

Plate Machining Setup Guide

Document 231010c

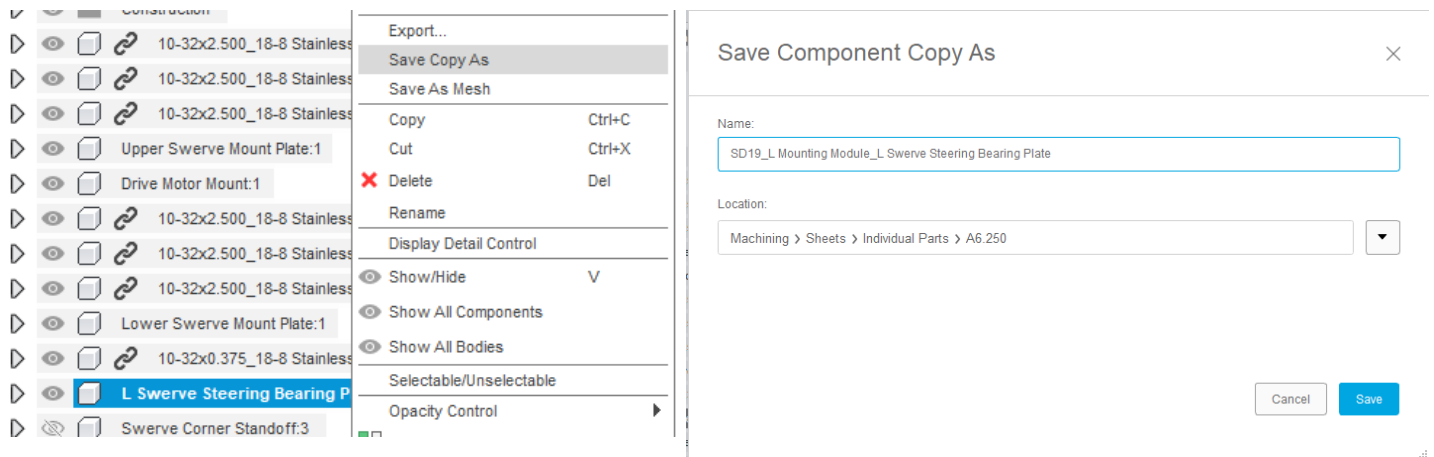
Revised Oct. 10, 2023



This guide will walk you through the process of setting up plate parts for machining using Autodesk Fusion 360.



Right click the component and click “Save copy as”, then save the copy into “Fusion 360/Machining/Sheets/Individual Parts/XX.YYY”. Replace “XX” with the desired material—A6 for 6061 aluminum, A5 for 5052 aluminum (only for bent material), or PC for polycarbonate), and replace “YYY” with the material thickness.



Go to the folder “Fusion 360/Machining/Sheets” and find the sheet with the right prefix (usually A6 or PC) that has the lowest letter designator that will fit your part(s). For example, if both A6.250a and A6.250b have room for your part, choose sheet A6.250a. In this case, it looks like A6.250b has room.

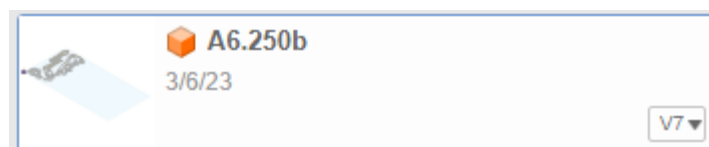


Plate Machining Setup Guide

Document 231010c

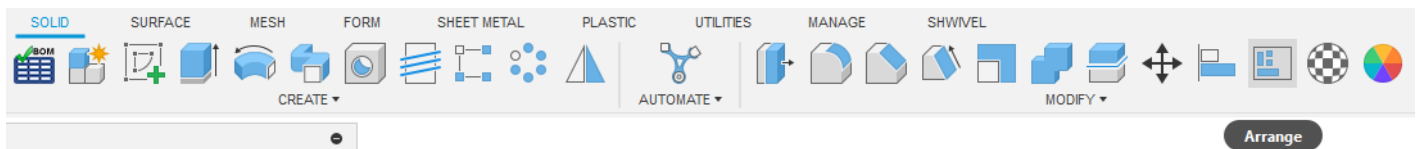
Revised Oct. 10, 2023



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.



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

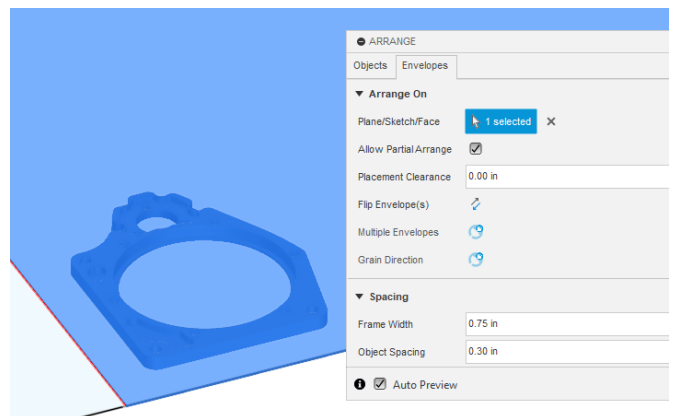
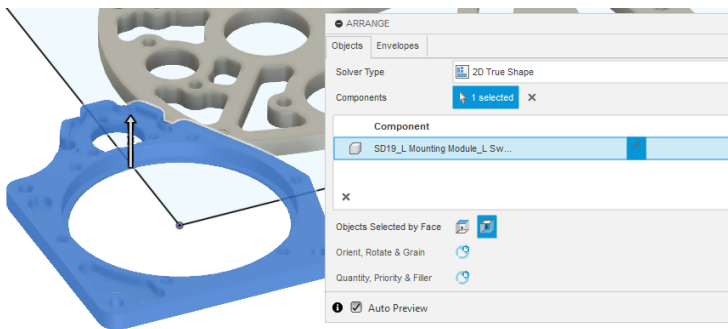


Plate Machining Setup Guide

Document 231010c

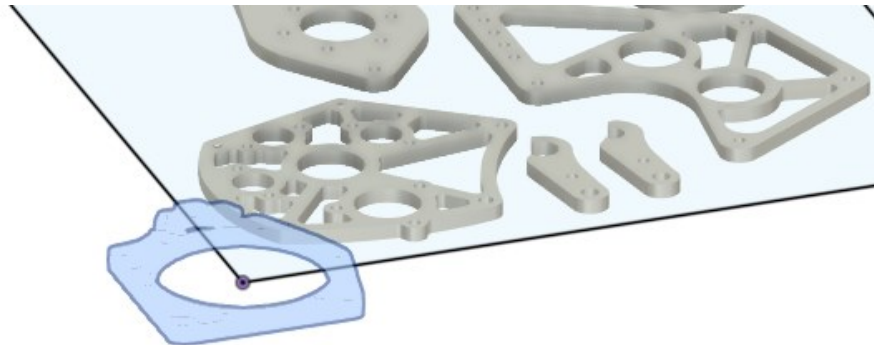
Revised Oct. 10, 2023



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. The blind (partial depth) pockets should be visible from the top so that they can be machined.



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



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

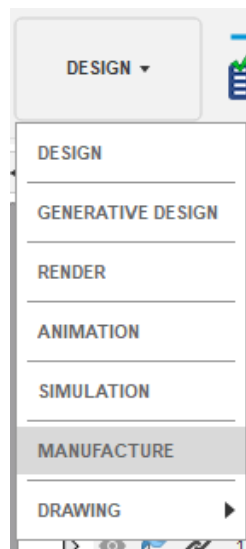
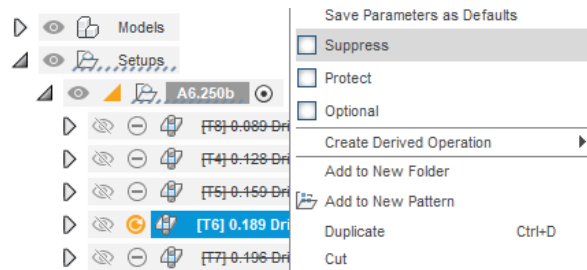


Plate Machining Setup Guide

Document 231010c

Revised Oct. 10, 2023



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 NCI [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.

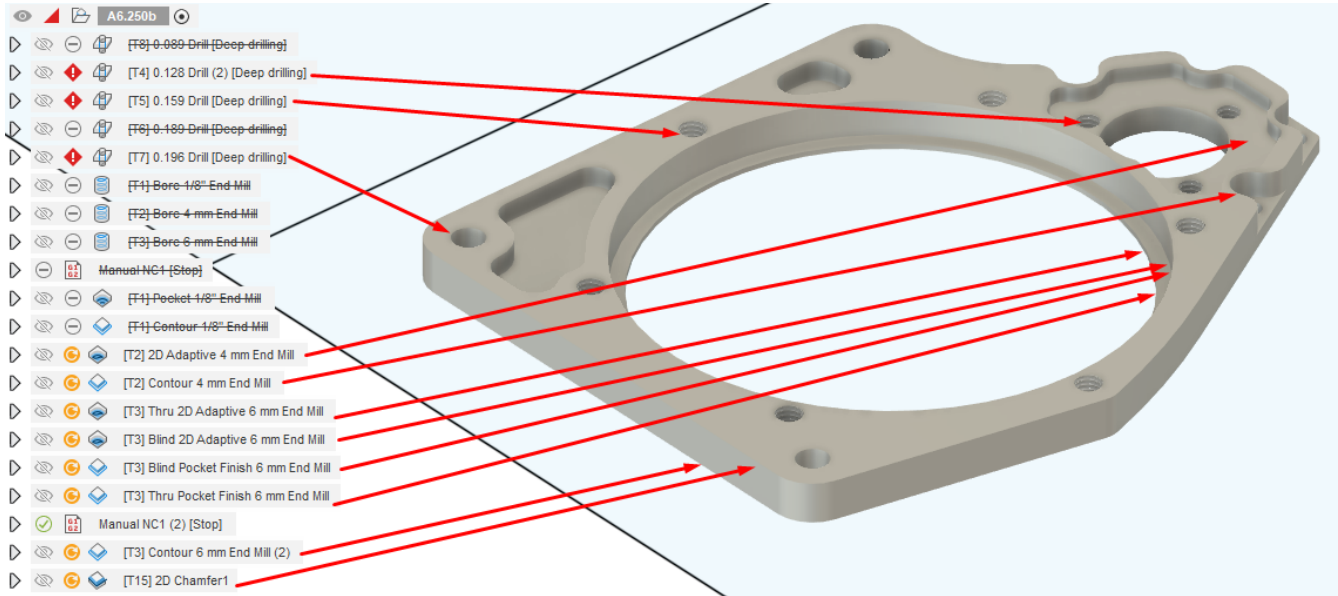
Plate Machining Setup Guide

Document 231010c

Revised Oct. 10, 2023



For the example part, “0.128 Drill” (close enough to the 0.136” drill used), “0.159 Drill”, “0.196 Drill”, “2D Adaptive 4 mm End Mill”, “Contour 4 mm End Mill”, “Thru 2D Adaptive 6 mm End Mill”, “Blind 2D Adaptive 6 mm End Mill”, “Blind Pocket Finish 6 mm End Mill”, “Thru Pocket Finish 6 mm End Mill”, “Contour 6 mm End Mill”, and “2D Chamfer” are un-suppressed. All other operations are suppressed. To suppress or un-suppress, 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.

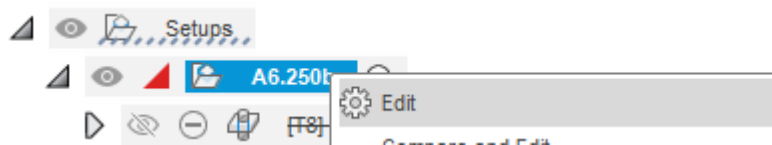


Plate Machining Setup Guide

Document 231010c

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.

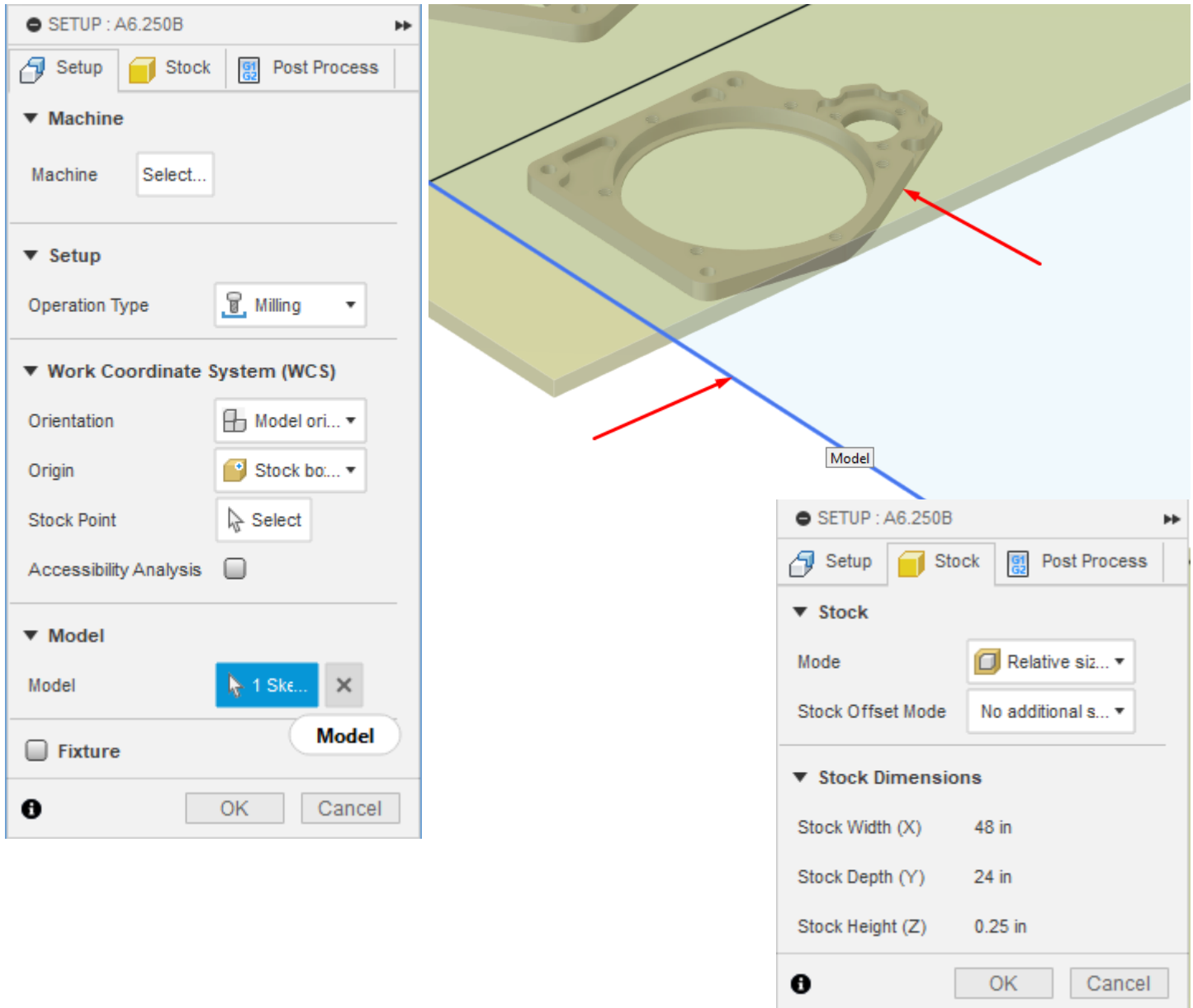


Plate Machining Setup Guide

Document 231010c

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 selection box to clear any existing selections, then select the correct 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”.

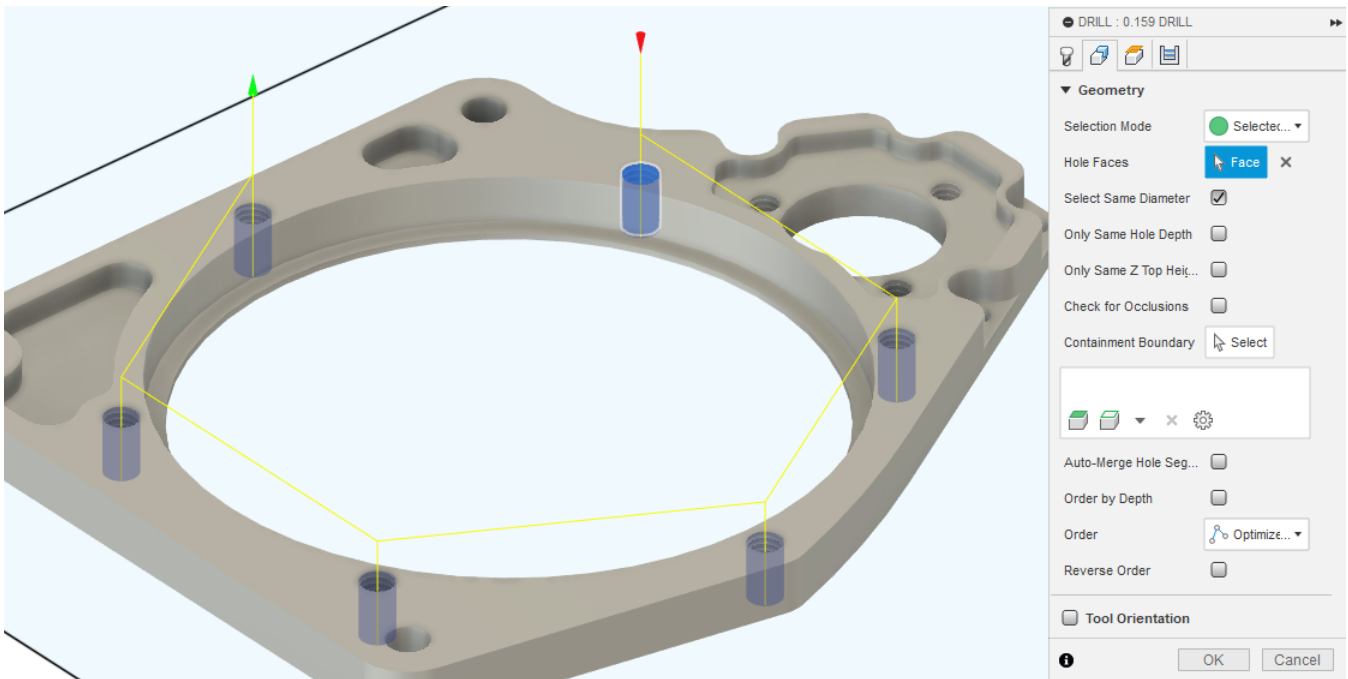


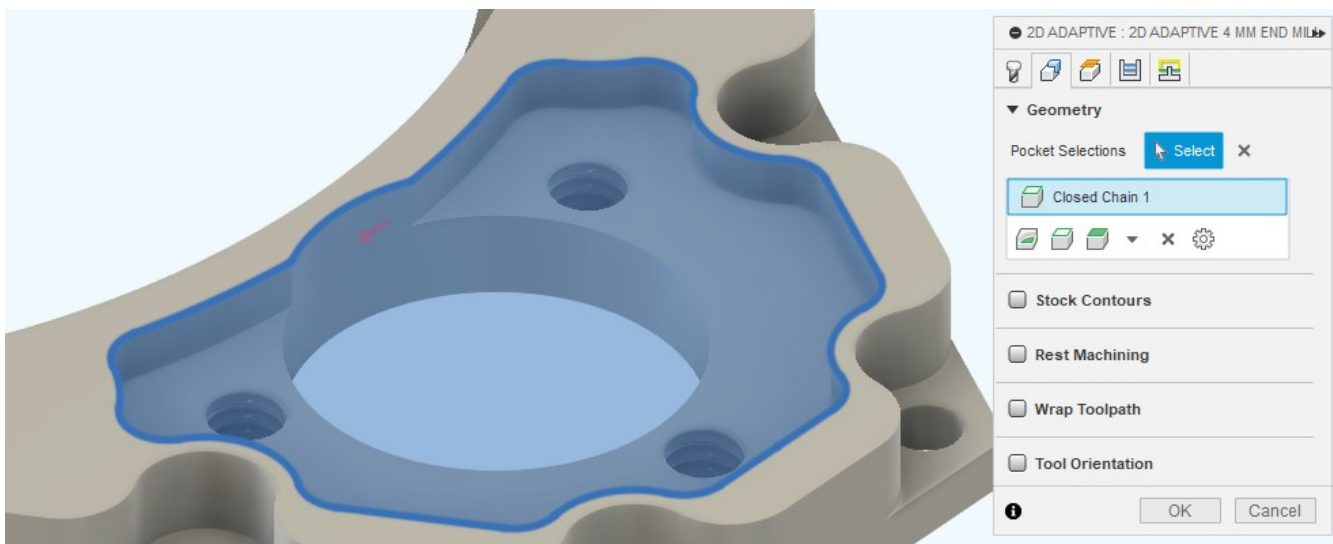
Plate Machining Setup Guide

Document 231010c

Revised Oct. 10, 2023



For **Pocket 1/8" End Mill** and **2D Adaptive 4 mm End Mill** operations, edit then click the "X" next to the selection 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. To machine the small counterbore pockets, I had to duplicate the 2D Adaptive 4mm End Mill operation and select the contours as the bottom height. This is because the larger contour doesn't have a complete profile at its base.



For blind (partial depth) operations, it is necessary to select the proper bottom height for this operation. Change the bottom height to be from "Selection", then select the face and ensure 0 offset as shown.

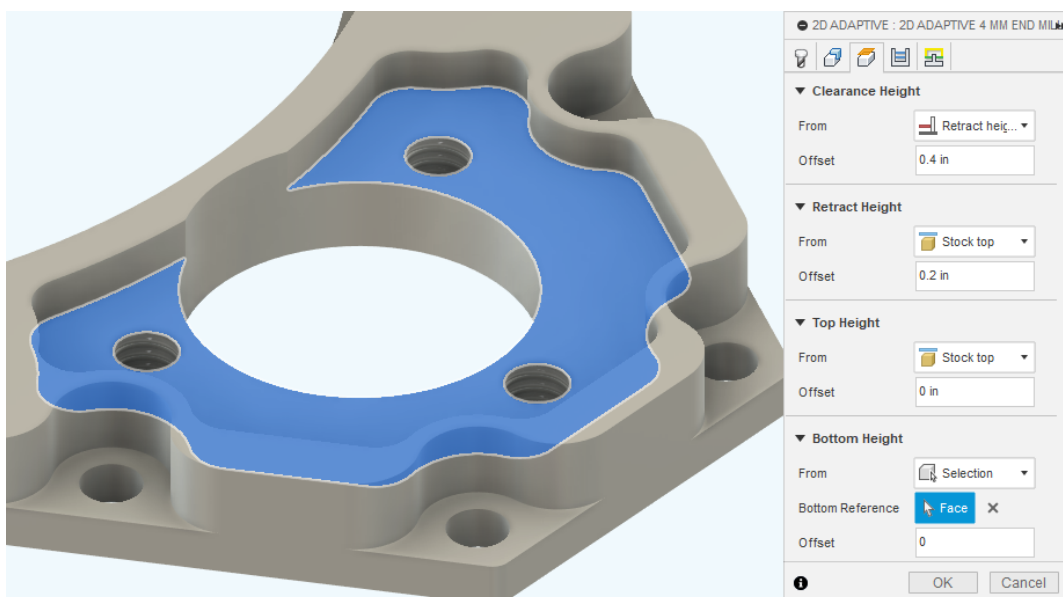


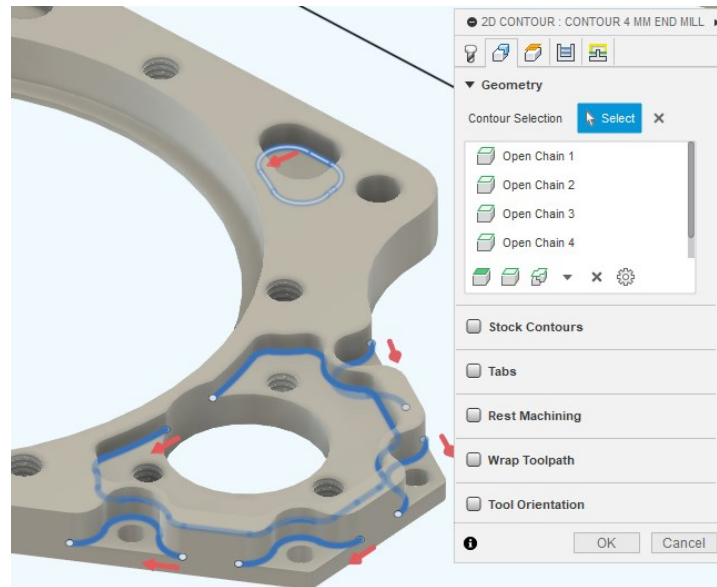
Plate Machining Setup Guide

Document 231010c

Revised Oct. 10, 2023



For **Contour 1/8" End Mill** and **Contour 4 mm End Mill** operations, click the "X" next to the selection box to clear any existing selections, then select the outline of the contour(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 the blue line to pick a partial contour if necessary.



For blind (partial depth) operations, it is necessary to select the proper bottom height for this operation. Change the bottom height to be from "Selected Contours" and ensure 0 offset as shown.

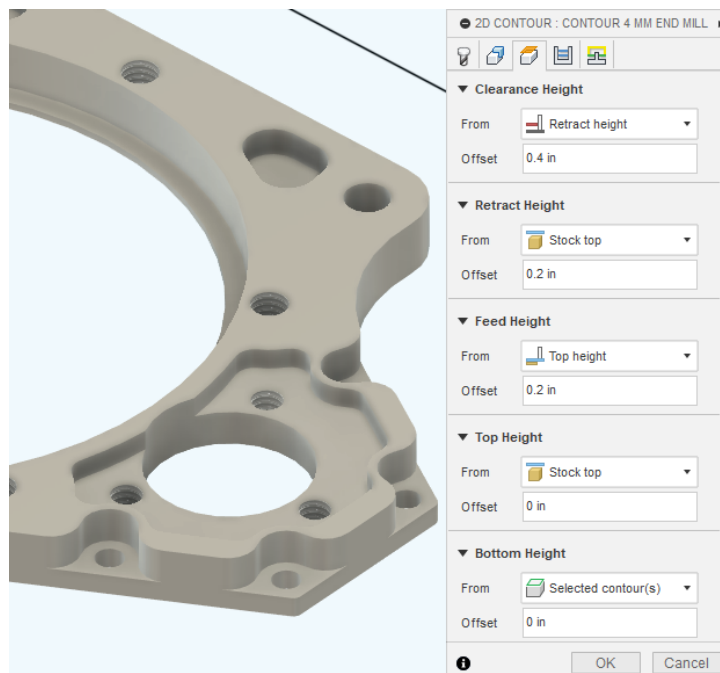


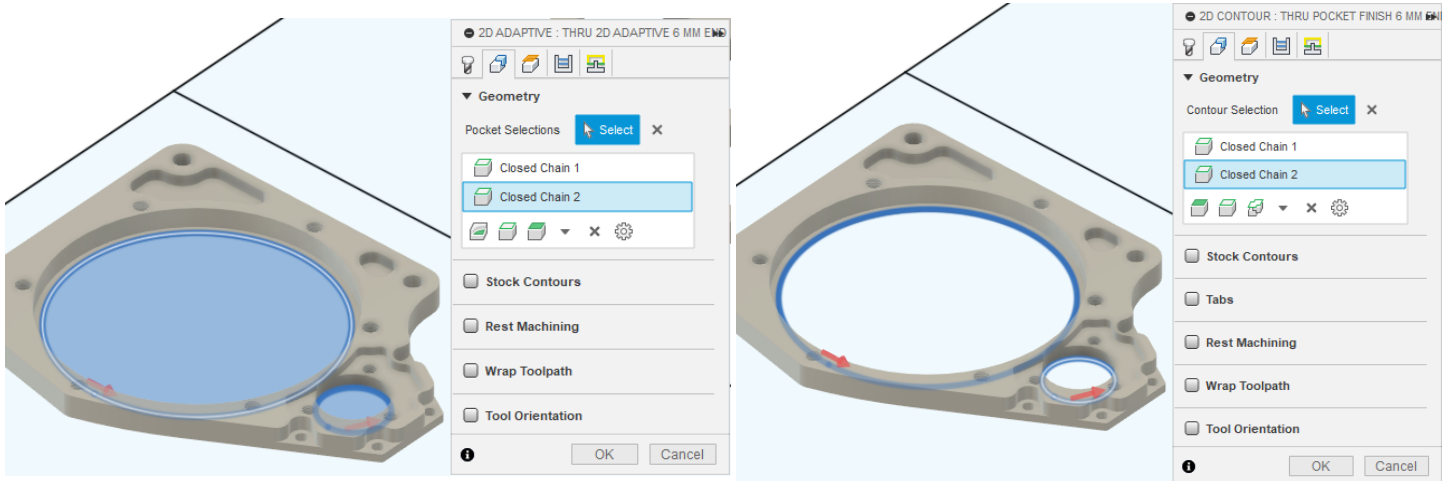
Plate Machining Setup Guide

Document 231010c

Revised Oct. 10, 2023



For **Thru 2D Adaptive** and **Thru Pocket Finish** operations, edit then click the “X” next to the selection 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.



For **Blind 2D Adaptive** and **Blind Pocket Finish** operations, edit then click the “X” next to the selection box to clear any existing selections, then select the faces for adaptive clearing and the outlines for finishing at the base 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.

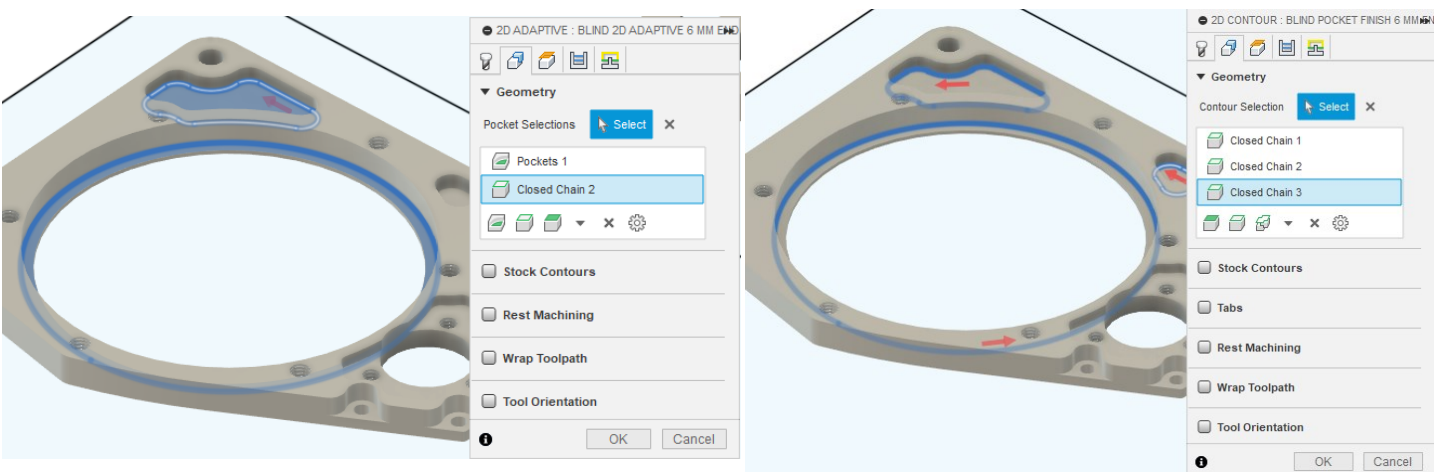


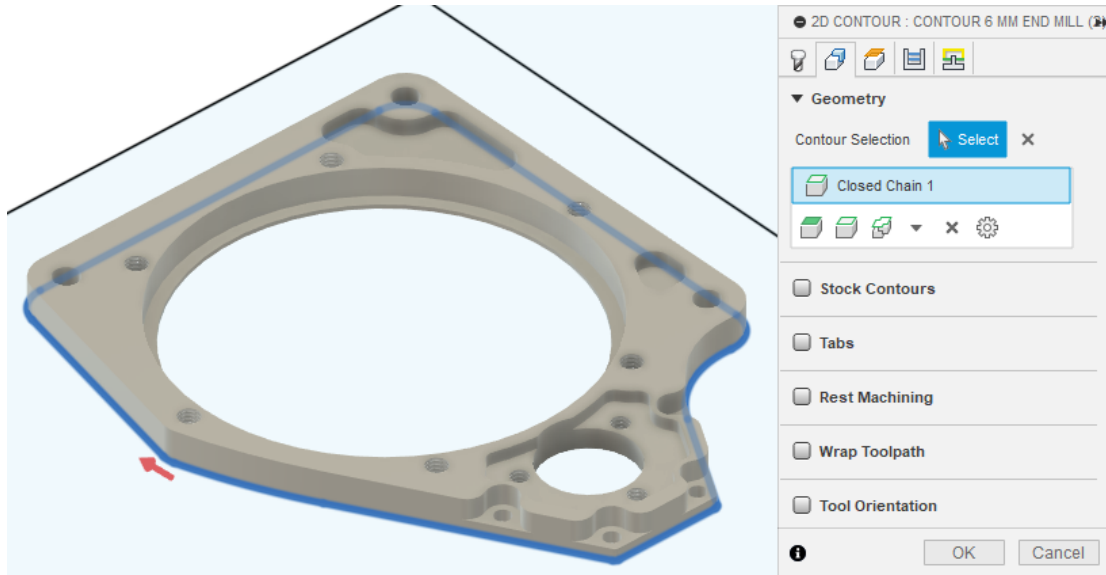
Plate Machining Setup Guide

Document 231010c

Revised Oct. 10, 2023



For **Contour 6 mm End Mill** operations, click the “X” next to the selection 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.



For **2D Chamfer** operations, click the “X” next to the selection box to clear any existing selections, then select any edges that can be chamfered to reduce the need to deburr the finished part. Blind pocket top edges in particular should be selected.

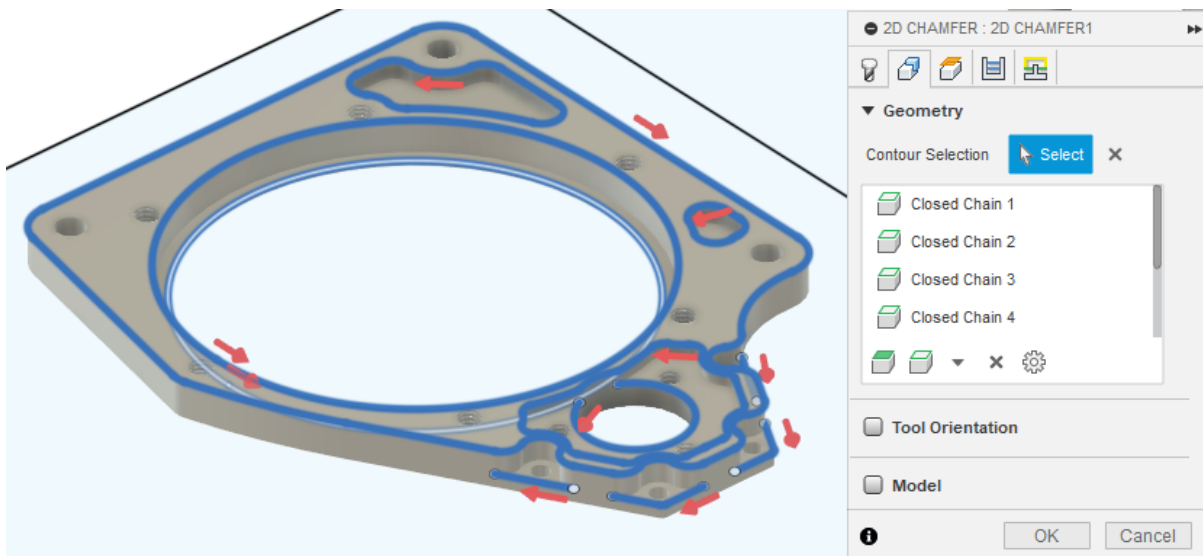


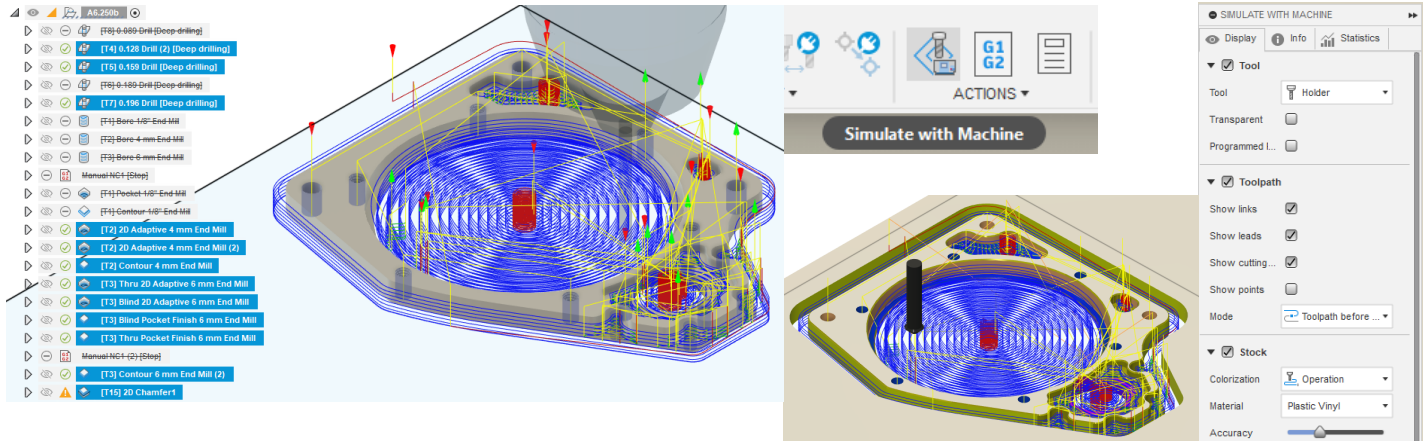
Plate Machining Setup Guide

Document 231010c

Revised Oct. 10, 2023



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.



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.

Machine and post	
Use machine configuration	<input type="checkbox"/>
Post	UCCNC Mill / UCCNC Mill
Use cascading post	<input type="checkbox"/>
Program	
Name/number	A6.250b
File name	A6.250b
Comment	
Output folder	s/JohnT/Documents/G Codes
Post to Fusion Team	<input type="checkbox"/>
NC extension	.tap
Unit	Inches
Open NC file in editor	<input type="checkbox"/>
Create in browser	<input checked="" type="checkbox"/>

Post properties	
General	
dwellInSeconds	<input checked="" type="checkbox"/>
optionalStop	<input checked="" type="checkbox"/>
preloadTool	<input type="checkbox"/>
separateWordsWithSpace	<input checked="" type="checkbox"/>
sequenceNumberIncrement	5
sequenceNumberStart	10
showSequenceNumbers	<input type="checkbox"/>
useG28	<input type="checkbox"/>
useM6	<input checked="" type="checkbox"/>
useRadius	<input type="checkbox"/>
writeMachine	<input checked="" type="checkbox"/>
writeTools	<input checked="" type="checkbox"/>
Built-in	