Penta 5-Axis Lab Guide

Lucas Rivers

May 1, 2025

Table of Contents

Required Materials 1
Step 1: Create a New Setup 1
Step 2: Create Sketch Geometry 1
2.1: Create Sketch 2
2.2: Project Geometry 3
2.3: Sketch From the Center of the Projected Circle 4
2.4: Project the Center Bore 5
2.5: Create Circular Pattern 6
Step 3: Mill Counterbores 6
3.1: Tool Tab 6
3.2: Multi-Axis Tab 7
3.3: Geometry Tab 8
3.4: Heights Tab 9
3.5: Cycle Tab 10
3.6: Linking Tab 10
3.7: Machine Remaining Counterbores 10
Step 4: Peck Drill the Angled Holes 11
4.1: Tool Tab 11
4.2: Multi-Axis Tab 12
4.3: Geometry Tab 12
4.4: Heights Tab 12
4.5: Cycle Tab 13
4.6: Linking Tab 13
4.7: Peck Drill the Other Two Holes 13
Step 5: Deburr Part 14
5.1: Tool Tab 14
5.2: Multi-Axis Tab 15

5.3: Geometry Tab 16
5.4: Heights Tab 16
5.5: Passes Tab 20
5.6: Linking Tab 21
Step 6: Mill The Undercut 22
6.1: Tool Tab 22
6.2: Multi-Axis Tab 23
6.3: Geometry Tab 24
6.4: Radii Tab 24
6.5: Passes Tab 25
6.6: Linking Tab 25
Step 6: Post Process Operation and Modify G-Code 26
6.1: Post Process the 5-Axis Operation 26
6.2: Search G-Code 27
6.3: Edit G-Code 29

Required Materials

All the files created during the completion of the “Penta Lab Setup Guide”,the “Penta 3-Axis Lab Guide”, and the “Penta 4-Axis Lab Guide”

Step 1: Create a New Setup

Create a new setup and name it appropriately for this lab. Except for the stock tab, the process is exactly the same as in the Lab Setup Guide.

We want the in-process stock from the 4-axis setup to carry over into this 5-axis setup. To do this, change the stock mode to “From preceding setup” and select “Continue Rest Machining”.

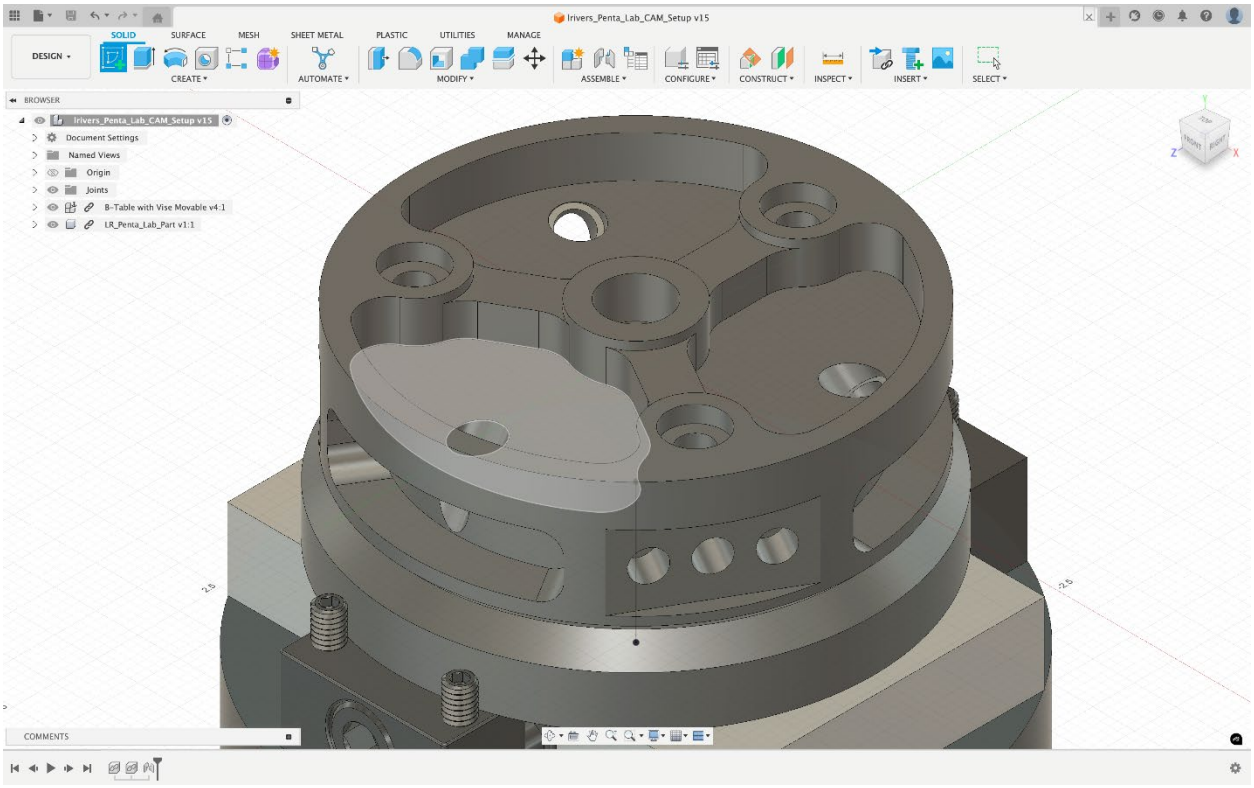
Finish creating the setup and click OK.

Step 2: Create Sketch Geometry

For this operation, we will use sketch geometry to define the Z and X axes.

While it is possible to pattern an operation that would machine all three counterbores on this specific part, it is important to know other methods. There are instances where parts do not have equal spacing for features or have no geometry that is suitable to select in the multi-axis tab.

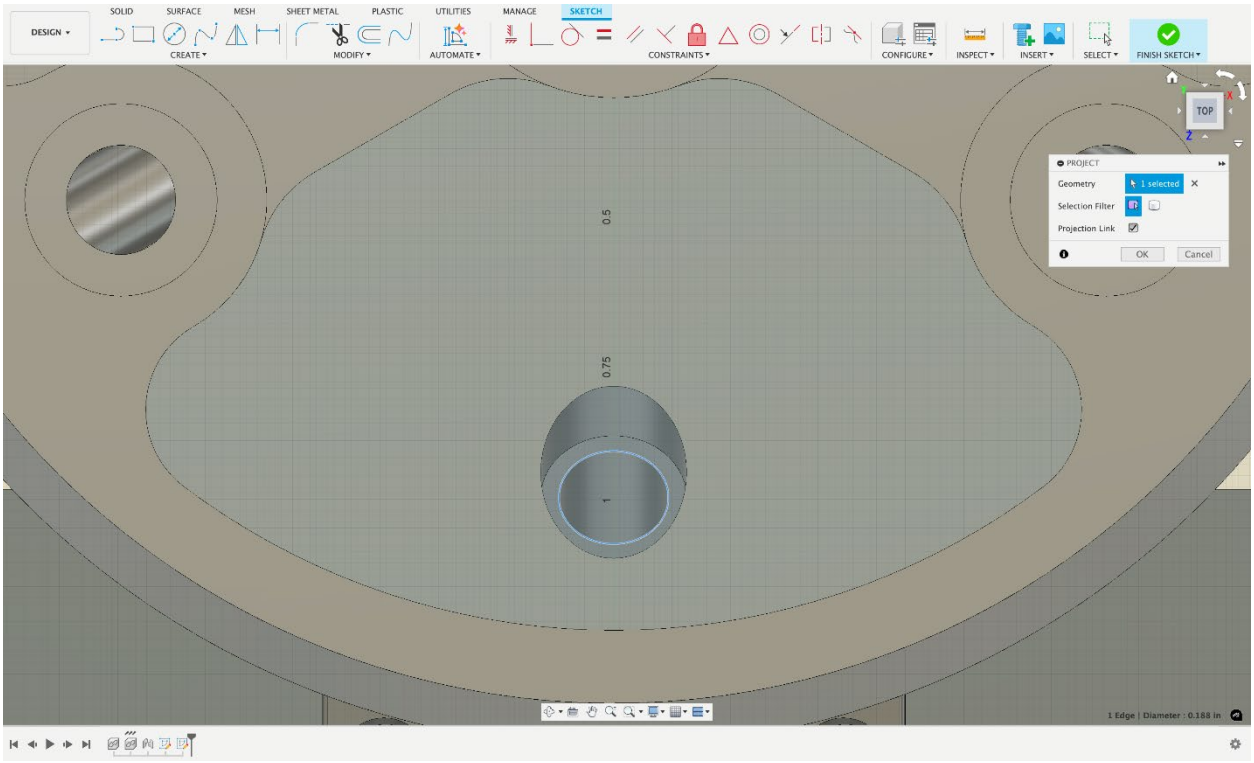
2.1: Create Sketch



Switch from the Manufacture Workspace" to the "Design Workspace" and create a sketch.

Select the floor of one of the pockets as the sketch plane.

2.2: Project Geometry

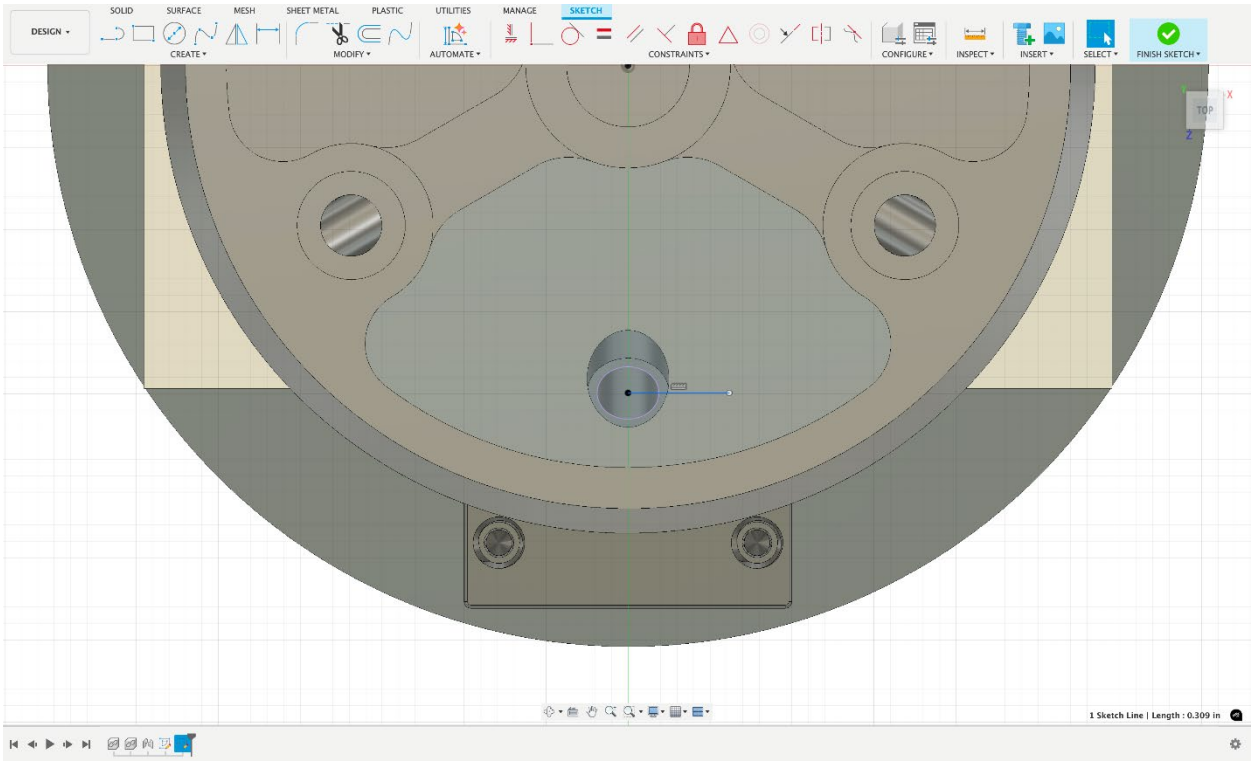


Find “Project/Include” under the “Create dropdown” and click on “Project” (keyboard shortcut “P”).

Click on either the edge of the counterbore or the edge of the drilled hole as shown in the image above and then click OK.

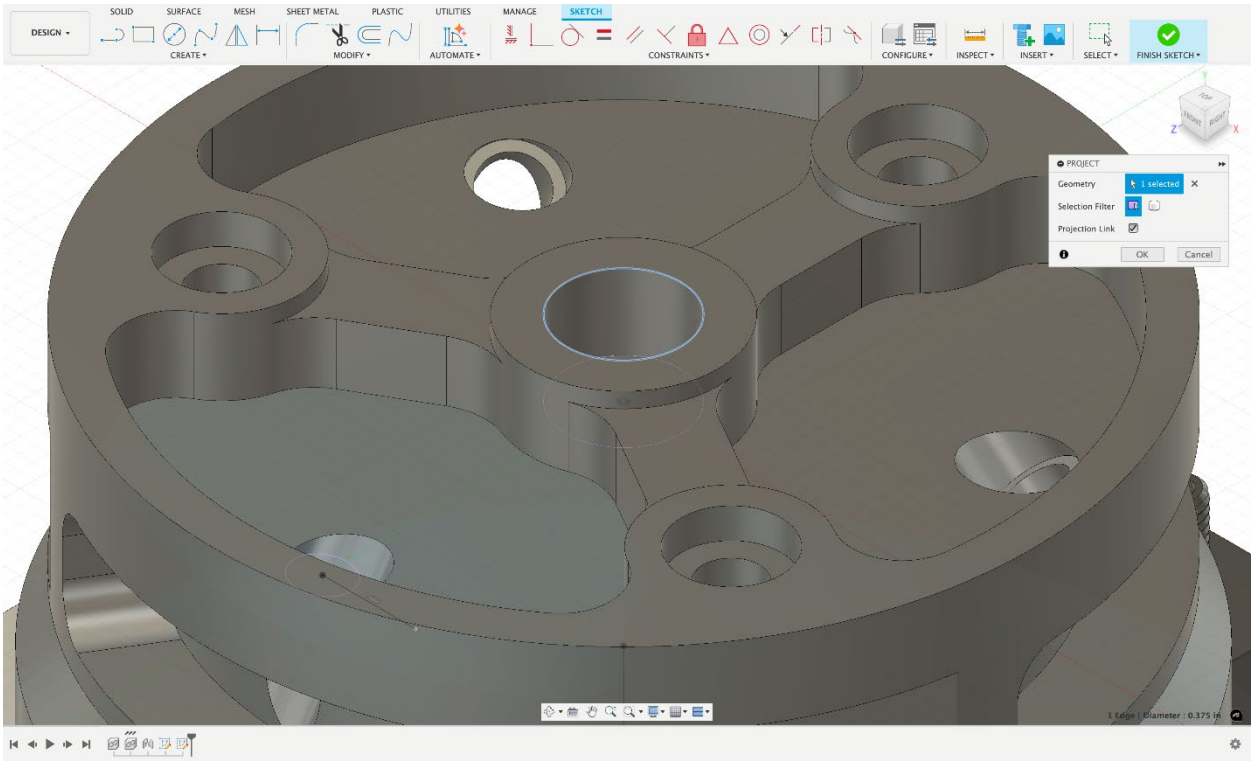
You will now see a purple circle with a center mark on the sketch plane.

2.3: Sketch From the Center of the Projected Circle



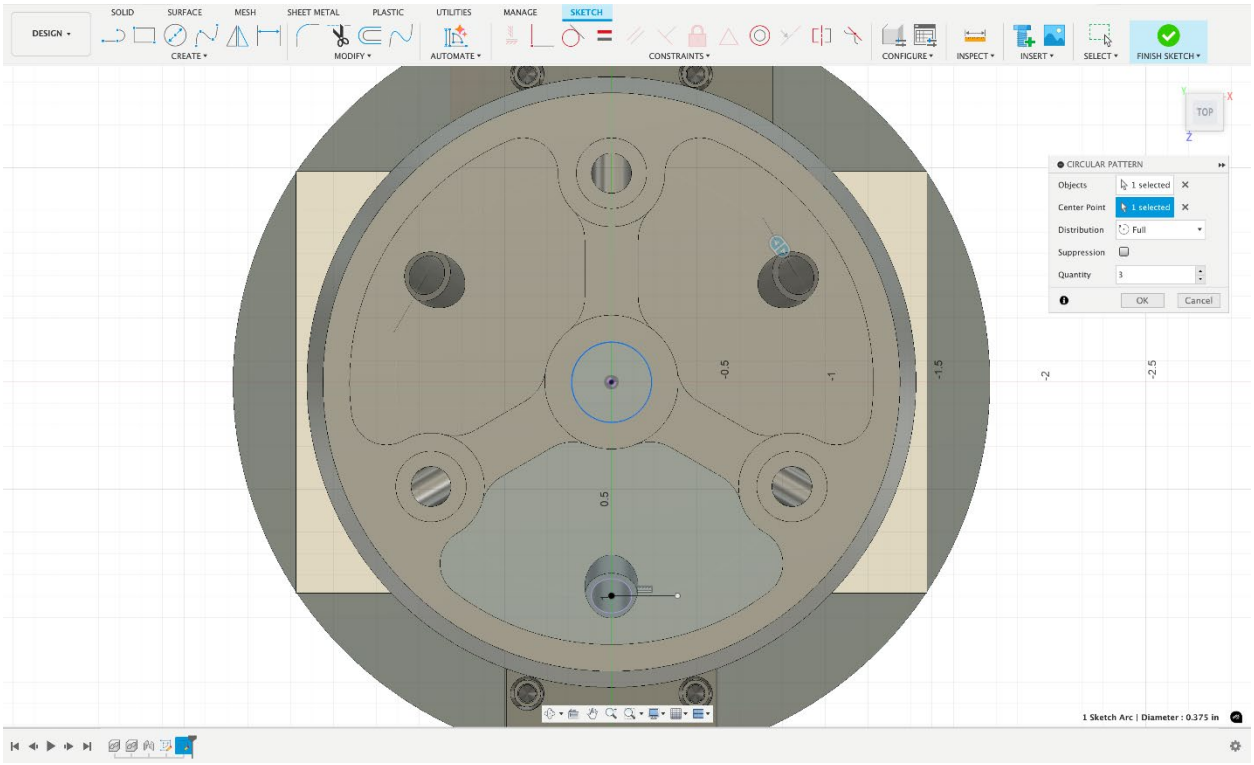
Draw a line (keyboard shortcut “L”) from the center of the projected circle as shown above. The line can go to the left or right, but it must be constrained horizontally.

2.4: Project the Center Bore



Use the project command to project the edge of the center bore of the part onto the sketch.

2.5: Create Circular Pattern



Select “Circular Pattern” in the create dropdown.

Select the horizontal sketch line as the object and the projected center bore sketch as the center point.

Make sure that the distribution is set to “Full”, and to and change the quantity to three.

Step 3: Mill Counterbores

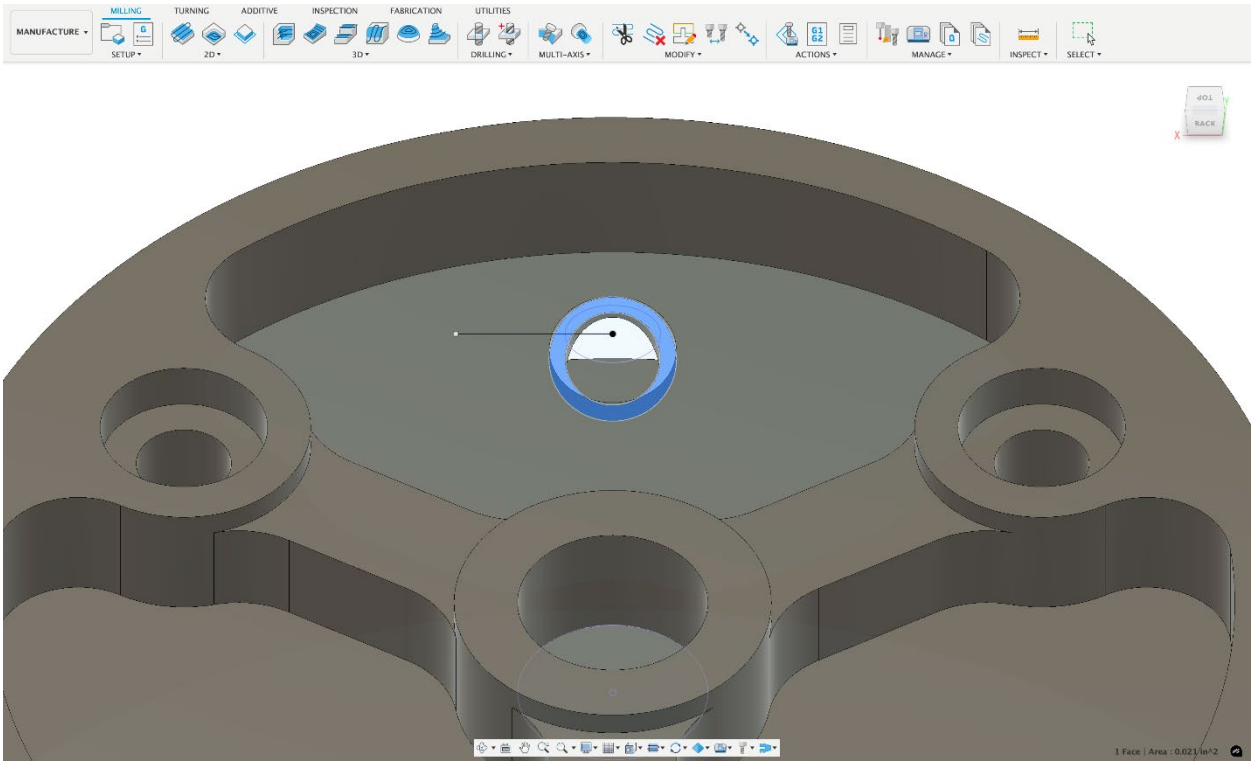
Go back to the “Manufacture Workspace” and create a drilling operation.

3.1: Tool Tab

- Tool #2
- Coolant disabled
- Default preset feeds and speeds
- Shaft clearance: 0.005 inches
- Holder clearance: 0.05 inches

The settings in the tool tab are the same as usual and should be used for every operation in this document unless stated otherwise.

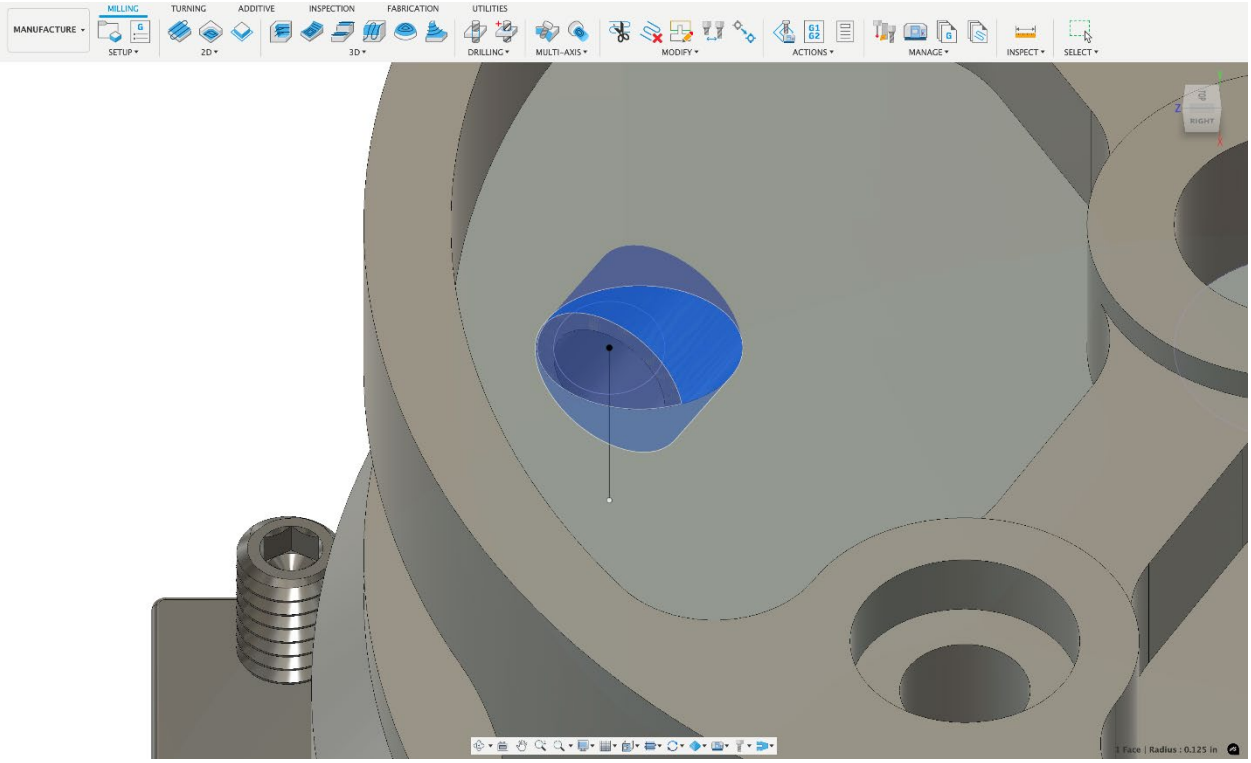
3.2: Multi-Axis Tab



Carefully select the floor of one of the counterbores of the angled holes in the bottom of a top pocket. The WCS should have the Z-axis pointing in the opposite direction of the counterbore face and at the same angle as the face.

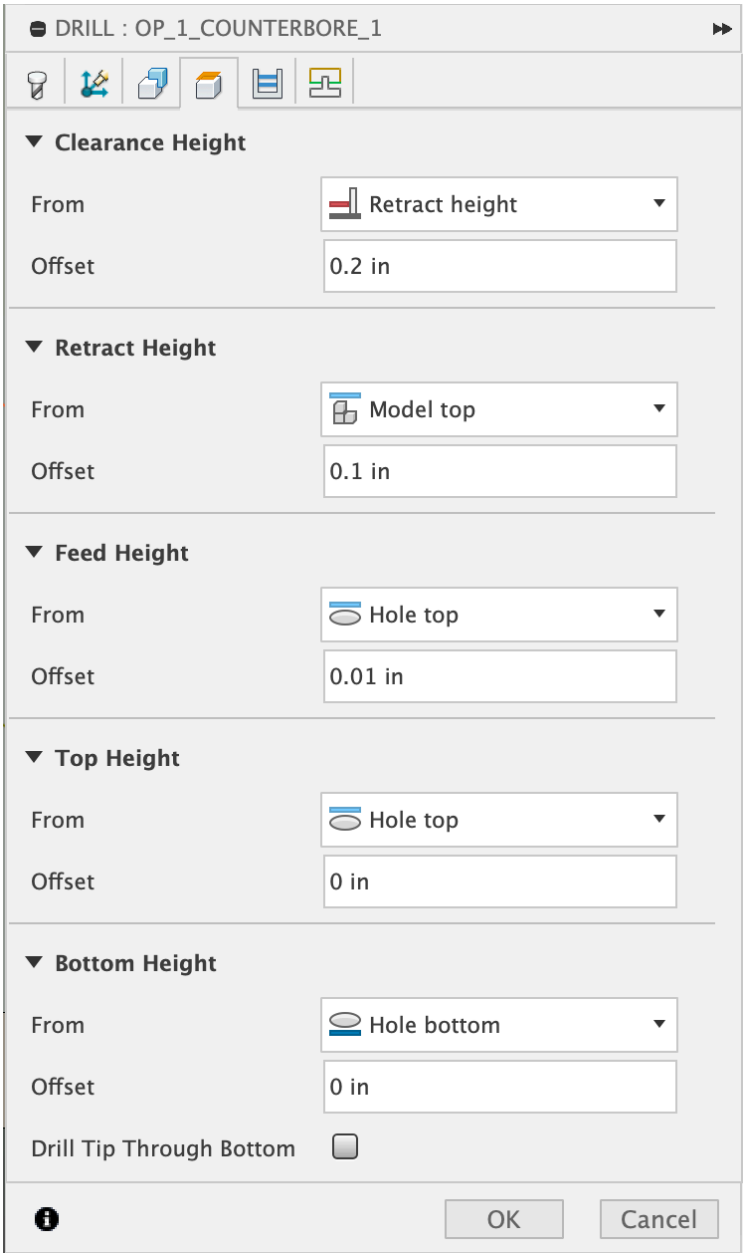
Next, select the line from the sketch we made for the X-axis. The WCS should have the Y-axis pointing at an angle in the general direction of the outer upper edge of the part.

3.3: Geometry Tab



Select the wall of the counterbore that you aligned the WCS with for the “Hole Face”.

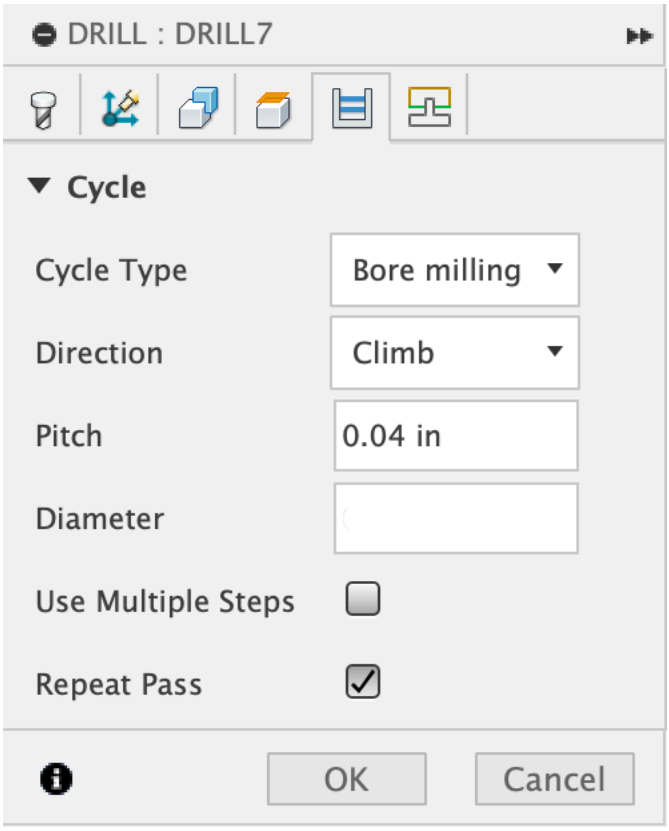
3.4: Heights Tab



Use the heights shown above.

Note: These values will be used for all drilling operations in this document.

3.5: Cycle Tab



Except for the diameter, use all the settings shown above.

The correct diameter of the hole can easily be found with the measure tool.

3.6: Linking Tab

There are no changes required in the linking tab for this operation.

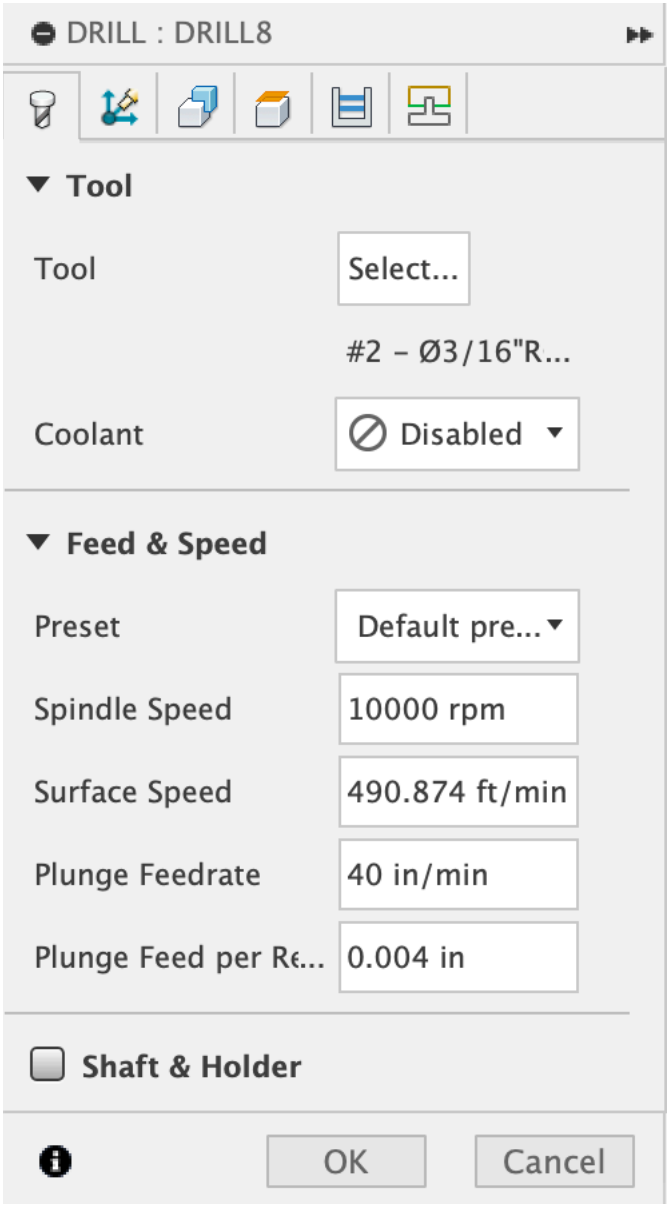
3.7: Machine Remaining Counterbores

Once finished with the counterbore operation, duplicate the operation twice and edit the WCS and geometry so that the other two counter bores are machined.

Step 4: Peck Drill the Angled Holes

Create another drilling operation. These holes will be made in a very similar way to the counterbores.

4.1: Tool Tab



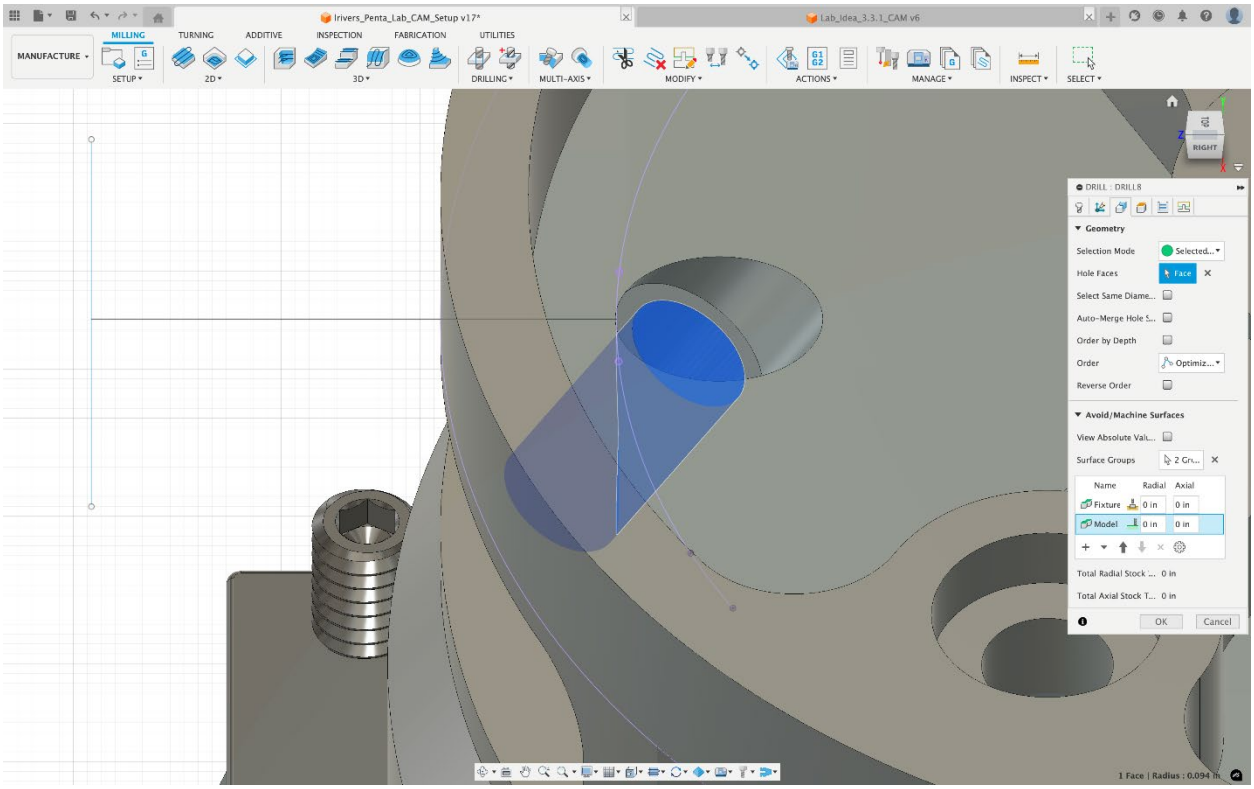
The machine barely has the reach to drill the angled holes in the part. Because of this, the “Shaft & Holder” checkbox must be unchecked for this operation.

4.2: Multi-Axis Tab

Make the same multi-axis selections that you did for the counterbore operation in 3.2: Multi-Axis Tab.

The counterbore floor is the Z-axis and the sketch line is the X-axis.

4.3: Geometry Tab

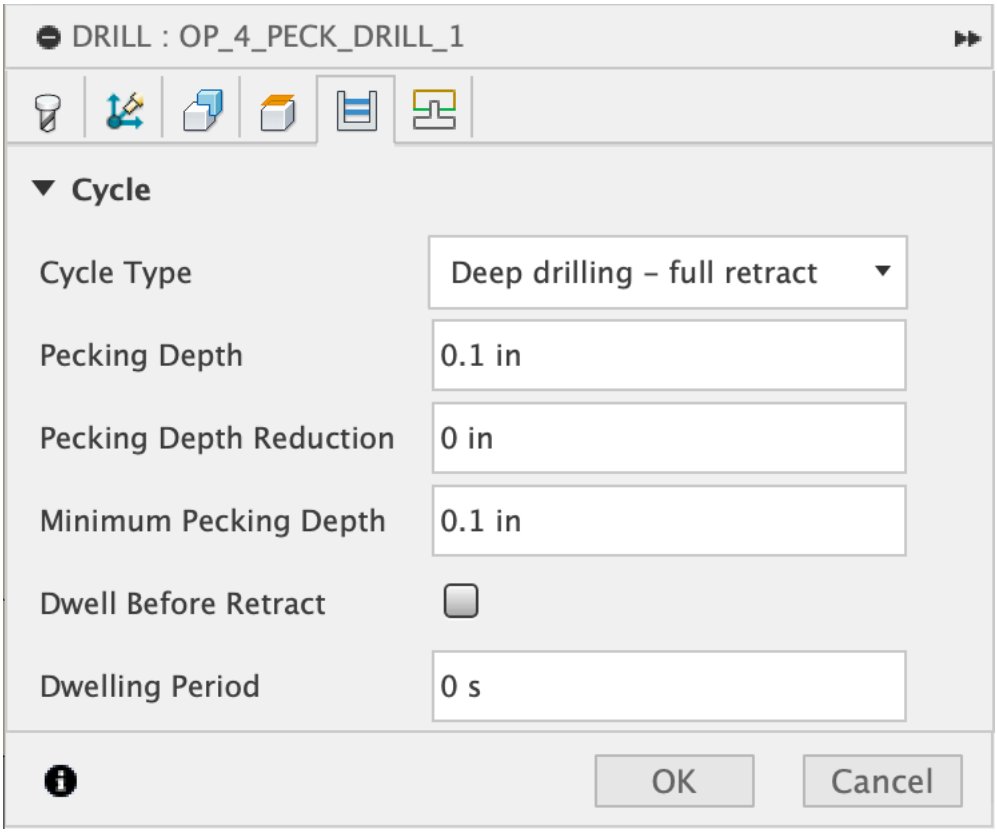


Select the wall of the angled hole as the “Hole Face”.

4.4: Heights Tab

Use the settings and selections from 3.4: Heights Tab.

4.5: Cycle Tab



Use the settings shown above for the cycle tab.

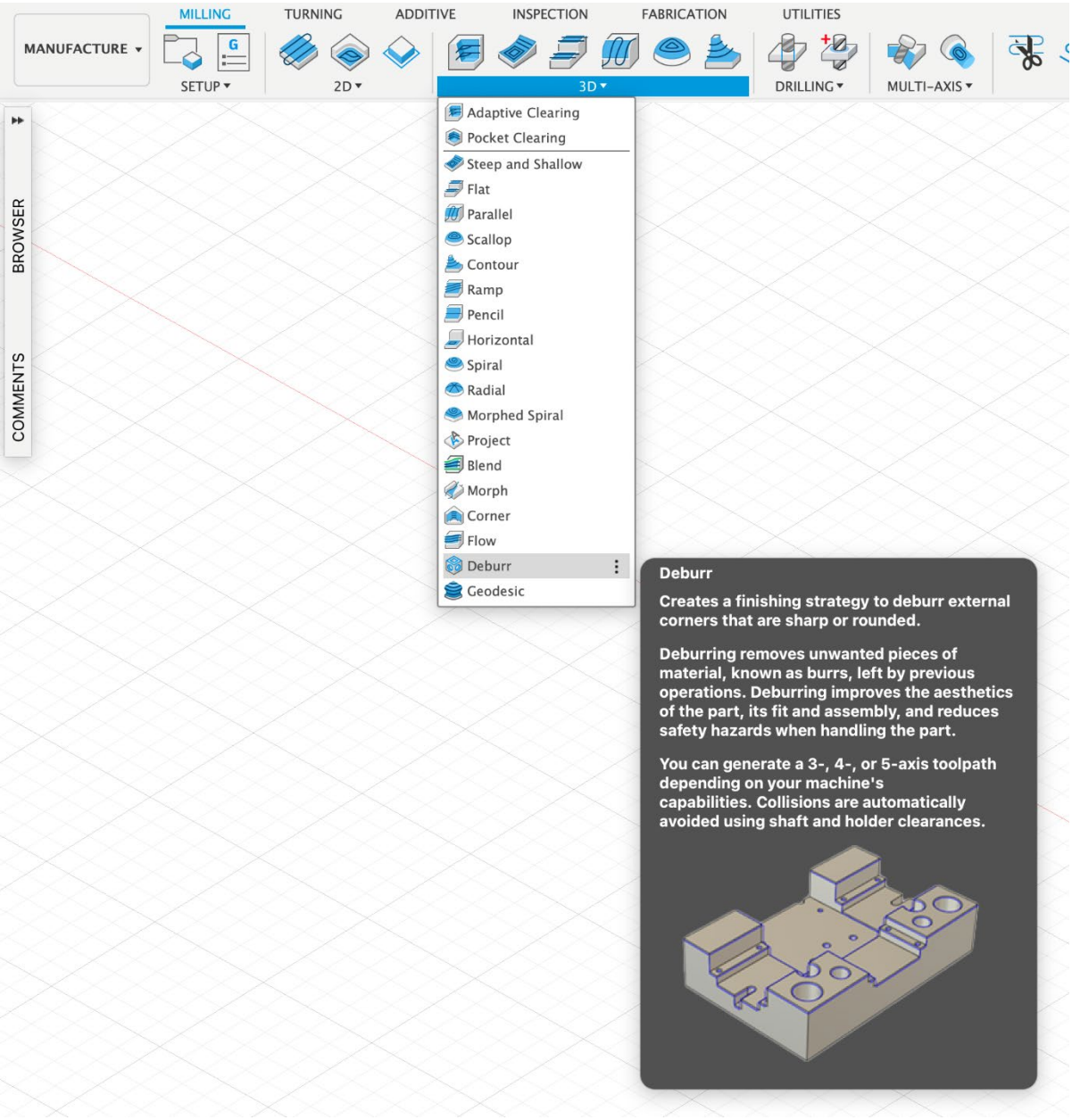
4.6: Linking Tab

There is no need to adjust the linking tab.

4.7: Peck Drill the Other Two Holes

Once finished with the peck drill operation, duplicate the operation twice and edit the WCS and geometry so that the other two angled holes are drilled.

Step 5: Deburr Part

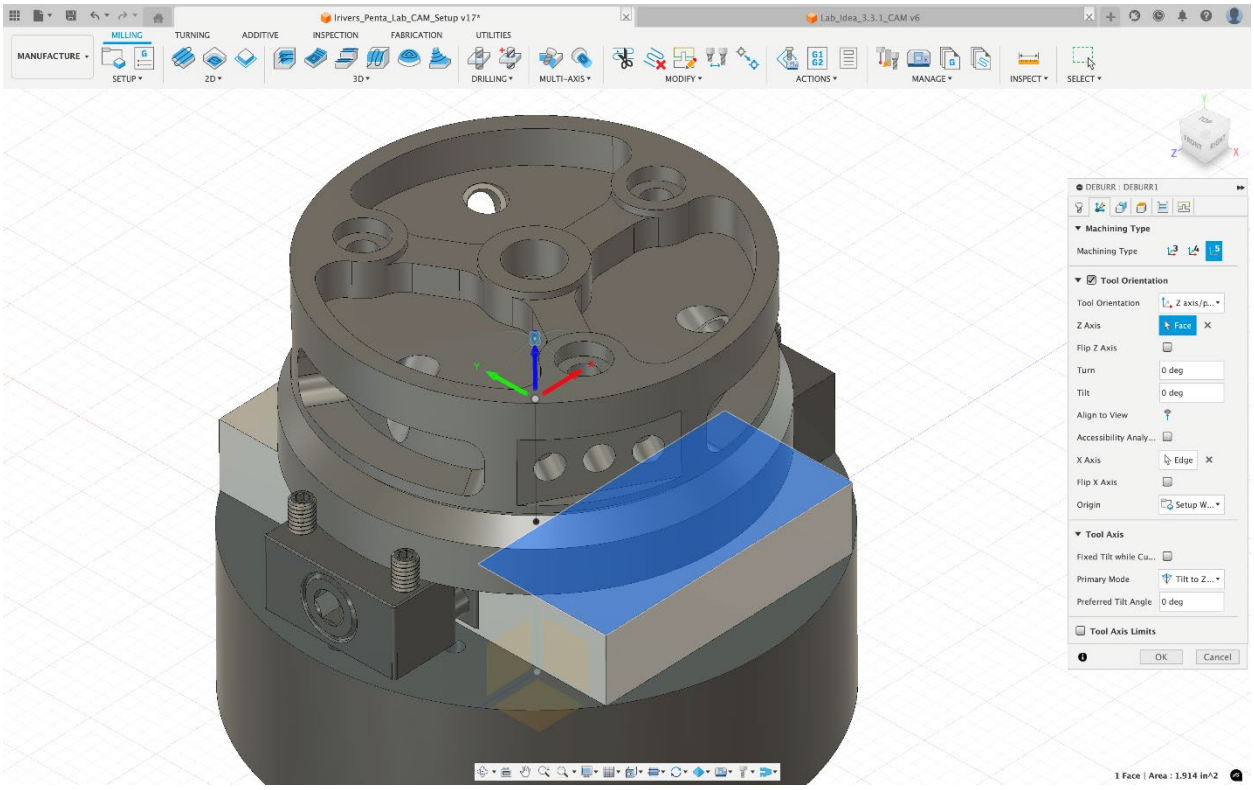


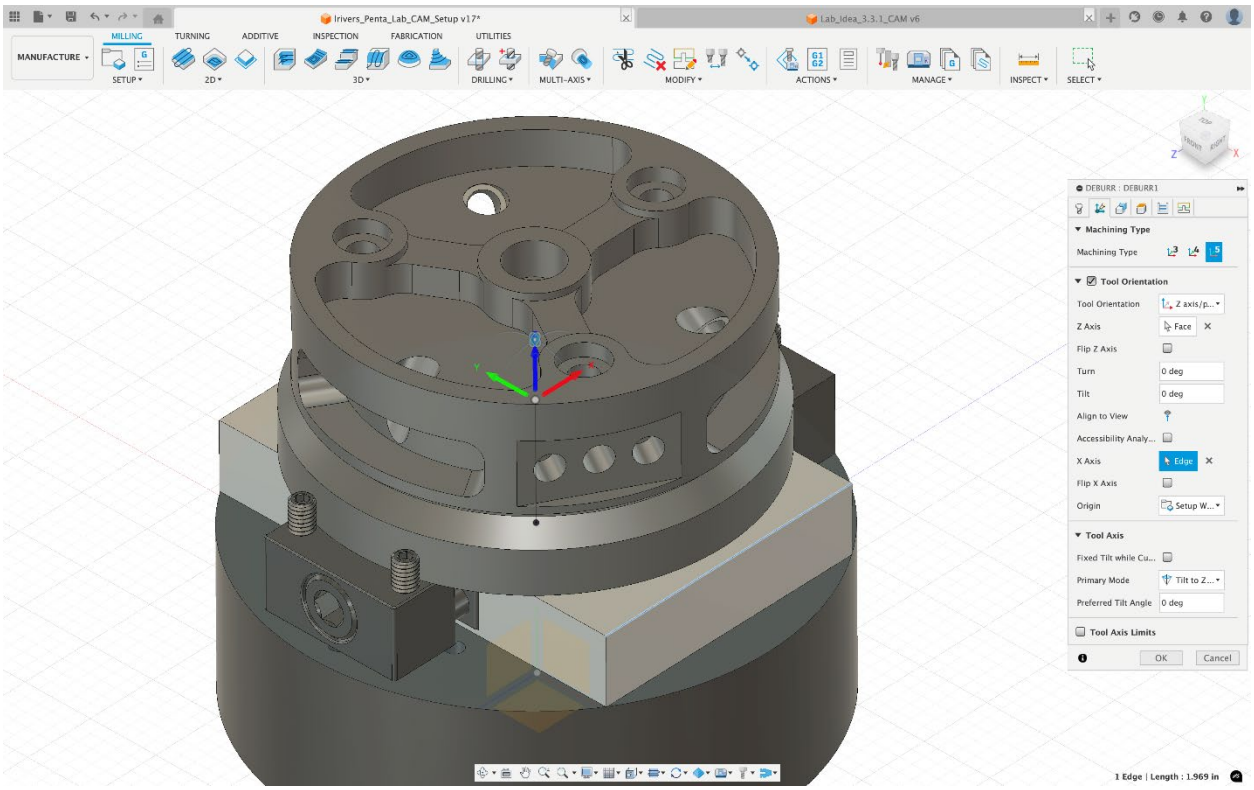
Under the 3D dropdown, create a deburr operation.

5.1: Tool Tab

Use the settings and selections from 3.1: Tool Tab.

5.2: Multi-Axis Tab



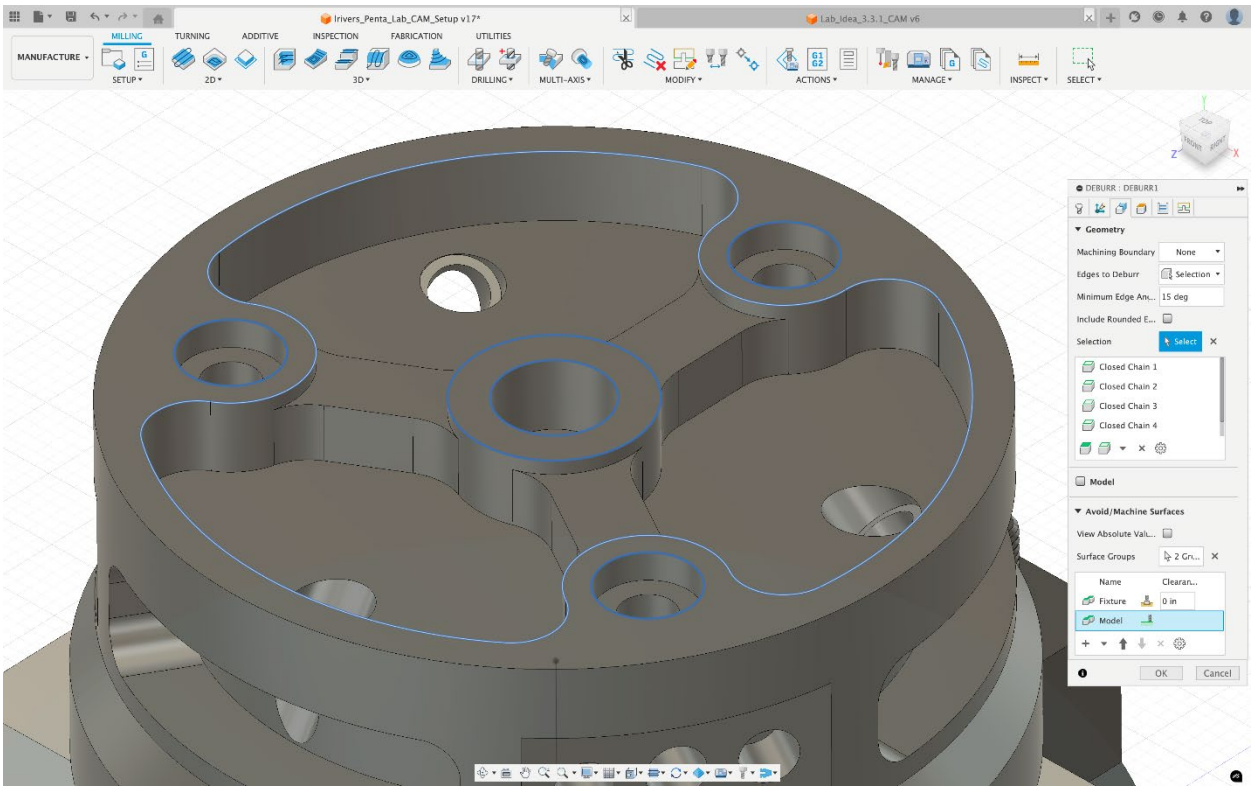


Change the machining type to 5-axis and make the same selections that you did in the 3-axis lab for the tool orientation.

The Z-axis should point up and the X-axis should point towards the back of the machine.

Change the tool axis “Primary Mode” to “Tilt to Z axis”.

5.3: Geometry Tab



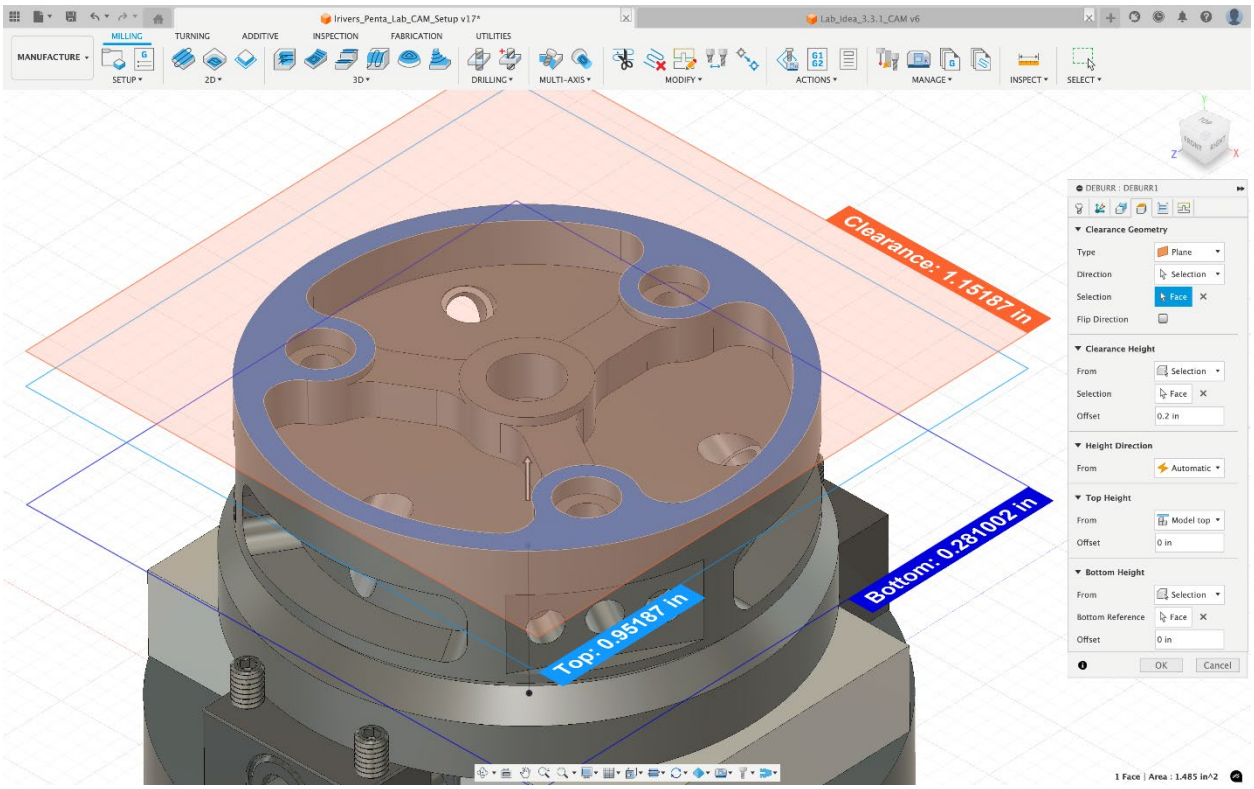
Change the “Edges to Deburr” to “Selection”.

Check that the “Minimum Edge Angle” is set to 15 degrees.

Select all the edges shown in the image above to deburr. There should be a total of six selections.

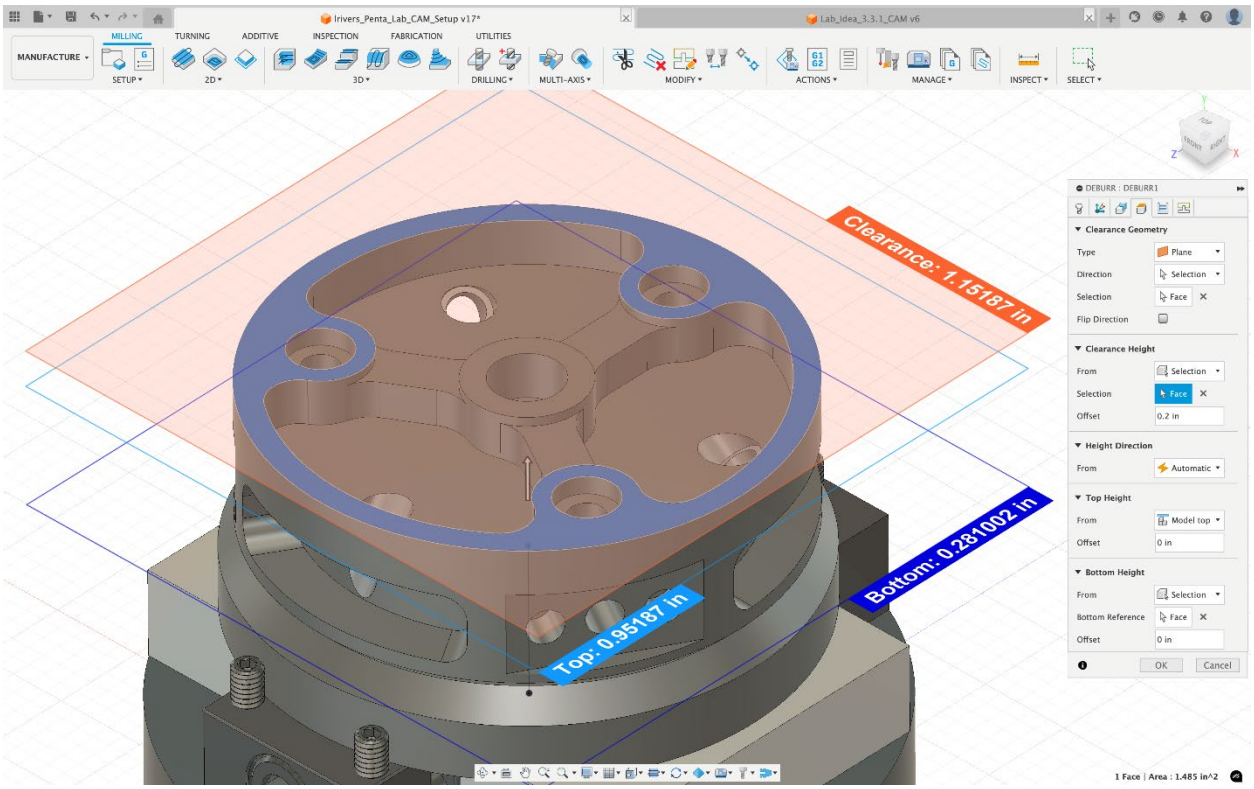
5.4: Heights Tab

For this part, the automatic heights are too large for the machine we are using. We will need to set them manually to avoid the machine stopping with an error.



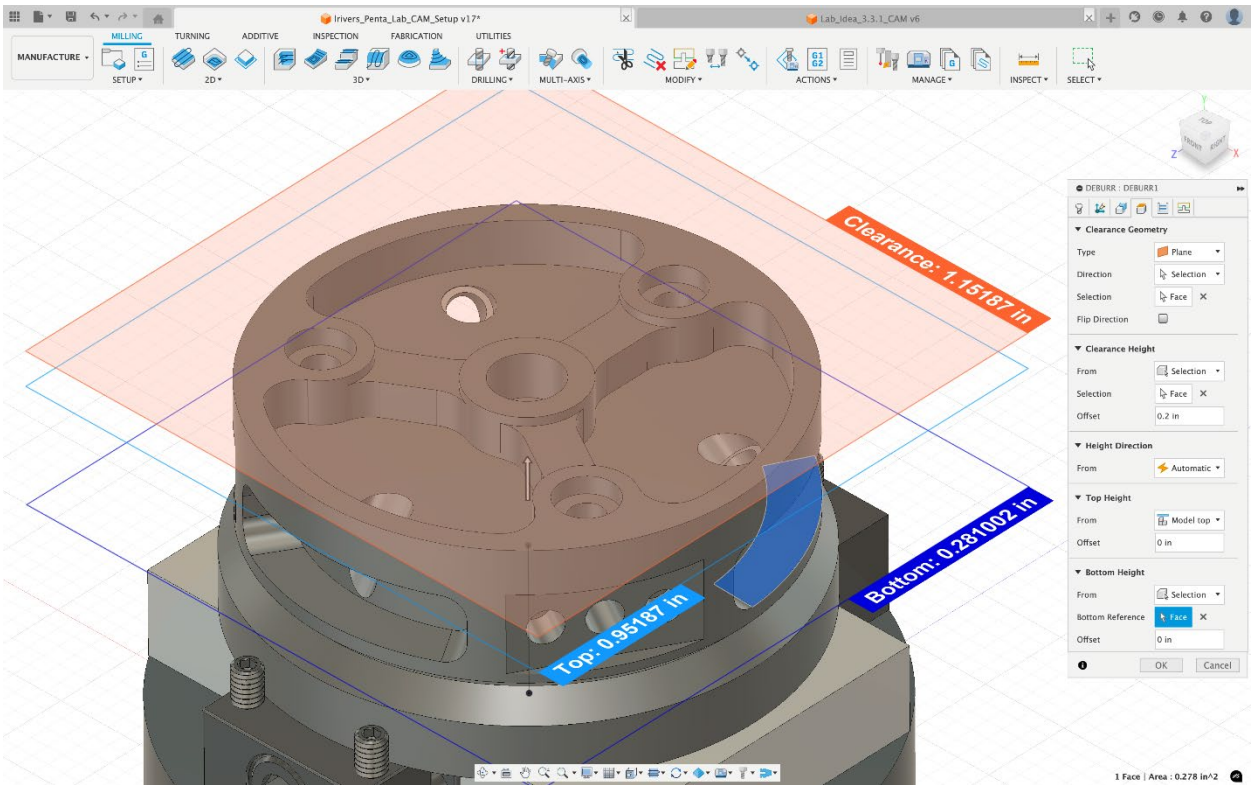
Set the clearance geometry type to “Plane” and the direction to “Selection”.

The selection should be the top face of the part. This is shown in blue in the image above.



Change the “From” selection in the clearance height to “Selection” and select the face shown in blue in the image above.

Adjust the offset to 0.2 inches.

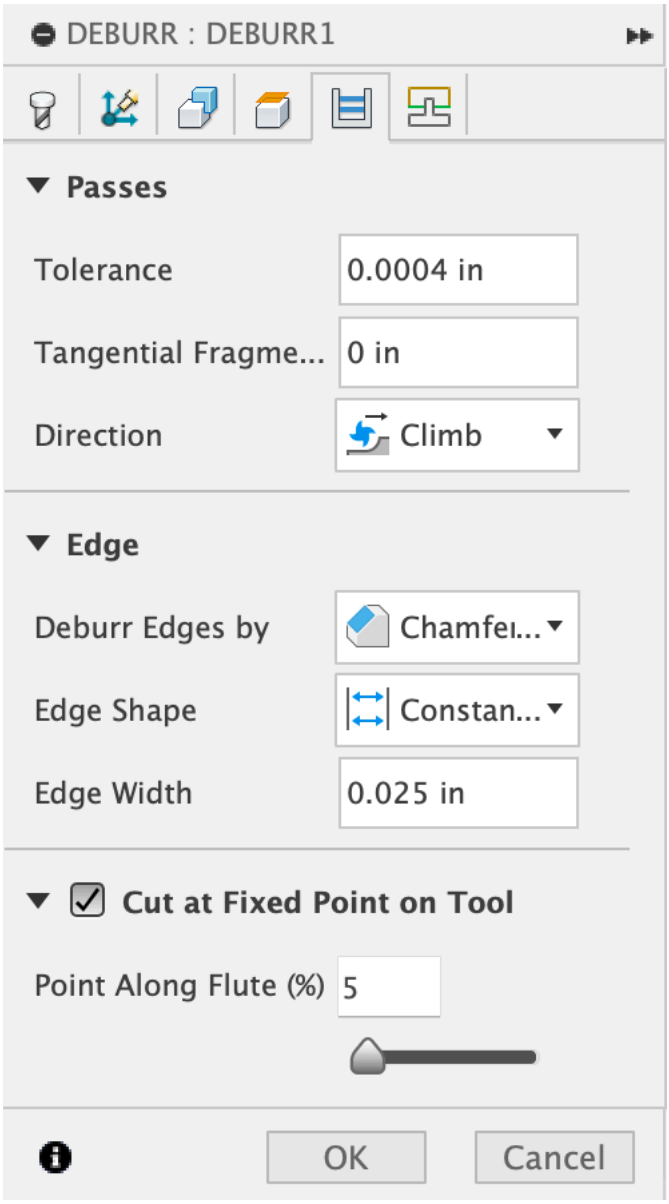


Change the top height to “Model Top”.

Next, change the bottom height to “Selection”. Select the lowest wall of a side pocket for this selection.

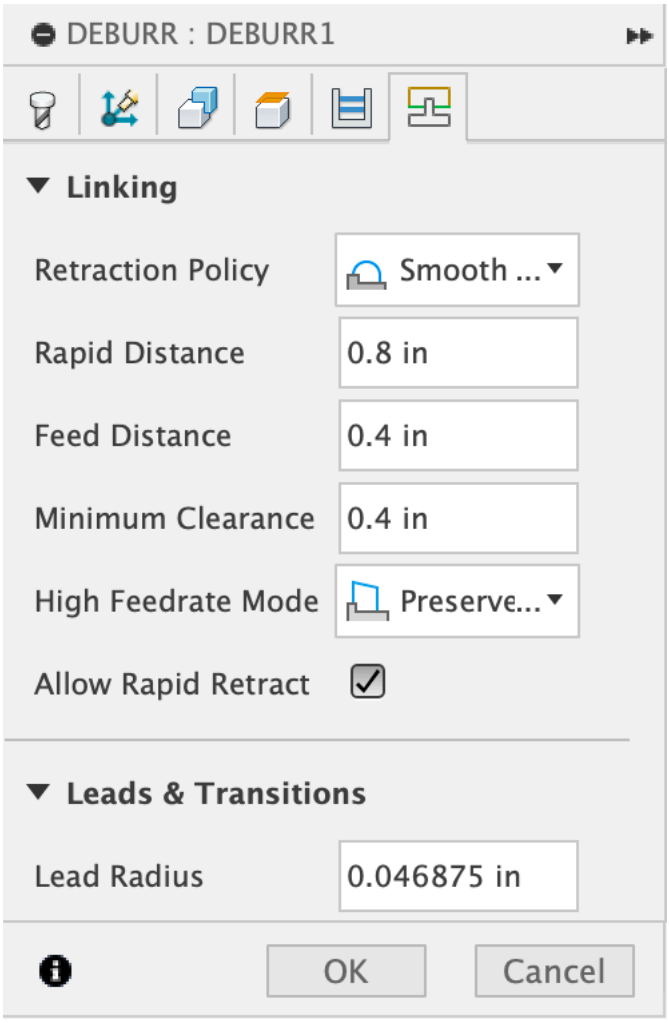
This bottom height adjustment is not strictly necessary, but we are doing it to force Fusion to keep the tool further away from the vise for safety.

5.5: Passes Tab



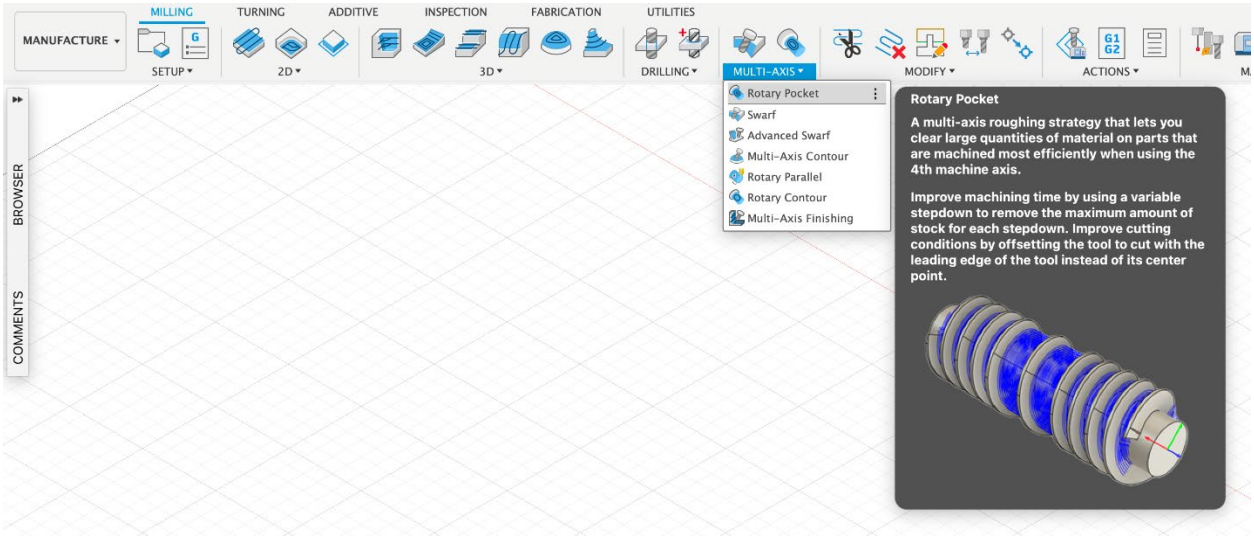
Use the settings above for the passes tab.

5.6: Linking Tab



Change the retraction policy to “Smooth retraction” in the linking tab.

Step 6: Mill The Undercut

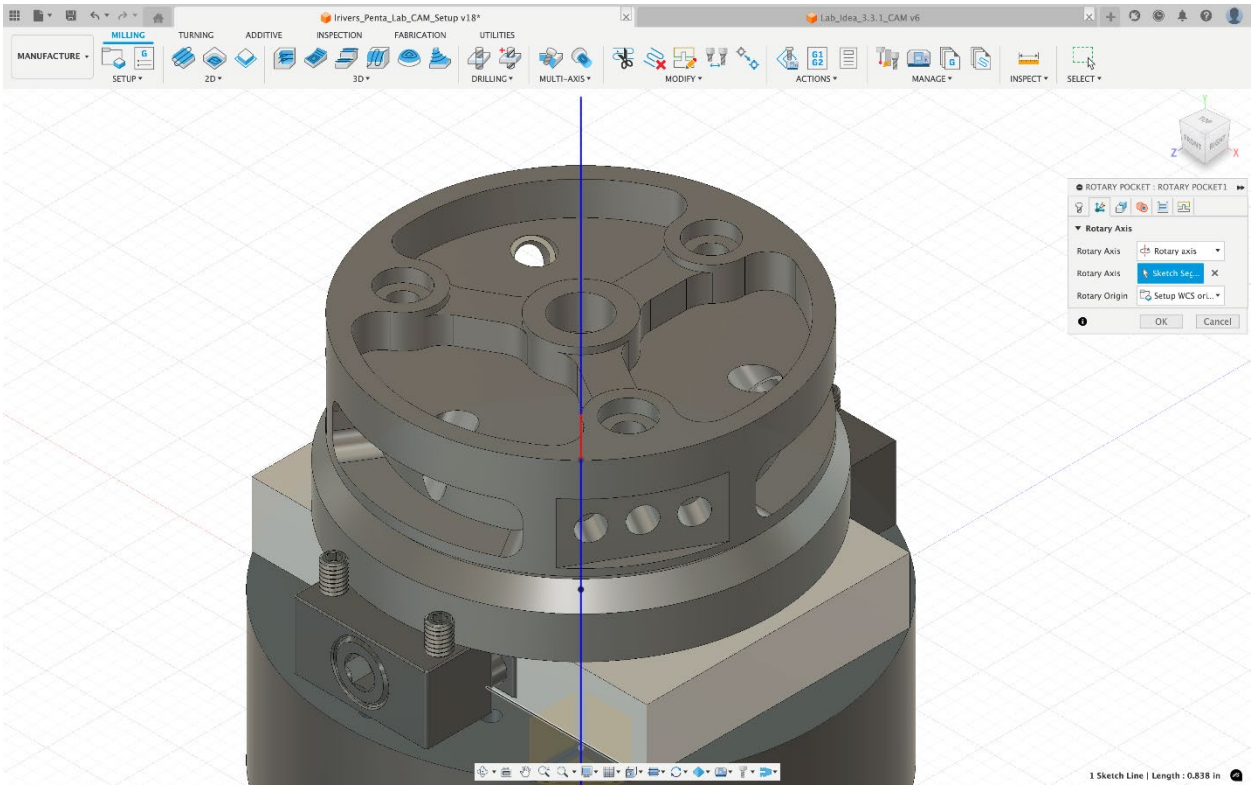


Create a rotary pocket operation. This operation type can be found in the “Multi-Axis” dropdown.

6.1: Tool Tab

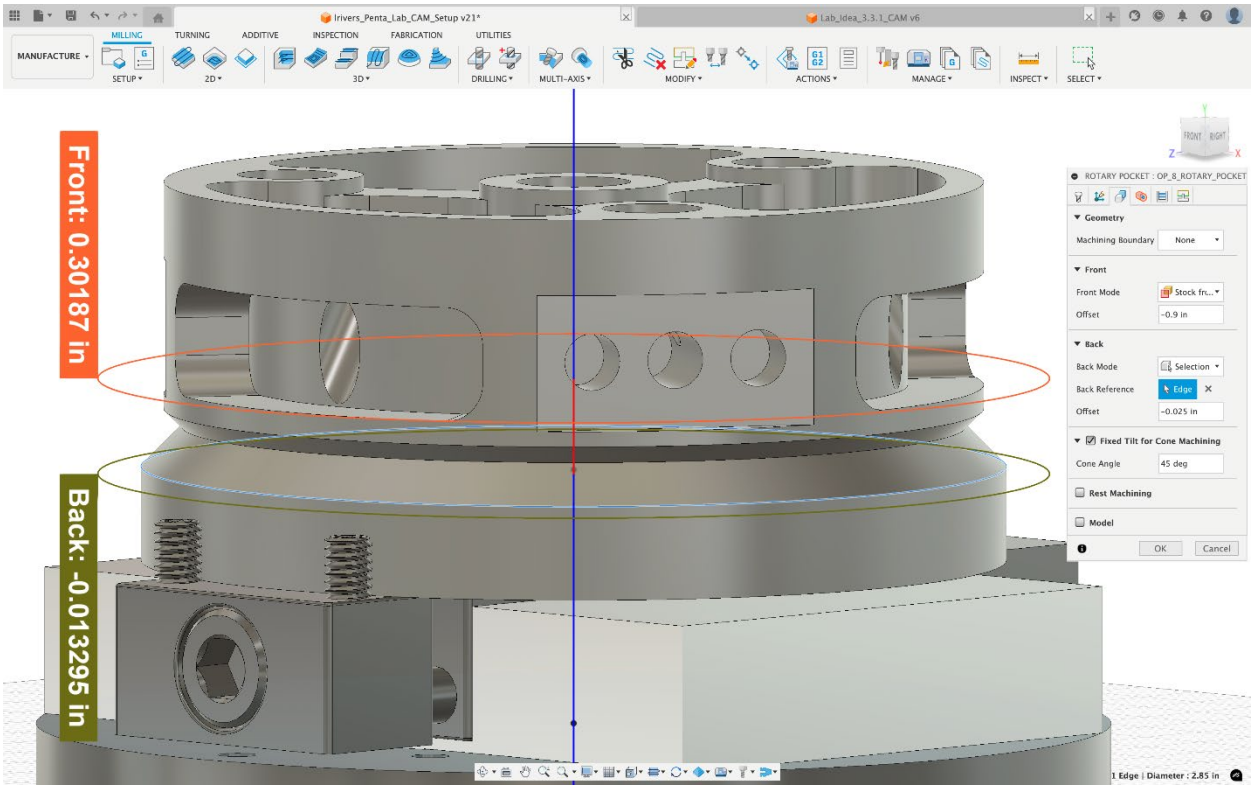
Use the settings and selections from 3.1: Tool Tab.

6.2: Multi-Axis Tab



Change the rotary axis from “Setup Z Axis" to “Rotary Axis” and select the B-table offset sketch as the rotary axis.

6.3: Geometry Tab



Next, we need to constrain the toolpath to only include the area we want to machine. There are multiple ways to do this, but the geometry tab is simple and works well for this part.

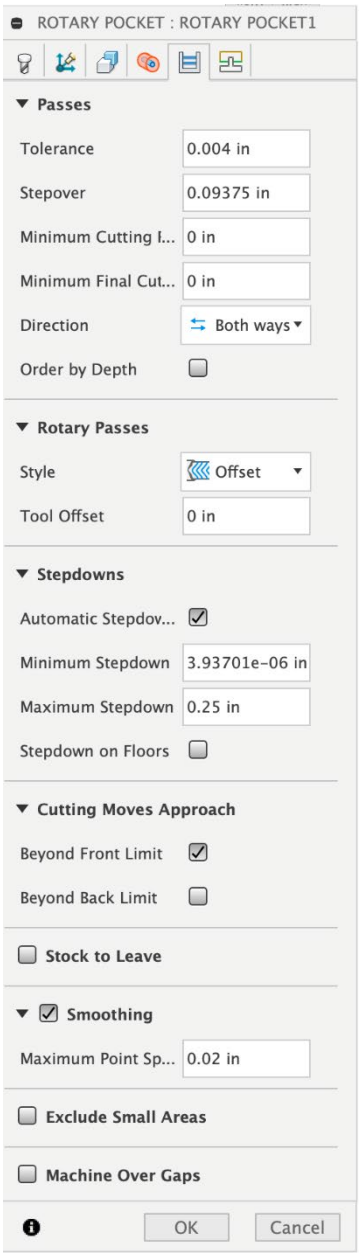
Make the following changes:

- Change the offset under the “Front” dropdown to –0.9 inches.
- Change the “Back Mode” to “Selection” and select the edge at the bottom of the undercut feature. This is shown in light blue in the image above.
- Change “Offset” under the “Back” dropdown to –0.025 inches.

6.4: Radii Tab

The radii tab can remain unchanged.

6.5: Passes Tab



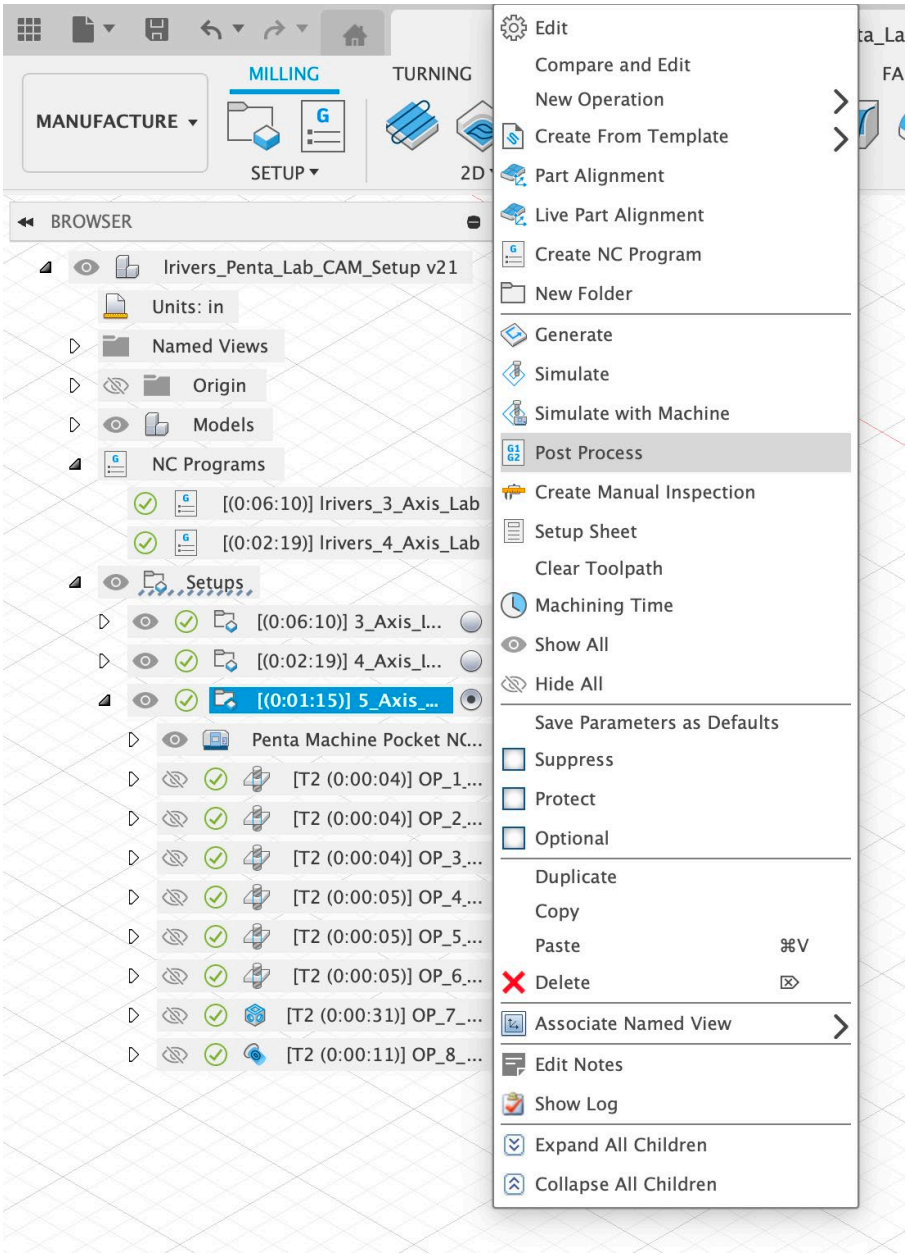
- Check that the stepover is set to 0.09375 inches
- Turn on “Automatic Stepdown” and set the “Maximum Stepdown” to 0.25 inches.
- Turn off “Stock to leave”.
- Turn on “Smoothing”.

6.6: Linking Tab

Change the retraction policy to “Minimum Retraction”.

Step 6: Post Process Operation and Modify G-Code

6.1: Post Process the 5-Axis Operation



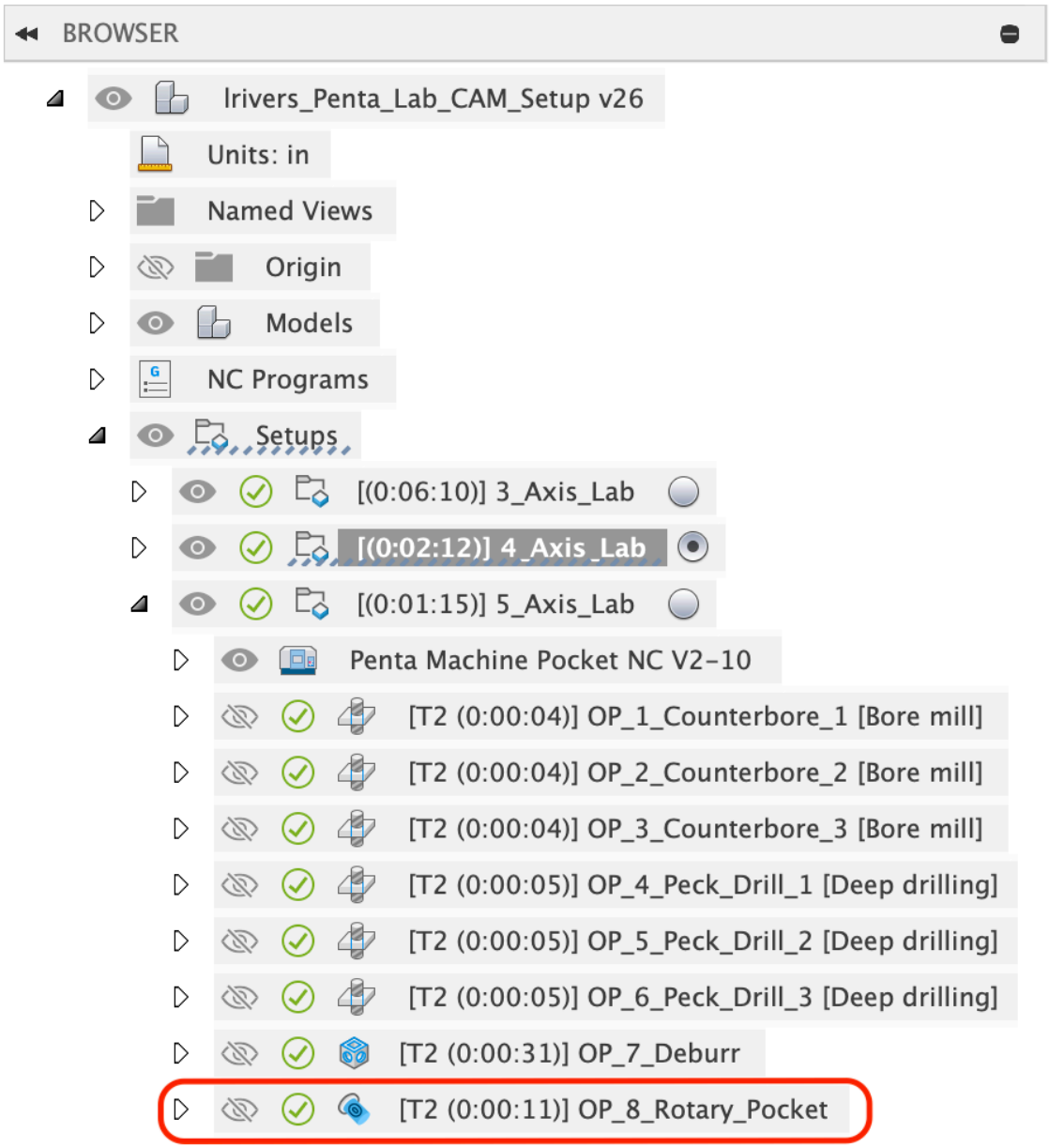
Post process your 5-axis lab as usual.

Your program is nearly ready for the machine, but there is one change to the G-code that will reduce machining time by over 30 seconds by getting rid of B-table unwind.

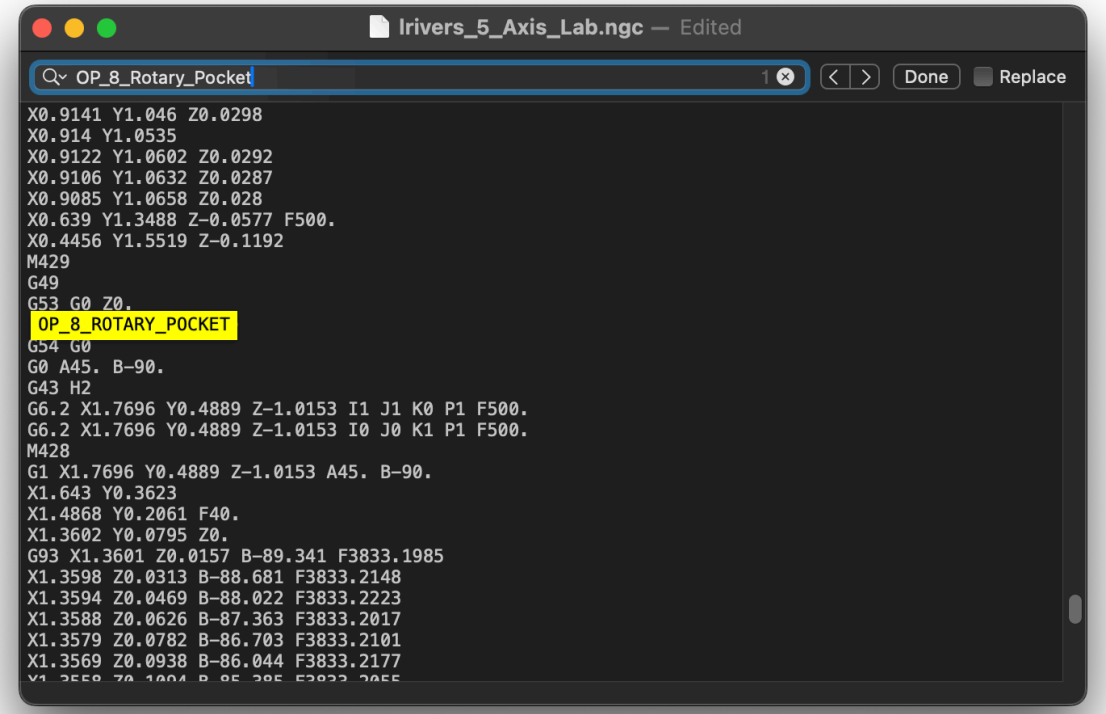
6.2: Search G-Code

Open the G-code that the post processor generated from your 5-axis operation in a text editor. Notepad works well for this (TextEdit for Mac users).

Once you have opened the G-code in a text editor, search the .ngc file for the name of your setup's last operation (keyboard shortcut control/command "F").

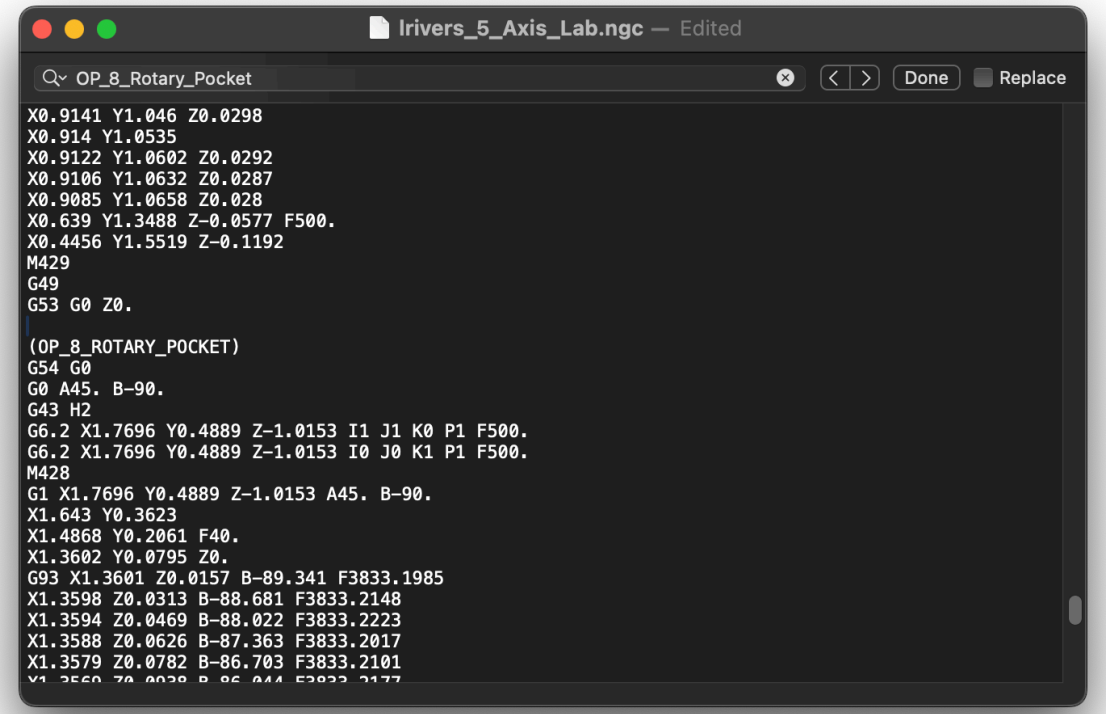


For the 5-axis lab setup, I named my last operation "OP_8_Rotary_Pocket", so this is what I will search for in the G-code.

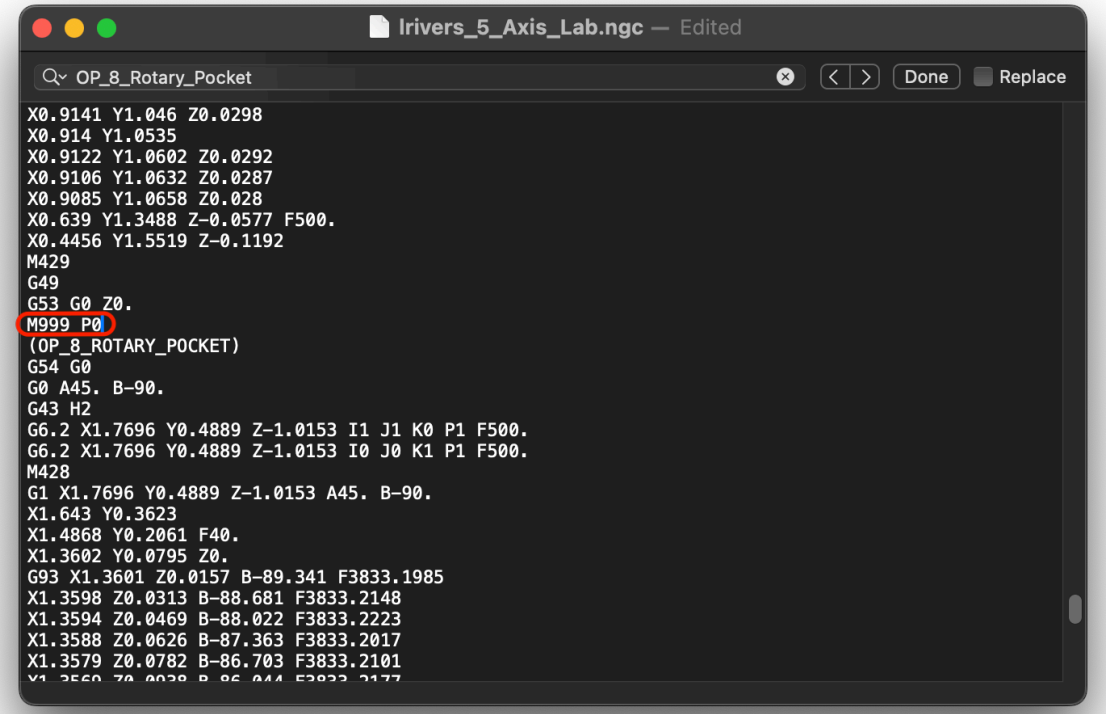


Since operation names are inserted into the program as comments, your search should find the start of the operation.

6.3: Edit G-Code



Once you have located the start of your last operation, look to the line above it. Click to the right of the period in the line “G53 G0 Z0.” and press enter. This will create a space for you to manually add a line of code.



After you have done this, type “M999 P0”.

Note: The “P” value is the desired position of the B-table and is a **zero**, **NOT** an “O”.

This will command the B-table to go to the zero position instead of having to make multiple revolutions to get rid of the accumulated windup that was produced during operation.

Make sure you did not inadvertently make any other changes to the program and save your changes before closing the text editor.

Use the modified .ngc file in lab.