EFICAD

SWOOD

SWOOD 2023

What's New

SWOOD 2023

New Features

This document is used to present the new features of SWOOD 2023, to define their parameters, and finally to propose a small exercise for each of them to test for real.

How does this document work?

For each software (SWOOD Design, SWOOD CAM, SWOOD Nesting, and SWOOD Report) you can click on a bookmark, and it will redirect you to the related chapter within this document.

How to download an exercise package?

Inside a chapter, go to "Exercise" and click on the Link or on the icon  .

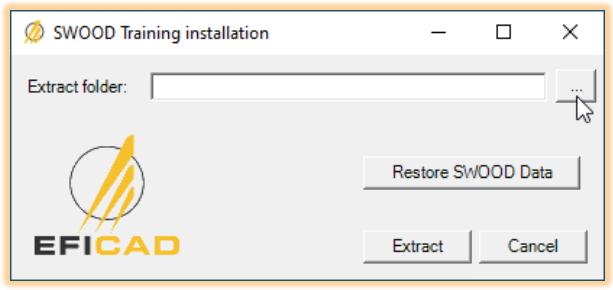
How do exercise packages work?

The exercise packages have different purposes:

- To automatically unzips the stored files (SWOODData, exercise files, etc.).
- To saves the actual SWOODData path.
- To changes the SWOODData to the exercise one.

The steps are:

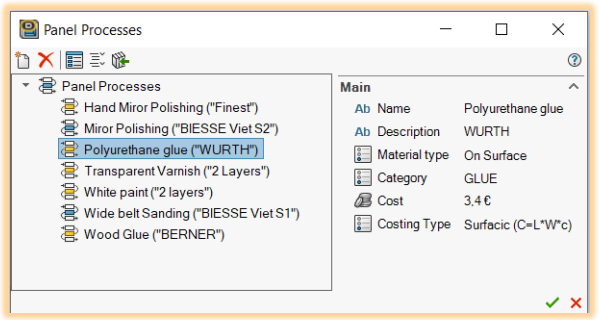
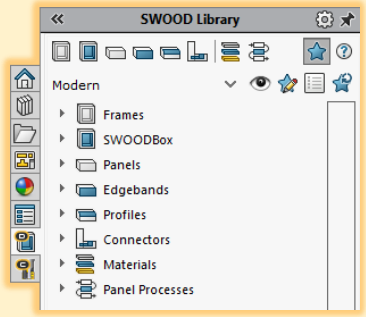
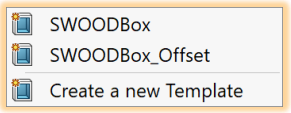
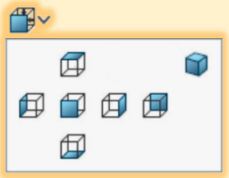
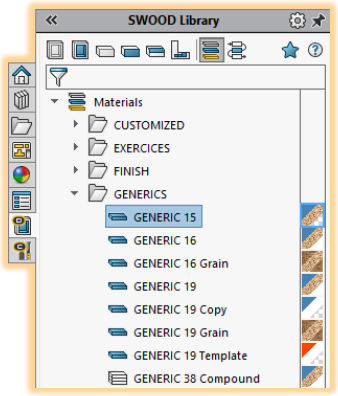
1. Open the  package already downloaded (in the Download folder).
2. Click on the  button to set the location where you need to unzip the package (it can be in the Document folder).

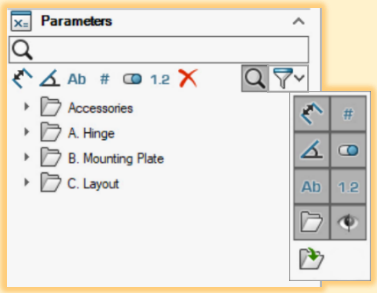
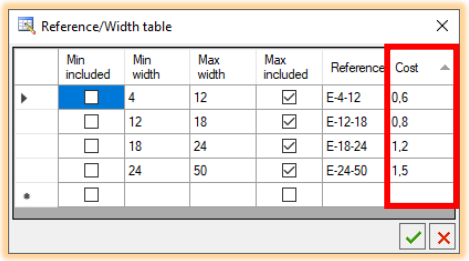
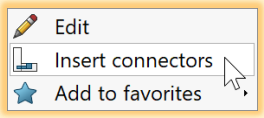
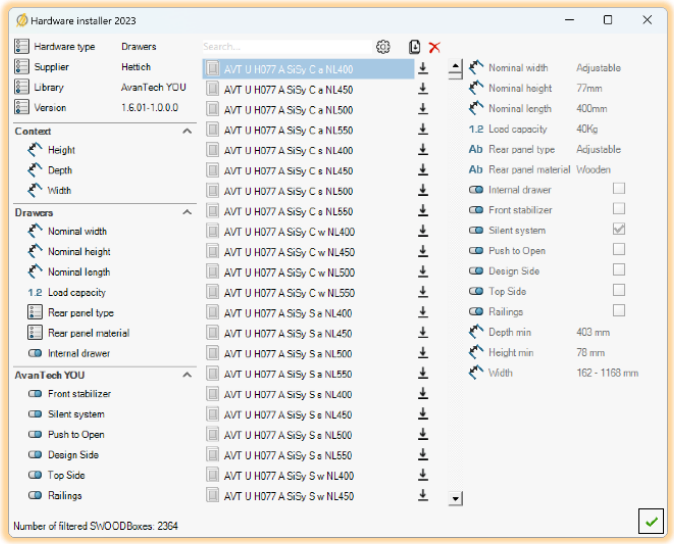


3. Click on Extract to make all the changes automatically.

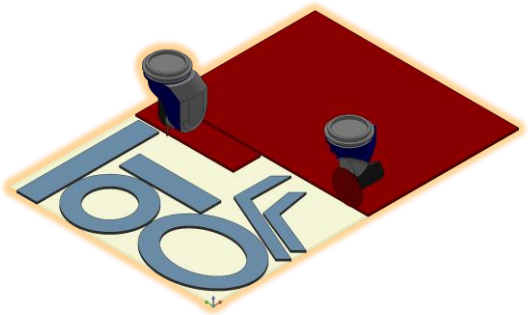
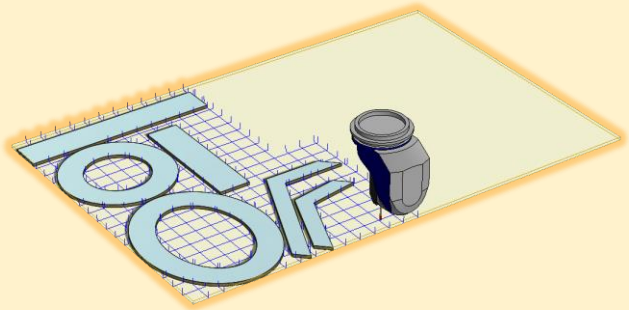
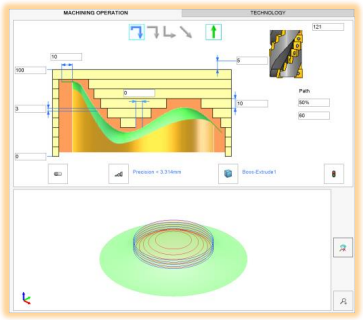
⚠ At the end of each exercise, do not forget to reopen the package (.exe file) and click on Restore SWOOD Data to restore the previous setup.

SWOOD Design

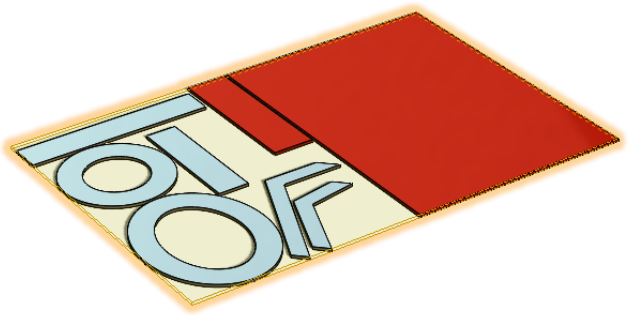
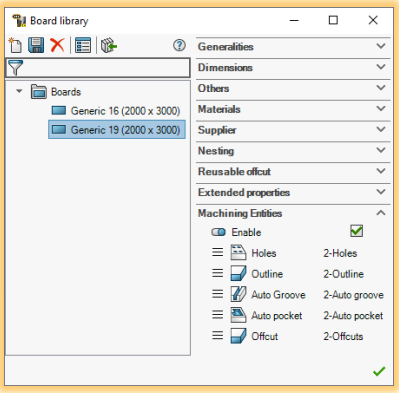
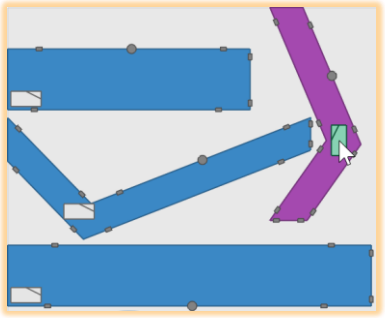
Feature	Description
Panel Processes Library	
Favorites	
Library Templates	
Connectors View Orientation	
Appearance Display in the Materials and Edgebands Library	

SWOODBox Parameters Display	
Cost in Edgeband Reference/Width table	
Insert Connector from Library	
Hardware Installer 2023	
SWOOD Connect	

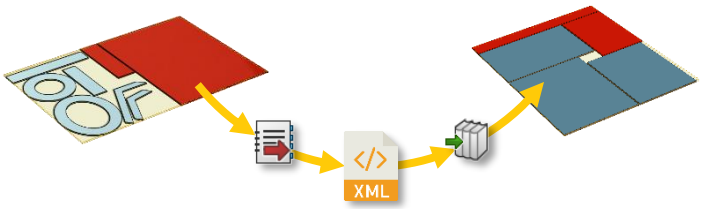
SWOOD CAM

Feature	Description
Offcutting	
Waste Cutting	
Upgraded Roughing Volume Operation	
3D Milling Surface and Graphic Bodies	

SWOOD Nesting

Feature	Description
Reusable Offcuts	
Machining Insertion from Boards Library	
Rotate Labels Independently	

SWOOD Report

Feature	Description
Offcut Export	

SWOOD Design – Panel Processes Library

This new feature was created for companies that apply special processes to their panels or laminates, such as gluing, varnishing, or polishing operations.

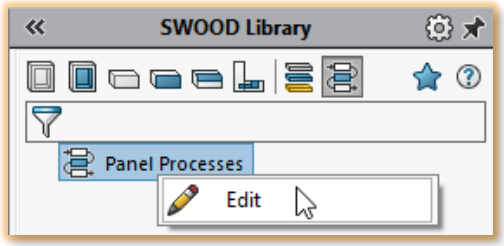
Once parameterized and applied on panels and laminates, this functionality , provides the costs of these processes at the project level.

You can see a new tab on your SWOOD Design Library.

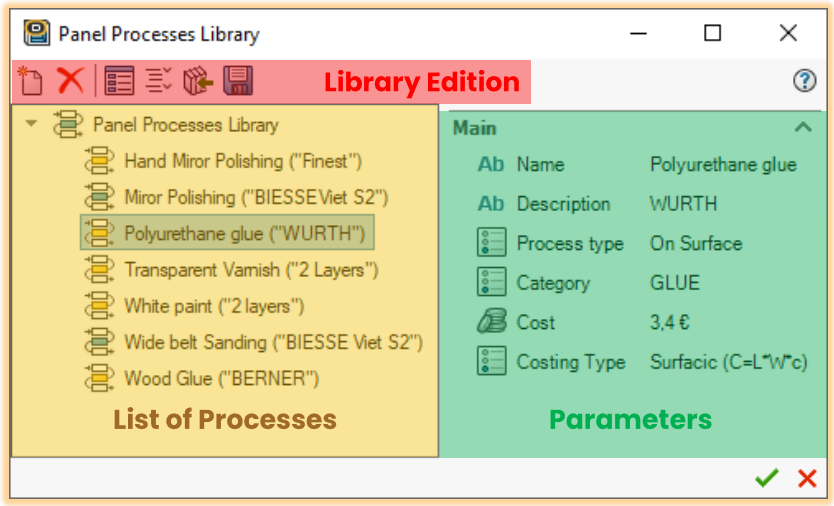
I. How to Set Up Processes

1. *Panel Processes Edition*

To access the processes settings, right-click on "Panel Processes" and click on "Edit".



Three areas in the Panel Processes edition window will show: The library edition toolbar, the list of processes, and the parameters.



a. *Library Edition Toolbar*

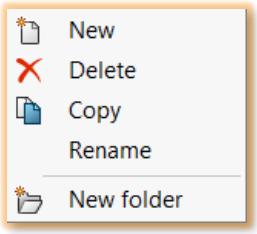
The toolbar contains the following buttons:

Parameter	*Description*	*Further explanations*
New (Process)	To create a new process	/
Delete (Process)	To delete an existing process	/
Extended Properties	To add extended properties to the process parameters	/
Category	To open the Category window	§ I.2
Import	To import a Panel Processes library from an external file	/
Save	To save the current Panel Processes library to an external file	/

b. *List of Processes*

In this area, all the existing Panel Processes are displayed. You can rearrange the Processes order by drag & drop.

By right-clicking onto a process, you can do the following actions:



c. *Parameters*

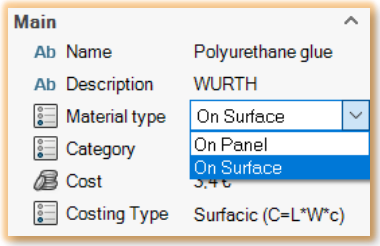
In this area, you can set the following parameters:

Parameter	*Description*	*Options*	*Further explanations*
Ab Name	Name of the process	*Text*	/
Ab Description	Short description of the process	*Text*	/

Material type	Defines what the process can be applied on	On surface / On panel	§ I.1.c.i
Category	The category of process	Created categories	§ I.2
Cost	The unit cost of the process	*Price*	/
Costing Type	How the process cost is calculated	Surfacic / Unitary	/

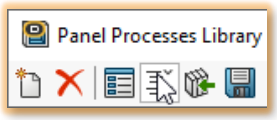
i. Material Type

There are two types of processes: one that is applied to a surface (such as a varnish), and one that is applied to the entire panel (such as running the panel through the wide belt sander).

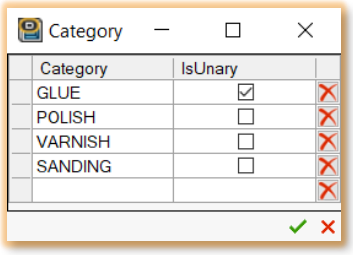


1. *Category Edition*

When you click on the "Category" button,

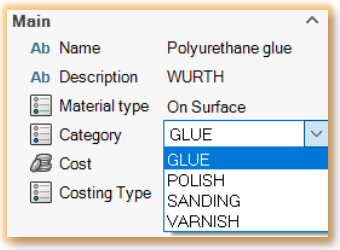


A window opens which allows you to create categories in order to group several processes together.



By clicking the "IsUnary" checkbox or not, you can define each category as unary or not. Unary simply means that the process can only be applied once on a panel, like gluing for example.

Once all the categories have been filled in, they can be found in the "Category" drop-down list of process parameters.



II. Exercise

To try this new feature, follow the instructions:

1. *Download the package*

Link

2. *Run the "panel_process.exe" program*

Running this program will make SWOOD point to another SWOODData, the one you'll need to practice.

3. *Open "simple.SLDASM" SolidWorks assembly (in "Exercise" folder)*

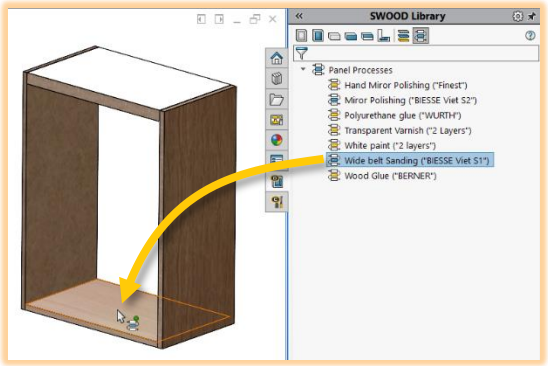
4. *Apply several Panel Processes to the frame panels*

There are different consequences to a drag & drop process on a panel.

a. *Process type*

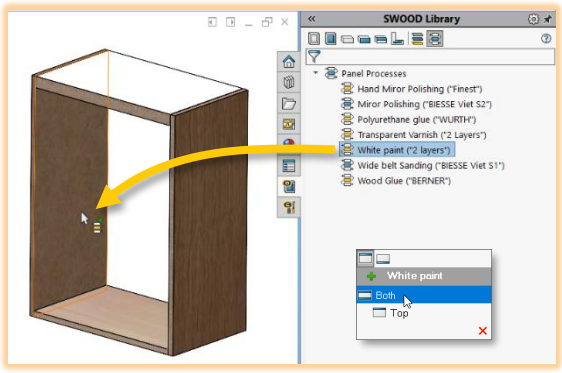
i. "On panel"

If the "material type" is set to "On panel", you'll have to do a simple drag & drop.



ii. "On surface"

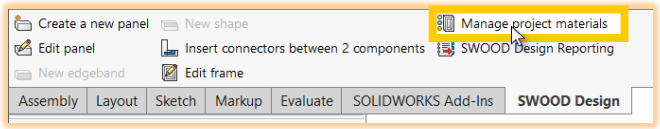
If the "material type" is set to "On surface", after your drag & drop you'll have to choose if you want to set it on the selected face or on both.

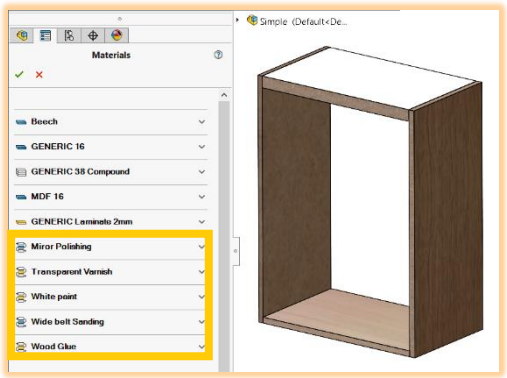


b. *"IsUnary"*

If the category of your Process is set to Unary, every time you apply a "Unary" process on a "already processed panel" it will replace the process.

Or else, the processes will be added to the previous ones.

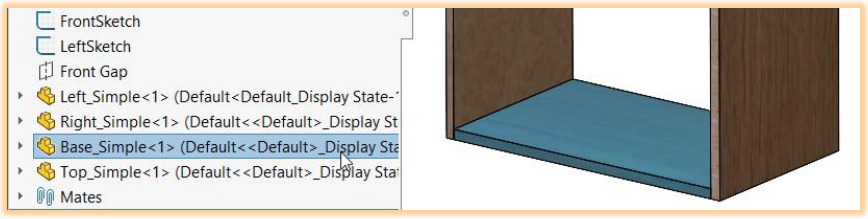


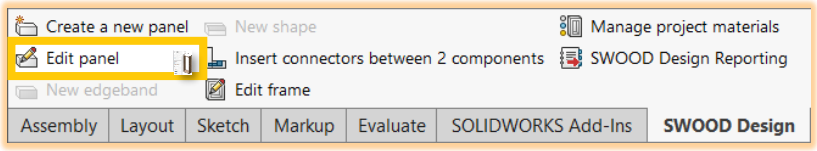


5. *Click on the "Manage project materials" button (on SWOOD Design SolidWorks tab)*

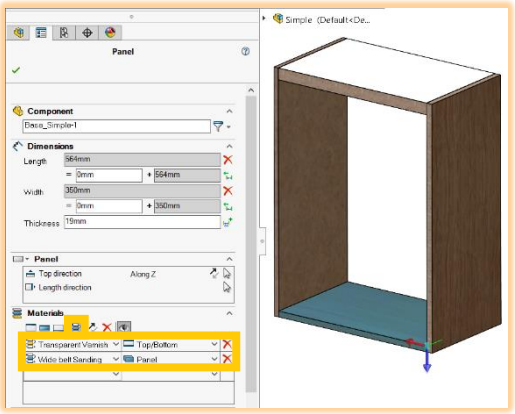
This will show the applied processes on the panels faces.

6. *Select any part on the SolidWorks FeatureManager Design Tree*





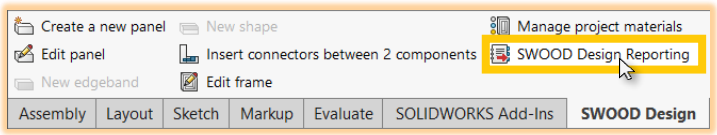
Then click on the "Edit panel" button (on SWOOD Design SolidWorks tab).



You will be able to manage the applied processes on the Processes tab of the Materials area.

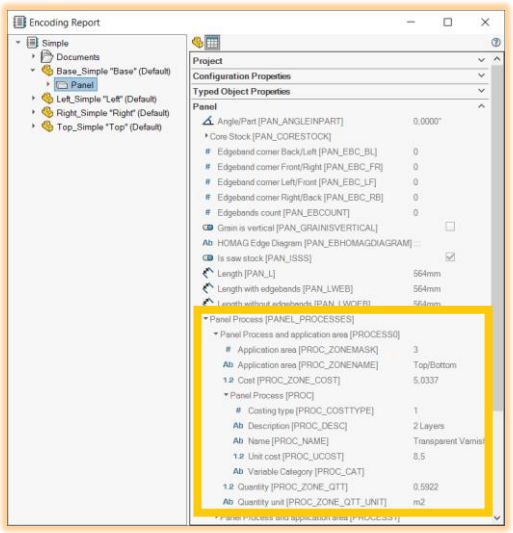
7. *Generate a Report*

By clicking on the "SWOOD Design Reporting" (SWOOD Design tab in the CommandManager).



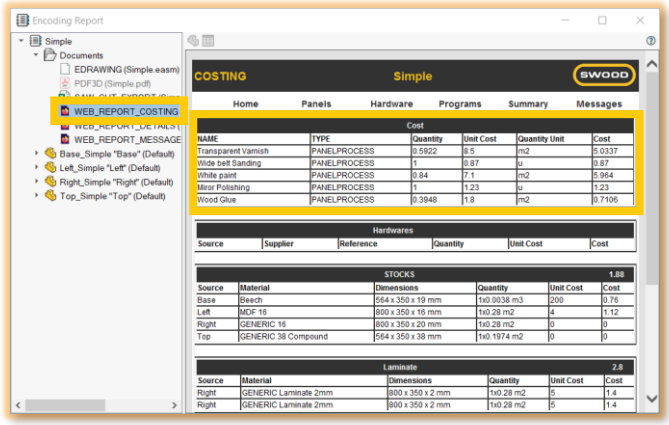
a. Reach the processes data

On the report "Simple" → Click on a Part → Panel → on the right side, click on "data".



b. View the Costing summary

On the report "Simple" → Click on the " WEB_REPORT_COSTING" → Click on "Summary".

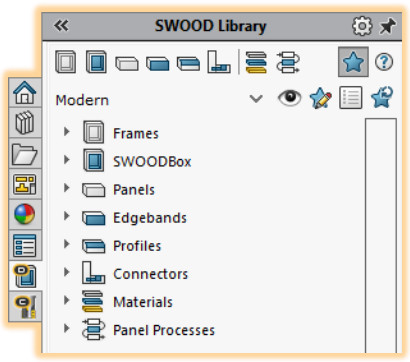


8. *Restore the SWOODData*

Reopen the package (panel_process.exe file) and click on Restore SWOOD Data to restore the previous setup.

SWOOD Design – Favorites

The **Favorites** give the user the ability to combine multiple items from different SWOOD Design libraries into a single favorite category for quicker access when designing the same style projects.



You can see a new tab on your SWOOD Design Library.

I. Interface

The Favorites library is divided into 3 sections:

- The Favorites Toolbar

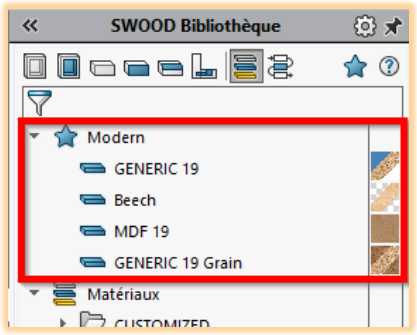
- The Favorites List
- The Appearances

1. *Sections*

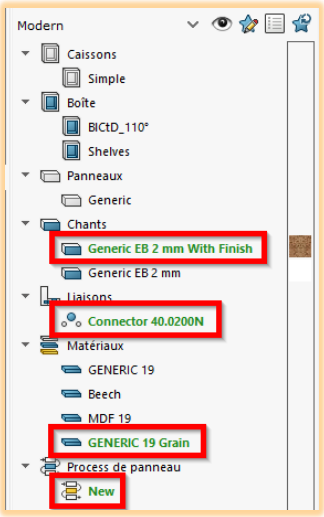
a. *Toolbar*

The Favorites toolbar is composed of 5 elements, which are from left to right:

- **The drop-down list of favorites categories**: Allows to select one of the existing categories of favorites. Once selected, this category of favorites will be displayed in the Favorites List.
- **Pin Favorites**: Allows to display in each SWOOD Design library the favorites of the selected category for faster access.



- **Edit Categories**: Allows to open the Favorites Categories window in order to edit them.
- **Change favorite display type**: Allows to display the favorites in list mode (sorted alphabetically) or in Folder mode (sorted by item type).
- **Refresh opened favorites** *(Ctrl+F5 shortcut)*: Displays in green which favorite library items are present in the active document (this does not work for Frames, SWOODBoxes, and Panels).



b. *Favorites List*

The Favorites list displays all library items that have been added to the selected favorites category.

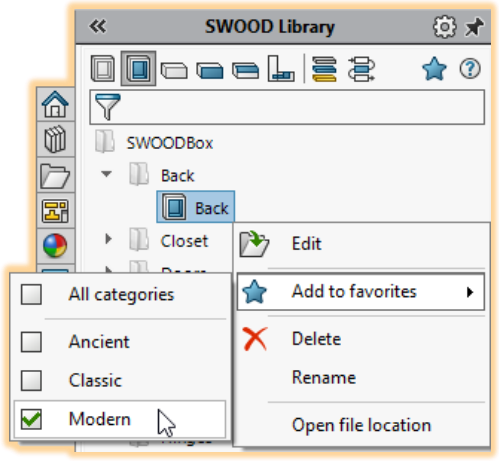
This list can be displayed in 2 ways using the "Change favorite display type" button: either the favorite items are sorted alphabetically, or they are sorted by type of library item.

When right-clicking on a favorite item, several actions are possible depending on the type of item selected:

- For all types of library items:
  - **Open favorites location**: Allows to go directly to the library with the selected item.
  - **Delete the favorites of this category**: Removes the selected item from the favorites of this category.
  - **Edit**: Allows to edit the selected item directly.
- For the Frames, SWOODBoxes, and Panels:
  - **Rename**: Renames the selected item. The item will also be renamed in the source library.
  - **Open file location**: Opens the windows explorer at the location where the selected item is saved.
- For the Frames, only when a SolidWorks assembly is open:
  - **Insert copy**: Allows to insert a copy of the selected frame in a layout.

- For the SWOODBoxes that have a .pdf with the same name as the SWOODBox in the same folder only:
  - **Show pdf**: Opens the explicative .pdf associated with the SWOODBox.
- For the Panels, only when a SolidWorks assembly is open:
  - **Insert copy**: Allows to insert a copy of the selected panel in a frame or a layout.

To add an item from any SWOOD Design library, simply right-click on the item, click on "**Add to favorites**" and then select the category or categories where the item needs to be inserted.



c. Appearances

As in the Edgebands and Materials libraries, the right-hand side allows visualizing the appearance of the elements.

- In the case of a Panel type material, the appearance will be represented divided into three:
  - ❶ The first layer represents the appearance of the top face.
  - ❷ The second layer represents the main appearance of the material.
  - ❸ The third layer represents the appearance of the bottom face.

- In the case of a Laminate type material, only the main appearance of the material will be displayed.

- In the case of a Compound type material, the appearance will be represented divided into three:

  ❶ The first layer represents the appearance of the top face of the top layer of the compound.

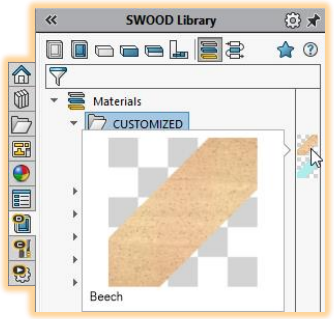
  ❷ The second layer represents the main appearance of the layers.

  ❸ The third layer represents the appearance of the bottom face of the bottom layer of the compound.

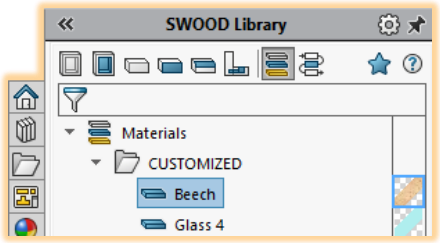
- In the case of an Edgeband, only the main appearance of the edgeband will be displayed.

If an appearance is missing, it will be represented by a light grey grid (like the third layer in the image above).

To see the appearance more precisely, simply hover over the element's appearance.

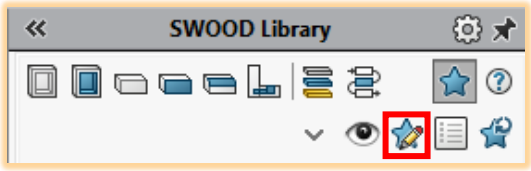


The appearance of the selected element is surrounded by a blue outline to associate the name of the element with its appearance.



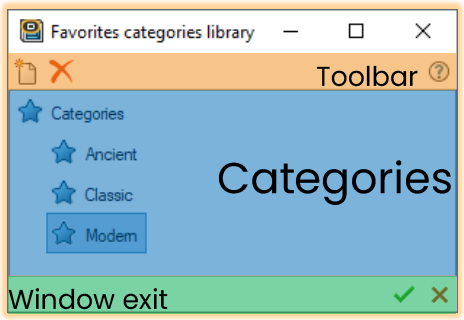
2. *Categories*

The Favorites categories are groups in which the user can aggregate SWOOD Design library items.



a. Interface

The editing window of the Favorites categories is divided into 3 sections:



i. Toolbar

The toolbar is composed of two tools:

- **New**: To create a new category.
- **Delete**: To delete the selected category.

ii. Categories

The created categories are listed under "Categories".

By right clicking on one of the categories, it is possible to do the following actions:

- **New**: To create a new category.
- **Copy**: To copy the selected category. It will duplicate the selected category adding a number after it.
- **Delete**: To delete the selected category.
- **Rename**: To modify the name of the selected category.

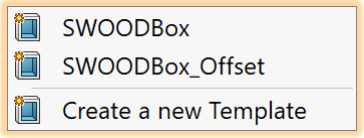
iii. Window Exit

At the very bottom of the Favorites Category Editing window, there are two options to exit the window:

- ✓ **Validate**: Exit the window, saving the modifications.
- ✗ **Cancel**: Exit the window, cancelling the modifications.

SWOOD Design – Library Templates

This new feature has been created for using different file templates when creating a new frame, SWOODBox, or panel.



I. How to Enable

1. *Location*

To add new templates to the lists, simply save the template files ("ASMDOT" file for assemblies and "prtdot" file for parts) in the following location:

SWOODData\SWOODDesign\Templates

II. How to Use

To use the templates:

- Go to the SWOOD Library pane.
- Right-click on a folder.
- Click on New.

III. Exercise

To try this new feature, follow the instructions:

1. *Download the package*

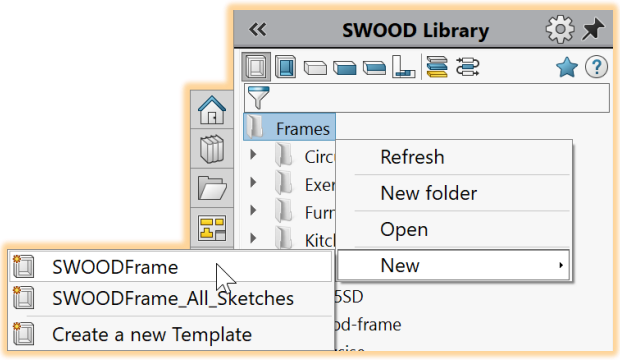
Link

2. *Run the "library_templates.exe" program*

Running this program will make SWOOD point to another SWOODData, the one you'll need to practice.

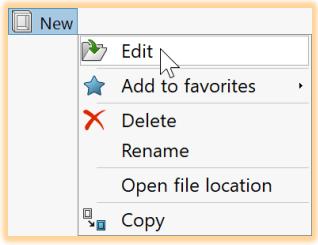
3. *Create a new frame based on the regular template*

Go to the Frames library, right click on the Frames folder, click on “New” and select the “New SWOODFrame” template.



4. *Open the new frame*

Do not rename the assembly, right-click on it and click on Edit.



5. *Create offset planes*

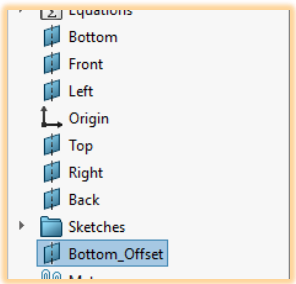
Select the Bottom plane.

Go to the Assembly tab in the CommandManager.

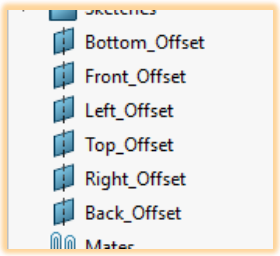
Click on the Reference Geometry button and select the Plane command.

Put an Offset distance of 20.00mm and validate.

Rename the new plane “Bottom_Offset”.



Repeat the procedure for the Front, Left, Top, Right, and Back planes.

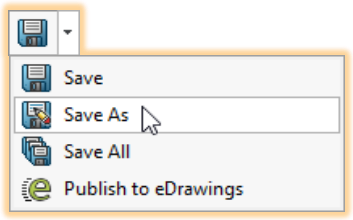


Select the six new planes, right-click and select "Add to New Folder".

Rename the folder "Offset Planes".

6. *Save the new template*

On the SolidWorks Menu Bar, click on "Save As".

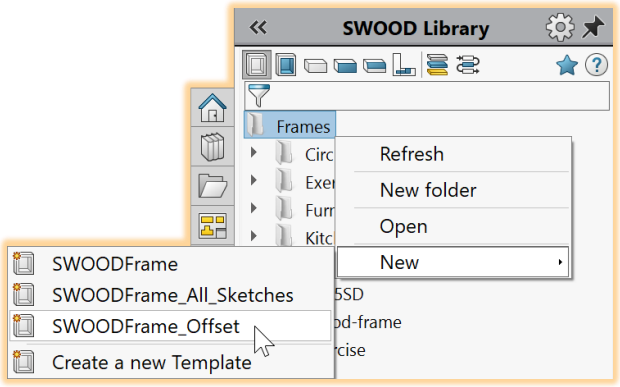


For the file type, select the "Assembly Templates (*.asmdot)".

Go to the following location:

SWOODData\SWOODDesign\Templates\SWOOD

Name the template "SWOODFrame_Offset.ASMDOT" and save.



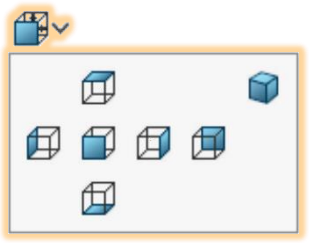
You can delete the frame "New" in the Frames library.

7. *Restore the SWOODData*

Reopen the package (library_templates.exe file) and click on Restore SWOOD Data to restore the previous setup.

SWOOD Design – Connectors View Orientation

This new feature has been created to better visualize the connector orientation. As in SolidWorks, it has all the basic views (*Front,* *Back,* *Left,* *Right,* *Top,* *Bottom*) and also the *Isometric View*.



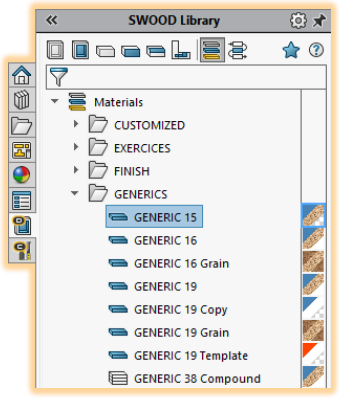
I. How to Use

To use the Connectors View Orientation:

- Go to the SWOOD Library pane.
- Go to the Connectors Library.
- Right-click on "Connectors" and click on "Edit".
- On the top of the Graphics Area (of any connector), click on the new command .

SWOOD Design – Appearance Display in the Materials and Edgebands Library

This new feature has been created to visualize the complete appearance of all materials or edgebands at a glance and make it easy to choose from the library.



I. How to Use

It is possible to see the new appearance display in the Edgebands Library and in the Materials Library.

Go on one of these libraries and open a folder to reveal some items.

The appearances will be shown on the right side of the library.

1. *Appearance Display*

The appearances are displayed according to the appearance of the item.

If the appearance of the item is not defined, it will be represented by a light grey grid, as if it were transparent.

a. *For the Edgebands*

For the edgebands, the appearance will be defined by the only appearance parameter of the library.

b. *For the Materials*

The material may contain several appearances, so they are represented in three oblique layers.

i. Panel Type

- The ❶ first layer represents the appearance of the top face (Top).
- The ❷ second layer represents the main appearance of the material.
- The ❸ third layer represents the appearance of the underside (Bottom).

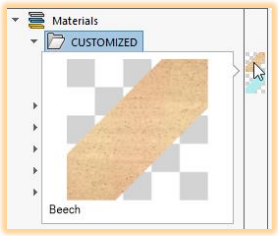
ii. Compound Type

- The ❶ first layer represents the appearance of the top face of the top layer of the compound.
- The ❷ second layer represents the main appearance of the layers.
- The ❸ third layer represents the appearance of the underside of the bottom layer of the compound.

2. *Interaction*

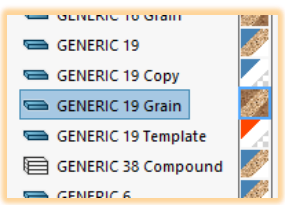
a. *Mouse Hover*

When the mouse hover over an appearance, a tooltip appears and shows the appearance in a bigger size.



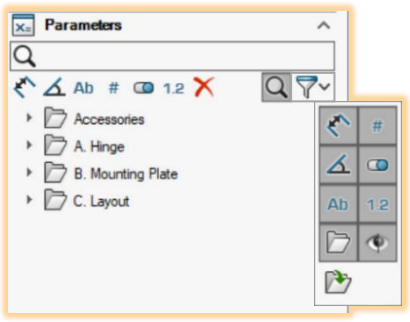
b. *Selection*

When an item is selected, a blue outline surrounds the appearance to associate the item and its appearance.



SWOOD Design – SWOODBox Parameters Display

This set of new features has been created to better organize the many SWOODBox parameters and facilitate their use.



I. How to Use

To be able to see the new SWOODBox parameters display, simply create or edit an existing SWOODBox.

1. *Search Bar*

To reach the search bar, simply click on the magnifying glass icon on the parameters toolbar. It will open the search bar above.

- **The search is <u>not</u> case-sensitive**. For example, searching "Oo" might result in words with "oo" or "OO".
- **The search is inclusive**. For example, searching "oo" might result in "Groove".
- **The search is active in parameters' Names, Descriptions, Aliases, and Categories**.

2. *Filters*

It is possible to show some parameters based on some of their properties.

a. *Parameters' Type*

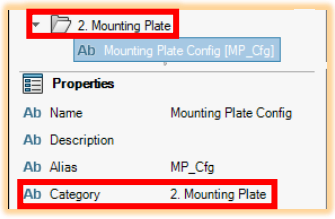
The parameters list can be filtered with the parameter's type: Distance, Angle, String, Integer, Boolean and Decimal.

b. *Exposed*

The parameters list may only display the exposed parameters when the "Exposed" filter is unchecked.

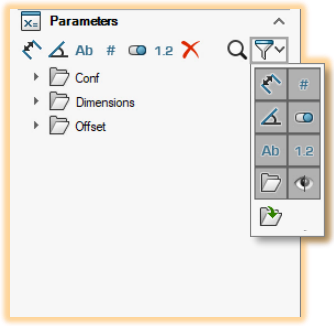
3. *Folder Organization*

From now on, parameters that have a category are organized in a folder, with the name of this category, in the list of parameters.

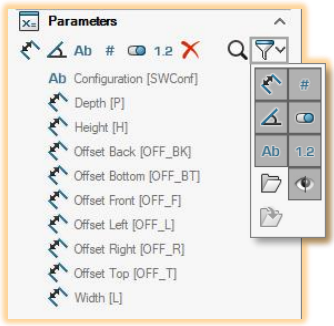


It is possible to assign a category to a parameter by dragging the parameter into the folder.

This folder organization can be disabled (old layout) by clicking on the "Show/Hide Repository" filter.

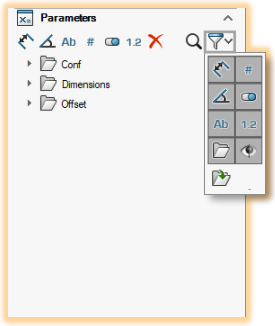


Folder Layout

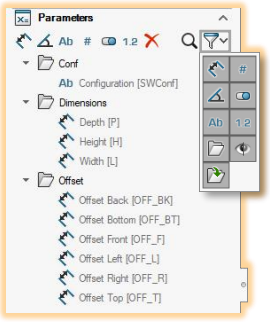


Regular Layout

If the Folder layout is selected, it is possible to open all the folders at once by clicking on the "Open/Close Directories" filter.



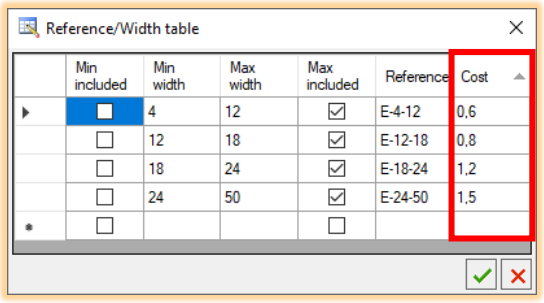
Close Directories



Open Directories

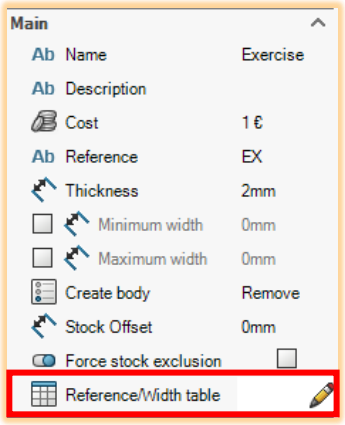
SWOOD Design – Cost in Edgeband Reference/Width Table

This new feature has been created to add a cost to each Reference inside a Reference/Width edgeband table.



I. How to Use

It is possible to reach the edgeband's "Reference/Width table" at the bottom of the Main parameters, by clicking on the "Edit" button.



The cost can be set at the sixth column of the table.

II. Exercise

1. *Download the package*

Link

2. *Run the "cost_edgeband.exe" program*

Running this program will make SWOOD point to another SWOODData, the one you'll need to practice.

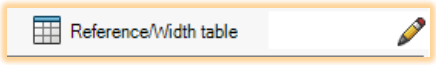
3. *Create an edgeband "Reference/Width table"*

a. *Reaching the Table*

Go to the SWOOD Library, then to the Edgebands library.

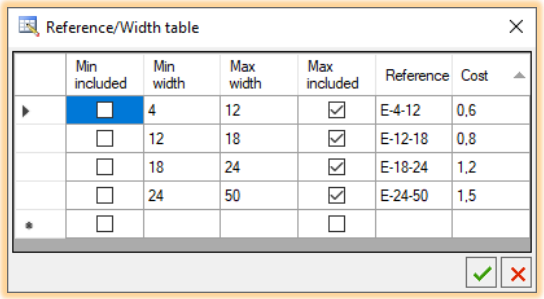
Right-click on the "Exercise" edgeband and select "Edit".

On the Main parameters, scroll down to "Reference/Width table" and on the right side of the box, click on the "" button.



b. *Filling the Table*

When the table window opens, fill the cells to get the following table:



	Min included	Min width	Max width	Max included	Reference	Cost
▸	☐	4	12	☑	E-4-12	0,6
	☐	12	18	☑	E-12-18	0,8
	☐	18	24	☑	E-18-24	1,2
	☐	24	50	☑	E-24-50	1,5
*	☐			☐		

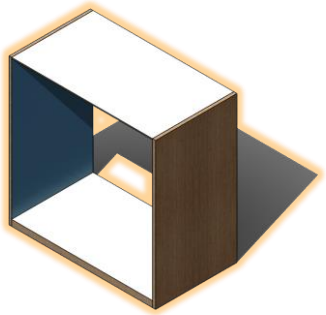
✓ Validate the table window.

✓ Validate the Edgebands library window in order to save the last modifications.

4. *Open the exercise frame*

Go to the SWOOD Library, then to the Frames Library.

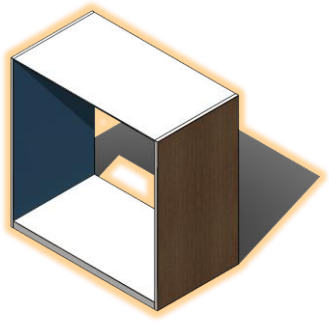
Right-click on the "Exercise" frame and select "Edit".



5. *Apply the edgeband to the panels*

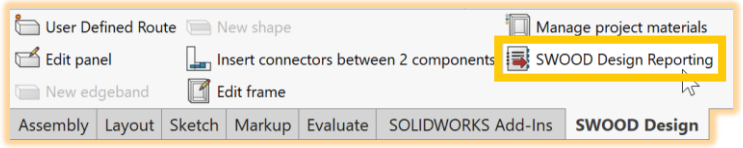
Go to the Edgebands library.

Drag & drop the "Exercise" edgeband to all the visible panel edges.



6. *Generate a report*

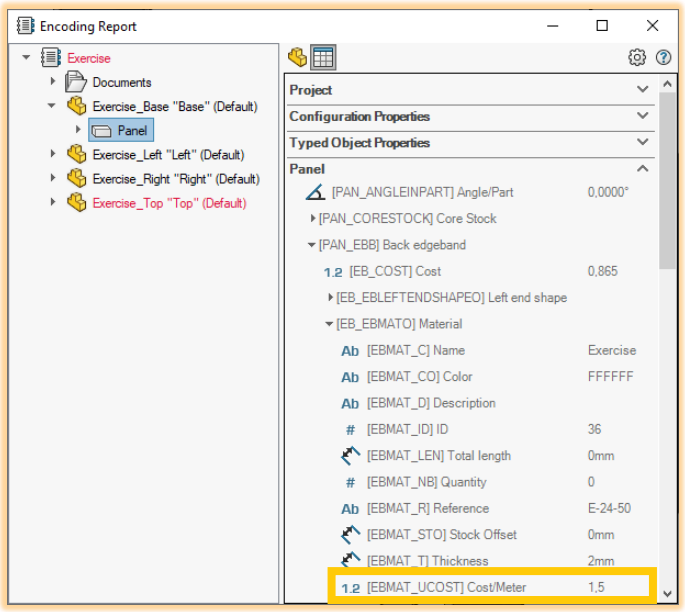
By clicking on the "SWOOD Design Reporting" (on SWOOD Design tab in the CommandManager).



a. *Reach the edgeband data*

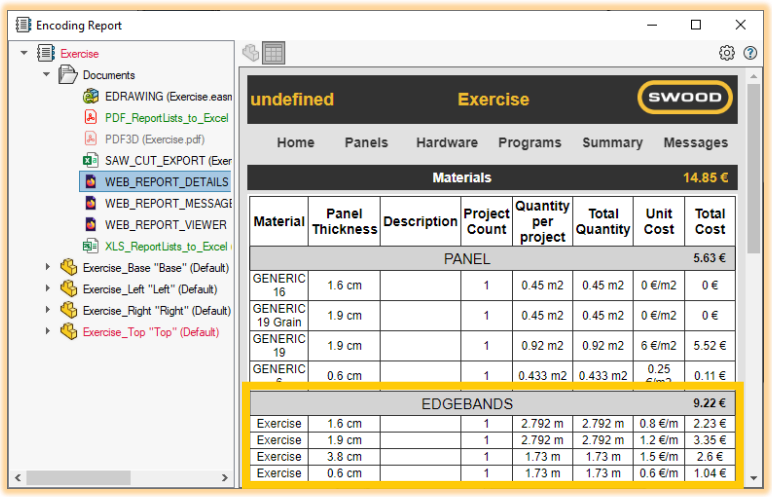
On the report "Exercise" → click on a Part → Panel → on the right side, click on "data".

On the data, expand the "[PAN_EBB] Back edgeband", then expand the "[EB_EBMATO] Material" and verify if the value of "[EBMAT_UCOST] Cost/Meter" is correct.



b. *View the Summary*

On the report "Exercise" → Click on the " WEB_REPORT_DETAILS" → Click on "Summary".

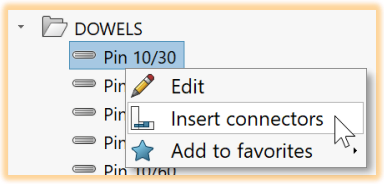


7. *Restore the SWOODData*

Reopen the package (cost_edgeband.exe file) and click on Restore SWOOD Data to restore the previous setup.

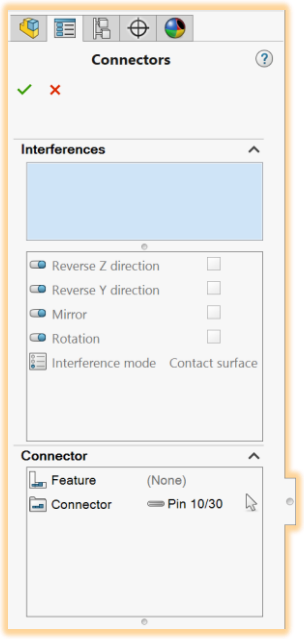
SWOOD Design – Insert Connector from Library

This new feature has been created to insert a selected connector directly from the library.



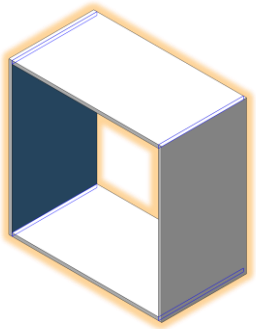
I. How to Use

It is possible to reach this feature by going in the SWOOD Library, then in the Connectors library, right-click on the wanted connector and click on "Insert connectors". Doing this will open the "Connectors" FeatureManager pane on the left side of the screen.



The "Connectors" pane will already include the selected connector.

On the Graphics Area, the interferences will appear in blue.



SWOOD Design – Hardware Installer 2023

This new feature has been created to download library items from a large data base.

I. How to Set Up Hardware Installer

The Hardware Installer is an external software that needs to be installed apart from SWOOD.

The latest version of the Hardware Installer can be downloaded from the following link:

Link

Unzip the compressed file.

Install the software by double clicking on Setup.exe.

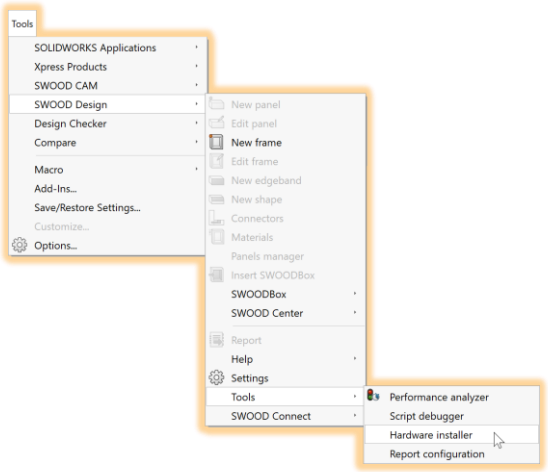
Restart SolidWorks.

II. How to Use the Hardware Installer

There are two ways to open the Hardware Installer:

- From the SWOOD interface: on the SolidWorks Menu Bar, click on:

  Tools → SWOOD Design → Tools → Hardware Installer.



- From the Windows explorer, go in:

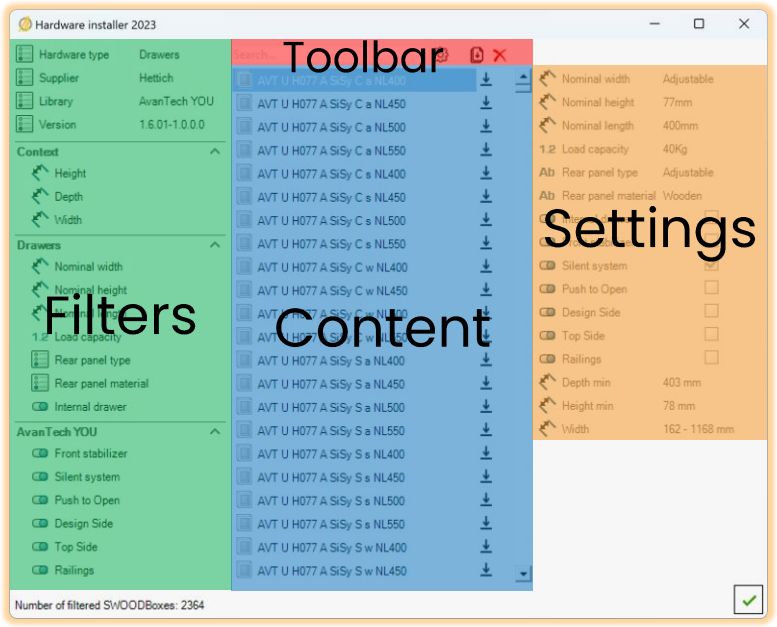
  C:\Program Files\EFICAD\HARDWARE_INSTALLER_2023\DLLS.

1. *Interface*

Once opened, the Hardware Installer window has 4 sections:

- The Toolbar
- The Filters
- The Content

- The Settings



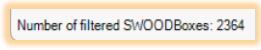
a. *Toolbar*

The toolbar is composed of 4 elements:

- A Search bar, to be able to find content writing its name.
- An Option button, to set the options of the Hardware Installer, like the downloading options, the language or the default values of the filters.
- A Download button, to be able to download all filtered content in one action.
- A Remove button, to be able to remove all the downloaded content.

b. *Filters*

As you can see at the bottom of the window, there is a lot of content available.



It is therefore important to be able to filter certain parameters in order to obtain only content that matches these parameters.

The types of filters are the following:

Drop-down lists: Allows to choose a value among several possible choices.

Dimensions: Allows to set a parameter dimension to a " = exact", " < minimum" or a " > maximum" value, or with a drop-down list.

1.2 **Decimal number**: Allows to set a numeric parameter value, which is not a dimension.

Boolean: Allows to set a parameter to a true or false value only.

c. *The content*

In this section, there is a list of all the items that match the filters.

There are several actions possible with the content list:

Download a content: clicking on the Download button.

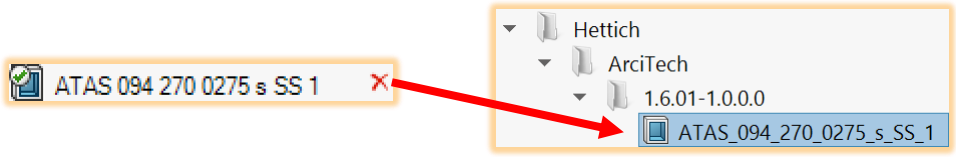
Remove a content: clicking on the Remove button.

The items have several states:

Not downloaded.

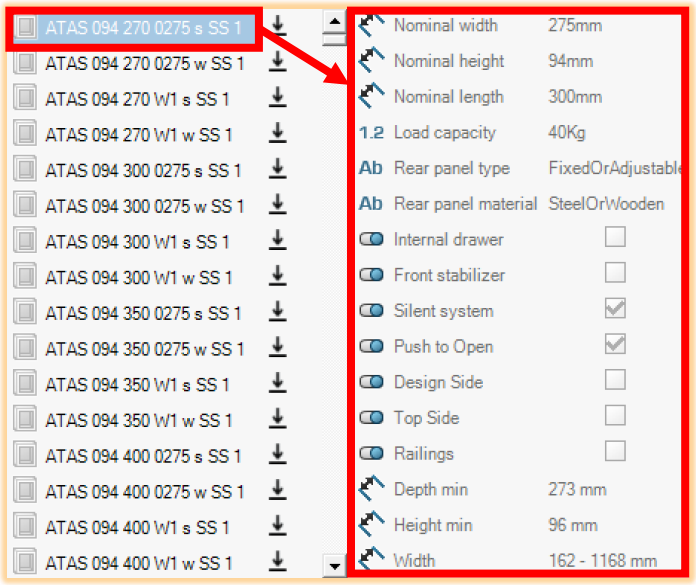
Downloading the item.

Downloaded item, the item is directly installed in your SWOODData.



d. *Settings*

Once an item is selected in the Content section, it is possible to see the values of its parameters. This can help choosing the needed item.



SWOOD Design – SWOOD Connect

SWOOD Connect has been created to connect SWOOD Design to external services in an optimal way.

I. Blum® Import

Through the Blum® e-services website, it is possible to:

- Configure "Product" or "Cabinet".
- Download the BXF file of the configuration

Then, with the SWOOD Design's "Blum® Import" tool, it is possible to import with a single operation:

- The cabinet assembly as a SolidWorks assembly.
- The panels as SWOOD panels with the SWOOD material.
- The Blum® hardware as SolidWorks parts recognized as hardware in the SWOOD report.

1. *How to Set Up Blum® Import*

a. *Registration at BLUM e-services*

To be able to create and export assemblies, it is necessary to register (for free) at the BLUM e-service website:

e-services blum

Fill in first name, last name, e-mail address and the name of the company, and finally click on “Register”.

b. *Download BXF File*

Either from the Product Configurator or the Cabinet Configurator, it is possible to generate a BXF file in the “RESULT” section.

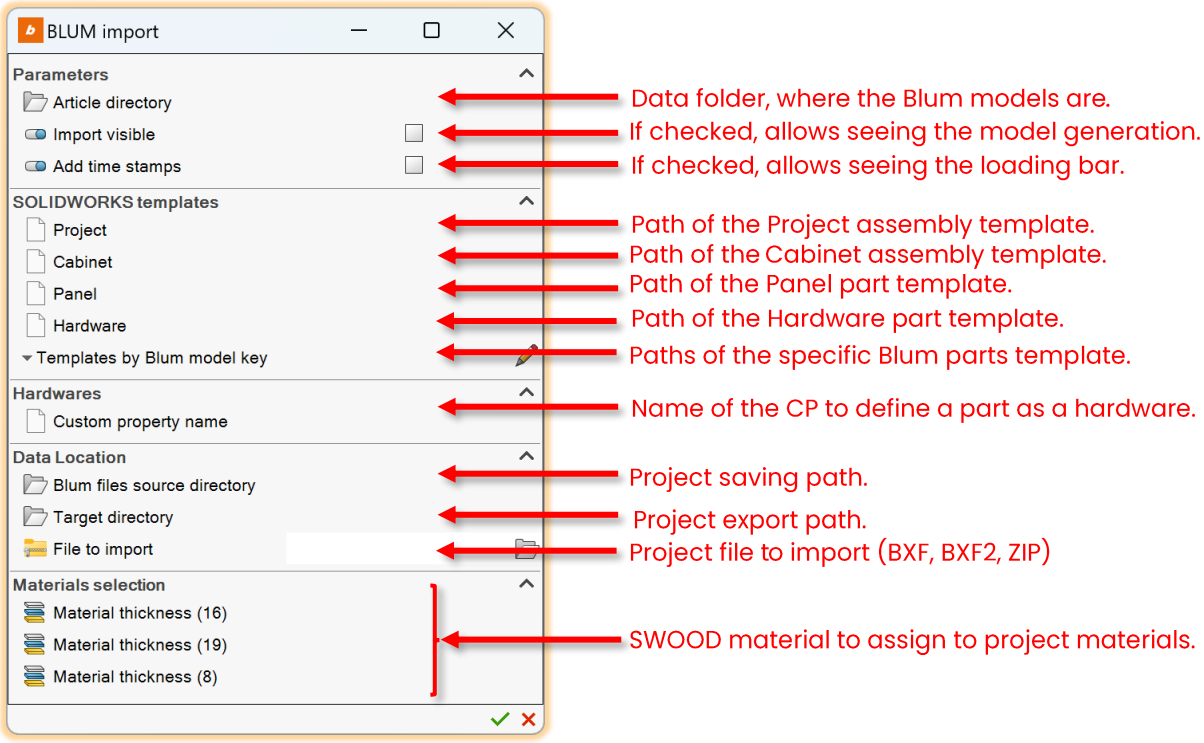
Before downloading the file, please make sure that the selected partner software is SWOOD.

c. *Import in SWOOD*

In the SolidWorks Menu Bar, click on:

Tools → SWOOD Design → SWOOD Connect → Blum → *Import*

A window opens.



Fill all the needed information and validate the window.

Blum Import makes calculations and produces a main assembly containing all the panels, and creates subassemblies with the hardware.

2. *Exercise*

a. *Download the package*

Link

b. *Run the "blum_import.exe" program*

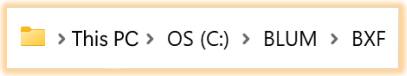
Running this program will make SWOOD point to another SWOODData, the one you'll need to practice.

c. *Create a new directory on your computer*

For the Blum® Import tool to work properly, it is necessary to create a new directory on your computer.

This directory can be located in C: (or on another place), and it can be called "BLUM".

Winthin this "BLUM" folder, create a new folder called "BXF".



d. *Fill up the Blum® Import fields*

Open the Blum® Import window from the SolidWorks Menu Bar :

Tools → SWOOD Design → SWOOD Connect → Blum → *Import*

i. Fill up the Parameters

The only mandatory parameter is the Article directory:

Select the "BLUM" directory you just created.

The other two parameters are optional.

ii. Fill up the SOLIDWORKS Templates

The 4 SolidWorks templates are part of the package, in the Material folder.

Select each template in the Blum® Import window from:

Package extract folder\blum_import\Material

iii. Fill up the Data Location

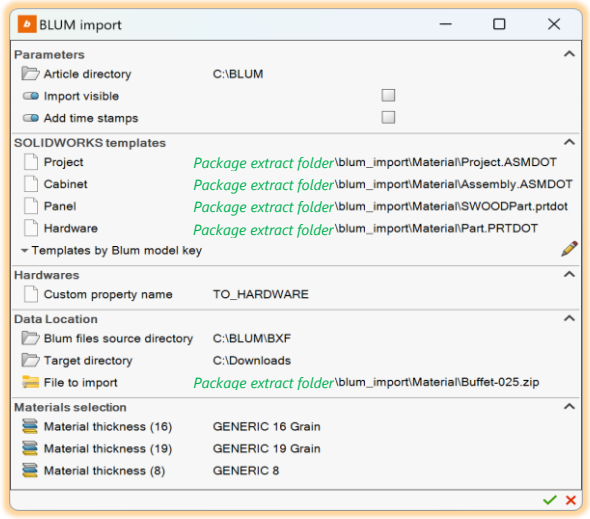
For the Blum files source directory, select the "BLUM/BXF" directory you just created.

For the Target directory, it is possible to select the Downloads folder.

For the File to import, you can select the "Buffet-025.zip" file.

Once the file to import is selected, the Blum import tool automatically recognizes the model's materials and asks what SWOOD Materials they are referring to:

- For the Material thickness (**16**), apply the **GENERIC 16 Grain**.
- For the Material thickness (**19**), apply the **GENERIC 19 Grain**.
- For the Material thickness (**8**), apply the **GENERIC 8**.

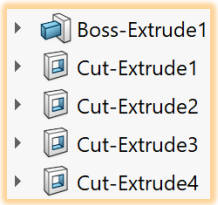


iv. Validate the window

Click on the ✔ Validate button.

The tool will compute all the cabinet's panels (this may take a while).

You can see that each imported panel has editable SolidWorks functions to form its shape, instead of impractical imported bodies.



v. Restore the SWOODData

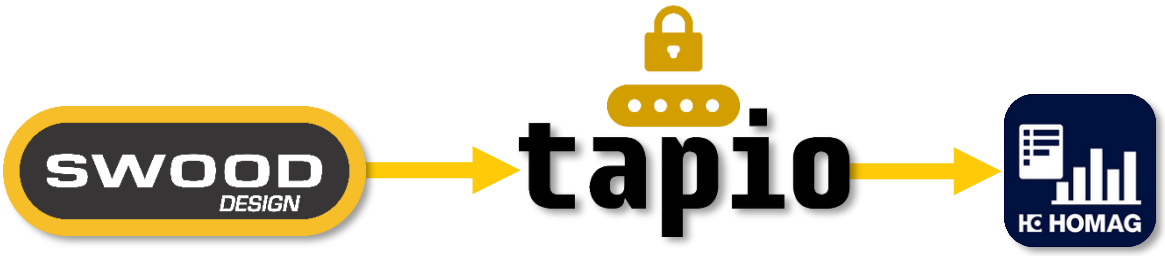
⚠ Reopen the package (blum_import file) and click on Restore SWOOD Data to restore the previous setup.

II. HOMAG productionManager

"HOMAG productionManager" is a web app that helps the company to maintain an overview and tracking of orders, as well as up to date information among all employees.
With this SWOOD Connect tool, it is possible to transfer any SWOOD Design project directly to the productionManager with all the project information.

1. *Tapio Connection*

To be able to transfer a project from SWOOD Design to HOMAG productionManager, this tool will pass through a service called Tapio.

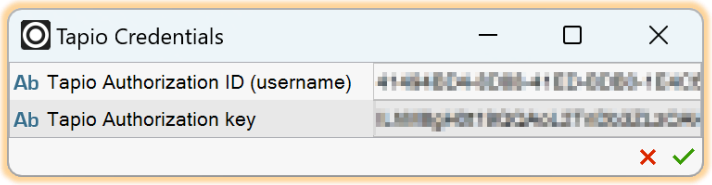


Tapio is a digital ecosystem of the wood industry and HOMAG productionManager is one of the available tool.
In our case, Tapio will be useful to login to the tool.
The first step is to create a Tapio account:

1. Register your company at my.tapio.one and invite your colleagues as users.
2. Go to the tapio.shop, select a package and get (for free) your productionManager license.
3. Assign the purchased productionManager licenses to your users.
4. Open productionManager at productionManager.homag.cloud and log in with your user account.

To enter your login informations, in the SolidWorks Menu Bar click on:

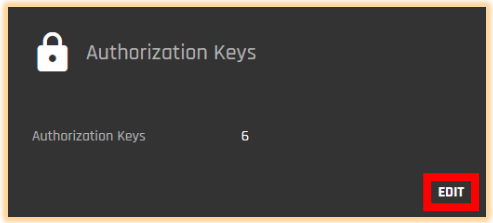
Tools → SWOOD Design → SWOOD Connect → HOMAG → 🔒 *Tapio Credentials*



To get the "**Tapio Authorization ID (username)**":

Once you receive an email from Tapio, confirming your "**Entreprise ID**",get the "**Tapio Authorization key**":

1. Login at my.tapio.one.
2. Go to " Applications".
3. Click on " HOMAG productionManager".
4. At the bottom of the page, click on the Add-on "SWOODConnect productionManager.
5. Click on "Edit" in " Authorization Keys".



6. In the pop-up that appears, click on the "**+ Add**" button.
7. Click on "**Confirm**" then on "**Copy**".

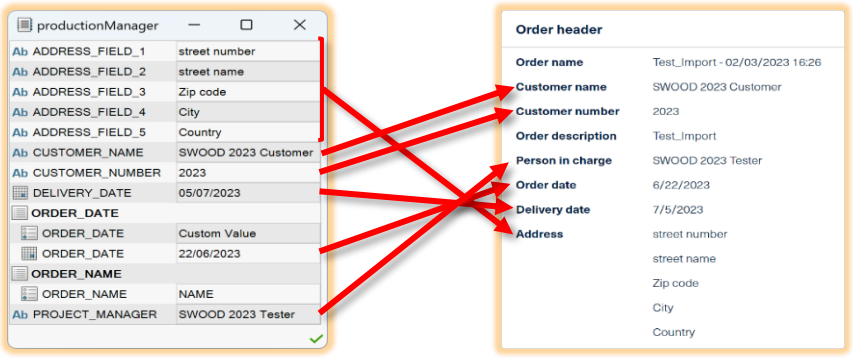
Warning: The Authorization key will only be displayed once (when we confirm the creation), copy and paste this key to save it.

2. *Transfer Configuration*

It is possible to add information to the SWOOD project that can also be transferred to the productionManager.

To reach the window where it is possible to add these information, in the SolidWorks Menu Bar, go to:

Tools → SWOOD Design → SWOOD Connect → HOMAG → Configure Transfer



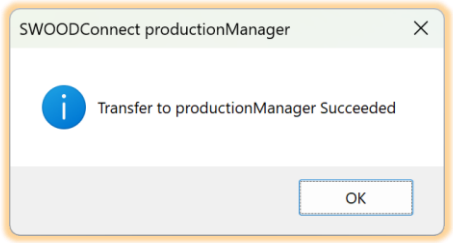
3. Project Transfer

When the "Tapio Credentials" are set, and when a SWOOD project is open in SolidWorks, it is possible to transfer it in the HOMAG productionManager web service.

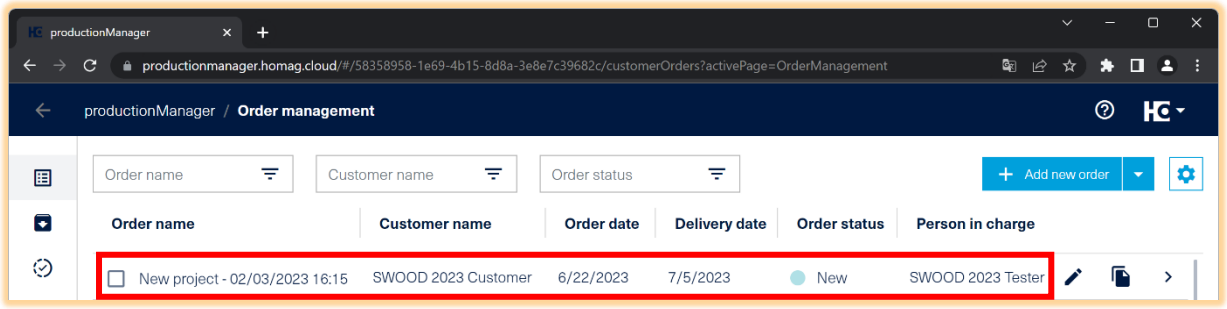
In the SolidWorks Menu Bar, click on:

Tools → SWOOD Design → SWOOD Connect → HOMAG →  *Transfer to productionManager*

If the transfer succeeds, the following message is displayed:

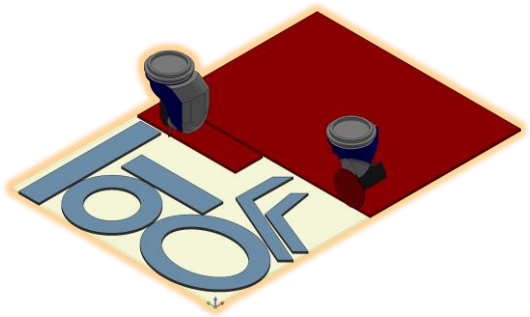


In the productionManager interface, the order will be present in the list.



SWOOD CAM – Offcutting

This new machining has been created to detect reusable offcuts in Nesting boards and to propose several operations to machine them.

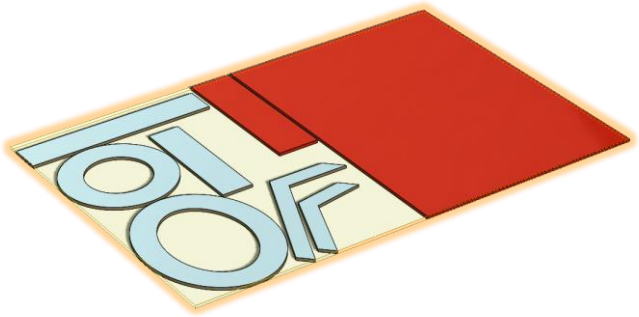


I. How to Use

The Offcutting machining will only detect the offcuts generated during the nesting assembly creation.

1. *Create a Nesting Assembly*

It is possible to use the Reusable offcuts feature documentation to create a Nesting assembly containing Offcuts.



2. *Offcutting Automatic Machining*

The Offcutting has an automatic detection of offcuts in an assembly, which means that no selection is necessary to use the machining.

3. *Choosing an operation*

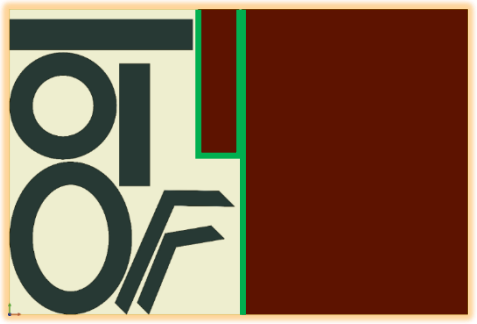
According to the used tool, some operations will be available:

- Move tool center
- Contour
- Partial contouring

- Saw

4. *Milling*

The Offcutting machining path is made only inside the nesting sheet, leaving the outside edges raw.



5. *New Parameter*

With this new parameter, if two offcuts are generated by the Nesting, it is possible to decide which one will be machined by the operation:

-  **Both offcuts**: Will machine the two offcuts.
-  **First offcut**: Will machine the first offcut.
-  **Second offcut**: Will only machine the second offcut.

II. Exercise

1. *Download the package*

Link

2. *Run the "offcutting.exe" program*

Running this program will make SWOOD point to another SWOODData, the one you'll need to practice.

3. *Open "nesting_result" SolidWorks assembly (in "Exercise/Panels" folder)*

4. Create an “ Offcutting” machining

a. Open the “SWOOD CAM” tab in the FeatureManager

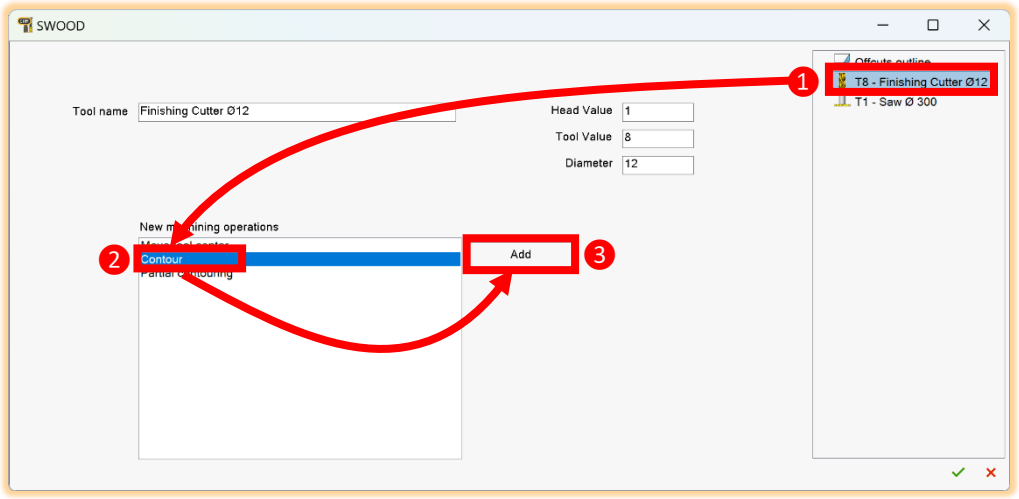
b. Click on the “ Offcutting” command

Click on Tools/SWOOD CAM /Offcutting.

5. Create a “ Contour” operation on the second offcut

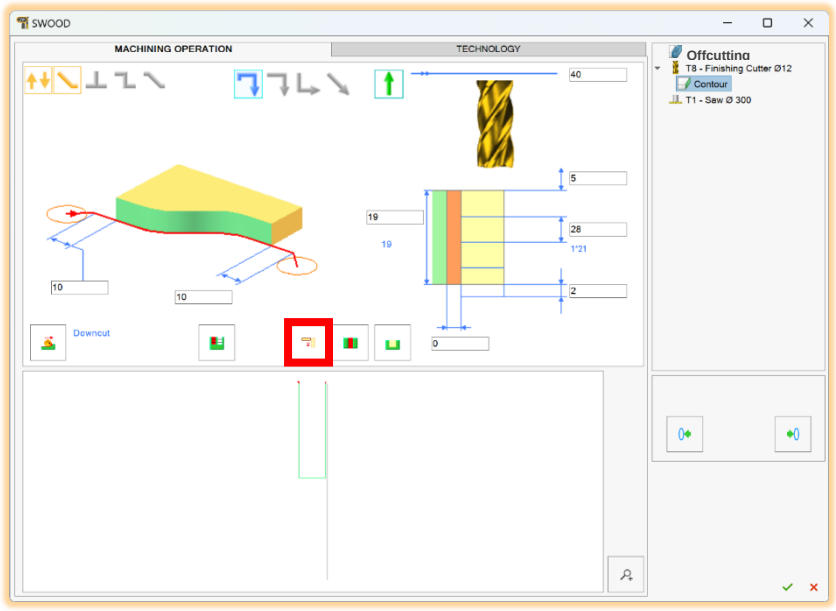
a. Create the operation

In the operation window that opens, select the Finishing cutter and add a Contour operation.



b. Filter the second Offcut

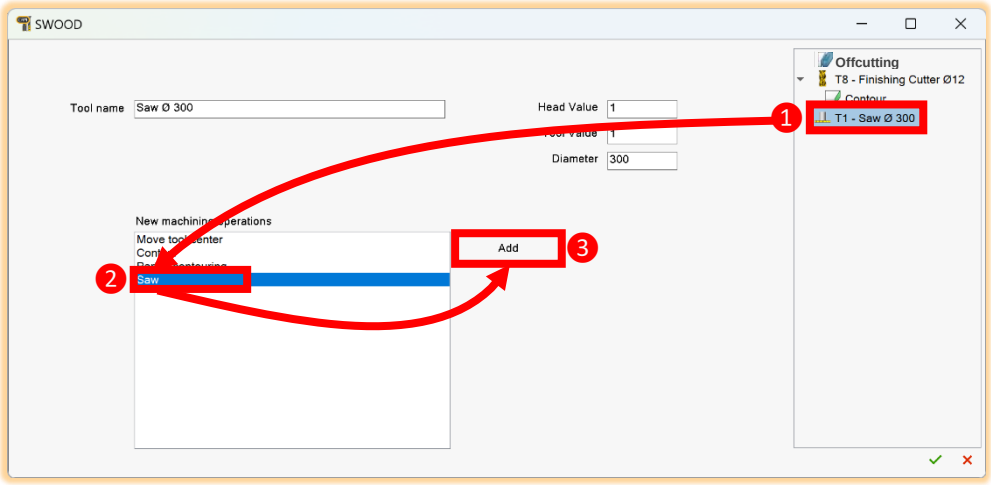
Click on the new parameter to reach the “ Second offcut”.



6. *Create a "Saw" operation on the first offcut*

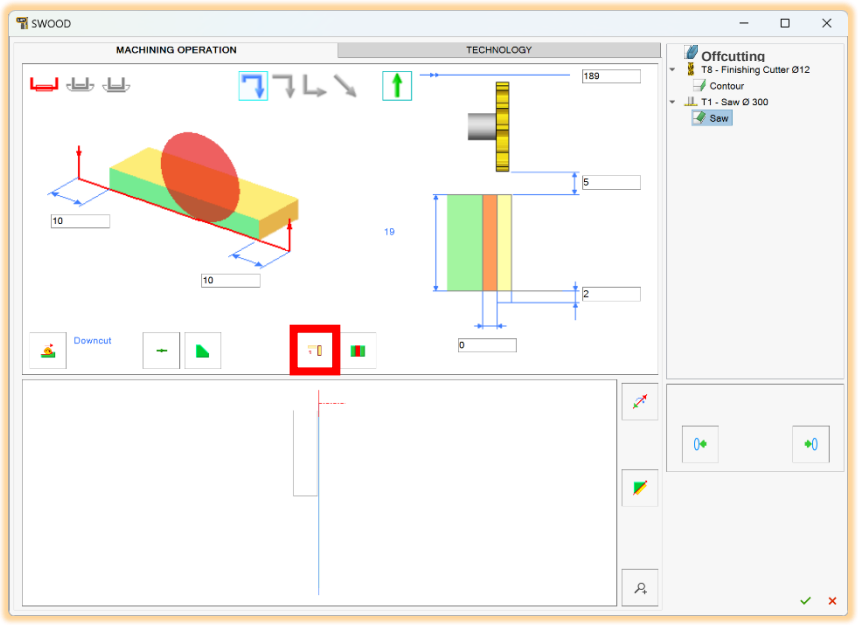
a. Create the operation

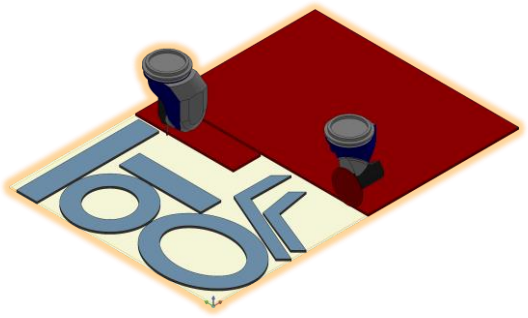
In the operation window that opens, select the Saw and add a Saw operation.



b. Filter the first Offcut

Click on the new parameter to reach the " First offcut" and ✓ validate the window.



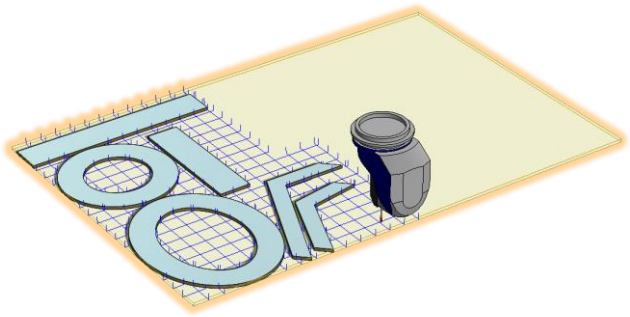


7. *Restore the SWOODData*

Reopen the package (offcutting.exe file) and click on Restore SWOOD Data to restore the previous setup.

SWOOD CAM – Waste Cutting

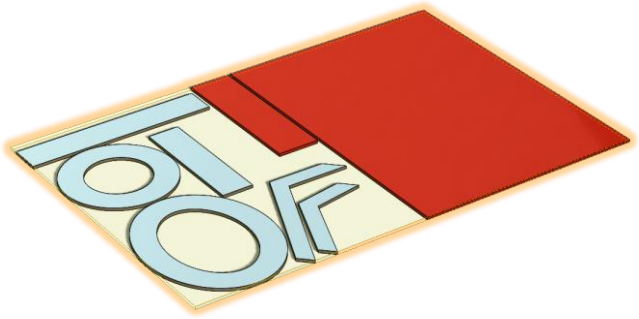
This new operation has been created to cut the waste between the parts on Nesting boards which, for example, can be useful for companies that use automated conveyors and want to avoid waste jamming.



I. How to Use

1. *Create a Nesting Assembly*

It is possible to use the Reusable offcuts feature documentation to create a Nesting assembly containing Offcuts.



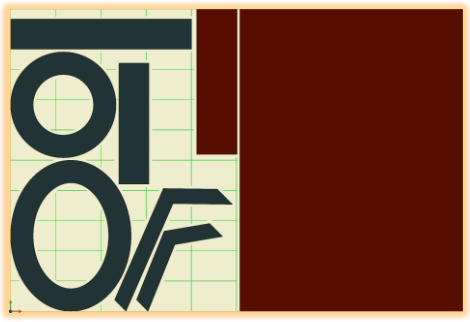
2. *Waste Cutting Operation*

Through the Outline machining, it is possible to reach the Waste cutting operation using the following tools:

- Roughing cutter
- Finishing cutter
- Grooving Shank-Type Cutter
- Shank-Type Cutter

3. *Milling*

The Waste cutting operation path is made only inside the nesting sheet, leaving the outside edges raw.



4. *New Parameters*

The Waste cutting operation has several parameters in order to set it up correctly:

- **Minimum distance between two horizontal lines**: Defines the horizontal grid spacing.
- **Minimum distance between two vertical lines**: Defines the vertical grid spacing.
- **Minimum line size**: Defines the minimum length below which a line will not be considered.
- **Offset**: Defines the distance between the parts and the grid.

Another parameter "Outer/Inners" allows applying the grid to all the edges of the part or to exclude the inner edges.

II. Exercise

1. *Download the package*

Link

2. *Run the "waste_cutting.exe" program*

Running this program will make SWOOD point to another SWOODData, the one you'll need to practice.

3. *Open "nesting_result" SolidWorks assembly (in "Exercise/Panels" folder)*

4. *Create a "Waste cutting" operation*

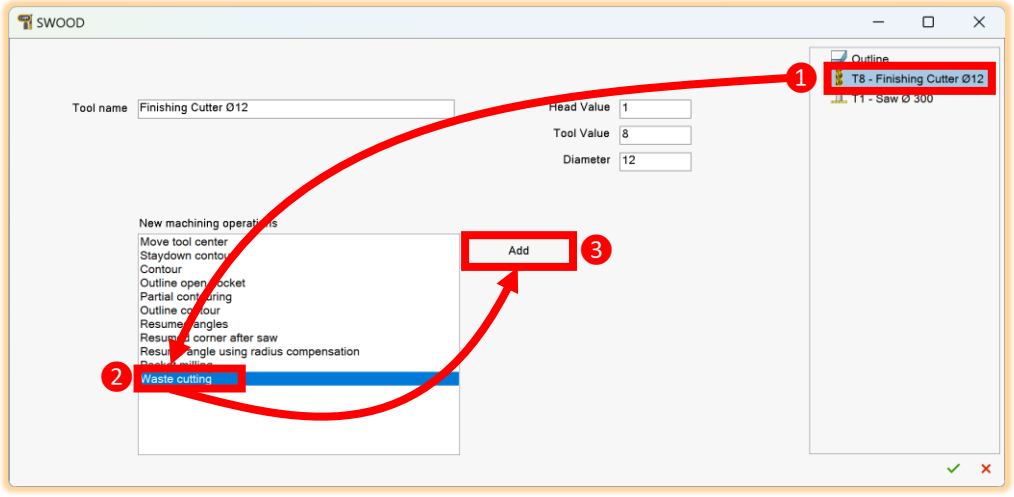
   a. *Open the "SWOOD CAM" tab in the FeatureManager*

   b. *Click on the "Outline" command*

Click on Tools/SWOOD CAM /Outline.

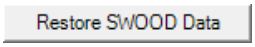
   c. *Add the "Waste cutting" operation*

Select the Finishing cutter, select the "Waste cutting" operation, and click on the "Add" button.



5. *Modify the operation parameters*

6. *Restore the SWOODData*

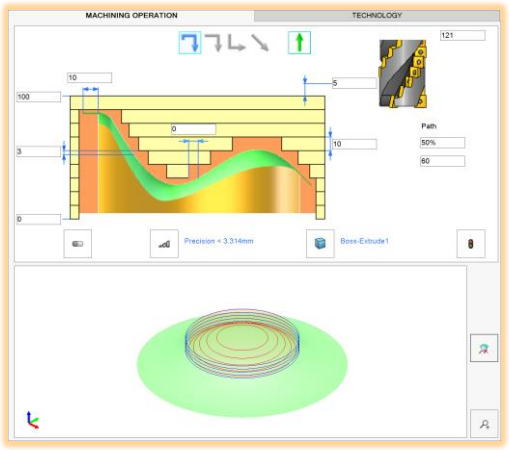
 Reopen the package (waste_cutting.exe file) and click on 

to restore the previous setup.

SWOOD CAM – Upgraded Roughing Volume Operation

The Roughing Volume operation has been upgraded to be more stable and to add some new options:

- The possibility to add a Contour to the roughing.
- The possibility to select a Precision to the calculation.
- The possibility to modify several parameters before rebuilding the calculation.
- The possibility to use stock, bodies or sketches as a Milling area.



I. How to Use

1. *Roughing Volume Operation*

It is possible to apply the Roughing Volume operation on any part volume without any selection.

Through the 3D Milling machining, it is possible to reach the Roughing Volume operation using the following tools:

- Roughing cutter
- Finishing cutter
- Shank-Type Cutter

2. *New way of working*

From SWOOD 2023, the recalculation of this operation won't be automatically applied after each parameter modification. The reason for this is that recalculation can take a long time with each parameter modification, especially with a complex volume.

Therefore, after every parameter modification, the new "Rebuild" button will appear with a warning sign on it . It will be necessary to click on it for the recalculation to be done.

3. *Removed Parameters*

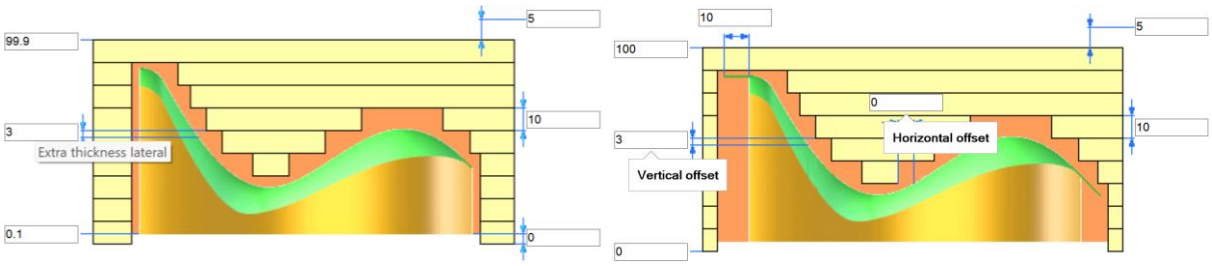
Some parameters of this operation, present in SWOOD 2022 and previous versions, have been removed:

- The "Roughing type" parameter has been removed to always define the roughing with an extension. If the user doesn't want an extension to the operation, the "Surface extension mm" parameter can be simply set to zero.
- The "Bottom Exceeding" parameter has been removed from the operation's interface. It is still possible to get a bottom exceeding to the roughing by setting a negative value to the "Zmin mm" parameter.

4. *New Parameters*

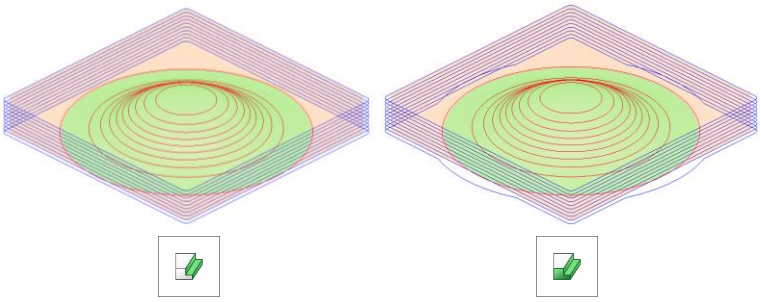
Some parameters of this operation have been added to improve the operation performance:

- The "Vertical offset" and the "Horizontal offset" have been created to replace the old "Extra thickness lateral" parameter (which has been removed).

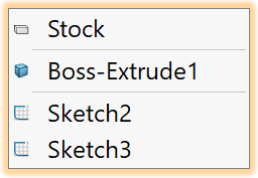


2022 ***2023***

- The "Enable contouring during 3-axis roughing" button has been created to add an offset to the part's volume. This parameter is different from the "Surface extension mm" parameter which adds an offset to the part's roughing.



- The  "Precision" button has been created to choose which level of accuracy should be used in detecting the reference volume for the operation. The greater the precision chosen, the more precise the reference volume will be, but the longer the computation time.
- The "Milling area" button has been created to choose which object will be used to define the Milling area.



By default, the  stock volume is selected, but it is possible to choose among the other  Solid Bodies of the part or existing  Sketches which are normal to the OP0 origin.

- The  "Rebuild" button has been created to launch the recalculation of the operation, after modifying one or several parameters. More precisions in the New way of working chapter.

II. Exercise

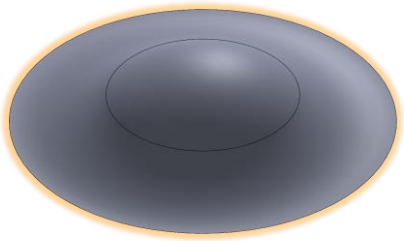
1. *Download the package*

Link 

2. *Run the "roughing_volume.exe" program*

Running this program will make SWOOD point to another SWOODData, the one you'll need to practice.

3. *Open the "3d_part" SolidWorks part (in "Exercise/Panels" folder).*

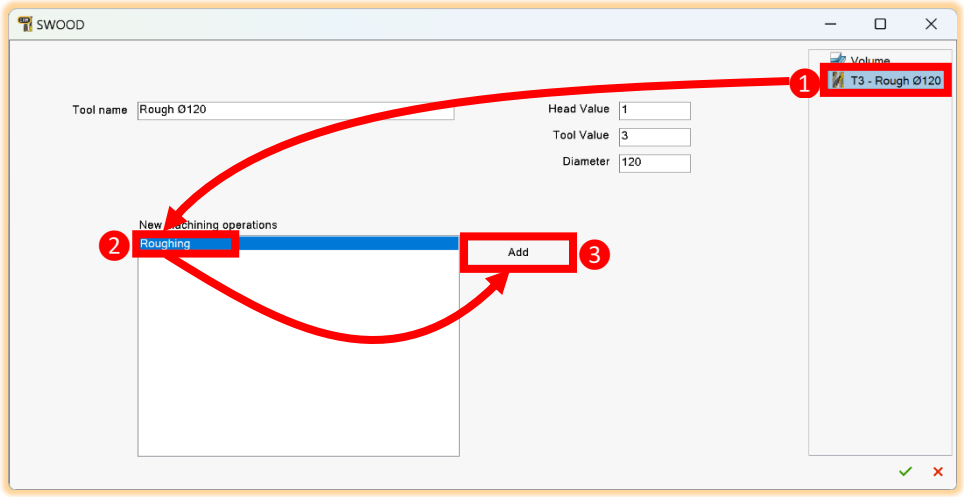


4. *Create a "Roughing" operation*

On the SolidWorks FeatureManager Design Tree, develop the Solid Bodies folder and select the "Revolve" body.

Click on Tools/SWOOD CAM / 3D Milling.

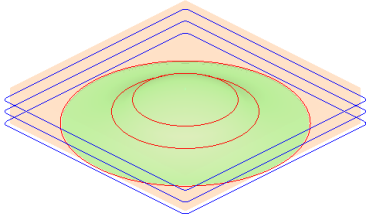
Select the Rough Ø120 tool, select the "Roughing" operation and click on the "Add" button.



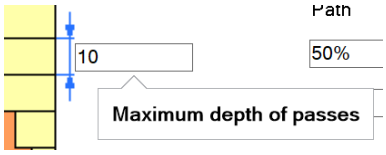
5. *Modify the parameters*

a. *Basic parameters*

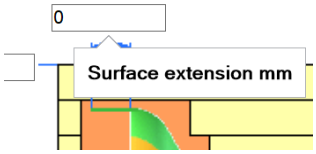
- Rebuild once the calculation is done to see the initial state.



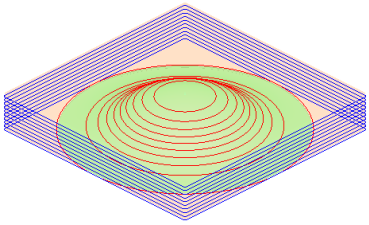
- Put a "Maximum depth of passes" at 10mm.



- Put a “Surface extension mm” at 0mm.

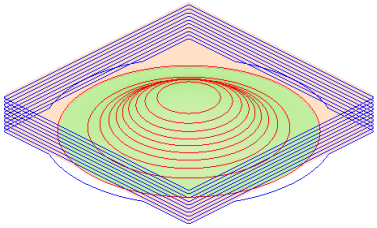


- Rebuild again to see the effect.



b. *Overlap strategy*

- Enable the “Overlap strategy”.
- Rebuild to see the effect.

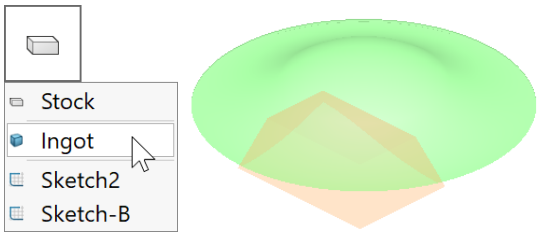


- Now, disable the “Overlap strategy”.
- Rebuild.

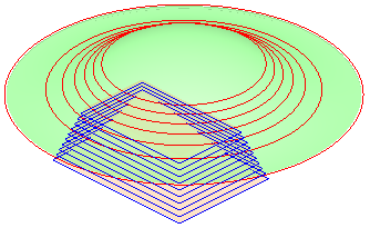
6. *Modify the Milling area*

a. *Ingot body*

- Click on the “Milling area” button, which is set at “stock” by default, and select the “Ingot” body.

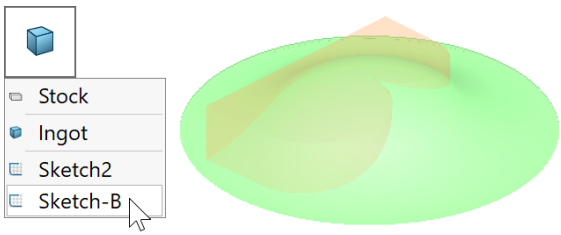


- Rebuild.

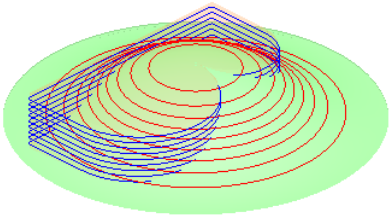


b. *Sketch B*

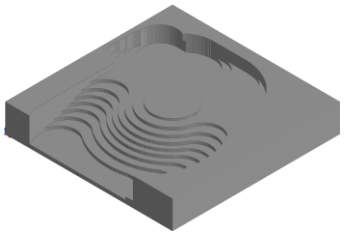
- Click again on the "Milling area" button, and this time select the "Sketch-B" sketch.



- Rebuild.



- Validate the operation window and simulate the operation.

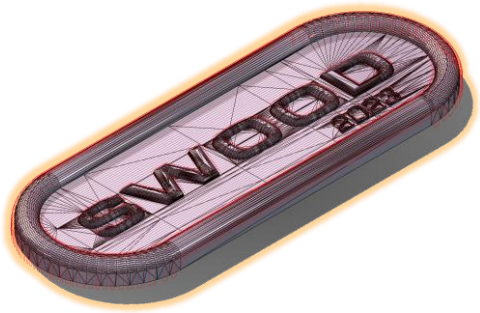


7. *Restore the SWOODData*

Reopen the package (roughing_volume.exe file) and click on Restore SWOOD Data to restore the previous setup.

SWOOD CAM – 3D Milling Surface and Graphic Bodies

This new feature has been created to allow applying a 3D Milling machining directly to Surface bodies and Graphic bodies; for example, for a STL file imported into SolidWorks.



I. How to Use

It is possible to apply the 3D Milling machining onto a Surface body or a Graphic body when selecting it on the FeatureManager Design Tree. Like regular 3D Milling machining, it is possible to apply the two operations:

- Roughing
- Finishing

Warning: If the part doesn't have a Solid body (for external file import, for example) it won't be possible to neither do any other automatic machining on it, nor Material Removal Simulation.

Warning: This specific feature doesn't work below SolidWorks 2022 version.

II. Exercise

This exercise only works in SolidWorks 2022 version and above.

1. *Download the package*

Link

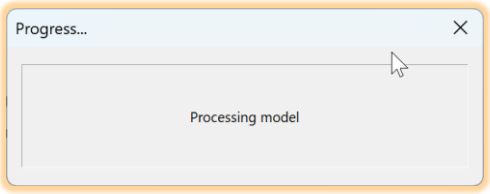
2. *Run the "surface_bodies.exe" program*

Running this program will make SWOOD point to another SWOODData, the one you'll need to practice.

3. *Import the "SWOOD_logo.STL" file in SolidWorks*

On the SolidWorks Menu Bar, click on the Open button.

Select the "SWOOD_logo.STL" file in the "Exercise" folder, and click on "Open".



4. *Apply the 3D Milling machining*

a. *Create a Milling phase*

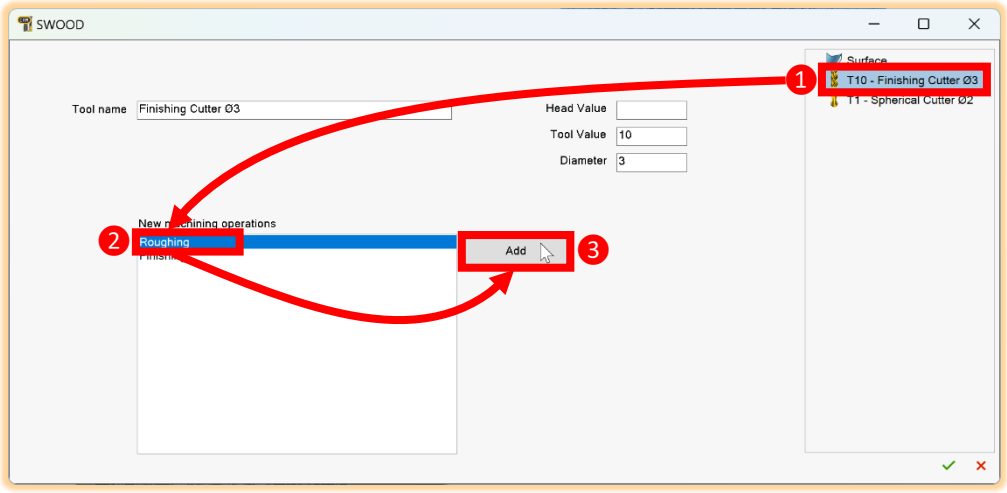
- Once the model has been imported, on the SolidWorks Menu Bar click on:

  Tools/SWOOD CAM/ Program Setting

- Create a program origin by clicking on the " Origins" button and just validate the "Positioning" pane.
- Go to the " SWOODCAM Library", the " Tool" sub-tab, and insert the " Finishing Cutter Ø3" and the " Spherical Cutter Ø2" tools.

b. *Add a Roughing operation*

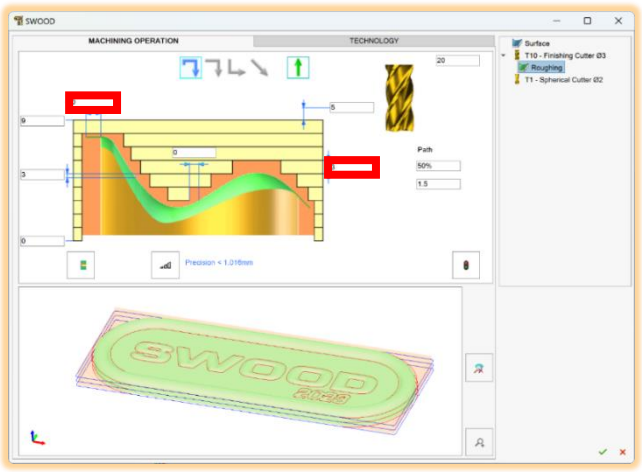
- Go back to the FeatureManager Design Tree, expand the " Surface Bodies" folder and select the " Surface-Imported1" surface.
- On the SolidWorks Menu Bar, click on: *Tools/SWOOD CAM/ 3D Milling*
- When the operation window opens, select the Finishing Cutter, click on the "Roughing" operation and click on "Add".



Set 0 to the parameter “Surface extension mm”.

Set 3mm to the parameter “Maximum depth of passes”.

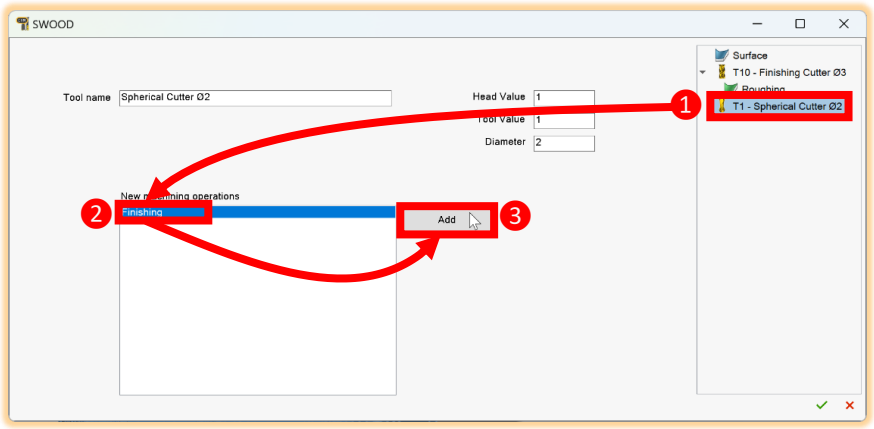
Click on the “ Rebuild” button.



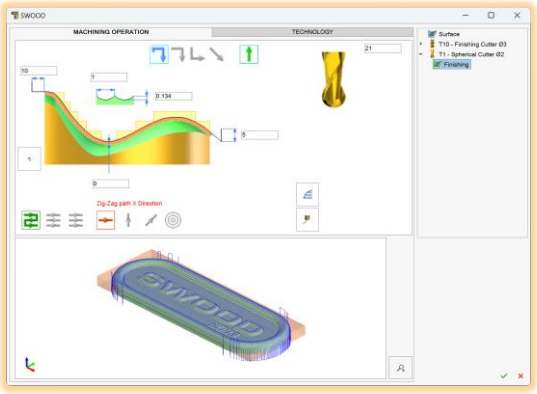
Do not validate the window.

c. *Add a Finishing operation*

Select the Spherical Cutter, click on the “Finishing” operation and click on “Add”.



Do not change any parameter and validate the window.

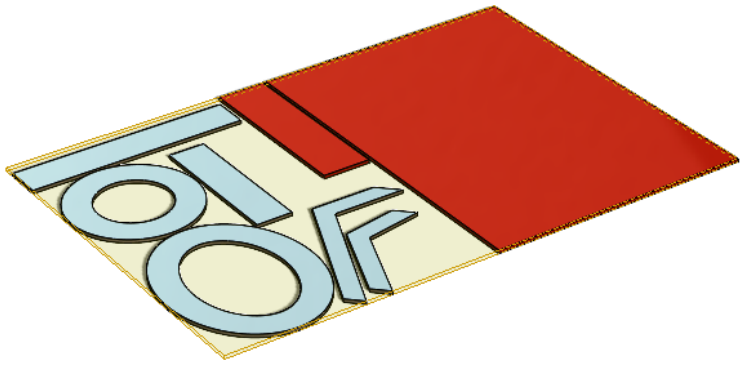


5. *Restore the SWOODData*

Reopen the package (surface_bodies.exe file) and click on Restore SWOOD Data to restore the previous setup.

SWOOD Nesting – Reusable Offcuts

This new feature has been created to generate one or more reusable offcuts from the remaining space on the Nesting boards.



I. How to Set Up Reusable Offcuts

1. *Boards Edition*

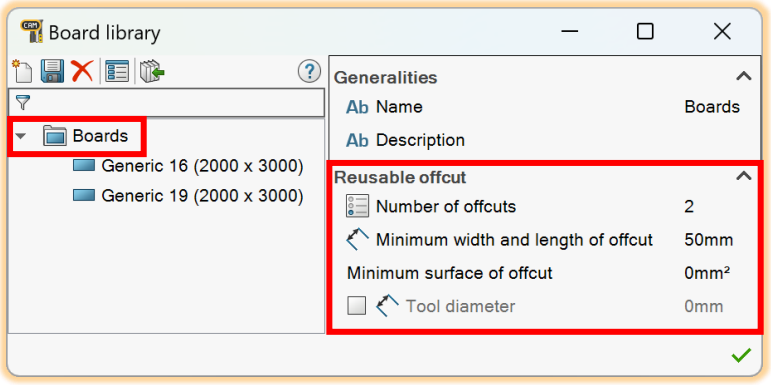
The Reusable offcuts parameters can be set in the Nesting board library.

On the SolidWorks Menu Bar, click on "Tools/SWOOD CAM/ Nesting", then click on the " Boards library" button.

a. Global Parameters

It is possible to set global parameters for all reusable offcuts on all boards.

On the Boards library, click on " Boards", and the Reusable offcut parameters will appear.



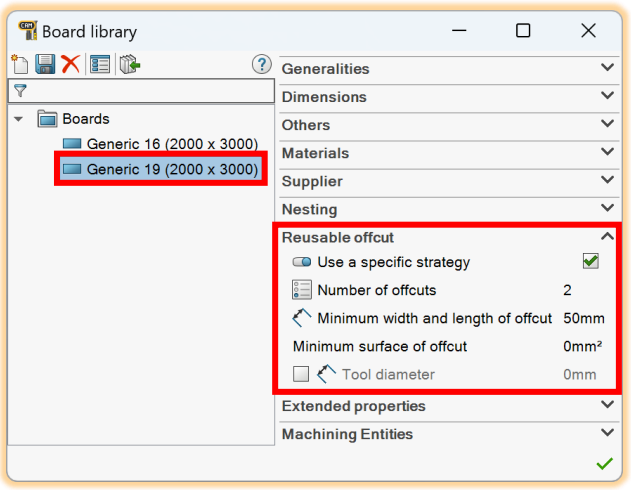
b. Specific Board Parameters

It is possible to set specific parameters for each board on the Boards library.

On the Boards library, click on a specific board.

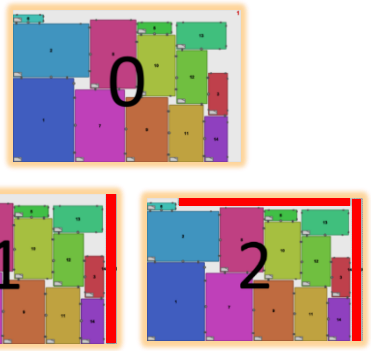
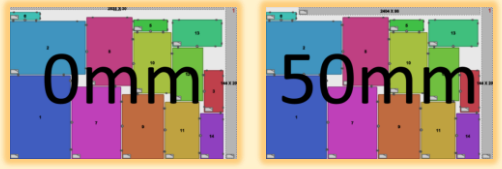
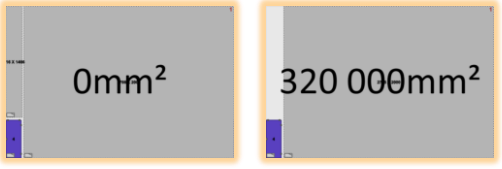
Under the "Nesting" parameters, you will find find the "Reusable offcut" parameters.

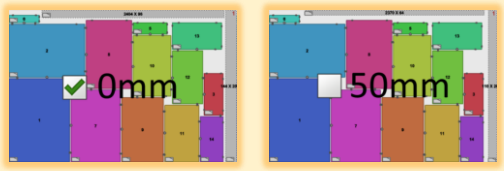
To activate the specific board parameters, click on the "Use a specific strategy" checkbox.



c. *Parameters*

There are four parameters to set up the Reusable offcuts:

Parameter	*Description*	*Options*
Number of offcuts	Defines the maximum number of reusable offcuts per board in the nesting. ⚠ If no offcut is desired, choose zero.	
Minimum width and length of offcut	Defines the minimum size at which a reusable offcut can be generated.	
Minimum surface of offcut	Defines the minimum surface (in mm²) at which a reusable offcut can be generated.	

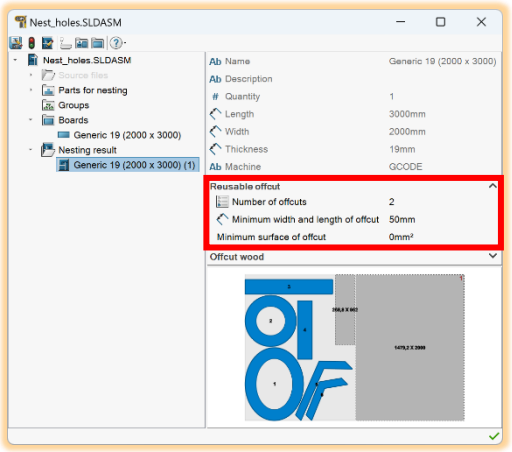
☐ Tool diameter	If checked, defines the diameter of the tool used to machine reusable offcut only.	

2. *Result Edition*

After Rebuilding nesting result, it is possible to reedit the Reusable offcuts sheet per sheet.

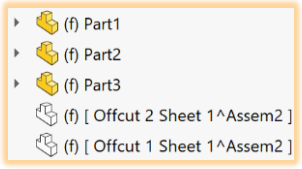
To do so, click on a sheet to reach its Reusable offcut's parameters.

Modifying these parameters will only take effect on the selected sheet.



II. How to Use

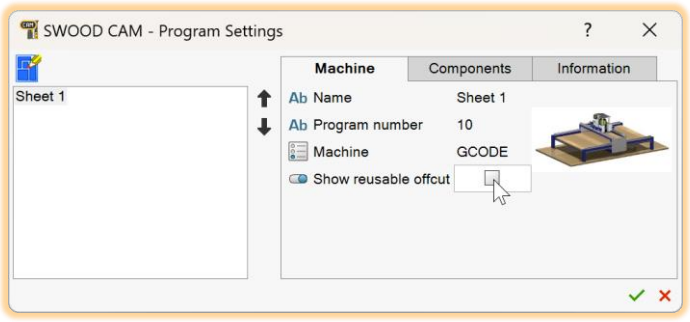
Once the Nesting result is validated, the Nesting assembly is generated adding the offcuts as virtual parts which are suppressed.



1. *Show the Reusable Offcuts*

By default, the offcuts are hidden.

To show them, go to the SWOOD CAM tab on the FeatureManager, then double-click on the phase. On the Program Settings window, click on the "Show reusable offcut" checkbox.



2. *Reusable Offcut Milling*

With the new Offcutting machining, it is possible to mill the offcuts. That will only detect the offcuts and will only machine the sides in contact with the inside of the nesting sheet.

III. Exercise

1. *Download the package*

Link

2. *Run the "reusable_offcut.exe" program*

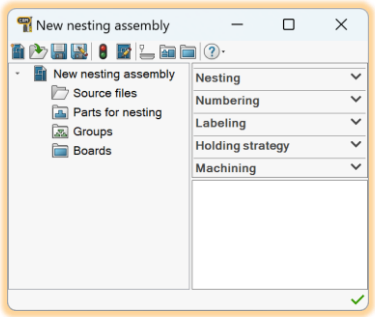
Running this program will make SWOOD point to another SWOODData, the one you'll need to practice.

3. *Create a Nesting project*

a. *Create a new project*

To reach the Nesting project window, on the SolidWorks Menu Bar, click on:

Tools → SWOOD CAM → Nesting

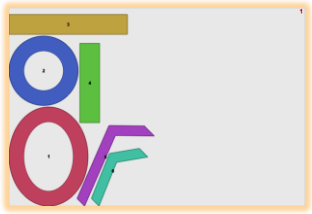


b. *Add source files*

To add parts to the Nesting project, right click on " Source file" and click on " Add source file".

When the Open window appears, select all the parts in the "Exercises/Panels" folder and click on "Open".

Rebuild the Nesting project once, to see the first nesting result without the offcuts.



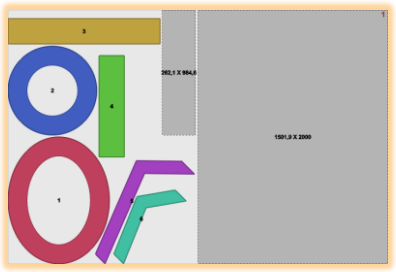
4. *Setting of reusable offcuts*

Click on the "Boards library" on the Toolbar, then click on "Boards" in the boards list.

Set the "Number of offcuts" to 2.

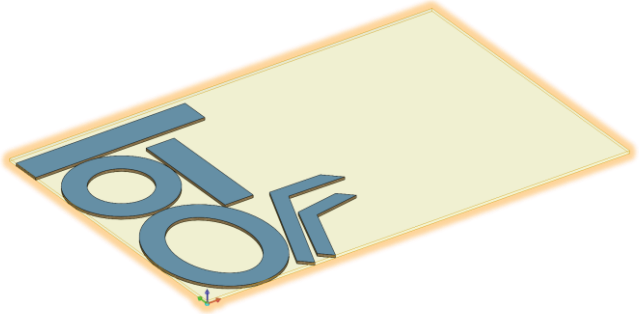
Save the Boards library and validate the window.

Rebuild the Nesting project, to see the nesting result with the offcuts included.

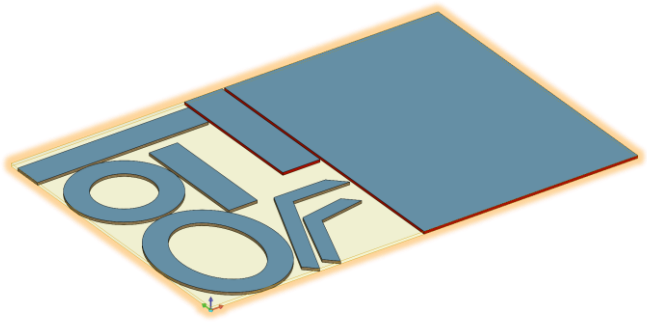


5. *Generate the Nesting assembly*

Validate the Nesting projects window in order to generate the Nesting assembly.



Go to the SWOOD CAM tab on the FeatureManager, then double-click on the milling phase. On the Program Settings window, click on the "Show reusable offcut" checkbox .

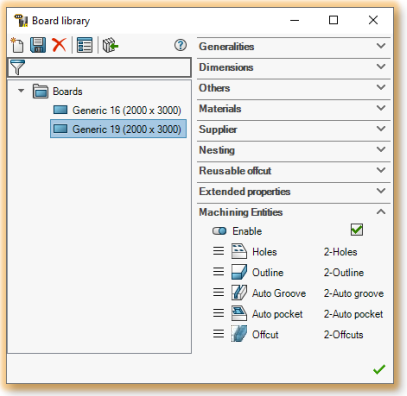


6. *Restore the SWOODData*

Reopen the package (reusable_offcut.exe file) and click on Restore SWOOD Data to restore the previous setup.

SWOOD Nesting – Machining Insertion from Boards Library

This new feature has been created to automatically insert specific machinings into the Nested assembly according to the used board.



I. How to Set Up the Machinings

1. *Reach Settings*

To reach the new boards machining setting, on the SolidWorks Menu Bar, click on:

Tools/SWOOD CAM/ Nesting

In the Nesting project window, click on the "Boards library" situated on the toolbar.

It is possible to reach the "Machinings" parameters by selecting any board in the boards list.

2. *Set Machinings*

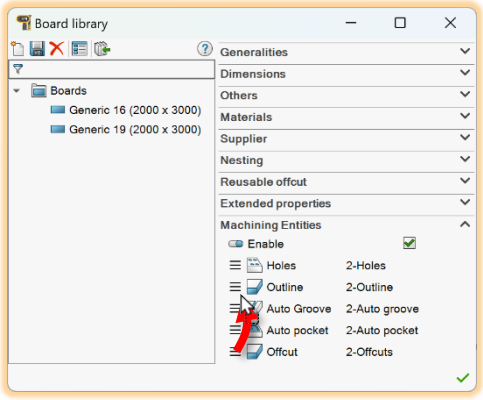
Once the "Enable" checkbox is checked, it is possible to see all the following automatic machinings:

- Holes
- Outline
- Auto Groove
- Auto pocket
- Offcut

To set the machinings, click on the list on the right, where it is written "Disable".

For each machining, it will be possible to select all the same type machinings from the "Machining Library".

It is possible to reorganize the machining by dragging the "☰" button of the machining, and dropping it on the wanted position.



II. How to Use

To see the effect of this new feature, it is necessary to create a Nesting project using the concerned board.

Once the Nesting result is ✓ validated, the Nesting assembly is generated adding the set machinings to the sheet Milling phase.

⚠ Although the "Automatic machining insertion" are already set, if the machinings are enabled in the Boards library, it will be the board machinings that will be inserted in the Nesting assembly.

III. Exercise

1. *Download the package*

Link

2. *Run the "machining_insertion.exe" program*

Running this program will make SWOOD point to another SWOODData, the one you'll need to practice.

3. *Setting of Machining insertion on a board*

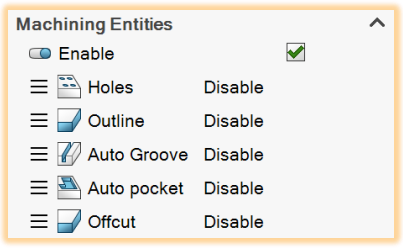
On the SolidWorks Menu Bar click on:

Tools/SWOOD CAM/ Nesting

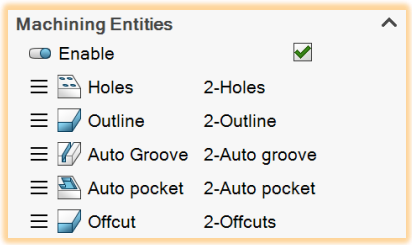
Click on the "Boards library" on the Toolbar, then click on the "Generic 19 (2000 x 300" board in the boards list.

On the last section of the parameters, called "Machinings", click on the "Enable" checkbox.

a. *Assign the machinings*



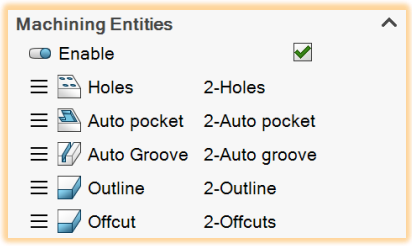
On every machining, click on the list on the right, where it is written "Disable", and select the "2-XXX" machining.



b. *Reorder the machinings*

Using the "☰" buttons, reorder the machinings:

- Placing the "Auto pocket" under the "Holes".
- Placing the "Auto Groove" under the "Auto pocket".



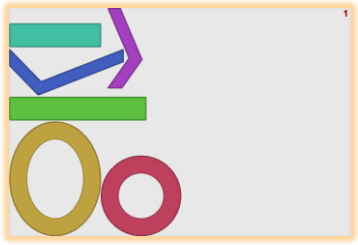
Save the Board library by clicking on the "Save" button, and ✓ validate the window.

4. *Create a Nesting project*

On the Nesting project window, right-click on the " Source files" and click on " Add source file".

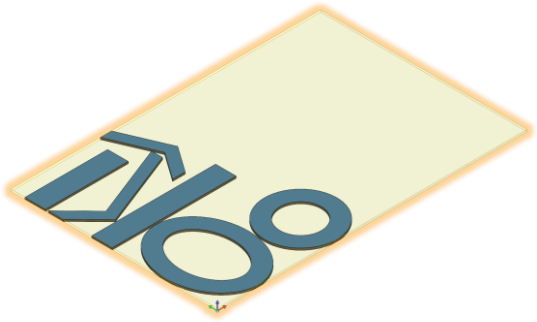
When the Open window appears, select all the parts in the "Exercise" folder and click on "Open".

Rebuild the Nesting project once, to see the nesting result.



5. *Generate the Nesting assembly*

Validate the Nesting projects window in order to generate the Nesting assembly.



Go to the SWOOD CAM tab on the FeatureManager, then double-click on the Milling phase. On the Program Settings window, click on the " Show reusable offcut" checkbox .

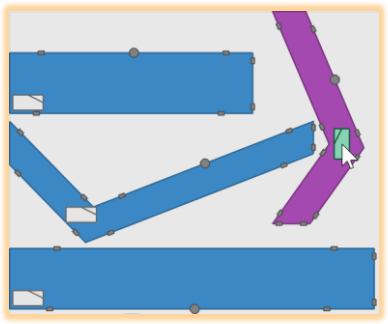
See the Milling phase and all its machining already inserted.

6. *Restore the SWOODData*

Reopen the package (machining_insertion.exe file) and click on Restore SWOOD Data to restore the previous setup.

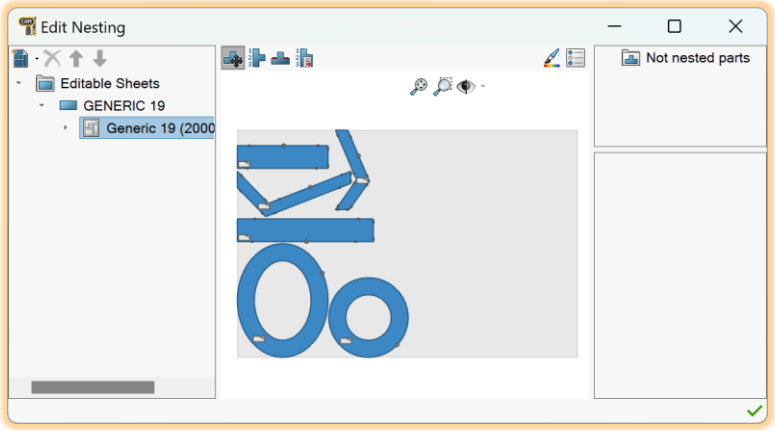
SWOOD Nesting – Rotate Labels Independently

This feature has been created to allow to rotate independently the labels of the nested parts in the Manual Nesting mode.

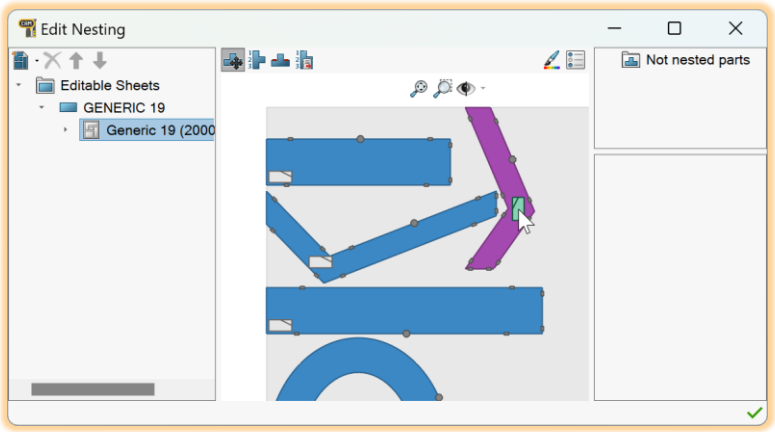


I. How to Use

When a Nesting project has been generated in the Nesting project window, it is possible to edit more precisely the result in the Manual Nesting window.

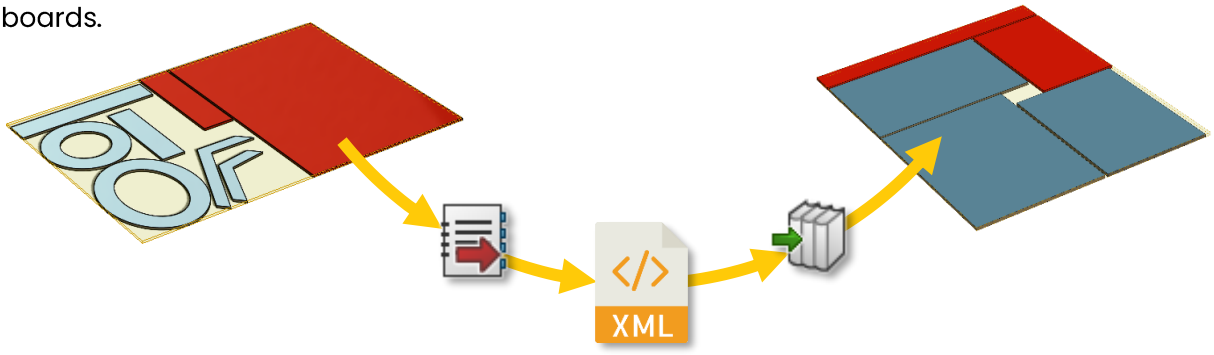


Select one of the labels which needs to be rotated individually and keep the click pressed. Press the rotation shortcut (R) to rotate it.



SWOOD Report – Offcut Export

This new feature has been created to be able to export the offcuts of a Nesting result in an external (XML) file, and to be able to reimport them in the Nesting Board library as new boards.



I. How to Set Up Offcut Export

1. *Activate Export*

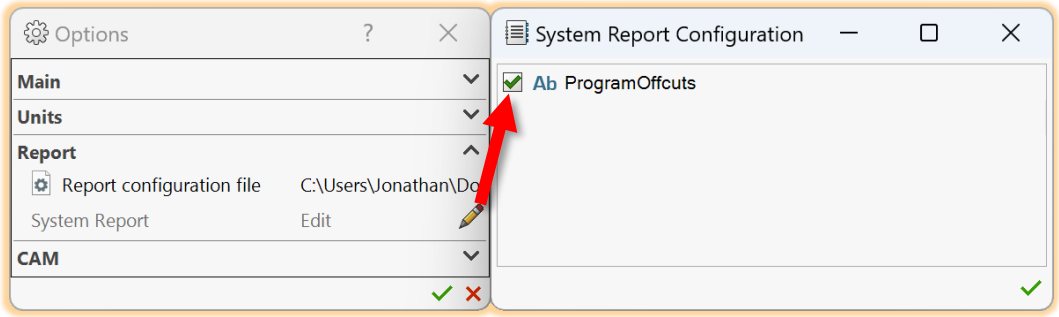
To be able to export the Nesting offcuts, it is necessary to activate the export in the report configuration.

To do so, in the SolidWorks Menu Bar, go to:

Tools/SWOOD CAM/ Settings

In the **Report** section, click on the System Report Edit button.

Finally, check the " ProgramOffcuts" option.

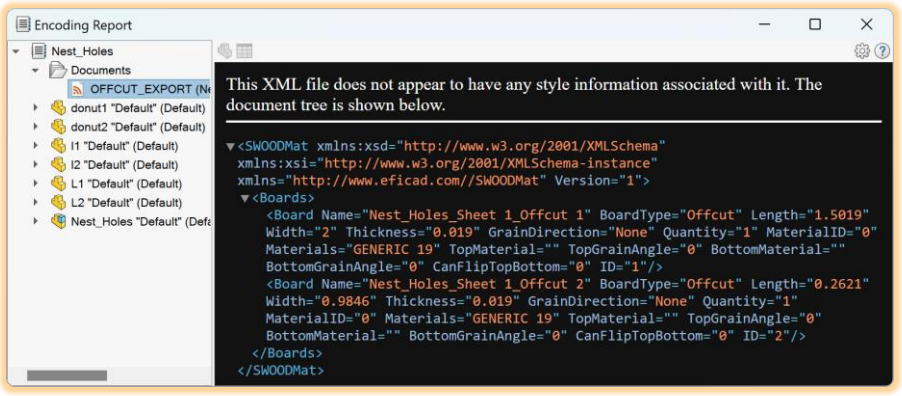


2. *Launch Report*

To launch a Report, on the SolidWorks Menu Bar, click on:

Tools/SWOOD CAM/ Report

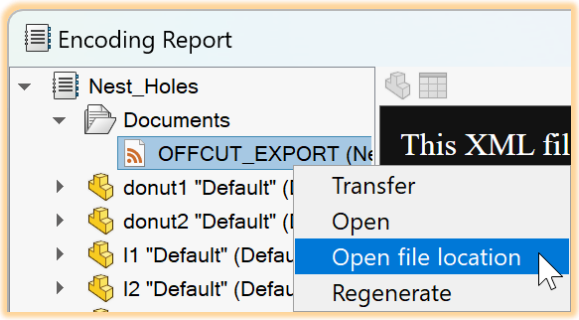
It is then possible to see the generated XML file in the Encoding Report window, under Documents.



II. How to Import the Offcuts in the Board Library

1. *Open XML File Location*

If the Encoding Report window is still open, it is possible to directly go to the XML file with a right click, and by selecting “Open file location”.



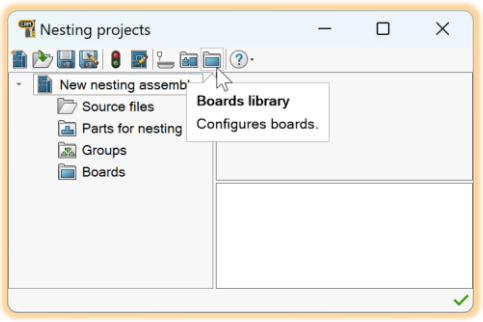
Otherwise, it will be necessary to go to the report folder which is saved at the same location as the Nesting assembly.

Then, go to the folder called “XMLs”.

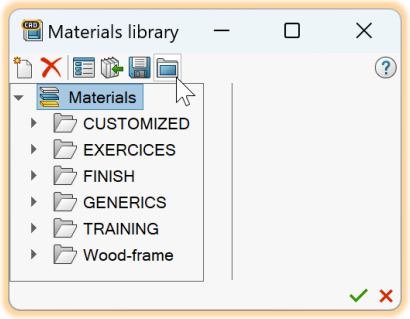
2. *Open Board Library*

There are two ways to import the Board library:

- From the Nesting interface: On the SolidWorks Menu Bar, click on: Tools/SWOOD CAM/ Nesting.
- Once the Nesting projects window is opened, it is possible to reach the “ Board library”.

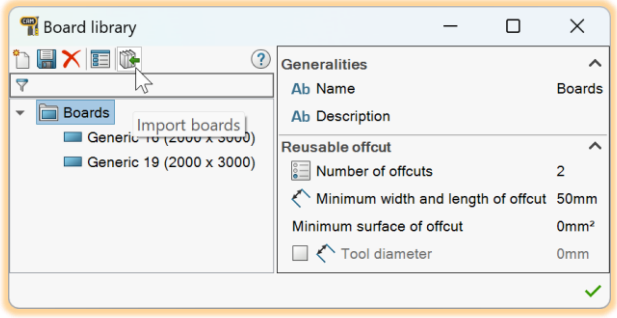


- From the Materials library: If SWOOD Design is enabled, go to the SWOOD Design Task Pane, and go to the Materials library. Right-click on the "Materials" icon, and click on "Edit". Once the Materials library window is opened, it is possible to reach the "Board library".



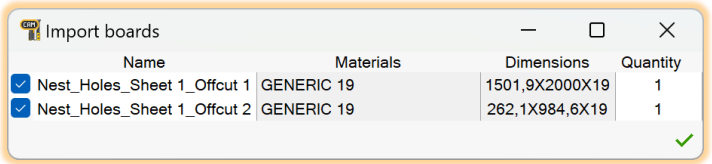
3. *Import XML File*

Once the Board library is opened, it is possible to reach the "Import boards" button.



This button leads to an open window filtered to only open XML files.

When a XML file is selected and the window is validated, an Import boards window opens and shows the offcuts information ready to be imported.



When the Import boards window is validated, the Board library window has a new folder called "▾ Imported boards" with the new boards generated from the offcuts.