SETUP & OPERATE TRAINING GUIDE

CNC-MILL-LESSON-2

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.

April, 2017

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.

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.

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.

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**.

3. 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.

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

5. 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.

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

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

7. 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 2:
JOG THE MACHINE

- ➲ Before setting the **Work Offset** 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.

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

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.

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.

Set Material in 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 Vice 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.

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 Vice 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 of 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 ¼" drill which is Tool 7 and the ½" End Mill which is Tool 8. We will also be using an edge finder and be putting it in tool slot #20 and therefore calling it Tool 20.

3. Close the Doors on the Machine

   - ➲ **Tool 6**

4. Press the **MDI Button** on the Control Panel Keyboard.

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

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

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

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

9. Once the Machine finishes it's cycle, Open the Doors.

10. Take **Tool 6** the Spot Drill and insert it (pull stud first) into the spindle.
    a. Push and Hold the **Tool Release** button.
    b. 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**.

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

    - ➲ Tool 7

12. Close the Machine Doors.

13. Press the MDI Button (if required).

14. Press **T** and then **7** [for tool 7] (letter T and number 7).

15. Press the ATC FWD button.

16. Open the Machine Doors.

17. Take **Tool 7** the ¼" Drill and insert it (pull stud first) into the spindle.
    a. Push and Hold the **Tool Release** button.
    b. 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**.

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

19. Close the Machine Doors.

    ➲ Tool 8

20. Close the Machine Doors.

21. Press the MDI Button (if required).

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

23. Press the ATC FWD button.

24. Open the Machine Doors.

25. Take **Tool 8** the ½" End Mill and insert it (pull stud first) into the spindle.
    a. Push and Hold the **Tool Release** button.
    b. 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**.

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

27. Close the Machine Doors.

28. MDI (if required).

29. Type in **T** then **2** then **0** [for the edge finder].

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

30. Press **ATC FWD**.

31. Open the Doors

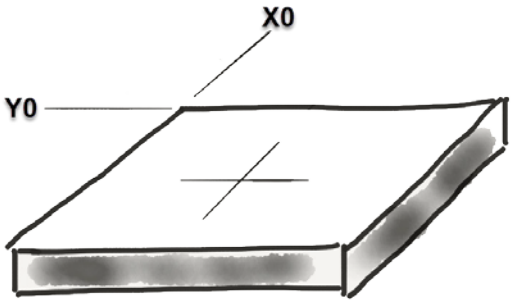
32. Now take the Edge Finder and insert it into the spindle.

TASK 4:
SET THE WORK OFFSET

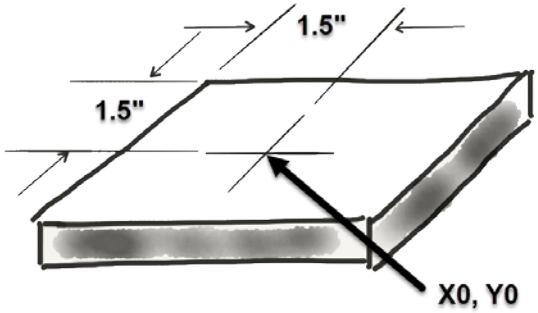
- In order to machine a workpiece (part), the mill needs to know where the part is located on the table. You can use an edge finder, an electronic probe, or many other tools and methods to establish part zero. For this lesson we are going to use the Edge Finder to set the Work Offset. Load an edge finder 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.

Step 1

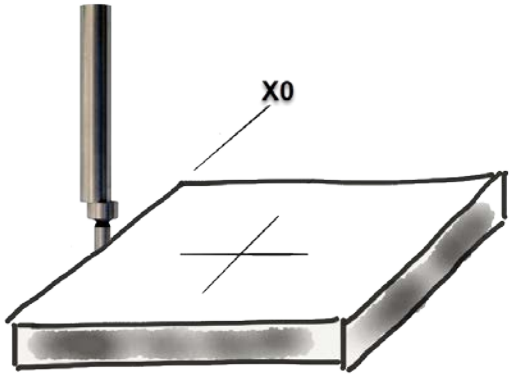


Step 2



SET THE X AXIS OFFSET

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



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).
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).
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.
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.
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.

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.

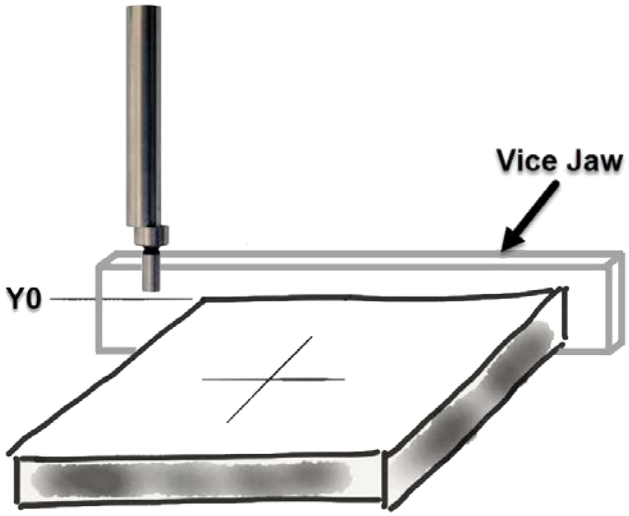
26. Press the **OFFSET** button until the **ACTIVE WORK ZERO OFFSET** panel is active (highlighted in white).

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

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

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.

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

   ➲ NOTE: We are using the Vice Jaw instead of the workpiece because the workpiece does not have enough material exposed to touch it with the Edge Finder.

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.

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.

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 **OFFSET** button.

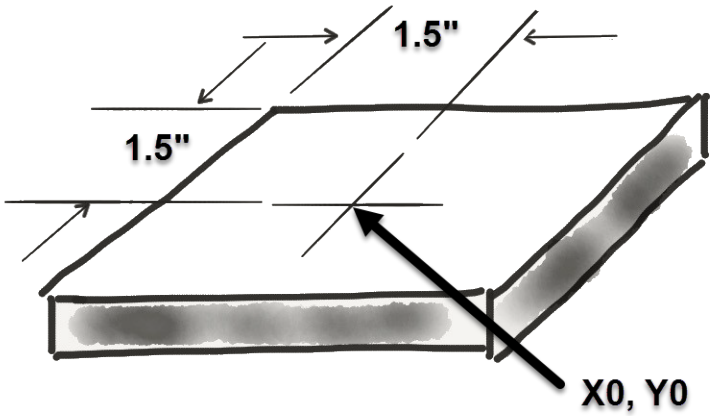
20. Until the **WORK ZERO OFFSET** panel is active (highlighted in white).

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

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

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

➲ 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.

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”**.

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”**.

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.

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.

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

11. 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 Enter to accept the warning.

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

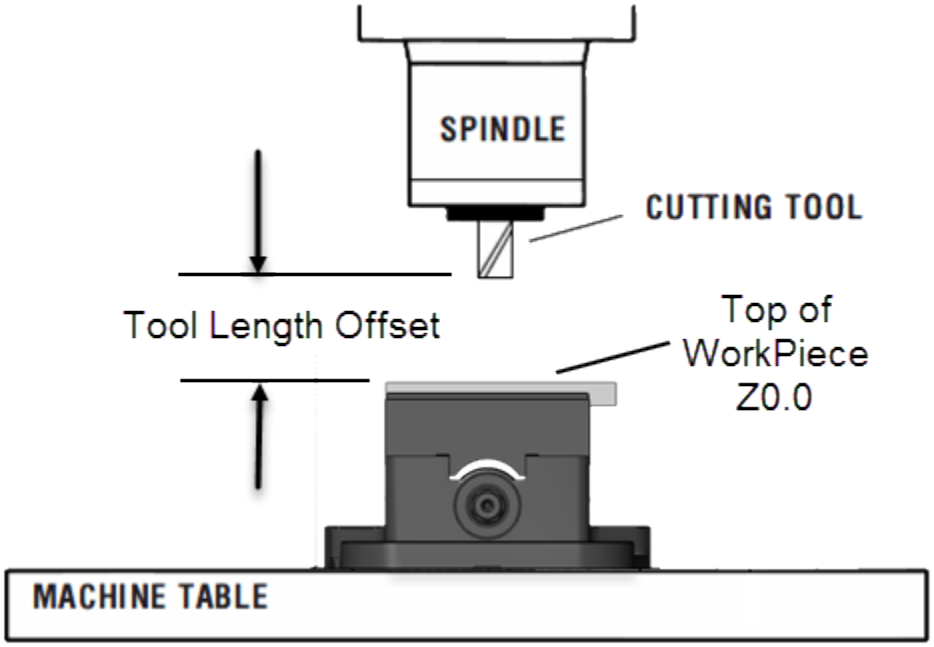
13. Select the **Z Axis** button and **Jog** the Spindle up and away from the part.

14. Select the **Handle Jog** button and set to **.01/10.**, then continue to raise the **Edge Finder** from the part.

TASK 5:
SETTING THE TOOL LENGTH OFFSETS

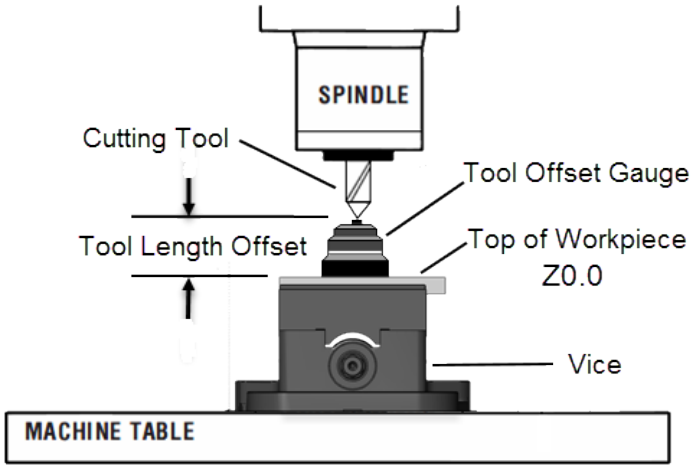
- Task 5 explains how to set the Tool Offsets which defines the distance from the tip of the tool to the top of the part. Another name for this is Tool Length Offset, which is designated as H in a line of machine code. The distance for each tool is entered into the Tool Offset Table. For this lesson we will be using a Tool Offset Gauge which provides a more accurate way of setting the Tool Offset. For more information on setting Tool Length Offsets and the Tool Offset Gauge please refer to the TIPS videos.

- 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.



TOOL 6

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



1. Put the **Tool Height Gauge** on top of the material.
2. Close the **Machine Doors**.

➲ 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.
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.
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.
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.

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 Appears. Press **Enter** on the Keyboard to accept.

17. Note the new **Offset** for **Tool 6** is 2.000" more. In our example it went from -5.4290 to -7.4290.

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

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

TOOL 7

1. Close the **Machine Doors**.

2. Press the **MDI** button.

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

4. Press the **ATC FWD** button.

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

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

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

8. **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.

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

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

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

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

13. Accept the warning by hitting **Enter**. Note the new **Offset** for **Tool 7** is 2.000" more. In our example it went from -8.9190 to -10.9190

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

TOOL 8

1. Close the **Machine Doors**.
2. Press the **MDI** button.
3. Press the letter **T** then the number **8.**
4. Press the **ATC FWD** button.
5. Press the **HANDLE JOG** button and press the **.01/10**. Button (The mill moves at a fast rate when the handle is turned).
6. Select between the **X** and **Y** and **Z** Axis and jog the tool near the center of the Tool Setter.
7. Press the **.001/.1** button.
8. **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.
9. Press the **OFFSET** button until the **TOOL OFFSET PAGE** is highlighted.
10. Make sure **Tool 8** is highlighted in the **GEOMETRY LENGTH** column.
11. Press the **TOOL OFFSET MEASURE** button.
12. We now need to subtract the 2.0" for the **Tool Setter**. Input **-2.0** and hit **Enter**.
13. Accept the warning by hitting **Enter**. Note the new **Offset** for **Tool 8** is 2.000" more. In our example it went from -9.0290 to -11.0290
14. Press the **Handle Jog** button and Jog **Z** away from the Tool Setter.
15. 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.

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

5. Use the **Right Cursor Key** to open the **USB DeviceTab** (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.

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.

15. Select **Cycle Start** to start simulation. Note, you may have to adjust the zoom **F2** to see the simulation.

16. 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.

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

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

➲ We are now ready to Run the Program.

➲ 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.

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

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

7. Press Cycle Start button to **Run** the full Program to the next tool change.

➲ Tool 7 Operation.

8. Press the **MEMORY** button and the **SINGLE BLOCK** Button.

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

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

11. Press Cycle Start button to **Run** the full Program to the finish.

- ➲ Tool 8 Operation.

12. Press the **MEMORY** button and the **SINGLE BLOCK** Button.

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

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

15. Press the **Cycle Start** button to **Run** the full Program to the finish.

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

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