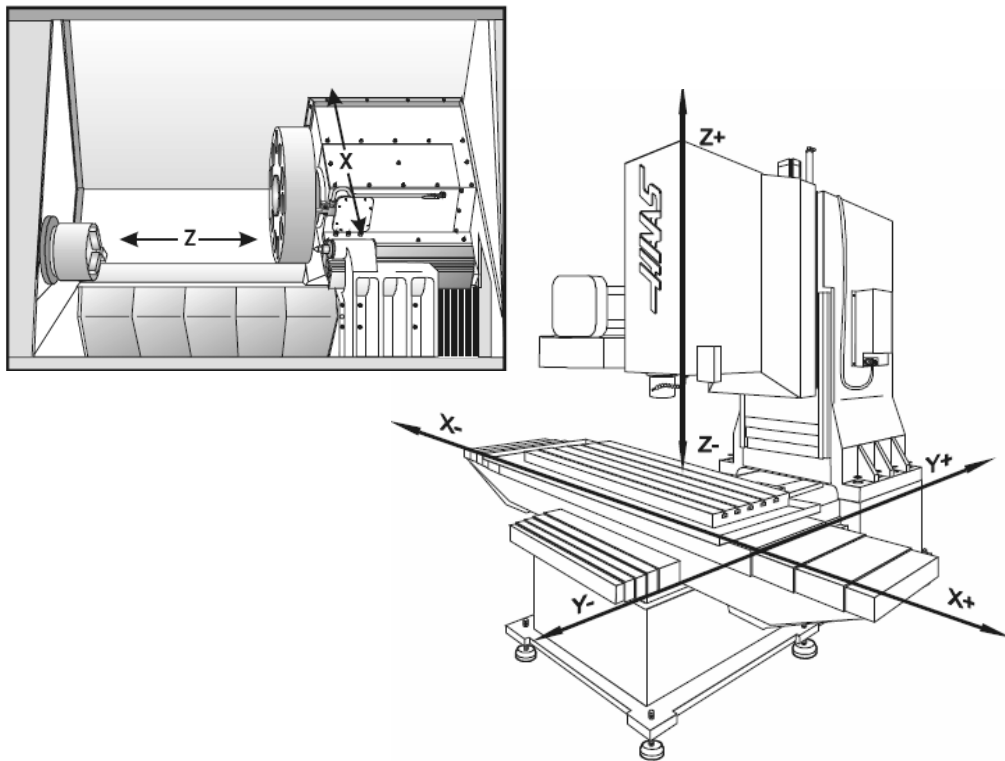


HAAS Machine WPI Quick Guide

June, 2012



Wei Tian, James Loiselle, Adam Sears

Torbjorn Bergstrom

Copyright 2012 Worcester Polytechnic Institute Haas Technical Education Center

Foreword

This document is intended to assist AUTHORIZED WPI students in operating the Haas CNC machine tools available in the Manufacturing Laboratories.

This document does not replace supervised training.

See one of the Manufacturing Laboratories Staff members for information on how to become AUTHORIZED.

Many of the figures in this document are taken from the Haas Automation VF/EC/HS Mill Series Control Panel Workbook available at:

http://www.haascnc.com/custserv_training.asp

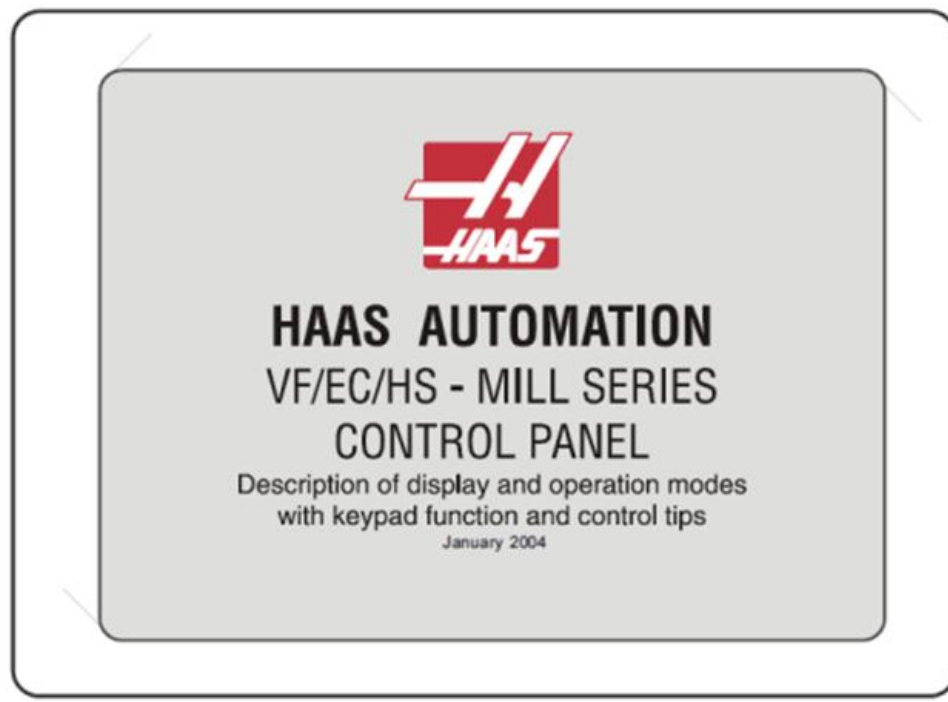


Table of Contents

Safe Operation Checklist.....	1
Power On.....	2
Send to Home Position	3
Simulate in CAM	4
Creating NC Code.....	5
Load NC Code from a Disk (Old Style Controller).....	6
Load NC Code with USB (New Style Controller)	7
Load NC Code with RS232.....	8
Select Program.....	10
Operate Tool Change	11
Load Tool(s)	12
Remove Tool(s).....	13
Handle Jog	14
Fixturing.....	15
Set Tool and Work Offsets	16
Simulate in Graphics	17
Reduce Rapid Rate.....	18
Before Running the Machine	19
Check Distance to Go	20
Cut Additional Parts	21
Remove Parts.....	22
Clean Work Area Mill	23
Appendix A Single Axis Zero Return	24
Appendix B Set up CAM Simulation	25
Appendix C Esprit Settings for File Locations	27
Appendix D Fixturing.....	28
Vise.....	28

Pneumatic Operated Collet Chuck on the Mill	29
Pneumatic Operated Collet Chuck on the Lathe	30
Appendix E Tram a Vice on a Mill	31
Appendix F Mill Tool Offsets	33
Set Tool Offset (Length) on a Mill with Gauge Block or Dowel Pin	34
Set Tool Offset on a Mill with the Tool Setting Probe	36
Appendix G Mill Work Offsets.....	45
Set Work Offset (X and Y) on a Mill with a Coaxial Indicator	46
Set Work Offset (X and Y) on a Mill with Dial Indicator	50
Set Work Offset (X and Y) on a Mill with an Edge Finder.....	52
Set Work Offset (Z) on a Mill with Gauge Block or Indicator	55
Set Work Offset on a Mill with the Spindle Probe	60
Appendix H Lathe Tool Offsets.....	74
Set Tool Offset on a Lathe with the Tool Setting Probe.....	75
Set Tool Offset on a Lathe with the Gauge Pin (for Z direction)	77
Set Tool Offset on a Lathe by Testing Cut (for X direction).....	79
Appendix I Lathe Work Offsets	81
Appendix J Set up Simulation in Graphics	82
Setting the Zoom for Simulation	83
Setting the Display for Positions	84
Setting the Display for Programs.....	85

Safe Operation Checklist

Operator

ID#

Start Time

Date

Machine Tool Name

End Time

Date

Operator Signature

All users must complete this checklist anytime they operate a machine tool at WPI. All of the steps must be repeated under the following circumstances:

1. any time a program has been modified,
2. any time the user has altered the setup in the machine tool since the program last ran successfully in that machine tool, or
3. any time the user has been away from the machine tool long enough for someone else to have altered the setup since the program last ran successfully in that machine tool.

Required Steps

Check here if you are completing a non-standard operation that precludes completing one or more of the steps below. *Details must be provided in the notes section below.* The current lab monitor must initial _____

[Simulate in the CAM Software](#)

- o This ensures that you know what the computer thinks the program will do.

[Set Tool and Work Offsets](#)

- o This records information about the length and diameter of tools loaded in the machine tool as well as the location of the workpiece.

[Simulate on Controller](#)

- o This ensures that you know what the machine tool thinks the program will do. It also allows you to catch some common programming and setup errors.

[Reduce Rapid Rate](#)

- o This gives you a chance to stop the machine tool before something bad happens if you have made a mistake in the previous steps.

[Check the Distance to Go as each tool is approaching the workpiece](#)

- o This is your last chance to catch setup and possible programming mistakes. Almost every crash that happens can be stopped if we pay attention and complete this step. Look into machine to see if is safe to proceed.

Notes:

See other Side

Power On

- (1) Press the green button (**POWER ON**) located on the upper left side of the controller

If the machine fails to start it may indicate that it has been locked out, or that the room emergency stop has been activated. Seek one of the Lab Staff for assistance

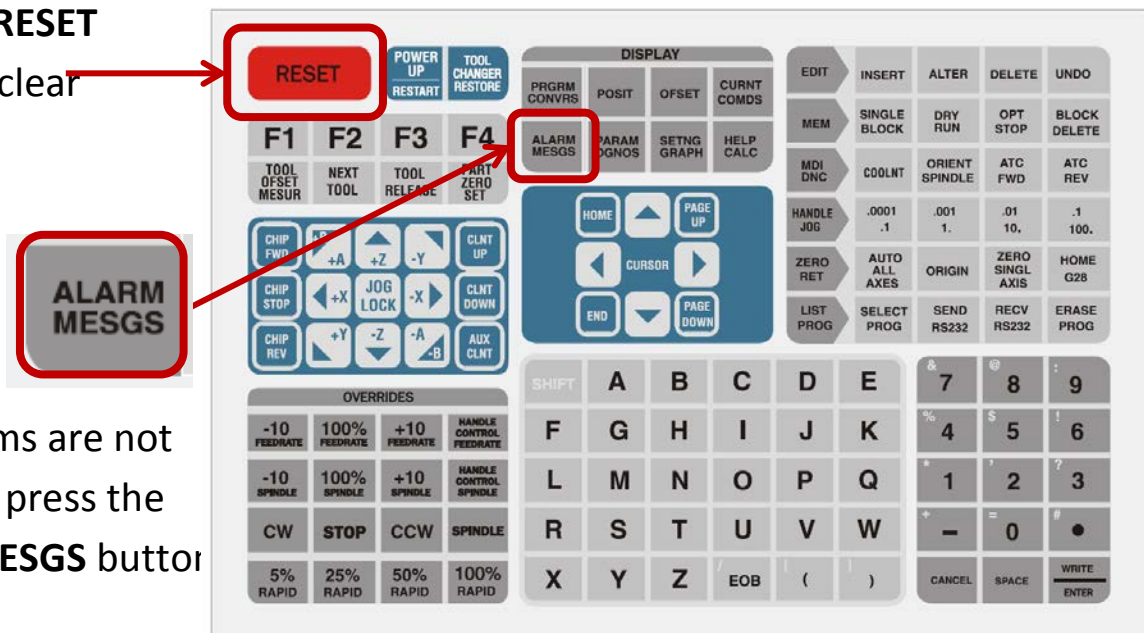


- (2) Reset the **Emergency Stop**

Twist it to the right and it will pop out (Pulling the button out will cause it to fail prematurely!)

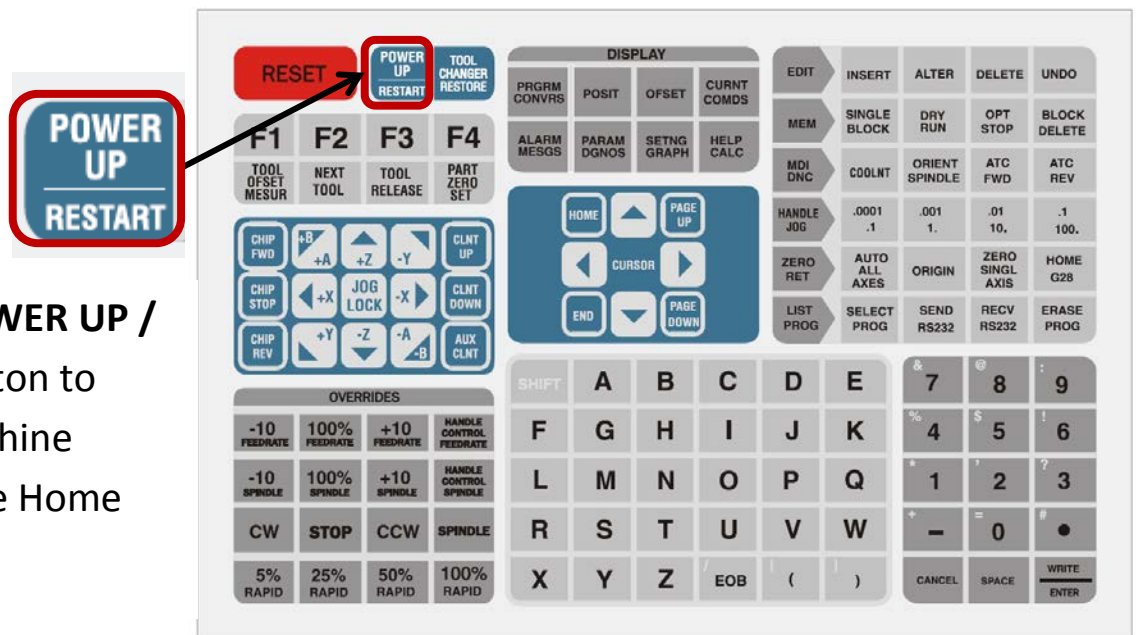
- (3) Reset the Alarms

Press the **RESET** button to clear all alarms



Send to Home Position

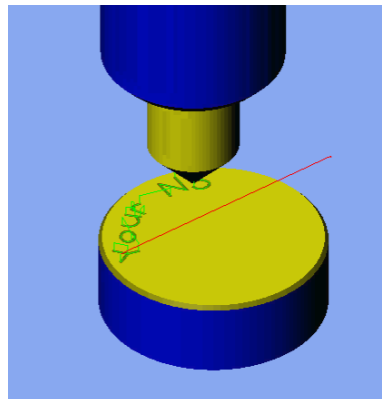
- If there is any obstruction (large fixturing, long tools, etc.), see [Appendix A Single Axis Zero Return](#).
- If all of the axes including the tool changer are clear to directly to the home position, Press the **POWER UP/RESTART** button and wait for the machine to complete operation



Simulate in CAM



- Watch the simulation, verify all moves are in accordance with your operations
 - The red lines indicate rapid motion;
 - The green lines indicate feed motion.
 - Ensure **Collision Detection** is set to “**Full**”, see Appendix H Set up CAM Simulation for more details



Troubleshooting:

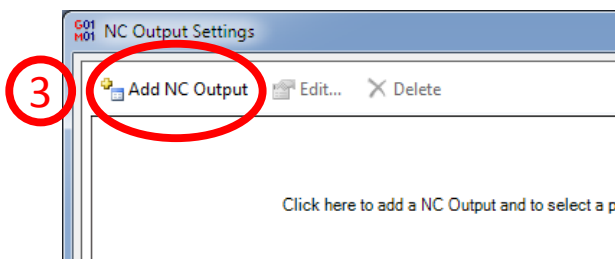
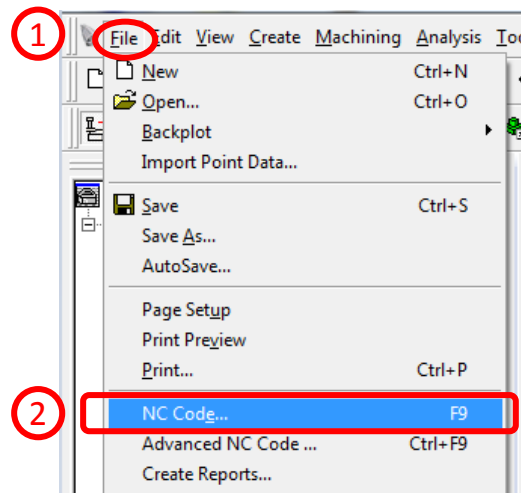
- If no stock appears when you simulate you need to set up the simulation (See [Appendix B Set up CAM Simulation](#) for more details).
- If a crash occurs in the simulation, the CAM program needs to be fixed accordingly before the NC codes are generated.

Creating NC Code

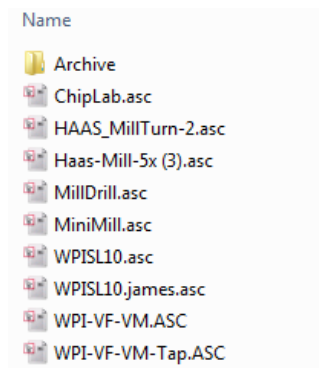
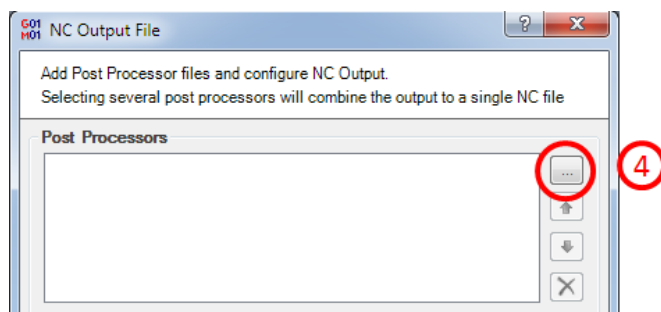
(1) On the menu bar, click **File**

(2) Select **NC Code**

(3) If there are no NC Outputs, also referred to as **Post Processors**, listed, click **Add NC Output**



(4) Browse for Post Processor Files and select the proper file for the machine you are using.



If the Post Processors do not show up, please refer to [Appendix C Esprit Settings for File Locations](#) for more details.

(5) Click **OK** and then **OK** again

Load NC Code from a Disk (Old Style Controller)

- Insert a 3.5” disk into the controller



- Select **EDIT**

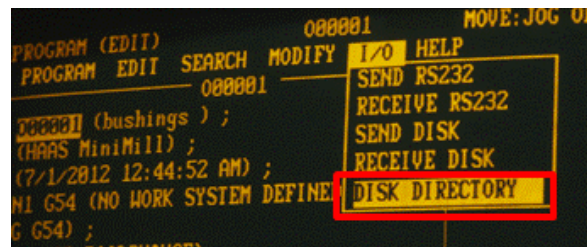
- Select **F1** to activate the menus



- Use the **Arrow Keys** to select I/O



- Select **Disk Directory**



- Press Enter Use **Arrow Keys** or **Jog Handle** to highlight the desired file

- Press **Write / Enter**



- If required press “**Y**” to indicate it is ok to overwrite the file currently in memory with the same Program Number

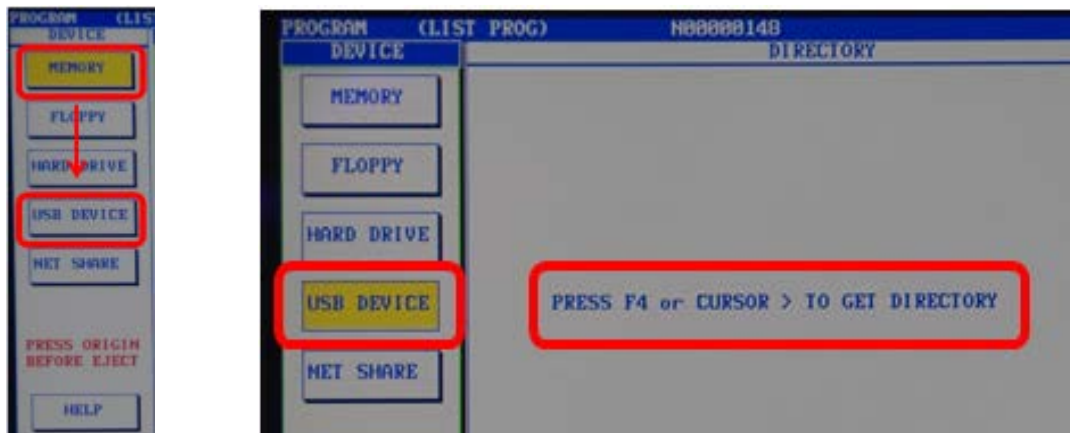
- Press **EDIT** or **MEM** to exit the **Disk Directory**

Load NC Code with USB (New Style Controller)

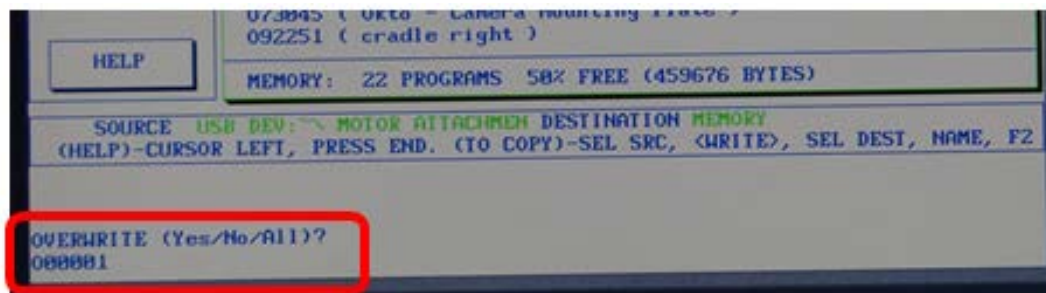
- Select **LIST PROG**



- Use the cursor arrows to select **USB DEVICE** and press the **Right Cursor Arrow** to select. If the list of devices isn't active, press the **Left Cursor Arrow**.



- Locate the program you are trying to load and press **Enter**
- To get back to the list of devices, press the **Left Cursor Arrow**
- Use the cursor arrows to select **MEMORY** and press the **Right Cursor Arrow**
- Press **F2** to copy the file to **MEMORY**. If the program number already exists, the operator will be prompted whether to overwrite or not. Press **Y** for Yes and **N** for No.

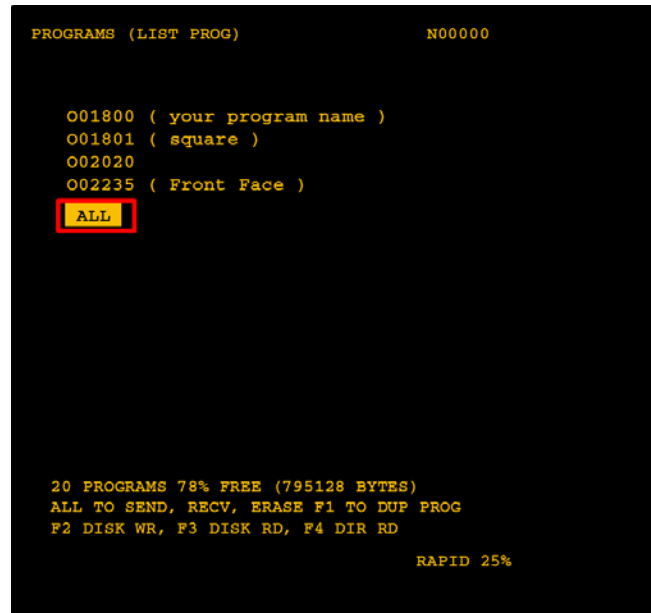
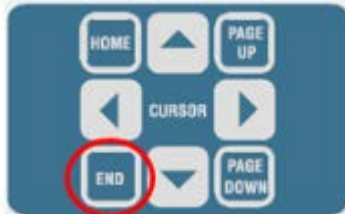


Load NC Code with RS232

- Select **LIST PROG** to view listed programs



- Press **END** to make sure **ALL** is highlighted

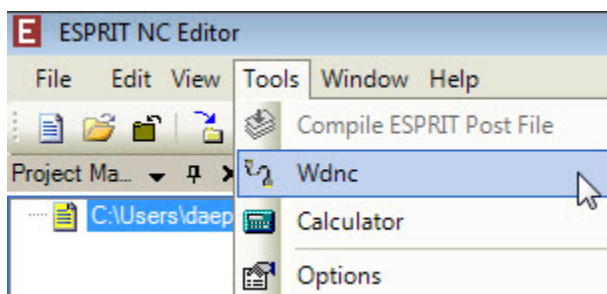


- Press **RECV RS232**

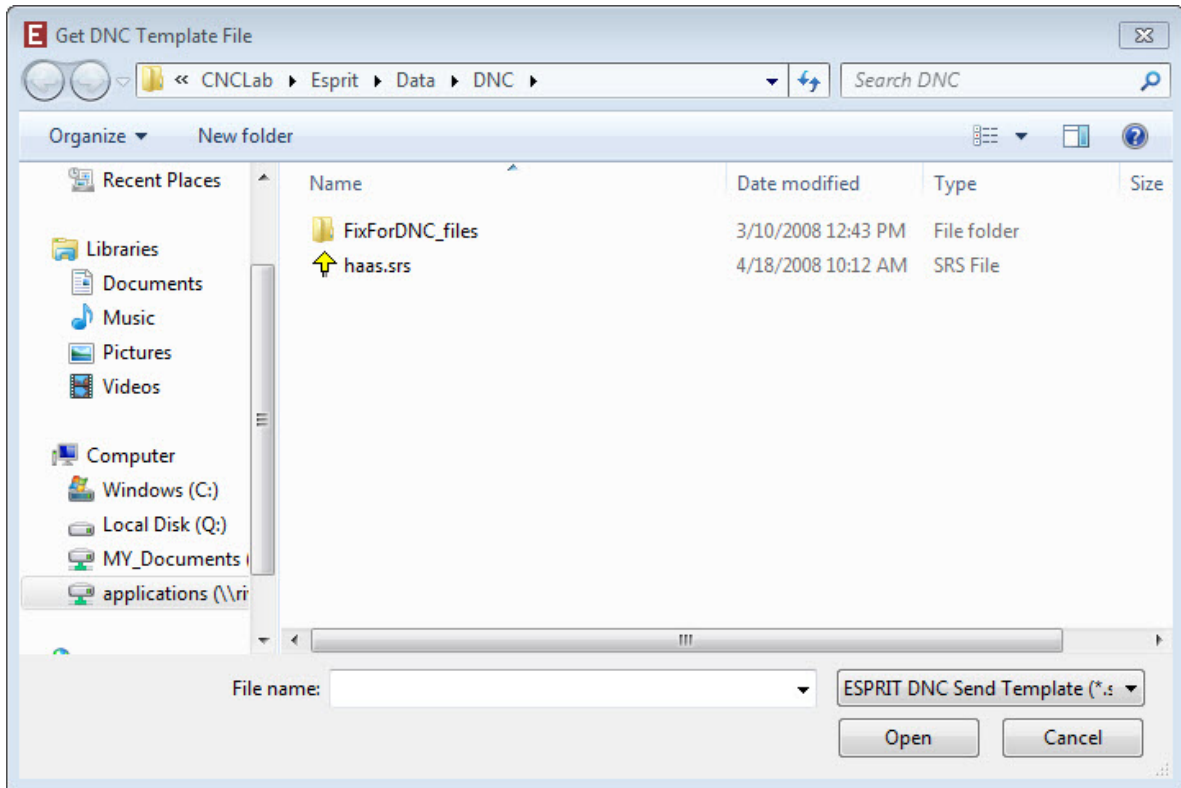


- The controller is now ready to receive NC code from Esprit

- In the NC Editor, click tools and WDNC.



- If a file-search box launches, browse to the DNC Template. Please refer to [Appendix C Esprit Settings for File Locations](#) for more details.



- With the machine controller set to receive code, press play in the WDNC window.



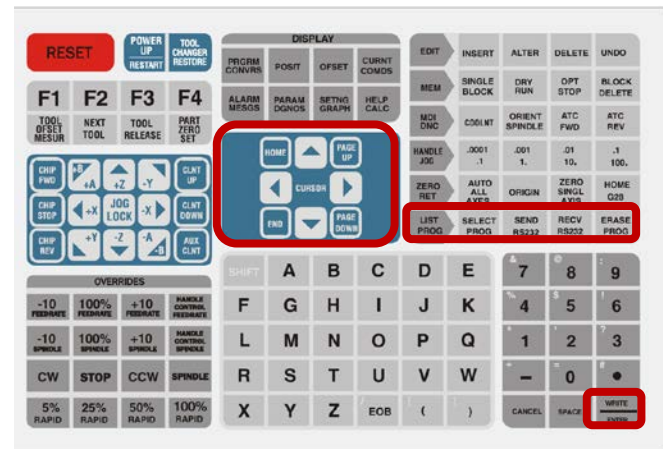
- If the program number already exists, the operator will be prompted whether to overwrite or not. Press **Y** for Yes and **N** for No.

```
OVERWRITE (Yes/No/All)?
001800
```

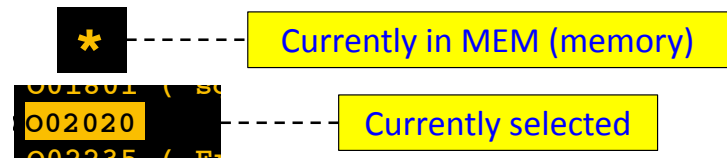
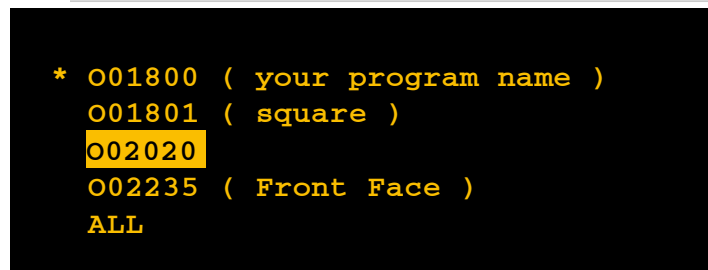
Select Program

Select the program that you downloaded to your machine tool that is associated with your operation.

- Select **LIST PROG**



- Use Arrow Keys to highlight your program



- Press **WRITE/ENTER** to confirm your selection



*e.g. the program numbered **002020** is going to be selected*



Operate Tool Change

(1) Check to ensure the tool changer can rotate without any of the tools impacting fixturing or parts ([Handle Jog](#) the table or turret as necessary to clear the tool change area)

(2) Select **MDI** mode

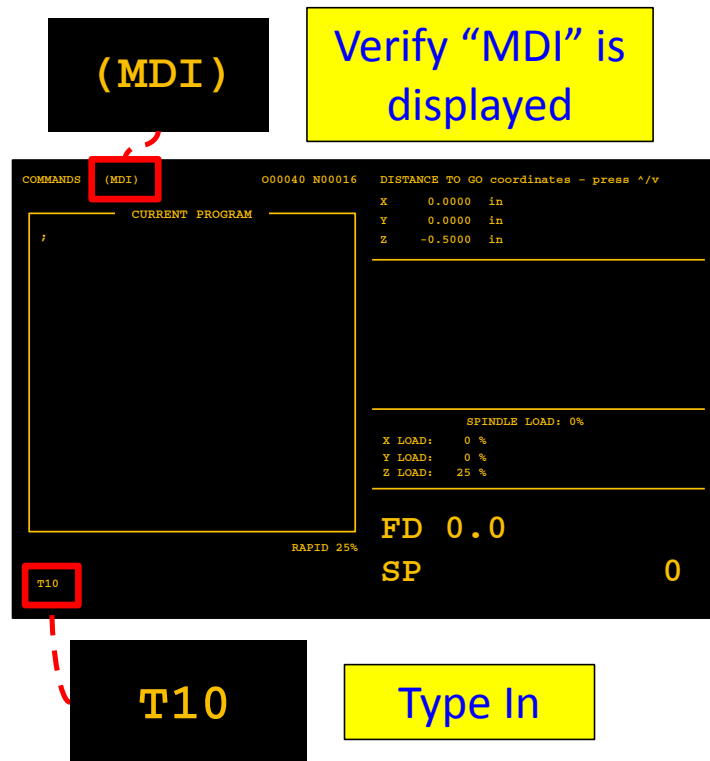


(3) Type “T” followed by the desired **Tool Number**

Example: Tool Number 10

Enter “T10”

Do Not Press Write/Enter



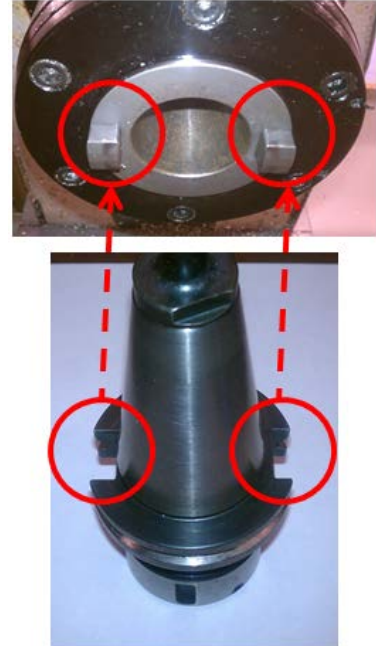
The screenshot shows the CNC control interface in MDI mode. At the top, a black box contains the text "(MDI)". To its right, a yellow box contains the text "Verify 'MDI' is displayed". The main screen displays "COMMANDS (MDI)" at the top left, "000040 N00016" at the top right, and "DISTANCE TO GO coordinates - press ^/v" on the right side. The central area shows "CURRENT PROGRAM" with a cursor. On the right, there are coordinates: X 0.0000 in, Y 0.0000 in, Z -0.5000 in. Below that, "SPINDLE LOAD: 0%" is shown, followed by "X LOAD: 0 %", "Y LOAD: 0 %", and "Z LOAD: 25 %". At the bottom right, "FD 0.0" and "SP 0" are displayed. A red box highlights the "T10" command entered in the command line at the bottom left. Below the screenshot, a black box contains the text "T10" and a yellow box contains the text "Type In".

(4) Select **ATC FWD (TURRET FWD)** or **ATC REV (TURRET REV)**



Load Tool(s)

- (1) [Tool Change](#) to the desired tool number.
- (2) If there is a tool in the spindle remove it. [Remove Tool\(s\)](#)
- (3) Ensure **MDI** mode is selected
- (4) With your left hand hold the tool in the spindle with the slots lined up with the spindle dogs pressing up firmly



- (5) With your right hand press and hold the **TOOL RELEASE** button until the tool holder move up into the spindle

Look at the controller and verify the MDI mode before pressing this button



- (6) Release the **TOOL RELEASE** button
- (7) Ensure the tool is held securely in the spindle before releasing it from your left hand
- (8) Repeat as necessary

Remove Tool(s)

- (1) [Tool Change](#) to the desired tool number.
- (2) Ensure **MDI** mode is selected
- (3) With your left hand hold the tool firmly
- (4) With your right hand press and hold the **TOOL RELEASE** button until the tool holder move down out of the spindle

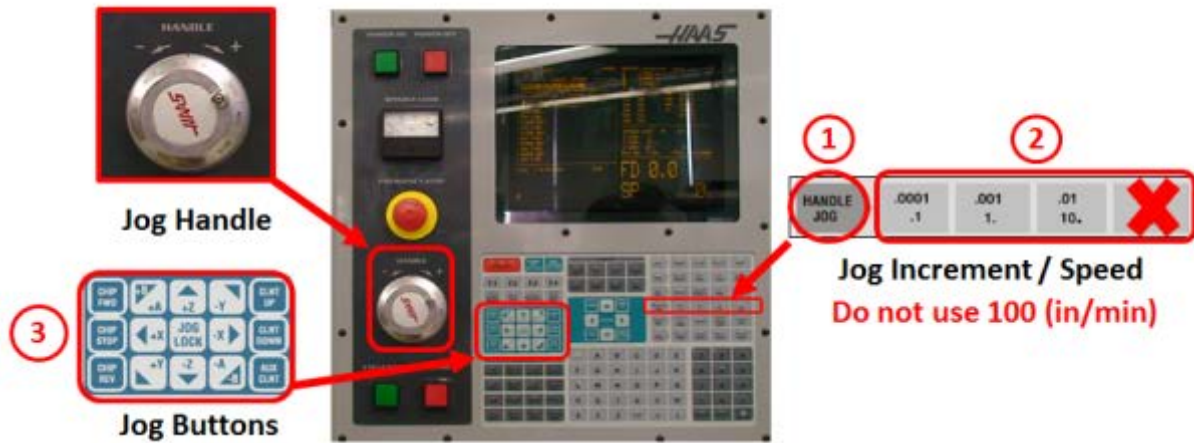


Look at the controller and verify the MDI mode before pressing this button

- (5) Release the **TOOL RELEASE** button
- (6) Repeat as necessary

Handle Jog

Handle Jog Mode is used to manually position the axis of the machine tool. You can use either the **Jog Handle** or the **Jog Buttons**



(1) Select the **HANDLE JOG** mode

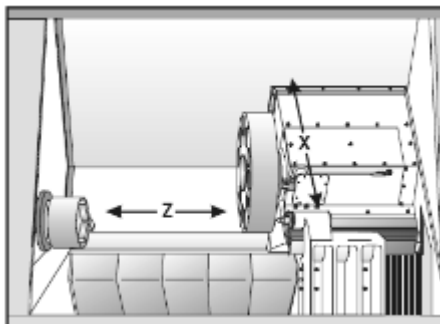
(2) Select an **Increment / Speed**

Do not use 100 (in/min)

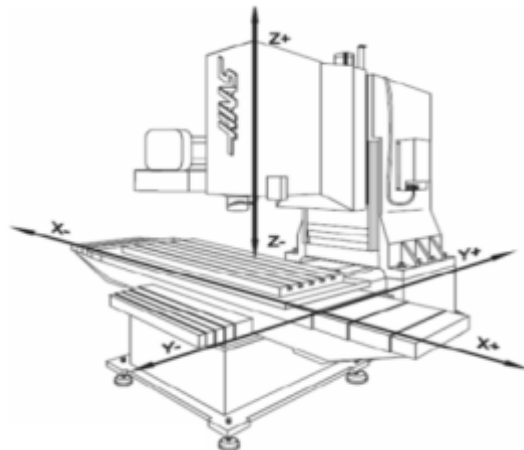
- The top number indicates the distance the selected axis will move for each “click” of the **Jog Handle**
- The bottom number indicates the speed the axis will move (inches/min) if the **Jog Buttons** are held in, or **JOG LOCK** is used

(3) Select the axis to jog

- **X, Y, Z**



Haas Lathe X and Z axis lines

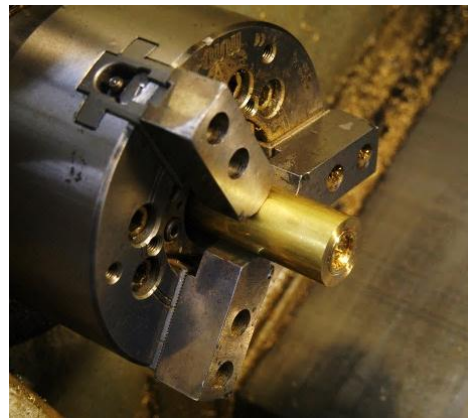
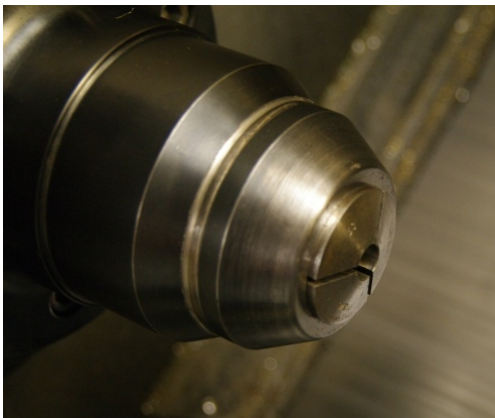
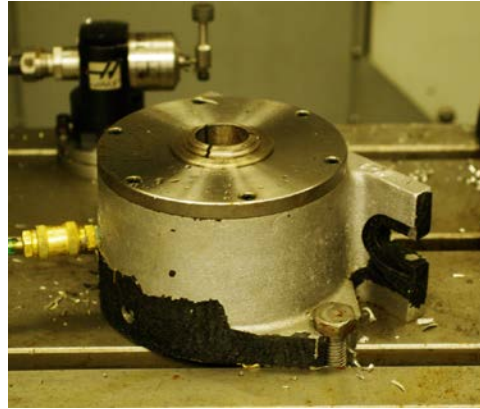
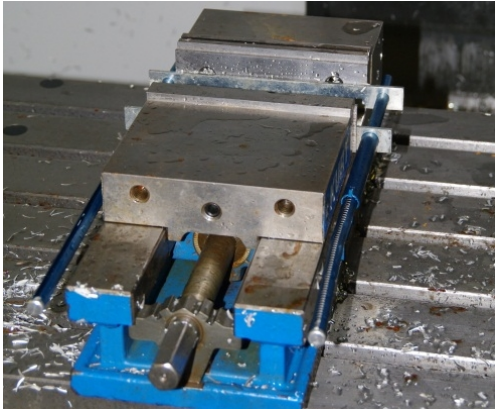


Haas Mill X, Y and Z axis lines

Fixturing

Fixturing is the process to secure the workpiece to the worktable or chuck so that the rigid position and orientation is maintained during the machining process.

- The workpiece needs to be properly secured to the worktable (mill) or chuck (lathe) to perform machining operations.

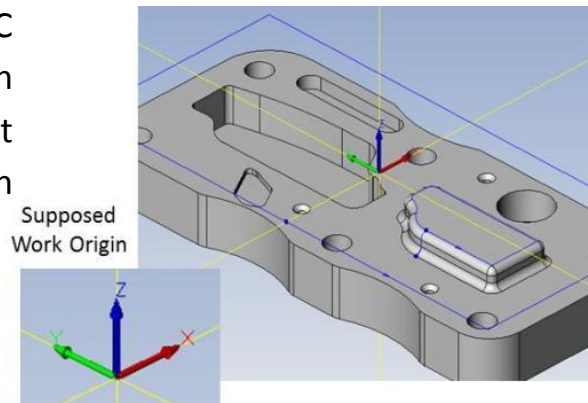


- For more details, see [Appendix D Fixturing](#)

Set Tool and Work Offsets

What is the Work Offset?

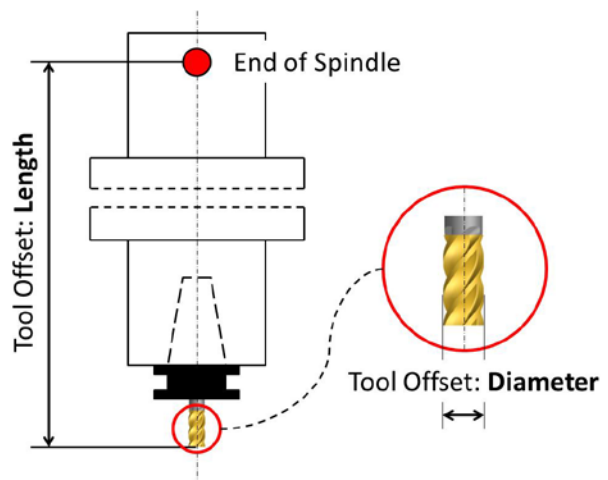
Work offset is the origin of the workpiece in referring to the machine tool coordinate system that your NC program uses. It describes the position of the workpiece in the machine tool. It correlates the workpiece origin with the machine tool coordinates.



What is the Tool Offset?

Tool offset is the coordinate systems that record the critical dimensions of the tools in referring to the machine tool coordinate system.

Only when the work offset and the tool offset are fully defined, the NC program could guide the cutting tool to perform machining operations on the correct location in order to get correct shape and precision.



Refer to the Appendices for detail information on how to set Tool and Work Offsets.

[Appendix F Mill Tool Offsets](#)

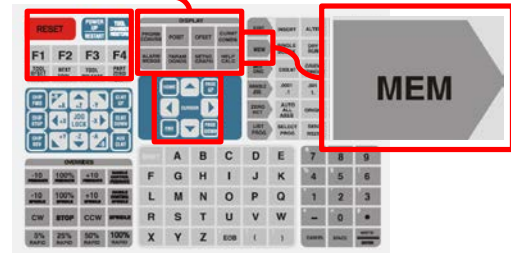
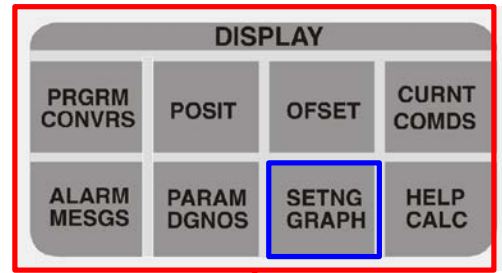
[Appendix G Mill Work Offsets](#)

[Appendix H Lathe Tool Offsets](#)

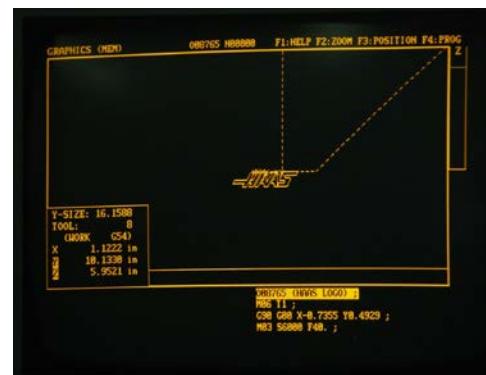
[Appendix I Lathe Work Offsets](#)

Simulate in Graphics

- Ensure the desired program is selected. Press the **MEM** button to select the **MEMORY** mode
- Press the **SETNG GRAPH** button above the arrow keys **TWICE** to enter the graphics display
- Press **CYCLE START** to start the simulation
- Look at the display to check if there are alarm messages or whether the tool path is as expected



PRESS
"CYCLE START"



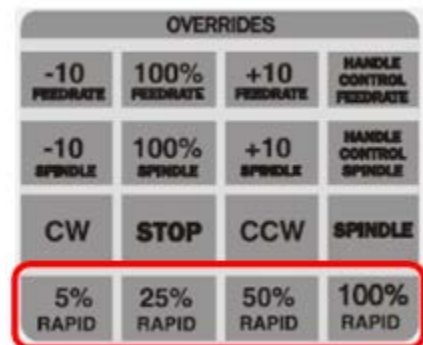
See [Appendix J Set up Simulation in Graphics](#) for more details.

Reduce Rapid Rate

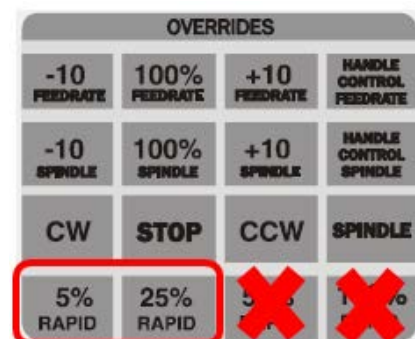
- Recall that rapid motion is positional movement between cutting operations. The rate of rapid motion is controlled by the machine tool.



- This rate can be controlled by the machine operator by using the **% of Rapid Buttons** located in the **OVERRIDES** section of the controller.

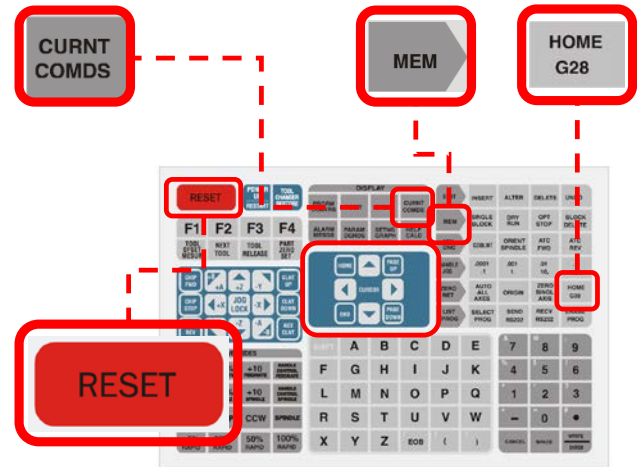


- When running a program for the first time, it is important to reduce the rapid rate to 5% or 25% at most. This is done by pressing the **5% RAPID** or **25% RAPID** buttons. The rapid rate may be increased once a program is proven to not crash.

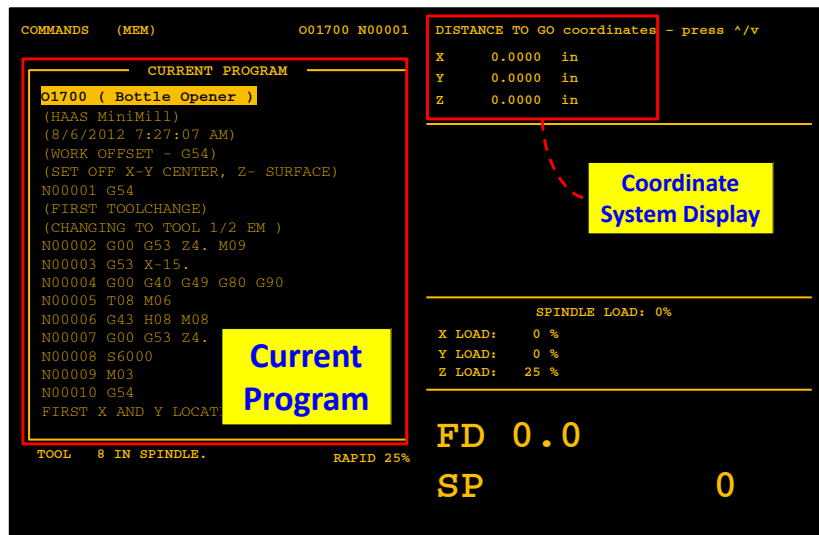


Before Running the Machine

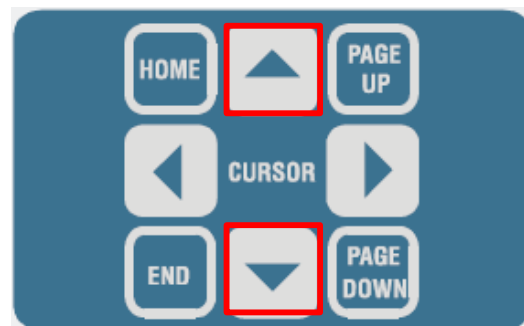
- Under **MEMORY** mode (press **MEM** button), Press the **CURNT COMDS** button to enter **Current Command view**
- Press **RESET**
- Look in the machine to verify the machine is at the **“Home Position”**



If not, send it to home by pressing **HOME G28**

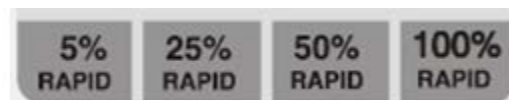


- Use Up and Down Arrows in Cursors to select **DISTANCE TO GO** on the **Coordinate System Display**



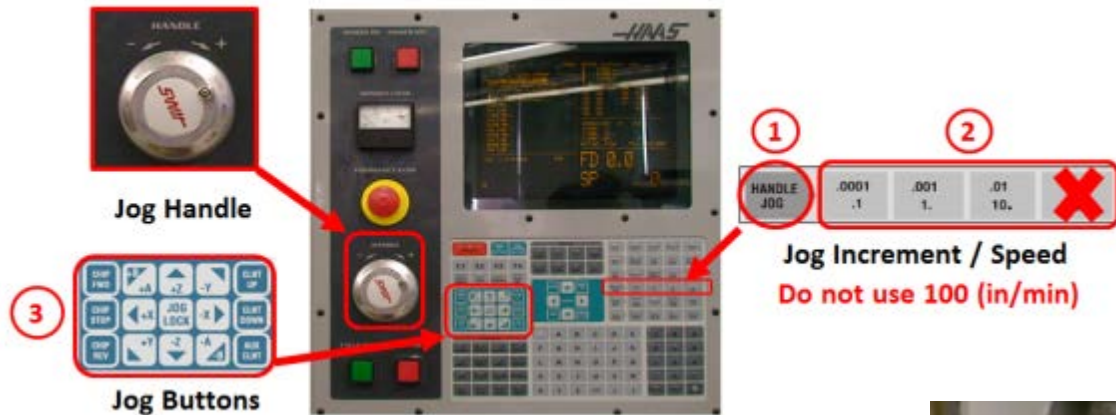
Cut Additional Parts

- (1) Once the first part has been completed you can load and run any additional parts.
- (2) Unless a workstop or other locating device was used when the first part was loaded and probed the new workpiece will need to be probed in order to set the [Work Offset](#)
- (3) Check **DISTANCE TO GO** on the first tool
- (4) If only the workpiece was changed between running the parts you can increase the [RAPID Rate](#) after the first tool has engaged the workpiece



Remove Parts

- (1) If necessary, Handle Jog the table so you have easy access to the fixturing.



- (2) You may use the air hose to blow **coolant** off of the workpiece
- Use air gently
 - **Do not use air to blow chips off of the part or tables in the mill**



- (3) Open the vise just enough to remove the part

- (4) Return the vise handle to the shadow board



Clean Work Area Mill

(1) Remove Part safely and brush off chips (Video)



(2) Use the coolant spray down hose to wash chips down to the bed of the machine tool (Video)



(3) Use chip rake to pull the chips into the chip bin (Video)



(4) Use a broom to sweep the area around the machine tool (Video)



(5) Wipe down the outside of the machine tool and the controller as necessary (Video)

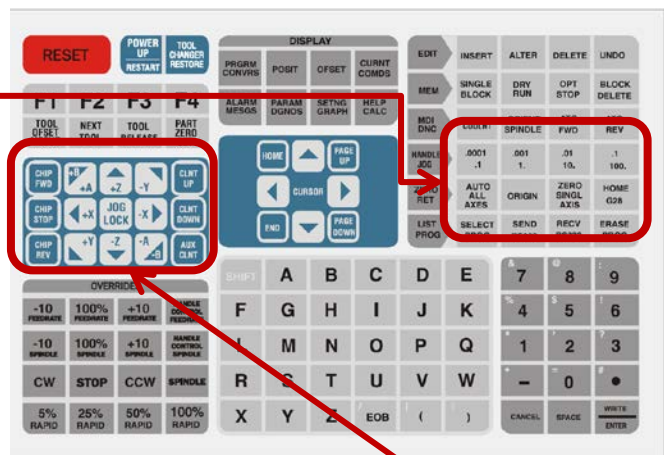


Appendix A Single Axis Zero Return

If the POWER UP/RESTART button was skipped, as the path from the current position to the home position is not clear, the axis need to be sent to home one at a time. For each axis, when the path is clear for it to move to the Home Position

- You may need to enable setting 53 (JOG W/O ZERO RETURN) For more information, refer to Haas Operator’s Manual page 257.
- Use the [Handle Jog](#) mode to move the axis so that they can safely move to the home position

MDI DNC	COOLNT	ORIENT SPINDLE	ATC FWD	ATC REV
HANDLE JOG	.0001 .1	.001 1.	.01 10.	.1 100.
ZERO RET	AUTO ALL AXES	ORIGIN	ZERO SINGL AXIS	HOME G28



- Change to the Zero Return Mode by pressing the **ZERO RET** button

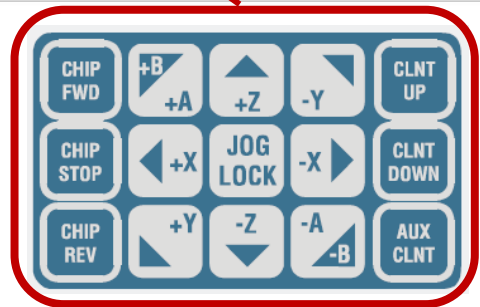


RET button

- Key in the axis name (**X, Y, or Z**)
- Press the **ZERO SINGL AXIS**



button



- Jog the X and Y axes so that the tool changer is clear to rotate
- Change to MDI mode by pressing **MDI/DNC** button



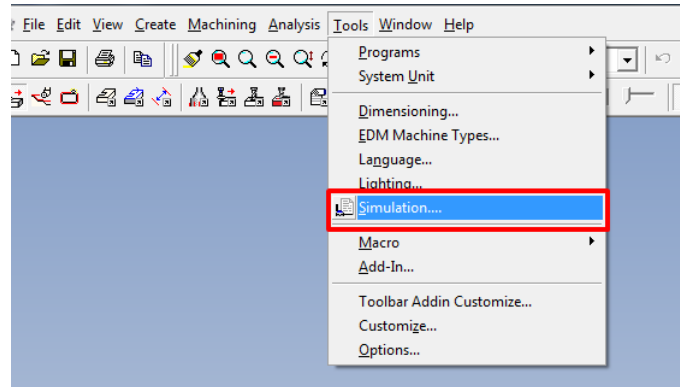
- Press **ATC FWD**



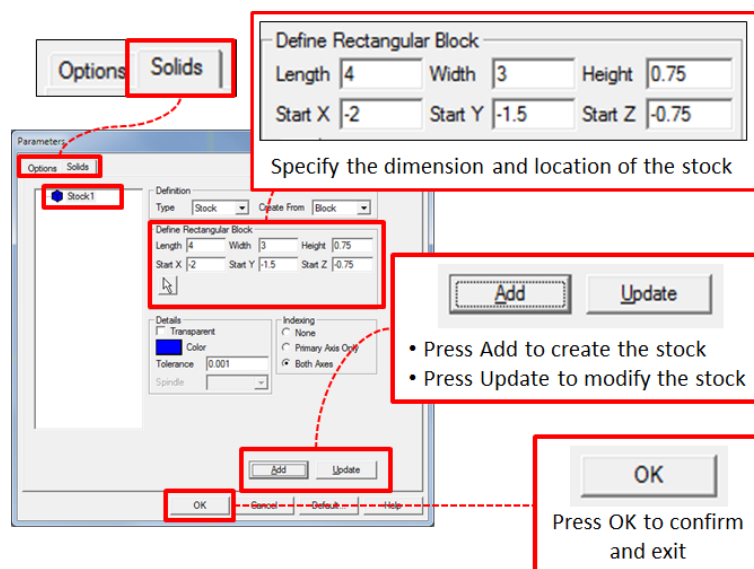
Appendix B Set up CAM Simulation

Setting up CAM simulation is to ensure that you know what to expect from the program. This is your first chance to see if anything in the program will crash.

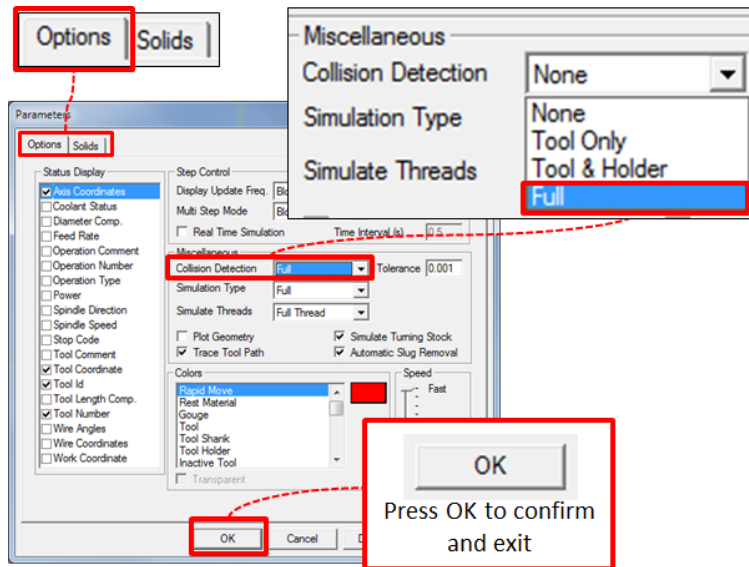
- Start the simulation set-up window



- Create stock for the simulation
 - Make sure the **Solids** tab is active
 - Specify the dimension and location of the stock
 - Press **Add** to create the stock
 - Or press **Update** to change your stock



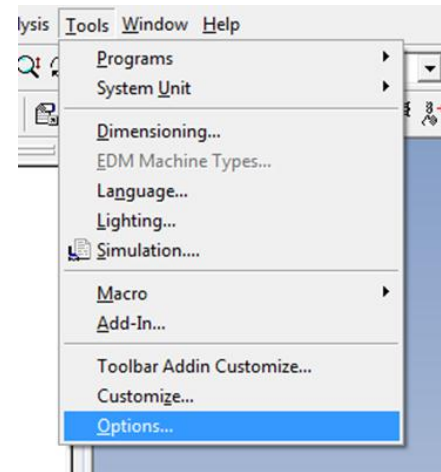
- Set the **Collision Detection** to **Full**
 - While the **Options** tab is active
 - On the dropdown menu of **Collision Detection**, select **Full**
 - Press **OK** to confirm the selection and exit



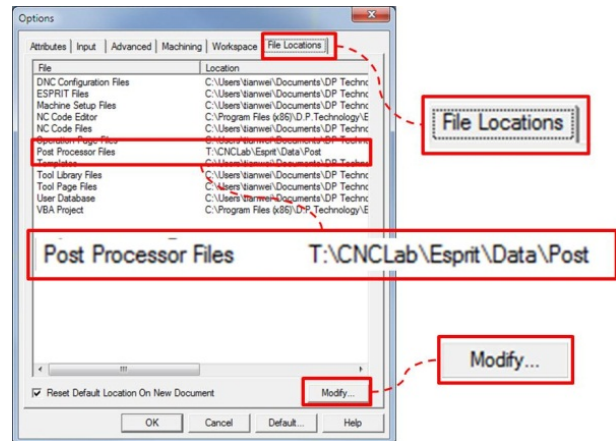
Appendix C Esprit Settings for File Locations

This appendix illustrates how to set proper file locations in the Esprit.

- On the main Esprit screen, locate the **Tools\Options** from the menu bars

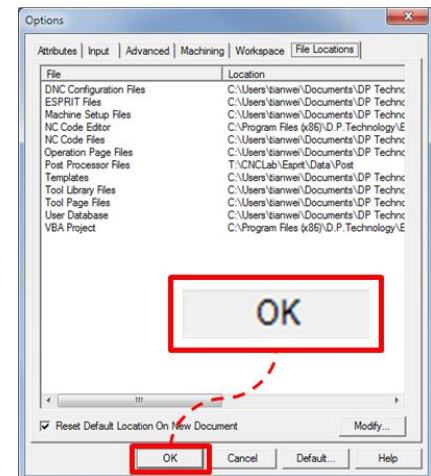


- Click the Tab of **File Locations**, highlight the **Post Processor Files**, double click on it or click **Modify** to modify the default locations



- The location should be [rivet.wpi.edu\CNCLab\Esprit\Data\Post](http://rivet.wpi.edu/CNCLab/Esprit/Data/Post)

- Click OK to confirm the change

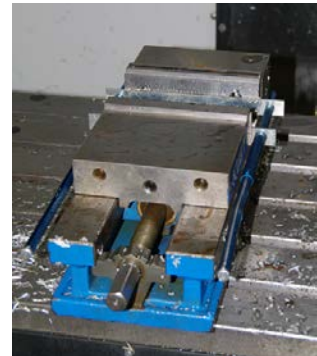


Appendix D Fixturing

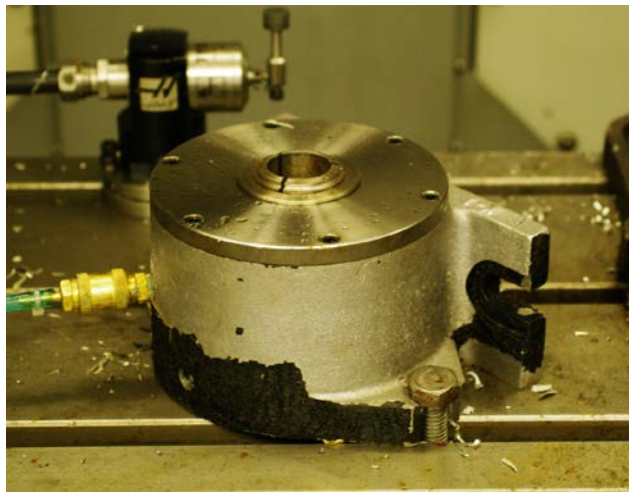
- Know the fixture you are using
- If using T-nuts, verify that there is no chips or debris on the thread and the thread is free of damage
The nut should rotate freely along the whole thread. Otherwise, DO NOT FORCE them and check the thread visually.

Vise

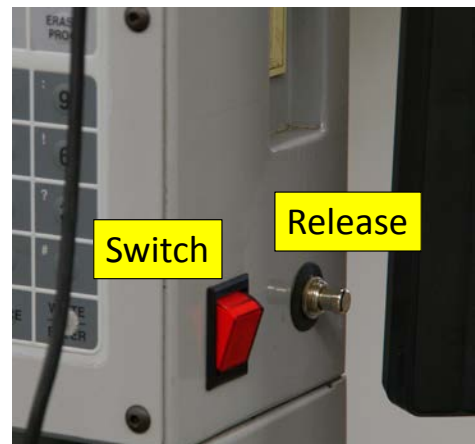
- **Make sure the vise is secured to the table**
- Use parallels if necessary (make sure the workpiece sits parallel to the bottom of the vise)
- Make sure the vise will NOT be cut unintentionally (check the maximum depth of operations)
- Make sure the vise is properly trammed (See [Appendix E Tram a Vice on a Mill](#) for more details)
- **Make sure the workpiece is secured to the vise**
- **DO NOT LEAVE THE VICE HANDLE IN THE MACHINE DURING THE OPERATION**



Pneumatic Operated Collet Chuck on the Mill



- **Make sure the fixture is secured to the table**
- To clamp:
 - Press the **Switch** button (mounted on the right side of the controller) once
- To unclamp:
 - Press the **Switch** button once; press the **Release** button once
- **Caution: Do NOT clamp the collet WITHOUT a workpiece in it.**



Pneumatic Operated Collet Chuck on the Lathe



- **Make sure the fixture is secured**

- To clamp/unclamp:
Press the yellow pedal once for clamping or unclamping

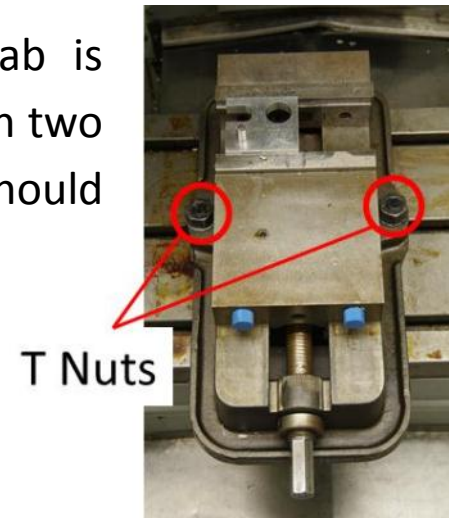


- **Caution: Do NOT clamp the collet WITHOUT a workpiece in it**

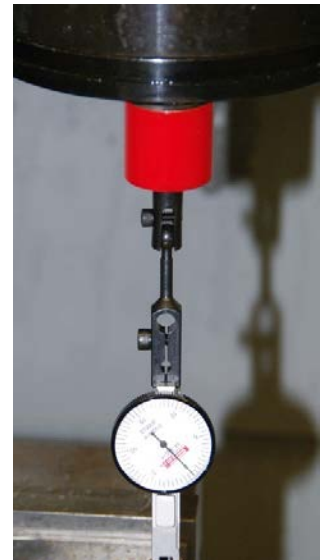
Appendix E Tram a Vice on a Mill

The process of tramping a vice is to align the X and Y axes of the machine tool with the X and Y axes of the vice.

- The manual vice available in this lab is typically secured to the worktable with two T-nuts. Prior to this process, they should be loose.

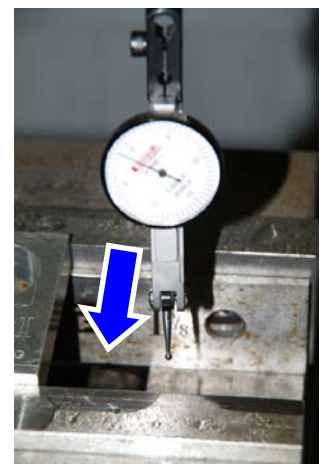


- Install the indicator
 - Install the indicator to a magnetic base
 - Attach the magnetic base to a clean smooth surface on the spindle head

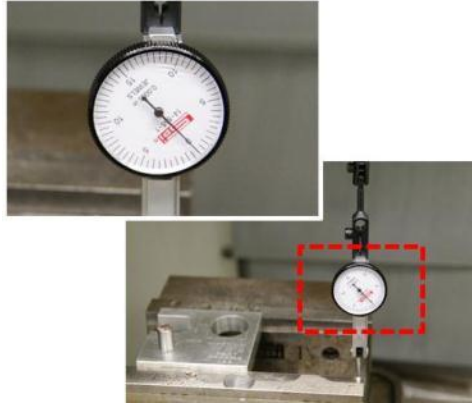


- Find a smooth, machined surface and Handle Jog the indicator towards it

- Slowly Handle Jog so the stylus of the indicator approaches the surface along the normal direction until the dial started to move



- As the dial started to move, keep jogging so until the dial rotates for about 180 degrees
- Jog the indicator to one end of the surface, set the dial to zero



- Jog the indicator along the probing surface and observe the movement of the dial



The probing point is closer to the indicator



The probing point is farther to the indicator

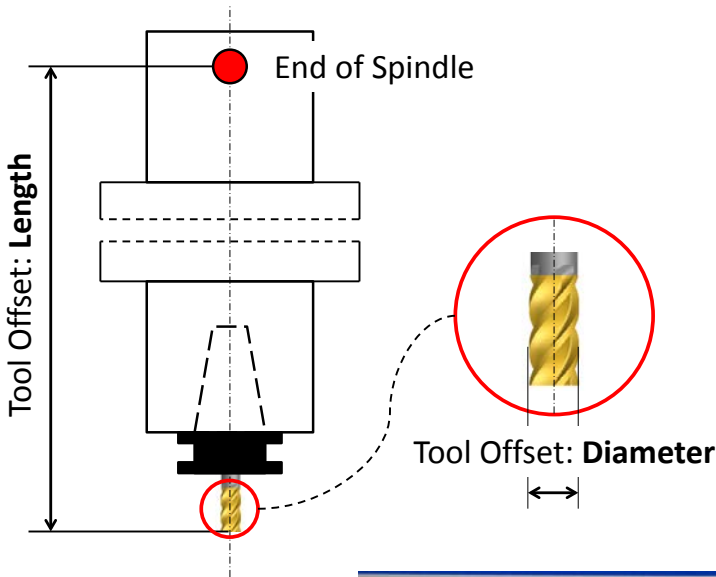
- Tap on the side of the vice as required
- Jog the indicator along the probing surface again to evaluate the adjustment, tap as required until the dial movement is within **+/-0.001''**
- Tighten the T-bolts on the vice after the tramming is completed

Appendix F Mill Tool Offsets

This appendix illustrates how to set mill tool offsets in two methods.

Set Tool Offset (Length) on a Mill with Gauge Block or Dowel Pin

Set Tool Offset on a Mill with the Tool Setting Probe



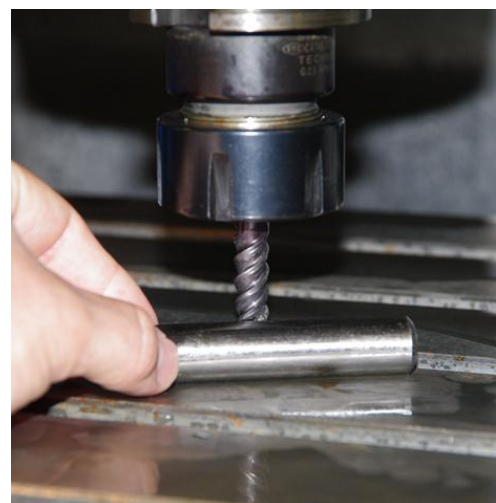
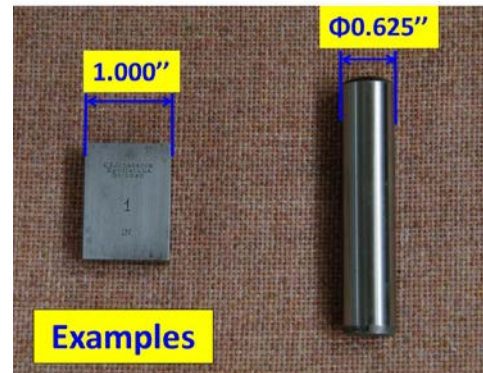
TOOL	COOLANT POSITION	---LENGTH---		---DIAMETER---		ACTUAL	
		GEOMETRY	WEAR	GEOMETRY	WEAR	FLUTES	DIAMETER
1	23	-22.8710	0.	0.	0.	2	0.
2	20	-3.3200	0.	0.	0.	1	0.
3	24	5.0343	0.	0.	0.	2	0.
4	15	3.5745	0.	0.5000	0.	2	0.
5	18	4.9809	0.	0.	0.	2	0.
6	18	4.2838	0.	0.	0.	2	0.
7	19	4.8808	0.	0.	0.	2	0.
8	17	4.0910	0.	0.	0.	2	0.
9	23	3.8586	0.	0.	0.	2	0.
10	9	11.8496	0.	0.	0.	2	0.
11	16	5.1676	0.	0.	0.	2	0.
12	14	3.3529	0.	0.	0.	2	0.

Set Tool Offset (Length) on a Mill with Gauge Block or Dowel Pin


The tool length needs to be known by the machine tool for the machining operations to be performed properly. In this section, we learn how to set this Tool Offset (Length) with a gauge block or dowel pin.

Any objects with well characterized dimensions can be used as gauge blocks. We use a $\Phi 0.6250''$ as an example.


- Find a smooth flat surface in the machine tool (in this document, the worktable is chosen)
- Change to the tool to be set
- Handle Jog the tool towards the common referring surface (worktable) where the block does **NOT** fit into the gap between the tip of the tool and the surface
- Handle Jog
 - Use **0.001''** increment
 - Move the tool **UP** a little bit
 - Try to fit the gauge block between the tool tip and the work table
 - When it just fits, move the gauge block away from the tool first, then Handle Jog the tool down for 2~3 clicks

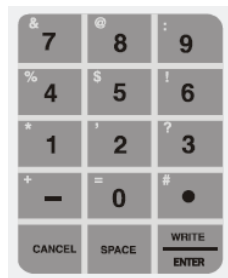


.0001
.1

- Switch to **0.0001"** increment
- Repeat this process until the point where the gauge block just fit into the gap
- Set the Tool Offset in the controller
 - Cycle through the **OFFSET**  button until the tool offset screen is displayed

COOLANT		---LENGTH---		---DIAMETER---		ACTUAL	
TOOL	POSITION	GEOMETRY	WEAR	GEOMETRY	WEAR	FLUTES	DIAMETER
1	23	0.	0.	0.	0.	2	0.
2	20	-3.3200	0.	0.	0.	1	0.
3	24	5.8343	0.	0.	0.	2	0.
4	15	3.5745	0.	0.5000	0.	2	0.
5	18	4.3089	0.	0.	0.	2	0.
6	18	4.2830	0.	0.	0.	2	0.
7	19	4.8809	0.	0.	0.	2	0.
8	17	4.8910	0.	0.	0.	2	0.

- Move the cursor to highlight the **LENGTH** of the tool to be set
- Press the **TOOL OFFSET MESUR**  button to record the current position into the tool offset
- Using the number pad, type the diameter of the pin we use **(-0.625)** and press **WRITE/ENTER**



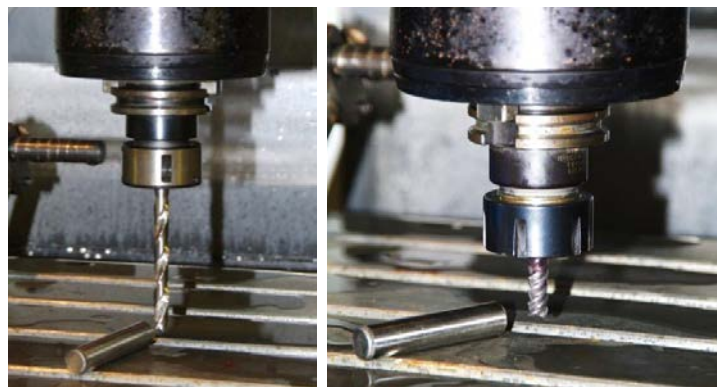
TOOL	POSITION	GEOMETRY	WEAR	GEOMETRY	WEAR
1	23	-22.8716	0.	0.	0.
2	20	3.3200	0.	0.	0.



- The number should update and the set for this tool is completed

TOOL	POSITION	GEOMETRY	WEAR	GEOMETRY	WEAR
1	23	-23.4966	0.	0.	0.
2	20	3.3200	0.	0.	0.

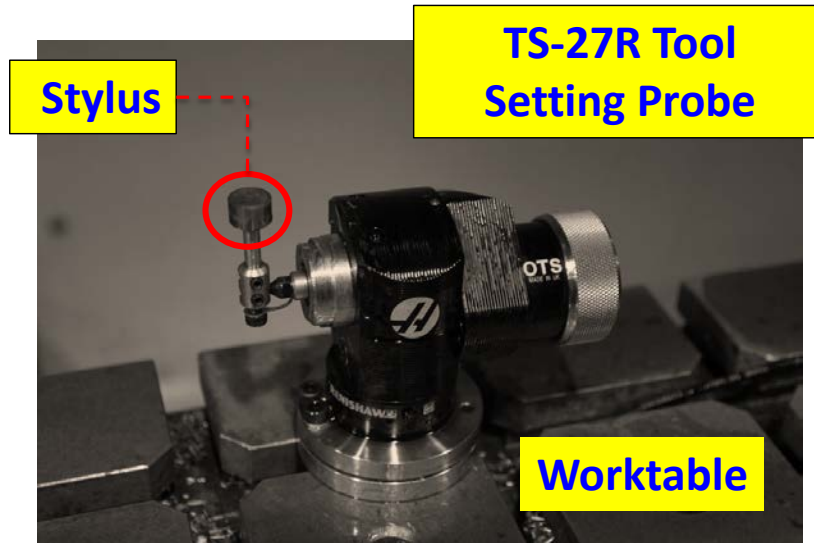
- Repeat this steps for all the tools involved in the operation



*. If the tools are probed in this way, the work offset needs to be set accordingly. Refer to the [Appendix G Mill Work Offsets Set Work Offset on a Mill with Gauge Block or Indicator \(Z\)](#) for more details.

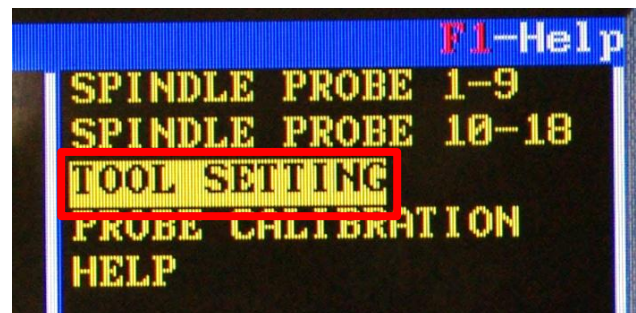
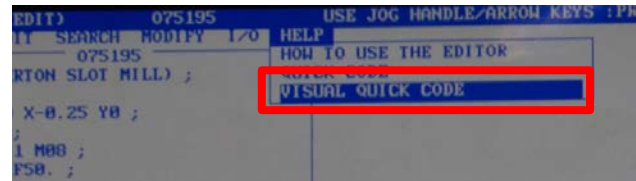
Set Tool Offset on a Mill with the Tool Setting Probe

The geometry of the tool needs to be known by the machine tool via “Tool Offset” for the machining operations to be performed properly. In this section, we learn how to set Tool Offset with the Tool Setting Probe.



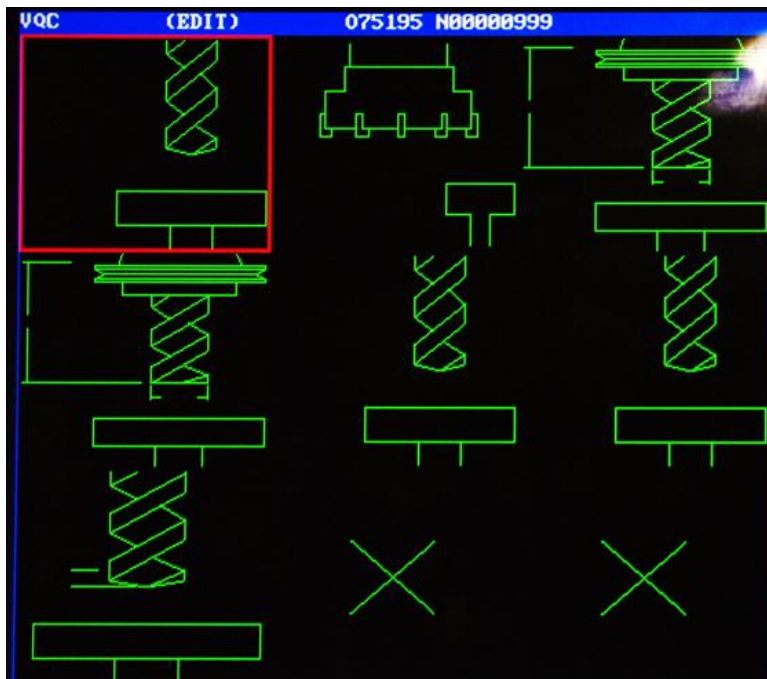
To utilize this tool setting probe, a procedure of selecting proper “Visual Quick Codes” needs to be followed.

- Press **EDIT**, use **F1** and cursors to highlight the **HELP** tab
- Use cursors to highlight the “**VISUAL QUICK CODE**” and press **WRITE/ENTER**
- User cursors to select “**TOOL SETTING**” on the upcoming screen and press **WRITE/ENTER**

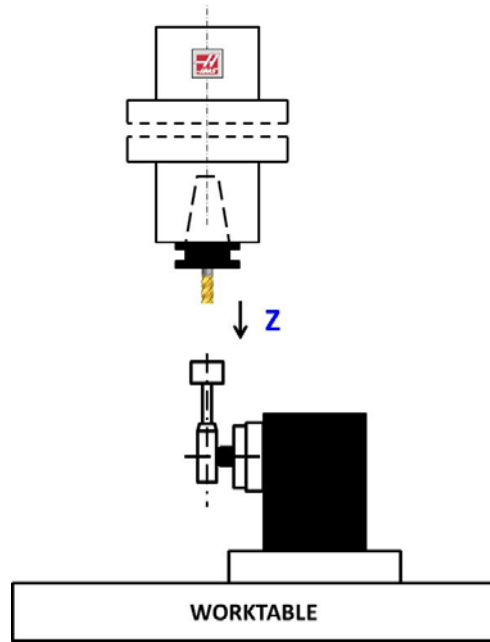


The following image should then appear on the screen, including 7 programmable sub-routines for tool probing.

- [Auto Length Only](#)
- [Auto Length Only Sequential Tools](#)
- [Auto Length Only Random Tools](#)
- [Automatic Length Rotating](#)
- [Manual Length Only](#)
- [Tool Breakage](#)
- [Automatic Length and Diameter](#)



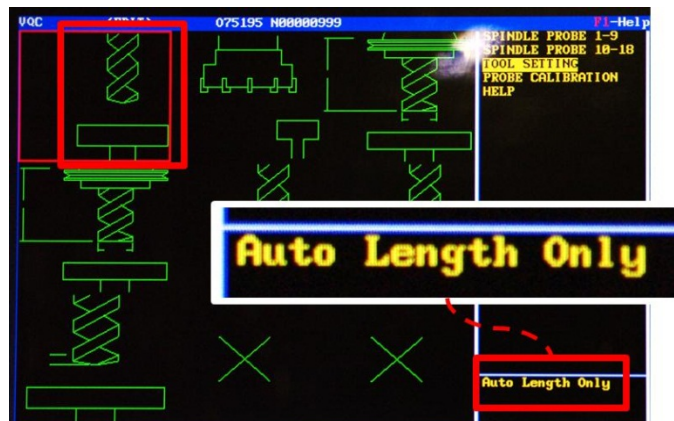
During the probing of the tool length, the machine tool will usually drive the spindle in the **Z**-direction so the tip of the tool approaches and eventually comes into contact with the top surface of the stylus.



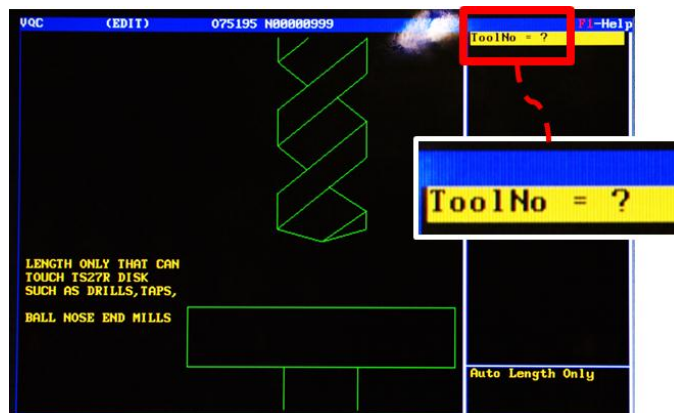
Auto Length Only

This sub-routine automatically measures the **LENGTH** of one tool, the tip of which needs to be able to touch the top of the stylus, like drills, taps, end mills, etc.

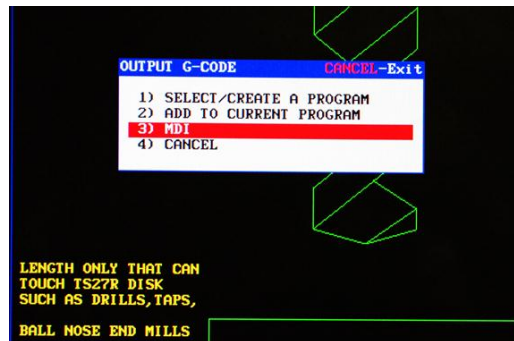
- On the screen where all of the sub-routines for tool setting are displayed, use cursors to highlight **Auto Length Only** option and press **WRITE/ENTER**



- Type in tool number as screen prompts and press **WRITE/ENTER**



- Output G-Code by moving cursors to select **MDI** and press **WRITE/ENTER**



- Press **CYCLE START** to execute the measurement

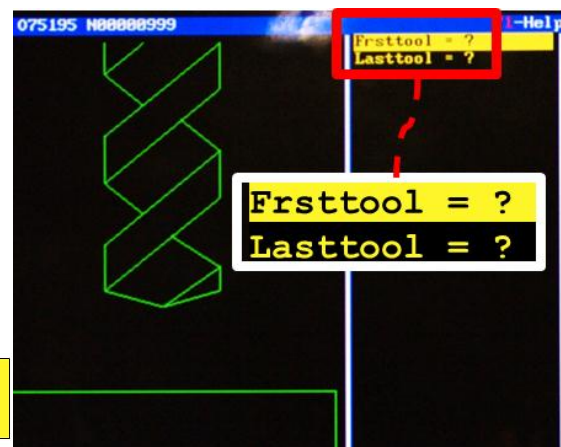
Auto Length Only Sequential Tools

This sub-routine measures the **LENGTH** of a series of tools in a sequential order.

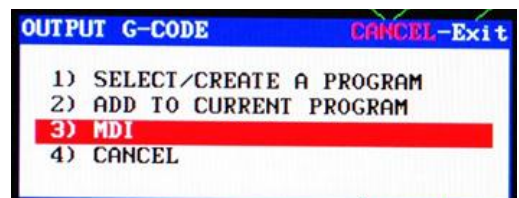
- On the screen where all of the sub-routines for tool setting are displayed, use cursors to highlight **Auto Length Only Sequential Tools** option and press **WRITE/ENTER**

- Type in the tools number of the first tool and last tool to be measured as screen prompts and press **WRITE/ENTER**

e.g. `Frsttool = 1`
`Lasttool = 5` → Probe Tool 1 through Tool 5



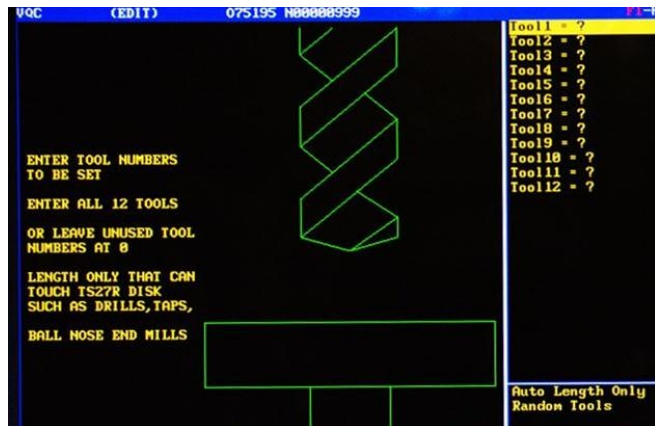
- Use cursors to select **MDI**, press **WRITE/ENTER** and press **CYCLE START** to execute the program



Auto Length Only Random Tools

This sub-routine measures the **LENGTH** of tools that are specifically specified.

- On the screen where all of the sub-routines for tool setting are displayed, use cursors to highlight **Auto Length Only Random Tools** option and press **WRITE/ENTER**
- Type in the number of the tools that needs to be probed and Press **WRITE/ENTER** to confirm
- The rest of the questions marks needs to be replaced with **ZEROS**



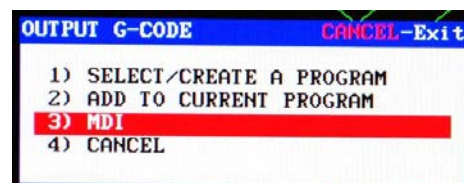
For example:

If the Tool 4 and Tool 6 are the tools to be probed, the input should be:

```
Tool1 = 4
Tool2 = 6
Tool3 = 0
Tool4 = 0
Tool5 = 0
Tool6 = 0
Tool7 = 0
Tool8 = 0
Tool9 = 0
Tool10 = 0
Tool11 = 0
Tool12 = 0
```

Please note all the spaces need to be filled.

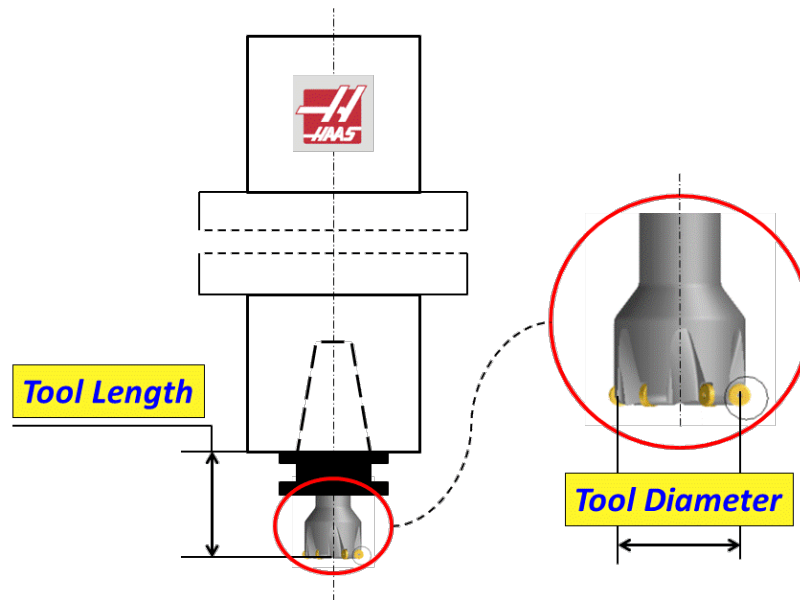
- Use cursors to select **MDI**, press **WRITE/ENTER** and press **CYCLE START** to execute the program



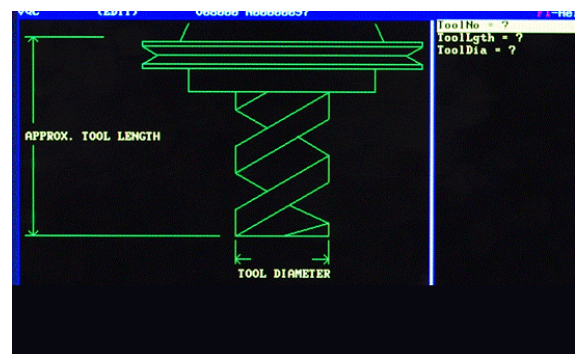
Automatic Length Rotating

This sub-routine measures the **LENGTH** of a non-center-cutting tool, the inserts of which are typically distributed on a circle.

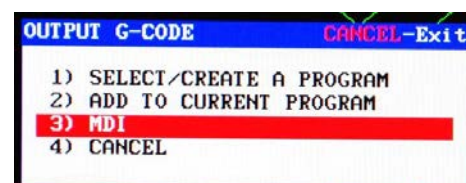
The rough measurement of the “tool length” and “tool diameter” are required in setting up this sub-routine.



- On the screen where all of the sub-routines for tool setting are displayed, use cursors to highlight **Automatic Length Rotating** option and press **WRITE/ENTER**
- Type in the tool number, length and diameter as prompted to do so, press **WRITE/ENTER** to confirm the input



- Use cursors to select **MDI**, press **WRITE/ENTER** and press **CYCLE START** to execute the program

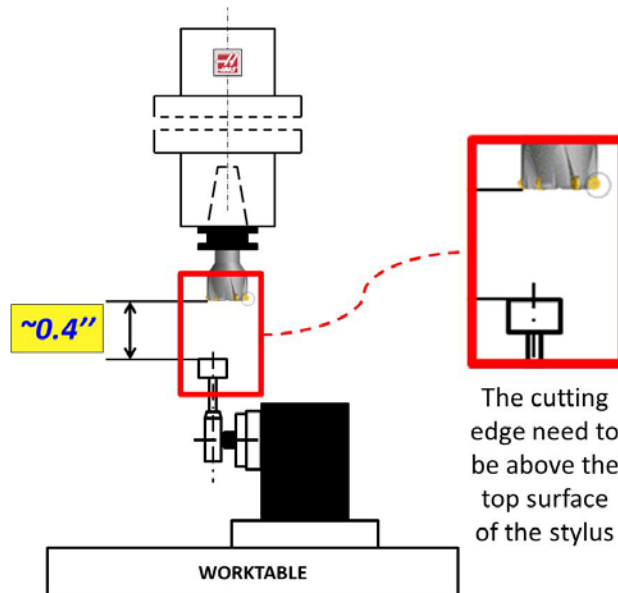


Manual Length

This sub-routine measures the **LENGTH** of a tool with more manual operation than other sub-routines.

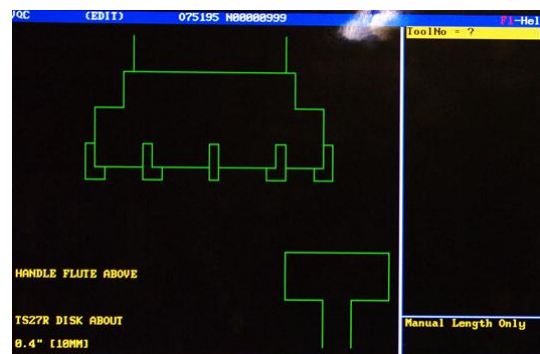
The operator needs to manually jog the spindle so the tip of the tool is slightly above the top of the stylus, and the cutting edges on the tool needs to be aligned with the stylus on the tool setting probe.

- **HANDLE JOG** the tool towards the tool setting probe

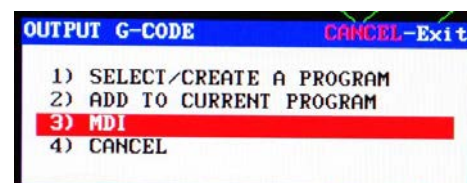


- On the screen where all of the sub-routines for tool setting are displayed, use cursors to highlight **Manual Length** option and press **WRITE/ENTER**

- Type in the too number as prompted



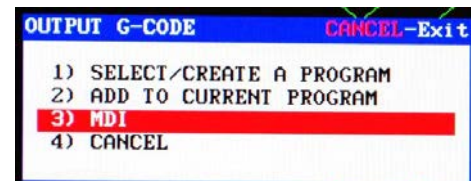
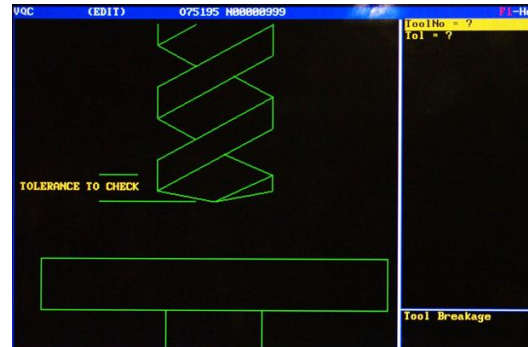
- Use cursors to select **MDI**, press **WRITE/ENTER** and press **CYCLE START** to execute the program



Tool Breakage

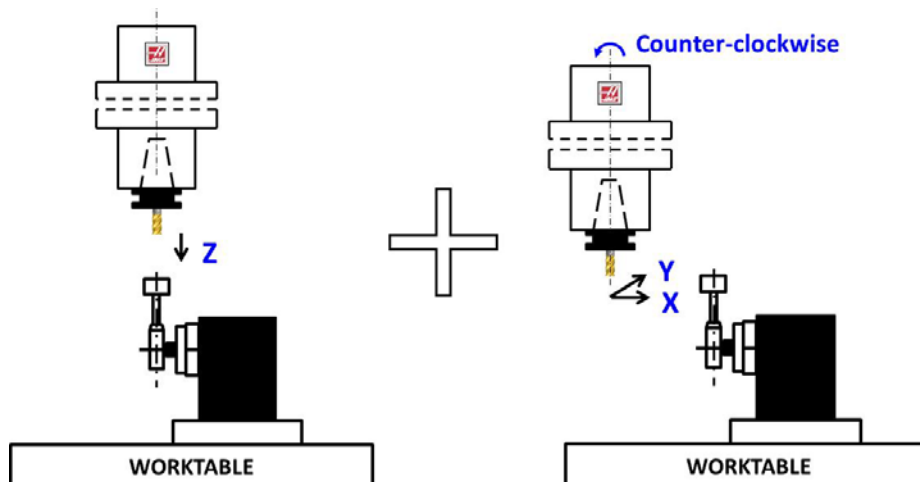
This subroutine measures the **WEAR** of a tool in **LENGTH**.

- On the screen where all of the sub-routines for tool setting are displayed, use cursors to highlight **Tool Breakage** option and press **WRITE/ENTER**
- Type in the tool number and the tolerance as prompted, press **WRITE/ENTER** to confirm
- Use cursors to select **MDI**, press **WRITE/ENTER** and press **CYCLE START** to execute the program

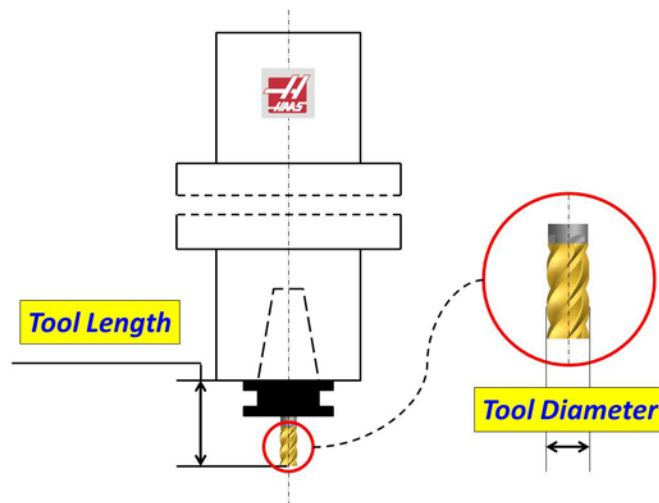


Automatic Length and Diameter

This subroutine measures the **LENGTH** and **DIAMETER** of a tool with the tool setting probe. The tool touches the top of the stylus to measure the diameter; then it moves downwards while rotating in the reverse direction followed by moving in the horizontal plane to touch the side of the stylus to measure the diameter.

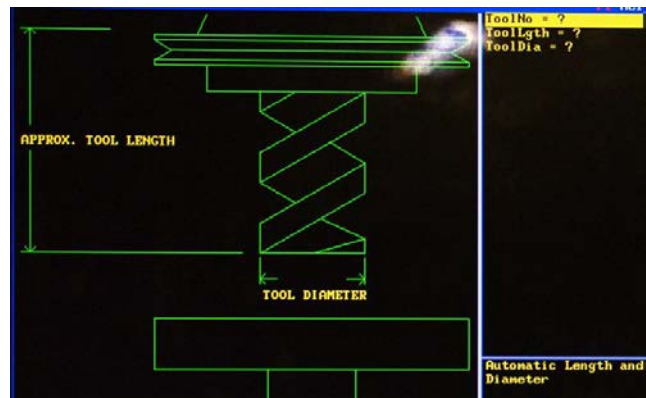


The rough measurement of the “tool length” and “tool diameter” are required in setting up this sub-routine.

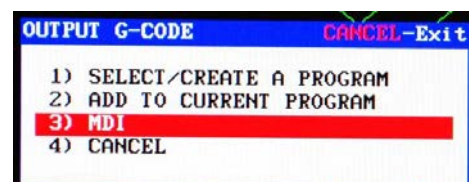


- On the screen where all of the sub-routines for tool setting are displayed, use cursors to highlight **Automatic Length and Diameter** option and press **WRITE/ENTER**

- Type in the tool number, length and diameter as prompted to do so, press **WRITE/ENTER** to confirm

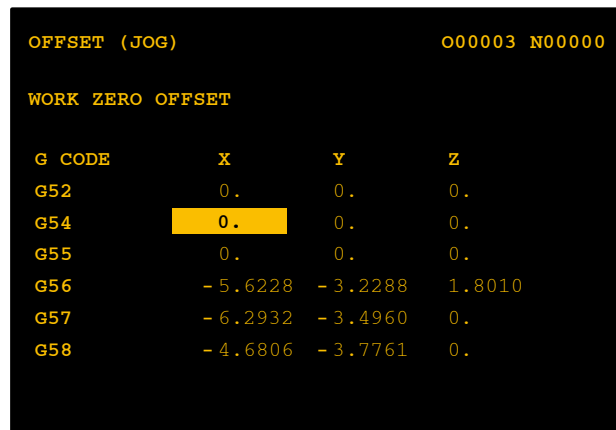


- Use cursors to select **MDI**, press **WRITE/ENTER** and press **CYCLE START** to execute the program



Appendix G Mill Work Offsets

On a mill, all parameters in the Work Zero Offset need to be defined for the machining operations to perform correctly. On a 3 axes system, X, Y, and Z need to be defined.

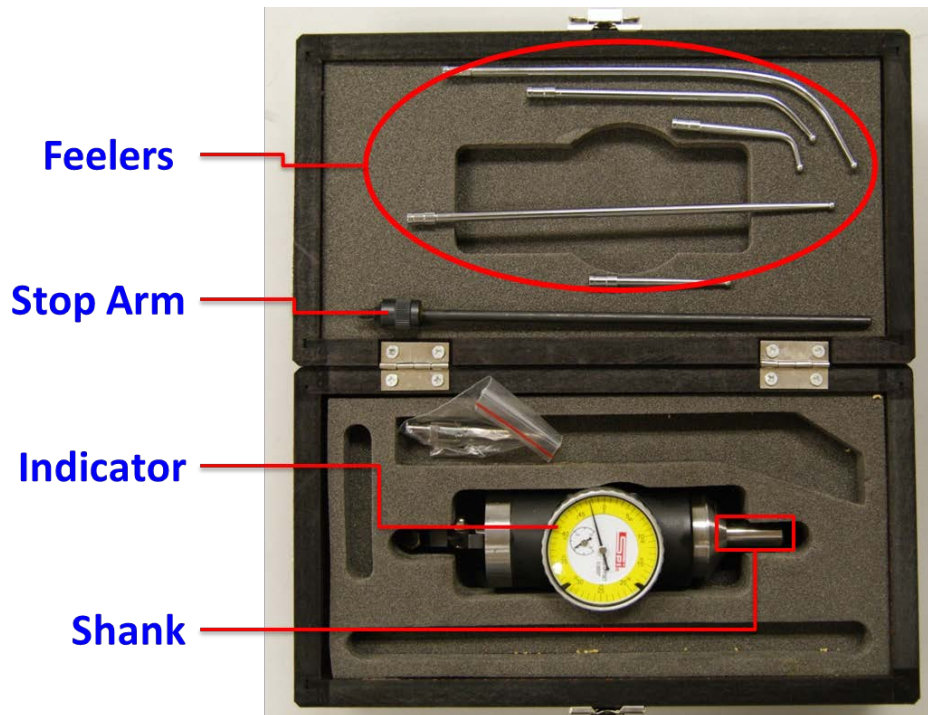


G CODE	X	Y	Z
G52	0.	0.	0.
G54	0.	0.	0.
G55	0.	0.	0.
G56	-5.6228	-3.2288	1.8010
G57	-6.2932	-3.4960	0.
G58	-4.6806	-3.7761	0.

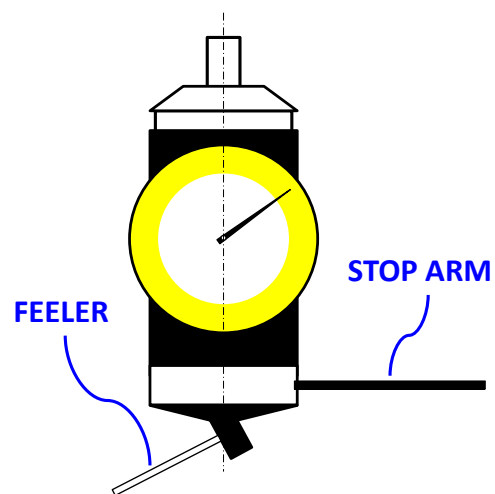
- [Set Work Offset on a Mill with a Coaxial Indicator \(X and Y\)](#)
- [Set Work Offset on a Mill with Dial Indicator \(X and Y\)](#)
- [Set Work Offset on a Mill with an Edge Finder \(X and Y\)](#)
- [Set Work Offset on a Mill with Gauge Block or Indicator \(Z\)](#)
- [Set Work Offset on a Mill with the Spindle Probe](#)

Set Work Offset (X and Y) on a Mill with a Coaxial Indicator

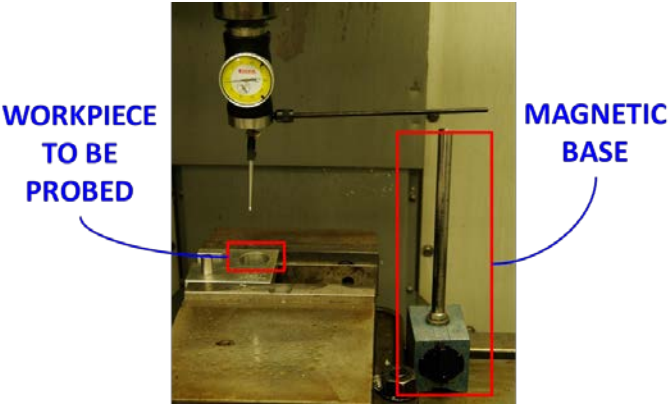
Co-axial indicator is a tool to find the center of inner or outer cylindrical surfaces. In this lab, we use it to find X and Y work offset on a mill.



- Assemble the stop arm and feeler to the indicator
 - Install the stop arm
 - Pick the proper feeler depending on the diameter of the bore or boss to be probed
 - Hand tight the feeler to the indicator



- Assemble the co-axial indicator to a tool holder and install it to the machine tool
 - The shank goes into a proper size collet (3/8" dia.)
- Attach a magnetic base to the table where the stop arm can apply to
 - So the indicator can remain stationary and face the operator

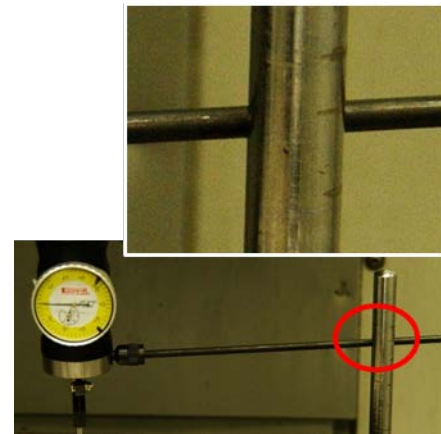


- Handle Jog the feeler near the feature to be probed, try roughly align the center of the spindle with that of the bore/boss

- Gently push or pull the feeler while jogging down so that the feeler shall touch the probing surface upon release




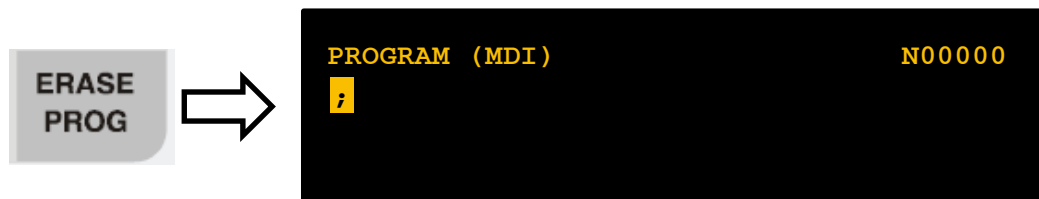
- Touch the magnetic base with the stop arm so the indicator remains stationary while the spindle turns



- Gently turn the spindle **BY HAND** for a whole revolution to make sure the feeler moves freely

- Start the spindle (50~100 RPM)

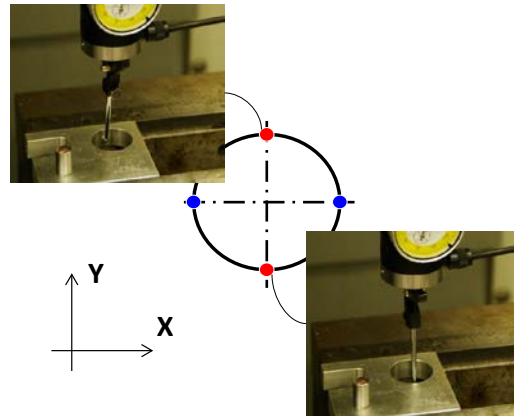
- Go to **MDI** mode 
- Press **ERASE PROG** until there is no existing lines on the screen



- Specify spindle RPM
e.g Type **“S60”** and press **“WRITE/ENTER”** for 60 RPM
- Close the door
- Press **“CYCLE START”**
- Press the **“CW”** (clockwise) or **“CCW”** (counter clockwise) button to start the spindle



- Observe the positions of the indicator dial at the four points
 - Two blue points – two utmost positions along the X axis
 - Two red points – two utmost positions along the Y axis

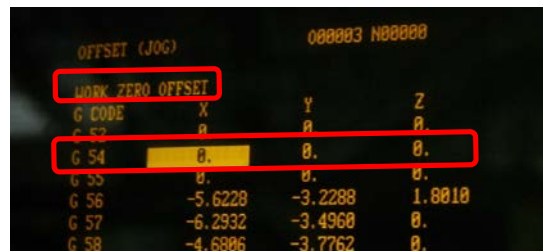


- Handle Jog

- Use **0.001"** increment .001
1.
- Along only in X/Y direction so the indicator points to the middle of the two utmost positions along this direction
- Jog the other direction so the indicator movement is minimized
- Switch to **0.0001"** increment .0001
.1
- Minimize the indicator dial movement

- Cycle **OFFSET** button until the **WORK ZERO OFFSET** is displayed

- Move your cursor to highlight the offset value to be changed



- Press **PART ZEROS SET** button to set the offset value for **X** and then **Y**



Set Work Offset (X and Y) on a Mill with Dial Indicator

A dial indicator held in the spindle can be used to find center of a round part.

- Setup Dial indicator as shown
- Indicator holder has $\frac{1}{4}$ " diameter post that must be held in either a collet holder or drill chuck
- Position indicator roughly over the center of the part. Rotate the spindle by hand to visually inspect positioning
- Lower the indicator down so that it can touch the side of the part
- Rotate the spindle by hand to ensure the indicator maintains contact with the part the whole way around
- Move X-position so that indicator reads the same number on both sides



- Once X-values are matching, move the position in Y to match that value as well.



Front View



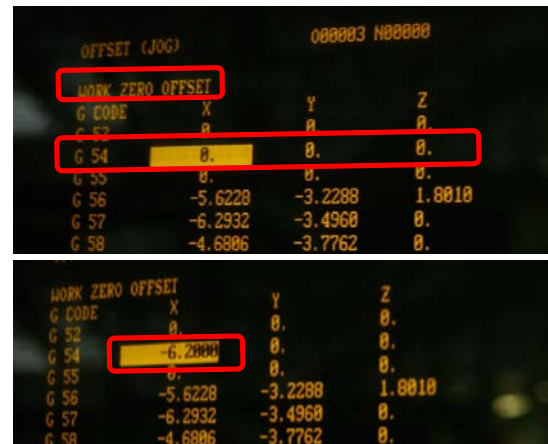
Side View

- Cycle **OFFSET** button until the **WORK ZERO OFFSET** is displayed

- Move your cursor to highlight the offset value to be changed

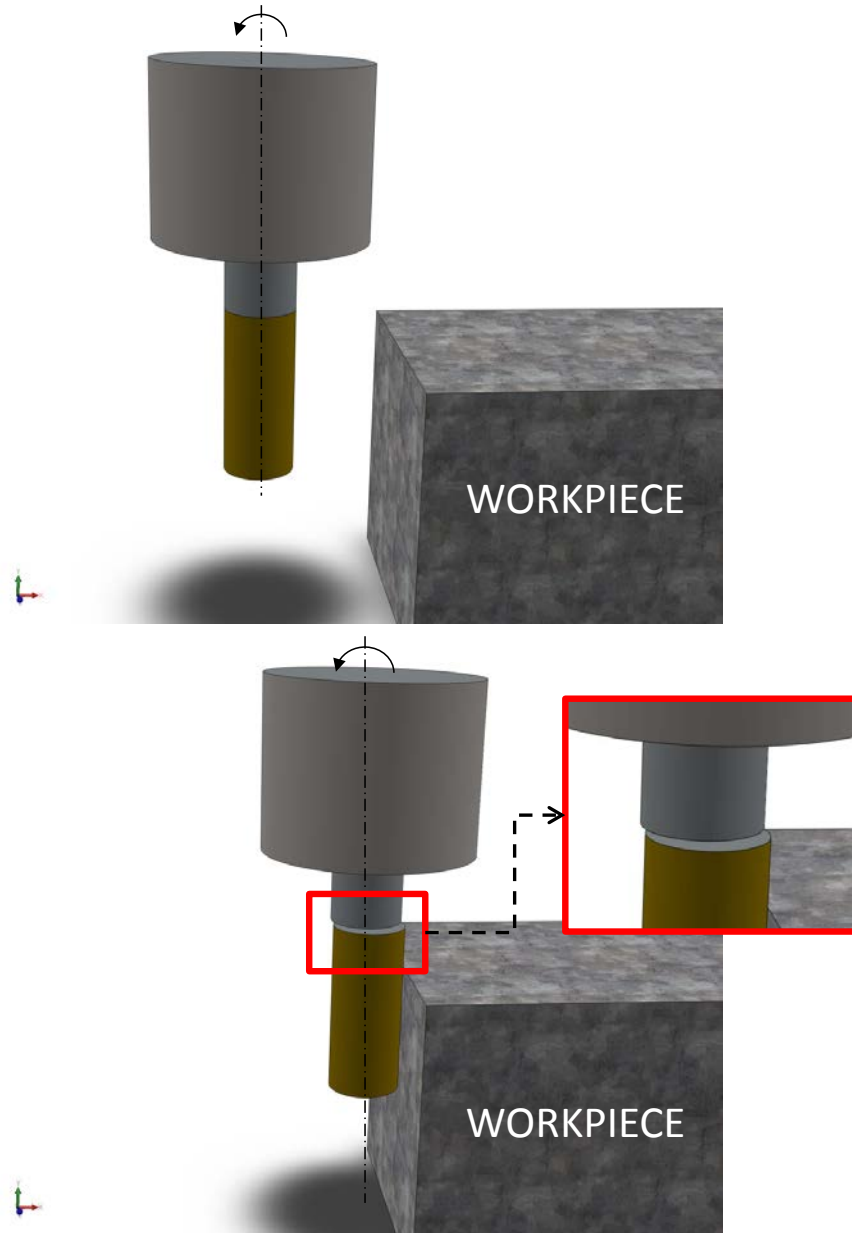
- Press **PART ZEROS SET** button to set the offset value for **X** and then **Y**

**PART
ZERO
SET**



Set Work Offset (X and Y) on a Mill with an Edge Finder

Edge finder is a spring loaded rotating tool that a kickoff of the cylinder occurred due to friction when it touches an edge. In this lab, we use it to find the X and Y work offset on a mill.



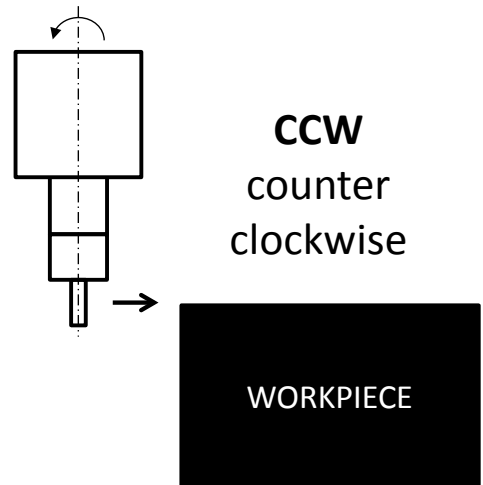
As the rotating edge finder touches the surface waiting to be probed, the friction causes a cylindrical kick off on the lower part of the edge finder, mainly along the tangential direction of the contact.

- Assemble the edge finder to a tool holder and install it to the machine tool



- Determine the “edge to be found”

- Start the spindle (<750 RPM)
 - When probing the left edge, use **CCW** (counter clockwise) so it breaks towards you
 - Otherwise, use **CW** (clockwise)



- Jog the edge finder near the “edge”



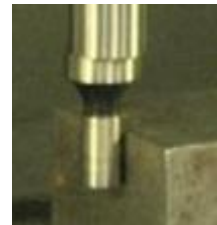
- Slowly Handle Jog the edge finder towards the workpiece



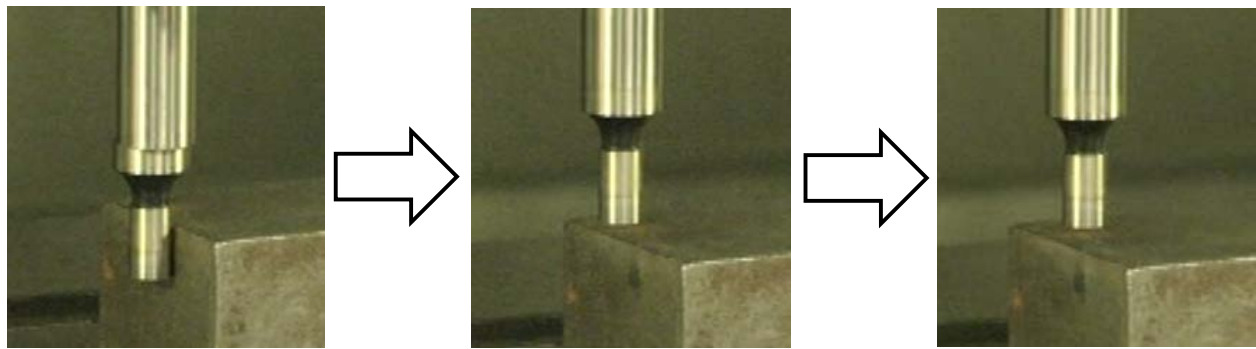
- Once edge starts to break, jog it away from the workpiece and select a smaller increment


.0001	.001	.01	.1
.1	1.	10.	100.

- Repeat the last two steps until you find the location where the edge JUST start to break



- Jog the edge finder UP to clear the workpiece and move towards the edge by the RADIUS of the edge finder



- Press **OFFSET** button  until the **WORK ZERO OFFSET** is displayed

- Move your cursor to highlight the offset value to be changed

OFFSET (JOG)		000003 N00000		
WORK ZERO OFFSET				
G CODE	X	Y	Z	
G 52	0.	0.	0.	
G 54	0.	0.	0.	
G 55	0.	0.	0.	
G 56	-5.6228	-3.2288	1.8818	
G 57	-6.2932	-3.4968	0.	
G 58	-4.6886	-3.7762	0.	

- Press **PART ZEROS SET** to set the offset value that has just been probed



WORK ZERO OFFSET				
G CODE	X	Y	Z	
G 52	0.	0.	0.	
G 54	-6.2880	0.	0.	
G 55	0.	0.	0.	
G 56	-5.6228	-3.2288	1.8818	
G 57	-6.2932	-3.4968	0.	
G 58	-4.6886	-3.7762	0.	

- Repeat the previous steps until all of the required values (X, Y, Z) for the specified offset are fully defined

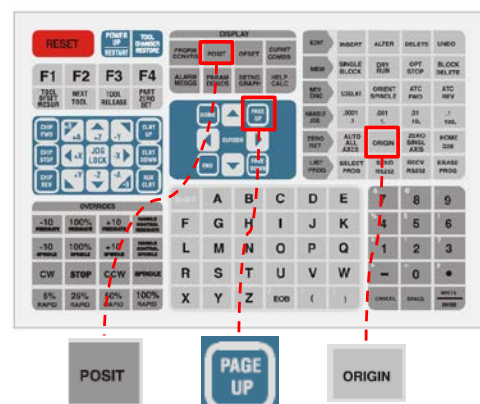
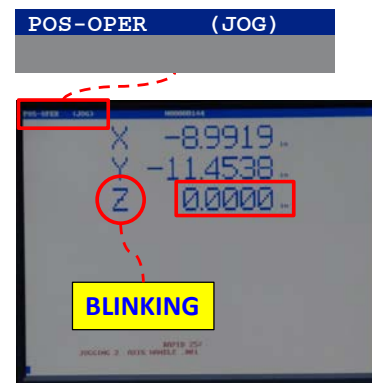
Set Work Offset (Z) on a Mill with Gauge Block or Dial Indicator

If the Tool Offset has been completed using gauge block, the Work Offset should be set with either one of those previously set tools + gauge block, or indicator.

The objective in this section in this step is to find the offset in referring to the reference surface that is used to set all the tool offsets.

Set Work Offset on a Mill with a Gauge Block

- Set the tools with the common reference surface (worktable) like described in [Appendix F Mill Tool Offsets](#) to find the position where the gauge block just fit into the gap between the tool tip and the reference surface
- Let the tool stay in that position
- Press the **POSIT** button once and cycle the **PAGE UP** button until the **POS-OPER** (Position – Operator) shows up on the upper left corner of the screen
- Make sure the Z is blinking
If not, press **HANDLE JOG** button and select Z axis then repeat the last step
- Press the **ORIGIN** button so the values for Z for the **Operator's Coordinate** is set to **0.0000''**



- Handle Jog

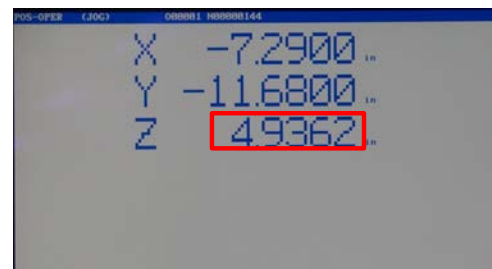
- Move the previously set tool over the workpiece



- Using the techniques similarly when setting the Tool Offset using the gauge block, locate the position where the gauge block just fits into the gap between the tool tip and the workpiece (the surface that is set as the Z origin in your program)



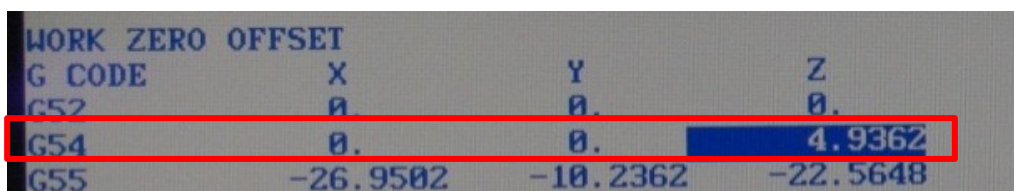
CAUTION: NEVER HANDLE JOG THE TOOL INTO THE GAUGE BLOCK.



- Record the value on the screen for Z

- Set the Work Offset in the Controller

- Cycle **OFFSET** button until the **WORK ZERO OFFSET** is displayed
- Move your cursor to highlight the offset value to be changed
- Use the number pad to type in proper numbers and press **WRITE/ENTER** (In this example, it is 4.9362".)



Set Work Offset (Z) on a Mill with Dial Indicator

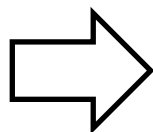
- Install the indicator
 - Install the indicator to a magnetic base
 - Attach the magnetic base to a clean smooth surface on the spindle head



- Handle Jog the indicator gently into the common reference surface (the worktable in this document) until the stylus starts to move for roughly half a revolution

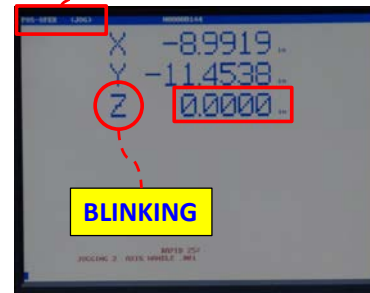


- Rotate the outer ring to set the dial to zero

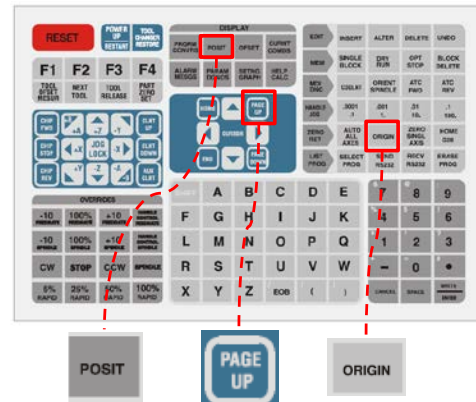


- Press the **POSIT** button once and cycle the **PAGE UP** button until the **POS-OPER** (Position – Operator) shows up on the upper left corner of the screen

POS-OPER (JOG)



- Make sure the Z is blinking
If not, press **HANDLE JOG** button and select Z axis then repeat the last step
- Press the **ORIGIN** button so the values for Z for the **Operator's Coordinate** is set to **0.0000''**



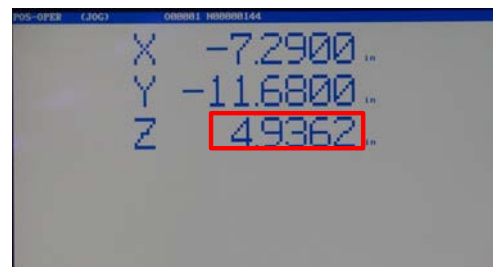
- Handle Jog the indicator over the workpiece (the surface that is set as the Z origin in your program)
- Handle Jog the indicator down gently as the stylus touches the workpiece
- Reduce the increment to **0.0001''** as the dial approaches zero



- Handle Jog carefully so that the dial points at zero



- Record the value on the screen for Z



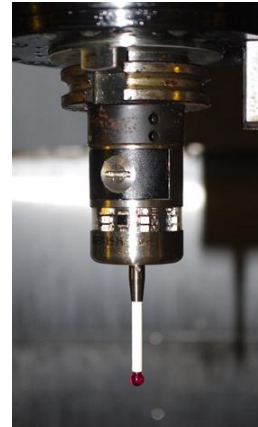
- Set the Work Offset
 - Cycle **OFFSET** button until the **WORK ZERO OFFSET** is displayed
 - Move your cursor to highlight the offset value to be changed
 - Use the number pad to type in proper numbers and press **F1** to replace the original number (In this example, it is 4.9362".)

WORK ZERO OFFSET			
G CODE	X	Y	Z
G52	0.	0.	0.
G54	0.	0.	4.9362
G55	-26.9502	-10.2362	-22.5648

Set Work Offset on a Mill with the Spindle Probe

The spindle probe is an equipment to measure the workpiece in the mill.

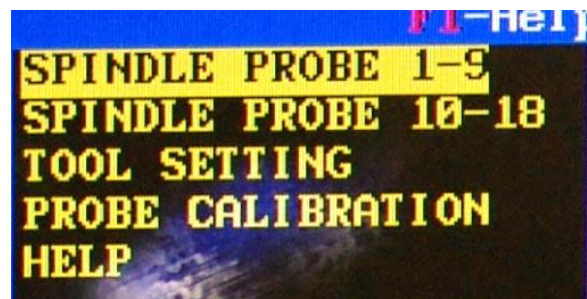
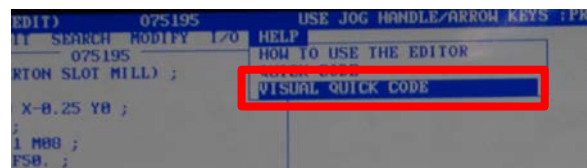
At the Washburn Shops, the spindle probes are typically installed as **Tool 10** on MiniMills, or **Tool 25** on VF4 and VM3.



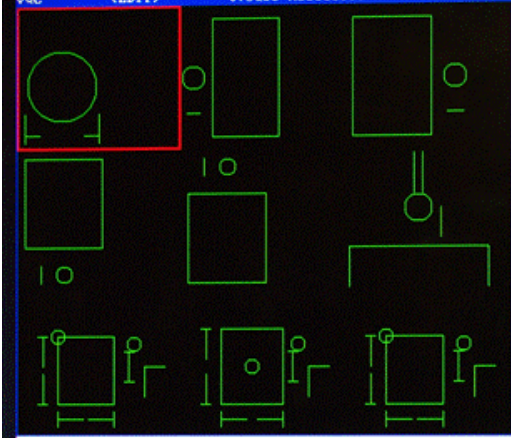
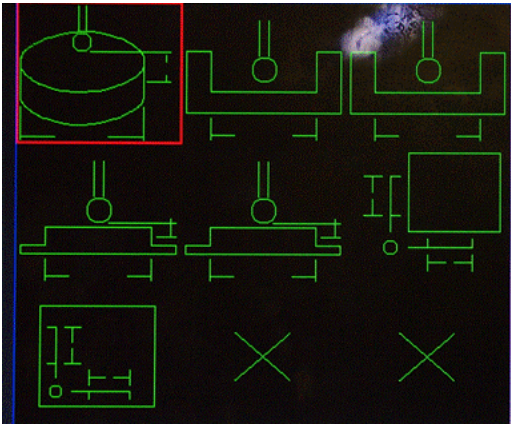
Usually, the operator needs to **HANDLE JOG** to probe in a reasonable proximity of a designated area near the workpiece to be probed. The detailed location is determined by the subroutines.

To utilize this spindle probe, a procedure of selecting proper “**Visual Quick Codes**” needs to be followed.

- Press **EDIT**, use **F1** and cursors to highlight the **HELP** tab
- Use cursors to highlight the “**VISUAL QUICK CODE**” and press **WRITE/ENTER**
- User cursors to select “**SPINDLE PROBE 1-9**” or “**SPINDLE PROBE 10-18**” on the upcoming screen and press **WRITE/ENTER**



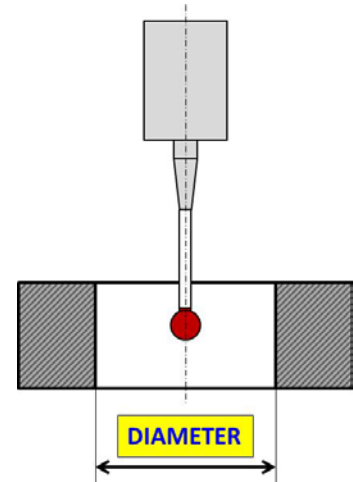
Under SPINDLE PROBE 1-9, there are 9 programmable subroutines. And under SPINDLE PROBE 10-18, there are 7 additional programmable subroutines.

	OBJECTIVE	SUBROUTINES
<p align="center">SPINDLE PROBE 1-9</p> 	<u>Center of Bore</u>	Probe Bore
	<u>Edge</u>	Probe X Plus Surface
		Probe X Minus Surface
		Probe Y Plus Surface
		Probe Y Minus Surface
	<u>Probe Z Surface</u>	
	<u>Probe Vise Corner</u>	
	<u>Center of Block</u>	Probe Center of Block Start from Center
		Probe Center of Block Start from Corner
<p align="center">SPINDLE PROBE 10-18</p> 	<u>Center of Boss</u>	Probe Boss
	<u>Center of Pocket</u>	Probe X Pocket
		Probe Y Pocket
	<u>Center of Web</u>	Probe X Web
		Probe Y Web
	<u>Corners</u>	Probe External Corner
		Probe Internal Corner

Find the Center of a Bore

This section illustrates how to find the center of a bore using the spindle probe.

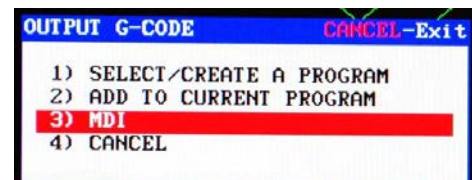
- Roughly measure the diameter of the bore to be probed
- Handle Jog the spindle probe approximately to the center of the bore.
- On the screen where all of the sub-routines for **SPINDLE PROBE 1-9** are displayed, use cursors to highlight **Probe Bore** option and press **WRITE/ENTER**



- Set up the subroutine:
 - As prompted, type in the register number for the **Work Zero Offset** (typically, 54 for G54); and the measurement / estimation for the diameter
 - Press **WRITE/ENTER** to confirm, or **CANCEL** to modify



- Use cursors to select **MDI**, press **WRITE/ENTER** and press **CYCLE START** to execute the program

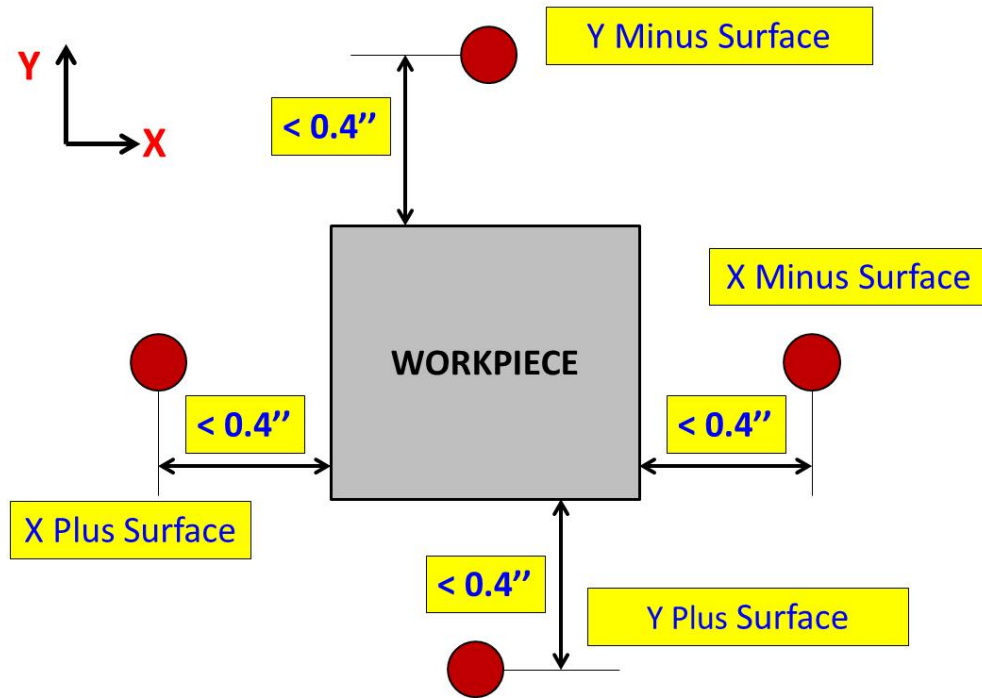


The Work Zero Offset number for X and Y coordinate of the designated register will change automatically when the subroutine completes.

Find an Edge

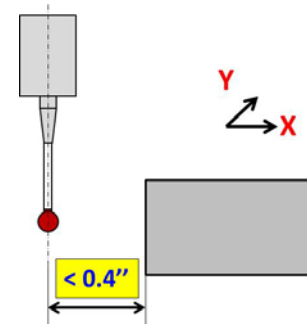
This section illustrates how to find the edge of a part using the spindle probe. There are 4 sub-routines to choose from to find an edge.

The operator needs to handle jog the probe to certain positions in order to perform the edge probing.



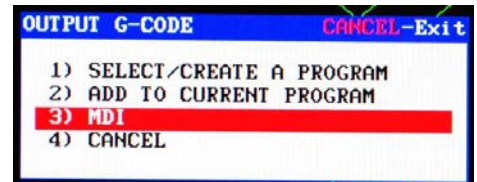
The subroutine **Probe X Plus Surface** is illustrated as an example.

- Handle Jog the spindle probe to the left side of the workpiece, the distance between center the probe stylus and the edge to be probed should be within 0.4''



- On the screen where all of the sub-routines for **SPINDLE PROBE 1-9** are displayed, use cursors to highlight **Probe X Plus Surface** option and press **WRITE/ENTER**

- Set up the subroutine:
 - As prompted, type in the register number for the **Work Zero Offset** (typically, 54 for G54);
 - Press **WRITE/ENTER** to confirm, or **CANCEL** to modify
- Use cursors to select **MDI**, press **WRITE/ENTER** and press **CYCLE START** to execute the program

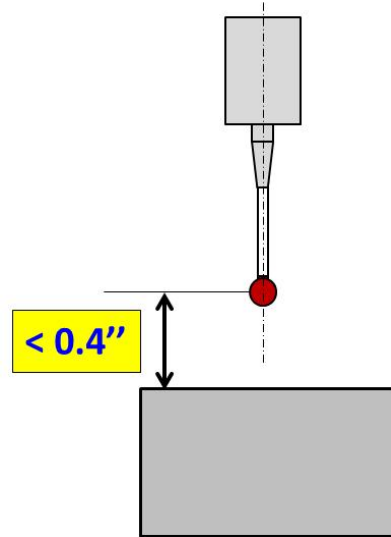


The Work Zero Offset number for X coordinate (if chose to Probe X Plus Surface or Probe X Minus Surface) for the designated register will change automatically when the subroutine completes.

Find the Z Surface

This section illustrates how to find the Z surface of a part using the spindle probe.

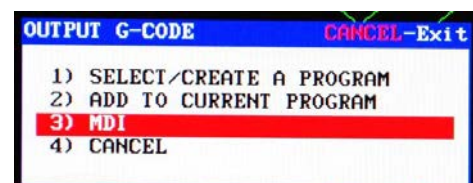
- Handle Jog the spindle probe to the left side of the workpiece, the distance between center the probe stylus and the edge to be probed should be within 0.4"
- On the screen where all of the sub-routines for **SPINDLE PROBE 1-9** are displayed, use cursors to highlight **Probe Z Surface** option and press **WRITE/ENTER**



- Set up the subroutine:
 - As prompted, type in the register number for the **Work Zero Offset** (typically, 54 for G54);
 - Press **WRITE/ENTER** to confirm, or **CANCEL** to modify



- Use cursors to select **MDI**, press **WRITE/ENTER** and press **CYCLE START** to execute the program

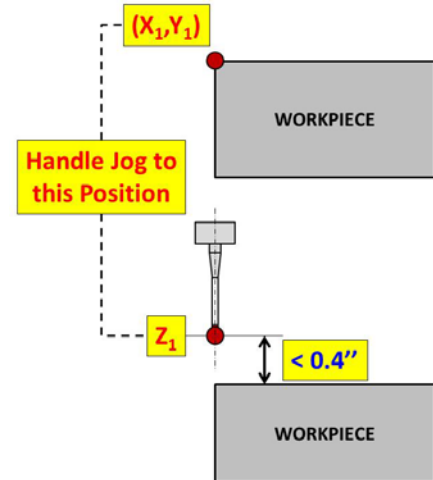


The Work Zero Offset number for Z coordinate for the designated register will change automatically when the subroutine completes.

Find the Vise Corner

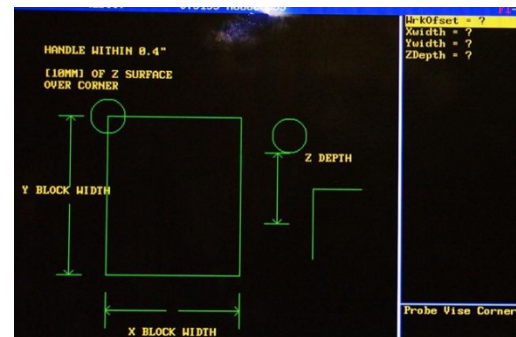
This section illustrates how to find the X, Y, and Z work coordinates of a corner all at once with the **Probe Vise Corner** subroutine.

- Handle Jog the spindle probe stylus OVER the corner of the piece to be probed (X_1, Y_1, Z_1)

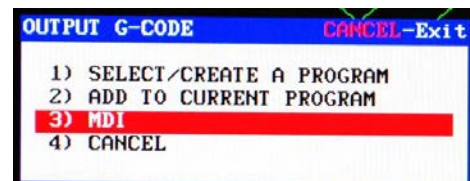


- On the screen where all of the sub-routines for **SPINDLE PROBE 1-9** are displayed, use cursors to highlight **Probe Vise Corner** option and press **WRITE/ENTER**

- Set up the Subroutine
 - As prompted, type in the register number for the **Work Zero Offset** (typically, 54 for G54); and the measurement for **X Width**, **Y Width** and **Z Depth***
 - Press **WRITE/ENTER** to confirm, or **CANCEL** to modify



- Use cursors to select **MDI**, press **WRITE/ENTER** and press **CYCLE START** to execute the program



*. Once the subroutine is set up, the spindle probe will perform the following procedure:

- a) Move horizontally to (X_2, Y_2, Z_1)

$$X_2 = X_1 + X \text{ Width}/2$$

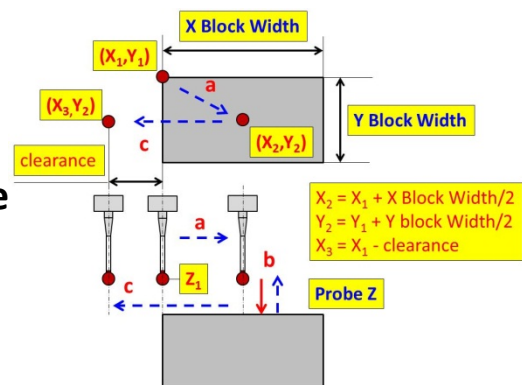
$$Y_2 = Y_1 + Y \text{ Width}/2$$

- b) Plunge down to **Probe the Z Surface** and back to Z_1

- c) Move horizontally back to (X_3, Y_2, Z_1)

$$X_3 = X_1 - \text{clearance}$$

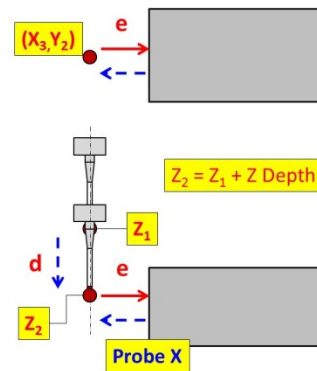
Clearance is a preset positive number in the machine.



- d) Move down to (X_3, Y_2, Z_2)

$$Z_2 = Z_1 + Z \text{ Depth}$$

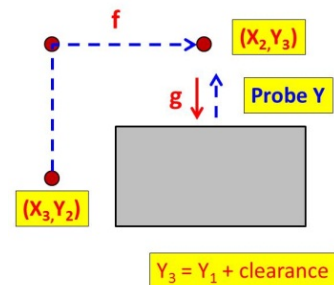
- e) **Probe the X Surface** and move horizontally back to (X_3, Y_2, Z_2)



- f) Move horizontally to (X_2, Y_3, Z_2)

$$Y_3 = Y_1 + \text{clearance}$$

- g) Probe the Y Surface and move back to (X_2, Y_3, Z_2)

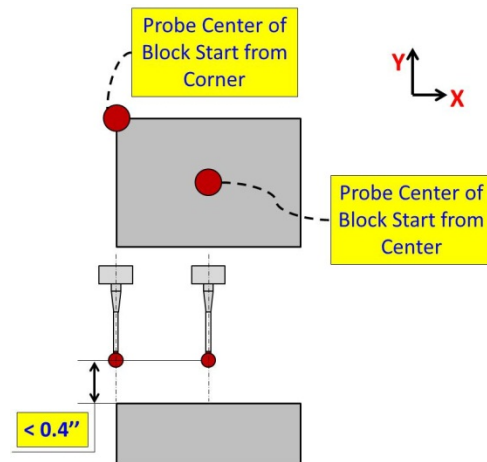


- h) The probing is completed and the X,Y,Z values are registered to the designated register

Find the Center of a Block

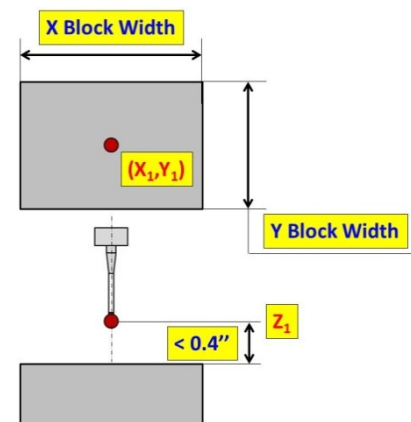
This section illustrates how to find the center of a block using the spindle probe which includes finding X, Y and Z coordinates all at once. There are 2 sub-routines to choose from.

The operator needs to handle jog the probe to certain positions in order to perform the edge probing.

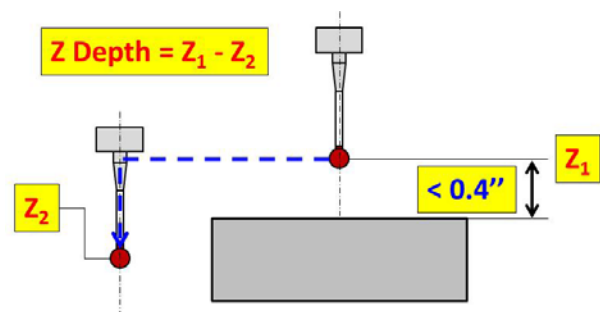


The subroutine **Probe Center of Block Start from Center** is illustrated as an example.

- Handle Jog the spindle probe approximately over the center of the block, the distance between the center of the stylus and the top surface of the part should be less than 0.4". (X_1, Y_1, Z_1) as illustrated



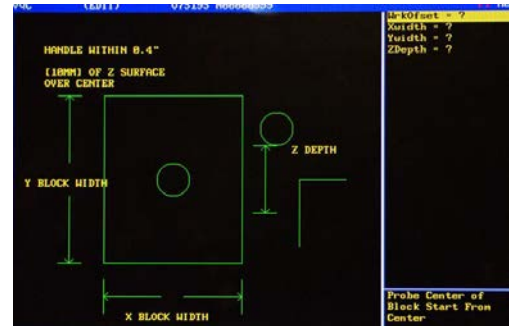
- Roughly measure the **X block width** and **Y block width** and estimate the **Z depth** by determining the proper height for **X** and **Y** probing



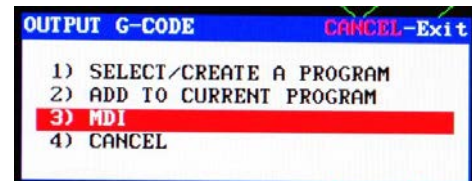
- On the screen where all of the sub-routines for **SPINDLE PROBE 1-9** are displayed, use cursors to highlight **Probe**

Center of Block Starting from Center option and press WRITE/ENTER

- Set up the Subroutine
 - As prompted, type in the register number for the **Work Zero Offset** (typically, 54 for G54); and the measurement for **X Width**, **Y Width** and **Z Depth**
 - Press **WRITE/ENTER** to confirm, or **CANCEL** to modify



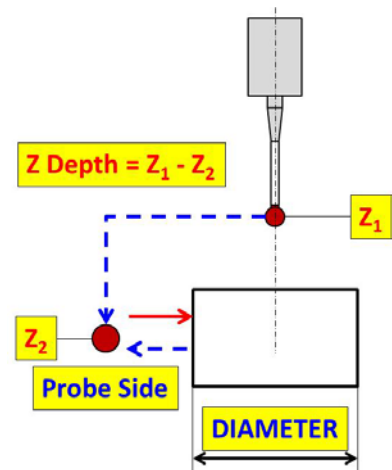
- Use cursors to select **MDI**, press **WRITE/ENTER** and press **CYCLE START** to execute the program



Find the Center of a Boss

This section illustrates how to find the center of a boss using the spindle probe.

- Handle Jog the spindle probe approximately over the center of the bore
- Roughly measure the diameter of the boss to be probed and estimate the **Z depth** by determining the proper height for X and Y probing

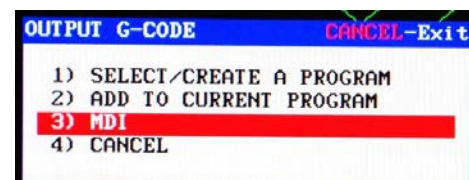


- On the screen where all of the sub-routines for **SPINDLE PROBE 10-18** are displayed, use cursors to highlight **Probe Boss** option and press **WRITE/ENTER**

- Set up the Subroutine
 - As prompted, type in the register number for the **Work Zero Offset** (typically, 54 for G54); and the measurement for **Diameter and Z Depth**
 - Press **WRITE/ENTER** to confirm, or **CANCEL** to modify



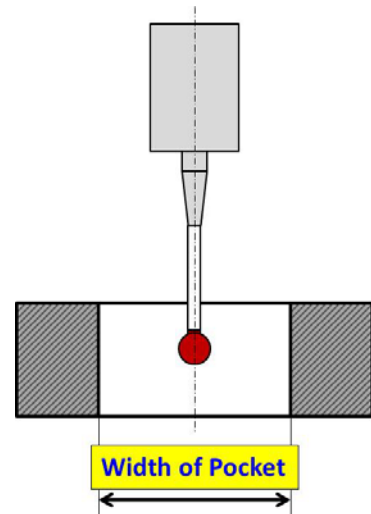
- Use cursors to select **MDI**, press **WRITE/ENTER** and press **CYCLE START** to execute the program



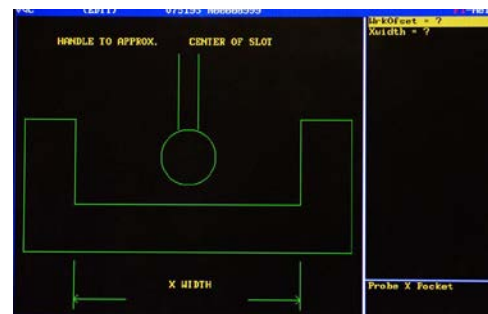
Find the Center of a Pocket

This section illustrates how to find the center of a X/Y pocket using the spindle probe. There are two subroutines to choose from.

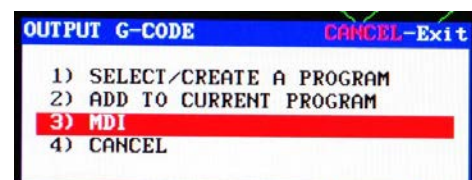
- Roughly measure the width of the pocket to be probed
- Handle Jog the spindle probe approximately to the center of pocket



- On the screen where all of the sub-routines for **SPINDLE PROBE 10-18** are displayed, use cursors to highlight **Probe X Pocket** or **Probe Y Pocket** option and press **WRITE/ENTER**
- Set up the Subroutine
 - As prompted, type in the register number for the **Work Zero Offset** (typically, 54 for G54); and the measurement for **X/Y width**
 - Press **WRITE/ENTER** to confirm, or **CANCEL** to modify



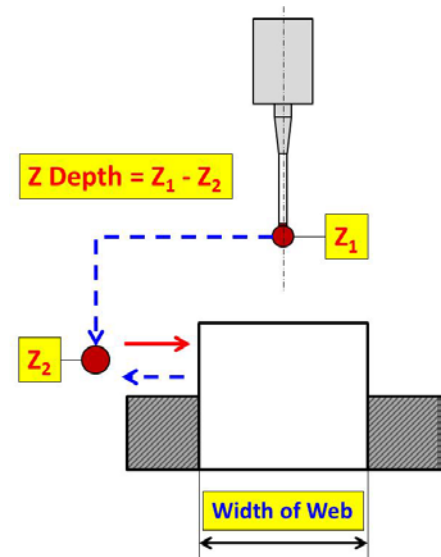
- Use cursors to select **MDI**, press **WRITE/ENTER** and press **CYCLE START** to execute the program



Probe the Center of a Web

This section illustrates how to find the center of a X/Y web using the spindle probe. There are two subroutines to choose from.

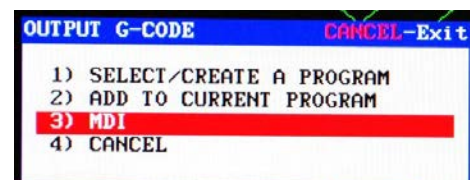
- Handle Jog the spindle probe approximately over the center of the web
- Roughly measure the width of the web to be probed and estimate the **Z depth** by determining the proper height for X and Y probing



- On the screen where all of the sub-routines for **SPINDLE PROBE 10-18** are displayed, use cursors to highlight **Probe X Web** or **Probe Y Web** option and press **WRITE/ENTER**
- Set up the Subroutine
 - As prompted, type in the register number for the **Work Zero Offset** (typically, 54 for G54); the measurement for **X/Y width** and the **Z depth**
 - Press **WRITE/ENTER** to confirm, or **CANCEL** to modify



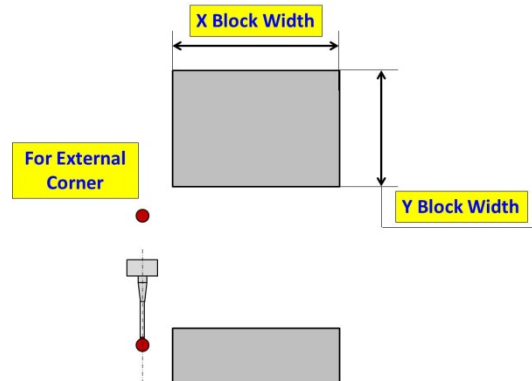
- Use cursors to select **MDI**, press **WRITE/ENTER** and press **CYCLE START** to execute the program



Probe Corners

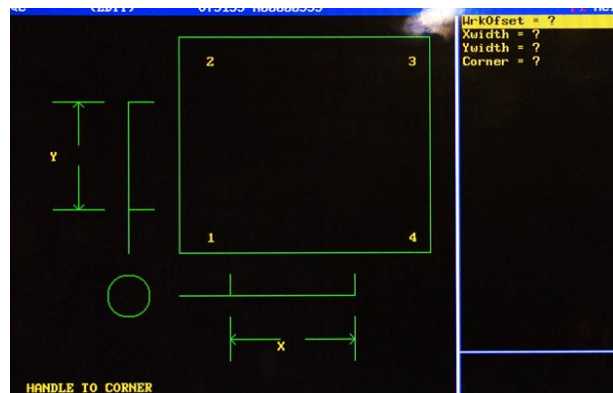
This section illustrates how to find the corner on a workpiece web using the spindle probe. There are two subroutines to choose from.

- Handle Jog the spindle probe near the corner to probe
- Roughly measure **X Width** and **Y Width** of the corner



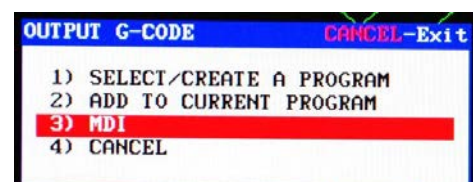
- On the screen where all of the sub-routines for **SPINDLE PROBE 10-18** are displayed, use cursors to highlight **Probe External Corner** or **Probe Internal Corner** option and press **WRITE/ENTER**

- Set up the Subroutine
 - As prompted, type in the register number for the **Work Zero Offset** (typically, 54 for G54); the measurement for **X/Y width** and the **number of the corner to probe**



- Press **WRITE/ENTER** to confirm, or **CANCEL** to modify

- Use cursors to select **MDI**, press **WRITE/ENTER** and press **CYCLE START** to execute the program



Appendix H Lathe Tool Offsets

- [Set Tool Offset on a Lathe with the Tool Setting Probe](#)
- [Set Tool Offset \(Z\) on a Lathe with the Gauge Pin](#)
- [Set Tool Offset \(X\) on a Lathe by Testing Cut](#)

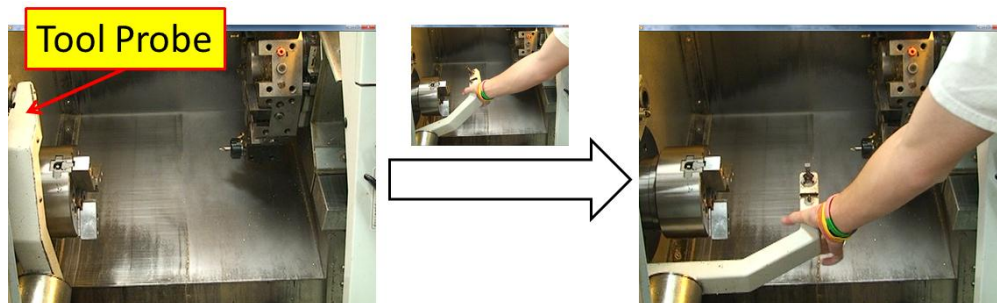
Set Tool Offset on a Lathe with the Tool Setting Probe

- Change to the tool to be probed

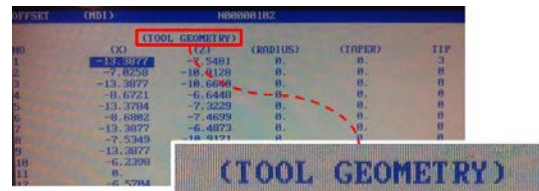
Always make sure the path of the tool turret rotation is clear before performing the tool change!

- **GENTLY AND SLOWLY** deploy the tool probe

Your hand leaves the probe arm **AFTER** it has been deployed!

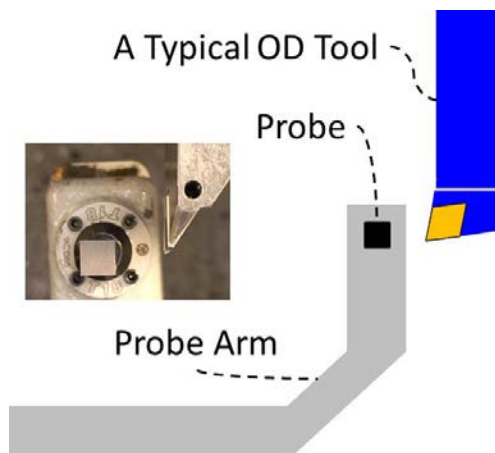


- Press **OFFSET** button until the tool offset is displayed



- Move the cursor to highlight the tool being probed
- Handle Jog the tool **NEAR** the probe

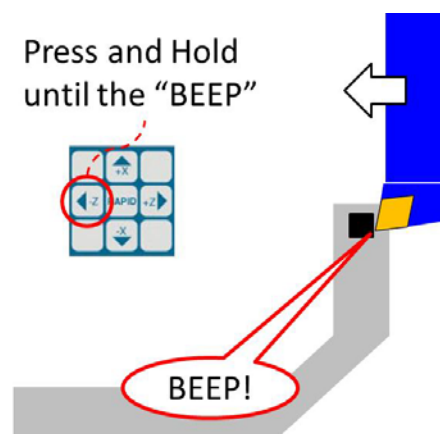
DO NOT HANDLE JOG INTO THE PROBE WITH JOG HANDLE!



- Select the increment of **0.0001"**
- Hold down the direction button to touch the edges that need to be probed



- You will hear a single beep as soon as the tip of the tool comes into contact with the probe. Then, the edge is probed



- Hold down the direction button to jog the tool away from the probe
- When there is sufficient room between the tip of the tool and the probe (e.g. 1"), use the jog handle
- Repeat the previous steps for other edges of the tool
- Secure the Probe Arm before performing tool changes

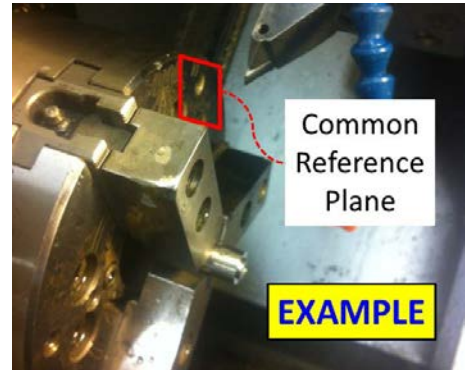
Set Tool Offset (Z) on a Lathe with the Gauge Pin

- Change to the tool to be probed

Always make sure the path of the tool turret rotation is clear before performing the tool change!

- Find a proper smooth and flat surface

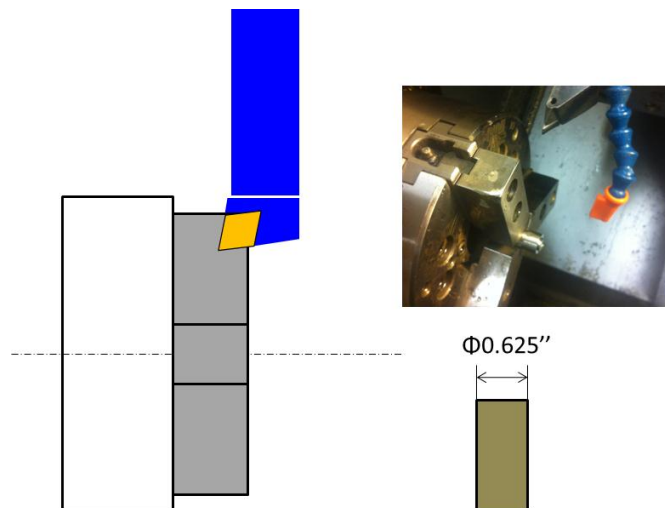
As a common reference for all tools to be probed, in this document, the end surface on the chuck is selected.



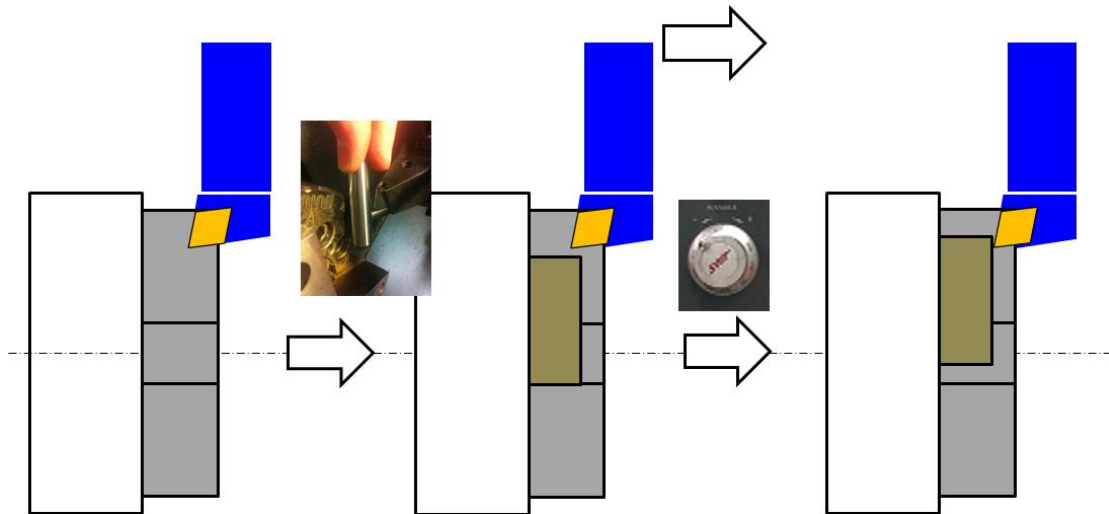
- Press **OFFSET** button until the tool offset is displayed
- Use the cursor to highlight the tool to be probed

NO	(X)	(Z)	(RADIUS)
1	-13.3877	-7.5481	0.
2	-7.8258	-10.9128	0.
3	-13.3877	-10.6640	0.

- Jog the tool towards the surface **WITHOUT** touching it



- Jog the tool away while trying to insert the gauge pin ($\Phi 0.625''$ as shown in this example) between the tool tip and the reference plane



- Verify sure the screen still displays tool offset and the tool being probed is highlighted

OFFSET		(JOG)	N00000102	
(TOOL GEOMETRY)				
NO	(X)	(Z)	(RADIUS)	
1	-13.3877	-7.5481	0.	
2	-7.0258	-10.9128	0.	
3	-13.3877	-10.6640	0.	

- Press **Z SURFACE MEASURE** when the gauge block just fits
- Offset the thickness of the gauge block
 - Type **-0.625''** and Press **WRITE/ENTER**

OFFSET		(JOG)	N00000102	
(TOOL GEOMETRY)				
NO	(X)	(Z)	(RADIUS)	
1	-13.3877	-11.4100	0.	
2	-7.0258	-10.9128	0.	
3	-13.3877	-10.6640	0.	

Set Tool Offset (X) on a Lathe by Testing Cut

- Change to the tool to be probed

Always make sure the path of the tool turret rotation is clear before performing the tool change!

- Handle Jog the tool near the face of the workpiece

Make sure the tool does NOT touch the workpiece unless the spindle is turning!

- Handle Jog the tool near the face of the workpiece

Make sure the tool does NOT touch the workpiece unless the spindle is turning!

- Start the Spindle

- Go to **MDI** mode



- Specify spindle RPM (e.g Type "**S500**" for 500 RPM) and press "**WRITE/ENTER**"

- Close the door and press "**CYCLE START**"

- Press the "**FWD**" button to start the spindle



- Jog along the Z axis and cut off a thin layer from the OD surface

While the spindle is spinning at specified RPM,

- Go to "**HANDLE JOG**" mode

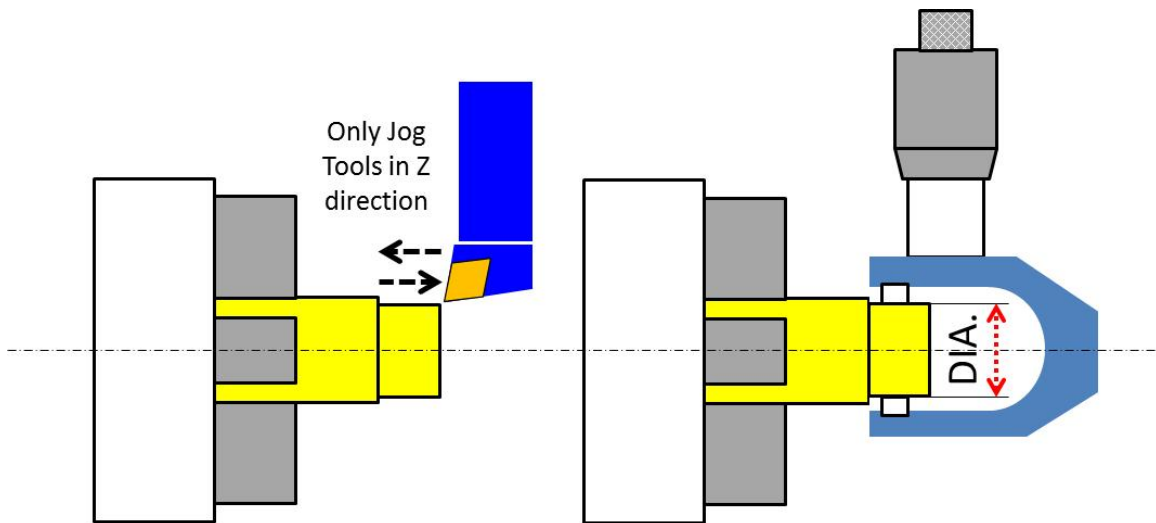


- Choose the increment of **0.001**

- Jog in the X direction till the tip of the tool is along the outer edge of the workpiece



- Jog along the Z direction to cut off a thin layer
- Jog along the negative Z direction to retreat the tool
- Measure the diameter of the machined surface with a micrometer and remember the reading



- Press **OFFSET** button until the tool offset is displayed
- Use the cursor to highlight the tool to be probed

OFFSET (JOG) N00000102			
(TOOL GEOMETRY)			
NO	(X)	(Z)	(RADIUS)
1	-13.3877	-11.4100	0.
2	-7.0258	-10.9128	0.
3	-13.3877	-10.6640	0.

- Press **X DIA MEASURE**
- Type in the number from the micrometer reading


Appendix I Lathe Work Offsets

- Change to a diamond OD tool
Always make sure the path of the tool turret rotation is clear before performing the tool change!




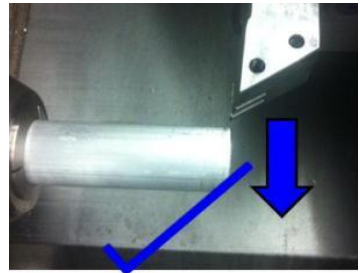
- Handle Jog the tool near the face of the workpiece

Make sure the tool does NOT touch the workpiece unless the spindle is turning!

- Start the Spindle
 - Go to **MDI** mode 
 - Specify spindle RPM
e.g Type **"S500"** and press **"WRITE/Enter"** for 500 RPM
 - Close the door and press **"CYCLE START"**
 - Press the **"FWD"** button to start the spindle



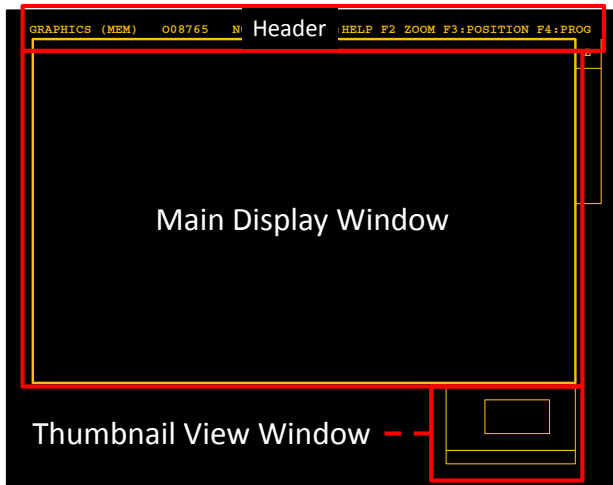
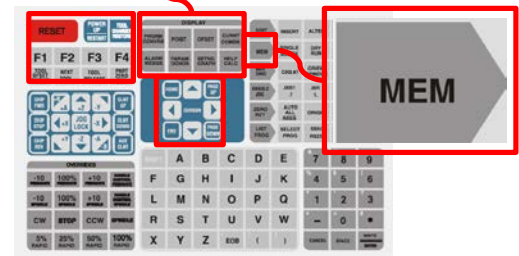
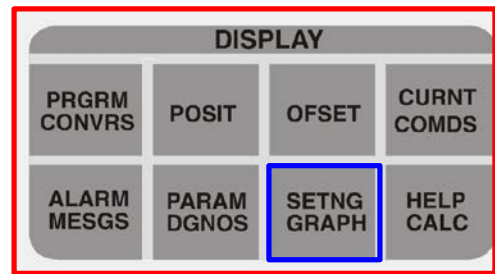
- Faceoff the workpiece
While the spindle is spinning at specified RPM,
 - Go to **"HANDLE JOG"** mode 
 - Choose the increment of **0.001"**
 - Jog in the Z direction until the tool just touches the workpiece
 - Change the axis selection to **X** and face off the workpiece



- Set Work Offset
 - Keep the tool at same Z coordinate of the last faceoff cut (Jog only in X direction)
 - Press **OFFSET** button until you see the screen of work offset
 - Use the cursor to highlight the Z offset for Work System G54
 - Press **Z Face Measure** to record the work offset

Appendix J Set up Simulation in Graphics

- Press **MEM** button once, then **SETNG GRAPH** button **TWICE** to activate the screen for simulation on the controller



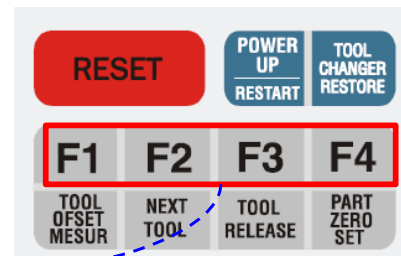
HEADER

GRAPHICS (MEM) 008765 N00000 F1:HELP F2 ZOOM F3:POSITION F4:PROG

Indicate the current display as **GRAPHICS** and the system on **MEMORY** mode

008764	: The number of the current program
N0000	: The current line number

Display Options



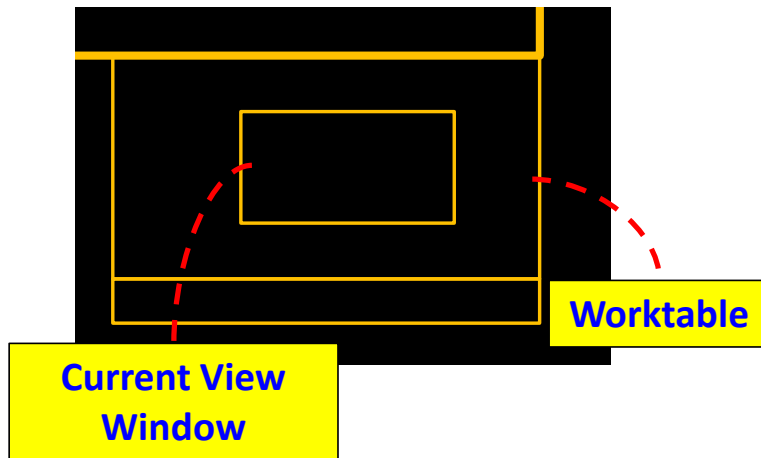
F1	: Switch for display the help information for current screen
F2	: Switch for the zoom settings
F3	: Switch for position display
F4	: Switch for program display

Setting the Zoom for Simulation

- Press **F4** until the thumbnail view shows up on the lower right corner
- Under the screen of Simulation in Graphics, press **F2** until the right end of the header displays

F1:HELP WRITE:ZOOM DONE

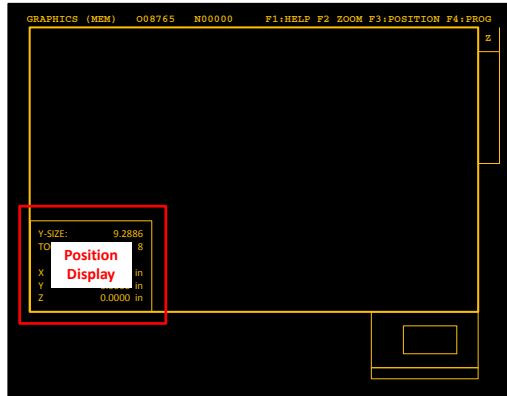
- Use the arrow buttons to move the view window, use **PAGE UP** or **PAGE DOWN** buttons to zoom out or zoom in. The changes should be instantly reflected on the thumbnail view



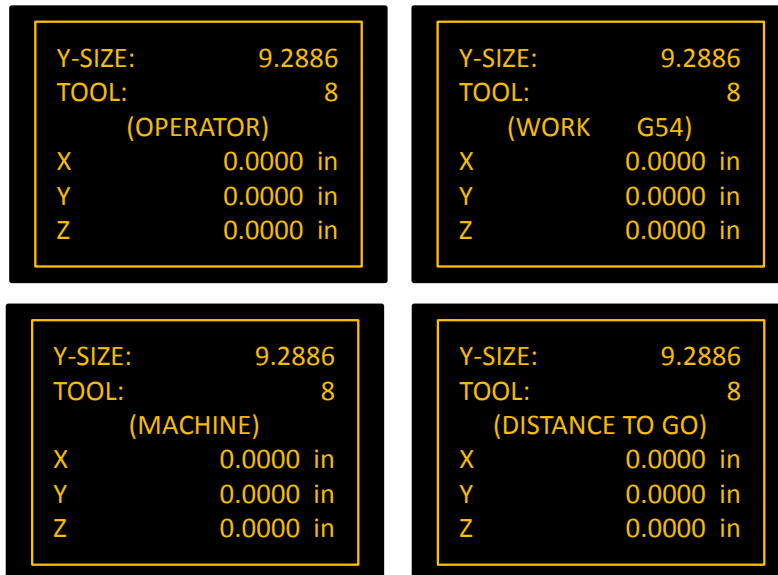
- Press **WRITE/ENTER** when the selection of view is completed to confirm the change

Setting the Display for Positions

- The position view can be activated by pressing **F3**



- Four different types of positions will cycle through by keeping pressing **F3**



Setting the Display for Programs

- The thumbnail view can be replaced with the program by pressing **F4** button

