

CAM Training *for the* CNC Lathe

An Introduction

Dylan Lucia



Tulane University MakerSpace

DRAFT

Created 03/19/2018. Modified 04/10/2018.

Table of Tools

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

Tool Name	Tool Number		Use
	<i>Conversational</i>	<i>CAM</i>	
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	
Jacob's Chuck	10	60	Center boring

Contents

Table of Tools	i
Contents	ii
1 Creating Toolpaths	1
1.1 Introduction	1
1.2 Setting up Fusion for CAM	2
1.3 Creating the Stock	5
1.4 The CAM Environment in Fusion	6
1.5 Creating the CAM Setup	6
1.6 Creating a Facing Operation	8
1.7 Creating a Turning Profile Operation	9
1.8 Checking Created Paths	13
1.9 Creating a Turning Groove Operation	13
1.10 Creating a Parting Operation	16
1.11 Simulating the Tooling Paths	16
2 Post-Processing	21
2.1 Introduction to Post-Processing	21
2.2 Post-Processing	21
2.3 Post-Processing Summation	23
3 Moving to the Lathe	25
3.1 Accessing the USB Flash Drive	25
3.2 Transferring the G-code onto the USB flash drive	26
3.3 Copying the G-code onto the Lathe	26
4 Inspection, Setup, and Zeroing of the Lathe	29
4.1 Path Inspection & Running in Air	29
4.2 Inserting Stock	31
4.3 Zeroing the Machine to the Stock	31
4.4 Adjusting Coolant	33
5 Running the Program on the Lathe	35

CONTENTS

iii

5.1 RH Turning Tool Operation 35
5.2 Changing the Tool 36
5.3 Conclusion 36

Index **39**

DRAFT

Chapter 1

Creating Toolpaths

1.1 Introduction

This training course will serve as a basic walkthrough for transitioning from using Conversational in PathPilot 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. There are always more methods and tricks to learn in CAM, and after this course, you are encouraged to explore online forums, books, and other resources to learn more methods for 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 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 face cylindrical stock or something similar. 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, as can be seen in [Figures 1.1 and 1.2](#). For simplicity, and to show you the similarities/differences to Conversational, we will be making the same part you created in your initial lathe training.

To begin, you will need to download and install a few files. First, you will need to install Fusion 360. In the future, you may choose to use other software, but for the purposes of this training, you will utilize Fusion.* Fusion is free for those in University (and by virtue of being a member of the MakerSpace, that means you can get it for free). 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 Tulane information to successfully get access to Fusion at no cost. Once downloaded, install

*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 constantly is updating, and as such, the version used in this tutorial (2.0.3803), will likely be different from yours.

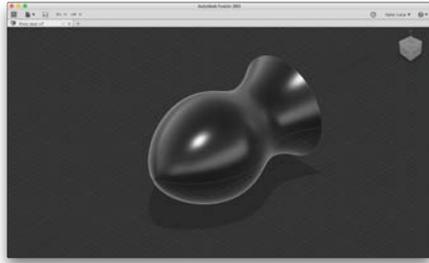


FIGURE 1.1: A BEZIER CURVE REVOLVED AROUND THE Z-AXIS.



FIGURE 1.2: THE RESULT OF CREATING FIGURE 1.1 ON THE LATHE USING CAM OPERATIONS.

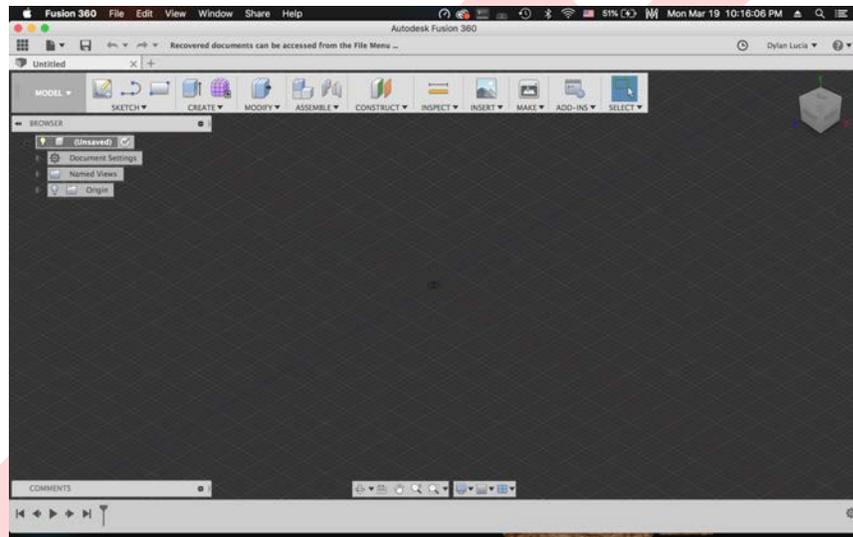


FIGURE 1.3: A BASIC FUSION WINDOW.

the software. Once you have done so, open it and make sure that it runs properly. You should be met with a screen similar to the one shown in Figure 1.3.

1.2 Setting up Fusion for CAM

Click on `Show Data Panel`, Figure 1.4. Click on the blue `New Project` button, Figure 1.5. Enter a name, such as `Lathe Project`. This will be for lathe-specific parts. Navigate to Fusion Preferences. On the left side of the window, select `CAM`. Check the box next to `Enable Cloud Libraries`, Figure 1.6. Then select `OK` to close the preferences window. Navigate to [this training's page](#) on the MakerSpace's Wiki and download the part, the tool library, and the post-processor. The part is the model you will be creating using CAM. The tool library allows you to instantly install the digital tools we have for the lathe into your Autodesk

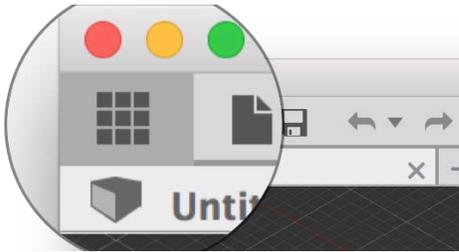


FIGURE 1.4: SHOW DATA PANEL.

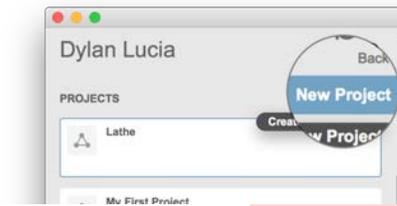


FIGURE 1.5: NEW PROJECT.



FIGURE 1.6: ENABLING CLOUD LIBRARIES.

account. The post-processor is the coded instructions that Fusion uses to convert the CAM paths into G-code specific to our lathe. Return to the Data Panel and if you are not currently in your Lathe Project, open it by double-clicking it. Press `Upload`, Figure 1.7, and navigate to and select the part file you downloaded from the MakerSpace Wiki. It should appear in your Project folder. Now, open your web browser, and navigate to [Autodesk A360](#), Autodesk's cloud system, to sign in and open up your A360 account. Open the *Assets* folder, and then the *CAMPosts* folder. Drag/drop or upload the post-processor file you downloaded from the MakerSpace Wiki into this folder in your A360 account. Once it has uploaded successfully, shown in Figure 1.8, return to the Assets folder. Open the *CAMTools* folder and upload the tool library file you downloaded from the MakerSpace Wiki. Once it has uploaded successfully, shown in Figure 1.9, 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 anymore after this.

FIGURE 1.7:
PROJECT UPLOAD
BUTTON.

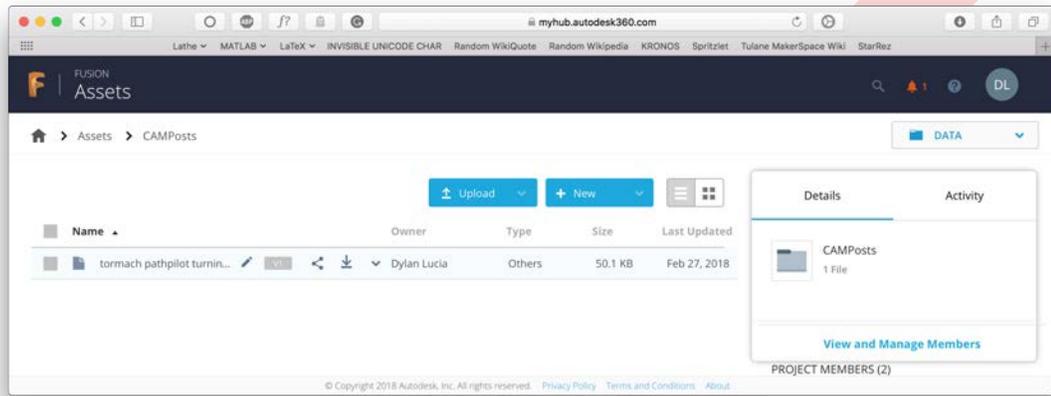


FIGURE 1.8: SUCCESSFUL UPLOAD OF THE POST TO THE CAMPOSTS FOLDER.

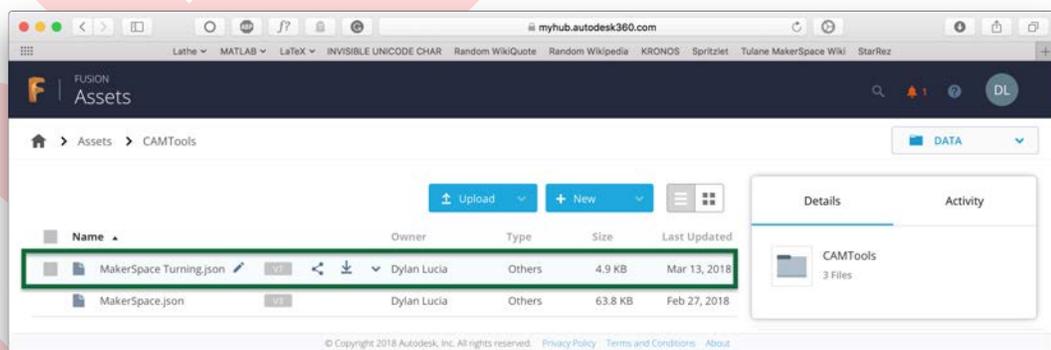


FIGURE 1.9: SUCCESSFUL UPLOAD OF THE TOOL LIBRARY TO THE CAMTOOLS FOLDER.

1.3 Creating the Stock

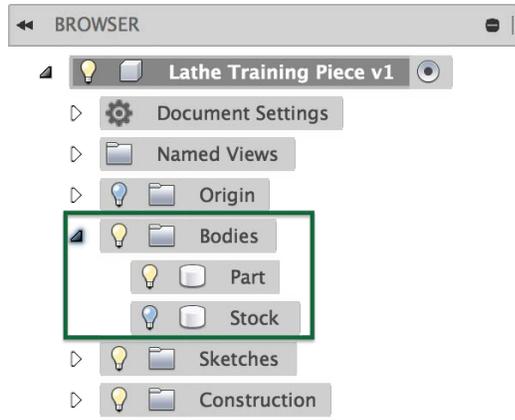


FIGURE 1.10: THE PROJECT BROWSER.

Open up the part file you uploaded to your project folder by double-clicking on it. It should look like Figure 1.11. Orbit around and inspect the part, getting to know it. Also ensure that your units are in imperial (inches). Something else you may notice is that in the part, we have not modeled threads or knurling. This will become clear later. If you open the **Bodies** dropdown in the Project Browser, shown in Figure 1.10, you can turn on the lightbulb next to the **Stock** body. This is a modeled body that represents the stock we will create the final part from, which is an important aspect of CAM. To lathe this part, much like the one created in basic lathe training, we will use 1" cylindrical stock. The length of the stock

modeled in Fusion does not matter, as long as it has about $1/10^{\text{th}}$ of an inch in front of the part, and at least an inch past the back of the part. You may notice through inspection of the sketches in the project that the stock is actually 1.005" in diameter, which accounts for variation in physical stock and gives us a little bit of a safety cushion.

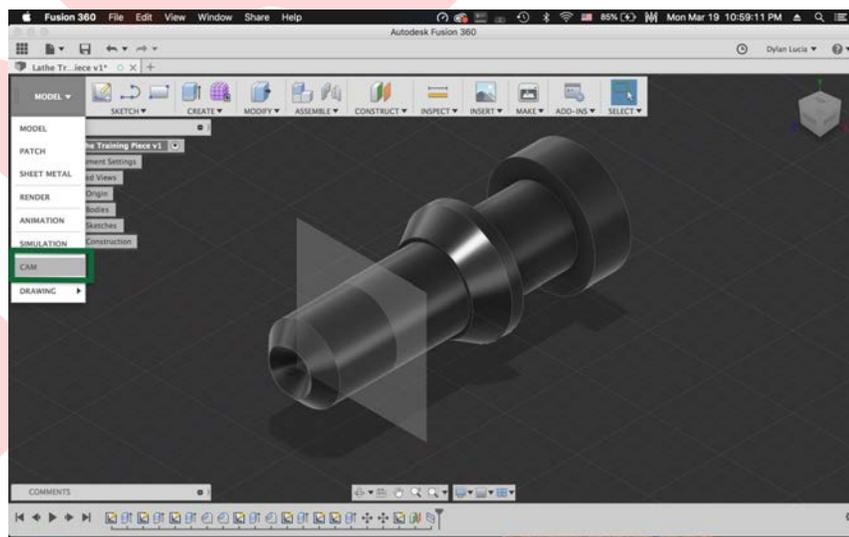


FIGURE 1.11: THE TRAINING PART IN THE FUSION WINDOW.

1.4 The CAM Environment in Fusion

To change our Workspace from Model to CAM, click `Workspace` in the Ribbon and select CAM, shown in Figure 1.11. 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 1.12, to position your view to the top view such that your view matches Figure 1.13. 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 foremost 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.

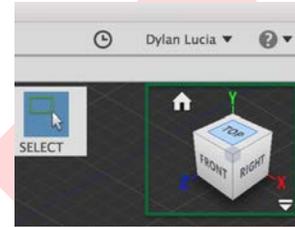


FIGURE 1.12: THE VIEW CONTROLLER.

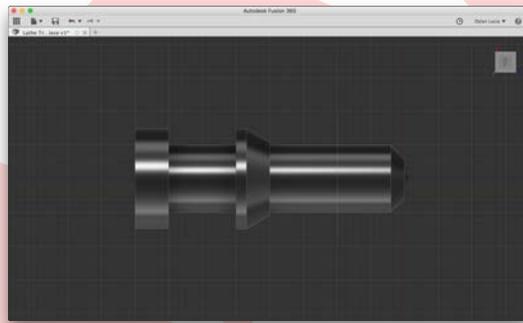


FIGURE 1.13: A GOOD VIEW FOR CREATING TOOLING PATHS.

1.5 Creating the CAM Setup

For any part, the first step in CAM is creating a setup. On the Ribbon, click `New Setup`, shown in Figure 1.14. Under “Setup”, change the operation type to `Turning or mill/turn`, shown in Figure 1.15. Flip your X-axis under Work Coordinate System by checking the box so that it matches Figure 1.16. You should see the red arrow on the model flip so that it faces down. This is because on our lathe, we tool in 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 tells Fusion that you want the tooling paths to be calculated based on this body. This can be seen in Figure 1.17.

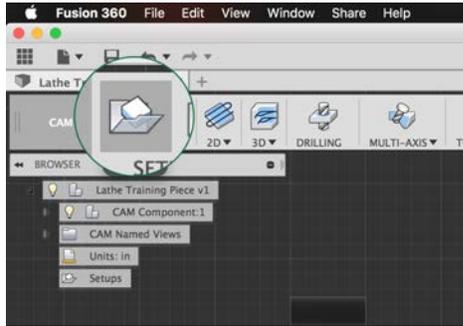


FIGURE 1.14: CREATING A NEW SETUP.

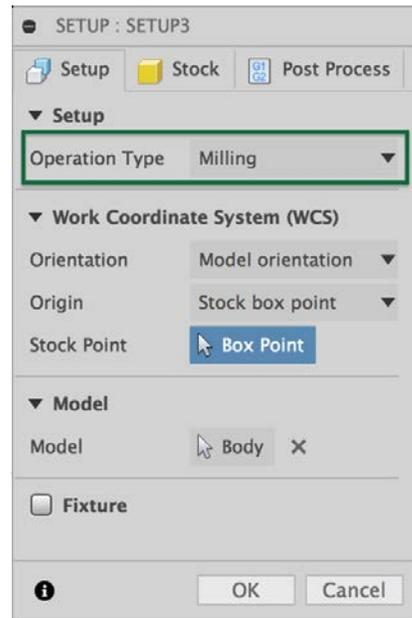


FIGURE 1.15: THE NEW SETUP'S SETTINGS DIALOG WINDOW.

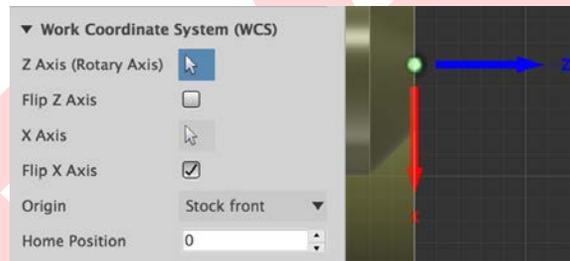


FIGURE 1.16: FLIPPED X-AXIS.

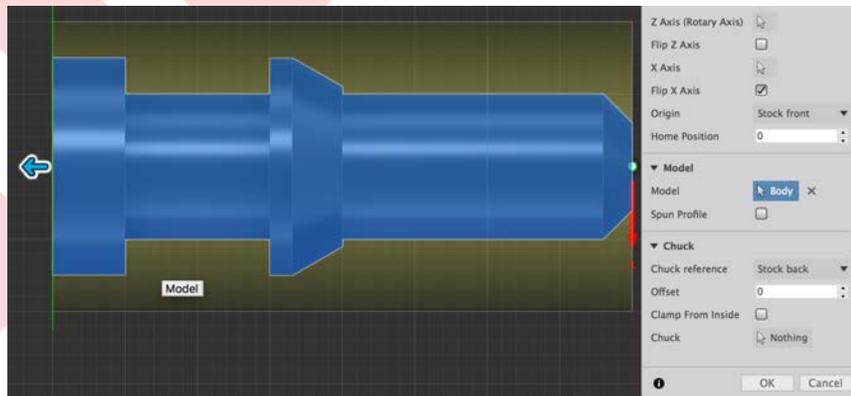


FIGURE 1.17: SELECTED MODEL FOR FUSION TO MACHINE.

Select the **Stock** tab, shown in Figure 1.18. 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 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 in the stock. However, this is optional. If you prefer not to model the stock as its own body for future projects, and instead want to set the stock settings in the Setup window, you may do so. It’s a personal preference. But for the purposes of this training, you will use the “from solid” option. 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 1.19. 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. **Probe WCS override** should automatically change to 2, as well. This setting is involved if using probing operations (which we are not), but it is a good habit to set this as well if it did not automatically. Select **OK**.

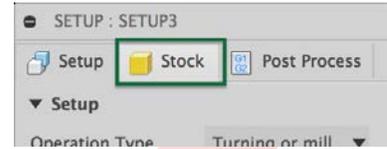


FIGURE 1.18: CHANGING THE SETUP SETTINGS TAB TO “STOCK.”

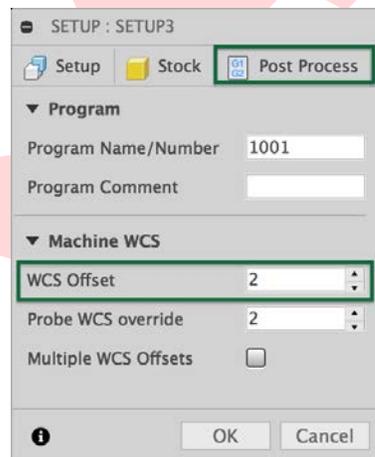


FIGURE 1.19: SETTING THE WORK COORDINATE SYSTEM (WCS).

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.

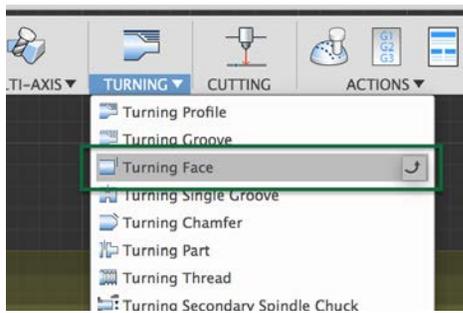


FIGURE 1.20: CREATING A TURNING FACE OPERATION.



FIGURE 1.21: SELECTING A TOOL FROM THE CLOUD LIBRARY WE INSTALLED EARLIER.

First, you will need to face the stock. On the ribbon, under the **Turning** dropdown, select **Turning Face**, shown in Figure 1.20. 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 1.21. Click **OK**.

Ensure that the rest of the settings for this tab match Figure 1.22. 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. Then ensure that you match your **Radii** and **Passes** tabs to Figures 1.23 and 1.24. Click **OK** to save these settings and then you’ve completed your facing operation.

1.7 Creating a Turning Profile Operation

The next step is to create the turning profile of the piece. To do this, return your attention to the Ribbon, and under the **Turning** tab, select **Turning Profile**, Figure 1.25. If it is not already selected, select **Tool 51**, just as you did for the creation of the Facing operation. Ensure your **Tool** and **Passes** tabs match Figure 1.27. 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 1.26. Another

[†]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.

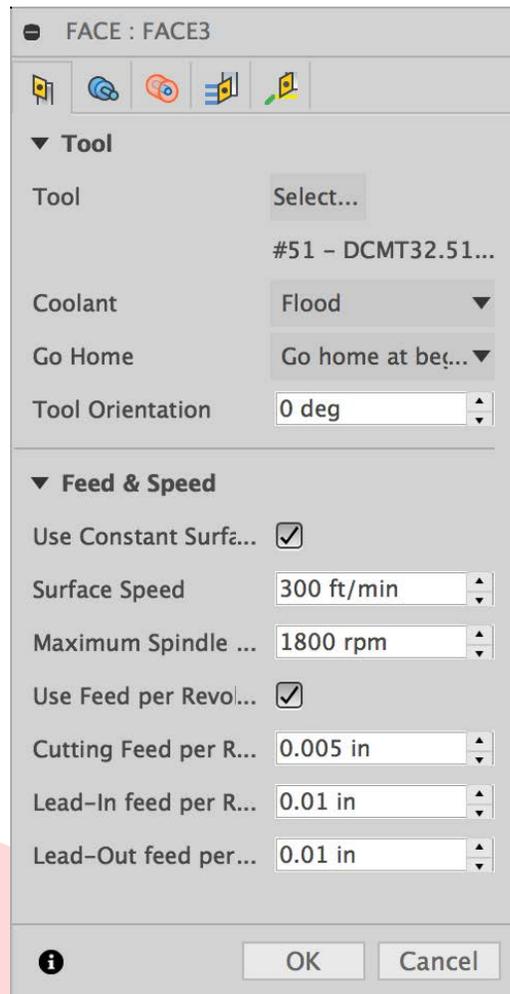


FIGURE 1.22: SETTING THE TOOL'S FEED AND SPEED SETTINGS.

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](#).

[‡]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. This varies the rotational velocity of the stock as the tool approaches the center of the stock, so that the point where the stock is making contact with the tool is experiencing the same speed. Feeds refer to how quickly the tool moves towards, away from, and during a cut.

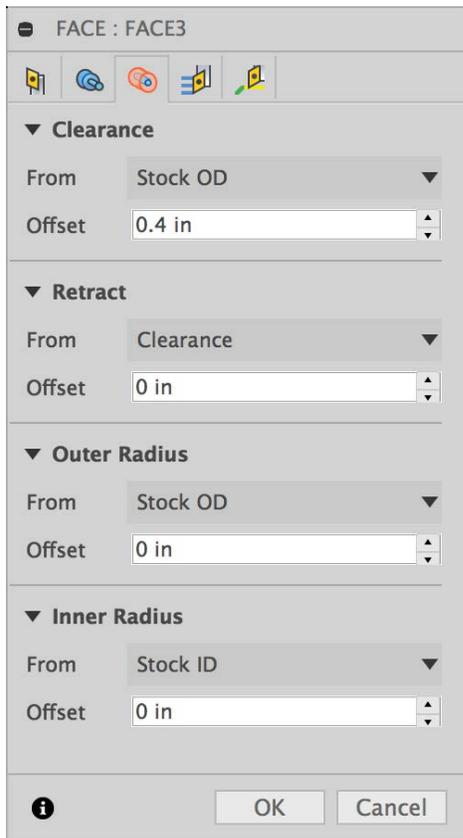


FIGURE 1.23: APPROPRIATE RADII SETTINGS FOR THE FACING PASS.

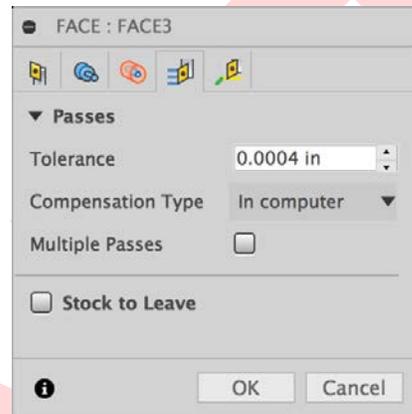
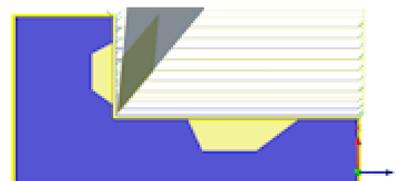


FIGURE 1.24: APPROPRIATE PASSES SETTINGS FOR THE FACING PASS.



FIGURE 1.25: CREATING A TURNING PROFILE OPERATION.



Don't allow grooving

FIGURE 1.26: SCHEMATIC SHOWING NO GROOVING IN THE TURNING PROFILE OPERATION.

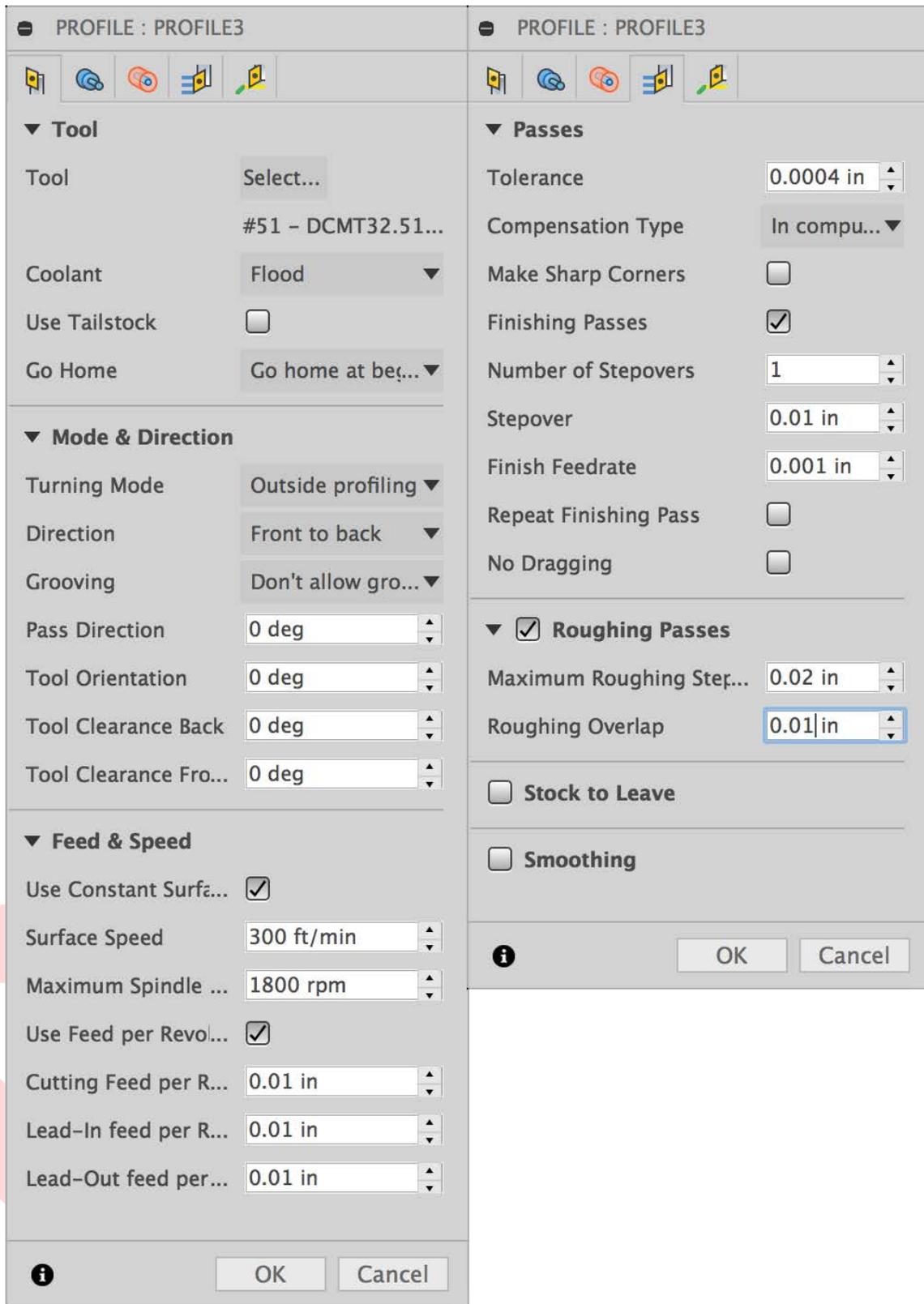


FIGURE 1.27: APPROPRIATE SETTINGS FOR THIS PROFILE'S TOOLING PATH.

1.8 Checking Created Paths

If you click on [T51] Profile in the Project Browser, you can see the tooling paths created for this profiling step. Your screen should match Figure 1.28. 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.

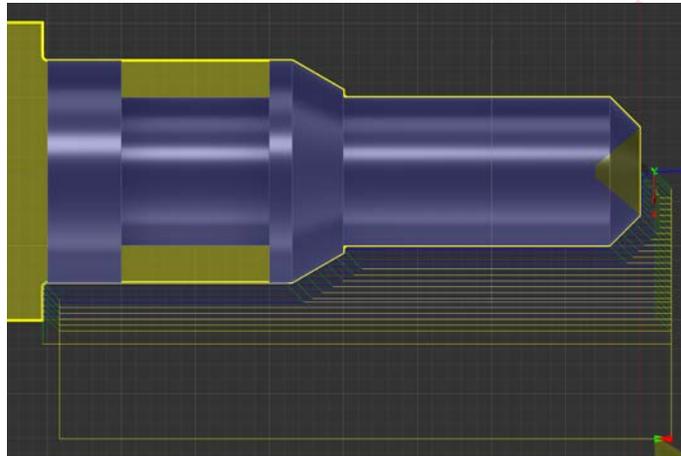


FIGURE 1.28: TOOLING PATHS IN THE FUSION WINDOW.

1.9 Creating a Turning Groove Operation

Next, we will perform the grooving operations. 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 1.29. Click [OK].

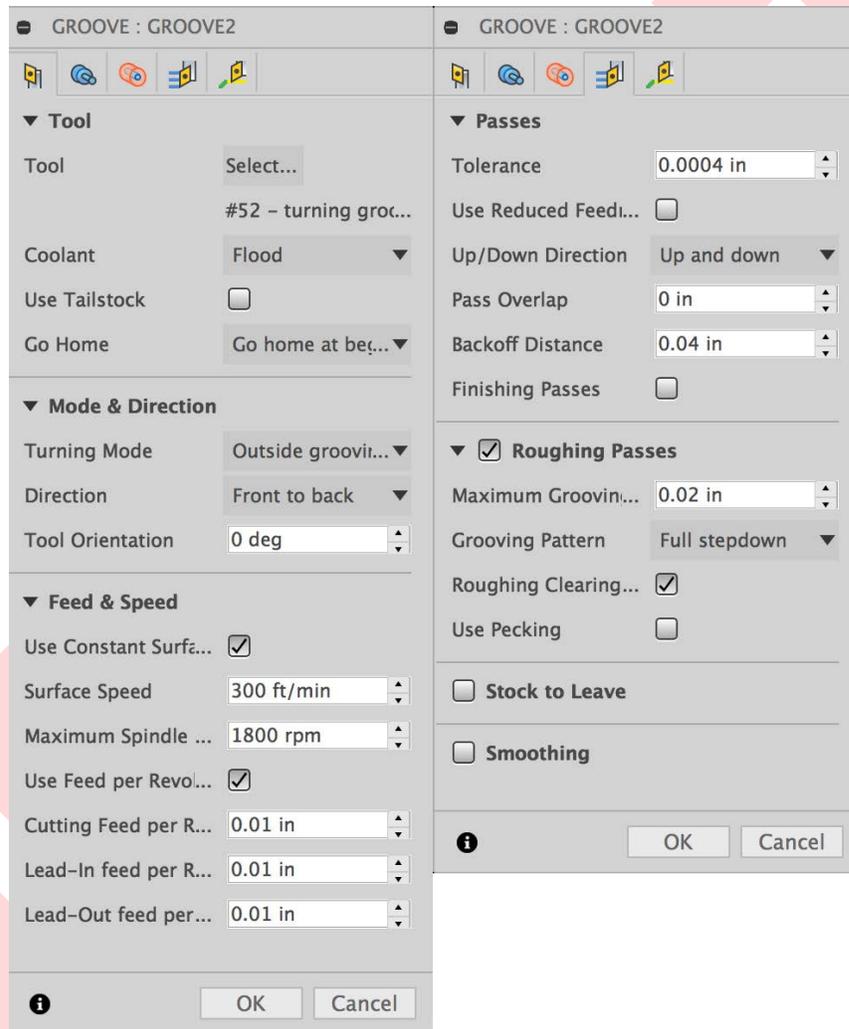


FIGURE 1.29: GROOVING OPERATION SETTINGS.

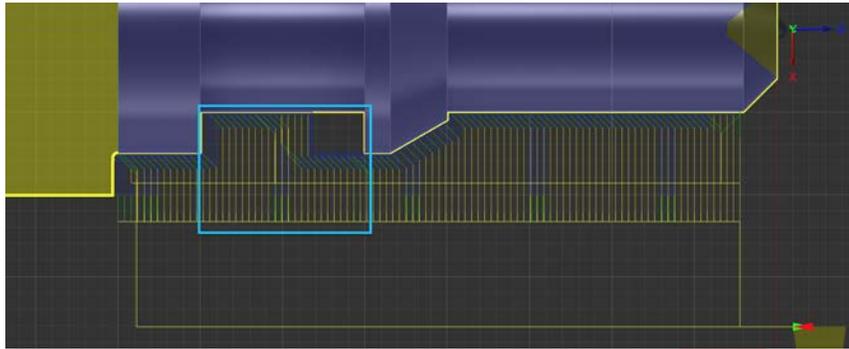


FIGURE 1.30: GROOVING OPERATION PATHS THAT NEED TO BE EDITED.

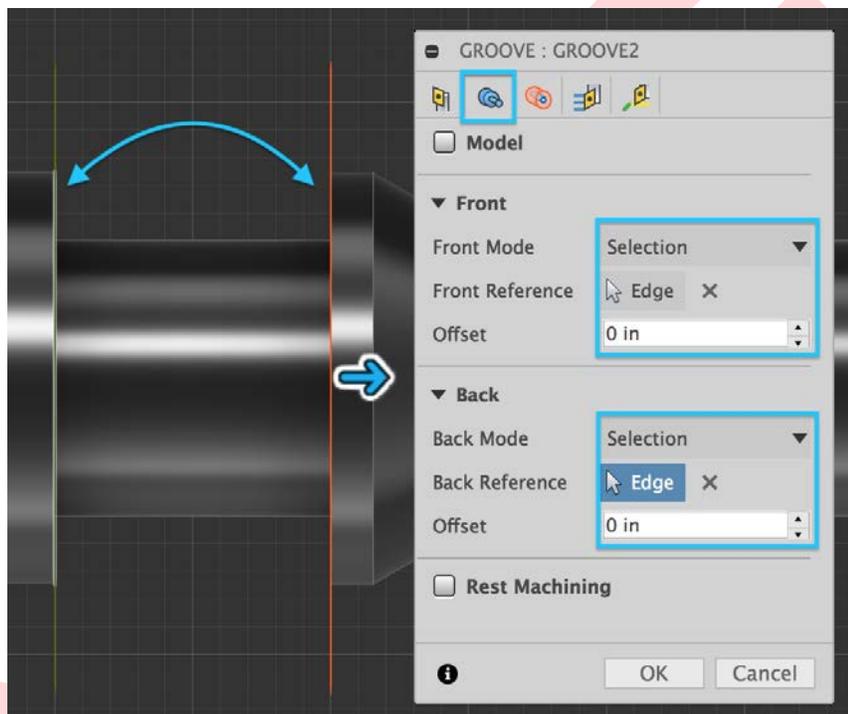


FIGURE 1.31: APPROPRIATE SETTINGS TO RESTRICT THE GROOVING TOOL TO ONE SPECIFIC AREA OF THE MODEL.

You may notice that the grooving tool is attempting to groove the entire part, Figure 1.30, rather than just the channel that we haven't grooved yet, marked in Figure 1.30 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, and 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

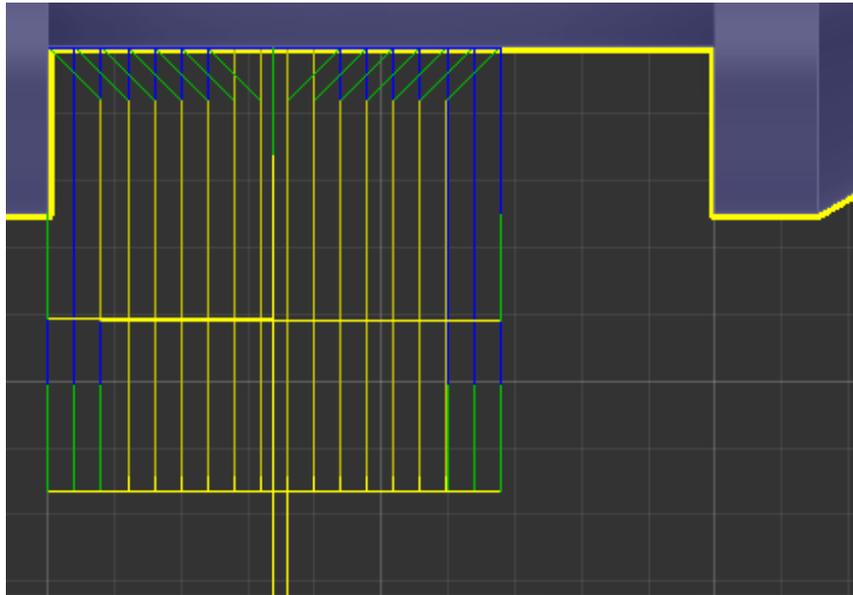


FIGURE 1.32: PROPER GROOVING PATHS.

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 1.31. Click . Does your grooving path now match Figure 1.32? If so, you may continue.

1.10 Creating a Parting Operation

The last operation is to part the piece. Similar to before, create a . Ensure that is selected. Match the settings to 1.33. Fusion should automatically select the back face of the piece. If it does not, go to the tab, and ensure “Back mode” is set to . Click .

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 identify issues and double-check the tool paths. Right-click on the we’ve been working on in the Project Browser. Select , 1.34. Turn off the lightbulb next to “Part” in the Project Browser. Match your simulation settings to 1.35. Click button, 1.36. 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

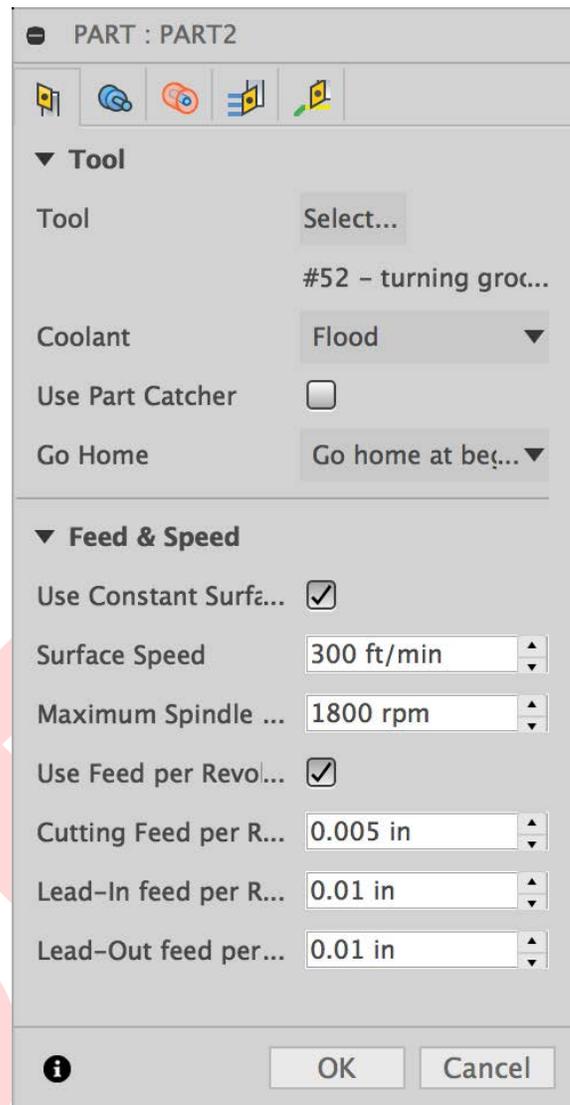


FIGURE 1.33: PROPER PARTING SETTINGS.

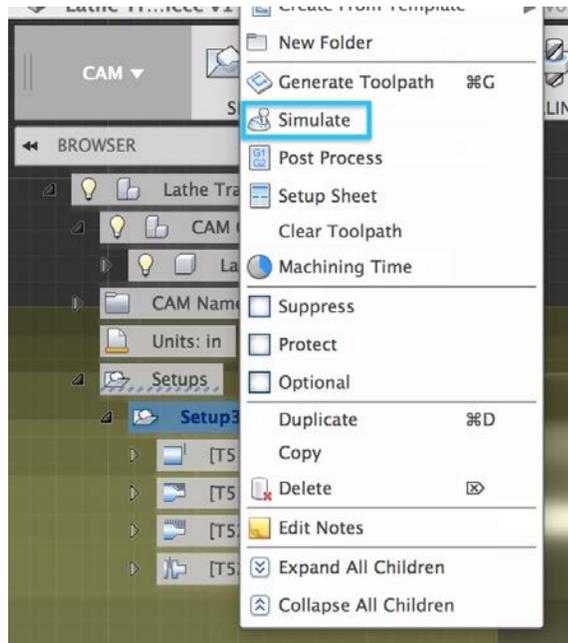


FIGURE 1.34: RIGHT-CLICK MENU OF A SETUP.

are done with your tool paths. This is the most complicated part of the tutorial; well done, so far!

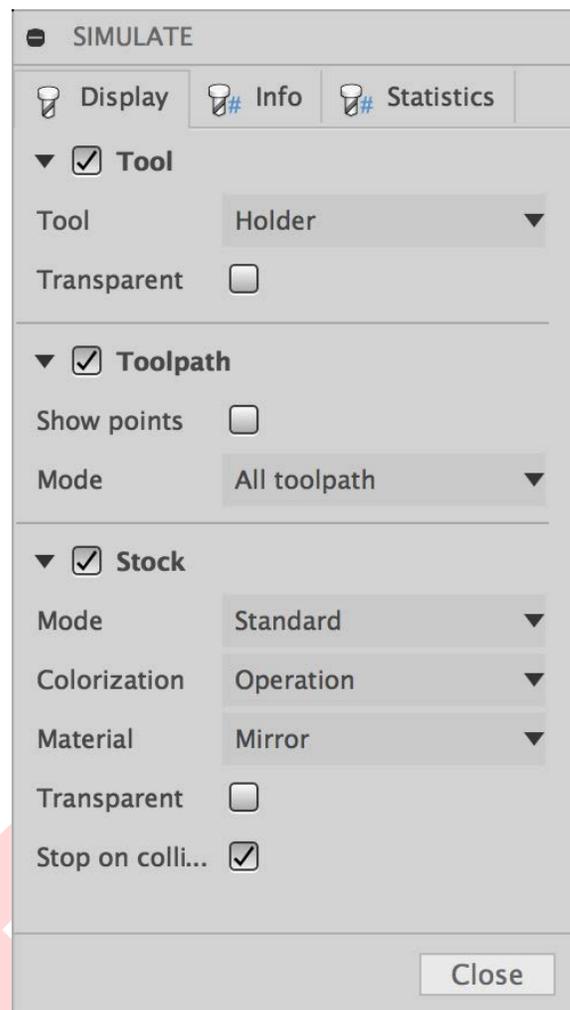


FIGURE 1.35: APPROPRIATE SIMULATION SETTINGS.

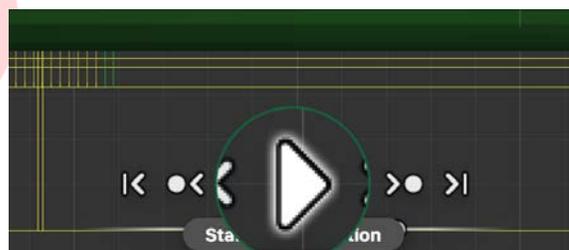


FIGURE 1.36: STARTING A SIMULATION.

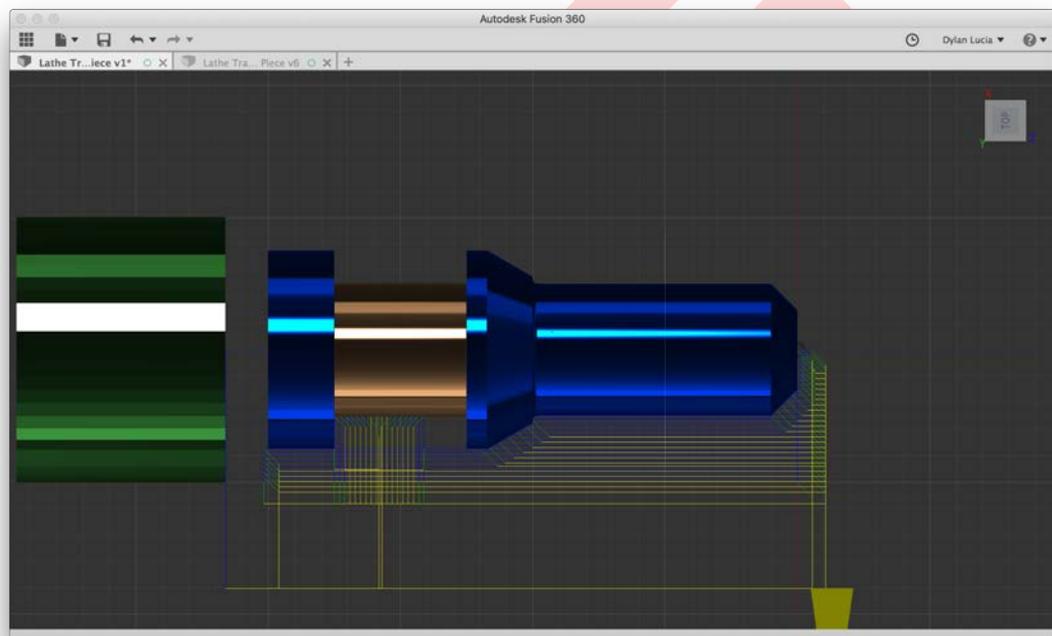


FIGURE 1.37: PROPER SIMULATION RESULT.

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

Click **Post-Process** in the Ribbon, Figure 2.1. In the popup window, change the “Source” to **My Cloud Posts**. If it does not automatically select, change the “Post Processor” to **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**.



FIGURE 2.1: POST-PROCESS BUTTON IN THE RIBBON.

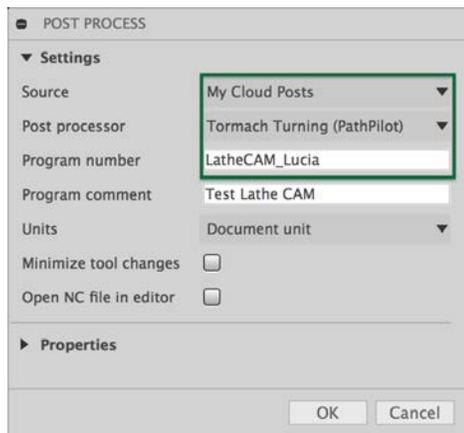


FIGURE 2.2: POST-PROCESS SETTINGS.

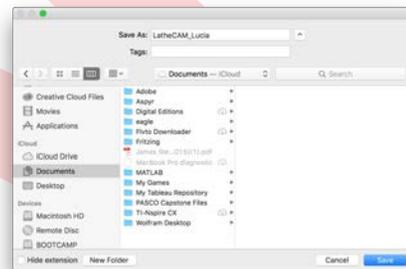


FIGURE 2.3: SAVING THE FILE TO THE COMPUTER.

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.

DRAFT

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.

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 to the fore of the machine, 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.

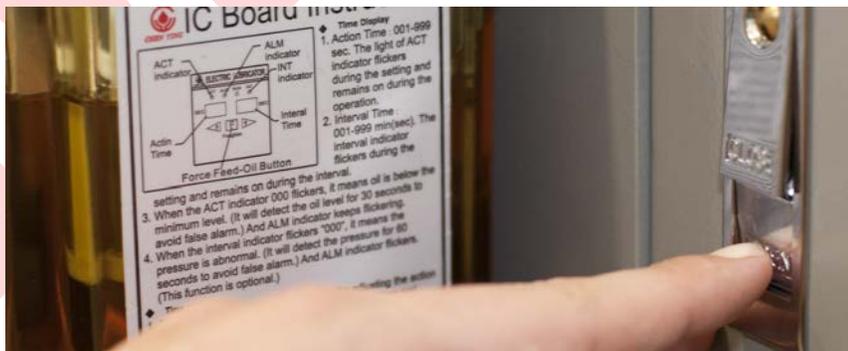


FIGURE 3.2: OPENING THE LOCKING MECHANISM ON THE COMPUTER CABINET.



FIGURE 3.4: BOTH FILE MANAGERS.



FIGURE 3.5: COPYING THE FILE FROM THE USB ONTO THE LATHE HARD DRIVE.

3.2 Transferring the G-code onto the USB flash drive

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.

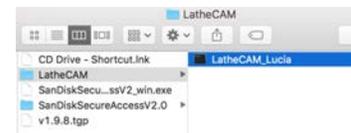


FIGURE 3.3: THE CORRECT FOLDER TO PLACE THE G-CODE IN THE USB.

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

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.

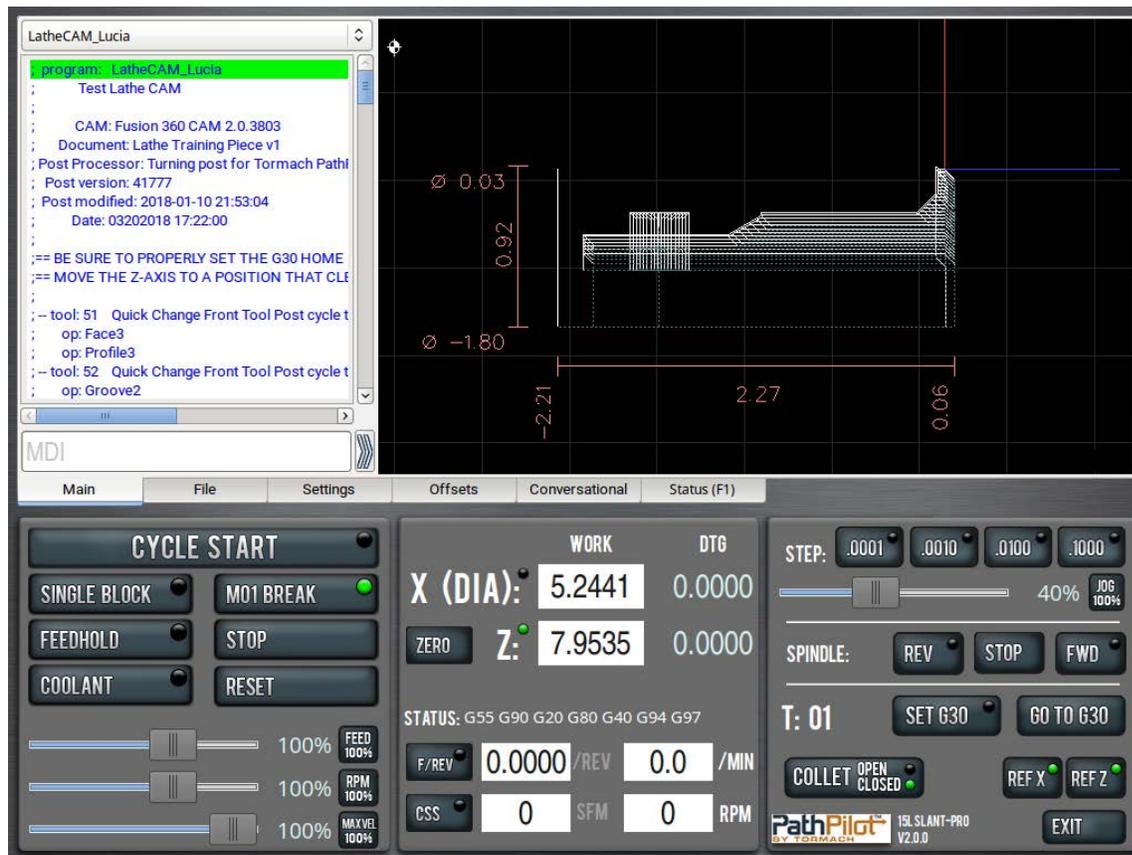


FIGURE 3.6: THE MAIN WINDOW WITH LOADED G-CODE.



FIGURE 3.7: GO TO G30 BUTTON.

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.

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, 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. 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.

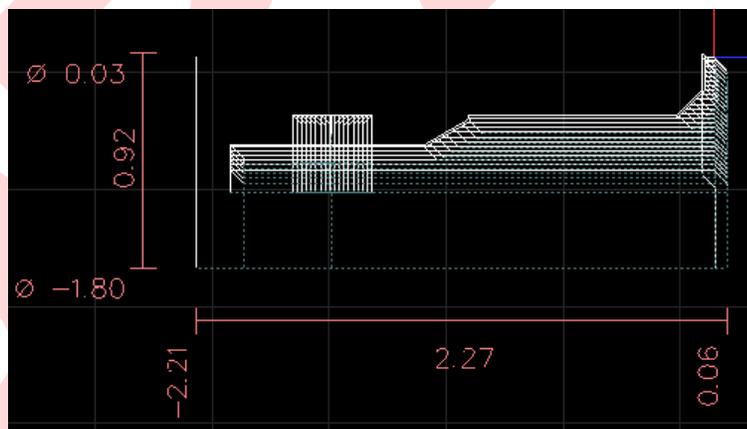


FIGURE 4.1: PROPER PATHS ON THE LATHE MONITOR.

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

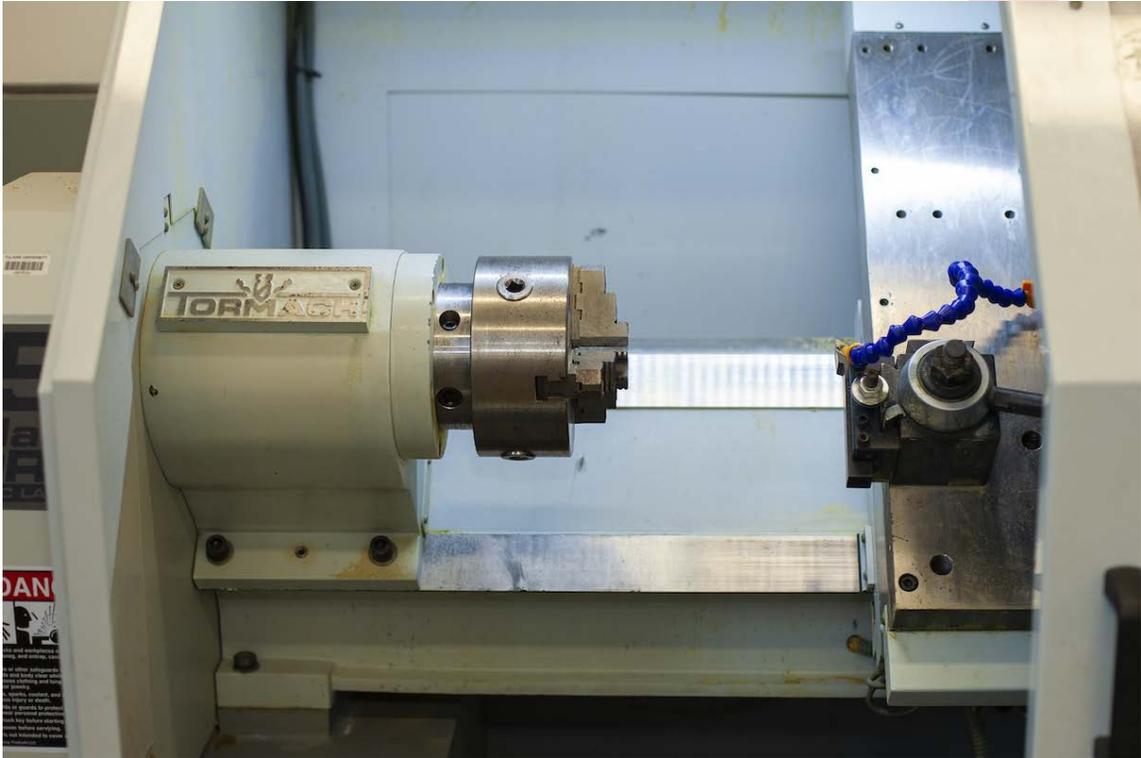


FIGURE 4.2: TOOL 51 APPROXIMATELY CENTERED IN THE MACHINE.



FIGURE 4.3: ZEROING THE AXES FOR RUNNING IN AIR.

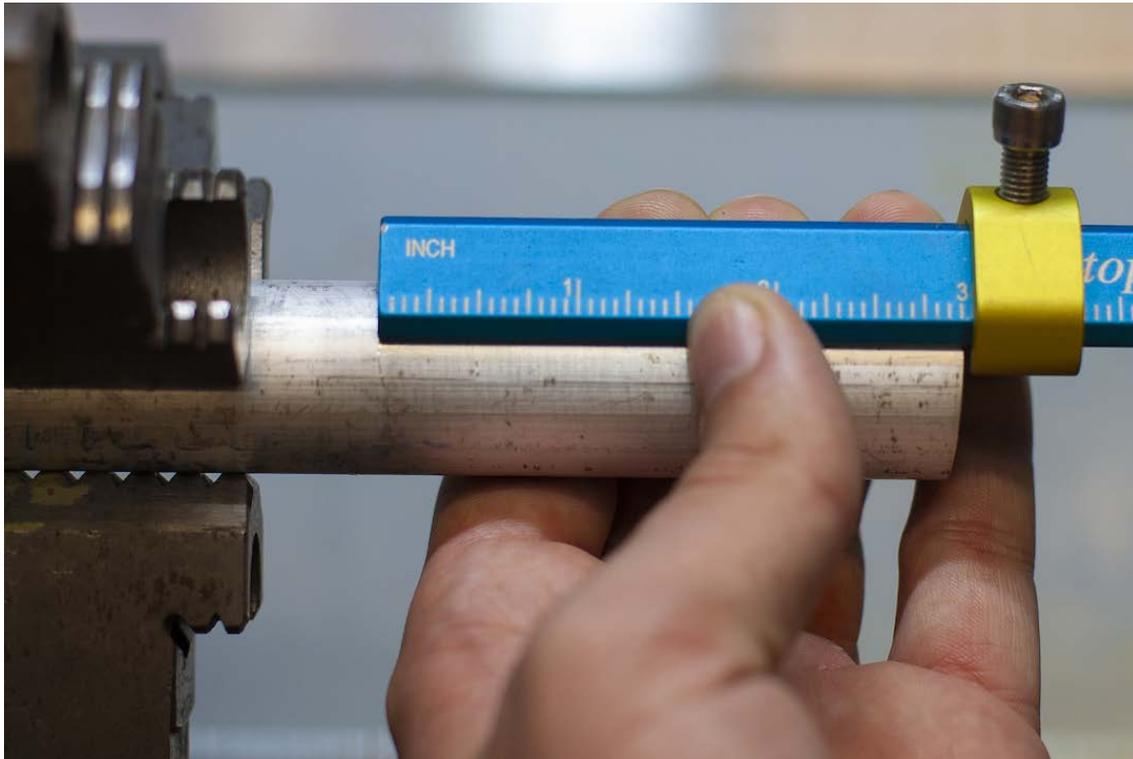


FIGURE 4.4: CHECKING STOCK/CHUCK CLEARANCE WITH THE STOP-LOC.

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, 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.

4.3 Zeroing the Machine to the Stock

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 $\frac{1}{10}$ th of an inch away. Next, use the stepping wheel to slowly pinch the feeler against the face of the stock, until it just barely supports its own weight, Figure 4.6. On the monitor, enter 0.0094 for the "Z" value, Figure 4.7. 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

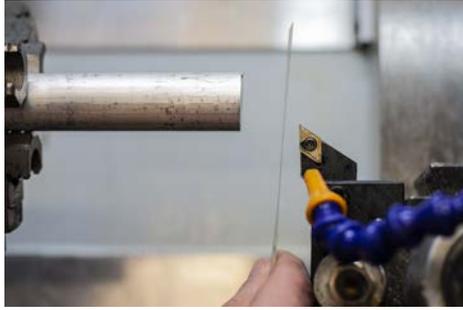


FIGURE 4.5: TOOL 51 1" FROM THE FACE OF THE STOCK.



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.

the stock as close to the point where the tool is touching as is possible, 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 .

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 . Then 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 and noting the point of rotation on the face, Figure 4.12.

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. You can navigate this window using the

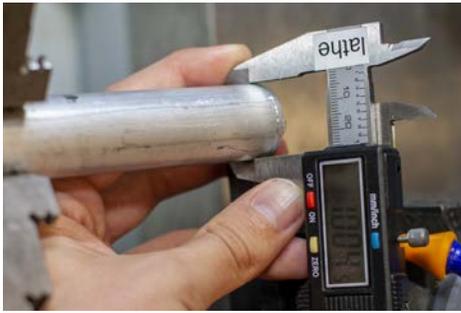


FIGURE 4.10: MEASURING THE DIAMETER OF THE STOCK.



FIGURE 4.11: ZEROING THE X-AXIS.



FIGURE 4.12: VERIFYING THE ZEROING ON THE STOCK.

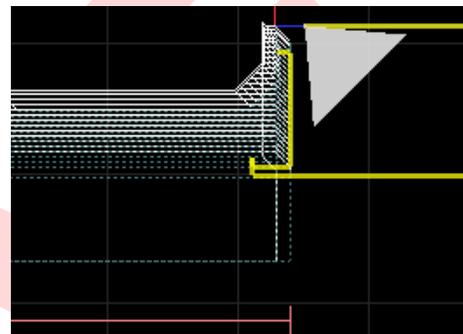


FIGURE 4.13: VERIFYING THE ZEROING ON THE PATHS.



FIGURE 4.14: THE COOLANT BUTTON ON THE MONITOR.

middle-mouse button to pan, and scrolling to zoom. Jog back to a safe distance and click `Go to G30`.

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.



FIGURE 4.15: APPROPRIATE COOLANT SPRAY FOR TOOL 51.

Chapter 5

Running the Program on the Lathe

5.1 RH Turning Tool Operation

Close the door. 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%. Then verify that the "Z" value on screen is approximately an inch. Raise the velocity to 10%. Allow the Lathe to return to G30 by itself and 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.

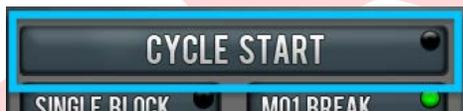


FIGURE 5.1: CYCLE START BUTTON.



FIGURE 5.2: THE HIGHLIGHTED G-CODE INDICATING THE FIRST REQUIRED TOOL.



FIGURE 5.3: WHAT THE PIECE SHOULD LOOK LIKE SO FAR.

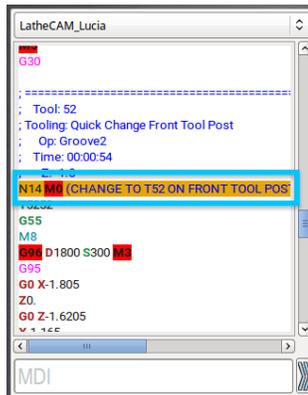


FIGURE 5.4: HIGHLIGHTED G-CODE INDICATING A TOOL CHANGE.



FIGURE 5.5: VERIFYING THE COOLANT SPRAY FOR TOOL 52.

5.2 Changing the Tool

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. Insert Tool 52 and aim the coolant hose, ensuring its spray is adequate by following the same process you did above, 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`. 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.

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.



FIGURE 5.6: THE FINAL PIECE AFTER PARTING.

Index

- .nc file, 23
- A360:CAMPosts, 3
- A360:CAMTools, 3
- Assets, 3
- Autodesk, 1, 2
- Autodesk A360, 3, 21
- Axes, 6
- Bezier, 1
- Bodies, 5
- Brackets, 21
- Cabinet, 25
- Calipers, 31
- CAM, 1
- CAM Workspace, 6
- Chips, 33, 36
- Chuck, 31
- Clearance, 31
- Cloud Libraries, 2, 9, 13
- Cloud Post-processor, 21
- Compressed air, 36
- Computer, 25
- Constant surface speed, 10
- Conversational, 1, 9
- Coolant, 33, 36
- Curve, 1
- Cutting feed per revolution, 9
- Data Panel, 2, 3
- Door, 35
- Facing operation, 1, 9
- Feeler, 31
- File manager, 26
- Flipping axes, 6
- Fusion 360, 1, 3, 5
- G-Code, 3, 8, 21, 23, 26
- G30, 27
- Geometry, 9
- Grooving, 9, 13
- Jog, 27, 31
- Knurling, 5
- Lathe, 1, 25
- Lead in/out, 13
- Lock, 25, 26
- Maximum spindle speed, 9
- MDI, 29, 36
- Model Workspace, 6
- Monitor, 25
- New Project, 2
- Offsets, 9, 16
- Operation Type, 6
- Part model, 2, 3, 5
- Parting, 16
- Passes, 9, 13
- PathPilot, 1
- Paths, 3, 13
- Post-Processing, 8
- Post-Processor, 2, 3
- Preferences, 2
- Probe WCS override, 8
- Program Number, 21
- Project, 2, 3, 5

Project Browser, 5, 8, 13, 16

Radii, 9

RH turning tool, 9

Rhino 3D, 1

Ribbon, 6, 9

Running in air, 29

Safety, 5

Setup, 6, 8, 16

Simulation, 16

Speeds/feeds, 10

Stepping wheel, 31

Stock, 1, 5, 8, 31

Stop-Loc, 31

Threading, 5

Tool Library, 2, 3

Tool numbers, 9

Tools, i, 2, 13

Tormach, 1

Troubleshooting, 15, 16

Turning Operations, 9

Turning Profile, 9

USB flash drive, 25, 26

Velocity slider, 35

View controller, 6

Work Coordinate System (WCS), 6, 8

Workspace, 6

Zeroing, 31, 32