3D Modeling
to Automated Machining

Fusion 360™ and the Tormach Slant Pro 15L™ Lathe

Dylan Lucia

Tulane University MakerSpace

Created 03/19/2018. Modified 05/21/2018.

Table of Tools

Below is a table of the tools that, as of the creation of this training, exist for use in the Tulane MakerSpace. This also shows you the distinction between tool numbers, easily computed with the following equation: $T_{\text{CAM}} = 50 + T_{\text{Conversational}}$. Refer to this table as necessary.

Tool Name	**Tool Number**		**Use**
	Conversational	*CAM*	
RH Turning	01	51	General front-to-back profiling, facing
RH Parting	02	52	Grooving, parting
RH Threading	03	53	Creating OD threads
Center Tool	04	54	Center drilling
XL RH Turning	05	55	Front-to-back profiling
LH Turning	06	56	General back-to-front profiling
	07	57	
	08	58	
	09	59	
Jacobs Chuck	10	60	Center boring

List of Figures

Table of Contents

Chapter 1

Creating Toolpaths

1.1 Introduction

This training course will serve as a basic walkthrough for transitioning from using Conversational mode in Tormach's PathPilot application on the lathe computer to using CAM on the MakerSpace's Tormach 15L Slant Pro Lathe.[*] CAM stands for Computer Aided Machining, and encompasses many different machines and methods. Essentially, CAM allows automated machining of many different parts on many different types of machines. There are always more methods and tricks to learn in CAM, and after completing this course, you are encouraged to explore online forums, books, and other resources to learn more methods and unique CAM possibilities.

You may wonder why you would choose to use CAM versus Conversational. Sometimes, you may want to just use Conversational instead of CAM. CAM requires 3D modeling a part in software such as Fusion[†] or Rhino,[‡] and then creating the machining tool paths. These must then be brought to the lathe, at which point there is very little left for you to do (we'll get to this later). As you can probably guess, this is far too much effort if you just want to square up the face of cylindrical stock material or something similar.

[*] http://bit.ly/LatheCAM1
[†] http://bit.ly/LatheCAM2
[‡] http://bit.ly/LatheCAM3

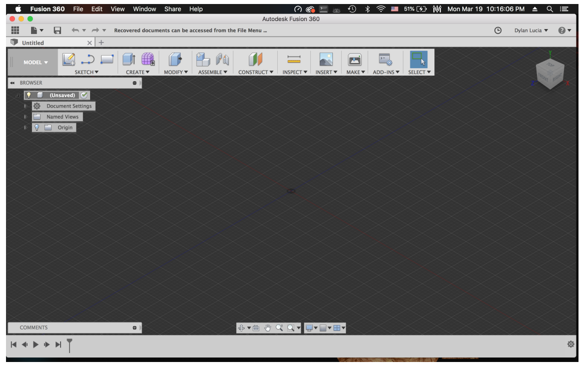


FIGURE 1.1: A BASIC FUSION WINDOW.

However, CAM offers many tooling options that are impossible to execute with just Conversational. For example, one could create a part with complex bezier curves which are not possible in Conversational. For simplicity, and to show you the similarities/differences to Conversational, we will be making a part similar to the part created in Tulane MakerSpace's initial lathe training.[§]

To begin, download and install the following files. First, install the 3D modeling software Fusion 360. In the future, you may choose to use other similar software,[¶] but for the purposes of this training, you will utilize Fusion.[‖] Fusion is free for those in University.

- Navigate to Autodesk's website[**] in your web browser and create an account (or sign in if you already have an Autodesk Education account).
- Use your University information to successfully get access to Fusion at no cost.
- Once downloaded, install the software.
- Open Fusion. You should be met with a screen similar to the one shown in Figure 1.1.

[§] http://bit.ly/LatheCAM4

[¶] http://bit.ly/LatheCAM5

[‖] Note that because Fusion (and most other software nowadays), is subscription-based, your interface may differ slightly from the one shown in this tutorial. This is because subscription-based software is constantly updating, and as such, the version used in this tutorial (2.0.3803), will likely be different from yours.

[**] http://bit.ly/LatheCAM6

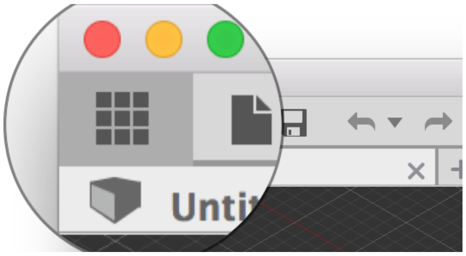


Figure 1.2: Show Data Panel.

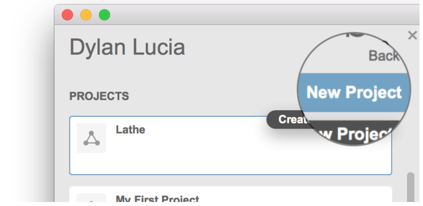


Figure 1.3: New Project.

Figure 1.4: Enabling Cloud Libraries.

1.2 Setting up Fusion for CAM

- Click on Show Data panel, Figure 1.2. Click on the blue New Project button, Figure 1.3. If you do not see this button, you may already be within a folder or project. Look for the back arrow button in the top left, and use it to navigate to the root level of the Data Panel.
- Enter a name, such as `Lathe Project`. This project will be for lathe-specific parts, rather than 3d-printed or milled ones.
- Navigate to Fusion Preferences. To do so, click on your name in the upper right corner of Fusion, and select Preferences. On the left side of the window, select CAM. Check the box next to Enable Cloud libraries, Figure 1.4. Then select OK to close the preferences window.

Navigate to this training's page[††] on the MakerSpace's Wiki and download the 3D part file, the digital tool library, and the post-processor. The part is the model you will be creating using CAM. The digital tool library allows you to easily install the digital tools we have for the lathe into your Autodesk account. The post-processor is the coded instructions that Fusion uses to convert the CAM paths into G-code that can be read by our lathe.

- Return to the Data Panel and if you are not currently in your Lathe Project, open it by double-clicking it.
- Click Upload, Figure 1.5, and navigate to and select the 3D part file you downloaded from the MakerSpace Wiki. It should appear with a preview in your Lathe Project folder.

Now, open your web browser, and navigate to Autodesk's cloud system,[‡‡] Autodesk A360, to sign in and open up your A360 account.

- Open the *Assets* library, and then the *CAMPosts* folder.
- Drag/drop or upload the post-processor file (tormach pathpilot turning.cps) you downloaded from the MakerSpace Wiki into this folder. Once it has uploaded successfully, return to the *Assets* folder.
- Open the *CAMTools* folder and upload the digital tool library file (MakerSpace Turning.tools) you downloaded from the MakerSpace Wiki. Once it has uploaded successfully, you may close your web browser.

Return to the Fusion application. This is the end of this important one-time setup process. You should not have to do this again.

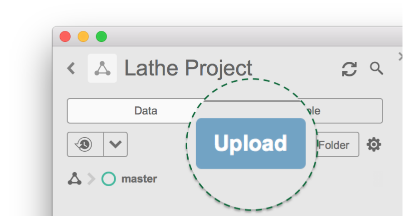


FIGURE 1.5: PROJECT UPLOAD BUTTON.

[††] http://bit.ly/LatheCAM7
[‡‡] http://bit.ly/LatheCAM8

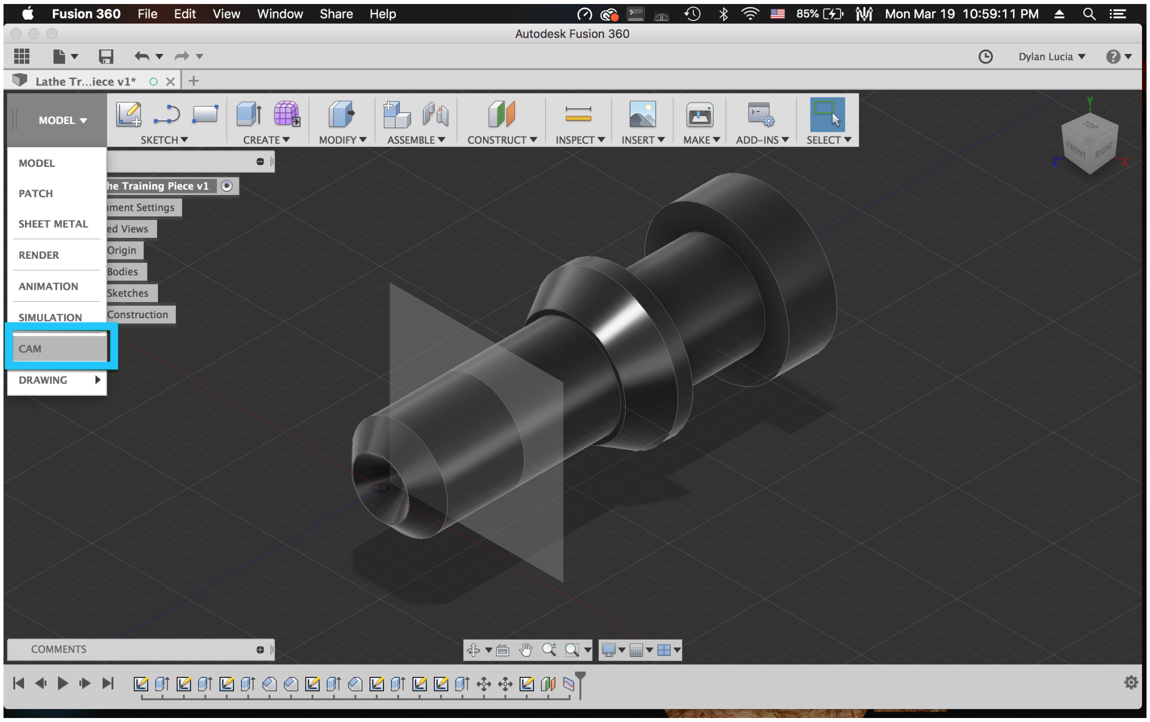


FIGURE 1.6: THE TRAINING PART IN THE FUSION WINDOW.

1.3 Creating the Stock

Fusion needs to know what you will be carving the model out of. This bulk physical material is called stock. Stock comes in many different materials, shapes, and sizes. Choosing the appropriate stock is an important aspect of the machining process. Often times, the type of material is dictated by structural or artistic specifications, such as how much stress a part will experience or the color of the part. Common materials include aluminum, wood, or plastics. An important step in the CAM process is letting Fusion know the specifications of the stock you will be machining.

- Open up the part file you uploaded to your project folder by double-clicking on it. It should look like Figure 1.6. Orbit around and inspect the part, getting to know it.
- Also ensure that your units are in imperial (inches). Do this by dropping down the *Document Settings*, Figure 1.7, and double clicking on the units and changing them.

Something else you may notice is that in the part, we have not modeled threads. This is because we will later create an operation that will define the threads, since threads themselves are not rotationally-symmetric at each z-axis point. Fusion will use the operation we create later to control the speed of the tool and spindle to create the threads automatically.

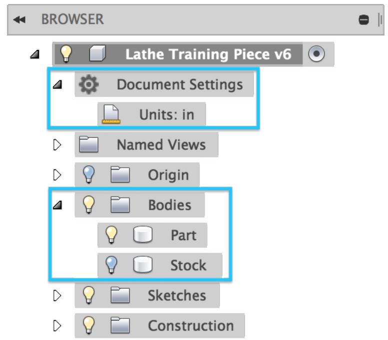


FIGURE 1.7: THE PROJECT BROWSER.

If you open the Bodies dropdown in the Project browser, shown in Figure 1.7, you can turn on the lightbulb next to the Stock body. This is a modeled body that represents the physical material (stock) we will turn the final part from, which is an important aspect of CAM. To create this part we will use 1” cylindrical stock. The length of the stock body modeled in Fusion does not matter, as long as it has about $^{1}/_{10}{}^{\text{th}}$ of an inch in front of the 3D part, and at least an inch past the back of the 3D part. You may notice through inspection of the sketches in the project that the stock body is actually 1.005” in diameter, which accounts for variation in physical stock material and gives us a little bit of a safety cushion.

1.4 The CAM Environment in Fusion

Right now, by default, Fusion is within the *Model* Workspace. We need to change to the *CAM* workspace to create our CAM operations.

- To change our Workspace from Model to CAM, click MODEL ▼ in the Ribbon, which opens up the workspace selector, and select CAM, shown in Figure 1.6. The biggest change you may notice is that the Ribbon at the top of the environment has changed to a different toolset.

- Use the View controller in the top right, Figure 1.8, to position your view to the top view such that your view matches Figure 1.9.

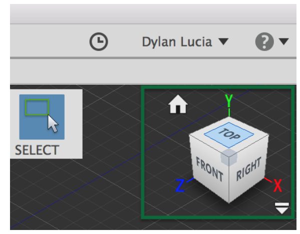


FIGURE 1.8: THE VIEW CONTROLLER.

This view orientation is helpful because it matches the orientation you see when operating the lathe, with the Z-axis horizontal and the X-axis vertical. It is important to note that the axis of rotational symmetry of the part is the Z-axis, because this is the axis that the lathe rotates the stock about. You should also note that the right-most point of the piece is exactly at the X-Y plane (the origin/0-point of the Z-axis). This makes the interface between Fusion and the lathe much simpler because you will not need to think about offsetting operations.

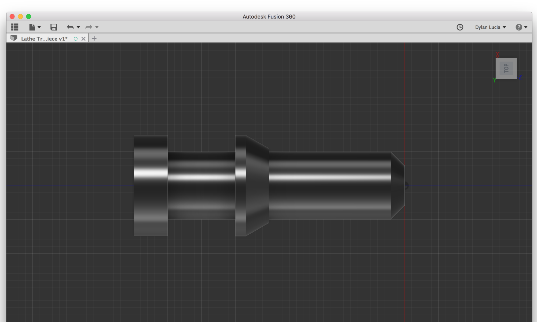


FIGURE 1.9: A GOOD VIEW FOR CREATING TOOLING PATHS.

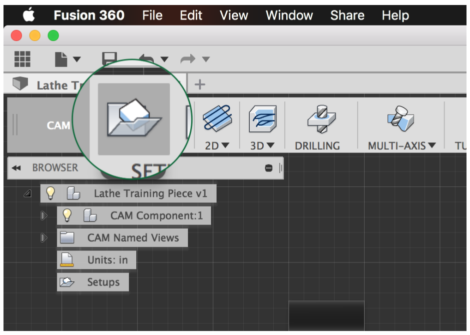


FIGURE 1.10: CREATING A NEW SETUP.

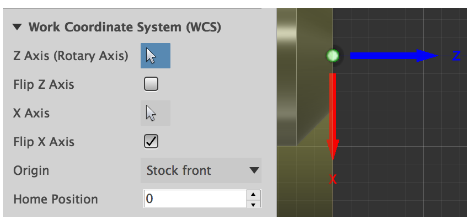


FIGURE 1.11: FLIPPED X-AXIS.

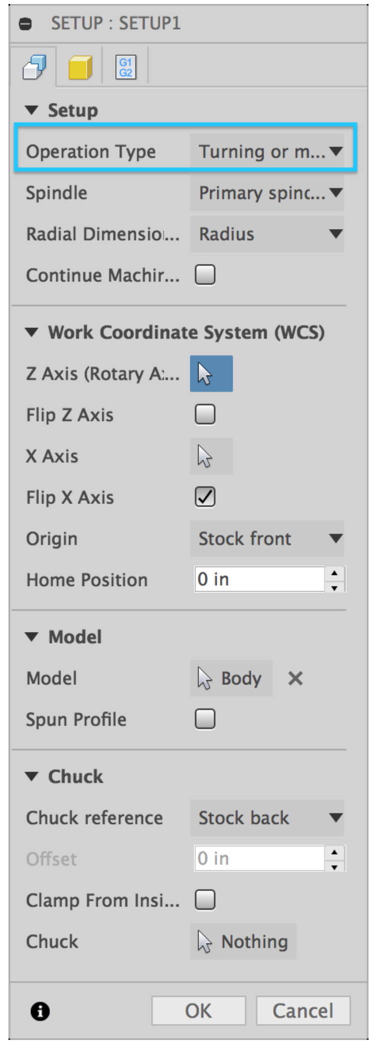


FIGURE 1.12: THE NEW SETUP'S SETTINGS DIALOG WINDOW.

1.5 Creating the CAM Setup

For any part, the first step in CAM is creating a new setup.

- On the Ribbon, click New Setup (it looks like a folder being opened), shown in Figure 1.10.
- Under the Setup tab, change the operation type to Turning or mill/turn, shown in Figure 1.12.
- Flip your X-axis under Work Coordinate System by checking the box so that it matches Figure 1.11.

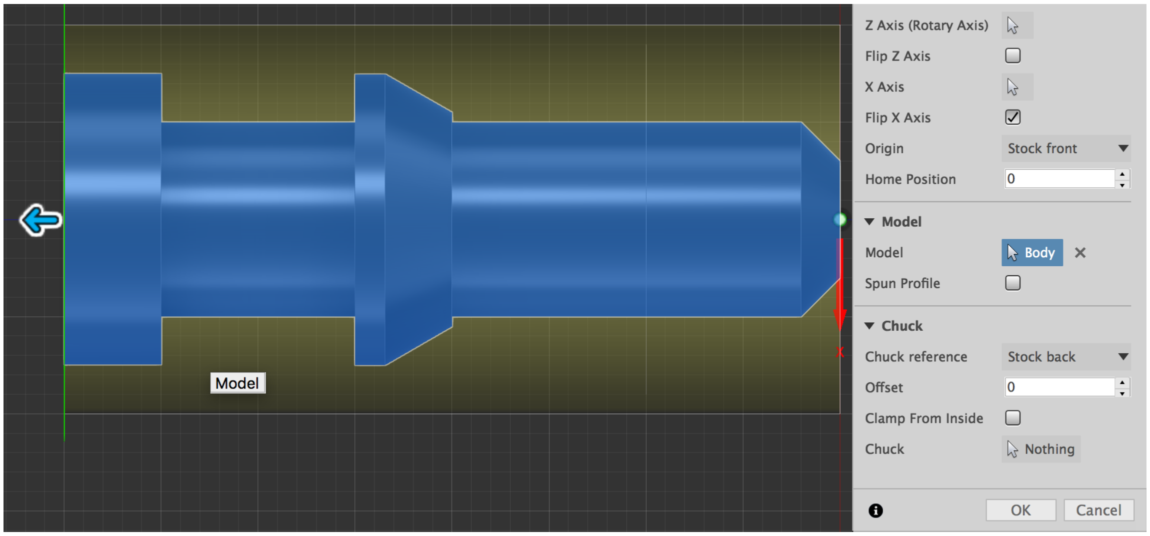


FIGURE 1.13: SELECTED MODEL FOR FUSION TO MACHINE.

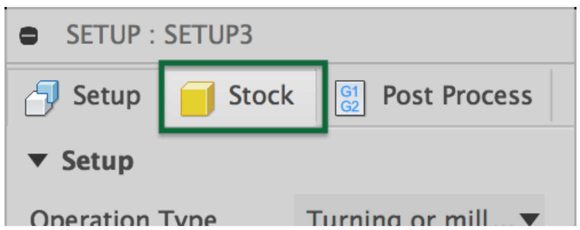


FIGURE 1.14: CHANGING THE SETUP SETTINGS TAB TO "STOCK."

You should see the red arrow on the model flip so that it faces down. This is because on our lathe, we approach from the bottom of the stock, rather than the top (which Fusion assumes by default). Ensure that the part model is selected under "Model."

- This can be done by clicking Model, and then selecting the body that is to be machined. This can be seen in Figure 1.13.

This tells Fusion that you want the tooling paths to be calculated based on this body.

- Select the Stock tab, shown in Figure 1.14.
- Select From solid under "mode."

This is where our previous modeling of the stock comes into play. Instead of defining the dimensions of the stock material in the stock settings here, the settings are extracted from the stock body that was previously modeled. This way, if we decide to change our stock, we can just change the parameters of the stock body, and still be able to visualize how the piece will fit within the stock.

- Click Body and navigate through the Project Browser until you find the Stock body, clicking on it to select it.
- Navigate to the Post-Process tab, and change "WCS offset" to 2, seen in Figure 1.15.

This setting tells the post-processor what coordinate system our lathe uses, and is *very* important to do for every setup you create because it tells the lathe where zero is in relation to its coordinate system and the G-code.

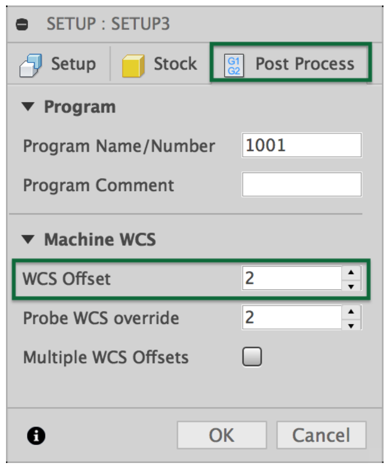
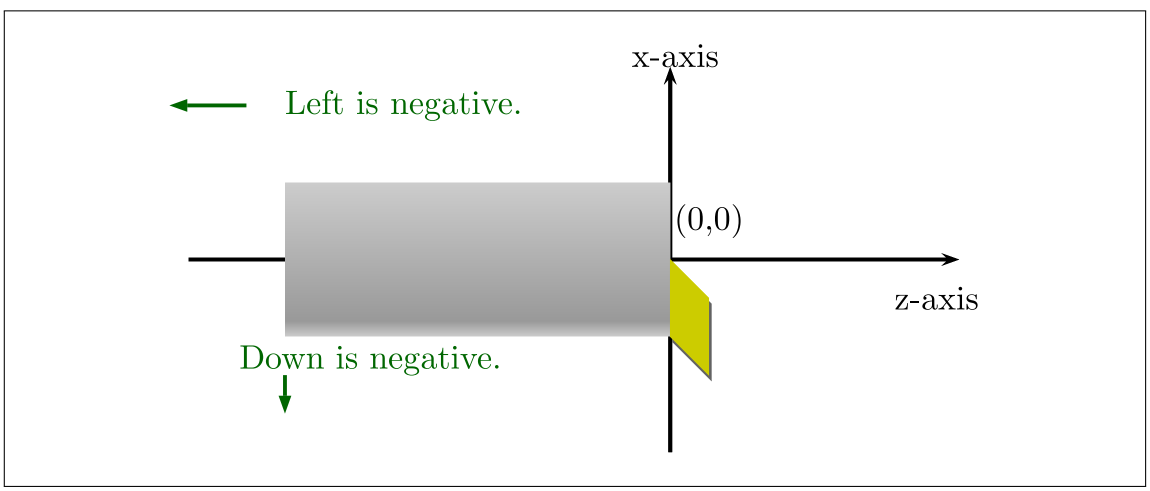


FIGURE 1.15: SETTING THE WORK COORDINATE SYSTEM (WCS).



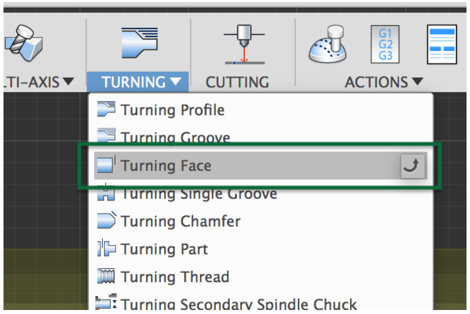


FIGURE 1.16: CREATING A TURNING FACE OPERATION.

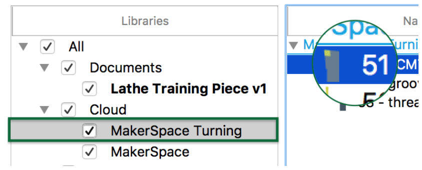


FIGURE 1.17: SELECTING A TOOL FROM THE CLOUD LIBRARY WE INSTALLED EARLIER.

1.6 Creating a Facing Operation

You've now created a Setup for the part, within which we will create the different machining operations. Take a minute to imagine the types of operations that will be necessary to create this part. You will need to face the stock, turn a profile of the model, address any deep grooves that the profile can't reach, and then part of the finished piece. First, you will need to create a facing operation on the stock. Facing stock involves shaving off a slight amount of the material from the front of the stock to square up the face of the stock. This is also useful because it removes any rough or unusable material.

- On the ribbon, under the Turning dropdown, select Turning Face, shown in Figure 1.16.
- In the resulting popup window, click Select... next to "Tool."
- In the next popup window, you will navigate to MakerSpace Turning and select Tool 51,[§§] our right-hand turning tool, Figure 1.17.
- Click OK.

[§§]You may be wondering why this is designated as Tool 51, rather than Tool 01, as it is in Conversational. This is because we have remade all of the same tools that already exist in the lathe, but added 50 to their tool number, designating these as the CAM-specific tools. While the physical tool is the same, the digital tool in the lathe's offset table and memory is different. This allows us to change the offsets for CAM separately than those for Conversational.

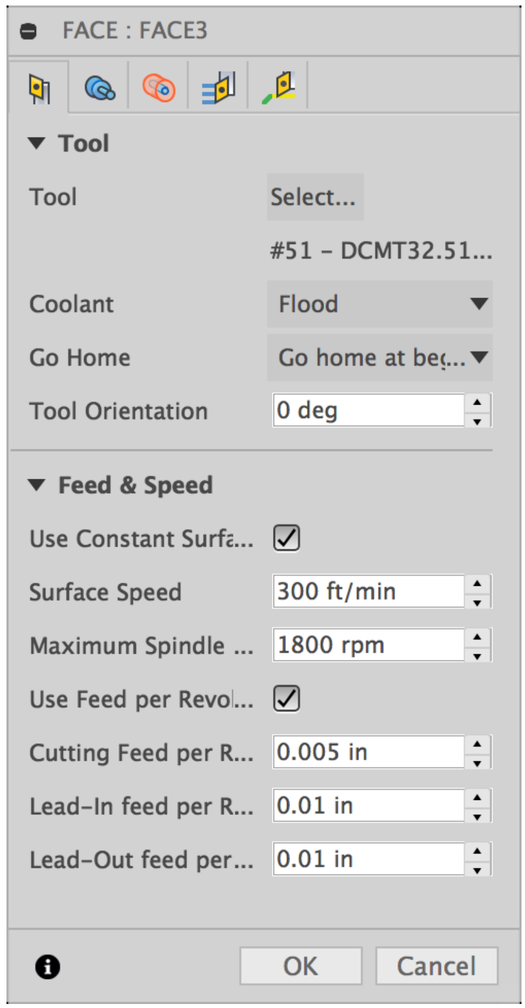


Figure 1.18: Setting the Tool's feed and speed settings.

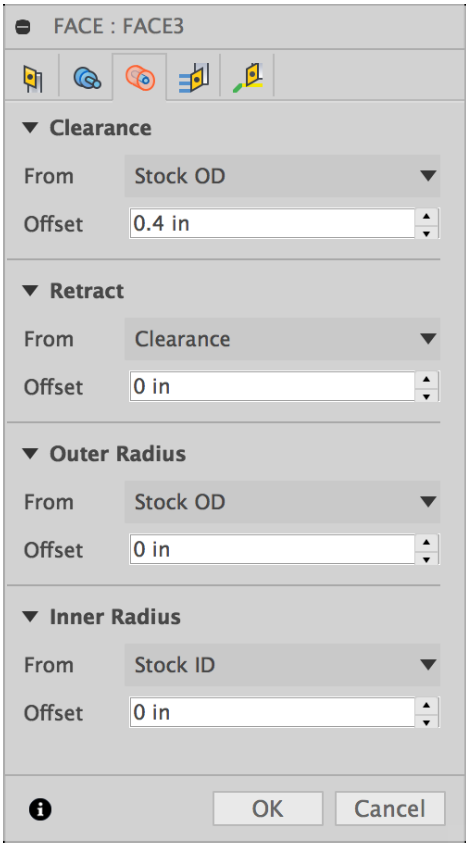


FIGURE 1.19: APPROPRIATE RADII SETTINGS FOR THE FACING PASS.

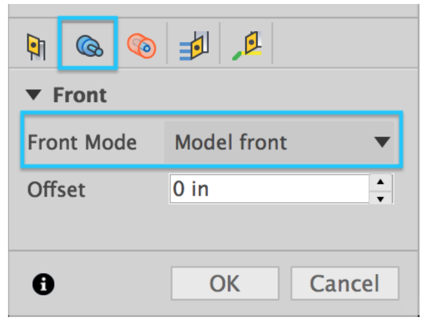


FIGURE 1.20: SELECTING THE FRONT OF THE MODEL AS THE REFERENCE FOR THE FACING OPERATION.

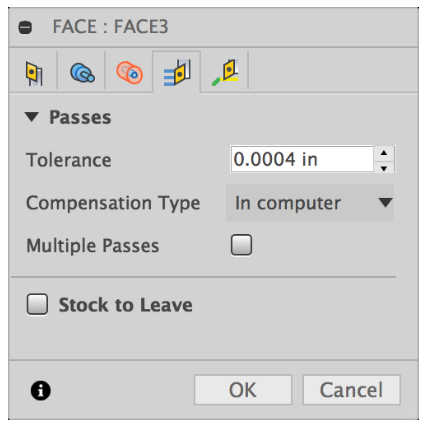


FIGURE 1.21: APPROPRIATE PASSES SETTINGS FOR THE FACING PASS.

Ensure that the rest of the settings for this tab match Figure 1.18. The biggest change here is the "Maximum spindle speed" (which limits the variable speed of the spindle), and "Cutting feed per revolution" (the distance the tool moves while cutting for each full revolution of the stock).

- Under the Geometry tab at the top of the window, ensure that Model Front is selected, Figure 1.20. This ensures that the facing operation will be created for the front of the model.
- Then ensure that you match your Radii and Passes tabs to Figures 1.19 and 1.21.
- Click OK to save these settings and then you've completed your setup for the facing operation.

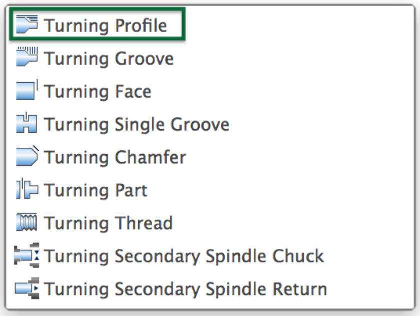


FIGURE 1.22: CREATING A TURNING PROFILE OPERATION.

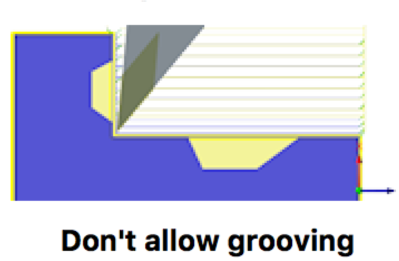


FIGURE 1.23: SCHEMATIC SHOWING NO GROOVING IN THE TURNING PROFILE OPERATION.

1.7 Creating a Turning Profile Operation

The next step is to create the turning profile of the piece.

- Return your attention to the Ribbon, and under the Turning tab, select Turning Profile, Figure 1.22.
- If it is not already selected, select Tool 51, the right-hand turning tool, just as you did for the creation of the Facing operation.
- Ensure your Tool and Passes tabs match Figure 1.24. These values are appropriate for a soft aluminum alloy, which you are going to use as stock material.

There are a couple of important settings to note:

- The first is Grooving, under the Tool tab. Ensure that this is set to Don't Allow Grooving. This prevents the tool from attempting to turn any types of grooves, shown schematically in Figure 1.23.
- Another important group of settings are the different speeds/feeds for roughing/finishing. These will vary with your stock material and tool, so be sure to research this appropriately for future projects.
- Click OK.

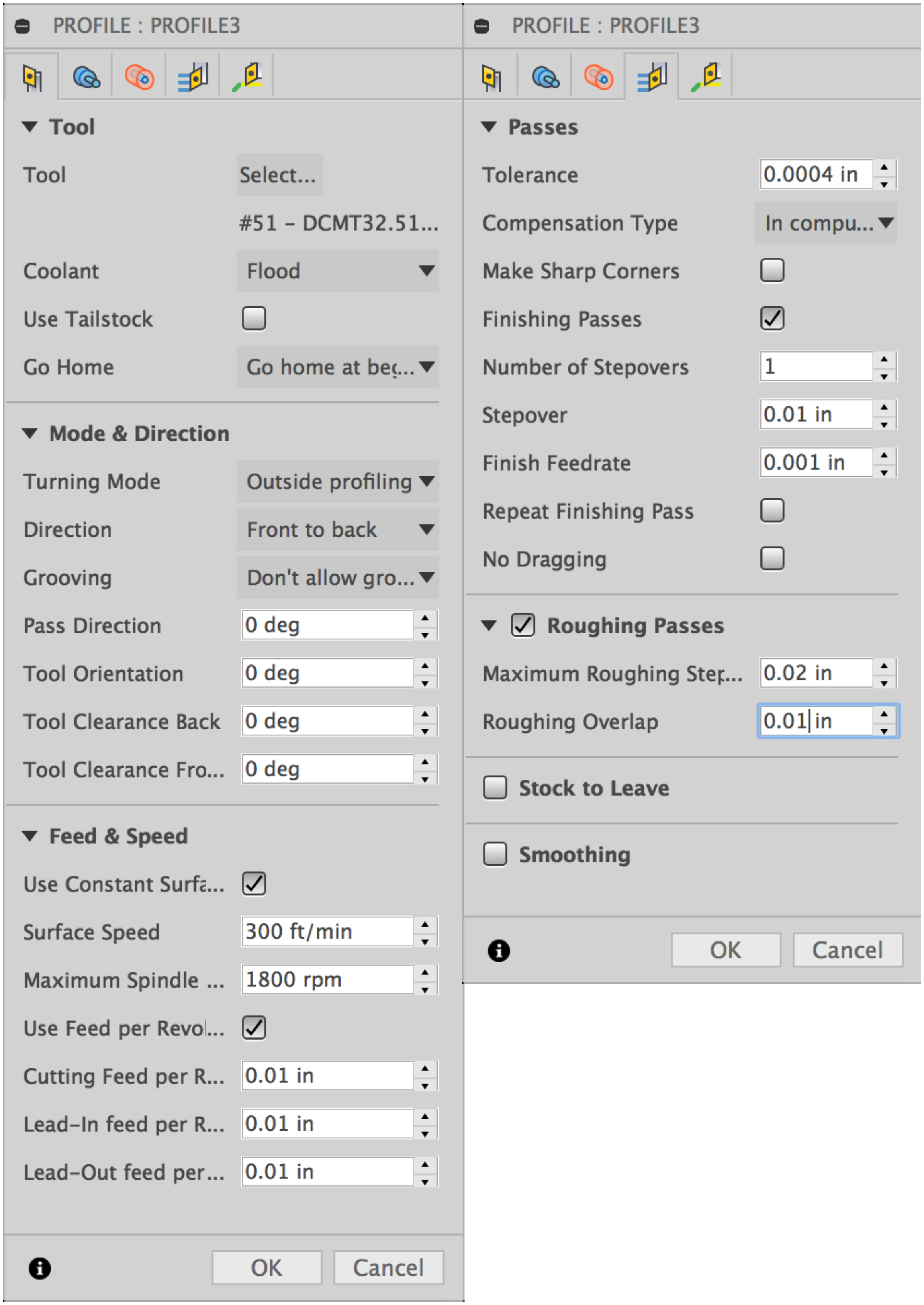


FIGURE 1.24: APPROPRIATE SETTINGS FOR THE PROFILE'S TOOLING PATH.

1.8 Checking Created Paths

If you click on [T51] Profile in the Project Browser (Figure 1.25), you can see the tooling paths created for this profiling step. Your screen should match Figure 1.26. The different lines you see around the part show different operations and movements. Yellow lines show generic tool movement, such as approaching, going home, or similar. Green lines show lead-in and lead-out movements. Blue lines show cutting movements.

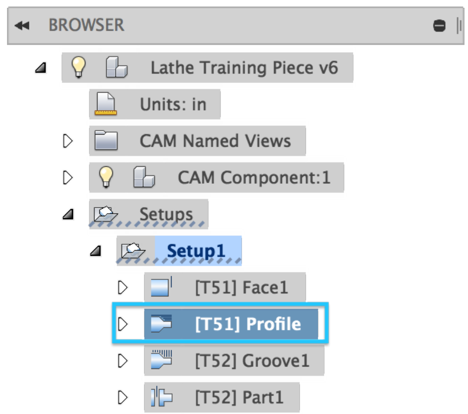


Figure 1.25: Opening the created Profile from the Project Browser.

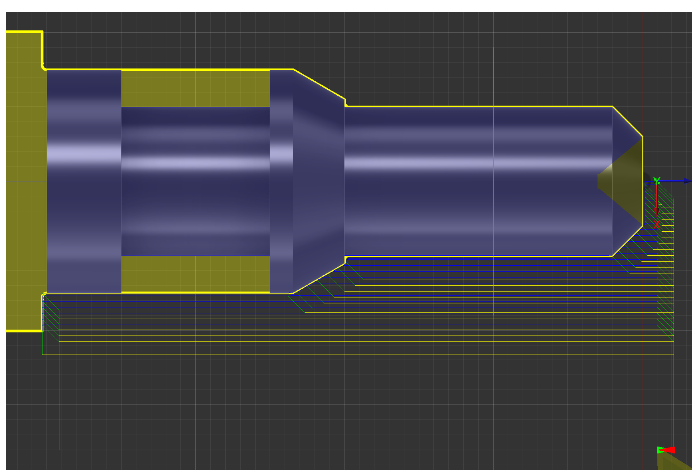


Figure 1.26: Tooling paths in the Fusion window.

1.9 Creating a Turning Groove Operation

Next, you will create the grooving operations. Earlier, you suppressed the profile operation from attempting to create any grooves to protect the physical tool from damage, and as such, you will now create those grooves with a grooving operation.

- Create a new Turning Groove operation, similar to how you did with the two previous operations, but this time selecting Turning Groove.
- For the tool, return to the MakerSpace Turning library, but this time select Tool 52.
- Match the settings for the Tool and Passes tabs to Figure 1.27.
- Click OK.

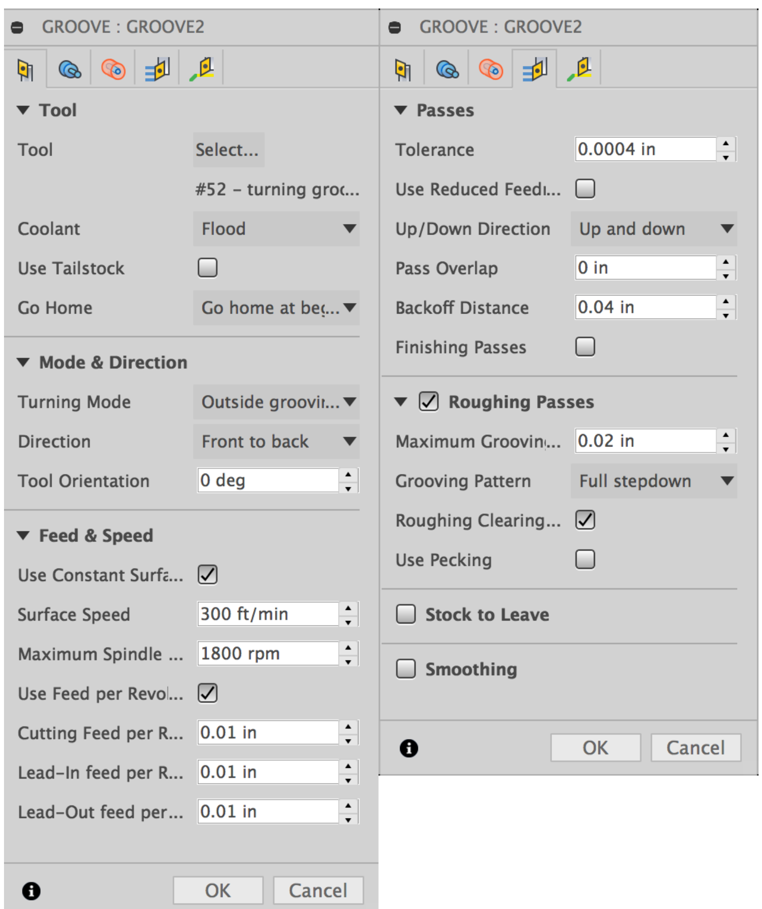


FIGURE 1.27: GROOVING OPERATION SETTINGS.

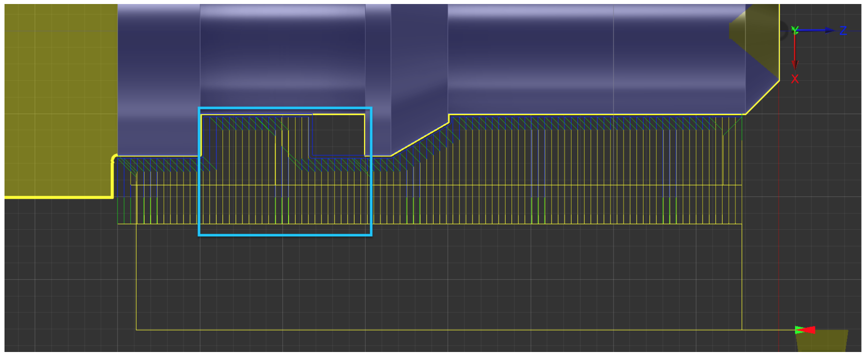


Figure 1.28: Grooving operation paths that need to be edited.

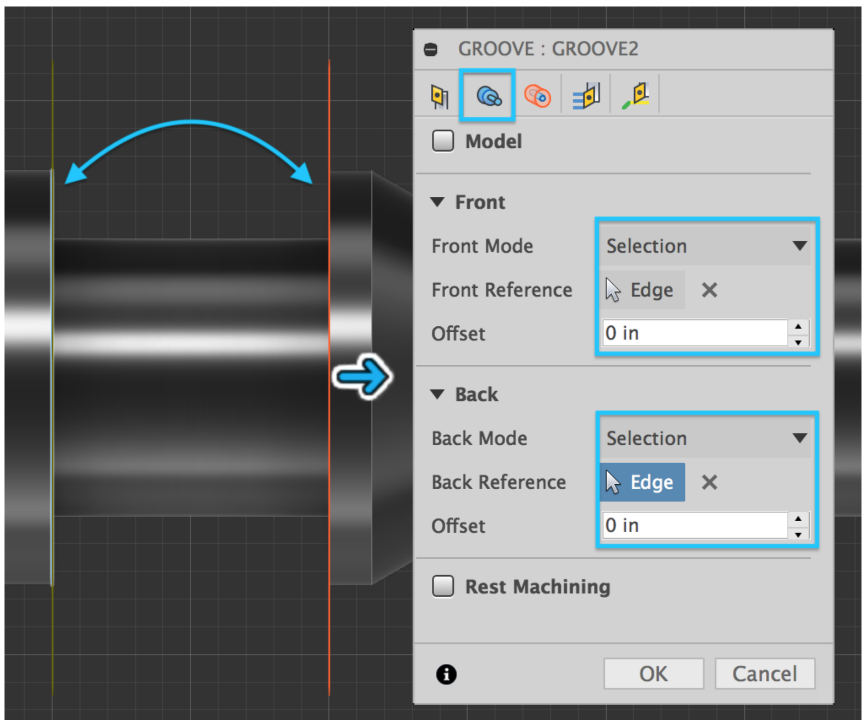


Figure 1.29: Appropriate settings to restrict the grooving tool to one specific area of the model.

You may notice that the grooving tool is attempting to groove the entire length of the part, Figure 1.28, rather than just the channel that we haven't grooved yet, marked in Figure 1.28 in blue. To fix this, we will edit the grooving path we just created.

- *Double*-click [T52] Groove in the Project Browser, and you will see the settings you were editing only moments ago.
- Go to the Geometry tab.

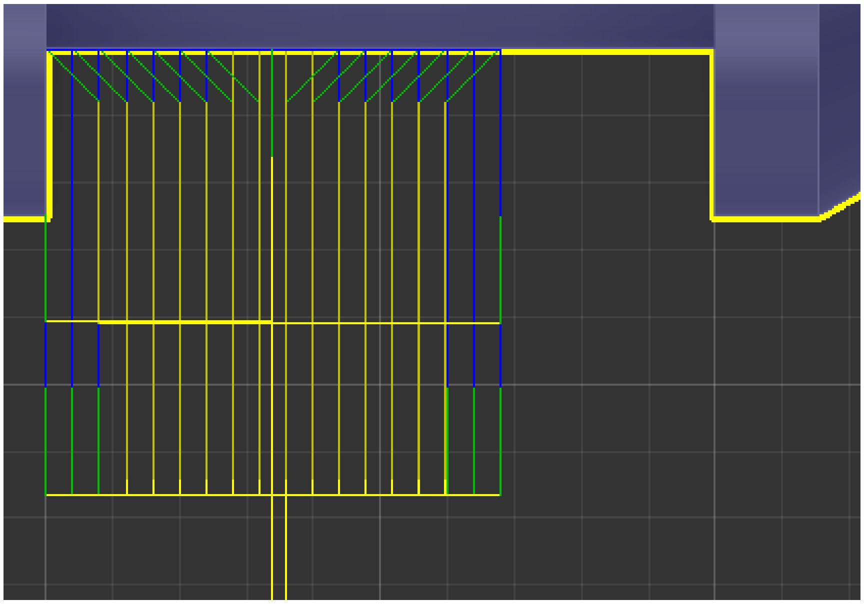


FIGURE 1.30: PROPER GROOVING PATHS.

- Change “front” and “back” mode both to Selection.
- Click the button next to “Front Reference,” and then select the front edge of the groove, which will then show as an orange indicator on the model.
- Do the same for the “Back Reference,” which will show as a dull yellow indicator on the model.
- Ensure the offsets are `0`. This means that there is no offset distance from the reference point.

All of this can be seen in Figure 1.29. Click OK. Does your grooving path now match Figure 1.30? If so, you may continue.

1.10 Creating a Parting Operation

The last operation is to part the piece. Parting refers to the operation that cuts through the entire diameter of the stock, removing the part from the stock.

- Similar to before, create a Turning part operation. If you need a reminder, follow these steps:
  - Return your attention to the Ribbon.
  - Under the Turning tab, select Turning part.
- Ensure that Tool 52 is selected.
- Match the settings to 1.31.

Fusion should automatically select the back face of the piece. If it does not, go to the Geometry tab, and ensure "Back mode" is set to Model Back, Figure 1.32. Click OK.

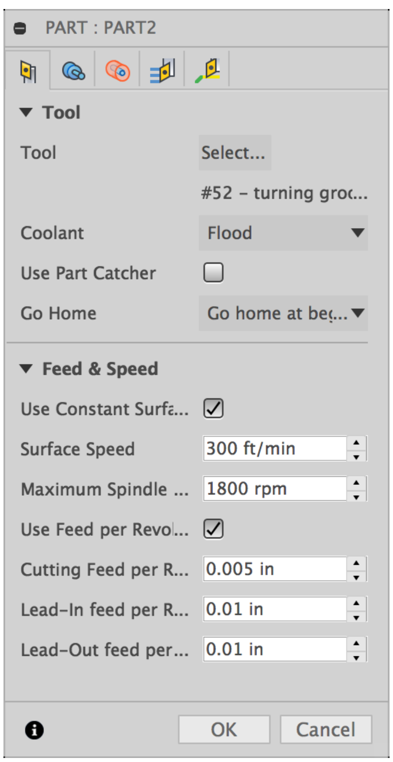


FIGURE 1.31: PROPER PARTING SETTINGS.

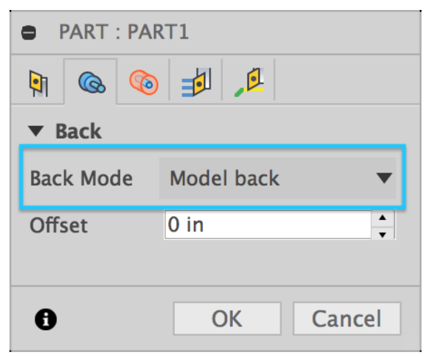


FIGURE 1.32: SELECTING THE BACK OF THE MODEL FOR PARTING IF NECESSARY.

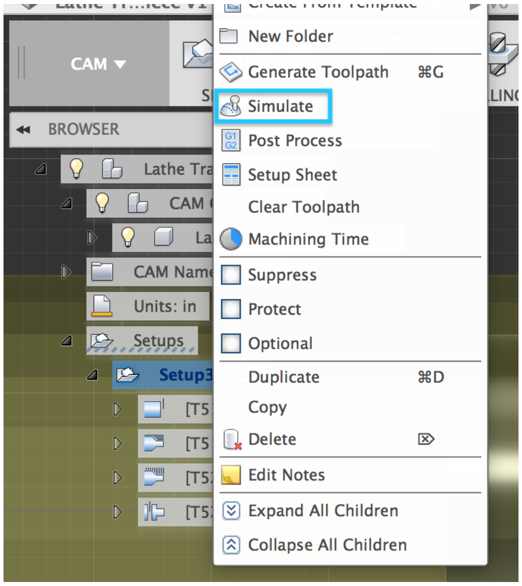


Figure 1.33: Right-click menu of a Setup.

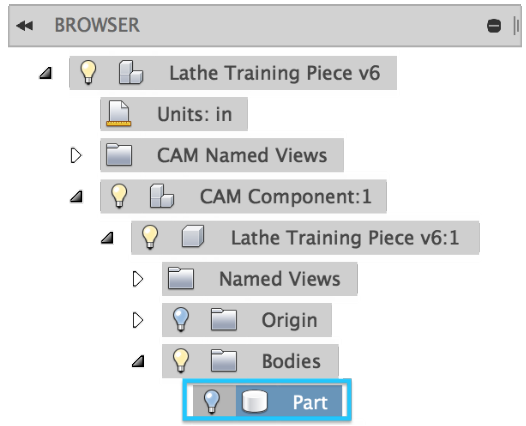


Figure 1.34: Turning off the visibility for the part body.

1.11 Simulating the Tooling Paths

You are now ready to simulate the machining process. This is a very powerful feature of Fusion, and will help you troubleshoot, identify issues, and double-check the tool paths.

- Right-click on the Setup we've been working on in the Project Browser.
- Select Simulate, Figure 1.33.
- Turn off the lightbulb next to "Part" in the Project Browser, Figure 1.34.
- Match your simulation settings to Figure 1.35.
- Click Start Simulation button, Figure 1.36.

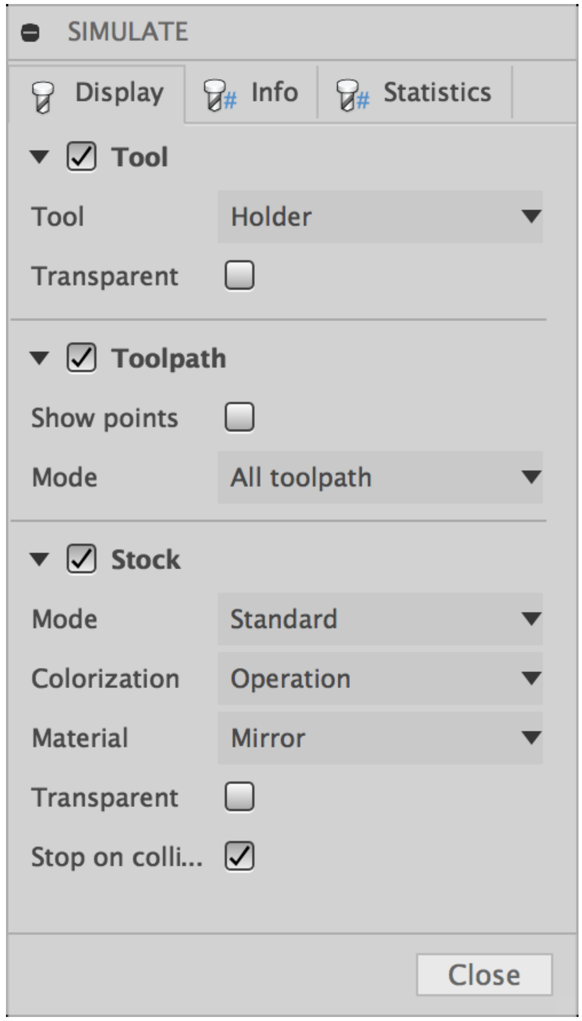


Figure 1.35: Appropriate simulation settings.

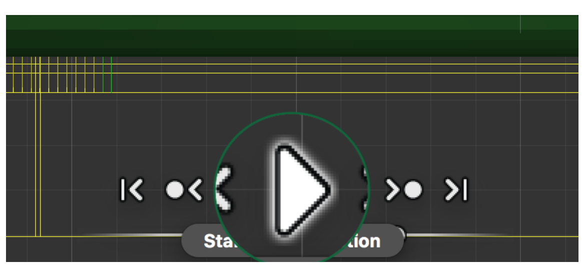


Figure 1.36: Starting a simulation.

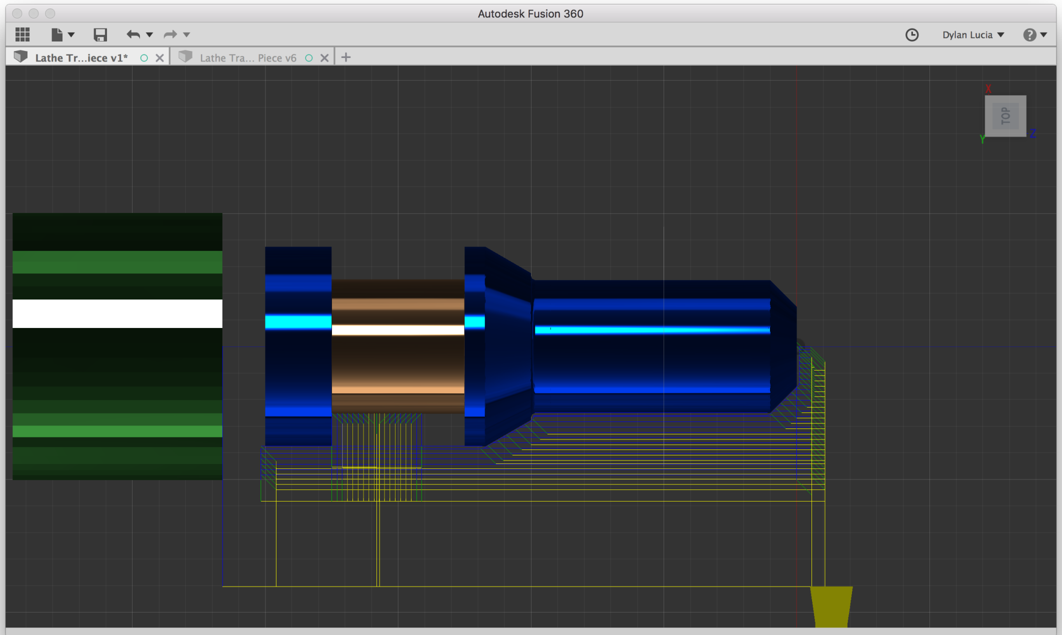


FIGURE 1.37: PROPER SIMULATION RESULT.

Watch through the entire simulation. You can adjust the speed of the simulation with the slider at the bottom of the screen. If there is a collision, Fusion will stop the simulation, and a red mark with information will appear on the scrubber, also at the bottom of the screen. If, when the simulation successfully finishes, your screen matches 1.37, you are done with your tool paths. This is the most complicated part of the tutorial; well done, so far!

Chapter 2

Post-Processing

2.1 Introduction to Post-Processing

Right now, you have a couple of tool-paths, but the lathe has no way of interpreting them. This job is left to the post-processor (often simply called a "post"). We will use a cloud post, which you already uploaded to your A360 account, and is accessible within Fusion already (as long as you're still connected to the internet).

2.2 Post-Processing

Turn your attention to the Ribbon at the top of the window.

- Click Post-Process in the Ribbon, Figure 2.1. In the popup window, change the "Source" to My Cloud Posts.
- Ensure the "Post Processor" is Tormach Turning (PathPilot).
- Change "Program Number" to `LatheCAM_Surname` which can be seen in Figure 2.2. You should not need to open the "Properties" dropdown. Click OK.
- In the window that pops up, choose a destination to save your file that you will remember, and if it did not happen automatically, type `LatheCAM_Surname` as the filename, Figure 2.3.

Fusion may offer to install software called Brackets, which serves as a way to read or edit G-code. This is optional. You shouldn't need to install Brackets if you already have code-editing software, such as Notepad++ or TextWrangler.*

*`http://goo.gl/52MNTt`

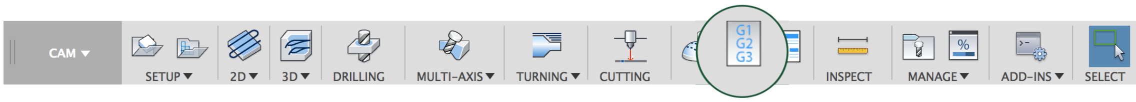


Figure 2.1: Post-Process button in the Ribbon.

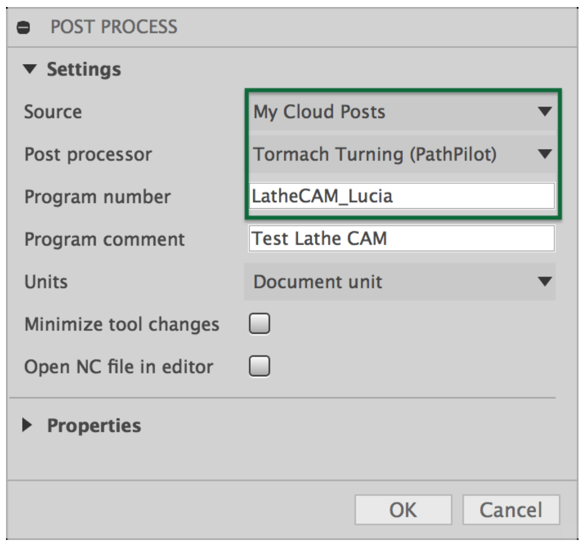


Figure 2.2: Post-process settings.

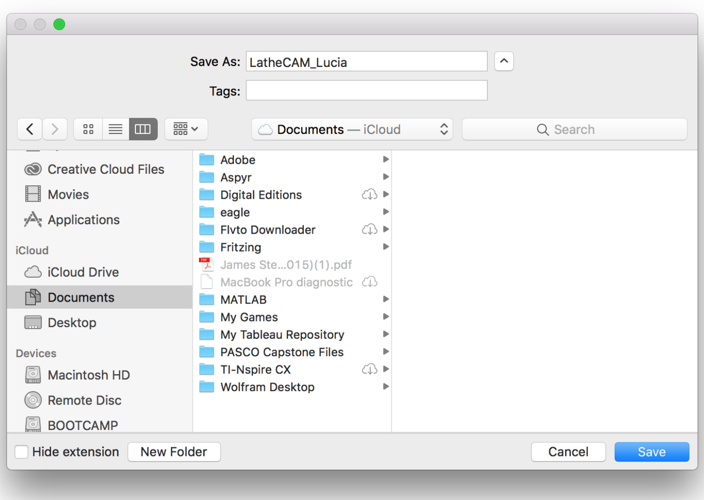


Figure 2.3: Saving the file to the computer.

2.3 Post-Processing Summation

Congratulations, you now have a fully-finished `.nc` file, which contains the G-code that the lathe will read to create your piece! The next chapter of this training will involve actually using the lathe, so if you have not done so already, go to the MakerSpace/setup a time with a Ninja to create the widget for your training.

Chapter 3

Moving to the Lathe

3.1 Accessing the USB Flash Drive

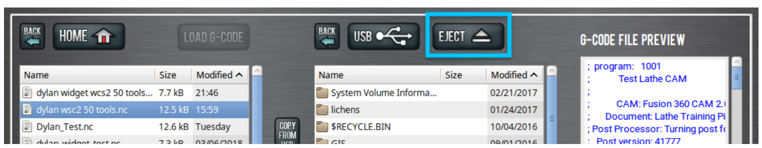


FIGURE 3.1: EJECTING THE USB FLASH DRIVE FROM THE LATHE SAFELY.

Now you will transition from the digital world to the physical one by beginning work on the lathe itself. First, you will need to get the digital file you just created into the lathe's hard drive.

- Adorn appropriate protective equipment and power on the lathe as normal so that it returns to its dormant state.
- On the monitor for the Lathe computer controller, navigate to the File tab and click Eject, Figure 3.1.
- Walk to the right side of the machine, move the computer arm as far forward on the machine as possible, and bend down towards the cabinet. There is a silver locking mechanism on the large panel of the machine. Use your finger to push the lower square labelled Open, Figure 3.2. The locking mechanism will pop out, and can be used as a handle to gently swing open the cabinet.
- Remove the USB flash drive which is inserted in the computer that is mounted on the cabinet door.

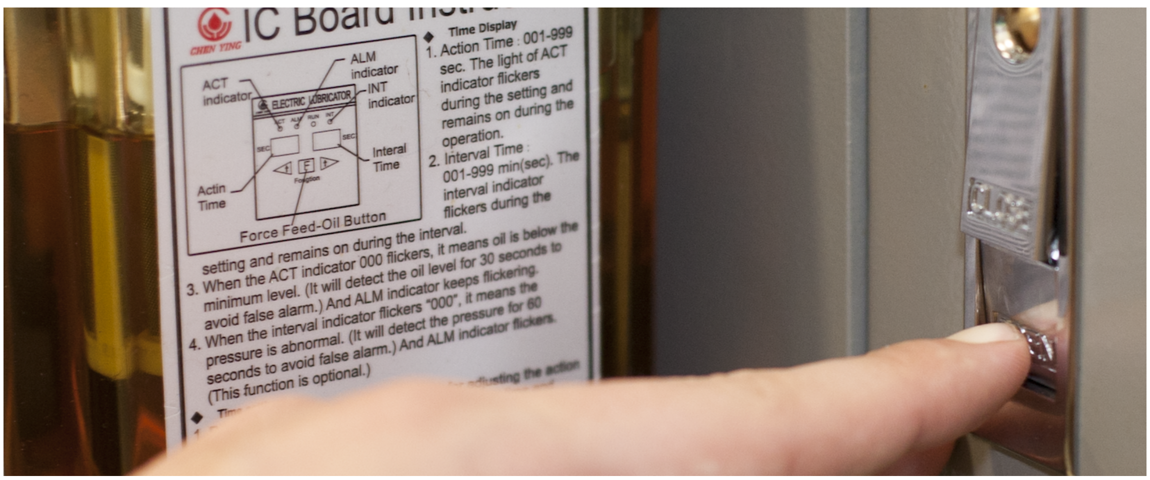


Figure 3.2: Opening the locking mechanism on the computer cabinet.

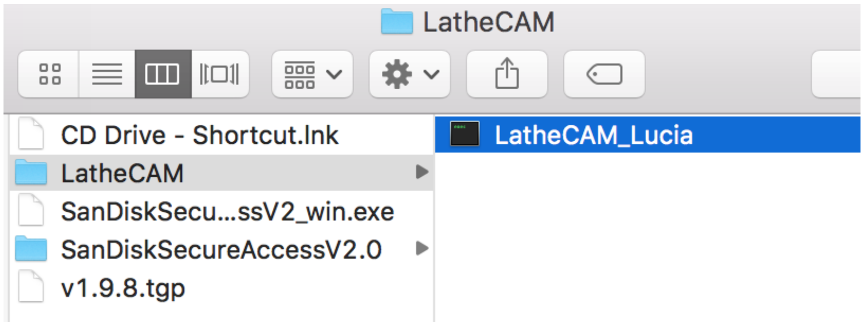


Figure 3.3: The correct folder to place the G-code in the USB.

3.2 Transferring the G-code onto the USB flash drive

Now, you will transfer the file to the lathe using the flash drive that you just got from the lathe computer.

- Insert the USB flash drive into the USB port on a computer where you can access your G-code file.
- Copy your G-code file into the "LatheCAM" folder, which is in the root of the flash drive, Figure 3.3.
- Safely eject the flash drive from the machine you are working on, and reinsert it in the Lathe's computer. Gently close the cabinet door and then press the locking mechanism so that it is flush. You should hear an audible click. This indicates that the cabinet is properly locked, which should be the case *whenever* the Lathe is operating.

If you return to the Lathe computer monitor, you should still see the File tab open. If it is not, click on it.

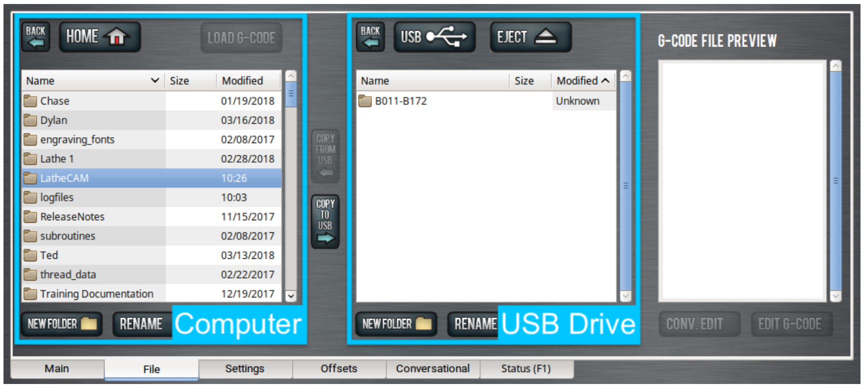


Figure 3.4: Both file managers.

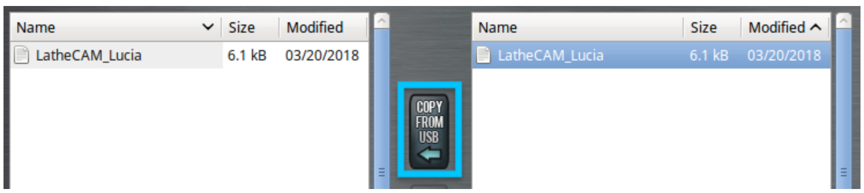


Figure 3.5: Copying the file from the USB onto the Lathe hard drive.

3.3 Copying the G-code onto the Lathe

On the monitor, you will see two file managers. One is the Computer hard drive (left), and one is the flash drive (right), shown in Figure 3.4.

- In the Computer file manager, click HOME and navigate to the "LatheCAM" folder.
- In the USB file manager, navigate to its "LatheCAM" folder as well, which should house your G-code.
- Select your file and click Copy From USB, Figure 3.5.

Figure 3.7: Go to G30 button.

Now, your G-code file is successfully written to the hard drive of the Lathe's computer. Double-click on your file in the computer side, and your G-code should open automatically in the Main tab, Figure 3.6.

Since you (or the user before you) has referenced the x-axis and z-axis already (and thus the tool post is as far from the chuck as possible), you can now click Set G30, and a green light on that button will illuminate. This tells the machine where the home is for CAM operations. Now, if you jog away from G30 an inch or so and click Go to G30, Figure 3.7, the machine should jog back to G30 by itself.

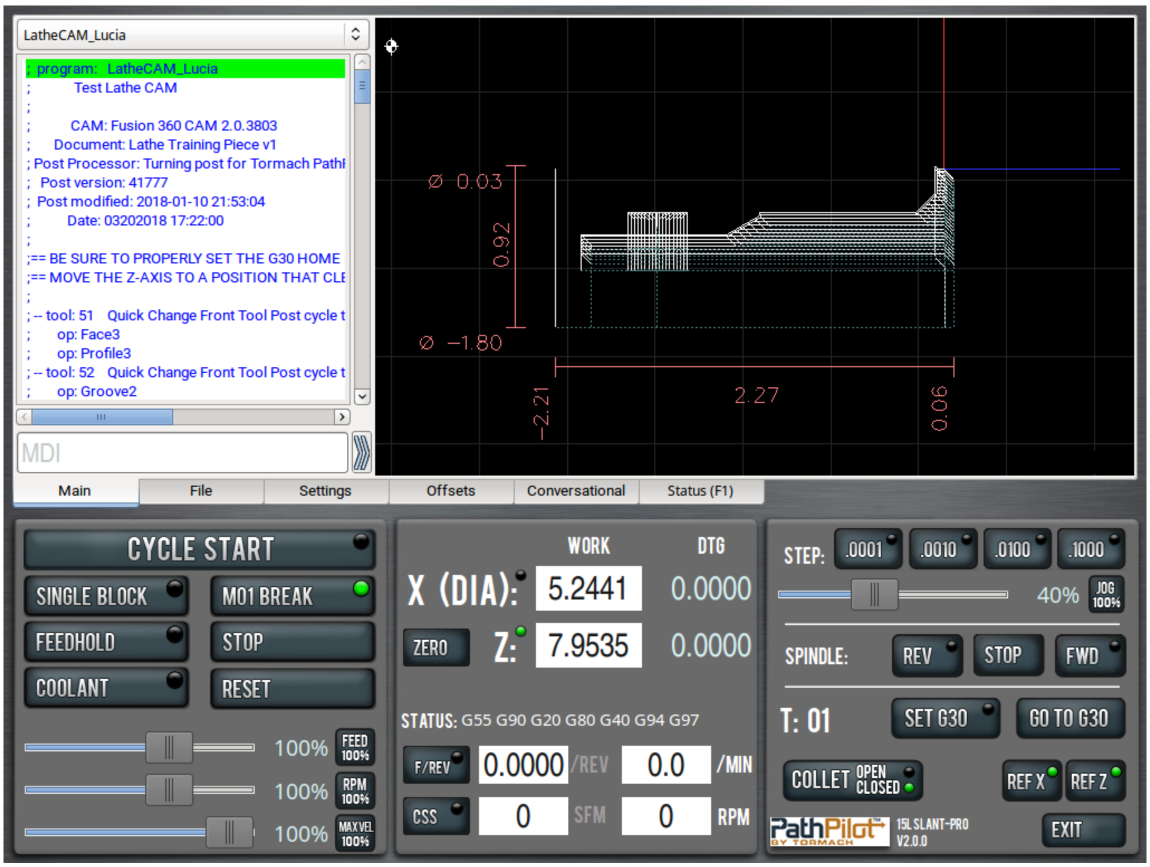


Figure 3.6: The main window with loaded G-code.

Chapter 4

Inspection, Setup, and Zeroing of the Lathe

4.1 Path Inspection & Running in Air

Inspect the path visualization, shown in Figure 4.1, and if you see any paths here that do not match the paths you created in Fusion, ask your training Ninja to evaluate the situation.

If all looks as it should, continue:

- Insert **Tool 51*** and type `T51` in the "MDI" box to tell the machine which tool you are using.
- Jog the machine in the Z-axis to about halfway, and *approximately* center the X-axis, Figure 4.2.

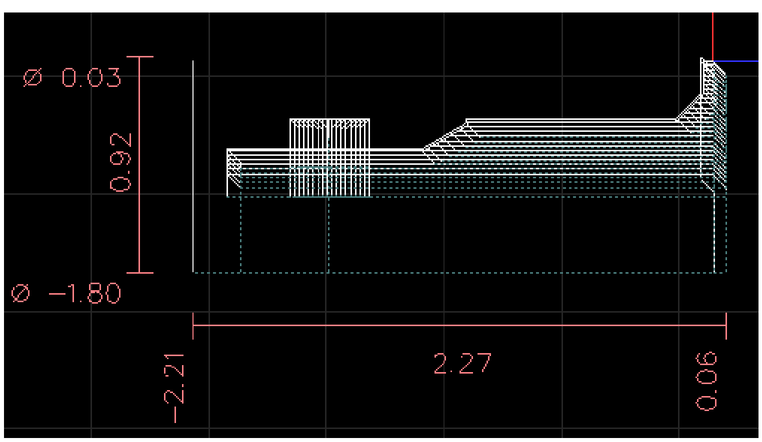


Figure 4.1: Proper paths on the Lathe monitor.

*Tools will be referred to as their CAM numbers. Refer to the Table of Tools if you need help remembering which tool is which.

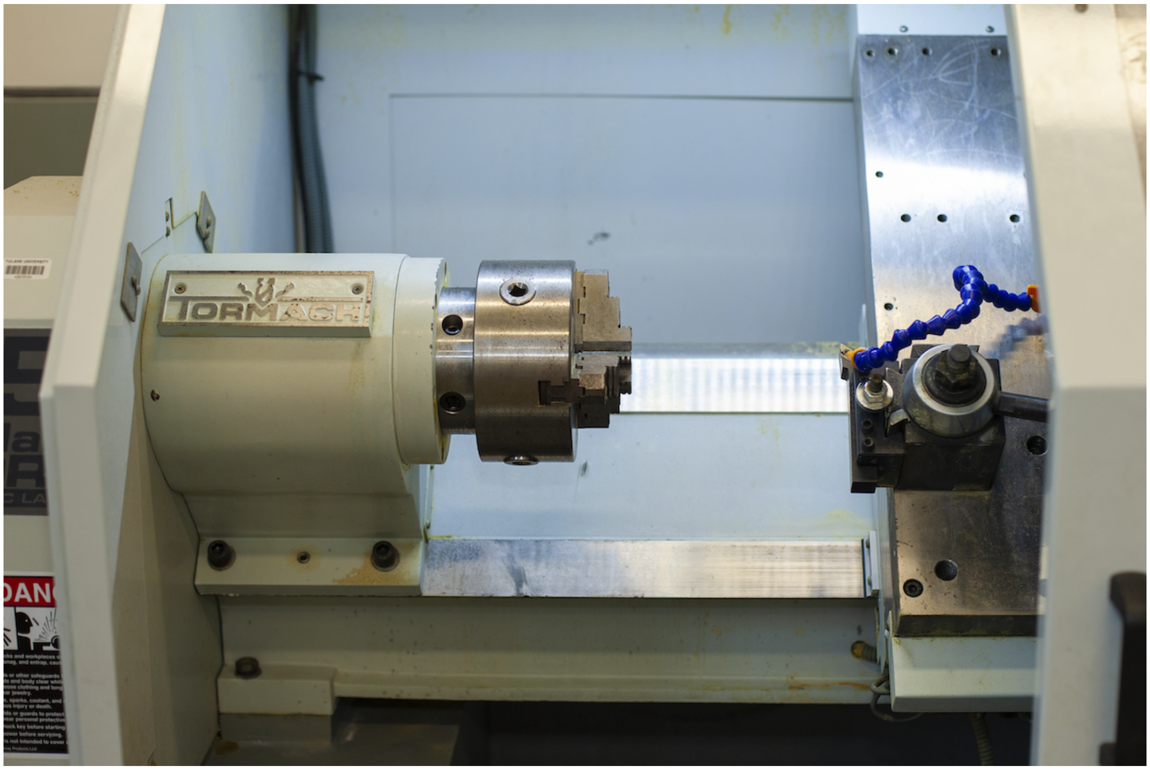


Figure 4.2: Tool 51 approximately centered in the machine.

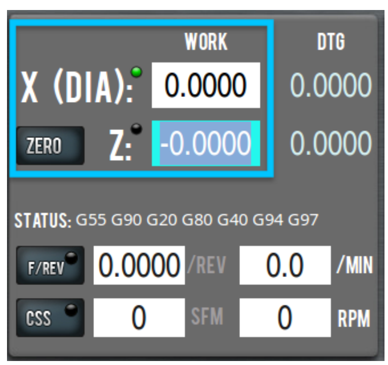


Figure 4.3: Zeroing the axes for running in air.

- Now set both the "X (DIA)" and "Z" values to `0`, Figure 4.3.
- Ask your training Ninja to show you how to run the file in air to evaluate it.

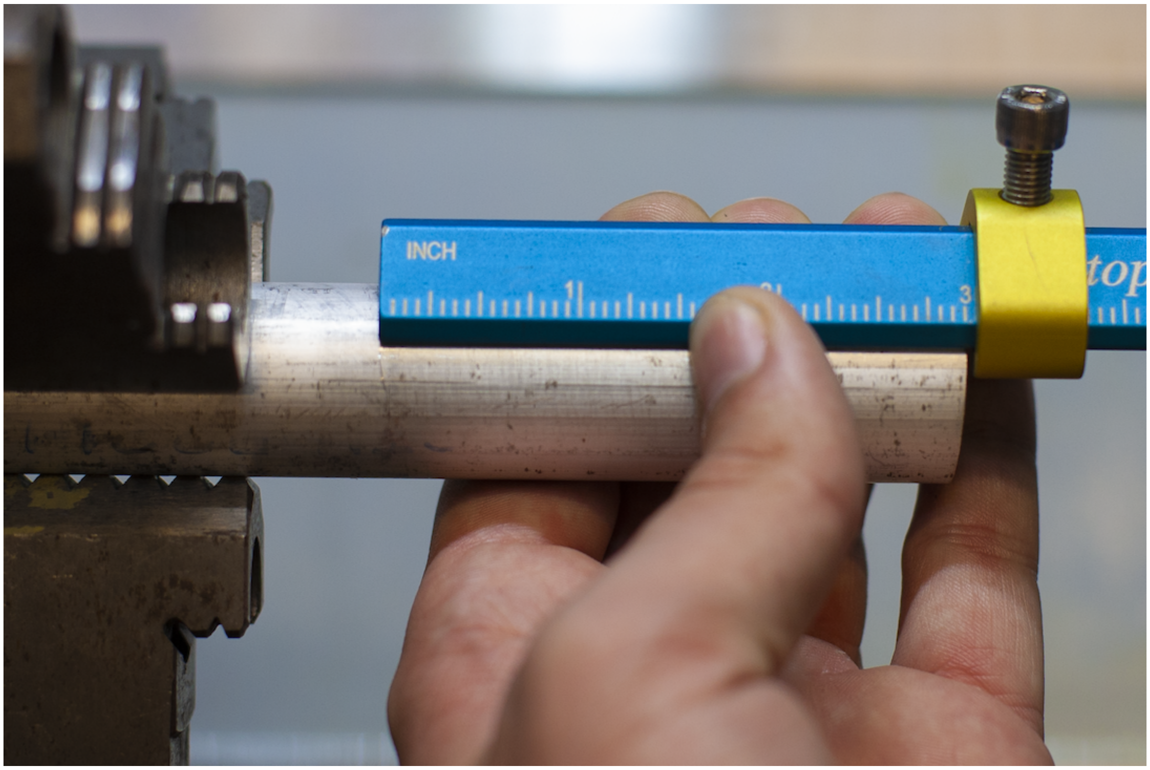


FIGURE 4.4: CHECKING STOCK/CHUCK CLEARANCE WITH THE STOP-LOC.

4.2 Inserting Stock

Next, you will prepare the stock to be turned.

- Using your knowledge from your first lathe training, as well as your experience gained since then, select an appropriate piece of 1” OD stock.[†]
- Insert and secure the stock in the chuck by evenly closing the jaws using the jaw key, and use the Stop-Loc™ to ensure there is a minimum of 3” of stock material from the chuck, Figure 4.4.
- Give the chuck a few spins by hand to make sure the stock is secured steadily and evenly. If it is not, remove the stock, ensure it is straight, and retry from the previous step.

[†]This stock should be approximately twice the length of your digital model, and should be free of any cracks, large divots, or other damage.

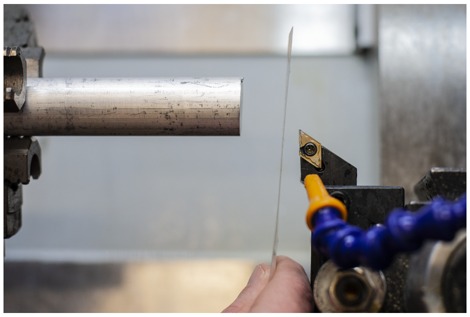


Figure 4.5: Tool 51 1" from the face of the stock.

Figure 4.6: Pinching the feeler to the face of the stock.

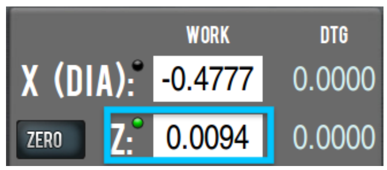


Figure 4.7: Entering the width of the feeler to zero the z-axis.

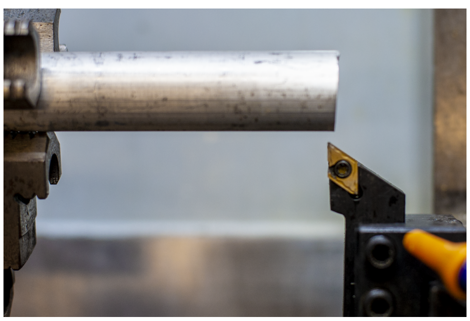


Figure 4.8: Tool 51 ready to approach the OD for zeroing.

Figure 4.9: A faint curl appearing at the tip of Tool 51.

4.3 Zeroing the Machine to the Stock

This marks the beginning of the extremely important (but also slightly tedious), process of zeroing the machine. Ensure you follow all steps very carefully and complete all steps thoroughly. If at any point you don't understand something, ask your training Ninja immediately.

- Jog **Tool 51** to approximately 1” from the face of the stock, Figure 4.5.
- Grab the feeler, and slowly jog towards the stock until you are approximately $^{1}/_{10}{}^{\text{th}}$ of an inch away.
- Next, use the stepping wheel to slowly pinch the feeler against the face of the stock, until the feeler just barely supports its own weight, Figure 4.6.

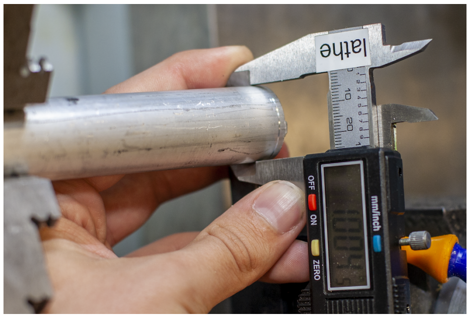


FIGURE 4.10: MEASURING THE DIAMETER OF THE STOCK.

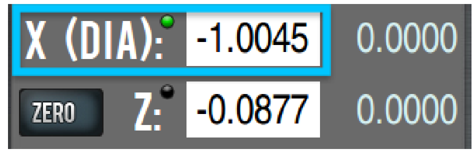


FIGURE 4.11: ZEROING THE X-AXIS.

- On the monitor, enter `0.0094` for the “Z” value, Figure 4.7. This is the exact width of the feeler, and so we enter this number to tell the lathe that the tip of the tool is exactly that far away from the front face.
- Carefully jog away from the stock, and replace the feeler. Next, jog to the position shown in Figure 4.8.
- Alternate between slowly approaching the stock in the X-axis and rotating the chuck one full rotation by hand. Do this until you notice a faint curl of material on the tool tip, Figure 4.9.
- Grab the digital calipers, zero them in imperial mode, and measure the diameter of the stock as close as is possible to the point where the tool is touching, Figure 4.10.
- Read the value shown on the display, and remove and replace the calipers.
- Type a negative in front of the measurement for the diameter that you just measured in the “X (Dia)” box and hit enter, Figure 4.11. For the stock measured in 4.10, we would enter a value of `-1.0045`. However, yours will likely be different.
- Slowly jog away to a safe z-distance, and move the tip of the tool in the x-axis until the value reads `-0.0000`.
- Slowly approach the face in the z-axis until the tool is about $^{1}/_{10}{}^{\text{th}}$ of an inch away.
- Verify that the tip of the tool appears to be in line with the center of the stock by rotating the chuck a few times by hand and noting the point of rotation on the face, Figure 4.12.

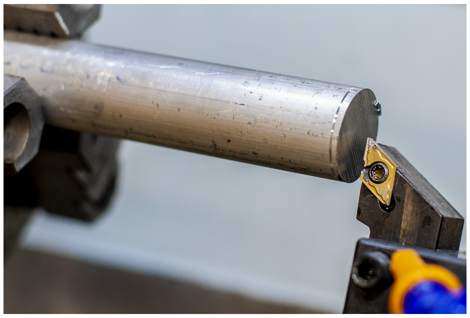


Figure 4.12: Verifying the zeroing on the stock.

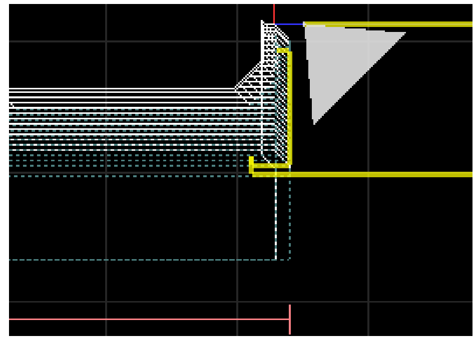


Figure 4.13: Verifying the zeroing on the paths.

- If all is good, verify that the path visualization window shows the tip of the tool in the location shown in Figure 4.13, verifying that the tooling paths will start at the zero-point of the stock that we just set.‡
- Jog back to a safe distance and click Go to G30.

‡You can navigate this window using the middle-mouse button to pan, and scrolling to zoom.

FIGURE 4.14: THE COOLANT BUTTON ON THE MONITOR.

FIGURE 4.15: APPROPRIATE COOLANT SPRAY FOR TOOL 51.

4.4 Adjusting Coolant

Arrange the blue gooseneck hose so that the coolant will spray the tip appropriately.[§] Verify this by clicking Coolant, Figure 4.14. Make minor adjustments to the coolant hose until the coolant is adjusted properly, Figure 4.15, then turn off the coolant by clicking its button again.

[§]"Appropriate" is dependent on the material, tool, and speeds. Generally, the coolant should be aimed such that it dislodges chips that gets caught on the tip while machining, as well as maintaining a cool temperature of the tip.

Chapter 5

Running the Program on the Lathe

5.1 RH Turning Tool Operation

Now, you will execute your first full operation on the stock in the machine!

- Close the door.
- On the screen, change the velocity slider to `10%`, and click Cycle Start, Figure 5.1.
- Nothing will seem to happen in the machine. However, the green dot on the button will start flashing, indicating your attention is needed. If you look at the G-code, you'll see a line highlighted indicating you need to change to Tool 51, Figure 5.2, if it isn't already mounted.
- Click Cycle Start. Allow the tool to approach the stock, and when it is about an inch away, slide the velocity down to `0%`.
- Verify that the "Z" value on screen is approximately an inch. If so, raise the velocity to `10%`.

The lathe will now follow the paths and turn the RH operations on the stock. Allow the Lathe to return to G30 by itself and completely stop moving before you open the door. Open the door and inspect the result of these operations. Make sure your piece matches Figure 5.3.

Figure 5.1: Cycle Start button.

Figure 5.2: The highlighted G-code indicating the first required tool.

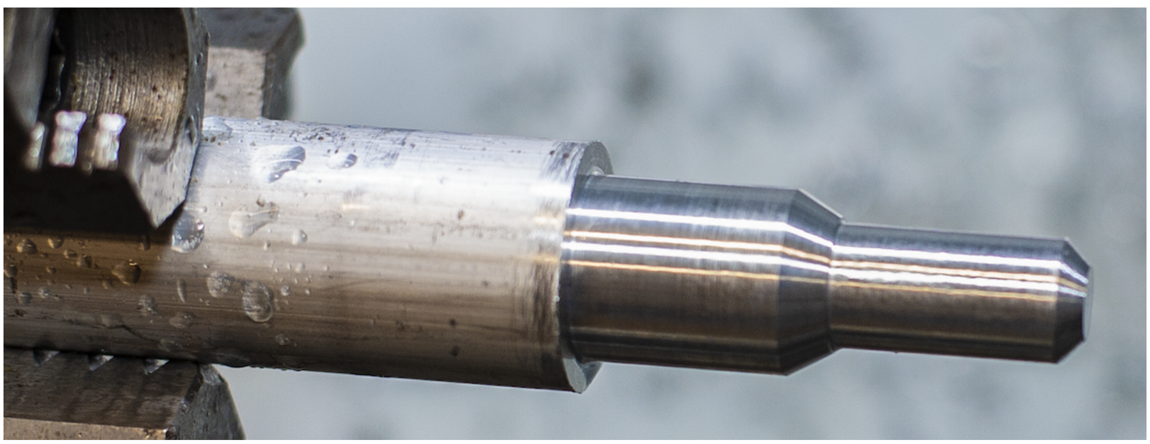


Figure 5.3: What the piece should look like so far.

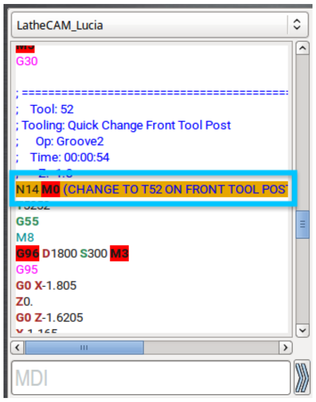


Figure 5.4: Highlighted G-code indicating a tool change.

Figure 5.5: Verifying the coolant spray for Tool 52.

5.2 Changing the Tool & Grooving/Parting Operations

On the monitor, you will notice that the G-code is highlighted, telling you to switch to **Tool 52**, Figure 5.4.

- Remove **Tool 51**, and blow out the chips around the tool post using compressed air which is on the reel on the front wall of the MakerSpace.
- Insert **Tool 52** and aim the coolant hose, ensuring its spray is adequate, Figure 5.5.
- Close the door and click Cycle Start. You'll notice that the machine doesn't move. If you look at the visualization window, you will see text reminding you to switch to Tool 52. You do *not* need to enter anything in MDI. Just verify that Tool 52 is in the post and ready to machine with appropriate coolant coverage.
- Click Cycle Start to tell the machine that **Tool 52** is ready to go.
- Stop the velocity when the tool is about 1" away to make sure the measurements are correct again. If so, proceed with caution.
- The machine will return itself to G30 after parting and come to a finish. The piece will fall into the bottom of the chamber.

Figure 5.6: The final piece after parting.

5.3 Conclusion

Open the door and examine your piece. If it matches 5.6, you are finished! Clean up your workspace and have your training Ninja take a photo of you with your finished* piece. Congratulations!

*You may want to remove the nub left behind by the parting tool and clean up any sharp corners using a grinder.

Index

Glossary

Autodesk A360 Cloud system created by Autodesk for storing, accessing, creating, and backing up project data.

bezier curve A parametric curve defined mathematically by multiple nodes rather than just a radius and angle. Often very organic.

CAM Computer Aided Manufacturing/Machining. Using software and digital representations of physical machines and tools to automate the complex creation of many parts.

cloud libraries A library for tools and the like that are stored in the cloud.

collision An unfavorable event where the machine touches the stock with something other than the cutting tip of a tool, such as the tool holder. Can be very damaging to the piece and the machine.

Conversational A part of PathPilot, proprietary CNC machine control system software created by Tormach. It allows "interactive" usage of the lathe, and is useful for simple operations. It lacks functionality such as complex curves and requires programming and measurements for every operation.

coordinate system The set of rules and code that define a 3D space and the actions within it. As an example, the command for returning to the home position in one coordinate system may be G54, G55. . . G59 which depends on how the machine is coded and its coordinate system.

cutting feed per revolution A setting that defines the distance the tool moves for each full revolution of the spindle while cutting.

data panel The main "file system" that Fusion accesses. Linked directly to the user's A360 account, and serves as a cloud drive.

facing An operation performed upon the stock where the front face of the stock is turned slightly, reducing the overall length of the stock. Serves as a way of perfectly squaring the face of the stock with the spindle.

feeds Refers to how quickly the tool moves towards, away from, and during a cut.

finishing Portions of a stock removal operation that occur after the roughing that very minutely and precisely shape the stock to its final size, leaving a desirable surface finish.

Fusion 360 Software created by Autodesk used for modeling 3D objects, creating CAM operations, and many more possibilities.

G-code Also known as RS-274. Layman term for Numerical Control programming language. Stored in .nc files, and contain the code that creates the commands for the CNC machine control system software to interpret.

grooving An operation used to create deep channels or grooves in the radius in the stock.

maximum spindle speed A setting that limits the speed of the spindle (the motored chuck that rotates the stock), to a maximum speed when performing operations where the tool has a constant surface speed (the tool's tip is always moving at the same speed in relation to the point it is touching, accomplished by varying the speed of the spindle depending on the tool's distance from the center).

part An operation that cuts through the entire radius of the stock, cutting the final piece away from the remaining stock.

part file A file that contains a 3D model of a part or body.

passes The group of settings controlling how many times the tool will step through an operation. More passes means the tool takes less material off each time to reach a desired depth, saving tool life and finish quality. Fewer passes means the tool takes more material off each time to reach a desired depth, saving time.

post-processor Often referred to as simply a "post." A driver that exports the toolpaths and instructions from Fusion into G-code specifically for a certain machine.

profile An operation that shapes the profile, or outer radial limits defining the shape of the piece.

project browser The "file tree" of sketches, setups, operations, bodies, and other components of a Fusion project.

radii The group of settings controlling and limiting movement for operations in the radial direction.

ribbon The strip of menu options and other software tools in Fusion's graphical user interface (GUI).

roughing Portions of a stock removal operation that remove the bulk of the material for the operation, getting close to the final dimensions.

scrubber The tick on the simulation timeline that represents the current timeframe of the simulation in regards to the whole process. Can be dragged to quickly move to a different timeframe of the simulation.

setup A specific set of CAM operations grouped together. Operations are performed in order within the setup.

simulate A function of Fusion that renders a 3D representation of the machining process for a setup from start to finish. Allows easy visualization and evaluation of a setup, and is useful for troubleshooting.

speeds Refers to the speed of the rotating stock. This can be controlled through a constant rotational velocity, or through a variable rotational velocity–called constant surface speed.

stock Raw material used for machining. Often aluminum, wood, or a polymer.

tool The physical tool that carves into the stock material, or the digital representation of that physical tool.

tool library A file that contains the different physical tools represented as digital tools for usage in CAM operations. Negates the need for each user to attempt to hand-create the tools digitally.

tool number The number that represents the tool. This number is defined in Fusion for each operation created. The lathe receives this tool number in the G-code, and knows how to adjust its movements for the specific tool specified by the tool number. In the Tulane MakerSpace, CAM and Conversational tools are physically the same, but represented digitally as two different sets, as shown in the Table of Tools.

turn The action of using a lathe to carve away material from the stock material to create some sort of rotationally symmetric piece.

WCS offset Defines the Work Coordinate System used by our lathe. Defines the G-code coordinate system of the target lathe, which the post-processor then uses to modify the outputted code to be used properly on our machine.