

<b>Workshop/Lesson Name:</b>	Tormach CNC Workshop
<b>Objective:</b>	Learn to properly and safely operate the machine. Create a simple project.
<b>Duration:</b>	2 HR
<b>Required Materials/Software:</b>	Fusion 360, Tormach Post Processor, PathPILOT

**Workshop/Lesson Structure:** Teaching steps and main points to cover; write out the entire workshop start to finish here.

Helpful YT Video: [📺 Tormach xsTECH CNC Router Complete Workflow Using Fusion 360: F...](#)

**Account Creation:** If you have not done so already, create an Autodesk account using your University of Wyoming email. (<https://www.autodesk.com/> ---->sign in ----> create account)  
It's important you use your UW email as this will enable you to access a fusion 360 education License for free, otherwise you will have to pay a high monthly rate for access to the software.

### **Setting up fusion 360 Libraries:**

To ensure everybody is working with the same tools to enable easier troubleshooting we will ensure everybody is using the same Fusion360 machine libraries. These libraries give Fusion 360 the proper tools to prepare files for machining on the Tormach XS CNC.

- 1) **First Download the Tormach XS CNC Tool Library and Machine file.**
  - a) **Tool Library: ([shorturl.at/otKV8](http://shorturl.at/otKV8))**



b) Machine File: ([shorturl.at/BGKOU](http://shorturl.at/BGKOU))



- 2) Open Fusion 360
- 3) In the top left in the toolbar there will be a large button that says “Design”, after clicking on it there will be a drop down menu. Select “Manufacturing”.
- 4) You’ll notice the toolbar now has multiple tabs: Milling, Turning, Additive,...etc
- 5) Select Milling (if it is not already selected) then under “Manage”, select “Tool library
- 6) In the left hand side of the window that opens is the tool library manager. There will be a section titled local. Right click on that title and in the drop menu select “import libraries”
- 7) Navigate to the “xsTECH FUSION TOOL LIBRARY V2” file and select it. This will add the CNC tools we have at the maker space to fusion 360 so that when you set up a file for machining it's only with tools that are available.
- 8) Close the tool library window, then select the icon next to “Tool Library” Labeled “Machine Library”.
- 9) Under “my machines” click on “Local”. In the middle window is the following options.



Click the circled option labeled “import”, then select the “xsTECH Router Fusion360” machine file.

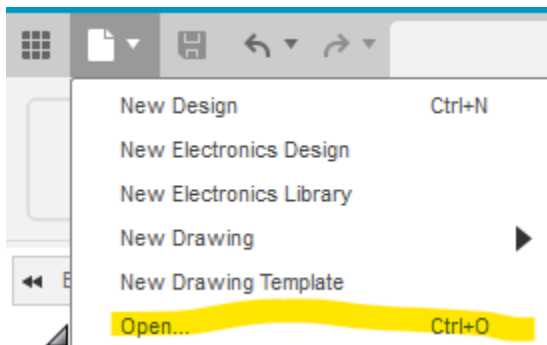
10) Close the menu. You are now prepared to move onto “File Prep”

### **File Prep:**

- 1) For this workshop we will be working with a Demo file, Download the Demo File using the following link: [shorturl.at/qwBEG](https://shorturl.at/qwBEG)



- 2) Select the File Icon, then open. In File explorer open the demo File you just downloaded.



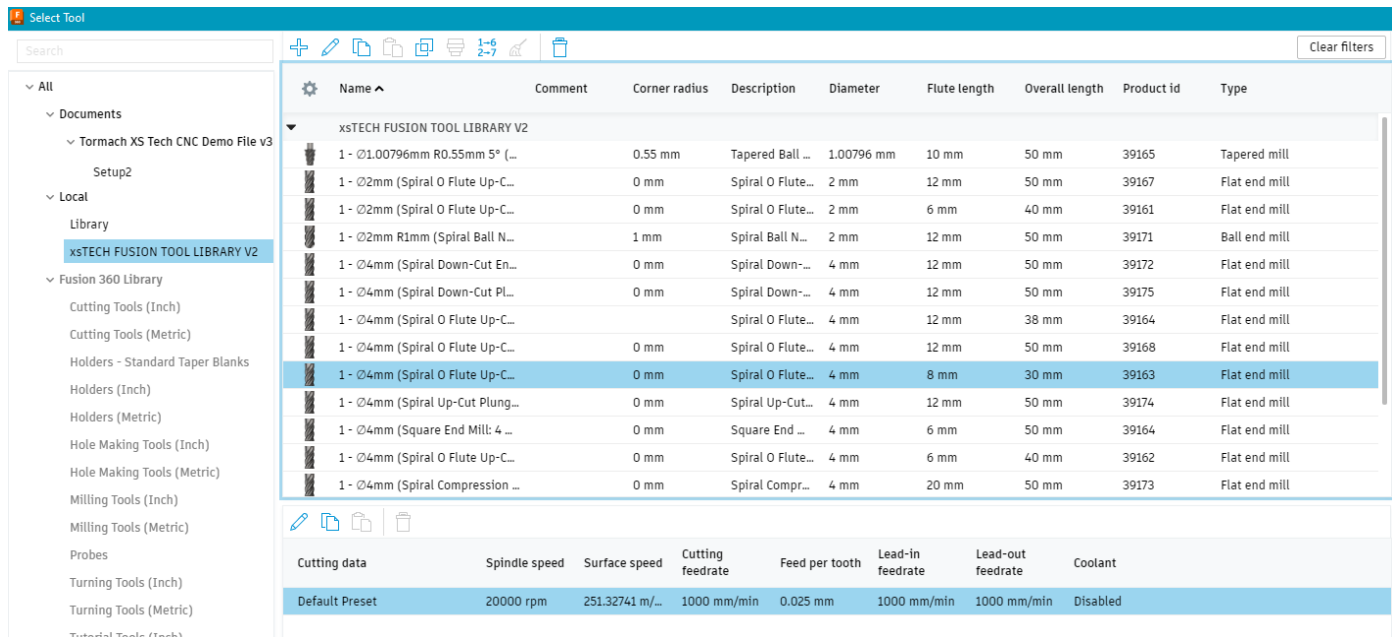
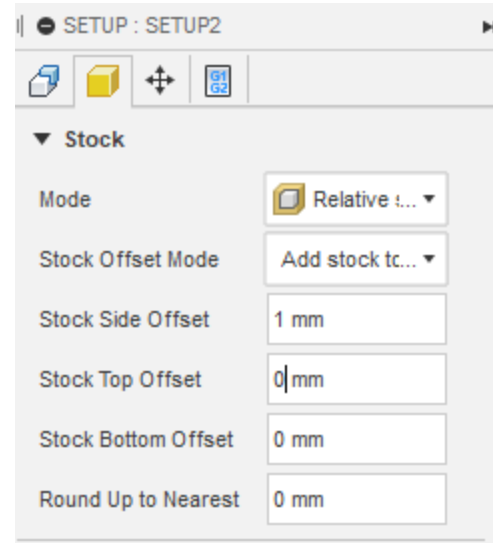
- 3) If you have not already, select the “Manufacturing” workspace like you did in the “setting up Fusion 360 libraries” section.
- 4) Next under Setup in the milling tab, select “Setup”, A menu will open where we will adjust some setting for milling
- 5) The first tab under “Machine” we will press select and then select the “Tormach xsTech CNC Router”

6) The next tab with the yellow cube defines our material we are cutting from or Stock. Since we are going to just mill a pocket out of our material we are going to adjust our stock to have only a side Offset.

7) Your First CNC Operation: Roughing the pocket:  
 a) Under the 3D tab select the “3D” drop down second option titled “Pocket Clearing”

b) A box very similar to the setup box will open with a bunch of settings that we can adjust for various materials and operations. This menu is incredibly adjustable but for now we are gonna work with the basics.

c) Under tool there is a select button, click this then navigate to the tool library that we imported at the beginning of the workshop

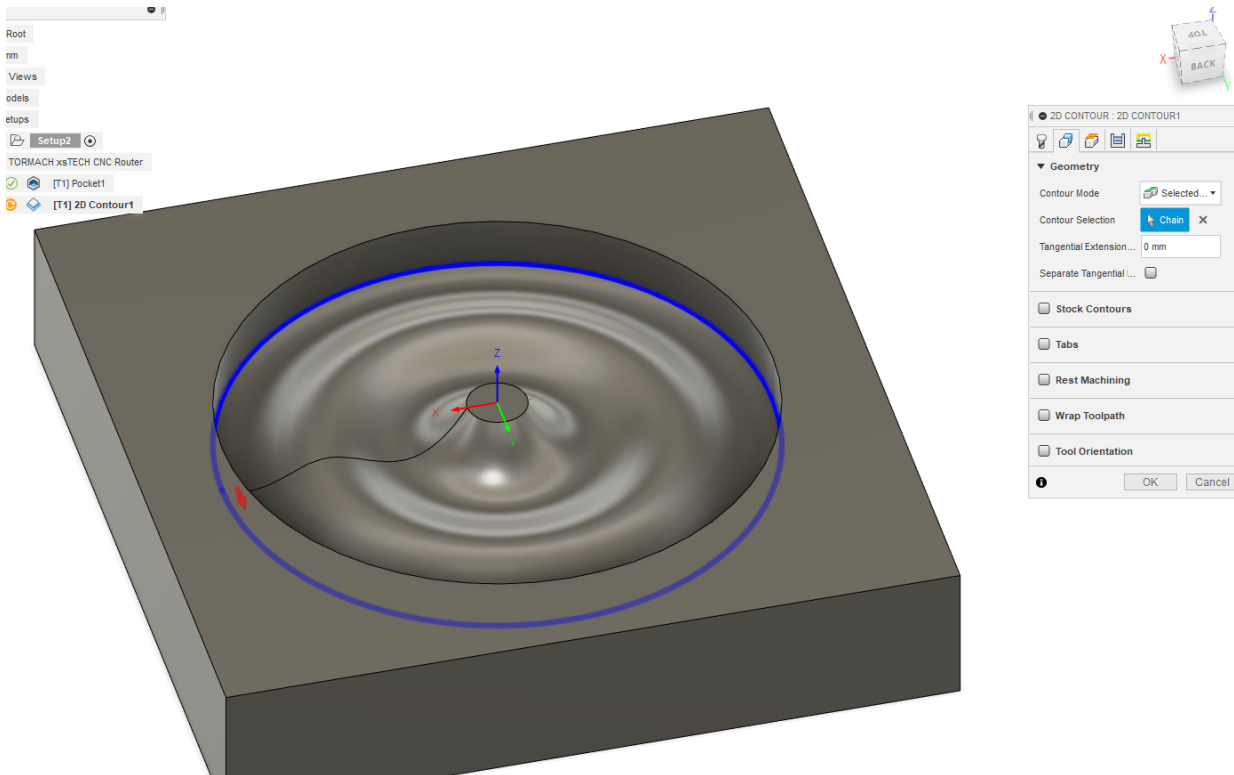


d) We are gonna use a basic 4mm diameter Flute Up-cut End Mill (39163) for roughing out the pocket in the block. The rest of the settings for this operation we are going to leave default

8) For our Second operation we are going to do a 2D contour, which outlines the ring of the pocket.

a) Under the 2D tab select “2D Contour”

b) You will be prompted with the same box and settings as the previous operation. This time we will go to the Geometry tab and make the following selection

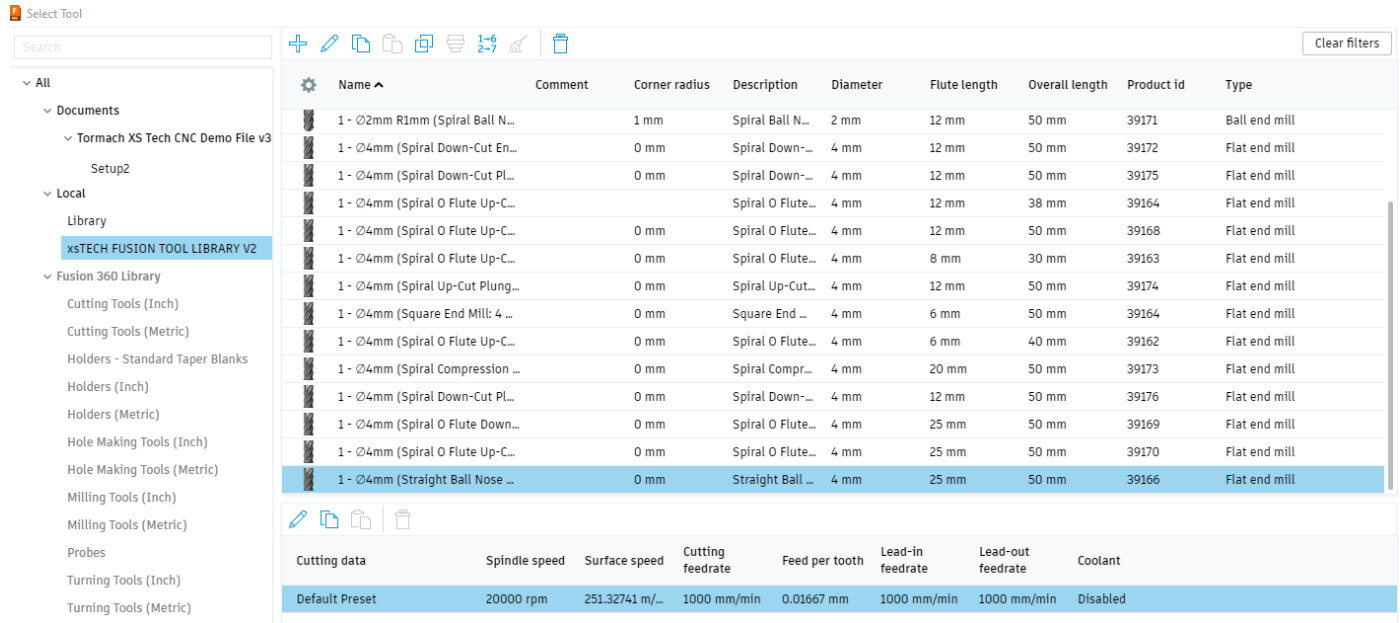


c) The rest of the settings we will leave default

9) Now we will do our final Finishing Pass. The previous 2 steps roughed out the main Geometry of our model but as you can probably see in the Green preview the model still doesn't quite look like the design. For this we will have to use a finer Ball nose bit to get a really clean finish.

a) Under the 3D tab click the "3D" drop down menu and select "Spiral"

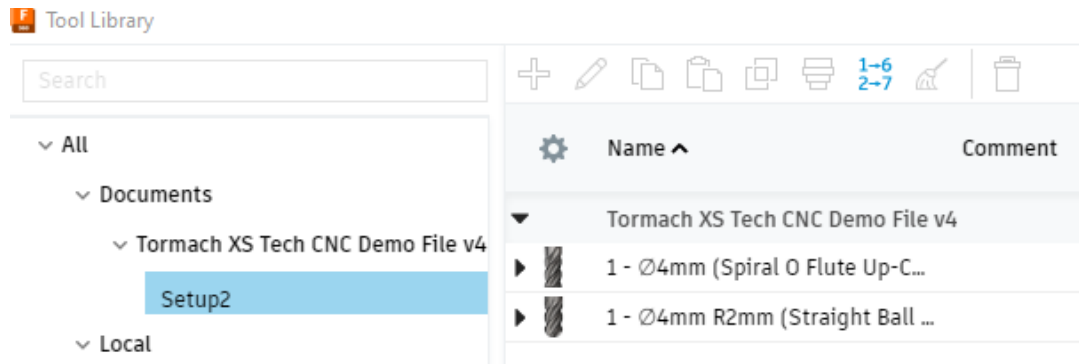
b) This will open the same setup box as before but this time we need to change our tool to a 4mm Straight Ball Nose End mill (39166) as seen below



c) We'll click select and then go over to the passes tab. Here we will go to step over and type 0.5 mm, this is the resolution of our passes which determines how clean the finish is but also how long this operation will take. After this you can click save.

10) We have one last step before we can transfer our machining file to the CNC and that is to number our tools. As of right now the CNC thinks we are using the same bit for all 3 operations, but we are in fact using 2 different bits. The flat bit first then the ball nose bit after.

a) To correct this open the Tool library tab. You will see the following



- b) In the top right you will see the number button highlighted blue, click this then click renumber
- c) Verify the bits are labeled in order they are used (flat is 1, ball is 2) then click save

### **General Workflow:**

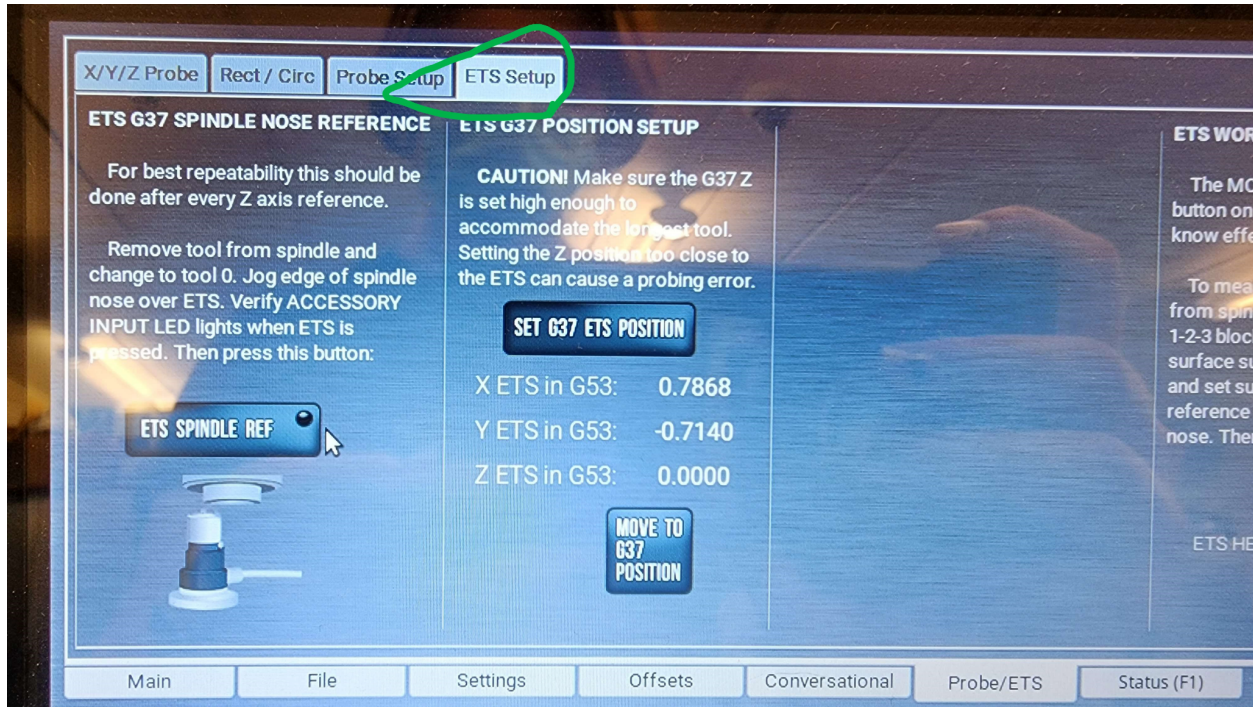
- 1) Make sure machine is out of e-stop (*spin the big red emergency stop button*)
- 2) Power on machine *using the power switch located on the back left hand side of the machine*
- 3) Clear the bed of mounts or material if applicable
- 4) Click reset
- 5) Reference all 3 axis, xyz, Remove Bit if one is in machine (utilize the 2 wrenches, ask a technician for assistance if this is your first time using the machine)
- 6) Set Tool number to Tool 0
- 7) Use shuttle to move axis



8) Set ETS

a) Tap the Probe ETS tab

b) Tap probe ETS Setup in the top left toolbar



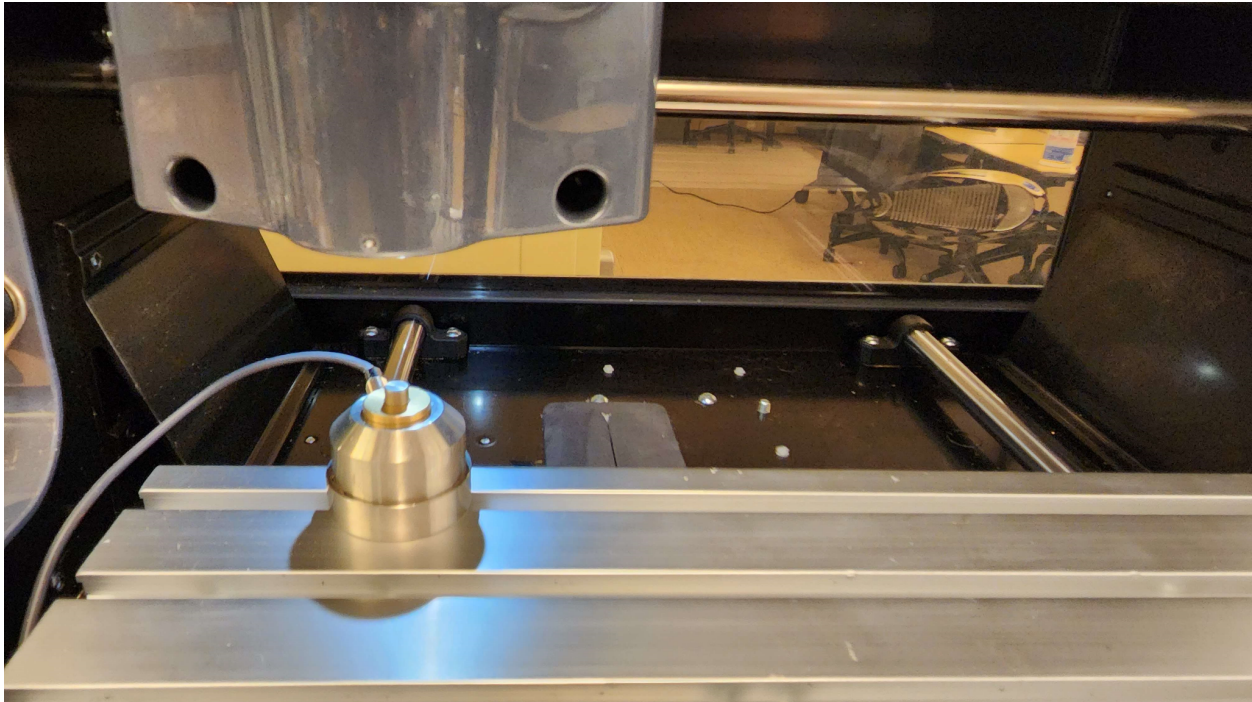
c) Move Spindle over the ETS and then select ets spindle Ref





**d) Install the 1st bit used in the Fusion 360 machine program we just created in the previous section. We will be using the 39163 flat end bit first, look through the bit catalog till you find the correct bit. MAKE SURE TO LEAVE THE EMPTY BIT CASING OUT SO WHEN YOU ARE DONE YOU CAN PUT THE BIT AWAY IN THE CORRECT CONTAINER**

**e) position the spindle over the ETS then raise the Z axis to its maximum**

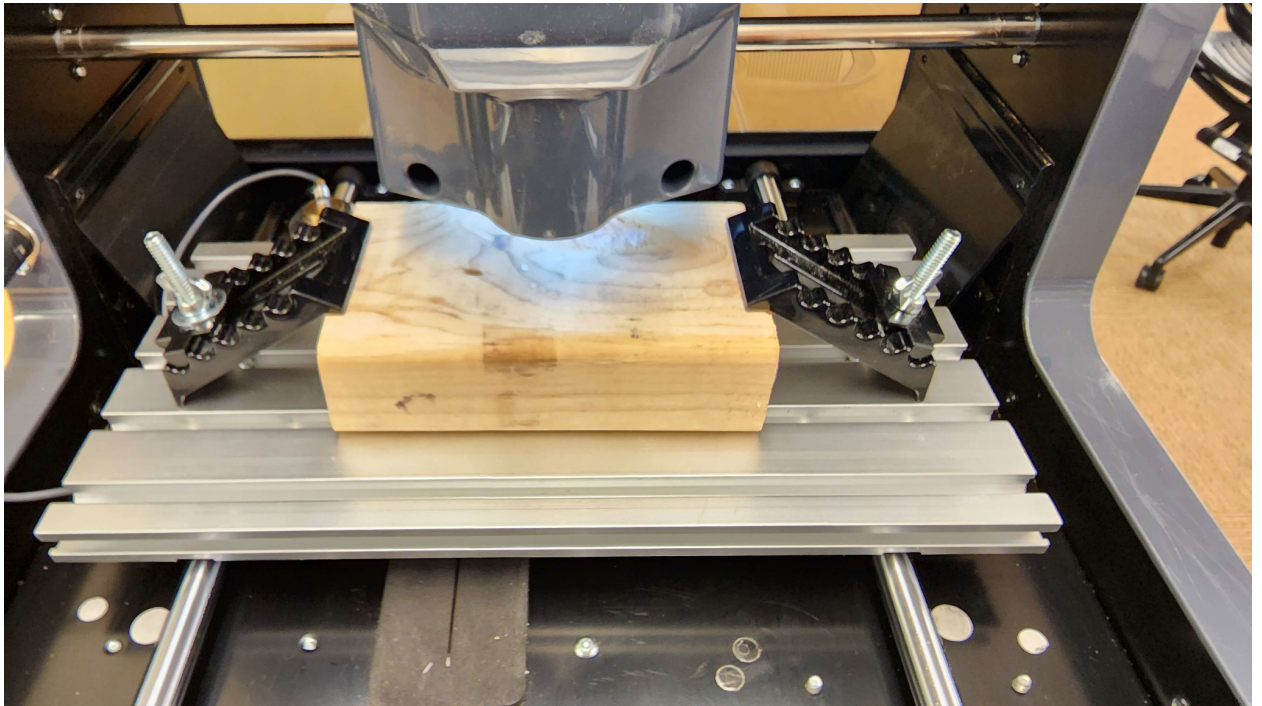


**f) in the main menu change the tool number to 1**

**9) Set Tool offsets**

- a) Navigate to offsets**
- b) Click tool 1 on the right side of the screen and with the current tool installed and the head over the ETS click “move and set tool length”**

### 10) Secure Stock on to bed securely Using the bolt-Wing Nut clamps



### 11) Set origin point

- ~ Use marker or pencil to mark origin on stock (usually the center)
- ~ Use shuttle to navigate center of bit on origin point
- ~ Lower the bit onto the origin point over a piece of paper, making sure the paper can still slide with friction
- ~ Zero your xyz

### 12) Load the NC program

- ~ Find on USB and copy
- ~ Click file and load gcode
- ~ Make sure program looks safe and correct on viewport
- ~ Use ETS to set T1 length
- ~ Make sure tool changes are in program and are accurate

### 13) Set tool length during tool change [Stop the program at tool change and ETS] To start again, select the tool change and right click and run [preparations optional]