

A large version of the TopSolid logo is centered on the page. It consists of the word "TopSolid" in a large, bold, italicized black font, with a red swoosh underline. Below the swoosh, the year "2016" is written in a smaller, grey, sans-serif font.

TopSolid v6.17
What's New

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Version 6.17 Rev.02

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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.

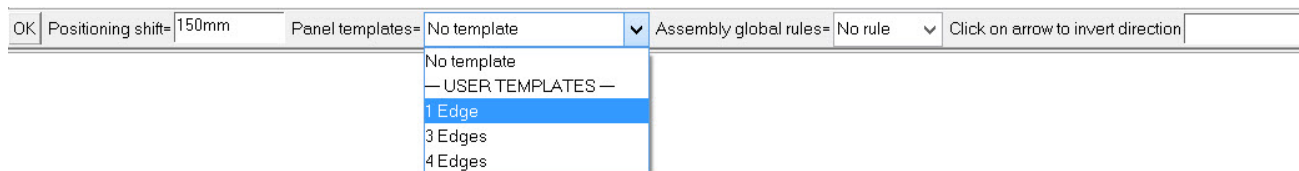
Note: The created panel will then be saved as a model. For example, if three edges are applied to this panel model, then three edges will also be applied to the part on which the model will be used. However, the model is for reference only. All the panel settings can be changed later on.

- 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.

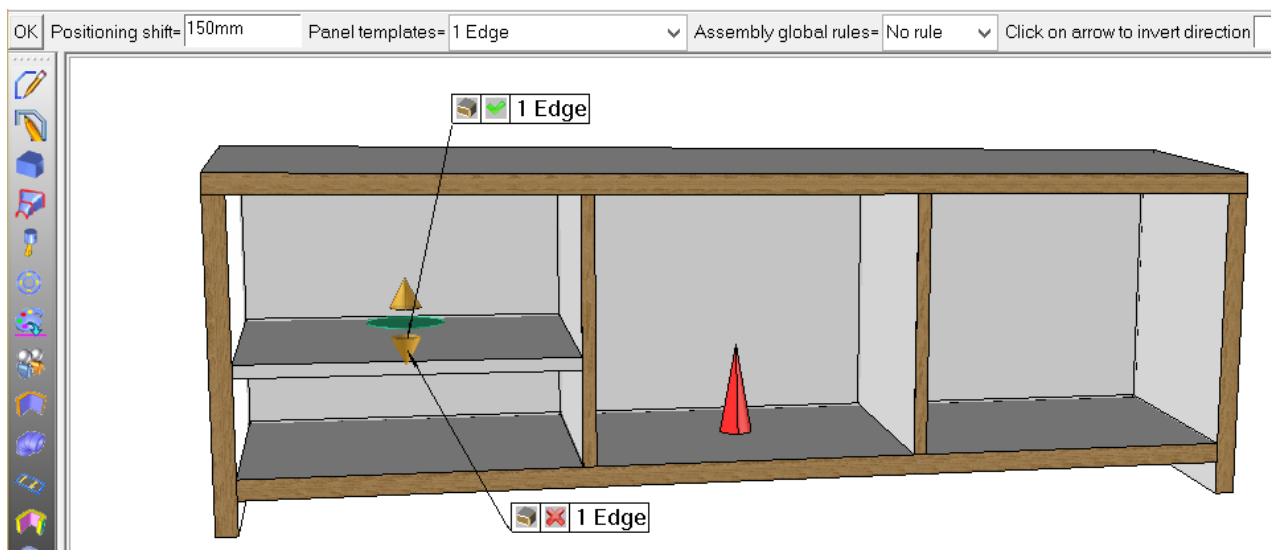


Once the panel model has been created, it can be used at the end of the **Constrained block** function.

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

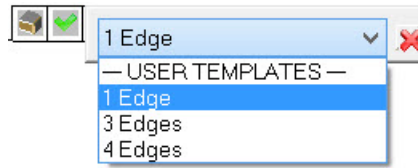


After selecting your panel model, two labels are displayed allowing you to select the reference face of the panel.



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:**

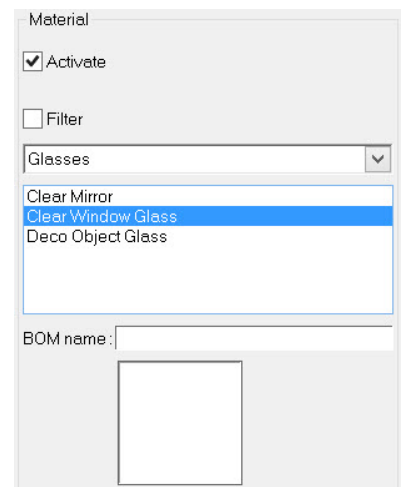
When this option is enabled, the created volume will be automatically placed on the level you specify.



- **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.



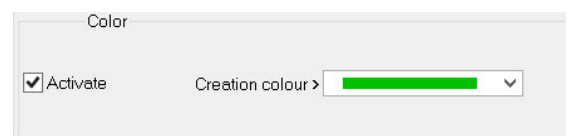
- **Transparency:**

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

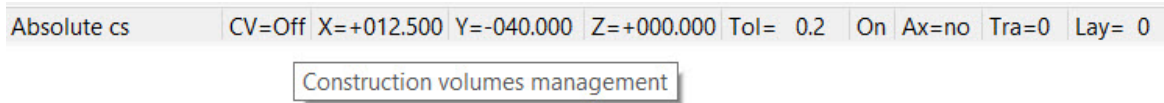


- **Color:**

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.

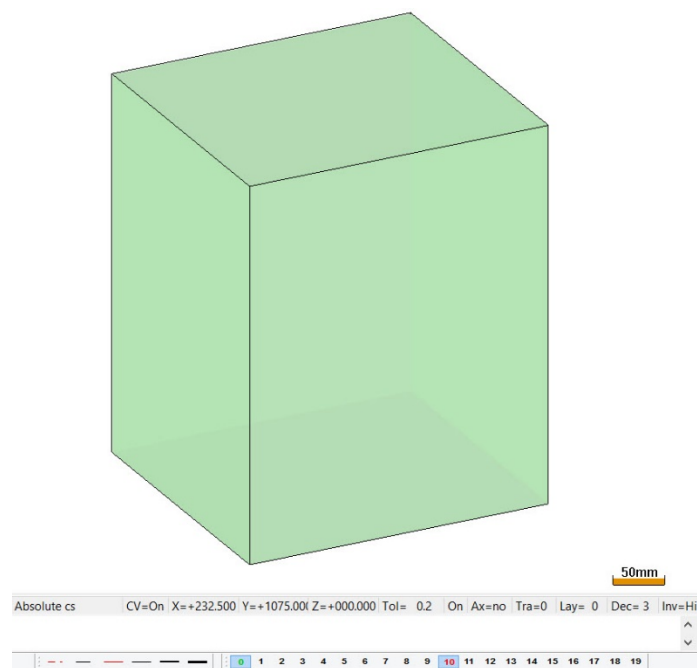


- Then create a shape.

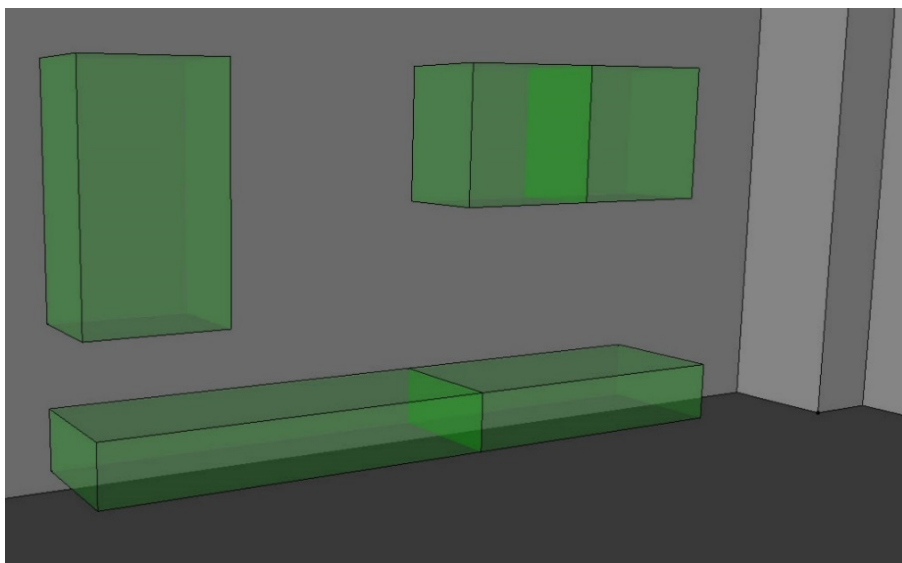
Note: Only the **Shape > Extruded** and **Shape > Block** functions can create **construction volumes**.

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

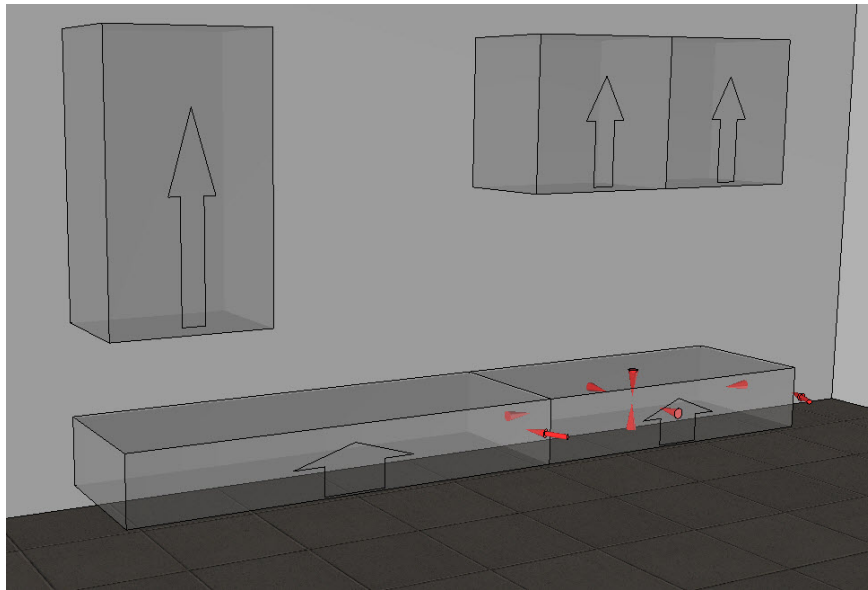
Here is an example of a **construction volume**. As defined in the **Tools > Options > Shape > Construction volumes** section, the extruded shape uses the **green color**, a **transparency of 7** and is created on **layer 10**.



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



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



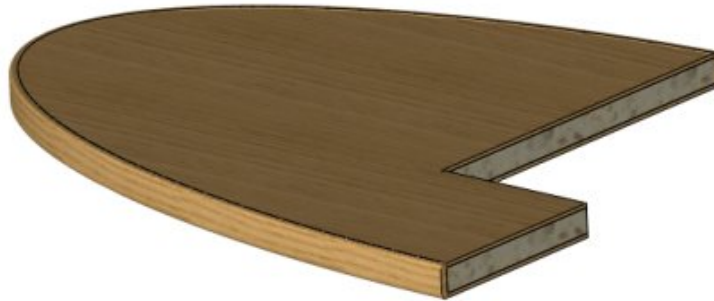
Here is the final result:



Panel

Elliptical panel

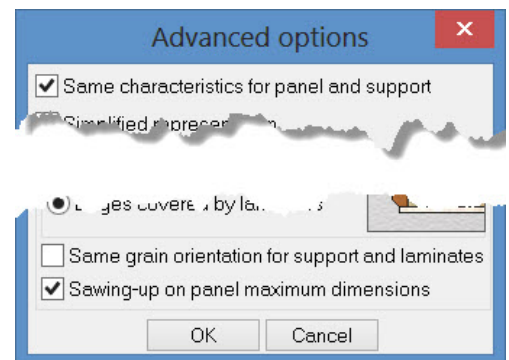
A panel entity can now be created on an elliptical shape.



Laminate sawing-up on panel's maximum dimensions

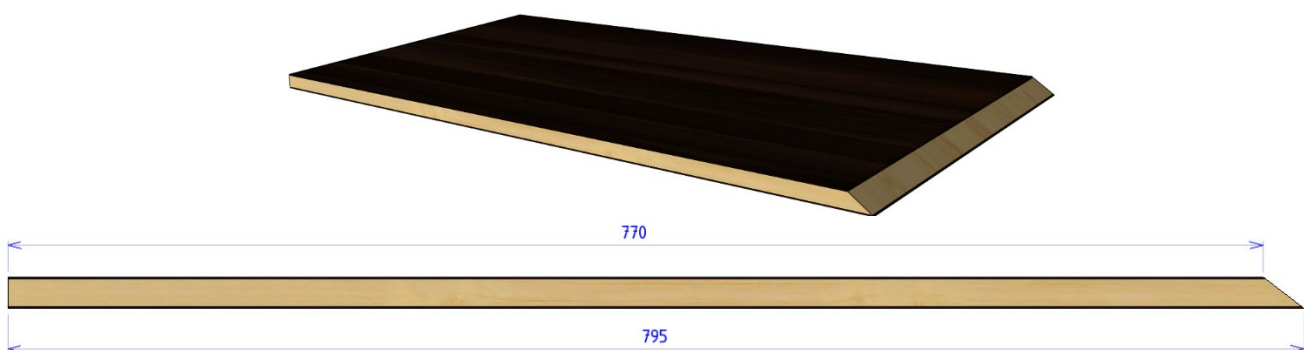
The panel's maximum dimensions can now be retrieved in the sawing-up of laminates. For a panel with a sloped edge, the panel's maximum length can be retrieved in the sawing-up of both laminates.

- 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.



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

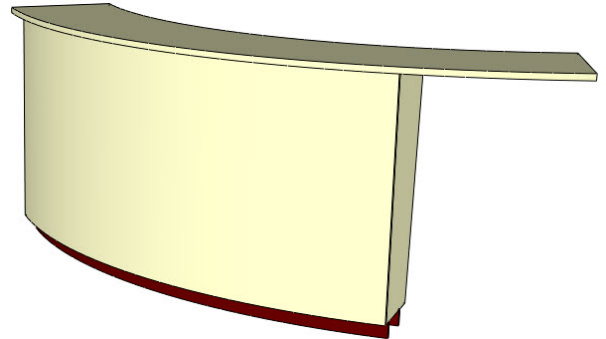
Bent part sawing-up

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.



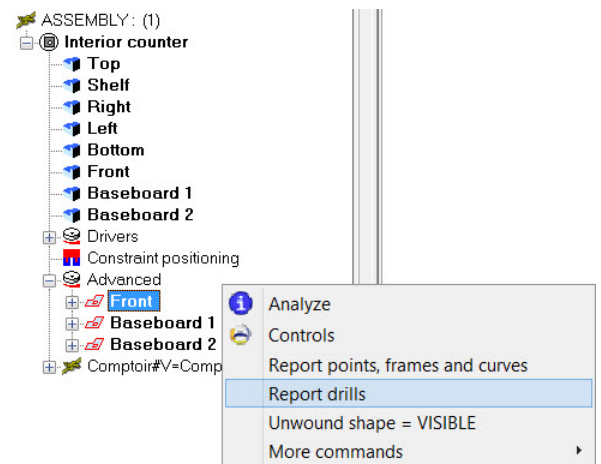
Note: The unwound part is included in the assembly with the component. It is automatically made invisible when inserted to prevent any errors.

- 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.



Note: 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**.

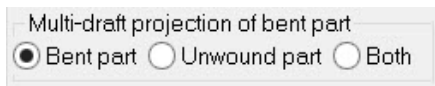
Cut/Machining export

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.

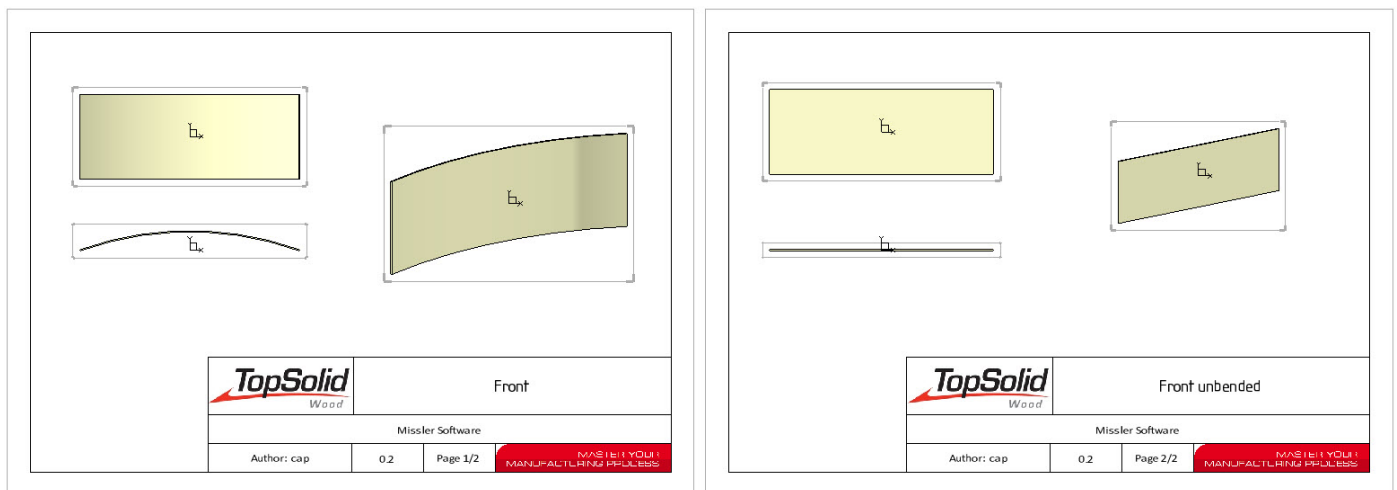


Note: As this setting has been made in **File > Properties**, it only applies to this document. To adjust the default value for all documents, define the projection mode of bent parts from the **Tools > Options > Drafting view > Initial values** section.

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.

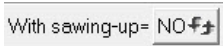


Note: The **Select axis automatically** and **Bent part** options must now be adjusted before you select the part to be defined.

A new **With sawing-up** button is available.



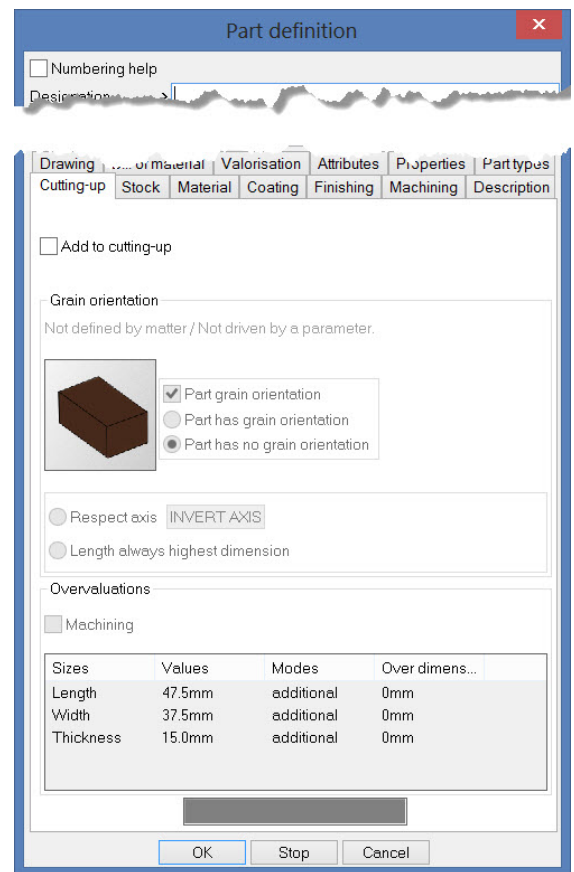
If this option is set to **Yes**, the function behaves the same way as in the previous version. The part sawing-up information will be displayed in the bill of material.



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.

Note: 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.

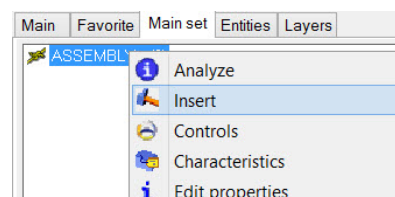
In the **Part definition** window, the **Add to cutting-up** box is cleared and the part sawing-up information is grayed out.



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.



Note: 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 over-dimension value.

Sizes	Values	Modes	Over dimens...
Length	1200.0mm	additional	OverL=2mm
Width	600.0mm	additional	OverW=2mm
Thickness	38.0mm	additional	0mm

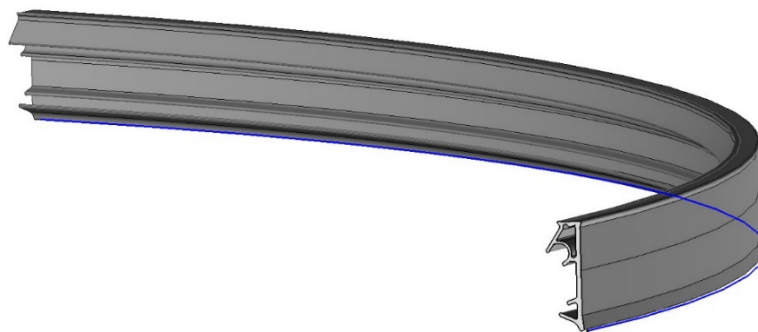
- Use the **Assembly > Define component > Define drivers** function to declare the over-dimension parameters as drivers.
- When including the component, the over-dimension values are required.





- Over-dimensions in **Fixed** mode:

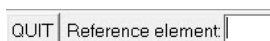
The **Fixed** mode is used to force the length, width or thickness of a part by specifying the desired dimension directly.

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).



- **Name** the parameter. Here, the parameter is named *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.

Name: Seal_Length

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

- 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.

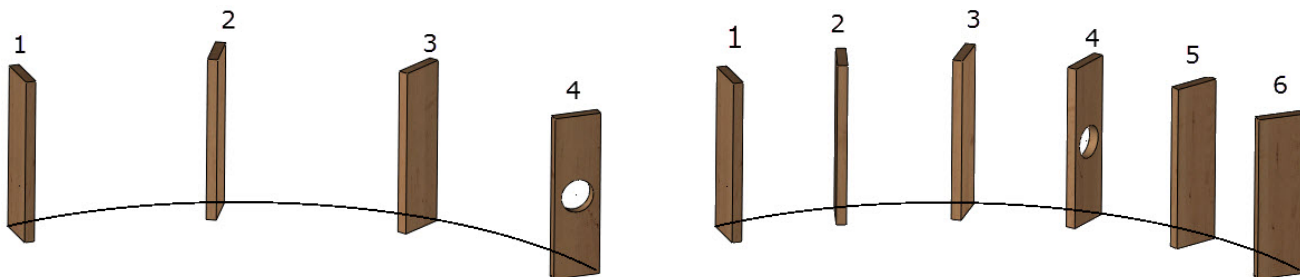


Repetition on curve: Give priority to ends

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.

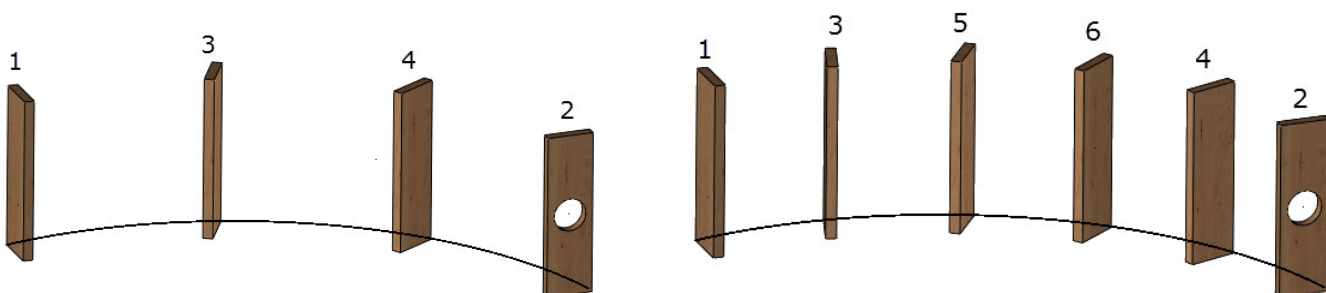
Give priority to ends= NO ↕

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

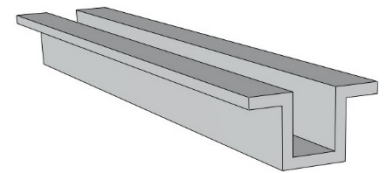
When a component layer is set to **Template attributes** or **Initial values of template** in the **Tools > Options > Component > Attributes** section, the corresponding layer in the assembly is automatically activated in order to see the component immediately when included.

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**.

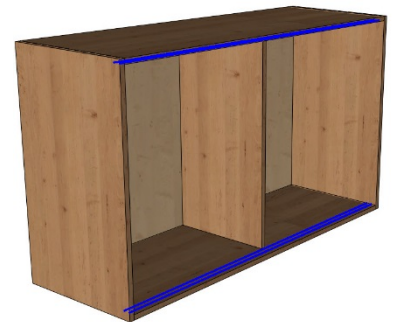


In this example, a rail component has been created. This component generates a groove and has two dimensional parameters to be specified when included (the **l** parameter being the length parameter required to create an extruded component).

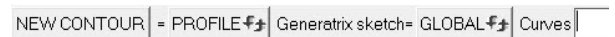


- In the assembly, launch the **Assembly > Include standard** function and select the extruded component. In our example, the rail is included in a cabinet in which sliding doors will be positioned.
- Specify the component's **driver parameter** values.
- Choose the insertion mode. In most cases, the 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:

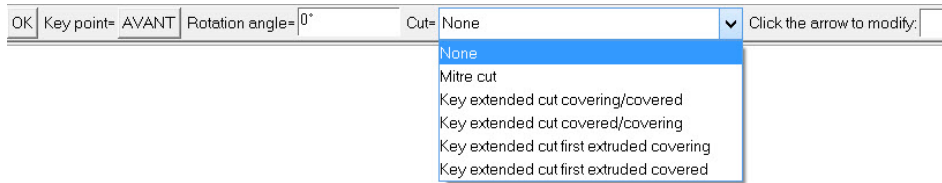


- **New contour:** This option allows you to create a new profile or sketch on which the extruded component will be inserted.
- **Generatrix sketch** selection mode: If the **Global** mode is selected, you can select the entire sketch by only clicking one of its segments. The **Local** mode allows you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

- 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. Mode= GLOBAL

- **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.

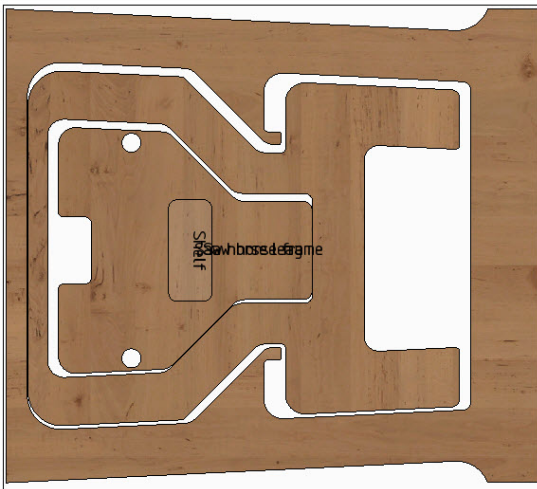


Nesting

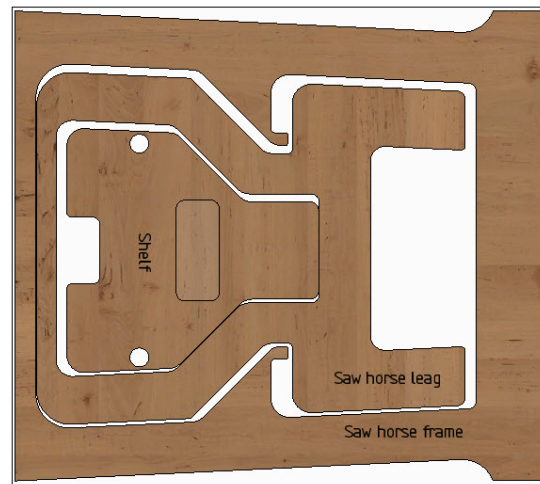
The position of label texts for parts and orientation symbols has been optimized. In previous versions, these texts were placed in the middle of the part's enclosing rectangle. Consequently, for the parts containing open areas or internal pockets, the text was placed outside the part, or worse, on another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



New method: Automatic 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.

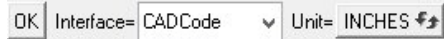


Wood machining

Machining export interfaces

2D½ DXF export

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



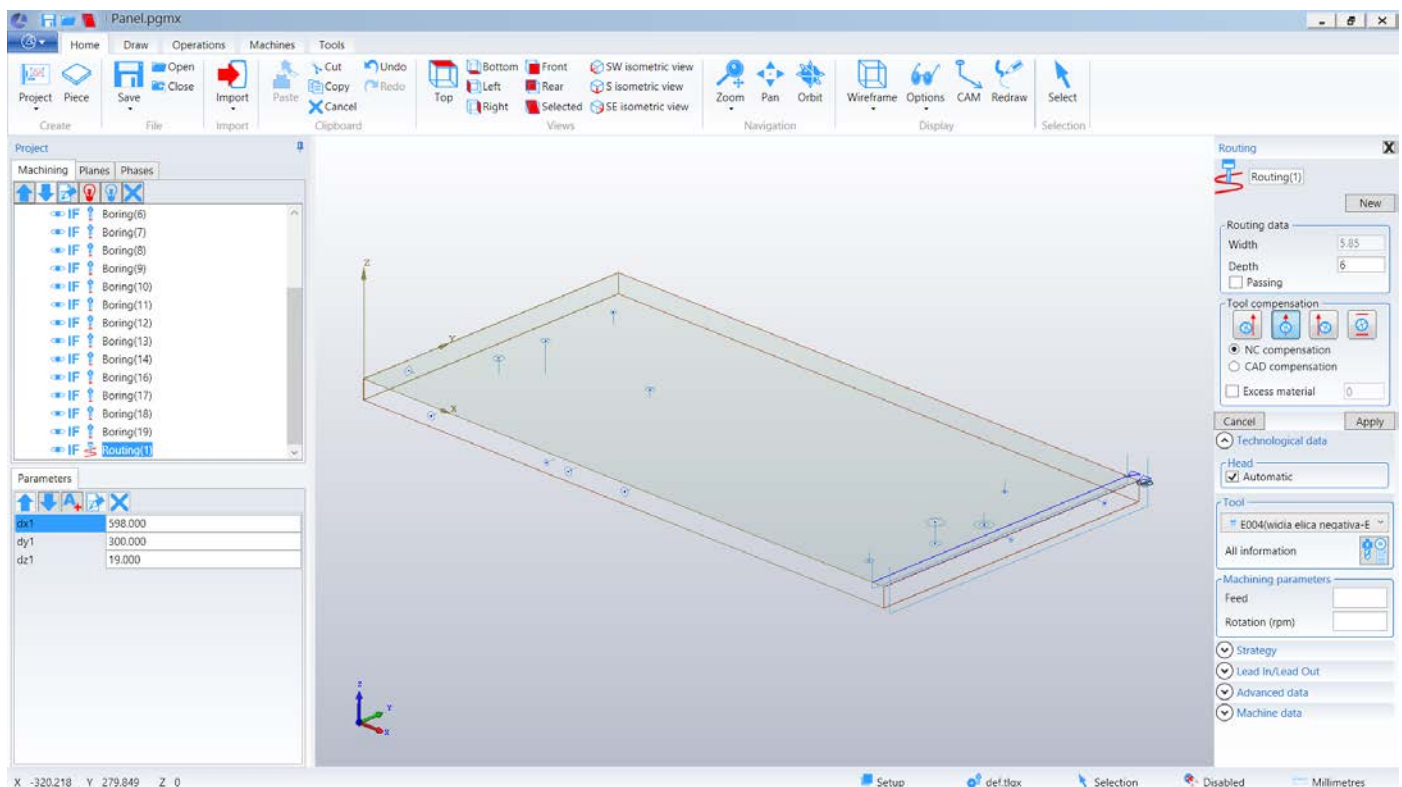
In the **TopSolid** configuration directory, the **cadcode.cfg** and **cadcode.bom** files are no longer used. Instead **TopSolid** adds two new configuration files whose names contain the export unit. There are now four configuration files for this interface in the **\$TOPCONFIG\dxf_wood** folder:

- 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...).

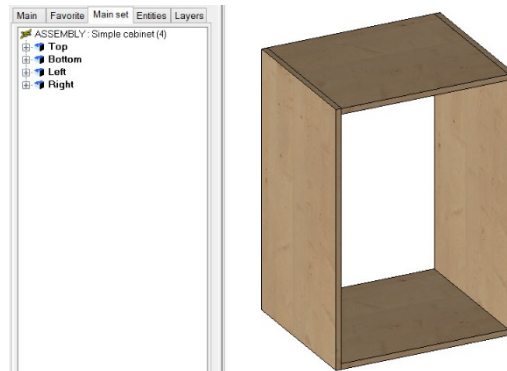


Bill of material

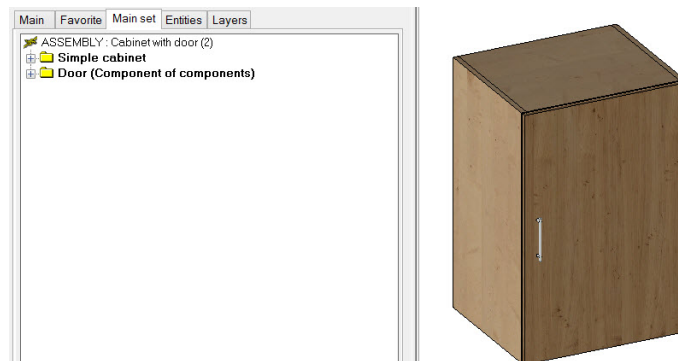
Recovery of parent information

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.

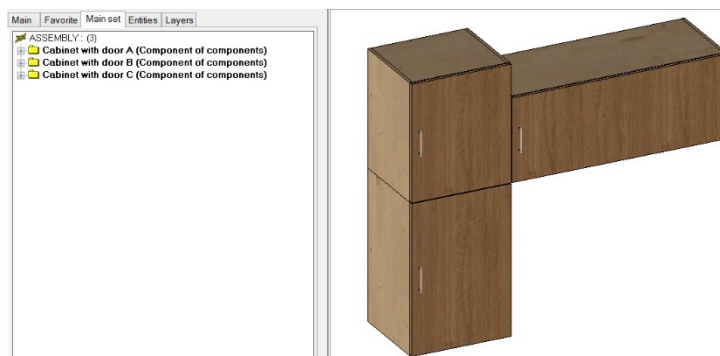
For example, a cabinet is created in a file.



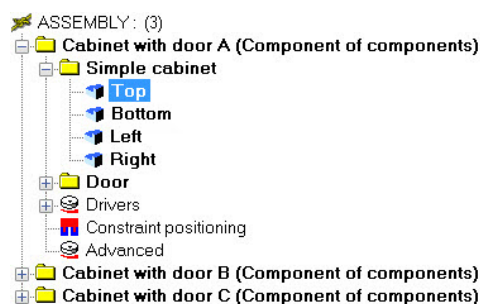
This cabinet is included in a new document with a door.



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**.



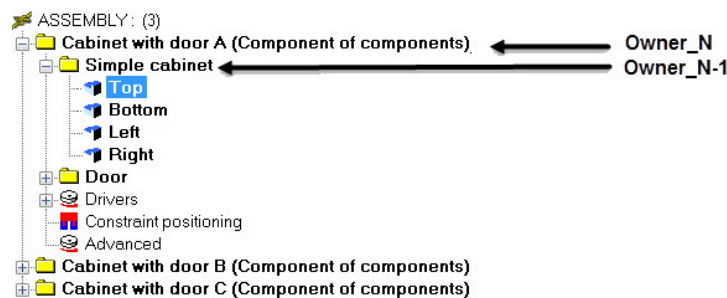
- Create a new bill of material file in .bom format.
- Open the file in with a text editor.
- Use the following BOM column to retrieve the designation of the **Cabinet with door A** assembly.

```

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	Top	Cabinet with door C	Simple cabinet	1262.00	500.00	19.00	1262.00	500.00	19.00
1	Top	Cabinet with door B	Simple cabinet	562.00	500.00	19.00	562.00	500.00	19.00
1	Top	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.

Note: 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:

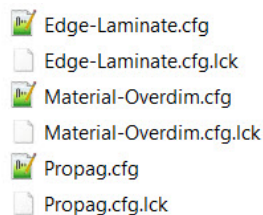
- **Edge-Laminate.cfg:** This file groups together all the settings done in the **Tools > Options > TopSolid'Wood Configuration > Edge/Laminate** section. Edge or laminate codifications created on the fly in the **Panel, Edge shape** and **Laminate shape** functions are also saved in this file.
- **Material-Overdim.cfg:**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

Note: 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.



Note: 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.

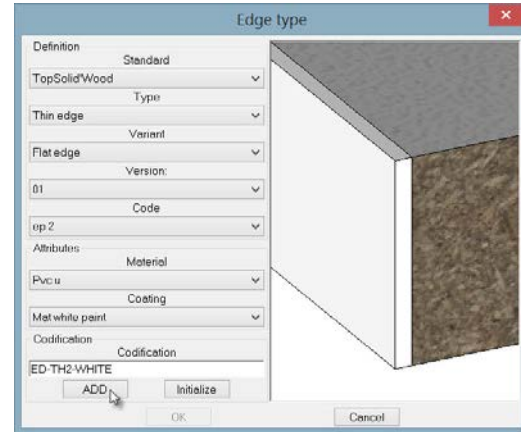
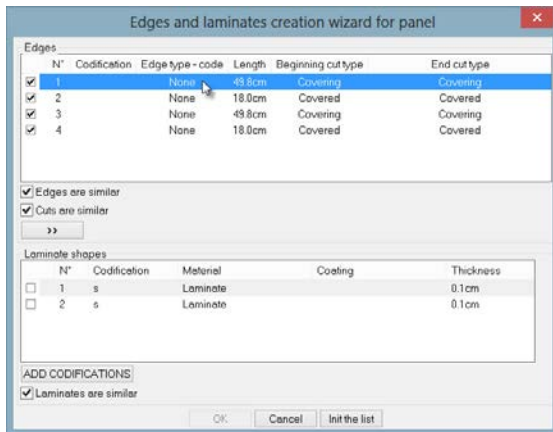
These files are locked in the following cases:

- **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.



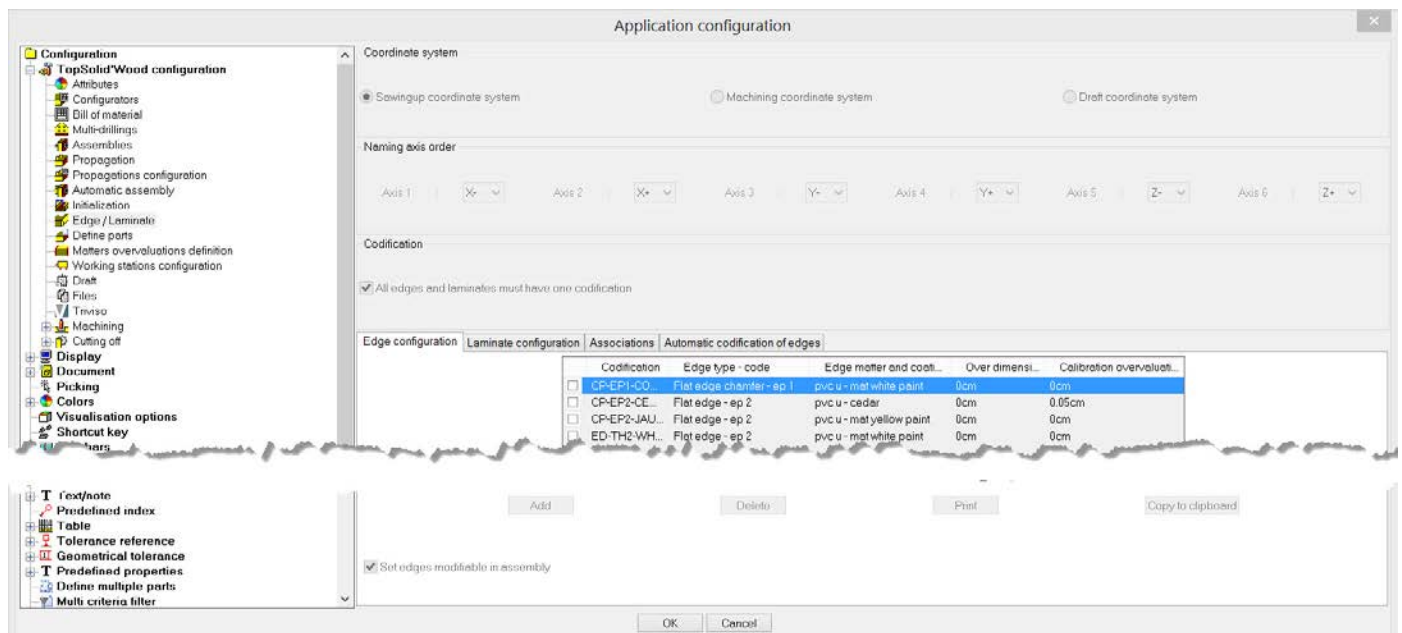
Note: A new **Init the list** button is available, allowing you to update the list of codifications for edges or laminates, and thus add a codification recently added by another user.

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



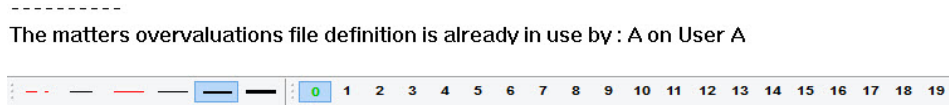
If the **Edge-Laminate.cfg** file is locked, the settings available in the **Tools > Options > TopSolid'Wood Configuration > Edge/Laminate** section are grayed out and the above message is displayed in the alpha bar.



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**

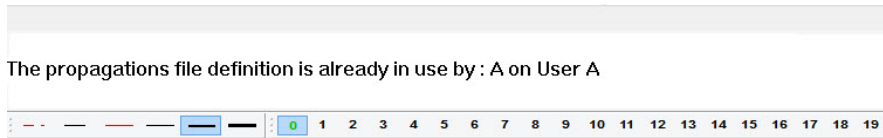
The file containing the **matter overvaluations** is locked only when a user defines matter overvaluations from the **Tools > Options > TopSolid'Wood Configuration > Matters overvaluations definition** section. The settings available are disabled on the other workstations and a message indicating that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



- **Propag.cfg**

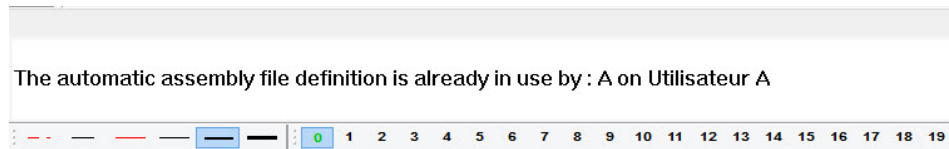
This file is locked only when a user creates, modifies or deletes an existing propagation from the **Tools > Options > TopSolid'Wood Configuration > Propagations configuration** section.

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.



- **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 **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

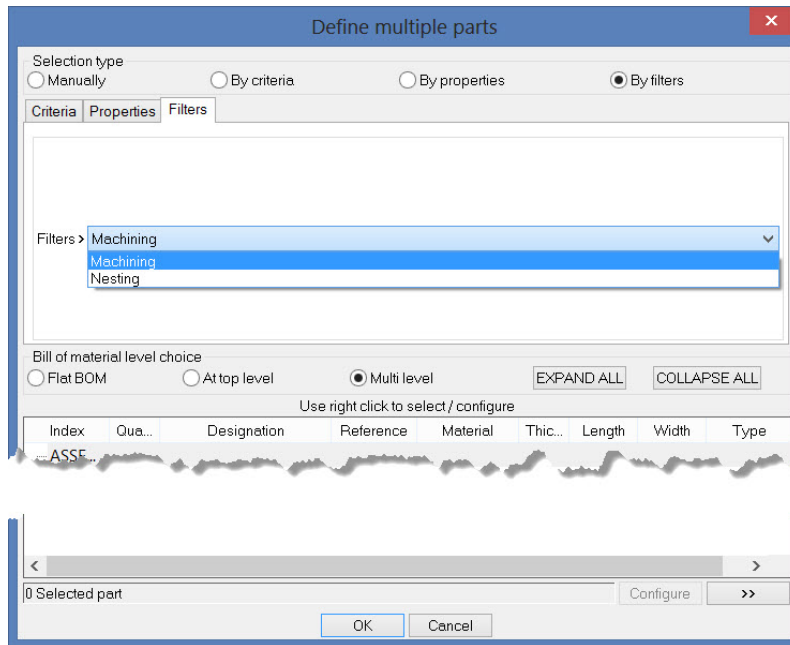
When you create a multi-criteria filter, you can now use the following operators in the **Tools > Options > Multi criteria filter** section:

- **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.



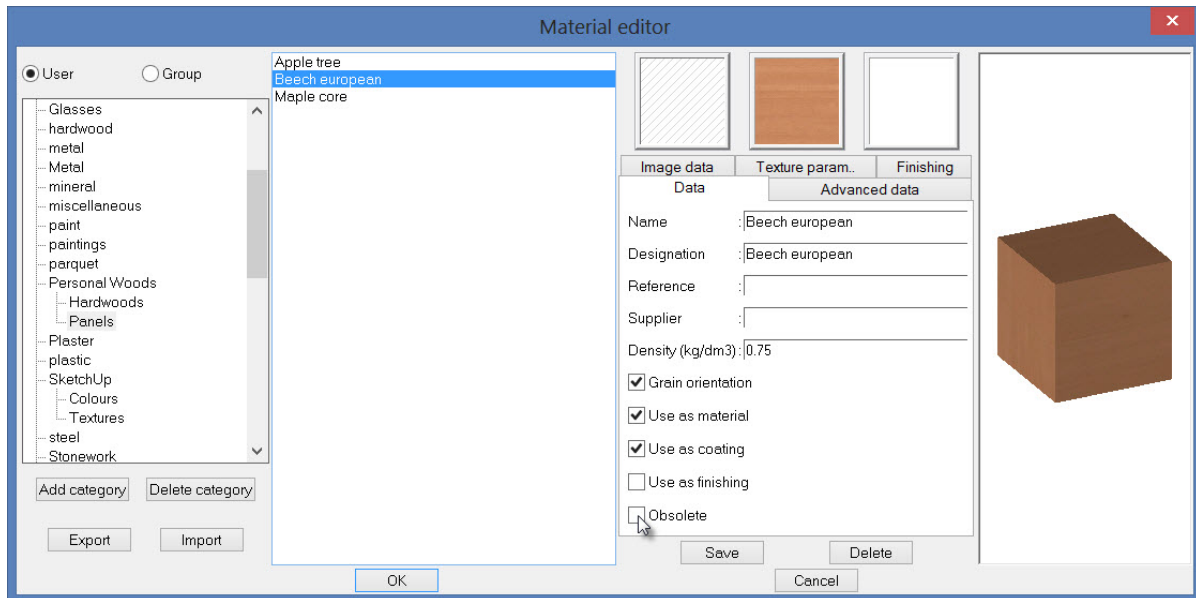
Management of materials

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.

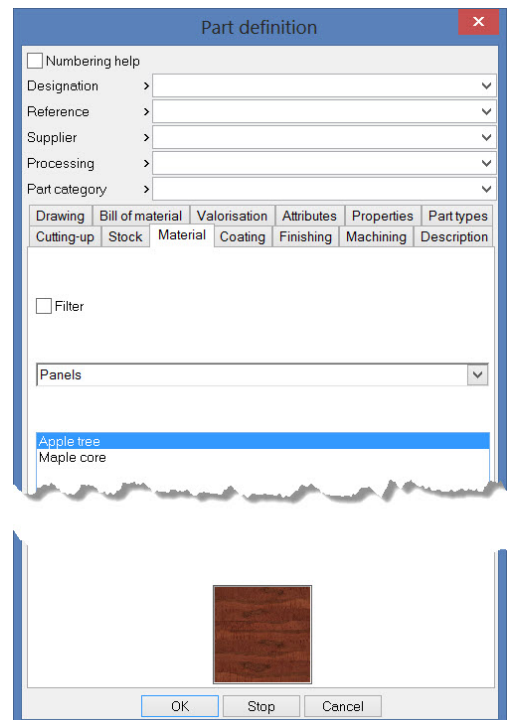
A new **Obsolete** option is available in the **Data** tab of the **Material editor**.



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.

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



Drafting

Views

Non-dynamic projections

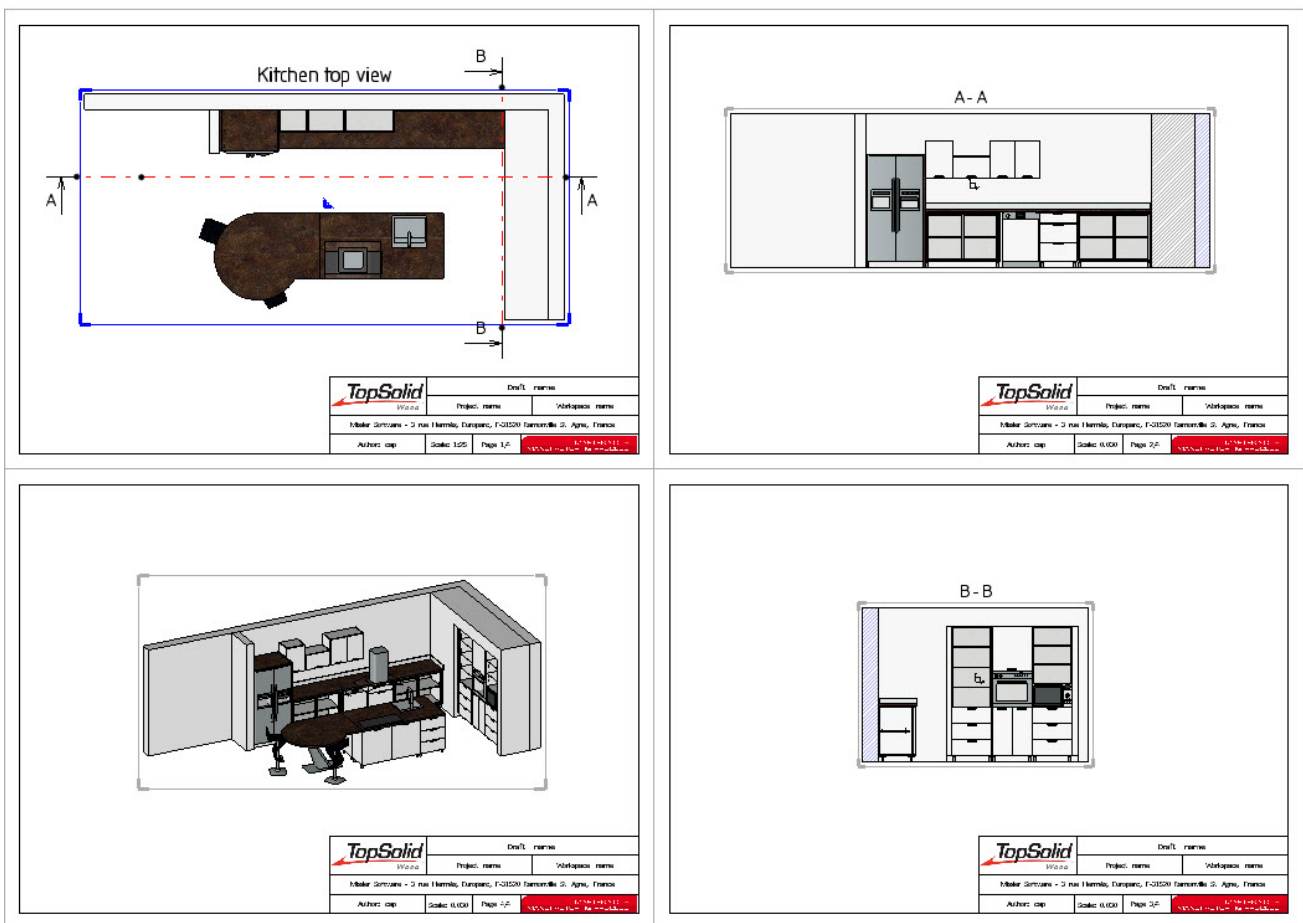
When creating an auxiliary view, if the **File > Properties > Projection parameters > Dynamic projection mode** option is cleared, TopSolid now displays the view frame without contents. The view is only calculated when positioned.

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.



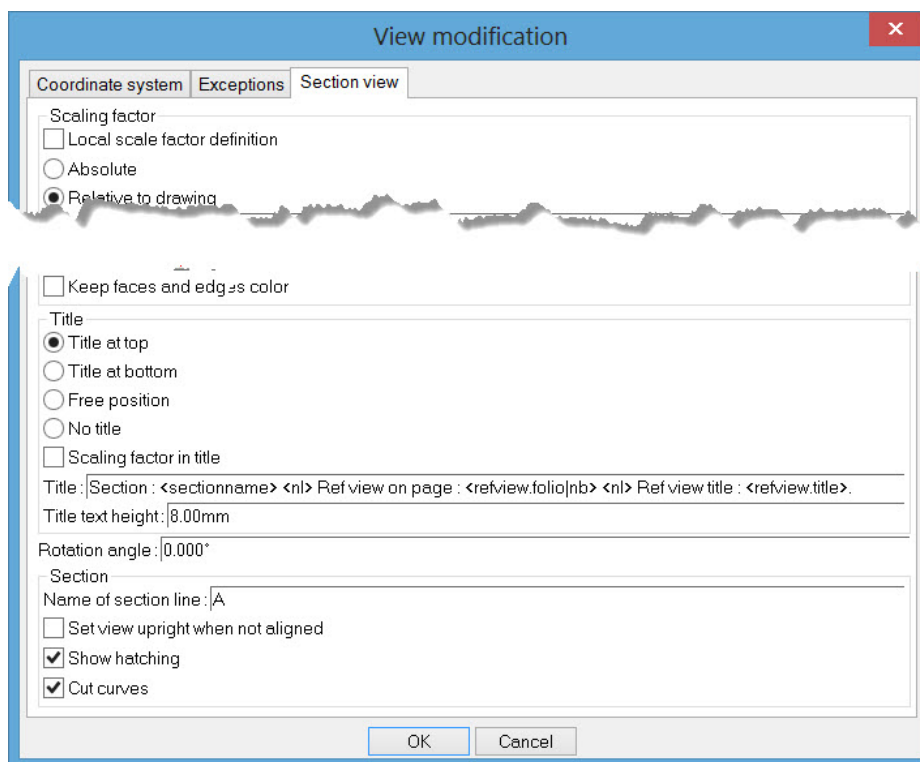
- Edit a view using the **Modify element** function.
- In the **Title** field, enter the syntax to retrieve the desired properties.

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

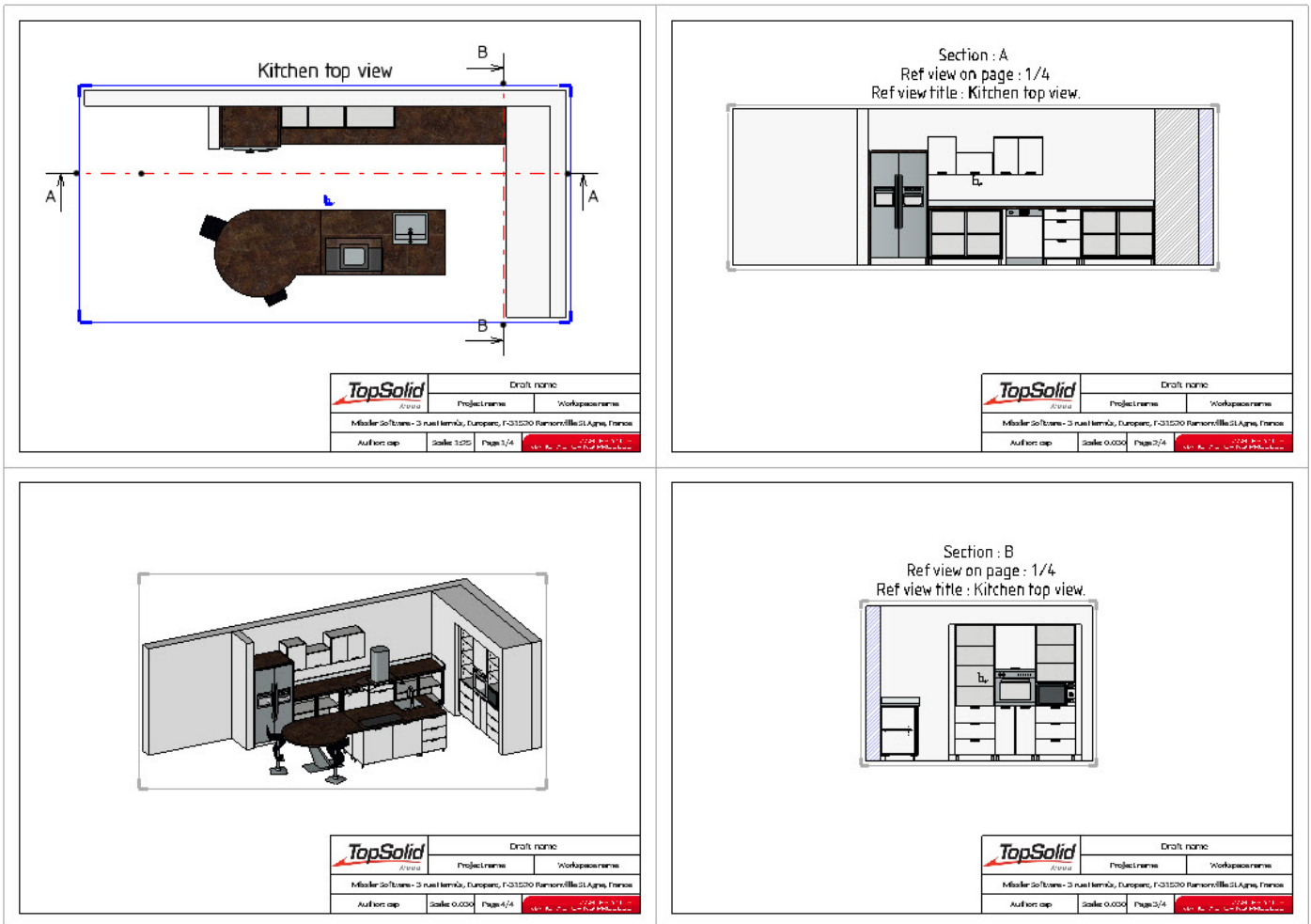
Note: To retrieve a property of the reference view, add **refview.** before the property.

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>.



Note: 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>.

Here is the final result in the drafting document:

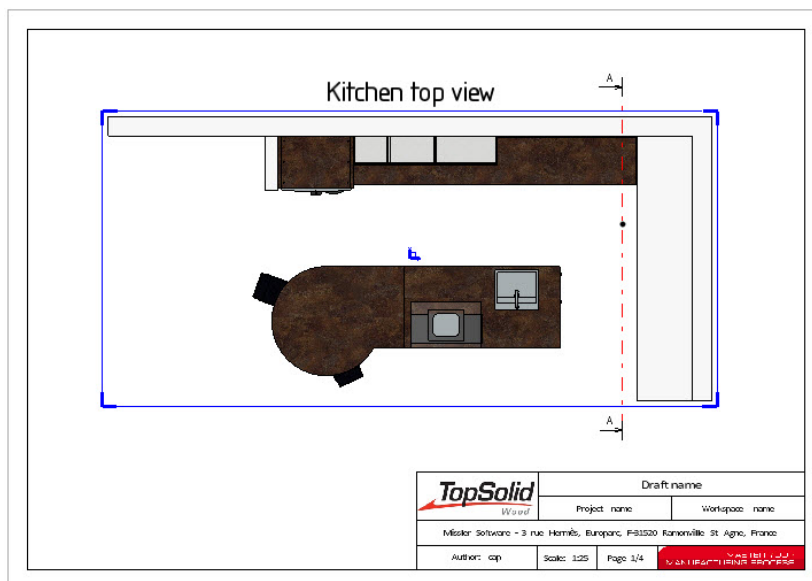


Elevation view

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

- Create a **main view**.

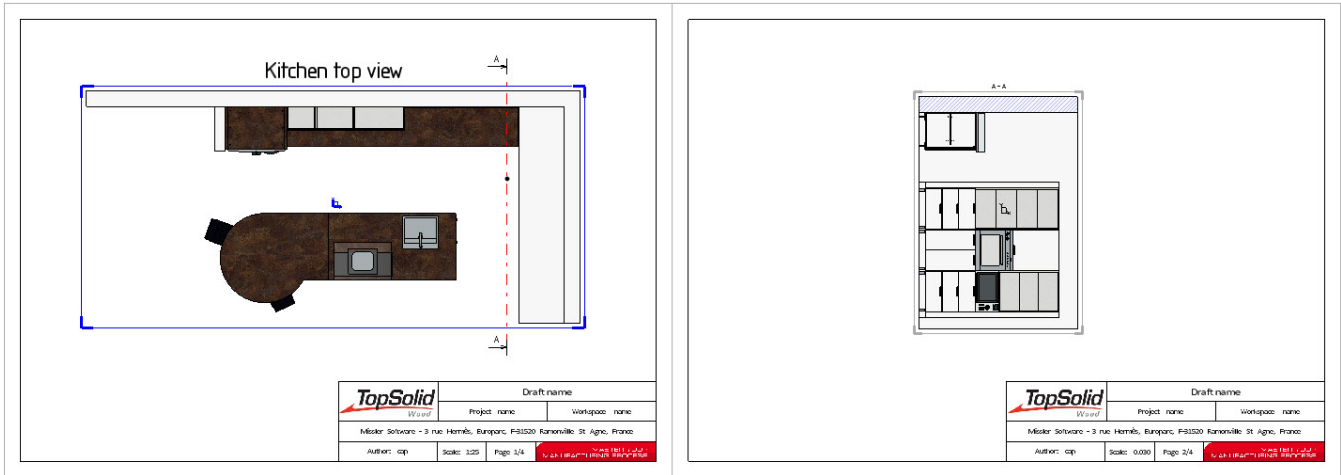
Again using the kitchen example, a section is created to better view one of the walls.



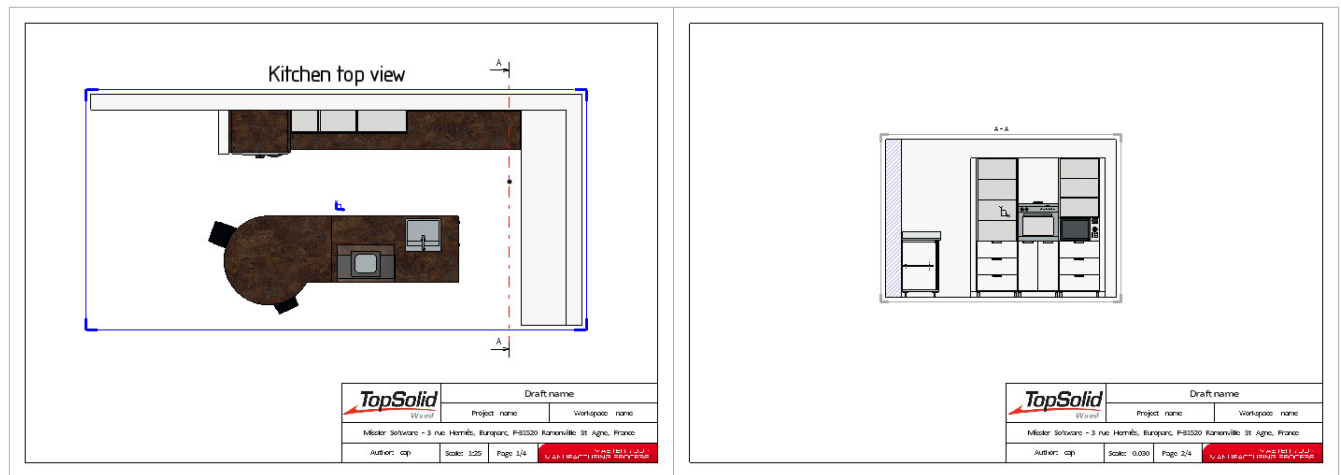
- Launch the **Full section** function and create the **cutting curve**.
- Before positioning the view, adjust **Alignment = NO**.
- The new **Set the view upright** option appears.

Note: 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

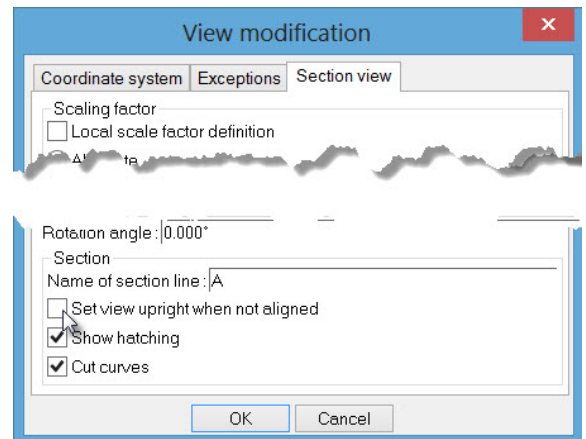


Set the view upright = YES



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.



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**.
 - The following options are available: Unit= Automatic
 - **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.



Note: 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

Miscellaneous

- 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  **Visibility** 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**.

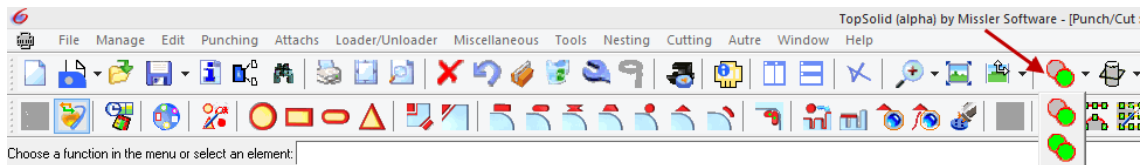
New features common to TopSolid'Punch and TopSolid'Cut

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.

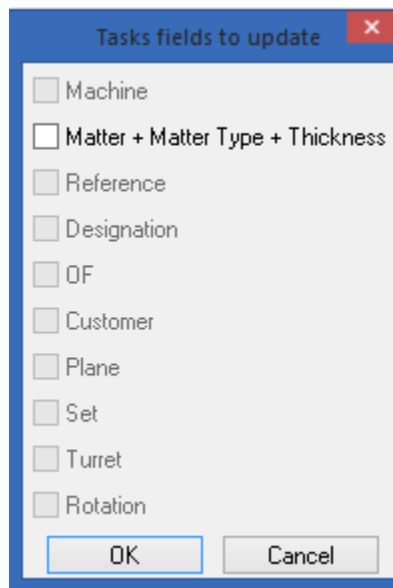


Production date

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

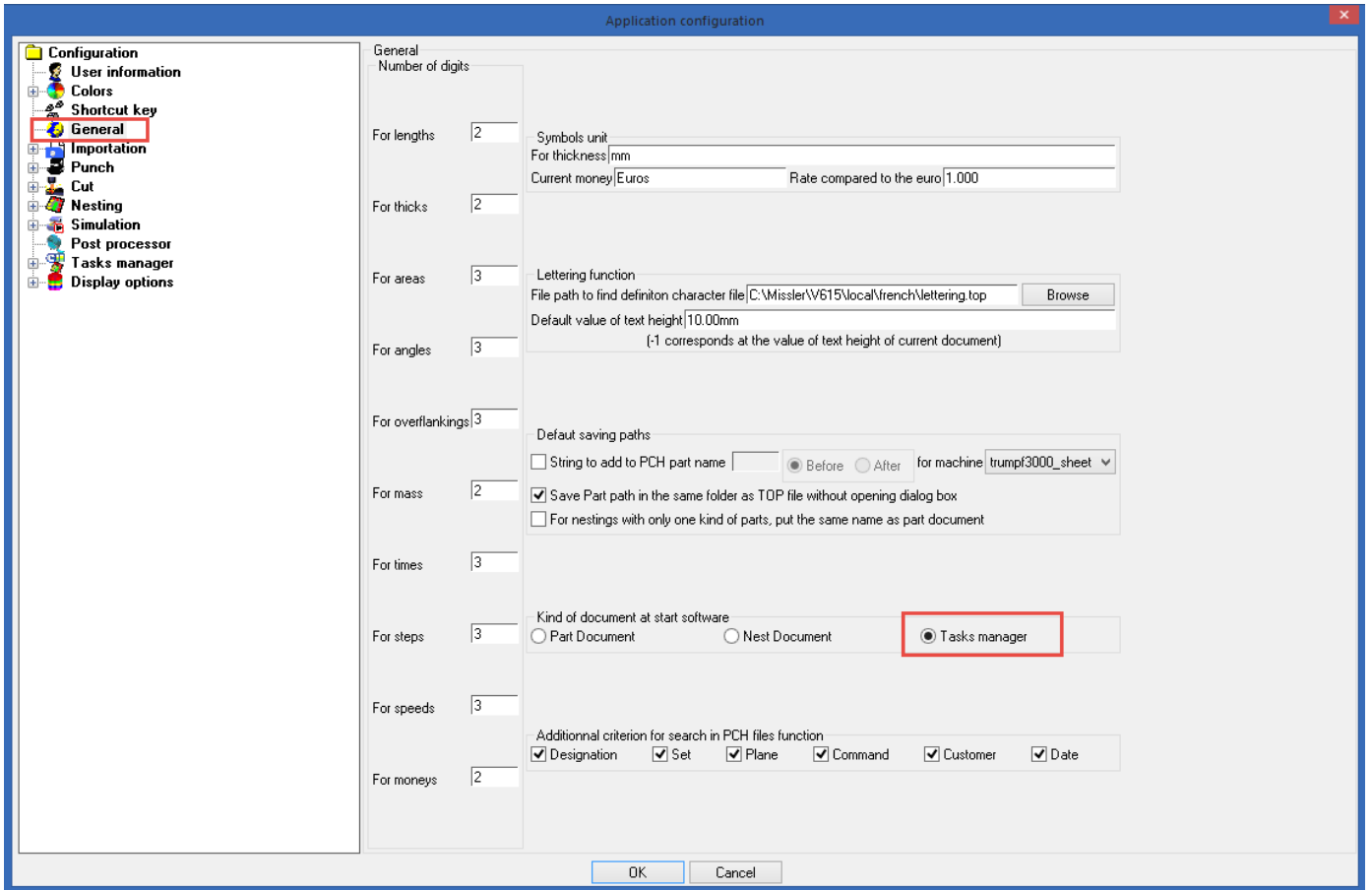
When opening the Task Manager from a Punch part document, the part data is checked to see whether it matches that of the Task Manager.

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



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**.



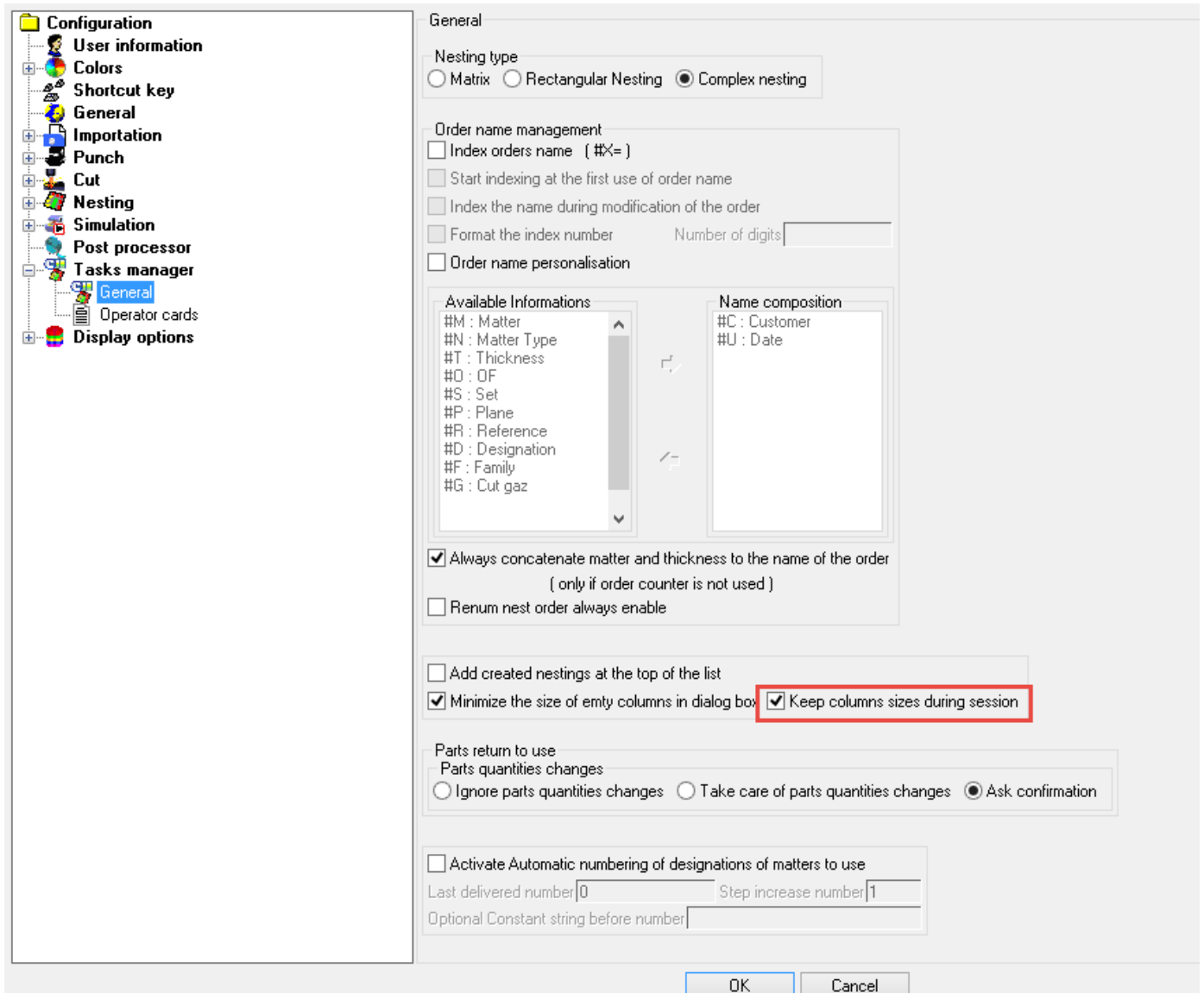
The option you choose is kept throughout the session, until you change it.

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.



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`.

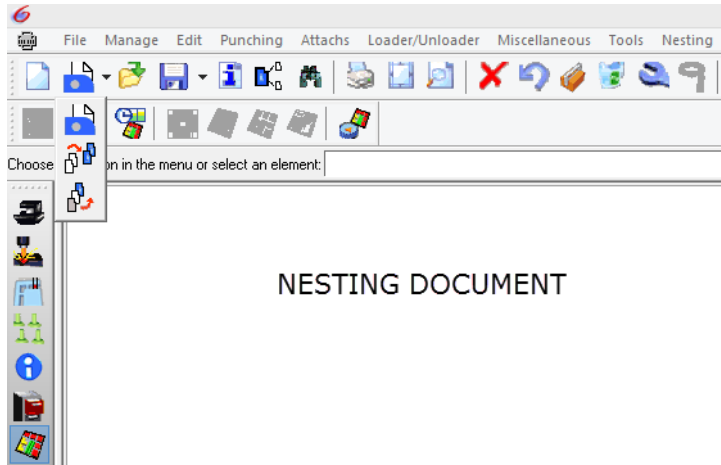
They are replaced every time the manager is saved.

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.

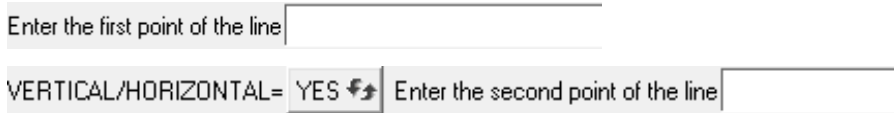
Part imports from nesting document

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

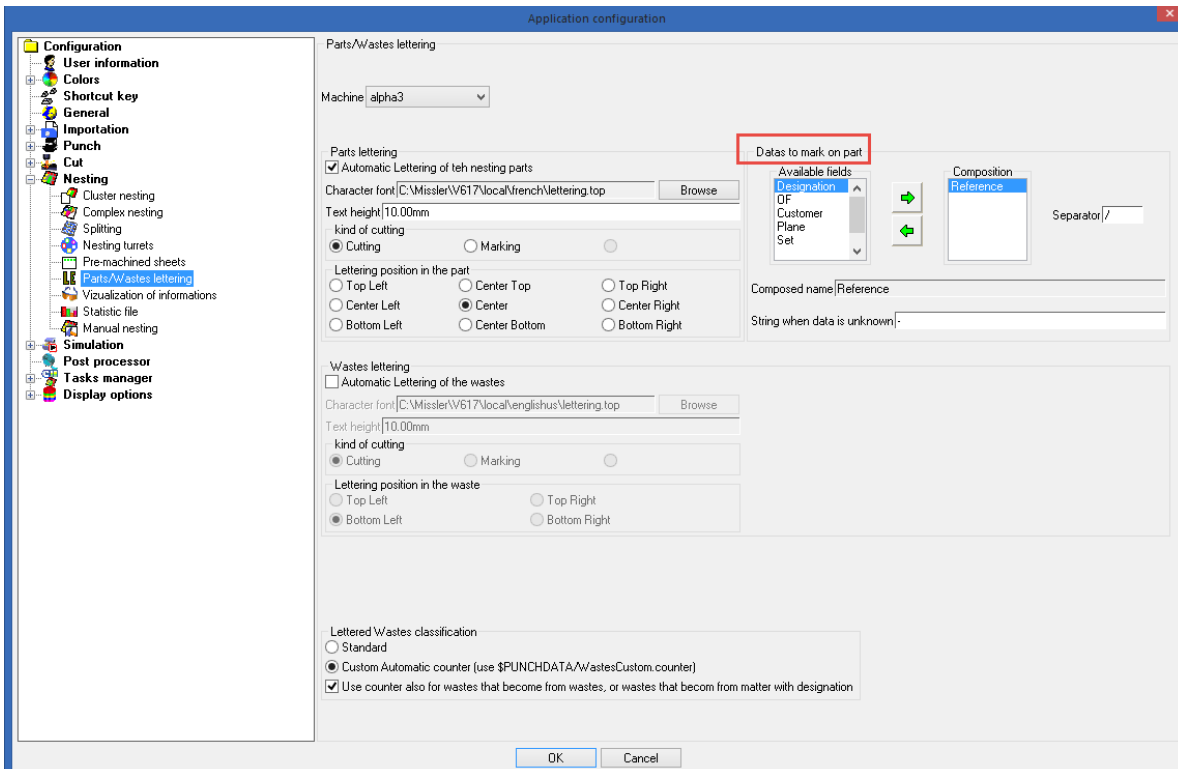
In the **Miscellaneous > Free contours > Free line** command, a button has been added to freeze the axes horizontally or vertically.



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**.



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

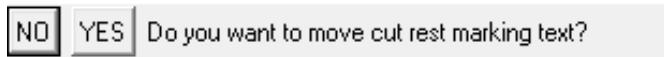
A new function has been added to the Punch and Cut modules to move a cut rest line.



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.

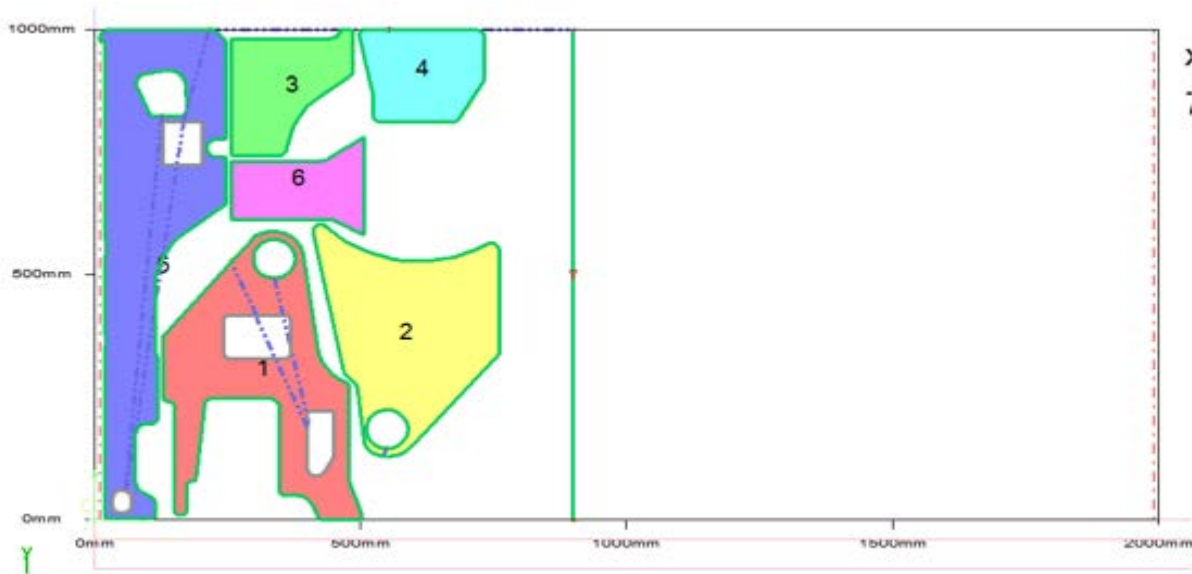


A lettering, which can be a waste marking, can now be moved dynamically, and it is represented by the oriented rectangle of the lettering box.

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):



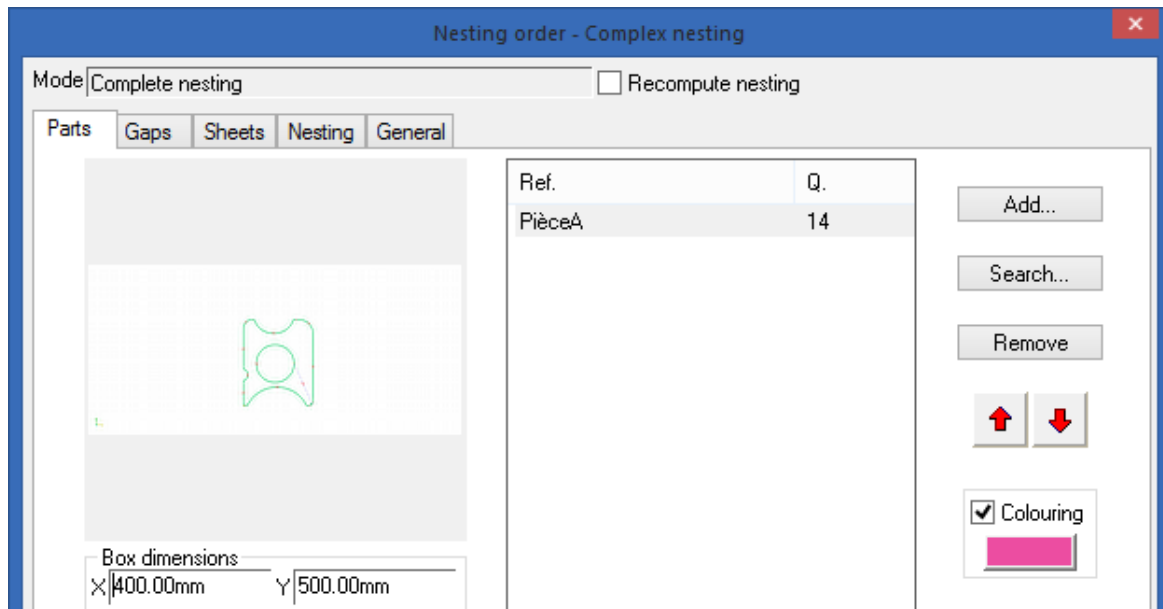
If you don't want this behavior, you can add the following configuration word manually to the *topzpch.cfg* file in order to return to the behavior of older versions.

```
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.

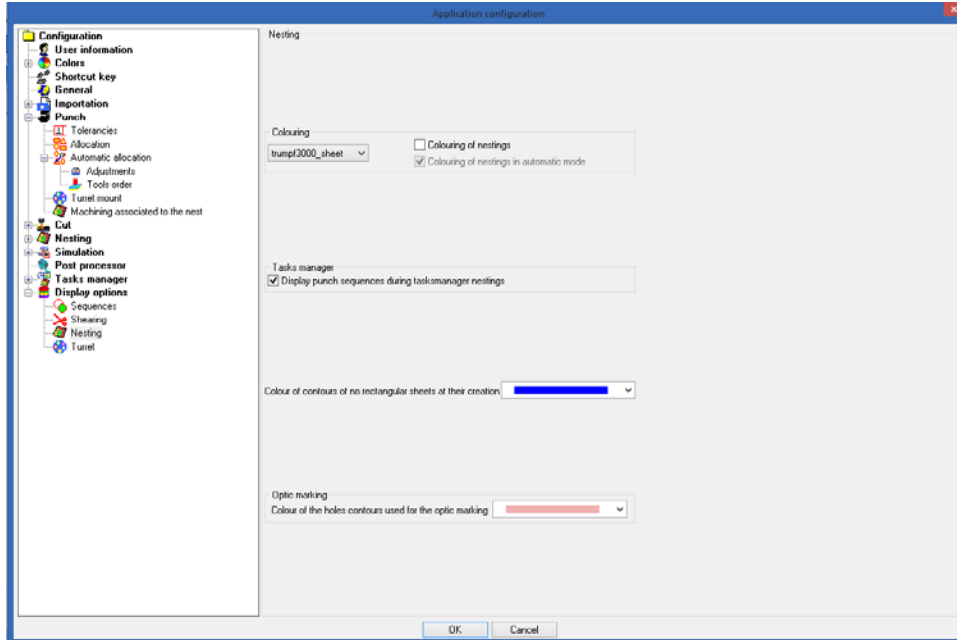


This gives each part a different color even if coloring was not requested during the nesting. Automatic color assignment is not applied if the nesting order contains only one part or in the case of unit part nesting, which makes it possible to keep the color defined in the part document.

TopSolid'Punch

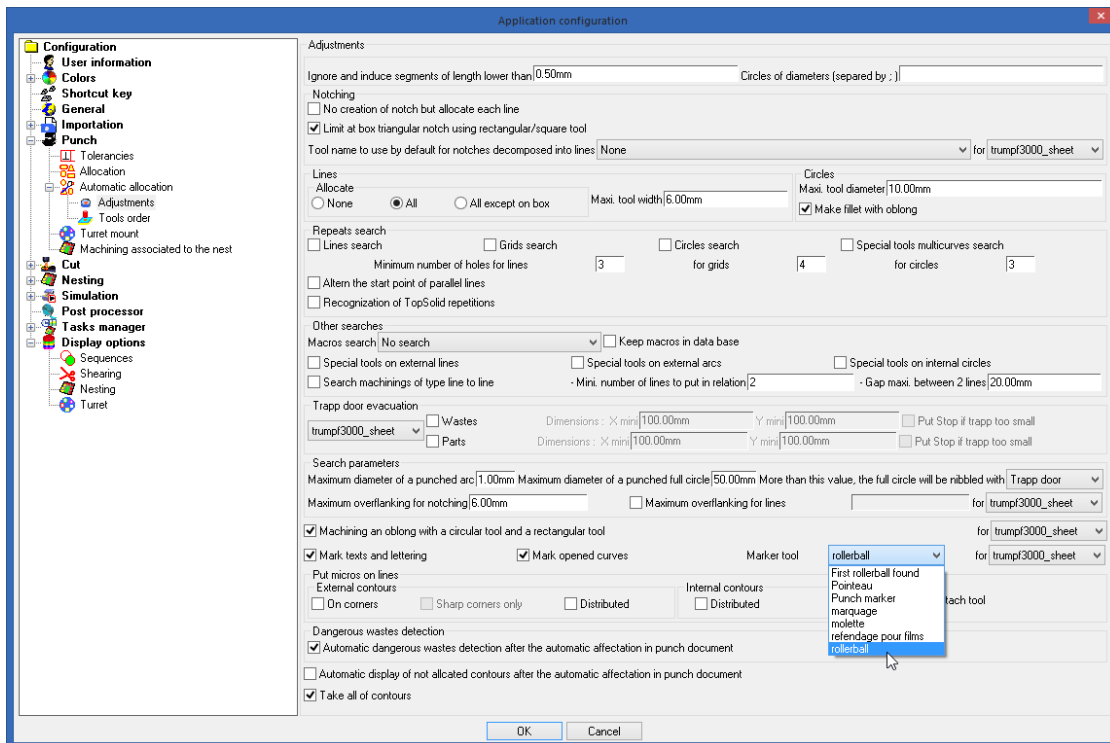
Punch sequence display

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



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**.



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**.

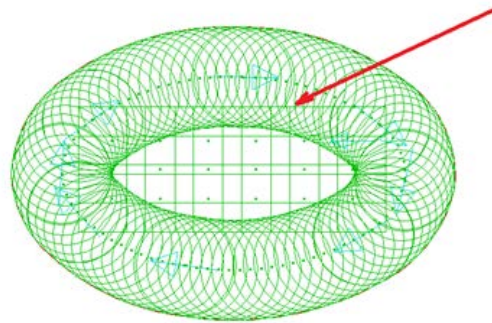
Copy of macro's free geometries

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.

Internal Free Macro



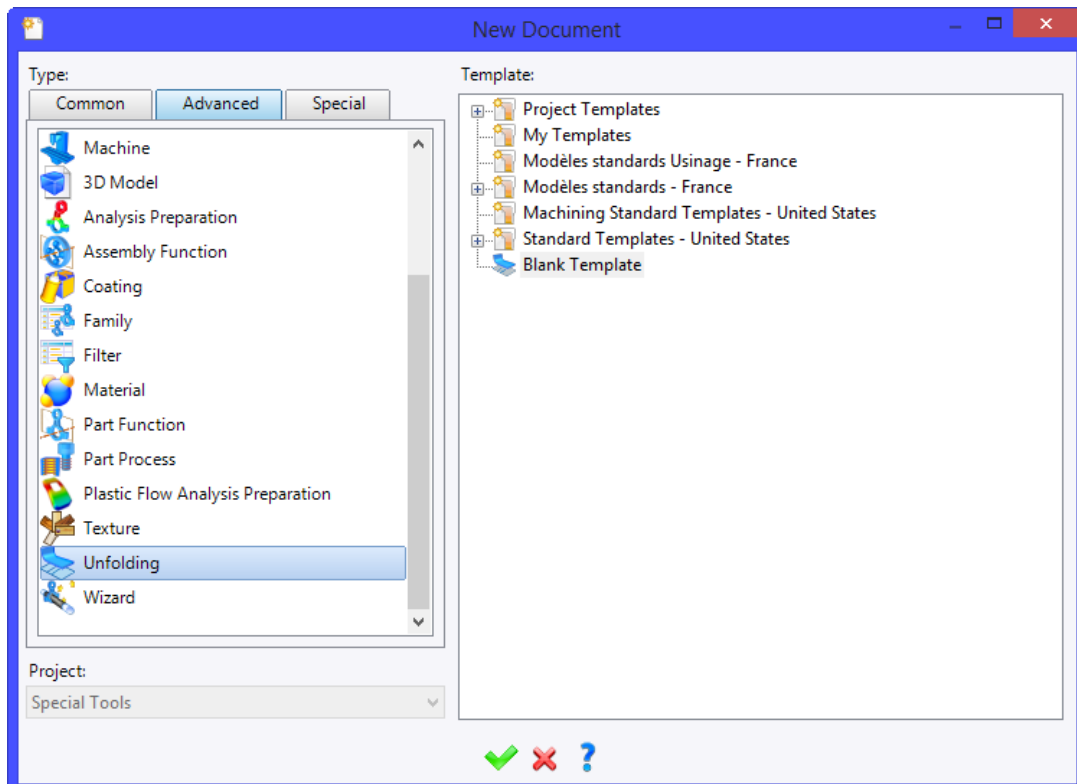
Note: In the machining document, the free geometries automatically created through macro recognition will be automatically removed (during update) when there is no more tool left on this free geometry.

Creation of special tools from TopSolid 7

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

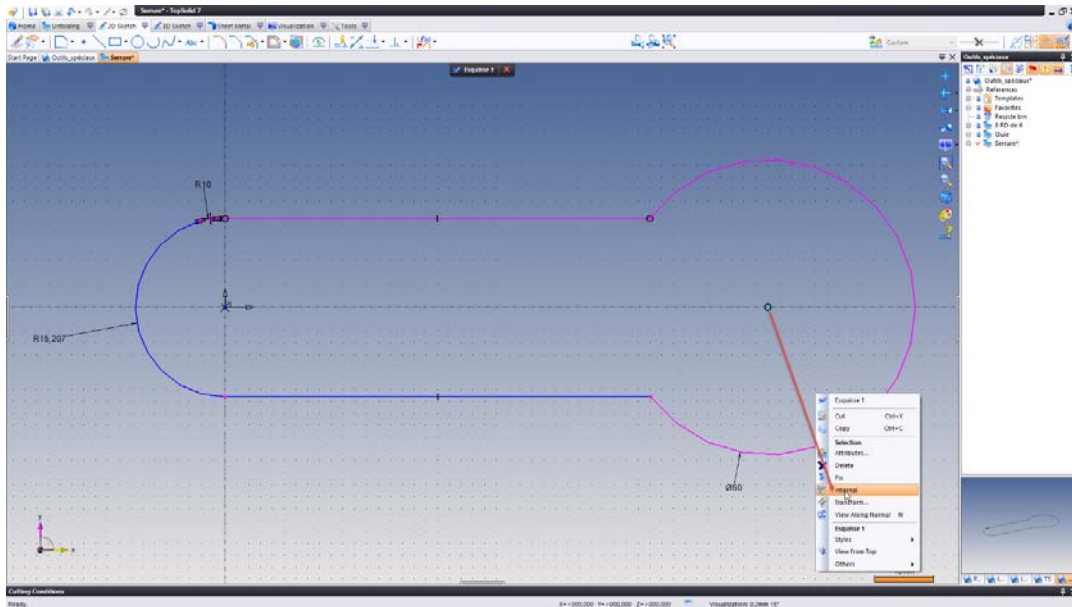
Methodology

- 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.

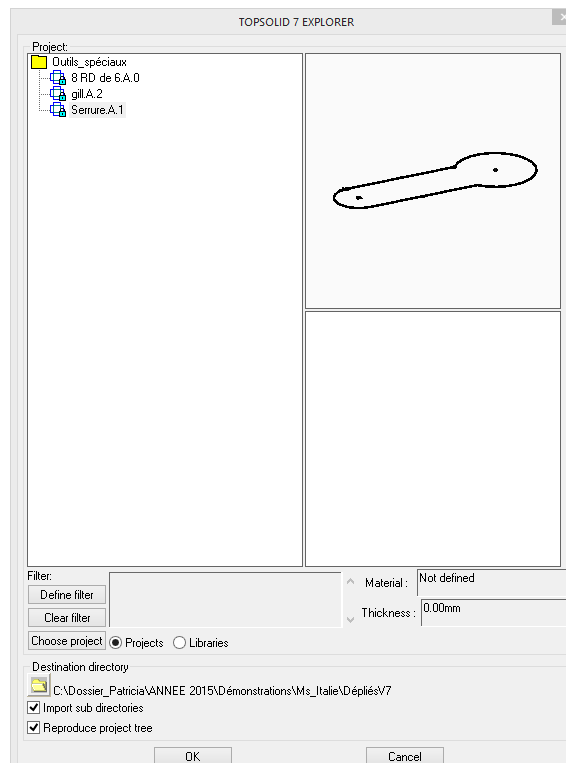
Example of a keyhole tool:



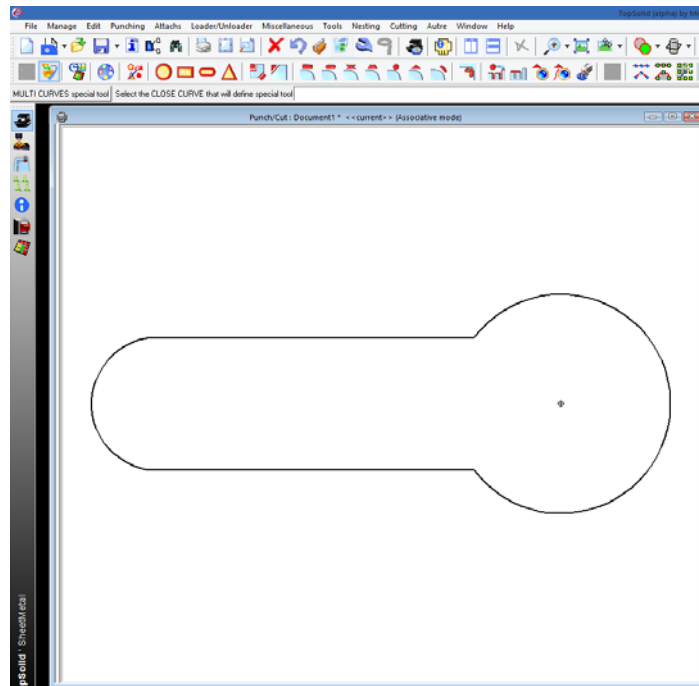
- Create the special tool in **TopSolid'Punch**:
 - Use the **Manage > Special tools creation** function.

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

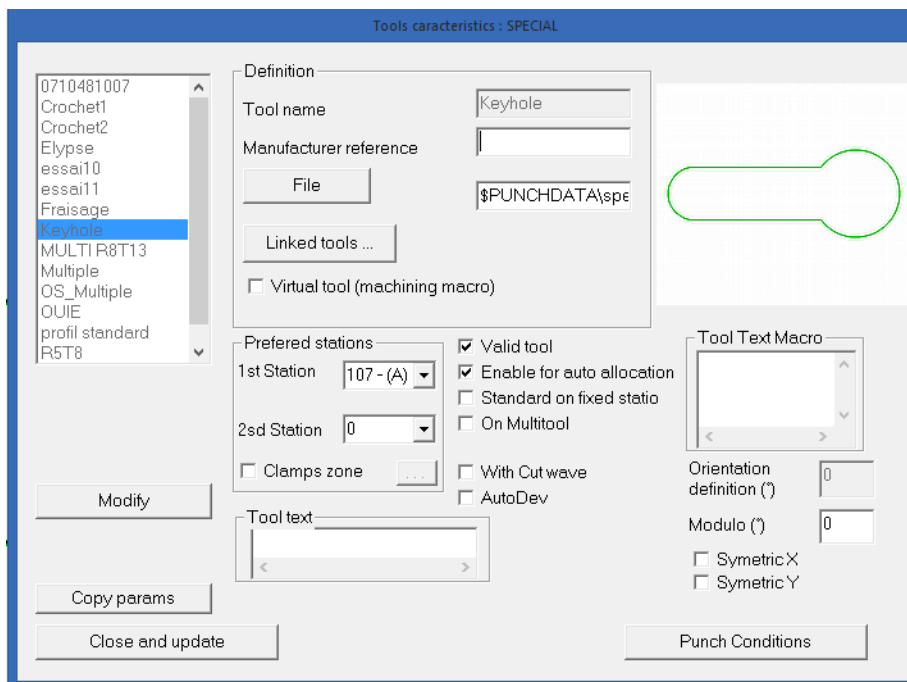
- Click the **V7 Explorer** button and select the project and the document for the tool.



- 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.

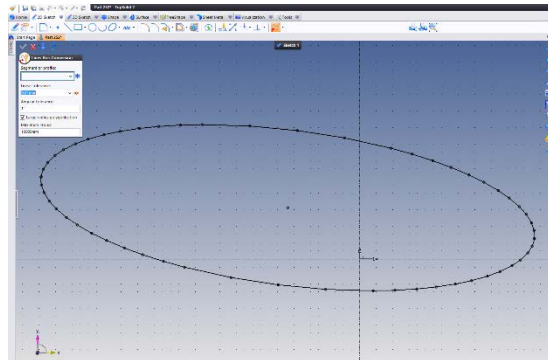


• Case of multi-curve tools:

- After choosing the document in the Project tree, click the **MULTI CURVES special tool** button.

Note: 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.

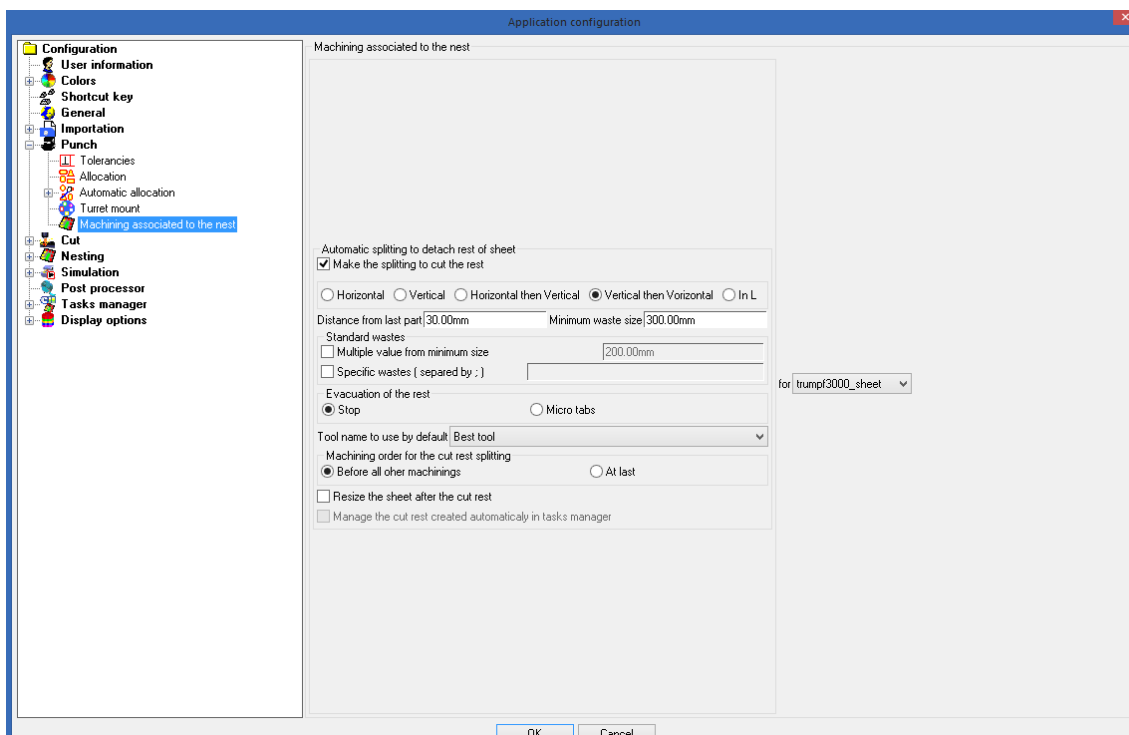


Cut rest manual function for punching

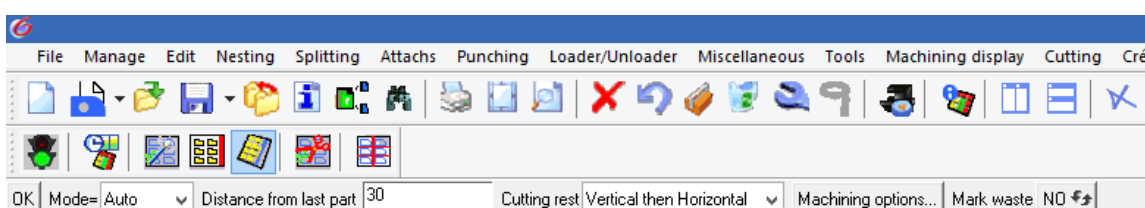
A new function is available in the **Splitting** menu to create either vertical, horizontal, mixed or in L-shape cut rest.



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



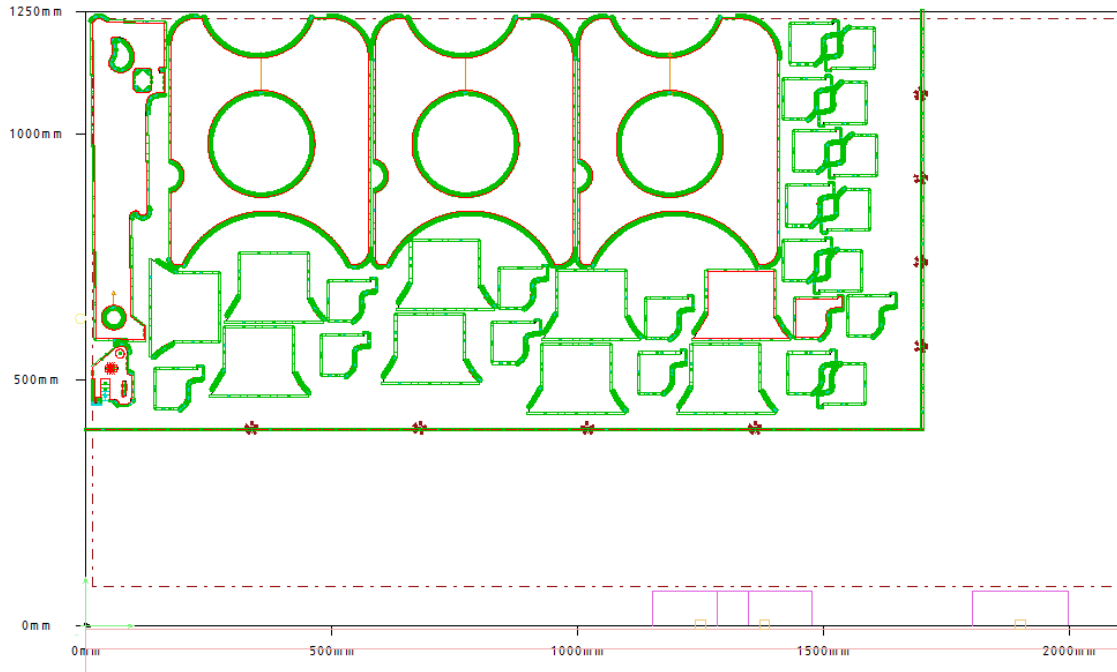
The **Auto** mode lets you create one or more wastes at a given distance from the last part.



The **Machining options** button lets you configure:

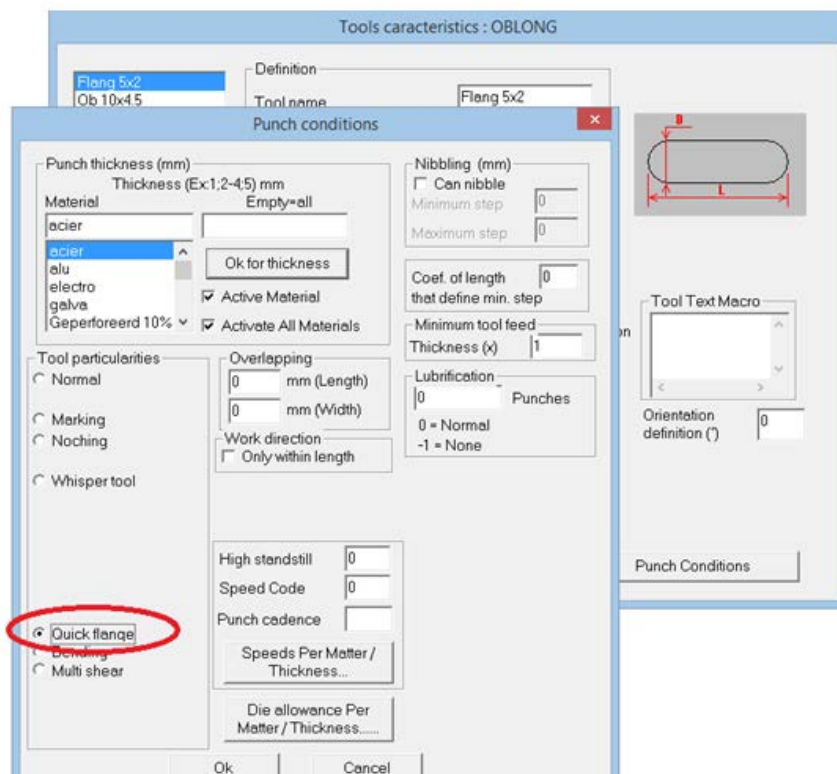
- whether the waste will be evacuated with a machine stop or held by micro-junctions;
- the tool selection to punch the lines;
- whether the waste cutting must be done at the end or before all other machinings.

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



Flange

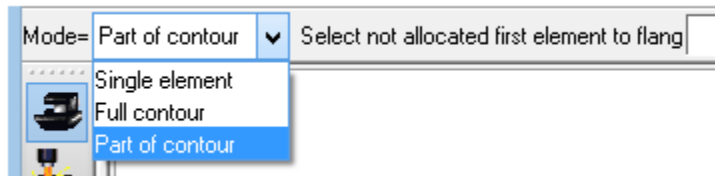
A new function in the **Punching** menu can be used to create flanges on geometries with Z elevation management. You need to define an **Oblong** tool and enable the **Quick flange** option in the punching conditions.



Functioning

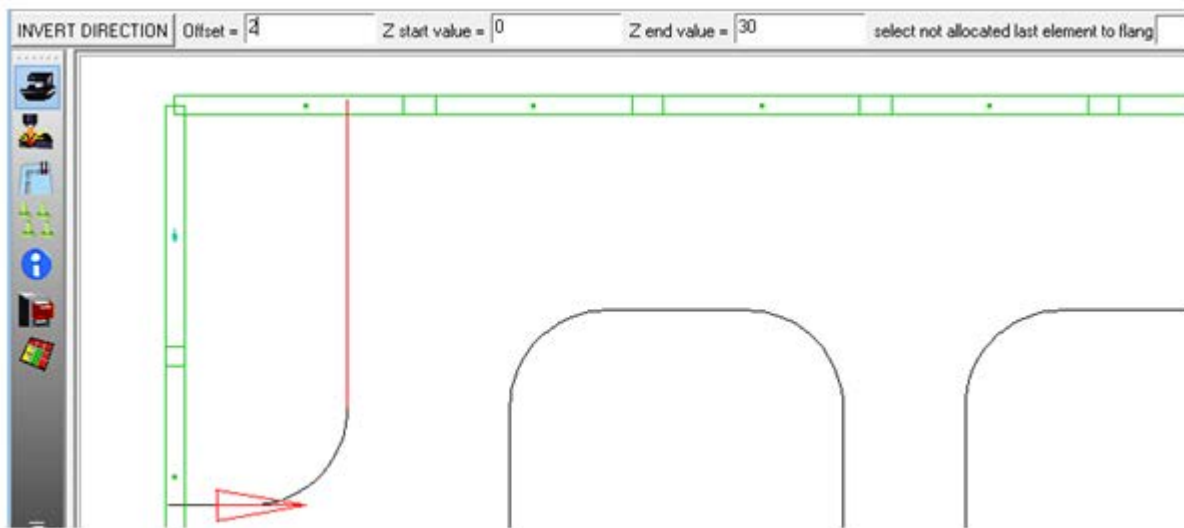
- Call up the function.

Several modes are available.

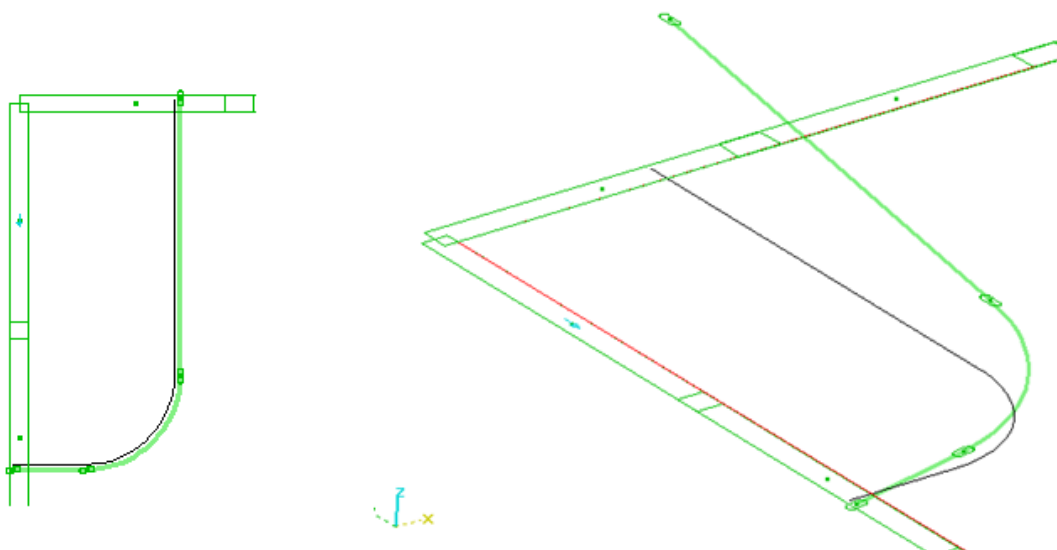


Part of contour mode:

- 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.



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.



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

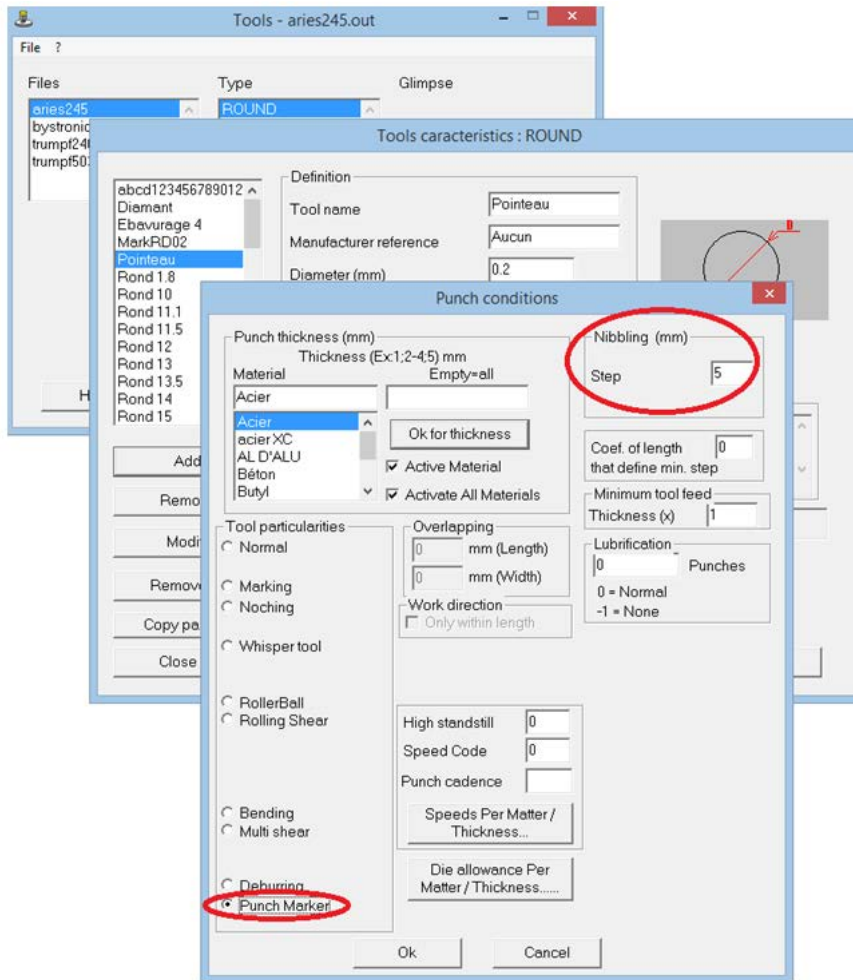
When the tool is shown dynamically on the mouse, its displayed orientation is now the angle of the mounting key on the turret + the definition orientation of the tool itself.



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.

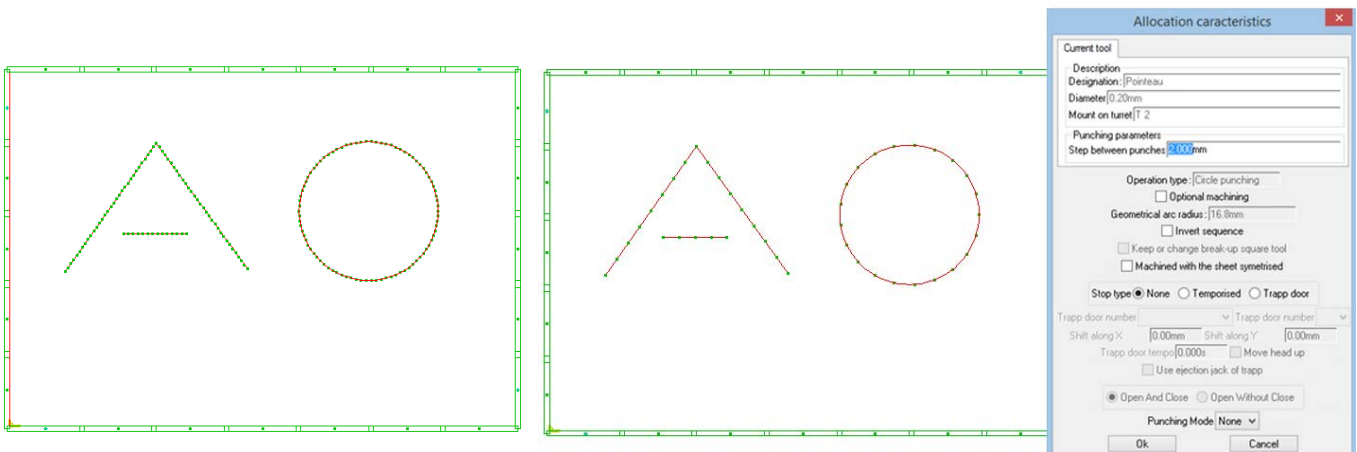
Punch marker tool

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.



Its allocation is done directly on the geometry using the  **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.

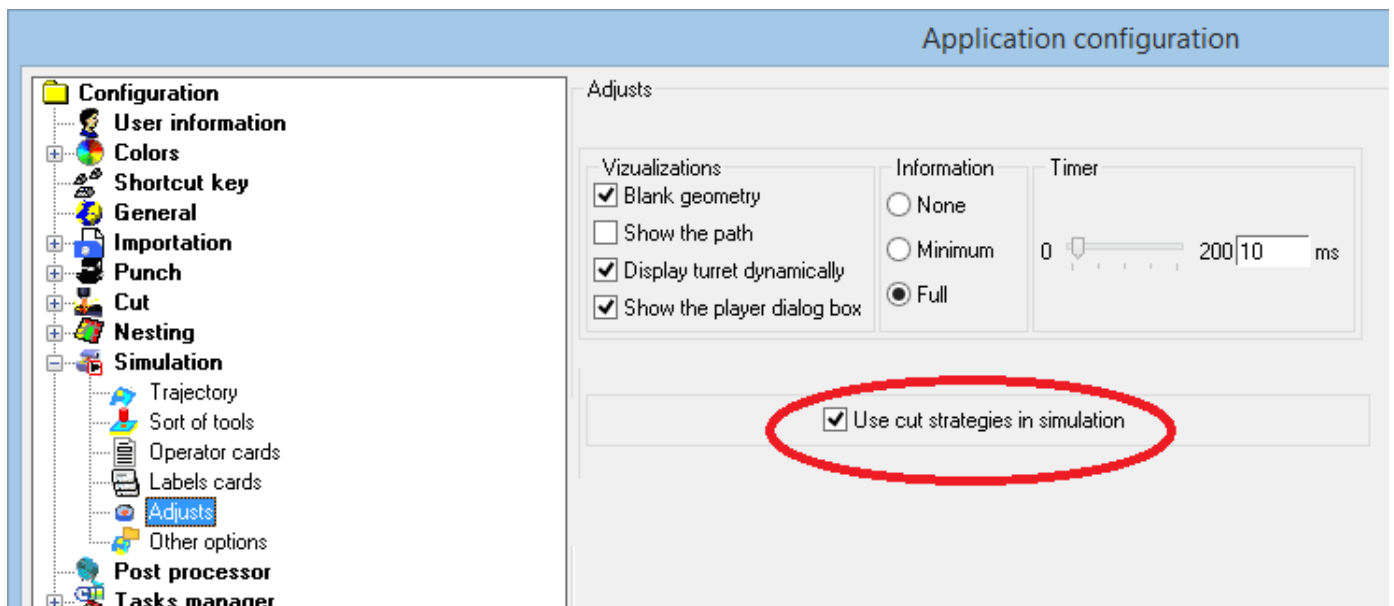
Consequently, no links must be defined between cuttings or parts. Keep only imposed links that cannot be modified by the simulation.

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

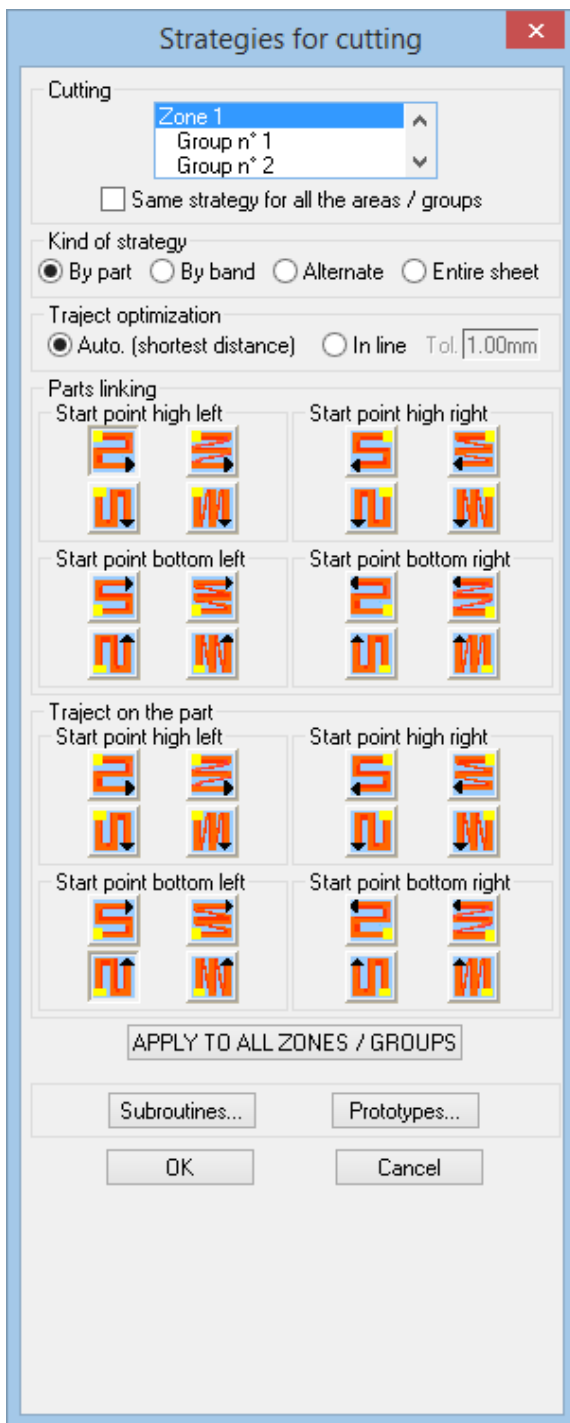
You can make changes to a link generated by simulation (shown in black), but it will turn into a blue-colored imposed link which is no longer impacted by the simulation.

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



Dialog box for cutting machines

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



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.

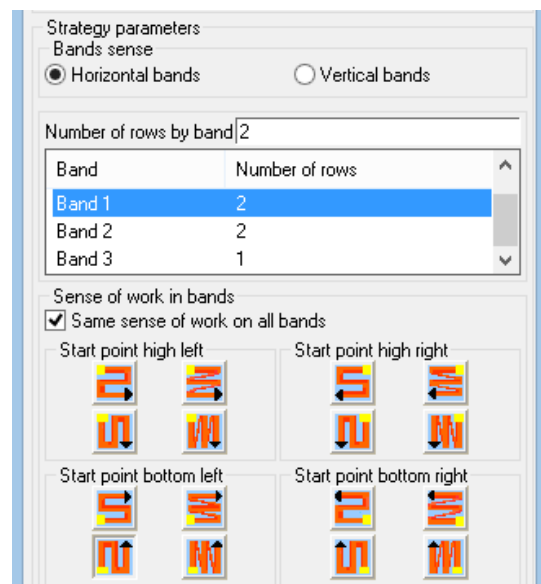
A different strategy can be applied to each of the groups.

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.

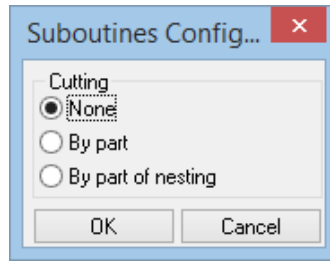
- **Strategy by horizontal or vertical bands**



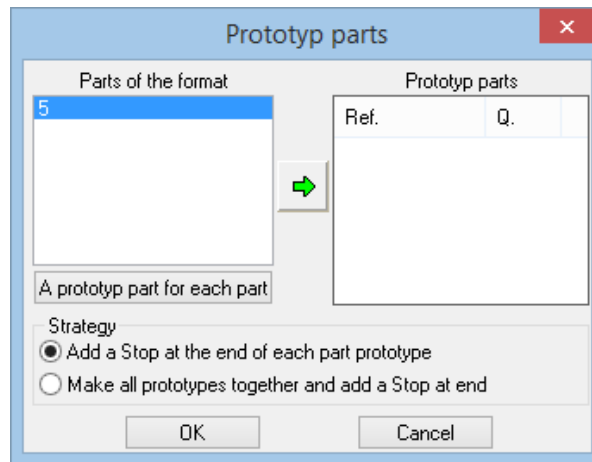
- **Entire sheet strategy**
All part's internal contours are processed before external contours.

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

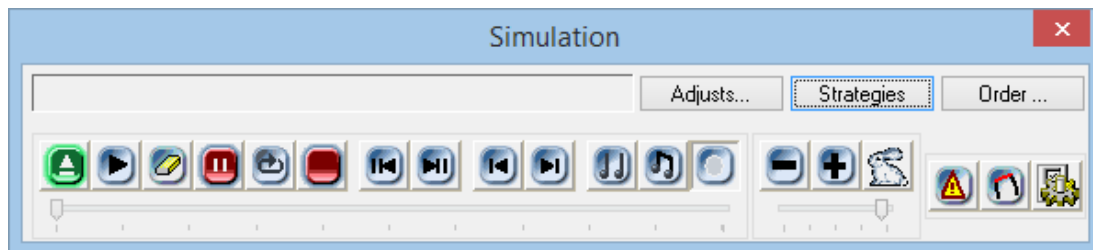
The **Subroutines** button lets you configure the way subroutines are generated.



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

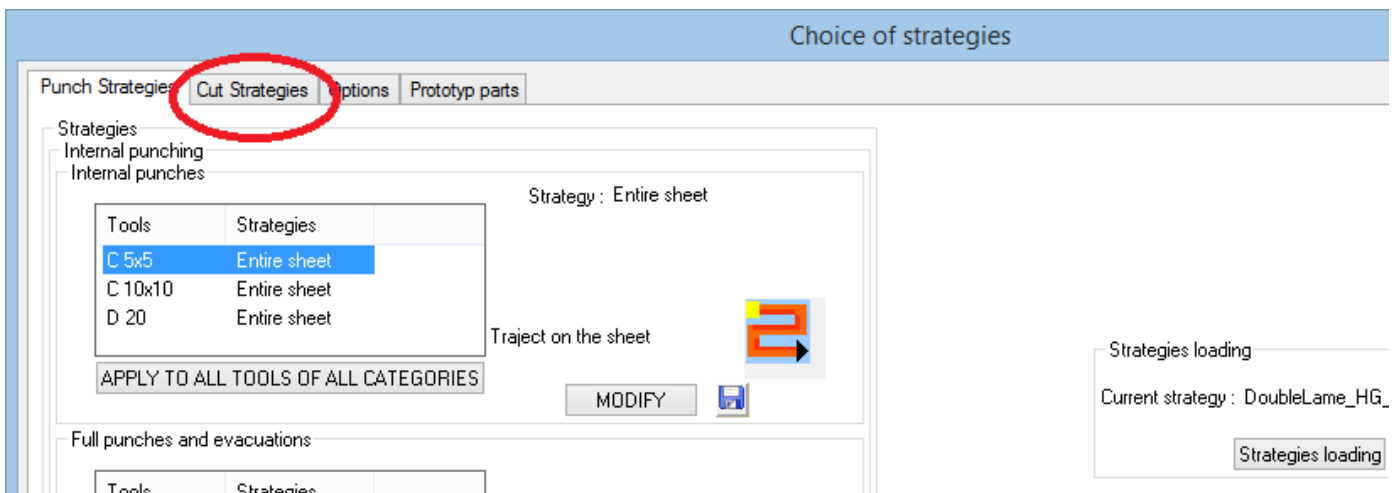


The simulation player displays the **Strategies** button which allows you to subsequently modify the cutting strategy.

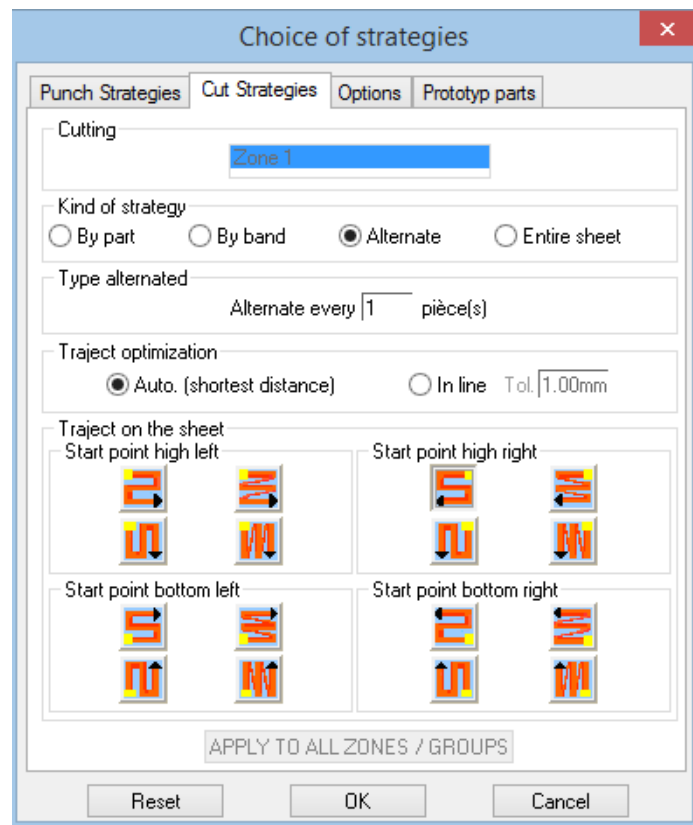


Dialog box for punching/cutting combined machined

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

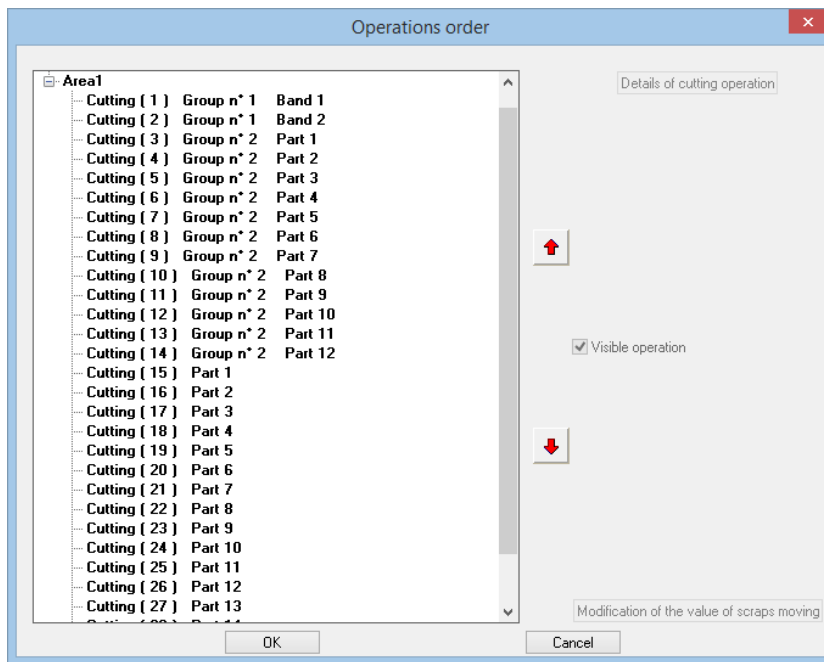


When clicking this tab, the dialog box changes, allowing you to define the cutting strategies.

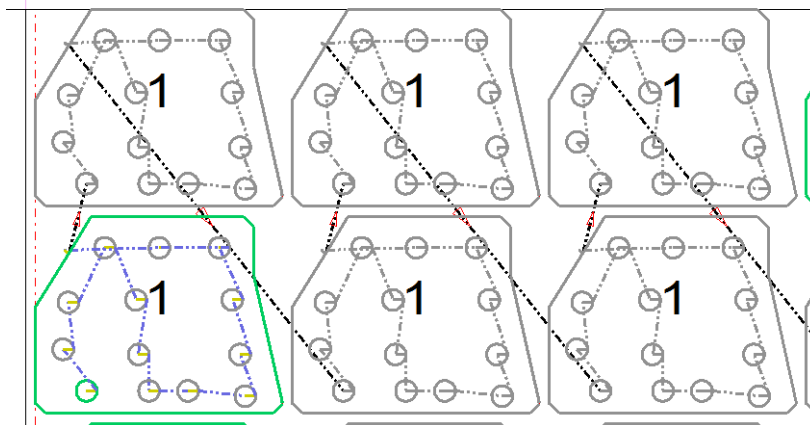


Operation order dialog box after simulation

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

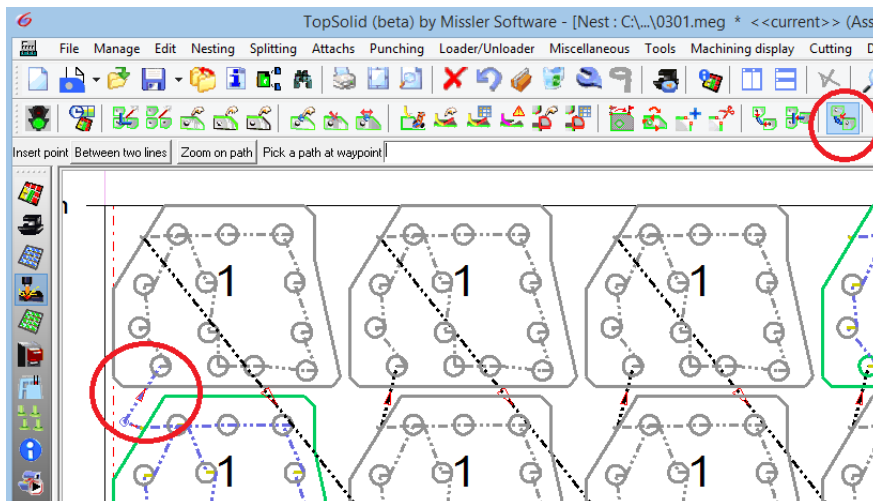


Example:



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**.

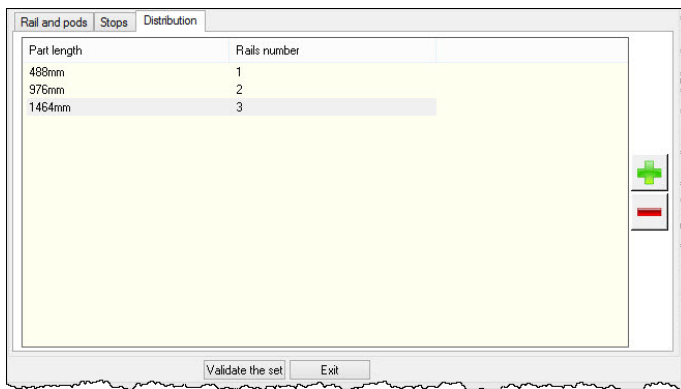
Rails and pods

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.

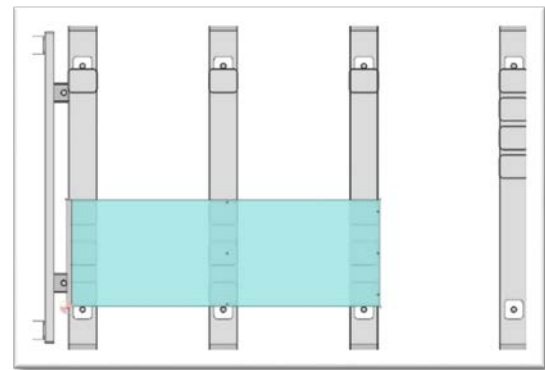
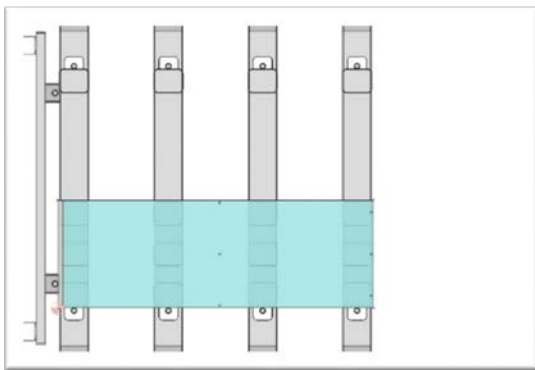
- To configure the distribution of rails, use the **Rails and pods definition** function. In the window, select a set of rails and pods, and then from the **Distribution** tab define the maximum number of rails for a given part length.

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



Adds a new line to the rail distribution table.

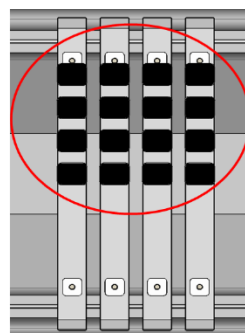
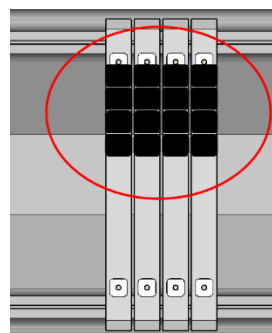
Deletes the current line in the rail distribution table.



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

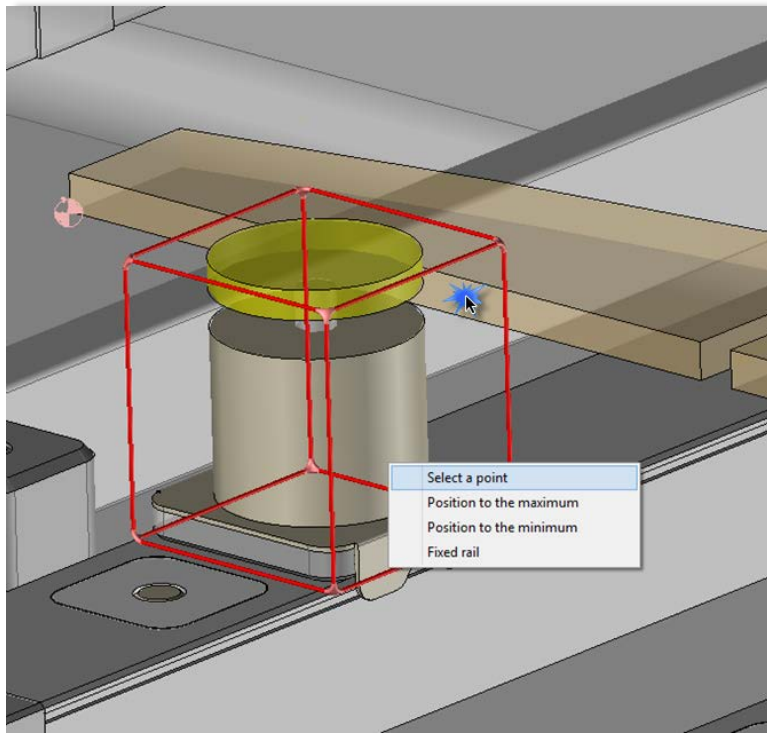
Stop mode:

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

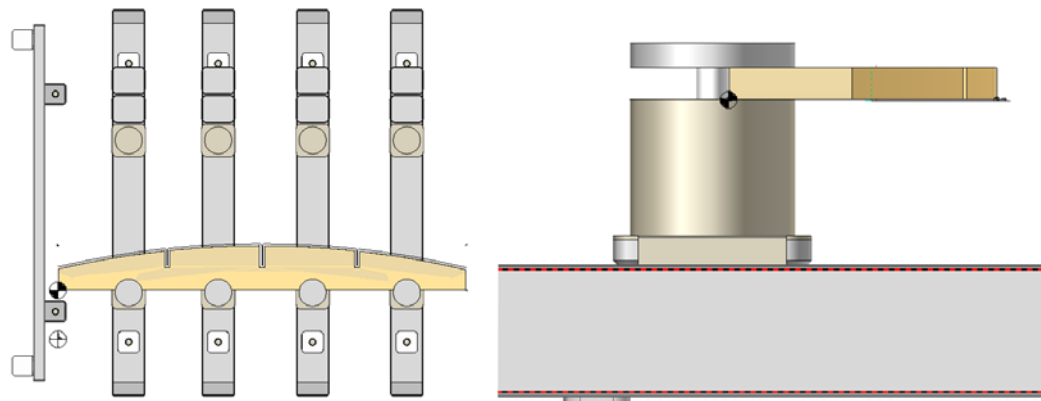
The minimum distances between rails and between pods are taken into account.

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.

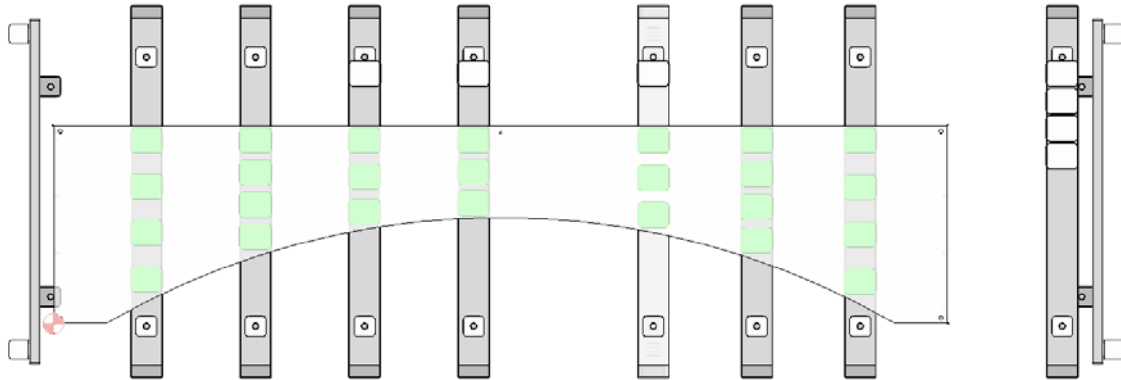


Automatic positioning of rails and pods in several areas

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.

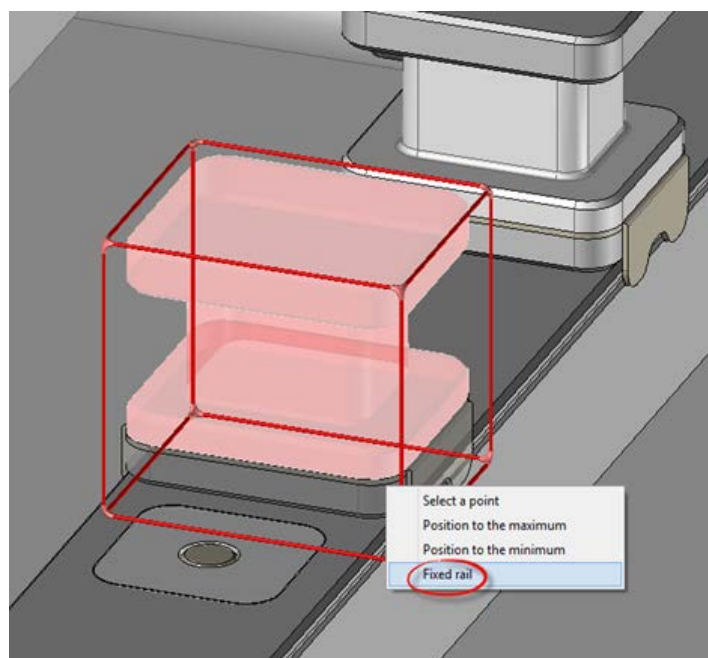
The stops that are related to each of the rail and pod sets are defined in the **Stops** tab when you create the set .

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



Fixed rails in a set

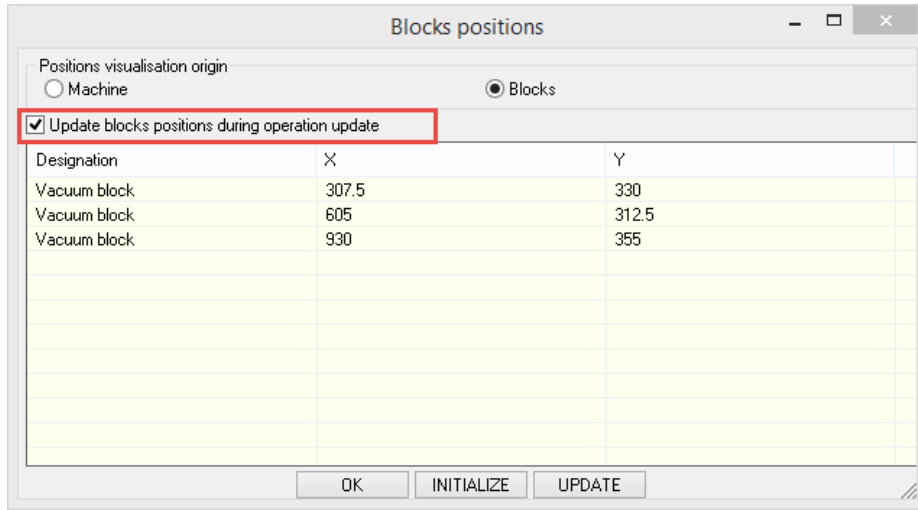
When the pods are positioned manually, it may be necessary to fix the position of rails. The **Fixed rail** function has been improved so that all the set's rails can be fixed.



Vacuum blocks

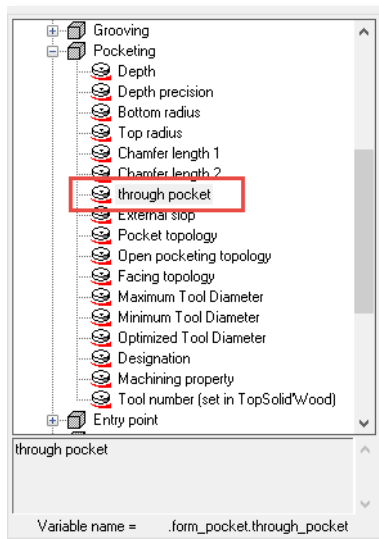
Updated vacuum block positions

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



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.

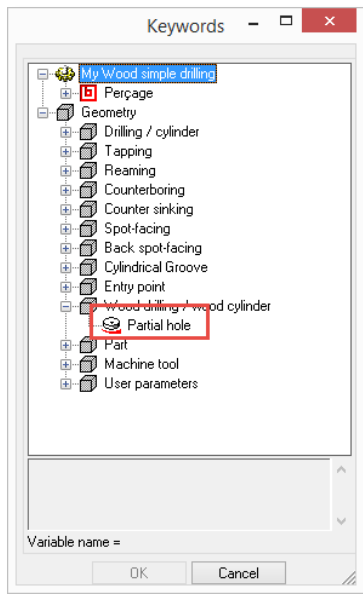
The pocket operations have the advantage of being processed independently of each other by the process assigned to them. In addition, these operations can be sorted using the **Operations sorting** CAD function, unlike apertures.

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 it is 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.

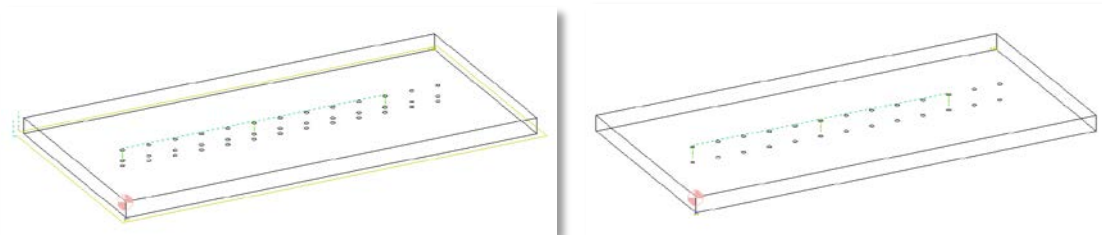
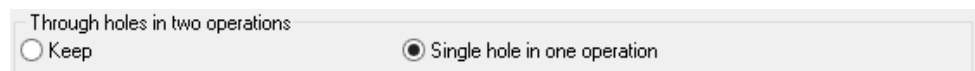


This boolean (**.form_woodHole.IsPartialHole**) displays **1** if it is a partial hole and **0** if not.

A different drilling depth can be given depending on whether it is a partial hole or not using the following syntax:

```
(.form_woodHole.IsPartialHole==0)?.form_hole.depth:.form_hole.depth*0.6
```

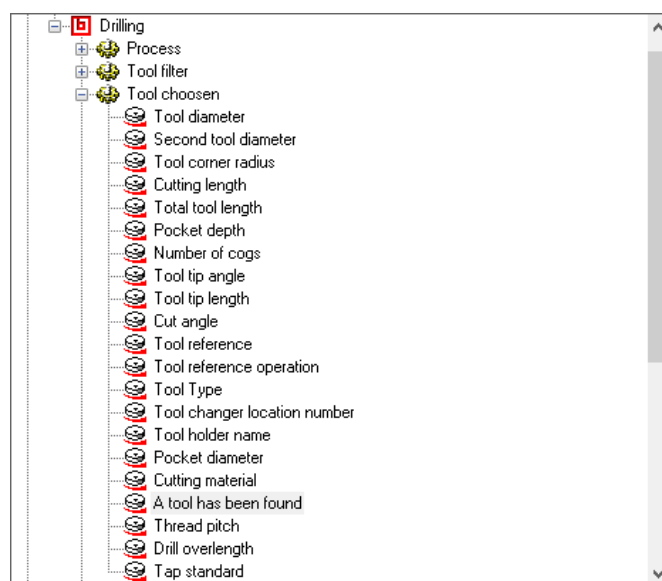
Then, from the **Properties > Routing configuration > Holes** section, you just need to choose whether through holes in two operations should be kept or processed as holes in a single operation.



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.

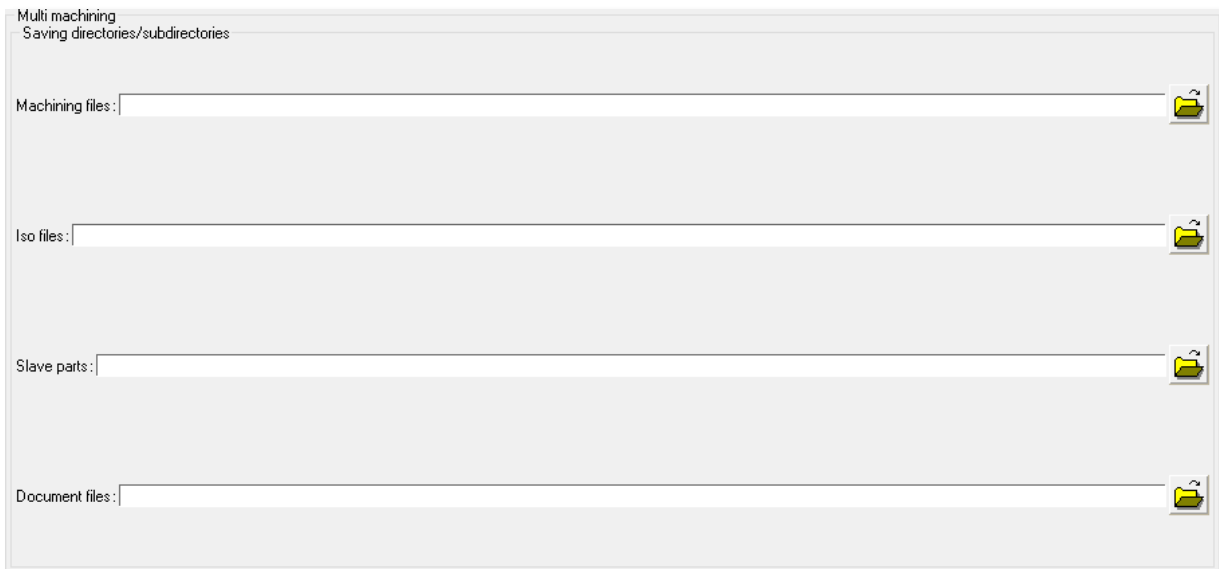
This option is saved and can be configured for the document using the **File >  Properties** function or directly in the options of the **analysis of part operations **.

Warning: 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.



Nesting

Abort the execution of nesting operations

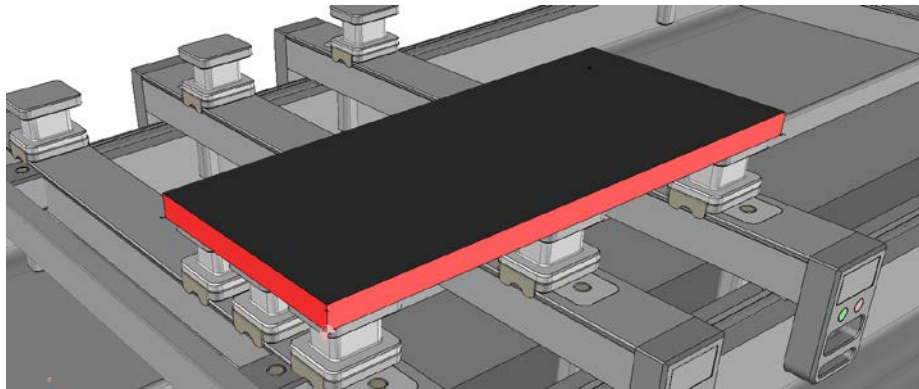
The **Esc** key lets you stop the analysis of all the nesting operations.

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

Reminder: 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

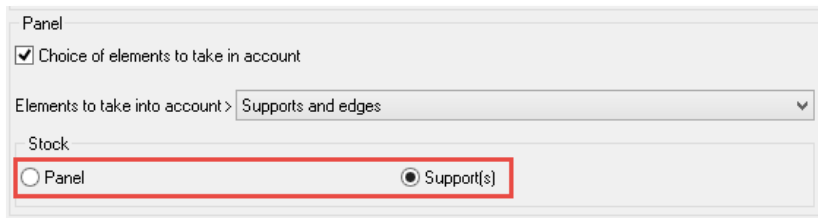
In this case the machinable part is not created and **TopSolid'WoodCam** work as in the 6.16 version.

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.



Different options are available:

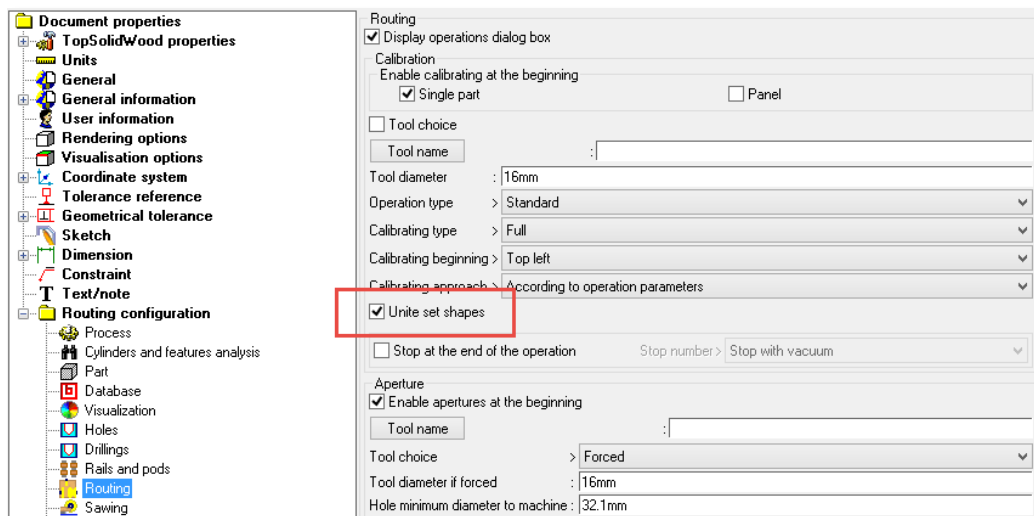
- Positioning of the incomplete panel: The panel's stock provided by TopSolid'Wood is used.
- Positioning of the panel and edges: The panel's stock provided by TopSolid'Wood or the enclosing shape of supports and edges is used.
- Positioning of the panel and laminates: The panel's stock provided by TopSolid'Wood 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).



5-axis machining

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 Theoretical 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.