

Machining Export

bSolid

bSolid

Makes it real. After you think it... before you do it!



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Note: If you are experiencing problems using this training guide, please feel free to send your feedback and comments at edition@topsolid.com.

Contents

Introduction	1
TopSolid'Wood configuration	2
General configuration	2
Configuring default machining options.....	3
Drilling settings.....	3
Tool path setting for grooves, rabbets and mouldings	3
Tool calibration over-thickness value.....	4
BOM configuration for part selection sorting.....	4
Export configuration using configuration keywords.....	5
General configuration keywords.....	5
Interface-specific configuration keywords.....	7
bSolid configuration	8
Making an export	8
From TopSolid'Wood	8
One part mode	8
Multi-parts mode	10
From bSolid	12
Notes	14

Introduction

bSolid is the machining software for programming **Biesse** numerically controlled machines. The interface sends the parts drawn in **TopSolid'Wood** to **bSolid** with machining definitions. **TopSolid'Wood** then produces a .cix file to be imported from **bSolid**.

TopSolid'Wood configuration

Export to **bSolid** can be configured in **Tools > Options > TopSolid'Wood configuration > Machining > Biesse CIX configuration**.



General configuration

- **Number of decimals:** Corresponds to the number of decimal places to be exported. Let's take a drilling 10.1234mm deep, for example, for 3 decimal places, the depth exported to **bSolid** will be 10.123mm.

Number of decimals :

- **Tolerance of discretization:** The tolerance of discretization defines the level of accuracy used to export the machining geometries, including curved and circular paths. The finer the tolerance, the closer the exported curve is to the drawn curve, but the longer the calculation. The value of 10^{-5} (in meters) is the most common average value used, because it provides a good compromise between precision and performance.

Tolerance of discretisation :

- **Files save path:** Lets you specify the path of the directory where the exported files will be saved.

Files save path :

Note: By default, if no directory is specified, the files will be saved in the folder of the exported project.

- The **Unit** section lets you configure the type of unit (**Millimeter** or **Inch**) for the machining file.

Unit

Millimeter
 Inch

Configuring default machining options

You can adjust some default machining settings in the **TopSolid'Wood** configuration.

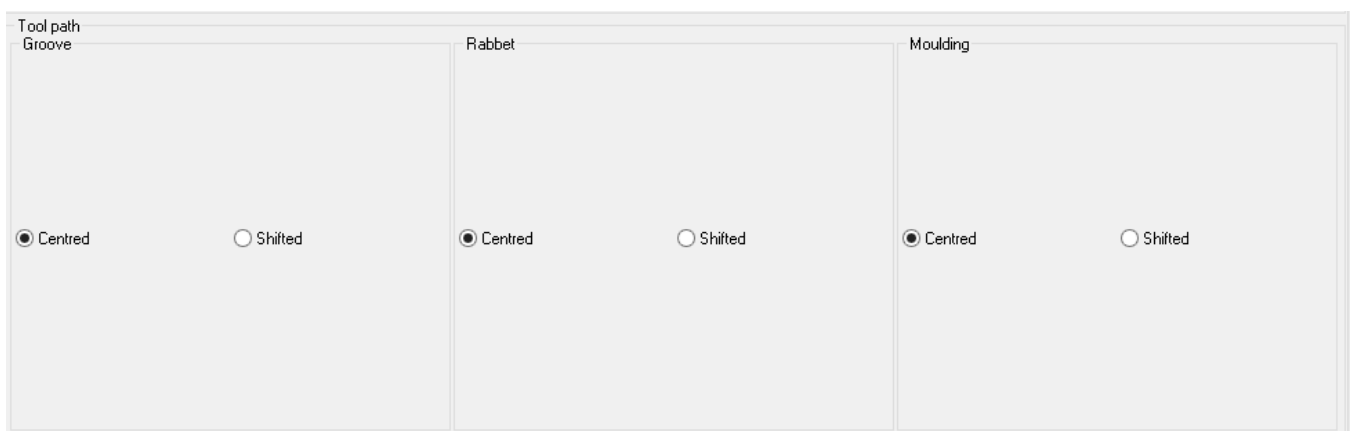
Drilling settings

- **Maximum diameter:** This option lets you specify a maximum drilling diameter for machining. If the drilling diameter is strictly greater than this setting value (in millimeters), the drilling is not exported to **bSolid**.

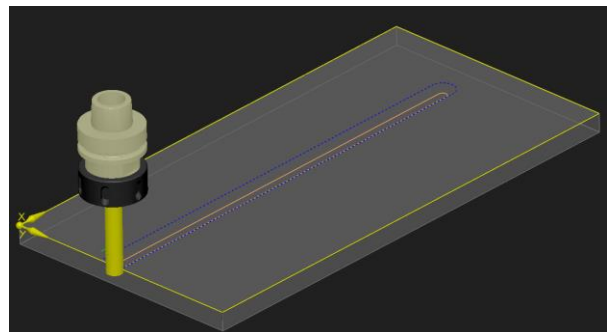
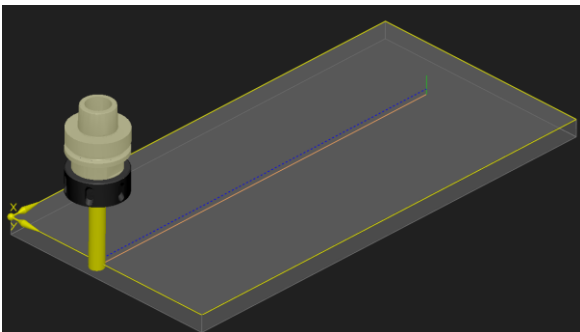
Tool path setting for grooves, rabbets and mouldings

You can choose to export the centered or shifted tool path for each operation (**Groove**, **Rabbet** and **Moulding**).

Tool path settings can be configured in **Tools > Options > TopSolid'Wood configuration > Machining > Biesse CIX configuration**.



Examples of groove exports with the **Centered** option (left) and **Shifted** option (right) enabled:

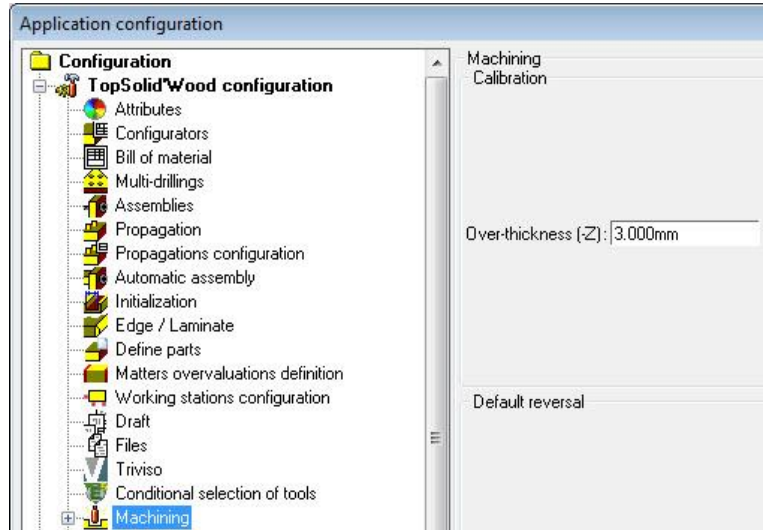


Note: This setting will be only applied to grooves and rabbets using a **router**. The grooves that go through the part are systematically exported in **Centered** mode.

Tool calibration over-thickness value

You are able to export an over-thickness value for calibration machining to **bSolid**. This value corresponds to the tool's exceeding value under the part.

This default value can be adjusted in **Tools > Options > TopSolid'Wood configuration > Machining > Over-thickness (-Z)**.



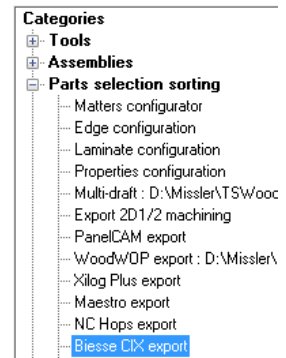
Note: This default value can be modified at each export in the part selection window.



BOM configuration for part selection sorting

You can set a default BOM file for part selection sorting in **TopSolid'Wood**.

The BOM file to be used for part sorting can be set in **Tools > Options > TopSolid'Wood configuration > Files > Part selection sorting**.



Export configuration using configuration keywords

The configuration keywords are used to set some export parameters that are less common than those available in **Tools > Options > TopSolid'Wood configuration**.

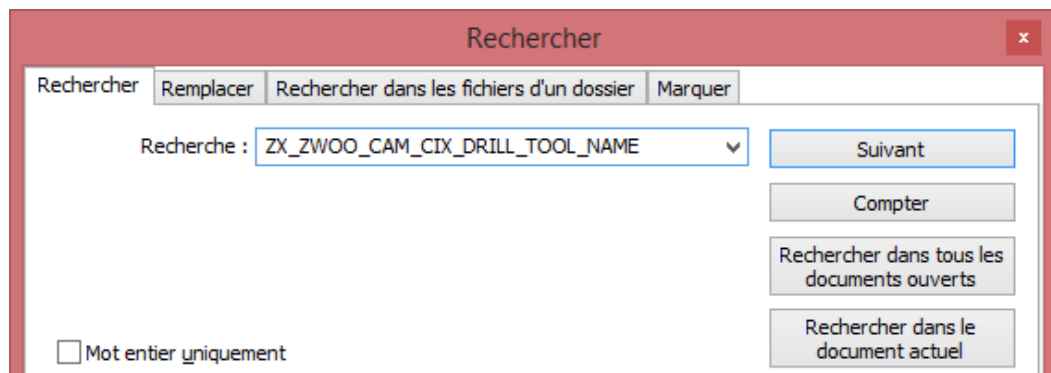
To modify/add configuration keywords, proceed as follows:

- Using the **Notepad** (or any other text editor), open the *topzwoo.cfg* file which is located in the folder *Missler/ConfigV6x/V6x/topzwoo.cfg*.

Note: When in a **Group** configuration, open the *Missler/GroupV6x/V6x/topzwoo.cfg* file.

In this file, copy and paste the configuration keywords followed by the rules (setting values) separated by a tab character.

- Before pasting a new configuration keyword, check that it does not already exist by using the **Search** function of the **Notepad** (**Edit > Search** or **Ctrl + F**).



General configuration keywords

<i>Configuration keyword</i>	<i>Value</i>	<i>Result</i>
ZX_ZWOO_CAM_NO_CALIBRATION	1	Automatically deactivates the part's calibration when the overdimension values are set to Edge shape or Abacus mode.
	0 (default value)	The part's calibration is always exported.
ZX_ZWOO_CAM_CALIBRATION_CHANGE_ORIGIN	1	The starting point of the part's calibration will be in the middle of a calibration segment.
	0 (default value)	The starting point of the part's calibration will be at the end of a calibration segment.
ZX_ZWOO_CAM_GROUP_DRILLS	1 (default value)	Groups together the drillings during export.
	0	Allows the drillings not to be grouped together during export.
D_SH_OP_POC_USE_MAC_PROPERTY	1	Allows for the correct export of overlapping operations with pockets.
	0 (default value)	

ZX_ZWOO_CAM_REMOVE_FACING_HOLE	1	Allows the through hole of a spot facing not to be exported.
	0	The spot facing and the through hole are exported.
ZX_ZWOO_CAM_THROUGH_DRILL_IN_FIRST_FILE	0 (default value)	Through holes are exported on their reference face.
	1	In Two files if necessary mode, the through holes are exported in the first machining file.
ZX_ZWOO_CAM_THROUGH_DRILL_ABOVE_ONE_FILE	1 (default value)	In One machining file mode, the bottom through holes are exported on the top face.
	0	Through holes are exported on their reference face.
ZX_ZWOO_CAM_THROUGH_POC_ABOVE	1 (default value)	Exports the bottom through pockets on the top face.
	0	Through pockets are exported on their reference face.
ZX_ZWOO_CAM_MPR_THROUGH_POC_TO_MILL	1 (default value)	Through pockets are exported as millings. Only the contour of the pocket will be exported.
	0	A through pocket is exported as a pocket. All of the material will be machined.
ZX_ZWOO_CAM_MPR_INSIDE_SAWING_TO_MILL	1	Exports the closed trimmings performed inside the part as a milling.
	0	The closed trimmings performed inside a part are not exported.
ZX_ZWOO_CAM_MPR_MILL_CHANGE_ORIGIN	1	The starting point of the part's internal milling will be in the middle of a milling segment.
	0 (default value)	The starting point of the part's internal milling will be at the end of a milling segment.
ZX_ZWOO_CAM_DRILL_DEPTH_WITHOUT_TIP	0 (default value)	The drilling depth is exported without the tip.
	1	The drilling depth is exported with the tip.
ZX_ZWOO_CAM_THROUGH_DRILL_IN_FIRST_FILE	0 (default value)	Through holes are exported on their reference face.
	1	In Two files if necessary mode, the through holes are exported in the first machining file.

Interface-specific configuration keywords

<i>Configuration keyword</i>	<i>Value</i>	<i>Result</i>
ZX_ZWOO_CAM_CIX_DRILL_TOOL_NAME	101 ""(default value)	Default tool number assigned to the drilling operation.
ZX_ZWOO_CAM_CIX_POCKET_TOOL_NAME	101 ""(default value)	Default tool number assigned to the pocket operation.
ZX_ZWOO_CAM_CIX_MILL_TOOL_NAME	101 ""(default value)	Default tool number assigned to the milling/calibration with router
ZX_ZWOO_CAM_CIX_SLOT_MILL_TOOL_NAME	101 ""(default value)	Default tool number assigned to the groove with router.
ZX_ZWOO_CAM_CIX_SLOT_SAW_TOOL_NAME	101 ""(default value)	Default tool number assigned to the groove with saw.
ZX_ZWOO_CAM_CIX_THROUGH_POC_DEPTH_OFFSET	0.003 ""(default value).	Default over-thickness dimensions (in meters) used for through pockets.

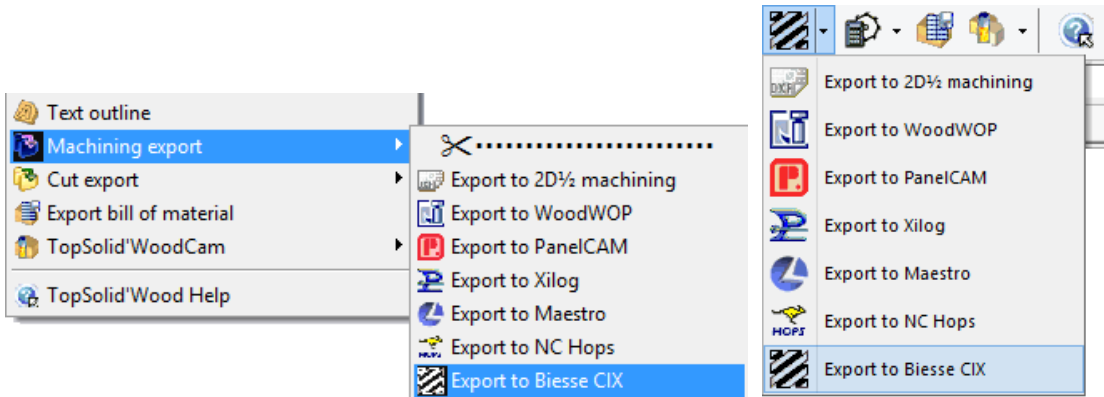
bSolid configuration

No special settings are required in **bSolid** to import the files generated by **TopSolid'Wood**.

Making an export

From TopSolid'Wood

- Select **Wood > Machining export > Export to Biesse CIX**.



Note: The **Machining export > Export to Biesse CIX** function can also be accessed from the icon bar of the **Wood** context (on the left side of the screen).

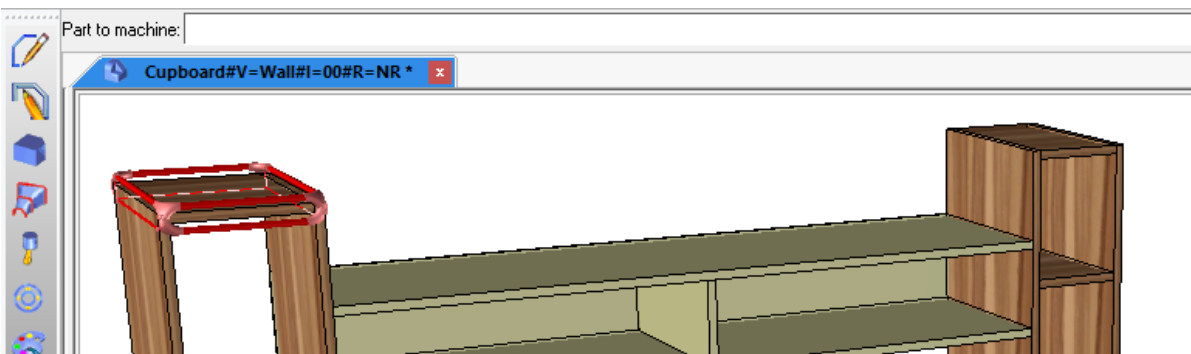
- Choose the **One part** or **Multi-parts** mode.
 - The **One part** mode lets you export only one part. You can also use this mode to add milling operations to the part manually.
 - The **Multi-parts** mode lets you export several parts of an assembly.

One part mode

- Select the **One part** mode.

The file explorer opens in the folder that is defined by default in **Tools > Options**.

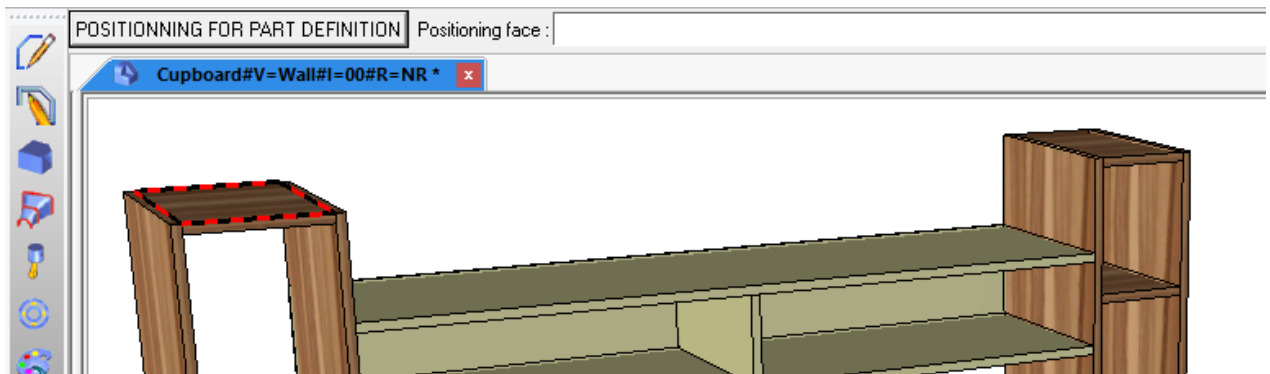
- Select the folder where you want your file saved and give the exported file a name (by default the name of the original Design file).
- Select the part to be exported to **bSolid**.



- If the selected part is set to **Optimized** machining mode, the following two options are available:
 - **OK** to continue without updating the machining face.
 - **Update machining face** to update the machining face of the exported part.

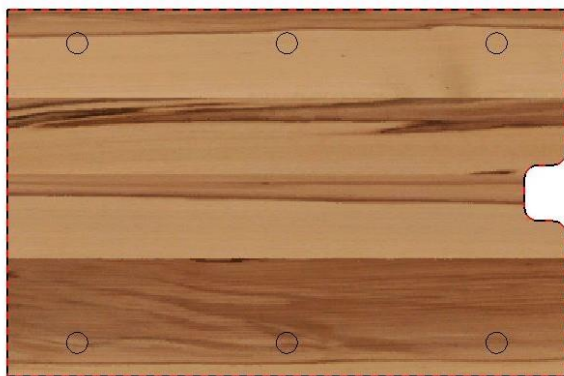
- Next pick the positioning face (CNC positioning face, the bottom in **bSolid**).
- Select the **Positioning for part definition** option to use the positioning face set in the part definition.

POSITIONNING FOR PART DEFINITION

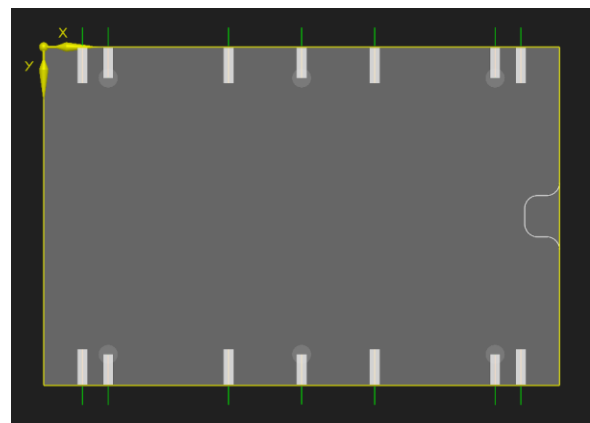


Note: The part is placed in top view in relation to the selected face.

- Select the part's positioning orientation for the **bSolid** machining export using the << and >> arrows from the dialog bar.
- Click **OK** to confirm the positioning.



Part displayed in **TopSolid'Wood**



Part positioning in **bSolid**

Note: In **TopSolid'Wood**, the part's horizontal axis from left to right will be the X+ axis in **bSolid** and the vertical axis from bottom to top will be Y+.

- The **Other routings of contour** button lets you add milling operations to the exported part:

OTHER ROUTINGS OF CONTOUR

- **Edges on planar face:** Lets you select several edges at once from a reference selected face.
- **All face edges:** Selects all edges of the selected face.
- **Relative depth:** Machining depth value (in millimeters).
- **Depth from face:** The **relative depth** value is calculated by default from the reference face you selected previously. To change the reference face for the machining depth, click **Depth from face** and select the new reference face for the machining depth.
- **Follow tangent edges = Yes/No:** You can choose to associate or not associate the tangent edges with the selected edge.
- **Reference edge for tool path:** Lets you pick the edges you want to add to the tool path.

- Once you have selected one or more edges, click **OK** to confirm.
- It is possible to select another reference face. If you want to keep the previously selected face, click **Stop**.

Note: All paths selected from the **Other routings of contour** option, except those selected from **Edges on planar face**, are not aggregated as a single machining. Each selected edge will be exported as one separate machining.

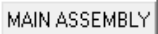
- Click **OK** to confirm the **other routings** in order to create the CIX file.


Multi-parts mode

- Select the **Multi-parts** mode. 

Several options for selecting parts are available in the dialog bar:

- **Main assembly:** Selects all parts of the main assembly.

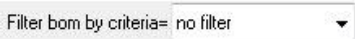


- If several **sub-assemblies** are created in the document, a drop-down list appears and lets you select a sub-assembly. 

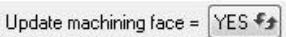
- **Depth:** Lets you select the depth type to display the elements of the selected assembly.




- **Filter bom by criteria:** Lets you select a filter to filter the parts in the selected assembly.



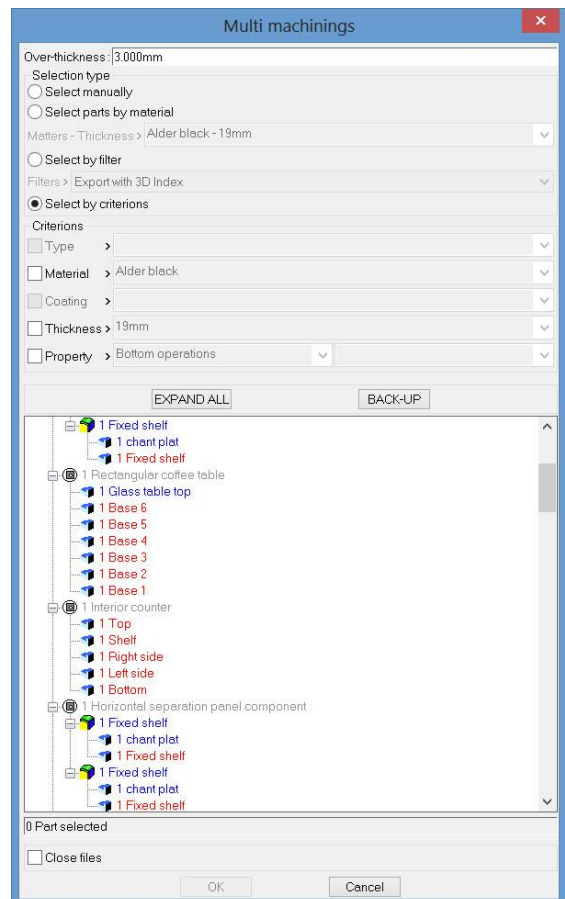
- **Update machining face:** This option is available if one part is in **Optimized** machining mode. It updates the machining faces of parts exported in **Optimized** mode.



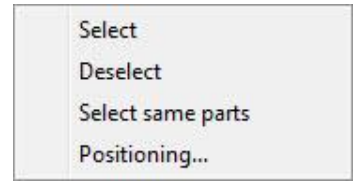
- **Select element to use:** Lets you select parts manually:
 - o **Left-click** on the parts to be selected. Once the parts are selected, they turn red.
 - o Click **STOP** to confirm the selected part(s). 

The **Multi machinings** window opens.

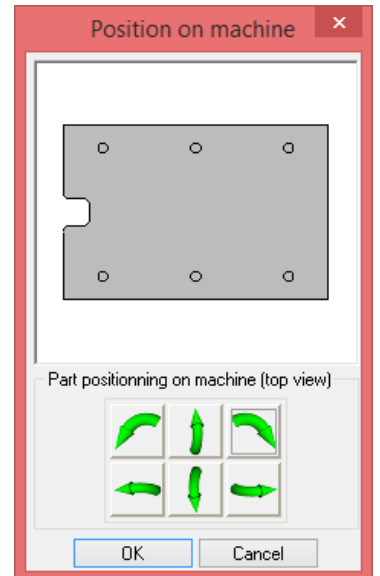
- **Over-thickness:** The default over-thickness value is provided in **Tools > Options**, but you can modify this value for the exported project.
- **Select type:** Part selection mode.
- **Expand all/Back-up:** Opens or closes all of the assembly in the BOM.



- Right-clicking on a part of the list displays several options:
 - **Select:** Selects the part.
 - **Deselect:** Deselects the part.
 - **Select same parts:** Selects all parts that are identical to the selected ones.
 - **Positioning:** Opens the setting window for the part positioning. This window allows you to change the positioning direction of the part in **bSolid**.



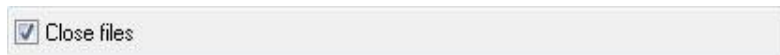
Note: This change in positioning will only apply to this export.



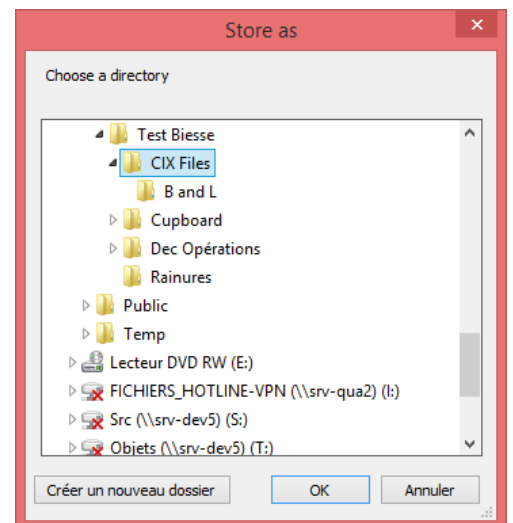
Note: At the bottom of the window, an information box indicates the number of selected parts.

1 Part selected

- Selecting the **Close files** option prevents the generated files from opening. Check this box when exporting a lot of parts.



- Click **OK** to start the export.
- The folder selection window opens on the folder configured in the **options**. Select a destination folder for the exported files.



TopSolid'Wood then generates the **CIX** files. Once the export is finished, the number of generated files is displayed in the alpha bar.

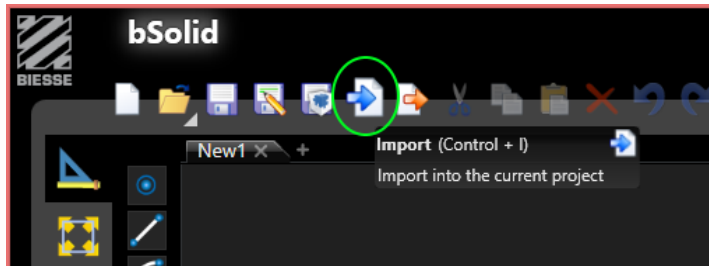
Machining export | Export to WoodWOP.

3 file(s) saved

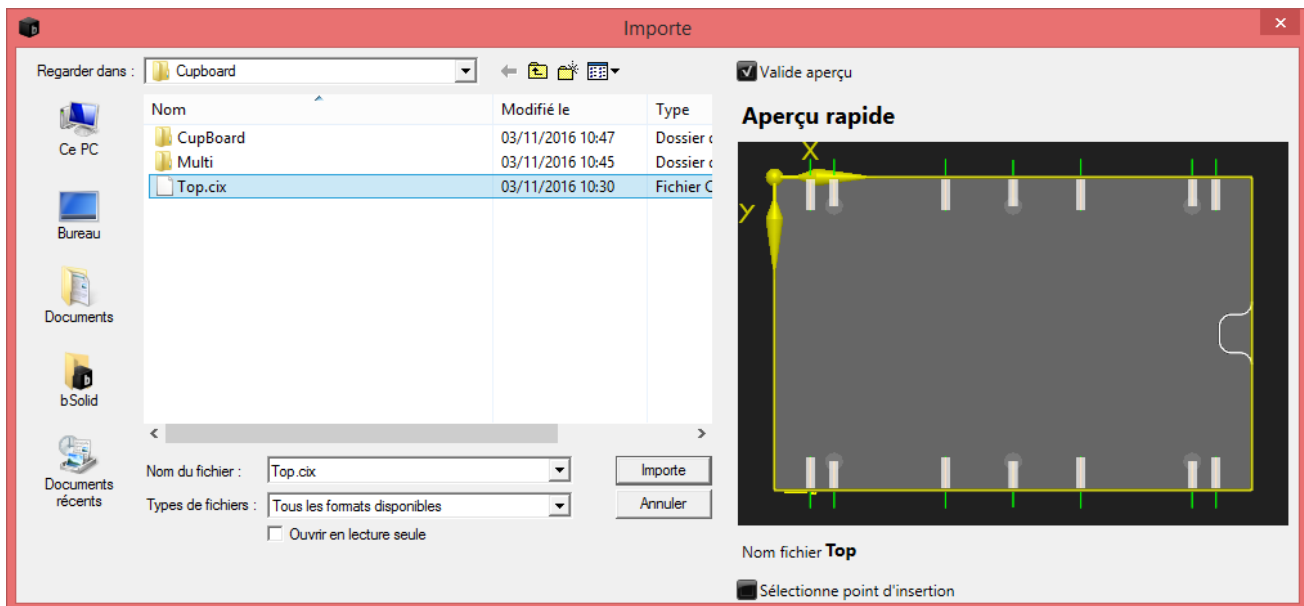
Note: If you did not check the **Close files** option, all the generated files are opened using the notepad.

From bSolid

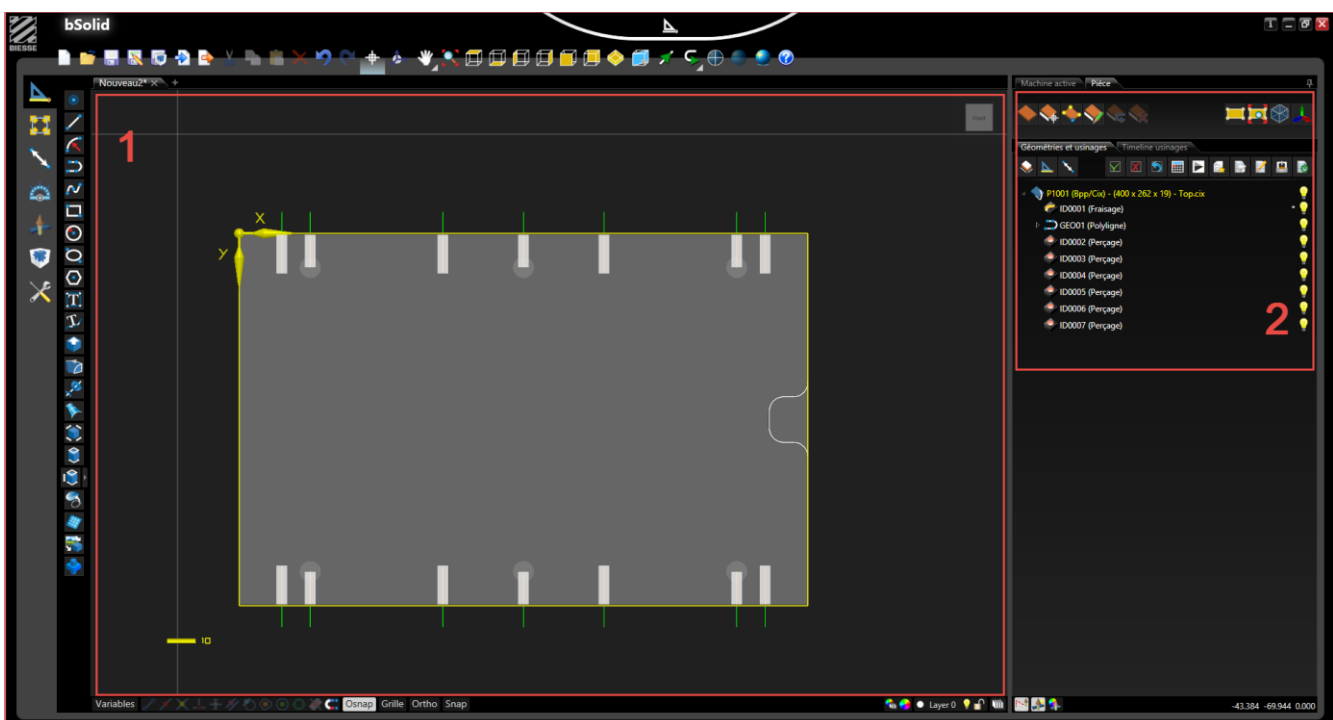
- Once the .cix files have been generated from TopSolid, they need to be imported to **bSolid**.
- From a new **bSolid** document, use the **Import** function.



- Next select the .cix file generated by TopSolid.



The **bSolid** interface is split into two areas:



1. Drawing area

This area includes the graphics area and the part display. Various display options are available in the toolbar such as the isometric perspective views and the 2D views of the part's faces.



2. Data area

This area is divided into 3 parts:

1- Commands:

Lets you define or edit part information.

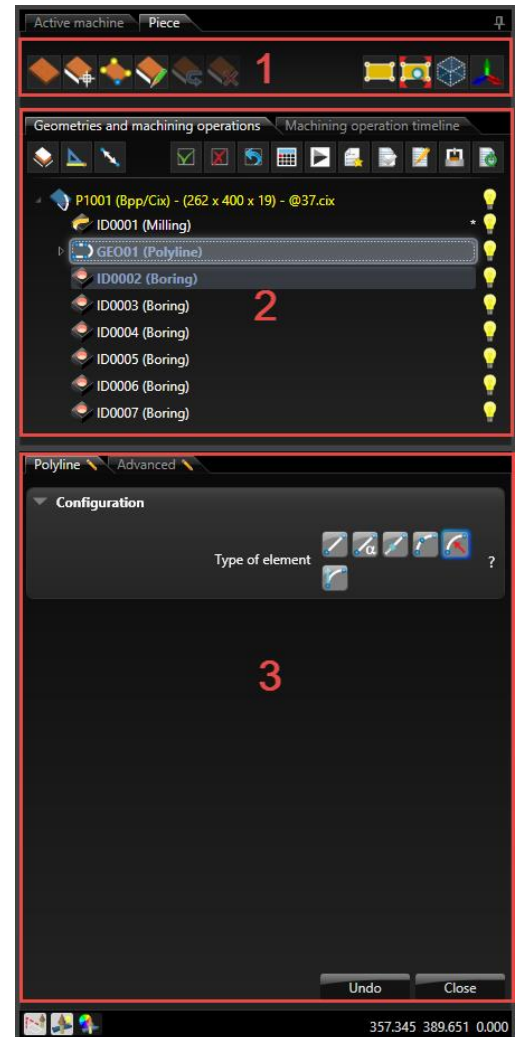
2- Geometries and machining operations:

The **Geometries and machining operations** tab displays the nodes of the geometries, the part and applied machining operations and lets you edit, show or hide the machinings in the graphics area.

The **Machining operation timeline** tab displays the nodes of the machining operations. From this area, the machining operations can be ordered and simulated, the ISO files can be generated, etc.

3- Parameters:

Lets you display and/or change the parameters of the selected item.



Notes

A series of horizontal dotted lines for taking notes.

A series of horizontal dotted lines for taking notes.