

Tulane University MakerSpace CNC Mill 1 Training



Tormach PCNC 770 Mill

Written by Chase Schober

Revised 2/26/2018

Contents

Introduction.....	3
Turning on the mill	3
Moving the mill.....	6
Loading the stock.....	8
Changing the tool.....	10
Referencing the stock.....	12
Setting up your first operation.....	15
Performing your first operation.....	17
Setting up a profile cut.....	21
Performing a profile cut.....	24
Setting up a pocket cut	24
Performing a pocket cut.....	26
Setting up a counter bore.....	27
Performing a counter bore cut.....	29
Setting up a spot drill operation.....	30
Performing a spot drill operation.....	32
Setting up a counter sink operation.....	32
Performing a countersink operation.....	34
Setting up a drilling operation.....	35
Performing a drilling operation.....	36
Setting up a second drilling operation.....	36
Performing a second drilling operation.....	37
Setting up an engraving operation.....	40
Performing an engraving operation.....	41
Setting up a second engraving operation.....	42
Performing a second engraving operation.....	42
Flipping the workpiece.....	43
Manually facing with the mill.....	46
Setting up a backside profile operation.....	48
Performing a backside profile operation.....	49
Setting up a backside spot drilling operation.....	49
Perform a backside spot drilling operation.....	49
Setting up a backside drilling operation.....	50
Perform a backside drilling operation.....	50
Setting up a second backside drilling operation.....	50
Perform a backside drilling operation.....	51
Finishing Up	51

Introduction

Welcome to the CNC Mill Training! This manual will walk you through the steps to perform several milling operations using the Conversational Mode of the Tormach PCNC 770 Mill. In this mode, you will input all the necessary information through the computer monitor attached to the mill. This is optimized for simplicity. For complicated parts, Conversational Mode may not be sufficient; more complex parts will require the use of Fusion 360 to design and orchestrate toolpaths. This tutorial will teach you how to operate the mill safely and effectively. It is important to note that this CNC mill is fully enclosed and never exposes any part of your body to a moving tool. Zero accidents have occurred with this CNC mill and we intend to keep it this way. Keep yourself and others safe by maintaining an active safety mentality throughout mill operation. Let's get started!

Turning on the mill

Ask the Training Ninja to double-check that Tool 11 is in the spindle, Tool 43 is loaded with a 3/8" drill bit and has its Z offset recorded, and Tool 42 is loaded with something other than a 1/8" drill bit.

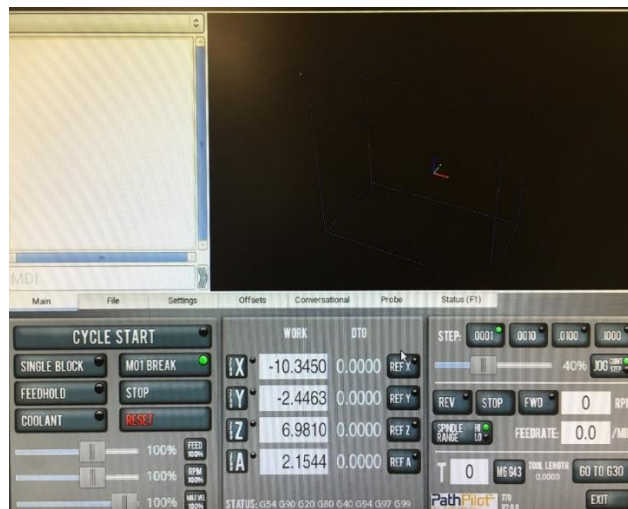
Check your PPE and make sure that you are safely dressed.

Open the mill enclosure door by sliding the two silver handles apart. Open them to their full width.

Inside the back-right of the mill enclosure, to the right of the Series 3 text, there is a yellow and red power switch. You may need to stick your head in the enclosure to see it. When in the "Off" position, the red switch will be pointed in the 12:00 position. To turn on the power, one needs to rotate the red switch 90 degrees clockwise to the 3:00 position. Note, the red part of the switch will fall off from time to time. If this happens, ask the Training Ninja how to re-assemble the piece.



Flipping this switch will cause the computer monitor to boot up. It will take about 30 seconds for the screen to display its various start-up screens before settling into the display screen shown below. Note the flashing red Reset button indicating that the system has not been initiated.

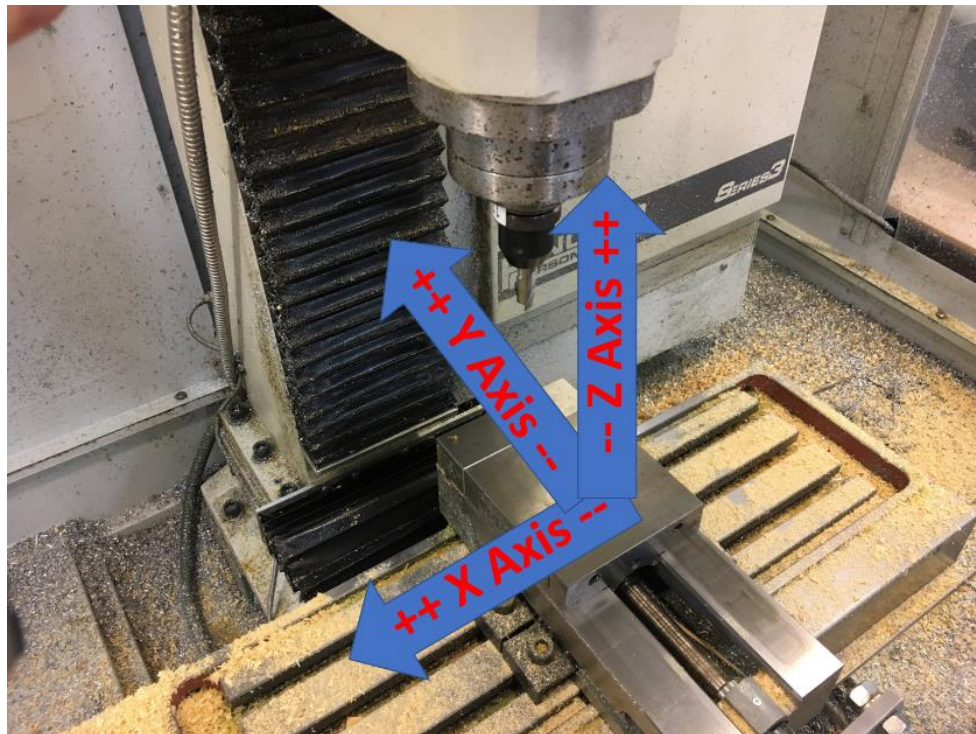


Whenever the CNC mill has been turned off, it must have been E-stopped first. E-stop stands for Emergency stop and is the button responsible for cutting power to the machine in case of an emergency. On this CNC mill, the E-stop is the red button to the right of the enclosure doors. To reset the machine, the E-stop needs to be released. Rotate the red button clockwise about 90 degrees. The button will release and spring out. Once this has occurred, press the green button to the right of the E-stop. You should see the enclosure lights flicker as power is returned to the machine. The E-stop button is covered later in this tutorial and is used to stop the machine in the event of an emergency. Quickly strike the E-stop button to see how it responds. Then, press the green button once more to restore power to the machine.

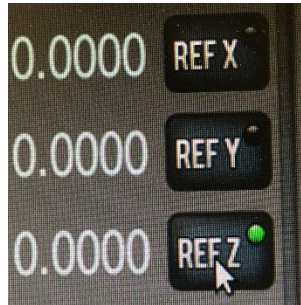


On the computer monitor, click the flashing red "RESET" button. You should hear an audible click and the "RESET" button will stop flashing red.

Now that the machine has been reset, the next step is to reference its axes. For this CNC mill in its particular arrangement, there are three axes, the X, Y, and Z. The X and Y axis are in the plane of the ground, with X referring to left-right movement and Y referring to the front-back movement. The Z axis moves the spindle up and down. For the mill to know where it is in space, we need to reference the machine. This occurs by moving the machine as far as is allowed in one direction until it encounters a limit switch. When the mill strikes this limit switch, it knows where the spindle is in relation to the enclosure bed.



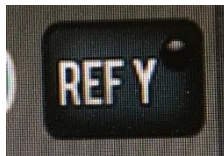
ALWAYS REFERENCE THE Z AXIS FIRST! It is very important to do so because there will never be any obstacles in the positive Z direction. If you had a workpiece loaded into the machine and erroneously chose to reference the X or Y axis first, you could crash the machine. Click the "REF Z" button in the middle of the computer screen. This will cause the machine to travel upwards in the positive Z direction until it encounters the limit switch. Once it stops, notice that the circle to the top right of the "REF Z" button is filled with green. This indicates that the Z-axis has been properly referenced.



Once the green LED next to "REF Z" is visible, click "REF X" and watch as the machine travels to its X-axis limit switch.



Once the green LED next to "REF X" is visible, click "REF Y" and watch as the machine travels to its Y-axis limit switch.

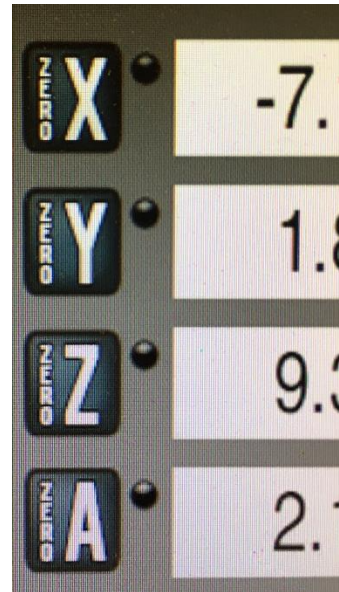


Verify that there are green circles above each of the REF buttons before continuing. Note that there is a fourth letter displayed on the screen, "A". This is used for when the CNC Mill is in its four-axis arrangement. It can be ignored for the purposes of this tutorial.

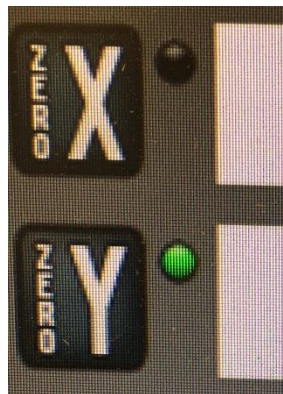


Moving the mill

Now we will get familiar with how to move the CNC workpiece (X,Y axes) and tool (Z). To the left of the keyboard, there is a black device known as a jog shuttle. This is the main method of moving the CNC tool and workpiece between operations. You'll notice that each of the four axes are represented on the jog shuttle as buttons. In the middle of your computer display, to the left of the work coordinates, notice that there is a circle to the top right of one of the four axes. By clicking an axis on the jog shuttle, a green circle illuminates in the corresponding axis on the monitor. This is how we know which axis the jog shuttle will move.



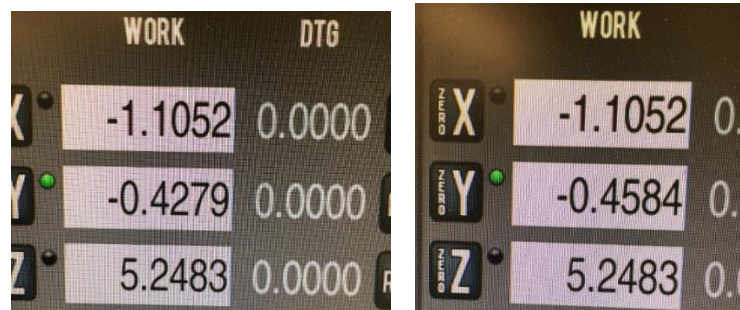
Press the "Y" button on the jog shuttle. Check the computer monitor to verify that a green circle is illuminated to the top-right of the 'ZERO Y' button.



In the center of the jog shuttle, is a rotatable wheel known as the shuttle ring. This is NOT the control with a dimple in it. One can move the mill along the chosen axis in the positive or negative direction by rotating the rubber shuttle ring clockwise or counterclockwise respectively. As shown on the jog shuttle, clockwise rotation means positive movement, counterclockwise rotation means negative movement. Right now, the Y-axis is at its maximum positive (forward) position. Therefore, it can only be moved in the negative direction (towards the back of the machine). Gently rotate the shuttle ring in the counterclockwise direction and then release it. Notice the speed at which the bed adjusts. More angular rotation means faster bed movement.

Ask your Ninja about moving the mill to its opposite Y-axis limit switch. You can move in this direction by rotating the shuttle ring in the counterclockwise direction. The way the vise is currently positioned, it may crash into the machine before the software recognizes that it has reached its limit switch. This will be fixed soon. To prevent this, we need to make sure that the vise is not fully closed, but rather open at least two inches. Ask the Ninja about how to perform this procedure. Once it is safe to do so, move towards the limit switch until the machine prevents any further movement.

Travel back and forth between the Y-axis limit switches to get comfortable driving the jog shuttle. You can see how far you move by monitoring the work coordinate values on the computer display. The images below show movement along the Y axis.



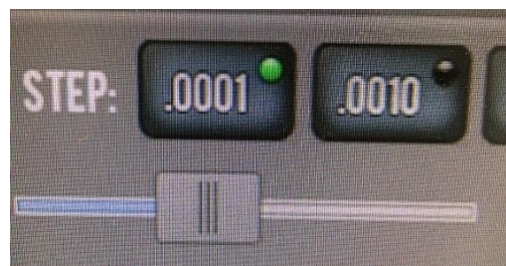
Once you feel comfortable, we will move to the X-axis between its limit switches. Click the “X” on the jog shuttle and verify that the green circle illuminates on the computer display to the top right of the “ZERO X” button. Just like the Y-axis, the mill is at its maximum negative X-axis position. Decide which action is needed to move the mill to its opposite limit switch and perform it.

Travel back and forth between the X-axis limit switches to understand the mill’s directions and limitations. The workpiece can move 14” along the X axis and 7.5” in the Y axis.

Now we will move the tool along the Z-axis. Unlike the X and Y movement, the software will not stop you from crashing in the Z direction. Therefore, we need to be careful as we move the Z axis towards the bed. As you can see, there is currently a tool loaded into the mill. We do not want to crash this tool into the vise or bed.

Press the Z button on the jog shuttle to inform the mill which axis will be moved. Once the green circle next to the “ZERO Z” button on the display has been illuminated, cautiously move the mill in the negative (downward) Z direction until the tool is roughly 3” above the vise. After this is accomplished, move the tool to its Z-axis positive limit switch.

There is one other way to move the tool and workpiece using the jog shuttle. In the center of the jog shuttle is a dimpled circle. This is known as the jog wheel. By rotating the jog wheel, one can move the mill in specific increments. The four possible increments are .0001”, .001”, .01”, .1”. To avoid crashes, we recommend that the step size be .0001”. This allows for precise movement and prevents the machine from lurching into a workpiece when working at close quarters. The step size can be cycled by pressing the “STEP” button on the jog shuttle. As this button is pressed, a green circle will illuminate to the top right of the specific increment on the right of the computer display. Press the “STEP” button until a green circle illuminates to the top-right of the “.0001” button.



Loading the stock

For this tutorial, we will be milling a piece of 2x4 wood stock. We will secure the stock by holding it with the vise.

Ask the Training Ninja to provide you with your piece of wood stock.

Now, we will measure the dimensions of the wood stock with calipers. On the bench against the right side of the mill, there is a black box labeled “CNC mill” that contains a set of calipers. Remove the calipers from the box.



Calipers are precision measuring tools that track the distance between the two metal tips. Slide the adjustable part of the calipers until there is no gap in between the two tips. Press the yellow “zero” button on the calipers, this makes sure that they are properly calibrated. If the units are displayed in mm, press the blue “mm/inch” button on the calipers to toggle to inches.



Open the calipers wide enough to accept the width of the wood stock. Press firmly with your thumb against the adjustable slider so that the two metal tips are pressed against the two faces of the stock. Ask your training Ninja to give you a note card and record the value displayed on the caliper screen as your width.



Repeat this procedure for the length and height of your stock. Record each value.

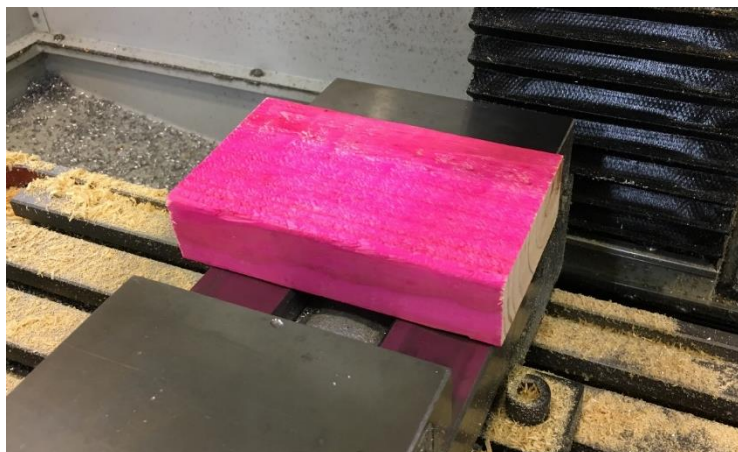


Approach the mill enclosure and inspect the vise that is secured to the mill bed. The vise can be opened or closed by rotating its handle. Counter-clockwise rotation opens the vise and clockwise rotation closes the vise. Open the vise to its full extent.

Each time stock is to be loaded into the vise, care needs to be taken to wipe the vise clean of chips and debris. Using a rag provided by the training Ninja, comprehensively wipe the inside faces of the vise clean.



Place your stock in the vise so that it sits as shown below (need image). Notice how only $\frac{1}{4}$ " of the stock is above the vise. It would be very difficult to perform an operation to trim the stock without risking a tool crash into the vise. Therefore, we will use parallels to raise the height of the stock in the vise. Parallels are thin, flat pieces of metal with parallel sides that come in exact pairs. They are used extensively while working with this CNC mill.

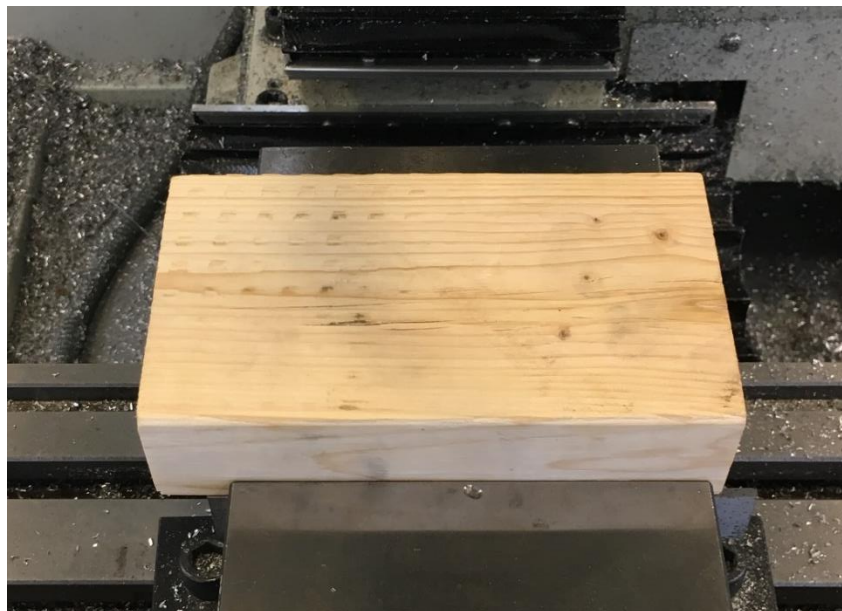


On the shelf behind the CNC mill, to the left of the wooden tool rack, there is a blue box. In it are a set of parallels that range in height from $\frac{1}{2}$ " to $1\frac{5}{8}$ ". Grab the $\frac{7}{8}$ " pair from the set (fourth pair).

Remove your wooden stock from the vise. Place one parallel on edge against the back face of the vise, and one against the front face as shown below. Make sure that the parallels aren't resting on any chips or debris in the vise.



Rest your wooden stock on top of the back parallel and close the vise until the wood is sitting on each parallel. Once the wood is in the correct position, place your palm on top of the wood and apply pressure to ensure that the wood is resting flat on the parallels. Rotate the vise until you feel resistance, then rotate a quarter turn farther to make sure that the stock is secure in the vise. In general, it is better to lean on the side of overtightening as this reduces the chance that your stock will loosen while milling. However, in the case of softer materials such as wood or aluminum, overtightening can cause deformation and decreased accuracy.



Changing the tool

The tool currently loaded into the spindle is a $\frac{1}{2}$ " flat end mill. For this mill, individual tool numbers are assigned to each tool so that they can be referenced by the machine and other computer programs. This is tool number 11. You will notice that the tools for this mill are kept in the wooden rack on the shelf behind the mill. Each tool position is labeled with its classification, and each tool is labeled with its number. For our purposes, we will swap Tool 1, the Offset Tool, into the spindle.

Remove Tool 1 from the tool-holding structure and place it on the bench across from the CNC mill.

The current tool is removed from the spindle by accessing the motor cover. On the right side of the motor cover, there is a black clasp that secures the door. Undo this clasp by lifting the near side with your thumb and using your middle finger to unlatch the clasp. Swing the motor cover door open once the clasp has been released.



Once the door has been opened, you can see the drawbar. This is connected to the spindle and translates the rotation of the belts to the tool. Notice how spinning the drawbar spins the tool. In order to loosen the tool, we will need to prevent the rotation of the spindle. Spin the drawbar until the flat side of its mounting cylinder is parallel to the enclosure doors. This will allow the spindle lock to engage. Pivot the black spindle lock from its resting position onto the drawbar mount to prevent spindle rotation.



Spindle Lock **Drawbar**



Take the brass-headed hammer from beside the keyboard. Its open side (wrench) fits onto the top of the drawbar and is used to loosen it from the spindle. Keep a hold of the $\frac{1}{2}$ " end mill as you loosen the spindle! When the spindle is fully loosened the tool could drop, damaging the tool or your workpiece.

Push the handle of the hammer in the counterclockwise direction while holding onto the tool. After rotating it $\frac{1}{4}$ " turn, reposition the hammer and perform another $\frac{1}{4}$ turn. This should sufficiently loosen the tool in the spindle. Flip the hammer and use the brass side to rap the top of the drawbar WHILE HOLDING THE TOOL. The tool should release into your hand. Take Tool 11 and place it in its designated toolholding spot.

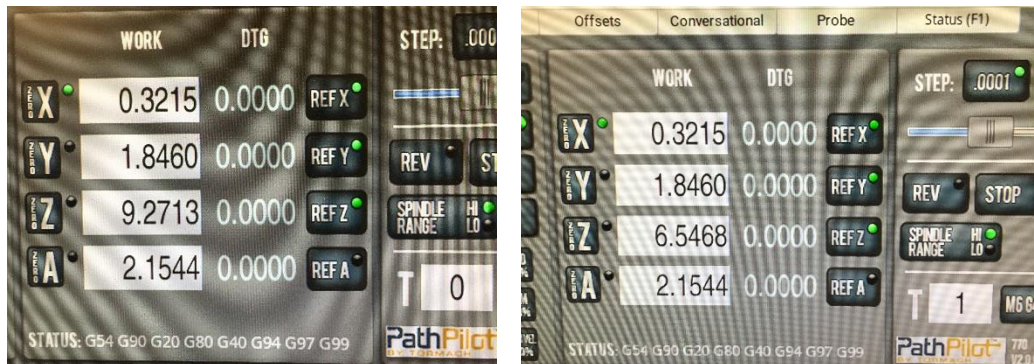
Now we will place Tool 1 into the spindle. With your left hand, lift Tool 1 firmly into the spindle and hold it there as you rotate the drawbar clockwise with the hammer-wrench. Perform two $\frac{1}{4}$ turns and lean on the side of overtightening in this case.

Once Tool 1 is secured, release the spindle lock by pivoting it back into its vertical resting position. Then, swing the motor cover closed and secure its black clasp.

Referencing the stock

As mentioned previously, each tool in the tool library is assigned a designated number. This is important because each tool has a different length when measured from top to bottom. In the computer, the height of each tool is recorded so that the mill can perfectly know the coordinate of the tool tip. Each time there is a tool change, it is vitally important to tell the mill this information. **NOT UPDATING YOUR TOOL NUMBER CAUSES THE MAJORITY OF CRASHES.**

In the bottom-right corner of the screen, there is a text box to the right of a "T". The number displayed in this text box is the tool that the mill thinks it is equipped with. Currently, the text box should display "0" because the mill was just reset and didn't retain this information. To update this, highlight the "0", type "1" since we just equipped Tool 1, and HIT ENTER on the keyboard. Notice that the Z coordinate text box updated to reflect the height of the new tool.



Now, we want to inform the machine the coordinates of our stock. To do this, we will use the offset tool to "touch off" on the stock.

Use the jog shuttle to move the X and Y axis so that the offset tool is roughly centered over the stock. Pro tip! Look through the side window to double-check your Y position.

Carefully lower the offset tool in the negative Z direction until the top of the tool is about 2" above the stock top. Re-center the stock in the X and Y direction if necessary.

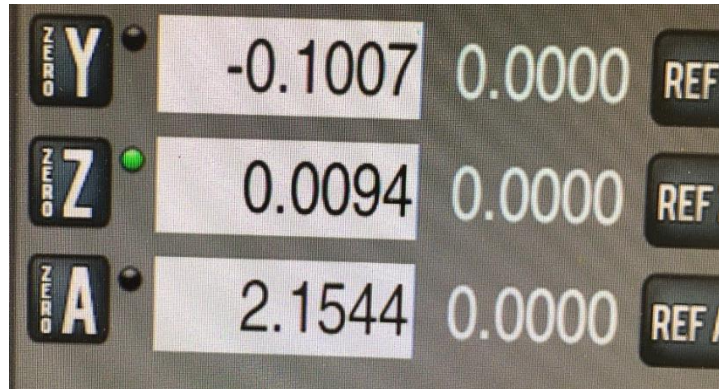
Now, we will move our tool above the highest part of the stock. In general, the stock will not be completely flat on its top face. Rather, it will have a varying topography. We want to set the highest part of the stock as our top face in the computer. This way, the tool won't crash into the stock.

Under the keyboard is a clear piece of plastic known as a feeler gauge. This piece of plastic is 0.0094" thick. To reference the stock, place this gauge between the stock and the offset tool. We will slowly lower the offset tool until it is exactly .0094" (the width of the plastic gauge) away from the stock. The label on the gauge is thicker than .0094" so use the other plastic area as your reference.

Begin SLOWLY lowering the offset tool in the Z direction while sliding the gauge back and forth. The idea is that as soon as you can no longer easily move the gauge back and forth, the offset tool is the width of the gauge (0.0094") away from the top of the stock. Be careful not to crash the tool by moving too quickly. As you begin to feel slight resistance, it is useful to use the jog wheel to make fine increments. Before you do, check to make sure that the step size is ".0001".

Adjust the height until you can barely move the gauge back and forth. Ask your Training Ninja to demonstrate the correct pressure. In general, the correct pressure is when the gauge slides as you pull it, but bends as you push it.

Highlight the text box to the right of the "ZERO Z" button. Type ".0094" and HIT ENTER on your keyboard. The mill now correctly thinks it is positioned .0094" above the top of the stock.

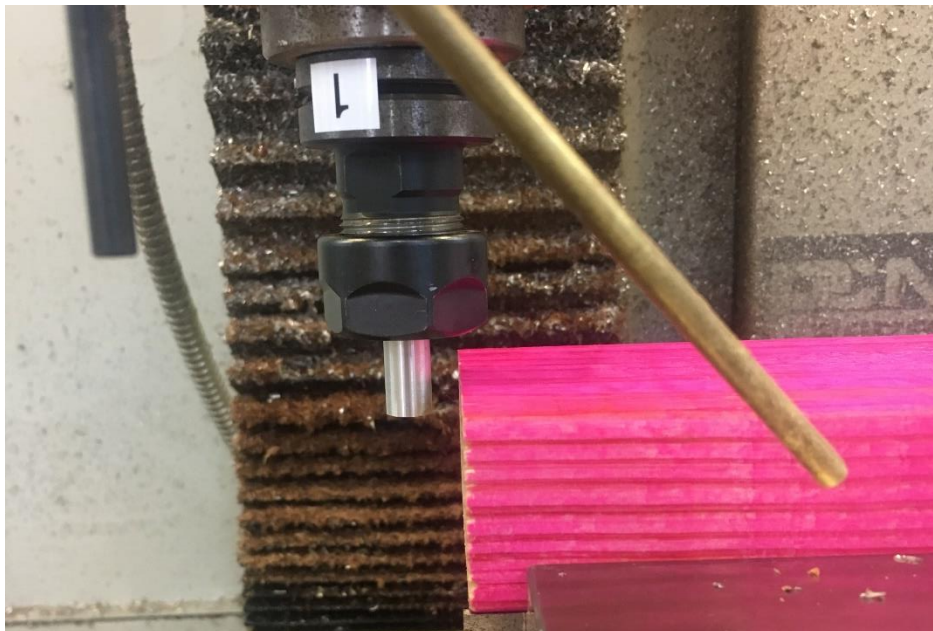


Raise the Z height until the Z-coordinate value is exactly 0.5000. This is a good chance to practice using the jog wheel to make precision adjustments.

Now, we want to reference the X and Y axes. In almost all instances, we want to set X and Y equal to zero at the exact center of the stock.

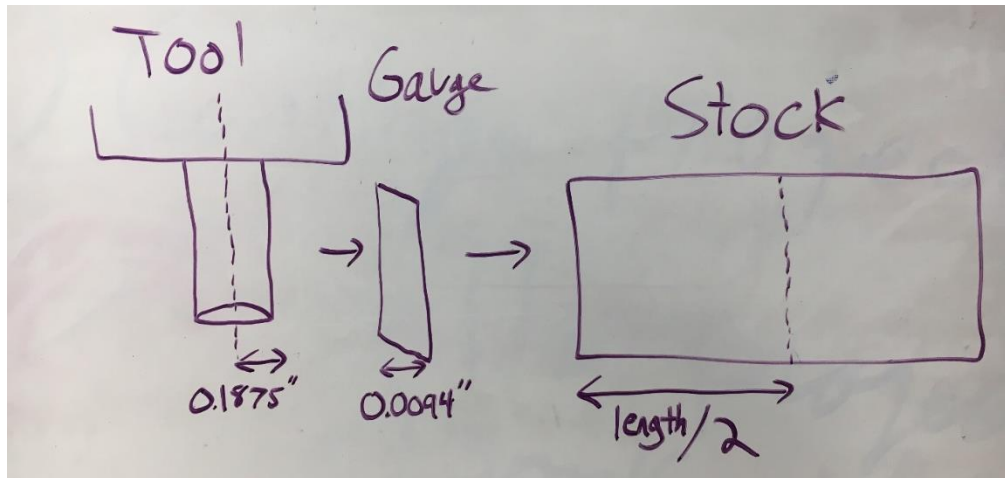
Drive the offset tool to the left of the stock until the metal cylinder on Tool 1 can lower without crashing into the stock. Give it a little room to spare for caution.

Lower the Z height until the coordinate value is -0.2500. This is a relatively trivial value and is expressed here to practice reaching a specified coordinate. A picture of the correct position is shown below



Now, we will perform the same gauge sliding action as we did when referencing the Z height. Move the tool in the positive X direction while sawing the gauge back and forth. Adjust until you can barely move the gauge back and forth. Adjust until you can barely move the gauge back and forth.

Now, we want to set the center of our stock to be equal to a work coordinate of zero. To do this, we need to perform a little math. First, we know that the center of our spindle is a certain distance away from the left face of our workpiece. This distance is the thickness of the plastic gauge (.0094") added with the radius of Tool 1's metal cylinder (.1875"). Then we know that the left face of our workpiece is $\frac{1}{2}$ the length of our workpiece away from the center of the workpiece. Therefore, we are $(.0094" + .1875" + (1/2 * \text{measured length of our workpiece}))$ away from the center of the workpiece. As leftward X axis movement is negative in this coordinate system, the value we enter will be negative. For example, if the workpiece length was 6", we would enter $-(.0094" + .1875" + 3")$ or (-3.1969) we would enter this value into the "ZERO X" text box. Highlight the "X ZERO" text box, type your calculated value and HIT ENTER. The X=0 coordinate is now directly in the center of your stock.



Now, we will perform the same operation to reference the Y axis. There are no instructions for this portion of the tutorial as it is important to practice this type of problem solving on your own. However, the training Ninja is available to answer any questions you have.

Once all axes have been referenced, move the mill to the X=0", Y=0", Z=0.1" position without crashing the tool. The tool should appear to be directly in the center of the stock, and .1" above the stock face. If this is not the case, consult your training Ninja to determine where the error occurred. If the tool looks centered, the stock is completely referenced! Raise the tool to the positive Z limit switch.



Setting up your first operation

Now that the stock has been referenced, it's time for the fun part! On the computer monitor, in the center of the screen, click on the tab labeled "Conversational". This is one of the ways that we can write operations for the mill to perform.

At the top of the screen, click the tab labeled "Face". A facing operation cuts away the top part of the stock. It is almost always the precursor to any other milling operation, and is used to flatten and smooth the top surface. In this operation, we will face 0.1" off the top of our stock.



There is a lot of information to cover in this section and so each text box on the screen will be broken down and explained individually.

The "Title" box in the upper-left of the screen is arbitrary and usually we don't bother to change the text (you can save the file as a custom name later).

The "Work Offset" box should be filled as "G54". The mill has several work coordinate offsets that can be used and all the information in its current arrangement is stored in G54.

The "Tool" box should be filled with "3". Tool 3 is known as the Shear Hog and is our go-to tool for removing large amounts of material. We will be using this tool to face the stock.

The "Spindle RPM" box should be filled with "6000". This value tells the tool how fast to spin while cutting through the material. The spin rates of each tool can be explained in-depth outside of this tutorial. Presently, all you need to know is the rotation per minute (RPM) values for each tool which will be provided.

The "Feedrate" box should be filled with "40.0". This value tells the tool how fast to move while cutting through the material. The unit for this value is inches per minute (IPM). As wood is a softer material, we can move more quickly while cutting. A harder material such as steel would need a slower feed rate.

The "Z Feedrate" box should be filled with "10.0". This value (IPM) tells the tool how fast to move when moving along the Z-axis. 10 IPM is the standard value for this as this tool can cut faster laterally rather than downwards.

The “Z Clear” box should be filled with “0.2000”. This value tells the tool how high above the stock to move between operations. We want to give the tool a little clearance for peace of mind. Verify that this part of the screen looks exactly like the image below. Note that these values are modeled after the speeds and feeds for cutting aluminum. This is because aluminum is the most common material being cut in the mill and this tutorial can serve as a reference guide moving forward.

TITLE:	Tutorial
WORK OFFSET:	G54
TOOL :	3
SPINDLE RPM:	6000
FEEDRATE:	40.0
Z FEEDRATE:	10.0
Z CLEAR :	0.2000

The “X START” box refers to the coordinate value of our workpiece’s left face. For this text box, input a negative sign followed by $(1/2 * \text{the length of your workpiece} + 0.1)$. For instance, if the length of your workpiece is 6”, you would enter “-3.1000”. The 0.1 is for a little extra margin. Record this value on your notecard.

The “X END” box should be filled with the absolute value of the “X START” box. (3.100” in the case of the example).

The “Y START” box should be filled with $(1/2 * \text{the width of your workpiece} + 0.1)$. For instance, if the width of your workpiece is 3.5”, you would enter 1.8500”.

The “STEPOVER” box should be filled with “0.7”. This value refers to how much of the tool overlaps the previous cut. A value of 1.0 would mean there is no overlap over the previous cut. Since wood is a soft material, we can be more aggressive in our stepover. The value for steel would be much less.

The “Y END” box should be filled with the negative of your Y START box. (-1.8500” in the case if the example).

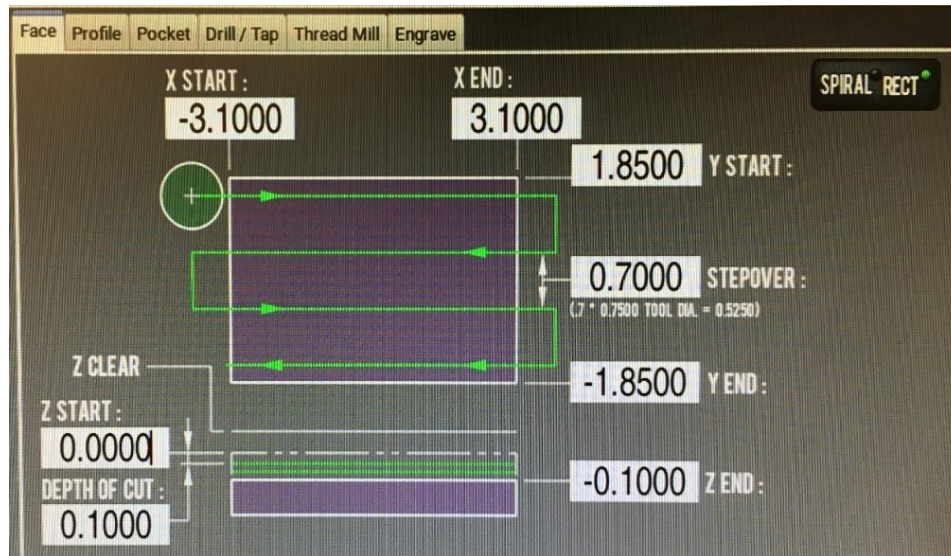
The “Z START” box should be filled with “0.0000”. We want to start cutting at the plane of the top face.

The “DEPTH OF CUT” box should be filled with “0.1000”. This value refers to how deep the tool cuts with each pass. This value will vary based on the material and on the tool. The shear hog is very powerful and can handle milling with a larger depth of cut. Again, the value for steel would be significantly lower.

“The Z END” box should be filled with “-0.1000”. This is because we only want to remove 0.1” of material from the top of our stock with one pass.

In the top right corner, toggle the “SPIRAL/RECT” button so that the green circle to the top right of RECT is illuminated. This is a more efficient pattern for facing operations.

Double-check through your values to make sure that the negative signs are correctly placed. This is a common mistake.

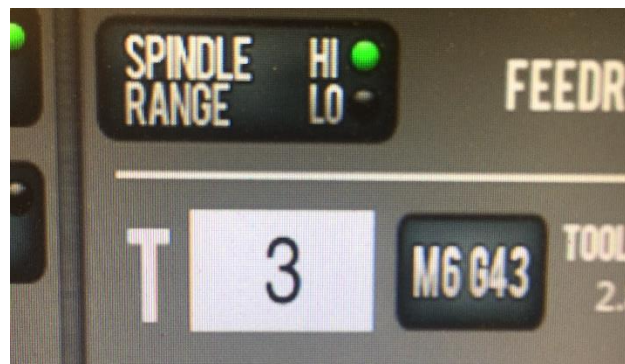


Click “POST TO FILE”. A pop-up window will appear in the center of your screen. Click the “HOME” button in the top left of the pop-up window. Double click the “CNC 101” folder. In the text box to the right of “NAME”, type 1.lastname (with lastname meaning your last name). Then click “SAVE”.

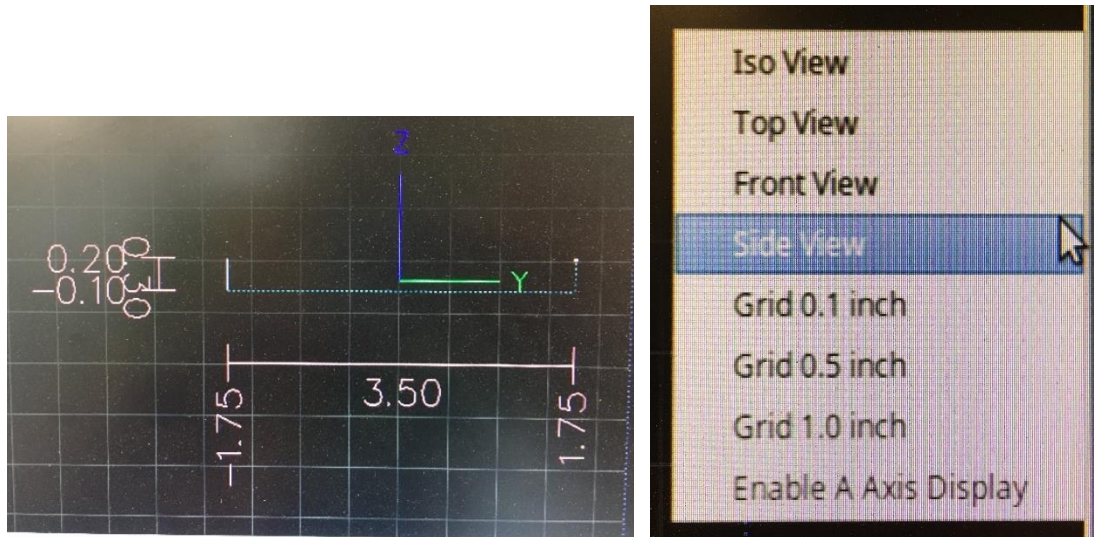
The operation will automatically load onto your screen. Congratulations, you just wrote the code for you first operation!

Performing your first operation

As we indicated in the previous section, this facing operation will be performed with Tool 3, the Shear Hog. Our first order of business is to load this tool into the spindle. Repeat the same operation described previously to remove Tool 1 and insert Tool 3 into the spindle. Return Tool 1 to its designated place in the tool rack. Highlight the text box to the right of “T”, type “3”, and HIT ENTER.

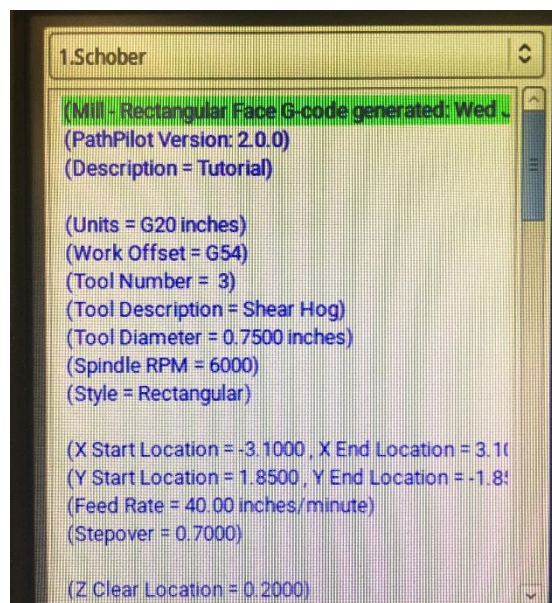


Now turn your attention to the computer monitor. In the top-right area, there is a gridded display showing the toolpath of the operation. This toolpath is labelled with coordinates that indicate the extreme values reached by the tool. The grid view can be changed from “top” to “side” or “iso” by right clicking with the mouse and selecting the preferred view. Change the view to “side” view and double-check the coordinates on the left of the display. These indicate the extreme Z positions of the toolpath. The top coordinate, 0.20, is the value that we inputted for our “Z CLEAR”. This determines the height at which the tool travels between passes. The bottom coordinate indicates the depth of the tool’s final pass. In this instance, this value is -0.10. It is recommended to check these values before starting any operation to ensure that the tool will behave as you expect it to.



Return the view to “top” view.

The top-left area of the screen displays the G-code for the operation. This code is the command script for the mill. Notice how this displays the inputted information from the conversational tab. The G-code holds additional information that will be described by the training Ninja in more detail throughout this tutorial.



During any operation on the mill, we will use the brass-and-blue attachment known as the fogbuster. This device will spray a user-defined ratio of compressed air and oil mixture onto the tool while milling. The oil mixture, known as coolant, is important to keep the tool lubricated and reduce heat build up associated with milling because of friction.

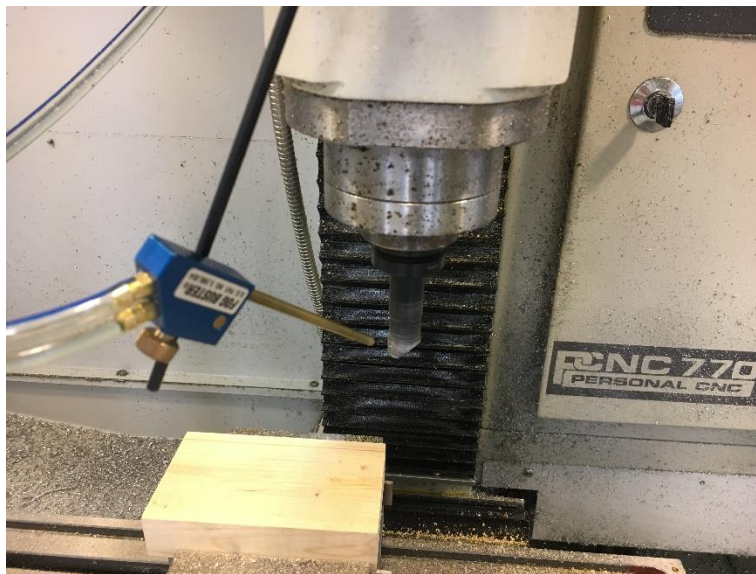
Furthermore, the compressed air helps to clear the leftover chips and debris from the milling surface. Both functions are vitally important while cutting metal because heat and chip build up affects the life of the tool. In the case of wood, no coolant is needed because the heat-build-up from friction is negligible. Compressed air is still used to clear the milling area of chips.



The Fog Buster is toggled on and off by pressing the “COOLANT” button in the computer display. Press the “COOLANT” button. The Fog Buster should emit a steady hissing sound. Test the compressed air output by placing your finger in front of the Fog Buster’s cylinder.

The amount of coolant in the compressed air is controlled by the brass screw on the Fog Buster. Clockwise rotation increases the amount of coolant and counter-clockwise rotation reduces the amount of coolant. Take the index card with your dimensions and hold it 2” away from the coolant nozzle. Rotate the brass screw clockwise until instructed to stop by your training Ninja (usually about one full rotation). This is the amount of coolant that is used while cutting metal. Rotate the brass screw counter-clockwise until movement is prevented.

Position the coolant so that it is aimed towards the tip of Tool 3. The Fog Buster is mounted with a magnetic base and rotatable arms. Aim the coolant as shown in the picture below. Test the coolant stream with your finger to make sure it is incident on the tool tip.



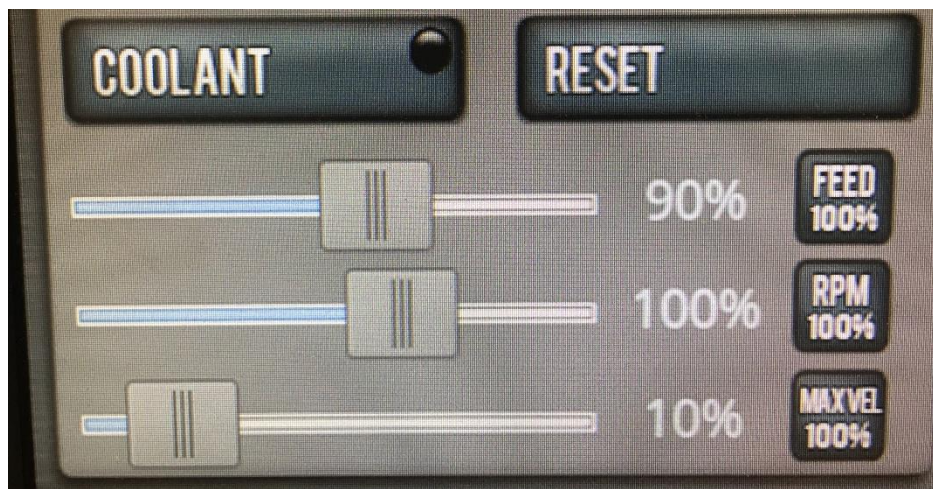
Click the “COOLANT” button to shut off the coolant. An issue with our coolant system can cause the coolant to remain blowing even after it is shut off. To rectify this, walk to the back-left of the mill, locate the black rectangle sitting on top of the coolant chamber, and flick it with your finger. The hissing should now stop.



On the bottom left of the computer display are three sliding bars. The top bar controls the Feed Rate of the mill. While writing the code for this operation, we designated this rate to be 40 IPM. At the beginning of an operation, it is common to reduce the feed rate to 90% of the specified feed rate, then to increase it to 100% if the mill appears to be cutting cleanly. Move the slider of the Feed rate to 90%.

Underneath the Feed Rate slider is the RPM slider. To begin, we will leave this slider at 100%.

Underneath the RPM slider is the MAX VEL slider. This controls the speed at which the mill travels when it is not cutting. The 100% speed is very fast, and for the first several operations in this tutorial, we will limit this slider to 10%. Move the MAX VEL slider to 10%. Any percentage below 10% will also impact the feed rate of the mill.



Once the operation begins, there are three ways to stop the mill. First, by sliding the MAX VEL slider to 0% or pressing the “FEEDHOLD” button, the mill will cease to move along the axes. However, the tool will continue to rotate. To stop the operation entirely, one can press the button “STOP” or hit the “ESC” key on the keyboard. An operation stopped this way can be easily restarted. To shut off power to the machine entirely, the red E-stop button can be pushed. This is the most effective option and should be performed immediately at the first sign of danger. However, it requires more effort to return the machine to its ready position. Therefore, we recommend to ALWAYS hit the E-stop if the mill is cutting and exhibits a dangerous low-pitched sound or unsteady position. During the first several operations of this tutorial, the training Ninja will be in control of when to hit the E-stop.

You are ready to begin the operation! Shut the enclosure doors fully. An operation cannot be run while the enclosure doors are ajar. When the operation begins, the tool will travel to its positive Z limit switch, initiate the coolant, begin spinning the tool, and move towards the workpiece. Keep your hand on the mouse and the mouse on the MAX VEL slider. Once the tool is moving towards the workpiece and is about an inch above the top face, you will slide the MAX VEL slider to zero and double-check the height. This is an important step in every operation you will perform.

Click the “CYCLE START” button to begin the operation. The training Ninja will explain the operation in depth as it occurs.



Once the operation has completed, the tool will travel to its positive Z limit switch. Once the movement stops, open the enclosure doors and inspect the workpiece. If the coolant doesn't shut off, flick the necessary box behind the CNC.

Setting up a profile cut

Now we will cut a profile out of our workpiece using the same tool. We are going to cut a .75" deep, 5" x 2.5" profile with .25" radius rounded corners into our stock.



Click on the “Conversational” tab in the middle of the screen. Then, click on the “Profile” tab on the top of the screen.

All the information in the left side of the screen will be preserved from the last operation. Therefore, none if it should need to be changed. Verify that it matches the image below.

TITLE:	Tutorial
WORK OFFSET:	G54
TOOL :	3
SPINDLE RPM:	6000
FEEDRATE:	40.0
Z FEEDRATE:	10.0
Z CLEAR :	0.2000

In the center of your screen, the following values need to be inputted.

Toggle the “RECT/CIRC” button in the top right of the screen so that the green circle is illuminated to the right of RECT.

Input “-2.5” into the “X PROFILE START” text box. As we want a symmetric 5” long profile, we want to start the profile 2.5” to the left of the center.

Input “2.5” into the “X PROFILE END” text box.

Input the same value into the “X START” text box as you did for the facing operation. This value should be negative with a magnitude = $(\text{length of your stock})/2 + .1$ ” factor of safety.

Input the same value into the “X END” text box as you did for the facing operation. This value should be positive with a magnitude = $(\text{length of your stock})/2 + .1$ ” factor of safety.

Input “0.25” into the “RADIUS” text box. This designates the radius of curvature for the profile edges.

Input “1.25” into the “Y PROFILE START” text box. As we want a symmetric 2.5” wide profile, we want to start the profile 1.25” to the back of the center.

Input “-1.25” into the “Y PROFILE END” text box. As we want a symmetric 2.5” wide profile, we want to start the profile 1.25” to the front of the center.

Input the same value into the “Y START” text box as you did for the facing operation. This value should be positive with a magnitude = $(\text{width of your stock})/2 + .1$ ” factor of safety.

Input the same value into the “Y END” text box as you did for the facing operation. This value should be negative with a magnitude = $(\text{width of your stock})/2 + .1$ ” factor of safety.

Input “0.7” into the “STEPOVER” text box.

Input “0.25” into the “Z DOC” text box. This is the depth of cut of the operation. For aluminum, we may not be this aggressive but for wood, the tool is comfortable with this depth of cut. This value will allow us to perform the operation in three passes.

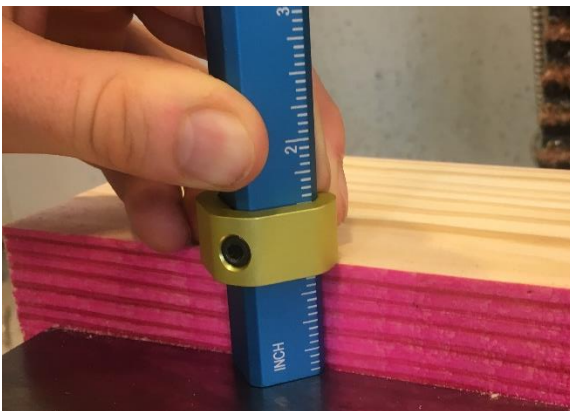
Input “-0.1” into the “Z START” text box. After our facing operation, this is the height of our stock face.

Input “-0.85” into the Z END” text box. We want to cut a 0.75” deep profile into our stock. This is an important value to be cognizant of. As you can see, the top face of the workpiece sticks out a certain distance above the vise. To avoid crashing into the vise, it is always good to check that the lowest Z coordinate the tool tip will reach is higher than the top face of the vise. To do this, we will use a tool known as the Stop-Loc. It is a quick and easy way to check the height between the workpiece and the stock.

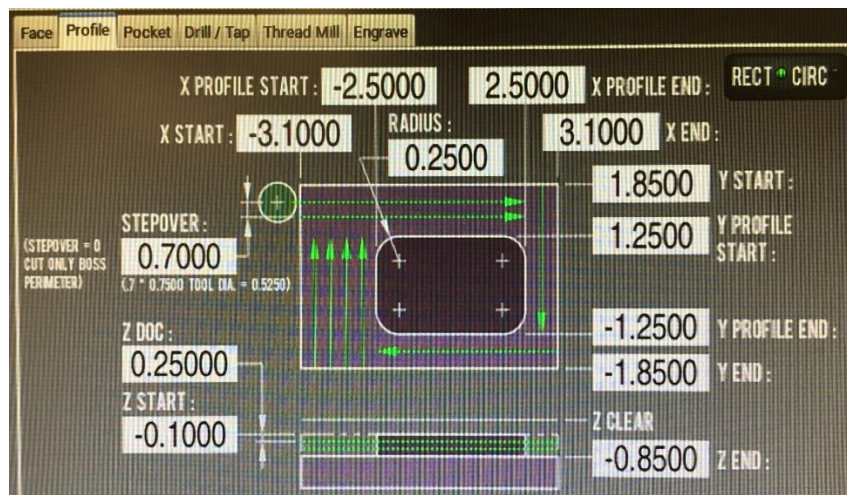
The yellow slider on the Stop-Loc can be loosened by turning the hex-cap screw in the yellow slider. Loosen the yellow slider so that it can be adjusted.



Position the Stop-Loc like the image shown below. Tighten the hex screw. In the case of the image below, it is shown that we have one inch of clearance between the top of the stock and the top of the vise. This is sufficient and shows that we will not crash into the vise.



Your screen should resemble the image below. Check your negative signs.

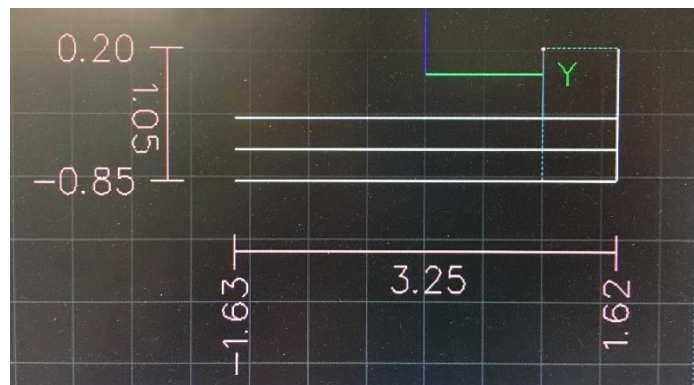


If all looks well, click “POST TO FILE”. Type “2.Lastname” into the “NAME” text box. Click “SAVE”.

Performing a profile cut

Right click the gridded display and select the “Side View” option. Verify that the left-hand values are the same as the image below. The right-hand values will change depending on the initial length of your stock and may differ from the image shown. This is ok.

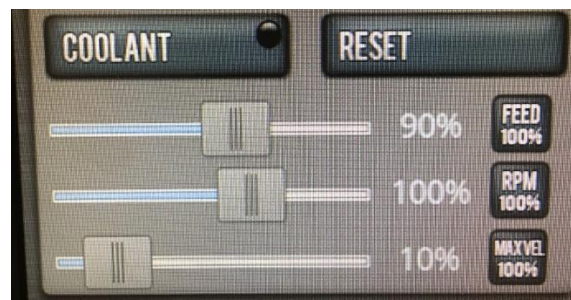
Display an “Side View” and notice how the operation contains three passes.



Scroll down the G-code on the computer monitor. Notice how it identifies the three different passes in the script.

Close the enclosure doors.

Double-check your slider values to make sure that they match the image below.



Confirm with the training Ninja. Hit “CYCLE START” and monitor the operation.

Once the operation has completed, inspect the workpiece and address the coolant if necessary.

Setting up a pocket cut

Now we will cut a .75” diameter, .5” deep pocket into the center of our workpiece.



Click on the “conversational” tab in the middle of the screen. Then, click on the “pocket” tab on the top of the screen.

For this operation we will use Tool 16, a ¼” flat end mill. Because of this change, we will need to update the information on the left side of the screen.

Input “16” into the “Tool” text box.

Input “6500” into the “SPINDLE RPM” text box. This is the RPM used for a ¼” end mill while cutting at this feed rate.

Verify that your screen looks like the following image.

TITLE :	Tutorial
WORK OFFSET :	G54
TOOL :	16
SPINDLE RPM :	6500
FEEDRATE :	40.0
Z FEEDRATE :	10.0
Z CLEAR :	0.2000

Toggle the “RECT/CIRC” button in the top right of the screen so that the green circle is illuminated to the right of CIRC.

Input “0.75” into the “POCKET DIA.” text box.

Input “0.5” into the “STEPOVER” text box. As this is a slimmer tool, we will perform a less-aggressive stepover.

Input “0.1” into the “Z DOC” text box. Same logic as the stepover.

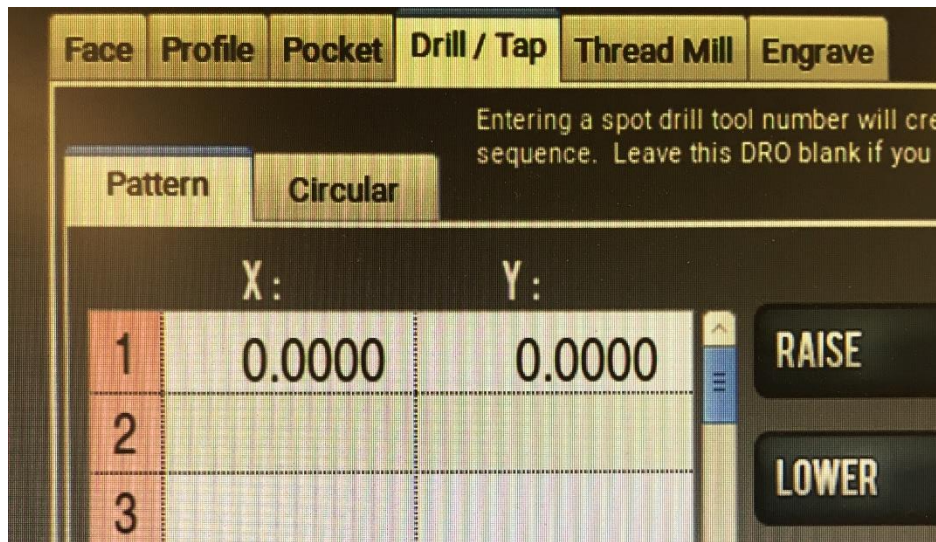
Input “-0.1” into the “Z START” text box. After our facing operation, the height of the top workpiece face is -0.1”.

Input “-0.6” into the “Z END” text box. We want a .5” deep pocket.

Verify that your screen looks like the image below.

		RECT CIRC <input checked="" type="radio"/>	
X (FROM DRILL TABLE)			
(STEPOVER = 0 CUT ONLY POCKET PERIMETER)		0.7500 POCKET DIA. :	
STEPOVER :		Y (FROM DRILL TABLE)	
0.5000			
(0.7 * 0.1875 TOOL DIA. = 0.1312)			
Z DOC :		Z CLEAR	
0.1000		-0.6000 Z END :	
Z START :			
-0.1000			

For circular pocket operations, we need to define the center point of our pocket in the “Drill/Tap” tab . Click on the “Drill/Tap” tab at the top of the screen. Click on the “Pattern” tab. In the first row of the chart, enter “0” into the X column and “0” into the Y column. This informs the machine that the center of the pocket will be at (0,0).



Return to the “Pocket” tab. Click “POST TO FILE”. Type “3.Lastname” into the “NAME” text box. Click “SAVE”.

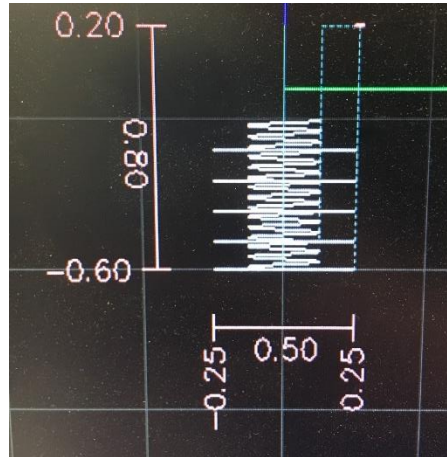
Performing a pocket cut

Remove Tool 3 from the spindle and use a rag to clean it free of sawdust. Return it to its position in the tool rack. Insert Tool 16. On the computer screen, highlight the “T” text box, type “16”, and HIT ENTER.

Re-aim the Fog Buster so that it is incident on the tool tip.



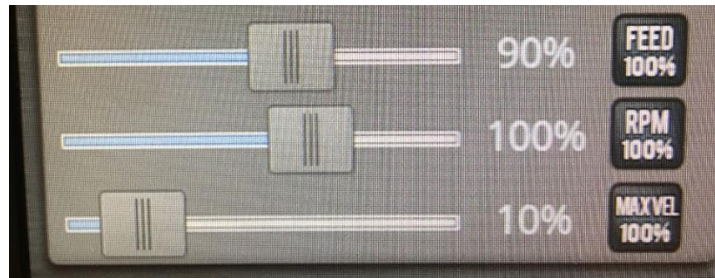
Select the “Side View” option and Verify that all coordinate values are the same as the image below. Also note how the operation contains five passes.



Change to an iso view.

Close the enclosure doors.

Double-check your slider values to make sure that they match the image below.



Confirm with the training Ninja. Hit “CYCLE START” and monitor the operation.

Once the operation has completed, inspect the workpiece and address the coolant if necessary.

Setting up a counter bore

In this tutorial, we will be drilling a set of four holes in a rectangular pattern about our workpiece. Above one of these holes, we will cut a counterbore. A counterbore is a flat-surfaced enlargement at the mouth of a drill hole that can fit the head of a screw. In this operation, we will mill a 0.5” diameter, 0.25” deep counterbore at the (2”,0.75”) position.



Click on the “conversational” tab in the middle of the screen. Then, click on the “pocket” tab on the top of the screen.

For this operation we will use Tool 15, a 3/16” flat end mill. Because of this change, we will need to update the information on the left side of the screen.

Input “15” into the “Tool” text box.

Input “7000” into the “SPINDLE RPM” text box.

Verify that your screen looks like the following image.

TITLE:	Tutorial
WORK OFFSET:	G54
TOOL:	15
SPINDLE RPM:	7000
FEEDRATE:	40.0
Z FEEDRATE:	10.0
Z CLEAR:	0.2000

Toggle the “RECT/CIRC” button in the top right of the screen so that the green circle is illuminated to the right of CIRC.

Input “0.5” into the “POCKET DIA.” text box.

Input “0.5” into the “STEPOVER” text box.

Input “0.1” into the “Z DOC” text box.

Input “-0.1” into the “Z START” text box. After our facing operation, the height of the top workpiece face is -0.1”.

Input “-0.35” into the “Z END” text box. We want a .25” deep pocket.

Verify that your screen looks like the image below.

Face	Profile	Pocket	Drill / Tap	Thread Mill	Engrave
------	---------	--------	-------------	-------------	---------

RECT CIRC

X (FROM DRILL TABLE)

Y (FROM DRILL TABLE)

(STEPOVER = 0 CUT ONLY
POCKET PERIMETER)

STEPOVER:
0.5000

(.25" ± 0.1875 TOOL DIA. =
0.0500)

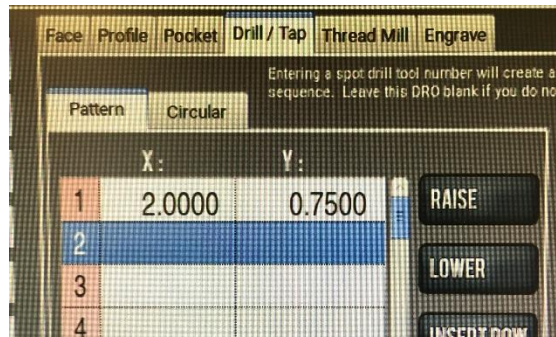
Z DOC:
0.1000

Z START:
-0.1000

0.5000 POCKET DIA.:

Z CLEAR
-0.3500 Z END:

For circular pocket operations, we need to define the center point of our pocket in the “Drill/Tap” tab. Click on the “Drill/Tap” tab at the top of the screen. Click on the “Pattern” tab. In the first row of the chart, enter “2.0” into the X column and “0.75” into the Y column. This informs the machine that the center of the pocket will be at (2,0.75).



Return to the “Pocket” tab. Click “POST TO FILE”. Type “4.Lastname” into the “NAME” text box. Click “SAVE”.

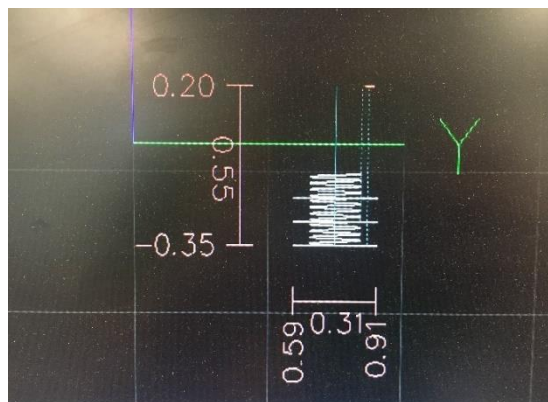
Performing a counter bore cut

Remove Tool 16 from the spindle and use a rag to clean it free of sawdust. Return it to its position in the tool rack. Insert Tool 15. On the computer screen, highlight the “T” text box, type “15”, and HIT ENTER.

Re-aim the Fog Buster so that it is incident on the tool tip.



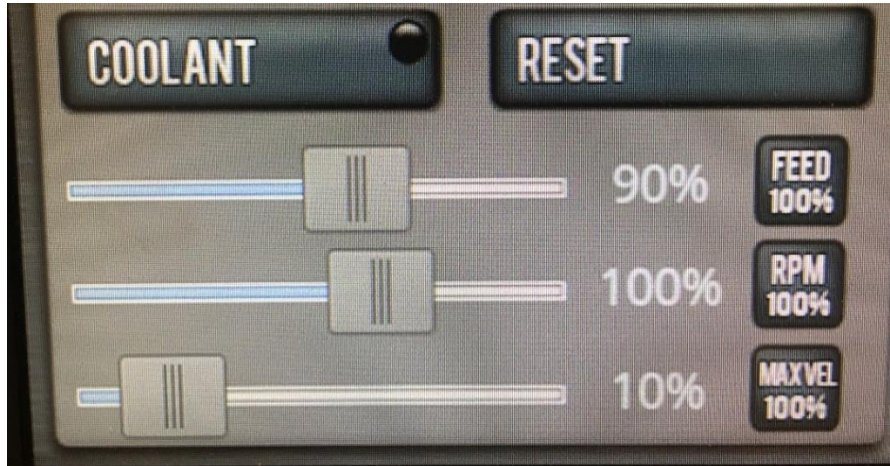
Select the “Side View” option and Verify that all coordinate values are the same as the image below. Also note how the operation contains three passes.



Change to an iso view.

Close the enclosure doors.

Double-check your slider values to make sure that they match the image below.



Confirm with the training Ninja. Hit "CYCLE START" and monitor the operation.

Once the operation has completed, inspect the workpiece and address the coolant if necessary.

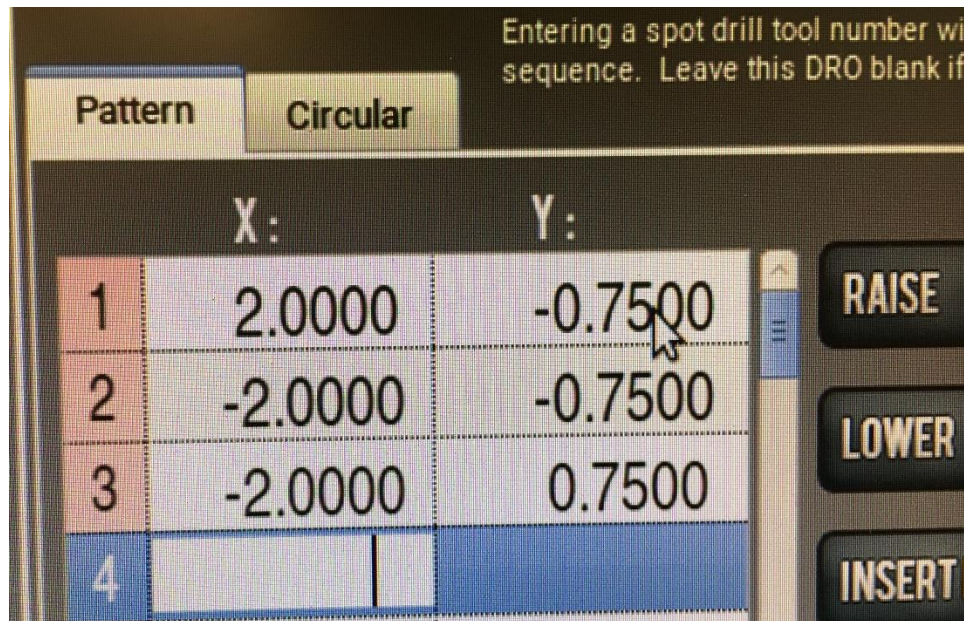
Setting up a spot drill operation

Before we drill into any material, it is recommended to preface the operation with a spot drill. This is a dimple cut into the material at the point of contact so that the drill bit is guided straight when it encounters the material. It is performed using Tool 44. In this operation, we will drill spot drill for the remaining three drill holes at (-2,0.75), (-2,-0.75), and (2,-0.75).



Click on the "conversational" tab in the middle of the screen. Then, click on the "Drill/Tap" tab on the top of the screen.

Click on the “Pattern” tab. In the first row of the chart, enter “2.0” into the X column and “-0.75” into the Y column. In the second row, enter “-2.0” into the X column and “-0.75” into the Y column. In the third row, enter “-2.0” into the X column and “0.75” into the Y column.



Input “44” into the “TOOL” text box.

Input “5000” into the “SPINDLE RPM” text box.

Input “5” into the “Z FEEDRATE” text box.

Make sure that the green circle next to “DRILL” is illuminated in the top right corner.

Leave the “SPOT TOOL #” and “SPOT TOOL DOC” empty. These spot drill text boxes are used only if another operation is being performed.

Input “0” into the “PECK” text box. Pecking is a method of chip clearing that we will not use in this tutorial.

Input “-0.1” into the “Z START” text box.

Input “-0.125” into the “Z END” text box. We want the depth of our spot drill to be 0.025”.



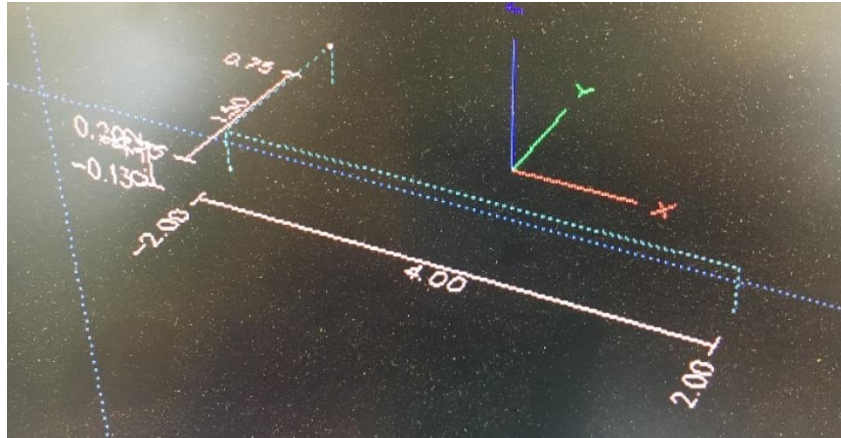
Click “POST TO FILE”. Type 5.Lastname” into the “NAME” text box. Click “SAVE”.

Performing a spot drill operation

Remove Tool 15 from the spindle and return it to its position in the tool rack. Insert Tool 44. On the computer screen, highlight the “T” text box, type “44”, and HIT ENTER.

Re-aim the Fog Buster so that it is incident on the tool tip.

Change to an iso view and verify the coordinate positions with the image below.



Close the enclosure doors.

Double-check your slider values to make sure that they match the image below.

Confirm with the training Ninja. Hit “CYCLE START” and monitor the operation.

Once the operation has completed, inspect the workpiece and address the coolant if necessary.

Setting up a counter sink operation

Now we will drill a .25” counter sink at the (2,-0.75) position using a 3/8” drill bit (Tool 43). A counter sink is similar to a counter bore except it is performed with a drill bit and so it has an angled bottom instead of a flat bottom.



Click on the “conversational” tab in the middle of the screen. Then, click on the “Drill/Tap” tab on the top of the screen.

Click on the “Pattern” tab. In the first row of the chart, enter “2.0” into the X column and “-0.75” into the Y column. Delete the entries for the second and third row.

Input "43" into the "TOOL" text box.

Input "2000" into the "SPINDLE RPM" text box.

Input "15" into the "Z FEEDRATE" text box.

TITLE: Tutorial

WORK OFFSET: G54

TOOL: 43

SPINDLE RPM: 2000

FEEDRATE: 40.0

Z FEEDRATE: 15.0

Z CLEAR: 0.2000

(Z FEED PER REVOLUTION: 0.0075)

Face

1

2

3

4

5

6

7

Leave the "SPOT TOOL #" and "SPOT TOOL DOC" empty. These spot drill text boxes are used only if another operation is being performed.

Input "0" into the "PECK" text box. Pecking is a method of chip clearing that we will not use in this tutorial.

Input "-0.1" into the "Z START" text box.

Input "-0.35" into the "Z END" text box. We want the depth of our spot drill to be 0.25".

Face Profile Pocket Drill / Tap Thread Mill Engrave

Entering a spot drill tool number will create a spot drilling sequence before the drilling sequence. Leave this DRC blank if you do not want to spot drill.

DRILL • TAP

Pattern Circular

X: 2.0000 Y: -0.7500

1 2 3 4 5 6 7 8 9

RAISE LOWER INSERT ROW DELETE CLEAR ALL

SPOT TOOL #: SPOT TOOL DOC:

PECK: 0.0000

Z START: -0.1000

Z END: -0.3500

Z CLEAR

(TOOL FEED PER REVOLUTION: 0.0075) (HOLE DIA: 0.2500)

Click "POST TO FILE". Type 6.Lastname" into the "NAME" text box. Click "SAVE".

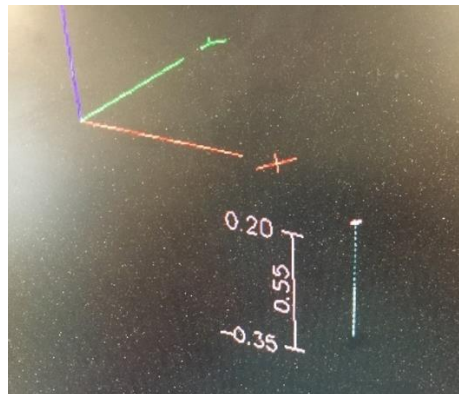
Performing a countersink operation

Remove Tool 44 from the spindle and return it to its position in the tool rack. Insert Tool 43. This time, do not update the tool number on the computer screen.

Re-aim the Fog Buster so that it is incident on the tool tip.



Change to an iso view and verify the coordinate positions with the image below.

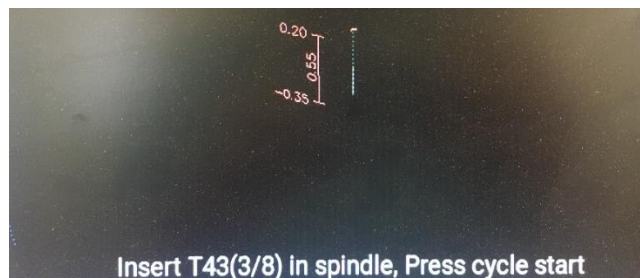


Close the enclosure doors.

Double-check your slider values.

Confirm with the training Ninja. Hit "CYCLE START".

Notice how a message appears telling you to insert Tool 43 into the spindle. This is a software safety feature that helps prevent a tool crash. Since we have already loaded in Tool 43, simply press "CYCLE START" and monitor the operation. Take extra care to verify the Z height of the tool before the operation begins. Drills are the most important to check for this as their height values are constantly being updated and re-arranged.



Once the operation has completed, inspect the workpiece and address the coolant if necessary.

Setting up a drilling operation

Now we will drill two .75" deep holes at the at the (2,0.75) and (-2,0.75) positions using the same 3/8" drill bit (Tool 43). Instead of drilling all the way through the part, we are going to drill halfway through the part, then flip the piece and drill through the other half.



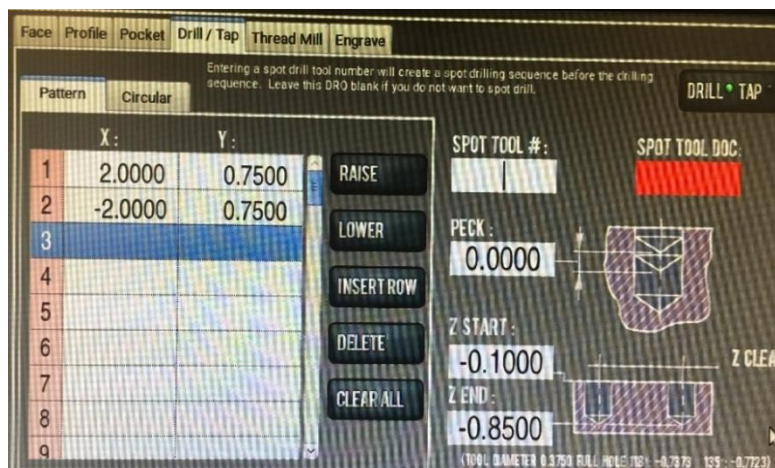
Click on the “conversational” tab in the middle of the screen. Then, click on the “Drill/Tap” tab on the top of the screen.

Click on the “Pattern” tab. In the first row of the chart, enter “2.0” into the X column and “0.75” into the Y column. In the second row of the chart, enter “-2.0” into the X column and “0.75” into the Y column.

As we are using the same tool as before, the left-hand side of the screen can be left unchanged.

Input “-0.1” into the “Z START” text box.

Input “-0.85” into the “Z END” text box. We want the depth of our spot drill to be 0.75”.

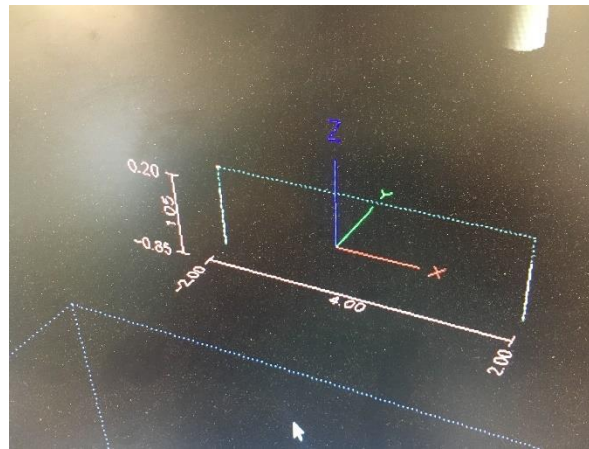
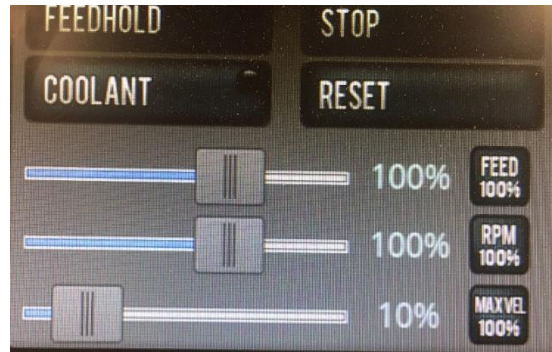


Click “POST TO FILE”. Type 7.Lastname” into the “NAME” text box. Click “SAVE”.

Performing a drilling operation

Close the enclosure doors.

Double-check your slider values. Update the feed rate to 100%.



Confirm with the training Ninja. Hit "CYCLE START" and monitor the operation.

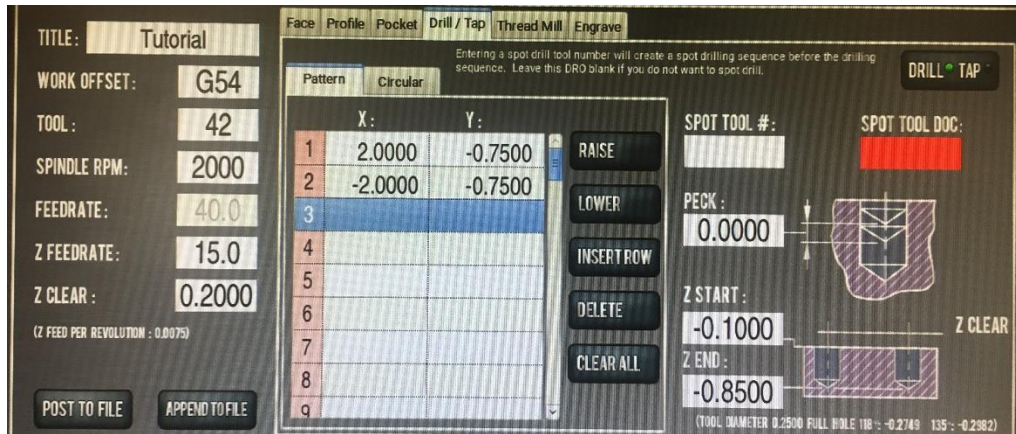
Once the operation has completed, inspect the workpiece and address the coolant if necessary.

Setting up a second drilling operation

Now we will drill two .75" deep holes at the at the (2,-0.75) and (-2,-0.75) positions using a 1/8" drill bit (Tool 42).



Follow the procedure to set up this operation in Conversational mode. The feedrate and RPM for this operation will be the same as the previous operation.



Click “POST TO FILE”. Type 8.Lastname” into the “NAME” text box. Click “SAVE”.

Performing a second drilling operation

Remove Tool 43 from the spindle and return it to its position in the tool rack.

For this CNC, the drill tools will constantly change their drill bits. Therefore, when we add a new drill bit, we need to update its information in the computer. To the left of the tool rack, there is a grey box filled with a set of drill bits. Remove the 1/8” drill bit from the box.

Tool 42, 43, and 44 use Jacob’s chucks to secure the drill bit. These can be loosened and tightened with a chuck key to swap drill bits. Using the smaller of the two chuck keys sitting next to the drill bit set, loosen Tool 42’s chuck and remove its bit.



Return the bit to its appropriate place in the drill bit set.

Insert the 1/8” drill bit into Tool 42 and tighten it. Take care that the drill bit is centered on the chuck. Ask the training Ninja advice on how to ensure this. Tighten the drill bit!

Now that the Tool 42 is ready, we need to update its information in the computer. The most important piece of information we need is its height offset. An incorrect value for could cause the tool to crash into the workpiece. Take Tool 42 over to the measuring block to the left of the blue box of parallels.



Position the calipers so that its measuring tips rests on the stone block. Hit the blue “ZERO” button. This ensures that the relative height will be correct.



Place the butt end of Tool 42 into the hole in the stone block. Raise the micrometer and position it so that it rests on the tip of Tool 42. Record the displayed number.



Return to the computer monitor. Click the “Offsets” tab in the middle of the screen.

The right-hand side of the screen displays the tools, their diameters, and their offsets. Scroll down until you see Tool 42.

41		0.0000	0.0000
42	7/32	0.2500	5.4050
43	3/8	0.3750	6.6345
44	Center Drill	0.2500	3.9735
45		0.0000	0.0000
46		0.0000	0.0000

Update the information in Tool 42's row. Input “1/8” into the Description, “1/8” into the Diameter, and your measured value into the Length

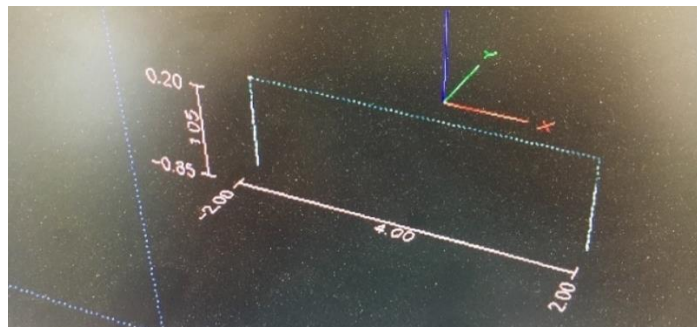
Tool	Description	Diameter	Length
41		0.0000	0.0000
42	1/8	0.1250	4.2215
43	3/8	0.3750	6.6345
44	Center Drill	0.2500	3.9735
45		0.0000	0.0000

Now, insert Tool 42 into the spindle and secure it. Update the tool number in the computer.

Re-aim the Fog Buster so that it is incident on the tool tip.

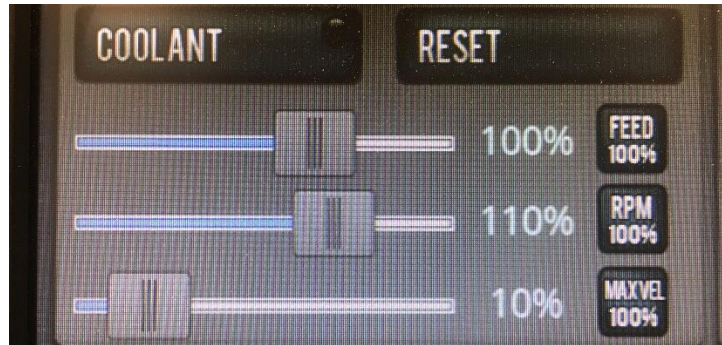


Change to an iso view and verify the coordinate positions with the image below.



Close the enclosure doors.

Update the Feedrate Slider to 100% and the RPM slider to 110%. As this is a smaller drill bit, we want it to be spinning a little faster.



Confirm with the training Ninja. Hit “CYCLE START” and monitor the operation.

Once the operation has completed, inspect the workpiece and address the coolant if necessary.

Setting up an engraving operation

Now we want to engrave “CNC Mill” above the center hole. This will be done with the engraving feature in Conversational mode.



Click on the “conversational” tab in the middle of the screen. Then, click on the “Engrave” tab on the top of the screen.

For engraving, we will use a 1/4” chamfer tool. This is Tool 22 in the library. Input “22” into the “TOOL” text box.

This tool cuts best at 2500 RPM. Input “2500” into the “RPM” text box.

For engraving, we tend to run at a slower feedrate. Input “20.0” into the “FEEDRATE” text box.

We will begin by engraving the top text, “CNC Mill” onto our workpiece. Input “CNC Mill” into the “TEXT” box. Scroll through the font list and select whichever font you like the most.

We want the height of our text to be 0.35”. Input “0.35” into the “HEIGHT” text box.

We want our text to be centered on the workpiece so enter “0” into the “X BASE” text box. Then, click the circle to the left of the “CENTER” text. This will center our text on the Y axis.

We want to align the center of our text to the center of our drill holes. As this Y-coordinate is 0.75”, the Y base should be $(0.75 - \text{Height}/2) = 0.575$. Input “0.575” into the “Y BASE” text box.

Input “-0.1” into the “Z START” text box.

Input “0.02” into the “Z DEPTH OF CUT” text box. This determines how deep the tool will cut. This value can differ based on preference but .02” is preferred for this tutorial.

“SN START” is used for engraving serial number and is not covered in this tutorial.



Click “POST TO FILE”. Type 9.Lastname” into the “NAME” text box. Click “SAVE”.

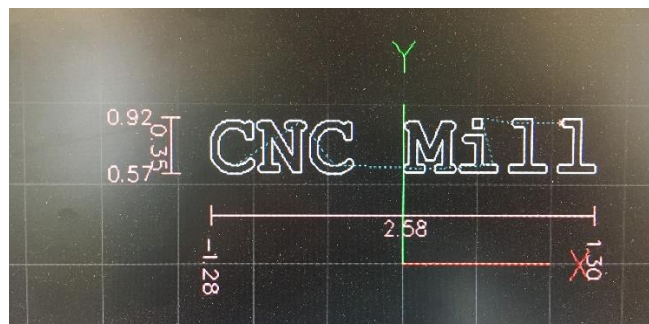
Performing an engraving operation

Remove Tool 42 from the spindle and return it to its position in the tool rack. Input Tool 22 and update the tool number in the computer.

Re-aim the Fog Buster so that it is incident on the tool tip.



Verify the coordinate positions with the image below.



Close the enclosure doors.

Confirm the slider values with the training Ninja. Hit “CYCLE START” and monitor the operation.

Once the operation has completed, inspect the workpiece and address the coolant if necessary.

Setting up a second engraving operation

Now we want to engrave your last name underneath the center hole in the workpiece.

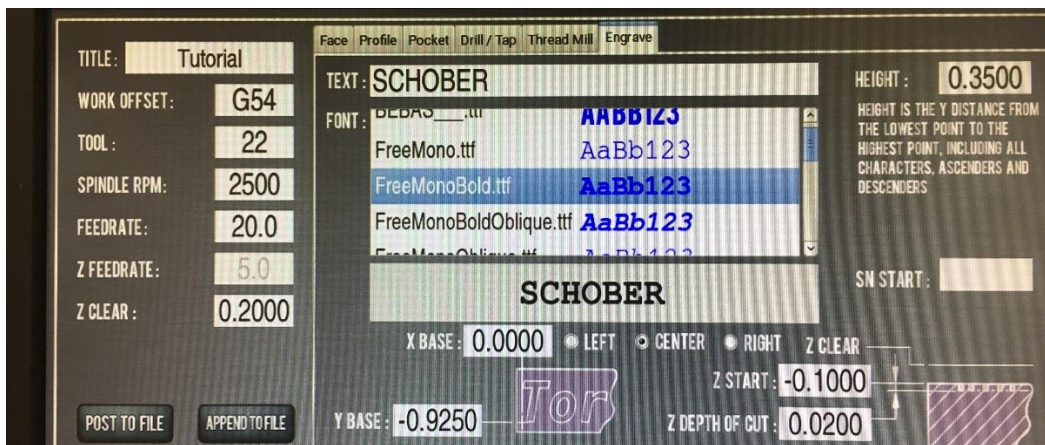


Click on the “conversational” tab in the middle of the screen. Then, click on the “Engrave” tab on the top of the screen.

Input “LAST NAME” into the “TEXT” box. Scroll through the font list and select “FreeMonoBold.ttf”.

We want to align the center of our text to the center of our drill holes. As this Y-coordinate is 0.75”, the Y base should be $(-0.75 - \text{Height}/2) = -0.925$. Input “-0.925” into the “Y BASE” text box.

Verify that your other text boxes match the image below.



Click “POST TO FILE”. Type 10.Lastname” into the “NAME” text box. Click “SAVE”.

Performing a second engraving operation

Close the enclosure doors.

Confirm with the training Ninja. Hit “CYCLE START” and monitor the operation.

Once the operation has completed, inspect the workpiece and address the coolant if necessary.

Flipping the workpiece

Now that all our operations have been completed on this face, we will flip the workpiece and mill the backside.

Rotate the vise handle counter clockwise to release the part.

Use a rag to clear the vise and parallels free of sawdust.

Flip the part, place the profile on top of the parallels and tighten the vise.



Now that we have moved the workpiece, we must re-reference it. Instead of using the “touching” method that we did originally, we will introduce a new tool called the Haimer probe. This is a pressure sensitive tool that can be used to measure extremely accurate work offsets. It is Tool 5 and is housed in a wooden box to the left of the tool rack labeled “TTS Zero Master”.



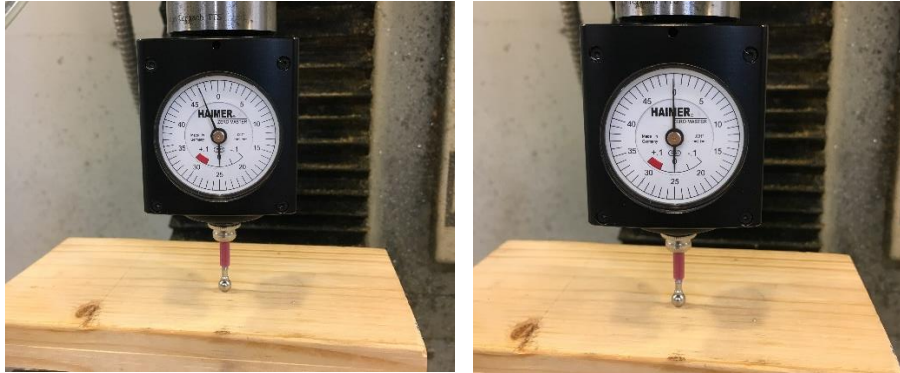
Carefully take the Haimer probe over to the mill. It is an **EXTREMELY** delicate and expensive tool that **WILL BREAK** if subjected to too much pressure.

Load the Haimer into the spindle. Update the tool number in the computer to Tool 5.



The Haimer has a pressure sensor display on its face. As its metal sphere at its tip is moved into the material and subjected to force, the hands will move in a clockwise direction. Once the large hand has completed exactly two travels around the sensor, the center of the spindle is positioned directly above the plane of the face. More than two rotations will subject too much pressure on the sensor and risk shattering the \$50 bit.

Turn the controls over to your training Ninja and watch as they demonstrate the correct procedure for measuring with the Haimer. They will slowly drive the probe in the negative Z direction towards the top face of the workpiece. Once the tip touches the material, they will slow the movement down to the lowest speed with the shuttle ring until the pressure reaches the image below. Then, they will use the jog wheel to complete the rotation. Once the pressure sensor displays the reading shown below, the tool is at the Z=0 position.



The training Ninja will drive the probe back to its positive limit switch and turn the controls back to you. Avoid any lurches and trust your abilities.

Once the pressure sensor reaches the desired position, highlight the “Z ZERO” text box, type “0” and HIT ENTER. Then, carefully back the Haimer away from the top face until it no longer touches the workpiece.

Now, it is time to reference the X and Y axes. The Haimer measures lateral pressure the same way it measures vertical pressure. First, we will drive the Haimer to the left-side workpiece face. Take care to move slowly and be sure that the Haimer has enough clearance for the movement. Now, lower the Haimer probe to the Z=-0.2 position. As long as the probe is low enough that the metal sphere is fully engaged on the face of the workpiece, but not too low that the red sleeve of the probe is in line with the top of the workpiece.



We will now repeat the same procedure to set the X-coordinate of the left-hand plane equal to “0” by moving the probe into the workpiece until the pressure sensor displays the image below.



	WORK	DTG
X	0.0000	0.0000
Y	0.4251	0.0000
Z	-0.1797	0.0000

After the left-hand plane has been set to zero, cautiously back the probe away from the workpiece. Then, raise the probe sufficiently above the top plane of the workpiece, and travel to the right-hand workpiece face. Carefully move the probe into the workpiece until the pressure displays the desired value.

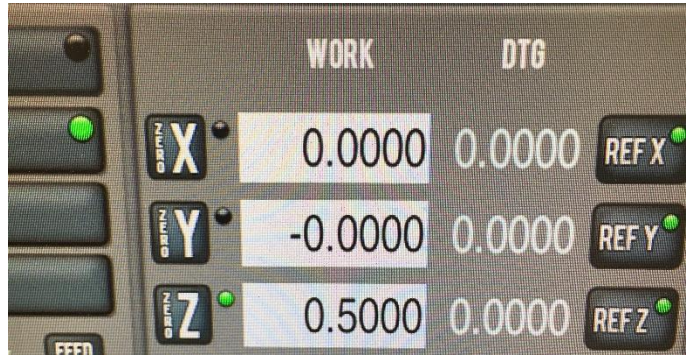


Observe the X-coordinate value. This is the overall length of the workpiece. To set the center of the workpiece as the origin, we will use a simple trick. By dividing the current value by two, we are telling the machine that it is half the length of the workpiece away from the origin. Thus, the origin is in the center of the workpiece. Highlight the “X ZERO” text box, Type “/2” and HIT ENTER. The coordinate will update to be $\frac{1}{2}$ the length of the workpiece.

X	6.0966	X	/2	X	3.0483
----------	--------	----------	----	----------	--------

Now, perform the same procedure to set the Y-axis origin to be in the center of the workpiece. Ask the training Ninja if you have any questions.

Once the piece has been fully referenced, carefully bring the Haimer Probe above the workpiece. Move it to $X=0$, $Y=0$, $Z=0.5"$. This should be in the center of your piece. Consult the training Ninja if it is not.



Return the tool to its positive limit switch.

Manually facing with the mill

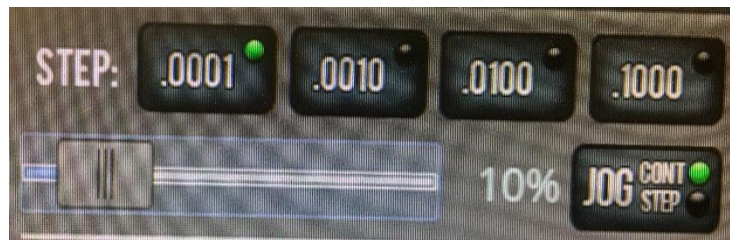
Now, we will perform a manual facing operation with the mill. Instead of writing an operation to move the machine, we will drive it with the keyboard. We will use the Superfly, Tool 2, for this operation.

CAREFULLY remove the Haimer Probe from the spindle. Make sure that it is secure in your hand while loosening the spindle. Return it to its designated box.

Take Tool 2 from the tool rack and insert it into the mill. Update the tool number in the computer.

The Superfly is used to cut small layers off the top face of a material and leaves a very smooth surface finish. Almost exclusively, it is operated manually.

The speed of manual milling is determined by the slider in the middle-right of the screen underneath the "STEP" button. Its default percentage is 40% which corresponds to a feed rate of 54.4 IPM. This is unnecessarily high and can lead to crashes. Move the slider to 10%. This corresponds to a feed rate of 13.7 IPM, which is what we will use when milling with the Superfly.



Manual milling is operated with the arrow keys on the keyboard. Take some time to move the workbench in the X and Y axes by holding the arrow keys. Familiarize yourself. Notice how the speed is perfectly consistent, unlike the jog shuttle which can vary speed. This allows for a smooth milling operation. A word of caution, the "END" key can perform functions depending on what is selected on the screen. For instance, if the slider is selected and "END" is pressed, the slider moves to 100% which is a 126 IPM feedrate! Take care to avoid pressing keys other than the arrow keys during an operation.

Once you are comfortable driving with the arrow keys locate the RPM text box underneath the slider. The Superfly cuts best when rotating at 1500 RPM. Input “1500” into the RPM text box and HIT ENTER.



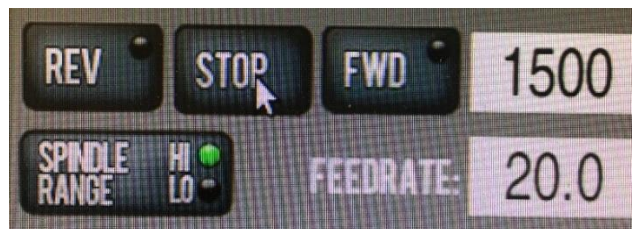
Using the jog shuttle, drive the Superfly to the right of the workpiece until its full rotational diameter can be lowered down without striking any material. Then, lower the superfly to the Z=-0.020 coordinate. Rotate the superfly fully to make sure that it will not strike the workpiece.

Position the Fog Buster so that it is incident on the tool tip when the Superfly is in the position where it extends farthest to the left.

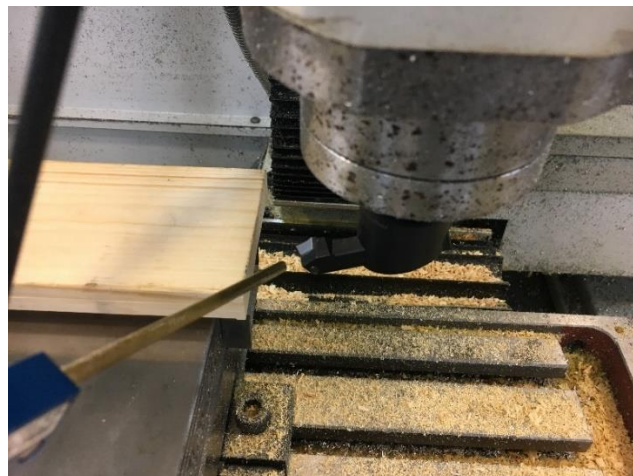
To rotate the spindle above 1000 RPM, the enclosure doors need to be closed. Close the enclosure doors.

Double-check that the RPM value is 1500. If you had neglected to change this and the RPM value was over 5000, serious damage could occur.

Now, we will turn on the spindle, observe, and turn it back off. Press the “FWD” button to the left of the “RPM” text box to start the spindle. After a brief moment, the Superfly will begin to rotate at 1500 RPM. Stop the rotation by pressing the “STOP” button to the left of the “FWD” button.



Once you have done this, it is time to face the part. We will need to perform two passes to achieve this as the Superfly isn't wide enough to cut the whole workpiece in one pass. Position the tool so that it looks like the following image.



Hit the “COOLANT” button to turn on the coolant.

Press “FWD” and hold the left arrow key to drive the Superfly across the workpiece. Once you have cleared the left side of the workpiece, press “STOP”.

Now, move the superfly towards the back of the machine so that it will mill the parts of the workpiece that it missed on the first pass. Reposition your coolant so that it is incident on the tool. Press “FWD”, and hold the right arrow key to drive the Superfly back across the material. Press “STOP” which it is fully clear of the workpiece.

Depending on the flatness of your stock, the Superfly may have left some areas on the workpiece top un-milled. This would occur if the stock was sloped or contained divots. The tutorial will continue but if you wish to perform another pass to rectify this, consult the training Ninja.

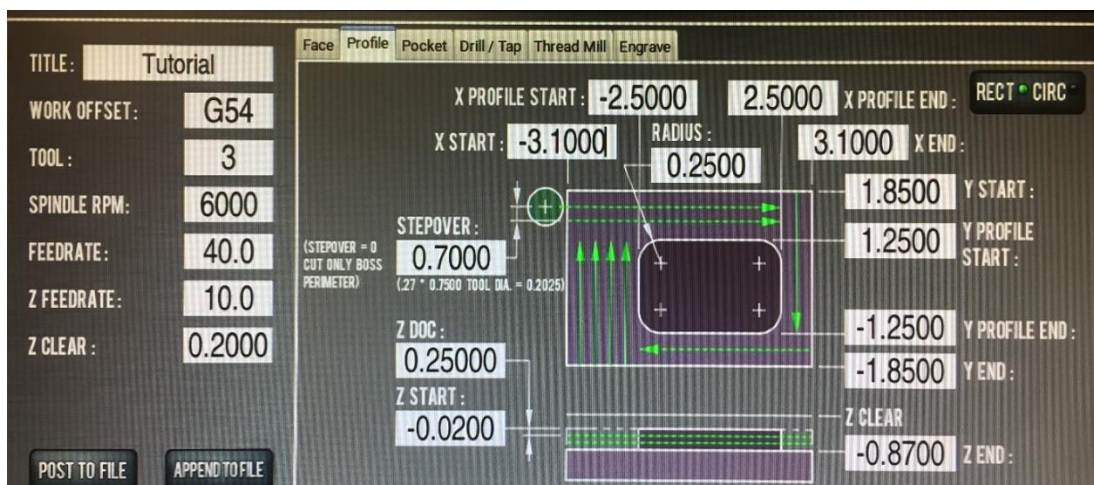
Raise the tool to its positive Z limit switch.

Setting up a backside profile operation

Now we will mill the same rectangular profile in the backside of our workpiece. It will be a .85” deep, 5” x 2.5” profile with .25” radius rounded corners into our stock with the Shear Hog. Use the Stop-Loc to make sure you have enough clearance over the top of the vise.



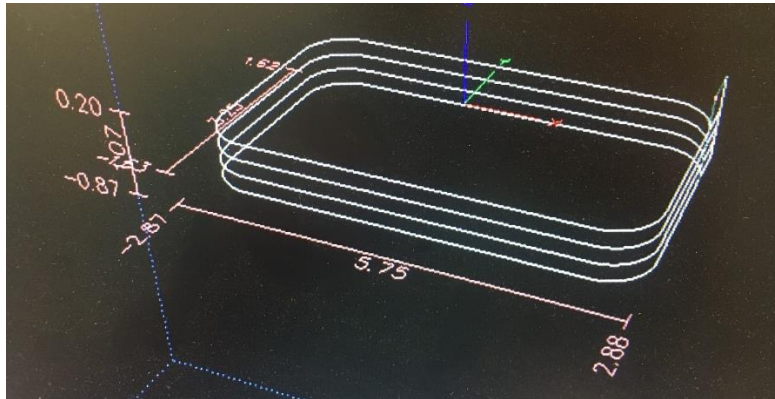
Do you best to set up the operation on your own. Refer to the image below if you need a refresher. Key things to remember are what to change for the “Z START”, “Z END”, “TOOL”, “RPM”, and “FEEDRATE”. Check your work below if necessary



Save the file as 11.Lastname.

Performing a backside profile operation

Do what needs to be done to perform this operation.

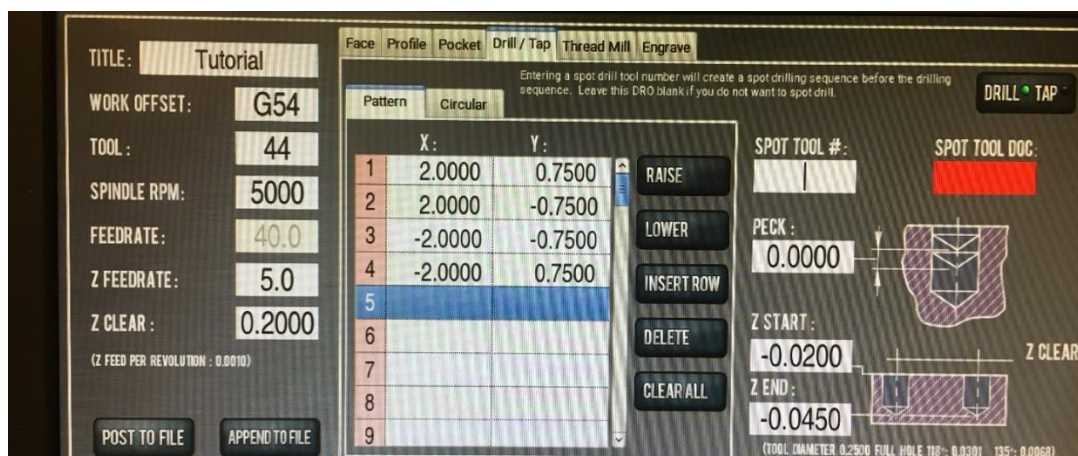


Setting up a backside spot drilling operation

Set up four spot drills at positions (2,0.75), (2,-0.75), (-2,0.75),(-2,-0.75) using Tool 44.



Double-check your values with the image below.



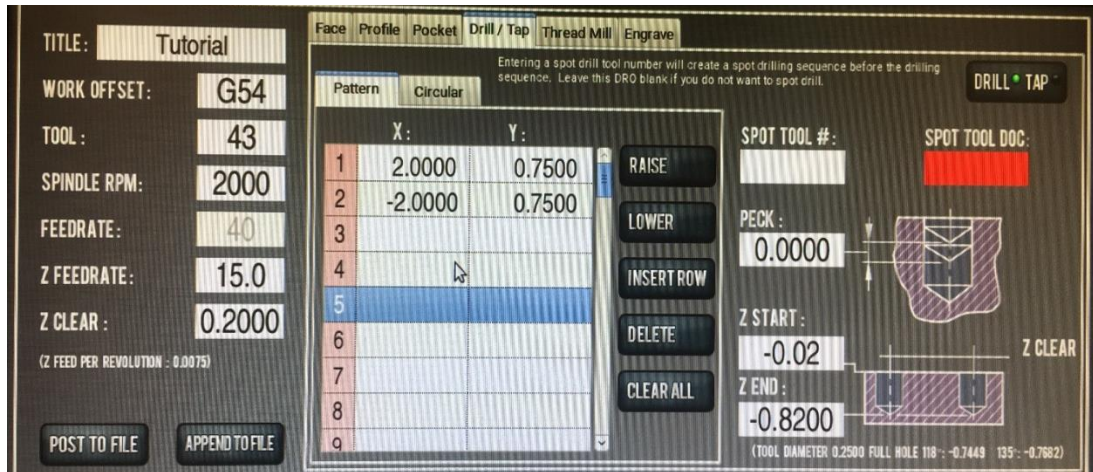
Perform a backside spot drilling operation

Do what is necessary to complete this operation. Consult the training Ninja if necessary.

Setting up a backside drilling operation

Set up two 0.8" deep drills at positions (2,0.75) and (-2,0.75) using the 3/8" drill (Tool 43).

Review the information below to double-check your work.



Perform a backside drilling operation



Do what is necessary to perform this operation.

Setting up a second backside drilling operation

Set up two 0.8" deep drills at positions (2,-0.75) and (-2,-0.75) using the 1/8" drill (Tool 42).



Review the information below to double-check your work.

TITLE: Tutorial

WORK OFFSET: G54

TOOL: 42

SPINDLE RPM: 2000

FEEDRATE: 40

Z FEEDRATE: 15.0

Z CLEAR: 0.2000

(Z FEED PER REVOLUTION : 0.0075)

POST TO FILE **APPEND TO FILE**

Face **Profile** **Pocket** **Drill / Tap** **Thread Mill** **Engrave**

Entering a spot drill tool number will create a spot drilling sequence before the drilling sequence. Leave this DRO blank if you do not want to spot drill.

Pattern **Circular**

	X:	Y:
1	2.0000	-0.7500
2	-2.0000	-0.7500
3		
4		
5		
6		
7		
8		

RAISE **LOWER** **INSERT ROW** **DELETE** **CLEAR ALL**

SPOT TOOL #:

SPOT TOOL DOC:

PECK: 0.0000

Z START: -0.02

Z END: -0.8200

(TOOL DIAMETER 0.1000 FULL HOLE 118°: -0.7900 135°: -0.7993)

DRILL **TAP**

Z CLEAR

Perform a backside drilling operation

Do what is necessary to perform this operation.

Finishing Up

Congratulations! You have completed the CNC mill training. To power down the machine, first hit the red E-stop button. Then, rotate the red power switch inside the mill enclosure to the 12:00 position.

Take your part out of the vise.

Remove Tool 42 and return it to its appropriate position in the toolholder.

Remove parallels, wipe them clean, and return them to their holder.

Return calipers to their case and position on the right side of the mill enclosure.

Discard the index card you were using to record measurements.

Ask the Training Ninja to find the vacuum for cleaning the CNC. Vacuum fully inside the enclosure.

Wipe the vise completely free of sawdust and debris.

Close enclosure doors.

Have your Training Ninja take a picture of you and your work in front of the CNC mill.