CNC Mill Computer Aided Manufacturing (CAM) Training

BJ Kravets

January 21, 2026

This document – which is meant to accompany the in-person CNC Mill CAM Training – will go over everything necessary to use CAM on the Makerspace's Tormach PCNC 770 Mill. It will go over importing the tool library and vise 3D Models from the Makerspace website, setting up your Manufacture workspace in Fusion360, programing your toolpaths, and exporting and running the .nc file to the mill.

Contents

1. Table of Tools 4
2. Downloading Necessary Files 5
   - 2.1. Tool Library and Vise 3D File 5
   - 2.2. Importing into Fusion360 5
     - 2.2.1. .tool File 5
     - 2.2.2. Vise .f3d File 6
   - 2.3. Downloading the training 3D Model. 8
3. Inserting the Parametric Vise & Creating our Stock 9
   - 3.1. Parametric Functionality 9
   - 3.2. Mating the Vise to the Part 10
   - 3.3. Insert Second Vise 10
   - 3.4. Mating the Second Vise 11
4. Creating our First Setup and Its Toolpaths 12
   - 4.1. Setup 1 (Op1) 12
   - 4.2. Tabs in each Toolpath 15
   - 4.3. 2D Face 16
   - 4.4. 2D Pocket (Perimeter and Interior) 18
     - 4.4.1. Perimeter vs. Interior Pocket 18
     - 4.4.2. Programing the Perimeter Pocket 19
     - 4.4.3. Programing the Interior Pocket 21
   - 4.5. Drilling 22
   - 4.6. End of Setup One 24
5. Creating our Second Setup and Its Toolpaths 25
   - 5.1. Setup 2 (Op2) 25
   - 5.2. 2D Face 26
   - 5.3. 2D Perimeter Pocket 27
   - 5.4. Chamfer 28
     - 5.4.1. A note on Programing Chamfers 29
   - 5.5. End of Setup 2 30
6. Exporting your Program as a .nc File to the Mill 31
   - 6.1. Exporting from Fusion360 31
   - 6.2. Loading File onto Tormach Mill 32
7. Running your CAM Programs 33
   - 7.1. Zero the Machine 33
   - 7.2. Insert the First Tool 33
   - 7.3. Cycle Start 33
   - 7.4. Tool Changes 33
   - 7.5. Continue the Process & Repeat 34
8. Conclusion 35

Index of Figures 36

1. Table of Tools

The table below contains the tools currently available for use in the Makerspace. Please refer to this table when needed. Please note that tools **110-122** require installation and into a collet, are are much more challenging to use.

Tool Name	Tool Number	Tool Type
Super Fly	2	Face Mill
Shear Hog	3	Flat End Mill
Haimer Probe	5	Probe
1/2″ 4 Flute	11	Flat End Mill
3/8″ 4 Flute	12	Flat End Mill
1/4″ 2 Flute	16	Flat End Mill
1/2″ 2 Flute	17	Flat End Mill
5/8″ 4 Flute Large	19	Flat End Mill
1/4″ Ball	21	Ball End Mill
1/4″ Chamfer	22	Chamfer/Engrave
1/8″ Drill	42	Drill
3/8″ Drill	43	Drill
1/4″ Spot Drill	44	Spot Drill
22mm Face Mill	46	Face Mill
1mm End Mill (HSS)	110	Flat End Mill
2mm End Mill (HSS)	111	Flat End Mill
3mm End Mill (HSS)	112	Flat End Mill
4mm End Mill (HSS)	113	Flat End Mill
6mm End Mill (HSS)	114	Flat End Mill
4mm Ball End Mill (HSS)	115	Ball End Mill
1mm End Mill (Carbide)	116	Flat End Mill
2mm End Mill (Carbide)	117	Flat End Mill
3mm End Mill (Carbide)	118	Flat End Mill
4mm End Mill (Carbide)	119	Flat End Mill
6mm End Mill (Carbide)	120	Flat End Mill
4mm Ball End Mill (Carbide)	121	Ball End Mill
4mm 45° Chamfer (Carbide)	122	Chamfer/Engrave

2. Downloading Necessary Files

2.1. Tool Library and Vise 3D File

The very first step to Computer Aided Machining (CAM) is downlaoding the necessary files. For our case, that is the **.tool** file that includes an up to digitized library of all the milling tools available in the Makerspace (See Section 1), as well as the **3D Model** of the vise that is installed in the mill. Both of these files are avilable here°, or via the Makerspace Website°. Download the .f3d file entitled '*Tulane_Tormach_5in_Vise*,' as well as the .tool file entitled '*Tulane CNC Mill Library*,' that has the most updated date.

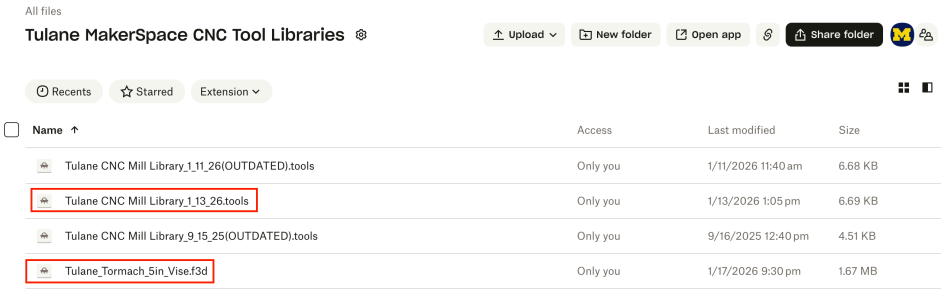


Figure 1: Highlighted are the two files that need to be downloaded

2.2. Importing into Fusion360

2.2.1. .tool File

Once you have the two files downloaded, open a new document in Fusion360. Once in the document, navigate to the manufacture workspace, pictured below:

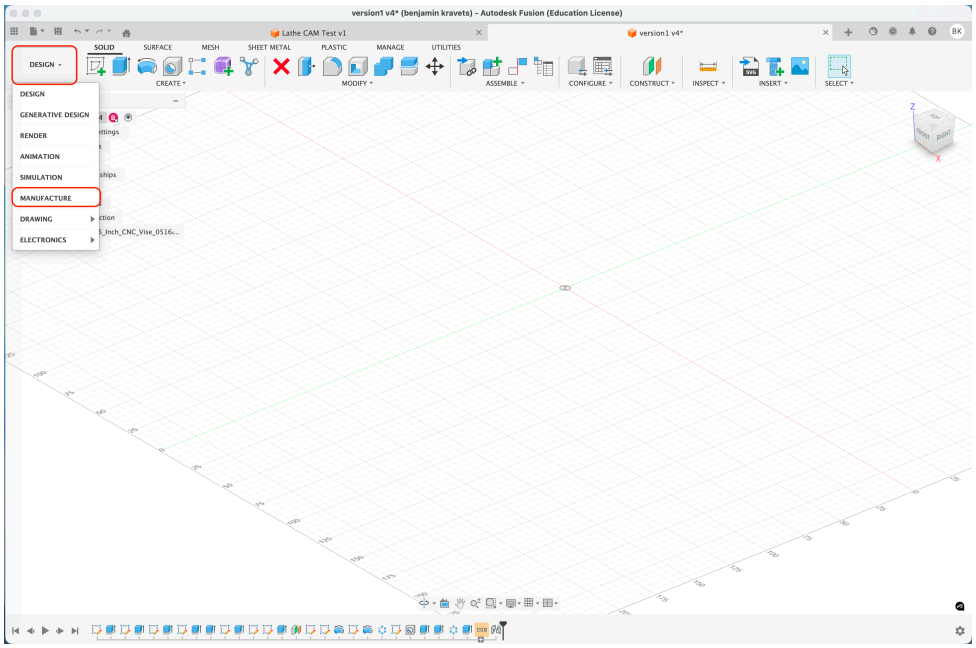


Figure 2: Highlighted are the steps to navigate to the manufacture workspace

The manufacture workspace is the space in which we will program our CAM Toolpaths, and adjust/modify anything CAM related. First, we will import the **.tool** file we previously downloaded. To do this, open the **Tool Library** window, right click on 'local,' then click 'import,' and select the **.tool** file from wherever you previously downloaded it.

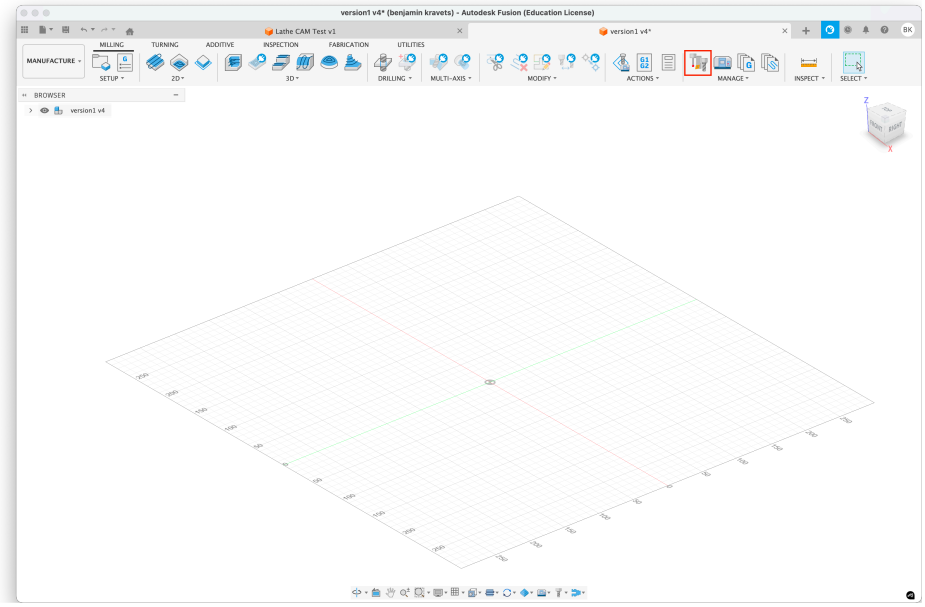


Figure 3: Highlighted is the **Tool Library** button

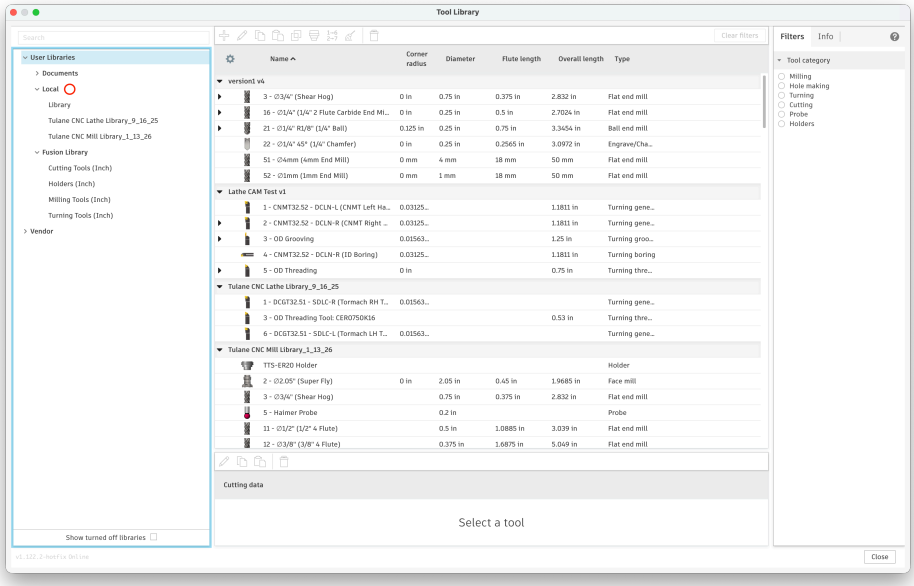


Figure 4: Highlighted is where the user should right click to import the **.tool** file

Once these steps are completed, you should see a new tool library appear on the left hand side named Tulane CNC Mill Library. This is also where you can go to see specific data about a tool, such as length, diameter, number of flutes, etc.

2.2.2. Vise .f3d File

Next we will be importing the .f3d file of the vise that is installed in the mill. The product page for the vise is available here°. The reason we import the 3D model of the vise into our document is so that we can make sure that the toolpaths we create will not collide with the vise.

To import the .f3d file, first navigate back to the Design Workspace. Then. click on the **Data Panel** button in the top left corner of Fusion. Once this is open, navigate to the location your

current document is. Once there, click the **upload** button, and upload the **.f3d** file of the vise that we previously downloaded.

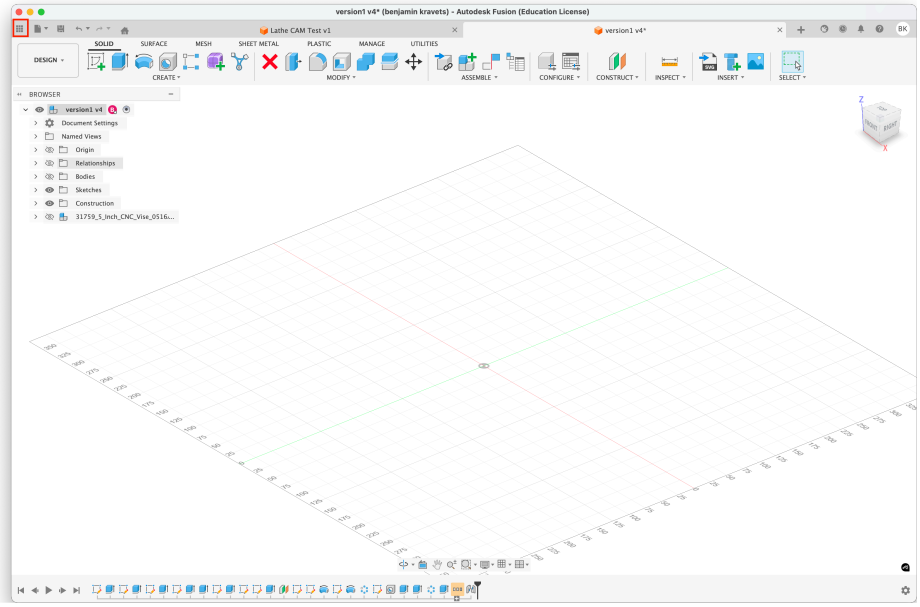


Figure 5: Highlighted is **Data Panel Button**

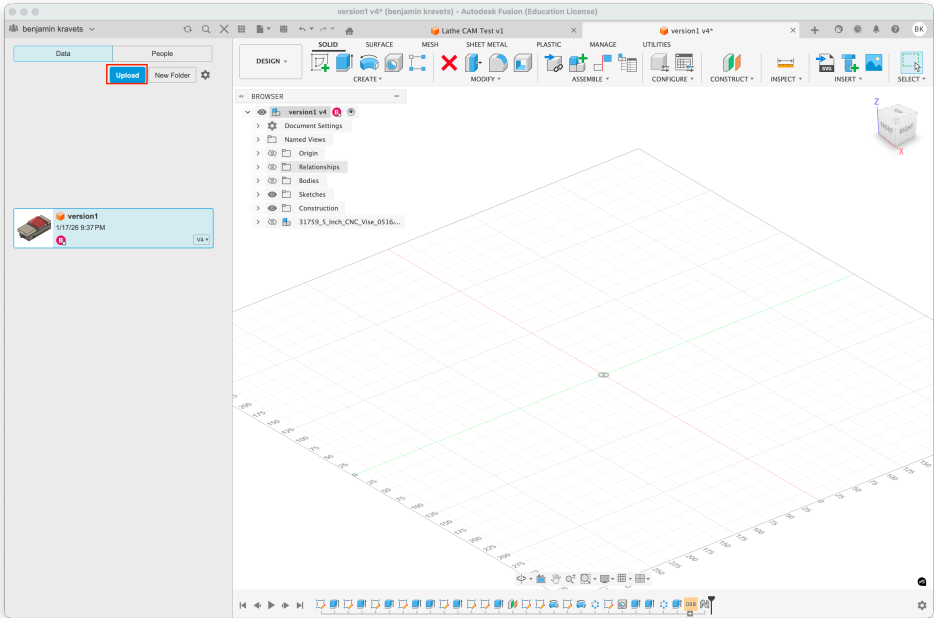


Figure 6: Highlighted is **upload** button that we will use to import the vise **.f3d** file

Once the vise .f3d file is finished uploading in the data panel, right click on it and select **Insert into Current Design**. Insert it anywhere into the document, and **ensure** that the hexagonal rod coming out of the vise is facing the **front.** Then, right click on it in the browser and click on '**Break Link**.' This will enable us to use the parametric functionality of the vise. When you click this, you may get three errors titled "CutPasteBody." These will not affect any functionality and can be ignored. Next, click '**Unground from Parent**.' Finally, toggle the visibility of the vise off, as it will not be used for now.

2.3. Downloading the training 3D Model.

This training does not cover 3D Modeling of a part in Fusion360. Thus, we provide the 3D Model of the training part. Follow the steps from Section 2.1 and 2.2 to download the training 3D Model from the Dropbox and import it into your Fusion360 File. The **.f3d** file is titled 'Makerspace_CAM_Tutorial_Part.f3d.' Once the part is imported into your Fusion360 file, use the **Joint** tool to mate it to the origin. It is important to ensure that the top of the part (the face with five holes), is facing up in the Z-Axis. Your document should look like this:

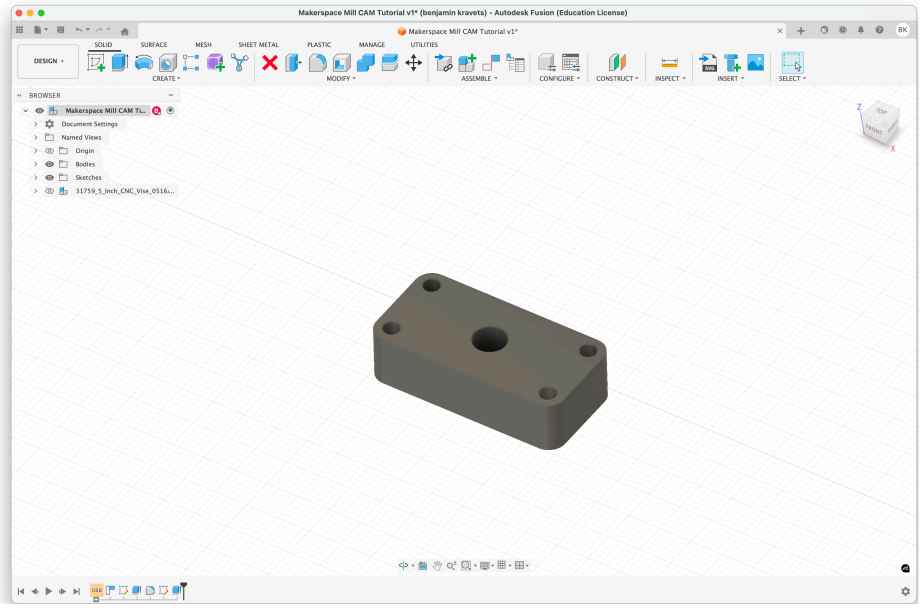


Figure 7: Example of what your Fusion360 document should currently look like

3. Inserting the Parametric Vise & Creating our Stock

3.1. Parametric Functionality

The vise that we downloaded has parametric functionality built in. This will help us easily create the virtual size of our stock, as well as make sure we use the right sized parallels. You should now enable visibility of the vise in the Browser. To access the parameters, click on the **Change Parameters** button under the modify tab, as shown below.

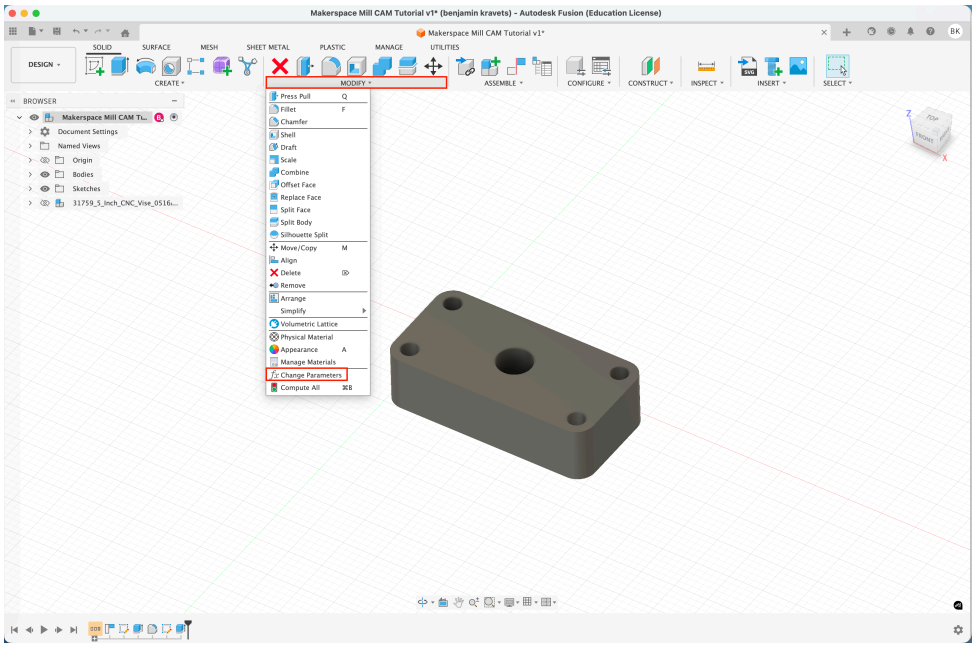


Figure 8: Highlighted are the steps to access the **Parameters** window

Once the parameters window is open, you should see seven different parameters that you can edit: StockX, StockY, StockZ, StockOffset, ParallelThickness, ParallelWidth, and ParallelHeight.

The StockX, StockY, and StockZ parameters edit the size of the stock, visualized by the transparent red box. Edit these parameters to see how the size of the stock changes accordingly.

The StockOffset parameter is used to offset the stock from the origin along the X-Axis. Edit this parameter to see how the location of the stock changes accordingly.

The ParallelThickness, ParallelWidth, and ParallelHeight parameters are used to edit the dimensions of the parallels. Edit these parameters to see how the size of the parallels changes accordingly.

For our use, please set StockX = 4in, StockY = 1.8in, StockZ = 1.0in, StockOffset = 0in, ParallelThickness = 0.125in, ParallelWidth = 6in, and ParallelHeight = 1in. Your parameters should match those in the image pictured below.

User Parameter	StockX	in	4 in	4.00
User Parameter	StockY	in	1.8 in	1.80
User Parameter	StockZ	in	1 in	1.00
User Parameter	StockOffset	in	0 in	0.00
User Parameter	ParallelThickness	in	0.125 in	0.125
User Parameter	ParallelWidth	in	6 in	6.00
User Parameter	ParallelHeight	in	1 in	1.00

Figure 9: Shown here are the proper parameters for our use

3.2. Mating the Vise to the Part

Next, we will use the **Joint** tool to mate the **Bottom Center** of the red stock visualization to the **Bottom Center** of our part. Add a **Z Offset** of 0.1in to the Joint. You should end up with a setup that looks like this:

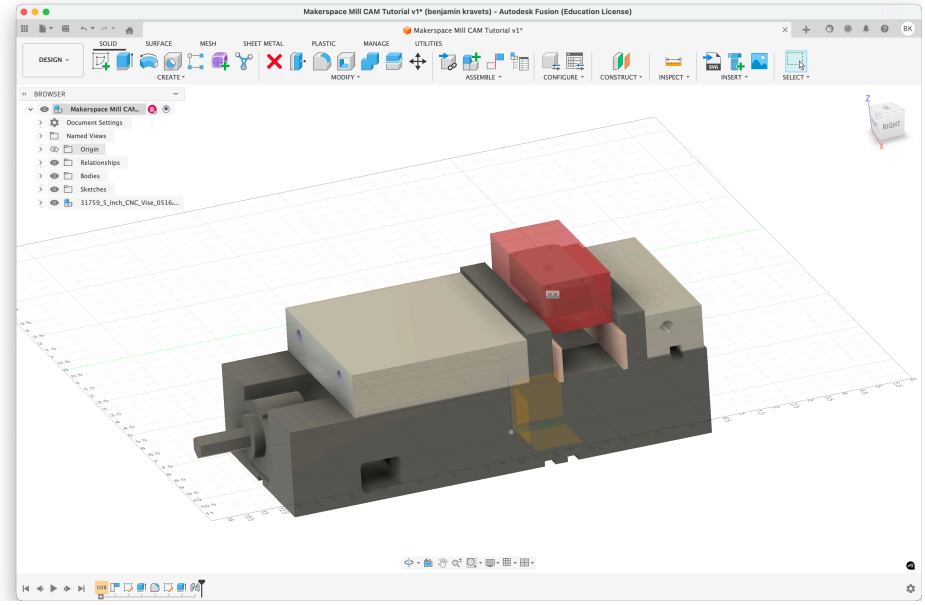


Figure 10: Shown here is what the current setup should look like

Please ensure that the part is offset 0.1in from the bottom of the stock.Once you do this, right click on the vise in the Browser and rename it to **'Op1Vise'**

3.3. Insert Second Vise

Because this part requires two setups to mill (similar to the training part for the basic CNC Mill Training), we will need two setups. Unfortunately, this means we have to insert and program two vises.

Still in the Design Workspace, use the Data Panel to insert another vise into the current design. However, before you fully import it, make sure to rotate it **180° along the Y-Axis**. You should end up with a vise that is upside down compared to the first vise, as shown below.

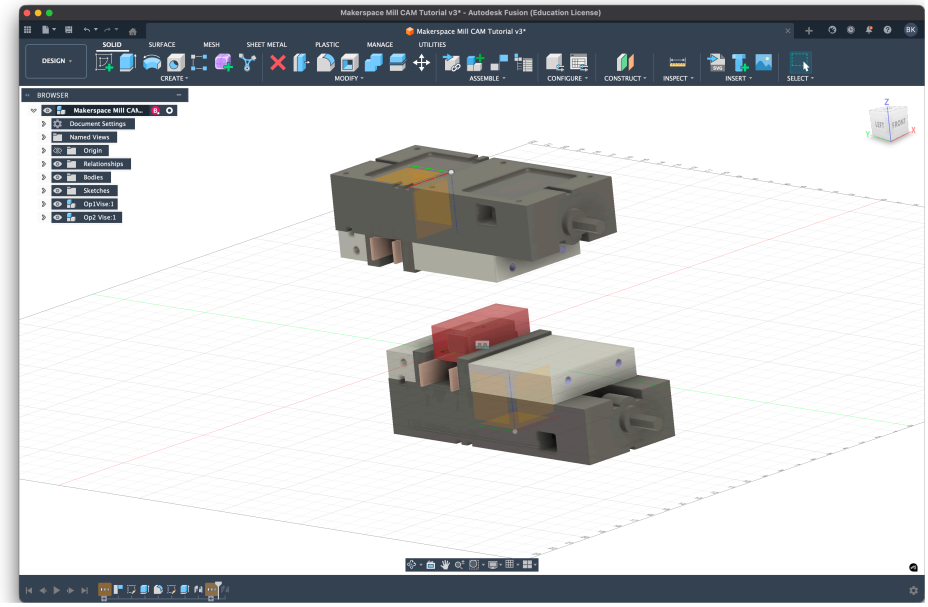


Figure 11: Shown here is what the current setup should look like

Next, right click on the second vise in the Browser and click **Break Link**. This will enable the parametric features of the second vise. Additionally, rename the new vise to **Op2Vise**. Now that we have the second vise imported, go into the **Parameters Window** and change **StockY_1** to be **1.5in** rather than 2in.

3.4. Mating the Second Vise

We will be mating the second vise differently than the first vise. This time, we are going to mate the **bottom of the stock from Op1Vise** to the **top of the stock from Op2Vise.** In the joint, make sure to add a **Z offset** of **0.6in** (note that this number may change in when programing CAM for different parts and is based on the dimensions of the part you are trying to mill. This will make more sense when we get to programing the toolpaths). Additionally, change **ParallelHeight_1** to **1.1in**. Your setup should look like this if everything is done correctly:

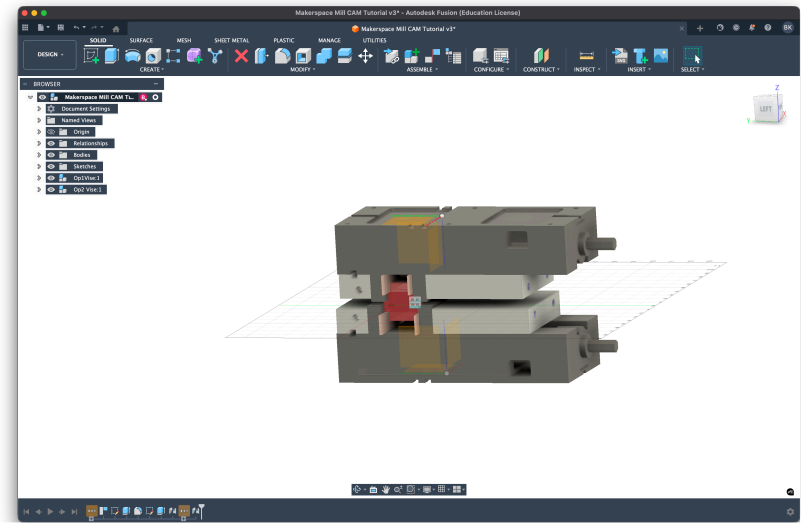


Figure 12: Shown here is what the current setup should look like

4. Creating our First Setup and Its Toolpaths

Next, we will be creating our setups and toolpaths. A **setup** (also sometimes refereed to as an **Operation** or **Op1**) the foundational step that defines what you're machining and how, establishing the context for all subsequent toolpath operations by defining the machine, operation type (milling, turning, etc.), Work Coordinate System (WCS) origin, stock material, and part orientation. A **toolpath** is the path that the tool will follow when cutting on the mill.

4.1. Setup 1 (Op1)

Navigate back to the manufacture workspace. Click on the **New Setup** button. Once you do this, the setup panel will appear on the right hand side of the screen. First, click the **Select** button under the 'Machine' menu and select the **Tormach 770M** from the Fusion Machine Library.

Next, ensure that the operation type is set to **Milling**, and not turning, cutting, or additive.

Under Work Coordinate System (WCS), ensure that you select **Model Orientation** under the 'Orientation' menu, and **Selected Point** under the 'Origin' menu. Once you choose **Selected Point**, click on the circle located on the top face of the model (note that you may have to hid the model of the vise to access this point). This should set the WCS such that the X-Axis is pointing to the right, the Y-Axis is pointing forwards, and the Z-Axis is pointing upwards. Ensure that your WCS matches the WCS shown in the image below.

Under the **Model** section, select the our training part that we previously imported (note that you may have to hid the model of the vise to access this point).

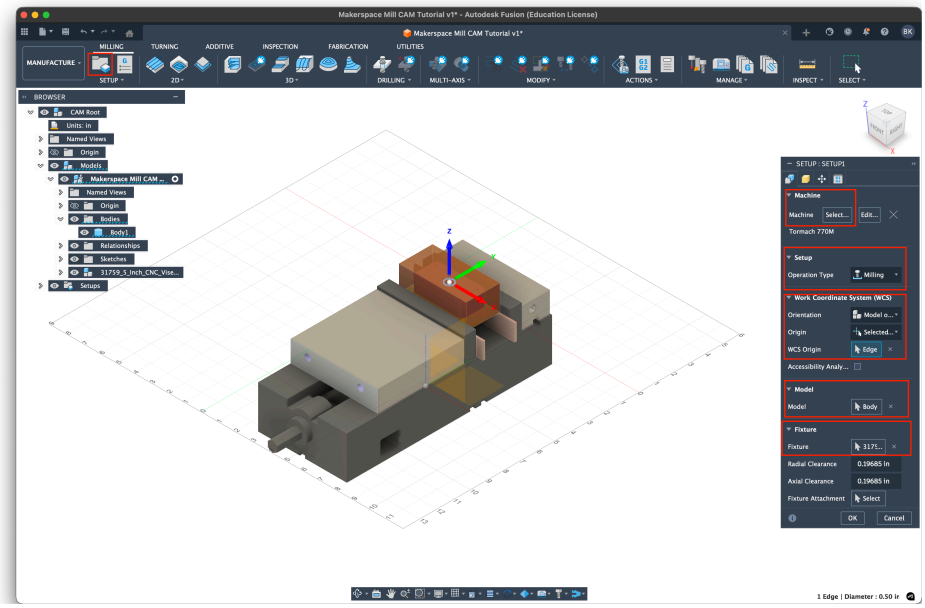


Figure 13: Shown here is what the current setup should look like

Under the **Fixture** section, we want to select the vise that we imported. However, since the parametric stock we set up is under the same model, we need to select all parts of the vise

besides the stock. Imaged below is what should be selected. This will inform Fusion that the vise is a piece of work holding equipment, and not something that we want to machine.

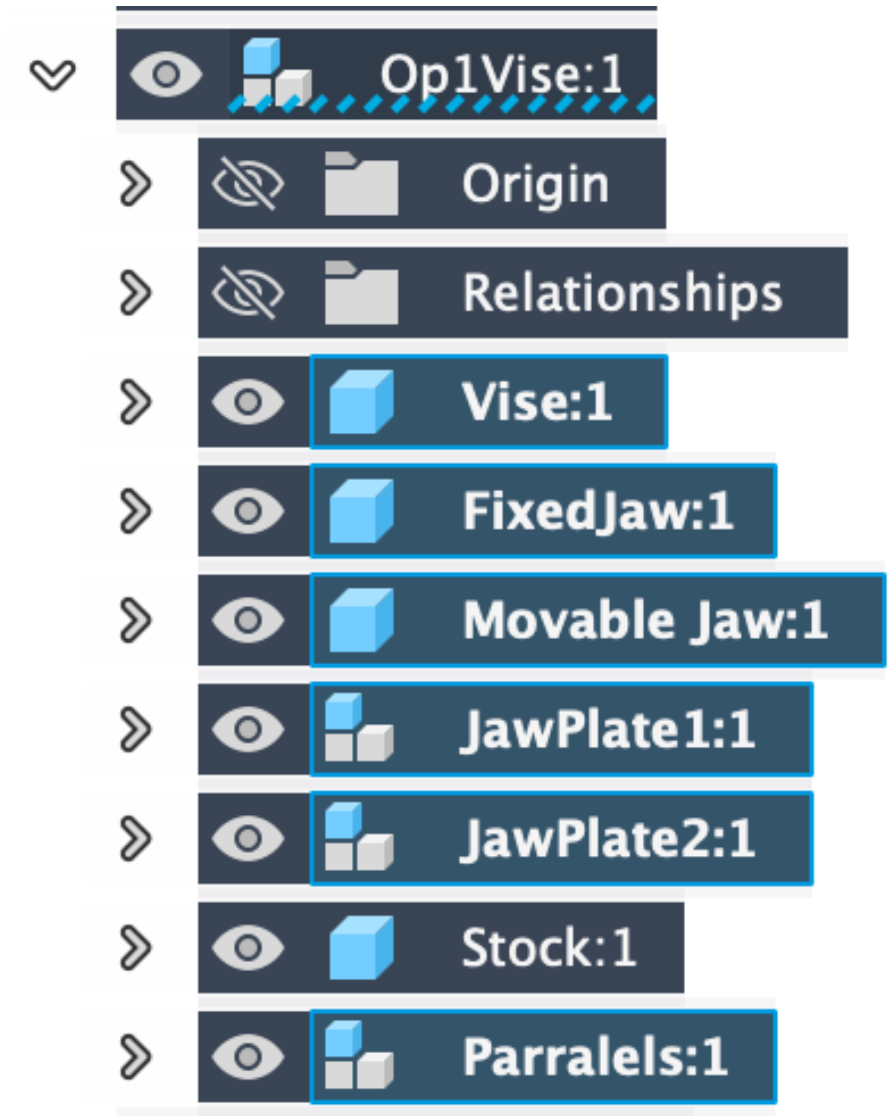


Figure 14: Shown here is should be selected when setting up the **Fixture** section

Move to the next tab in the setup window. In the 'Stock' section, choose **From Solid** from the drop down menu. Then select the red translucent box. This will set the stock in the manufacture workspace as the stock that we defined earlier. Ensure that the stock dimensions match what we set them as in Section 3.1.

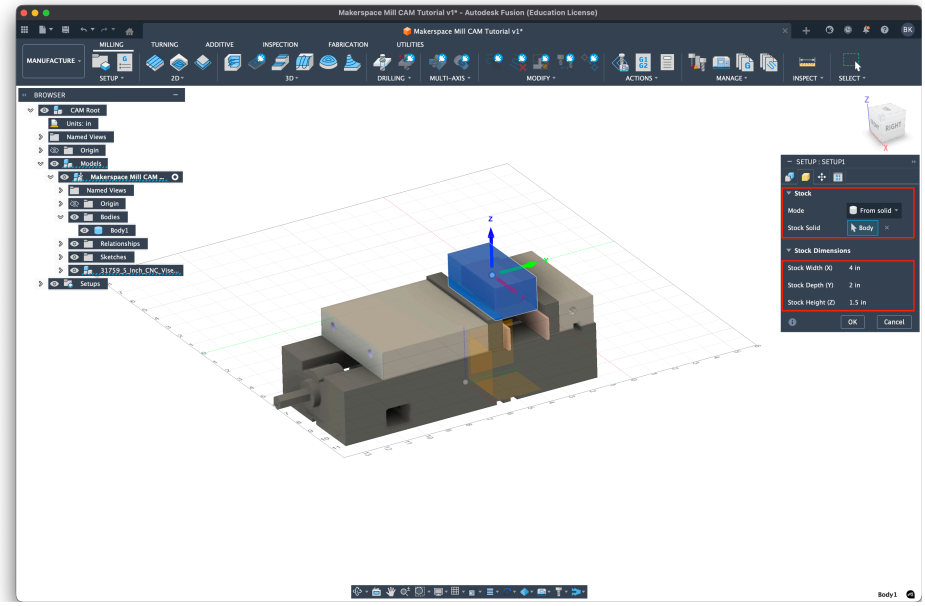


Figure 15: Shown here is what the current setup should look like

Next, move to the third tab in the setup window. Set the **Y Distance** to **–2**. This will move the vise to be in the same spot as it is in the mill.

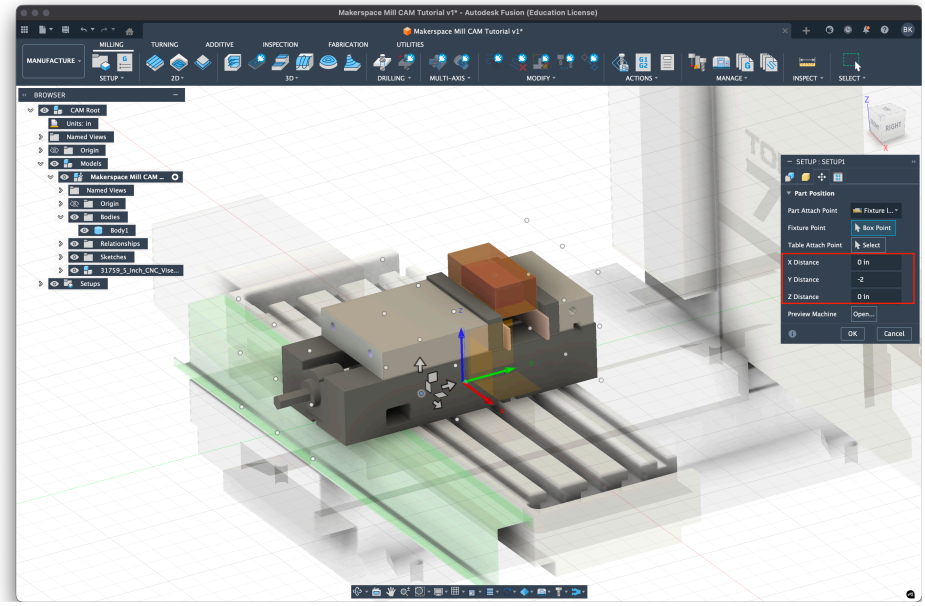


Figure 16: Shown here is what the current setup should look like

Next, move to the fourth and final tab in the setup window. Ensure that your **Program Name/ Number** is set to **1001**, and that your **WCS** is set to **54**. Once this is complete, click **OK** to close the setup window. Lastly, toggle off the visibility of the vise in the Browser window. Shown below is what your current setup should look like:

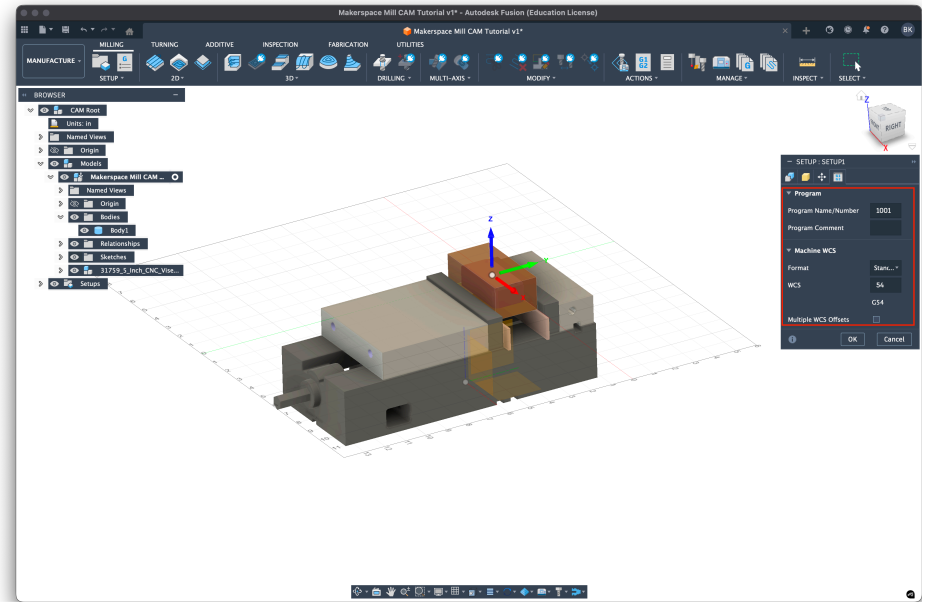


Figure 17: Shown here is what the current setup should look like

4.2. Tabs in each Toolpath

Each toolpath that we will program has some common tabs and settings. The first tab is the **'Tool'** tab. This is where we will select which tool we will use for the operation, program in the feeds and speeds, as well as state what type of coolant we want. For our purposes, we will leave the feeds and speeds alone, and **always select flood coolant.**

The second tab is the **'Multi-axis'** tab. Since our mill does not have a permanent 4th or 5th axis, we will not use this tab.

The third tab is the **'Geometry'** tab. This is the tab we will use to select the geometry we want to reference for each toolpath. When in this tab, you can either use your mouse to select geometry, or use the 'Face Contours' and 'Chain' tools to select more precise geometry.

The fourth tab is the **'Heights'** tab. This tab is where we will define the Z-axis limits for each toolpath, as well as define the clearance height, retract height, and feed height. For most cases, you will only need to modify the **Top Height** and **Bottom Height** options.

The fifth tab is the **'Passes'** tab. This is where we can define things like whether or not we want to take multiple passes, if you want to leave axial or radial stock, or if you would like to take a finishing pass or a roughing pass. We will be using this tab often.

The sixth and last tab is the **'Linking'** tab. This is where you can modify the lead-in/lead-out points of a toolpath; however, this is fairly complicated, and Fusion is good at doing this automatically, so we will not be modifying anything in this tab.



Figure 18: Shown here are the tabs in each toolpath operation window

4.3. 2D Face

The first toolpath we want to program is going to be a 2D Facing operation. This will face the stock down to a desired height. To do this, select **Face** from the '2D' drop down menu in the toolbar.

Once in the 2D Face window, go to the **Tool** tab and use the selection menu to select **Tool 3, the Shear Hog**.

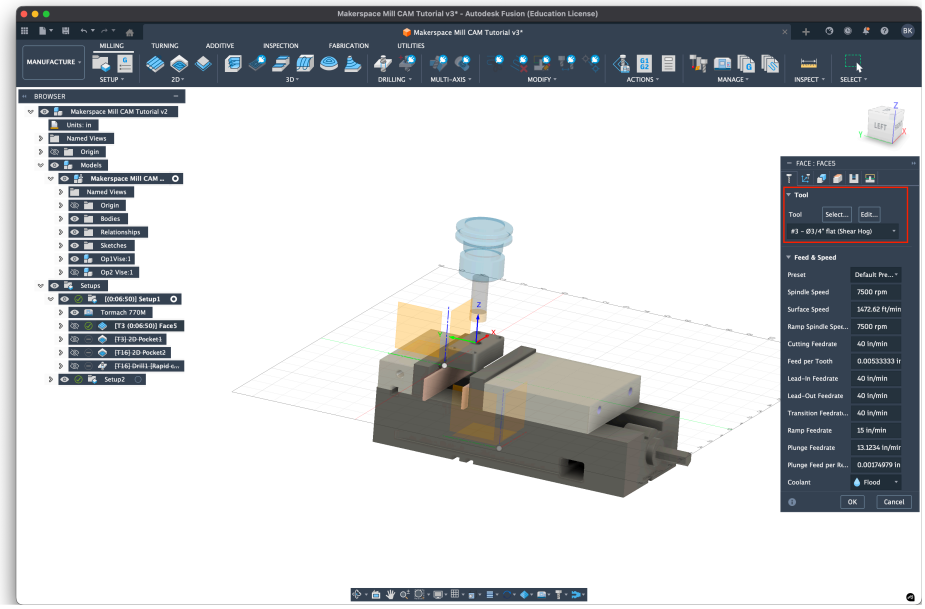


Figure 19: Shown here are the steps to select a tool in a toolpath

For this operation, we want to face the entire piece of stock, and not just a specific section. Thus, we will not select any geometry in the **Geometry** tab, and Fusion will automatically face the entire piece of stock.

Next, navigate to the **Heights** tab. Ensure that the **Top Height** is set to 'Stock Top' and that the **Bottom Height** is set to 'Model Top'. This means that the toolpath will start cutting at the top of the stock, and mill down until it gets to the top of the model.

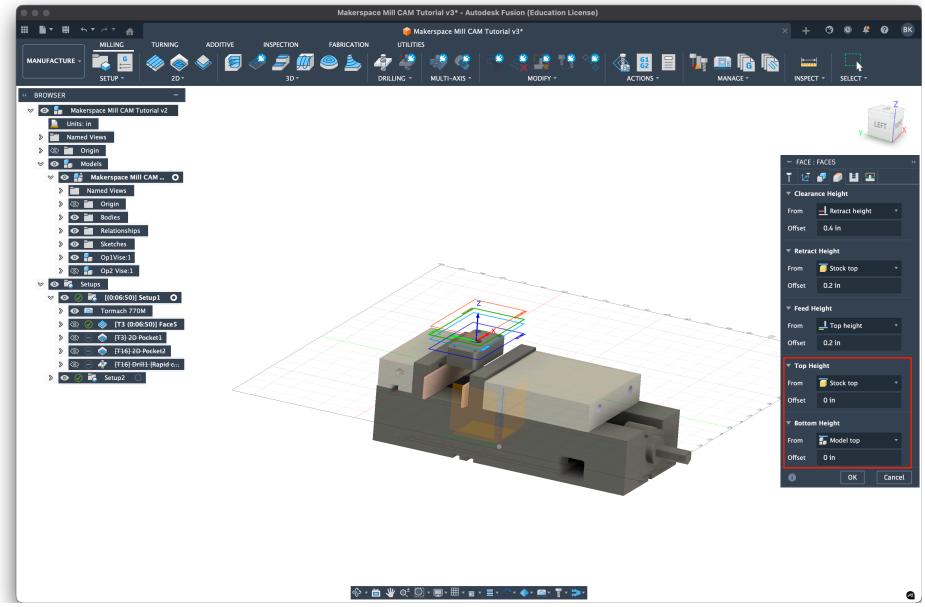


Figure 20: Shown here are the steps to program the heights for a **2D Facing** operation

Next, move to the **Passes** tab. In this tab, we want to make sure to enable **Multiple Depths**, with a Maximum Stepdown of **0.04in**. This will tell Fusion to have the toolpath take multiple passes to face the material, ensuring that we do not wear out the tool. Additionally, make sure that Stock to Leave is **disabled**. If enabled, it would not mill all of the material we told it to. This is useful if you want to go in for a finishing pass with a more precise tool, but we are not going to do that.

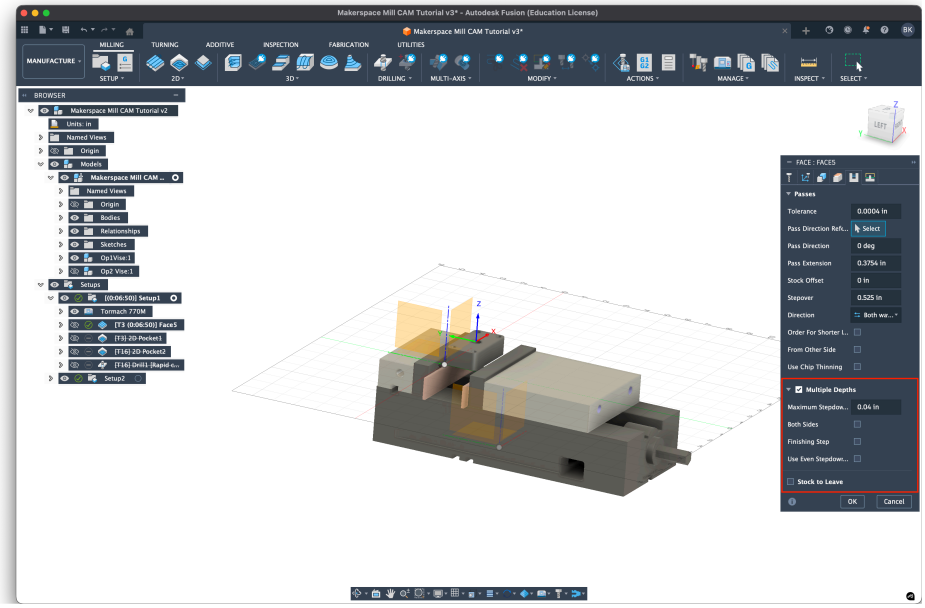


Figure 21: Shown here are the steps to program the passes for a **2D Facing** operation.

Once you complete this step, click **OK** at the bottom of the window. If everything was done correctly, you should see a visualization of the toolpath similar to the below image:

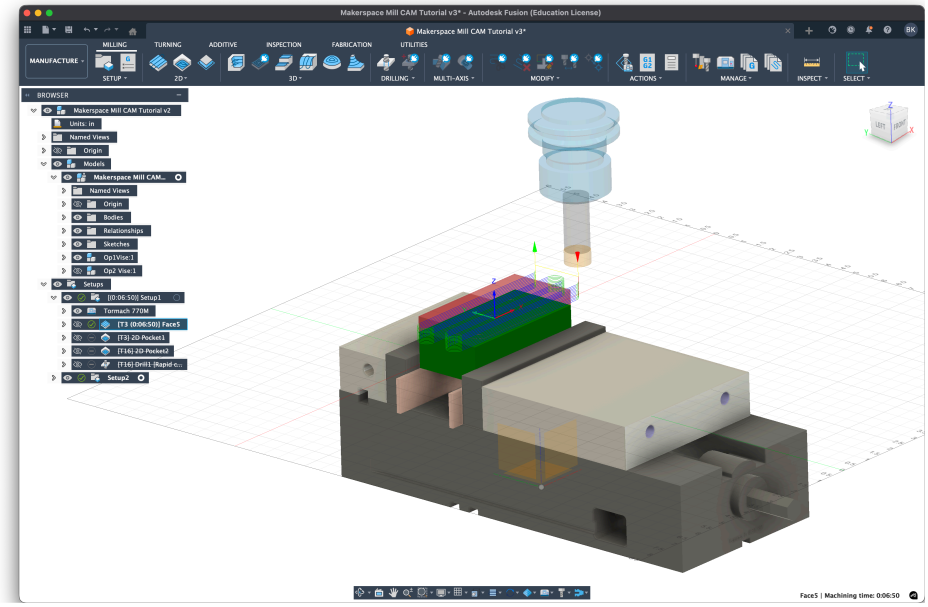


Figure 22: Shown here is what the screen should display after sucessfully programing a **2D Facing** operation.

4.4. 2D Pocket (Perimeter and Interior)

Next, we will program a **2D Pocket** to first mill out the exterior geometry of the part, then to mill out the hole in the center of the part. However, it is important to note that there are two distinct uses of the **2D Pocket** tool: Perimeter and Interior

4.4.1. Perimeter vs. Interior Pocket

A perimeter **2D Pocket** can be used when you want to mill out the outer geometry of a shape (e.g. milling your specific part's shape out of a standard rectangular stock). A interior **2D Pocket** can be used to mill out a shape on the interior of a part (such as the hole in the center of our part that we will mill out). However, note that the shape does not have to be perfectly circular or rectangular. The **2D Pocket** tool is able to mill out any shape so long as it is physically possible for the tool.

We can note the difference between the two types of **2D Pockets** with the red arrow, shown below, that appears when geometry is selected in the **Geometry** tab of a **2D Pocket** operation. When the red arrow is outside of the selected geometry, it is a Perimeter Pocket. When the red arrow is inside of the selected geometry, it is a Interior Pocket.

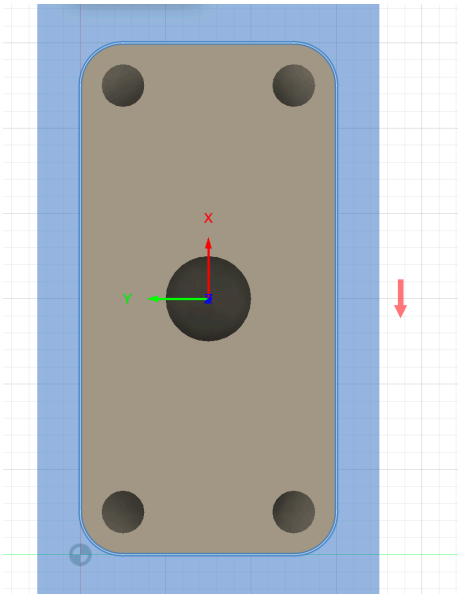


Figure 23: Example of a **Perimeter Pocket**

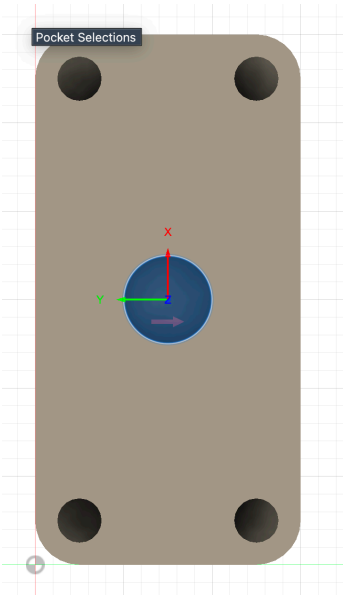


Figure 24: Example of a **Interior Pocket**

Now that the difference between the types of **2D Pockets** is established, we can move on to programing the first pocket.

4.4.2. Programing the Perimeter Pocket

Under the 2D Menu in toolbar, select **2D Pocket**. In the **Tools** tab, select **Tool 3, the Shear Hog**. Next, navigate to the **Geometry** tab. We want to select the perimeter of the top face. Ensure that the red arrow is outside of the selected geometry. If done correctly, your screen should look like this:

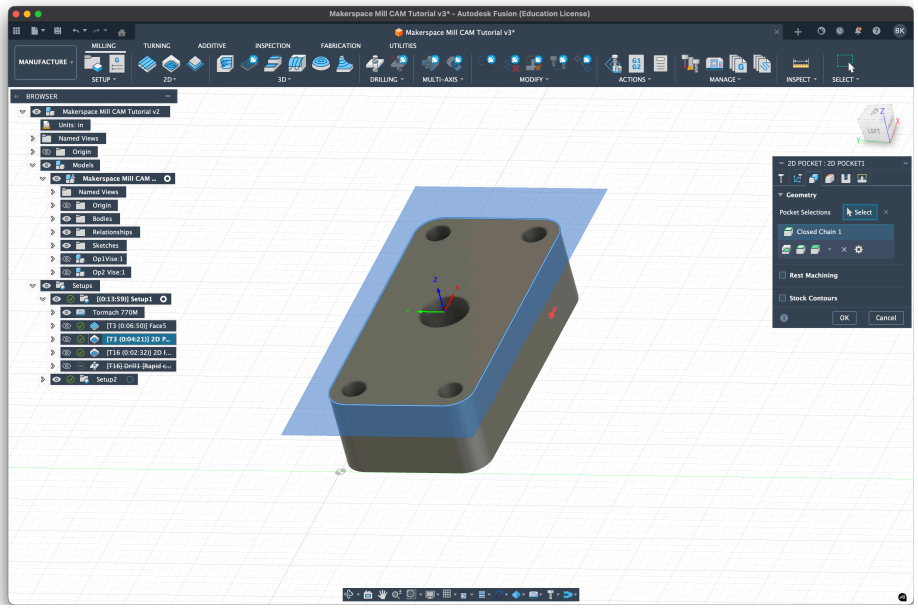


Figure 25: Shown here is what the screen should display after successfully programing the **Geometry** tab of a **2D Perimeter Pocket**

Next, go the the **Heights** tab. Ensure that **Top Height** is set to 'Model Top' (**Not** 'Stock Top'). For **Bottom Height**, we want to leave it as 'Selected Contours,' but we want to add a offset of

–0.40in. We do this because we want to mill as much of the material away as we can before colliding with the vise. Your **Heights** tab should look as follows:

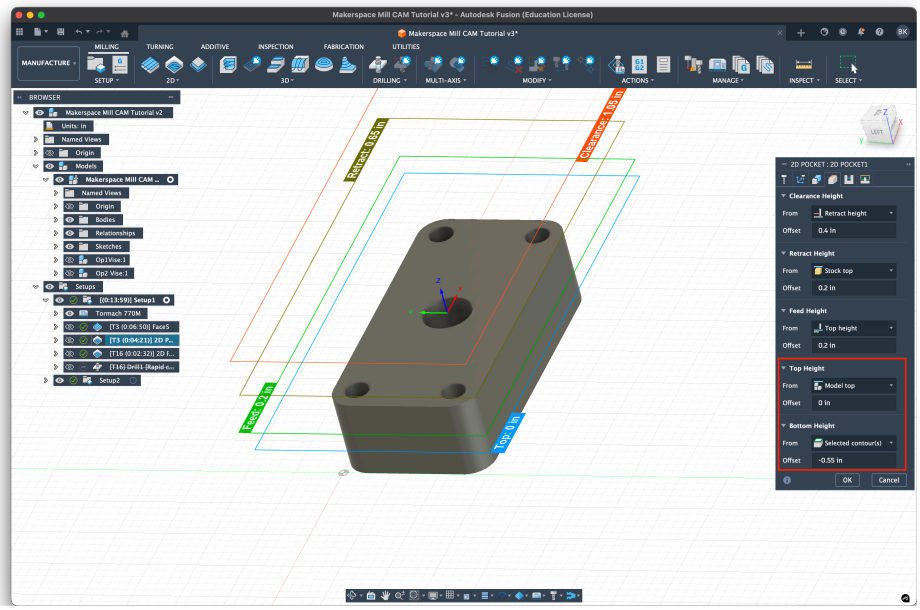


Figure 26: Shown here is what the screen should display after successfully programing the **Heights** tab of the **2D Pocket**

Lastly, navigate to the **Passes** tab and ensure that **Multiple Passes** is enabled with a **0.075** Maximum Roughing Depth, and that **Stock to Leave** is **disabled**. Once this step is completed, click **OK** on the **2D Pocket** window. If everything was done successfully, it should look as follows:

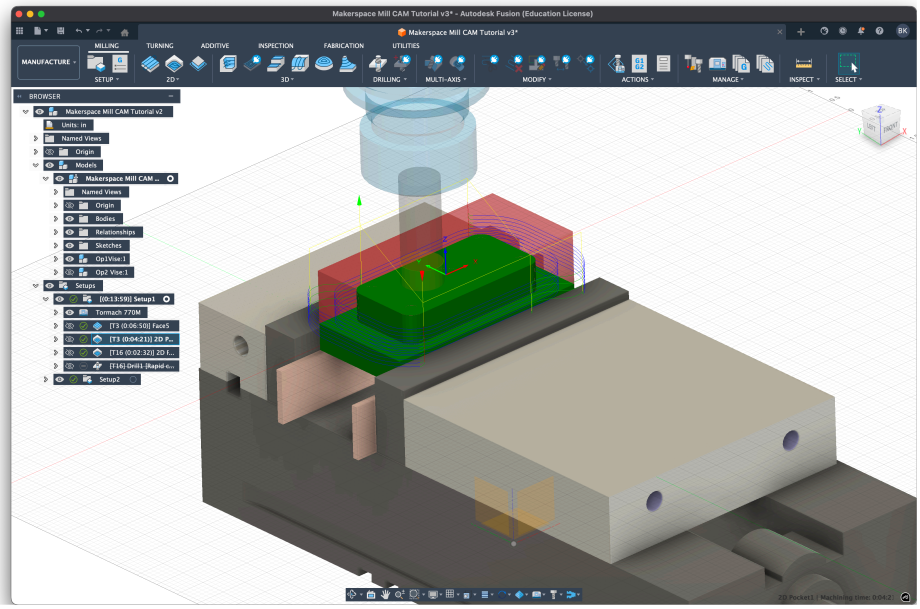


Figure 27: Shown here is what the screen should display after successfully programing the **2D Perimeter Pocket**

As you can see, when we enable visibility of **Op1Vise**, we have milled down the stock to just above the vise. This is ideal, and will limit the amount of milling needed in second setup.

4.4.3. Programing the Interior Pocket

Next, we will program the **Interior 2D Pocket**. The process will be practically the same as for the perimeter pocket, we will just select different geometry and use a different tool.

Create a new **2D Pocket** toolpath from the **2D** menu in the toolbar. In the **Tools** tab, change to **Tool 16, the 1/4″ 2 Flute End Mill**.

Next, navigate to the **Geometry** tab, and select the **bottom face** of the hole in the center of the part. If selected correctly, it should look as follows:

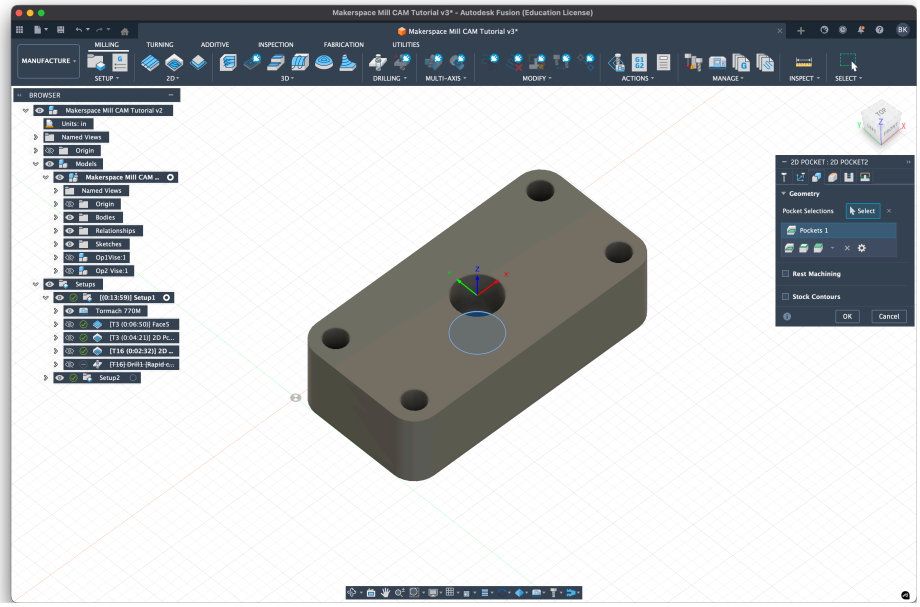


Figure 28: Shown here is what the screen should display after successfully programing **Geometry** tab of the **Interior 2D Pocket**

Next, move to the **Heights** tab. Ensure that the **Top Height** is set to 'Model Top,' with a **0.05in** offset. The purpose of the offset is to give some leeway for the tool to get to speed before it starts drilling. Next, make sure that the **Bottom Height** is set to 'Selected Contour.' If it is not, the mistake was likely made in setting up the **Geometry** tab.

Lastly, go to the **Passes** tab and enable **Multiple Depths** with a Maximum Roughing Depth of **0.025in**, and ensure that **Stock to Leave** is **disabled**. Once this is completed, click **OK** at the bottom of the **2D Pocket** window. If everything was done correctly, the toolpath should look as follows:

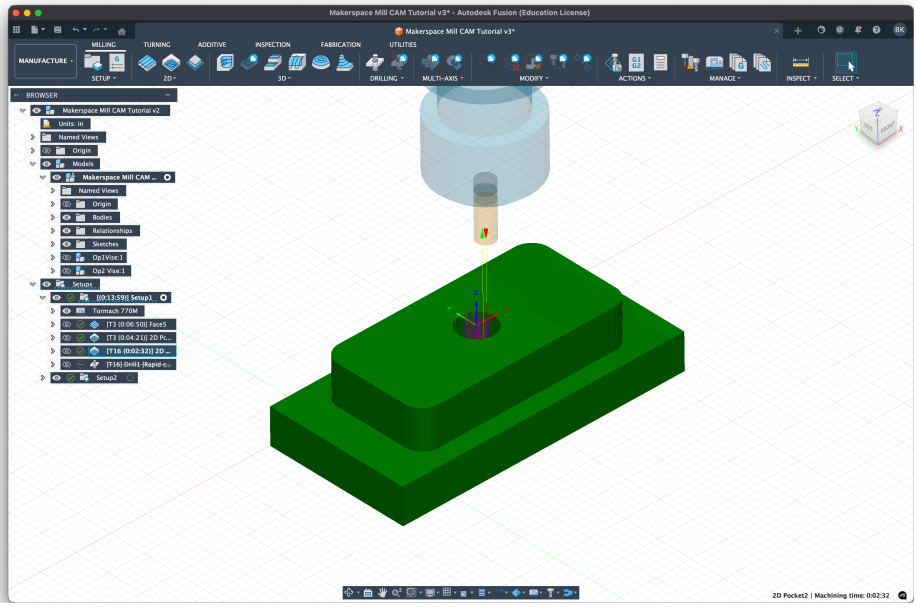


Figure 29: Shown here is what the screen should display after successfully programing the **2D Interior Pocket**

4.5. Drilling

Next, we are going to program the toolpath to drill out the four ∅0.25in holes in the four corners of our part. Instead of using the **2D Pocket** tool (which would work) we are going to use the **Drilling** toolpath instead. To start, select this operation from the center of the toolbar.

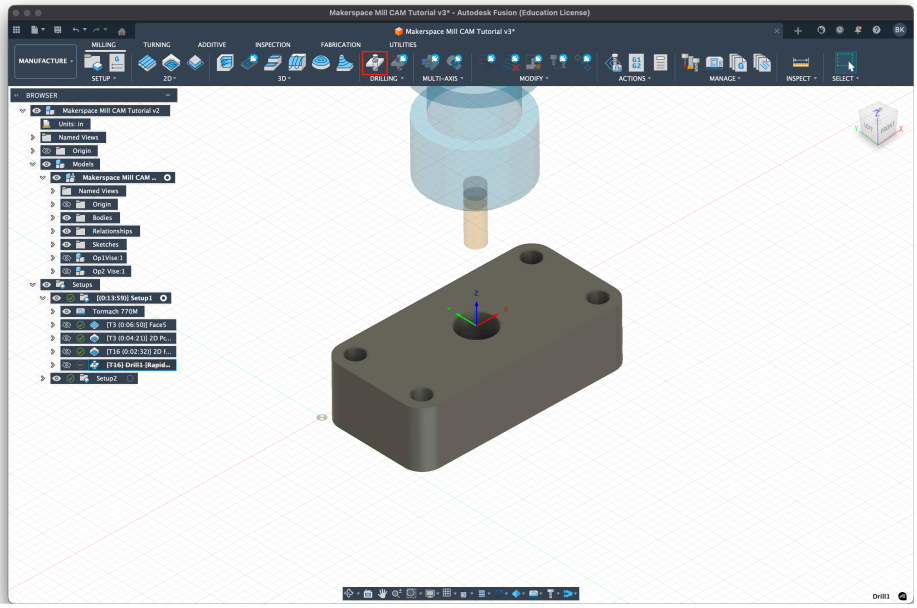


Figure 30: Shown here is where to click to select the **Drilling** toolpath

Once the **Drilling** toolpath opens, ensure that **Tool 16, the 1/4″ 2 Flute End Mill** is selected. Under the **Geometry** tab, select the cylindrical walls of each of the four ∅0.25 holes. If done correctly, it should look as follows:

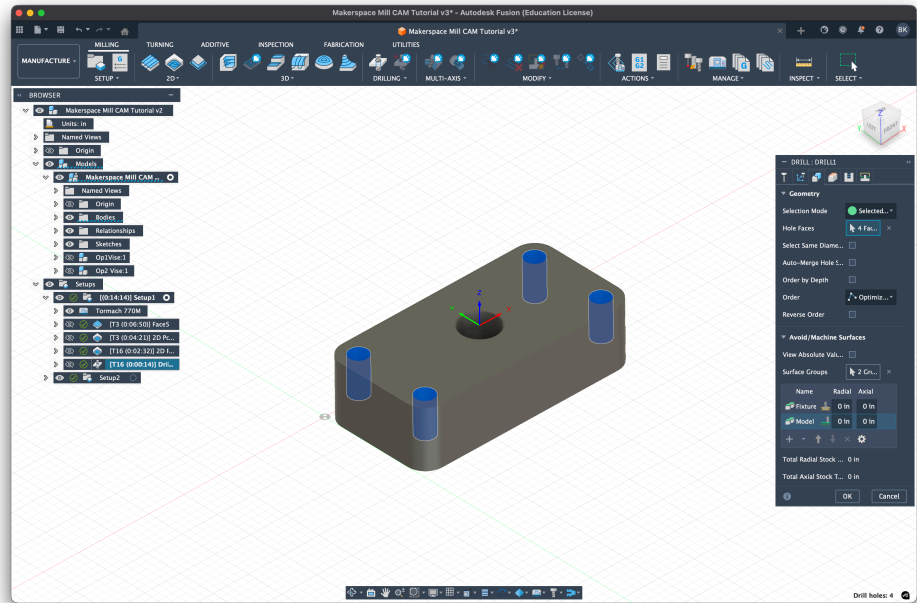


Figure 31: Shown here is what it should look like after successfully selecting the four ∅0.25in holes in the **Geometry** tab of the **Drilling** toolpath.

Next, go to the **Heights** tab and ensure that **Top Height** is set to 'Hole Top' and **Bottom Height** is set to 'Hole Bottom.' If these do not automatically populate, an error was likely made when setting up the **Geometry** tab.

Next, you will notice that the second to last tab, which has previously been called **Passes** is now called **Cycle**. This is because, for the **Drilling** operation, instead of setting up multiple passes or stock to leave, you instead choose what type of Drilling Cycle you would like. For our purposes, select **Drilling - rapid out**.

Next, go to the **Linking** tab and enable **Ignore Gouges**. Once this step has been completed, click **OK** at the bottom of the **Drilling** toolpath window. If everything was done correctly, your screen should show a toolpath similar to the following.

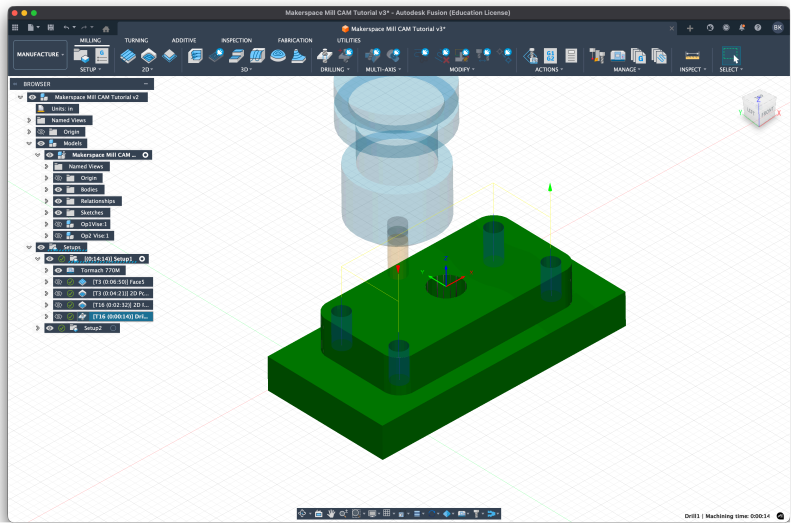


Figure 32: Shown here is what it should look like after successfully programing the **Drilling** toolpath

4.6. End of Setup One

That concludes the programing for Setup One. It is now good practice to simulate the program in Fusion360 to ensure that there is no errors that we can see. To do this, click the **Simulate with Machine** button located at the right side of the toolbar at the top of the screen.

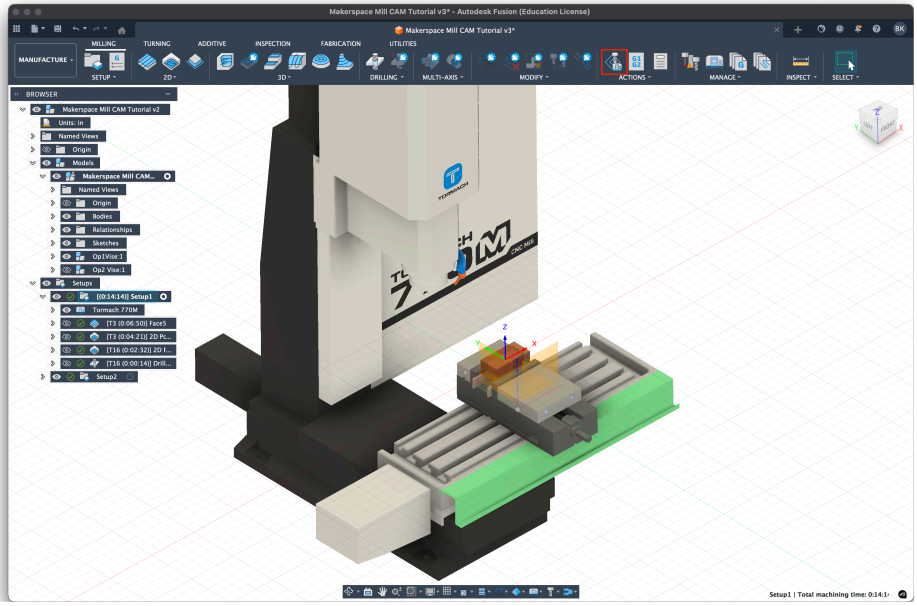


Figure 33: Shown here is where the **Simulate with Machine** button is located

Once you enter the simulation environment, enable visibility for **Op1Vise**. On the right hand side of your screen, you can adjust the visibility and transparency of certain elements, such as the machine, the stock, and the model. Once the settings are set up how you want, click play and watch the simulation.

At the timeline at the bottom of the simulation environment, there will be red or yellow warnings if Fusion360 detects any errors. You should not see any errors.

5. Creating our Second Setup and Its Toolpaths

Next we will configure our second **setup** and its **toolpaths**. This will be largely the same as the first setup, except we will be milling the bottom of the stock this time.

5.1. Setup 2 (Op2)

Setup 2 will be largely the same as Setup 1; in fact, much of the data will auto populate for us. There are only a few things we have to change.

First, disable visibility of **Op2Vise** and of **Op1Vise**. Orient your screen so that the vise is facing upwards.

In the **Work Coordinate System** section, set **Orientation** to 'Z axis/plane & X axis.' Ensure that the point selected is the middle of the top face of the yellow stock box. If needed, click on the tip of the blue Z-Axis arrow so that it is facing 'upwards.' If done correctly, it should look as follows:

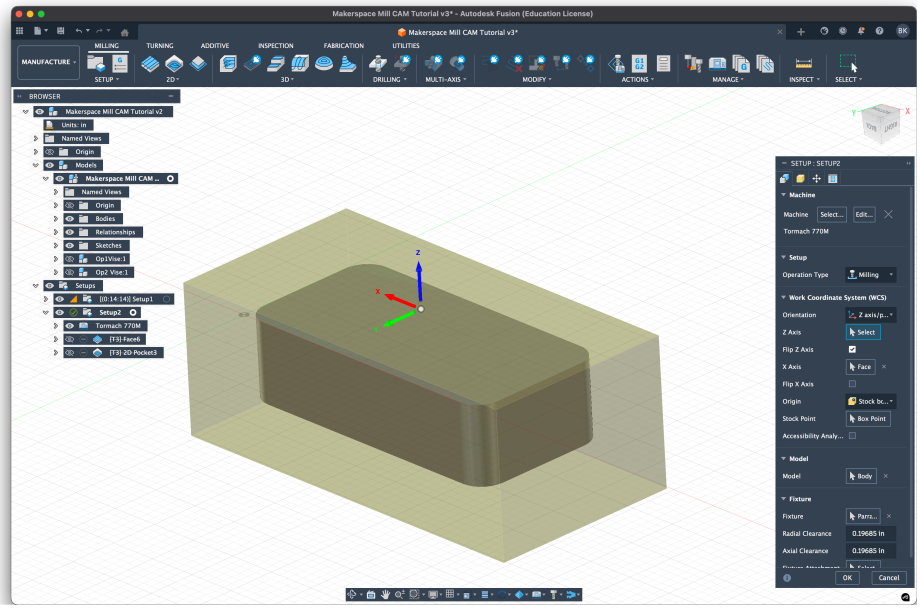


Figure 34: Shown here is what it should look like after successfully programing the **WCS** section in **Setup 2**

Next, we need to select the correct **Fixture**. Follow the same steps as in Section 4.1 for selecting the correct components of the vise. Ensure you are using **Op2Vise**.

Lastly, navigate to the **Stock** tab and ensure that **Mode** is set to 'From Previous Setup,' and **Continue Rest Machining** is enabled. This will use the leftover stock from the previous setup in this stock. It should look as follows when complete.

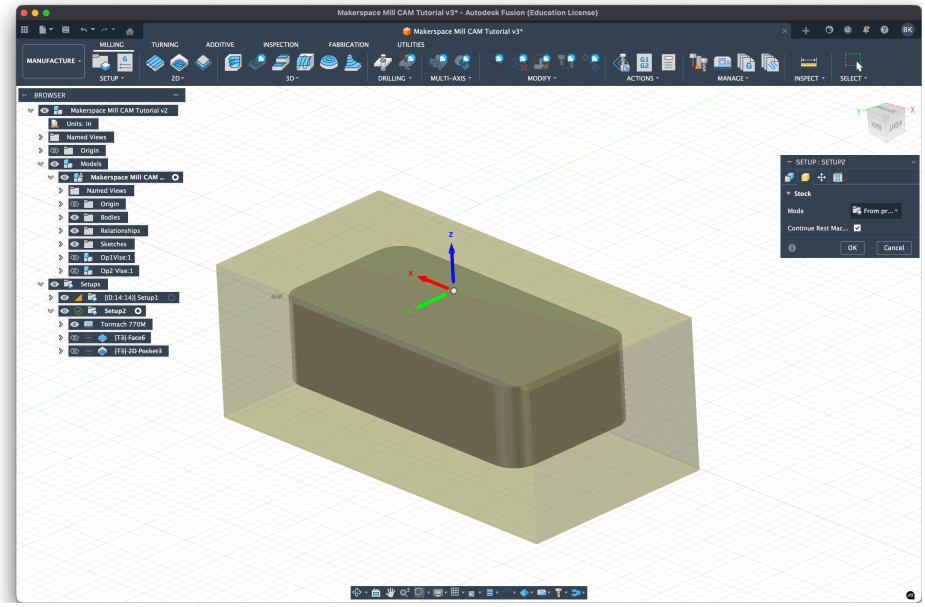


Figure 35: Shown here is what it should look like after successfully programing the **Stock** tab in **Setup 2**

If this is all completed correctly, click **OK** at the bottom of the **Setup2** window. We are now ready to program the first operation.

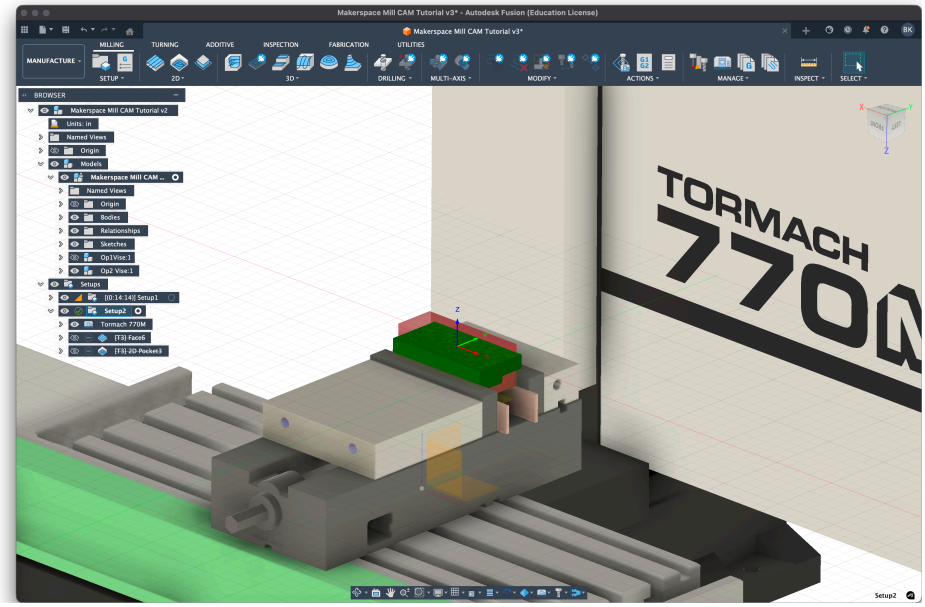


Figure 36: Shown here is what it should look like after successfully programing **Setup2**

5.2. 2D Face

Next, we will be facing this side of the stock. Follow the exact steps from Section 4.3 to do this. When done correctly, it should look as follows:

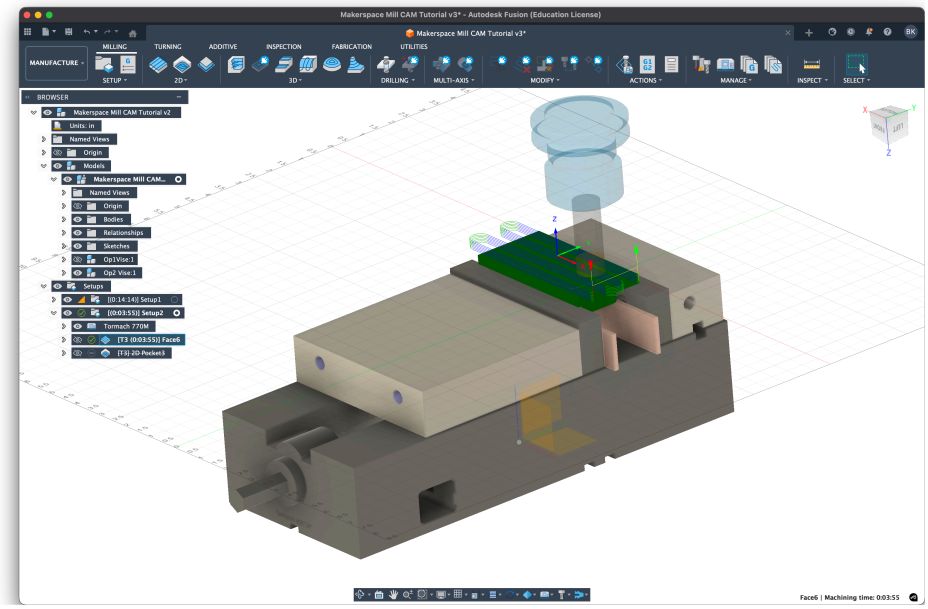


Figure 37: Shown here is what it should look like after successfully programing the **2D Face** in **Setup2**

5.3. 2D Perimeter Pocket

For our next operation, we will be doing a perimeter pocket, the similar to the one from **Setup1**. Follow the steps from Section 4.4.2 to create the perimeter pocket for **Setup2**. However, ensure to add a **–0.4in** offset to the **Bottom Height** in the **Heights** tab. If done correctly, it should look as follows:

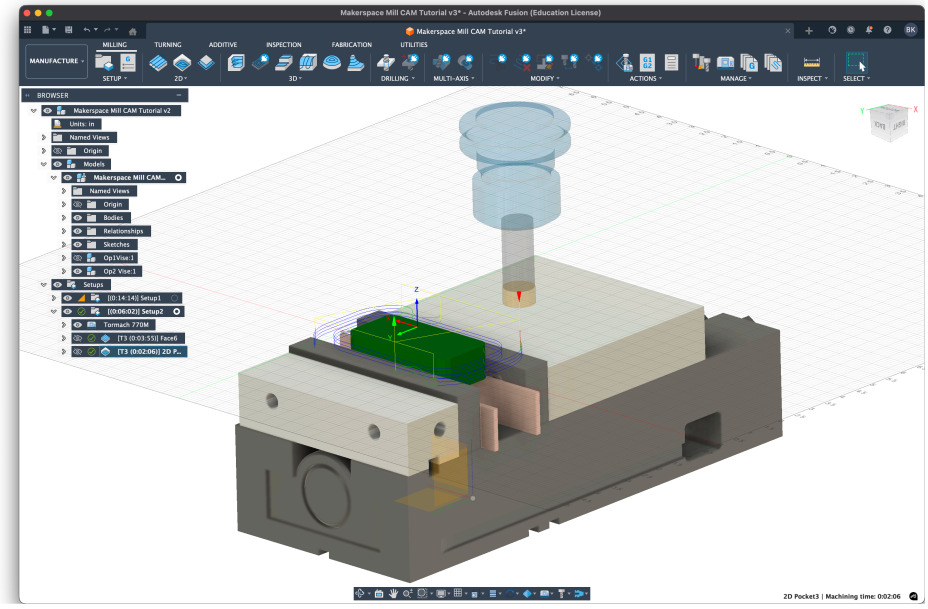


Figure 38: Shown here is what it should look like after successfully programing the **2D Perimeter Pocket** in **Setup2**

It is good practice to simulate this operation to ensure that we do not collide with the vise. If your simulation shows that you do, double check your **Heights** and **Geometry** tabs. You may

see some warnings that you “Overtraveled in Y Axis.” You can ignore these errors, as they will be fixed once you actually zero the stock in the mill.

5.4. Chamfer

Next, we are going to chamfer the bottom edge of the part. To do this, select the **2D Chamfer** toolpath under the **2D** menu in the toolbar.

Once open, select **Tool 22, the ∅0.25 45° Chamfer Bit.** Under the **Geometry** tab, select the perimeter edge of the face currently oriented upwards. If done correctly, it should look as follows:

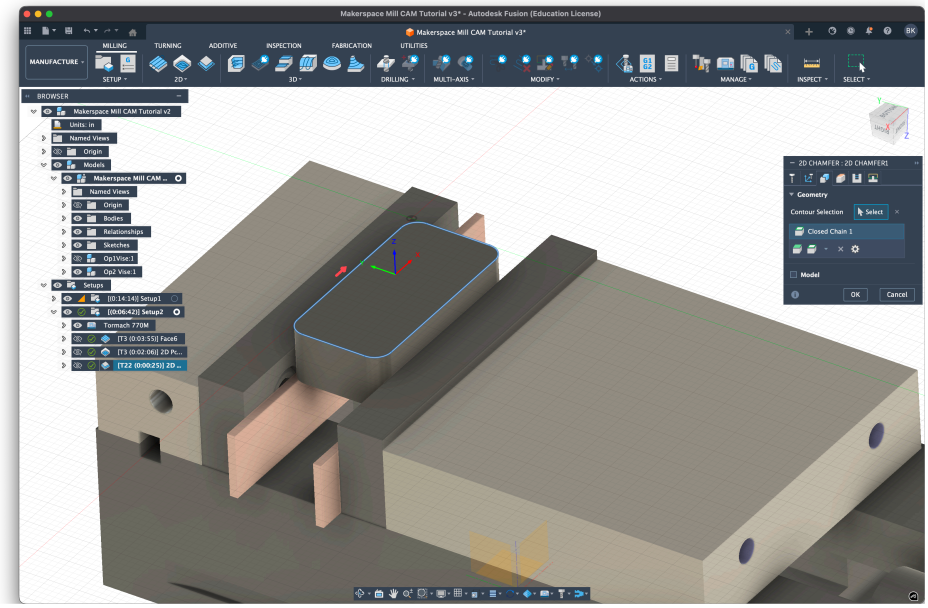


Figure 39: Shown here is what it should look like after successfully programing the **Geometry** tab in the **Chamfer** toolpath

Next, navigate to the **Passes** tab. In the ‘Chamfer’ section, ensure that **Chamfer Width** is set to **0.04in**, **Chamfer Tip Offset** is **0.04in**, and that **Chamfer Clearance** is **0.025in,** as shown below:

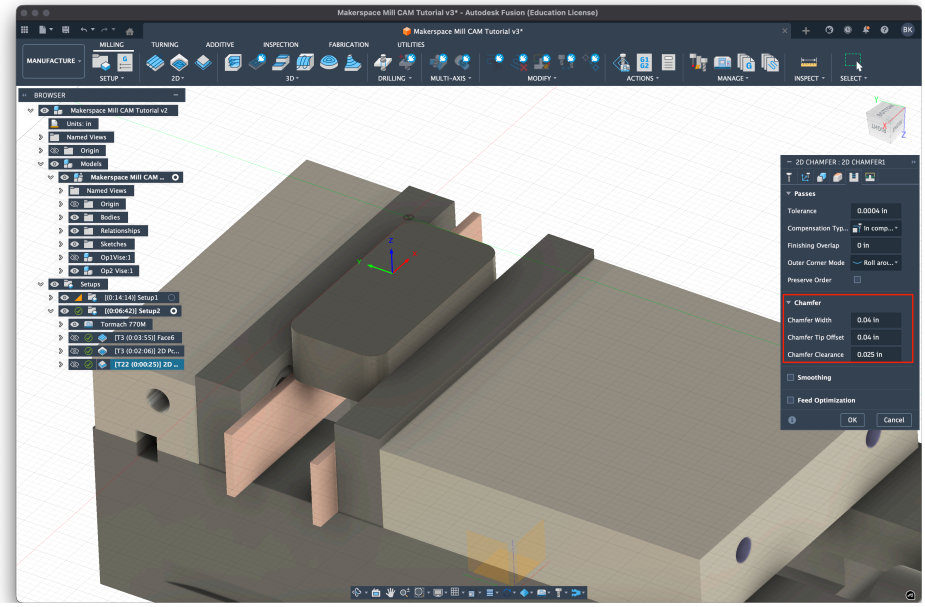


Figure 40: Shown here is what it should look like after successfully programing the **Passes** tab in the **Chamfer** toolpath

Once these steps are complete, click the **OK** button at the bottom of the **Chamfer** toolpath window. Your toolpath should look as follows.

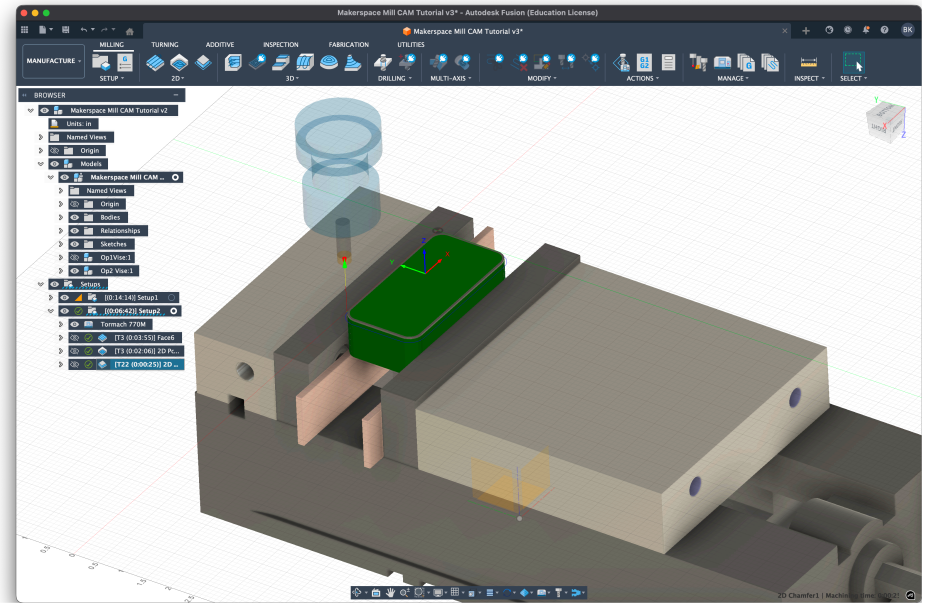


Figure 41: Shown here is what it should look like after successfully programing the **Chamfer** toolpath

5.4.1. A note on Programing Chamfers

There are technically two ways to program chamfers in CAM. The first is what is shown above, where you program in a chamfer on an edge that **does not** have a pre-designed chamfer.

The other way to do is the pre-design the chamfer to your specifications, then program the chamfer around that.

5.5. End of Setup 2

This is the end of programming for Setup Two. As we did with setup one, use the **Simulate with Machine** functionality to preview your program.

6. Exporting your Program as a .nc File to the Mill

6.1. Exporting from Fusion360

Next, we will export our program to a USB-A thumb drive to run it on the computer. To do this, click on the **Post Process** button in the toolbar. Ensure that you have Setup1 selected when you click **Post Process.**

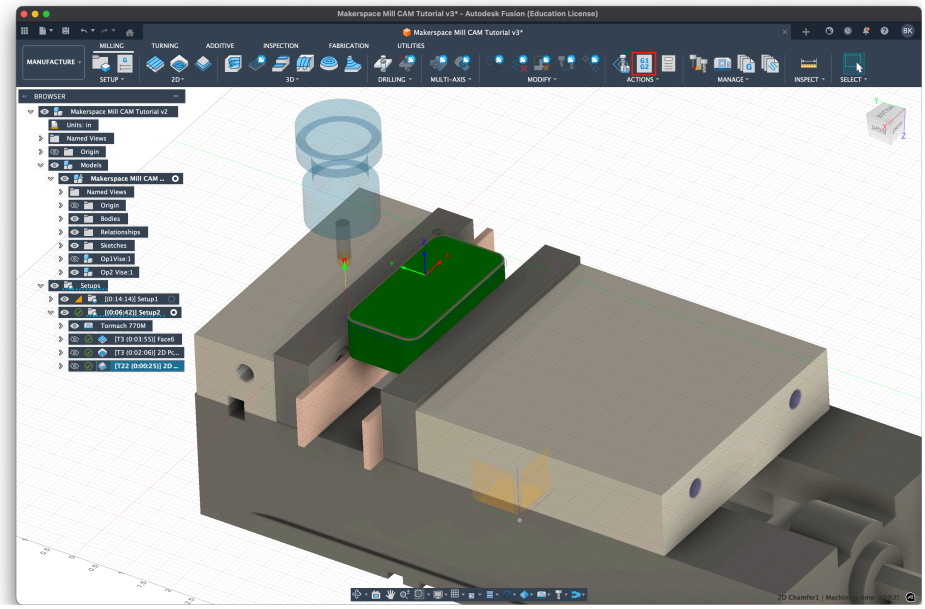


Figure 42: Shown here is where the **Post Process** button is located

Once in the **Post Process** menu, ensure that the machine selected is the **Tormach 770M**, and that the **Post** is set to **Tormach PathPilot**. This is the processing engine that the Tormach PCNC770 Mill uses. Lastly, ensure that you name the file so that it is identifiable on the USB-B thumb drive.

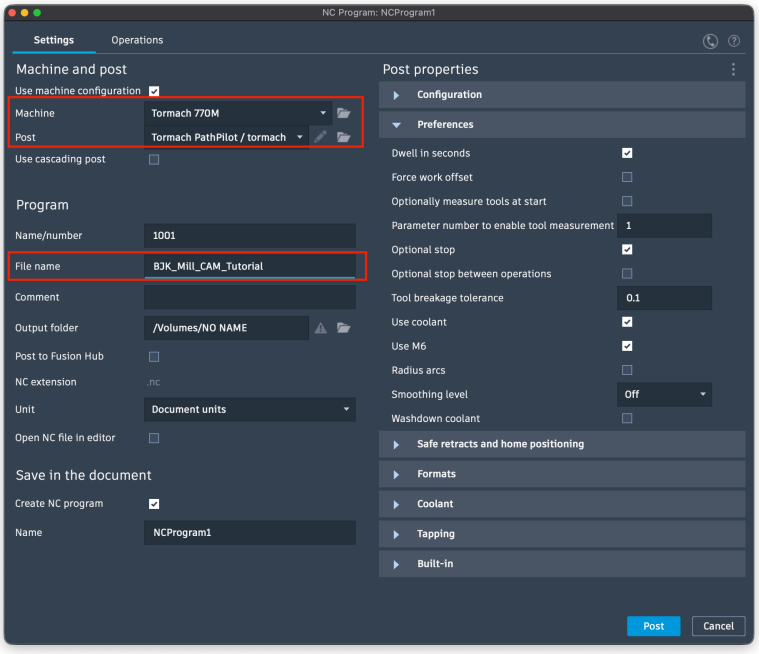


Figure 43: Shown here is the correctly configured **Post Process** menu

Once these steps are complete, select the output folder as the USB-B thumb drive and click **post**. Repeat these steps to export **Setup2** onto the thumb drive.

6.2. Loading File onto Tormach Mill

Once you have both .nc files loaded onto the USB-B thumb drive, take the drive and plug it into the USB port on the Tormach PCNC770 Mill.

Once the mill is powered on, and all axes are referenced, navigate to the **Files** tab on the computer. There, you should see a section listing all the files on the USB thumb-drive on the right side, as well as a section listing all of the folders that are local to the computer. Locate your folder and **double click** to open it. Then select your .nc file and click the **copy** button to copy it from the USB thumb drive onto the local storage. do this for both of your .nc files. Then when complete, double click on the **Setup1 .nc file** to open it.

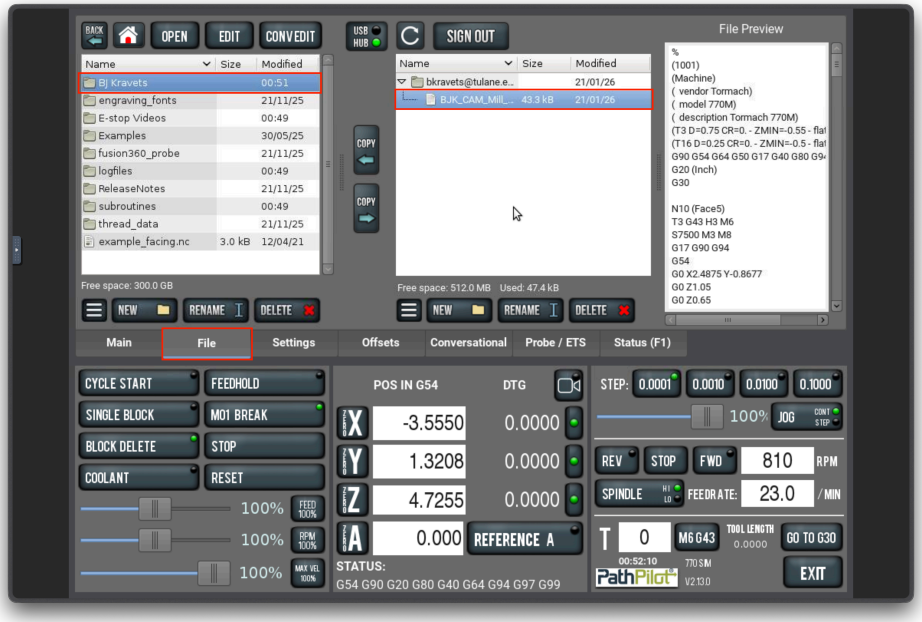


Figure 44: Shown here is the **Files** tab on the Tormach PathPilot computer

7. Running your CAM Programs

Once the first .nc file is opened, all that is left to do before you start milling is zeroing the machine and inserting the correct tool.

7.1. Zero the Machine

The next step is to zero the machine according to where we set the **Work Coordinate System** in Sections 4.1 and 5.1 (refence Section 4.1 for **Setup1** and Section 5.1 for **Setup2**). Zero the machine using the Haimer Probe (**Tool 5**) using the methods taught in the basic CNC Mill Training.

Important: Make sure that you set the top surface of the stock as **Z = 0.1in,** not as Z = 0in. This is because we have our origin point set as the top surface of the *model,* not the top surface of the stock.

7.2. Insert the First Tool

Once the machine is zeroed on the X, Y, and Z axes, you need to insert the first tool. For us, we will be using **Tool 3, the Shear Hog** first. Once the shear hog is installed, set the **Feed Rate** to **20%** and the **Max Velocity** to **20%**. We do this as a general precaution at the start of each toolpath.

7.3. Cycle Start

We are now ready to start the program. Click **Cycle Start** and watch the machine. If the toolpath seems to be working properly, increase **Feed Rate** and **Max Velocity** back to **100%**. However, if there are two operations that use the same tool back-to-back – as ours does – the machine **will not** stop between each toolpath, so, it is the duty of the machine operator to watch for the start of a new toolpath and to again decrease the **Feed Rate** and **Max Velocity** to **20%** until you can verify the toolpath is safe.

7.4. Tool Changes

When the program requires a tool that is not currently installed, it will stop running and a blue banner will appear asking for a specific tool. This is a **safe** time to open the machine and insert a new tool. Once the new tool is in, press **Cycle Start**, and the next toolpath will run. It is not necessary to fill in the new tool number manually as it is in conversational mode – the mill will automatically update this number for you

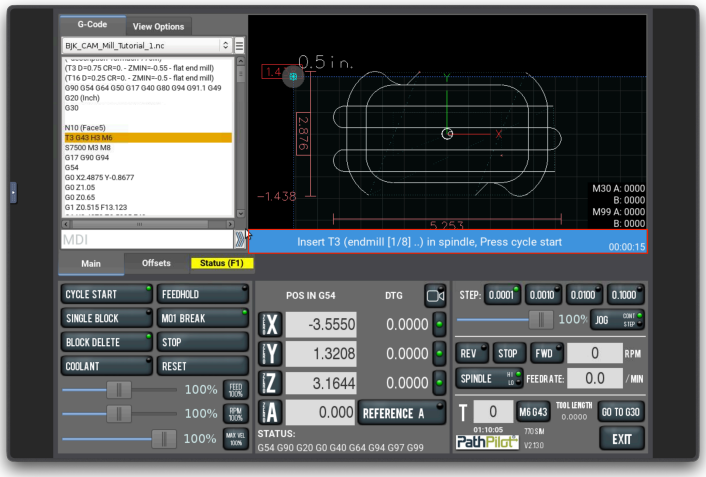


Figure 45: Shown here is the **blue banner** that will appear when the machine requests a new tool

7.5. Continue the Process & Repeat

Follow the steps of starting and observing each new toolpath and inserting new tools when necessary until **Setup1's .nc file** is complete. Once the first setup is complete, remove the stock, and reorient for **Setup2**. Follow the steps to re-zero the stock, and run the second **.nc file**. Once the second setup is ran and completed, the part is complete.

8. Conclusion

Once all setups are complete, remove your part. You are now done with you CNC Mill CAM Training is complete. Extra resources are avilable at makerspace.tulane.edu°. If you have any further questions, please reach out to any FabTechs via email or in the MakerSpace, or the author of this document bkravets@tulane.edu°.

Index of Figures

Figure 1 Highlighted are the two files that need to be downloaded 5
Figure 2 Highlighted are the steps to navigate to the manufacture workspace 5
Figure 3 Highlighted is the **Tool Library** button 6
Figure 4 Highlighted is where the user should right click to import the **.tool** file 6
Figure 5 Highlighted is **Data Panel Button** 7
Figure 6 Highlighted is **upload** button that we will use to import the vise **.f3d** file 7
Figure 7 Example of what your Fusion360 document should currently look like 8
Figure 8 Highlighted are the steps to access the **Parameters** window 9
Figure 9 Shown here are the proper parameters for our use 9
Figure 10 Shown here is what the current setup should look like 10
Figure 11 Shown here is what the current setup should look like 11
Figure 12 Shown here is what the current setup should look like 11
Figure 13 Shown here is what the current setup should look like 12
Figure 14 Shown here is should be selected when setting up the **Fixture** section 13
Figure 15 Shown here is what the current setup should look like 14
Figure 16 Shown here is what the current setup should look like 14
Figure 17 Shown here is what the current setup should look like 15
Figure 18 Shown here are the tabs in each toolpath operation window 16
Figure 19 Shown here are the steps to select a tool in a toolpath 16
Figure 20 Shown here are the steps to program the heights for a **2D Facing** operation . . 17
Figure 21 Shown here are the steps to program the passes for a **2D Facing** operation. . . . 17
Figure 22 Shown here is what the screen should display after sucessfully programing a **2D Facing** operation. 18
Figure 23 Example of a **Perimeter Pocket** 19
Figure 24 Example of a **Interior Pocket** 19
Figure 25 Shown here is what the screen should display after successfully programing the **Geometry** tab of a **2D Perimeter Pocket** 19
Figure 26 Shown here is what the screen should display after successfully programing the **Heights** tab of the **2D Pocket** 20
Figure 27 Shown here is what the screen should display after successfully programing the **2D Perimeter Pocket** 20
Figure 28 Shown here is what the screen should display after successfully programing **Geometry** tab of the **Interior 2D Pocket** 21
Figure 29 Shown here is what the screen should display after successfully programing the **2D Interior Pocket** 22
Figure 30 Shown here is where to click to select the **Drilling** toolpath 22
Figure 31 Shown here is what it should look like after successfully selecting the four ∅ 0.25in holes in the **Geometry** tab of the **Drilling** toolpath. 23
Figure 32 Shown here is what it should look like after successfully programing the **Drilling** toolpath 23
Figure 33 Shown here is where the **Simulate with Machine** button is located 24
Figure 34 Shown here is what it should look like after successfully programing the **WCS** section in **Setup 2** 25

Figure 35 Shown here is what it should look like after successfully programing the **Stock** tab in **Setup 2** 26
Figure 36 Shown here is what it should look like after successfully programing **Setup2** . 26
Figure 37 Shown here is what it should look like after successfully programing the **2D Face** in **Setup2** 27
Figure 38 Shown here is what it should look like after successfully programing the **2D Perimeter Pocket** in **Setup2** 27
Figure 39 Shown here is what it should look like after successfully programing the **Geometry** tab in the **Chamfer** toolpath 28
Figure 40 Shown here is what it should look like after successfully programing the **Passes** tab in the **Chamfer** toolpath 29
Figure 41 Shown here is what it should look like after successfully programing the **Chamfer** toolpath 29
Figure 42 Shown here is where the **Post Process** button is located 31
Figure 43 Shown here is the correctly configured **Post Process** menu 31
Figure 44 Shown here is the **Files** tab on the Tormach PathPilot computer 32
Figure 45 Shown here is the **blue banner** that will appear when the machine requests a new tool 34