SWOOD by EFICAD

WHAT'S NEW SWOOD 2025

EFICAD

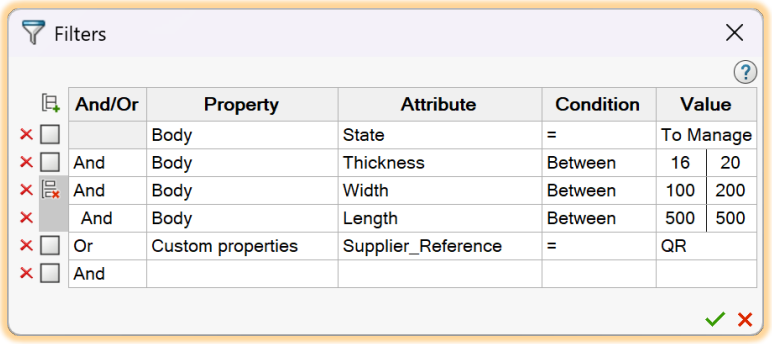
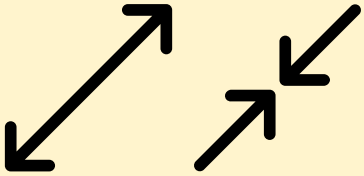
What's new in SWOOD 2025

The purpose of this document is to present the new features of SWOOD 2025, and to define their parameters so that they can be better understood.

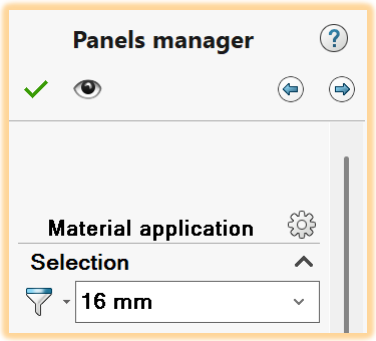
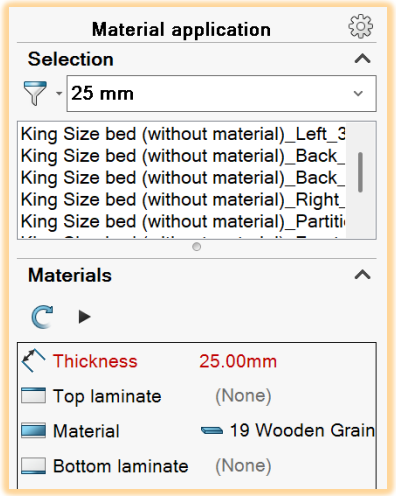
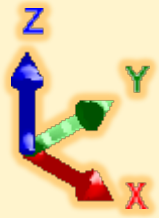
How does this document work?

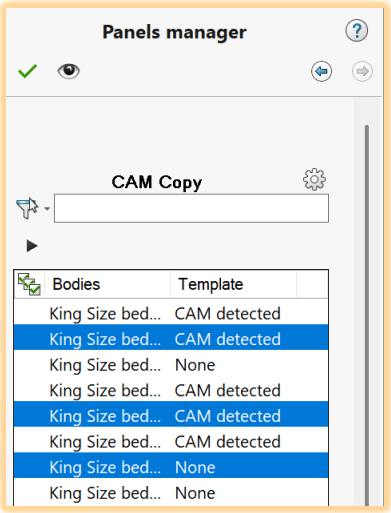
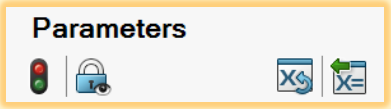
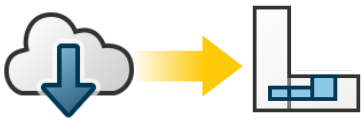
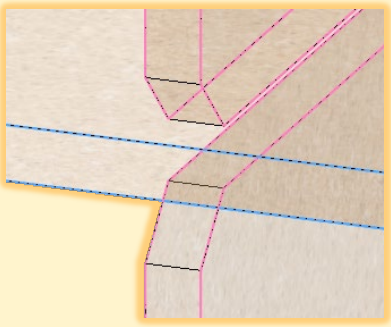
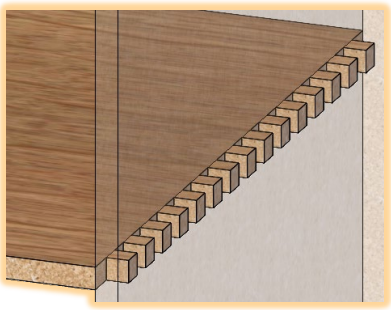
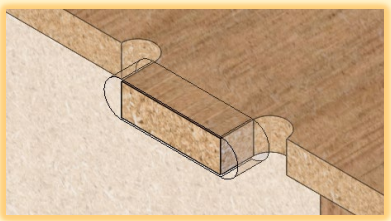
For each software (SWOOD Design, SWOOD CAM, SWOOD Nesting and SWOOD Report), you can click on a **bookmark**, which will redirect you to the corresponding chapter in this document.

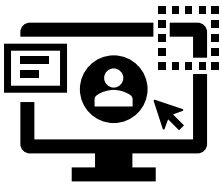
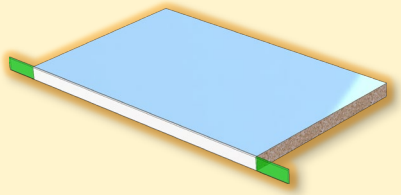
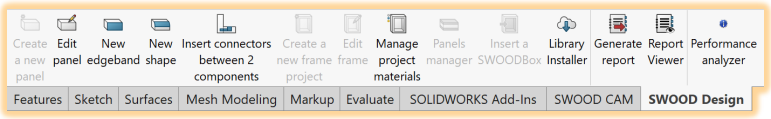
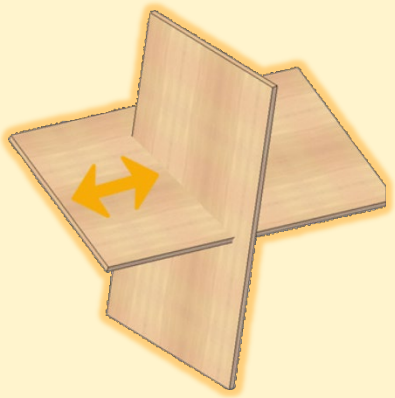
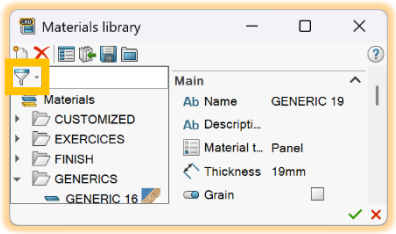
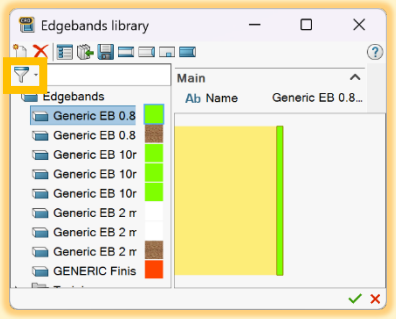
SWOOD

New feature	
Filters	
User Experience: Shortcuts for Expand/Collapse	

SWOOD Design

New feature	
Panels Manager: Material Application Filters	
Panels Manager: Inch Management	
Panels Manager: Forced Material Application	
Panels Manager: Reorientation of SolidWorks Assembly for Front View	

Panels Manager: CAM Copy	
SWOODBox: Default Parameters	
Library Installer: Connector Integration	
Panel Joint: Notch Flaring	
Panel Joint: Mortise and Tenon Repetition	
Panel Joint: New Mortise and Tenon Shapes	

[Panel Joint: Interface Enhancements]	
[Edgeband Overlength]	
[Enhanced User Interface: New CommandManager]	
[Enhanced User Interface: Texture Stability]	
[Filters: Materials Library]	
[Filters: Edgebands Library]	

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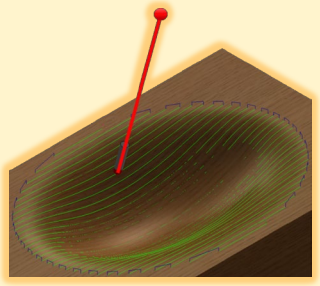
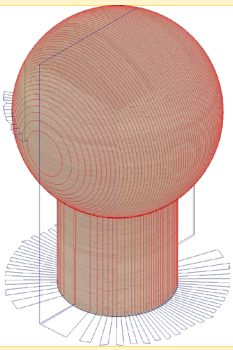
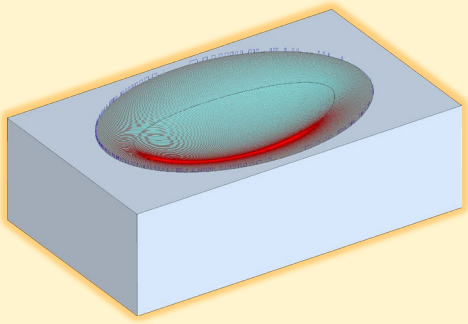
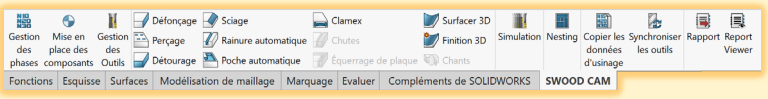
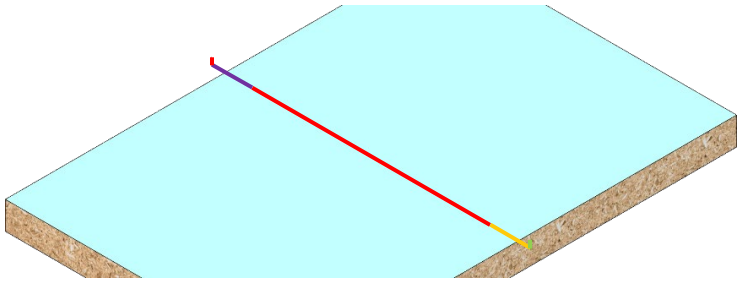
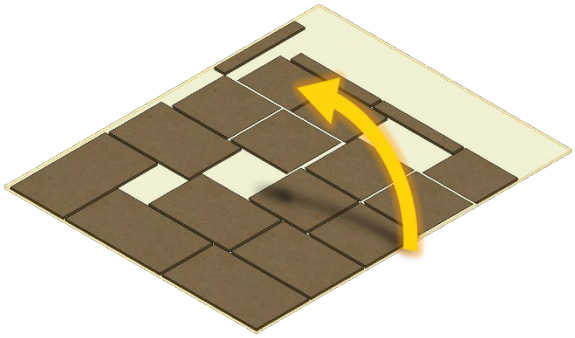
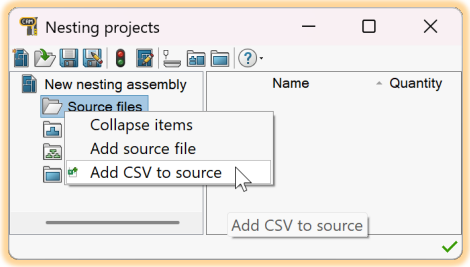
SWOOD CAM

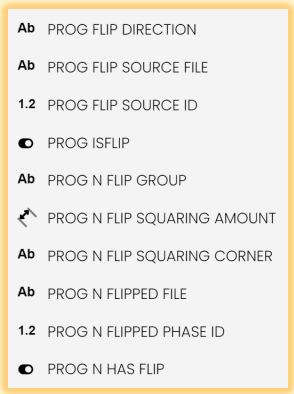
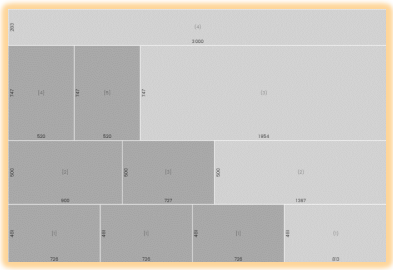
New feature	
Stock management in component positioning	
3D Finishing: New Tool Tilting Types	
3D Finishing: Mill Undercuts	
3D Finishing: New Trajectory Display	
Tools Synchro: New Features	
Enhanced User Interface: New CommandManager	

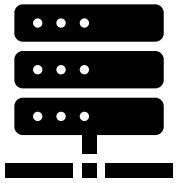
Entry / Exit Machining Feedrate	
Sheet Squaring	

SWOOD CAM Nesting

New feature	
Nesting Flip	
Filters: Parts for Nesting List	
Nesting from a CSV file	

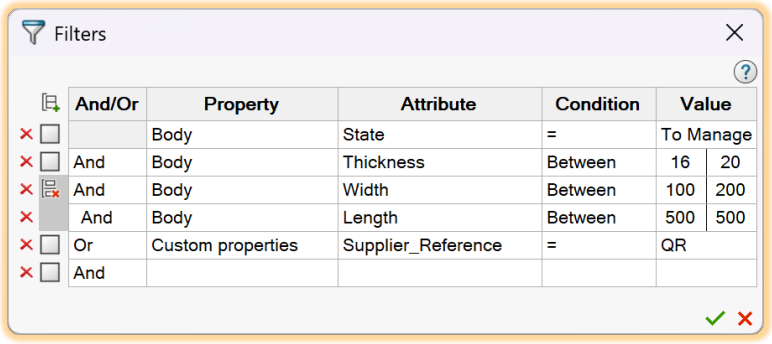
SWOOD Report

New feature	
New Activation	
New variables related to the flip	
Cutting Pattern New Features	
« Group By »	
Column Data Aggregation	
User Profiles	

Report Sharing on Local Server	
Report Sharing in Compressed Format	
Version Update Procedure	

SWOOD – Filters

SWOOD 2025 introduces a major cross-functional feature: filters. This powerful new tool is designed to offer greater control and precision when selecting elements within your projects. It allows users to define advanced selection criteria by combining several conditions based on various properties such as the status of a part, its dimensions (thickness, width, etc.), or even custom properties. These filters can be constructed using "And" or "Or" logic and can be grouped to further refine selections. For example, you can now easily select all panels of a specific thickness and width exceeding a certain value in order to apply a particular material or machining operation to them.



Example of a filter

How to use

1. *Access to filters*

Filters are available in many places in SWOOD.

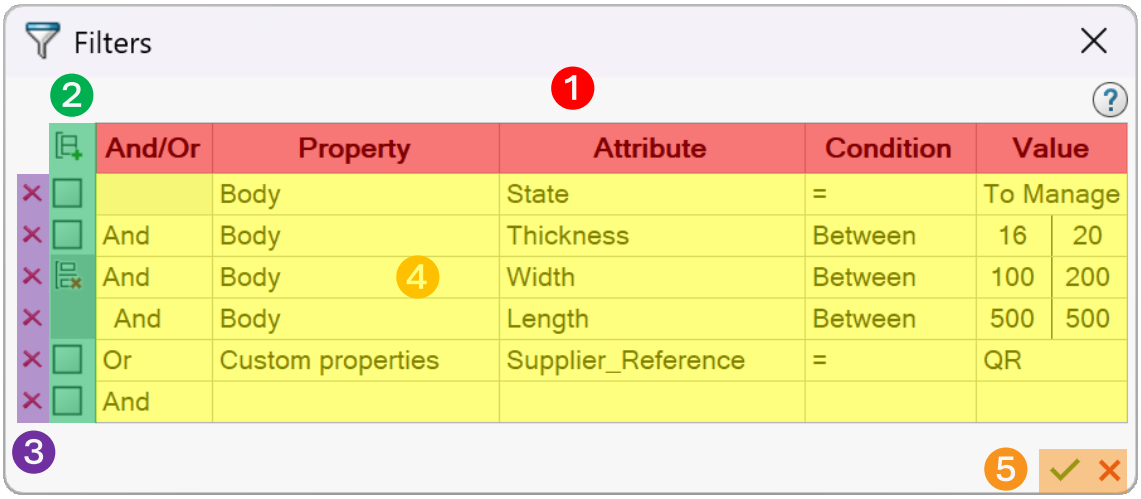
a. In the material application of Panels Manager (SWOOD Design)
b. In the CAM Copy of the Panels Manager (SWOOD Design)
c. In the Materials library (SWOOD Design)
d. In the Edgebands library (SWOOD Design)
e. In the parts for Nesting list (SWOOD Nesting)

2. *Filter application interface*

A filter is defined as follows:

Click on the arrow next to the filter to display the filter edit icon.

Next, the filter editing window opens.



The filter window is divided into several areas:

1. Columns
2. Groups
3. Filter deletion
4. Filter list
5. Validation

a. List of columns

Depending on where the filter window is opened, the columns may differ:

Source	Et/Ou	Property	Attribute	Condition	Value
Material application	Yes	Yes	Yes	Yes	Yes
CAM Copy	Yes	Yes	Yes	Yes	Yes
Materials library	Yes	No	Yes	Yes	Yes
Edgebands library	Yes	No	Yes	Yes	Yes
Parts for Nesting list	Yes	Yes	Yes	Yes	Yes

- **And/Or**

- Defines whether the condition should be added to the previous condition (with "And") or whether it can replace it (with "Or").

- For example: when you have "A And B," both conditions A AND B must be satisfied for the filter to be active.

- Another example: when you have "A Or B," condition A must be satisfied OR condition B must be satisfied for the filter to be active.

- **Property and attribute**

Depending on the origin of the filter, the properties may differ and their attributes.

  - In the material application of Panels Manager (SWOOD Design)
  - In the CAM Copy of the Panels Manager (SWOOD Design)
  - In the Materials library (SWOOD Design)
  - In the Edgebands library (SWOOD Design)
  - In the parts for Nesting list (SWOOD Nesting)

- **Condition**

Depending on the attribute selected, there are several types of conditions:

Sign	The condition is satisfied when:
=	The number is **equal** to the Value
!=	The number is **different** from the Value
›	The number is **greater** than the value
›=	The number is **greater than or equal** to the Value
‹	The number is **less** than the Value
‹=	The number is **less than or equal** to the Value
Between	The number is between a **minimum and maximum** value.

- Text: Allows text manipulation

Sign	The condition is satisfied when:
=	The text is **equal** to the Value
Contains	The text **contains** the Value
Not Contains	The text does **not contain** the Value
Start With	The text **starts with** the Value
End With	The text **ends with** the Value

- Date: Allows date manipulation

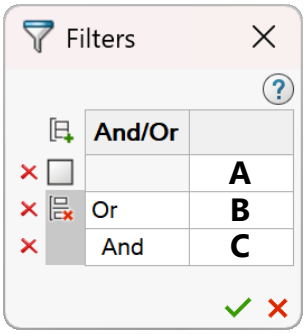
Sign	The condition is satisfied when:
=	The date is **equal** to the Value
Before the	The date is **before the** value
After the	The date is **after the** Value

- o Boolean: Allows values to be manipulated with only two possibilities

Sign	The condition is satisfied when:
=	The element is **equal** to the Value
!=	The element is **different** from the Value.

b. Groups

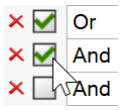
It is possible to group several lines together to create parentheses. For example, if you want to write: A or (B and C), you can use the following filter:



- Group Creation

To create a group, it is necessary to:

- Select two lines that are next to each other.



- Click on the "New Group" button.

- Group Deletion

To delete an existing group, it is necessary to:

- Click on the "Delete group" button.

c. Filter Deletion

To delete a filter, it is necessary to:

- Click on the delete button for the line you want to delete.

d. Filter list

A filter is read **from left to right**. For example, in the case of the following filter:

Property	Attribute	Condition	Value	
Body	Thickness	Between	16	20

It is written: The **Body** (the property), its **thickness** (the attribute) is **between 16 and 20 mm** (the value).

e. Validation

There are two buttons at the bottom right of the window:

- The validation button ✓ allows to close the window, save the state of the filter window, and apply the filters.

- The cancel button ✗ allows to close the window and return to the previous state of filter settings.

Consequences of filter activation

When a filter is **active**, the filter button is grayed out.

When a filter is **inactive**, the filter button is not grayed out.

- General case

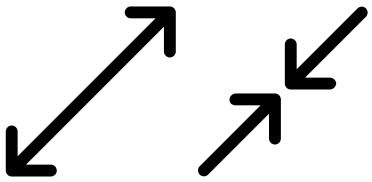
In most cases, when a filter is applied, it will display only the items affected by the filter and hide the others.

- CAM Copy Case

For CAM Copy from Panels Manager, activation has a special effect, see related chapter.

SWOOD – User Experience: Shortcuts for Expand/Collapse

The organization of SWOOD libraries within SolidWorks Task Pane, through a structure of folders and subfolders, is essential for effective component management. However, navigating complex tree structures can require numerous clicks. SWOOD 2025 simplifies this interaction by introducing keyboard shortcuts and a new context-sensitive command to quickly expand or collapse these structures. For example, a user working with a large hardware library can now, with a single action, collapse the entire tree structure to quickly access a different product category, thereby optimizing design time.



Icons

How to use

1. *Keyboard shortcuts usage*

SWOOD 2025 uses keyboard shortcuts configured in SolidWorks for tree expansion and collapse actions. These shortcuts are therefore synchronized with the user's SolidWorks settings.

a. Keyboard shortcut "Expand/Collapse"

To use the "Expand/Collapse" shortcut, it is necessary to:

- Go to the SWOOD task pane (Design or CAM).
- Go to a library.
- Select a folder (or a subfolder).

Click on the keyboard shortcut to expand/collapse (by default, the "C" key).
Using the shortcut "Expand/Collapse" will expand a collapsed folder and collapse an expanded folder.

b. Keyboard shortcut "Collapse all Items"

To use the "Collapse all Items" shortcut, it is necessary to:

- Go to the SWOOD task pane (Design or CAM).
- Go to a library.
- Click on the keyboard shortcut to collapse everything (by default, the key sequence is "Shift + C").

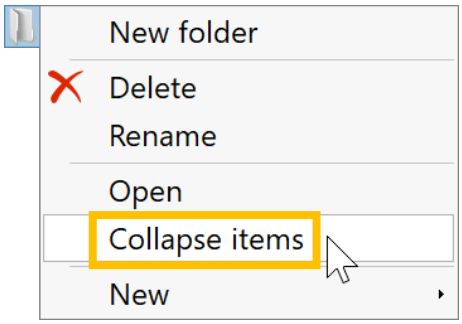
Using the "Collapse all Items" shortcut will collapse all folders and subfolders in the library at once.

2. *Using the "Collapse items" context command*

To use the "Collapse items" context command, it is necessary to:

- Go to the SWOOD task pane (Design or CAM).
- Go to a library.

- Right-click on a folder or item.
- Click on the "Collapse items" command.



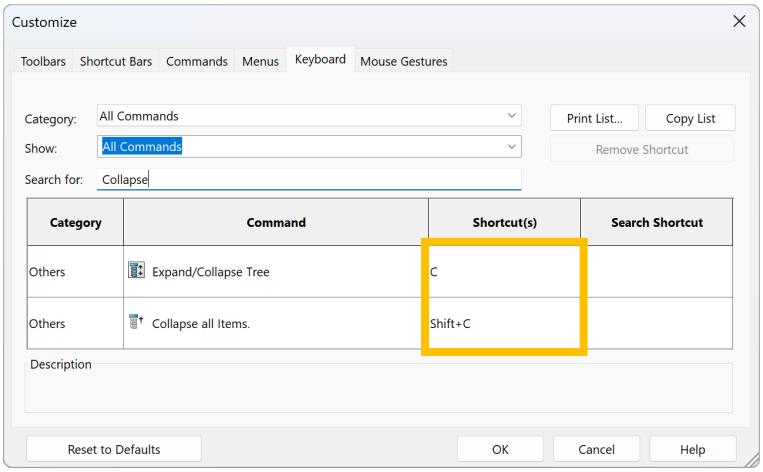
3. *Keyboard shortcut customization*

It is possible to modify the default keyboard shortcuts in SolidWorks for expanding and collapsing. To do this, it is necessary to:

- Go to the SolidWorks menu bar.
- Click on:

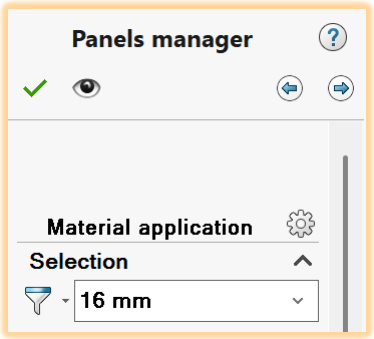
Tools → Customize…

- Go to the "Keyboard" tab.
- In the "Search for" text box, type "Collapse".
- Edit the items in the "Shortcut(s)" column.



SWOOD Design – Panels Manager: Material Application Filters

Applying materials via Panels Manager is a proven feature of SWOOD Design, facilitating the assignment of SWOOD properties to SolidWorks parts. With SWOOD 2025, this feature is even more powerful thanks to the integration of filters. Now, it is possible to refine the selection of panels, whether in the Graphics Area or in the list, based on specific criteria. This improvement is particularly beneficial for large-scale projects, where the number of panels can be high, allowing users to quickly isolate all panels of a certain thickness and specific condition to assign them a new material in a single operation.



Panels Manager PropertyManager, Materials Application tab

How to use

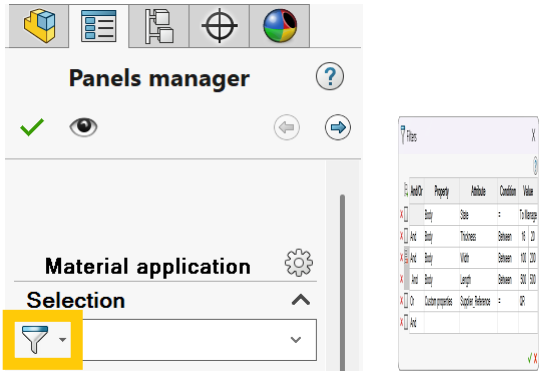
1. *Access to filters*

In order to apply a filter in the material application, it is necessary to:

- Open Panels Manager from the SolidWorks menu bar:

Tools → SWOOD Design → Panels Manager

- Go to the "Material application" tab.
- Click on the arrow next to the filter button to access the "Configure filters" button.



2. *Properties and attributes*

The properties and their attributes specific to material application in Panels Manager are as follows:

Property	Attribute	Attribute definition	Condition type
Body	State	Defines whether the part is to be processed	Boolean
	Thickness	Defines the **thickness** of the part	Range
	Width	Defines the **width** of the part	Range
	Length	Defines the **length** of the part	Range
Custom properties	*Depends on part properties*	Ø	*Depends on attributes*

SWOOD Design – Panels Manager: Inch Management

With a growing international presence, SWOOD recognizes the need to adapt to the different measurement standards used around the world. SWOOD 2025 meets this demand by integrating full support for the imperial unit system (inches) within the Panels Manager. This improvement allows users who work primarily in inches to manage their projects in a more intuitive and native way. Now, whether viewing the dimensions detected on imported parts or defining panel thicknesses, values can be displayed and entered directly in inches, simplifying the workflow and reducing the risk of errors associated with manual conversions.

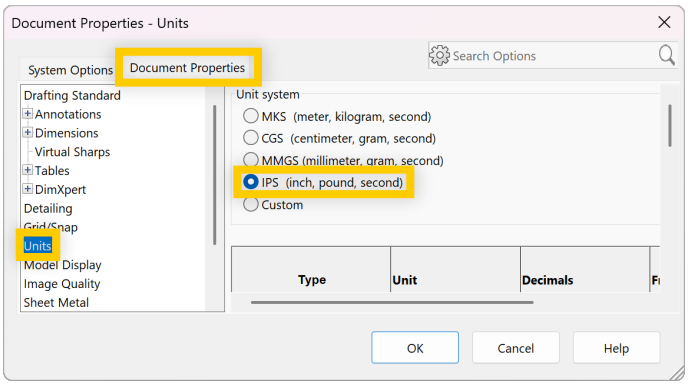
Imperial units

How to use

1. *Prerequisites: SolidWorks document configuration*

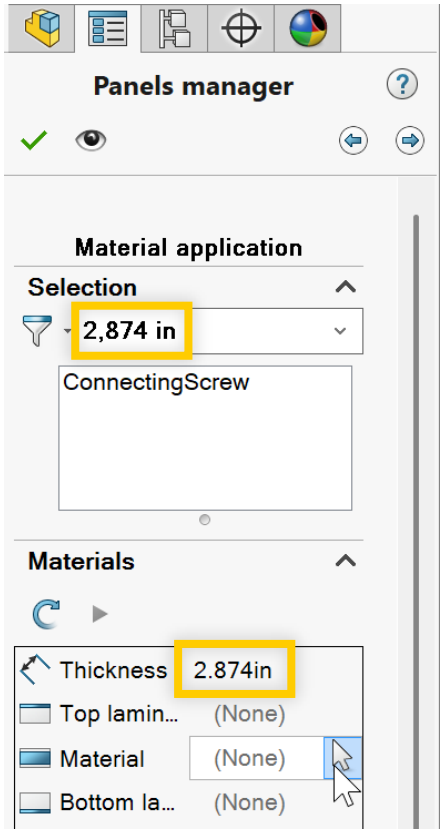
In order for SWOOD's Panels Manager to display and manage dimensions in inches, it is essential that the active SolidWorks document be configured to use the imperial unit system. To do this, it is necessary to:

- Go to Tools → Options → "Document Properties" tab → Units
- Select the "IPS" (inch, pound, second) unit system



2. *Display and use in Panels Manager*

Once the SolidWorks document has been correctly configured in imperial units, SWOOD's Panels Manager will automatically adjust its display.


Panels manager
Material application
Selection
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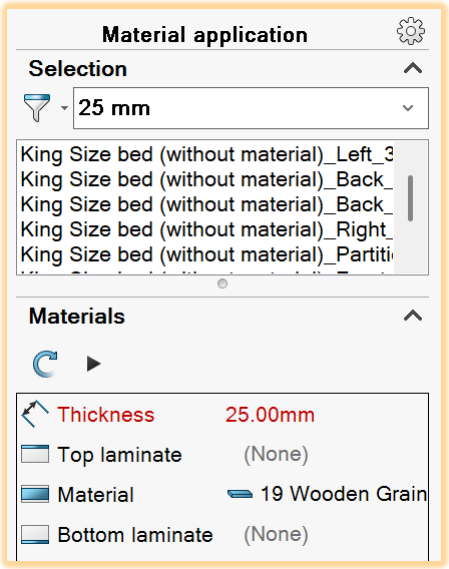
SWOOD Design - Panels Manager: Forced Material Application

Until now, applying materials via SWOOD Design's Panels Manager required a strict match between the thickness of the material and that of the SolidWorks body. SWOOD 2025 offers greater flexibility with the introduction of forced material application. This new option now allows a material to be assigned to a body even if their thicknesses do not match perfectly. To assist the user in this process and prevent errors, a color-coding system has been implemented, indicating the match between the thickness of the body and that of the selected material, in relation to a configurable thickness tolerance threshold. This can be particularly useful when resuming projects where the thicknesses of the bodies do not exactly match the material standards, or for managing minor dimensional variations.



Example of forced material application with a thickness different from that of the selected panels

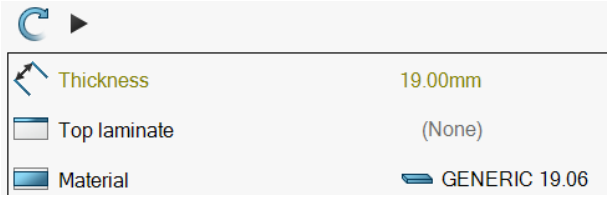
How to use

1. *Understanding thickness color indicators*

When selecting a material in the Panels Manager (Material Application tab) for a given SolidWorks body, SWOOD displays a color indicator to inform the user of the relationship between the thickness of the body and that of the selected material:

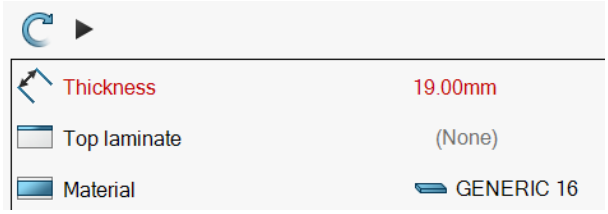
- **Black:** The thickness of the selected material is equal to the thickness of the SolidWorks body. This is the ideal case.

- **Yellow:** The thickness of the selected material differs from that of the body, but this difference is within the thickness tolerance threshold defined by the user. For example, if the body thickness is 18.5 mm, the thickness tolerance is 1 mm, and a material thickness of 18 mm or 19 mm is selected.

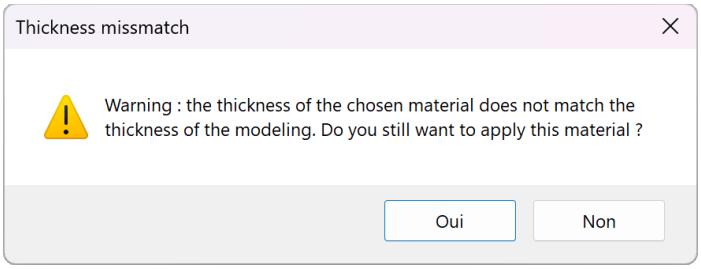


If the application of a material is forced **within the thickness tolerance limits**, no warning message will be displayed.

- **Red:** The thickness of the selected material differs from that of the body, and this difference exceeds the thickness tolerance limits. For example, if the body thickness is 22 mm, the tolerance is 1 mm, and a material thickness of 18 mm is selected.



If the application of a material is forced **beyond the thickness tolerance limits**, a warning message will be displayed.

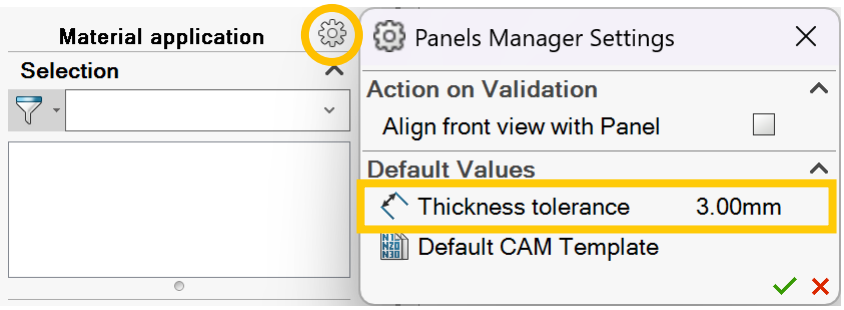


Even if the indicator is yellow or red, it is still possible to "force" the material to be applied.

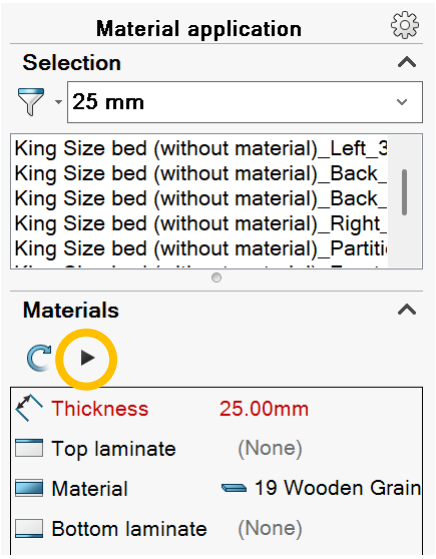
2. *Thickness tolerance configuration*

The thickness tolerance allows to define the acceptable margin for a thickness difference to be reported in yellow (tolerated but not ideal) rather than in red (strongly discouraged). To adjust this tolerance, it is necessary to:

- Open **Panels Manager**.
- Go to the **Material Application** tab.
- Click on the ⚙ Panels Manager settings button.
- Find and adjust the **thickness tolerance** setting.
- Enter the desired value (e.g., 1 mm, 0.5 mm).
- Confirm the changes.



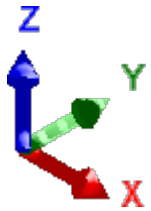
3. *Force a material to apply*
   - Select the SolidWorks body or bodies to which the material is to be applied.
   - Select the desired material from the library.
   - Click on the icon to launch the material application macro.
   - The SWOOD material will then be assigned to the body, even if its thickness does not match that of the SolidWorks model.



Important: When forcing material application with a thickness that differs from that of the SolidWorks body, the **SWOOD material thickness** will be used for all subsequent operations (bills of materials, quantity calculations, machining, etc.). The geometry of the SolidWorks body is not modified by this material application operation. It is the user's responsibility to ensure the consistency of this decision.

SWOOD Design – Panels Manager: Reorientation of SolidWorks Assembly for Front View

Consistency in the orientation of parts between the native SolidWorks environment and SWOOD conventions is crucial for the correct application of SWOOD features, including the generation of drawings. By default, SolidWorks considers its "Front Plane" to be the XY plane, while SWOOD interprets this same XY plane as the "Top Plane" (the main face on which machining operations are typically applied). Although preconfigured SWOOD document templates handle this distinction, projects imported or created without these templates may exhibit an orientation mismatch. SWOOD 2025 introduces an option in the Panel Manager to harmonize the SolidWorks front view with the orientation defined in SWOOD. This ensures better integration, predictable behavior of SWOOD functions, and, in particular, allows the standard views used in SolidWorks drawings to be correctly aligned with the panel orientations defined in SWOOD.



Reference Triad

How to use

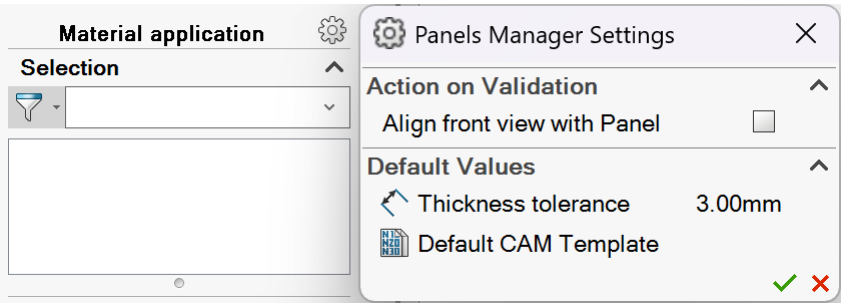
1. *Understanding the need for reorientation*

When working with imported parts or assemblies, or SolidWorks files that do not originate from SWOOD models, the orientation perceived by SWOOD for the application of its features (such as the definition of reference faces for materials, edgebands, or machining operations) may not correspond to the standard SolidWorks "Front View." This new feature aims to realign the SolidWorks front view to match the face that SWOOD considers to be the main face after any reorientation via the Panels Manager.

2. *Enable front view alignment*

To activate this automatic redirection option, it is necessary to:

- Open **Panels Manager**.
- Click on the Panels Manager Settings button.
- In the settings window, check the box labeled "**Align front view with Panel**".
- Confirm the changes.



3. *Operation when closing Panels Manager*

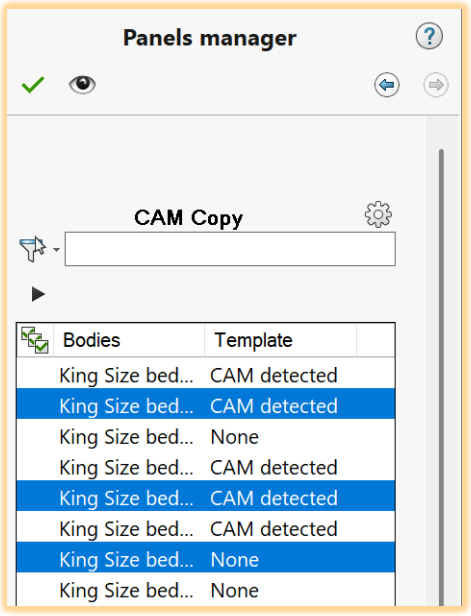
Once this option is enabled:

- After completing the operations in Panels Manager (applying materials, reorienting the main faces of the panels if necessary, etc.) and when closing the Panels Manager interface.
- SWOOD will automatically identify panels whose SolidWorks front view needs to be updated to match the orientation defined in SWOOD.
- These panels will then be processed so that their SolidWorks "Front View" is realigned.

Warning: This realignment process requires SWOOD to temporarily open each affected part file in the background to modify its standard view orientation. On assemblies with a large number of parts to be reoriented, this operation may take some time. It is therefore important to be aware of this, especially before closing the Panels Manager on large projects.

SWOOD Design - Panels Manager: CAM Copy

SWOOD 2025 reinforces the link between design and manufacturing by integrating a key feature of SWOOD CAM directly into SWOOD Design's Panels Manager: CAM Copy. Previously available as a separate tool, this new tab allows users with both SWOOD Design and SWOOD CAM licenses to quickly and efficiently apply predefined machining models and strategies to SolidWorks parts or assemblies. This integration aims to streamline the CAM preparation process by providing a unified interface for identifying parts requiring machining and assigning them the appropriate operations, while taking advantage of powerful filters for targeted selections.



CAM Copy tab in Panels Manager

How to use

1. *Prerequisites and access to the tab*

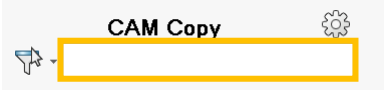
- **Required licenses:** To use this feature, it is necessary to have an active license for SWOOD Design and SWOOD CAM.
- **Access:** In Panels Manager, a new tab called "CAM Copy" (or similar wording) is now available.
- **Part visibility**: Only parts to which a SWOOD material has already been applied via the "Material Application" tab in the Panels Manager will be listed and available for CAM Copy.

2. *Panel selection*

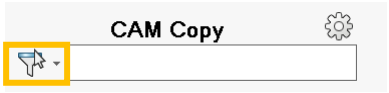
The "CAM Copy" tab offers several methods for selecting the panels to which the milling strategy is to be applied or modified:

- **Direct selection:** Click on panels directly in the displayed list or in the SolidWorks 3D Graphics Area.

- **Search by name:** Use the search bar provided to filter and select panels based on their name.



- **Filters:** Access filters (via the usual filter icon) to make advanced selections based on various part properties (thickness, material, custom properties, etc.), in the same way as for material application.



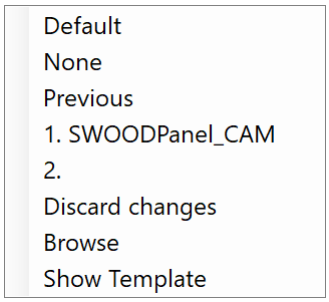
- **Select/Deselect All:** A button is available to quickly select or deselect all listed panels.

3. *CAM template application*

Once the panels have been selected, a CAM template can be assigned to them:

- **Right-click** on one of the panel lines selected in the list.
- OR click on the **dedicated icon** (located at the end of the line) for the selected panel.

A pop-up menu will open, offering the following options for the selected panels:

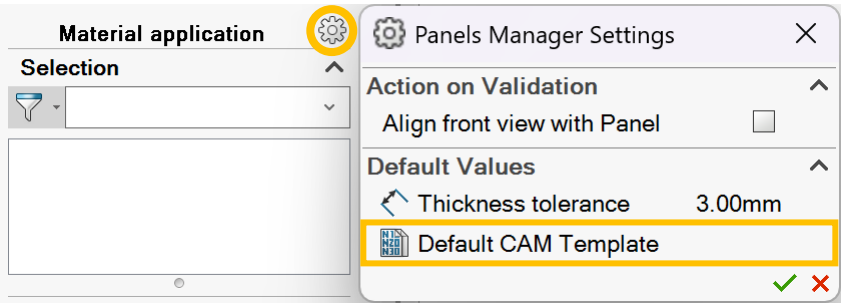


- **Default:** Applies the CAM template that has been defined as default in the Panels Manager settings.
- **None:** Does not apply any machining patterns. If a pattern was previously applied to these panels, it will be removed.
- **Previous:** Reapplies the last CAM template that was specifically selected for this panel (this option is more relevant for a single selection).
- **1. or 2.:** Applies one of the last two CAM templates used globally in this session of Panels Manager.
- **Discard changes:** Returns to the CAM template that was applied to the selected panels when the "CAM Copy" tab was opened.
- **Browse:** Opens a standard file explorer allowing the user to browse and select a SolidWorks part with a SWOOD CAM CAM template to apply.
- **Show Template:** Provides a preview of the CAM template that would be applied. Note: This option works best when only one panel is selected (usually the last one clicked).

4. *Default CAM Template definition*

To configure the CAM template that will be applied when selecting the "Default" option:

- Click on the Panels Manager settings button.
- Navigate to the corresponding section and select or specify the path to the default CAM template.
- Confirm the changes.



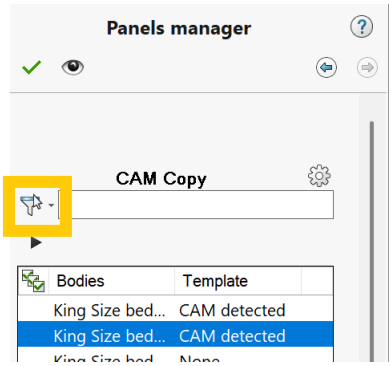
5. *Filter specifications*

a. Filters access

The specific filters for CAM Copy are accessible from the SolidWorks menu bar:

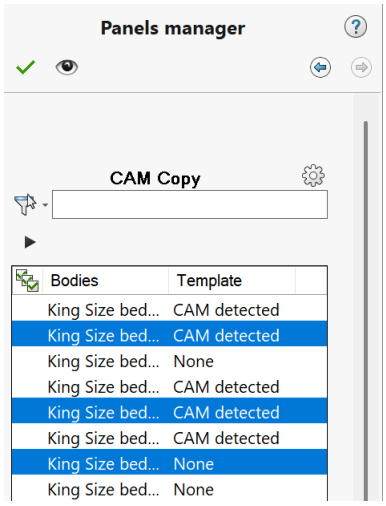
Tools → SWOOD Design → Panels Manager

Then go to the "CAM Copy" tab.



b. Consequences of activating a filter

In the case of the " CAM Copy" filter, the filter will only select the panels affected by the application of the filter, in order to apply machining data.



c. Properties and attributes

The properties and their attributes specific to CAM Copy in Panels Manager are as follows:

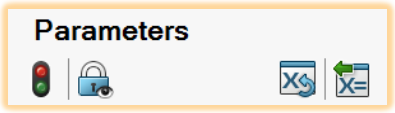
Property	Attribute	Attribute definition	Type of condition
Custom properties	*Depends on part properties*	Ø	*Depends on attributes*
Materials	Name	Allows to select a material from the materials library	Boolean
Template	Ø	Allows selection of a CAM template from among those used	List of templates

SWOOD Design - SWOODBox: Default Parameters

In SWOOD Design 2025, the SWOODBox Test Mode has been enhanced with two ingenious buttons for more intuitive configuration.

- **« Restore default values »** instantly resets all parameters to their default settings, ideal for risk-free testing and returning to the original configuration.
- **« Set all by default »** allows to save the values modified in Test Mode as new default references, thereby simplifying the optimization of SWOODBox.

With these tools, quickly adjust the configurations and increase efficiency in the design of furniture.



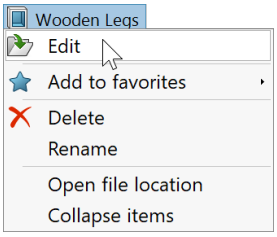
Parameters toolbar with two default parameter buttons

How to use

1. *Usage*

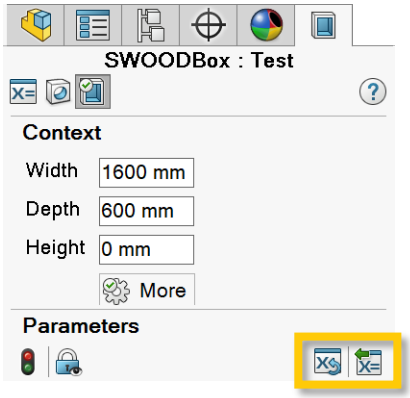
To access the default parameter settings for SWOODBoxes, a SWOODBox must be edited. To do this, it is necessary to:

- Go to the SWOOD library .
- Go to the SWOODBox library .
- Right-click on a SWOODBox, then click on "Edit."



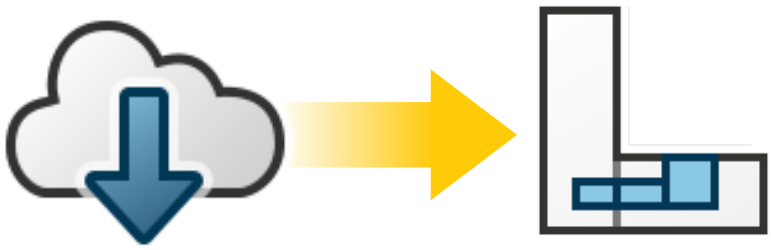
- Go to the SWOODBox PropertyManager .
- Go to the "SWOODBox: Test Mode" tab ».

In the toolbar of the "Parameters" section, there are two new buttons:



SWOOD Design - Library Installer: Connector Integration

Originally, the Library Installer was designed to facilitate the download of complex SWOODBoxes offered by various suppliers, including official hardware manufacturers. Given its growing popularity among users, it continues to evolve to further simplify access to the essential elements of SWOOD Design. With SWOOD 2025, the Library Installer has taken a new step forward by integrating **Connectors**, allowing you to directly download precise models of assembly systems provided by leading brands such as Lamello, Festool, Fastenlink, Ovvo, Lockdowel, and many others. There is no longer any need to recreate these components manually: in just a few clicks, they are ready to be used in your designs. This saves considerable time for faster, more accurate assemblies that comply with manufacturers' recommendations.



Integrating connectors into the Library Installer

How to use

1. *Access to the Library Installer*

There are many ways to access the Library Installer:

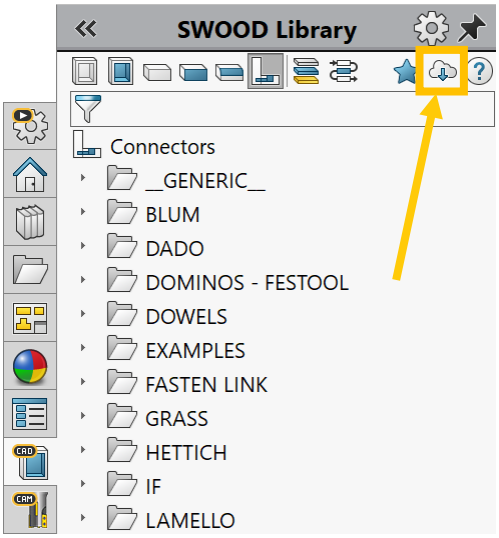
a. From the SolidWorks menu bar

The Library Installer command can be accessed from the SolidWorks menu bar:

Tools → SWOOD Design → Connect → Library Installer

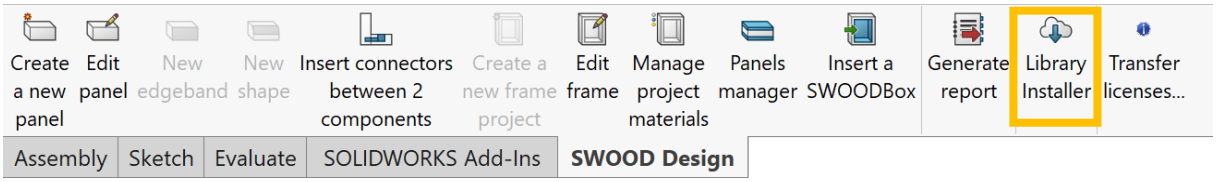
b. From the SWOOD Design task pane

In the SWOOD Design task pane, in the toolbar, there is a shortcut to the Library Installer.



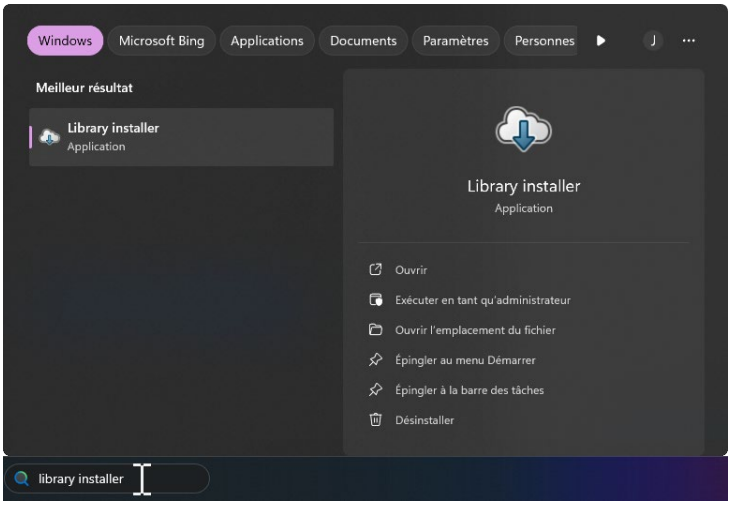
c. From the SWOOD Design order manager

When a file is opened in SolidWorks (a part or assembly), it is possible to access the Library Installer command using the new SWOOD Design 2025 command manager.



d. From Windows Search

In the Windows search bar, simply type "Library Installer."

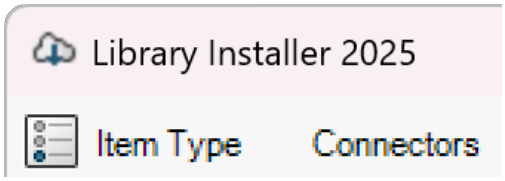


2. *Download a connector*

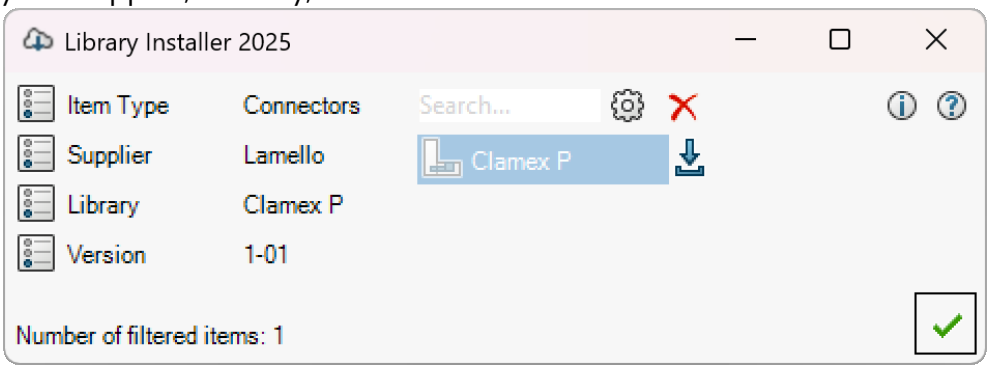
a. Download from the Library Installer

Once the Library Installer window is open, to download a connector, it is necessary:

- Select "Connector" as the type of hardware.



- • Select your "Supplier," "Library," and "Version".

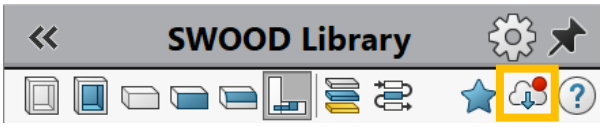


- In the list of available connectors, click on the download icon.
- As long as the connector icon is greyed out with a timer, the download is in progress.
- When the download is complete, the Library Installer window can be closed.

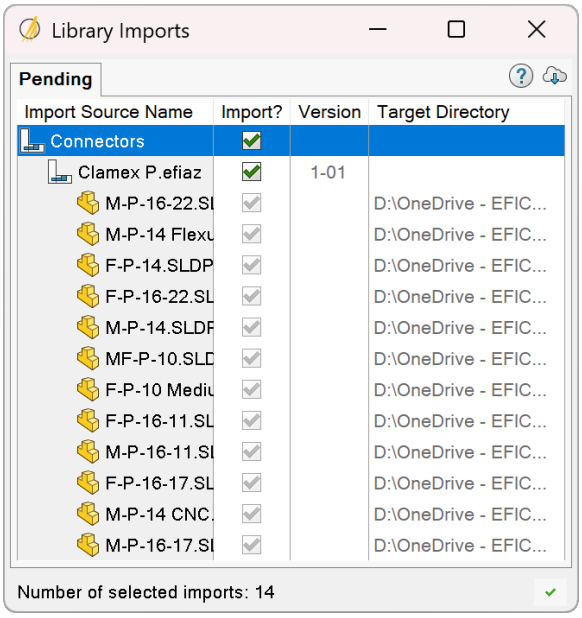
b. Import to library

Unlike SWOODBoxes, connectors are not imported directly into the connector library, but via an import interface.

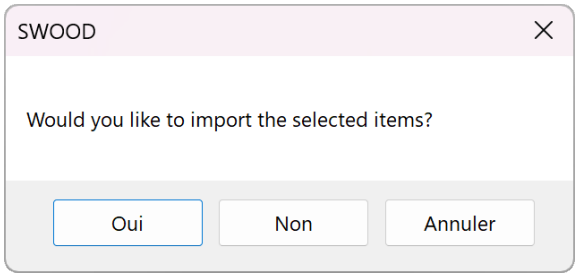
If one or more elements are waiting to be imported, the Library Installer icon in SWOOD Design's task pane looks different.



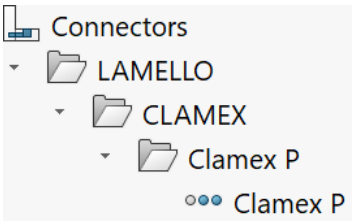
Clicking on this icon opens the import window.



- Click on the validation arrow .
- Validate the confirmation window.

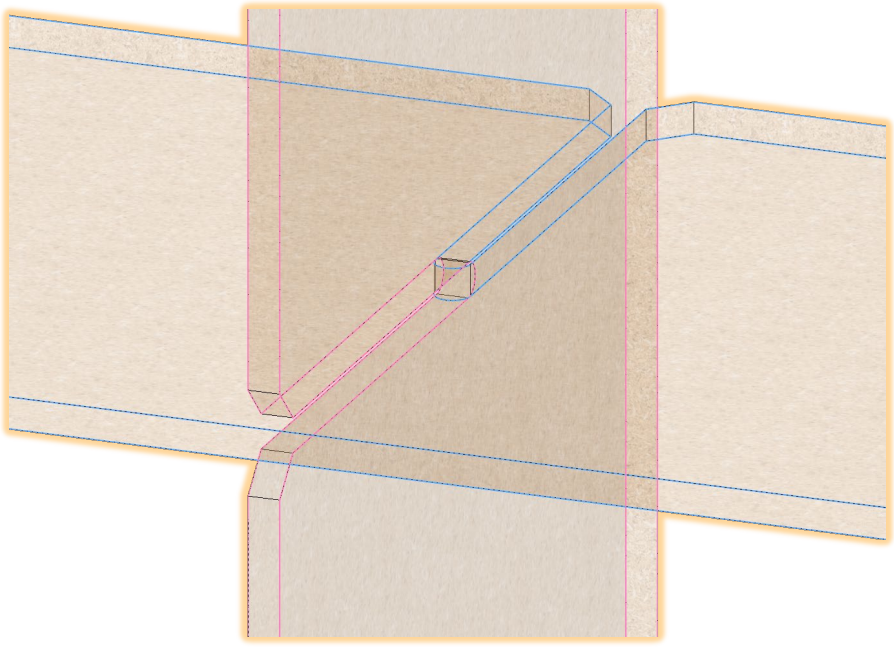


It is possible to check that the connector has been imported into the Connectors library.



SWOOD Design - Panel Joint: Notch Flaring

The introduction of panel joints in SWOOD Design 2024 enabled the automated creation of complex joints, such as mortise-and-tenon or halved joints, by volume interference detection. SWOOD Design 2025 enhances this functionality with the introduction of notch flaring, specifically for halved joints. This new tool makes it possible to add a bevel, whose angle and depth can be parameterized, directly to notches. The aim of this enhancement is to facilitate panel insertion during assembly, particularly when fits are tight.



Example of halved joint with notch flaring

How to use

1. *Notch flaring parameters*

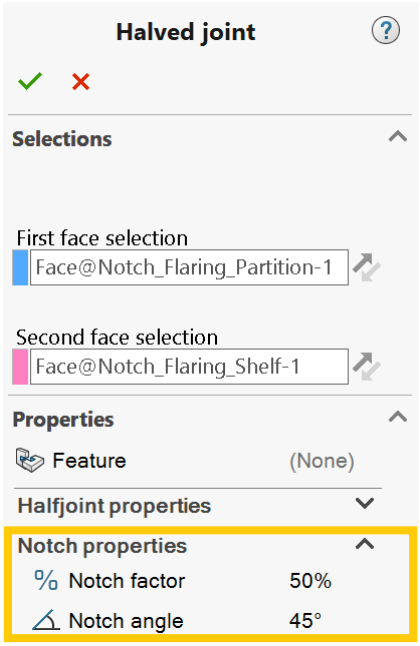
To create and set the notch flaring, first create a halved joint panel assembly:

- In the SolidWorks menu bar, go to:

Tools → SWOOD Design → Panel Joint → Create halved Joint

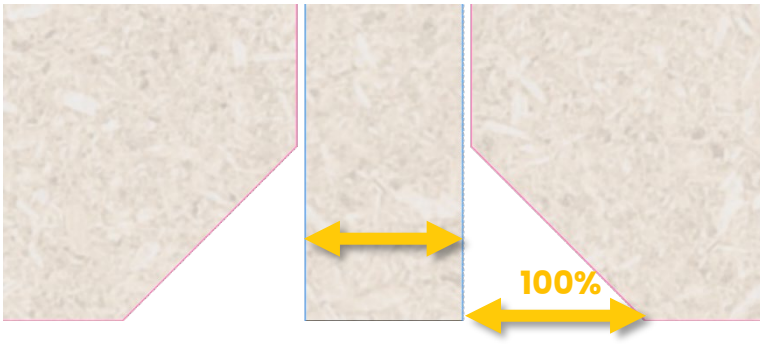
- Select the two faces of the panels to be joined.

Next, you can access the notch parameters by going to the "Notch properties" section.



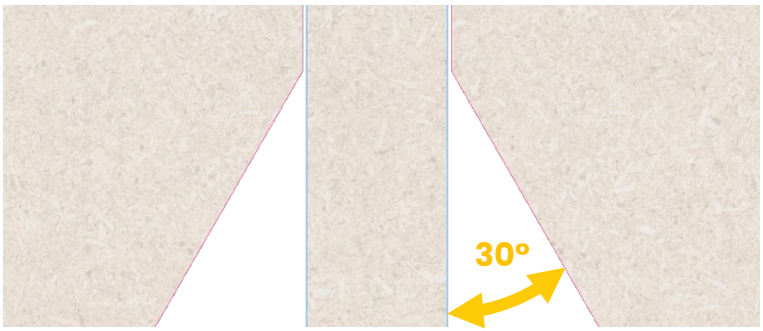
a. Notch factor

The notch factor represents the size of the notch in relation to the size of the panel to be joined.
In the example below, the notch factor is equal to 100%, i.e. the thickness of the blue panel.



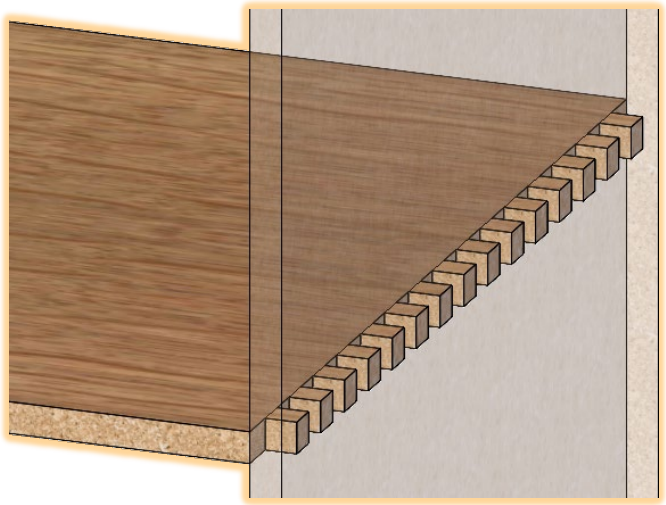
b. Notch angle

The notch angle represents the angle the notch will take.
In the example below, the notch angle is 30°.



SWOOD Design - Panel Joint: Mortise and Tenon Repetition

SWOOD Design 2025 extends the capabilities of mortise and tenon joints by introducing an automatic repetition feature. This new feature allows mortise and tenon joints to be duplicated in a variety of configuration modes. Users can define repetitions based on a fixed pitch and number, or opt for automatic distribution according to a given number. Some repetition options also allow dynamic adaptation to panel dimensions or specific limits. The objective is to offer a configurable method for rapidly generating series of mortise and tenon joints, thus optimizing the design process.



Example of mortise and tenon repetition to form a set of straight shanks

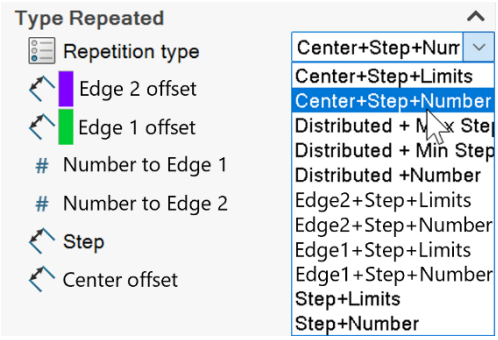
How to use

1. *Repetition creation*

To make a mortise and tenon repetition, it is necessary to:

- Open a SolidWorks assembly (a Frame, for example)
- In the SolidWorks menu bar, click on :

  Tools → SWOOD Design → Panel joint → Create Mortise and Tenon
- Select the two faces involved in this mortise and tenon joint. First select the face of the mortise panel (in blue), then the face of the tenon panel (in pink).
- • In the "Repetition Type" section, select the type of mortise and tenon repetition.



2. *Repetition Types*

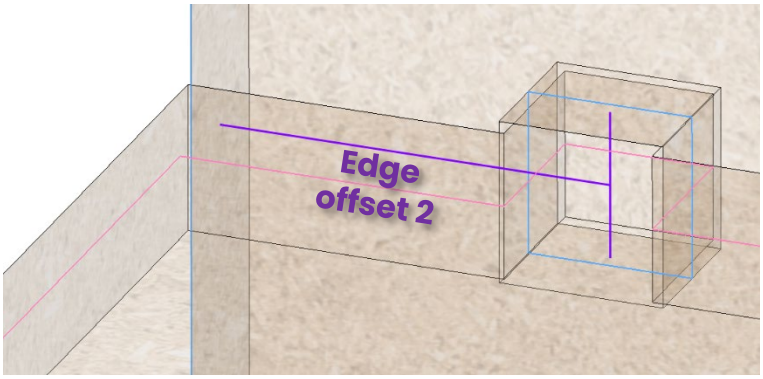
There are 11 different repetition types (the same repetition types as for "Component Repetition" of connectors).

a. Parameters description

Depending on the repetition type selected, certain parameters are visible. Here is a description of them:

- Edge offset 2

This parameter defines the **offset of edge 2 of the interference** in **distance**.



Edge 2 offset

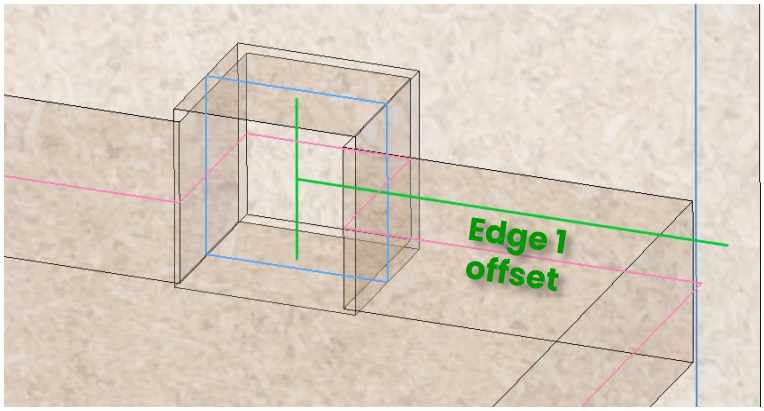
This parameter can be located in the Graphics Area using the "Selected object 3" color.

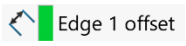
To change the color of this parameter, go to the SolidWorks menu bar and click on :

Tools → Options... → in the "System Options" tab → in the "Colors" category

- Edge offset 1

This parameter defines the **offset of intersection edge 1** in **distance**.





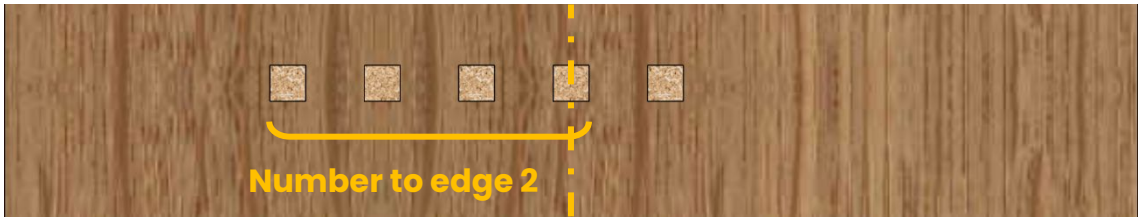
This parameter can be located in the Graphics Area using the "Selected object 4" color.

To change the color of this parameter, go to the SolidWorks menu bar and click on :

Tools → Options... → in the "System Options" tab → in the "Colors" category

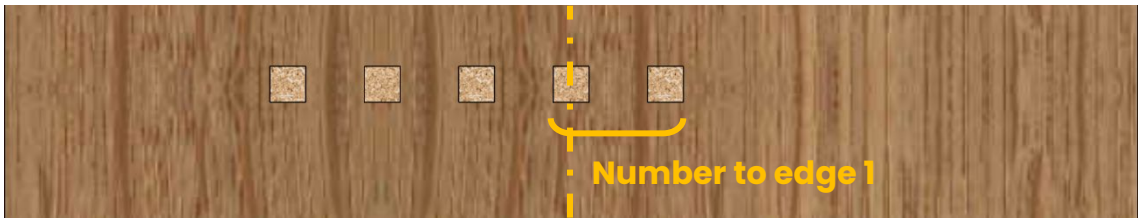
- # Number to edge 2

This parameter defines the **number of elements to edge 2** (including the center element).



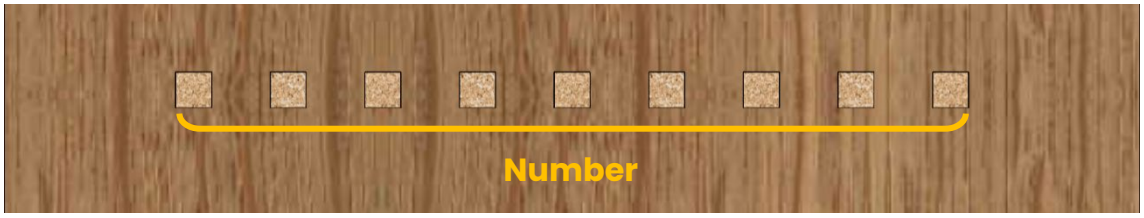
- # Number to edge 1

This parameter defines the **number of elements to edge 1** (including the center element).



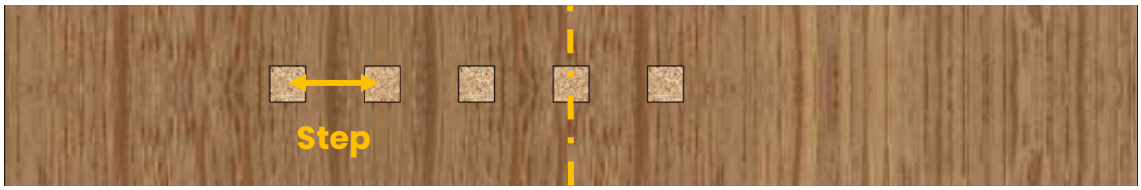
- # Number

This parameter defines the **total number of elements** in the repetition.



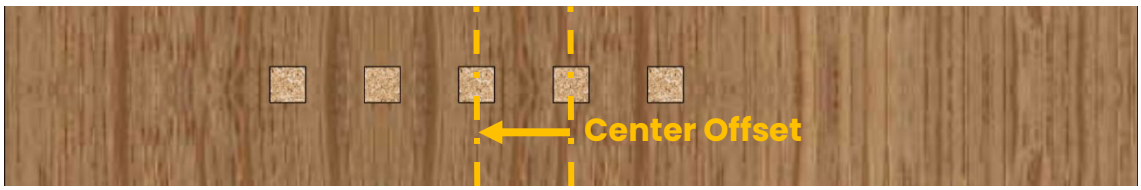
- Step

This parameter defines the **distance between two elements** (center distance).



- Center Offset

The center offset defines the **distance of the center offset**.



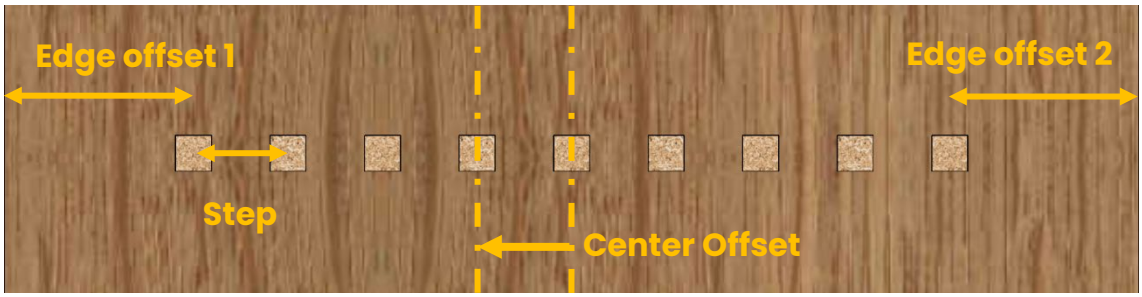
b. Description of repetition types

Mortise and tenon repetition must be adapted to the dimensions of the panels and joints themselves, which is why there are so many repetition types.

They work as follows:

- Center + Step + Limits

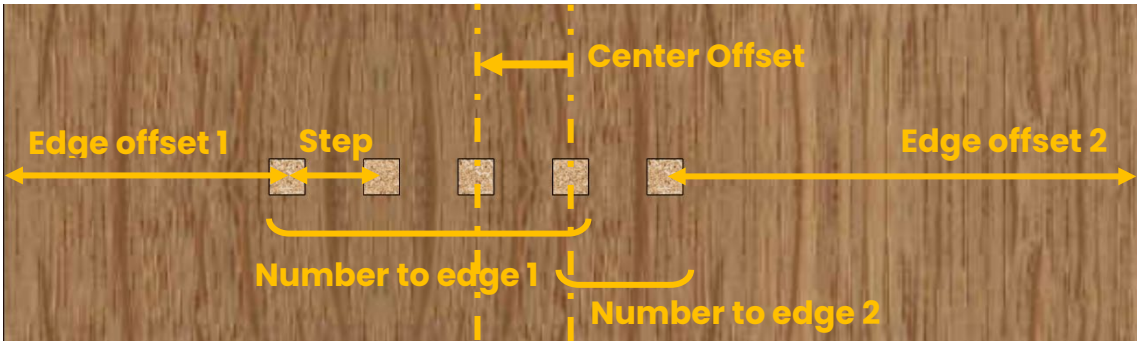
The "**Center + Step + Limits**" repetition type will distribute the mortise and tenon joints according to its center, which may have been offset by the **center offset**, according to the **step** and the **offsets of edge 2 and edge 1**.



- Center + Step + Number

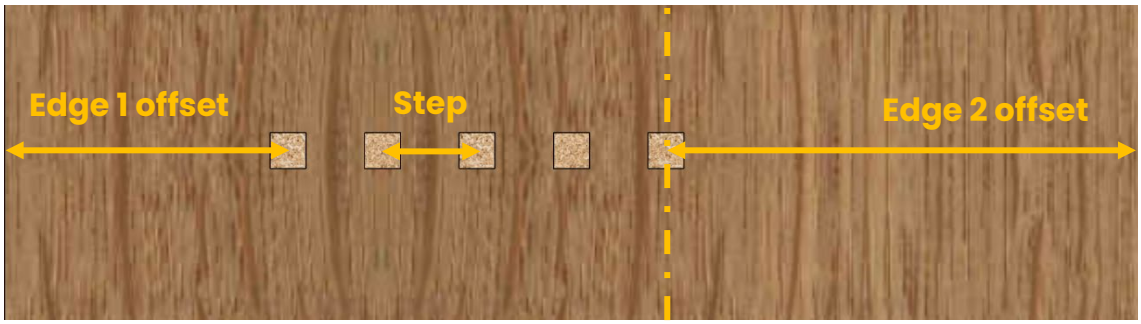
The "**Center + Step + Number**" repetition type, in addition to the "Center + Step + Limits" parameters, takes into account the number of repetitions in both directions.

Warning: the number to edge 1 and 2 take into account the center mortise and tenon.



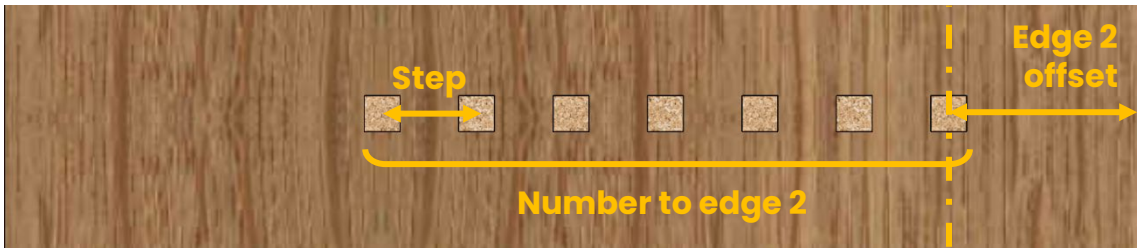
- Edge 2 + Step + Limits

The "**Edge 2 + Step + Limits**" repetition type will distribute the mortise and tenon joints from the **edge 2 offset**, according to the **step**, and will stop at the **edge 1 offset**.



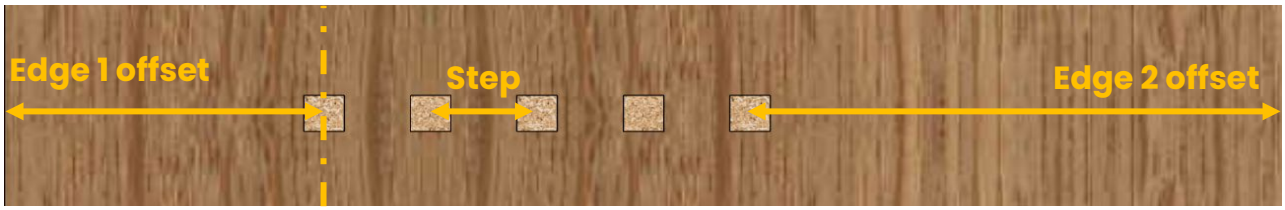
- Edge 2 + Step + Number

The "**Edge 2 + Step + Number**" repetition type will distribute the mortise and tenon joints starting from the **offset of edge 2**, according to the **step**, and will stop once the **number to edge 2** has been reached.



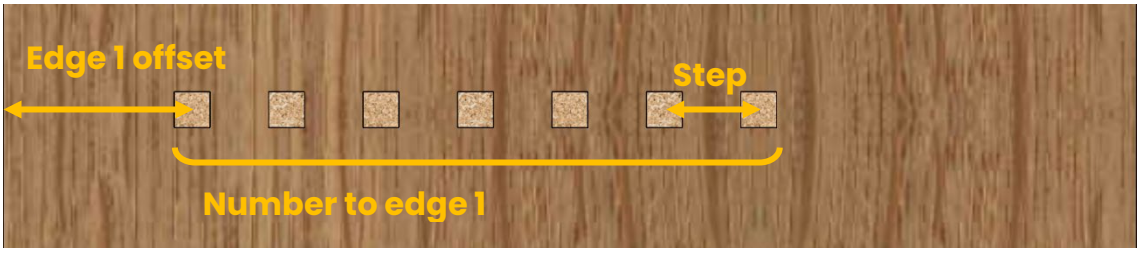
- Edge 1 + Step + Limits

The "**Edge 1 + Step + Limits**" repetition type will distribute the mortise and tenon joints from the **edge 1 offset**, according to the **step**, and will stop at the **edge 2 offset**.



- Edge 1 + Step + Number

The "**Edge 1 + Step + Number**" repetition type will distribute the mortise and tenon joints starting from the **edge 1 offset**, according to the **step**, and will stop once the **number to edge 1** has been reached.

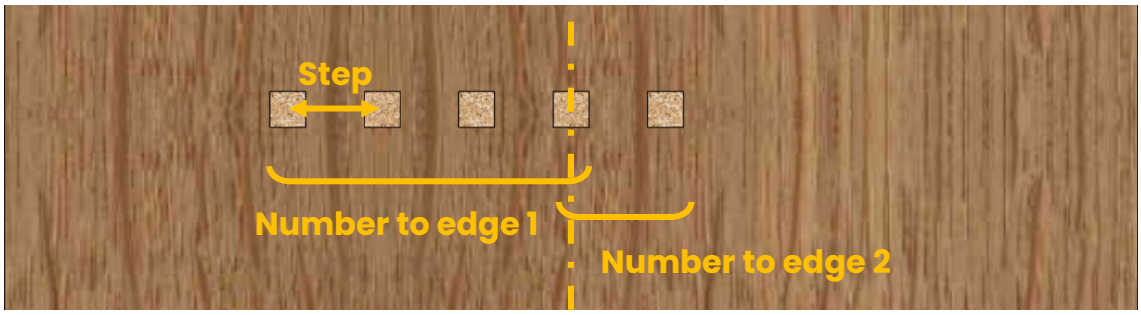


- Steps + Limits

The "**Step + Number**" repetition type will distribute the mortise and tenon joints by a defined **step**, with the **edge 2 and edge 1 offsets** as limits.

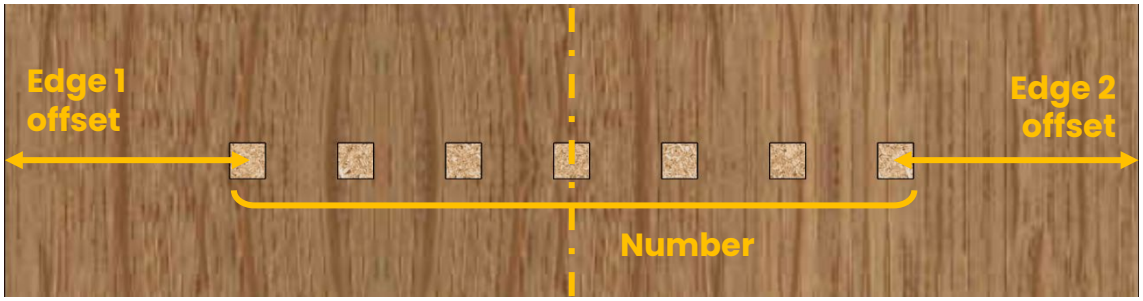
- Step + Number

The "**step + number**" repetition type will distribute the mortise and tenon joints according to **step**, **number to edge 2 and to edge 1**.



- Distributed + Number

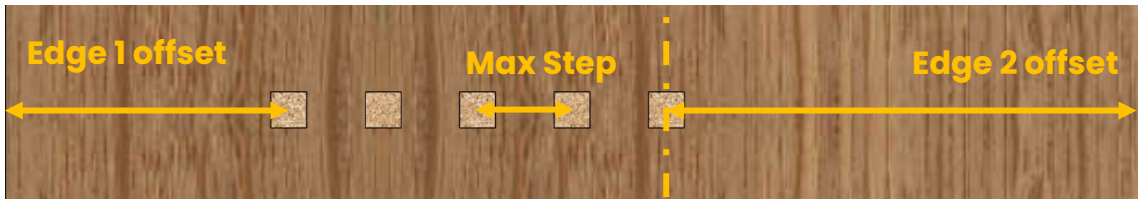
The "**Distributed + Number**" repetition type will distribute mortise and tenon joints according to **number**, with **edge 2 and edge 1 offset** as limits.



- Distributed + Max Step

The "**Distributed + Max Step**" repetition type repeats mortise and tenon joints from **edge offset 2 to edge offset 1**, without exceeding the **maximum step size**.

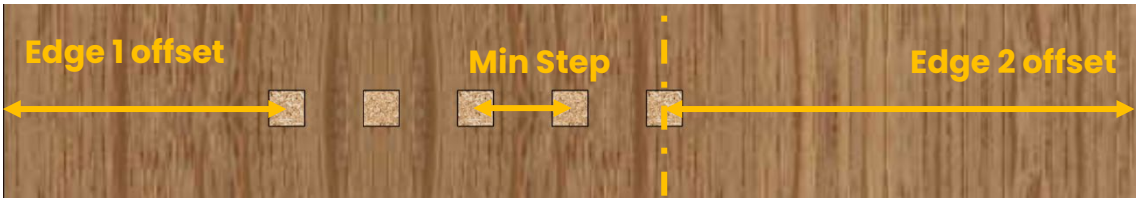
Warning: The actual step may be smaller than the input step.



- Distributed + Min Step

The "**Distributed + Min Step**" repetition type repeats mortise and tenon joints from **edge 2 offset to edge 1 offset**, without going below the **minimum step**.

Warning: The actual step may be greater than the input step.



c. Parameter availability

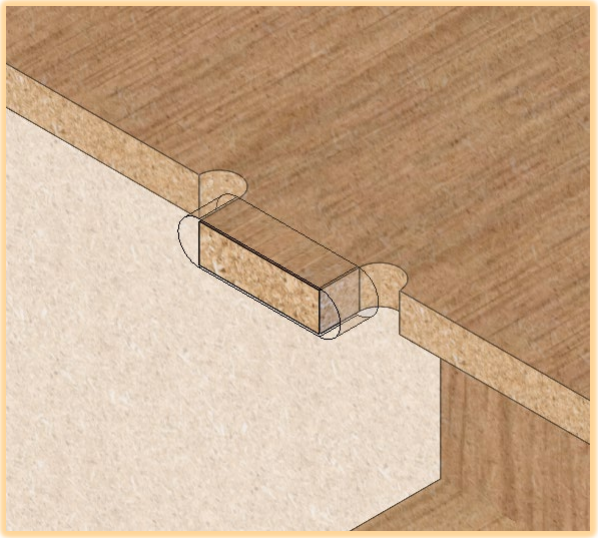
Depending on the selected repetition type, some parameters are available and others are not.

Parameter availability is as follows:

Repetition type	Parameters						
	Edge 2 offset	Edge 1 offset	Number to edge 2	Number to edge 1	Number	Step	Center offset
Center + Step + Limits	Yes	Yes	No	No	No	Yes	Yes
Center + Step + Number	Yes	Yes	Yes	Yes	No	Yes	Yes
Edge 2 + Step + Limits	Yes	Yes	No	No	No	Yes	No
Edge 2 + Step + Number	Yes	No	Yes	No	No	Yes	No
Edge 1 + Step + Limits	Yes	Yes	No	No	No	Yes	No
Edge 1 + Step + Number	No	Yes	No	Yes	No	Yes	No
Step + Limits	Yes	Yes	No	No	No	Yes	No
Step + Number	No	No	Yes	Yes	No	Yes	No
Distributed + Number	Yes	Yes	No	No	Yes	No	No
Distributed + Max Step	Yes	Yes	No	No	No	Yes	No
Distributed + Min Step	Yes	Yes	No	No	No	Yes	No

SWOOD Design - Panel Joint: New Mortise and Tenon Shapes

The introduction of mortise and tenon joints in SWOOD Design 2024 opened up new possibilities for connecting wood panels. SWOOD Design 2025 expands on this functionality by integrating three new mortise and tenon shapes. These geometries have been specifically designed for machining on 3-axis machines and are intended to be applied simultaneously to the tenon and mortise. The aim of these new shapes is to minimize the need for manual rework after machining, enabling assemblies to be produced that are suitable for 3-axis equipment and ready for assembly.



Example of a mortise and tenon shape

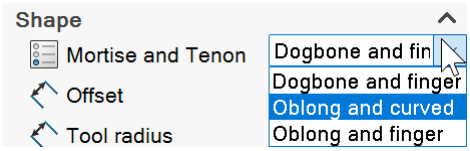
• How to use

1. *Shape selection*

To choose the mortise and tenon shape, it is necessary to:

- Open a SolidWorks assembly (a Frame, for example)
- In the SolidWorks menu bar, click on :

  Tools → SWOOD Design → Panel joint → Create Mortise and Tenon
- Select the two faces concerned by this mortise and tenon joint. First select the mortise panel face (blue), then the tenon panel face (pink).
- In the "Profile" section, select the mortise and tenon shape.



2. *Types of shapes*

a. Tenon shapes

The types of **tenon** shapes are as follows:

- Finger

The "**Finger**" tenon shape is a shape that highlights the entire tenon and mills the panel a little deeper to clear its inner angle. This shape is useful when the inner angle needs to be marked.

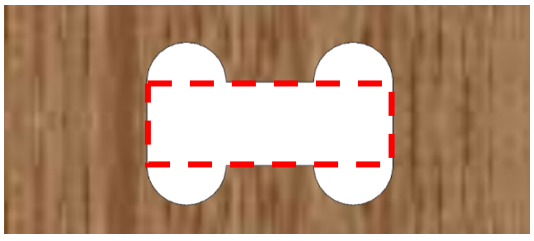
- Curved

The "**Curved**" tenon shape is a shape that machines the tenon, leaving a rounded edge on its inner angle. This shape involves leaving a margin on the mortise to allow for the rounded edge.

b. Mortise shapes

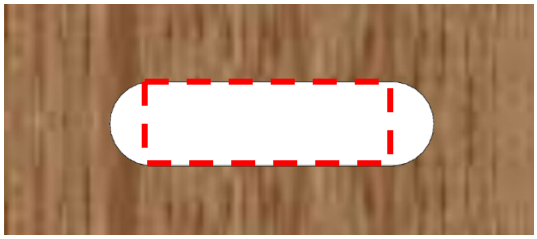
- Dogbone

The "**Dogbone**" mortise shape is a shape that reveals its inner angle by machining the sides a little more.



- Oblong

The "**Oblong**" mortise shape is a shape that reveals its inner angle by machining the length a little more.

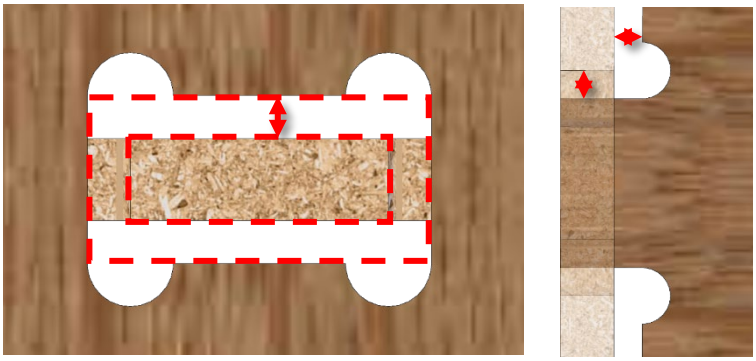


3. *Shape settings*

Regardless of the shape selected, several parameters must be defined to adjust the shape of the mortise and tenon joints.

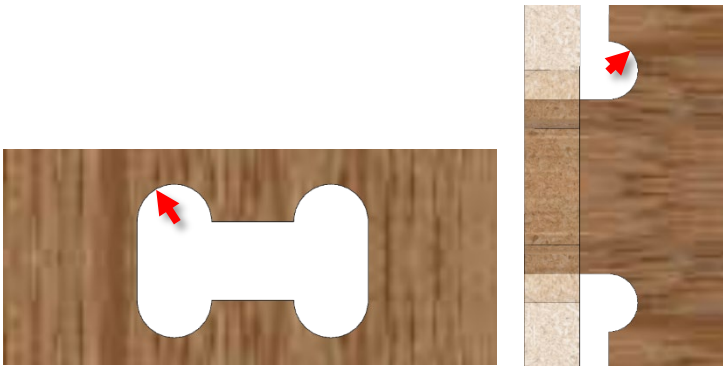
a. Offset

Play between the two panels, both around the joint and between the two panels.



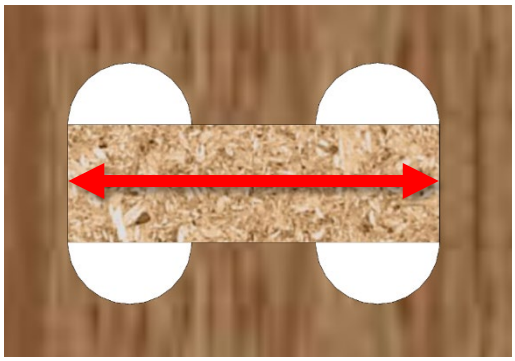
b. Tool radius

Radius of the arcs of circles for tenon and mortise machining.



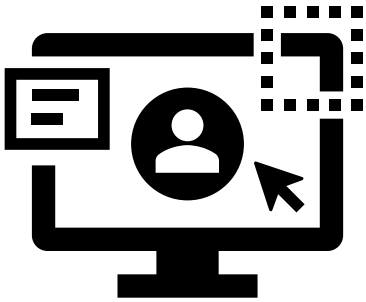
c. Mortise length

Length of the mortise, without taking into account the offset parameter.



SWOOD Design – Panel Joint: Interface Enhancements

In addition to new features dedicated to panel assemblies, SWOOD Design 2025 includes several interface improvements aimed at optimizing ergonomics and efficiency when creating and managing these joints. These adjustments are designed to simplify users' daily workflow.

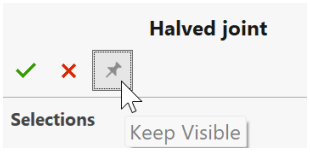


Interface

How to use

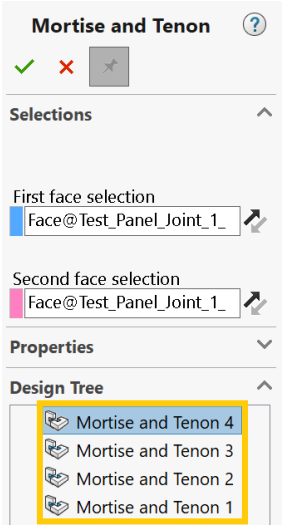
1. *Pin icon*

A new pin icon has been added to the joint creation window. When activated, this icon keeps the creation window open after a joint has been validated, allowing you to create several consecutive joints without having to reopen the function.



2. *Panel Joint history*

A history of the panel joints created is now visible in the interface. This history, often presented in tree form at the bottom of the function window, facilitates the monitoring and management of joints already implemented in the project.

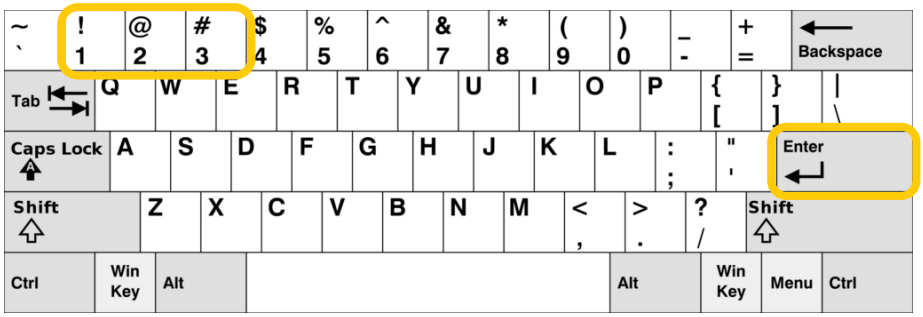


3. *Keyboard shortcuts*

New keyboard shortcuts have been added to speed up certain common actions:

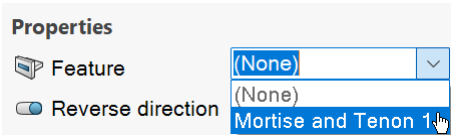
Changing the joint type: The numeric keys **1**, **2**, **3** can be used to quickly switch between the different joint types available.

Validation: The "**Enter**" key on the keyboard can now be used to validate the creation of the current assembly.



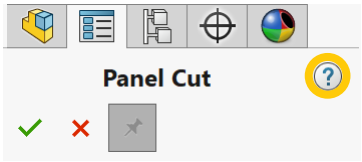
4. *Visualization of parent-child relationships*

The interface provides a better visualization of the dependency relationships (parent-child) between the various joints functions. This helps to understand the structure and links between the joints created.



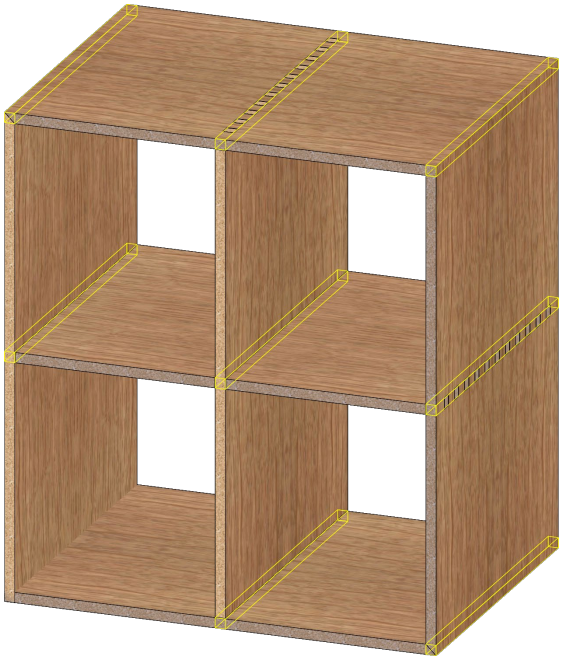
5. *Contextual help button*

A help button has been added, providing direct access to documentation specific to the panel joint function currently in use.



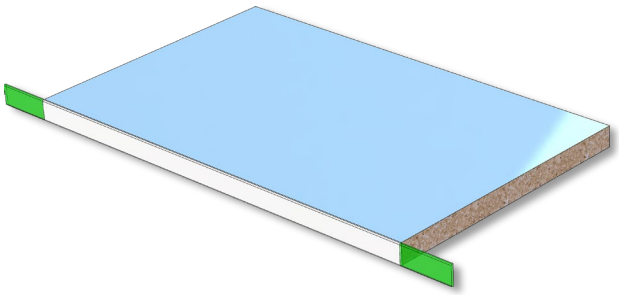
6. *Highlighting existing interferences and joints*

When creating a new joint, the interface highlights potential interferences or existing joints. This highlighting (e.g., in yellow) allows user to quickly identify areas that have already been processed or potential conflicts.



SWOOD Design – Edgeband Overlength

Applying edgebands is a crucial step in panel finishing. SWOOD Design 2025 introduces edgeband overlength management, a feature designed to improve the accuracy of material estimates and anticipate manufacturing needs. Configurable at the edgeband library level or directly during application, this overlength is then taken into account in reports. It allows for a more realistic estimate of the quantities of edgeband required, including a margin for cuts. For example, a workshop can ensure that it orders the right amount of edgeband for a batch of fronts by allowing for the additional material that will be removed during final trimming.



Example of a panel with a visual representation of edgeband overlengths

How to use

1. *Overlength configuration*

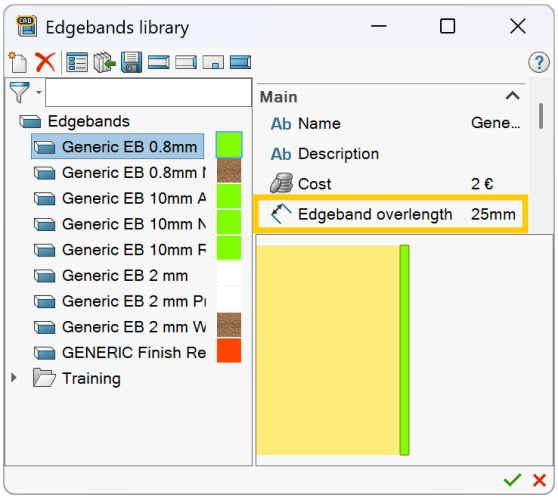
The edgeband overlength can be configured at two levels:

a. **From the edgeband library**

It is possible to define a default overlength value directly in the properties of an edgeband within the SWOOD Design library. This value will then be automatically applied each time this specific edgeband is used.

To change this value, go to:

- The SWOOD Design task pane.
- The Edgebands library.
- Select an edgeband.
- In the general settings.

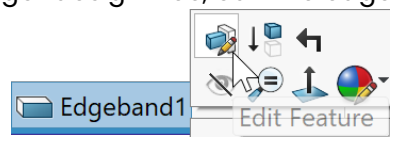


b. **When applying via the edgeband function**

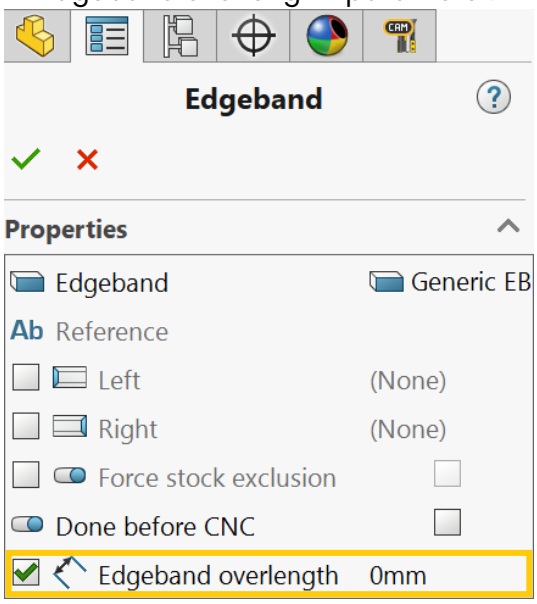
When applying an edgeband to a panel using the dedicated function in SWOOD Design, the user can specify or modify the overlength for that particular application, overriding the default value in the library if necessary.

To change this value in the feature, it is necessary to:

- Apply an edgeband to a panel.
- In the panel's FeatureManager design tree, edit the edgeband.



- Enter a value in the "Edgeband overlength" parameter.

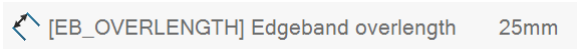


2. *Inclusion in the report*

The configured overlength information is then integrated and taken into account when generating reports.

a. Overlength parameter

The parameter **[EB_OVERLENGTH]** represents the edgeband overlength in the standard report.



b. Information in the system report

- Parameter name

In the system report, the name of the parameter is called "**Overlength**".

- On the Summary page

On the "Summary" page, in the "Edgebands" group, the length displayed is the total length of the edgebands **WITH** the total overlength. This is therefore taken into account in the cost calculation.

- On the Edgebands page

However, on the "Edgebands" page, the length displayed is only the total length of the edgebands **WITHOUT** the total overlength.

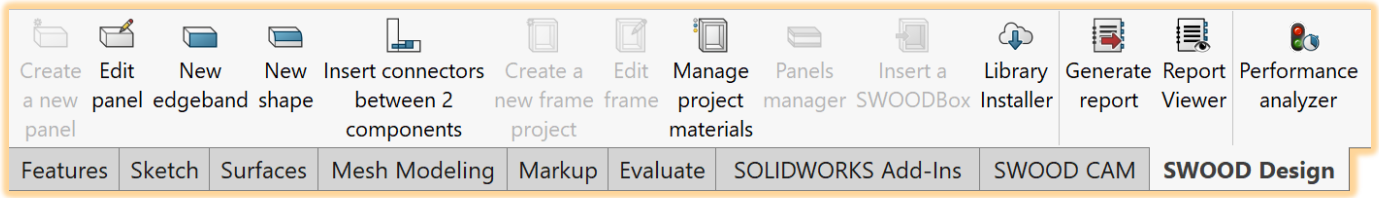
It is possible to add a column with the parameter "**Overlength**." To do this, it is necessary to:

- Go to the "Edgebands" page.
- Click on the "Modify columns" button.
- Go to the "Add/Remove" tab.
- In the "Edgebands" category, click on the "**Overlength**" parameter.
- Click on the "**Apply**" button (bottom right).
- If necessary, reposition the parameter in the list to choose the position of the column.
- Click on the table to exit the column editing interface.

It is possible to create a calculated parameter in the report configuration that is the sum of the length of an edgeband and its overlength.

SWOOD Design – Enhanced User Interface: New CommandManager

SWOOD Design 2025 introduces a new Command Manager specifically dedicated to its features, directly integrated into the SolidWorks interface. This improvement aims to centralize and organize all SWOOD Design tools in a more accessible way. Users can now find key commands such as panel creation and editing, edgeband and profile management, connector functions, access to the Panels Manager, SWOODBox insertion, and shortcuts to the Library Installer and Report Viewer.



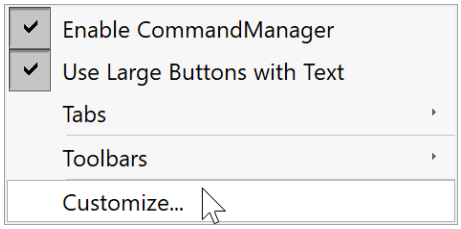
New Command Manager

How to use

1. *Access and customization*

The new SWOOD Design Command Manager appears as a separate tab in the SolidWorks interface, grouping together icons for the various features.

According to SolidWorks standards, this Command Manager is fully customizable. Users can add, delete, or rearrange commands according to their preferences and usual workflow. To do so, simply right-click on the Command Manager and select "Customize."



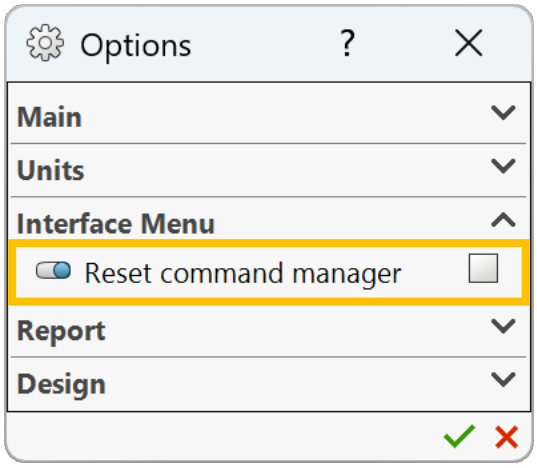
2. *Resetting the Order Manager*

In the event that the SWOOD Design Order Manager does not appear or displays inconsistencies, or if the user wishes to revert to the default configuration, a reset option has been added.
To do this, it is necessary to:

- Go to the SWOOD Design options. From the SolidWorks menu bar, click on:

  Tools → SWOOD Design → Settings

- In the "Interface Menu" section, check the " Reset Command Manager" setting.

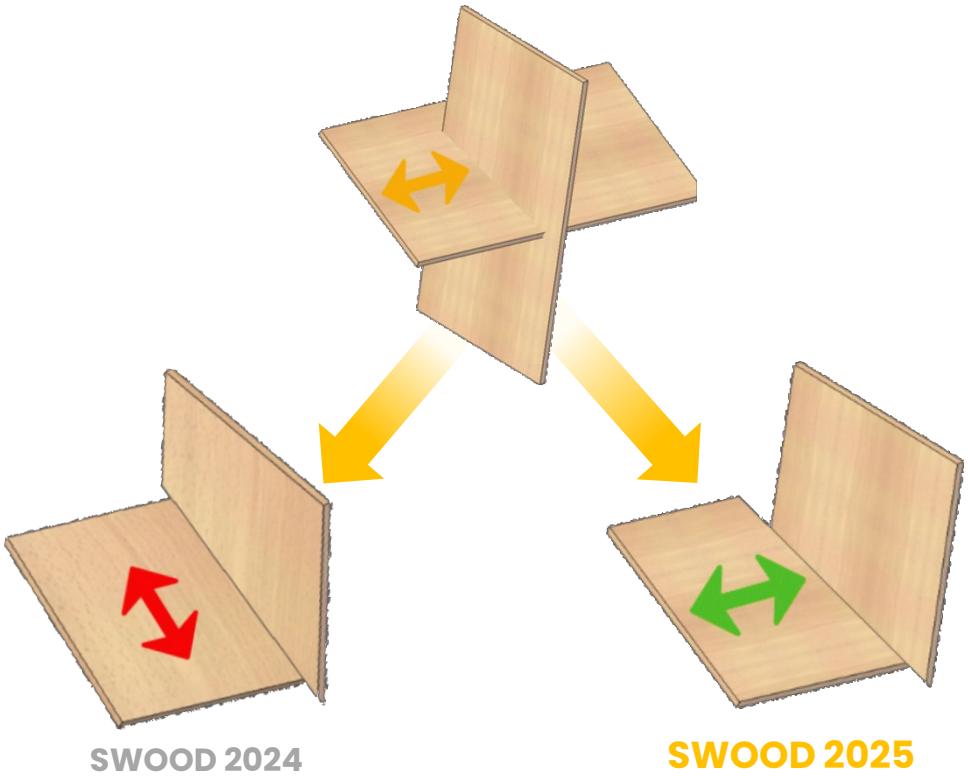


- Restart SolidWorks to see the changes.

Warning: Resetting the Command Manager will also reset all customizations.

SWOOD Design – Enhanced User Interface: Texture Stability

Accurate representation of materials, particularly those with a distinct grain direction such as solid wood, is crucial in furniture design. SWOOD Design 2025 brings a significant improvement to the stability of texture orientation applied to panels. Previously, certain geometric modification operations, such as panel cuts, could occasionally cause unwanted reorientation of the texture. This new version incorporates optimized texture management, ensuring that the orientation initially defined by the user is maintained. Thus, when creating a series of cabinet doors that need to have a continuous horizontal wood grain, the necessary cuts will no longer alter the visual alignment of the textures.

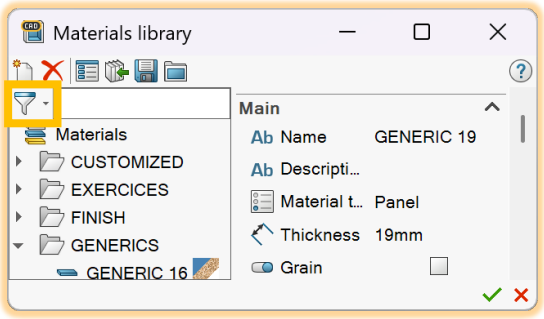


How the upgrade works

Improved texture stability in SWOOD Design 2025 is based on optimized internal management of texture application and tracking. The software now ensures that the initial texture orientation defined by the user (e.g., wood grain alignment) is consistently preserved even when panels undergo geometric changes such as cutting or machining. This approach ensures a more reliable and predictable visual rendering throughout the design process, reducing the need for manual adjustments and providing better control over the final appearance of the parts. No specific action is required on the part of the user to benefit from this improvement, as it is integrated into the standard behavior of the software.

SWOOD Design – Filters: Materials Library

The introduction of filters in SWOOD 2025 provides greater control over the selection of elements. This feature has now been extended to material library management, allowing users to refine their search and display of available materials. By combining criteria based on properties such as designation, thickness, material type, or even extended properties, it becomes easier to navigate through provided libraries. For example, a designer looking for a solid wood panel of a specific thickness, with a particular grain direction and belonging to a certain cost category, will be able to quickly isolate the corresponding materials using these filters, whether from the main library or when assigning a material to a panel being edited.



Filters in the material editing window

How to use

1. *Access to filters*

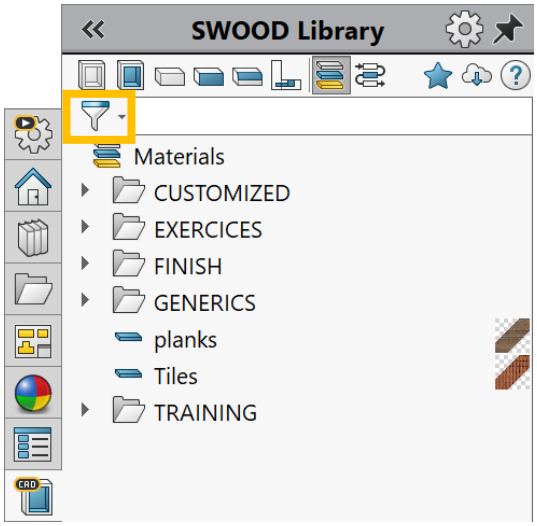
There are two different ways to access the Materials Library:

a. Access to filters in the main library

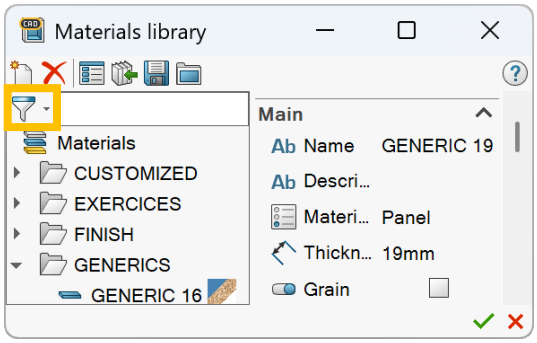
To access the filters in the main material library, it is necessary to:

- Go to the SWOOD Design task pane.
- Go to the Materials Library.

The filters are located at the top left.



The filters are also accessible in the material editing window.



b. Access to filters in the panel editing material library

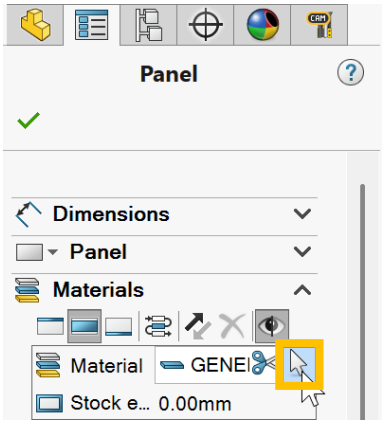
It is possible to access the elements in the material library when editing a panel and assigning a material to it.

To access the filters in the secondary material library, it is necessary to:

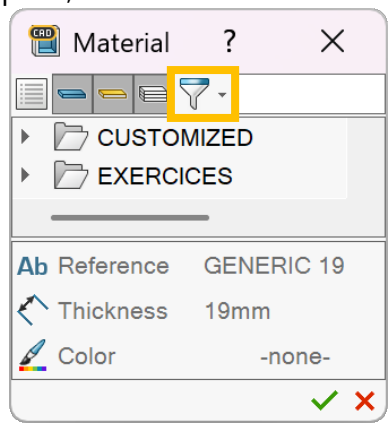
- Go to the SolidWorks menu bar and navigate to:

  Tools → SWOOD Design → Edit panel

- In the Materials section, click on the material selector.



- In the "Material" window that opens, the filter is located in the toolbar.



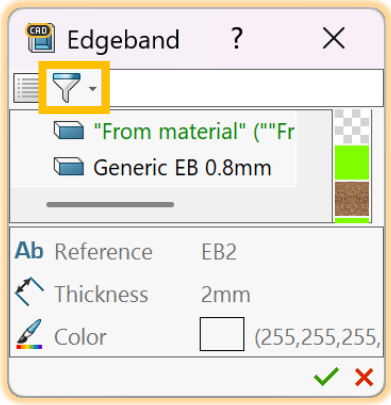
2. *Properties and attributes*

The properties and their specific attributes in the Materials Library are as follows:

Attributes	Attribute definition	Condition type
Name	Material name	List
Description	Material description	Text
Material type	Panel / Laminate / Compound	Text
Thickness	Material thickness	Number
Grain	Grain direction or not	Boolean
Cost	Material cost	Number
Color	Material color (selector)	List
Extended properties	*Depends on extended properties*	*Depends on attributes*

SWOOD Design – Filters: Edgebands Library

The cross-functional filter feature introduced in SWOOD 2025 also applies to edgeband library management. This tool allows users to target edgebands precisely based on their characteristics, making it easier to navigate and select from potentially large libraries. By defining criteria based on properties such as name, thickness, reference, or extended properties, users can quickly isolate the desired edgebands. For example, when searching for an edgeband with a specific thickness and finish for a project, applying the appropriate filters will display only the relevant options, whether from the main edgebands library or when assigning an edgeband to a panel.



Filters in the edgeband editing window

How to use

1. *Access to filters*

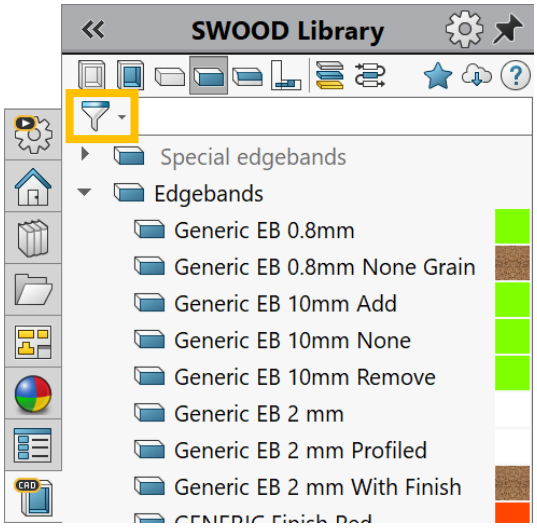
There are two different ways to access the Edgebands Library:

a. Access to filters in the main library

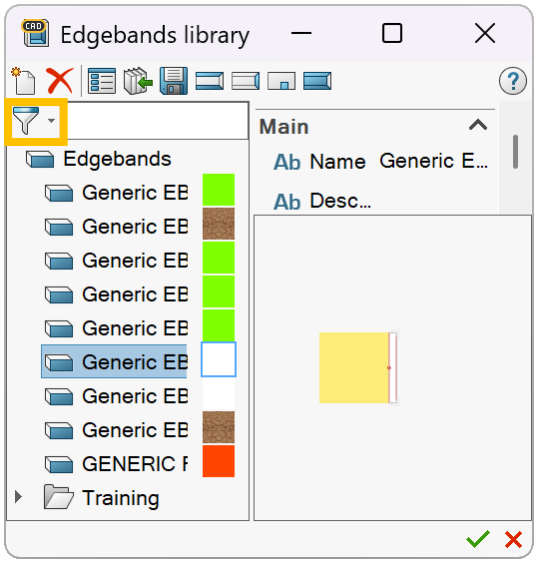
To access the filters in the main Edgeband Library, it is necessary to:

- Go to the SWOOD Design Task Pane.
- Go to the Edgeband Library.

The filters are located at the top left.



Filters are also accessible in the edgeband editing window.

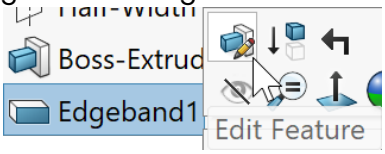


b. Access to filters in the Edgebands Library for editing edgebands

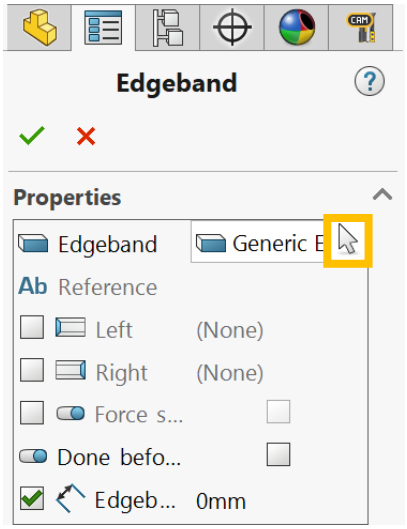
You can access the elements in the Edgebands Library when editing an edgeband that you want to select.

To access the filters in the secondary Edgebands Library, it is necessary to:

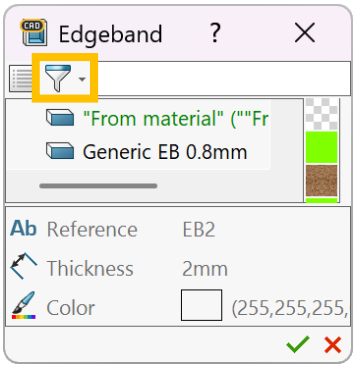
- Go to the FeatureManager design tree and right-click on an edgeband to edit.



- For the Edgeband parameter, click on the edgeband selector.



- In the "Edgeband" window that opens, the filter is located in the toolbar.



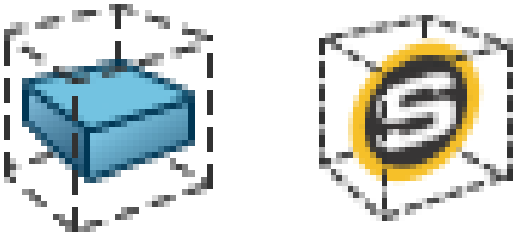
2. *Properties and attributes*

The properties and their attributes specific to the Edgebands Library are as follows:

Attributes	Attribute definition	Condition type
Name	Edgeband name	List
Description	Additional information will be written in parentheses after the name of the edgeband	Text
Cost	Edgeband cost	Number
Reference	Edgeband reference (for the report)	Text
Thickness	Edgeband thickness	Number
Create body	Defines the impact of the edgeband on the panel volume with three options: Remove, None, Add	List
Shaping	Defines the shape of the edgeband (Chamfer/Round) from the edgeband shape library.	List
End shaping	Defines the end shape of the edgeband (Chamfer/Round) from the End Shape library.	List
Couleur	Material color (selector)	List
Extended properties	*Depends on extended properties*	*Depends on attributes*

SWOOD CAM - Stock management in component positioning

The precise definition of the stock is a fundamental step in preparing for CNC machining, as it determines the positioning of the part on the machine and the generation of tool paths. SWOOD CAM 2025 improves this step by introducing new modes for defining the stock when setting up components, accessible when positioning is based on the SWOOD Design panel definition. These new options offer greater flexibility in determining the exact dimensions of the stock, taking into account either the bounding box of the pure geometry or information from SWOOD Design such as extensions, edgebands, or laminates. For example, a user can now choose to define the stock for a panel by specifically including edgeband overhangs for precise machining of the edgebands.

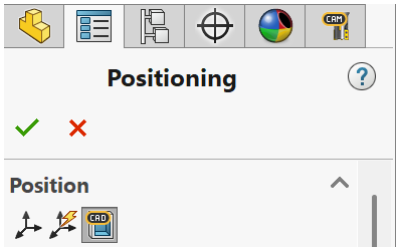


Stock definition icons

How to use

1. *Prerequisites for choosing stock definition methods*

The new stock definition modes are only available in the "Based on panel definition in SWOOD Design" position mode.



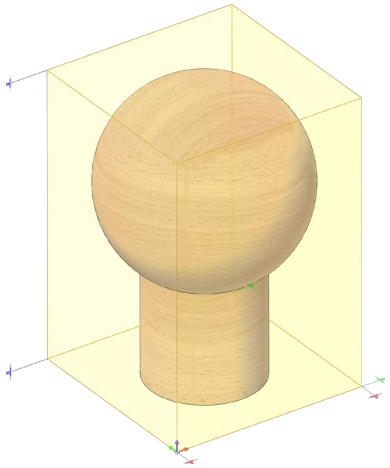
Therefore, it is necessary to have a license for **SWOOD CAM** and **SWOOD Design**.

2. *Methods of defining stock*

To define the stock for a part, in order to position it on the CNC machine, several positioning modes may be required:

c. Stock based on bounding box

When the stock is defined by the bounding box, this will take into account the smallest parallelepiped that contains the part.



d. Stock based on SWOOD Design panel

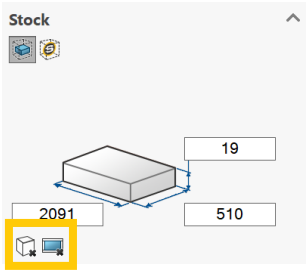
When the stock is defined by the SWOOD Design panel, this will take into account all SWOOD Design elements that could affect the stock of the panel (extension, edgebands, laminates).

3. *Additional settings*

When a stock definition mode is selected, certain additional settings are available.

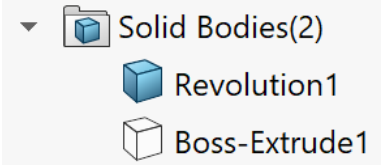
e. When stock is based on bounding box

When the stock is defined by the bounding box, there are two additional parameters available:



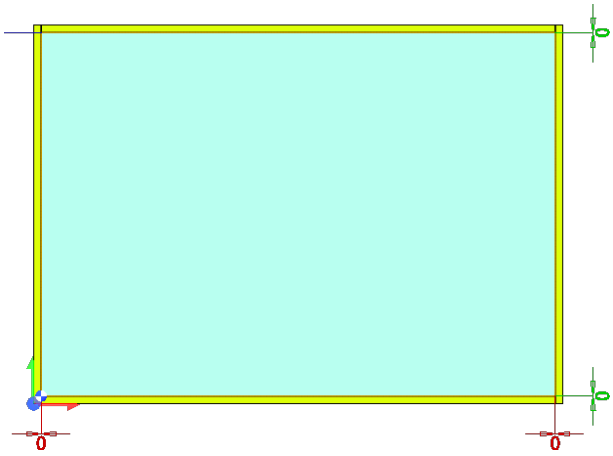
- Exclude hidden bodies

If the part being used contains bodies that have been hidden (in the FeatureManager design tree), clicking this option **will only consider the bodies that are visible** in the stock definition.



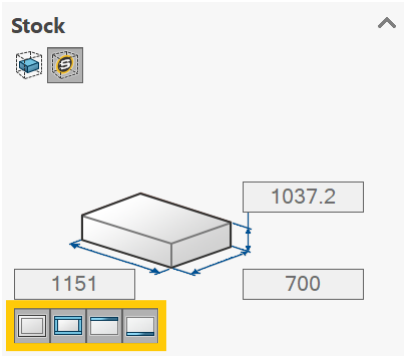
- Ignore edgeband bodies

If the part contains edgebands, clicking this option **will not take the edgebands into account** in the stock definition.



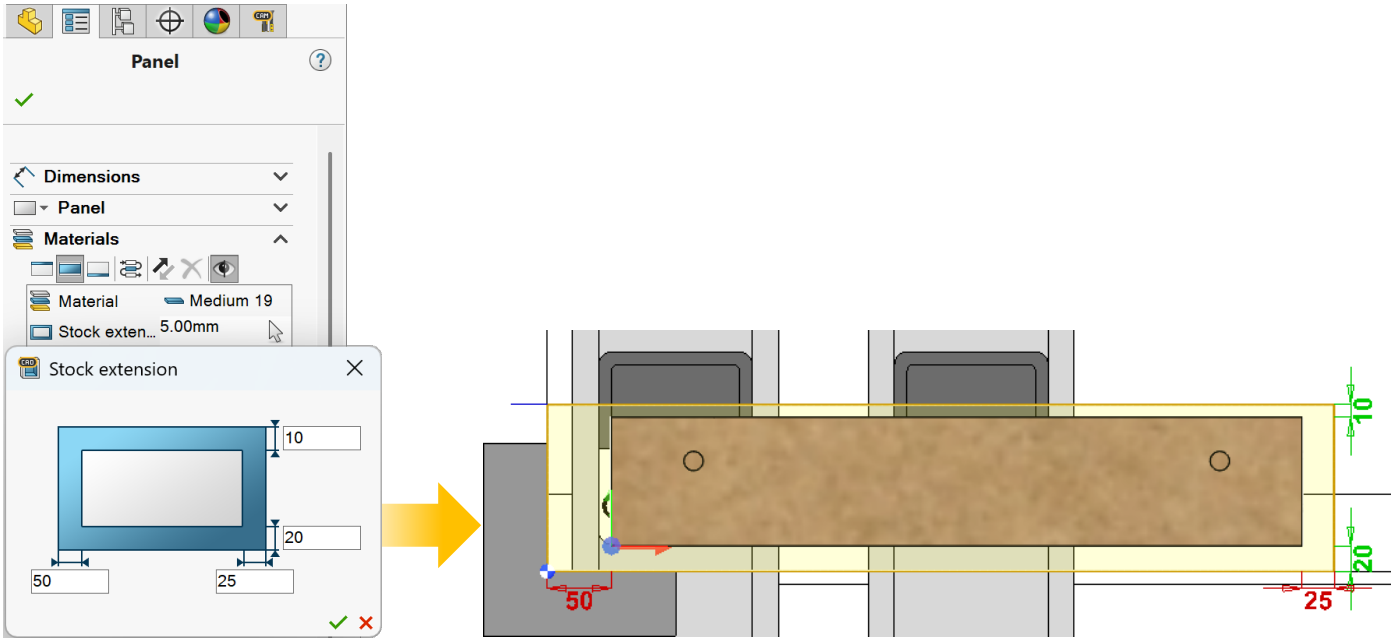
f. When stock is based on the SWOOD Design panel

When the stock is based on SWOOD Design panel, there are four additional parameters available:



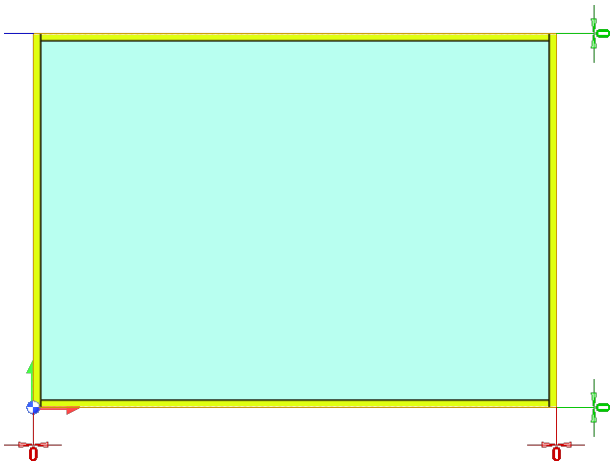
- Use stock extension

If the SWOOD panel has extensions, clicking on this option will take them into account when defining the stock.



- Use panel with edgebands

If the part contains edgebands, clicking this option **will take the edgebands into account** in the stock definition.



- Include top laminate

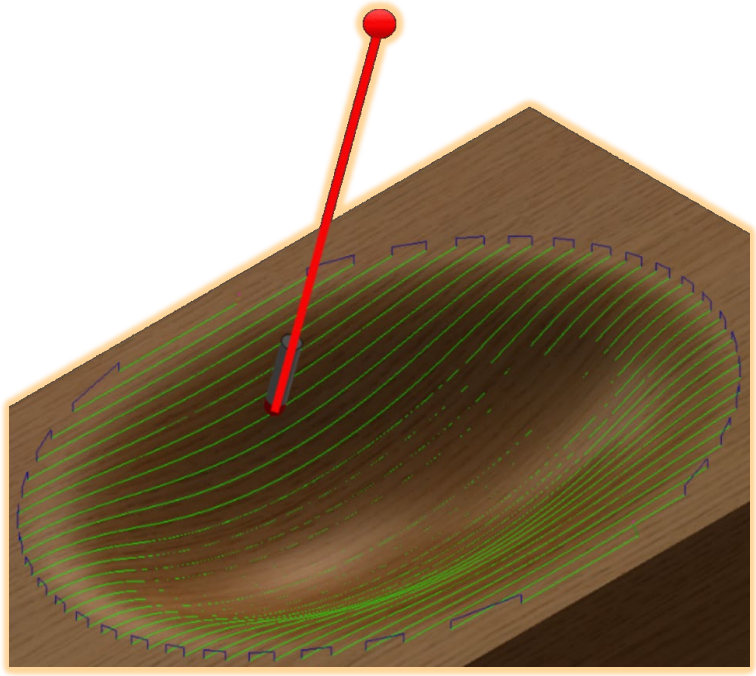
If the part contains a laminate on its top face, clicking this option **will take the top laminate into account** when defining the stock.

- Include bottom laminate

If the part contains a laminate on its bottom face, clicking this option **will take the bottom laminate into account** when defining the stock.

SWOOD CAM – 3D Finishing: New Tool Tilting Types

The 3D Finishing machining, introduced in SWOOD CAM 2024, provided a basis for detailed management of 3D machining operations. SWOOD CAM 2025 extends this capability by offering increased control over tool path and orientation with the introduction of five new tilting types. These options allow for fine adjustment of the tool angle relative to the machined surface, going beyond simple perpendicular orientation. For example, when machining a complex curved surface, the user can now select a tilting type that maintains the tool at an optimal angle of attack, improving surface quality and potentially extending tool life.



Example of 3D finishing with one of the new tilting types

How to use

1. *Access to tilting types*

To be able to choose from the new tilting types, it is necessary to:

a. Create a 3D finishing

To do this, go to the SolidWorks menu bar and click on:

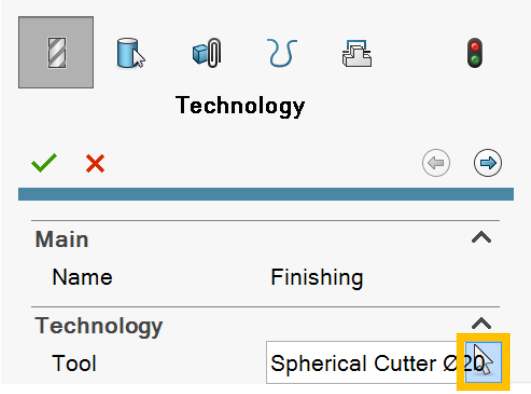
Tools → SWOOD CAM → 3D Finishing

b. Create a new "3D Finishing" operation

In the window that opens, click on the " 3D Finishing" button to create a new operation.

c. Assign a tool to the operation

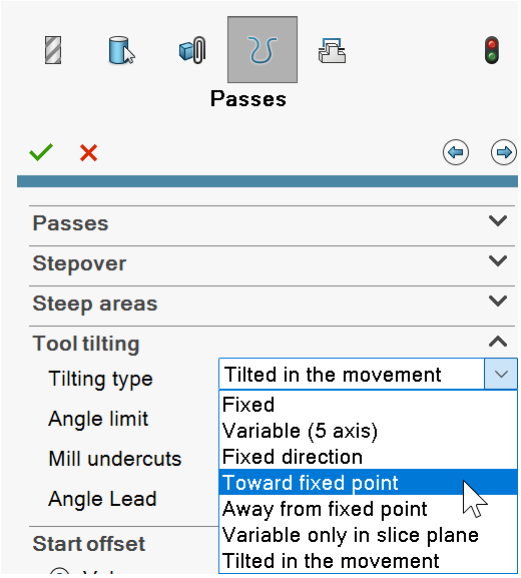
In the " Technology" tab, select a tool from the drop-down list or from the selector (in the "Technology" section).



Tool selector location

d. Change the tilting type

Dans l'onglet « Passes », sélectionnez le type d'inclinaison voulu (dans la section « Inclinaison de l'outil »).



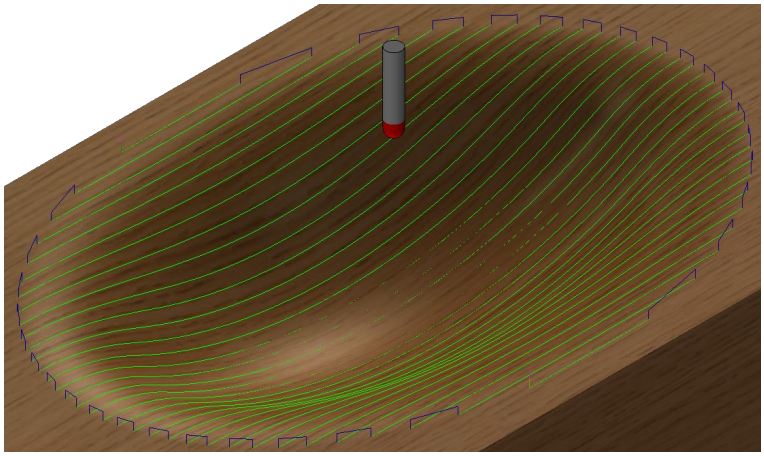
Available Tilting Types

2. *Descriptions of tilting types*

The tilting types can be described as follows:

a. Fixed (default value) [SWOOD 2024]

Mill the part keeping the tool vertical at all times.



Example of 3D finishing with a fixed angle

- • Parameters

None

Advantages:

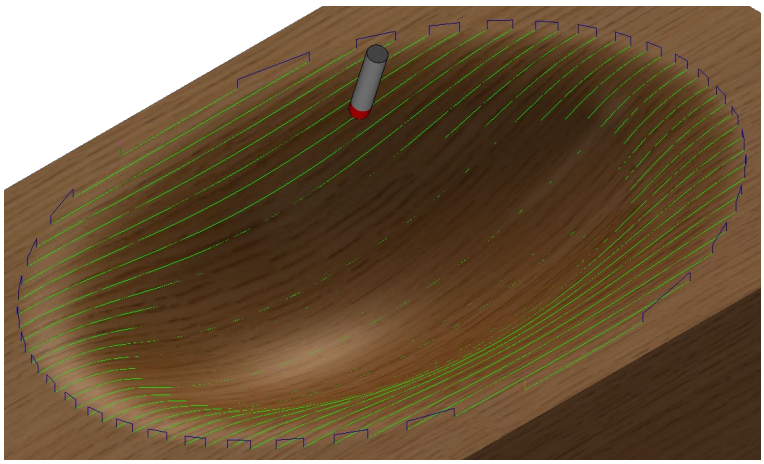
- Enables machining with a 3-axis CNC.
- Faster machining because there is no rotation of the B and C axes.

Disadvantages:

- Does not allow undercuts to be machined.
- Uneven surface quality.
- Less effective machining on steep slopes.

b. Variable (5 axis) [SWOOD 2024]

Machine the part by positioning the tool perpendicular to the surface.



Example of 3D finishing with a Variable inclination (5 axes)

- Parameters

- Angle limit: Defines the maximum angle B that can be used to lock the angle that the machining head can take.

- Undercut machining: Allows machining of overhanging parts, if the angle limit permits.

Advantages:

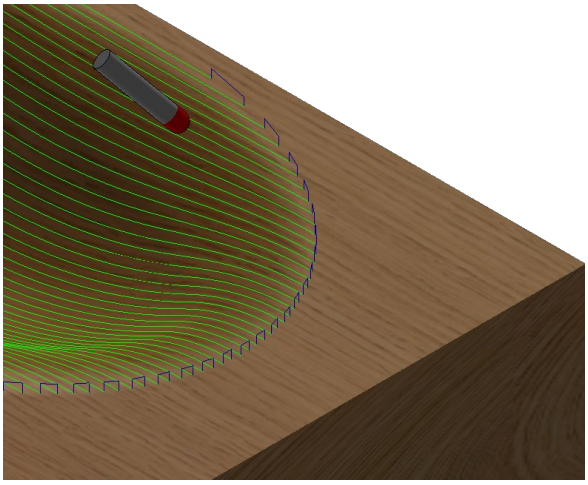
- Standardizes surface quality.
- Good machining on all slopes.
- Enables machining of undercuts.

Disadvantages:

- Slower machining.
- Uneven surface quality.

c. Fixed direction [SWOOD 2025]

Machine the part keeping the tool always parallel to a fixed direction.



Example of 3D finishing with a fixed direction

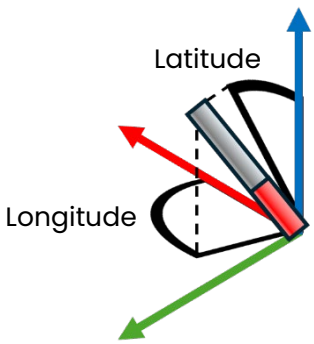
Parameters

- Angle limit
- Mill undercuts
- Selection (direction): Select a face or edge to specify the direction.

If the selection is an edge, the direction will be along that edge.

If the selection is a face, the direction will be perpendicular to that face.

- Latitude & Longitude (direction): If you do not use the selection (edge or face), you can specify the angle using latitude (angle in the XZ plane) and longitude (angle in the XY plane).

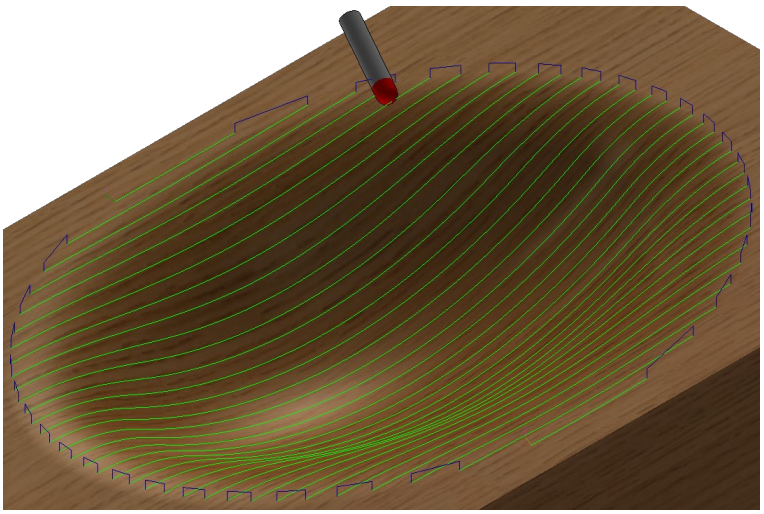


Advantages:

- Enables safe machining of complex areas.

d. Towards fixed point [SWOOD 2025]

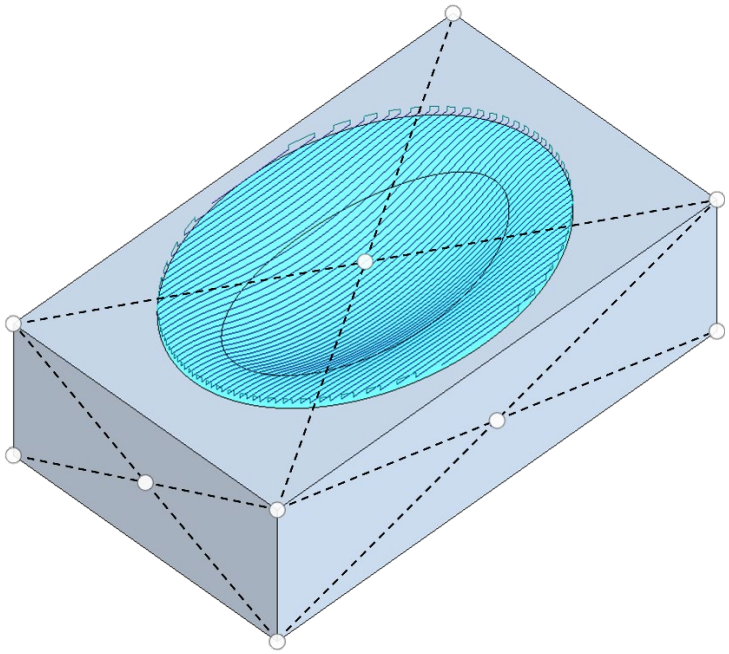
Machine the part with the tool always pointing toward a fixed point.



Example of a 3D finishing tool pointing to a fixed point

Parameters

- Angle limit
- Mill undercuts
- Point (From): Defines the position of the point to be fixed according to a reference ("From"), either a vertex of the blank, or the center of one of its 6 faces, or OP0, or the center of the blank.



Example of reference to the center of a face (here the rear face)

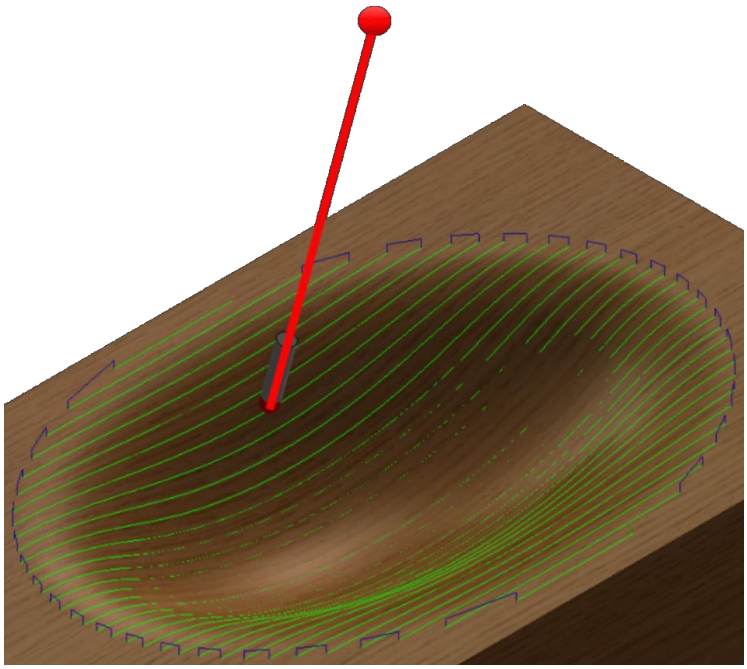
- Point (X, Y, and Z Offsets): Defines the offset on the 3 axes relative to the selected reference point.

Advantages:

- Can enable better uniformity of a concave round volume (e.g., a hemisphere).

e. Away from fixed point [SWOOD 2025]

Machine the part with the tool always pointing away from a fixed point.



Example of a 3D finishing operation where the tool points away from a fixed point

Parameters

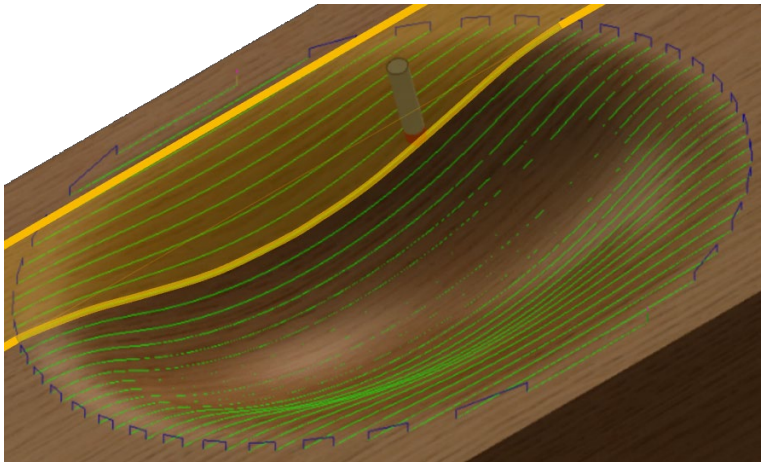
- Angle limit
- Mill undercuts
- Point (From)
- Point (Offsets X, Y and Z)

Advantages:

- Can enable better uniformity of a convex round volume (e.g., a sink).

f. Variable only in slice plane [SWOOD 2025]

Machine the part by positioning the tool perpendicular to the surface (or to OP0 or the center of the blank) while remaining in the plane of the path.



Example of 3D finishing variable only in slice plane

Parameters

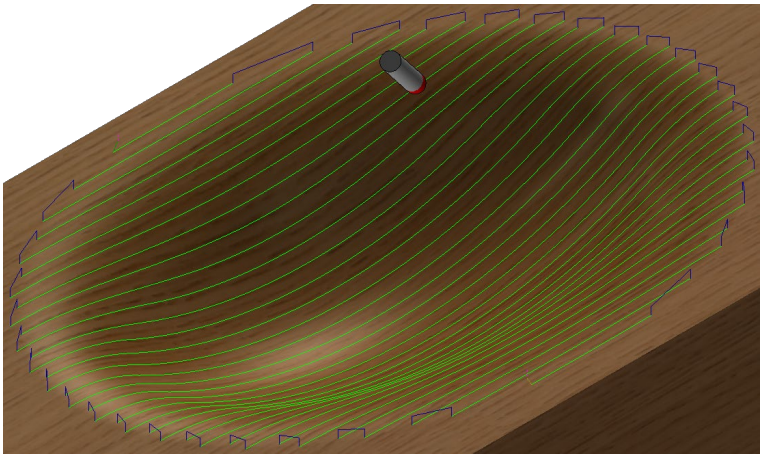
- Angle limit
- Mill undercuts

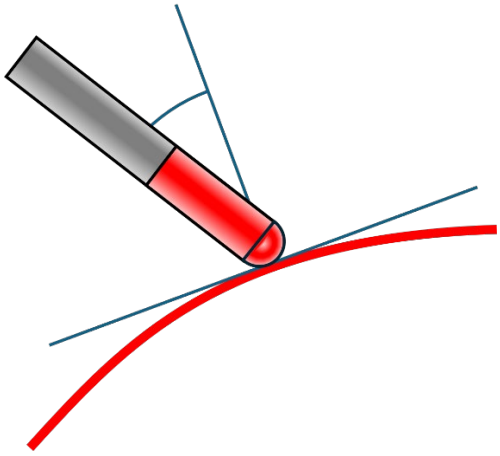
Advantages:

- Improves safety by ensuring that the head rotates only within the sluce plane.

g. Tilted in the movement

Machine the part by positioning the tool perpendicular to the surface while maintaining an angle of attack.





Example of 3D finish tilted in the movement

Parameters

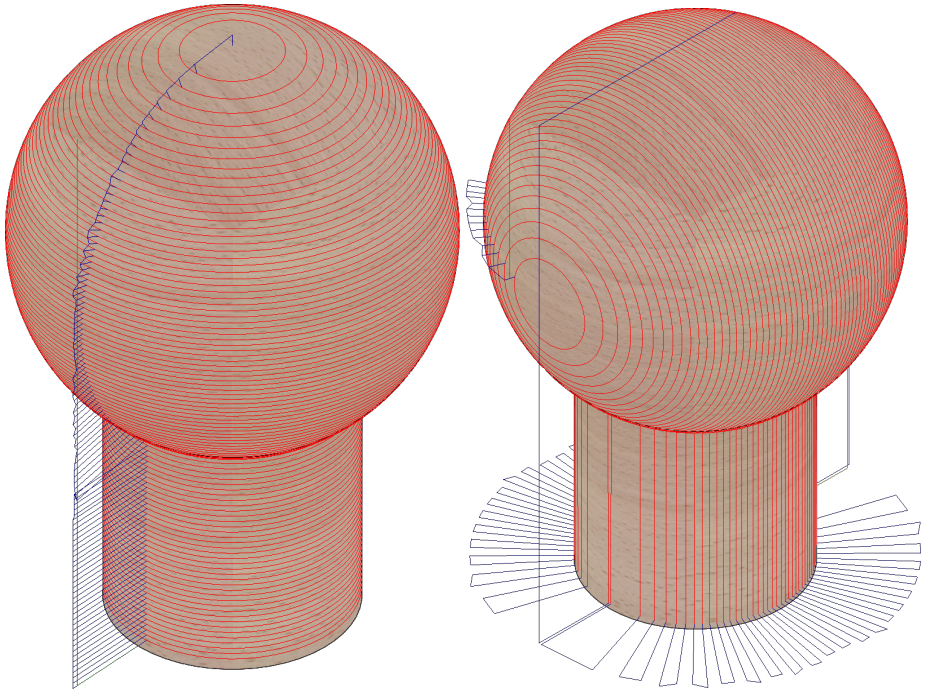
- Angle limit
- Mill undercuts
- Angle lead: Additional angle perpendicular to the surface.

Advantages:

- Allows the surface to be machined with a part of the tool that is sharper than its tip.

SWOOD CAM – 3D Finishing: Mill Undercuts

Since its introduction in 2024, the 3D Finishing machining in SWOOD CAM has provided options for configuring 3D machining operations. SWOOD CAM 2025 extends these capabilities by introducing undercut milling for "Parallel" pass strategies, in addition to improvements for "Contour" passes. By activating a dedicated option and using an appropriate 5-axis tilt strategy, the software generates the necessary trajectories to machine these overhanging areas, while managing clearances to ensure safe machining. For example, the manufacture of a complex turned part with an internal groove can now be programmed for complete finishing, including undercuts, using parallel passes.



Milling with undercut and "Contour" pass type (left) and "Parallel" pass type on the left

How to use

1. *Enabling undercut milling*

To mill the undercuts of a 3D part, it is necessary to:

a. Create a 3D Finishing

To do this, go to the SolidWorks menu bar and click on:

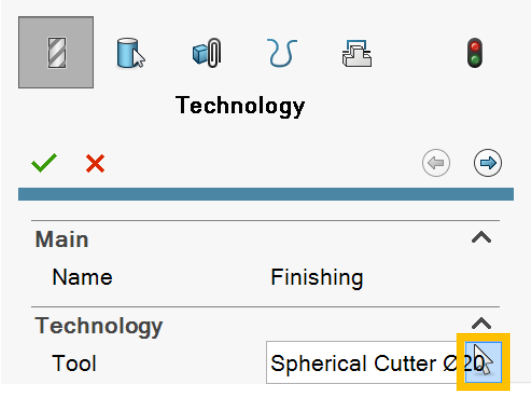
Tools → SWOOD CAM → 3D Finishing

b. Create a new "3D Finishing" operation

In the window that opens, click on the " 3D Finishing" button to create a new operation.

c. Assign a tool to the operation

In the " Technology" tab, select a tool from the drop-down list or from the selector (in the "Technology" section).



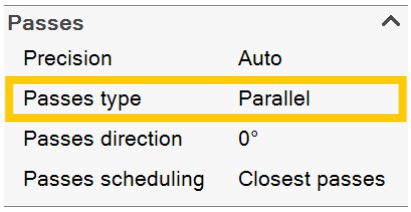
Location of the tool selector

d. Change the type of passes

Starting with SWOOD CAM 2024, it was possible to mill undercuts with a "Contour" passes type.

SWOOD CAM 2025 adds the ability to mill undercuts with the "**Parallel" passes type.**

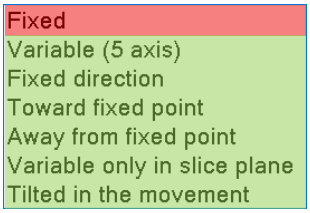
In the " Passes" tab, select the '**Parallel**' passes type (in the "Passes" section).



"Parallel" passes type

e. Assign a tilting type to the tool 5 Axes

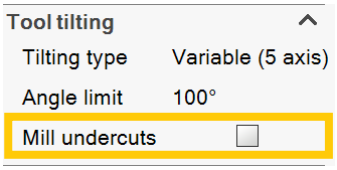
In the tilting types, select a 5-axis tilting (all except "Fixed").



Assign an angle limit.

f. Enable undercut milling

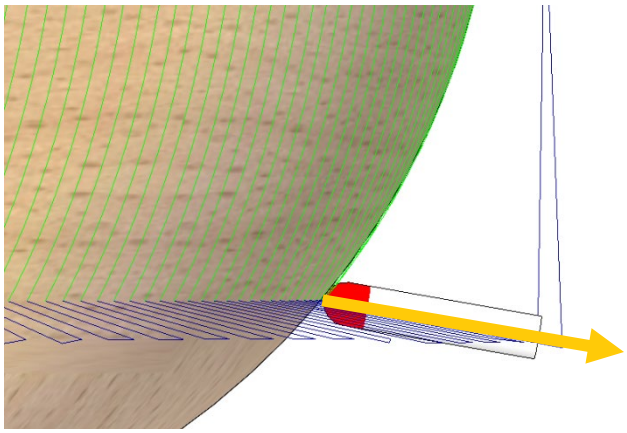
In the "Tilting type" section, click on "Mill Undercut ".



g. Enable tool clearance in transitions

Warning: When milling undercuts, the tip of the tool is below the raw material, and a vertical transition could cause a collision.

It is therefore essential to activate **clearance** during transitions.



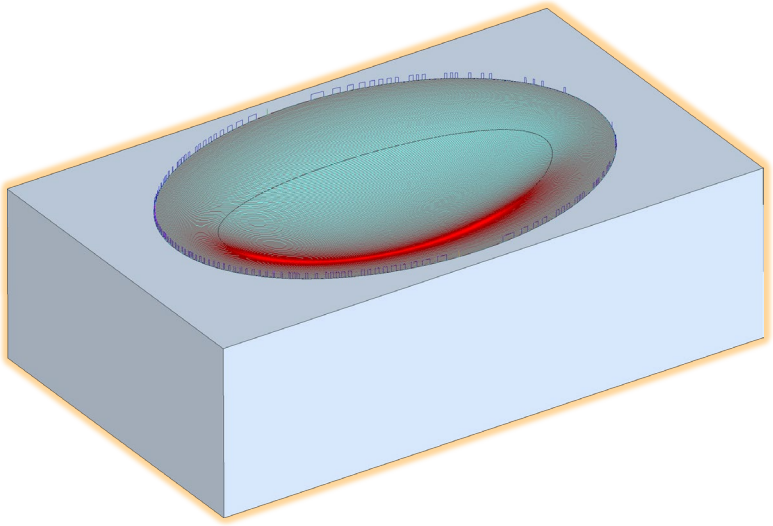
Example of clearance outside the raw part

To do this, it is necessary to:

- Go to the "Transitions" tab.
- Check the "Clearance" option.

SWOOD CAM – 3D Finishing: New Trajectory Display

When setting up complex 3D finishing operations, it is essential to have a clear and reliable visualization of tool paths directly in the editing window. SWOOD CAM 2025 introduces a new tool path display option called "Smart Trajectory," which is specifically optimized for the 3D finishing editing window. This new representation is designed to provide a more robust and efficient display of tool paths, particularly for complex machining, allowing for better anticipation of the result. For example, when fine-tuning the parameters of a finish on a sculpted surface, "Smart Trajectory" can help you better understand the sequence of passes and surface coverage before running a full simulation.



New trajectory display

How to use

1. *Context of application*

- This display option is available when editing a 3D finishing machining in SWOOD CAM.
- The representation of "Smart Trajectory" trajectories is, along with SWOOD 2025, the default display.

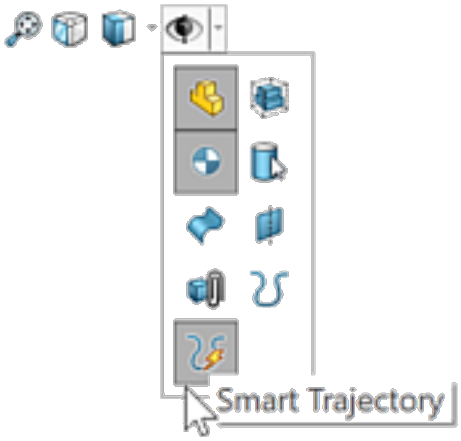
2. *Choice of trajectory display*

In order to choose the trajectory display, it is necessary to:

- Create a 3D Finishing machining by going to the SolidWorks menu bar:

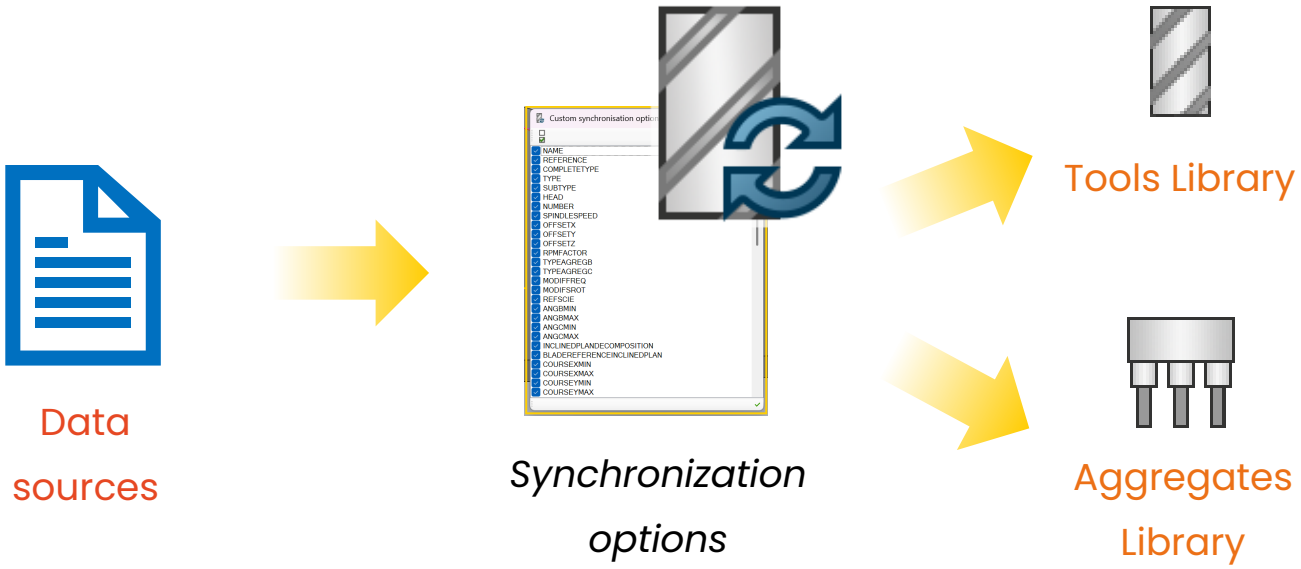
  Tools → SWOOD CAM → 3D Finishing

- Add a tool to the machining.
- Rebuild the machining.
- In the Graphics Area toolbar, click on the "Hide/Show Items" command.
- • Select the "Trajectory" or "Smart Trajectory" mode.


Smart Trajectory

SWOOD CAM - Tools Synchro: New Features

The "Tool Synchro" feature in SWOOD CAM allows users to create and maintain a library of tools and aggregates based on files from CNC controllers (such as HOMAG, HOPS, Maestro, etc.) or structured files (CSV, XML). SWOOD CAM 2025 brings several significant improvements to this tool, aimed at making it more robust, flexible, and accurate. These developments facilitate data import, offer better control over tool creation and updating, and improve library organization. For example, a user importing a tool database from a HOMAG machine will now benefit from more tolerant file reading and more refined options for managing aggregates.



Key improvements

1. *More robust and flexible file import*
   - **Improved reading of manufacturer files**

The tool is now more tolerant when reading files, particularly those from HOMAG, and can continue importing even if some final parameters (not essential to the tool definition) are missing.

   - **Improved management of file types during import**
     - For certain formats (e.g., HOMAG), the file type to be loaded may be grayed out to guide the user.
     - Filters on file type can be applied during upload to facilitate selection.

   - **Correction of the default dive angle**

When importing, the tool dive angle is now correctly initialized (for example, to 90° when relevant), avoiding potential errors when creating 5-axis finishing operations.

2. *Refined management of tools and aggregates*

- **Tool numbering based on reference**

A new option allows tool and corrector numbers to be generated based on the tool reference (for example, a reference "E012" can generate the number 12, I12, or I012 depending on the configuration).

- **More accurate identification and import of tool types**

Improvements have been made to better detect the actual type of tools when importing from certain files (e.g., correct distinction between a saw and a roughing cutter for Dati files).

- **Importing and organizing aggregates**
  - Aggregates (and their spindles) are imported with the correct tool type.
  - It is now possible to import aggregates into a specific folder within the aggregate library, named for example after the source library, for better organization and visibility.

- **Control over the creation and updating of tools**
  - Options are available to allow or disallow the creation of new tools during synchronization if a corresponding tool is not found.
  - A feature allows existing tools to be synchronized (based on criteria such as tool type and name) rather than systematically adding new tools.
  - Additional options are available for selectively deleting tools or refreshing specific properties such as diameter.

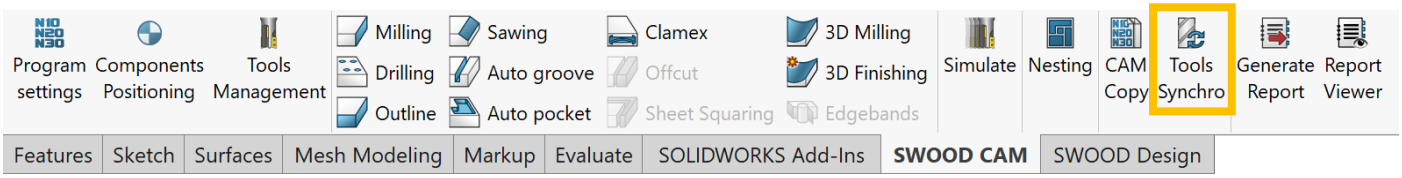
- **Data column management**

  The user has more control over which data columns are read and which can be updated during synchronization.

3. *Improvements to the interface and usability*

- **Easy access to Tool Synchro**

Shortcuts to the synchronization tool have been added to the SolidWorks "Tools → SWOOD CAM" menu and directly from the SWOOD CAM command manager for more direct access.

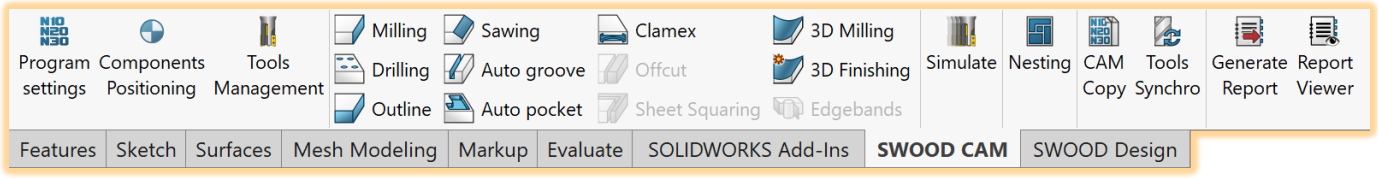


- **Visibility of modifications**

  The interface more clearly indicates modified, corrupted, existing, or new tools, as well as specific fields that have been modified.

SWOOD CAM- Enhanced User Interface: New CommandManager

SWOOD CAM 2025 introduces a new CommandManager specifically dedicated to its CAM programming features, directly integrated into the SolidWorks interface. This development aims to centralize and organize all SWOOD CAM tools in a more accessible way. Users can now find key commands such as milling phase management, component positioning, access to different machining (routing, drilling, 3D finishing, etc.), simulation, nesting management, as well as shortcuts to CAM Copy and tool synchro.

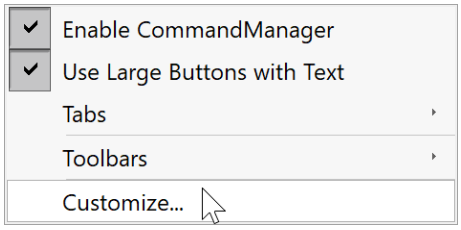


New Command Manager

How to use

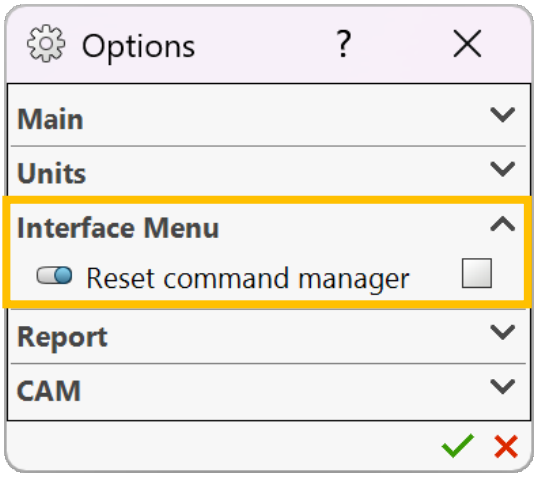
1. *Access and customization*

- The new SWOOD CAM Command Manager appears as a separate tab in the SolidWorks interface, grouping together icons for the various features.
- According to SolidWorks standards, this Command Manager is fully customizable. Users can add, delete, or rearrange commands according to their preferences and usual workflow. To do this, simply right-click on the Command Manager and select "Customize."



2. *Resetting the Command Manager*

- In the event that the SWOOD CAM Command Manager does not appear or displays inconsistencies, or if the user wishes to revert to the default configuration, a reset option has been added.
- To reset:
  - Access the SWOOD CAM settings. From the SolidWorks menu bar, click:
    Tools → SWOOD CAM → Settings
  - In the "Interface Menu" section, check the " Reset Command Manager" setting.
  - Confirm the changes.

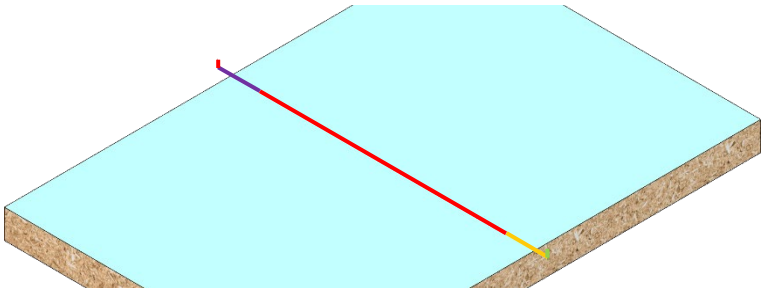


- Restart SolidWorks to see the changes.

Warning: Resetting the Command Manager will also result in the loss of all customizations previously made by the user on this tab.

SWOOD CAM – Entry / Exit Machining Feedrate

Machining operations are at the heart of SWOOD CAM, with numerous features designed to fine-tune each parameter with precision. With SWOOD CAM 2025, a new setting has been introduced: it is now possible to define different feedrates at the entry and exit points of the material. These two crucial moments can generate chips, and this feature allows you to significantly reduce or even completely eliminate them. This gives you a perfect finish while protecting your parts and optimizing the final quality of your machining operations.



Representation of entry and exit feedrates for machining operations

How to use

1. *Tool edition*

The entry and exit feedrate parameters for machining operations can be entered directly in the tool settings. This means that they will be applied each time a tool is used.

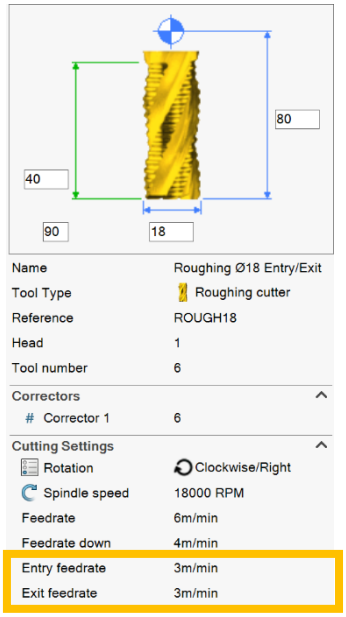
a. Access to parameters

To add entry and exit feedrate settings in the tool settings, it is necessary to:

- Go to the SWOOD CAM library in the task pane.
- Go to the " Tools" tab.
- Select a tool, then right-click and select " Edit."
- In the "Cutting Settings" section, you can find the entry and exit feedrate settings for machining operations.

b. Parameter definition

The entry and exit feedrate parameters for machining operations are grouped together in the "Cutting Settings" section:



Feedrate parameters for machining entry and exit in the tool parameters

- **Entry feedrate**

Speed at which the tool will enter the material during the operation (trajectory section starting from the moment the tool emerges from the dive)

- **Exit feedrate**

Speed at which the tool will exit the material during the operation.

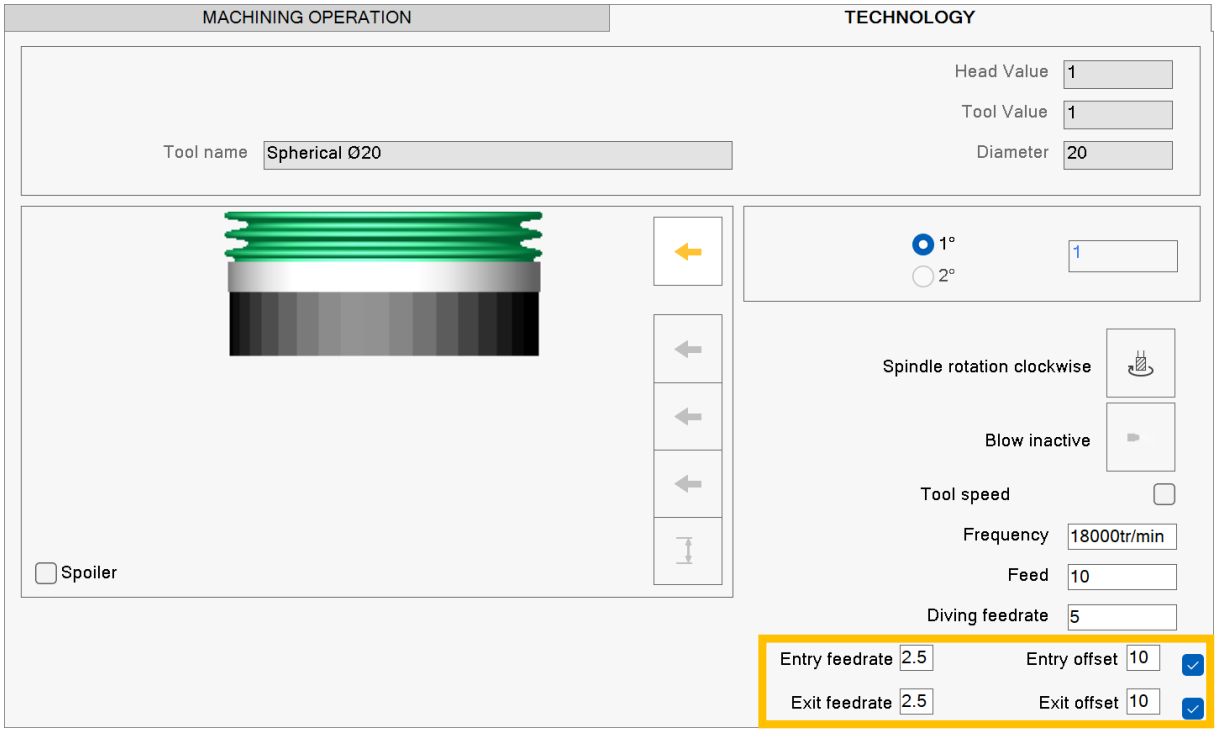
2. *Operation edition*

In a compatible machining operation, it is possible to add or modify entry and exit feedrates.

a. Access to parameters

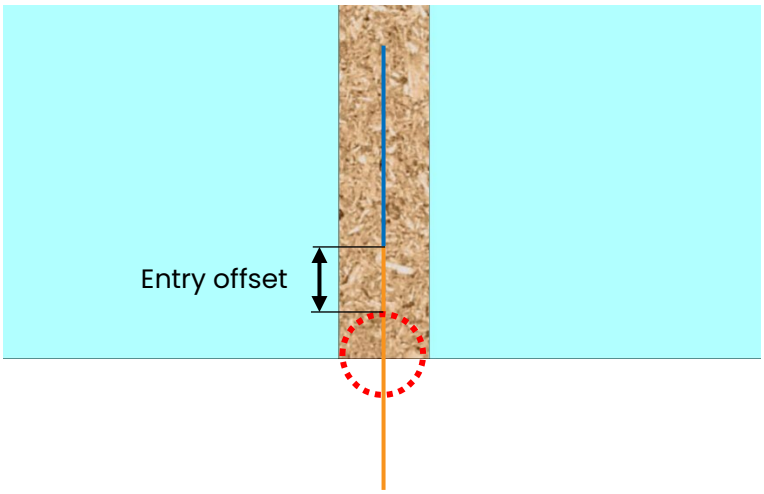
- In the SWOOD CAM FeatureManager, double-click on the operation to be edited.
- When the window opens, select the "Technology" tab.

At the bottom right, you can find the entry and exit feedrate settings for machining operations.



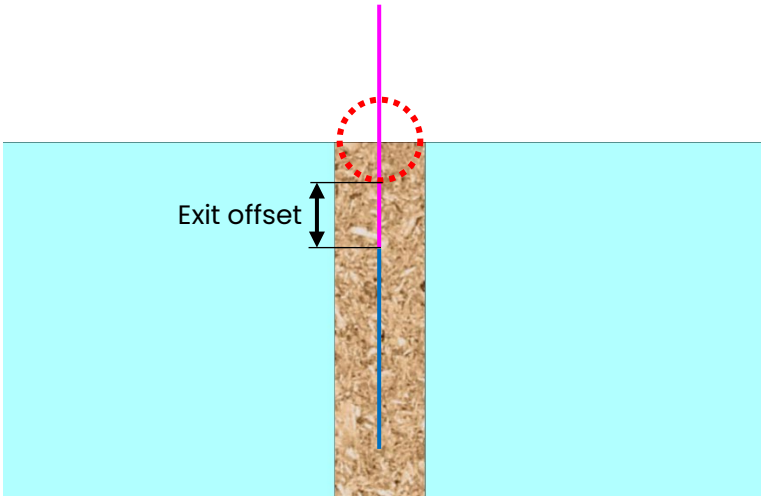
- **Entry offset**

Additional distance added to the tool **radius** to define the distance inside the material at the feedrate on entry.



- **Exit offset**

Additional distance added to the tool **radius** to define the distance inside the material at the feed rate on exit.



To activate the entry and exit feedrates, check the ☑ boxes.

b. Operations concerned

The operations for which entry and exit feedrates are available are as follows.

- Contours: Contour, Staydown contour, Partial contouring, Outline contour.
- Move tool center: closed and open.
- Interpolate Axis C.
- Continuous 5-axis.

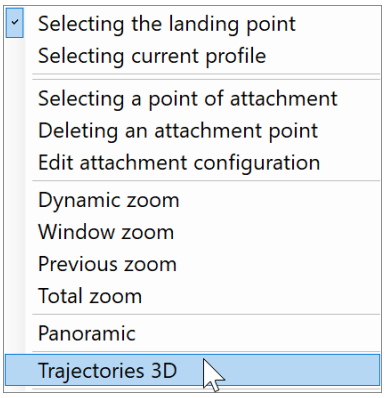
3. *Display of entry and exit feedrates*

It is possible to view the entry and exit feedrates.

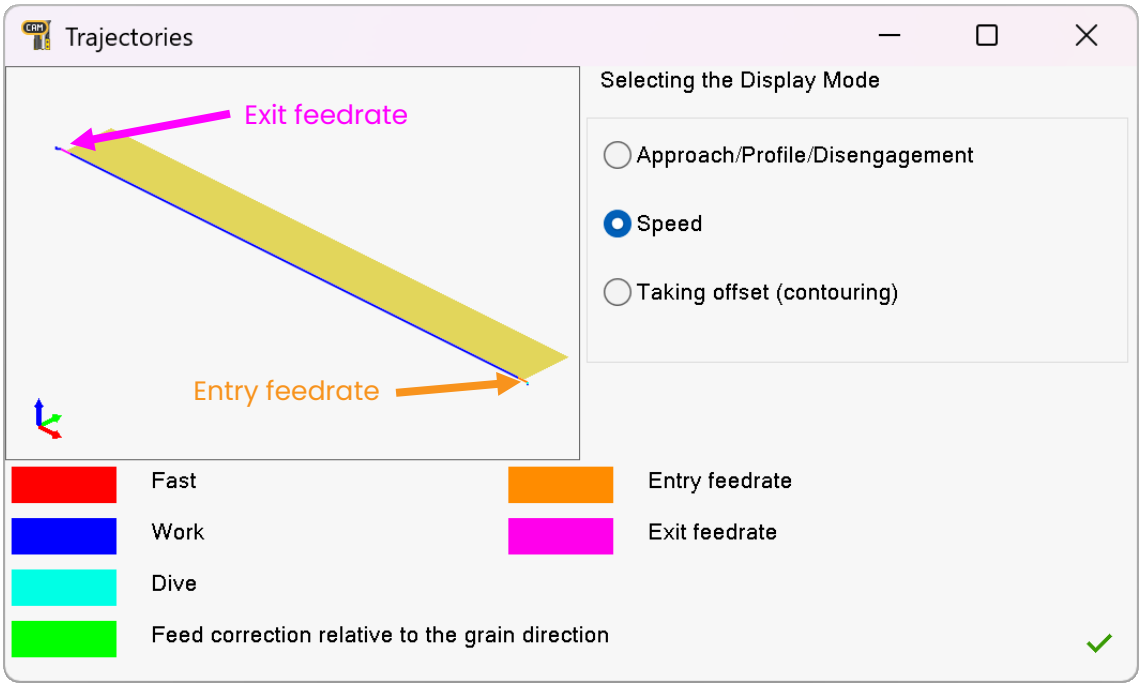
a. In the operation

To view the entry and exit feedrates in the operation, it is necessary to:

- Edit the operation.
- Right-click on the graphics view of the operation.
- Select "Trajectories 3D"



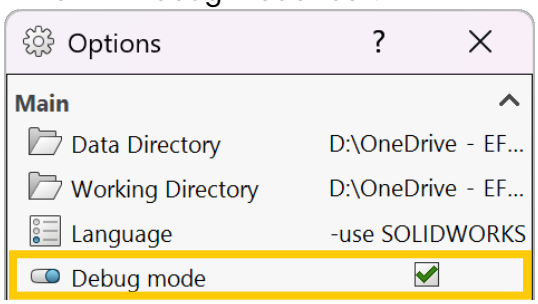
- In the new "Trajectories" window, click on " Speed."



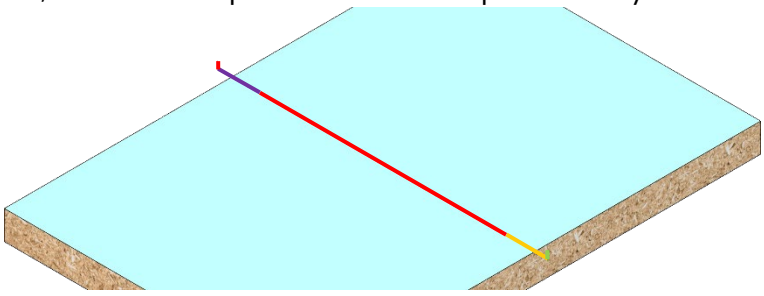
b. In the Graphics Area

To view entry and exit feedrates in the SolidWorks Graphics View, it is necessary to:

- Open the SWOOD CAM settings from the SolidWorks menu bar:
  Tools → SWOOD CAM → Settings
- In the main settings, check the "☑ Debug mode" box.



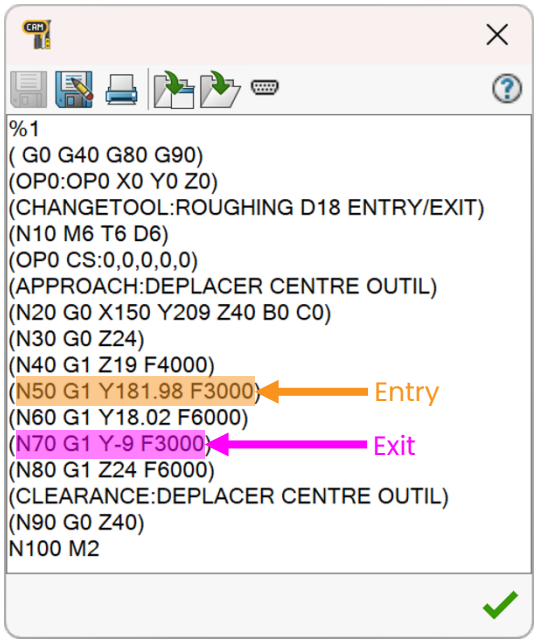
- In the milling phase, click on the operation that has specific entry and exit feedrates.



Different colors can be observed, corresponding to different milling feedrates.

c. In the program

When encoding the program, the application of entry and exit feedrates can be observed.


%1
(G0 G40 G80 G90)
(OP0:OP0 X0 Y0 Z0)
(CHANGETOOL:ROUGHING D18 ENTRY/EXIT)
(N10 M6 T6 D6)
(OP0 CS:0,0,0,0,0)
(APPROACH:DEPLACER CENTRE OUTIL)
(N20 G0 X150 Y209 Z40 B0 C0)
(N30 G0 Z24)
(N40 G1 Z19 F4000)
(N50 G1 Y181.98 F3000)
Entry
(N60 G1 Y18.02 F6000)
(N70 G1 Y-9 F3000)
Exit
(N80 G1 Z24 F6000)
(CLEARANCE:DEPLACER CENTRE OUTIL)
(N90 G0 Z40)
N100 M2

SWOOD CAM – Sheet Squaring

Machining parts on several sides of a nesting board requires flipping the board. To ensure accurate positioning after this flip, SWOOD CAM 2025 introduces a new dedicated machining entity: **Sheet Squaring**. This highly specific feature is designed to machine two edges of the stock board to form a clean corner during the milling phase following a flip. The objective is to create two precise reference surfaces on the flipped board, thus ensuring exact repositioning for machining the second side. For example, after machining the first side of a series of doors, the operator flips the board; the Sheet Squaring entity is then used to machine a reference edgeband before proceeding to machine the second side.

Sheet squaring machining icon

How to use

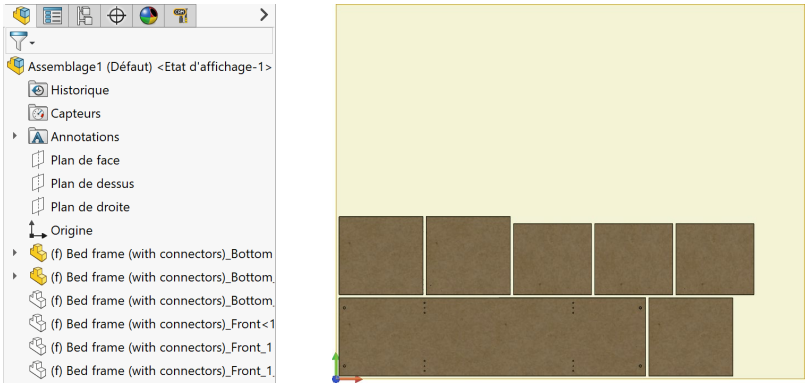
1. *Conditions of access*

The sheet squaring machining is very specific to the Nesting flip milling phases and can only be used under very specific conditions.

In all other cases, this machining is grayed out and unusable.

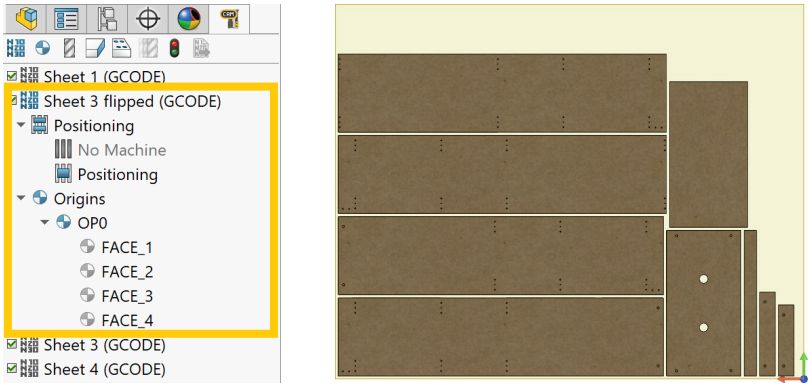
d. Nesting Assembly

The first condition for accessing the Sheet Squaring machining is **to be editing a Nesting assembly**.



e. From a flip milling phase

The second condition is that you must **be editing a flip milling phase**.



2. *Insertion*

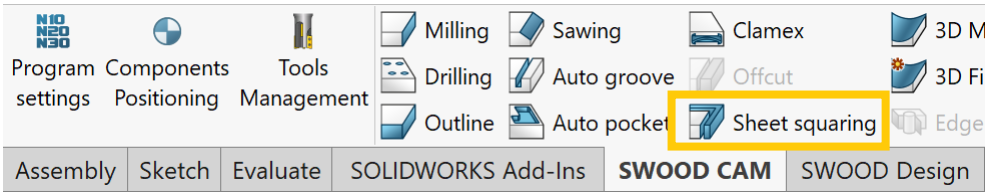
Once the conditions are met, there are several ways to insert this machining.

f. By inserting classic machining

Once the Nesting assembly has been generated, it is possible to insert this machining from the SolidWorks menu bar:

Tools → SWOOD CAM → Sheet squaring

This button can also be accessed from the Command Manager.



g. Through automatic machining insertion of Nesting

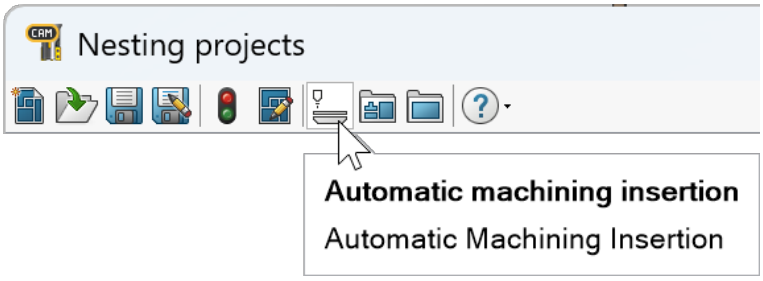
Even before generating the Nesting assembly, it is possible to force the automatic insertion of this machining directly from the Nesting interface.

There are two methods:

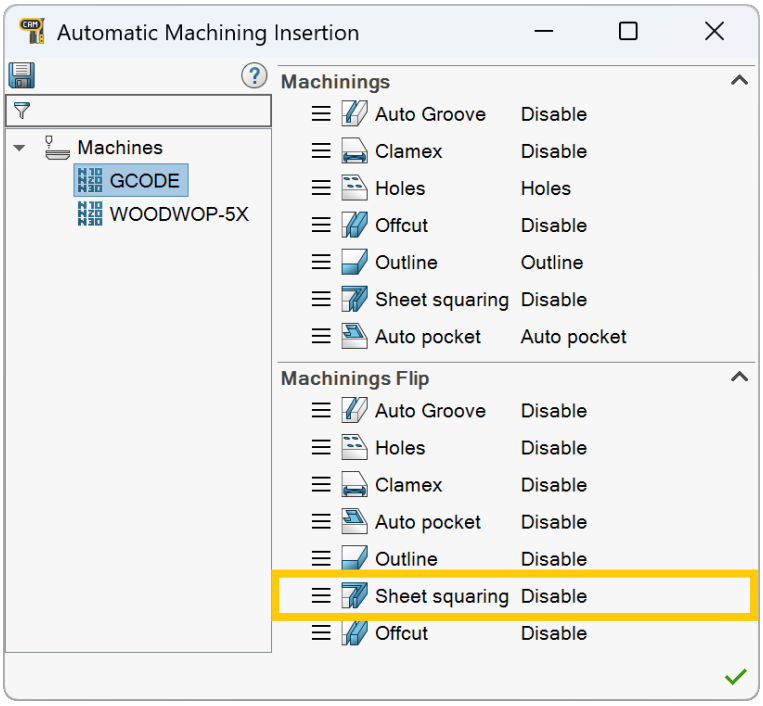
- From automatic machining insertion

To do this, from the Nesting projects window, it is necessary to:

- Click on the automatic machining insertion button (in the toolbar).



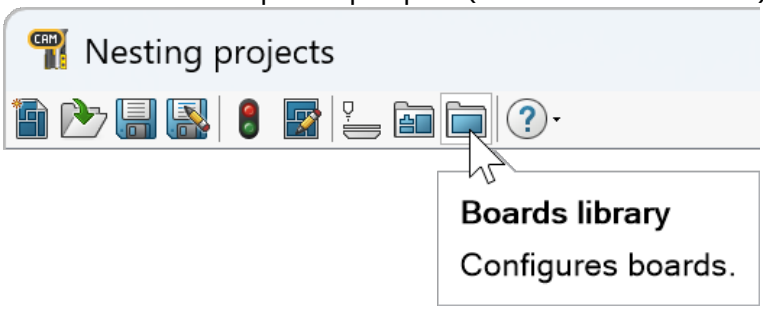
- Select a Post-Processor.
- In the "Machining Flip" section, select a Sheet squaring machining previously saved in the Machining Library.



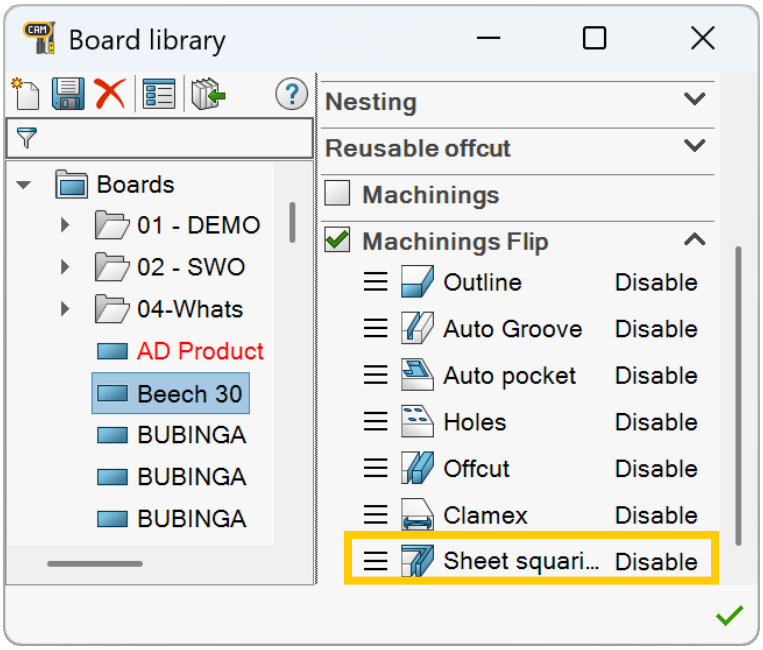
- From the Boards library

To do this, from the Nesting projects window, it is necessary to:

- Cliquer sur le bouton de la bibliothèque de plaques (dans la barre d'outils).



- Select a board.
- At the bottom of the settings, check the "Machining Flip" section.
- Select a Sheet Squaring machining previously saved in the machining library.



3. *Machining parameters*

a. Compatible tools

As this is a machining operation from the **Contours** family, the tools compatible with the Sheet Squaring are the same. However, it is advisable to use a **roughing cutter**, a **finishing cutter** or a **saw**.

b. Available operations

As this is a machining operation from the **Contours** family, the operations available with the Sheet Squaring are the same (such as "**Contour**," "**Move Tool Center**," "**Saw**," etc.).

SWOOD Nesting - Nesting Flip

SWOOD Nesting optimizes part nesting to minimize waste and maximize efficiency. SWOOD Nesting 2025 introduces a major feature for managing parts that require double-sided machining. It is now possible to group these parts on the same board, machine one side without contouring, then flip the board to machine the second side and perform the final cut. This approach simplifies the workflow and improves accuracy. The configuration includes activating the flip, defining the parts to be flipped by creating opposite milling phases or manually, setting automatic machining insertions for each side, and managing the grouping of flipped parts on a minimum number of boards. The result is a nesting assembly with separate milling phases for each side of the board.

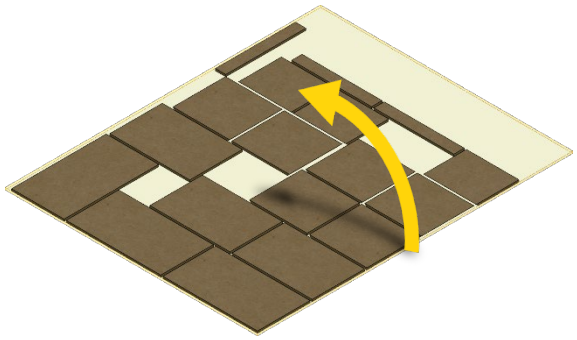


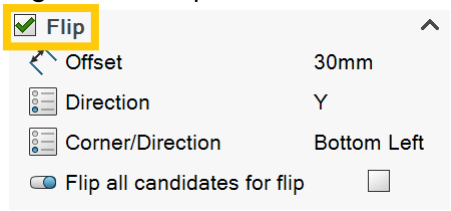
Illustration of board flip

How to use

1. *Flip activation*

To benefit from the flip feature in Nesting, you must first enable it in the general Nesting settings.
To do this, it is necessary to:

- Create a Nesting project (*Tools → SWOOD CAM →* *Nesting*).
- In the Nesting project window, go to the "Flip" section and check the activation box.



2. *Create parts to flip*

There are several ways to force certain parts to be flipped:

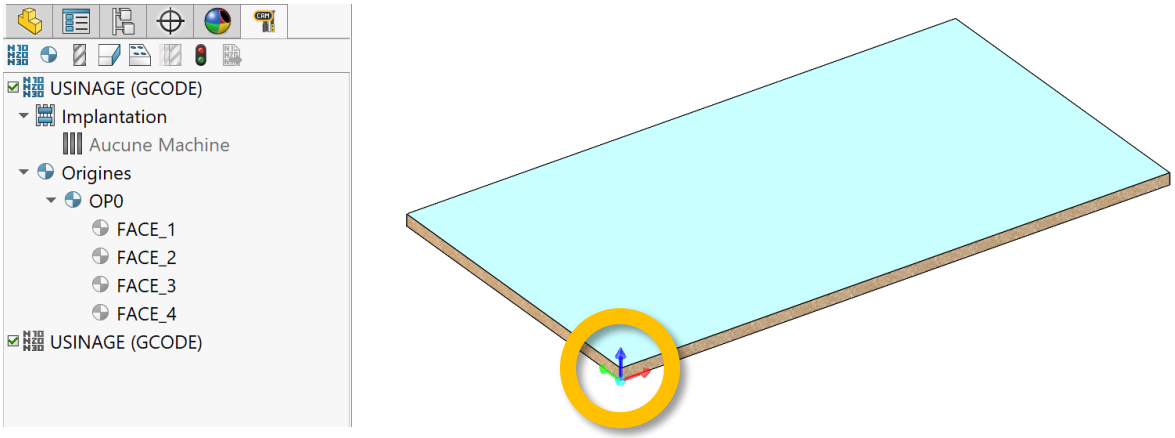
a. By creating two milling phases

To indicate to Nesting that you want to flip the board, you must create **two milling phases with opposite origins on each part**.

The steps are as follows:

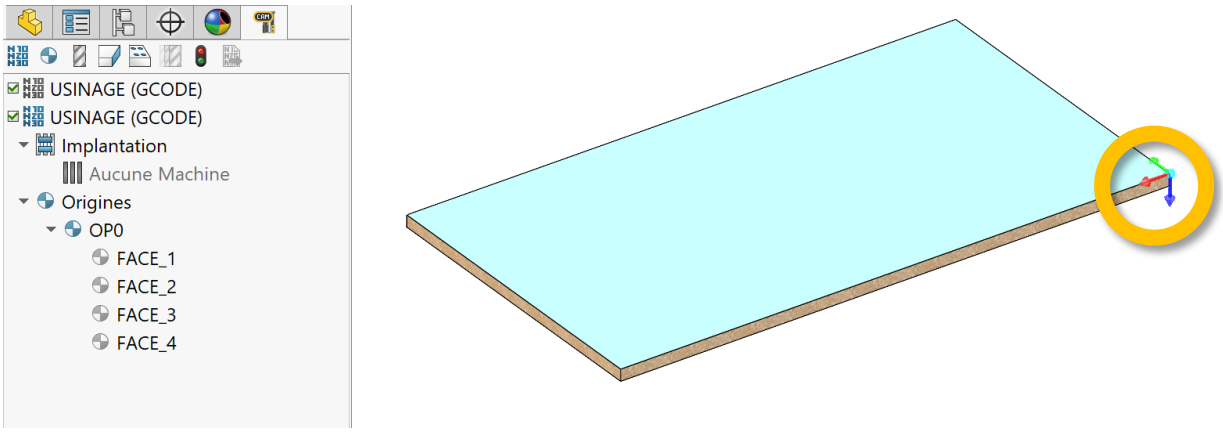
1. Edit the part

In a part, you just need to create an initial milling phase and an origin with the Z axis pointing in one direction.



First milling phase with the Z in one direction

Thena second milling phase must be created with an origin whose Z axis points in the opposite direction.

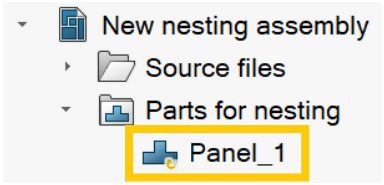


Second milling phase with the Z in the opposite direction

b. Individually from the Nesting project window

When creating a nesting project, the algorithm automatically detects parts that have opposite machining operations (non-through) on the top and bottom faces.

In fact, in the parts for nesting list, they are considered "Flip candidates" identifiable by the icon with the yellow arrow.

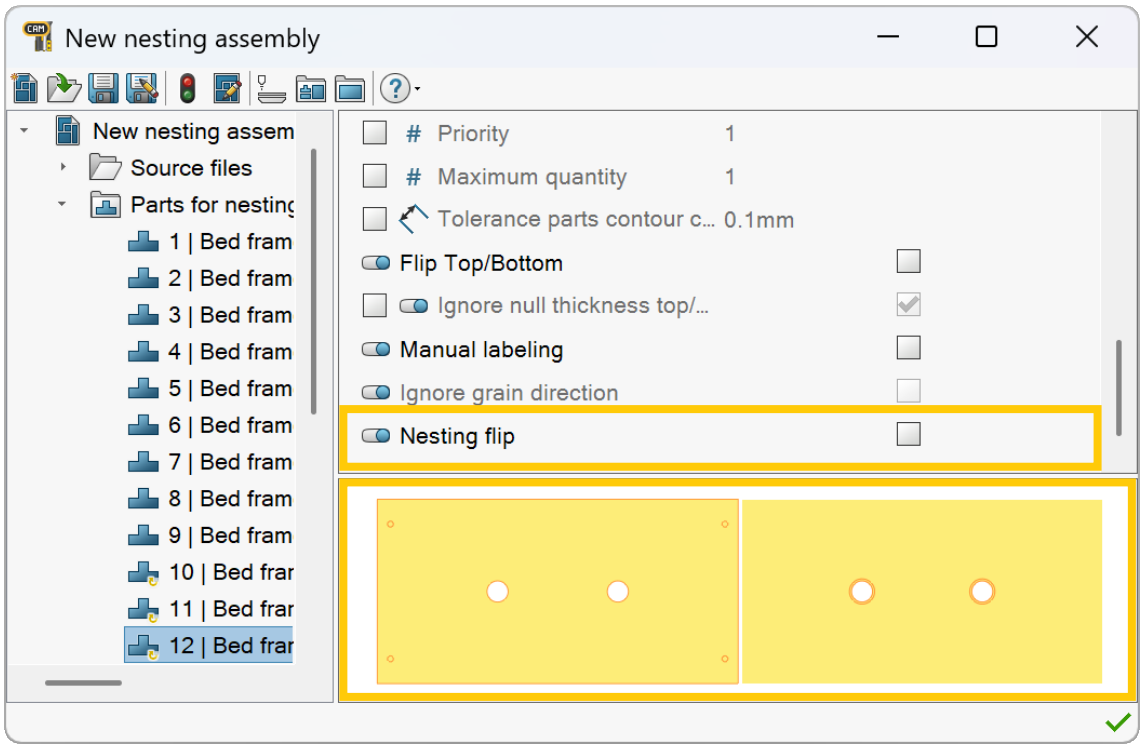


There are two ways to manually convert a candidate flip part into a flipped part:

- From the part for nesting

If a user clicks on a part for nesting that has a yellow arrow, they can see in the graphics view that instead of having a single graphic element, there are two: one for each side of the part concerned, with blind machining on both sides.

This allows the geometry of each part to be checked individually and false positives to be avoided.

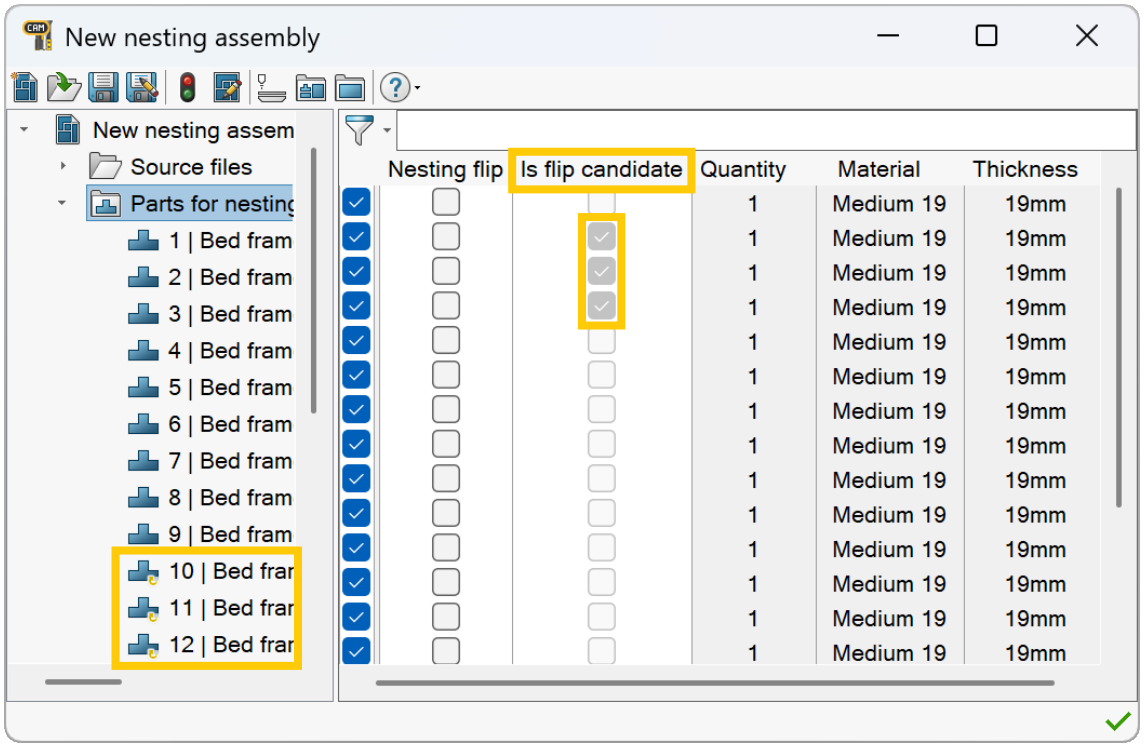


To force this part to flip, simply check the "Nesting flip" parameter.

- From the parts for nesting list

If you click on the "Parts for nesting" list, a table appears with a column labeled "is flip candidate".

In this column, parts for which the algorithm has detected opposite, non-through machining operations on the top and bottom faces are marked (and cannot be modified).
It is therefore possible to check the part(s) for which you want to force the flip in the "Nesting flip" column.



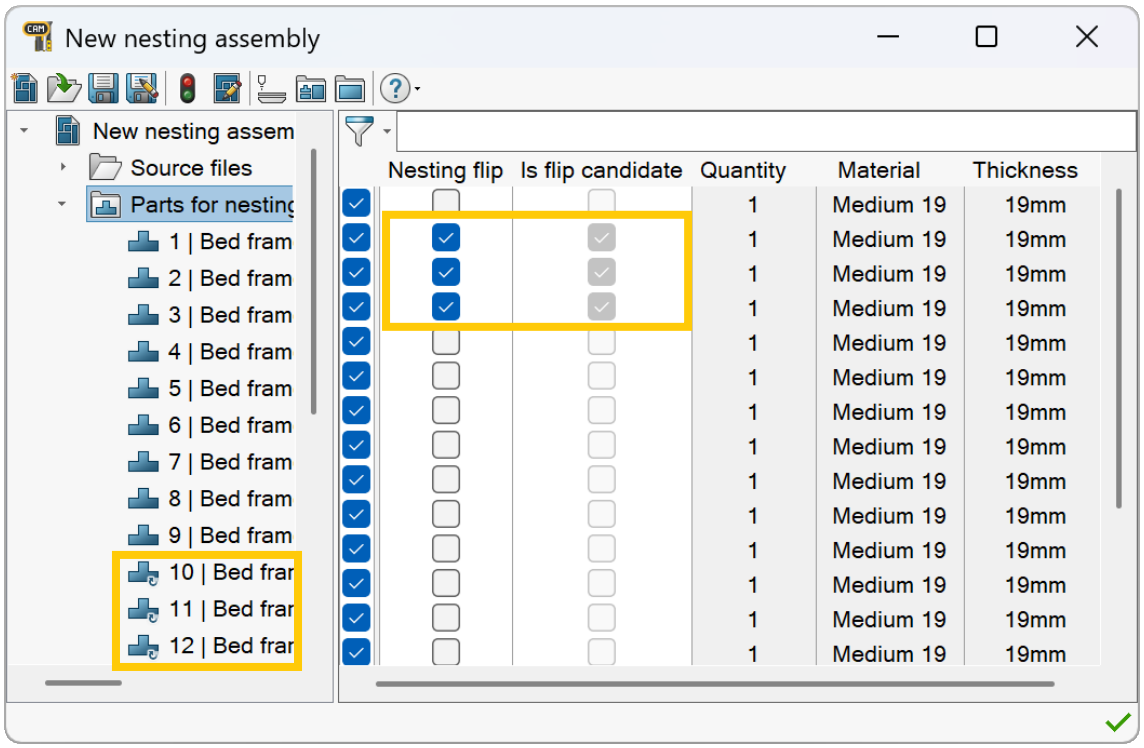
c. Automatically from the Nesting project window

For more advanced automation, it is possible to force all candidates to flip.

To do this, it is necessary to:

- Create a Nesting project (*Tools → SWOOD CAM →* *Nesting*).
- In the nesting settings specific to flipping, check the box "Flip all candidates for flip."

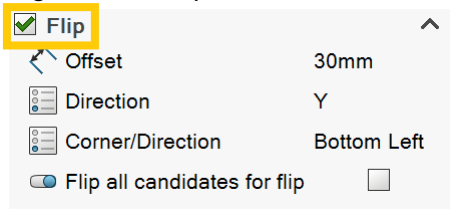
This will automatically force all candidates to flip.



3. *Enabling the flip in a Nesting project*

To activate the flip in Nesting, it is necessary to:

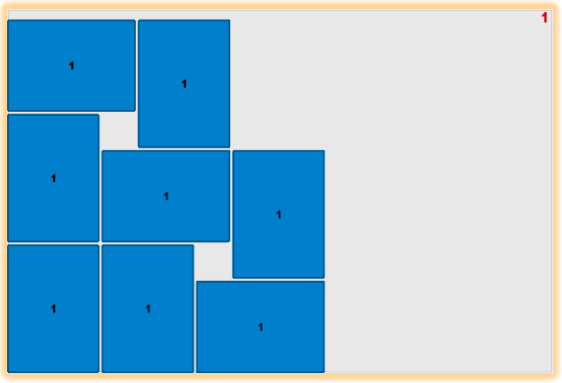
- Create a Nesting project (*Tools → SWOOD CAM →* *Nesting*).
- In the Nesting project window, go to the "Flip" section and check the activation box.



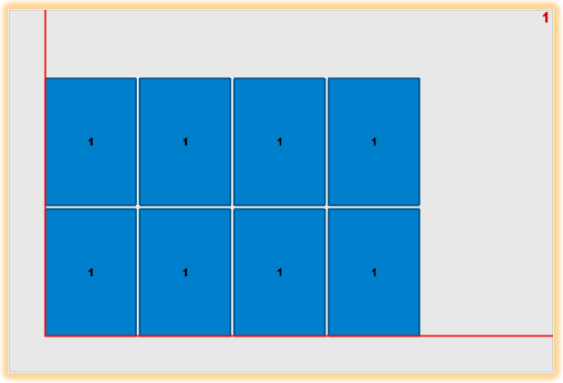
From there, board flip is active, and the flip settings are visible:

a. Offset

Reduction of the board size, in X and Y, in order to provide for board squaring machining for board flip. The positioning of the offsets depends on the **Direction** and **Corner/Direction**.



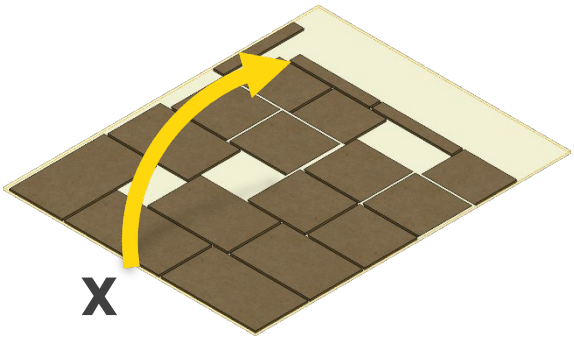
Example without offset

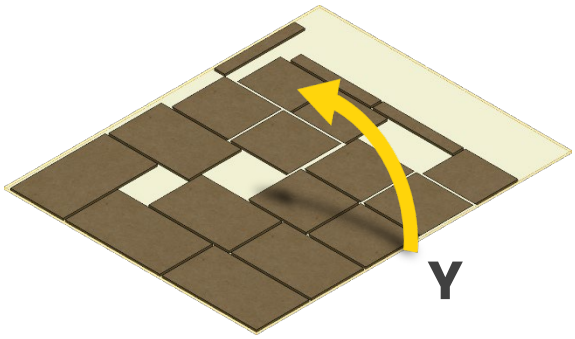


Example with a 200 mm offset

b. Direction

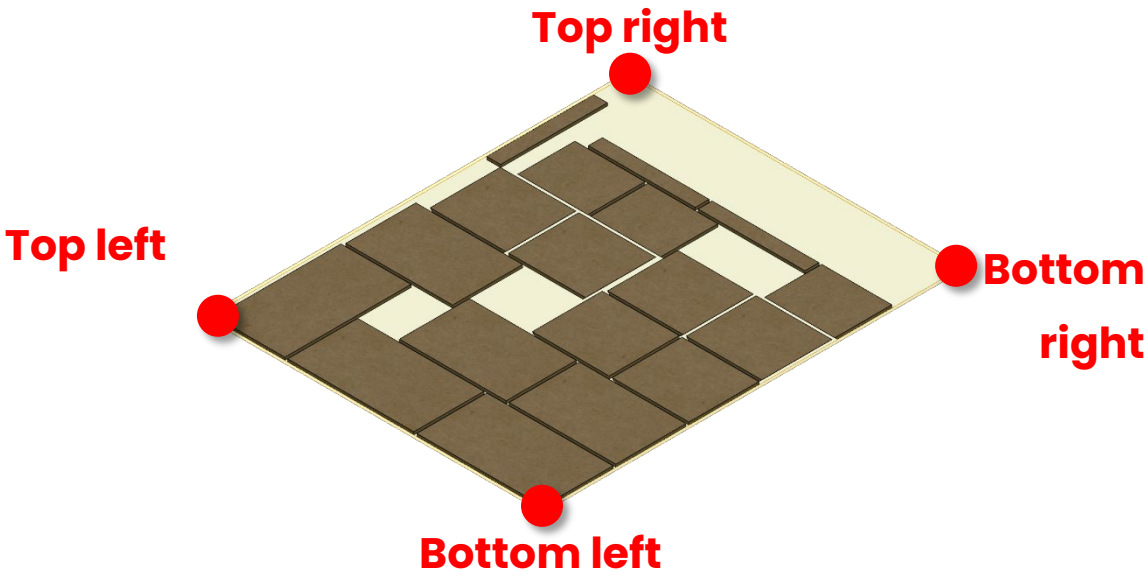
Direction of the board flip. According to **X**, the board is flipped lengthwise. According to **Y**, the board is flipped widthwise.





c. Corner/Direction

Defines the stop corner of the nesting machine for the main milling phase.



d. Flip all candidates for flip

Allows to force all candidate parts to flip at once.

Warning: some candidates may not need to be flipped, so it is important to check them.

Name	Nesting flip	Is flip candidate	Quantity	Material	Thickness
Bottom	☐	☐	1	HARDWOOD_Oak 19mm	19mm
Left	☑	☑	1	HARDWOOD_Oak 19mm	19mm
Right	☐	☐	1	HARDWOOD_Oak 19mm	19mm
Top	☑	☑	1	HARDWOOD_Oak 19mm	19mm

4. *Automatic machining insertion*

With the flip, there are now two sides to machine, instead of one previously.

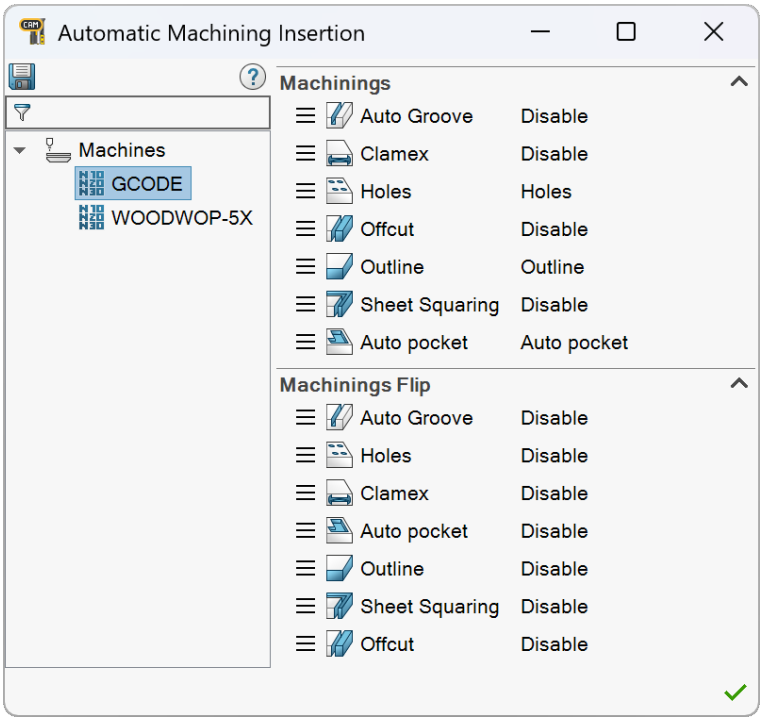
As a result, there is the possibility of adding automatic machining on both sides (before and after flipping).

a. By machine

It is possible to assign automatic machining entities on both sides, based on the choice of Post-Processor.

To do this, it is necessary to:

- • Click on the automatic machining insertion button.
- Select a Post-Processor.
- Fill in the desired machining operations on both sides.

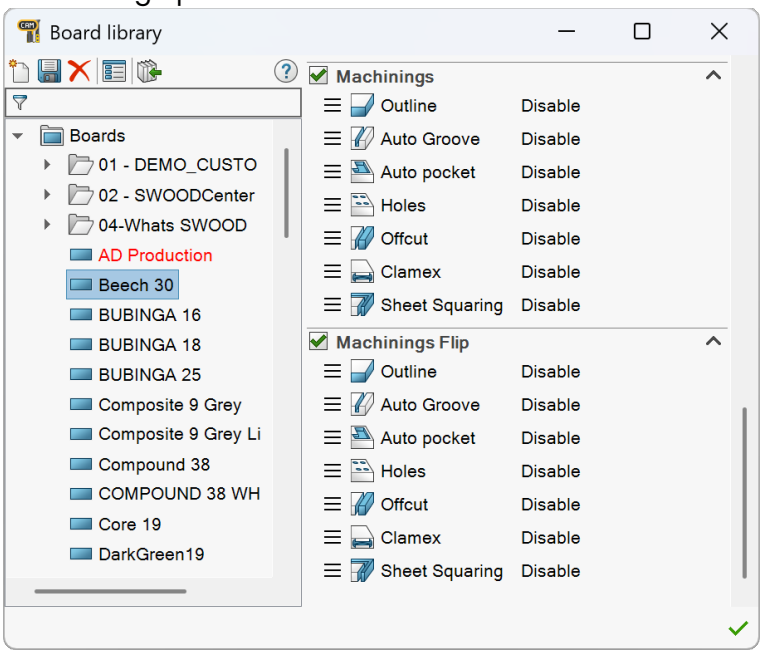


b. By board

It is possible to assign automatic machining entities on both sides, based on the choice of board used (insertion from the board will take over from insertion by machine).

To do this, it is necessary to:

- Click on the boards library button.
- Select a board.
- At the bottom of the settings, check the "Machinings" and "Machinings Flip" sections.
- Fill in the desired machining operations on both sides.

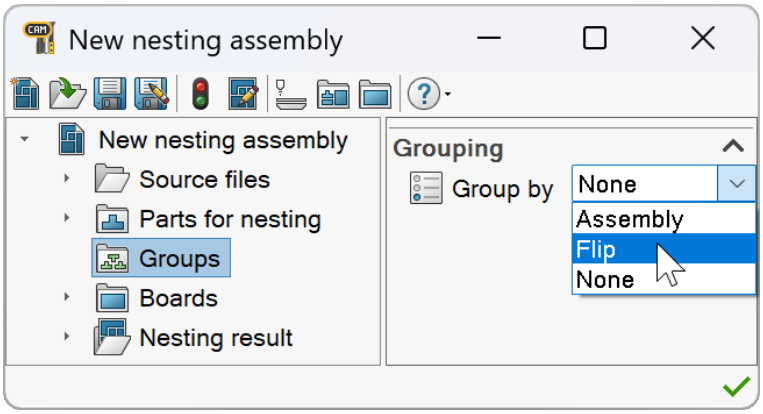


5. *Groups*

By default, pieces to be flipped can be found on several different boards (all of which will be flipped). To minimize the number of boards to be flipped, it is possible to group all the pieces to be flipped so that they are placed, ideally, on the same board.

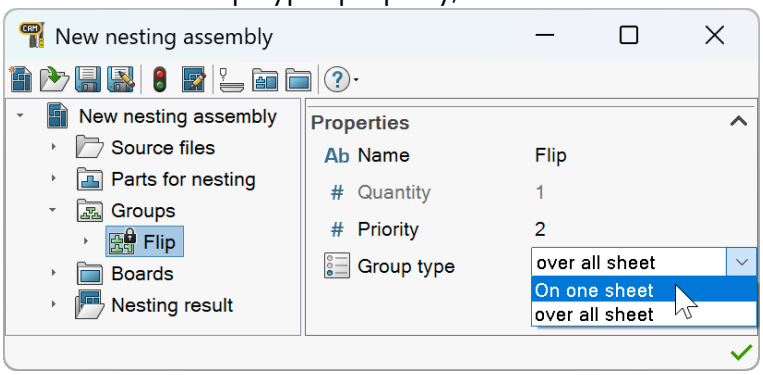
To do this, it is necessary to:

- Click on Groups.
- Select "Flip" from the "Group by" drop-down list.

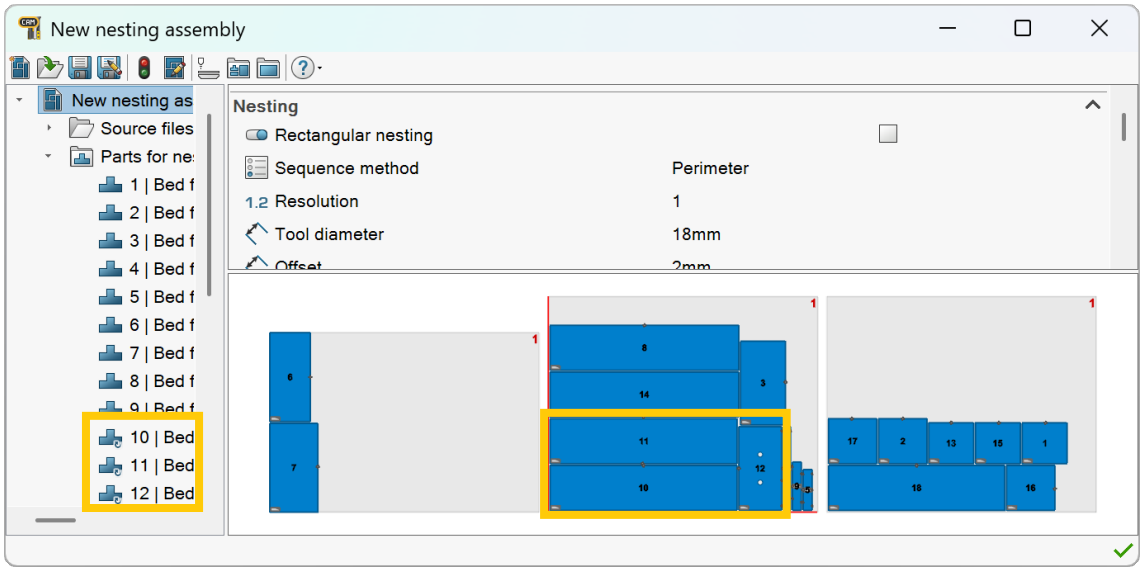


- Rebuild the Nesting for the first time.
- Expand the groups and click on Flip.

- In the drop-down list for the "Group Type" property, select "**On one sheet**".



- Rebuild the Nesting again.
- All parts to be flipped are located on a single board.



6. *Nesting result*

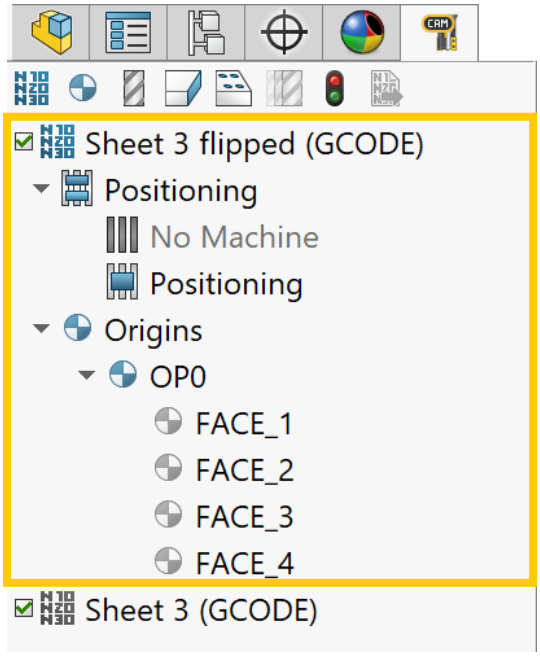
Once the nesting project meets your expectations, you can ✓ confirm the nesting project window to create the nesting assembly.

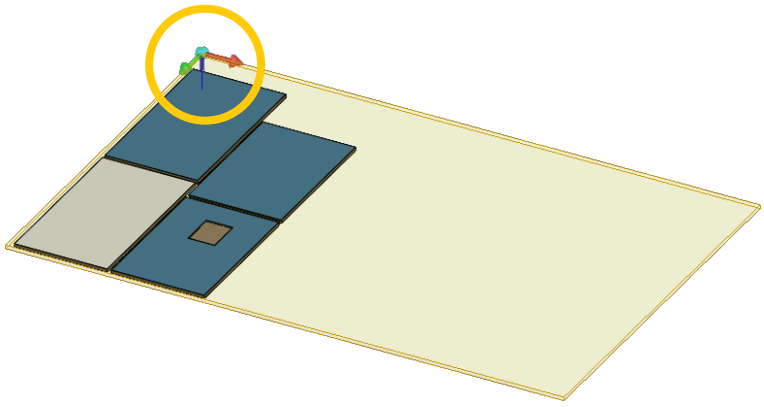
a. Milling phases

Nesting boards that are flipped have two milling phases (instead of just one for a non-flipped board). These milling phases include automatic machining operations, if they have been set at the machine or board level.

- "Sheet XX flipped" milling phase

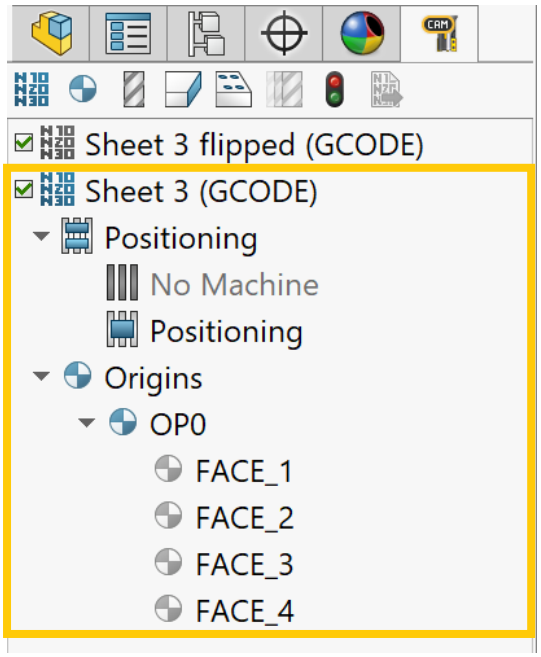
This new milling phase, which represents the flipped board, places the origin (with Z pointing downwards) according to the flip parameters (**Direction** and **Corner**).

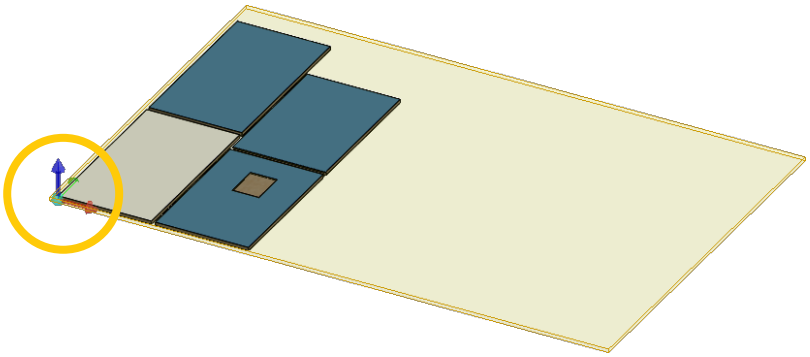




- "Sheet XX" milling phase

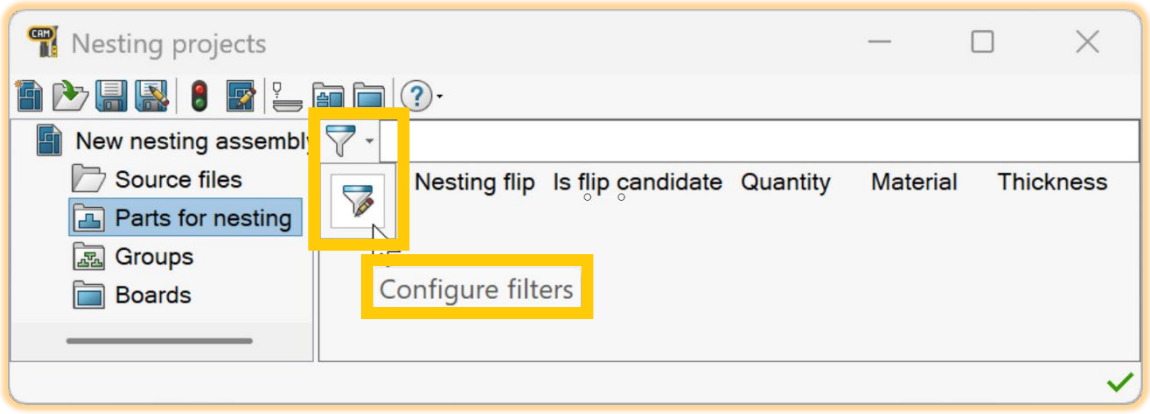
The milling phase is the same as for boards that do not have a flip. The origin is the same (with Z pointing upwards).





SWOOD Nesting – Filters: Parts for Nesting List

The cross-functional filter feature, introduced in SWOOD 2025 for greater control over element selection, now extends to the list of parts to be nested in SWOOD Nesting. This tool allows users to refine part selection before nesting by defining specific criteria based on their properties. It is thus possible to filter parts according to their materials, their flip status (whether they are candidates for flipping or whether flipping is forced), or even using the extended data and properties of the materials. For example, a user can quickly isolate all parts of a specific material that are also candidates for flipping in order to check their configuration before starting nesting.



Filters in the nesting window, in the table of parts for nesting

How to use

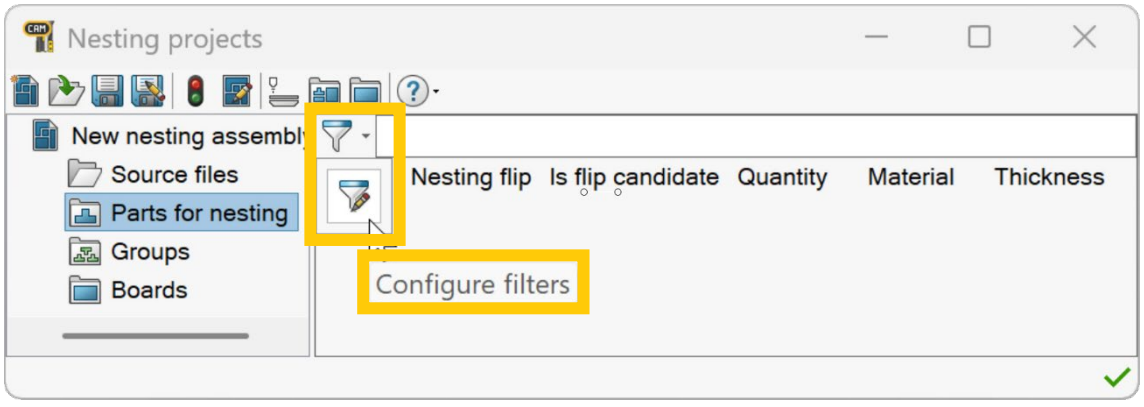
1. *Access to filters*

The Nesting window can be opened from the SolidWorks menu bar:

Tools → SWOOD CAM → Nesting

Then click on "Parts for nesting"

The filters are located at the top of the list of parts for nesting, on the right side of the window.



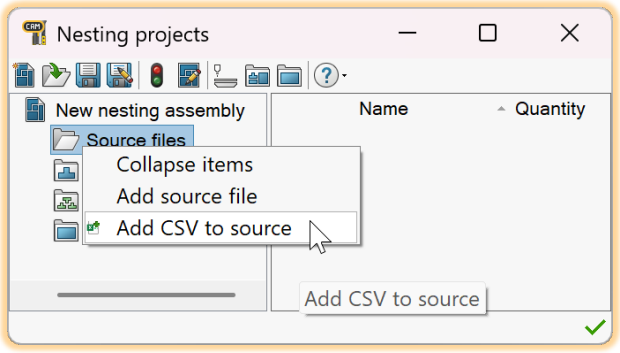
2. *Properties and attributes*

The properties and their attributes specific to the parts for nesting list are as follows:

Property	Attributes	Attribute definition	Condition type
Materials	Ø	Ø	List
Flip	Ø	Ø	Boolean
Flip candidate	Ø	Ø	Boolean
Materials data	See material attributes		
Materials Extended Properties	*Depends on extended properties*		*Depends on attributes*
Custom properties	*Depends on custom properties*		*Depends on attributes*

SWOOD Nesting – Nesting from a CSV file

SWOOD Nesting 2025 introduces a new method for creating nesting projects by directly importing parts lists from a CSV (Comma Separated Values) file. This feature is designed to simplify and speed up the creation of nesting batches, particularly when data comes from external systems, databases, or production lists generated by ERP systems. By providing a structured CSV file containing essential part information (file path and quantity), users can automate the generation of their nesting projects in SWOOD Nesting, thereby reducing manual data entry and the risk of errors.



Access to import from a CSV file

How to use

1. *Preparing the CSV file*

The CSV file must be structured with two specific columns so that SWOOD Nesting can interpret the data correctly. The required columns and their order are crucial.

a. CSV file structure

- Column 1: Absolute file path
  - Contains the full path to the SolidWorks part or assembly file (.SLDPRT or .SLDASM) to be nested.
  - Must be written between quotation marks " ".
- Column 2: Quantity
  - Contains the number of copies of this part/assembly to be nested.

b. Format

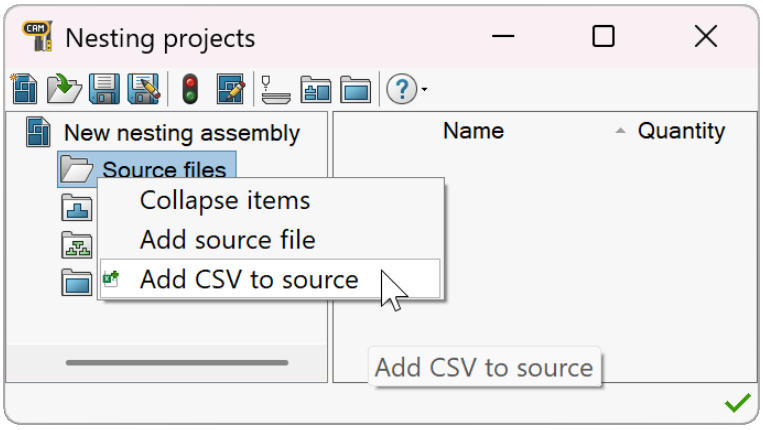
- Must contain **two columns**.
- Use a **semicolon** (;) as a column separator.
- Use **UTF-8** encoding.

c. Example of CSV file content

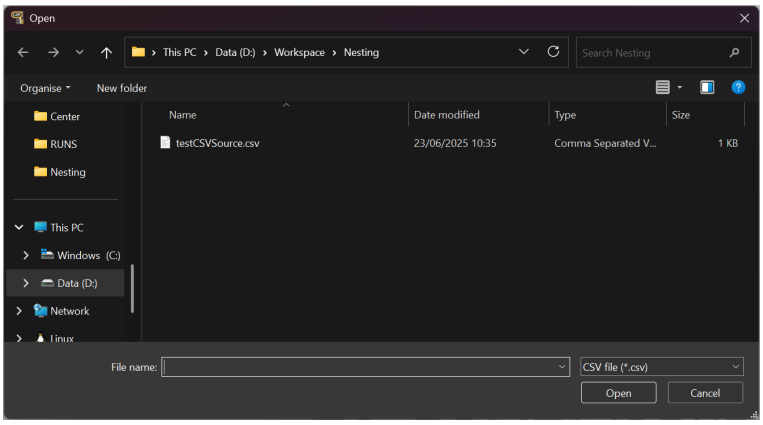
```
"D:\Workspace\Center\Part1.SLDPRT";6
"D:\SWOODData\SWOODDesign\Panels\WIP\Generic_WIP.sldprt";3
```

2. *Access to the CSV import feature*

- Open the Nesting Projects window (the window where you manage your nesting projects).
- Right-click on the Source Files folder and select "Add CSV to Source."

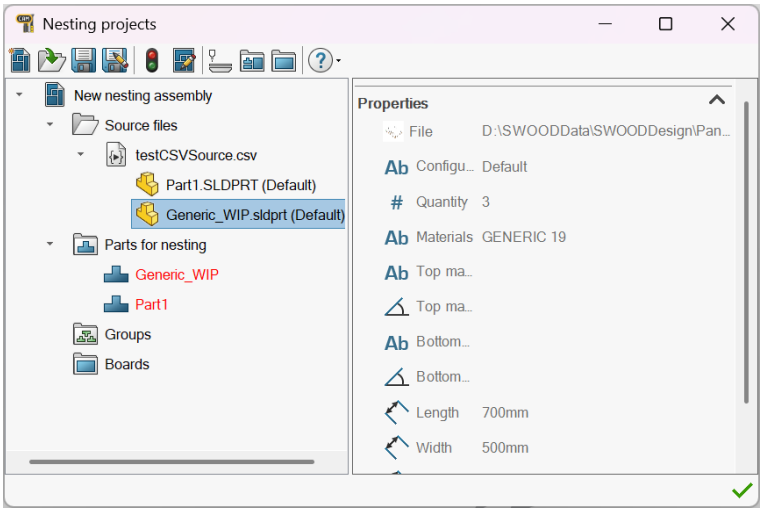


- A file open dialog box will appear. Navigate to the location of your prepared .csv file and select it.
- Click "Open."



3. *Generation of the nesting project*

- SWOOD Nesting will read the information from the CSV file.
- For each line in the CSV file, SWOOD Nesting will locate the part specified by the absolute path of the file.
- The nesting project will be updated, containing the parts listed with the specified quantities.


Nesting projects
New nesting assembly
Source files
testCSVSource.csv
Part1.SLDPRT (Default)
Generic_WIP.sldprt (Default)
Parts for nesting
Generic_WIP
Part1
Groups
Boards
Properties
File D:\SWOODData\SWOODDesign\Pan...
Configu... Default
Quantity 3
Materials GENERIC 19
Top ma...
Top ma...
Bottom...
Bottom...
Length 700mm
Width 500mm

System Report – New Activation

The SWOOD System Report is a central tool for consolidating and visualizing project data. With SWOOD 2025, this feature has been significantly improved to increase its flexibility, power, and ease of use. These improvements include the method for activating and conditioning its generation, visualization options, sharing reports within the company via a dedicated server, and customizing the display using profiles. The overall objective is to provide a more powerful reporting tool that is better suited to the diverse needs of users. We will begin by detailing the new procedure for activating and configuring the generation of the System Report.

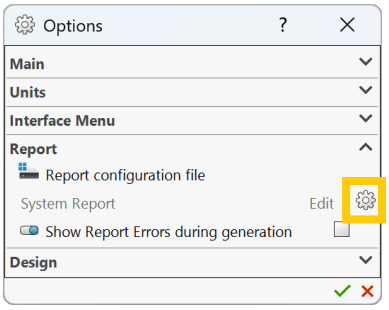
SWOOD Report Viewer icon

How to use

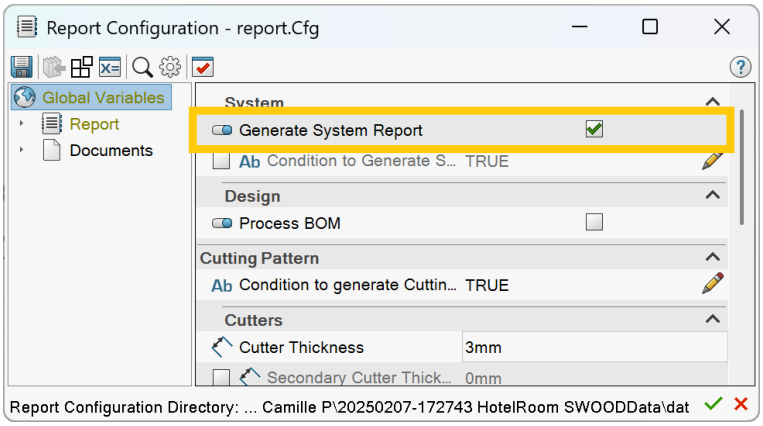
1. *Enabling system report generation*

The System Report is now activated via the report configuration:

- Access the SWOOD software "Settings" In the SolidWorks menu bar, click on:
  - Tools
  - SWOOD Design or SWOOD CAM
  - Click on Settings
- In the "Report" section, click on the gear icon next to "System Report".



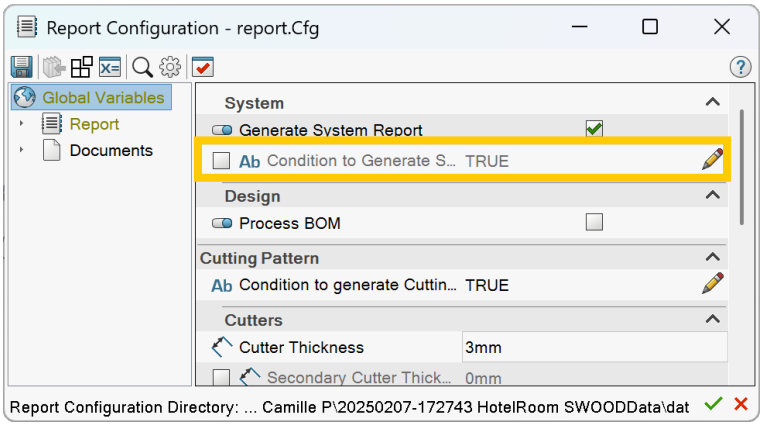
- In the "Report Configuration" window that opens, under Global Variables, check the option: "Generate System Report"



2. *System report generation conditions*

By default, the condition for generating the system report is set to "TRUE." This means that the report is generated automatically as soon as the "Generate system report" option is checked.

If the "Condition to generate the system report" option is enabled (the checkbox to its left is active), the generation condition can be modified. This condition uses the PROJECT variables and logical operators to build the conditional syntax.

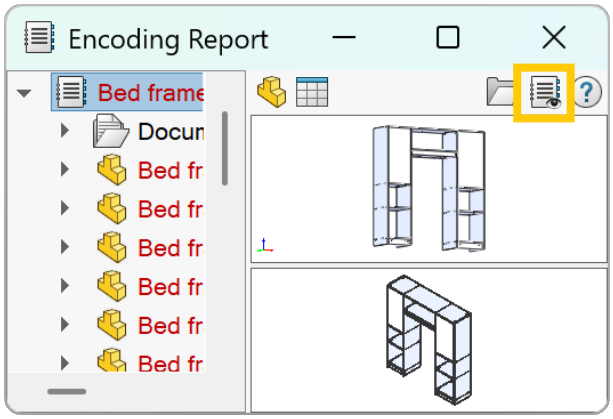


Warning: Only "PROJECT" variables can be used to define the condition for generating the system report.

3. *Generate and view the system report results*

When the System Report is generated in SWOOD 2025, only a compressed data file in **.swr** format (optimized for speed) is created for the System Report itself. This file is no longer listed among the "documents" in the legacy report results window.

Once the system report has been generated, the results can be viewed by clicking on the eye button (the "Report Viewer" icon) that appears in the report generation results window.



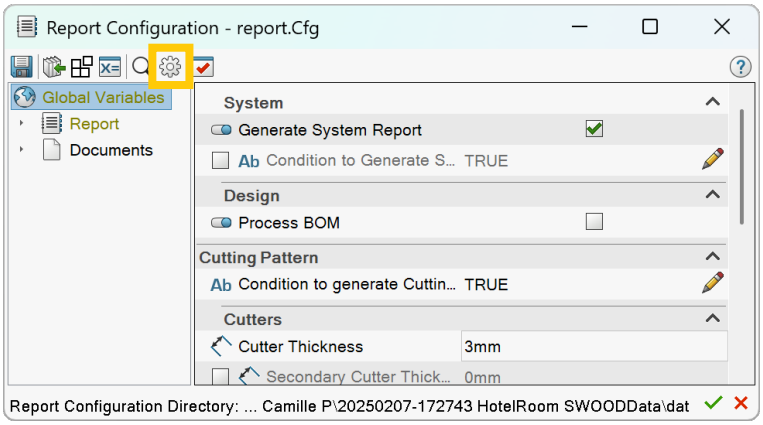
4. *Choosing how to view the report results*

You can choose how the results window is displayed at the end of report generation. Two modes are available:

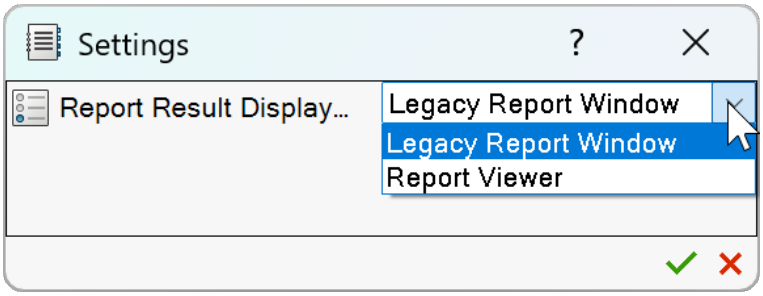
- **Legacy Report Window:** The legacy report window appears, displaying all report objects, variables, and a button to display the System Report (via the "Report Viewer").
- **Report Viewer:** A new report window appears, displaying only the results of the System Report.

To configure this option, it is necessary to:

- Access the SWOOD software "Settings". In the SolidWorks menu bar, click on:
  - SWOOD Design or SWOOD CAM
  - Click on Settings
- In the "Report" section, click on the gear icon next to "System Report."
- In the "Report Configuration" window that opens, click on the "Settings" icon.

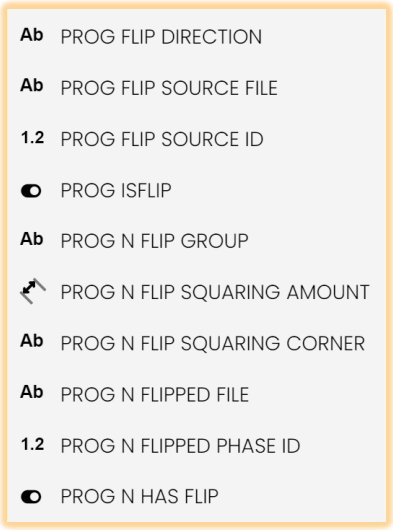


- In the "Settings" window that opens, select the desired mode from the "Report Result Display Options" drop-down list (choose between "Legacy Report Window" and "Report Viewer"), then confirm.


Settings
Report Result Display...
Legacy Report Window
Legacy Report Window
Report Viewer

System report – New variables related to the flip

The introduction of the board flip feature in SWOOD Nesting 2025 is followed by the creation of new dedicated variables within the System Report. These variables are intended to enrich the information available, specifically on the "Programs" page, in order to enable more accurate monitoring and traceability of flip-related operations. They can be used to create new information columns, filters, or groupings, providing better visibility of flip parameters such as direction, stop corner, or squaring offset. For example, a user can now easily list or filter all milling programs that involve flipped sheets or identify the specific parameters applied during the squaring phase.



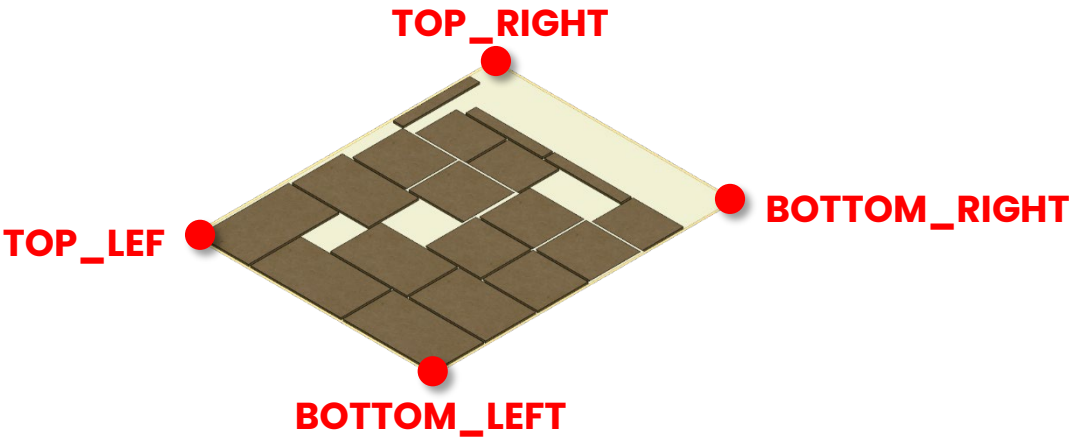
List of new variables related to the flip

Variable definitions

The new variables related to the flip can only be used in the "**Programs**" page of the system report to create new columns and add information. They can also be used to create filters or groupings on the same page.

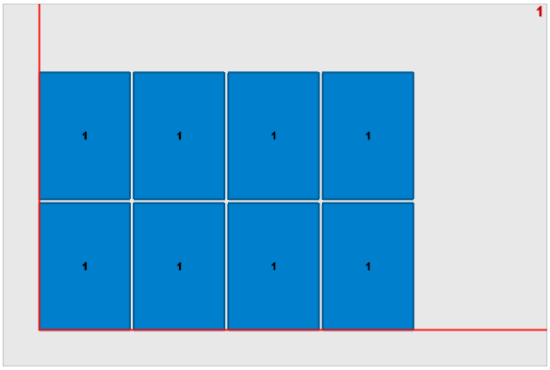
5. Ab *PROG_N_FLIP_SQUARING_CORNER*

The text variable **PROG_N_FLIP_SQUARING_CORNER** corresponding to "**Corner/Direction**" returns the stop corner of the nesting machine for the main milling phase.



6. *PROG_N_FLIP_SQUARING_AMOUNT*

The dimension variable **PROG_N_FLIP_SQUARING_AMOUNT** corresponds to the "**Offset**" and returns the distance between the edge of the board and the Sheet Squaring machining operation.
The positioning of the offsets depends on the **Direction** and the **Corner/Direction**.



7. 1.2 *PROG_N_FLIPPED_PHASE_ID*

The numerical variable **PROG_N_FLIPPED_PHASE_ID** is a variable that only applies to classic milling phases.
It returns the number of the associated flipped milling phase.

8. *PROG_N_HAS_FLIP*

The Boolean variable **PROG_N_HAS_FLIP** is a variable that only applies to classic milling phases.
If PROG_N_HAS_FLIP is **Yes**, the milling phase is on a board that **contains** a flip milling phase.
If PROG_N_HAS_FLIP is **No**, the milling phase is on a board that **does not contain** a flip milling phase.

9. *PROG_ISFLIP*

The Boolean variable **PROG_ISFLIP** defines the type of milling phase.
If PROG_ISFLIP is **Yes**, the program comes from a flip milling phase.
If PROG_ISFLIP is **No**, the program comes from a classic milling phase.

10. 1.2 *PROG_FLIP_SOURCE_ID*

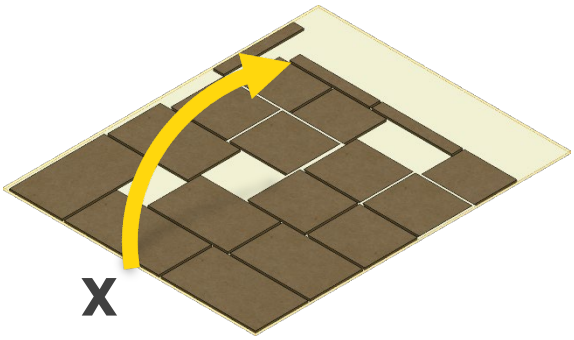
The numerical variable **PROG_FLIP_SOURCE_ID** is a variable that only applies to flip milling phases.
It returns the number of the classic milling phase associated with this flip.

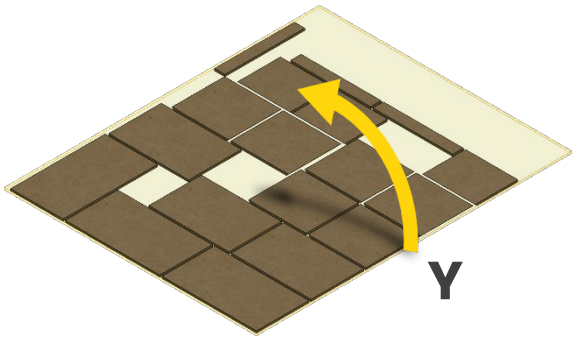
11. Ab *PROG_FLIP_DIRECTION*

The text variable **PROG_FLIP_DIRECTION** corresponding to "**Direction**" returns the direction in which the board is flipped.

On **X**, the board is flipped along its length.

On **Y**, the board is flipped along its width.





12. Ab *PROG_N_FLIP_GROUP*

The text variable **PROG_N_FLIP_GROUP** is a variable that defines the pair: classic phase / flipped phase. (<source>-<flip>).

For example, on a flipped board, if the ID of the classic phase is 1 and the ID of the flipped phase is 2, then the PROG_N_FLIP_GROUP variable will return "**1-2**".

13. Ab *PROG_N_FLIPPED_FILE*

The text variable **PROG_N_FLIPPED_FILE** is a variable that only applies to classic milling phases.

It returns the file path of the linked flip program (if it exists).

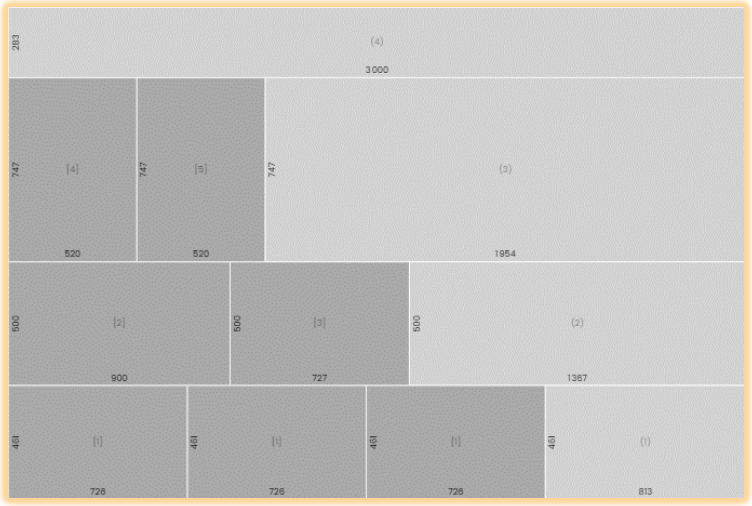
14. Ab *PROG_FLIP_SOURCE_FILE*

The text variable **PROG_FLIP_SOURCE_FILE** is a variable that only applies to flip milling phases.

It returns the file path of the classic program linked to this flip phase.

System Report – Cutting Pattern New Features

The cutting pattern generated by SWOOD's System Report provides an estimate of the number of boards required and suggests an optimized cutting strategy for table saws. SWOOD 2025 enhances this functionality by introducing two new parameters that further refine this optimization and better adapt it to production constraints. For example, a workshop can now require that the first cuts on a board always be made lengthwise to suit its machine, or limit the number of band rotations to simplify handling and speed up the cutting cycle.



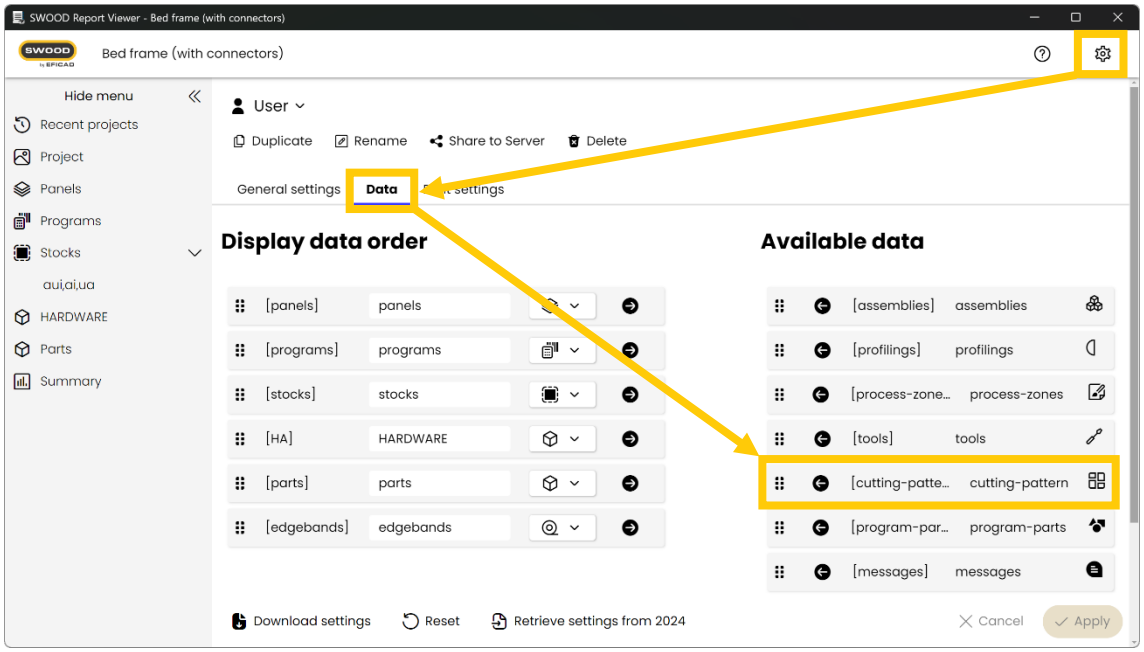
Example of a cutting pattern with a horizontal full cut strategy

How to use

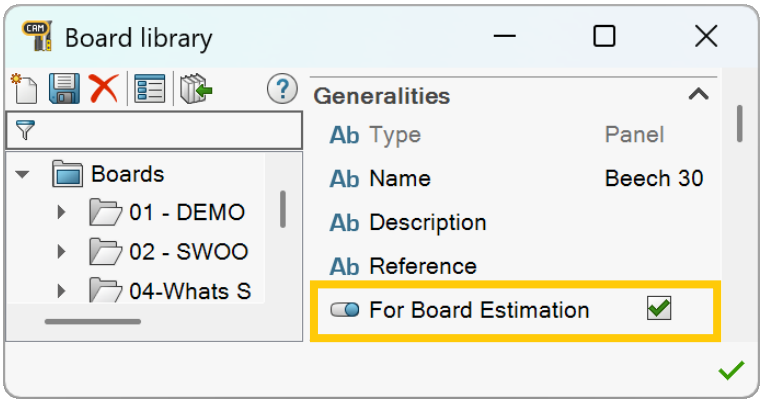
1. *Prerequisites for cutting pattern generation*

The conditions for generating the "Cutting pattern" sheet remain the same as for SWOOD 2024. The following conditions must be met:

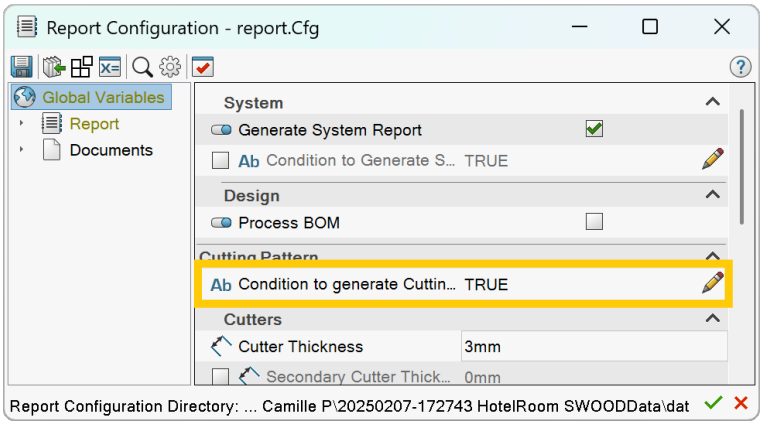
- The **Report System must be enabled** (see previous chapter on activation).
- The "Cutting Pattern" sheet is **added to the System Report view** that you generate.



- The "**For Board Estimation**" option is checked in the board library.



- The **condition for generating** the "Cutting Pattern" sheet (in the report configuration) is set to **TRUE** (or a logical condition that evaluates to TRUE).



- The condition **CUTTING_PATTERN_IGNOREPART** is set to **FALSE** (which is the default). This variable allows you to exclude certain parts from the cutting pattern if necessary.

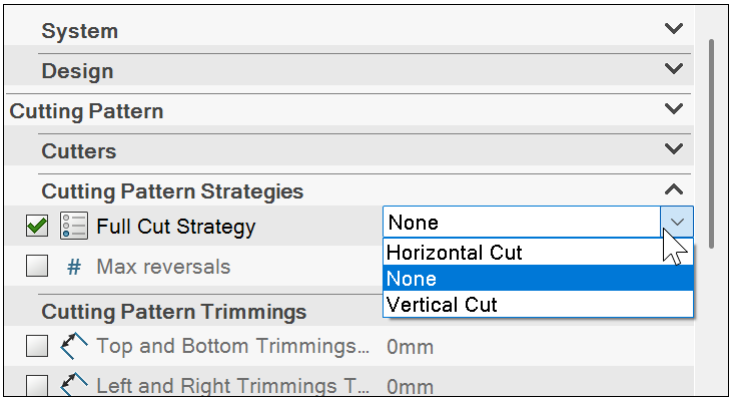
2. *New cutting pattern parameters in SWOOD 2025*

These new parameters are configured in the global variables of the report, within the "Report Configuration" window, in the "Cutting Pattern Strategies" subsection.

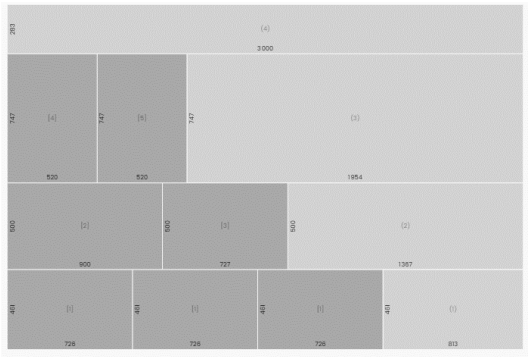
a. Full Cut Strategy

This parameter allows you to set a priority for the direction of the first cuts on the board, across its entire size.

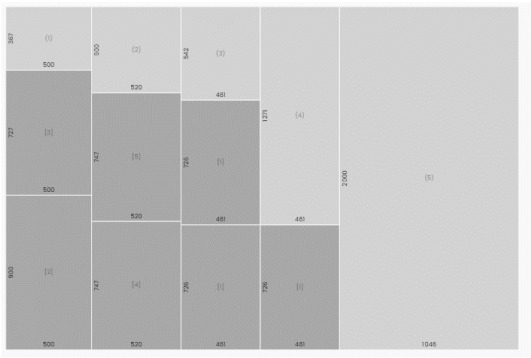
- **Principle:** It is possible to specify whether cuts should be made primarily in the horizontal or vertical direction of the board.
- **Options:**



  - **None:** There will be no full cut strategy.
  - **Horizontal section:** The first large cuts will be mostly parallel to the **length** of the board.



  - **Vertical cut:** The first large cuts will be mostly parallel to the **width** of the board.



- **Use:** This choice may be dictated by the capabilities of the saw, the grain direction for certain optimizations, or workshop practices.

b. **Max reversals**

This parameter limits the number of times a band (or a piece resulting from a cut) will need to be rotated during the cutting process.

- **Principle:** By controlling this number (minimum 1), it is possible to simplify operations in the workshop.
- **Objective:** Reduce overall cycle time by minimizing rotations, which can be time-consuming and sources of error.
- **Use:** Particularly useful for workshops that want to standardize and speed up their manual or semi-automated cutting processes.

System Report – « Group By »

The SWOOD 2025 System Report improves data visualization and analysis by introducing a grouping feature. It is now possible to group rows in a table according to the value of a specific variable, providing a more concise and organized view of the information. This capability is particularly useful for analyzing large datasets, such as viewing all panels belonging to the same product or all parts of a specific nesting program. This enables a better understanding of project structures and facilitates the identification of trends or the consolidation of information.

"Group by" button in a data table

How to use

To illustrate how grouping works, we will use the example of panels grouped by product.

You can download the sample source files:

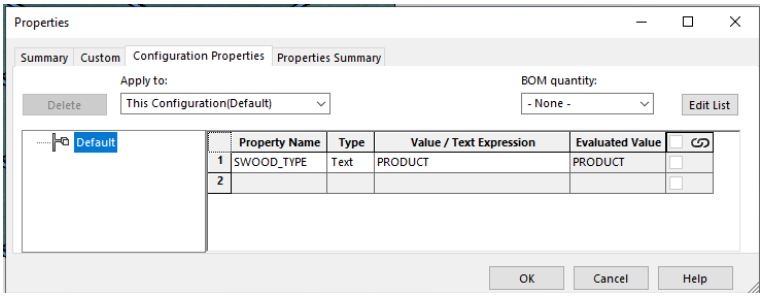
- Report configuration
- Sample model (AsmBOM)

You can also watch the steps in this example by following this video.

1. *Prerequisite: Assigning a Group by Property*

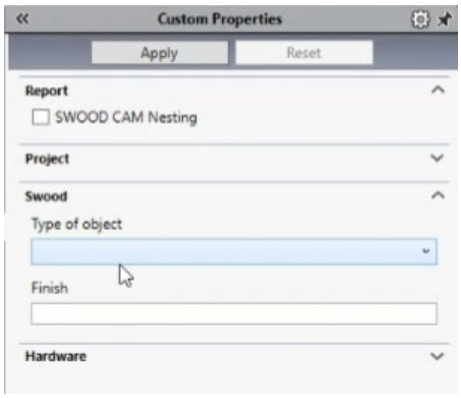
To be able to group items (such as panels) by a logical entity (such as a "Product"), you must first assign a custom property that will serve as the grouping criteria.

a. In SolidWorks
   - Open the relevant subassembly or part.
   - Access the file's custom properties (the "Custom" or "Configuration-specific" tab, as applicable).
   - Create a new property. For example:
     - **Property Name:** SWOOD_TYPE
     - **Type:** Text
     - **Value / Text Expression:** PRODUCT (or the name of the specific product, e.g., "Upper_Kitchen," "Lower_Office").



- Repeat this process for all assemblies/parts that you wish to identify as separate products.

Note: Using a SolidWorks property form can also make it easier to assign this value.



2. *System Report Configuration*

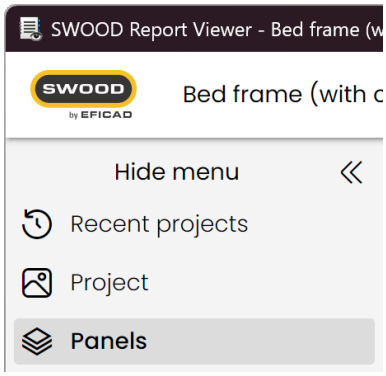
Ensure that your report configuration file is correctly defined in the SWOOD settings. This configuration file must include the view and variables required to display the panels and their SWOOD_TYPE property.

3. *System Report Generation*

- Open the main assembly model.
- Generate the System Report.

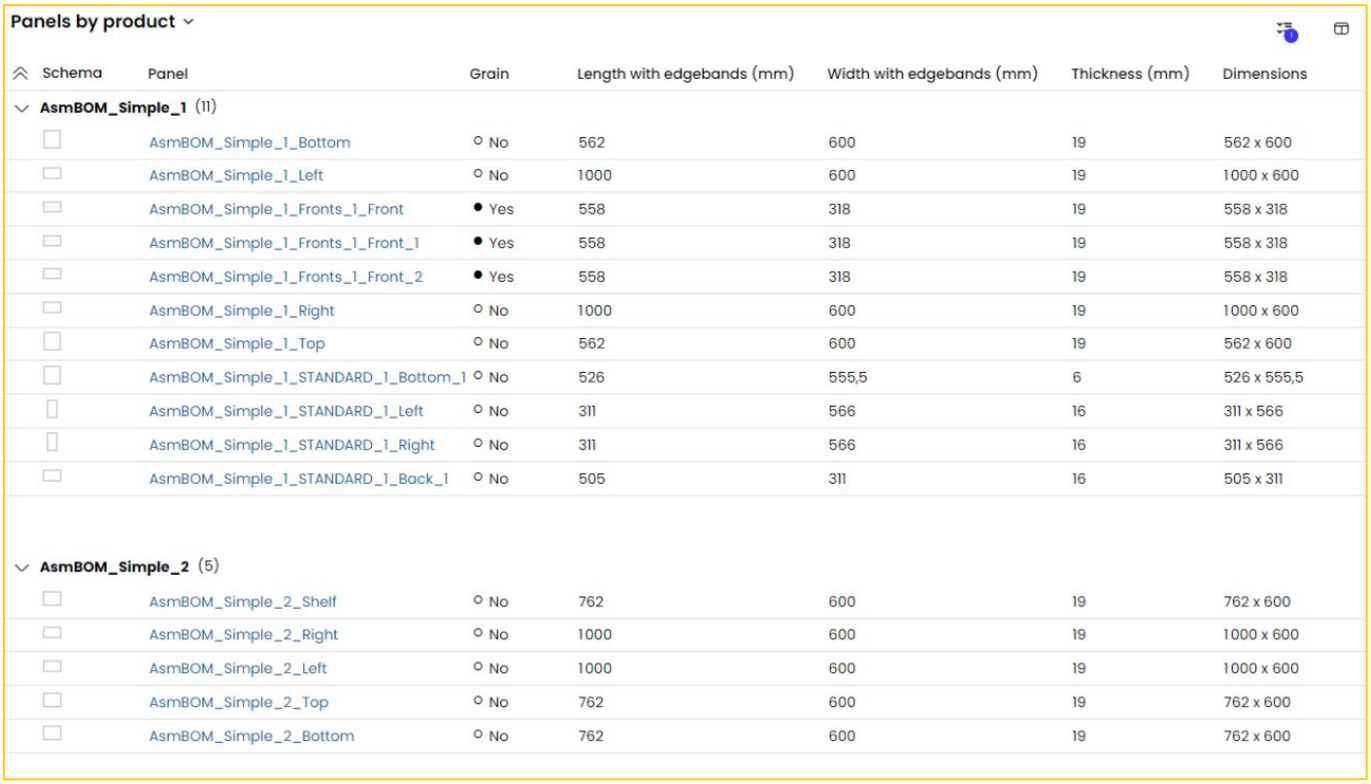
4. *Applying the "group by" in the System Report interface*

- Once the System Report has been generated and displayed (for example, in the report viewer), navigate to the table containing the list of panels.



- Click on the "Group by" button.

- Select the variable that will be used as the grouping criteria. In our example, this would be the variable corresponding to SWOOD_TYPE (the exact name in the list may vary, for example TO_PRODUCT_NAME or the name of the column that displays this value).
- The table will then be reorganized, displaying the panels grouped under headers corresponding to the different values of the grouping property (for example, "PRODUCT: AsmBOM_Simple_1").



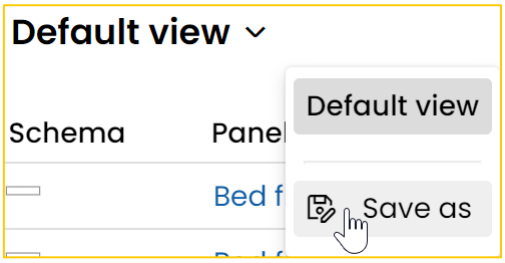
Panels by product

Schema	Panel	Grain	Length with edgebands (mm)	Width with edgebands (mm)	Thickness (mm)	Dimensions
AsmBOM_Simple_1 (11)						
	AsmBOM_Simple_1_Bottom	No	562	600	19	562 x 600
	AsmBOM_Simple_1_Left	No	1000	600	19	1000 x 600
	AsmBOM_Simple_1_Fronts_1_Front	Yes	558	318	19	558 x 318
	AsmBOM_Simple_1_Fronts_1_Front_1	Yes	558	318	19	558 x 318
	AsmBOM_Simple_1_Fronts_1_Front_2	Yes	558	318	19	558 x 318
	AsmBOM_Simple_1_Right	No	1000	600	19	1000 x 600
	AsmBOM_Simple_1_Top	No	562	600	19	562 x 600
	AsmBOM_Simple_1_STANDARD_1_Bottom_1	No	526	555,5	6	526 x 555,5
	AsmBOM_Simple_1_STANDARD_1_Left	No	311	566	16	311 x 566
	AsmBOM_Simple_1_STANDARD_1_Right	No	311	566	16	311 x 566
	AsmBOM_Simple_1_STANDARD_1_Back_1	No	505	311	16	505 x 311
AsmBOM_Simple_2 (5)						
	AsmBOM_Simple_2_Shelf	No	762	600	19	762 x 600
	AsmBOM_Simple_2_Right	No	1000	600	19	1000 x 600
	AsmBOM_Simple_2_Left	No	1000	600	19	1000 x 600
	AsmBOM_Simple_2_Top	No	762	600	19	762 x 600
	AsmBOM_Simple_2_Bottom	No	762	600	19	762 x 600

Note: The interface may display the name of the grouping property followed by the name of the grouped item and the number of items in that group (e.g., TO_PRODUCT_NAME: AsmBOM_Simple_1 (**11**)).

5. *Saving the view (Optional)*

If you want to keep this grouped view for future reference, you can save it (for example, under the name "Panels by product").



System Report – Column Data Aggregation

In addition to the grouping feature, SWOOD 2025 allows you to perform aggregation calculations directly on column values within the System Report tables. This capability provides a quick way to obtain summary information such as the total number of parts, the sum of certain dimensions, the average of numerical values, or the count of specific values. Whether the table is grouped or not, these calculations provide key indicators without requiring data export or external processing. For example, a user can quickly find out the total number of panels in a project or the average length of parts of a certain type.

Bed frame (with connectors)_Front_1_Single Front_2	No	533,89	989	19
Bed frame (with connectors)_Shelf	No	454,428	500	19
Count 18	No 18/18	Average 1004,035	Min 80	Max 19

Example of multiple aggregated columns in a table

How to use

To illustrate the use of data aggregations, we can use the example of the "Panels by product" view created earlier, or use any other table from the System Report.

You can download the sample source files:

- Report configuration
- Sample model (AsmBOM)

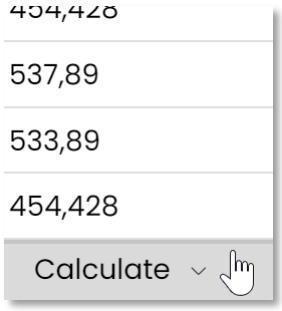
You can also watch the steps in this example by following this video.

1. *View preparation (if a specific context is desired)*
   - Open the sample template (AsmBOM.zip).
   - Define the report on the desired configuration (Report2025.Cfg).
   - If you want to perform aggregations on specific groups, first apply the "Group by" feature as described in the previous chapter to obtain, for example, the "Panels by product" view.

Important note: The aggregation feature is available on any table, whether grouped or not. Using a grouped view allows you to obtain aggregations by group in addition to aggregations across the entire table.

2. *Access to the aggregation feature*
   - In the System Report visualization interface (for example, the report viewer), display the table on which you want to perform calculations.
   - Move the mouse pointer below the last row of the table, in the column for which you want to perform a calculation.

- An option labeled "Calculate" will appear in this cell.



3. *Choice of the type of aggregation calculation*

Once you click on the "Calculate" option, a list of options will open, offering different types of aggregation calculations.

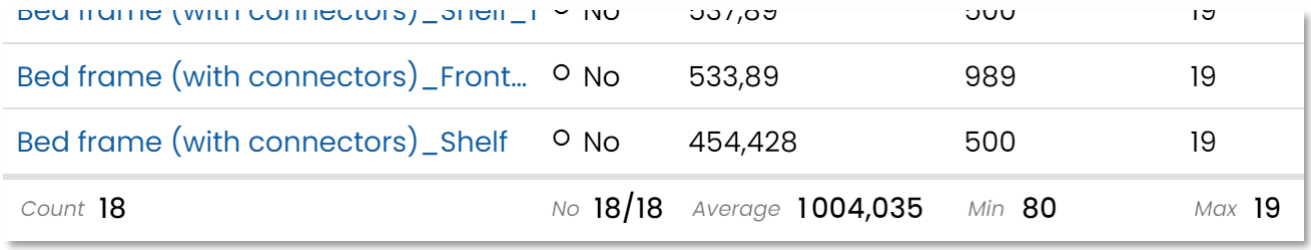
The options available depend on the type of data in the selected column (Boolean, numeric, text, etc.).

4. *Examples of possible calculations (non-exhaustive list)*

Boolean column (Yes/No)	Digital column	Text column
None	None	None
Count	Count	Count
Yes	Average	
No	Sum	
	Min	
	Max	

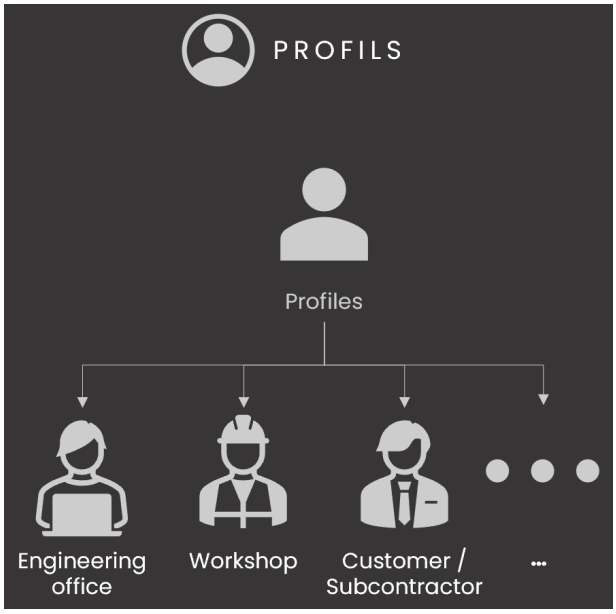
5. *Result display*

Once the calculation type has been selected, the aggregation result will be displayed at the bottom of the corresponding column. If the table is grouped, aggregations may also appear for each group.



System Report – User Profiles

SWOOD 2025 enriches the user experience of the System Report by introducing the concept of Profiles. This feature allows you to define and save custom display configurations tailored to the specific needs of different users or departments within the company. A profile can thus determine the available views, the columns displayed in tables, the groupings applied, and other display options. For example, a "Design Office" profile could focus on cost and assembly information, while a "Workshop" profile could focus on parts lists, schedules, and cutting patterns, giving each user direct and relevant access to the data they need.



Schematic representation of different user profiles

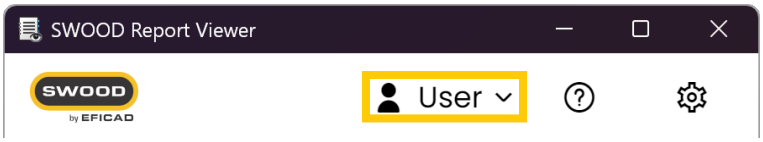
How to use

You can watch the steps in this example by following this video.

1. *Profile Concept*
   - A profile saves a set of System Report display settings (active views, table configuration, groupings, filters, etc.).
   - Each user can switch between different profiles to adapt the data display to their current task.
   - Profiles can be created, modified, shared, and managed directly from the **Report Viewer** interface.

2. ***Profile Management in the Report Viewer***

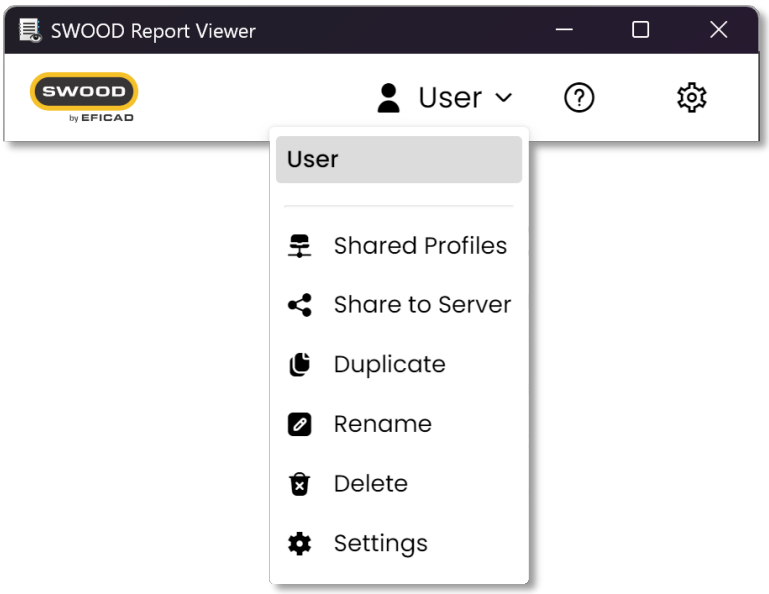
The profile-related commands are accessible from the top banner of the **Report Viewer**, on the right, next to the active profile drop-down list.



3. *Actions available on Profiles*

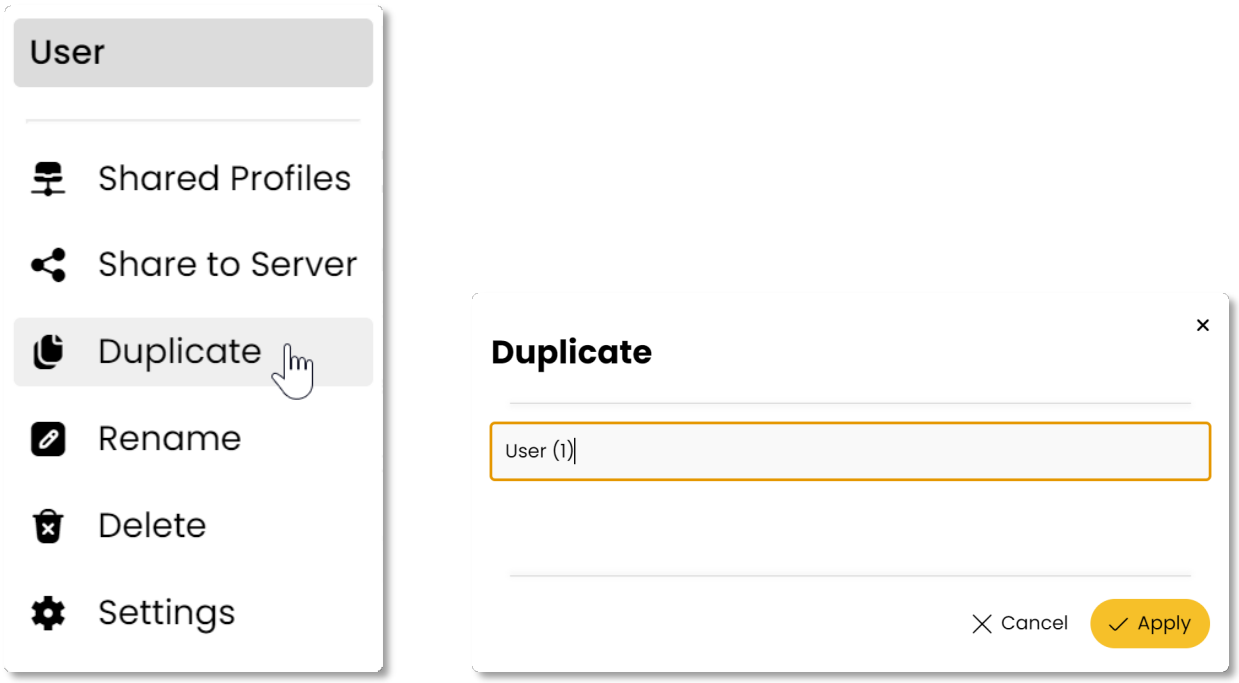
a. Select an existing profile

- Click on the drop-down list displaying the name of the active profile.
- Select the desired profile from the list to apply it instantly.



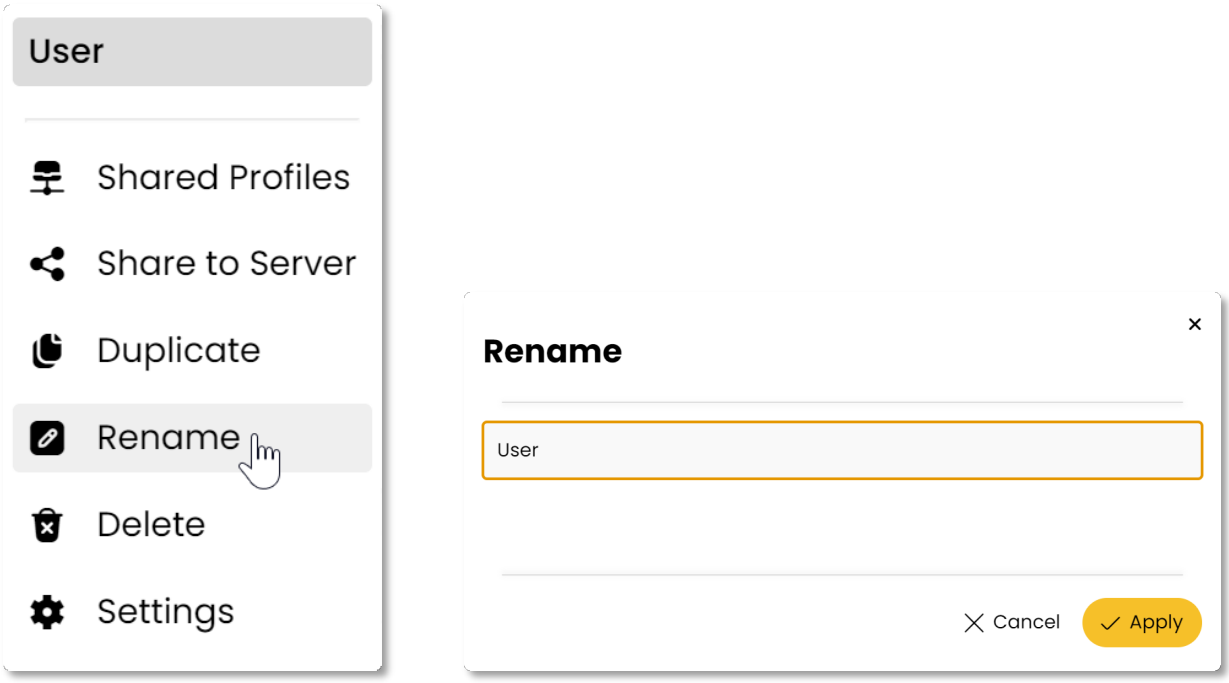
b. Create a new profile (by duplicating an existing profile)

- Display the profile you want to use as a basis.
- Select the "**Duplicate**" action.
- Give the new profile a name. It will inherit the settings from the original profile, which you can then modify.



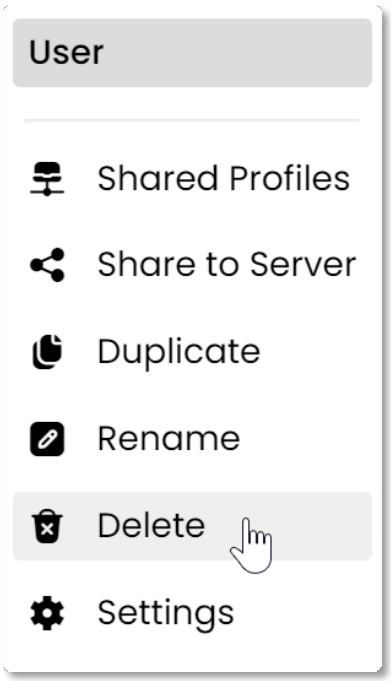
c. Renaming a profile

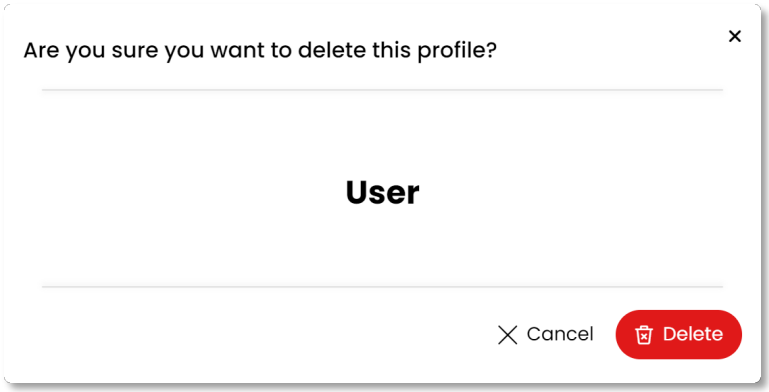
- Access actions on profiles.
- Select the "**Rename**" action and enter the new name.



d. Delete a profile

- Select the profile to delete.
- Select the "**Delete**" action.

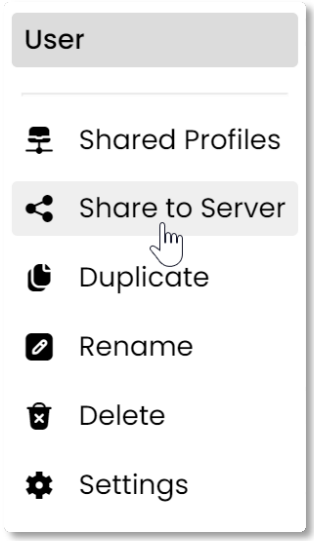


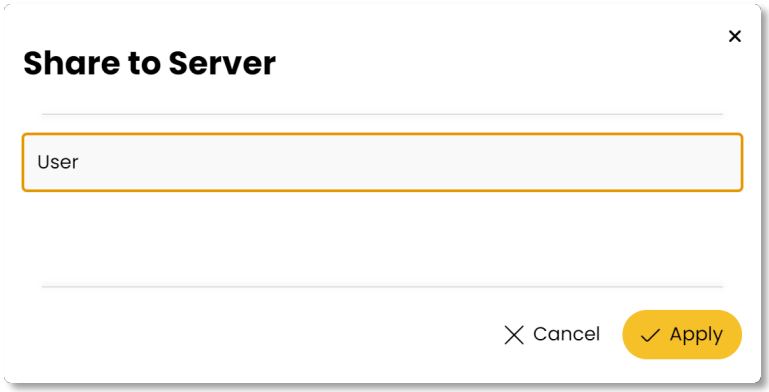


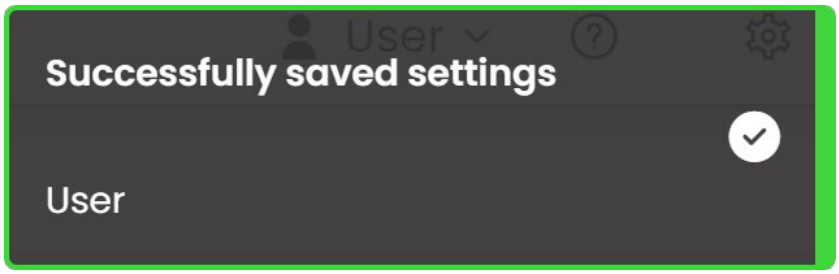
e. Share a profile

- Select the profile you want to share.
- Access actions on profiles.
- Click on the "**Share**" action.

If the **SWOOD Report Server** is configured, the profile will be published on the server, making it accessible to other users connected to that server.

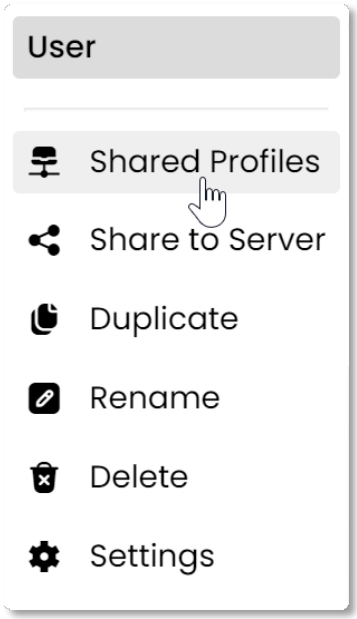


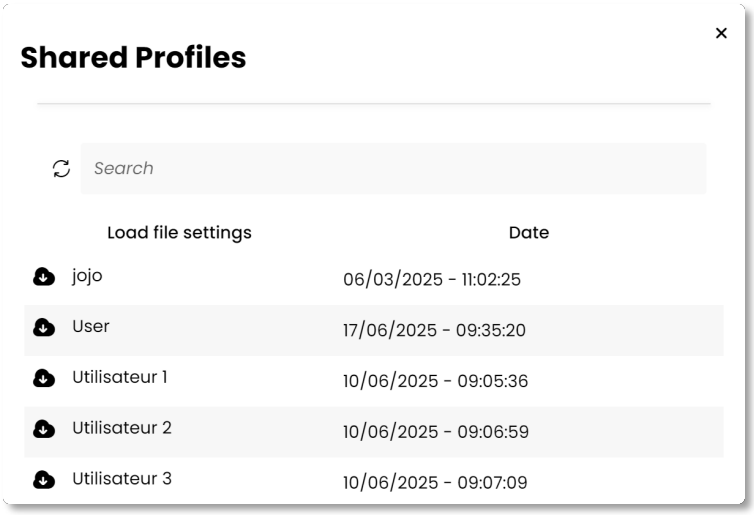




f. View the list of shared profiles

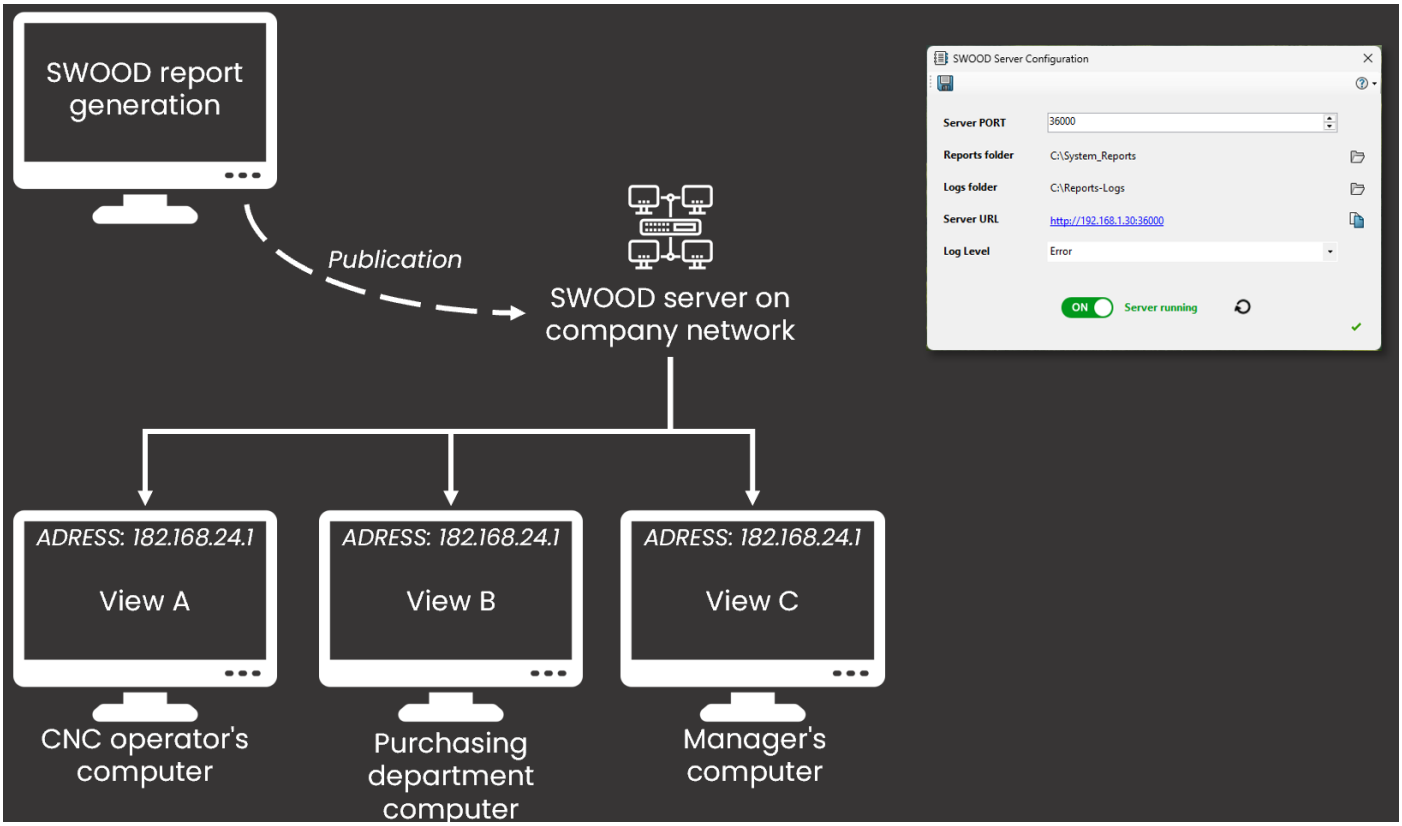
- Access actions on profiles.
- Click on the "**Shared Profiles**" action.
- This will display the list of profiles available on the **SWOOD report server**, allowing you to import or use them.





System Report – Report Sharing on Local Server

SWOOD 2025 introduces a solution to centralize and facilitate the sharing of System Reports within the company's local network using a dedicated report server: the SWOOD Report Server. This feature allows reports generated on a server machine to be published, making project information accessible to other network users via a web browser or directly in the SWOOD Report Viewer, without requiring a full installation of SWOOD on each workstation. This simplifies data distribution, avoids duplication of reports, and ensures that all employees (workshop, design office, purchasing department, etc.) have a single, up-to-date source of information.



Schematic visualization of SWOOD report sharing via a local server

How to use

1. *Server installation prerequisites*

The PC on which the SWOOD report server will be installed must:

- Be **connected** to the company's local network.
- **Remain on continuously** (avoid standby mode) to ensure server accessibility.
- Require **Windows 10 minimum** (the configuration has been validated on Windows 10 and 11).

2. *Installing the SWOOD report server*

- Download the SWOOD Report Server installation files from the usual source provided by your reseller (email, dedicated portal).

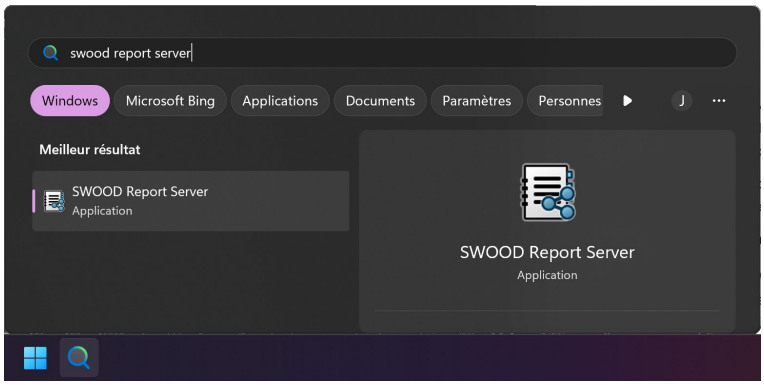
Note: It is important to always use the latest version available to benefit from fixes and improvements.

- Follow the standard installation steps, similar to those for SWOOD.

3. *Access to the SWOOD Report Server Configurator*

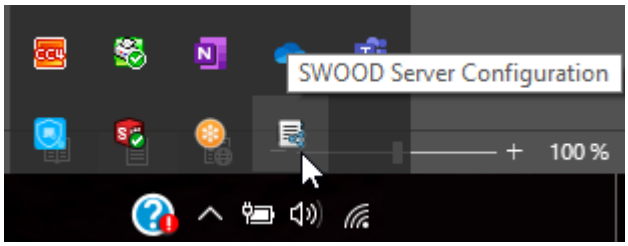
Once the server is installed, its configuration application (not the server itself, which can run in the background) is accessible for configuration. To open the configurator:

- o Search for "**SWOOD Report Server**" in the Windows search bar.

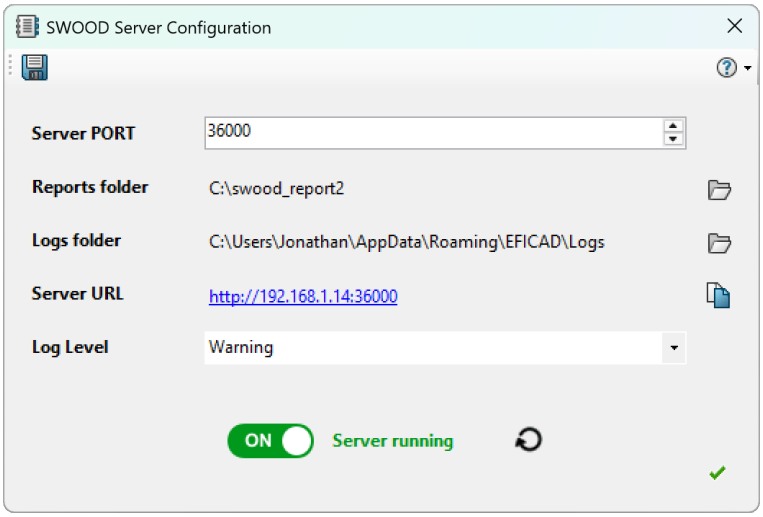


- Or access it via the "Start" menu → 'EFICAD' → "**SWOOD Report Server**".

Important: The icon in the notification area (if visible) corresponds to the server **configurator**, not the server itself. The server may be running even if this icon is not visible or if the configurator is closed. The server is designed to run as a service and, in principle, restarts automatically in the event of a problem.



4. *SWOOD server configuration*



In the "SWOOD Server Configuration" interface:

a. Server PORT

Enter the port number on which the server will listen for requests (for example, 36000). Make sure that this port is not already in use by another application on the server and that it is open in the server's firewall if necessary.

b. Reports folder

Specify the path to the folder on the server where the .swr report files (published from SWOOD workstations) will be stored.

c. Logs folder

Path where the server activity logs will be saved.

d. Server URL

This URL will be used by customers to access reports. You can copy it by clicking on the copy button to share it.

e. Log Level

Select the level of detail for the information that will be recorded in the server activity logs. The available options are:

- **None**: No information is recorded.
- **Warning**: Only warnings and errors (normal and critical) are logged.
- **Error**: Only errors and critical events are logged (recommended for normal use so as not to overload the logs).
- **Critical**: Only critical events that prevent the server from functioning properly are logged.

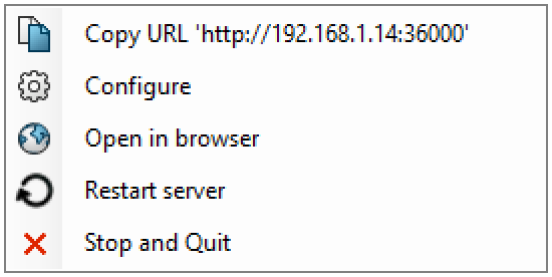
5. *Starting the Server*

- In the configurator, click the server start button to launch the service. The status will change from **OFF** to **ON**.

- If the configured port is blocked by a firewall, a message may appear asking for permission to open it. Confirm by clicking "Yes."

6. *Commands from the Windows notification area*

If you right-click on the Windows notification icon corresponding to the SWOOD report server, you will have access to various commands:



1. **Copy URL:** Copies the URL to connect to the system report page.
2. **Configure:** Opens the report server configuration window.
3. **Open in browser:** Opens the system report in your default browser.
4. **Restart server:** Restarts the license server. Use this in case of a bug or if the system report does not respond.
5. **Stop and Quit:** Stops the report server service and closes the configuration window.

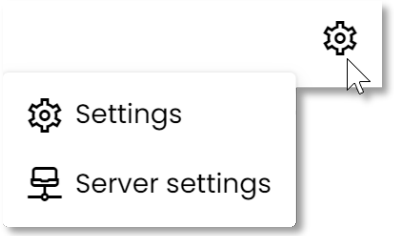
7. *Publishing reports from SWOOD*

- On SWOOD user workstations, in the System Report settings (Report Configuration window → Settings), you must configure the SWOOD Report Server URL (copied in step 4.d) in the appropriate field for publication.
- When a System Report is generated on a SWOOD workstation, if it is configured for publication, the .swr file will be sent to the reports folder on the server.

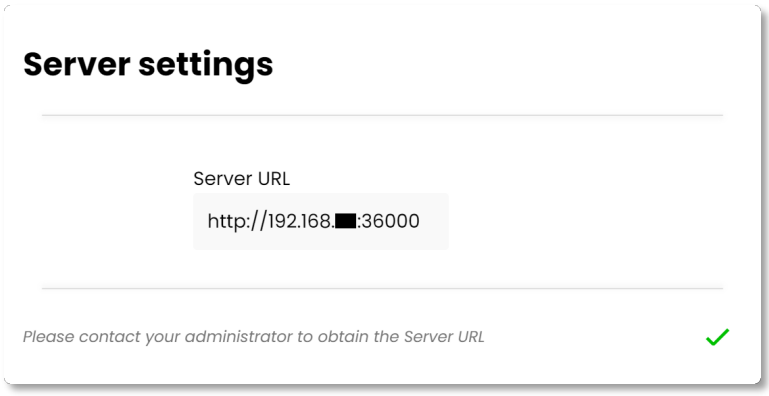
8. *User access to reports*

Other network users can access published reports in two ways:

- **Via a web browser:** By entering the server URL (copied in step 4.d) into the address bar of their browser.
- **Via the SWOOD report viewer:**
  - In the Report Viewer (on a computer where SWOOD is installed or via the standalone Report Viewer), click on the "Server Settings" icon.



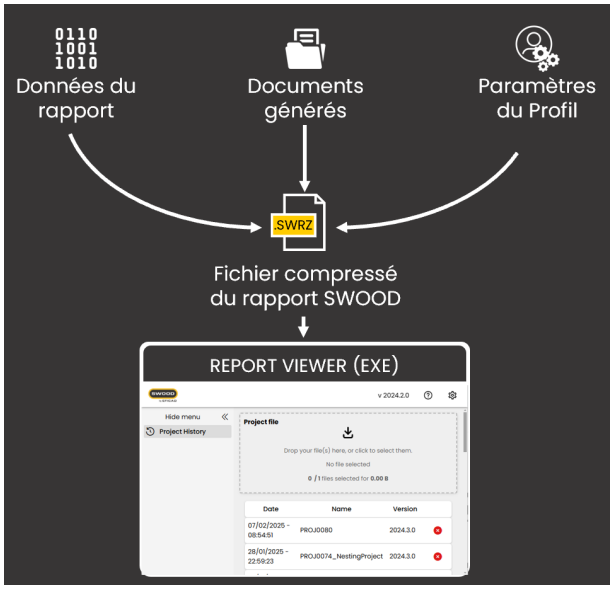
- Paste the server URL into the appropriate field.



- The reports available on the server will be listed and can be opened.

System Report – Report Sharing in Compressed Format

SWOOD 2025 facilitates the complete sharing of System Reports with users outside the company, such as subcontractors or customers, thanks to a new compressed file format: .swrz. This single file encapsulates not only all report data (table information, variables), but also generated documents (such as parts lists) and the settings of the active display profile at the time of export. The recipient can then view all of this content using the standalone **SWOOD Report Viewer**, without requiring a full installation of SWOOD. This greatly simplifies the communication of complete and contextual project information.



Schematic visualization of SWOOD report sharing in a compressed format

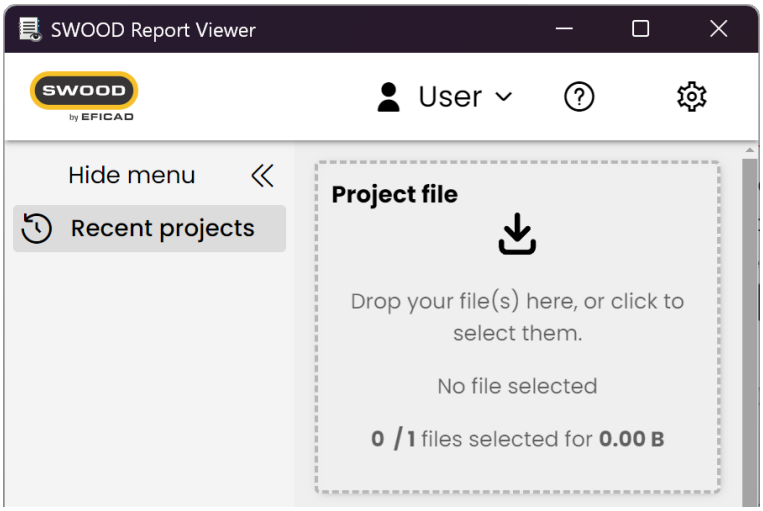
How to use

You can watch the steps in this example by following this video.

1. *Preparation of the report to be shared*
   - Open the System Report you want to share in the **SWOOD Report Viewer** (either after generating it or by opening an existing .swr file).
   - Ensure that the active display profile is the one you want to send to the receiver (for example, a "Subcontractor" profile that does not display costs but includes part drawings). The settings for this profile will be included in the .swrz file.
   - Go back to "**Project**".
   - Click on the "**Download**" button in the toolbar.
   - Select a folder where the .swrz file will be created.

2. *Download the report in compressed format (.swrz)*

- In the **Report Viewer** interface, go to "**Recent Projects**".
- On the right side, in the "Project File" box, you can either **drag and drop** an .swr file or **click on** it to open the Windows file open window and select a file.



3. *Sharing the .swrz file*

Once the .swrz file has been saved, share it with the external user using your preferred method (email, file transfer service, etc.).

4. *Viewing by the receiver (external user)*

The external user must have the standalone SWOOD Report Viewer (Report Viewer EXE).

- **Installing the Report** Viewer (if necessary): If the receiver does not already have it, they will need to install the SWOOD **Report Viewer**. Installation files are downloadable with this link.
- **Opening the .swrz file:**
  - The receiver opens the SWOOD **Report Viewer** application.
  - To open the .swrz file, refer to part 2 of this chapter.
  - The receiver will then have access to all data, documents, and display settings as they were at the time the .swrz file was created.

System Report – Version Update Procedure

Updating components related to the SWOOD System Report follows a standard procedure that requires the existing version to be uninstalled before installing the new one. This method ensures a clean transition and avoids potential conflicts between versions. This procedure applies to: the SWOOD software itself, the **SWOOD report server** (if used), and the standalone **SWOOD Report Viewer**.

SWOOD software update

How to use

To update any of the products listed above, please follow these steps:

1. *Uninstalling the current version*
   - Before installing the new version, it is essential to uninstall the version currently installed on the workstation or server.
   - Use the Windows Program Manager ("Add/Remove Programs" or "Apps & Features") to uninstall the relevant component (**SWOOD**, **SWOOD report server**, or **SWOOD report viewer**).

Note: If you have a lot of programs installed, you can sort the list by "Installed on" to bring up the most recently installed programs, or sort by 'Editor' and search for "EFICAD."

2. *Installing the new version*
   - Make sure you have downloaded the installation files for the new version from the official source (provided by your reseller or EFICAD) prior to installation.
   - Run the installation program for the new version.

Recommendation: It is advisable to right-click on the installation file and select "Run as administrator" to ensure that the installation has all the necessary permissions.