



# TopSolid v6.17 What's New

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# What's new in TopSolid'Wood v6.17



This section describes the new features in the version **6.17** of **TopSolid'Wood**.

# Design

### **Constrained block**

A panel can now be created at the end of the **Constrained block** function, after the distribution.

To do this, start by defining one or more **panel models**.

- In a **new document**, create an extruded shape.
- Launch the **Wood** > **Panel** function, and then create the panel on the extruded shape.

**Note**: The created panel will then be saved as a model. For example, if three edges are applied to this panel model, then three edges will also be applied to the part on which the model will be used. However, the model is for reference only. All the panel settings can be changed later on.

- Launch the **Wood** > **Panel** function again.
- Click MODELS and then DEFINE NEW MODEL
- Select the panel you have just created.
- Specify whether the model has to be saved in the **User** or in the **Group** configuration.
- Finally, name the model and click OK to confirm.

OK Model= USER + Model name= 4 Edges

Once the panel model has been created, it can be used at the end of the **Constrained block** function.

- Create a constrained block.
- At the end of the function, select the panel model you want to use.



After selecting your panel model, two labels are displayed allowing you to select the reference face of the panel.



Note: These labels consist of two separate areas:

- The first area lets you select the panel's reference face.
- The second area lets you change the panel model.



• Once you have created the panel on the constrained block, you can modify it using the **Modify element** function as with all other panel entities.

#### **Construction volume**

Construction volumes can now be created. They can then be used to position a driver block component.

First, you need to define the attributes of the future construction volumes.

- Launch the **Tools > Options > Shape > Construction volumes** function.
- Define the characteristics of the construction volumes.

#### - Layer:

When this option is enabled, the created volume will be automatically placed on the level you specify.

Layer		
✓ Activate	Creation level : 10	l

- Material:

Check this box to define the material you want to apply to the volume. If this box is cleared, the created volume will use the current material.

**Note**: Make sure to choose a transparent material in order to have a transparent construction volume in realistic rendering.

Material
✓ Activate
Filter
Glasses
Clear Mirror Clear Window Glass
Deco Object Glass
Deco Object Glass

#### - Transparency:

This option lets you define a transparency for the construction volume. A value from 0 to 10 can be entered.

✓ Activate	Creation transparency : 7
Color	
<ul> <li>Activate</li> </ul>	Creation colour >

Turnersense

**Color**: If the box is checked, the construction volume will use the color selected in the drop-down list.

- Once you have defined these values, close the Options window.
- Switch to the **Construction volumes** mode by selecting the **Tools** > **Construction volumes** function or by clicking directly on **CV=Off** in the status bar.

```
Absolute cs CV=Off X=+012.500 Y=-040.000 Z=+000.000 ToI= 0.2 On Ax=no Tra=0 Lay= 0
```

• Then create a shape.

**<u>Note</u>**: Only the **Shape** > **Extruded** and **Shape** > **Block** functions can create **construction volumes**.

• The created shape automatically takes the attributes that you have defined.

Here is an example of a construction volume. As defined in the Tools > Options > Shape > Construction volumes section, the extruded shape uses the green color, a transparency of 7 and is created on layer 10.



In the following example, several construction volumes have been created.



### Driver block components are then included in each construction volume.



Here is the final result:



### Panel

### **Elliptical panel**

A panel entity can now be created on an elliptical shape.



#### Laminate sawing-up on panel's maximum dimensions

The panel's maximum dimensions can now be retrieved in the sawing-up of laminates. For a panel with a sloped edge, the panel's maximum length can be retrieved in the sawing-up of both laminates.

- Launch the **Wood** > **Panel** function.
- Open the **Advanced Options** window by clicking the >> button.
- Check the new Sawing-up on panel maximum dimensions option.
- Finish creating the panel.

Advanced options	x
Same characteristics for panel and support	
● ⊾ ges cover∈ a by las Same grain orientation for support and lamina ✓ Sawing-up on panel maximum dimensions	ates
OK Cancel	

In the following example, the panel contains a sloped edge. If the **Sawing-up on panel maximum dimensions** option is checked, both laminates will be 795mm long.

If the box is cleared, the top laminate will be 770mm long and the bottom laminate 795mm long.



# Definition of parts/sets

#### Bent part sawing-up

It is no longer necessary to define the unwound shape of a bent part at each assembly level. The sawing-up information is automatically retrieved.

• Define a **bent part** in a document.

A counter will be used as an example since the front and baseboards are bent parts.

• Include this component in an assembly.



<u>Note</u>: The unwound part is included in the assembly with the component. It is automatically made invisible when inserted to prevent any errors.

• All the document's unwound shapes are grouped together in the unwound shape set.

#### Additional operations

When drilling operations are performed on the bent part, they have to be added to the unwound shape.

- Edit the component in the symbolic tree.
- Expand the **Advanced** node.
- **Right-click** the line of the unwound shape on which the drillings have been created.
- Select Report drills.
- From the graphics area, select the drillings to be added or deselect the drillings that are not to be reported on the unwound shape.
- Click **OK** to end the function.



<u>Note</u>: Only drilling operations can be reported on the unwound shape. However it is possible to report curves, frames or points. To do this, **right-click** on the unwound shape line and select **Report points, frames and curves**.

#### Cut/Machining export

From now on, when a bent part is selected for a cut or machining export, the unwound shape is exported.

#### **Drafting**

You can now define the projection mode of bent parts (i.e., choose whether you want to project the bent part, the unwound part, or both).

- Create a new **Draft** document.
- Launch the File > Properties > Drafting view function.
- Select the projection mode of bent parts.
  - Multi-draft projection of bent part Bent part Unwound part Both

<u>Note</u>: As this setting has been made in **File** > **Properties**, it only applies to this document. To adjust the default value for all documents, define the projection mode of bent parts from the **Tools** > **Options** > **Drafting view** > **Initial values** section.

Again using the counter example, the projection mode of bent parts has been set to **Both** in the multi-drafting template.

**Note**: In this projection mode, the bent part and its unwound shape are projected in different views.

This is the result of the multi-drafting operation for the **Front** part:



#### Management of parts without sawing-up

The **Define part** function has been improved to make the definition of parts without sawing-up more intuitive.

#### Definition of a shape

• Launch the **Define part** function.

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

<u>Note</u>: The **Select axis automatically** and **Bent part** options must now be adjusted before you select the part to be defined.

A new With sawing-up button is available.

With sawing-up= YES+1

If this option is set to **Yes**, the function behaves the same way as in the previous version. The part sawing-up information will be displayed in the bill of material.

With sawing-up= NO+

If this option is set to **No**, the defined parts will have no sawing-up information. This mode is useful when, for example, you define hardware parts.

<u>Note</u>: Once the option has been set to **No**, the adjustment of the **Select axis automatically** and **Bent part** options is no longer taken into account.

In the **Part definition** window, the **Add to cutting-up** box is cleared and the part sawing-up information is grayed out.



#### Definition of a non-shape element

Sometimes non-shape elements need to be inserted in the main set (profiles, coordinate systems, lights...).

- Open the symbolic tree.
- Right-click the **Assembly** line and select **Insert**.
- Select the element you want to insert in the assembly.



<u>Note</u>: Unlike the previous version, the axes are no longer required when adding the following elements to an assembly:

- Curves or sketches
- Lights
- Coordinate systems
- Background images
- Texts
- Dimensions
- Notes
- The **Part definition** windows opens and the **Add to cutting-up** box is automatically cleared. All you need to do is click **OK** to add the element to the assembly.

#### Set over-dimensions

When a part uses the **Additional** or **Fixed** over-dimension modes, the over-dimension can now be set by specifying a parameter that will act as the over-dimension value. This parameter can then be defined as a driver so that its value can be changed when including the component.

- Over-dimensions in Additional mode:

For a part with over-dimensions defined in **Additional** mode, specifying a parameter as the value allows you to include a component with the possibility of changing the over-dimension value.

- Create **parameters** for the over-dimensions.
- In the **part definition**, enter the parameter name as the overdimension value.

Uvervaluation	ns		
Machining			
Sizes	Values	Modes	Over dimens
Length	1200.0mm	additional	OverL=2mm
Width	600.0mm	additional	OverW=2mm
<b>T</b> 1 · · ·	38.0mm	additional	Omm

- Use the Assembly > Define component > Define drivers function to declare the over-dimension parameters as drivers.
- When including the component, the over-dimension values are required.

OK MEASURE Overlength= 2mm

- Over-dimensions in Fixed mode:

The **Fixed** mode is used to force the length, width or thickness of a part by specifying the desired dimension directly.

Below is an example of how this mode can be used on a component using a driver profile. Here, the component length cannot be known automatically since the profile is not necessarily straight. You need to use a parameter to retrieve the length of the driver profile.



The shape section has been extruded along the blue curve. This guide curve matches the length of this seal component.

- Create a length parameter by clicking the 🛄 icon, then click the 🎦 icon.
- Choose the guide curve as the **reference element** (the blue curve here).

   QUIT Reference element
- Name the parameter. Here, the parameter is named Seal\_Length. Name: Seal\_Length
- **Define** the part.
- In the **Over dimensions** column, set the length to **Fixed** mode, and then enter the parameter name as the value.
- As the width and thickness do not change, you can also set their over-dimensions to **Fixed** mode in order to enter their exact value manually.

Sizes	Values	Modes	Over dimensions
Length	246.5mm	fixed	Seal_Length=459mm
Width	18.0mm	fixed	0mm
Thickness	355.3mm	fixed	0mm

Overvaluations

• Once the component has been inserted in the assembly, its length is updated and matches the length of the curve on which the component was included.





### Repetition on curve: Give priority to ends

The new **Give priority to ends** option is available when performing a repetition on curve. It allows you to change the order in which the repetition instances are created.

Give priority to ends= NO 🗫

When this option is disabled, the instances are created one after the other along the curve. Consequently, the last instance is not always the same if the quantity changes.





Give priority to ends= YES 🗫

When this option is enabled, the instances are created alternatively, starting from each end of the curve. This means that, even if the quantity is modified, the instances placed at the ends of the curve are always the same.



This mode is useful when, for example, the instances at the ends undergo further geometry changes, such as a drilling in the above example.

# Assembly

# Components

#### Automatic activation of the component layer

When a component layer is set to **Template attributes** or **Initial values of template** in the **Tools** > **Options** > **Component** > **Attributes** section, the corresponding layer in the assembly is automatically activated in order to see the component immediately when included.

# **Extruded** components

#### Drivers and processes of an extruded component

The drivers and processes are now required when you include an extruded component. It is also possible to perform its processes.

- Create the **extruded component**.
- Declare the component's tools and driver parameters.

In this example, a rail component has been created. This component generates a groove and has two dimensional parameters to be specified when included (the I parameter being the length parameter required to create an extruded component).

✓ TOOLS SET : (1)
☆ Groove : Groove (1)
★ groove\_1 : groove =\*

DRIVERS SET: (3)
 Rail\_Width : Rail Width = 6mm
 Rail\_Depth : Rail Depth = 8mm
 I: length = 100mm

- In the assembly, launch the **Assembly** > **Include standard** function and select the extruded component. In our example, the rail is included in a cabinet in which sliding doors will be positioned.
- Specify the component's driver parameter values.
- Choose the insertion mode. In most cases, the extruded components are inserted by clicking directly on two points in the graphics area or by using the On curves mode.

As shown opposite, two sketches, each containing two lines (blue) indicating the locations of the rails in this cabinet, have been drawn. Consequently, the rails are included using the **On curves** mode.

- The **On curves** mode provides several options:
  - **New contour**: This option allows you to create a new profile or sketch on which the extruded component will be inserted.
  - Generatrix sketch selection mode: If the Global mode is selected, you can select the entire sketch by only clicking one of its segments. The Local mode allows you to select only a segment of the sketch.

**Note**: In **Global** mode, if a segment is subsequently added to the selected sketch, a component will be automatically added.

For inserting the rails in this cabinet, the **Global** mode is used so that the rails can be included on all the selected sketch's segments.



13

NEW CONTOUR = PROFILE + Generatrix sketch= GLOBAL+ Curves



• Then define the key point to be used. A rotation angle and cuts can also be defined.



- Click **OK** to confirm and to complete the insertion.
- If processes were defined in the component model, you can activate them by clicking on the **Automatic** button or perform them manually (as with any other component).

Note: The Automatic mode provides an additional option.

- **Global** mode: The process will be performed for all the components you inserted.
- Local mode: This mode is most often used when processes are performed once the components are inserted using the Assembly > Use process function. Indeed, it allows you to perform only the processes of the selected component.



#### TopSolid 2016

### Nesting

The position of label texts for parts and orientation symbols has been optimized. In previous versions, these texts were placed in the middle of the part's enclosing rectangle. Consequently, for the parts containing open areas or internal pockets, the text was placed outside the part, or worse, on another part placed in the internal offcut of the first part.

**TopSolid** now provides an automatic mode that detects the part's operated areas and prevents the text and symbol from being positioned in these areas.

#### **Old method: Centered position**





Note: Label texts always use the automatic positioning method.

For symbols, the **Automatic** positioning mode must be specified in the **Tools** > **Options** > **Nesting** > **Nesting default parameters** section.

Label 🖌 Generate labe	symbol	
Position		
Automatic 🗸 🗸	1	Browse

# Wood machining

# Machining export interfaces

### 2D½ DXF export

In the **CADCode** interface of the 2D½ DXF export, you can now choose the export unit.

#### OK Interface= CADCode 🗸 Unit= INCHES 🗫

In the **TopSolid** configuration directory, the **cadcode.cfg** and **cadcode.bom** files are no longer used. Instead **TopSolid** adds two new configuration files whose names contain the export unit. There are now four configuration files for this interface in the **\$TOPCONFIG\dxf\_wood** folder:

- For exports in millimeters: cadcode#mm.bom and cadcode#mm.cfg
- For export in inches: cadcode#inch.bom and cadcode#inch.cfg

**Note**: If the contents of the previous configuration files were customized, these changes must be applied in the new corresponding files.

# Maestro interface

A new machining interface is available, allowing you to export parts designed in **TopSolid'Wood** to **Maestro**. **TopSolid'Wood** generates a .xcs file that contains all the machining information of the part (toolpath, position, tools...).



# **Bill of material**

# Recovery of parent information

It is now possible to retrieve information on the parent below the final parent. For example, you can retrieve the designation of the assembly to which the part belongs on several levels.

For example, a cabinet is created in a file.



This cabinet is included in a new document with a door.



To finish, this **Cabinet with door** component is included several times in an assembly. Their **designation** has been changed so that they can be quickly identified.



In this example, thanks to the **OWNER\_N** BOM property, it will be possible for the **Top** part to retrieve the designations of assemblies in which this part is included: **Simple cabinet**, **Cabinet with door A**.



- Create a new bill of material file in .bom format.
- Open the file in with a text editor.
- Use the following BOM column to retrieve the designation of the Cabinet with door A assembly.

NAME=DESIGNATION\_PARENT "DEF=<DESIGNATION|OWNER\_N>" TYPE=STRING ALIGN=LEFT TITLE\_ALIGN=LEFT WIDTH=0.015 FORMAT=02|mm VISIBLE=YES .

• To retrieve the designation of the sub-assembly level, create a new column and add the assembly level of the set after the **OWNER\_N** property.

In this example, the **OWNER\_N** property of the **Top** part matches the **Cabinet with door A** set. To retrieve the designation of the **Simple cabinet** assembly, use the **OWNER\_N-1** property. Similarly, you can retrieve the properties of assemblies on X level (N-1, N-2, N-3, etc.).



In the BOM file:

NAME=DESIGNATION\_PARENT\_N-1 "DEF=<DESIGNATION|OWNER\_N-1>" TYPE=STRING ALIGN=LEFT TITLE\_ALIGN=LEFT WIDTH=0.015 FORMAT=02|mm VISIBLE=YES ;

Here is the BOM result:

1	Тор	Cabinet with door C	Simple cabinet	1262.00	500.00	19.00	1262.00	500.00	19.00
1	Тор	Cabinet with door B	Simple cabinet	562.00	500.00	19.00	562.00	500.00	19.00
1	Тор	Cabinet with door A	Simple cabinet	562.00	500.00	19.00	562.00	500.00	19.00
1	Right	Cabinet with door C	Simple cabinet	500.00	500.00	19.00	500.00	500.00	19.00
1	Right	Cabinet with door B	Simple cabinet	700_00	500.00	19.00	700.00	500.00	19.00
1	Right	Cabinet with door A	Simple cabinet	900.00	500.00	19.00	900.00	500.00	19.00
1	Left	Cabinet with door C	Simple cabinet	500.00	500.00	19.00	500.00	500.00	19.00
1	Left	Cabinet with door B	Simple cabinet	700.00	500.00	19.00	700.00	500.00	19.00
1	Left	Cabinet with door A	Simple cabinet	900_00	500.00	19.00	900.00	500.00	19.00
1	Door	Cabinet with door C	-	496.00	1296.00	44.00	496.00	1296.00	44.00
1	Door	Cabinet with door B	-	696.00	596.00	44.00	696.00	596.00	44.00
1	Door	Cabinet with door A	-	896.00	596.00	44.00	896.00	596.00	44.00
1	Bottom	Cabinet with door C	Simple cabinet	1262.00	500.00	19.00	1262.00	500.00	19.00
1	Bottom	Cabinet with door B	Simple cabinet	562.00	500.00	19.00	562.00	500.00	19.00
1	Bottom	Cabinet with door A	Simple cabinet	562.00	500.00	19.00	562.00	500.00	19.00
NB	DESIGNATION	DESIGNATION_OWNER	DESIGNATION_OWNER_N-1	PART_LENGTH	PART_WIDTH	PART_THICKNESS	OVER_DIMENSION_LENGTH	OVER_DIMENSION_WIDTH	OVER_DIMENSION_THICKNESS

# Configuration

# **Optimized Group configuration**

This new version comes with an improved **Group** configuration, allowing users working in group configuration to manage the configuration files more safely, including through cfg file **locking**.

#### New configuration files

In order to facilitate working with configuration files, three new cfg files have been created. These cfg files contain settings that were previously found in the **topzwoo.cfg** file. They are located in the same folder as the **topzwoo.cfg** file, in the **Group/V6X** folder.

<u>Note</u>: The **autoasm.cfg** file can now be stored in the **Group/V6X** folder like any other **TopSolid'Wood** configuration file.

The three new configuration files are the following:

- Edge-Laminate.cfg: This file groups together all the settings done in the Tools > Options > TopSolid'Wood
   Configuration > Edge/Laminate section. Edge or laminate codifications created on the fly in the Panel, Edge shape and Laminate shape functions are also saved in this file.
- Material-Overdim.cfg: All settings made in the Tools > Options > TopSolid'Wood Configuration > Matters overvaluations definition section are saved in this file.
- **Propag.cfg**: Propagations created from the **Tools** > **Options** > **TopSolid'Wood Configuration** > **Propagations configuration** section are saved in this file.

<u>Note</u>: All other settings available in the **Tools** > **Options** > **TopSolid'Wood Configuration** section are always saved in the **topzwoo.cfg** file and in the **autoasm.cfg** file for automatic assemblies.

#### Functioning of new configuration files

These three new configuration files and the **autoasm.cfg** file are automatically locked when a change is made from **TopSolid**. This prevents a user from unintentionally overwriting settings configured by another user.

A .lck file is automatically created when the .cfg file is locked. The image below indicates that the three new configuration files are locked.



<u>Note</u>: The **topzwoo.cfg** file is never locked by **TopSolid**. However, **Windows** safety features can be defined on the file to prevent some users from making changes to it.

These files are locked in the following cases:

#### - Edge-Laminate.cfg

The configuration file for edges and laminates is locked when a user clicks the **Edge/Laminate** line in the **Tools** > **Options** > **TopSolid'Wood Configuration** section. This prevents two users from creating different codifications at the same time.

This file is also locked when codification is created in the **Panel**, **Edge shape** and **Laminate shape** functions.

For example, in the **Panel** function, a user **A** creates a new codification by double-clicking a line of the **Edge type - code** column, and then clicking the **Add** button.

		140		000000		historie -	Defeition	N	
Edg	jes	Codification F	doet.me-code	Length	Reginning cid time	Endicithme	Standard	1	
	1	Counceion C	None N	49.8cm	Covining	Caverna	TopSolidWood	× \\	
M	2		None	18.0cm	Covered	Covered	Туре		
1	3		None	49.8cm	Covering	Covering	Thin edge	~	
2	4		None	18.0cm	Covered	Covered	Variant		11-
							Flatedge	~	12-53
							Version:		The second
VE	dges	are similar					01	~	19.00
VC	uts en	e similar					Code		1000
	>>						op 2	*	
Lan	ninete	shanes					Attributes		and the second se
MONT 1	N	Codification	Material		Coating	Thickness	Moterial		
	1	s	Laminate			0.1cm	Pvcu	*	
	2	6	Laminate			0.1cm	Coating		- 10 M
							Mat white paint	~	
							Codification		
ADD	0 000	FICATIONS					ED-TH2-WHITE		
V L	amina	tes are similar					ADD Initia	lize	
			04	in the	Cancel Init the list		OK		Cancel

**Note**: A new **Init the list** button is available, allowing you to update the list of codifications for edges or laminates, and thus add a codification recently added by another user.

When the codification is created, the file is locked, which prevents another user from creating the same codification or overwriting a change. On the user **B**'s workstation, the following message is displayed in the alpha bar if he attempts to add or modify a codification when the file is locked.

The edge and laminate file definition is already in use by : A on User A

-- -- -- -- -- -- 0 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19

If the **Edge-Laminate.cfg** file is locked, the settings available in the **Tools** > **Options** > **TopSolid'Wood Configuration** > **Edge/Laminate** section are grayed out and the above message is displayed in the alpha bar.

		Applica	tion configuration				×
Configuration	Coordinate system						
TopSolid'Wood configuration     Attributes     Configurators     Bill of material	Sawingup coordinate system		Machining cod	ordinate system		Draft coordinate system	
Multi-drillings     Assemblies     Propagation     Propagation	Naming axis order						
Automatic assembly Linitalization Edge/Laminate	Axis 1 X Y Ax	e 2 X+	Axis 3	Ye w Axig 4	Y+ ~~	Axis 5 2- 9	Axis 6
Matters avervaluations definition	Codification						
- Ag Uran - Q Files - V Troiso - M Acchising	All odges and laminates must have on	e codification					
Cutting off	Edge configuration Laminate configurati	on Associations	Automatic codification of e	daes			
🕀 🕎 Display		0-48-4				Contraction and the	
E Ocument		Codification	Edde Mbe - code	Edge matter and coat	. Over dimensi_	Calibration overvaluat	
T Picking		CP-EPI-CO	Flat edge chamter - ep 1	pvcu-matwhite paint	0 cm	0cm	
E Colors		CP-EP2-CE_	Flat edge - ep 2	pvc u - cedar	0cm	0.05cm	
Shortcut key	and any second performance	ED-TH2-WH	Flat edge - ep 2 Flat edge - ep 2	pvc u - mat yellow paint pvc u - mat white paint	Ocm Ocm	0cm 0cm	and the service
i T come	the proof of the second s			G			
Prodefined index      Table      Correct Control Contro Control Control Contro Control Control Control Control Control Co	Add		Deleto		Print	Copy to clip	board
Geometrical tolerance     T Predefined properties     Define multiple parts     Multiple parts	Set edges modifiable in assembly						

The file is unlocked when the user exits the **Tools** > **Options window** or finishes creating the codifications from the **Panel**, **Edge shape** and **Laminate shape** functions.

#### Material-Overdim.cfg

The file containing the **matter overvaluations** is locked only when a user defines matter overvaluations from the **Tools** > **Options** > **TopSolid'Wood Configuration** > **Matters overvaluations definition** section. The settings available are disabled on the other workstations and a message indicating that the file is locked by another user is displayed in the alpha bar.

The matters overvaluations file definition is already in use by : A on User A

#### - Propag.cfg

This file is locked only when a user creates, modifies or deletes an existing propagation from the **Tools** > **Options** > **TopSolid'Wood Configuration** > **Propagations configuration** section.

When the **Propag.cfg** file is locked, all settings available in the **Tools** > **Options** > **TopSolid'Wood Configuration** > **Automatic assembly**, **Propagations configuration**, **Propagation**, **Assemblies** and **Multi-drillings** sections are grayed out. As for the two previous files, a message indicating that the file is locked is displayed in the alpha bar.

The propagations file definition is already in use by : A on User A

#### - Autoasm.cfg

This file is locked only when the user goes to the **Tools** > **Options** > **TopSolid'Wood Configuration** > **Automatic assembly** section. As for previous files, a message indicating that the file is locked is displayed in the alpha bar.

The automatic assembly file definition is already in use by : A on Utilisateur A

The Material-Overdim.cfg, Propag.cfg and autoasm.cfg files are unlocked when the user exits the Tools > Options window.

# Dialog box resizing

You can now resize certain dialog boxes. Like in Windows, you just have to click and drag one edge of the dialog box to adjust its size.

You can resize the windows of the following functions:

- File > Export project
- File > Import project
- File > Links
- Parameter > Edit list
- Assembly > Include standard
- Tools > Text > Several lines
- Tools > Note
- Attribute > Properties > Properties editor

**Note**: The window size adjustment is kept for the full **TopSolid** session, i.e. as long as the software is left open. When the software is closed, the windows go back to default size.

# Multi-criteria filters

#### **New operators**

When you create a multi-criteria filter, you can now use the following operators in the **Tools** > **Options** > **Multi** criteria filter section:

- Does not start with
- Does not contain
- Ends with
- Does not end with

#### **Selection through filters**

Filter selection can now be used in the **Define multiple parts** function.

- Launch the **Wood > Define > Define multiple parts** function.
- Select parts in the graphics area or click Main assembly.
- To choose the filter selection, enable the **By filters** option, or click in the **Filters** tab directly.

		1	Define multip	ole parts				×
Selection	n type ally	O By criteria	() E	By properties		• E	ly filters	
Criteria	Properties	Filters						
Filters >	Machining Machining Nesting							~
Flat B	ateriai level OM	At top level	<ul> <li>Multi levi</li> </ul>	əl	EXPA	ND ALL	COLLAP	SE ALL
		Us	e right click to se	ect / configure	9			
Index	Qua	Designation	Reference	Material	Thic	Length	Width	Туре
ASSE		a george and a geo	- Januariana	* <b>#</b> ** #* (			an gran	-
<							c	>
U Selecter	d part		ОК	Cancel			ontigure	>>

### Management of materials

#### **Obsolete materials**

Some materials can now be defined as **obsolete** so that they are no longer displayed when defining a part. This saves the material from being deleted so that parts of old projects don't lose their material.

• Launch the Attribute > Material > Manage materials function.

A new **Obsolete** option is available in the **Data** tab of the **Material editor**.

	Material	editor	×
User     Group     Glasses     hardwood     metal     Metal     mineral     miseral     paint     paintings     parquet     Personal Woods     Hardwoods     Panels     Plaster     plaster     plaster     plaster     SketchUp     Colours     Textures     steel     Stonework     Add category     Delete category     Export     Import	Apple tree Beech european Maple core	Image data       Texture param       Finishing         Data       Advanced data         Name       :Beech european         Designation       :Beech european         Reference       :         Supplier       :         Density (kg/dm3): [0.75         Image data       Obsolete         Image data       Texture param         Finishing       Obsolete         Save       Delete	
	and the second s		

**Note**: Once the **Obsolete** box is checked, the material can no longer be used on a part. However, the old parts that used this material will always be linked to it, and the material will always appear in the bill of material.

 The Beech european material has been defined as obsolete. When a part is defined, this material is no longer visible in the list of materials.

**<u>Note</u>**: If all materials of a category are defined as obsolete, then the category is not displayed in the list of materials.

		Р	art defi	nition		×
Numbering h	elp					
Designation	>					~
Reference	>					~
Supplier	>					~
Processing	>					~
Part category	>					~
Drawing Bill	ofmateria	Va	lorisation	Attributes	Properties	Part types
Cutting-up Ste	ock Mat	erial	Coating	Finishing	Machining	Description
Panels						~
Apple tree Maple core		4.9	- A.			
		K	Circ		nad	

# Drafting

#### Views

#### Non-dynamic projections

When creating an auxiliary view, if the **File** > **Properties** > **Projection parameters** > **Dynamic projection mode** option is cleared, TopSolid now displays the view frame without contents. The view is only calculated when positioned.

**Note**: This mode is useful if, for example, the size of the set to be projected is large and requires long calculation times when generating the view.

#### Formatted view title

You can now retrieve information in a view title. For files with multiple drawings, this makes it possible to display in a section view title the number of the page where the reference view appears.

In this example, the sections views are created on other pages than the main view.



- Edit a view using the **Modify element** function.
- In the **Title** field, enter the syntax to retrieve the desired properties.

Here is a list of properties that can be retrieved in the title, as well as the syntax to be used. The properties must be entered between the < and > characters.

Property	Purpose		
title	View title		
scale	View scale		
drawingscale	Drawing scale		
paper	Drawing format (A4, A3, etc.)		
folio	View drawing's folio number		
folio nb	Retrieves the drawing's folio number in 2/4 format		
depth	Cross section/section depth		
sectionname	Cross section polygon name		
nl	Skips a line between two properties		

**<u>Note</u>**: To retrieve a property of the reference view, add **refview.** before the property.

In this example, the **section name**, the **reference view folio number** and the **reference view title** will be retrieved in the title of a section view. The properties to be used are in the following order: **<sectionname> <refview.folio|nb> <refview.title>**.

View modification	×
Coordinate system Exceptions Section view	
Scaling factor	_
CLocal scale factor definition	
Adsolute     Belative to drawing	
	ь.
Keep faces and edg=s color	
Title	
Title at top	
◯ Title at bottom	
◯ Free position	
O No title	
Scaling factor in title	
Title : Section : <sectionname> <nl> Ref view on page : <refview.folio nb> <nl> Ref view title : <refview.title>.</refview.title></nl></refview.folio nb></nl></sectionname>	-
Title text height: 8.00mm	_
Botation angle (0.000*	-1
Section	
Name of section line: A	
Set view upright when not aligned	
Show hatching	
Cut curves	
OK Cancel	

<u>Note</u>: For further clarity, plain text has been added (Section:, Ref view on page: and Ref view title:) and a line is skipped after each property using **<nl>**.

#### TopSolid 2016

#### Here is the final result in the drafting document:



# **Elevation view**

It is now possible to straighten section views or sections in order to create elevation views.

• Create a main view.

Again using the kitchen example, a section is created to better view one of the walls.



- Launch the **Full section** function and create the **cutting curve**.
- Before positioning the view, adjust Alignment = NO.
- The new **Set the view upright** option appears.

**<u>Note</u>**: This option is used to force the view orientation to be vertical. It is only available in **Alignment = NO** mode.

#### Set the view upright = NO



#### Set the view upright = YES



The Set the view upright option can be modified after the view is created.

- Launch the **Modify** function.
- Select the view to make upright.
- Check/Uncheck the Set view upright when not aligned option.

Coordinate system	Exceptions	Section view	
Scaling factor	or definition	سی س	~
Rotation angle: 0.00 Section Name of section lir	00* ne:[A		10 <mark></mark>

# Interfaces

### SketchUp import

A new mode for importing **SketchUp** files has been developed. You can now import objects in polyhedral mode that are smaller in size than if exact shapes were generated during import. The **SketchUp** files with the **.skp** extension are most frequently used to import decorative objects into **TopSolid**.

- Launch the File > Open function.
- Choose the .**skp** file you want to import, then click **OK** to confirm.
- Two import mode are available. Select Import polyhedrons. MPORT SHAPES MPORT POLYHEDRONS
- The following options are available: OK Unit= Automatic v Import free curves= NO+ Import textures= NO+
  - **Unit**: Used to choose the export unit. By default, this option is set to **Automatic**, i.e. the file will be imported in the unit with which it was created.
  - **Import free curves**: Used to import the curves of the .skp file. If the option is set to **No**, only volumes will be imported.
  - Import textures: Used to retrieve the textures applied on the faces.
- Click **OK** to confirm and to end the function.

Here is an example of a file that was imported in polyhedral mode.

<u>Note</u>: Once the file is imported, the resulting shapes are stored in the **Polyhedral shapes** set.

In this example, the polyhedral file size is more than seven times smaller than the file containing the exact shapes.

Plant (polyhedrons).top	652 Ko
Plant (shapes).top	5 025 Ko
Plant.skp	4 370 Ko



- An ellipse can now be selected as the guide curve in the **Shape** > **Pipe** function.
- In the **Wood** > **User machining** function, a new **Centered tool = YES/NO** option enables the centered tool path of the machining to be exported.
- The new configuration word D\_DRIVER\_HOUSING\_DEFAULT\_NAME "default\_name" lets you define the default name of a driver block. This line must be added to the top.cfg file. If this configuration word is available, you will not be asked to enter the name of the driver block.
- The **Include standard** and **Include sub assembly/part** functions are now available in neutral mode by clicking in the screen background.
- The Make visible function, which is available in the same menu as the Sibility function, only displays the invisible elements. You then need to click the elements you want to make visible.
- The current coordinate system is now shown in green in the symbolic tree.
- In a **distribution**, a component can now be inserted in gaps (for example, inserting a pole between two distributed partitions).
- The new **Automatic for all** button is available when including a component with driver parameters. This automatically connects all the component's driver parameters to the corresponding parameters in the assembly.
- In the **Automatic** mode of the **Wood** > **Constrained block** function, when a face with non-parallel edges is selected, a **constrained block with non-parallel faces** is automatically created.

# What's new in TopSolid'SheetMetal v6.17



This section describes the new features in the version 6.17 of TopSolid'Punch and TopSolid'Cut.

# New features common to TopSolid'Punch and TopSolid'Cut

#### Sequence visualization

The access to this functionality is now faster.

From the toolbar, you just need to activate the icon's drop-down menu in order for sequences (displayed in green) to regenerate more quickly without interrupting the current LIP macro. The view filter on a given tool remains active.

 Image: File
 Manage: Edit
 Punching: Attachs
 Loader/Unloader
 Miscellaneous
 Tools
 Nesting: Cutting: Autre
 Window
 Help

 Image: File
 Image: File

### **Production date**

In this new version, the production date has been removed from the update dialog box.

When opening the Task Manager from a Punch part document, the part data is checked to see whether it matches that of the Task Manager.

If there are differences, a dialog box appears allowing you to select the data fields you want to update.

Tasks fields to update	×				
Machine					
Matter + Matter Type + Thickness					
Reference					
Designation					
OF					
Customer					
Plane					
Set					
Turret					
Rotation					
OK Cancel					

# Task Manager at startup by default

This new version lets you start the application with a Task Manager document by default. This can be configured under **Tools** > **Options**.

Application configuration					
Configuration 2 User information 3 Colors 4 Chotrout Key	General Number of digits				
Shotch Key	For lengths 2				
Posting     Simulation     Post processor	For thicks 2				
u-∽r lasks manager ⊡-∰ Display options	For areas 3	Lettering function File path to find definiton character file [C:\Missler\V615\local\french\lettering.top Browse Default value of text height[10.00mm			
	For angles 3	[-1 corresponds at the value of text height of current document]			
	For overflankings 3	Defaut saving paths           String to add to PCH part name              Before After         for machine         trumpf3000_sheet V			
	For mass 2	✓ Save Part path in the same folder as TOP file without opening dialog box ☐ For nestings with only one kind of parts, put the same name as part document			
	Fortimes  3	Kind of document at start software			
	For speeds 3	Part Document     Nest Document     I asks manager			
	For moneys 2	Additionnal criterion for search in PCH files function           Image: Comparison of Set         Image: Command         Image:			
		OK Cancel			

The option you choose is kept throughout the session, until you change it.

# Column sizes in Task Manager

This new version makes it possible to preserve any manual changes made to the column sizes of the various Task Manager's lists.

These changes are saved throughout the TopSolid session.

This new feature, which can be configured under **Tools** > **Options**, is also available in the favorite part and nesting dialog boxes.

Configuration	General					
9 User information						
	Nesting type					
A Shortcut key	O Matrix O Rectangular Nesting O Complex nesting					
- A General						
H Importation	Order name management					
Punch	Index orders name (#X=)					
🗄 📲 Cut	Start indexing at the first use of order name					
🗄 🦉 Nesting	Index the name during modification of the order					
🗄 📲 Simulation						
- 🛒 Post processor	Format the index number of digits					
🛓 😽 Tasks manager	Order name personalisation					
General	Augilable Informations					
Dperator cards	#M : Matter					
🗄 📲 Display options	#N : Matter Type #U : Date					
	#T : Thickness					
	#0:0F					
	HD: Dane					
	#B : Reference					
	#D : Designation					
	#F: Family					
	#G : Cut gaz					
	v					
	Always concatenate matter and thickness to the name of the order					
	( only if order counter is not used )					
	Renum nest order always enable					
	Add created nestings at the top of the list					
	Minimize the size of emty columns in dialog box					
	Parts return to use					
	Parts quantities changes					
	Ignore parts quantities changes 🔘 Take care of parts quantities changes 💿 Ask confirmation					
	Activate Automatic numbering of designations of matters to use					
	Last delivered wimber 0					
	Uptional Lonstant string before number					
	OK Canad					

### Task Manager recovery

The backup files of the Task Manager are regularly backed up when the manager is saved. These backup files can be found in the *\$PUNCHDATA\Archives\_TMan* folder. By default, there are three files with the following extension: *#B=-1.bak*, *#B=-2.bak* and *#B=-3.bak*. They are replaced every time the manager is saved.

If a problem occurs when saving or reading the manager, a new system allows you to prevent the saving of backup files so that you don't overwrite a valid backup file with wrong data.

In addition, when the Task Manager dialog box is open, the system will launch the data recovery procedure using the backup files.

# Part imports from nesting document

A drop-down menu has been added to the nesting document to help you import parts without having to create an "empty part" document.



## Free line

In the **Miscellaneous** > **Free contours** > **Free line** command, a button has been added to freeze the axes horizontally or vertically.

Enter the first point of the line				
VERTICAL/HORIZONTAL=	YES 🗫	Enter the second poir	nt of the line	

Pressing the z or Z key (not case-sensitive) changes the status of the button.

# Custom marking of parts in the nesting

In this version, you can customize the marking of parts that are created directly during a nesting. The data to be marked can be configured under **Tools** > **Options**.

	Application configuration	
Configuration User information Colors Shortcut key General General	Parts/Wastes lettering Machine alpha3 v	
Punch     Cut     Cut     Cuter nesting     Cuter nesting     Complex nesting     Permachined sheets     Permachined sheets     Part/Vartes lattering     Statistic file     Manual nesting     Post processor     Display options	Pats lettering     Datas to mark on patt       ✓ Automatic Lettering of teh nesting patts     Available fields       Character fork/CLMissler/V617Vloca/trenct/vlettering.top     Browsee       Text height 10:00mm     Marking       Lettering position in the patt     Center Top       Or p Left     Center Flight       Bottom Left     Center Bottom       Wastes lettering     Automatic Lettering of the wastes       Character ford CLMissler/V617Vloca/tenglishus/lettering.top     Browsee	n Separator//
	Lettered Wastes classification Standard © Custom Automatic counter (use \$PUNCHDATA/WastesCustom counter) ✓ Use counter also for wastes that become from wastes, or wastes that becom from matter with designation OK Cancel	

By default, only the part reference is marked. It is now possible to mark the **Designation**, the **Plane**, the name of the **Set**, the name of the **Customer** and the **Order**.

You can also compose the information you want to mark and the order of data using the arrows.

You can use a separator character as a field delimiter.

If a data is unknown, it can be replaced by special characters.

#### Moving a cut rest line

A new function has been added to the Punch and Cut modules to move a cut rest line.

You can click on the horizontal or vertical line to move it dynamically.

If wastes were marked, a message will appear to indicate that the marking texts may be moved. If only one waste is marked, you will be asked whether you want to move the marking text.

NO YES Do you want to move cut rest marking text?

A lettering, which can be a waste marking, can now be moved dynamically, and it is represented by the oriented rectangle of the lettering box.

### Management of completion parts

This new version gives priority to waste cutting in order for completion parts to be placed only in the free space corresponding to priority parts.

Example of a nesting without completion parts (preset and recovered cutting waste):



Example of a nesting with completion parts (preset and recovered cutting waste):



If you don't want this behavior, you can add the following configuration word manually to the *topzpch.cfg* file in order to return to the behavior of older versions.

ZPCH\_NEST\_COMPLEX\_FAVOR\_CUTREST\_ON\_COMPLETIONPARTS 0

# Improved part coloring feature

From now on, a color is always assigned automatically to each of the nesting parts, even if coloring is not necessarily requested.

This function can be enabled/disabled in the nesting order.

Nes	ting order - Complex r	nesting	×
Mode Complete nesting	Recor	mpute nesting	
Parts Gaps Sheets Nesting General			
	Ref.	Q.	
	PièceA	14	Add
			Search
ŏ			Remove
ь.			•
Box dimensions			
× 400.00mm Y 500.00mm			

This gives each part a different color even if coloring was not requested during the nesting.

Automatic color assignment is not applied if the nesting order contains only one part or in the case of unit part nesting, which makes it possible to keep the color defined in the part document.

# **TopSolid'Punch**

# Punch sequence display

The punch sequences can be displayed immediately in nesting documents from the Task Manager. This feature can be enabled under **Tools** > **Options**.

	Application configuration	
Calors Class Consideration Class General Class General Class General Class Cla	Nerling         Colouring         Tasks manage         ✓ Colouring of nestings in automatic mode         Tasks manage         ✓ Display punch sequences during tasksmanager nestings         Colour of contours of no inclangular thesis at their creation         Colour of contours of no inclangular thesis at their creation         Option motiong         Colour of contours used for the optic mathing         Option motiong         Colour of the holes contours used for the optic mathing	

## Default marking tool selection

During automatic punch allocation, you can configure the default tool to be used for marking. This feature can be enabled under **Tools** > **Options**.

	Application configuration								
Configuration	Adjustments								
🔤 🖉 User information		1							
🗄 😍 Colors	Ignore and induce segments of length lower than [U-50mm Circles of diameters (separed by ; )]	4							
Shortcut key	Notching	1							
General	No creation of notch but allocate each line								
Importation	✓ Limit at box triangular notch using rectangular/square tool								
Puncn     Tolerancies	Tool name to use by default for notches decomposed into lines None v for trumpf3000_sheet								
	Lines Circles	1							
E 🔀 Automatic allocation	Allocate Maxi teal with 5 00eeee Maxi teal with 5 00eeee								
Adjustments     Lools order	None ● All All except on box max. Up with blong								
- 30 Turret mount	Repeats search	1							
- 🏹 Machining associated to the nest	Lines search Grids search Circles search Special tools multicurves search								
🗊 🍒 Cut	Minimum number of holes for lines 3 for grids 4 for circles 3								
🗄 🍓 Nesting	Altern the start point of parallel lines								
E Best processor	Recognization of TopSolid repetitions								
Tasks manager	Other searches	1							
Display options	Macros search No search v Keep macros in data base								
- Gequences	Special tools on external lines Special tools on external arcs Special tools on internal circles								
Shearing	Search machinings of type line to line • Mini. number of lines to put in relation 2 • Gap maxi. between 2 lines 20.00mm								
Turret	Trapp door evacuation	1							
	The second secon								
	Utumpicouo_sneet  Parts Dimensions : X mini 100.00mm Y mini 100.00mm Dut Stop if trapp too small								
	Search parameters	4							
	Maximum diameter of a punched arc 1.00mm Maximum diameter of a punched full circle 50.00mm More than this value, the full circle will be nibbled with Trapp door 🗸								
	Maximum overflanking for notching[6.00mm Maximum overflanking for lines for trumpf3000_sheet V								
	Machining an obligg with a circular tool and a rectangular tool for Immor@000_sheet	i.							
	The state and billing and the second summer Herbert and the second	í.							
	Mark toxis and receiving Mark opened curves Mark tool Tolerada	4							
	External contours Pointeau								
	On corners Sharp corners only     Distributed     Distributed     Punch marker     tach tool     marker     tach tool								
	Dangarous wastes detection molecte								
	Datageness values detection after the automatic affectation in punch document     refendage pour films     refendage pour films     refendage pour films								
	Automatic dividau of not alloated contours after the automatic affectation in sunch document	1							
	Taken all construct a subject constants and the duration of plantary participation and construct a subject and the duration of the durati								
·	DK Cancel	Ĩ							
		1							

When manual lettering is created using the **Miscellaneous** > **Free contours** > **Lettering** function, you can choose the tool to be used for lettering in the case of several tools.

The types of tools included are Rollerball, Rolling shear and Punch marker.

# Copy of macro's free geometries

When an internal macro is modified, free geometries can be added to it.

When the macro is automatically recognized, the free geometries and their machinings are copied into the machining document.

For example, in order for the ellipse to be fully machined, it may be useful to add a free geometry rectangle to punch the inside of the rectangle.

During automatic allocation, the free geometries are automatically created when the macro is recognized.



**Note**: In the machining document, the free geometries automatically created through macro recognition will be automatically removed (during update) when there is no more tool left on this free geometry.

# Creation of special tools from TopSolid 7

It is possible to create special tools in **TopSolid 7** that will be recognized in **TopSolid'Punch**.

#### **Methodology**

• From **TopSolid 7**, create a new unfolding document.



- Create a new 2D sketch.
  - Draw the curve of the tool.
  - In the sketch, create the center point of the tool (make sure this point is **internal**).
  - Optionally, draw a curve of the rectangle type that will be used as the tool's safety zone.
- Confirm the sketch.
- Save and check-in the document.

Example of a keyhole tool:

Cutific apréciaux Tennare*	The second	and the lost	Carton	× KOL
	Lange 1 18			A Contra spectrum S I I' to a S Martin a Contra spectrum
				A coloradores to a coloradores
RĮD				1.1
all a	<b>0</b>	ď		2
	4			
18 302		<b>^</b>		
			Expanse 1	
			Copy CH+C Selection	
	a second s	a a contract of the second of the second	Attitutes Delete	100 P
		esto	and the second s	
			Verr Along Romal N Esquitor 1	1
₽ <b>→</b> ,			Styles     Vess from Top     Others	

- Create the special tool in **TopSolid'Punch**:
  - Use the Manage > Special tools creation function.

EXPLORER V7 EXPLORER CURRENT V7 SPECIAL TOOL DOCUMENT Select a CLOSED CURVE, a GROUP or a COMPONENT from the original document

- Click the V7 Explorer button and select the project and the document for the tool.

TOPSOLID 7	EXPLORER
Project:	
Ga 8 Pio de 6A.0 Ga gil A.2 Ga Serure.A.1	$\overline{\cdot}$
Pirter: Define filter	Material: Not defined
Clear filter Choose project Projects Libraries	V Thickness:
Destination directory	
C:\Dossier_Patricia\ANNEE 2015\Démonstrations\M Import sub directories Reproduce project tree	ls_Italie\DépliésV7
OK	Cancel

- In the current document, select the curve, and then the center point of the tool.



- Specify the tool name.
- Select the segments to be used as supports for automatic allocation (all in this example).
- Select **NONE** for the safety zone.
- Specify the tool-specific data.

	Tools caracter	istics : SPECIAL	
0710481007 Crochet1 Crochet2 Elypse essai10 essai11 Fraisage Keyhole MULTI R8T13 Multiple OS_Multiple OS_Multiple	Definition Tool name Manufacturer reference File Linked tools	Keyhole	
profil standard R5T8	Prefered stations	✓ Valid tool ✓ Enable for auto allocati Standard on fixed static On Multitool	ion
Modify	Tool text	With Cut wave AutoDev	Orientation definition (") Modulo (") Symetric X
Copy params Close and update			Punch Conditions

- Case of multi-curve tools:
  - After choosing the document in the Project tree, click the **MULTI CURVES special tool** button.

<u>Note</u>: If no direct link exists with **TopSolid 7**, make an export of the special tool with **TopSmi** conversion in **TopSolid 7** and use the **EXPLORER** button to retrieve the **TopSmi** document.

- Case of special tools with complex curves (example of an elliptical-shaped special tool created in **TopSolid 7**):
  - Create a new 2D sketch.
  - Convert the curve into arcs and lines using the **2D Sketch** > **Operations** > **Lines arcs conversion** function.
  - Create the center point of the tool.
  - Validate the sketch.
  - Save and check-in the document.



# Cut rest manual function for punching

A new function is available in the **Splitting** menu to create either vertical, horizontal, mixed or in L-shape cut rest.

The default settings can be adjusted under **Tools** > **Options**.

The Auto mode lets you create one or more wastes at a given distance from the last part.

0															
File	Manage	Edit	Nesting	Splitting	Attachs	Punching	Loader/U	Inloader	Miscellar	neous	Tools	Machin	ing display	Cutting	Cré
	💾 <b>-</b> 🖻	<mark>۶</mark>	- 🌔	i 0°	M	🌢 🚺	🖻   🗙	9	🧳 👿	2	9		<b>%</b>   🔟		K
8	8	200	8												
ок м	ode= Auto	¥ [	Distance fro	om last part	30	Cutti	ing rest Vert	ical then l	Horizontal	• М	achining	options	Mark waste	NO 🖘	

The Machining options button lets you configure:

- whether the waste will be evacuated with a machine stop or held by micro-junctions;
- the tool selection to punch the lines;
- whether the waste cutting must be done at the end or before all other machinings.

Example of L-shaped cut rest with micro-junctions:



# Flange

A new function in the **Punching** menu can be used to create flanges on geometries with Z elevation management. You need to define an **Oblong** tool and enable the **Quick flange** option in the punching conditions.

Flang 5x2 Ob 10x4.5	Definition Tool name	Flang 5x2	
	Punch conditions	X	<u>_</u>
Punch thickness (mm) Thickness (Ex Material acier alu electro galva Geperforeerd 10% v Tool particularities Normal Marking Noching Whisper tool	1:2-4:5) mm Empty=all Ok for thickness Active Material Activate All Materials Overlapping 0 mm (Length) 0 mm (Width) Work direction	Nibbling (mm)         Can nibble         Minimum step         Moximum step         0         Coet. of length         0         that define min. step         Minimum tool feed         Thickness (x)         1         Lubrification         0         Punches         0 = Normal         -1 = None	n Tool Text Macro Orientation definition ()
Ouick flange Bonding Multi shear	High standstill 0 Speed Code 0 Punch cadence Speeds Per Matter / Thickness Die allowance Per Matter / Thickness		Punch Conditions

#### Functioning

• Call up the function.

Several modes are available.

Mode=	Part of contour	~	Select not allocated first element to flang
3	Single element Full contour		
	Part of contour		1

Part of contour mode:

- Select the start segment (an arrow indicates the direction).
- If necessary, click the **INVERT DIRECTION** button to change the direction.
- If necessary, specify an offset, the Z start and the Z end values of the last flange segment.
- Select the end segment.



• You can rotate the view to put the result into perspective.



#### TopSolid 2016

The flange is displayed as a green thick line. The elevation is done progressively along the segments. Once the flange has been created, any changes to it will be made using the wrench.

#### **Modification**

It is possible to change the offset, the Z start and the Z end values of the flange. Depending on the segment selected, it is possible to force the Z end value of this segment.

OK INTERMEDIATE POINTS... Offset = 2 Z start value = 0 Z end value = 30 Selected segment Z end value FREE 4p

The elevation is done progressively along the segments. The Z end value of the first segment is free.

• To force a new value, click the **FREE** button. The displayed value corresponds to the Z value considering the progressiveness.

#### Addition of intermediate points

To vary the elevation before the end of a segment, an intermediate point can be added by cutting the segment in half.

• Use the wrench on the flange and click the **Intermediate points** button.

Two modes are available: Manual or Position value.

• Click the black geometry in front of the flange.

A new segment has been inserted and its Z end value can be modified like any other segment. The cutting point appears as a dotted line.

- To change the position of the point, use the wrench and click on the dotted line.
- If necessary, you can specify a specific value in relation to the beginning of the line.
- To delete the segment created, check the **Delete it** box.

## Mouse tool orientation

When the tool is shown dynamically on the mouse, its displayed orientation is now the angle of the mounting key on the turret + the definition orientation of the tool itself.

If the tool is not yet mounted or it is mounted on an indexing station, only the definition orientation is taken into account.

This is a fast way to determine if a geometry can be machined with the tool.

# Punch marker tool

The new **Punch Marker** feature has been added to the round tool management.

In the tool management, enabling the **Punch Marker** option allows you to define the step between each punch mark.

2	Too	ls - aries245.out	×
File ?			
Files	Туре	G	Glimpse
aries245	ROUN	ND 🔨	
trumpf24		Too	ols caracteristics : ROUND
tumpt50:	abcd123456789012 ^ Diamant Ebavurage 4 MarkR002 Pointeau Rond 10 Rond 11.8 Rond 11.8 Rond 11.5 Rond 12 Rond 13.5 Rond 13.5 Rond 14 Rond 15 Add Remo Modir C No Remov Copy pa Copy pa	Definition Tool name Manufacturer refer Diameter (mm) Thickness (Ex:1; erial er D'ALU D'ALU particularities rmal rking ching isper tool	Pointeau erence Aucun 0.2 Punch conditions 1.2-45) mm Empty-all Ok for thickness Active Material Activate All Materials Overlapping Overlapping Minimum tool feed Thickness (x) 1 Ubrification 0 Punches 0 - Normal - 1 - None
		IlerBall Iling Shear F nding Iti shear humng nch Marker	High standstill       High standstill       Speed Code       Punch cadence       Speeds Per Matter /       Thickness       Die allowance Per       Matter / Thickness       Ok

Its allocation is done directly on the geometry using the source with ROUND tool or tool machining on profile function.

#### Example:



# TopSolid'Cut

## Part groups

A new function is available in laser to define machining zones in order to limit overheating of very thick sheets. This function can be accessed from the icon bar in a **Nesting** document.



These groups are defined by the user.

The following three options are available: CREATE, MODIFY and REMOVE.

In order to make the selection easier, cutting sequences are hidden and parts can be colored as soon as the function is launched.

#### **Group creation**

Parts can be selected freely or using a filter on the names of the nesting parts. Existing groups can be hidden or visible.

Multiple selection is possible.

If only one part is selected, it can be included in the group or removed from the group if it already exists. If only one part is selected and this part belongs to a cutting path, all parts of this path will be included in the group (each part must have only one cutting instance).

When confirming the group by clicking OK, a group number is provided and, if there are several zones (shifts) in the part selection, a group number is created for each zone.

The parts that overlap several zones will be contained in a specific group.

To associate a part that overlaps several zones with a group of parts that only includes parts of a given zone, you need to use the **Modify a group** function.

### **Group modification**

A drop-down list allows you to select the group to be modified.

The other existing groups can be hidden or visible. When the other groups are visible, a part can be easily transferred from a group to another.

The **END** button allows you to quit the change function.

### **Group deletion**

A drop-down list allows you to select the group to be deleted. An option is also available to delete all groups.

# **Cutting strategies**

In this version, simulation allows path strategies to be taken into account for cutting.

#### **Functioning**

For cutting, the path is given by the links defined between cuttings in the part document and/or nesting document. These links are generated using the **Automatic cutting and linking** function or the **Linking path between cutting path** function. The links set a path between cuttings.

Cutting strategies involve creating or recreating automatically the links between cuttings or between parts during the simulation.

Consequently, no links must be defined between cuttings or parts. Keep only imposed links that cannot be modified by the simulation.

Imposed links are shown in blue. They are created by the user with the **Automatic cutting and linking** or **Linking path between cutting** function.

The links created automatically by the simulation are shown in black and depend on the required strategies. When changing the strategy, these links will be readapted.

You can make changes to a link generated by simulation (shown in black), but it will turn into a blue-colored imposed link which is no longer impacted by the simulation.

This can be configured under **Tools** > **Options**.

		Applica	tion configuration
Configuration User information Colors Shortcut key General Importation Umportation Unch Cut Nesting Simulation Trajectory Sort of tools	Adjusts Vizualizations Blank geometry Show the path Display turret dynamically Show the player dialog box	Applica Information None Minimum Full se cut strategies	tion configuration
Labels cards     Adjusts     Post processor     Tasks manager			

#### **Dialog box for cutting machines**

When the simulation is launched, the following dialog box appears.

Strategies for cutting			
Cutting Zone 1 Group n* 1 Group n* 2 Same strategy for all	the areas / groups		
● By part ○ By band ○ A	Alternate 🔘 Entire sheet		
Traject optimization Auto. (shortest distance)	OIn line Tol. 1.00mm		
Parts linking Start point high left	Start point high right		
Start point bottom left	Start point bottom right		
Traject on the part Start point high left	Start point high right		
Start point bottom left	Start point bottom right		
APPLY TO ALL ZO	NES / GROUPS		
Subroutines	Prototypes		
OK	Cancel		

A drop-down list containing the different groups defined in the zone is available at the top of the window, with a line for each zone.

The paths that match the selection in the graphics area are highlighted.

A different strategy can be applied to each of the groups.

The following strategies are available:

#### - Strategy by part

Definition of the desired path between each part. The path can be defined between the internal and external cuttings for each part, providing that no blue links were imposed in the part. The cuttings are performed in the internal order, and then in the external order part by part.

#### - Alternate strategy

Same concept as the strategy by part by skipping a part to prevent overheating in strong thicknesses.

#### - Strategy by horizontal or vertical bands



#### Entire sheet strategy

All part's internal contours are processed before external contours.

The **Apply to all zones/groups** button makes it possible to apply the same strategy to all zones/groups.

The **Subroutines** button lets you configure the way subroutines are generated.

Suboutines Config	×
Cutting None By part By part of nesting	
OK Cance	el

The **Prototypes** button allows you to graphically select the parts to be cut before the rest of the sheet.

Prototyp parts ×			
Parts of the format		Prototyp p	parts
5		Ref.	Q.
A prototyp part for each part	=>		
Strategy Add a Stop at the end of each part prototype Make all prototypes together and add a Stop at end			
ОК		Cancel	

The simulation player displays the **Strategies** button which allows you to subsequently modify the cutting strategy.



### Dialog box for punching/cutting combined machined

A new **Cut Strategies** tab lets you set the cutting strategies independently of the punching strategies.

			Choice	e of strategies
Pur - S	nch Strategies itrategies Internal punching Internal punches	Cut Strategies Options Proto	typ parts	
			Strategy : Entire sheet	
	lools	Strategies		
	C 5x5	Entire sheet		
	C 10x10	Entire sheet		
	D 20	Entire sheet	Traject on the sheet	Strategies loading
	APPLY TO A	ALL TOOLS OF ALL CATEGOR	IES MODIFY	Current strategy : DoubleLame_HG_
	Full punches and	d evacuations Strategies		Strategies loading

When clicking this tab, the dialog box changes, allowing you to define the cutting strategies.

Choice of strategies			
Punch Strategies Cut S	trategies Options	Prototyp parts	
Cutting	one 1		
Kind of strategy	pand  Alter	nate O Entire s	heet
Type alternated Al	ternate every 1	pièce(s)	
Traject optimization O Auto. (shortes	t distance)	O In line Tol. 1.00	mm
Traject on the sheet Start point high left	- Star	t point high right	
Start point bottom left	Star	t point bottom right	
	M		
APPLY TO ALL ZONES / GROUPS			
Reset	OK	Cance	el

#### Operation order dialog box after simulation

Further information has been added about the group number on the part and/or the band.



Example:



Once the simulation using cutting strategies is performed, you can see the black links that have been automatically created through the simulation process.

If a **waypoint** is created on one of these links, this link will turn blue and will be imposed.



# What's new in TopSolid'WoodCam v6.17



This section describes the new features in the version **6.17** of **TopSolid'WoodCam**.

# **Rails and pods**

### Distributed rails under the part

This feature allows you to change the number of rails to be used under a part according to its length.

To configure the distribution of rails, use the <sup>#+</sup> Rails and pods definition function. In the window, select a set of rails and pods, and then from the Distribution tab define the maximum number of rails for a given part length.

The distribution values are saved in the document because they are included in the rail and pod definition.



On the left, pods are positioned without rail distribution. On the right, pods are positioned with rail distribution.

## Parking positioning of rails and pods

The rails and pods that are arranged during a positioning now take account of the minimum distances between rails and between pods. These values can be defined in the document properties or in the configuration of the pod positioning function.





# Push the rails and pods to their stroke limits

The rails and pods can now be pushed independently of each other to their stroke limits by pressing the following keys:

#### Step-by-step mode:

- Left/Right Arrow
- Up/Down Arrow

#### Stop mode:

- Ctrl + Left/Right Arrow
- Ctrl + Left/Right Arrow

The minimum distances between rails and between pods are taken into account.

# Element selection for clamp positioning

The clamp positioning function enables the selection of a point or a face.



For example, if a face is selected, the guiding axis of the clamp will be positioned against the face.



### Automatic positioning of rails and pods in several areas

When a machine contains multiple sets of rails and pods and a part positioned on a stop encroaches on various areas, **TopSolid'WoodCam** automatically positions the rails and pods for the areas concerned.

The stops that are related to each of the rail and pod sets are defined in the **Stops** tab when you create the set **#**.

When the **Use sets linked to the stop** and **Automatic positioning of rails and pods** options are enabled in the **i document properties**, **TopSolid'WoodCam** positions all the rail and pod sets related to the stop being used.



### Fixed rails in a set

When the pods are positioned manually, it may be necessary to fix the position of rails. The **Fixed rail** function has been improved so that all the set's rails can be fixed.



# Vacuum blocks

#### Updated vacuum block positions

The new **Update blocks positions during operation update** option lets you update the block positions when **re-executing** <sup>10</sup> the operation (without editing the block positioning operation).

	Blocks po	ositions	- □ ×
Positions visualisation origin Machine		Blocks	
Update blocks positions du	ring operation update		
Designation	×	Y	
Vacuum block	307.5	330	
Vacuum block	605	312.5	
Vacuum block	930	355	
		OFFAIL	1.

# Processes

#### Through pocket parameter



When a part contains several apertures created through Boolean operations, **TopSolid'WoodCam** retrieves a set of curves for machining purposes. The disadvantage of this method is that all apertures are processed in the same way.

The pocket operations have the advantage of being processed independently of each other by the process assigned to them. In addition, these operations can be sorted using the **Operations sorting** CAD function, unlike apertures.

To dissociate through pockets from blind pockets, the new **Through pocket** parameter is available in the **keywords**. This Boolean parameter displays **1** if it is a through pocket or **0** if is it blind.

It is now possible to activate a contouring for a through pocket and a pocketing for a blind pocket.

# Partial hole

A new keyword is available in the drilling processes, making it possible to detect whether a hole is the result of two holes facing each other in CAD mode.



# Found tool

A new keyword lets you know whether a tool is compatible with an operation.

In the keywords of the **Tool chosen** > **A tool has been found** operation, this Boolean displays **1** if the tool has been found and **0** if not.



# **Analysis**

# Tool selection window display

By default, during automatic analysis, if no matching tool is found, the process execution is interrupted with an error message. To avoid this happening, the **Show dialog if tool request failed** option is now available.

This option is saved and can be configured for the document using the **File** > 1 **Properties** function or directly in the options of the **analysis of part operations** 1.

<u>Warning</u>: The analysis is no longer stopped if a tool is missing, so you have to check that all of the operations have been machined. In multi-machining mode, errors are recorded in the log file. Consequently, it is up to you to check whether operations are missing in there.

# **Multi-machining**

# Directory explorer

An **Explorer** icon has been added to choose each directory in the **Tools** > **Options** > **Routing configuration** > **Multi machining** section.

Multi machining Saving directories/subdirectories	
Machining files:	- 🚔
Iso files :	- 🛋
Slave parts :	- 🛋

# Nesting

# Abort the execution of nesting operations

The **Esc** key lets you stop the analysis of all the nesting operations.

# **Machinable part**

# Simple panel and multilayer panel

The panel entity and the multilayer panel are now processed as a single part during positioning and machining.

This makes it possible to machine the edges and laminates and the operations that affect several parts of a multilayer panel without being detected several times for one single entity.

The elements to be positioned can still be separated in the document properties. Consequently, the multilayer panel can be machined in multi-machining mode. Edges and laminates are represented in a simplified way on the machine.



# Mounted set

The mounted set uses the concept of machinable part. As a result, all of the CAD set's parts are considered as one part in **TopSolid'WoodCam**. This removes unreachable operations between two parts and thus avoids detecting several times operations that affect several parts and are unreachable.

# Ghost drillings

**<u>Reminder</u>**: In the CAD system, ghost drillings can be defined to lighten the 3D model.

When a part contains ghost drillings, it is automatically converted into a machinable part.

During conversion, the drillings are recreated (in real mode and not in ghost mode), allowing them to be normally machined.

The machinable part can be disabled using the following configuration word:

#### ZMI\_WOOD\_PART\_CREATE\_MACHINING\_SHAPE 0

In this case the machinable part is not created and **TopSolid'WoodCam** work as in the 6.16 version.

# **Panel positioning**

# Intermediate stock for panel positioning

When an incomplete panel (supports with edges, supports with laminates or simple supports) is positioned, **TopSolid'WoodCam** can use the enclosing shape of the positioned elements, or the panel's stock provided by TopSolid'Wood.

The type of panel or support(s) can now be selected in the document's **Properties > Part positioning** section.

Panel	
$\checkmark$ Choice of elements to take in account	
Elements to take into account > Supports and edges	•
Stock	
O Panel	Support(s)

Different options are available:

- Positioning of the incomplete panel: The panel's stock provided by TopSolid'Wood is used.
- Positioning of the panel and edges: The panel's stock provided by TopSolid'Wood or the enclosing shape of supports and edges is used.
- Positioning of the panel and laminates: The panel's stock provided by TopSolid'Wood or the enclosing shape of supports and laminates is used.
- Positioning of simple supports: If there is only one support, the panel's stock provided by TopSolid'Wood is used. Otherwise, the enclosing shape of supports is used.

This option also works if the machinable part is disabled by the configuration word.

# Set machining

#### Unite set shapes

The **Unite set shapes** option has been added to the document properties in order to activate the **Unite shapes** function by default when a set of parts is calibrated (but only if these parts were mounted in **Create all parts** mode).

Document properties	Routing	
🖶 🖓 TopSolidWood properties	Display operations dialog box	
- Units	Calibration	
🚽 📲 🖓 General	Enable calibrating at the beginning	
👜 🐌 General information	Single part Panel	
📲 🖉 User information	Tool choice	
Rendering options	Testana	
- 🗂 Visualisation options		
🗄 🛃 Coordinate system	Tool diameter : 16mm	
Tolerance reference	Operation type > Standard	~
Geometrical tolerance	Calibrating tupe	
		*
	Calibrating beginning > Top left	~
Constraint	Calibrating approach According to operation parameters	
T Text/note	g the second ing to operation parameters	· · ·
E Bouting configuration	✓ Unite set shapes	
Process		
🖬 Cylinders and features analysis	Stop at the end of the operation Stop number > Stop with vacuum	~
Part Diate		
Database	Aperture	
	Tool name :	
	Tool choice > Forced	~
Rails and pods	Test Second IC-	
	I ool diameter if forced :   16mm	
	Hole minimum diameter to machine : 32.1mm	

# Generation of orthogonal angles from real kinematics

Certain 5-axis machines only accept the programming of theoretical axes. For these machines whose rotation axes are not orthogonal, programming in **TopSolid'WoodCam** with the real kinematics is now possible as well as the generation of programs in an orthogonal coordinate system, while keeping the angular limits of the real kinematics. Available for Head/Head machines with a C-A configuration from the kit later than 1.15.41.

#### [POST PROCESSOR]

#LABEL = Woodwop Theorical angles on real kinematic #COMPANY = TopSolid'WoodCam #MACHINE TOOL = #NUMERICAL CONTROL = T1 #POST PROCESSOR NAME = TopPpWoodwopV135 #FILENAME PARAMETER = TopPpWoodwop#V=T1#I=13.cfg

[SPECIFIC] #Orthogonal\_Axis = YES

# **Ergonomics**

# Multi-machining

The multi-machining window has been modified in order to be able to read the sort criteria of parts correctly.

TopSolid 2016