HAAS FACTORY OUTLET
A DIVISION OF PRODUCTIVITY INC

HAAS MILL OPERATOR

OPERATOR MANUAL

Revised 01/25 JP

This material is to be used as a guide to the operation of the machine tool. The Operator is responsible for following Safety Procedures as outlined by their instructor or manufacturer's specifications.

NOTE: Downloading and/or other use of this manual does not certify completion of the Training course. This manual is for reference only.

For more information on
Additional Training Opportunities or our Classroom Schedules
Contact the Productivity Inc Applications Department
Minneapolis: ☎ 763.476.8600
Cedar Rapids: ☎ 319.734.3403
Omaha: ☎ 402.330.2323
Denver: ☎ 800-947-8665
Salt Lake City: ☎ 801.886.2221
Visit us on the Web: www.productivity.com

HAAS Interactive Mill Operator Manual

https://www.haascnc.com/service/online-operator-s-manuals/mill-operator-s-manual/mill---introduction.html

Note: Some of the content, images and screen shots included in this manual are taken from Haas manuals, controllers and web information with permission from Haas Automation Inc. 2800 Sturgis Road Oxnard CA 93030-8933

CONTENTS

INTRODUCTION TO BASIC VERTICAL MILL OPERATION

Welcome to Productivity, Inc., your local Haas Factory Outlet (H.F.O.) for the Haas Mill Operator Class. This class is intended to give a basic understanding of the set-up and operation of a Haas Machining Center.

After 1945 the wing designs for the US Air Force became extremely complex and hard to manufacture using conventional machine tools. MIT **(Massachusetts Institute of Technology)** developed a machine that was able to control a cutting tool path with a series of straight lines defined by axial coordinates at prescribed feed rates. The first NC machine tool was introduced to the defense and aerospace industry by MIT in 1952. The contour of a constantly changing curvature could be described by a series of short lines determined by a series of coordinates in three axes.

The first machine tools were run with instructions or programs punched out on paper tape. The files of the early machine tools were often in the format which later became called G-code. The reason for the name being that many of the lines of text began with the letter G.

In an NC machine, the tool is controlled by a code system that enables it to be operated with minimal supervision and with a great deal of repeatability. "CNC" (Computerized Numerical Control) is the same type of operating system, with the exception that a computer monitors the machine tool.

The same principles used in operating a manual machine are used in programming an NC or CNC Machine. The main difference is that instead of cranking handles to position a slide to a certain point, the dimension is stored in the memory of the machine control **once.** The control will then move the machine to these positions each time the program is run.

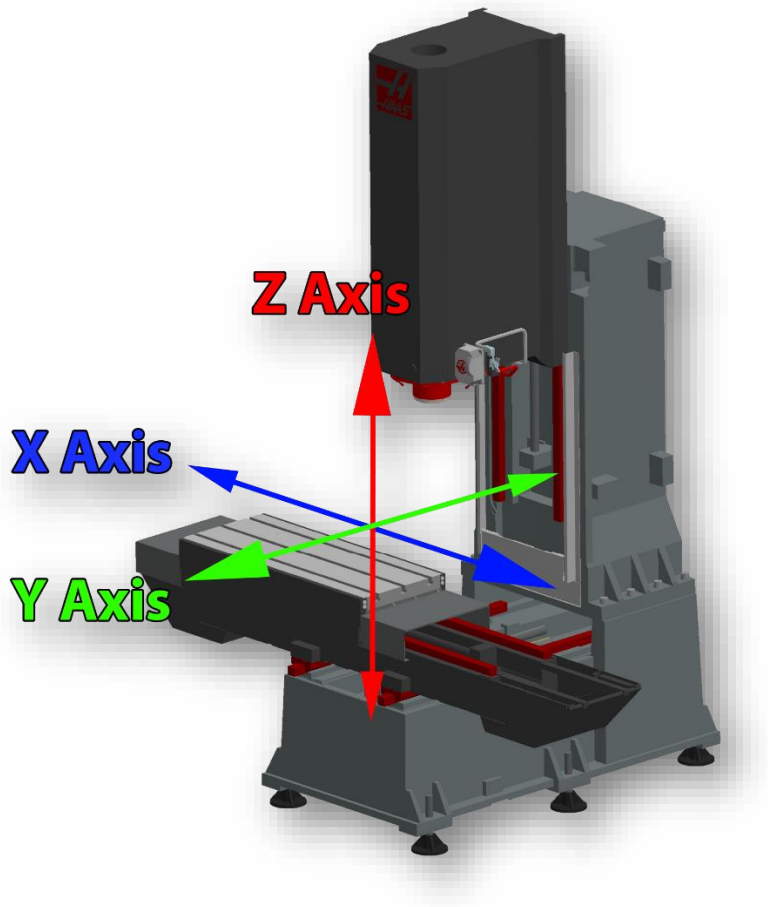
The operation of the VF-Series Vertical Machining Center requires that a part program be designed, written, and entered the memory of the control. There are two options for getting these programs to the control. Ethernet and USB are both viable ways to transmit and receive programs.

To operate and program a CNC controlled machine, a basic understanding of machining practices and a working knowledge of math are necessary. It is also important to become familiar with the control console and the placement of the keys**,** switches, displays, etc., that are pertinent to the operation of the machine.

This manual can be used as both an operator's manual and as a programmer's manual. It is intended to give a **basic** understanding of CNC programming and its applications. It is not intended as an in-depth study of all ranges of machine use, but as an overview of common and potential situations facing CNC programmers. Much more training and information is necessary before attempting to program on the machine.

The programming section of this manual is meant as a supplementary teaching aid to users of the HAAS Vertical Machining Center. The information in this section may or may not apply in whole or in part to the operation of other CNC machines. Its use is intended only as an aid in the operation of the HAAS Vertical Machining Center.

VERTICAL MACHINING CENTER TRAVELS



HAAS VMC (Vertical Mill Center) shown with the X, Y, and Z axis

The machine illustration shows three directions of travel available on a vertical machine center. Now to carry the number line idea a little further, imagine such a line placed along each set of travels (or axis) of the machine.

X-Axis moves the table moving left and right. Positive X values would move the spindle to the right on our part, and negative would move the spindle to the left.

Y-Axis moves the table front and back. Moving the spindle towards the operator would be negative values in the Y axis.

Z-Axis moves the spindle head up and down. A positive Z value will move the spindle up towards the tool change position, and negative values would move the spindle towards our part.

All axis of HAAS VMCs has a max resolution of .0001” inches (or .001mm). Each has its own max range of motion. See tables below for each model.

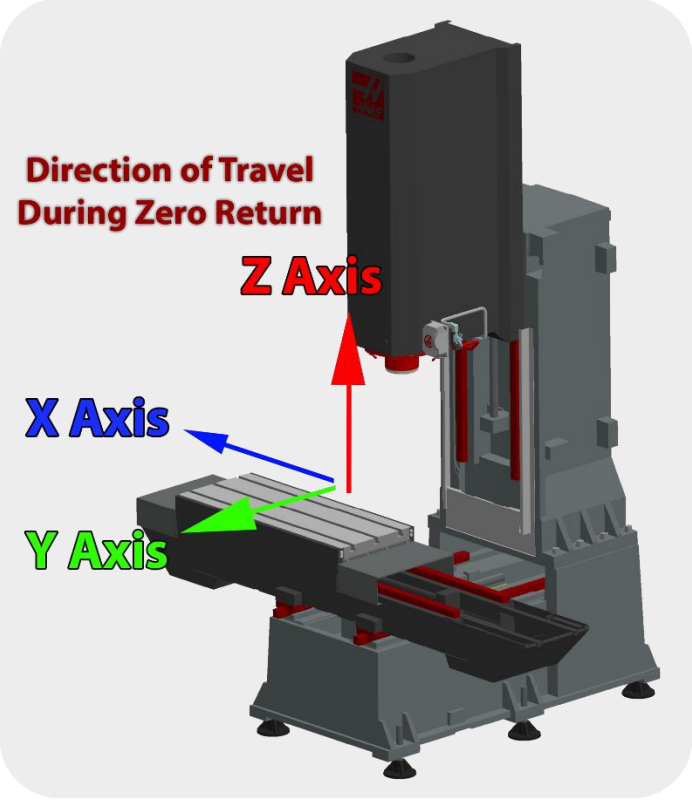
VMC MAX TRAVELS

	X-axis	Y-axis	Z-axis
VF-0/ VF-1	20”	16”	20
VF-2	30”	16	20
VF-3	40”	20	25
VF-4	50”	20	25
VF-6	64”	32	30
VF-8	64”	40	30
VF-10	120”	32	30

The Machine Coordinate System - Machine Home Position

All HAAS machines require all axes to be home out to be operational. This process is performed with **POWER UP RESTART**. All 3 axes (X, Y and Z) will move towards 0 position until it triggers the home switch; see direction of travel in illustration below.

4th and 5th Axes will rotate based on direction of travel.
*** Note that Rotaries and Trunnions can move in either direction based on parameters.



Notes

NEXT GENERATION CONTROL

INTUITIVE ONLINE HAAS NGC CONTROL

https://www.haascnc.com/content/haascnc/en/service/online-operator-s-manuals/mill-operator-s-manual/mill---introduction.html

OPERATOR'S AND SERVICE MANUALS

https://www.haascnc.com/content/dam/haascnc/en/service/manual/operator/english---mill-ngc---operator%27s-manual---2023.pdf

FRONT PANEL CONTROLS

Name	Image	Function
POWER ON		Powers the machine on.
POWER OFF		Powers the machine off.
EMERGENCY STOP		Press to stop all axis motion, disable servos, stop the spindle and tool changer, and turn off the coolant pump.
HANDLE JOG		This is used to jog axes (select in HANDLE JOG Mode). Also used to scroll through program code or menu items while editing.
CYCLE START		Starts a program. This button is also used to start a program simulation in graphics mode.
FEED HOLD		Stops all axis motion during a program. The spindle continues to run. Press CYCLE START to cancel.

PENDANT RIGHT SIDE, AND TOP PANELS

Name	Image	Function
USB		Plug compatible USB devices into this port. It has a removable dust cap.
Memory Lock		In the locked position, this key switch prevents alterations to programs, settings, parameters, and offsets.
Setup Mode		In the locked position, this key switch enables all machine safety features. Unlock allows setup (refer to “Setup Mode” in the Safety section of this manual for details).
Second Home		Press to rapid all axes to the coordinates specified in settings 268 - 270. (Refer to “Settings 268 - 270” in the Settings section of this manual for details).
Auto Door Override		Press this button to open or close the Auto Door (if equipped).
Work light		These buttons toggle the internal work light and High Intensity Lighting (if equipped).

PENDANT TOP PANEL

Beacon Light	
Provides quick visual confirmation of the machine’s current status. There are five different beacon states:	
Light Status	**Meaning**
Off	The machine is idle.
Solid Green	The machine is running.
Flashing Green	The machine is stopped but is in a ready state. Operator input is required to continue.

Flashing Red	A fault has occurred, or the machine is in Emergency Stop

ICON GUIDE

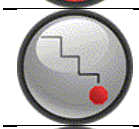
	**Setup**	Setup mode is locked; the control is in Run mode. Most machine functions are disabled or limited while the machine doors are open.
	**Setup**	Setup mode is unlocked; the control is in Setup mode. Most machine functions are available but may be limited while the machine doors are open.
	**Cycle door**	The door must be cycled at least once to ensure that the door sensor is working. This icon appears after **[power up]** if the user has not yet cycled the door.
	**Door open**	Warning, door is open.
	**Pallet load door open**	The pallet load station is open.
	**Light curtain breach**	This icon appears when the machine is idle, and the light curtain is triggered. It also appears when a program is running, and the light curtain is running. This icon disappears when the obstacle is removed from the light curtain line of sight.
	**Light curtain hold**	This icon appears when a program is running, and the light curtain is triggered. This icon will clear the next time **[cycle start]** is pressed.
	**Running**	The machine is running a program.
	**Jog**	An axis is jogging at the current jog rate.
	**Apl mode**	This icon appears when the machine is in apl mode.
	**Power saving**	The power-saving servos-off feature is active. Setting 216, servo and hydraulic shutoff, designates the time allowed before this feature activates. Press a key to activate the servos.

	**Jog**	This icon appears while the control returns to the workpiece during a run-stop-jog-continue operation.
	**Jog**	You have pressed **[FEED HOLD]** during the return portion of a run-stop-jog-continue operation.
	**Jog**	This icon prompts you to jog away during a run-stop-jog-continue operation.
	**Feed hold**	The machine is in feed hold. Axis motion has stopped, but the spindle continues to turn.
	**Feed**	The machine is executing a cutting move.
	**Rapid**	The machine is executing a non-cutting axis move (G00) at the fastest possible rate. Overrides can affect the actual rate.
	**Dwell**	The machine is executing a dwell (G04) command.
	**Restart**	The control scans the program before a restart if Setting 36 is **ON**.
	**Singbk stop**	**Single block** mode is active, and the control needs a command to continue.
	**Door hold**	Machine motion has stopped because of door rules.
	**Jog lock**	Jog lock is active. If you press an axis key, that axis moves at the current jog rate until you press **[jog lock]** again, or the axis reaches its limit.
	**Remote jog**	The optional remote jog handle is active.
	**Vector jog**	For five-axis machines, the tool will jog along the vector defined by the rotary positions.
	**Low gearbox oil flow**	This icon appears when low gearbox oil flow persists for 1 minute.
	**Low gearbox oil**	The control detected a low gearbox oil level.

	**Rotary lube**	Check and fill the rotary table lubrication oil reservoir.
	**Dirty tsc/hpfc filter**	Clean the through-spindle coolant or high-pressure flood coolant filter.
	**Low coolant concentrate**	Fille the concentrate reservoir for the coolant refill system.
	**Pulsejet Low Oil**	This icon appears when the system detects a low oil condition on the pulsejet oil reservoir.
	**Low lube**	The spindle lubrication oil system detected a low oil condition, or the axis ball screw lubrication system detected a low oil or low pressure condition.
	**Low oil**	The rotary brake oil level is low.
	**Residual pressure**	Before a lubrication cycle the system detected residual pressure from the grease pressure sensor. This can be caused by an obstruction in the axes grease lubrication system.
	**Mist filter**	Clean the mist extractor filter.
	**Vise clamp**	This icon appears when the vise is commanded to clamp.
	**Low coolant (warning)**	Coolant level is low.
	**Low pulsejet oil**	The pulsejet oil level is low.
	**Mist condenser**	This icon appears when the mist condenser is turned on.
	**Low air flow**	Inch mode - air flow is not sufficient for correct machine operation.
	**Low air flow**	Metric mode - air flow is not sufficient for correct machine operation.

	**Spindle**	When you press **[HANDLE SPINDLE]**, the jog handle varies the spindle override percentage.
	**Feed**	When you press **[HANDLE FEED]**, the jog handle varies the feed rate override percentage.
	**Handle scroll**	When you press **[handle scroll]**, the jog handle scrolls through the text.
	**Mirroring**	Mirroring mode is active. Either G101 is programmed or Setting 45, 46, 47, 48, 80, or 250 (mirror image of axis X, Y, Z, A, B, C) is set to **ON**.
	**Brake**	A rotary axis brake, or a combination of rotary axis brakes, is unclamped.
	**Brake**	A rotary axis brake, or a combination of rotary axis brakes, is clamped.
	**Hpu oil low**	The hpu oil level is low. Check the oil level and add the recommended oil for the machine.
	**Hpu oil temperature (warning)**	The oil temperature is too high to reliably operate the hpu.
	**Spindle fan failed**	This icon appears when the spindle fan stops operating.
	**Electronics overheat (warning)**	This icon appears when the control has detected cabinet temperatures are approaching levels that are potentially dangerous to the electronics. If the temperature reaches or exceeds this recommended level alarm **253 electronics overheat** will be generated. Inspect the cabinet for clogged air filters and correctly operating fans.
	**Electronics overheat (alarm)**	This icon appears when the electronics remains in the overheat state for too long. The machine will not operate until the condition is corrected. Inspect the cabinet for clogged air filters and correctly operating fans.
	**Transformer overheat (warning)**	This icon appears when the transformer is detected to be overheated for more than 1 second.

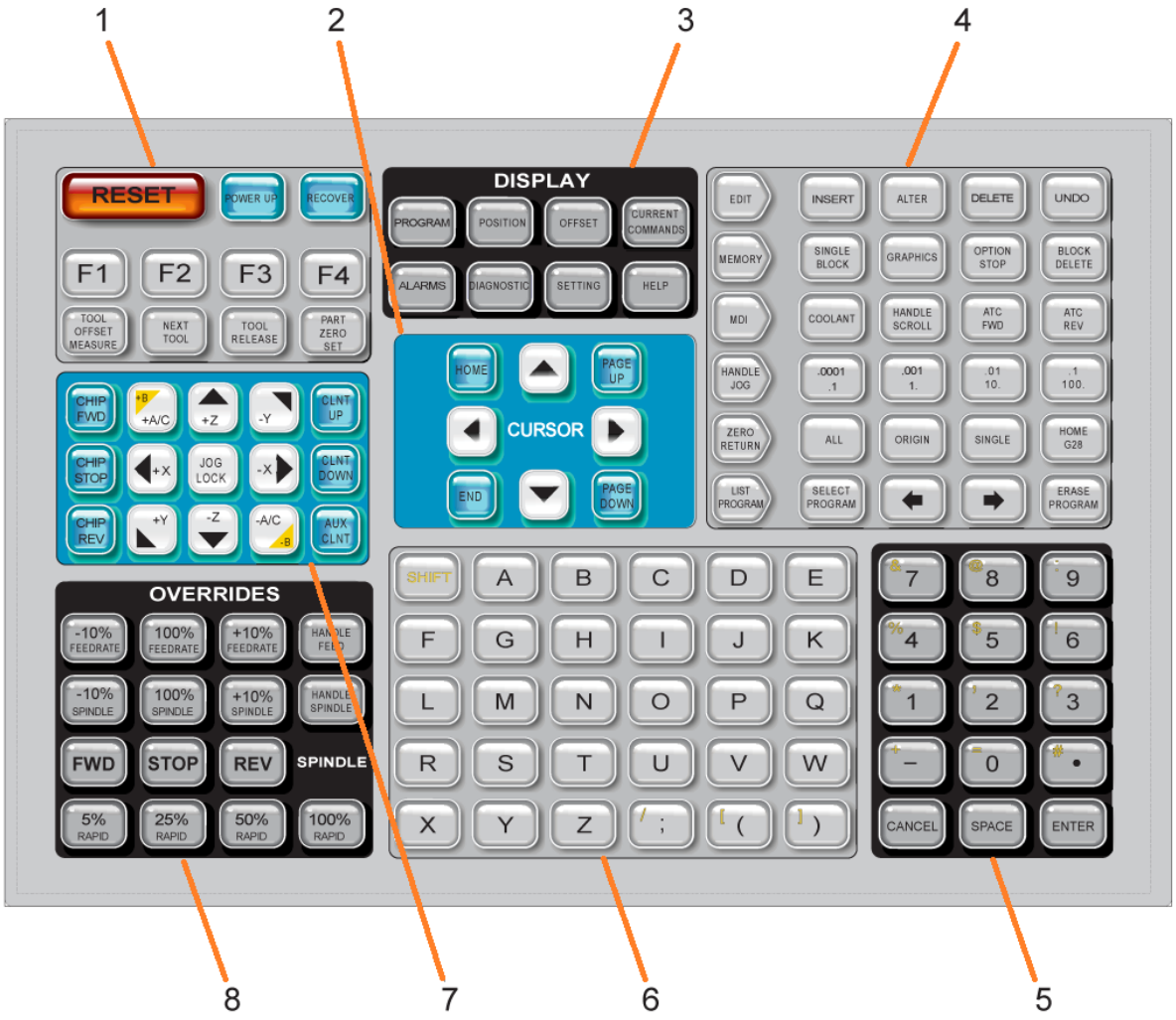
	**Transformer overheat (alarm)**	This icon appears when the transformer remains in the overheat state for too long. The machine will not operate until the condition is corrected.
	**Low voltage (warning)**	The pfdm detects low incoming voltage. If the condition continues, the machine cannot continue to operate.
	**Low voltage (alarm)**	The power fault detect module (pdfm) detects incoming voltage that is too low to operate. The machine will not operate until the condition is corrected.
	**High voltage (warning)**	The pfdm detects incoming voltage above a set limit, but still within operating parameters. Correct the condition to prevent damage to machine components.
	**High voltage (alarm)**	The pfdm detects incoming voltage that is too high to operate and could cause damage to the machine. The machine will not operate until the condition is corrected.
	**Surge protector fault detected**	Indicates a surge protector fault has been detected. This icon is active until the fault has been cleared. **warning:** if continuing to use the machine in this state. The electronics are capable of being damaged due to any electrical surge.
	**Robot Battery is Low**	Robot Battery is Low. Please replace the pulse coder batteries as soon as possible. Do **NOT** turn off the robot, otherwise it may require remastering.
	**Low air (warning)**	The air pressure to the machine is too low to reliably operate pneumatic systems. Correct this condition to prevent damage to or incorrect operation of pneumatic systems.
	**Low air (alarm)**	The air pressure to the machine is too low to operate pneumatic systems. The machine will not operate until the condition is corrected. You may need a higher-capacity air compressor.
	**High air (warning)**	The air pressure to the machine is too high to reliably operate pneumatic systems. Correct this condition to prevent damage to or incorrect operation of pneumatic systems. You may need to install a regulator at the machine's air input.

	**High air (alarm)**	The air pressure to the machine is too high to operate pneumatic systems. The machine will not operate until the condition is corrected. You may need to install a regulator at the machine's air input.
	**Pendant e-stop**	**[emergency stop]** on the pendant has been pressed. This icon disappears when **[emergency stop]** is released.
	**Apc e-stop**	**[emergency stop]** on the pallet changer has been pressed. This icon disappears when **[emergency stop]** is released.
	**Tool changer e-stop**	**[emergency stop]** on the tool changer cage has been pressed. This icon disappears when **[emergency stop]** is released.
	**Auxiliary e-stop**	**[emergency stop]** on an auxiliary device has been pressed. This icon disappears when **[emergency stop]** is released.
	**Remote jog handle-xl (rjh-xl) e-stop**	**Remote jog handle-xl (rjh-xl) e-stop** **[emergency stop]** on the rjh-xl has been pressed. This icon disappears when **[emergency stop]** is released.
	**Single blk**	**Single block** mode is active. The control executes programs (1) block at a time. Press **[cycle start]** to execute the next block.
	**Tool life (warning)**	The tool life remaining is below setting 240, or the current tool is the last one in its tool group.
	**Tool life (alarm)**	The tool or tool group has expired, and no replacement tools are available.
	**Opt stop**	**Optional stop** is active. The control stops the program at each m01 command.
	**Blk delete**	**Block delete** is active. When block delete is on, the control ignores (does not execute) the code following a forward slash (/), on that same line.
	**Tc door open**	The side-mount tool changer door is open.

	**Tc manual mode**	This icon appears when the tool carousel is in manual mode via the auto/manual switch. This switch is only on machines with tool cages.
	**Tl ccw**	The side-mount tool changer carousel is rotating counter-clockwise.
	**Tl cw**	The side-mount tool changer carousel is rotating clockwise.
	**Tool change**	A tool change is in progress.
	**Tool unclamped**	The tool in the spindle is unclamped.
	**Probe**	The probe system is active.
	**Conveyor fwd**	The conveyor is active and moving forward.
	**Conveyor rev**	The conveyor is active and moving in reverse.
	**Tsc**	The through-spindle coolant (tsc) system is active.
	**Tab**	The tool air blast (tab) system is active.
	**Air blast**	The auto air gun (aag) is active.
	**Hil light**	Indicates the optional high intensity (hil) are turned on and the doors are open. Duration is determined by setting 238.
	**Coolant**	The main coolant system is active.

Notes

KEYBOARD AND CONTROLS



Keyboard keys are grouped into these functional areas	
1	Function
2	Cursor
3	Displays
4	Modes
5	Numeric
6	Alpha Keys
7	Jog Operations
8	Overrides

SPECIAL SYMBOL INPUT

Some special symbols are not on the keypad.

Symbol	Name
_	underscore
^	caret
~	tilde
{	open curly brackets
}	closed curly brackets
\	backslash
|	pipe
<	less than
>	greater than

Do these steps to enter special symbols:

1. Press LIST PROGRAMS and select a storage device.
2. Press F3.
3. Select Special Symbols and press ENTER.
4. Enter a number to copy the associated symbol to the INPUT: bar.

For example, to change a directory's name to MY_DIRECTORY:

1. Highlight the directory with the name that you want to change.
2. Type MY.
3. Press F3.
4. Select SPECIAL SYMBOLS and press ENTER.
5. Press 1.
6. Type DIRECTORY.
7. Press F3.
8. Select RENAME and press ENTER.

FUNCTION KEYS

List of Function Keys and How They Operate

Name	Key	Function
Reset	**[RESET]**	Clears alarms. Clears input text. Sets overrides to default values if Setting 88 is ON.
Power up	**[POWER UP]**	Zero returns all axes and initializes the machine control.
Recover	**[RECOVER]**	Enters tool changer recovery mode.
F1- F4	**[F1 - F4]**	These buttons have different functions depending on the tab that is active.
Tool Offset Measure	**[TOOL OFFSET MEASURE]**	Records tool length offsets during part setup.
Next Tool	**[NEXT TOOL]**	Selects the next tool from the tool changer.
Tool Release	**[TOOL RELEASE]**	Releases the tool from the spindle when in MDI, ZERO RETURN, or HAND JOG mode.
Part Zero Set	**[PART ZERO SET]**	Records work coordinate offsets during part setup.

CURSOR KEY

The cursor keys let you move between data fields, scroll through programs, and navigate through tabbed menus.

Name	Key	Function
Home	**[HOME]**	Moves the cursor to the top-most item on the screen; in editing, this is the top left block of the program.
Cursor Arrows	**Up, Down, Left, Right**	Moves one item, block, or field in the associated direction. The keys depict arrows, but this manual refers to these keys by their spelled-out names.
Page Up, Page Down	**[PAGE UP] / [PAGE DOWN]**	Used to change displays or move up/down one page when viewing a program.
End	**[END]**	Moves the cursor to the bottom-most item on the screen. In editing, this is the last block of the

DISPLAY KEYS

You use the Display keys to see the machine displays, operational information, and help pages.

Name	Key	Function
Program	**[PROGRAM]**	Selects the active program pane in most modes.
Position	**[POSITION]**	Selects the positions display.
Offsets	**[OFFSET]**	Displays the Tool Offset and Work Offset tabbed menu.
Current Commands	**[CURRENT COMMANDS]**	Displays menus for Devices, Timers, Macros, Active Codes, Calculators, Advanced Tool Management (ATM), Tool Table, and Media.
Alarms	**[ALARMS]**	Displays the Alarm viewer and Message screens.
Diagnostics	**[DIAGNOSTIC]**	Displays tabs for Features, Compensation, Diagnostics, and Maintenance.
Settings	**[SETTING]**	Displays and allows changing of user settings.
Help	**[HELP]**	Displays help information.

MODE KEYS

Mode keys change the operational state of the machine. Each mode key is arrow shaped and points to the row of keys that perform functions related to that mode key. The current mode is always displayed in the top left of the screen, in Mode:Key display form.

NOTE: EDIT and **LIST PROGRAM** can also act as display keys, where you can access program editors and the device manager without changing the machine mode. For example, while the machine runs a program, you can use the device manager (LIST PROGRAM) or background editor (EDIT) without stopping the program.

Name	Key	Function
EDIT MODE KEYS		
Edit	**[EDIT]**	Lets you edit programs in the editor. You can access the Visual Programming System (VPS) from the EDIT tabbed menu.
Insert	**[INSERT]**	Enters text from the input line or the clipboard into the program at the cursor position.
Alter	**[ALTER]**	Replaces the highlighted command or text with text from the input line or the clipboard. **NOTE:** ALTER does not work for offsets.
Delete	**[DELETE]**	Deletes the item that the cursor is on, or deletes a selected program block.
Undo	**[UNDO]**	Undoes up to the last 40 edit changes, and deselects a highlighted block. **NOTE**: UNDO does not work for deleted highlighted blocks or to recover a deleted program.

Name	Key	Function
MEMORY MODE KEYS		
Memory	**[MEMORY]**	Selects memory mode. You run programs in this mode, and the other keys in the MEM row control the ways in which the program is run. Shows OPERATION:MEM in upper left display.
Single Block	**[SINGLE BLOCK]**	Toggles single block on or off. When single block is on, the control runs only one program block each time you press CYCLE START.
Graphics	**[GRAPHICS]**	Opens Graphics mode.
Optional Stop	**[OPTION STOP]**	Toggles optional stop on or off. When optional stop is on, the machine stops when it reaches M01 commands.
Block Delete	**[BLOCK DELETE]**	Toggles Block Delete On or Off. When Block Delete is On, the control ignores (does not execute) the code following a Forward Slash (/), on that same line.

Name	Key	Function
MDI MODE KEYS		
Manual Data Input	**[MDI]**	In MDI mode, you run unsaved programs or blocks of code entered from the control. Shows EDIT:MDI in upper left display.
Coolant	**[COOLANT]**	Turns the optional coolant on and off. Also, SHIFT + COOLANT turns on and off the optional Auto Air Gun / Minimum Quantity Lubrication functions.
Handle Scroll	**[HANDLE SCROLL]**	Toggles Handle Scroll mode. This lets you use the jog handle to move the cursor in menus while the control is in jog mode.
Automatic Tool Changer Forward	**[ATC FWD]**	Rotates the tool carousel to the next tool.
Automatic Tool Changer Reverse	**[ATC REV]**	Rotates the tool carousel to the previous tool.

Name	Key	Function
HANDLE JOG MODE KEYS		
Handle Jog	**[HANDLE JOG]**	Enters Jog mode.
.0001/.1 .001/1 .01/10 .1/100	**[.0001] /[.1], [.001] / [1.], [.01] / [10.], [.1] / [100].]**	Selects the increment for each click of the jog handle. When the mill is in MM mode the first number is multiplied by ten when jogging the axis (e.g., .0001 becomes 0.001 mm). The bottom number sets speed after you press JOG LOCK and an axis jog key or you press and hold an axis jog key. Shows SETUP:JOG in the upper left of the display.

Name	Key	Function
ZERO RETURN MODE KEYS		
Zero Return	**[ZERO RETURN]**	Selects Zero Return mode, which displays axis location in four different categories: Operator, Work G54, Machine, and Dist (distance) To Go. Select the tab to switch between the categories. Shows SETUP: ZERO in the upper-left display.
All	**[ALL]**	Returns all axes to machine zero. This is like POWER UP, except a tool change does not occur.
Origin	**[ORIGIN]**	Sets selected values to zero.
Single	**[SINGLE]**	Returns one axis to machine zero. Press the desired axis letter on the Alpha keyboard and then press SINGLE
Home G28	**[HOME G28]**	Returns all axes to zero in rapid motion. HOME G28 will also home a single axis in the same manner as SINGLE. **CAUTION**: Make sure the axis motion paths are clear when you press this key. There is no warning or prompt before axis motion begins.
LIST PROGRAM MODE KEYS		
List Programs	**[LIST PROGRAM]**	Accesses a tabbed menu to load and save programs.
Select Programs	**[SELECT PROGRAM]**	Makes the highlighted program the active program.
Back	**[BACK ARROW]**	Navigates to the screen you were on before the current one. This key operates like the BACK button on a web browser.
Forward	**[FORWARD ARROW]**	Navigates to the screen you went to after the current screen, if you have used the back arrow. This key operates like the FORWARD button on a web browser.
Erase Program	**[ERASE PROGRAM]**	Deletes the selected program in List Program mode. Deletes the entire program in MDI mode.

NUMERIC KEYS

Use the numeric keys to type numbers, along with some special characters (printed in yellow on the main key). Press SHIFT to enter the special characters.

Name	Key	Function
Numbers	**0-9**	Types numbers.
Minus sign	-	Adds a minus (-) sign to the input line.
Decimal point	.	Adds a decimal point to the input line.
Cancel	**[CANCEL]**	Deletes the last character typed.
Space	**[SPACE]**	Adds a space to input.
Enter	**[ENTER]**	Answers prompts and writes input.
Special Characters	Press **[SHIFT]**, then a numeric key	Insert the yellow character on the upper left of the key. These characters are used for comments, macros, and certain special features.
+	**[SHIFT]**, then -	Inserts +
=	**[SHIFT]**, then 0	Inserts =
#	**[SHIFT]**, then .	Inserts #
*****	**[SHIFT]**, then 1	Inserts *
‘	**[SHIFT]**, then 2	Inserts ‘
?	**[SHIFT]**, then 3	Inserts ?
%	**[SHIFT]**, then 4	Inserts %
$	**[SHIFT]**, then 5	Inserts $
!	**[SHIFT]**, then 6	Inserts !
&	**[SHIFT]**, then 7	Inserts &
@	**[SHIFT]**, then 8	Inserts @
:	**[SHIFT]**, then 9	Inserts :

ALPHA KEYS

Use the alpha keys to type the letters of the alphabet, along with some special characters (printed in yellow on the main key). Press SHIFT to enter the special characters.

Name	Key	Function
Alphabet	**A-Z**	Uppercase letters are the default. Press SHIFT and a letter key for lowercase.
End-of-block (EOB)	**;**	This is the end-of-block character, which signifies the end of a program line.
Parentheses	**(,)**	Separate CNC program commands from user comments. They must always be entered as a pair.
Shift	**[SHIFT]**	Accesses additional characters on the keyboard or shifts to lower case alpha characters. The additional characters are seen in the upper left of some of the alpha and number keys.
Special Characters	Press **[SHIFT]**, then an alpha key	Insert the yellow character on the upper left of the key. These characters are used for comments, macros, and certain special features.
Forward Slash	**[SHIFT]**, then ;	Inserts /
Left Bracket	**[SHIFT]**, then (	Inserts [

Right Bracket	**[SHIFT]**, then)	Inserts]

JOG KEYS

Name	Key	Function
Chip Auger Forward	**[CHIP FWD]**	Starts the chip removal system in the forward direction (out of the machine).
Chip Auger Stop	**[CHIP STOP]**	Stops the chip removal system.
Chip Auger Reverse	**[CHIP REV]**	Starts the chip removal system in the "reverse" direction.
Axis Jog Keys	**+X/-X, +Y/-Y, +Z/-Z, +A/C/-A/C and +B/-B ([SHIFT] +A/C/-A/C)**	Jog axes manually. Press and hold the axis button, or press and release to select an axis and then use the jog handle.
Jog Lock	**[JOG LOCK]**	Works with the axis jog keys. Press JOG LOCK, then an axis button, and the axis moves until you press JOG LOCK again.
Coolant Up	**[CLNT UP]**	Moves the optional Programmable Coolant (P-Cool) nozzle up.
Coolant Down	**[CLNT DOWN]**	Moves the optional P-Cool nozzle down.
Auxiliary Coolant	**[AUX CLNT]**	Press this key in MDI mode to toggle the Through-Spindle Coolant (TSC) system operation, if equipped. Press SHIFT + AUX CLNT to toggle the Through Tool Air Blast (TAB) function, if equipped. Both functions also work in Run-Stop-Jog-Continue mode.

OVERRIDE KEYS

Overrides let you temporarily adjust the speeds and feeds in your program. For example, you can slow down rapids while you prove out a program, or adjust the feedrate to experiment with its effect on part finish, etc.

You can use Settings 19, 20, and 21 to disable the feedrate, spindle, and rapid overrides, respectively.

FEED HOLD acts as an override that stops rapid and feed moves when you press it. FEED HOLD also stops tool changes and part timers, but not tapping cycles or dwell timers.

Press CYCLE START to continue after a FEED HOLD. When the Setup Mode key is unlocked, the door switch on the enclosure also has a similar result but displays Door Hold when the door is opened. When the door is closed, the control is in Feed Hold and CYCLE START must be pressed to continue. Door Hold and FEED HOLD do not stop any auxiliary axes.

You can override the standard coolant setting by pressing COOLANT. The coolant pump remains either on or off until the next M-code or operator action (see Setting 32).

Use Settings 83, 87, and 88 to have M30 and M06 commands, or RESET, respectively, change overridden values back to their defaults.

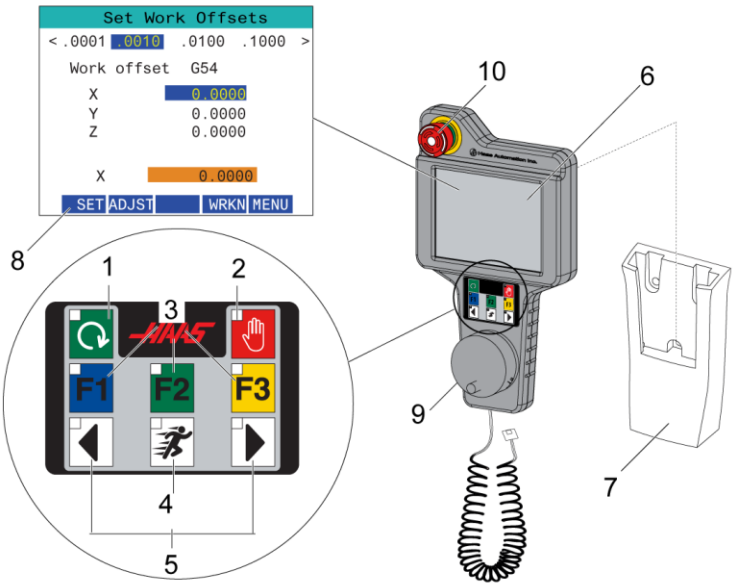
Name	Key	Function
-10% Feedrate	**[-10% FEEDRATE]**	Decreases the current feedrate by 10%.
100% Feedrate	**[100% FEEDRATE]**	Sets an overridden feedrate back to the programmed feed rate.
+10% Feedrate	**[+10% FEEDRATE]**	Increases the current feedrate by 10%.
Handle Control Feed Rate	**[HANDLE FEED]**	Lets you use the jog handle to adjust the feedrate in 1% increments.
-10% Spindle	**[-10% SPINDLE]**	Decreases the current spindle speed by 10%
100% Spindle	**[100% SPINDLE]**	Sets the overridden spindle speed back to the programmed speed.
+10% Spindle	**[+10% SPINDLE]**	Increases the current spindle speed by 10%.
Handle Spindle	**[HANDLE SPINDLE]**	Lets you use the jog handle to adjust the spindle speed in 1% increments.
Forward	**[FWD]**	Starts the spindle in the clockwise direction.
Stop	**[STOP]**	Stops the spindle.
Reverse	**[REV]**	Starts the spindle in the counterclockwise direction.
Rapids	**[5% RAPID]/ [25% RAPID]/ [50% RAPID] / [100% RAPID]**	Limits machine rapids to the value on the key.

RJH-TOUCH XL OVERVIEW

The Remote Jog Handle (RJH-Touch XL) is an optional accessory that gives you hand-held access to the control for faster and easier setups.

Your machine must have Next Generation Control software 100.21.000.1000 or higher to use all the RJH-Touch XL functions. The next sections explain how to operate the RJH-Touch.

The illustration shows these components:



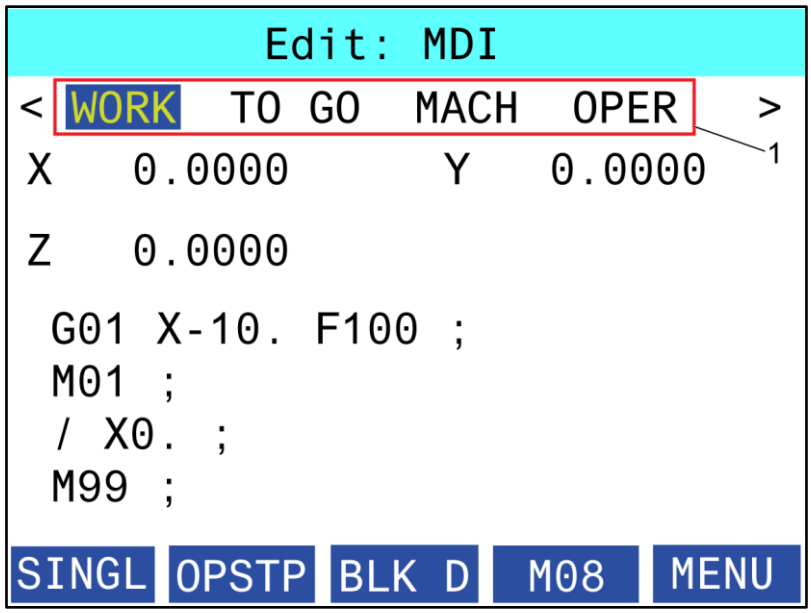
1. Cycle Start. Has the same function as **[CYCLE START]** on the pendant.
2. Feed Hold. Has the same function as **[FEED HOLD]** on the control pendant.
3. Function Keys. These keys are for future use.
4. Rapid jog button. This key doubles the jogging speed when pressed simultaneously with one of the jog direction buttons.
5. Jog Direction Keys. These keys work the same as keypad jog arrow keys. You can press and hold to jog the axis.
6. LCD Touch Screen Display.
7. Holster. To activate the RJH-XL, lift it out of the holster. To deactivate the RJH-XL, put it back into the holster.
8. Function Tabs. These tabs have different functions in different modes. Press the function tab that corresponds to the function you want to use.
9. Handle Jog Wheel. This Handle Jog works like the jog handle on the control pendant. Each click of the handle jog moves the selected axis one unit of the selected jog rate.
10. E-Stop. Has the same function as **[EMERGENCY STOP]** on the pendant.

Most RJH functions are available in Handle Jog mode. In other modes, the RJH screen displays information about the active or MDI program.

Note: The RJH-XL cannot be removed when the machine is on.

Note: The images show MDI, but the following instructions apply to both MDI and MEM.

When the MDI or MEM is pressed on the pendant there are 4 main tabs [1] on the RJH: **WORK**, **TO GO**, **MACH**, and **OPER**.



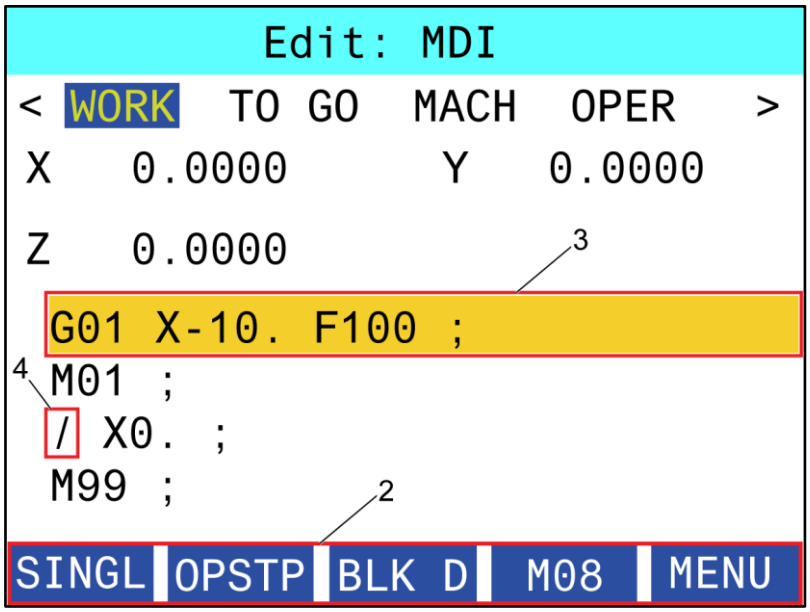
When **[WORK]** is highlighted, the screen shows the axis positions in relation to part zero.

When **[TO GO]** is highlighted, the screen shows the distance remaining before the axes reach their commanded position.

When **[MACH]** is highlighted, the screen displays the axes positions in relation to machine zero.

When **[OPER]** is highlighted, the screen displays the distance the axes have been jogged.

At the bottom of the screen there are 5 buttons [2]: **SINGL**, **OPSTP**, **BLK D**, **M08**, **MENU**.



When **SINGL** is pressed it will execute the highlighted line [3] and stop and when **[CYCLE START]** is pressed it'll execute the next line and stop, and so on.

OPSTP is optional stop, when this is pressed the program will stop at every M01 encountered.

Note: On machines with an autodoor, **OPSTP** will stop at every M01 and open the door(s).

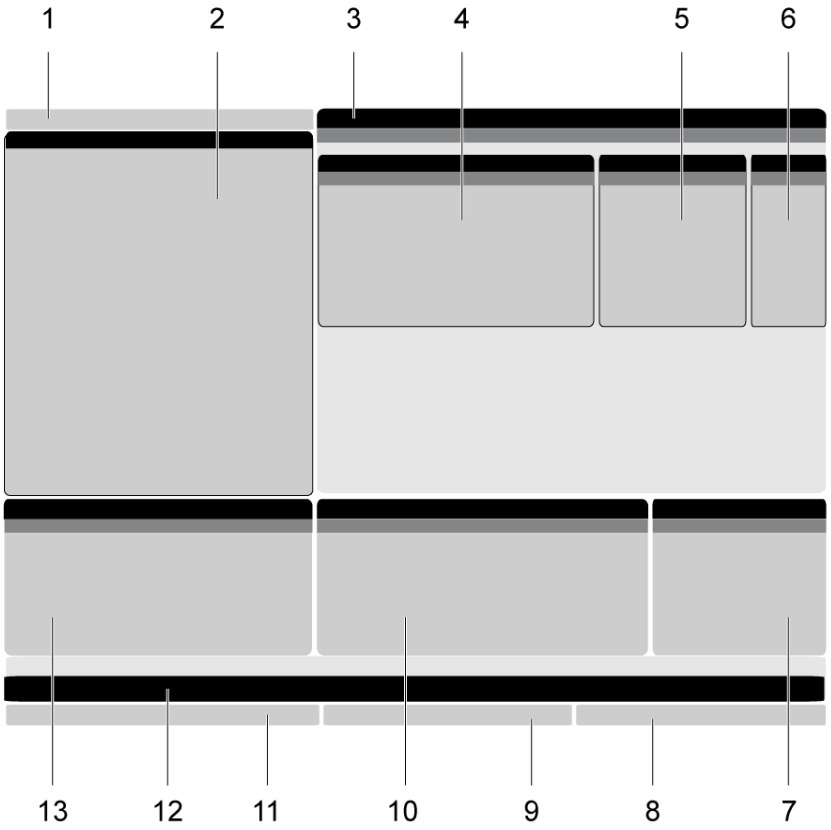
BLK D is block delete, when this is pressed any line beginning with a forward slash [4] will be skipped when the program is ran.

When **M08** is pressed, the coolant turns on and the button will then read **M09** which will turn the coolant off when pressed.

CONTROL DISPLAY

The control display is organized into panes that change with the different machine and display modes.

Basic Control Display Layout in Operation: Mem Mode (While a Program Runs)



1	Mode, Network, and Time Status Bar
2	Program Display
3	Main Display (size varies)/ Program/ Offsets/Current Commands/ Settings/ Graphics/ Editor/ VPS/ Help
4	Active Codes
5	Active Tool
6	Coolant
7	Timers, Counters / Tool Management
8	Alarm Status
9	System Status Bar
10	Position Display / Axis Load
11	Input Bar
12	Icon Bar
13	Spindle Status

The active pane has a white background. You can work with data in a pane only when that pane is active, and only one pane is active at a time. For example, when you select the Tool Offsets tab, the

offsets table background turns white. You can then make changes to the data. In most cases, you change the active pane with the display keys.

TABBED MENU BASIC NAVIGATION

The Haas control uses tabbed menus for several modes and displays. Tabbed menus keep related data together in an easy-to-access format. To navigate these menus:

1. Press a display or mode key.

The first time you access a tabbed menu, the first tab (or sub-tab) is active. The highlight cursor is at the first available option in the tab.

2. Use the cursor keys or the HANDLE JOG control to move the highlight cursor within an active tab.

3. To choose a different tab in the same tabbed menu, press the mode or display key again.
NOTE: If the cursor is at the top of the menu screen, you can also press the UP-cursor arrow key to select a different tab.

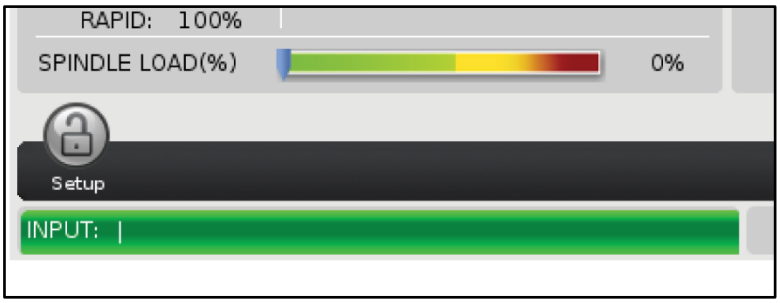
The current tab becomes inactive.

4. Use the cursor keys to highlight a tab or a sub-tab and press the DOWN cursor arrow key to use the tab.

NOTE: You cannot make the tabs active in the POSITIONS tabbed display.

5. Press a different display or mode key to work with a different tabbed menu.

INPUT BAR

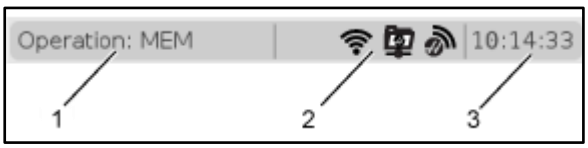


The input bar is the data entry section located in the bottom-left corner of the screen. This is where your input appears as you type it.

Mode, Network, and Time Status Bar

This status bar in the top left of the screen is divided into three sections: mode, network, and time.

The Mode, Network, and Time Status bar shows [1] the current machine mode, [2] network status icons, and [3] the current time.



MODE, KEY ACCESS, AND MODE DISPLAY

Mode [1]

The Haas control organizes machine functions into three modes: Setup, Edit, and Operation. Each mode shows on one screen all of the information you need to do tasks under that mode. For example, in Setup mode, you have access to the work offset table, the tool offset table, and position information. Edit mode gives you access to the program editor and optional systems like Visual Programming (VPS) (which contains Wireless Intuitive Probing (WIPS)). Operation mode includes Memory (MEM), the mode in which you run programs.

Mode	Keys	Display [1]	Function
Setup	ZERO RETURN	SETUP: ZERO	Provides all control features for machine setup.
	HANDLE JOG	SETUP: JOG	
Edit	EDIT	ANY	Provides all program editing, management, and transfer functions.
	MDI	EDIT: MDI	
	LIST PROGRAM	ANY	
Operation	MEMORY	OPERATION: MEM	Provides all control features necessary to run a program.
	EDIT	OPERATION: MEM	Provides background editing of active programs.
	LIST PROGRAM	ANY	Provides background editing of programs.

NETWORK

If you have networking installed on your Next Generation Control, icons in the center networking partition of the bar give you networking status. See the table for meanings of the networking icons.

ICON	Network Status
	The machine is connected to a wired network with an Ethernet cable.
	The machine is connected to a wireless network with 70 - 100% signal strength.
	The machine is connected to a wireless network with 30 - 70% signal strength.
	The machine is connected to a wireless network with 1 - 30% signal strength.
	The machine is connected to a wireless network, but it is not receiving data packets.
	The machine is successfully registered with MyHaas and is communicating with the server.
	The machine had previously registered with MyHaas and has a problem connecting to the server.
	The machine is connected to a remote net share.

SETTINGS DISPLAY

Press SETTING, then select the SETTINGS tab. Settings change the way the machine behaves; refer to the “Settings” section for a more detailed description.

COOLANT DISPLAY

The coolant display appears in the upper-right of the screen in OPERATION:MEM mode.
The first line tells you if the coolant is ON or OFF.
The next line shows the position number of the optional Programmable Coolant Spigot (P-COOL). The positions are from 1 to 34. If the option is not installed, no position number appears.
In the coolant gauge, a black arrow shows the coolant level. Full is 1/1 and empty is 0/1. To avoid coolant flow problems, keep the coolant level above the red range. You can also see this gauge in DIAGNOSTICS mode under the GAUGES tab.

OFFSET DISPLAY

To access the offset tables, press **OFFSET** and select the **TOOL** tab or the **WORK** tab.

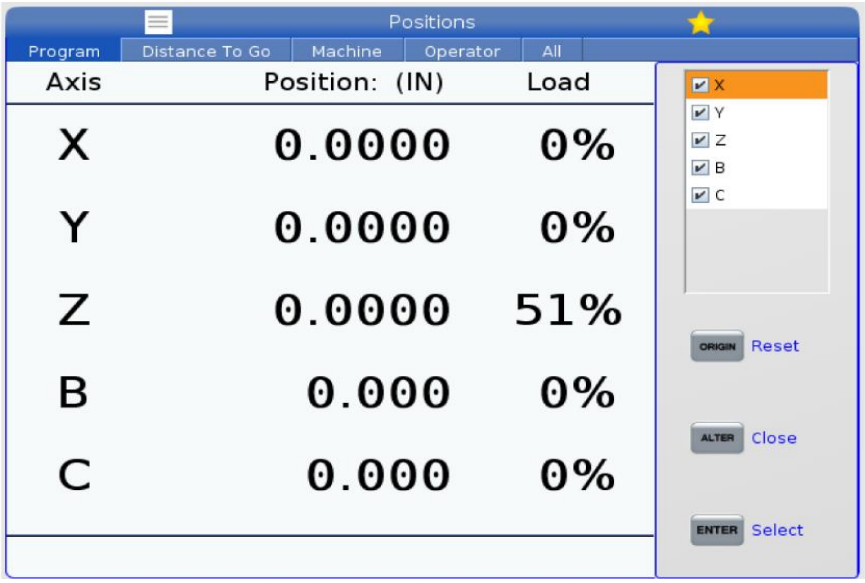
Name	Function
TOOL	Display and work with tool numbers and tool length geometry.
WORK	Display and work with part zero locations.

POSITION DISPLAY

The Position display shows the current axis position relative to four reference points (Work, Distance-to-go, Machine, and Operator). In any mode, press POSITION and use cursor keys to access the different reference points displayed in tabs. The last tab display shows all the reference points on the same screen.

Coordinate Display	Function
WORK (G54)	This tab displays the axis positions relative to part zero. On power-up, this position uses work offset G54 automatically. It displays the axis positions relative to the most recently-used work offset.
DIST TO GO	This tab displays the distance remaining before the axes reach their commanded position. When in SETUP: JOG mode, you can use this position display to show a distance moved. Change modes (MEM, MDI) and then switch back to SETUP: JOG mode to zero this value.
MACHINE	This tab displays the axis positions relative to machine zero.
OPERATOR	This tab shows the distance you have jogged the axes. This does not necessarily represent the actual distance the axis is from machine zero, except when the machine is first powered on.
ALL	This tab displays all reference points on the same screen.

AXIS DISPLAY SELECTION



You can add or remove axes in the Positions displays. While a Positions display tab is active, press ALTER.

The axis display selection window comes in from the right side of the screen.

Use the cursor arrow keys to highlight an axis, and press ENTER to toggle it on and off for display. The positions display will show axes that have a check mark.
Press ALTER to close the axis display selector.
NOTE: You can display a maximum of (5) axes.

CURRENT COMMANDS

This section describes the Current Commands pages and the types of data they show. The information from most of these pages also appears in other modes.

Press **CURRENT COMMANDS** to access the tabbed menu of available Current Commands displays.

Devices -The Mechanisms tab on this page shows hardware devices on the machine that you can command manually. For example, you can manually extend and retract the Parts Catcher or Probe Arm. You can also manually rotate the spindle clockwise or counterclockwise at a desired RPM.

Timers Display -This page shows:

- The current date and time.
- The total power on time.
- Total cycle start time.
- Total feed time.
- M30 counters. Each time the a program reaches an M30 command, both of these counters increment by one.
- Macro variable displays.

You also see these timers and counters in the lower-right section of the display in the OPERATION:MEM, SETUP:ZERO, and EDIT:MDI modes.

Macros Display -This page shows a list of the macro variables and their values. The control updates these variables as programs run. You can modify the variables in this display.

Active Codes -This page lists the active program codes. A smaller version of this display is included on the OPERATION:MEM and EDIT:MDI mode screens. Also when you press PROGRAM in any Operation mode, you see the active program codes.
Advanced Tool Management -This page contains information the control uses to predict tool life. Here is where you create and manage tool groups, and where you enter the maximum tool load percentage expected for each tool. For more information, refer to the **Advanced Tool Management** section in the Operation chapter of this manual.

Calculator -This page contains the Standard, Milling/Turning, and Tapping calculators.

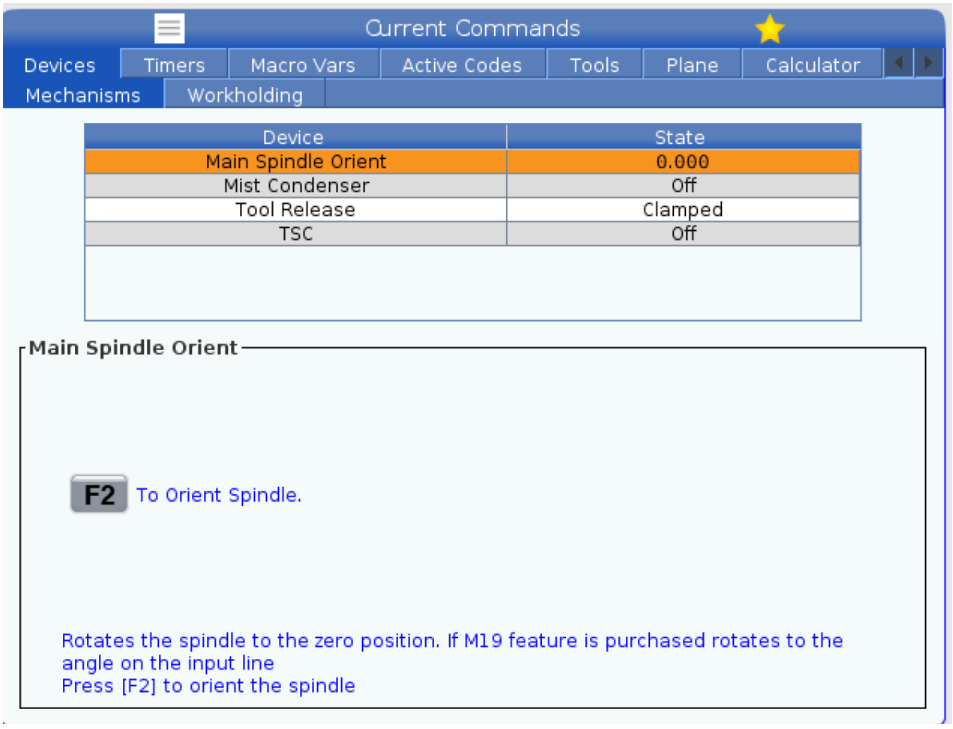
Media -This page contains the Media Player.

DEVICES - MECHANISMS

The Mechanisms page displays possible machine components and options on your machine. Select the listed mechanism using the UP and DOWN arrows for more information on its operation and use. Pages give detailed instructions on the functions of the machine components, quick tips, as well as links to other pages to help you learn about and utilize your machine.

- Select the Devices tab in the Current Commands menu.
- Select the Mechanisms that you want to use.

The Main Spindle option in Devices allows you to rotate the spindle clockwise or counterclockwise at a chosen RPM. The maximum RPM is limited by the machine's maximum RPM settings.



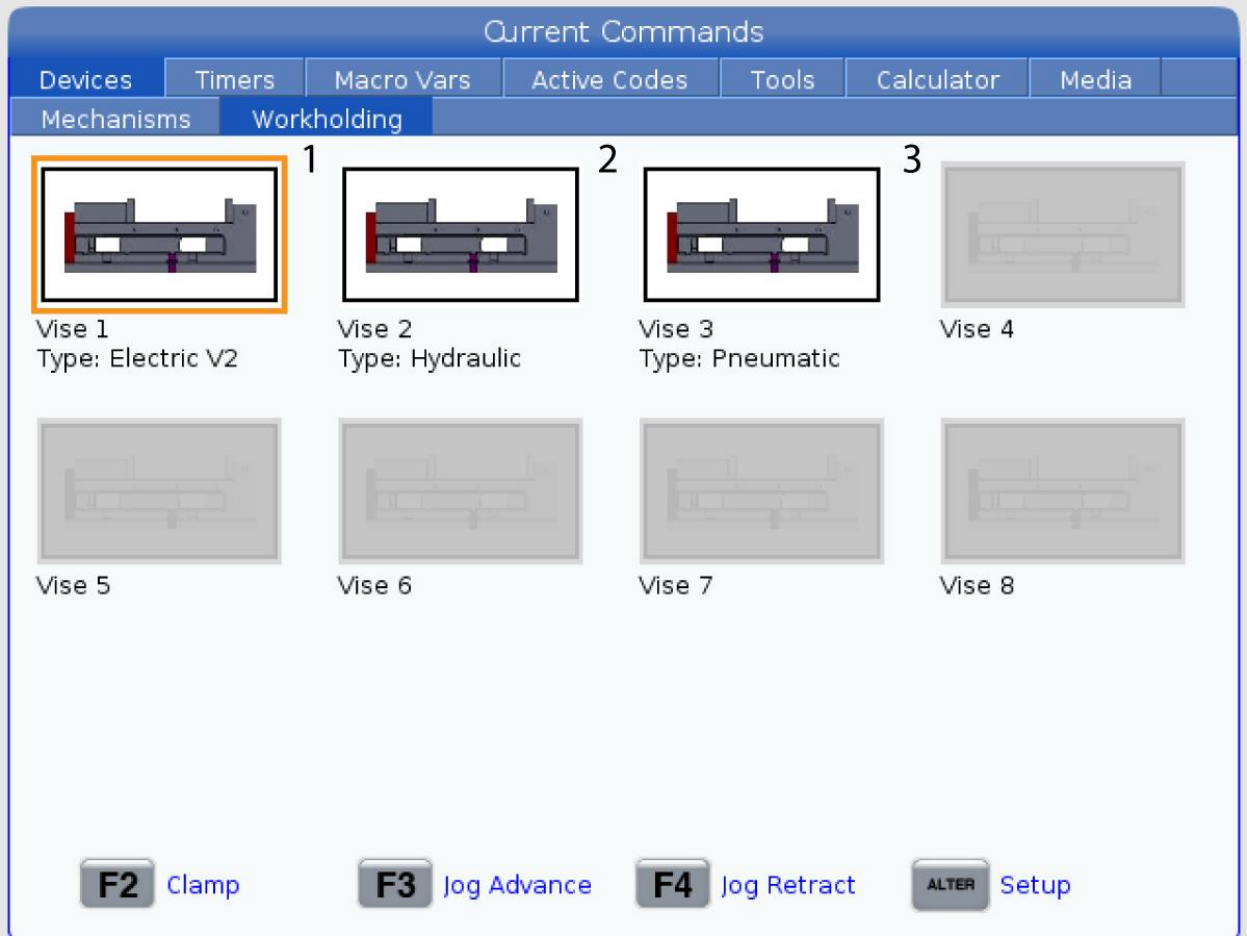
- Use the cursor arrow keys to move from field to field.
- Enter the RPM you want to rotate the spindle and press **F2**.
- Hold down **F3** to rotate the spindle clockwise. Hold down **F4** to rotate the spindle counterclockwise. The spindle comes to a stop when the button is released.

DEVICES - WORKHOLDING

Starting in software version **100.20.000.1110** a workholding tab was added to the control to support multiple workholding devices. The control will support Haas E-Vise [1], Hydraulic [2] and Pneumatic [3] vises.

The machine supports up to 3 foot pedals each would toggle Vise1, Vise2, and Vise3 respectively. If you have a single pedal you will need to enable Vise1 to the vise you want be actuated by the foot pedal.

NOTE: The E-vise is used on the Mill APL and Robot systems but can also be used as a stand-alone product.



You can actuate up to 8 workholding devices.

To access the Workholding page press Current Commands and navigate to **Devices > Workholding.**

From the Workholding display tab you will be able to:

- Setup the workholding devices
- Enable and Disable workholding devices
- Clamp and Unclamp

- Jog Advance / Retract (E-Vises only)

TIME ADJUSTMENT

Follow this procedure to adjust the date or time.

1. Select the Timers page in Current Commands.
2. Use the cursor arrow keys to highlight the Date:, Time:, or Time Zone field.
3. Press **[EMERGENCY STOP]**.
4. In the Date: field, type the new date in the format **MM-DD-YYYY**, including the hyphens.
5. In the Time: field, type the new time in the format **HH:MM**, including the colon.

Press **[SHIFT]** and then 9 to type the colon.

6. In the Time Zone: field, press **[ENTER]** to select from the list of time zones. You can type search terms in the pop-up window to narrow the list. For example, type PST to find Pacific Standard Time. Highlight the time zone you want to use.
7. Press **[ENTER]**.

TIMER AND COUNTER RESET

You can reset the power-on, cycle-start, and feed cutting timers. You can also reset the M30 counters.

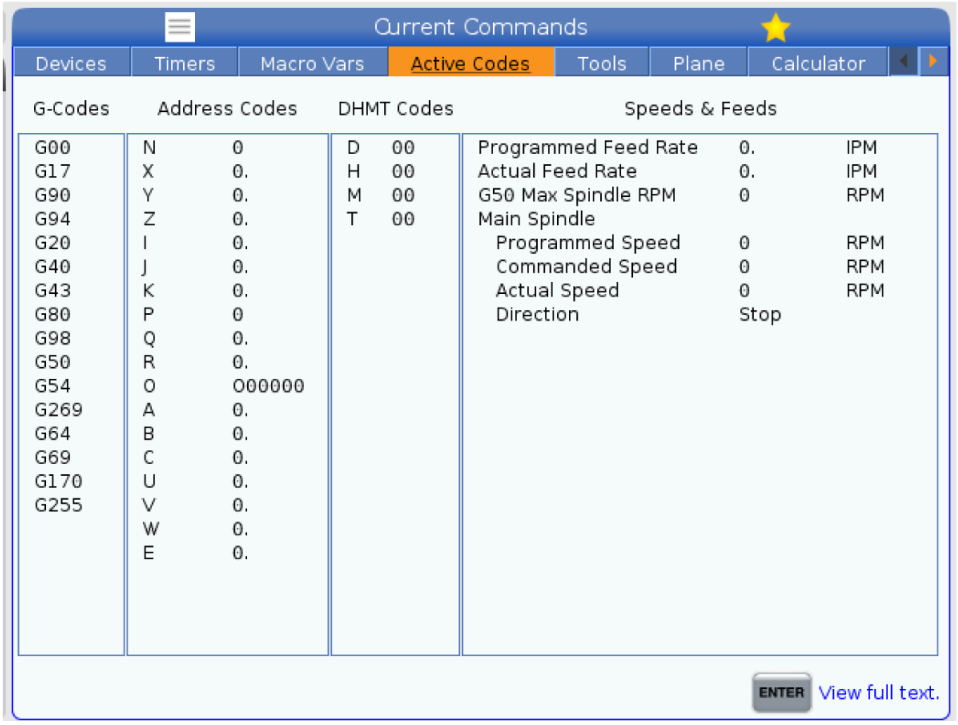
1. Select the Timers page in Current Commands.
2. Use the cursor arrow keys to highlight the name of the timer or counter that you want to reset.
3. Press ORIGIN to reset the timer or counter.

tip: You can reset the M30 counters independently to track finished parts in two different ways; for example, parts finished in a shift and total parts finished.

CURRENT COMMANDS - ACTIVE CODES

This display gives read-only, real-time information about the codes that are currently active in the program; specifically,

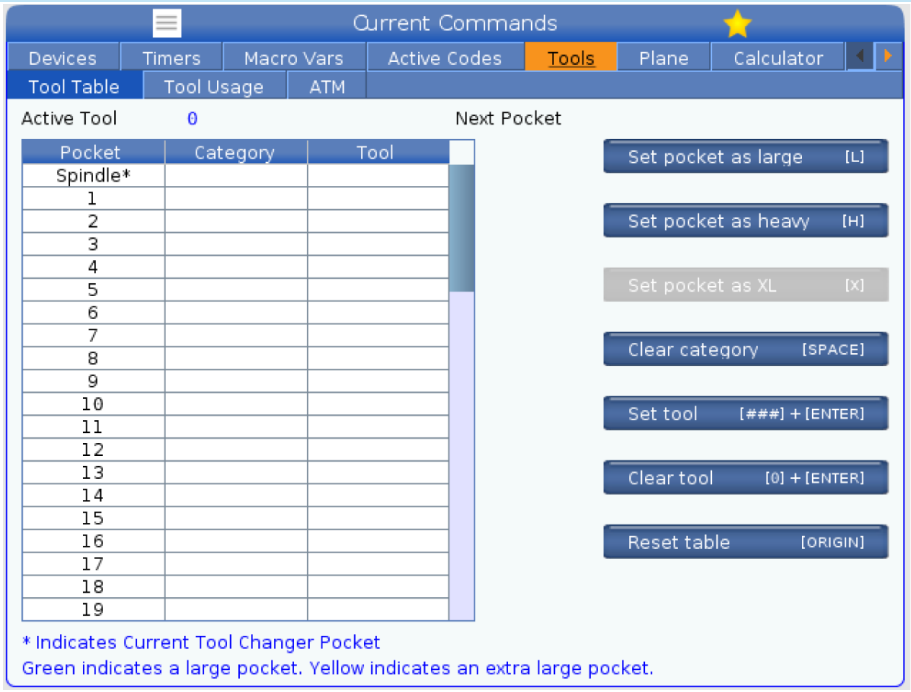
- the codes that define the current motion type (rapid vs linear feed vs circular feed)
- positioning system (absolute vs incremental)
- cutter compensation (left, right or off)
- active canned cycle, and work offset.



This display also gives the active Dnn, Hnn, Tnn, and

most recent M-code. If an alarm is active, this shows a quick display of the active alarm instead of the active codes.

TOOLS - TOOL TABLE



This section tells you how to use the tool table to give the control information about your tools.

To access the Tool Pocket Table, press CURRENT COMMANDS and choose the Tool Table tab.

Active Tool- Tells you the tool number that is installed in the spindle.

Active Pocket - this shows you the next pocket number.

Set pocket as Large [L] - Use this flag when a large tool has a diameter greater than 3" for 40-taper machines and greater than 4" for 50-taper machines. Scroll to the pocket of interest and press L to set the flag.

CAUTION: You cannot place a large tool in the tool changer if one or both surrounding pockets already contain tools. Doing so causes the tool changer to crash. Large tools must have the surrounding pockets empty. However, large tools can share adjoining empty pockets.

Set pocket as heavy [H] - Use this flag when a Heavy, Small Diameter 40-taper tool (4 lb. or heavier) or a 50-taper tool (12 lb. or heavier) is loaded into the spindle. Scroll to the pocket of interest and press H to set the flag.

Set pocket as XL [X] - Use this flag when two adjacent pockets at each side of the tool are needed. Scroll to the pocket of interest and press X to set the flag.
NOTE: This option only appears if your machine is a 50-taper.
Clear category [Space] - Highlight the desired tool and press SPACE to clear the flag.

Set tool [###] + [ENTER]- Highlight the desired pocket and type in the tool number + Enter to set the desired tool number.

NOTE: You cannot assign a tool number to more than one pocket. If you enter a tool number that is already defined in the tool pocket table, you see an Invalid tool error.

Clear tool [0] + [ENTER]- Highlight the desired pocket and press 0 + Enter to clear out the tool number.

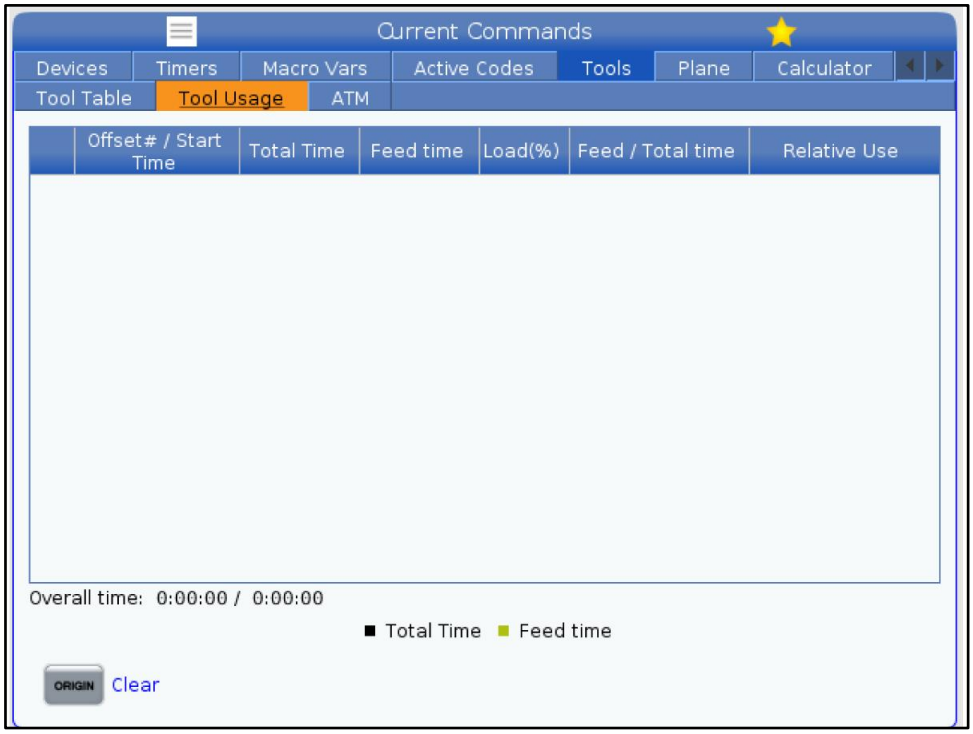
Reset table [Origin] - Press ORIGIN with the cursor in the center column to use the ORIGIN menu. This menu lets you:

Sequence All Pockets - Makes all of the tool numbers sequential based on their pocket location, starting with 1.

Zero All Pockets - Removes all of the tool numbers from all of the pocket numbers.

Clear Category Flags - Removes the category designations from all the tools.
* Indicates current tool changer pocket.

TOOLS – TOOL USAGE



The Tool Usage tab contains information about the tools used in a program. This display will tell you information about each tool used in a program and statistics about each time it was used. It starts to collect information when user Main program start and clear information when meet codes M99, M299, M199.

To get to the Tool Usage display press CURRENT COMMANDS then go to Tools and then Tool Usage tab.

Start Time - When the tool was inserted to the spindle.

Total Time - The total time the tool has been in the spindle.

Feed time - Tool usage time.

Load% - The maximum load of the Spindle during a tool usage.

NOTE: This value is retrieved every second. The actual load compared to the recorded may vary.

Feed/Total time - A graphic representation of the feed time of the tool over the total time.

Engagement:

- Black Bar- The tool usage versus another tools.
- Gray Bar - This bar shows how long the tool was used in this usage related to other usages.

MACROS INTERFACE

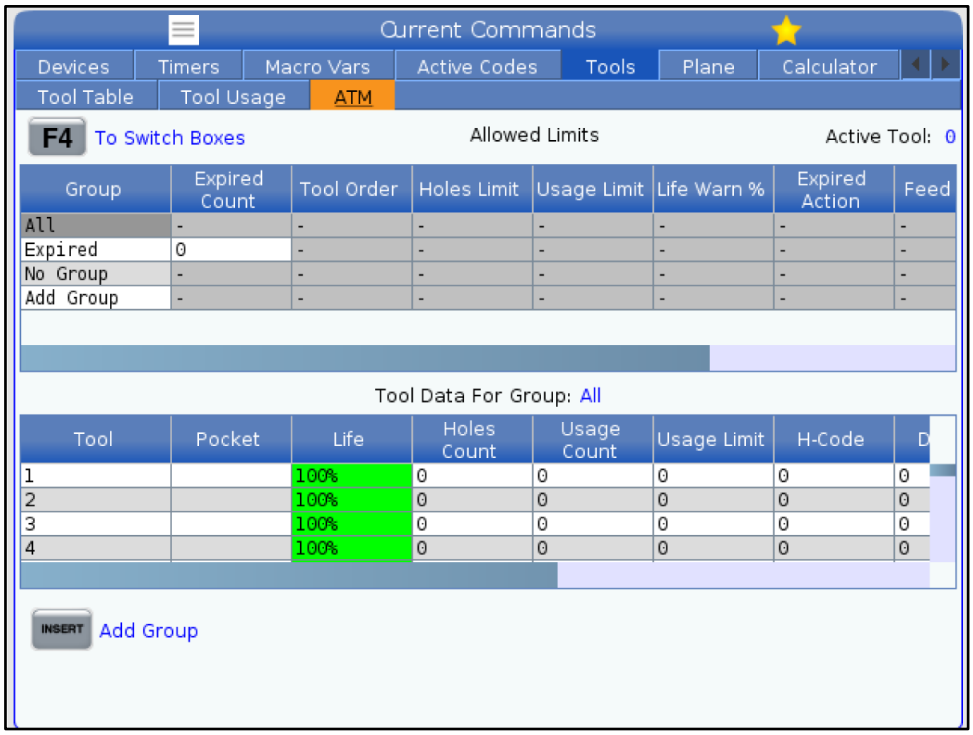
You can use these macros variables to set and collect the tool usage data.

Macro Variable	Function
#8608	Set the desired tool
#8609	Current tool number - if result more 0 (the tool was used)
#8610	Total time mentioned in #8609 tool number
#8611	Feed time of mentioned tool number
#8612	Total time
#8605	Next usage of a tool
#8614	Usage start time stamp
#8615	Usage Total time
#8616	Usage Feed time
#8617	Usage Max load

ADVANCED TOOL MANAGEMENT

Advanced Tool Management (ATM) lets you set up groups of duplicate tools for the same or a series of jobs.

ATM classifies duplicate or backup tools into specific groups. In your program, you specify a group of tools instead of a single tool. ATM tracks the tool use in each tool group and compares it to your defined limits. When a tool reaches a limit, the control considers it "expired." The next time your program calls that tool group, the control chooses a non-expired tool from the group.



When a tool expires:

- The beacon will flash.
- ATM puts the expired tool in the EXP group
- Tool groups that contain the tool appear with a red background.

To use ATM, press CURRENT COMMANDS, and then select ATM in the tabbed menu. The ATM window has two sections: Allowed Limits and Tool Data.

ALLOWED LIMITS

This table gives data about all of the current tool groups, including default groups and user-specified groups. ALL is a default group that lists all of the tools in the system. EXP is a default group that lists all of the tools that are expired. The last row in the table shows all of the tools that are not assigned to tool groups. Use the cursor arrow keys or END to move the cursor to the row and see these tools.

For each tool group in the ALLOWED LIMITS table, you define limits that determine when a tool expires. The limits apply to all tools assigned to this group. These limits affect every tool in the group.

The columns in the ALLOWED LIMITS table are:

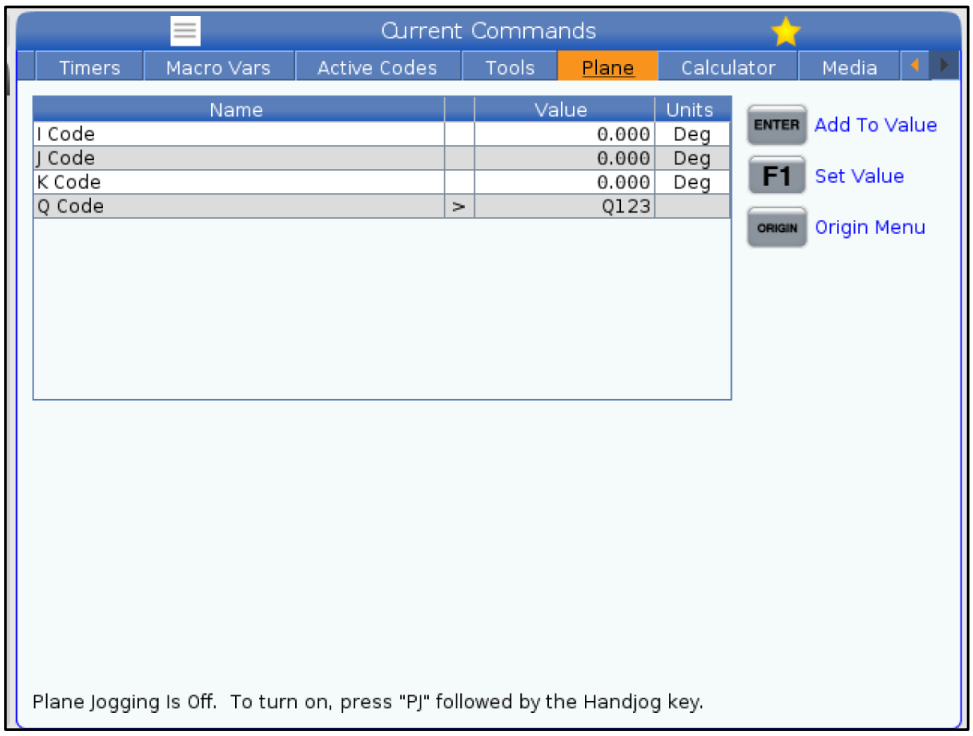
- **GROUP** - Displays the tool group's ID number. this is the number you use to specify the tool group in a program.
- **EXPIRED COUNT** - Tells you how many tools in the group are expired. If you highlight the ALL row, you see a list of all of the expired tools in all groups.
- **TOOL ORDER** - Specifies the tool to use first. If you select ORDERED, ATM uses the tools in tool number order. You can also have ATM automatically use the NEWEST or OLDEST tool in the group.
- **HOLES LIMIT** - The maximum number of holes a tool is allowed to drill before it expires.
- **USAGE LIMIT** - The maximum number of times the control can use a tool before it expires.
- **LIFE WARN %** - The minimum value for tool life remaining in the group before the control gives a warning message.
- **EXPIRED ACTION** - The automatic action when a tool reaches its maximum tool load percentage. Highlight the tool action box to change and press ENTER. Use the UP and DOWN cursor keys to select the automatic action from the pull down menu (ALARM, FEEDHOLD, BEEP, AUTOFEED, NEXT TOOL).
- **FEED LIMIT** - The total amount of time, in minutes, that the tool can be in a feed.
- **TOTAL TIME LIMIT** - The total amount of time, in minutes, that the control can use a tool.
- **MAIN SPINDLE LOAD LIMIT** - The allowed load limit for tools in the group before the control issues a warning.
- **SUB SPINDLE LOAD LIMIT** - The allowed load limit for tools in the group before the control issues a warning.
- **LIVE TOOL LOAD LIMIT** - The allowed load limit for tools in the group before the control issues a warning.

TOOL DATA

This table gives information about each tool in a tool group. To look at a group, highlight it in the ALLOWED LIMITS table, and then press F4.

- **TOOL** - Shows the tool numbers used in the group.
- **POCKET** - Shows the pocket numbers used in the group.
- **LIFE** - The percentage of life left in a tool. This is calculated by the CNC control, using actual tool data and the allowed limits the operator entered for the group.
- **HOLES COUNT** - The number of holes the tool has drilled/ tapped/ bored.
- **USAGE COUNT** - The total number of times that a program has called the tool (number of tool changes).
- **USAGE LIMIT** - The total number of times that a tool can be used (number of tool changes).
- **FEED TIME** - The amount of time, in minutes, the tool has been in a feed.
- **TOTAL TIME** - The total amount of time, in minutes, the tool has been used.
- **MAIN SPINDLE MAX LOAD** - The maximum load, in percent, exerted on the tool.
- **MAIN SPINDLE LOAD LIMIT** - The maximum load allowed for the tool before warning issued.
- **SUB SPINDLE MAX LOAD** - The maximum load, in percent, exerted on the tool.
- **SUB SPINDLE LOAD LIMIT** - The maximum load allowed for the tool before warning issued.
- **LIVE TOOL MAX LOAD** - The maximum load, in percent, exerted on the tool.
- **LIVE TOOL LOAD LIMIT** - The maximum load allowed for the tool before warning issued.

PLANE



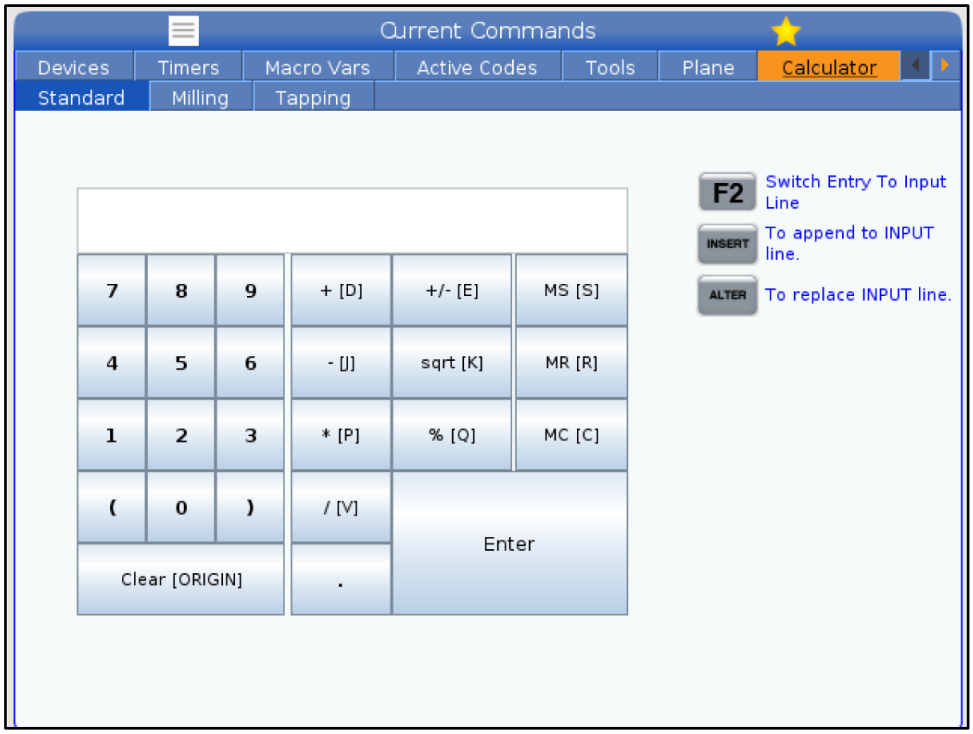
The Plane tab allows for a machine with a Gimbal Spindle define custom planes for jogging.

The plane tab can be used in conjunction with G268 run in a program or from filling out the required fields.

Each one of the required fields has a help text at the bottom of the table.

To enter plane jogging mode type in "**PJ**" followed by **[HAND JOG]**.

CALCULATOR



The calculator tab includes calculators for basic mathematical functions, milling, and tapping.

Select the calculator tab in the Current Commands menu.

Select the calculator tab that you want to use: Standard, Milling, or Tapping.

The standard calculator has functions like a simple desktop calculator; with available operations such as addition, subtraction, multiplication, and division, as well as square root and percentage. The calculator lets you easily transfer operations and results to the input line so that you can put them into programs. You can also transfer results into the Milling and Tapping calculators.

Use the number keys to type operands into the calculator.

To insert an arithmetic operator, use the letter key that appears in brackets next to the operator you want to insert.

These keys are:

KEY	FUNCTION	KEY	FUNCTION
D	Add	K	Square Root
J	Subtract	Q	Percentage
P	Multiply	S	Memory Store (MS)
V	Divide	R	Memory Recall (MR)
E	Toggle sign (+/-)	C	Memory Clear (MC)

After you have entered data into the calculator input field, you can do any of the following:

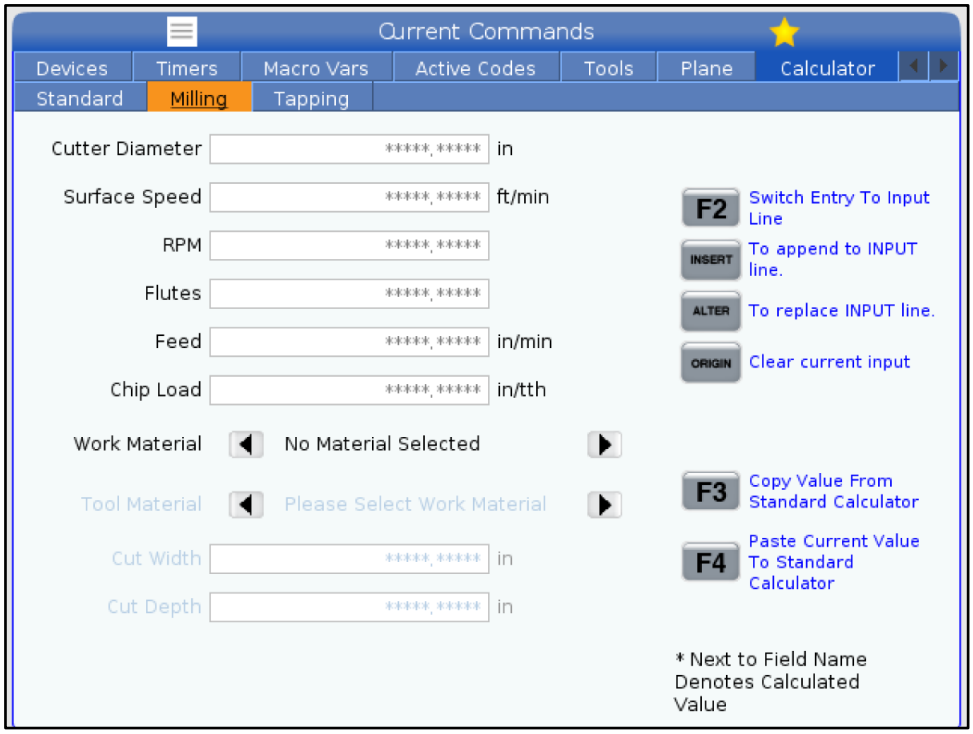
NOTE: These options are available for all calculators.

- Press ENTER to return the result of your calculation.
- Press INSERT to append the data or the result to the end of the input line.
- Press ALTER to move the data or the result to the input line. This overwrites the current contents of the input line.
- Press ORIGIN to reset the calculator.

Keep the data or the result in the calculator input field and select a different calculator tab. The data in the calculator input field remains available to transfer into the other calculators.

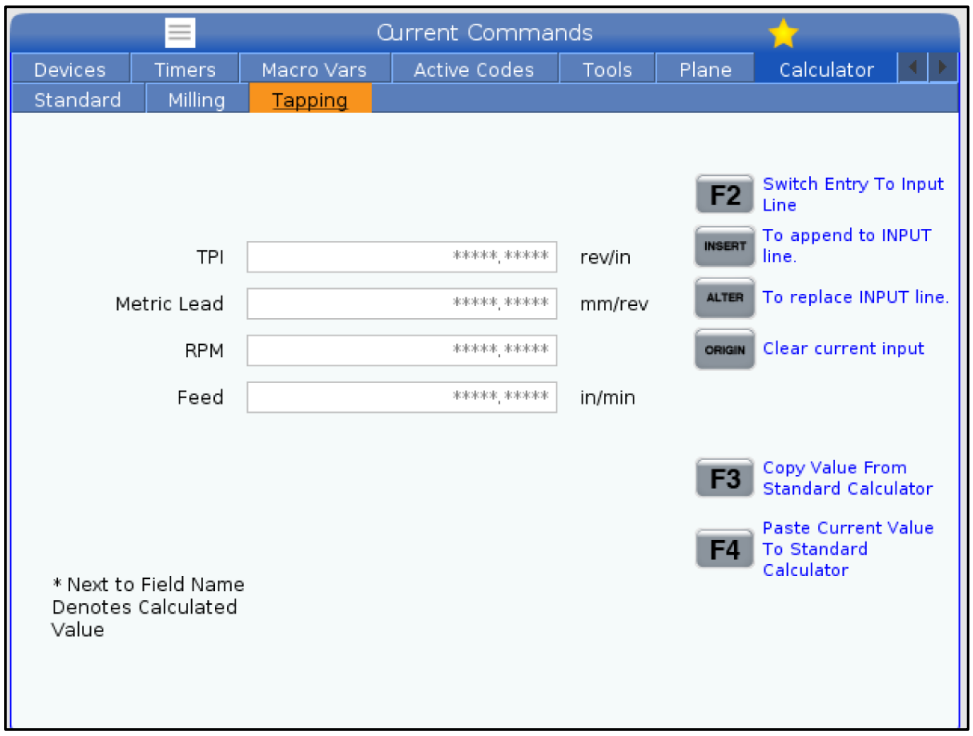
MILLING /TURNING CALCULATOR

The milling/turning calculator lets you automatically calculate machining parameters based on given information. When you have entered enough information, the calculator automatically displays results in the relevant fields. These fields are marked with an asterisk (*).



- Use the cursor arrow keys to move from field to field.
- Type known values in the appropriate fields. You can also press F3 to copy a value from the standard calculator.
- In the Work Material and Tool Material fields, use the LEFT and RIGHT cursor arrow keys to choose from the available options.
- Calculated values appear highlighted in yellow when they are outside of the recommended range for the workpiece and tool material. Also, when all of the calculator fields contain data (calculated or entered), the milling calculator displays the recommended power for the operation.

TAPPING CALCULATOR



The tapping calculator lets you automatically calculate tapping parameters based on given information. When you have entered enough information, the calculator automatically displays results in the relevant fields. These fields are marked with an asterisk (*).

- Use the cursor arrow keys to move from field to field.
- Type known values in the appropriate fields. You can also press F3 to copy a value from the standard calculator.
- When the calculator has enough information, it puts calculated values in the appropriate fields.

MEDIA DISPLAY

M130 Lets you display video with audio and still images during program execution. Some examples of how you can use this feature are:

Providing visual cues or work instructions during program operation

Providing images to aid part inspection at certain points in a program

Demonstrating procedures with video

The correct command format is M130(file.xxx), where file.xxx is the name of the file, plus the path, if necessary. You can also add a second comment in parentheses to appear as a comment in the media window.

Example: M130(Remove Lifing Bolts Before Starting Op 2)(User Data/My Media/loadOp2.png);

note: M130 uses the subprogram search settings, Settings 251 and 252 in the same way that M98 does. You can also use the Insert Media File command in the editor to easily insert an M130 code that includes the file path.

$FILE Lets you display video with audio and still images outside of program execution.

The correct command format is ($FILE file.xxx), where file.xxx is the name of the file, plus the path, if necessary. You can also add a comment between the first parentheses and the dollar sign to appear as a comment in the media window.

To display the media file, highlight the block while in memory mode and press enter. $FILE media display block will be ignored as comments during program execution.
Example: (Remove Lifting Bolts Before Starting Op 2 $FILE User Data/My Media/loadOp2.png);

Standard	Profile	Resolution	Bitrate
MPEG-2	Main-High	1080 i/p, 30 fps	50 Mbps
MPEG-4 / XviD	SP/ASP	1080 i/p, 30 fps	40 Mbps
H.263	P0/P3	16 CIF, 30fps	50 Mbps
DivX	3/4/5/6	1080 i/p, 30fps	40 Mbps
Baseline	8192 x 8192	120 Mpixel/sec	-
PNG	-	-	-
JPEG	-	-	-

NOTE: For the fastest loading times, use files with pixel dimensions divisible by 8 (most unedited digital images have these dimensions by default), and a maximum resolution of 1920 x 1080.

Your media appears in the Media tab under Current Commands. The media displays until the next M130 displays a different file, or M131 clears the media tab contents.

ALARMS AND MESSAGES DISPLAY

Use this display to learn more about machine alarms when they occur, view your machine's entire alarm history, look up definitions of alarms that can occur, view created messages, and show the keystroke history.

Press ALARMS, then select a display tab:

The ACTIVE ALARM tab shows the alarms that currently affect machine operation. Use PAGE UP and PAGE DOWN to see the other active alarms.

The MESSAGES tab shows the messages page. The text you put on this page stays there when you power the machine off. You can use this to leave messages and information for the next machine operator, etc.

The ALARM HISTORY tab shows a list of the alarms that have recently affected machine operation. You can also search for an alarm number or alarm text. To do this type in the alarm number or the desired text and press F1.

The ALARM VIEWER tab shows a detailed description of all the alarms. You can also search for an alarm number or alarm text. To do this type in the alarm number or the desired text and press F1.

The KEY HISTORY tab shows up to the last 2000 keystrokes.

ADD MESSAGES

You can save a message in the MESSAGES tab. Your message stays there until you remove it or change it, even when you turn the machine off.

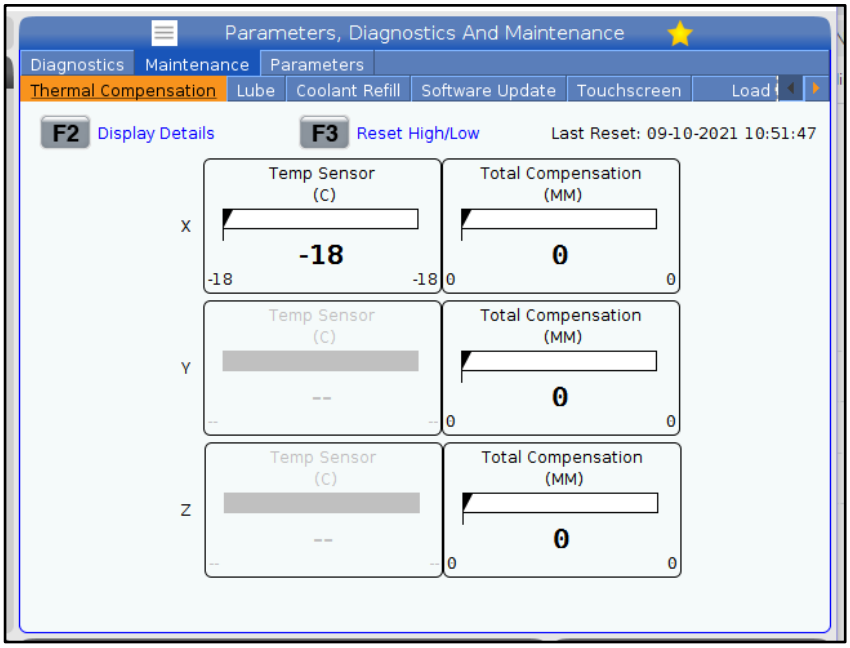
1. Press ALARMS, select the MESSAGES tab, and press the DOWN cursor arrow key.
2. Type your message.
   Press CANCEL to backspace and delete. Press DELETE to delete an entire line. Press ERASE PROGRAM to delete the entire message.

MAINTENANCE

Thermal Compensation tab under Maintenance in Diagnostics that was released in software version **100.21.000.1130.**

This tab has two options to switch between, a simple gauge version and a more detailed view.

Note: For now, this tab is purely for informational purposes.



DEVICE MANAGER (LIST PROGRAM)

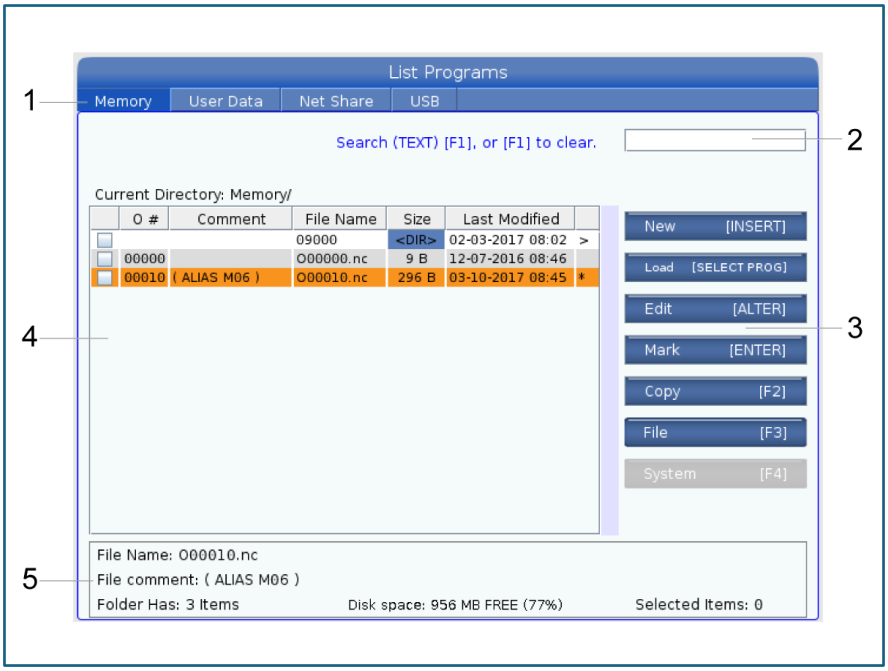
You use the device manager (LIST PROGRAM) to access, save, and manage data on the CNC control, and on other devices attached to the control. You also use the device manager to load and transfer programs between devices, set your active program, and back up your machine data.
In the tabbed menu at the top of the display, the device manager (LIST PROGRAM) shows you only the available memory devices. For example, if you do not have a USB memory device connected to the control pendant, the tabbed menu does not show a USB tab. For more information on navigating tabbed menus, refer to Chapter 5.1.

The device manager (LIST PROGRAM) shows you the available data in a directory structure. At the root of the CNC control are the available memory devices in a tabbed menu. Each device can contain combinations of directories and files, many levels deep. This is like the file structure you find in common personal computer operating systems.

DEVICE MANAGER OPERATION

Press LIST PROGRAM to access the device manager. The initial device manager display shows the available memory devices in a tabbed menu. These devices can include machine memory, the User Data directory, USB memory devices connected to the control, and files available on the connected network. Select a device tab to work with the files on that device.

Device Manager Initial Screen Example:
[1] Available Device Tabs,
[2] Search Box,
[3] Function Keys,
[4] File Display,
[5] File Comments (only available in Memory).



Use the cursor arrow keys to navigate the directory structure:

Use the UP and DOWN cursor arrow keys to highlight and interact with a file or a directory in the current root or directory.

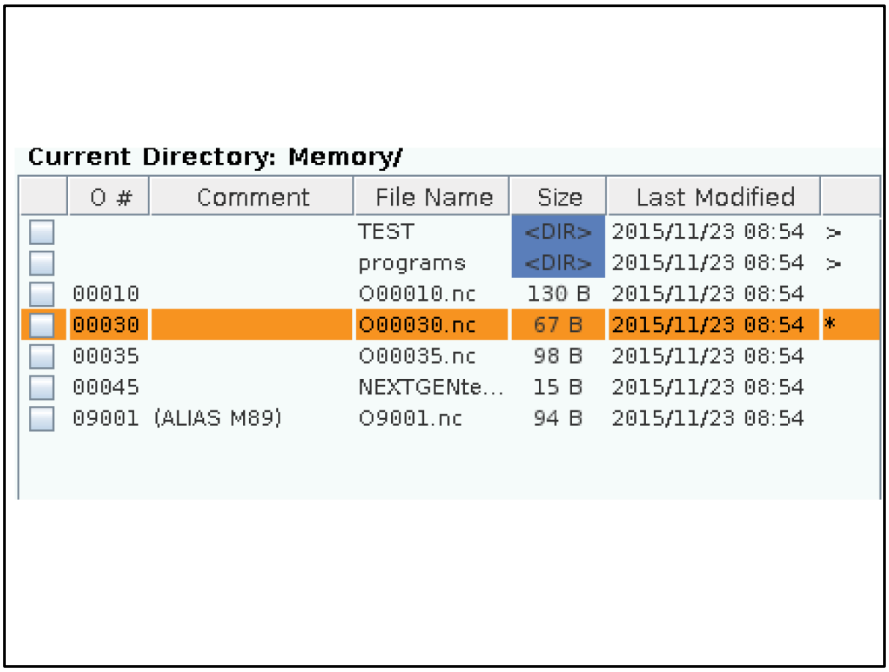
Roots and directories have a right-hand arrow character (>) in the far-right column of the file display. Use the RIGHT cursor arrow key to open a highlighted root or directory. The display then shows the contents of that root or directory. Use the LEFT cursor arrow key to return to the previous root or directory. The display then shows the contents of that root or directory

The CURRENT DIRECTORY message above the file display tells you where you are in the directory structure; for example: MEMORY/CUSTOMER 11/NEW PROGRAMS shows that you are in the subdirectory NEW_PROGRAMS inside the directory CUSTOMER 11, in the root of MEMORY.

FILE DISPLAY COLUMNS

When you open a root or directory with the RIGHT cursor arrow key, the file display shows you a list of the files and directories in that directory. Each column in the file display has information about the files or directories in the list

The columns are:



File selection check box (no label): Press ENTER to toggle a check mark on and off in the box. A check mark in a box indicates that the file or directory selected for operations on multiple files (usually copy or delete).

Program O Number (O #): This column lists the program numbers of the programs in the directory. The letter 'O' is omitted in the column data. Only available in the Memory tab.

File comment (Comment): This column lists the optional program comment that appears in the first line of the program. Only available in the Memory tab.

File Name (File Name): This is an optional name that the control uses when you copy the file to a memory device other than the control. For example, if you copy program O00045 to a USB memory device, the filename in the USB directory is NEXTGENtest.nc.

File Size (Size): This column shows the amount of storage space that the file takes up. Directories in the list have the designation <DIR> in this column.

NOTE: This column is hidden by default, press the F3 button and select Show File Details to display this column.

Date Last Modified (Last Modified): This column shows the last date and time when the file was changed. The format is YYYY/MM/DD HR:MIN.

NOTE: This column is hidden by default, press the F3 button and select Show File Details to display this column.

Other Information (no label): This column gives you some information about a file's status. The active program has an asterisk (*) in this column. A letter E in this column means that the program is in

the program editor. A greater-than symbol (>) indicates a directory. A letter S indicates that a directory is part of Setting 252. Use the RIGHT or LEFT cursor arrow keys to enter or exit the directory.

CHECK MARK SELECTION

The check box column on the far left of the file display lets you select multiple files.

Press ENTER to place a check mark in the file's check box. Highlight another file and press ENTER again to put a check mark in that file's check box. Repeat this process until you have selected all the files you want to select.

You can then do an operation (usually copy or delete) on all those files at the same time. Each file that is part of your selection has a check mark in the check box. When you choose an operation, the control does that operation on all the files with check marks.

For example, if you want to copy a set of files from the machine memory to a USB memory device, you will put a check mark on all of the files that you want to copy, then press F2 to start the copy operation.

To delete a set of files, put a check mark on all of files you want to delete, then press DELETE to start the delete operation.

NOTE: A check-mark selection only marks the file for further operation; it does not make the program active.

NOTE: If you have not selected multiple files with check marks, the control does operations only on the currently highlighted directory or file. If you have selected files, the control does operations only on the selected files and not on the highlighted file, unless it is also selected.

SELECT THE ACTIVE PROGRAM

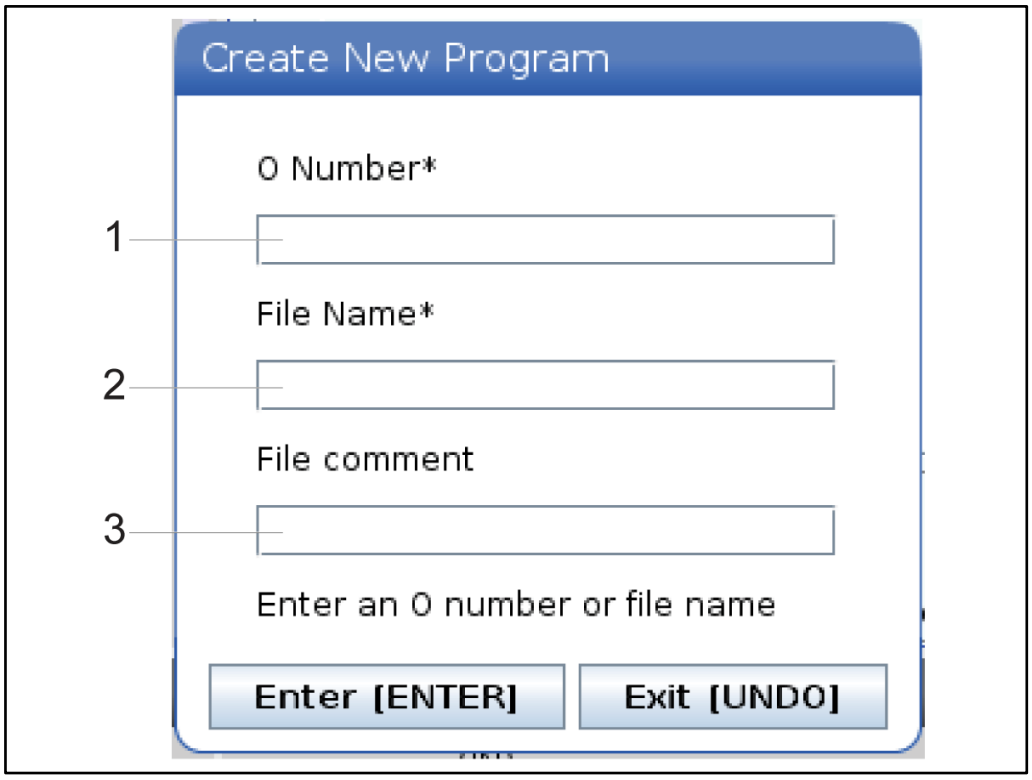
Highlight a program in the memory directory, then press SELECT PROGRAM to make the highlighted program active.

The active program has an asterisk (*) in the far-right column in the file display. It is the program that runs when you press CYCLE START in OPERATION: MEM mode. The program is also protected from deletion while it is active.

CREATE A NEW PROGRAM

Press INSERT to create a new file in the current directory. The CREATE NEW PROGRAM popup menu shows on the screen:

Create New Program Popup Menu Example: [1] Program O number field, [2] File Name field, [3] File comment field.



Enter the new program information in the fields. The Program O number field is required; the File Name and File comment are optional. Use the UP and DOWN cursors to move between the menu fields.

Press **UNDO** at any time to cancel program creation.

Program O number (required for files created in Memory): Enter a program number up to (5) digits long. The control adds the letter O automatically. If you enter a number shorter than (5) digits, the control adds leading zeros to the program number to make it (5) digits long; for example, if you enter 1, the control adds zeros to make it 00001.

NOTE: note: Do not use O09XXX numbers when you create new programs. Macro programs often use numbers in this block and overwriting them may cause machine functions to malfunction or stop working.

File Name (optional): Type a filename for the new program. This is the name the control uses when you copy the program to a storage device other than memory.

File comment (optional): Type a descriptive program title. This title goes into the program as a comment in the first line with the O number.
Press ENTER to save your new program. If you specified an O number that exists in the current directory, the control gives the message File with O Number nnnnn already exists. Do you want to replace it? Press ENTER to save the program and overwrite the existing progrpam, press CANCEL to return to the program name popup, or press UNDO to cancel.

EDIT A PROGRAM

Highlight a program, and then press **ALTER** to move the program into the program editor.

The program has the designation E in the far-right column of the file display list when it is in the editor, unless it is also the active program.

You can use this function to edit a program while the active program runs. You can edit the active program, but your changes do not take effect until you save the program and then select it again in the device manager menu.

COPY PROGRAMS

This function lets you copy programs to a device or a different directory.

To copy a single program, highlight it in the device manager program list and press **ENTER** to assign a check mark. To copy multiple programs, check-mark all the programs you want to copy.

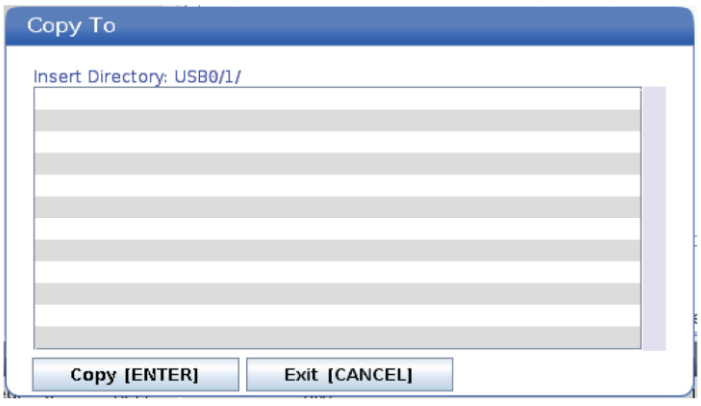
Press **F2** to start the copy operation.

The Select Device popup appears.

Select Device

Use the cursor arrow keys to select the destination directory. RIGHT cursor to enter the chosen directory.

Press **ENTER** to complete the copy operation, or press **CANCEL** to return to the device manager.



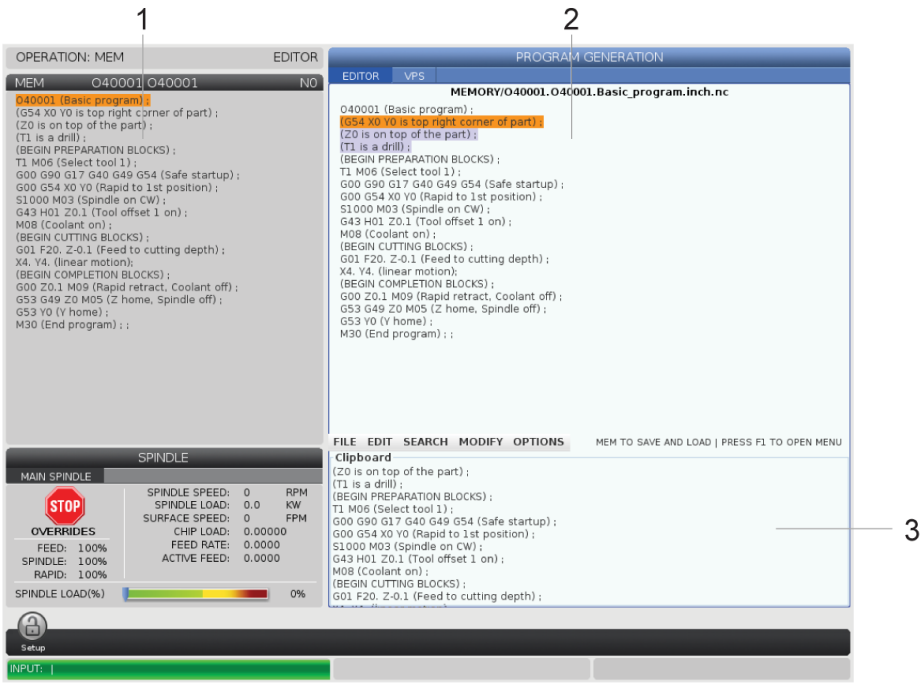
CREATE / SELECT PROGRAMS FOR EDITING

You use the Device Manager (LIST PROGRAM) to create and select programs for editing. Refer to the CREATE, EDIT, COPY A PROGRAM tab to create a new program.

PROGRAM EDIT MODES

The Haas control has (2) program edit modes: The program editor or manual data input (MDI). You use the program editor to make changes to numbered programs stored in an attached memory device (machine memory, USB, or net share). You use MDI mode to command the machine without a formal program.

The Haas control screen has (2) program edit panes: The Active Program / MDI pane, and the Program Generation pane. The Active Program / MDI pane is on the left side of the screen in all display modes. The Program Generation pane appears only in EDIT mode.



Example Edit Panes.

[1] Active Program / MDI Pane,

[2] Program Edit Pane,

[3] Clipboard Pane

BASIC PROGRAM EDITING

This section describes the basic program editing functions. These functions are available when you edit a program.

1) To write a program, or make changes to a program:
To edit a program in MDI, press MDI. This is EDIT: MDI mode. The program is displayed on the Active pane.
To edit a numbered program, select it in the Device Manager (LIST PROGRAM), then press EDIT. This is EDIT: EDIT mode. The program is displayed in the Program Generation pane.

2) To highlight code:
Use the cursor arrow keys or the jog handle to move the highlight cursor through the program.
You can interact with single pieces of code or text (cursor highlighting), blocks of code, or multiple blocks of code (block selection). Refer to the Block Selection section for more information.

3) To add code to the program:

- Highlight the code block you want the new code to follow.
- Type the new code.
- Press INSERT. Your new code appears after the block you highlighted.

4) To replace code:

- Highlight the code you want to replace.
- Type the code you want to replace the highlighted code with.
- Press ALTER. Your new code takes the place of the code you highlighted.

5) To remove characters or commands:

- Highlight the text you want to delete.
- Press DELETE. The text that you highlighted is removed from the program.

6) Press UNDO to reverse up to the last (40) changes.

NOTE: You cannot use UNDO to reverse changes that you made if you exit EDIT: EDIT mode.

NOTE: In EDIT: EDIT mode, the control does not save the program as you edit. Press MEMORY to save the program and load it into the Active Program pane.

BLOCK SELECTION

When you edit a program, you can select single or multiple blocks of code. You can then copy and paste, delete, or move those blocks in one step.

To select a block:

- Use the cursor arrow keys to move the highlight cursor to the first or last block in your selection.

NOTE: You can start a selection at the top block or the bottom block, and then move up or down as appropriate to complete your selection.

NOTE: You cannot include the program name block in your selection. The control gives the message GUARDED CODE.

- Press F2 to start your selection.
- Use the cursor arrow keys or the jog handle to expand the selection.
- Press F2 to complete the selection.

ACTIONS WITH A BLOCK SELECTION

After you make a text selection, you can copy and paste it, move it, or delete it.

NOTE: These instructions assume you have already made a block selection as described in the Block Selection section.

NOTE: These are actions available in MDI and the Program Editor. You cannot use UNDO to reverse these actions.

1) To copy and paste the selection:

- Move the cursor to the location where you want to put a copy of the text.
- Press ENTER.

The control puts a copy of the selection on the next line after the cursor location.

NOTE: The control does not copy the text to the clipboard when you use this function.

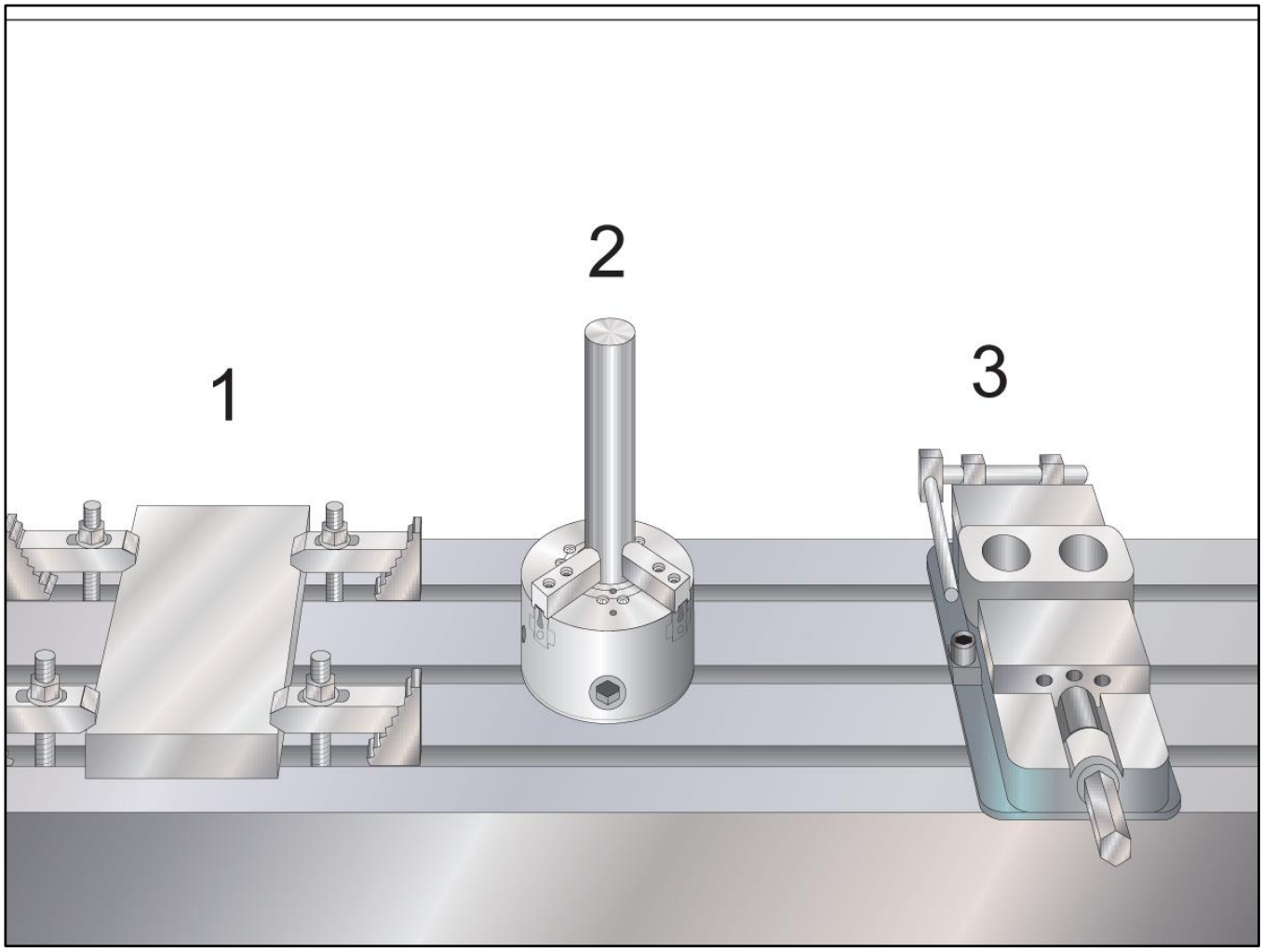
2) To move the selection:

- Move the cursor to the location where you want to move the text.
- Press ALTER.

The control removes the text from its current location and puts it in the line after the current line.

3) Press DELETE to delete the selection.

PART SETUP



Part Setup Examples: [1] Toe clamp, [2] Chuck, [3] Vise.

Correct work holding is very important for safety, and to get the machining results that you want. There are many work holding options for different applications. Contact your HFO or work holding dealer for guidance.

JOG MODE

Jog mode lets you jog the machine axes to a desired location. Before you can jog an axis, the machine must establish its home position. The control does this at machine power-up.

To enter jog mode:
Press **HANDLE JOG**.
Press the desired axis (+X, -X, +Y,-Y, +Z, -Z, +A/C or -A/C, +B, or -B).
There are different increment speeds that can be used while in jog mode; they are .0001, .001, .01 and .1. Each click of the jog handle moves the axis the distance defined by the current jog rate. You can also use an optional Remote Jog Handle (RJH) to jog the axes.

Press and hold the handle jog buttons or use the jog handle control to move the axis.

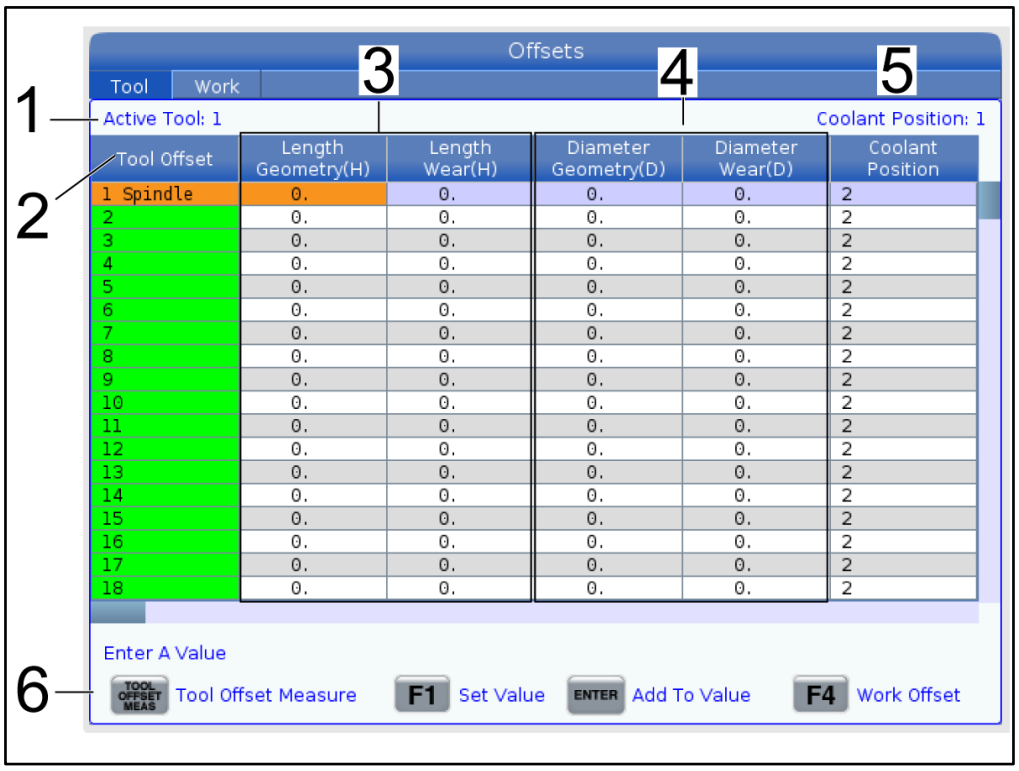
TOOL OFFSETS

To machine a part accurately, the mill needs to know where the part is located on the table and the distance from the tip of the tools to the top of the part (tool offset from home position).

The tool offset behavior has been modified on Haas machines in the following ways:

By default, tool offsets will now always be applied, unless a G49/H00 (Mill) or Txx00 offset (Lathe) is explicitly specified.
On mills, when a tool change occurs, the tool offset will automatically update to match the new tool. This behavior already existed on lathes.

Press the OFFSET button to view the tool offset values. The tool offsets can be entered manually or automatically with a probe. The list below will show how each offset setting works.



1) Active Tool: - This tells you which tool is in the spindle.

2) Tool Offset (T) - This is the list of tool offsets. There is a maximum of 200 tool offsets available.

3) Length Geometry (H), Length Wear (H) - These two columns are tied to the G43 (H) values in the program. If you command a
G43 H01;

from within a program for tool #1, the program will use the values from these columns.

NOTE: The Length Geometry can be set manually or automatically by the probe.

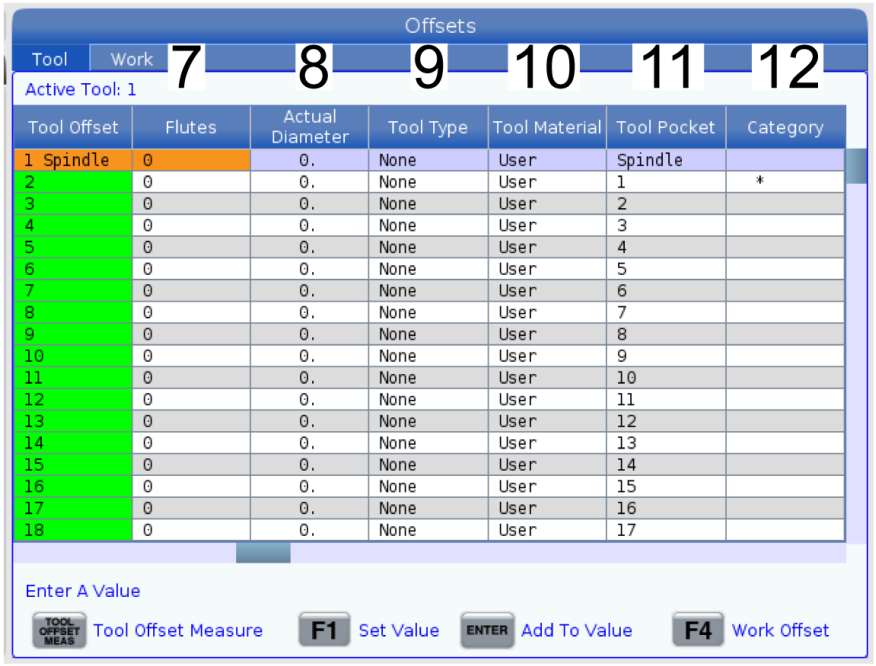
4) Diameter Geometry (D), Diameter Wear (D) - These two columns are used for cutter compensation. If you command a G41 D01; from within a program, the program will use the values from these columns.

NOTE: The Diameter Geometry can be set manually or automatically by the probe.

5) Coolant Position - Use this column for setting the coolant position for the tool in this row.

NOTE: This column will only show if you have the Programmable Coolant option. **6)** These functions buttons allow you to set the offset values.

7) Flutes - When this column is set to the correct value, the control can calculate the correct Chip Load value displayed at the Main Spindle screen. The VPS feeds and speeds library will also use these values for calculations.



Tool Offset	Flutes	Actual Diameter	Tool Type	Tool Material	Tool Pocket	Category
1 Spindle	0	0.	None	User	Spindle	
2	0	0.	None	User	1	*
3	0	0.	None	User	2	
4	0	0.	None	User	3	
5	0	0.	None	User	4	
6	0	0.	None	User	5	
7	0	0.	None	User	6	
8	0	0.	None	User	7	
9	0	0.	None	User	8	
10	0	0.	None	User	9	
11	0	0.	None	User	10	
12	0	0.	None	User	11	
13	0	0.	None	User	12	
14	0	0.	None	User	13	
15	0	0.	None	User	14	
16	0	0.	None	User	15	
17	0	0.	None	User	16	
18	0	0.	None	User	17	

NOTE: The values set on the Flute column will not affect the operation of the probe.

8) Actual Diameter - This column is used by the control to calculate the correct Surface Speed value displayed at the Main Spindle screen.

9) Tool Type - This column is used by the control to decide which probe cycle to use to probe this tool. Press F1 to view the options: None, Drill, Tap, Shell Mill, End Mill, Spot Drill, Ball Nose and Probe. When this field is set to Drill, Tap, Spot Drill, Ball Nose and Probe, the probe will probe along the tool centerline for the length. When this field is set to Shell Mill or End Mill the probe will probe at the tools edge.

10) Tool Material - This column is used for calculations by the VPS feeds and speeds library. Press F1 to view the options: User, Carbide, Steel. Press Enter to set the material, or press Cancel to exit.

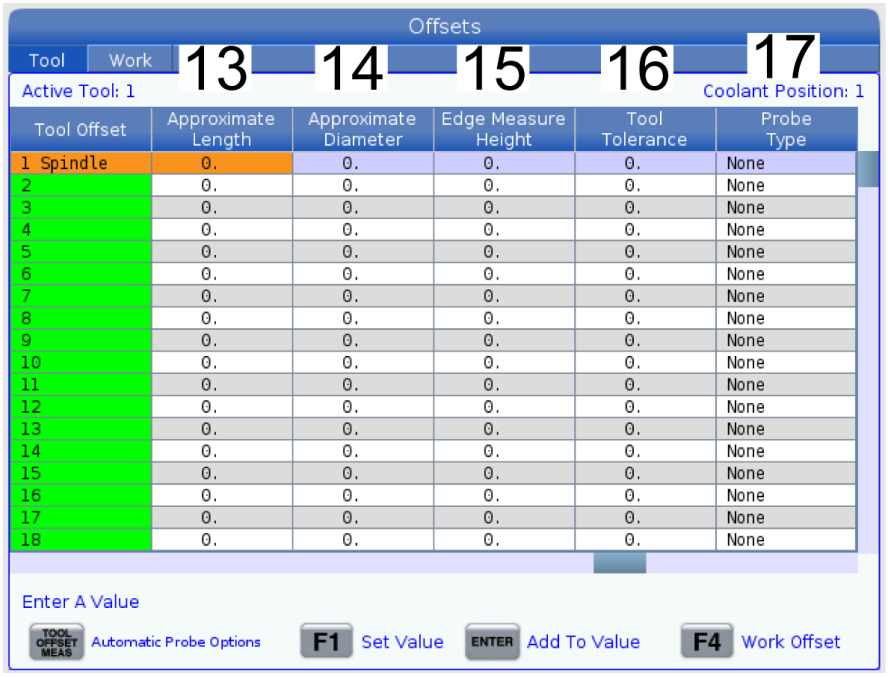
11) Tool Pocket - This column shows you what pocket the tool is currently in. This column is read only.

12) Tool Category - This column shows if the tool is set up as large, heavy or extra-large. To make a change, highlight the column and press ENTER. The Tool Table will be displayed. Follow the instructions on the screen to make tool table changes.

13) Approximate Length - This column is used by the probe. The value in this field tells the probe the distance from the tip of the tool to the spindle gauge line.

NOTE: If you probe the length of a drill or a tap, or some tool that is not a shell mill or an end mill you can leave this field blank.

14) Approximate Diameter - This column is used by the probe. The value in this field tells the probe the diameter of the tool.



15) Edge Measure Height - This column is used by the probe. The value in this field is the distance below the tip of the tool that the tool needs to move, when the tool diameter is probed. Use this setting when you have a tool with a large radius or when you are probing a diameter on a chamfer tool.

16) Tool Tolerance - This column is used by the probe. The value in this field is used for checking tool breakage and wear detection. Leave this field blank if you are setting the length and diameter on the tool.

17) Probe Type - This column is used by the probe. You can select the probe routine you want to perform on this tool.

The choices are: 0 - No tool probing to be performed., 1- Length probing (Rotating)., 2 - Length probing (Non-Rotating)., 3 - Length and Diameter probing (Rotating). Press TOOL OFFSET MEASURE to set automatic probe options.

SET A TOOL OFFSET

Tool Offsets

The tool offset behavior has been modified on Haas machines in the following ways:

By default tool offsets will now always be applied, unless a G49/H00 (Mill) or Txx00 offset (Lathe) is explicitly specified.
Press the [OFFSET] button to view the tool offset values. The tool offsets can be entered manually or automatically with a probe. The list below will shows how each offset setting works.

1. Active Tool: - This tells you which position is active turret.

2. Tool Offset (T) - This is the list of available tool offsets. There is a maximum of 99 tool offsets available.

3. Turret Location- This column is used to help the operator remember which tool is on the turret station. This is useful when you have a tool holder that has tools mounted on the front and on the back. You want to remember what offset each of the tools is using and where it is located.

4. X and Z Geometry - Each offset contains values for the distance from machine zero to the tip.

5. Radius Geometry - This offset is used to compensate for the radius on the tool tip when cutter compensation is used. Check the radius specification on the tool inserts and enter the value on this offset.

6. Tip Direction - Use this to set the direction of the tool tip when cutter compensation is used. Press [F1] to view the options.

7. These functions buttons allow you to set the offset values. Pressing [F1] enters the number in the selected column. Entering a value and pressing [ENTER] adds the amount entered to the number in the selected column.

8. X and Z Wear Geometry - The values entered here are intended for minute adjustments to offset that are required to compensate for normal wear during the course of a job.

9. Radius Wear - The values entered here are intended for minute adjustments to offset that are required to compensate for normal wear during the course of a job.

10. Tool Type - This column is used by the control to decide which probe cycle to use to probe this tool. Press [F1] to view the options.

11. Tool Material - This column is used for calculations by the VPS feeds and speeds library. Press [F1] to view the options.

12. Live Tool Radius - This offset is used to compensate for the radius on the live tool tip. Check the radius specification on the tool inserts and enter the value on this offset.
13. Live Tool Wear - The values entered here are intended for minute adjustments to offset that are required to compensate for normal wear during the course of a job.

14. Flutes - When this column is set to the correct value, the control can calculate the correct Chip Load value displayed at the Main Spindle screen. The VPS feeds and speeds library will also use these values for calculations. **Note: The values set on the Flute column will not affect the operation of the probe.**

15. Actual Diameter - This column is used by the control to calculate the correct Surface Speed value displayed at the Main Spindle screen.

16. Approximate X and Z - This column is used by the ATP or Tool Setting Probe. The value in this field tells the probe the approximate position of the tool being probed.

17. Approximate Radius - This column is used by the ATP probe. The value in the field tells the probe the approximate radius of the tool.

18. Edge Measure Height - This column is used by the ATP probe. The value in this field is the distance below the tip of the tool that the tool needs to move, when the edge is probed. Use this setting when you have a tool with a large radius or when you are probing a diameter on a chamfer tool.

19. Tool Tolerance - This column is used by the probe. The value in this field is used for checking tool breakage and wear detection. Leave this field blank if you are setting the length and diameter on the tool.

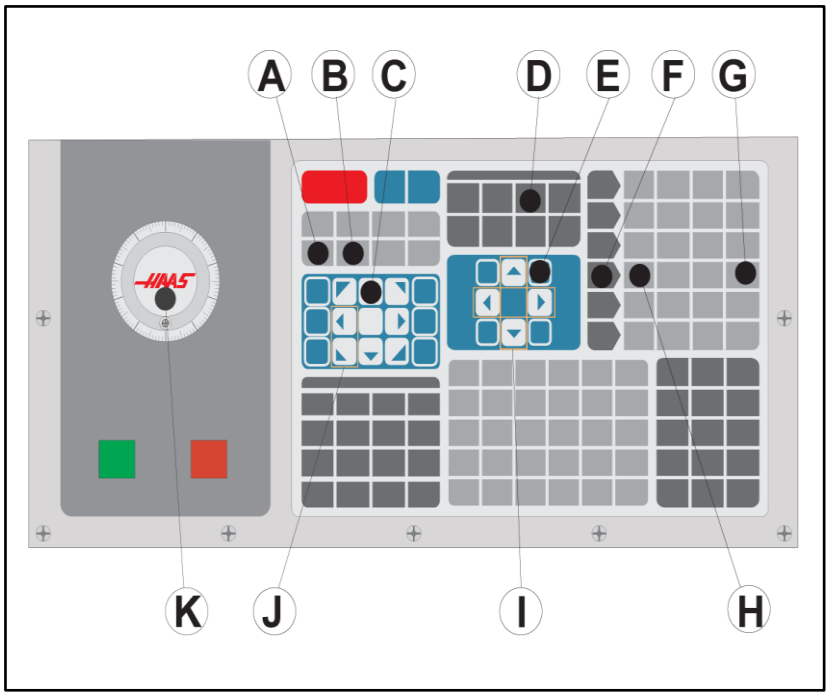
20. Probe Type - This column is used by the probe. You can select the probe routine you want to perform on this tool. Press [X DIAMETER MEASURE] to view the options.

The next step is to touch off the tools. This defines the distance from the tip of the tool to the top of the part. Another name for this is Tool Length Offset, which is designated as H in a line of machine code. The distance for each tool is entered into the TOOL OFFSET table.

NOTE: When touching off tools or working on the stationary table, make sure the tilt axis is at 0 degrees (A0° or B0°).

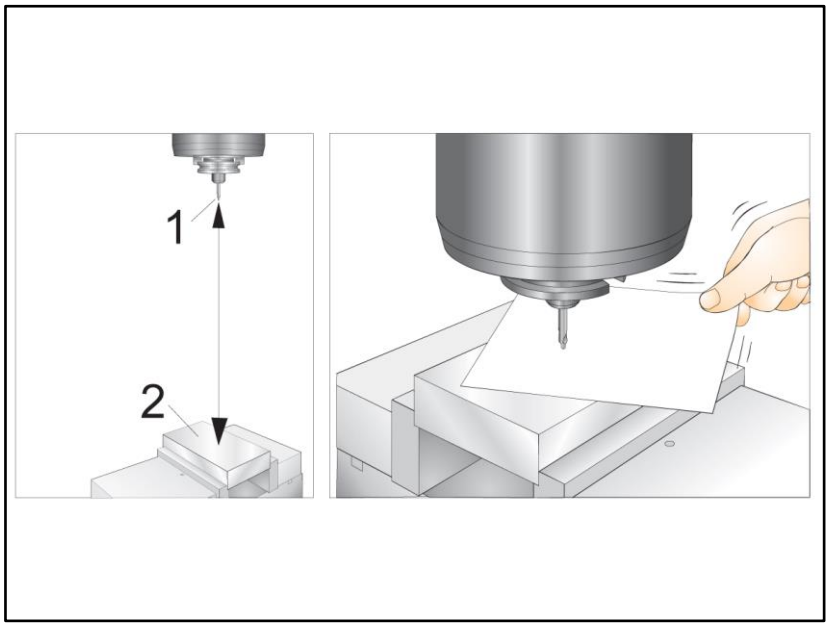
1

Setting Tool Offset. With the Z Axis at its home position, Tool Length Offset is measured from the tip of the tool [1] to the top of the part [2].
Load the tool in the spindle [1].
Press HANDLE JOG [F].
Press .1/100. [G] (The mill moves at a fast rate when the handle is turned).
Select between the X and Y axes [J], and with the use the jog handle [K] to move the tool near the center of the part.
Press +Z [C].
Jog the Z Axis approximately 1" above the part.
Press .0001/.1 [H] (The mill moves at a slow rate when the handle is turned).



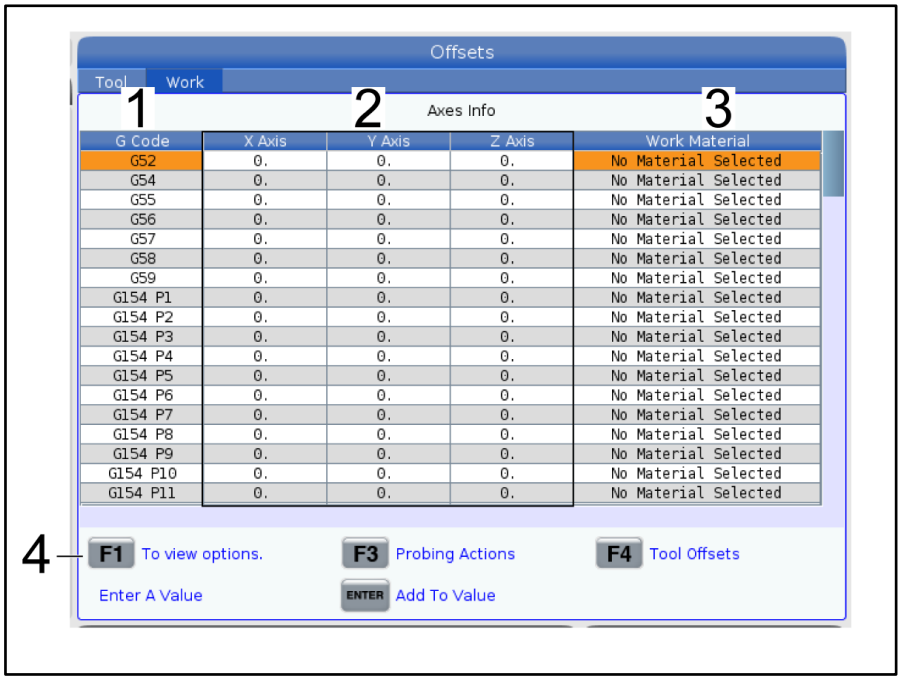
2

Place a sheet of paper between the tool and the work piece. Carefully move the tool down to the top of the part, as close as possible, and still be able to move the paper.
Press OFFSET [D] and select the TOOL tab.
Highlight the H (length) Geometry value for position #1.
Press TOOL OFFSET MEASURE [A]. A pop box may appear if the change is Greater Than Setting 142! Accept (Y/N).
Press Y to accept.
CAUTION: The next step causes the spindle to move rapidly in the Z Axis.
Press NEXT TOOL [B].
Repeat the offset process for each tool.



WORK OFFSETS

Press the OFFSET, then the F4 to view the work offset values. The work offsets can be entered manually or automatically with a probe. The list below will show how each work offset setting works.



1) G Code - This column displays all the available work offset G-codes. For more information on these work offsets, refer to the G52 Set Work Coordinate System (Group 00 or 12), G54 Work Offsets, G92 Set Work Coordinate Systems Shift Value (Group 00).

2) X, Y, Z, Axis - This column displays the work offset value for each axis. If rotary axis(es) are enabled the offsets for these will be displayed on this page.

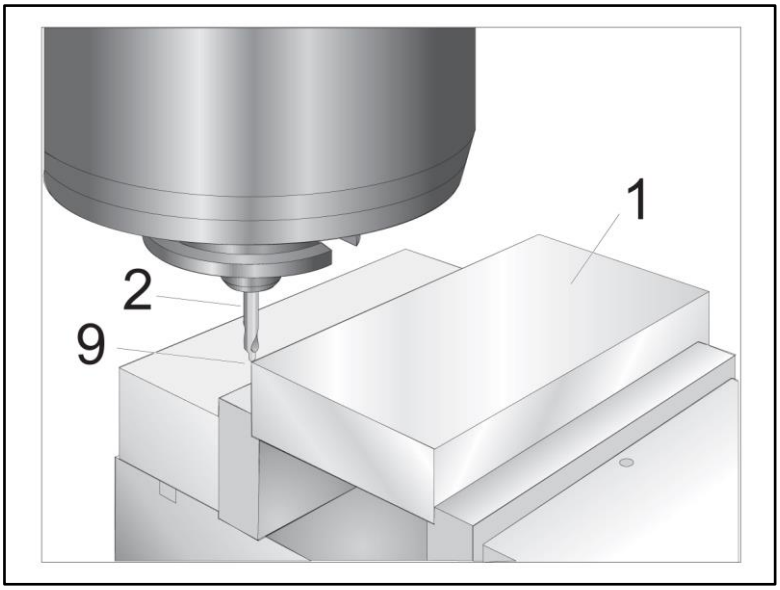
3) Work Material - This column is used by the VPS feeds and speeds library.

4) These functions buttons allow you to set the offset values. Type in the desired work offset value and press F1 to set the value. Press F3 to set a probing action. Press F4 to toggle from work to tool offset tab. Type in a value and press Enter to add to the current value.

SET A WORK OFFSET

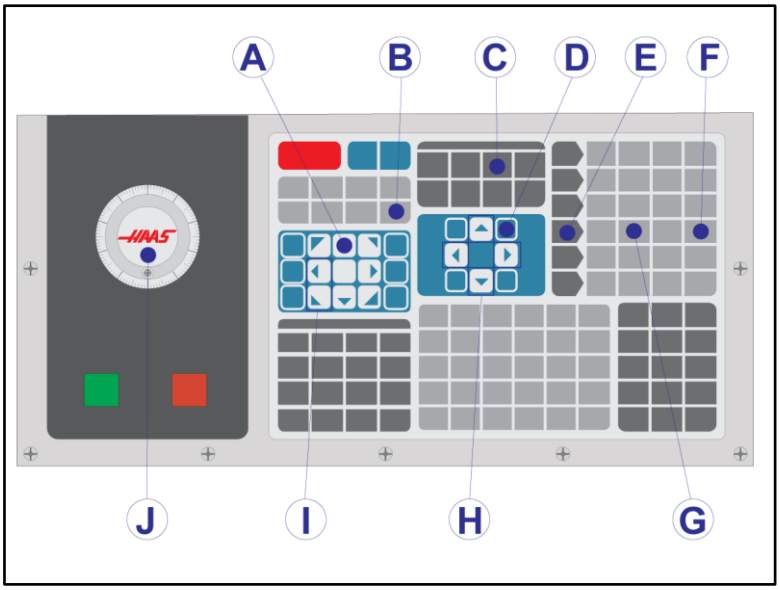
To machine a workpiece, the mill needs to know where the workpiece is located on the table. You can use an edge finder, an electronic probe, or many other tools and methods to establish part zero. To set the part zero offset with a mechanical pointer:

1



Place the material [1] in the vise and tighten.
Load a pointer tool [2] in the spindle.
Press HANDLE JOG [E].
Press .1/100. [F] (The mill moves at a fast speed when the handle is turned).
Press +Z [A].
Use the jog handle [J] to move the Z-Axis approximately 1" above the part.
Press .001/1. [G] (The mill moves at a slow speed when the handle is turned).
Jog the Z-Axis approximately. 0.2" above the part.
Select between the X and Y axes [I] and jog the tool to the upper left corner of the part (see illustration [9])

2



Navigate to the OFFSET>WORK [C] tab and press the DOWN cursor key [H] to activate the page. You can press F4 to toggle between Tool offsets and Work offsets.
Navigate to the G54 X Axis location.
CAUTION: In the next step, do not press PART ZERO SET a third time; this loads a value into the Z AXIS column. This causes a crash or a Z-Axis alarm when the program is run.
Press PART ZERO SET [B] to load the value into the X Axis column. The second press of PART ZERO SET [B] loads the value into the Y Axis column.

Notes

WIPS - OPERATORS MANUAL

Please refer to the WIPS operator's manual, operation section, for instructions on how to set tool offsets and work offsets using a probing cycle.

Clink on the link below to be redirected to that page.

https://www.haascnc.com/service/online-operator-s-manuals/wips---interactive-operator-s-manual-supplement/wips---operation.html

AUTOMATIC TOOL CHANGERS

There are (2) types of mill tool changers: the **Umbrella Style (UTC)**, and the **Side-Mount Tool Changer (SMTC)**. You command both tool changers in the same way, but you set them up differently.

Make sure the machine is zero returned. If it is not, press POWER UP.

Use TOOL RELEASE, ATC FWD, and ATC REV to manually command the tool changer. There are (2) tool release buttons; one on the spindle head cover and another on the keyboard.
Loading the Tool Changer

CAUTION: Do not exceed the maximum tool changer specifications. Extremely heavy tool weights should be distributed evenly. This means heavy tools should be located across from one another, not next to each other. Ensure there is adequate clearance between tools in the tool changer; this distance is 3.6" for a 20-pocket and 3” for a 24+1 pocket. Check your tool changer specifications for the correct minimal clearance between tools.

NOTE: Low air pressure or insufficient volume reduces the pressure applied to the tool release piston and will slow down tool change time or will not release the tool.

WARNING: Stay away from the tool changer during power up, power down, and during tool changer operations.

Always load tools into the tool changer from the spindle. Never load a tool directly into the tool changer carousel. Some mills have remote tool changer controls to let you inspect and replace tools at the carousel. This station is not for initial loading and tool assignment.

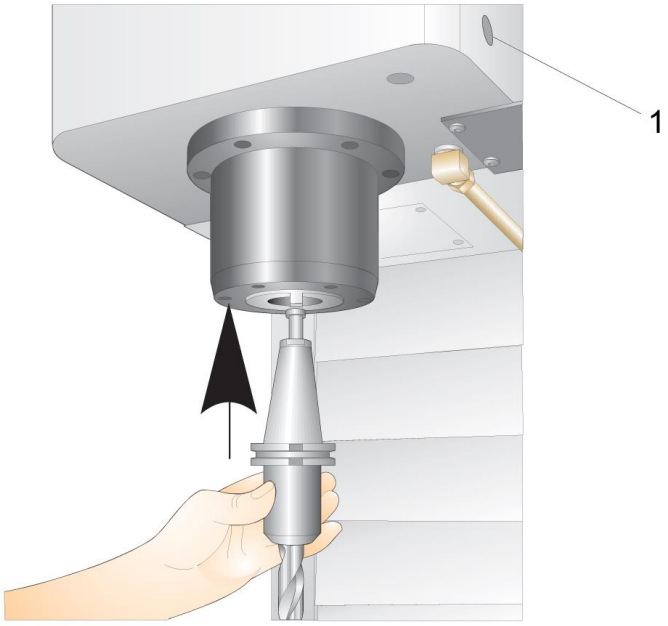
CAUTION: Tools that make a loud noise when released indicate a problem and should be checked before serious damage occurs to the tool changer or spindle.

TOOL LOADING FOR AN UMBRELLA TOOL CHANGER

1

This section tells you how to load tools into an empty tool changer for a new application. It assumes that the pocket tool table still contains information from the previous application.

Make sure your tool holders have the correct pull stud type for the mill.



2

Tools are loaded into the umbrella tool changer by first loading the tool into the spindle. To load a tool into the spindle, prepare the tool and then follow these steps:

Ensure the tools loaded have the correct pull stud type for the mill.

Press MDI/DNC for MDI mode.

Organize the tools to match to the CNC program.

Take tool in hand and insert the tool (pull stud first) into the spindle. Turn the tool so that the two cutouts in the tool holder line up with the tabs of the spindle. Push the tool upward while pressing the Tool Release button. When the tool is fitted into the spindle, release the **Tool Release** button.

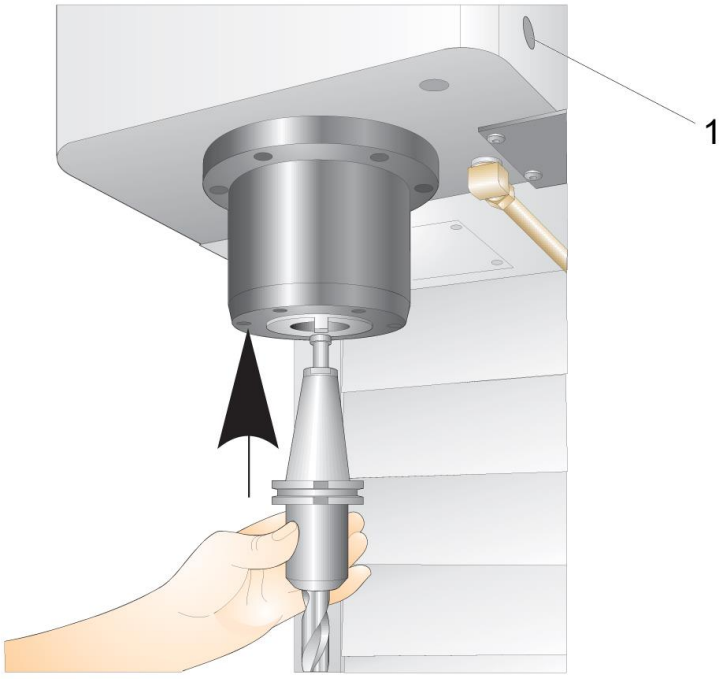
Press ATC FWD.

TOOL LOADING FOR A SIDE-MOUNT TOOL CHANGER

1
This section tells you how to load tools into an empty tool changer for a new application. It assumes that the pocket tool table still contains information from the previous application.

Make sure your tool holders have the correct pull stud type for the mill.

Press CURRENT COMMANDS then navigate to the TOOL TABLE tab and press the DOWN cursor. Refer to the Tool Table to set the correct tool information into the tool table.



2
Insert tool 1 (pull stud first) into the spindle.

Inserting a Tool into the Spindle: [1] Tool release button.

Turn the tool so that the two cutouts in the tool holder line up with the tabs of the spindle.
Push the tool upward and press the tool release button.
When the tool is inserted into the spindle, release the Tool Release Button.

HIGH-SPEED SIDE-MOUNT TOOL CHANGER

The high-speed side-mount tool changer has an additional tool assignment, which is “Heavy”. Tools that weigh more than 4 pounds are considered heavy. You must designate heavy tools with H (Note: All large tools are considered heavy). During operation, an “h” in the tool table denotes a heavy tool in a large pocket.

As a safety **PRECAUTION**, the tool changer will run at a maximum of 25% of the normal speed when it changes a heavy tool. The pocket up/down speed is not slowed down. The control restores the speed to the current rapid when the tool change is complete. Contact your HFO for assistance if you have problems with unusual or extreme tooling.

H - Heavy, but not necessarily large (large tools require empty pockets on either side).

h - Heavy small diameter tool in a pocket designated for a large tool (must have empty pocket on both sides). The lower case “h” and “l” are placed by the control; never enter a lower case “h” or “l” into the tool table.

l - Small diameter tool in a pocket reserved for a large tool in the spindle.

Large tools are assumed to be heavy.

Heavy tools are not assumed to be large.

On non-high speed tool changers, “H” and “h” have no effect.

Using ‘0’ for a Tool Designation

In the tool table, enter 0 (zero) for the tool number to label a tool pocket "always empty". The tool changer does not "see" this pocket, and it never tries to install or retrieve a tool from pockets with a ‘0’ designation.
You cannot use a zero to designate the tool in the spindle. The spindle must always have a tool number designation.

MOVING TOOLS IN THE CAROUSEL

If you need to move tools in the carousel, follow this procedure.
CAUTION: Plan the reorganization of the tools in the carousel ahead of time. To reduce the potential for tool changer crashes, keep tool movement to a minimum. If there are large or heavy tools currently in the tool changer, ensure that they are only moved between tool pockets designated as such.

UMBRELLA TOOL CHANGER RECOVERY

If the tool changer jams, the control will automatically come to an alarm state. To correct this:

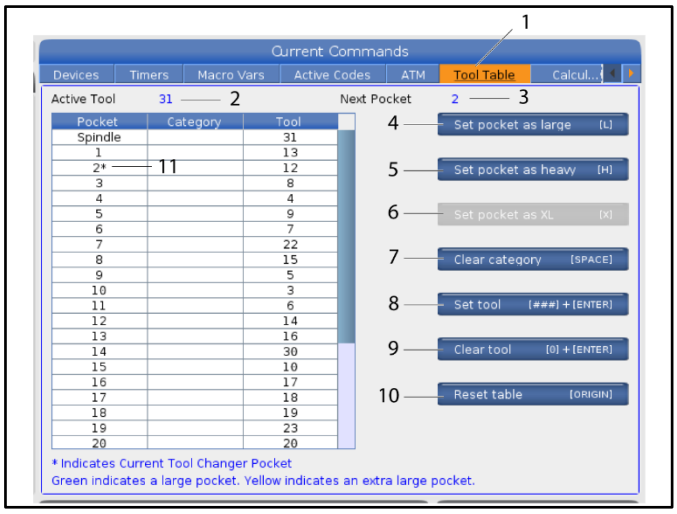
WARNING: Never put your hands near the tool changer unless there is an alarm displayed first.

Remove the cause of the jam.
Press RESET to clear the alarms.
Press RECOVER and follow the directions to reset the tool changer.

TOOL TABLE

This section tells you how to use the tool table to give the control information about your tools.

NOTE: If your machine has an umbrella-style tool changer, you will not use the tool pocket table.

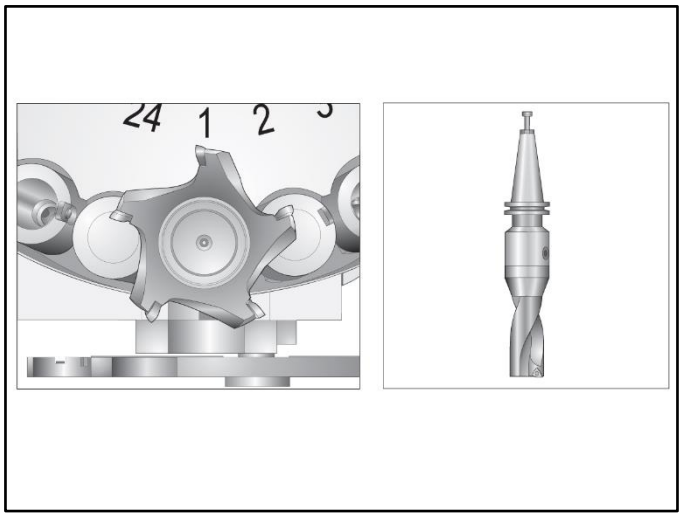


To access the Tool Pocket Table, press CURRENT COMMANDS and choose the Tool Table tab.

2) Active Tool - Tells you the tool number that is in the spindle.
3) Active Pocket - this shows you the next pocket number.
4) Set pocket as Large [L] - Use this flag when a large tool has a diameter greater than 3" for 40-taper machines and greater than 4" for 50-taper machines. Scroll to the pocket of interest and press L to set the flag.

CAUTION: You cannot place a large tool in the tool changer if one or both surrounding pockets already contain tools. Doing so causes the tool changer to crash. Large tools must have the surrounding pockets empty. However, large tools can share adjoining empty pockets.

A Large (left), and a Heavy (not Large) Tool (right)



5) Set pocket as heavy [H] - Use this flag when a Heavy, Small Diameter 40-taper tool (4 lb. or heavier) or a 50-taper tool (12 lb. or heavier) is loaded into the spindle. Scroll to the pocket of interest and press H to set the flag.

6) Set pocket as XL [X] - Use this flag when two adjacent pockets at each side of the tool are needed. Scroll to the pocket of interest and press X to set the flag.

NOTE: This option only appears if your machine is a 50-taper.

7) Clear category [Space] - Highlight the desired tool and press SPACE to clear the flag.

8) Set tool [###] + [Enter]- Highlight the desired pocket and type in the tool number + Enter to set the desired tool number.

NOTE: You cannot assign a tool number to more than one pocket. If you enter a tool number that is already defined in the tool pocket table, you see an Invalid tool error.

9) Clear tool [0] + [Enter]- Highlight the desired pocket and press 0 + Enter to clear out the tool number.
10) Reset table [Origin] - Press ORIGIN with the cursor in the center column to use the ORIGIN menu. This menu lets you:
Sequence All Pockets - Makes all the tool numbers sequential based on their pocket location, starting with 1.
Zero All Pockets - Removes all the tool numbers from all the pocket numbers.
Clear Category Flags - Removes the category designations from all the tools.
11) * Indicates current tool changer pocket.

SMTC RECOVERY (SIDE MOUNT TOOL CHANGER)

If a problem occurred during a tool change, a tool changer recovery needs to be performed. Enter the tool changer recovery mode by:

Press RECOVER and navigate to the TOOL CHANGER RECOVERY tab.

Press ENTER. If there is no Alarm, the control first attempts an automatic recovery. If there is an alarm press RESET to clear the alarms and repeat from step 1.

At the VMSTC TOOL RECOVERY screen, press A to begin automatic recovery or E to exit.

If the automatic recovery fails, press M to continue for a manual recovery.

In manual mode, follow the instructions and answer the questions to perform a proper tool changer recovery. The entire tool changer recovery process must be completed before exiting. Start the routine from the beginning if you exit the routine early.

OPERATION

MACHINE POWER-ON

This section tells you how to power-on a new machine for the first time.

Press POWER ON until you see the Haas logo on the screen. After a self-test and boot sequence, the display shows the startup screen.

The startup screen gives basic instructions to start the machine. Press CANCEL to dismiss the screen.

1 Turn EMERGENCY STOP to the right to reset it.

2 Press RESET to clear the startup alarms. If you cannot clear an alarm, the machine may need service. Contact your Haas Factory Outlet (HFO) for assistance.

3 If your machine is enclosed, close the doors.

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

4 Press POWER UP.

5 After the first POWER UP, the axes move toward their home positions. The axes then move slowly until the machine finds the home switch for each axis. This establishes the machine home position.

Press any of the following:

- **CANCEL** to dismiss the screen.
- **CYCLE START** to run current program.
- **HANDLE JOG** for manual operation.

SPINDLE WARM-UP

If your machine's spindle has been idle for more than (4) days, run the spindle warm-up program before you use the machine. This program brings the spindle up to speed slowly to distribute the lubrication and let the spindle reach a stable temperature.

Your machine includes a 20-minute warm-up program (O09220) in the program list. If you use the spindle at consistent high speeds, you should run this program every day.

SCREEN CAPTURE

The control can capture and save an image of the current screen to an attached USB device or the User Data memory.

Enter a filename if desired. If no filename is entered, the system will use the default filename (see note).

Press SHIFT.
Press F1.

NOTE: The control uses the default filename snapshot#.png. The # starts from 0 and increments each time you capture a screen. This counter resets at power off. Screen captures that you take after a power cycle overwrite previous screen captures that have the same filename on the User Data memory.

Result: The control saves the screen capture to your USB device or control memory. The message Snapshot saved to USB or Snapshot saved to User Data appears when the process finishes.

ERROR REPORT

The control can generate an error report that saves the state of the machine that is used for analysis. This is useful when helping the HFO troubleshoot an intermittent problem.

1. Press SHIFT.
2. Press F3.

NOTE: Be sure to always generate the error report with the alarm or the error is active.

Result: The control saves the error report to your USB device or control memory. The error report is a zip file that includes a screen capture, the active program, and other information used for diagnostics. Generate this error report when an error or an alarm occurs. E-mail the error report to your local Haas Factory Outlet.

BASIC PROGRAM SEARCH

You can use this function to quickly find code in a program.

NOTE: This is a quick-search function that finds the first match in the search direction that you specify. You can use the Editor for a more full-featured search. Refer to Chapter 6.5 for more information on the Editor search function.

NOTE: This is a quick-search function that finds the first match in the search direction that you specify. You can use the Editor for a more full-featured search. Refer to The Search Menu for more information on the Editor search function.

1. Type the text you want to find in the active program.
2. Press the UP or DOWN cursor arrow key.

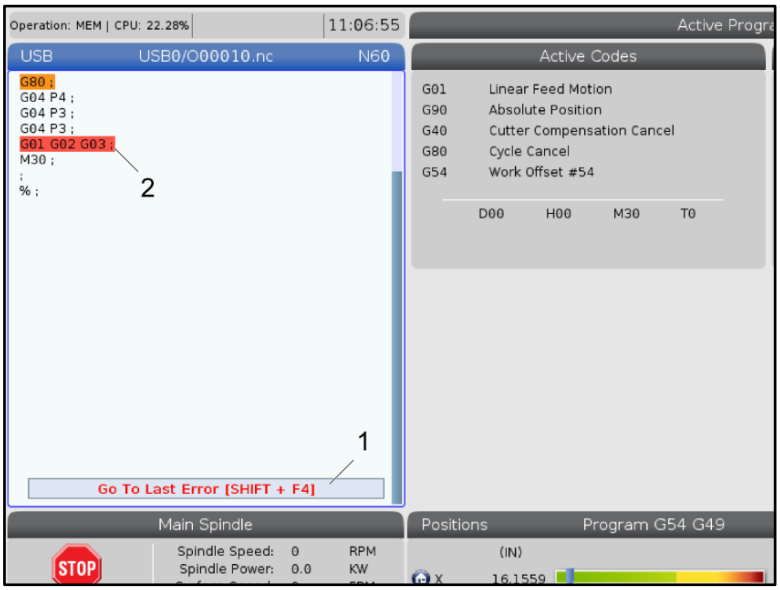
Result:
The UP-cursor arrow key searches from the cursor position to the start of the program. The DOWN cursor arrow key searches to the end of the program. The control highlights the first match.

NOTE: Putting your search term within parentheses (), will search only within comment lines.

LOCATE THE LAST PROGRAM ERROR

Starting in software version **100.19.000.1100** the control can find the last error in a program.

Press **SHIFT + F4** to display the last line of G-code that generated the error.



SAFE RUN MODE

The purpose of Safe Run is to reduce damage to the machine in the event of a crash. It does not prevent crashes, but it raises an alarm sooner and backs off from the crash location.

NOTE: The Safe Run feature is available starting in software version 100.19.000.1300.

NOTE: If the machine is equipped with a spindle head accelerometer, and when safe mode is enabled and active (i.e. during rapids), the accelerometer alarm behavior will be suppressed so the machine can do the normal back off behavior, then the **9941 Alarm** will generate. The G force value will still be recorded at the time of the event.

Safe Run Supported Machines

- VF-1 through VF-5
- VM-2/3
- UMC-500/750/1000
- All DM's
- All DT's
- All TM's
- ST-10 through ST-35

Common causes for crashes are:

- Incorrect tool offsets.
- Incorrect work offsets.
- Wrong tool in the spindle.

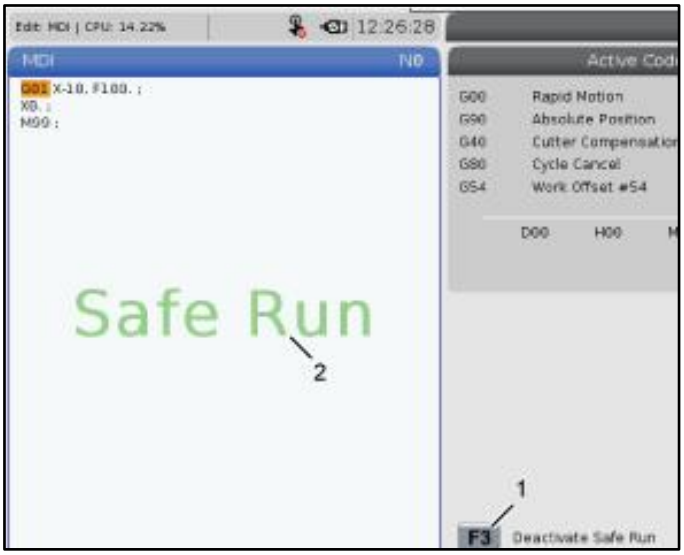
NOTE: The Safe Run feature will only detect a crash in handle jog and rapid (G00), it will not detect a crash in a feed move.

Safe Run does the following:
Slow down the speed of the motion.
Increases the position error sensitivity.
When a crash is detected, the control will immediately reverse the axis by a small amount. This will prevent the motor from continuing to drive into the object it has crashed into as well as relieve pressure from the crash itself. After Safe Run has detected a crash, you should be able to easily fit a piece off paper between the two surfaces that crashed.

NOTE: Safe Run is intended for running a program for the first time after writing or changing it. Is it not recommended to run a reliable program with Safe Run, as it increases cycle time significantly. The tool may break and the work piece may still be damaged in a crash.

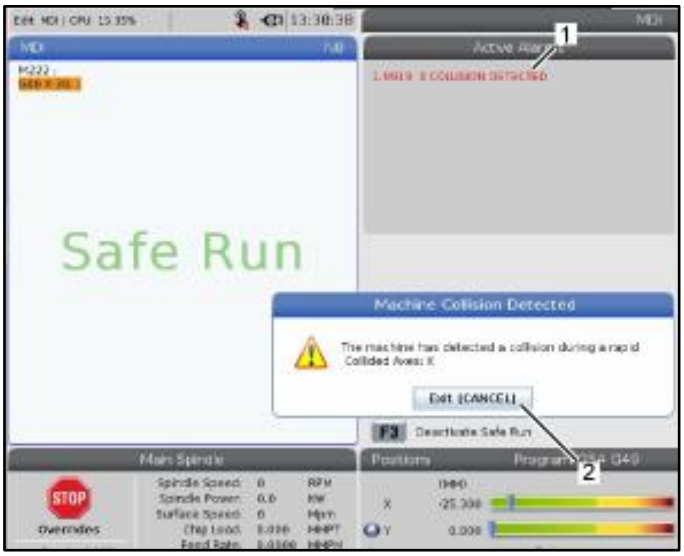
Safe Run is active during jogging as well. Safe Run can be used during job setup to protect against accidental crashes due to operator error.

If your machine supports Safe Run, you will see a new icon in MDI with the text F3 Activate Safe Run [1]. Press F3 to turn Safe Run on/off. Safe Run Active state is noted by a water mark [2] in the program panel. It is only active during rapid motions. Rapid motions include G00, Home G28, moving to tool changes, and the non-machining motions of canned cycles. Any machining motion such as a feed or tap will not have safe mode active.



Safe Run is not active during feeds due to the nature of crash detection. Cutting forces cannot be discerned from crashes.

When a crash is detected, all motion is brought to a stop, an alarm [1] is generated, and a popup [2] is generated letting the operator know that a crash was detected, and which axis it was detected on. This alarm can be cleared by reset.



In certain cases, the pressure against the part may not have been relieved by the Safe Run back-off. In the worst case, an additional crash may be generated after you have reset the alarm. If this happens, turn Safe Run off and jog the axis away from the crash location.

RUN-STOP-JOG-CONTINUE

This feature lets you stop a running program, jog away from the part, and then start the program again.

1 Press **FEED HOLD**.

Axis motion stops. The spindle continues to turn.

2 Press X, Y, Z, or an installed Rotary Axis (A for A Axis, B for B Axis, and C for C Axis), then press HANDLE JOG. The control stores the current X, Y, Z, and rotary axes positions.

3 The control gives the message Jog Away and displays the Jog Away icon. Use the jog handle or jog keys to move the tool away from the part. You can start or stop the spindle with FWD, REV, or STOP. You can command optional Through Spindle Coolant on and off with the AUX CLNT key (you must stop the spindle first). Command optional Through Tool Air Blast on and off with SHIFT + AUX CLNT keys. Command Coolant on and off with the COOLANT key. Command the Auto Air Gun / Minimum Quantity Lubrication options with SHIFT + COOLANT keys. You can also release the tool to change inserts.

CAUTION: When you start the program again, the control uses the previous offsets for the return position. Therefore, it is unsafe and not recommended to change tools and offsets when you interrupt a program.

4 Jog to a position as close as possible to the stored position, or to a position where there is an unobstructed rapid path back to the stored position.

5 Press MEMORY or MDI to return to run mode. The control gives the message Jog Return and displays the Jog Return icon. The control continues only if you return to the mode that was in effect when you stopped the program.

6 Press CYCLE START. The control rapids X, Y, and rotary axes at 5% to the position where you pressed FEED HOLD. It then returns the Z Axis. If you press FEED HOLD during this motion, axis motion pauses and the control gives the message Jog Return Hold. Press CYCLE START to resume the Jog Return motion. The control goes into a feed hold state again when the motion is finished.

CAUTION: The control does not follow the same path that you used to jog away.

7 Press CYCLE START again, and the program resumes operation.

CAUTION: If Setting 36 is ON, the control scans the program to make sure the machine is in the correct state (tools, offsets, G- and M-codes, etc.) to safely continue the program. If Setting 36 is OFF, the control does not scan the program. This can save time, but it could cause a crash in an unproven program.

GRAPHICS MODE

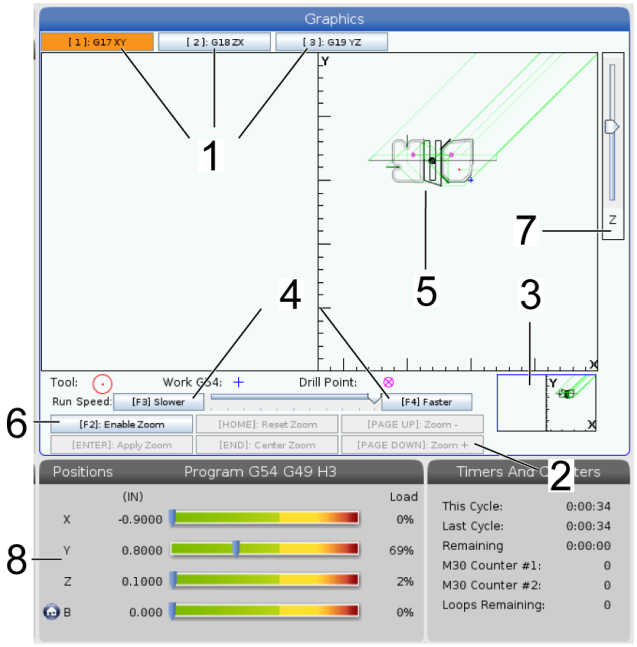
A safe way to troubleshoot a program is to press GRAPHICS to run it in graphics mode. No movement occurs on the machine, instead the movement is illustrated on the screen.

1) Axis Planes Press 1 to view the graphics in G17 plane, press 2 forG18 plane or press 3 to view in G19 plane.

2) Key Help Area The lower-left part of the graphics display pane is the function key help area. This area shows you the function keys that you can use, and a description of what they do.

3) Locator Window The lower-right part of the pane displays the simulated machine table area, and it shows where the simulated view is zoomed and focused.

4) Graphics Speed Press f3 or f4 to run the desired graphics speed.



5) Tool Path Window The large window in the center of the display gives a simulated view of the work area. It displays a cutting tool icon and simulated tool paths.

NOTE: Feed motion appears as a black line. Rapid moves appear as a green line. Drill cycle locations appear with an X.

NOTE: If Setting 253 is ON, the tool diameter is shown as a thin line. If it is OFF, the tool diameter specified in the Tool Offsets Diameter Geometry table is used.

6) Zoom Press F2 to display a rectangle (zoom window) that shows the area that the zoom operation will move to. Use PAGE DOWN to decrease the size of the zoom window (zoom in) and use PAGE UP to increase the size of the zoom window (zoom out). Use the cursor arrow keys to move the zoom window to the location you want to zoom, and press ENTER to complete the zoom. The control scales the tool path window to the zoom window. Run the program again to display the tool path. Press F2 and then HOME to expand the Tool Path window to cover the entire work area.

7) Z-Axis Part Zero Line The horizontal line on the Z-Axis bar at the top-right corner of the graphics screen gives the position of the current Z-axis work offset plus the length of the current tool. While a program simulation runs, the shaded portion of the bar indicates the depth of the simulated Z-Axis motion relative to the Z-Axis work zero position.

8) Position Pane The position pane displays axis locations just as it would during a live part run

MILL G-CODES

This page gives detailed descriptions of the G-codes that you use to program your Mill machine.

CAUTION: The sample programs in this manual have been tested for accuracy, but they are for illustrative purposes only. The programs do not define tools, offsets, or materials. They do not describe workholding or other fixturing. If you choose to run a sample program on your machine, do so in Graphics mode. Always follow safe machining practices when you run an unfamiliar program.

NOTE: The sample programs in this manual represent a very conservative programming style. The samples are intended to demonstrate safe and reliable programs, and they are not necessarily the fastest or most efficient way to operate a machine. The sample programs use G-codes that you might choose not to use in more efficient programs.

*** Click on **Description** in **BLUE** for more information ***

Code	Description	Group
G00	Rapid Motion Positioning	01
G01	Linear Interpolation Motion	01
G02	Circular Interpolation Motion CW	01
G03	Circular Interpolation Motion CCW	01
G04	Dwell	00
G09	Exact Stop	00
G10	Set Offsets	00
G12	Circular Pocket Milling CW	00
G13	Circular Pocket Milling CCW	00
G17	XY Plane Selection	02
G18	XZ Plane Selection	02
G19	YZ Plane Selection	02
G20	Select Inches	06
G21	Select Metric	06
G28	Return To Machine Zero Point	00
G29	Return From Reference Point	00
G31	Feed Until Skip	00
G35	Automatic Tool Diameter Measurement	00
G36	Automatic Work Offset Measurement	00
G37	Automatic Tool Offset Measurement	00
G40	Cutter Compensation Cancel	07
G41	2D Cutter Compensation Left	07
G42	2D Cutter Compensation Right	07
G43	Tool Length Compensation + (Add)	08
G44	Tool Length Compensation - (Subtract)	08
G47	Text Engraving	00
G49	G43/G44/G143 Cancel	08
G50	Cancel Scaling	11
G51	Scaling	11

G52	Set Work Coordinate System	00 or 12
G53	Non-Modal Machine Coordinate Selection	00
G54	Select Work Coordinate System #1	12
G55	Select Work Coordinate System #2	12
G56	Select Work Coordinate System #3	12
G57	Select Work Coordinate System #4	12
G58	Select Work Coordinate System #5	12
G59	Select Work Coordinate System #6	12
G60	Uni-Directional Positioning	00
G61	Exact Stop Mode	15
G64	G61 Cancel	15
G65	Macro Subprogram Call Option	00
G68	Rotation	16
G69	Cancel G68 Rotation	16
G70	Bolt Hole Circle	00
G71	Bolt Hole Arc	00
G72	Bolt Holes Along an Angle	00
G73	High-Speed Peck Drilling Canned Cycle	09
G74	Reverse Tap Canned Cycle	09
G76	Fine Boring Canned Cycle	09
G77	Back Bore Canned Cycle	09
G80	Canned Cycle Cancel	09
G81	Drill Canned Cycle	09
G82	Spot Drill Canned Cycle	09
G83	Normal Peck Drilling Canned Cycle	09
G84	Tapping Canned Cycle	09
G85	Boring Canned Cycle	09
G86	Bore and Stop Canned Cycle	09
G89	Bore In, Dwell, Bore Out Canned Cycle	09
G90	Absolute Position Command	03
G91	Incremental Position Command	03
G92	Set Work Coordinate Systems Shift Value	00
G93	Inverse Time Feed Mode	05
G94	Feed Per Minute Mode	05
G95	Feed per Revolution	05
G98	Canned Cycle Initial Point Return	10
G99	Canned Cycle R Plane Return	10
G100	Cancel Mirror Image	00
G101	Enable Mirror Image	00
G103	Limit Block Buffering	00
G107	Cylindrical Mapping	00
G110	#7 Coordinate System	12
G111	#8 Coordinate System	12
G112	#9 Coordinate System	12
G113	#10 Coordinate System	12

G114	#11 Coordinate System	12
G115	#12 Coordinate System	12
G116	#13 Coordinate System	12
G117	#14 Coordinate System	12
G118	#15 Coordinate System	12
G119	#16 Coordinate System	12
G120	#17 Coordinate System	12
G121	#18 Coordinate System	12
G122	#19 Coordinate System	12
G123	#20 Coordinate System	12
G124	#21 Coordinate System	12
G125	#22 Coordinate System	12
G126	#23 Coordinate System	12
G127	#24 Coordinate System	12
G128	#25 Coordinate System	12
G129	#26 Coordinate System	12
G136	Automatic Work Offset Center Measurement	00
G141	3D+ Cutter Compensation	07
G143	5-Axis Tool Length Compensation +	08
G150	General Purpose Pocket Milling	00
G154	Select Work Coordinates P1-P99	12
G156	Broaching Canned Cycle	09
G167	Modify Setting	00
G174	CCW Non-Vertical Rigid Tap	00
G184	CW Non-Vertical Rigid Tap	00
G187	Setting the Smoothness Level	00
G234	Tool Center Point Control (TCPC)	08
G253	G253 Orient Spindle Normal To Feature Coordinate System	00
G254	Dynamic Work Offset (DWO)	23
G255	Cancel Dynamic Work Offset (DWO)	23
G266	Visible Axes Linear Rapid % Motion	00
G268	Enable Feature Coordinate System	14
G269	Disable Feature Coordinate System	14

MILL M-CODES

This page gives detailed descriptions of the M-codes that you use to program your machine.

CAUTION: The sample programs in this manual have been tested for accuracy, but they are for illustrative purposes only. The programs do not define tools, offsets, or materials. They do not describe workholding or other fixturing. If you choose to run a sample program on your machine, do so in Graphics mode. Always follow safe machining practices when you run an unfamiliar program.

NOTE: The sample programs in this manual represent a very conservative programming style. The samples are intended to demonstrate safe and reliable programs, and they are not necessarily the fastest or most efficient way to operate a machine. The sample programs use G-codes that you might choose not to use in more efficient programs.

M-codes are miscellaneous machine commands that do not command axis motion. The format for an M-code is the letter M followed by two to three digits; for example, M03. Only one M-code is allowed per line of code. All M-codes take effect at the end of the block.

*** Click on **Description** in **BLUE** for more information ***

M - Code	Description
M00	Stop Program
M01	Optional Program Stop
M02	Program End
M03	Spindle Foward Command
M04	Spindle Reverse Command
M05	Spindle Stop Command
M06	Tool Change
M07	Shower Coolant On
M08 / M09	Coolant On / Off
M10 / M11	Engage / Release 4th Axis Brake
M12 / M13	Engage / Release 5th Axis Brake
M16	Tool Change
M19	Orient Spindle
M21-M25	Optional User M Function with M-Fin
M29	Set Output Relay with M-Fin
M30	Program End and Reset
M31	Chip Conveyor Forward
M33	Chip Conveyor Stop
M34	Coolant Increment
M35	Coolant Decrement
M36	Pallet Part Ready
M39	Rotate Tool Turret
M41 / M42	Low / High Gear Override
M46	Qn Pmm Jump to Line

M48	Validate That The Current Program is Appropriate for Loaded Pallet
M50	Pallet Change Sequence
M51-M55	Set Optional User M-codes
M59	Set Output Relay
M61-M65	Clear Optional User M-codes
M69	Clear Output Relay
M70/M71	Workholding Clamp / Unclamp
M73 / M74	Tool Air Blast (TAB) On / Off
M75	Set G35 or G136 Reference Point
M78	Alarm if Skip Signal Found
M79	Alarm if Skip Signal Not Found
M80 / M81	Auto Door Open / Close
M82	Tool Unclamp
M83 / M84	Auto Air Gun On / Off
M86	Tool Clamp
M88 / M89	Through-Spindle Coolant On / Off
M90 / M91	Fixture Clamp Input On / Off
M95	Sleep Mode
M96	Jump If No Input
M97	Local Sub-Program Call
M98	Sub-Program Call
M99	Sub-Program Return or Loop
M104 / M105	Probe Arm Extend/ Retract
M109	Interactive User Input
M116 / M117	Vise Air Chips Blast On/Off
M130 / M131	Display Media / Cancel Display Media
M138 / M139	Spindle Speed Variation On/Off
M158 / M159	Mist Condenser On/Off
M160	Cancel Active PulseJet
M161	PulseJet Continuous Mode
M162	PulseJet Single Event Mode
M163	PulseJet Modal Mode
M180 / M181	Auto Window Open / Close
M199	Pallet / Part Load or Program End
M300	M300 - APL/Robot Custom Sequence

MILL SETTINGS

This page gives detailed descriptions of the settings that control the way that your machine works.

Inside the **SETTINGS** tab, the settings are organized into groups. Use the **[UP]** and **[DOWN]** cursor arrow keys to highlight a setting group. Press the **[RIGHT]** cursor arrow key to see the settings in a group,. Press the **[LEFT]** cursor arrow key to return to the setting group list.

To quickly access a single setting, make sure the **SETTINGS** tab is active, type the setting number, and then press **[F1]** or, if a setting is highlighted, press the **[DOWN]** cursor.
Some settings have numerical values that fit in each range. To change the value of these settings, type the new value and press **[ENTER].** Other settings have specific available values that you select from a list. For these settings, use the **[RIGHT]** cursor to display the choices. Press **[UP]** and **[DOWN]** to scroll through the options.
Press **[ENTER]** to select the option.

List of Settings

*** Click on **Description** in **BLUE** for more information ***

Setting Number	Description
1	Auto Power Off Timer
2	Power Off at M30
4	Graphics Rapid Path
5	Graphics Drill Point
6	Front Panel Lock
8	Prog Memory Lock
9	Dimensioning
10	Limit Rapid at 50%
15	H and T Code Agreement
17	Opt Stop Lock Out
18	Block Delete Lock Out
19	Feedrate Override Lock
20	Spindle Override Lock
21	Rapid Override Lock
22	Can Cycle Delta Z
23	9xxx Progs Edit Lock
27	G76 / G77 Shift Dir.
28	Can Cycle Act w/o X/Y
29	G91 Non-modal
31	Reset Program Pointer
32	Coolant Override
33	Coordinate System
34	4th Axis Diameter
35	G60 Offset
36	Program Restart

39	Beep @ M00, M01, M02, M30
40	Tool Offset Measure
42	M00 After Tool Change
43	Cutter Comp Type
44	Min F Radius CC%
45	Mirror Image X Axis
46	Mirror Image Y Axis
47	Mirror Image Z Axis
48	Mirror Image A Axis
52	G83 Retract Above R
53	Jog w/o Zero Return
56	M30 Restore Default G
57	Exact Stop Canned X-Y
58	Cutter Compensation
59	Probe Offset X+
60	Probe Offset X-
61	Probe Offset Y+
62	Probe Offset Y-
63	Tool Probe Width
64	Tool Offset Measure Uses Work
71	Default G51 Scaling
72	Default G68 Rotation
73	G68 Incremental Angle
74	9xxx Progs Trace
75	9xxx Progs Single BLK
76	Tool Release Lockout
77	Scale Integer F
79	5th-Axis Diameter
80	Mirror Image B Axis
81	Tool At Power Up
82	Language
83	M30/Resets Overrides
84	Tool Overload Action
85	Maximum Corner Rounding
86	M39 Lockout
87	Tool Change Resets Override
88	Reset Rests Override
90	Max Tools To Display
101	Feed Override -> Rapid
103	Cyc Start/Fh Same Key
104	Jog Handle to SNGL BLK
108	Quick Rotary G28
109	Warm-Up Time in Min.
110	Warmup X Distance
111	Warmup Y Distance

112	Warmup Z Distance
113	Tool Change Method
114	Conveyor Cycle Time (minutes)
115	Conveyor On-Time (minutes)
117	G143 Global Offset
118	M99 Bumps M30 Cntrs
119	Offset Lock
120	Macro Var Lock
130	Tap Retract Speed
131	Auto Door
133	Repeat Rigid Tap
142	Offset Chng Tolerance
143	Machine Data Collection Port
144	Feed Override -> Spindle
155	Load Pocket Tables
156	Save Offsets with Program
158	X Screw Thermal Comp%
159	Y Screw Thermal Comp%
160	Z Screw Thermal Comp%
162	Default To Float
163	Disable .1 Jog Rate
164	Rotary Increment
165	Ssv Variation (RPM)
166	Ssv Cycle
188	G51 X Scale
189	G51 Y Scale
190	G51 Z Scale
191	Default Smoothness
196	Conveyor Shutoff
197	Coolant Shutoff
199	Backlight Timer
216	Servo and Hydraulic Shutoff
238	High Intensity Light Timer (minutes)
239	Worklight Off Timer (minutes)
240	Tool Life Warning
242	Air Water Purge Interval
243	Air Water Purge On-Time
245	Hazardous Vibration Sensitivity
247	Simultaneous XYZ Motion in Tool Change
249	Enable Haas Starup Screen
250	Mirror Image C Axis
251	Subprogram Search Location
252	Custom Subprogram Search Location
253	Default Graphics Tool Width
254	5-Axis Rotary Center Distance

255	MRZP X Offset
256	MRZP Y Offset
257	MRZP Z Offset
261	DPRNT Store Location
262	DPRNT Destination File Path
263	DPRNT Port
264	Autofeed Step Up
265	Autofeed Step Down
266	Autofeed Minimum Override
267	Exit Jog Mode After Idle TIme
268	Second Home Postion X
269	Second Home Position Y
270	Second Home Position Z
271	Second Home Position A
272	Second Home Position B
273	Second Home Position C
276	Workholding Input Monitor
277	Lubrication Cycle Interval
291	Main Spindle Speed Limit
292	Door Open Spindle Speed Limit
293	Tool Change Mid Position X
294	Tool Change Mid Position Y
295	Tool Change Mid Position Z
296	Tool Change Mid Position A
297	Tool Change Mid Position B
298	Tool Change Mid Position C
300	MRZP X Offset Master
301	MRZP Y Offset Master
302	MRZP Z Offset Master
303	MRZP X Offset Slave
304	MRZP Y Offset Slave
305	MRZP Z Offset Slave
306	Minimum Chip Clear Time
310	Min User Travel Limit A
311	Min User Travel Limit B
312	Min User Travel Limit C
313	Max User Travel Limit X
314	Max User Travel Limit Y
315	Max User Travel Limit Z
316	Max User Travel Limit A
317	Max User Travel Limit B
318	Max User Travel Limit C
323	Disable Notch Filter
325	Manual Mode Enabled
330	MultiBoot Selection Time out

335	Linear Rapid Mode
356	Beeper Volume
357	Warm Up Cycle Start Idle Time
369	PulseJet Injection Cycle Time
370	PulseJet Single Squirt Count
372	Parts Loader Type
375	APL Gripper Type
376	Light Curtain Enable
377	Negative Work Offsets
378	Safe Zone Calibrated Geometry Reference Point X
379	Safe Zone Calibrated Geometry Reference Point Y
380	Safe Zone Calibrated Geometry Reference Point Z
381	Enable Touchscreen
382	Disable Pallet Changer
383	Table Row Size
389	Vise Unclamped Safety Check
396	Enable / Disable Virtual Keyboard
397	Press and Hold Delay
398	Header Height
399	Header Tab
400	Pallet Ready Beep Type
403	Change Popup Button Size
408	Exclude Tool From Safe Zone
409	Default Coolant Pressure
416	Media Destination
420	ATC Button Behavior
421	General Orient Angle
422	Lock Graphics Plane
423	Help Text Icon Size
424	Mist Extractor Condenser Time Out
430	Enable - Broken Tool Detection
431	Region of Interest - Broken Tool Detection
433	Behavior - Broken Tool Detection

MILL – OTHER EQUIPMENT

Interactive Manuals

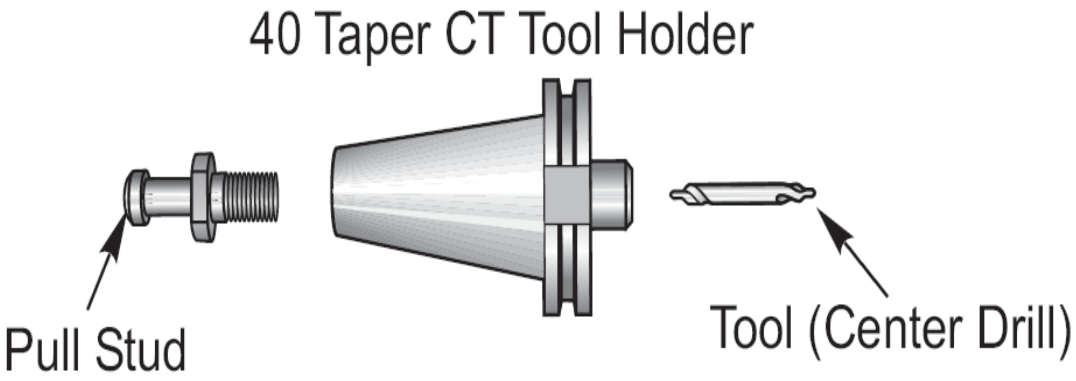
Product	Mill Operator's Manual Supplements	Service Manual
Desktop Mill	Desktop Mill - Interactive Operator's Manual Supplement	N/A
Desktop Lathe	Desktop Lathe - Interactive Operator's Manual Supplement	N/A
Compact Mill	Compact Mill - Interactive Operator's Manual Supplement	N/A
Gantry - Series	Gantry-Series - Interactive Operator's Manual Supplement	N/A
Mill APL	Mill - APL - Interactive Operator's Manual Supplement	Haas Automatic Parts Loader - Interactive Service Manual
Compact APL	Compact APL - Interactive Operator's Manual Supplement	
Pallet Pool	Pallet Pool - Interactive Operator's Manual Supplement	Pallet Pool - Interactive Service Manual
VF Pallet Pool	VF-Pallet Pool - Interactive Operator's Manual	
Rotary	Rotary - Interactive Operator's Manual Supplement	Rotary - Interactive Service Manual
UMC-Series	UMC-Series - Interactive Operator's Manual Supplement	UMC-Series - Interactive Service Manual
VR-Series	VR-Series - Interactive Operator's Manual Supplement	N/A
DC-Series	DC- Series Operator's / Service Manual	DC- Series Operator's / Service Manual

Other Equipment	Operator's Manual	Service Manual
Autodoor	N/A	Autodoor - Interactive Service Manual
Haas Air Compressor	Haas Air Compressor - Operators/Service Manual	Haas Air Compressor - Operators/Service Manual
Haas Cobot Package	Haas Cobot Package - Operator's/Service Manual	Haas Cobot Package - Operator's/Service Manual
Haas Laser Engraver	Haas Laser Engraver - Operator's/Service Manual	Haas Laser Engraver - Operator's/Service Manual
Haas Robot Package	Haas Robot Package - Interactive Operator's Manual	Haas Robot Package - Interactive Service Manual
Haas Robot Pallet Loader	Haas Robot Pallet Loader - Operator's/Service Manual	Haas Robot Pallet Loader - Operator's/Service Manual
HSF-325	HSF-325 - Interactive Operator's/Service Manual	HSF-325 - Interactive Operator's/Service Manual
HSF-450	HSF-450 - Interactive Operator's/Service Manual	HSF-450 - Interactive Operator's/Service Manual
HTS400	HTS400 - Interactive Operator's/Service Manual	HTS400 - Interactive Operator's/Service Manual
Haas Tooling and Workholding	N/A	Haas Tooling and Workholding - Interactive Service Manual
Lubrication Systems	N/A	Lubrication Systems - Interactive Service Manual
Chip Removal and Coolant	N/A	Chip Removal and Coolant - Interactive Service Manual
WIPS and WIPS-L	WIPS - Interactive Operator's Manual Supplement	N/A
CAN Bus Systems	N/A	CAN Bus Systems - Interactive Service Manual
Haas Spindle Chiller	Haas Spindle Chiller - Operator's/Service Manual	Haas Spindle Chiller - Operator's/Service Manual

TOOLING AND WORKHOLDING

CAT 40 TOOL HOLDER

All of the Haas TM series machines are equipped with a “CAT-40” spindle. This is a standardized quick change tooling design and is the most popular design amongst the ones offered in the machine tool market (vs. HSK, BT, Capto, etc.....)



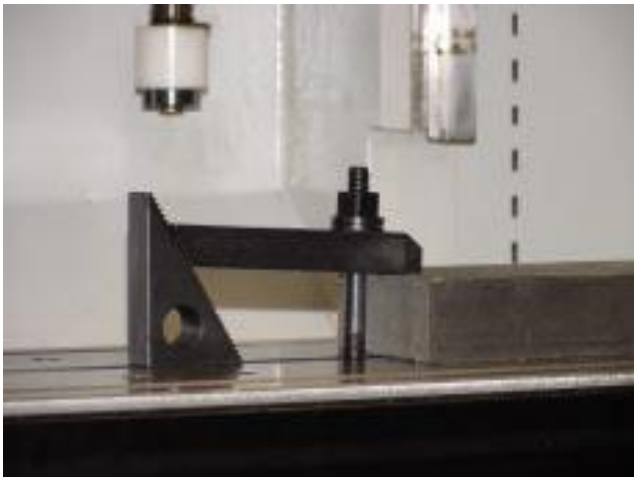
There is a wide variety of tooling available for CT-40, as it is the most popular tooling option for machining centers.

MILL WORK-HOLDING OPTIONS

As far as holding on to parts, a VMC has a great deal of not only spindle power but servo (axis thrust) and we can actually generate more pressure than that of a manual machine. Care needs to be taken to make sure that we have a SOLID GRIP on our parts!

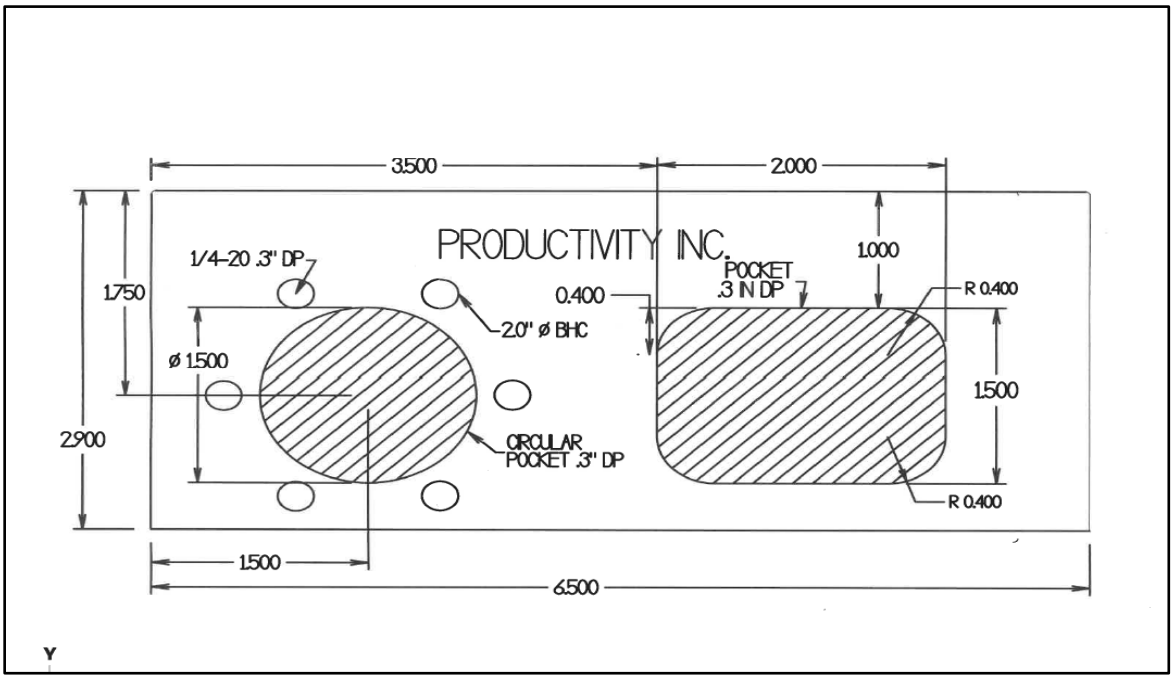
On a machining center there are almost an endless number of different ways of holding onto our work pieces. Most of the map can be covered by 3 different ways:

Vise Toe Clamp



3-Jaw Chuck

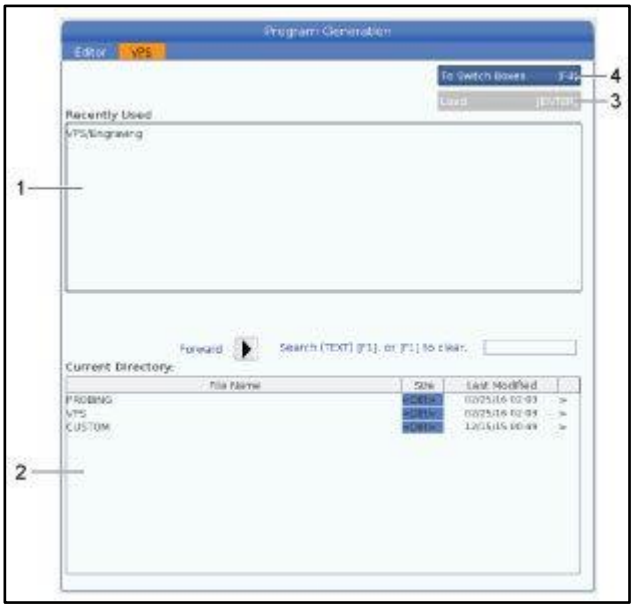
Example Part
The rough stock is 3.3in x 7in x 3in aluminum
The part will be held by a Kurt Vise



VISUAL PROGRAMMING SYSTEM (VPS)

https://www.haascnc.com/service/online-operator-s-manuals/mill-operator-s-manual/mill---options-programming.html

VPS lets you quickly build programs from program templates. To access VPS, press EDIT and then select the VPS tab.



VPS Starting Screen.
[1] Recently Used Templates,
[2] Template Directory Window,
[3] ENTER to Load a Template,
[4] F4 to Switch Between Recently Used and Template Directory.

VPS Example

When you use VPS, you choose a template for the function you want to program, and then enter variables to create a program. The default templates include probing and part features. You can also create custom templates. Contact the Applications department at your HFO for help with custom templates.

In this example, we use a VPS template to program the engraving cycle from the G47 program example in this manual. The G47 description starts on page 5. VPS templates all work in the same way: You first fill in values for the template variables, and you then output a program.
1. Press EDIT, and then select the VPS tab.

2. Use the cursor arrow keys to highlight the VPS menu option. Press the RIGHT cursor arrow key to select the option.

3. Highlight and select the Engraving option from the next menu.

VPS Engraving Program Generation Window Example.
[1] Variable Illustration,
[2] Variables Table,
[3] Variable Description Text,
[4] Template Illustration,
[5] Generate G-code F4,
[6] Run in MDI CYCLE START,
[7] Clear ORIGIN,
[8] Default value was changed indicator.

In the Program Generation window, use the UP and DOWN cursor arrow keys to highlight the variable rows.

5. Type a value for the highlighted variable and press ENTER. The control will display an asterisk (*) next to variable if the default value is changed. To set the variable back to default press the ORIGIN button. Press the DOWN cursor arrow key to move to the next variable.

To generate the example engraving cycle, we use these variable values. Note that all the position values are given in work coordinates.

Variable	Description	Value
WORK_OFFSETS	Work Offset Number	54
T	Tool Number	1
S	Spindle Speed	1000
F	Feedrate	15
M8	Coolant (1 - YES / 0 - NO)	1
X	Starting X Position	2
Y	Starting Y Position	2
R	R-Plane Height	0.05
Z	Z Depth	-0.005
P	Text or Serial Number Switch (0 - Text, 1 - Serial Number)	0
J	Text Height	0.5
I	Text Angle (Degrees from Horizontal)	45
TEXT	Text to Engrave	TEXT TO ENGRAVE

6. With all the variables entered, you can press **[CYCLE START]** to immediately run the program in MDI, or F4 to output the code to either the clipboard or MDI without running the program.

This VPS template creates a program with the specified variables to engrave the text:

```
%
O11111 ;
(Engraving) ;
( TOOL 1 ) ;
( SPINDLE 1000 RPM / FEED 15. ) ;
( DEPTH -0.005 ) ;
T1 M06 ;
G00 G90 G54 X2. Y2. S1000 M03 ;
G43 Z0.05 H1 ;
M08 ;
G00 G90 G54 X2. Y2. ;
( TEXT ENGRAVING : TEXT TO ENGRAVE ) ;
G47 E7.5000 F15. I45. J.5 P0 R0.05 Z-0.005 (TEXT
TO ENGRAVE) ;
G0 Z0.05 M09 ;
M05 ;
G91 G28 Z0. ;
G91 G28 Y0. ;
M01 ( END ENGRAVING ) ;
%
```