CAM 101 WITH FUSION 360

Developed for CS450HO– Robotic Design and Fabrication Honors

Published by FRC7407 in November 2021

Revised February 2022

WHAT IS CAM?

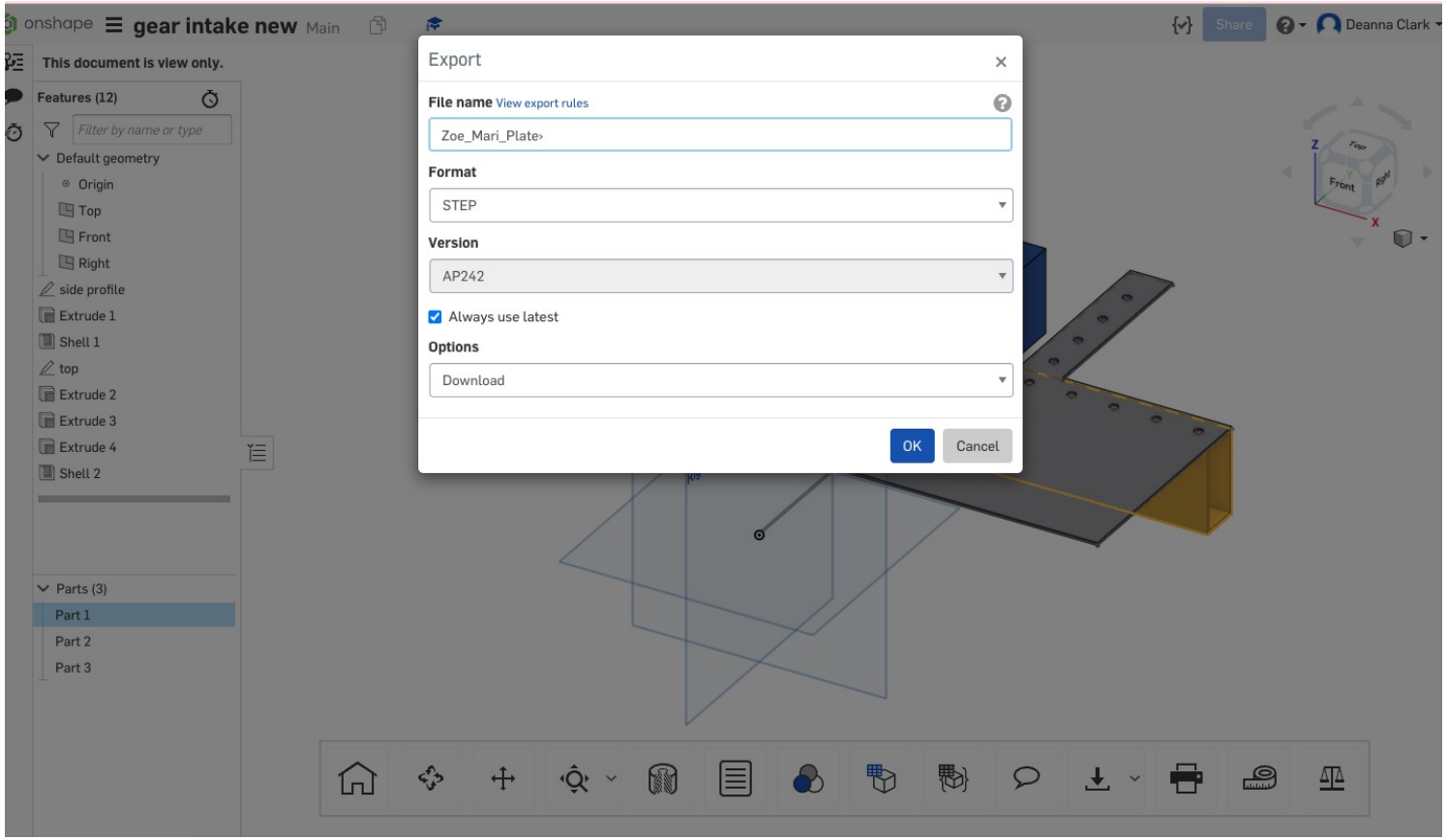
- CAM = computer aided manufacturing
- When we use the term “CAM” a part we are creating tool paths that the CNC router uses to cut out our designed part
  - CNC = computer numerical control
- CAM software create g-code that the CNC read
  - G-Code is essentially a series of X,Y & Z coordinates that tell the router bit where to travel to at each point in the cutting process

FUSION 360 LICENSES

- Autodesk Fusion 360 is a CAD and CAM software
- It is very user friendly and has useful features such as automatic tabs to hold the piece down on final cutting passes
- Need to verify student status at Choate with Autodesk to get a license
  - https://www.autodesk.com/education/edu-software/overview?sorting=featured&filters=individual

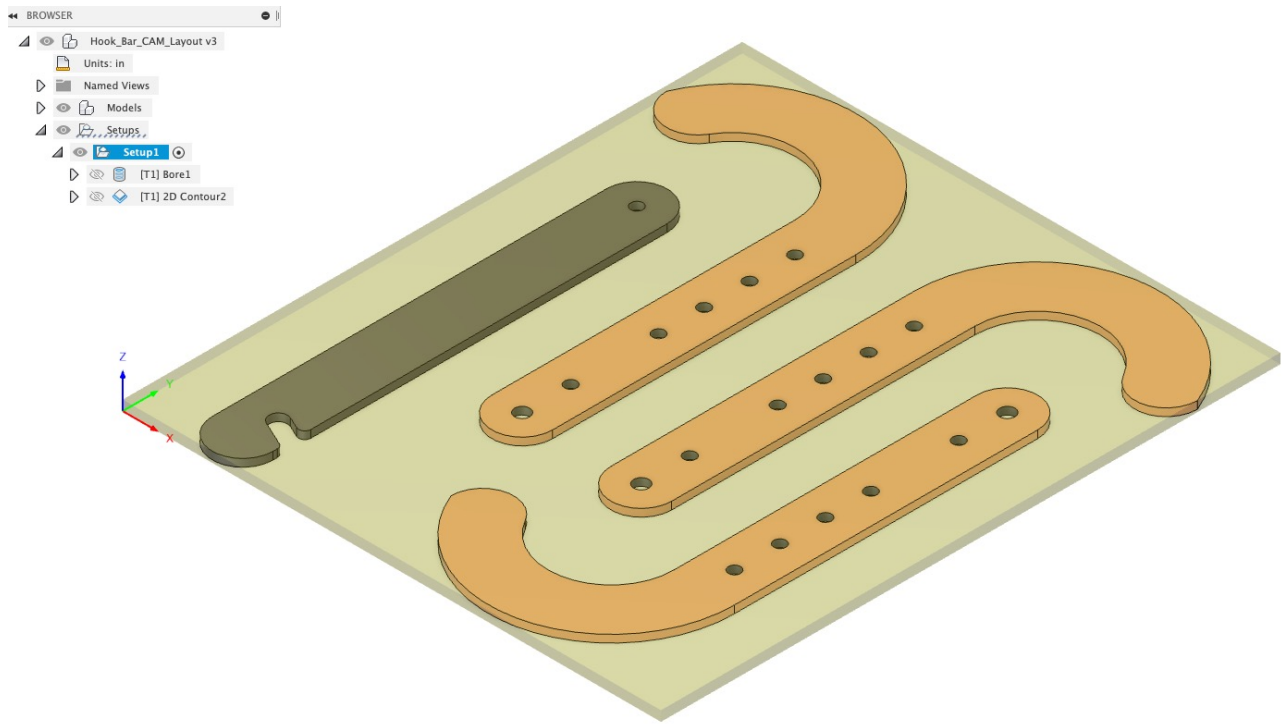
EXPORTING CAD FROM ONSHAPE

- Right click on part in Onshape and select “Export”
- Export as a .STEP file



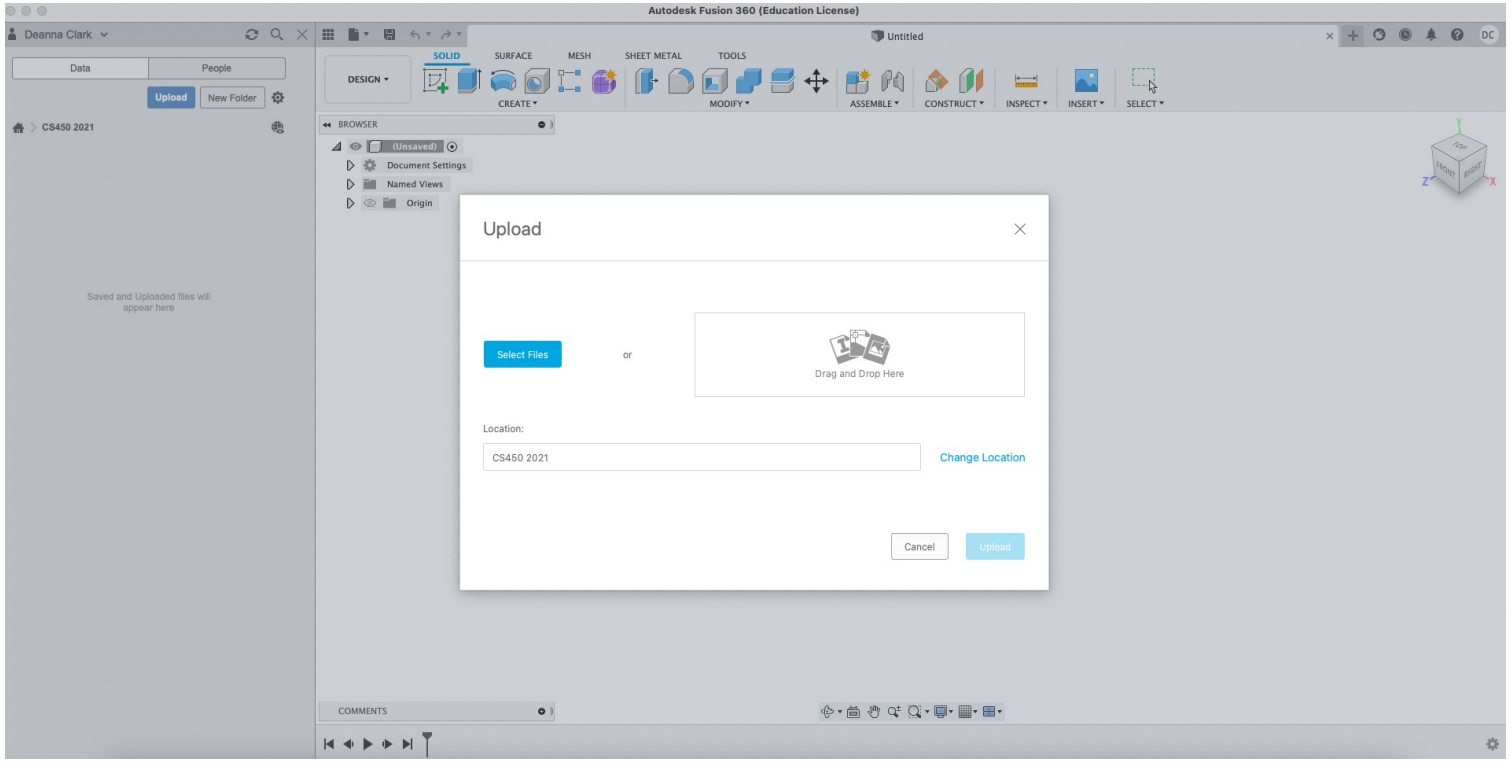
EXPORTING CAD FROM ONSHAPE

- If you are cutting multiple pieces in one operation, you will need to create a flat sheet assembly in Onshape of all your parts (mate all to same place) and layout in most material efficient manner
- Export entire assembly and individual parts, import all of these into Fusion 360
  - See example of a CAM layout assembly to the right



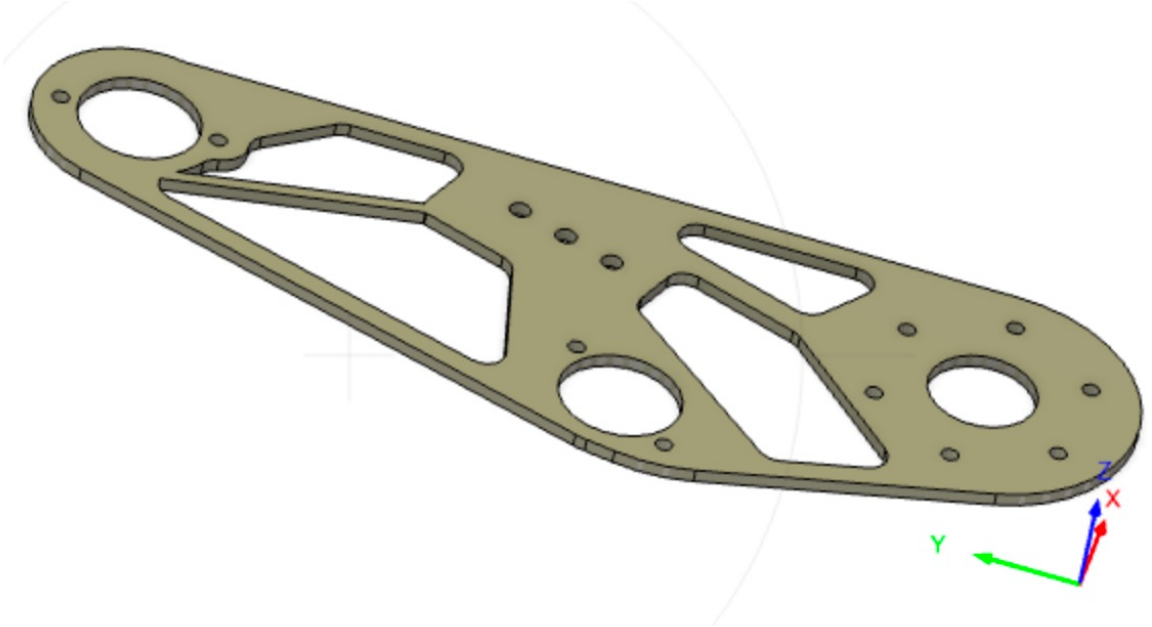
IMPORT INTO FUSION 360

- Open Fusion 360
- Select “Upload” and choose .STEP file



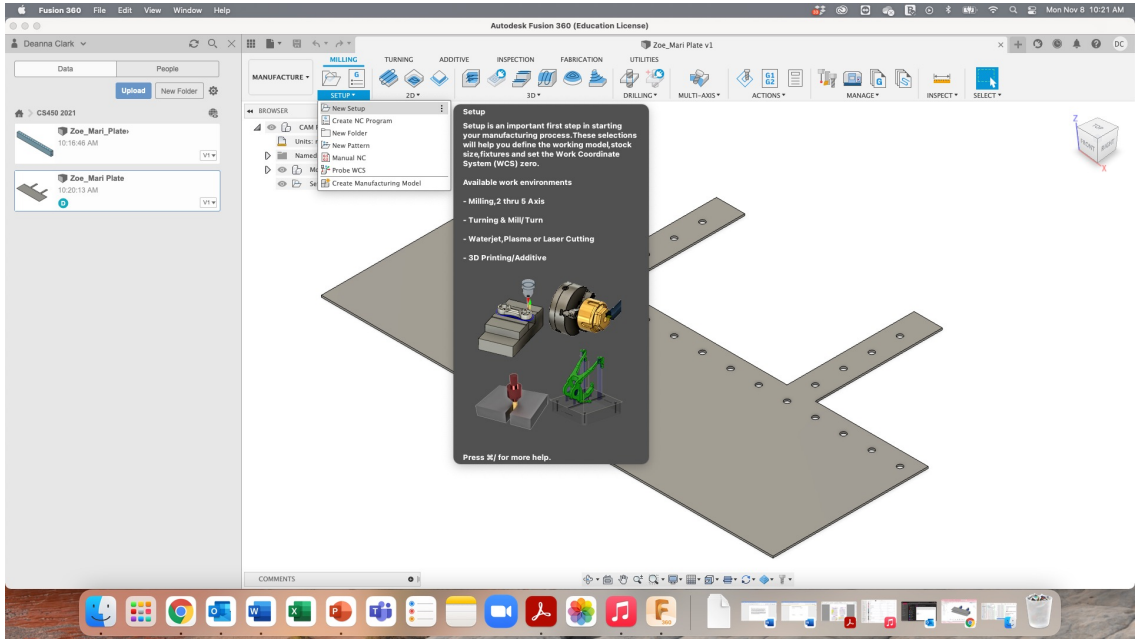
CUTTING PLATES ON THE OMIO

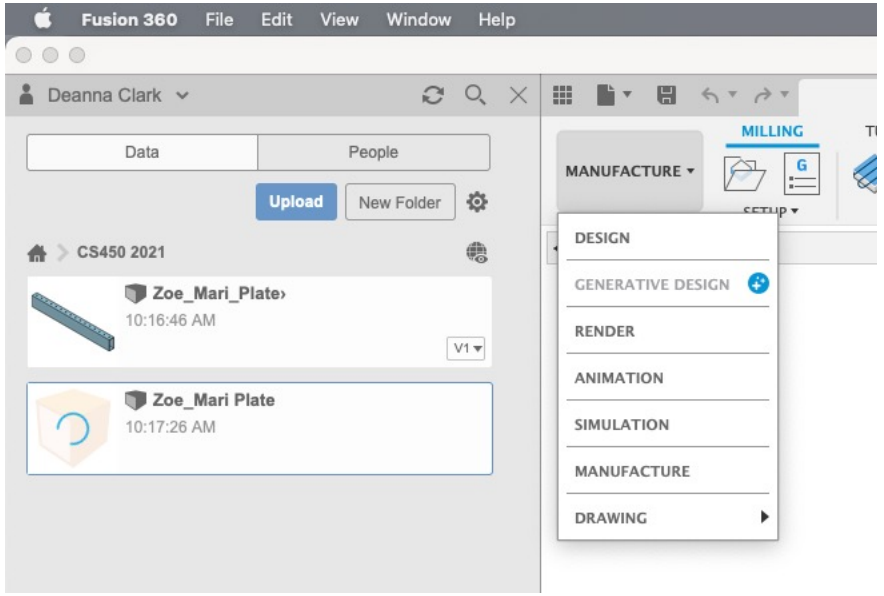
This section will discuss how to cut plates of different materials on the Omio using Fusion 360 and Mach3.



CREATE A NEW SETUP

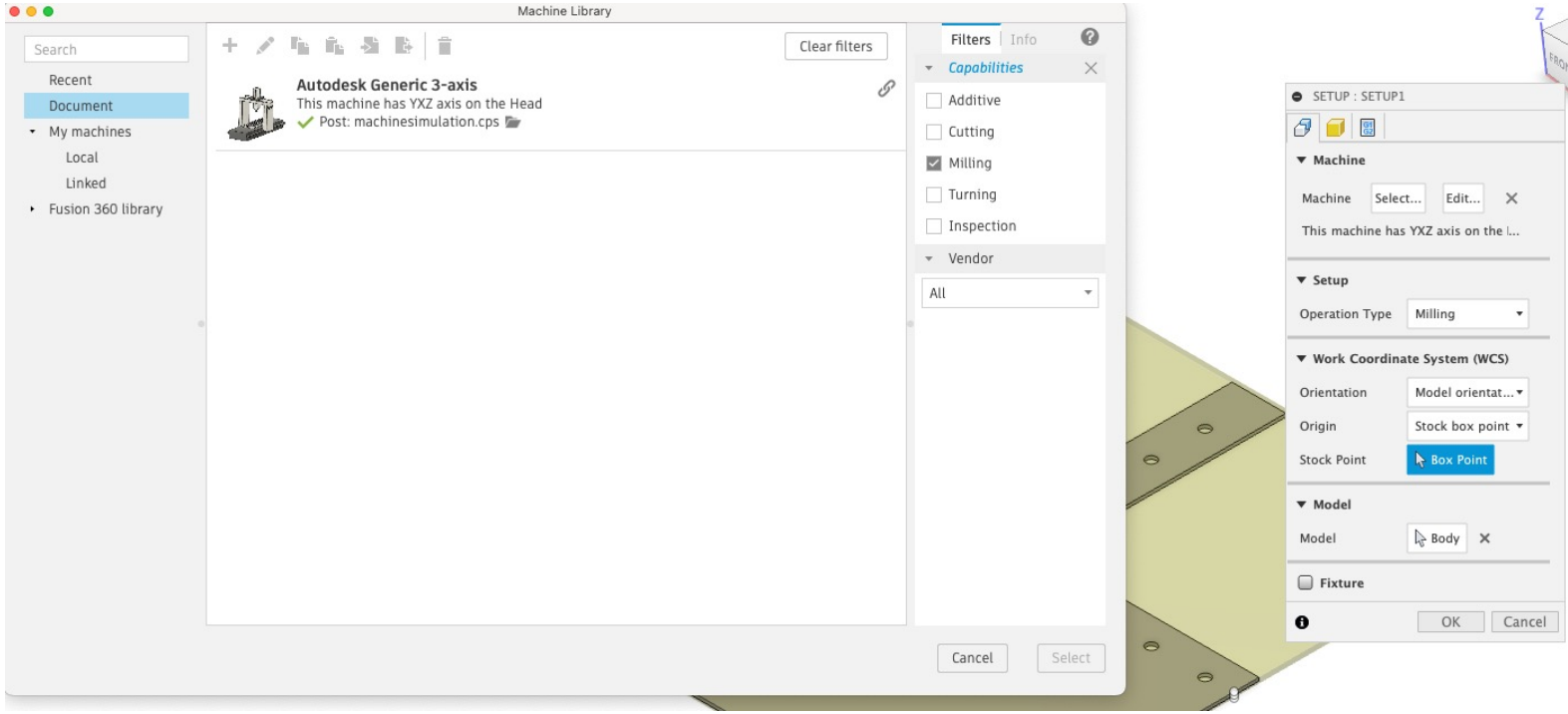
- Navigate to “Manufacture” tab
- Select “Setup” → “New Setup”





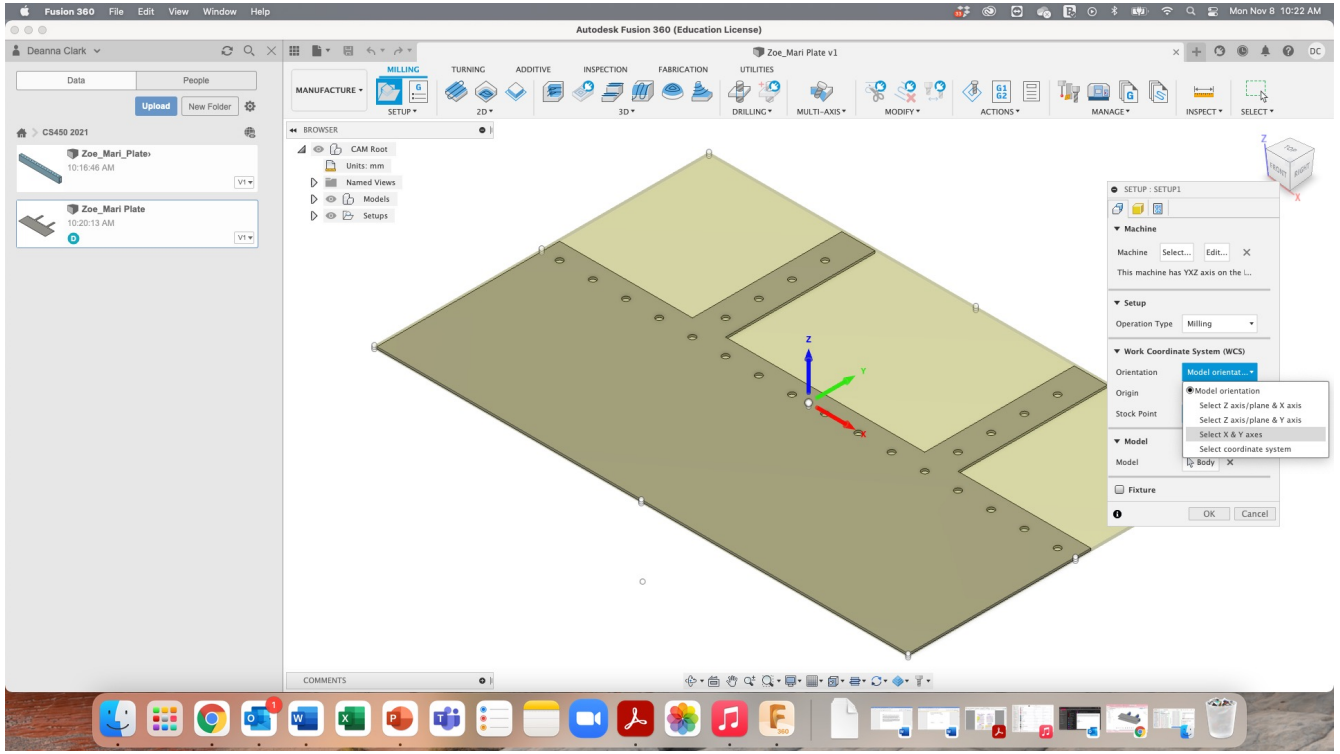
CREATE A NEW SETUP

- Select “Autodesk Generic 3-axis” as machine
  - X,Y,Z on head as that is how our router works



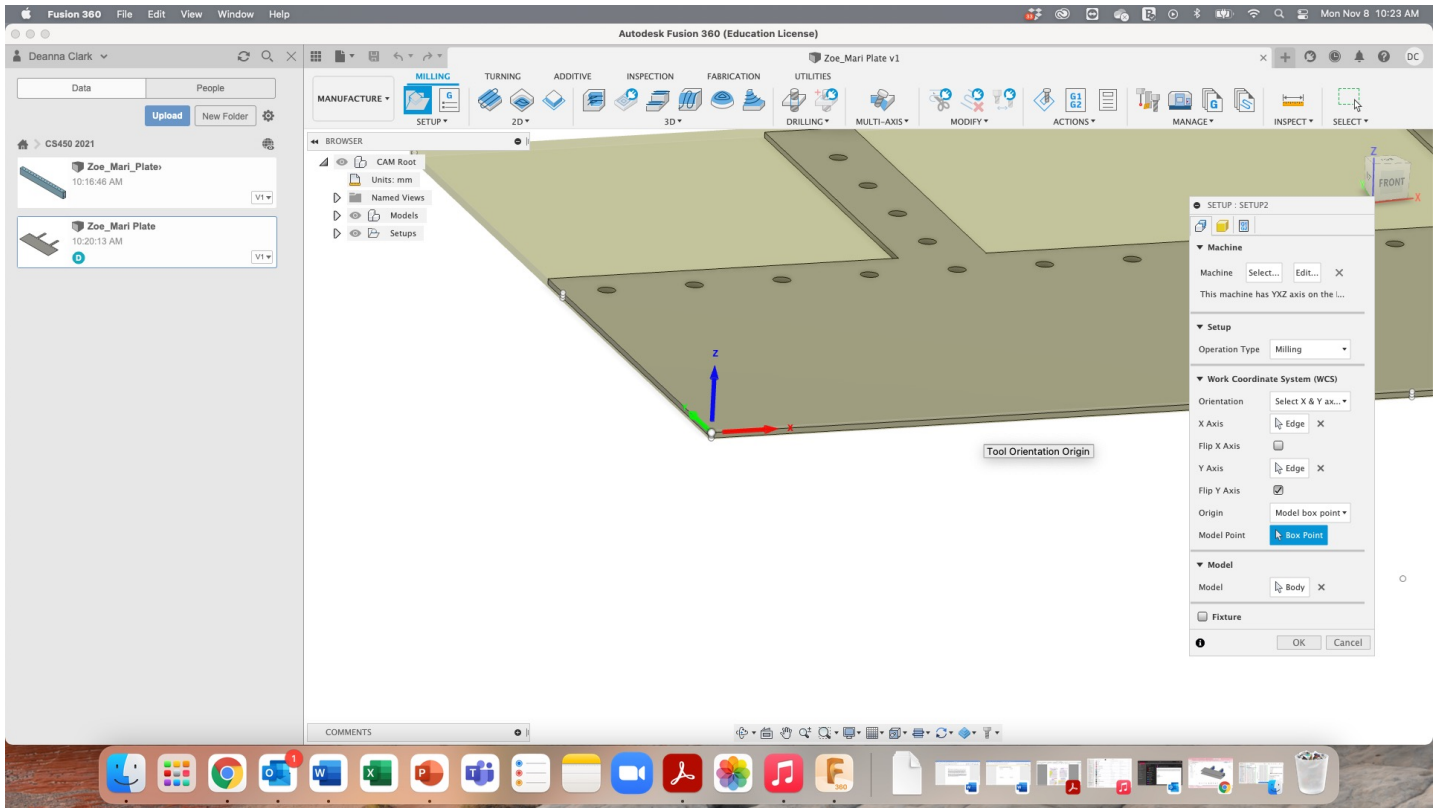
CREATE A NEW SETUP

- Need to select X & Y axes on part → Select x & y axes
  - If your part doesn't have straight lines need to use stock geometry as your axes
  - May need to flip X,Y& Z axes to get right orientation



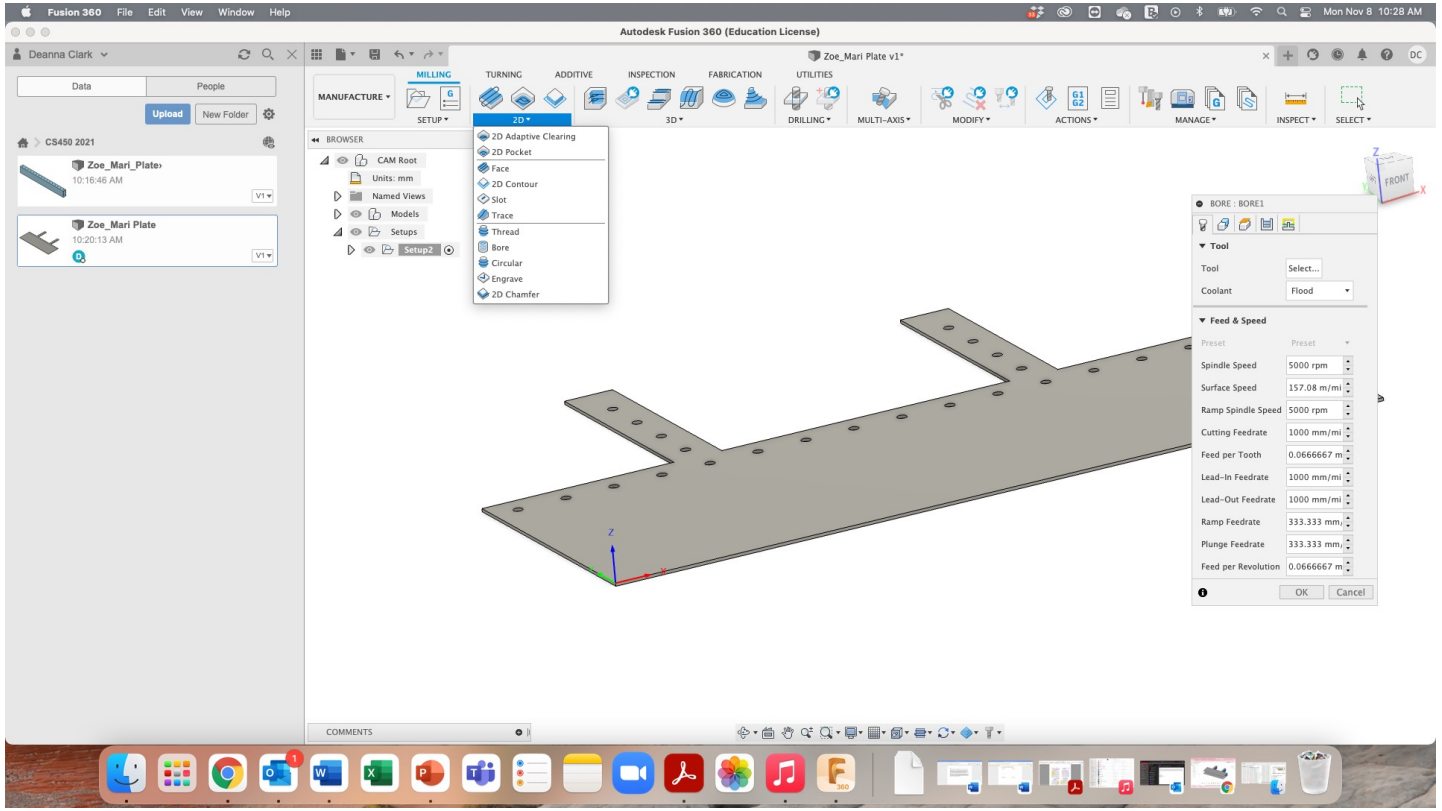
CREATE A NEW SETUP

- Select “model box point” to place origin
  - Place origin in bottom left corner, topmost dot in Z direction



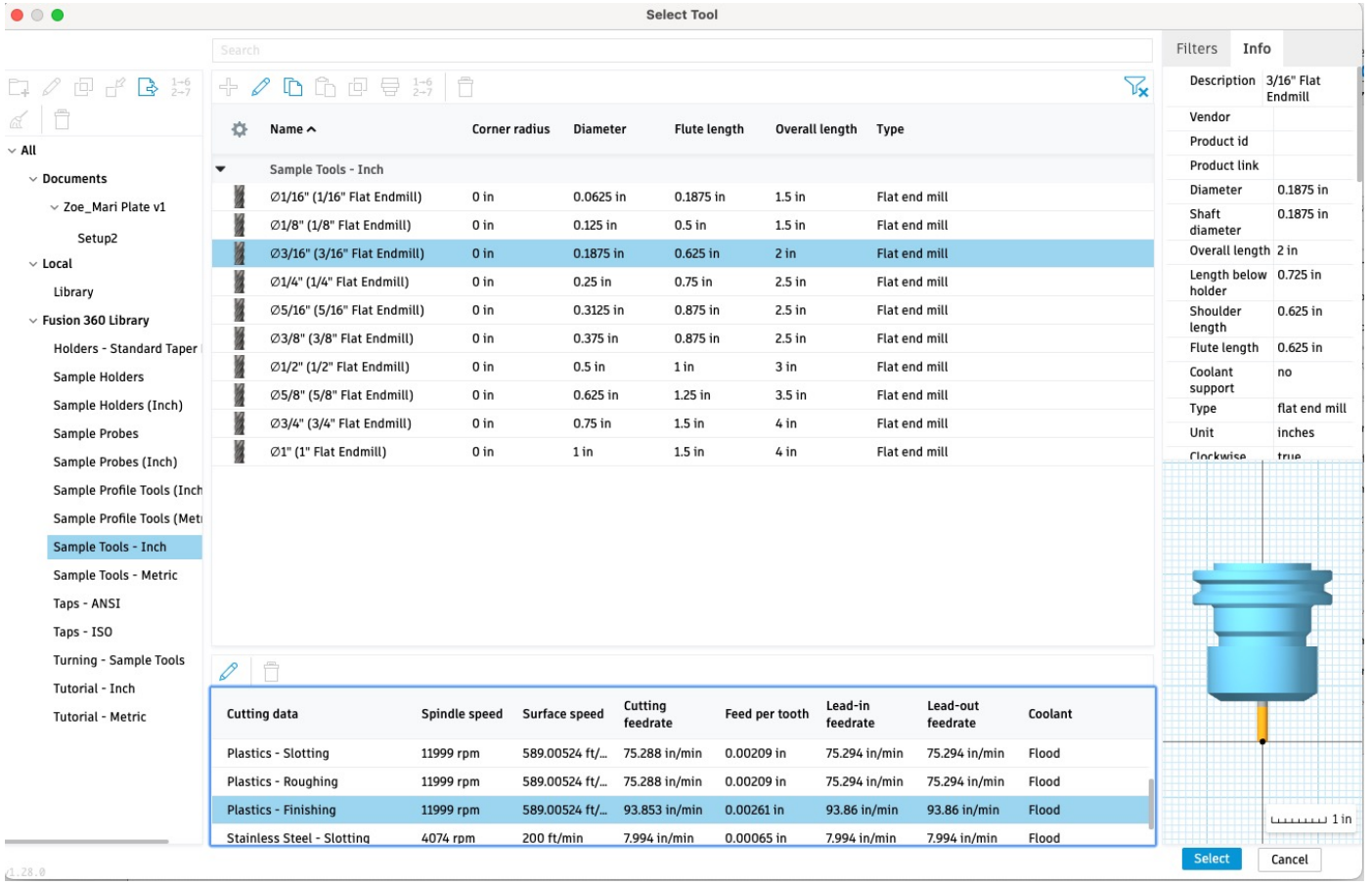
CREATE A 2D BORE OPERATION

- Select 2D → Bore



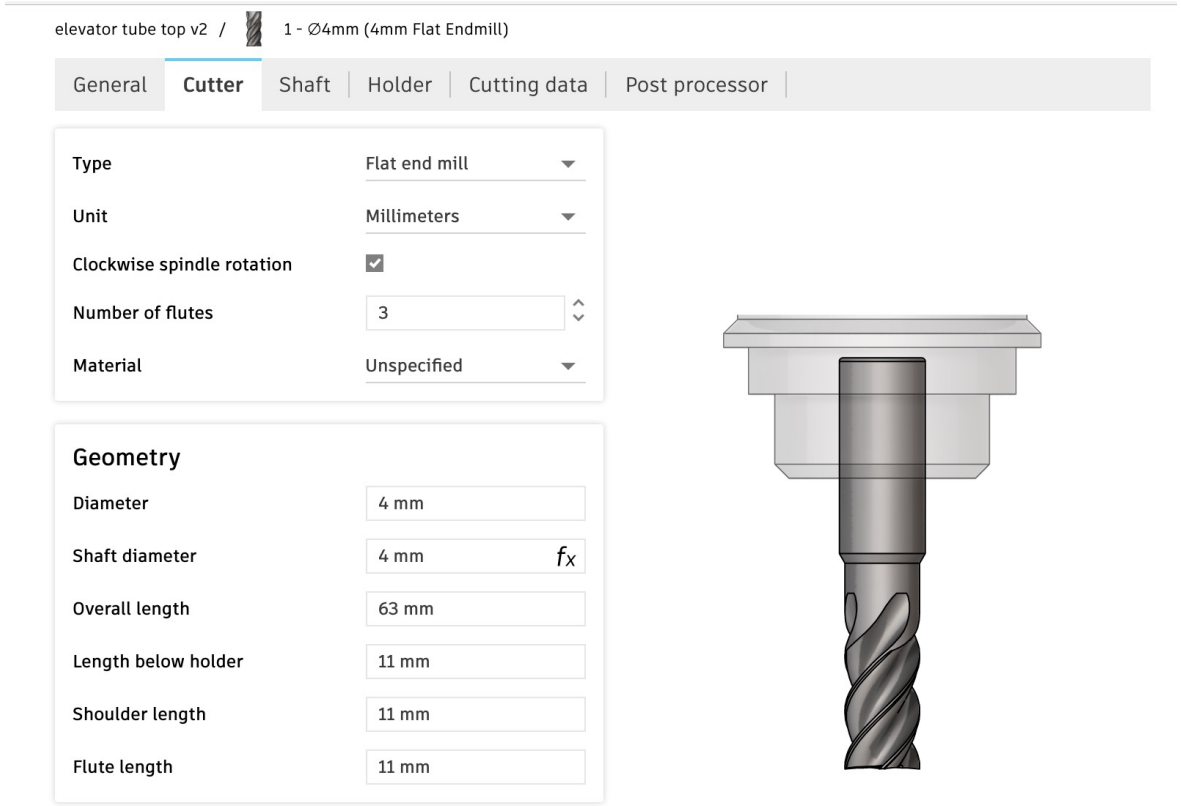
CREATE A 2D BORE OPERATION

- Select cutter from Fusion 360 Tool Library
  - Select appropriate material (plastics or aluminum finishing)
  - Speeds and feeds automatically populated for you!
    - How fast the cutter spins on different operations



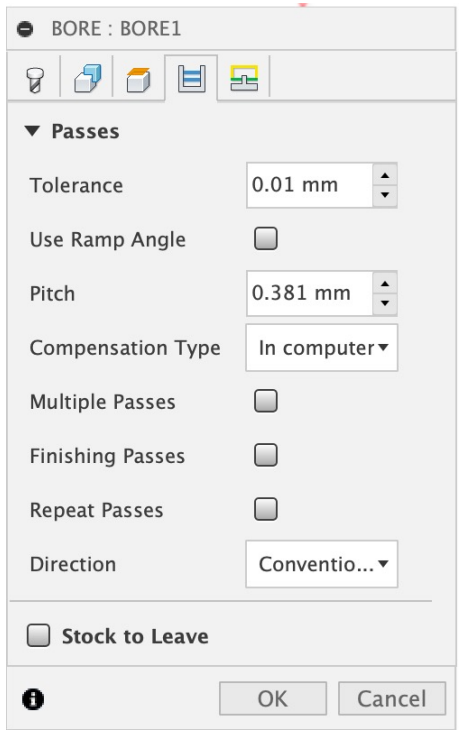
CREATE A 2D BORE OPERATION

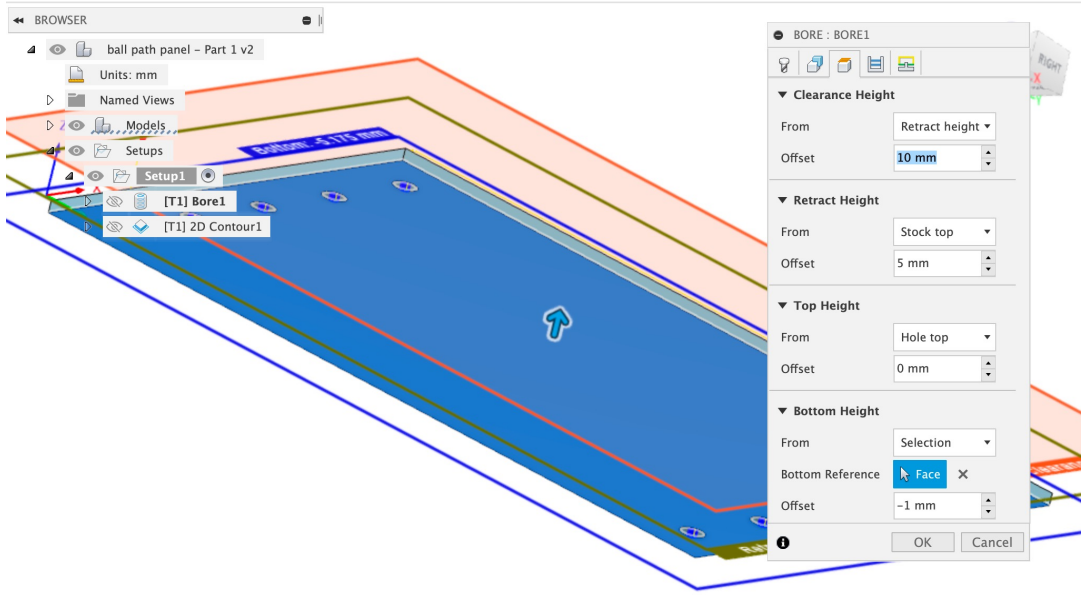
- Modify tool if needed to match your specs
  - Ensure length below holder, shoulder length and flute length match your tool
  - Change number of flutes to match your bit
  - **Fewer flute bits are better for the router**



CREATE A 2D BORE OPERATION

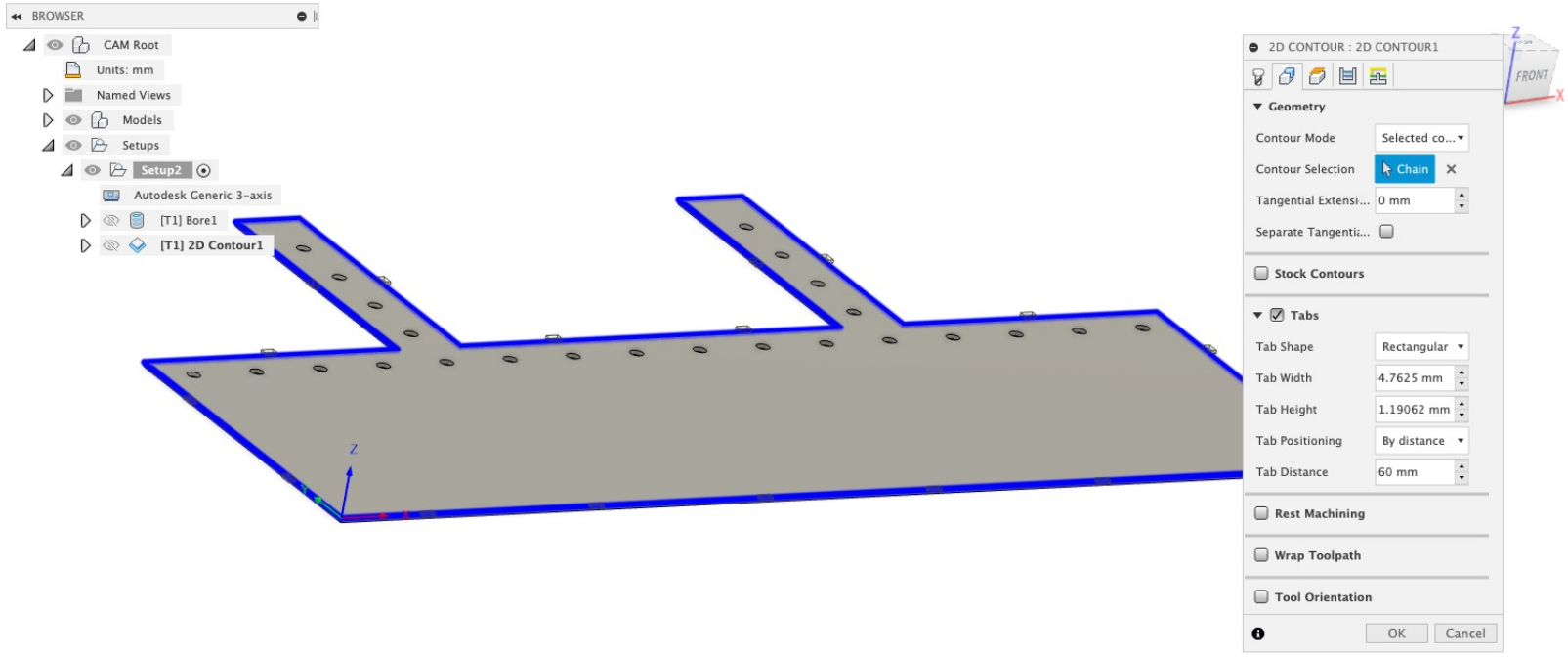
- On third tab of contour operation, change bottom face to “Selection” and select bottom face of part
- Set the tool to go -1mm beyond bottom face – this will ensure part gets cut completely if spoil board is not totally flat
- Set pitch to 0.381mm
- Set direction to “conventional”





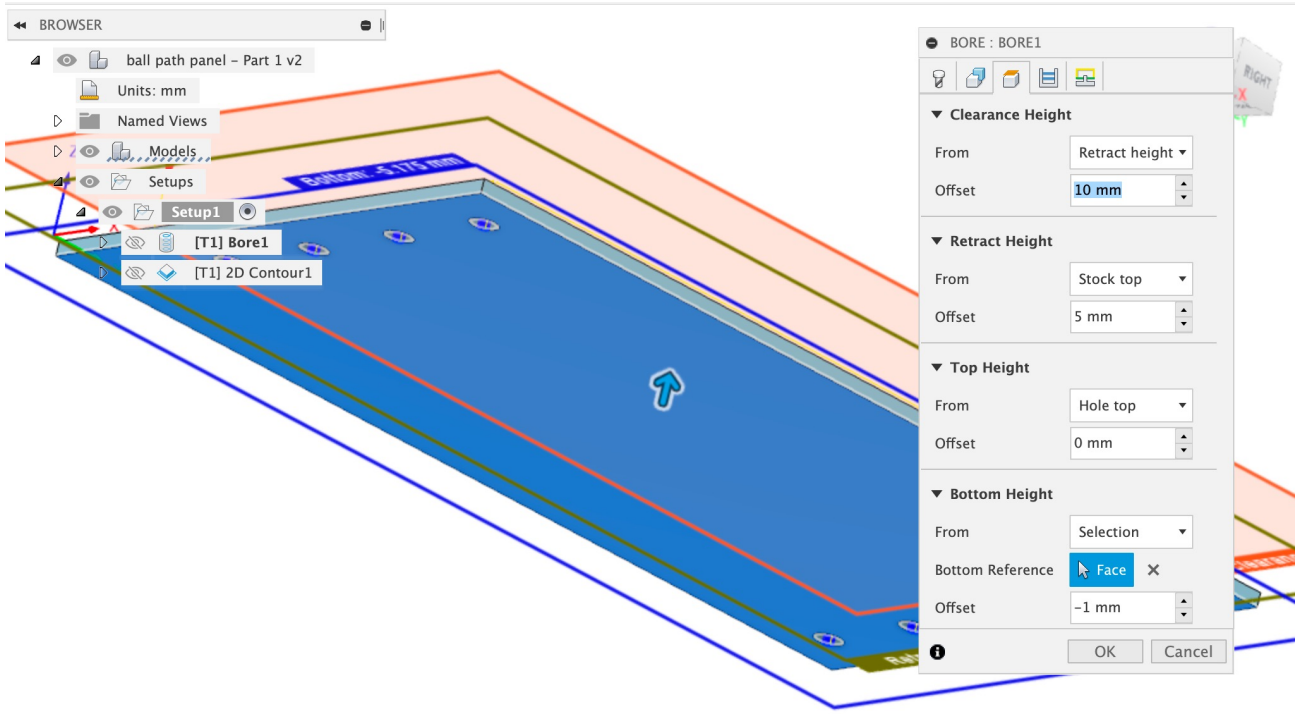
CREATE A 2D CONTOUR OPERATION

- Select 2D contour from 2D menu
- Using same tool
- Select top edge of part, will highlight blue
- Add tabs – these will prevent piece from flying off router after final cut
  - Not enough holes to retain entire part with screws
  - Space tabs out along perimeter of part



CREATE A 2D CONTOUR OPERATION

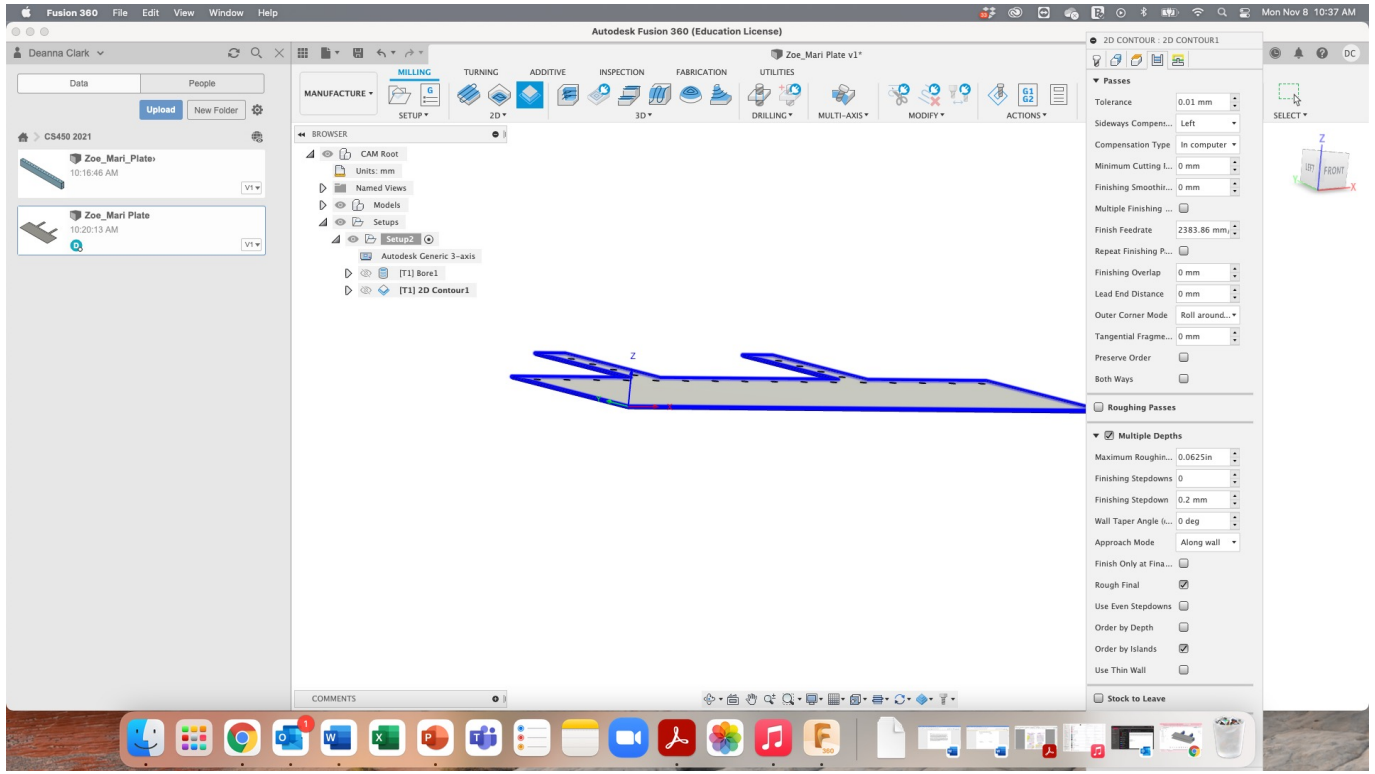
- On third tab of contour operation, change bottom face to “Selection” and select bottom face of part
- Set the tool to go -1mm beyond bottom face – this will ensure part gets cut completely if spoil board is not totally flat



CREATE A 2D CONTOUR OPERATION

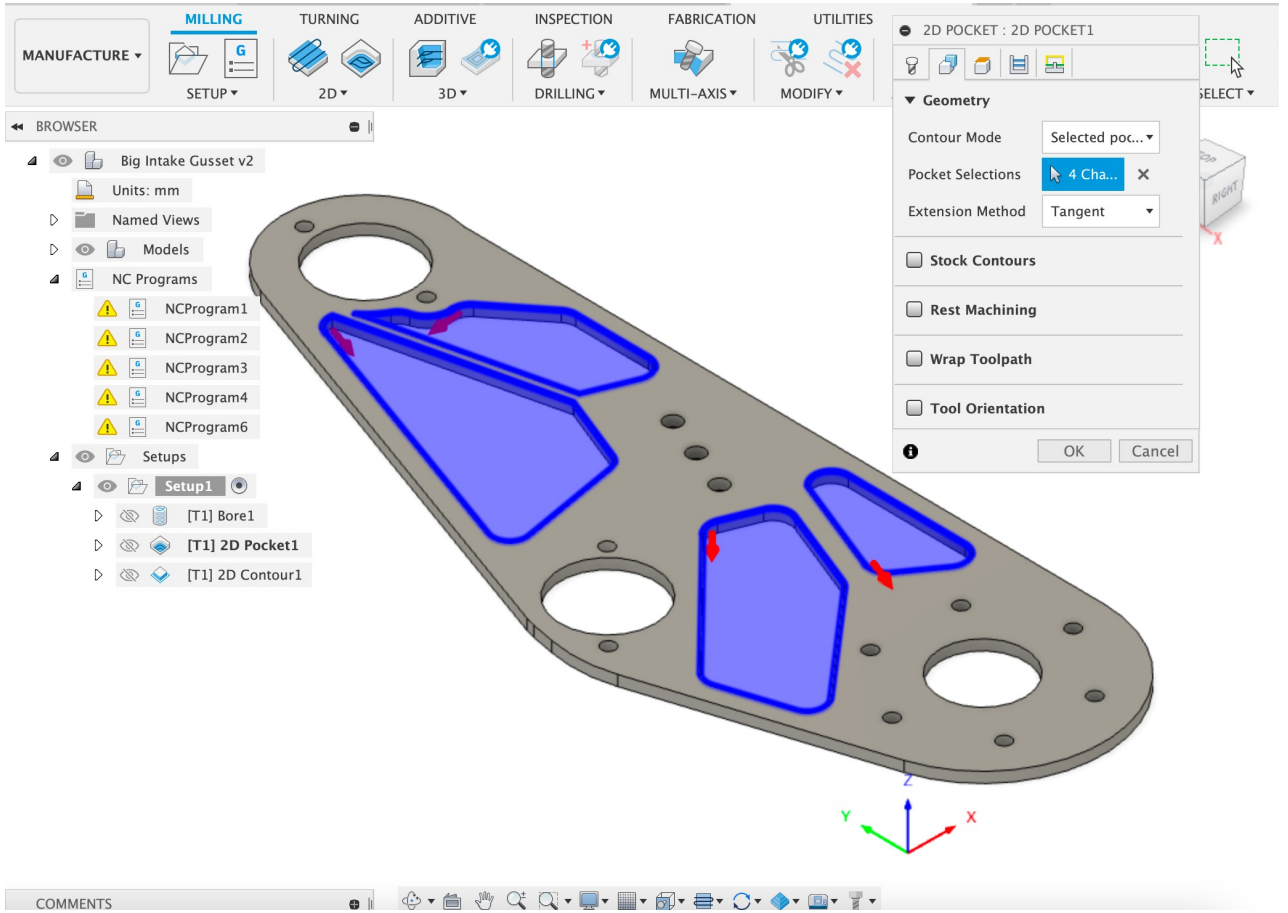
- Go to fourth tab
- Enable “Multiple Depths”
  - “Roughing stepdown” is how much material the machine takes off in the Z-direction on each pass

Material	1/8” 2 flute flat endmill stepdown	4mill 1 flue flute endmill stepdown	6mill 1 flute endmill stepdown
Polycarbonate	1mm	1mm	1mm
Aluminum	0.25mm- 0.4mm depending on complexity of the part	0.5mm	0.5mm



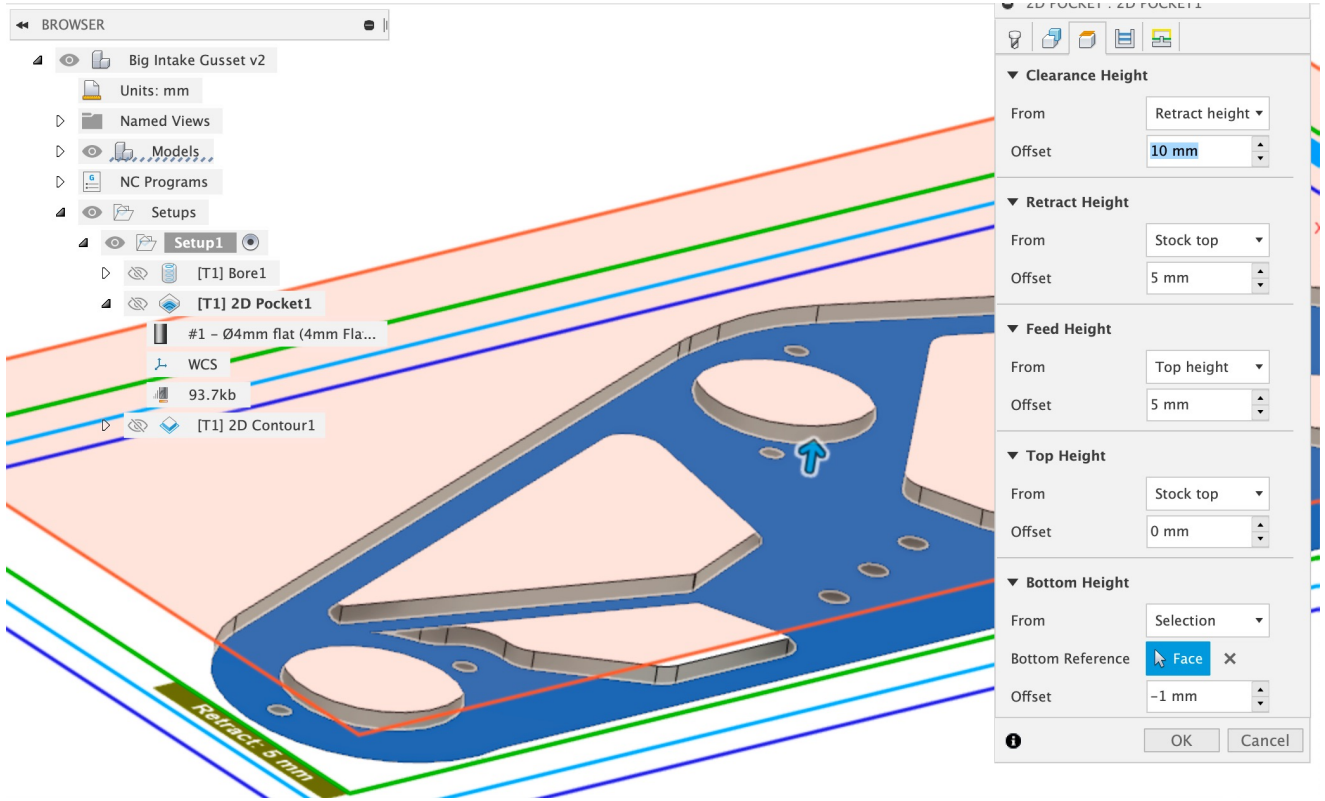
CREATE A 2D POCKET OPERATION

- Pockets can be used when cutting lightening geometry into parts, such as triangles or holes
- Pockets can go all the way through or partially through the material, depending on how you set your faces
- Use same stepdown settings under Multiple Depths as 2D contours



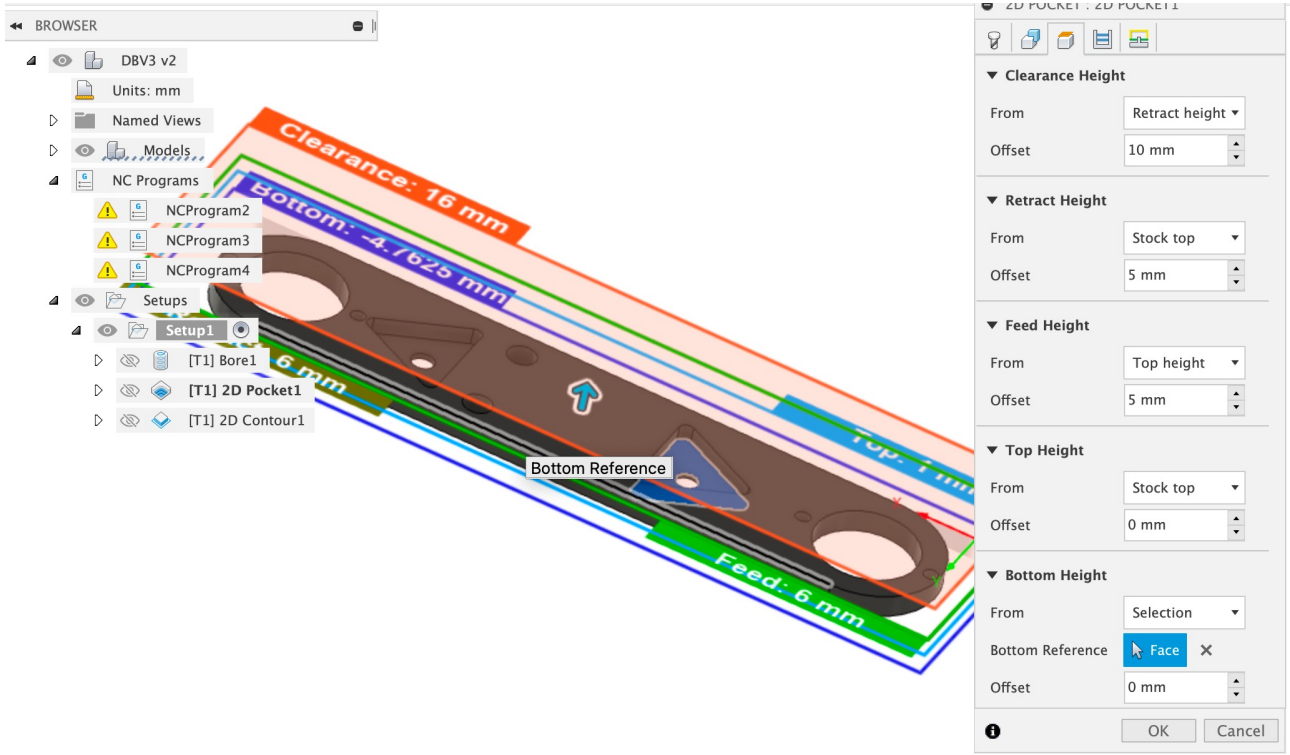
CREATE A 2D POCKET – ALL THE WAY THROUGH

- On third tab of contour operation, change bottom face to “Selection” and select bottom face of part
- Set the tool to go -1mm beyond bottom face – this will ensure part gets cut completely if spoil board is not totally flat



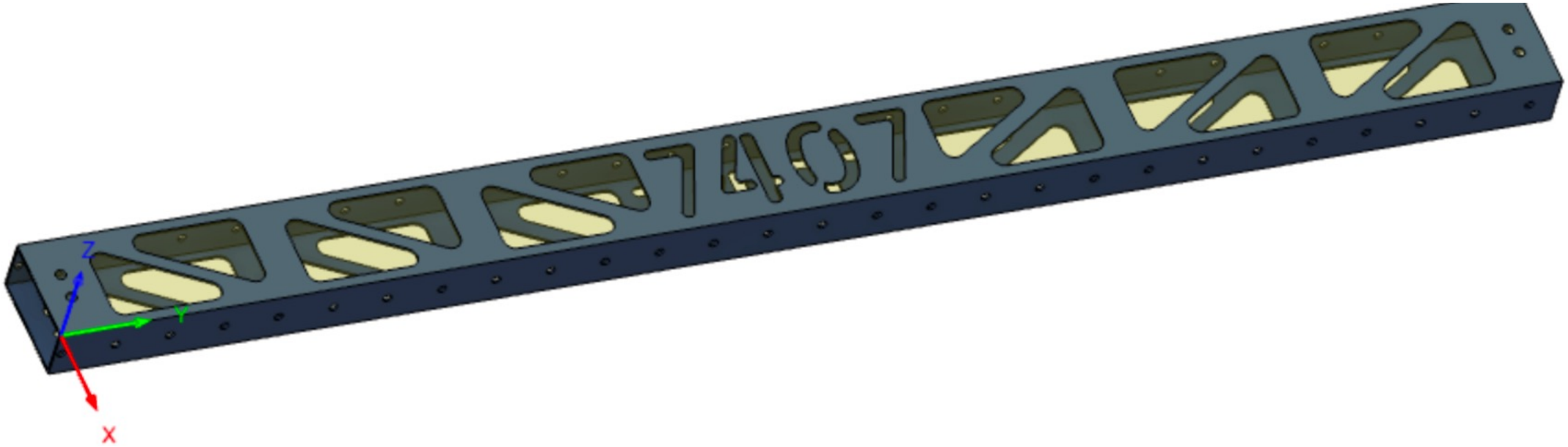
CREATE A 2D POCKET – PARTIALLY THROUGH

- On third tab of contour operation, change bottom face to “Selection” and select bottom face of part
- Set the tool to go 0mm beyond bottom face – this should leave the desired material in the pocket



CUTTING TUBES ON THE OMIO

This section will discuss how to cut metal tubes on the Omio using Fusion 360 and Mach3. You will need a tube jig for this.

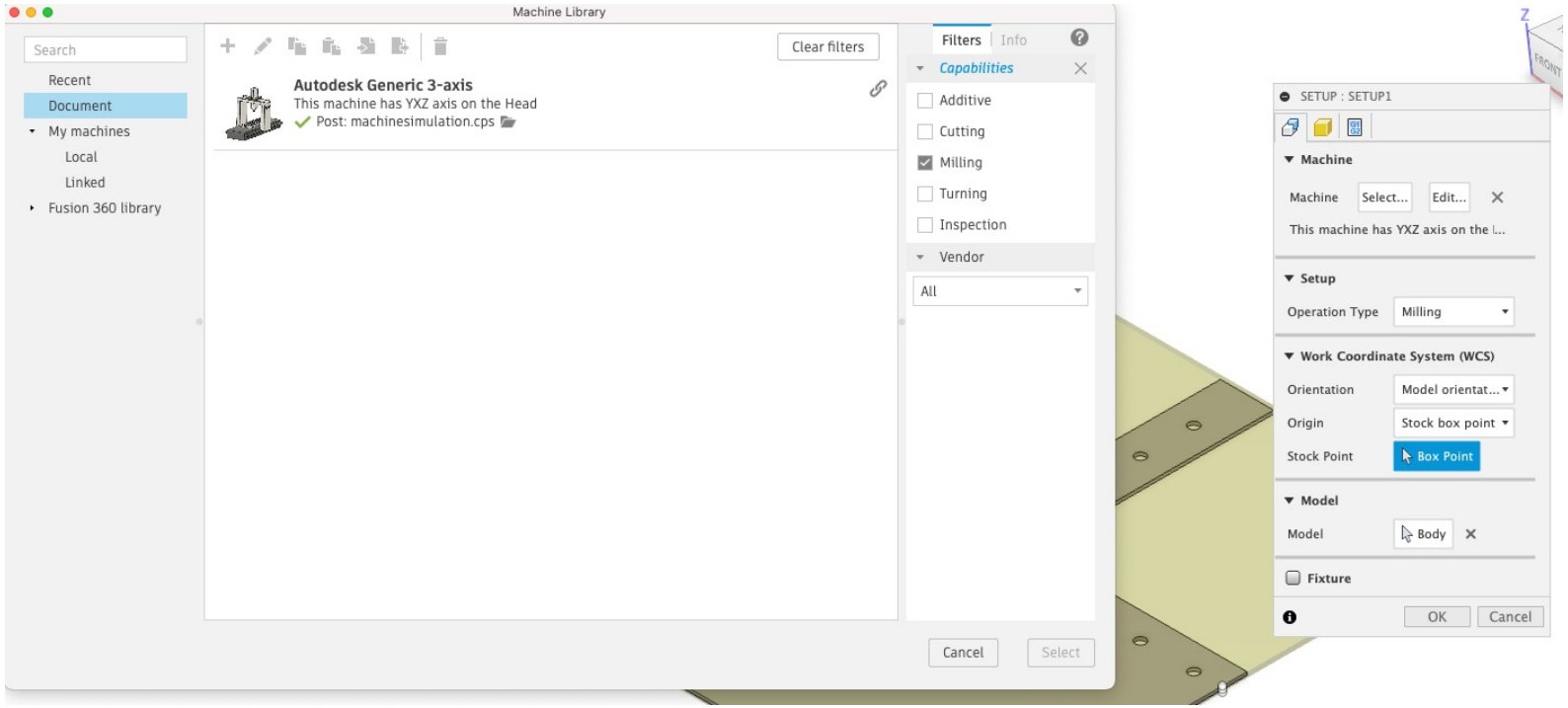


CREATE A NEW SETUP

- Navigate to “Manufacture” tab
- Select “Setup” → “New Setup”
- Will need a new setup for each face you are machining on

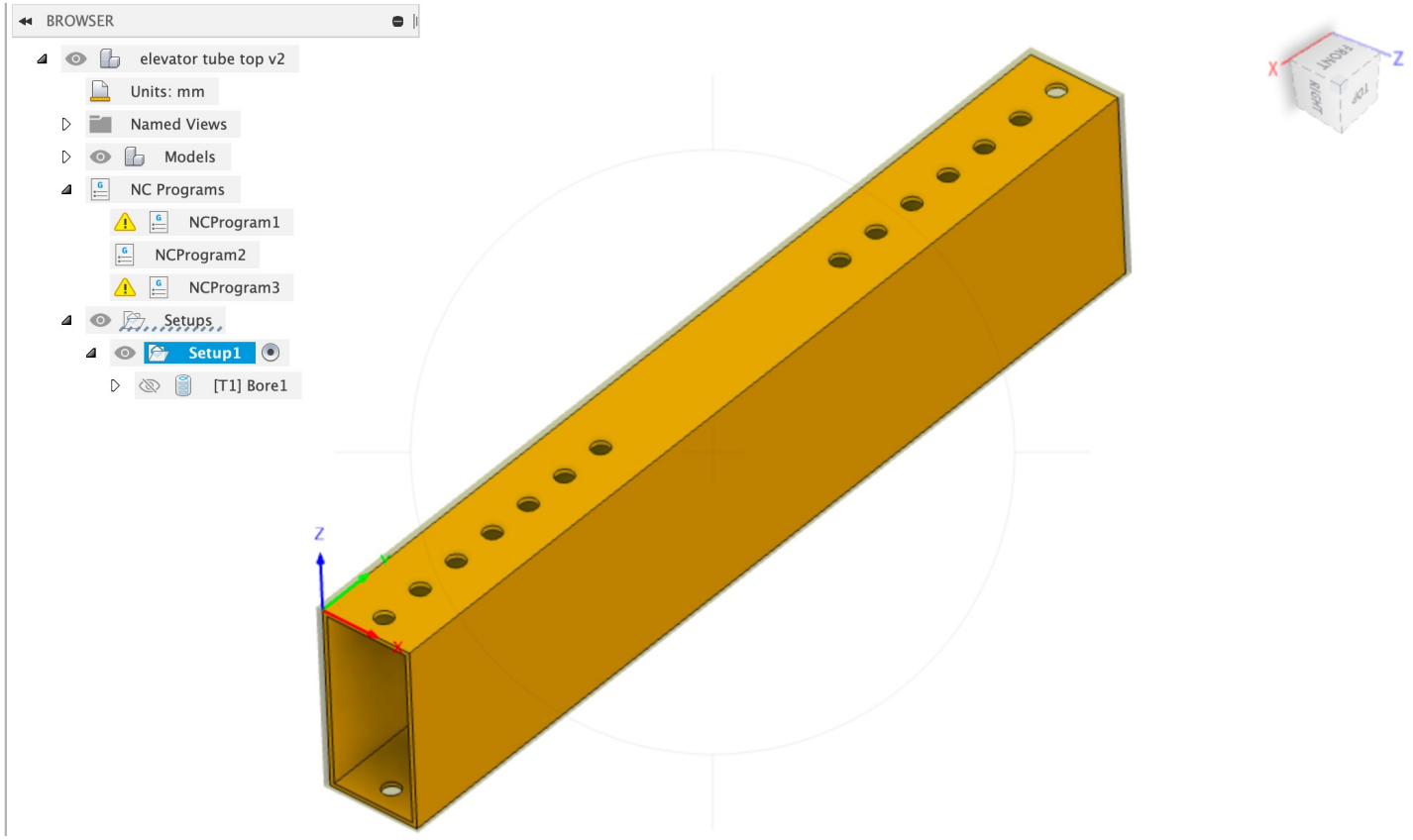
CREATE A NEW SETUP

- Select “Autodesk Generic 3-axis” as machine
  - X,Y,Z on head as that is how our router works



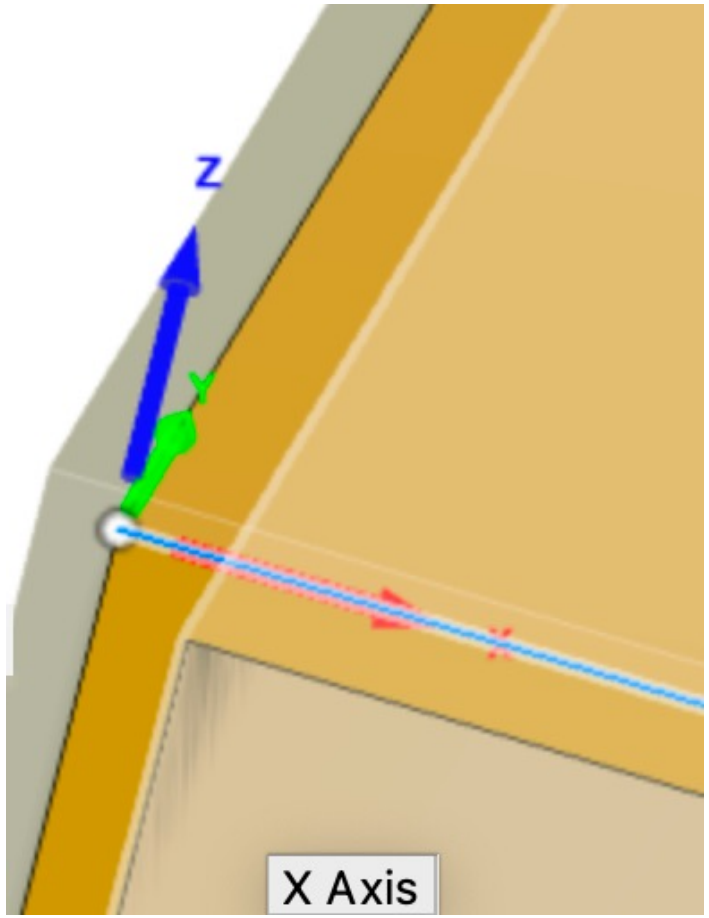
CREATE A NEW SETUP

- Need to select X & Y axes on part → Select X & Y axes
  - Ensure your X & Y match the tube jig setup you have



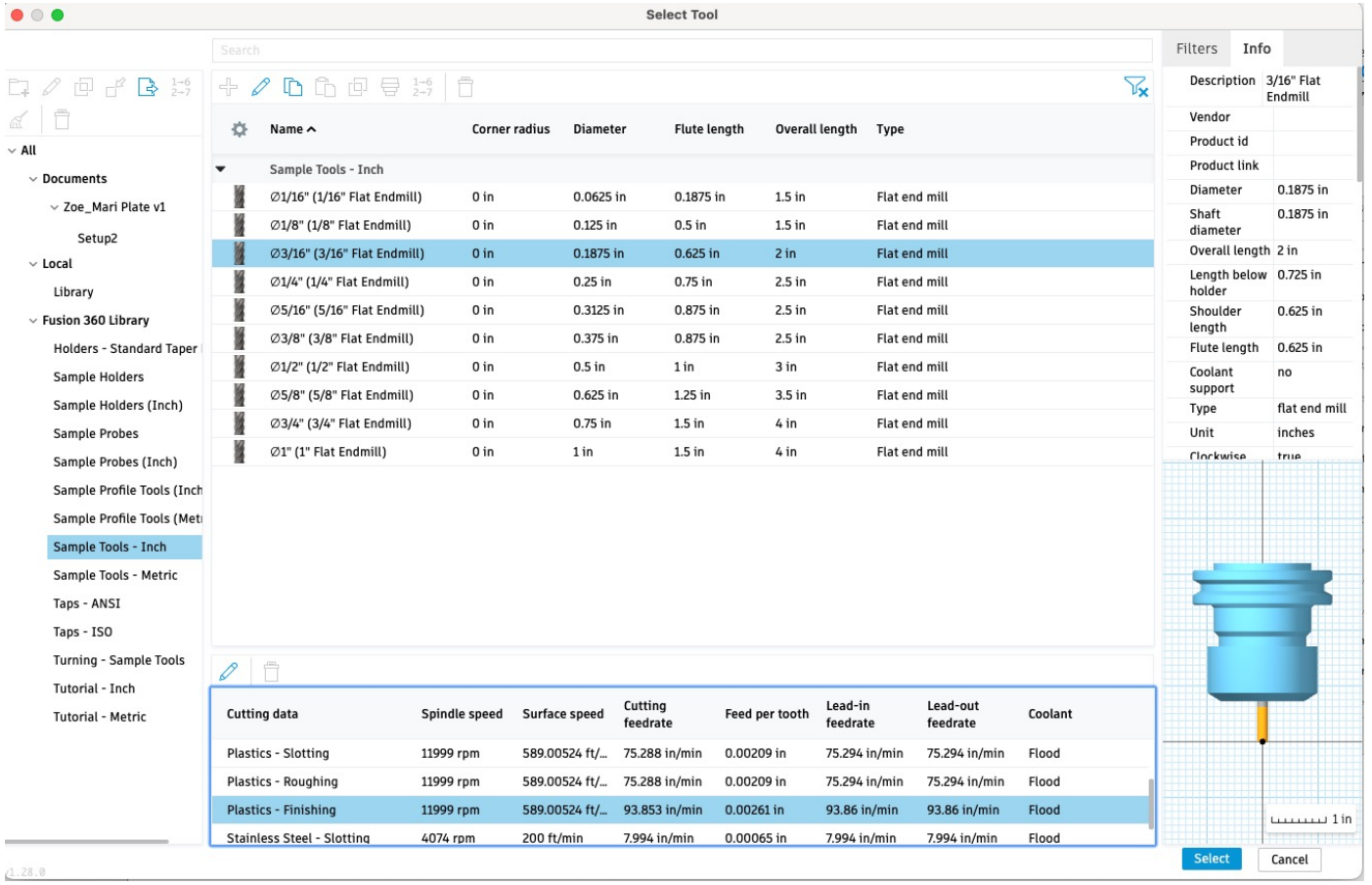
CREATE A NEW SETUP

- Select “model box point” to place origin
  - Place origin at one corner of the tube, with Z-axis going up



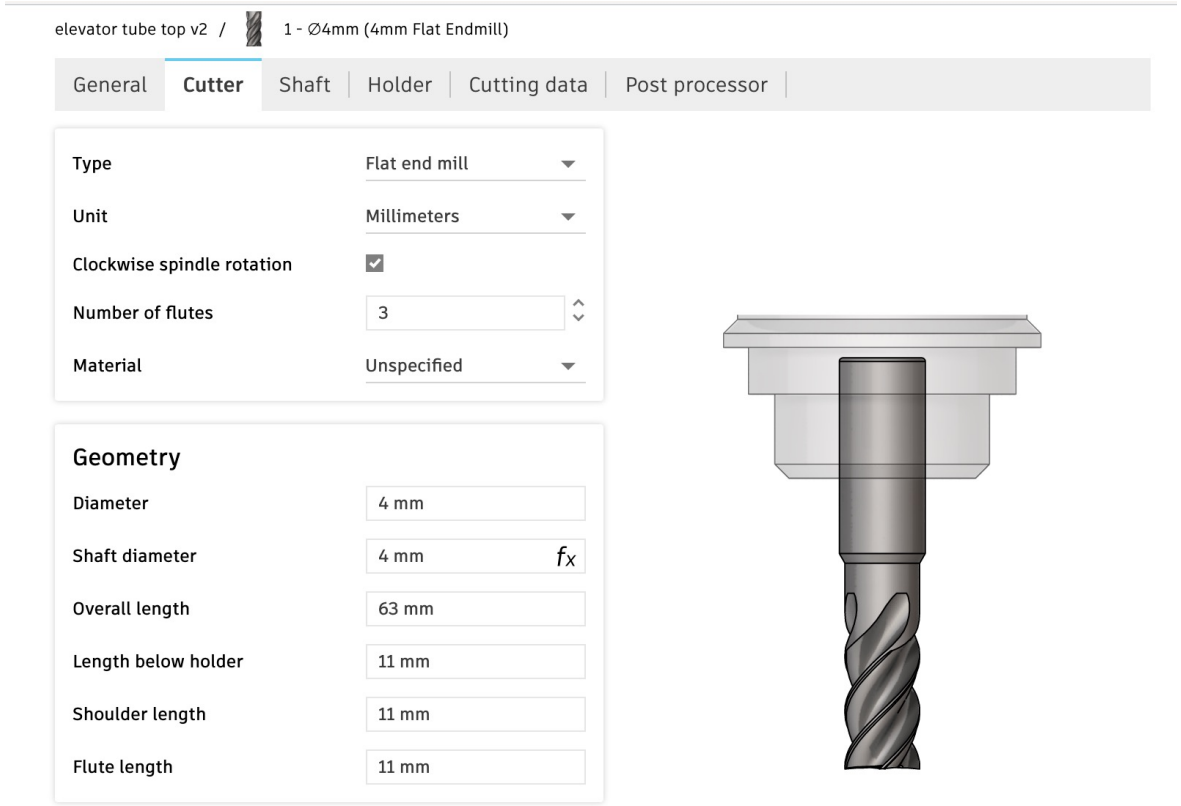
CREATE A 2D BORE OPERATION

- Select cutter from Fusion 360 Tool Library
  - Select appropriate material (plastics or aluminum finishing)
  - Speeds and feeds automatically populated for you!
    - How fast the cutter spins on different operations



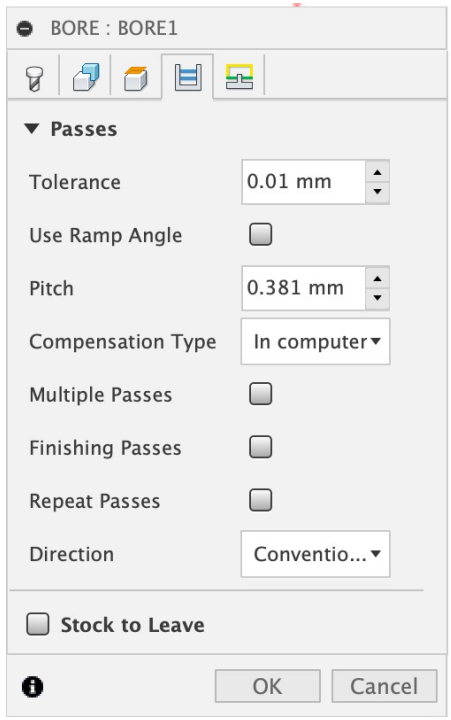
CREATE A 2D BORE OPERATION

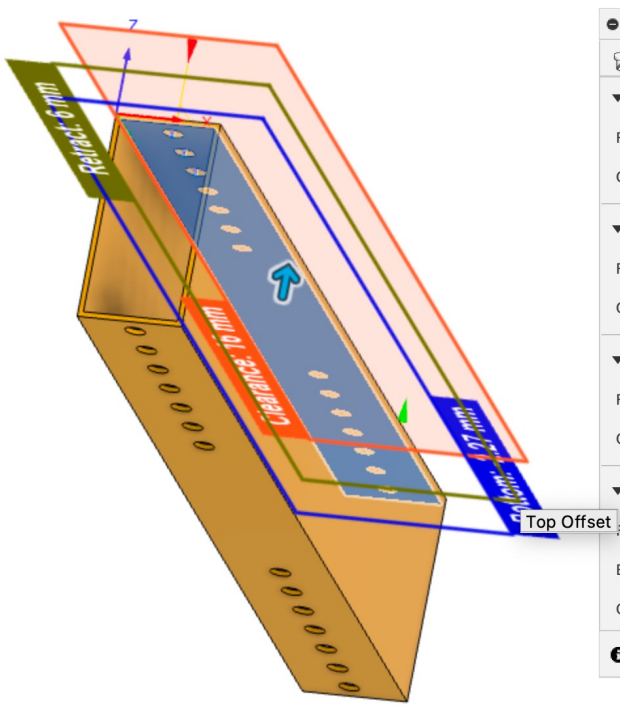
- Modify tool if needed to match your specs
  - Ensure length below holder, shoulder length and flute length match your tool
  - Change number of flutes to match your bit
  - **Fewer flute bits are better for the router**

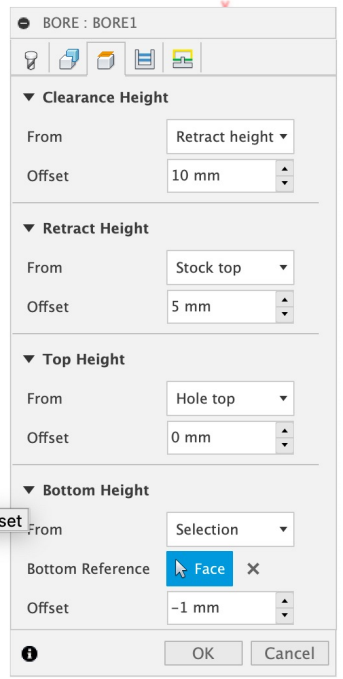


CREATE A 2D BORE OPERATION

- On third tab of contour operation, change bottom face to “Selection” and select bottom face of part
- Set the tool to go -1mm beyond bottom face – this will ensure part gets cut completely through the tube
- Set pitch to 0.381mm
- Set direction to “conventional”

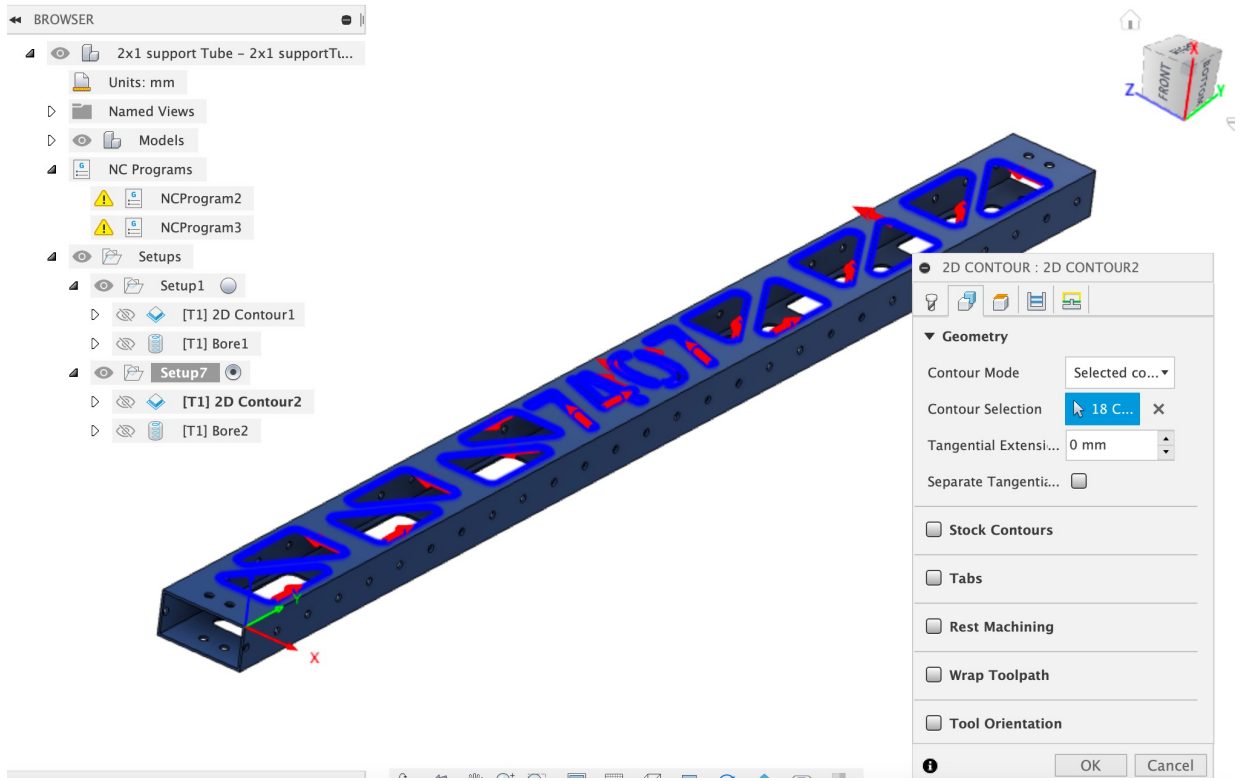






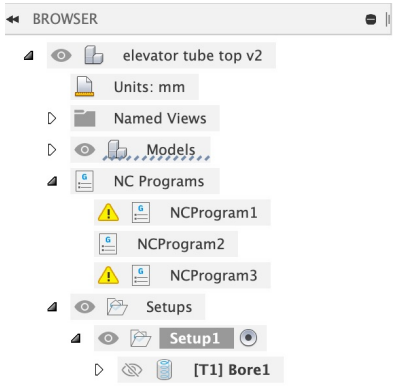
CREATE A 2D CONTOUR OPERATION

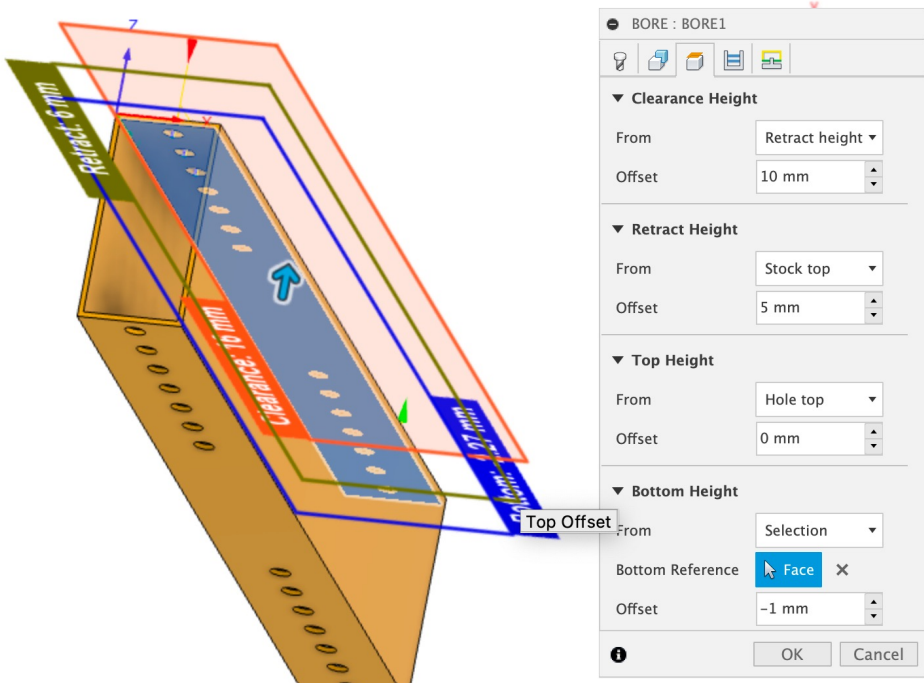
- Select 2D contour from 2D menu
- Select appropriate tool and adjust settings as needed
- Select desired contours, may need to select partial contour if making cutouts in tube



CREATE A 2D CONTOUR OPERATION

- On third tab of contour operation, change bottom face to “Selection” and select bottom face of part
- Set the tool to go -1mm beyond bottom face – this will ensure part gets cut completely through tube

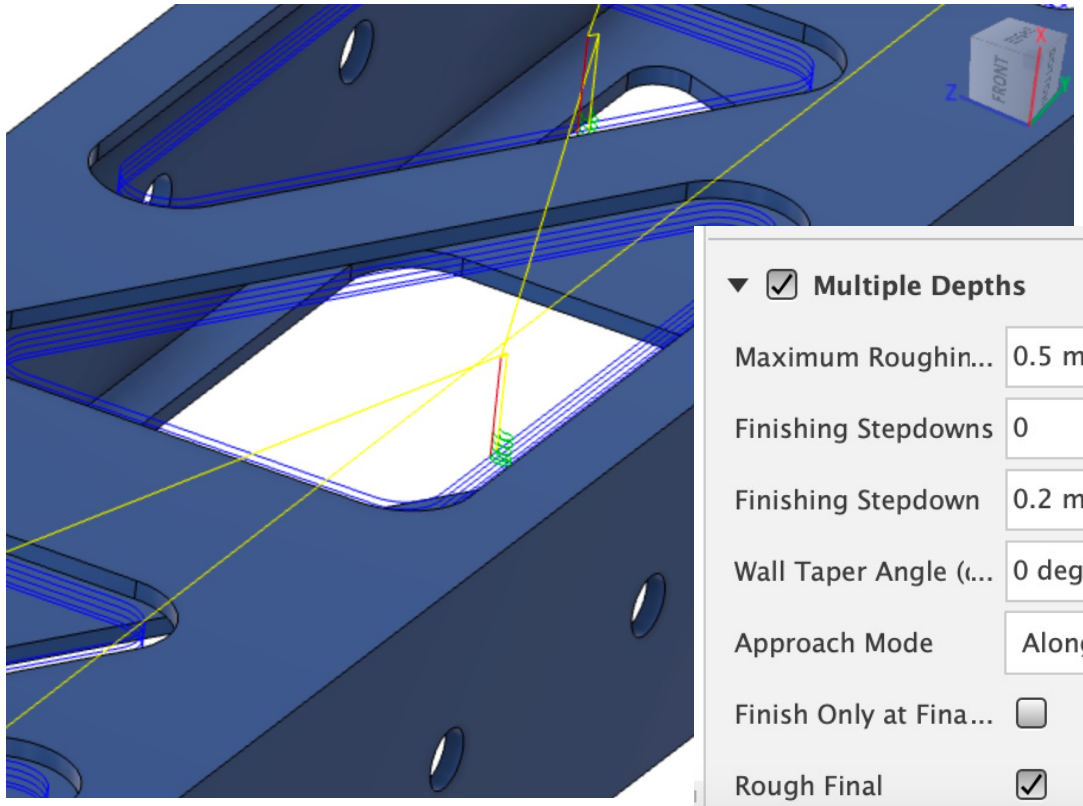




CREATE A 2D CONTOUR OPERATION

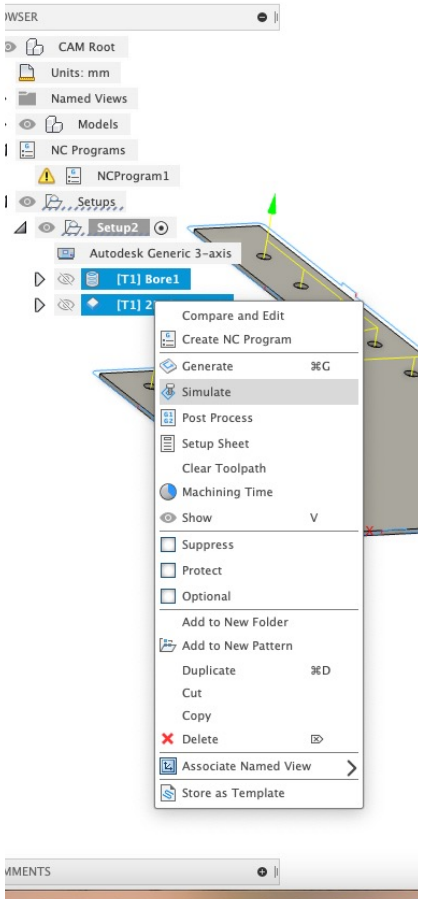
- Navigate to “Passes” Tab
- Enable “Multiple Depths”
  - “Roughing stepdown” is how much material the machine takes off in the Z-direction on each pass

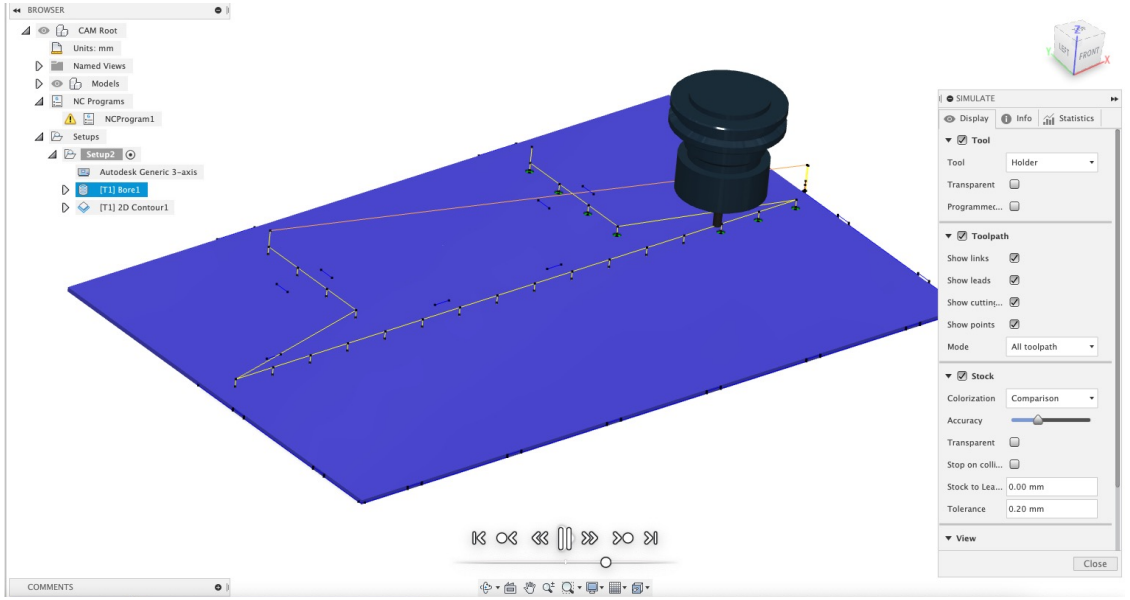
Material	1/8” 2 flute flat endmill stepdown	4mill 1 flue flute endmill stepdown	6mill 1 flute endmill stepdown
Polycarbonate	1mm	1mm	1mm
Aluminum	0.25mm- 0.4mm depending on complexity of the part	0.5mm	0.5mm



SIMULATE OPERATIONS

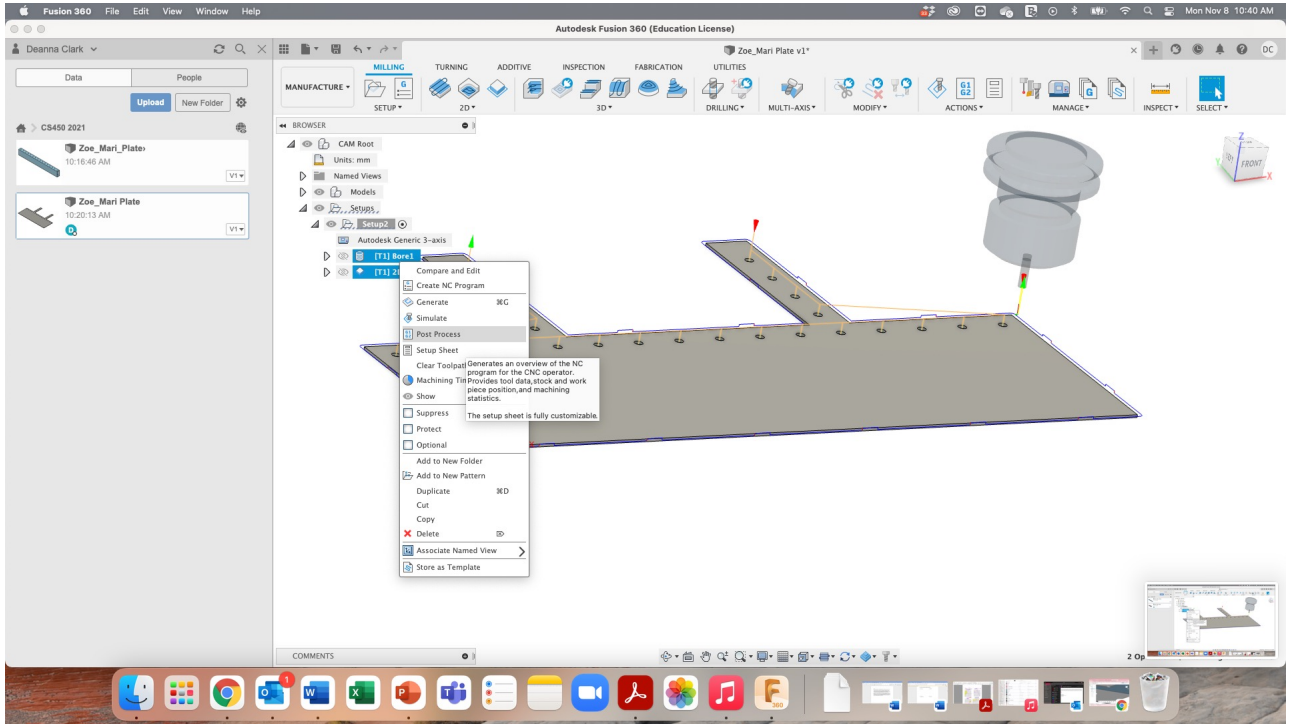
- Select both toolpaths, right click → simulate
- Press play and make sure tool path is simulated as expected





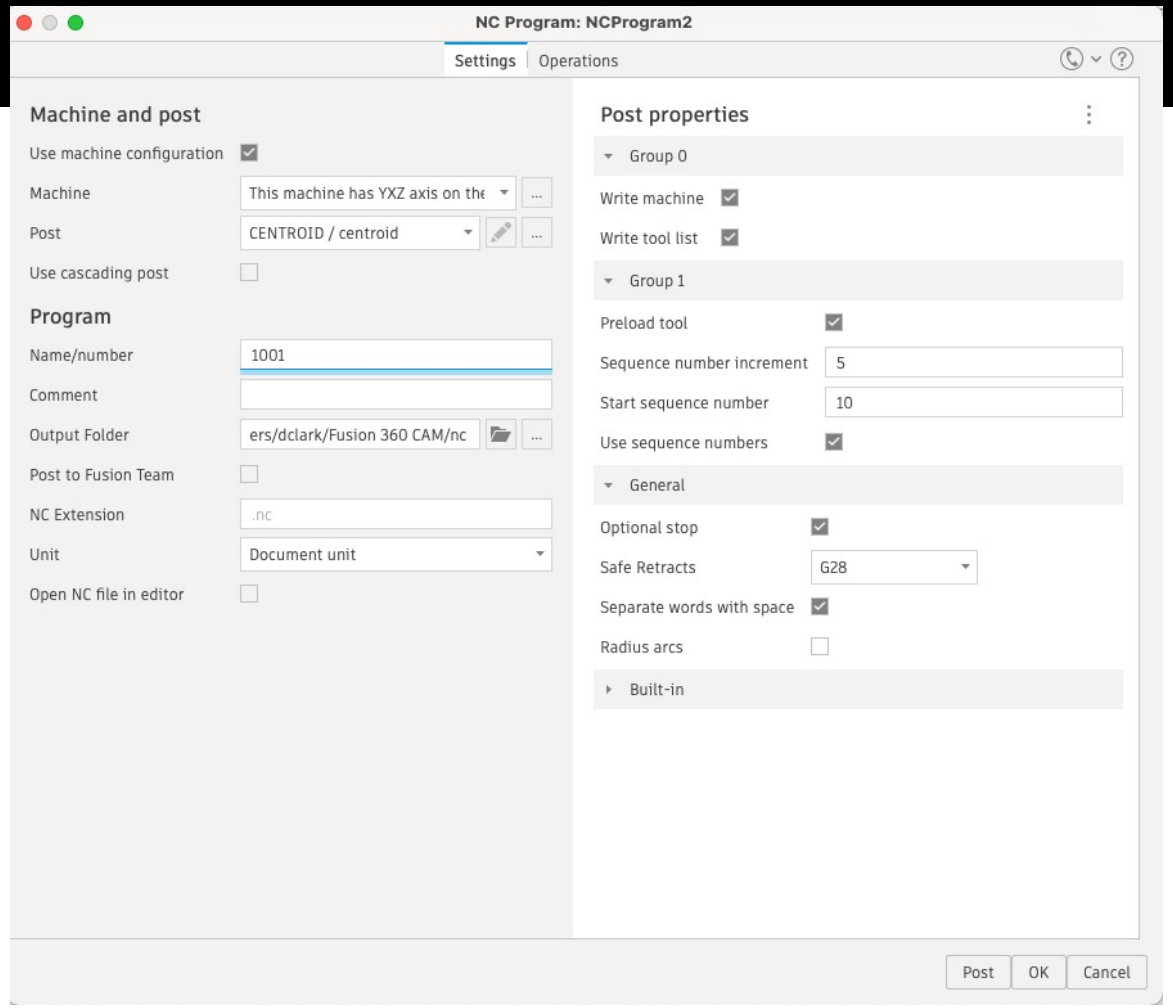
POST PROCESS CODE

- Post processing the code enables it to be run on the CNC Router
- Can export g-code files as individual files or combine into one
  - Depends on the part and how you are retaining it → if you use screws you will need to pause before 2D contour passes to install screws
- Right click toolpaths and select post-process



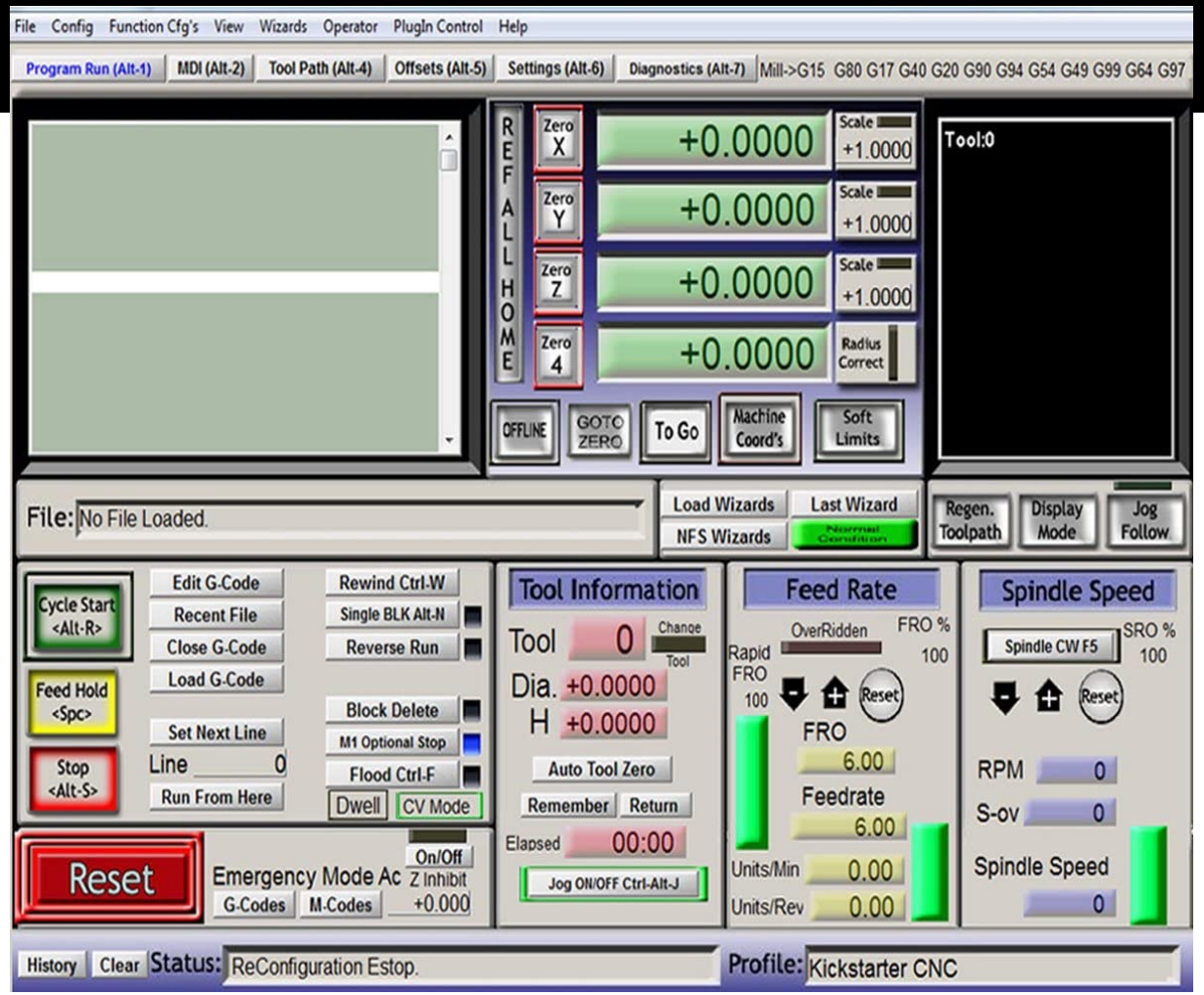
POST PROCESS CODE

- Select “CENTROID” as post type for Omio
- Need to give a numerical file name – no letters
- Select post and save file to a flash drive to open on the CNC computer



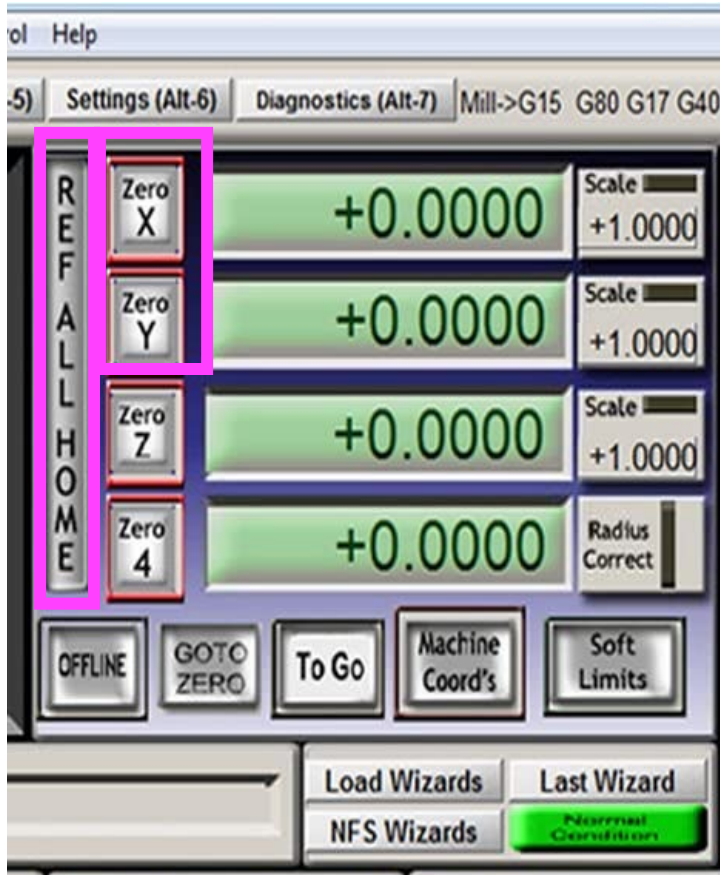
LOAD FILES INTO MACH3

- Load your g-code into Mach3 on CNC desktop computer
- An image of your tool path should appear in the software



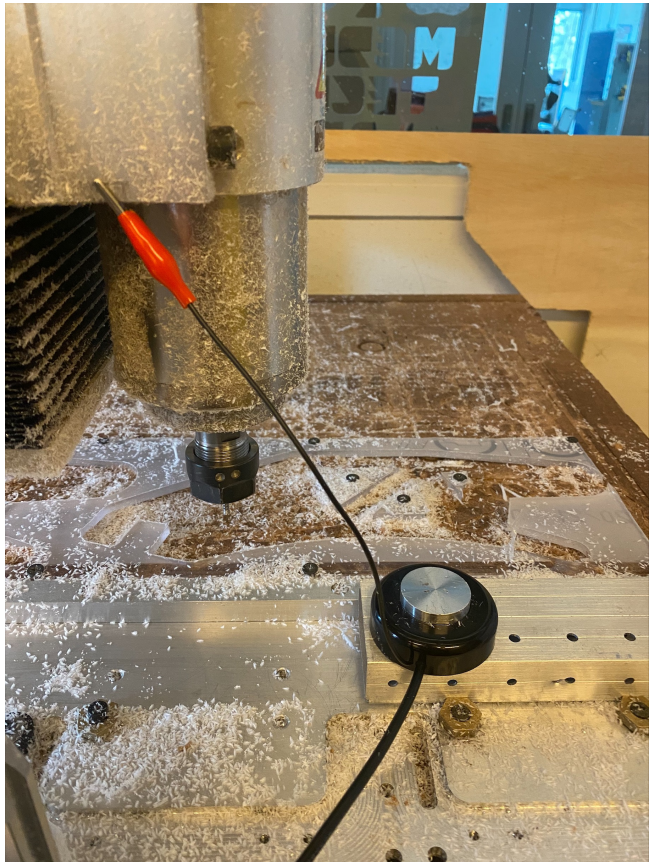
PREPARE CNC ROUTER FOR PLATE

- To begin prepping the machine for cutting, we need to make sure it has a proper 0,0,0 point to run the g-code from
- Begin by selecting “REF ALL HOME” in Mach 3
  - The router will go to its machine 0 points in X, Y and Z directions
  - Note machine 0 is different from the 0 position you set for the piece!
- Using remote jogger, move spindle to the desired 0,0 (X,Y) location. Once there select ”ZERO X” and “ZERO Y”



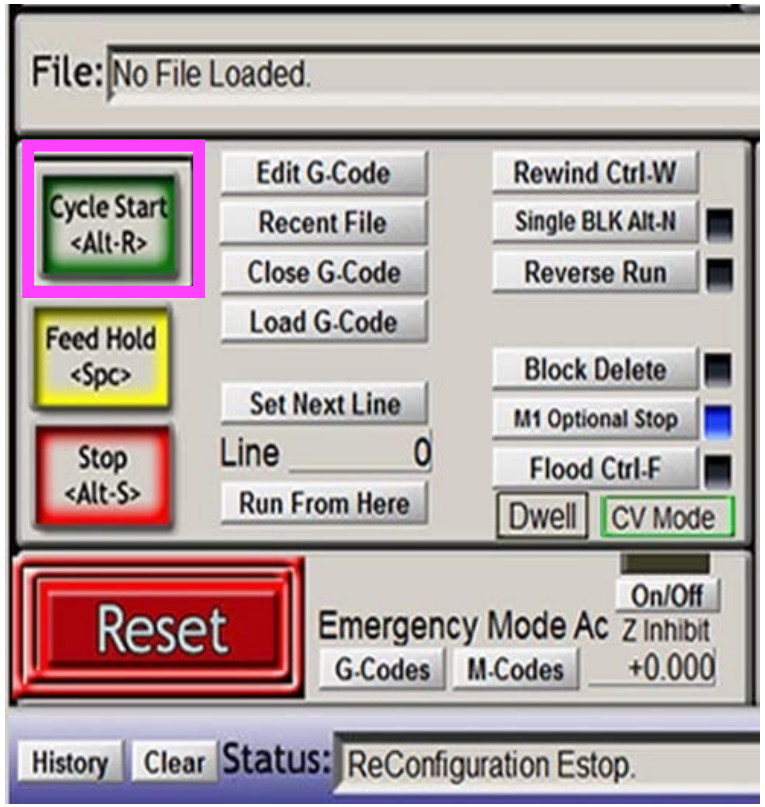
PREPARE CNC ROUTER FOR PLATE

- Move router bit over where you will be cutting to prepare to zero Z
  - Connect touch pad alligator clip to side of spindle and place pad directly underneath the bit; hold in place.
  - Have a second person press ”auto tool zero” in Mach3; bit will move down until circuit is completed by the metal bit touching the plate
  - Machine will automatically set Z-zero for you, ensure Z-offset for touch pad is accurate in Mach 3 or this operation will not work



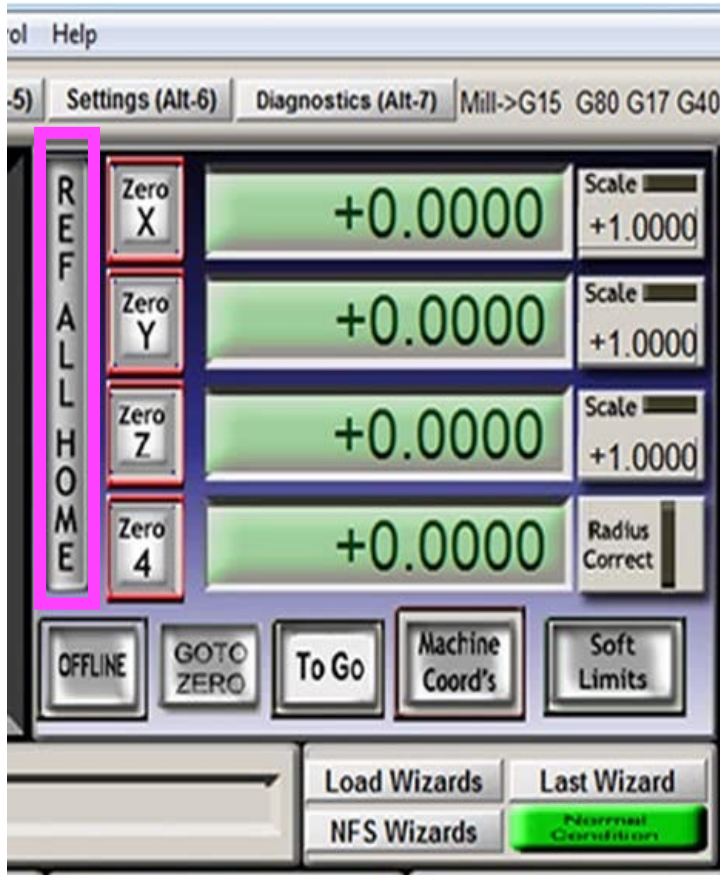
PREPARE CNC ROUTER FOR PLATE

- Ensure correct tool is in router, and dust collection system running (follow toolpaths with vacuum by hand if no chip clearing is present on your machine
  - Chip loading with aluminum can compromise quality of your parts and break or wear out bits quickly
- Once you are zeroed and the machine is ready to safely run, start your program by pressing “cycle start”!



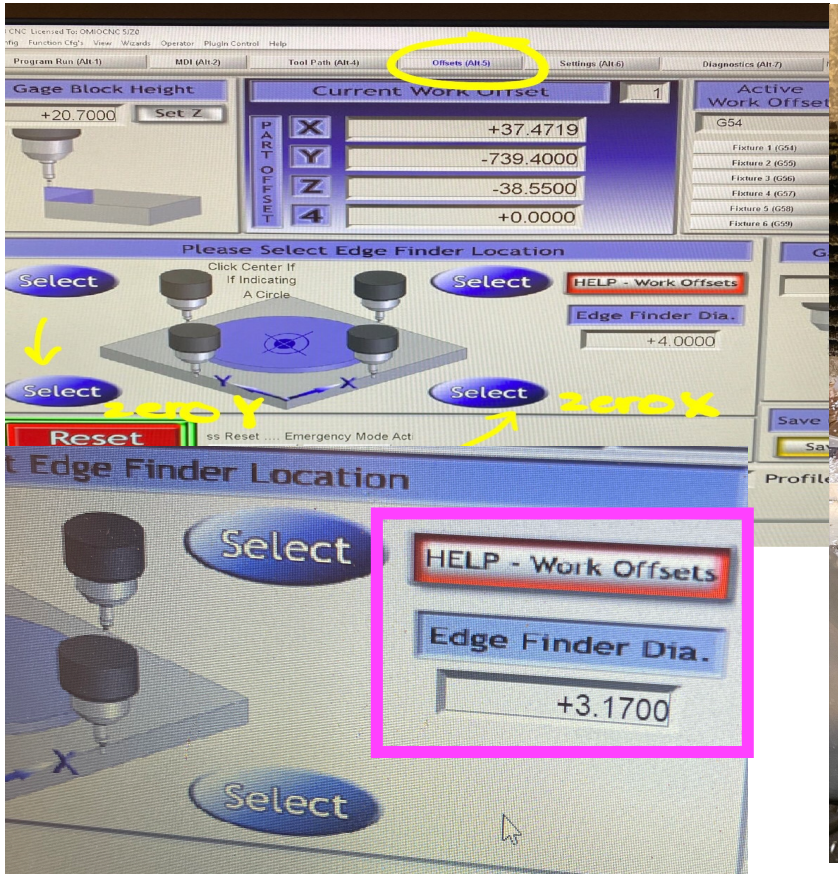
PREPARE CNC ROUTER FOR TUBE

- To begin prepping the machine for cutting, we need to make sure it has a proper 0,0,0 point to run the g-code from
- FRC7407 has written special code to make zeroing tubes in Mach 3 a breeze! Please see next slides for instructions on importing the code.
- Begin by selecting “REF ALL HOME” in Mach 3
  - The router will go to its machine 0 points in X, Y and Z directions
  - Note machine 0 is different from the 0 position you set for the piece!



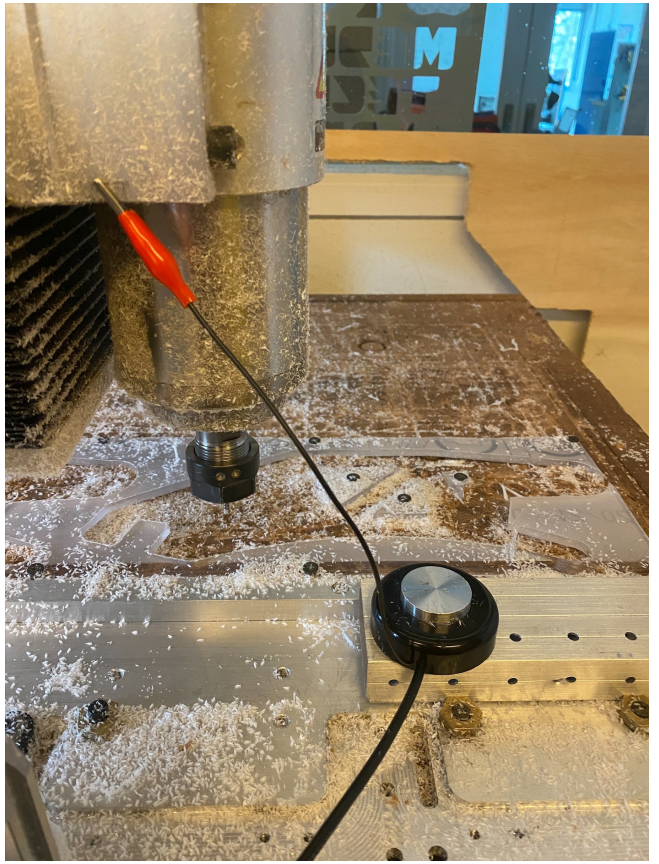
PREPARE CNC ROUTER FOR TUBE

- Using alligator clips, clip one end to the touch pad and the other end onto the tube. Take the touch pad alligator clip and secure it to the side of the spindle
  - Move the bit to the side of the y-axis where you want your y-zero to be. Navigate to Offsets tab in Mach 3 and select “XYZ”
  - The bit should move towards the Y-axis until the circuit is completed when it touches the metal. This will automatically set y-zero for you.
  - Repeat the same procedure for x
  - MAKE SURE YOUR EDGE FINDER DIAMETER MATCHES YOUR BIT DIAMETER!



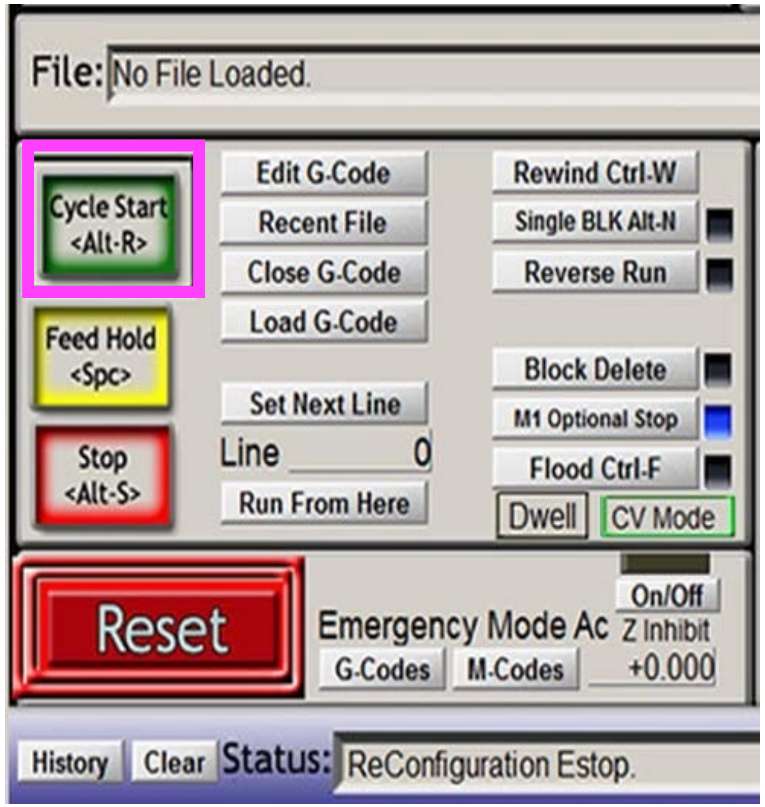
PREPARE CNC ROUTER FOR TUBE

- Move router bit over where you will be cutting to prepare to zero Z
  - Connect touch pad alligator clip to side of spindle and place pad directly underneath the bit; hold in place.
  - Have a second person press ”auto tool zero” in Mach3; bit will move down until circuit is completed by the metal bit touching the plate
  - Machine will automatically set Z-zero for you, ensure Z-offset for touch pad is accurate in Mach 3 or this operation will not work



PREPARE CNC ROUTER FOR TUBE

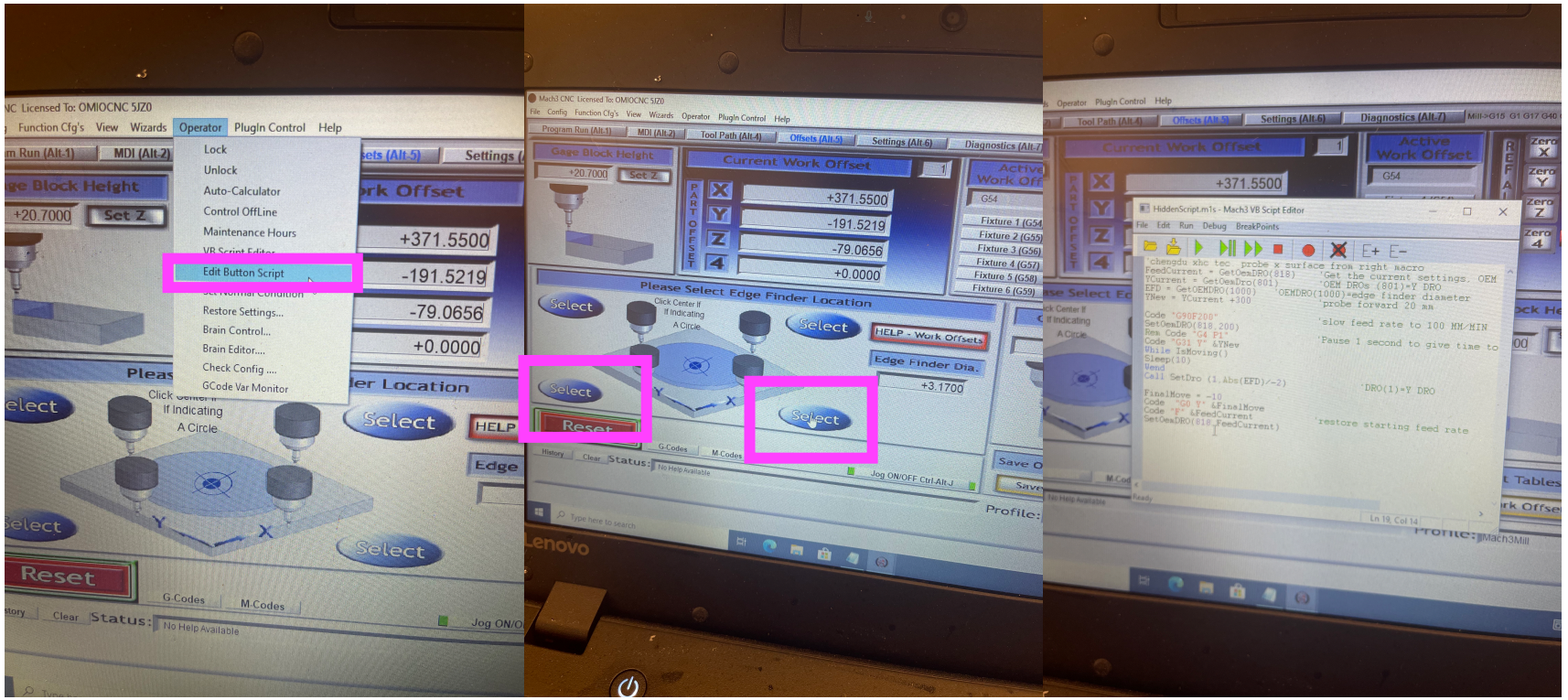
- Ensure correct tool is in router, and dust collection system running (follow toolpaths with vacuum by hand if no chip clearing is present on your machine
  - Chip loading with aluminum can compromise quality of your parts and break or wear out bits quickly
- Once you are zeroed and the machine is ready to safely run, start your program by pressing “cycle start”!



MACH3 TUBE ZEROING CODE

Google Drive Folder with Zeroing Scripts

- To load these scripts, navigate to Operator dropdown menu and select “Edit Button Script”
- Upload appropriate script for each axis into the following:
  - X&Y zeroing buttons on offsets tab
  - Auto too zero button on program run tab



THINGS TO LOOK OUT FOR

- Make sure your tool path doesn't hit any screw heads → bit will likely break
- Make sure your parts are secured to the router via screws in the spoil board, inside the tube jig or clamped down in a secure fashion
- If you try to cut too much material at once or use too fast or too slow speeds based on material, you may damage the bit or your part
- It takes lots of practice to make good g-code, so don't worry if you make mistake –> that is how we learn!
- Always wear safety glasses and watch cuts as they happen → be ready to emergency stop if something goes wrong
- Always CNC with a buddy!

QUESTIONS OR COMMENTS?

Please feel free to email robotics@choate.edu with any questions you have!