TopSolid

TopSolid 2015

TopSolid v6.16

What's New

MASTER YOUR MANUFACTURING PROCESS

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What's new in TopSolid'Wood v6.16

This section describes the new features in the version **6.16** of **TopSolid'Wood**.

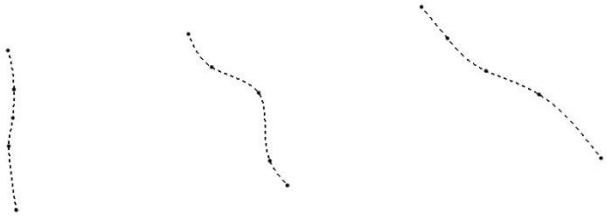
Design

Curves and sketches

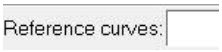
Curve morphing

A curve morphing function is now available in **Curve** > **Other curves** > **Morphing**, and allows you to automatically create a series of curves from several reference curves.

- After creating several reference curves, select **Curve** > **Other curves** > **Morphing**. 



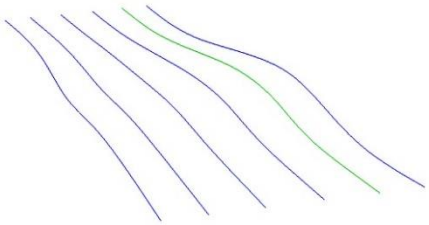
- Select the reference curves/sketches (at least 2 curves).



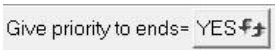
Note: It is then possible to add or remove reference curves.

- Set the **Morphing** options. 

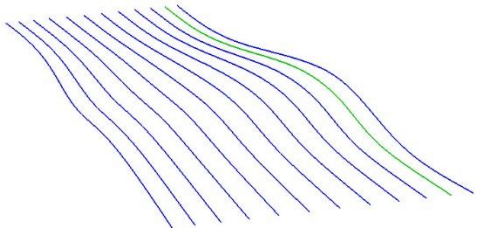
Give priority to ends: When activated, this option keeps the curves located at the ends of the morphing when adding or removing a reference curve.
For example, on the morphing here, the green resulting curve shows the difference between having this option enabled or disabled.

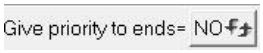


Once the number of curves is modified, the result is the following:

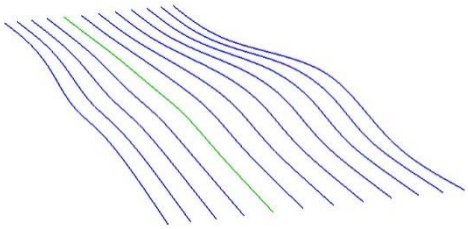


The green curve is still the second-to-last curve of the morphing.

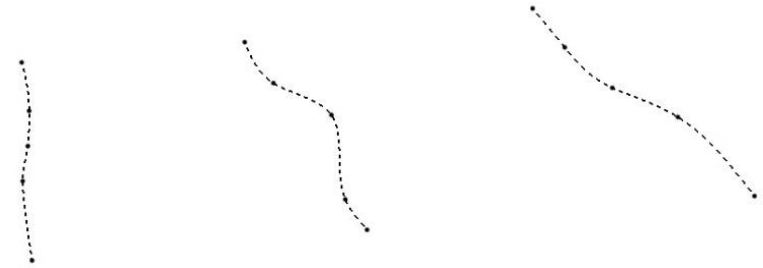


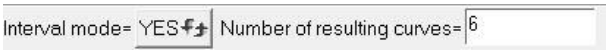


The green curve has not kept its position.

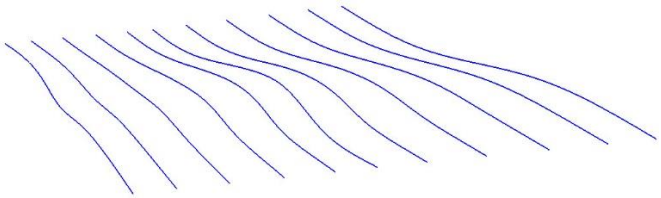


Interval mode: Specifies how the number of curves is calculated, either between each reference curve when the mode is enabled, or between the two curves at both ends.
For example, a morphing created from 3 reference curves:

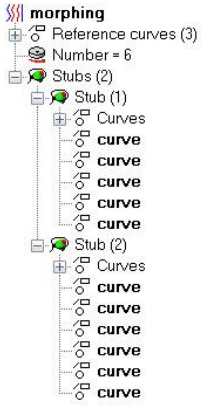


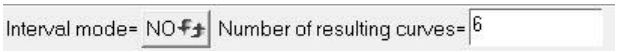


6 curves are created between each reference curve.

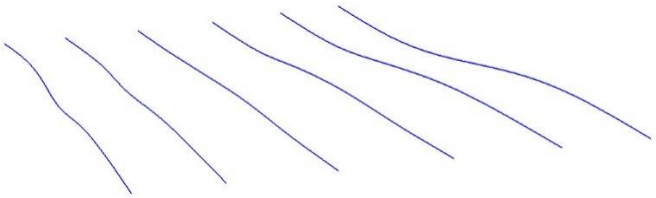


In the tree, a **Stubs** node appears under the **Morphing** section, indicating which curves delimit the stubs and resulting curves.

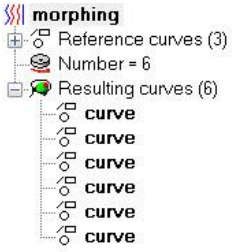




6 curves are generated between both ends.



In the tree, a **Resulting curves** node contains the curves generated by the morphing.



- Set the **number of resulting curves**.

Once the morphing is created, it is possible to modify the number of resulting curves, or add/remove reference curves.

- Edit the morphing from the tree.
- To add a reference curve, **right-click** on **Reference curves** > **Insert**, and click the curve(s) to be inserted.
- To remove a reference curve, right-click on the curve line and select **Extract**.
- To modify the number of resulting curves, double-click on the **Number** line.

Note: Once the morphing is created, the **Interval mode** and **Give priority to ends** options cannot be modified. They can only be accessed when creating the morphing.

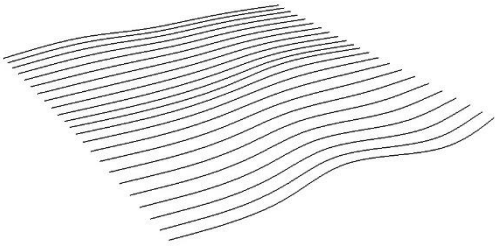
Connection

It is now possible to connect several curves together using the **Connection** command. This command is very useful for creating a tool path, and then applying a machining on the generated curves.

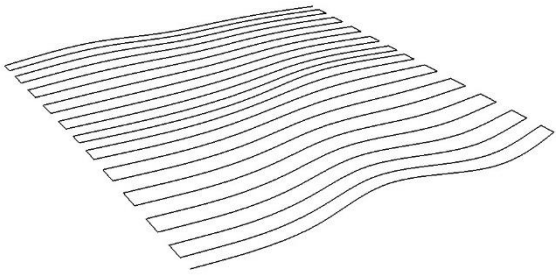
- Launch **Curve** > **Other curves** > **Connection** 
- Select the **reference coordinate system**. The best is to select the coordinate system on which curves have been drawn.
- Then click the curves to be connected and validate with **OK**.
- Set the options:
  - **Keep curves orientation**: Used to keep the orientation of the selected curves when creating the connection. If the option is set to **NO**, the orientation of some curves will be modified during the connection.
  - **Link type**: Used to select the connection link type: **Line** or **Bspline**. 

The connections created between the curves are lines.

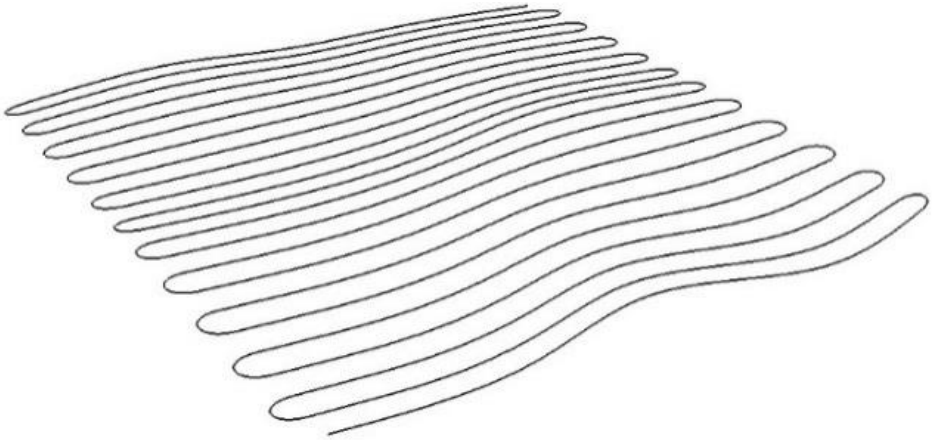
Origin curves:



After connection:



The Bspline mode creates curves to connect the selected curves. A loop length is to be entered; this value corresponds to the curve's offset value. If this value is 0mm, new lines will be generated. It is also possible to use a Z minimum to prevent the connection from falling below a certain value.



Note: All these values can be modified after the creation of the connection. To do this, use the **Modify** function on one of the connections.

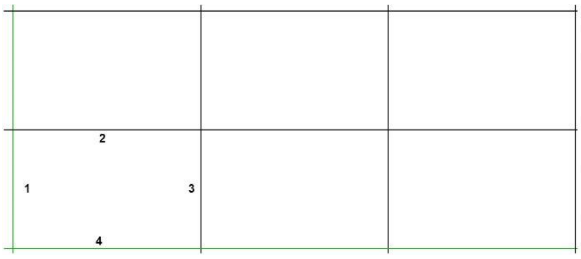
- Once the connections are created, operations can be applied to these curves (groove, user machining…).

Cell

When four open curves intersect each other, the new **Cell** function allows you to create a closed curve automatically.

- After drawing open curves, select **Curve** > **Other curves** > **Cell**.

Here are two green lines that are repeated on X and Y in order to generate a grid.



- Select the four curves defining the cell in the order given above.

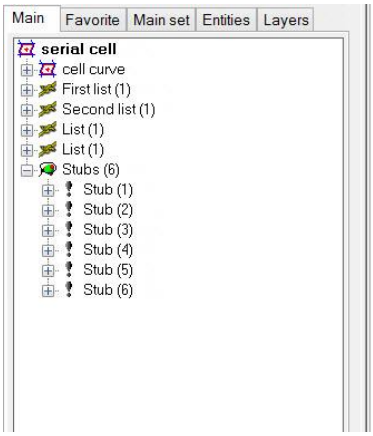
The **Projected mode** can be used to project the generated cell onto a different coordinate system.

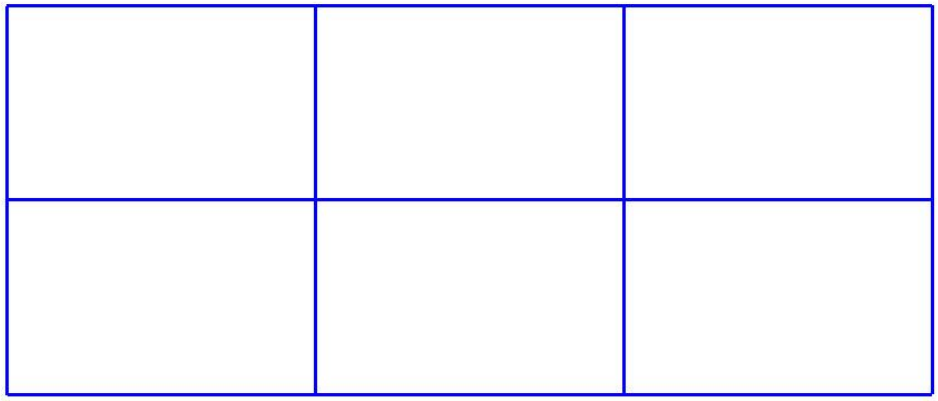
After selecting the fourth curve, the cell is automatically created.

- It is possible to stop the function or make a **serial copy** of this cell.

- If the previously selected curves have been generated through **repetition**, the **Automatic** button automatically selects the curves in order.

- In this example, the **serial copy** mode to be used is **Interval**. Then select **Automatic** and the cells will be repeated (blue curves). The tree displays the curves used to create the cells, as well as the generated **stubs** (cells).





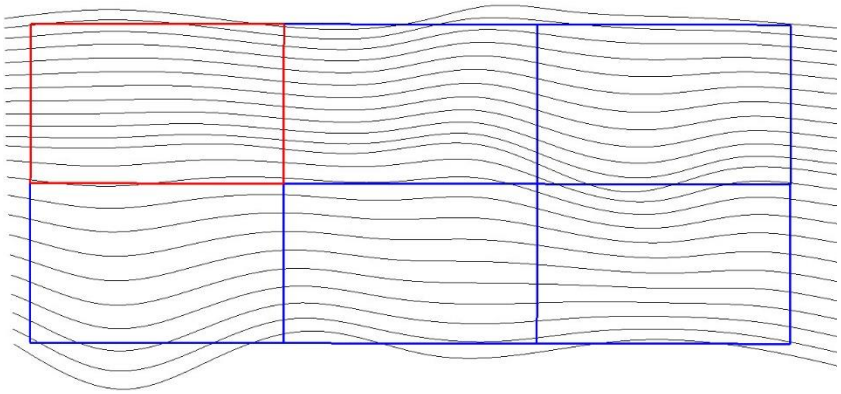
These curves can then be used to divide a part into several panels.

Trimming

It is now possible to trim a batch of curves by a closed curve using the **Trimming** command.

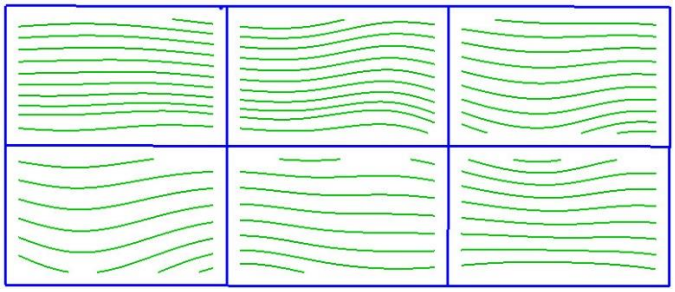
- Launch **Curve** > **Other curves** > **Trimming**. 
- Select the trimming curve.
- Select the curves to be trimmed.

In the example below, the black curves will be trimmed by the six rectangular curves.



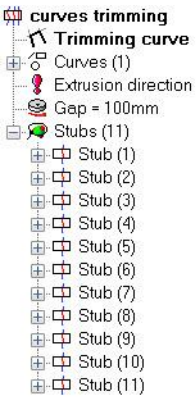
- If necessary, enter a gap between the trimming curve and the trimmed curves.

Note: It is not required to specify an extrusion direction; this option is only useful if the extrusion direction is not perpendicular to the trimming curve's plane.
Still using the same example, the curves generated by the trimmings are shown in green. A gap has also been added.



Once the trimming is created, some parameters can be modified.

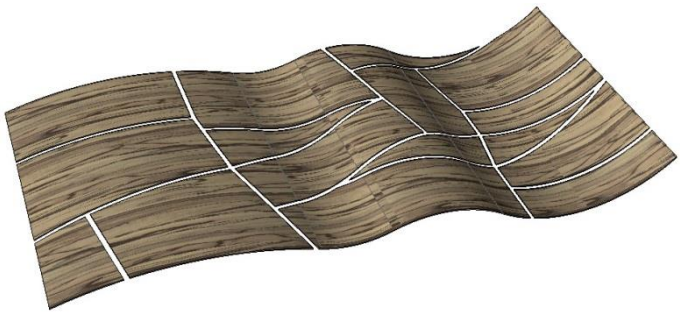
- Edit one of the resulting curves from the tree.
- To add a curve to be trimmed, **right-click** > **Insert** on **Curves**.
- A double-click on the **Gap** line modifies the gap value.



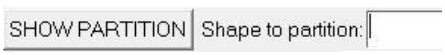
The **stubs** correspond to each trimmed curve.

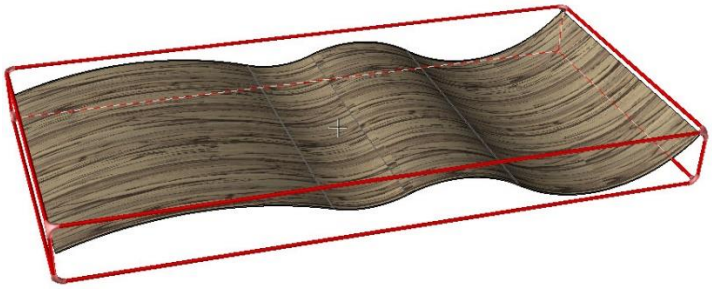
Part partition

The new **Partition** function command makes it possible, from an original part and closed curves, to automatically create the resulting parts.



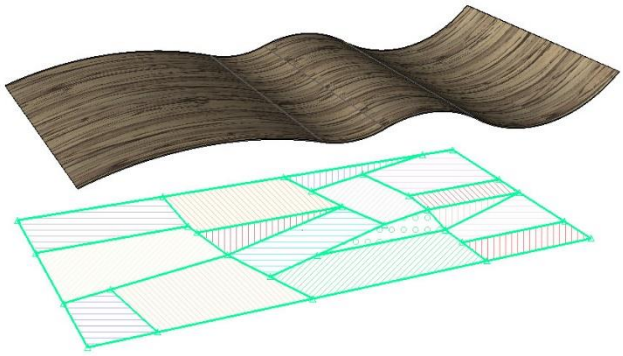
- Launch **Shape** > **Other shapes** > **Partition**. 
- Select the part for the partition.





- Select closed curves to partition the part.

Note: Here it is possible to select one or more sketches. The partition will be performed on the closed curves of sketches.

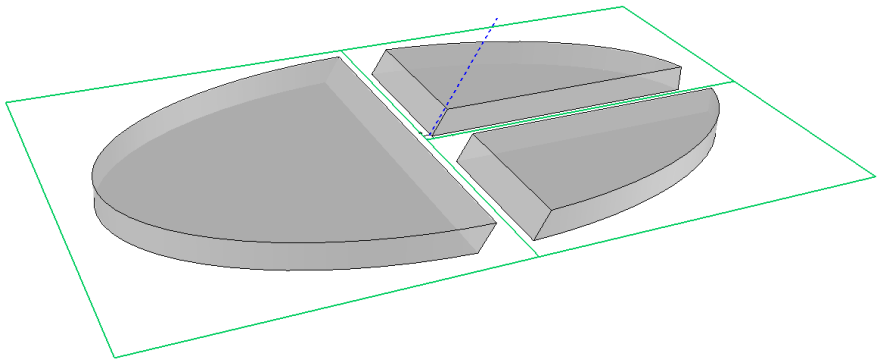


- Once all curves are selected, validate with **OK**.

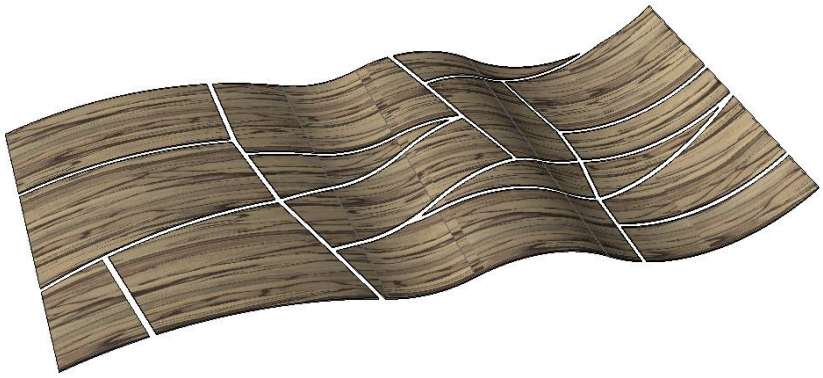
Note: It is then possible to add or remove curves from the partition.

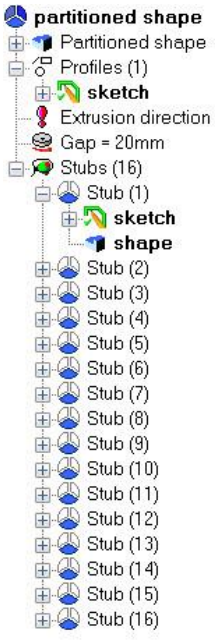
- Enter a **gap** between each partitioned part, and then validate with **OK**.

Note: As with the extrusion function, the **Extrusion direction** option modifies the partition direction of the original part. By default, the **extrusion direction** is perpendicular to the curves.



The partitioned shapes are then generated.





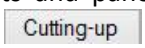
- Edit the partitioned shapes from the tree. The **Partitioned shape** operation then contains:
  - The partitioned original shape.
  - The list of curves used for the partition.

Note: It is possible to insert new curves by a **right-click** > **Insert**, and extract curves with a **right-click** > **Extract**.

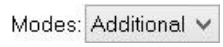
- The extrusion direction. If the default extrusion direction is used, no direction is then displayed here, but it is possible to reconnect it.
- The gap parameter between partitioned parts.
- The different parts generated as stubs, with the shape and the curve used to create it.

Define parts/sets

Default matter overvaluation mode management

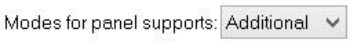
The matter overvaluation modes for parts and panels are now managed in **Tools** > **Options** > **TopSolid'Wood configuration** > **Define parts** > **Cutting-up**. 

- Set the overvaluation mode for the definition of standard parts.



Note: This overvaluation mode will only be used for new defined parts.

- Set the overvaluation mode for the definition of panel supports.



Note: This overvaluation mode will only be used for new supports defined when creating a panel on a non-defined part.

Machining length always highest dimension

A new option is available for machining and cut exports; it allows you to position the length axis of selected parts on their highest dimension without having to edit the part definition.

Note: This setting is only available in the WoodWop, NCHops, Xilog and PanelCAM interfaces and Ardis XML exports. In addition, the option is only visible in multi-part export.

- Start one of the compatible interfaces.
- Choose **Multi-parts**.
- Set the **Length always highest dimension** option to **YES**.

Note: This setting does not affect the part definition; it is considered during export on an occasional basis only.

Texture frame

When the **Add to sawing-up** option is unchecked during the part definition, a frame must now be selected to properly orient the texture.

- When defining a part, uncheck **Add to sawing-up**.
- Select an existing frame or create a new frame by clicking directly on a face or using the **Wizard** button.

Note: The frame is requested only if the part has grain orientation.

Define multiple parts

Various changes have been made to the **Define multiple parts** function to improve usability.

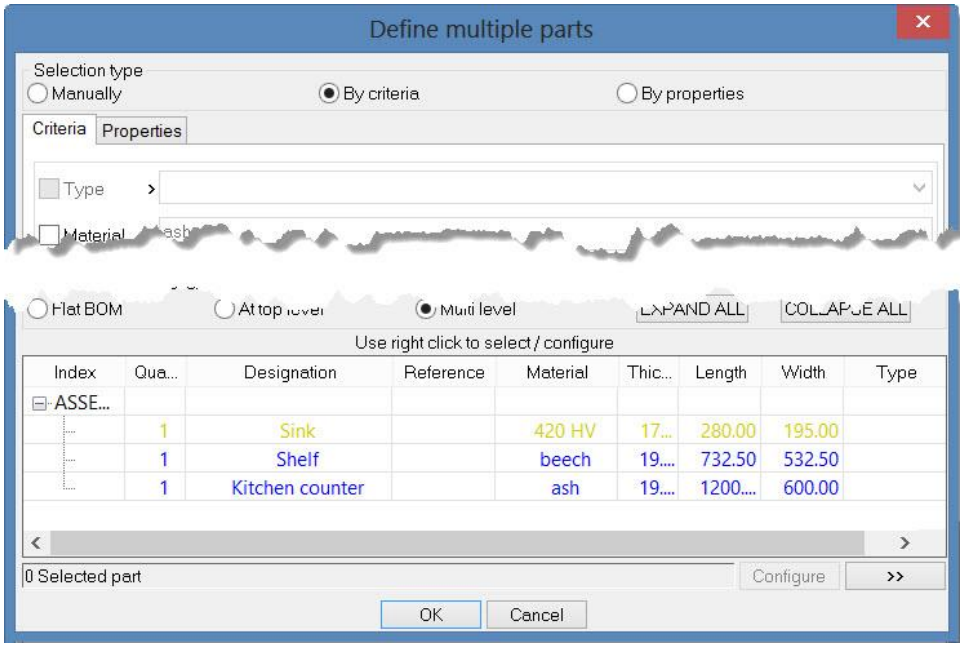
This includes the possibility to expand or collapse the configuration section by clicking the 

button. Material and texture modifications have also been enhanced.

Management of parts without sawing-up

Parts without sawing-up are no longer grayed out in the **Define multiple parts** window and can therefore be configured. Parts with no sawing-up are displayed in yellow in the list of parts.

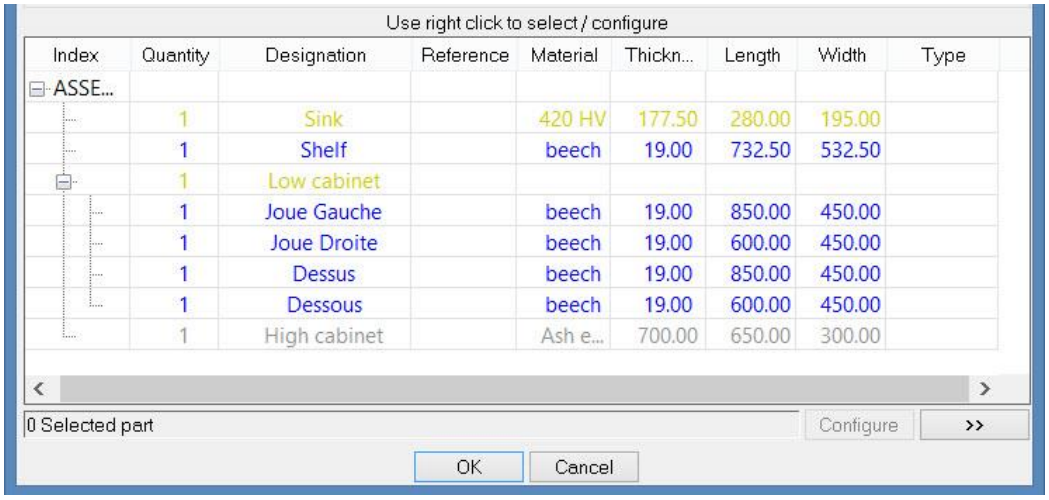
Note: To avoid any errors, it is not possible to select a part with sawing-up and a part without sawing-up simultaneously. In addition, parts without sawing-up cannot be selected by criteria or by properties; they can only be selected manually.



Management of assemblies and components

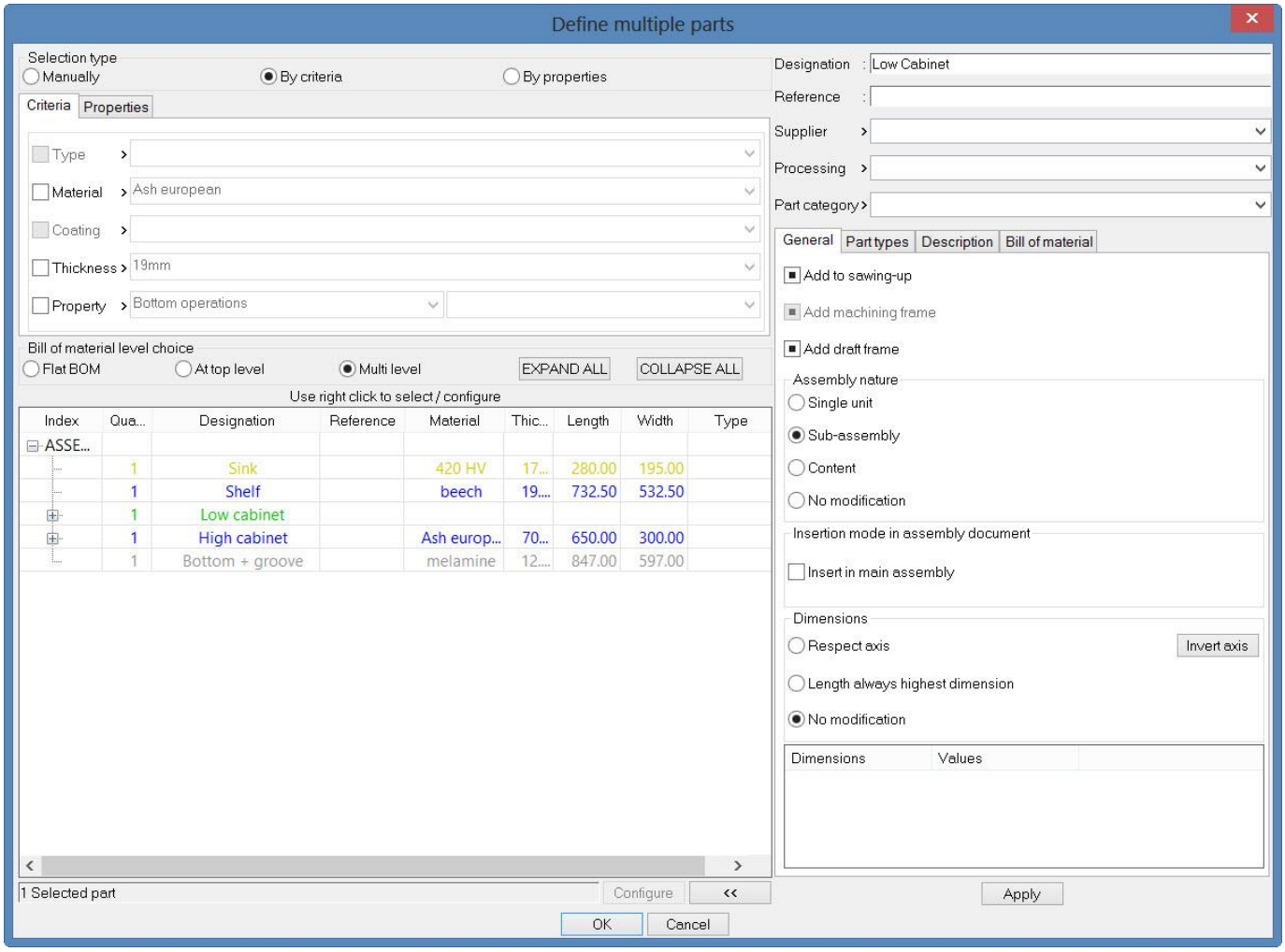
Assemblies and components can now be configured in the **Define multiple parts** window. Like for parts without sawing-up, they are displayed in yellow in the list of parts.

Note: Assemblies and components works the same way as parts without sawing-up; they cannot be selected at the same time as parts and cannot be selected by criteria or properties.
In the example below, the **Low cabinet** assembly is displayed in yellow in the list, just like the **Sink** part which has no sawing-up.



When one or more assemblies are selected, the **Define assembly** function's dialog box is displayed. Single units are not configurable and are therefore grayed out. To be able to configure them, they must be set to **Sub-assembly** or **Single unit**.

In the example below, the **Low cabinet** assembly is selected. The definition of the assembly is displayed on the right side of the window. The **Bottom + groove** assembly has been defined as **Single unit**, it is therefore grayed out.



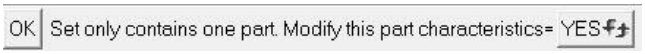
Elements that make up the single units can be displayed in the list of parts in order to be configured.

- Select **Tools** > **Options** > **TopSolid'Wood configuration** > **Configurators**. It is then possible to choose in which functions the constituents of the single units will appear.
- Check **Define multiple parts**.

Still in the same example, the part belonging to the **Bottom + groove** single unit appears in the list of parts of the **Define multiple parts** window and can be configured.

Define set

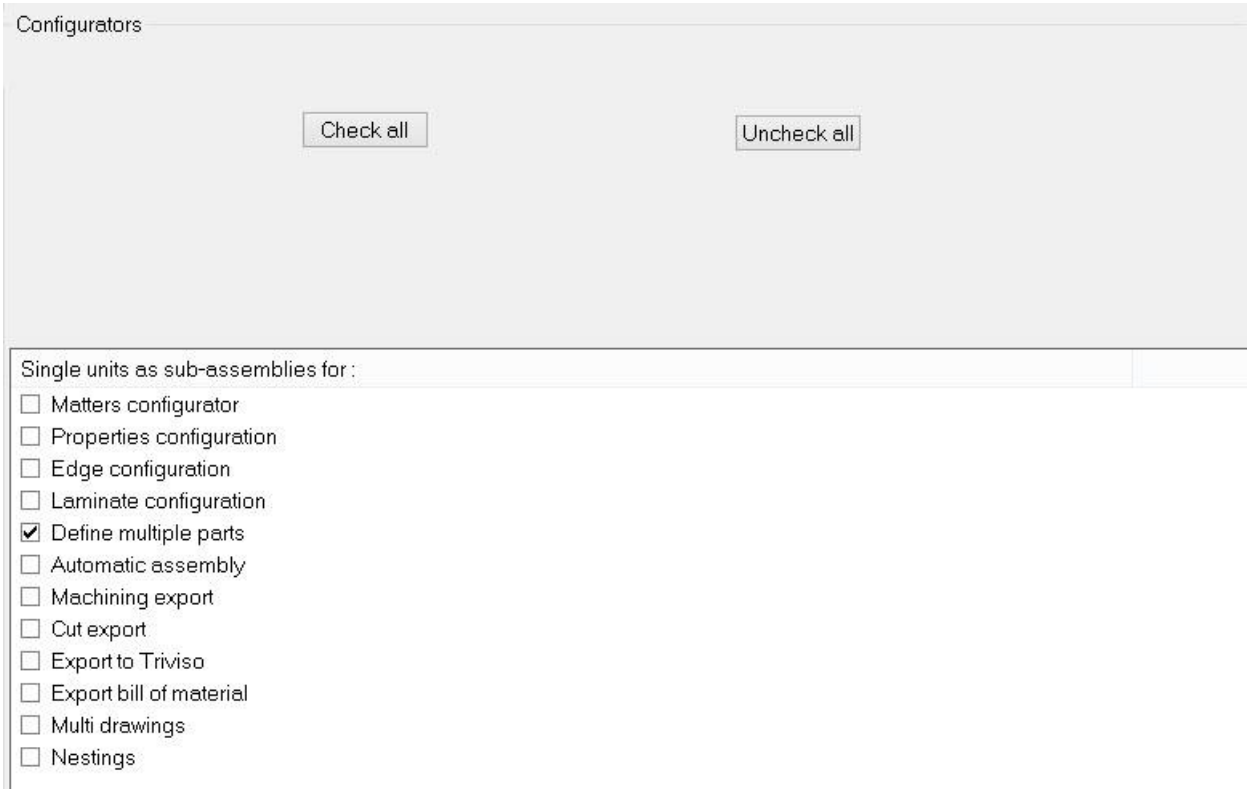
Now, when the **Define set** function is launched in a document with only one defined part, a question appears.



If the answer is set to **YES**, the **Define part** dialog box opens automatically and the definition of the part contained in the set is modified.

When a single unit is displayed in a part selection dialog box (**Define multiple parts**, **Configurators**…), it is possible to display its constituent part(s). This setting needs to be made for each function.

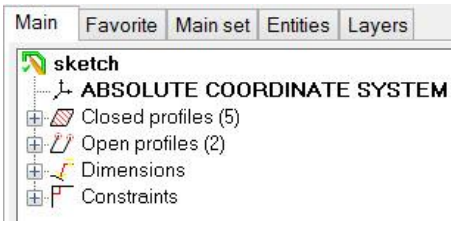
- Go to **Tools** > **Options** > **TopSolid'Wood configuration** > **Configurators**.
- Check the functions in which the constituents of the single units must be displayed.

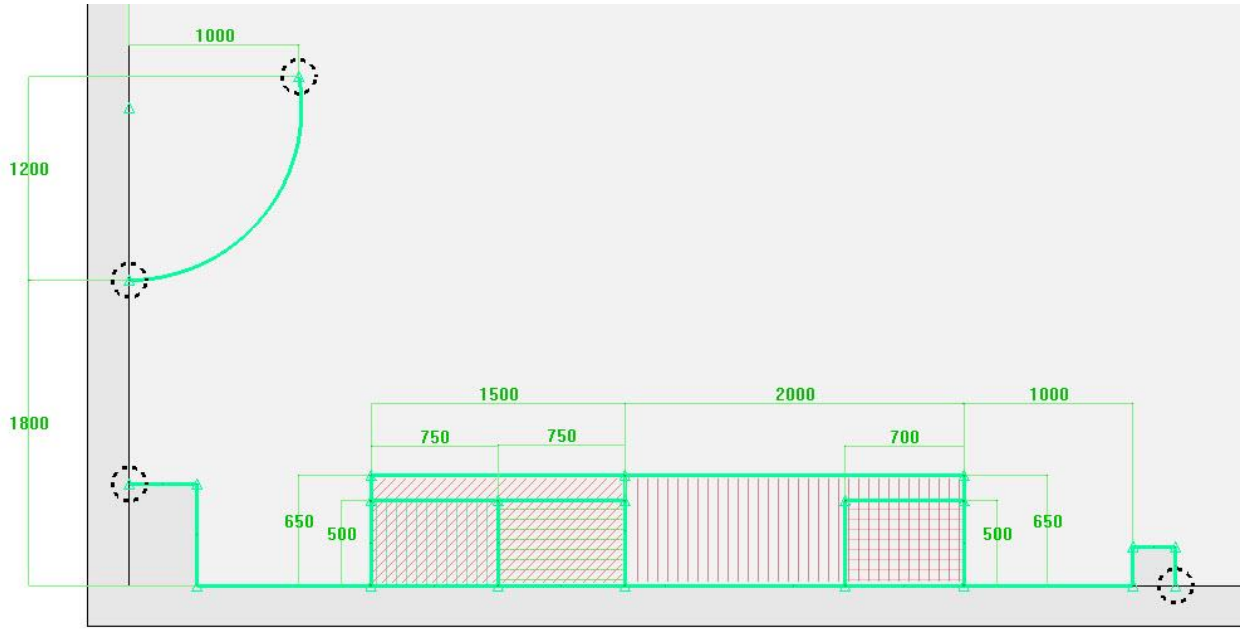


Improved sketch display

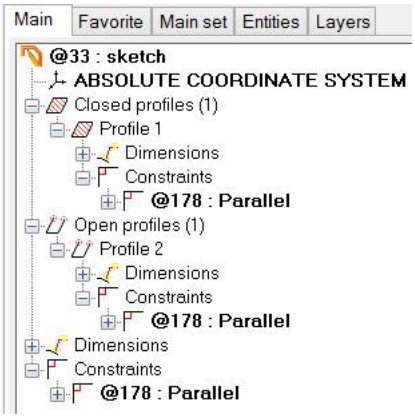
The display of sketches from the tree has been improved to make their comprehension easier.
The editing of a sketch now displays:

- The sketch **creation coordinate system**.
- **The closed curves** drawn in the sketch, with their associated dimensions and constraints.
- **The open curves** drawn in the sketch, with their associated dimensions and constraints.
- The sketch **dimensions**.
- The sketch **constraints**.





If a dimension or a constraint is placed between two different curves, the element is then displayed in both curves as well as in the list of sketch dimensions/constraints.



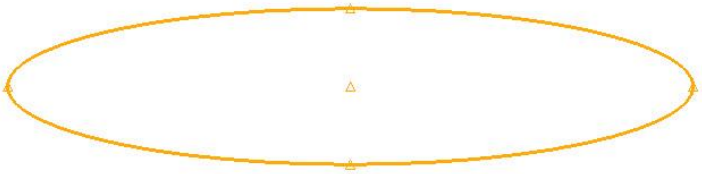
Ellipse in the sketch

Ellipses can now be created in a sketch.

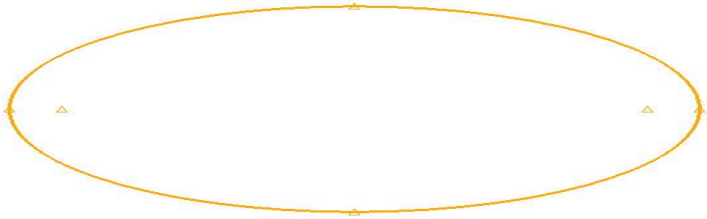
- Start a new sketch. 
- Select the **Ellipse** function. 

It is possible to draw two different types of ellipses:

- With a center point.

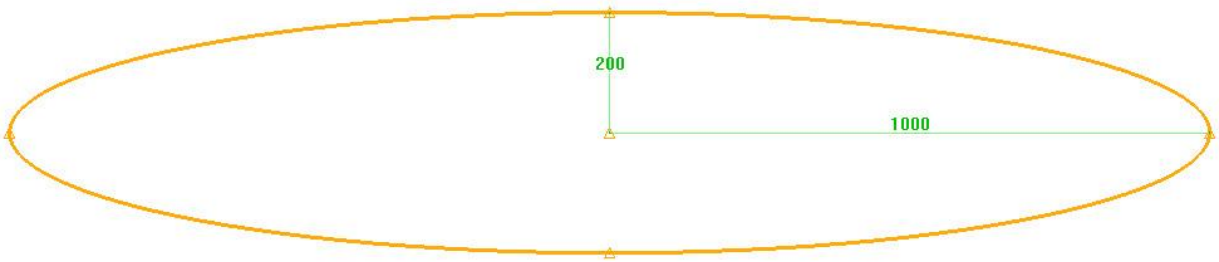


- With two focuses.



Ellipse with a center point

- Select the center point of the ellipse to be drawn.
  - Then select a point towards **X** and a point towards **Y** to generate the ellipse.
  - Set a **radius** or a **diameter** on **X**, and then on **Y** to generate a dimensioned ellipse.

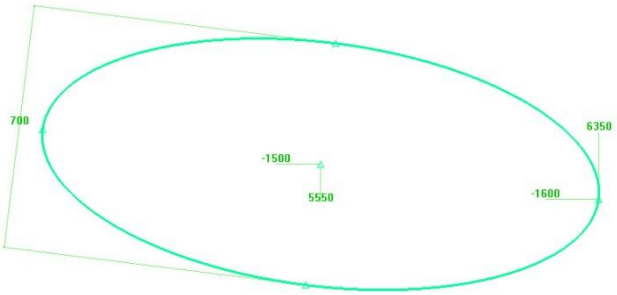


Ellipse with two focuses

- Select the **2 focus** option.
- Select the **first** and **second focus**.
- Then select a passing point to generate the ellipse.

The **Center point** or **2 focus** ellipses can be dimensioned by:

- Their center point or their two focuses.
- Their two control points on **X**.
- Their two control points on **Y**.



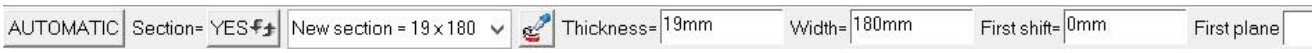
Constrained block

Given section

It is now possible to create constrained blocks using a given section. This section is then saved to be reused.

- Launch the **Constrained block** command. 
- Set **Section = YES**.
- Enter the section's thickness and width values.

The new created section appears in the drop-down list.



Note: The **Share thickness**  function allows you to retrieve the dimensions of an existing constrained block. Then, when the dimension of one of the two blocks will be modified, the other block will be automatically modified.

- Choose the planes of the constrained block and finish creating it.

The previously created section then appears in the drop-down list. It is possible to have several predefined sections.

Note: After a section has been created, it is automatically added to the list. To remove sections from the list, you need to open the *top.cfg* file, and then delete the lines corresponding to the sections. These lines begin with *D_SH_CBLOCK_SECTION*, followed by the section dimension in meters.

top.cfg

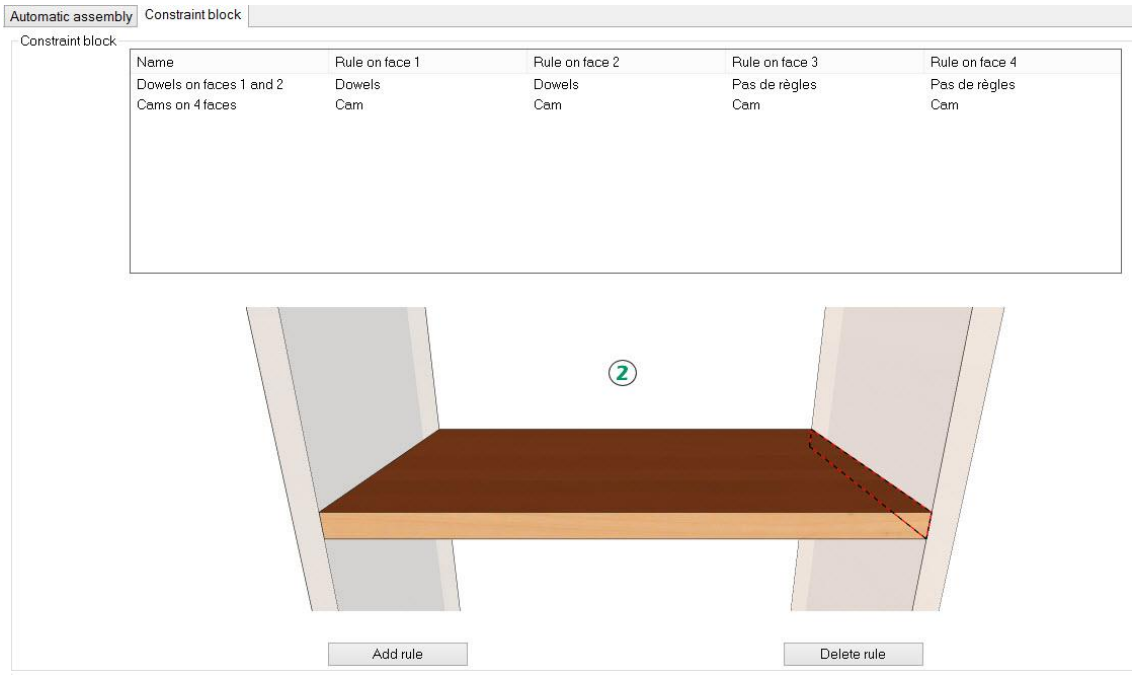
```
175  D_SH_BLOCK_CONSTRAINT_THICKNESS 0.019
176  D_SH_CBLOCK_SECTION 0.019    0.18
177  D_SH_CBLOCK_SECTION 0.08     0.08
178  D_SH_CBLOCK_SECTION 0.08     0.16
```

Automatic assembly

It is now possible to automatically assemble the constrained blocks using automatic assembly rules. Assemblies are created after completion of the function, after distribution.
It is possible to use predefined assembly rules or manually select an assembly to be applied.

- To use a predefined rule, assembly rules for constrained block must be declared first in **Tools** > **Options**.
- Go to **Tools** > **Options** > **TopSolid'Wood configuration** > **Automatic assembly**, and click the **Constrained block** tab.
- Create a new rule using the **Add rule** button.
- Rename the rule by double-clicking in the created line's **Name** column.
- Then define the rule to be applied to each face of the constrained block. The rules available in the drop-down lists of each face are existing automatic assembly rules.

Note: An animated image allows the four faces to be identified.



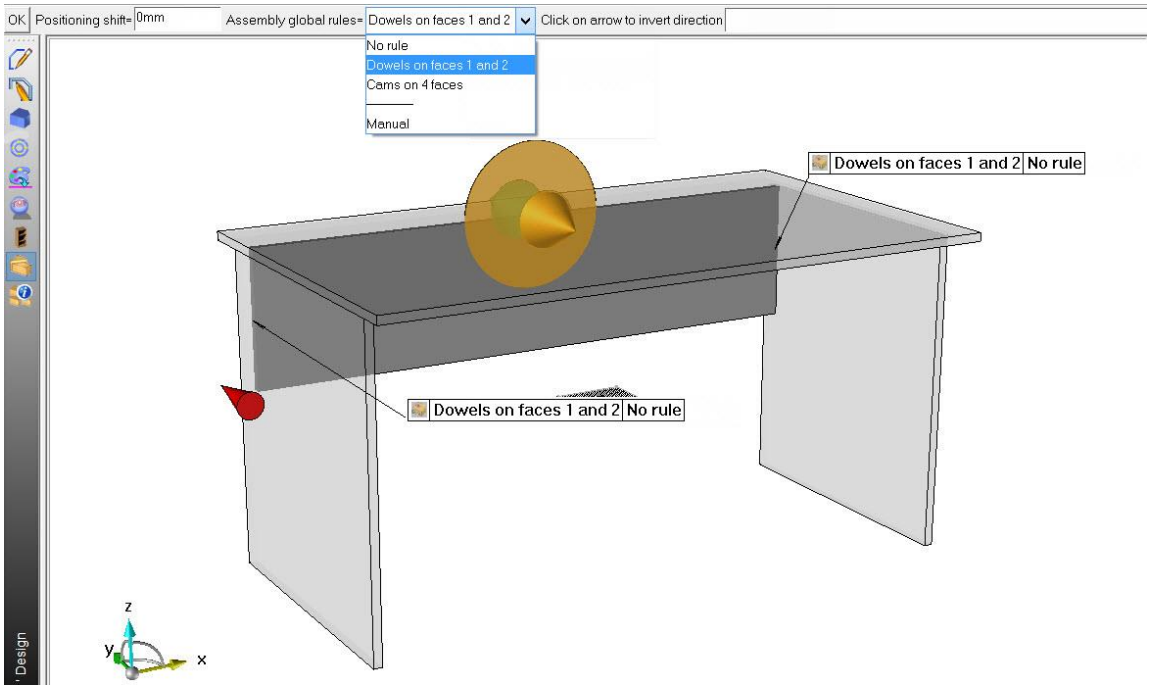
Once in the assembly, these rules can be used after completion of the **Constrained block** function. These assembly functions are mainly used for in-place design.

- Create a **constrained block**.

Note: When creating the constrained block, the **Invert direction** button is now available in the second question.

- On completion of the function, choose the rule to be used in the drop-down list.

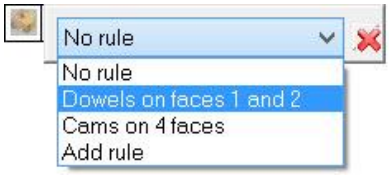
Labels appear on the faces used by the selected rule.



Note: These labels consist of two areas:

- The first area corresponds to the general rule applied to all faces. The assembly rules for constrained block are available.
- The second area allows you to modify the applied rule locally; only the face identified by the label will be modified. Only the automatic assembly rules are available.

A double-click on one of these areas opens a drop-down list to modify the rule used. To close the drop-down list, click the red cross icon  .



It is also possible to manually select the rules to apply to each face.

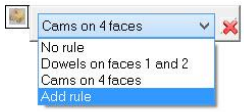
- Select **Manual** in the drop-down list
- Define the rule to be applied to each face.

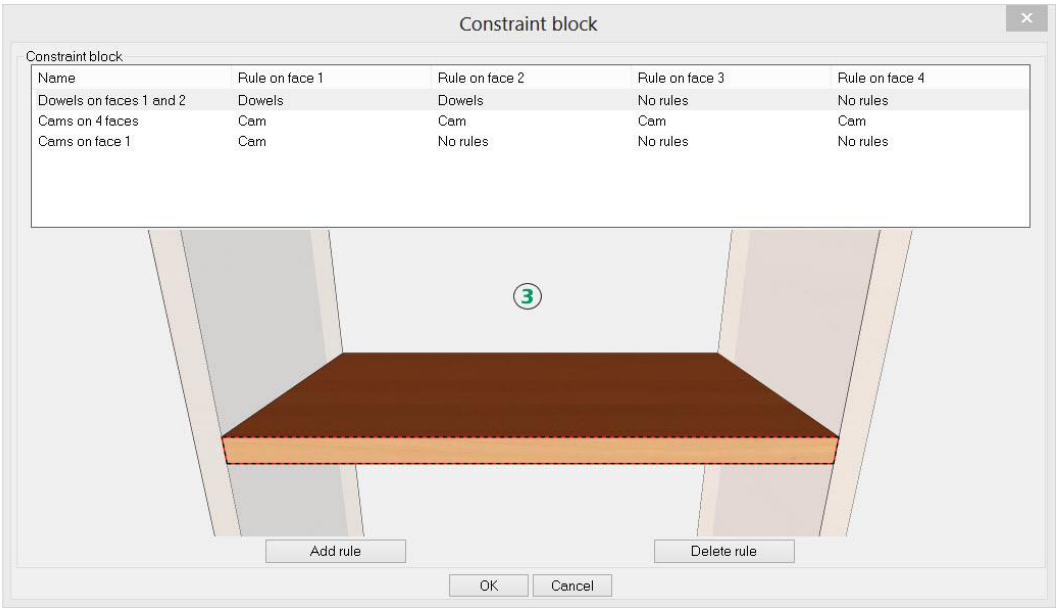
Rules can be created directly from labels.

- Choose a rule in the drop-down list at the end of the function.
- Double-click in the first area of one of the labels.
- Select **Add rule**.

A creation dialog box opens. This dialog box works similarly to the rule creation dialog box in **Tools** > **Options**.

Note: The rule will be added to those created in **Tools** > **Options** > **TopSolid'Wood configuration** > **Automatic assembly** > **Constrained block**.

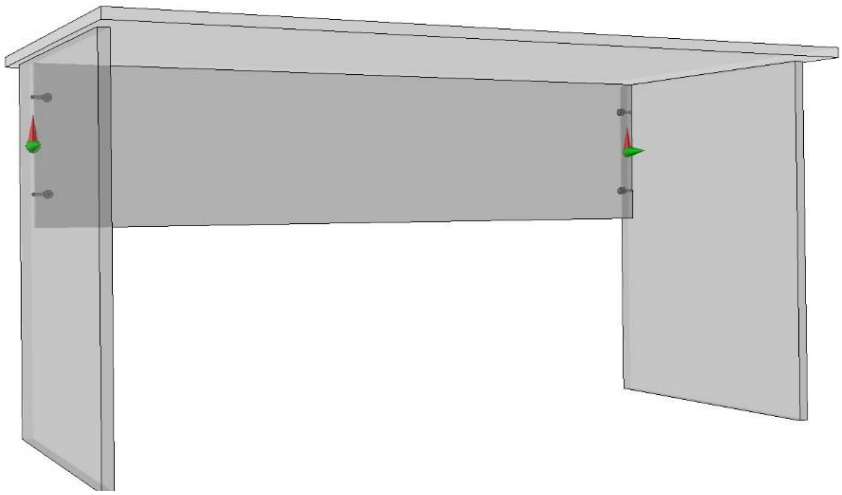




- Once the rules are selected, validate with **OK**.

Arrows appear, allowing you to change the direction of components. They work the same way as those displayed in the **Automatic assembly** function.

- Finish creating the components with **OK**.



Note: After validating the creation of the constrained block and the components, the created part is automatically defined in order to perform the component processes. If parts in contact with the components are not defined, no process is created. However, the machining will be automatically generated after the part definition.

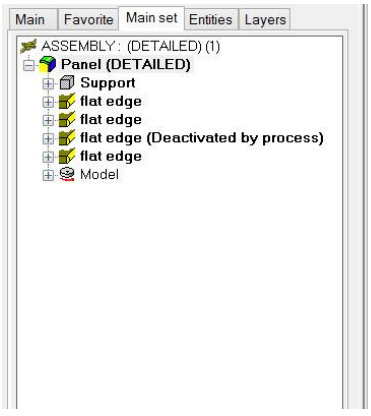
Automatic deactivation of cut parts

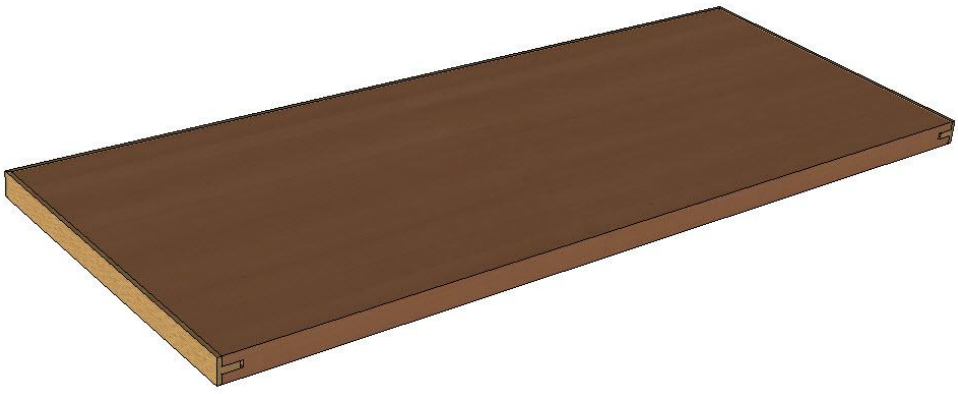
If after operating a part (trimming, subtraction, pocket, etc.) there is no material left, this part is now automatically deactivated. It will no longer be visible when projecting it into draft, bill of material, and in higher level sub-assemblies.

For example, a panel with four edges on which a sawing operation must be performed (according to the black line). In this case, the edge does not intersect the curve, but it must be disabled during the operation.

Note: This applies to **serial sets** and **panels**.

- Once the operation is completed, the edge is deactivated and the **Deactivated by process** information appears on the edge line.





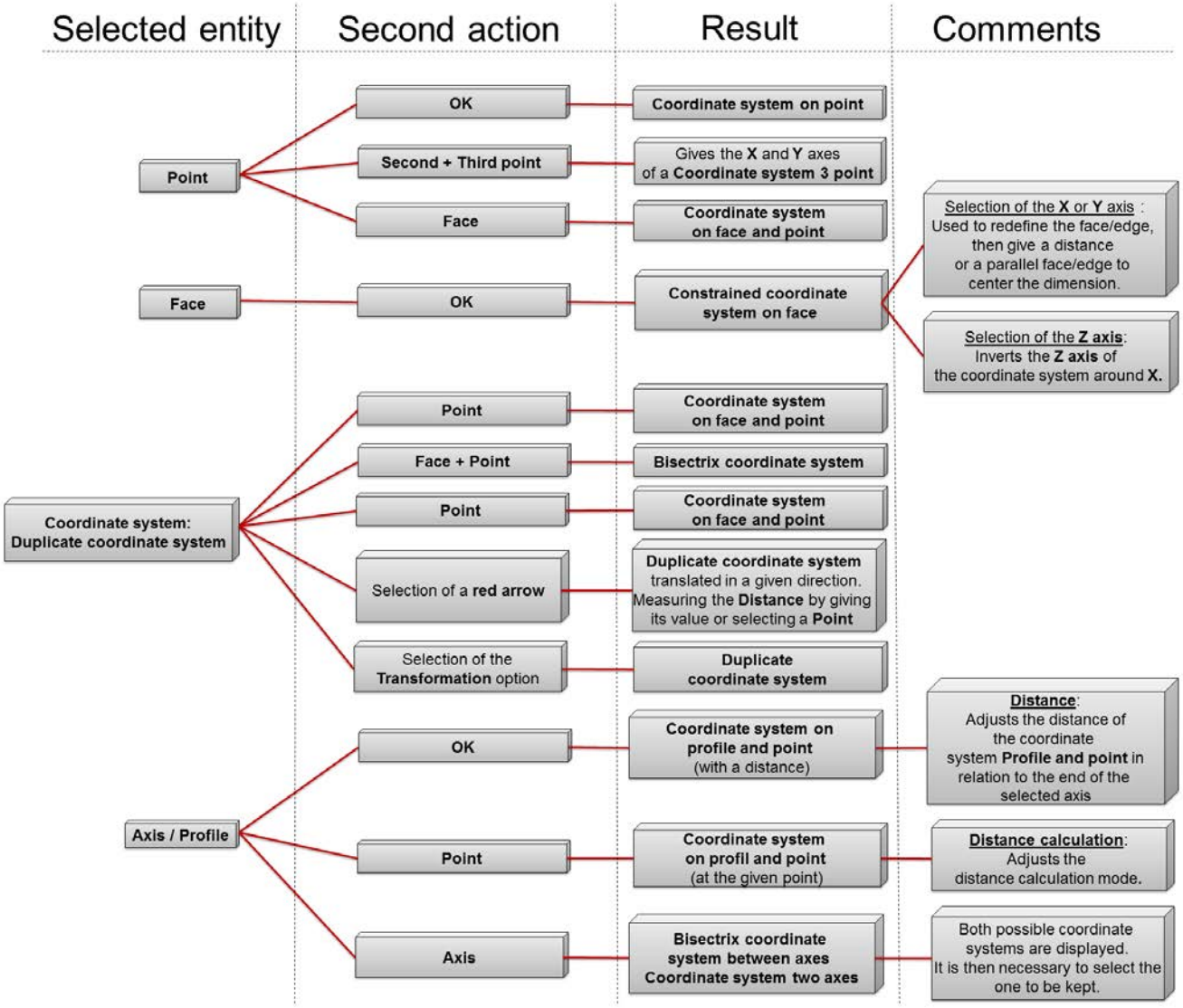
Other CAD improvements

- Now, when removing an operation, you are asked whether tools are to be deleted, made visible or hidden.
- When creating an extrusion, several curves can be selected one after the other without using the **Selection** function.
- The **Move** function can be used on a relative point; the coordinate system on which the point was created is automatically activated.
- Using the **Modify** function on a point modifies its position on the X,Y and Z axes thanks to the **+** and **–** buttons.
- A text can now be used as a reference for a tabulated parameter.
- A **distribution** can now be defined as a driver by **right-clicking** on the distribution line > **Define driver**.
- A stop can now be turned into a **main stop** with a **right-click** > **Main stop**.

Coordinate system creation wizard

The new coordinate system wizard allows you to directly select the elements to be used to create the coordinate system without having to select the type of coordinate system first.
Based on the selected elements, the appropriate coordinate system is then created. If the selected elements allow you to create two different coordinate systems, you will need to choose the coordinate system to be kept.

- Start **Tools** > **Coordinate system**. 
- Set **Wizard = Yes**.



Assemblies

Improved assembly functions

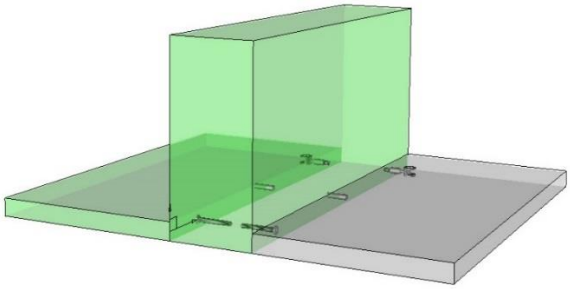
The **Dowel assembly**, **Screw**, **Len** and **Assembly kit** functions now use the automatic processes of components.
For example, this allows you to assemble more than two parts, while ensuring that the processes will be performed.
Before, these functions could only operate two parts.

Non-symmetrical kit

The assembly kit definition function has been improved. This now makes it possible to define non-symmetrical assembly kits. This feature also manages the notion of thickness which was necessary for screws.
This new definition of the assembly kit can contain two distinct portions: the main portion and the opposite portion. Each portion includes a set giving the elements to be included, a key coordinate system and a set of tools.

- In a new document, draw three parts in contact, representing the two insertion possibilities.
- Next assemble them using standard components.

In this example, the two green parts correspond to the parts to be assembled. The kit includes a screw, a dowel and a cam. As the kit is not symmetrical, the gray part will be used to define the opposite portion; this opposite portion must also be assembled by components.

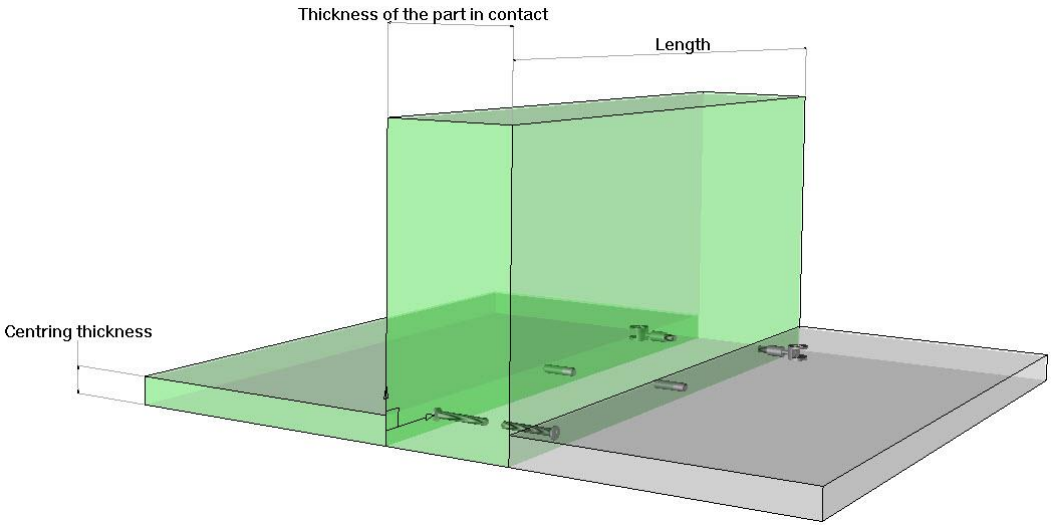


- Select **Wood** > **Define** > **Define assembly kit**. 
- Choose **Advanced assembly kit**.
- Select the **key coordinate system**.

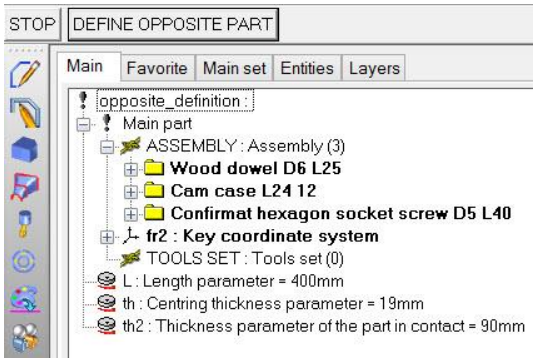
In the example, the key coordinate system is the coordinate system that is centered on the part thickness.

- Select the **length parameter**, the **centring thickness parameter** and the **thickness parameter of the part in contact**.

These three parameters are shown in the following example.



Once all parameters are provided, the first portion of the kit is created. Elements that make up the assembly kit appear in the construction tree. The **Tools set** node contains elements only if tools have been defined in the document.
In this example, tools are inherited from components and therefore do not appear in the list.

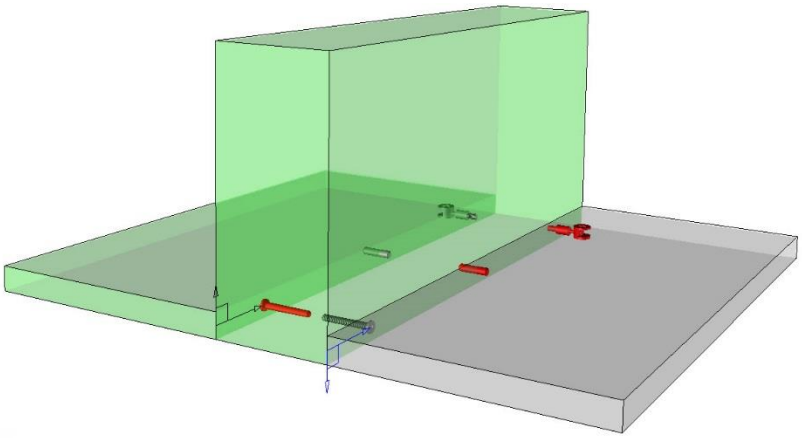


- If the kit is symmetrical, choose **STOP**. Otherwise, choose **Define opposite part**.
- Select the key coordinate system of the opposite portion.

In the example, the blue coordinate system has been created.

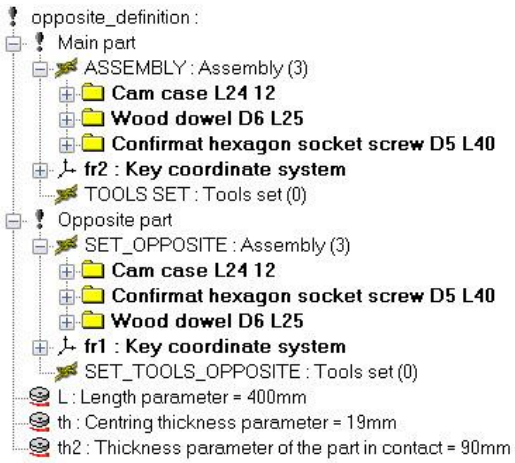
Note: The key coordinate system of the opposite definition must be rotated by 180° along the X axis.

- Select the components used in the opposite portion.



The selected elements and the key coordinate system are then visible in the construction tree under **Opposite part**.

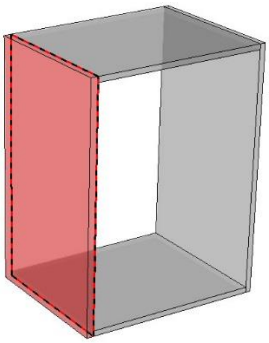
Note: Elements can be added to the kit. To do this, **right-click** > **Insert** on the **Set** line of the main or opposite part.

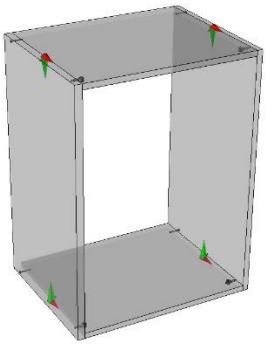


- Once completed, save the assembly kit in a standard library.

In an assembly, these kits can be inserted using the **Include standard**, **Automatic assembly** (if a rule using the kit has been created) or **Assembly kit** commands.

- Next insert the assembly kit. As in previous versions, select a start face, and then define the centering direction.
- Click all the faces on which to insert the kit.



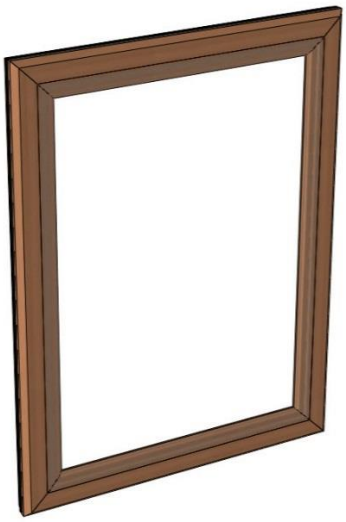


Management of multi-body extruded components

Multi-body extruded components are now managed when creating miter or planar cuts. These extruded components are created and used in the same way as a simple extruded component.

- Insert several shapes in the assembly when creating the component.

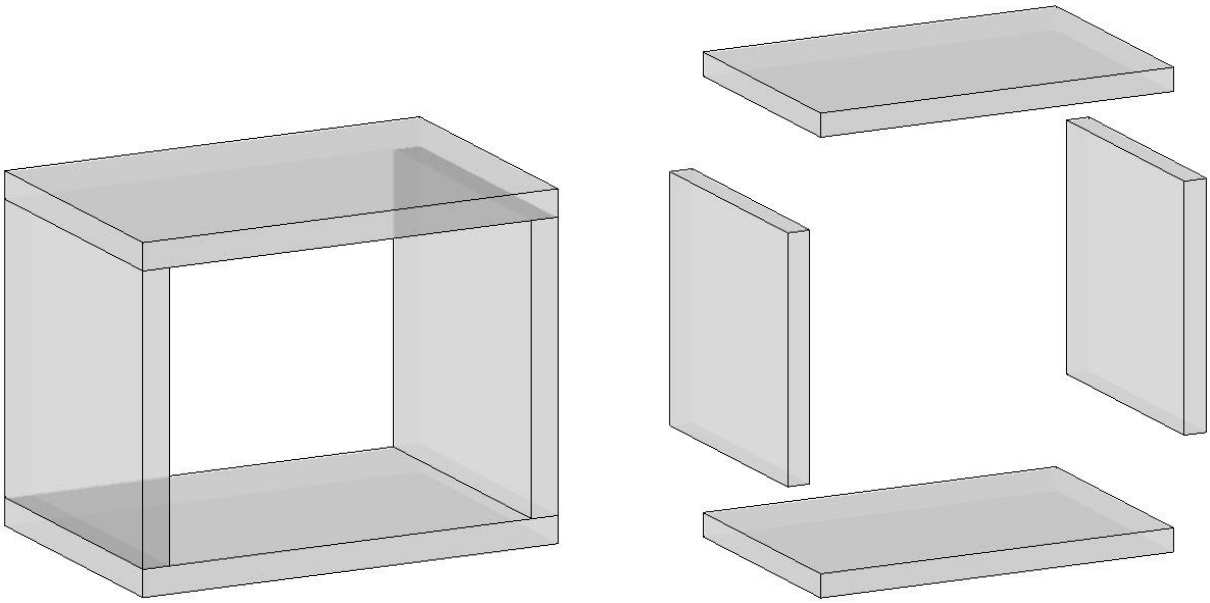
- Include the component in an assembly, and then create the cuts.



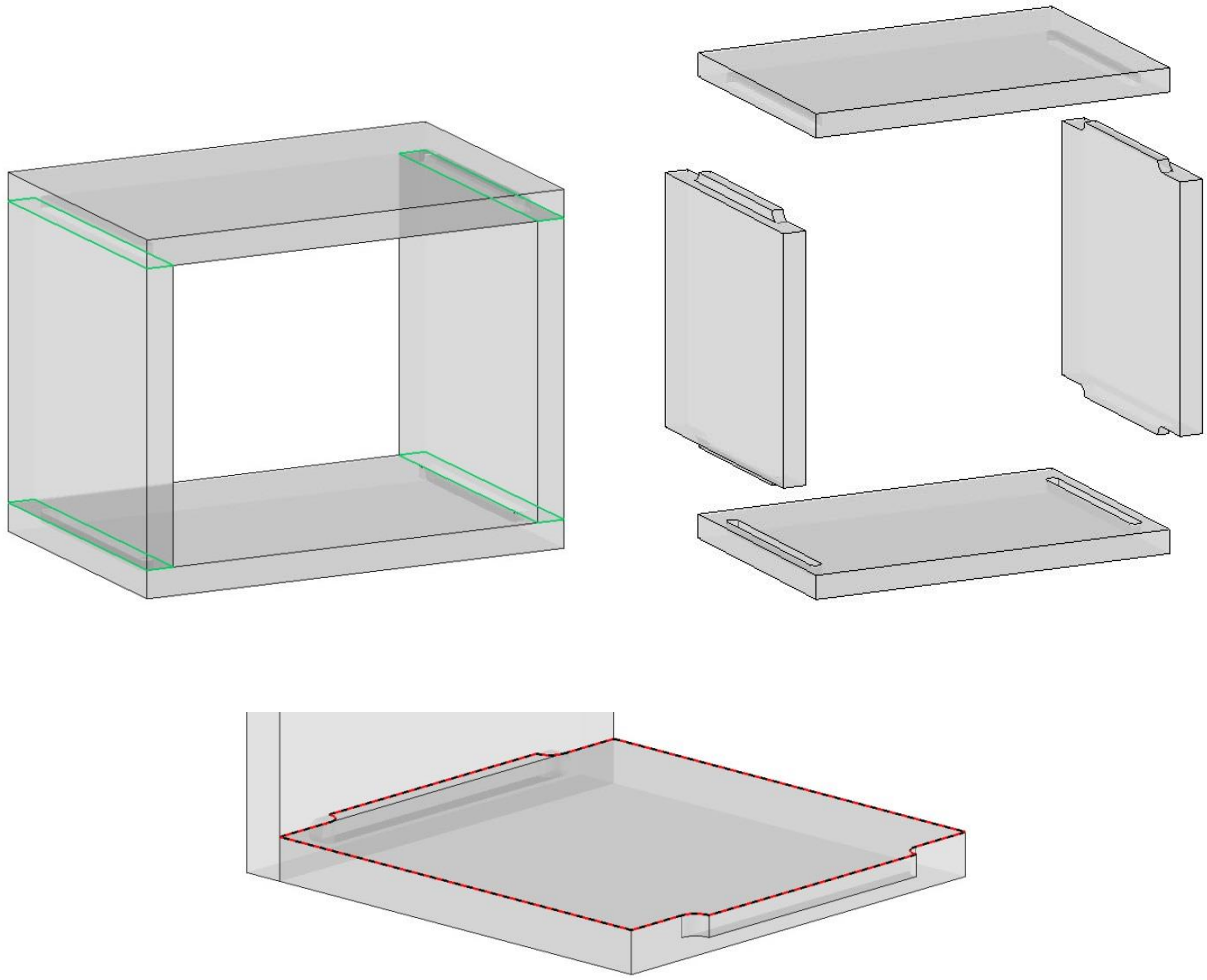
Automatic material addition using component process

It is now possible to automatically add material to a part by using an automatic tool process.

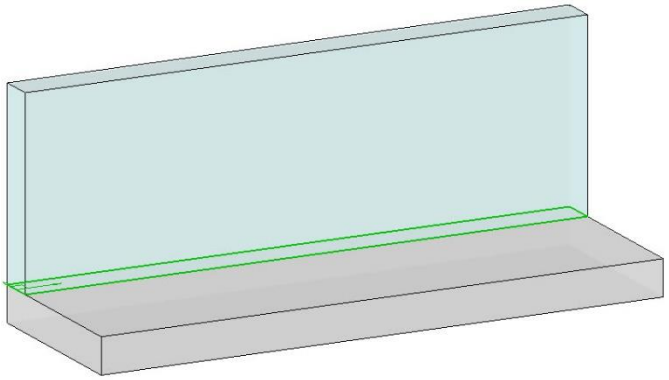
Example of a cabinet with butt joint parts.



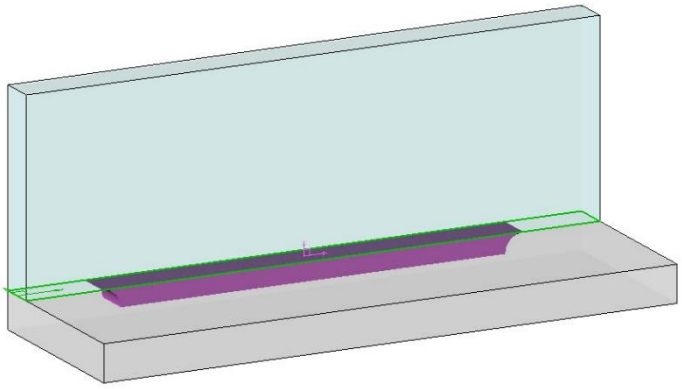
Result after the parts have been automatically assembled using a kit with an automatic union process.



- Create the component.
  The example is an advanced assembly kit.

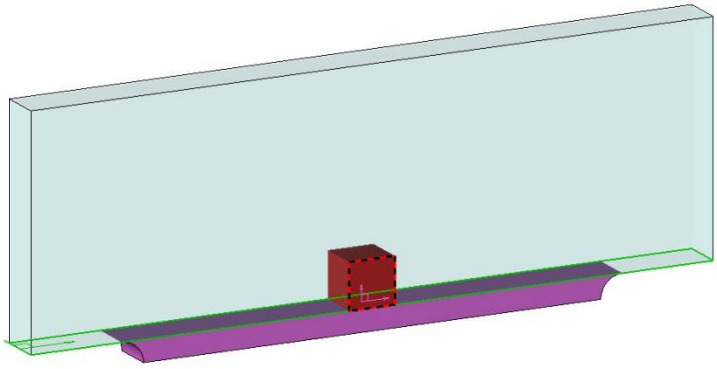


- Create the shape for the union. Here, this is a shape extruded from a sketch (in purple).

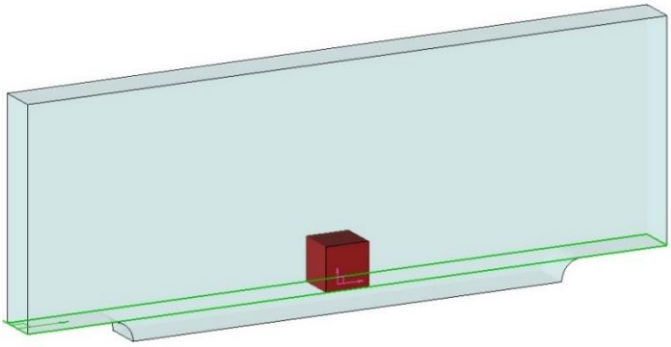


Note: The automatic union can be done in a process only if the union has a **clash shape**. In a simple process (drilling, pocket…), the tool's **clash shape** facilitates and/or limits the search for parts to be operated.
In the case of an automatic union, the **clash shape** can be used to determine the part with which the **union** must be done.

- Create an extruded part (red) which is in contact only with the blue part.

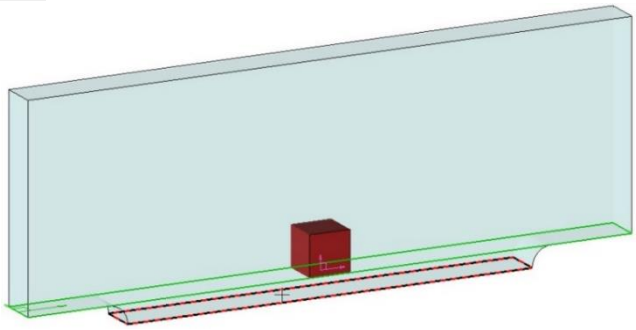


- Make the part union using the **Shape** > **Unite**  command, and select the martyr part (blue) as **Shape(s) to modify** and the part to be united (in purple) as **Tool shape(s) to use**.



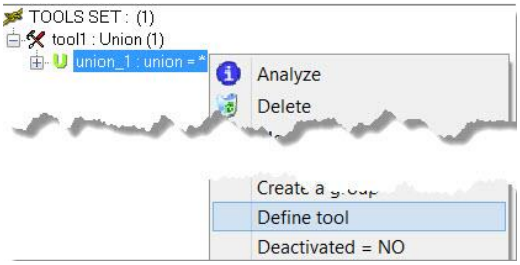
- Create a tool in the component using the **Assembly** > **Define component** > **Define tool** command.
  The tool must be created in **Operation type = Local operation on shapes** mode.

- When selecting the **local operation to insert in tool**, select one of the united shape's faces.



Note: The union is now declared as a tool. However, it can be used only if the clash shape is declared.

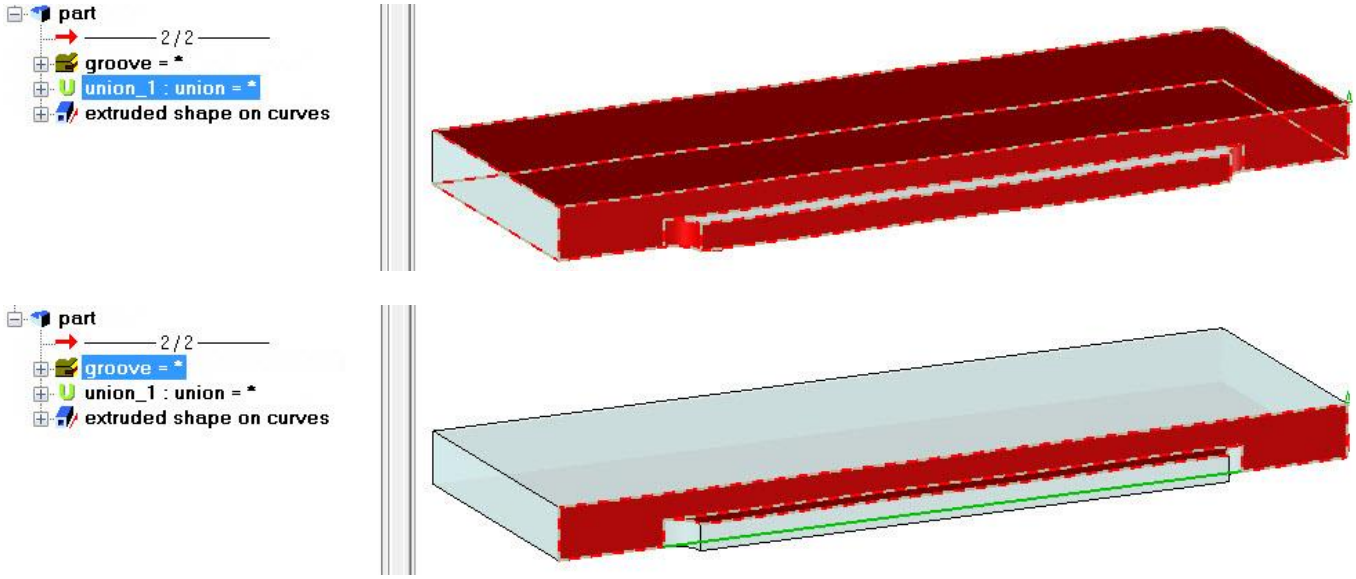
- From the construction tree, edit the tool set.
- On the union line, **right-click** > **Define tool**.
- Select **Define clash shape**.



- Select the previously extruded part as the **clash shape**.

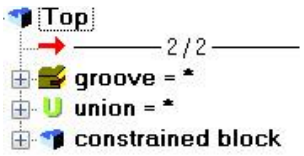
Note: In the component tools, it is possible to create a tool which will subsequently operate the united shape. In the tool set, this operation must be located under the union process.

Preview of the martyr part



Order of the component tools

Result in the assembly



Note: The automatic union in a component process, just like the manual union, is not allowed on panel entities.

Constrained positioning

3D labels are now displayed when creating or modifying the constrained positioning.

- Insert a component in an assembly.

When a positioning constraint is created, a label appears, pointing to the constrained face.

After creating the constraint, the constraint labels already created are displayed.

These labels are divided into four areas 

:

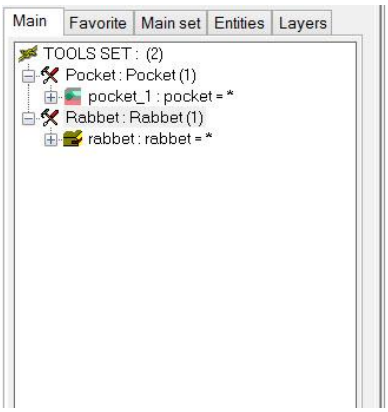
- The first area is the icon of the created constraint (plane on plane, axis on axis…). When the cursor is positioned on this icon, a graphic echo shows the constrained elements.
- The second area can be used to modify the distance between the two constrained elements. For example, in the case of a plane on plane constraint, this allows you to move the planes away from each other or bring them closer. Clicking in these areas allows you to enter the distance manually, use the **+** or **–** buttons, or move the cursor. Before using the + or – buttons or the cursor, a step must be entered.

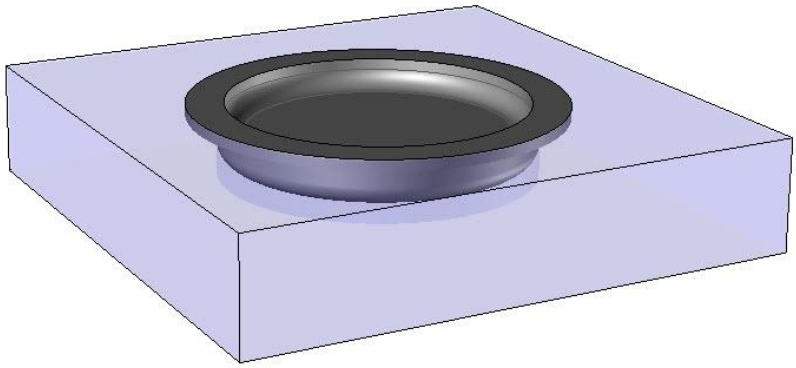
Note: On some constraints, it is not possible to enter an offset value, like for an axis on axis constraint for example. This area therefore does not exist.

- The third icon inverts the constraint.
- The fourth icon deletes the constraint.

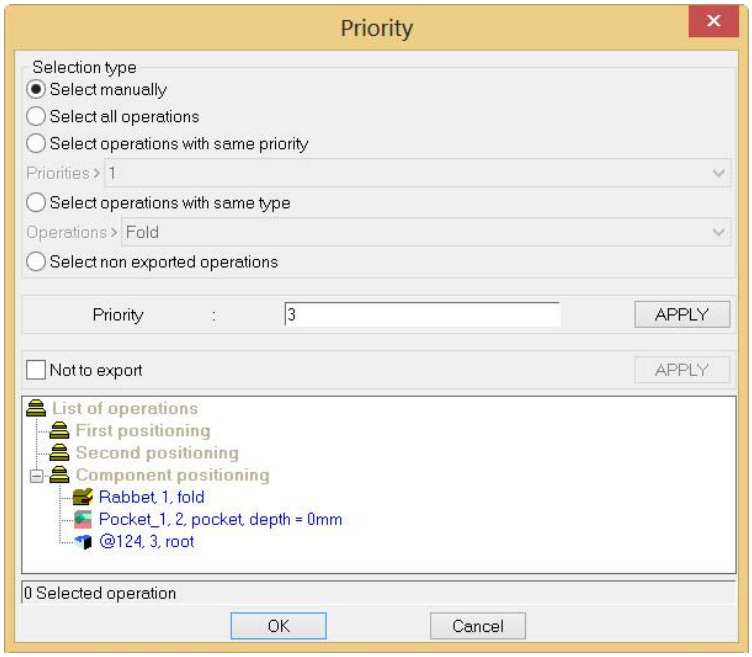
Operation priority: Component positioning

The **Operations priority** function now provides an additional positioning that allows you to assign priorities to the operations generated by the component, while respecting the positioning when it is assembled on different faces of a part.
Below is a handle component that generates a pocket and a rabbet when it is inserted.



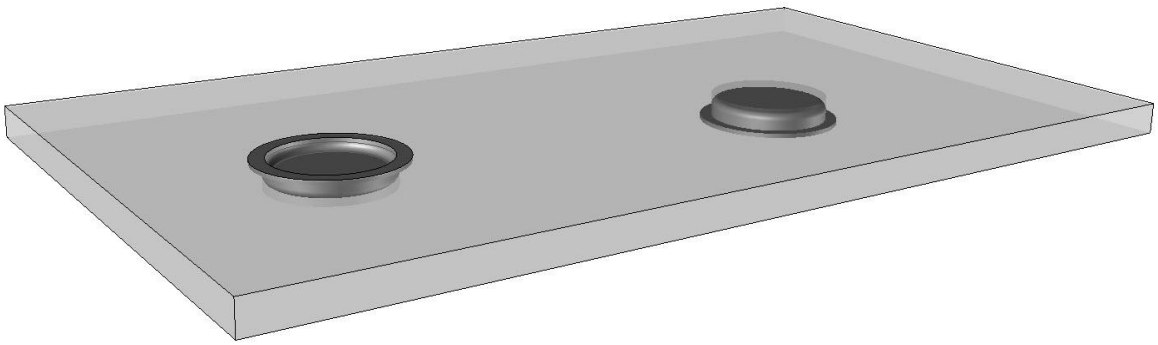


- In the **Operations priority** function, place the operations that will be generated by the component in **Component positioning**.

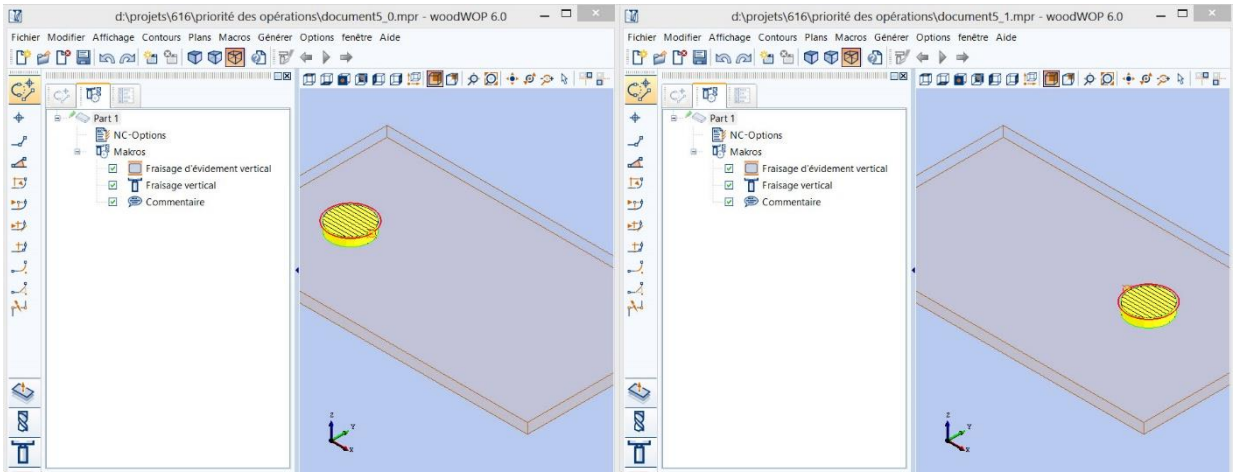


When using this component in an assembly, the operations will be exported according to the priorities defined in the component model, while taking into account the face on which it is positioned.

In the example below, the handle is inserted on two opposite faces.

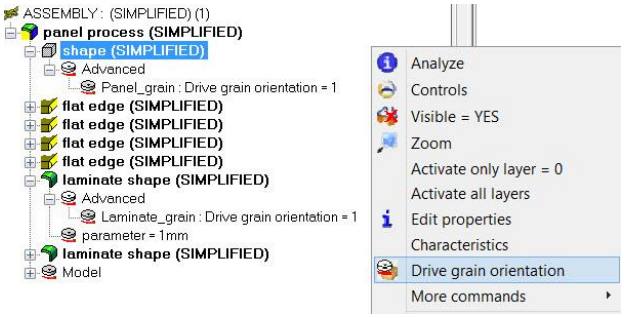


During export, the operations will be generated in the order defined in the model and on two different faces.

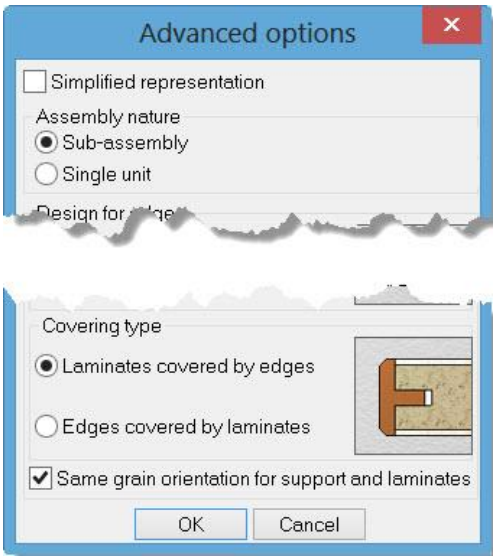


Panel grain orientation

It is now possible to drive the grain orientation of a panel's supports and laminates by **right-clicking** > **Drive grain orientation** on the support or a laminate of the panel entity. The driver parameter is then displayed in the tree under the concerned element.



If the grain orientation of laminates must always follow the support's grain orientation, the **Same grain orientation for support and laminates** option must be checked in the panel's **Advanced options**.



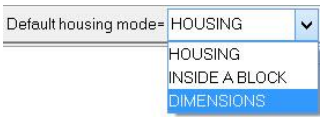
Settings

Driver block in Dimensions mode

Driver blocks can now be inserted in **Dimensions** mode. This makes it possible to specify the block's dimensions, and then position it using constraints. These dimensions can be driven by a catalog or manually entered during inclusion.

Note: The driver block is not compatible with key points/coordinate systems; only the constrained positioning can be used.

When creating the driver block, a new default housing mode is available.



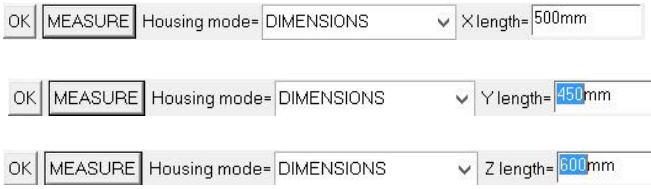
The three dimensions of the driver block can be driven by a catalog. To do this, once the driver block is created, do the following:

- Select **Assembly** > **Define component** > **Edit catalog header** and choose **All parameters and texts**.
- An Excel file opens. Enter the values for the X, Y and Z dimensions of the driver block.
- **Save** the file.

	A	B	C	D
1	$code	bp.x	bp.y	bp.z
2	H=600 L=500 P=450	500	450	600
3	H=650 L=550 P=450	550	450	650
4	H=700 L=600 P=500	600	500	700

- In an assembly, select the code to be used when including the component.

By default, the X, Y and Z values of the driver block match the values you specified in the catalog.
Still using the same catalog, the first code has been selected during the inclusion. The component dimensions are as follows.

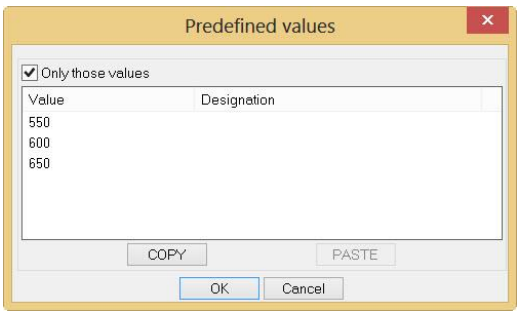


Note: These dimensions correspond to the created catalog. However, these values are not fixed. It is possible to enter different values from those in the component catalog.

Predefined values can also be set for each of the parameters.

- **Edit** the driver block from the tree.
- Double-click one of the three lines corresponding to the block's dimensions.
- Click on **Predefined values**. 

- In the dialog box, enter the different possible values. If the **Only those values** option is checked, a drop-down list will appear during insertion, and it will not be possible to enter a value manually.



Wood machinings

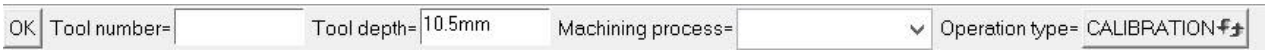
User machining on simplified panels

User machinings can now be created on simplified panels.

Calibration deactivation option

User machinings can now be defined as calibration or milling operations.

- Create a part and define it.
- Launch **Wood** > **User machining**. 
- Click the reference face, and then the tool path.
- Select the operation type: **Calibration** or **Milling**.



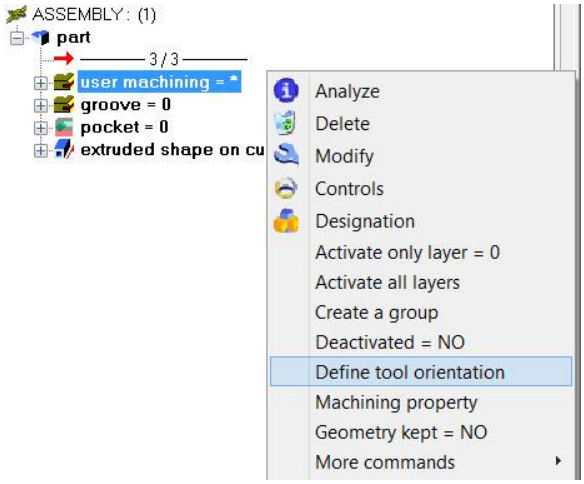
During a machining export, the **Calibration** user machining is subject to the calibration rules: proper positioning for part definition, deactivation of automatic calibration.
However, a **Milling** user machining is considered as a pure milling operation.

Note: The user machining created with the previous versions is automatically considered as a calibration, but its type can be modified.

User machining orientation

It is now possible to define the orientation of the tool used by a user machining. To do this, **right-click** > **Define tool orientation** on the operation. The given direction will then correspond to the tool axis.

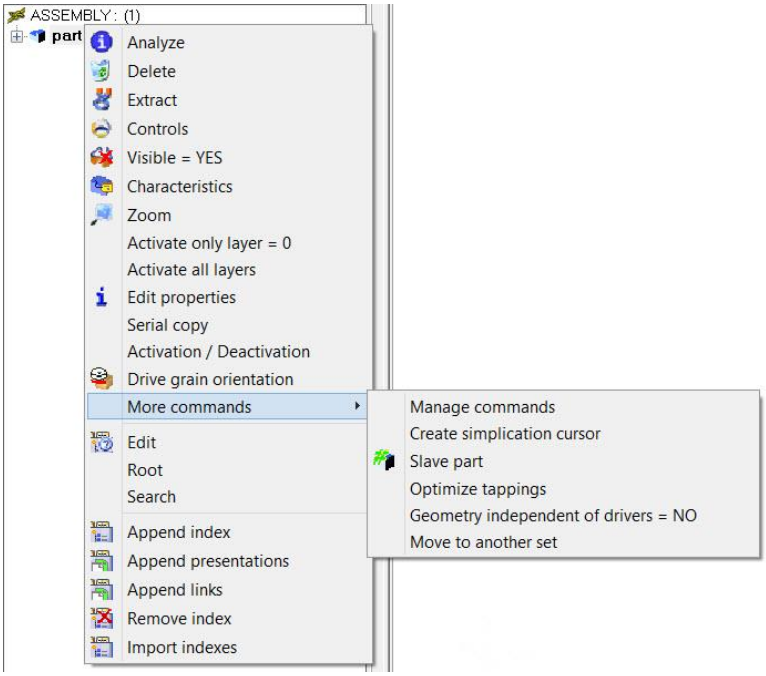
Note: This function has been developed mainly for TopSolid'WoodCam users.



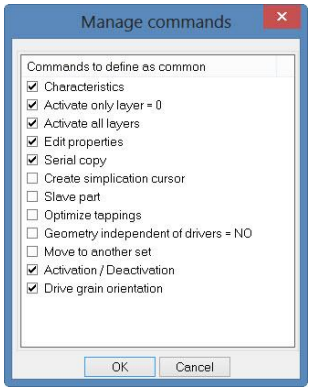
Configuration

Customized contextual commands

It is now possible to simplify the construction tree's contextual menu. This provides an easily navigable menu and quick access to commonly used features.
When right-clicking an assembly or a part, the **More commands** submenu has been added and lists the non-common commands.



- To define commands as common, right-click on a part or an assembly, and select **More commands** > **Manage commands**.
- Check/Uncheck the functions. All unchecked functions will be listed in the **More commands** submenu.



Note: Some commands cannot be defined as non-common and are therefore not listed: **Modify**, **Delete**, **Analyze**, **Visible**, **Control**, **Zoom**, **Break associativity**, **Replace**, **Extract**, **Insert** and **Name**.

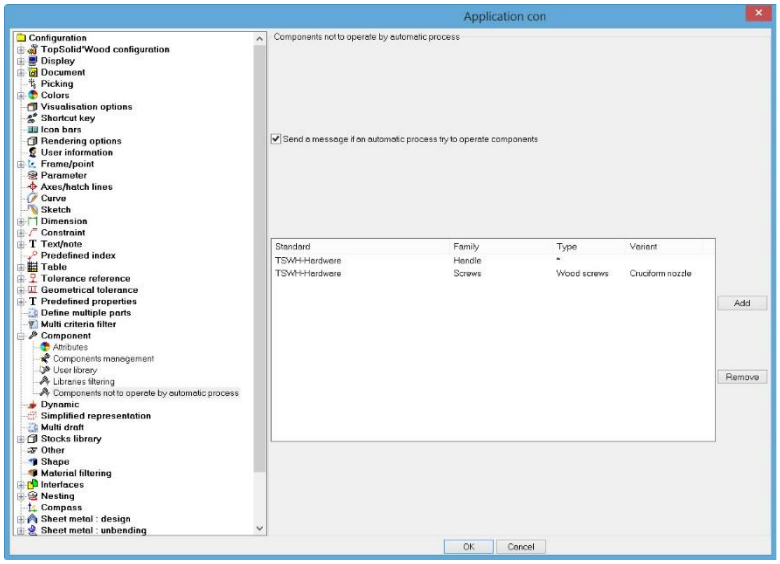
This setting is stored in a **macro.cfg** configuration file contained in the **Config** directory.

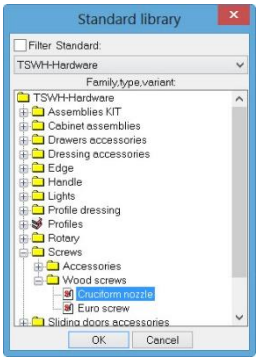
Note: It is possible to manually place the **macro.cfg** file in the **Group** folder to have the same setting on several stations. If the **macro.cfg** file is located in both **Group** and **Config** folders, only the file in the **Group** folder will be loaded.

Definition of elements not to operate

It is now possible to define standard components that should not be operated by automatic processes. For example, this prevents a dowel from operating a runner.

- To define the standards not to operate, go to **Tools** > **Options** > **Component** > **Components not to operate by automatic process**.



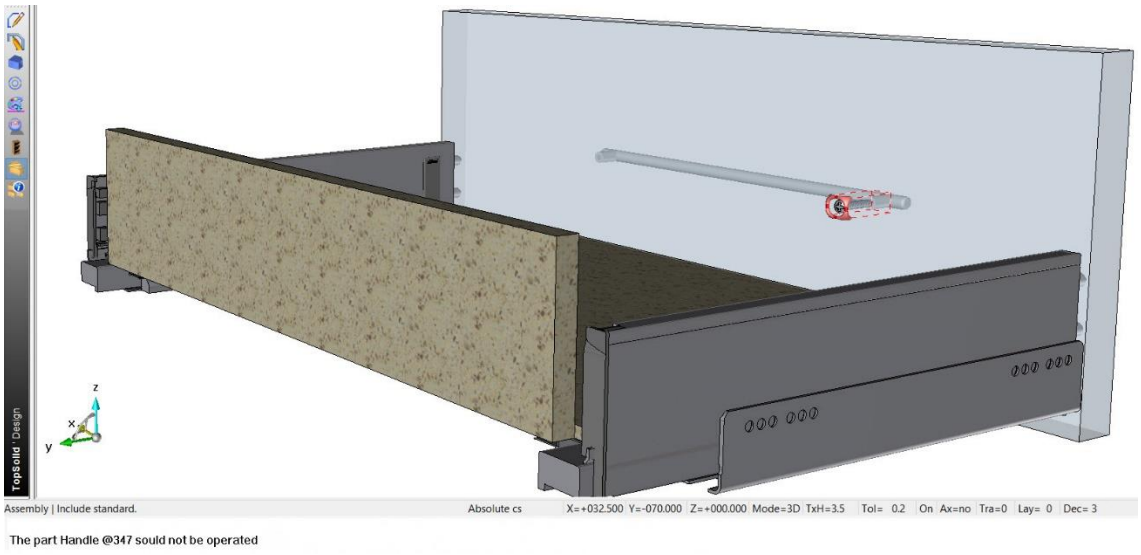


- Click on **Add** to define a standard not to operate.
- A dialog box for selecting components appears. Select the component not to operate, and then **validate**.

Note: It is possible to select the component directly or point to a folder or subfolder of the library. In this case, all components located in the folder will not be operated in assemblies.

- Once the standard is added, it is possible to remove a component from the list using the **Remove** button.
- Check or uncheck the option that sends a message if an automatic process attempts to operate the selected components. This only displays information in the alpha bar. In both cases, components are not operated.

In the example below, the screw attempts to operate the handle declared as a component not to operate. A message is displayed in the alpha bar.



Assembly order for automatic numbering

During automatic numbering, it is now possible to modify the order of components in the bill of material. This order can be modified using drag-and-drop directly from the construction tree.

Note: Only the consecutive identical elements will be grouped together.

- In an assembly, open the construction tree, and move the elements by **drag-and-drop** operation.
- So that the order displayed in the tree is the same in the bill of material, open the .bom file of the bill of material using Notepad.
- Modify the **ORDER_BY** paragraph. The syntax must be the following:

```
ORDER_BY {

        ORDER_BY=NULL
}
```

For example, an assembly with the following structure. As identical parts are not consecutive, they are not grouped together in the bill of material.

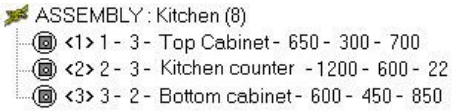
Assembly:

Bill of material:

Now if parts are reordered by drag-and-drop in the assembly and identical parts are consecutive, then these parts are grouped together in the bill of material and during numbering.

Reorganized assembly:

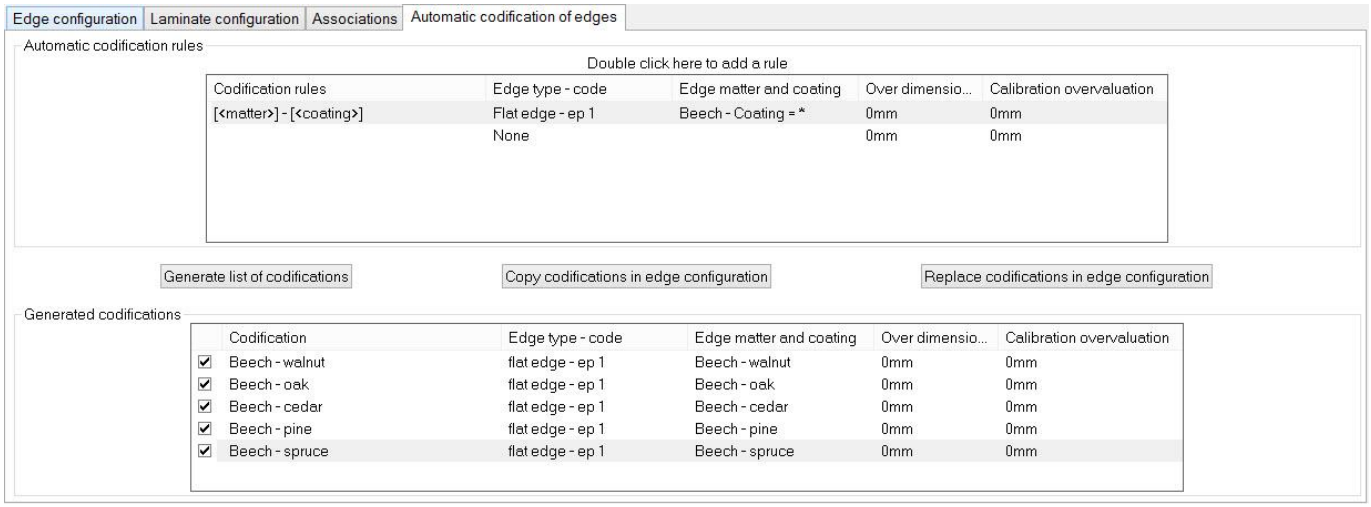
Bill of material:



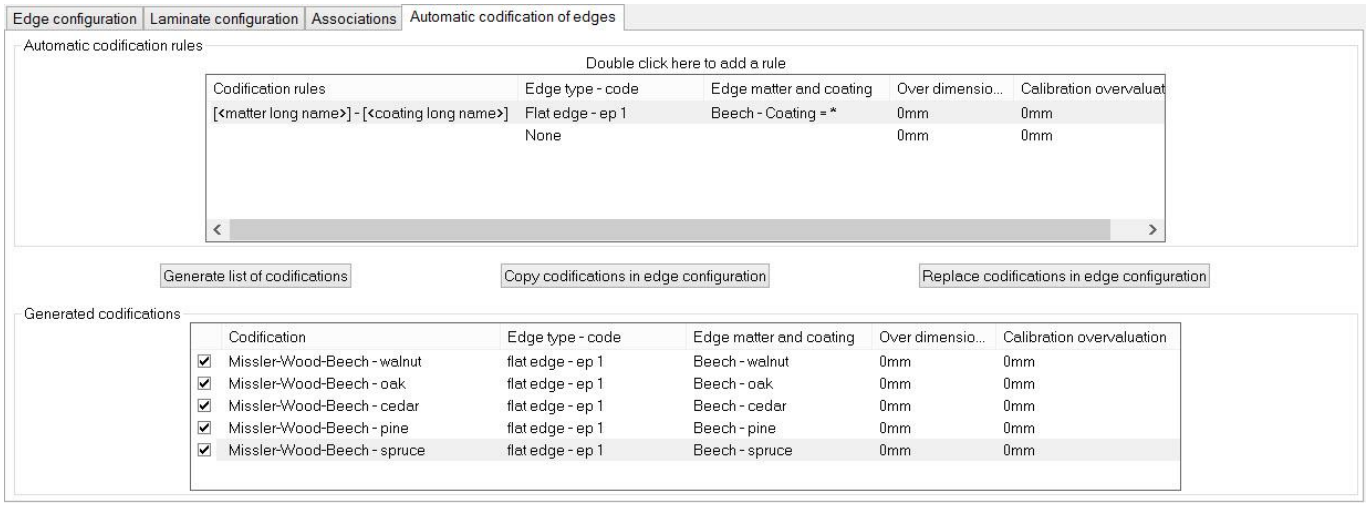
Material names in edge codification

User matters and coatings are now properly translated during automatic generation of codifications. The syntax for displaying them has not changed; [<matter>] can be used to retrieve the matter and [<coating>] the coating.

- Go to **Tools** > **Options** > **TopSolid'Wood configuration** > **Edges/Laminates** > **Automatic codification of edges** tab.
- Specify a codification rule, and then generate the list of codifications.



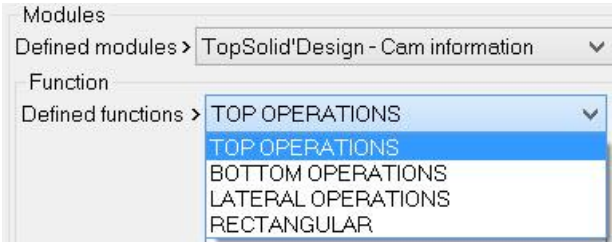
It is also possible to display the long name of matters and coatings in the codifications. This will show the category and the subcategory to which the matter or the coating belong. To do this, use **[<matter long name>]** and **[<coating long name>]**.
For example, for the **Beech** matter used in the previous example, the long name will be **Missler-Wood-Beech**, since this matter is in the **Missler** category, and in the **Wood** subcategory.



Note: Only user matters and coatings are compatible with this feature.

New BOM properties

Properties that retrieve machining information have been added. These properties all have a **YES** or **NO** value.
These new properties have been gathered into the **TopSolid'Design - Cam information** module.



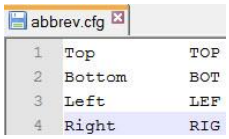
Wood Codification: Limited number of characters

This setting is used by the **WOO_CODIFICATION**, **WOO_CAM_FILE_NAME**, **WOO_NESTING_IDENTIFICATION** and **WOO_TREE_IDENTIFICATION** properties.
There are two ways to limit the properties, either using a correspondence table, or specifying the number of characters to be displayed.

Correspondence table: It is defined in a configuration file named **abbrev.cfg**. This file can be placed in the **Config** and/or **Group** directory. If two configuration files are found, they are automatically concatenated. In this file, you need to define the correspondence between the value provided in TopSolid and the value displayed in the bill of material. Then, in the bill of material, use the syntax *<WOO_CODIFICATION|$BOM property|Upper|Abbrev$>.*

For example, a part with the **Top** designation, and **Top** has to be replaced by **TOP**:

- Open the file **abbrev.cfg**.
- Write *Top*.
- Press the **Tab** key.
- Write the replacement value: *TOP.*

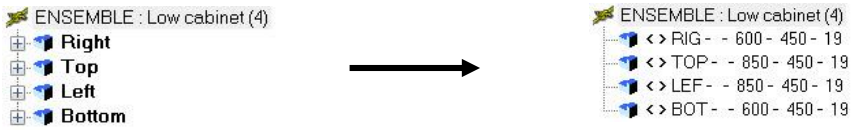


- Once the correspondences are defined, open the .bom file of the bill of material used.
- Enter the following syntax: *<WOO_CODIFICATION|$DESIGNATION|Upper|Abbrev$>.*

```
COLUMNS {

    NAME=DESIGNATION
    "DEF=<WOO_CODIFICATION|$DESIGNATION|Upper|Abbrev$>"
    TYPE=STRING
    ALIGN=LEFT
    TITLE_ALIGN=LEFT
    WIDTH=0.015
    VISIBLE=YES
    ;
```

- Display the bill of material of a file containing a part with a **Top** designation. Result: All designations using a text contained in the correspondence table are replaced.



Number of characters to display: It is possible to limit the number of characters to be displayed. To do this, use the following syntax in the BOM file:
<WOO_CODIFICATION|$BOM property|Upper|Trim(i,j,X)$>

In the **Trim(i,j,X)** property, the three parameters correspond to:

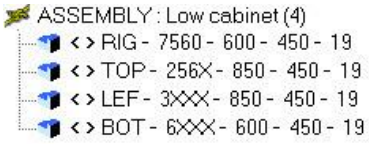
- **i**: Number of the first character to be kept. If this number is negative, the counting is done from the end.
- **j**: Number of characters to be kept.
- **X**: Fill character if the property has less characters than **j**.

Still using the previous example, the **Right** part has a reference **P75608**. In the bill of material, the first letter must not appear and only four characters must be displayed. The following syntax should therefore be used:

```
NAME=REFERENCE
"DEF=<WOO_CODIFICATION|$REFERENCE|Upper|Trim(2,4,X)$>"
TYPE=STRING
ALIGN=LEFT
TITLE_ALIGN=LEFT
WIDTH=0.015
VISIBLE=YES
;
```

Note: If both **Abbrev** and **Trim** are found in the syntax, the abbreviation will be trimmed.

Result in the bill of material: The display of the reference starts at the second character and the next four characters are displayed.



Note: As the **Left** and **Bottom** parts have a reference with only two characters, this reference is completed by the fill character (**X** in the example).

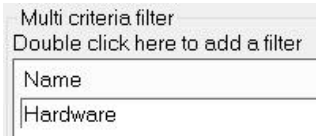
Improved custom filters

Reminder: Creation of a multi-criteria filter

Prior to addressing the various improvements made to the multi-criteria filters, here is a reminder of the procedures that need to be followed to create a filter.

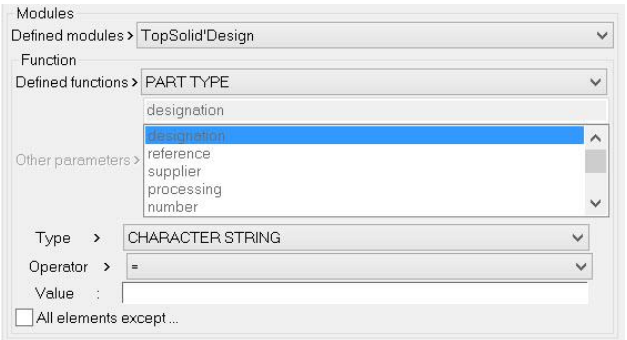
- Open the multi-criteria filters in **Tools** > **Options** > **Multi criteria filter**. 

- Double-click an empty line in the **Name** field and rename the filter.



For this example, a filter to select hardware parts will be created.

- Define the property to be used to filter the parts.



For this example, the property used is the part type. Therefore, you need to select the **TopSolid'Design** module and the **PART TYPE** defined function.

Note: The selection of the function used by the filter is similar to the selection of a BOM property. You first need to choose the module where the property is located, and then select it in the drop-down list.

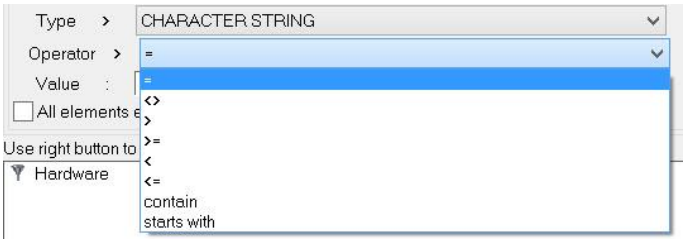
After that, the value type is requested. Three selections are available: **CHARACTER STRING**, **INTEGER** or **REAL**.

- In this example, select **CHARACTER STRING** since the part type is a text.

Note: The **INTEGER** and **REAL** types are reserved for numerical properties.

- Define the operator to be used.

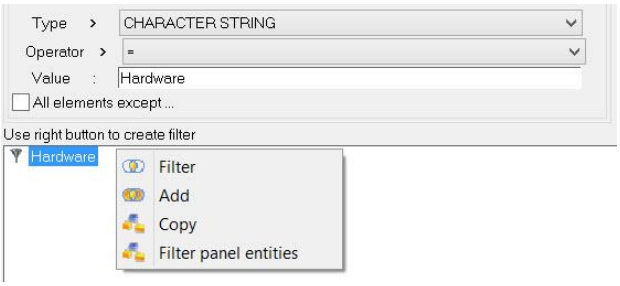
Note: If the selected type is **CHARACTER STRING**, only the **=** , **contain** and **start with** operators can be used.



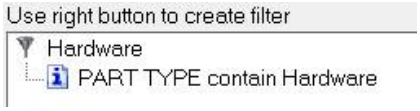
Note: In the case of a filter on a part type, it is recommended that you choose the **contain** operator. In this way, the part will be selected even if it has several types. The **=** operator is more restrictive and will select the part only if it has the concerned type (only).

- In the **Value** field, write the value to be compared.
  In this example, the value is **Hardware** since the parts to filter will have this type.

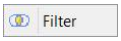
Warning: The PART TYPE value is case sensitive.



- Finish creating the filter by right-clicking the name line, and selecting **Filter**.

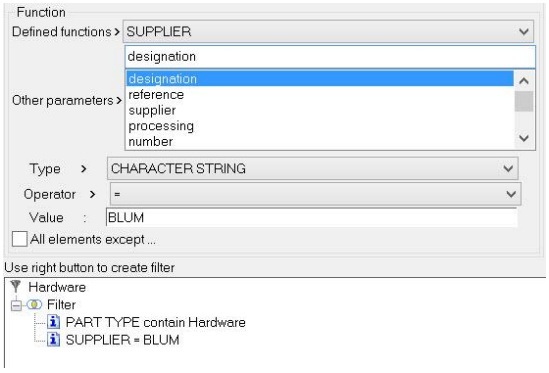


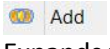
Note: When creating the filter, four options are available:



Used to refine the filter further.

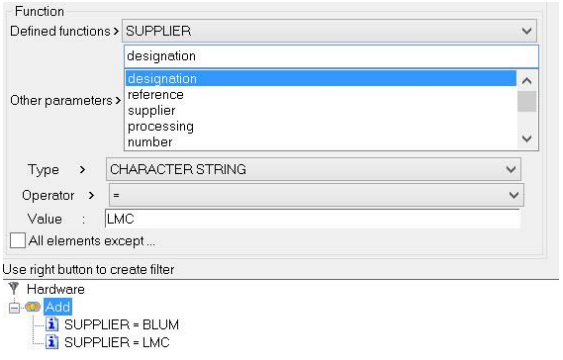
Example: A part A with the BLUM supplier and the Hardware type, and then a part B with the LMC supplier and the Hardware type. In the previously created filter, the Supplier property with the BLUM value is inserted using the **Filter** button. So when this filter is used, only parts with the Hardware type and the BLUM supplier will be filtered. In this example only, the part A is selected.





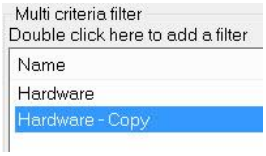
Expands the filter possibilities.

Still using the same example of the parts A and B, this time the first Supplier property is inserted with the **Filter** button; its value is BLUM. The second Supplier property with the LMC value is inserted using the **Add** button. When this filter is used, parts with the BLUM or LMC supplier will be selected. So in this example the parts A and B are selected.



Copy

Creates a copy of this filter by adding - *Copy* after the filter name; then you just need to modify it.



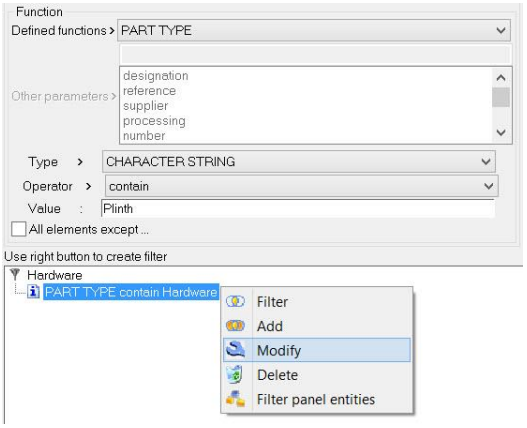
Filter panel entities

Automatically filters the panel entities.

Modify

Allows you to modify the filter after its creation.

- To do this, you first need to specify the new property to be filtered, and then right-click on a property of the filter and select **Modify**.



Delete

Deletes a property of the filter.

Using filters in part selections

Multi-criteria filters can now be used to select the parts. This option is available in the following dialog boxes: **Matters configurator**, **Automatic assembly**, **Multi-draft**, and all **Machining** and **Cut** exports. These filters must have been created first in **Tools** > **Options** > **Multi criteria filter**.

- In the part selection dialog box, choose **Select by filter**, and then select the filter to be used in the drop-down list.

Improved management of materials and textures

A new material definition window has been created. Materials and textures can now be set in the same window. This window works similarly as in the previous version.

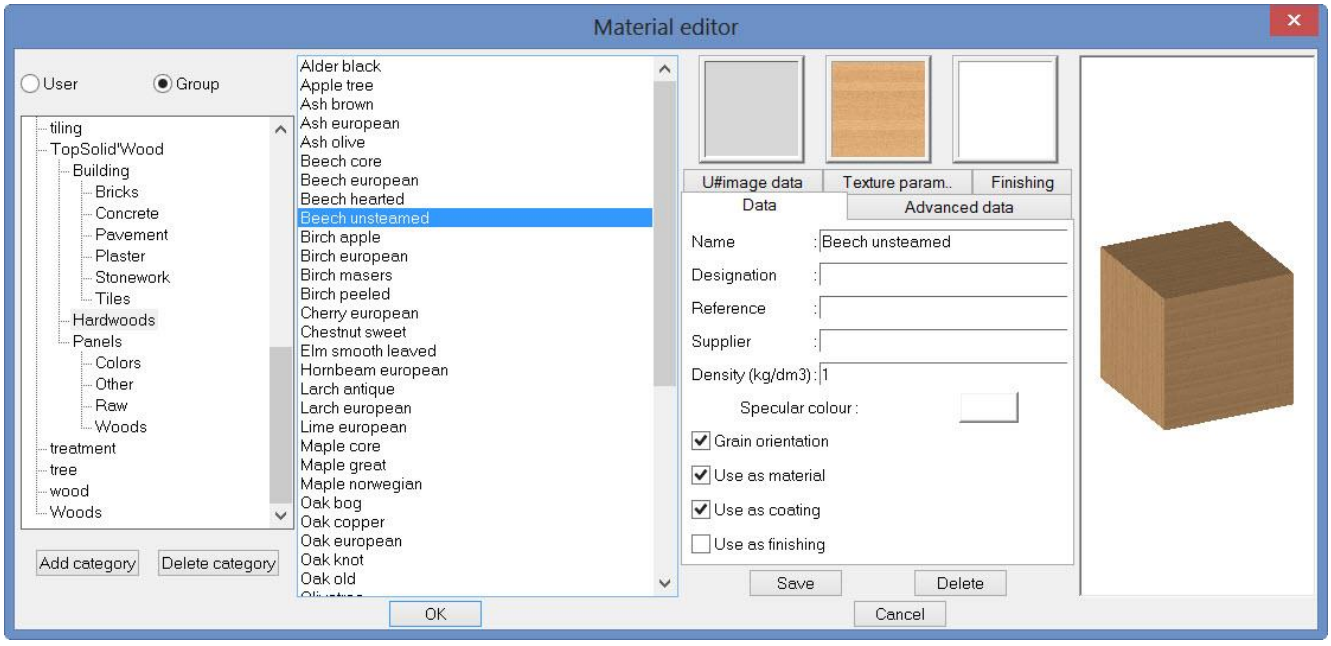
Note: The **Attributes** > **Textures** function is now only used to import textures; textures can be directly set (reflection, transparency…) using the **Attributes** > **Matters** > **Manage materials** function in the **Texture param.** and **Image data** tabs.

A new material file replacing the **top.mat** file has also been created: **topmaterials.xml**. The materials of the previous version will therefore need to be migrated.

- Launch **Attributes** > **Matters**.
- Click the **Migrate** button.

The materials are then automatically migrated to the new format and only the **topmaterials.xml** file will be used later.

When creating or modifying a material or a texture, a preview is displayed on the right side of the window.



Improved imaging

Light management

In order to be compatible with the **RedWay** module, lights have been redefined. It is now possible to:

- Create **OpenGL** lights.
- Create a **Sky** light that provides more options than those contained in the previous versions.
- Import an **IES** light file.

The existing **Positional** and **Spot** lights have simply been modified to be compatible with **RedWay**.

New RedWay module

A new **RedWay** imaging module has replaced LightWorks. The images generated are of better quality than in previous versions.

- Select **Image** > **Display**.
- Set the options.
- Click in the graphics area.

Draft

Multi-draft

Scale factor calculation according to the border

A new option in the multi-draft document calculates the view's scale factor based on the dimension of the view in relation to its border.
This provides a more adjusted scale factor and a view that is better positioned on paper.

Note: The old scale factor calculation mode took account of the view dimension according to the document margins, title blocks, and other views.

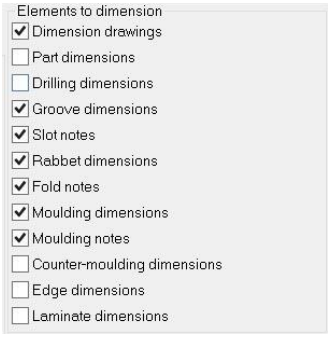
Free scale factor rounding

The view scale factor calculation has been improved to allow the scale factor to be rounded as close as possible.

Wood machining dimensions

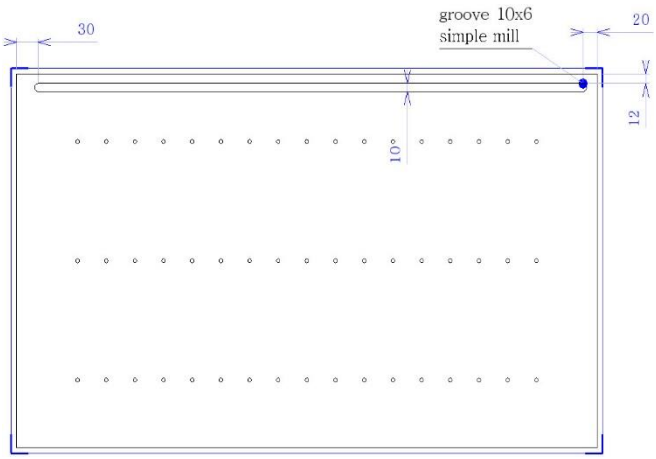
The automatic dimensions of **grooves**, **rabbets** and **mouldings** have been enhanced to display more information on these operations. In the **File** > **Properties** > **TopSolid'Wood properties** > **Draft** command, you can choose to project the notes and/or the dimensions for these three operations. The projected dimensions for each operations are the following:

- **Grooves**: The width, depth and position dimensions and the entry and exit dimensions.
- **Rabbets**: The width and depth dimensions and the entry and exit dimensions.
- **Mouldings**: The entry and exit dimensions.



The notes display the operation name and the name of the standard used as a tool.

Below is an example of dimensions on a part with a groove. In the document properties, the **Groove dimensions** and **Slot notes** options have been checked.



Note: The tool designation can be displayed in the note. To do this, the 

option must be checked in **File** > **Properties** > **TopSolid'Wood properties** > **Draft**.

Component detailing projection

Just as it was possible to project a schematic representation of components, it is now possible to project detailing elements.
For example, this allows you to automatically project dimensions with the drafted assembly.

Elements to be projected must first be defined in the component template.

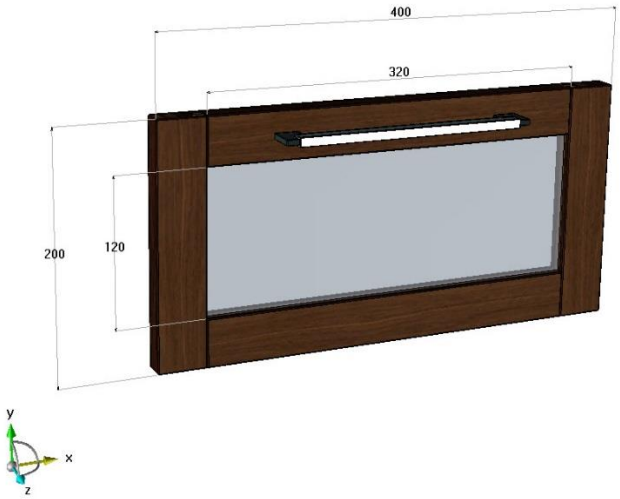
- On the **main assembly** (or on an alternative set), **right-click** > **Define draft detailing**.
- Then set the **viewing direction**. Two modes are available depending on the behavior you want.

  - 

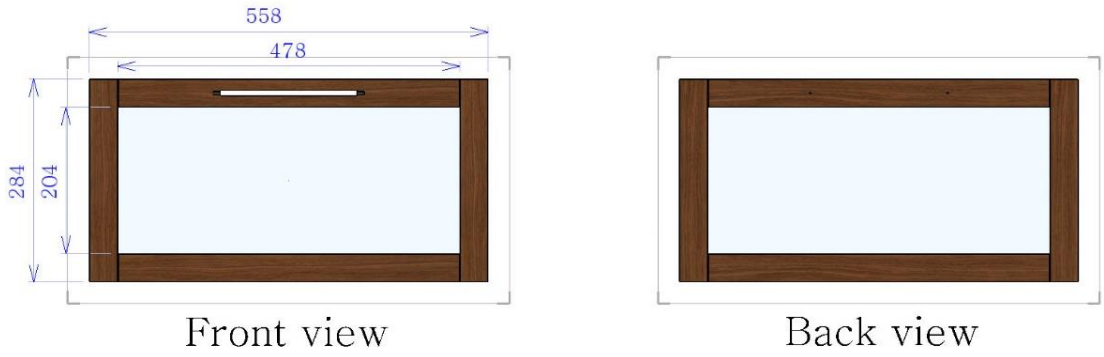
    This mode allows you to project different elements according to the view orientation in the draft. You therefore need to specify the orientation in which detailing must be projected.

In the example below, if dimensions only have to be projected in front view, the **Z+** direction must be selected.

Note: In order to select the correct orientation, it is best to use the compass orientation in the graphics area.



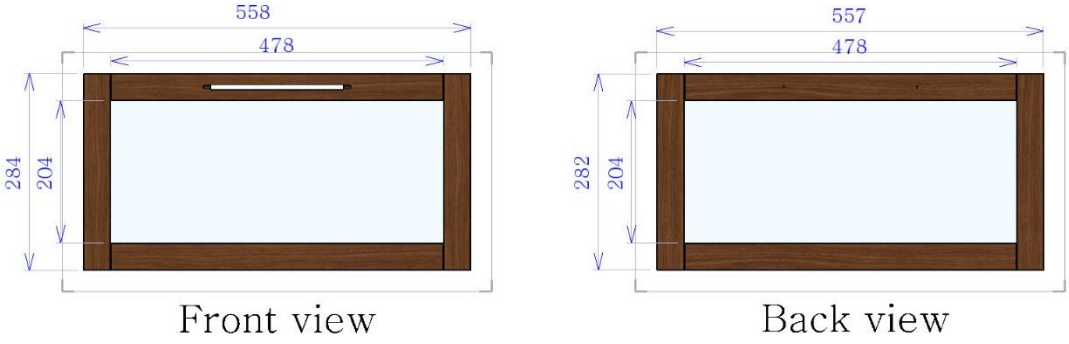
Result in the draft document:



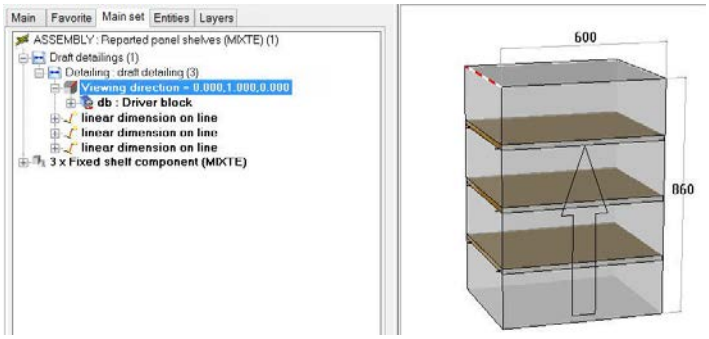
- 

  This mode is used when no specific orientation is needed.

Still using the same sample file, if this time the selected direction is **Z**, the result in the draft is the following.

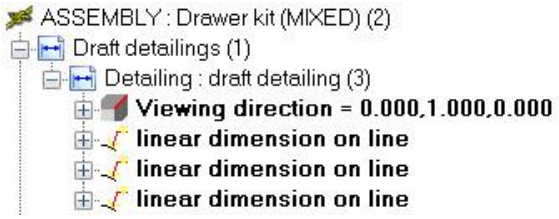


Note: In the case of a driver block component, it is recommended that you select an edge of the driver block.



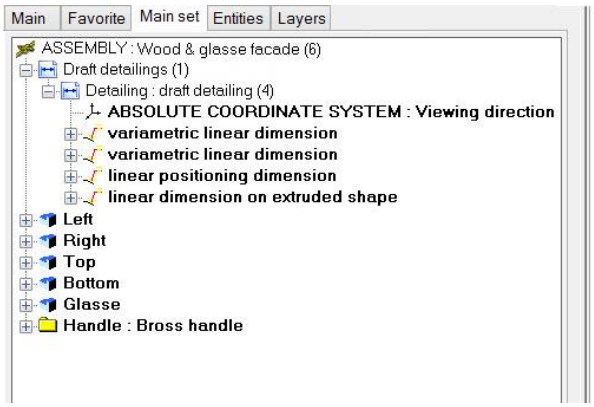
- Next, select the elements to be projected in the draft.

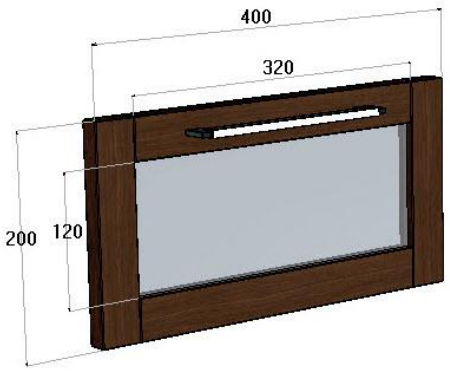
Note: It is possible to select curves, dimensions, shapes, texts.



- Once the selection is completed, you can give a name and a designation to the draft detailing or finish by clicking on **OK**.

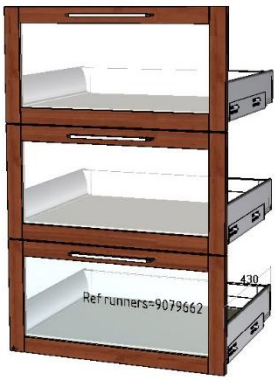
Once created, a **Draft detailings** line is added under the concerned assembly.





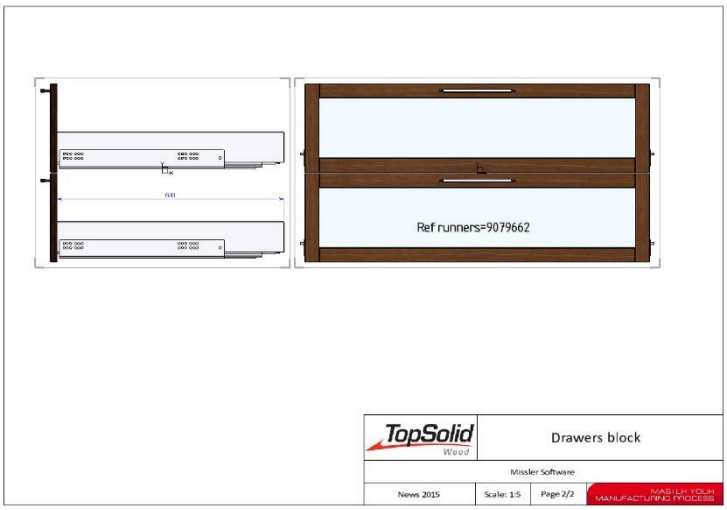
- After defining the draft detailing in the component template document, include it in an assembly, and then create the draft of this assembly.

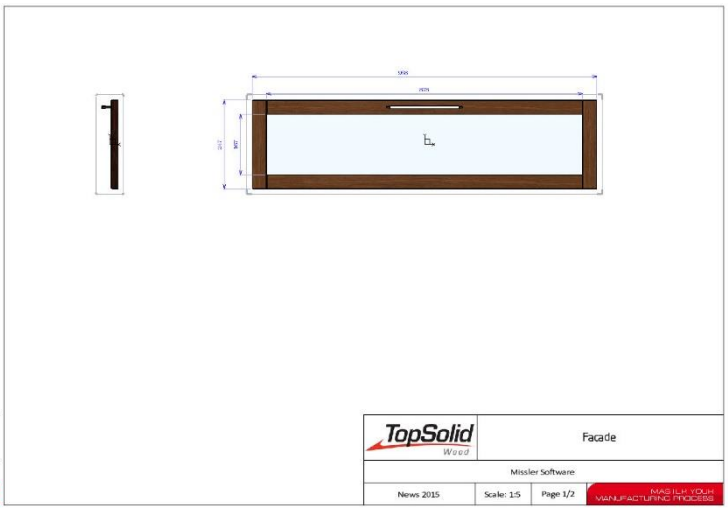
This front is then assembled to form a complete drawer set.
A formatted text that retrieves the reference of runners and a depth dimension are defined as draft detailings.
Two detailings are created since the two elements are not visible in the same direction.



The drawer set component is then included in an assembly.
When creating the multi-draft document, the drawer set and the front are selected in order to generate a drawing of each of them.

On the multi-draft result, both components are detailed.





Once assembled, the drawer is projected in the draft.

- To display the detailing elements in a draft, check the **Components detailing projection** option in:
  - **Tools** > **Options** > **Projection parameters** if you still want to project the component detailing.

Note: This only applies to drafts that do not use any existing templates because, in that case, the setting made in **File** > **Properties** is taken into account.

- **File** > **Properties** > **Projection parameters** if you want to project the detailing elements only in a specific document (a draft template for example).
- The creation or modification dialog box of a view if only a specific view must project the detailing elements.

Note: The draft detailing in only inherited at the first assembly level.

Machining export interfaces

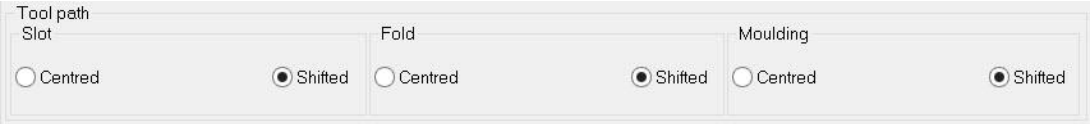
Export of shifted machining tool paths

It is now possible to export in the **TopSolid'Wood** machining interfaces the shifted machining tool paths for the **mouldings**, the **slots** and the **folds**.
This type of tool path can be configured for the following interfaces:

- **WoodWop**
- **Xilog Plus**
- **NC Hops**
- **Ardis**
- **EnRoute**

Note: For slots and folds, this setting will only be done on machinings performed with a mill tool.

- Open the options from **Tools** > **Options** > **TopSolid'Wood configuration**.
- Open the settings of the concerned interface from the **Machining** or **Cutting off** category (for **Ardis**).
- From the **Tool path** section, configure the following:
  - **Slot**: **Centred / Shifted**
  - **Fold**: **Centred / Shifted**
  - **Moulding**: **Centred / Shifted**



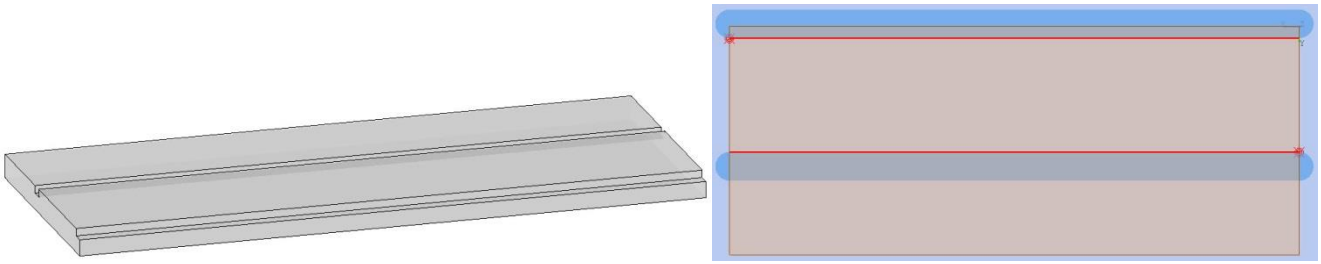
Note: The **Centred** tool path type is the export mode that is currently used and selected by default.

EnRoute cut export

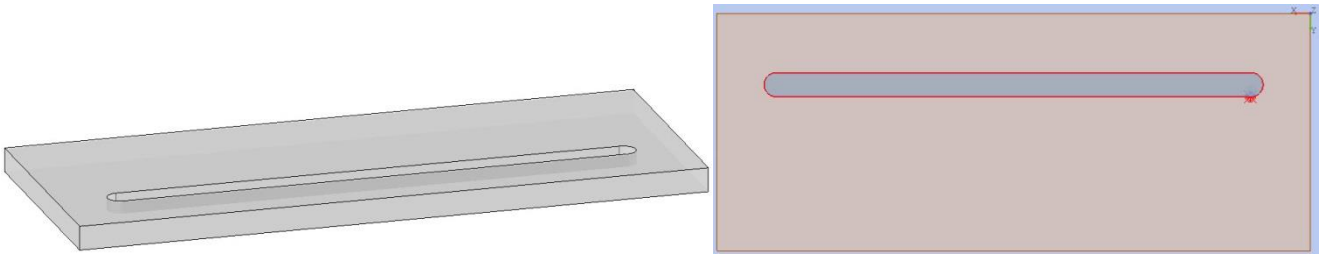
For the export to **EnRoute**, the shifted tool paths are set via the following configuration words.

Configuration word	*Value*	*Effect*
ZX_ZWOO_CAM_DXF_FOLD_CENTRE	1 (by default)	Export of the centered tool path for folds.
	0	Export of the shifted tool path.
ZX_ZWOO_CAM_DXF_SLOT_CENTRE	1 (by default)	Export of the centered tool path for slots.
	0	Export of the shifted tool path.
ZX_ZWOO_CAM_DXF_MOULD_CENTRE	1 (by default)	Export of the centered tool path for mouldings.
	0	Export of the shifted tool path.

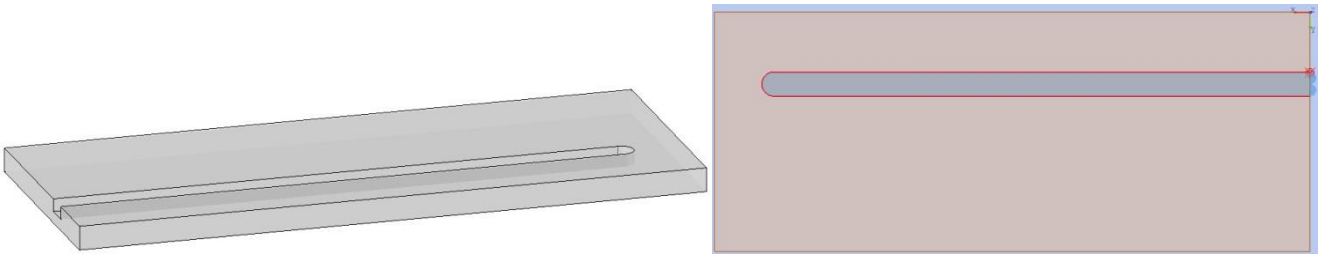
- For a machining that goes through the part, on an edge or within the part, the shifted tool path is exported.



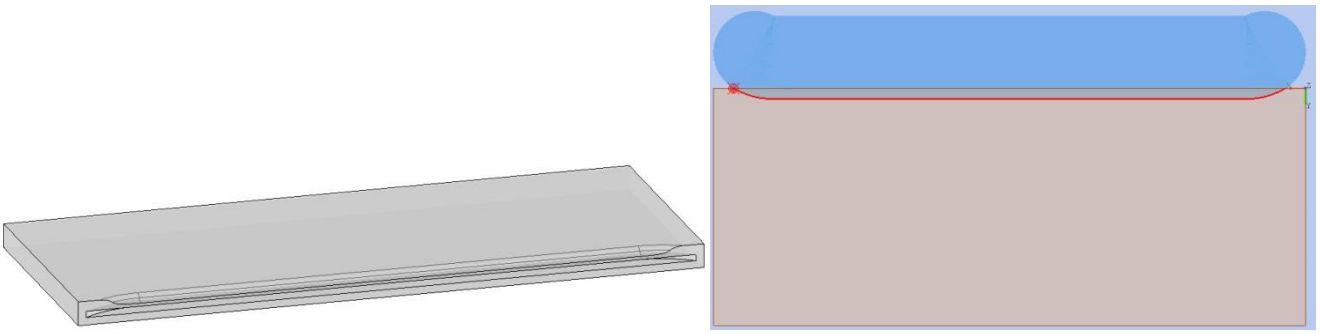
- For a machining stopped in the part, the closed contour of the machining is exported.



- For a machining which both is stopped in the part and goes through the part, the open contour of the machining is exported.

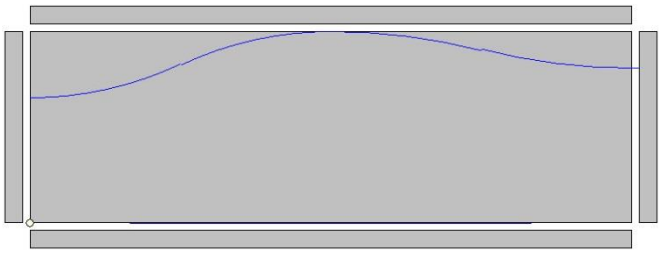


- For a machining stopped on the part edge, the open contour of the machining is exported.



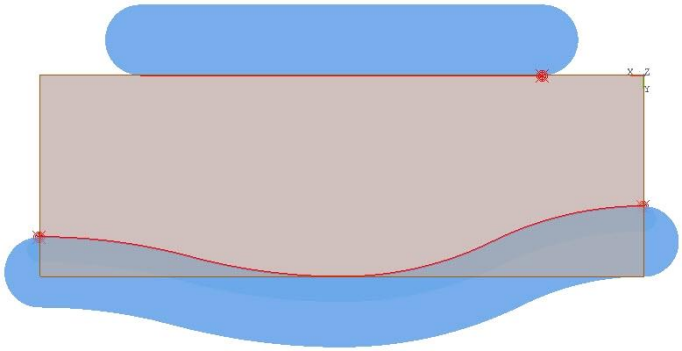
Moulding

Result imported in **Ardis**.



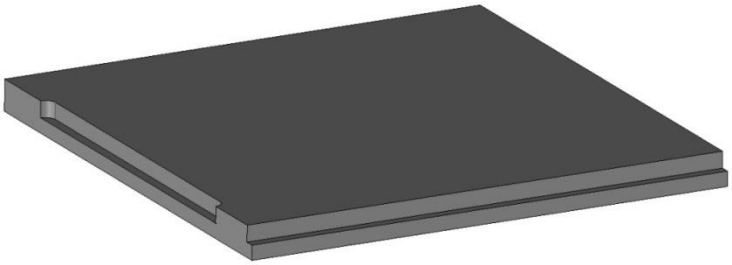
Function	Operation	Shift	Tool Number	Operation Number	Order
ARC	Groove	Right	T="101"	"2"	1
LINE	Groove	Right	T="101"	"3"	1
LINE	Groove	Right	T="101"	"3"	2
LINE	Groove	Right	T="101"	"3"	3
LINE	Groove	Right	T="101"	"3"	4

Result imported in **WoodWop**.

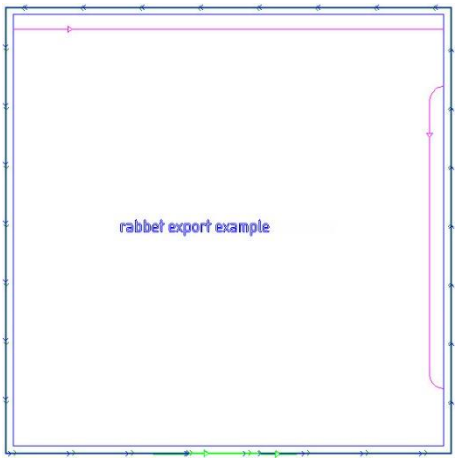


Fold

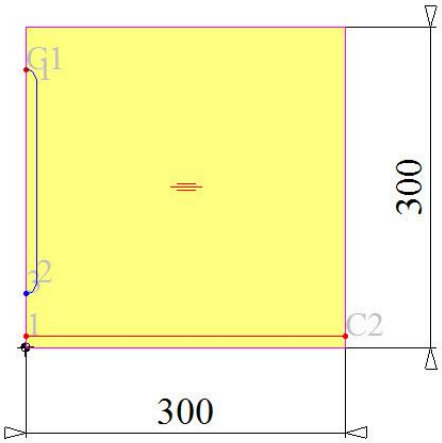
Part exported from **TopSolid'Wood**.



Result imported in **EnRoute**.

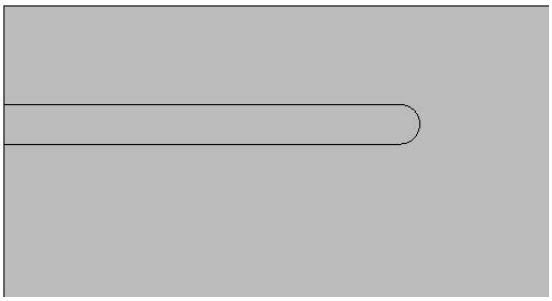


Result imported in **NC Hops**.



Slot

Part exported from **TopSolid'Wood**.



Result imported in **Xilog**.

New Cyflex configuration for Xilog

The **Xilog** export is now compatible with the **Cyflex** machine.

- The machine for the export can be selected from **Tools** > **Options** > **TopSolid'Wood configuration** > **Machining** > **Xilog Plus configuration**.
- Select **SCM Cyflex** from the **Machine** drop-down list.

The following configuration words are also available for **SCM Cyflex** export.

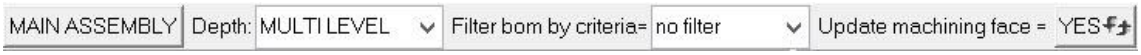
Configuration word	Value	Effect
ZX_ZWOO_CAM_XXL_CYFLEX_POCKET_TOOL_NB	By default 140.	Default tool number assigned to pockets.
ZX_ZWOO_CAM_XXL_CYFLEX_SLOT_MILLING_TOOL_NB	By default 101.	Default tool number assigned to slots/folds performed with a mill.
ZX_ZWOO_CAM_XXL_CYFLEX_SLOT_SAW_TOOL_NB	By default 101.	Default tool number assigned to slots/folds performed with a saw.
ZX_ZWOO_CAM_XXL_CYFLEX_MOULDING_TOOL_NB	By default 101.	Default tool number assigned to mouldings.

Update of machining faces

When a project contains parts whose machining is in **Optimized** mode, it is possible, during exports to the **machining interfaces** and **TopSolid'WoodCam**, to update these machining faces according to the operations on the part.

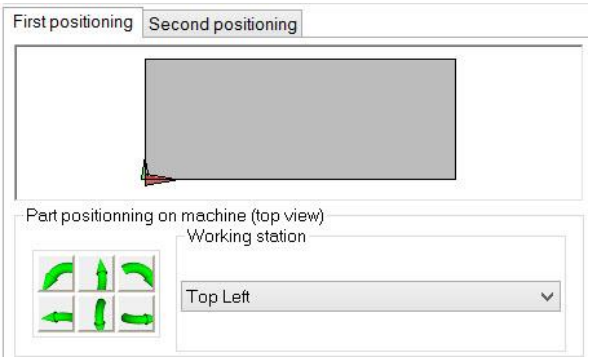
This update option is now available for the two sawing-up interfaces which also deal with machining operations: **Ardis** and **EnRoute**.

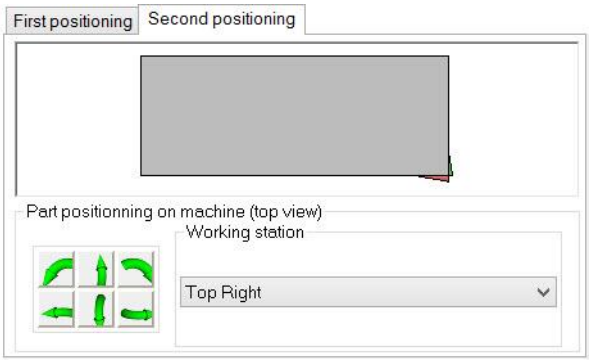
Note: This option for updating machining faces is only available if at least one part of the assembly is in **Optimized** machining mode.



Machining stop export

The machining stops defined when defining a part are now exported to the machining interfaces.





The name of the machining stop defined from **Tools** > **Options** > **TopSolid'Wood configuration** > **Working stations configuration** is exported to the interface.
If a name and a designation are specified in the machining stop, the designation will be displayed in the part definition and the name will be exported.

In interfaces where two machining files can be generated (**WoodWop**, **Xilog**, **NCHops**), only the name of the stop corresponding to the file machining positioning is exported.
If two machining files are generated for the part, the machining stop name of the first positioning is exported in the first file and the machining stop name of the second positioning is exported in the second file.

WoodWop

Machining stops are exported as comments on the line #33 of the **mpr** file.

First machining file

```
1  [H
2  VERSION="6.0"
3  MAT="HOMAG"
4  OP=1."
32 KM="X-=;X+=;Y-=;Y+=;"
33 KM="Top Left"
```

Second machining file

```
1  [H
2  VERSION="6.0"
3  MAT="HOMAG"
4  OP=1."
32 KM="X-=;X+=;Y-=;Y+=;"
33 KM="Top Right"
```

Xilog

Machining stops are exported in the **xxl** file header.

Note: The name of the exported machining stop must correspond to a work area in Xilog. Otherwise, the file may not be read in **Xilog**.
Names of machining stops allowed in **Xilog**: **A**, **B**, **C**, **D**, **AB**, **BA**, **CD**, **DC**, **AD**, **DA**.

NC Hops

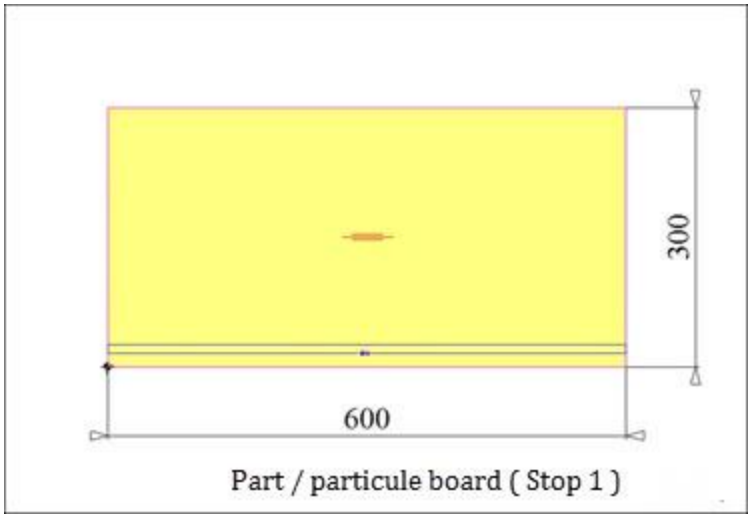
Machining stops are exported in a variable of the **hop** file.

Note: As the **hop** file contains a new variable, the **NC Hops** import macros must be updated.

The name of the variable in which the machining stop is exported depends on the language set in **TopSolid'Wood**.

- **French**: **_PosteTravail**
- **English**: **_WorkingStation**
- **German**: **_Arbeitsstation**

This variable can then be used in **NC Hops** like any other variable. The new macros allow the machining stop to be displayed in brackets after the part material.



Ardis

As all machinings of all parts are sent to the same **Ardis** file, the two machining stops of each part are contained in the generated **xml** file. Both of these are exported in the columns:

- First stop: **PartCNCOriginMain - Origin main**
- Second stop: **PartCNCOriginSecond - Origin secondary**

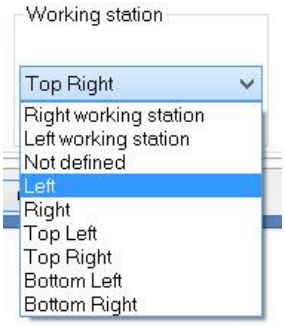
To ensure that machining stops can be properly read in **Ardis**, the **TopSolid'Wood** stop names and designations must be set as follows:

TopSolid'Wood stop name	TopSolid'Wood stop designation	Reading in Ardis
0	Undefined	**Undefined**
1	Left	**Left**
2	Right	**Right**
3	Left front	**Left front**
4	Right front	**Right front**
5	Left rear	**Left rear**
6	Right rear	**Right rear**
Other value	/	**Undefined**

Configuration of stops in the options:

Name	Designation	Work station 1	Work station 2
0	Not defined	-	-
1	Left	X	-
2	Right	-	X
3	Top Left	-	-
4	Top Right	-	-
5	Bottom Left	-	-
6	Bottom Right	-	-

Stop selection from the part definition:

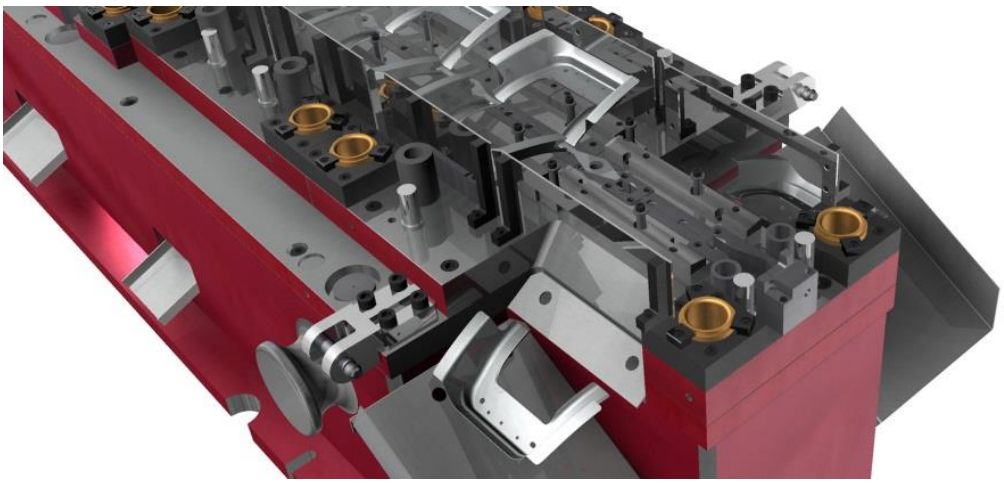


Result in **Ardis**:

Parts (F9)

	Ref	Origin secondary	Origin main
	Piece1	Right rear	Left rear
	Piece2	Right front	Left front
	Piece3	Right	Left
	Piece4	Undefined	Undefined

What's new in TopSolid'Progress v6.16

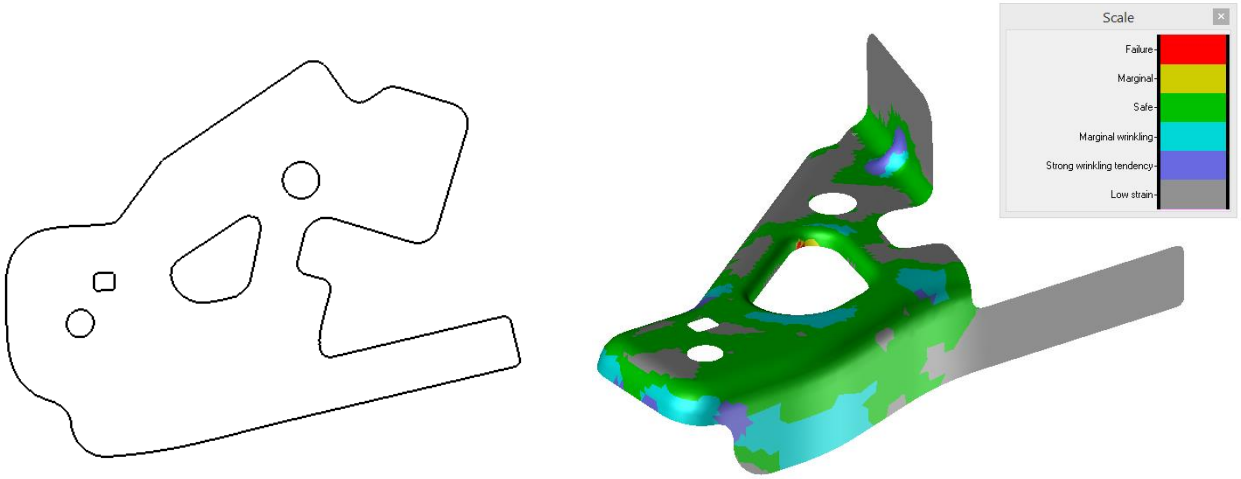


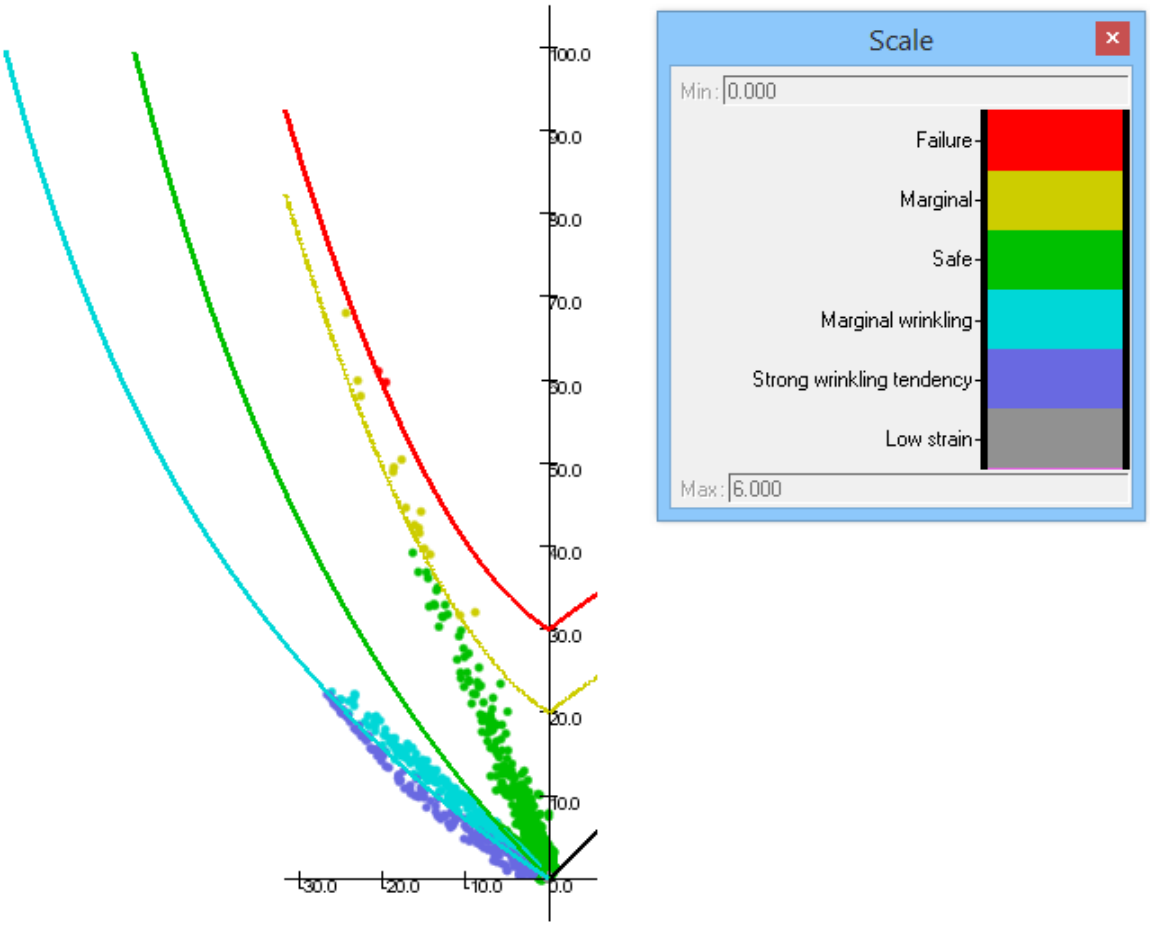
This section describes the new features in the version **6.16** of **TopSolid'Progress**.

Part preparation

Analyze unstamping results

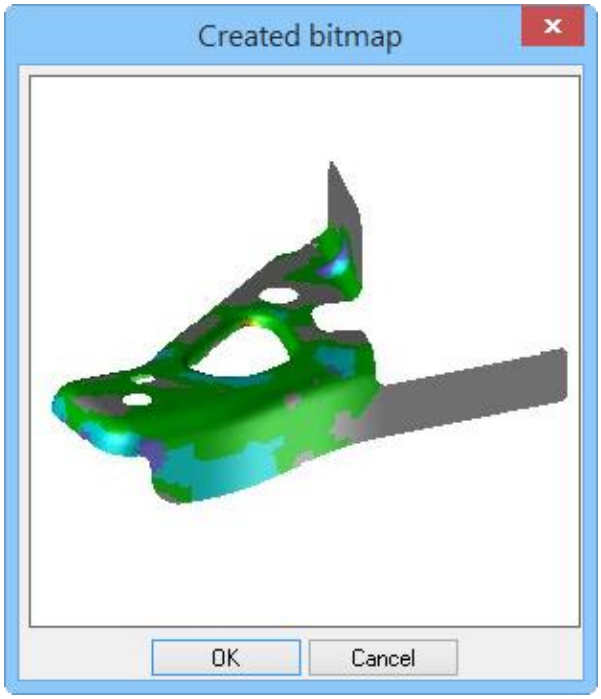
A new type of analysis is now available: the **Forming Limit Diagram** (or **FLD**).
It takes into account the data resulting from the analysis of **minor strains**, **major strains** and **safety zones**, and displays them as a diagram of points.
The abscissa axis represents the minor strains and the ordinate axis represents the major strains.





The various colored curves identify the areas of characteristic behaviors where the analysis points (the part's meshing nodes) are displayed in the corresponding color.
This diagram allows you to quickly evaluate the part's formability by immediately viewing the nature and distribution of the behaviors the part will be subjected to during the forming process.

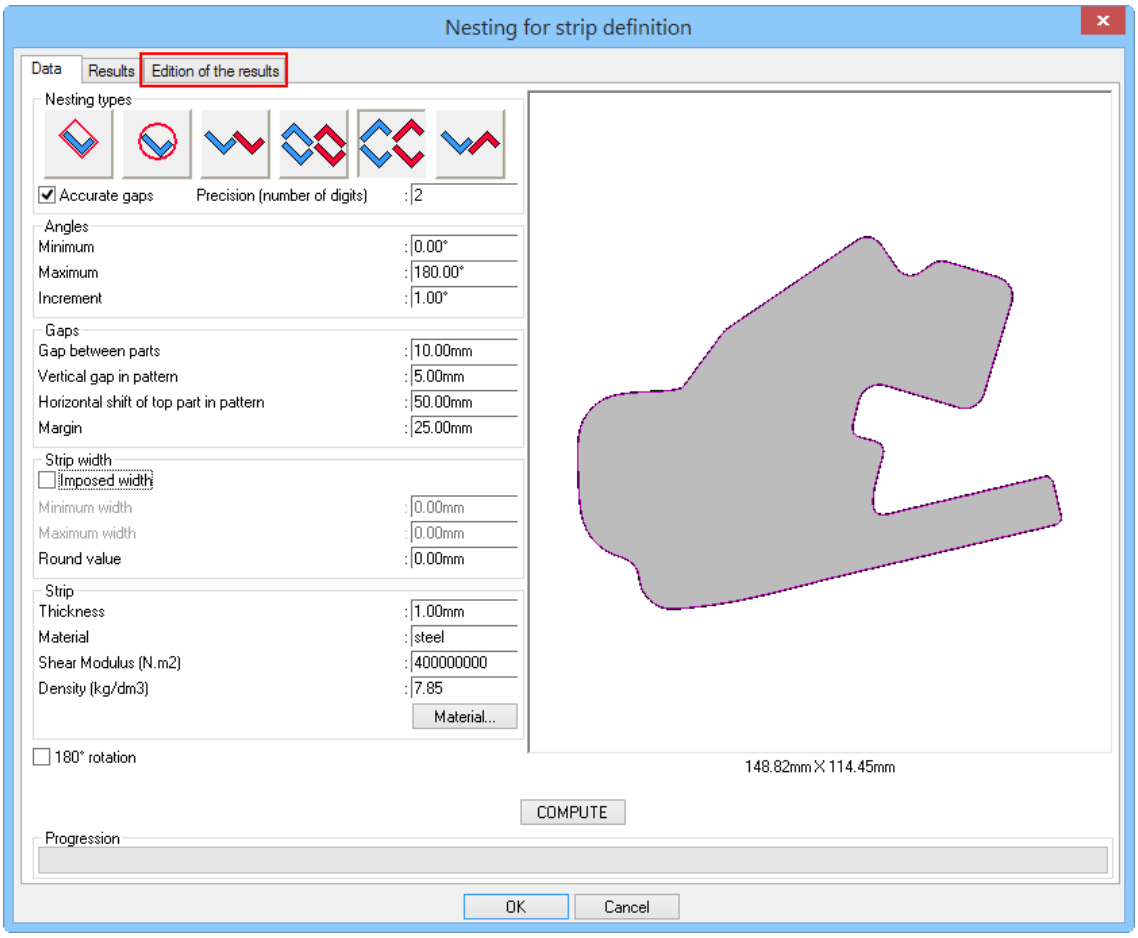
In addition, a new option allows you to capture images when editing the currently viewed analysis. It is possible to adjust the capture settings (size, resolution) and specify whether the scale is to be embedded in the image. Once captured, a preview of the resulting image is provided.



When validated, the image is automatically saved in *.jpg format in a subdirectory of the part's directory.
These images are then available to users in the report creation stage of the **Nesting** command (see next chapter).

Nesting

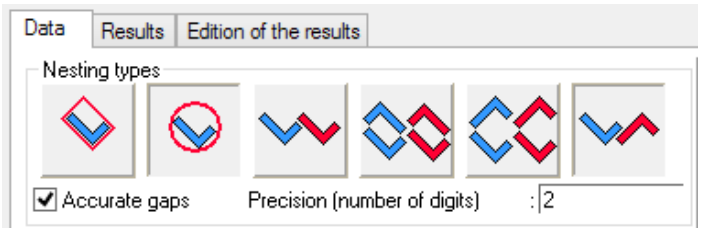
Many improvements have been made to this command.
The overall dialog has been reorganized and includes a new tab for editing the nesting results.



Data tab

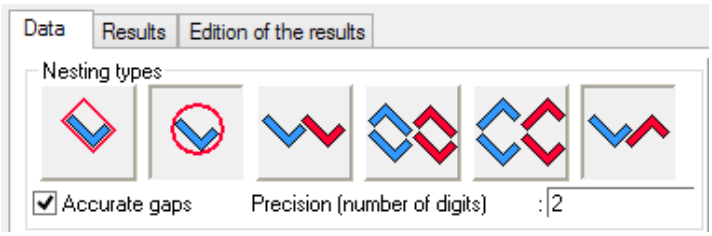
Two new nesting types have been added:

- **Minimum circle**
- **With symmetric part (horizontal)**



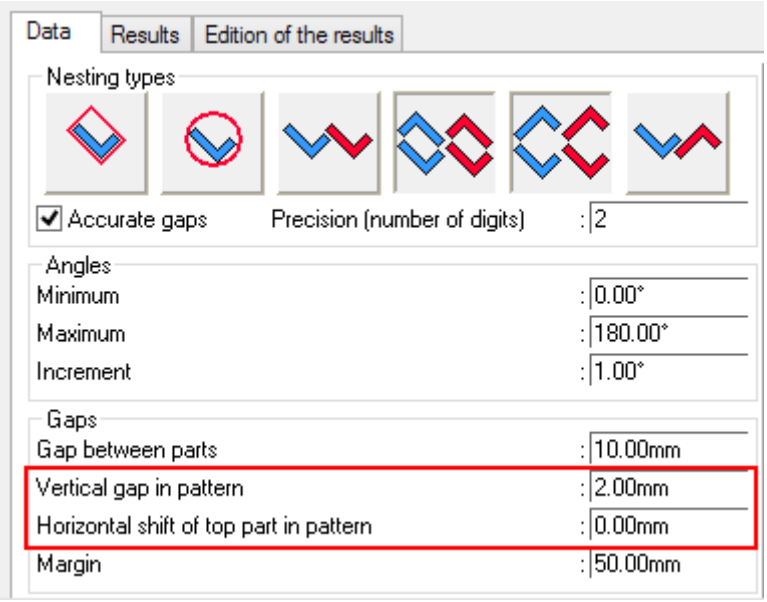
The **Minimum circle** nesting type computes the part's minimum bounding circle and applies the step and strip width values resulting from this computation.
The **With symmetric part (horizontal)** nesting type computes, for each requested part orientation, the pattern of the part and its symmetric part offset by the minimum possible distance in the strip progressive direction. From this pattern, it applies the step and strip width values resulting from the computation.

A new **Accurate gaps** option allows you to compute the nesting results by fully respecting the requested gaps between parts (based on the precision specified by the user).

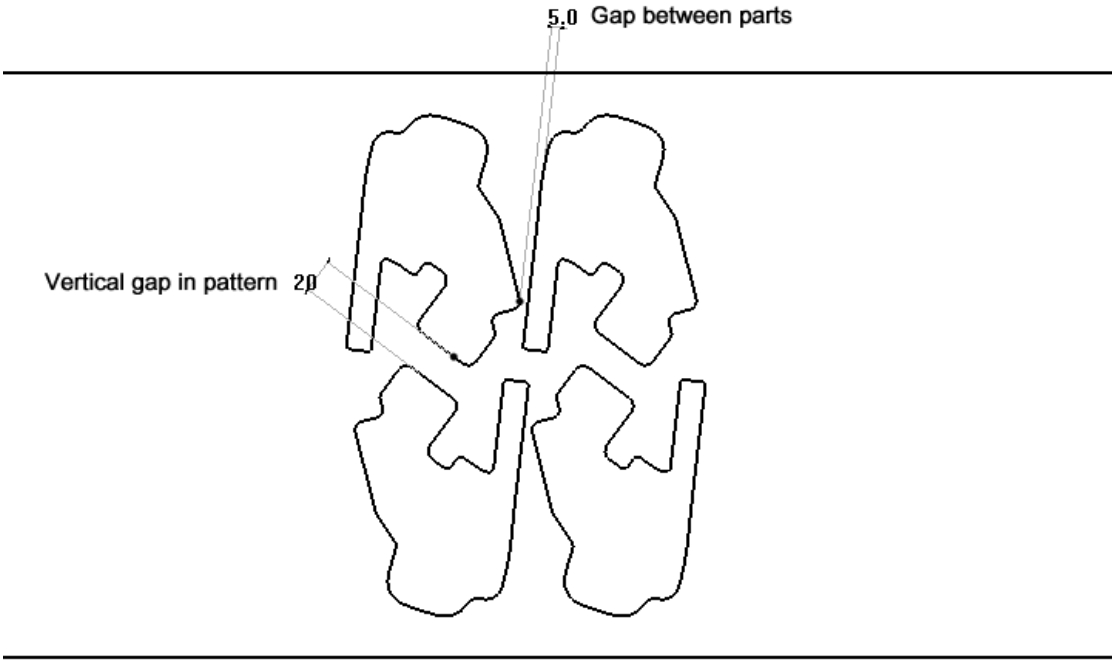


This option uses a new calculation mode for the distances between parts, based on the parallelization of the part's exact external profile (the polygonized profile is used when this option is not selected), and ensures accurate steps, strip width and part positions during calculations and drafting.
When this option is activated, it is also possible to specify an increment angle value less than 1°.

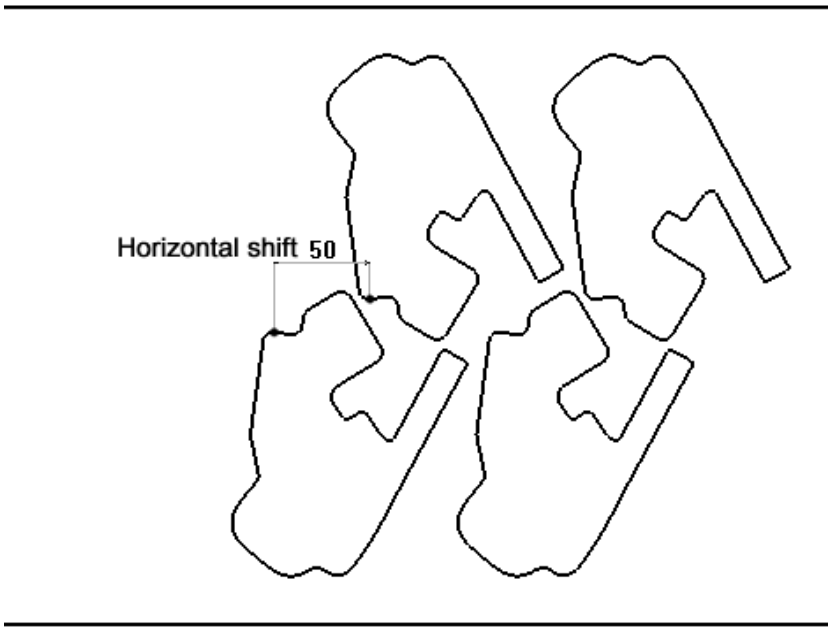
In the **Gaps** section, two new parameters have been added: **Vertical gap in pattern** and **Horizontal shift of top part in pattern**.



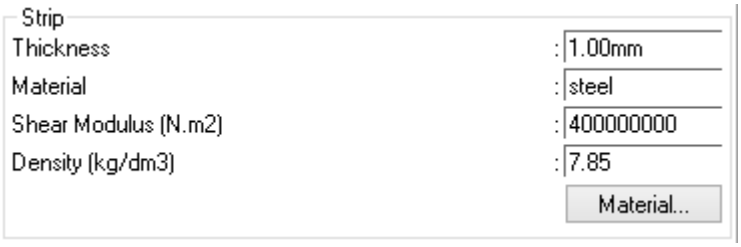
These two parameters are only active if the **Accurate gaps** option and the **With rotated part** and/or **With symmetric part** nesting types are required.
The **Vertical gap in pattern** parameter allows you to define the gap between the top and bottom parts that make up the pattern. This gap may therefore be different from the **gap between parts** which is used to calculate the position of parts in the strip progressive direction.



The **Horizontal shift of top part in pattern** parameter allows you to specify a translation for positioning the top part of the pattern which will be used for the nesting computation.

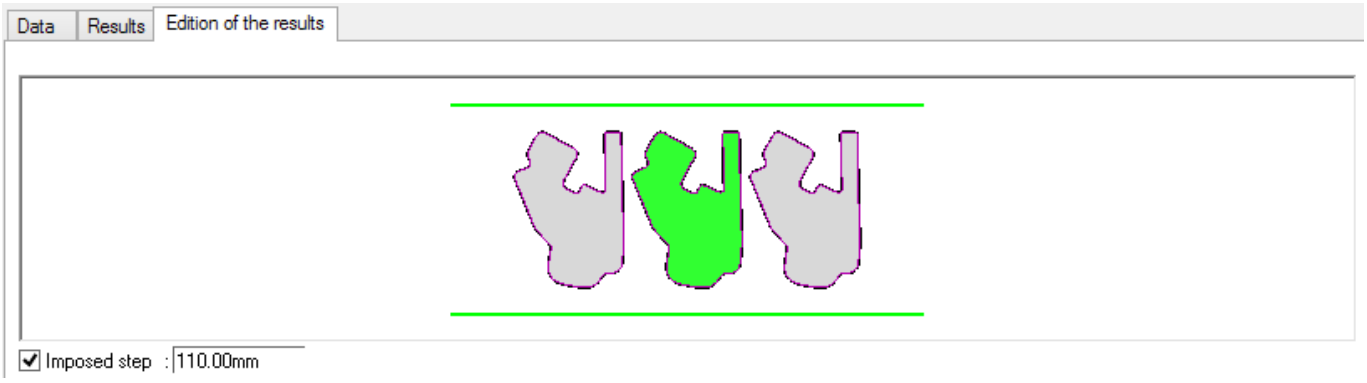


In the **Strip** section, the material data are now initialized from the part used to generate the unfolding. However, it is always possible to define and impose these data manually.

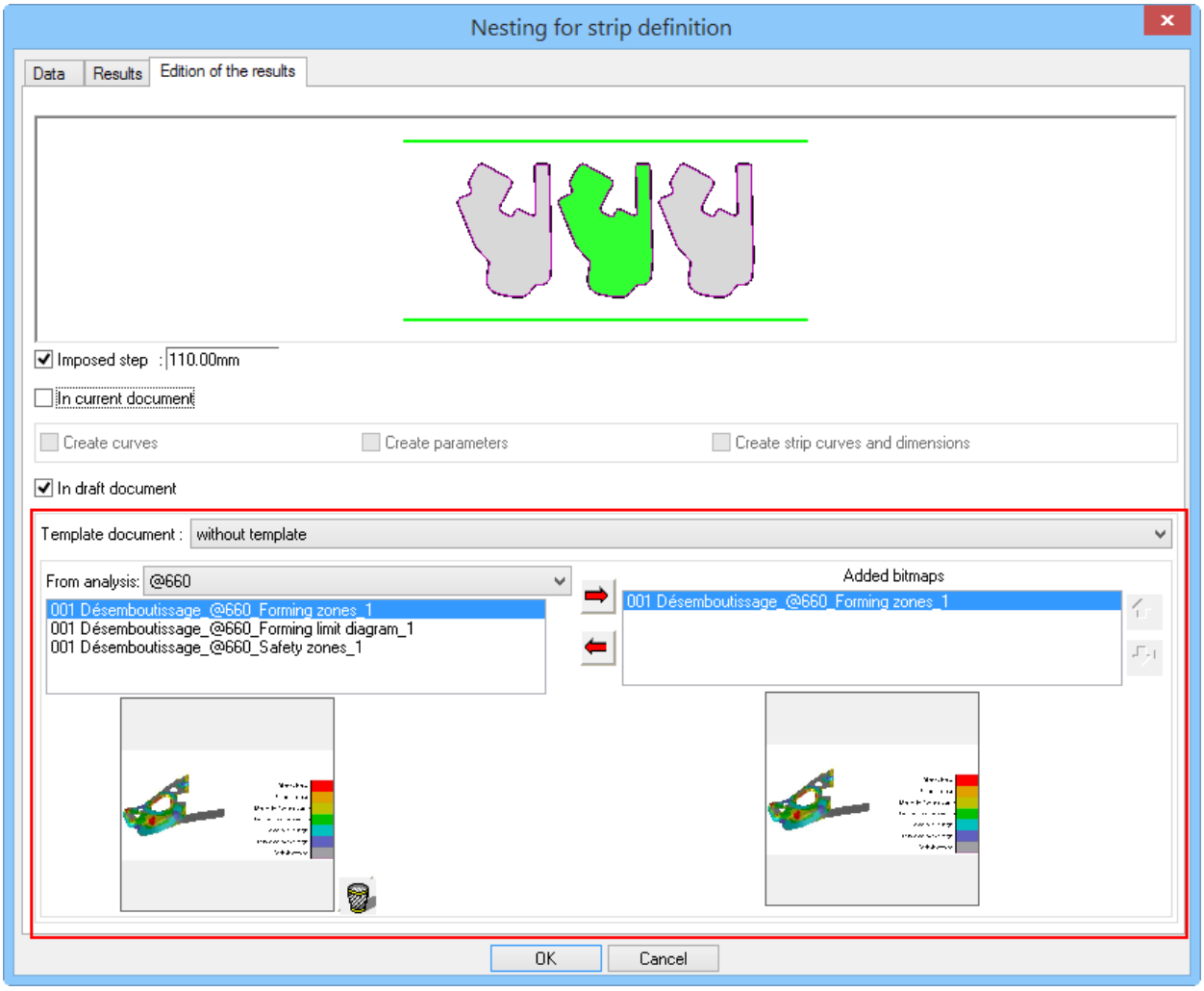


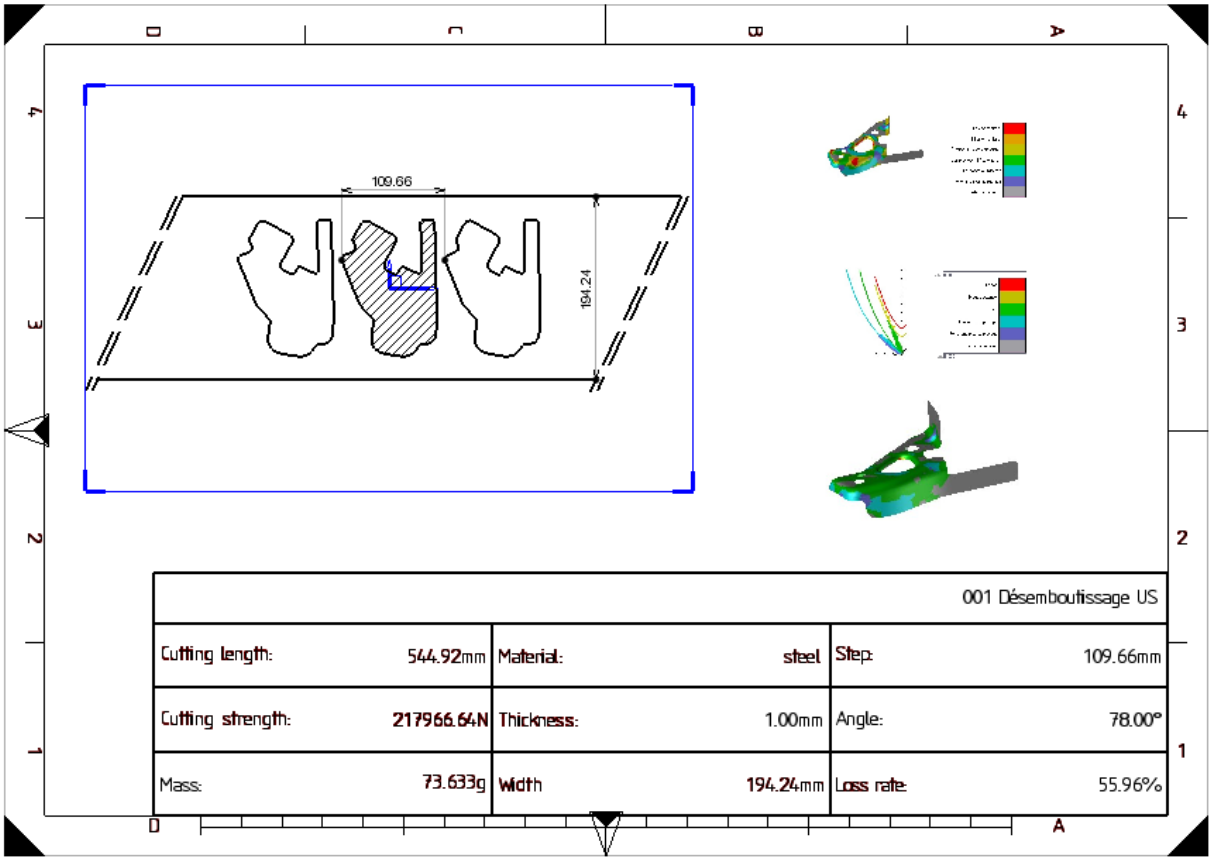
Edition of the results tab

It is now possible to set a given step between parts thanks to the **Imposed step** option. It allows the use of a rounded step value rather than that calculated by the application.



In the lower area of the dialog box, you can now select the images to be inserted in the report in Draft format. The available images result from the captures that can be taken in the **Analyze unstamping results** command (see previous chapter). The selected images will be inserted in the Draft document, instead of the bitmaps inserted in the selected template (**Tools** > **Bitmap** in the Draft document). In the template, these bitmaps must be identified using the **Progress die** > **Nesting report configuration** command of the Draft document. You only need to select each bitmap and specify an index value to use in the list of selected bitmaps. A default Draft template document for the report is provided (in English).

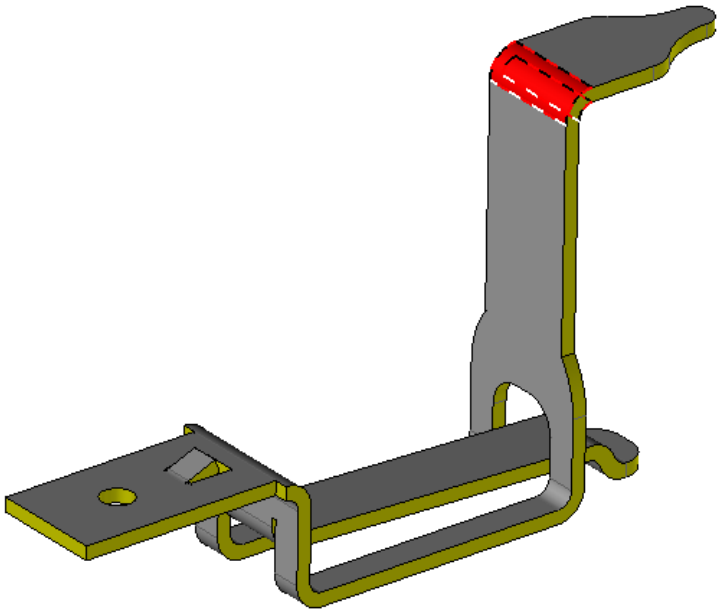

Nesting for strip definition
Data
Results
Edition of the results
Imposed step : 110.00mm
In current document
Create curves
Create parameters
Create strip curves and dimensions
In draft document
Template document : without template
From analysis: @660
001 Désemboutissage_@660_Forming zones_1
001 Désemboutissage_@660_Forming limit diagram_1
001 Désemboutissage_@660_Safety zones_1
Added bitmaps
001 Désemboutissage_@660_Forming zones_1
OK
Cancel

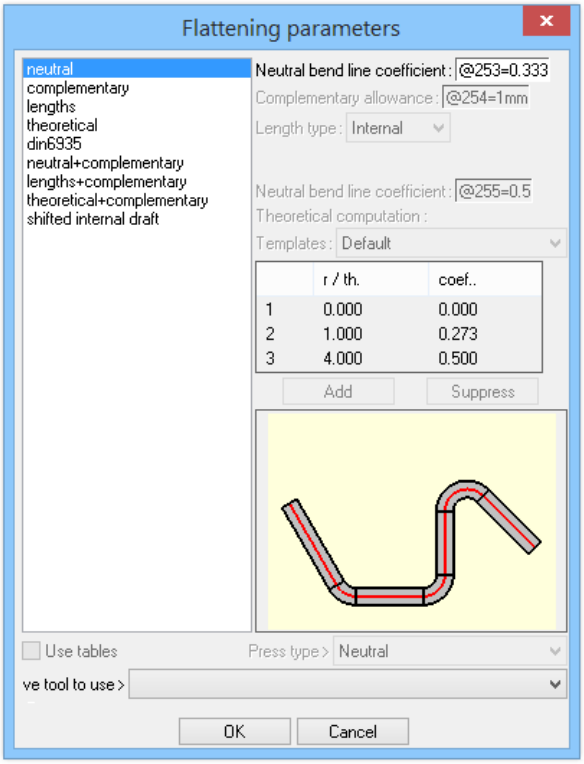

109.66
194.24
001 Désemboutissage US
Cutting length: 544.92mm
Material: steel
Step: 109.66mm
Cutting strength: 217966.64N
Thickness: 1.00mm
Angle: 78.00°
Mass: 73.633g
Width 194.24mm
Loss rate: 55.96%

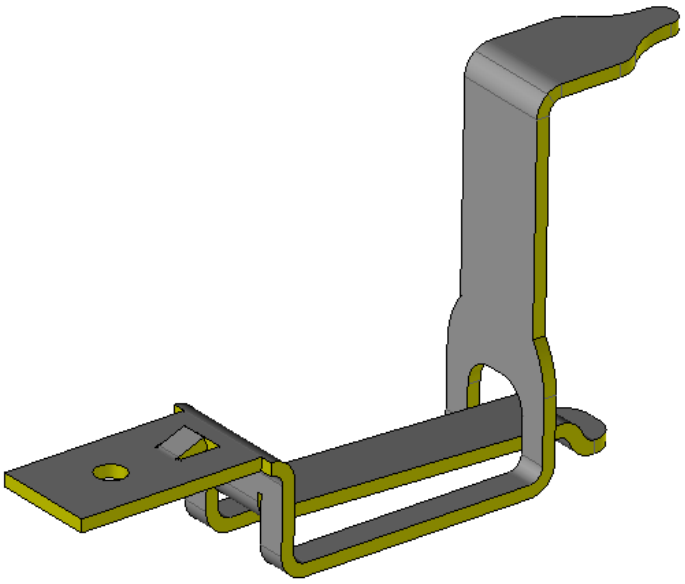
In the Draft document templates of the nesting reports, it is now possible to get the cutting length and cutting strength parameter values.
These values can be accessed using the **Progress die** > **Nesting report configuration** command of the Draft document.

Bend division

The bend division line can now be positioned by specifying a value corresponding to the flat bend length, and taking into account the bending allowance computation rules.
After selecting the new **division mode = GIVEN LENGTH ON FLAT BEND** and specifying the **Distance on flat bend** value, the system will ask for the bending allowance computation method to use in order to calculate the portion of angle this length will match on the bended part.







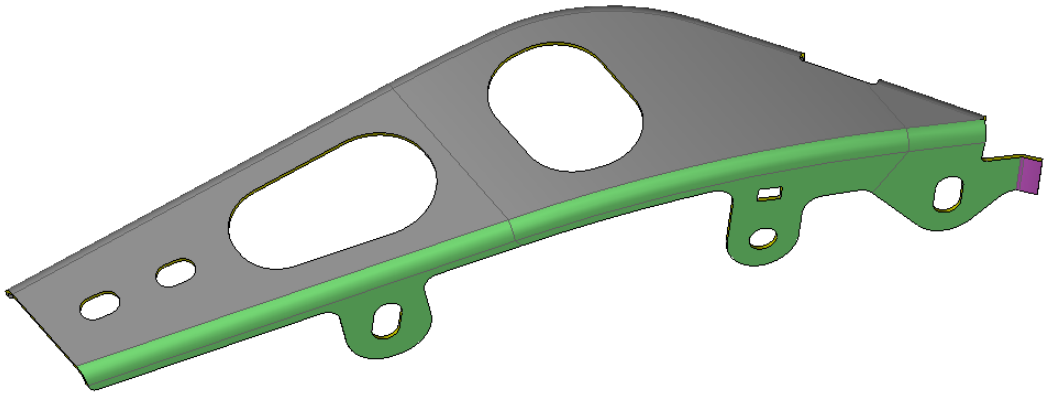
Body management

In the **DIVIDE** mode, the smooth edges generated by the trimming of bodies are now kept.

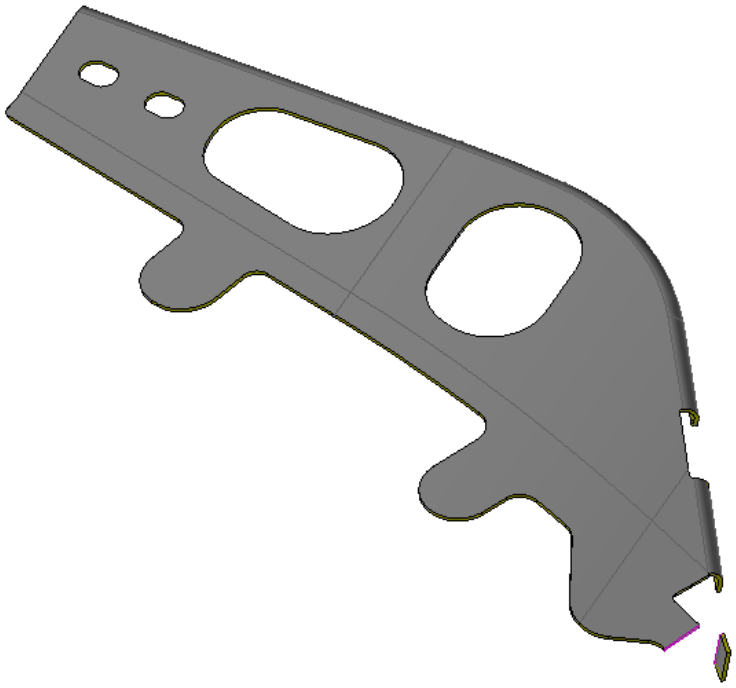
In the **MOVE BODIES** mode, a new **POSITION BODY** submode has been added.
This new positioning mode is particularly useful for repositioning bodies resulting from a flange unfold operation whose some subareas have been ignored and left in place (see next chapter).

Flange unfold

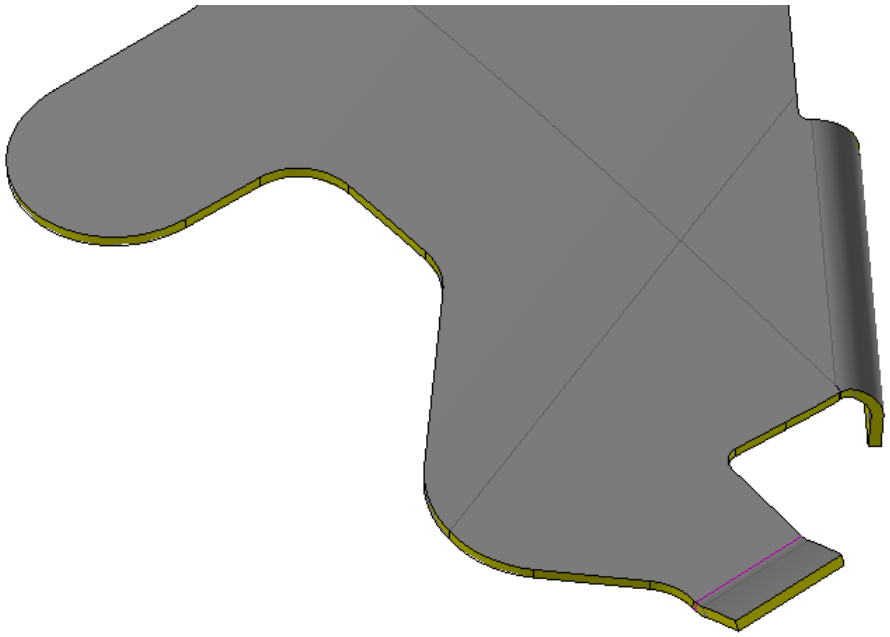
It is now possible to define a sub-area to be ignored within the area to unfold. The geometries of this part's sub-area will be left in place. It will be up to the user to reposition this isolated body manually on the unfolded part.
A typical use of this configuration concerns the small bended tabs hooked to a forming that separates them from the main body of the part.
For example, for the part shown below, the purple area must be treated by bending before the flange unfold operation is performed on the green area.



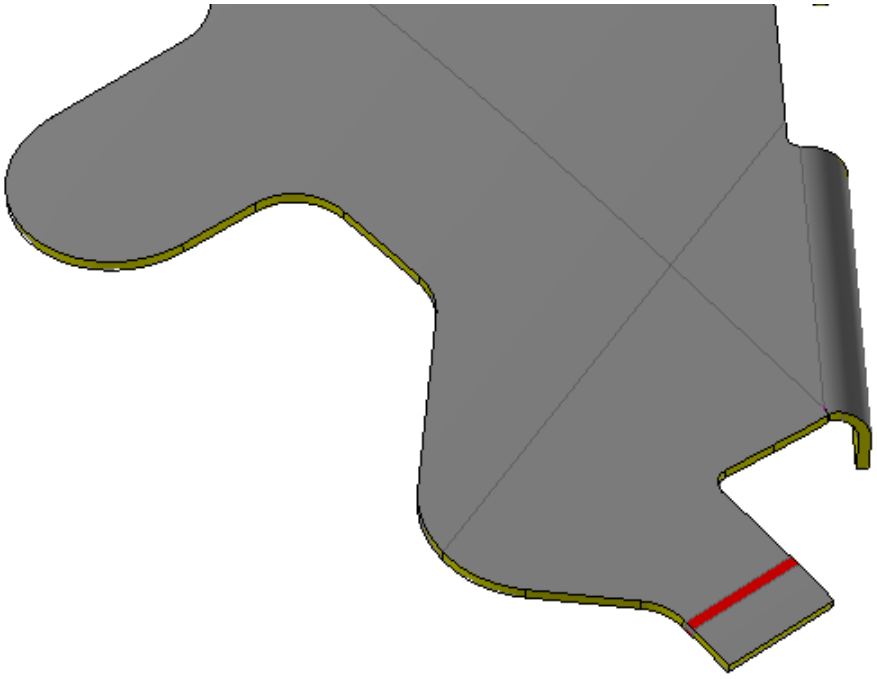
With this new functionality, the flange unfold operation can be done on the green area while leaving the purple area in its original position in order to obtain the following result.



Thanks to the **Bodies management** > **Move bodies** > **Position body** command, it is then easier to reposition the body that was left in place.



Using the same command in **MERGE** mode, these two bodies are merged into a single body.
It is then possible to treat the bend using the **Unbend for intermediate stage** command.



For more complex configurations, the definition of the transition area between the repositioned body and the body resulting from the flange unfold operation may be more complicated. It could be necessary to apply other design methods, including the surface modeling commands.

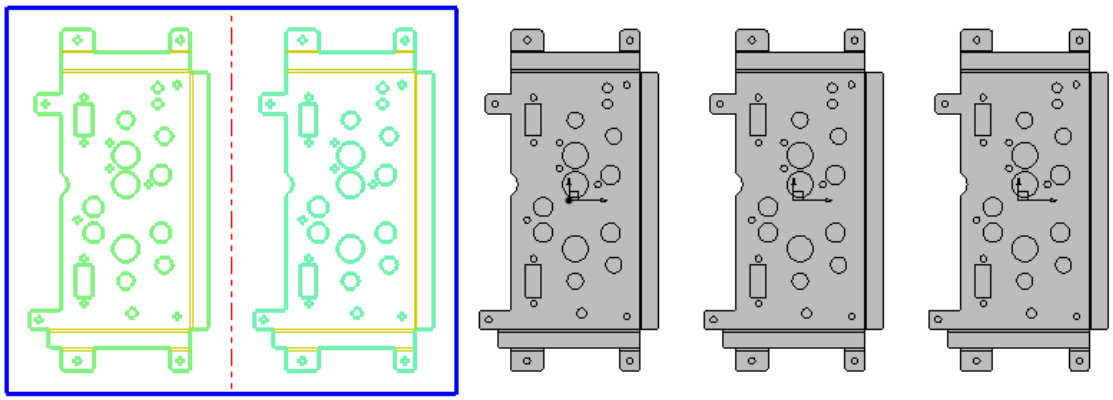
Deep drawn wizard

Ten lines are now available in the calculation form of the number of drawing steps.

Strip design

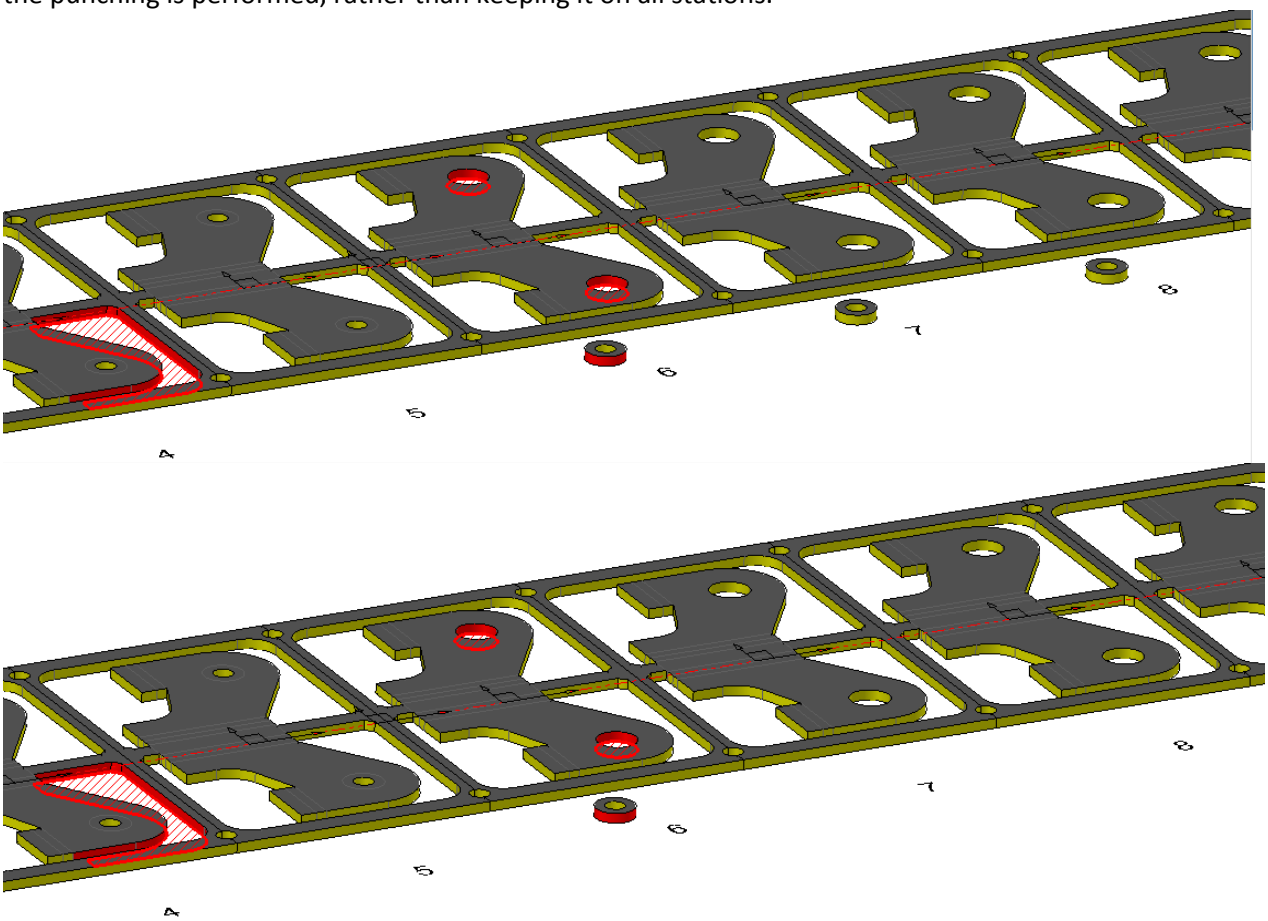
Create strip

In **No strip of matter around part** mode, it is now possible to fill the part's internal holes by checking the **Fill holes in part** option.



Punching

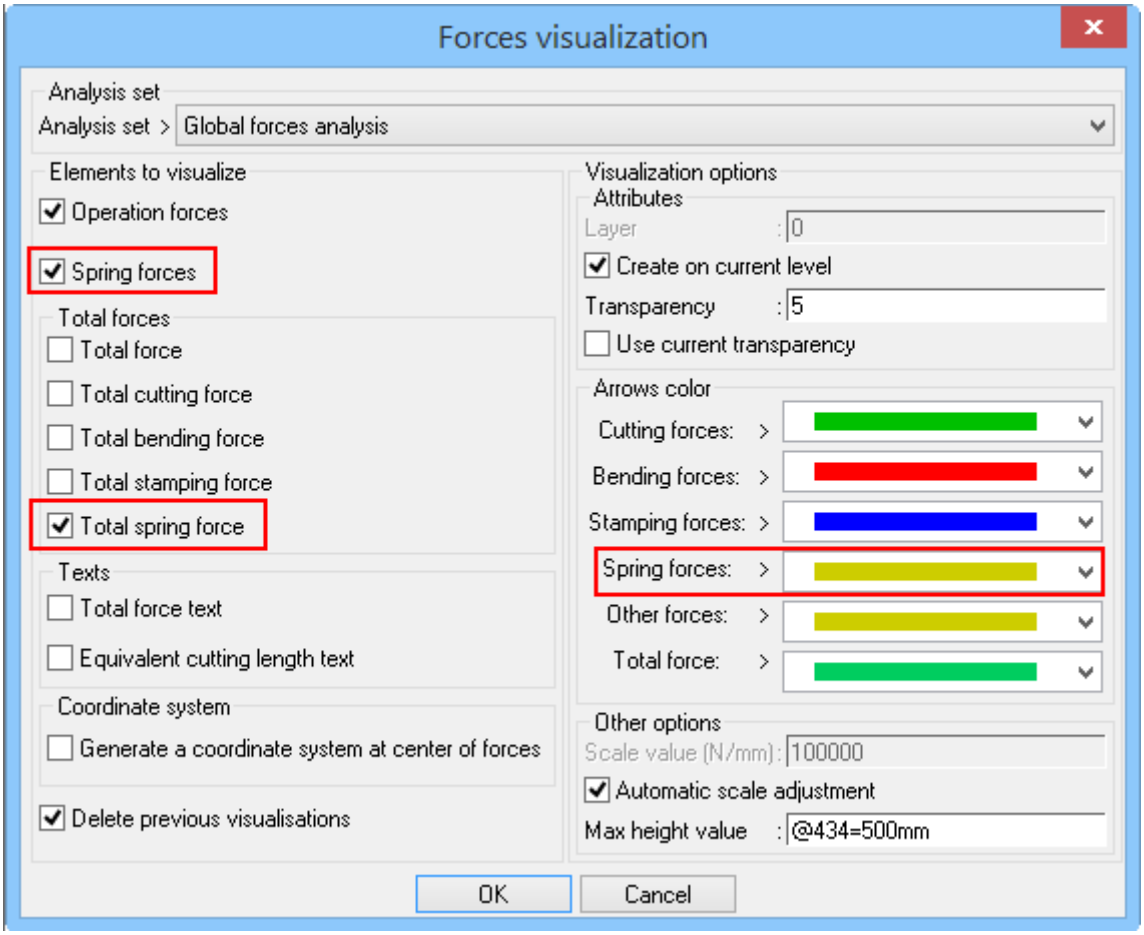
Different loss treatment modes are now available. This makes it possible to keep the loss only on the station where the punching is performed, rather than keeping it on all stations.



*Example of the same punching operation performed with the **Keep loss** option **on all stations** (top) and with the new option **on station only** (bottom).*

Force visualization

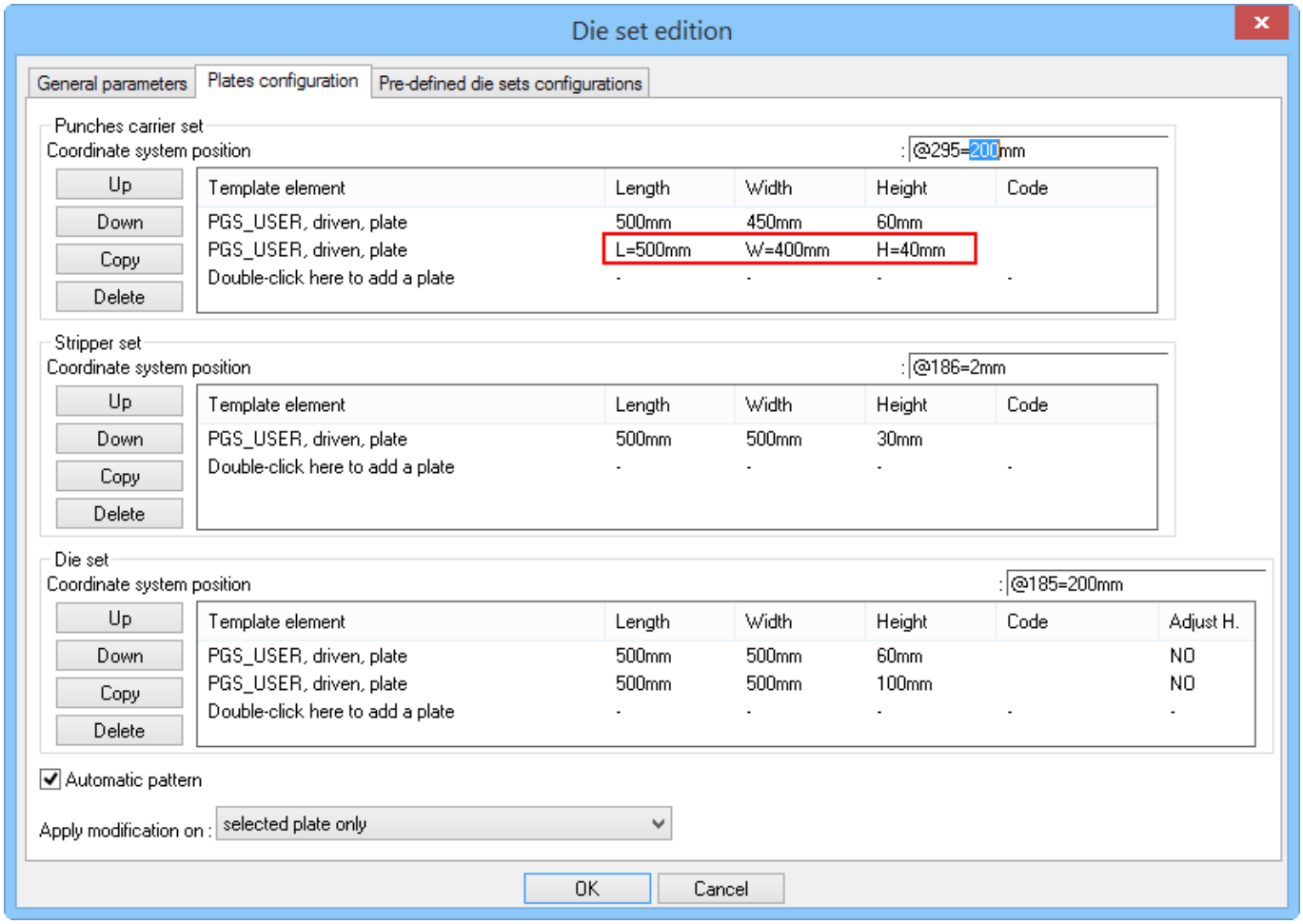
The forces generated by the springs can now be displayed.
For the computation, only the **Progress standard components that are identified as springs** are considered.
For your information, the compression stroke used to calculate the forces is given when creating the spring component.

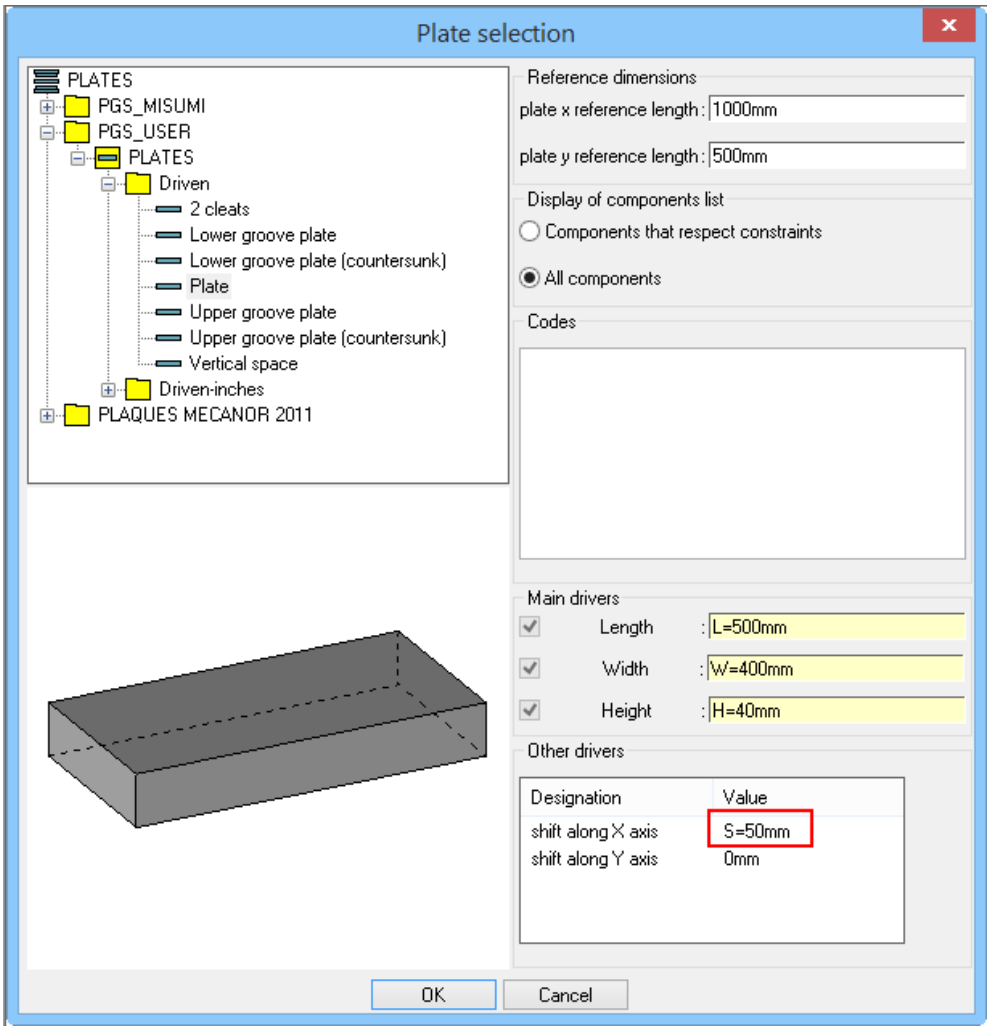


Die set

Create die set

It is now possible to use named parameters to specify the **main dimensions of plates** in the die set editing dialog box, or in the plate editing dialog box. Until now, it was only possible to use values to set these dimensions.
The same applies to the **other plate drivers** in the plate editing dialog.

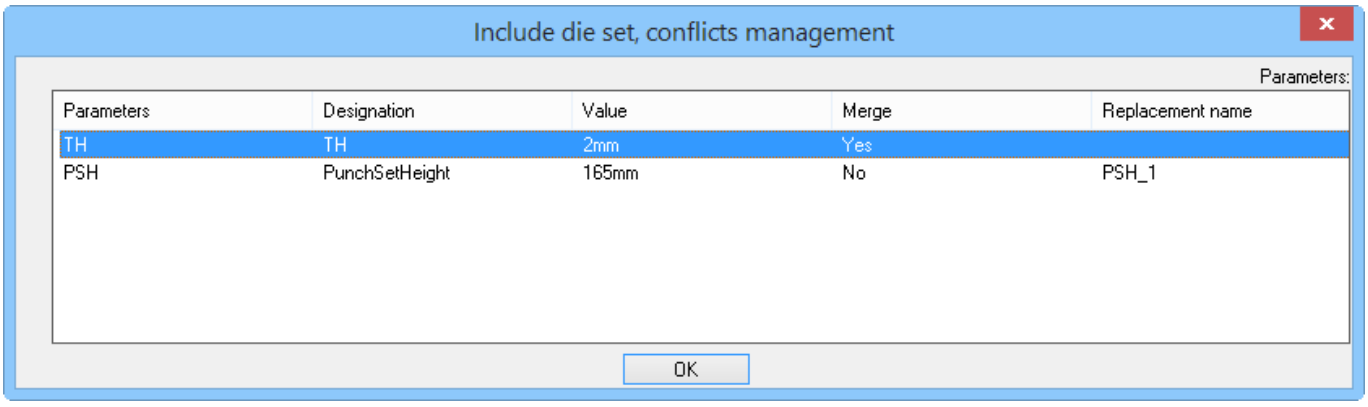




Consequently, in the **Copy plate parameters** command, now you will be asked if you want to **COPY VALUES** of parameters or if you want to **LINK NAMED PARAMETERS**.

Include die set, Include standard die set

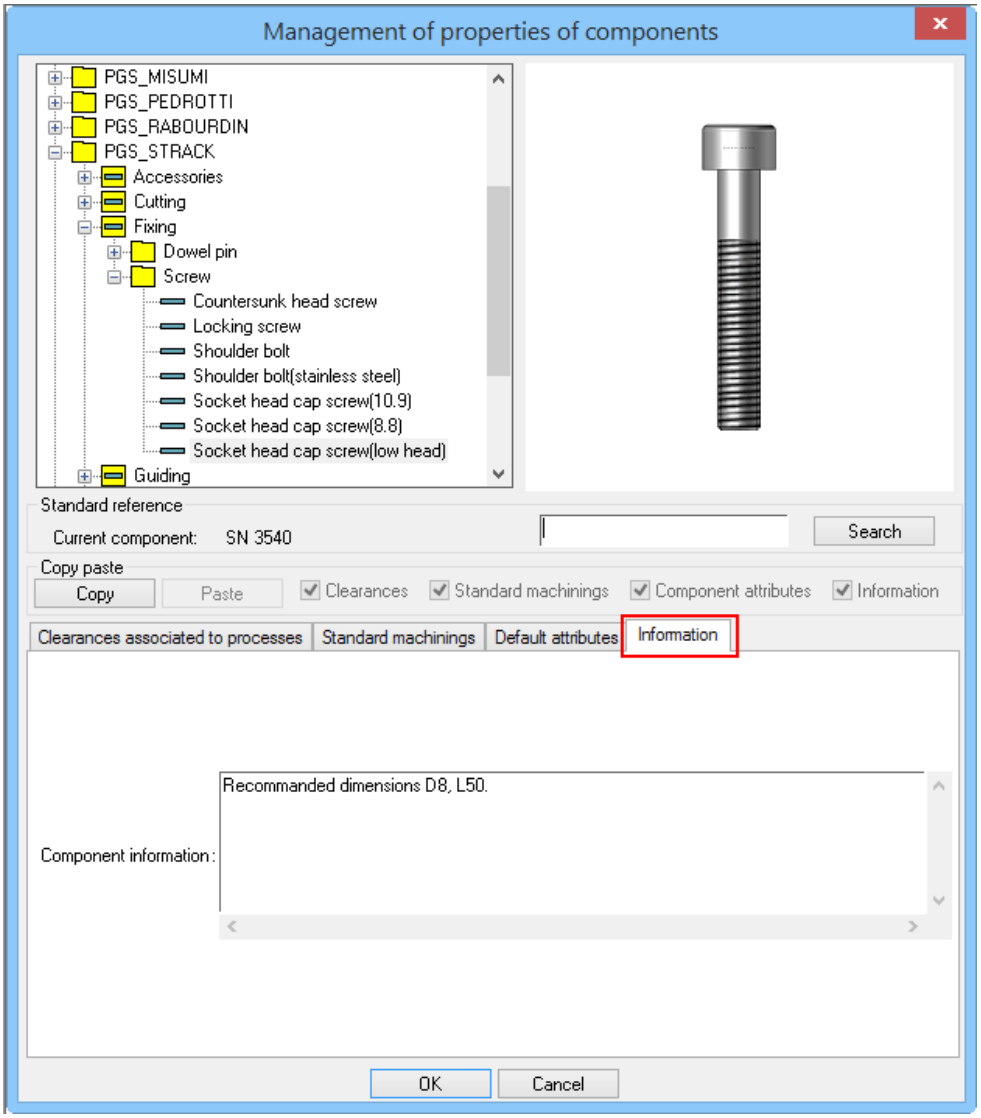
In these two commands, if the template document of the die set contains parameters whose name is already used in the destination document, the user can now choose to duplicate these parameters (by giving them a new name since a default name is proposed), or merge them with the existing parameters of the destination document.



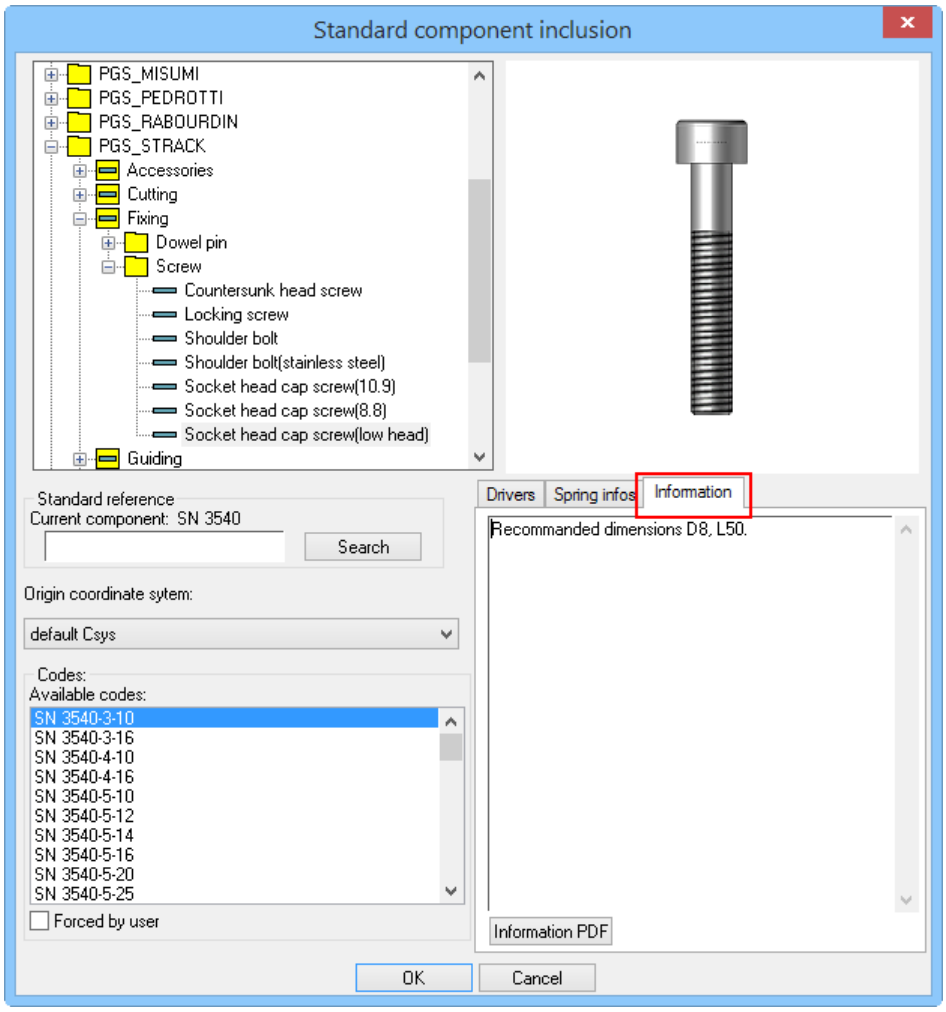
Components

Components > Configuration, *Components > Create*, *Component > Modify*

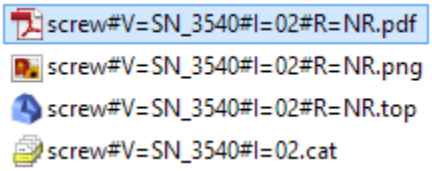
In the **Component configuration** command, a new **Information** tab allows you to associate an information text with each component.



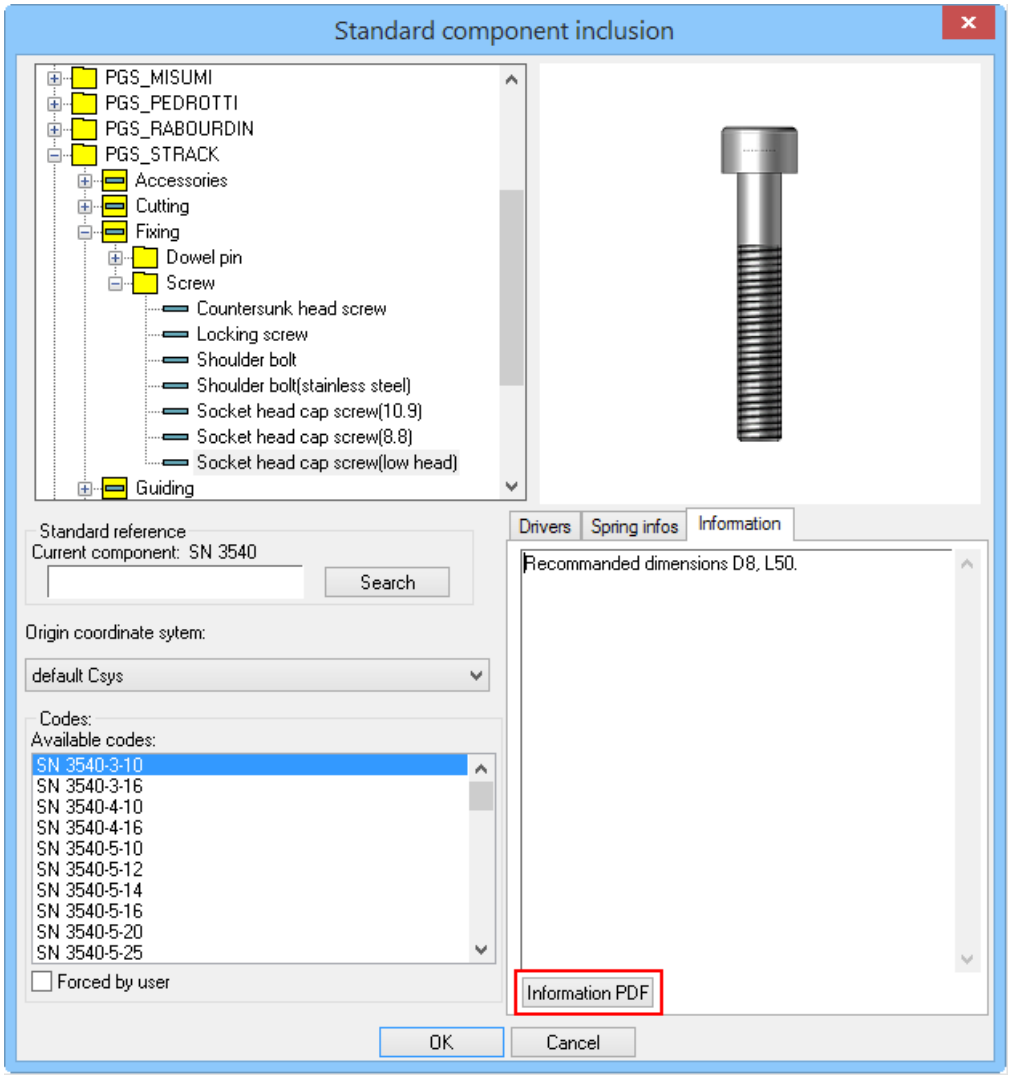
This text is shown in the component selection dialog box, as well as in the component modification dialog box.



Similarly, in component libraries, a PDF information file can be associated with each template.

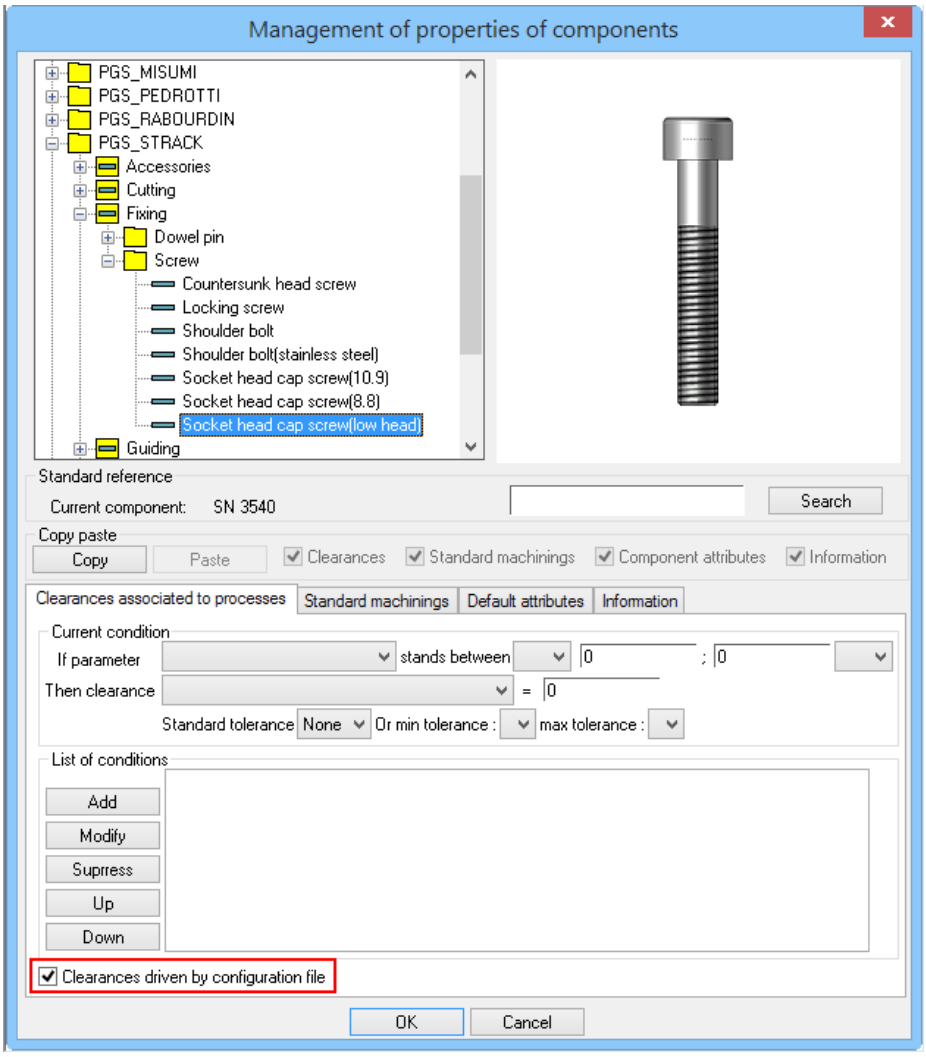


If such a file is found, it is possible to ask for it to be displayed in the component selection dialog box, and in the component modification dialog box by clicking the **Information PDF** button.

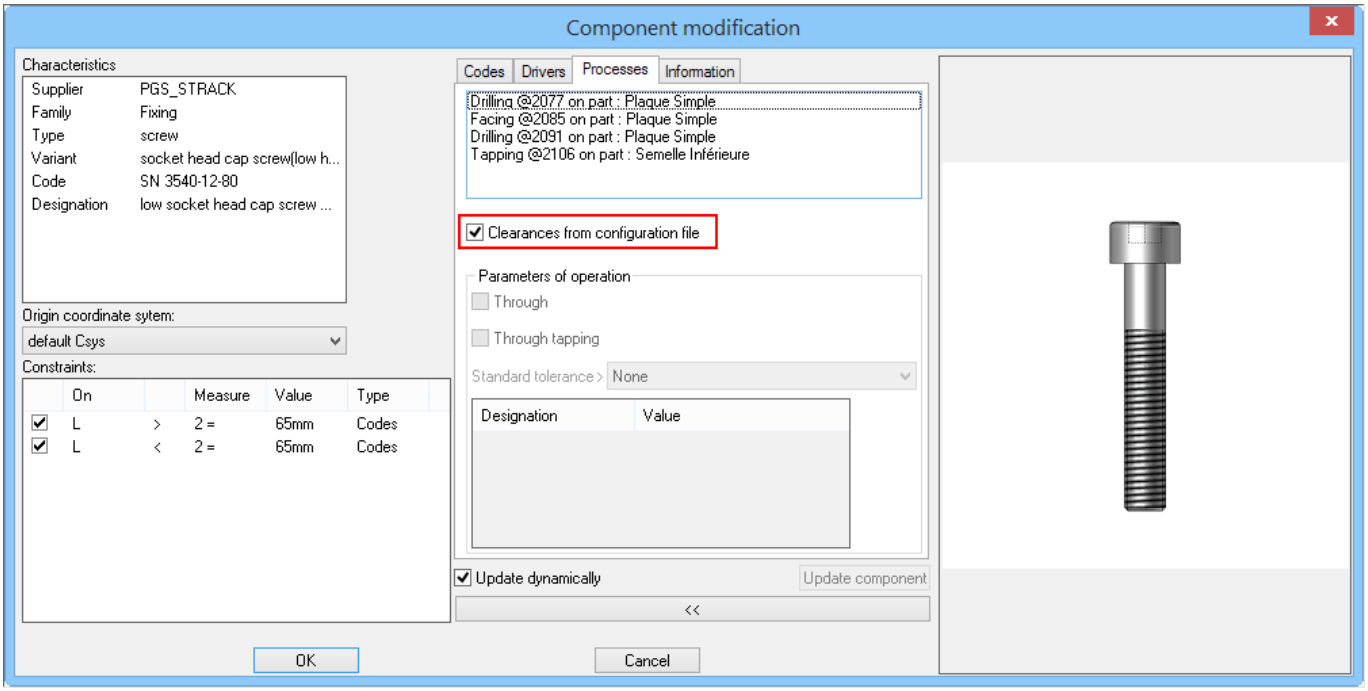


Component > Configuration , Driven clearances

For calculating the Progress component processes, it is possible to request the associative calculation of clearances when the component dimensions are modified.
This option can be activated individually, for each component template, via the dialog box for editing the Progress component configuration.



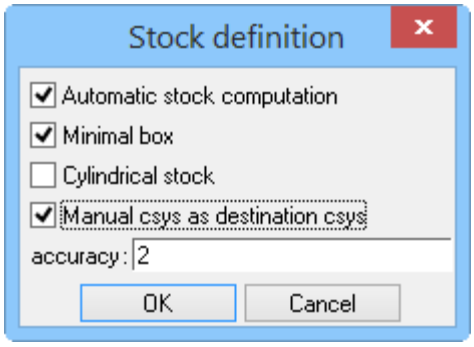
In addition, this option can be activated or deactivated for existing components via the dialog box for modifying the standard Progress components.



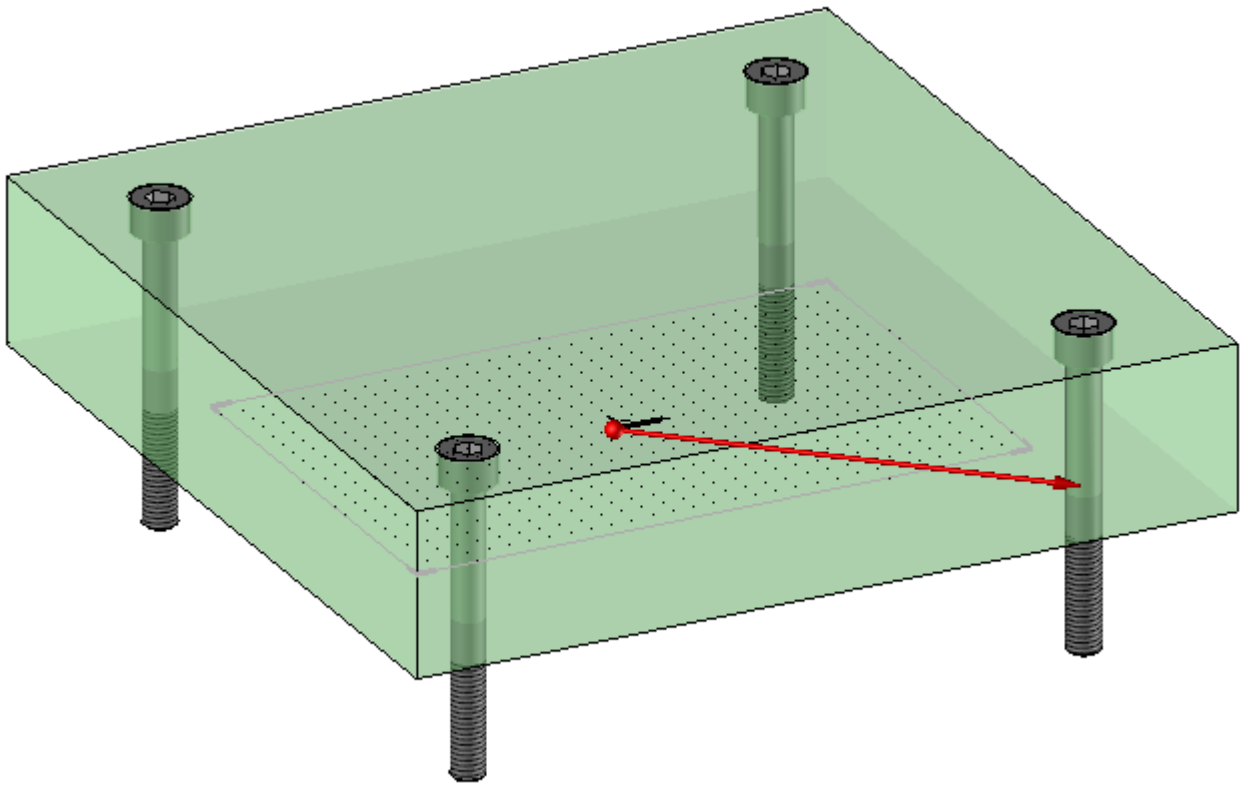
If this option is enabled, when changing the component dimensions, the clearances associated with the processes generated by the component will be recalculated based on the values given in the component's configuration file.

Progress die > Component definition > Stock definition 

The **Stock definition** command has been added to the Progress component definition commands available from a Design document.
This command provides access to two new features.
The **DEFINE DEFAULT STOCK PROPERTIES** option allows you to predefine the stock computation options for the current component.



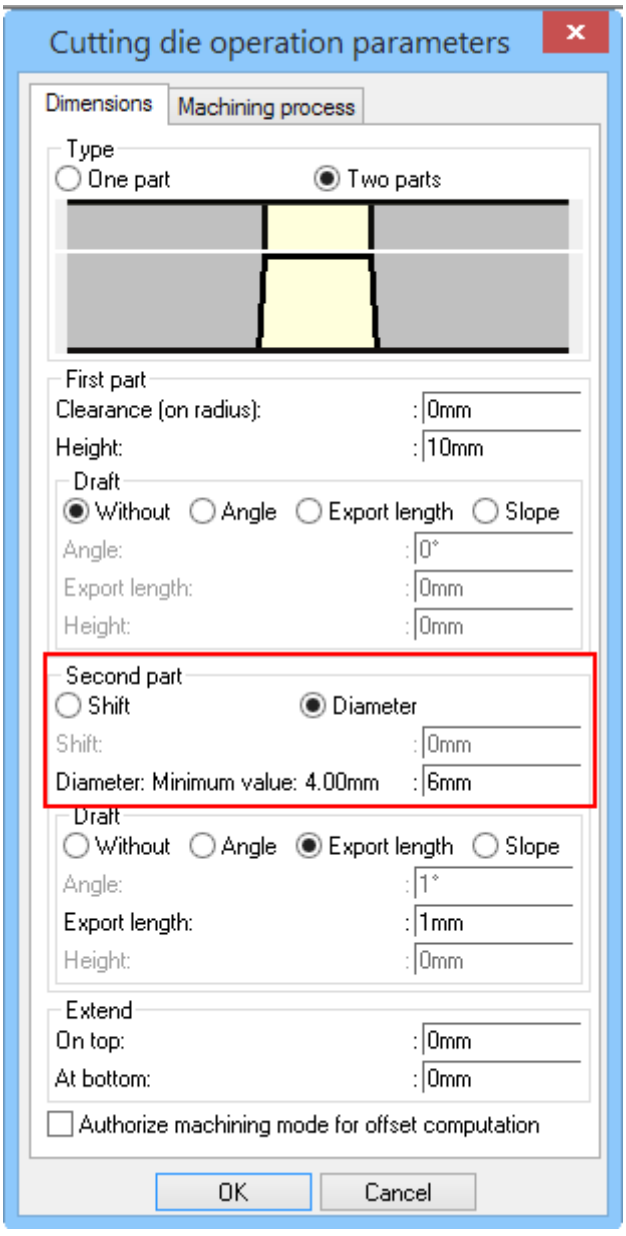
The **SHAPES TO IGNORE IN STOCK COMPUTATIONS** option allows you, for subassembly components, to identify the subcomponents that are to be ignored for the stock computation.
In the example below, it is typically used to ignore the screws.



Punches and dies

Cutting die 

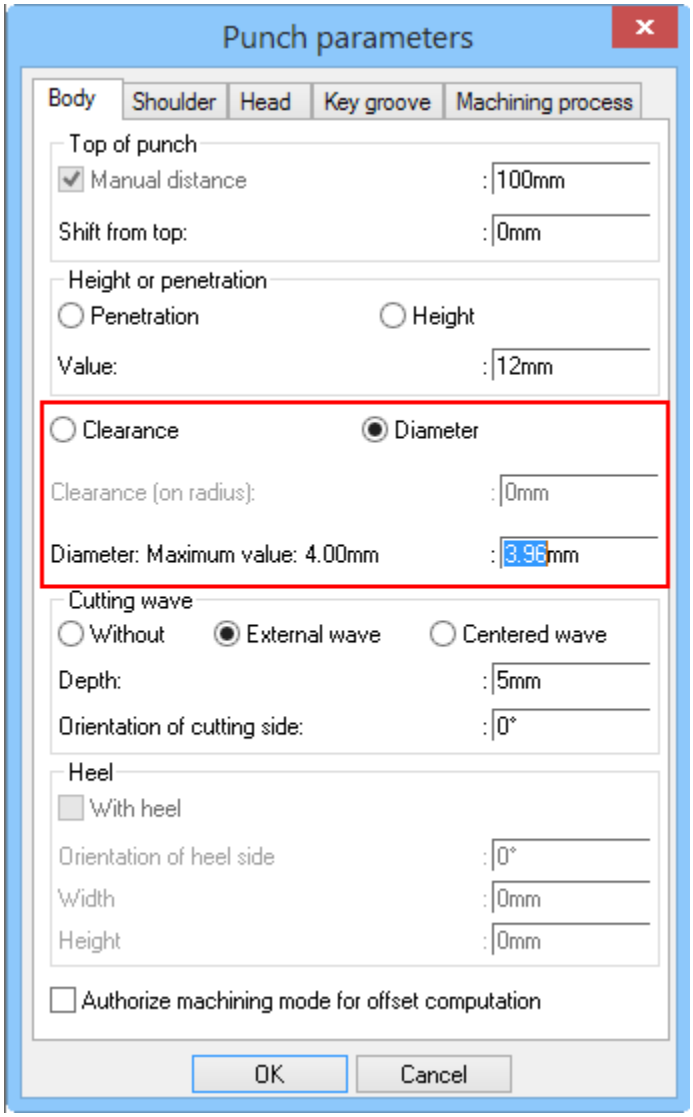
When the base profile of a cutting die was circular, it was already possible to define the dimensions of the die's second part as a diameter value.



This feature is now available when creating cutting dies based on multiple selection of several circular profiles.

Cutting punch

When the base profile of a cutting punch was circular, it was already possible to set its final diameter rather than giving a clearance value (in order to obtain a rounded value for the punch dimension).



This functionality is now available when creating cutting punches based on multiple selection of several circular profiles.

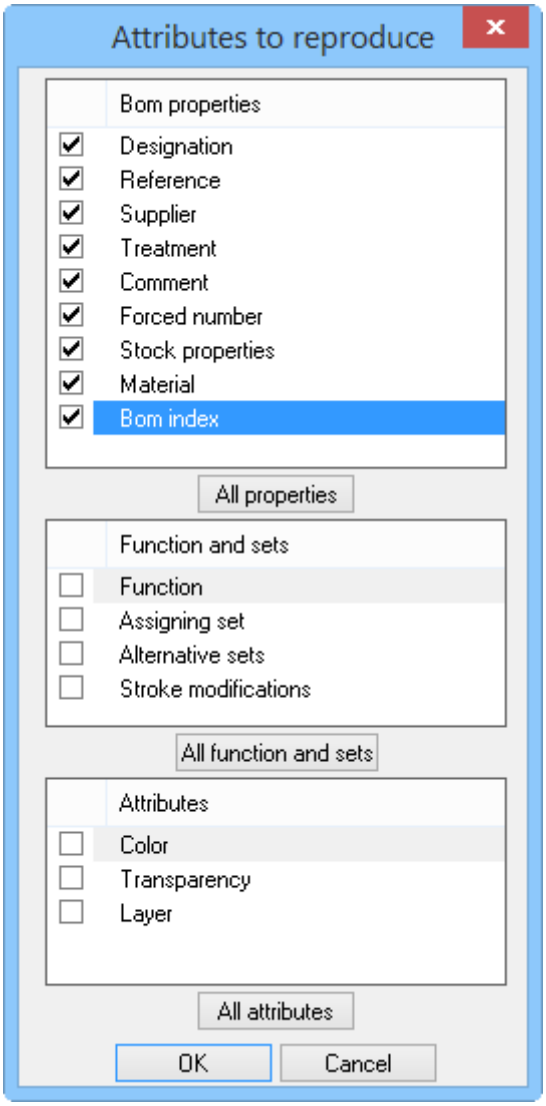
Miscellaneous

Progress properties

In the stock computation results, it is possible to still display the trailing zeros.

Reproduce bom properties

It is now possible to reproduce the **Bom index** information of template parts on the parts to be modified.
When this option is checked, the copy of all the other BOM properties is automatically activated, considering that parts with the same BOM indexes must be identical. However, the copy can be manually disabled for each of the other selected properties.

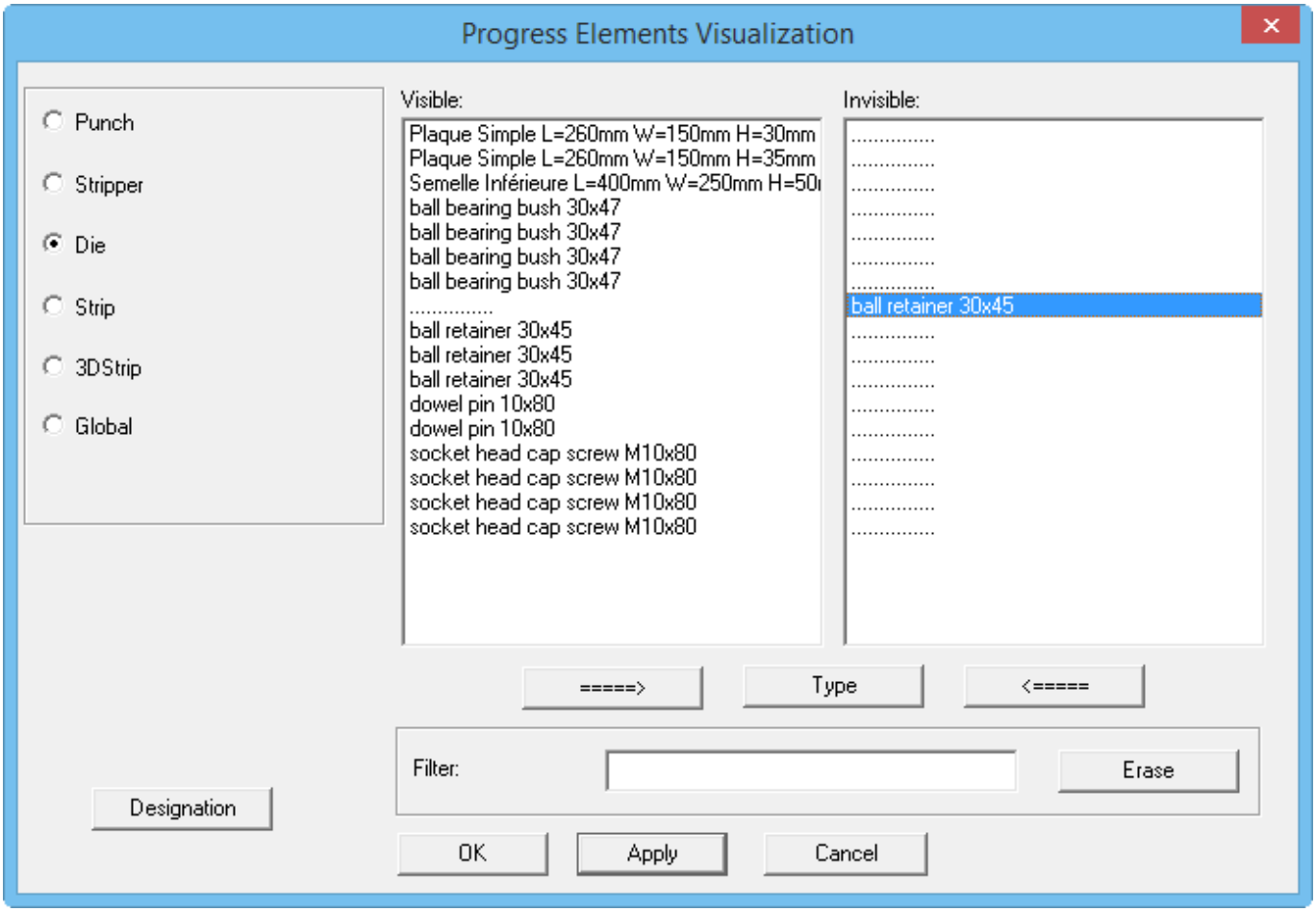


Edit > Duplicate

The parts created with the **Edit** > **Duplicate** command are now automatically assigned to the set of their template part.

TopSolid'Viewer > Progress elements visualization

In TopSolid'Viewer, a new **Progress elements visualization** dialog is now available.
It is used to quickly show/hide parts by manual selection, or according to the Progress tooling set they belong.

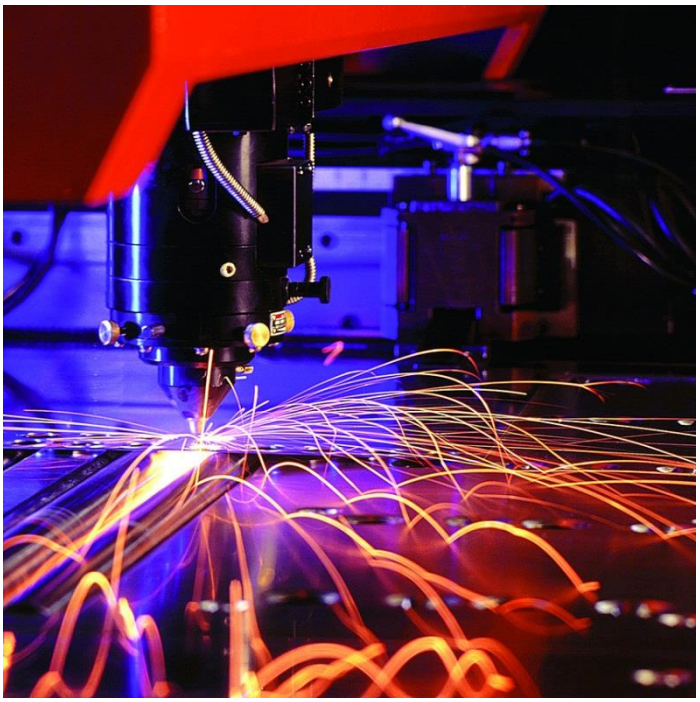


Draft

Component index

It is now possible to move a component index created with the **Component index** command after its creation by using the **Modify** command.

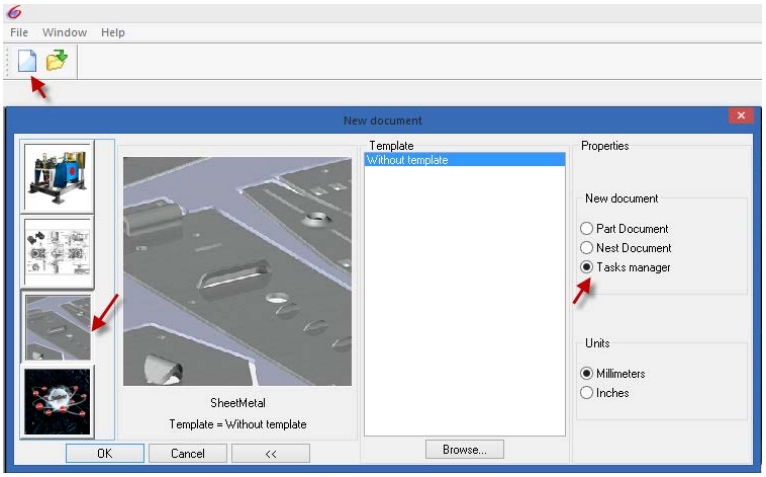
What’s new in TopSolid’SheetMetal v6.16



This section describes the new features in the version **6.16** of **TopSolid'SheetMetal**.

General

Direct access to the Tasks manager



Purpose of the function

The Tasks manager can now be accessed directly from the TopSolid main window.

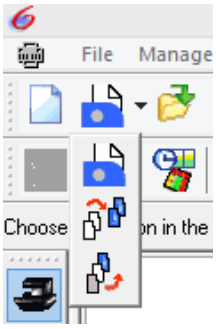
Triggering the function

Access from **File** > **New** > **SheetMetal** > **Tasks manager**.

Import menu

Purpose of the function

The functions that enable you to import multiple documents can be accessed directly from the system icon bar via a drop-down menu.



Update status of Part documents

Purpose of the function

A new update status is available.

Triggering the function

When the Part document results from a TopSolid 7 unfolding, a new update status specifies whether the v7 unfolding document no longer exists or it could not be found.

 The PCH part is up to date with its v7 unfolding.

 The v7 unfolding has been modified and the PCH part needs to be updated.

 Unknown update status when the connection to TopSolid 7 is disabled  .

 The v7 part could not be found.

Update status of Nest documents

Purpose of the function

New update status.

Triggering the function

The nesting update has been simplified. The context of updated functions (handwheel icon) has been removed. Now, when you want to update the nesting, you only need to click the traffic light icon.
The referenced PCH parts are first updated with their respective TOP file, and then the nesting is updated.

Traffic lights are now more readable.


Up to date nesting.


Some machinings of the nesting are not up to date (in relation to the machinings of referenced parts).


Geometries are not up to date.

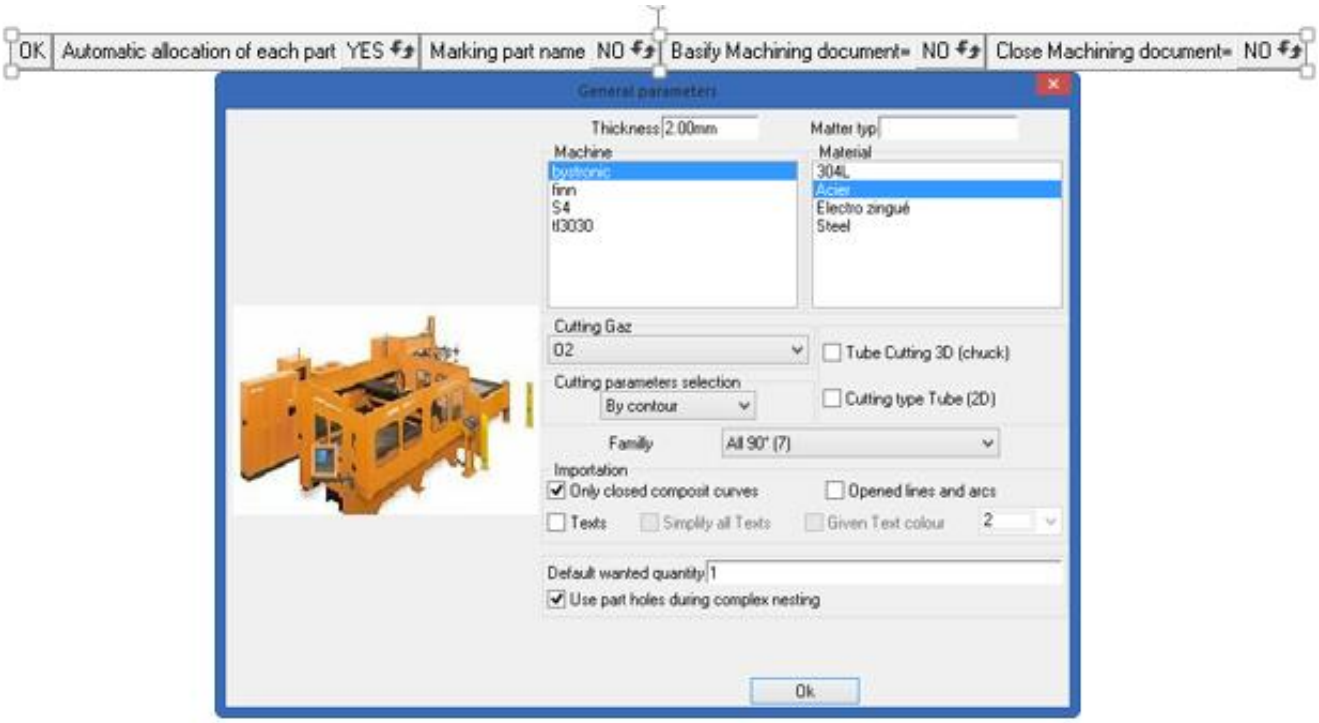
Importing multiple documents - Managing part families

Purpose of the function

Possibility of providing a family for a set of parts.

Triggering the function

During multi-document imports in which you are not asked to confirm the general parameters for each part, the **General parameters** dialog box appears only once for the set of documents. In this case, it is possible to specify the family which will be the same for all parts of the set.



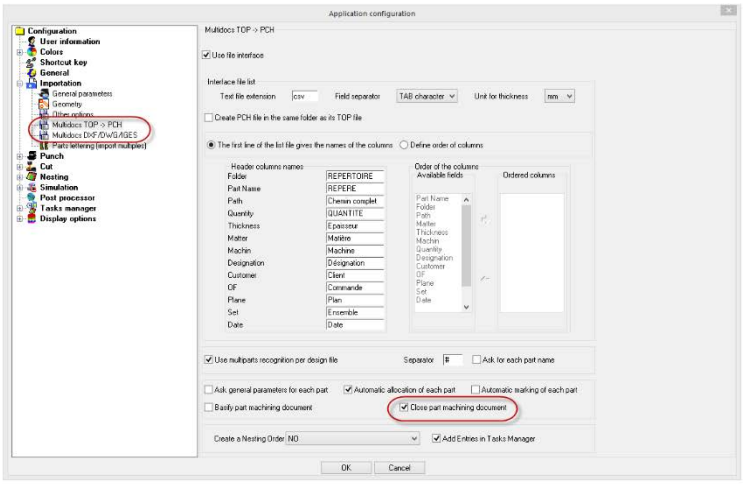
Importing multiple documents - Closing PCH documents

Purpose of the function

Possibility of closing the PCH documents created when importing multiple documents TOP → PCH and DXF/IGES/DWG → TOP → PCH

Triggering the function

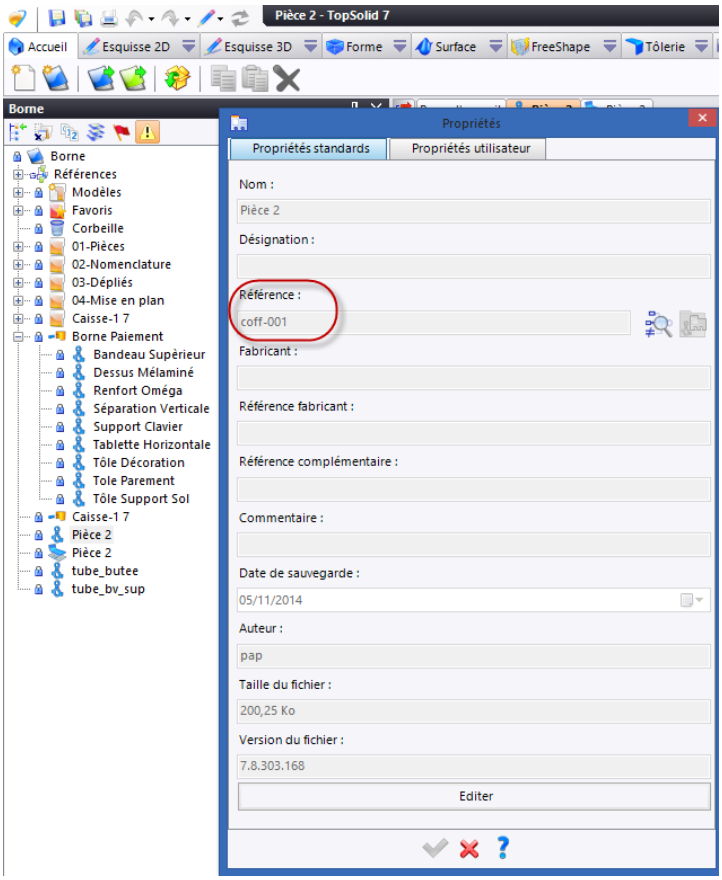
The setting is made in **Tools** > **Options** > **Importation** > **Multidocs TOP → PCH**
Multidocs DXF/DWG/IGES



TopSolid 7 import - Retrieving references

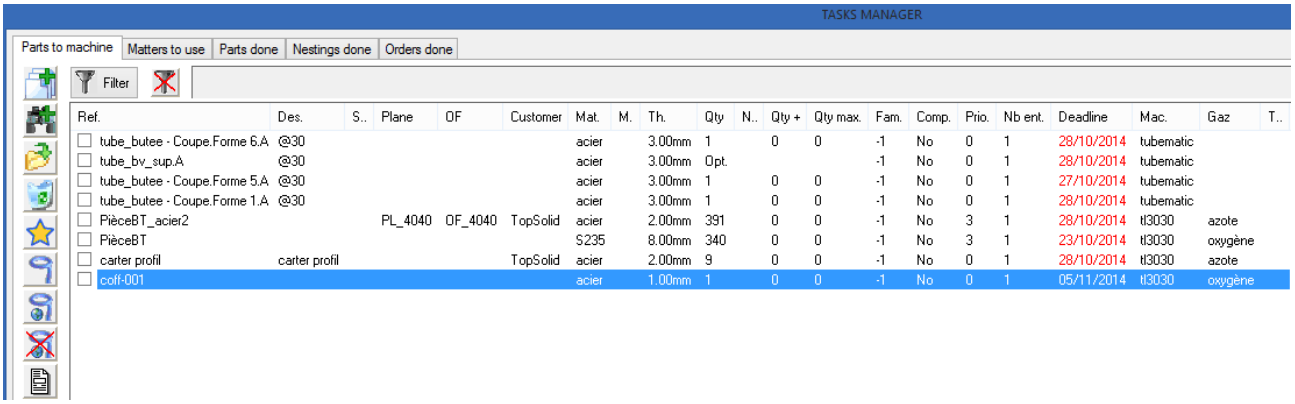
Purpose of the function

Possibility of retrieving the reference of the sheet metal part designed in TopSolid 7 during CAM import.



Triggering the function

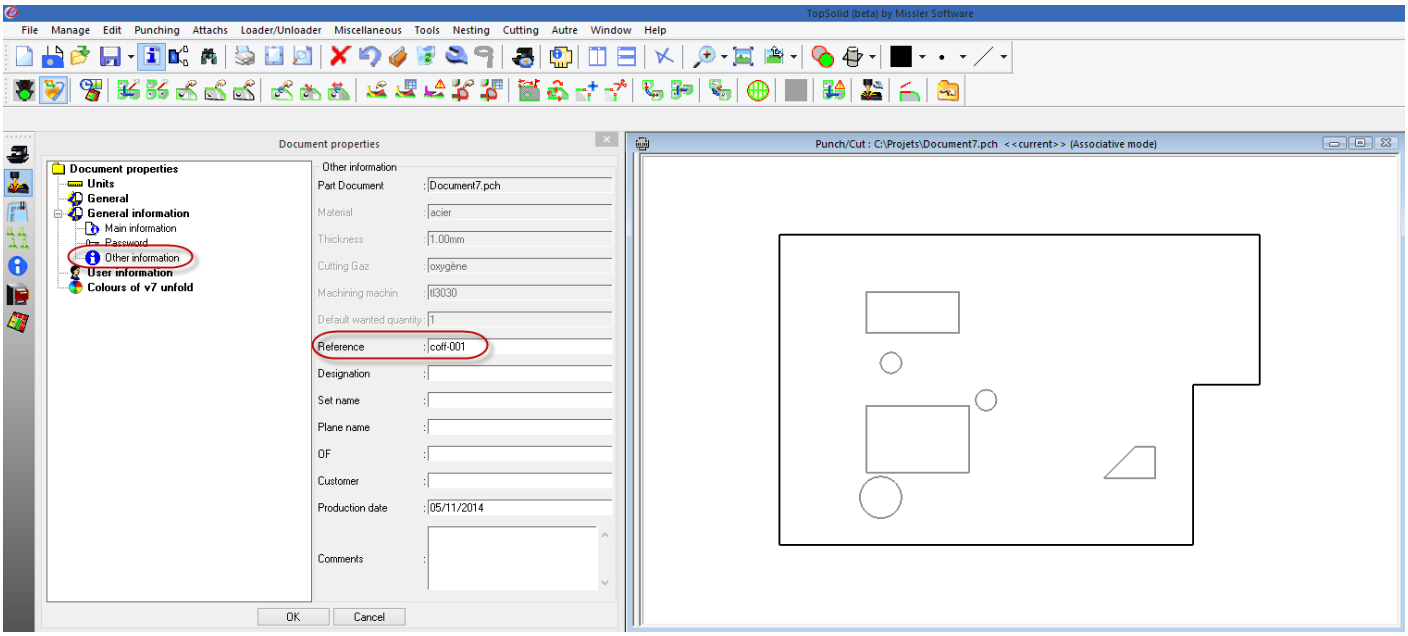
The reference is displayed in the document properties and available in the Tasks manager's **Reference** field.



Ref.	Des.	S..	Plane	OF	Customer	Mat.	M.	Th.	Qty	N..	Qty +	Qty max.	Fam.	Comp.	Prio.	Nb ent.	Deadline	Mac.	Gaz	T..
tube_butee - Coupe.Forme 6.A	@30					acier		3.00mm	1		0	0	-1	No	0	1	28/10/2014	tubematic		
tube_bv_sup.A	@30					acier		3.00mm	Opt.				-1	No	0	1	28/10/2014	tubematic		
tube_butee - Coupe.Forme 5.A	@30					acier		3.00mm	1		0	0	-1	No	0	1	27/10/2014	tubematic		
tube_butee - Coupe.Forme 1.A	@30					acier		3.00mm	1		0	0	-1	No	0	1	28/10/2014	tubematic		
PièceBT_acier2			PL_4040	OF_4040	TopSolid	acier		2.00mm	391		0	0	-1	No	3	1	28/10/2014	tl3030	azote	
PièceBT						S235		8.00mm	340		0	0	-1	No	3	1	23/10/2014	tl3030	oxygène	
carter profil	carter profil				TopSolid	acier		2.00mm	9		0	0	-1	No	0	1	28/10/2014	tl3030	azote	
coff-001						acier		1.00mm	1		0	0	-1	No	0	1	05/11/2014	tl3030	oxygène	

Note: If the 3D sheet reference is not specified, but the unfolding reference is provided, the transferred value will be that of the unfolding. However, if both references are specified, the 3D sheet value takes priority.

Since this reference may be modified in the PCH document's properties, there is no associativity on this data over time (same for the designation).



Displaying TopSolid 7 CAD parameters in the operator card

Purpose of the function

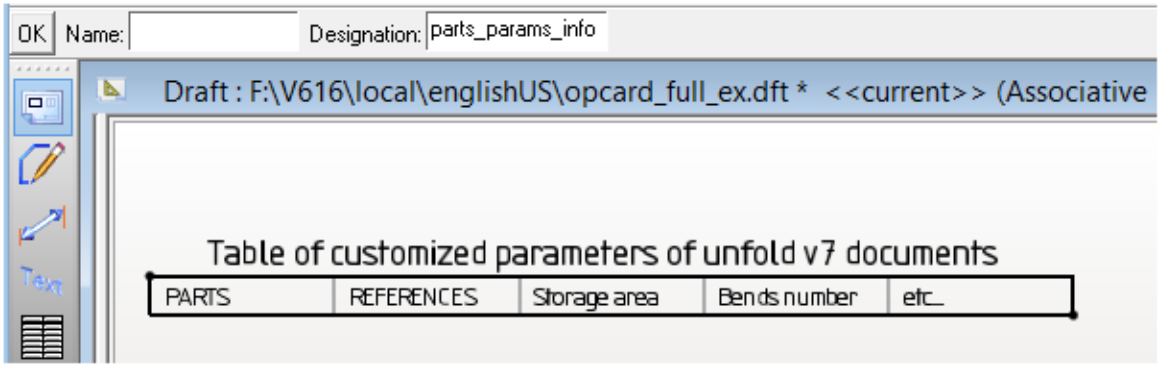
Possibility of displaying in the operator card, in the table describing the parts, the parameters defined in the v7 unfolding document.

Triggering the function

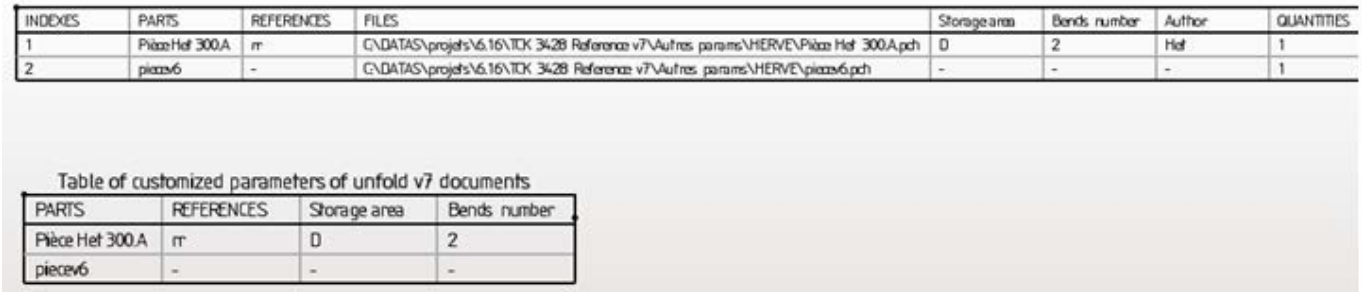
- Define the parameter name as the column name.

A sample table is available in the operator card: *opcard_full_ex.dft*.
This table has the following designation: *parts_params_info*.

Example of table in the template operator card:



Result:



INDEXES	PARTS	REFERENCES	FILES	Storage area	Bends number	Author	QUANTITIES
1	Pièce Het 300.A	rr	C:\DATAS\projets\v6.16\TCK 3428 Reference v7\Autres params\HERVE\Pièce Het 300.A.pch	D	2	Het	1
2	piecev6	-	C:\DATAS\projets\v6.16\TCK 3428 Reference v7\Autres params\HERVE\piecev6.pch	-	-	-	1

Table of customized parameters of unfold v7 documents

PARTS	REFERENCES	Storage area	Bends number
Pièce Het 300.A	rr	D	2
piecev6	-	-	-

Note: These columns (whose names are customized to match the parameter names) can be added to the *parts_info* table.

The same can be done in the complete order operator card.

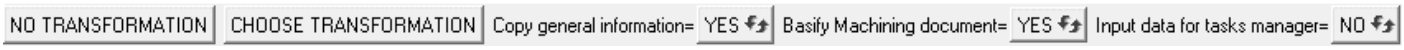
Import with basification

Purpose of the function

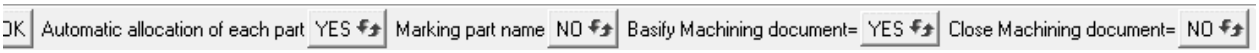
Basification of the Part document during import.

Triggering the function

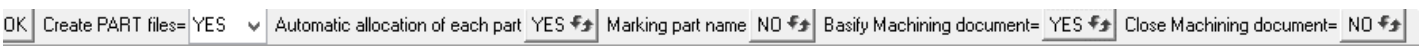
- **Single import**



- **Import of multiple documents Top → PCH**

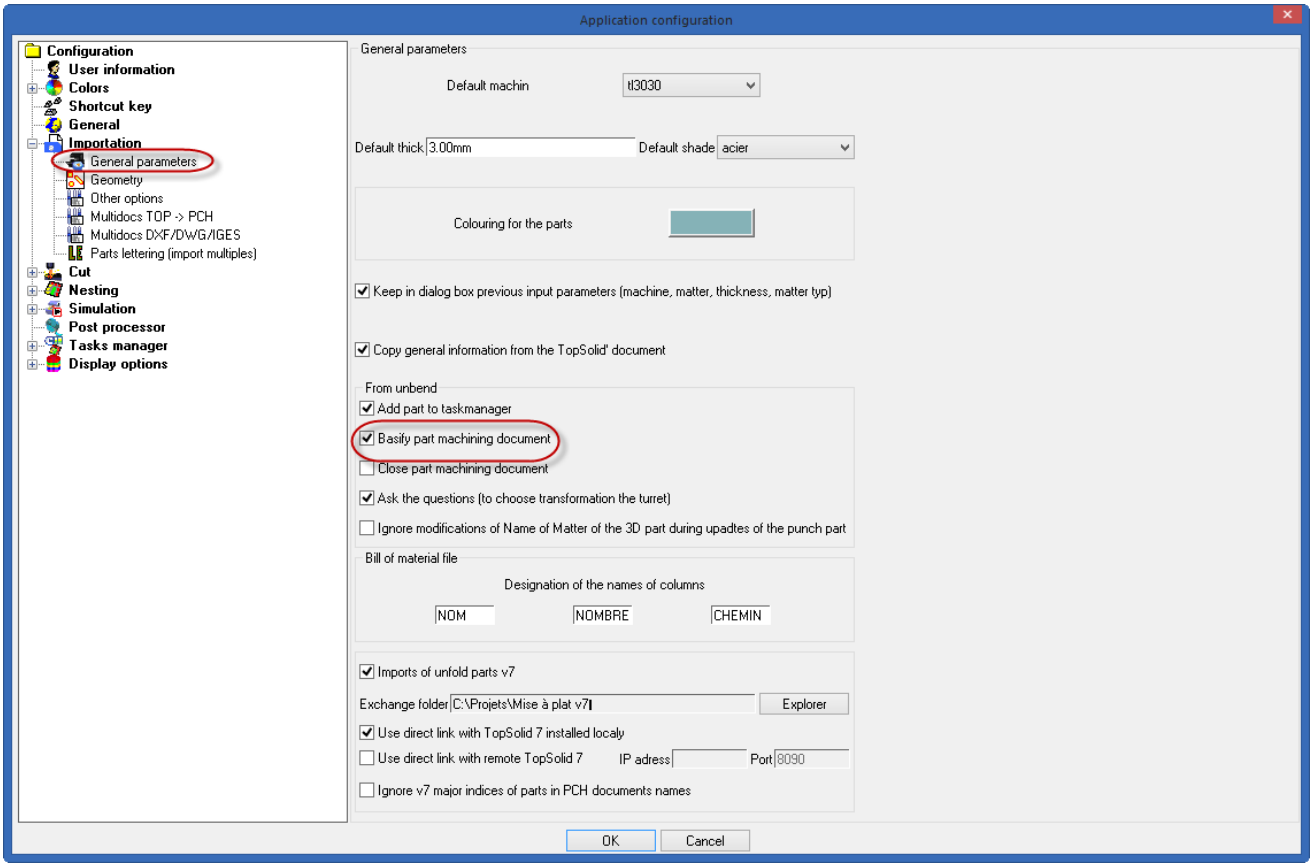


- **Import of multiple documents DXF - IGES → TOP → PCH**

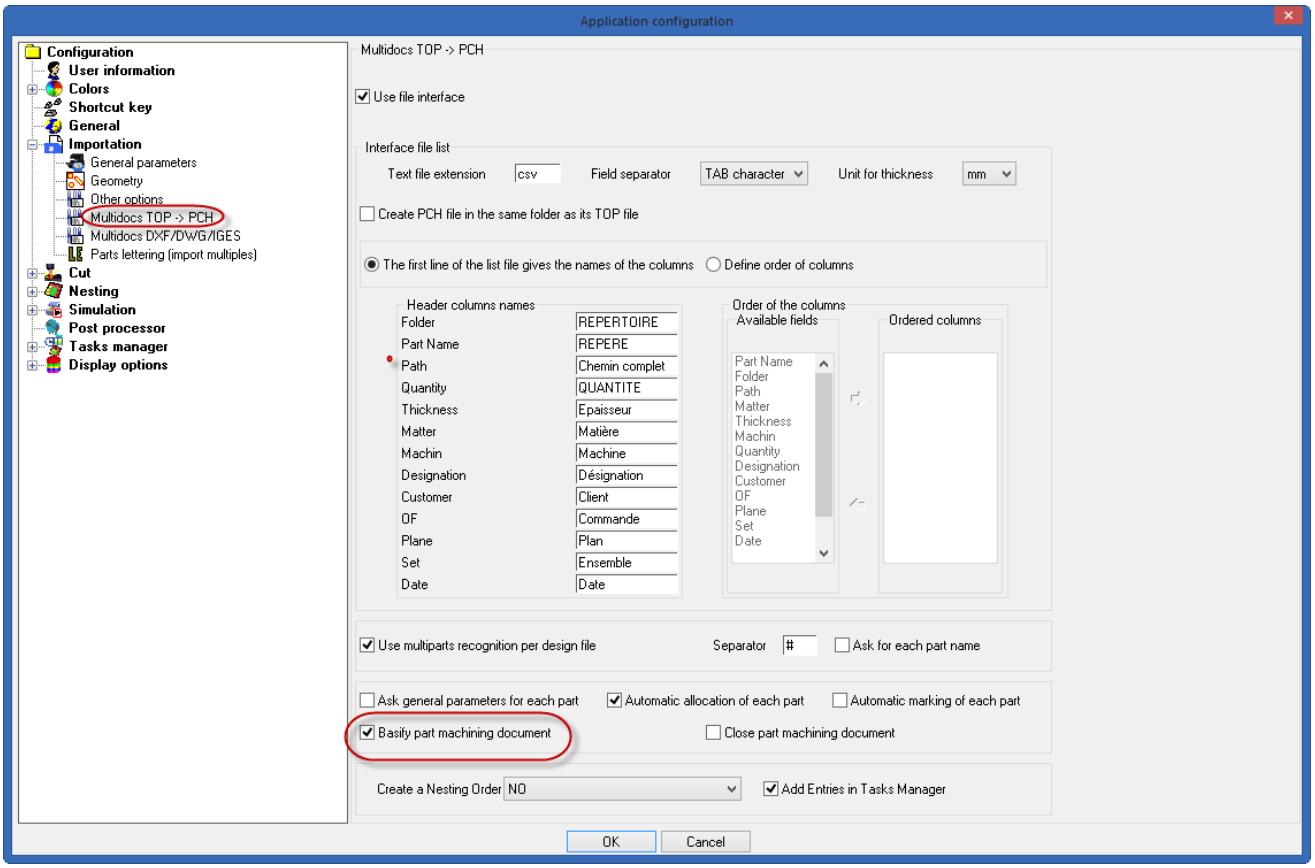


This setting is available by default in **Tools** > **Options**.

For single importation:



When importing multi-documents TOP → PCH:



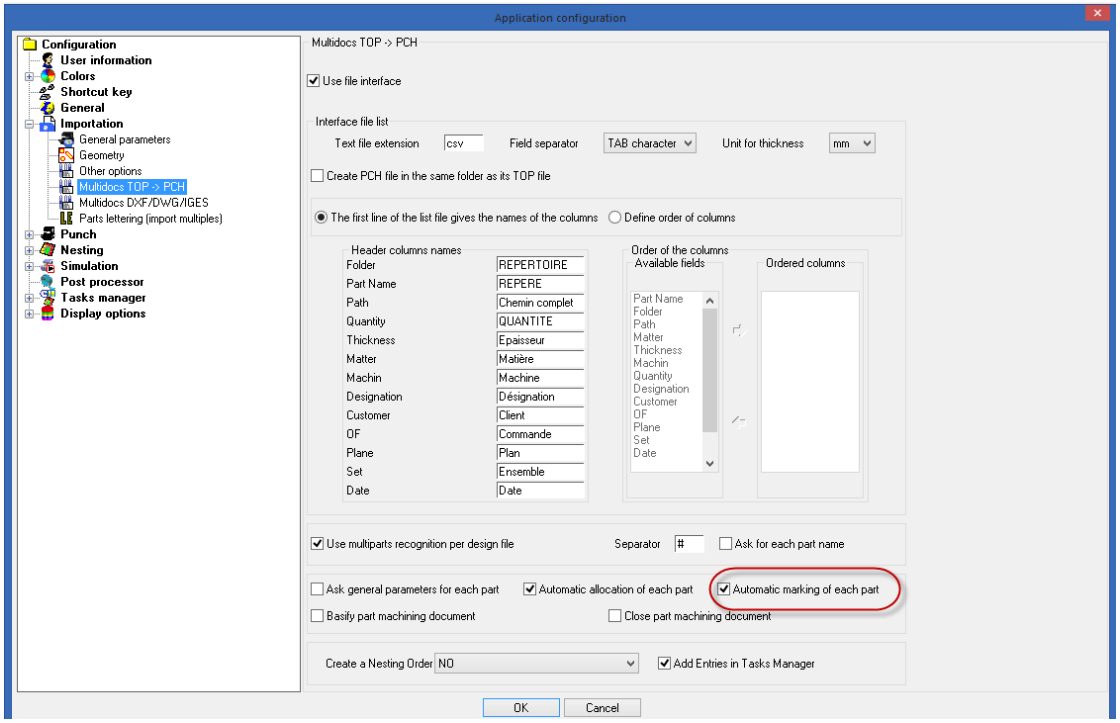
Automatic part lettering in multi-document import

Purpose of the function

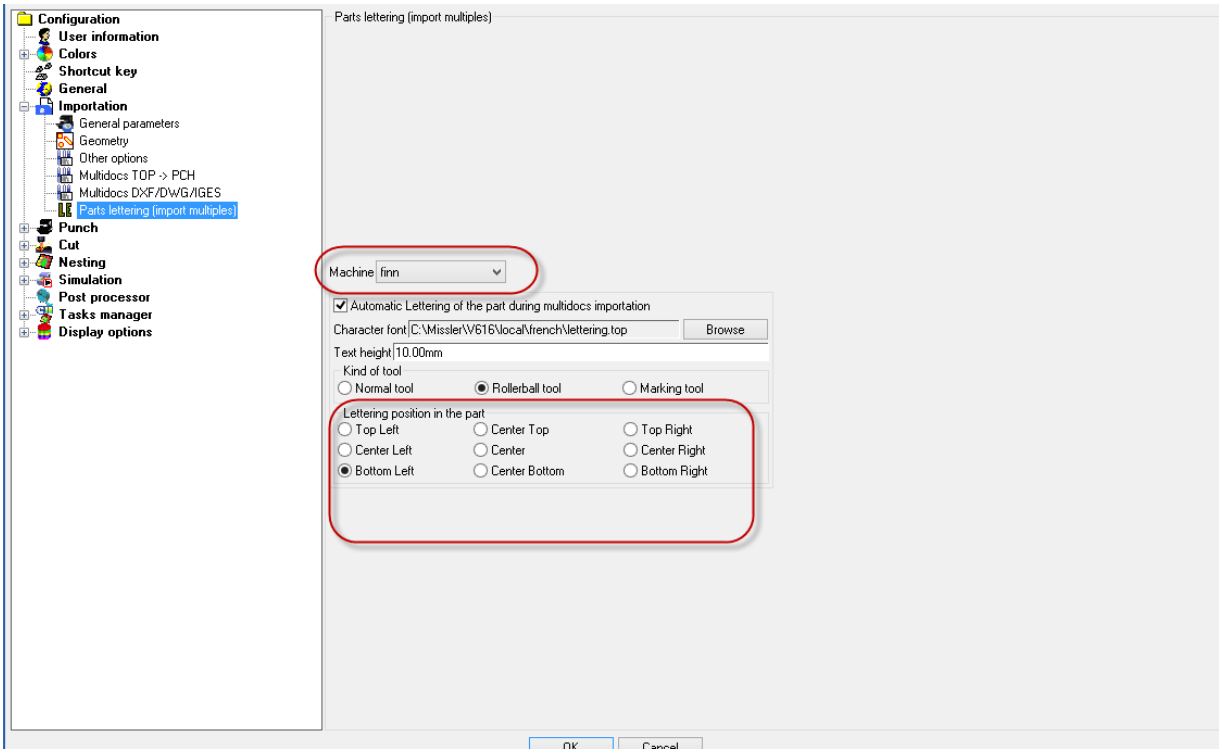
Possibility of automatically lettering the part name when importing multiple documents **TOP → PCH** and **DXF/DWG/IGES → TOP → PCH**.

Triggering the function

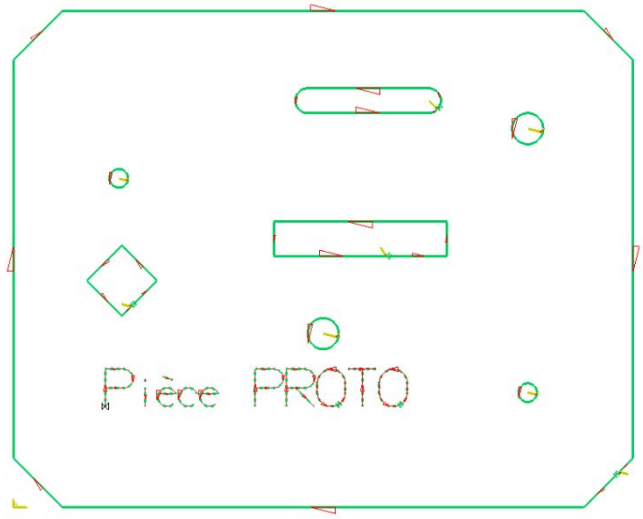
The automatic lettering can be configured in **Tools** > **Options** in each multi-document import section.



For each machine, you also need to configure whether the lettering is to be performed when creating the part during multi-document imports, as well as the font settings, the text height, the machining position and type (punching or laser lettering cutting).



Example:



Note: There are also settings to perform automatic lettering during linked actions for part nesting.
The settings for a given machine can be different.

- If part lettering is requested during linked actions for nesting, it will not be performed if it has already been created on the Part document. The part lettering of the PCH document will be given priority.
- If lettering is done using the **Miscellaneous** > **Free contours** > **Lettering** function and the **Generate PartName as Text** open is selected, this lettering is considered as part lettering, and in this case, the automatic lettering during linked actions for nesting will not be recreated for the part.

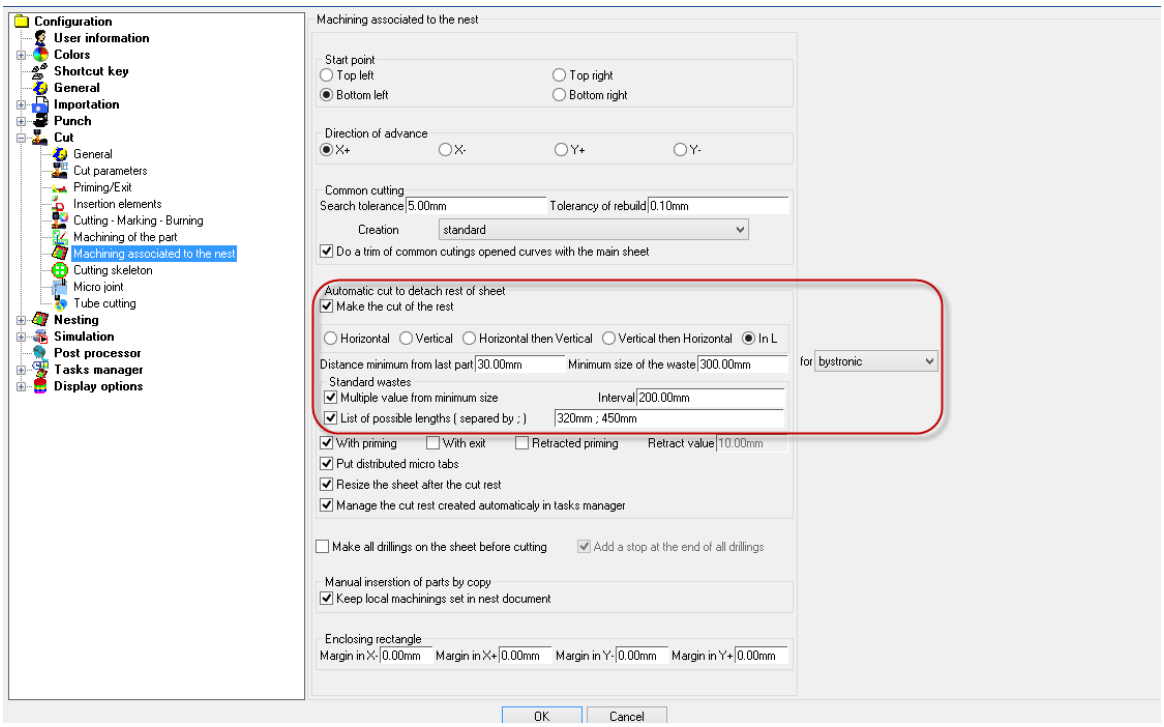
Configuring waste sizes

Purpose of the function

Possibility of standardizing the waste sizes.

Triggering the function

Waste sizes can be configured in **Tools** > **Options**.



Two cumulative options are proposed:

- **Multiple value from minimum size**. In the example above, the waste sizes can be 300mm, 500mm, 700mm, 900mm, etc.
- **List of possible lengths (separated by ;)**. In the example above, the waste sizes can be 320mm and 450mm.

If one or both boxes are checked, the system will adjust the waste size to the value closest to the set of values, while taking into account the minimum distance from the last part and the minimum waste size.

Improved manual function

Accessible from **Cutting** > **Common** > **Rest cut**.

Two modes are available:

- **Automatic mode**: The waste is calculated by taking full account of the distance from the last part.
- **Manual mode**: A waste size is imposed.

Once this function is launched, the system calculates the optimal waste size according to the standard sizes defined in **Tools** > **Options** and proposes the manual mode.

Note: The same configurations are found in **Tools** > **Options** for punching machines.

Mixed waste cuttings

Purpose of the function

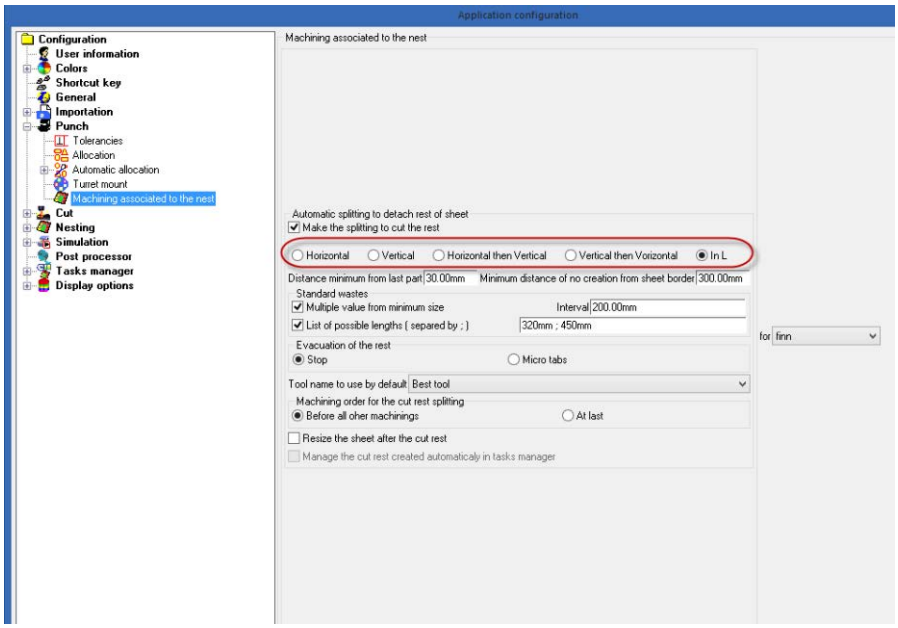
Possibility of cutting rests automatically both vertically and horizontally on the same sheet.

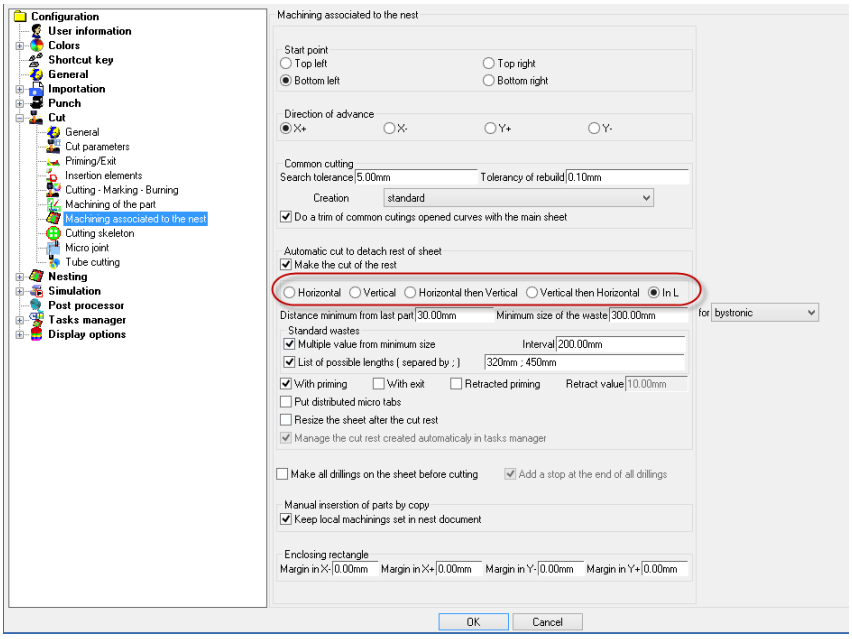
Triggering the function

Configuration in **Tools** > **Options** for cutting and punching machines.

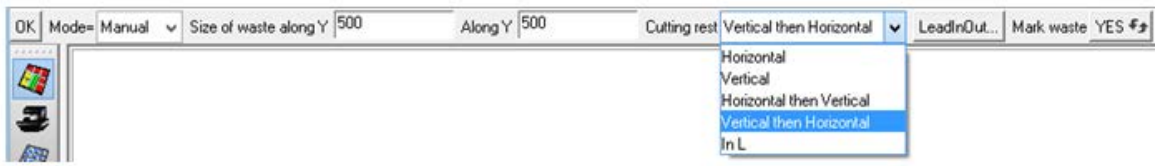
The various choices are:

- **Horizontal**
- **Vertical**
- **Horizontal then Vertical**
- **Vertical then Horizontal**
- **In L** in one go

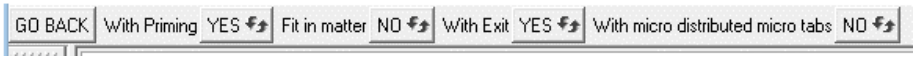




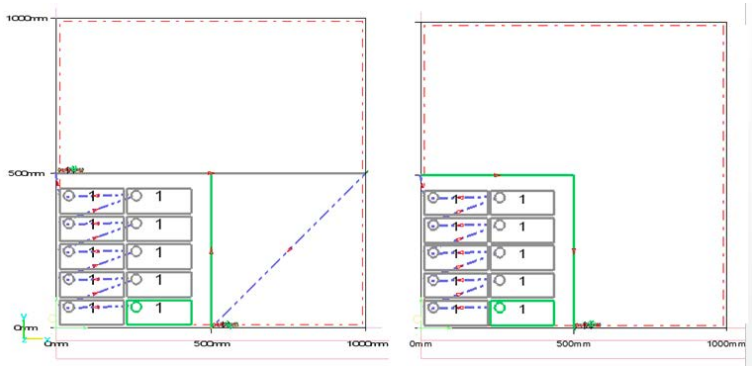
For cutting machines, the menu of the **Cutting rest** feature has been modified.



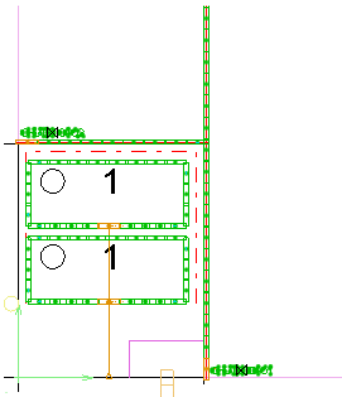
A new **LeadInOut** button allows you to configure the primings/exits, as well as maintain wastes with micro-tabs.



Cutting examples:



Punching example with automatic resizing:



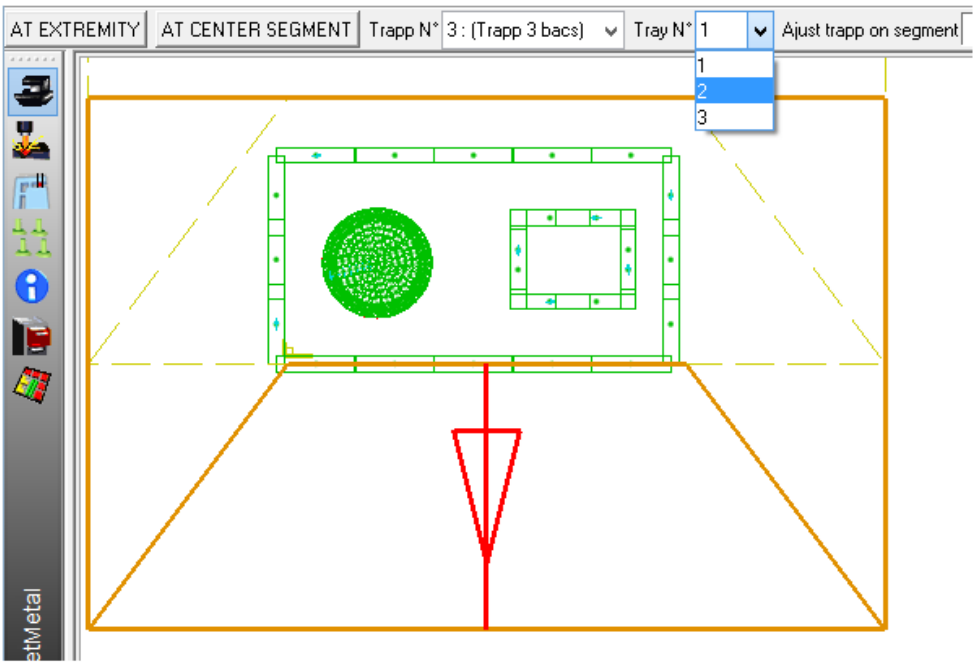
Trapp door evacuation function - Tray number

Purpose of the function

Possibility of selecting the number of the evacuation tray.

Triggering the function

Trapp door evacuation menu.

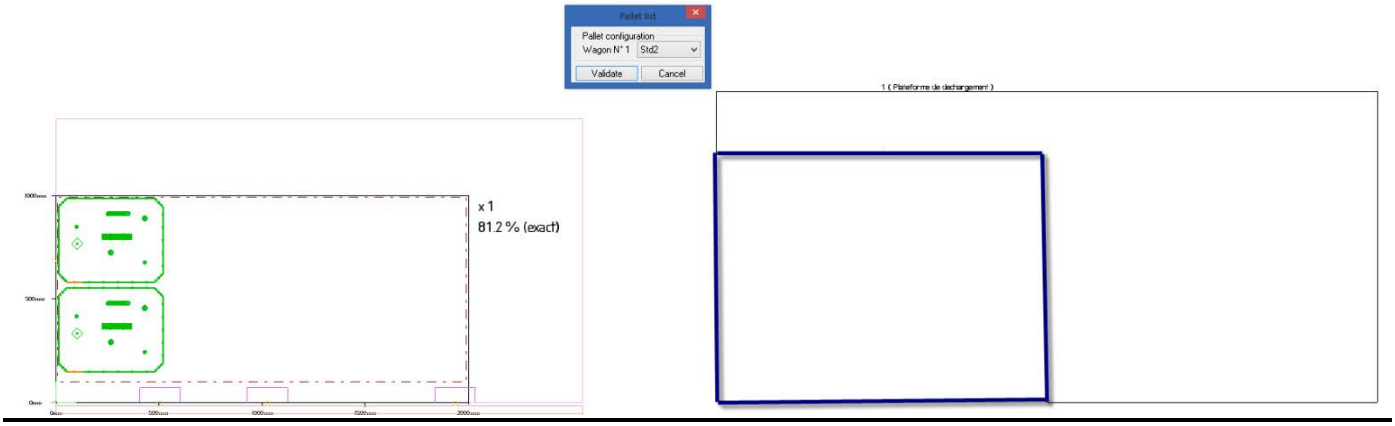


Pallet preview

Purpose of the function

Preview of the pallet selected in the drop-down list displayed on the wagon.
This enables the pallets to be visually adapted to the parts to unload.

Triggering the function



Dynamic zoom can be done in the dialog box and the dialog box can also be cancelled.

Simplified visualization of the trapp door

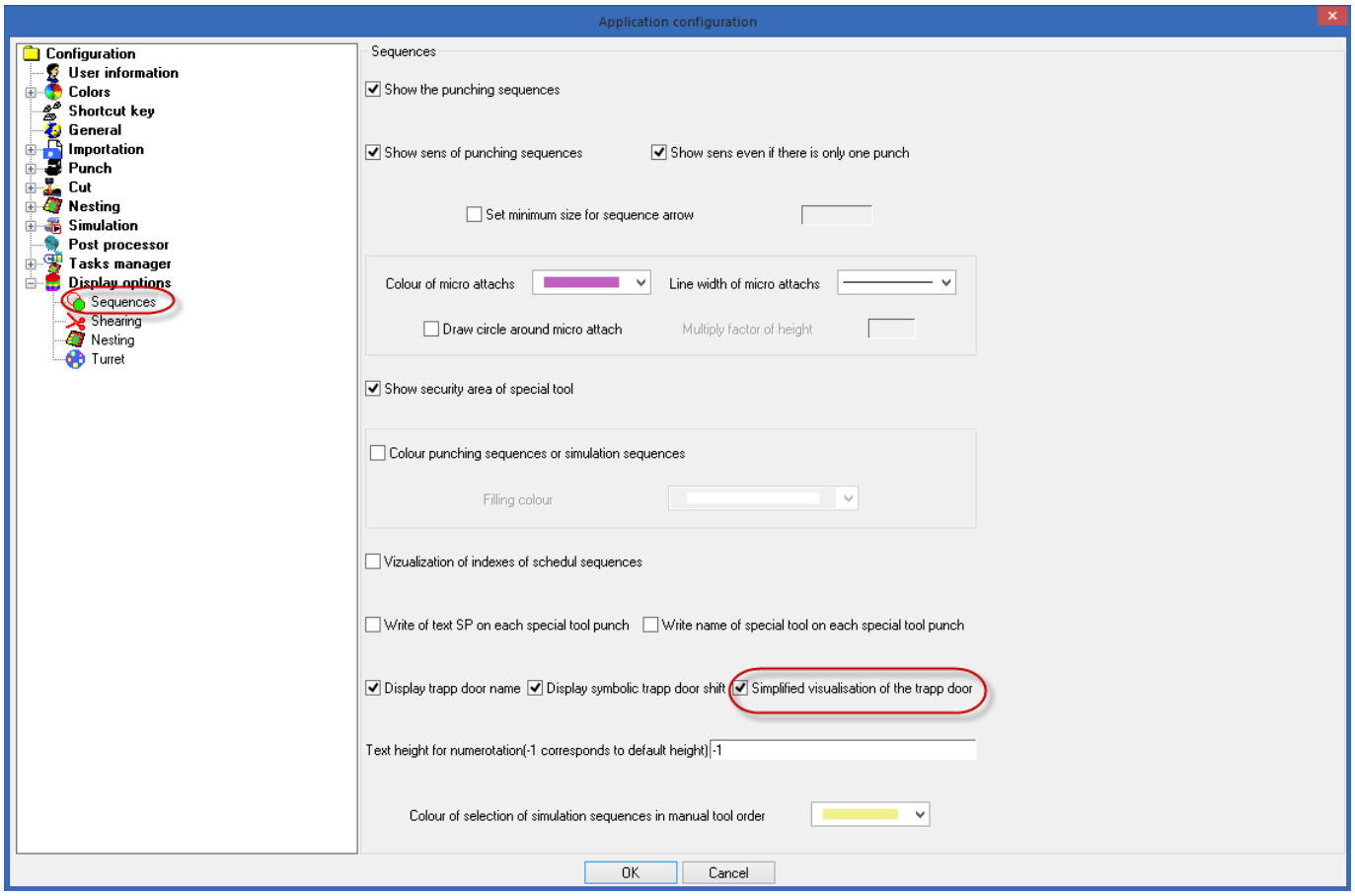
Purpose of the function

The trapp door visualization can be simplified.
Possibility of not viewing the evacuation trapezoid.

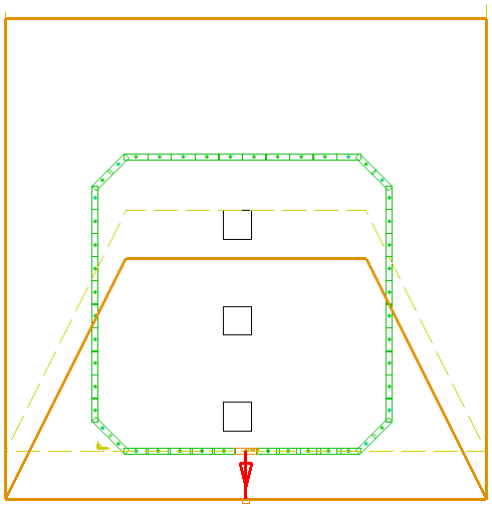
Triggering the function

Configuration in **Tools** > **Options** > **Display options** > **Sequences**.

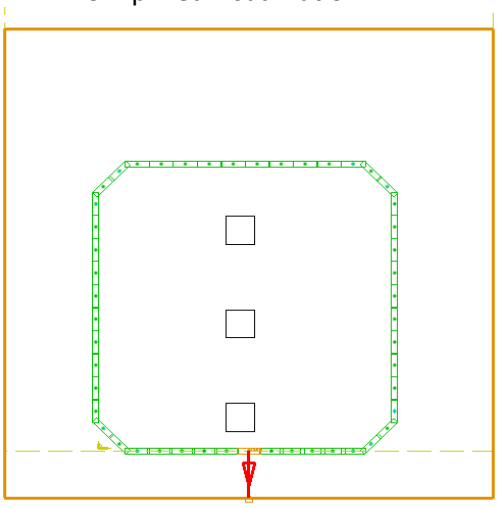
- Check the **Simplified visualisation of the trapp door** box.



Full visualization



Simplified visualization



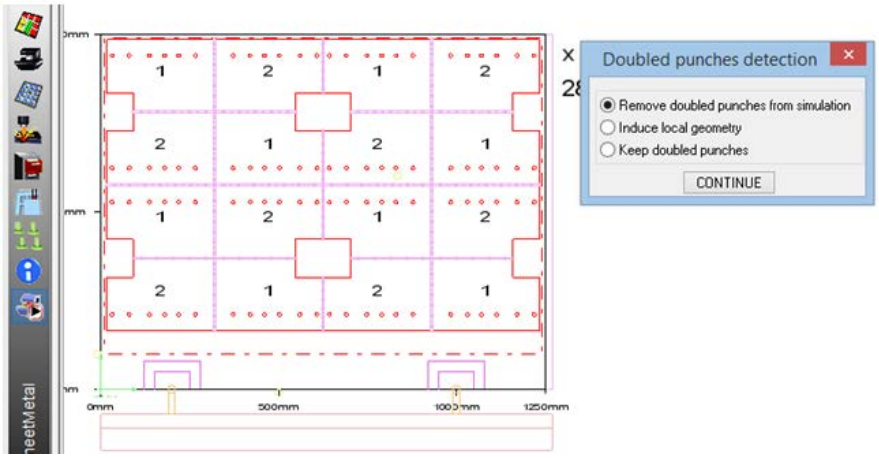
Visualization of doubled punches before simulation

Purpose of the function

Visualization of doubled punches.

Triggering the function

Visualization of doubled punches when displaying the simulation dialog box.



Additional information in the operator card

Purpose of the function

Additional information in the operator card.

Triggering the function

In a nesting done from the Tasks manager, if parts not contained in the Tasks manager are added manually, the data of the operator card's table such as the order, the customer, etc. are now provided with the values specified in the Part document's properties.

The same applies to nestings that are done outside the Tasks manager.

This information is also displayed in the part operator card and in the complete order operator card.

Tasks manager

Adding nestings to an order

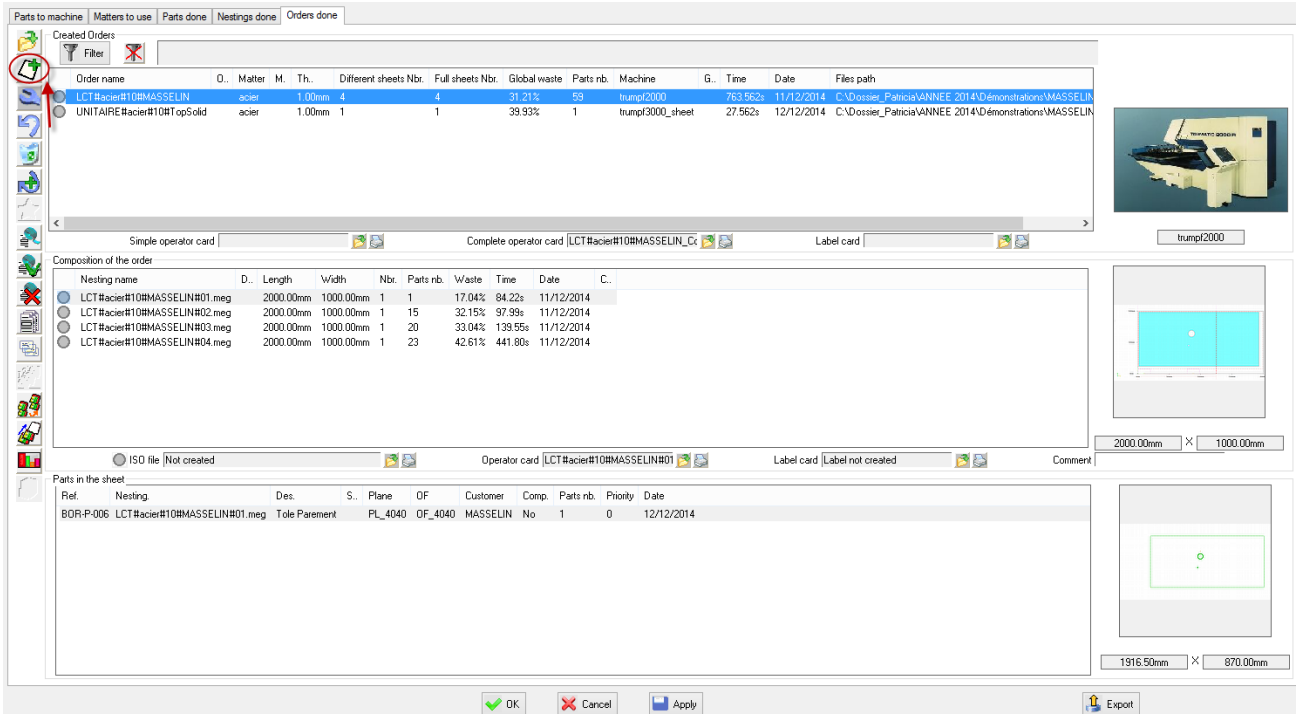
Purpose of the function

This function allows you to add one or more nestings to an existing order without changing the nestings already done in the order.

The new created nestings can be performed automatically or manually.

Triggering the function

- From the Tasks manager's **Orders done** tab, activate the **Add new nestings in order** icon. 



Waste rotation in Tasks manager

Purpose of the function

Rotates the geometry of a waste.

Triggering the function

- Select a waste (whose preview is available) and use the two buttons  .

The geometry of the drawing file corresponding to the waste is rotated 90° or -90°.
The preview is updated again.

TopSolid'Punch - Punching

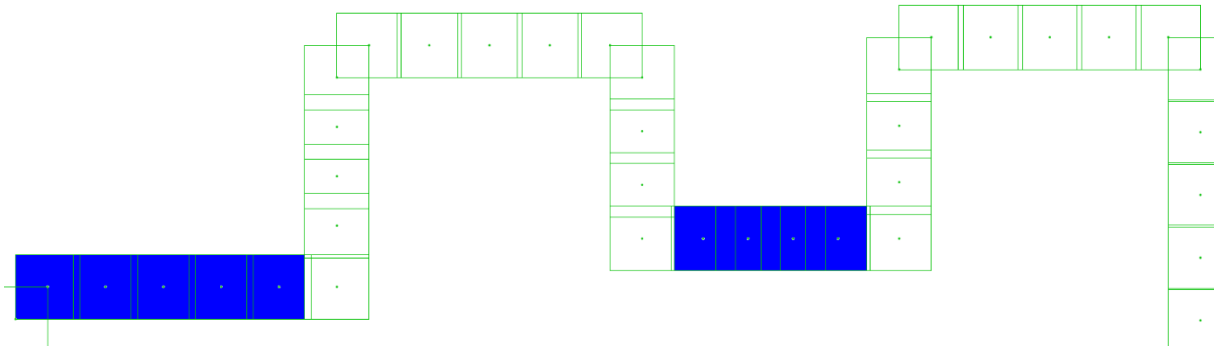
Notches decomposed into lines

Purpose of the function

Performing an automatic allocation while avoiding doubled punches.

Triggering the function

During automatic allocation, if notches are split into lines and lines are machined with a square or round tool, the system reduces the machining to avoid doubled punches where two lines intersect.



Automatic allocation - Arc with two oblongs

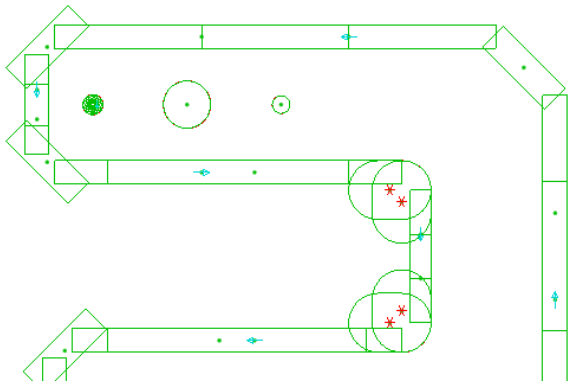
Purpose of the function

A new feature allows you to place a tangential oblong on each side.

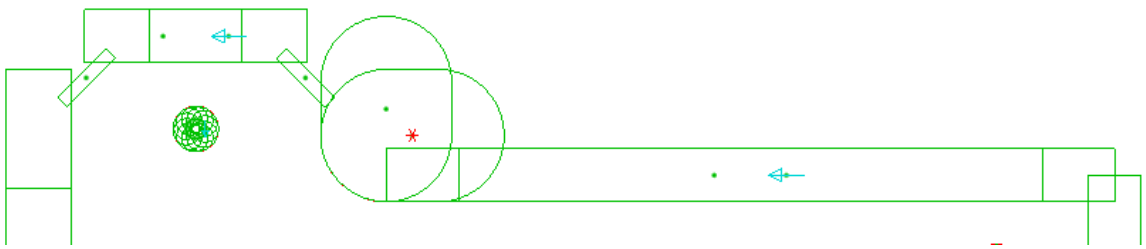
Triggering the function

The selected oblong has the same diameter as the arc and the shortest length of the tool library.

A tool on point associative with the line is positioned provided that the line is greater than the tool. Otherwise, the oblong tool is positioned on the line.

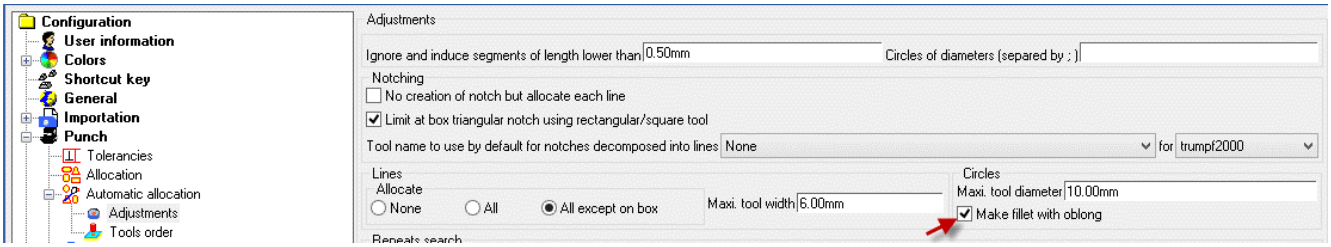


The arc then turned into an induced machining.

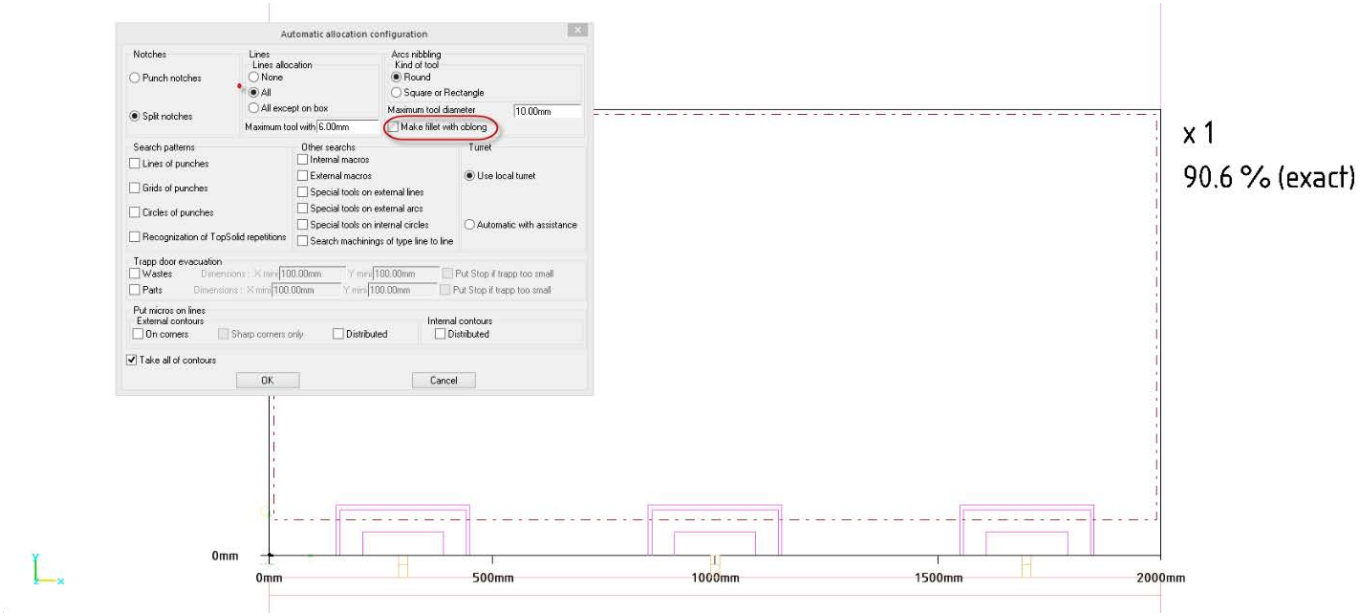


Configuration

Accessible from **Tools** > **Options** > **Punch** > **Automatic allocation** > **Adjustments**.



Activation in automatic allocation



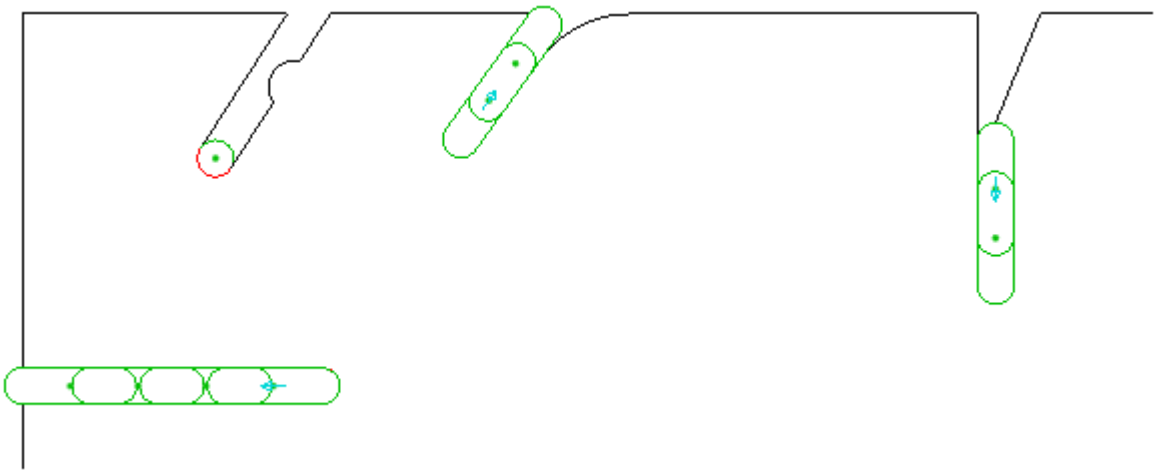
Activation in manual punching

A function of the **Punching** menu can be used to perform this type of machining by clicking an arc.

Automatic allocation - Oblong groove

Purpose of the function

Half-oblong grooves are automatically recognized and the oblong tool with the relevant diameter is positioned.



The machining is placed on the longest line if there is no collision, otherwise on the shortest line. The arc and the other completely covered line are placed as induced machinings.

To disable the automatic recognition of oblong grooves, the following line must be added manually into the *C:\Missler\Config\Topzpch.cfg* file:

ZPCHAFAUTO_HALF_OBLONG_GROOVE_WITH_OBLONG 0

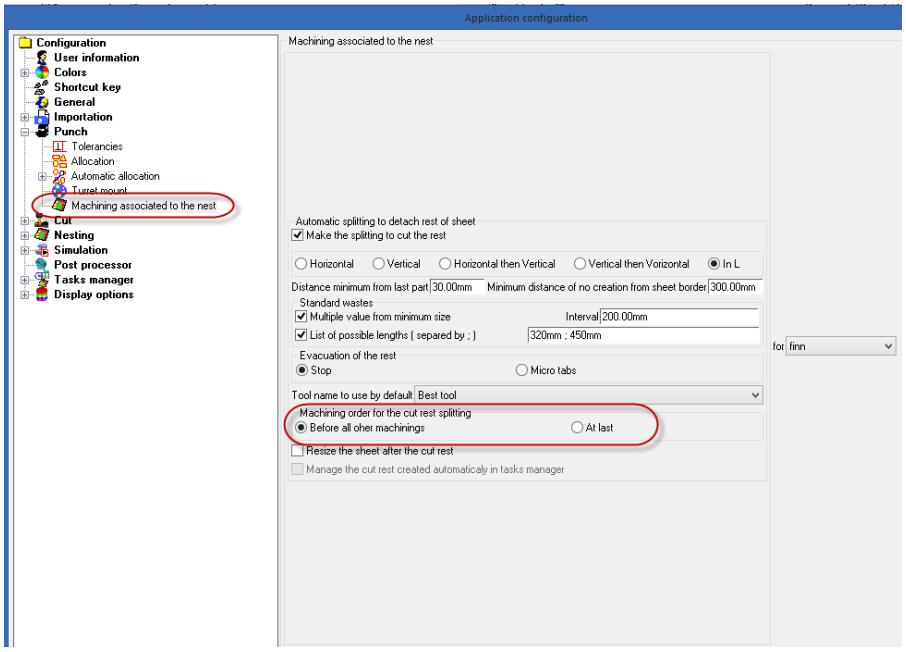
Punched rest cutting - Before or last

Purpose of the function

Possibility of configuring the machining order for the cut rest splitting in punching.

Triggering the function

Configuration in **Tools** > **Options** > **Punch** > **Machining associated to the nest**.



You can also modify this information using the wrench on the splitting.

Applying the modified turret of a nesting to all the order's nestings

Purpose of the function

If the current document is a nesting belonging to an order which was created from the Tasks manager, and the turret is in **Manual tool put on** mode, a new button then appears: **Nests order turret change**.

This new feature can modify the turret of the current nesting document and apply this turret to all the order's nestings.

Triggering the function

Requirements/Operating processes:

A preliminary check is done (by opening each document of the order) on a station-by-station basis to make sure that all the turrets are identical.

The dialog box with the turret picture allows you to change the place of the tool (for example).

After validating the dialog box, the turret of the current document will be copied to all the order's nestings with the usual checks that may make sequences for which the tool is no longer appropriate (wrong orientation) invalid. Messages are launched in the alpha bar, nesting by nesting.

The simulation of each nesting is obsolete.

TopSolid'Cut - Cutting

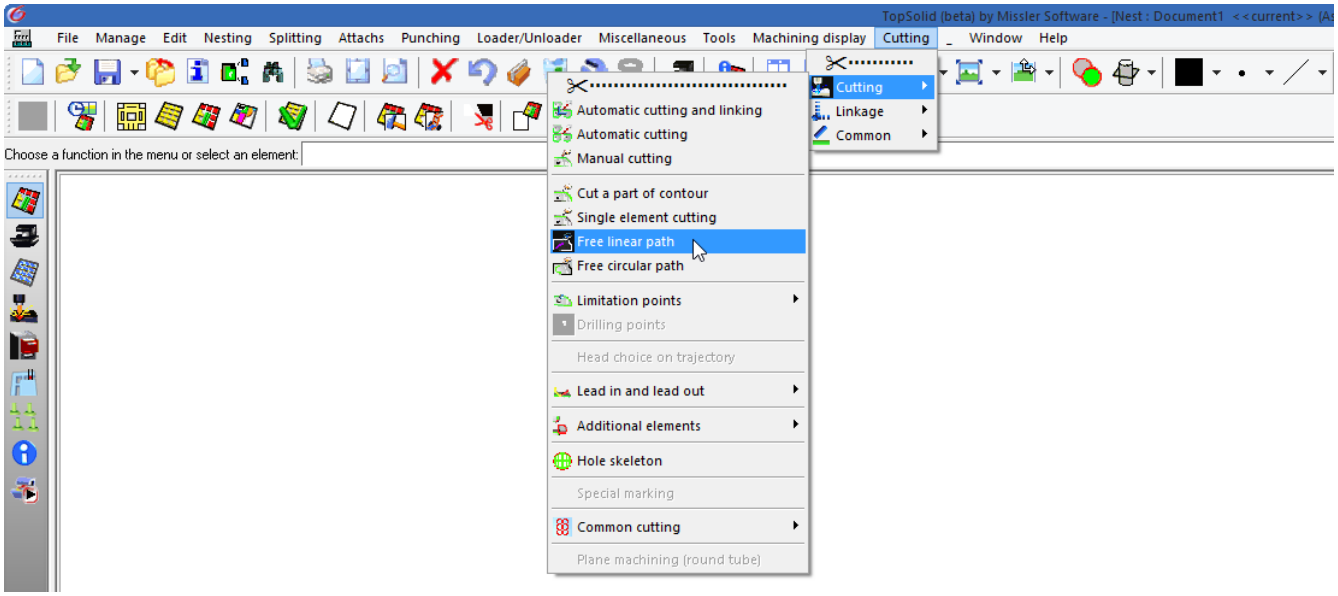
Free cuttings in the Part document

Purpose of the function

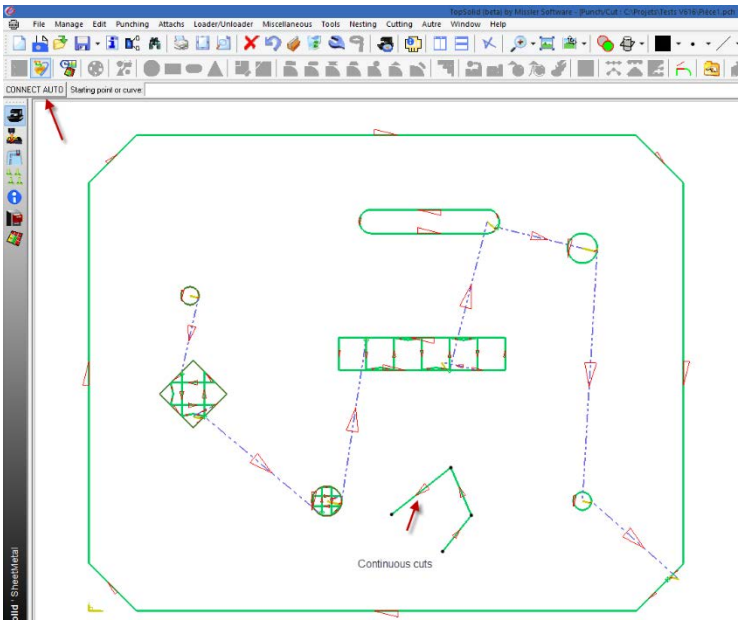
Possibility of performing free linear and circular cuttings in the Part document.

Triggering the function

Accessible from the **Cutting** menu > **Cutting** > **Free linear path** or **Cutting** menu > **Cutting** > **Free circular path**.



The **Connect auto** button allows various lines to be connected in order to have a single cutting operation.



Internal skeletons

Purpose of the function

Creating a hole skeleton in the Part Document.

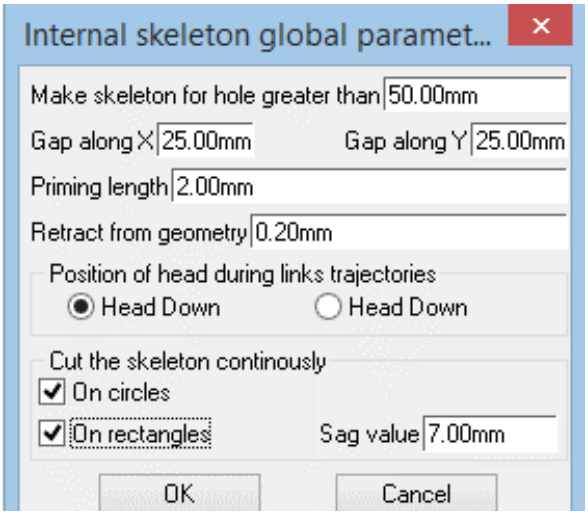
Triggering the function

Accessible from the **Cutting** menu > **Cutting** > **Hole skeleton**  .

This function creates lines inside the hole; these vertical and/or horizontal lines are spaced by a gap given in both directions. The starting point is one of the four corners of the hole's enclosing box.
If the hole already has a cutting allocation, it is possible to calculate the starting corner automatically to ensure that the last skeleton line is as close as possible to the priming on the contour.

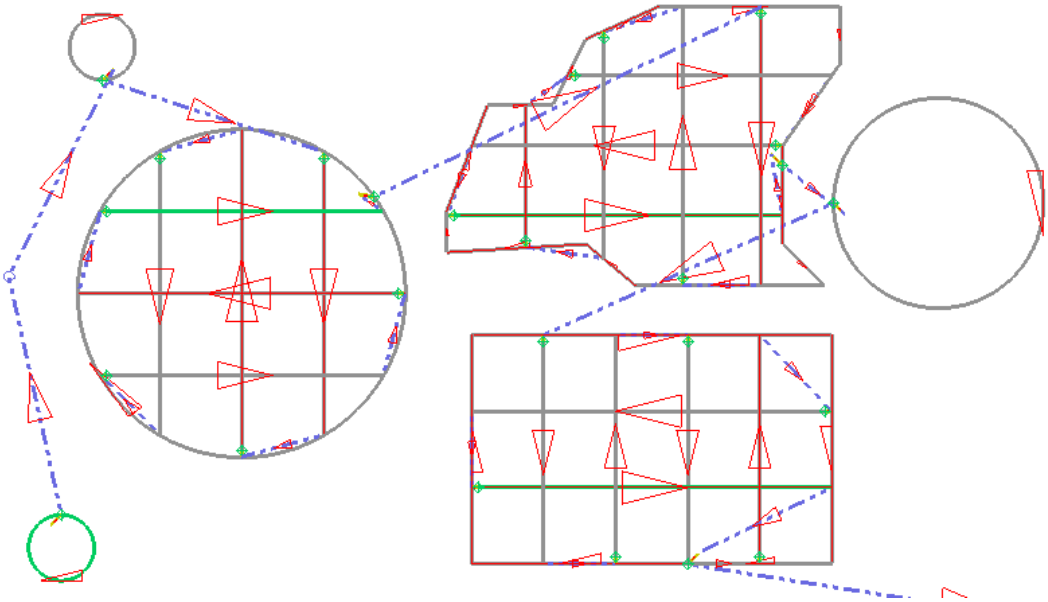
An **Internal skeleton global parameters** dialog box allows you to configure:

- The minimum size of holes.
- The gap between vertical lines and the gap between horizontal lines.
- The priming length.
- The retract value from the geometry.
- The head position during link trajectories.
- For full circles, you can choose whether the skeleton is to be cut continuously.
- For rectangles, you can choose whether the skeleton is to be cut continuously and specify a sag value between lines.



Question bar:

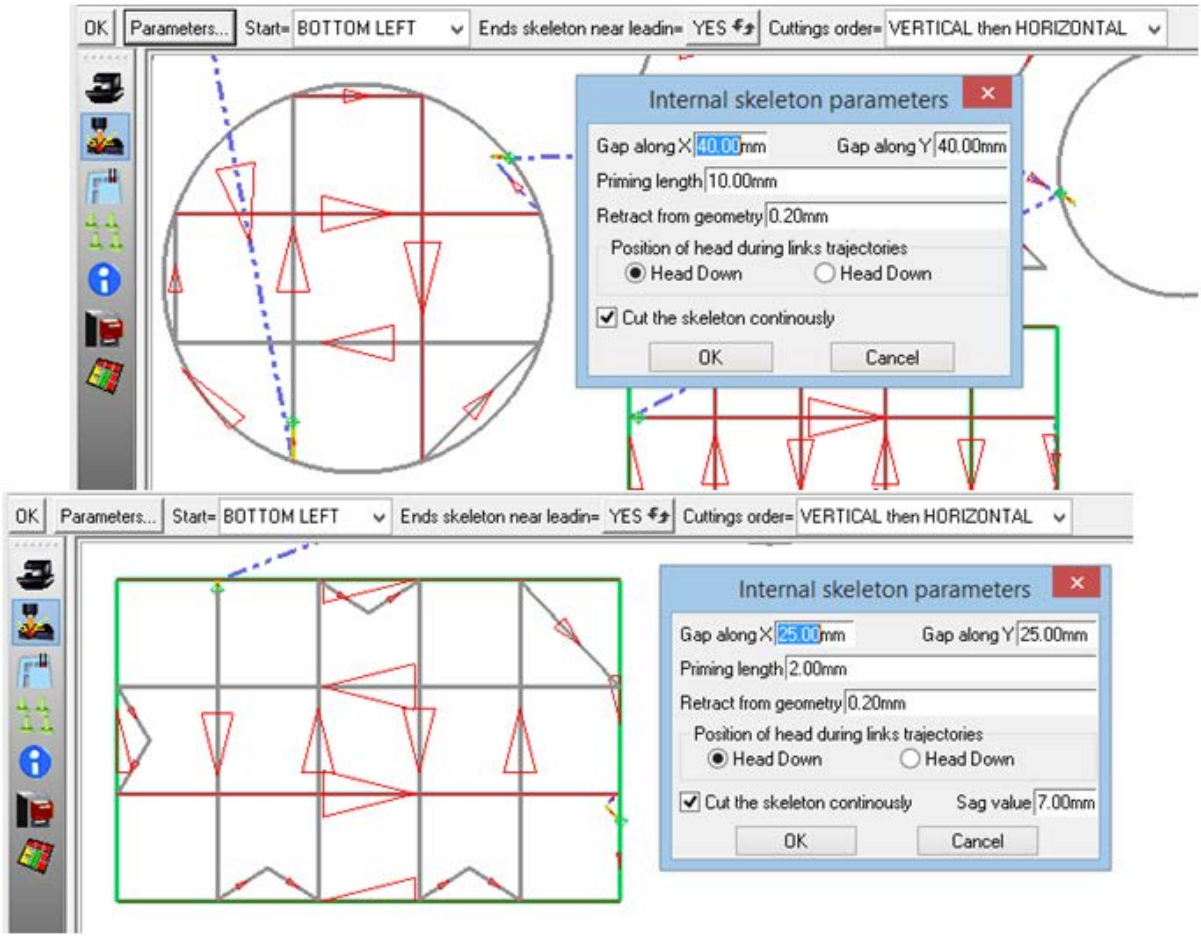
Example:



If the hole is already cut in a cutting path, when creating the internal skeleton on the hole, trajectories are automatically inserted before the hole cutting.

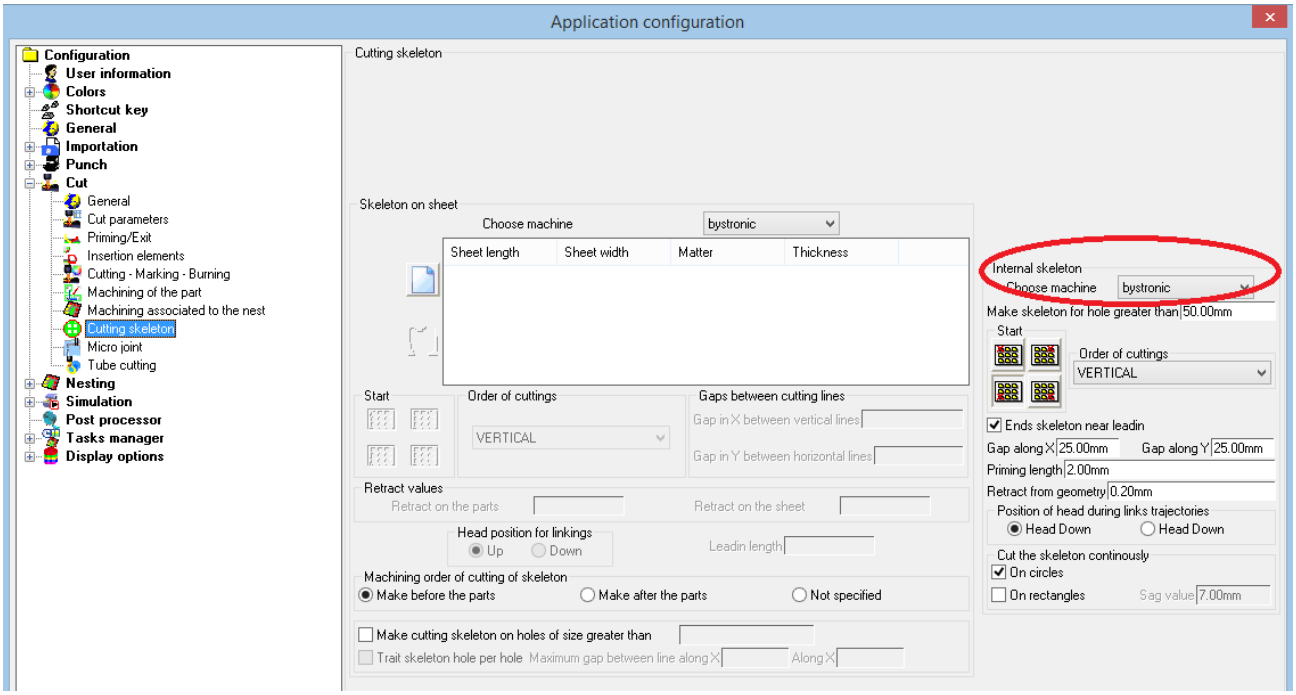
Example of continual cutting on the circle and the rectangle (in order to limit the primings):

Using the wrench  on the internal skeleton displays the question bar and the parameters that can be adjusted locally on the given hole.



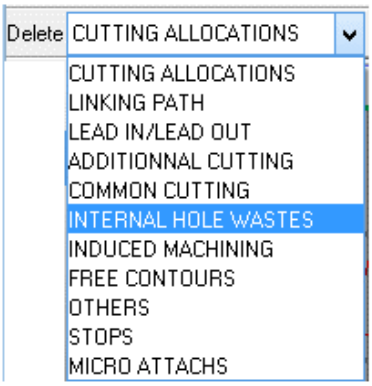
The internal skeleton cutting is associative to its geometry. If the support geometry changes, lines are recalculated, can be deleted, or new lines can be added.
The starting corner may also vary depending on the new number of lines in order for the last line to be as close as possible to the priming.

The configuration is done via a machine selection in **Tools** > **Options**.



Note that the configuration of the skeleton cutting relating to the sheet can now be accessed here (function of the nesting menu **Cutting** > **Common** > **Skeleton cut**).

Internal hole wastes can be deleted via the icon .



Note: The internal skeletons created in the Part document cannot be deleted in the nesting document.

Moreover, the holes on which an internal skeleton has been created with the function are ignored by the sheet skeleton cutting function.

Moving lead in/lead out dynamically along contour

Purpose of the function

A new function can be used to move the lead in/lead out or the limitation point dynamically along a closed contour.

Triggering the function

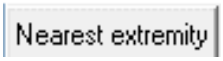
- Select the  icon, and click a lead in, a lead out or a limitation point.

Note: It is not possible to select a lead in/lead out on attach.

If the lead in is at one end and the limitation point does not exist, a new limitation point will be automatically created for the dynamic movement.

When the mouse cursor is located at one end of the segment, the lead in /lead out automatically moves to the next or previous segment.

- Click the new selected position.

The 

button enables the lead in/lead out to be placed at the segment end that is closest to the mouse cursor. The limitation point is then automatically removed.

There are restrictions on contours with micro-joints:

The dynamic segment change is not possible here.
The movement of the lead in/lead out will then be limited to the segment length if this segment does not have attach, or to the length limited to the attaches if the lead in to be moved is located between two attaches.

Multi-head machine

Head changes

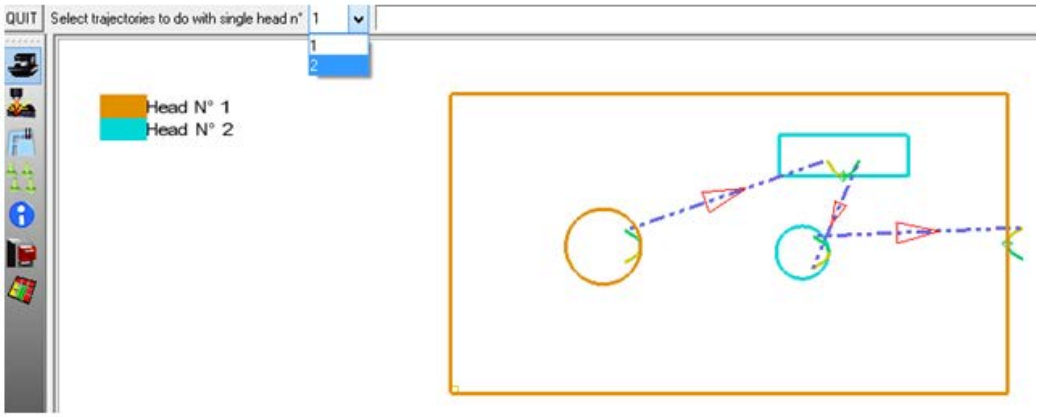
Purpose of the function

Possibility of choosing the head number on a given trajectory.

Triggering the function

Accessible from the **Cutting** menu > **Cutting** > **Head choice on trajectory**.

A caption appears at the top left of the part indicating colors for each head available on the machine.

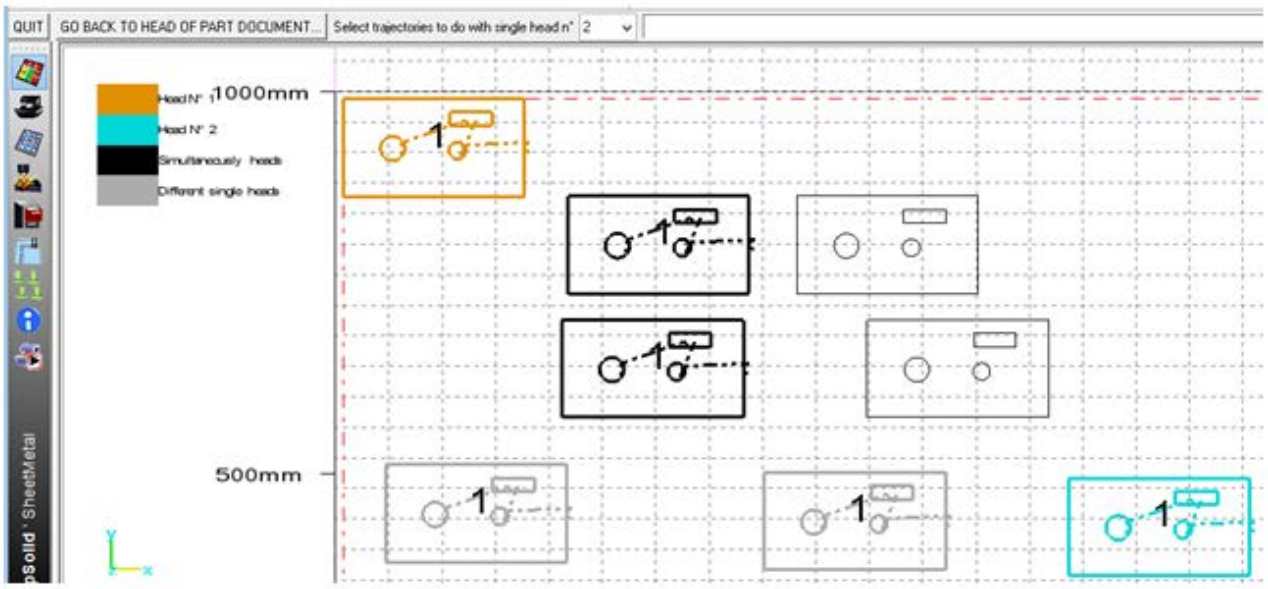


- Select the head number from the drop-down menu, and click the trajectory to be modified, which will then be colored as the selected head.

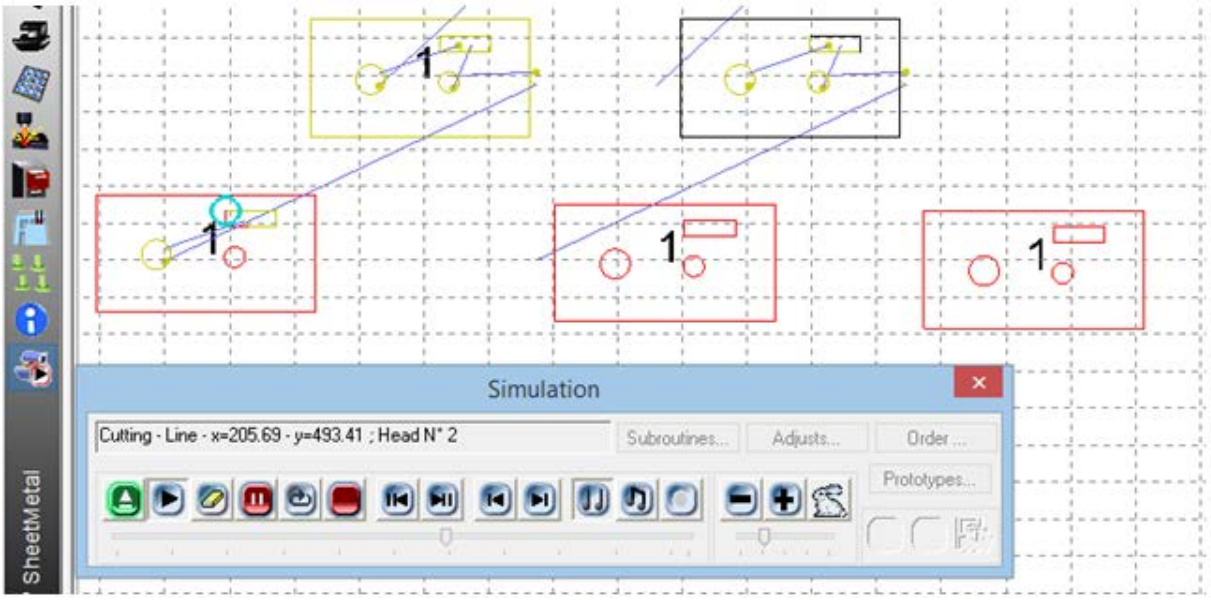
This function is available in the nesting document. Simultaneous head trajectories are displayed in black.
Part cutting instances performed with different heads are shown in dark gray.

In the nesting document, it is possible to overload on an instance the head number for all the instance's trajectories.

A button allows you to return to the part document's head configuration (on a trajectory or an instance).



Note: During simulation, the cutting head changes to the referenced color. The information is also displayed in the player.



Configuration

Head colors can be redefined manually in the *topzpchcut.cfg* configuration file.

```
# In multi-heads, when defining the head: identification colors of trajectories
# Word                                  Head_N°     Color_Index
ZCUT_TRAJ_MULTI_HEADS_COLOUR_HEAD       1       6   # ORANGE        Head N°1
ZCUT_TRAJ_MULTI_HEADS_COLOUR_HEAD       2       26  # CYAN          Head N°2
ZCUT_TRAJ_MULTI_HEADS_COLOUR_HEAD       3       18  # BROWN         Head N°3
ZCUT_TRAJ_MULTI_HEADS_COLOUR_HEAD       4       8   # YELLOW        Head N°4
ZCUT_TRAJ_MULTI_HEADS_COLOUR_HEAD       5       30  # MAGENTA       Head N°5
ZCUT_TRAJ_MULTI_HEADS_COLOUR_HEAD       6       10  # GREEN         Head N°6
ZCUT_TRAJ_MULTI_HEADS_COLOUR_HEAD       7       4   # RED           Head N°7
ZCUT_TRAJ_MULTI_HEADS_COLOUR_HEAD       8       27  # LIGHT CYAN    Head N°8
ZCUT_TRAJ_MULTI_HEADS_COLOUR_HEAD       9       14  # VIOLET        Head N°9
ZCUT_TRAJ_MULTI_HEADS_COLOUR_HEAD       10      17  # LIGHT GRAY    Head N°10
ZCUT_TRAJ_MULTI_HEADS_COLOUR_HEAD       0       2   # BLACK         Simultaneous multi-torches
ZCUT_TRAJ_MULTI_HEADS_COLOUR_HEAD       -2      3   # GRAY          Different head numbers in the instance
```

TopSolid'CutTubes - Tube cutting

Tube import (TopSolid 7)

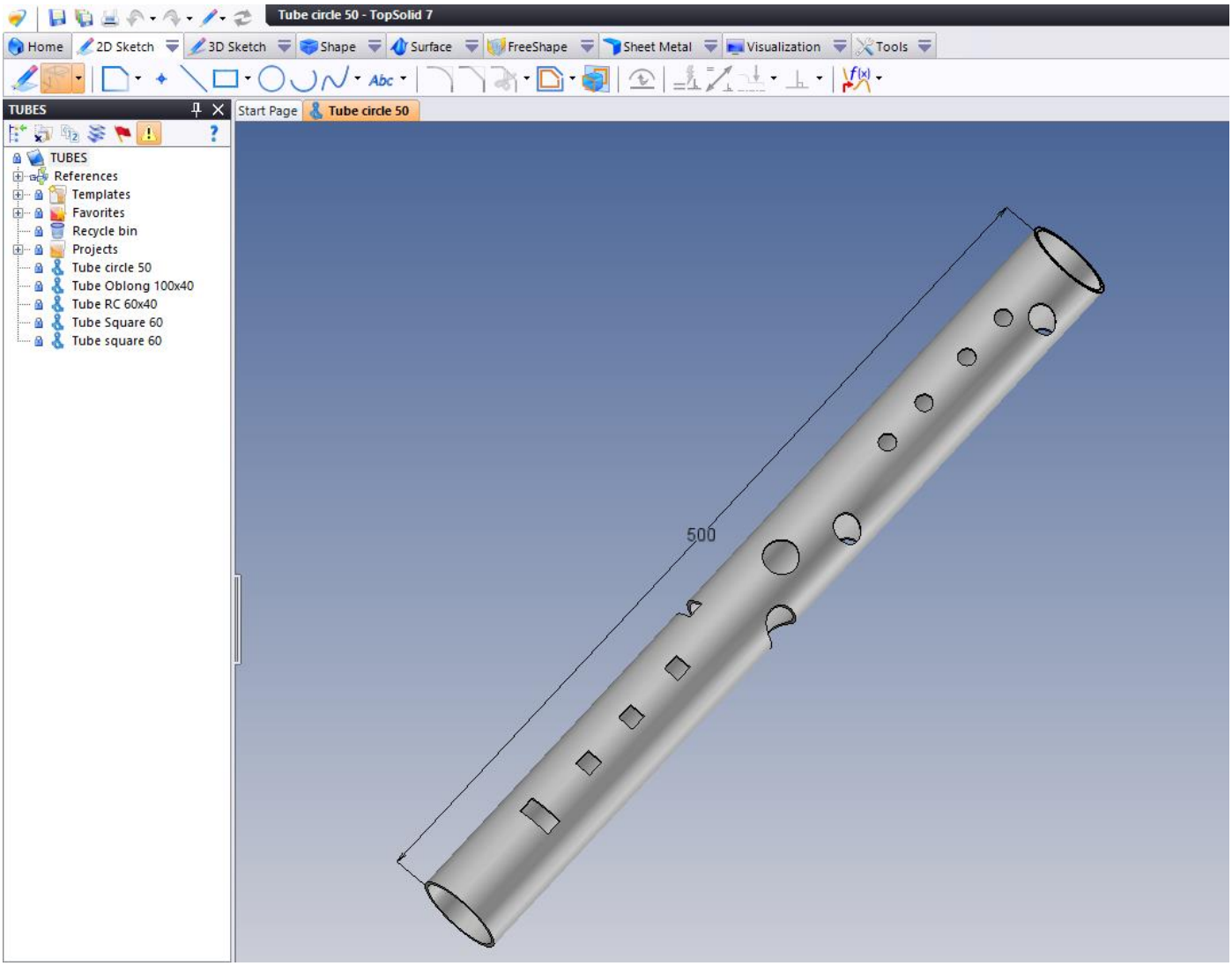
Purpose of the function

Possibility of importing in TopSolid'CutTubes tubes that were directly designed in TopSolid 7.

Triggering the function

In **Tools** > **Options**, the 3D TUBE machines must be defined and the **import of v7 unfolded parts** must be allowed.

Example of a tube designed in TopSolid 7:



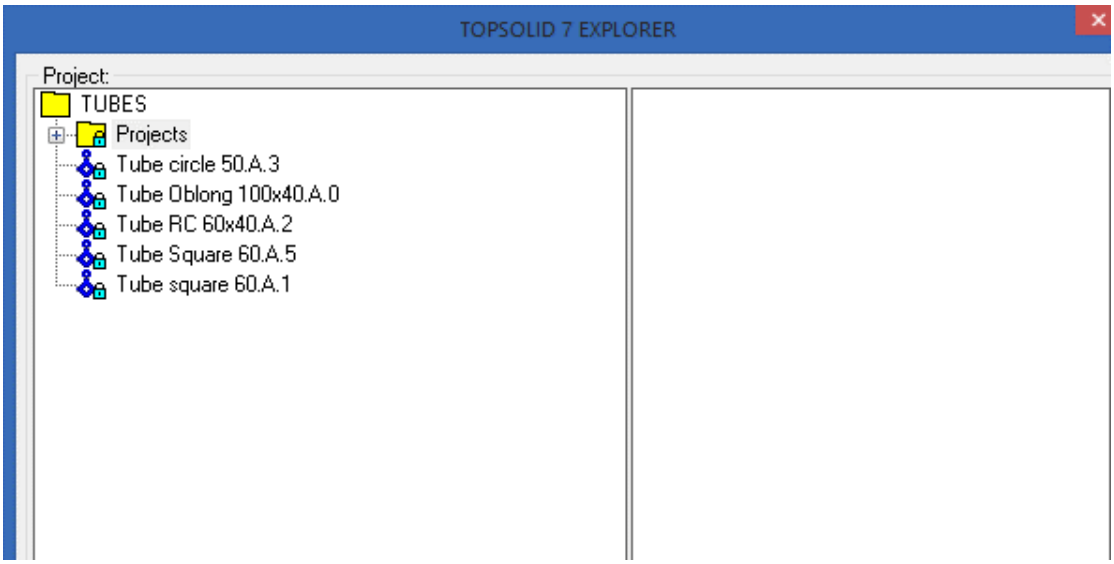
Importing the tube in the TopSolid'Tube CAM

- Use the **New task** function.

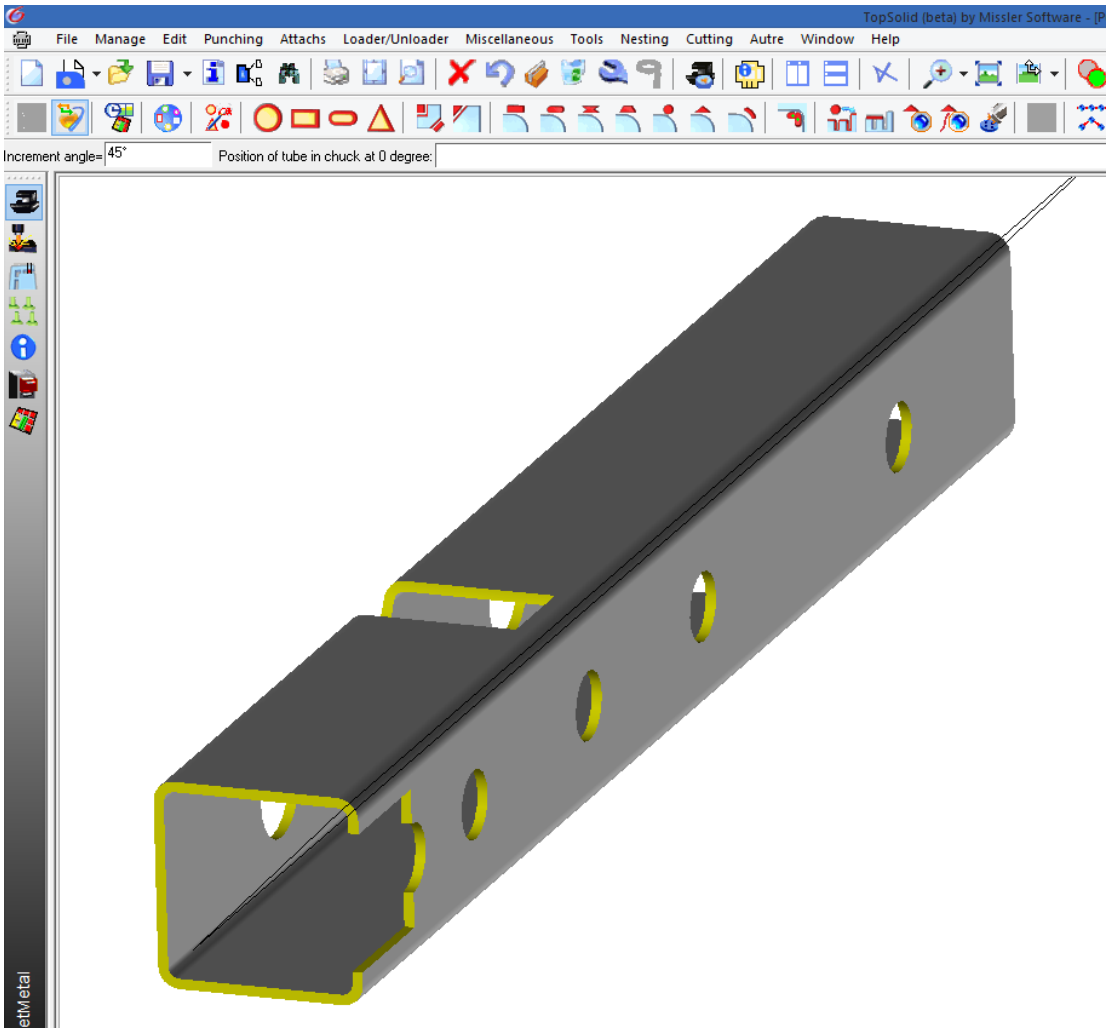
EXPLORER | V7 EXPLORER | CURRENT V7 PROJECT | V7 TUBE EXPLORER

- Choose **V7 TUBE EXPLORER**.

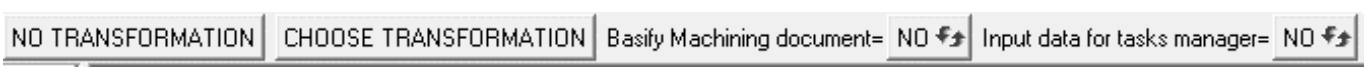
The Project Explorer appears. Only checked-in parts are available.



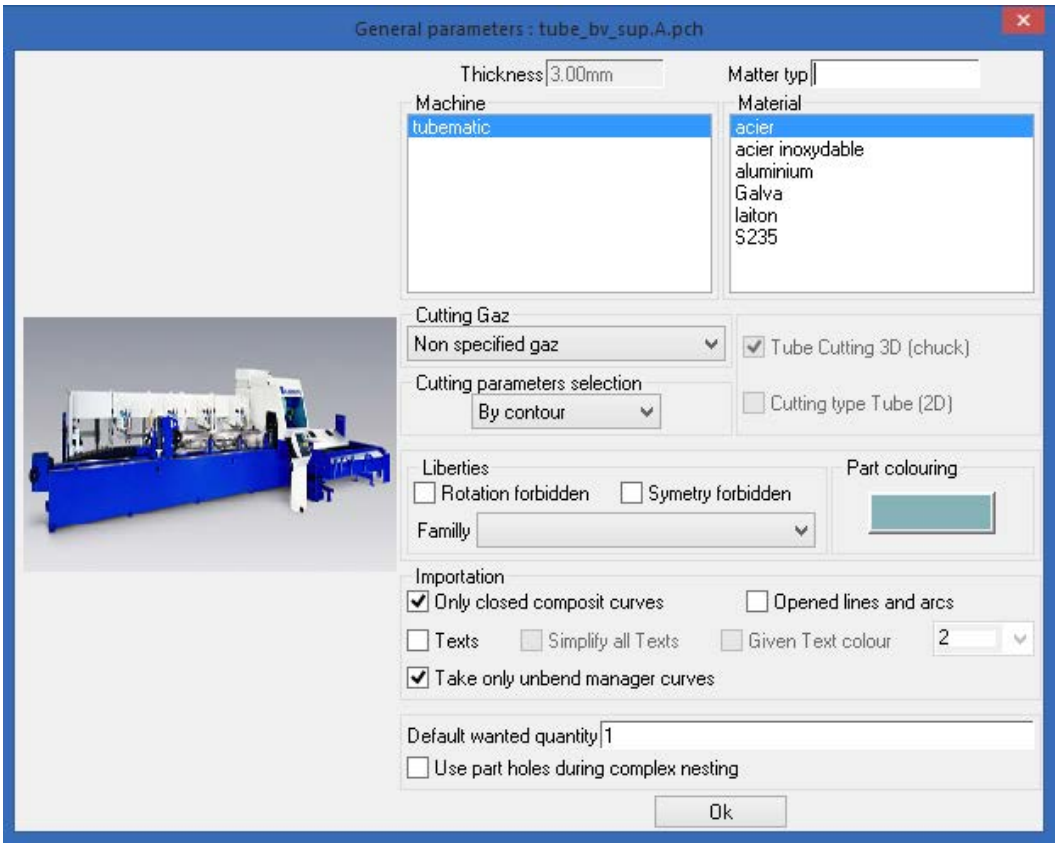
The tube is displayed on the screen to choose its position at 0° in the chuck.



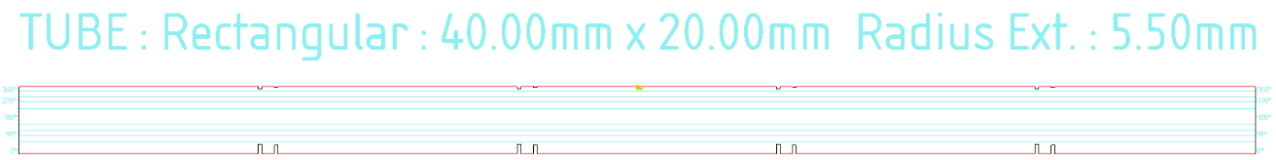
- Choose **NO TRANSFORMATION**.



- Select the machine.

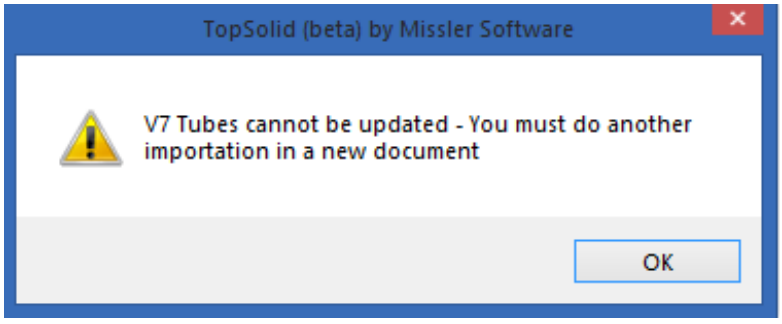


- Creation of the curves in the Part document.



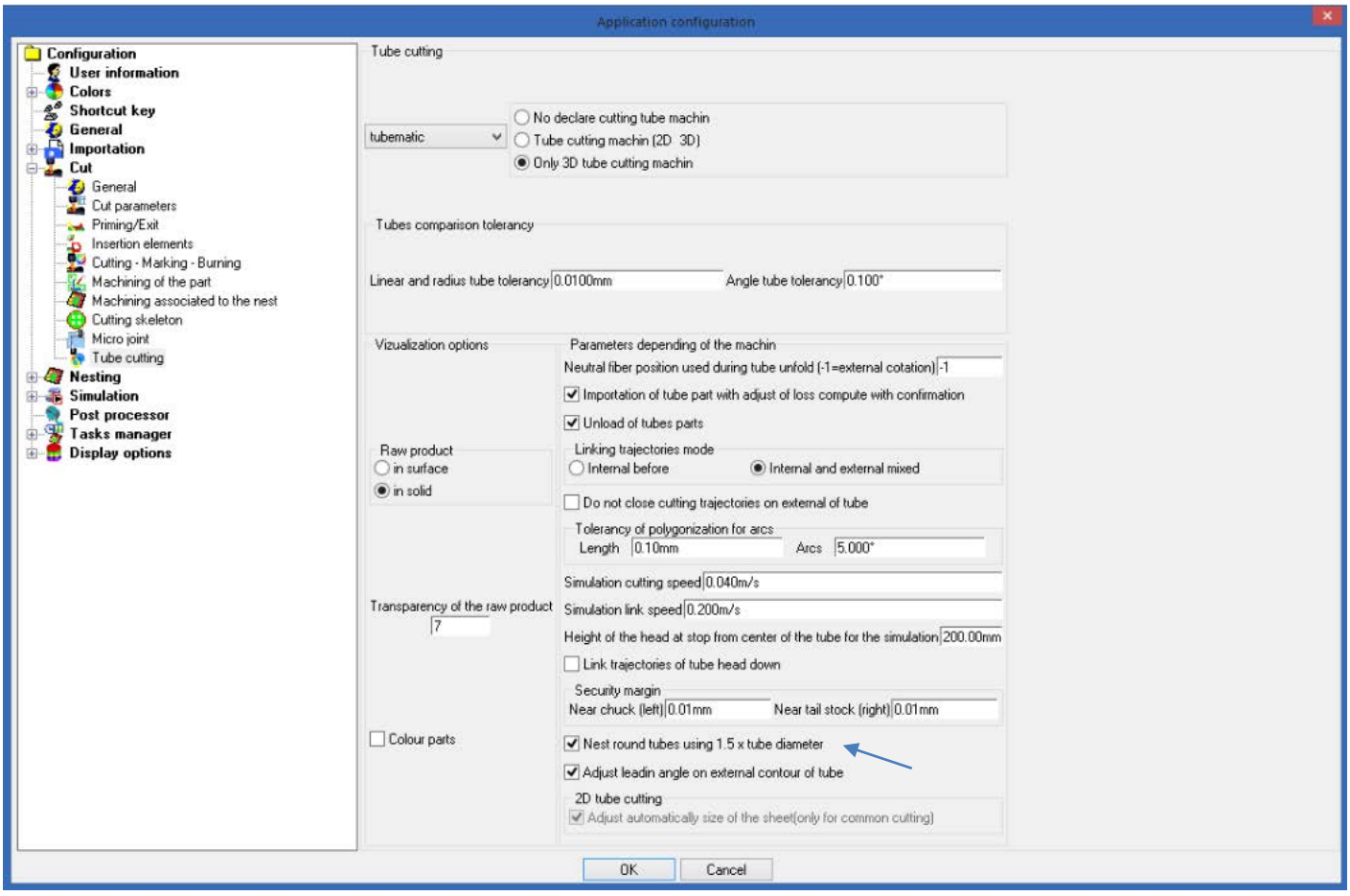
Note on v7 tube updates

If a change is made in TopSolid 7, the light turns red to indicate that the PCH part is not up to date. However, it cannot be updated. The following message is displayed:



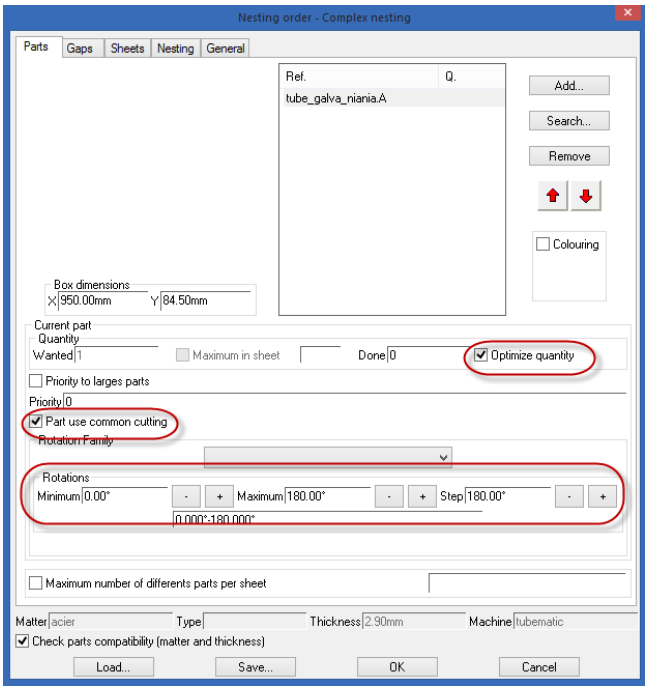
Improved common cutting nesting for round tubes

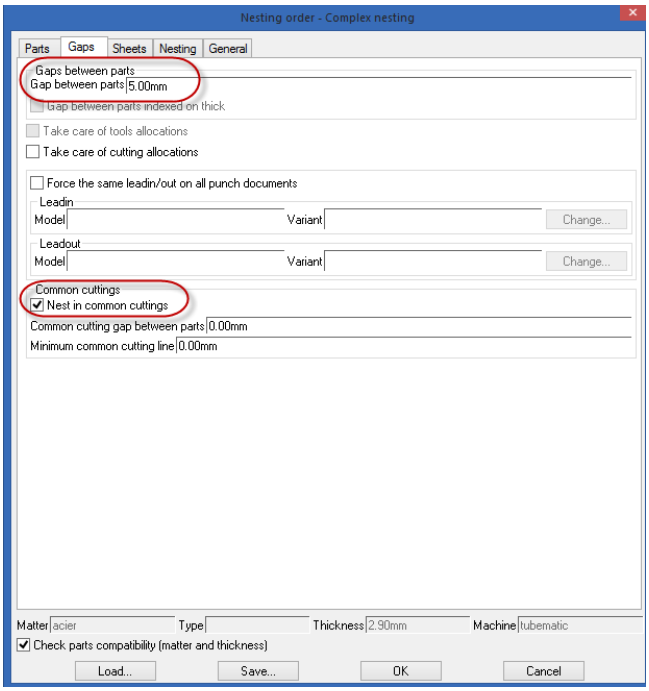
In order for the tubes to better nest within each another, you can choose to nest the round tubes on 1.5 times the tube diameter in **Tools** > **Options**.

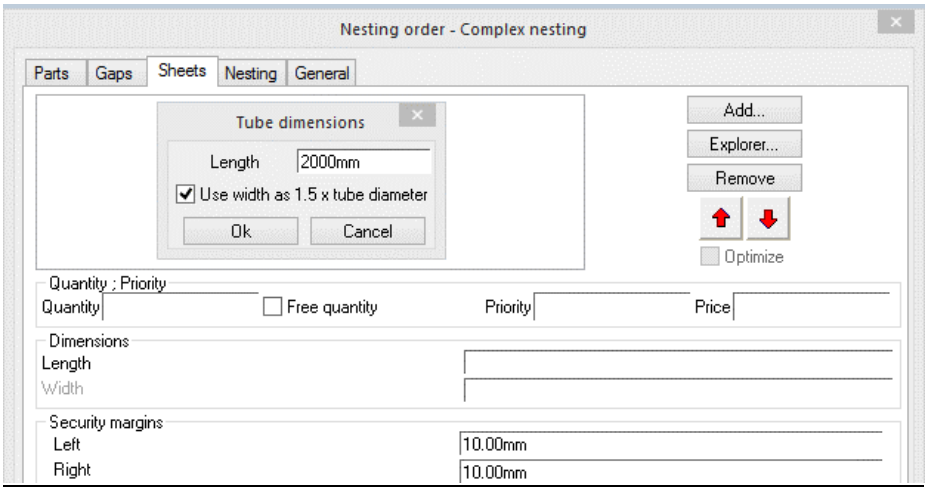


Improved complex nesting for round tubes

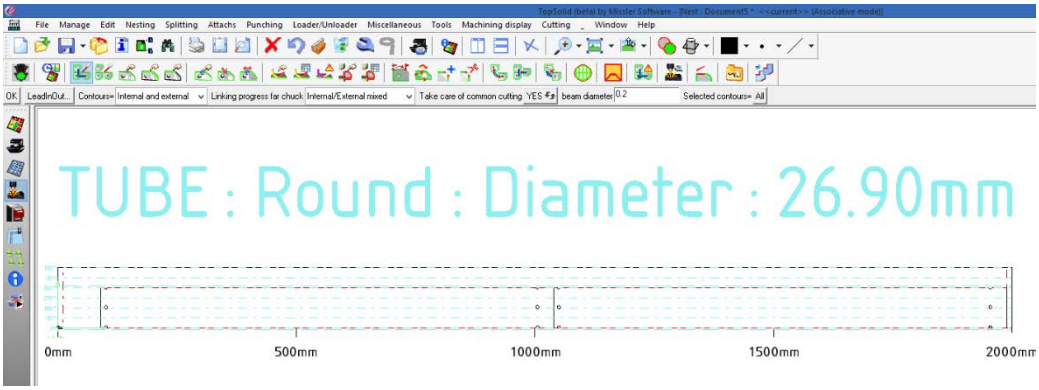
When the tube can rotate to 180°, it can be nested upside-down in common cuttings on 1.5 times the tube diameter.







A new option is available in the automatic path creation function and allows you to manage the tube common cuttings.



What's new in TopSolid'WoodCam v6.16

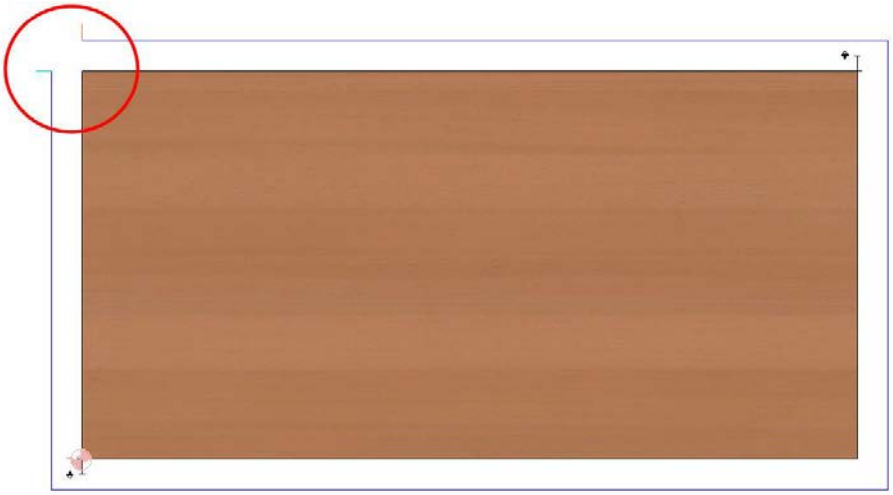
This section describes the new features in the version **6.16** of **TopSolid'WoodCam**.

Machine kinematics associative to 3D modeling

In previous versions, a change in the machine geometry required the kinematics to be redefined. The machine kinematics is now associative to 3D modeling. This allows the position of any machine elements to be modified without redefining the kinematics. For old machine models, tool holders must be redefined.

Radiusing and intersection between the closed curve's last and first elements

In the version 6.16, the path of a closed calibration is no longer extended by a tool radius; the starting point of the contouring operation is at the beginning of the path's first element and the ending point is at the end of the last element.



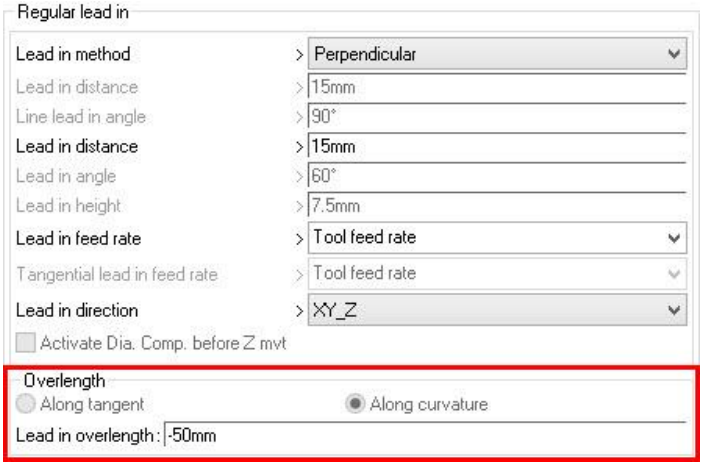
This path can be extended by using the **lead in** and **lead out overlengths**.

Negative overlength value in contouring operations

In all contouring derived operations (contouring, calibrating, sawing, groove, rabbet,…), it is possible to enter negative **lead in** and **lead out overlengths**.

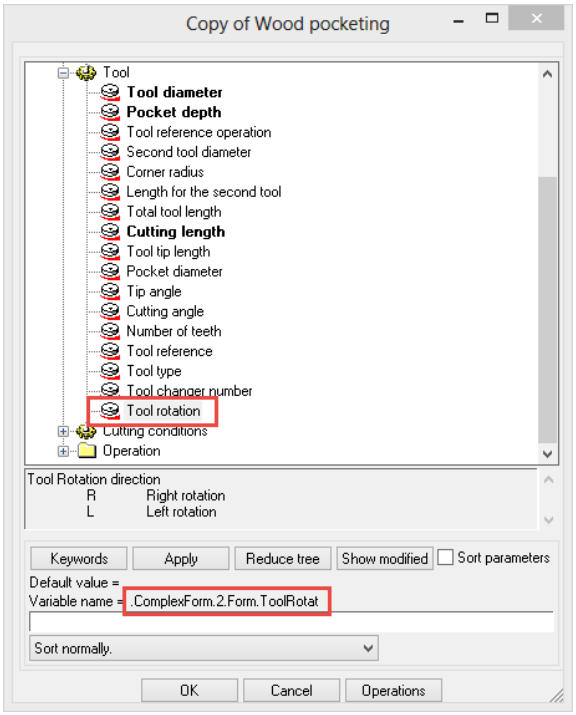
These negative values are necessarily defined in **Along curvature** mode.

The **Along curvature** mode is automatically selected when validating a negative value in the input field by pressing the **Tab** key.



Tool rotation direction variable

A new variable is now available in the processes and allows the rotation direction to be forced when selecting the tool.



.ComplexForm.1.Form.ToolRotat = "R" -> The selected tool has a clockwise rotation.

.ComplexForm.1.Form.ToolRotat = "L" -> The selected tool has a counterclockwise rotation.

.ComplexForm.1.Form.ToolRotat = " " -> The selected tool has a clockwise or counterclockwise rotation according to the first matching tool.

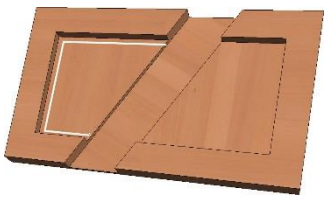
It is also possible to retrieve the value of this variable (for a formula or a comment).

Contouring cut by another operation

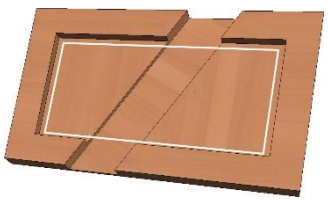
The concept developed in the version 6.14 for pockets is now available for contouring operations; a pocket crossed by another operation is now fully contoured.

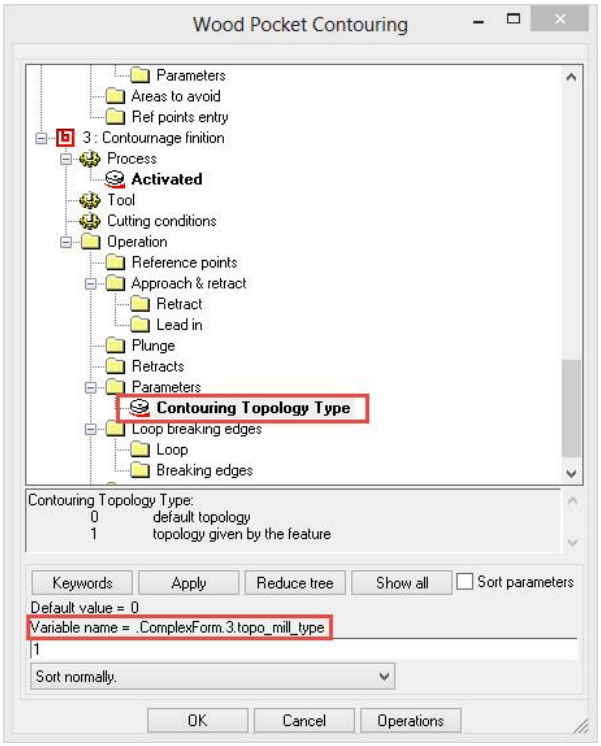
It is possible to choose in the process the type of topology to be used.

Topo_mill_type = 0 -> Default topology (the machining will be performed only on a portion of the cut pocket).



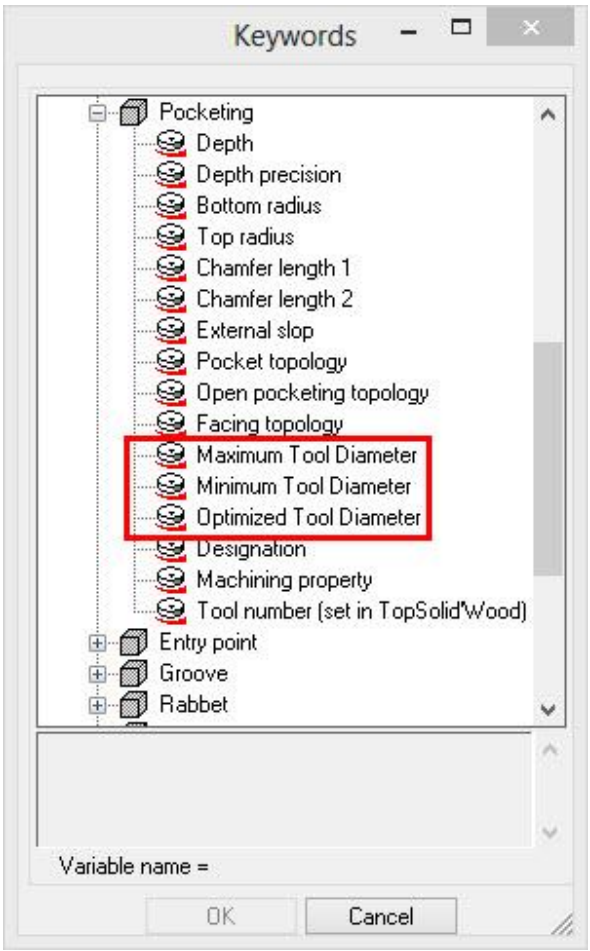
Topo_mill_type = 1 -> Topology given by the feature (the machining is performed as if no operation crossed the part).



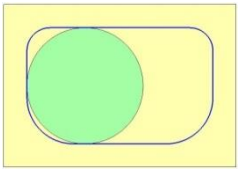


Publishing of the maximum, minimum and optimized tool diameter values for pockets

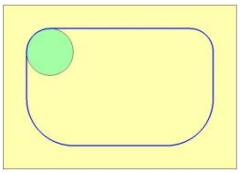
Three new variables have been added to the already existing pocketing list. They are available when editing the process parameters via the **Keywords** button.
These variables provide the maximum, minimum and optimized diameters of the pocket profile.



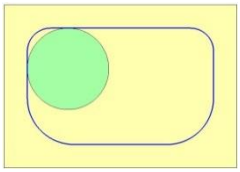
- **Maximum Tool Diameter** = Maximum circle diameter (different from zero) which is bitangent or tritangent to the sketch elements.



- **Minimum Tool Diameter** = Minimum circle diameter (different from zero) which is bitangent or tritangent to the sketch elements. The result is zero if the pocket is not radiused.



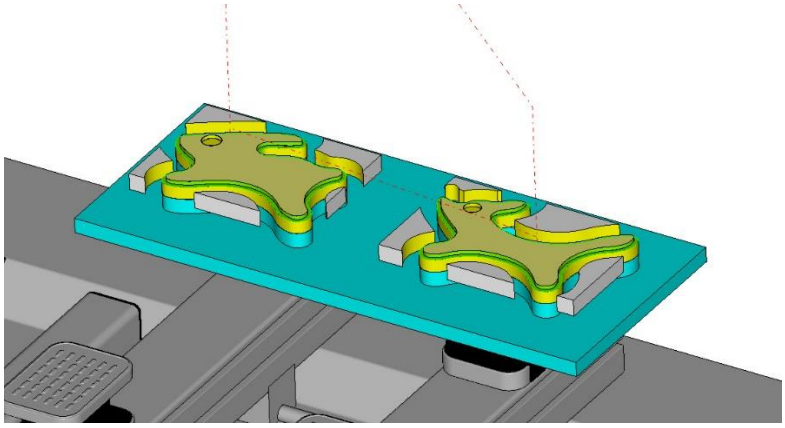
- **Optimized Tool Diameter** = Arithmetical mean of maximum and minimum values, i.e. (maximum diameter + minimum diameter)/2.



The values of the above parameters can be accessed directly via the **Info** button available after the topological analysis of a face (if this face has a pocket topology).

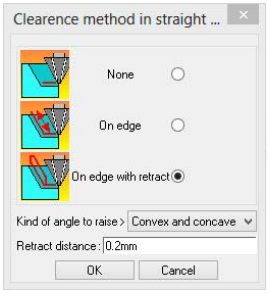
Verification on several parts

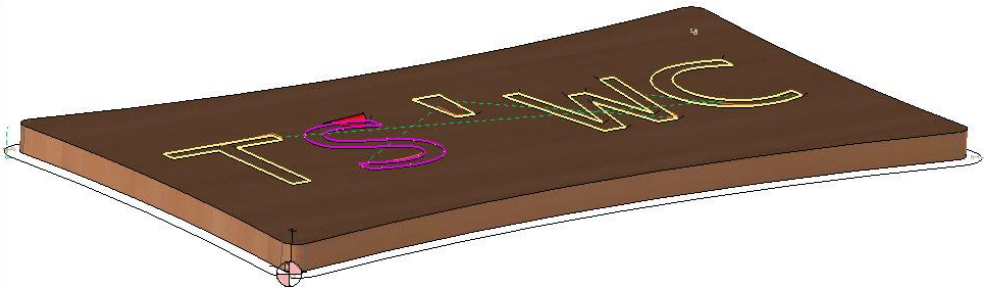
This new version allows several parts to be **verified** . This is very useful for verifying machinings on nested parts or on parts of a set.



Clearance method in straight angles for engraving

The feature that allowed clearance in straight angles of a **multi-contouring** in the version 6.15 is now available in the **Engraving** function, and works the same way.



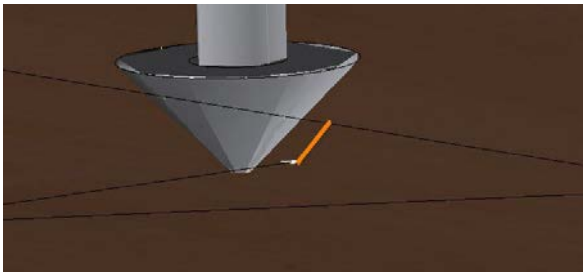


Two clearance modes are available: **On edge** and **On edge with retract**. It is also possible to choose the kind of angle to be raised: **Concave**, **Convex** or **both**.

The **Clearance method in straight angles** option can be accessed from the **Main** tab of the **Engraving** and **Multi engraving** operations.

Slope plunge and retract for engraving

In the **Engraving** function, the slope plunge and retract can now be configured.

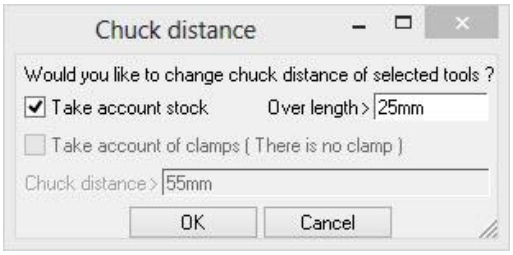


This works the same way as for the contouring.

Minimal tool length for all operations

The **Minimal tool length** option allows you to modify the chuck distance of tools for some or all operations in the **Operations manager**.
It can be accessed by right-clicking the list of operations.

It is also possible to calculate the minimal tool length considering the height of the stock and/or clamps.



Retrieval of nested part's properties

When repositioning multi-nested parts, it is possible to define the name of the machining file via a BOM file which must include the **WOO_CAM_FILE_NAME2** property.
This BOM file must be declared in **Tools** > **Options** > **Configuration** > **TopSolid'Wood configuration** > **File** > **Parts selection sorting** > **TopSolid'WoodCam export**.

If the WOO_CAM_FILE_NAME2 property is not defined in the BOM file used, or if no BOM file is declared, the designation of the turned-over part is taken into account as: *Designation_turned-over.wod*.

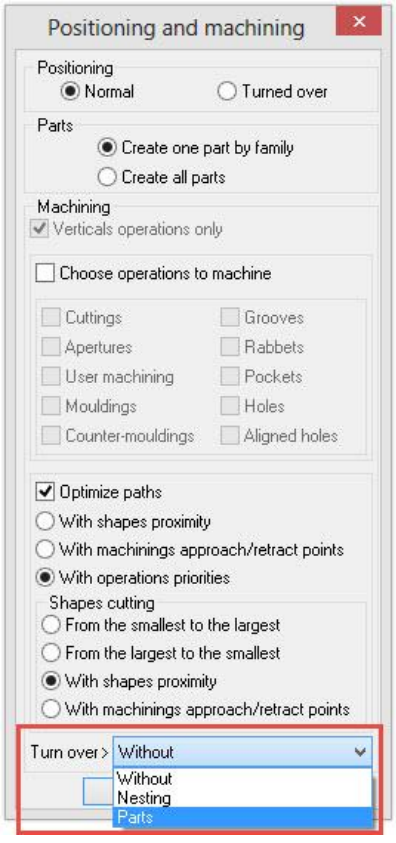
Nesting turn-over method

In multi mode, the turn-over method can be configured. This window is accessible by right-clicking the nesting, and then selecting **Positioning and machining**.

In the red box below, three turn-over options are available:

- **Parts**: Each part with operations positioned on faces other than those on top (side and bottom) is automatically turned over.
- **Nesting**: The whole panel is turned over.
- **Without**: No element is turned over.

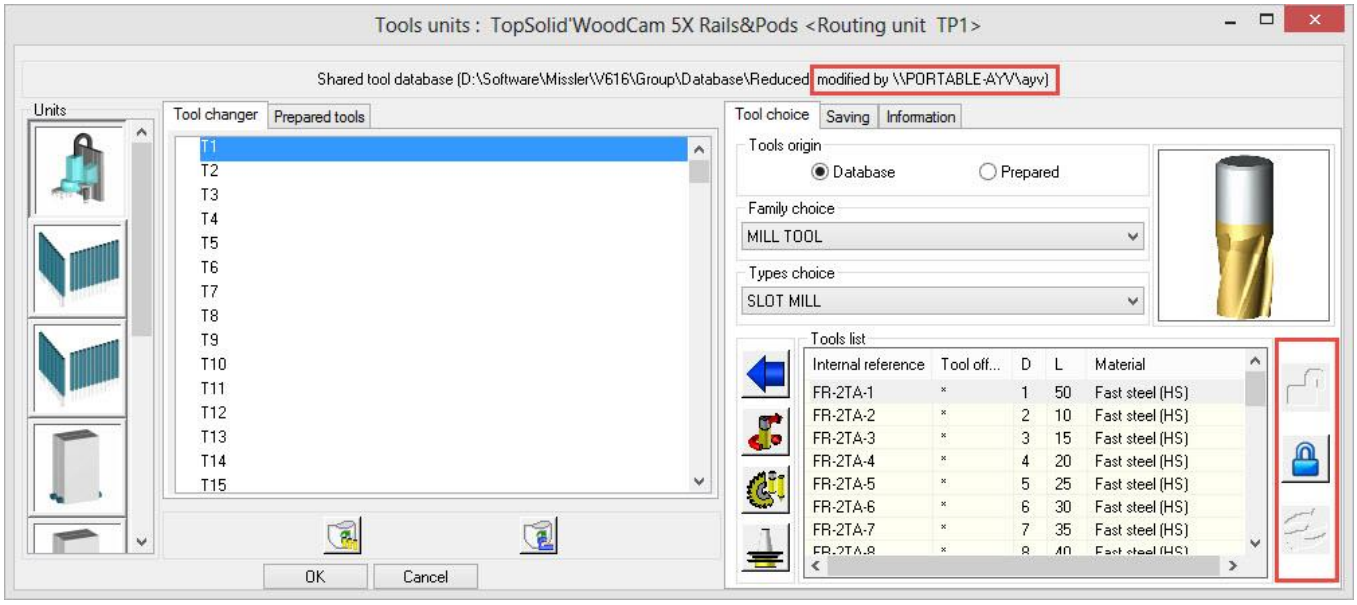
This way, the method can be configured for each nesting of the list of parts.



Shared reduced tool database (Group)

When the reduced database is stored in **Group**, three new icons appear at the bottom right of the **Tool manager** .

By default, the user cannot modify the tool database (no change in cutting conditions from the tool changer or the abacus, no change in materials, no saved prepared tool lists).

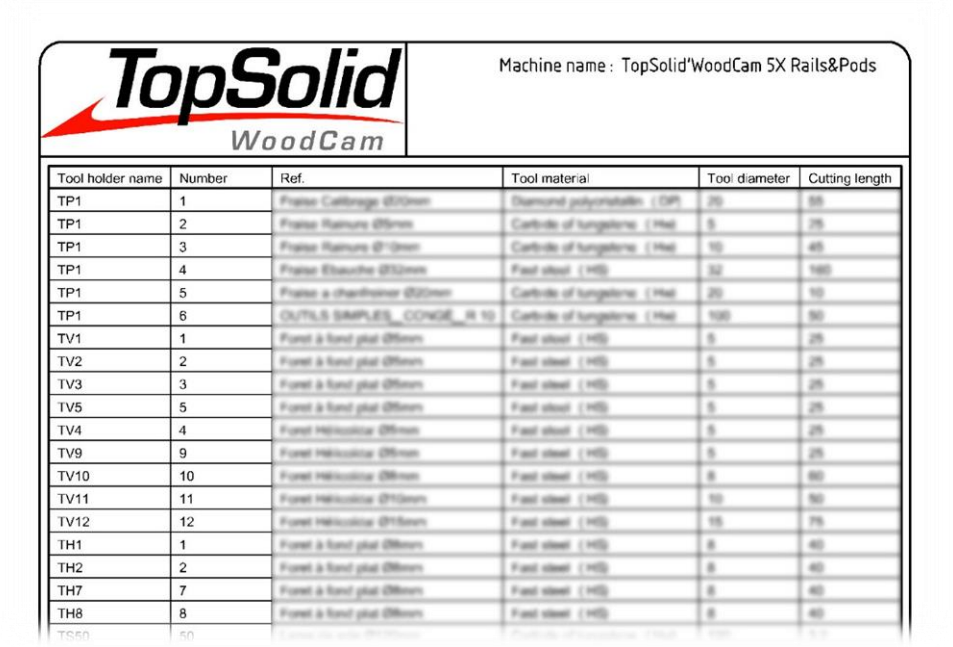


These three icons will help you manage the changes made to the group database.

- The **Allow access to the database** icon gives users access to the reduced database for modification. The user ID is displayed in the upper section of the window for all the users connected to this database (the **user 1** can modify the database, the **other users** can only read it).
- The **Close access to database** icon allows the database to be closed after modification and allows any users to load the new database (the **user 1** closes the access to the database, **all users** can access the database again via the **Allow access to database** function).
- The **Update database** icon allows all users connected to this database to update it in the current session (**all users** can load the database modified by the **user 1**).

Sorting by tool unit in workshop documents

In workshop documents, the tool tables are sorted by tool unit, and then by tool number.
Spindles of old machine models must be initialized to make sure the tool units are properly sorted.



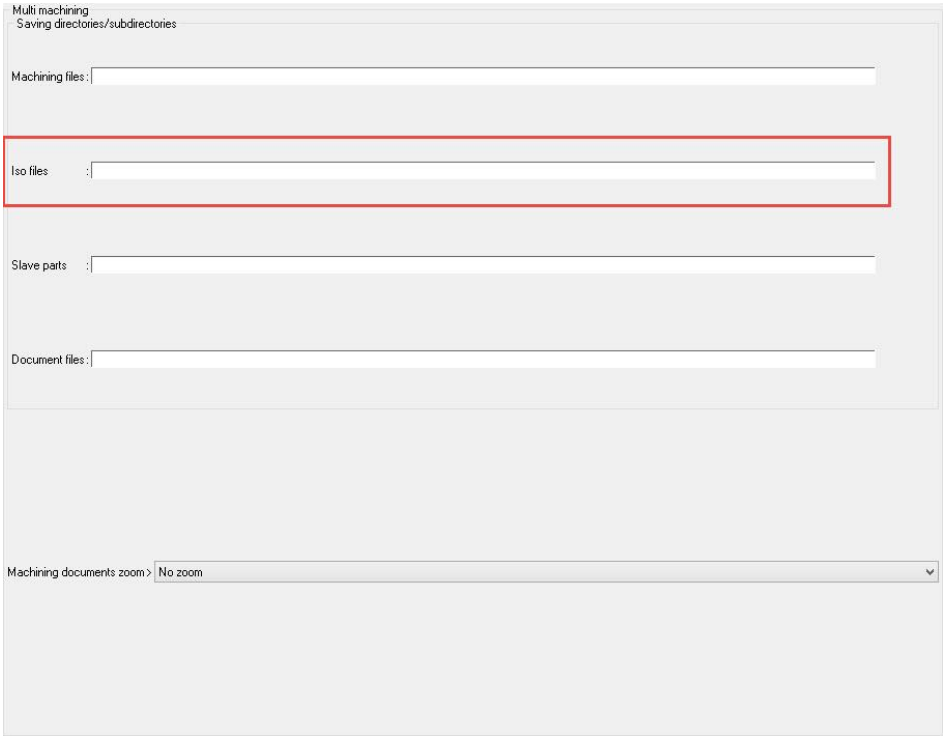
TopSolid WoodCam

Machine name : TopSolid'WoodCam 5X Rails&Pods

Tool holder name	Number	Ref.	Tool material	Tool diameter	Cutting length
TP1	1				
TP1	2				
TP1	3				
TP1	4				
TP1	5				
TP1	6				
TV1	1				
TV2	2				
TV3	3				
TV5	5				
TV4	4				
TV9	9				
TV10	10				
TV11	11				
TV12	12				
TH1	1				
TH2	2				
TH7	7				
TH8	8				

ISO directory in multi-machining mode

The "ISO" destination directory is no longer created by default in multi-machining mode.
A default directory or path can be defined in **Tools** > **Options** > **Routing configuration** > **Multi machining**.



Operation parameters for post-processor

The post-processor can enable the generation of operation parameters via a **PDB_CONFIG**.

The list of parameters is created before the start of the tool path, since the number of parameters can be high. If the size of the event exceeds 1024 characters, the following parameters will be sent in a new **PARAMETERS=OPER** line. Each parameter contains a key and a value; they are available in the PDB with the prefix **Db.Oper**.

```
*N0006  OPER=1  CLASS=OP_MILLING_AXIAL_MILL OFFSET=0  CUTCOM_OPER=NO  COMM=Surfaçage de la
*N0007  PARAMETERS=OPER Db.Oper.dpas=43.54  Db.Oper.offset_side=-1  Db.Oper.offset_side_isl
*N0008  PARAMETERS=OPER Db.Oper.plunge_radius_min_in=0.0311 Db.Oper.full_z_down_type_out=2
*N0009  PARAMETERS=OPER Db.Oper.facing_island_minimize_nb_pass=1  Db.Oper.facing_island_dpa
*N0010  PARAMETERS=OPER Db.Oper.facing_island_plunge_radius_min_in=0.0311 Db.Oper.facing_is
*N0011  PATH_APPROACH=BEGIN TOOL_APPROACH=CTUR  FACE_APPROACH=XY-Z  NAME_APPROACH=DEFAULT
```