

# Sheet Metal Machining Setup Guide

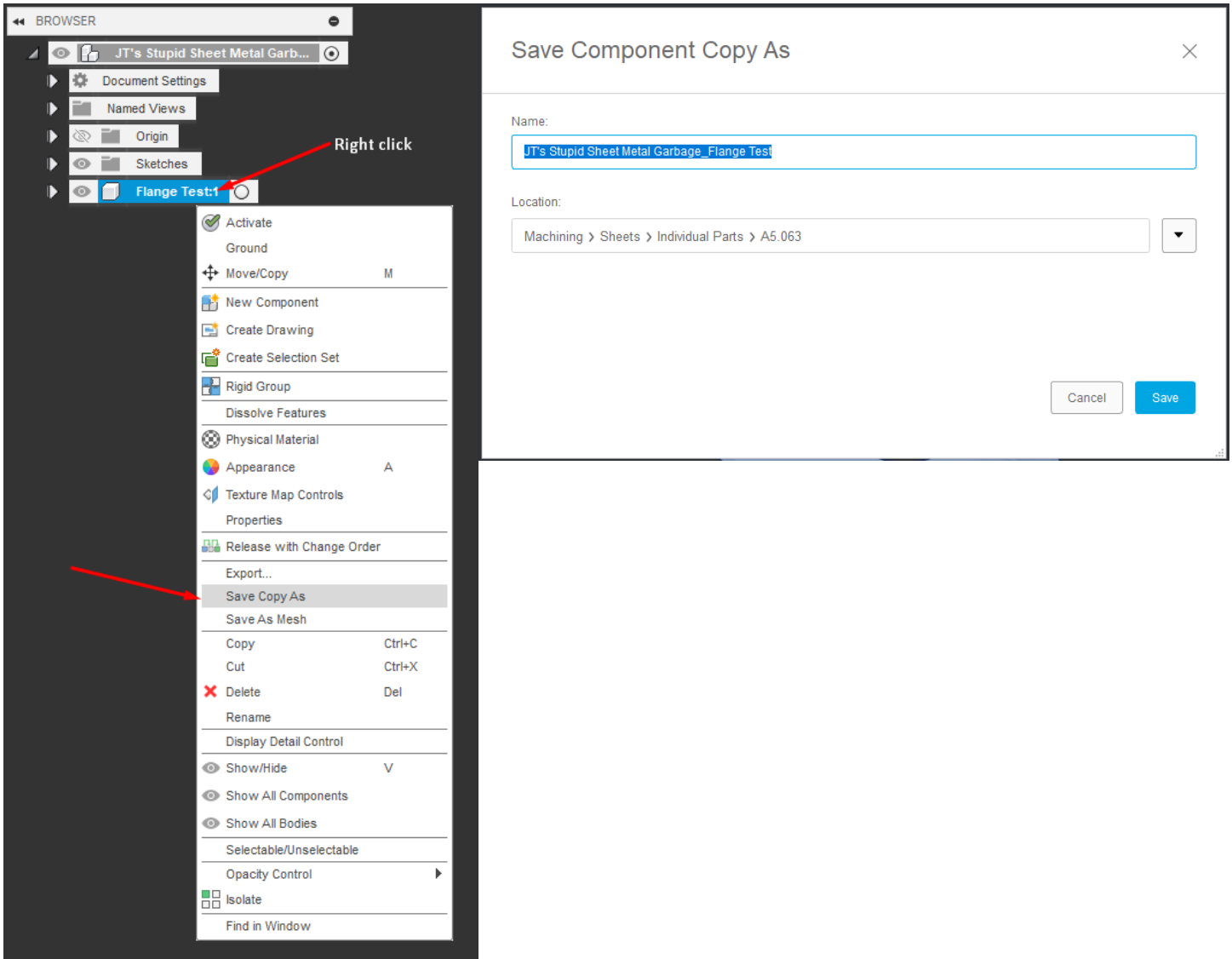
Document 231010b

Revised Oct. 10, 2023

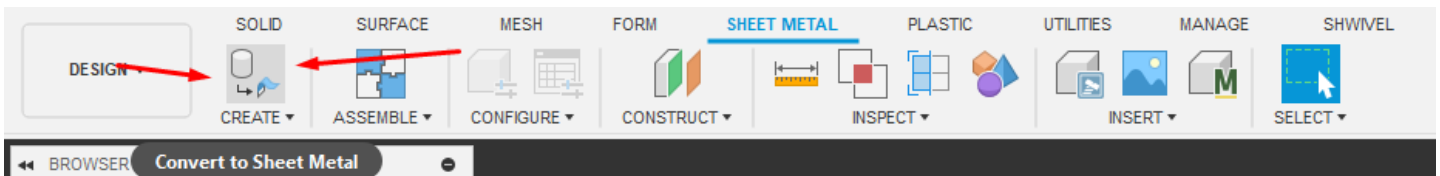


This guide will walk you through the process of machining sheet metal parts using Autodesk Fusion 360.

Right click the component and click “Save copy as”, then save the copy into “Fusion 360/Machining/Sheets/Individual Parts/A5.XXX”. Replace “XXX” with the material thickness. “A5” means “Aluminum 5052”. Use a “PC.XXX” folder for polycarbonate parts.



Open the part, navigate to the “SHEET METAL” tab, and click “Convert to sheet metal” as shown.



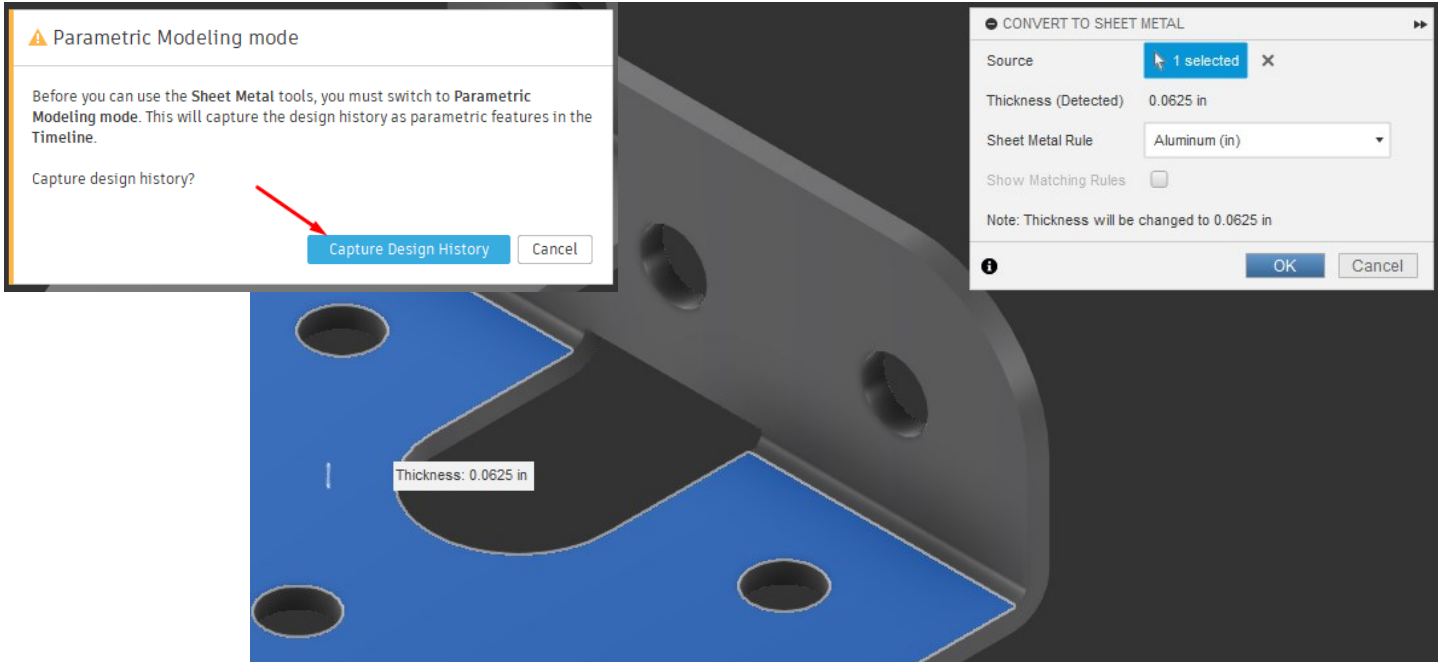
# Sheet Metal Machining Setup Guide

Document 231010b

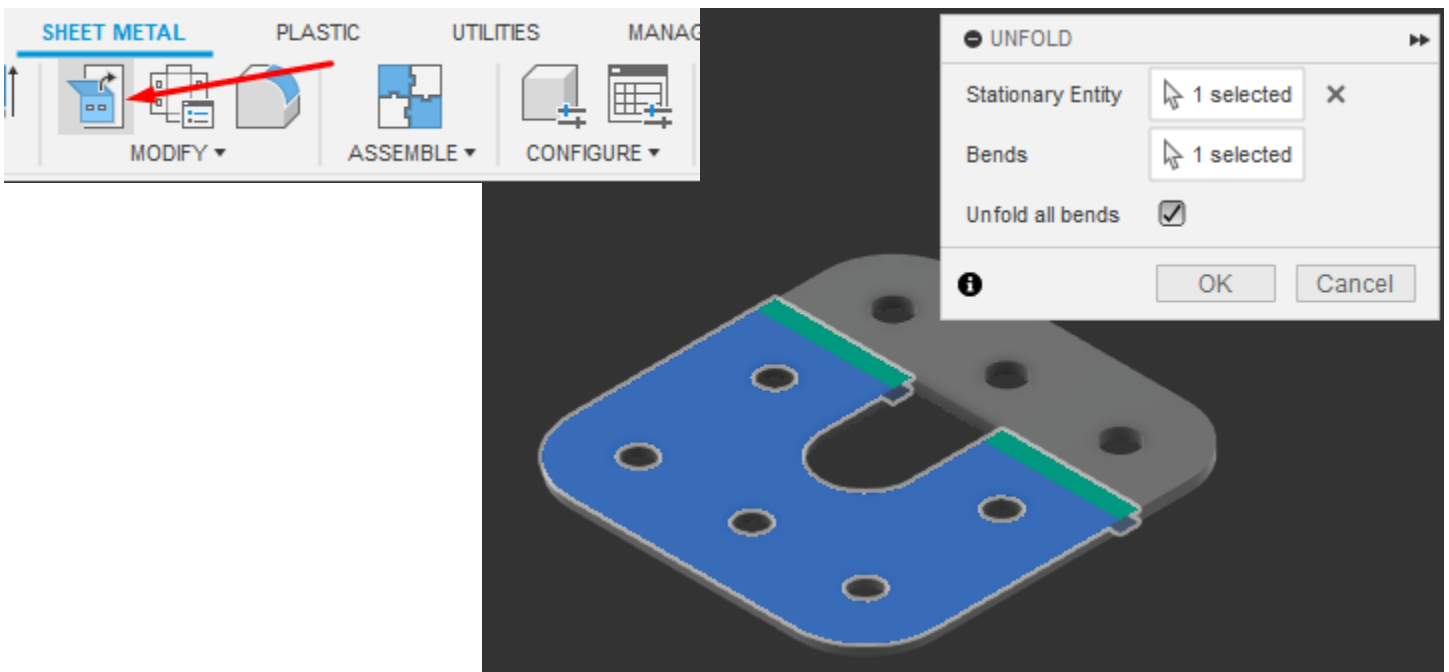
Revised Oct. 10, 2023



“Capture design history”, select any flange face, and select “Aluminum (in)” for the Sheet Metal Rule., then click “OK”.



Click “Unfold”, click any flange face again, check “Unfold all bends” and click “OK” to unfold the part. The part should now look like the image below. Save when this is finished.



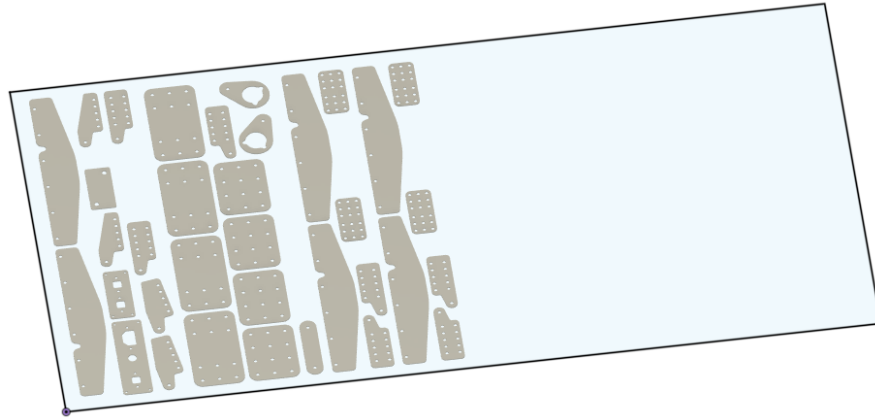
# Sheet Metal Machining Setup Guide

Document 231010b

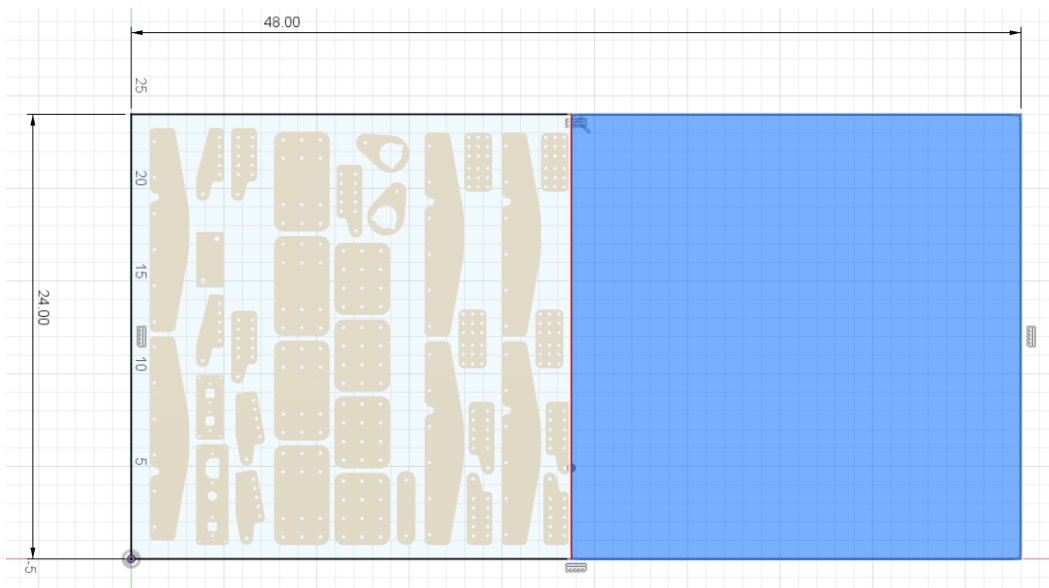
Revised Oct. 10, 2023



Go to the folder “Fusion 360/Machining/Sheets” and find the sheet with the right prefix (A5 or PC) that has the lowest letter designator that will fit your part(s). For example, if both A5.063a and A5.063b have room for your part, choose sheet A5.063a. In this case, it looks like A5.063a has room.



Since the sheet is partially used, arranging the new parts onto the sheet with the full rectangle won't work. Modify the original sketch or create a new sketch to draw the shape of the left-over material before using the arrange tool. Geometry from the existing parts can be projected for use as reference points/lines for line placement.



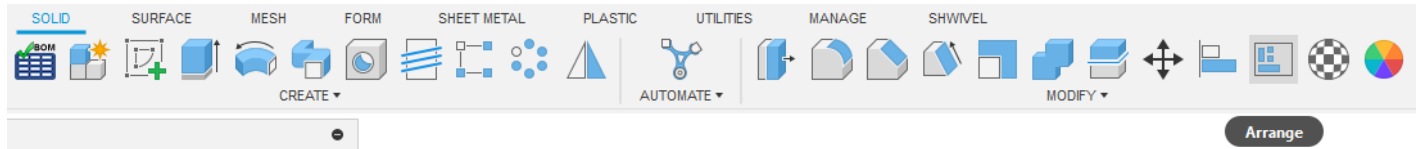
# Sheet Metal Machining Setup Guide

Document 231010b

Revised Oct. 10, 2023

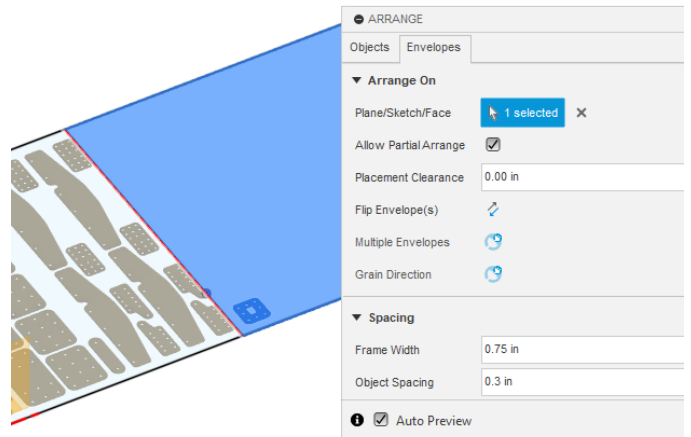
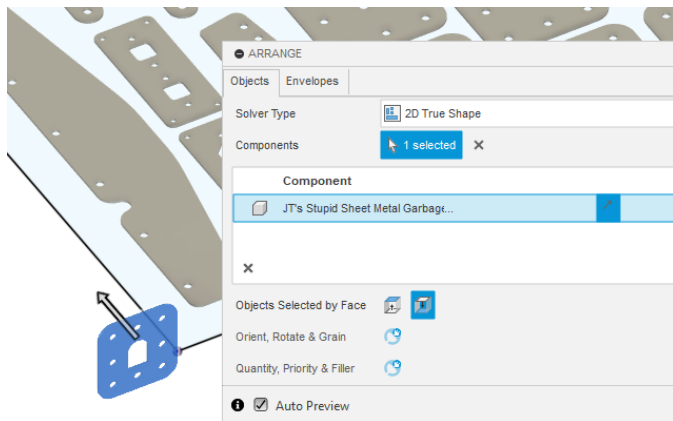


Drag the new part(s) into the sheet file, then use the “Arrange” tool with the settings shown. TIP: Drag the arrange tool from the dropdown menu up to the Toolbar so it's more accessible. If you don't you'll have to open the “MODIFY” dropdown menu to select the tool.



Components: Select the parts to be machined.

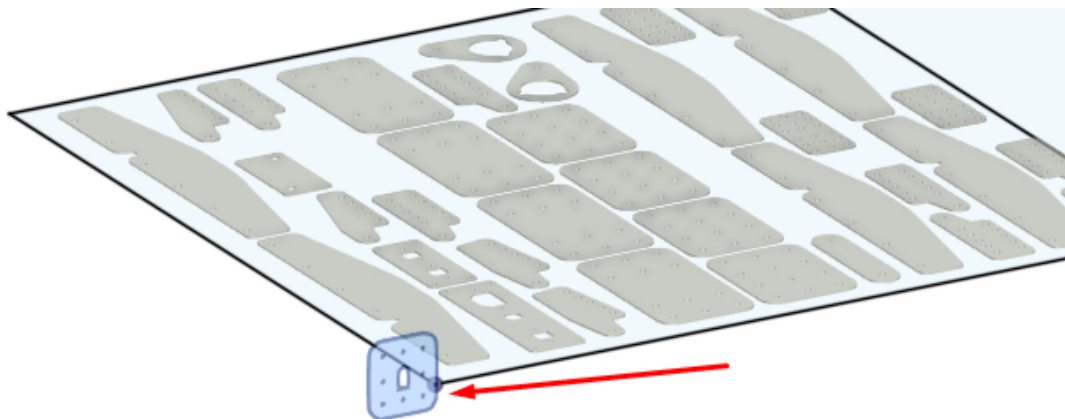
Envelopes: Select the sketch profile that represents the remaining material. Use 0.75 in Frame Width and 0.30 in Object Spacing



If the parts are on the wrong side of the sketch when compared to the other parts like shown below, click “Flip Envelope(s)”. Parts should be above the sketch.



Hide the original parts that are floating in space once the arrangement has been made.



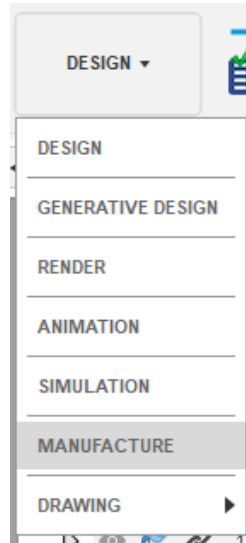
# Sheet Metal Machining Setup Guide

Document 231010b

Revised Oct. 10, 2023



Move to the manufacturing workspace to begin creating G codes for the CNC router.



Suppress and unsuppress operations as necessary. Here is an overview of what each operation is used for:

- **X.XXX Drill:** Drill holes in diameter “X.XXX”. Ask a mentor if a needed hole size is missing.
- **Bore xxx End Mill:** Bores holes of any diameter slightly larger than the end mill—useful for holes larger than around 1/4” in diameter.
- **Manual NC1 [Stop]:** Used to stop the machine between operations to add hold down screws. The machine stops at tool changes, so this is only needed if only bored holes are used.
- **Pocket/2D Adaptive 1/8”/4 mm End Mill:** Used to create internal pockets with the 1/8” or 4 mm end mill—rarely used.
- **Contour 1/8”/4 mm End Mill:** Used to create external contours with the 1/8” or 4 mm end mill—rarely used.
- **Thru 2D Adaptive & Thru Pocket Finish 6 mm End Mill:** Used to machine and finish full-depth pockets in materials—always used together.
- **Blind 2D Adaptive & Blind Pocket Finish 6 mm End Mill:** Used to machine and finish partial-depth pockets in materials—always used together—rarely used for sheet metal.
- **Contour 6 mm End Mill:** Used to machine outer full-depth contours for parts—sometimes used for large internal contours.
- **2D Chamfer:** Deburrs selected upper perimeters.

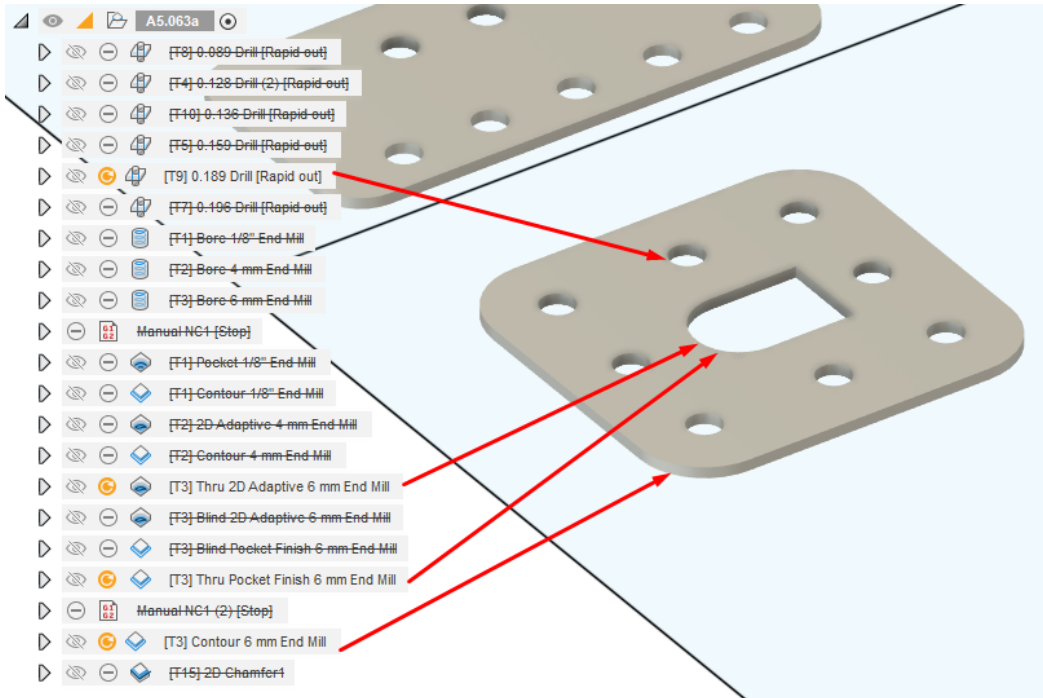
# Sheet Metal Machining Setup Guide

Document 231010b

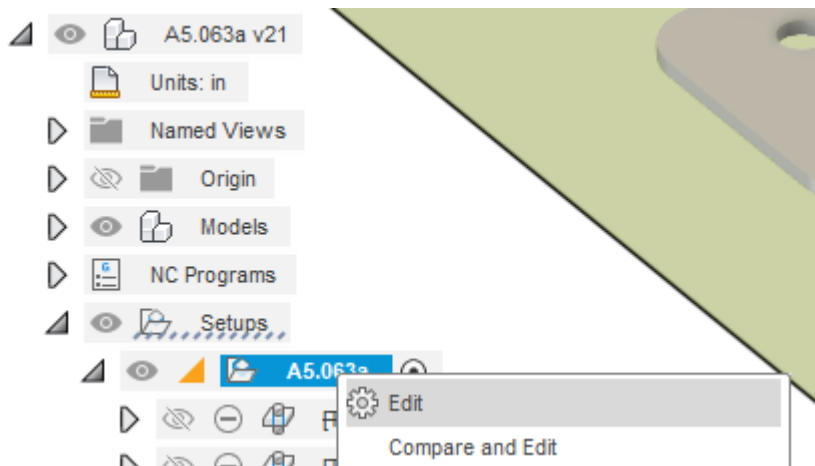
Revised Oct. 10, 2023



For the example part, “0.189 Drill”, “Thru 2D Adaptive 6 mm End Mill”, “Thru Pocket Finish 6 mm End Mill”, and “Contour 6 mm End Mill” are unsuppressed. All other operations are suppressed. To suppress or unsuppress, right click an operation and click “Suppress”.



Right click the sheet setup, click “Edit”, then reselect the model. Often you will be presented with a message “This operation has missing references, do you want to clear these selections?” Yes, the selections should be cleared. The same note applies nearly every time one is presented with that message.



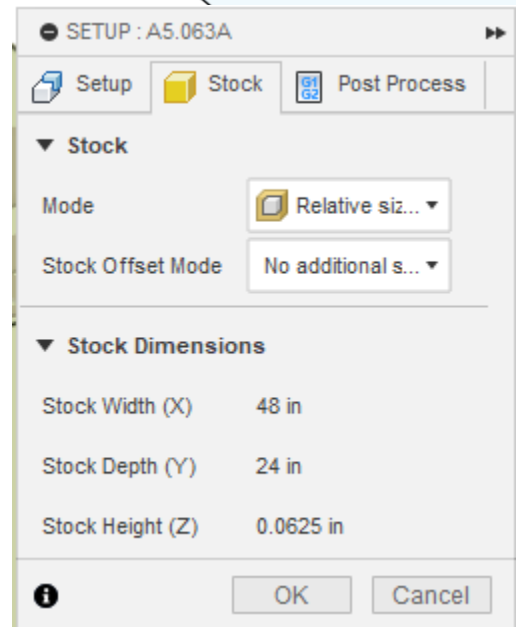
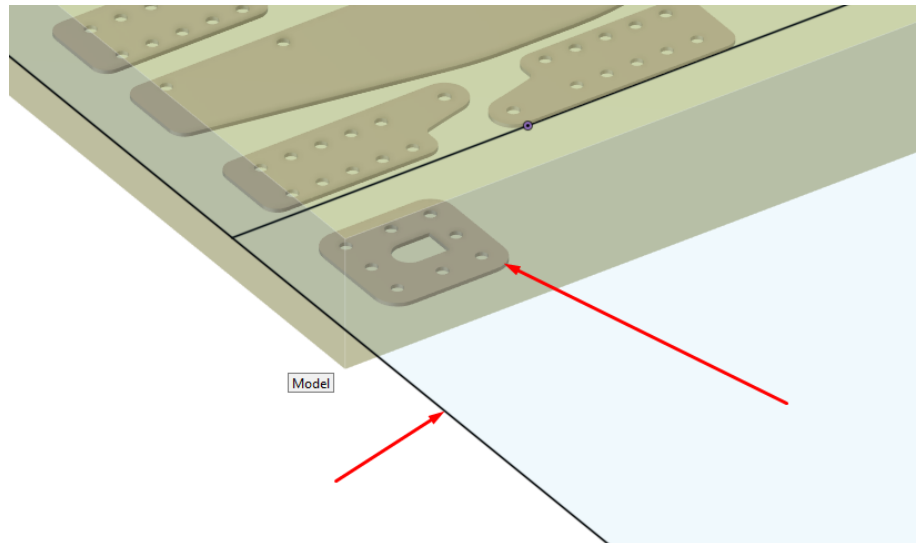
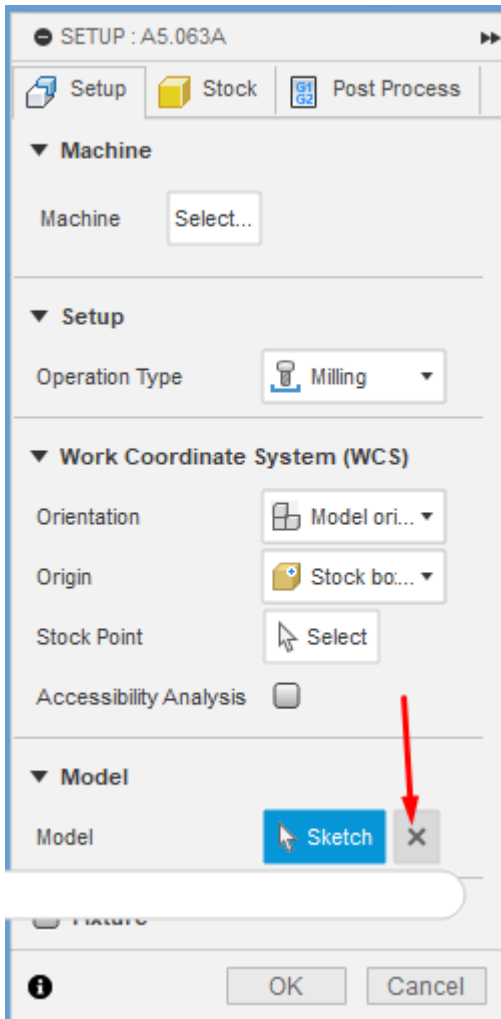
# Sheet Metal Machining Setup Guide

Document 231010b

Revised Oct. 10, 2023



Clear existing models in the “Setup” tab and select one of the sketch lines and the parts to be machined as the new model. To confirm things are correct, go to the “Stock” tab and verify the width, depth, and height are the same as the stock sheet being used. If there are differences, be sure the settings used are the same as what’s shown in the images below. Click “OK” when finished.



# Sheet Metal Machining Setup Guide

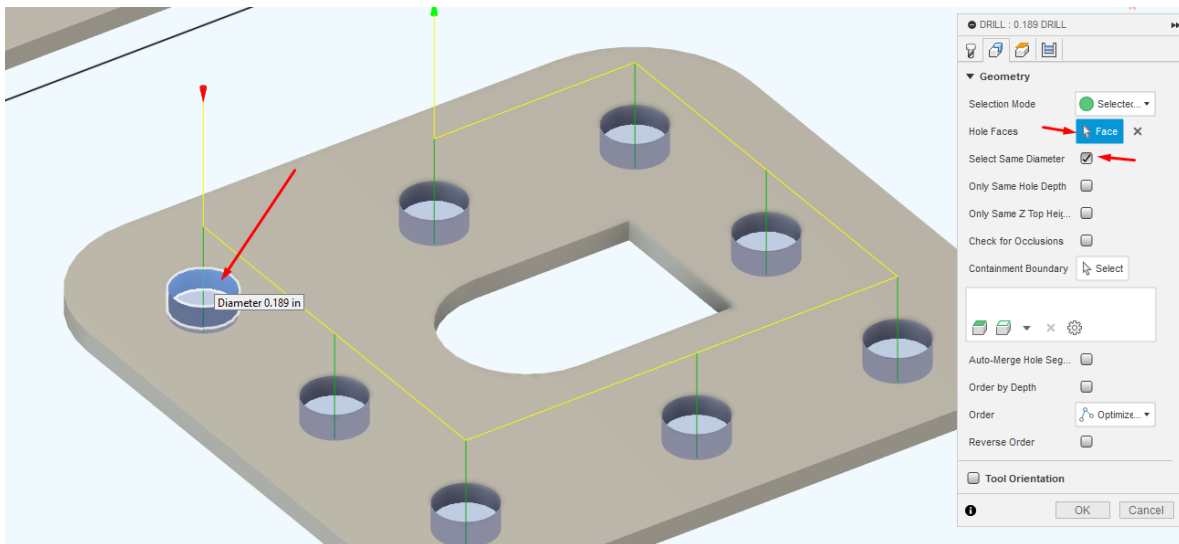
Document 231010b

Revised Oct. 10, 2023

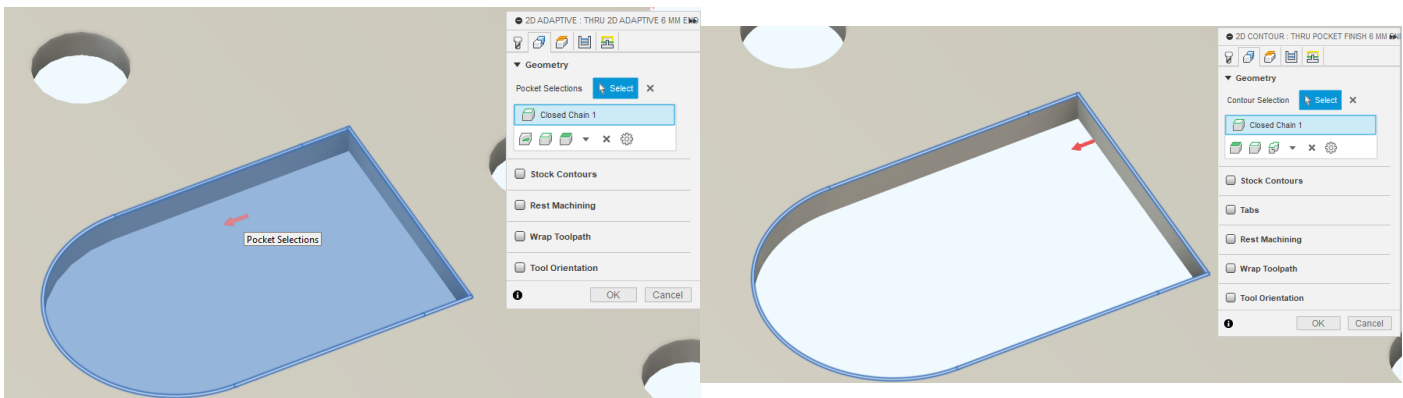


Right click the first operation used, click “edit”, then go to the geometry tab.

For **Drill** operations, be sure “select same diameter” is checked so all the same sized holes automatically select, click the “X” next to the section box to clear any existing selections, then select the hole faces (not lines, faces). You may have to click the “Select” box next to “Hole faces” if you just checked the “Select Same Diameter” checkbox. Be sure the blue shaded volumes appear to drill through the part appropriately and click “OK”.



For **Thru 2D Adaptive** and **Thru Pocket Finish** operations, click the “X” next to the section box to clear any existing selections, then select the outline of the pocket(s) in the geometry tab. Existing geometry selections will need to be removed before adding the new ones being currently machined. The red arrow indicates which side of the line the tool will machine on. If it is on the incorrect side, click the red arrow to correct it. This happens rarely. Click “OK” once the selection(s) have been made.



It is a good idea upon completion of each step, to inspect the toolpath lines that show up to verify the end mill is on the correct side and the tool will not machine to an odd depth among other possible issues.



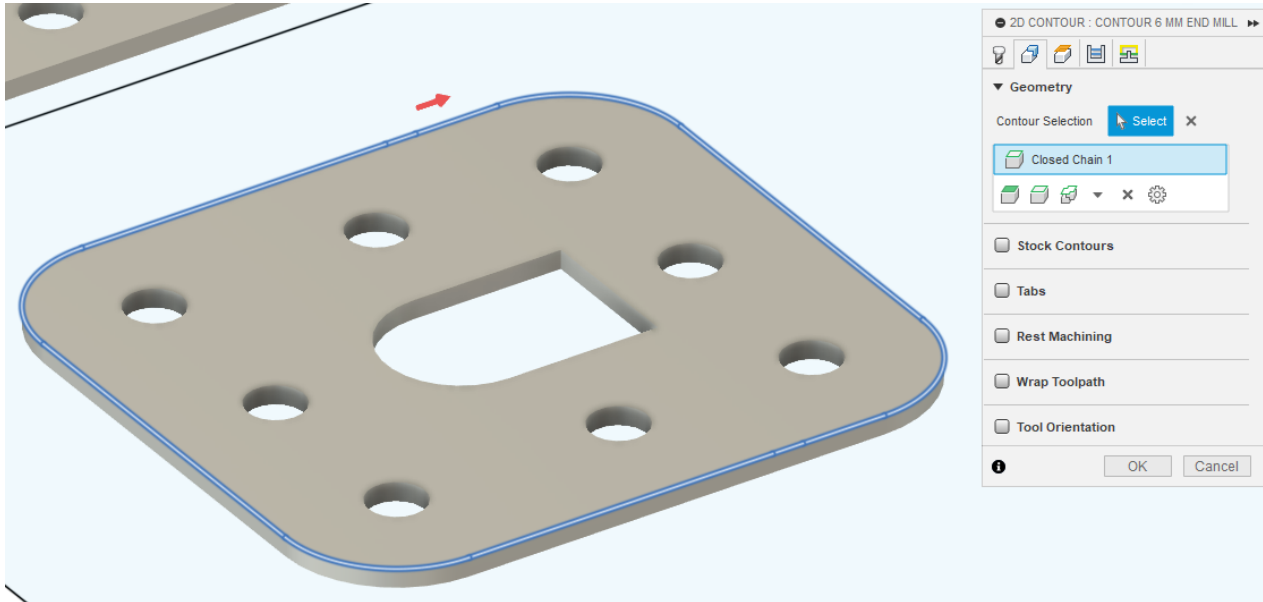
# Sheet Metal Machining Setup Guide

Document 231010b

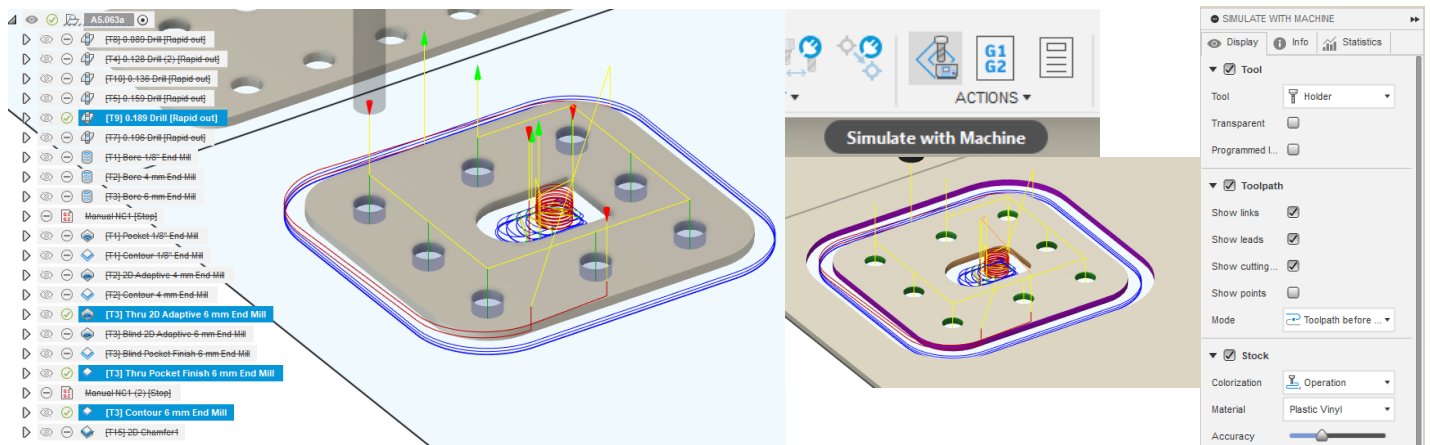
Revised Oct. 10, 2023



For **Contour** operations, click the “X” next to the section box to clear any existing selections, then select the outline of the part(s) in the geometry tab. Existing geometry selections will need to be removed before adding the new ones being currently machined. The red arrow indicates which side of the line the tool will machine on. If it is on the incorrect side, click the red arrow to correct it. This happens rarely. Click “OK” once the selection(s) have been made.



Hold “ctrl” and select all the used operations to inspect the toolpaths to ensure things look sensible—as a mentor if you are in doubt regarding any operation. Then “Simulate with machine” and play the simulation with the settings shown to ensure the part looks like it should.



# Sheet Metal Machining Setup Guide

Document 231010b

Revised Oct. 10, 2023



NOTE: THE FOLLOWING STEPS CAN ONLY BE DONE ON JOHN TAYLOR'S COMPUTER UNLESS YOUR POST LIBRARY HAS SPECIFICALLY BEEN CONFIGURED TO INCLUDE OUR POST PROCESSOR

Right click the sheet setup, then click "POST PROCESS" to generate G codes. Ensure settings are *identical* to what's shown below besides the "Name/number", "File name", and "Output folder" before posting. Overwrite the file if prompted. You will damage the machine if you don't take care during this step of the process. For example, "useG28" will make the spindle launch to a point that is likely not even on the machine between each operation, causing stalling, crashes, and other issues.

The screenshot shows the 'Settings | Operations' dialog box for 'NC Program: NCPProgram6'. It is divided into two main sections: 'Machine and post' and 'Post properties'.

**Machine and post**

- Use machine configuration:
- Post: UCCNC Mill / UCCNC Mill
- Use cascading post:
- Program**
- Name/number: A5.063a
- File name: A5.063a
- Comment: (empty)
- Output folder: s/JohnT/Documents/G Codes
- Post to Fusion Team:
- NC extension: .tap
- Unit: Inches
- Open NC file in editor:
- Create in browser:

**Post properties**

- General**
- dwelInSeconds:
- optionalStop:
- preloadTool:
- separateWordsWithSpace:
- sequenceNumberIncrement: 5
- sequenceNumberStart: 10
- showSequenceNumbers:
- useG28:
- useM6:
- useRadius:
- writeMachine:
- writeTools:
- Built-in**

Buttons: Post, Cancel

Congratulations, you have written a G code to machine sheet metal parts!