CAM Guide

SIMBA SIMPLE v0.2

Cutting Strategy

Our goal is to cut the chair parts from an 18 mm (¾”) plywood sheet. There are multiple ways to perform such a cut, but describing all of them would be out of scope of this document. If you have a working strategy to perform this kind of cut, you can skip the next section. Otherwise, use the following recommendations as a starting point for your experiments.

It is strongly recommended to use a single flute 3.175 mm (⅛”) compression end mill with at least 20mm (0.79”) flute length for this job. With such a narrow cut, wood chips will perfectly hold the parts in place while it’s being cut, so there will be no need to use tabs, and therefore, no need to sand them off.

Notice: *If your router/spindle doesn’t support such an end mill, use a 6 mm (¼”) one and make sure you use tabs when creating paths.*

Feeds and Speeds

Why does speed matter? - If your feed rate is too slow, the tool will overheat and get blunt very quickly. If the speed is too high, the cut won’t be accurate. Think of the tool as it was a car wheel - it is not efficient if it’s spinning faster than the road surface moves below it, and it will get damaged if it spins slower than the surface. As a CNC machinist, not only can you adjust the wheel rotation speed and the car speed independently, you can choose different wheel diameter and depth as well. Your goal is to find a correct balance between multiple variables without overstressing the system to get an accurate and fast cut and longer tool life.

To calculate an optimal cutting feed rate, multiply the maximum RPM of your router/spindle by the chip load of your tool:

Optimal Feed Rate = Spindle RPM x Tool Chip Load

Here is the table of the most common tool chip loads for plywood:

	Number of Flutes	
Tool Diameter	**1**	**2**
3.175 mm (⅛”)	0.125 mm (0.005”)	0.25 mm (0.01”)
6 mm (¼”)	0.3 mm (0.012”)	0.6 mm (0.024”)

As the result of such multiplication, you will get the suggested feed rate for the selected tool either in millimeters per minute or inches per minute.

If the speed seems too high or low for your machine, you can change it by reducing/increasing the RPMs or choosing a tool with a larger/smaller chip load.

When cutting at optimal speed and RPM, you will see decent chips (not dust) coming from the cut. There should be no long burns during the cut! Even a single burn can dull the end mill enough to start overstressing the system and all your calculations will not work as intended.

Depth of Cut

Once you have the feed rate, you will need to find the best depth of cut. It is generally recommended to start experimenting from a single tool radius as the depth and gradually increase it until your machine is not able to cut smoothly and consistently. Once the limit is found, use the last depth at which your machine was cutting confidently.

Known Good Settings

	Tool	**RPM**	**Feed Rate**	**Depth of cut**	**Comment**
WorkBee 1515 (Screw Driven)	3.175 (⅛”)	24000	3000 mm/min (118 ipm)	3.175 mm (⅛”)	Confident cut at cruising speed without overheats

If you found settings that work well for your setup, please send them to ajay@better.furniture so that I can add them to the table.

Importing and Calibrating

What file to use?

Import the STEP file if your CAM supports them. If it doesn't, go with the SVG file.

Calibration

When importing files to your CAM, make sure to set the import unit to millimeters.

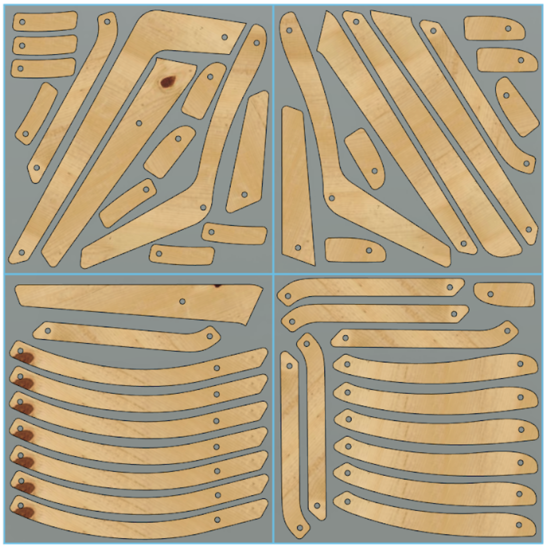
After importing, make sure that the inner diameter of the holes is 12 mm (0.47”). If you have troubles measuring the circles, try drawing a 12 mm circle and make sure that it has the same size as the holes in the imported file.

If the hole has different size, you will need to measure it as accurately as possible and calculate the scaling factor:

Scaling Factor = 12 mm (0.47”) / Measured Hole Diameter

Once calculated, re-import the file with the scaling factor set.

Creating Stock(s)



This pattern is 120 x 120 cm (4 by 4 ft) and it has all the parts for making half of the chair. Therefore to make a whole chair you will need to **cut this pattern twice**.

Because plywood usually has a nicer side, you will need to make sure you cut this pattern once with the nicer side of the plywood facing down, and once with the nicer side facing up. This way the chair will look consistently from side to side.

If your machine is smaller than the pattern, you can use it as four 60 x 60 cm (2 by 2 ft) tiles, in this case, you will have to cut out eight tiles in total - four facing down, and four facing up.

The recommended depth (thickness) of stock should be set to 19-20 mm (0.75 - 0.79”) so that you can use the same GCODE for any plywood that fits in that box. Those couple of additional millimeters on top will also help dealing with plywood that is not exactly flat or has slightly inconsistent thickness.

The height and width of stock should be set to the size of the tile you chose - either 60 x 60 cm (2 by 2 ft) or 120 x 120 cm (4 by 4 ft).

Creating Tool Paths

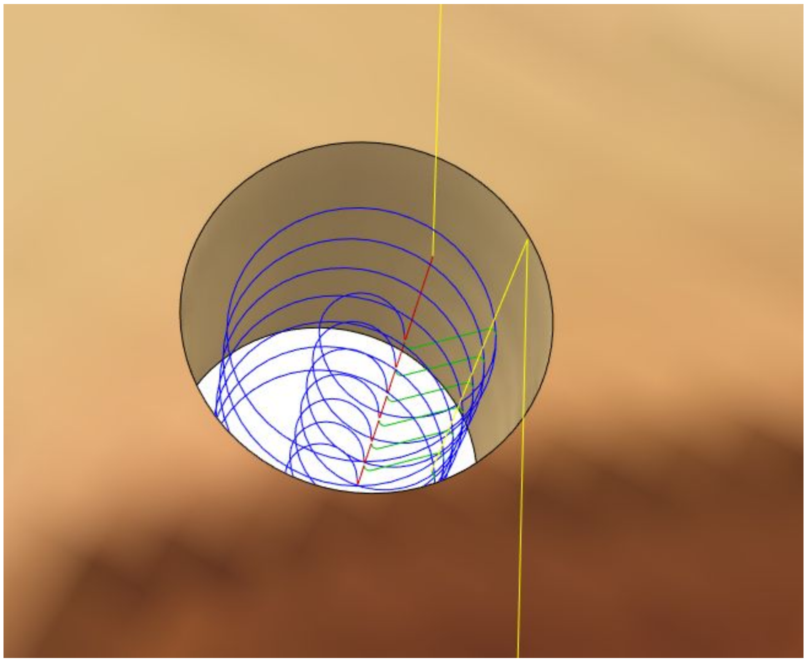
Simba Simple requires only cut-through operations. You can also go -0.5 mm deeper into the spoil board to make sure there are no plywood layers left, because sanding them off is what we want to avoid at the cost of the spoilboard that was designed to be spoiled.

Set the depth and speeds of all the cuts to the values determined earlier.

The path direction recommendations assume your spindle is rotating clockwise.

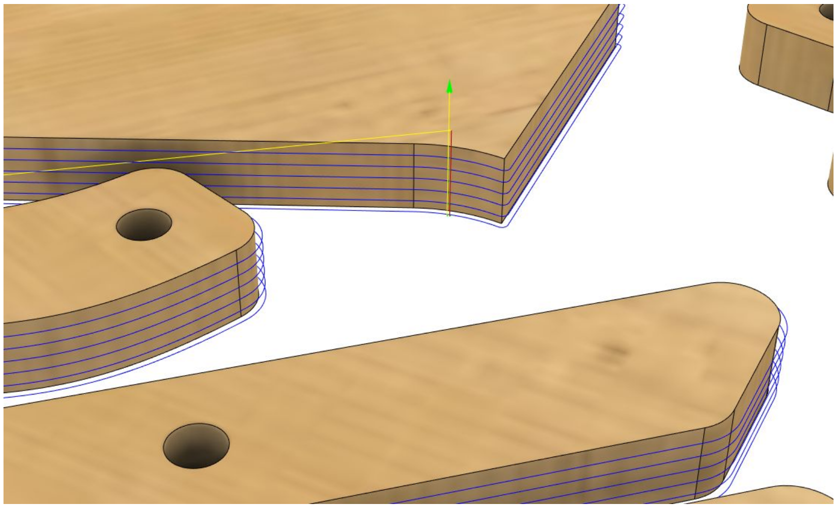
Holes

- To prevent stock leftovers dangerously flying around, it is recommended to use a pocket operation when cutting out the holes.
- Use **Left (climb milling)** if prompted.



Parts Contours

- Use contour operation.
- Make sure the tool is on the proper side of the contour.
- Use **Right (conventional milling)** if prompted.
- Disable **Stock to Leave** if it was enabled.
- Do not use tabs and finishing passes if using a 3.175 mm (⅛”) tool.
- Do use tabs if using a 6 mm (¼”) end mill. You can also use finishing passes since the part is fixed securely.



Simulating

Simulate the whole setup and make sure that everything looks as expected. Not the machining time for future reference.

Post Processing

Save your work as GCODE or Post Process the paths.

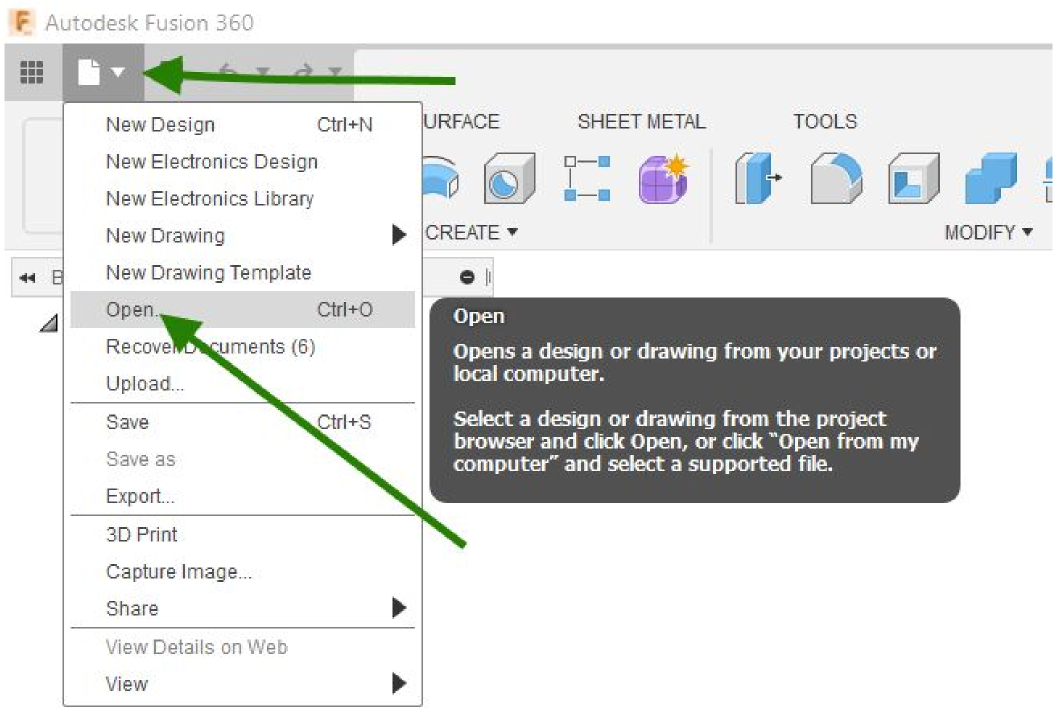
Fusion 360 step by step Workflow

From here, I will provide detailed instructions for Fusion 360 only. But you should be able to perform similar tasks in a CAM software of your choice.

Fusion 360 is a very powerful and easy to use design and CAM tool. And it **is free** for personal use and startups! It has post processors for any kind of machines, including Mach3, GRBL, X-Carve, Shapeoko and WorkBee.

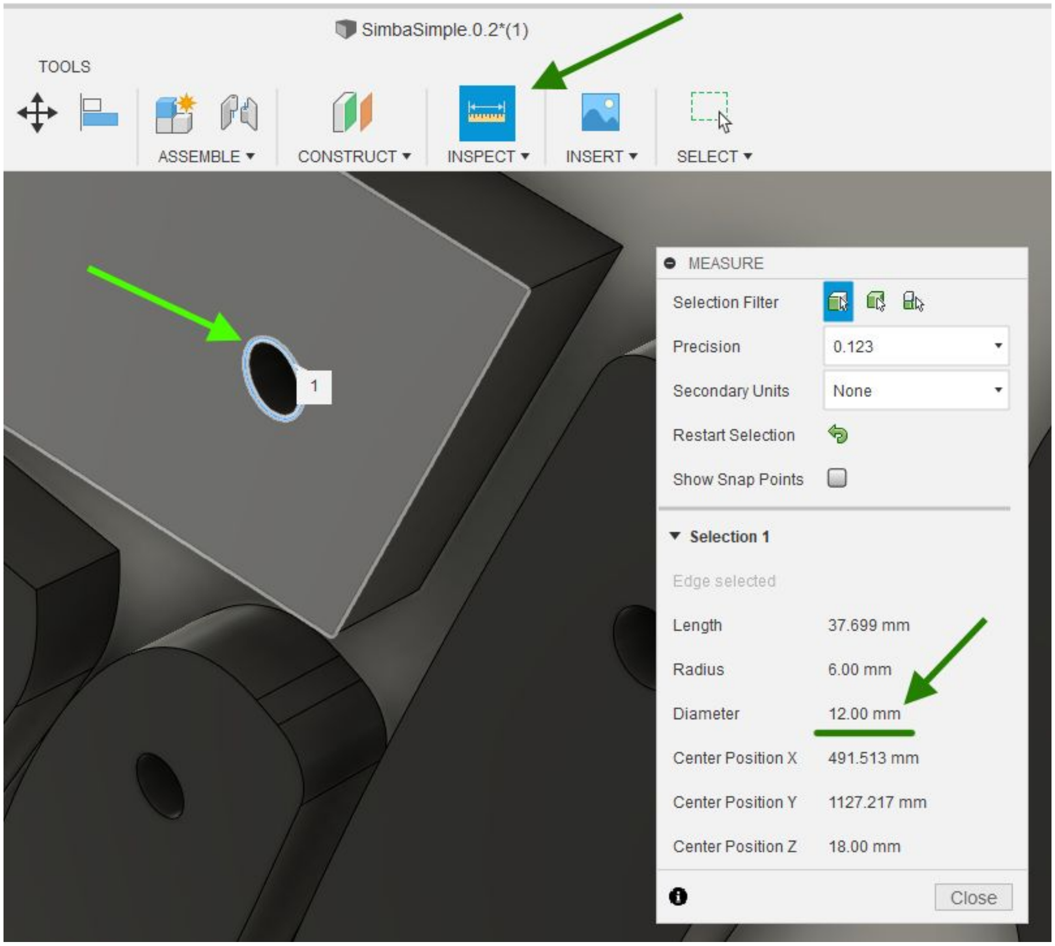
1. Importing the File

1.1 Open the Simba.Simple.step file in Fusion 360:

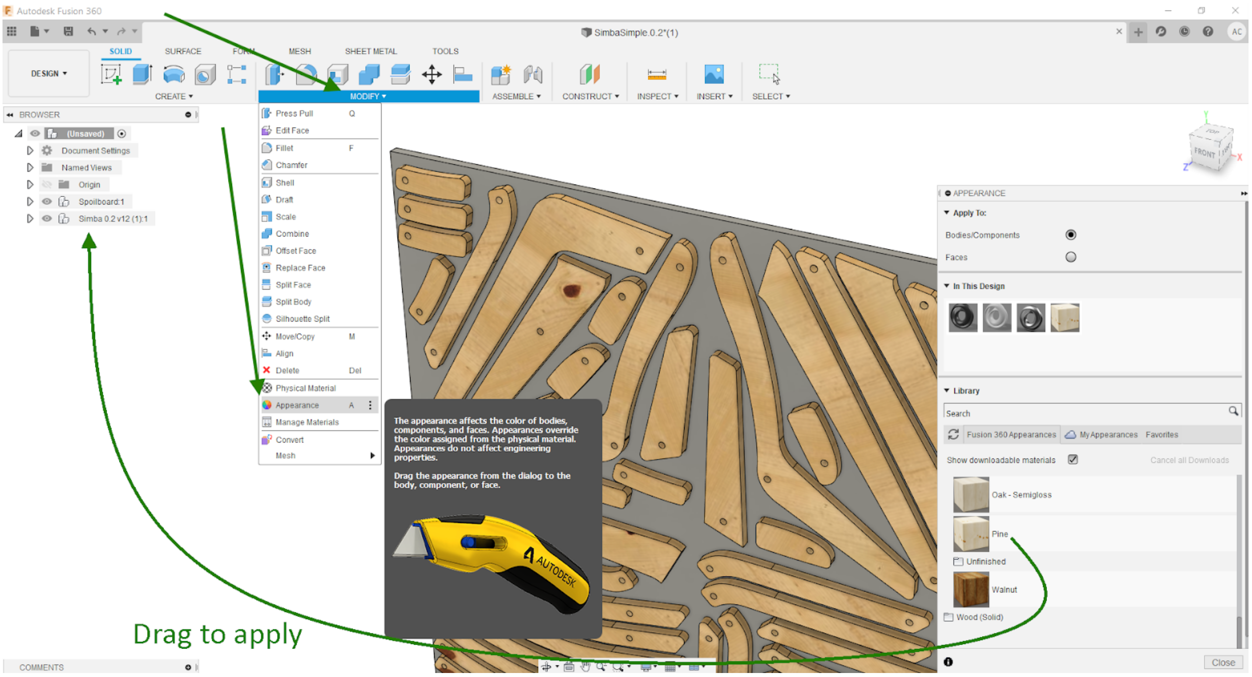


1.2 In the file open popup window, select “Open From Computer” and locate the SimbaSimple.0.2.step file in your system.

1.3 Once the file is loaded, use the Inspect tool to measure any of the holes and make sure it has a proper scale of 12mm (0.47”):

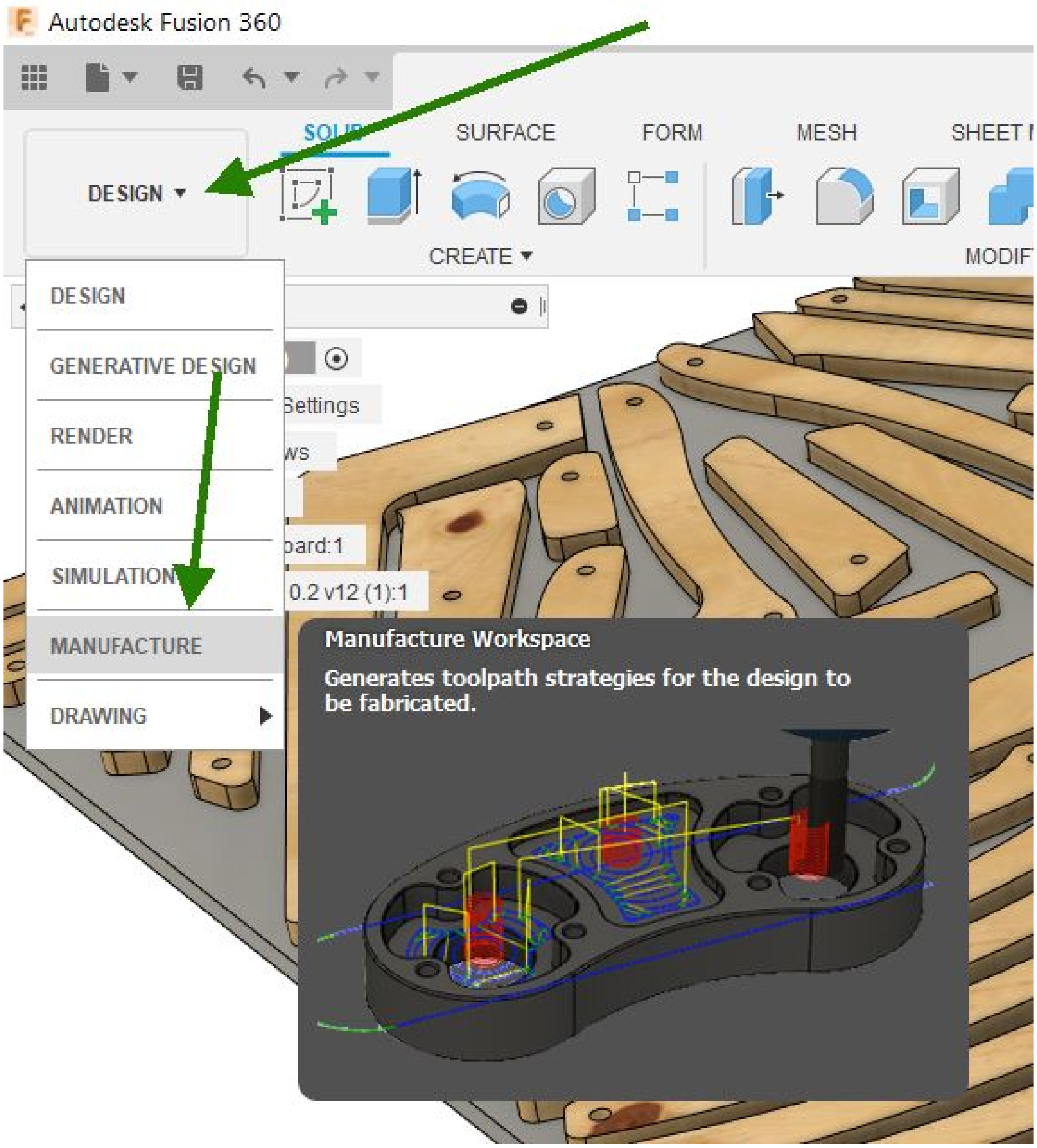


1.4 Optionally change the Simba Component appearance to wood:

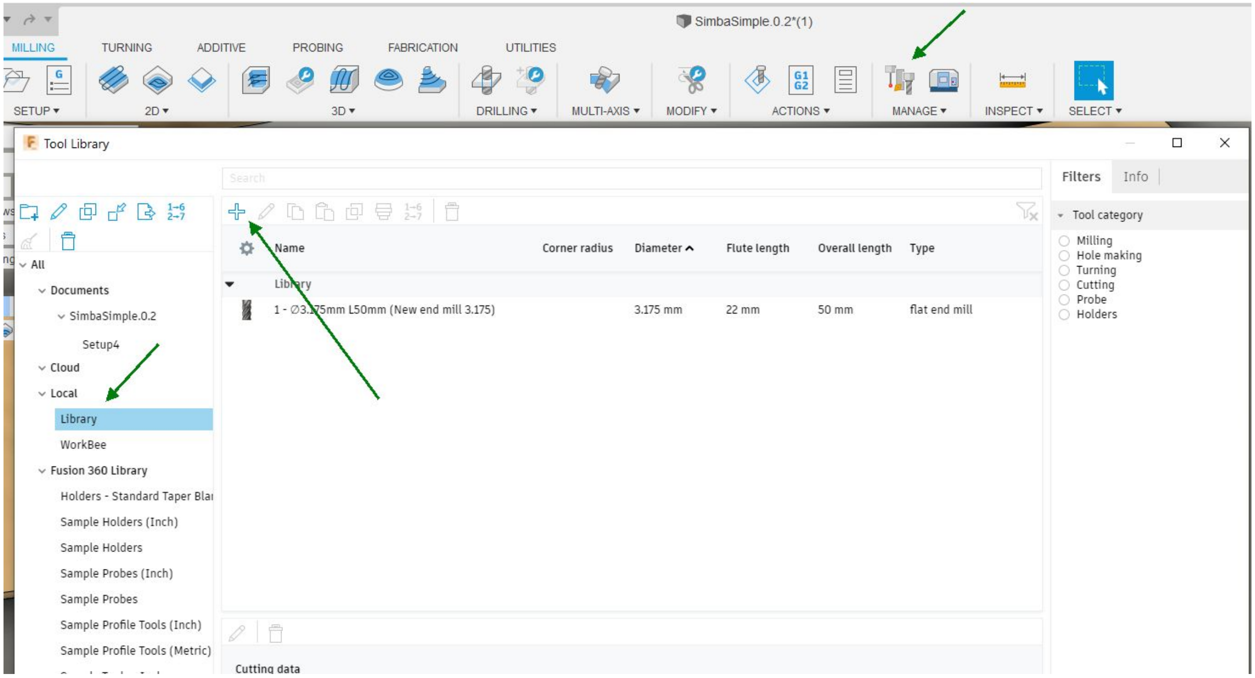


2. Creating the Tool

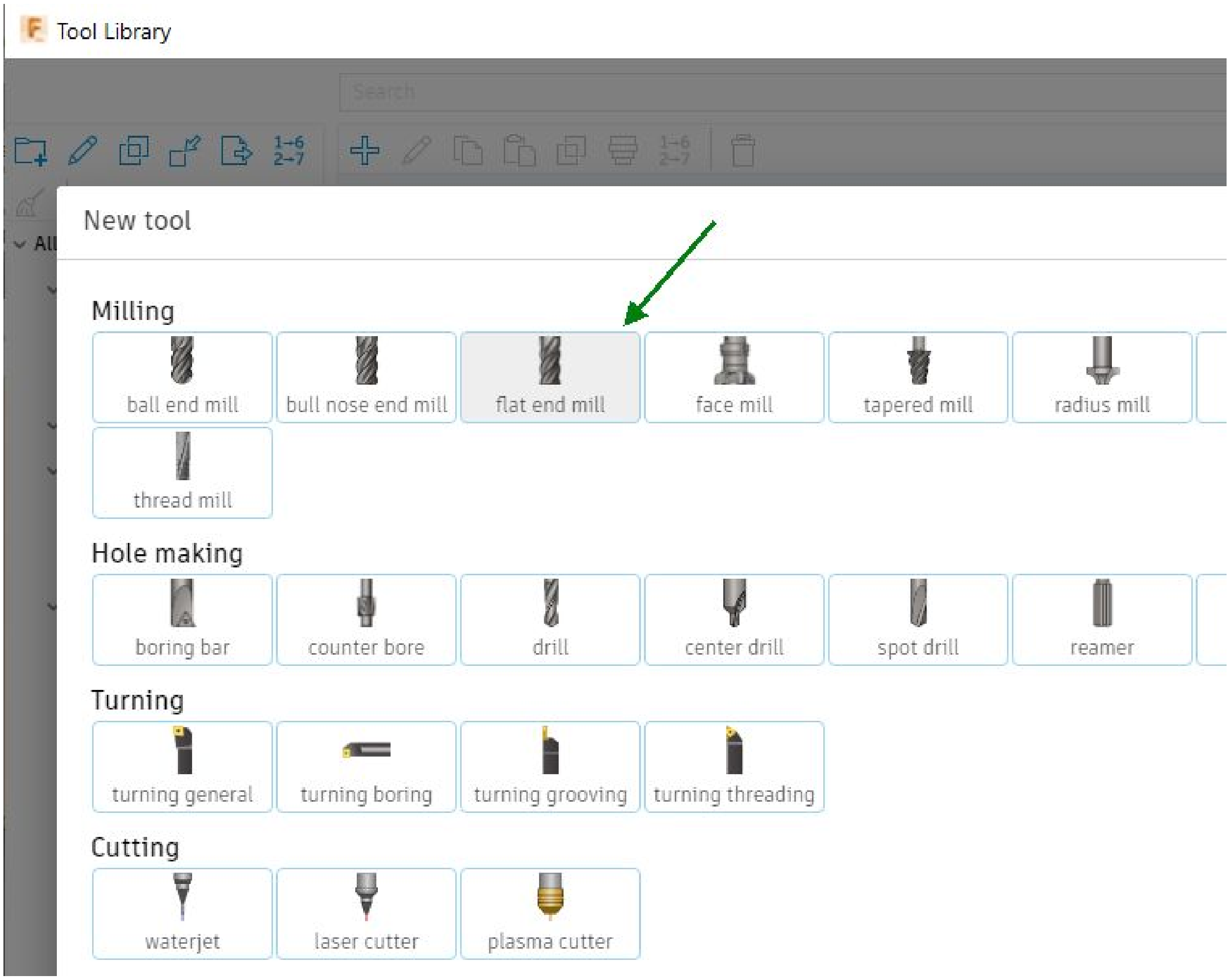
2.1 Switch to the Manufacture mode:



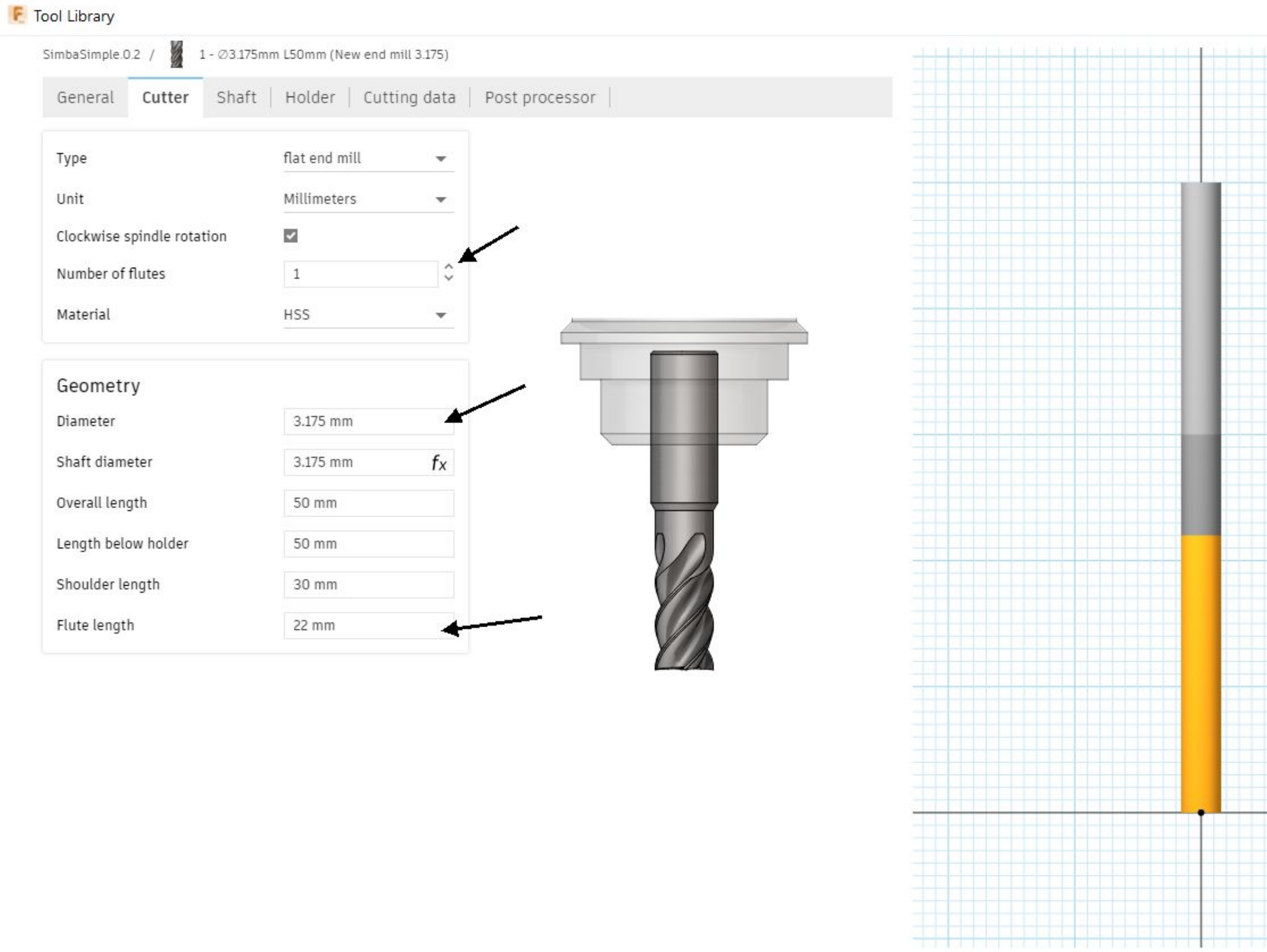
2.2 Click Tool Database Icon, go to Local Library and Click the plus icon to add new tool:



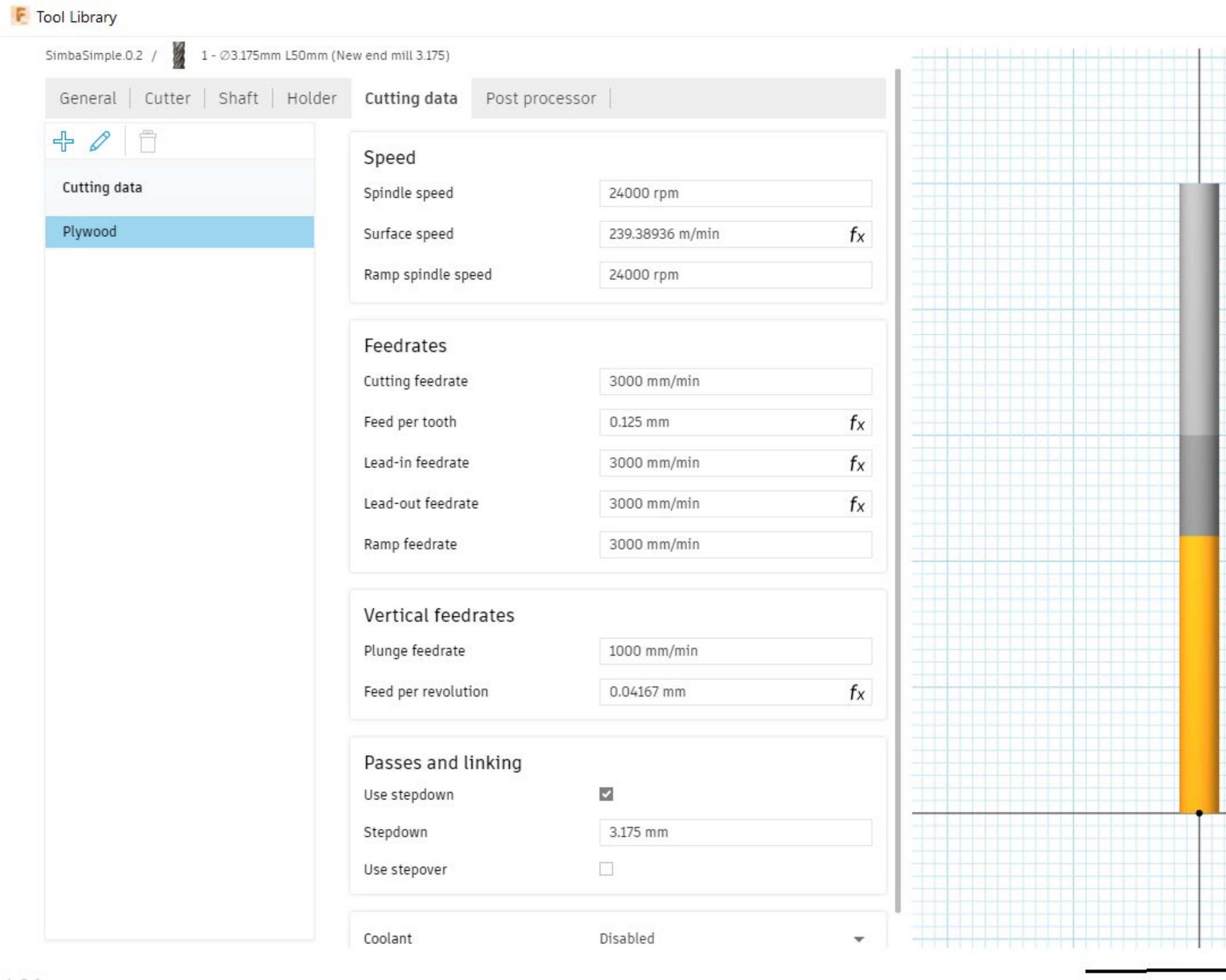
2.3 Select Flat end mill:



2.4 Measure your tool and enter the data in the corresponding fields:



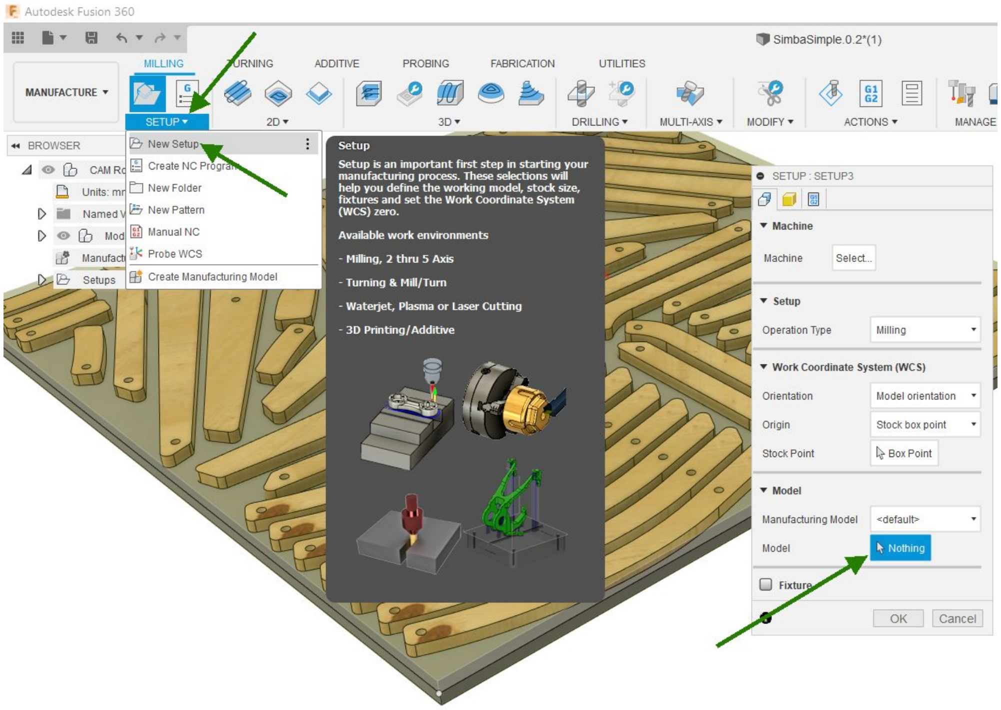
2.5 Go to the cutting tab and enter the speeds, calculated earlier:



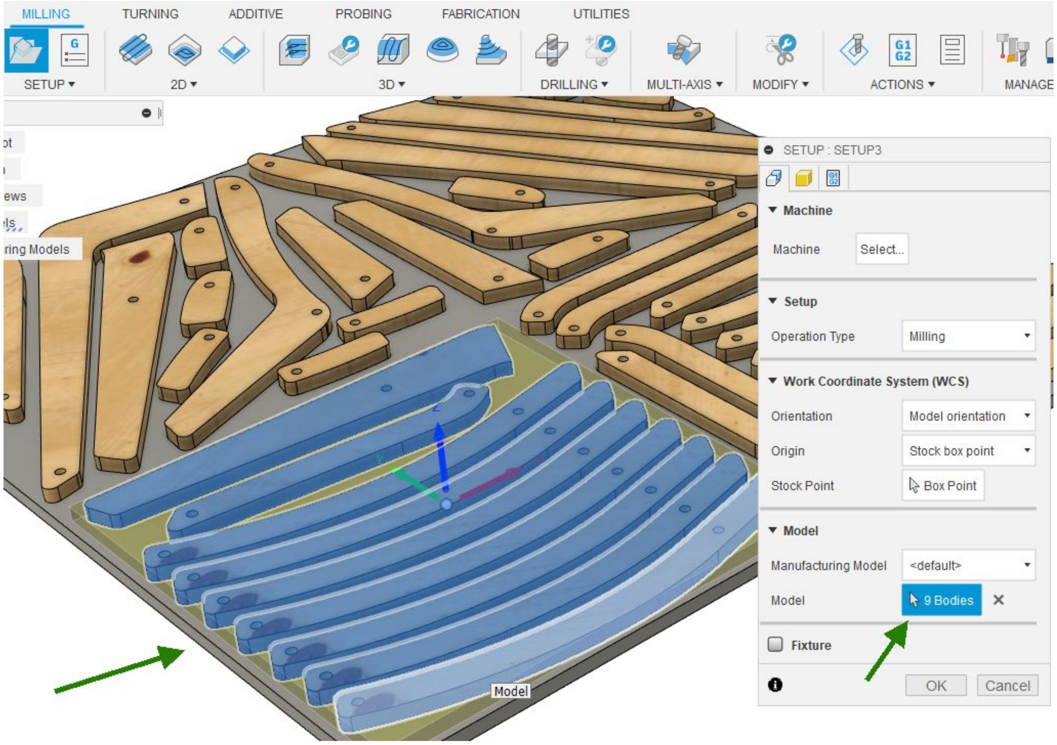
2.6 Accept to save the tool.

3. Creating a Setup

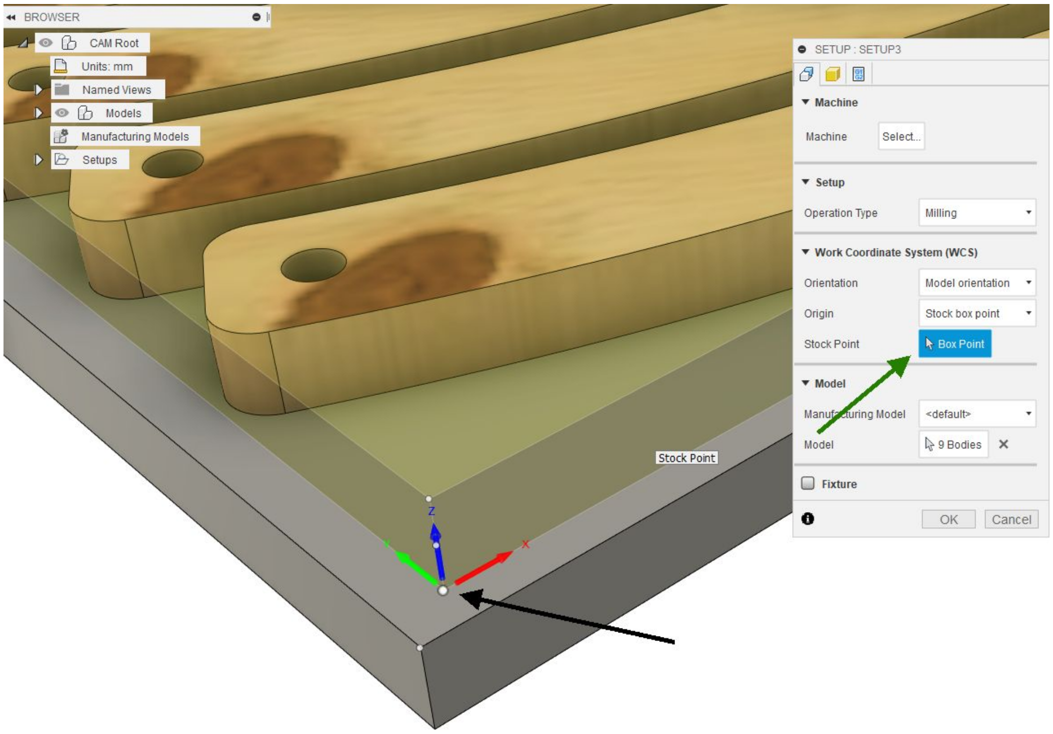
3.1 Create a new Setup and Click “Nothing” in the model section tab:



3.2 Click on the parts that you want to include in the first cut:

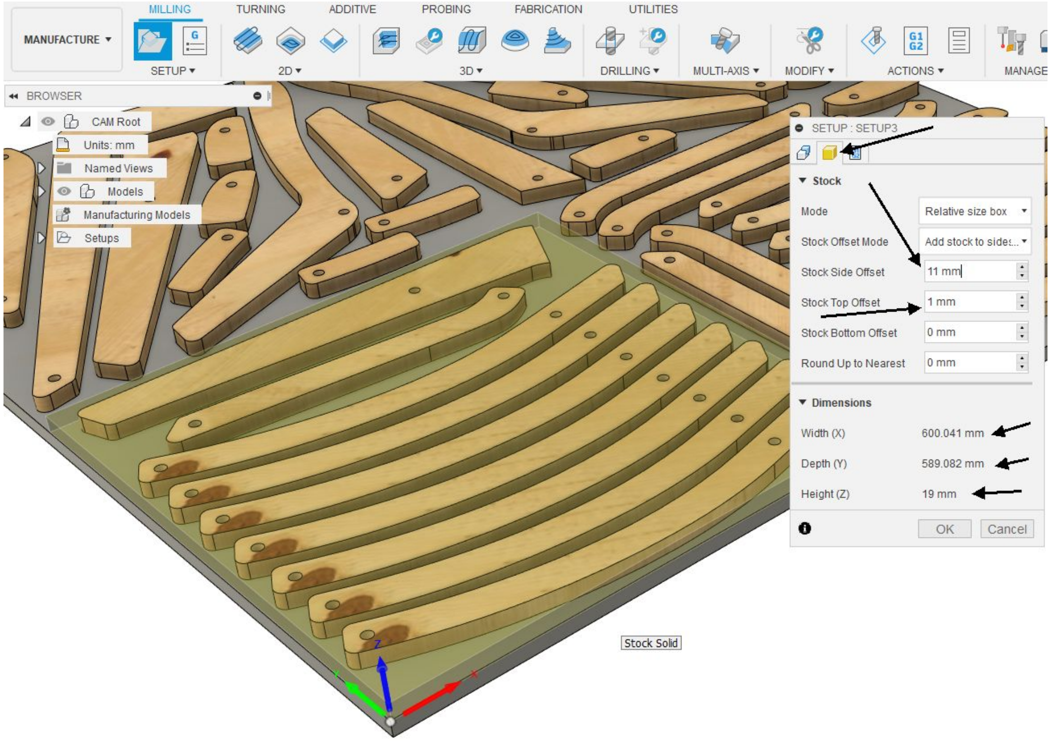


3.3 Click "Box Point" button, move the point to the corner and make sure the axises point according to your machine's axises:



3.4 If the axises are positioned wrong, fix them by selecting one of the options in the “Mode Orientation” drop-down.

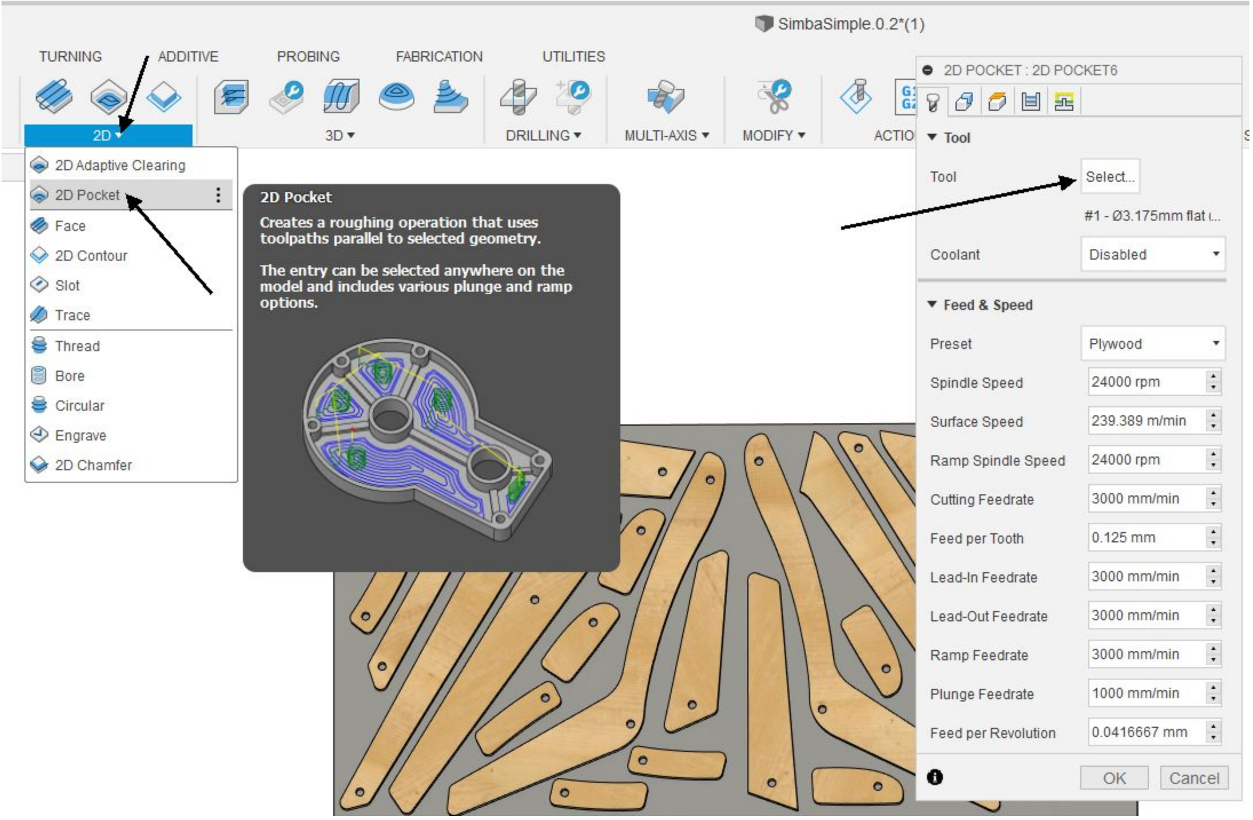
3.5 Go to the Stock tab and select “Relative size box”. Top up the side and top offsets to be as close as possible to a 600 x 600 x 19 mm box or 1200 x 1200 x 19 in case your machine can handle 4 tiles at once:



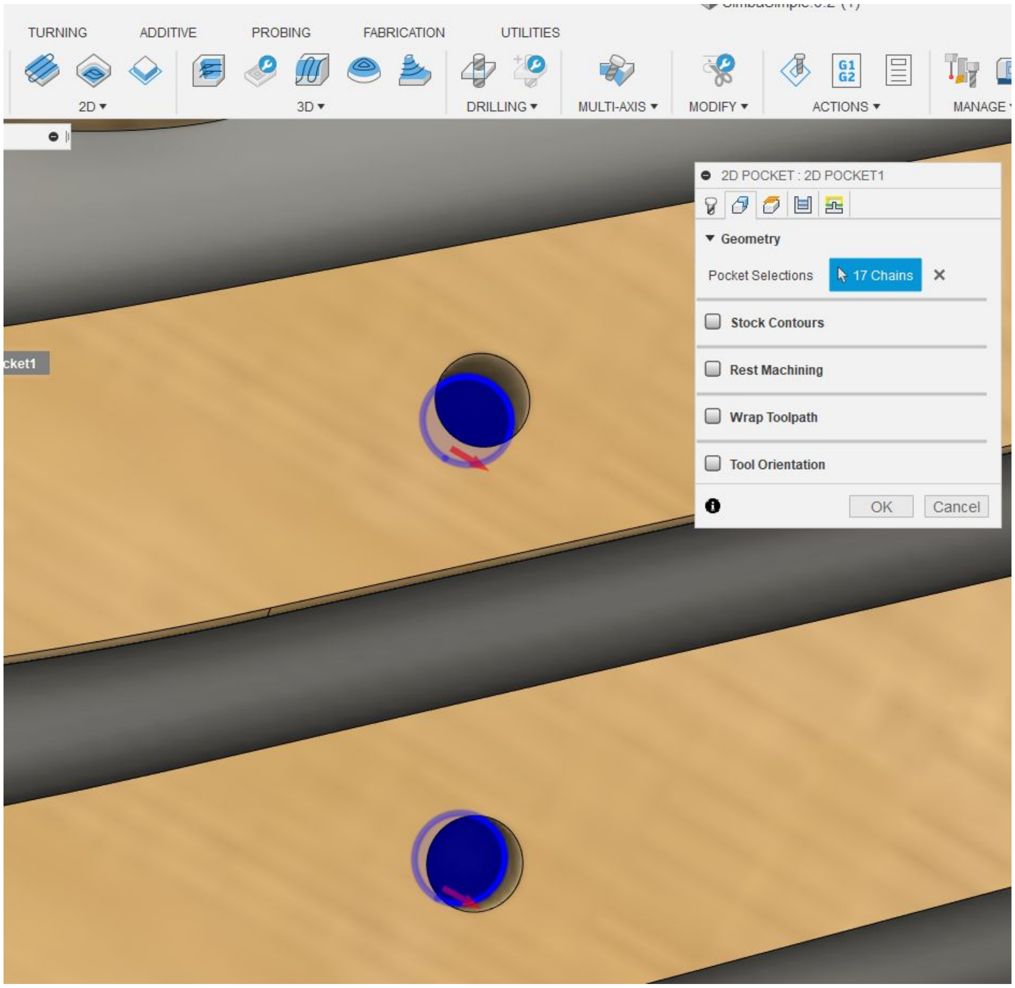
3.6 Click OK to save the setup.

4. Creating Paths for the Holes

4.1 Select 2D Pocket operation and choose the tool, created earlier:



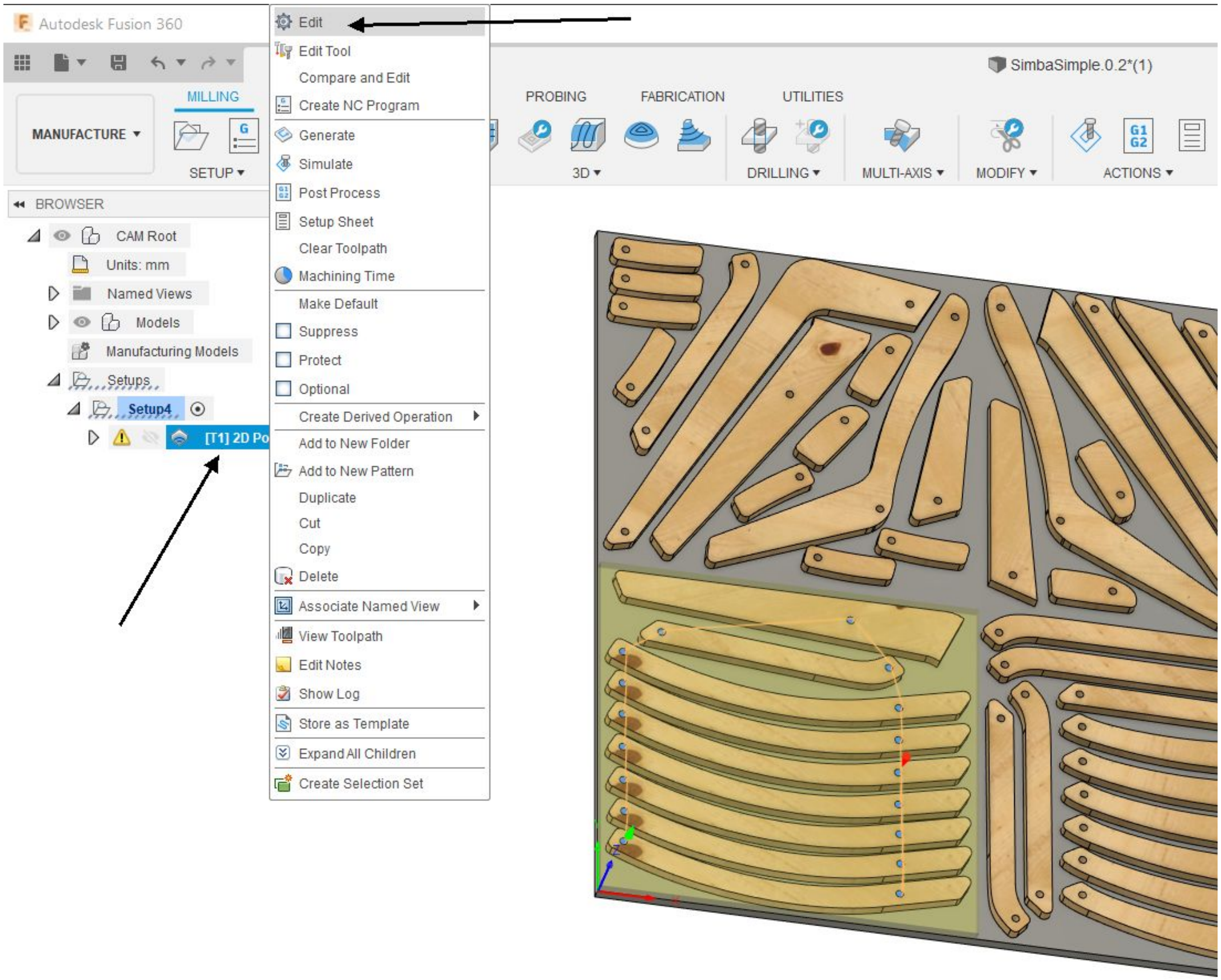
4.2 Click on the holes bottom contours of the holes:



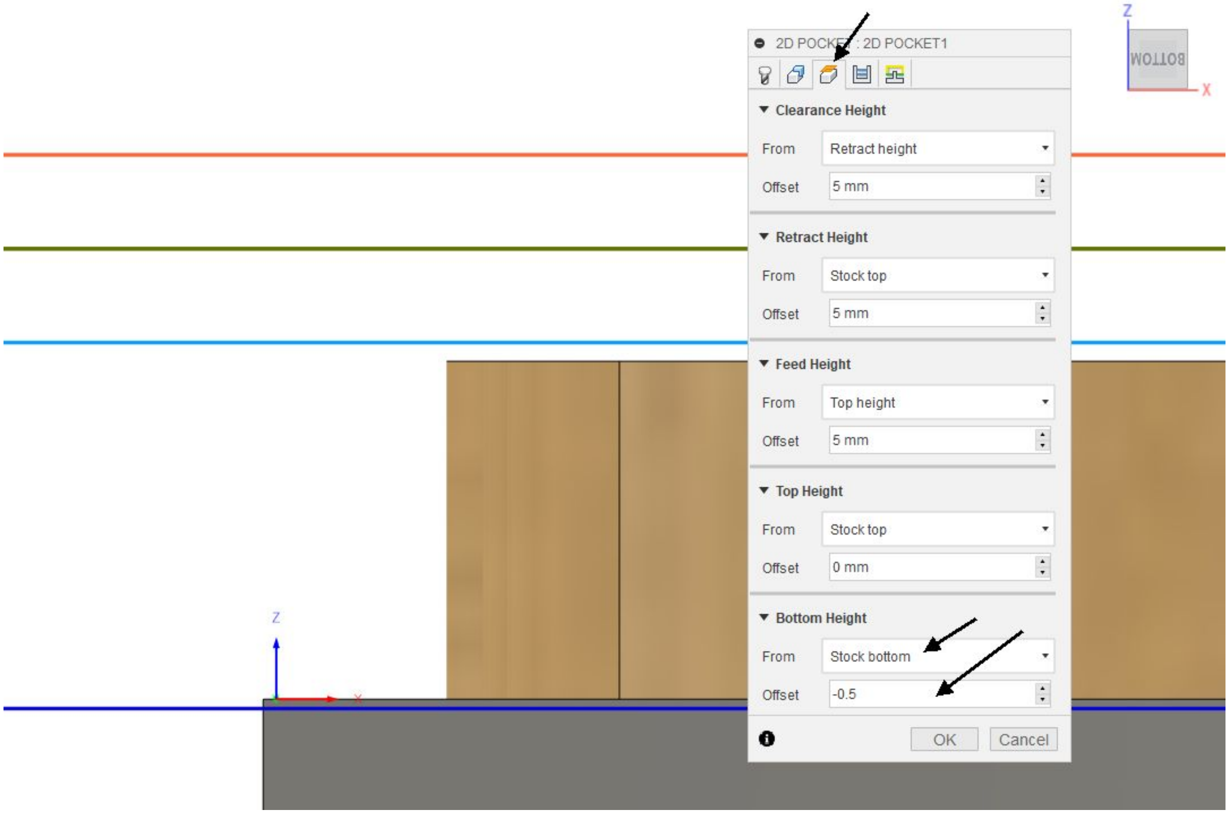
4.3 Make sure that the red arrow is inside the circles, otherwise the cut will be performed outside of the circles.

4.4 Save the operation, ignoring any warnings for now. The operation is saved at this point to prevent loss of geometries selection. Select the tool created earlier when prompted.

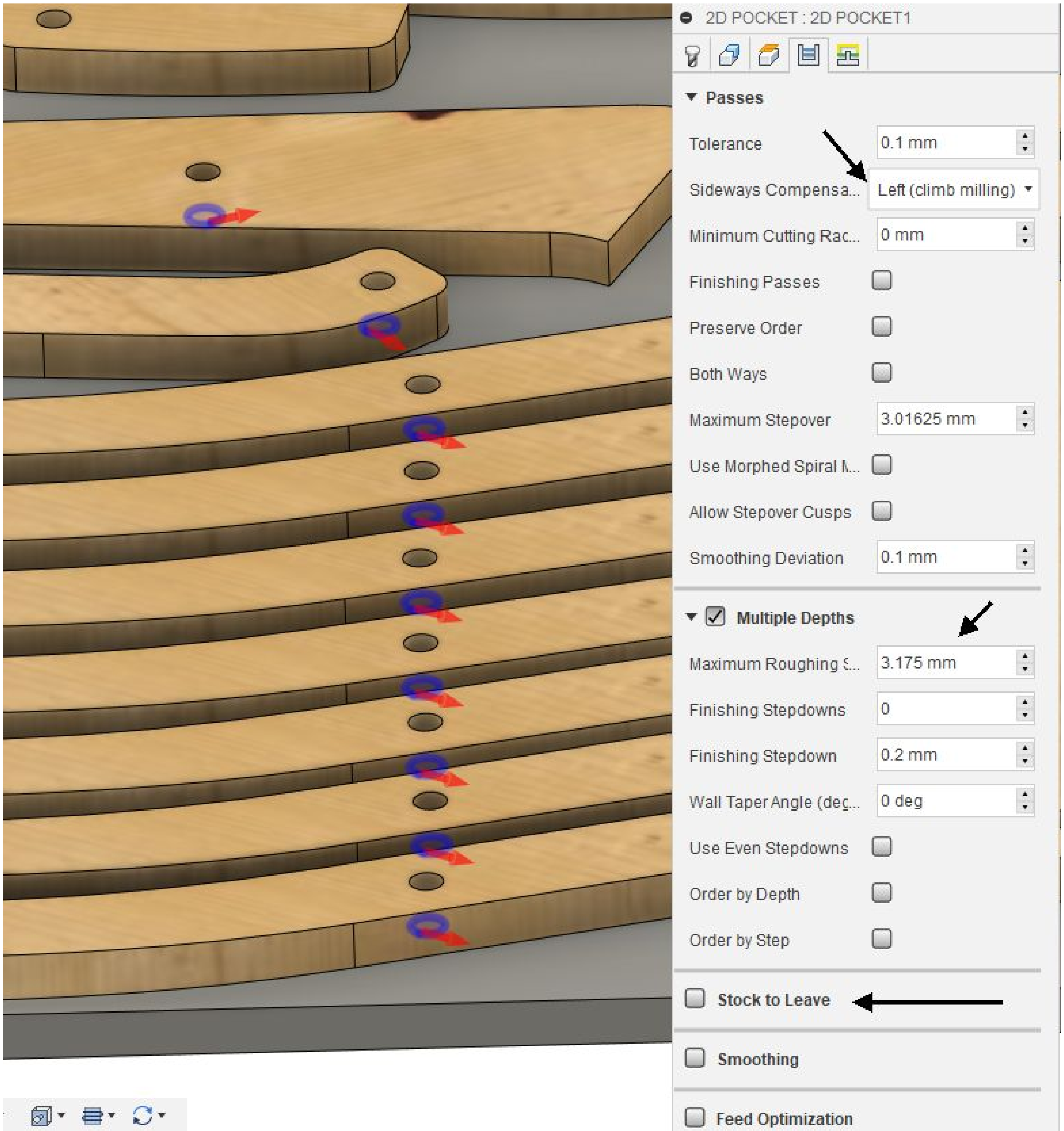
4.5 Click to edit the created operation:



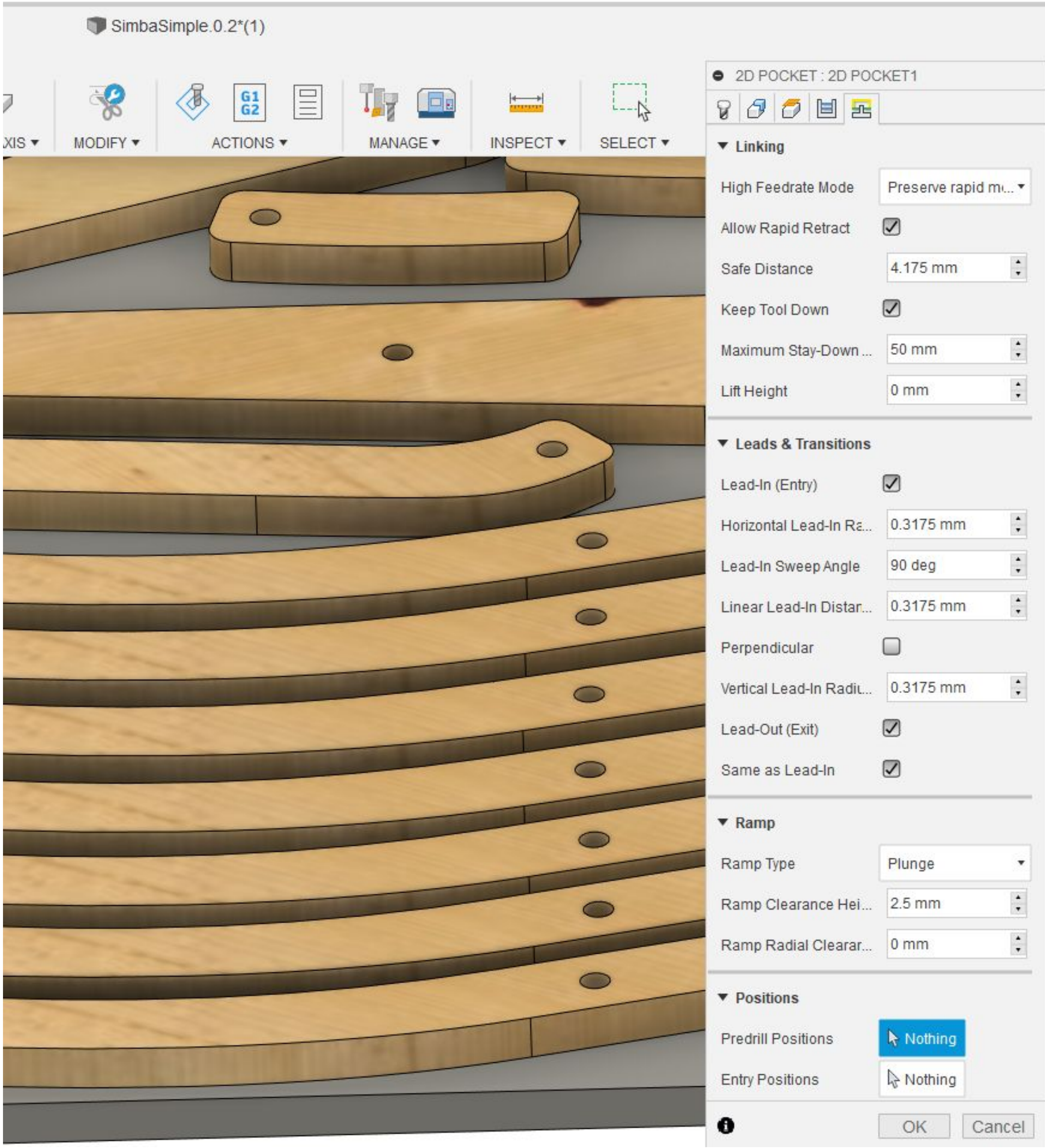
4.6 Go to the “Heights” tab and make sure the Bottom Height is set to stock bottom. It is safer to go -0.5 mm deeper into the spoil board to make sure all parts will be easy to remove after the cut:



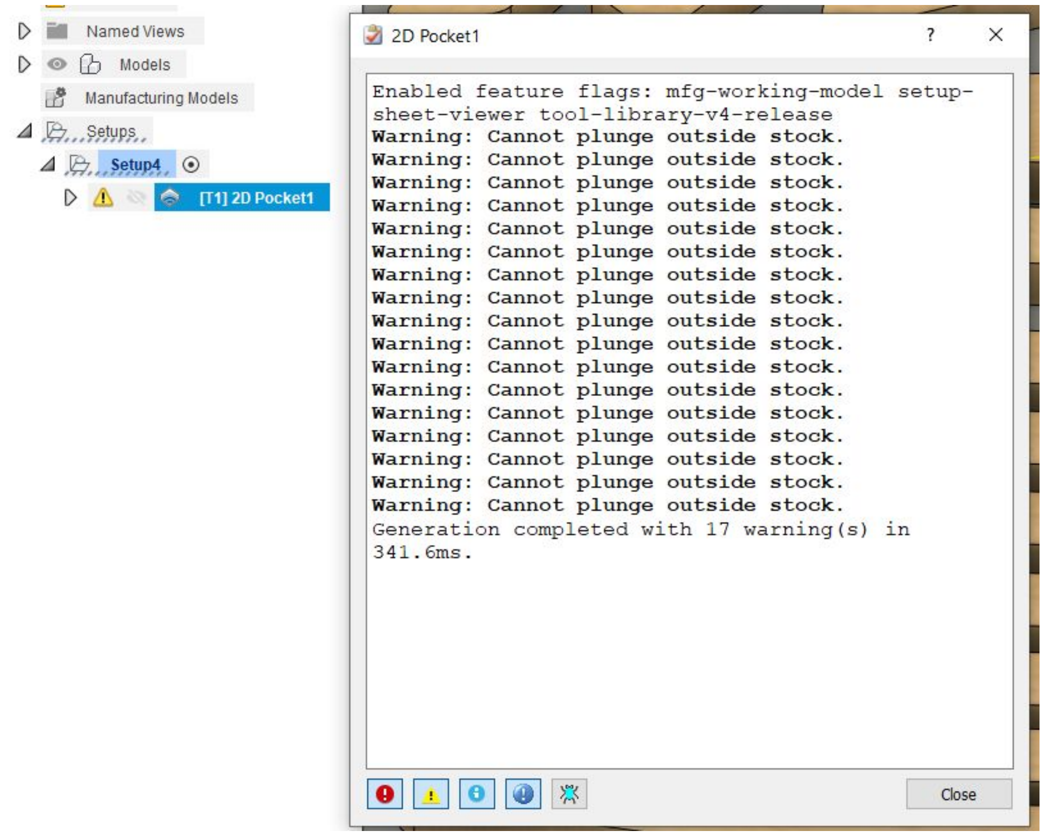
4.7 Go to the passes tab and make sure “Stock to Leave” is disabled. Set the rest of the setting as follows:



4.8 Linking tab settings:

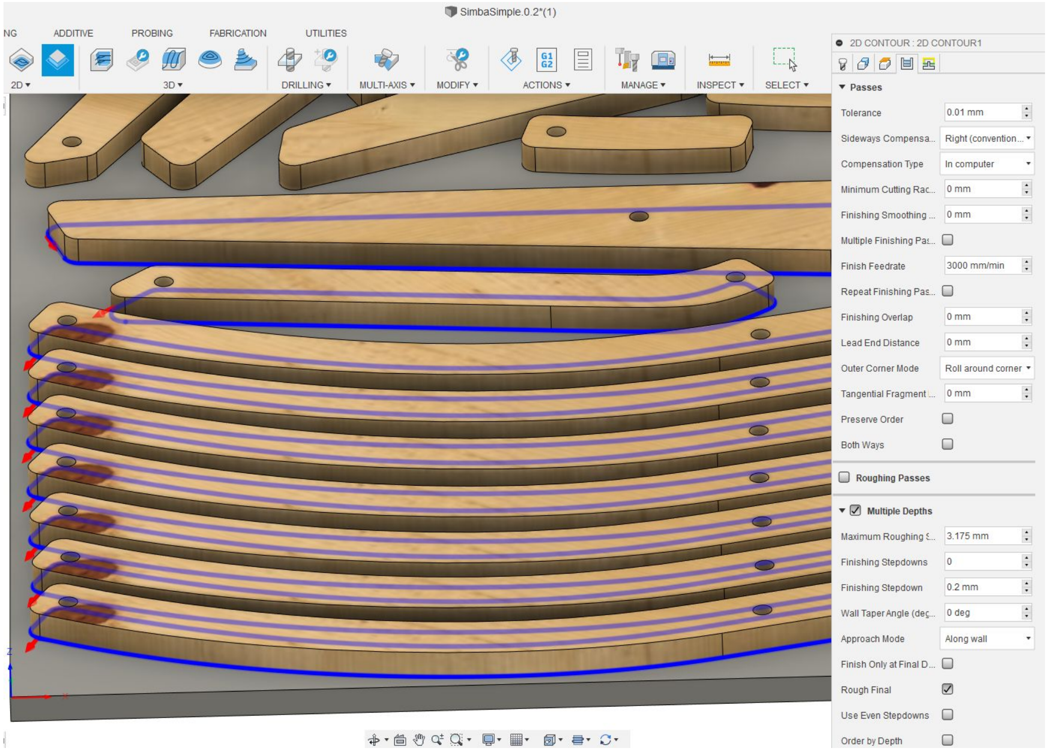


4.9 Click OK to save the operation. There is a chance you will get these warnings when the path is generated. It is OK to ignore them:

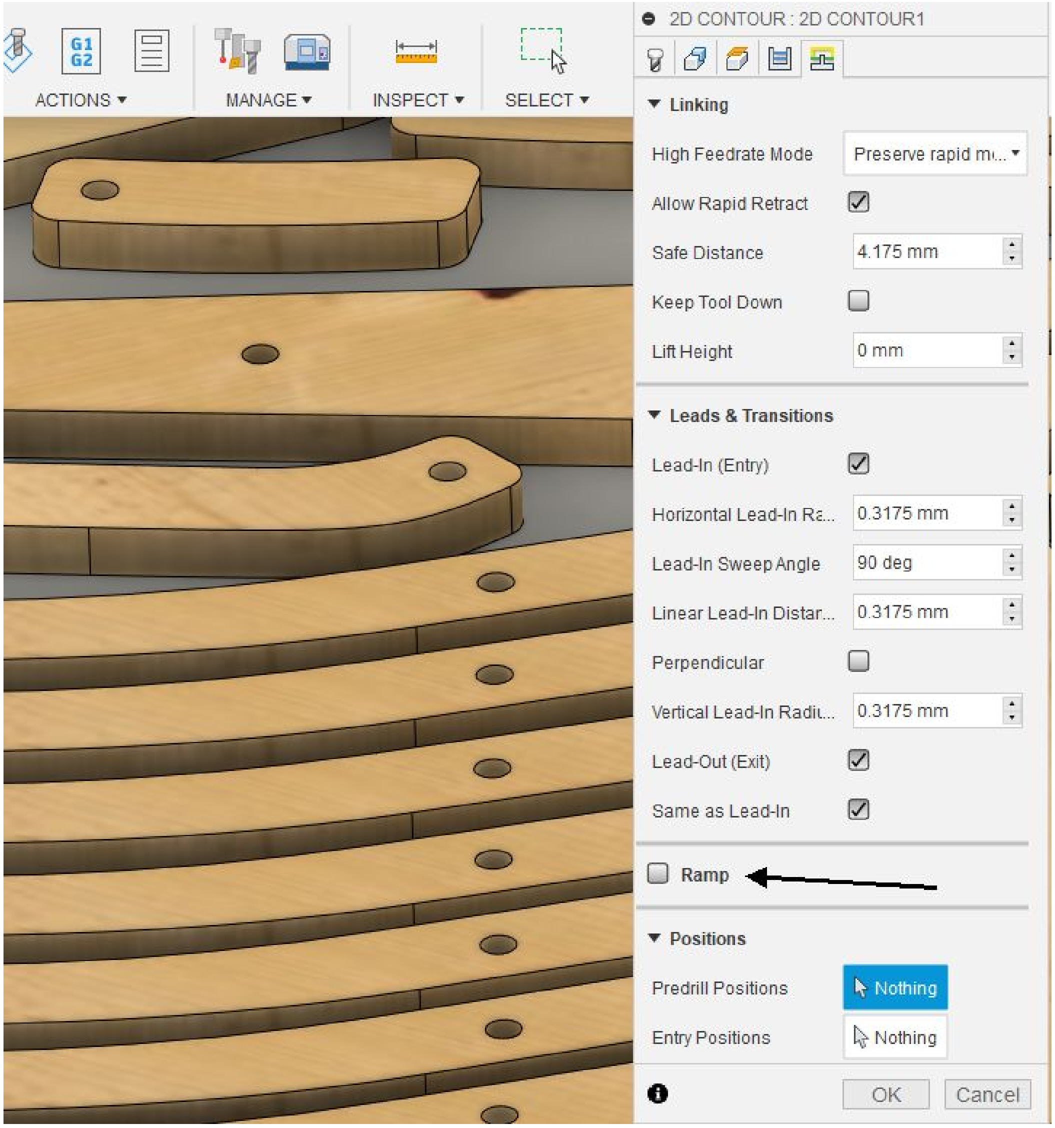


5. Creating Paths for the contours

5.1 Create a 2D contour operation, select all the contours of the parts and use the same settings as for the pockets. The only difference is the sideways compensation in the passes tab, that should be set to “Right (Conventional milling)”:

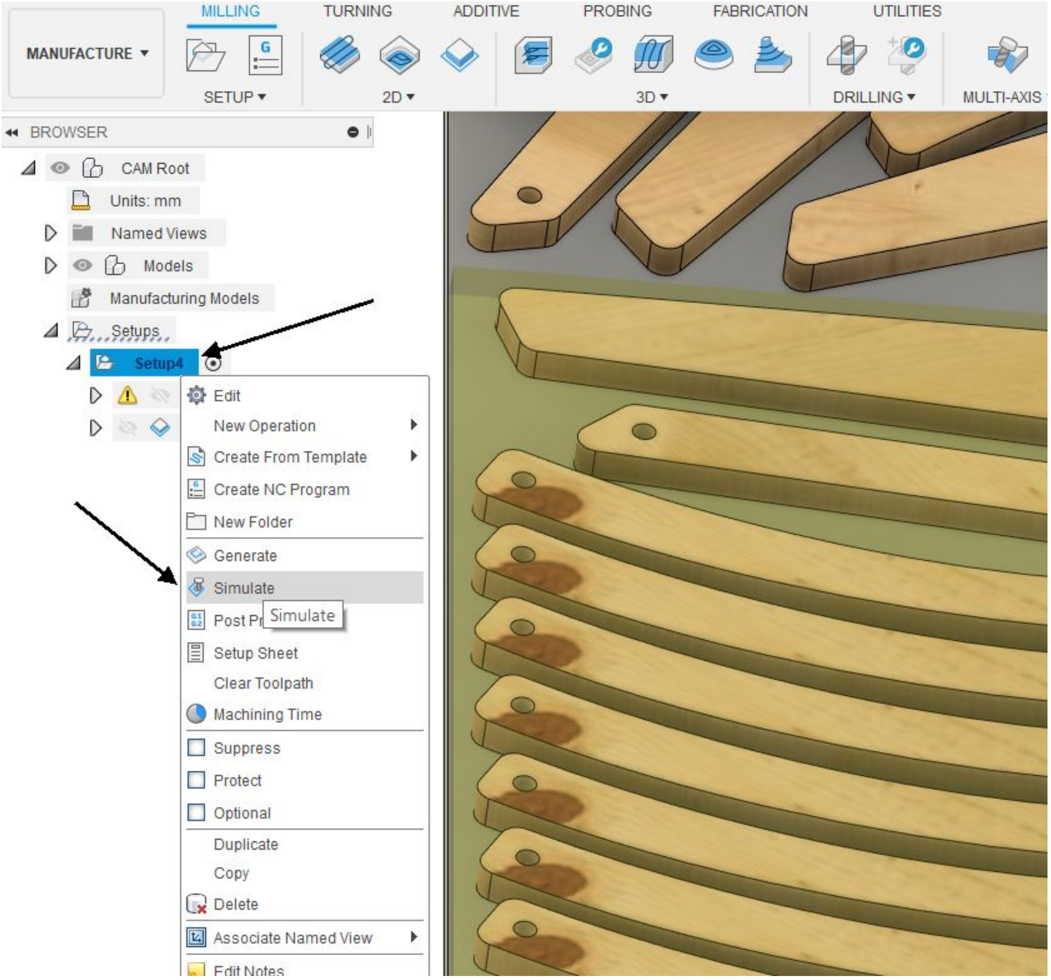


5.2 Disable Ramp in the linking tab:

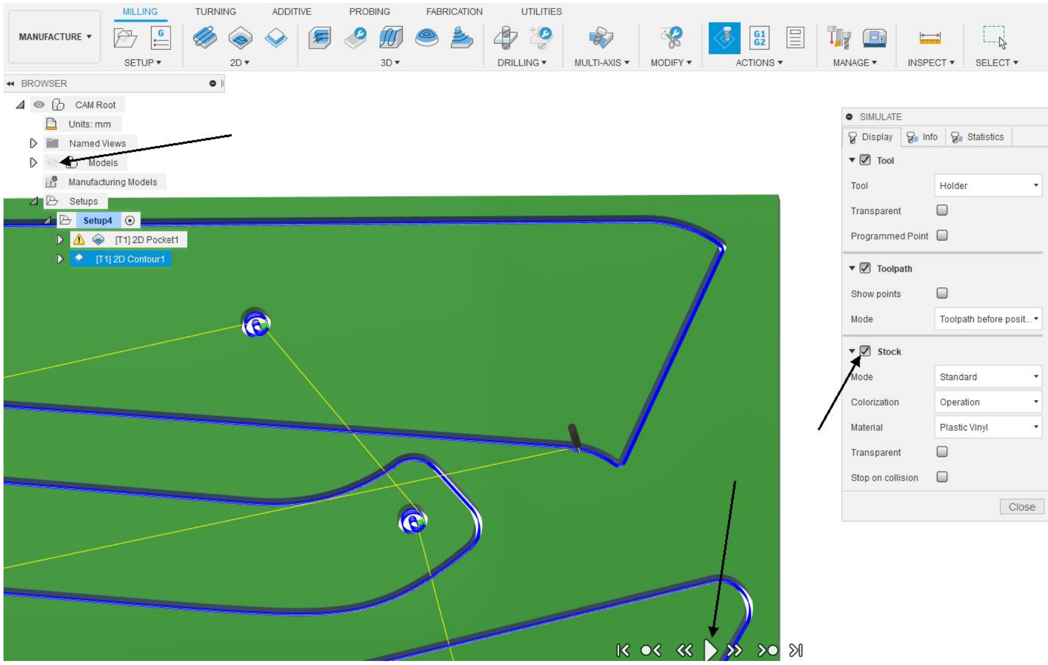


6. Simulating

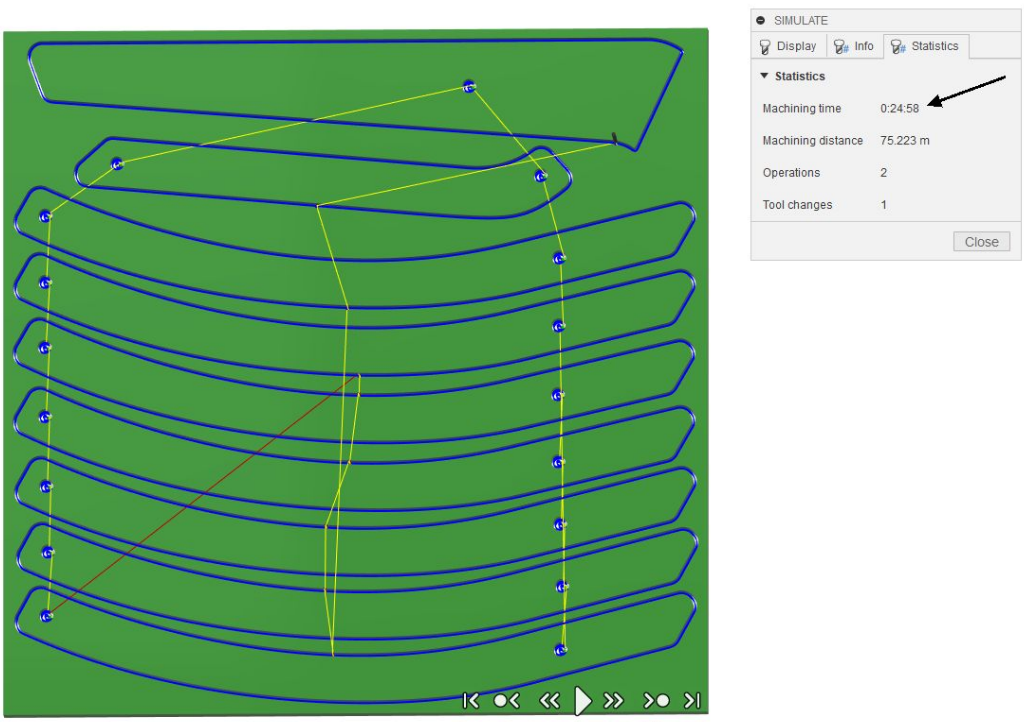
6.1 Click OK to save the operation and simulate the setup:



6.2 Enable Stock, watch the simulation:

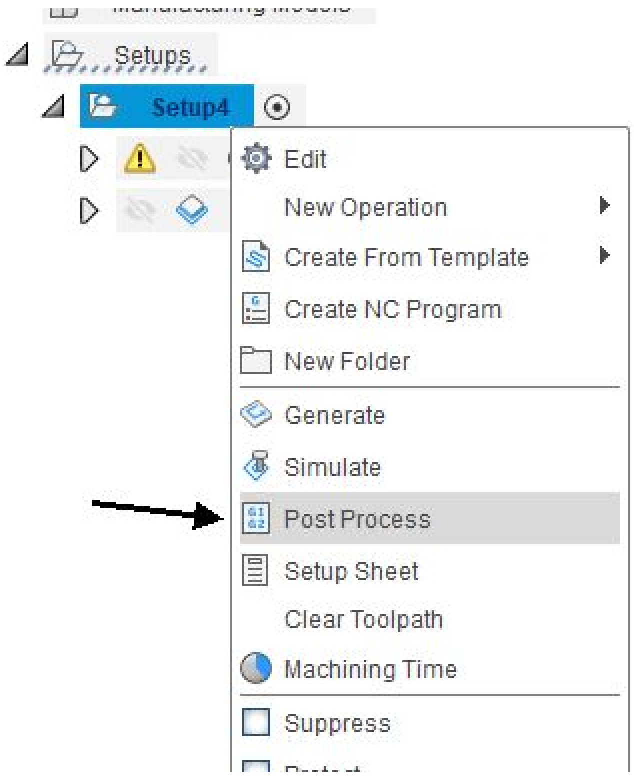


6.3 Check that the machining time is reasonable:

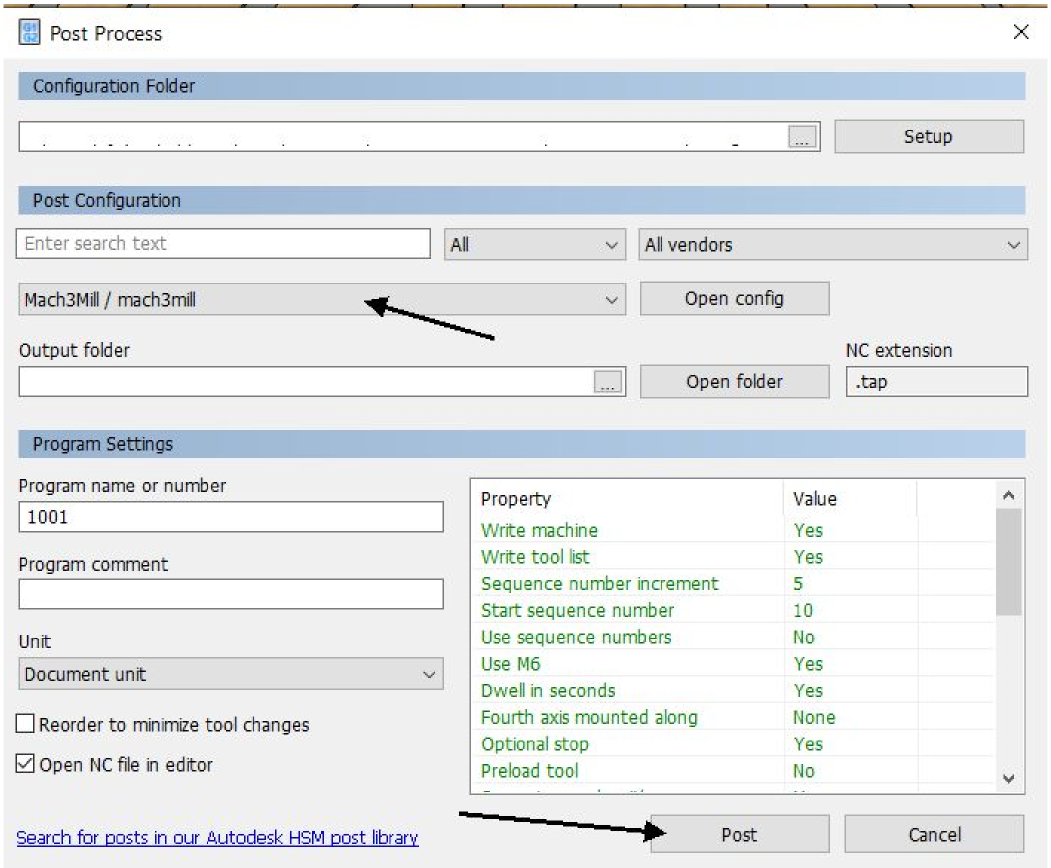


7. Post Processing

7.1 Post Process the created Setup:



7.2 Save the GCODE:



You can install a post processor for your machine following these links: Mach3, GRBL, X-Carve, Shapeoko and WorkBee.

7.3 Repeat the steps for the rest of the tiles.