Haas Control

The Haas control is shown in Figures 2 and 3. Familiarize yourself with the location of buttons and controls. Detailed instructions on the following pages show how to operate the control.

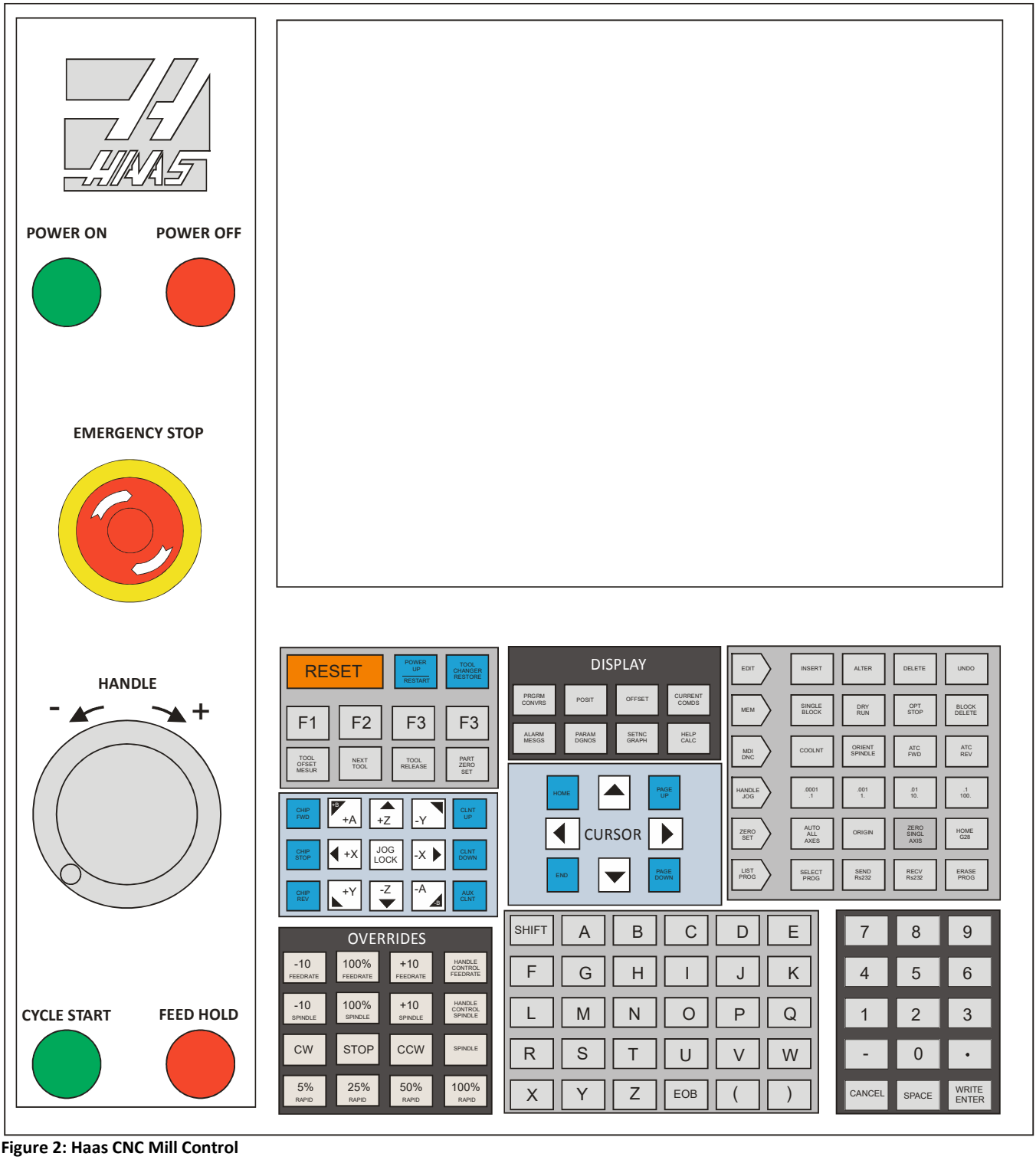


Figure 2: Haas CNC Mill Control

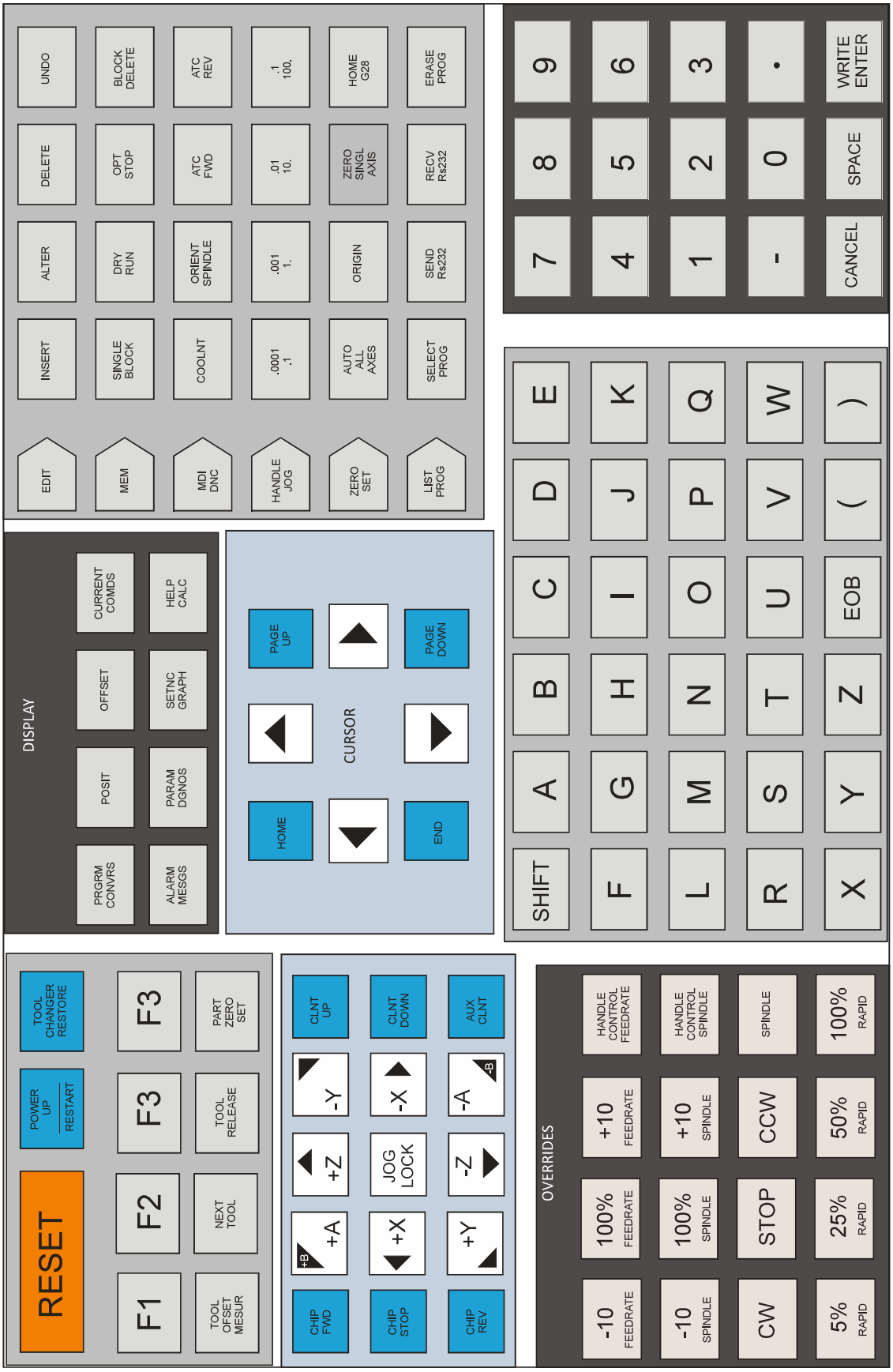
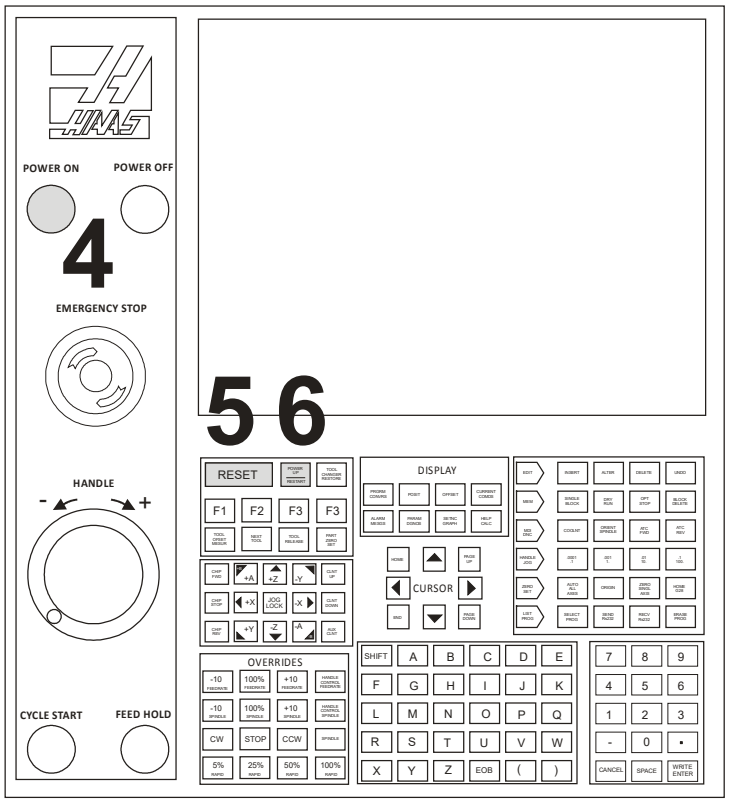


Figure 3: Haas CNC Control Buttons

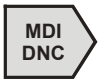
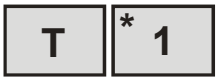
Start/Home Machine

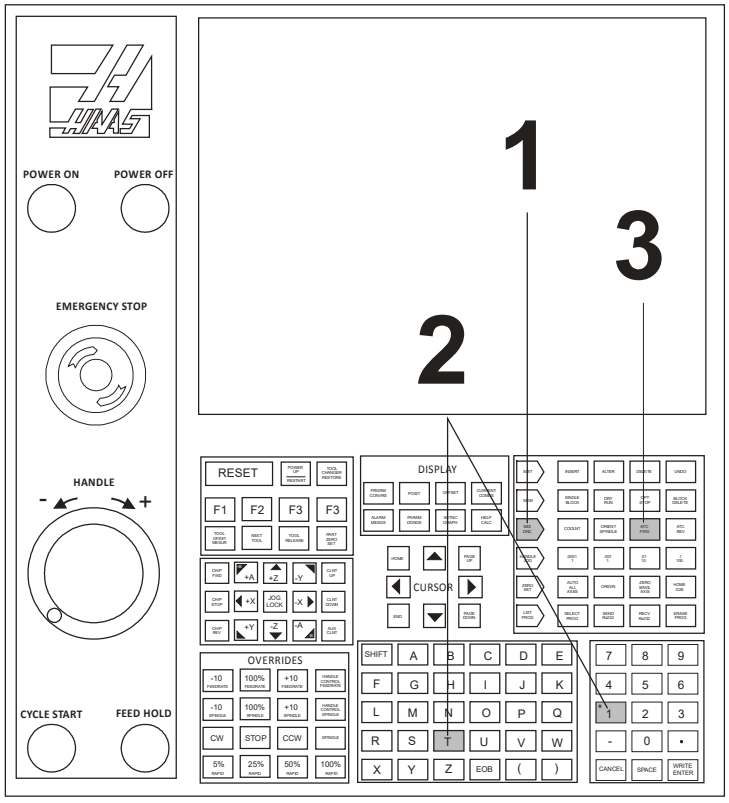
	Check List	Buttons
1	**Work Area:**Clear	
2	**Air Supply:** On/Correct ▪ Haas requires at least 70PSI for tool changer to operate.	
3	**Main Breaker:** On	
4	**POWER ON:** Press ▪ Ensure Emergency Stop is not tripped. If it is, twist red knob right to release. ▪ Wait until message **102 SERVOS OFF** appears before proceeding.	POWER ON
5	**RESET:** Press	RESET
6	**Power On Restart:** Press ▪ Ensure doors are closed and work area is clear. ▪ Allow all machine axes to home before proceeding.	POWER UP / RESTART



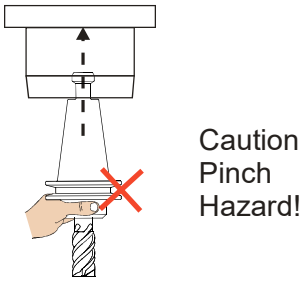
Start/Home ***Sheet 1 of 1***

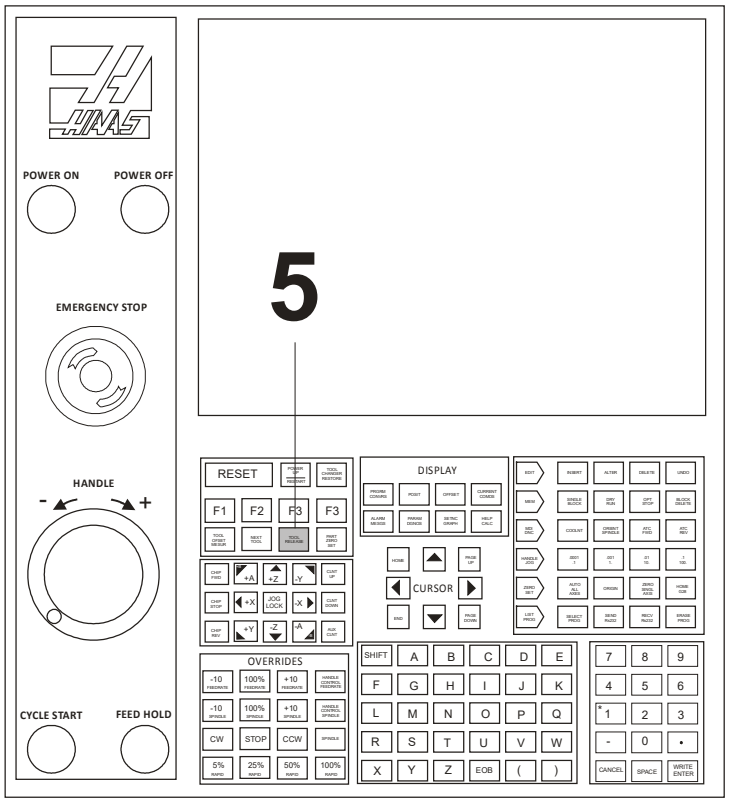
Load Tools

	Check List	Buttons
1	**MDI/DNC Key:** Select	
2	**Tool Number:** Enter ▪ For example, to position the tool changer to T1, Press the T and then the 1 buttons.	
3	**ATC FWD:** Select ▪ Tool carousel will index to T1 position.	



Load Tool ***Sheet 1 of 2***

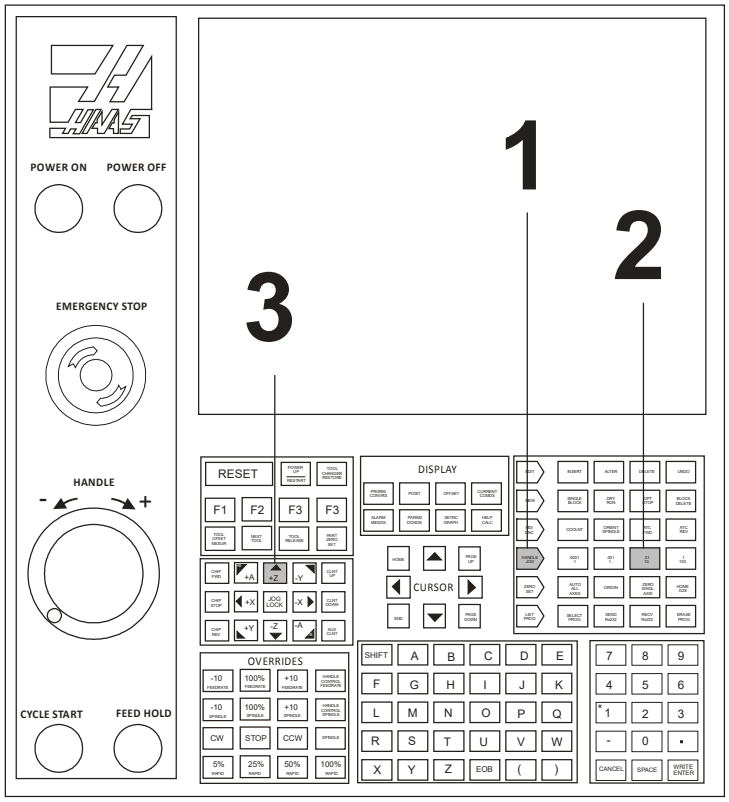
4	**Position Tool in Spindle** ▪ **Do not grip by tool cutting flutes!** ▪ Ensure tool taper is clean. ▪ Grip tool holder below V-flange to prevent pinching. ▪ Push tool into spindle. ▪ Ensure "dogs" on spindle line up with slots on tool holder.	
5	**Tool Release:** Select ▪ Machine will blow air thru spindle to clear debris. ▪ Gently push the tool upward and then release the Tool Release button. ▪ Ensure tool is securely gripped by spindle before releasing it.	
6	**Repeat** ▪ Repeat steps 2-5 until all tools are loaded.	



Load Tool ***Sheet 2 of 2***

Set Tool Length Offset (TLO)

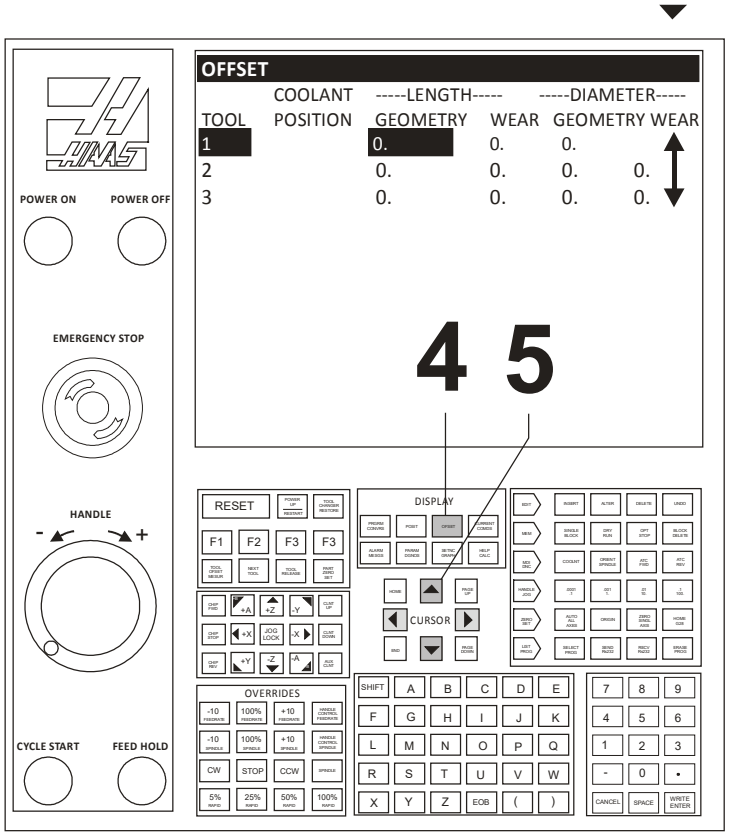
	Check List	Buttons
1	**Handle Jog Mode:** Select ▪ This sets machine to be controlled by the hand wheel.	HANDLE JOG
2	**Jog Increment:**.01 ▪ This sets the job increment so each click of the hand wheel moves the tool .01 inches in the jog direction.	.01 10.
3	**Jog Direction:** Z ▪ This sets the tool to move in Z when the jog handle is moved.	Z



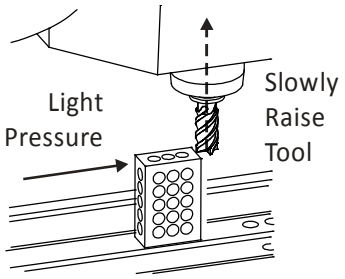
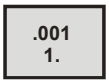
Note: Machine parameter 64 (T OFS MEAS USES WORK) must be set to OFF to use this method.

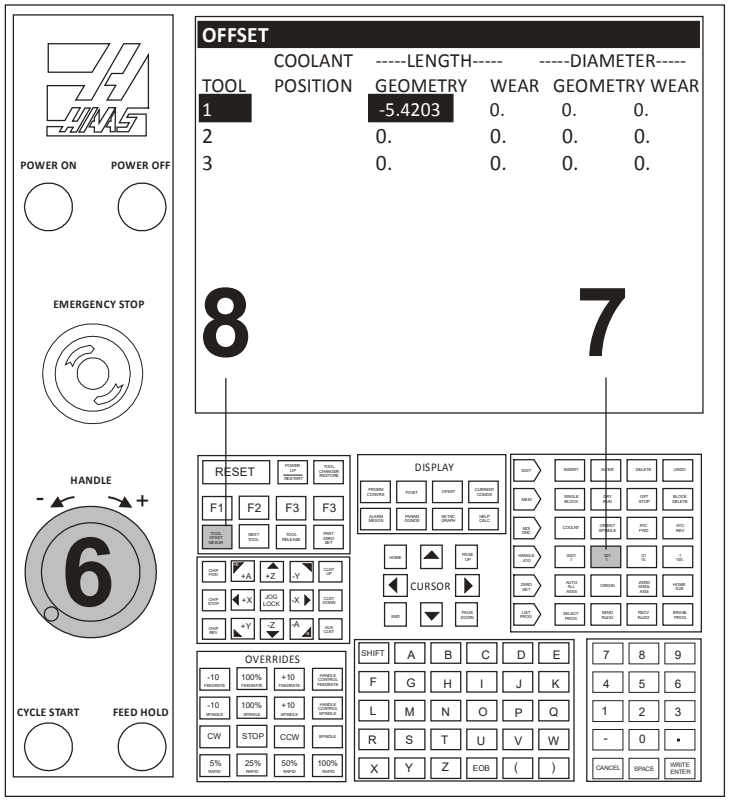
Set TLO ***Sheet 1 of 4***

4	**Offsets:** Select ▪ Press this button until the Tool Offset page displays.	
5	**Cursor Arrows:** Align for active tool ▪ Use the Up-Dn cursor keys (if needed) to move the highlighted bar on the graphics display over the offset values for the currently active tool.	



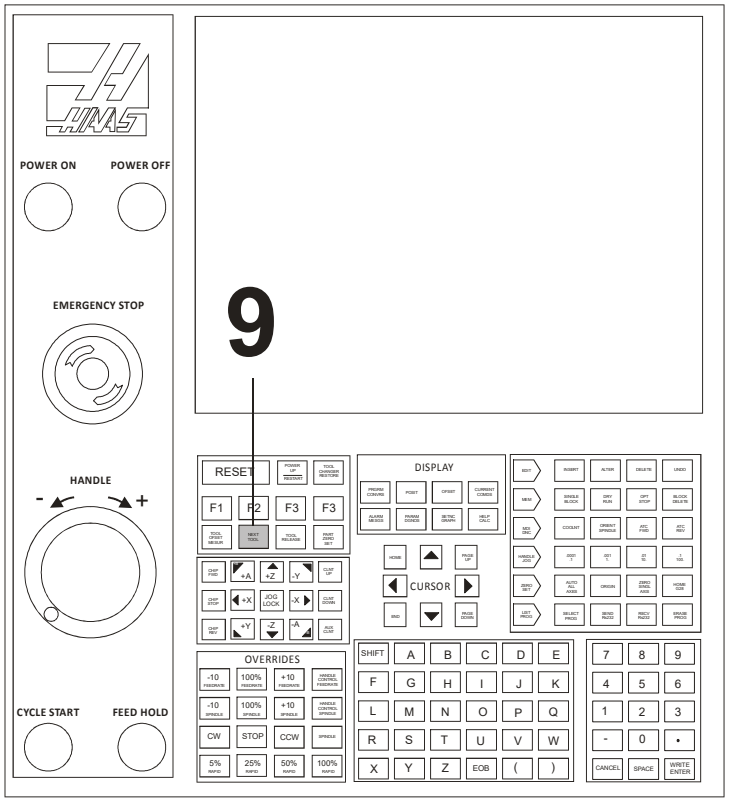
Set TLO ***Sheet 2 of 4***

6	**Use 1-2-3 Block to Set Tool Length** ▪ Jog so tool is below top block. ▪ Apply slight pressure to block against tool. Use Jog Wheel to raise tool until the block just slides underneath it. ▪ Move block out of way and then move tool back down .01 inches below top of block.	
7	**Jog Increment:** .001 ▪ Reduce jog increment and use jog handle to raise tool in .001 increments until it just slides under the block again. Repeat this process with the .0001 jog increment if needed.	
8	**Tool Offset Measure:** Select ▪ This causes the controller to enter the current position of the tool in the length offset register. ▪ Make sure the tool length number updates before proceeding.	



Set TLO ***Sheet 3 of 4***

9	**Next Tool:** Select ▪ This causes the current tool to be put away and the next tool to be loaded. ▪ Repeat steps 1 thru 9 until all tools are set.	NEXT TOOL



Tips

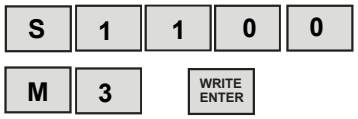
- Do not press any buttons between steps 8-9 or **NEXT TOOL** button will not load next tool.
- If you should press any buttons and interrupt this process, load the next tool by selecting the **ATC FWD** button, followed by **HANDLE JOG**.

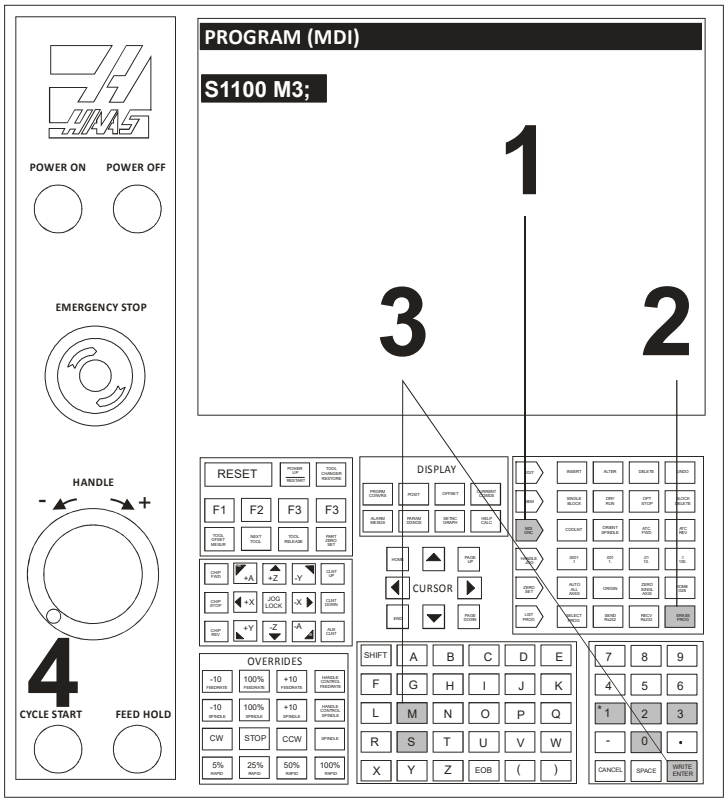
Set TLO ***Sheet 4 of 4***

WARNING: Setting tools requires manually jogging the machine with hands in the machine work envelope. Use extreme caution and observe the following rules:

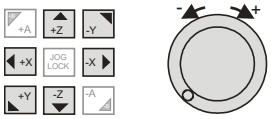
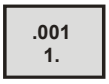
- The spindle must be off.
- Never place your hand between the tool and the 1-2-3 block.
- Ensure the correct axis and jog increment are set before jogging.
- Move the handle slowly and deliberately. Keep your eyes on your hands and the tool position at all times.
- Never allow anyone else to operate the control when your hand is in the work area.

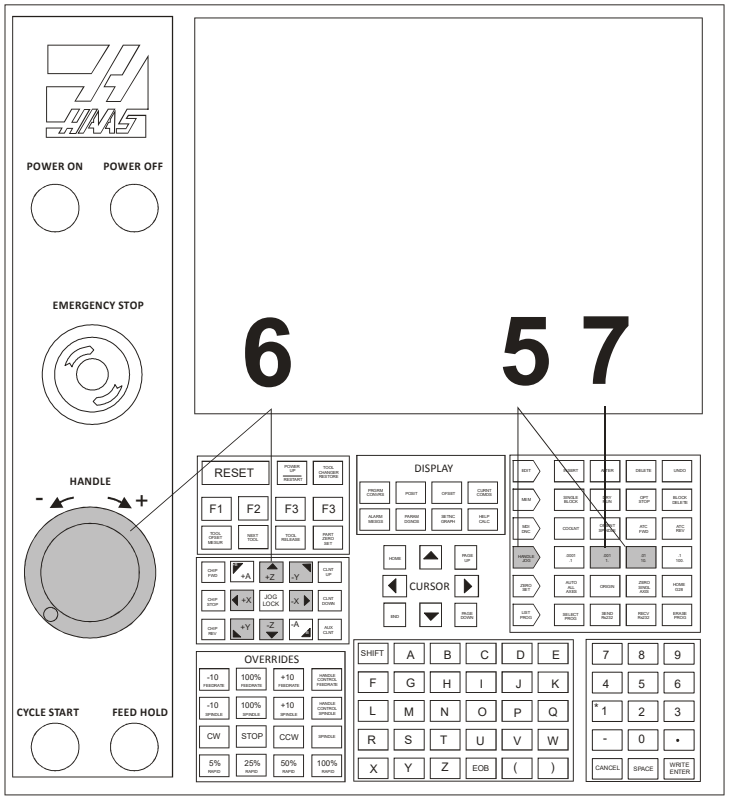
Set Fixture Offset XY

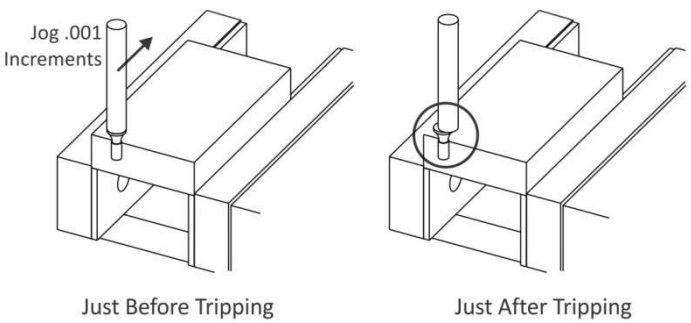
Checklist		Buttons
1	**MDI/DNC Key:** Select Erase Prog: Select (to clear any commands)	
2	**Erase Prog:** Select (to clear any commands)	
3	**Spindle Speed:** S1100 ▪ **S1100 M3:** Input ▪ **Write/Enter:** Select	
4	**Cycle Start:** Select ▪ Spindle will start CW at 1100 RPM	



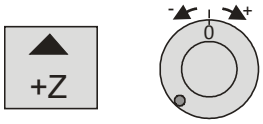
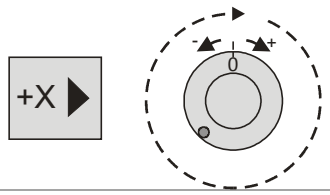
Fixture Offset XY ***Sheet 1 of 4***

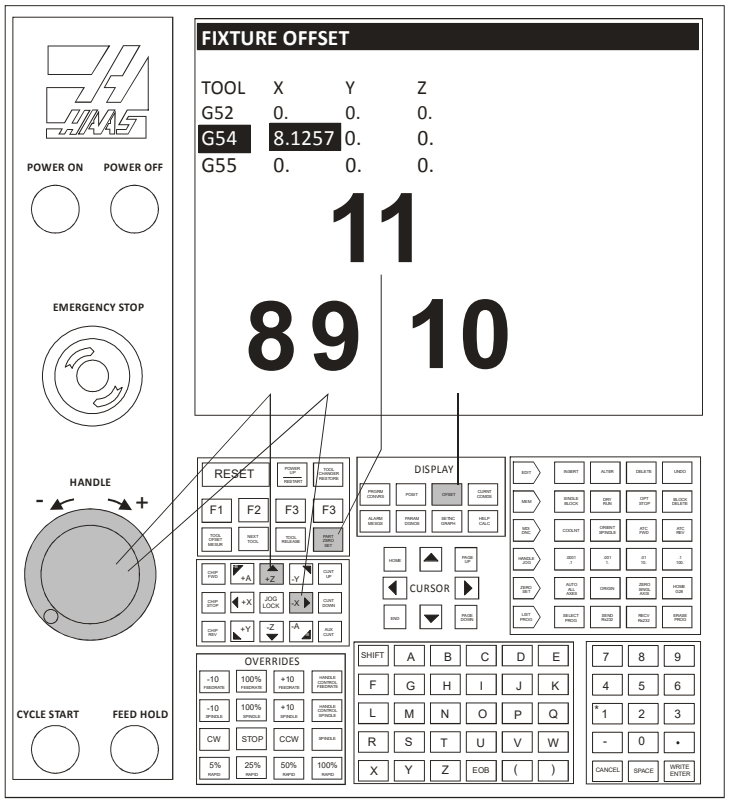
5	**Handle Jog:** Select **Jog Increment:**.01	
6	**Jog Handle:** As Needed ▪ Select jog direction and use handle as required to place edge finder stylus alongside the left part edge.	
7	**Jog Increment:**.001 ▪ Move edge finder slowly until it just trips off center as shown below. ▪ This places the center of the spindle exactly .100 from the part edge.	





Fixture Offset XY ***Sheet 2 of 4***

8	**Jog Handle:** Retract in Z ▪ Jog straight upward in Z until edge finder is above part and jog handle reads zero on the dial.	
9	**Jog Handle:** Set jog direction to +X and rotate handle one full turn clockwise. ▪ Since the control is in .001 increment mode, rotating the dial exactly one full turn places the center of the spindle directly over the left part edge.	
10	**Offset Page: Select** ▪ Select Offset button and PgUp/PgDn buttons until Work Zero Offset page appears. Use Arrow keys to highlight G54 (or whatever fixture offset is to be set).	
11	**Part Zero Set:** Select ▪ This sets the G54 X value to the current spindle position.	

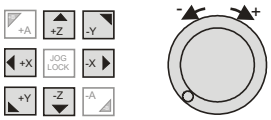
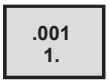
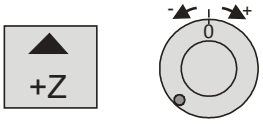
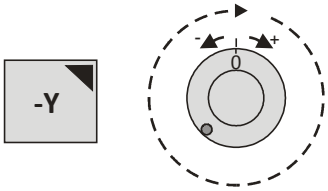
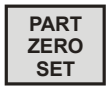


Tips

- To shift the datum RIGHT in relation to the machine operator, **ADD** a shift amount to the offset X-value. For example, to shift X+.1, input **.1WRITE/ENTER**.
- To shift the datum CLOSER to the machine operator, **SUBTRACT** a shift amount from the offset Y-value. For example, to shift Y-.1, input **-.1 WRITE/ENTER.**

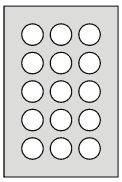
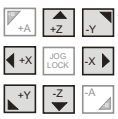
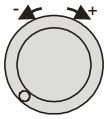
Fixture Offset XY ***Sheet 3 of 4***

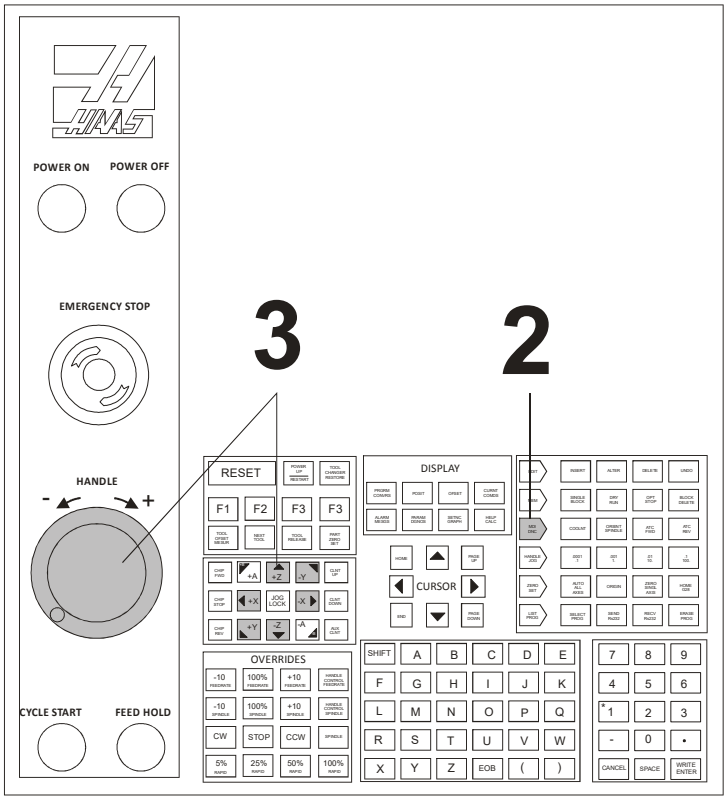
The following instructions repeat steps 4-10 but for setting the Y-axis.

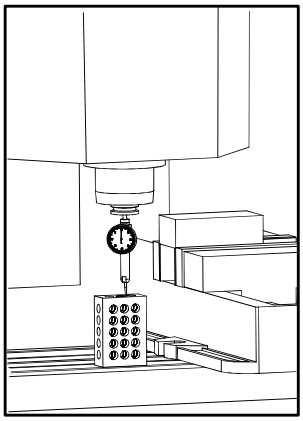
12	**Jog Handle:** As Needed ▪ Select jog direction and use handle as required to place edge finder stylus alongside the back part edge.	
13	**Jog Increment:** .001 ▪ Move edge finder slowly until it just trips off center. ▪ This places the center of the spindle exactly .100 from the part edge.	
14	**Jog Handle:** Retract in Z ▪ Jog straight upward in Z until edge finder is above part and jog handle reads zero on the dial.	
15	**Jog Handle:** Set jog direction to -Y and rotate handle one full turn clockwise. ▪ Since the control is in .001 increment mode, rotating the dial exactly one full turn places the center of the spindle directly over the back part edge.	
16	**Offset Page:** Use arrow keys to highlight G54 field (or whichever fixture offset you are setting).	
17	**Part Zero Set:** Select ▪ This sets the G54 Y value to the current spindle position.	
18	**Spindle Stop:** Select	

Fixture Offset XY ***Sheet 4 of 4***

Set Fixture Offset Z

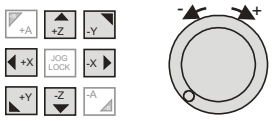
	Check List	Buttons
1	**Prepare:** ▪ Ensure tip angle of dial indicator is at about 15 degrees from a horizontal surface. ▪ Ensure gauge block is flat on table, no chips underneath, and not floating on coolant.	 Dial Indicator  1-2-3 Block
2	**MDI Mode: Select** ▪ Ensure machine is in MDI Mode	
3	**Handle:** As Needed ▪ Select jog direction and use handle as required to place dial indicator stylus on top of gauge block and the dial reads zero.	 

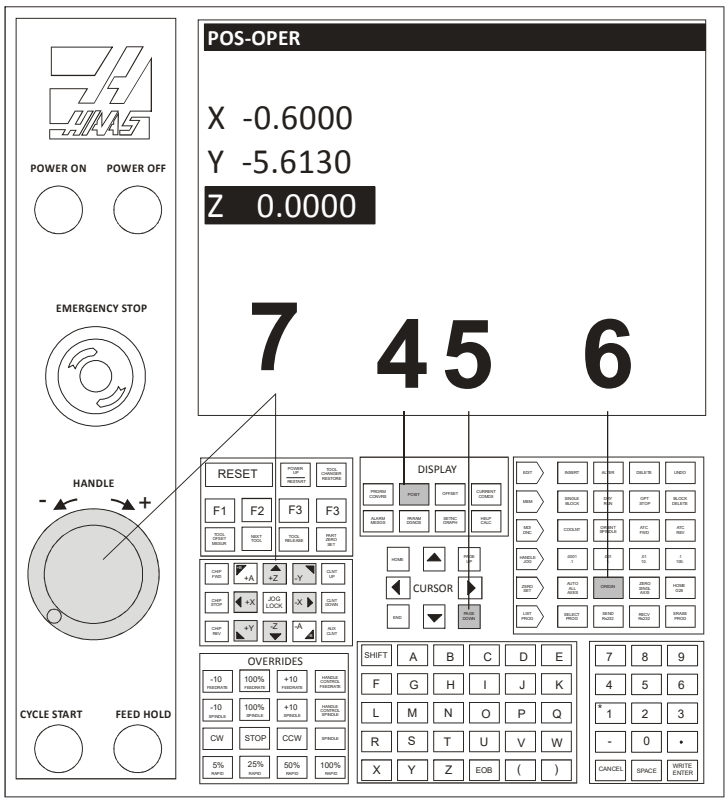


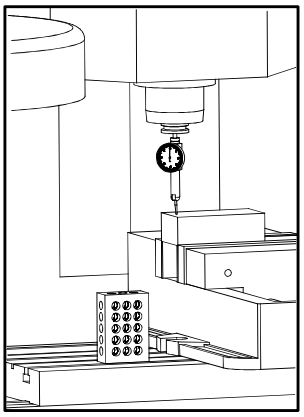


Note: Machine parameter 64 (T OFS MEAS USES WORK) must be set to OFF to use this method.

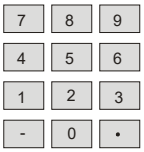
Fixture Offset Z ***Sheet 1 of 3***

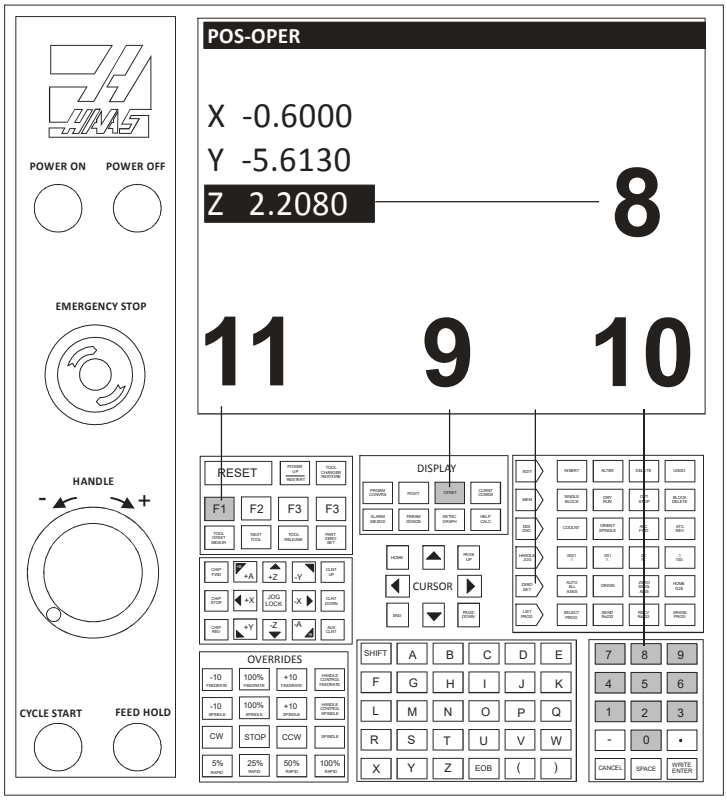
4	**Posit Button:** Press	
5	**Operator Screen:** Select ▪ Use PG DN Button (5 times) until POS-OPER screen appears.	
6	**Origin:** Set ▪ Press Origin button to set Z-value on operator screen to zero.	
7	**Handle:** As Needed ▪ Select jog direction and use handle as required to place dial indicator stylus on top of part stock and the dial reads zero.	





Fixture Offset Z ***Sheet 2 of 3***

8	**Operator Z Value:** Read ▪ Read the operator Z value from the graphic area. This is the incremental distance between the top of the 1-2-3 block and the top of the part. Ex: 2.2080	
9	**Offset Page:** Set G54 Z Value ▪ Press Offset button twice to get to Offsets Page	
10	**Numeric Keypad:** Enter Z Value ▪ Input the value using the numeric keyboard. ▪ To set Z below the part face, reduce the value by the amount of material removed by the face cut.	
11	**F1 Button:** Press ▪ Select the F1 button to enter the value input in Step #7 into the Z offset field.	



Tip

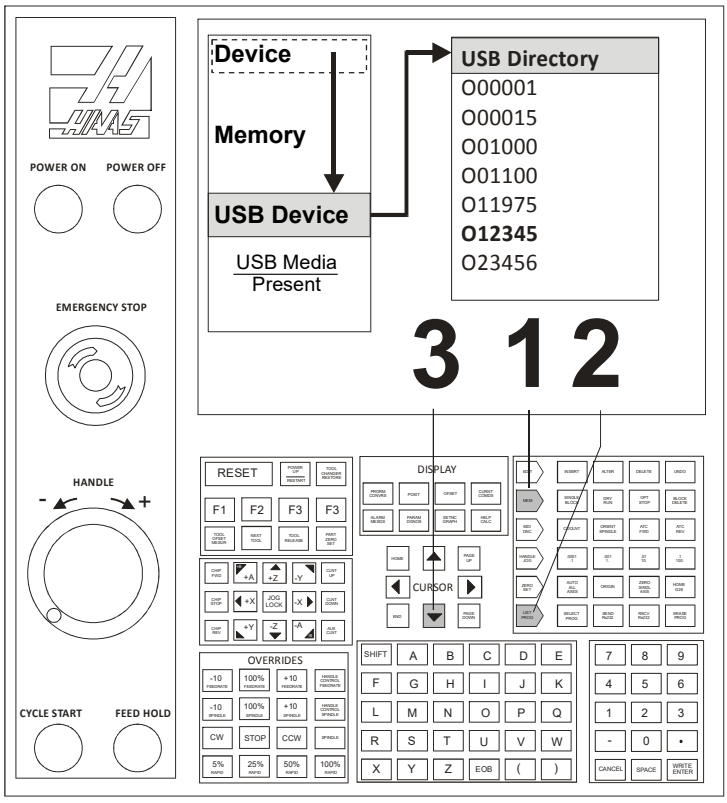
Shift the datum DOWN by an incremental distance from the top of the part to remove stock from the top-face by **subtracting** the amount of stock to remove from the Fixture Offset Z Value.

Example: To set the datum to Z-.02 below the top of the part:

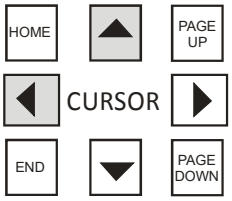
Z2.2080
-.020
= 2.1880

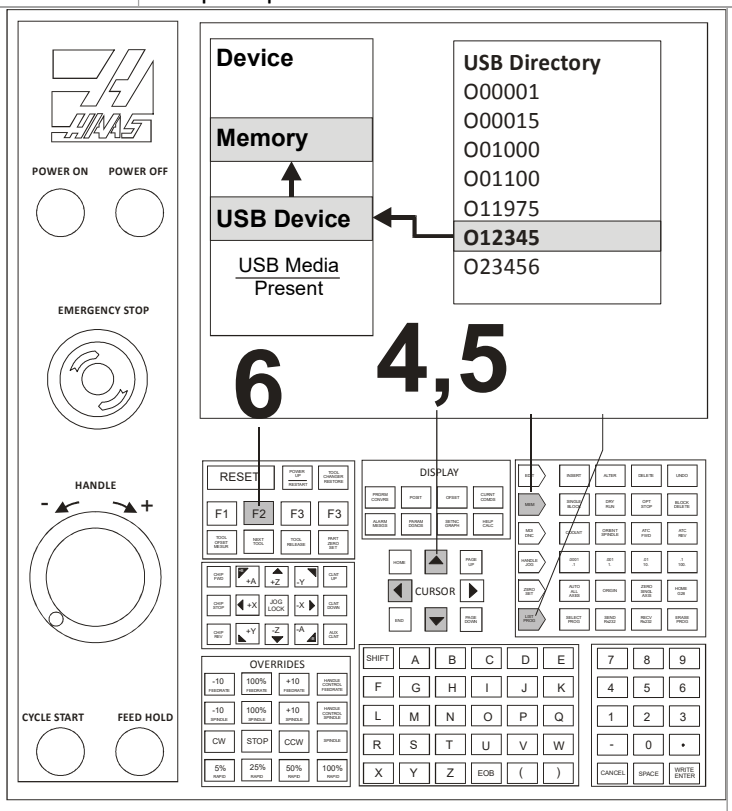
Load CNC Program

	Check List	Buttons
1	**Memory:** Select	MEM
2	**List Programs:** Select	LIST PROG
3	**USB Device:** Select ▪ Use the DN arrow to highlight USB Device on the graphics screen. (This sets the program list to display files on the USB Device). ▪ Select F4 button.	CURSOR F4



Load CNC Program ***Sheet 1 of 2***

	Checklist	Buttons
4	**Program Name:** Highlight ▪ Use RT button to move highlighted bar over USB Directory List ▪ Use the DN arrow keys to highlight the program to read.	
5	**Write To Device (Memory):** Select ▪ Press the Left arrow key and then the Up arrow key to move the highlight bar back over the Memory option in the Device list.	
6	**F2:** Select ▪ Press the F2 button to copy the program from the USB Device to Memory. ▪ If the program already exists, select (Y) when prompted to overwrite.	

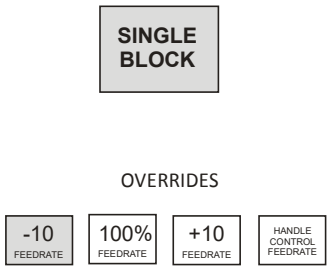


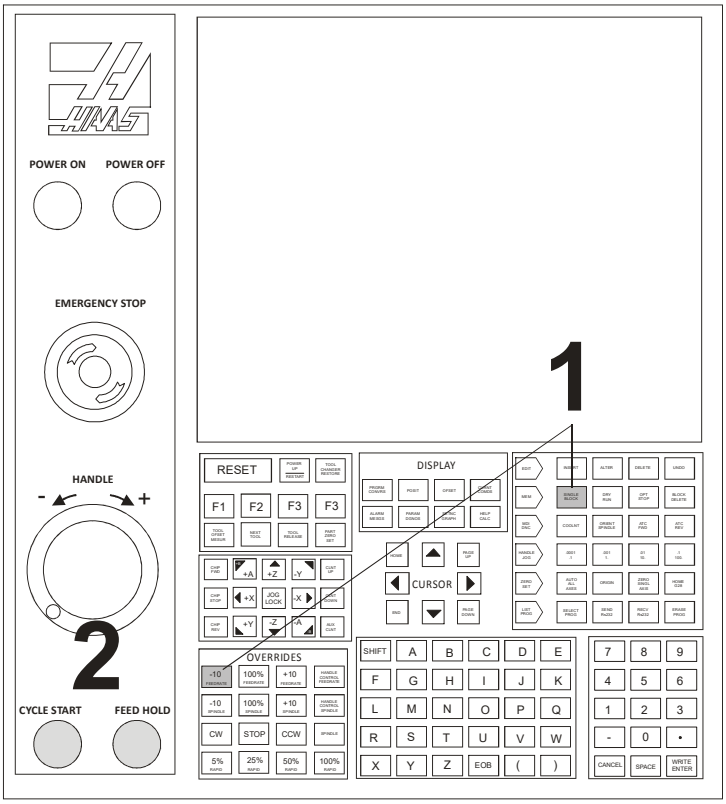
Note: All programs must begin with the letter “O” (NOT the number zero). The program name must be an integer up to five digits long. No decimal point, letters, or special characters are allowed.

Load CNC Program ***Sheet 2 of 2***

Run CNC Program

This is the preferred process for running a new program. Once a program is proven, all feed rates can be set to 100% and single block mode can be set to off.

	Check List	Buttons
1	**Pre-Start** ▪ Ensure vise or fixture is secure and that you have a safe setup. ▪ There should be no possibility that the work holding will fail to perform as required. ▪ Remove vise handles. ▪ Clear the work area of any tools or other objects. ▪ Close the machine doors. ▪ Turn **Single Block** Mode On. ▪ Press Rapid Feedrate **-10** button eight times to set rapid Feed Rate Override to 20% of maximum.	
2	**Start** ▪ Place one hand on Feed Hold button and be ready to press it in case there are any problems. ▪ Press Cycle Start Button.	



Run CNC Program ***Sheet 1 of 1***

Caution: A common error is setting the Fixture or Tool Length offset incorrectly. When running a program for the first time, set the machine to Single block mode. Reduce rapid feed rate to 25%, and proceed with caution. Once the tool is cutting, turn off single block mode and let the program run. Do not leave the machine unattended, and keep one hand on the feed hold button. Listen, watch chip formation, and be ready to adjust cutting feed rates to suite cutting conditions.

Adjusting CDC Offsets

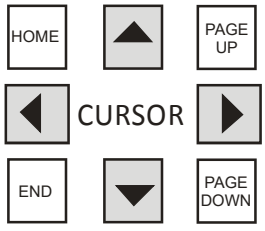
Machining operations that use Cutter Diameter Compensation (CDC: G41/G42) can be adjusted to account for tool wear and deflection. Measure across a finished feature on part and compare it with the desired value. Subtract the Actual from the Target sizes and enter the difference into the CDC register on the control for that tool. For example:

Target Feature Size: 1.0000
Actual Feature Size: 1.0120
Wear Value: -0.0120*

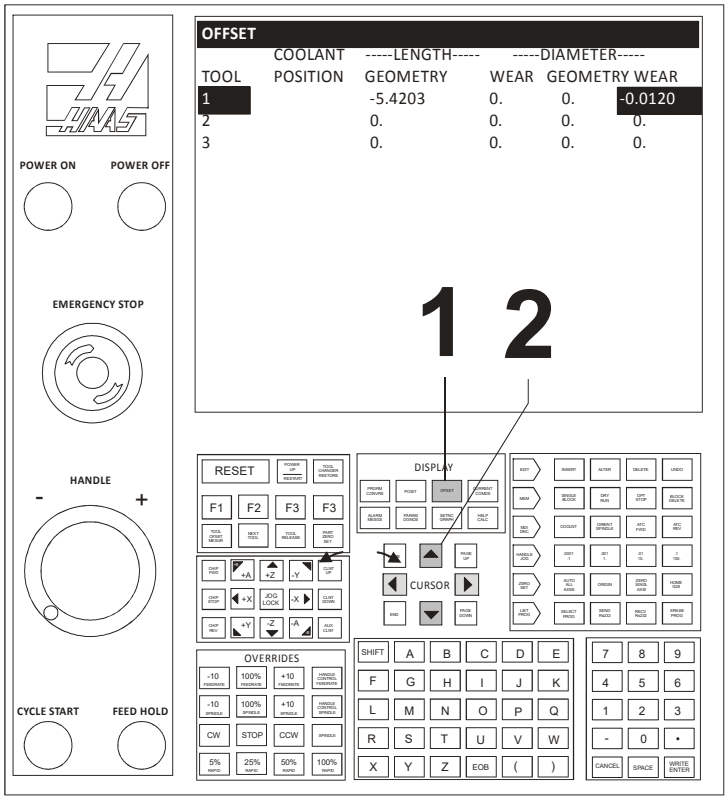
The tool path will now be compensated for the size difference. Running the same operation again should result in the feature being exactly the target size.

*Wear compensation is used only on contour passes. It is not used for face milling, 3D milling, or drill cycles. Select the Wear Compensation option in your CAD/CAM software and, if needed, set a **Tool Diameter Wear** value as shown above. When used, the wear value is always a negative number.*

*Always set **Tool Diameter Geometry** to zero for all tools since CAD/CAM software already accounts for the tool diameter by programming the tool center line path.*

	Check List	Buttons
1	**Offset Page:** Select	
2	**Adjust Diameter Offset** ▪ Pg Up/Dn to highlight the tool to be adjusted. ▪ Enter a value using the numeric keypad. ▪ Select	

Adjusting Offsets ***Sheet 1 of 2***



Adjusting Offsets ***Sheet 2 of 2***

Shut Down CNC

	Check List	Buttons
1	**Remove tool from spindle:** ▪ Enter the number of an empty tool carousel. ▪ Select ATC FWD	
2	**Jog Machine to Safe Area:** ▪ **Select Jog**	
3	**Shut Down Button:** Press	**POWER OFF** 

Post Power-Down Checklist:

- Wipe spindle with a soft clean rag to remove coolant and prevent rusting.
- Put away tools.
- Clean up work area.
- Always leave the machine, tools, and equipment in the same or better condition than when you found them.

Shut Down CNC ***Sheet 1 of 1***

Note: It is important to clean the machine after each use to prevent corrosion, promote a safe work environment, and as a professional courtesy to others. Allow at least 15-30 minutes at the end of each day for cleaning. At the very least, put away all unused tools and tooling, wash down the machine with coolant, remove standing coolant from the table, and run the chip conveyor.