



# TopSolid v6.16 What's New

MASTER YOUR MANUFACTURING PROCESS

#### TopSolid 2015

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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).
Reference curves:

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

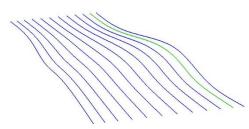
• Set the Morphing options. OK Give priority to ends= YES + Interval mode= YES + Number of resulting curves= 6

**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.

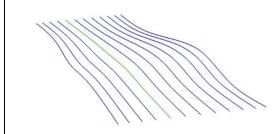
Once the number of curves is modified, the result is the following:

Give priority to ends= YES +

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

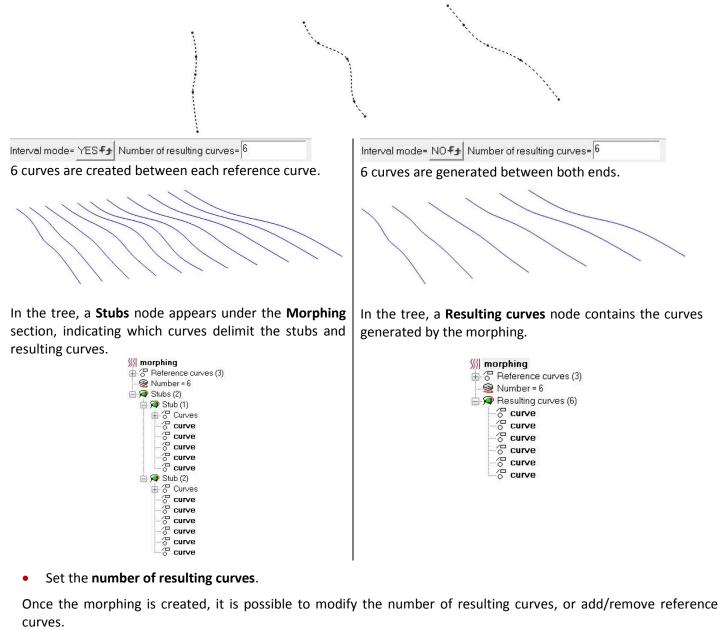


Give priority to ends= NO**fy** The green curve has not kept its position.



**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:



- 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.

<u>Note</u>: 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.

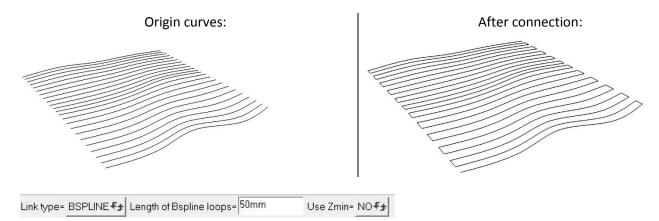
# Connection

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.

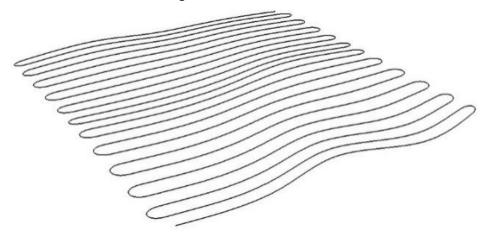
- Launch Curve > Other curves > Connection
- NU
- Select the **reference coordinate system**. The best is to select the coordinate system on which curves have been drawn.
- Then click the curves to be connected and validate with **OK**.
- Set the options:
  - **Keep curves orientation**: Used to keep 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.

OK Keep curves orientation= NO++ Link type= BSPLINE++ Length of Bspline loops= 0mm Use Zmin= NO+

The connections created between the curves are lines.



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.

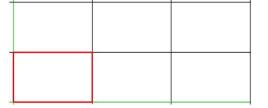
Here are two green lines that are repeated on X and Y in order to generate a grid.

	2			
4		3		
		3		
	4			

• 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.

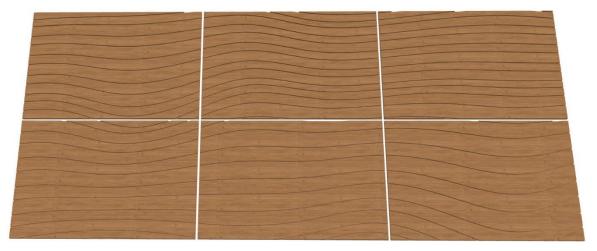


- It is possible to stop the function or make a **serial copy** of this cell.
- If the previously selected curves have been generated through repetition, the Automatic button automatically selects the curves in order.

• 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).

Main     Favorite     Main set     Entities     Layers          ☐		
⊕       \$ Stub (1)         ⊕       \$ Stub (2)         ⊕       \$ Stub (3)         ⊕       \$ Stub (4)         ⊕       \$ Stub (5)         ⊕       \$ Stub (5)         ⊕       \$ Stub (6)		

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

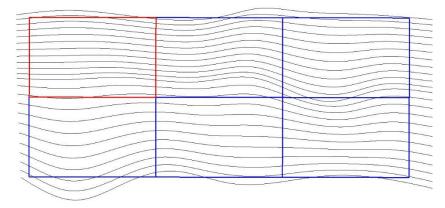


#### Trimming

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.

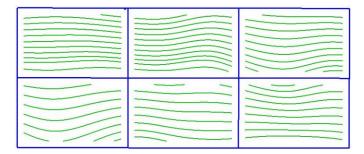


• If necessary, enter a gap between the trimming curve and the trimmed curves.

OK EXTRUSION DIRECTION Gap= 100mm

**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.

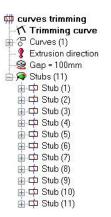
Still using the same example, the curves generated by the trimmings are shown in green. A gap has also been added.



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

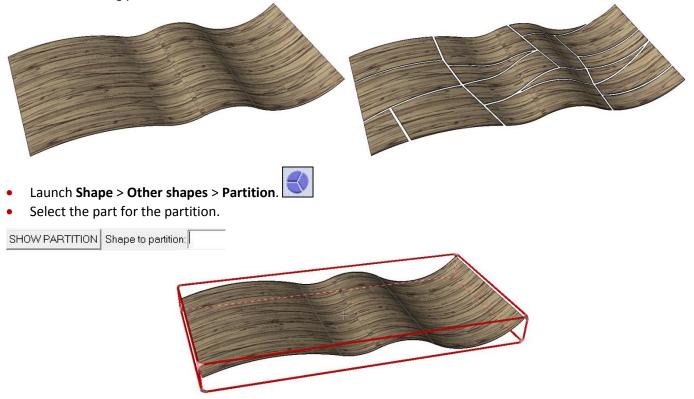
- Edit one of the resulting curves from the tree.
- To add a curve to be trimmed, right-click > Insert on Curves.
- A double-click on the **Gap** line modifies the gap value.

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.



What's new in TopSolid'Wood v6.16

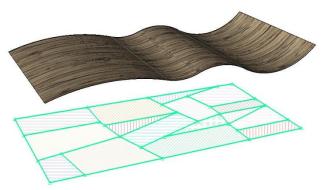
• Select closed curves to partition the part.

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

Curves:

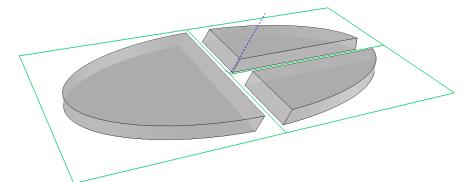
• Once all curves are selected, validate with **OK**.

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

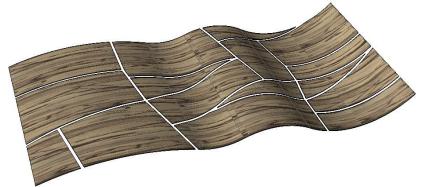


Enter a **gap** between each partitioned part, and then validate with **OK**.
 EXTRUSION DIRECTION Gap= 20mm

**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.



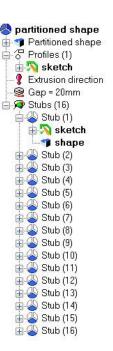
The partitioned shapes are then generated.



- 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.

<u>Note</u>: 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.

Modes: Additional 🗸

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

Set the overvaluation mode for the definition of panel supports.

```
Modes for panel supports: Additional 🗸
```

**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.

MAIN ASSEMBLY Depth: MULTI LEVEL V Filter bom by criteria= no filter V Length always highest dimension = YES + Select elements to use:

**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.

WIZARD Named coordinate system= ABSOLU	TE COORDINATE SYSTEM SAWING-UP FI	ME New texture frame:
--	-----------------------------------	-----------------------

**<u>Note</u>**: 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.

<u>Note</u>: 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.

Define multiple parts										
Selection type O Manually O By criteria O By properties										
Criteria Properties										
Type Materia		() Attop isver	(•) Muni lev		<b></b>		COLAP	JE ALLI		
		- ·	e right click to se		1,0000000000000000000000000000000000000		10028 # 0			
Index	Qua	Designation	Reference	Material	Thic	Length	Width	Туре		
- ASSE										
	1	Sink		420 HV	17	280.00	195.00			
	1	Shelf		beech	19	732.50	532.50			
	1	Kitchen counter		ash	19	1200	600.00			
<								>		
0 Selected p	0 Selected part Configure									
			ОК	Cancel						

#### 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.

Index	Quantity	Designation	Reference	Material	Thickn	Length	Width	Туре
ASSE								
-	1	Sink		420 HV	177.50	280.00	195.00	
1	1	Shelf		beech	19.00	732.50	532.50	
<u>.</u>	1	Low cabinet						
	1	Joue Gauche		beech	19.00	850.00	450.00	
	1	Joue Droite		beech	19.00	600.00	450.00	
	1	Dessus		beech	19.00	850.00	450.00	
	1	Dessous		beech	19.00	600.00	450.00	
L.	1	High cabinet		Ash e	700.00	650.00	300.00	
C								>
Selected	part						Configure	>>

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**.

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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.

						Define n	nultiple	parts	
Selection ty Manually		By criteria			() Ву рі	roperties			Designation : Low Cabinet
Criteria Pr	operties								Reference :
Туре	>							~	
Materia	Ash	european						ý	Processing >
Coating	• •							~	Part category>
Thickne		172						ý	General Part types Description Bill of material
									Add to sawing-up
Proper	y > Bom	om operations		~				~	Add machining frame
Bill of mate		choice At top level	Multi lev		EVD	AND ALL	COLLAF		Add draft frame
	1		~ ~	elect / configure	-		COLLAP	OE ALL	Assembly nature
Index	Qua	Designation	Reference	Material	Thic	Length	Width	Туре	Sub-assembly
ASSE	1	Sink		420 HV	17	280.00	195.00		Content
-	1	Shelf		beech	19		532.50		No modification
÷.	1	Low cabinet High cabinet		Ash europ	70	650.00	300.00		Insertion mode in assembly document
	1	Bottom + groove		melamine	12		597.00		
									Insert in main assembly
									Dimensions
									Respectaxis
									C Length always highest dimension
									No modification
									Dimensions Values
<								>	
1 Selected p	part					С	onfigure	۰۰	Apply
						ОК	Can	cel	

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.

- ASSEMBLY						
-	1	Sink	420 HV	17	280.00	195.00
-	1	Shelf	beech	19	732.50	532.50
÷.	1)	Low cabinet		86	619.00	450.00
÷.	1	High cabinet	Ash europ	70	650.00	300.00
	1	Bottom + groove	melamine	12	847.00	597.00
La	1	Bottom	melamine	12	847.00	597.00

#### **Define set**

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

OK Set only contains one part. Modify this p	part characteristics= YES <b>#</b>
--	------------------------------------

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.

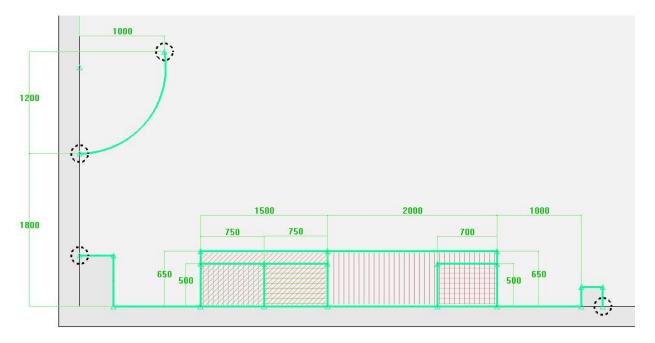
Configurators			
	Check all	Uncheck all	
Single units as sub-assembl	ice for :		
Matters configurator	les for :		
Properties configuration			
Edge configuration			
Laminate configuration			
Define multiple parts			
Automatic assembly			
Machining export			
Cut export			
Export to Triviso			
<ul> <li>Export bill of material</li> <li>Multi drawings</li> </ul>			
Nestings			

#### Improved sketch display

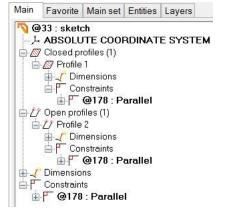
The display of sketches from the tree has been improved to make their comprehension easier. The editing of a sketch now displays:

- 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.



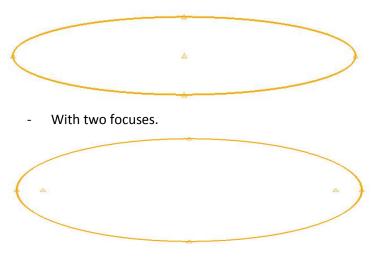
# Ellipse in the sketch

Ellipses can now be created in a sketch.

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

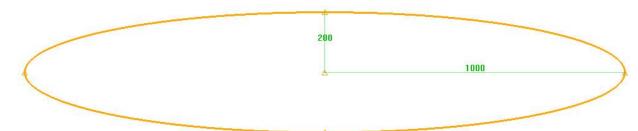
It is possible to draw two different types of ellipses:

- With a center point.



#### 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.

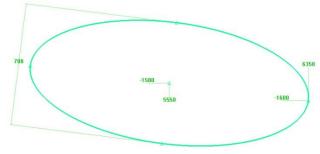


#### Ellipse with two focuses

- 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.



**<u>Note</u>**: The **Share thickness** in 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.





<u>Note</u>: 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_THICH	KNESS 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

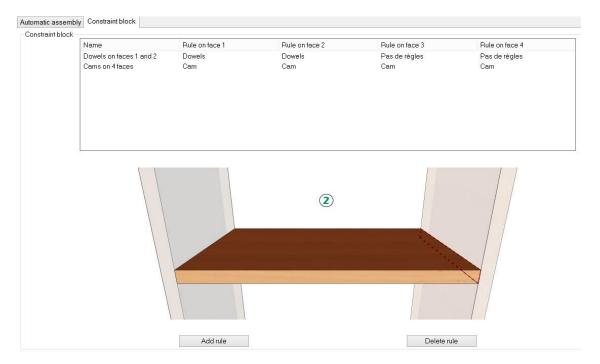
It is now possible to automatically assemble the constrained blocks using automatic assembly rules. Assemblies are created after completion of the function, after distribution.

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.

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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.

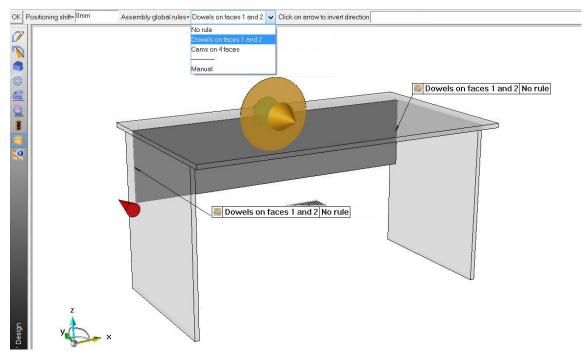
#### • Create a constrained block.

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.

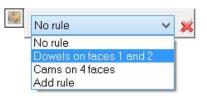


Note: These labels consist of two areas:

- 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.

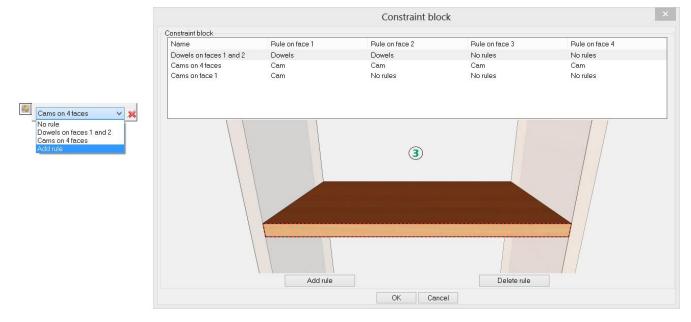
- Select Manual in the drop-down list
- Define the rule to be applied to each face.

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**.

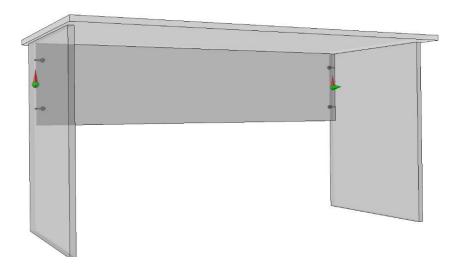
<u>Note</u>: The rule will be added to those created in Tools > Options > TopSolid'Wood configuration > Automatic assembly > Constrained block.



• 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**.



<u>Note</u>: 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**

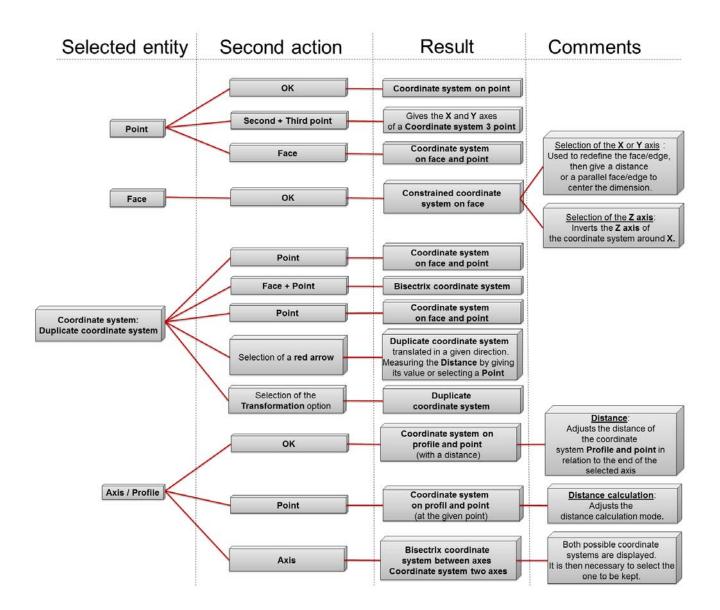
- Now, when removing an operation, you are asked whether tools are to be deleted, made visible or hidden.
- When creating an extrusion, several curves can be selected one after the other without using the **Selection** function.
- The **Move** function can be used on a relative point; the coordinate 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

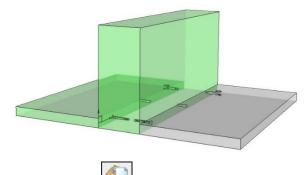
The **Dowel assembly**, **Screw**, **Len** and **Assembly kit** functions now use the automatic processes of components. For example, this allows you to assemble more than two parts, while ensuring that the processes will be performed. Before, these functions could only operate two parts.

#### 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.

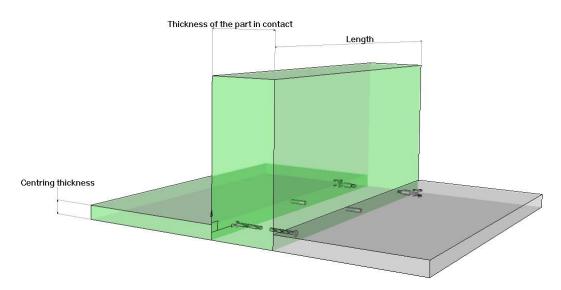


- Select Wood > Define > Define assembly kit.
- Choose Advanced assembly kit.
- Select the key coordinate system.

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

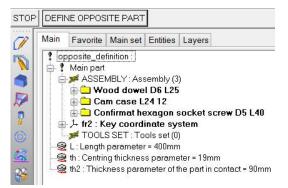
• Select the length parameter, the centring thickness parameter and the thickness parameter of the part in contact.

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.

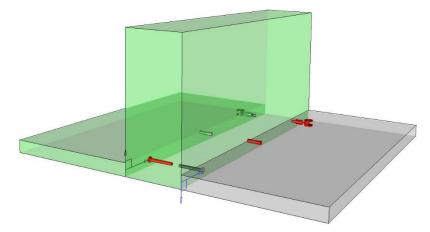


- If the kit is symmetrical, choose **STOP**. Otherwise, choose **Define opposite part**.
- Select the key coordinate system of the opposite portion.

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.

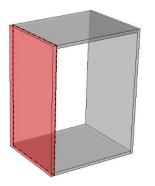


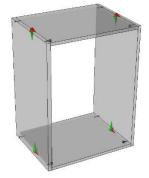
The selected elements and the key coordinate system are then visible in the construction tree under **Opposite part**. **Note**: Elements can be added to the kit. To do this, **right-click** > **Insert** on the **Set** line of the main or opposite part.

- ? opposite\_definition : 📋 📍 Main part SSEMBLY : Assembly (3) 🖶 🧰 Cam case L24 12 🗄 🗀 Wood dowel D6 L25 🗄 🛄 Confirmat hexagon socket screw D5 L40 🛓 , 🕂 fr2 : Key coordinate system 🗯 TOOLS SET : Tools set (0) 🖶 📍 Opposite part 🛓 🗯 SET\_OPPOSITE : Assembly (3) 🗄 🗀 Cam case L24 12 🗄 🗀 Confirmat hexagon socket screw D5 L40 🗄 🗀 Wood dowel D6 L25 🛓 쳐 fr1 : Key coordinate system SET\_TOOLS\_OPPOSITE : Tools set (0) 🗟 L : Length parameter = 400mm Sth: Centring thickness parameter = 19mm 👻 th2 : Thickness parameter of the part in contact = 90mm
- 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.

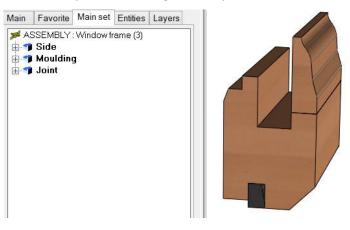




# Management of multi-body extruded components

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.



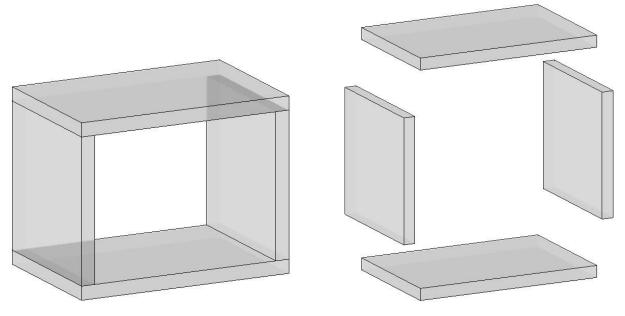
• Include the component in an assembly, and then create the cuts.



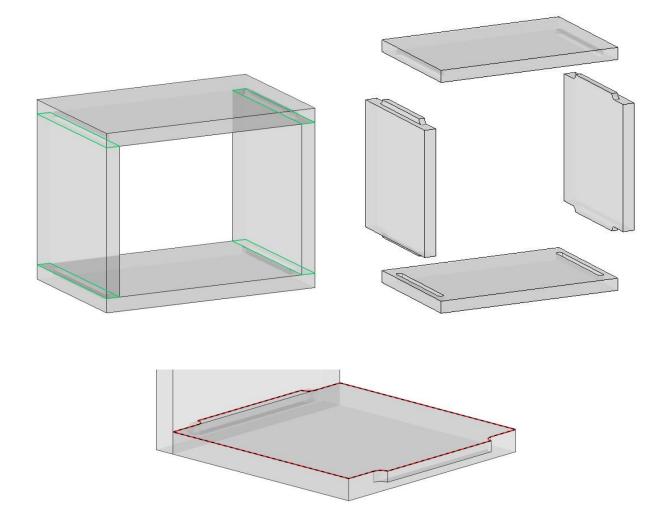
# Automatic material addition using component process

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.

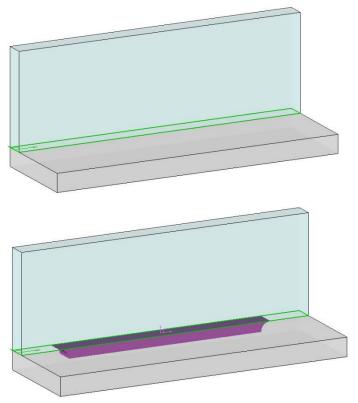


Result after the parts have been automatically assembled using a kit with an automatic union process.



#### TopSolid 2015

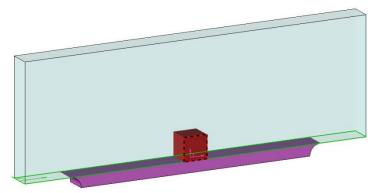
• Create the component. The example is an advanced assembly kit.



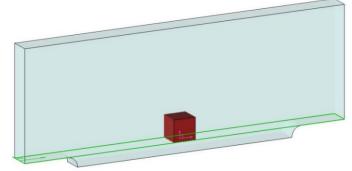
• Create the shape for the union. Here, this is a shape extruded from a sketch (in purple).

**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.



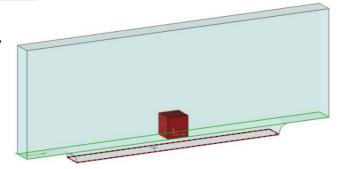
Make the part union using the Shape > Unite command, and select the martyr part (blue) as Shape(s) to modify and the part to be united (in purple) as Tool shape(s) to use.



 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.

Operation type= LOCAL OPERATION ON SHAPES v Name of tool element. tool1

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



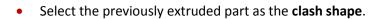
⊯ TOOLS SET : (1) ⊡ 🛠 tool1 : Union (1)

由 U union 1:u

**<u>Note</u>**: 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.

DEFINE CLASH SHAPE





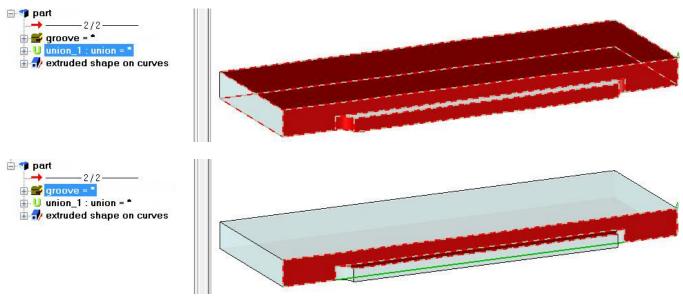
Analyze

\delta Delete

Create a group Define tool Deactivated = NO

**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

📬 Ta	p]
-	2/2
±.#	groove = *
<u>ل</u>	union = *
÷	constrained block

**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 **■** 0mm **G ③**:

- 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.



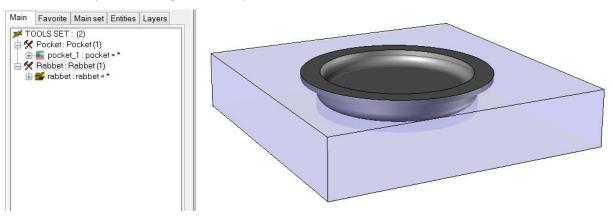
<u>Note</u>: On some constraints, it is not possible to enter an offset value, like for an axis on axis constraint for example. This area therefore does not exist.

- 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.

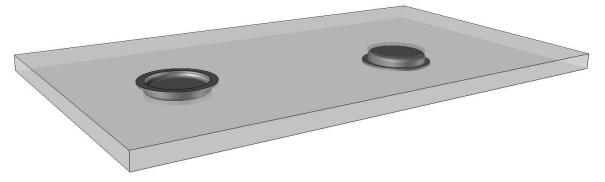


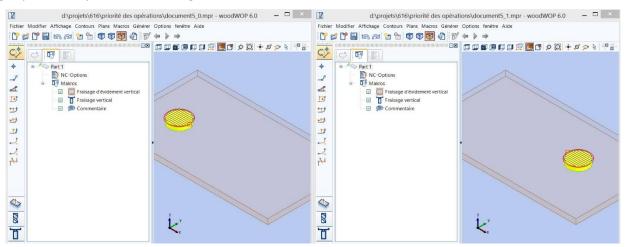
 In the Operations priority function, place the operations that will be generated by the component in Component positioning.

Priority	×
Selection type Select manually Select all operations Select operations with same priority	
Priorities > 1	~
O Select operations with same type	
Operations > Fold	~
◯ Select non exported operations	
Priority : 3	APPLY
Not to export	APPLY
<ul> <li>List of operations</li> <li>First positioning</li> <li>Second positioning</li> <li>Component positioning</li> <li>Rabbet 1, fold</li> <li>Pocket, 1, 2, pocket, depth = 0mm</li> <li>@124, 3, root</li> </ul>	
0 Selected operation	- 24
OK	

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.

Stape (SIMPLIFIED)     Stape (SIMPLIFIED)     Stape (SIMPLIFIED)     State dge (SIMPLIFIED)     flat edge (SIMPLIFIED)     state dga (SIMPLIFIED)     flat edge (SIMPLIFIED)     f	0 8 1 1	Analyze Controls Visible = YES Zoom Activate only layer = 0 Activate all layers Edit properties Characteristics	
⊞- <mark>S</mark> Model	1	Drive grain orientation More commands	

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.

	Advanced	d options	×
Simplified I Assembly na Sub-asse		on	
Single un			
Design for	'de		s-1-1
Counciliants	-		
Covering typ	be		The second second
Laminate	s covered by	/ edges	
The Desired States			
O Edges co	vered by lar	ninates	Prod Production
	-		and laminates

# 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.

<u>Note</u>: 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.

1	А	В	С	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

HOUSING

INSIDE A BLOCK

• 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.

OK MEASURE Housing mode= DIMENSIONS	✓ ×length= 500mm
OK MEASURE Housing mode= DIMENSIONS	✓ Y length= 450 mm
OK MEASURE Housing mode= DIMENSIONS	✓ Z length= 600 mm

<u>Note</u>: 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.
   OK × length= 000
   PREDEFINED VALUES=3
- 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.

	Predefine	d values	×
✔ Only those ∨	alues		
Value	Designatio	n	
550 600 650			
L.	COPY	PASTE	
	ОК	Cancel	

🗇 housing element

bp : Driver block

🚰 bp.y : Longueur Y du bloc pilote bp = 500

🕐 bp.z : Lonqueur Z du bloc pilote bp = 860

TopSolid 2015

#### 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.

OK Tool number=	Tool depth= 10.5mm	Machining process=	✓ Operation type= CALIBRATION +
-----------------	--------------------	--------------------	---------------------------------

During a machining export, the **Calibration** user machining is subject to the calibration rules: proper positioning for part definition, deactivation of automatic calibration.

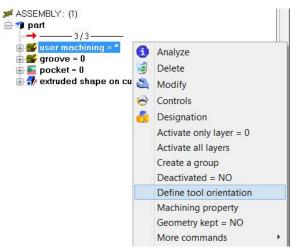
However, a **Milling** user machining is considered as a pure milling operation.

**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.

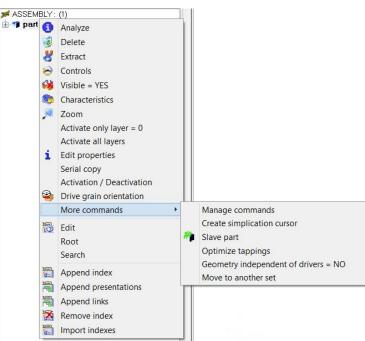


## Configuration

### 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.

<u>Note</u>: 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.

<u>Note</u>: 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.

		Application of	nc		×
Configuration     Configuration     Configuration     Control     Contro     Control     Control     Control     Control	Components not to operate by a	sulametic process	5		
Constraint     Texholo     Texholo	Standard TSWH-Hodware TSWH-Hodware	Family Hendle Screws	Type Wood screws	Veriant Cruciform nozale	Add
Instruction		OK Cencel			

- 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.



The part Handle @347 sould not be operated

Standard library

Filter Standard

Dressing accessorie
Edge

Accessories Wood screws Si Cructorm n Euro screw

OF

Cancel

TSWH-Hardware Family.type TSWH-Hardware Assemblies KIT Cabinet assemblies

### Assembly order for automatic numbering

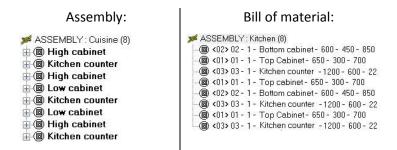
During automatic numbering, it is now possible to modify the order of components in the bill of material. This order can be modified using drag-and-drop directly from the construction tree.

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

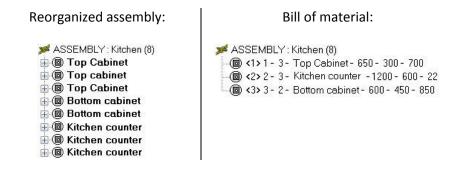
- In an assembly, open the construction tree, and move the elements by drag-and-drop operation.
- So that the order displayed in the tree is the same in the bill of material, open the .bom file of the bill of material using Notepad.
- Modify the ORDER\_BY paragraph. The syntax must be the following:

```
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.



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.



### 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.

Automatic codificatio						
	7 <u>.</u>	Double cli	ck here to add a rule		19	
	Codification rules	Edge type - code	Edge matter and coating	O∨er dimensio	Calibration overvaluation	
	[ <matter>]-[<coating>]</coating></matter>	Flat edge - ep 1	Beech - Coating = *	0mm	0mm	
		None		0mm	0mm	
	Concepts list of codifications	Comune differentieuro in a	day configuration	Daplace		tion
	Generate list of codifications	Copy codifications in e	dge configuration	Replace	codifications in edge configur	tion
Generated codificatio		Copy codifications in e	dge configuration	Replace	codifications in edge configur	tion
Generated codificatio		Copy codifications in e Edge type - code	dge configuration Edge matter and coating	Replace Over dimensio		tion
Generated codificatio	tions					tion
Generated codificatio	ions Codification	Edge type - code	Edge matter and coating	Over dimensio	. Calibration overvaluation	tion
Generated codification	Codification	Edge type - code flat edge - ep 1	Edge matter and coating Beech - walnut	Over dimensio Omm	. Calibration overvaluation Omm	tion
Generated codificatio	Codification Codification Beech - walnut Beech - oak	Edge type - code flat edge - ep 1 flat edge - ep 1	Edge matter and coating Beech - walnut Beech - oak	Over dimensio Omm Omm	Calibration overvaluation 0mm 0mm	tion

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.

matic codification rules -					
1		Double clic	k here to add a rule		
	Codification rules	Edge type - code	Edge matter and coating	Over dimensio	Calibration overvaluat
	[ <matter long="" name="">] - [<coating long="" name="">]</coating></matter>	Flat edge - ep 1	Beech - Coating = *	0mm	0mm
		None		0mm	0mm
	<				>
	٢				>
L		Conv codifications in ad	ap configuration	Replace co	diffections in adda configu
L		Copy codifications in ed	ge configuration	Replace co	difications in edge configu
L		opy codifications in ed	ge configuration	Replace co	difications in edge configu
Genera	ate list of codifications C	copy codifications in ed	ge configuration		difications in edge configu
Genera	Codification E			Over dimensio	
Genera Genera	Codifications C Codification E Missler-Wood-Beech - walnut f	Edge type - code	Edge matter and coating	Over dimensio Omm	Calibration overvaluation
Genera enerated codifications	Codifications C Codification E Missler-Wood-Beech - walnut f Missler-Wood-Beech - oak f	Edge type - code lat edge - ep 1	Edge matter and coating Beech - walnut	Over dimensio Omm Omm	Calibration overvaluation 0mm
Generated codifications	Codifications C Codification E Missler-Wood-Beech - walnut f Missler-Wood-Beech - oak f Missler-Wood-Beech - cedar f	Edge type - code lat edge - ep 1 lat edge - ep 1	Edge matter and coating Beech - walnut Beech - oak	Over dimensio Omm Omm Omm	Calibration overvaluation 0mm 0mm

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.

Modules		
Defined modules >	TopSolid'Design - Cam information	~
Function		
Defined functions	TOP OPERATIONS	Y
	TOP OPERATIONS	
	BOTTOM OPERATIONS LATERAL OPERATIONS RECTANGULAR	Ĩ

#### 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\$>.

🔚 abbrev.cfg 🗵

1 Top

4 Right

3 Left

Bottom

TOP

BOT

LEF

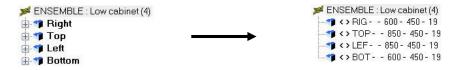
RIG

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\$>.

COL	JMNS {
	NAME=DESIGNATION
	"DEF= <woo_codification \$designation upper abbrev\$>"</woo_codification \$designation upper abbrev\$>
	TYPE=STRING
	ALIGN=LEFT
	TITLE_ALIGN=LEFT
	WIDTH=0.015
	VISIBLE=YES
	1

• 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=REFER	ENCE
"DEF= <woo_< td=""><td>CODIFICATION   \$REFERENCE   Upper   Trim (2,4,X) \$&gt;"</td></woo_<>	CODIFICATION   \$REFERENCE   Upper   Trim (2,4,X) \$>"
TYPE=STRIN	G
ALIGN=LEFT	
TITLE_ALIG	N=LEFT
WIDTH=0.01	5
VISIBLE=YE	S
;	

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.



<u>Note</u>: 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. 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.

Defined modules >	TopSolid'Design	~
Function		
Defined functions >	PART TYPE	~
	designation	
	designation	~
Other parameters >	reference	
	supplier processing	
	number	~
Type > Cł	HARACTER STRING	~
Operator > =		~
Value :		

Multi criteria filter

Name

Hardware

Double click here to add a filter

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.

Type >	CHARACTER STRING	¥
Operator >		~
Value :		
All elements	() >	
Use right button to	>=	
🍸 Hardware	<	
	contain starts with	

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.

Туре 🔸	CHARACTER STRING	~
Operator >	=	~
Value :	Hardware	
All elements	except	
se right button ti	o create filter	
se right button ti Mardware	o create filter	
-		
-	① Filter	

Hardware

PART TYPE contain Hardware

Finish creating the filter by right-clicking the name line, and selecting Use right button to create filter Filter.

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

#### ① Filter

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.

0	Add					
Exp	oands	the	filter	possi	bilitie	es.

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.

Function		
Defined functions >	SUPPLIER	¥
	designation	
Other parameters >	designation reference supplier processing number	~
Type > Cł	HARACTER STRING	~
Operator > =		~
Value : BL	UM	
All elements exc	cept	
Jse right button to cr	eate filter	
Y Hardware		
Filter	PE contain Hardware	
SUPPLIEI		

Defined functions >	SUPPLIER	~
	designation	
	designation	^
Other parameters >	reference supplier processing number	~
Type > Cł	HARACTER STRING	~
Operator > =		~
Value : LN	1C	
All elements exc	cept	
se right button to cr	eate filter	
Hardware		
- 🧐 Add		

Automatically filters the panel entities.

#### Same Modify

🐔 Сору

modify it.

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.

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

#### 😼 Delete

Deletes a property of the filter.

Defined functions >	PART TYPE		~
Other parameters >	designation reference supplier processing number		^ ~
Type > Cl	HARACTER STRING		~
Operator > c	ontain		~
Value : Pli	inth		
All elements exi	cept		
Jse right button to cr	eate filter		
Hardware	contain Hardware		
		Filter	
		Add	
	3	Modify	
	1	Delete	
		Filter panel entities	

Function

#### 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.

Selection type Select manually Select parts by material	
Matters - Thickness >	~
Select by filter	
Filters > Cabinet	~
O Sele Cabinet Hardware Criterio Panel	

Double click here to add a filter

Multi criteria filter

Name

Hardware

#### 🐔 Filter panel entities

### 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.

<u>Note</u>: 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.

	Material	editor	×
User Group - tiling - TopSolid'Wood - Building - Bricks - Concrete - Pavement - Plaster - Stonework - Tiles - Hardwoods - Panels - Colors - Other - Raw - Woods - treatment - tree - wood - Woods Add category Delete category	Alder black Apple tree Ash brown Ash european Ash olive Beech core Beech european Beech hearted Beech unsteamed Birch apple Birch apple Birch peeled Cherry european Chestnut sweet Elm smooth leaved Hombeam european Larch antique Larch european Lime european Lime european Maple core Maple great Maple great Maple great Maple great Maple great Maple great Maple great Maple core Maple great Maple ore Oak koog Oak koop Oak koot Oak koot Oak old	U#image data       Texture param.       Finishing         Data       Advanced data         Name       ;Beech unsteamed         Designation       ;         Reference       ;         Supplier       ;         Density (kg/dm3): [1       ;         Specular colour:       ;         Image data       ;         Use as material       ;         Use as finishing       ;         Use as finishing       ;         Save       Delete	
		Calicer	

## **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

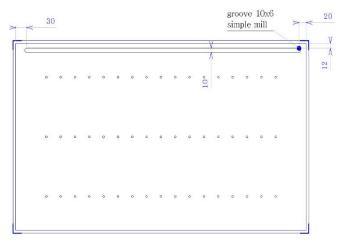
The automatic dimensions of **grooves**, **rabbets** and **mouldings** have been enhanced to display more information on these operations. In the **File** > **Properties** > **TopSolid'Wood properties** > **Draft** command, you can choose to project the notes and/or the dimensions for these three operations.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

Elements to dimension	
Dimension drawings	
Part dimensions	
Drilling dimensions	
Groove dimensions	
Slot notes	
Rabbet dimensions	
Fold notes	
Moulding dimensions	
Moulding notes	
Counter-moulding dimensions	
Edge dimensions	
Laminate dimensions	

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.



<u>Note</u>: The tool designation can be displayed in the note. To do this, the Print tool designation in notes 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.

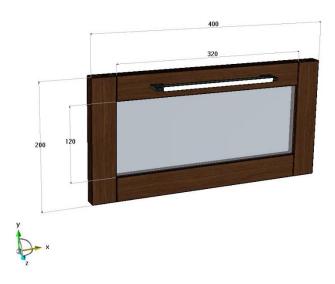
Elements to be projected must first be defined in the component template.

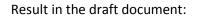
- On the main assembly (or on an alternative set), right-click > Define draft detailing.
- Then set the **viewing direction**. Two modes are available depending on the behavior you want.
  - Oriented= YES Fs X+ X- Y+ Y- Z+ Z- Viewing direction:

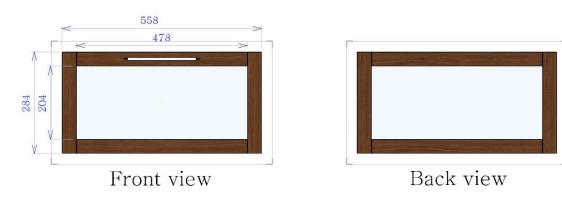
This mode allows you to project different elements according to the view orientation in the draft. You therefore need to specify the orientation in which detailing must be projected.

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.



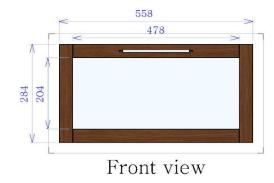


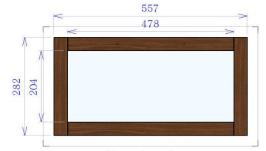


Oriented= NO + X Y Z Viewing direction:

This mode is used when no specific orientation is needed.

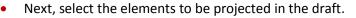
Still using the same sample file, if this time the selected direction is **Z**, the result in the draft is the following.





Back view

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



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

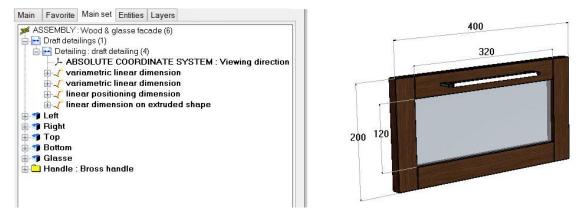
ASSEMBLY : Reported panel shelves (MKTE) (1) Draf detailings (1) Detailings (1) Detailing (3) Verwing direction = 0.000,1000,000 Detailing is the circle shell be compared by the compared by	600
	860
ASSEMBLY : Drawer kit	

- $\frac{1}{4}$  inear dimension on line
- 🗄 🧹 linear dimension on line
- Once the selection is completed, you can give a name and a designation to the draft detailing or finish by clicking on **OK**.

Main Favorite Main set Entities Layers

\*

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.

TopSolid Wood Drawers block TopSolid Wood Pacade
Missler Software         Missler Software           News 2015         Scale: 15         Page 1/2         VMANUFACTURED FOR CODES

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.

<u>Note</u>: 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**.

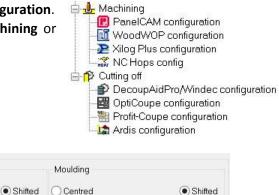
Shifted

- 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

Tool path

Centred

Slot



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

Fold

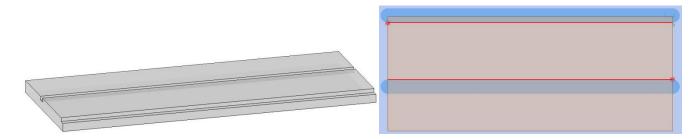
O Centred

#### **EnRoute cut export**

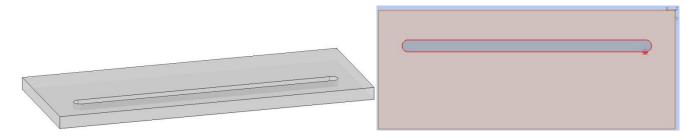
For the export to EnRoute, the shifted tool paths are set via the following configuration words.

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.

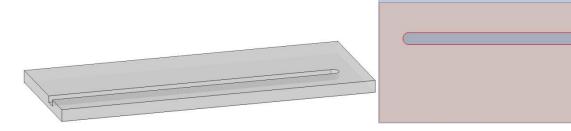
- For a machining that goes through the part, on an edge or within the part, the shifted tool path is exported.



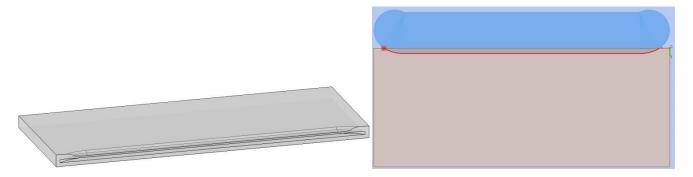
- For a machining stopped in the part, the closed contour of the machining is exported.



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



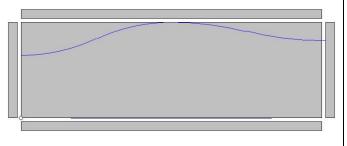
- For a machining stopped on the part edge, the open contour of the machining is exported.



#### Moulding

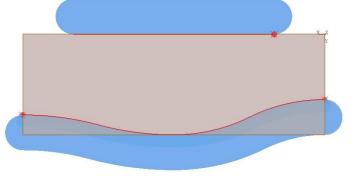


Result imported in Ardis.



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**.

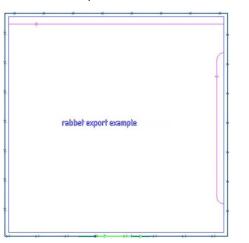


#### <u>Fold</u>

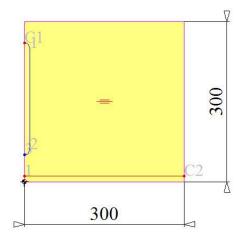
#### Part exported from TopSolid'Wood.



Result imported in **EnRoute**.

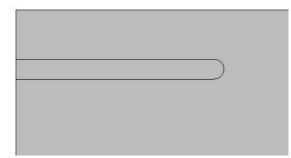


Result imported in **NC Hops**.



#### <u>Slot</u>

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.

Machine		
Machine >	SCM CYFLEX	

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	Default tool number assigned
	140.	to pockets.
ZX_ZWOO_CAM_XXL_CYFLEX_SLOT_MILLING_TOOL_NB	By default	Default tool number assigned
	101.	to slots/folds performed with a
		mill.
ZX_ZWOO_CAM_XXL_CYFLEX_SLOT_SAW_TOOL_NB	By default	Default tool number assigned
	101.	to slots/folds performed with a
		saw.
ZX_ZWOO_CAM_XXL_CYFLEX_MOULDING_TOOL_NB	By default	Default tool number assigned
	101.	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**.

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

MAIN ASSEMBLY Depth: MULTI LEVE	- v	Filter bom by criteria= no filter	~	Update machining face =	YESF
---------------------------------	-----	-----------------------------------	---	-------------------------	------

#### Machining stop export

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

irst positioning Second positioning	First positioning Second positioning
Part positionning on machine (top view) Working station	Part positionning on machine (top view) Working station
	Top Right

The name of the machining stop defined from **Tools** > **Options** > **TopSolid'Wood configuration** > **Working stations configuration** is exported to the interface.

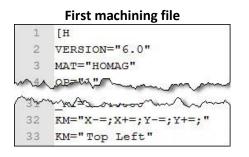
If a name and a designation are specified in the machining stop, the designation will be displayed in the part definition and the name will be exported.

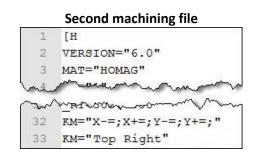
In interfaces where two machining files can be generated (**WoodWop**, **Xilog**, **NCHops**), only the name of the stop corresponding to the file machining positioning is exported.

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.





#### <u>Xilog</u>

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

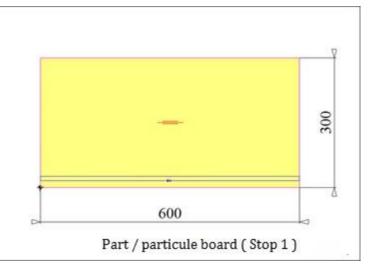
Machining stops are exported in a variable of the **hop** file.

**<u>Note</u>**: 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.



#### <u>Ardis</u>

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

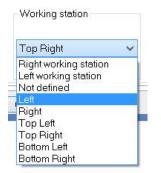
To ensure that machining stops can be properly read in **Ardis**, the **TopSolid'Wood** stop names and designations must be set as follows:

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	88 <del>5</del> 4	2552
1	Left	×	020
2	Right	83 <del>5</del> 4	×
3	Top Left	1121	(22)
4	Top Right	10 <b>5</b> 4	83 <del>5</del> 8
5	Bottom Left	020	020
6	Bottom Right	20 <b>5</b> 2	25 <del>5</del> 2

#### Stop selection from the part definition:

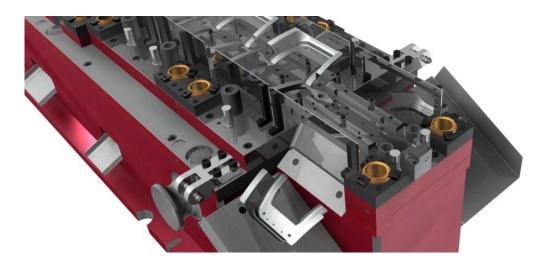


#### Result in Ardis:

#### 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**.

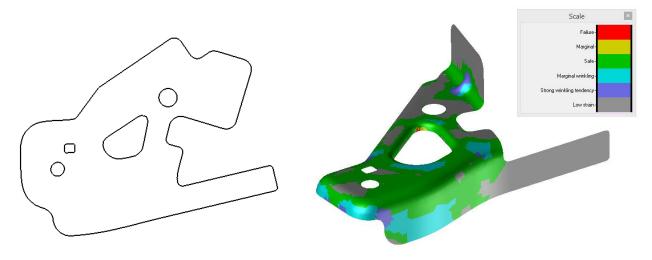
## **Part preparation**

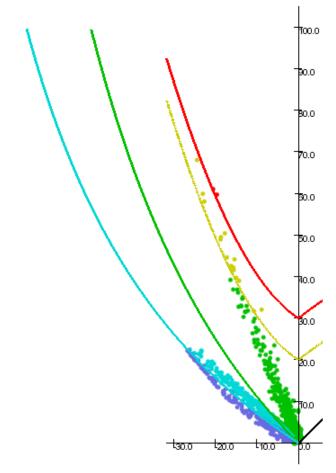
### Analyze unstamping results

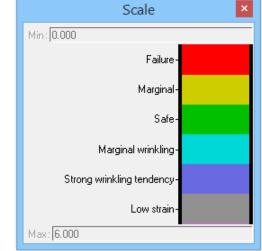
A new type of analysis is now available: the Forming Limit Diagram (or FLD).

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.



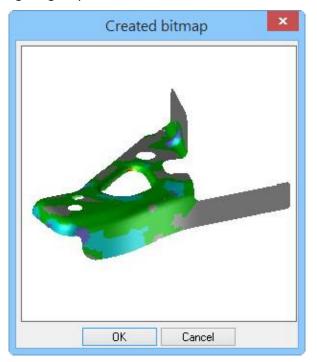




The various colored curves identify the areas of characteristic behaviors where the analysis points (the part's meshing nodes) are displayed in the corresponding color.

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).



Many improvements have been made to this command.

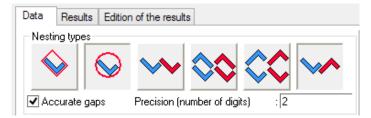
The overall dialog has been reorganized and includes a new tab for editing the nesting results.

Nesting for strip definition					
Data Results Edition of the results Nesting types Accurate gaps Precision (number of digits) Angles Minimum Maximum Increment Gaps Gap between parts Vertical gap in pattern Horizontal shift of top part in pattern Margin	:[2 :[0.00* :[180.00* :[1.00* :[1.00* :[5.00mm :[50.00mm :[25.00mm]				
Strip width Imposed width Minimum width Round value Strip Thickness Material	: 0.00mm : 0.00mm : 0.00mm : 1.00mm : steel				
Shear Modulus (N.m2) Density (kg/dm3)	: 40000000 : 7.85 Material				
Progression					
	10	Cancel			

#### 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).

Data	Resu	Its Edition	n of the result	s		
Nesti	ng type	es				
	>		$\checkmark$	$\land \land$		
	×		••	$\mathbf{V}$	$\mathbf{V}$	
🖌 Ac	curate	gaps	Precision (n	umber of dig	its) : 2	

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.

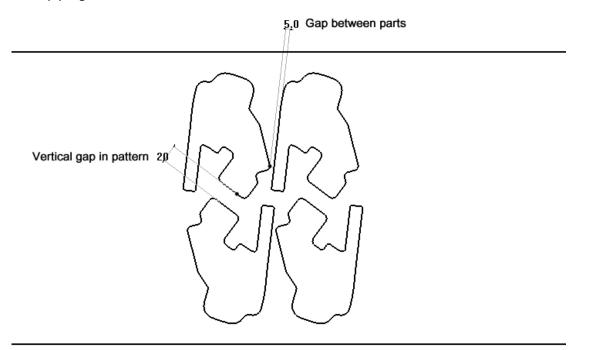
When this option is activated, it is also possible to specify an increment angle value less than 1°.

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

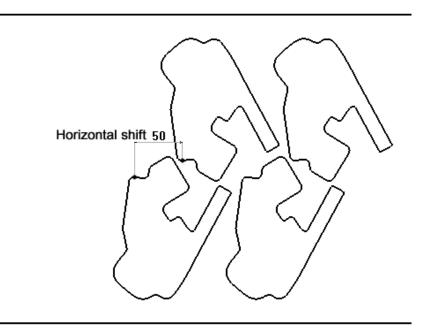
Data Results Edition of the results							
Nesting types							
$  \diamond   \diamond   \diamond   \diamond   \diamond   \diamond   \diamond   \diamond   \diamond   \diamond$	<						
Accurate gaps Precision (number of digits)	:2						
Angles Minimum	: 0.00*						
Maximum	: 180.00*						
Increment	: 1.00*						
Gaps							
Gap between parts	: 10.00mm						
Vertical gap in pattern	: 2.00mm						
Horizontal shift of top part in pattern	: 0.00mm						
Margin	: 50.00mm						

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.

Strip	]
Thickness	: 1.00mm
Material	: steel
Shear Modulus (N.m2)	: 40000000
Density (kg/dm3)	: 7.85
	Material

#### Edition of the results tab

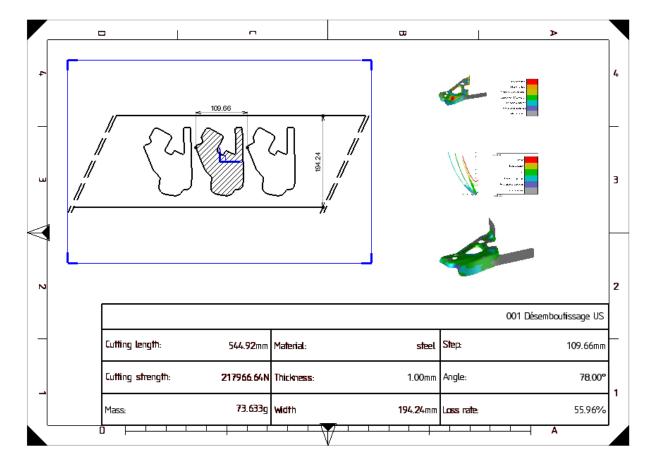
It is now possible to set a given step between parts thanks to the **Imposed step** option. It allows the use of a rounded step value rather than that calculated by the application.

Data	Results	Edition of the results	

Imposed step : 110.00mm

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 stri	o definition	×
Data Results Edition of the results			
, ✓ Imposed step : 110.00mm			
Create curves	Create parameters	Create strip curves and dimensions	
✓ In draft document			
Template document : without template			~
	v	Added bitmaps	~
Template document :       without template         From analysis:       @660         001 Désemboutissage       @660_Forming zones_1         001 Désemboutissage_@660_Forming limit diag       001 Désemboutissage_@660_Safety zones_1		Added bitmaps 001 Désemboutissage_@660_Forming zones_1	✓
From analysis: @660 001 Désemboutissage_@660 Forming zones ; 001 Désemboutissage @660 Forming limit diar	gram_1		۲



In the Draft document templates of the nesting reports, it is now possible to get the cutting length and cutting strength parameter values.

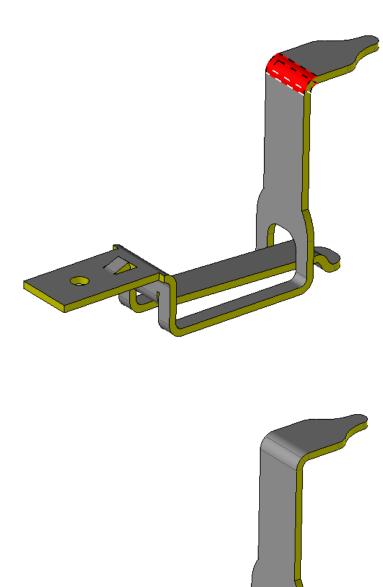
These values can be accessed using the Progress die > Nesting report configuration command of the Draft document.

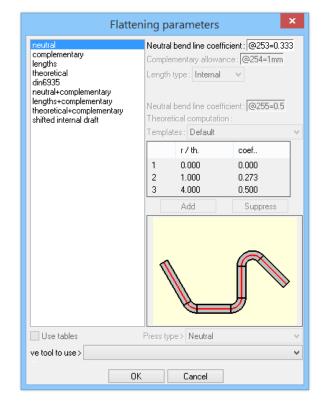




The bend division line can now be positioned by specifying a value corresponding to the flat bend length, and taking into account the bending allowance computation rules.

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).





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.

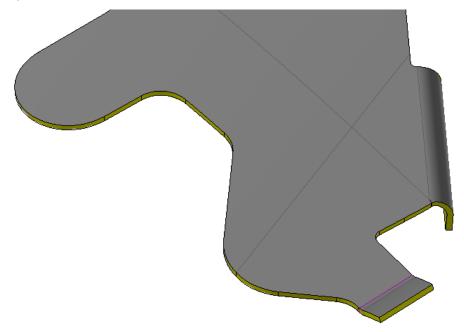
For example, for the part shown below, the purple area must be treated by bending before the flange unfold operation is performed on the green area.



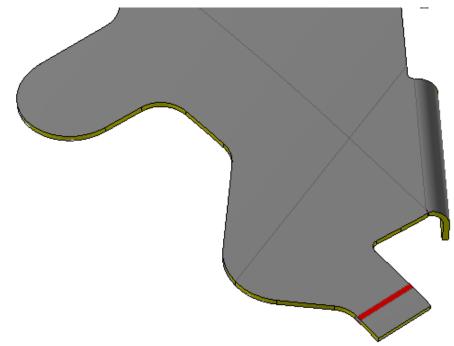
With this new functionality, the flange unfold operation can be done on the green area while leaving the purple area in its original position in order to obtain the following result.



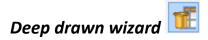
Thanks to the **Bodies management > Move bodies > Position body** command, it is then easier to reposition the body that was left in place.



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.

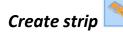


For more complex configurations, the definition of the transition area between the repositioned body and the body resulting from the flange unfold operation may be more complicated. It could be necessary to apply other design methods, including the surface modeling commands.

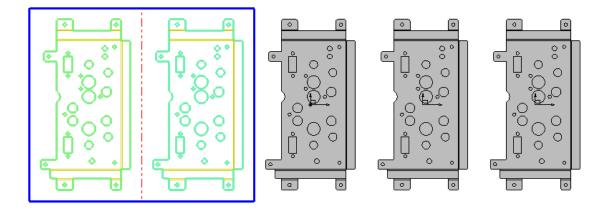


Ten lines are now available in the calculation form of the number of drawing steps.

## Strip design

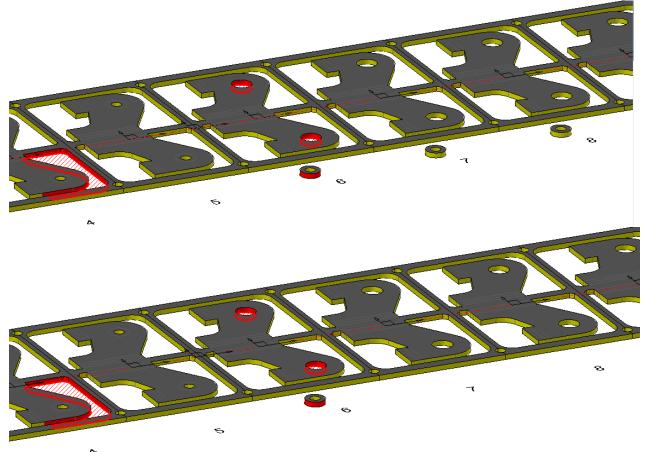


In **No strip of matter around part** mode, it is now possible to fill the part's internal holes by checking the **Fill holes in part** option.



## 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.

Forces vi	sualization ×
Analysis set Analysis set > Global forces analysis	~
Elements to visualize  C Operation forces  Total forces  Total force	Visualization options Attributes Layer : 0 ✓ Create on current level Transparency : 5 Use current transparency
<ul> <li>Total cutting force</li> <li>Total bending force</li> <li>Total stamping force</li> <li>Total spring force</li> </ul>	Arrows color Cutting forces: > Bending forces: > Stamping forces: >
Texts Total force text Equivalent cutting length text	Spring forces:       >         Other forces:       >         Total force:       >
Coordinate system Generate a coordinate system at center of forces Delete previous visualisations	Other options Scale value (N/mm): 100000 Automatic scale adjustment Max height value : @434=500mm
ОК	Cancel

### **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.

Die set edition							
General parameters Plates configuration Pre-defined die sets configurations							
Punches carrier set Coordinate system position : @295=200mm							
Up	Template element	Length	Width	Height	Code		
Down	PGS_USER, driven, plate	500mm	450mm	60mm			
Сору	PGS_USER, driven, plate	L=500mm	W=400mm	H=40mm	J		
Delete	Double-click here to add a plate	•	•	-			
Stripper set							
Coordinate system p	position			:@186=	2mm	_	
Up	Template element	Length	Width	Height	Code		
Down	PGS_USER, driven, plate	500mm	500mm	30mm			
Сору	Double-click here to add a plate	-		-			
Delete							
Die set							
Coordinate system p	position				: @185=200mm		
Up	Template element	Length	Width	Height	Code	Adjust H.	
Down	PGS_USER, driven, plate	500mm	500mm	60mm		NO	
Сору	PGS_USER, driven, plate	500mm	500mm	100mm		NO	
Delete	Double-click here to add a plate	•	·	•	-	-	
✓ Automatic pattern							
Apply modification on : selected plate only							
OK Cancel							

Plate selection ×				
PLATES PGS_USER PLATES PLAT	Reference dimensions         plate x reference length : 1000mm         plate y reference length : 500mm         Display of components list         Components that respect constraints         Image: All components         Codes         Main drivers         Image: Length         Image: Length </td			
	shift along Y axis Omm			
ОК	Cancel			

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.



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.

Include die set, conflicts management				
				Parameters:
Parameters	Designation	Value	Merge	Replacement name
TH	TH	2mm	Yes	
PSH	PunchSetHeight	165mm	No	PSH_1
ОК				

# Components



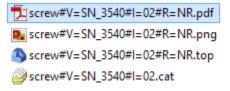
In the **Component configuration** command, a new **Information** tab allows you to associate an information text with each component.

Manager	ment of properties of c	omponents ×
PGS_MISUMI     PGS_PEDROTTI     PGS_RABOURDIN     PGS_STRACK     PGS_STRACK	s steel) ew(10.9) ew(8.8)	
Standard reference		Search
Current component: SN 3540	ŀ	Joach
	arances 🛛 🗹 Standard machining	gs 🗹 Component attributes 🗹 Information
Clearances associated to processes Stan	ndard machinings Default attribut	tes Information
Component information :	mensions D8, L50.	∧
	OK Cancel	

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

Standard comp	onent inclusion
PGS_MISUMI     PGS_PEDROTTI     PGS_RABOURDIN     PGS_STRACK     Accessories     Cutting     Cutting     Dowel pin     Screw     Countersunk head screw     Screw     Scoket head cap screw(10.9)     Socket head cap scr	
Standard reference Current component: SN 3540 Search Origin coordinate sytem: default Csys Codes: Available codes: SN 3540-3-10 SN 3540-3-10 SN 3540-3-10 SN 3540-3-10 SN 3540-3-10 SN 3540-5-10 SN 3540-5-12	Drivers Spring infos Information Recommanded dimensions D8, L50.
SN 3540-5-14 SN 3540-5-16 SN 3540-5-20 SN 3540-5-25 Forced by user	Information PDF Cancel

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.

Standard component inclusion	×
PGS_MISUMI PGS_PEDROTTI PGS_RABOURDIN PGS_STRACK Accessories Cutting Cutting Dowel pin Screw Countersunk head screw Locking screw Shoulder bolt Shoulder bolt Socket head cap screw(10.9)	
Standard reference     Drivers     Spring infos     Information       Current component:     Search     Pecommanded dimensions D8, L50.       Origin coordinate sytem:     Information	^
default Csys	
Codes: Available codes: SN 3540-3-16 SN 3540-3-16 SN 3540-4-10 SN 3540-4-16 SN 3540-5-10 SN 3540-5-12 SN 3540-5-16 SN 3540-5-20 SN 3540-5-25 V Information PDF	~
OK Cancel	

# 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.

	Management of propert	ties of components
Lockir Shouk Shouk Socke Socke	ersunk head screw ng screw	
Standard reference		
	N 3540	Search
Copy paste Copy Paste	e 🔽 Clearances 🗹 Standard	d machinings 🗹 Component attributes 📝 Information
Clearances associated to p	rocesses Standard machinings Def	fault attributes Information
Current condition		
If parameter	✓ stands betwee	
Then clearance		✓ = 0
Standard	tolerance None 👻 Or min tolerance	: 🗸 max tolerance : 🗸
List of conditions		
Add		
Modify		
Suprress		
Up		
Down		
Clearances driven by co	onfiguration file	
	OK	Cancel

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

	Component modification	×
Characteristics Supplier PGS_STRACK Family Fixing Type screw Variant socket head cap screw[low h Code SN 3540-12-80 Designation low socket head cap screw	Codes       Drivers       Processes       Information         Drilling @2077 on part : Plaque Simple       Facing @2085 on part : Plaque Simple         Drilling @2091 on part : Plaque Simple       Tapping @2106 on part : Semelle Inférieure         Image: Clearances from configuration file       Image: Clearances for operation	
Origin coordinate sytem:	Through	
default Csys 🗸	Through tapping	
Constraints:	Standard tolerance > None v	
On Measure Value Type	Defector Vila	
<ul> <li>✓ L &gt; 2 = 65mm Codes</li> <li>✓ L &lt; 2 = 65mm Codes</li> </ul>	Designation Value	
	Update dynamically Update component	
OK	Cancel	

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.

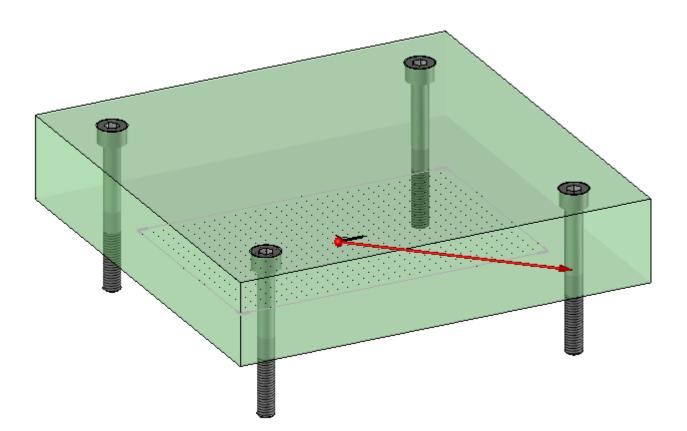
This command provides access to two new features.

The **DEFINE DEFAULT STOCK PROPERTIES** option allows you to predefine the stock computation options for the current component.

Stock definition
<ul> <li>Automatic stock computation</li> </ul>
✓ Minimal box
Cylindrical stock
<ul> <li>Manual csys as destination csys</li> </ul>
accuracy: 2
OK Cancel

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.

Cutting die operati	ion parameters 🛛 💌
Dimensions Machining pro	ocess
Туре	
One part	Two parts
First part	
Clearance (on radius):	: Omm
Height:	: 10mm
Draft	
	Export length Slope
Angle: Export length:	: [0 : [0mm
Height:	: Omm
Second part	
	Diameter
Shift	: Omm
Diameter: Minimum value:	4.00mm : 6mm
Draft Without O Angle (	Export length OSlope
Angle:	:1°
Export length:	: 1mm
Height:	: Omm
Extend	
On top:	: Omm
At bottom:	: Omm
Authorize machining mo	de for offset computation
ОК	Cancel

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).

Punch parameters			
Body Shoulder Head Key gro	ove Machining process		
Top of punch Manual distance	:100mm		
Shift from top:	: Omm		
Height or penetration	) Height		
Value:	:12mm		
O Clearance 💿	Diameter		
Clearance (on radius):	: Omm		
Diameter: Maximum value: 4.00mm	: <mark>3.96</mark> mm		
Cutting wave Without  External wave	Centered wave		
Depth:	: 5mm		
Orientation of cutting side:	: 0*		
Heel With heel			
Orientation of heel side	: 0*		
Width	: Omm		
Height	: Omm		
Authorize machining mode for off	set computation		
ОК	Cancel		

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

# **Miscellaneous**



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



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.

A	ttributes to reproduce 🛛 🗙
V F V S V F V S V F	Bom properties Designation Reference Supplier Treatment Comment Forced number Stock properties Material
	Rom index
	All properties
F	Function and sets Function Assigning set Alternative sets Stroke modifications
	All function and sets
	Attributes Color Transparency Layer
	All attributes
	OK Cancel

## 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.

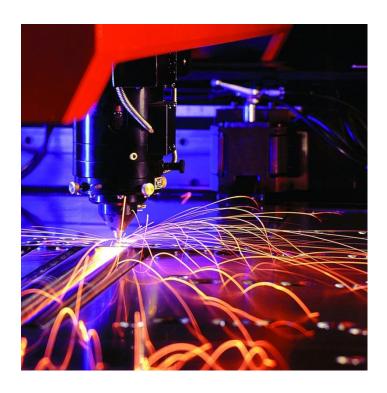
	Progress Elements Visualization	on ×
<ul> <li>Punch</li> <li>Stripper</li> <li>Die</li> <li>Strip</li> <li>3DStrip</li> <li>Global</li> </ul>	Visible: Plaque Simple L=260mm W=150mm H=30mm Plaque Simple L=260mm W=150mm H=35mm Semelle Inférieure L=400mm W=250mm H=50r ball bearing bush 30x47 ball bearing bush 30x47 ball bearing bush 30x47 ball retainer 30x45 ball retainer 30	Invisible:
Designation	Filter:	ancel

# Draft



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

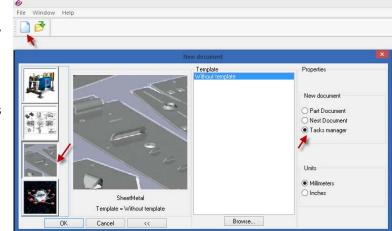
# Direct access to the Tasks manager

#### 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.

×r.

	General parameters Thickness (2.00mm N		
	Machine Machine	Matter typ Material	
	bystonic	304L	
	finn \$4 t(3030)	Electro zingué Steel	
	Cutting Gaz	Tube Cutting 3D (chuck)	_
1 Jan Na	Cutting parameters selection By contour V	Cutting type Tube (2D)	
	Family All 90*	(7) 🗸	
	Importation  Impor	Opened lines and arcs	
	Texts Simplify all Texts	Given Text colour 2	1.
	Default wanted quantity 1		
	Use part holes during complex	nesting	

# Importing multiple documents - Closing PCH documents

#### Purpose of the function

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

#### Triggering the function

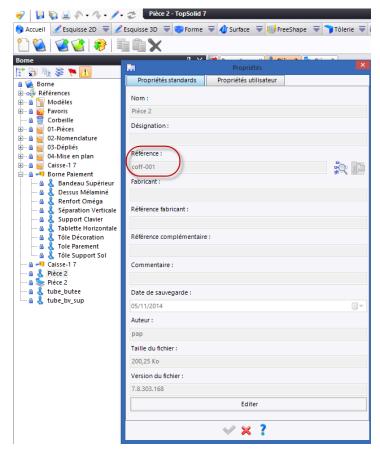
The setting is made in Tools > Options > Importation > Multidocs TOP → PCH Multidocs DXF/DWG/IGES

		Application config	juration			1
iguration Iser information Jolons Ihortout key Ienoral	Multidocs TOP → PCH					
nportation	Interface file list					
General parameters Geometry Other options Multidocs TOP > PCH	Test file extension cov Field reparator TAB character v Unit for thickness nm v Ceate PCH file in the same folder as in TOP file					
Multidocs DXF/DW/G/IGES	The first line of the lat file gives	the names of the column	s 🔿 Define order of	columns		
lut	Header columns names		Order of the o			
lesting	Folder	REPERTOIRE	Available field	ds	Ordered columns	
imulation ost processor	Part Name	REPERE	Part Name			
asks manager	Path	Chemin complet	Folder	^		
hisplay options	Quentity	QUANTITE	Path Mater	1.2		
	Thickness	Epaisseu	Thickness			
	Matter	Matièro	Machin Quantite			
	Machin	Machine	Designation			
	Designation Customer	Désignation	Customer			
	OF	Commande	Plane	1		
	Pane	Plan	Set			
	Set	Ensemble	Date	v		
	Date	Date				
	Use multiparts recognition per de	sign file	Separator 🖡		kak for each part name	
	Ask general parameters for each	pat 🗹 Automatic a	allocation of each part		utomatic marking of each part	
	Create a Nesting Order NO		✓ ✓ Add	Entries in	Tasks Manager	

# 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.

													TASKS N	IANAG	ER						
Parts to r	machine Matters to use Parts don	e Nestings	done	Orders do	ne																
	Tilter																				
<b>#</b>	Ref.	Des.	S	Plane	OF	Customer	Mat.	М.	Th.	Qty	N	Qty +	Qty max.	Fam.	Comp.	Prio.	Nb ent.	Deadline	Mac.	Gaz	Τ
	🔲 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		
8-11	L tube_butee - Coupe.Forme 5.A	@30					acier		3.00mm	1		0	0	-1	No	0	1	27/10/2014	tubematic		
3	L 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	
$\bigstar$	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						No			05/11/2014	tl3030	oxygène	
<u> </u>																					
91																					
$\mathbf{X}$																					

**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).

6				TopSolid (beta) by Missler Software	
File	Manage Edit Punching Attachs Loader/Un	loader Miscellaneous Tools Nesting Cutting	Autre Window Help		
				▶ - 🖂 🖆 -   💊 ᠿ -   🔳 - • ∕ -	
8	کے اگ کے کہ کہ 🖌 😽 ≷	3 🚲 👗 🗳 🖳 🗳 🎜 🕌	🔹 👉 💕 🗞 🍽	🗞 🕕 🔳 👪 🎥 🚖 🔕	
*****					
3	Do	ocument properties		Punch/Cut : C:\Projets\Document7.pch < <current>&gt; (Associative mode)</current>	- • ×
-	Document properties	Other information			
-		Part Document : Document7.pch			
(F***)	🚽 🗘 General 🚍 🗘 General information	Material : acier			
44	Main information	Thickness : 1.00mm			
	C Other information				
0	User information	Cutting Gaz : oxygène			
	Colours of v7 unfold	Machining machin : tt3030			
<b>/7</b>		Default wanted quantity:			
~					
		Reference : coff-001		$\bigcirc$	
		Designation :		$\bigcirc$	
		Set name :			
		Plane name :			
		OF .			
		or			
		Customer :			
		Production date : 05/11/2014			
		Comments :			1
		Comments .			
			~		
		DK. Cancel			

## 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:

OK Na	ame:	De	esignation: parts_par	ams_info			
		Draft : F:\V61	6\local\english	US\opcard_full	_ex.dft * < <cu< th=""><th>irrent&gt;&gt; (Asso</th><th>ciative</th></cu<>	irrent>> (Asso	ciative
-		. Table of	customized pa	arameters of	unfold v7 doc	uments	_
Text		PARTS	REFERENCES	Storage area	Bendsnumber	etc_	1
							•

#### Result:

NDEXES	ARTS	REFERENCES	FILES		Storage arm	Bends number	Author	QUANTITIES
	Nace Hat 300,A	m	C\DATAS\	projets\6.16\TCK 3428 Reference v7\Autres params\HERVE\Pilaze Het 300A.pch	0	2	Hat	1
2	ians/6	-	C\DATAS\	projets/6.16/,TCK 3428 Reference v7/Autres params/HERVE/pieze6.pch	-	-	+ -	1
Table of r	istomizari n	arameters of	f unfold v7	7 documents				
				7 documents				
PARTS	REFEREN	CES Stora	f unfold v7 age area	Bends number				
	REFEREN							

**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

NO TRANSFORMATION CHOOSE TRANSFORMATION Copy general information= YES 🖅 Basify Machining document= YES 🖅 Input data for tasks manager= NO 🖅

#### - Import of multiple documents Top $\rightarrow$ PCH

DK Automatic allocation of each part YES 🖘 Marking part name NO 🖘 Basify Machining document= YES 🖘 Close Machining document= NO 🖘

#### - Import of multiple documents DXF - IGES $\rightarrow$ TOP $\rightarrow$ PCH

OK Create PART files= YES 🗸 Automatic allocation of each part YES 🗲 Marking part name NO 🗲 Basify Machining document= YES 🗲 Close Machining document= NO 🗲

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

For single importation:

	Application configuration	
Configuration User information Colors Shortcut key General Colors General parameter Colter options Multidocs DXF/DWG/IGES Multidocs DXF/DWG/IGES Cut Simulation Post processor Cut Display options	General parameters         Default machin         Utility         Default thick 3.00mm         D	
	Ask the questions (to choose transformation the turret)         Ignore modifications of Name of Matter of the 3D part during upadtes of the punch part         Bill of material file         Designation of the names of columns         NOM       NOMBRE	
L	OK Cancel	

When importing multi-documents TOP  $\rightarrow$  PCH:

		Application config	uration					
Configuration User information Colors Shortcut key General Importation Geometry Dther options Multidocs TDP > PCH Multidocs DXF/DWG/IGES	Multidocs TOP -> PCH ✓ Use file interface Interface file list Text file extension csv Create PCH file in the same folder	Field separator r as its TOP file	TAB character V Unit for thickness mm V					
Mulados DX-F/DW3/1625 Cut Cut Simulation Post processor Tasks manager Display options	The first line of the list file gives to Header columns names Folder Path Name Path Quantity Thickness Matter Machin Designation Customer OF Plane Set Date	he names of the columns REPERTOIRE REPERE Chemin complet QUANTITE Epaisseur Matière Machine D ésignation Client Commande Plan Ensemble Date	S Define order of columns					
	Date       Date         Use multiparts recognition per design file       Separator         Ask general parameters for each part       Automatic allocation of each part         Ask general parameters for each part       Automatic allocation of each part         Basify part machining document       Close part machining document         Create a Nesting Order NO       V Add Entries in Tasks Manager							

## Automatic part lettering in multi-document import

#### Purpose of the function

Possibility of automatically lettering the part name when importing multiple documents TOP  $\rightarrow$  PCH and DXF/DWG/IGES  $\rightarrow$  TOP  $\rightarrow$  PCH.

#### Triggering the function

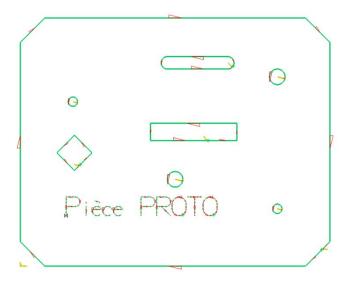
The automatic lettering can be configured in **Tools > Options** in each multi-document import section.

		Application configu	ation		
Configuration User information Colors Shortcut key General General Javameters Cometry Uther options Multidocs DVP/DVG/IGES Uther options Multidocs DVP/DVG/IGES Uther options Multidocs DVP/DVG/IGES	Multidocs TOP → PCH  ✓ Use file interface  Interface file list Text file extension Csv  Create PCH file in the same folder  ● The first line of the list file gives th			or thickness mm v	
<ul> <li>Factor</li> <li>Simulation</li> <li>Simulation</li> <li>Tasks manager</li> <li>Display options</li> </ul>	Header columns names Folder Path Name Path Quantity Thickness Matter Machin Designation Customer OF Plane Set Date	REPERTOIRE REPERE Dremin complet QUANTITE Epsisseur Machine Désignation Client Commande Plan Ensemble Date	Order of the columns Available fields Part Name Folder Path Matter Thickness Machin Quantity Designation Octomer Plane Set Date	Ordered columns	
	Use multiparts recognition per desit Ask general parameters for each p Basily part machining document Create a Nesting Order NO	art I Automatic al			$\mathbf{D}$

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).

Image: Solution and constraints       Parts lettering (import multiple)         Image: Solution keys       General parameters         Image: Solution keys       Beneral parameters         Image: Multiples       Multiples         Multiples       Parch         Cotd       Machine Image: Solution throughers         Tasks manager       Automatic Lettering of the part during multiples         Tasks manager       Automatic Lettering of the part during multiples         Tasks manager       Automatic Lettering of the part during multiples         Tasks manager       Automatic Lettering of the part during multiples         Tasks manager       Automatic Lettering of the part during multiples         Tasks manager       Automatic Lettering of the part during multiples         Tasks manager       Display options         Biotom Lett       Center         Top Flight       Botom Lett         Center Botom       Botom Flight		
Colors Shotcut key General Impotation Security Dither options Multidocs TDP > PCH Multidocs TDP > PCH Mult	Configuration	Parts lettering (import multiples)
Shortcut key General Display options Several parameters General Other options Display options Display options Machine finn Machine fi		
General parameters General parameters General parameters Dither options Multidocs TDP > PCH Multidocs TDP > PCH Multidocs DXP / PWGA/GES P Punch Cut Simulation P Parts lattering (import multiples) Tasks manager Display options Tasks manager Display options Machine finn P Automatic Lettering of the part during multidocs importation Character fort [C:Missler/W616/local/tench/Vettering top Tosk height [10.00mm Kind of too] Denter Top Lett Center Top Character fort	🗈 😌 Colors	
General parameters General parameters Dither options Multidocs TOP > PCH Multidocs DXP/TOV/G/IGES Ponch Cut Nesting Tasks manager Display options Machine finn Post processor Tasks manager Display options Machine finn Post processor Character fort [C:Missler/W616/local/french/Uettering top Rowse Text height [10.00mm Kind of too Normal tool Rollerball tool Normal tool Center Top Charecter Top Charecter Top Charecter Top Charecter Top Charecter Top Charecter Top Charecter Top Center Top Right	🚕 Shortcut key	
General parameters General parameters General parameters Dither options Multidocs TDP > PCH Multidocs TDP > PCH Multidocs DXP / PWGA/GES P Punch Cut Simulation P Parts lattering (import multiples) Tasks manager Display options Tasks manager Display options Machine finn P Automatic Lettering of the part during multidocs importation Character fort [C:Missler/W616/local/tench/Vettering top Tosk height [10.00mm Kind of too] Denter Top Lett Center Top Character fort	🗌 🚭 🚱 General	
General parameters General parameters General parameters Dither options Multidocs TDP > PCH Multidocs TDP > PCH Multidocs DXP / PWGA/GES P Punch Cut Simulation P Parts lattering (import multiples) Tasks manager Display options Tasks manager Display options Machine finn P Automatic Lettering of the part during multidocs importation Character fort [C:Missler/W616/local/tench/Vettering top Tosk height [10.00mm Kind of too] Denter Top Lett Center Top Character fort	📄 💼 Importation	
Other options     Multidocs DX PDW5/DW5/AGES     Punch     Cut     Simulation     Post processor     Tasks manager     Display options     Text height[1:00mm     Kind of tool     Normal tool     Rollerball tool     Marking tool     Center Top     Top Right     Center Top     Center Right	🗌 🚽 🐻 General parameters	
Multidoss TOP > PCH Multidoss DXF/DWG/IGS Punch Simulation Post processor Tasks manager Display options Machine finn Machine finn Mac	- 🔂 Geometry	
Muldidees DXF/DWG/IGES         Parts         Parts         Simulation         Past processor         Tasks manager         Display options         Machine finn         Automatic Lettering of the part during multidocs importation         Character form[C:\Missler\V616\Ucoal\trench\Uettering top         Bisplay options         Text height [10.00mm         Normal tool         O Rollerball tool         Marking tool         Lettering position in the part         Top Left       Center Top         Center Top         Contert Left	🕂 🛗 Other options	
Parts lettering (import multiples)     Punch     Cut     Machine finn     Post processor     Tasks manager     Display options     Text height 10.00mm     Kind of tool     Normal tool     Rollerball tool     Marking tool     Lettering position in the part     Center Top     Top Right     Center Right	Multidocs TOP -> PCH	
Punch     Cut     Machine finn     Post processor     Simulation     Post processor     Display options     Machine finn     VAutomatic Lettering of the part during multidocs importation     Character fort/C:\Missler\W616\local\trench\lettering top     Browse     Text height[10.00mm     Kind of tool     Normal tool     Order Top     Center Top     Top Right     Center Top     Center Right	Multidocs DXF/DWG/IGES	
Cut Nesting Post processor Tasks manager Display options Character fort[C:\Misslet\V616\local\french\lettering top Text height[10.00mm Kind of tool Normal tool Center Top Center Top Center Top Center Left Center Top Center Right		
Nesting     Simulation     Post processor     Tasks manager     Display options     Vesting (100mm     Kind of tool     Normal tool     Center Top     Conter Left     Center Top     Center Right		
Simulation     Post processor     Post processor     Display options     Character tort C:\Missler\V616\Jocal\Irench\Uettering top     Text height 10.00mm     Kind of tool     Normal tool     Rollerball tool     Marking tool     Lettering position in the part     Top Left     Center Top     Top Right     Center Left     Center Right		
Simulation         Post processor         Tasks manager         Display options         Character fork[C:Missler/W616/local/french/lettering top         Browse         Text height[10:00mm         Kind of tool         Normal tool         Center Top       Top Right         Center Left       Center         Center Left       Center		Marshine Finn y
Tasks manager       Image: Automado Lettering of the part during mutadocs importation         Display options       Character fork[C:\Missler\V616\local\french\lettering top         Bisplay options       Tasks manager         Character fork[C:\Missler\V616\local\french\lettering top       Browse         Text height[10:00mm       Mormal tool         Normal tool       Image: Rollerball tool         Lettering position in the part       Top Left         O Center Left       Center         Center Left       Center		
Tasks manager       Character fort [C:\Misslet\V616\local\french\lettering.top       Browse         Text height       10.00mm       Kind of tool       Normal tool       Rollerball tool       Marking tool         Lettering position in the part       Top Left       Center Top       Top Right       Center Keit       Center Right		Automatic Lettering of the part during multidocs importation
Text height [10.00mm Kind of tool Normal tool  Rollerball tool  Marking tool Lettering position in the part Top Left  Center Top  Top Right Center Left  Center Center Right		
Kind of tool       Image: Rollerball tool       Marking tool         Normal tool       Image: Rollerball tool       Marking tool         Lettering position in the part       Top Lett       Center Top         Top Lett       Center Top       Top Right         Center Left       Center       Center Right	🗄 🚍 Display options	Character font C:\Missler\V616\local\french\lettering.top Browse
Normal tool     Rollerball tool     Marking tool      Lettering position in the part     Top Lett     Center Top     Top Right     Center Left     Center Conter Right		Text height 10.00mm
Lettering position in the part Top Left O Center Top Top Right Center Left Center Conter Right		Kind of tool
Lettering position in the part Top Lett Center Top Top Right Center Lett Center Conter Right		Normal tool  Rollerball tool Marking tool
O Top Left     O Center Top     O Top Right       O Center Left     O Center     O Center Right		
Center Left Center Center Right		
Bottom Left     Center Bottom     Bottom Right		
		Bottom Left Center Bottom O Bottom Right
		OK Creed

#### 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**.

Configuration	Machining associated	to the nest						
🖉 User information								
🖶 🍊 Colors	Start point							
🝰 Shortcut key	<ul> <li>Top left</li> </ul>		O Top right					
🕹 General								
	Bottom left		<ul> <li>Bottom right</li> </ul>					
E Punch								
therefore the second s	Direction of advanc							
📕 🧑 General	• X+	⊙×-	○ Y+	O Y-				
- 🚰 Cut parameters								
Priming/Exit	Common cutting							
- Insertion elements	Search tolerance 5.0	Mmm	Tolerancy of reb	uild 0.10mm		-		
- Stating - Marking - Burning - Stating of the part			,					
	Creation	standard			~			
— Machining associated to the nest	🖌 🖌 Do a trim of comm	non cutings opened	I curves with the main sl	neet				
Cutting skeleton								
- Micro joint	Automatic cut to del	tach rest of sheet				_		
- Tube cutting	Make the cut of t							
🗄 🏹 Nesting								
🗄 🚜 Simulation	O Horizontal O V	ertical O Horizor	ntal then Vertical 🔘 Ve	ertical then Ho	izontal 💿 In L			
Sector Post processor		-	_		-			
🗄 🧏 Tasks manager	Distance minimum fro	om last part 30.00m	m Minimum size	e of the waste	300.00mm	for bystronic	~	
E Display options	Standard wastes							
	Multiple value fro	om minimum size	Interva	al 200.00mm				
	↓ ✓ List of possible le	engths ( separed by	; ) 320mm ; 450m	m				
	✓ With priming	With exit	Retracted priming	Betract va	lue 10.00mm	-		
	<ul> <li>Put distributed mid</li> </ul>		Includeded plinning	mediace va				
	<ul> <li>Resize the sheet</li> </ul>	after the cut rest						
	Manage the cut r	est created automa	ticaly in tasks manager					
					<ul> <li>II. 1.202</li> </ul>			
	Make all drillings o	n the sheet before	cutting 🗹 Addia s	top at the end	or all drillings			
	Manual inserstion of							
	<ul> <li>Keep local maching</li> </ul>	nings set in nest do	cument					
	- Enclosing rectangle							
			0mm Margin in Y-0.1	00mm Margi	n in Y+ 0.00mm			
		2 [2.			,			
		_	OK Can					

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.

ΟK	Mode= Manual 🔍	Size of waste along Y	1900	Along Y 500	Cutting rest	InL 🗸	LeadInOut	Mark waste	YES 🗫
----	----------------	-----------------------	------	-------------	--------------	-------	-----------	------------	-------

**<u>Note</u>**: 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.

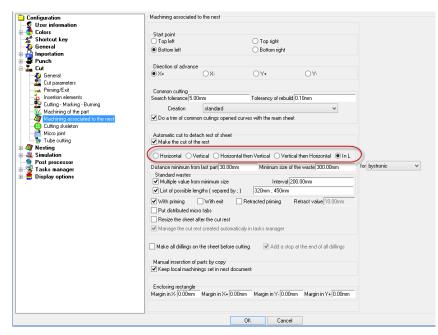
#### **Triggering the function**

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

	Application configuration	
Configuration	Machining associated to the nest	
Shortout key General Punch Punch Automatic allocation Unit Toterancies Automatic allocation Unit function Cut Cut Network Net	Automatic splitting to detach rest of sheet ☑ Make the splitting to cut the rest	
	Horizontal Vertical Horizontal then Vertical Vertical then Voizontal I in L Distance minimum from last part [30.00mm Minimum distance of no creation from sheet border [300.00mm Standard weate Multiple value from minimum size Interval [200.00mm List of possible lengthst [espeed by ; ] [20mm : 450mm Evacuation of the rest	for finn v
	Stop     Officerotabs Tool name to use by default Best tool     Micro tabs     Micro tabs     Mochning order for the cut rest splitting     Before alignments     Officerotabs     At last	
	Control and the indicating of the cut rest     Control and the cut rest     Manage the cut rest created automatically in tasks manager	

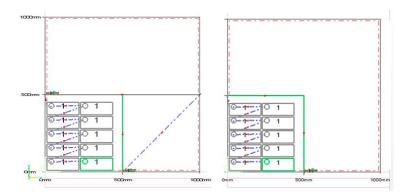


For cutting machines, the menu of the **Cutting rest** feature has been modified.

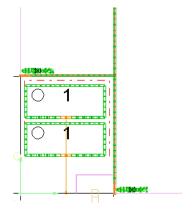


A new LeadInOut button allows you to configure the primings/exits, as well as maintain wastes with micro-tabs.

Cutting examples:



Punching example with automatic resizing:



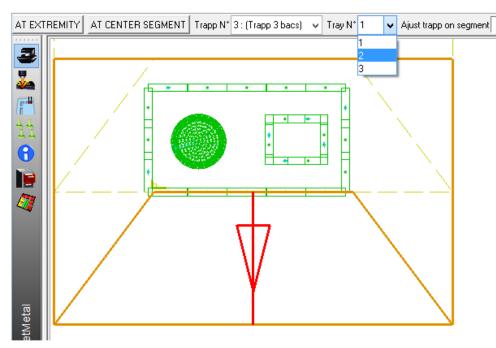
# Trapp door evacuation function - Tray number

#### Purpose of the function

Possibility of selecting the number of the evacuation tray.

#### Triggering the function

#### Trapp door evacuation menu.

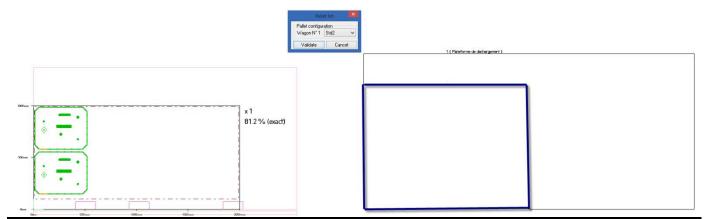


# Pallet preview

#### 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.

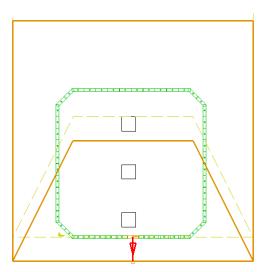
#### Triggering the function

Configuration in Tools > Options > Display options > Sequences.

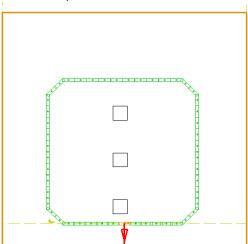
• Check the Simplified visualisation of the trapp door box.

	Application configuration
Configuration 2 User information 3 Colors 3 Shortcut key 6 General	Sequences           Show the punching sequences
←	✓ Show sens of punching sequences ✓ Show sens even if there is only one punch
Simulation     Sost processor     Tasks manager     Display options	Colour of micro attachs
Sequences Shearing Westing Turret	Draw circle around micro attach Multiply factor of height
	✓ Show security area of special tool Colour punching sequences or simulation sequences
	Filling colour
	<ul> <li>☐ Vizualization of indexes of schedul sequences</li> <li>☐ Write of text SP on each special tool punch</li> <li>☐ Write name of special tool on each special tool punch</li> </ul>
	Display trapp door name      Display symbolic trapp door shift      Simplified visualisation of the trapp door
	Text height for numerotation(-1 corresponds to default height)
	Colour of selection of simulation sequences in manual tool order
	OK Cancel

#### Full visualization



#### Simplified visualization



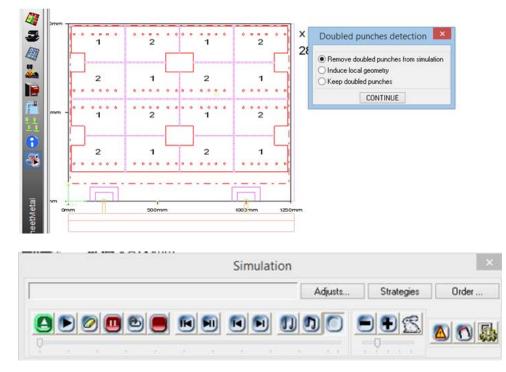
# Visualization of doubled punches before simulation

#### 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.

Order name			Th				br. Global was			G	Time	Date	Files path				
UNITAIRE#acier#10#MASSE			1.00mm 1.00mm			<u>4</u> 1	31.21% 39.93%	59	trumpf3000_sheet		763.562s 27.562s			_Patricia\ANNEE 2014' _Patricia\ANNEE 2014'			The second se
< Simple ope	rator card			<b>2</b>	1	Com	olete operator ca	rd LCT#aci	er#10#MASSELIN_C	<b>1</b>	5	Lat	bel card		 [2] [3]	>	trumpf2000
Composition of the order																	
LCT #acier#10#MASS LCT #acier#10#MASS LCT #acier#10#MASS	ELIN#03.meg	2000.00	Imm 10	00.00mm 1 00.00mm 1 00.00mm 1	I 15 I 20 I 23	33.04%	97.99s 11/ 139.55s 11/ 441.80s 11/	12/2014									2000.00mm × 1000.
ISO file No	ot created				10	0	perator card LC	T#acier#10	#MASSELIN#01 📑	2		Label card L	abel not cre	ated 📑 👌	3	Comment	
Parts in the sheet							pordior ouro juio	- Haddon Fre		99		eases care le	00011101 010		2	Common (	
Ref. Nesting. BOR-P-006 LCT#acier#10	#MASSELIN#0	Des. 1.meg Tole	Paremen	S Pla		Custo 4040 MAS			Priority Date 0 12/12/2014								ò.

## 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**

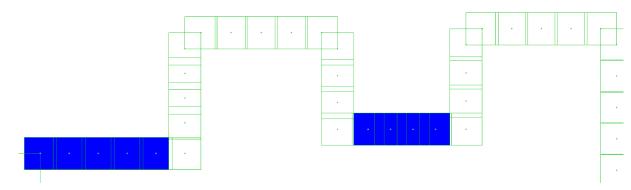
# Notches decomposed into lines

#### 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

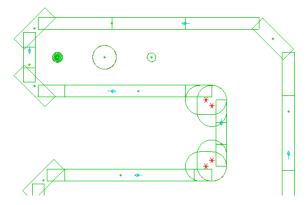
#### Purpose of the function

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

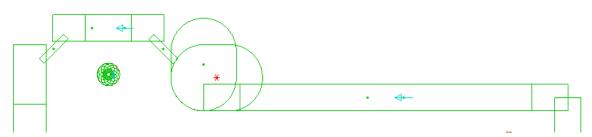
#### Triggering the function

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.

Configuration	Adjustments	
User information	Ignore and induce segments of length lower than 0.50mm Circles of diameters (separed I	by ; )
🔮 Shortcut key	Notching	
Importation  Punch  Tolerancies	Limit at box triangular notch using rectangular/square tool     Tool name to use by default for notches decomposed into lines None	✓ for trumpf2000 ✓
Allocation	Lines Allocate O None O All O All except on box Maxi. tool width 5.00mm	
- Jools order	Repeats search	

#### 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.

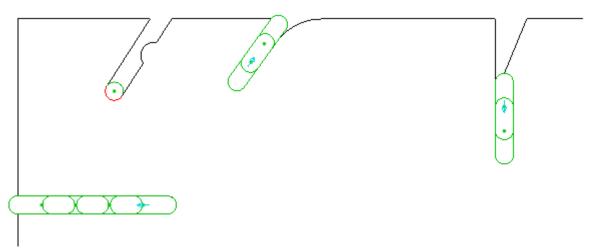


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## Automatic allocation - Oblong groove

#### 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**.

	Application configuration	
Configuration	Machining associated to the nest	
User information Shortcut key General Punch Toterancies Allocation Comment Co	Automatic splitting to detach riest of sheet Make the splitting to detach riest of sheet Horizontal Vertical Horizontal then Vertical Vertical then Vorizontal In L Distance minimum from last part (30.00mm Minimum distance of no creation from sheet border (300.00mm) Just of Distance sheet and the state of the sheet last of the sheet last of the sheet at the sheet	for finn v

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**

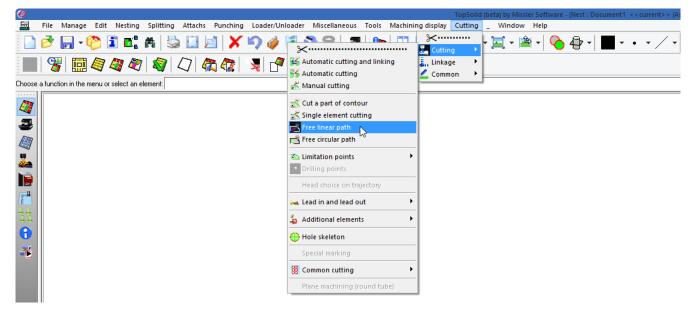
# Free cuttings in the Part document

#### Purpose of the function

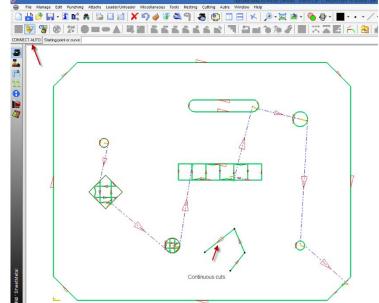
Possibility of performing free linear and circular cuttings in the Part document.

#### **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:

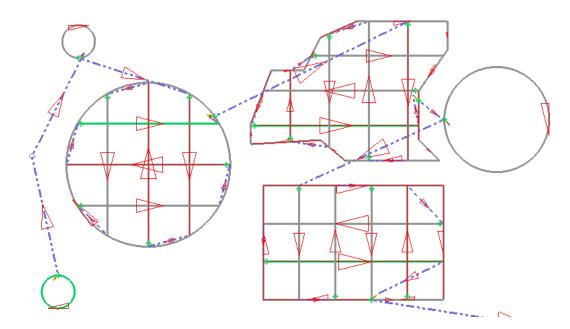
- The minimum size of holes.
- The gap between vertical lines and the gap between horizontal lines.
- The priming length.
- The retract value from the geometry.
- The head position during link trajectories.
- For full circles, you can choose whether the skeleton is to be cut continuously.
- For rectangles,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

Internal skeleton glo	obal paramet 💌							
Make skeleton for hole greater than 50.00mm								
Gap along×25.00mm	Gap along Y 25.00mm							
Priming length 2.00mm								
Retract from geometry 0.20mm								
<ul> <li>Position of head during line</li> <li>Head Down</li> </ul>	nks trajectories O Head Down							
Cut the skeleton continously								
On rectangles	Sag value 7.00mm							
OK	Cancel							

#### Question bar:

Pa	rameters	Start= BOTTOM LEFT	Ends skeleton near leadin= Y	ESF	Cuttings order= VERTICAL	~	Select holes to cut as waste
10000						0.000	

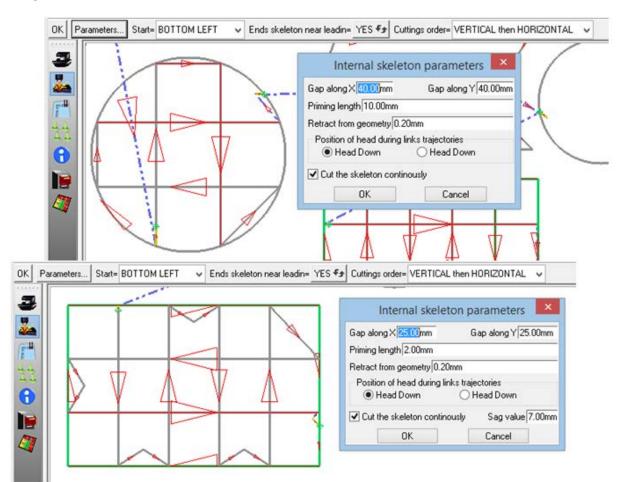
#### Example:



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 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.

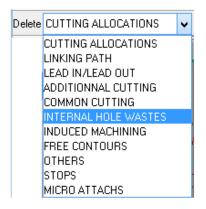
The starting corner may also vary depending on the new number of lines in order for the last line to be as close as possible to the priming.

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

	Application configuration	×		
Configuration	Cutting skeleton Skeleton on sheet Choose machine bystronic Sheet length Sheet width Matter Thickness	Internal skeleton Choose machine bystronic Make skeleton for hole greater than 50.00mm Start		
Image: Constraint of the second se	Start Urder of cuttings Gaps between cutting lines Gap in X between vertical lines Gap in Y between horizontal lines	VERTICAL ✓ VERTICAL ✓ VERTICAL ✓ Control Statement (Control Control Contro		
	Hetract values         Retract on the sheet           Head position for linkings         Leadin length	Retract from geometry[0.20mm Position of head during links trajectories		
	Machining order of cutting of skeleton  Make before the parts  Not specified  Make cutting skeleton on holes of size greater than  Trait skeleton hole per hole Maximum gap between line along X  Along X	On circles On rectangles Sag value 7.00mm		

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** (2)).

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 Nearest extremity 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

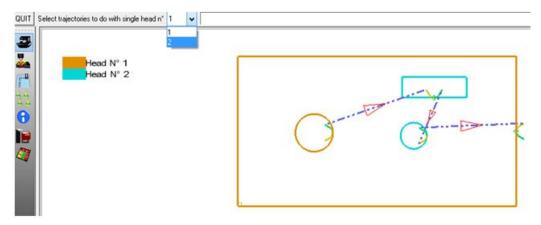
#### Purpose of the function

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.

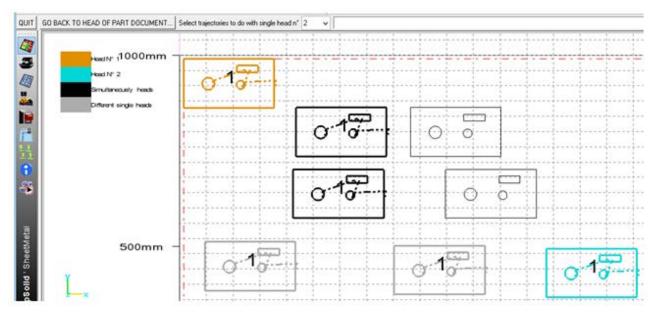


 Select the head number from the drop-down menu, and click the trajectory to be modified, which will then be colored as the selected head.

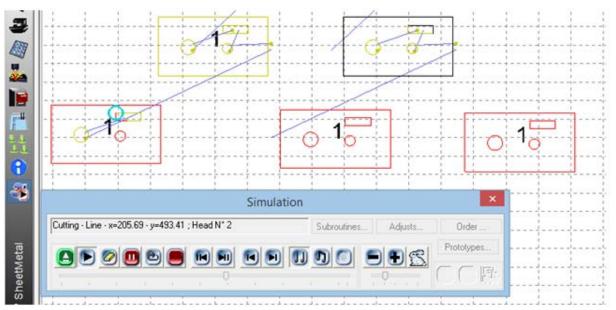
This function is available in the nesting document. Simultaneous head trajectories are displayed in black. Part cutting instances performed with different heads are shown in dark gray.

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)

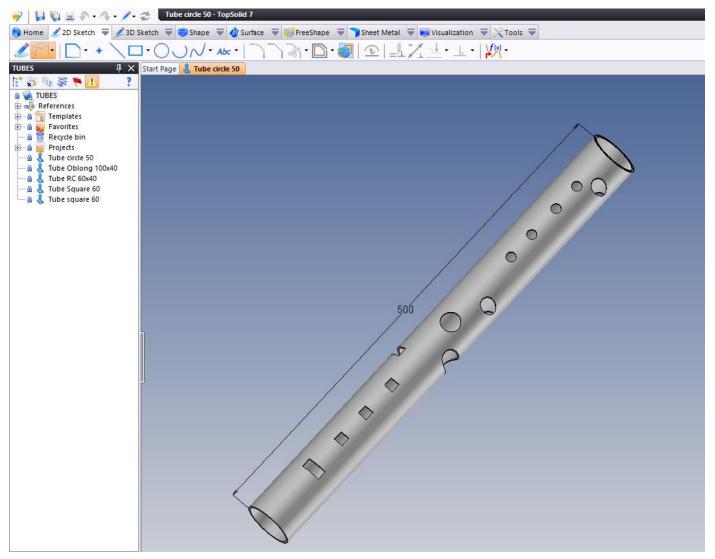
#### Purpose of the function

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** </u> function.

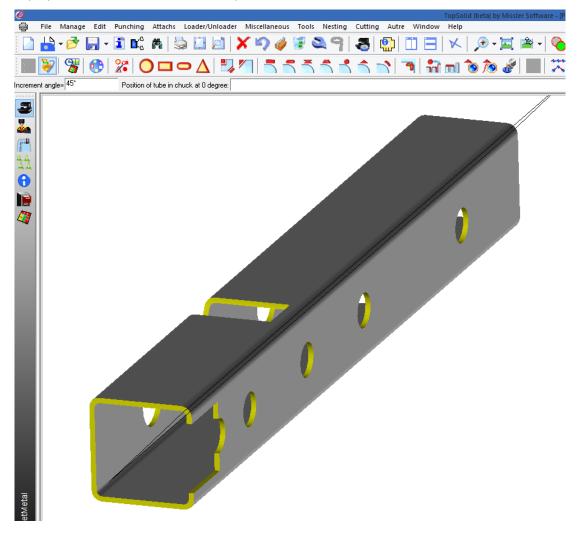
EXPLORER V7 EXPLORER CURRENT V7 PROJECT V7 TUBE EXPLORER

• Choose V7 TUBE EXPLORER.

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

TOPSOLID 7 EXP	.ORER
Project: TUBES Tube circle 50.A.3 Tube Oblong 100x40.A.0 Tube RC 60x40.A.2 Tube Square 60.A.5 Tube square 60.A.1	

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



• Choose NO TRANSFORMATION.

NO TRANSFORMATION CHOOSE TRANSFORMATION Basify Machining document= NO 🗫 Input data for tasks manager= NO 🗫

#### • Select the machine.

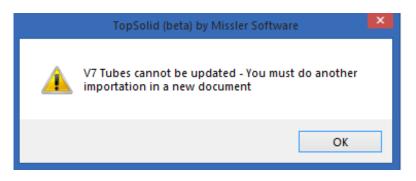
Gen	eral parameters : tube_bv_sup.A.pch	×
	Thickness 3.00mm Machine tubematic	Matter typ
	Cutting Gaz Non specified gaz Cutting parameters selection By contour	✓ Tube Cutting 3D (chuck)
	Liberties Rotation forbidden Symetry for Family	Part colouring
	Importation           Importation	Opened lines and arcs Given Text colour
	Default wanted quantity 1	-

• Creation of the curves in the Part document.

# TUBE : Rectangular : 40.00mm x 20.00mm Radius Ext. : 5.50mm

#### 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**.

		Application configuration	
Configuration 9 User information 9 Colors 2 Shortcut key 3 General		declare cutting tube machin	
General     General     General		e cutting machin (2D_3D) 3D tube cutting machin	
Cut parameters Cut parameters Priming/Exit Cuting - Marking - Burning Cuting - Marking - Burning Machining associated to the nest	Tubes comparison tolerancy	0.0100mm Angle tube tolerancy 0.100*	
Cutting skeleton Micro joint Tube cutting B 4 Nesting	Vizualization options	Parameters depending of the machin Neutral fiber position used during tube unfold (-1=external cotation)]-1	
<ul> <li>Base Simulation</li> <li>Post processor</li> <li>Base Tasks manager</li> <li>Display options</li> </ul>	Raw product in surface in solid	Importation of tube part with adjust of loss compute with confirmation     Unload of tubes parts     Linking trajectories mode     Internal before     Internal before     Do not close cutting trajectories on external of tube	
	Transparency of the raw product	Tolerancy of polygonization for arcs       Length     [0.10mm       Arcs     [5.000*   Simulation cutting speed [0.040m/s	
	Colour parts	Image: Security margin         Near chuck (left)         Image: Security margin         Image: Security margin         Near chuck (left)         Image: Security margin         Image: Security margin	
		OK Cancel	

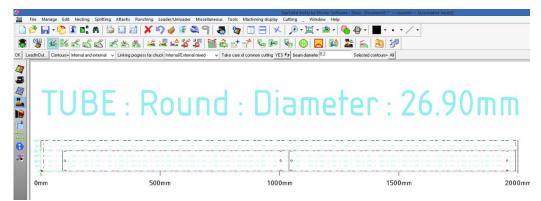
#### Improved complex nesting for round tubes

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

Nesting order - Complex nesting	Nesting order - Complex nesting
Pats       Gaps       Sheets       Nesting       General         Ref.       Q.       Add         Nube_galva_niania A       Search         Remove <ul> <li> <li> <li> </li> <li> <li> </li> <li> </li> <li> </li> <li> <li> </li> <li> <!--</th--><th>Pats       Gaps       Neets       Neeting         Gaps between pats       Gaps between pats       Gaps between pats       Gaps between pats         Gaps between pats       Modes       Interview       Interview         Take care of tools allocations       Interview       Interview       Interview         Take care of tools allocations       Interview       Interview       Interview         Leadin       Variant       Change         Model       Variant       Change         Common cuttings       Interview       Interview         Common cutting gap between pats       0.00mm         Minimum common cutting line       0.00mm</th></li></li></li></li></li></ul>	Pats       Gaps       Neets       Neeting         Gaps between pats       Gaps between pats       Gaps between pats       Gaps between pats         Gaps between pats       Modes       Interview       Interview         Take care of tools allocations       Interview       Interview       Interview         Take care of tools allocations       Interview       Interview       Interview         Leadin       Variant       Change         Model       Variant       Change         Common cuttings       Interview       Interview         Common cutting gap between pats       0.00mm         Minimum common cutting line       0.00mm
Matter[acier Type Thickness]2.90mm Machine[tubematic Check parts compatibility (matter and thickness)	Matter[acier Type Thickness]2:90mm Machine[lubematic ✓ Check parts compatibility (matter and thickness)
Load Save OK Cancel	Load Save OK Cancel

	Nesting order	- Complex nesting		
arts Gaps	Sheets Nesting General			
	Tube dimensions		Add	
			Explorer	
	Length 2000mm		Remove	
	Use width as 1.5 x tube diameter		● <b>●</b> Optimize	
Quantity ; Prior Quantity	ity Free quantity	Priority	Price	
Dimensions — _ength ⊮idth				
Security margin Left	15	10.00mm		
Right		10.00mm		

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.

Lead in method	>	Perpendicular	~
Lead in distance	>	15mm	
Line lead in angle	>	90*	
Lead in distance	>	15mm	
Lead in angle	>	60°	
Lead in height	>	7.5mm	
Lead in feed rate	>	Tool feed rate	~
Tangential lead in feed rate	>	Tool feed rate	~
Lead in direction	>	XY_Z	¥
🔲 Activate Dia. Comp. before Z	mvt		
Overlength		6.3508000 /35	
Along tangent		Along curvature	

#### **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.

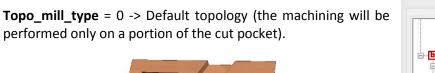
**.ComplexForm.1.Form.ToolRotat = " "** -> The selected tool has a clockwise or counterclockwise rotation according to the first matching tool.

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.

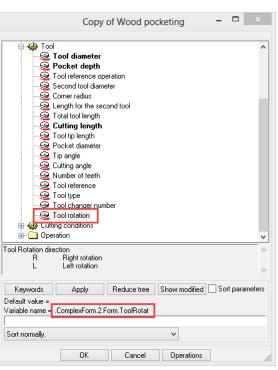
It is possible to choose in the process the type of topology to be used.



**Topo\_mill\_type** = 1 -> Topology given by the feature (the machining is performed as if no operation crossed the part).



	Wood	Pocket Cont	ouring -	
Procee     Proceee     Proceeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeee	s <b>tivated</b> g conditions tion eference points oproach & retrac ] Retract ] Lead in unge etracts arameters	<b>Topology Type</b> ges	]	^
Contouring Topolog		<u> </u>		~
0 1	default topolog topology given			~
Keywords	Apply	Reduce tree	Show all Sort	parameters
Default value = 0				
Variable name = .0	ComplexForm.3.t	opo_mill_type		
1				
Sort normally.			¥	
			2 <u>2</u> 2	
	OK	Cancel	Operations	

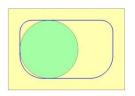


# Publishing of the maximum, minimum and optimized tool diameter values for pockets

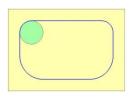
Three new variables have been added to the already existing pocketing list. They are available when editing the process parameters via the **Keywords** button.

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.

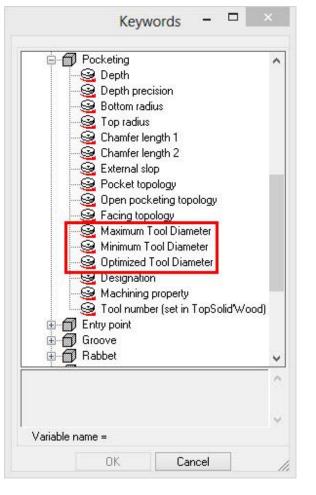


 Minimum Tool Diameter = Minimum circle diameter (different from zero) which is bitangent or tritangent to the sketch elements. The result is zero if the pocket is not radiused.



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

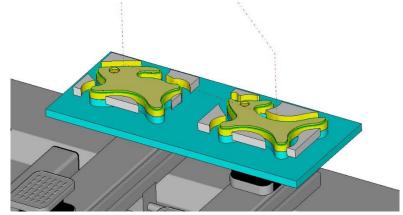




The values of the above parameters can be accessed directly via the **Info** button available after the topological analysis of a face (if this face has a pocket topology).

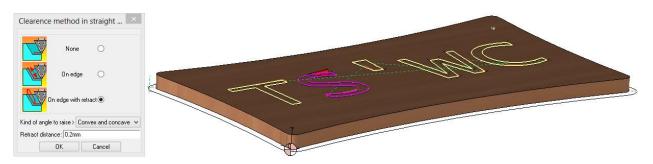
#### Verification on several parts

This new version allows several parts to be **verified** *w*. 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** <sup>34</sup> in the version 6.15 is now available in the **Engraving** function, and works the same way.

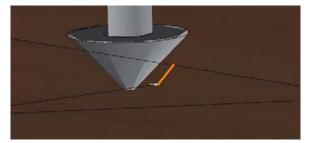


Two clearance modes are available: **On edge** and **On edge with retract**. It is also possible to choose the kind of angle to be raised: **Concave**, **Convex** or **both**.

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** <sup>1</sup> 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.

Chuck of	distanc	e –		
Would you like to c	hange cl	huck distance of	selecte	d tools ?
☑ Take account s	stock	0 ver length >	25mm	
Take account o	of clamps	( There is no cla	mp)	
Chuck distance > 5	55mm			
	ОК	Cancel		/

# **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.

Positioning	
🖲 Normal	<ul> <li>Turned over</li> </ul>
Parts	
Create one p	oart by family
Create all pa	arts
Machining	25
<ul> <li>Verticals operations or</li> </ul>	nly
Choose operations to	machine
Cuttings	Grooves
	Rabbets
User machining	Pockets
Mouldings	Holes
Counter-mouldings	Aligned holes
✓ Optimize paths ○ With shapes proximit	,
With machinings app	
With operations priori	ties
Shapes cutting	
From the smallest to	8
O From the largest to the	
With shapes proximi	
	proach/retract points
<ul> <li>With machinings app</li> </ul>	
With machinings app	

# 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).

	Shared tool database (D:\Software	e\Missler\V616\Group\Database\Reduc	ed modified by \\POF	RTABLE AY	V\ay\	1			
nits	Tool changer Prepared tools	Tool choi	ce Saving Informa	ation					
R î	T1	Tools of	origin				10		
	T2		Database	OF	repar	ed			1
- ব্য	ТЗ	E1	- Lotor					1.15	
	Τ4	Family	14 3000 F 14 0 M2						
	T5	MILL T	00L				~		
	T6	Types	choice						
	T7	SLOT	14.55.514.44.5148				~		
	Т8	32011							
	Т9		Tools list						F
	T10		Internal reference	Tool off	D	Ŀ	Material	^	
	T11		FR-2TA-1	×	1	50	Fast steel (HS)		ſ
*	T12		FR-2TA-2	×	2	10	Fast steel (HS)		
	T13	<b>5</b>	FR-2TA-3	×	3	15	Fast steel (HS)		
	T14		FR-2TA-4	8	4	20	Fast steel (HS)		1
	T15	<b>&amp;</b>	FR-2TA-5	×	5	25	Fast steel (HS)		1-
			FR-2TA-6	×	6	30	Fast steel (HS)		3
	. (a) (	9 - 1	FB-2TA-7	×	7	35	Fast steel (HS)		

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       Machine name : TopSolid'WoodCam 5X Rails&Pode         WoodCam       WoodCam						
Tool holder name	Number	Ref.	Tool material	Tool diameter	Cutting length	
TP1	1	Fraise Calibrage (EDDren	Diamond polyoniatallin ( DP)	20	55	
TP1	2	Fraise Rainure Ø5mm	Carbide of lungations ( Hel	5	25	
TP1	3	Fraise Rainure @10mm	Carthole of tungations ( Hel	10	45	
TP1	4	Fraise Ebauche 252mm	Faul shoul (HS)	32	180	
TP1	5	Praise a chardroiner @20nm	Carbole of longatione ( Hel	20	10	
TP1	6	OUTLS SMPLES_CONGE_R 10	Carbole of lungations (Hell	190	50	
TV1	1	Ponet à font plut Otimes	Faul shoul (HS)	5	25	
TV2	2	Foret & ford plat Offeren	Faul sheet (HS)	5	25	
TV3	3	Foret & fond plat Offeren	Faul sheet (HIS)	5	25	
TV5	5	Foret à fant plat Offeren	Faed shoul (145)	5	25	
TV4	4	Fond Hélicolitar (Minus	Faul aloui (HS)	5	25	
TV9	9	Forel Helicolita (Miner	Faul steal (HD)	6	25	
TV10	10	Forel Hilliositar (36mm	Faul sheet (HS)	8	60	
TV11	11	Foret Helicolital (21Genes	Faul siles (HS)	10	50	
TV12	12	Foret Helicoldar (215mm)	Fast steel (HS)	15	75	
TH1	1	Foret a fond plat Others	Fast steel (HS)	8	40	
TH2	2	Foret a fond plat Offmen	Fast size (HS)	8	40	
TH7	7	Foret à fant plat 28mm	Fant steel (HS)	8	40	
TH8	8	Format is found plast Officeurs	Fant stant (140)		40	

## 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**.

Multi machining Saving directories/subdirectories	
Machining files :	
so files :	
Slave parts :	Ü
Document files:	Ĩ
fachining documents zoom > No zoom	~

#### **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=N0 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 PARAMETERS=OPER Db.Oper.facing\_island\_plunge\_radius\_min\_in=0.0311 Db.Oper.facing\_is