

SETUP & OPERATE TRAINING GUIDE

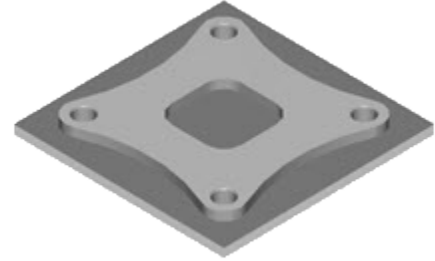


CNC-MILL-LESSON-2

Setup & Operate Training Guide

Objectives

You will machine the following part on the Haas CNC Mill. You will be working through all the steps required to complete this objective from machine power up to part inspection.



☞ To machine this part, you must:

Assemble tools, tool holders, material and work holding attachments.

Power the machine up.

Load the tools into the machine.

Load the material into the machine.

Set the offsets for the tools and material.

Load the program and verify it.

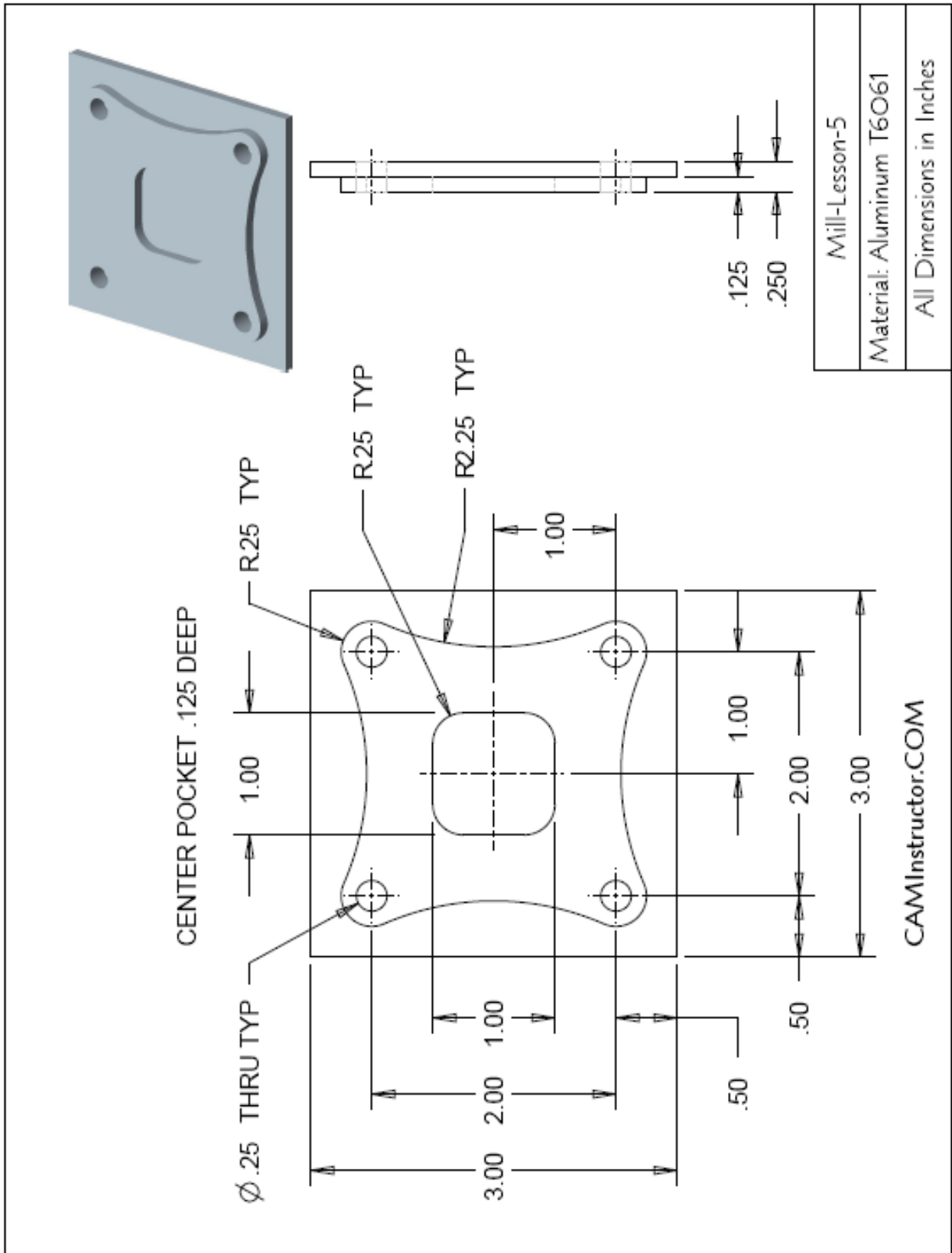
Run the program.

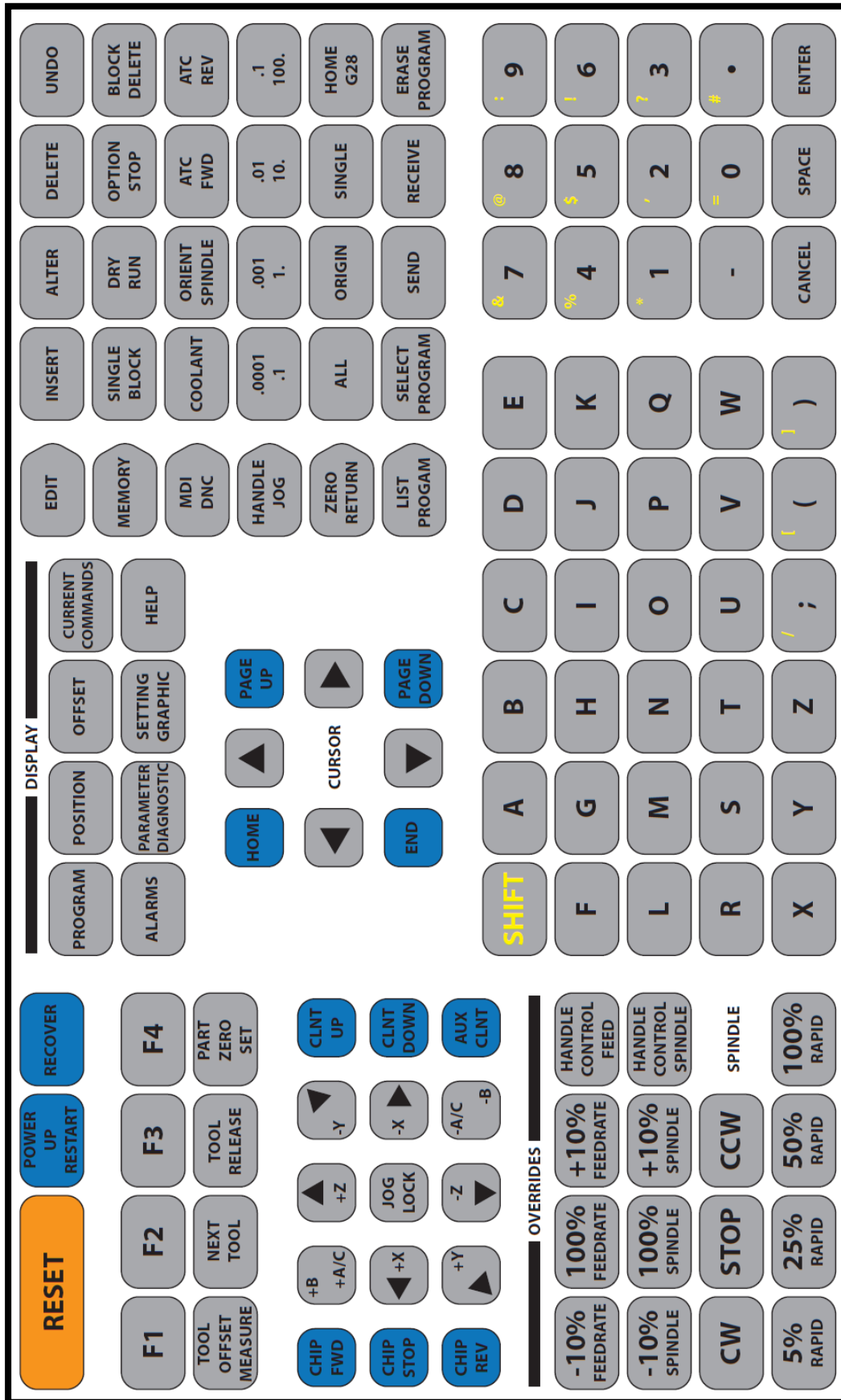
Inspect the completed part.

Disclaimer

1. The machine that will be used in this lesson is the **Haas V F 2 CNC Milling Machine**. Although all Haas machines are very similar, make note that some buttons or machine features may vary on your machine.
2. Before attempting to complete the tasks in this lesson, first get the permission from your instructor/supervisor.
3. The instructions in these lessons are to be used as a guide only. They do not cover all aspects of safety that need to be considered when working with tools and machinery of this type. Therefore, it is your responsibility to ensure that you understand and follow the safety procedures necessary to operate this type of equipment.
4. If you're not sure about a task or step, ASK your instructor/supervisor for help. These machines are expensive and safety is of paramount concern.

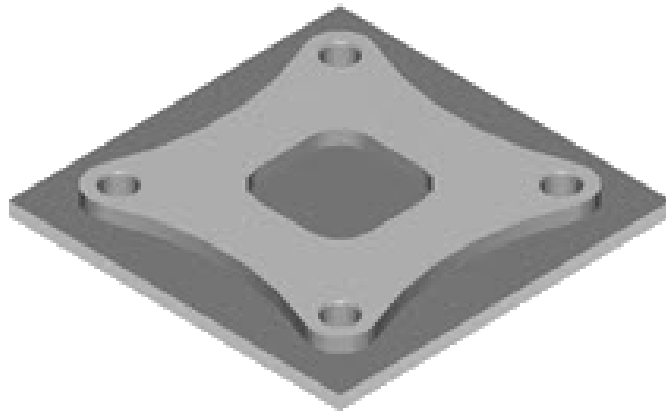
CNC MILL LESSON 2- DRAWING





LESSON-2 - THE PROCESS

- TASK 1:** Assemble Tooling, Material and Workholding
- TASK 2:** Machine Power On and Jogging
- TASK 3:** Loading Tools in Tool Changer
- TASK 4:** Set the Work Offset
- TASK 5:** Setting the Tool Offsets
- TASK 6:** Load, Verify and Run the Program

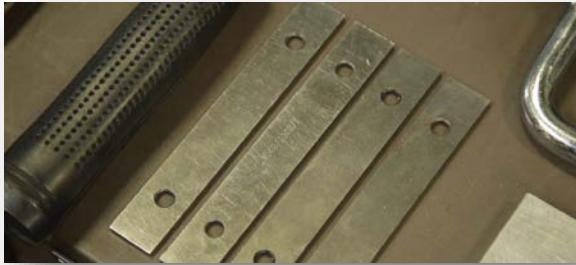


TASK 1: ASSEMBLE TOOLING, MATERIAL AND WORKHOLDING

Assemble Tooling

Before starting this lesson, you will need the following items.

1. The **CNC program** on a **USB stick**. You can use your own program or download it from our website. Whichever program you choose to use, it should still be checked before running it on the CNC machine. Note, the CNC Program for this lesson can be generated from Lesson 5 in CamInstructor's Mastercam Training Guide - Mill 2D www.caminstructor.com/products/mastercam-2017-mill-2d
2. A **Setup Sheet** for the part. This can be found at the end of this Lesson or can be downloaded from the CamInstructor course site.
3. **Parallels** that are high enough that the top of the part is slightly above the top of the vise.



4. The **Material**. This is a piece of **3" by 3" by 1/4" thick 6061 aluminum**. Our part has been previously cut to size and deburred. This can be seen in the above photo.

5. A **3/8" high speed steel spot drill** and a holder for it. We have chosen a drill chuck to hold this tool but a collet holder setup could be used as well.



NOTE: Ensure the Tool Holders are the correct type for this Machine.



6. A **1/4" high speed steel drill** and a holder for it.



Setup & Operate Training Guide

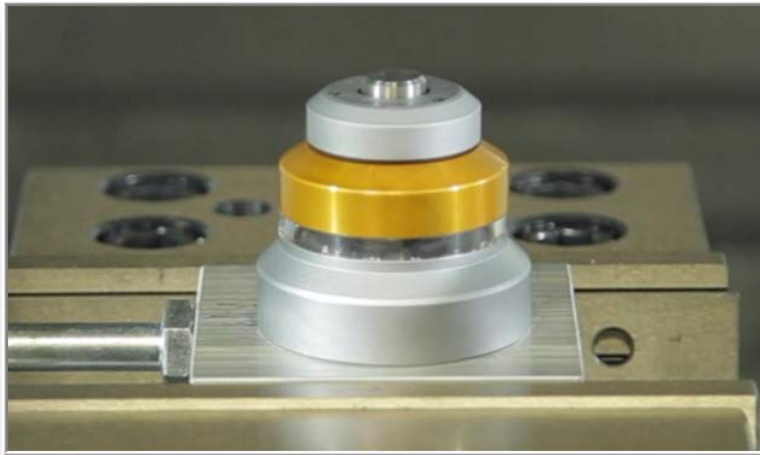
7. A 1/2" Flat End Mill and a holder.



8. An edge finder and a holder.

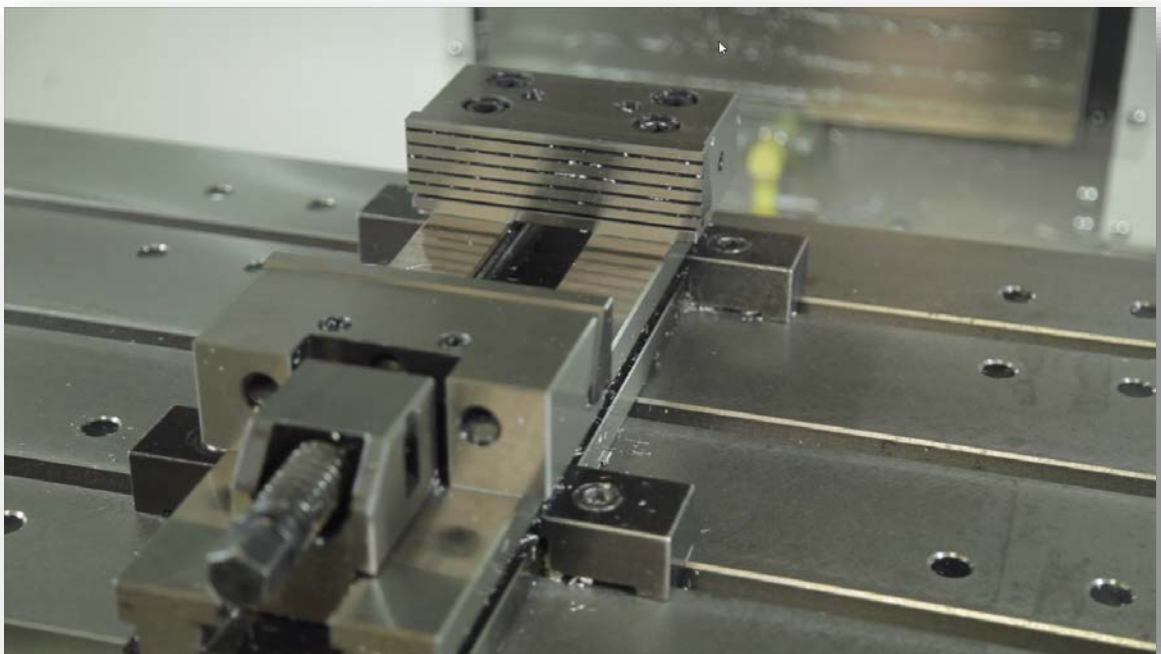


9. A **touch off gauge** for setting tool lengths. An electronic setting gauge is used in this Lesson. For instructions on using alternative methods for setting the Tool Length offset go to the Tips section.



10. Miscellaneous hand tools such as a soft faced hammer, Allen keys, wrenches and measuring calipers.

11. A vise to hold the material.



Setup & Operate Training Guide

Assemble Tools into their Tool Holders

- ☞ You can use the Tool Holding Vise on the front of the machine to perform this task. If the Machine does not have a Tool Holding Vise ask your instructor/supervisor for an alternative. Ensure that the holder is being held securely before tightening with a chuck key or collet wrench.

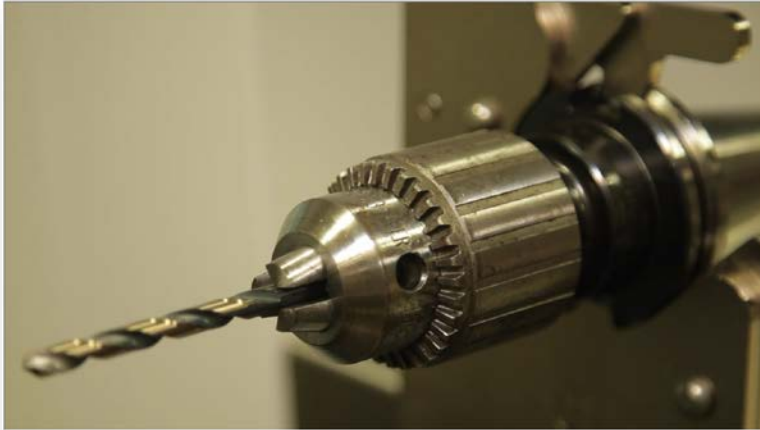


- ☞ Remember to be careful with the sharp tools and ensure all parts are clean before assembling.
1. Ensure both drive dogs are engaged into the holder before attempting to tighten a tool. Take the 3/8" spot drill and install it into the drill chuck. Insert at least 1" of the tool into the holder so it will be held firmly. Keep in mind, the less the tool sticks out the more rigid that tool will be.



2. Use the correct size chuck key to tighten the drill chuck.

3. Repeat this same process for the 1/4" drill. Insert the drill so that the flutes do not come into contact with the holding portion of the tool holder. Again, use the correct sized key to tighten the chuck.



4. Repeat this same process for the 1/2" End Mill. Insert the End Mill so that the flat lines up with the set screw in the Holder.



5. If necessary, repeat these same steps for the edge finder.

Setup & Operate Training Guide

6. These tools are now ready to be loaded into the machine.



7. Continue to **TASK 2** to learn how to power up the machine.

TASK 2: MACHINE POWER UP

➡ In this task, we show you the procedure for Powering Up the Machine.

1. Walk around to the back of the machine and **Turn on the Main Power.**



2. Ensure the **Compressed Air** is running and set to the proper **PSI**.



Setup & Operate Training Guide

- On the Front Control Panel press the **POWER ON** button. The Haas logo should appear on the Control screen. The machine will do a self-test and boot sequence and the display will show the startup screen. This may take about 20 to 30 seconds to boot up.



➡ The startup screen gives basic instructions **(1)** to start the machine.

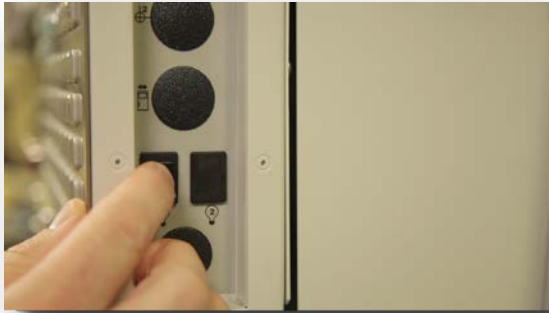
- Turn the **EMERGENCY STOP** button **(2)** to the right to reset it.



- Now press **RESET** as shown in **(4)** above, to clear the startup alarms. If you cannot clear an alarm, the machine may need service. **BE SURE** to let the instructor/supervisor know and do not continue.

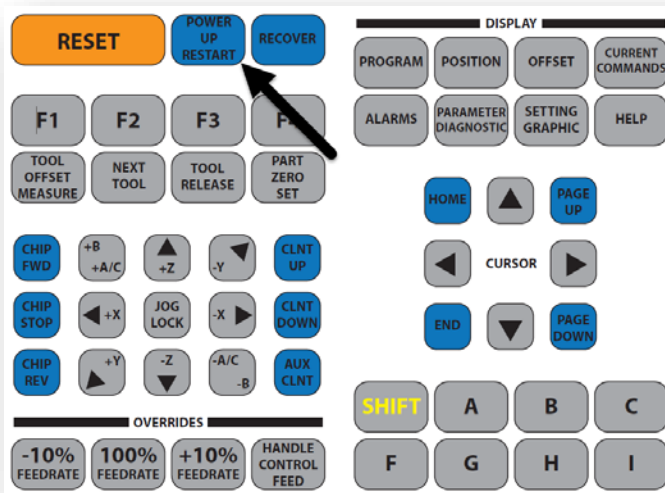
WARNING: Before you proceed with the next step, remember that automatic motion begins immediately when you press **POWER UP/RESTART**. Make sure the motion path is clear. Stay away from the spindle, machine table, and tool changer.

- Turn on the interior Light (if it is not on).



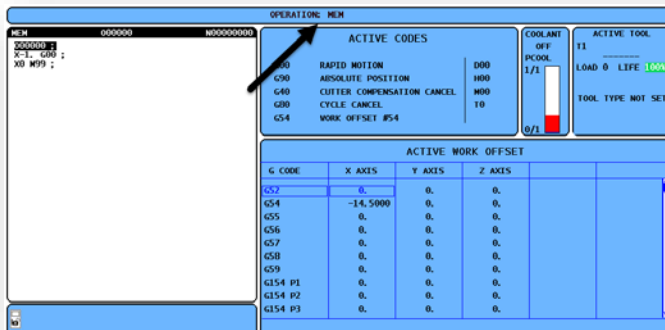
- For this next step, the Doors on the Machine need to be closed.

- Press **POWER UP/RESTART**.



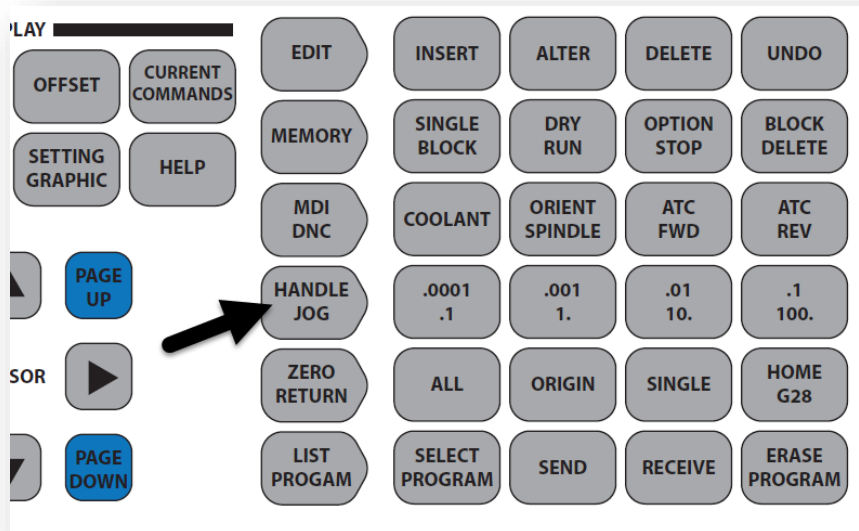
- The Axis will rapid toward their home positions and then move slowly until the machine finds the home switch for each axis. This establishes the machine home position.

- Notice that the control is now in **OPERATION:MEM** mode.

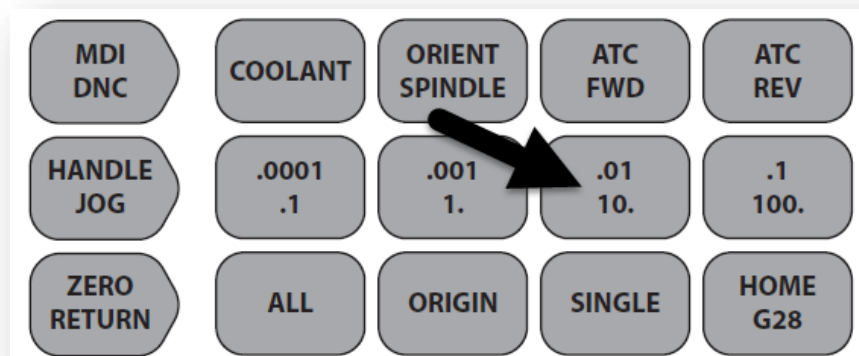


TASK 2B: JOG THE MACHINE

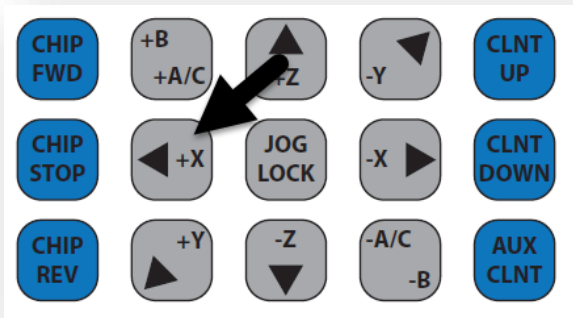
- Before setting the **Work and Tool Offsets** we need to go over the basics of Jogging the Machine.
 - Jog Mode lets you jog the machine Axis to a desired location. Before you can jog an axis, it must have the home position established. The control establishes the home position at machine power-up which we did at the beginning of this Task.
1. Press the **HANDLE JOG** button on the Control Panel.



- There are different increment speeds that can be used while in jog mode; they are .0001, .001, .01 and .1. We will NOT be using the .1/100 speed setting as the machine moves too quickly on this setting.
2. Select the **.01** button



3. Select the **X Axis Button** in the Jog Panel



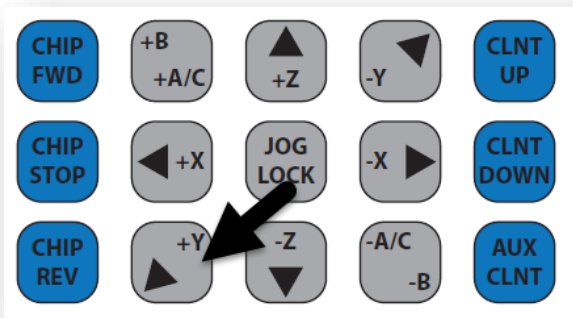
4. Use the HANDLE JOG control to move the axis.



- Notice as the HANDLE JOG is turned the machine table moves left and right.

WARNING: Make sure the Spindle doesn't collide with the Table and/or Vice.

5. Press the **Y Axis Jog Button** in the Jog Panel.

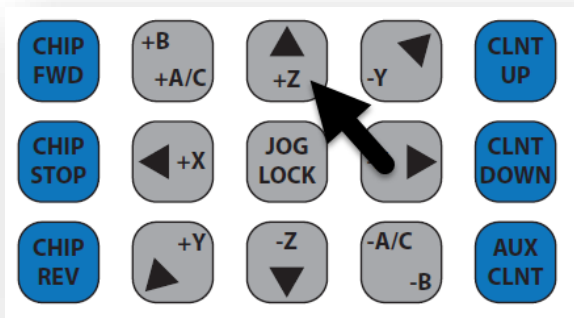


Setup & Operate Training Guide

6. Use the **HANDLE JOG** control to move the **Y axis**. Notice the table moves in and out.



7. Select the Z Axis Button in the Jog Panel.



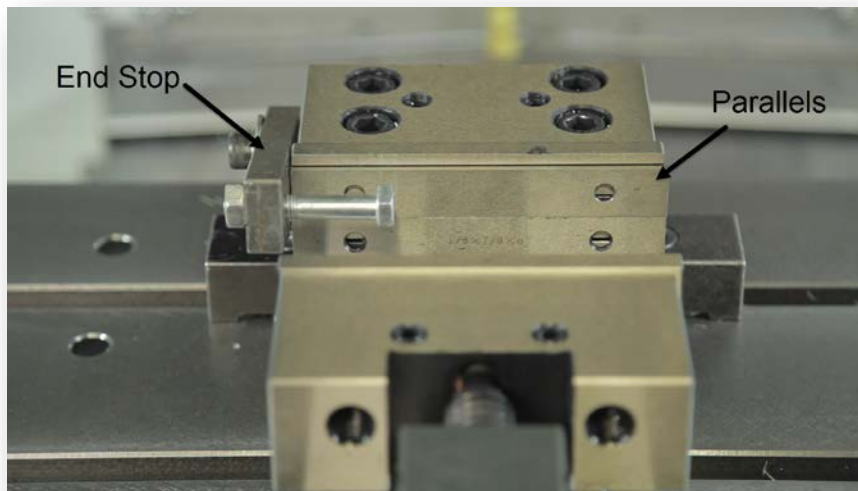
8. Use the **HANDLE JOG** control to move the **Z axis** being careful not to move the spindle into the table, and/or vice. Notice the spindle moves up and down. **Be careful not to run the Spindle into the Table or Vice.**



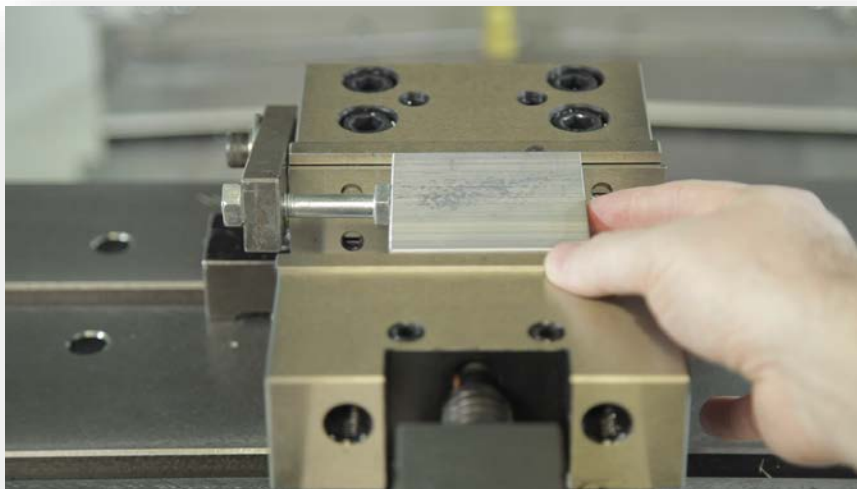
9. Practice moving the **Table (X and Y)** the **Spindle (Z)** in different directions and at different jog modes to get an idea of how it works.

SETTING THE MATERIAL IN THE VICE

- For the following steps, we will assume the Vice has already been fastened and squared onto the Milling Machine Bed.
 - For instructions on how to do this go to the TIPS section and view setting up a Vice on a CNC Milling Machine Table.
1. Ensure the **Parallels** and a material stop are positioned properly. For this lesson, we will be using an end stop on the **Vise** to ensure the part always goes in the same position. That way we don't have to set the work offsets each time we load a new piece of material.



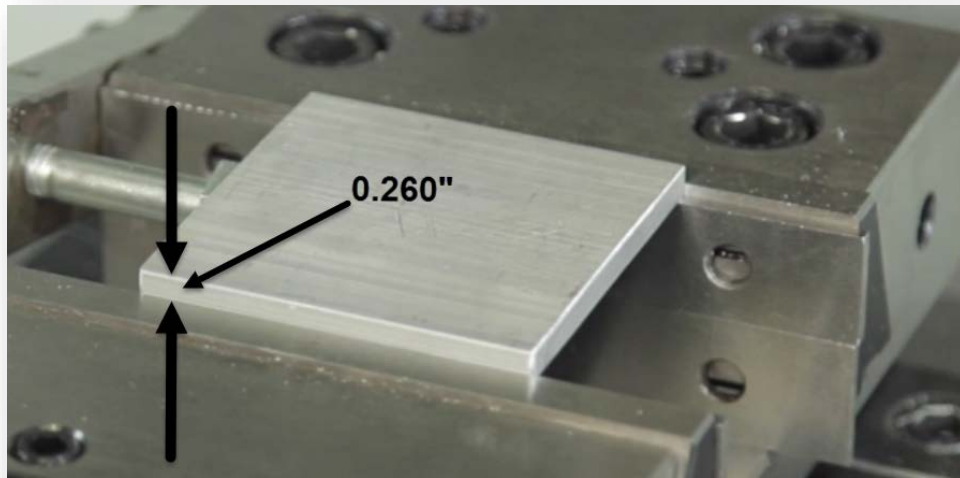
2. Slide the **Material** up against the stop and then close the **Vice Jaws**.



Setup & Operate Training Guide

NOTE:

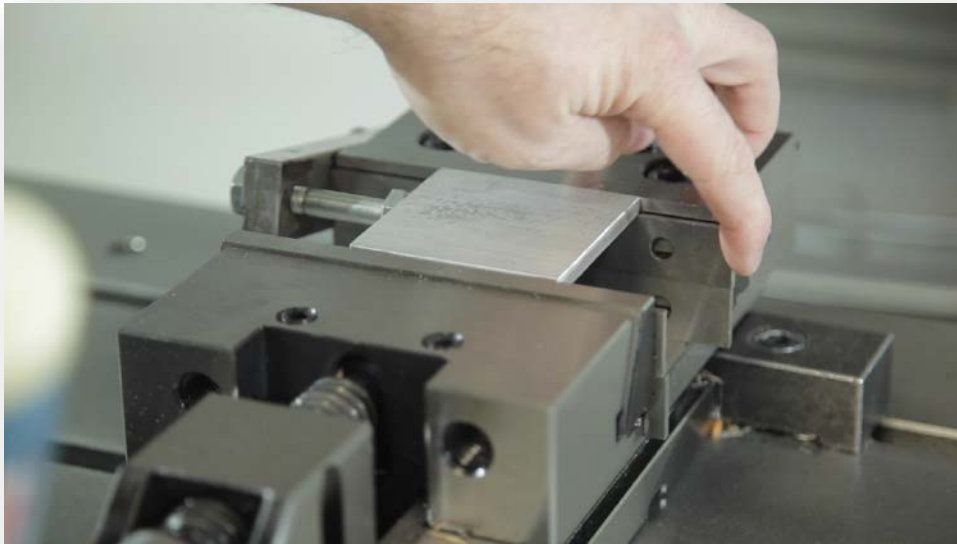
The material needs to be sitting 0.260 inches above the vice jaws because the program machines around the outside of the part. Check with a measuring tool that the distance above the workpiece is sufficient.



3. Using a **Soft Faced Hammer**, lightly tap the material down onto the parallels to ensure they are firmly resting on the **Parallels**.



4. You can check the **Parallels** by trying to move them. If they move then the material is not firmly positioned and will require another tap from the hammer.



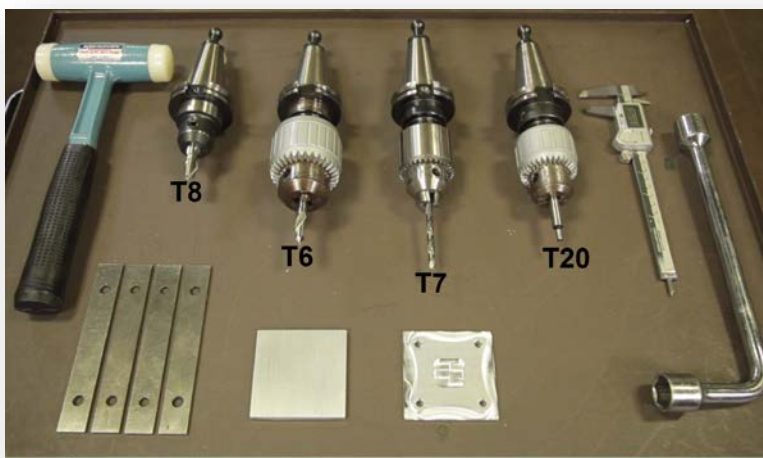
5. Remove the **Vise Handle** and **Close the Doors** on the Machine.

TASK 3: LOAD THE CUTTING TOOLS IN THE TOOL CHANGER

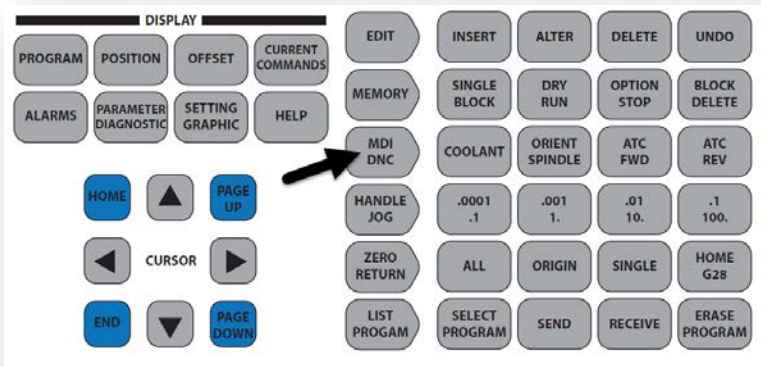
- Before proceeding with this Task, check that your instructor/supervisor has given the okay to proceed.
 - The Tools and edge finder are loaded into the umbrella tool changer by first loading the tool into the spindle. Be sure that all the Tools are in their holders and within reach.
 - NOTE: The Tool numbers in this Lesson do not correspond to the Mill 2D Lesson 5 Tool numbers. They have been changed to correspond to this Lesson.
1. Ensure the **Tool Carousel** is empty of all Tools. If it is not, DO NOT PROCEED. Check with your instructor/supervisor.



2. Organize the tools to match to the CNC program. In this case we are using 3 tools, the **Spot Drill** which is **Tool 6**, the $\frac{1}{4}$ " **Drill** which is **Tool 7** and the $\frac{1}{2}$ " **End Mill** which is **Tool 8**. We will also be using an **edge finder** and be putting it in **Tool 20**.



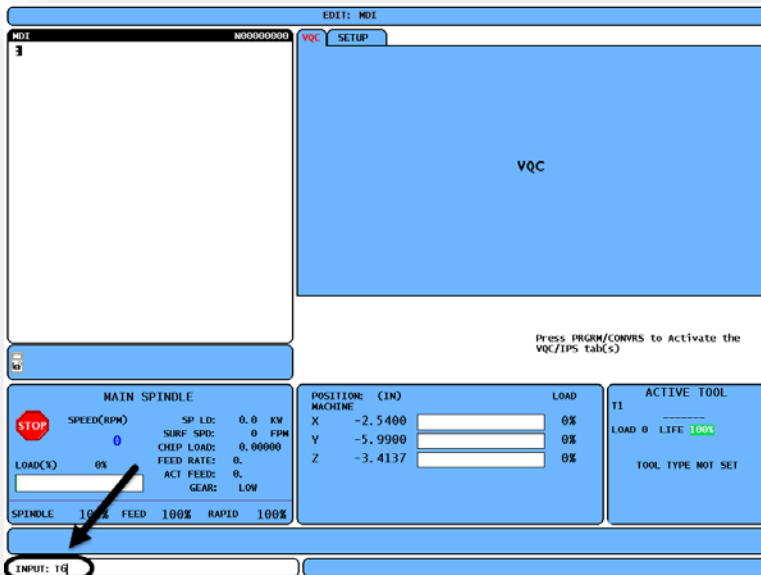
- With the **Machine Doors Closed**, Press the **MDI Button** on the Control Panel Keyboard.



- Press **T** and **6** [for Tool 6] (letter T and number 6).



- Notice on the Controller that **T6** is listed in the comment line.



Setup & Operate Training Guide

WARNING: Before Proceeding make sure there are NO tools in the tool changer.

- Press the **ATC Forward** Button to advance the tool changer.



- The Tool Changer will cycle to the **Tool 6** Pocket.

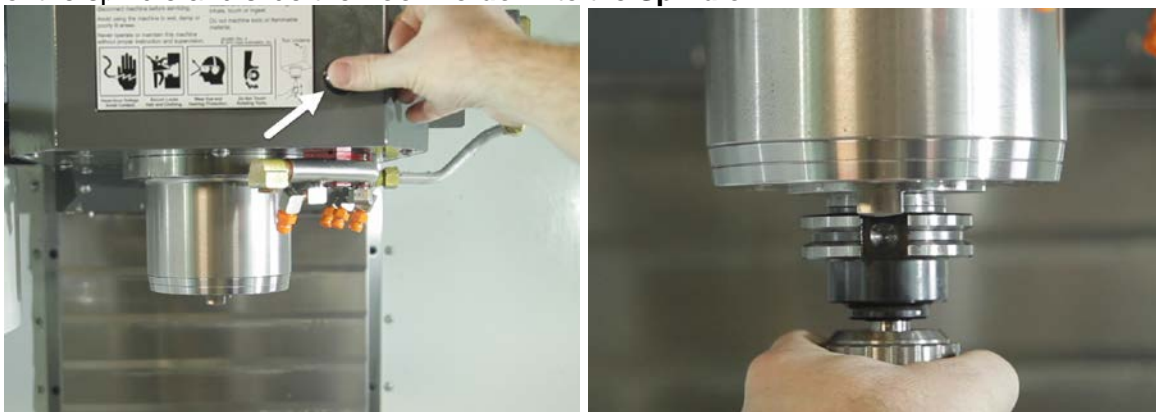


- Once the Machine finishes its cycle, **Open the Doors**.

WARNING: In the next step, make sure your hand does not come between the Holder and the Spindle.



9. Take **Tool 6**, the Spot Drill, in one hand and using the other hand **Push and Hold** the **Tool Release** button. Turn the tool so that the two cutouts in the tool holder line up with the tabs of the spindle and slide the **Tool Holder** into the **Spindle**

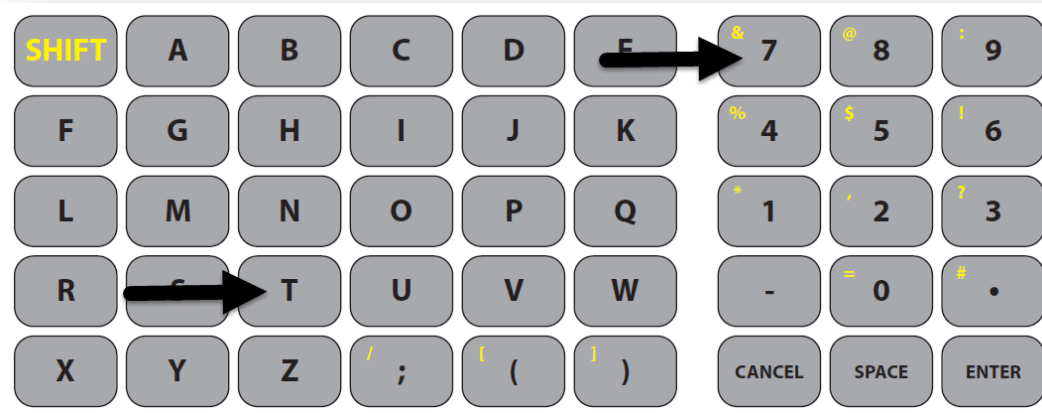


10. When the tool is fitted into the spindle, release the **Tool Release** button.

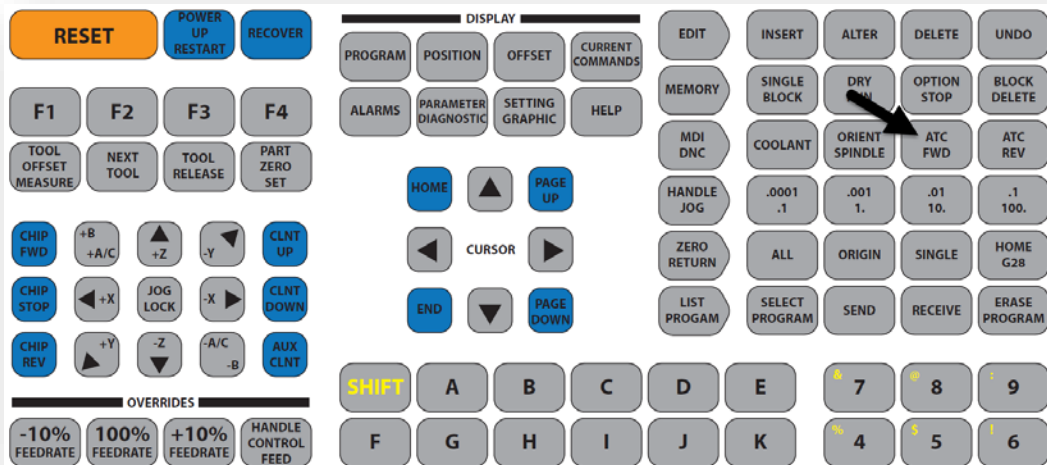


Setup & Operate Training Guide

11. Close the Machine Doors.
12. Press the **MDI** Button (if required).
13. Press **T** and then **7** [for tool 7] (letter T and number 7).

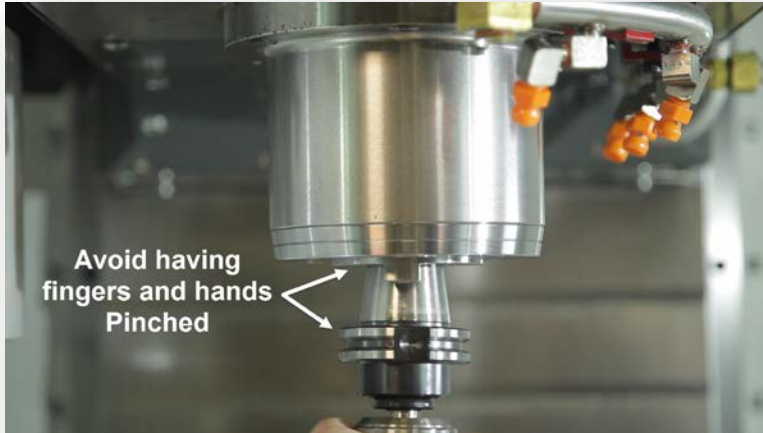


14. Press the ATC FWD button.



15. Open the Machine Doors.

WARNING: In the next step, make sure your hand does not come between the Holder and the Spindle.



16. Take **Tool 7** the ¼" Drill in one hand and using the other hand **Push and Hold** the **Tool Release** button. Turn the tool so that the two cutouts in the tool holder line up with the tabs of the spindle and slide the **Tool Holder** into the **Spindle**



Setup & Operate Training Guide

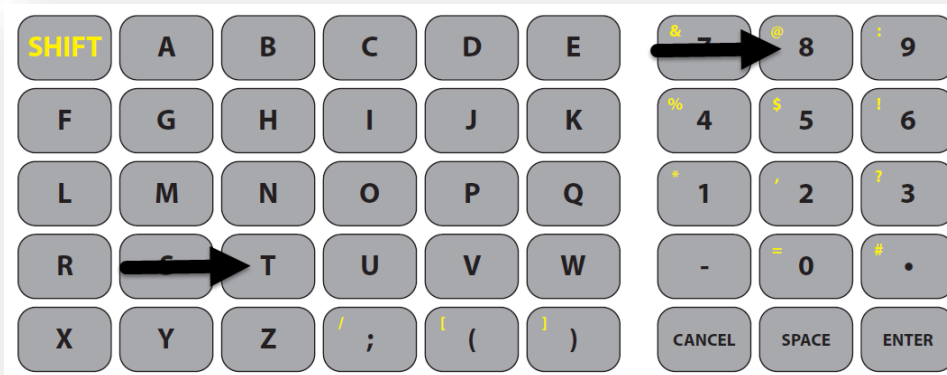
17. When the tool is fitted into the spindle, release the **Tool Release** button.



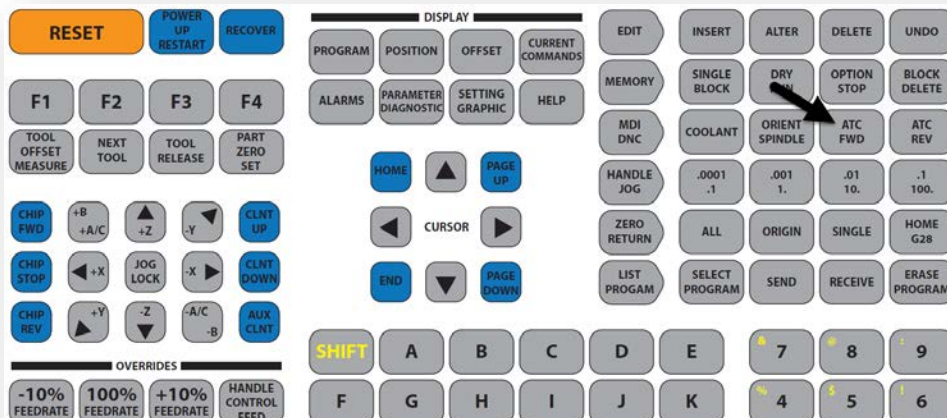
18. Close the Machine Doors.

19. Press the **MDI** Button (if required).

20. Press **T** and then **8** [for tool 8].



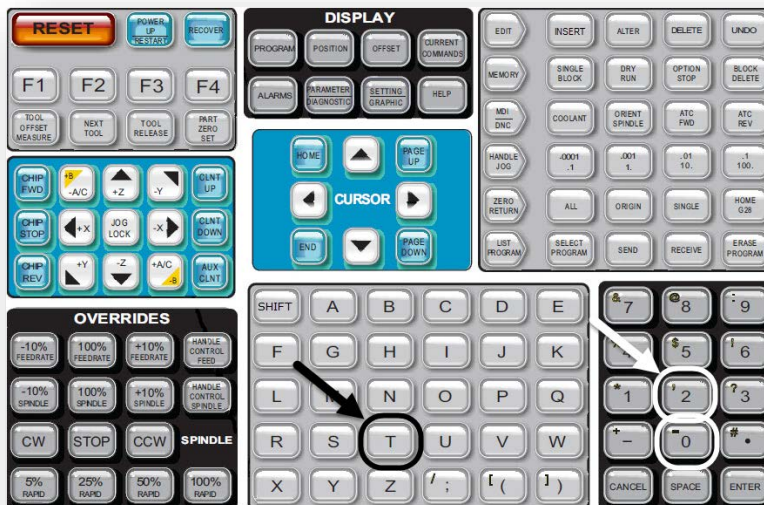
21. Press the **ATC FWD** button.



WARNING: In the next step, make sure your hand does not come between the Holder and the Spindle.



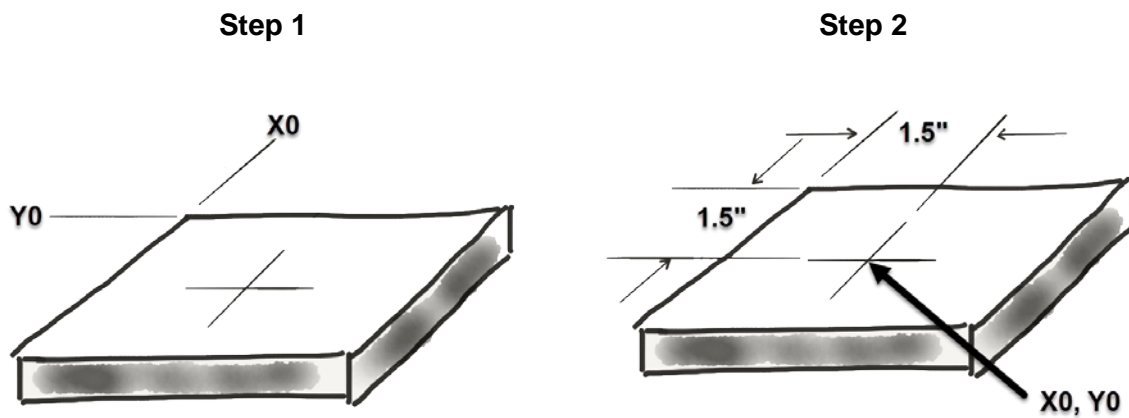
22. Take **Tool 8**, the ½” End Mill, and load it into the spindle as was done with the previous 2 tools.
23. Tool change to **Tool 20** and load the **Edge Finder** into the machine. Use the same procedure as was done with the previous 3 tools.



☞ **NOTE:** We use Tool 20 for the edge finder so it is well apart from the other tools being used in the machining operation.

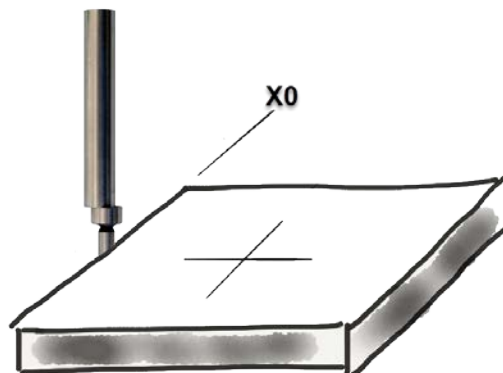
TASK 4: SET THE WORK OFFSET

- To machine a workpiece (part), the mill needs to know where the part is located on the table. You can use an edge finder, an electronic probe, or many other tools and methods to establish part zero. For this lesson, we are going to use the Edge Finder to set the Work Offset. Load an edge finder in the spindle.
- For this lesson, the work offset location is at the center of the part as shown in the image below. Therefore, the procedure to set the work offset will be done in 2 steps. **Step 1** will be to locate the corner of the workpiece. **Step 2** will be to locate the work offset at the center of the workpiece.



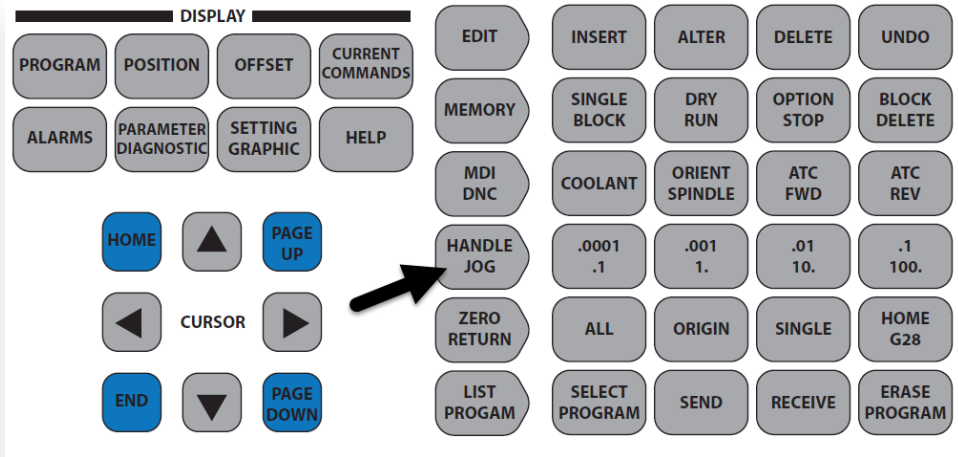
SET THE X AXIS OFFSET

- We will start by setting the X Axis offset using the left side of the workpiece.

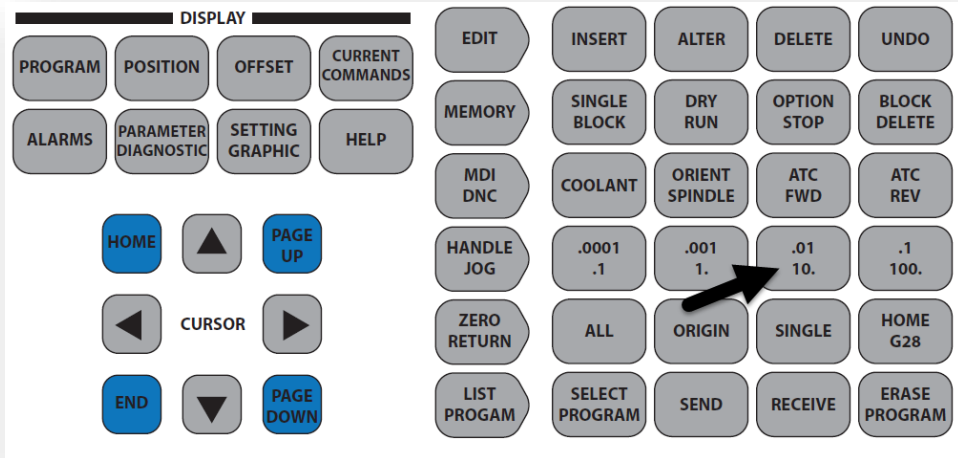


PAUSE:
 Tool 20 (the edge finder) should be in the Spindle. If it is not, do not proceed and check with your instructor/supervisor.

1. Close the Machine Doors.
2. Press the **HANDLE JOG** Button.

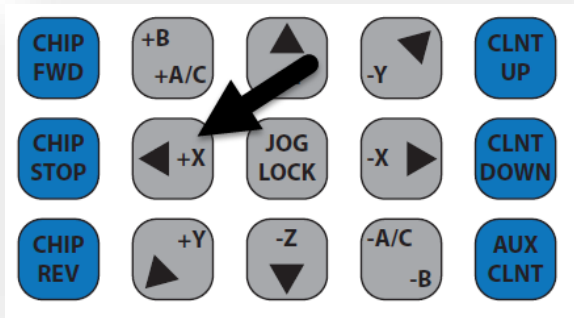


3. Press **.01/10**. (We use this setting so that the machine will move at a reasonable speed as we don't want to cause any damage).

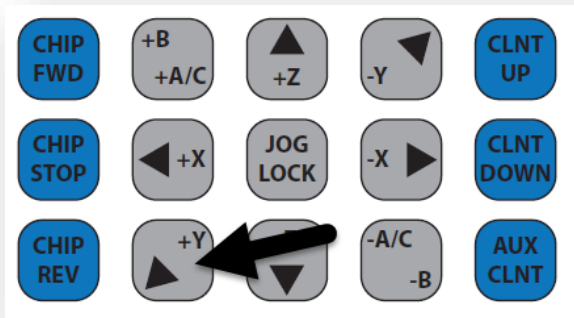


Setup & Operate Training Guide

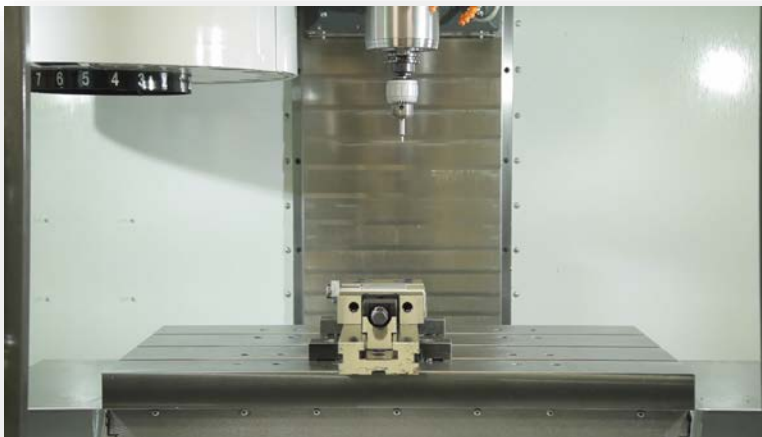
4. Select the **X Axis button** and Jog the Edge finder so it is over the top of the workpiece.



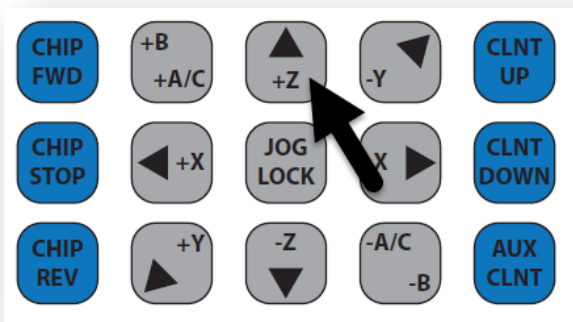
5. Select the **Y Axis button**.



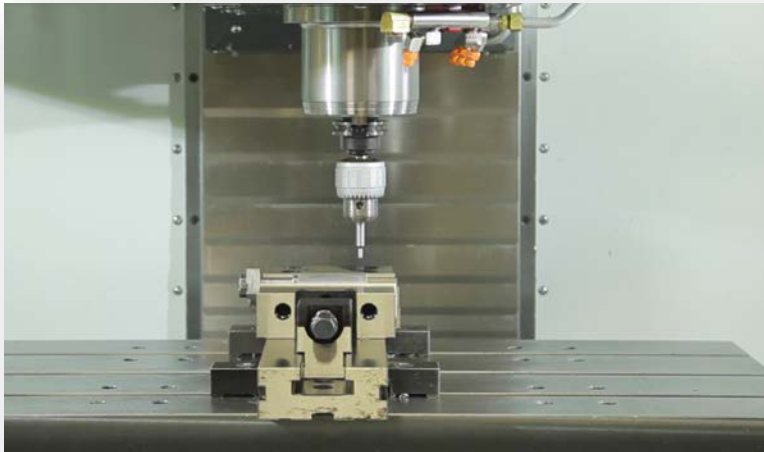
6. Jog the **Edge Finder** until it is over top of the workpiece.



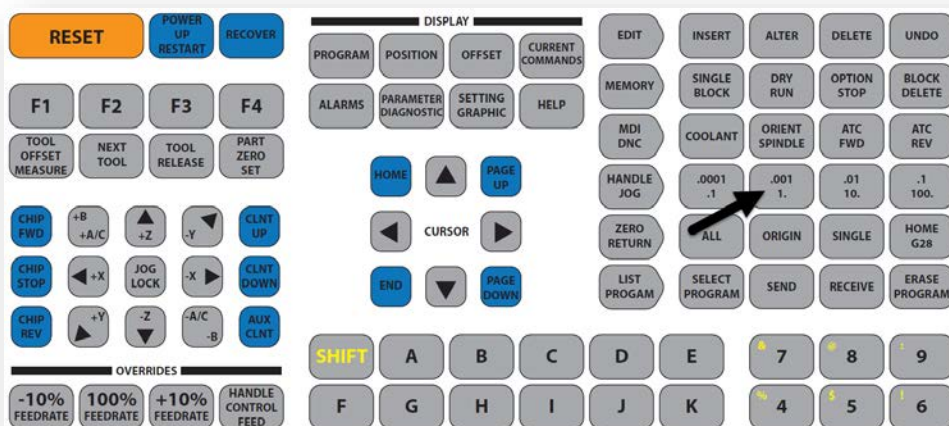
7. Select the Z Axis button.



8. Jog the Z Axis so the bottom of the edge finder is approximately 1" above the part.



9. Press the .001/1 button. (The mill will move at a slow speed when the handle is turned).

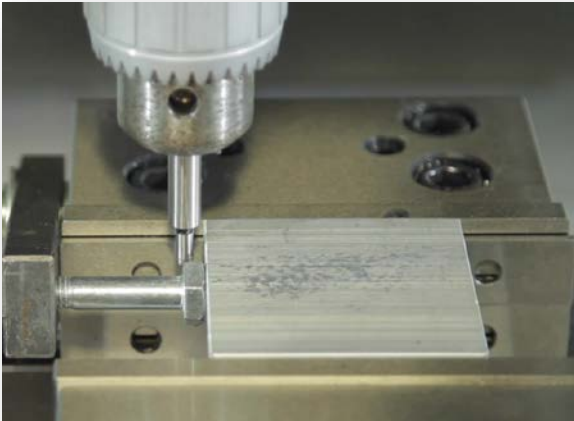


Setup & Operate Training Guide

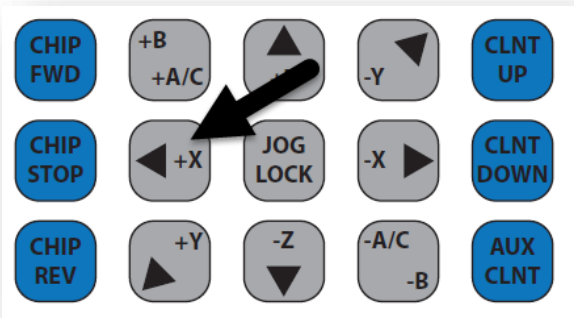
10. Jog the **Edge Finder** in X and Y so it is very close to the left edge of the workpiece.



11. Jog the **Z-Axis** approximately **0.2"** below the part.



12. Select the **X Axis**.



13. Set Spindle Speed to **750 RPM** by pressing the **7 5 0** Buttons on the numeric key pad followed by pressing the **CW** Button (for clock wise).



14. Bump the **Edge Finder** with a pencil so it starts to wobble.

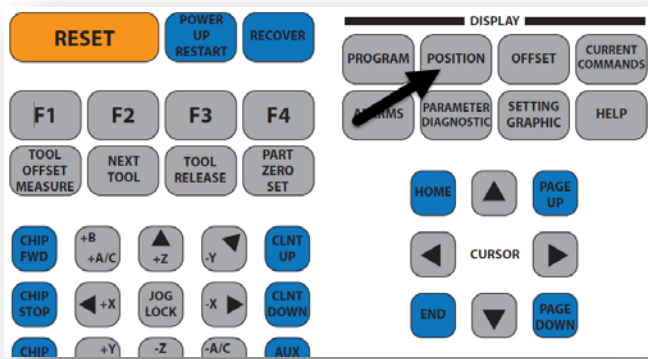


Setup & Operate Training Guide

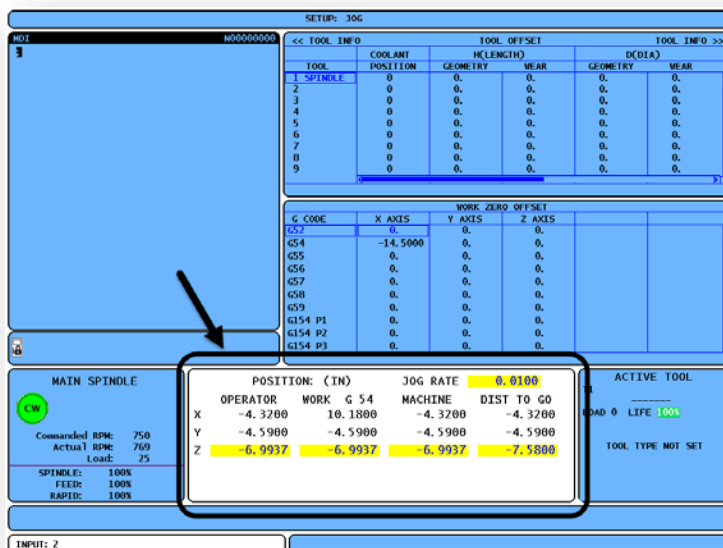
15. Jog the **Edge Finder** until it touches the part and rotates smoothly and then kicks out.



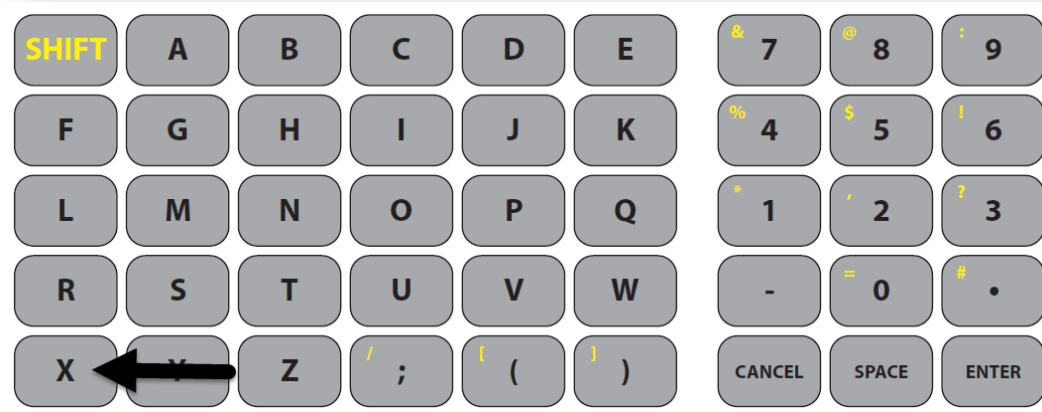
16. Press the **POSITION** Button.



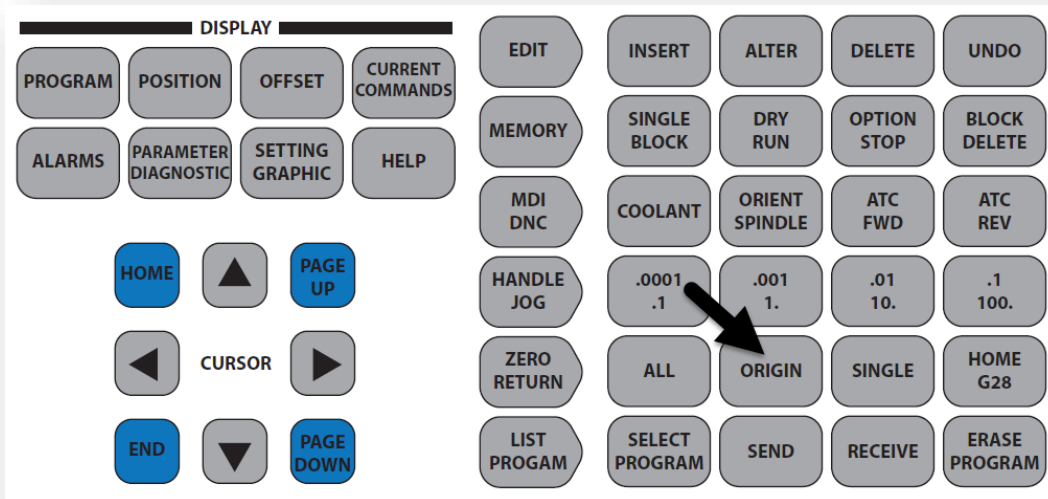
17. The **Position Screen** should be active (white).



18. Press the **X** button on the Keyboard.

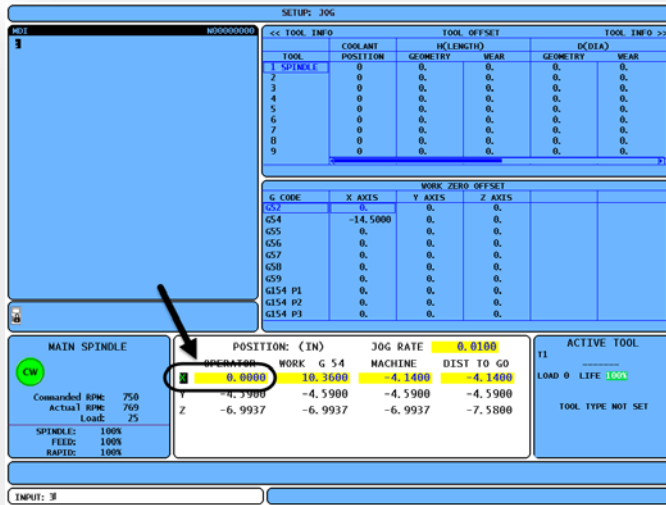


19. Press the **ORIGIN** button.



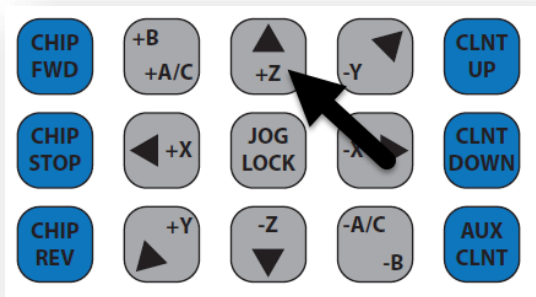
Setup & Operate Training Guide

20. It should indicate that X is **0.000**" under **Operator Column**.

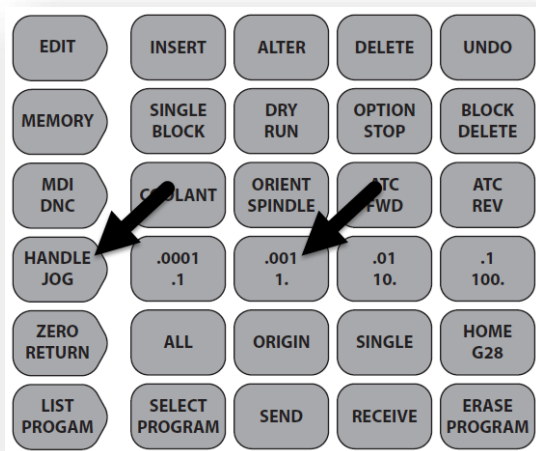


☞ You have successfully told the Machine's Controller that X 0.00 is at the left edge of the workpiece.

21. Select the **Z Axis**.



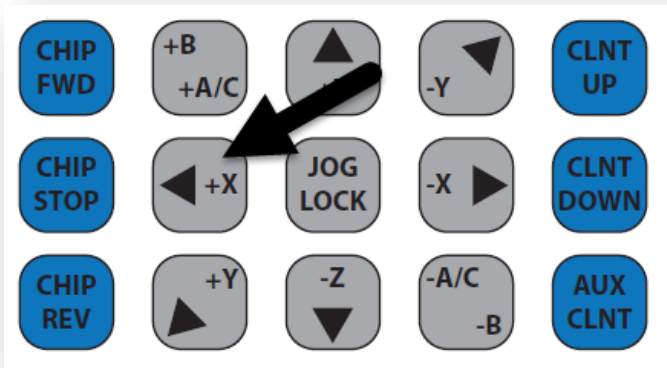
22. Press the **Handle Jog** button and the **.001/1**. Speed button.



23. Jog the Z axis up so the **Edge Finder** is above the workpiece.

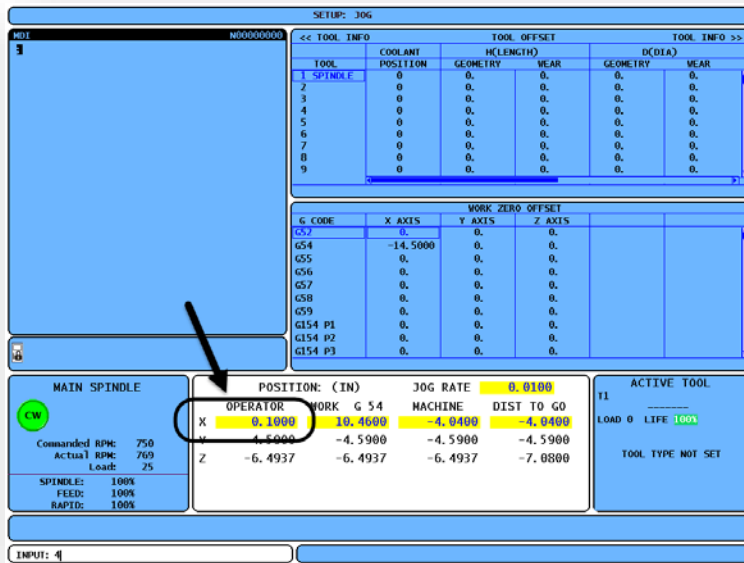


24. Select the **X Axis** button.

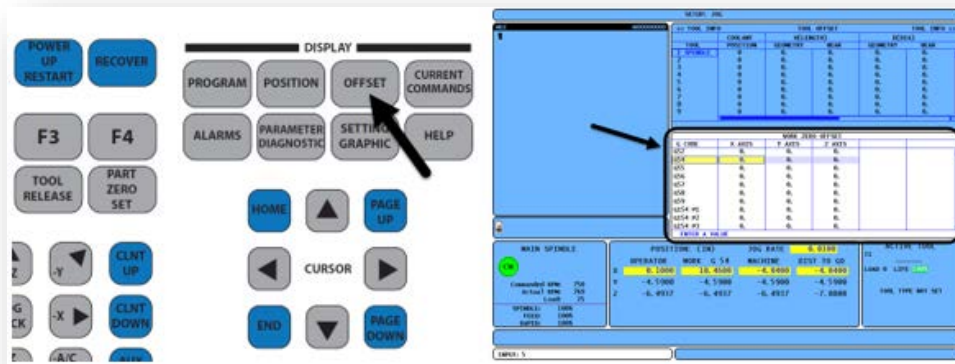


Setup & Operate Training Guide

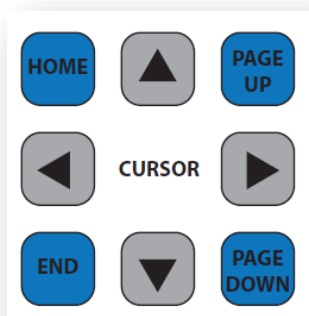
25. Jog 0.100" in the plus direction. Note the value of X in the **Position Panel** under the **Operator Column** should read **0.1000** as shown below.



26. Press the **OFFSET** button until the **ACTIVE WORK ZERO OFFSET** panel is active (highlighted in white).



27. **Cursor** to **G54** – (note G54 may not be visible on the screen. Keep pushing the cursor key until it becomes visible.)

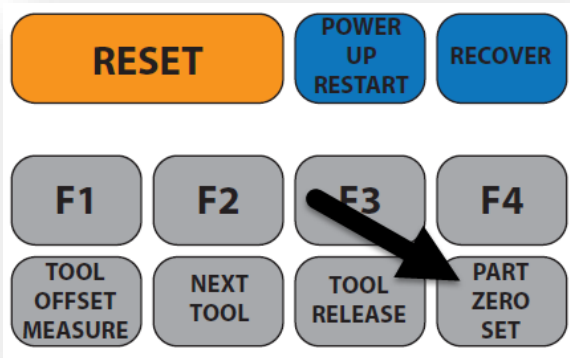


28. Ensure the **G54** and **X Axis Column** are highlighted as shown below.

WORK ZERO OFFSET			
G CODE	X AXIS	Y AXIS	Z AXIS
G52	0.	0.	0.
G54	0.	0.	0.
G55	0.	0.	0.
G56	0.	0.	0.
G57	0.	0.	0.
G58	0.	0.	0.
G59	0.	0.	0.
G154 P1	0.	0.	0.
G154 P2	0.	0.	0.
G154 P3	0.	0.	0.

ENTER A VALUE

29. Press **PART ZERO SET** to load the value into the X-Axis column.



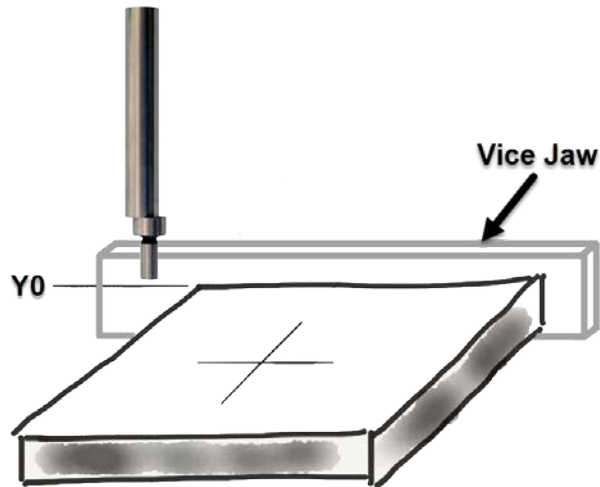
30. The **X AXIS OFFSET** figure should be updated.

WORK ZERO OFFSET			
G CODE	X AXIS	Y AXIS	Z AXIS
G52	0.	0.	0.
G54	-4.0400	0.	0.
G55	0.	0.	0.
G56	0.	0.	0.
G57	0.	0.	0.
G58	0.	0.	0.
G59	0.	0.	0.
G154 P1	0.	0.	0.
G154 P2	0.	0.	0.
G154 P3	0.	0.	0.

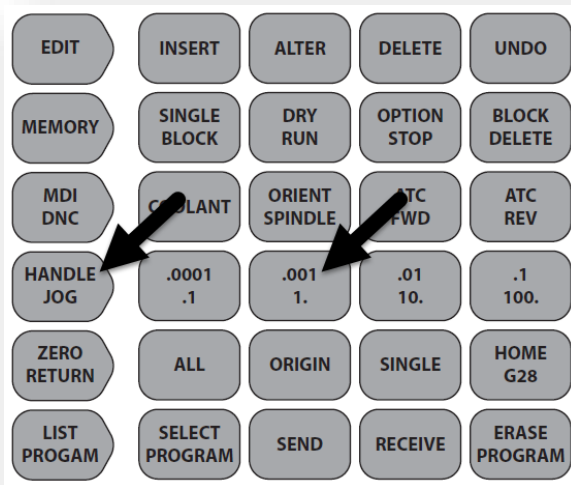
ENTER A VALUE

SET THE Y AXIS OFFSET

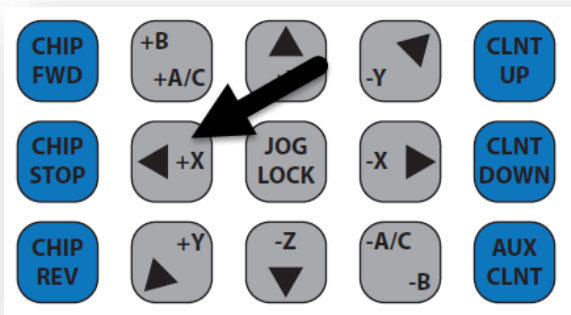
⇒ We will now set the Y Axis offset and will use the front face of the fixed Vice Jaw.



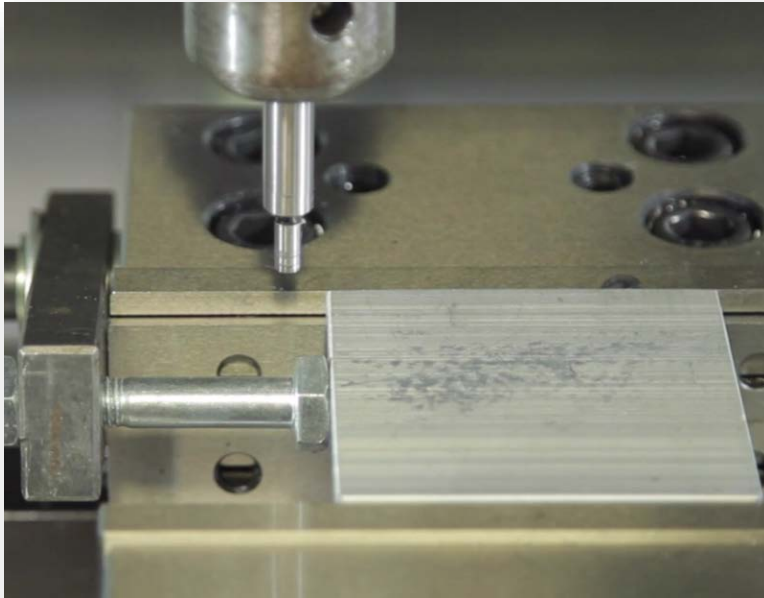
1. Press the **Handle Jog** button and the **.001/1.** Speed button.



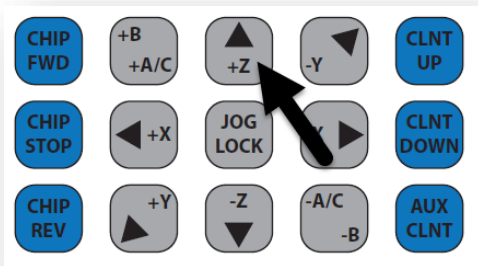
2. Select the **X Axis** button.



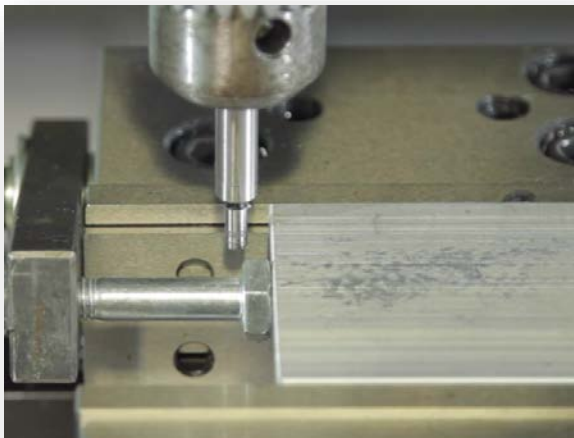
3. **Handle Jog** the Edge Finder away from the workpiece [X-].



4. Select the **Z Axis** button.

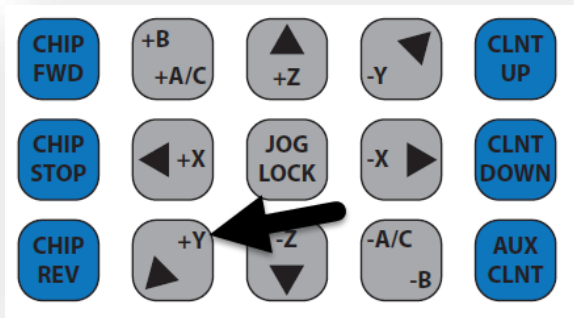


5. **Jog** the **Edge Finder** down [Z-] so it is below the top of the workpiece but still above the top of the parallel.



Setup & Operate Training Guide

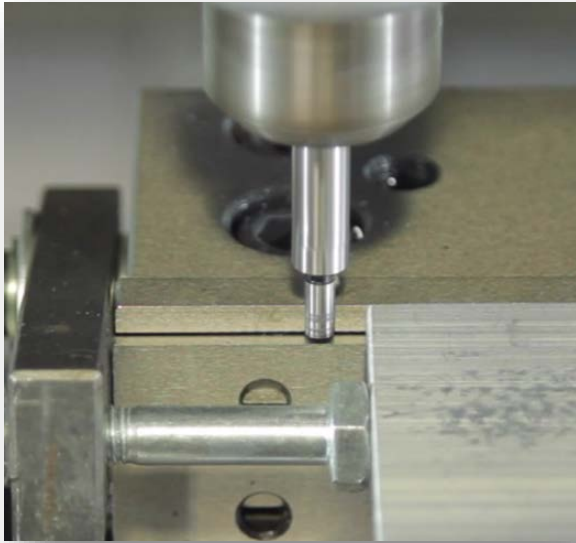
6. Push the **Y Axis** button.



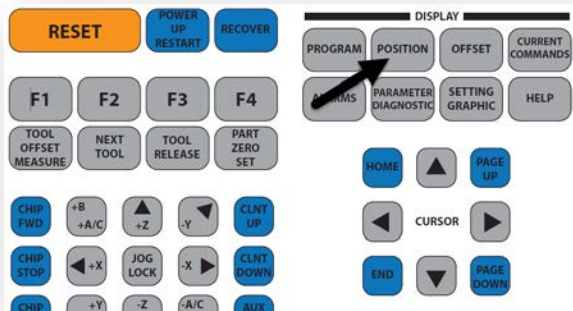
7. **Jog** the **Edge Finder** towards the fixed **Jaw** of the **Vice**. (NOTE: The Spindle should be turning. If it is not, press **CW** and **750** on the numeric keypad).
8. Bump the **Edge Finder** with a pencil so it starts to wobble.



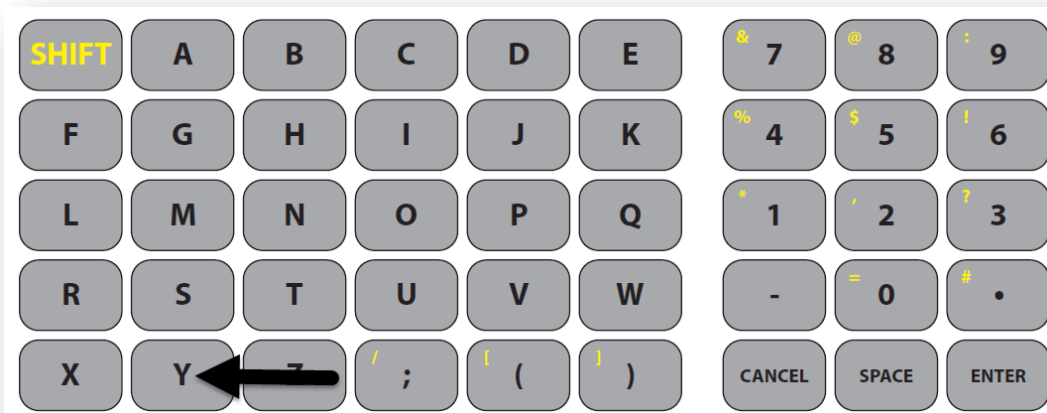
9. Jog the **Edge Finder** until it touches the Vice Jaw and rotates smoothly and then kicks out.



10. Press **POSITION** to Highlight the Position Screen.

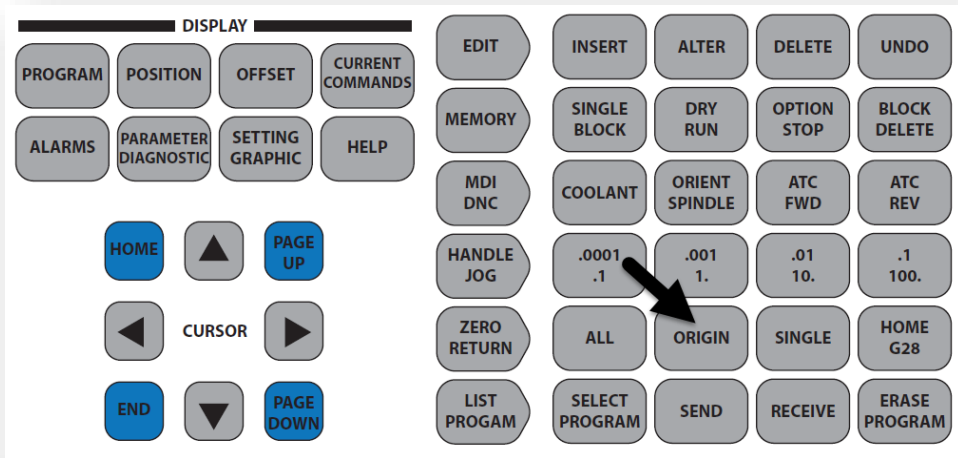


11. Press the **Y Axis** button on the Keyboard.

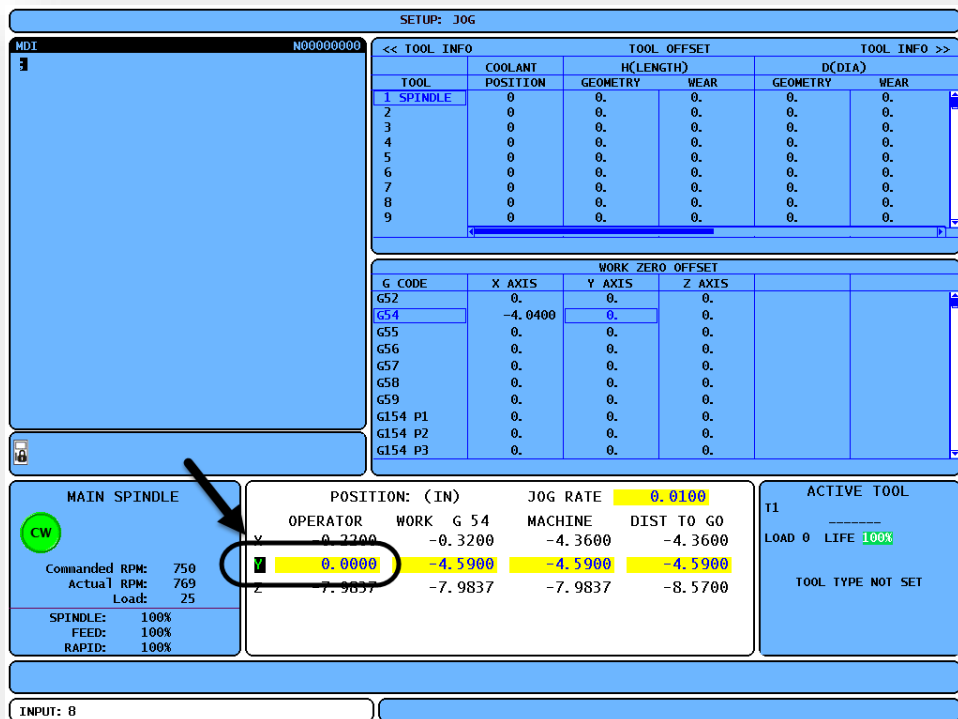


Setup & Operate Training Guide

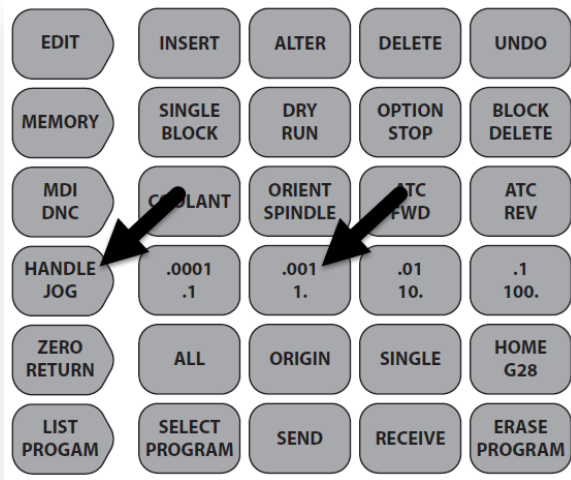
12. Press the **ORIGIN** button.



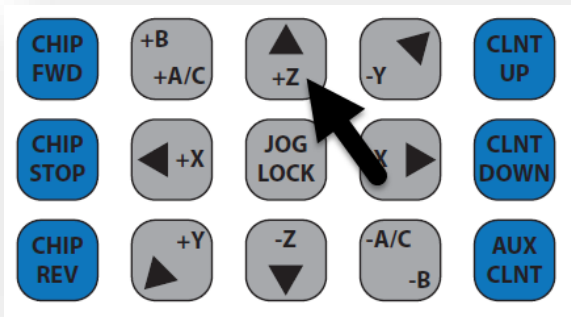
13. Under the **Operator Column**, Y should read **0.000**.



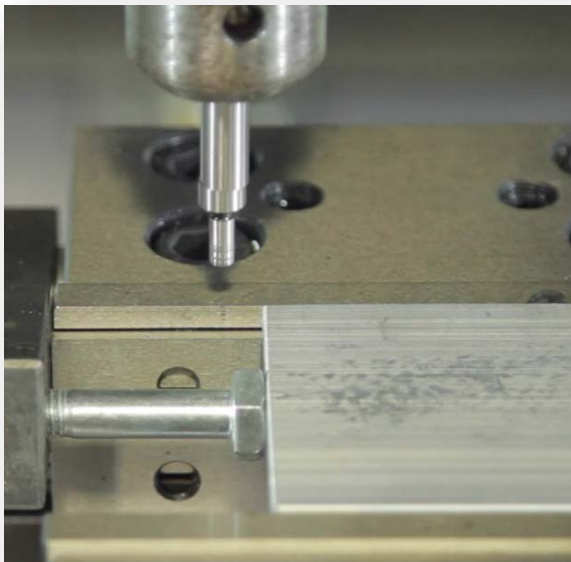
14. Press the **Handle Jog** button and the **.001/1.** Speed button.



15. Select the **Z Axis** button.

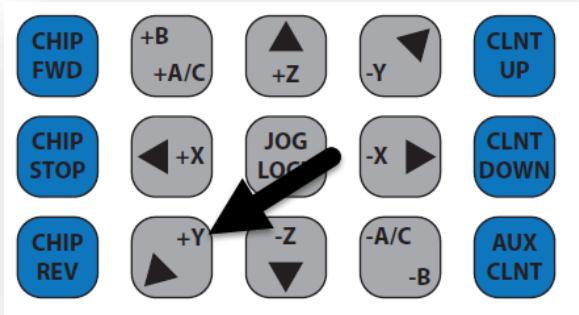


16. **Jog Z up [Z+]** so it is above the workpiece.

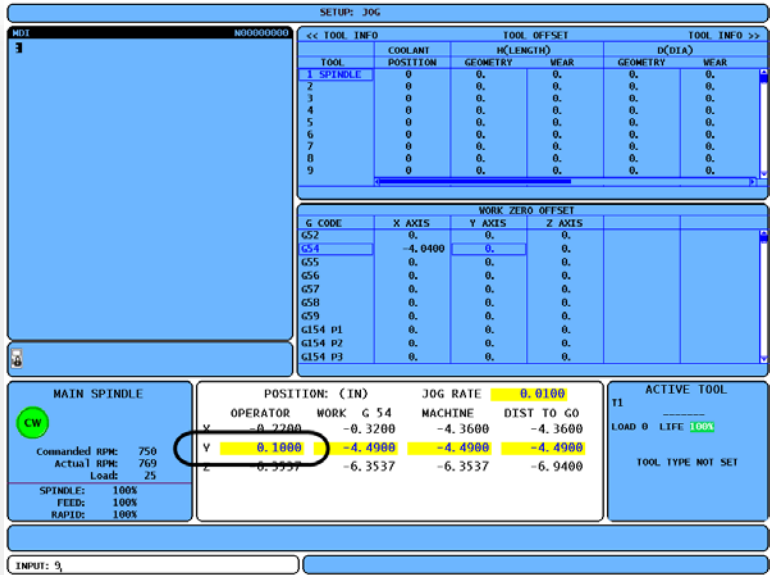


Setup & Operate Training Guide

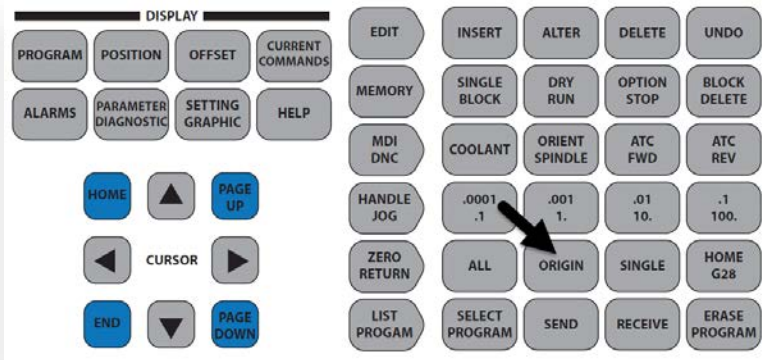
17. Select the Y Axis button.



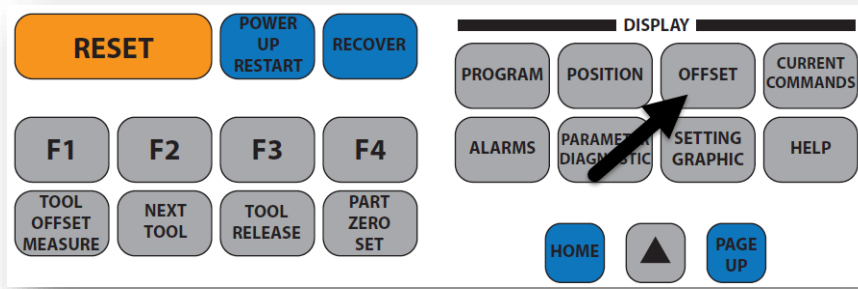
18. Jog Y in the plus direction 0.100". Note the value of Y in the Position Panel under the Operator Column.



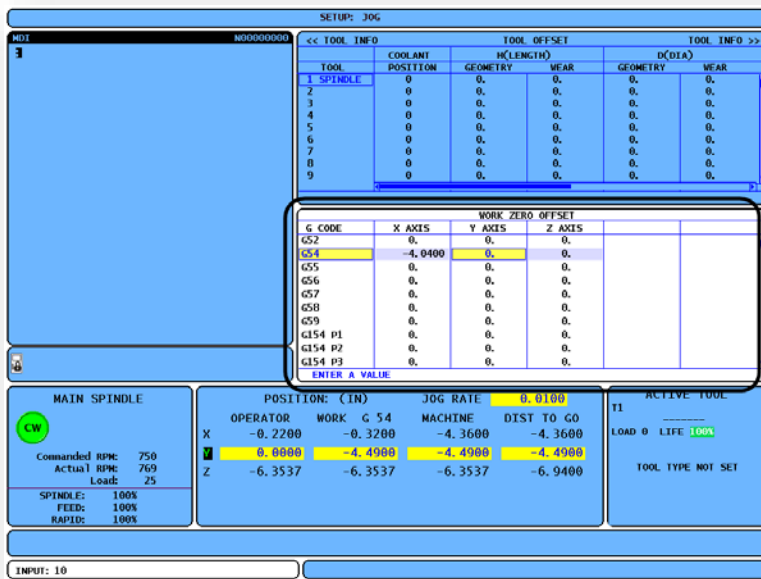
19. Press the Origin button.



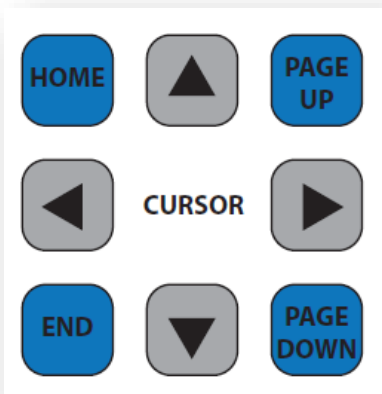
20. Press the **OFFSET** button.



21. ...until the **WORK ZERO OFFSET** panel is active (highlighted in white).

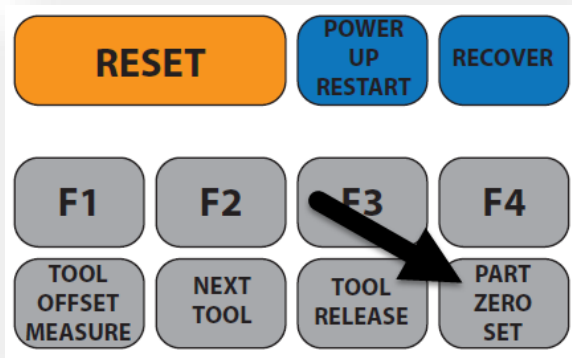


22. Cursor to G54.



Setup & Operate Training Guide

23. Press **PART ZERO SET** (ensuring the Y column is selected) to load the value into the Y-Axis column.

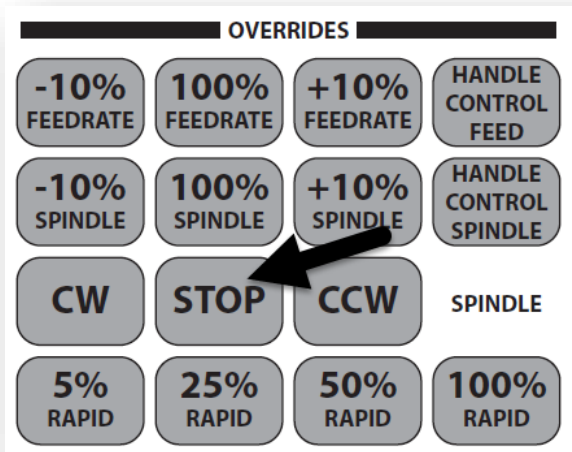


24. The Y AXIS OFFSET should be updated.

WORK ZERO OFFSET				
G CODE	X AXIS	Y AXIS	Z AXIS	
G52	0.	0.	0.	
G54	-4.0400	-4.4900	0.	
G55	0.	0.	0.	
G56	0.	0.	0.	
G57	0.	0.	0.	
G58	0.	0.	0.	
G59	0.	0.	0.	
G154 P1	0.	0.	0.	
G154 P2	0.	0.	0.	
G154 P3	0.	0.	0.	

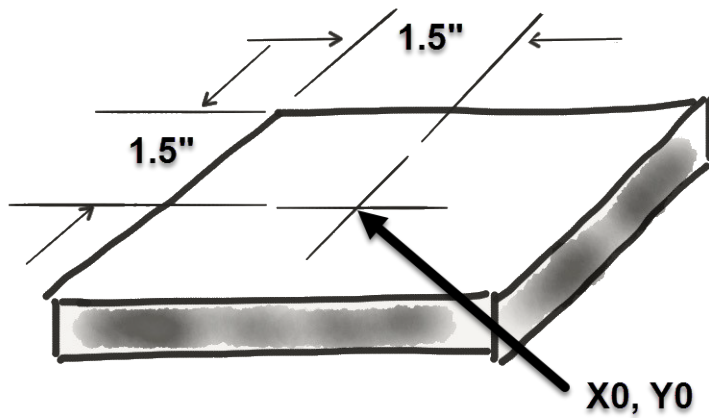
ENTER A VALUE

25. Turn the **SPINDLE OFF** by pressing the **STOP** key.



CHANGE THE WORK OFFSET TO THE CENTER OF THE WORKPIECE

- We now need to tell the machine that the work offset is at the center of the part, not the corner. This next set of instructions explains one way of doing that.

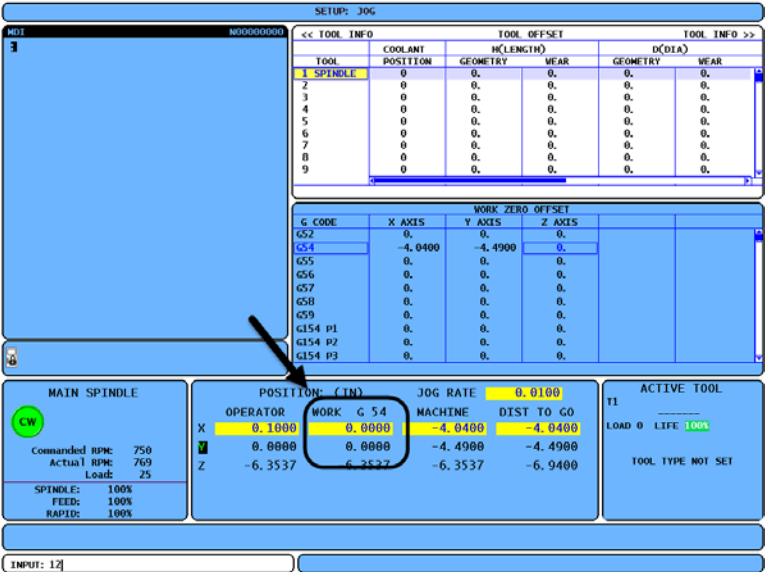


- The Edge Finder should be positioned at the back left corner of the workpiece above the workpiece.

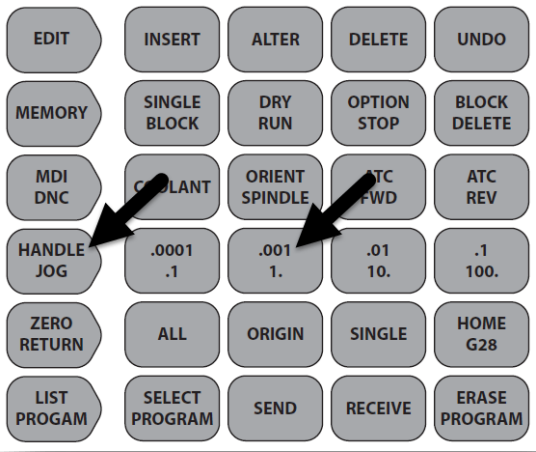


Setup & Operate Training Guide

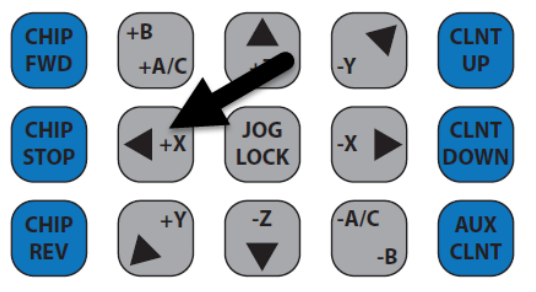
1. Note in the **POSITION PANEL** under the **WORK COLUMN** both the **X** and **Y** settings should read **0.0000**.



2. Press the **Handle Jog** button and set to .001/1. Speed.



3. Select the **X Axis** button.



4. Move the **Edge Finder** in **X + 1.5"** (half the part width) while watching the **WORK COLUMN G54 X** coordinate until it reads **1.500"**.

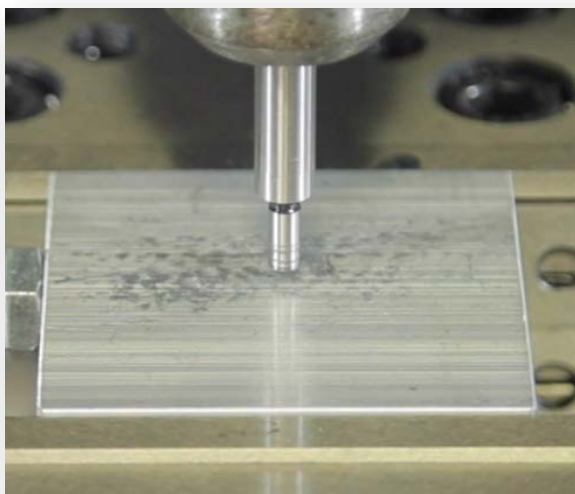
	POSITION: (IN)		JOG RATE	0.0100
	OPERATOR	WORK G 54	MACHINE	DIST TO GO
X	1.6000	1.5000	-2.5400	-2.5400
Y	0.0000	0.0000	-4.4900	-4.4900
Z	-6.3537	-6.3537	-6.3537	-6.9400

5. Select the **Y Axis** button.
6. Move the **Edge Finder** in **Y +1.5"** while watching the **WORK COLUMN G54 Y** coordinate until it reads **-1.500"**.

	POSITION: (IN)		JOG RATE	0.0100
	OPERATOR	WORK G 54	MACHINE	DIST TO GO
X	1.6000	1.5000	-2.5400	-2.5400
Y	-1.5000	-1.5000	-5.9900	-5.9900
Z	-6.3537	-6.3537	-6.3537	-6.9400

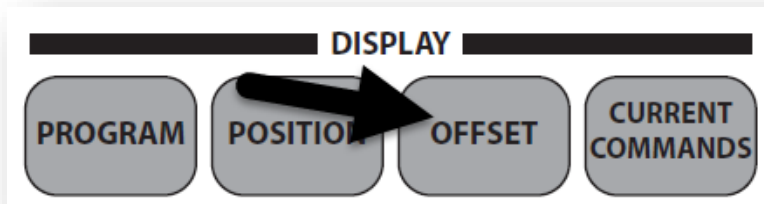
7. The **Edge Finder** should be at the center of the part in the X and Y axis. This is the Work Offset for this part.

NOTE:
If the Edge Finder is NOT at the middle of the workpiece check with your instructor/supervisor before proceeding.

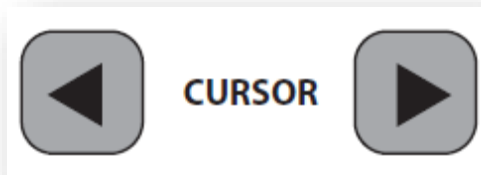


Setup & Operate Training Guide

8. Press **OFFSET** to highlight the **WORK ZERO OFFSET** screen.



9. Press the left or right **Cursor** button to highlight the **X Column** in the **G54** Section.



WORK ZERO OFFSET			
G CODE	X AXIS	Y AXIS	Z AXIS
G52	0.	0.	0.
G54	-4.0400	-4.4900	0.
G55	0.	0.	0.
G56	0.	0.	0.
G57	0.	0.	0.
G58	0.	0.	0.
G59	0.	0.	0.
G154 P1	0.	0.	0.
G154 P2	0.	0.	0.
G154 P3	0.	0.	0.

ENTER A VALUE

10. Press the **PART ZERO SET** button to load the value in the **X-Axis**.
11. Press **PART ZERO SET** again to load the value into the **Y-Axis**.

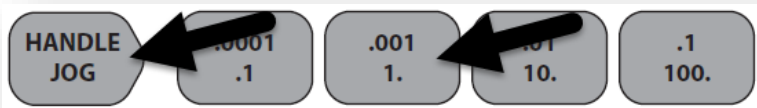


WORK ZERO OFFSET			
G CODE	X AXIS	Y AXIS	Z AXIS
G53	0.	0.	0.
G54	-2.5400	-5.9900	0.
G55	0.	0.	0.
G56	0.	0.	0.
G57	0.	0.	0.
G58	0.	0.	0.
G59	0.	0.	0.
G154 P1	0.	0.	0.
G154 P2	0.	0.	0.
G154 P3	0.	0.	0.

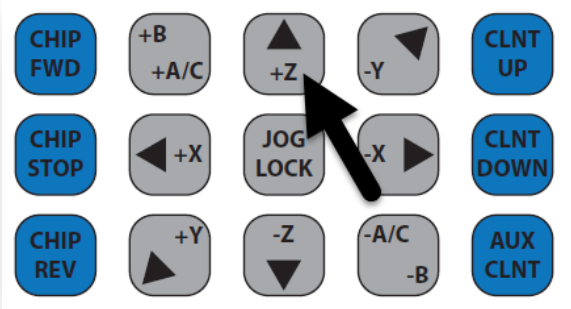
ENTER A VALUE

12. NOTE: The **Z Axis** setting should be **0.000**. If it is not then highlight the Z axis (use the cursors if necessary) then Press the 0.0 Keys on the numbers pad and Press F2 and **Y** or **ENTER** to accept the warning.

13. Press the **Handle Jog** button and set to **.01/10**.

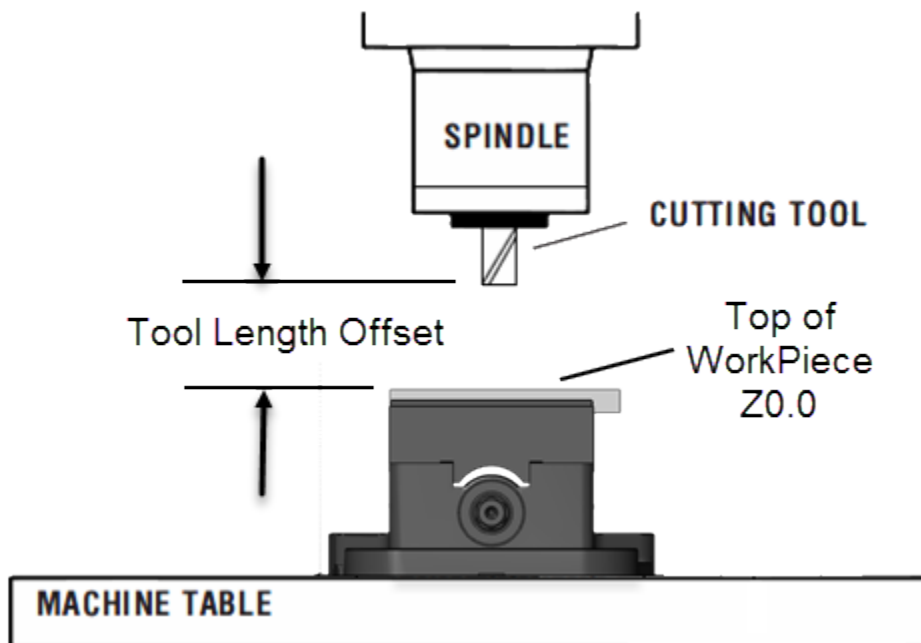


14. Select the **Z Axis** button and **Jog** the Spindle up and away from the part.

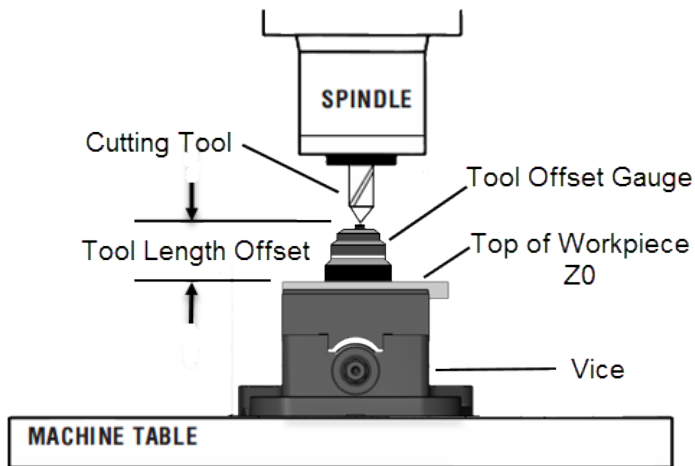


TASK 5: SETTING THE TOOL LENGTH OFFSETS

- Task 5 explains how to set the Tool Offsets which defines the distance from the tip of the tool to the top of the part. Another name for this is Tool Length Offset, which is designated as H in a line of machine code. The distance for each tool is entered into the Tool Offset Table.
- With the Z Axis at its home position, Tool Length Offset is the distance from the tip of the Cutting Tool to Z 0.0 which for this Lesson is the top of the Workpiece.

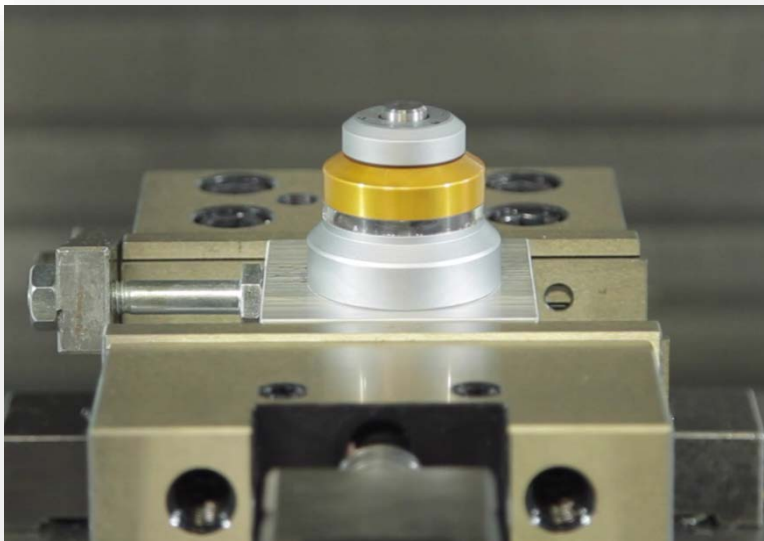


- For this lesson, we will be using a Tool Offset Gauge which provides a more accurate way of setting the Tool Offset.



TOOL 6 - SPOT DRILL

1. Put the **Tool Height Gauge** on top of the material.

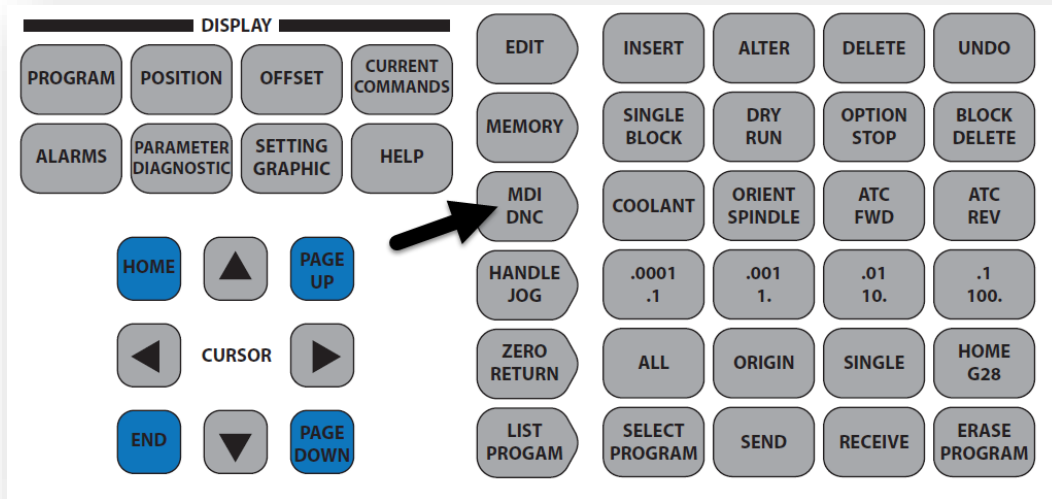


2. Close the **Machine Doors**.

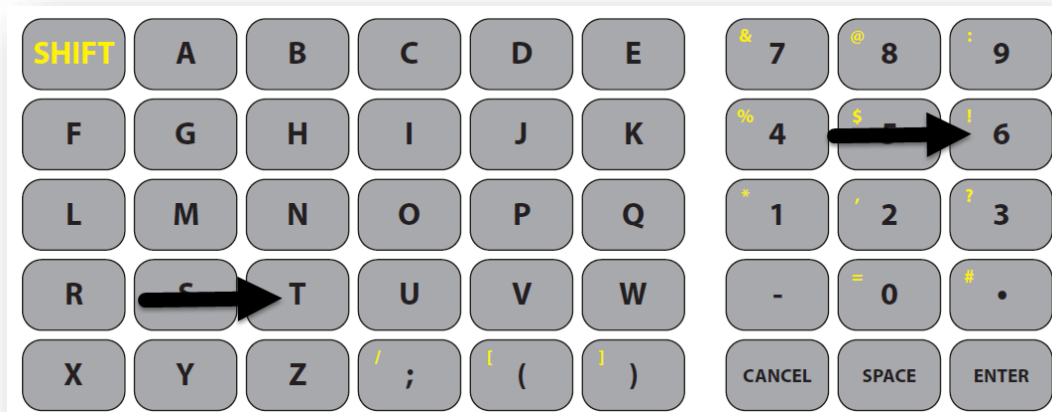
Setup & Operate Training Guide

➤ Set Tool Length Offset for Tool 6.

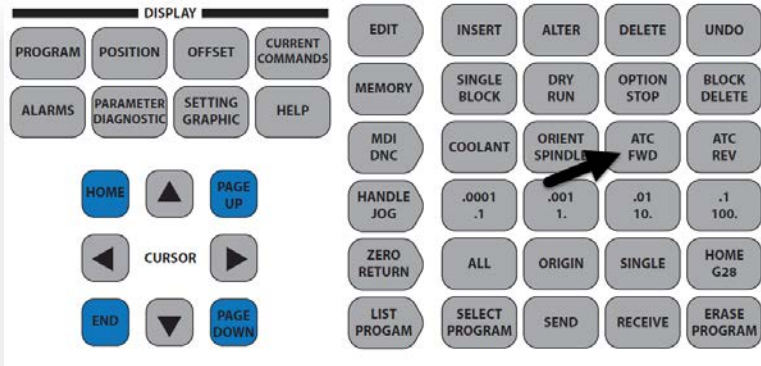
3. Press the **MDI** button.



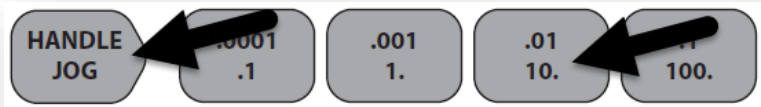
4. Press the letter **T** then the number **6**.



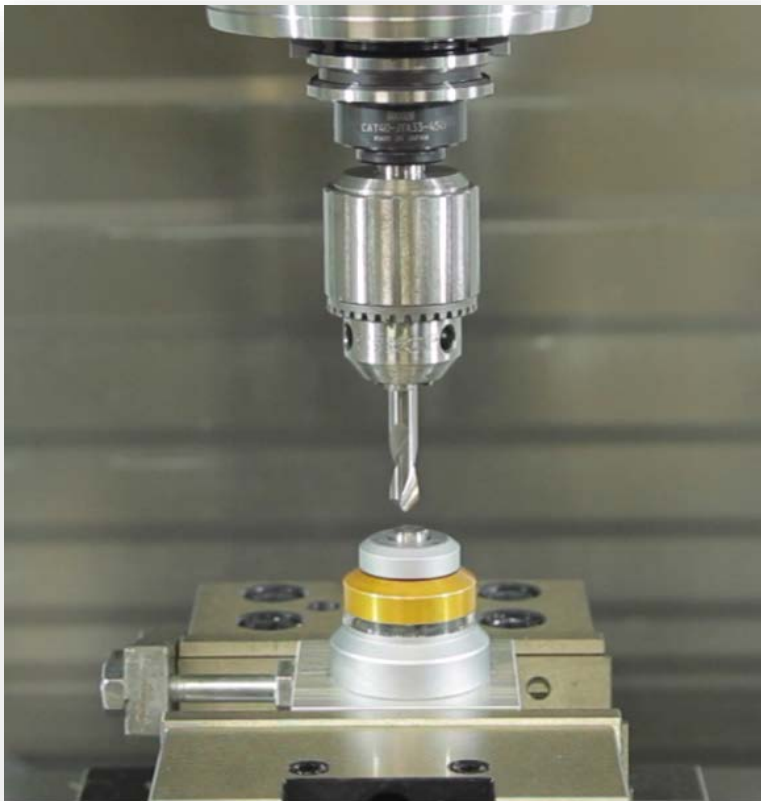
5. Press the **ATC FWD** button.



6. Open the Machine Doors.
7. Press the **HANDLE JOG** button and the **.01/10.** button.



8. Select between the **X** and **Y** and **Z** Axis and jog the tool near the center of the Tool Setter.

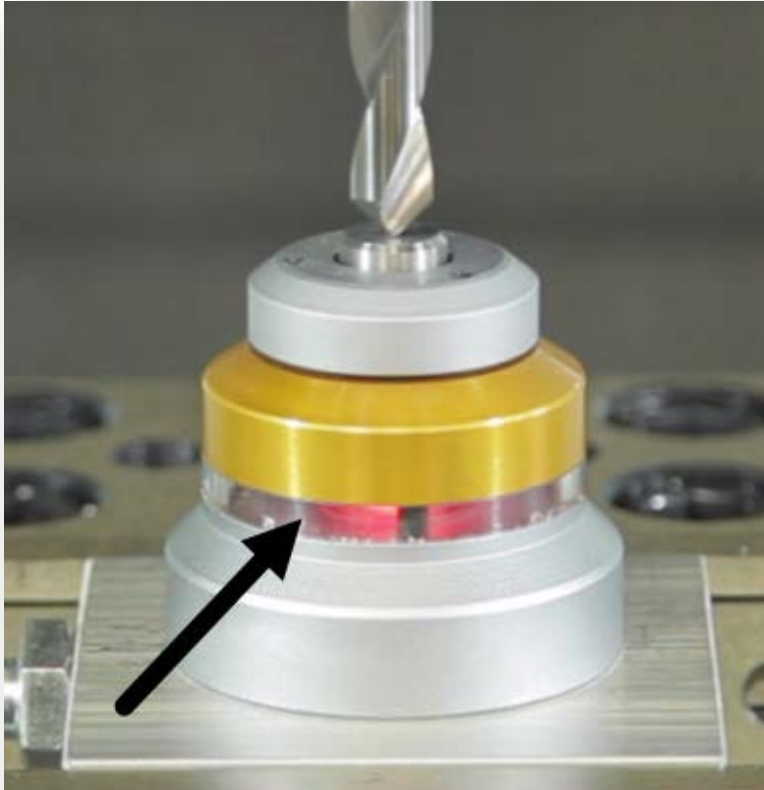


Setup & Operate Training Guide

9. Press the **.001/1.** button. (The mill moves at a slow rate when the handle is turned).



10. Jog the **Z Axis** down until the tip of the tool touches the gauge. A light will appear. The tool is now 2" exactly above the Part.



11. Press the **OFFSET** button until the **Tool Offset Page** is highlighted.

The screenshot shows the 'TOOL OFFSET' page with the following data:

TOOL	COOR. INT.	TOOL POSITION	GEOM. LIST	GEOM. NO.	GEOM. TYPE	GEOM. DATA
1	0	0	0	0	0	0
2	0	0	0	0	0	0
3	0	0	0	0	0	0
4	0	0	0	0	0	0
5	0	0	0	0	0	0
6	0	0	0	0	0	0
7	0	0	0	0	0	0
8	0	0	0	0	0	0
9	0	0	0	0	0	0

G CODE	X AXIS	Y AXIS	Z AXIS
G02	0	0	0
G14	-2.5400	-5.9900	0
G05	0	0	0
G06	0	0	0
G07	0	0	0
G08	0	0	0
G09	0	0	0
G154 P1	0	0	0
G154 P2	0	0	0
G154 P3	0	0	0

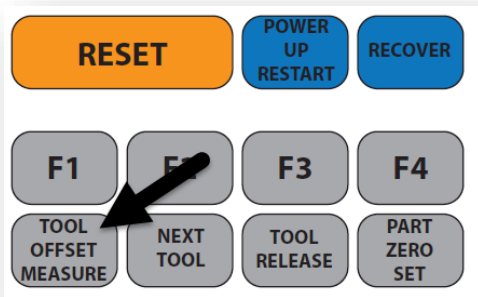
MAIN SPINDLE	POSITION (IN)	JOG RATE	ACTIVE TOOL
STOP	OPERATOR: G. 54 MACHINER	0.0010	LOAD 0 LIFE 100%
Commanded rpm: 750	X 1.6000 0.0000 -2.5400 0.0000	DIST TO GO	TOOL TYPE MET MET
Actual RPM: 0	Y -1.5000 0.0000 -5.9900 0.0000		
Lead: 0	Z -11.3770 -11.3770 -11.3770 -11.3770		
SPINDLE: 100%			
FEED: 100%			
RPM: 100%			

12. Look at the **Controller Screen** to make sure the **GEOMETRY COLUMN** under **TOOL OFFSET H (LENGTH)** is highlighted. If it is not, press the left or right cursor keys until it is.

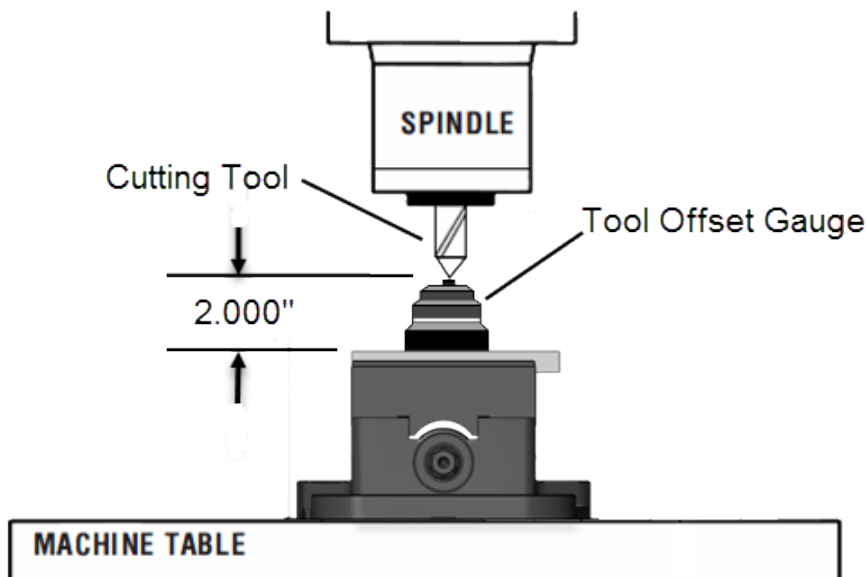
<< TOOL INFO		TOOL OFFSET				TOOL INFO >>	
TOOL	COOLANT POSITION	H(LENGTH)		D(DIA)			
		GEOMETRY	WEAR	GEOMETRY	WEAR		
1	0	0.	0.	0.	0.		
2	0	0.	0.	0.	0.		
3	0	0.	0.	0.	0.		
4	0	0.	0.	0.	0.		
5	0	0.	0.	0.	0.		
6 SPINDLE	0	0.	0.	0.	0.		
7	0	0.	0.	0.	0.		
8	0	0.	0.	0.	0.		
9	0	0.	0.	0.	0.		

ENTER A VALUE

13. Check to see that **Tool 6** is highlighted.
 14. Press the **TOOL OFFSET MEASURE** button.

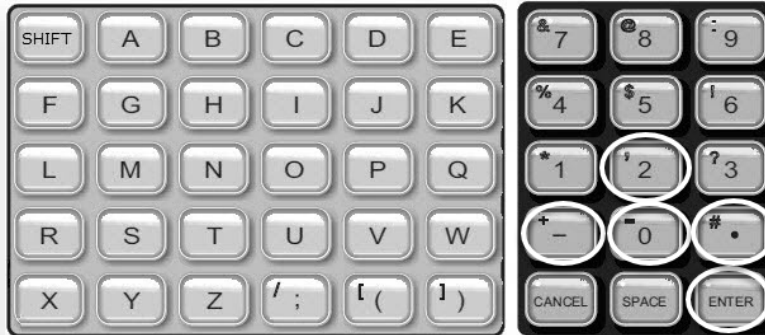


- ☞ Since the Tool Offset Gauge is exactly 2.000" in height we need to adjust the setting in the control accordingly. In this case add the extra 2 inches to the settings in the control.

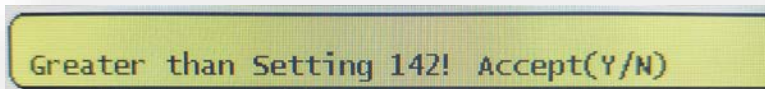


Setup & Operate Training Guide

15. We now need to subtract the 2.0" for the Tool Setter. Input **-2.0** and hit **Enter**.



16. A Warning on the Control Screen may appear. Press **Y** on the Keyboard to accept.



17. Note the new **Offset** for **Tool 6** is 2.000" more. Our example, it went from -11.3770 to -13.3770.

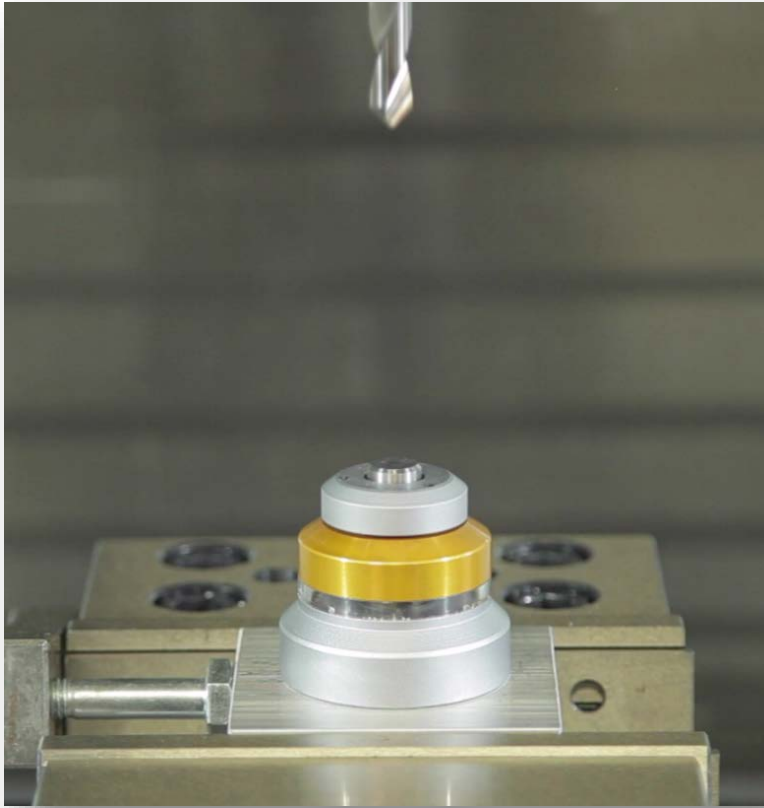
<< TOOL INFO		TOOL OFFSET			TOOL INFO >>	
TOOL	COOLANT POSITION	H(LENGTH)		D(DIA)		
		GEOMETRY	WEAR	GEOMETRY	WEAR	
1	0	0.	0.	0.	0.	
2	0	0.	0.	0.	0.	
3	0	0.	0.	0.	0.	
4	0	0.	0.	0.	0.	
5	0	0.	0.	0.	0.	
6 SPINDLE	0	-13.3770	0.	0.	0.	
7	0	0.	0.	0.	0.	
8	0	0.	0.	0.	0.	
9	0	0.	0.	0.	0.	

ENTER A VALUE

18. Press the **Handle Jog** button and set speed to **.001/1**.



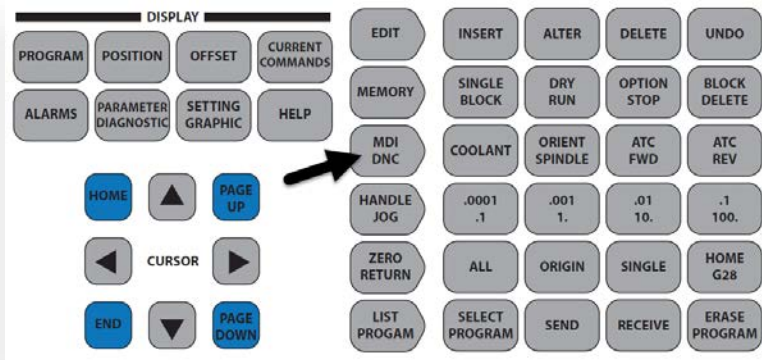
19. Jog +Z up and away from the Tool Setter.



WARNING: The next step causes the spindle to move rapidly in the Z Axis.

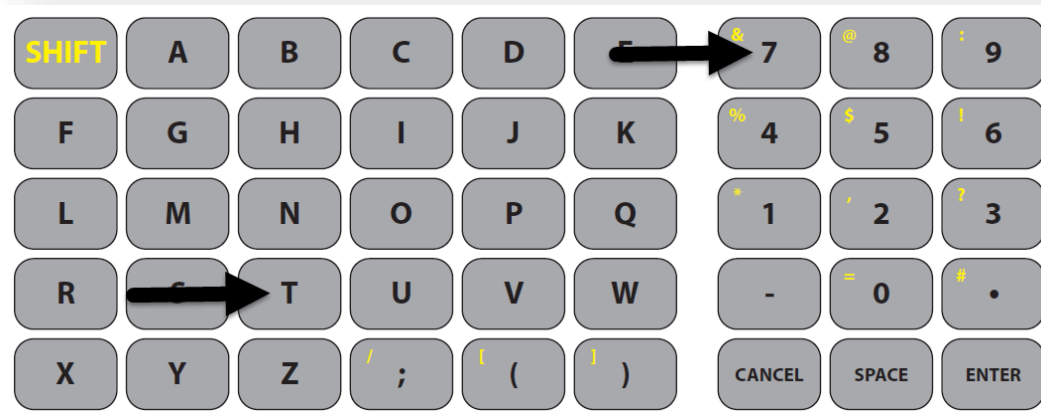
TOOL 7 - 1/4" DRILL

1. Close the **Machine Doors**.
2. Press the **MDI** button.

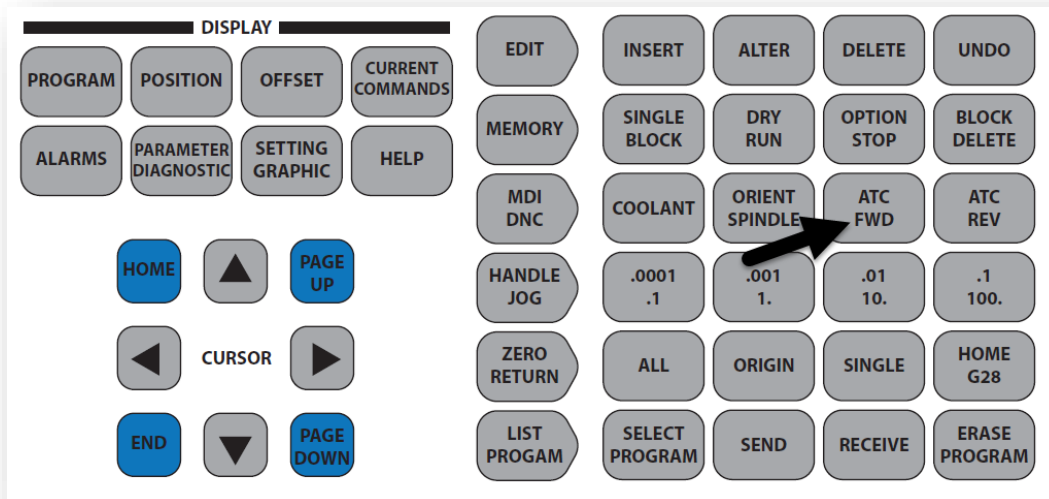


Setup & Operate Training Guide

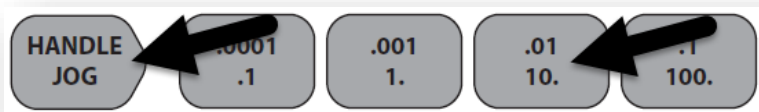
- Press the letter **T** then the number **7**.



- Press the **ATC FWD** button.

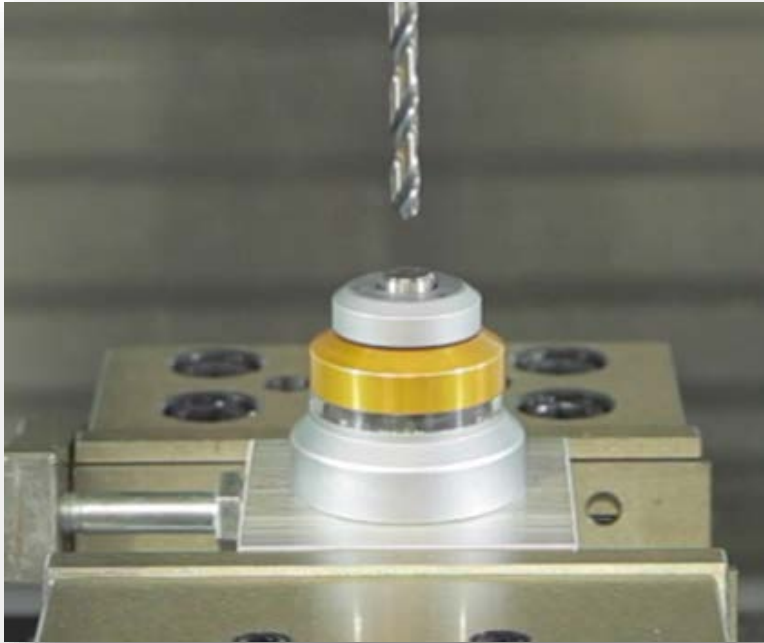


- Press the **HANDLE JOG** button and press the **.01/10.** Button (The mill moves at a fast rate when the handle is turned).

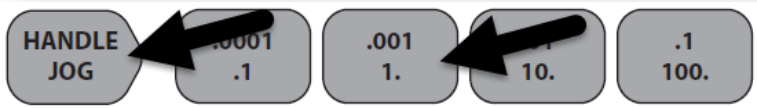


- Open the **Machine Doors**.

7. Select between the **X** and **Y** and **Z** Axis and jog the tool near the center of the Tool Setter.

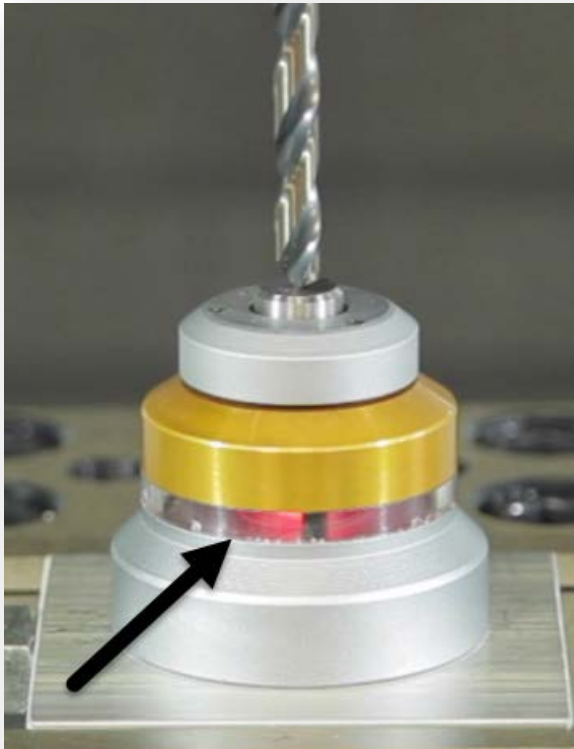


8. Press the **.001/.1** button.

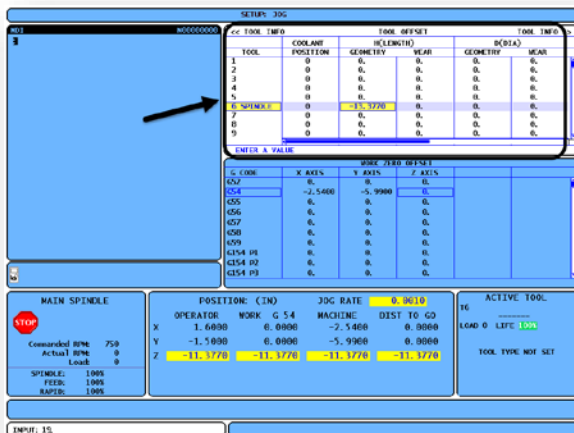
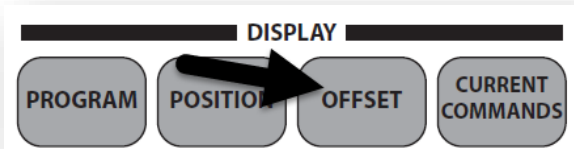


Setup & Operate Training Guide

9. **Jog** the **Z** axis down until the tip of the tool touches the gauge. A light will appear. The tool is now 2" exactly above the Part.



10. Press the **OFFSET** button until the **TOOL OFFSET PAGE** is highlighted.

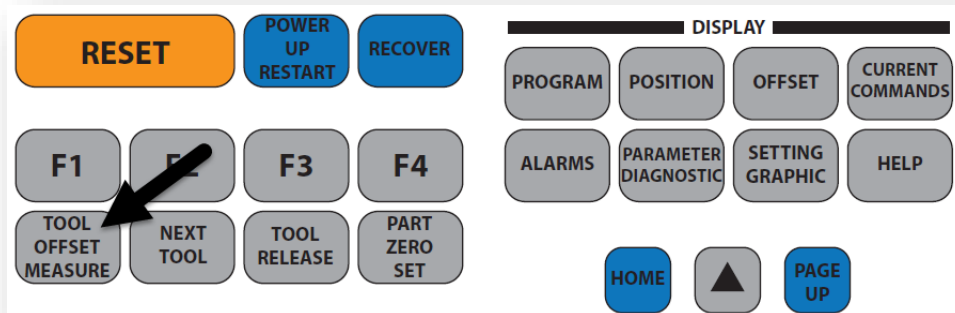


11. Make sure **Tool 7** is highlighted in the **GEOMETRY LENGTH** column.

<< TOOL INFO		TOOL OFFSET				TOOL INFO >>	
TOOL	COOLANT POSITION	H(LENGTH)		D(DIA)			
		GEOMETRY	WEAR	GEOMETRY	WEAR		
1	0	0.	0.	0.	0.		
2	0	0.	0.	0.	0.		
3	0	0.	0.	0.	0.		
4	0	0.	0.	0.	0.		
5	0	0.	0.	0.	0.		
6	0	-13.3770	0.	0.	0.		
7 SPINDLE	0	0.	0.	0.	0.		
8	0	0.	0.	0.	0.		
9	0	0.	0.	0.	0.		

ENTER A VALUE

12. Press the **TOOL OFFSET MEASURE** button.



13. We now need to subtract the 2.0" for the **Tool Setter**. Input **-2.0** and hit **Enter**.

14. Accept the warning by hitting **Y**. Note the new **Offset** for **Tool 7** is 2.000" more. In our example, it went from **-9.9280** to **-11.9280**

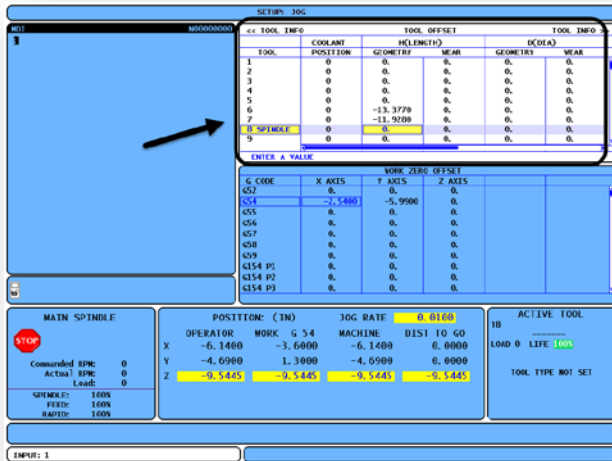
<< TOOL INFO		TOOL OFFSET				TOOL INFO >>	
TOOL	COOLANT POSITION	H(LENGTH)		D(DIA)			
		GEOMETRY	WEAR	GEOMETRY	WEAR		
1	0	0.	0.	0.	0.		
2	0	0.	0.	0.	0.		
3	0	0.	0.	0.	0.		
4	0	0.	0.	0.	0.		
5	0	0.	0.	0.	0.		
6	0	-13.3770	0.	0.	0.		
7 SPINDLE	0	-11.9280	0.	0.	0.		
8	0	0.	0.	0.	0.		
9	0	0.	0.	0.	0.		

ENTER A VALUE

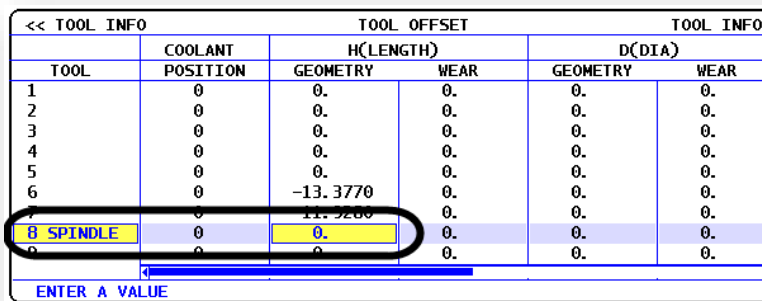
15. Press the **Handle Jog** button and Jog **Z** away from the Tool Setter.

WARNING: The next step causes the spindle to move rapidly in the Z Axis.

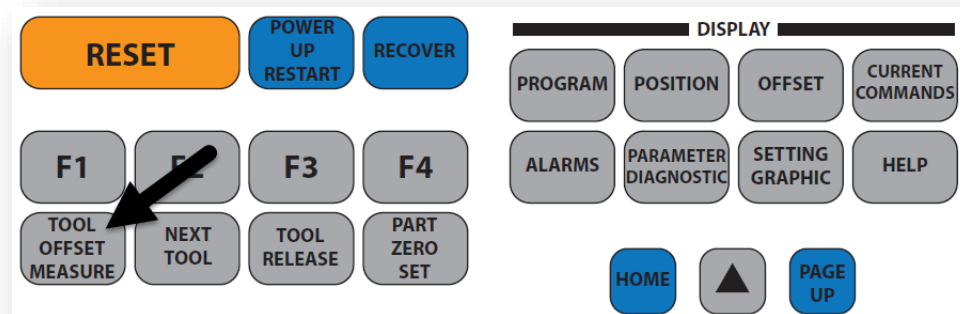
- Press the **OFFSET** button until the **TOOL OFFSET PAGE** is highlighted.



- Make sure **Tool 8** is highlighted in the **GEOMETRY LENGTH** column.



- Press the **TOOL OFFSET MEASURE** button.



Setup & Operate Training Guide

10. We now need to subtract the 2.0" for the **Tool Setter**. Input **-2.0** and hit **Enter**.
11. Accept the warning by hitting **Y**. Note the new **Offset** for **Tool 8** is 2.000" more. In our example, it went from -9.5445 to -11.5445

<< TOOL INFO		TOOL OFFSET		TOOL INFO >	
TOOL	COOLANT	H(LENGTH)		D(DIA)	
	POSITION	GEOMETRY	WEAR	GEOMETRY	WEAR
1	0	0.	0.	0.	0.
2	0	0.	0.	0.	0.
3	0	0.	0.	0.	0.
4	0	0.	0.	0.	0.
5	0	0.	0.	0.	0.
6	0	-13.3770	0.	0.	0.
7	0	-11.9280	0.	0.	0.
8 SPINDLE	0	-11.5445	0.	0.	0.
9	0	0.	0.	0.	0.

ENTER A VALUE

12. Press the **Handle Jog** button and Jog **Z** away from the Tool Setter.
13. Remove the **Tool Offset Gauge** from the workpiece.

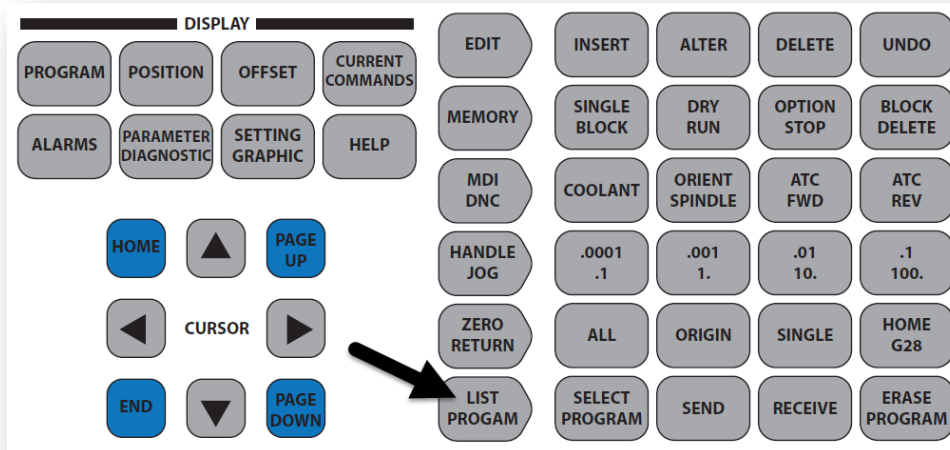
TASK 6: LOAD, CHECK AND RUN THE PROGRAM

☞ In this task, we will copy the NC program from the computer to the Haas Controller using a USB Memory stick.

1. Take the USB memory stick with the NC Program saved on it and insert it into the USB slot at the side of the controller.



2. Press the **LIST PROGRAM** button.

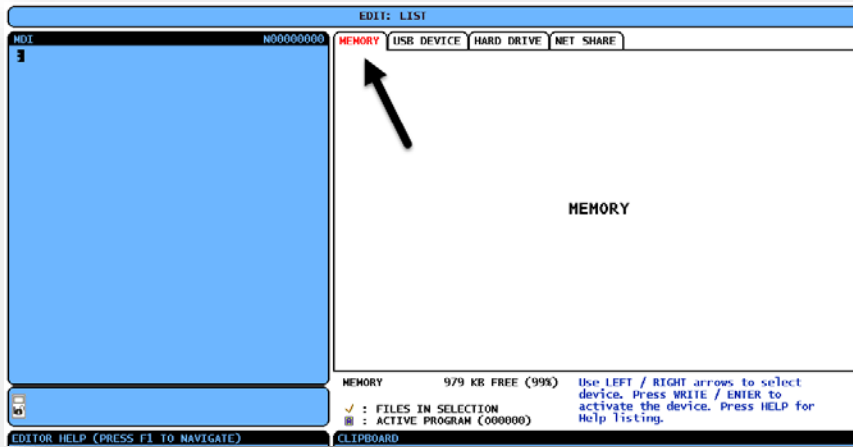



3. Use the **Up-Cursor Key** to highlight the **Tabs** at the top of the screen.

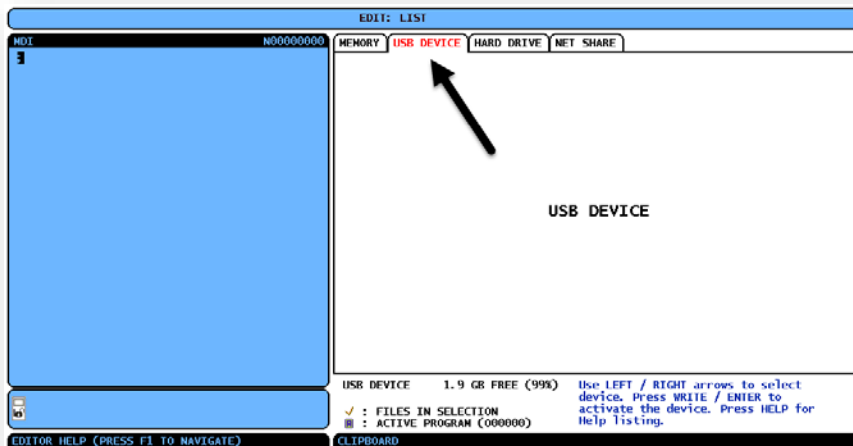



Setup & Operate Training Guide

4. The **Memory Tab** is now Red.



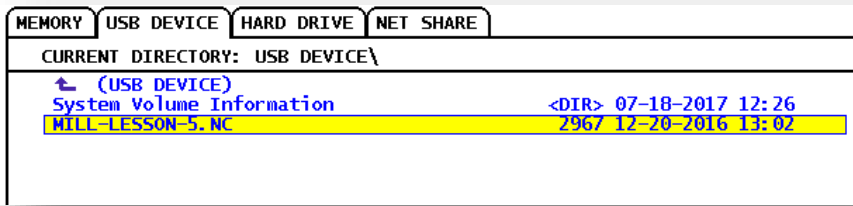
5. Use the **Right Cursor Key**  to open the **USB Device Tab** (window).



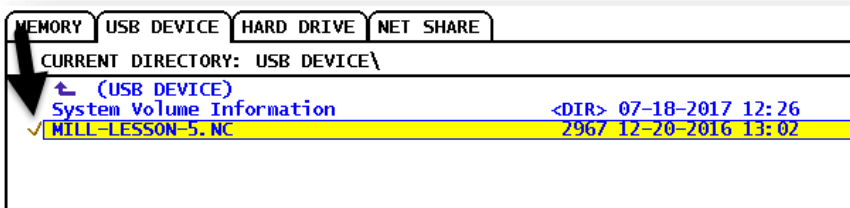
6. Press the **Down**  or **Enter** Key. The programs on the USB Stick will be displayed on the screen.



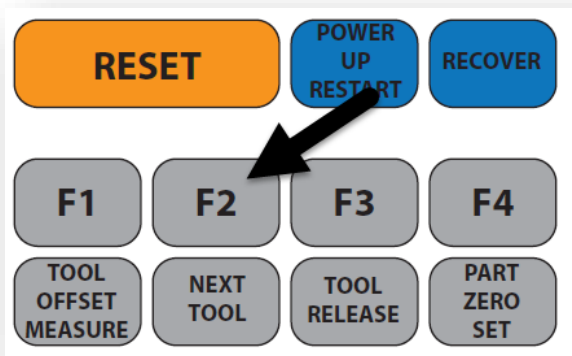
7. Press the **Down Cursor Key**  until **MILL-LESSON-5.NC** Program is highlighted.



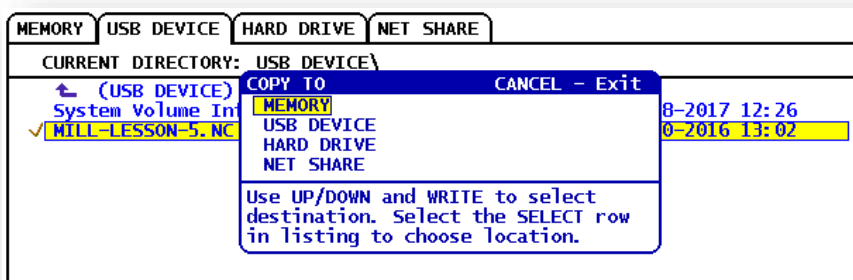
8. Press the **Enter Key** to select **MILL-LESSON-5.NC** Program. A checkmark will now appear beside the program.



9. Press the **F2** button.

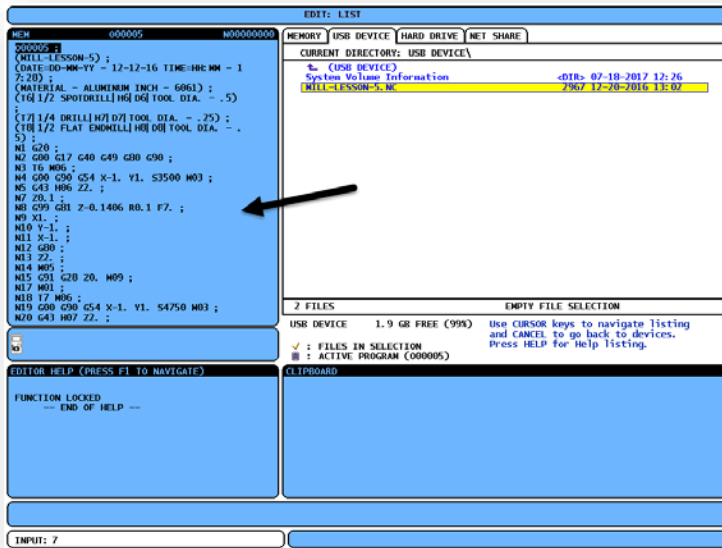


10. The **COPY TO** screen appears with **MEMORY** highlighted.



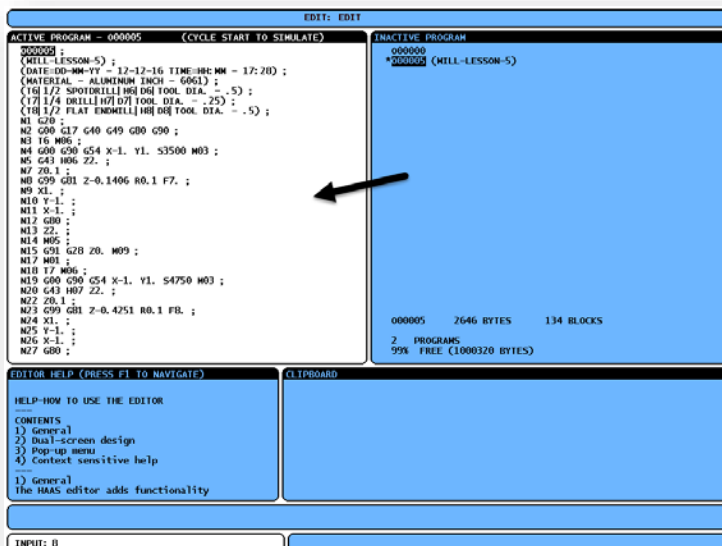
Setup & Operate Training Guide

11. Press the **Enter** button to copy the program from the **USB** to the **Machine Memory**. Note the program is now loaded in the **EDIT PANEL**.

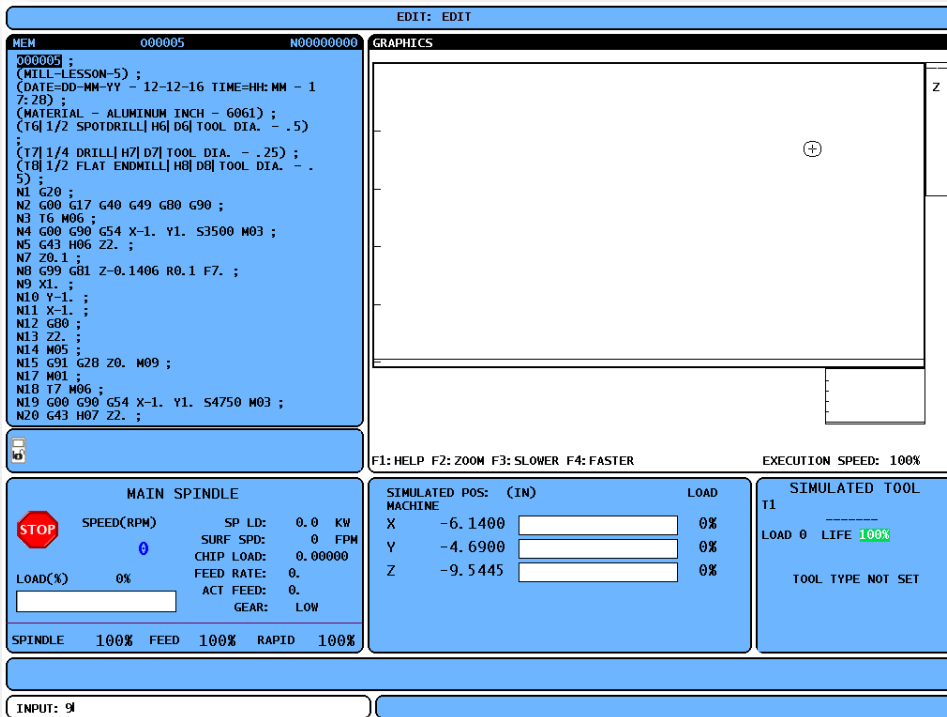
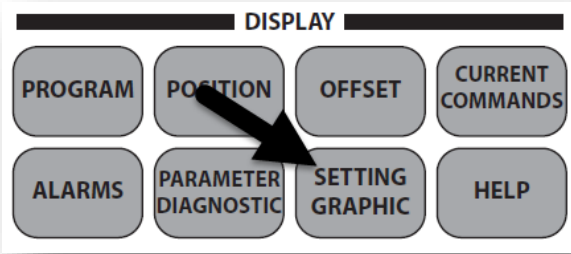


➡ The program is loaded into the memory of the machine. We can now **Verify** the Program using the graphics screen on the Haas Controller.

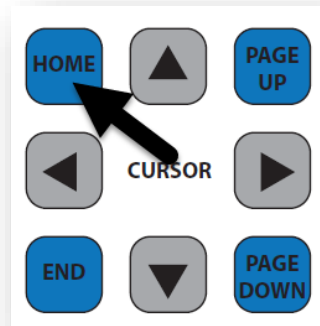
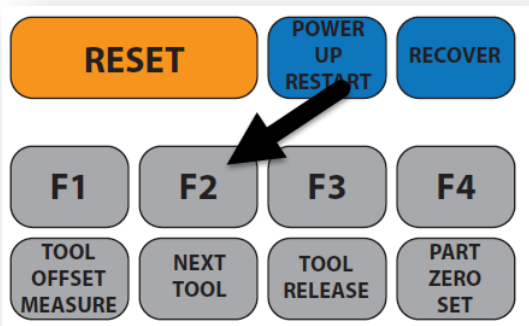
12. Press the **Edit** button.



13. Press the **Setting Graphic** Button.



14. Press the **F2** button and **Home** button to ensure you are at the **Default** view.



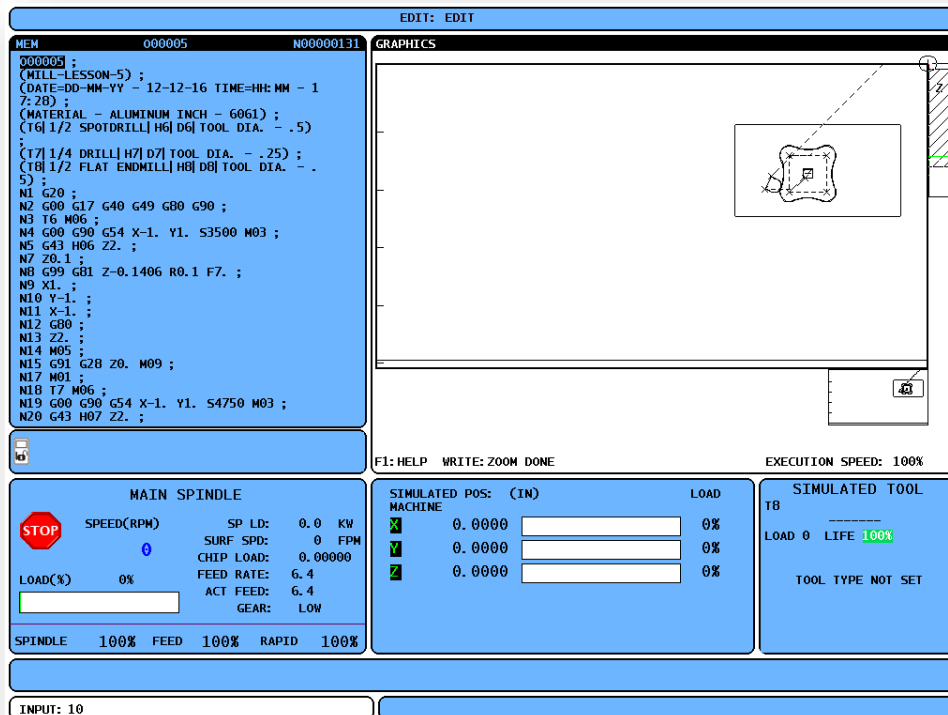
Setup & Operate Training Guide

15. Select **Cycle Start** to start simulation.



16. The simulation may be small. To Zoom and adjust the simulation area push the **F2** button.

17. Use the **Page Up** key to zoom out, **Page Down** key to zoom in to view the complete toolpath.



18. Use the **Cursor Keys** to move the window left and right to capture the full Toolpath.

19. Press the **Enter** key when you are satisfied with the display.

The screenshot shows the CNC control interface with the following details:

- EDIT: EDIT** header at the top.
- MEM** window on the left containing G-code:


```

000005 ;
(MILL-LESSON-5) ;
( DATE=DD-MM-YY - 12-12-16 TIME=HH:MM - 1
7:20) ;
(MATERIAL - ALUMINUM INCH - 6061) ;
(T6) 1/2 SPOTDRILL| H6| D6| TOOL DIA. - .5)
;
(T7) 1/4 DRILL| H7| D7| TOOL DIA. - .25) ;
(T8) 1/2 FLAT ENDMILL| H8| D8| TOOL DIA. - .
5) ;
N1 G20 ;
N2 G00 G17 G40 G49 G80 G90 ;
N3 T6 M06 ;
N4 G00 G90 G54 X-1. Y1. S3500 M03 ;
N5 G43 H06 Z2. ;
N7 Z0.1 ;
N8 G99 G81 Z-0.1406 R0.1 F7. ;
N9 X1. ;
N10 Y-1. ;
N11 X-1. ;
N12 G80 ;
N13 Z2. ;
N14 M05 ;
N15 G91 G28 Z0. M09 ;
N17 M01 ;
N18 T7 M06 ;
N19 G00 G90 G54 X-1. Y1. S4750 M03 ;
N20 G43 H07 Z2. ;
      
```
- GRAPHICS** window on the right showing a blank coordinate system with Z-axis.
- EXECUTION SPEED: 100%** indicator.
- MAIN SPINDLE** section:
 - STOP button
 - SPEED(RPM): 0
 - SP LD: 0.0 KW
 - SURF SPD: 0 FPM
 - CHIP LOAD: 0.00000
 - FEE RATE: 6.4
 - ACT FEED: 6.4
 - GEAR: LOW
 - LOAD(%): 0%
 - SPINDLE 100% FEED 100% RAPID 100%
- SIMULATED POS: (IN) MACHINE** table:

MACHINE	POS	LOAD
X	0.0000	0%
Y	0.0000	0%
Z	0.0000	0%
- SIMULATED TOOL** section:
 - T8
 - LOAD 0 LIFE 100%
 - TOOL TYPE NOT SET
- INPUT: 11** at the bottom.

20. Slow the **Backplot** by pressing the F3 key until the setting is set to 30%.

The screenshot shows the CNC control interface with the following details:

- MEM** window on the left showing:


```

- .375) ;
DIA. - .5)
DIA. - .375)
. S1500 M03
      
```
- GRAPHICS** window on the right showing a blank coordinate system with Z-axis.
- EXECUTION SPEED: 30%** indicator.
- MAIN SPINDLE** section:
 - SP LD: 0.0 KW
 - SPD: 0 FPM
 - CHP LD: 0.00000
 - FE: 8.
 - ED: 8.
 - AR: LOW
 - RAPID 100%
- SIMULATED POS: (IN) MACHINE** table:

MACHINE	POS	LOAD
X	0.0000	0%
Y	0.0000	0%
Z	0.0000	0%
- SIMULATED TOOL** section:
 - T9
 - LOAD 0 LIFE 100%
 - TOOL TYPE NOT SET
- INPUT:** (empty)

Setup & Operate Training Guide

21. Press the **Cycle Start** button to review the toolpaths.

The screenshot displays a CNC control interface with the following sections:

- EDIT: EDIT** (Title bar)
- MEM** (Memory): 000005, N000001.31
- GRAPHICS**: A window showing a 2D toolpath for a part with a square hole and rounded corners. A tool icon is visible in the bottom right corner of the graphics area.
- EXECUTION SPEED: 30%**
- MAIN SPINDLE**:
 - STOP button
 - SPEED(RPM): 0
 - SP LD: 0.0 KW
 - SURF SPD: 0 FPM
 - CHIP LOAD: 0.00000
 - FEED RATE: 6.4
 - ACT FEED: 6.4
 - GEAR: LOW
 - LOAD(%): 0%
- SIMULATED POS: (IN) MACHINE**:
 - X: 0.0000, LOAD: 0%
 - Y: 0.0000, LOAD: 0%
 - Z: 0.0000, LOAD: 0%
- SIMULATED TOOL**:
 - TB
 - LOAD 0 LIFE 100%
 - TOOL TYPE NOT SET
- SPINDLE 100% FEED 100% RAPID 100%**
- INPUT: 13**

WARNING:
Ask your instructor/supervisor to check the backplot of your program. Once you and the instructor/supervisor are satisfied the program is ready to run proceed to the next Step.

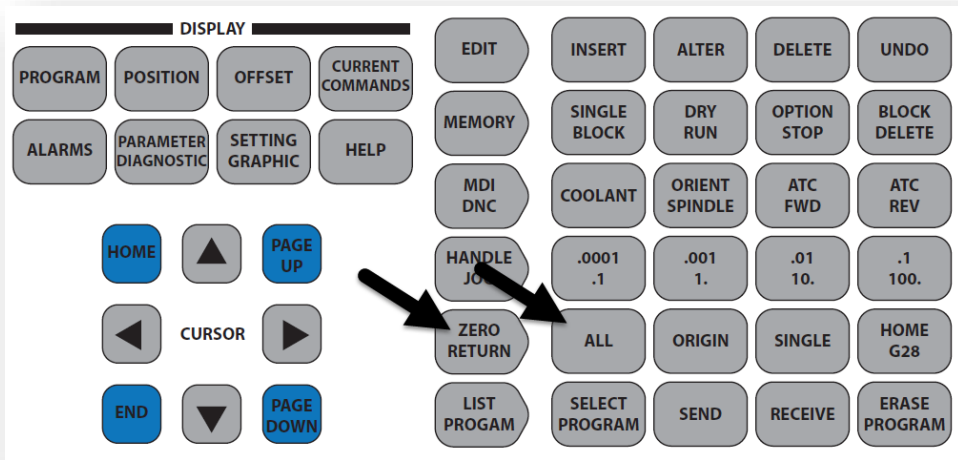
RUNNING THE PROGRAM

☞ NOTE: Watch the video: “Running the Program” before proceeding.

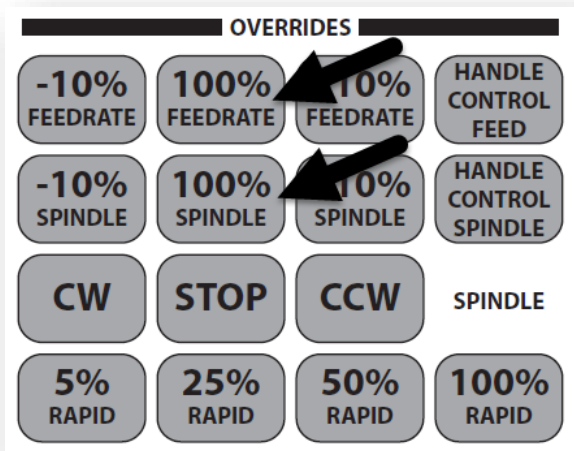
WARNING:
Ask your instructor/supervisor to check the backplot of your program. Once you and the instructor/supervisor are satisfied the program is ready to run proceed to the next Step.

☞ Note, we recommend that you watch the Running the Program video before proceeding.

1. Send the Machine to the Home Position by pressing the **ZERO RETURN** button followed by the **ALL BUTTON**.

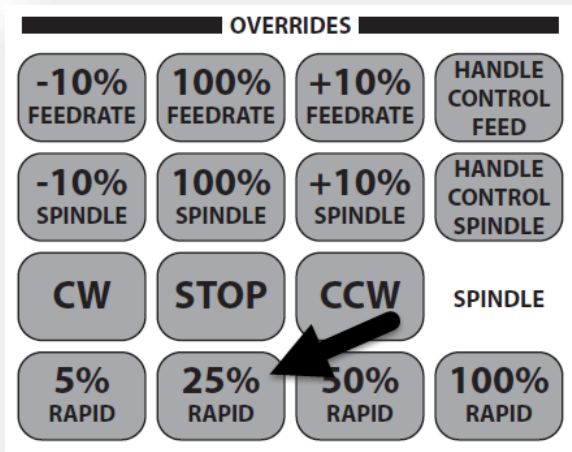


2. Ensure the **Spindle** and **Feeds** are all set to **100%** by pressing the **100% Spindle** button and the **100% Feedrate** button.

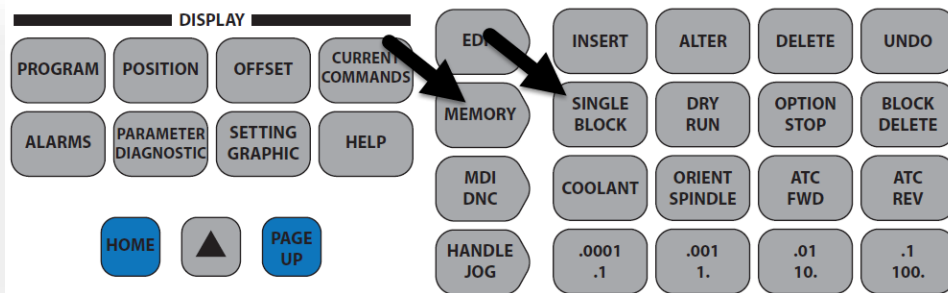


Setup & Operate Training Guide

3. Set **Rapid** override to **25%** by pushing the **25% RAPID** button.



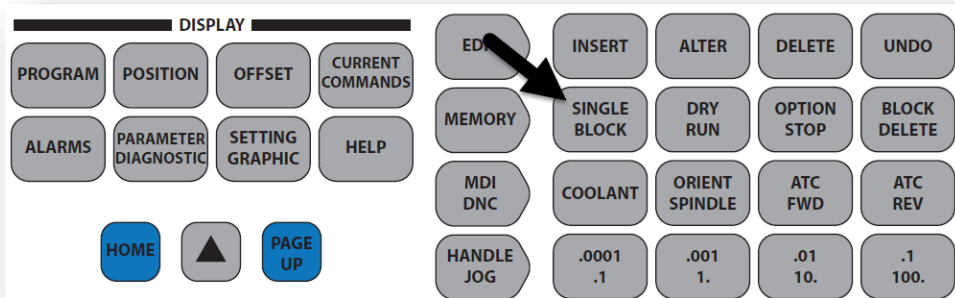
- For this lesson, we will run each program by starting in **SINGLE BLOCK** Mode. This will allow us to stop the machine before any potential problems occur, just in case there is a mistake in the program.
 - Single Block mode performs one line of code and then stops. **Cycle Start** needs to be pushed for the machine to proceed.
4. Press the **MEMORY** button and the **SINGLE BLOCK** Button.



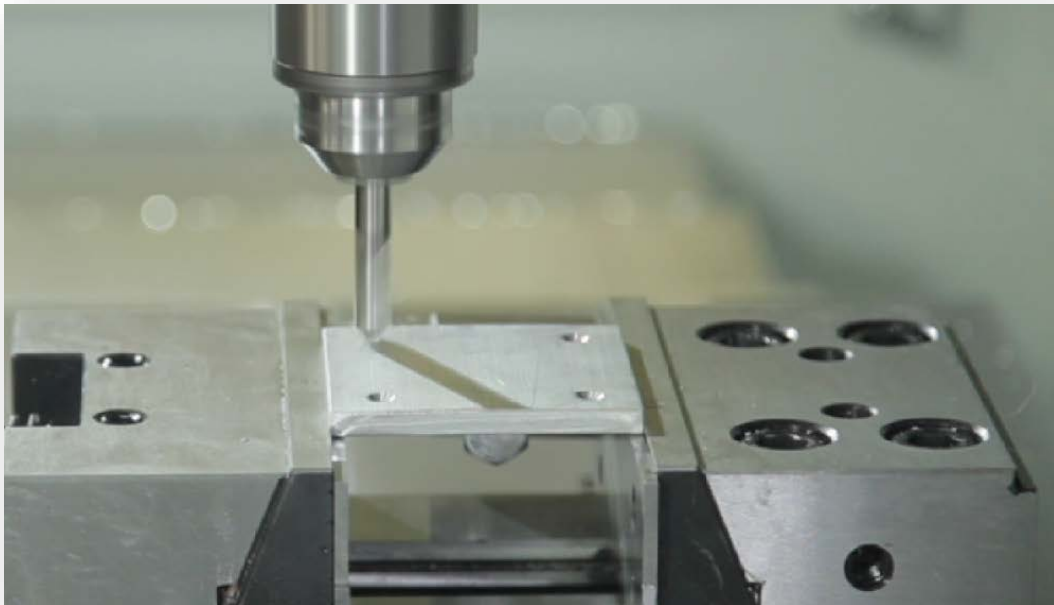
5. Press the **Cycle Start** button repeatedly until the machine starts to move. Hold your finger over the Feed Hold Button so you can press it if something is not correct.



- Once the Spindle turns on and the cutter starts feeding down towards the workpiece, be ready to push the feed hold.
- Pause the program and compare the Distance to Go values to actual distances within the machine
- The **Cutter** should stop before it starts cutting into the workpiece. NOTE: if the cutter doesn't stop before cutting the workpiece push the Feed Hold button and call over your instructor or supervisor.
- Push **Cycle Start** again to resume the program.
- The cutter should start drilling into the workpiece.
- If all looks correct, press the **Single Block** button to turn **Single Block OFF**.



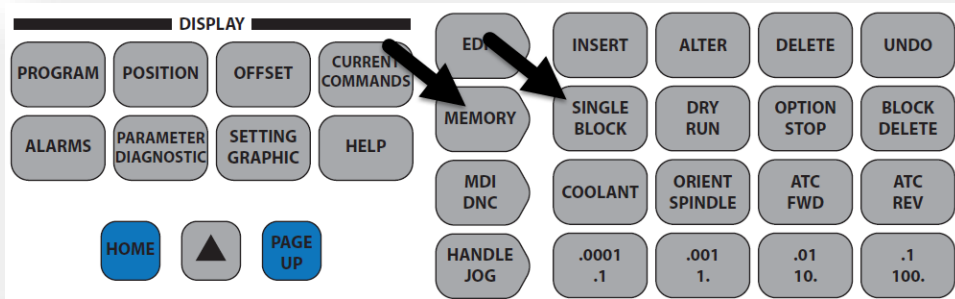
- Press Cycle Start button to **Run** the program to the next tool change.



Setup & Operate Training Guide

➤ Tool 7 Operation.

13. During the **Tool Change**, Press the **MEMORY** button and the **SINGLE BLOCK** Button.

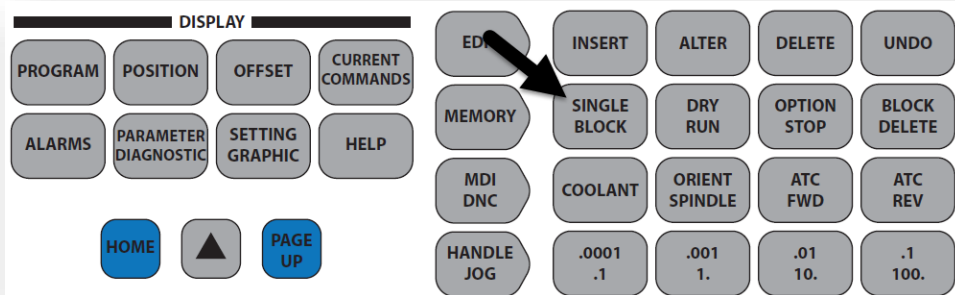


14. Press the **Cycle Start** while having your finger over the Feed Hold Button so you can press it if something is not correct.



15. The **Cutter** should stop before it starts cutting into the workpiece. **NOTE** if the cutter doesn't stop before cutting the workpiece push the Feed Hold button and call over your instructor or supervisor

16. If all looks correct, press the **Single Block** button to turn **Single Block OFF**.



17. Press Cycle Start button to **Run** the program to the next tool.



➤ **Tool 8 Operation.**

18. During the **Tool Change**, Press the **MEMORY** button and the **SINGLE BLOCK** Button.

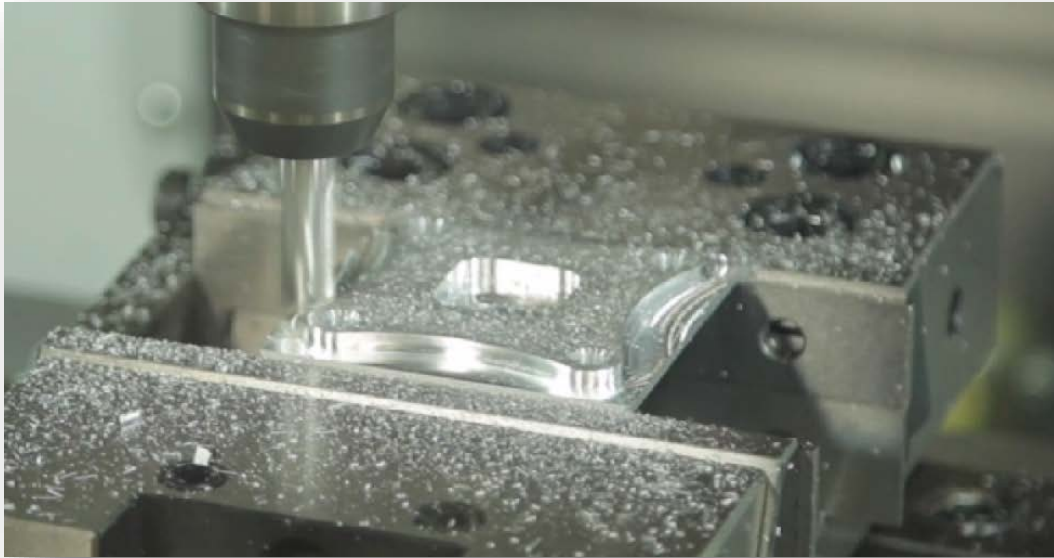
19. Press the **Cycle Start** while having your finger over the Feed Hold Button so you can press it if something is not correct.

20. The **Cutter** should stop before it starts cutting into the workpiece.

21. If all looks correct, press the **Single Block** button to turn **Single Block OFF**.

Setup & Operate Training Guide

22. Press the **Cycle Start** button to **Run** the program to the finish.



23. Before removing the workpiece, make sure the machine has stopped moving and the spindle has stopped turning.

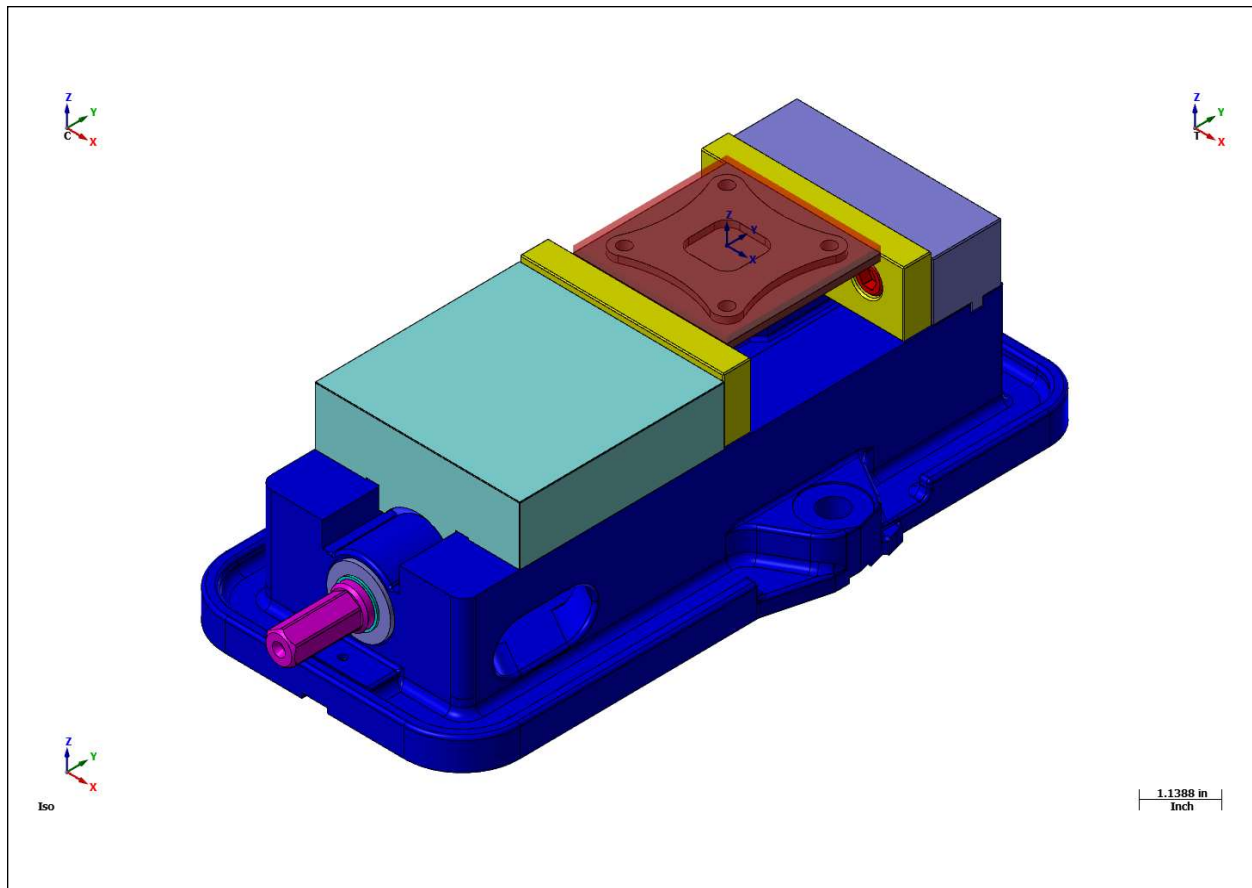
24. Carefully remove the workpiece and clean off the vice and table, leaving the machine ready for the next operator.

GENERIC HAAS 4 - AXIS VMC

GENERAL INFORMATION

PROJECT NAME: VMC Setup Operate Lesson 2
CUSTOMER NAME: CamInstructor
PROGRAMMER: MM
DRAWING: Mastercam Mill Programming Lesson 5

MILL-LESSON-5-VISE-3X3.MCAM



COMMENTS

Part is held in vise.

Locate against end-stop

Locate part on parallels

X0 Y0 at center of part Z0 is at the top

Material 3"x3"x1/4" Aluminum 6061

MILL-LESSON-5.NCI

TOTAL CYCLE TIME: 0 HOURS, 4 MINUTES, 36 SECONDS

OPERATION LIST

OP #	OPERATION NAME	TOOL #	MIN-Z	MAX-Z	CYCLE TIME
1	1 - Drill/Counterbore	1	-0.1406	2.0	00:00:08
2	2 - Drill/Counterbore	2	-0.4251	2.0	00:00:16
3	3 - Pocket (Standard)	3	-0.125	2.0	00:01:28
4	4 - Contour (2D)	3	-0.125	2.0	00:02:42

TOOL LIST

SORTED: ASCENDING

	<p>TYPE: Spot Drill</p> <p>MFG CODE:</p> <p>HOLDER: H4V4A1000</p> <p>NUMBER: 1</p> <p>LENGTH OFFSET: 1</p> <p>DIAMETER OFFSET: 1</p> <p>DIAMETER: 0.5</p> <p>CORNER RADIUS: 0.0</p> <p>TIP ANGLE: 90.0</p> <p>FLUTE LENGTH: 2.0</p> <p>OVERALL LENGTH: 3.0</p> <p># OF FLUTES: 2</p> <p>#1 - 0.5000 SPOT DRILL - 1/2 SPOTDRILL</p>
	<p>TYPE: Drill</p> <p>MFG CODE:</p> <p>HOLDER: H4V4A1000</p> <p>NUMBER: 2</p> <p>LENGTH OFFSET: 2</p> <p>DIAMETER OFFSET: 2</p> <p>DIAMETER: 0.25</p> <p>CORNER RADIUS: 0.0</p> <p>TIP ANGLE: 118.0</p> <p>FLUTE LENGTH: 2.0</p> <p>OVERALL LENGTH: 3.0</p> <p># OF FLUTES: 2</p> <p>#2 - 0.2500 DRILL - 1/4 DRILL</p>
	<p>TYPE: Endmill1 Flat</p> <p>MFG CODE:</p> <p>HOLDER: B2E4-0500</p> <p>NUMBER: 3</p> <p>LENGTH OFFSET: 3</p> <p>DIAMETER OFFSET: 3</p> <p>DIAMETER: 0.5</p> <p>CORNER RADIUS: 0.0</p> <p>TIP ANGLE: NA</p> <p>FLUTE LENGTH: 1.0</p> <p>OVERALL LENGTH: 1.5</p> <p># OF FLUTES: 4</p> <p>#3 - 0.5000 ENDMILL1 FLAT - 1/2 FLAT ENDMILL</p>