

TopSolid

2017

TopSolid v6.18

What's New

© 2017, Missler Software.
7, Rue du Bois Sauvage
F-91055 Evry, FRANCE
Web: www.topsolid.com
E-mail: info@topsolid.com
All rights reserved.

This information is subject to change without warning.

No material may be reproduced or transmitted, regardless of the manner, electronic or mechanical means used or purpose, without formal written consent from Missler Software.

TopSolid® is a registered trademark of Missler Software.

TopSolid® is a product name of Missler Software.

The information and the software contained within this document are subject to change without prior warning and should not be construed as a commitment by Missler Software.

The software covered by this document is supplied under license, and may only be used and duplicated in compliance with the terms of this license.

Version 6.18 Rev.02

Contents

- What's New in TopSolid'Wood v6.18..... 1**
 - Improved user interface 2
 - Design 6
 - Wood machining..... 10
 - Bill of material..... 11
 - Configuration 18
 - Drafting 28
 - Interfaces 42
 - Miscellaneous 43

- What's New in TopSolid'SheetMetal v6.18 45**
 - TopSolid'Punch and TopSolid'Cut 46
 - Task Manager 50

- What's New in TopSolid'WoodCam v6.18 57**
 - Template machine 58
 - Group work..... 60
 - Script 60
 - Link movements..... 61
 - Aggregates 62
 - Tool management..... 63
 - Contouring cycles..... 65
 - Analysis 65
 - Scoring 65
 - Nesting machining 66
 - Multi-machining..... 66
 - User machining 67
 - Ergonomics 67
 - Workshop documents 67
 - Stop parameter modification 68
 - Additional parameters for post-processor 68

What's New in TopSolid'Wood v6.18

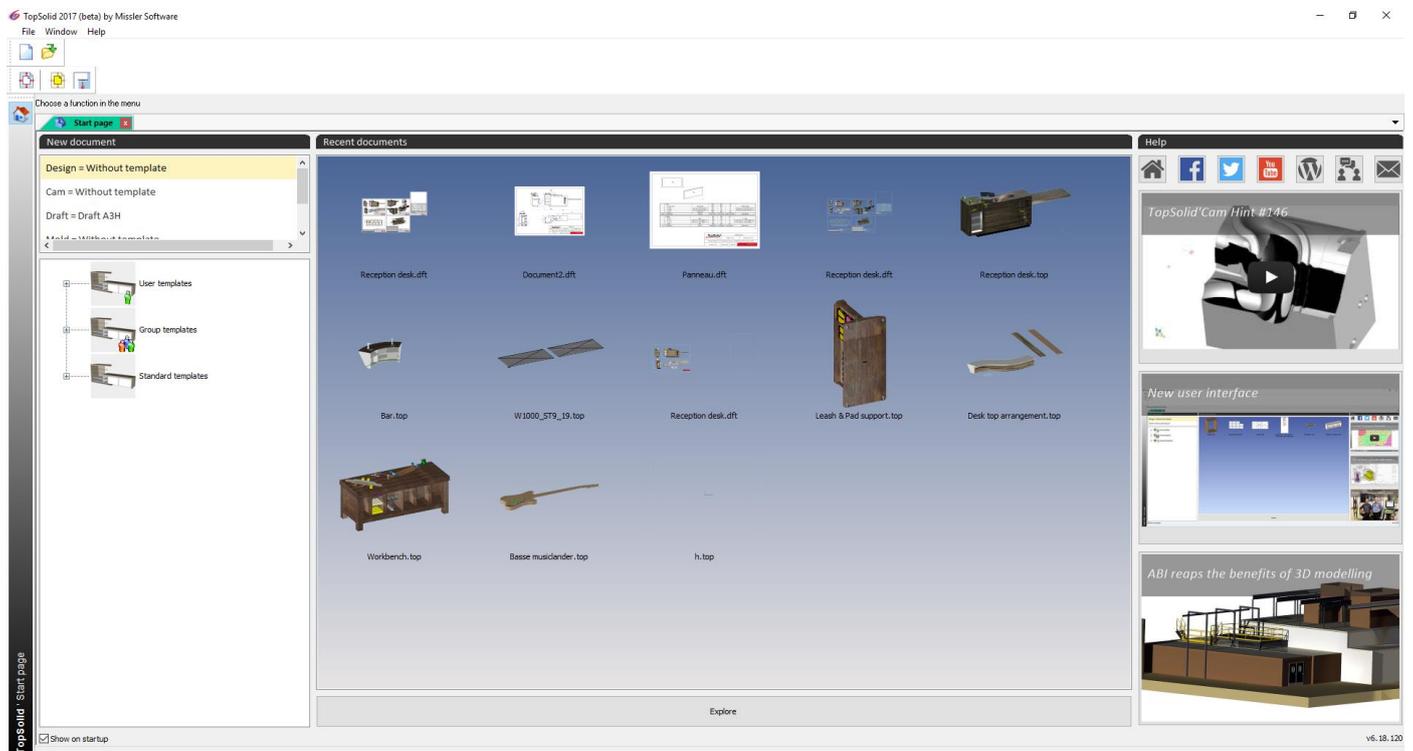


This section describes the enhancements made to the version **6.18** of **TopSolid'Wood**.

Improved user interface

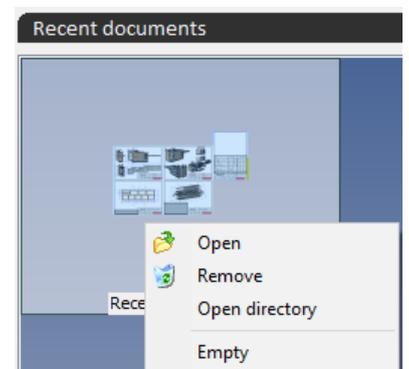
The **TopSolid 2017** user interface has been completely redesigned to be more user-friendly and intuitive.

A start page enables faster and easier access to recent documents.



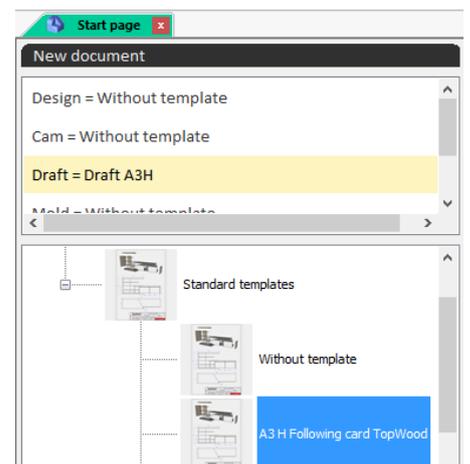
Right-clicking on a preview thumbnail allows the following actions:

- **Open** the document
- **Remove** the document from the list of recent documents
- **Open** the file's Windows **directory**
- **Empty** the whole list of recent documents

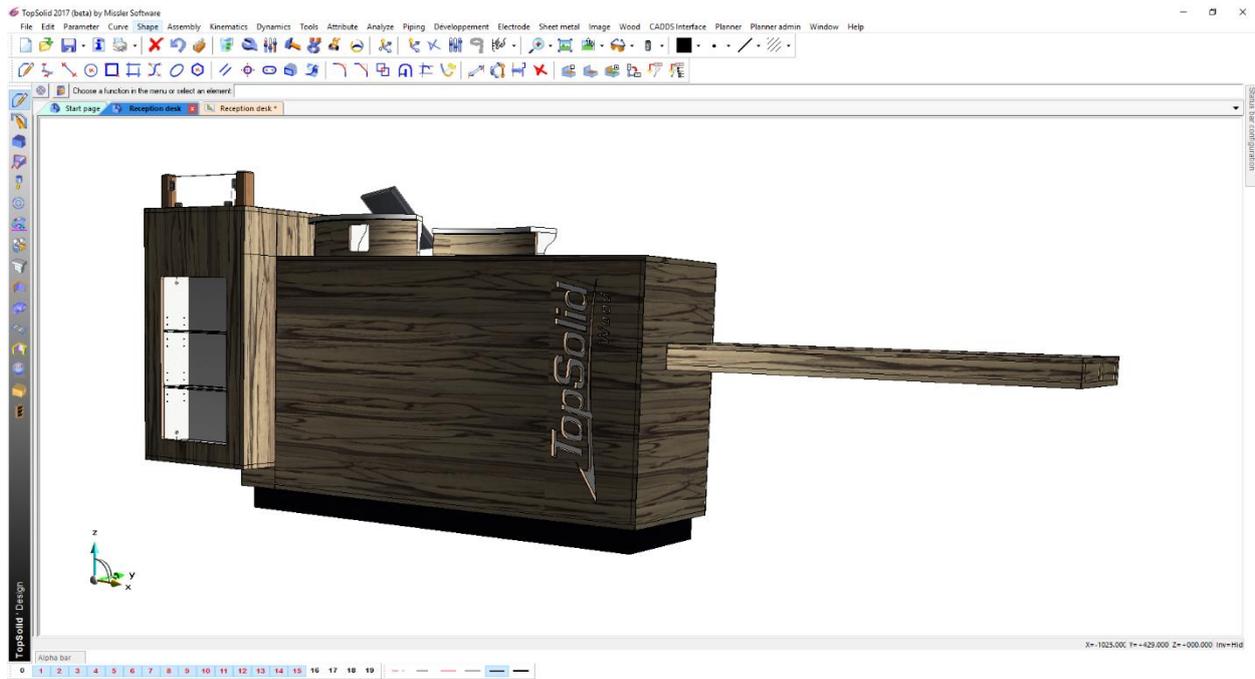


For new documents, the different types available (design, drafting...) are shown on the left of the **start page**.

The template documents are grouped in the lower section. Double-clicking on a document type or a template name creates a new document.

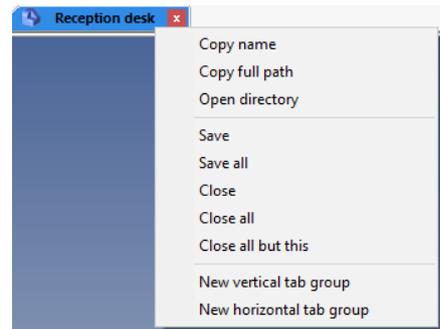


All open documents are now displayed in tabs.

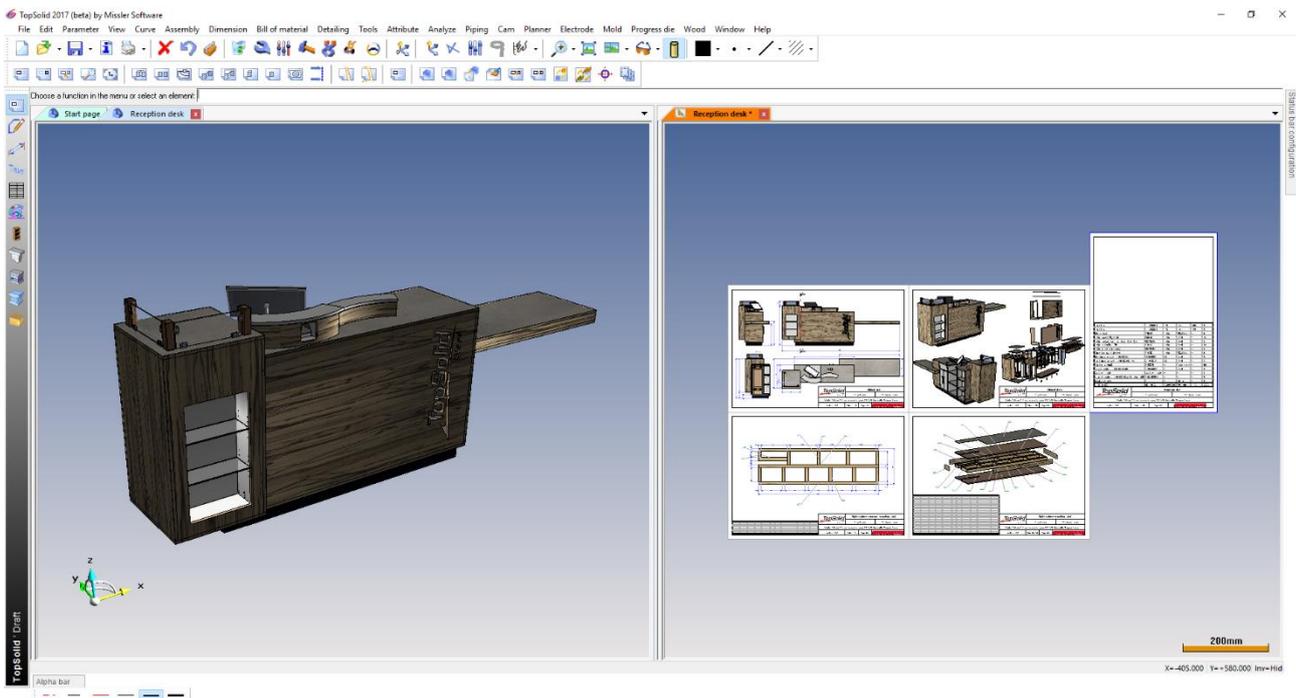


Right-clicking on a tab provides access to various functions:

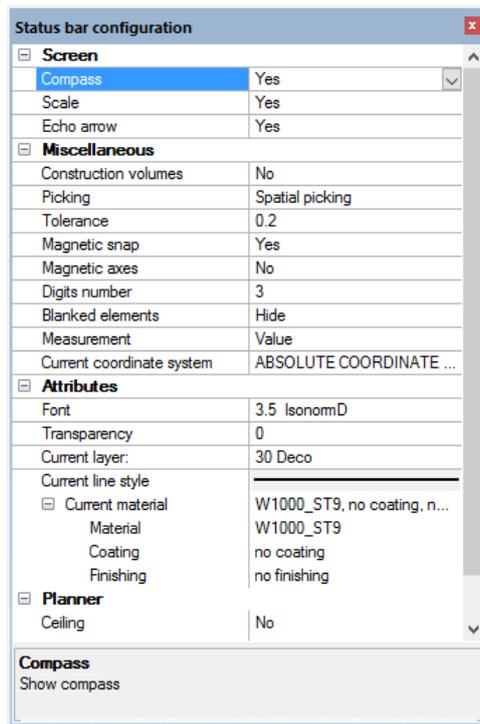
- Copy the file name
- Copy full path
- Open directory
- Save
- Save all
- Close
- New vertical tab group
- Move to previous/next tab group
- Back to one group



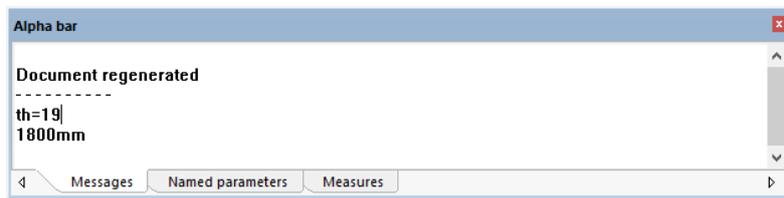
Note: The **Move to previous/next tab group** and **Back to one group** functions are available only when there are at least two groups on the screen. Below is an example of a two-group display.



The **Status bar configuration** dialog box includes all the settings that were previously available in the lower section of the graphics area (**Current material, Transparency, Current coordinate system...**).



The **alpha bar** is now divided into three tabs to make it easier to find the information you need. The **Messages** tab includes all the messages displayed (as was the case with the alpha bar previously). The **Named parameters** tab displays the name and value of the parameters created or changed by the user. The **Measures** tab contains the values of the measurements carried out from the **Analysis** menu.

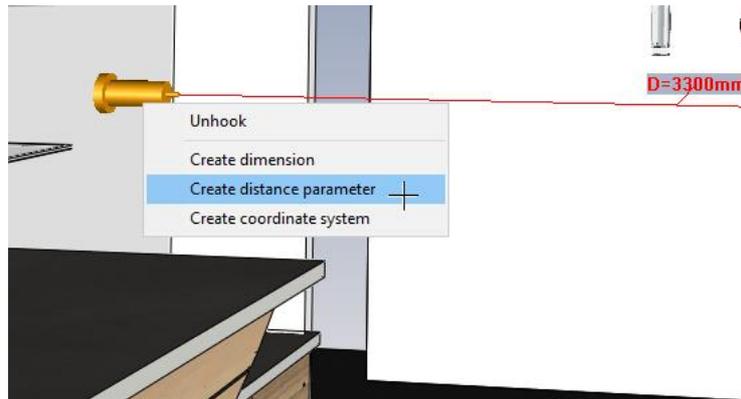


You can change the position and size of the **alpha bar** and the **status bar configuration** by clicking and dragging the dialog box's title bar (like a Windows dialog box).

TopSolid provides a **measurement tool** based on the scale. To use it, you have to **click and drag** the sphere from the scale at bottom right to a flat face. This measures the distance to the opposite plane.



Once the measure is displayed, you can **right-click** on the symbol to bring up a contextual menu that lets you **create the dimension** displayed, **create a distance parameter** that can be reused later, or **create a coordinate system** on the selected face. The **Unhook** option closes the measurement tool.



Design

New sketch constraints

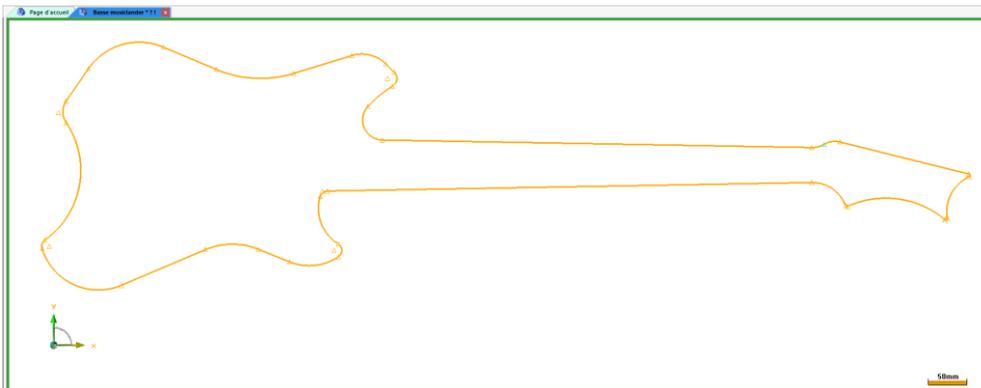
Two new **fixity** and **rigidity** constraints have been added to the sketch.

Fixity constraint

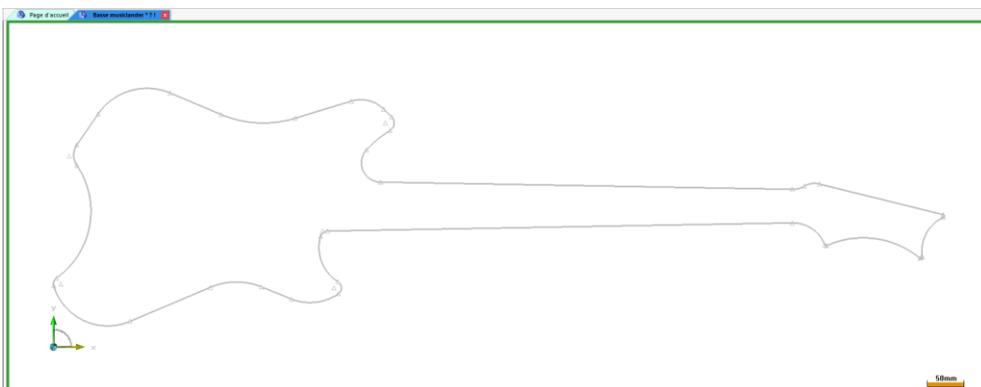
The **fixity** constraint lets you fix sketch elements at their position without having to create dimensions.

- In a sketch, select **Constraint > Fixity**.

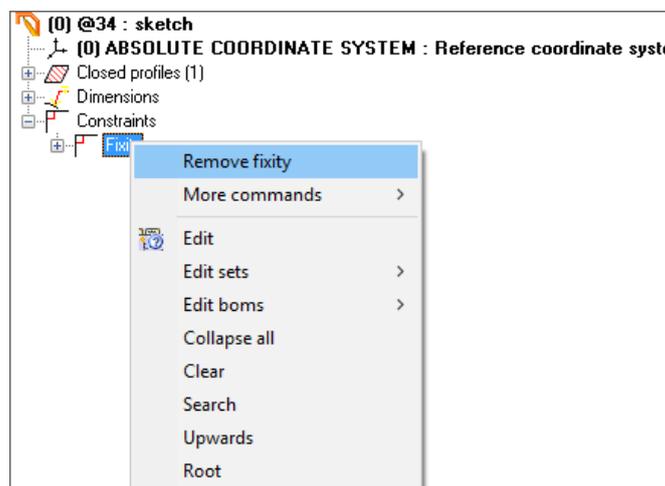
In the sketch below, the drawn contour is not constrained.



- Select the elements you want to fix.
- Once selected, the fixed elements turn gray and they cannot be moved anymore.



- To remove a fixity, launch **Constraint > Fixity** and pick the elements you want to unfix. You can also remove it by editing the sketch from the tree, and then **right-clicking** and selecting **Remove fixity** on the constraint to be deleted.

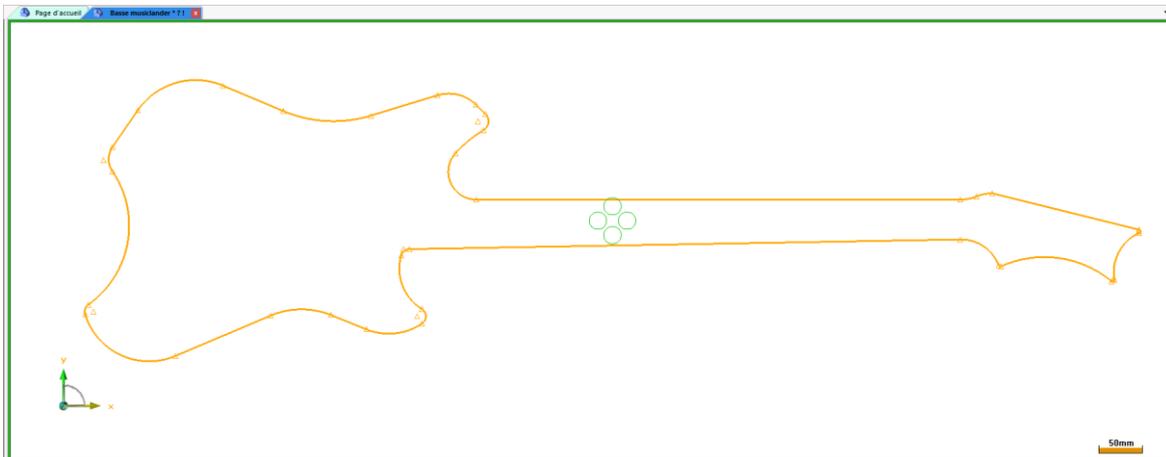


Rigidity constraint

The **Rigidity** constraint is used to group sketch elements so that they can be moved together for example.

- In a sketch, start **Constraint > Rigidity**.
- Select the components you want to group.
- When you're finished, click **STOP**.

The rigidity constraint is shown by the symbol  .



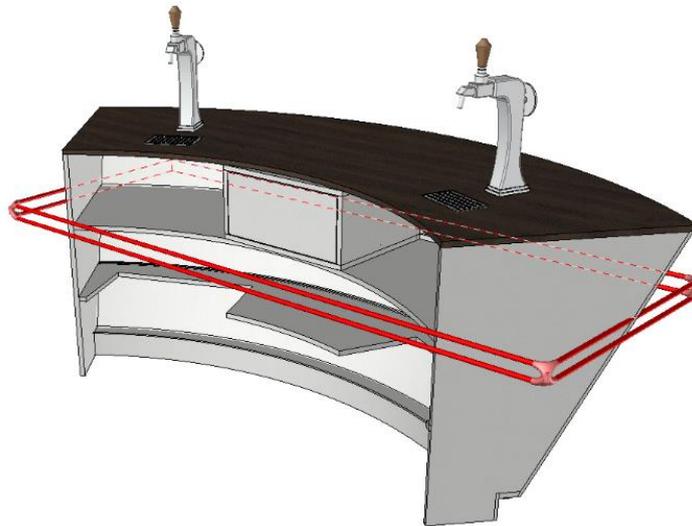
- To remove **rigidity**, use **Delete element** and click the constraint symbol. You can also remove it by editing the sketch from the tree, and then **right-clicking** and selecting **Delete** on the rigidity constraint.

Rough nesting

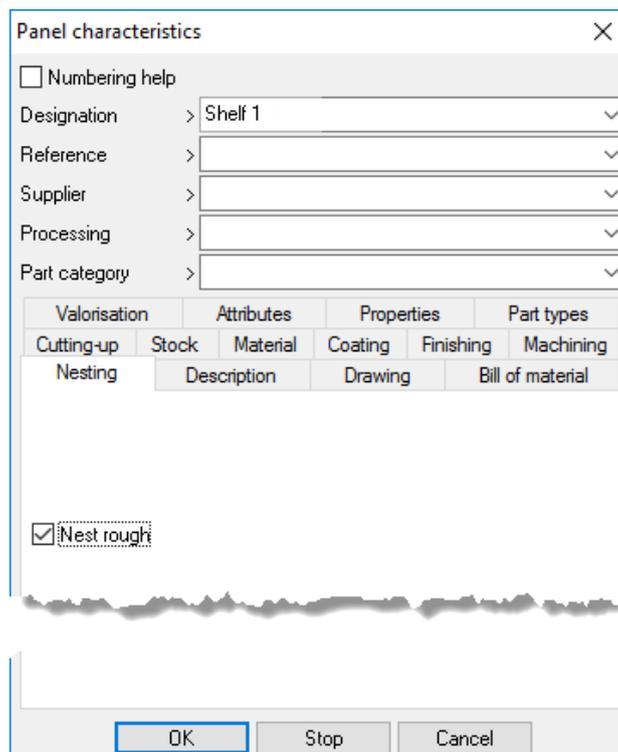
You can nest the rough of parts during **nesting**.

- In a document, select **Wood > Define part**.
- Click the part with the rough to be nested.

In the following example, the rough of the two shelves will be nested.



- In the **Nesting** tab, select the **Nest rough** box.

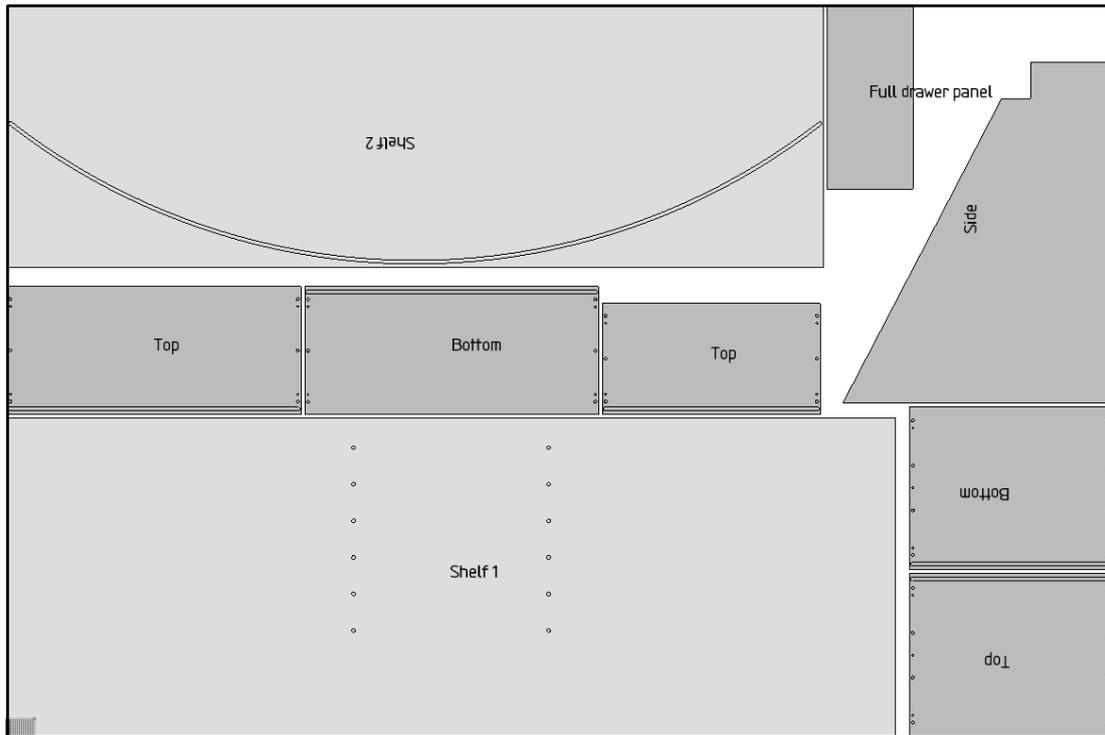


Note: If you want this option selected by default, select **Tools > Options > TopSolid'Wood configuration > Define parts** and open the **Nesting** tab where you can specify the default setting of the parts and the panel support.

- Select **Wood > Automatic nesting**.
- Pick the parts to be nested.
- Configure the nesting settings.

After the nesting is generated, the parts with nested rough appear transparent.

In the example below, the counter's parts have been nested and the **Shelf 1** and **Shelf 2** parts are transparent.

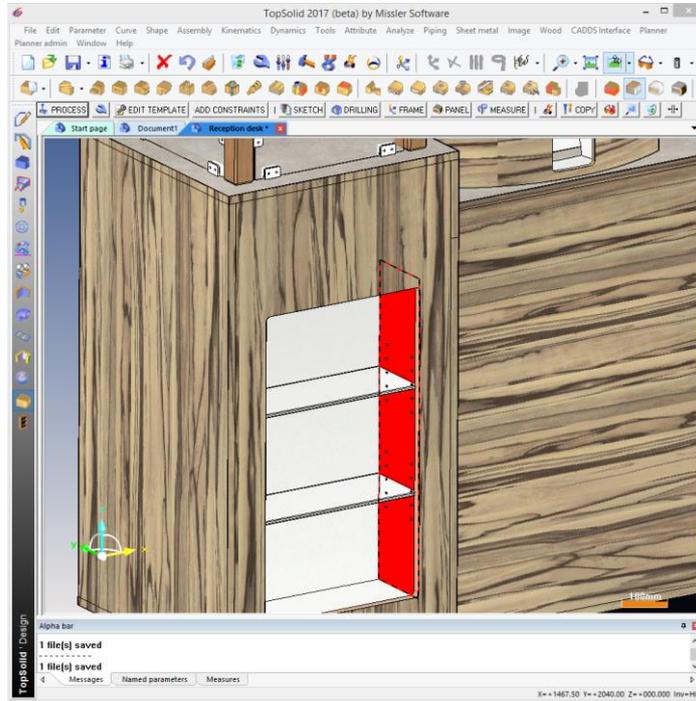


Note: When the rough of a part is positioned, only the operations performed on the top face are retrieved during nesting (one groove and drillings in the example).

Wood machining

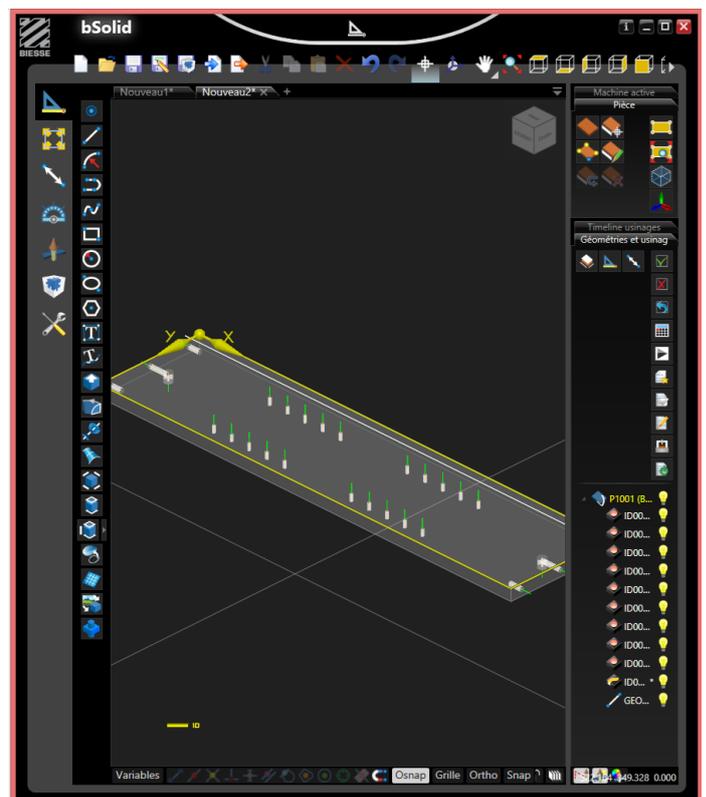
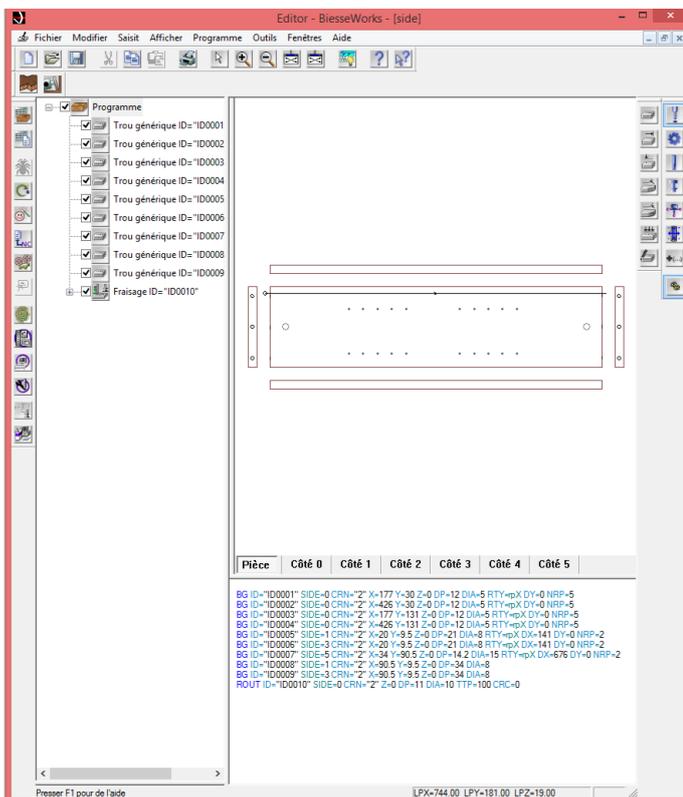
Biesse CIX interface

A new machining interface includes the ability to export parts designed in **TopSolid'Wood** to CIX format. The resulting file contains all the part's machining information (toolpath, position, tools...) and can be opened with **BiesseWorks** (version 3.4.14 or later) and **BSolid**.



BiesseWorks

BSolid



Bill of material

Generic codification of edges and laminates

You can define a **generic codification** for edges and laminates. These codification rules predefined upon configuration are then applied to edges and laminates.

This lets you quickly create codifications without the need to assign a codification to each edge type.

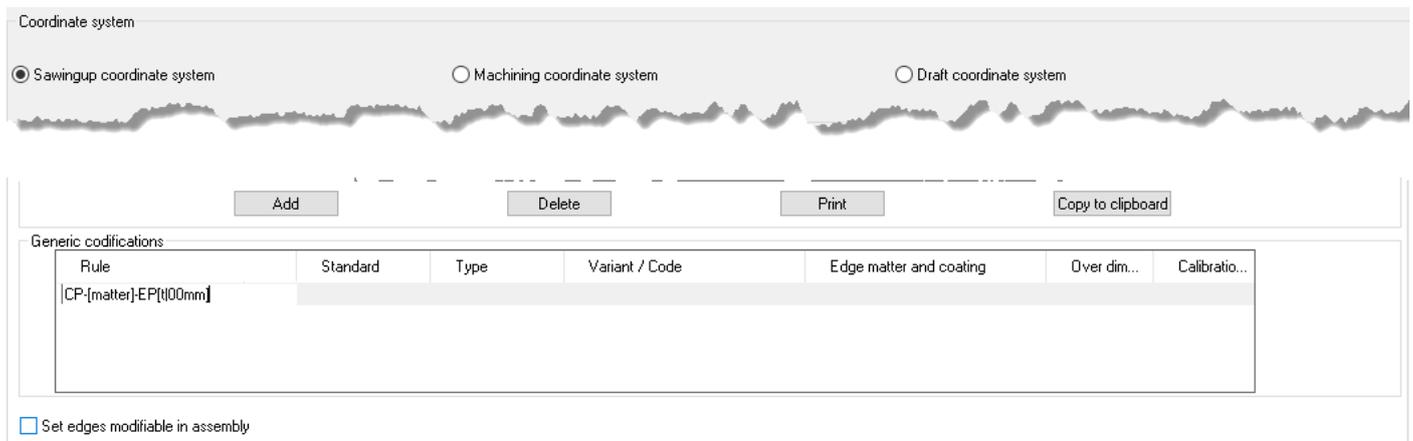
Creating a generic codification

- To create an **edge generic codification**, select **Tools > Options > TopSolid'Wood configuration > Edge/Laminate**.
- In the **Generic codifications** section of the **Edge configuration** tab, double-click in the **Rule** column to create a new codification rule.
- Enter the **generic codification**.

Note: Some of the **edge** or **laminates** parameters can be retrieved in the codification:

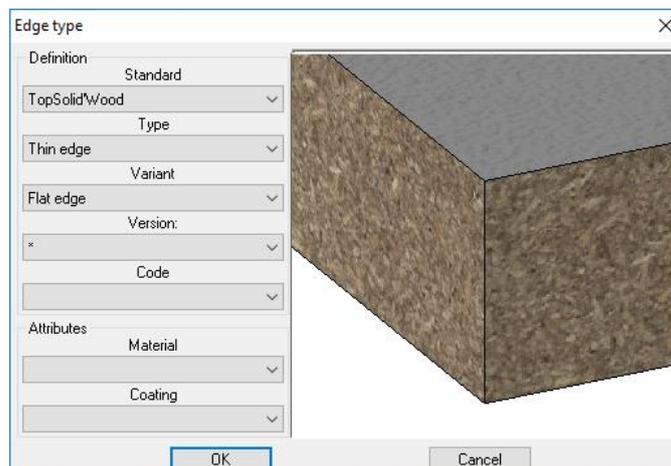
- Edge or laminate matter: [matter]
- Edge or laminate coating: [coating]
- Edge catalog code: [code]
- Edge or laminate thickness: [t]
- Edge length: [l]
- Edge height: [h]
- Laminate width: [w]

In the example below, the **CP-[matter]-EP [t|00mm]** codification is created. The **codification** will display the **matter** and **thickness** of the edge in **millimeters** and rounded to the nearest whole number.



- Next select the edges where you want to apply the codification by double-clicking in the **Standard** column.

In the following example, the codification will be applied to all **flat edges** of the **TopSolid'Wood** library.



Note: If no selection is made in one of the drop-down lists (Type, Variant, Version...), the codification will be applied to all edges of these categories.

- Specify the **over dimension length** and the **calibration overvaluation**.

In this example, no **over dimension length** is specified. For the **calibration overvaluation**, the **[cat:ep]** syntax retrieves the **ep** parameter value from the edge catalog. This ensures that the calibration overvaluation always matches the thickness of the edge used.

Rule	Standard	Type	Variant / Code	Edge matter and coating	Over dim...	Calibratio...
CP-[matter]-EP{t{00mm}}	TopSolid'Wood	thin edge	flat edge / *	* - *	0mm	[cat:ep]

Double clic to add a generic codifi...

- To create a **generic codification** for **laminates**, repeat the steps in the **Laminate configuration** tab.

In the example below, the **ST-[matter]** codification will be applied to all laminates using a matter from the **TopSolid'Wood** library's **Hardwoods** category, regardless of the coating or thickness.

Rule	Material	Coating	Thickness
ST - [matter]	TopSolid'Wood/ Hardwoods	*	*

Using generic codifications

- Start **Wood > Panel** and click a face of an existing part to create a new **Panel** entity.
- Select the **edge type** you want to apply by double-clicking in the **Edge type - code** column.

Edges and laminates creation wizard for panel

Edges						
N°	Codification	Edge type - code	Length	Beginning cut type	End cut type	
<input checked="" type="checkbox"/>	1	Flat edge - ep 2	800.0mm	Mitre cut	Mitre cut	
<input checked="" type="checkbox"/>	2	Flat edge - ep 2	500.0mm	Mitre cut	Mitre cut	
<input checked="" type="checkbox"/>	3	Flat edge - ep 2	800.0mm	Mitre cut	Mitre cut	
<input checked="" type="checkbox"/>	4	Flat edge - ep 2	500.0mm	Mitre cut	Mitre cut	

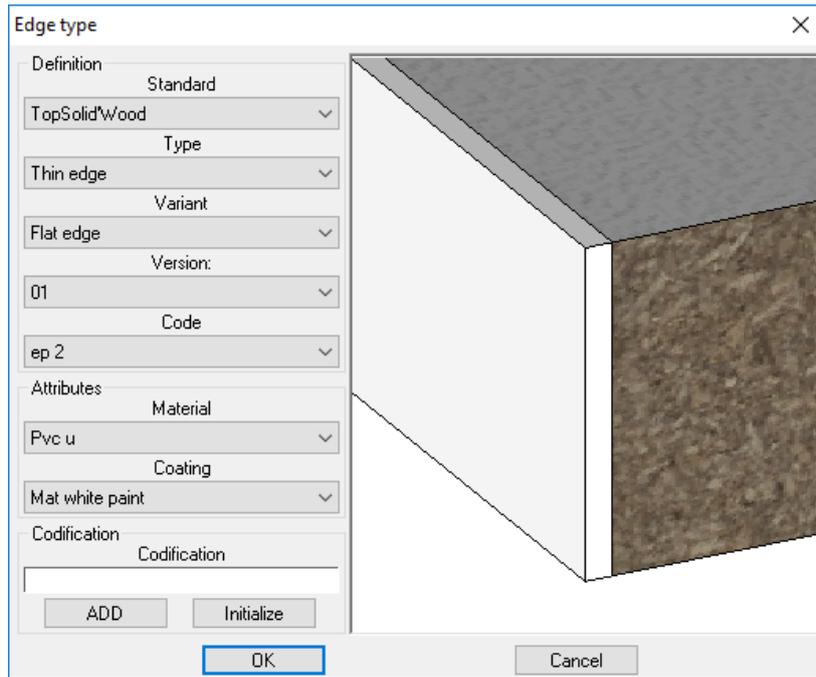
Edges are similar
 Cuts are similar

Laminate shapes				
N°	Codification	Material	Coating	Thickness
<input type="checkbox"/>	1	Ash olive		@82=1mm
<input type="checkbox"/>	2	Ash olive		@83=1mm

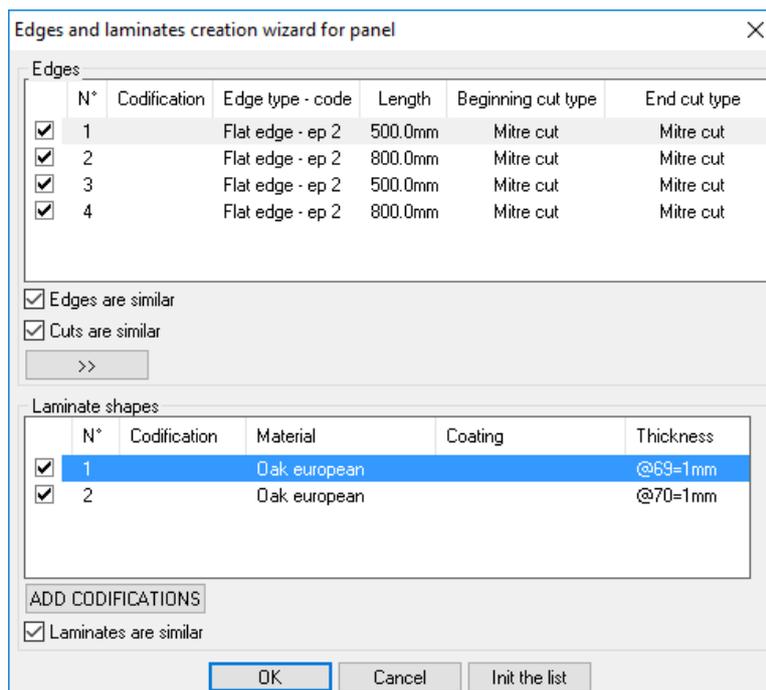
Laminates are similar

Note: In order for the **generic codification** to be applied, you need to select the edge to be applied directly. The codification is then automatically generated based on the parameters defined when creating the **generic codification**.

In the following example, a PVC flat edge from the **TopSolid'Wood** library is applied.



A 1mm thick oak laminate is also applied.



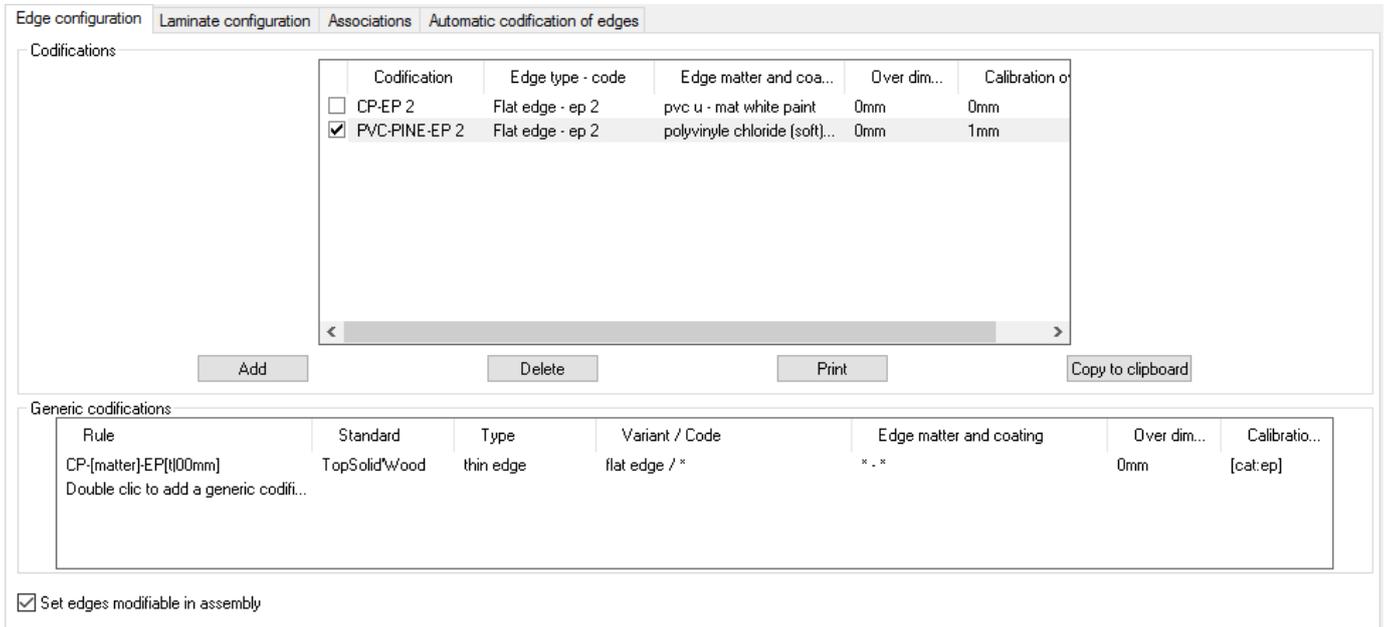
Using the generic codifications that were defined previously, here is the codification of the edges and the laminate displayed in a bill of material:

1	Panel	-	804.0	504.0	19.0	-
2	laminate shape	Oak european	800.0	500.0	1.0	ST-Oak european
2	flat edge	PVC P	500.0	19.0	2.0	CP-PVC P-EP2
2	flat edge	PVC P	804.0	19.0	2.0	CP-PVC P-EP2
1	Support	W1000_ST9	800.0	500.0	18.0	-
COUNT	DESIGNATION	MATERIAL	PART_LENGTH	PART_WIDTH	PART_THICKNESS	CODIFICATION

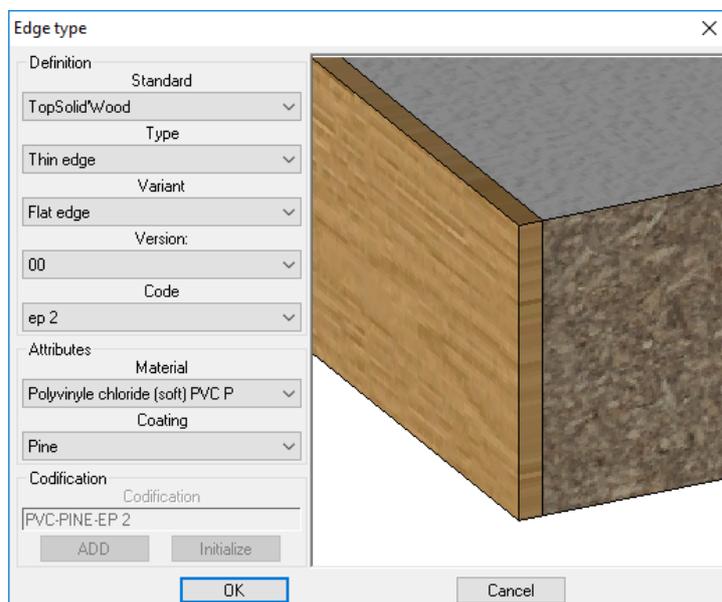
Special situations

- When there are codifications other than generic codifications, the codification assigned to the edge has priority over the generic codification.

In the example below, the 2mm thick PVC flat edge with pine coating uses the **PVC-PINE-EP 2** codification. The generic codification points to the flat edges too.



The flat edge with the **PVC-PINE-EP 2** codification is applied to a panel.



In the bill of material, you'll notice that the codification assigned to the edge is used, not the generic codification.

1	Panel	-	804.0	504.0	19.0	-
2	laminate shape	Oak european	800.0	500.0	1.0	ST-Oak european
2	flat edge	polyvinyle chloride (soft) PVC P	500.0	19.0	2.0	PVC-PINE-EP 2
2	flat edge	polyvinyle chloride (soft) PVC P	804.0	19.0	2.0	PVC-PINE-EP 2
1	Support	W1000_ST9	800.0	500.0	18.0	-
COUNT	DESIGNATION	MATERIAL	PART_LENGTH	PART_WIDTH	PART_THICKNESS	CODIFICATION

Note: To use only generic codifications, you need to delete the codifications assigned to the relevant edges.

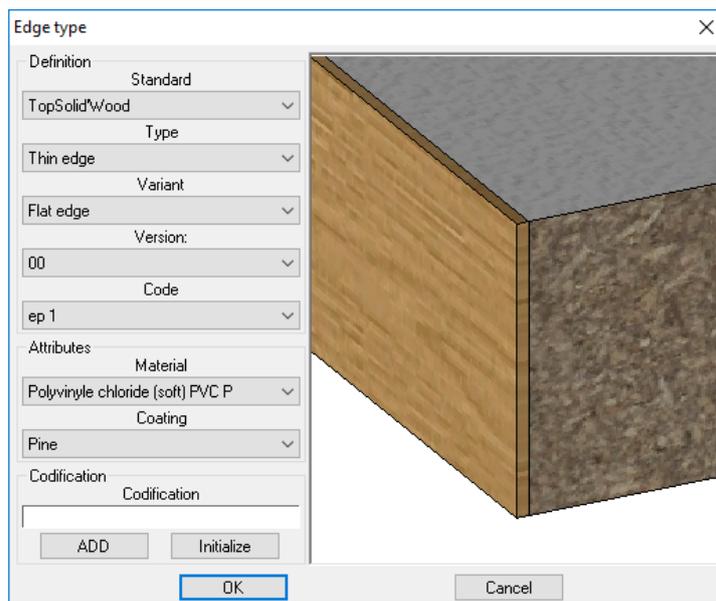
- When two **generic codifications** point to the same library, the list of **generic codifications** is read from top to bottom and **TopSolid** will use the first true codification.

In the following example, the two generic codifications point to the same **TopSolid'Wood** library. The first one is applied to edges from the **Thin edge/Flat edge** category, and the second one is applied to all edges from the **Thin edge** category.

Rule	Standard	Type	Variant / Code	Edge matter and coating	Over dim...	Calibratio...
CP-[matter]-EP[t 00mm]	TopSolid'Wood	thin edge	flat edge / *	* . *	0mm	[cat:ep]
EDGE-[matter]	TopSolid'Wood	thin edge	* / *	* . *		

Double clic to add a generic codifi...

A flat edge from the **Thin edge** category is applied to a panel. In this case, the two generic codifications can work as they both point to the **Thin edge** category.



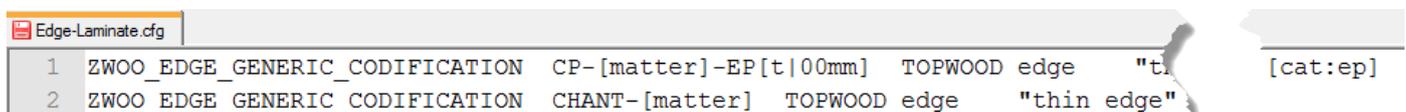
In the bill of material, you'll notice that the first codification of the list has been used.

1	Panel	-	802.0	502.0	19.0	-
2	laminate shape	Oak european	800.0	500.0	1.0	ST-Oak european
2	flat edge	polyvinyle chloride (rigid) PVC U	500.0	19.0	1.0	CP-PVC U-EP1
2	flat edge	polyvinyle chloride (rigid) PVC U	802.0	19.0	1.0	CP-PVC U-EP1
1	Support	W1000_ST9	800.0	500.0	18.0	-
COUNT	DESIGNATION	MATERIAL	PART_LENGTH	PART_WIDTH	PART_THICKNESS	CODIFICATION

If the order of generic codifications was reversed, the second codification would be used. Below is the BOM result after reversing the generic codifications in the list.

1	Panel	-	802.0	502.0	19.0	-
2	laminate shape	Oak european	800.0	500.0	1.0	ST-Oak european
2	flat edge	polyvinyle chloride (rigid) PVC U	500.0	19.0	1.0	EDGE-PVC U
2	flat edge	polyvinyle chloride (rigid) PVC U	802.0	19.0	1.0	EDGE-PVC U
1	Support	W1000_ST9	800.0	500.0	18.0	-
COUNT	DESIGNATION	MATERIAL	PART_LENGTH	PART_WIDTH	PART_THICKNESS	CODIFICATION

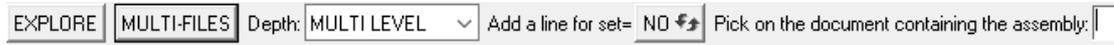
Note: To change the order of generic codifications, open the **Edge-Laminate.cfg** file from the **Group/V6X** folder. Next, change the order of lines beginning with **ZWOO_EDGE_GENERIC_CODIFICATION**.



BOM export to xlsx file

You can export a bill of material to an **Excel** file that contains macros.

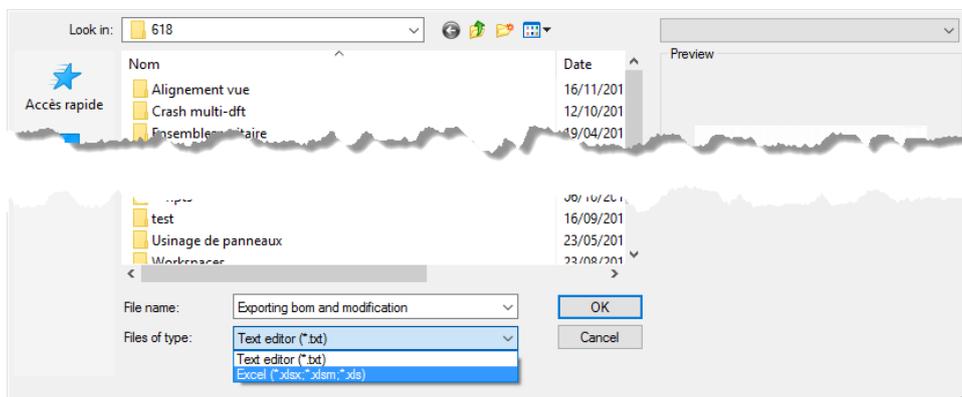
- Start **Wood > Export bill of material**.
- Select the **BOM template** you want to use.
- Specify the BOM depth and select the document containing the **set to export**.



- Select a **filter by criteria** if necessary.
- Adjust the **Use an Excel template = NO** option to **YES**.
- Select the xlsx file that will be used as a template.



- Click **NAVIGATOR** to specify the name of the file and where to save it.
- From the **Files of type** drop-down list, select the **Excel** format (*.xlsx; *.xlsxm; *.xls).



- **Name** the file and click **OK** to confirm.
- **Select** the parts to be exported and click **OK** to confirm.

The file is then generated in xlsxm format.

Numbering by type of part

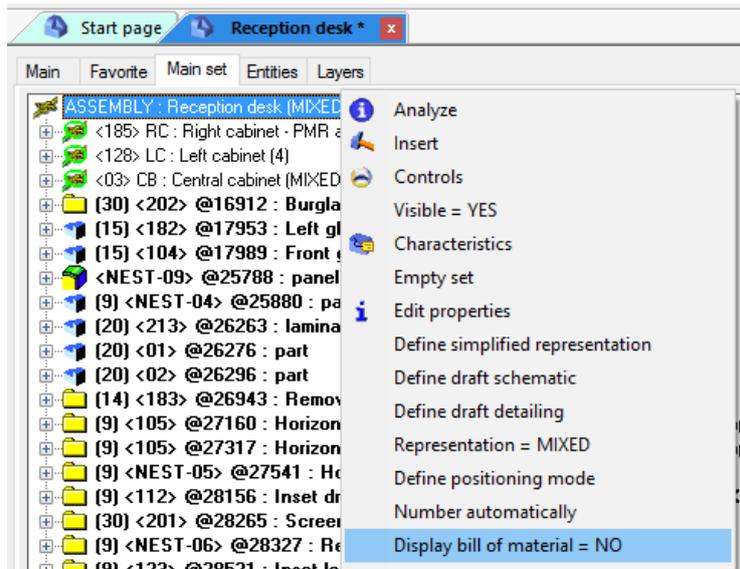
When you number a bill of material in **Use element type** mode, you are now able to **number parts without type**. In previous releases, you had to launch the **Numbering** function twice to achieve this.

- In **Tools > Options > Predefined index**, define BOM indexes for each part type.

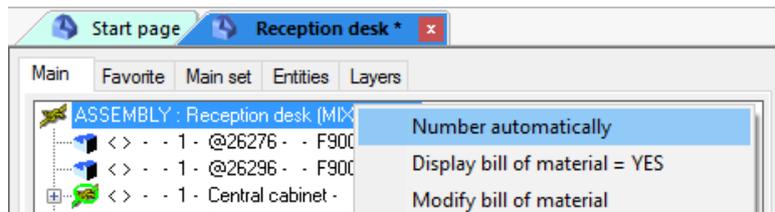
In the example below, the **NEST-** prefix is assigned to the **Nesting** part type.



- In a **Design** document, **right-click** on the **main assembly** line and select **Display bill of material**.

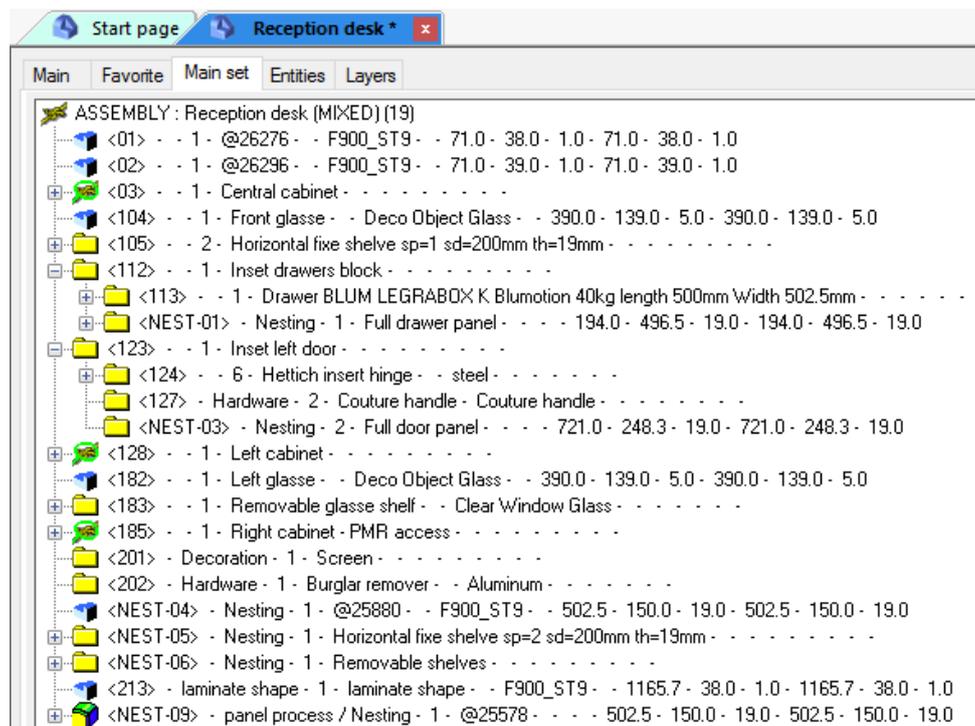


- Select a **BOM template** and click **OK** to confirm.
- Once you can see the bill of material, **right-click** on the **main assembly** line and select **Number automatically**.



- Choose **Use element type** as the **numbering mode** and set the **Number parts without type** option to **YES**.
- When you're finished completing the settings, click **OK** to confirm.

All the BOM's parts are numbered and the **Nesting** parts have a number starting with **NEST-**.



Configuration

Script documents

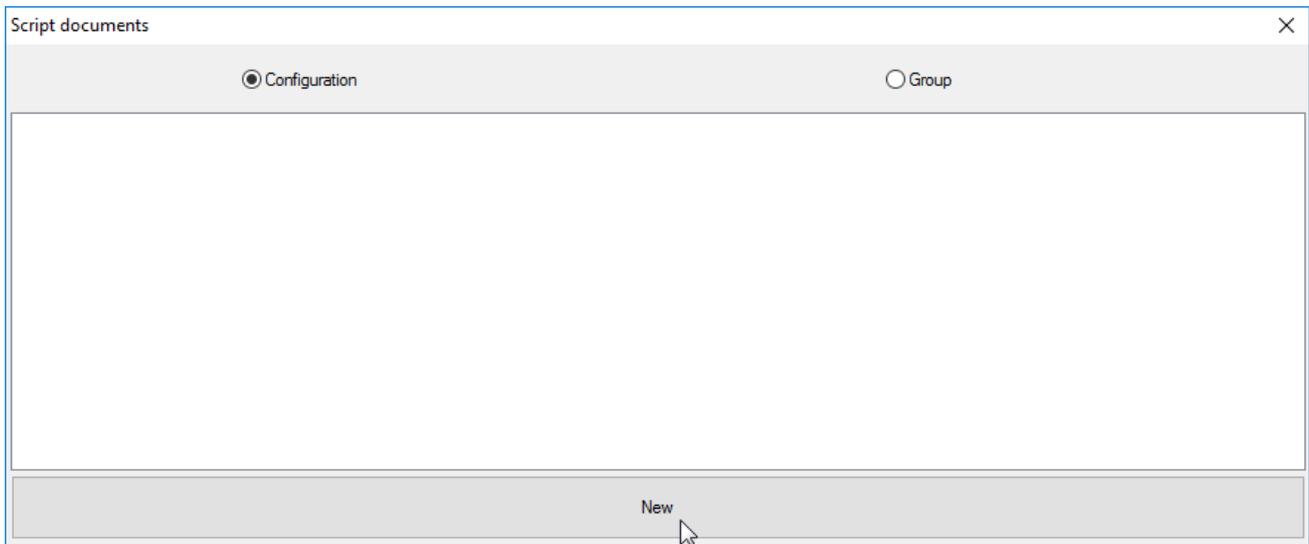
Configuration improvements include the ability to automate repetitive tasks using **Script** documents. This makes it possible to execute and string together various actions automatically.

Creating a script

- To create a new **Script** document, launch **Wood > Script**.

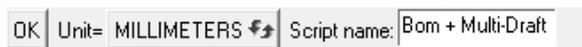
The **Script documents** dialog box opens.

- Choose whether the **script** must be created in the **local configuration** or the **group configuration**.
- Click **New**.



Note: The **Script** documents have the **.topscript** extension and are saved in the **Config/Template/Scripts** or **Group/Template/Scripts** folder.

- Select the **unit** to be used in the **Script** document and give the script a **name**.

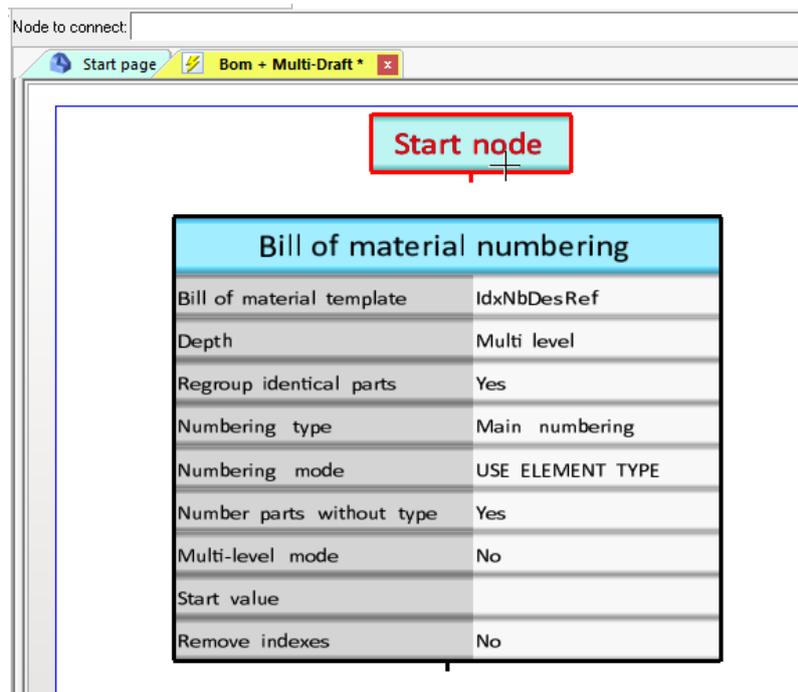


- Add the tasks to be executed by the **script**. You can use the following features:
 - **BOM numbering**
 - **Cut export**
 - **BOM export**
 - **Multi-drafting**
 - **Machining export**
 - **Multi-machining**

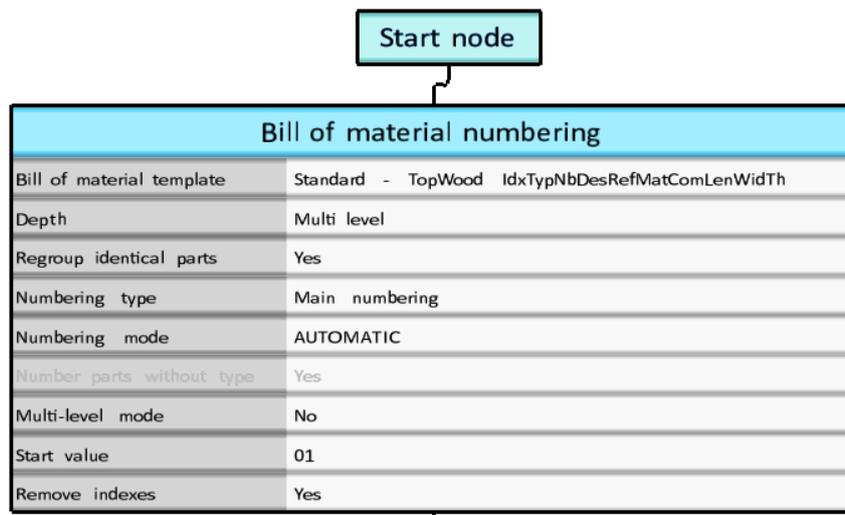
Note: Each task is shown by a node containing all the necessary settings for each function. These nodes are linked with each other and will be executed in order from top to bottom.

For example, here is the procedure on how to create a **script** in order to do **numbering**, **multi-drafting**, and then a **WoodWop export**.

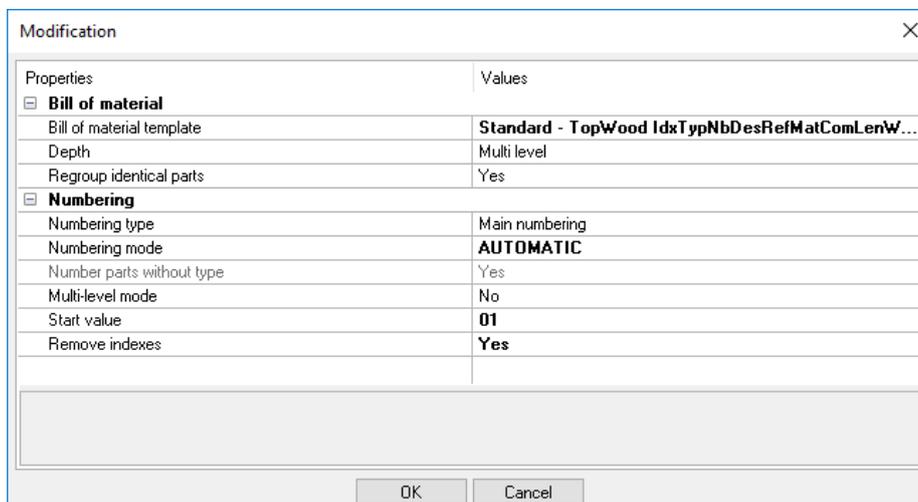
- In a new **Script** document, start **Wizard > Numbering**.
- The node is attached to the cursor. Click the blank page to position the node.
- The **Node to connect** option lets you choose the node after which the task being created will be executed. As there is yet no node, click **Start node**.



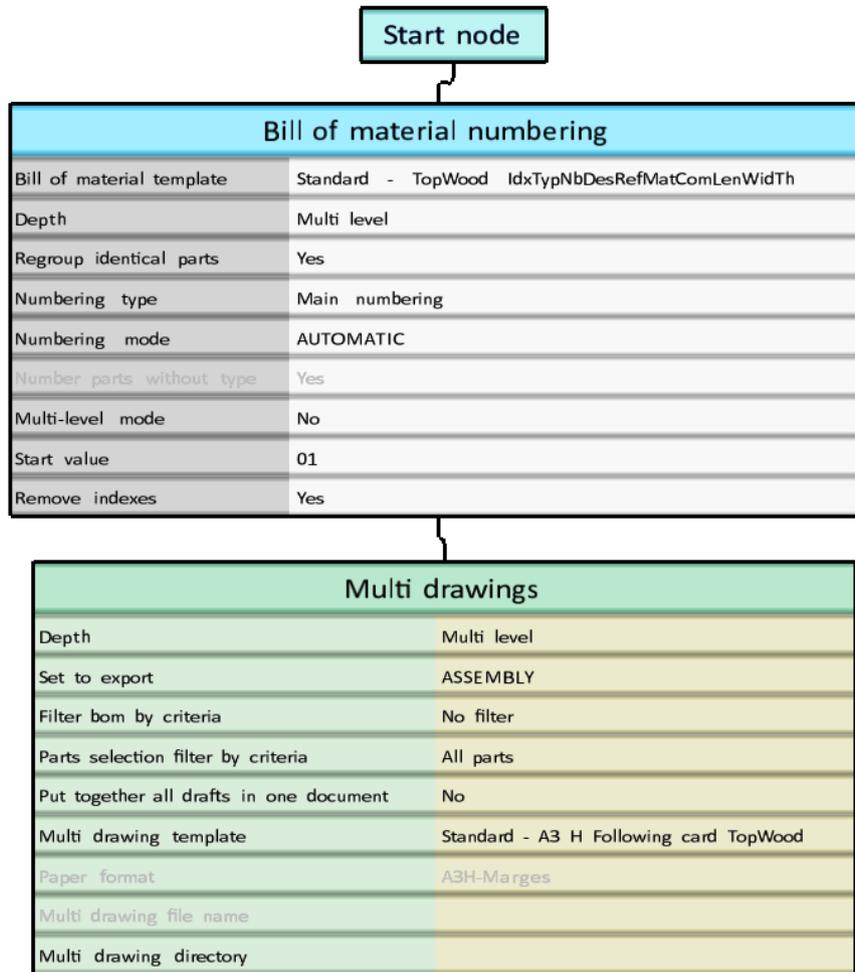
Result:



- To configure the **numbering**, select **Modify element** and click the created node.
- Specify the **numbering** parameters.



- Click **OK** to confirm.
- Start **Wizard > Multi-draft**.
- Place the node on the document and click the **Bill of material numbering** node to produce multiple drawings after numbering.



- Use **Modify element** on the created node in order to configure the multi-drawings.
- Specify the BOM depth for the part selection.
- Select the **elements to export**.

You can export parts from any set of the document (alternative set or in-place subset) by entering the name of the set to be used in the **Set to export** field. You can also filter the bill of material and the part selection using **multi criteria filters**. If no filter is used, all parts contained in the set will be exported.

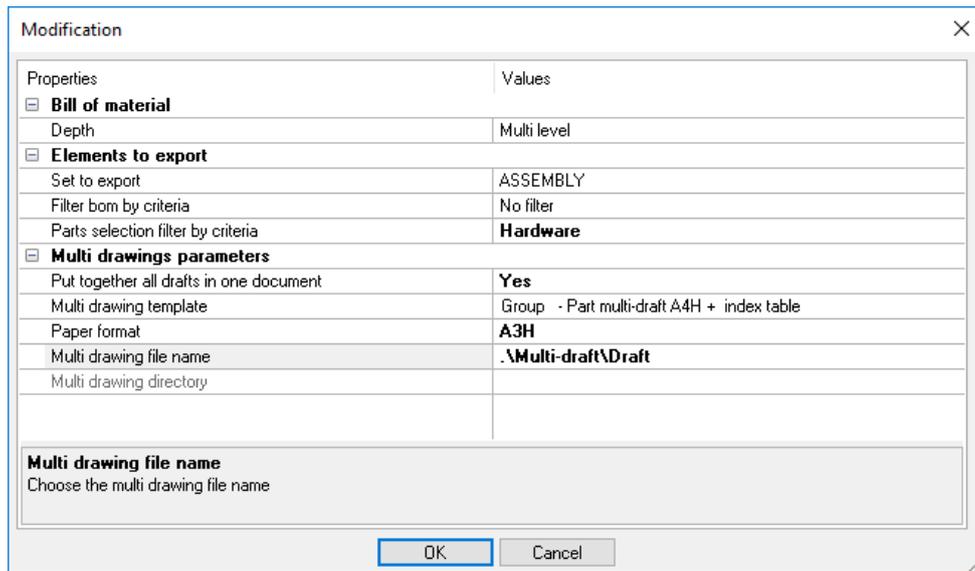
Note: These settings are available in the nodes of every function that requires part selection (machining export, sawing-up export, etc.).

- Configure the **multi-drawing parameters**.

Note: When all the drawings are placed on the same document, you need to define the **multi drawing file name**. This file will be saved in the same location as the document from which the **script** has been run. However, you can add subfolders using the syntax **./folder name/draft name** before the file name.

If a file is generated for each drawing, you need to specify a generation directory. The file names will be given by the **WOO_DFT_FILE_NAME** used in the BOM file specified in **TopSolid'Wood configuration > Files > Parts selection sorting > Multi-draft**.

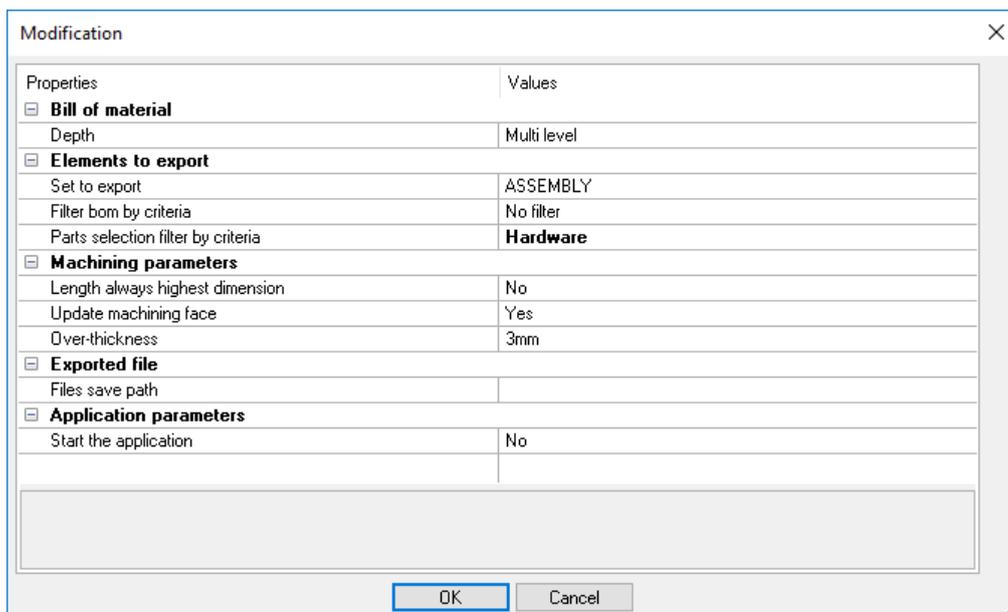
In this example, the **multi-draft** is configured to export all parts except the **Hardware** ones. All drawings will be placed in the same **Drawings** file that will be saved in a **Drafting** subfolder created in the same location as the document from which the **script** will be run.



- Click **OK** to confirm the settings.
- To add the **WoodWop** export, select **Wizard > Machining export > WoodWop**.
- Place the node in the document.
- Click the **Multi-drawings** node to perform the export after multi-drafting.
- Configure the **WoodWop** export.
- As with multi-drafting, you can filter the bill of material and the part selection.

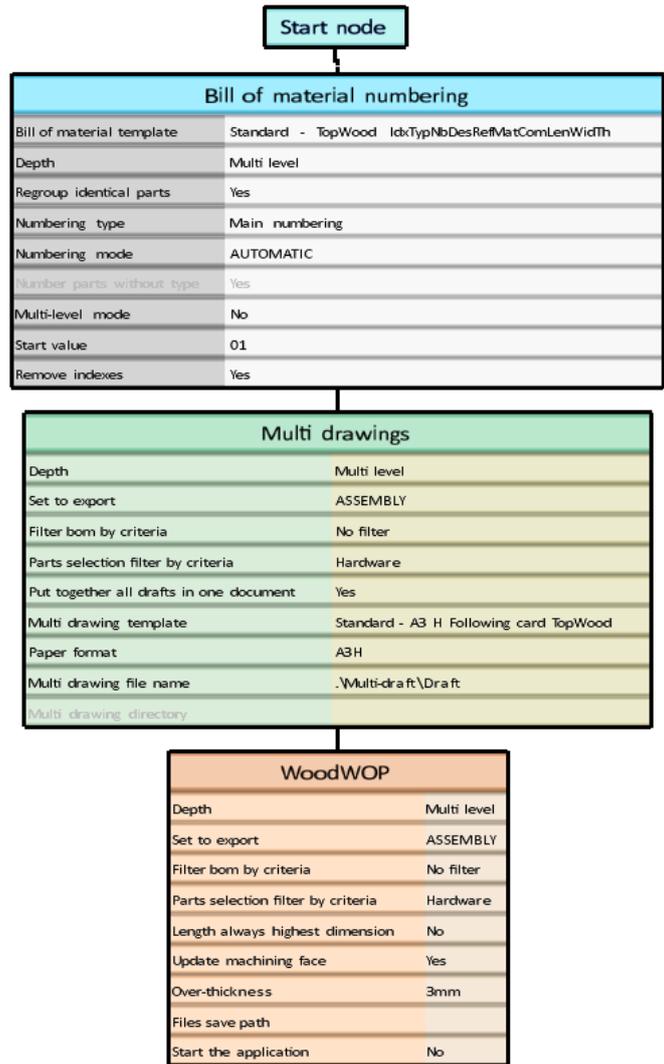
The settings available in **Machining parameters** are those available during a **WoodWop** export performed from the **Wood > Machining export > WoodWop** function.

In this example, all parts except the **Hardware** ones contained in the main set will be exported. The other settings keep their default values.



Note: For **machining exports**, the generated files are automatically saved to the location defined in **Tools > Options > TopSolid'Wood configuration > Machining > WoodWop, Maestro**, etc. The file names will be formatted by the **WOO_CAM_FILE_NAME** used in the BOM file specified in **TopSolid'Wood configuration > Files > Parts selection sorting > WoodWop export**.

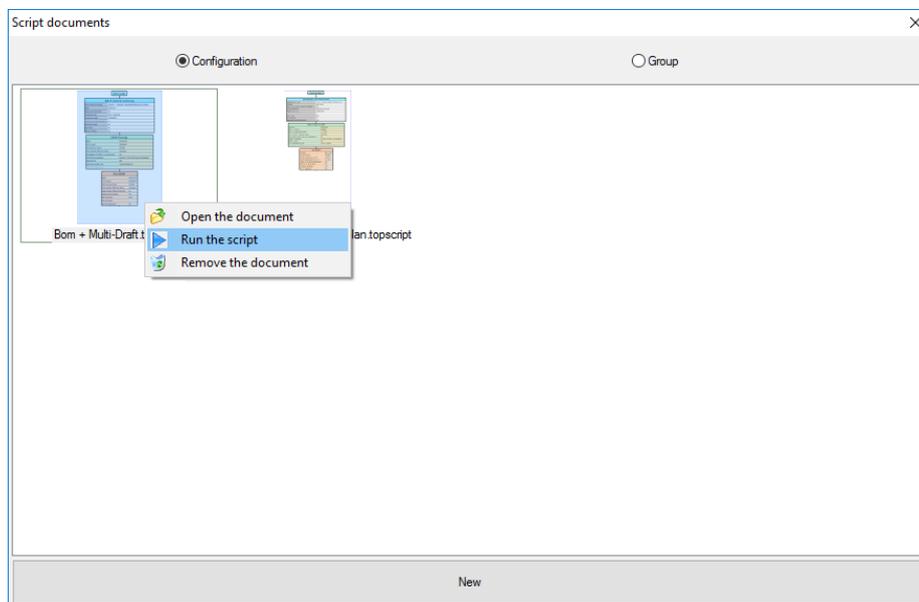
Once you have added and configured the **WoodWop** export, the following script document is created:



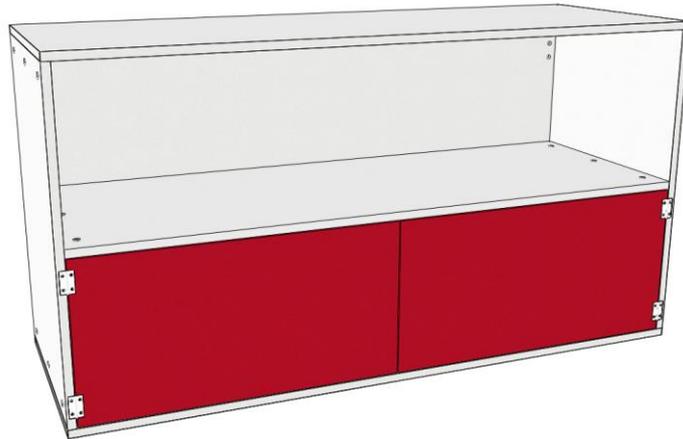
- **Save** the file.

Using the script

- To use a **script**, open a **Design** document of the project.
- To launch the **script**, **right-click** on the script to be executed and select **Run the script**.



In the example, the previously created **script** is executed on the following document.



- During the **script** execution, messages are displayed in the alpha bar which let you know whether the **script** has started, which node is being executed and when it is finished.

After launching the **script** in the example document, here are the messages displayed in the alpha bar for each step of the script:

Beginning of the **script**.

```
Alpha bar
Start of the script Bom + Multi-Draft.topscript
```

Bill of material numbering.

```
Alpha bar
Start of the script Bom + Multi-Draft.topscript
Bill of material numbering
```

Multi-drafting and the file containing all the drawings is saved.

```
Alpha bar
Start of the script Bom + Multi-Draft.topscript
Bill of material numbering
1 file(s) saved
Multi-Draft
```

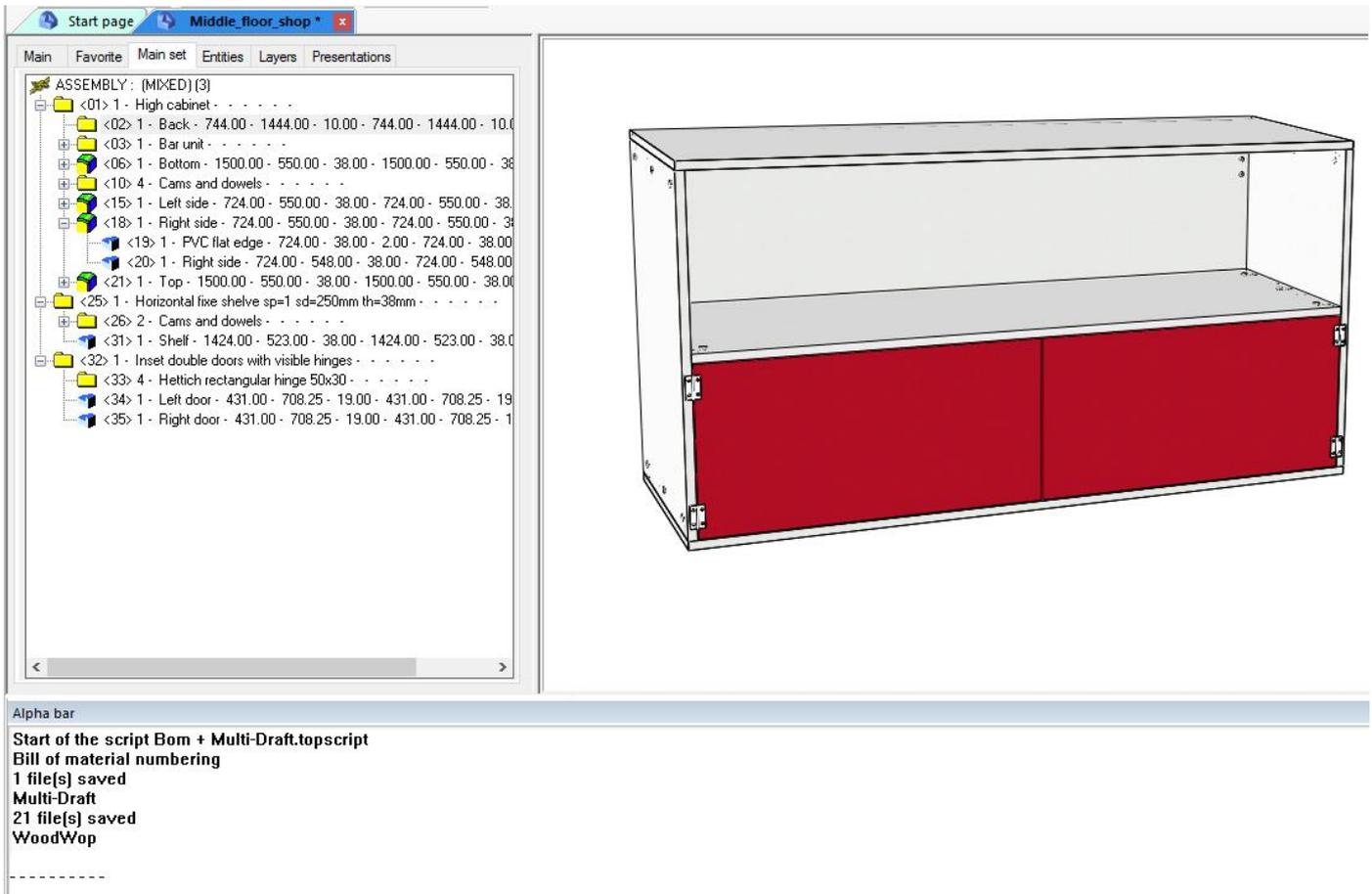
WoodWop export and number of generated files.

```
Alpha bar
Start of the script Bom + Multi-Draft.topscript
Bill of material numbering
1 file(s) saved
Multi-Draft
21 file(s) saved
WoodWop
```

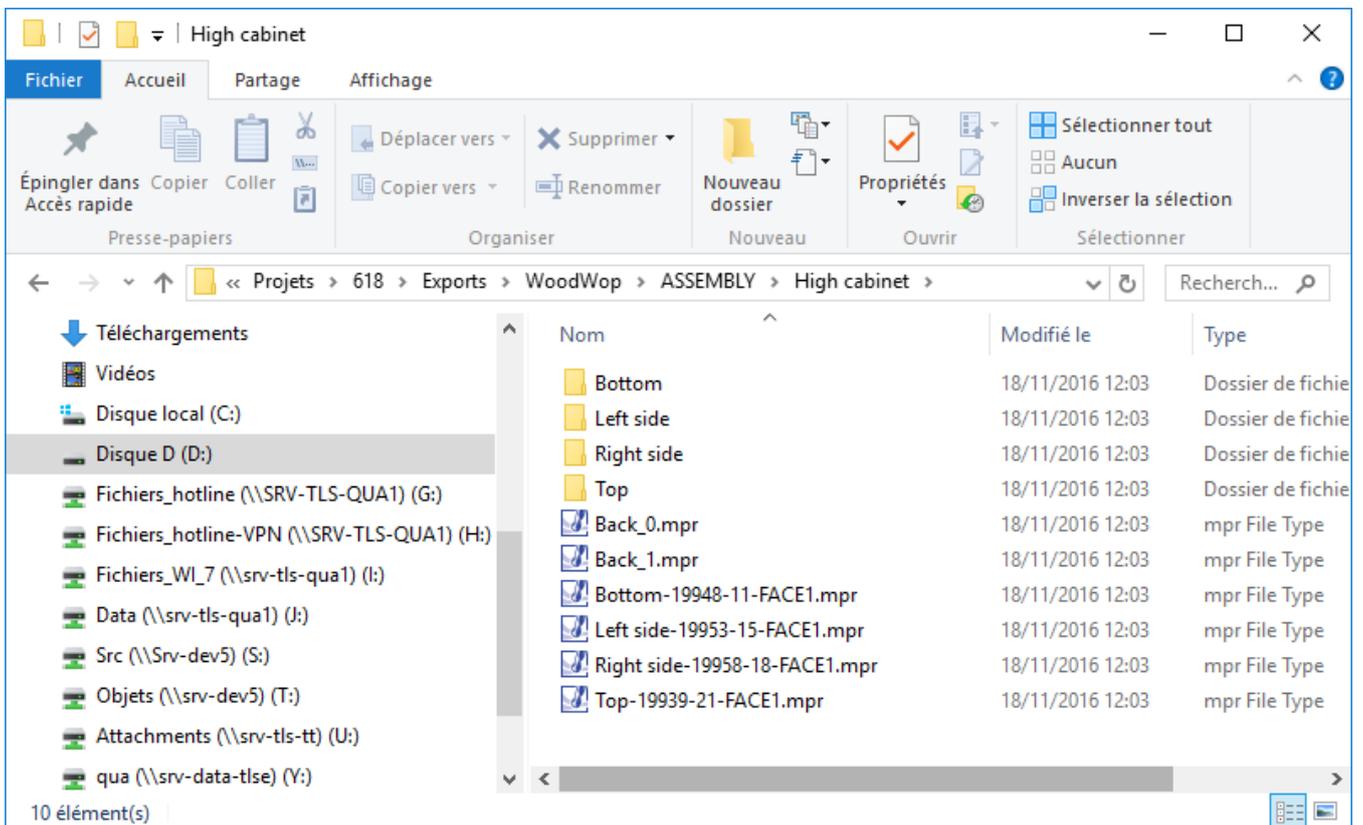
End of the **script**.

```
Alpha bar
Start of the script Bom + Multi-Draft.topscript
Bill of material numbering
1 file(s) saved
Multi-Draft
21 file(s) saved
WoodWop
End of the script Bom + Multi-Draft.topscript
```

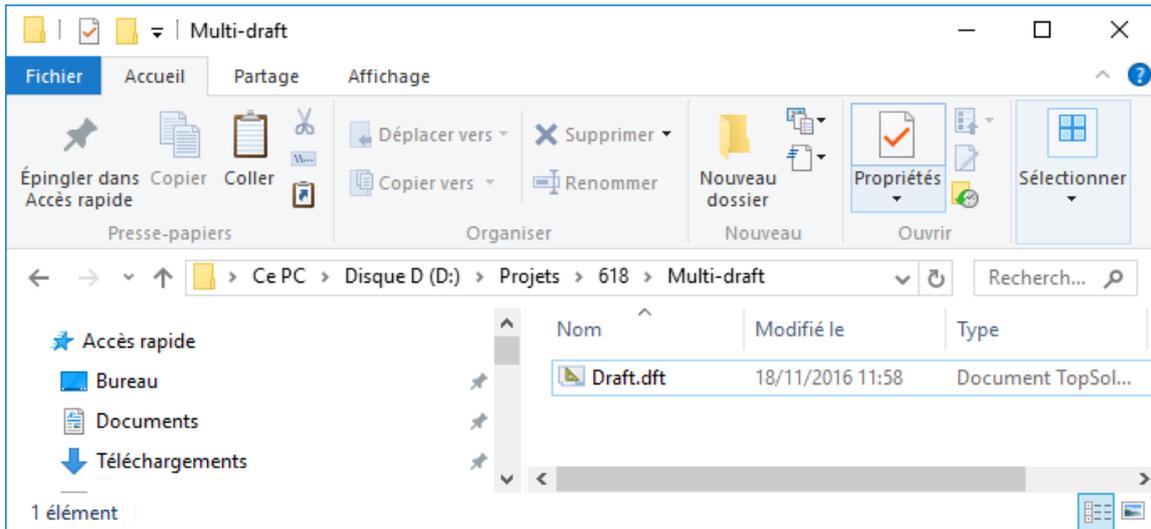
In the example document, you'll notice that the parts have been numbered and displayed as a bill of material in the tree.



The **WoodWop** files have been generated to your specified location.

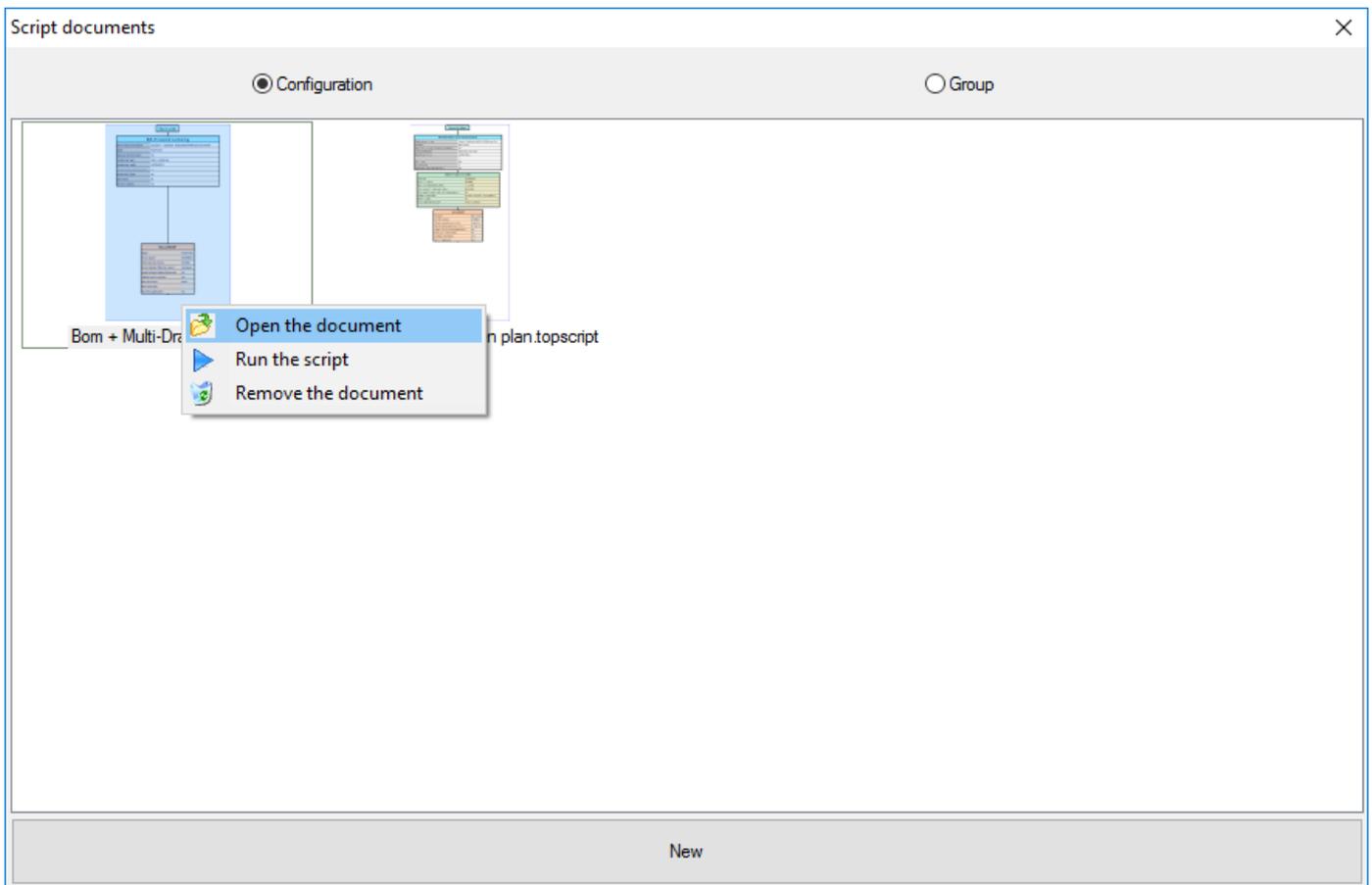


Similarly for the file generated by the **multi-drafting**.



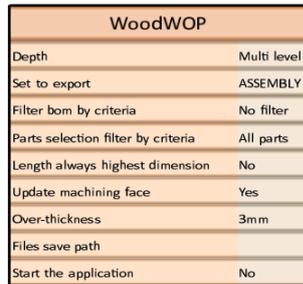
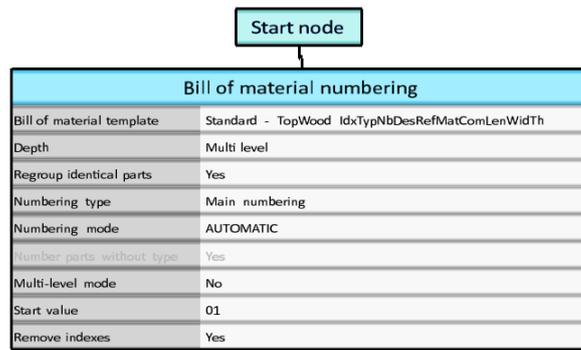
Modifying a script

- To modify an existing **script**, select **Wood > Script**.
- From the list of **scripts**, **right-click** and select **Open the document**.

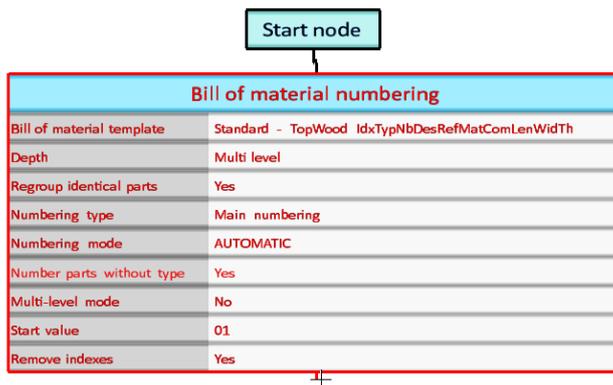


- Use **Modify element** or **Delete element** and click the node to change the settings or delete the node.

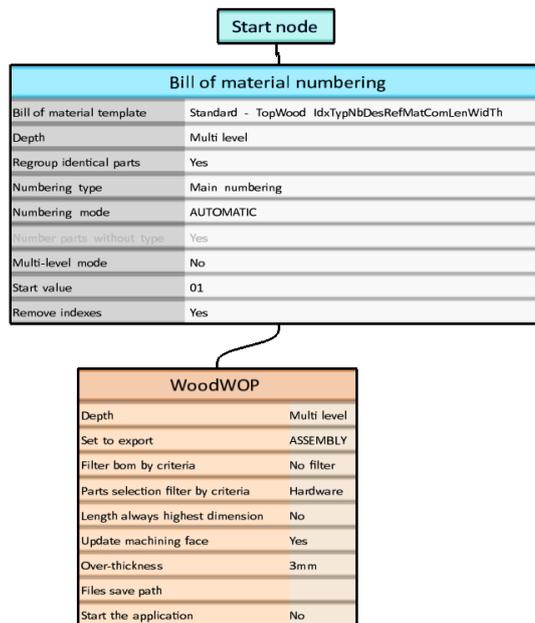
In the following example, the node which appeared in second place has been removed. In order for the **Script** document to work, you need to connect the two remaining nodes.



- To connect two nodes, select **Modify element**.
- Click the lower part of the node that'll need to be above the connected node.



- Click the **node to connect**.



This also has to be done when a new node must be inserted between two existing nodes.

Parameter edit dialog box

When you edit a parameter, a dialog box appears with which you can, for example, modify dimensions more intuitively.

- Edit a parameter.

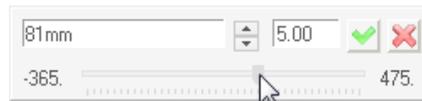
The dialog box for editing parameters opens.



- You can then modify the parameter in one of the following ways:
 - By typing the desired value into the appropriate box directly.
 - By entering a modification step and clicking the arrows to vary the step value.



- By sliding the cursor along the bar to quickly vary the values.



Note: Select **Tools > Options > Other** to display or hide this dialog box and change its opacity.

Drafting

Drilling table and index table in multi-drawings

You can include a **drilling table** and/or an **index table** in the **multi-drawing template**. These tables are then automatically completed based on the projected element.

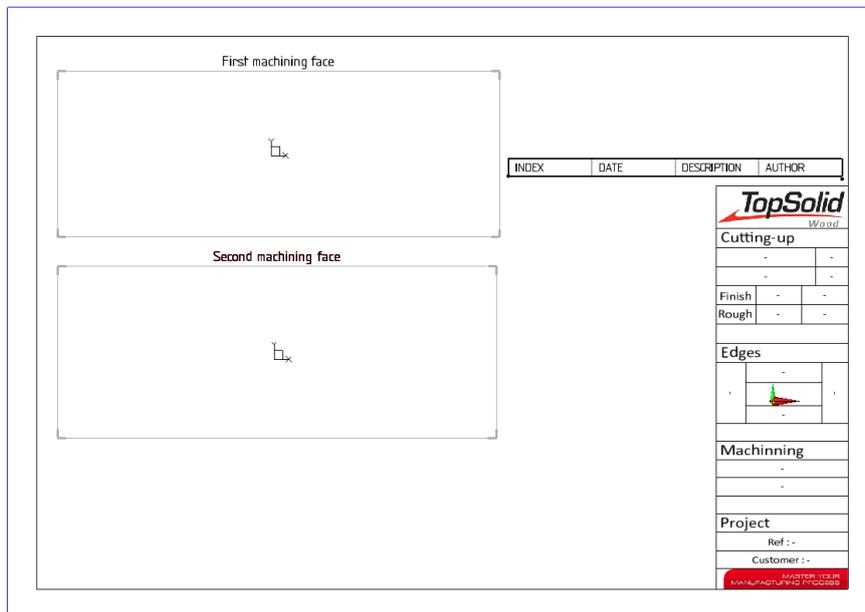
- Create a new **Drafting** document and open an existing **multi-drawing template**.
- Select **Wood > Multi-draft** and choose **Create template**.

Two new functions are available: **Create table of indices** and **Create table of drills**.

– **Create table of indices:**

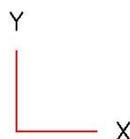
- Select the index table template.
- Position the table by clicking two points in the drawing.

Note: If the drawings are placed in the same document during multi-drafting, the index table will be local to each drawing and will display the indices of the projected element.



– **Create table of drills:**

- Choose the drilling table template.
- Select the **reference view** on which the drilling indices will be placed.
- Select the origin point or coordinate system for the drilling's coordinates. Choose **Use reference view origin** to define the coordinate origin as the 2D view coordinate system. Set **Show xy coordinates origin** to **Yes** to view the origin coordinate system.



- The **Allow inverted drill = Yes/No** option lets you display/hide the drillings on the opposite face to the projected face in the view. Define the origin face to calculate the drilling depth by setting **Origin for z coordinates = Top face/Bottom face**.
- Click **OK** to confirm these two options.

- Specify how the table has to be sorted:
 - **By type of element:** Groups the drillings by type. For example, the table will display the drilled holes, then the tapped holes, then the countersunk holes, etc.
 - **By operation:** Sorts the drillings by operation. For example, if there are tapped holes and countersunk holes, the table will display the drilling and its tapping, then the drilling and its countersinking, etc.
- Next select the types of drillings to be displayed in the table: **All drills, Not through drills only** or **Through drills only**.
- In the advanced options >>, you can choose to display the **diameter** or the **radius** of the drilling and use a **diameter/radius constraint** to see only the drillings with a diameter greater or less than 10mm.
- Click two points to position the drilling table and finish creating it.

Result of multi-drawings using a template containing a drilling table and an index table:

2	07/10/2016	Multi-Drilling	CAP
1	07/10/2016	Top/Bottom Assembly	CAP
0	07/10/2016	No machining	CAP
INDICE	DATE	LIBELLÉ	AUTEUR

TopSolid Wood		
Cutting-up		
Right	1	
Alder black	19	
Finish	900	500
Rough	900	500
Edges		
Machining		
Rigide-Projet (refer: 00:1)		
-		
Project		
Ref : Project reference		
Customer : Customer name		

RK2	50		Hole	0.00	12.000	5.00
RK1	6	↓	Hole	0.00	9.000	10.00
IDENTITY	NB	SYMBOL	TYPE	ZDP	DEPTH	DIAMETER

View style

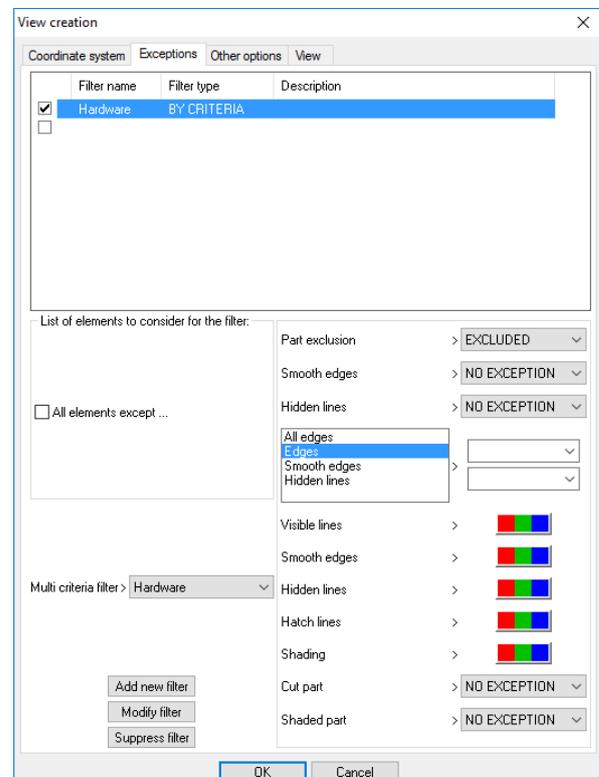
The ability has now been added to define **view styles** in order to quickly adjust all the projection parameters. A **view style** is a drafting document containing a view in which various projection parameters are set. This style is then applied to a drafting document or to existing views.

Creating the view style

- To create a new **view style**, select **File > New > Draft document**.
- Select **Without template**.
- Create a blank view. Select **View > Main view**.
- Click **No element**.
- The **View creation** window opens. Specify the different projection parameters that you want to apply to the view style. The view style can use the parameters from the following sections:
 - **Edges/Rendering**
 - **Colors**
 - **Title**
 - **Representation/Schematic**
 - **Exceptions**

In the example below, the view style will apply to the following parameters:

- **Edges/Rendering:** The smooth edges will be displayed in half-intensity and the view will be shaded.
- **Colors:** The visible lines will be projected in black.
- **Title:** No title will be displayed on the view.
- **Representation/Schematic:** The shapes will be projected in detailed representation.
- **Exceptions:** A multi-criteria filter will be used to prevent the **Hardware** parts from being projected.



- Place the view in the document.

Note: A **View style** document can contain only one view and must not reference other documents.

- Select **View > Define style**.
- Choose whether the view style will be saved in **local** or **group** configuration.
- Give the style a **name**.



Note: The view style documents have the same extension as the drafting files (.dft) and are saved in **Config/Template/ViewStyle** or **Group/Template/ViewStyle**.

- For each parameter, specify whether you want to **forbid any modification**.

In the following example, changing the title and the representation is not permitted. Once the **style view** is applied, the **Representation/Schematic** and **Title** settings will be disabled when modifying a view.



Note: This ensures that all drawings produced using this view style will have the same projection parameters.

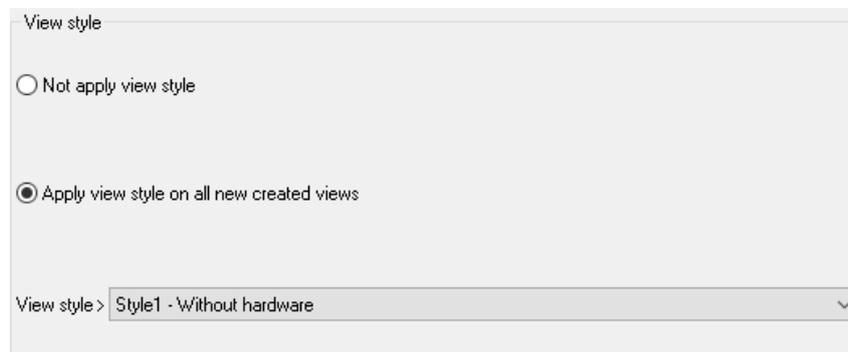
Using the view style

The view styles can be used either through drafting templates, or directly in an existing drawing.

Creating a drafting template using a view style

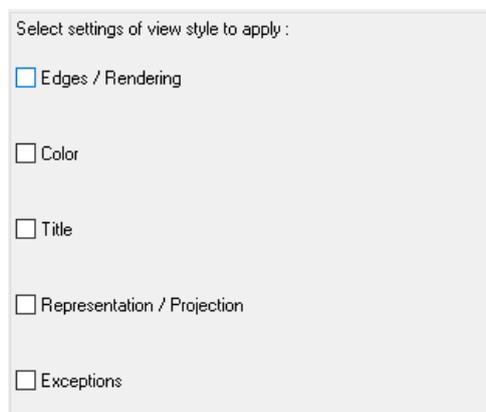
When a view style must be used each time a drafting template is used, you need to define the default style in the drafting template document.

- Select **Config/Template** or **Group/Template** and open the drafting template to be modified.
- Open **File > Properties > Drafting view > View style**.
- Click **Apply view style on all new created views**.
- Select the default **view style** to be used with the drafting template.



- Choose the **settings of the view style to be applied** by selecting/deselecting the corresponding boxes.

In the example, all settings are selected except the **title**. The views that will be created will not retrieve the settings defined in the **Title** section of the **view style**. All other settings will be applied.

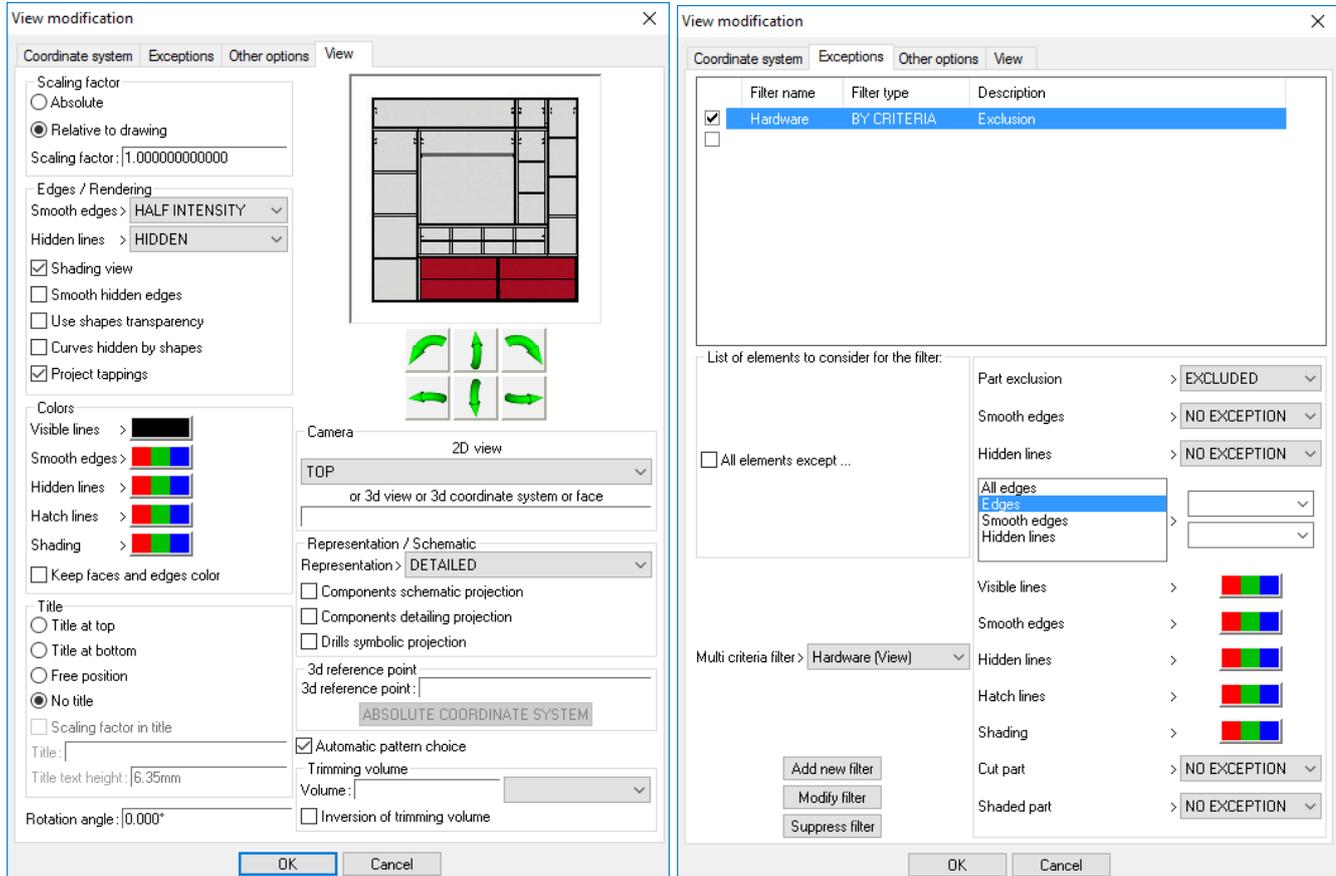


- Click **OK** to confirm the settings.

Using a drafting template with a view style

- Create a new drafting document and create a view.

In the following example, the parameters are those set in the **Style1 – Without hardware** view style that you created previously. The view is **shaded**, the **hidden lines** are **hidden**, the **visible lines** are **black**, no **title** is specified, the projected shapes are displayed in **detailed representation** and an **exception** excludes the **Hardware** parts.



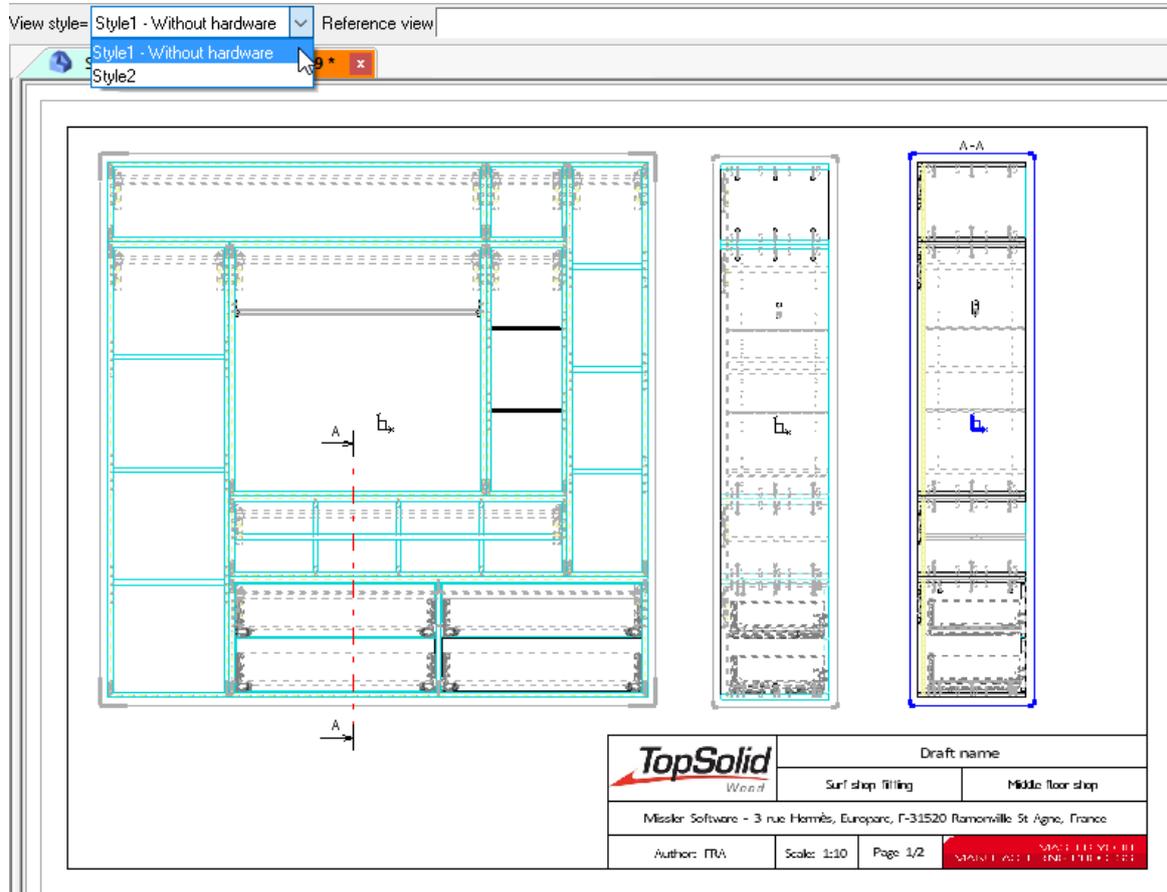
Note: The **Representation/Schematic** parameters are disabled since it has been specified in the style definition that no changes to these parameters are permitted. Changing the **title** is also forbidden, but no title must appear on the view as defined in the view style.

Using a view style in an existing drawing

A **view style** can also be applied to existing views.

- In a drafting document containing one or more views, select **View > Apply style**.
- From the drop-down list, select the **view style** you want to apply.

For example, you can use the previously created view style.



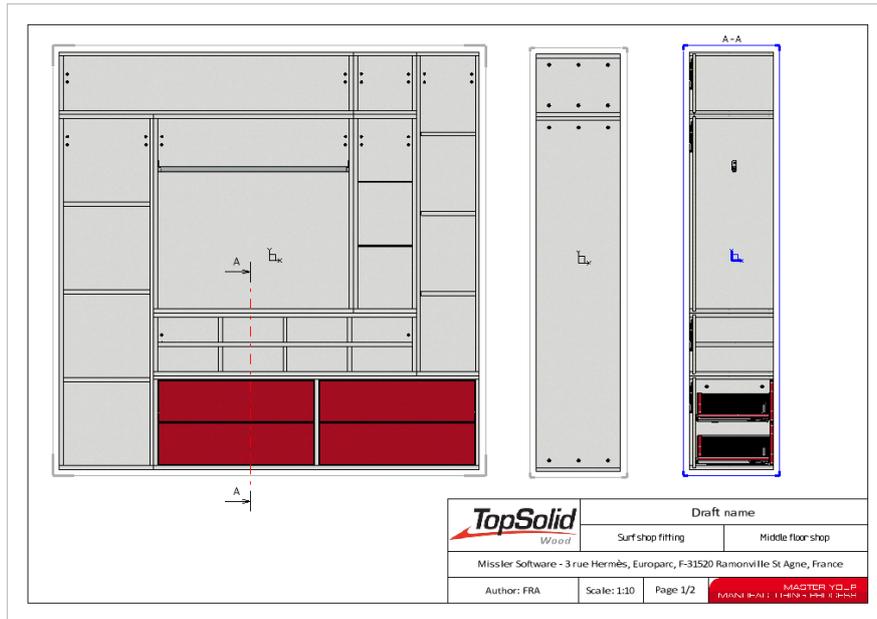
- Specify the **settings to be applied**.

In the example, only the selected **Edges/Rendering** and **Colors** settings of the **view style** will be applied. The view will be **shaded**, the **hidden edges** will be **invisible** and the **visible lines** will be displayed in **black**.



- Select the views on which to apply the style.

In the example below, all the document's views have been selected.

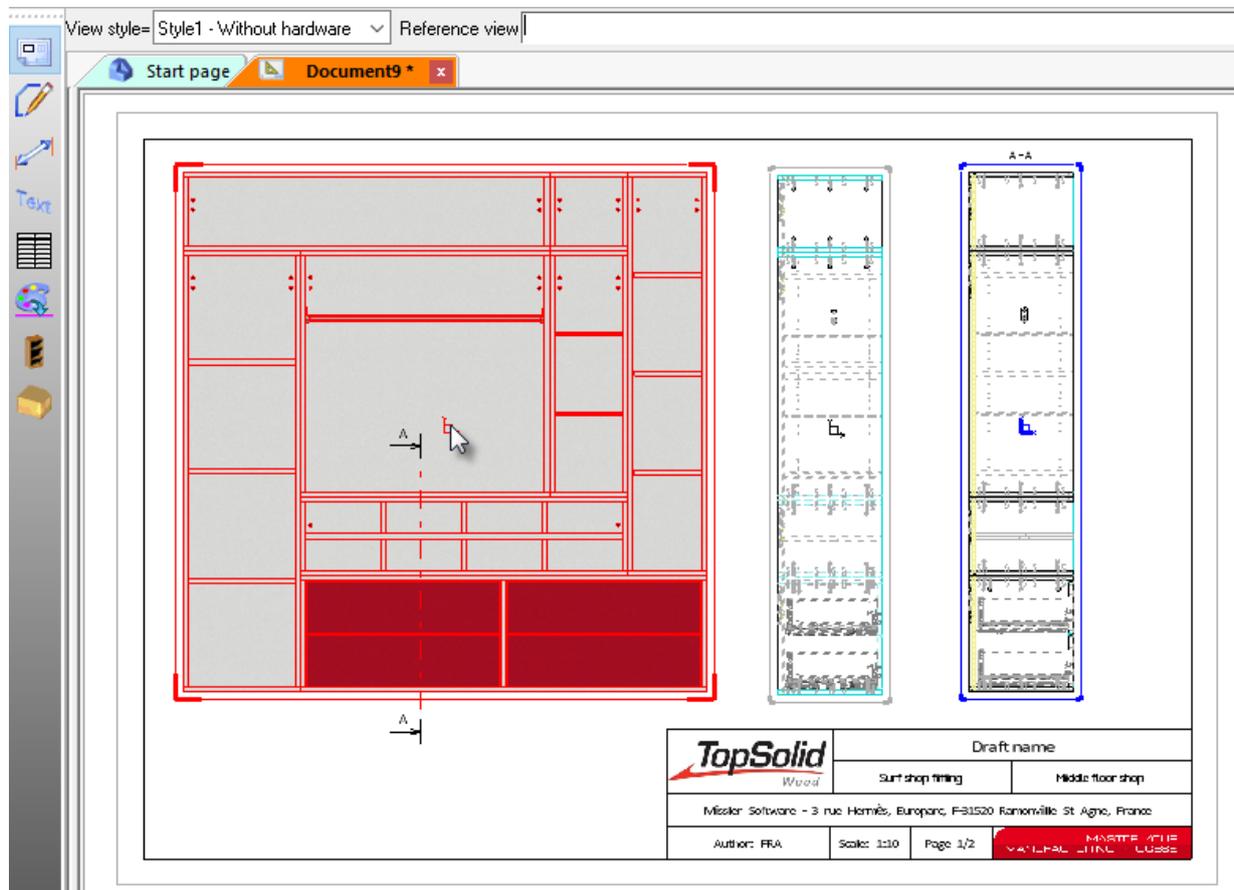


You can also **copy the projection settings of an existing view** to apply them to other views of the drawing or another drafting document.

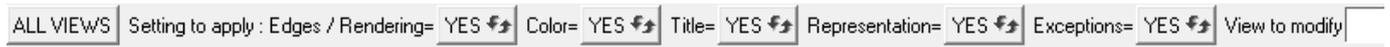
- In a drafting document containing one or more views, select **View > Apply style**.
- Select the **reference view** containing the settings to be copied.

Note: To click a view of another drafting document, click the corresponding tab and select the view you want to use.

In the following example, the shaded view is selected to apply its projection settings to the other two wireframe views.

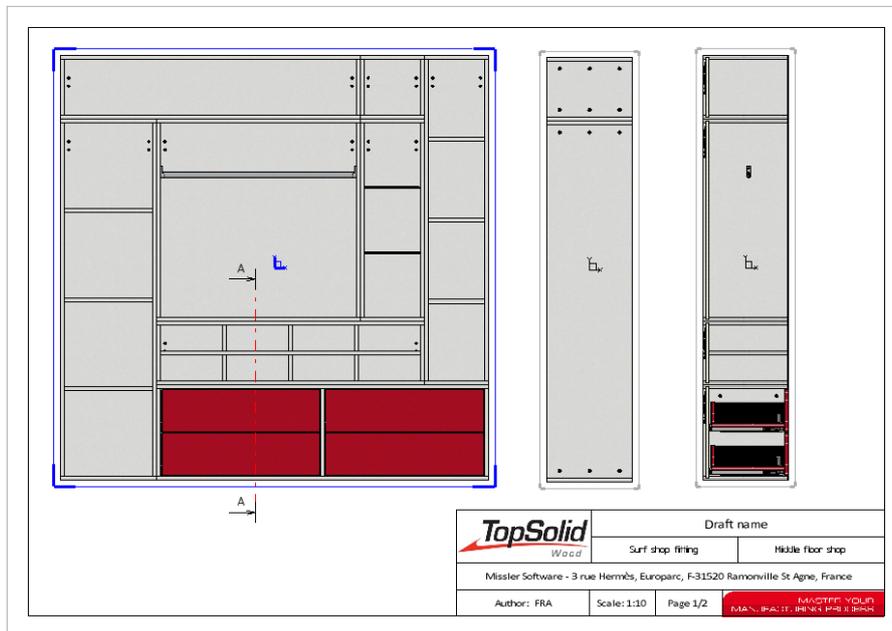


- Once you have selected the view, define the settings to apply.



- Finally, click the views to be modified.

Result:



Modifying the view style

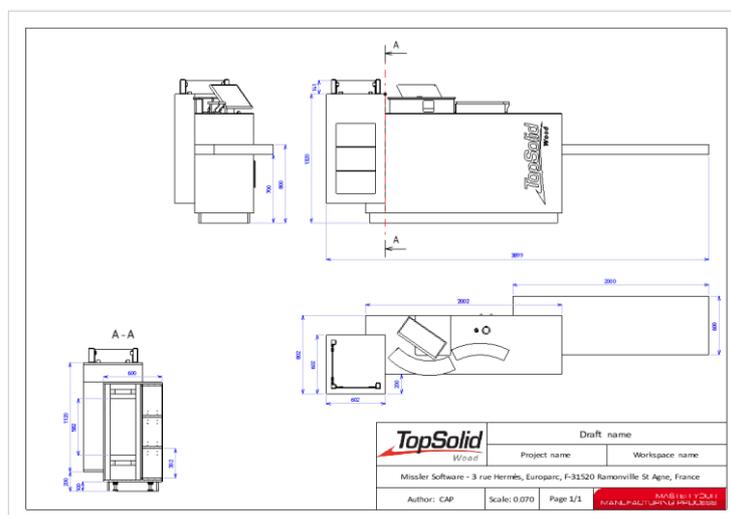
- Go to **Config/Template/ViewStyle** or **Group/Template/ViewStyle**.
- Open the document containing the **view style** to be modified.
- Use **Modify** on the view to edit the projection settings.
- To change the forbidden settings, select **Main view > Define style**.
- When you're finished making changes, click **OK** to confirm and save the document.

Note: A **view style** is not associative, which means that if the **view style** you have modified was already used, the views on which the style was applied will not change.

View alignment

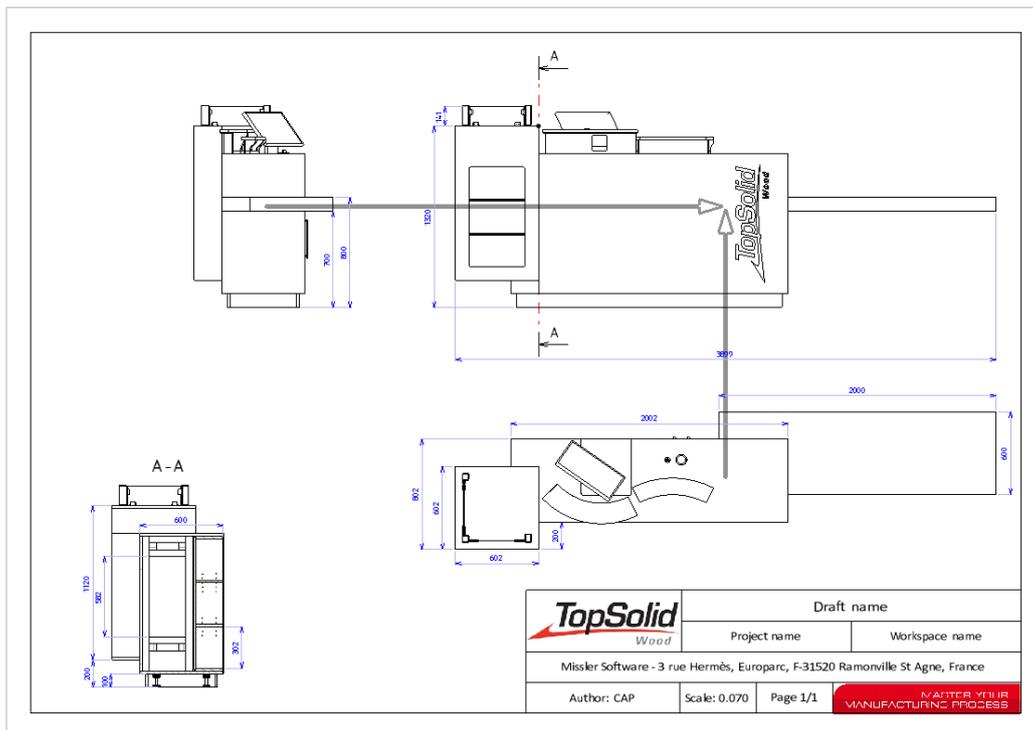
Views can be aligned quickly using **Adjust alignment**.

For example, let's use the following drawing:



- Select **View > Adjust alignment**.

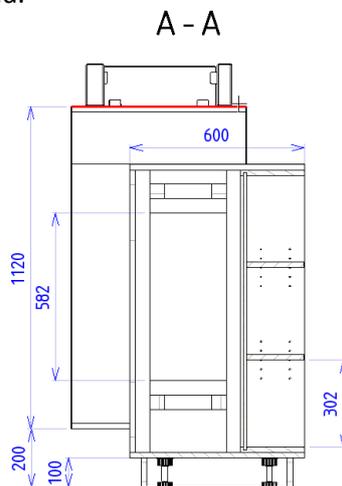
Gray arrows show the existing alignments such as the alignment between a reference view and its auxiliary views.



Note: The alignments indicated by gray arrows cannot be changed. However, the views can be moved along the axis of this alignment.

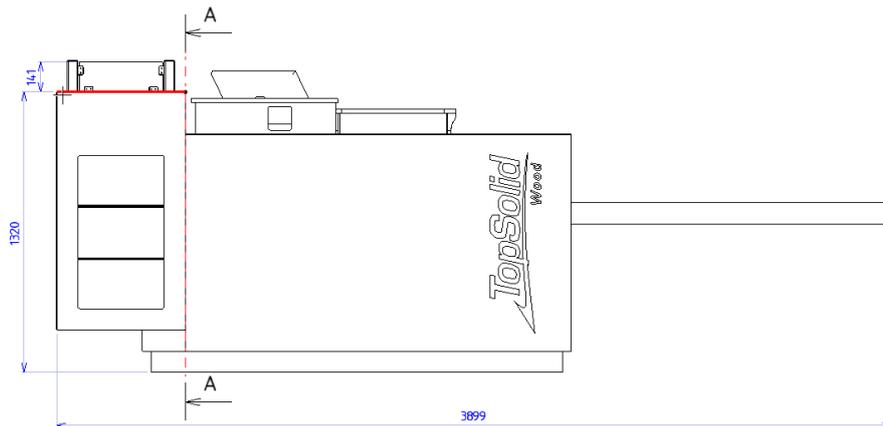
- Click an edge of the view to be aligned.

In the example, the cross section on the bottom left of the drawing will be aligned with the drawing's reference view. An edge of the cross section is selected.

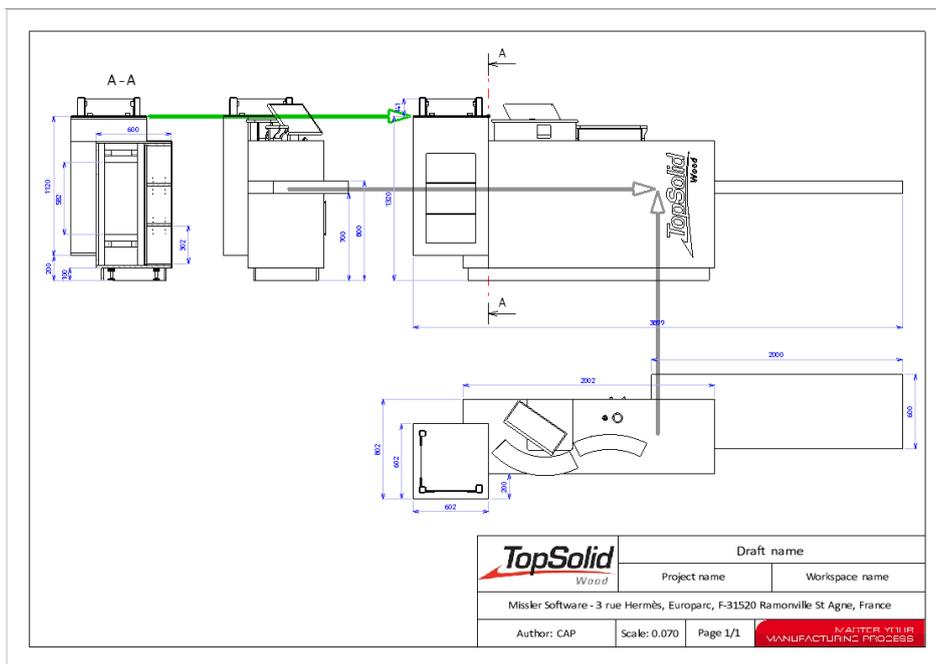


- Next click an edge of the reference view for alignment.

The corresponding edge is selected in the reference view.



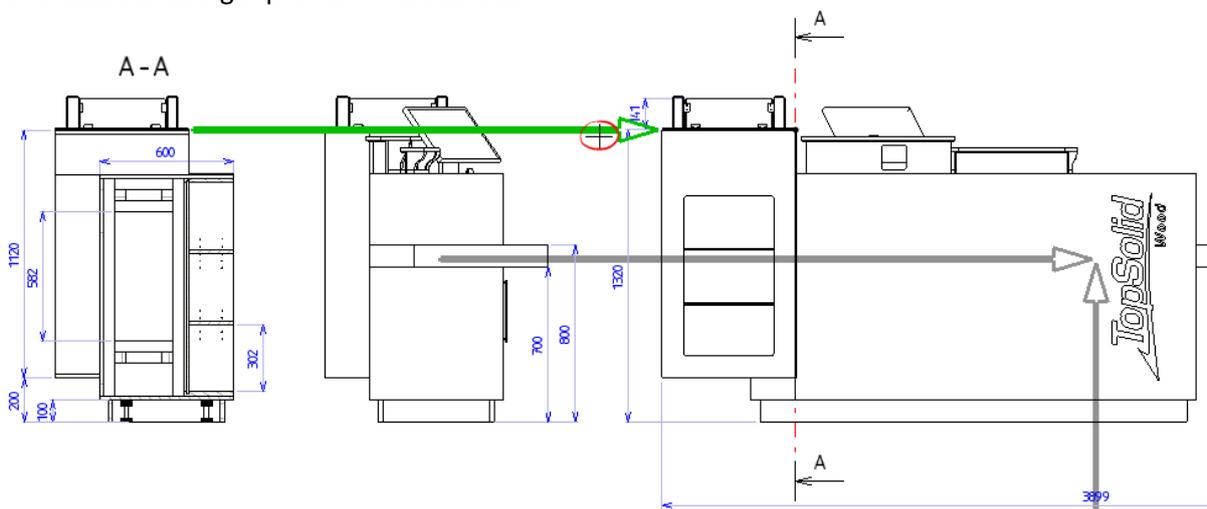
- Once the views are aligned, a green arrow appears.



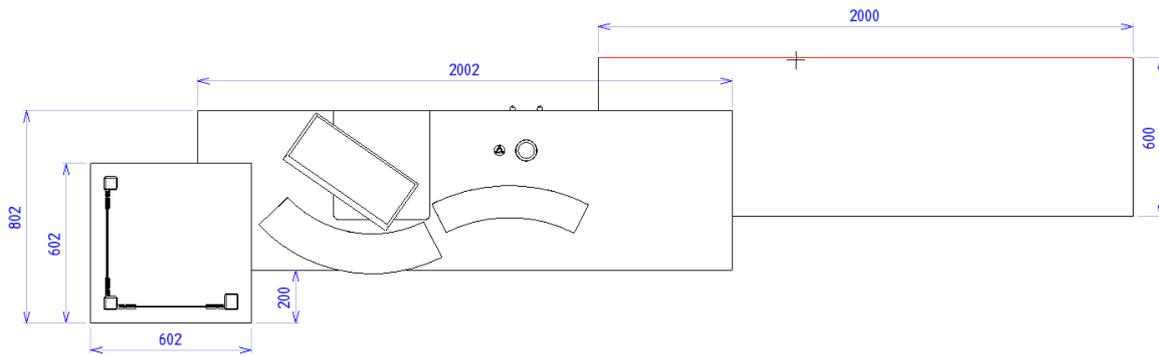
Note: The green arrows represent the alignments that you can change.

- To edit an existing alignment, start **View > Adjust alignment**.
- Click the arrow of the alignment to be modified, on the side of the edge to be replaced.

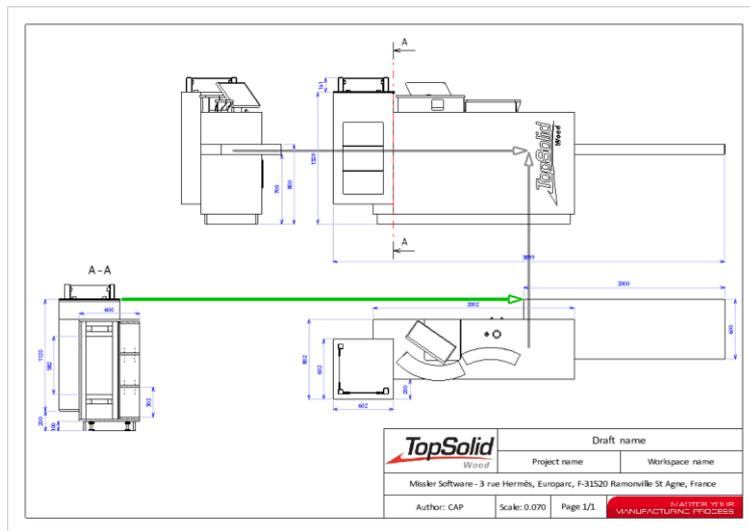
In the example, the created alignment will be changed to align the cross section with the top view of the item, so you need to click on the right portion of the arrow.



- Select **Reconnect**.
- Click the reference edge for alignment.



Result:

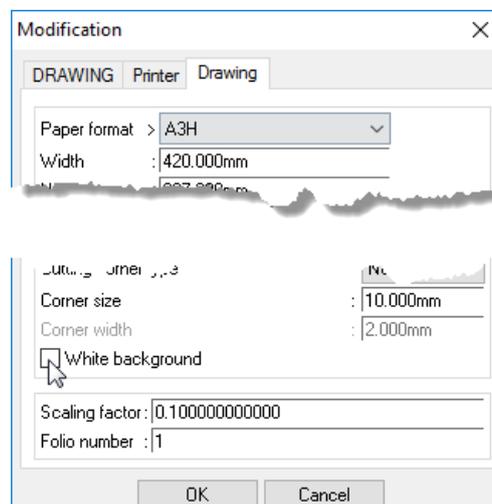


Note: Arrows may appear in red, which means that one of the two geometries used for alignment no longer exists. You need to select **View > Adjust alignment**, click the red arrow, and **reconnect** it to an existing geometry.

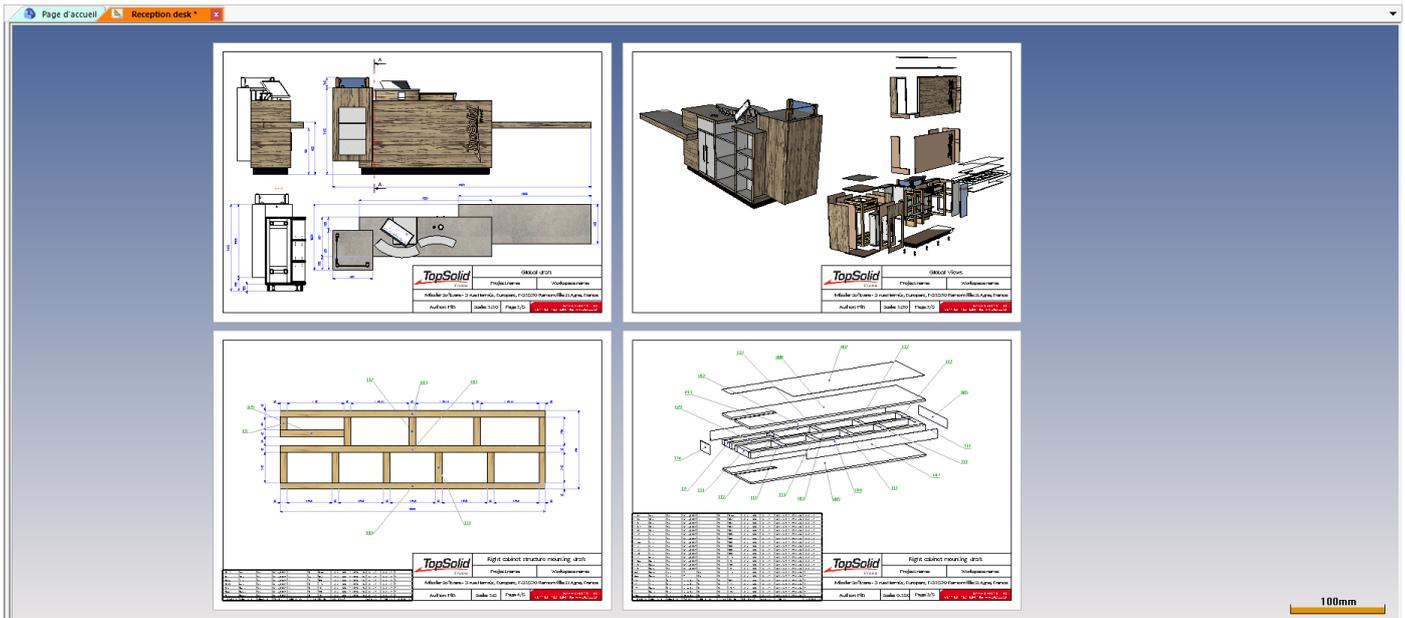
White background

The drawing's backgrounds contained in a drafting document are now white.

- To change a drawing's background to white, select **Modify element**.
- Click a drawing's border.
- Select **White background**.



The modified drawing is on a white background.



Note: All TopSolid templates are now using a white background. If you want the existing user templates to have this white background too, you need to open each template document and make the change.

Margins in multi-drawings

You can use the margins defined in the paper format for multi-drafting.

- From a **Drafting** document, select **Tools > Options > Drawing**.
- Specify the **dimensions**, the **margins** and the **name** of the paper format.

Drawing

Paper format > A4V-Margin

Width : 210.000mm

Height : 297.000mm

Name : A4V-Margin

Left margin : 8.000mm

Right margin : 8.000mm

Bottom margin : 10.000mm

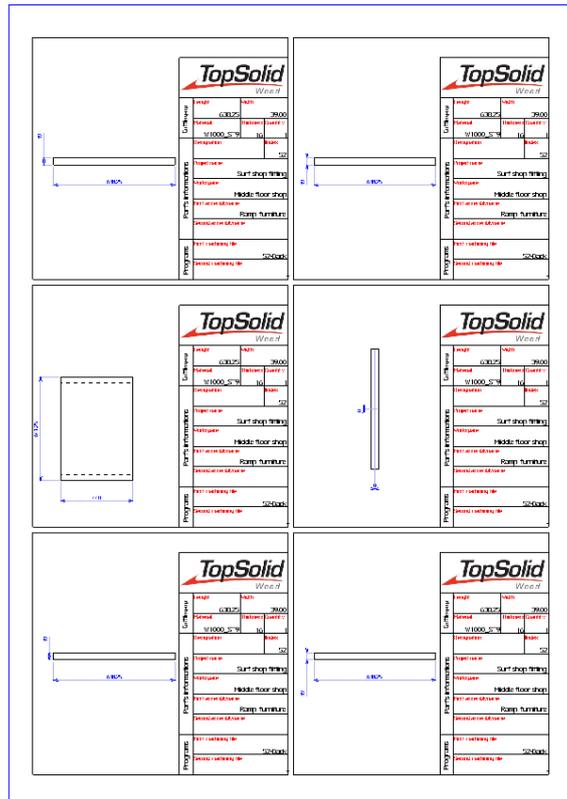
Top margin : 10.000mm

Nb of horizontal divisions : 4

Nb vertical divisions : 4

- To be able to use these margins, check **Use drawing's margins in multi-draft** in **Tools > Options > TopSolid'Wood configuration > Draft**.
- Create **multi-drawings** and set **Put together all drafts in one document** to **YES**.

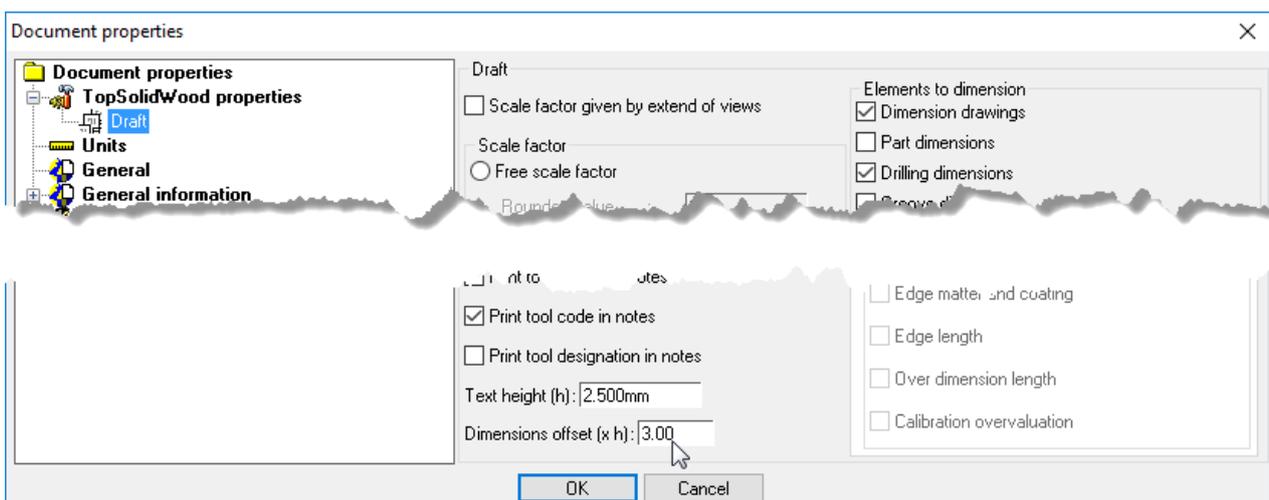
Result: In the resulting drawings, the projection is based on the defined margins.



Offset between dimensions

The offset between dimensions defined in the **multi-drawing template** or in **Tools > Options** is no longer based on the drawing scale factor, providing improved readability of dimensions.

- Open a **multi-drawing template**.
- Specify the **dimension offset** in **File > Document properties > TopSolid'Wood properties > Draft**.



- Create **multi-drawings** from a project.

Interfaces

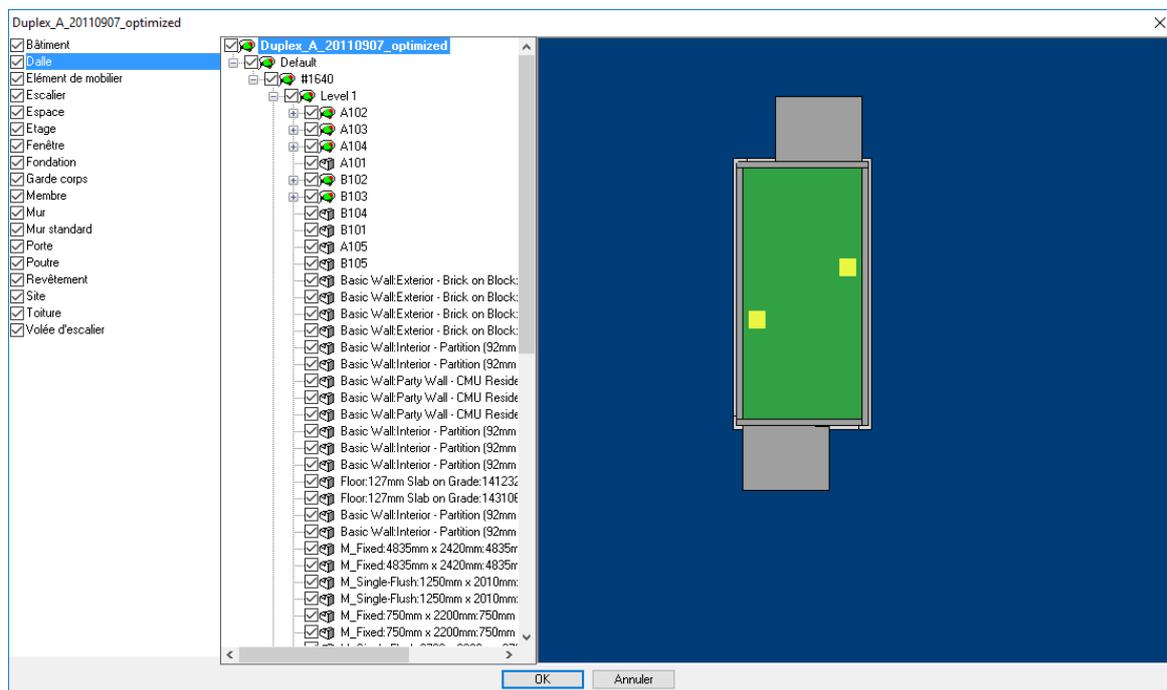
BIM IFC import/export

Import

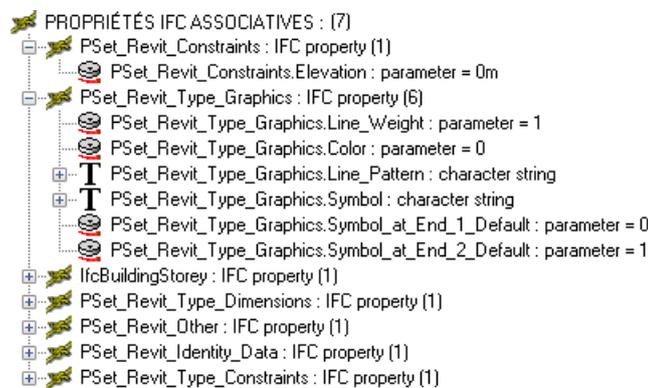
Enhancements include the ability to import **IFC** files into TopSolid. The shapes and properties contained in these **IFC** files are retrieved in a .top file tree.

- To **import** an **IFC** file, select **File > Open**.
- Select the file and click **OK** to confirm.
- Choose the directory where you want to save the .top files resulting from the import.

A window providing a document overview appears. The check boxes let you choose whether you want to import the selected shapes, by building, by floor, etc.



Note: The retrieved shapes are **polyhedral** shapes. The imported properties are copied into the set of **associative IFC properties**.



Export

You can export the shapes and the **IFC properties** to **IFC** format. To do this, select **File > Save as** and choose the **IFC** format (*.ifc).

Miscellaneous

- The **Part category** property can be used in the part selection dialog boxes.
 - You can import the windows of a **Sky** light with a component. To achieve this, you need to include the **Sky** light in the component's main set, with the windows. The windows will be added to those of the assembly's **Sky** light.
 - When using a **workspace**, you can define **context sets** to group elements to be used as contexts in multiple workspaces. These sets make the **workspaces** much easier to draft, as well **final assembly**.
 - Using **Shape > Other shapes > From bitmap**, you can create a surface from an image inserted in TopSolid.
 - When you create a **sawing operation**, a **groove operation** or a **user machining**, you can select several guide curves.
-

What's New in TopSolid'SheetMetal v6.18



This section describes the enhancements made to the version **6.18** of **TopSolid'Punch** and **TopSolid'Cut**.

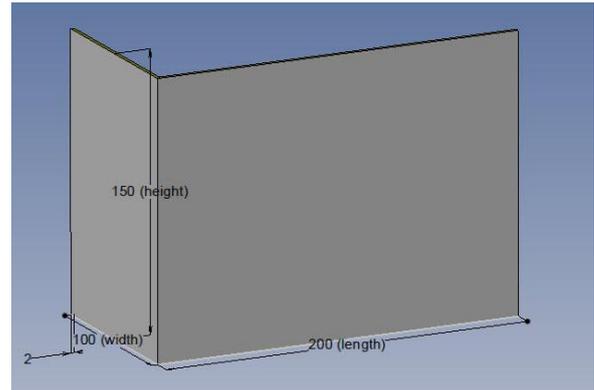
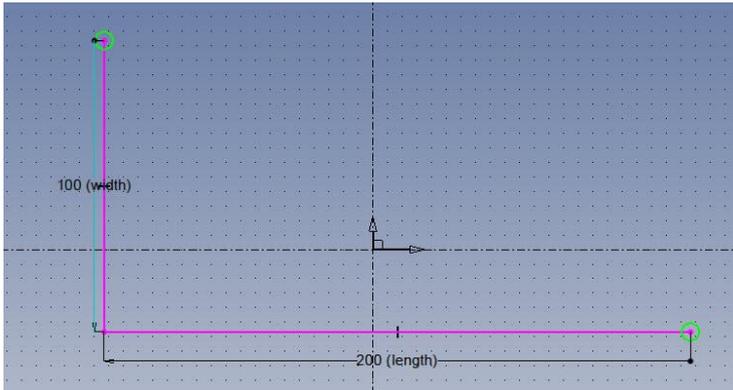
TopSolid'Punch and TopSolid'Cut

Importing a family of V7 unfoldings

A new menu function lets you automatically create all the PART documents for the various variants defined in a V7 family document.

Creating a family of parts in TopSolid 7

- Example of a sheet metal part created from a sketch on which parameters have been defined.



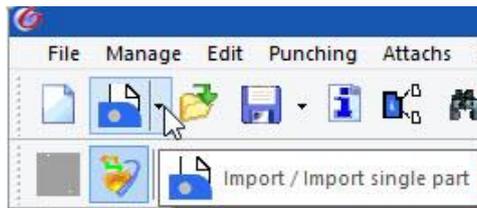
- Consider the following family document linked to the sheet metal part in which catalog codes and parameters values have been defined.

Code	length	width	height
A	200mm	400mm	50mm
B	400mm	500mm	250mm
C	500mm	800mm	450mm
D	800mm	1000mm	650mm

- Unfold the part.
- Check the document into the vault.

Importing the unfolding and preparing the part's machinings

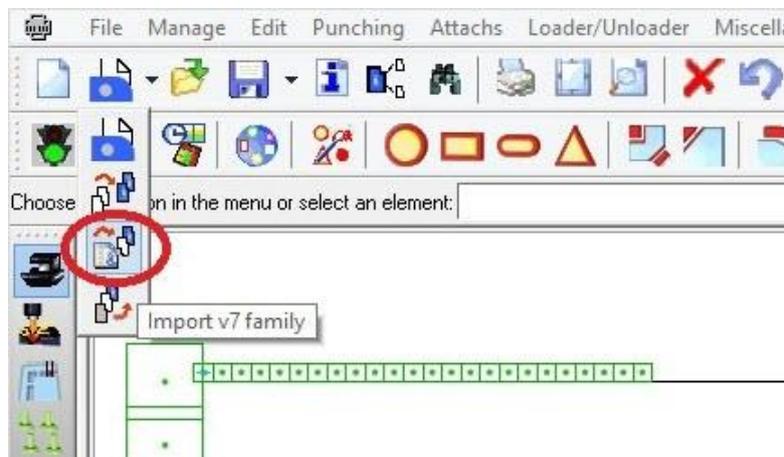
- In TopSolid'SheetMetal, create a part document based on the unfolding using **Import single part**.



- Place the machinings and save the document.

Creating the family's part documents while retaining the machinings updated

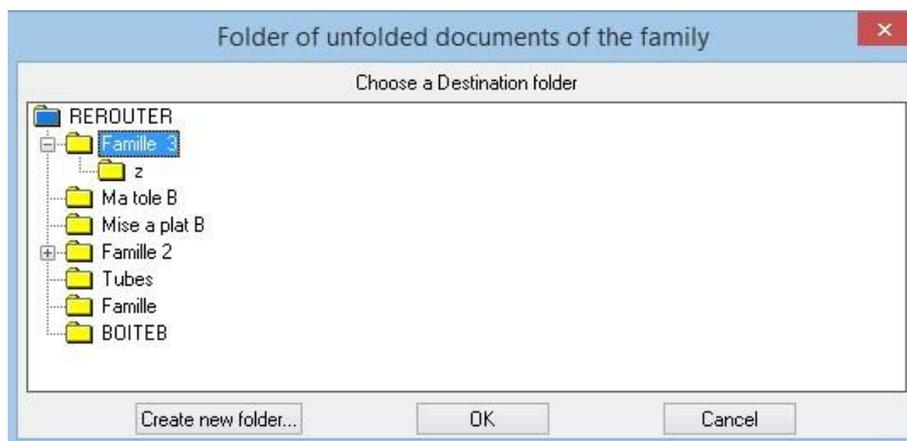
- From the current document, select **Import v7 family**.



- The system will ask you whether you want to complete the existing machinings with automatic allocation and whether the family's parts must be included in the Task manager.



- Choose the directory where you want to save the unfoldings derived from variants.

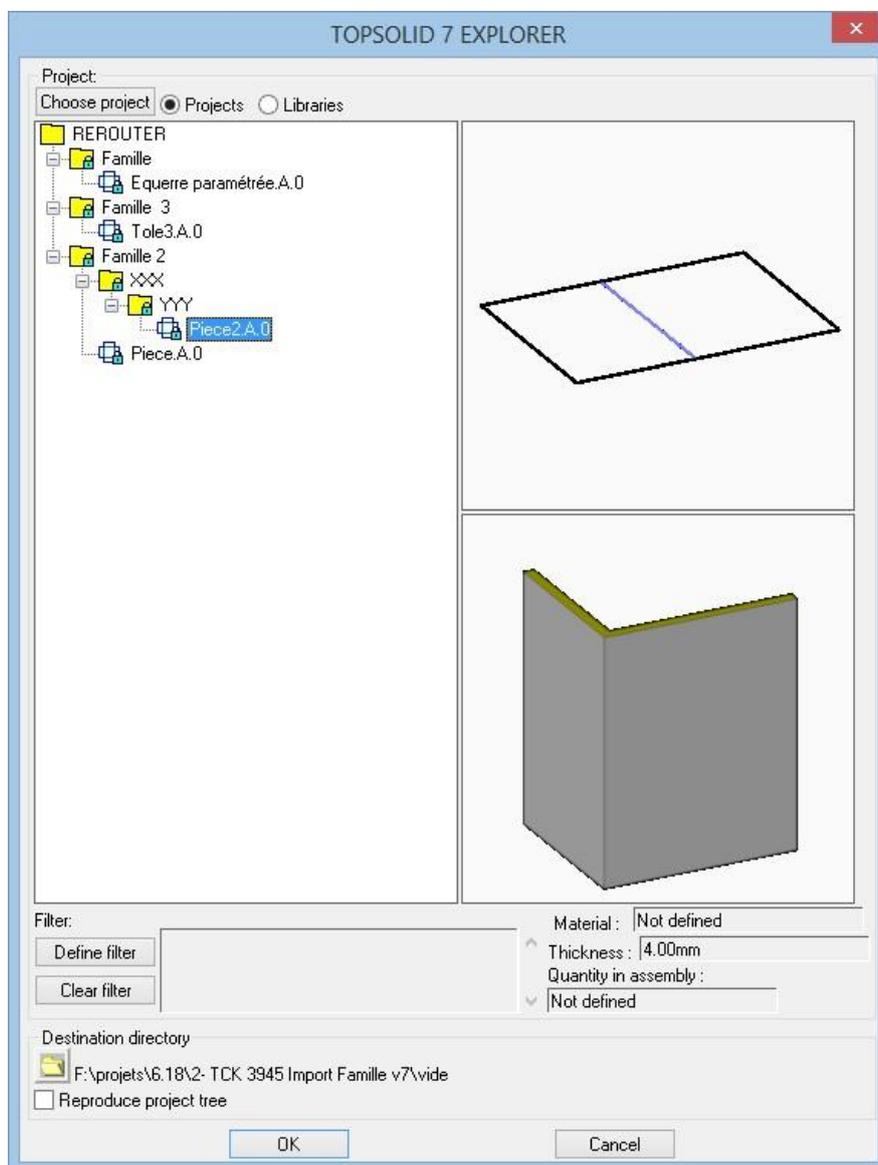


- The PCH documents are generated and named by concatenating the catalog code and the document's original name.
- Each PCH document is automatically linked to the unfolding created in TopSolid 7.



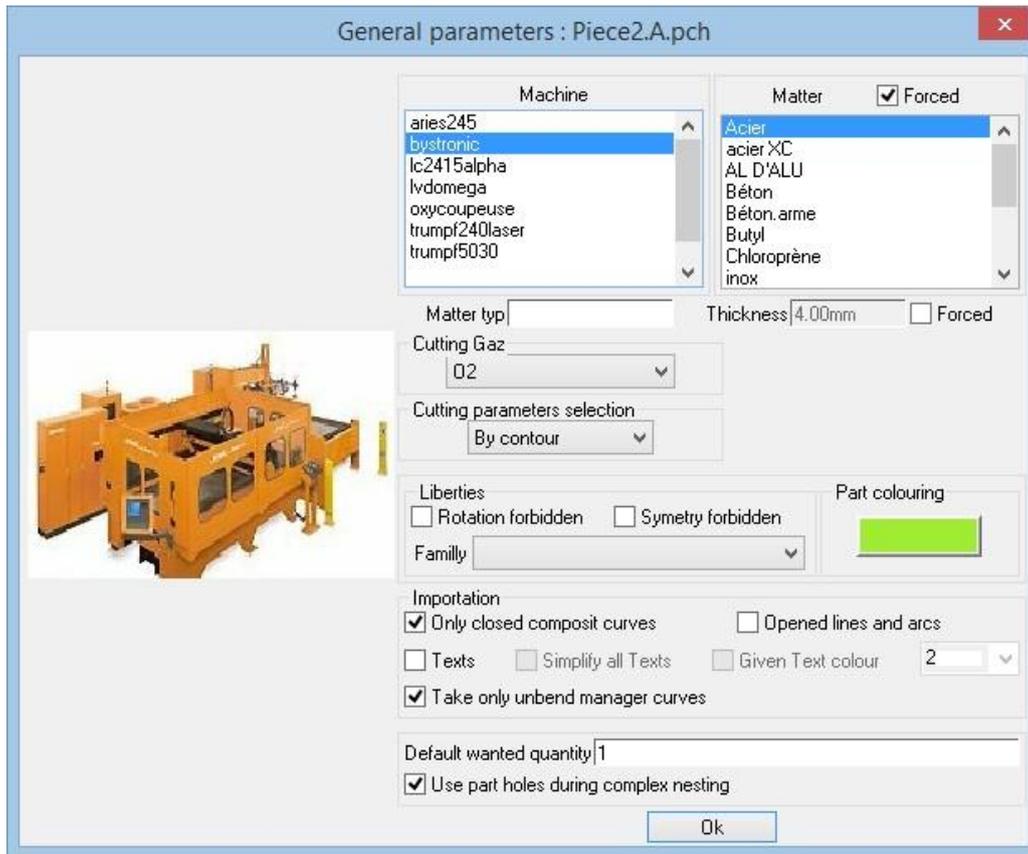
Creating a part family without original document

- When you call up the **Import v7 family** function from an empty PCH document, you are asked to choose an unfolding whose 3D part has at least one family. In this case, all the project's unfoldings are not available in the Project Explorer.

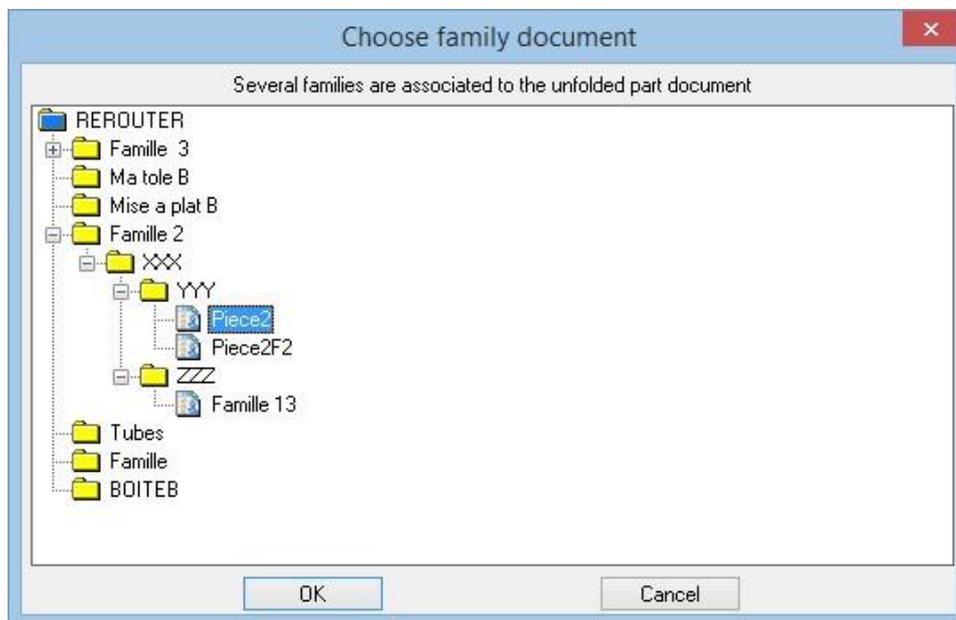


- Select the unfolding and the destination directory.

- Specify the machine and the other general parameters.



- Choose the directory where you want to save the unfoldings.



- All PCH documents have been created and named by concatenating the catalog code and the unfolding name.

Task Manager

Comment column

A new **Comment** column has been added to the Task Manager's **Parts** and **Matters** tabs.

Ref.	Des.	Set	Plane	Com.	Customer	Mat.	MatTyp	Th.	Qty	Nb set	Qty +	Qty max.	Fam.	Comp.	Prio.	Nb ent.	Deadline	Mac.	Gaz	Tur.	Comment
<input type="checkbox"/> 1						Acier		1.50mm	5		0	0	-1	No	0	1	03/12/2015	aries245		aries245	
<input type="checkbox"/> 1				Counters		Acier		1.50mm	1		0	0	-1	No	0	1	06/07/2015	aries245		aries245	
<input type="checkbox"/> 2				Counters		Acier		1.50mm	1		0	0	-1	No	0	1	06/07/2015	aries245		aries245	
<input type="checkbox"/> 2						Acier		1.50mm	6		0	0	-1	No	0	1	03/12/2015	bystronic	02		

Matter	Matter Type	Th.	Des.	Length	Width	Q.	Free qty	Waste	Stored sheet	Area	Date	Comment
<input checked="" type="checkbox"/> Acier		3.00mm		1000.00mm	1000.00mm	Yes	No	No		1000000.000mm²	28/06/2016	
<input type="checkbox"/> Acier		2.00mm		1000.00mm	1000.00mm	Yes	No	No		1000000.000mm²	17/12/2015	
<input checked="" type="checkbox"/> Acier		2.00mm		2000.00mm	1000.00mm	Yes	No	No		2000000.000mm²	21/07/2015	
<input type="checkbox"/> Acier		1.00mm		2000.00mm	1000.00mm	Yes	No	No		2000000.000mm²	25/07/2016	

The comment information is retrieved from the PCH document when the part is inserted into the Task Manager, and it can be edited by double-clicking in the **Comment** column of the desired line.

Mac.	Gaz	T..	Comment
t13030	HS		Part modify
t13030	HS		Part modify
t13030	HS		Part modify

The part comments are already displayed in the standard operator card.

If the Task Manager's comment column is empty, the comment of the PCH document is displayed instead.

The matter comment can be displayed in both the simple order operator card and the complete order operator card (*orderopcardcompl.full*).



Date	22-08-2016
Files path	F:\projets\6.18\7-TCK 3958 Colonne commentaire
Machin	aries245
Matter	Acier
Thickness	10.00mm
Author name	

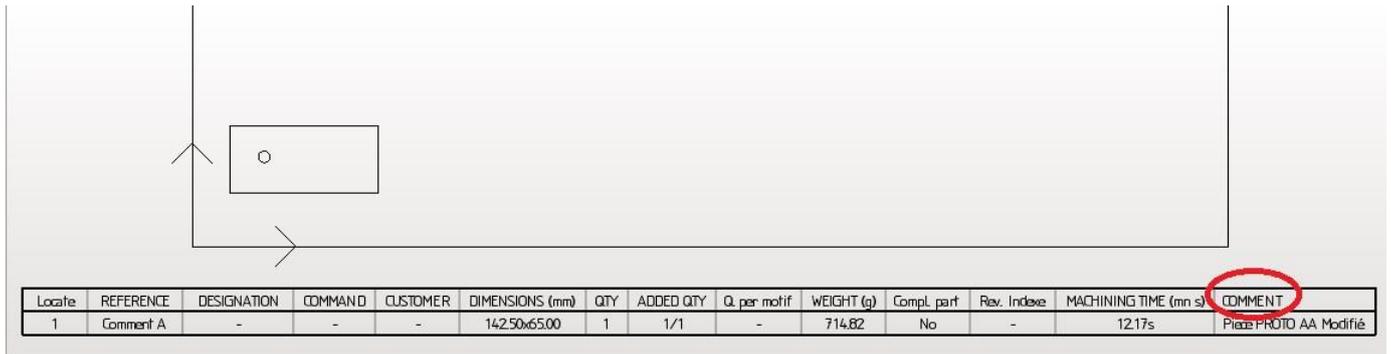
ORDER DATAS

Weight : 714.82g	Machining time : undetermined	Global waste rate : 99.1 %	Sheets qty : 1	Parts qty : 1
------------------	-------------------------------	----------------------------	----------------	---------------

MATTERS DATAS

MATTER	MATTER TYPE	THICKNESS (mm)	DESIGNATION	DIMENSIONS (mm)	AREA (mm²)	QTY	COMMENT
Acier	-	10.00	-	1000.00x1000.00	1000000.000	1	FORTE

Each part's comment is also available in the complete order operator card, in the table of parts for each nesting.



New columns in the archive file

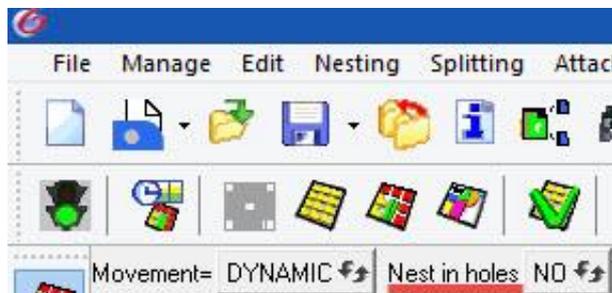
New columns have been added to the Task Manager's archive file:

- In the **Nestings** tab:
 - o Column **O: From a waste**
 - If the nesting has been performed from a waste, the value **Yes** will appear in this column.
 - o Column **T: Extracted waste**
 - If a waste has been created from this sheet, the value **Yes** will appear in this column.
 - o Column **U: Comment**
 - Nesting comment.

- In the **Parts** tab:
 - o Column **U : Average time per part**
 - Warning: It is only available for orders created from the 6.18 release.
 - o Column **V : Weight**
 - Weight of the single part.
 - o Column **W : Comment**
 - Part comment.

Nesting in holes

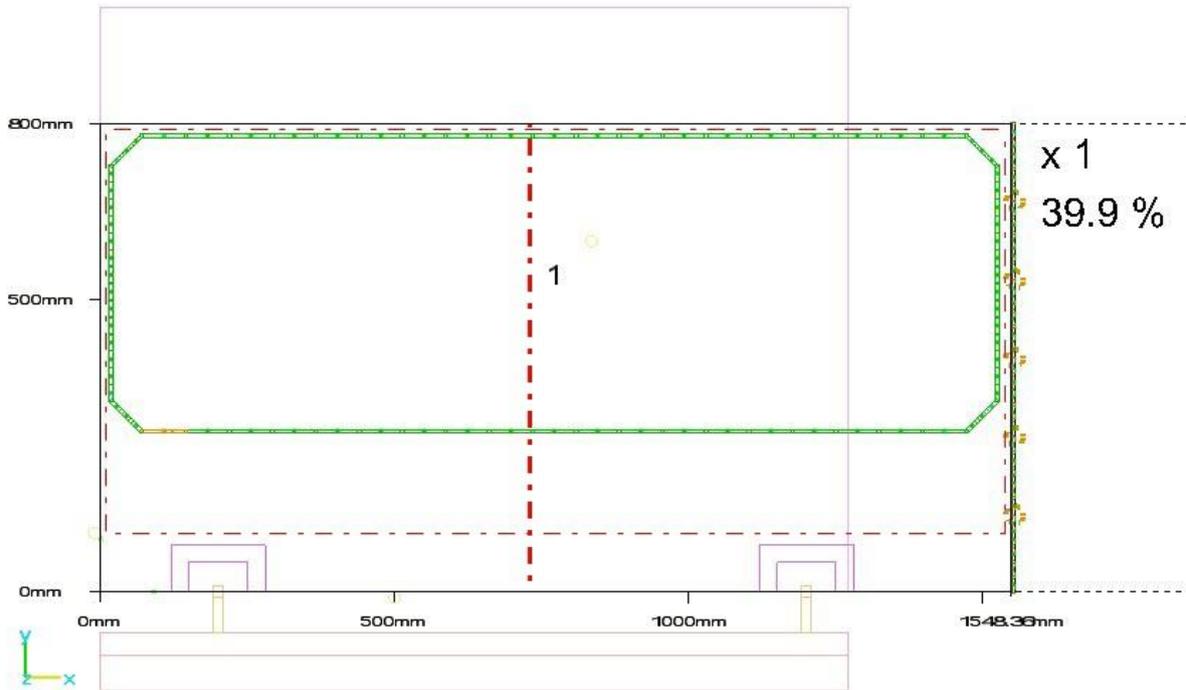
In both **Insert parts** and **Move parts** manual nesting functions, an option that lets you nest in part's holes has been added at the first level.



In the previous release, this option was available in the **Parameters** dialog box.

Displaying sheets with dotted extracted wastes

When a waste has been created and the sheet has been resized, the wastes are now displayed with dotted lines.



Waste surface displayed in order operator card

In the complete order operator card, a new word allows you to display the surfaces of wastes extracted from the sheet.

#waste_surf_matter#

The reference template operator card can be found in \Missler\V618\Local\EnglishUS\OrderOpcardCompl_full.dft.

Below is an example of result for a sheet with an extracted L-shaped waste:

Name: #name_matter# #name_iso#		Used waste: #matter_designation#	#name_prog_iso#	N#name_prog_iso#E0	Nb: #nbpas
Gross dim : #grossdims_matter#		Waste dim : #waste_des_matter#		#waste_dims_matter#	
Useful dim. : #usefuldims_matter#		Waste des. : #waste_surf_matter#			
Machining time : #time_matter#	Matter : #matter_sheet#	Total parts qty : #qtyparts_matter#			
Total machining time : #totaltime_matter#	Thickness : #thickness_sheet#				

Name: surf#01.meg surfT01	Used waste: -		Nb: 1
Gross dim : 2000.00mmx1000.00mm	Waste dim : surf#01#M=Acier#T=2.00mm#I=0		
Useful dim. : 400.00mmx500.00mm	Waste des. : 2000.00mm x 1000.00mm		
	Waste area. : 1785800.000mm²		
Machining time : 55.21s	Matter : Acier	Total parts qty : 1	
Total machining time : 55.21s	Thickness : 2.00mm		

The diagram shows a rectangular sheet with a diagonal hatching pattern. In the top-left corner, a rectangular area is shaded with a different pattern and labeled 'SURF' in pink. This shaded area represents the waste surface extracted from the sheet. The rest of the sheet is unshaded. The text 'JE ADRESINE' is visible at the bottom left corner of the sheet.

Multi-order operator card

You can create a common operator card for several orders.

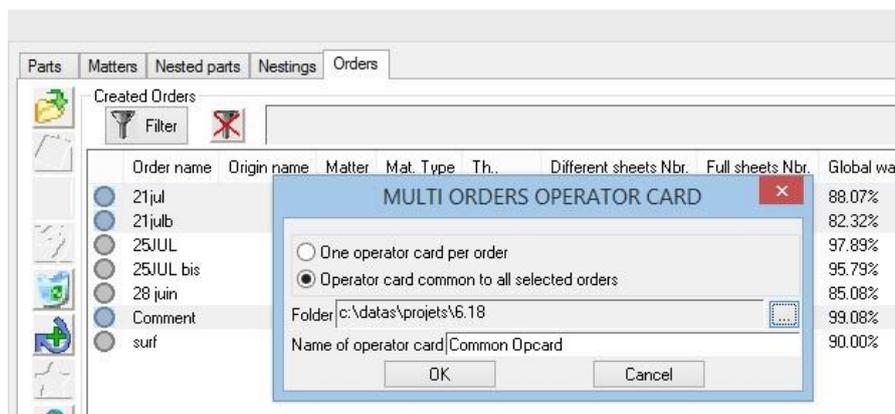
A template can be found in `\Missler\V618\Local\EnglishUS\MultiOrdersOpcard.dft`.

The image shows a template for an operator card. It features the TopSolid Sheetmetal logo at the top left. To the right of the logo are two input fields: 'Date' with the placeholder '#date_order#' and 'Author name'. Below this is a section titled 'ORDERS DATAS' containing a table with five columns: 'Weight : #weight_order#', 'Machining time : #time_order#', 'Global waste rate : #waste_order#', 'Sheets qty : #nbsheets_order#', and 'Parts qty : #nbparts_order#'. The next section is 'MATTERS DATAS', which is a table with columns: 'MATTER', 'MATTER TYPE', 'THICKNESS (mm)', 'DESIGNATION', 'DIMENSIONS (mm)', 'AREA (mm²)', 'QTY', and 'COMMENT'. This is followed by 'NESTINGS DATAS', which is a table with columns: 'N FILE', 'DES.', 'GROSS DIM. (mm)', 'USEFUL DIM. (mm)', 'NB MACHINING TIME (mm s)', 'TOTAL MACHINING TIME (mm s)', 'WASTE RATE', 'PARTS QTY', 'TOTAL PARTS QTY', and 'MACHINE'. The final section is 'FULL DATAS COMMANDS', which is a table with multiple columns for command details. At the bottom, there is a status bar with various icons and text.

This template lists the matter data, the nesting documents, and the details of each command for each order in summary tables.

To enable this feature, you need to copy the operator card template into the **\$PUNCHDATA** folder.

In the Task Manager's **Orders** tab, if you select several orders and you enable the  function to create the multi-order operator card, a new dialog box lets you choose whether you want to create an operator card per order, or an operator card common to all the selected orders.



After this, you have to specify a name for the resulting operator card as well as the folder where it should be saved.

Example:

O R D E R S D A T A S				
Weight : 28.43kg	Machining time : 52mn 41.93s	Global waste rate : 87.2 %	Sheets qty : 7	Parts qty : 157

M A T T E R S D A T A S							
MATTER	MATTER TYPE	THICKNESS (mm)	DESIGNATION	DIMENSIONS (mm)	AREA (mm²)	QTY	COMMENT
Acier	-	2.00	-	2000.00x1000.00	2000000.000	6	-
Acier	-	10.00	-	1000.00x1000.00	1000000.000	1	FORTE

N E S T I N G S D A T A S									
N FILE	DES.	GROSS DIM. (mm)	USEFUL DIM. (mm)	NB MACHINING TIME (mn s)	TOTAL MACHINING TIME (mn s)	WASTE RATE	PARTS QTY	TOTAL PARTS QTY	MACHINE
1 21jul#01.meg	-	2000.00x1000.00	1003.80x874.00	1 14mn 10.00s	14mn 10.00s	77.2 %	35	35	aries245
2 21jul#02.meg	-	2000.00x1000.00	357.72x874.01	1 5mn 34.60s	5mn 34.60s	90.6 %	9	9	aries245
3 21jul#03.meg	-	2000.00x1000.00	236.90x878.00	1 1mn 48.82s	1mn 48.82s	96.4 %	2	2	aries245
4 21julb#01.meg	-	2000.00x1000.00	680.00x1000.00	1 13mn 52.32s	13mn 52.32s	75.2 %	48	48	bystronic
5 21julb#02.meg	-	2000.00x1000.00	480.00x1000.00	1 9mn 56.79s	9mn 56.79s	82.2 %	37	37	bystronic
6 21julb#03.meg	-	2000.00x1000.00	280.00x1000.00	1 6mn 38.22s	6mn 38.22s	89.6 %	25	25	bystronic
7 Comment#01.meg	-	1000.00x1000.00	142.50x65.00	1 4.117s	4.117s	99.1 %	1	1	aries245

F U L L D A T A S C O M M A N D S															
COMMAND	NB PARTS	REP PART	BOX (mm)	UNINARY TIME (mn s)	TOTAL TIME PARTS (mn s)	UNINARY MASS (kg)	TOTAL MASS PARTS (kg)	MATTER	THICKNESS (mm)	TOTAL QTY DHD	FULL TIME DHD (mn s)	TOTAL MASS DHD (kg)	CUSTOMERS	ORDER	MACHINE
Arks	35	BT1	142.50x55.00	9.60s	5mn 38.00s	0.13kg	4.78kg	Acier	2.00	46	18mn 35.00s	11.17kg		21jul	aries245
	5	BT2	50.00x50.00	1mn 3.86s	5mn 19.25s	0.09kg	0.14kg								
	1	P1	388.30x54.87	2mn 24.46s	2mn 24.46s	1.51kg	1.51kg								
	1	P6	995.80x237.00	1mn 52.11s	1mn 52.11s	2.09kg	2.09kg								
	1	P3	349.12x72.60	1mn 35.29s	1mn 35.29s	1.33kg	1.33kg								
	1	P4	258.00x220.00	41.26s	41.26s	0.64kg	0.64kg								
	1	P5	236.90x180.00	22.23s	22.23s	0.61kg	0.61kg								
	1	P2	201.00x250.00	24.37s	24.37s	0.51kg	0.51kg								
	1	Comment A	142.50x65.00	12.17s	12.17s	0.71kg	0.71kg	Acier	10.00	1	12.17s	0.71kg	Comment		aries245
ByS	76	BT1	142.50x55.00	15.71s	9mn 22.32s	0.13kg	8.89kg	Acier	2.00	110	28mn 34.79s	16.59kg		21jul	bystronic
	30	BT2	50.00x50.00	12.66s	6mn 19.89s	0.09kg	0.89kg								
	1	P1	388.30x54.87	51.06s	51.06s	1.51kg	1.51kg								
	1	P6	995.80x237.00	1mn 2.11s	1mn 3.11s	2.09kg	2.09kg								
	1	P3	349.12x72.60	23.46s	23.46s	1.33kg	1.33kg								
	1	P4	258.00x220.00	12.20s	12.20s	0.64kg	0.64kg								
	1	P5	236.90x180.00	11.08s	11.08s	0.61kg	0.61kg								
	1	P2	201.00x250.00	11.77s	11.77s	0.51kg	0.51kg								

What's New in TopSolid'WoodCam v6.18



This section describes the new features in the version **6.18** of **TopSolid'WoodCam**.

Template machine

Post-processor's default values in the machine template creation

In the **Misc. machine definition** window, the option to display the post-processor's dialog box is selected by default. In the case of a mixed machine pool including a number of post-processors, this feature prevents you from selecting a post-processor which is incompatible with the machine template.



Warning message if link movements and machine template are not linked

If user link movements have been assigned to a unit of the machine and this macro can no longer be found by TopSolid - for example because the directory has been changed or the file has been renamed or deleted - an error message appears in the alpha bar.



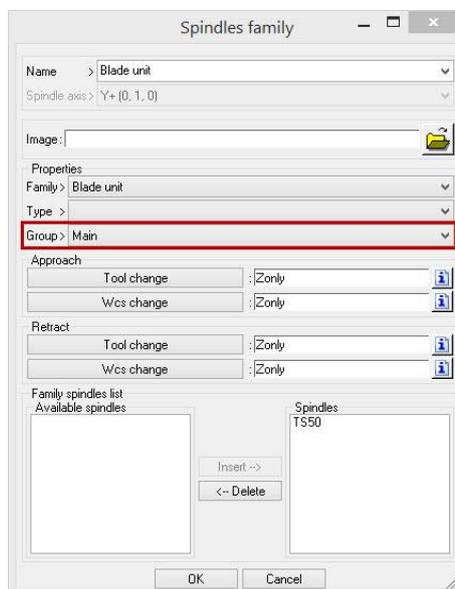
Saw machining on the bottom face

Some small-footprint machines make it possible to machine above and below the part. This strategy requires you to choose the blade unit to be used, since the frame of the saw machining is the same for an operation performed above and an operation performed below.

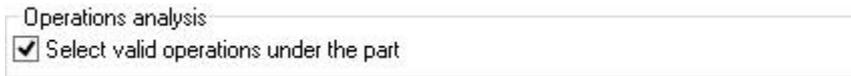
The blade unit is selected according to the priorities of operations. If an operation is defined on the main positioning, the main unit will be used.

Alternatively, if the operation is defined on the secondary positioning, the secondary unit will be used.

The selection is done in **Misc > Machine > Tool units**.

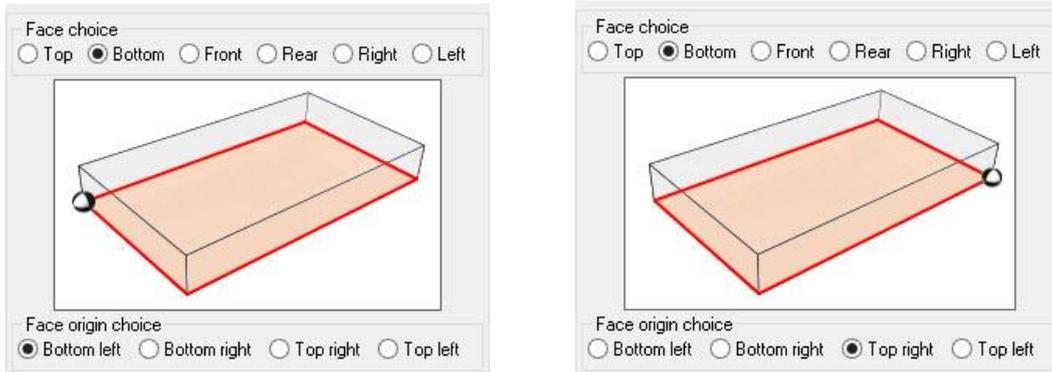


To enable the operations under the part, you need to check **Select valid operations under the part** in **Misc > Machine > Misc modifications**.



Configuring the bottom face's origin

The **Origins**  function lets you choose how to position the origin on the bottom face. As with other faces, you need to select the face to be modified (**Bottom**) and choose the position of the origin on the face (**Bottom left, Bottom right...**). The positions on the bottom face are described as if the part had rotated 180° around X. The new images show you where the origin is located on the 3D part.



When positioning the part, if the **Misc > Machine > Misc modifications > Options > Create frame under the part (if kinematics allow it)** option is selected, a frame is created on the part's bottom face.

Frames projected on the stock or finish

A new option available in **Misc > Machine > Misc modifications** allows you to offset the frame's origins to the nearest face of the stock or the finished part's bounding box. This option ensures the compatibility of **TopSolid'WoodCam** with BiesseWorks and Maestro systems which do not tolerate the tilted planes with an origin outside the part.

Group work

Group processes

Processes can now be used in a group configuration. The list of available processes is divided into three categories:

- ***** STANDARD PROCESSES *****: These processes are provided by Missler Software.
- ***** USER PROCESSES *****: These processes are created by the user locally and are stored in \$TOPCONFIG.
- ***** GROUP PROCESSES *****: These processes are created and shared by a user in group configuration. They are stored in \$TOPGROUP.

Group link movements

The principle of user or group processes has been extended to link movements. You can share link movements from TopSolid without moving folders from \$TOPCONFIG to \$TOPGROUP.

- ***** STANDARD APPROACHES *** / *** STANDARD RETRACTS *****: These link movements are provided by Missler Software.
- ***** USER APPROACHES *** / *** USER RETRACTS *****: These link movements are created by the user locally and are stored in \$TOPCONFIG.
- ***** GROUP APPROACHES *** / *** GROUP RETRACTS *****: These link movements are created and shared by a user in group configuration. They are stored in \$TOPGROUP.

Script

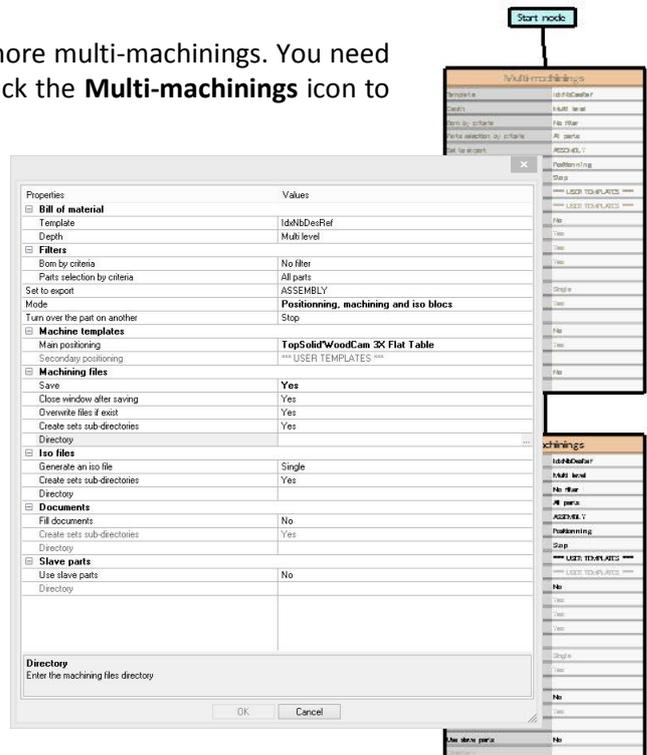
Multi-machining node

The **Script** function lets you configure and execute one or more multi-machinings. You need to create a new script or edit an existing script, and then click the **Multi-machinings** icon to create a **Multi-machinings** node.



Once the node has been included in the script, you only have to configure the BOM, filter and machining parameters.

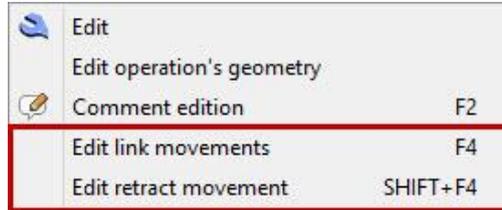
You can string together several **Multi-machinings** nodes using different filters, different bills of materials, different machines, etc.



Link movements

Calling up link movements

You can call up retract movements by right-clicking on an operation. The F4 or Shift+F4 shortcuts still work.



Comments in approach and retract macros

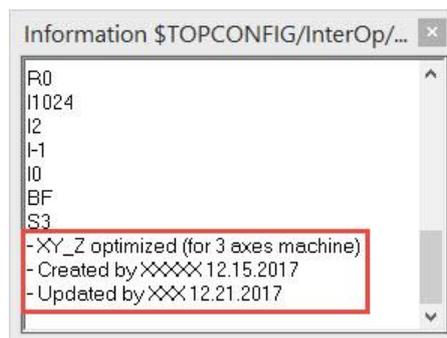
You can save a comment related to a link movement.



When this link movement is assigned to a tool unit, the comment appears when you click the **Information** icon.



The comment is displayed at the end of the ASCII file in the following form.

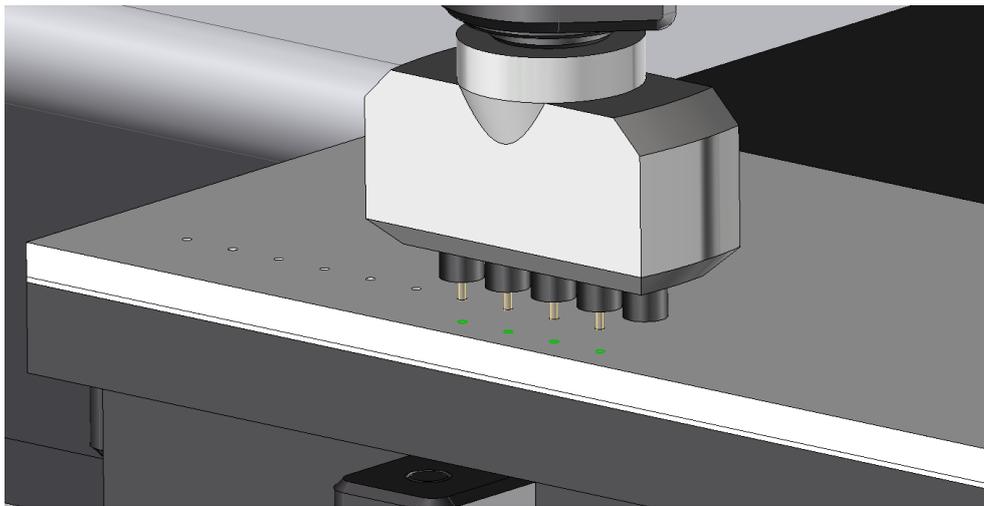


Aggregates

Managed multi-spindle drilling aggregates

To machine holes aligned with a multi-drilling aggregate when the number of holes is not a multiple of the number of tools available on the aggregate, you need to move the aggregate backwards to X times the distance per instance. For the following 5-spindle aggregate, which only has 4 mounted tools and must machine 10 holes, the aggregate will move down 3 times.

- Step 1: 4 holes are drilled, 6 holes left to be done.
- Step 2: 4 holes are drilled again, 2 holes left to be done.
- Step 3: The driven tool (1) moves down into the hole 7 of the propagation and the tools 3 and 4 of the ramp drill the holes 9 and 10 of the propagation.



This method is also available for the fixed ramps of the machine. To enable it, select **Document properties > Routing configuration > Holes** and check **Shift last hole** in the **Aligned holes** field.

Tool management

New images for tool units

New images have been added to simplify the use of the tool magazine and help differentiate the various machining units in the magazine. They are different depending on the unit type.

Below are a few examples:

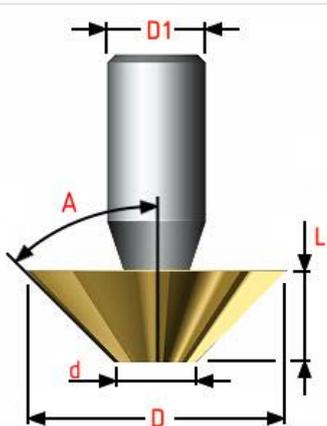
	Routing units 3X/5X
	Vertical drilling units X/Y
	Horizontal drilling units X+/X-
	Horizontal drilling units Y+/Y-
	Blade units
	Indexable units
	Aggregate

New images for tools

New images have been added to make it easier to create tools, allowing you to view the different parameters of a tool.

Each parameter listed on the left of the window appears on the tool diagram on the right.

Tool diameter (D)	> 6mm
Cutting length (L)	> 30mm
Cut angle (A)	> 65°
Second tool diameter (d)	> 0.001mm
Number of cogs	> 4
Tool shank diameter (D1)	> 6mm
Tool direction	> Right
Cutting material	> Fast steel (HS)
Machining function	> Roughing, Semi-finishing and Finishing
Internal reference	> FC-25°-D6



Tool magazine displayed as a table

The tool magazine is now displayed as a table to improve readability and allow you to change the size and position of columns, or sort the tools according to various criteria.

In the example below, the tools are sorted by diameter in descending order.

Tool	Type	Internal reference	D	L	Tool offsets
6	SLOT MILL	FR-2TA-16	16	80	6
5	SLOT MILL	FR-2TA-10	10	50	5
7	CHAMFER MILL	FC-45°	10	20	7
3	SLOT MILL	FR-2TA-8	8	40	3
4	SLOT MILL	FR-2TA-8	8	40	4
2	SLOT MILL	FR-2TA-5	5	25	2
1	SLOT MILL	FR-2TA-2	2	10	1
8					
9					
10					
11					
12					
13					
14					

Modifying tool primitives

The tool primitives called up by **TopSolid'WoodCam** have been renamed in the reduced database, which allows them to be isolated from the mechanical reduced database.

All primitives used by **TopSolid'WoodCam** are prefixed with **TSWC_XXXXXX**.

A fixed parameter has been added for the shank length, which prevents the profiles from intersecting when the tool length is equal to the tool cutting length.

Saving the PP word for the tool

The PP word assigned to a tool in the current tool magazine is taken into account when saving tool magazines. It is recovered when a magazine is reloaded by the user.

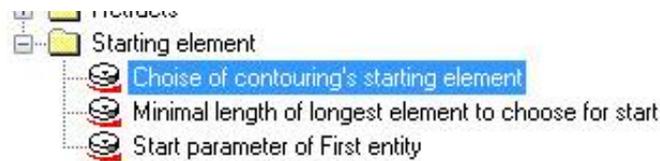
Recalculating the conical tool's dimensions

For conical tools such as chamfer mills, reverse chamfer mills, conical countersinks etc., the diameter and angle parameters are recalculated in relation to each other. This calculation is primarily based on the angle.

Contouring cycles

New starting element category

A new **Starting element** category including all the parameters related to the starting element of a contouring has been added.



Selecting the starting element for contouring

The new **Choice of contouring's starting element** parameter helps select the starting element:

- If this parameter is 1, TopSolid picks the largest linear element or the largest circular element if no segment is available.
- If this parameter is 0, TopSolid behaves as in previous releases.

Minimum length of longest element to choose for start

This parameter lets you specify a minimum length to choose the starting element. For example, you can remove from the list of starting elements the elements with a length less than twice the tool diameter. If a linear element has a length less than $2 \times \text{Tool.diam}$ and a circular element has a length greater than $2 \times \text{Tool.diam}$, TopSolid will choose the circular element.

Analysis

Searching for a tool in millimeters in an inch database

For an operation, you can search for a tool by its diameter in the database with a tolerance. As a result, the tool entered in millimeters will be found in an inch database even if the diameter is not strictly the same.

Scoring

Saw machining with scoring

A new pre-scoring depth parameter is available for all saw machinings: sawing, grooving and rabbeting.

This parameter can be configured at several levels:

- in the default values
- in the processes of the groove, rabbet and sawing operations
- in the manual sawing operation's dialog box

To avoid scoring, you just need to set the parameter value to 0.

Note: This pre-scoring parameter is processed directly by the post-processor, which means that the post-processor needs to be updated to use this parameter and the scoring pass is not simulated in **TopSolid'WoodCam**.

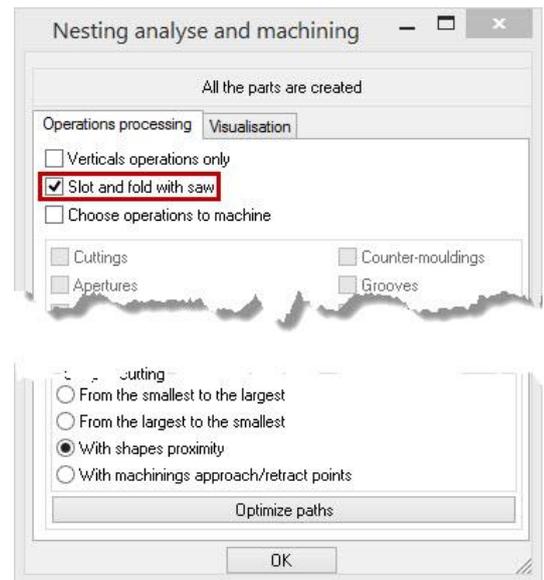
Nesting machining

Sawing operations

The 2017 release includes the ability to perform saw machinings, such as groove or rabbet operations, in nesting.

To achieve this, you need to select the **Slot and fold with saw** box. Accordingly, the frames of the groove and rabbet operations will be created on the nesting.

This option is also available for a nesting multi-machining.



Multi-machining

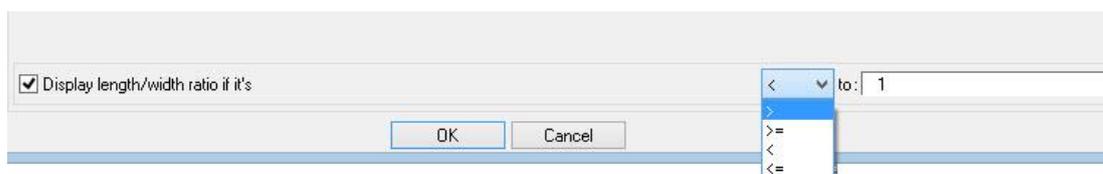
Displaying the length/width ratio

Information on the part's length/width ratio can be displayed in the list of parts to be machined.



You can configure this ratio in **Tools > Options > Routing configuration > Multi-machining**.

This information can be displayed according to whether the ratio is < ; > ; < or = ; > or = to a user value.



Displaying the ratio makes it easy for you to identify the parts whose positioning is to be modified by right-clicking and selecting **Positioning**.

Default options for nesting multi-machining

The **Positioning and machining** window's default options are those defined in the properties of the selected template.

General machining journal

The machining journal now displays the list of:

- inactive machinings
- unupdated machinings
- machinings with no path

Nesting machining journal

The nesting multi-machining journal now displays the list of unexecuted operations.

Filters in part selection (6.17)

Selection by filter has been added during the version 6.17, allowing you to filter the parts to be displayed in multi-machining.

Additional operations for nesting multi-machining (6.17)

New choices have been added to the nesting multi-machining: **Slope faces user machinings/Continuous 5 axis user machinings**.

User machining

Managed calibration/milling operation type

The operation type for multi-machining is taken into account by **TopSolid'WoodCam**. If a part contains a user machining like a calibration operation, the **TopSolid'WoodCam** calibration operation does not appear in the analysis and will not be created at the execution stage.

Ergonomics

New CAD interface supported in CAM

The tile mode is now managed by window groups. To return to a behavior similar to what it was in previous versions, when you position a part, two window groups are created when less than three documents are open, which lets you directly select the part to be positioned.

For multi-export with less than three opened documents, two groups are created: one including the design document and the start page and another one including all the generated files.

Workshop documents

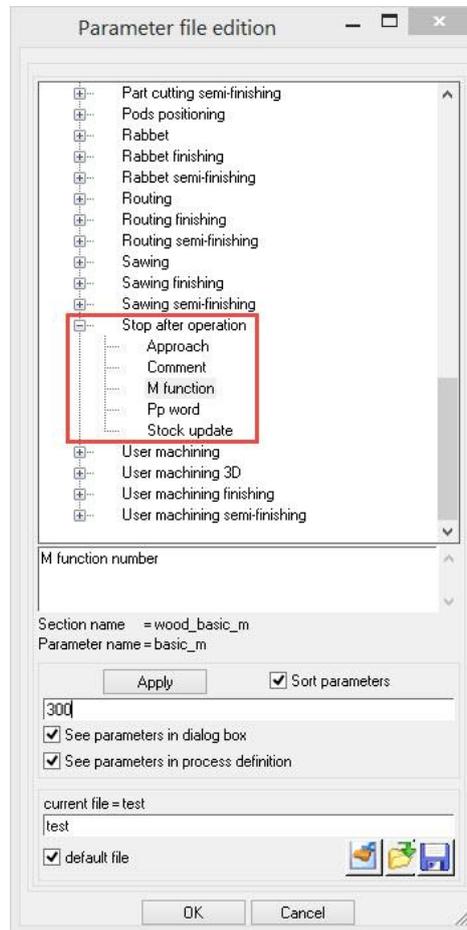
White background

As with drafting documents, you can set a white background for workshop documents. The workshop documents come standard with the white background enabled by default.

Stop parameter modification

Modifying the basic_m parameter in the wood_basic_m_break section (6.17)

In the default values, you can configure the **M function** for the basic stop operation.



Additional parameters for post-processor

Operation geometry and parameters in the PDB

CREATE PARAMETERS=YES

This PDB configuration displays the following parameters in the PDB:

- The geometry index if there are several geometries in a single operation -> **Db.Geometry.Index**
- The reference Z of the operation -> **Db.Geometry.z_ref**
- The operation depth -> **Db.Geometry.depth**
- The final pass depth -> **Db.Geometry.final_depth**
- The number of passes -> **Db.Geometry.nb_passes**
- The material height -> **Db.Geometry.h_mat**

MACHINING FEATURE=YES

This parameter lets you write the geometry in the **GEOMETRY** section of the operation.

Position of a tool holder element's coordinate system in the PDB

The PP configuration word **USE CHANGER LOC POS / NOT USE CHANGER LOC POS** has been added. The values are 999999 if the origin coordinate system of the face is not defined. The position of this coordinate system is associative. If the position of the spindle (and thus the position of the coordinate system) changes in the **TopSolid'WoodCam** document, it changes in the PDB too.

Rails and pods (6.17/6.18)

An extra line has been added to specify all the sets used:

```
*N0008 TYPE=SET      NAME=Rails&Pods Left      MACHINE_REF=L
  *N00XX TYPE=RAIL    ORDER=1      NAME=ZmiWoodMachine#Rail[1]
  *N00XX TYPE=RAIL    ORDER=1      NAME=ZmiWoodMachine#Rail[2]
  *N00XX TYPE=RAIL    ORDER=1      NAME=ZmiWoodMachine#Rail[3]
  *N00XX TYPE=RAIL    ORDER=1      NAME=ZmiWoodMachine#Rail[4]
*N00XX TYPE=SET      NAME=Rails&Pods Right     MACHINE_REF=R
  *N00XX TYPE=RAIL    ORDER=1      NAME=ZmiWoodMachine#Rail[5]
  *N00XX TYPE=RAIL    ORDER=1      NAME=ZmiWoodMachine#Rail[6]
```

The **Generate only used set positions in ISO code** option is no longer displayed in the 6.18 version. Consequently, all the sets used will be displayed.
