

TopSolid

2021

TopSolid v6.22

What's New

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7, Rue du Bois Sauvage
F-91055 Évry, France
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Email: contact@topsolid.com
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Version 6.22 Rev.03

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Welcome to TopSolid 2021

Throughout this document, you will discover the main enhancements and learn about the new features and benefits of the latest **2021** release of **TopSolid v6**. The innovations described here only represent a small portion of all the new functionalities.

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

What's New in TopSolid'Wood v6.22



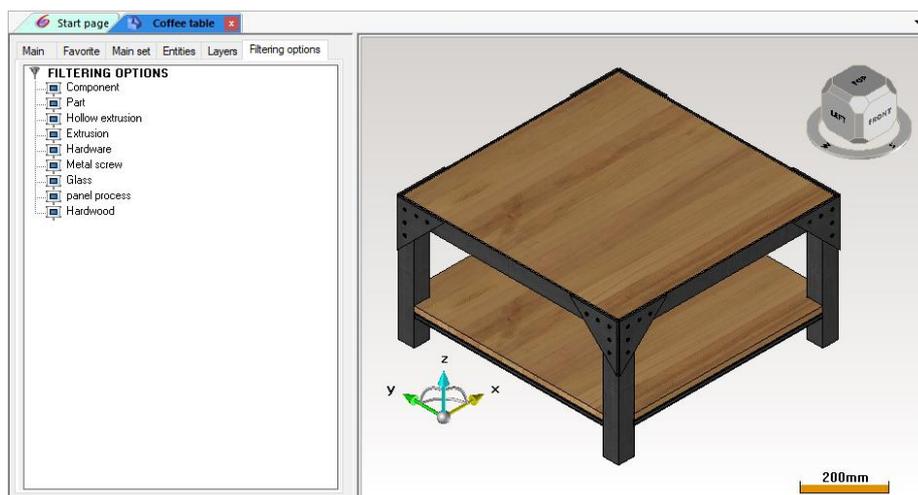
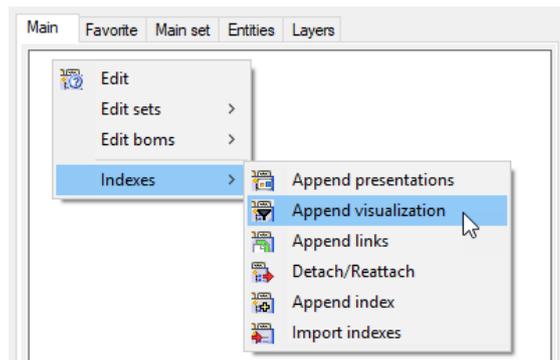
This section describes the enhancements made to version **6.22** of **TopSolid'Wood**.

User Interface

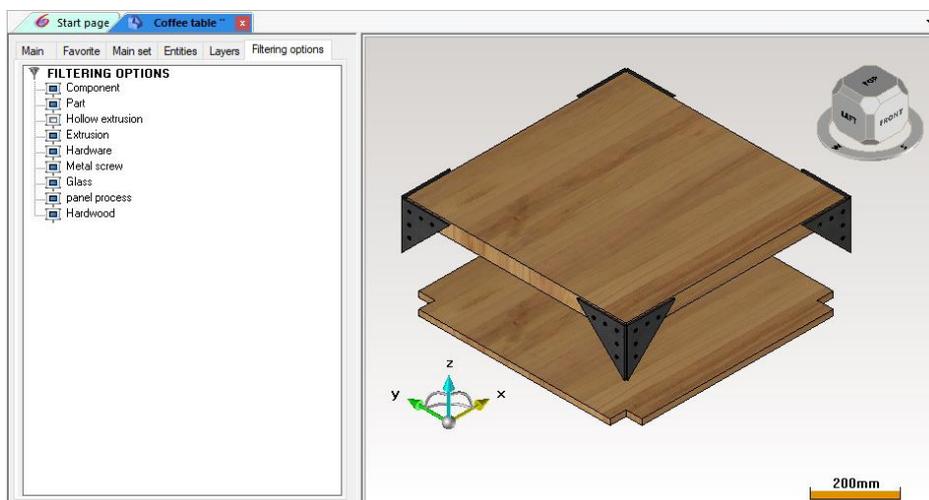
Tree

Filtering options

In the tree, you can add a tab by **right-clicking** in the tree and selecting the **Indexes > Append visualization** command.



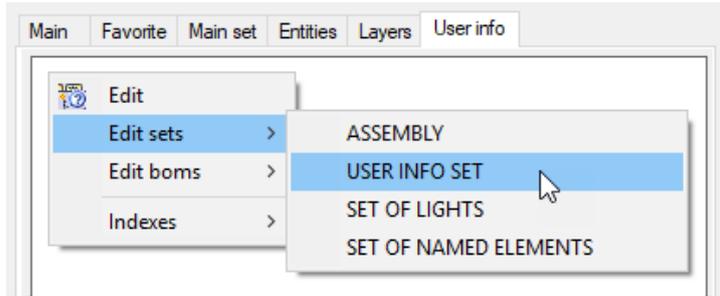
This tab allows you to quickly **enable** or **disable** display filters. To do this, you only have to double-click on the desired filter or right-click on the filter and select **Displayed = NO** to disable the corresponding elements.



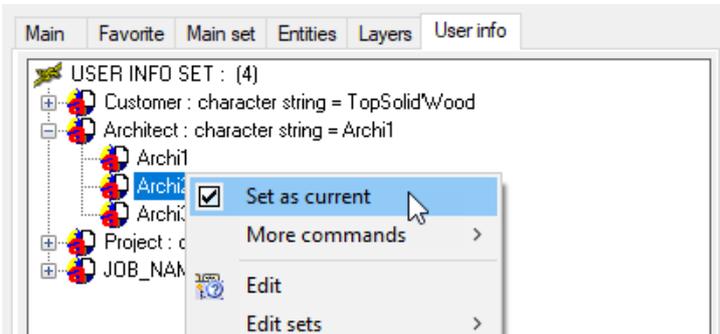
Note: When a filter is applied in the document, the symbol " is displayed after the document name and the filter icon is disabled in the tree.

General user information

You can also display and edit the general user information from the tree. To do this, you only have to **right-click** in the tree and select the **Edit sets > User info set** command.



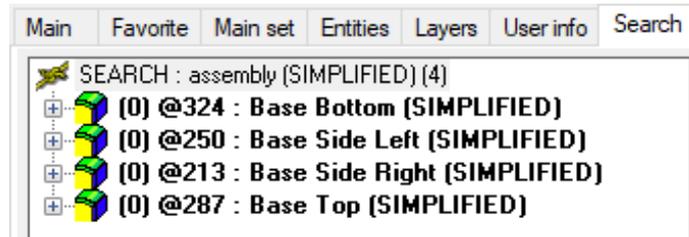
To enter the values, you simply have to **right-click** on the information and select the **Modify** command. You can also expand the information node, **right-click** on a predefined value, and then select the **Set as current** command.



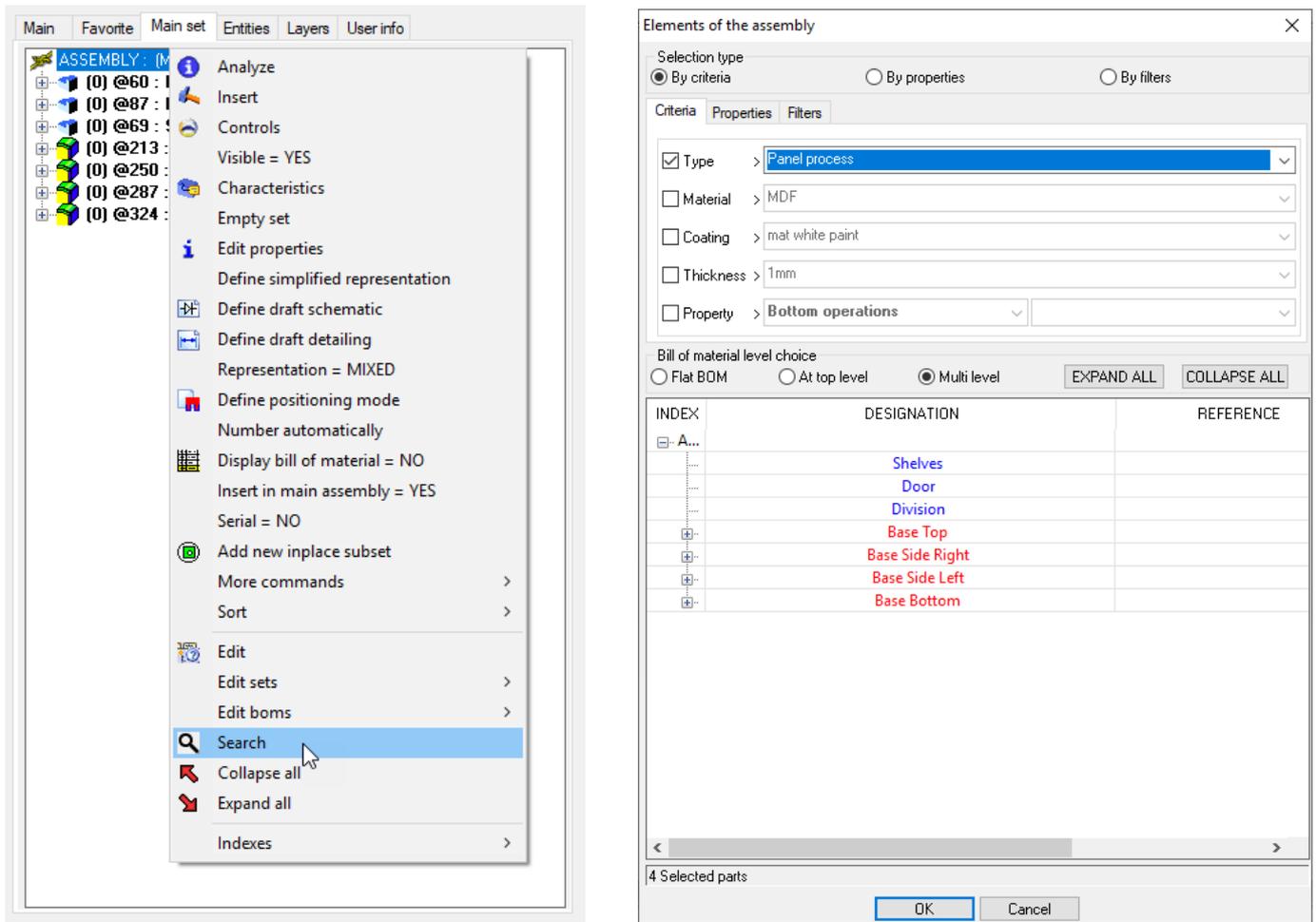
Reminder: This user information and its default values can be configured with the **Tools > Options > User information** command. You can also retrieve this information in a drafting document.

Search in the tree

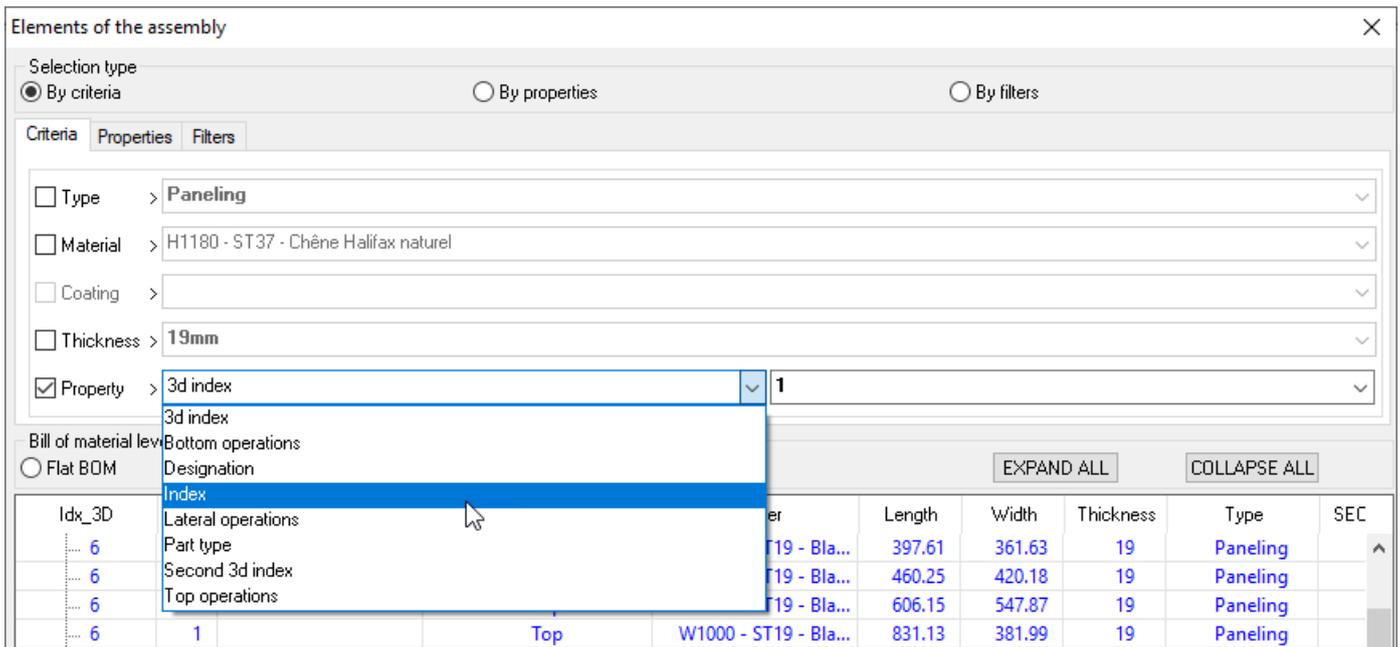
From the tree, you can launch a search to display only the desired elements in the new **Search** tab.



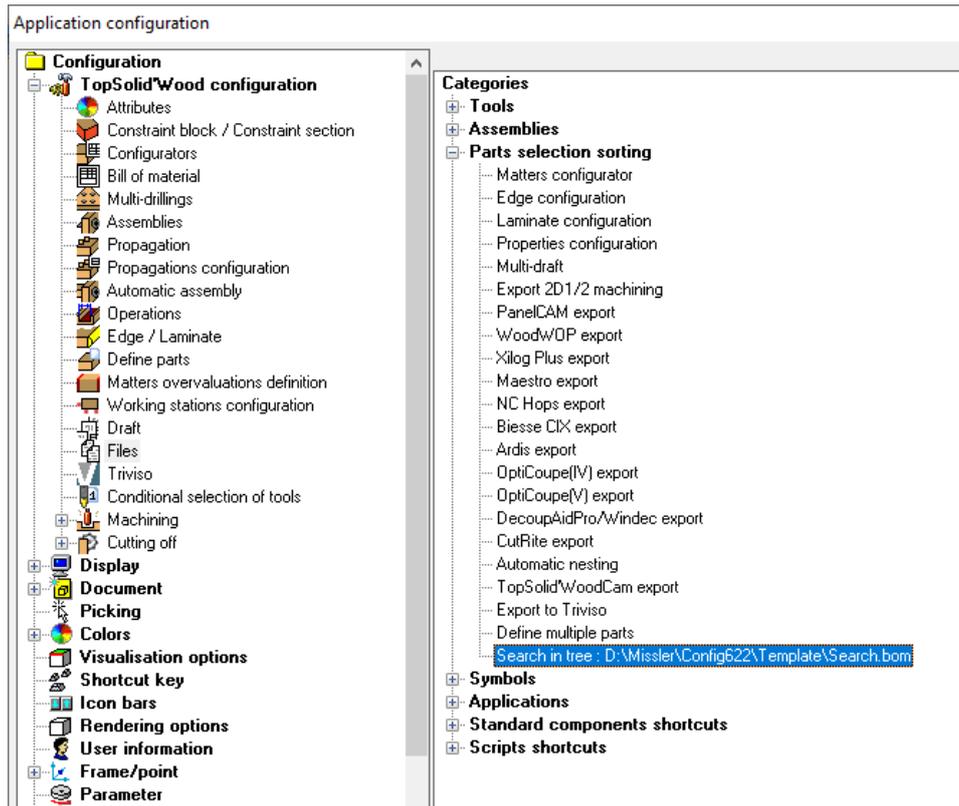
To do this, you only have to **right-click** on the assembly and select the **Search** command. In the dialog box that appears, you can then choose a selection type by criteria, by properties or by filters.



Three new properties allow you to search for elements from their 3D index. The first property indicates the presence of a BOM index (1 = presence of an index and 0 = no index), the second property concerns the BOM index and the third property concerns the secondary BOM index.

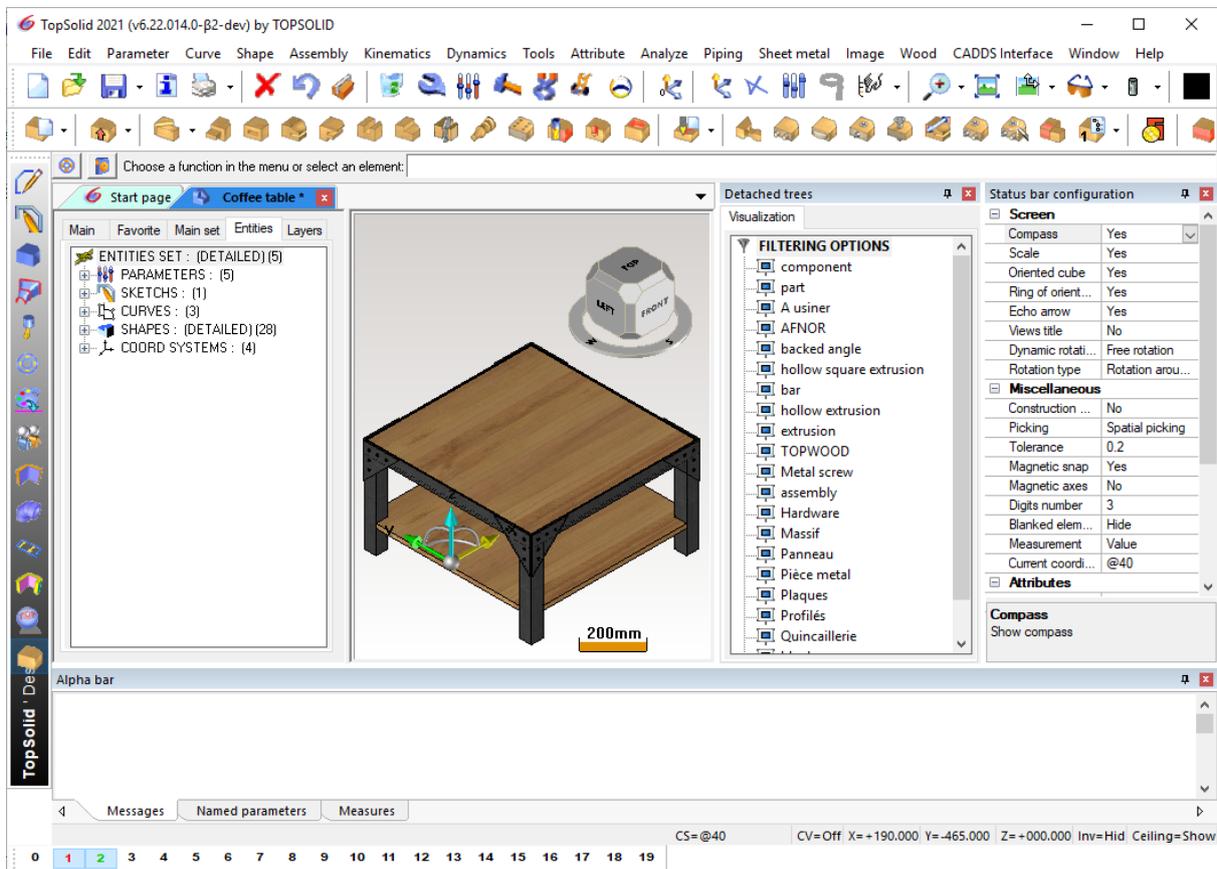


Note: You can set a bill of materials for the search in the **Tools > Options** command.

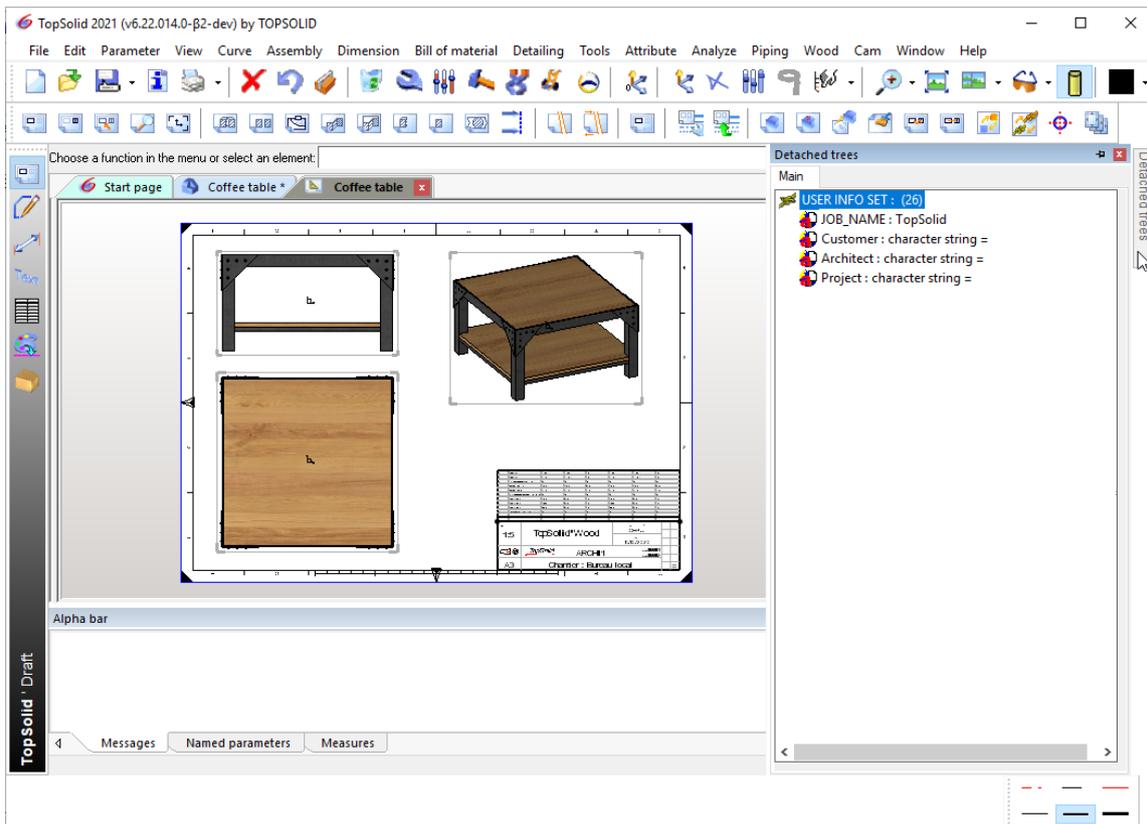


Detached trees

Since version 6.21, the trees can be detached. The tree positions are now saved by document type.



Example in a .top document: the detached tree is pinned to the left of the status bar configuration.



Example in a .dft document: the detached tree is located on the right side of the window and is retractable. The general tree is closed.

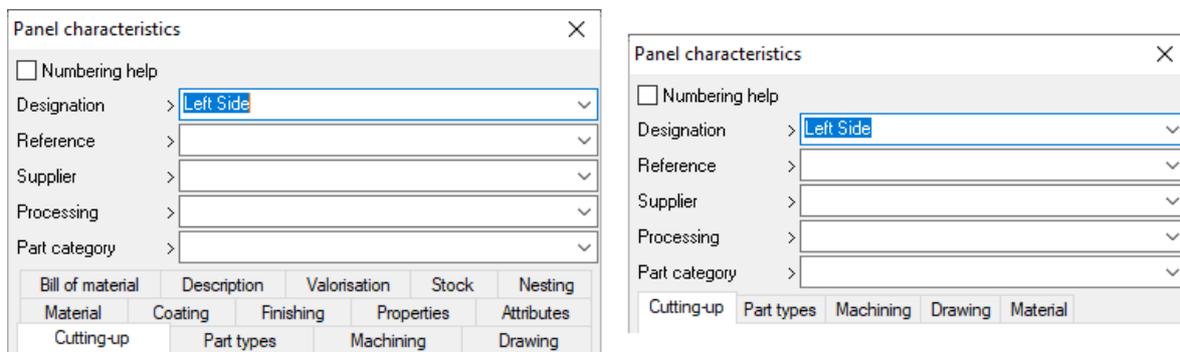
Note: This setting also affects the positioning of the status bar, but does not affect the contents of the detached tree.

Define part

Tab filtering

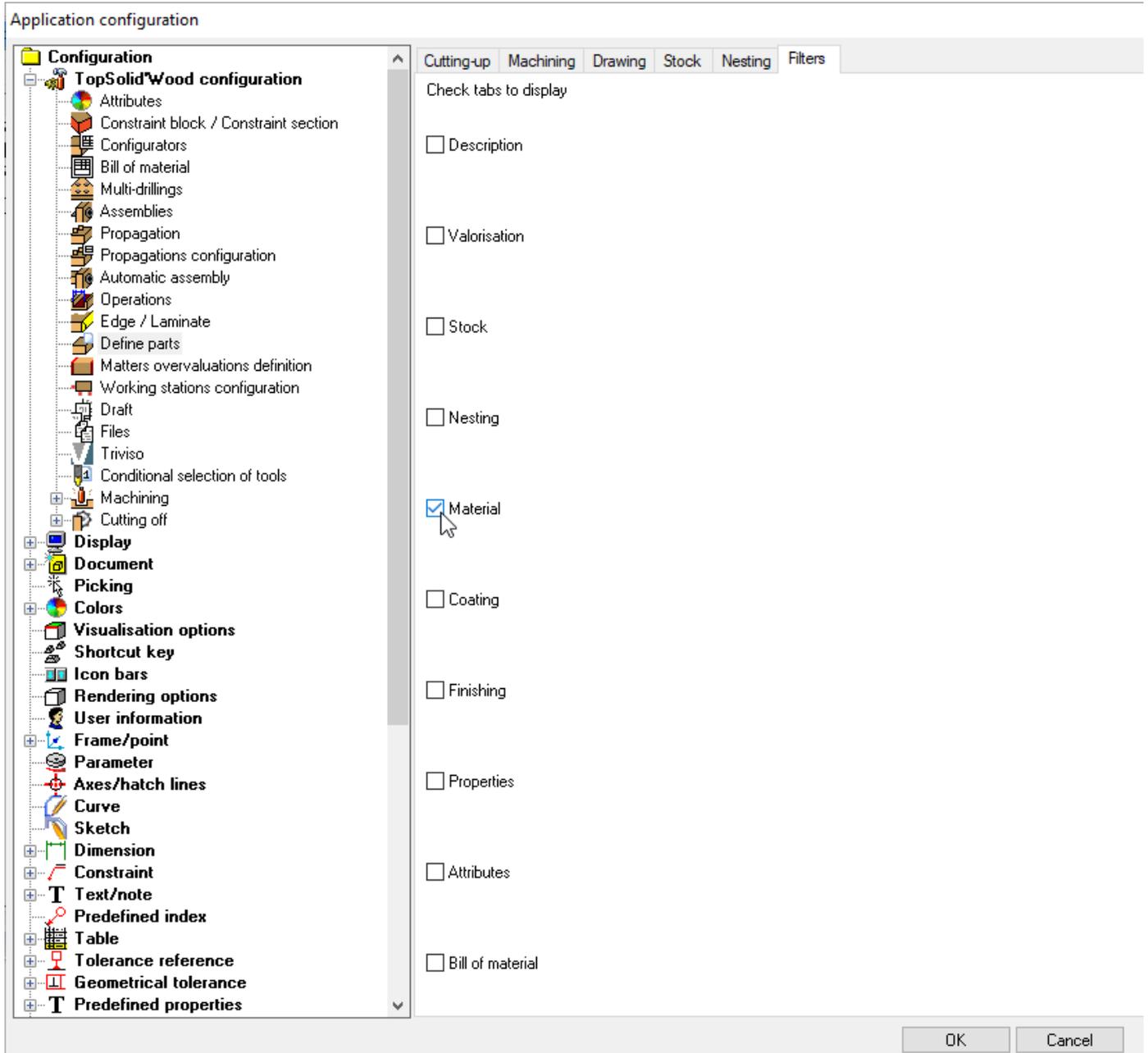
In the **Tools > Options > TopSolid'Wood configuration > Define parts** command, you can now filter the tabs available in the part definition dialog box. To be able to display the tabs in filtered mode, you only have to adjust the **Advanced mode** option to **NO** when selecting the **Assembly > Define part** command.

Assembly= MAIN ASSEMBLY | With sawing-up= YES | Select axis automatically= YES | Bent part= NO | Advanced mode= YES | Part(s) to define:



On the left, the **Advanced mode** option is set to **YES**. On the right, the **Advanced mode** option is set to **NO**.

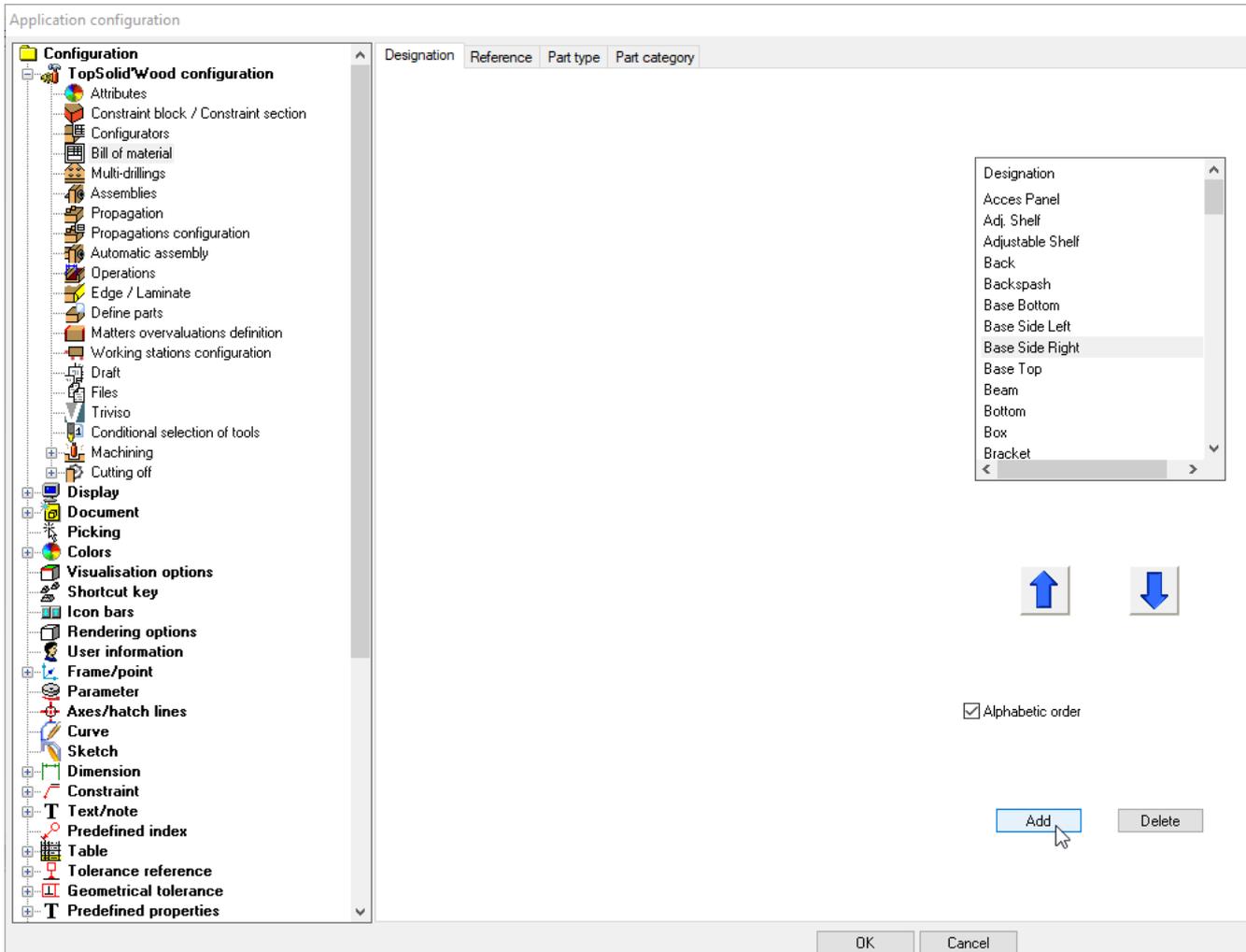
The tabs to be filtered must be unchecked in the **Tools > Options > TopSolid'Wood configuration > Define parts > Filters** command.



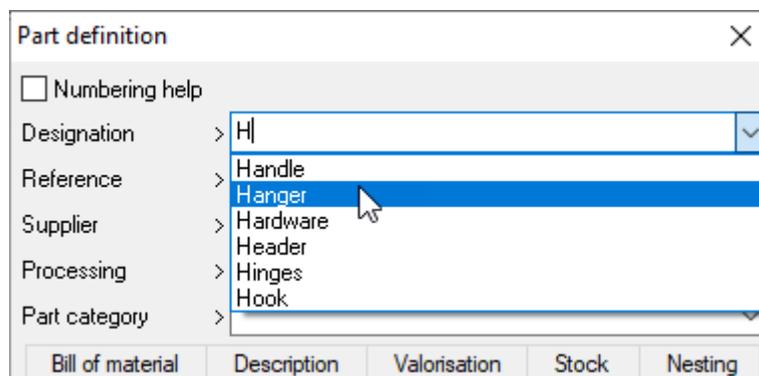
Note: When displaying the part characteristics from the Entities tree, the advanced mode is enabled and all tabs are therefore visible.

Semi-automatic data entry

In the **Tools > Options** command, you can add frequently used texts for the **Designation**, **Reference**, **Part type** and **Part category** properties in the **TopSolid'Wood configuration > Bill of material** section, as well as for the **Supplier** and **Processing** properties in the **Predefined properties** section.



These properties can now be filled in semi-automatically in the **Part definition** dialog box. To do this, you simply have to enter the first few characters of the predefined property and it will automatically appear in the drop-down list.



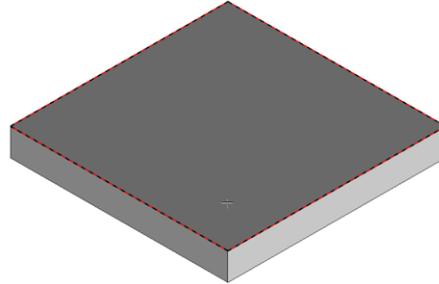
Note: You can use the **arrows**, the **Tab** and the **Esc** keys to move around, select a line or go backwards.

Neutral mode

The commands available in the neutral mode have been reorganized to provide quick access to the most frequently used commands, taking into account the selected elements.

Here are some examples of the commands available in this new version when selecting the following elements:

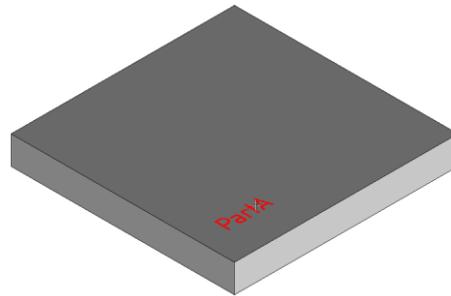
- A face on a flat part:



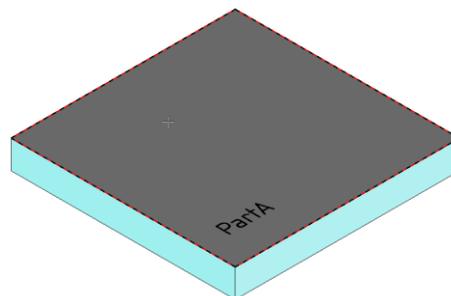
Note: You can show or hide additional commands by clicking on the + or – button.



- A text:



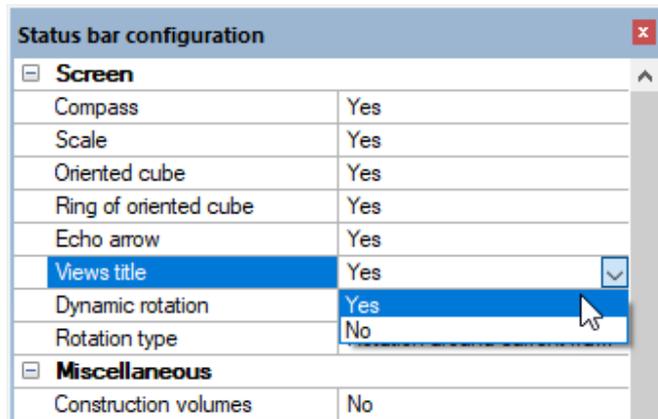
- A side of a panel:



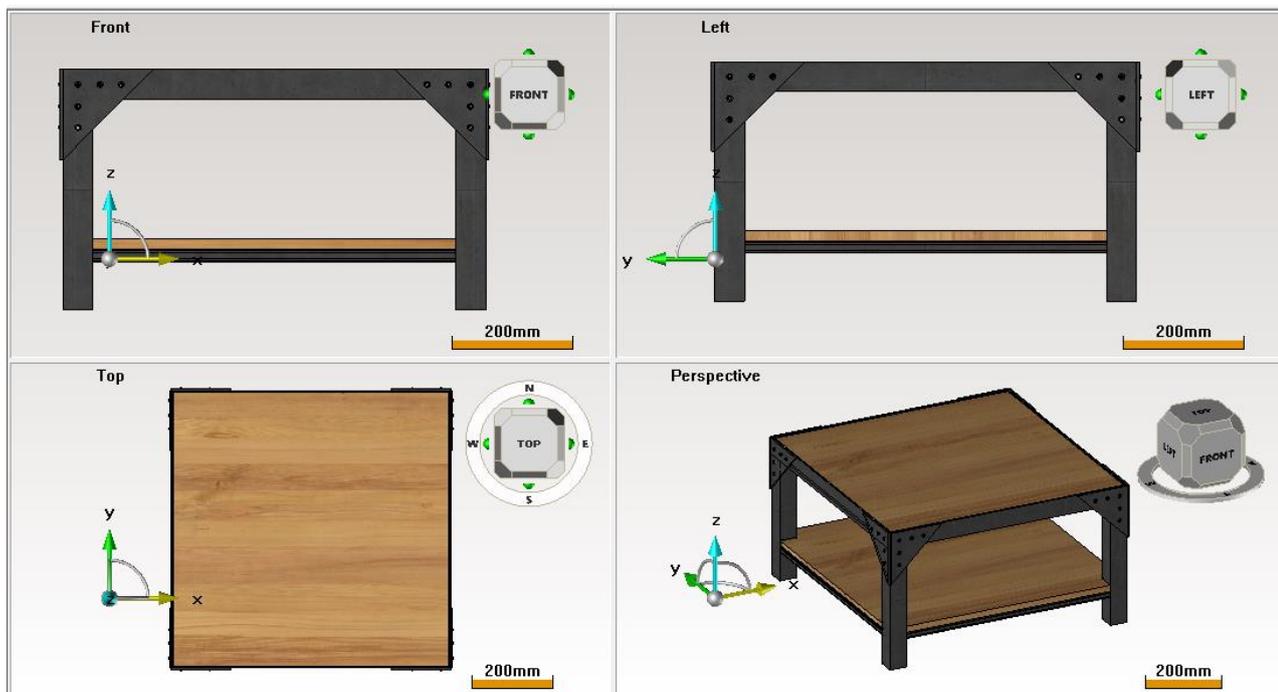
Graphics area

View titles

You can now display the titles of the views in the graphics area. This can be configured from the **Status bar configuration** dialog box by selecting **Views title = Yes**.



Note: This option is useful when the screen is split into multiple views.

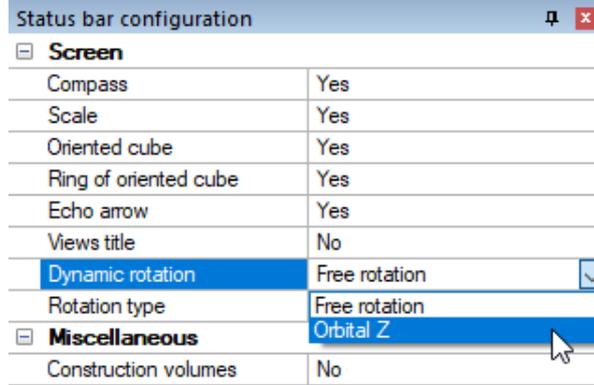


Rotation

There are now two modes of rotation in the space:

- **Free rotation:** This mode allows the element to be rotated along all three axes equally.
- **Orbital Z:** The absolute Z axis is predominant for the rotation. This mode is to be preferred for layout.

You can change the mode from the **Status bar configuration** dialog box.



The **Rotation type** option offers the following two modes:

- **Rotation around point:** You only have to click on a physical point in the 3D file to rotate to that point.
- **Rotation around current frame:** The rotation is performed around the absolute frame.

Note: You can use the **Rotation around point** mode via the **Ctrl + mouse wheel click** shortcut.

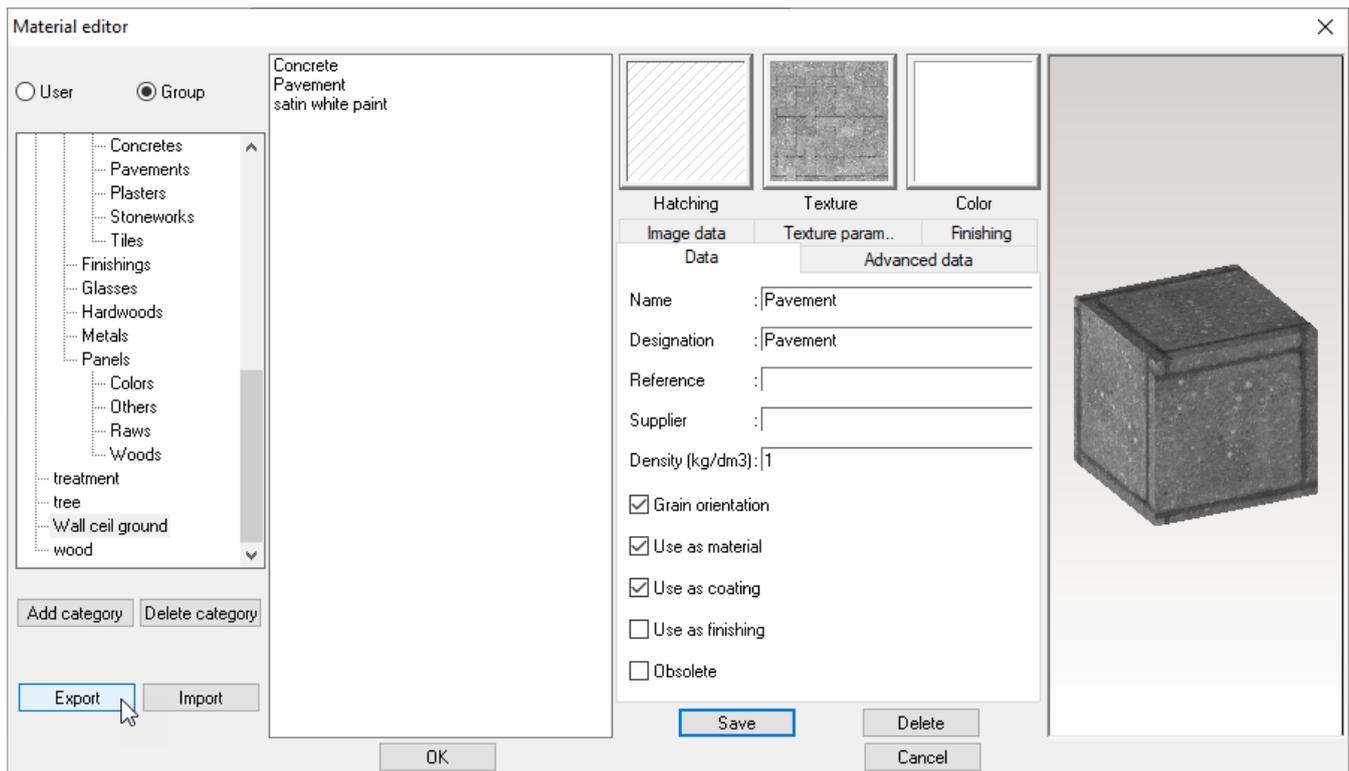
Design

Layout

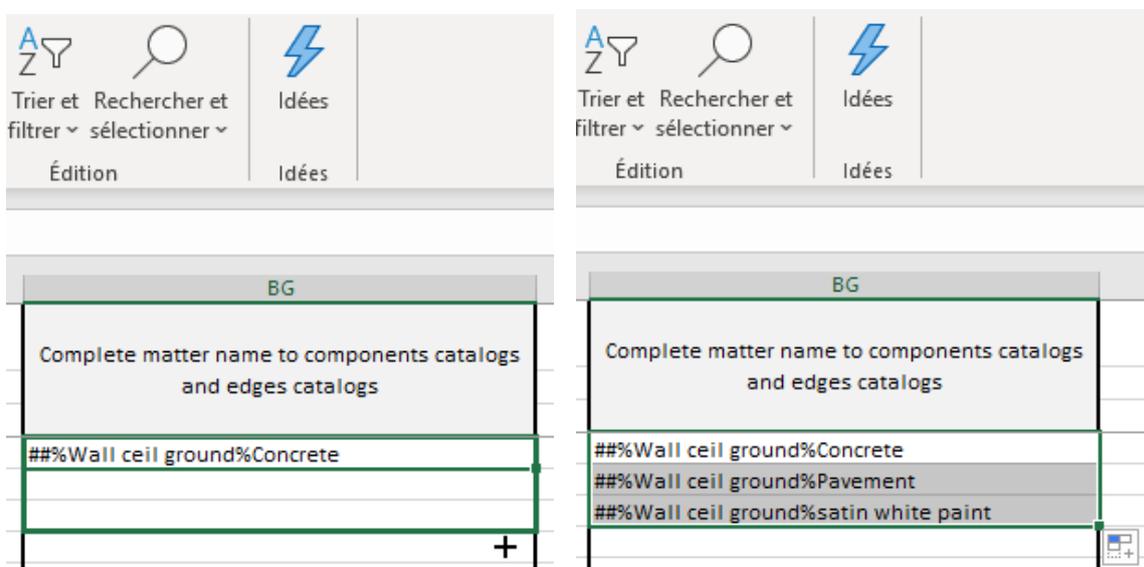
Wall, ground and ceiling: Default material and coating

You can adjust the default material and coating for the walls, grounds and ceilings using configuration words to be added to the *top.cfg* file. To do this, you simply have to proceed as follows:

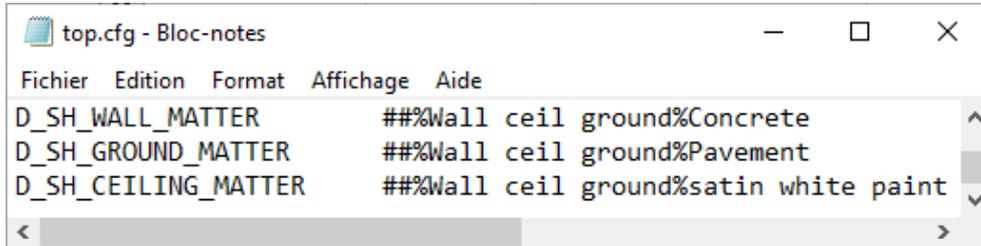
- Select the **Attribute > Material > Manage materials** command and create the materials and coatings.
- Click on the **Export** button to generate an Excel file containing the materials and coatings.



- In the resulting Excel file, go to the last column and drag the mouse cursor to display all the names of the materials and coatings.



- From the Excel file, copy the full name of the generated material or coating and paste it into the *top.cfg* file after the corresponding configuration word:
 - **D_SH_WALL_MATTER**: Allows you to adjust the default material of the walls.
 - **D_SH_WALL_COATING**: Allows you to adjust the default coating of the walls.
 - **D_SH_GROUND_MATTER**: Allows you to adjust the default material of the grounds.
 - **D_SH_GROUND_COATING**: Allows you to adjust the default coating of the grounds.
 - **D_SH_CEILING_MATTER**: Allows you to adjust the default material of the ceilings.
 - **D_SH_CEILING_COATING**: Allows you to adjust the default coating of the ceilings.



- You need to restart **TopSolid** for these settings to take effect.

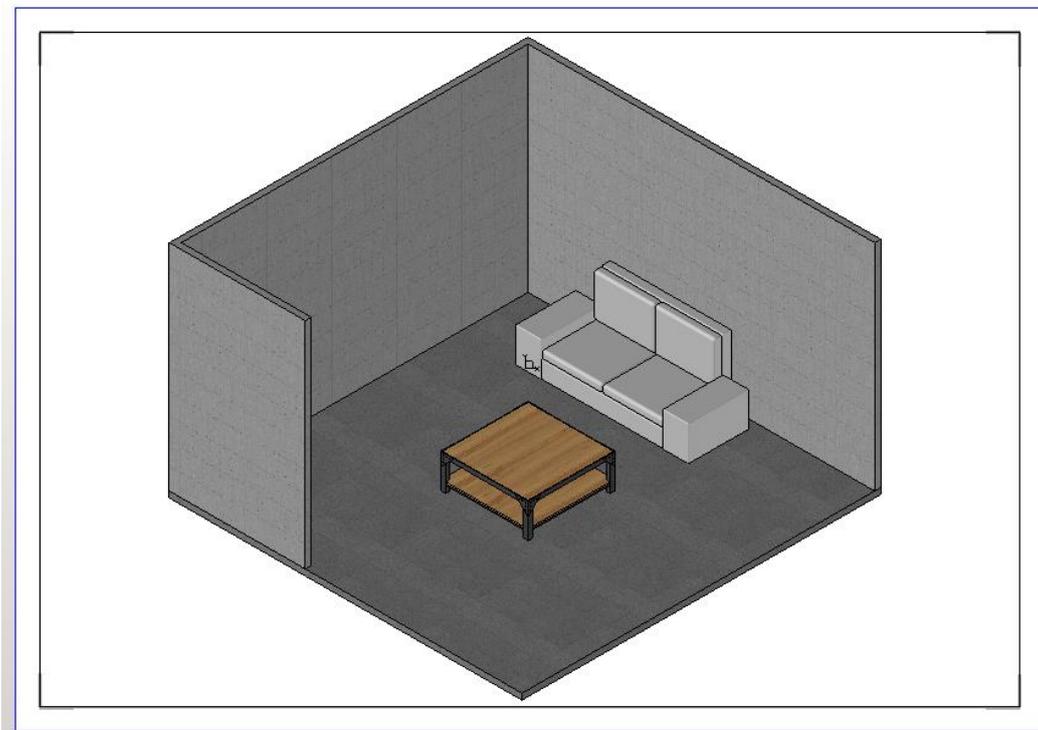
Note: Each configuration word must be separated from its value by one or more tabs.

Wall, ground and ceiling: Main set

The new **Add to main set** option allows you to add the walls, ground and ceiling to the main set. To do this, as soon as you select the **Wall** or the **Floor/Ceiling** command, you only have to adjust the **Add to main set** option to **YES**.



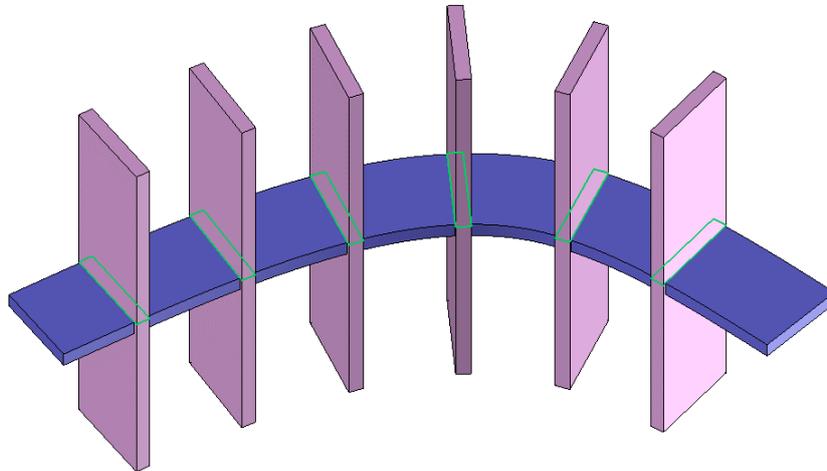
Adding these elements to the main set will allow you to display them in the drafting document, for a complete view for example.



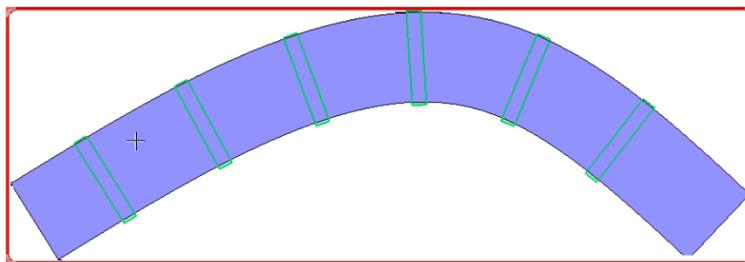
Shape

Division: Keep the outside of the curves

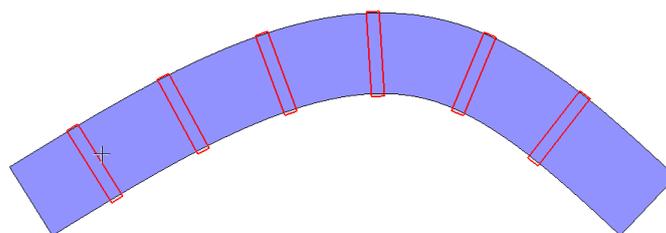
In addition to the existing **Keep the inside of curves** mode, a new **Keep the outside of curves** division mode is available.



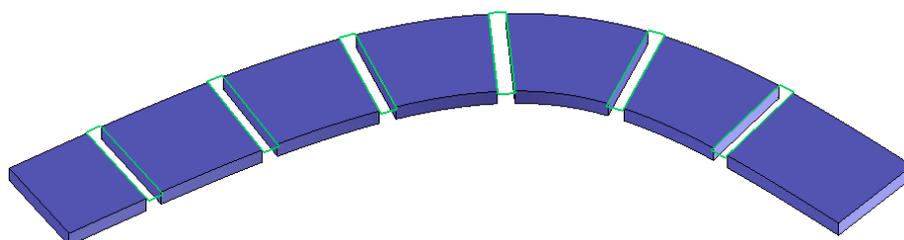
- Select the **Shape > Other shapes > Division** command.
- Select the shape to be divided.



- Select the profiles that will be used to perform the division.

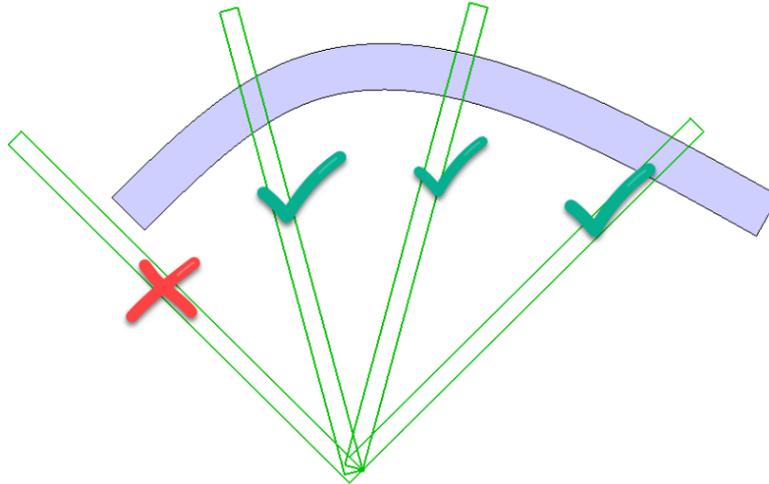


- Select the **keep the outside of curves** mode to divide the part by removing the material inside the profiles.



Note: The division using the **keep the outside of curves** mode is possible provided that the following conditions are met:

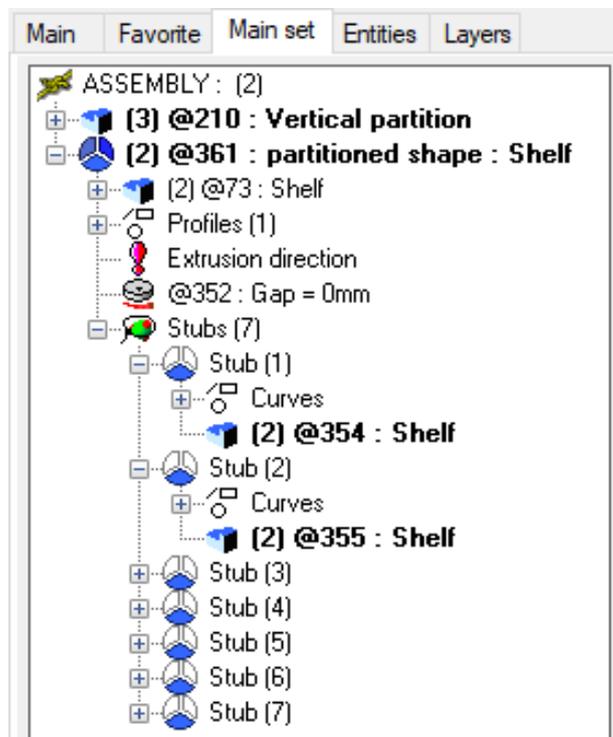
- The curves must belong to a list (repetition/serial list).
- The curves must be closed.
- The curves must be larger than the part.
- The curves must be inside the part.



If these conditions are not met, the error message **Bad list of curves, one curve does not cut the shape into two pieces** is displayed in the alpha bar.

Division: Visualization in the tree

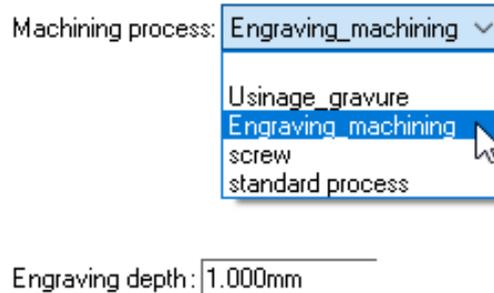
In the tree's **Main set** tab, the designation is now displayed on the line of the partitioned shape, on the line of the basic shape and on the line of each stub.



Note: If the model designation is modified, the designations of all other elements will also be modified, unless the designation has been changed manually on one of the stubs.

Engraving

In the **Tools > Options > TopSolid'Wood configuration > Operations > Engraving** command, you can now adjust a machining process and a depth for the engraving operation.



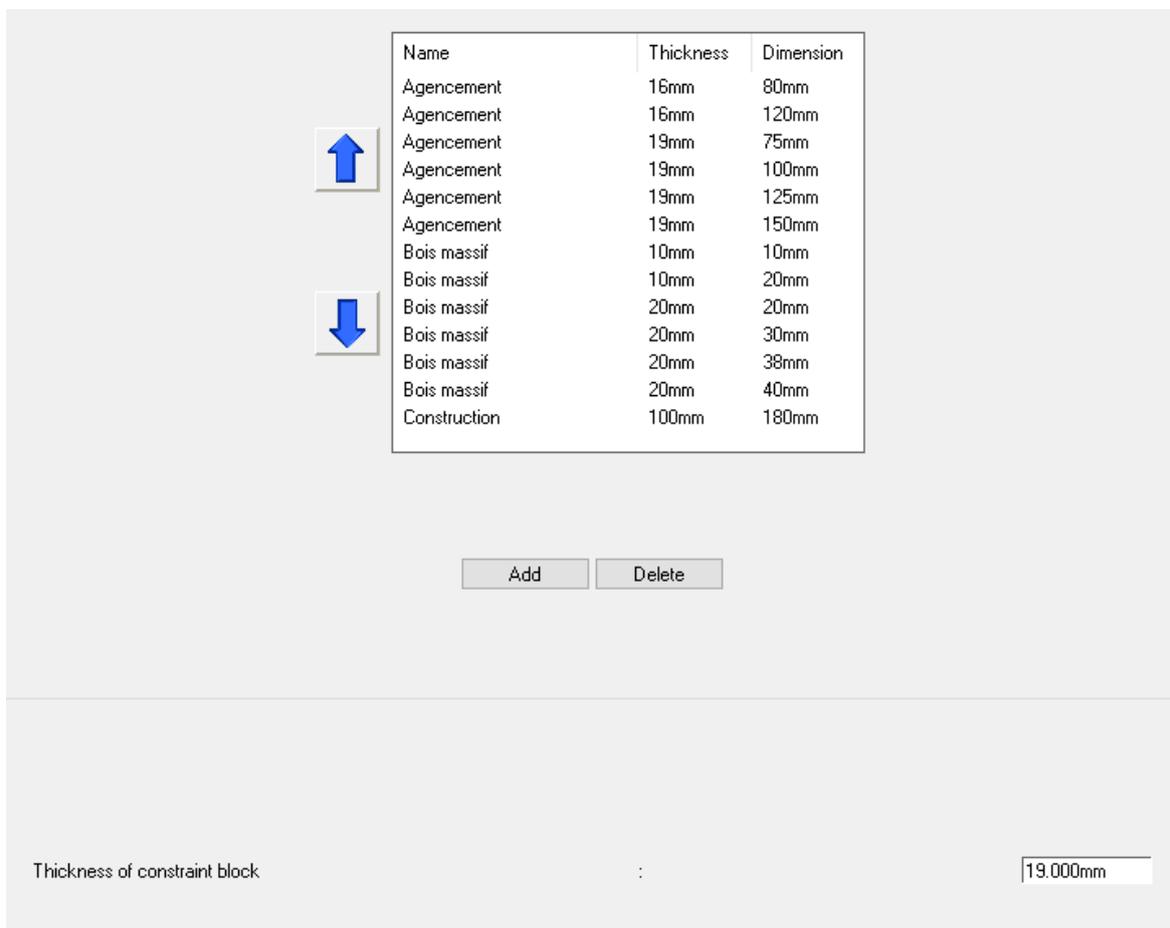
These options will be preset when using the **Wood > Engraving > Engraving > Engrave the definitions** or **New text** command.



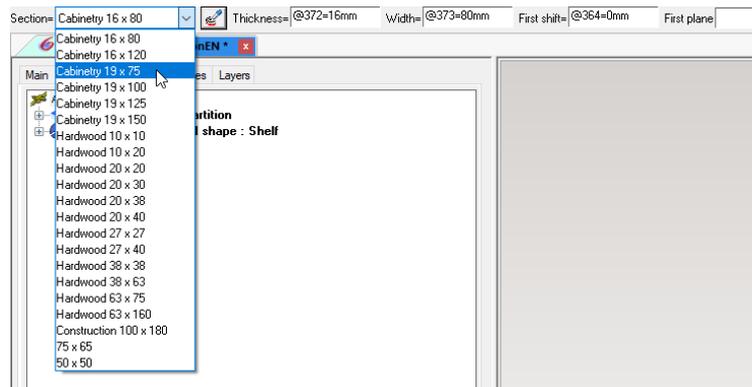
Reminder: The machining processes are defined in the *top.mac* file located in the personal configuration folder of the software.

Constrained section

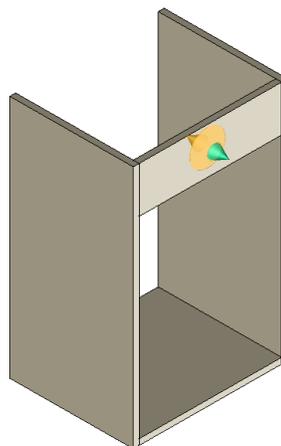
As for the thickness of the constrained block, you can now establish a list of constrained sections in the **Tools > Options > TopSolid'Wood configuration > Constrained block/Constrained section** command.



This list will then be available in the **Section** drop-down list of the **Wood > Constrained section** command.



Note: The constrained sections entered on the fly will always be available at the end of the drop-down list, after the sections defined in the **Tools > Options** command.

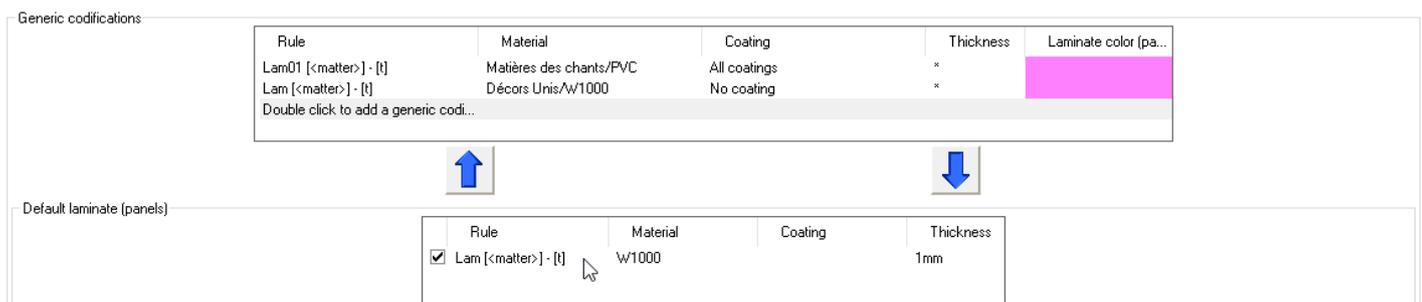


Panels

Default laminate

If you are always using the same laminate, it is useful to set a default laminate. It is now possible to do this in the options.

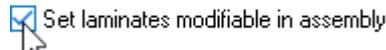
- Select the **Tools > Options > TopSolid'Wood configuration > Edge/Laminate** command and click on the **Laminate configuration** tab.
- Double-click in the **Rule** column.
- In the dialog box that appears, select the **material**.
- In the second dialog box that appears, select the **coating**.
- Enter the **thickness** of the laminate.
- **TopSolid** automatically recognizes the **generic codification** set just above.



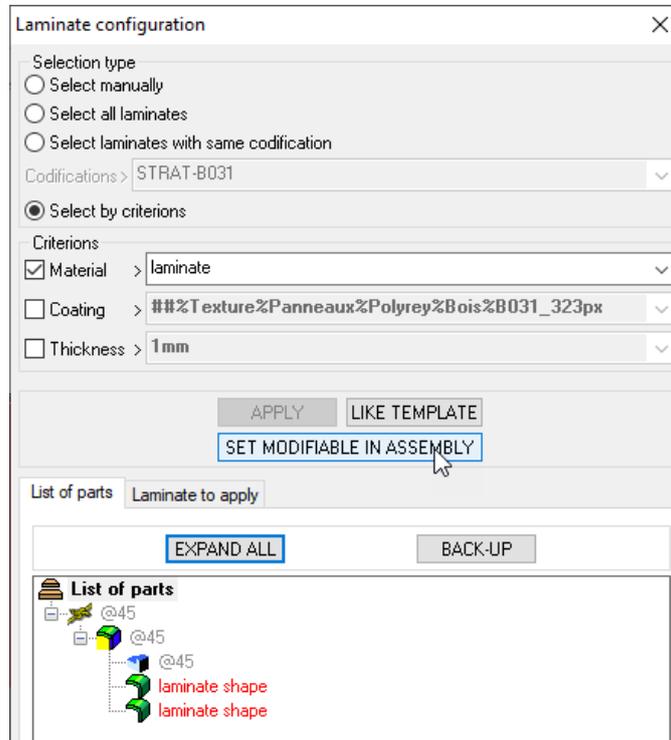
Note: The default laminate will only be available when using the **Panel** command.

Setting the laminates modifiable in the assembly

In the **Tools > Options > TopSolid'Wood Configuration > Edge/Laminate** command, you can now check the **Set laminates modifiable in assembly** box for all laminates.



This option corresponds to the **Set modifiable in assembly** button available in the laminate configurator.



User machining on a sloped face

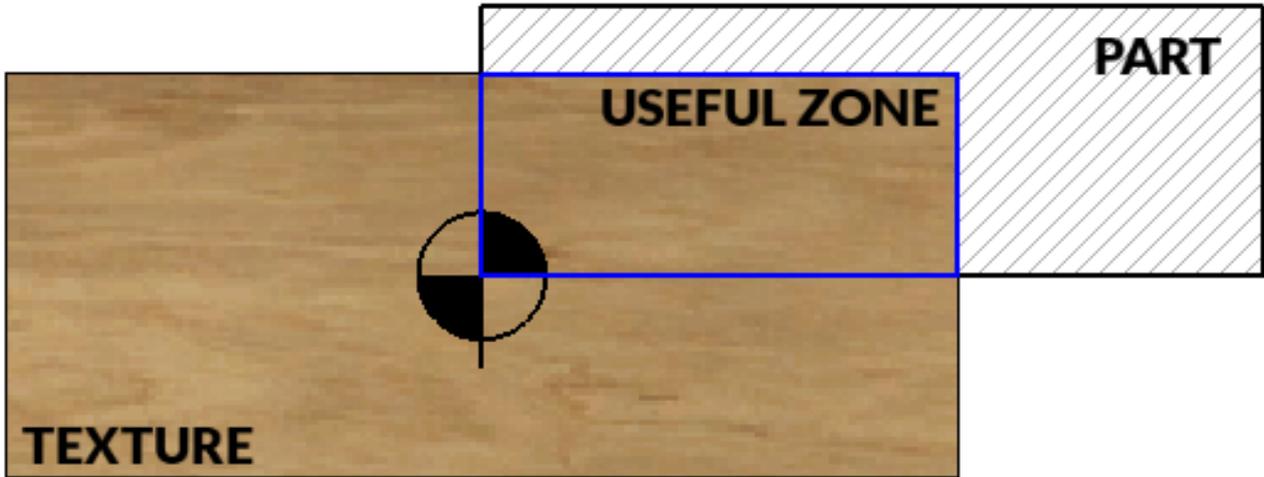
When performing a user machining on a sloped face, you can now enter the tool depth. This value is automatically filled in by detecting the part thickness when performing the operation.



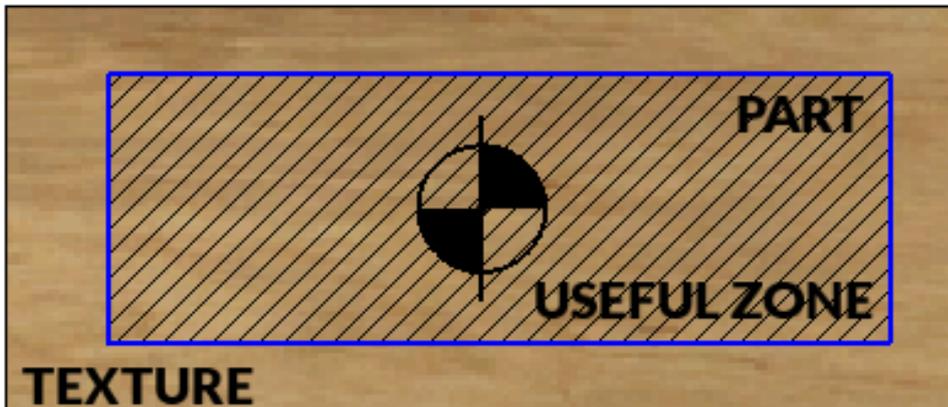
Note: The depth is calculated when the user machining is performed. This value is not associative with the part thickness. This is why it is advisable to use a parameter.

Materials

For the creation of the materials, the positioning of the texture has been reviewed to limit the demarcations between the texture repetitions on large parts.



2020



2021

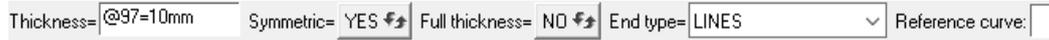
Until the 2020 version, the center of the texture was placed in a corner of the part. In the 2021 version, the center of the texture is located at the origin of the part. The entire surface of the texture is now used.

Sketch and Curve

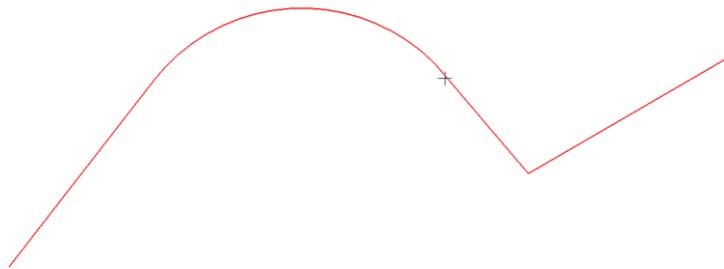
Thickened curve

For the contours consisting of several curves, you can now serialize the construction of the thickened curve. This new feature allows you for example to create elements on these curves. These elements will then follow the changes made to the curves.

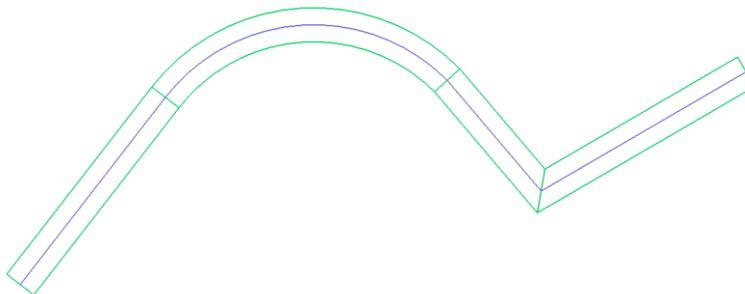
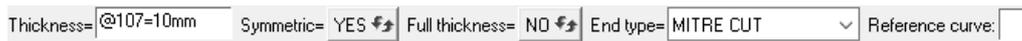
- Select the **Curve > Serial curves > Thickened curve** command.



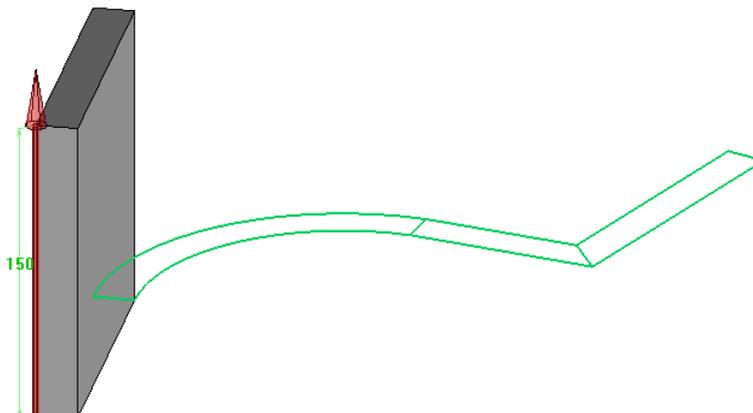
- Select the **reference curve**.



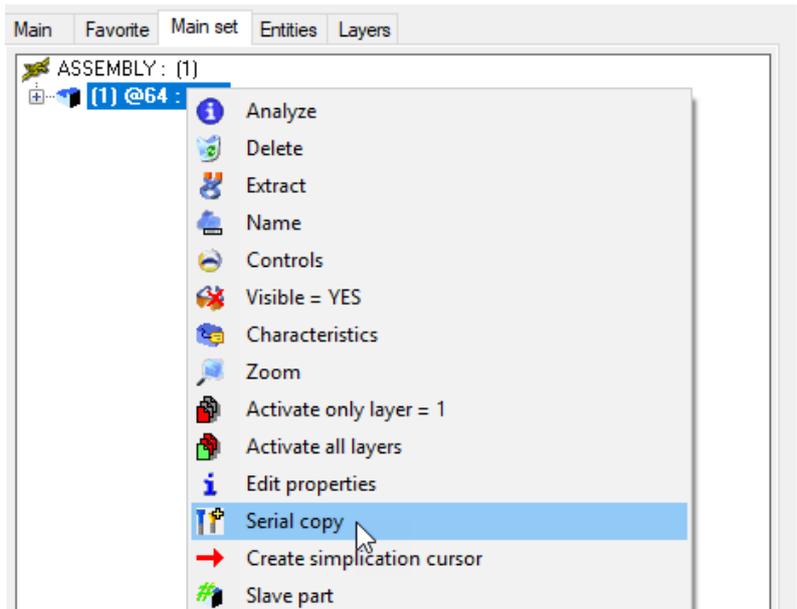
- Adjust the **thickness** and the **end type**, then adjust all the necessary options to create the thickened curve.



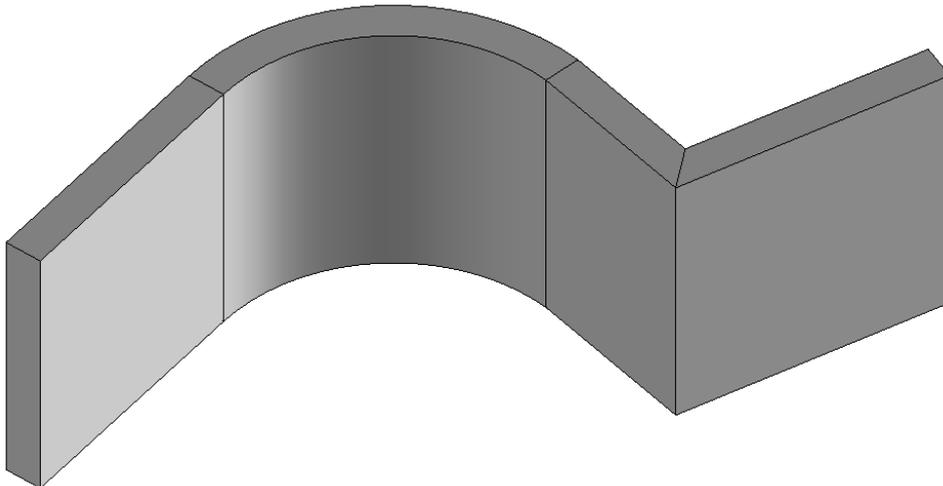
- Select the **Shape > Extruded** command and select one of the closed curves.



- Define the part, then **right click** on the part from the tree and select the **Serial copy** command.

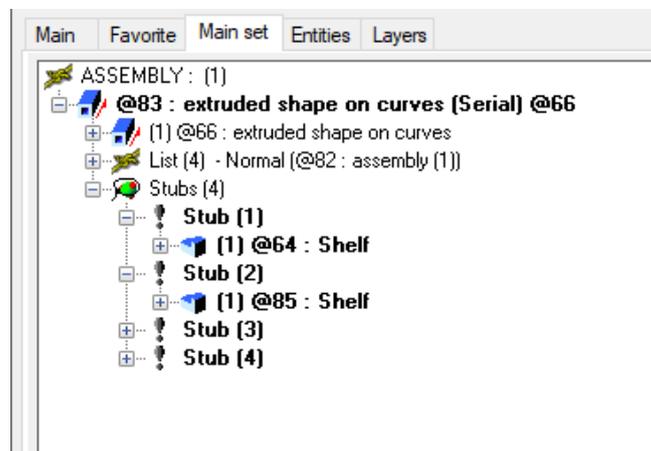


- Select the **Automatic** mode.



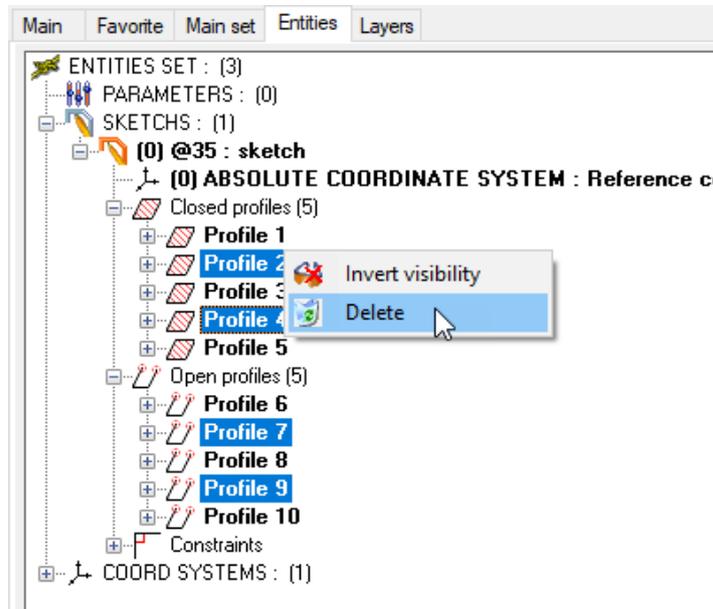
Notes: You can also launch this command from the **Curve > Thickened curve > SERIAL** command.

After the copying operation, the available stubs in the tree will have the same designation as the part defined before copying.



Deletion in the sketch

From the Entities tree, you can now delete or make invisible several profiles at the same time. To do this, the sketch must be in edit mode. Then you simply have to select the different open or closed profiles, **right-click** and select the **Delete** command.

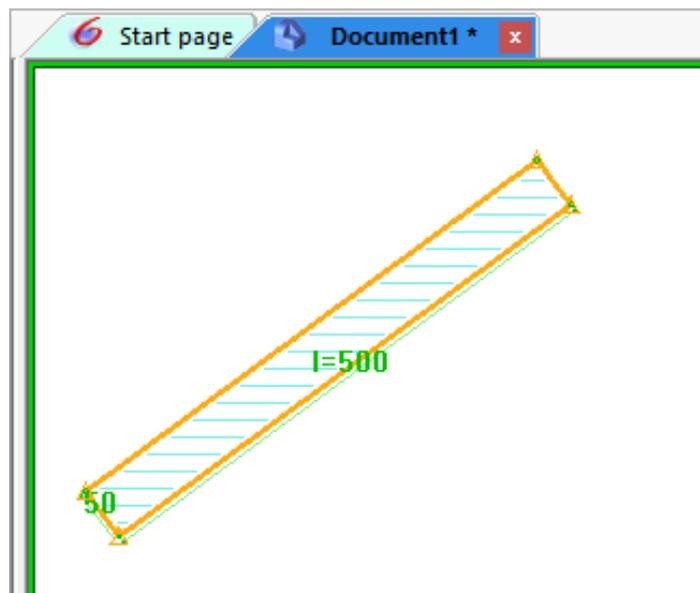


Note: Any constraints that depend on these profiles are also removed.

Three-point rectangle and value

When creating a three-point rectangle, you can now enter a value or parameter instead of the third point.

- Create a contour.
- Select the **RECTANGULAR** mode, then select the **3 POINTS** mode.
- Select the first two points.
- Enter a value or parameter for the third point.



Notes: This new mode is available in the **sketch** and in the **Curve** menu.

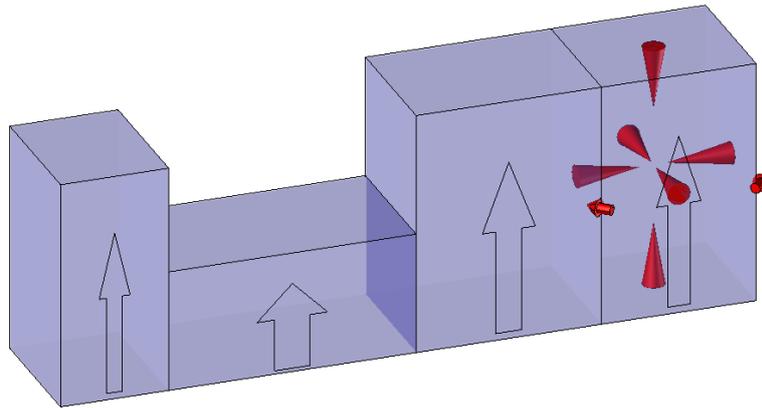
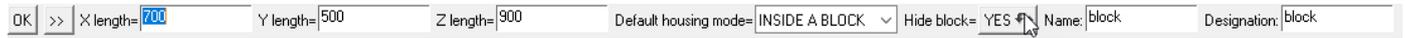
For a rectangle created from the **Curve** menu, you can modify the value of the third point from the tree.

Components

Driver block

Visibility of the driver block

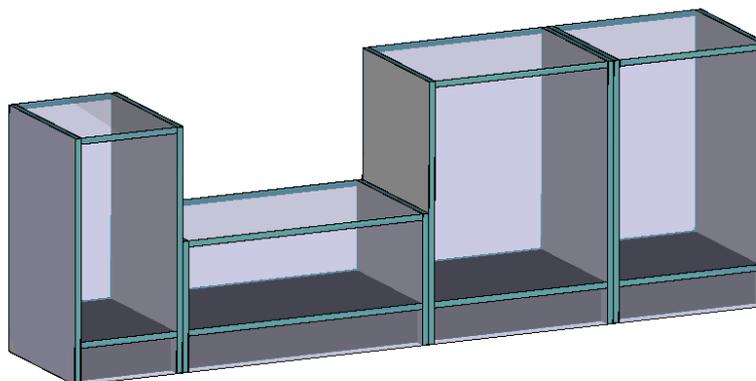
When creating the driver block, the new **Hide block** option allows you to hide the block when inserting the component.



Result after inserting the driver block in **Hide block = YES** mode:



Result after inserting the driver block in **Hide block = NO** mode:

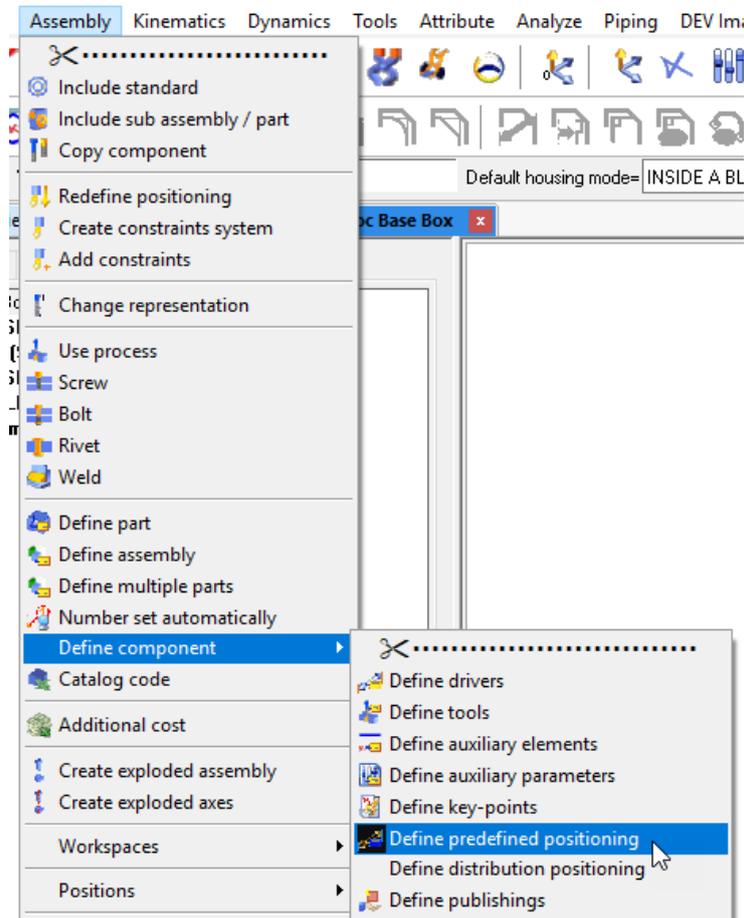


Reminder: The **D_DRIVER_HOUSING_DEFAULT_NAME** configuration word is used to set the default name of the driver block.

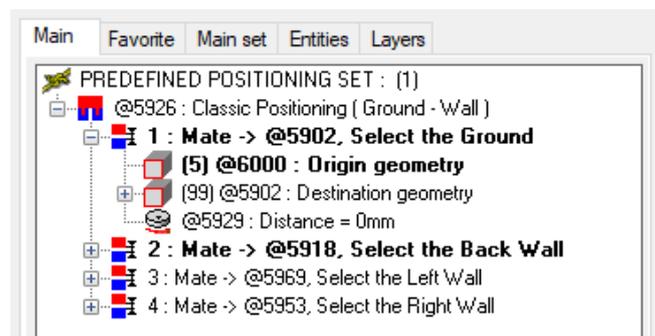
Positioning of the driver block

When inserting a component containing a driver block into the assembly in **DIMENSIONS** mode, you can now insert it with its predefined positionings.

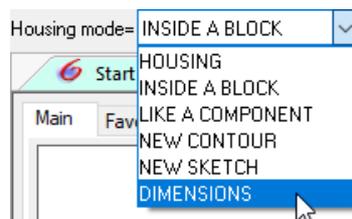
- In the component, create a predefined positioning via the **Assembly > Define component > Define predefined positioning** command.



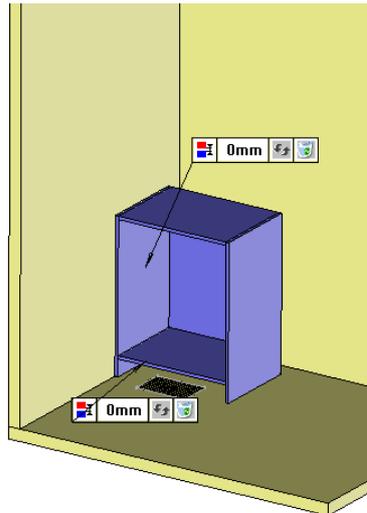
- Select the origin and destination faces of the positionings, then name these positionings.



- In the assembly, insert the component in **DIMENSIONS** mode, then enter the component dimensions.



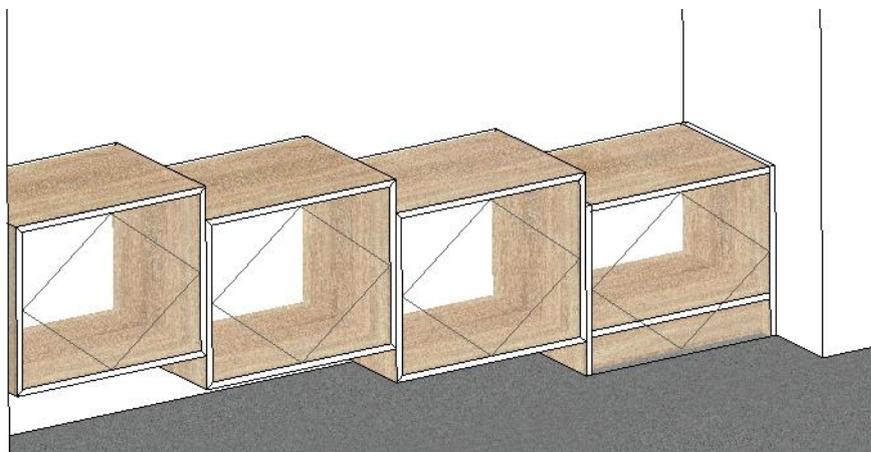
- Select the **Predefined positioning** option, then select the destination geometries that will constrain the positioning of the component.



Note: To create the predefined positioning, the origin geometries must be elements located in the main set. Do not choose the faces of the driver block.

Serial copy

When inserting a component with a driver block into an assembly containing a repeated or distributed block, the new **SERIAL COPY** option available at the end of the **Assembly > Include standard or assembly > Include sub assembly/part** command allows you to make a copy in the other blocks.



Curved component

Construction: Coordinate systems

When using the **Curved component driver** command, a coordinate system is created at each end of the curve. The second coordinate system is used to stabilize the construction of the component when it is inserted into the assembly.



Construction: Simplified shape

You can now simplify the construction of the curved geometry created from the **Shape > Pipe** command.

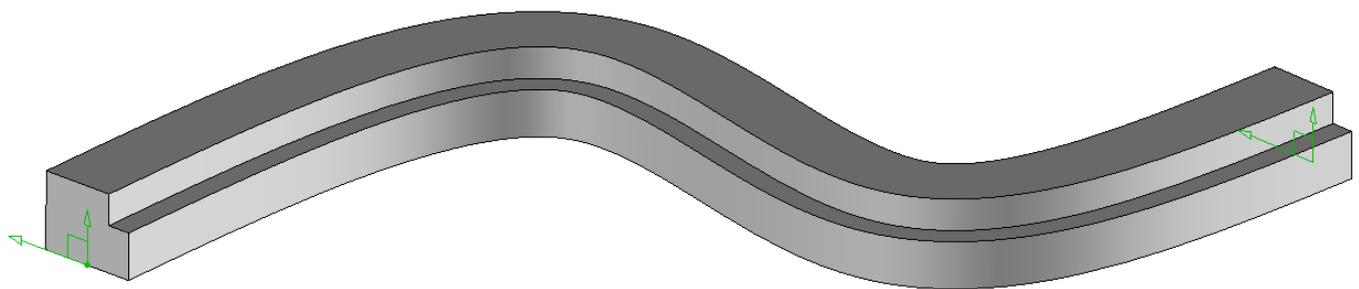
- Create a curve that will drive the pipe shape.
- Draw the section.
- Select the **Shape > Pipe** command and select the **ON CURVES** mode.



- Adjust the **Simplify geometry** option to **YES**.



Simplifying the geometry will allow you to perform operations on the shape that are not allowed on a curved surface, such as groove or rabbet operations.



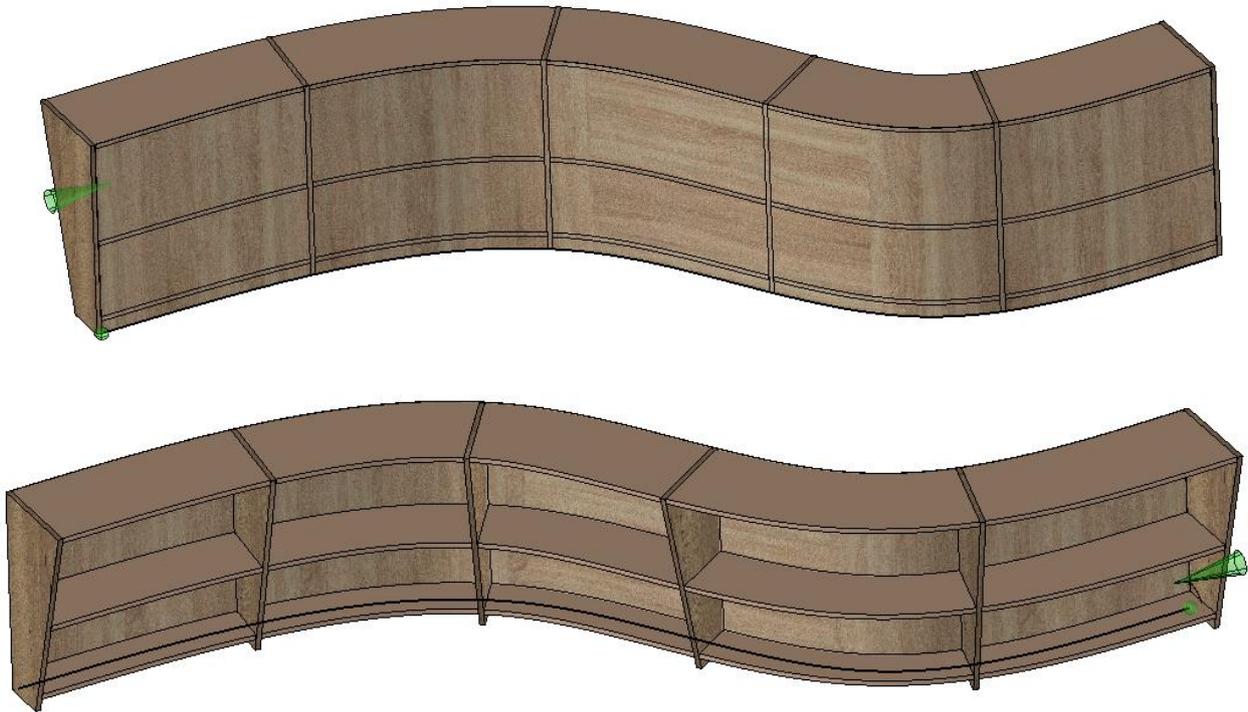
As another example, assemblies have been added onto the united parts that come from a pipe shape that has been divided.



Notes: This simplification mode only exists for the pipe shapes built on curves. You can modify the already built pipe shapes to apply the simplification option.

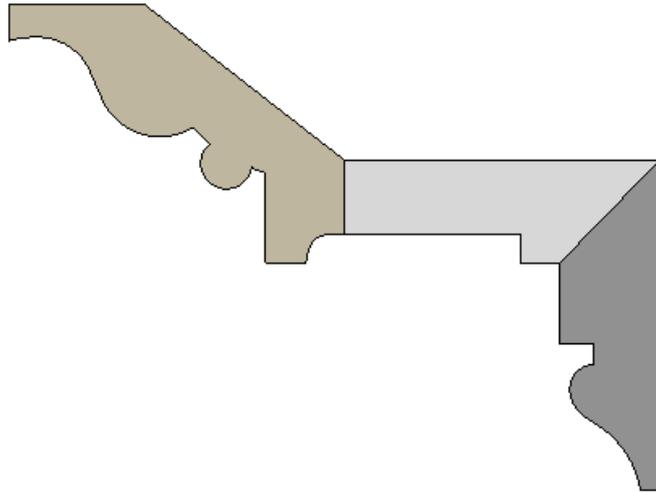
Positioning

In the assembly, when including a curved component, an arrow now allows you to adjust its direction and to reverse the start face and end face.

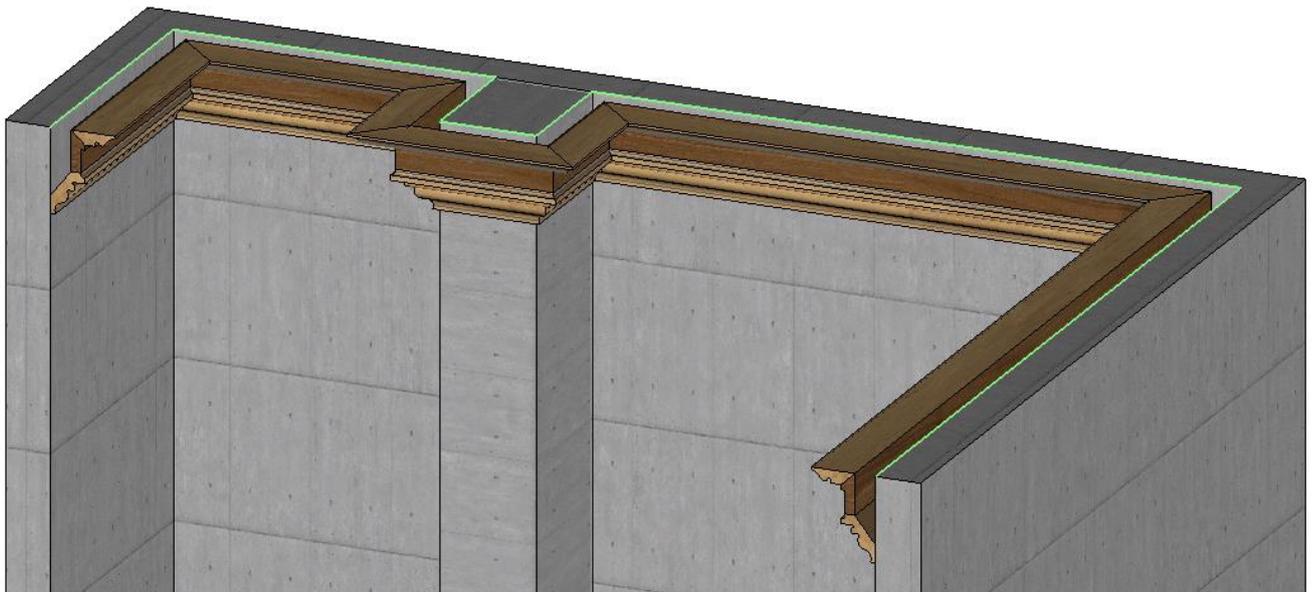


Multiple cuts

You can now perform mitre cuts on multi-body components.



Component section: three shapes.



The component is inserted on the green profile and the mitre cuts have been enabled.



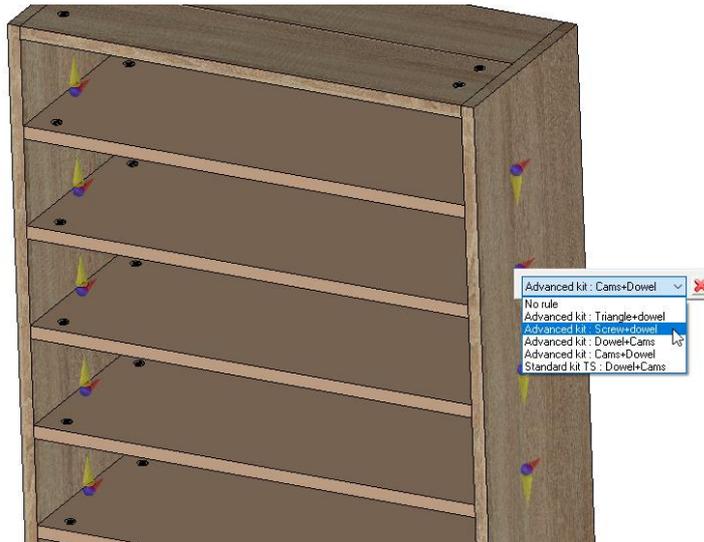
Detail view of the cut.

Assembly kit

Changing the kit

A new handle represented by a blue sphere is available when positioning an assembly kit. This allows you to change the kit template directly from the graphics area.

- Select the **Wood > Other processes > Automatic assembly** command.
- Select the **MAIN ASSEMBLY** option for the assembly.
- Select an assembly **rule**.
- Click on the **blue spheres** and select the different rules to be applied from the drop-down list according to the parts to be assembled.

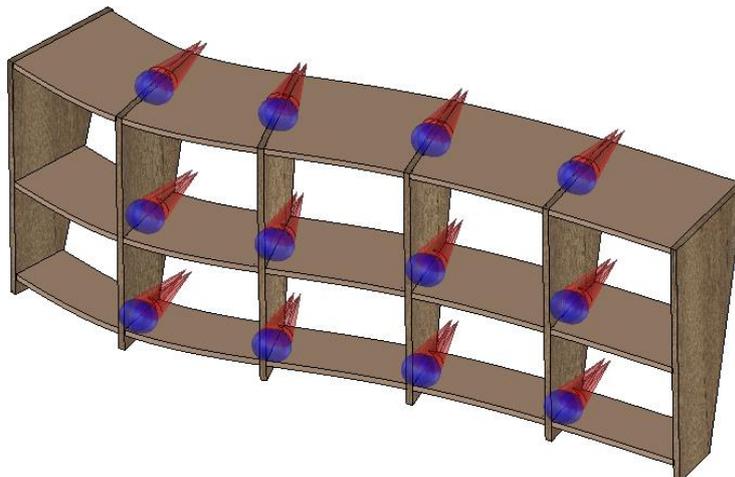
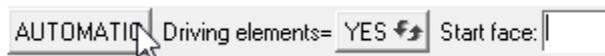


Note: To be able to interchange the kits, the kit must be an advanced kit and be set to **Non-parallel faces management = YES**.

Multi-directional positioning

You can now insert assembly kits in several automatically identified directions.

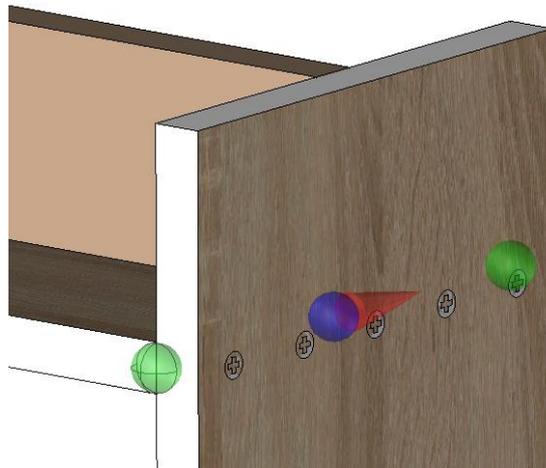
- Select the **Wood > Other processes > Automatic assembly** command.
- Select the **MAIN ASSEMBLY** option, then select the rule to be applied.
- Select the **AUTOMATIC** mode so that the start/end and centering faces are automatically identified.



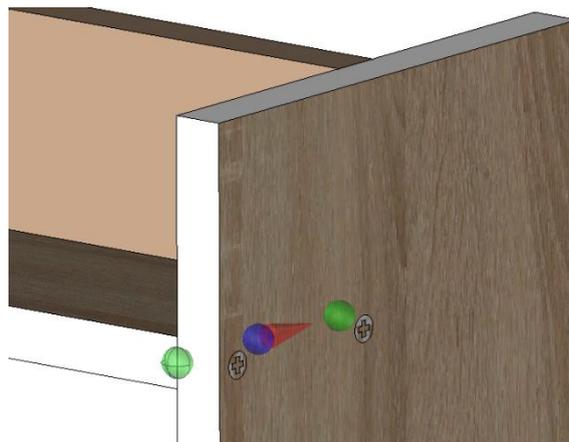
Positioning: Start and end faces

New handles represented by green spheres allow you to modify the start and end faces of the kit directly from the graphics area.

- Select an assembly command from the **Wood > Other processes** menu.
- Select the parts to be assembled.
- Adjust the **Driving elements** option to **YES** to display the spheres that will allow you to modify the start or end face.



- Click on the start sphere to reposition it on the central crossbar.



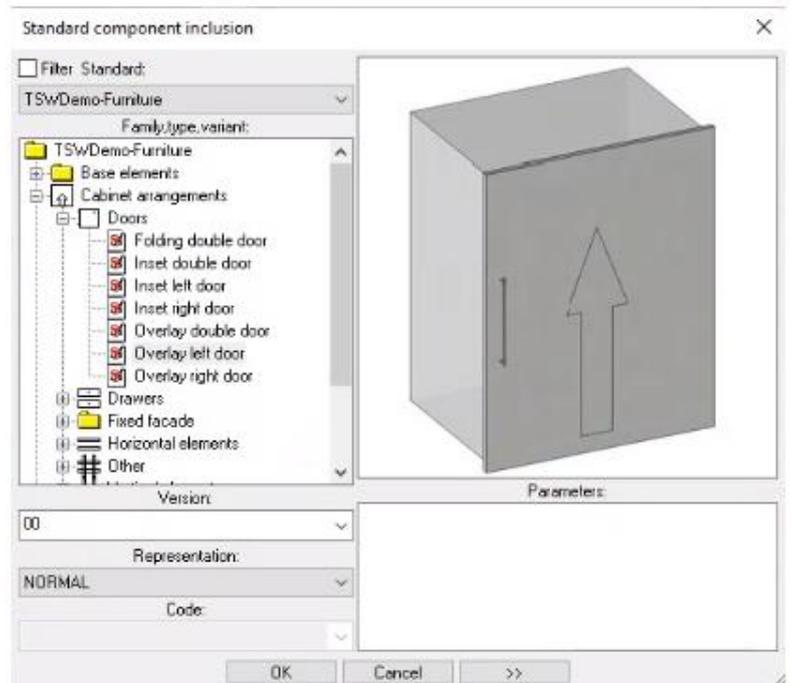
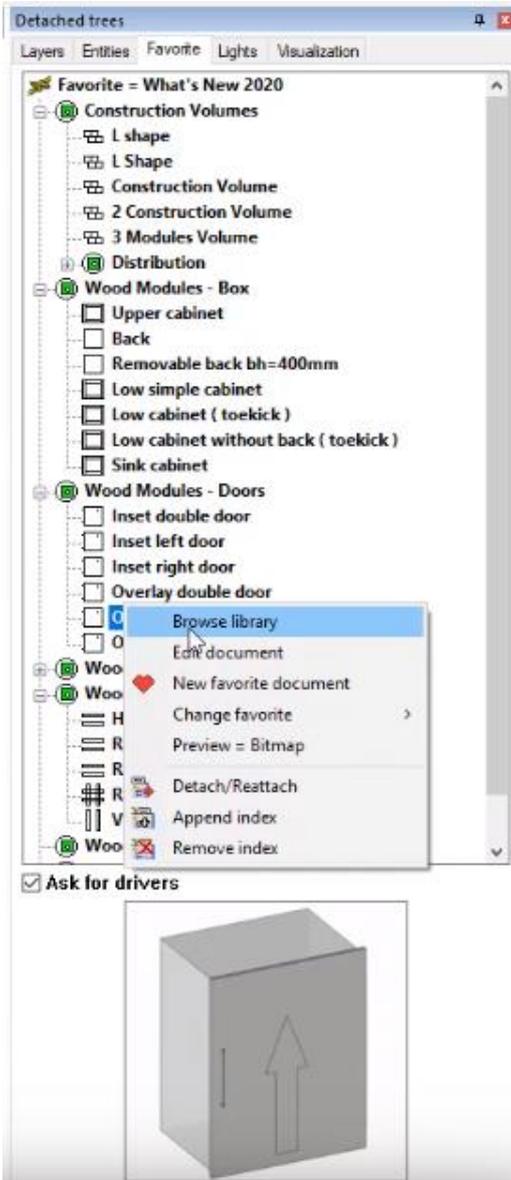
- The start face is repositioned and the propagation is recalculated.



Inclusion

Favorites

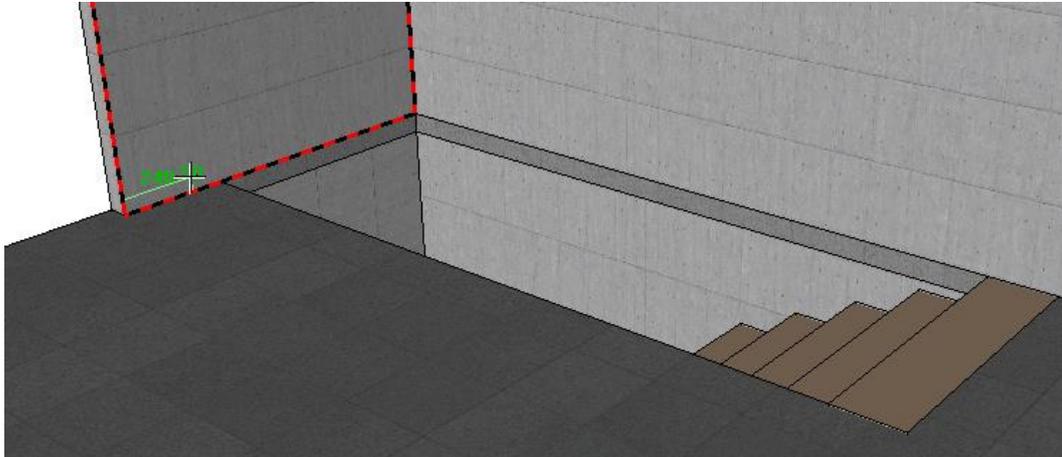
You can now access the standard library of a component directly from the Favorites tree. To do this, you only have to **right-click** on the component and select the **Browse library** command.



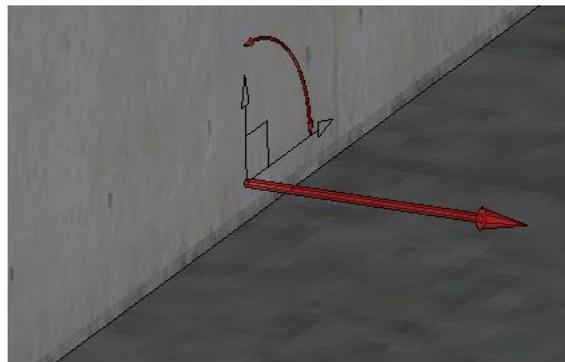
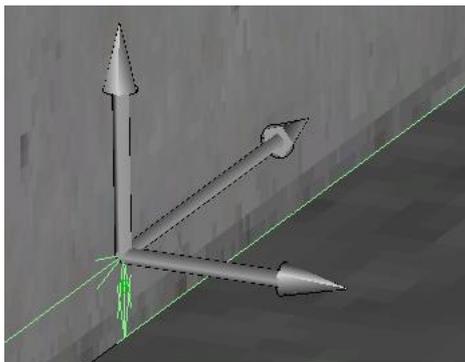
Distributed component

When including a component by distribution, you can now use a **wizard** to position the reference coordinate system on the fly.

- Select the command to include a component and select the **DISTRIBUTION** mode.
- Adjust the **Wizard** option to **YES**.
- Select the reference element to position the coordinate system.



- Use the arrows to adjust the position and direction of the reference coordinate system.



- Select the second face for the distribution.
- Adjust the distribution options.



TopSolid'Store

You can now access the **TopSolid'Store** website from the **Help > TopSolid'Store** command and directly from the start page.



TopSolid'Store regroups the component libraries of our hardware and material partners.

Version 6

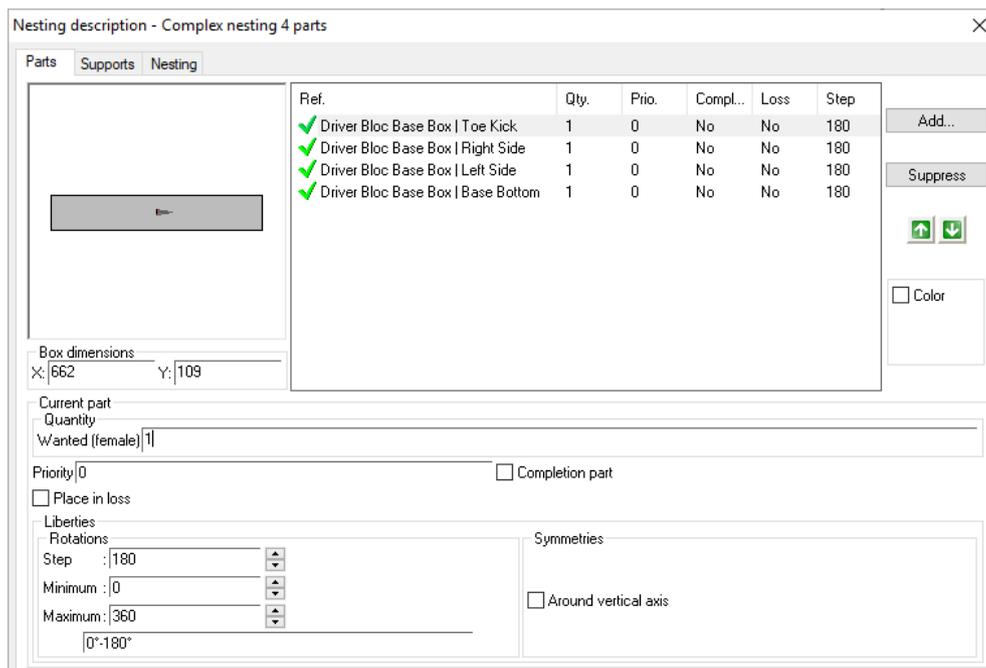
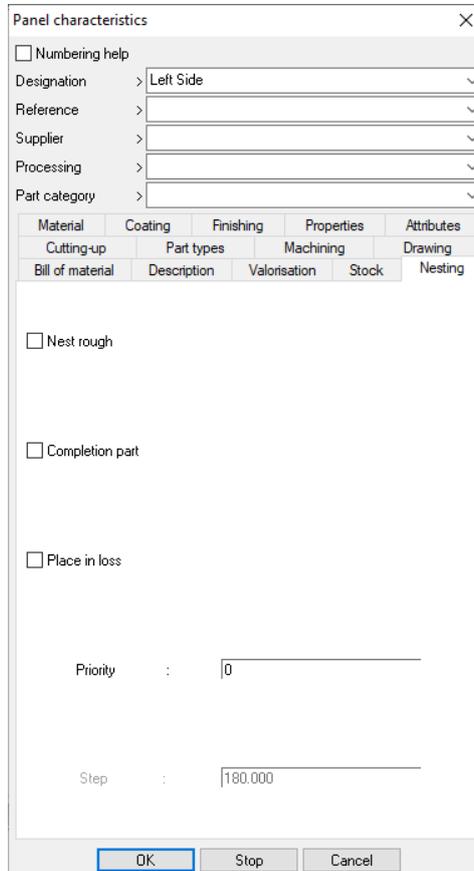
SORT BY: ↑ VIEW AS: 5 Item(s) SHOW:

				
TopSolid Material Database EGGER 2020-22	TopSolid'Wood BLUM Library 2020	TopSolid'Wood GRASS Library 2020	TopSolid'Wood Hettich Library 2020	TopSolid'Wood OVVO Library 2020
VIEW DETAILS	VIEW DETAILS	VIEW DETAILS	VIEW DETAILS	VIEW DETAILS

Nesting

Options

In the **Nesting** tab of the part characteristic definition dialog box, the new **Completion part**, **Place in loss**, **Priority** and **Step** options allow you to set as much information as possible before starting the nesting.

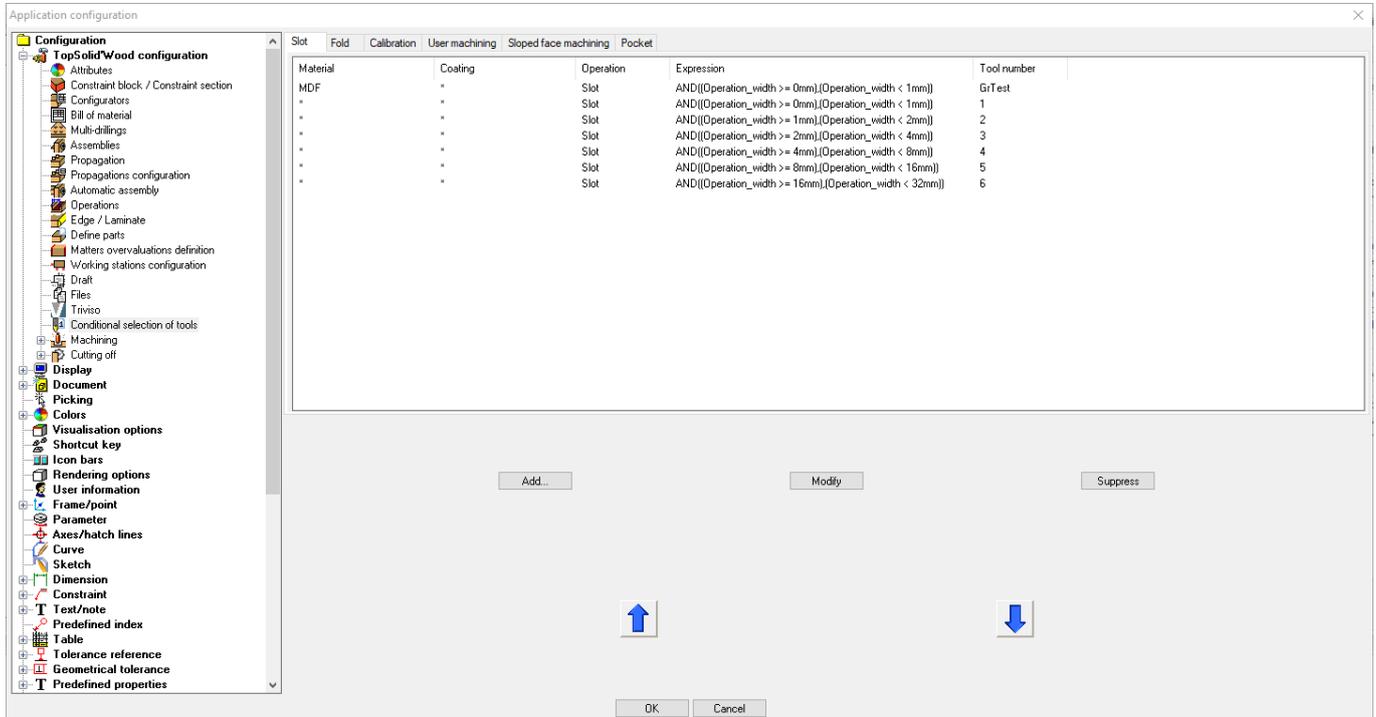


Machining Export

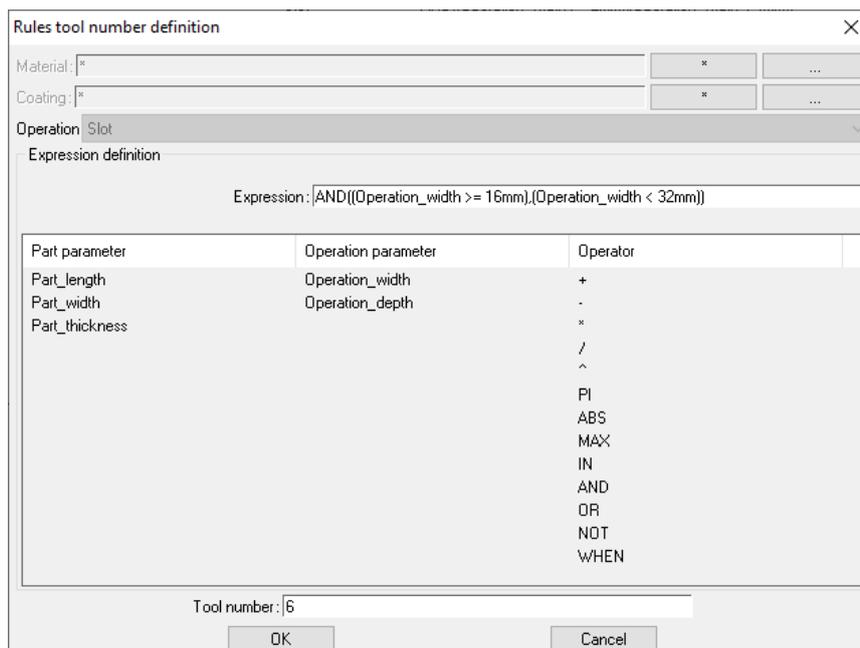
Conditional tool selection

For the export interfaces, you can now set tool numbers that meet conditions.

- Select the **Tools > Options > TopSolid'Wood configuration > Conditional selection of tools** command.



- Click on one of the **Slot, Fold, Calibration, User machining, Sloped face machining** or **Pocket** tabs.
- Click on the **Add** button to enter a condition.
- In the dialog box that appears, enter the desired **expression**. The parameters and operators to be used are listed in the lower section of the dialog box.



In the above example, if a slot is between 16mm and 32mm wide, then it will be exported with tool number 6.

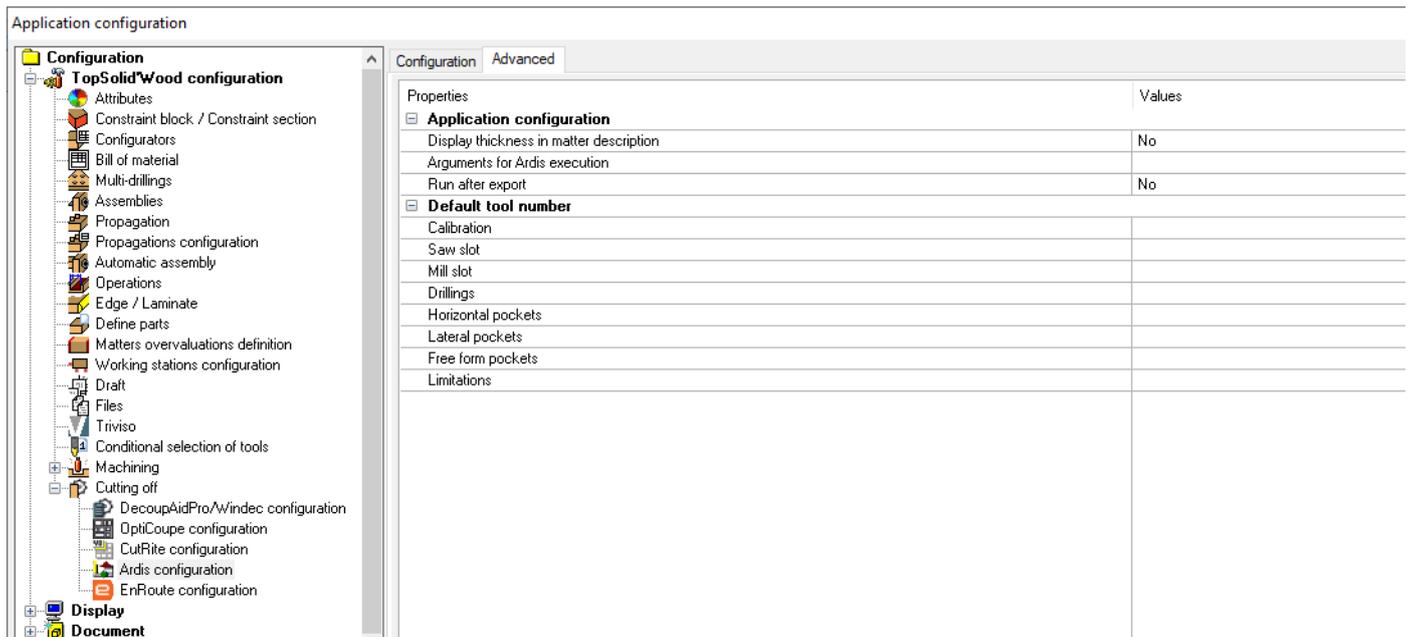
- Move the conditions using the blue arrows to order them.

Notes:

- If a criterion meets several conditions, the tool in the first line of the list will be selected. It is therefore important to check the order of the conditions.
- For Xilog, only the numbers are recognized as tool numbers. If the tool number is not a number, then tool number 101 will be exported.
- If the expression contains an error, a yellow panel is displayed next to the line .

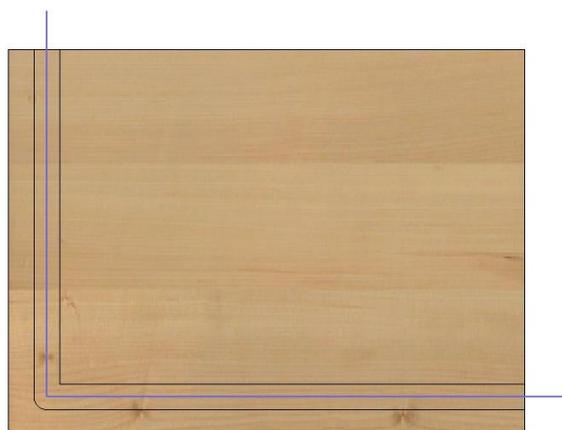
Ardis

All the settings for the **Ardis** exports can now be adjusted in the **Tools > Options > TopSolid'Wood configuration > Cutting off > Ardis configuration** command.

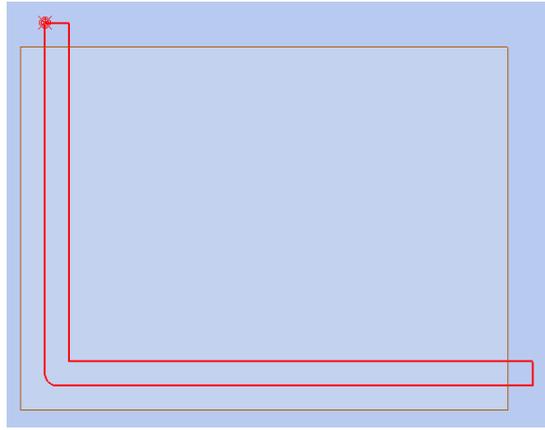


Operation in offset mode

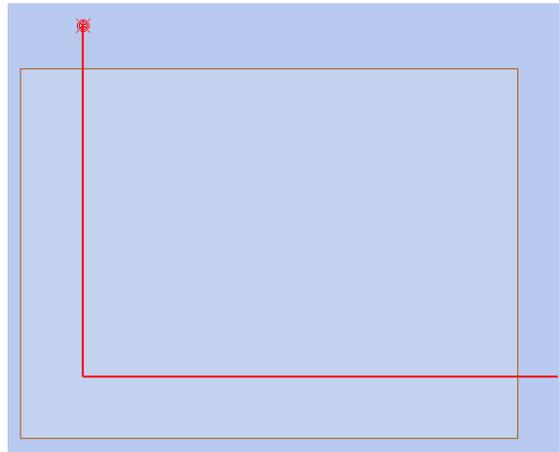
When exporting a machining in offset mode, the cases that were not previously managed are now supported, for example the through slots.



Here, the case of a through slot on two faces.



In WoodWOP, the contour of the slot will be exported and not its path.

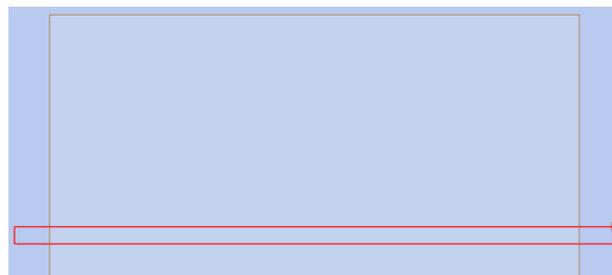


As a reminder, here is the result in version 6.21.

Note: The **ZWOO_DSHOPSHAPING_SLOT_EXTEND_LENGTH** configuration word can be added to the *topwoo.cfg* file so that you can adjust the offset value of the part.



Another example of a through slot.



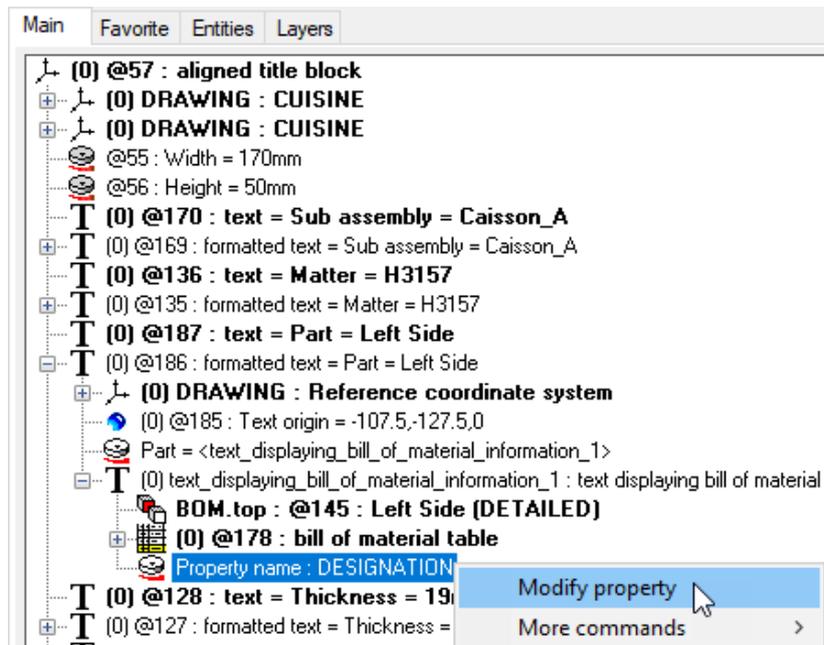
Result in WoodWOP.

Drafting

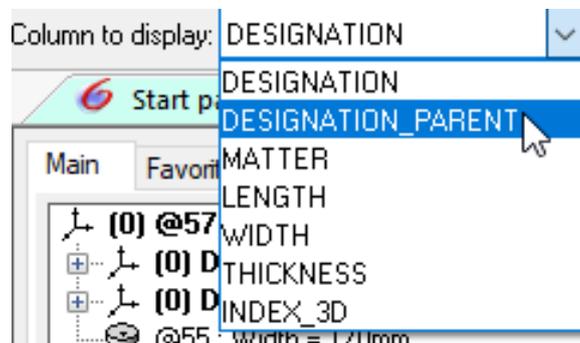
Title block

You can now reconnect a title block property to another BOM property from the tree.

- In the tree, **right-click** on the property name, select the **Edit** command, and then click on the title block.
- Expand the formatted text to get to the property name.
- **Right-click** on the property name and select the **Modify property** command.



- Select the replacement property from the drop-down list.



Length = 900mm	Width = 500mm	Thickness = 19mm
Part = Left Side		
Matter = H3157	Emplacement = Cabinet_A	

Here, "Left Side" has been replaced by "Cabinet_A".

When creating a title block in a multi-draft template, you sometimes have to add properties from the BOM. From now on, when the bill of materials is changed, the properties available in the two bills of materials are automatically reconnected in the title block.

BOM :

Left Side	Cabinet_A	H3157	0.9	0.5	0.02
DESIGNATION	DESIGNATION_PARENT	MATTER	LENGTH	WIDTH	THICKNESS

Title block with BOM properties

Length = 900mm	Width = 500mm	Thickness = 19mm
Part = Left Side		
Matter = H3157	Sub assembly = Cabinet_A	

In this example, the title block is set with a six-column bill of materials.

BOM :

Left Side	Cabinet_A	H3157	0.9	0.5	0.02	A_01
DESIGNATION	DESIGNATION_SUB_ASS	MATTER	LENGTH	WIDTH	THICKNESS	ID

Title block with BOM properties

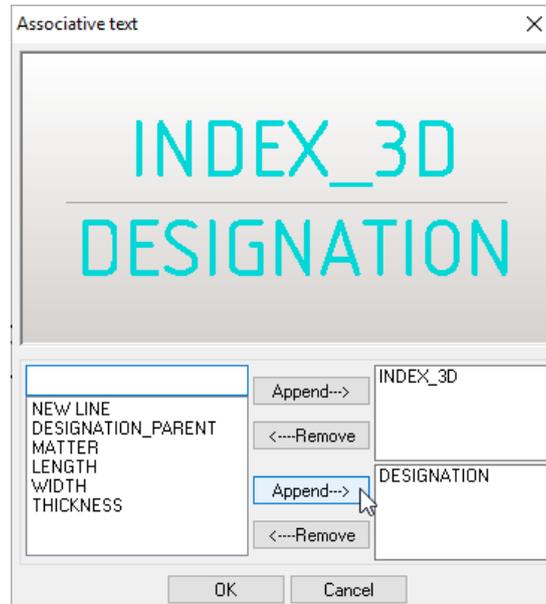
Length = 900mm	Width = 500mm	Thickness = 19mm
Part = Left Side		
Matter = H3157	Sub Ass = Cabinet_A	ID = A_01

The bill of materials is changed to add the properties in blue. A column has been added and the name of a column has been modified. The title block does not go into error and the properties are correctly reconnected.

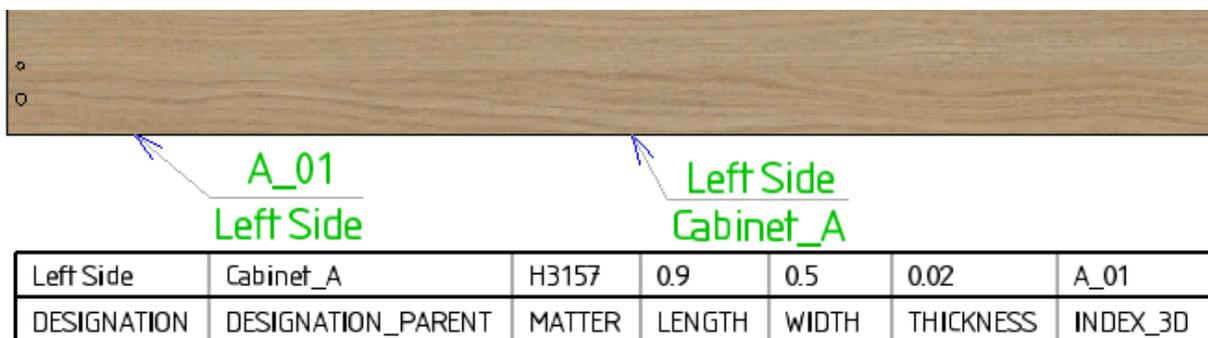
Text with information

A new **Text with information** command is now available. This command has the same behavior as a note with information but, unlike the note, it allows you to put the same text on several different parts or to put different texts with information on the same part.

- Select the **Text with information** command.
- If there is no bill of materials in the drafting document, create a bill of materials.
- Select the **NEW DEFINITION** option to create a definition on the fly.
- In the dialog box that appears, select the BOM properties you want to display in the text with information.

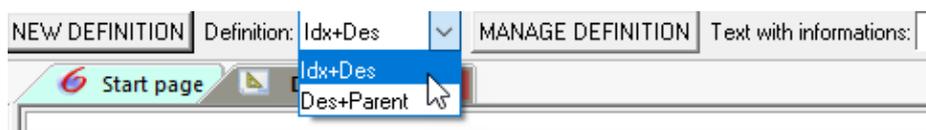


- Select the type of positioning: **Free**, **Magnetic** or **Automatic**.



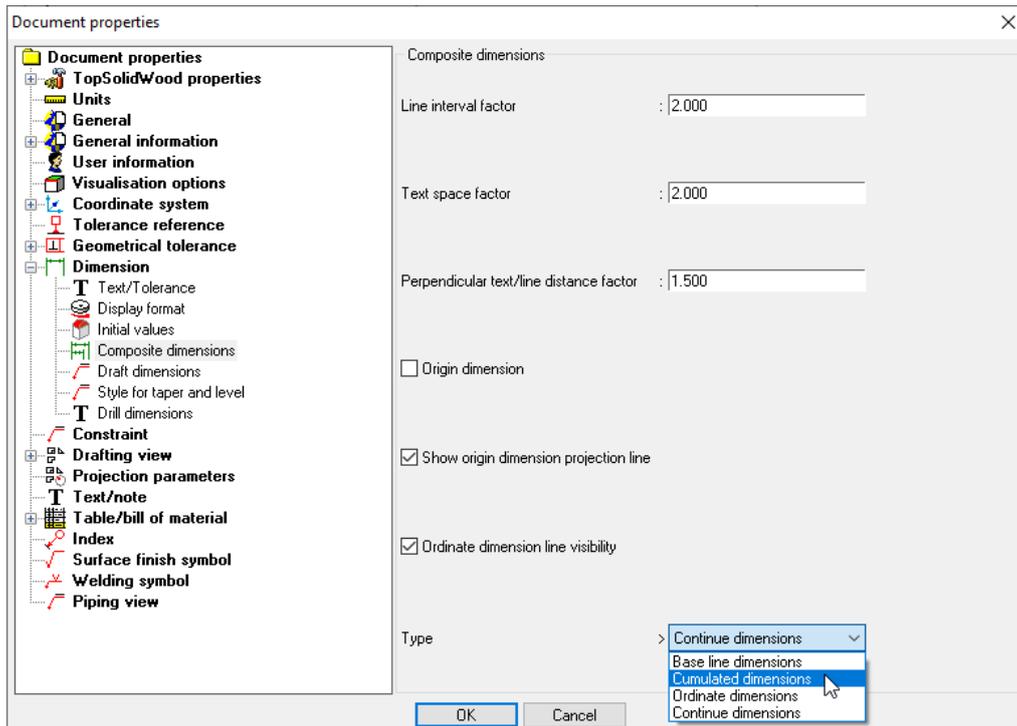
Note: You can create predefined texts that will be available in the drop-down list. To do this, you simply have to proceed as follows:

- Create a **text with information** on the fly.
- Click on the **MANAGE DEFINITION** option.
- Give a name to the definition.
- Select an existing text with information.



Dimensions

You can now adjust the type of composite dimensions directly in the **document properties** as well as in the **Tools > Options** command.



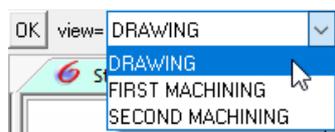
Notes:

- The adjustments made in the **Tools > Options** command apply for all future documents without template.
- The adjustments made in the **document properties** are taken into account for the current document.
- If you manually modify the type of dimensions directly from the **Composite dimensions** command, all the dimensions created after the modification will be of this type until the command is confirmed.

View

Nesting

When creating a nesting draft template, you can now choose between three definitions via a drop-down list: **Drawing**, **First machining** and **Second machining**. As a reminder, these definitions are set in the part characteristics.



Updating views

In case the view update is not enabled in the **document properties**, a new icon allows you to update the drafting document with a single click. This icon is available in the **TopSolid'Wood Light** style.

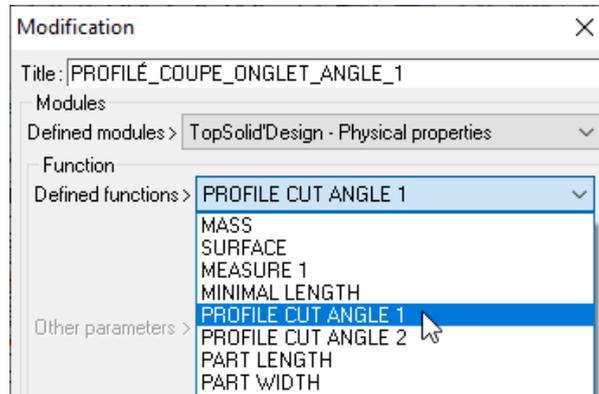


Note: If you want to add the icon to your style, it is available in the **View > Automatic update of views** command (system bar).

Bill of Materials

Cutting angle

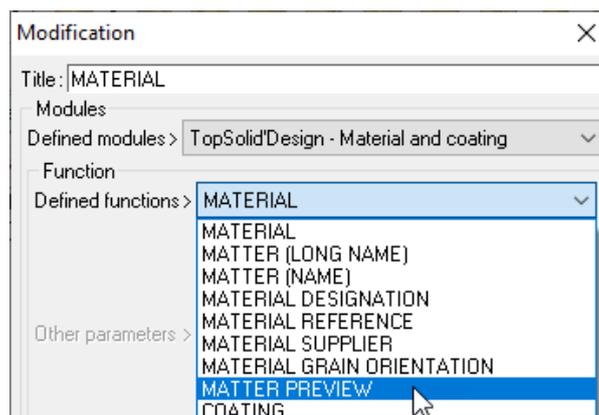
Two new BOM properties named **Profile cut angle 1** and **Profile cut angle 2** allow you to display the cutting angles of the parts. When creating the BOM template, these properties are available in the **TopSolid'Design - Physical properties** module.



poutrelle IPE 80	45	45
cornière à ailes égales 20x3	0	0
cornière à ailes égales 20x3	21	21
cornière à ailes égales 20x3	26	26
cornière à ailes égales 20x3	29	29
PART NAME	PROFIL_CUT_ANGLE_1	PROFIL_CUT_ANGLE_2

Preview

In the bill of materials, you can now preview the texture of the material and the coating thanks to the new **Matter preview** and **Coating preview** properties. When creating the BOM template, these properties are available in the **TopSolid'Design - Material and coating** module.



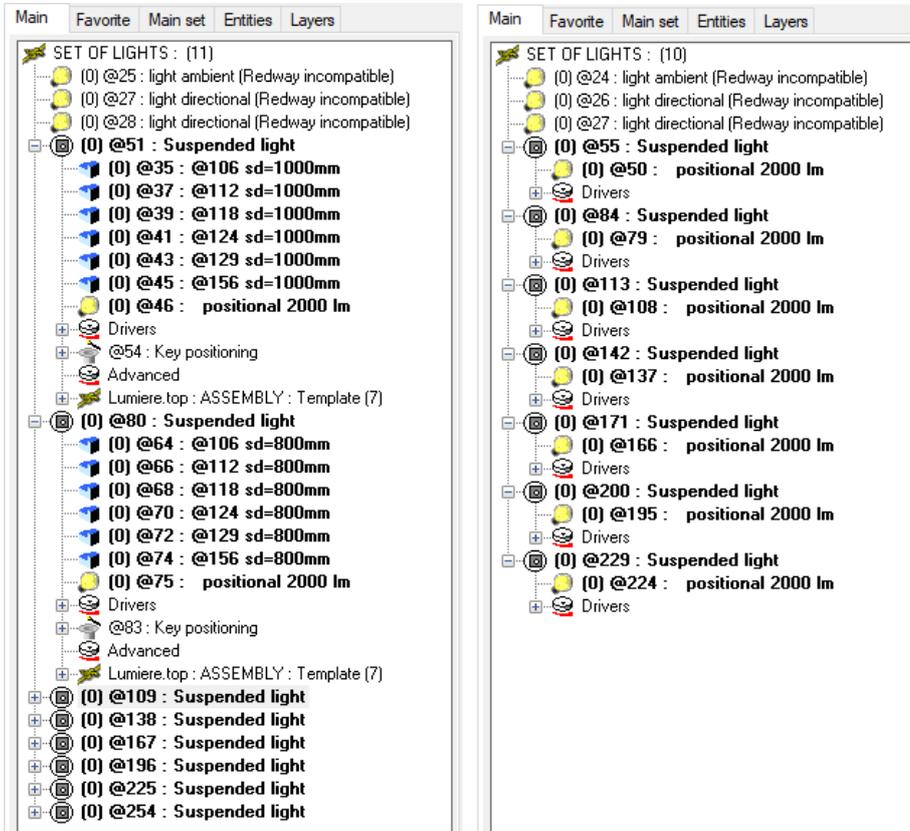
PartD			MDF	Beech core
PartC			MDF	Chestnut sweet
PartB			MDF	Wild service tree
PartA			MDF	Ash european
DESIGNATION	MATTER_PREVIEW	COATING_PREVIEW	MATTER	COATING

Image

Lights

Set of lights

The set of lights has been reduced to display only the component lights.

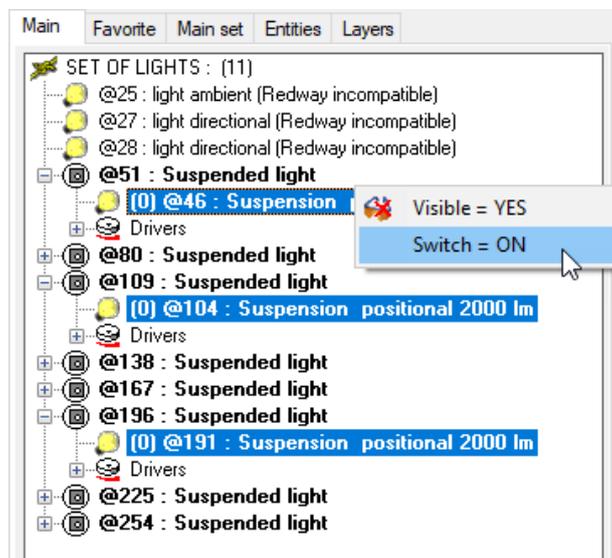


On the left, the set of lights in version 6.21. On the right, the set of lights in version 6.22.

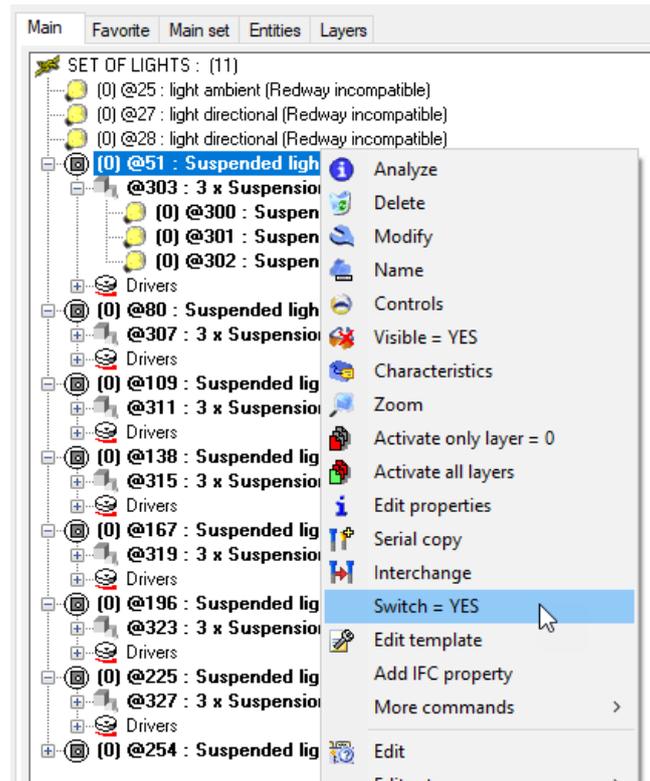
Turning multiple lights on/off

Multiple selection is now possible to turn lights on or off:

- either by selecting multiple lights, then **right-clicking** and selecting **Switch = ON/OFF**;

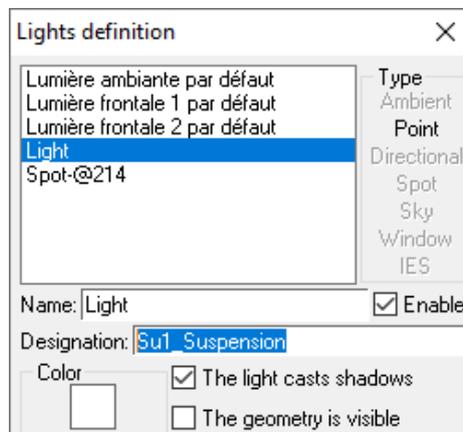


- or by selecting the line of a set containing several lights, then **right-clicking** and selecting **Switch = YES/NO**.



Light designation

You can now enter a designation when defining a light.



This designation is available in the **Lights definition** dialog box of the assembly. This can be useful to easily distinguish between several different lights.

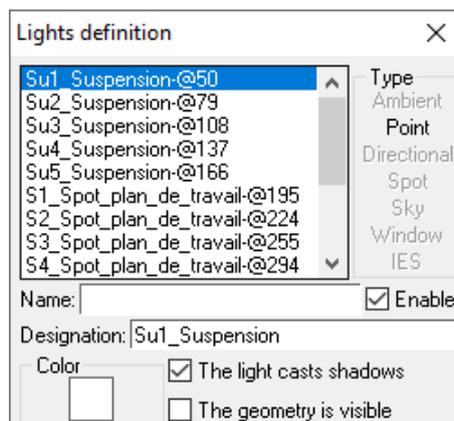
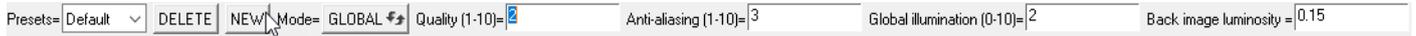
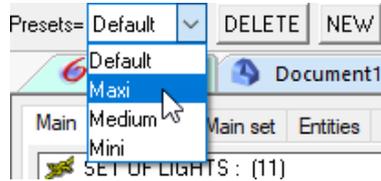


Image calculation preset

You can now define presets for image exports. You only have to enter all the desired values, then click on the **NEW** button to name each preset.



The presets are then available in the **Presets** drop-down list. These are arranged in order of creation and the last created preset appears at the end of the list.



To delete a preset, simply select it from the list and click on the **DELETE** button.

Visit mode

In the **Visit mode** command, the new **Height** parameter allows you to modify the height of the eye.



Interfaces

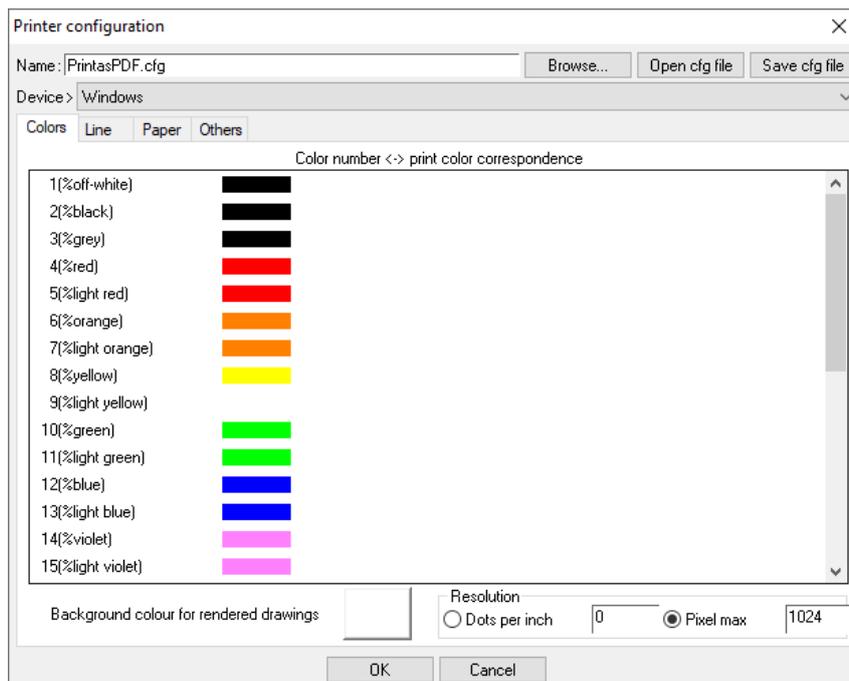
PDF 3D

You can adjust the print settings of a PDF 3D export (colors, line thicknesses, etc.). To do this, you only have to select the **Tools > Options > Interfaces > Acrobat Pdf 3D > Export** command, click on the **Browse** button, then indicate the path to an existing print configuration file.

The **Open cfg file** button opens the configuration in a notepad.



Note: The configuration file must have been previously set up and saved from the **printer configurator (Page setup > Configure)**.



Once the file has been saved using the **Save as > PDF 3D** command, you can now perform a search in the PDF file. To do this, simply use the **Ctrl + F** keyboard shortcut or the **Edit > Search** command. When the search is confirmed, the results are highlighted.

06	Vertical Partition	Structure	H1180 - ST37 - Oak Halifax	8
05	Top	Structure	H1180 - ST37 - Oak Halifax	1
04	Shelves	Structure	W1001 - Premium white	27
03	Right End Panel	Structure	H1180 - ST37 - Oak Halifax	1
02	Ribs	Structure	H1180 - ST37 - Oak Halifax	8
01	Left End Panel	Structure	H1180 - ST37 - Oak Halifax	1
INDEX_3D	DESIGNATION	DESIGNATION_PARENT	MATTER	TOTAL_COUNT

Searching for the word "shelves" in Acrobat provides three results.

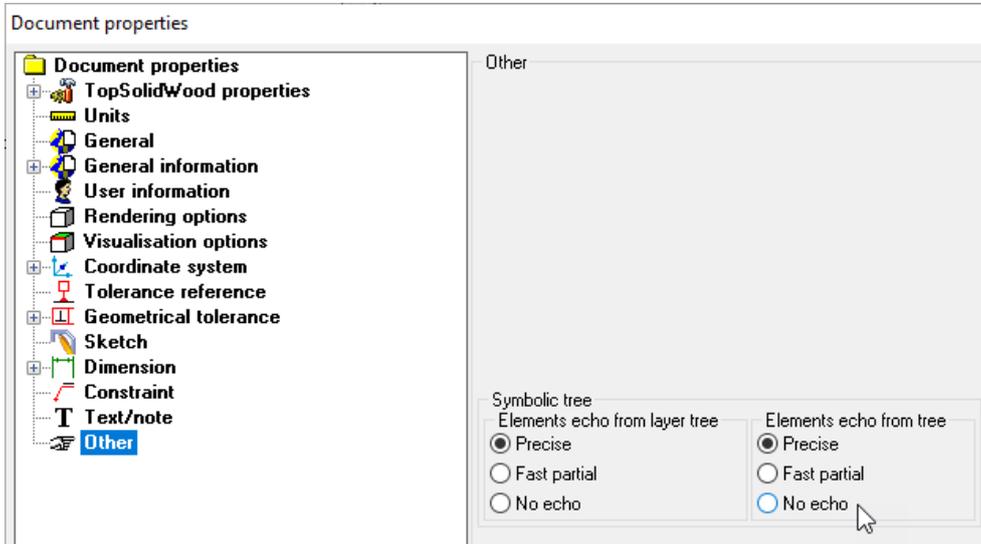
06	Vertical Partition	Structure	H1180 - ST37 - Oak Halifax	8
05	Top	Structure	H1180 - ST37 - Oak Halifax	1
04	Shelves	Structure	W1001 - Premium white	27
03	Right End Panel	Structure	H1180 - ST37 - Oak Halifax	1
02	Ribs	Structure	H1180 - ST37 - Oak Halifax	8
01	Left End Panel	Structure	H1180 - ST37 - Oak Halifax	1
INDEX_3D	DESIGNATION	DESIGNATION_PARENT	MATTER	TOTAL_COUNT

When you click on the matches, the words are highlighted in Acrobat.

Miscellaneous

Selection echo

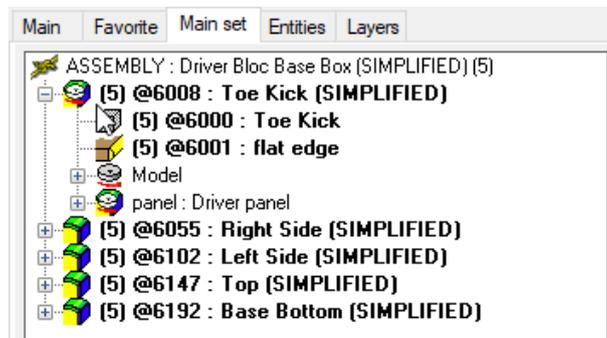
The echo, which was previously adjusted from the **Tools > Options** command, can now be adjusted in the **Document properties > Other** command, allowing you to change the settings only in the current document. The **No echo** option allows you to disable the highlight from elements, which can be very useful on very heavy assemblies.



Panels

Several improvements have been made to the panels:

- For the panels defined as drivers, icons have been added both in the tree and in the wood selection dialog boxes to quickly differentiate them.



- When the panel is in **Model** attribute, you can now select it graphically to change its layer without having to display the panel template. All entities in the panel will then change layer.
- When the panels are displayed in unit mode in the wood selection dialog boxes, you can select them by clicking directly in the graphics area.
- When there are only panels in a sub-set, you can now change the visibility of the set by **right-clicking** and selecting **Visible = YES/NO** from the Entities tree.



Favorites

The name displayed for the in-place sub-sets in the favorites now shows the designation of the in-place sub-set, whereas previously it displayed its name.

Laminate configuration

If filters are used for material selection, the **Filtered/Not filtered** option is now saved in the laminate configuration dialog box.

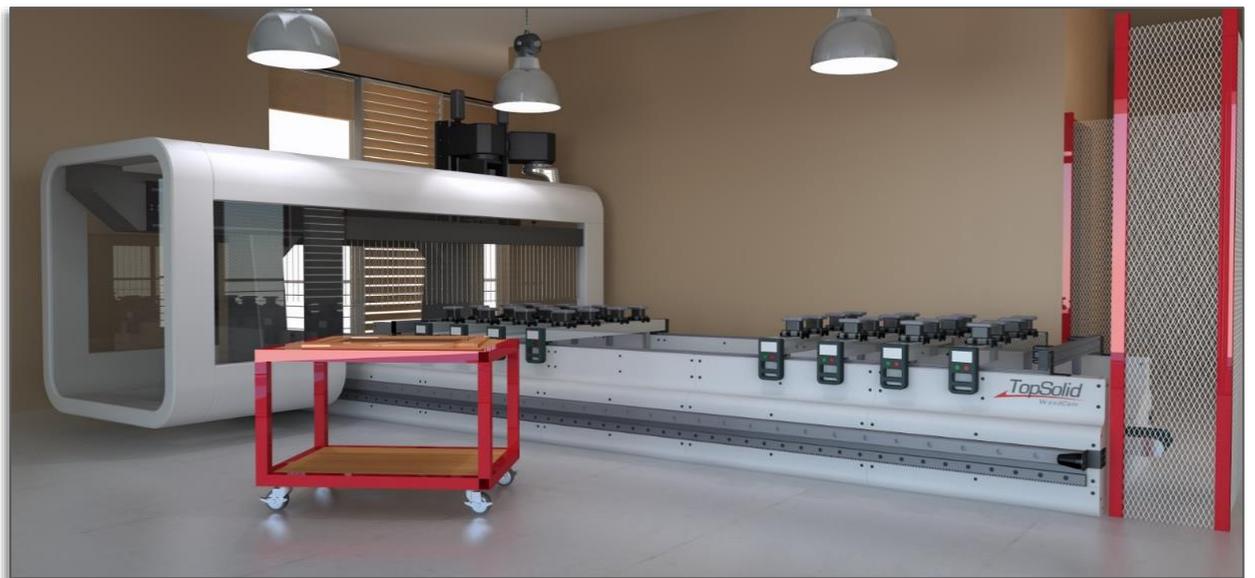
Visibility of a sub-set

To unify behaviors, when all the elements of a sub-set are hidden, it no longer appears in bold in the Entities tree.

Updating indexes

In the **Reroute** command, the new **Update indexes** option allows you to check in all sub-sets whether a component has a more recent index.

What's New in TopSolid'WoodCam v6.22

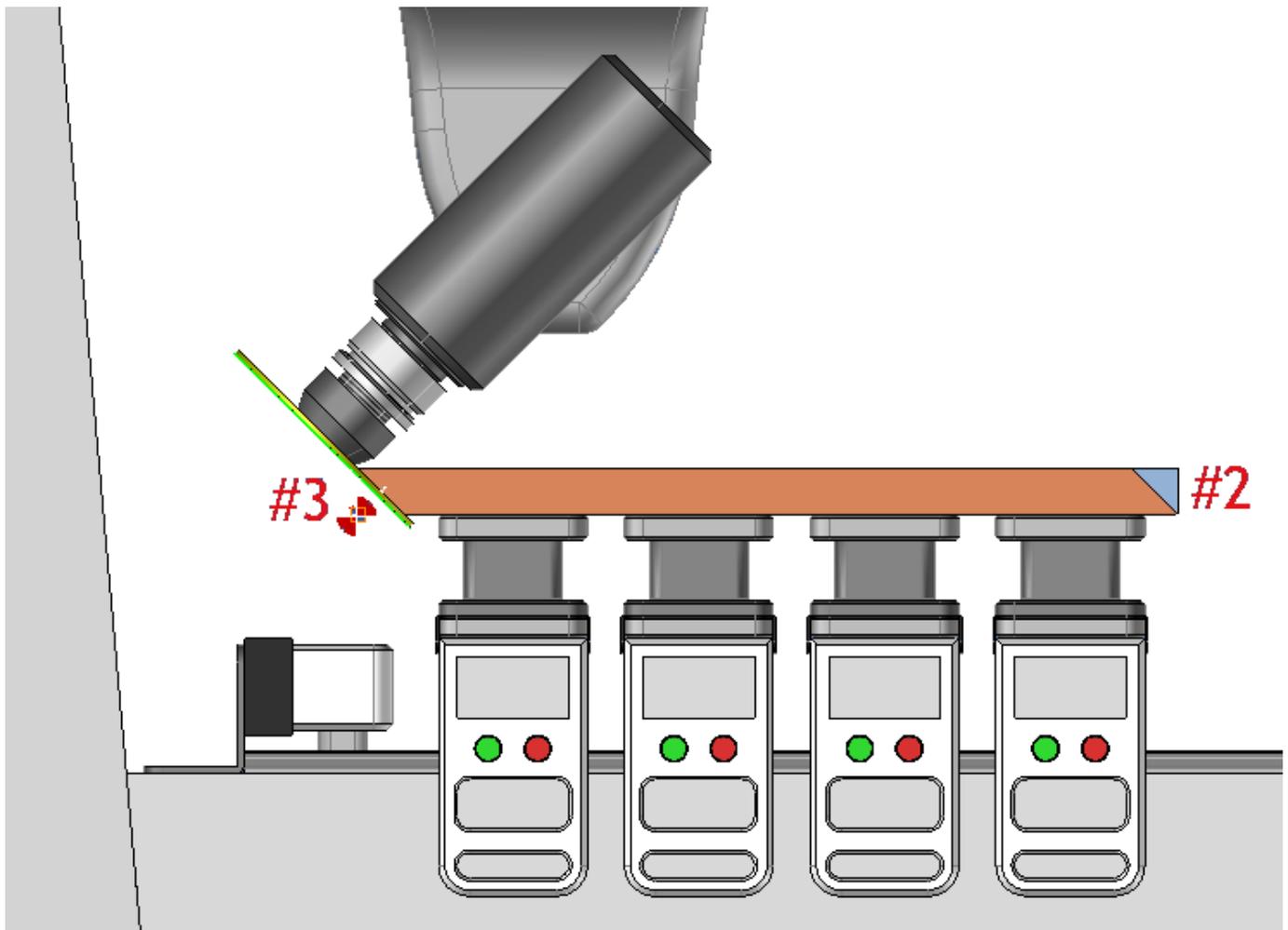
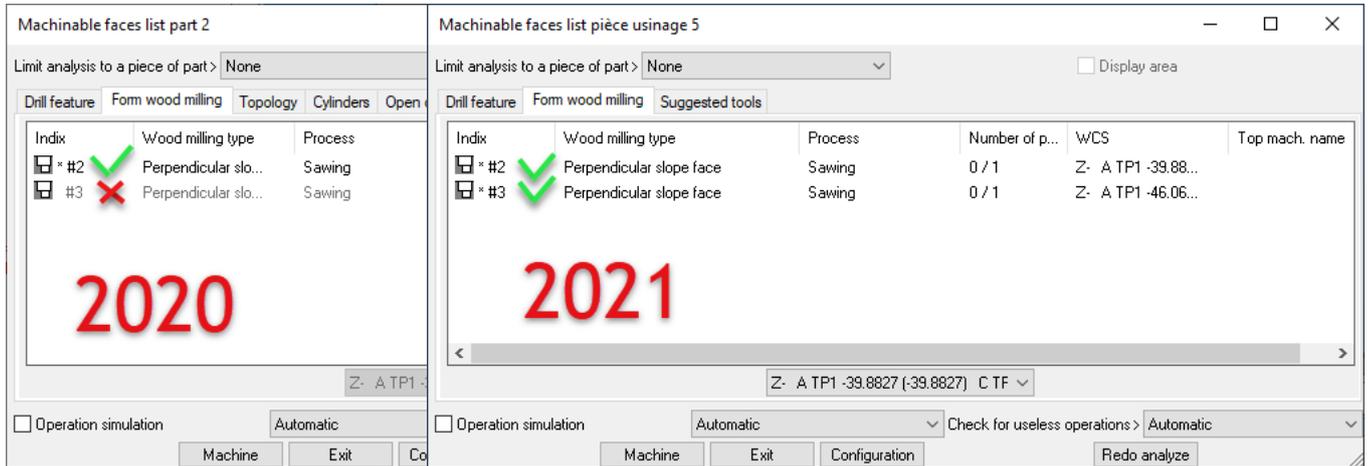


This section describes the enhancements made to version 6.22 of TopSolid'WoodCam.

Machining

Sawing on undercut faces in multi-machining

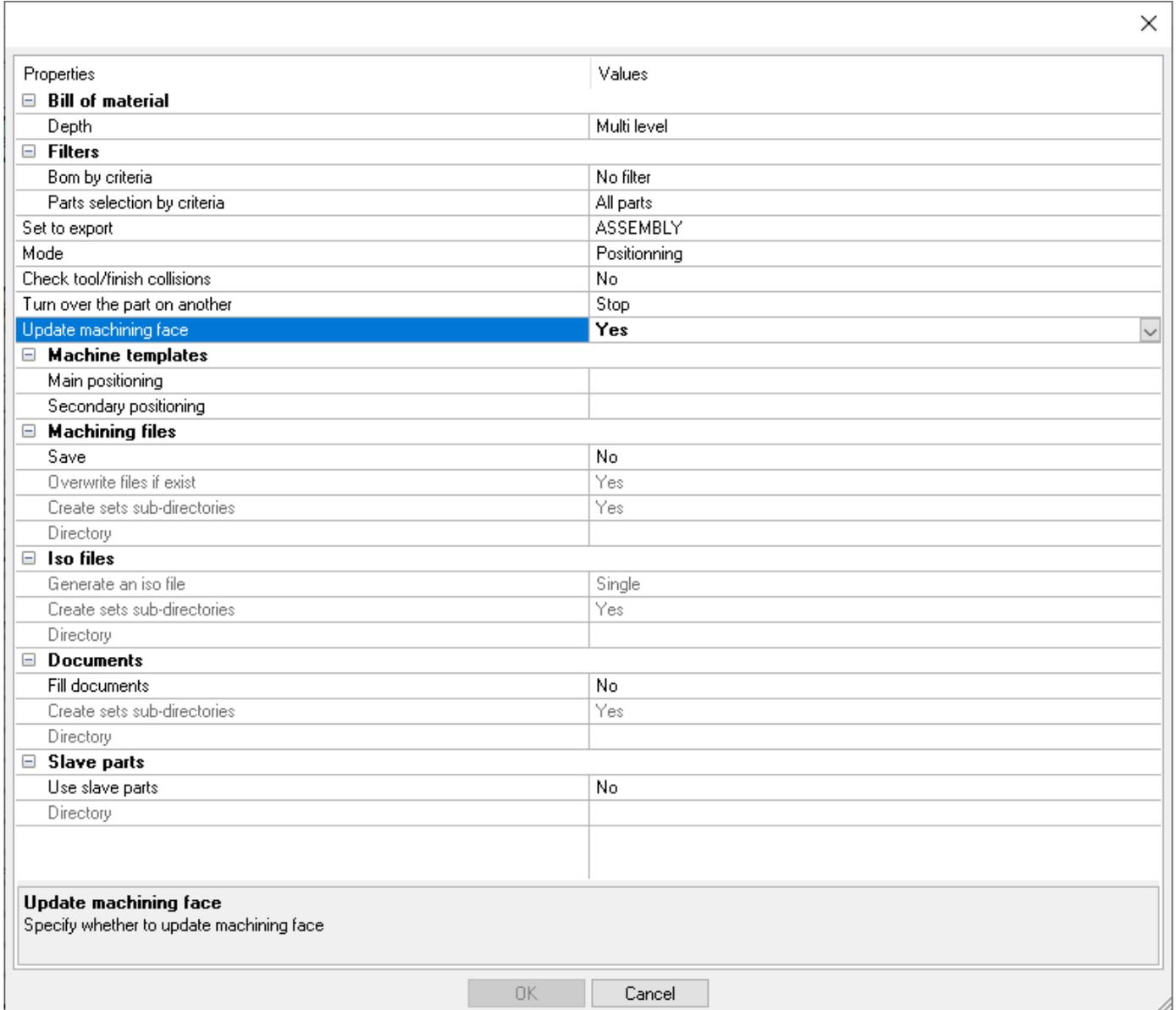
Sawing on undercut faces is now automatically performed in multi-machining mode. To carry out this machining operation, you only have to perform a user machining operation on a sloped face in perpendicular mode.



Script

Updating the machining face

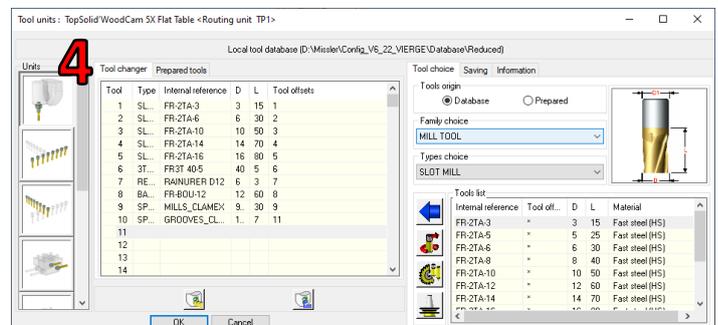
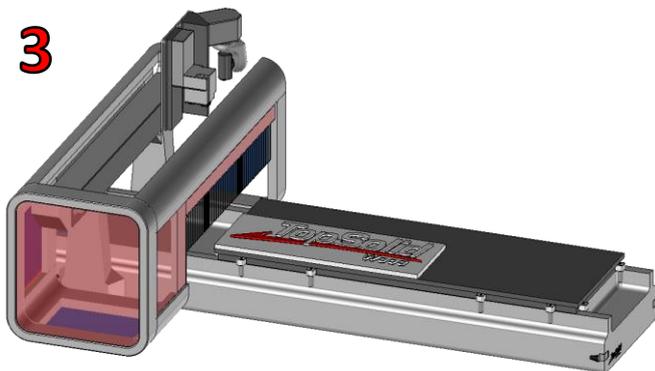
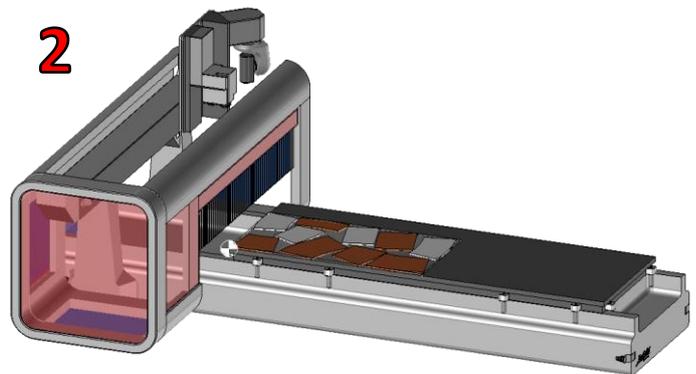
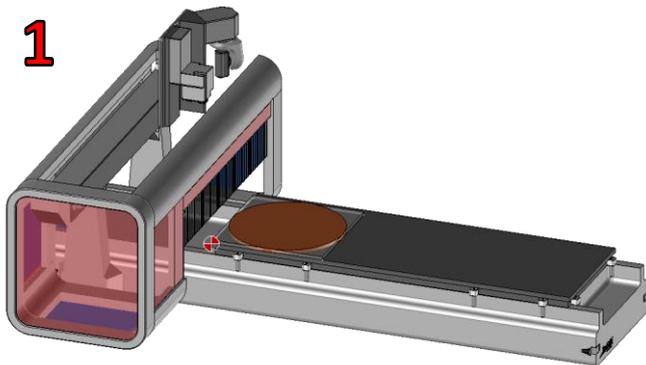
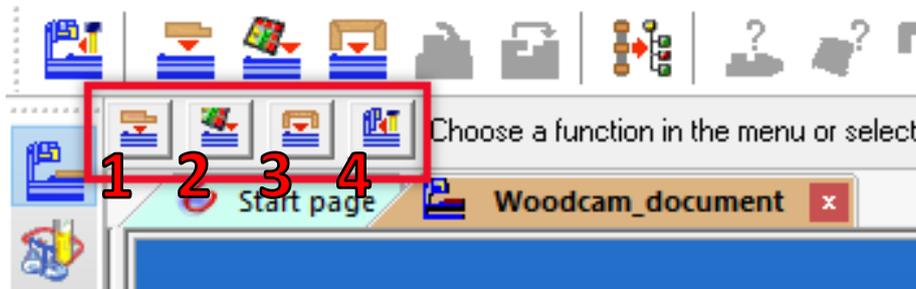
From a multi-machining node, the new **Update machining face** option allows you to update all the machining faces of the parts in the project that will be machined.



Ergonomics

Neutral mode

A new neutral mode is available directly when opening a **WoodCam** file. It allows you to quickly position a part, a nesting or a set, or to modify the Tool Manager.

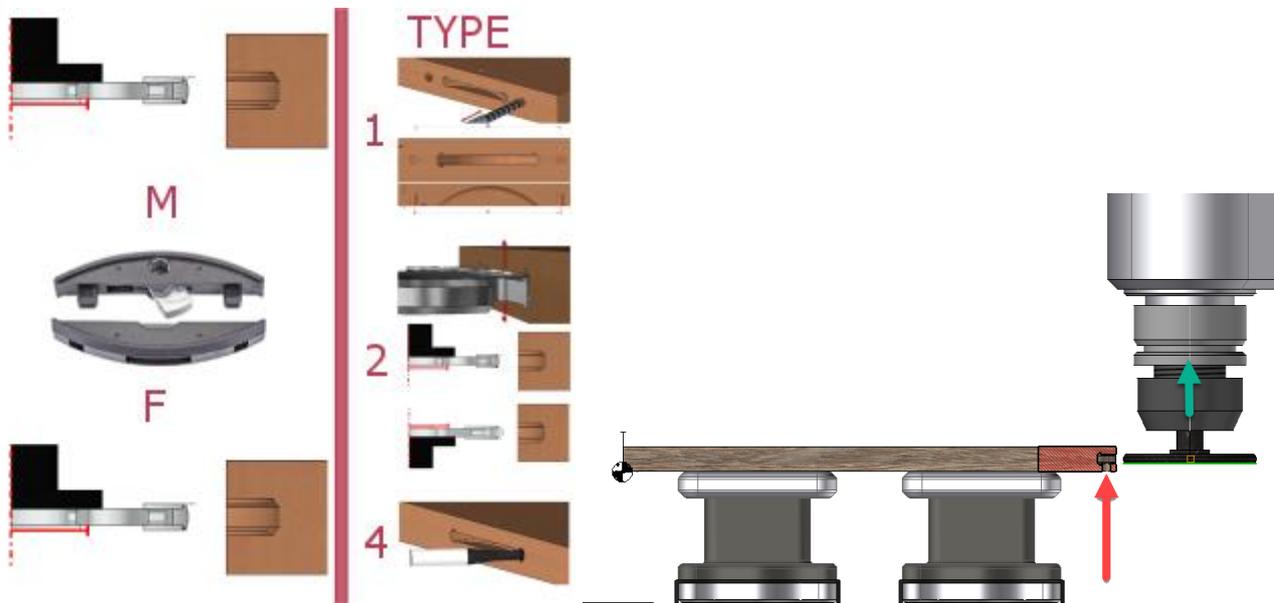


Clamex

Component improvement

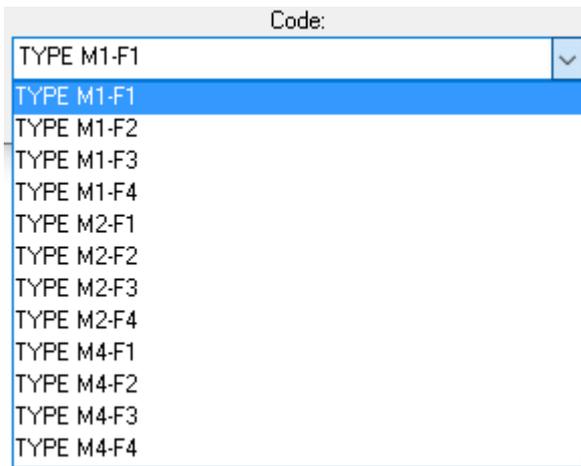
A new revision of the Clamex components brings a simplification of use and a gain in automation.

Clamex P-10 pair / Clamex P-14 pair / Clamex P-14-10 / Clamex P-15 pair / Clamex P-Medius 14-10 / Divario P-18 / Tenso P-10 pair / Tenso P-14 pair not centered / Tenso P-14 pair.

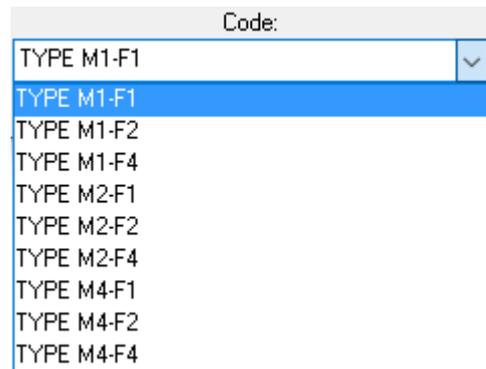


An optimization of the machining face allows a reduction in the number of catalog codes and thus makes the component easier to use. The positioning of the blade is now independent of the drilling position.

TopSolid 2020



TopSolid 2021



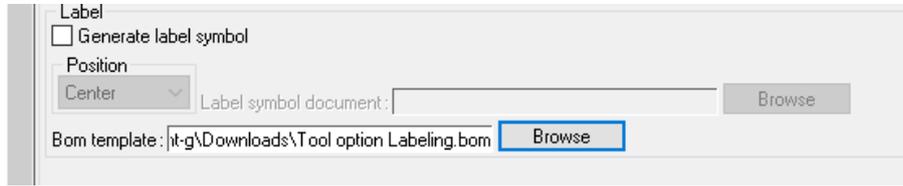
The resulting operations are thus enhanced for a better integration with the various CNC machines.

Note: A repositioning of the existing programs will be necessary to benefit from these optimizations.

Labels

Customizing the label table

When a bill of materials is used to create the nesting document, information such as the part material, the part dimensions, or other information can be added in some columns of the label operation.



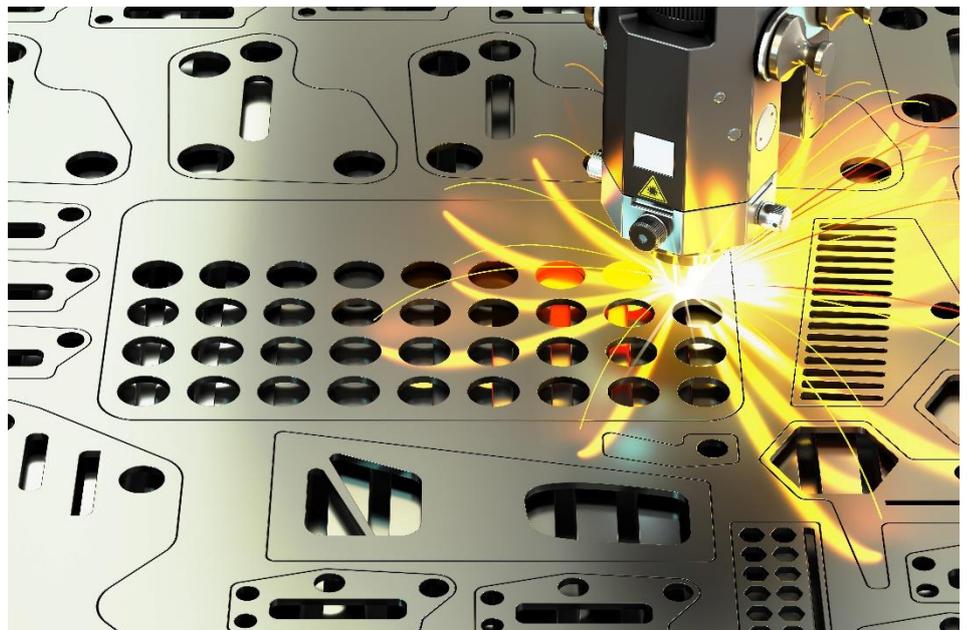
Labels positions

Positions visualisation origin: Machine Part

Proxi...	Text	Part	X	Y	Z	Angle	Label_Name	Label_X	Label_Adress	Name	Elem...	Matter	Program1...	Program2...	Width	Height	Thick...	Owner_N_Desig...	Orig...	UL_Job...	
1	6	Partition_top_2	part 21	1700.755	207.275	19	0	TopSolid-33792	0	.\MDF_201_w...	Partition_top_2	42	MDF	Left worki...	Left worki...	0.6	0.41	0.02	Division by curves	33792	TopSolid
2	5	Partition_med_high_2	part 22	1825.187	712.685	19	90	TopSolid-40759	0	.\MDF_201_w...	Partition_med_hi...	44	MDF	Left worki...	Left worki...	0.57	0.35	0.02	Division by curves	40759	TopSolid
3	10	Partition_top_6	part 23	221.423	197.174	19	180	TopSolid-33934	0	.\MDF_201_w...	Partition_top_6	46	MDF	Left worki...	Left worki...	0.44	0.39	0.02	Division by curves	33934	TopSolid
4	1	Partition_med_high_6	part 24	166.592	778.396	19	90	TopSolid-40763	0	.\MDF_201_w...	Partition_med_hi...	48	MDF	Left worki...	Left worki...	0.44	0.33	0.02	Division by curves	40763	TopSolid
5	2	Partition_med_low_2	part 25	593.791	890.322	19	180	TopSolid-33780	0	.\MDF_201_w...	Partition_med_lo...	50	MDF	Left worki...	Left worki...	0.53	0.22	0.02	Division by curves	33780	TopSolid
6	4	Partition_top_1	part 26	1433.104	740.501	19	270	TopSolid-33791	0	.\MDF_201_w...	Partition_top_1	52	MDF	Left worki...	Left worki...	0.52	0.48	0.02	Division by curves	33791	TopSolid
7	3	Partition_med_high_1	part 27	1063.988	747.553	19	270	TopSolid-40758	0	.\MDF_201_w...	Partition_med_hi...	54	MDF	Left worki...	Left worki...	0.5	0.42	0.02	Division by curves	40758	TopSolid
8	7	Partition_top_5	part 28	1235.306	236.88	19	270	TopSolid-33933	0	.\MDF_201_w...	Partition_top_5	56	MDF	Left worki...	Left worki...	0.47	0.46	0.02	Division by curves	33933	TopSolid
9	8	Partition_top_3	part 29	782.421	308.586	19	270	TopSolid-33793	0	.\MDF_201_w...	Partition_top_3	58	MDF	Left worki...	Left worki...	0.62	0.58	0.02	Division by curves	33793	TopSolid
10	9	Partition_med_high_5	part 30	487.95	539.052	19	270	TopSolid-40762	0	.\MDF_201_w...	Partition_med_hi...	60	MDF	Left worki...	Left worki...	0.47	0.42	0.02	Division by curves	40762	TopSolid

OK Cancel INITIALIZE

What's New in TopSolid'SheetMetal v6.22



This section describes the enhancements made to version 6.22 of TopSolid'SheetMetal.

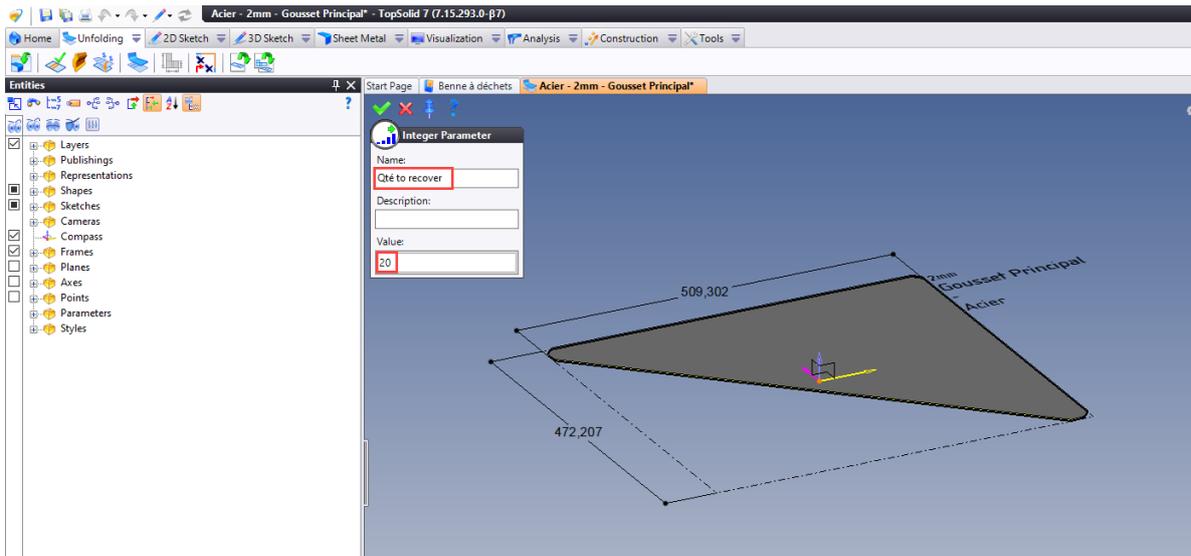
V7-V6 Link

Retrieving the quantity defined in the V7 part

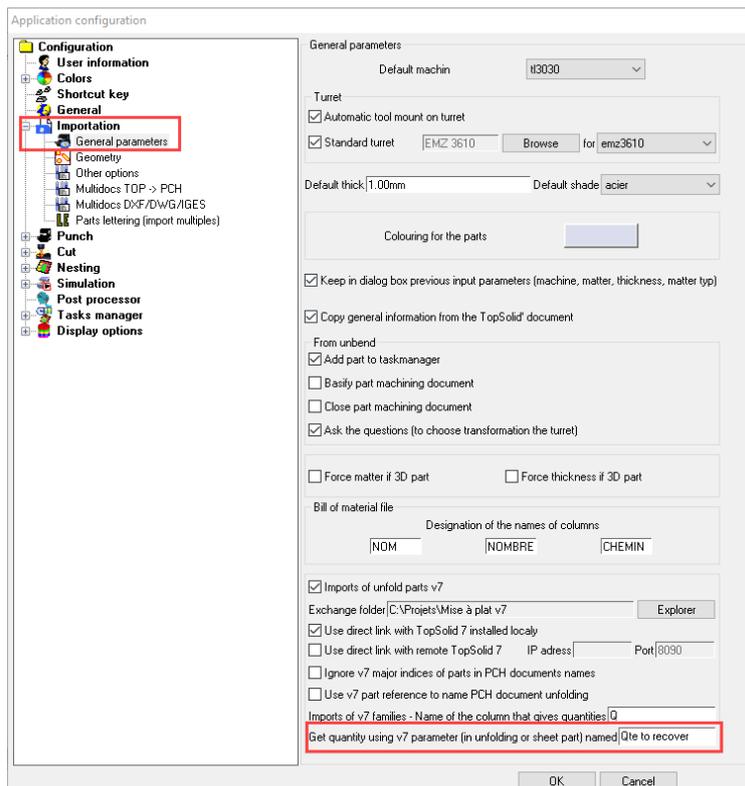
When importing V7 unfolding documents, you can retrieve the quantity even if the name of the assembly has not been defined.

Review the method:

- In the **TopSolid 7** unfolding document, create a new **integer parameter**.
- Enter the quantity to be produced.

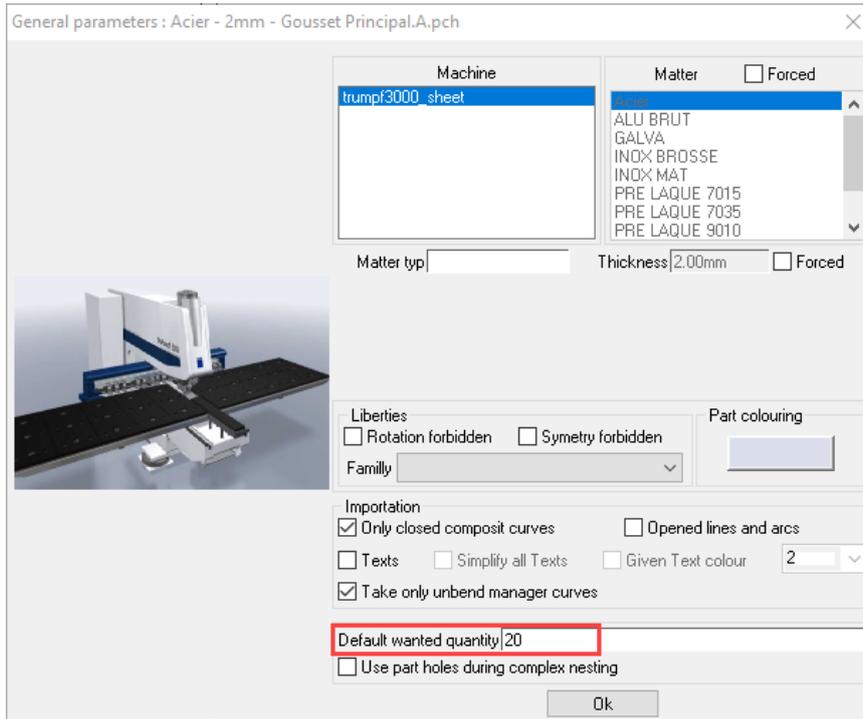


- Check the unfolding document into the vault.
- In **TopSolid V6**, configure the parameter name in the **Get quantity using v7 parameter (in unfolding or sheet part)** named field of the **Tools > Options > Importation > General parameters** command.



- Import the unfolding document in **TopSolid V6**.

The quantity is retrieved in the general parameters.



If the parameter does not exist in the unfolding document, the system will search for the parameter in the V7 sheet metal part.

If the parameter exists both in the unfolding and in the sheet metal part, the value used will be that of the unfolding.

Important: This feature is only available with the V7-V6 direct link. The quantity is not retrieved until the PCH document is created.

Multi-document Import

Retrieving the general data from a single text or the DXF file name

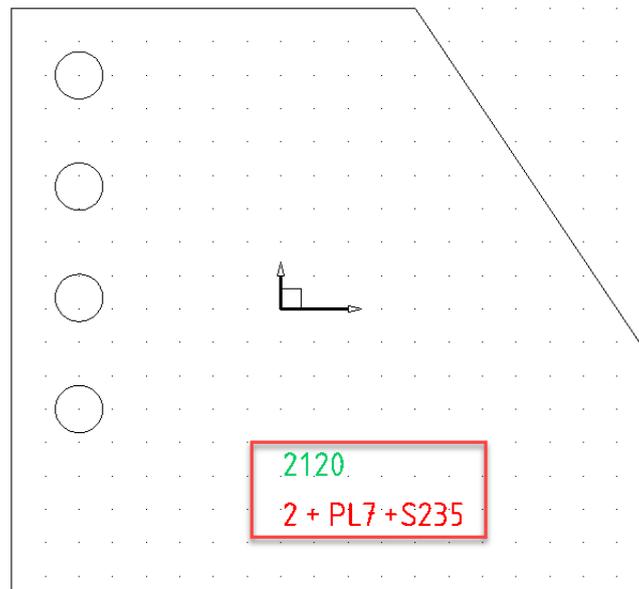
Two new configurations enhance the search for general part data (material, thickness, quantity and reference) when importing the DXF/Iges/DWG multi-documents.

Information can be found in a single text or directly in the DXF file name.

These new configurations are available at the following locations:

- C:\Missler\v622\local\englishUS\template\pch_import_mdocs\Config_4_ex.cfg
- C:\Missler\v622\local\englishUS\template\pch_import_mdocs\Config_5_ex.cfg

In the example below, the quantity, thickness and material of the file are automatically retrieved.



V7 Import

V7 import based on a set resulting from a set family

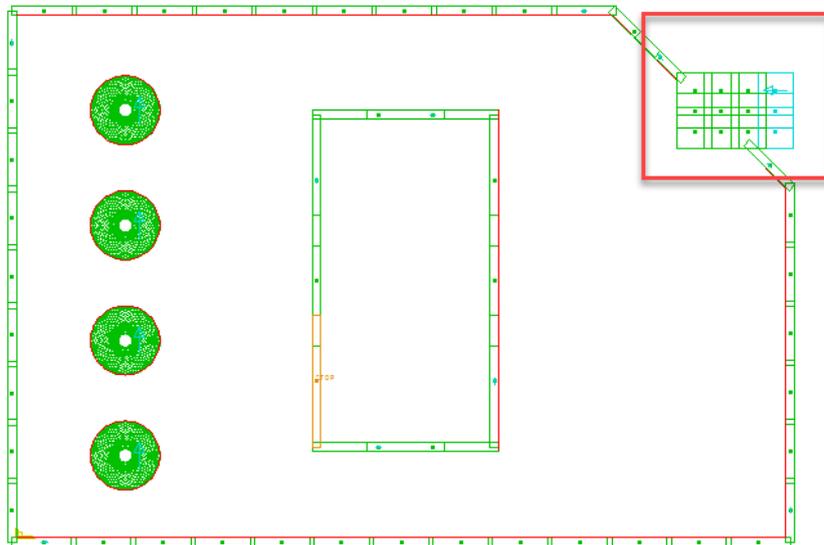
When importing a V7 unfolding document, the filter on the set definition allows you to view the sets contained in the V7 set families.

The assemblies contained in the family document are available in V6.

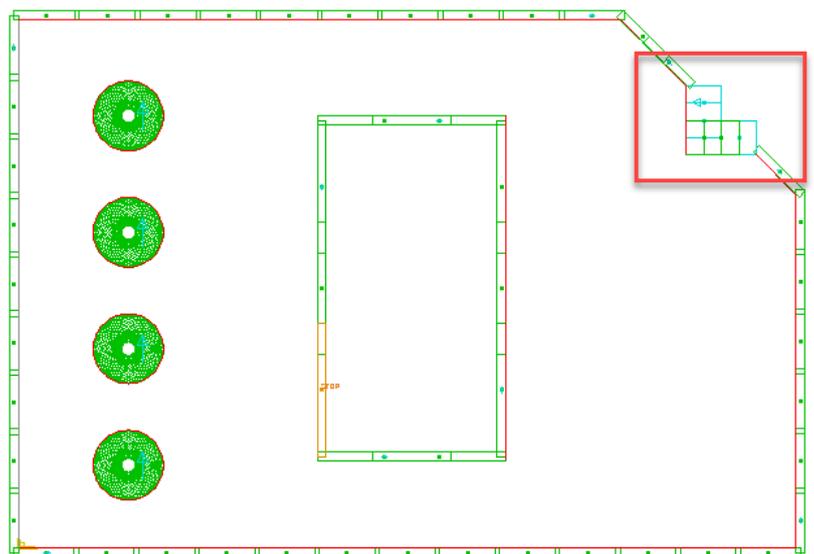
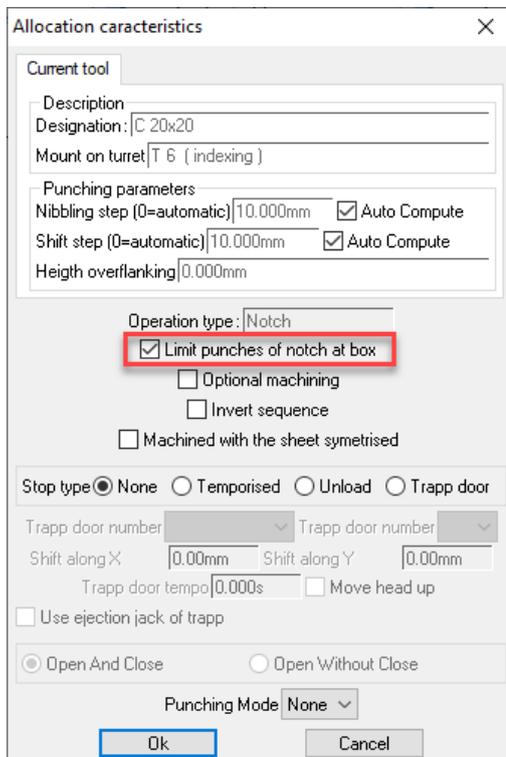
Punching

Triangular shaped notches on the edge of a part

The new **Limit punches of notch at box** option allows you to limit the inclined notches resting on two segments along the edge of the part.



To do this, you simply have to modify the sequence using the  wrench icon, then check the **Limit punches of notch at box** box in the dialog box.

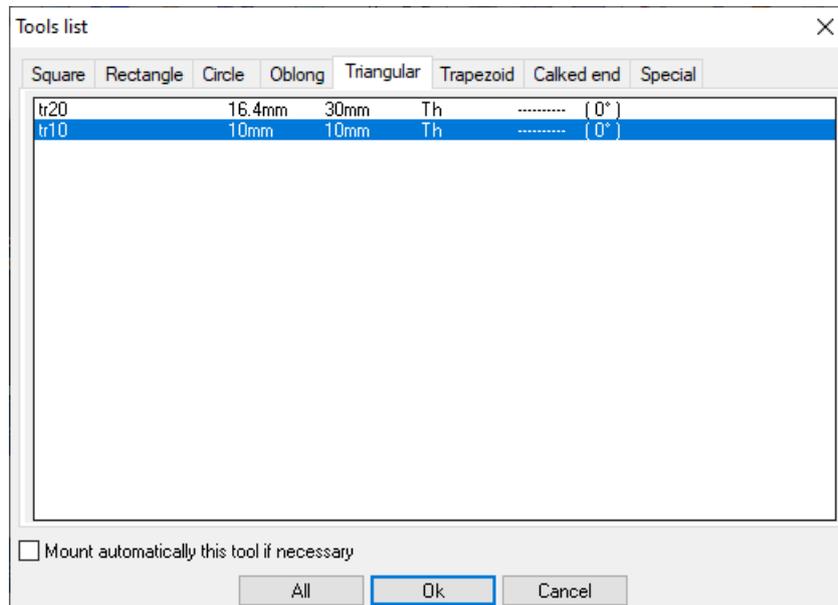


Tool on point along a profile: Triangular tools



The **Put a tool on point along profile** command has been enhanced for the triangular tools.

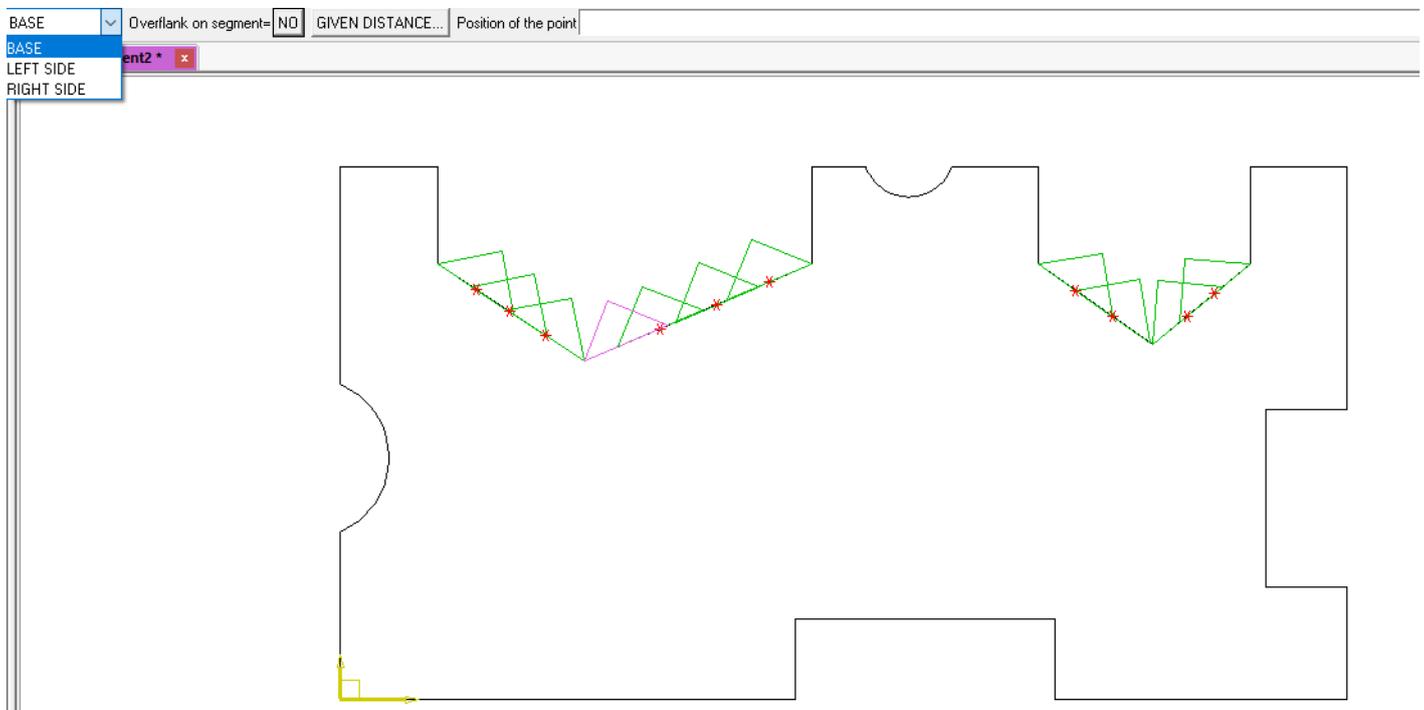
- Select the desired tool from the list.



- Click on the **Along segment** button.



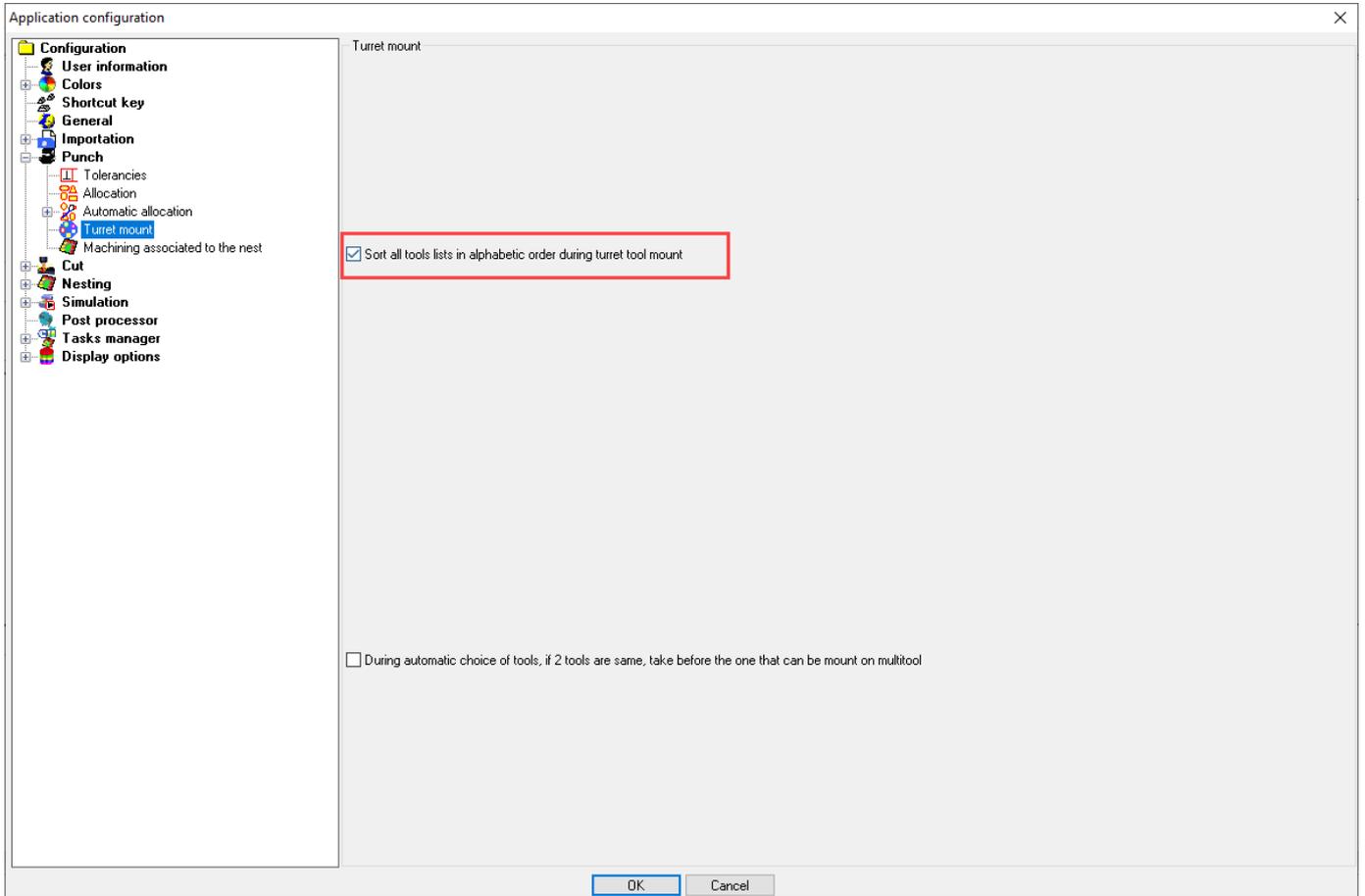
- Select the segment. You can use the base or one of the sides of the triangle.



Lists of tools

By default, when mounting the tools on the turret, the tools are sorted by size, from the largest to the smallest. The new **Sort all tools lists in alphabetic order during turret tool mount** box allows you to sort the tool lists alphabetically in the same order as in the tool management.

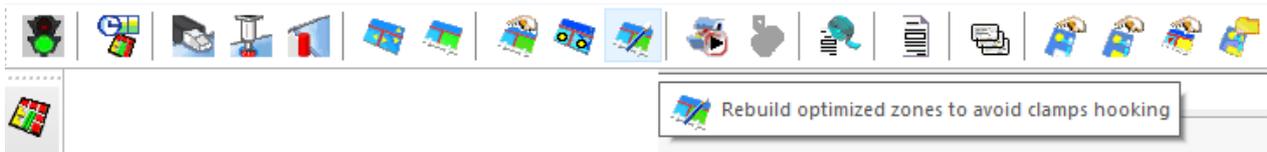
This option is available in the **Tools > Options > Punch > Turret mount** command.



Automatic zone management

The new **Rebuild optimized zones to avoid clamps hooking** command prevents the machine clamps from hooking during the repositioning operations.

Prerequisite: The machine must have the clamps free on the rule and the number of clamps is limited to 2.



After selecting the command, repositioning lines will be added so as not to hook the machining operations that have just been performed. The positions of the clamps are readjusted accordingly.

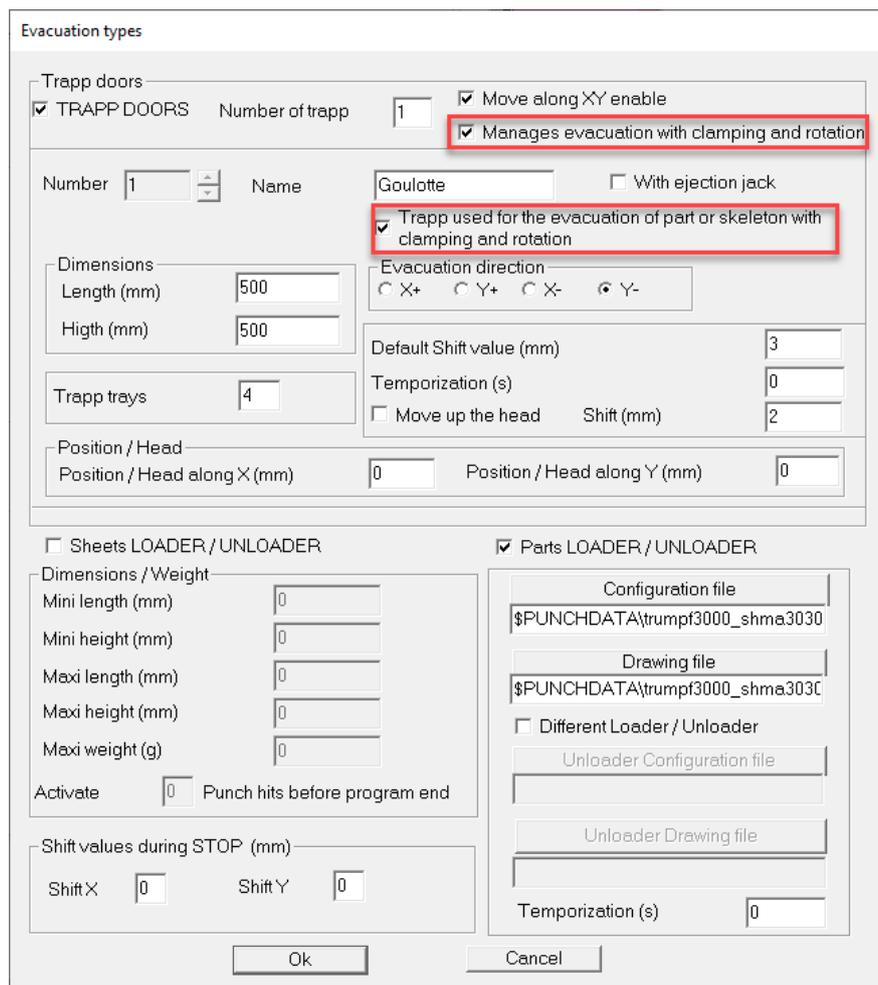
When it is not necessary to perform a repositioning to machine the part, this command is useful to find a clamp position that does not cause any punches under the clamps.

Trap door evacuation with clamping and rotation

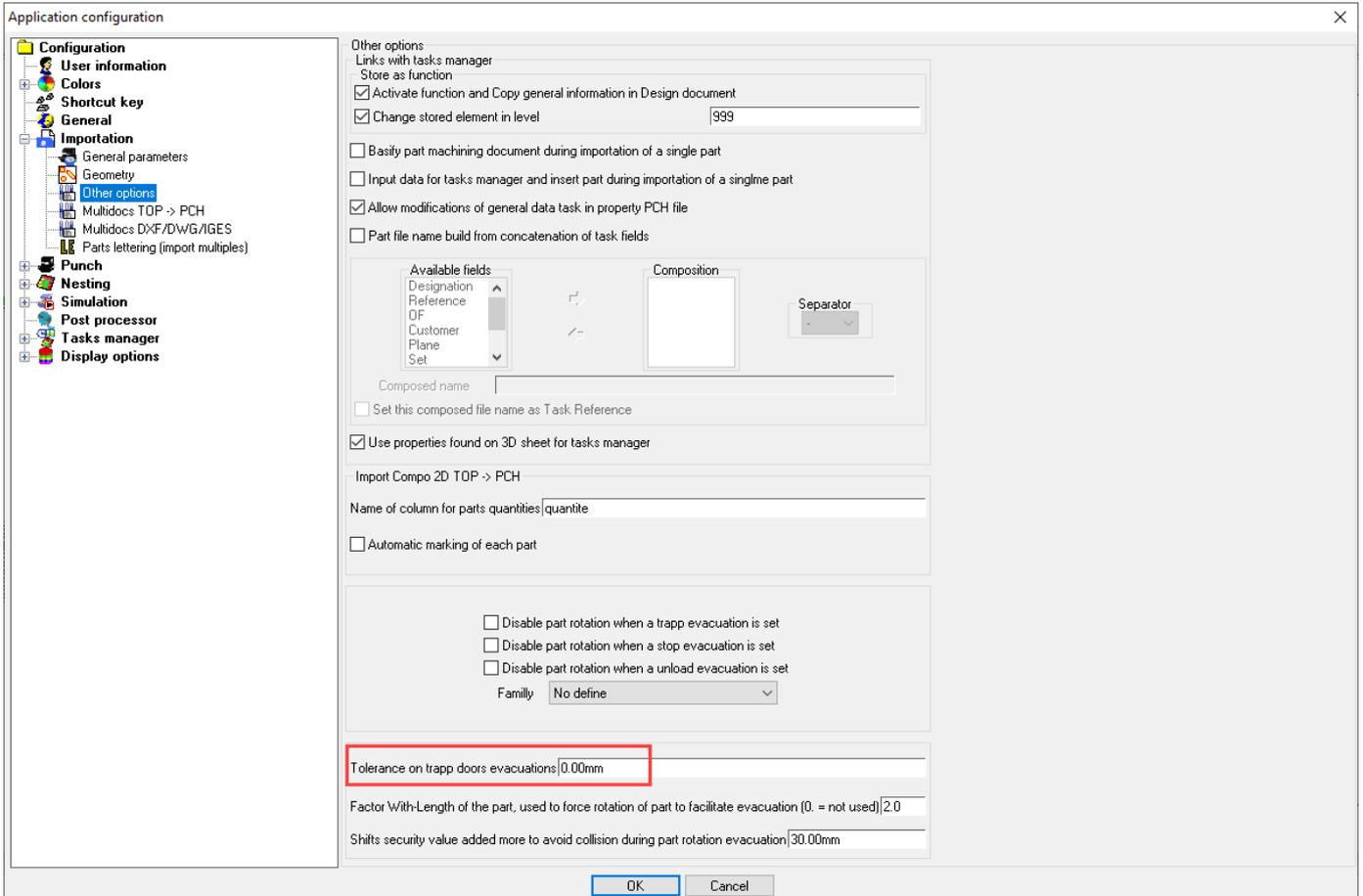
In the **Manage > Machines > Evacuation** command, the new **Manages evacuation with clamping and rotation** box allows you to indicate whether the machine manages the evacuation with clamping.

This option is available in the machine management for the punching machines only.

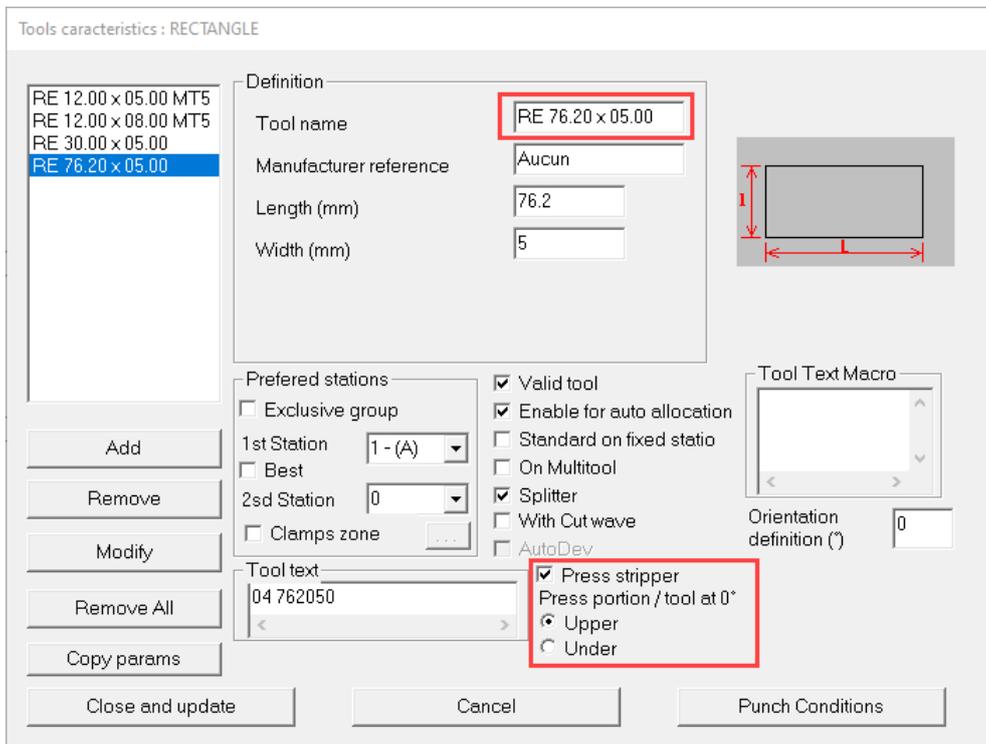
You can also indicate on one of the trap doors whether it can be used for the evacuation with prior rotation.



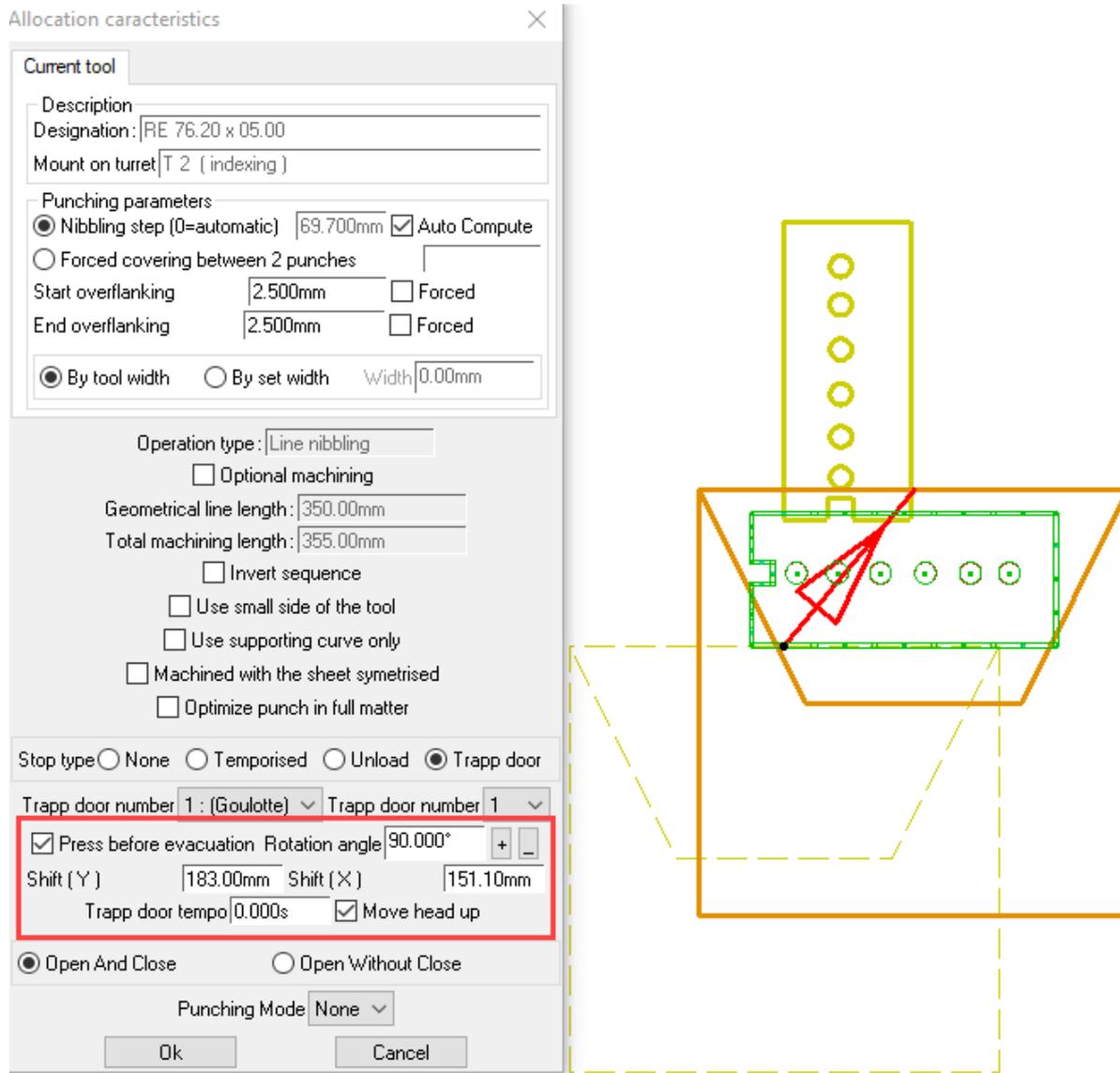
You can enter a **tolerance value on the trap door evacuations** in millimeters. By default, the value is 0, which allows the part to freely pass through the trap door.



The tool on the evacuation line must have a special press stripper. The new **Press stripper** box is available in the **Manage > Tools** command.



You can enable the rotation by using the wrench icon on the allocation.

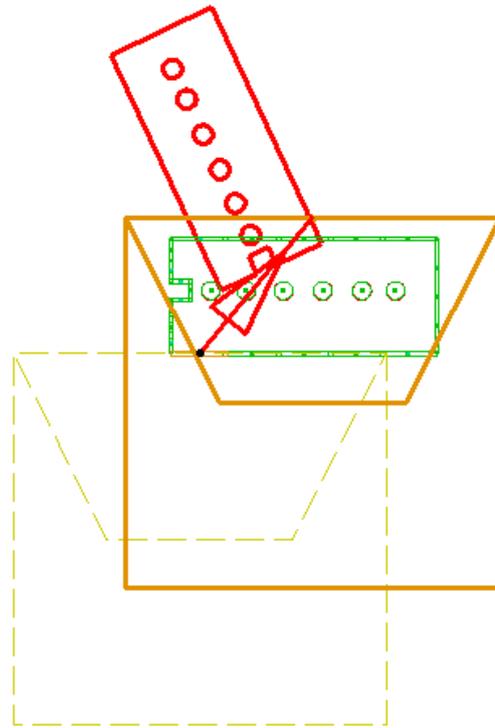


When selecting the **Trap door** option and selecting a trap door from the drop-down list, a pre-calculation is performed to determine the value of the rotation, as well as the Y and X shift values.

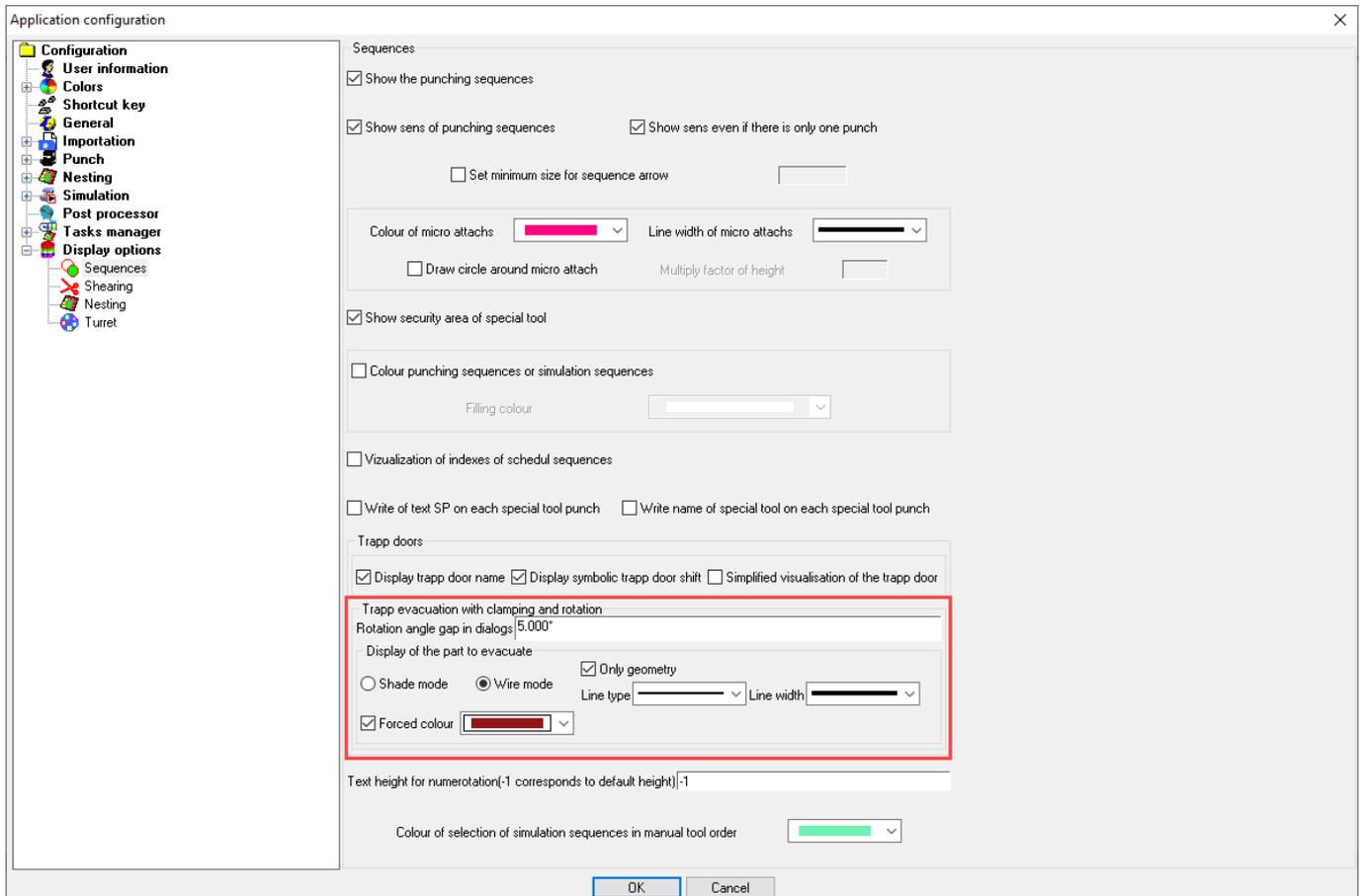
The part is shown in its final position.

The + and - buttons allow you to increment or decrement the rotation of a given step.

The part is displayed in red in the case where a collision is detected if the shifts are not large enough to rotate correctly.



The display options can be adjusted in the **Tools > Options > Display options > Sequences** command.



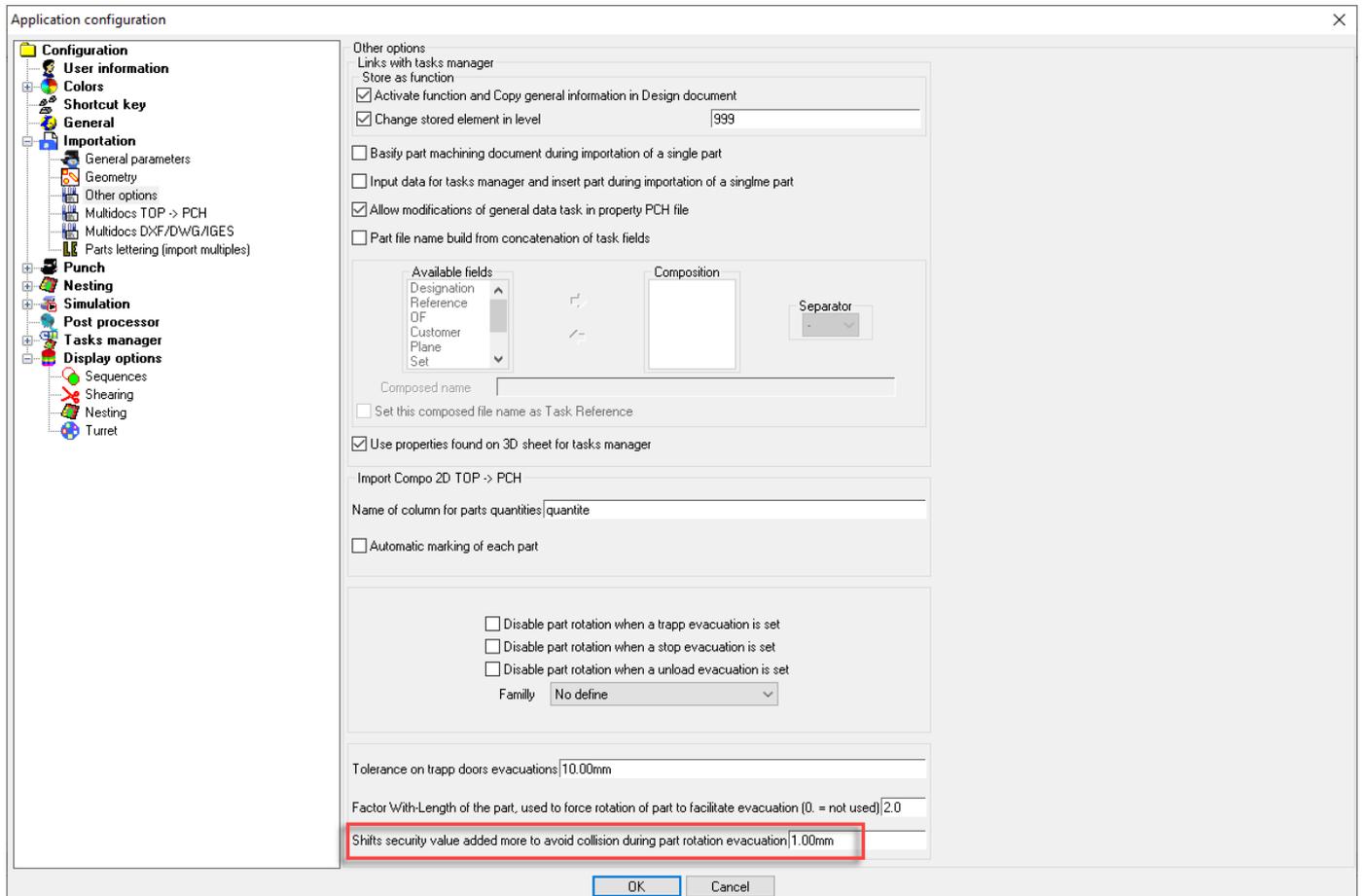
Configuration words

Configuration words are used to determine the position of the trap door. The last value indicates the position of the punch when the part is rotated before being evacuated.

Configuration word:

```
# Position of the trapp door evacuation on a given machine; for a given trapp door
# Parameter 1 = Name of the machine
# Parameter 2 = N° of the trapp (starting from 1)
# Parameter 3 = Position for standard evacuation (1 = bottom, 2 = right, 3 = top, 4 = left)
                (1 = bottom by default)
# Parameter 4 = Position for evacuation with rotation (if machine allows it) (1 = bottom, 2 =
right, 3 = top, 4 = left) (4 = left by default)
#ZPCHAFAUTO_TRAPP_POSITION"aries245"          1          1          4
```

An additional parameter allows you to define a **security value to be added to the calculated shifts to avoid collisions during the part rotation before the evacuation.**



Configuration word:

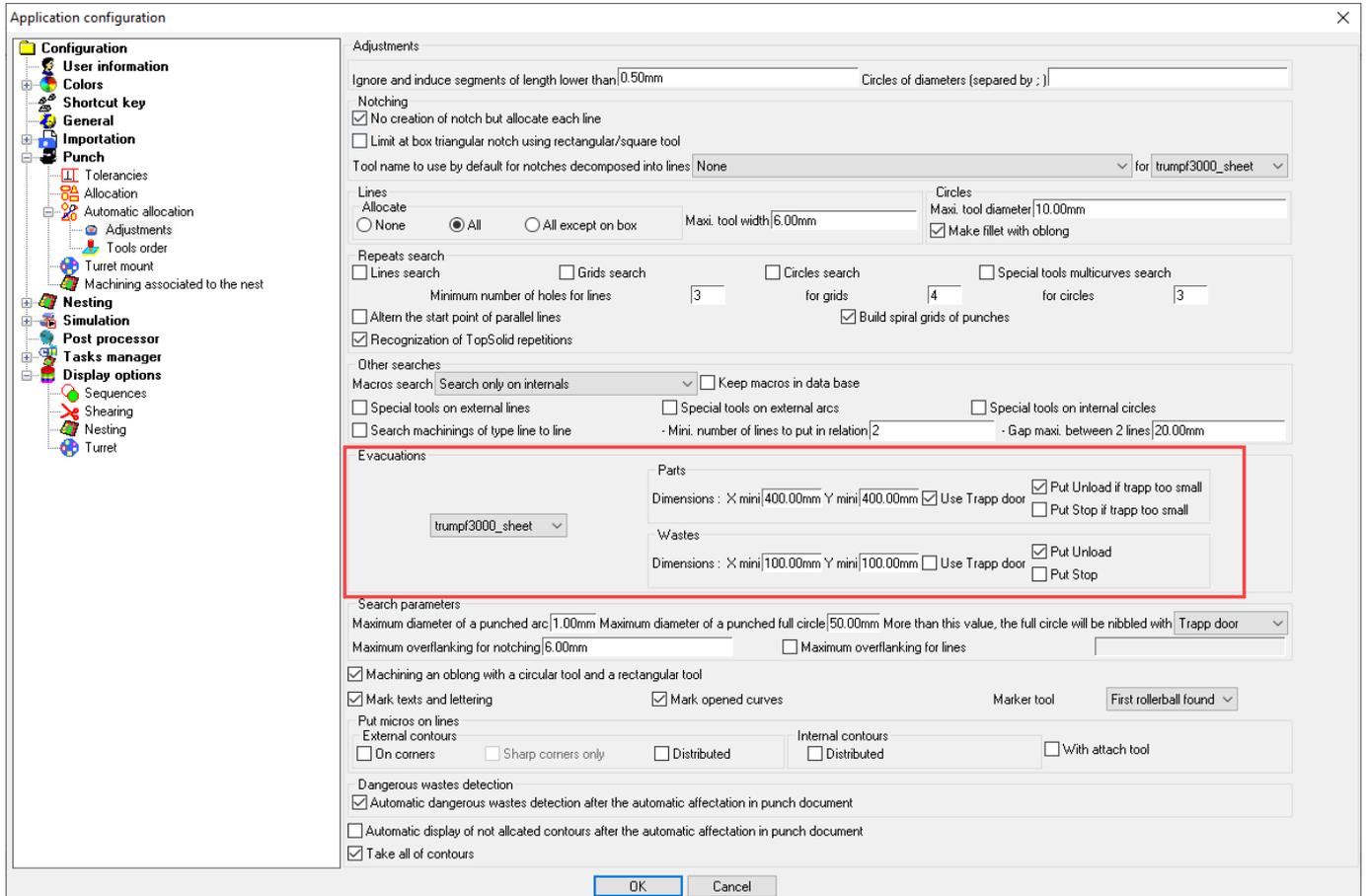
```
# Trapp door with rotation before evacuation : security value added to shifts X and Y (computed
automatically) needed before rotation and to avoid collision during part rotation (default =
30mm)
PCH_TRAPP_ROT_SHIFT_SECURITY          30.
```

Automatic allocation

In the **Tools > Options** command, the **Automatic allocation** section has evolved.

You can now specify dimension ranges to trigger the part evacuation.

You can also indicate whether you want to use the trap door or not (and thus to force, for example, an evacuation by unloading with suckers). If the trap door is used, it will have priority for the evacuation. If the trap door is too small, an unloader with suckers takes over.



If the machine is equipped with a sucker unloader, the following configuration word allows the unloading punch to be placed at a given position on the part or waste.

Configuration word:

```
# Position of the unload with suckers on the part or on a waste
# Parameter 1 = Name of the machine
# Parameter 2 = Put in horizontal (1) ; Or in vertical (0) (1 by default)
# Parameter 3 = Position : 1 = Top Left, 2 = Top Right, 3 = Bottom Right, 4 = Bottom Left (4 by default)
#ZPCHAFAUTO_UNLOAD_POSITION "trumpf500" 1 4
```

Evacuation of the clamp skeleton using the trap door

The new **Evacuate skeleton in clamps using trap door** command allows each piece of the skeleton to be evacuated with the trap door using rotation.



Depending on the available trap doors, the system will propose a maximum size of the skeleton pieces, taking into account the evacuation tolerance.

The system will create small splittings (linked by a scheduling) with an evacuation with rotation, with the eventual opening of the clamp for the release of the piece.

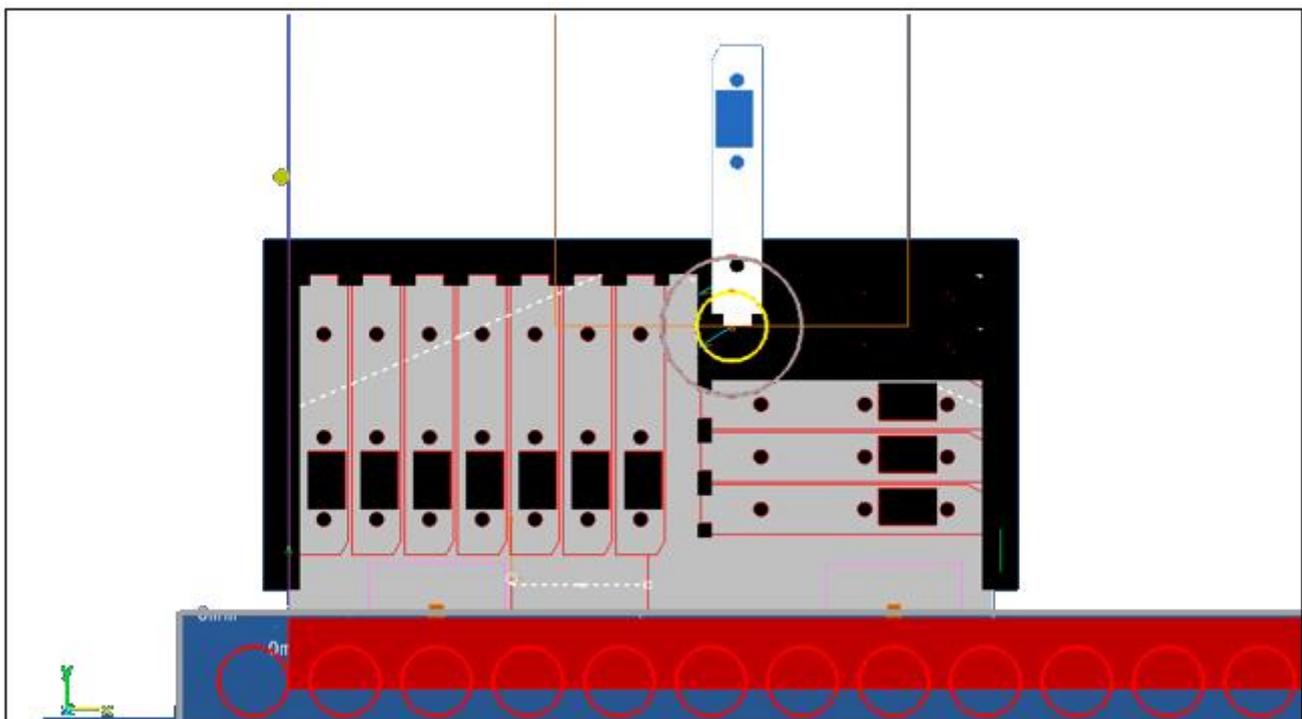
The length of each piece will be adapted according to the position of the clamps and the security area of the station.

The scheduling ends with a passing point to evacuate the last piece.

Note: When deleting one of the splittings in **Contour** mode, all of the splittings resulting from the cut of the skeleton pieces are deleted.

When the **Evacuate skeleton in clamps using trap door** command is selected again, the previous parameters are redisplayed. The old splittings are then replaced.

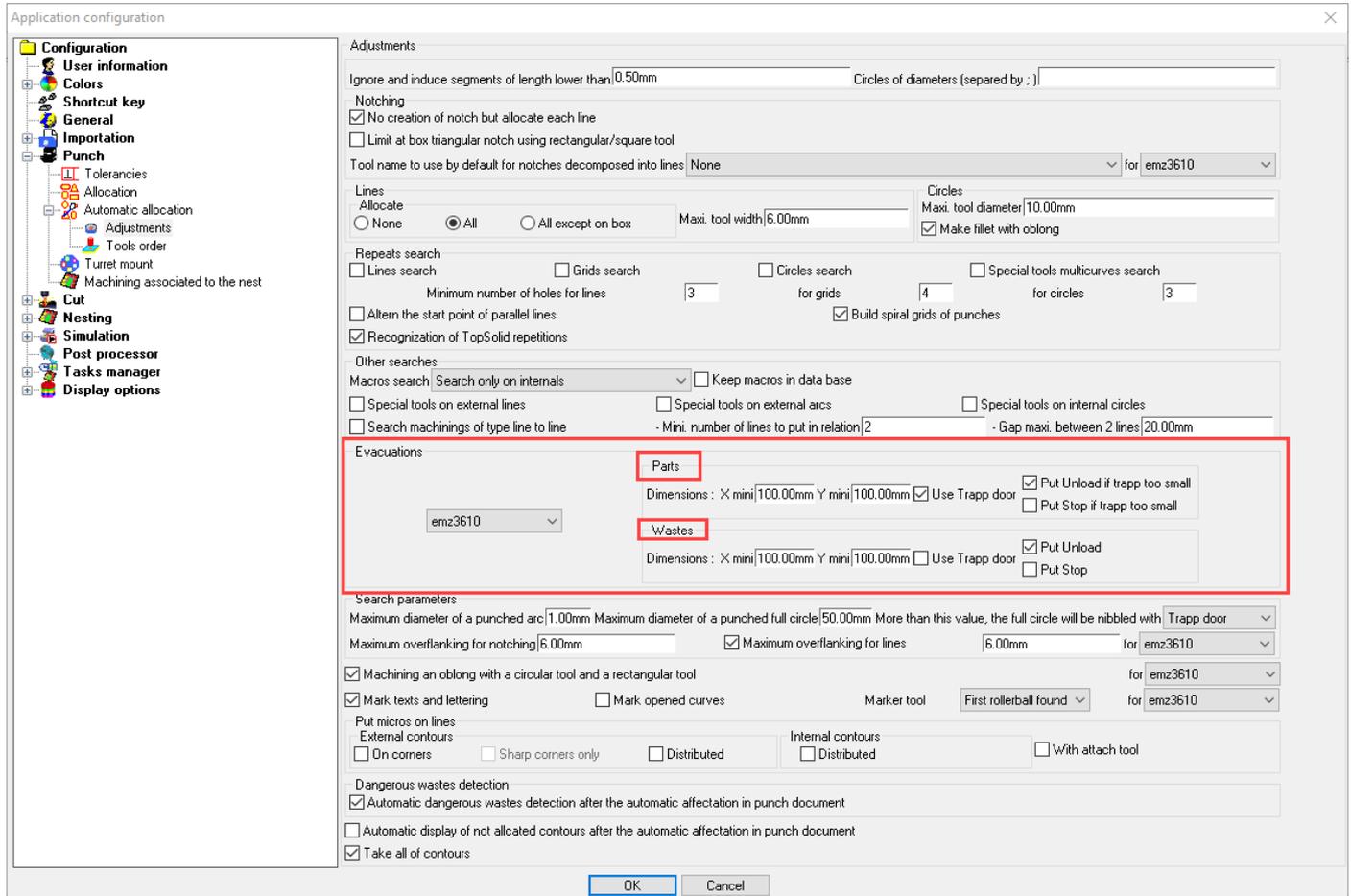
The simulation in shaded mode now shows the parts evacuated through the black trap door, as well as the parts evacuated by the sucker unloader.



Automatic unloading with suckers in automatic punching allocation

When the machine allows the part or waste to be unloaded with a sucker unloader, new options make it possible to automatically position the unloading and prepare the sucker selection.

This can be configured by default in the **Tools > Options** command.



The **Put unload** option is available for machines that have a part unloader.

If one or more trap doors exist, the system will first try to evacuate the part/waste through one of the trap doors. If all the trap doors are too small, the system will place an unloading.

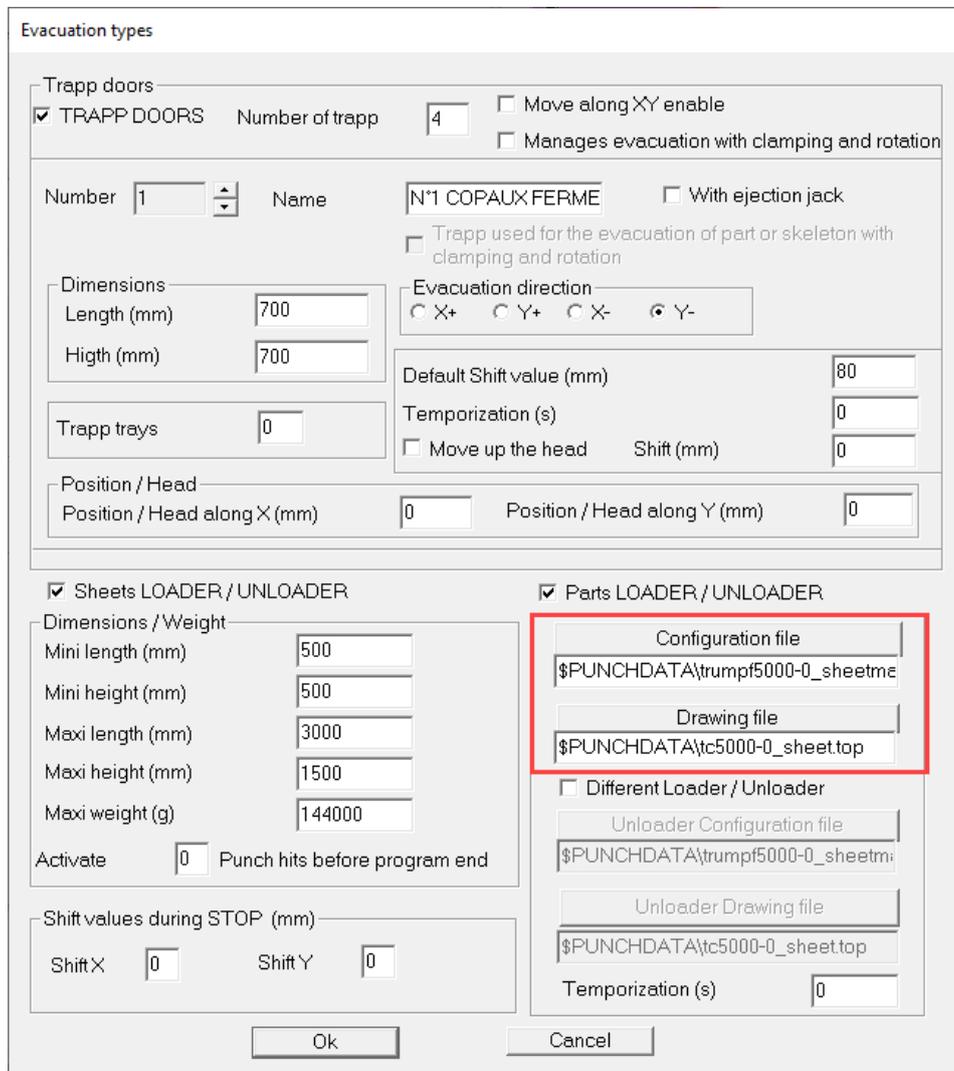
In the **Automatic allocation** command, these parameters can be changed on the fly.

Punching and Cutting

Configuring multiple unloaders

For the machines equipped with an automatic part unloader, you can now define several usable unloaders.

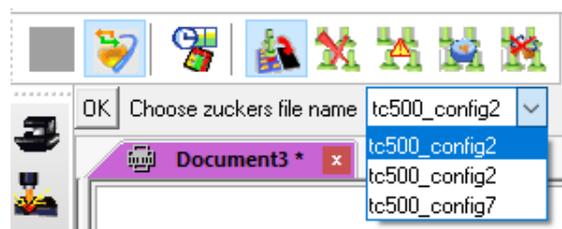
Prerequisite: The loader and unloader must be identical in the machine management.



To configure multiple unloaders, you simply have to create the .TOP files of the different drawings and indicate their path using the following configuration word in the loader/unloader configuration file.

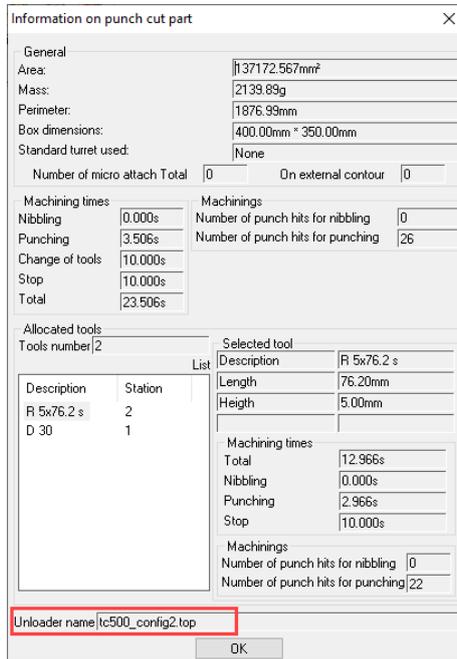
```
# Other unloader configurations
ZPCHSM_OTHER_UNLOADERS_PATHS      $PUNCHDATA\tc500_config2.top      $PUNCHDATA\tc500_config7.top
```

When using a loading/unloading command, you now have to select the unloader to be used.



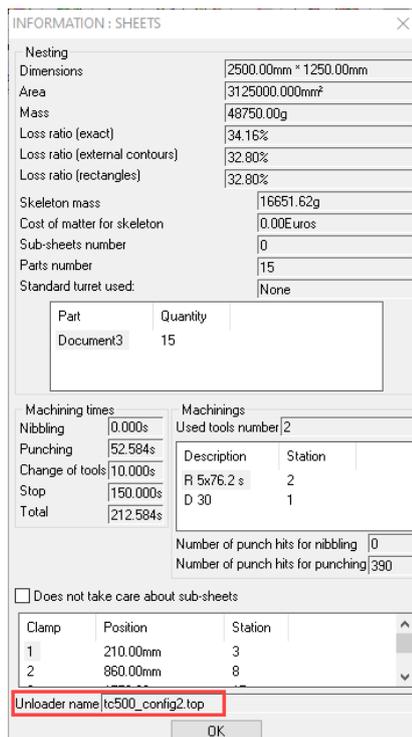
In a PCH document (part):

- The choice of the unloader is requested at the first unloading of the part or waste.
- This choice will be proposed again if all the unloadings (wastes and/or parts) have been removed.
- You can find out which unloader is being used in the **Information on parts** command.



In a MEG document (nesting):

- When the nesting document is created, the system checks whether the unloaders are the same for all the parts in the nesting whose suckers have been predefined in each part document. If a conflict arises, a message is displayed in the alpha bar (non-blocking message).
- The unloader selected for the nesting corresponds to that of the first nested part.
- You can find out which unloader is being used in the **Information on sheets** command.
- In case of a conflict, you can overload the position of the suckers on a local part that will take the unloader selected on the nesting.



An information message also appears in the alpha bar when inserting an incompatible unloader part.

The **Loader/Unloader > Load sheet** command, which automatically loads the sheet, will take the default unloader.

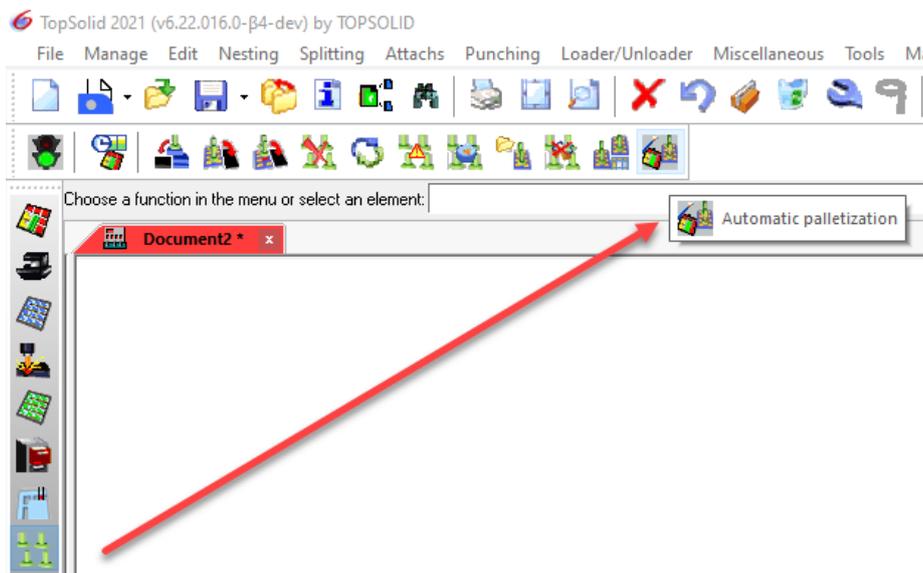
In the simulation, an alert is raised if the unloaders used are different on the unloadings of the nesting.

To remove the alert, you only have to position the new sucker configuration on these local parts.

With the **Automatic allocation** command, the default unloader will be the one defined in the machine parameters.

Automatic palletization

The new **Automatic palletization** command allows the automatic unloading of the parts on the pallets.

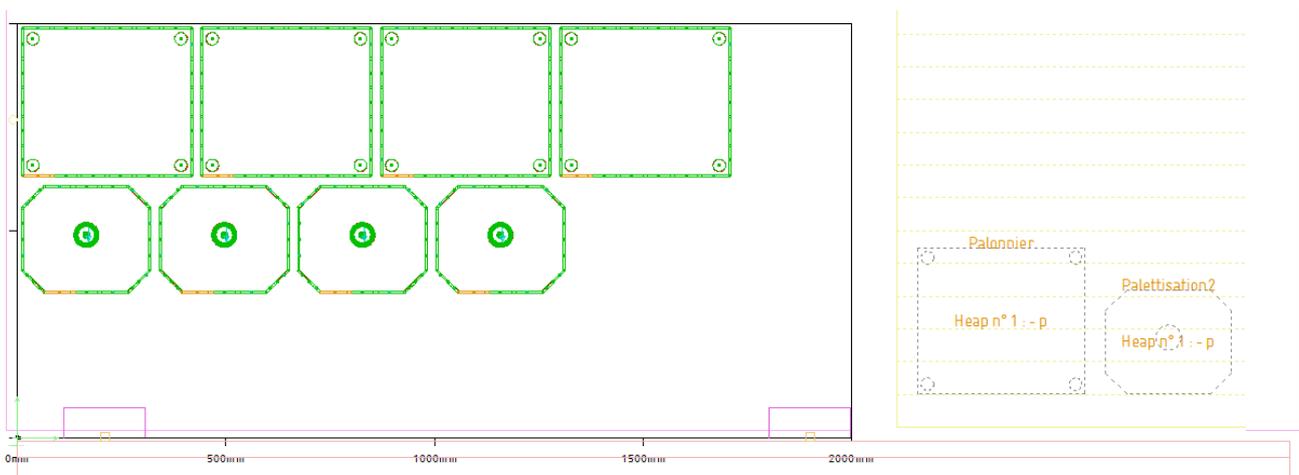


The parts to be unloaded can have only the unloading positioned and possibly the suckers selected.

If the suckers have not been defined on certain parts, the automatic palletization command will automatically select the suckers.

Simple nesting palletization

The system chooses the most suitable pallet configuration to support the largest parts and creates as many heaps as required for a given type of part.



If the parts have already been unloaded and other types of parts are added to the nesting, you have to restart the automatic palletization command that will calculate the unloading of the missing parts.

The automatic palletization command can be combined with the manual unloading command.

To adjust a position calculated by the palletization command, you have to use the **Parameters** command on a given part.

Order palletization

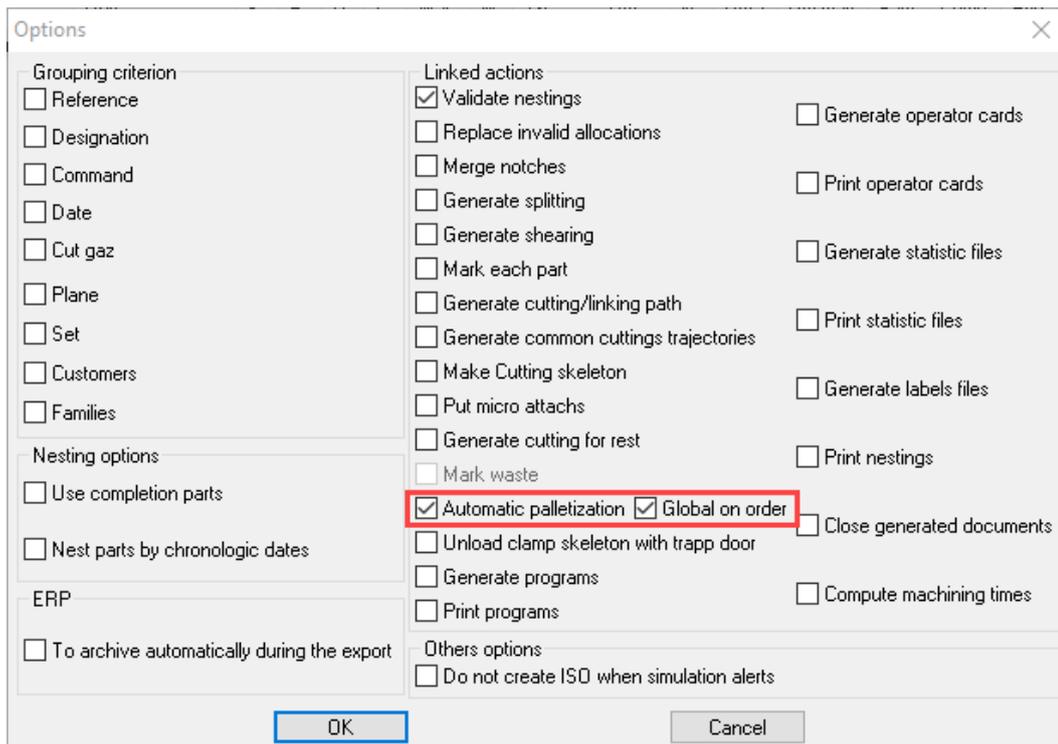
If the order consists of several sheets, you can start an automatic palletization "local to each nesting" or a palletization "global to the order".

The local palletization to the nesting will only unload the nesting parts by taking a pallet configuration adapted only to the nesting parts. The pallets will have to be removed after machining each nesting in the workshop.

The global palletization will reserve a place on the pallets for each type of part of the order. The pallet configuration will be the same for all the order's nestings.

Automatic palletization via the Task Manager

The linked actions can be configured in the Task Manager's options.



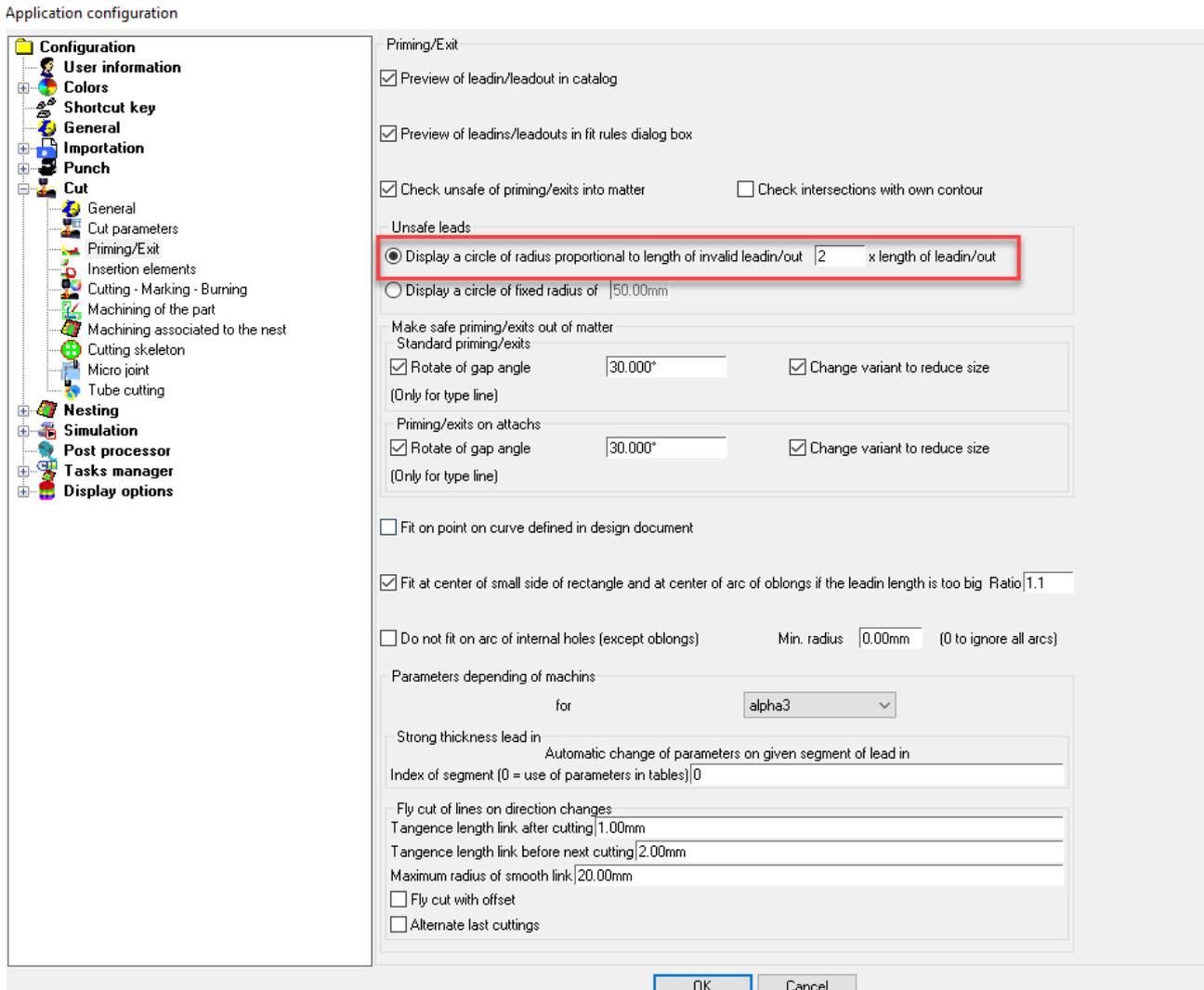
If only the **Automatic palletization** box is checked, the system will perform the automatic palletization "nesting by nesting" on the fly, as each nesting is created.

If the **Global on order** box is checked, the system first creates all the nestings and then performs the global palletization. If the nestings were closed, they are reopened and each simulation (program generation, if requested) is possibly redone.

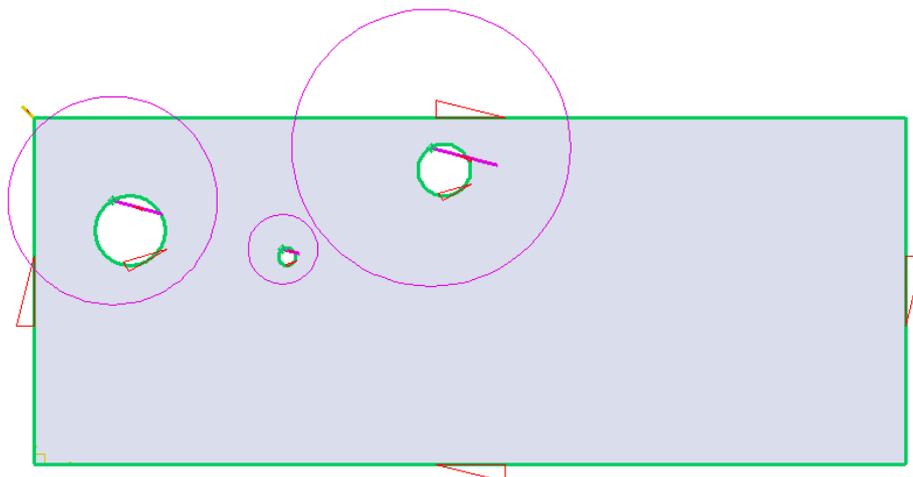
Cutting

Setting the size of the circle around the unsafe lead ins/lead outs

In the **Tools > Options > Cut > Priming/Exit** command, a new option allows you to configure the size of the circle in proportion to the size of the lead in/lead out.



You can also adjust the factor. The radius of the circle will then be equal to the length of the unsafe lead in /lead out multiplied by this factor.

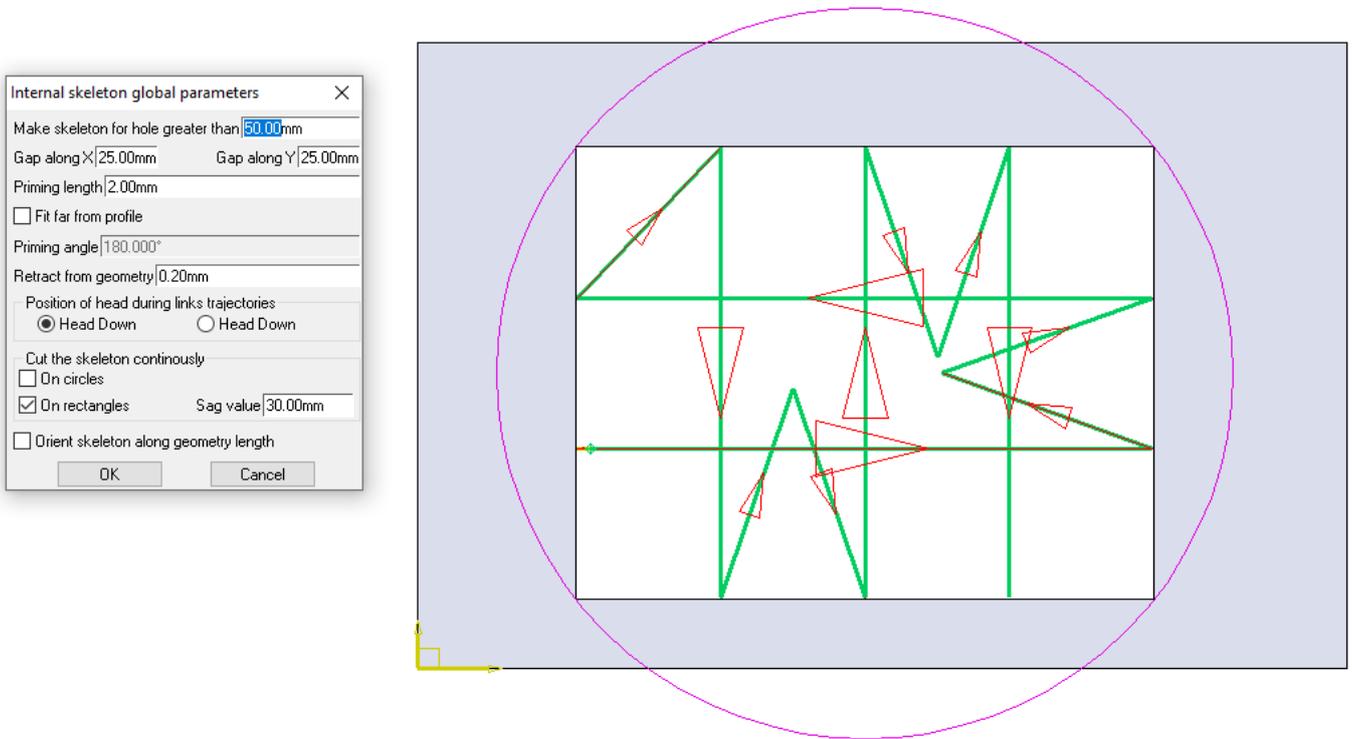


Continuous internal skeleton

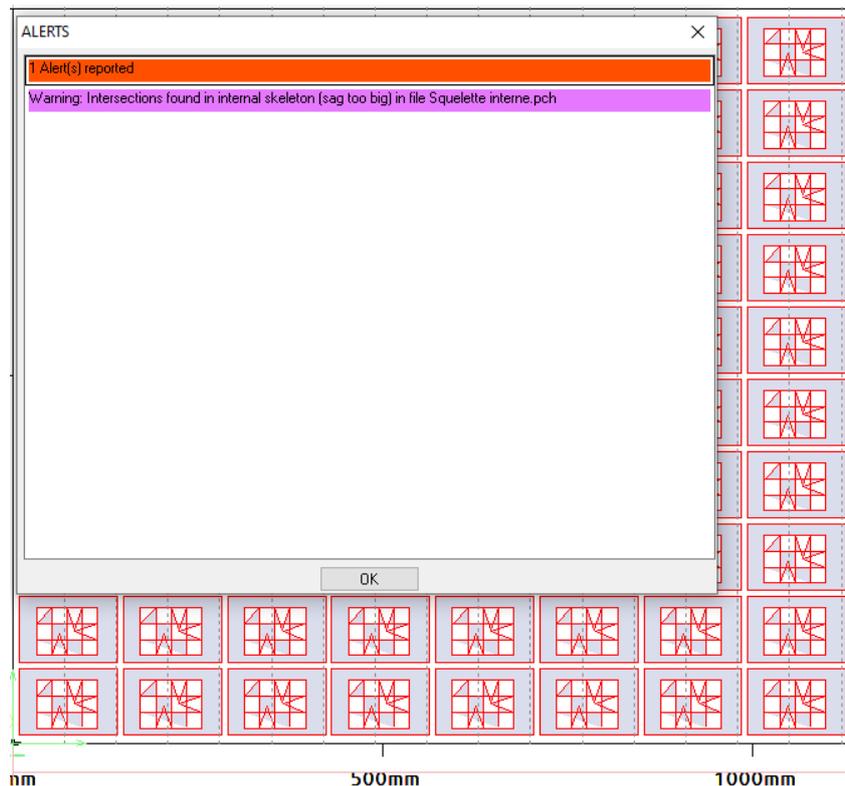
Sag value

When performing an internal skeleton cutting on a rectangle and using the continuous cutting option, if the value is too large, the lines may overlap and result in dangerous cuts.

The **Internal skeleton** command performs a check and, in case of overlaps, surrounds the affected contour with a purple circle. You can enter a **sag value** to remove these overlaps.

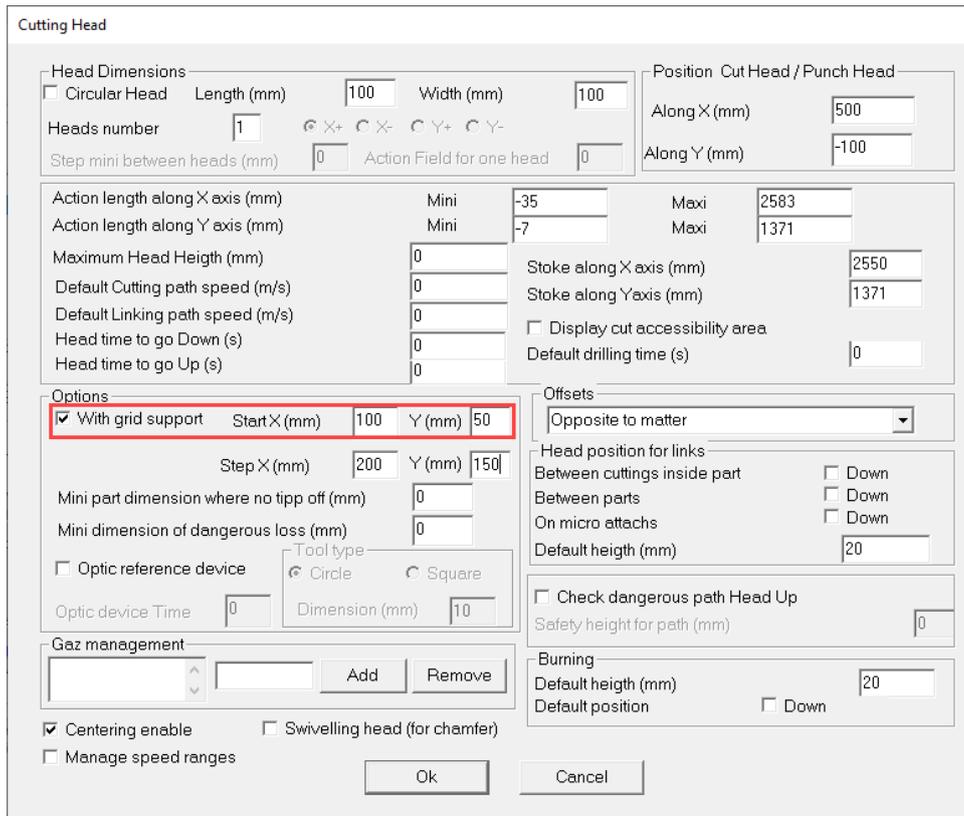


During the simulation, an alert is also displayed, indicating the name of the affected part.



Support grid visualization

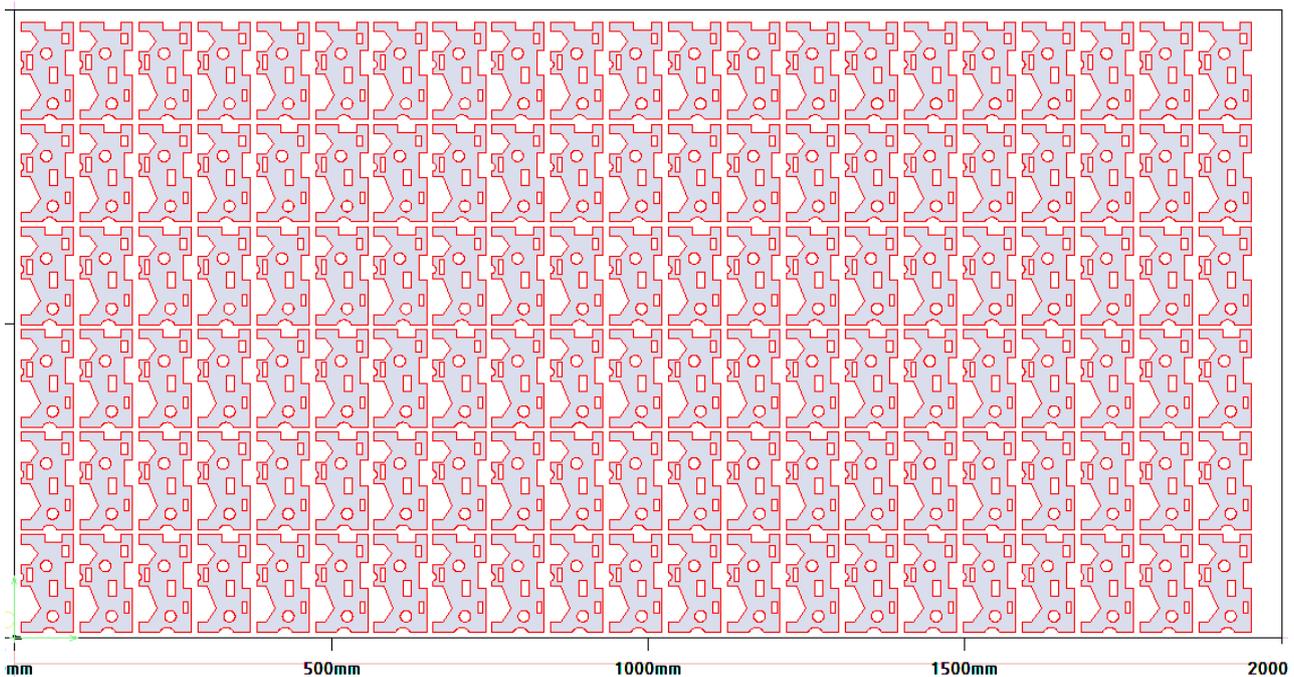
In the machine management, new options allow you to enter a X start value and a Y start value other than 0.



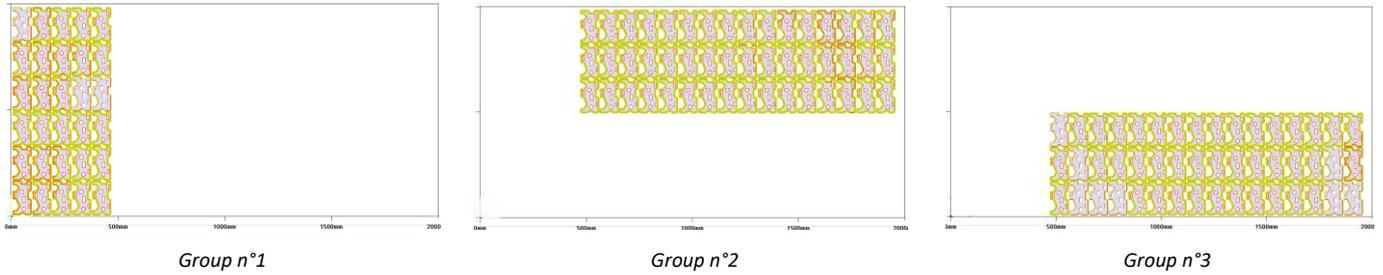
Saving and loading a cutting strategy

From a nesting simulation, you can now save a cutting strategy and then reload it to another nesting.

Here is an example of a nesting:

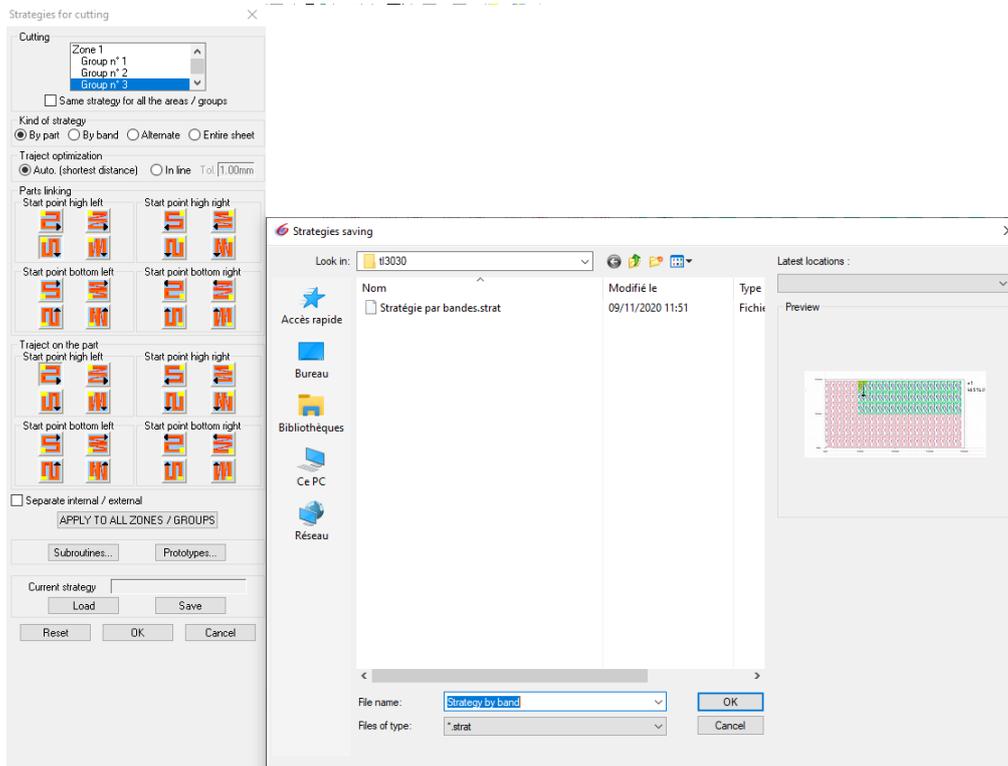
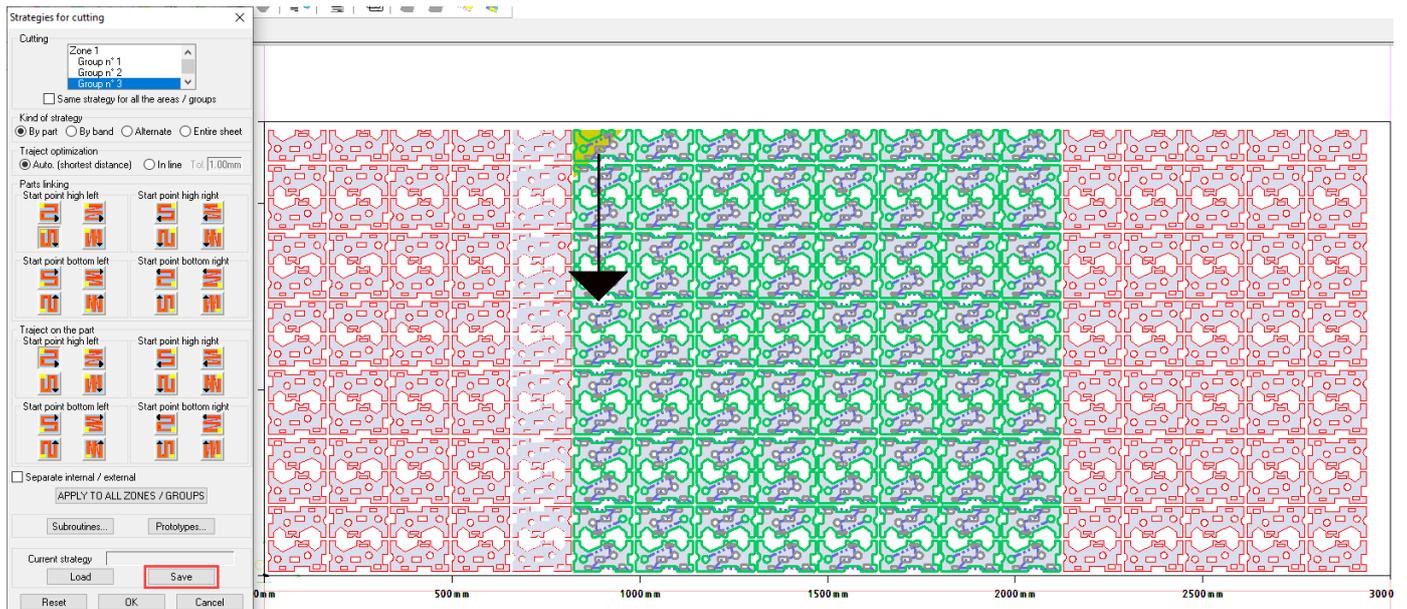


Group management:



Saving the strategy

In the simulation, when defining the cutting strategies, you simply have to click on the **Save** button.

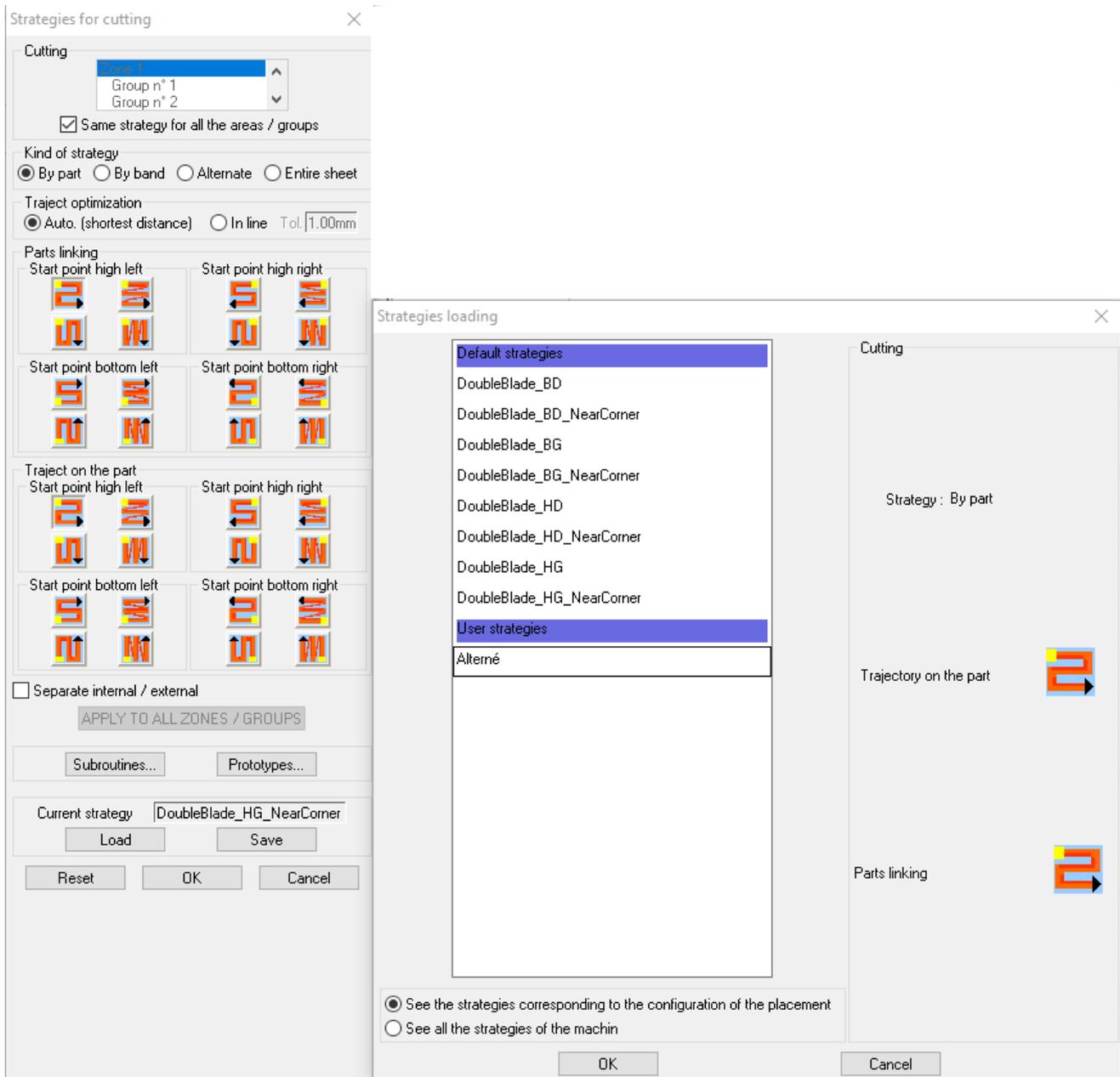


The saved strategy corresponds to that of the selected group. If the **Same strategy for all areas/groups** box is checked, the global strategy will be saved.

Loading the strategy

In the simulation, you simply have to click on the **Load** button.

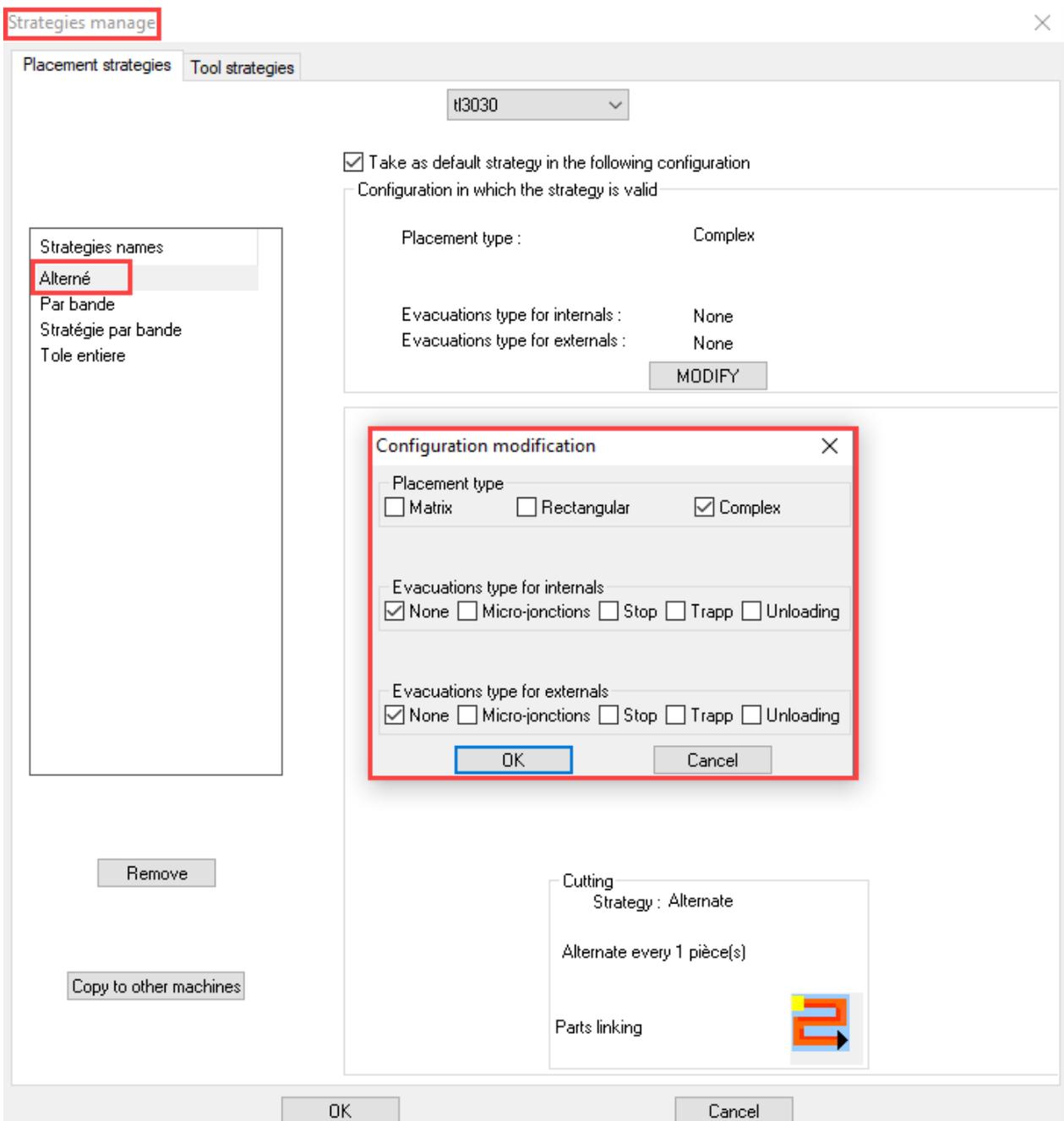
The loaded strategy will be applied to all areas/groups if the **Same strategy for all areas/groups** box is checked, or will only be applied to the selected area or group.



By default, the system only proposes the strategies that are compatible with the criteria of the current nesting.

Managing the strategies

The strategies can be managed via the **Manage > Strategies** command in the nesting document.



For pure cutting machines, only the **placement strategy** management can be used.

The **tool strategy** management is only valid for the punching machines or combined machines. If the environment only has pure cutting machines, the **Tool strategies** tab does not appear.

The **Cutting** section allows you to view the selected strategy type.

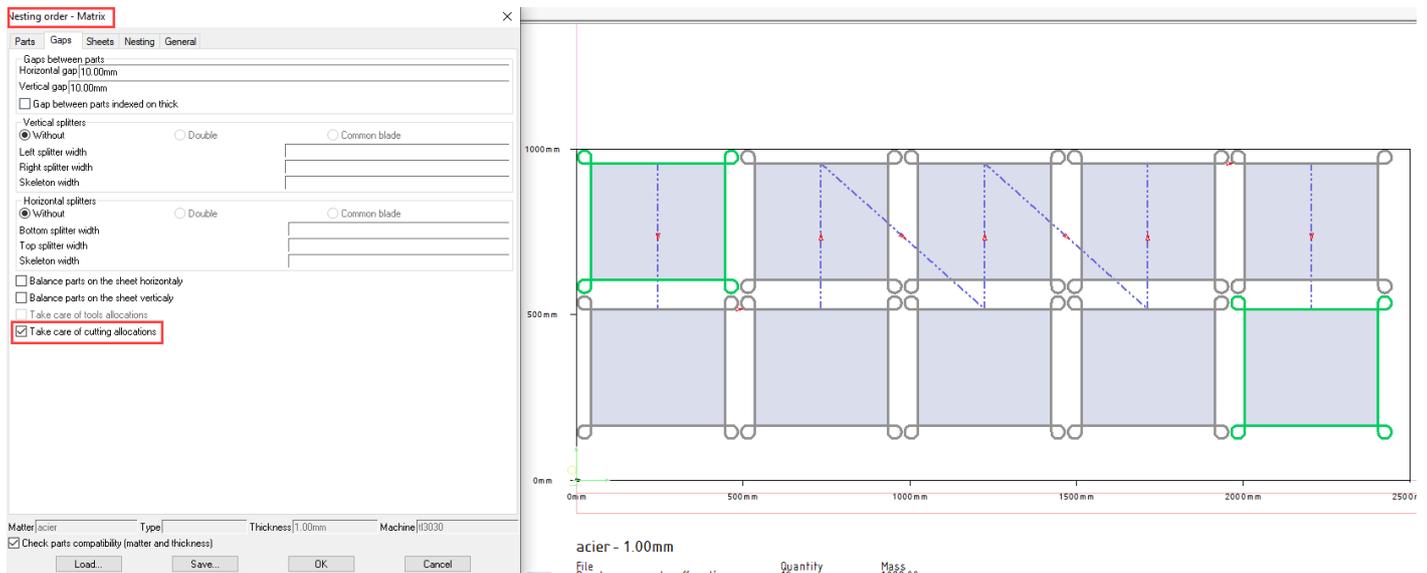
The **Modify** button allows you to define the **nesting type** as well as the **types of internal** and **external evacuations** for the applied cutting strategy.

If the **Take as default strategy in the following configuration** box is checked, the system will automatically load this strategy the first time a new nesting is simulated or when "resetting" the strategies of an old nesting.

If no user strategy is defined for the machine, the system will take a strategy **per part** by default at the start of any new simulation and will apply it to all areas/groups.

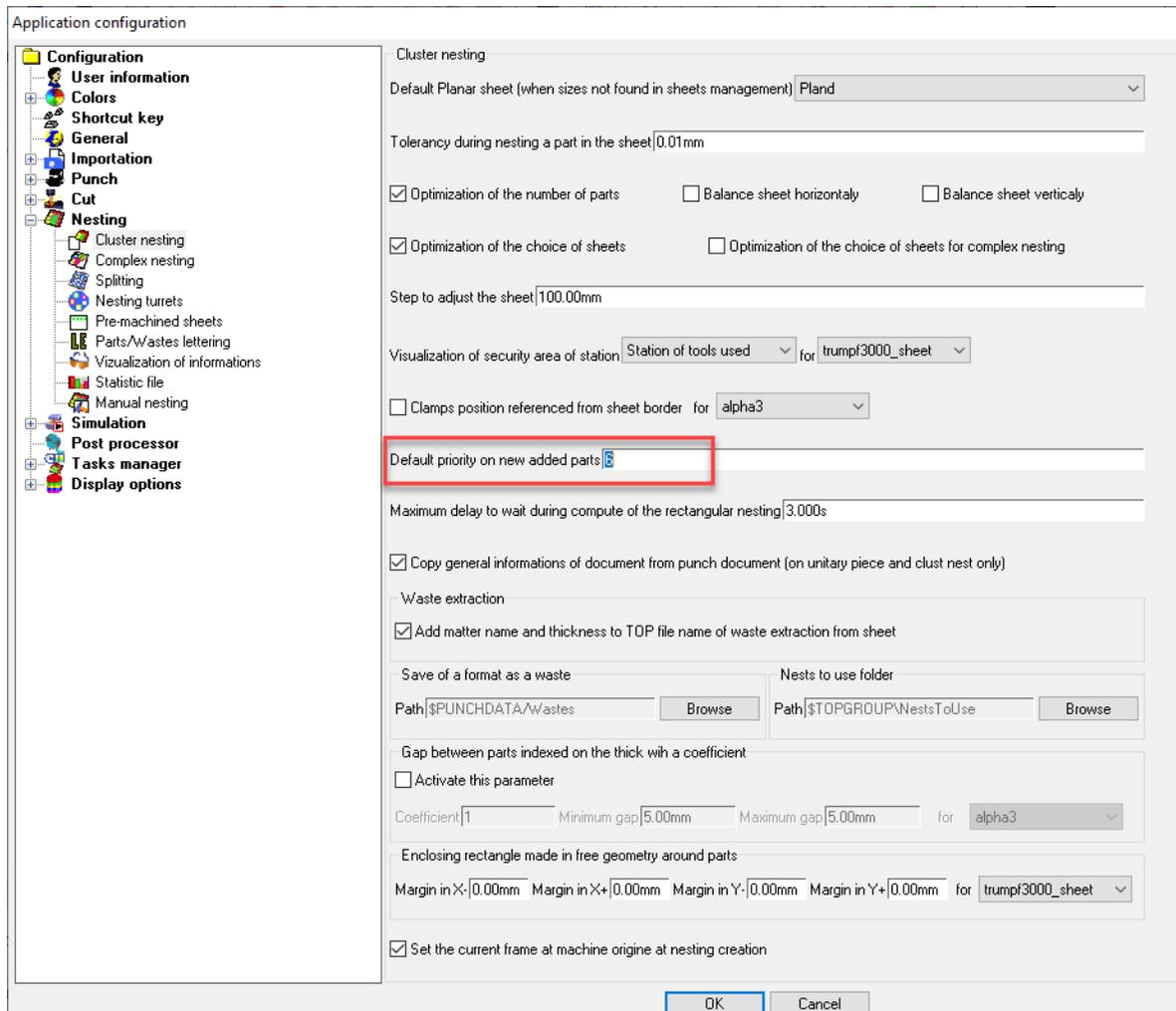
Nesting: Taking the cutting allocations into account

On the cutting machines or combined machines, you can now **take the cutting allocations into account** when performing a cluster, rectangular or unitary nesting.

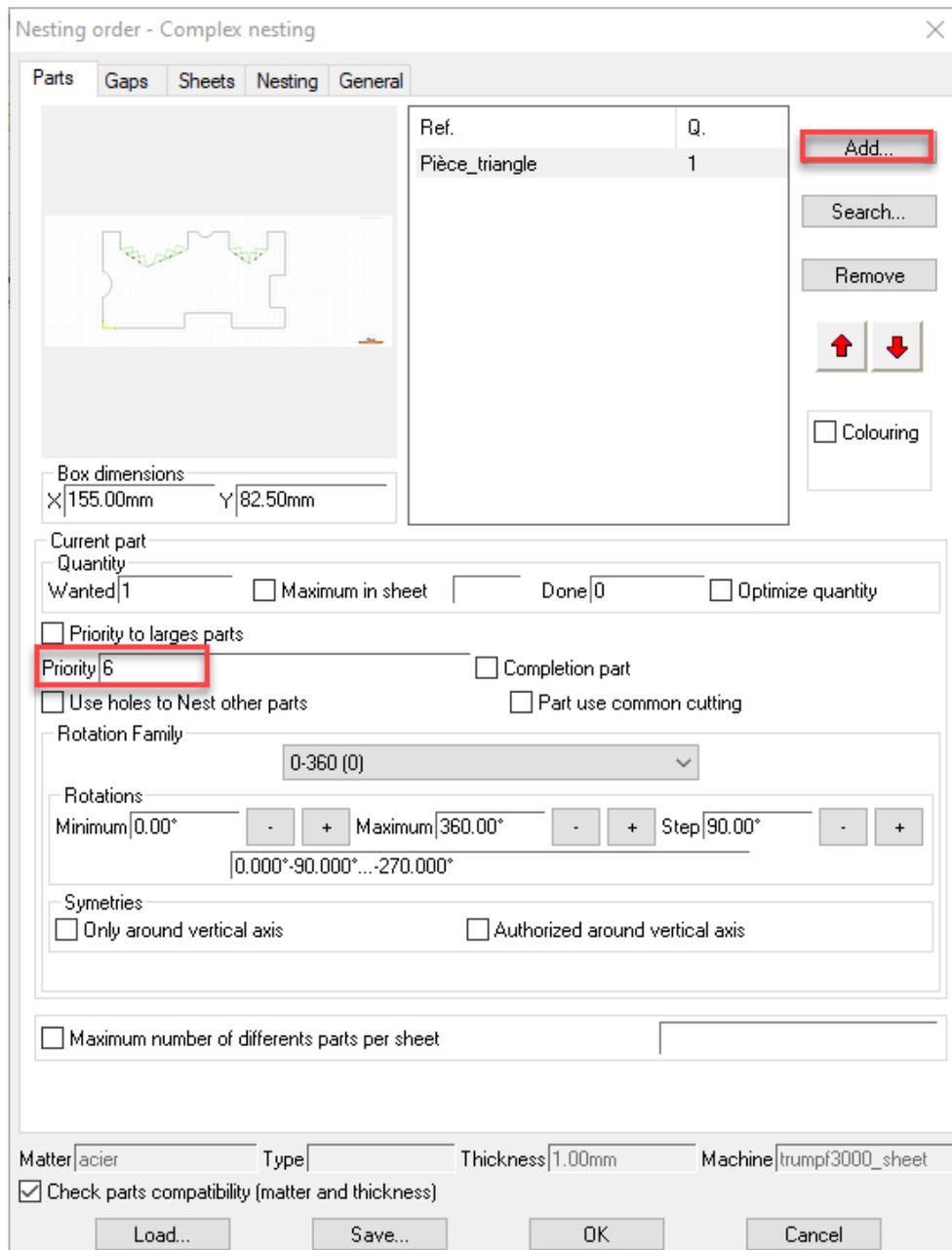


Nesting: Priority on the added parts

In the **Tools > Options > Nesting > Cluster nesting** command, the new **Default priority on new added parts** option allows you to configure a default value for the nesting priority of the new added parts.



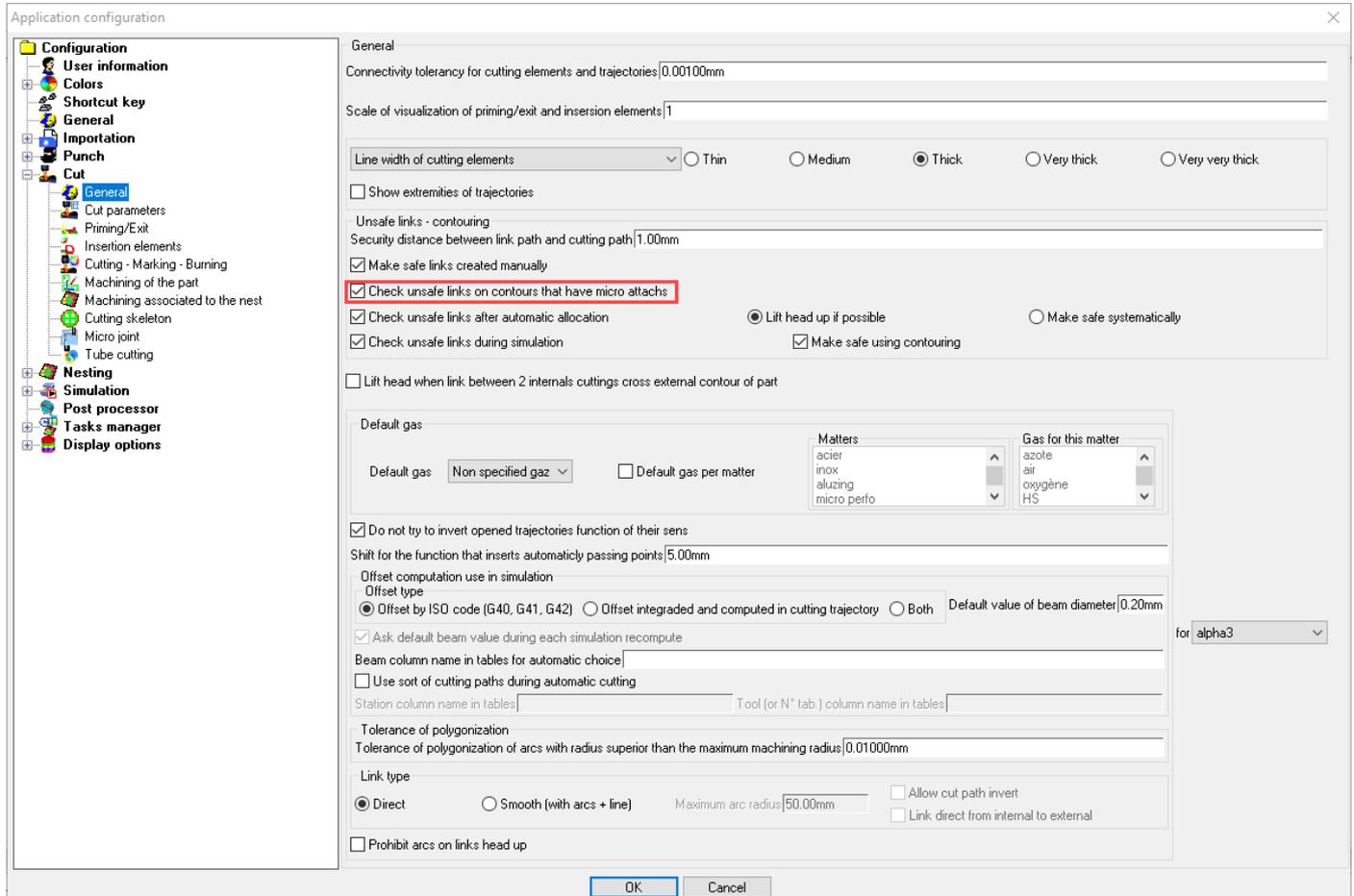
This option applies to the new parts added to a new nesting order and also to the new parts added to the Task Manager.



Unsafe link management

Checking the unsafe links on contours with micro-junctions

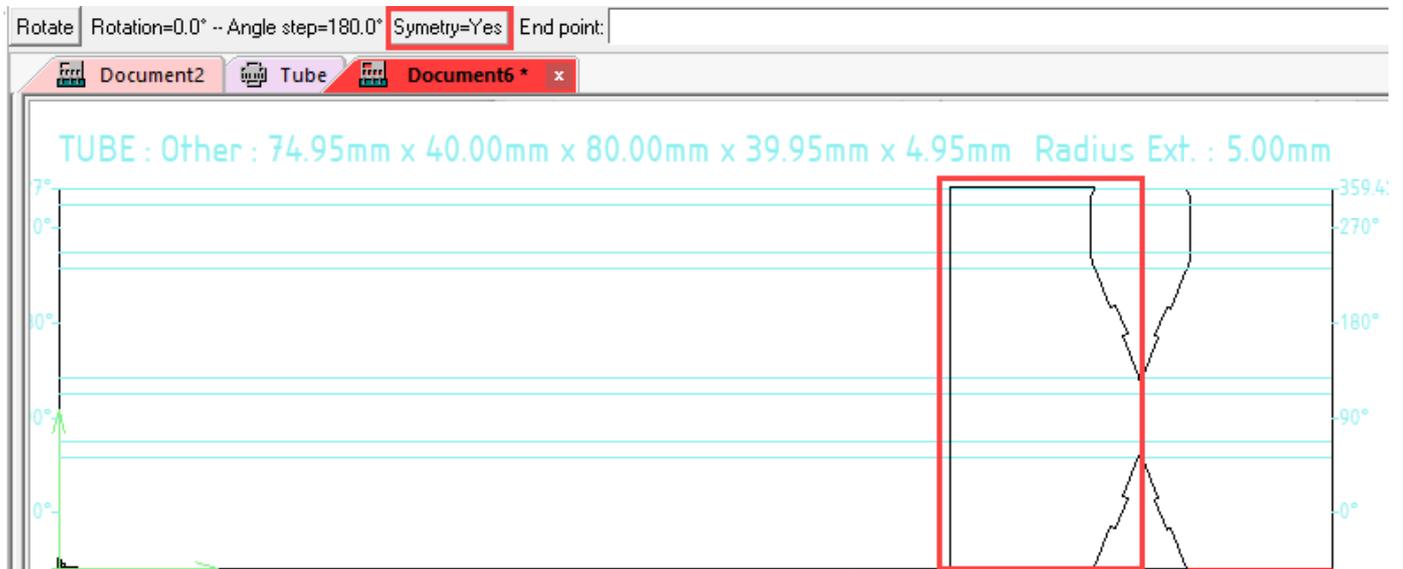
In the **Tools > Options > Cut > General** command, the new **Check unsafe links on contours that have micro-attaches** box allows you to consider the links as dangerous if they cross a contour with micro-junctions.



Manual Nesting

Tube symmetry

When performing a manual nesting, the **Insert** command allows you to perform the symmetry of a tube.



Rotation of parts using the keyboard

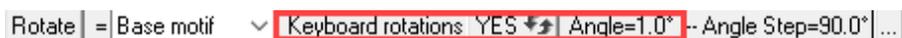
In order to facilitate the manipulation of the parts during a manual nesting, keyboard shortcuts can now be configured and used directly in the  **Move parts** and  **Insert parts** commands.

A new **Keyboard rotations** button is available in the question bar:



When the part is located at the mouse cursor:

- Pressing the **+** key on the keyboard allows the part to be rotated by 1° and the question bar takes the rotation value into account.



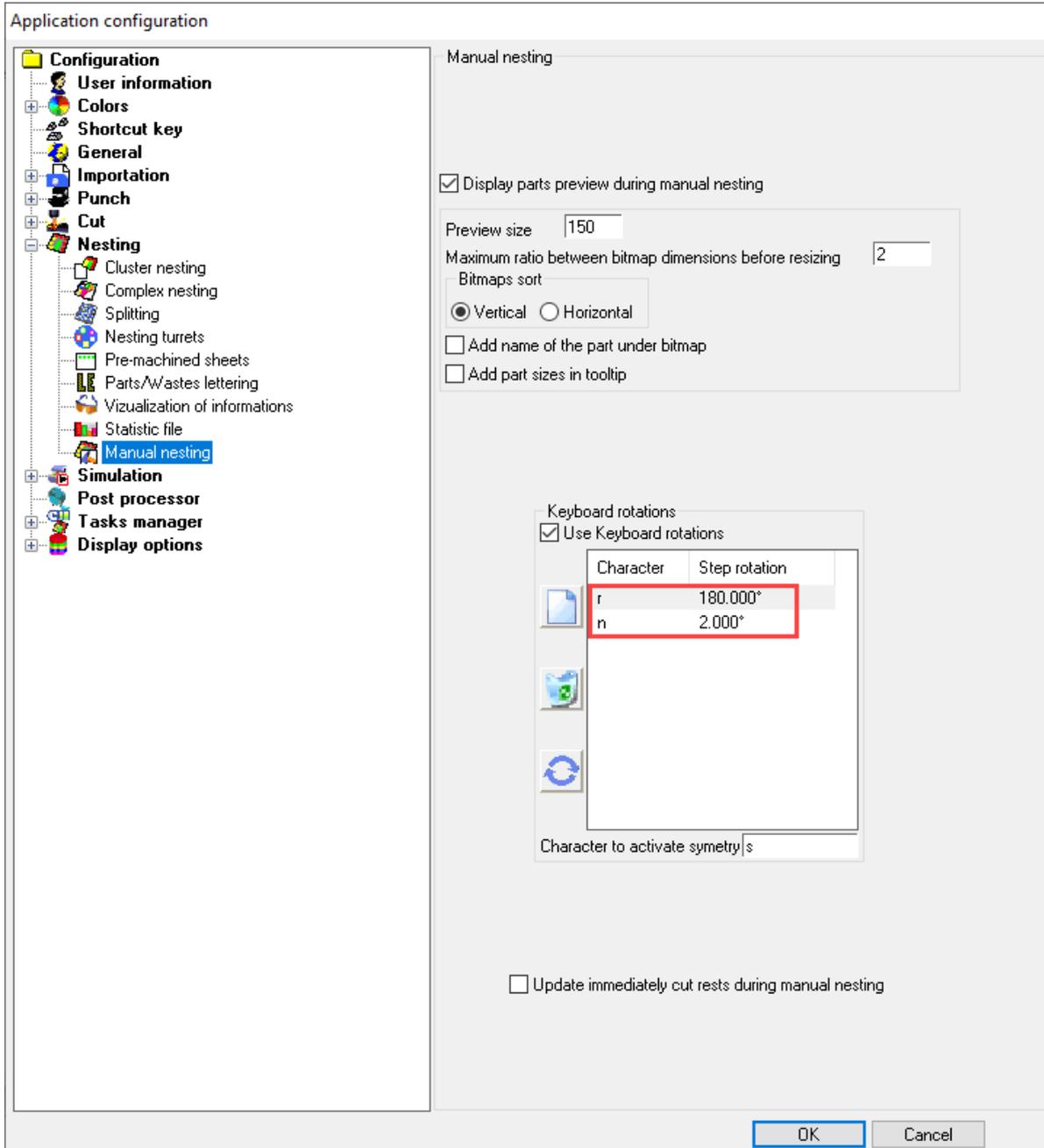
- Pressing the **/** key on the keyboard allows the part to be rotated by 45° in addition to its current rotation and the question bar takes the angular value into account.



- Pressing the **S** key on the keyboard allows you to create the symmetry of the part.

To disable the keyboard shortcuts and return to the classic input fields, the **Keyboard rotations** button must be adjusted to **NO**. The status of this button is kept during the **TopSolid** session.

Adjustments can be made in the **Tools > Options > Nesting > Manual nesting** command.



The  **New** button allows you to add a new **character** associated with a **rotation step**. The **n** character and a rotation step of **2°** are displayed by default.

To modify the character, you only have to double-click on the **n** character (only one character is allowed). The tab key allows you to confirm the modification.

To change the value of the rotation step, you simply have to double-click on it. Negative values are allowed, but 0° is not.

The  **Reload by default** button allows you to reload default values.

The **s** character allows you to create a symmetry of the part.

Operator Cards

Surface display

New keywords have been added to the operator cards to display the sheet surface.

For the **nesting operator card**, please refer to the template in *C:\Missler\v622\local\englishUS\opcard_full.ex.dft*. The new word is **#sheet_surface#**.

If the **#sheet_surface#** keyword is used alone, the default unit will be **m²**. To modify the unit, you have to create a parameter named **sheet_surface** and change the unit via the **Parameter > Edit list** command.

For the **order operator card**, please refer to the template in *C:\Missler\v622\local\englishUS\orderopcard_full.ex.dft*. The new words are the following:

- #gross_surface_matter#**
- #useful_surface_matter#**

Attributes on the part index texts

In the **Tools > Options > Simulation > Operator cards** command, new options allow you to display the texts of the part indexes in framed or underlined mode.

