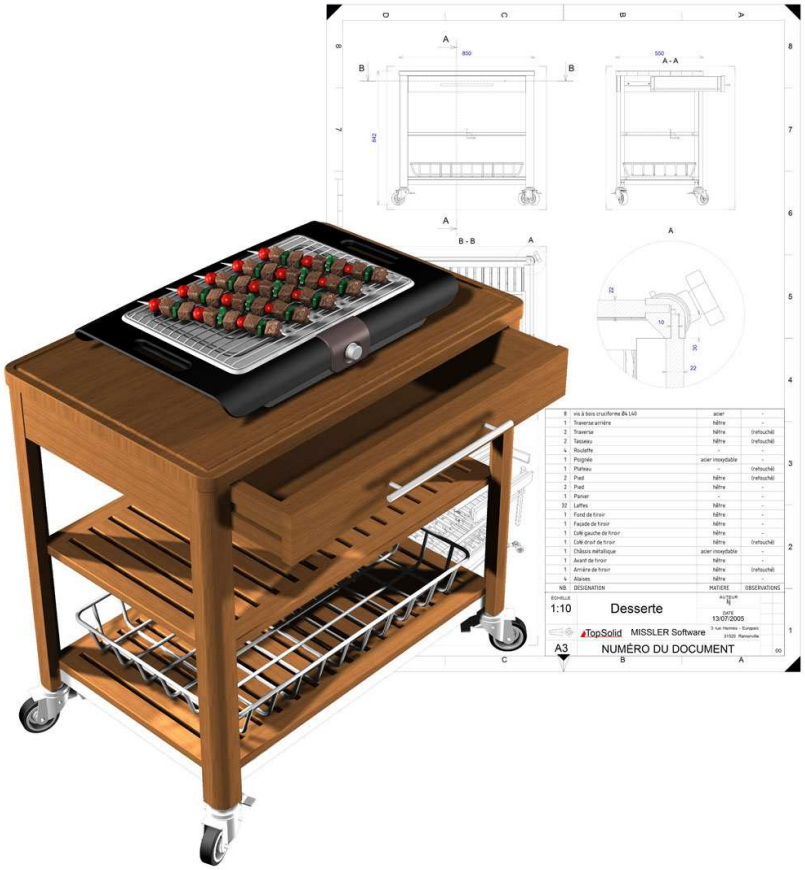
TopSolid’Wood 2011: What’s New



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Part definition– Identical coordinate systems

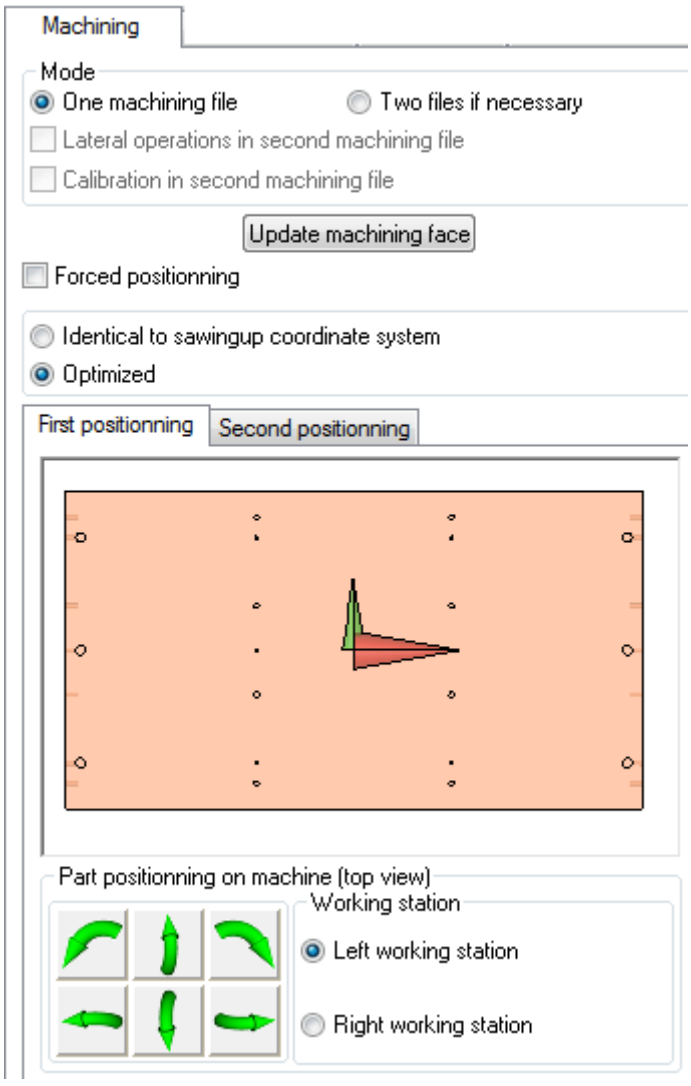
The wood define part function offer new options in order to manage the part orientation in the Sawing,Drawing and machining tabs.

Now you can set some rules for the part orientation and position for draft and machining.

The first time you define à part with the wood function, the software create 2 axis, one for the length, one for the width.
These two axis are displayed with two arrows, one red for length, one green for width (we call this the sawing up coordinate system).

Now in 6.12 the fist definition optimises the Z orientation of this sawing up coordinate system.
(The face containing the maximum of operations is show as a top view in the window).

Machining tab:



"Optimized":

The software locks the orientation (green arrows) but allows the update of the machining reference face for positioning in case of new further operations.

There is two ways to optimise:

- In define function with the option "Update machining face".
- When you export à list of part

"Identical to sawing-up coordinate system":

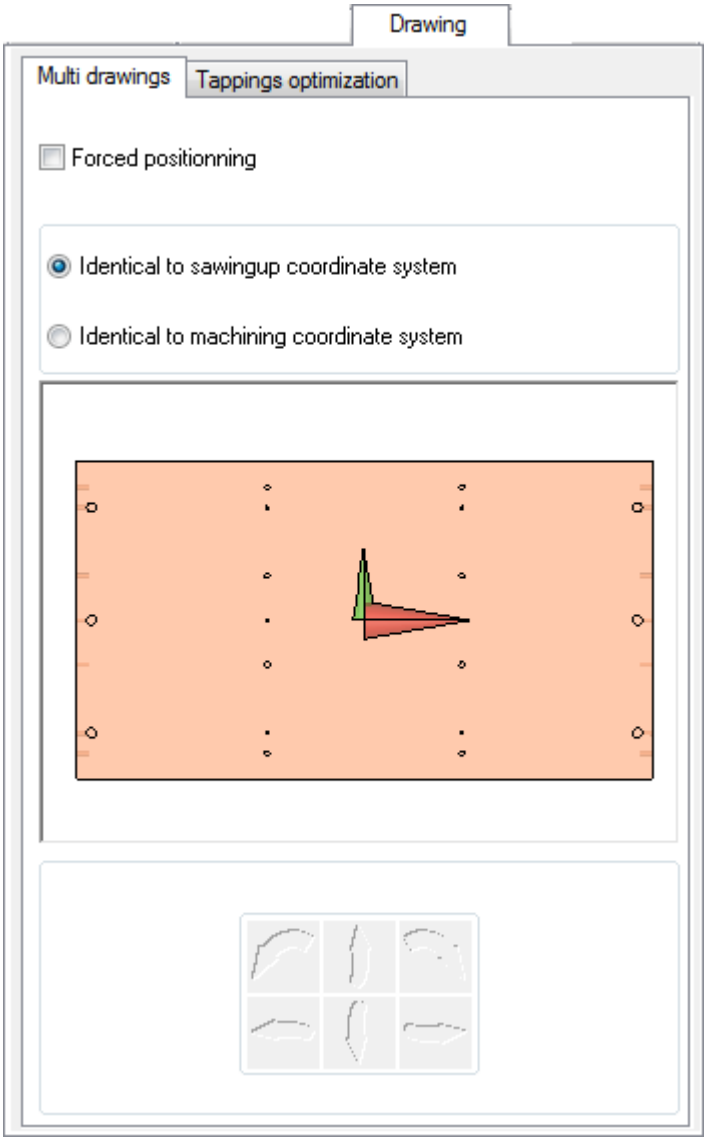
The orientation will be associative to the sawing up orientation.

"Forced positioning":

You are free to set the orientation you need by using the green arrows.

All Optimisation of machining face will not change part defined in "Forced positioning".

In the drawing tab:



"Identical to sawing-up coordinate system":

The orientation will be associative to the sawing up orientation.

"Identical to machining coordinate system":

The orientation will be associative to the machining orientation.

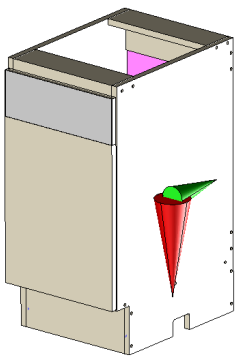
"Forced positioning":

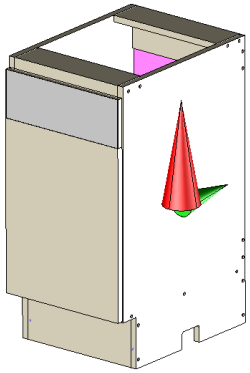
You are free to set the orientation you need by using the green arrows.

All Optimisation of machining face will not change part defined in "Forced positioning".

It's now very important to control and manage the first sawing up coordinate system.

These two examples below will provide the same cutting list but not the same orientations for machining and positioning.





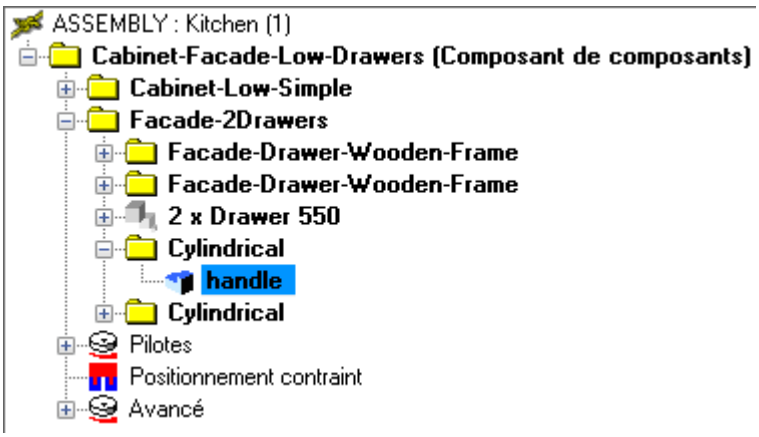
The former option "Orient main view from coordinate system" has been removed but can be put back with the configuration word ZWOO_D_DEFPART_DFT_KEEP_ISDRAFT at 1.

Define part on a component

The function "define part on a component" has been clarified:

1- When a part of a component is Defined with Detection  , the part is no longer inserted in the assembly. This process was also rectified for in-place sub-set parts.

2- A component part composed of a single part is now visible in the tree**.**

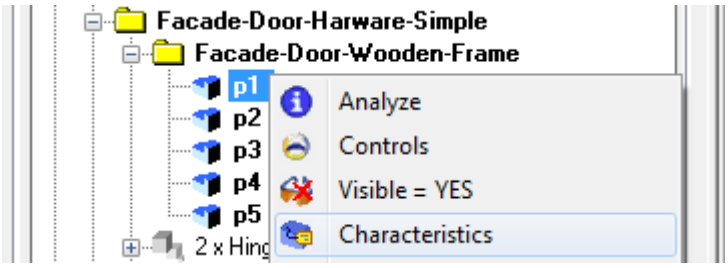


The configuration word associated is the following:

D_COMPO_ADD_CONSTITUENT_IN_TREE

It is by default at 0 in the top.cfg folder. To be visible in the tree, it has to be switched to 1.

3- The « characteristics » macro is now available on the different parts of a component and the panels.

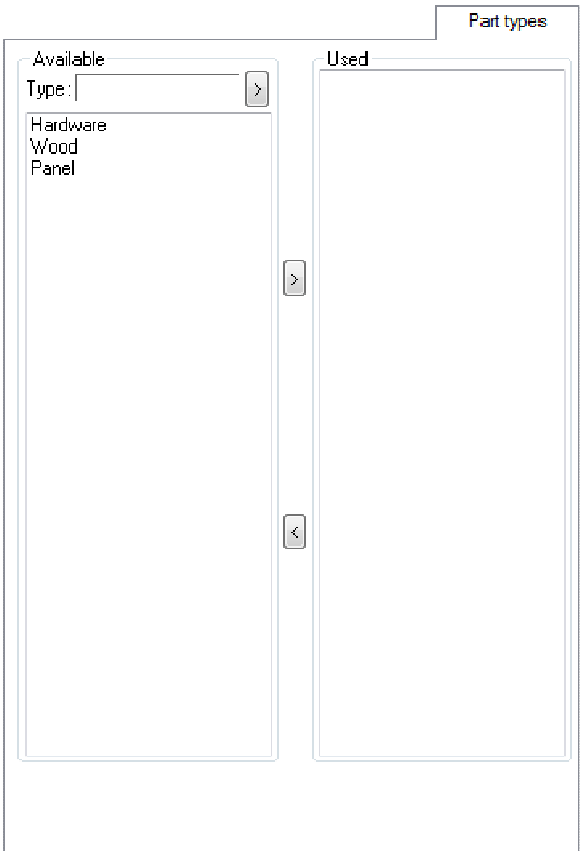


4- User properties of define part function like property systems.

Multiple part types

It is now possible to attribute several types to a single part.

 In the part definition, a tab was added:

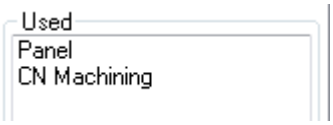


The "Available" column allows to selct predefined types in a configurable list from **Tools | Options** in **TopSolid'Wood configuration | Bill of material | Part type**, or to create one manually:

The "Used" column enables to see the different types applied to the part.

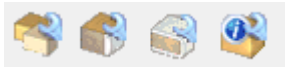
The arrows  and  enable to shift from a column to the other.

Several types can thus be applied to the same part:



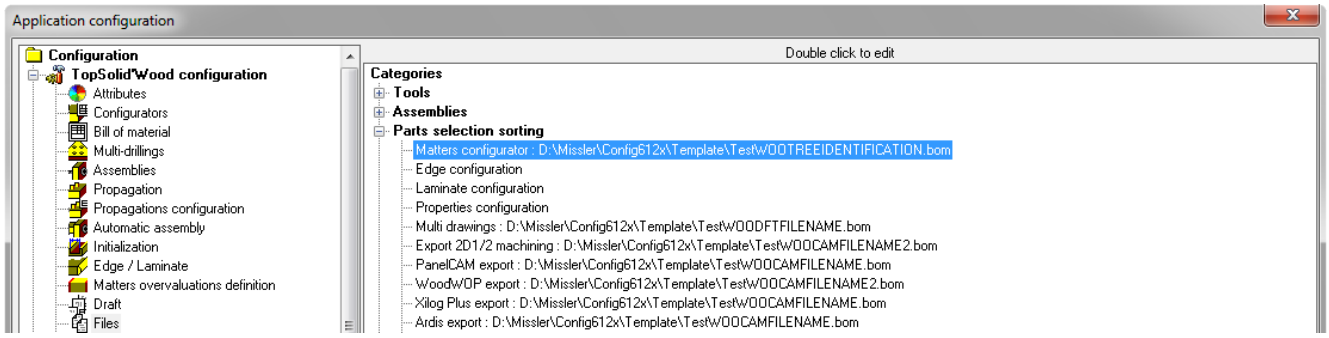
The typing of parts allows a more precise and quick selection during the multi draft, the sawing-up files or machining files exports...

Improvement of woods configurator

 It is possible to filter the bill of materials by criteria, using multi-criteria filters defined in **Tools |Options** for all the configurators, multi draft, nesting, and machining export.

The filter can be selected in the lip bar after starting off the function:

The choice of the parts name is the second improvement.
The part name can be defined in the column of bill of materials "WOO_TREE_IDENTIFICATION". This column must be in the bill of material, declared in **Tools | Options** in **TopSolid'Wood Configuration | Files | Parts selection sorting**.



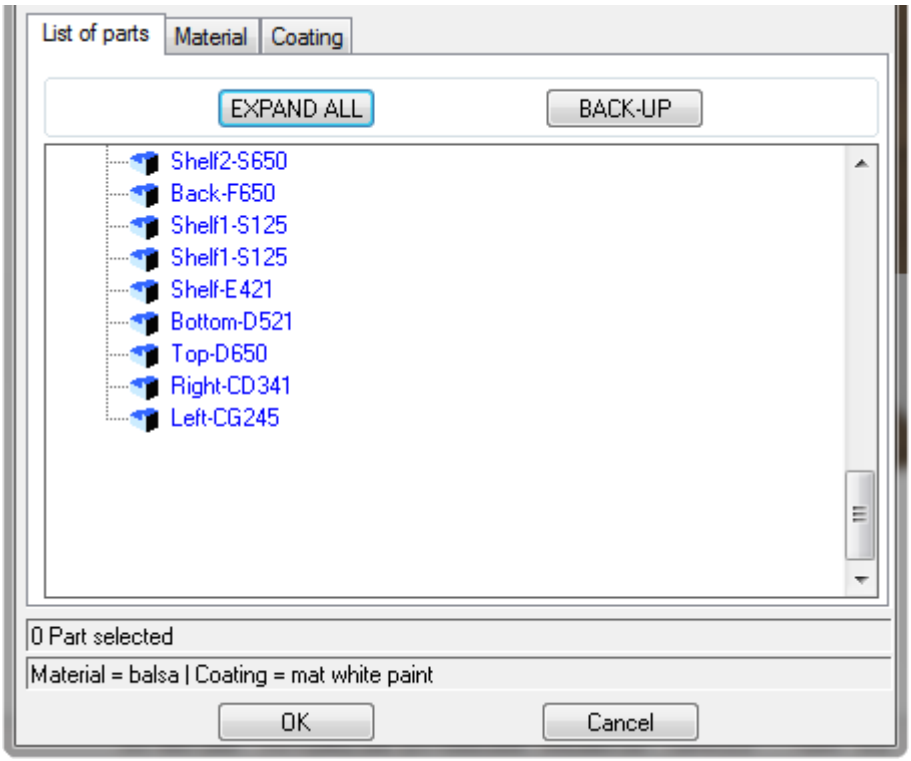
If no bill of material is declared in **Tools | Options**, the designation used is the one from the previous version.

The definition of the column is based on the same principle as « WOO_CAM_FILE_NAME » or « WOO_CODIFICATION » (see What's New 2008 p.19).

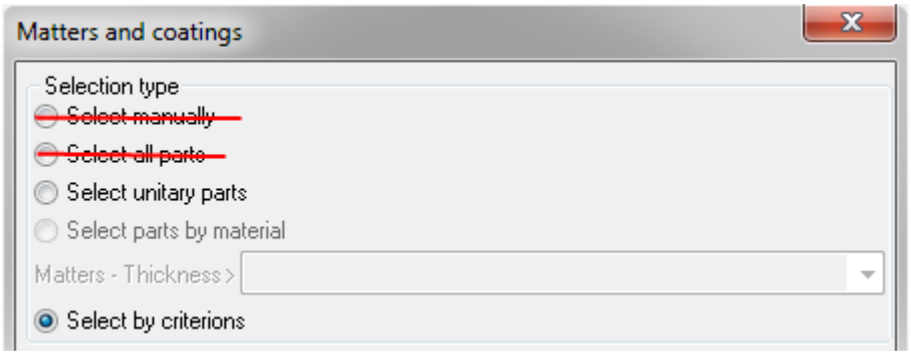
To view "designation – reference" in the selection window, the bill of material column must be made manually in the bom file as follow:

```
NAME=Part_name_configuration
"DEF=<WOO_TREE_IDENTIFICATION|$DESIGNATION$-$REFERENCE$>"
TYPE=STRING
ALIGN=LEFT
TITLE_ALIGN=LEFT
WIDTH=0.015
VISIBLE=YES
;
```

Thus, in the configurator selection window, parts are represented by their designations and their references:



Another modification was made to the different selection windows, the buttons "Select manually" and "Select all parts" have been removed.

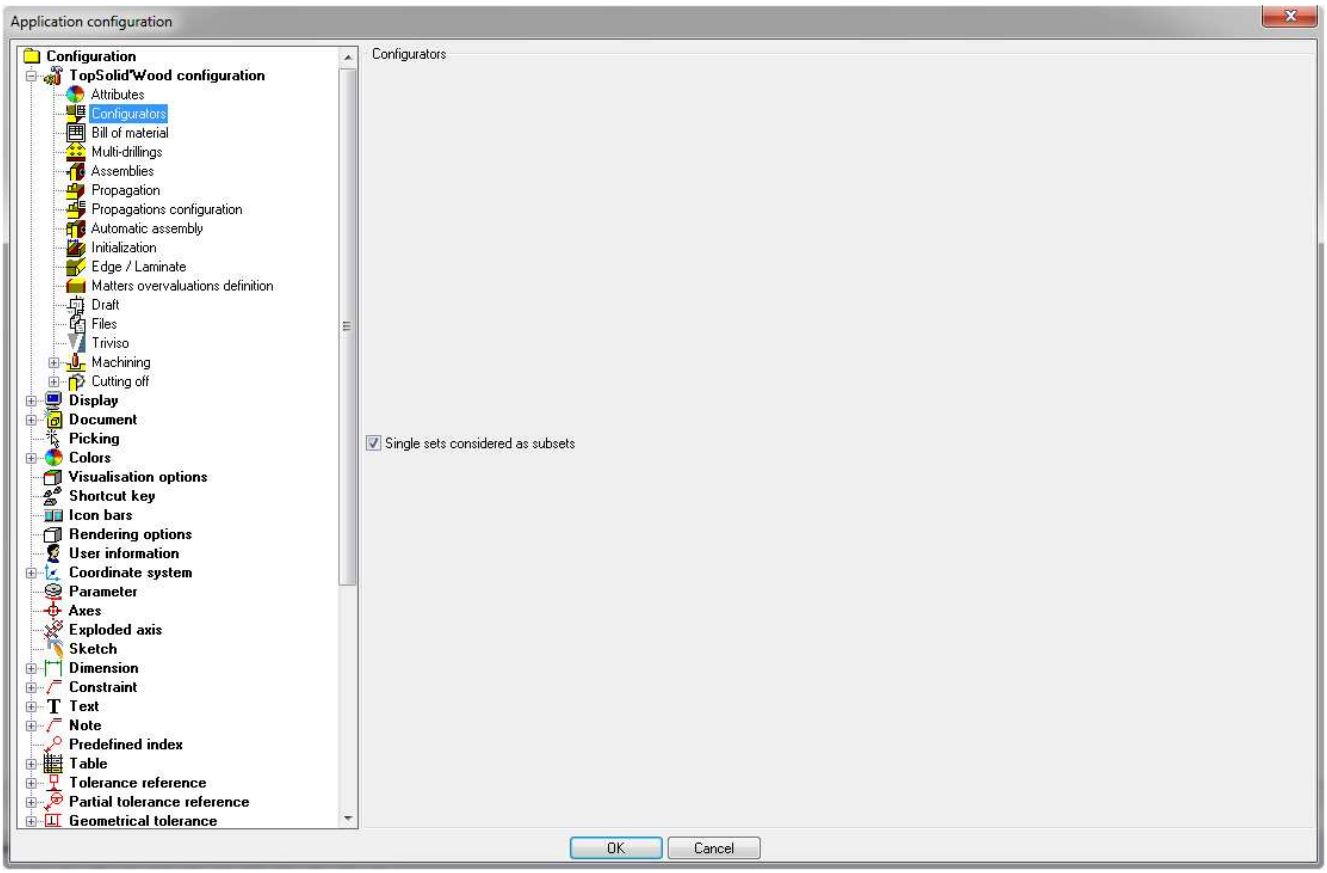


The configuration word ZWOO_U_DIAL_EXP_SELECTION_ENABLE_ALL_AND_UNITARY_PARTS (by default at 0) can restore these two options if set up at 1.

Visibility of single unit components in selection windows

All parts of a single unit set can be seen in selection windows (matters, edges, laminates, properties). This function also exists in selection windows of multi draft, machining export, and nesting.

In **Tools | Options**, in the "**Configurations**" tab, a mode has been added: "**Single sets considered as sub-sets**". When this mode is checked, the sub-sets in single units can also be unfolded.

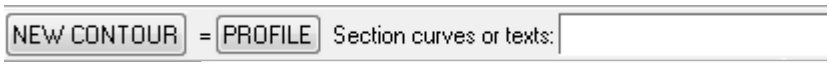


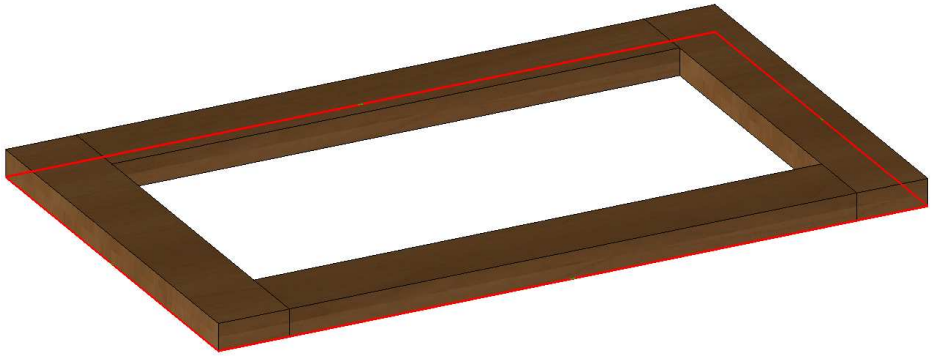
Panel on profile

 It is now possible to create a panel on one or several parts, or on unconnected supports using a profile.

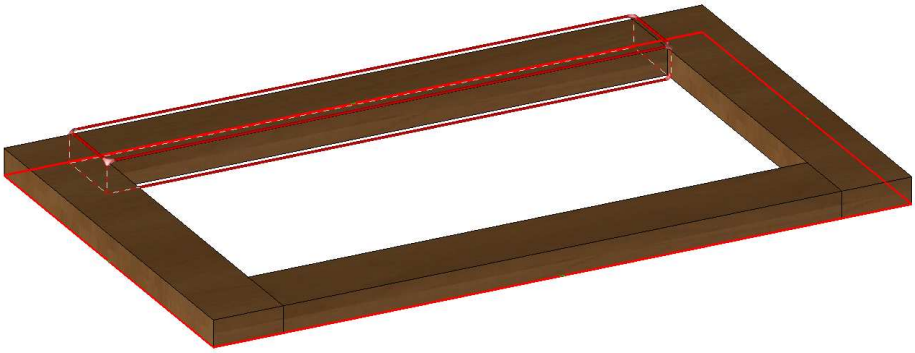
To do so, an option "On face" was added in the interface of the function panel.

By clicking on this function, the user can create a new contour in a profile or sketch mode, or by selecting one in the document.





Supports can then be selected. At least one support must compose the panel.

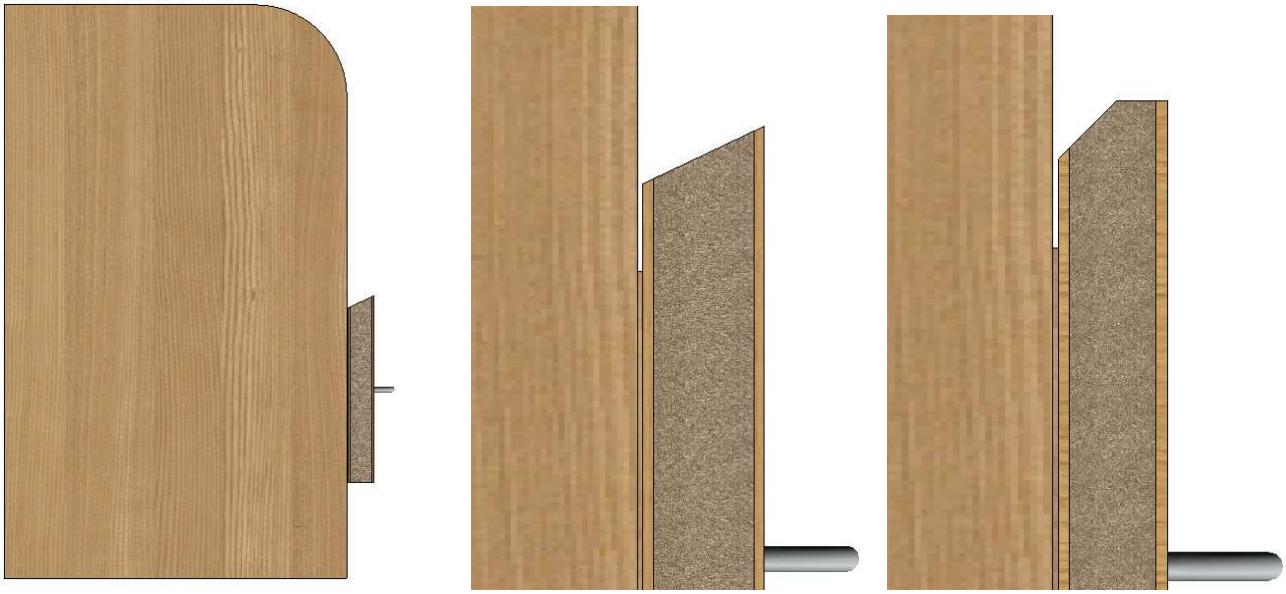


The user will then get back to the standard function panel, with the faces to laminate if the option was checked, and the edge configuration window.

 Edges and laminates are created from the profile previously selected. The profile must be drawn on the base of the panel.

Panel improvements

 The problem of laminates with inclined edges is now fixed. A part laminated with the panel function, with an inclined edge or a chamfer, will have a reported slope on the stratifies:



Over dimensions for panel supports

 It is now possible to choose the mode by default of over dimensions of the panel supports.

This setting can be made using the configuration word in the file **topzwoo.cfg** :

ZWOO_PANEL_PROP_OVERDIMS_MODE

By default, the configuration word is at 0, therefore Topsolid process will not be modified. It must be changed to 106 for instance, if the "edge shape" mode wants to be used.

Consequently, during the creation of a panel, the over dimension mode is automatically defined.

Sizes	Values	Modes	Over dime...
Length	350.0mm	edge shape	0mm
Width	250.0mm	edge shape	0mm
Thickness	19.0mm	edge shape	0mm

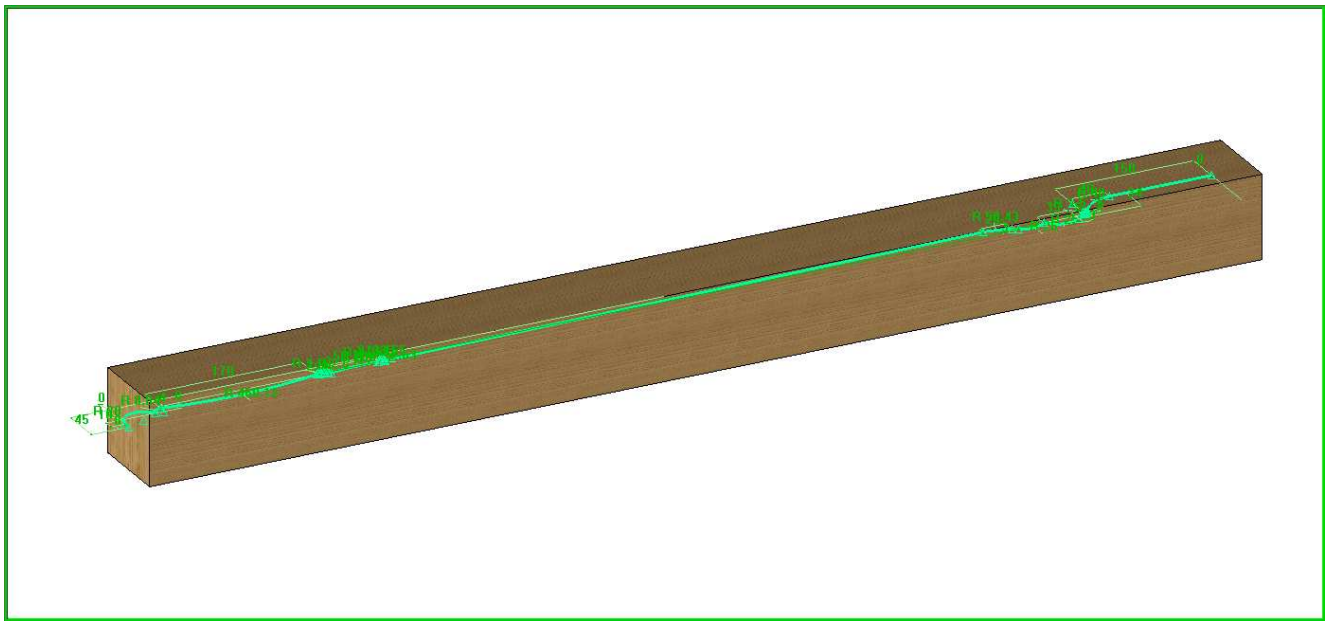
The numbers for each mode are as follow:

> 0 Additional
> 1 Proportional
> 2 Fixed
> 100 Cutting-up abacus
> 103 Machining abacus
> 106 Edge shape

 This configuration word functions even if the configuration word ZWOO_PROP_OVERDIMS_MODE (corresponding to the chosen mode during the part definition) is used.

Turning from a sketch

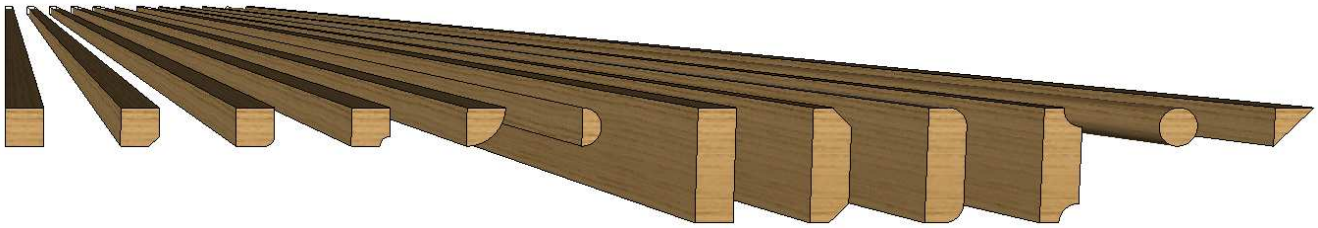

It is now possible to create a turning part using sketches. Until now, only profiles were managed.



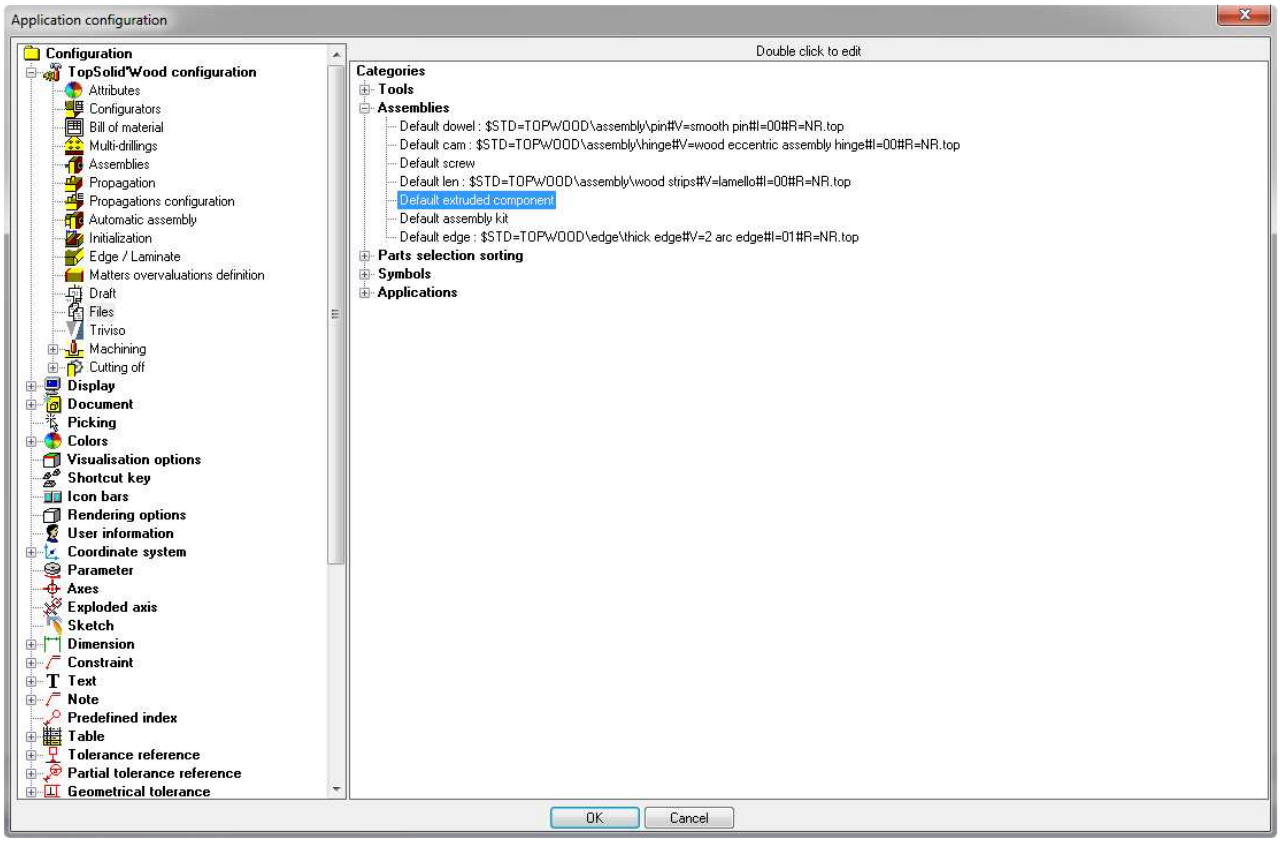
Insert of extruded components

 A new function enables to insert extruded components. This function is in **Wood | Other process | Extruded components.**

The Topsolid'Wood library was completed with many different extruded parts.



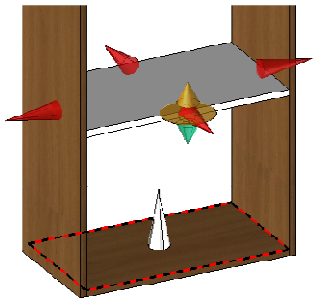
The extruded components used by default can be changed in **Tools | Options** in **TopSolid'Wood configuration | Files | Assemblies**:



Improvement of the constrained block

The function **CONSTRAINED BLOCK** has now an automatic mode 

enabling to position a block by selecting a single reference face.

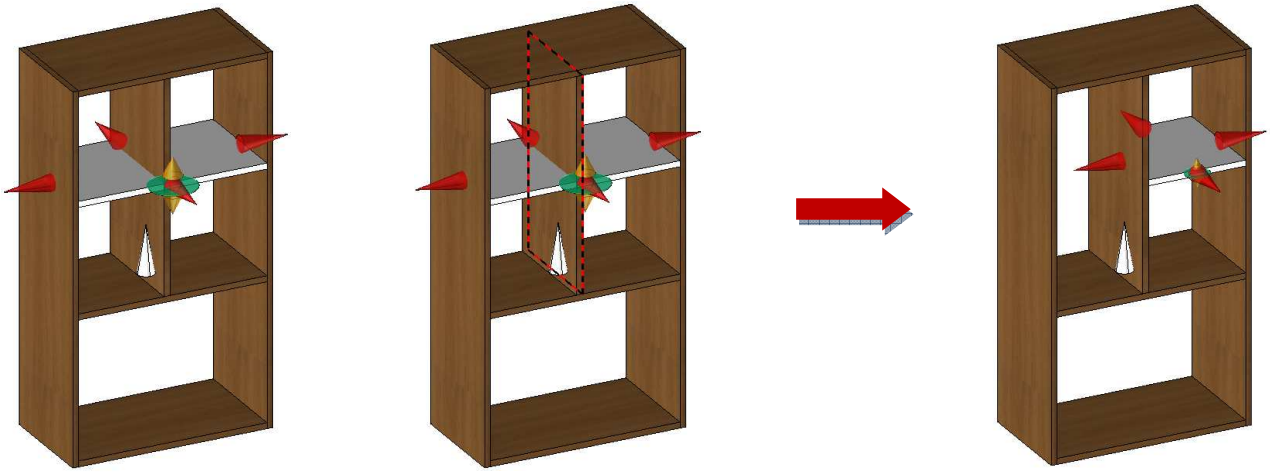


The block is automatically trimmed by the closest faces. A shelf can be designed in one click for instance.

If necessary, graphic handles allow to select manually trimmed faces and to fix the position of the block.

By clicking the left handle  , the shelf can be automatically resized by selecting another face.

Clic on handle  then redefine 



You can use Wood definition function after creation with :

D_SH_BLOCK_ZWOO_CONSTR_BOX 1

You can use points to dimension the bloc.

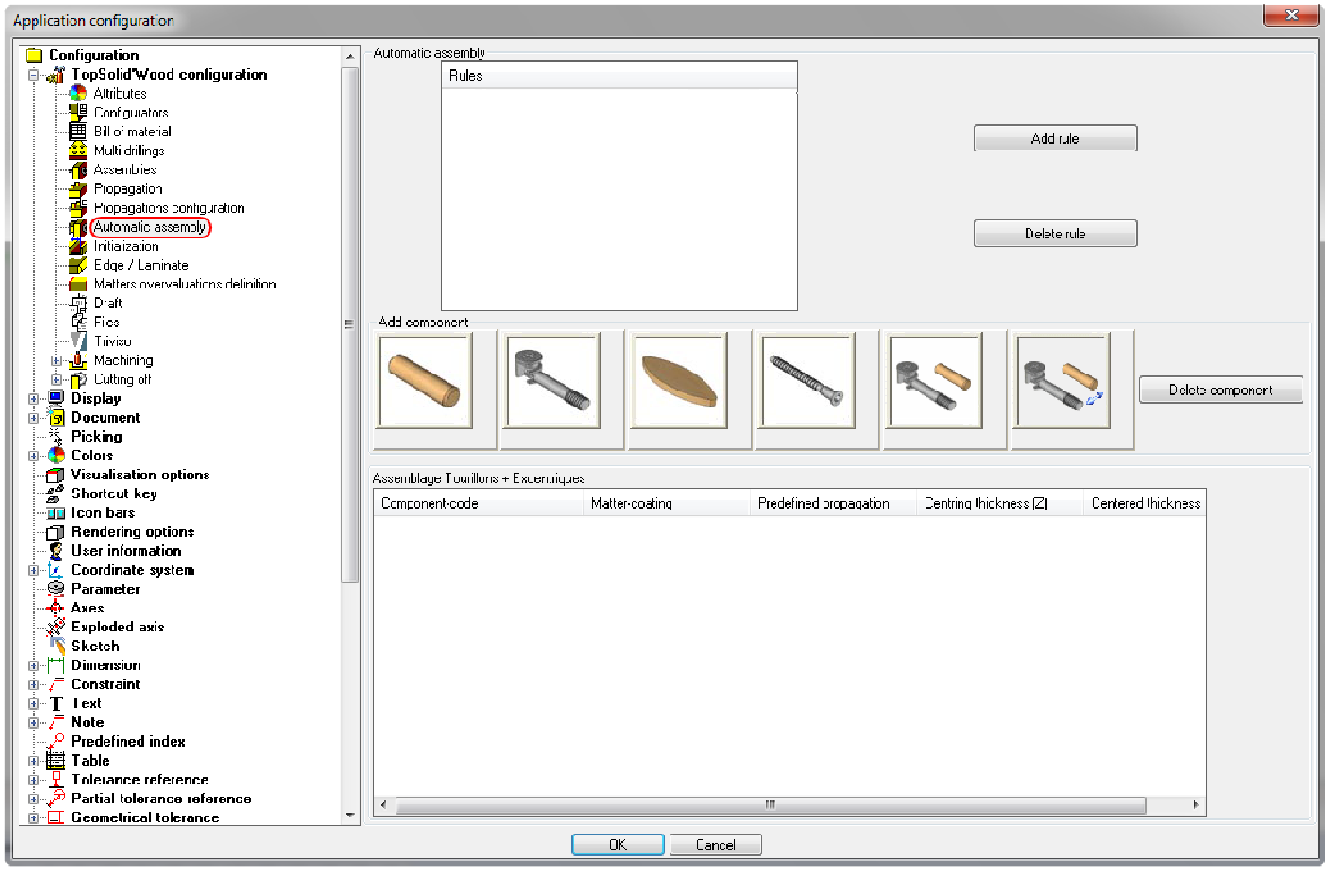
You can define the two distribution faces as drivers to use distribution in components.

Automatic assembly

 This new version contains a new function, "**Automatic assembly**", which can be found in **Wood | Other processes**.

This function works with the same operational principles of TopSolid'Wood's assembly functions (Dowel assembly, screw ...). Moreover, it enables to assemble several parts at the same time, using several components associated with several propagations.

Assemblies' configuration must be done in **Tools | Options** in **Automatic assembly**.

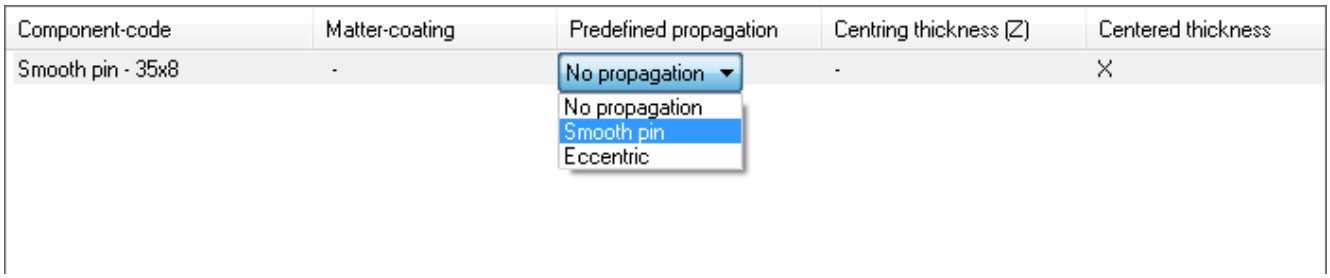


The button  enables to add and to name a rule.

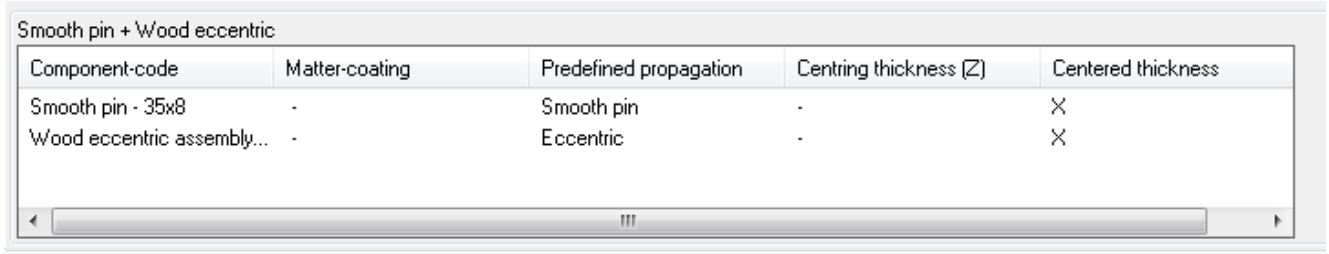
It is then possible to choose a component to use in the rule:

It is possible to use all TopSolid assembly components, as well as the assembly kits and nonlinear assembly templates (see above, p.12).

Once the component is selected, it is necessary to choose a propagation and the position of the component (centred or not).

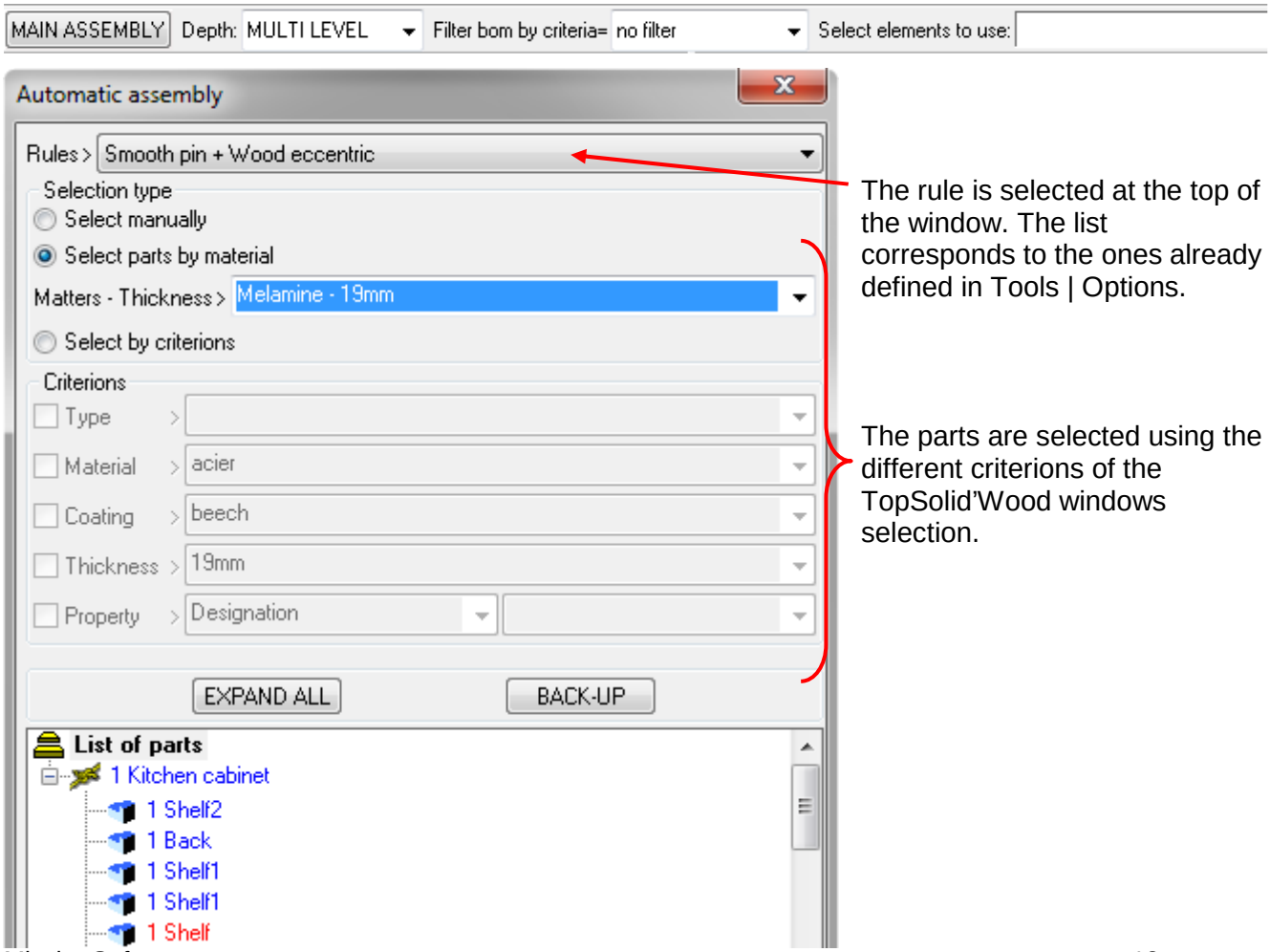


The predefined propagation list is defined in **Tools | Options | TopSolid'Wood Configuration | Propagations configuration**.
Hence, it is possible to select several components using several propagations:



Once the rule is created, the function "Automatic assembly" can be used.

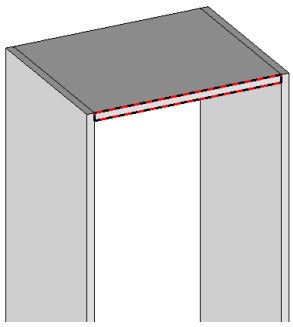
 Selection of parts is done at the beginning of the function, in the selection window commonly used in the various configurators of TopSolid'Wood.



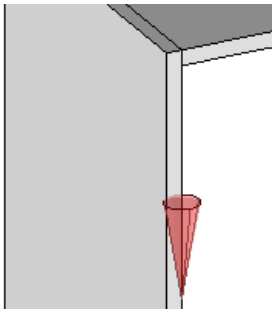
The rule is selected at the top of the window. The list corresponds to the ones already defined in Tools | Options.

The parts are selected using the different criterions of the TopSolid'Wood windows selection.

After selecting the parts to assemble, the user selects the start face for the propagation:



The centring sense should then be chosen :



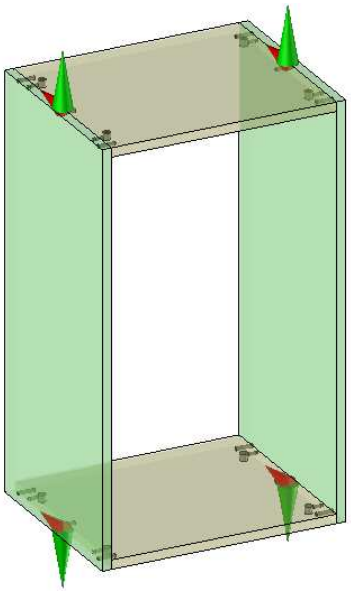
In this case, the centring sense will be vertical, it can be defined:

- **Manually:** if the arrow is going down, the reference faces will be the top faces of the parts ; if the arrow is going up, the reference faces will be bottom faces of the parts.

- **Automatically:** by using the button "AUTOMATIC CENTRING".

The user will either click Ok to confirm or outside faces/ insides faces like reference faces.

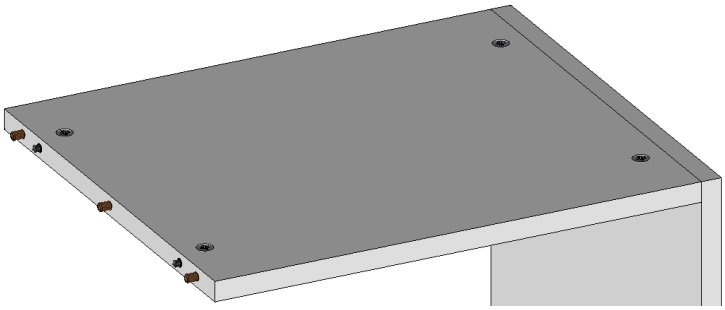
A cabinet preview allows to approve the different faces selected:



The green arrows enable to choose reference faces on a case by case basis, and eccentrics direction when they are used.

The red arrows allow to choose and confirm the direction of each propagation.

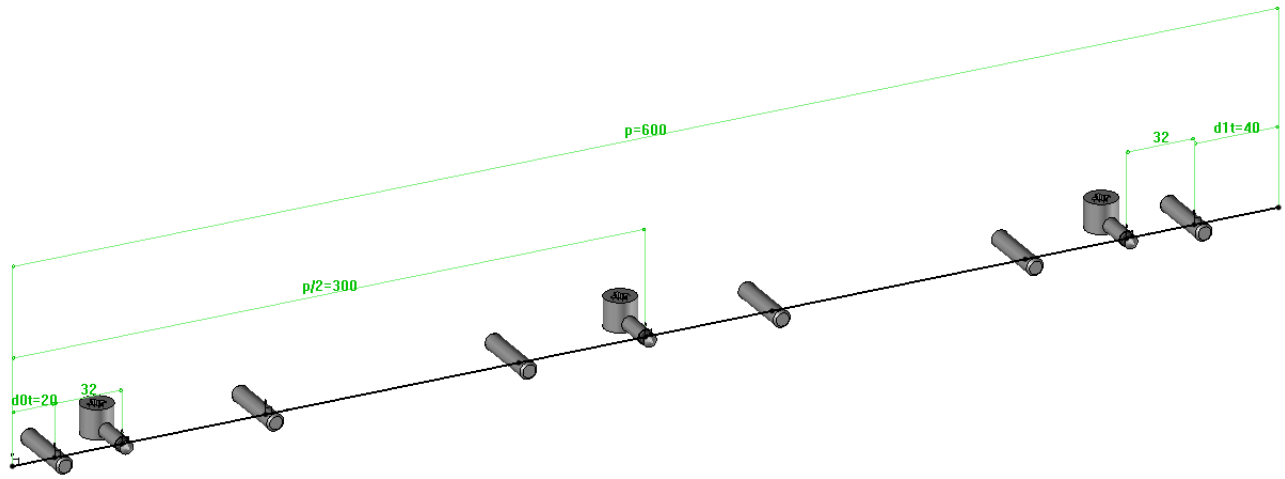
After validation every selected parts are assembled.



Nonlinear assembly

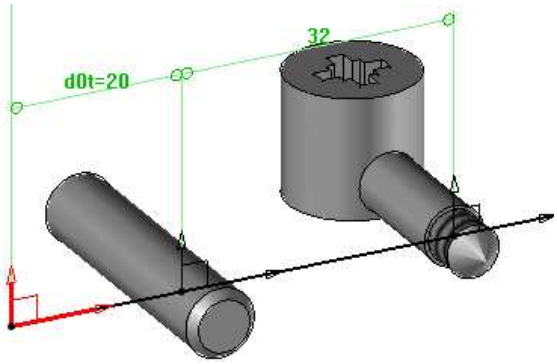
From this version, it is possible to assemble parts using nonlinear propagation templates.

The user has to build the assembly template in a new design file, creating a new parameter for the cabinet depth:



 In the function **Wood | Define | Define assembly template**, the button ADVANCED has been added.
This button enables to define the set and the driver for the cabinet depth once the assembly template is created. These two operations must be done in the component making.

The key-point must then be defined with the help of the function **Assembly | Define component | Define Key-points**:



The assembly template must be saved like a standard component using the function **Assembly | Define component | Edit/save template**.

Afterwards, this component will be able to be utilized with the functions:

 Multi-component and  Automatic assembly.

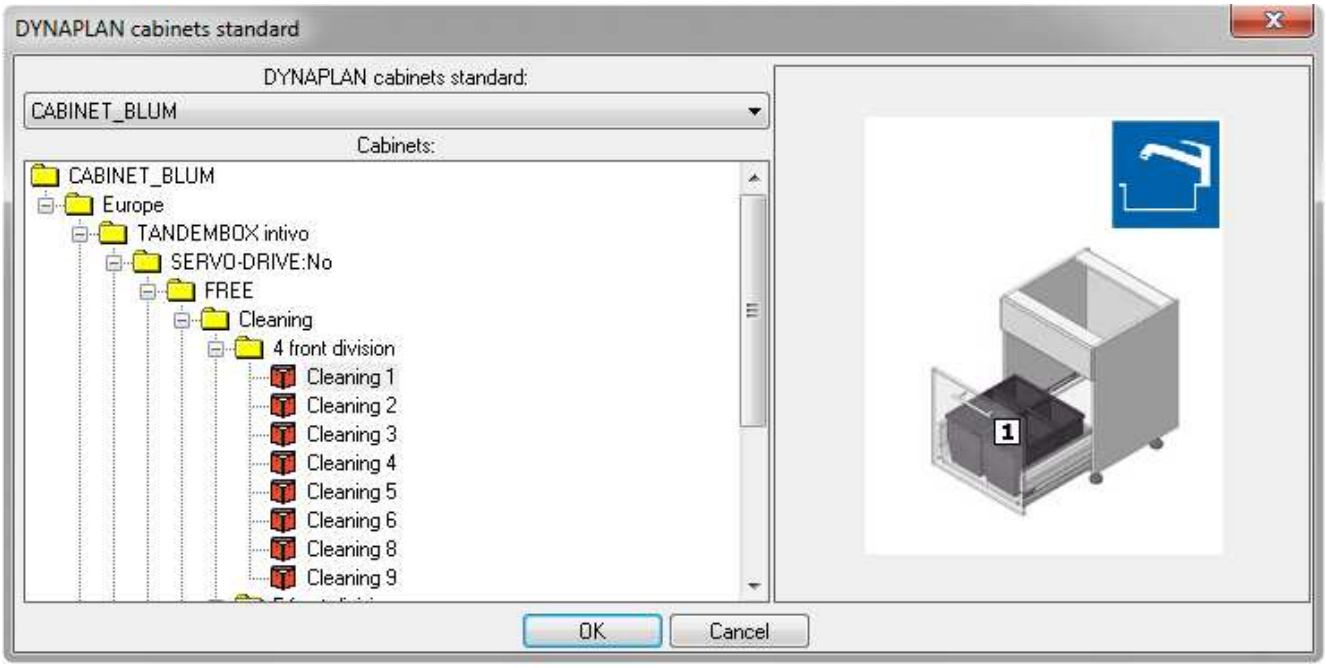
The multi-component function has been converted to include this kind of component, considering the depth parameter of the parts to assemble.

Blum library

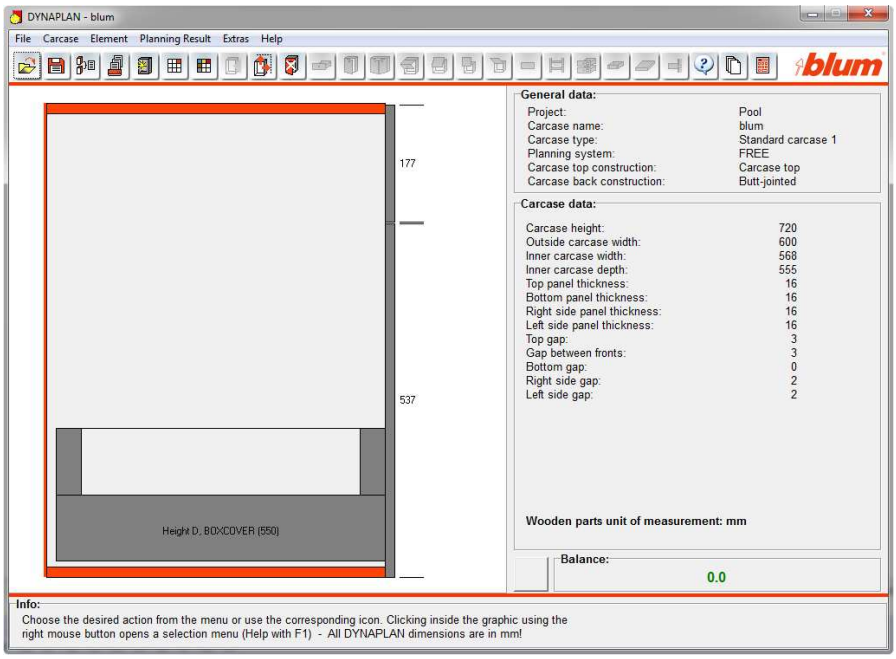
A new component library "BLUM HARDWARE" is available in this version. It can be used like a standard library, and is essential to make use of the new function « DYNAPLAN ».

 This new function can be found into **Wood | Dynaplan.** It will enable to import standard cabinets or user cabinets made in Dynaplan into TopSolid.

When starting the function, the following window opens to choose the cabinet to insert:



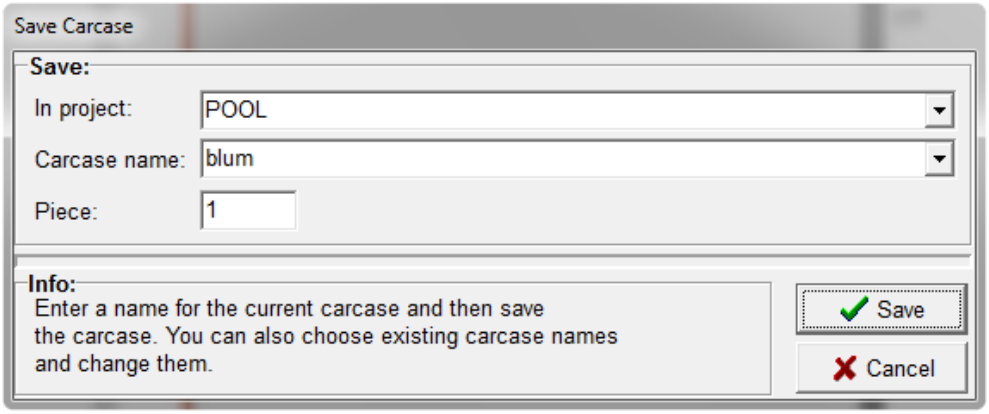
This library contains all the standards furniture of the company. Once the piece of furniture is selected, Dynaplan opens automatically.



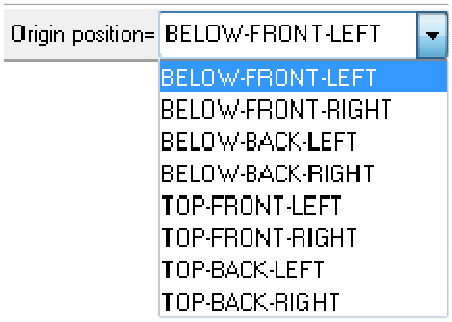
All modifications can be done from the Dynaplan interface (cabinet dimension, hardware...).

 However, the user must save the cabinet when modifications are done (non saved modifications will not be imported into TopSolid).

It must be saved in the repertory proposed by Dynaplan without modification of the name.



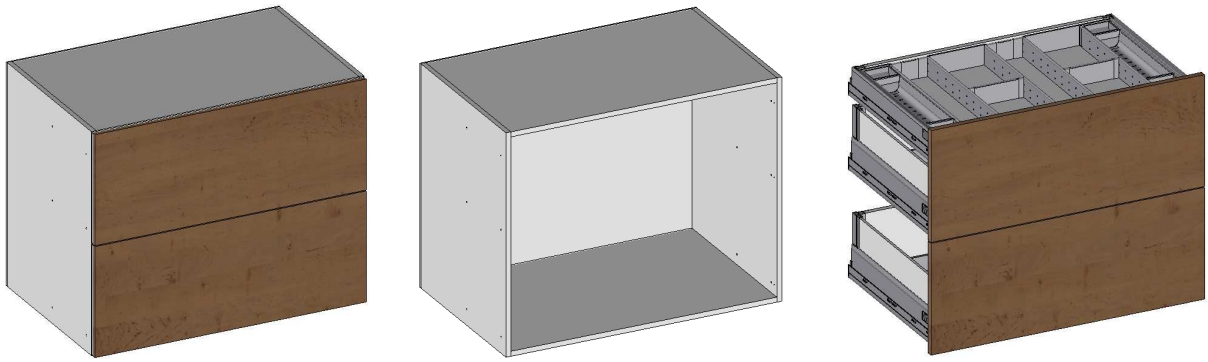
After saving the cabinet, Dynaplan can be closed. The cabinet is then positioned in TopSolid from an origin point:



And a destination point:

If there is another cabinet in the document, TopSolid'Wood offers an automatic mode enabling to select a face near the origin point.

The cabinet is then inserted in TopSolid'Wood:



Once inserted, all parts are defined and machined, and hardware is positioned.
Coordinate systems related to the imported cabinet are created to allow the placement of other cabinets.

It is also possible to create your own cabinet library.
The library declaration is done by adding a configuration word into the topzwoo.cfg :

```
ZWOO_DPROCESS_BLUM_STANDARD   BEB   "D:\Projects612\Tests\New functions\Dynaplan\BEB_CABINET"

Configuration word Standard name "Standard path"
```

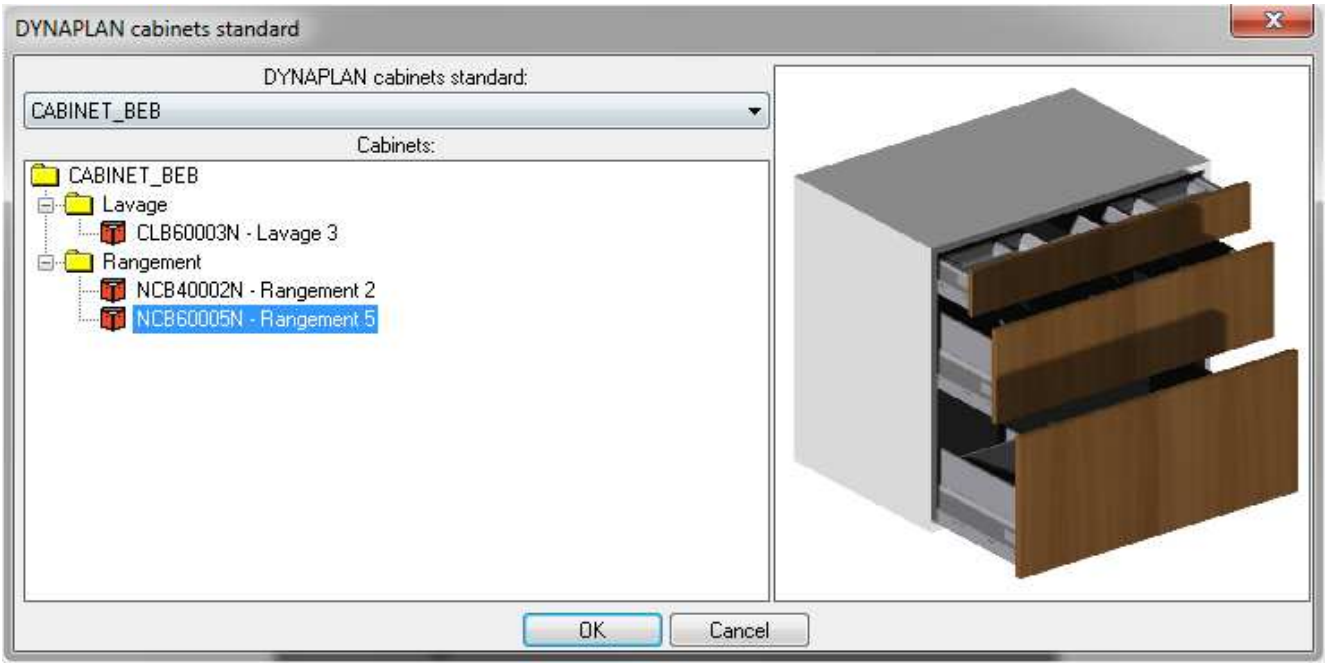
The associated repertory must contain .bpf files (Dynaplan format files) or .bxf files (Dynaplan export format files).
If the cabinet is saved in the library under a .bxf format, it must not contain hardware. Otherwise, it will not be possible to upload it in the interface.

If a bitmap file with the same name exists, it will be visible in the window selection.

By default, the realised files with the Dynaplan interface are saved in the following repertory:

C:\Program Files\DYNALOG\Dynaplan\Kommissionen\POOL

.bpf and .bxf files must be moved manually in the library created previously.
During the function starting off, the user library will be visible in the drop-down menu:



The library structure is defined by the user in Windows. The number of folders and sub folders is not limited and the components are not necessary at the same level.

 For all libraries, the cabinet modification offers to either change the definition coordinate system or to change the placement position point, or to modify a cabinet using the function CABINET.

 Caution! In the last case, Dynaplan is started off automatically. However, the modified cabinet in Dynaplan is considered as a brand new cabinet and all modifications done initially in TopSolid are lost (Edges, matters, operations...).

The designation, type, matter and coating can be set up by configuration using the following configuration words:

Designation:

```
ZWOO_DPROCESS_BLUM_TOP_DESIGNATION (défaut : “Dessus”)
ZWOO_DPROCESS_BLUM_BOTTOM_DESIGNATION (défaut : “Dessous”)
ZWOO_DPROCESS_BLUM_LEFTSIDE_DESIGNATION (défaut : “Coté gauche”)
ZWOO_DPROCESS_BLUM_RIGHTSIDE_DESIGNATION (défaut : “Coté droit”)
ZWOO_DPROCESS_BLUM_BACK_DESIGNATION (défaut : “Fond”)
```

Type:

```
ZWOO_DPROCESS_BLUM_PANEL_TYPE (défaut : “Panneau”)
ZWOO_DPROCESS_BLUM_BACK_TYPE  (défaut : “Fond”)
ZWOO_DPROCESS_BLUM_FRONT_TYPE (défaut : “Façade”)
```

Matter and coating of the cabinet structure:

```
ZWOO_DPROCESS_BLUM_PANEL_MATTER (défaut : “$melaminate”)
ZWOO_DPROCESS_BLUM_PANEL_COATING (défaut : “”)
```

Matter and coating of the cabinet back:

```
ZWOO_DPROCESS_BLUM_BACK_MATTER
ZWOO_DPROCESS_BLUM_BACK_COATING
```

Matter and coating of the cabinet facades:

```
ZWOO_DPROCESS_BLUM_FRONT_MATTER
ZWOO_DPROCESS_BLUM_FRONT_COATING
```

Matter and coating of the cabinet other parts:

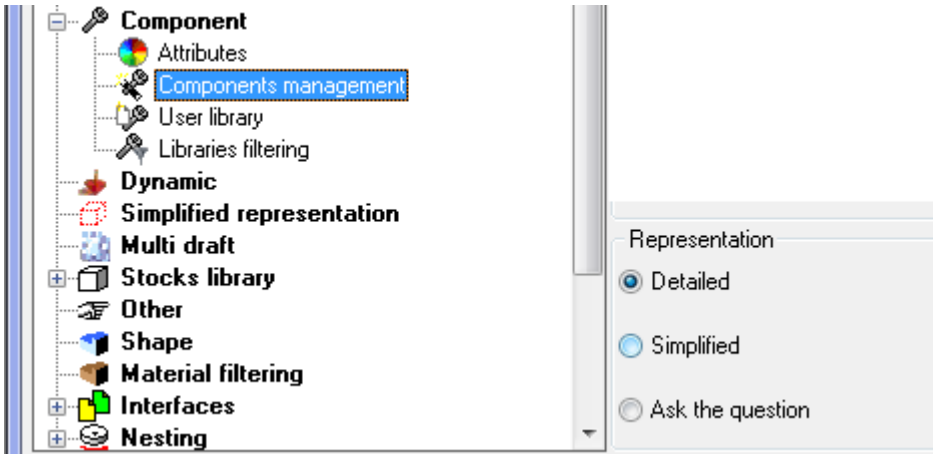
```
ZWOO_DPROCESS_BLUM_DEFAULT_MATTER
ZWOO_DPROCESS_BLUM_DEFAULT_COATING
```

These configuration words are to be placed into the topzwoo.cfg file, which is by default in: C:\Missler\Config\V612

 This function can be utilized from the time the Dynalog 2.8 version is installed.

Assembly simplification

In tool option you can configure the default inclusion representation value.
Of course you must have defined a simplified representation to be able to see it.



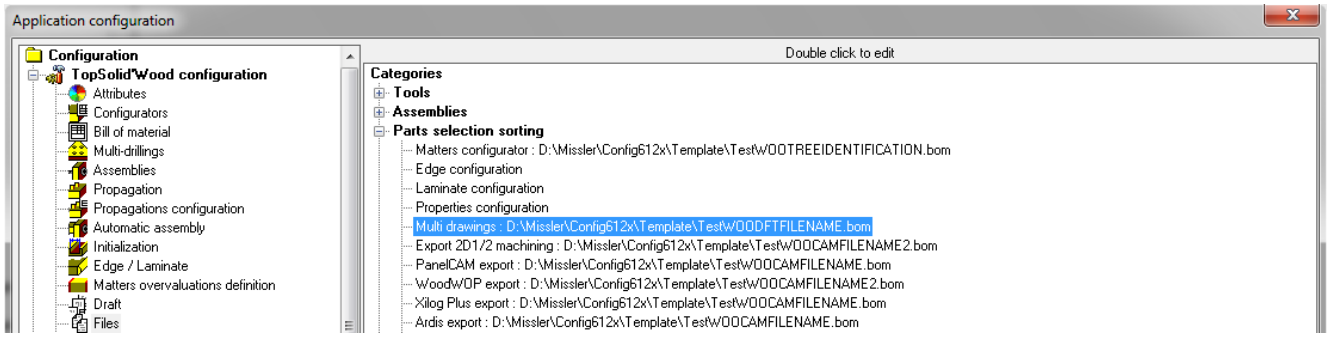
Draft file name

 It is now possible to set the multi draft file name.

Its functioning is the same as the one of the column « WOO_CAM_FILE_NAME » and so identical to the functioning of « WOO_TREE_IDENTIFICATION », as explained below.

The draft file name will be made by using a new bill of material column « **WOO_DFT_FILE_NAME** ».

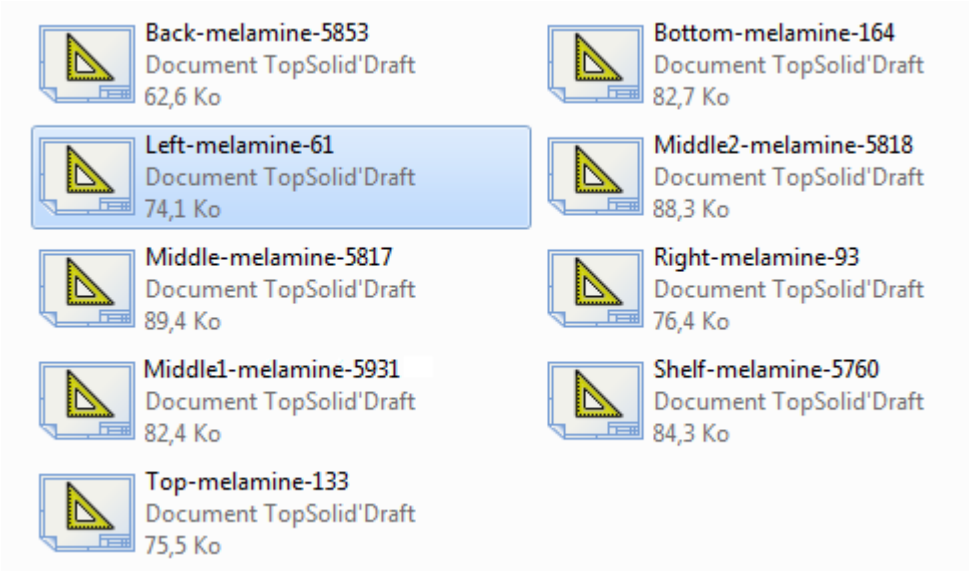
This bill of material must be declared in **Tools | Options** in **TopSolid'Wood Configuration | Files | Parts selection sorting**.



Here is an example of a draft file formatting name (designation – matter – part ID):

```
NAME=DFT_FILE_NAME
"DEF=<WOO_DFT_FILE_NAME|$DESIGNATION$-$MATTER$-$ELEMENT_IDENTIFIER$>"
TYPE=STRING
ALIGN=LEFT
TITLE_ALIGN=LEFT
WIDTH=0.015
VISIBLE=YES
;
```

Hence, all drawings will have automatically the same name:



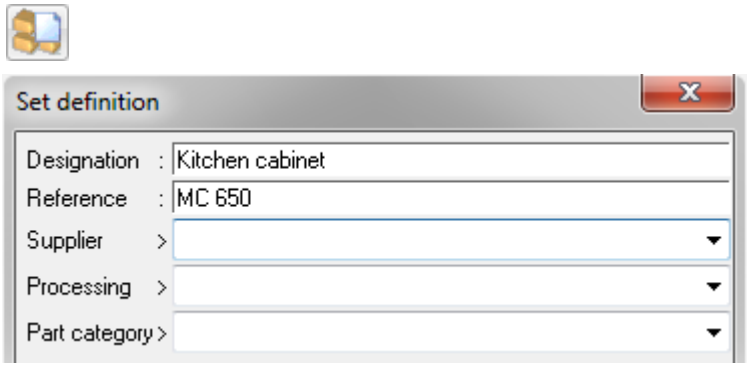
Owner designation and reference for in-place parts

It is now possible to use the owner information (designation, references ...) for in-place designed parts.

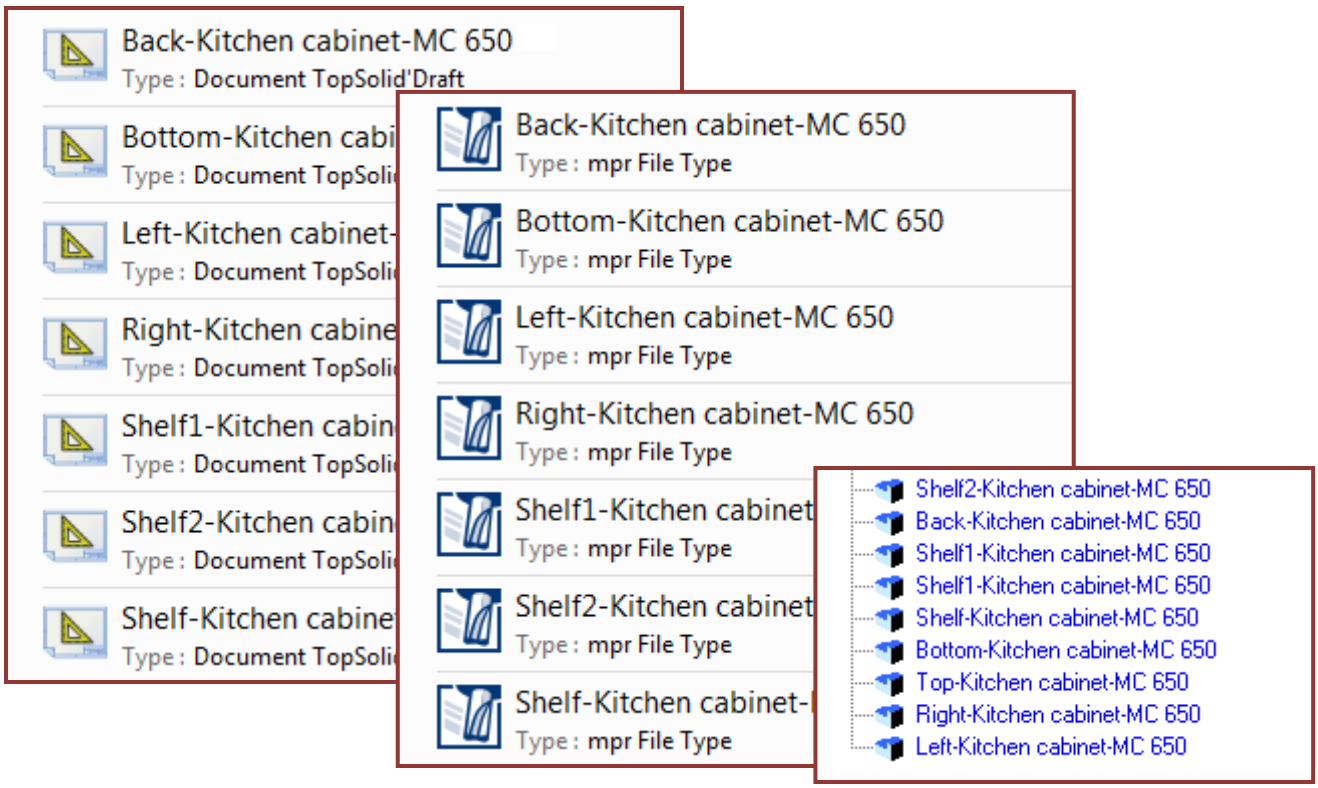
The owner notion corresponds most of the time to the name or reference of the project or to the cabinet. Until now, this information could not be retrieved from the exports, configurators, and multi drafts.

This notion corresponds to the properties of root (or direct) owner designation (or reference).

Hence, this property enables to determine draft file names, machining file names or tree identification:



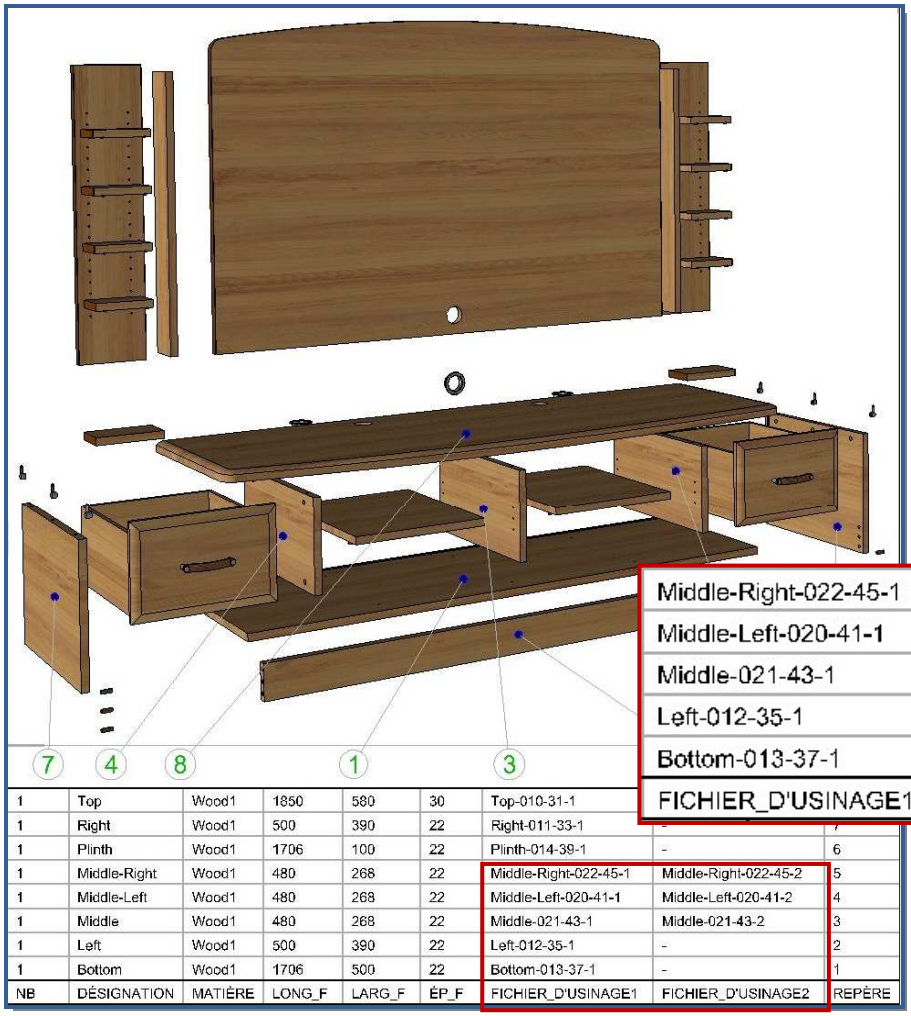
WOO_DFT_FILE_NAME|**$DESIGNATION$-$DESIGNATION|OWNER_PRJ$-$REFERENCE|OWNER_PRJ$**
WOO_CAM_FILE_NAME| " " " "
WOO_TREE_IDENTIFICATION| " " " "



File name recovery for the 2nd machining file name

TopSolid'Wood 2011 enables the user to manage the name of the second machining file the same way as the first file name.
The file name is calculated in a bill of material column with the following property:
DEF=<WOO_CAM_FILE_NAME2|. . .>

Its functioning is identical to the one of the column « WOO_CAM_FILE_NAME ».
The column set up defines the file name structure.
Hence, the user can choose the machining file name for parts with two machining files:



NB	DÉSIGNATION	MATIÈRE	LONG_F	LARG_F	ÉP_F	FICHIER_D'USINAGE1	FICHIER_D'USINAGE2	REPÈRE
1	Top	Wood1	1850	580	30	Top-010-31-1		
1	Right	Wood1	500	390	22	Right-011-33-1		
1	Plinth	Wood1	1706	100	22	Plinth-014-39-1	-	6
1	Middle-Right	Wood1	480	268	22	Middle-Right-022-45-1	Middle-Right-022-45-2	5
1	Middle-Left	Wood1	480	268	22	Middle-Left-020-41-1	Middle-Left-020-41-2	4
1	Middle	Wood1	480	268	22	Middle-021-43-1	Middle-021-43-2	3
1	Left	Wood1	500	390	22	Left-012-35-1	-	2
1	Bottom	Wood1	1706	500	22	Bottom-013-37-1	-	1

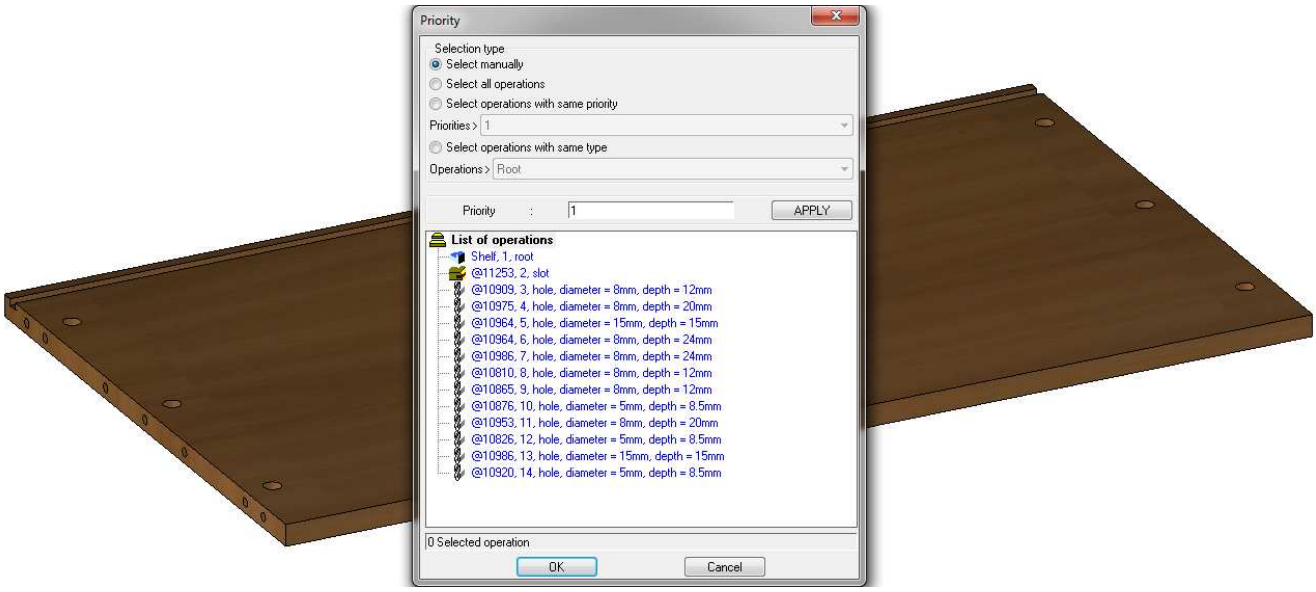
FICHIER_D'USINAGE1	FICHIER_D'USINAGE2	REPÈRE
Middle-Right-022-45-1	Middle-Right-022-45-2	5
Middle-Left-020-41-1	Middle-Left-020-41-2	4
Middle-021-43-1	Middle-021-43-2	3
Left-012-35-1	-	2
Bottom-013-37-1	-	1

Below is the extract of the bom file enabling to manage the name of the two machining files:

```
NAME=Machining_file_1
"DEF=<WOO_CAM_FILE_NAME|$DESIGNATION$-$REFERENCE$-$ELEMENT_IDENTIFIER$-1>"
TYPE=STRING
ALIGN=LEFT
TITLE_ALIGN=LEFT
WIDTH=0.015
VISIBLE=YES
;
NAME=Machining_file_2
"DEF=<WOO_CAM_FILE_NAME2|$DESIGNATION$-$REFERENCE$-$ELEMENT_IDENTIFIER$-2>"
TYPE=STRING
ALIGN=LEFT
TITLE_ALIGN=LEFT
WIDTH=0.015
VISIBLE=YES
;
```

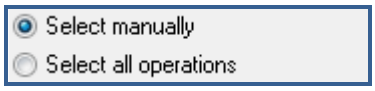
Operations priority for machining and exports

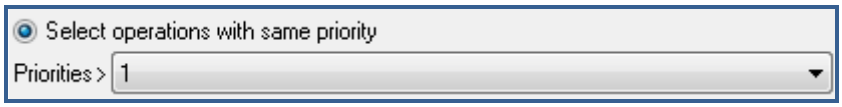
In the 2011 version of TopSolid'Wood, a new function "Operations priority" was added. It allows to organise the operation priority for machining. This function can be found into **Wood | Operations Priority.**

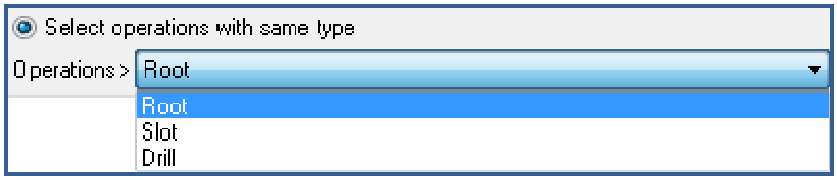


This function starts with the selection of the part on which operations will be prioritized:

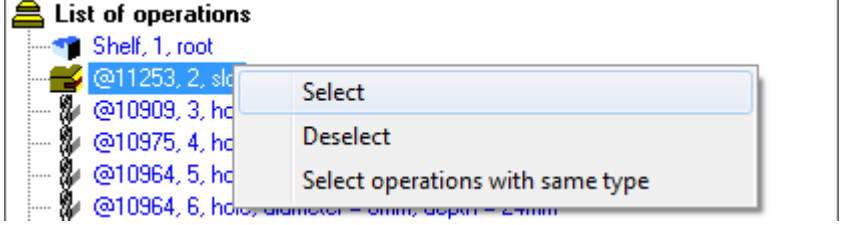
Operations can then be selected manually, by priority or by type:



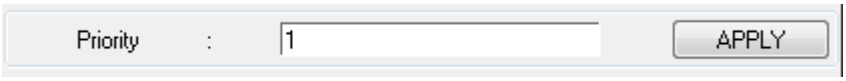




The manually selection (and deselection) can be done by right clicking in the operation list:



The priority number is then attributed from the button APPLY.

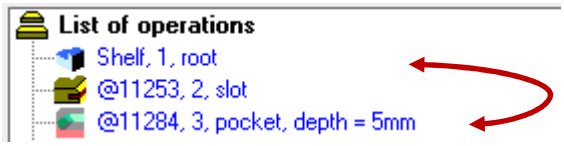


Priority 1 corresponds to the first operation to be done.
Priority 0 enables to cancel the priority applied to the part.

The tree operation list is sorted in increasing priorities, either the most prioritized or the least prioritized. It is also updated at each priority modification.

If two operations use the same priority number, they will be completed according to the way they are sorted in the tree.

The "drag and drop" also enables to sort the operations in the tree.



Machining exports export operations according to their priorities. Operations without priority are the last to be exported.

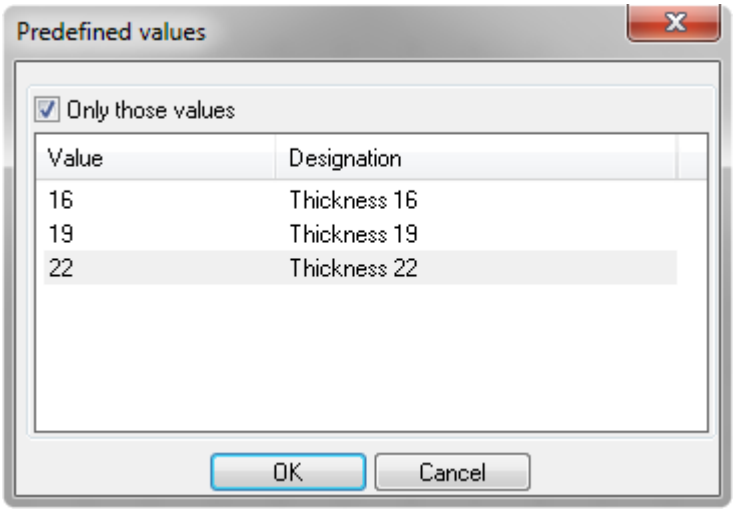
This function allows to prioritize the machining order during the export to TopSolid'WoodCam, machining interfaces (Woodwop, Xilog, Panelcam) and DXF 2D$^{1/2}$ interfaces.

Design – Parameters improvements

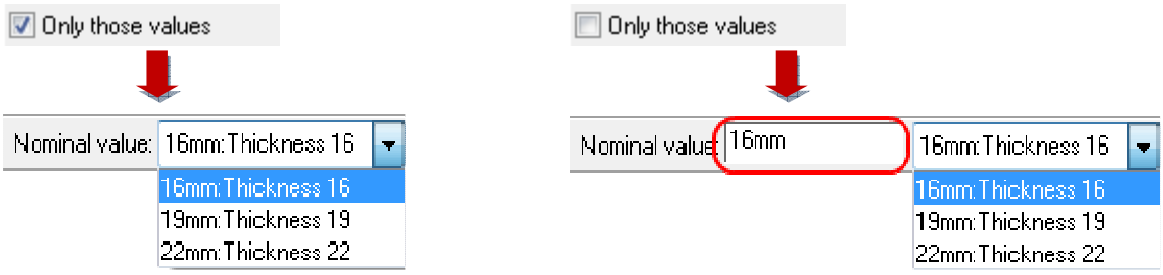
List of predefined values:

It is now possible to predefine the values of the parameters
During the creation or modification of a parameter in the advanced options a new button PREDEFINED VALUES=0 was added.

It enables to define a list of predefined values for a parameter, and to associate a designation to this value:



The option Only those values allows to choose among predefined values only, during the modification of a parameter or the insertion of a component containing drivers with predefined values.



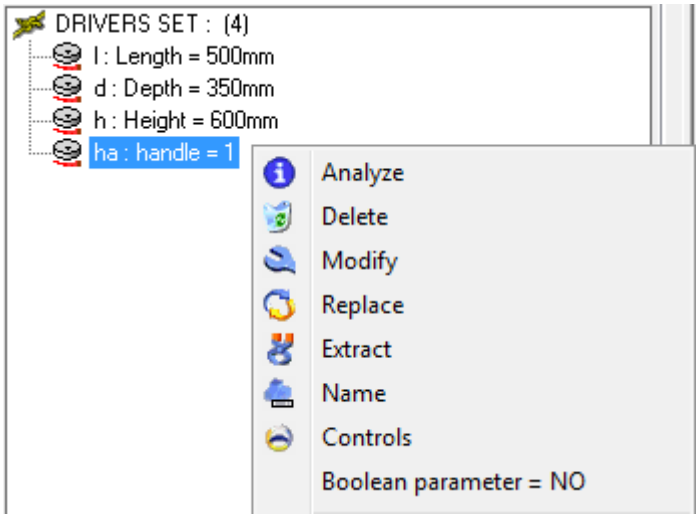
Predefined value of parameters can also be determined from the function **Parameters | Edit list** using the button Predefined values.

Expression analyser : See Design news documentation:

Boolean parameters:

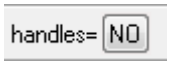
The Boolean driver parameter enables to manage a driver parameter at two values (True/False).

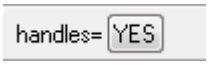
This type of parameter is particularly interesting to deactivate a component or an operation.
Boolean driver parameter defines itself in the construction tree when editing the drivers set:



To be defined as Boolean, the driver parameter must not have unities.

When the drivers is asked, you'll now have a button (Yes/No).

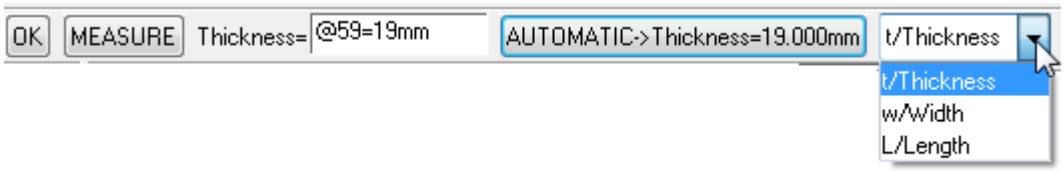




Automatic Drivers:

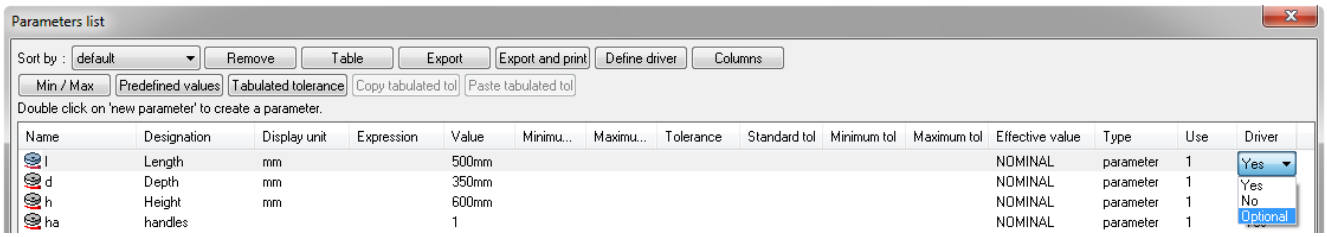
When a asked driver finds the same name in the assembly you'll find a new option AUTOMATIC.

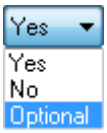
You'll also find a new list with some parameters of the document.



Optional driver parameter:

In the parameters list, it is now possible to make optional to fill in the driver parameter.





Consequently, the driver will not be enquired during the component insertion. However, it will appear in the drivers set of the tree, and could thus be modified.

Design – Automatic reference (TopSolid'Quote need this)

A bill of material can help you to generate references, in this example it's possible to use the property part type to say how to generate references for parts.

Like with the woo codification you can create rule to make your own reference.

```
NAME=REFERENCE
"DEF=<REFERENCE><WOO_CODIFICATION|$MATTER$_$PART_THICKNESS|Unit:4|Prec:2$TY
PE(Panel)><WOO_CODIFICATION|-$DESIGNATION$-
$PART_LENGTH|Unit:4|Prec:2$TYPE(Profile)><WOO_CODIFICATION|$DESIGNATION$-
$COMPO_CODE$TYPE(Profile)>"
TYPE=STRING
ALIGN=LEFT
TITLE_ALIGN=LEFT
WIDTH=0.015
VISIBLE=YES
;
```

In this example below:

> If part type= Profile > the reference of the part will be the Designation + Length (in mm 2 digits).

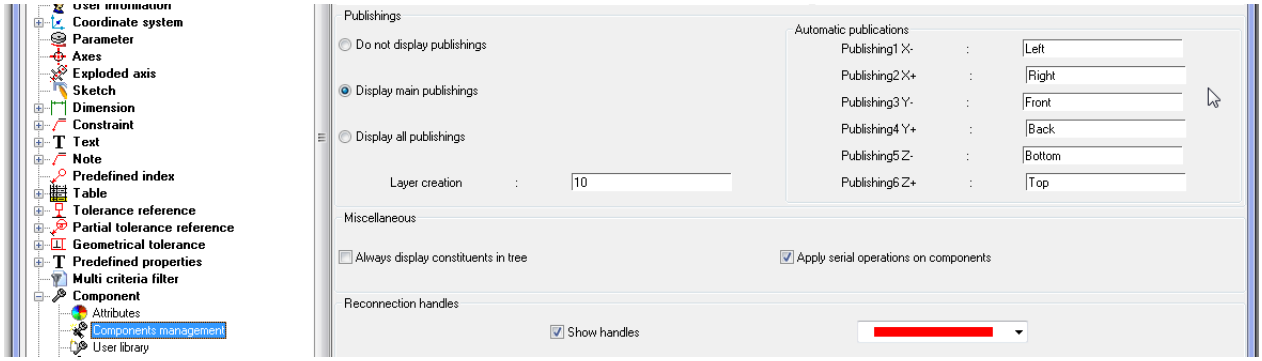
> If part type= Panel > the reference will be : Matter + Thickness (mm 2 digits).

> If part type = Hardware the reference will be : Designation + catalog code.

Shelve	Oak-copper_19.00	Oak-copper	0.41	0.25	0.02	-	Panel
Screw	Screw 3,5x9,5	steel	-	-	-	3,5x9,5	Hardware
Profil Alu small	Profil Alu small –696.00	steel	0.7	0.02	0.02	-	Profile
Mounting plate Expando Dis 0	177H5400E	steel	-	-	-	-	
DESIGNATION	REFERENCE	MATIÈRE	LENGTH	WIDTH	THICKNESS	CODE	TYPE

Design – Publishings

Tool | Option allows to set the default values for the visibility, the Layer and the names of Auto publishing.

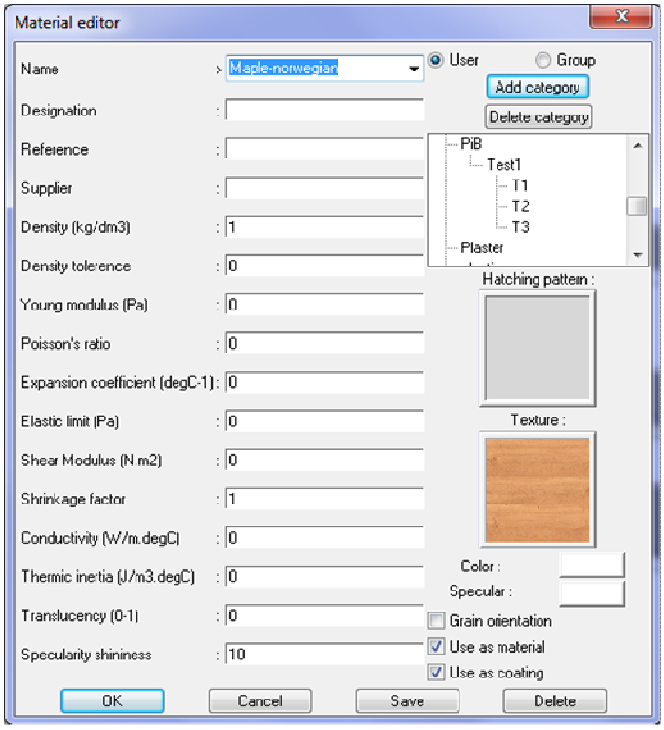


Do not forget that its possible to use automatic publishing on a main set but also in a alternative set to improve performances.

Design – Database material

It's now possible to manage :

- Multi-level material management.
- Grain information in definition.
- User or Group configurations.

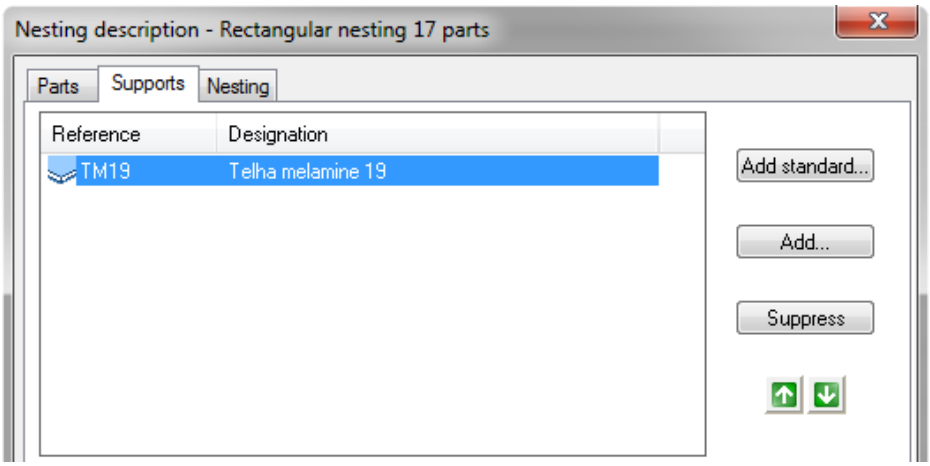


A database saved in 6.11 will not be compatible with previous versions of Topsolid.

Nesting – Supports selection

From this version, supports are automatically offered in the selection window.

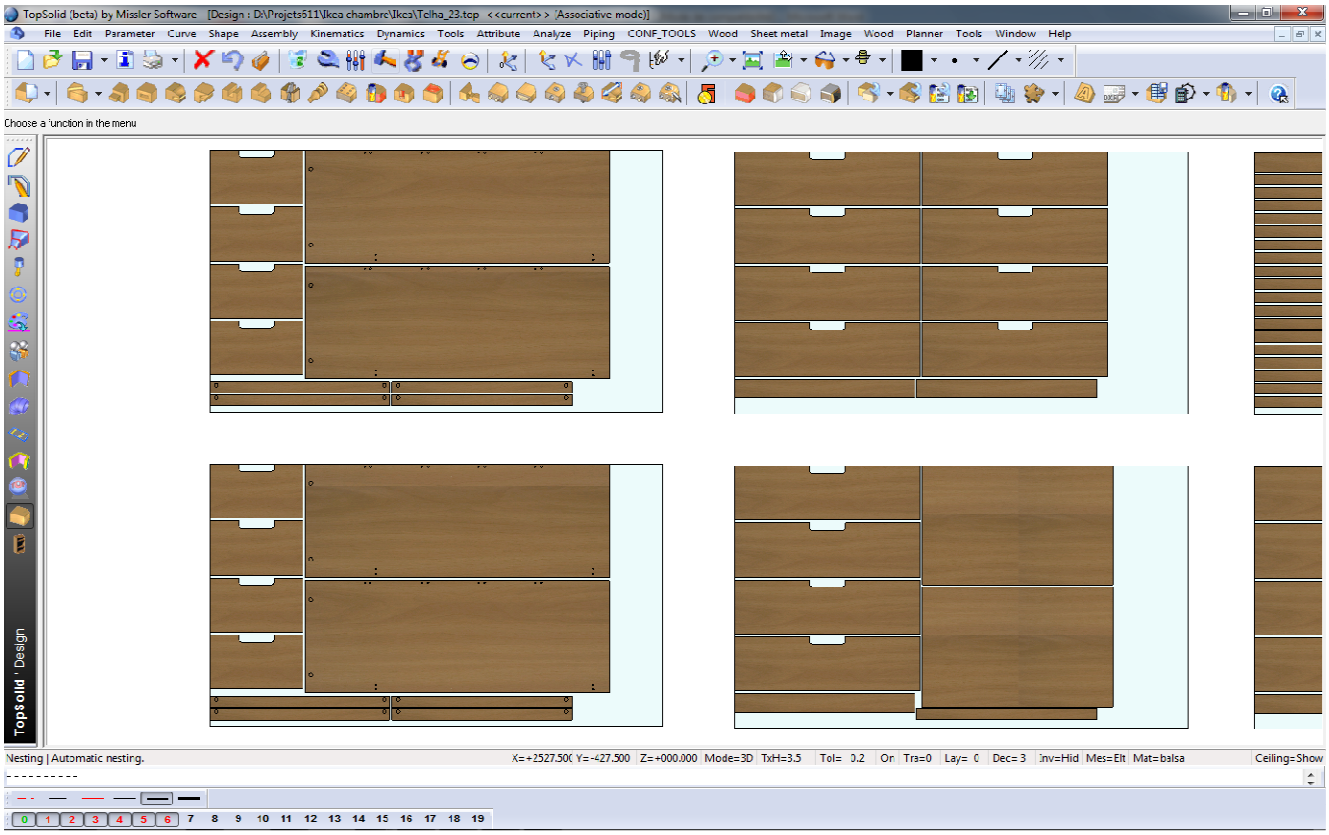
Supports are automatically chosen according to the parts' matter and thickness to nest. If several are proposed, the user will have to choose from the list.



The user does not need anymore to choose the support while creating the nests.

Nesting – Boards management

In the result file of the nesting, boards from the same group are moved in X and Y and all levels used are activated.



Nesting – Many improvements

Manual nesting in "rectangular" mode. During a manual nesting, rectangular mode is now available.

No need to save after each level modification in « one file result » mode.

Close the result file in "multi files" mode. The user has now the possibility to close the files created in "multi file" using the configuration word **D_NESTING_CLOSE_FILES** at 1.

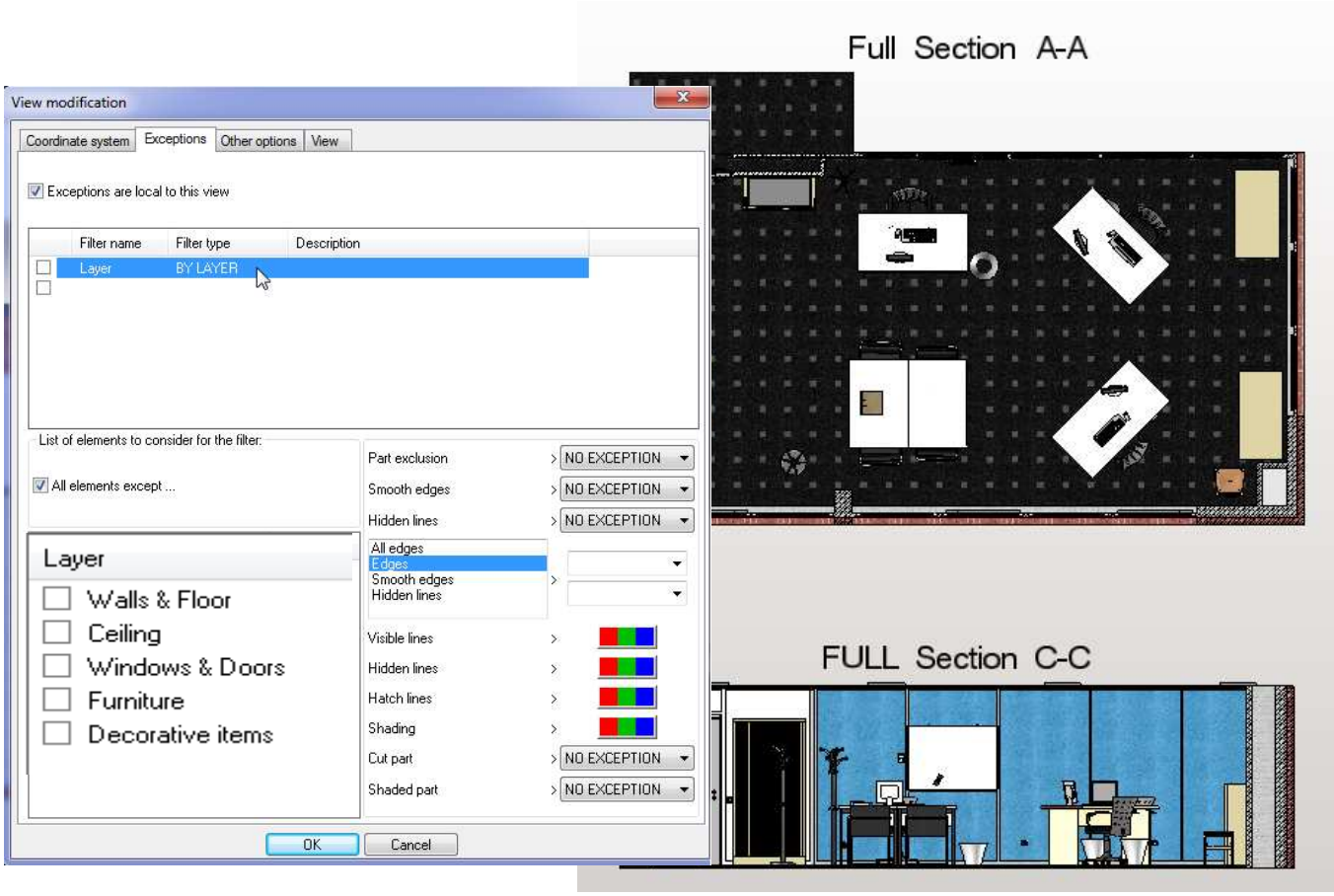
Addition of a new configuration word **D_NESTING_COMPLEX_ONE_PART_STRATEGY** allowing, if at 0, to switch off the "one part only" strategy which can present performance issues in some cases (big parts).

By default this configuration word is at 1, meaning that the "one part" strategy is automatically used when a single type of part must be placed (improvement for "U" or "L" shaped parts).

Draft improvements / Exceptions

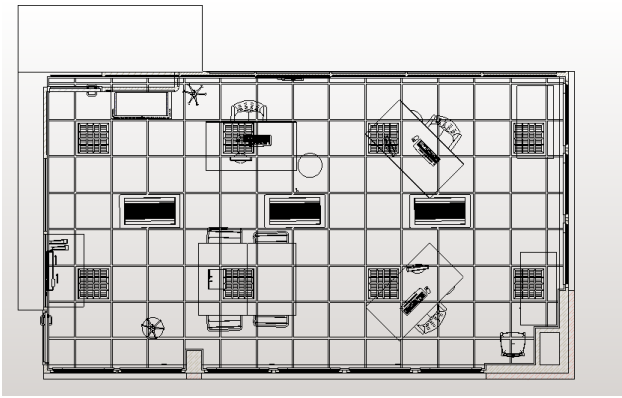
It's possible to use a new layer filter in Draft exceptions.
Layers used in a 3d document can now be useful for showing/hiding element in a 2d view.

Exceptions are now available in section and full section.



Draft improvements / Ceiling view

A new view is now available to interior design room, it allows to project the ceiling of a top view inside the room.



Design – Rendering

New kind of lights are available (SKY, AREA and WINDOW):

These three additional types were added to create images of better quality.

The sky will be used to symbolize the lighting of the sun, the area type to light up the entire face of an object, and the window type to symbolize the outside lightening through a window.

New rendering techniques were added to have a larger choice: