Haas Mill Operator

Revised 1/2020 JP

Productivity Inc®

This Manual is the Property of Productivity Inc

The document may not be reproduced without the express written permission of Productivity Inc, nor may it be sold. The content must not be altered, nor may the Productivity Inc name be removed from the materials.

This material is to be used as a guide to 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.

To obtain permission, please contact trainingmn@productivity.com.

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

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

Haas CNC Mill Operator Manual
Table of 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 design of wings for the US Air Force were becoming extremely complex and hard to manufacture using conventional machine tools. MIT 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 coordinate 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 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 a 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 into the memory of the control. There are several options for getting these programs to the control. RS-232 (serial port with a computer), 3.5" Floppy Disk, Ethernet / Networking/ and USB are all viable ways to transmit and receive programs.

In order 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 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.

Updated CK 12/27/18

Notes

The Cartesian Coordinate System

The first diagram we are concerned with is called a NUMBER LINE. This number line has a zero reference point location that is called an ABSOLUTE ZERO and may be placed at any point along the number line.

Horizontal number line
"X" axis
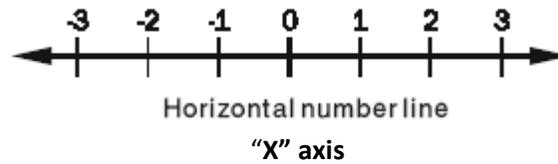
The number line also has numbered increments on either side of absolute zero.
Moving away from zero to the right are positive increments. Moving away from zero to the left are negative increments. The "+", or positive increments, are understood, therefore no sign is needed. We use positive and negative signs along with increment values to indicate its relationship to zero on the line.

Our concern is the distance and the direction from zero and is labeled as "Absolute Programming"

Remember that zero may be placed at any point along the line, and that once placed, one side of zero has negative increments and the other side has positive increments.
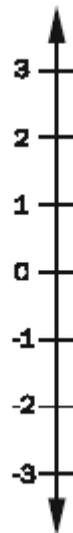
Vertical Number Line known as the "Y" axis

7

Absolute and Incremental Positioning

There are two different systems used in positioning our machine. Both will "steer" the machine where we need it to go, both will net the same results. The reason we use one more than the other, is flexibility. Below we will talk about both, and they are the first two "G-Codes"

Absolute Positioning:
With absolute positioning, we tell the machine where to move referenced to a common point, called X0 Y0 and Z0. Every time we need to move to a certain position, the ending point of that move is in direct relationship to this "common point"
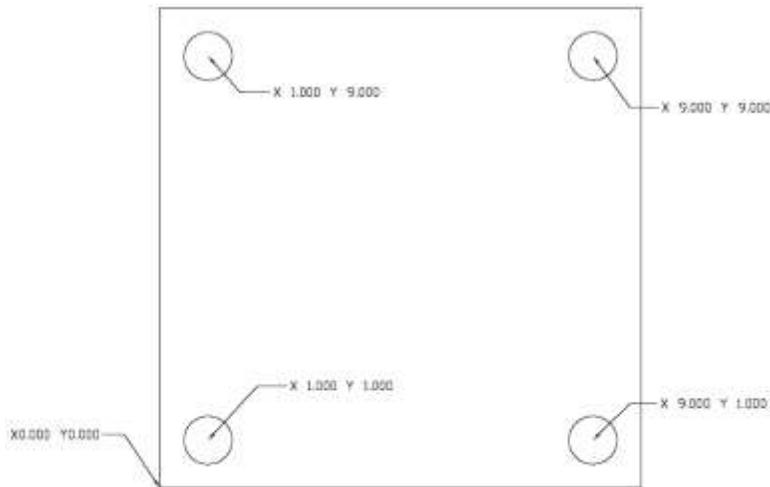
G90 Absolute Positioning

Program to move the machine to these 4 hole locations when using G90 (Abs.)

X 1.0000 Y 1.0000
X 9.0000 Y 1.0000
X 9.0000 Y 9.0000
X 1.0000 Y 9.0000

Incremental Positioning:
With incremental positioning, we are telling the machine where to go in relationship to where it currently is at. Basically like a set of directions given from where the machine stopped last.

G91 Incremental Positioning

Program to move the machine to the same 4 hole locations using G91 (Incr.)

X 1.0000 Y 1.0000
X 8.0000
Y 8.0000
X -8.0000
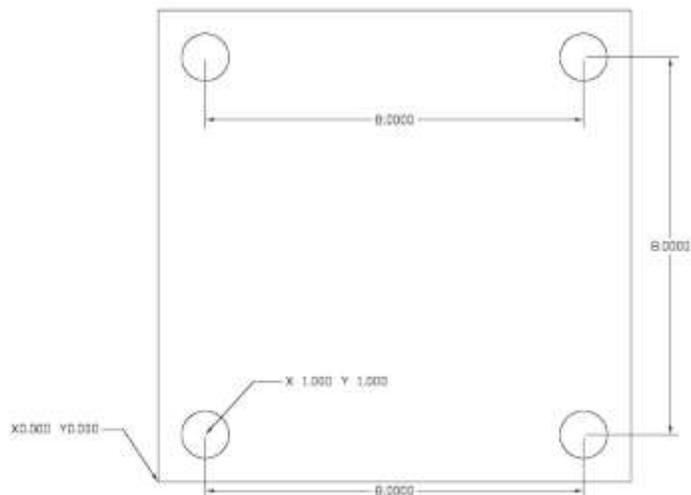
When do we decide which to use?
We switch between the two when it is more convenient. One example is look at the above 2 prints. Sometimes the print doesn't call out the hole-locations, but will give the distance between the holes.

Absolute and Incremental Exercise

G90 ABSOLUTE
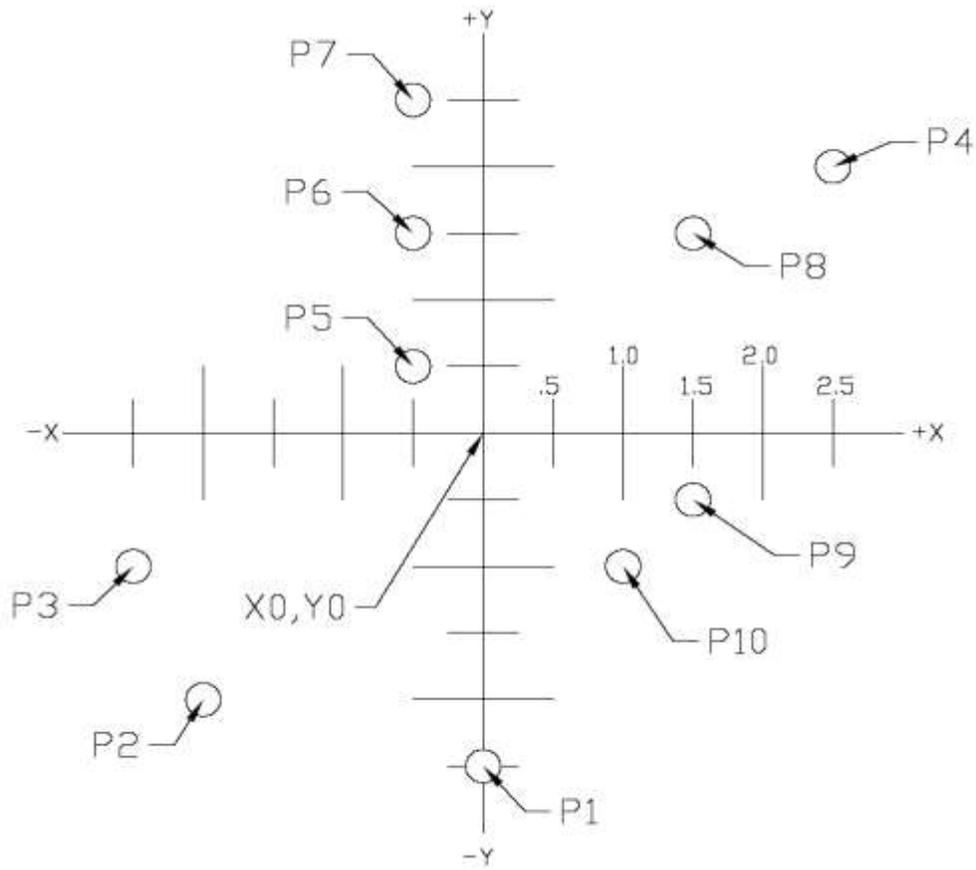
P1	X 0	Y -2.5
P2		
P3		
P4		
P5		
P6		
P7		
P8		
P9		
P10		

G91 INCREMENTAL

P1	X 0	Y -2.5
P2		
P3		
P4		
P5		
P6		
P7		
P8		
P9		
P10		

9

This diagram shows a front view of the grid as it would appear on the mill. This view shows the X and Y axes as the operator faces the mill. Note that at the intersection of the two lines, a common zero point is established.
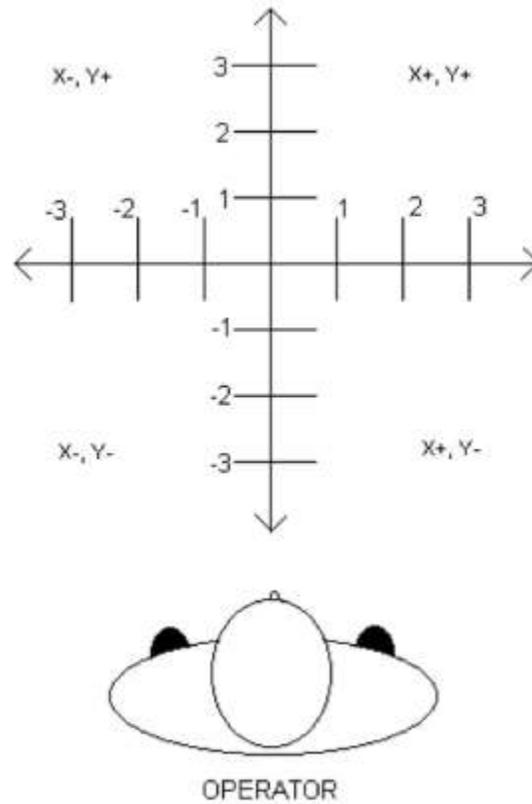
Fig. 1-4: Operator's working grid.

Whenever we set a zero somewhere on the X axis and somewhere on the Y axis, we have automatically caused an intersection of the two lines. This is known as "zero", where our X and Y lines would both be at a value of 0. We will move this point to a "Zero" position on our part so that we can steer the machine to locations that relate to our print. Another term for this "Zero" position is the origin of the part.

The following illustration shows the X & Y directions of travel on a vertical machining center with respect to a part print. It shows the positive and negative positions that our spindle would have to move with respect to the x and y axis. Also note where these two axis meet, they create a common point of "zero" where they both are at the value of "0" at the same point.
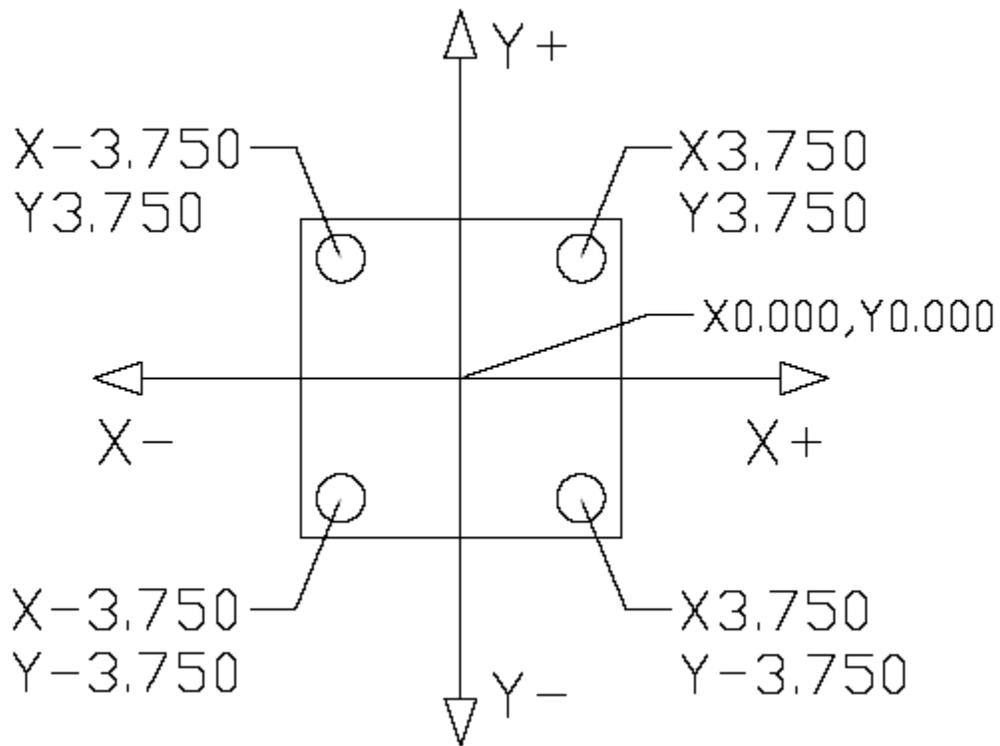
Vertical machining centers have 3 axis of travel. We have talked about two of them, the X (left to right movement), the Y (moving the part towards and away from the operator), but the last to discuss is the Z. The Z axis moves the milling spindle up or away from the part in the positive direction, and towards or into the part in the negative direction. Now that we have discussed all 3 of the axis, we can take a look at this picture of a Haas VF-8 VMC.

Vertical Machining Center Travels
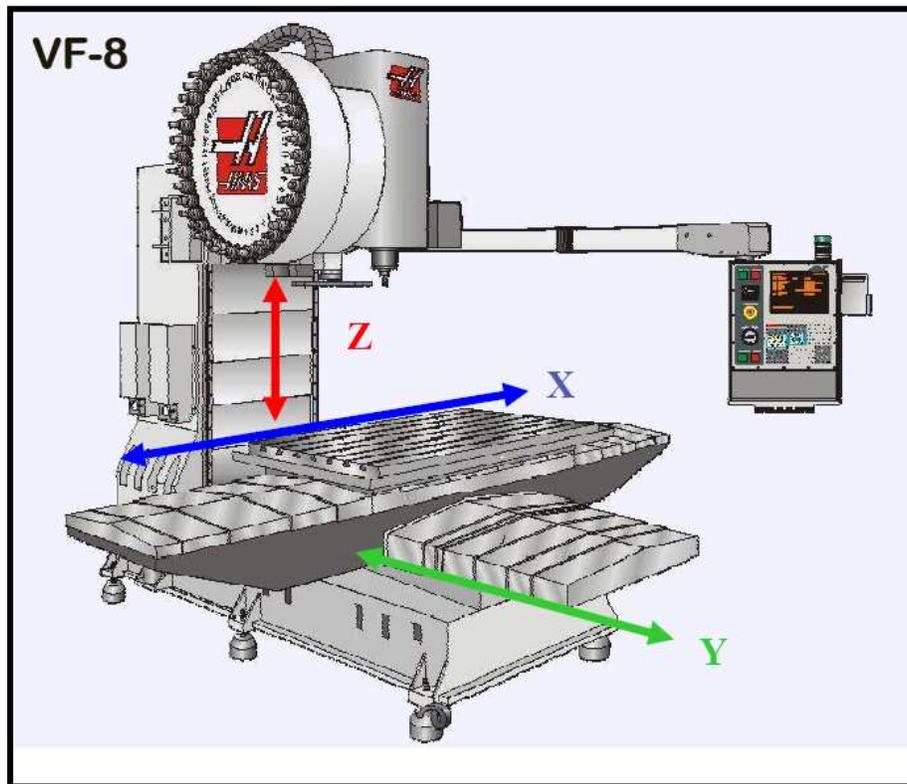
Haas VMC (VF-8) shown with the X, Y, and Z axis

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.

The first number line would be the left-to-right, or "X", axis of the machine. Positive X values would move the spindle to the right on our part, and negative would move the spindle to the left.

If we place a similar number line along the front-to-back, or "Y" axis, and wanted to move the spindle to the back of the table, these would be positive values. Moving the spindle towards the operator would be negative values in the Y axis.

The third axis of travel on our machine is the up-and-down, or "Z" axis. 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 have a resolution of .0001" inches (or .001mm).

Now theoretically all our number lines for each axis are infinite in length, we are limited to the travels of the machine we are using. Below is an example of the travels of different Haas VMCs showing how much movement we have on each model.

VMC 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"

Remember, when we are moving the machine, we are concerned with positioning the spindle. Although the machine table is the moving part, our programs are written as if the spindle was moving.

Keep in mind that the zero position may be placed at any point along each of the three number lines, and in fact, will probably be different for each setup of the machine.

The Machine Coordinate System - Machine Home Position

The principle of machine home may be seen when doing a reference return of all machine axes at machine start-up. A zero return (POWER UP/RESTART) is required when you power on machine. All three axes are moved to extreme positive locations until limit switches are reached. The reason the machine does this is to double check its position with the "Home" switches of the machine. On power up/restart the machine moves first up in the z axis to home and then moves to the x and y axis home positions at the same time.

At home position the X, Y and Z axis are all at machine 0.
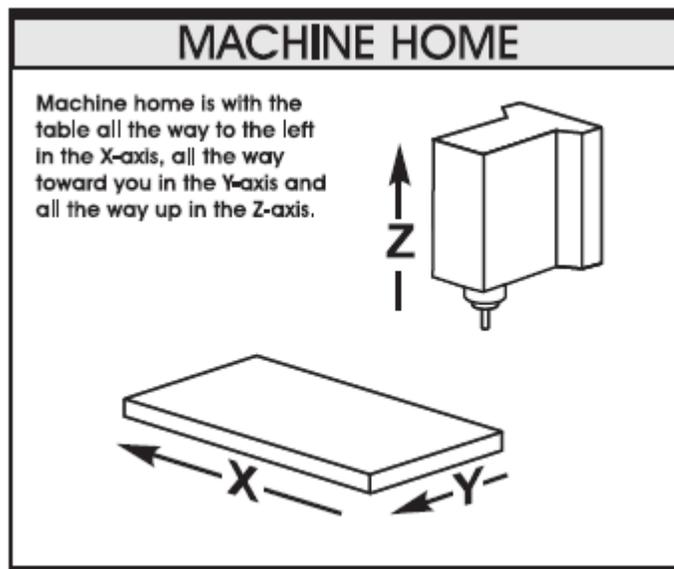
This is crucial to the operation and function of a CNC machine as all of our programs, fixtures, and tooling are based off of the **machine home position described as X0, Y0, Z0**.

Work Coordinate System

What is a "Work Coordinate"?

A work coordinate (also known as a part offset) is how we tell the machine where our part(s) are located with respect to the machine home position. Under the Work Offsets page in the control, we put the machine in jog and hand wheel the machine to the X & Y "Zero" location for our part, and use the "Part Offset Measure" key under the Reset key to set the corresponding work offset from our program (G54, G55, G56, etc.....)
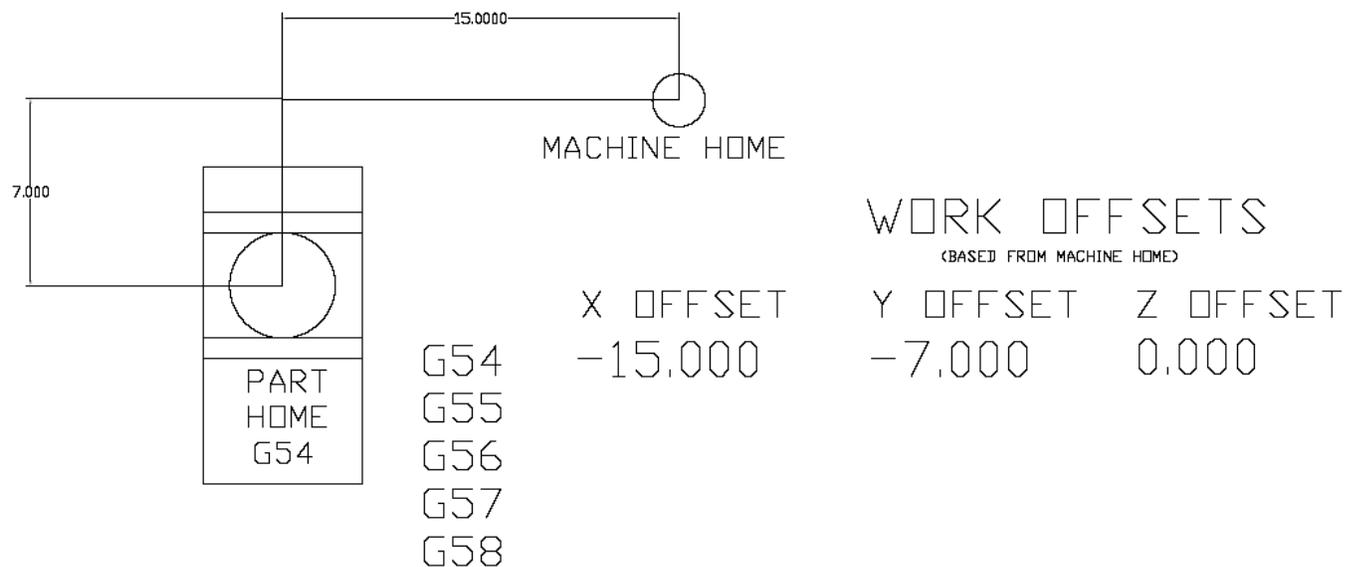
Above: The relationship of machine home to "work home", otherwise known as "work offset"

Note: Because the location of machine home zero is in the upper right-hand corner of the machine table our **values for X and Y will always be negative**.

G54 – G59 Work Offsets

These are the first G-Codes that were assigned to work coordinates. This how we tell the machine that we are working on part #1, part #2, etc. through part #6. Originally no one thought we would need more than 6 part offsets, but through time and the invention of new types of machines more were needed.

G154 P1 – G154 P99 Work Offsets

These codes are the same as **G54** to **G59**, they add more places as X & Y zero. We now can set up to 105 different "zeros" within the travels of our machine. On older Haas machines the extra work offsets were G110 to G129.

Other Work Coordinate Offsets

G52 Work Coordinate Shift

G52 will "shift" all work offsets that are set in the machine. In the Work Offsets page of the control, if we input a value of X +1.0000, all of the offsets will have a value of 1.0000 added to X. This is most commonly used in casting and forging work where we have core movement.

> **Note:** The G52 command works differently depending on the value of Setting 33. This setting selects the FANUC, HAAS, or YASNAC style of coordinates, which are listed below.

G53 Positioning in Reference to Machine Home

G53 is used when we want to move the machine a certain distance and location from Machine Home. This is quite often used if we want to establish a safe tool change position because we have large parts or tools and need to clear the tool changer. G53 is non-modal and only applies to the block which it is in.

G92 Set Work Coordinate System

G92 Can be used to set our work offsets while "on the fly" in our program. G92 was used back when machines only had one offset to choose from. We had to cut our first part, move the spindle over to the second part X&Y zero, and then call G92 X0Y0 in our program. Our work offset is now set around the second part. Using G54 – G154 P99 is much faster, more tunable, and easier to use.

Multiple Work Coordinate System
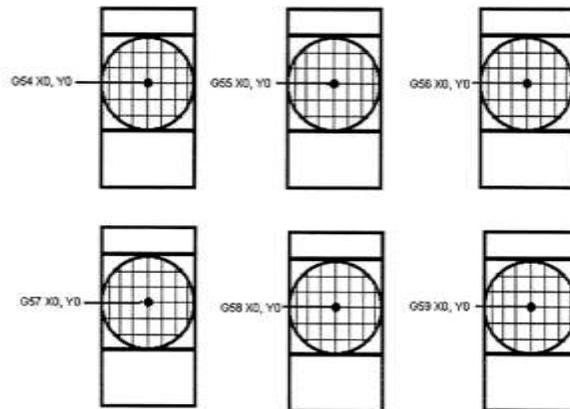
Multiple fixture setup

The figure above represents a multiple-fixture setup. Each vise will have an absolute zero once it is specified in the program. This is done by using G54 through G59. G54 through G59 and
G154 P1 through G154 P99 gives a total of 105 different work offsets which may be used.

Tool Length Offset

The tool length offset is how we tell the machine where the top face of our part is located in the Z direction with respect to machine home. The tool length offset gives the distance from the end of the tool at home position to the top face of our part or other plane that the programmer has determined as the Z zero reference point. This information is stored in the Tool Offset Memory.

Each tool in the machine will have its own defined tool length stored in the tool offset register determined by the operator during set up. Other information about each tool is stored in the *Tool Offset Register*. For each tool, the coolant tube position and the diameter or radius are also stored. In the wear section, small alterations to the tool length and diameter or radius are stored. If you cursor to the right in the tool register, additional information about the tool may be stored: the number of flutes, the actual diameter, the tool type, and tool category with respect to size and weight.

In the illustration below the spindle is sitting at the Z home position and shows the distance the spindle must go to reach +.100 above the face of the part. G43 code with an H-number tells the machine which tool length offset to use.
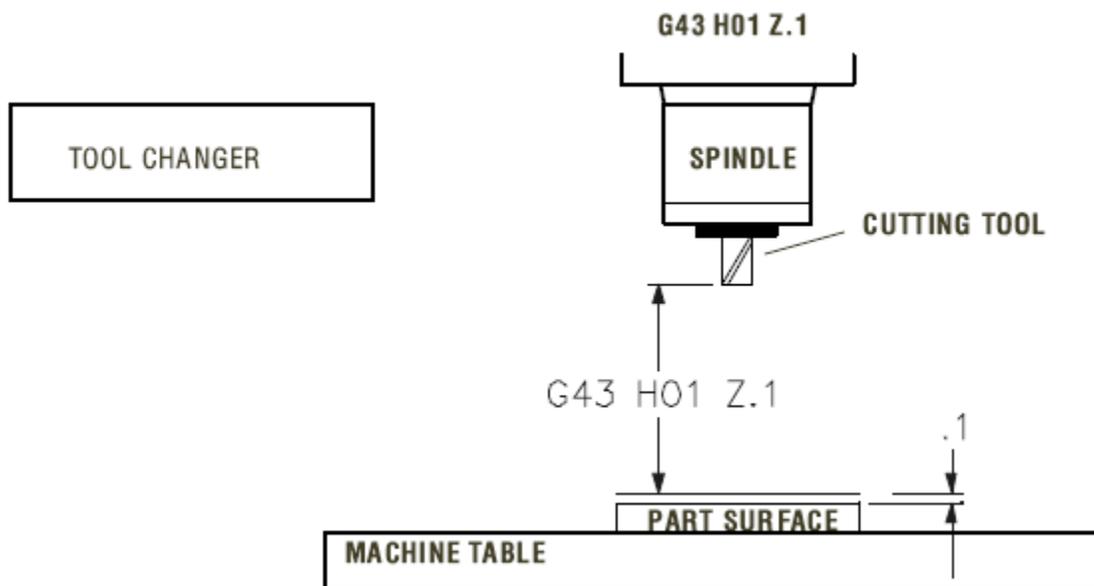
The Haas Next Generation Control

Powering on the machine

To power up a Haas machine press **POWER ON**. First the software to run the machine is loaded up. A turning Haas logo appears while the software is loading up. Then a Start Up menu appears. The door must be cycled and the [Emergency Stop] must be reset. The Haas machine tool must first find its fixed machine zero reference point before any operations can occur. After the software is loaded up pressing the [**RESET],** then the [**POWER UP]** key will send the machine to its machine zero reference location.

This establishes the machine home position.

Press any of the following: [Cancel] to clear the startup screen, [CYCLE START] to run current program or [HANDLE JOG] for manual operation.

General Machine Keys

Power On - Turns CNC machine on.

Power Off - Turns CNC machine tool off.

Emergency Stop - Stops all axis motion, stops spindle, tool changer and turns off coolant pump.

Jog Handle – Jogs axis selected, also may be used to scroll through programs, menu items while editing and also altering feeds and speeds.

Cycle Start – Starts program in run mode or graphics mode.

**Feed Hold –** Stops all axis motion. Spindle will continue to turn**.**

Reset – Stops machine, will rewind program.

Power Up/Restart – Axis will return to machine zero and tool change will occur per Setting 81
Recover – If a tool change is stopped in middle of a cycle an alarm will come up. Push the **Recover** button and follow the instructions to bring the tool change cycle to the beginning.

16 Software Control Display
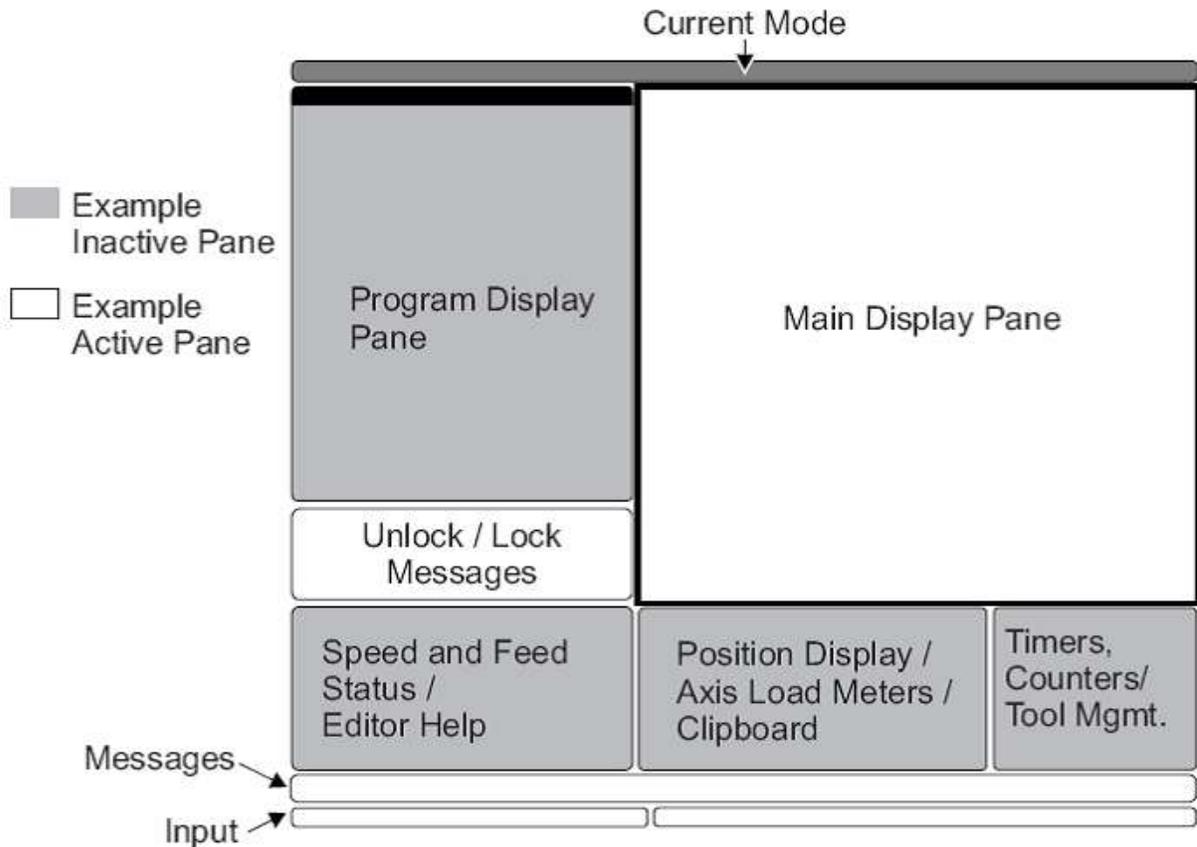
The new 16 software has a larger display and more panes than older versions. Above is the basic display layout. What is displayed depends on which display keys have been used. The only pane active is the one with the white background. Only when a pane is active may changes be made to data.

Control functions in Haas machine tools are organized in three modes: **Setup, Edit** and **Operation**

Access Modes using the mode keys as follows:

Setup: ZERO RET, HAND JOG keys. Provides all control features for machine setup.

Edit: EDIT, MDI/DNC, LIST PROG keys. Provides all program editing, management, and transfer functions.

Operation: MEM key. Provides all control features necessary to make a part.

Current mode is displayed at top of display.

Functions from another mode can still be accessed within the active mode. For example, while in the Operation mode, pressing OFFSET will display the offset tables as the active pane in the Main Display Pane and offsets may be altered; press OFFSET to toggle the offset display. While running a part in operation mode another program may be edited in the Main Display Pane. Press PROGRM CONVRS in most modes to shift to the edit pane for the current active program.

NGA Control Display

Active G Codes

Program Display

Mode Bar

Tool Info

Main Display Panel

Coolant Level

Input Bar

Spindle and Feed Status

Icon Bar

Override Status

Axis Servo Loads

Spindle Load
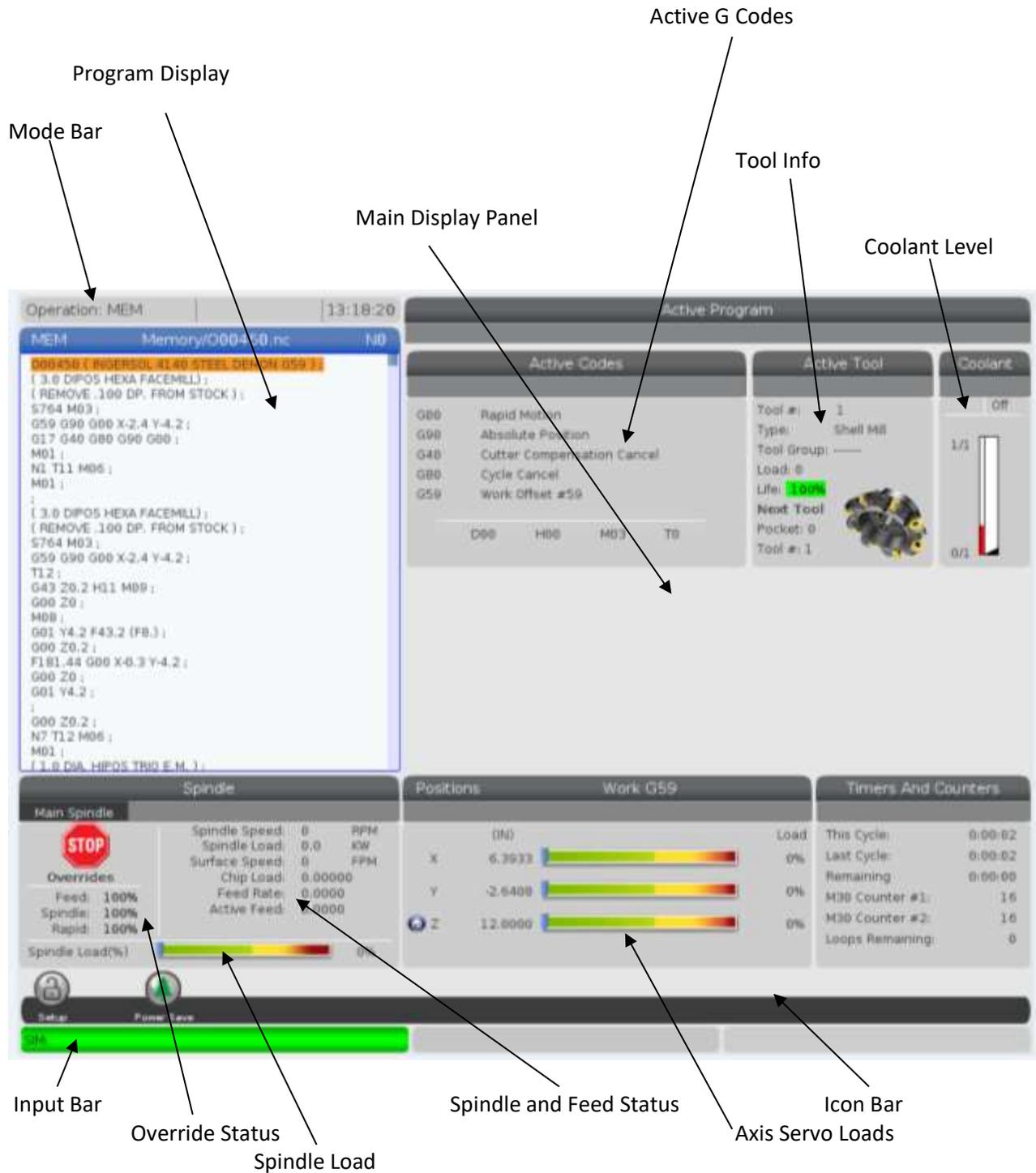
Above is a view of the control panel in Memory Mode with the Program Key active on the Display Key. The Main Display Panel varies according to which of the DISPLAY keys are active.

Keyboard Introduction

The keyboard is divided into eight different sectors: Function Keys, Jog Keys, Override Keys, Display Keys, Cursor Keys, Alpha Keys, Number Keys and Mode Keys. In addition, there are miscellaneous keys and features located on the pendant and keyboard which are described briefly on the following pages.
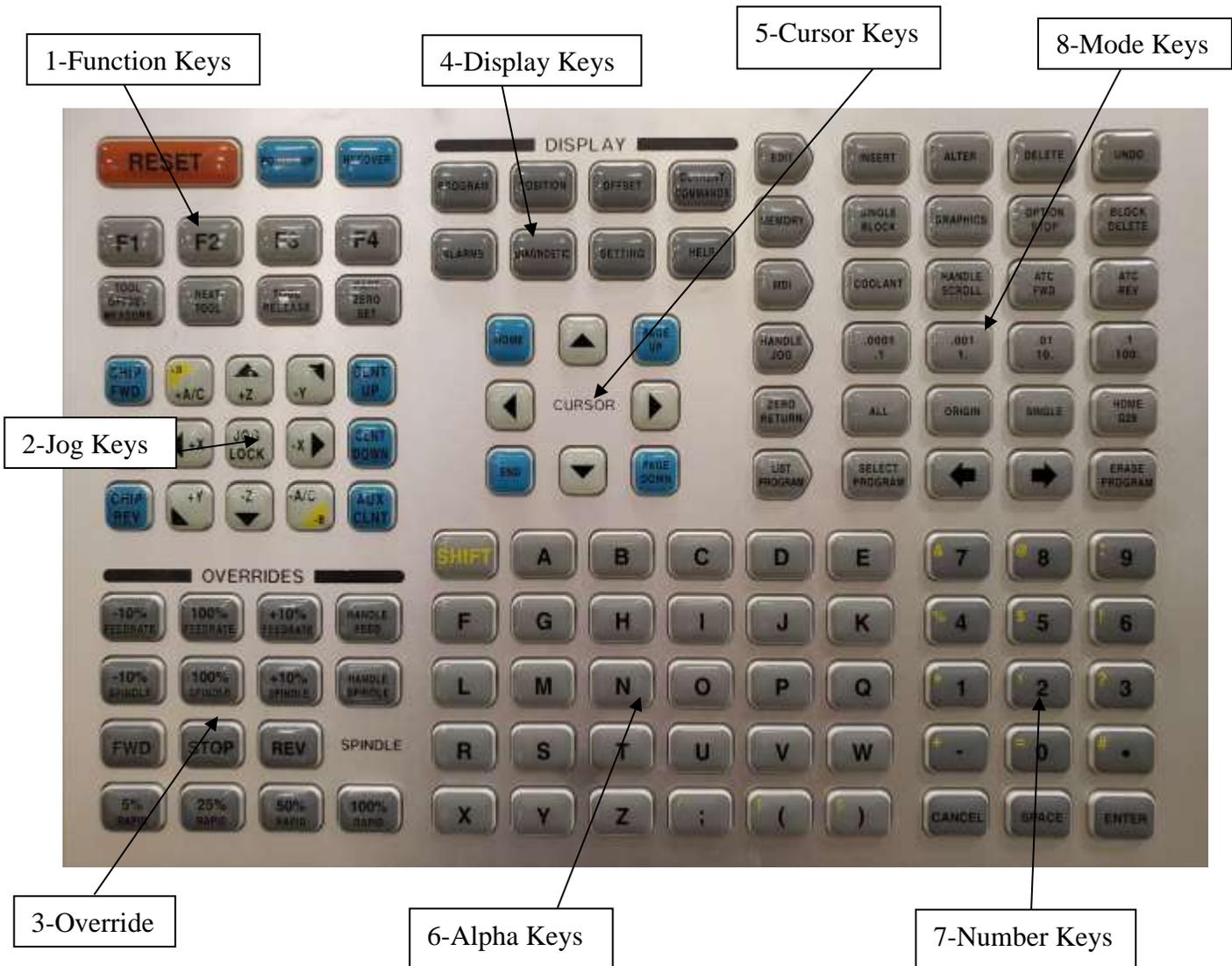
Next Generation Keyboard Changes

Only four keys have been changed on the keyboard with the NGA Control

The [DRY RUN] KEY has been changed to the [GRAPHICS] KEY.

The NGC does not provide the Dry Run option as in older controls.

The [ORIENT SPINDLE] KEY has been changed to the [HANDLE SCROLL] KEY.

This allows the handle jog wheel to be used to scroll thru offsets or a program.

If you want to orient the spindle, the M19 command must be run in the MDI mode.

The [SEND] AND [REC] KEYS have been changed to FORWARD and BACK KEYS.

These keys allow navigation forward and backward thru page history.

NGC does not provide for a RS232 port.

Only Ethernet connections are available or an upgrade to WIFI.

Display Keys

Display keys determine what is shown in the Main Display Panel.
Below the **[POSIT] Display Key** is active in the Memory Mode.
Tabs highlighted in orange tell you what axis you will see.   Below the All tab is active.
Machinist's when they are setting up like to have the All tab highlighted. It shows where they are at with respect the part zero. Also the distance is it going to go before the next block.

By depressing the [ALTER] key the display may be customized as shown below:

The **[Offset Key]** is active below.

Tool Length Geometry values are shown along with Tool Diameter. If tool life management is being used red will indicate the tool life has expired, yellow is a warning.   Also Wear values may be entered for H(Length) and D(Tool Diameter).

Operation: MEM		09:20:06

MEM   Memory/O04040.nc   N0

```
O04040 (ATM);
T1 M06 (#7 DRILL) ;
G00 G90 G54 X0. Y0. ;
S4000 M03 ;
G43 H01 Z0.1 M88 ;
G83 Z-0.1 R0.1 F11. Q0.25 L0 ;
X1. L8 G91 ;
G00 G80 Z0.1 M09 ;
G53 Z0 ;
M99 ;
```

Offsets

Tool | Work

Active Tool: 1

Tool Offset	Length Geometry	H(Length) Wear	Diameter Geometry	Diameter Wear
1 Spindle	3.0000	0.	2.0000	0.
2	3.0000	0.	0.5000	0.
3	3.0000	0.	0.6250	0.
4	3.0000	0.	0.	0.
5	3.0000	0.	0.	0.
6	3.0000	0.	0.	0.
7	3.0000	0.	0.	0.
8	3.0000	0.	0.	0.
9	3.0000	0.	0.	0.
10	0.	0.	0.	0.
11	0.	0.	0.	0.
12	0.	0.	0.	0.
13	0.	0.	0.	0.
14	0.	0.	0.	0.
15	0.	0.	0.	0.
16	0.	0.	0.	0.
17	0.	0.	0.	0.
18	0.	0.	0.	0.

Enter A Value

TOOL OFFSET MEAS   Tool Offset Measure   **F1** Set Value   **ENTER** Add To Value   **F4** Work Offset

Spindle

Main Spindle

STOP

Overrides
Feed: 100%
Spindle: 100%
Rapid: 100%

Spindle Load(%)   0%

Spindle Speed:	0	RPM
Spindle Load:	0.0	KW
Surface Speed:	0	FPM
Chip Load:	0.00000	
Feed Rate:	0.0000	
Active Feed:	0.0000	

Positions   **Operator**

	(IN)		Load
X	0.0000		0%
Y	0.0000		0%
Z	0.0000		0%

Timers And Counters

This Cycle:	0:00:11
Last Cycle:	0:00:00
Remaining	0:00:00
M30 Counter #1:	167
M30 Counter #2:	167
Loops Remaining:	0

Setup   Power Save   Tool Wear

SIM: |

23

Cursor to the right gives a 2nd page >   Information on tool type, material and category may be entered.

Tool Offset	Flutes	Actual Diameter	Tool Type	Tool Material	Tool Pocket	Category
1 Spindle	5	2.0000	Shell Mill	Carbide	Spindle	Heavy
2	4	0.5000	End Mill	Carbide	2	
3	4	0.6250	End Mill	Carbide	1	*
4	0	0.	Spot Drill	H.S. Steel	3	
5	0	0.	Drill	H.S. Steel	4	
6	0	0.	Drill	H.S. Steel	5	
7	0	0.	Drill	H.S. Steel	6	
8	0	0.	Drill	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	

Offsets — Tool / Work — Active Tool: 1

Enter A Value

TOOL OFFSET MEAS Tool Offset Measure F1 Set Value ENTER Add To Value F4 Work Offset

Hand Jog

Cursor to the right again will give a probing page. On the far-right different tool probing options can be selected. The information needed for probing is entered and probing may be performed from the offsets page.

Tool Offset	Approximate Length	Approximate Diameter	Edge Measure Height	Tool Tolerance	Probe Type
1 Spindle	3.0000	2.0000	0.	0.	1-L Rotatng
2	2.5000	0.5000	0.	0.	3-Len & Dia
3	2.8000	0.6250	0.	0.	3-Len & Dia
4	0.	0.	0.	0.	2-L Non Rot
5	0.	0.	0.	0.	2-L Non Rot
6	0.	0.	0.	0.	2-L Non Rot
7	0.	0.	0.	0.	2-L Non Rot
8	0.	0.	0.	0.	None
9	0.	0.	0.	0.	None
10	0.	0.	0.	0.	None
11	0.	0.	0.	0.	None
12	0.	0.	0.	0.	None
13	0.	0.	0.	0.	None
14	0.	0.	0.	0.	None
15	0.	0.	0.	0.	None
16	0.	0.	0.	0.	None
17	0.	0.	0.	0.	None
18	0.	0.	0.	0.	None

Tool / Work — Active Tool: 1

Enter A Value

TOOL OFFSET MEAS Automatic Probe Options F1 Set Value ENTER Add To Value F4 Work Offset

Hand Jog

Position: (IN)   Jog Rate: 0.0010

Going back to the first offset page curser to the top to highlight **Tool.** Then highlight **Work.** This opens the work offsets page as below.
One may also toggle between work and tool offsets pages by using the **[F4]** key.

[Current Commands] key gives many different options. Note the **Timers** tab is highlighted. On this page current date and time, cycle time, M30 counters for keeping track of the number of parts and Macro values may be indicated.   One may cursor down into the page to reset the M30 counter by pressing the [ORIGIN] key.

Current Commands						
Timers	Macro Vars	Active Codes	ATM	Tool Table	Calculator	

Date:	11-04-2016	Loops Remaining:	0
Time:	15:05:24	M30 Counter #1:	167
Time Zone:	GMT	M30 Counter #2:	167
Power On Time:	715:05:47	Macro Label #1:	LABEL 1
Cycle Start Time:	0:08:53	Macro Assign #1:	
Feed Cutting Time:	0:02:21	Macro Label #2:	LABEL 2
This Cycle:	0:00:11	Macro Assign #2:	
Last Cycle:	0:00:00		

Enter Date In The Format MM-DD-YYYY

ENTER Set Value

[Macro Vars] key gives the values of different variables # used in Macro Programming. Note 10000 must be added to the old legacy numbers to find the equivalent variable. Variable #188 becomes #10188. Once inside the variable table the variable # needed can be keyed in and the down cursor key will do a search for the particular variable.

Current Commands						
Timers	**Macro Vars**	Active Codes	ATM	Tool Table	Calculator	

Macro Variables		10400 - 10999 (Global) ▶	
(Local) 1 - 33	(Global) 10000 - 10199	(Global) 10200 - 10399	

Var	Value	Var	Value	Var	Value
1		10000	0.000000	10200	0.000000
2		10001	0.000000	10201	0.000000
3		10002	0.000000	10202	0.000000
4		10003	0.000000	10203	0.000000
5		10004	0.000000	10204	0.000000
6		10005	0.000000	10205	0.000000
7		10006	0.000000	10206	0.000000
8		10007	0.000000	10207	0.000000
9		10008	0.000000	10208	0.000000
10		10009	0.000000	10209	0.000000
11		10010	0.000000	10210	0.000000
12		10011	0.000000	10211	0.000000
13		10012	0.000000	10212	0.000000
14		10013	0.000000	10213	0.000000

*Legacy 3 digit macros begin at 10000 Range. i.e. Macro 100 will be displayed as 10100.

Highlighting the **Active Codes** tab gives the below panel.   It may be used for trouble shooting a program. All current alphabet address codes are shown with their values. Below the T and M values are 1000. This indicates that the machine is using ATM (Advanced Tool Management)

Current Commands					
Timers	Macro Vars	Active Codes	ATM	Tool Table	Calculator

| G-Codes	Address Codes		DHMT Codes		Speeds & Feeds	
G00	N	0	D	00	Programmed Feed Rate	11.0000
G17	X	0.	H	1000	Actual Feed Rate	11.0000
G90	Y	0.	M	88	Programmed Spindle Speed	4000
G94	Z	-0.1000	T	1000	Commanded Spindle Speed	4000
G20	I	0.			Actual Spindle Speed	3966
G40	J	0.				
G43	K	0.				
G83	P	0				
G98	Q	0.2500				
G50	R	0.1000				
G54	O	O04040				
G64	A	0.				
G69	B	0.				
G255	C	0.				
	U	0.				
	V	0.				
	W	0.				
	E	0.				

With the **ATM** tab highlighted the Advanced Tool Management panel is shown below. To use first a 1000 group must be set up and how the tool is managed (below the number of holes drilled is used). The **[F4]** key is used to switch boxes to denote the tools being used. Below Tools 5, 6 and 7 are used.

Current Commands					
Timers	Macro Vars	Active Codes	ATM	Tool Table	Calculator

F4 To Switch Boxes   Allowed Limits   Active Tool: 6

Group	Exp#	Order	Usage	Holes	Warn	Load	Action	Feed	Total
All									
Exp	0								
1000	1/3	Ordered	0	100	30%	0%	Alarm	00:00	00:00

Tool Data For   Group: 1000

Tool#	Life	Usage	Holes	Load	Limit	Alarm	Feed	Total	H-Code	D-Code
5	0%	2	104	0%	0%	0	0:02:01	0:05:11	5	5
6	36%	3	64	0%	0%	0	0:01:37	0:57:54	6	6
7	100%	1	0	0%	0%	0	0:00:02	0:00:41	7	7

Displays the number of holes the tool has drilled. ENTER Set Value ORIGIN To Clear.

Highlighting the **Tool Table** tab shows which tools are in which pockets.

Pocket	Category	Tool
Spindle		6
1		3
2		2
3		4
4		7
5	Heavy	1
6*		5
7		8
8	-	
9	Large	10
10	-	
11		12
12		13
13		14
14		15
15		16

Buttons:
- Set pocket as large [L]
- Set pocket as heavy [H]
- Set pocket as XL [X]
- Clear category [SPACE]
- Set tool [###] + [ENTER]
- Clear tool [0] + [ENTER]
- Reset table [ORIGIN]

Active Tool 6   Next Pocket 6

* Indicates Current Tool Changer Pocket
Green indicates a large pocket. Yellow indicates an extra large pocket.

With side mount tool changing machines the pot position of each tool is saved. Above tool number 6 is in the spindle. The asterisk indicates which tool and pocket are staged for the next tool index. In this example it is tool 5 in pocket 6.

With Cat 40 tools any tool greater than 4 lbs. must be denoted as **Heavy** tool. With the **Heavy** tool designation, the machine will only index that particular tool at 25% rapid. This will prevent the tool from inadvertently being thrown. Also, the **spindle probe** must be denoted as a heavy tool because a tool change at 100% may compromise the calibration.

Large tools are considered any tool with a diameter of 3" or larger. If two large tools are put in pockets adjacent to each other a crash may occur possibly damaging the pot holder. This is prevented by designating one tool a Large tool. Note above that Tool 10 in pocket 9 is designated as a **Large** tool.   It is always stored in pocket #9. Also, no tools will be stored in the pockets either side of pocket #9. Large tools also index only at a 25% rapid.

The Calculator tab opens 3 different calculators, Standard, Milling and Tapping. Below the Milling is tab is open. The calculator will give feed rates and spindle speeds for different surface speeds and diameters. If the work material and the tool material (tool steel or carbide) are defined it will give recommended speeds and feeds. If the chip load or surface speed are outside its recommended values the box will appear yellow. Below the .002 chip load is outside its recommended range. The Haas library recommendation is .004 to .008 in/rev-tooth for low carbon unalloyed steel.

[HELP] key basically leads you to the Operators Manual. It is stored in the control. Use the cursor key to highlight the topic you want more information from. Below SETTINGS is highlighted. Pressing the [ENTER] key will bring up a list of settings and a description. The total number of settings in a Haas is 277.
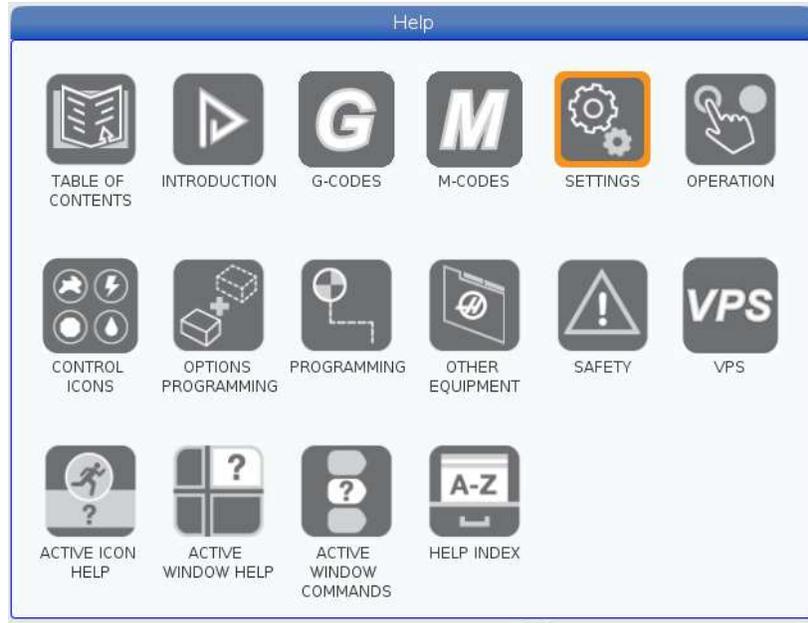
Highlighting Control Icons will give all the different icons which indicate the status of your machine. Below the open lock on the Icon Bar indicates the machine is in the Set Up Mode.
Highlighting other icons will bring up other locations in the manual.

[SETTINGS] key brings up all the different groups of settings. See below. Highlight the group which is needed and press the right cursor key [>] to open up the different settings. Below the Control Panel group is highlighted.
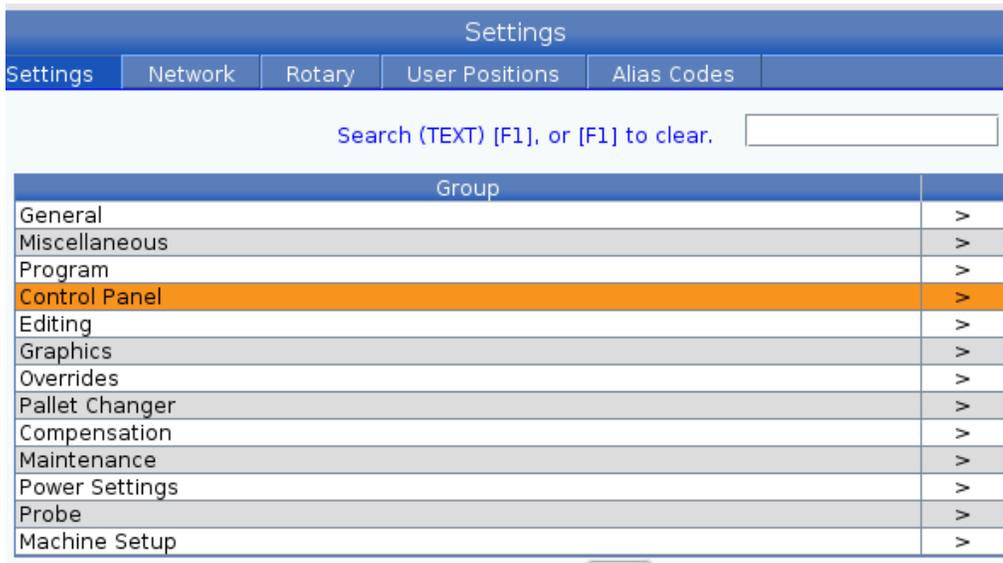
Below setting 76 is highlighted. Pressing the right cursor key [>] gives a drop down. Select one of the options by highlighting it and press [ENTER]. It is recommended that the Tool Release key is locked out if it is not used. Inadvertent pressing this key and the tool in the spindle will fall out.

Also note setting 163 [DISABLE .1 JOG RATE] is set to ON. This prevents an accidental .1 jog rate set and crash. At .1 jog rate one click on the HANDLE JOG will move the machine .1.
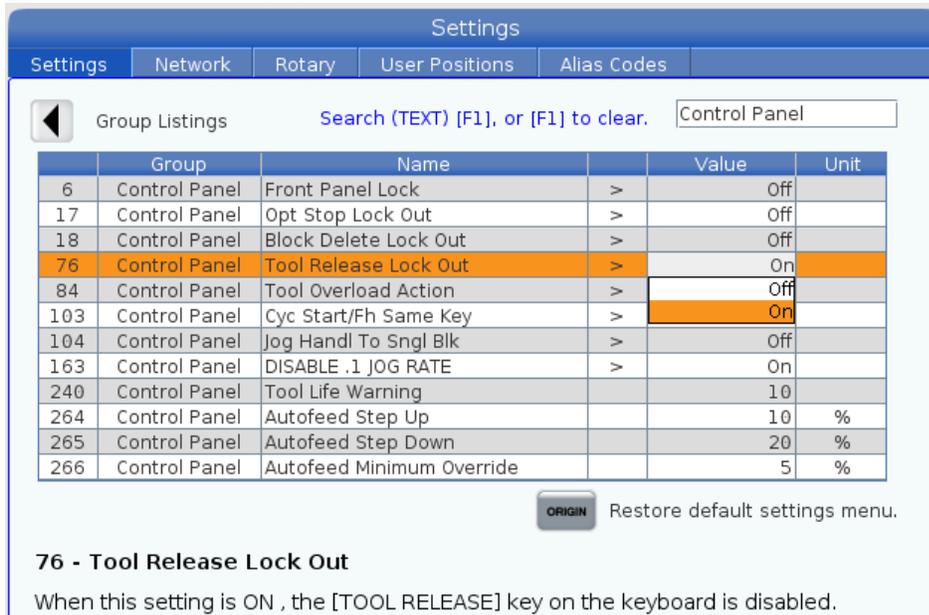
On the Settings page other tabs at the top are seen. The Network tab is used to set up an Ethernet connection. The Rotary Tab to specify a rotary table, and also a User Positions and Alias codes tabs.

Settings List

Scroll through Settings with the cursor keys or the Jog Handle.
To do a search, key in the setting number in the input line then press the [F1] key as shown below.

Above setting 15 was searched using the F1 key.   Note a short description is given on the particular setting. Full text may be seen by pressing the [HELP] key. Some settings are changed by selecting one of the drop down options. Other settings are numbers which can be changed by keying the new number.

There are many settings which give the user various options over the control of their machine tool. Read the Settings section of the operator's manual for all the possible options. Here are some of the more useful settings.

Setting 1   **AUTO POWER OFF** – This turns the machine off after it is idle for the number of minutes defined in this setting.

Setting 2   **POWER OFF AT M30** – This option will power off the machine tool when an M30 command is executed. In addition, for safety reasons, the control will turn itself off if an overvoltage or overheat condition is detected for longer than four minutes.

Setting 8   **PROG MEMORY LOCK** – When this is Off, control program memory can be modified. When this setting is turned On, memory edits cannot be done and programs cannot be erased.

Setting 9   **DIMENSIONING** – This changes the machine control from inch to metric, which will change all offset values and position displays accordingly. This setting will not change your program to either inch or metric.

Setting 15 **H & T CODE AGREEMENT** – When this is OFF, no special functions occur. When it's ON, a check is made to ensure that the H offset code matches the tool presently in the spindle. Usually you have one offset per tool, and it's usually the same number as the tool number. If it's not the same and this setting is ON, you will get an alarm of H AND T NOT MATCHED. This check can help prevent crashes. If you need to use a different offset number or more than just one, this setting will need to be switched OFF. In program restart this check is not done until motion begins.

Setting 23 **9XXX PROGS EDIT LOCK** – This is an On/Off setting. When it is On, the 9000 series programs (usually the Quick Code source file or macro programs) are invisible to the operator and cannot be uploaded or downloaded. They also cannot be listed, edited, or deleted.

Setting 30 **4TH AXIS ENABLE** – This is selected when using a rotary axis.

Setting 31 **RESET PROGRAM POINTER** – When this is On, the RESET key will send the cursor (program pointer) back to the beginning of the program. Normally set to on.

Setting 32 **COOLANT OVERRIDE** – This setting controls how the coolant pump operates. The settings are: Normal, Ignore and Off. When it is set on Normal, coolant commands respond as programmed. If set on Ignore, an M08 or M88 command in the program will not turn the coolant on (i.e., the command will be ignored), but it can be turned on manually using the COOLNT key. If this setting is Off, the coolant cannot be turned on at all, and the control will give an alarm when it reads an M08 or M88 command in a program.

Setting 33 **COORDINATE SYSTEM** – This setting changes the way the G92/G52 offset system works. It can be set to Fanuc, Haas, or Yasnac. Normally it is set to Fanuc.

Setting 36 **PROGRAM RESTART** – When it is OFF, starting a program from anywhere other than the beginning of a program or a tool sequence may produce inconsistent results or crashes. When it is ON, you are able to start a program from the middle of a tool sequence. You cursor onto the line you want to start on and press CYCLE START. It will cause the entire program to be scanned to ensure that the correct tools, offsets, G codes, and axes positions are set correctly before starting and continuing at the block where the cursor is positioned. Some alarm conditions are not detected prior to motion starting. You could leave this setting ON all the time if you want, but it might do some things unnecessarily, so you would probably prefer to turn it OFF when you're done using it.

Setting 40 **TOOL OFFSET MEASURE** – This setting selects how tool size is specified for cutter compensation: radius or diameter.

Setting 51 **DOOR HOLD OVERRIDE** – This setting is no longer available to use in new machines. On older machine when it is off, a program cannot be started if the doors are open, and opening the doors will cause a running program to stop – just like a feed hold. When this setting is On, the door condition is ignored. This setting will always be Off when the control is powered up.

Setting 56 **M30 Restores Default Codes –** Recommended to set to ON. Reset will also return the machine to the default codes.

Setting 76 **TOOL RELEASE LOCK OUT** – When this is On, the TOOL RELEASE button on the control keypad is disabled.

Setting 84 **TOOL OVERLOAD ACTION** – This is used to determine tool overload conditions as defined by the Tool Load monitor page in the CURNT COMDS display (use PAGE DOWN to get there). A tool overload condition can result in one of four actions by the control, depending on Setting 84. ALARM will generate an alarm when overload occurs; FEED HOLD will stop with a Feed Hold when overload occurs; BEEP will sound an audible alarm when overload occurs; or AUTOFEED will automatically decrease the feed rate.

Setting 85 **MAX CORNER ROUNDING** – This setting is used to set the corner rounding accuracy required by the user. The accuracy defined in Setting 85 will be maintained even at maximum feed rate. The control will only slow at corners when it is needed.   For most applications it is set to .025. **Setting 191** works along with this setting. It is normally set to medium.

Setting 88 **RESET RESETS OVERRIDE** – When this is On, the RESET key sets all overrides back to 100%.

Setting 101 **FEED OVERRIDE > RAPID** – When this setting is OFF, the machine will behave normally. When it is ON and HANDLE CONTROL FEED RATE is active, the jog handle will affect both the feed rate override and the rapid rate override simultaneously. That is, changing the feed rate override will cause a proportional change to the rapid rate. The maximum rapid rate will be maintained at 100% or 50% according to setting 10. (Any Mill Control Ver. 10.22 and above; any Lathe Control Ver. 4.11 and above.)

Setting 103 **CYC START / FH SAME KEY** – When this setting is ON, the CYCLE START button functions as the Feed Hold key as well. When CYCLE START is pressed and held in, the machine will run through the program; when it's released, the machine will stop in a Feed Hold. This gives you much better control when testing a new program. When you are done using this feature, turn it Off. This setting can be changed while running a program. It cannot be ON when Setting 104 is ON. When one of them is turned ON, the other will automatically turn OFF. (Any Mill Control Ver. 9.06 and above; any Lathe Control Ver. 4.11 and above.)

Setting 104 **JOG HANDL TO SNGL BLK** – When running a program in MEM mode in the Program or Graphics display, you can use the SINGLE BLOCK key to cycle through your program one line at a time with each press of the CYCLE START button, when the machine is running or you are in Graphics. If you turn Setting 104 ON, and SINGLE BLOCK has been selected. You first press the CYCLE START button, then each counterclockwise click of the jog handle will step you through a program line by line. Turning the handle clockwise will cause a FEED HOLD. This setting can be changed while running a program. It cannot be ON when Setting 103 is ON. When one of them is turned ON, the other will automatically turn OFF. (Any Mill Control Ver. 9.06 and above; any Lathe Control Ver. 4.11 and above.)

Setting 114 CONVEYOR CYCLE (MIN) – If this is set to zero, the conveyor will operate normally. If another number is entered, it defines how long (in minutes) each cycle will be when the chip conveyor is turned on. The chip conveyor cycle is started with either an M code (M31 or M32) or with the control CHIP FWD/REV keys. It will stay on for the time defined in Setting 115, then turn off and not restart until the cycle time in Setting 114 has elapsed.

Setting 115 CONVEYOR ON TIME (MIN) – This setting works with Setting 114, which defines the conveyor cycle time. Setting 115 defines how long the chip conveyor will stay on during each cycle.

Setting 130 RIG. TAP RETRACT SPEED – This feature augments one introduced in version 10.13, the Quick reversal out of a G84 rigid-tapped hole. If it is set to 0 or 1, the machine behaves normally. Setting it to 2 is the equivalent of a G84 command with a J value of 2. That is, the spindle will retract twice as fast as it went in. If this setting is set to 3, the spindle will retract three times as fast. Note that specifying a J value in a G84 command for rigid tapping will override Setting 130. (Any Mill Control Ver. 10.18 and above)

Setting 163 DISABLE .1 Jog Rate – Most shops disable .1 jog rate to reduce tool and probe crashes. To disable set to **ON**.

The **[DIAGNOSTICS]** key is primarily used by service technicians.
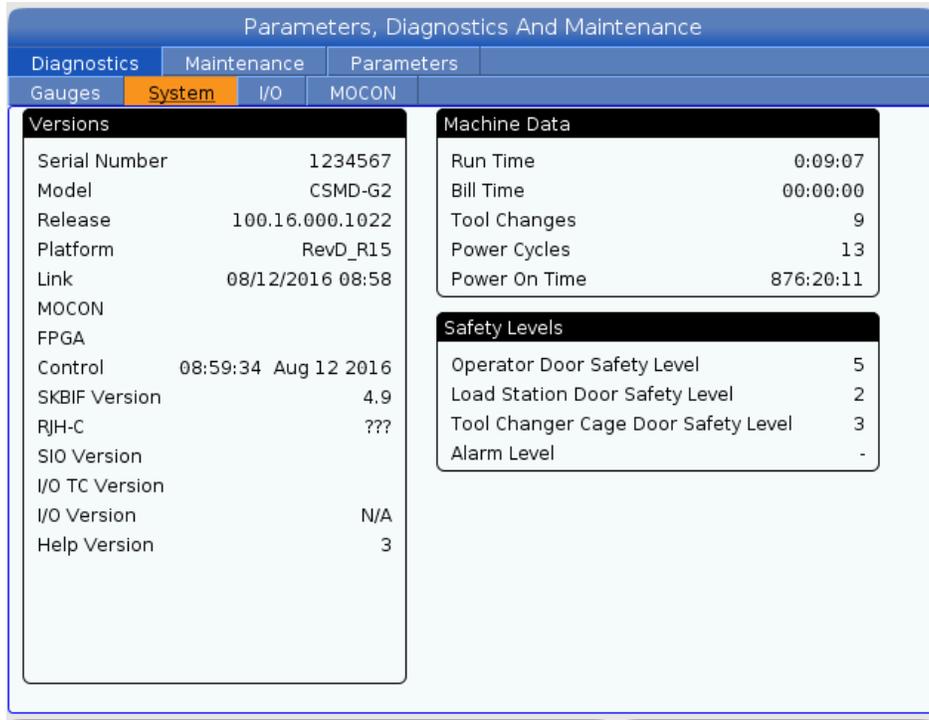
Highlighting the Parameters Tab shows which options have been purchased. Also some options have a Try Out option for 200 hours. The Activation tab is used to reactivate the machine.

Pressing the **[ALARMS]** key will give a description of any active alarms under the Active Alarms tab. Other tabs will give Alarm History and Key History. It also has an Alarm Viewer. The Haas manual does not give a list of alarm codes. One must use the Alarm Viewer to get a descriptions of a peculiar alarm. Below the Alarm Viewer tab is open. Alarm 332 is highlighted.
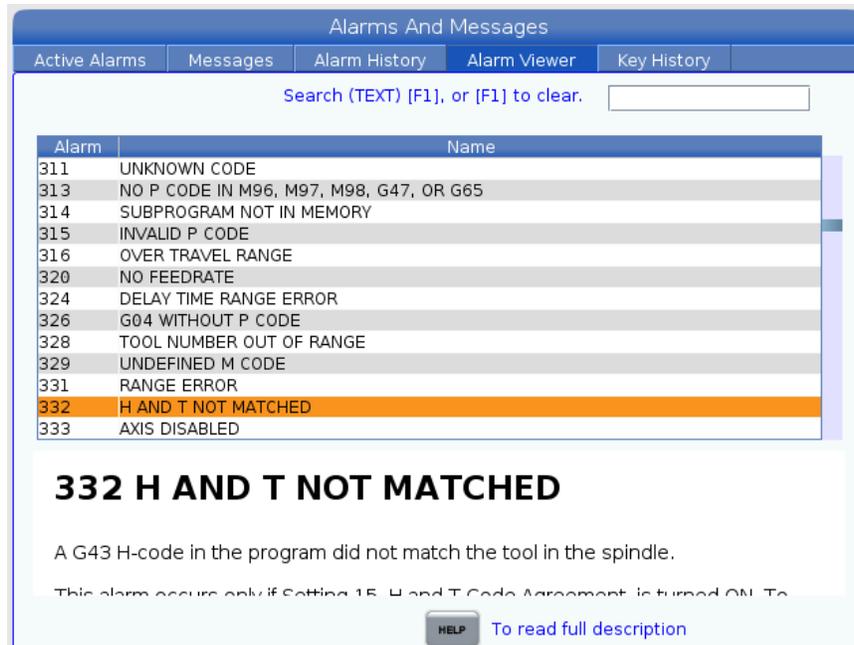
1 – Function Keys

F1 – F4 – Perform different functions depending on which mode the machine is in. Example in offsets mode **F1** will directly enter value that you give it into to offset register.

TOOL OFFSET MEASURE – Will take machine Z position readout at bottom of offset screen and load it in to the highlighted tool offset register.

NEXT TOOL – After pressing Tool Offset Measure button in a set up this will select the next tool and make a tool change

TOOL RELEASE - Releases tool from spindle in MDI, Zero Return or Handle mode. A button on the front of the spindle will do the same thing. Most users disable this key by turning **on Setting 76 (Tool Release Lock Out).**

PART ZERO SET – Records work coordinate offsets into the highlighted register.

2 – Jog Keys

Chip FWD *(Chip Auger Forward)* – Turns the optional chip auger in a direction that removes chips from the work cell.

Chip Stop *(Chip Auger Stop)* – Stops auger movement.

Chip REV *(Chip Auger Reverse)* – Turns the chip auger in reverse.

CLNT UP *(Coolant Up)* – Pressing this key will position the coolant stream one position higher.

CLNT DOWN *(Coolant Down)* **–** Pressing this key positions the coolant stream one position lower. Coolant stream position will appear in tool length offset register when position is highlighted.

AUX CLNT *(Auxiliary Coolant)* – Turns on the optional Through-the-Spindle (TSC) coolant (in MDI mode).

+X, -X *(Axis)* Selects the X axis for continuous motion when depressed.

+Y, -Y *(Axis)* Selects the Y axis for continuous motion when depressed.

+Z, -Z *(Axis)* Selects the Z axis for continuous motion when depressed.

+A, -A *(Axis)* Selects the A axis. This key selects the B axis when used with the **SHIFT** key if the machine is configured with a fifth-axis option.

Jog Lock – When this is pressed prior to one of the jog keys given above, the axis moves in a continuous motion without the need to hold down the axis key. Depressing this key again stops jogging motion. Feed rate is determined by the selection in **HAND JOG** mode keys.

3 – Override Keys

The overrides are at the lower right of the control panel. They give the user the ability to override the speed of rapid traverse motion, as well as programmed feeds and spindle speeds.

-10 FEED RATE	Decreases current feed rate in increments of 10 percent.
100% FEED RATE	Resets the control feed rate to the programmed feed rate.
+10 FEED RATE	Increases current feed rate in increments of 10 percent.
HANDLE CONTROL FEED RATE	Hand wheel will control feed rate at 1% increments.
-10 SPINDLE	Decreases current spindle speed in increments of 10 percent.
100% SPINDLE	Sets the control spindle speed at the programmed spindle speed
+10 SPINDLE	Increases current spindle speed in increments of 10 percent.
HANDLE CONTROL FEED	Hand wheel will control feed rate at 1% increments.
CW	Starts the spindle in the clockwise direction.
STOP	Stops the spindle.
CCW	Starts the spindle in the counterclockwise direction.
5% RAPID	Limits rapid to 5 percent of maximum.
25% RAPID	Limits rapid to 25 percent of maximum.
50% RAPID	Limits rapid to 50 percent of maximum.
100% RAPID	Allows rapid traverse to feed at its maximum.

Override Usage

Feed rates may be varied from 0% to 999%. Feed rate override is ineffective during G74 and G84 tapping cycles. Spindle speeds may be varied from 0% to 999%. Depressing Handle Control Feed rate or Handle Control Spindle keys, the jog handle movement varies by +/-1% increments.

Setting 10 will limit rapid movement to 50%.

Settings 19, 20, 21 make it possible to disable override keys.

Coolant may be over rode by depressing **COOLNT** button.

Feed Hold - Stops rapid and feed moves. **Cycle Start** button must be depressed to resume machine feeds. Similar situation applies when Door Hold appears. Door must be closed and **Cycle Start** pressed to continue running program.

Overrides may be reset to defaults with a M06, M30 or pressing **RESET** by changing Settings 83, 87 and 88 respectively.

5 – Cursor Keys
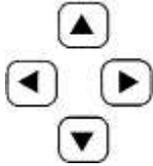
Cursor Keys The cursor keys are in the center of the control panel. They give the user the ability to move to and through various screens and fields in the control. They are used extensively for editing and searching CNC programs. They may be arrows or commands.

Up/Down	Moves up/down one item, block or field.
Page Up/Down	Used to change displays or move up/down one page when viewing a program.
HOME	Will move the cursor to the top-most item on the screen; in editing, this is the top left block of the program.
END	Will take you to the bottom-most item of the screen. In editing, this is the last block of the program.

6 and 7 – Alpha Keys and Numeric Keys

The **Alpha Keys** allow the user to enter the 26 letters of the alphabet along with some special characters. Depressing any Alphabet Key automatically puts that character in the Input Section of the control (lower left-hand corner).

SHIFT key provides access to the yellow characters shown in the upper left corner of some of the alphanumeric buttons on the keyboard. Pressing **SHIFT** and then the desired white character key will enter that character into the input buffer.

[;]	Key enters the end-of-block semicolon character and signifies the end of a programming block. It also moves the cursor to the next line.
()	Parentheses are used to separate CNC program commands from user comments. They must always be entered as a pair. Example: (T1 ½" End Mill) NOTE: Also any time an invalid line of code is received through the RS-232 port, it is added to the program between parentheses.
(–) and **(.)**	These keys are used to define negative numbers and give decimal position.
+ = # * [ ]	These symbols are accessed by first pressing the **SHIFT** key and then the key with the desired symbol. They are used in macro expressions (Haas option) and in parenthetical comments within the program.
, ? % $! & @ :	These are additional symbols, accessed by pressing the **SHIFT** key, that can be used in parenthetical comments.

The **Numeric Keys** allow the user to enter numbers and a few special characters into the control. Depressing any number key automatically puts it into the Input Section of the Control.

Cancel

The **Cancel** key will delete the last character put into the Input Section of the control display.

Space

Is used to format comments placed into the Input Section of the control display.

Write/
Enter

General purpose "Enter" key. It inserts code from the input section into a program when the program display is in EDIT mode. With offsets pages active, pressing the **WRITE/ENTER** key adds a number in the Input Section to the highlighted cell. Pressing the **F1** key will input the number into the cell.

-

The (Minus Sign) is used to enter negative numbers.

.

The (Decimal Point) is used to note decimal places.

8 – Mode Keys

Mode keys set the operational state of the machine tool. Once a mode is set the keys to the right may be used. The current operation mode of the machine is displayed at the top thin pane of the CRT.

EDIT

The edit mode is used to make changes in the program stored in the active memory.
In the edit mode you are able to use the edit keys in the **same row** as the **EDIT** key.
INSERT: Enters commands keyed into the input panel in lower left pane of CRT after the cursor highlighted word in a program.
ALTER: Highlighted words are replaced by text input into the input panel.
DELETE: Highlighted words are deleted from a program.
UNDO: Will undo up to the last 9 separate edit changes**.**

Above is the screen in the edit mode. To edit a word highlight it and then press DELETE, INSERT, ALTER or UNDO keys to change the program. Note on the bottom of the screen the program is not saved until the [MEMORY] key is pressed. Text may be selected to put in the Clipboard by using the [F2] key. [F4] allows you to paste code from the Clipboard to your program.

F1 KEY   While in the edit mode pressing **F1** will bring up different menus. These may be accessed by using the right and left cursor keys. High lighting File, Edit, Search or Modify the different options are shown in drop downs. The different options are shown below.

In the File drop down a **New** program may be created by high lighting New and pressing enter. Any editing changes can be saved by lighting **Save** and pressing enter. If a program needs to be duplicated the **Save As** function can be used. The Discard Changes function will undo all the current edits.

41

F2 KEY

Several lines of code can be selected by using the F2 key. First highlight the first line of code and press F2. Then cursor down to the last line of code that needs to selected and press Enter. This will select several lines of code and they will be highlighted in a light blue. This selected text can be either cut or copied to the Clipboard. From the Clipboard it may be pasted anywhere in the current program of another program using the Paste from Clipboard option.

Highlighting the Find and Replace key and pressing enter will give a pop up below. Here you can find and replace text thru out the whole program.
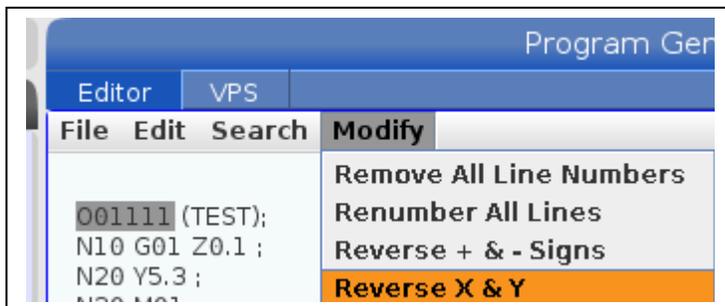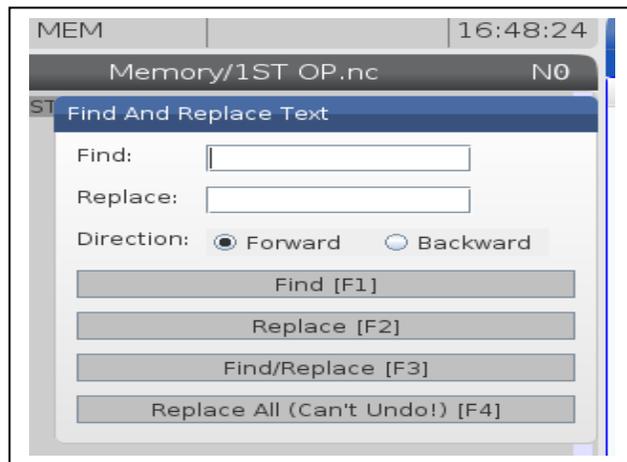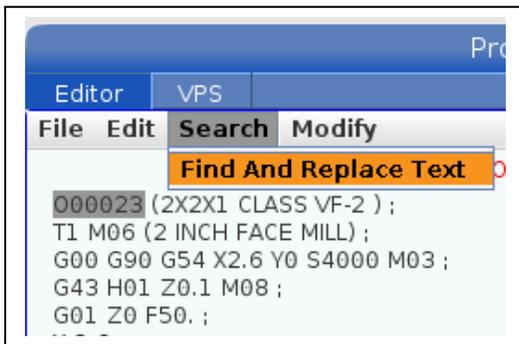
The Modify tab gives the option of putting in block numbers in a program. Also +/- signs and the X and Y may be reversed.

MEM   The memory mode is the mode used when running the machine and making a part. The active program is shown in the Program Display Pane. Keys in the memory mode line reflect different ways of running a part in memory. When the keys to the right are depressed they will show up highlighted in black on the bottom right of the CRT.

SINGLE BLOCK   When depressed **SINGLE BLOCK mode becomes active and it is indicated** in black and will appear on the   bottom of the display. When the machine is in SINGLE BLOCK mode only one block of the program is executed every time the cycle start button is depressed. Used when first test running a program or temporarily stopping a program when it is running.

GRAPHICS   On older controls the graphics were opened by the Settings/Graphics display key. In NGA controls it is found on the Memory line beneath the [Alter] key.   On the NGA controls a separate Graphics key is designated and replaces the DRY RUN key. NGA controls no longer have the Dry Run feature. Graphics of the active program may be run in either Memory mode or Edit mode. After the [CYCLE START] is depressed the graphics will be run as shown below.
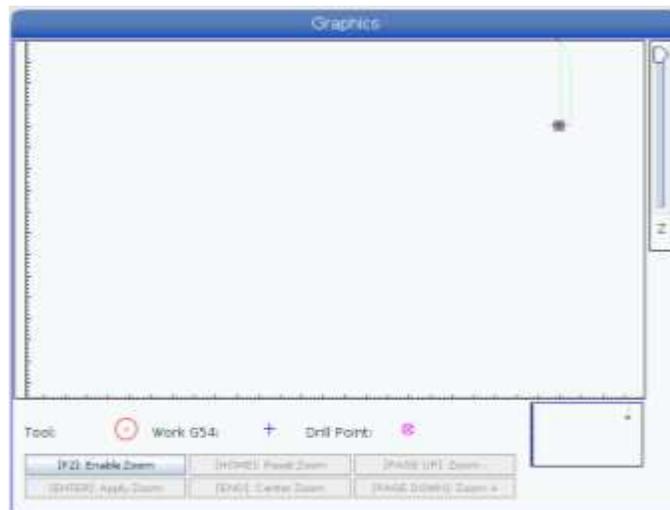
Note the graphics shown is the XY Plane, as if one was looking at the whole table from above.   Zoom into the part by pressing [F2]
Then [END] to center and [PAGE DOWN] several times until the box is just barely outside of the part as below.

Then press [ENTER] to apply zoom.

Run again in graphics by pressing [CYCLE START] to get graphics as shown below.

Rapid moves are indicated by the green lines. The grey lines represent tool paths with respect to the center of each tool. The circles with X's inside represent drill holes.
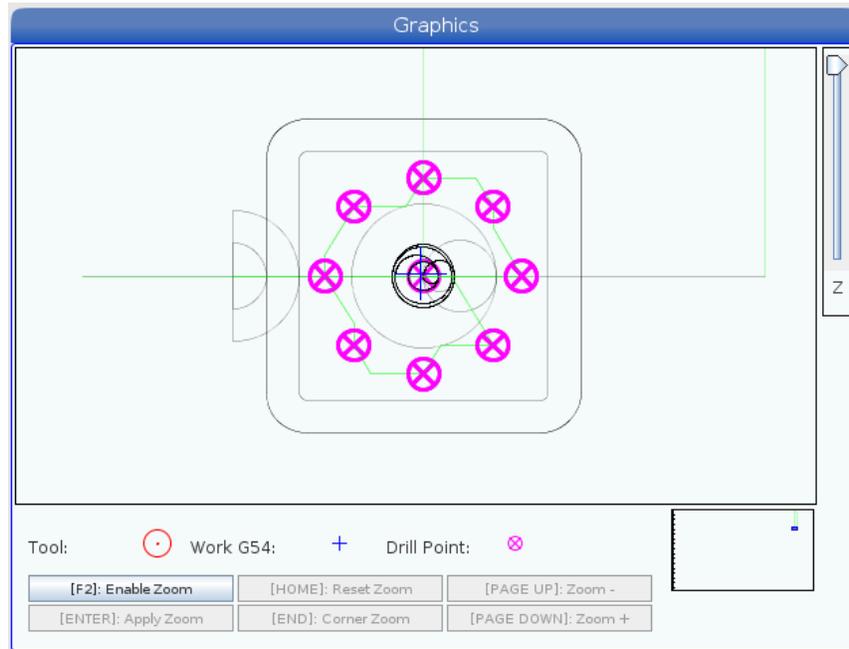
In the graphics mode the tool paths run instantaneously. To see the individual tool paths the program must be run in [SINGLE BLOCK] or [OPTION STOP]. The program must be run step by step. This is done by pressing the [CYCLE START] key repeatedly.

Settings #4, 5, and 253 give options on what you are seeing. Below setting #253 (Default Graphics Tool Width) is set to off. The tool path is determined by the Tool Offset Diameter Geometry. This represents what material is being cut off by the tool. In the Default mode the tool path is indicated by a Line
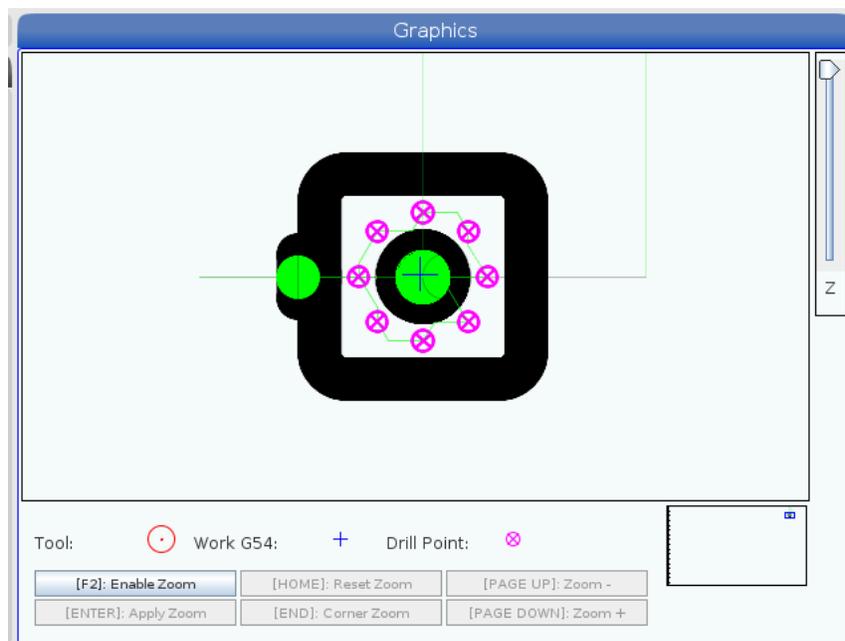
**OPTION
STOP**   When the **OPTION STOP** key is depressed the program will stop at any M01 in

44

the program. Normally M01's are placed after each tool has completed its operation in a program. When a job is being set up the operator may put machine in op stop mode to check dimensions after every tool has completed cutting. After that the optional stop is turned off.

BLOCK DELETE   When this button is depressed any block with a slash (/) in it is ignored of skipped.

MDI DNC   (MANUAL DATA INPUT mode) – Usually short programs are written in MDI but are not put into memory. DNC mode allows large programs to be drip fed from a computer into the control.

COOLNT   Turns coolant on and off manually

HANDLE   Allows the handle jog to cursor thru a program.

ATC FWD   Rotates turret to next tool and performs tool change - also used to call up specific tools or pots. Enter tool number (T1) or (P1) and press **ATC FWD**.

ATC REV   Rotates turret to previous tool and performs tool change - also used to call up specific tools or pots. Enter tool number (T1) and press **ATC REV**.

HAND JOG   Puts machine in jog mode for set ups. A particular axis, X, Y, or Z is selected and the handle Jog is turned clockwise for positive and counter clockwise for negative to move the tool. Top values (.0001, .001, .01, .1) represent distance traveled per click of jog handle. Bottom values (.1, 1., 10., 100) represent feed in inches/minute when jogging axis using jog buttons.

ZERO RET   On pressing position display becomes highlighted in Zero Return mode.

ALL   Returns all axes to machine home similar in similar fashion as a Power Up/Restart. First the Z axis then the X and Y axis.

ORIGIN   Sets selected displays to zero or other functions.

SINGL   Returns a single axis to machine home. Select desired axis (X, Y, or Z) then press **Singl** axis button.

Home/G28   Rapid motion to machine home; will make a rapid move in all axes at once - may also be used for a rapid home in one-axis. Press axis to home then **G28.**
Caution must be used that fixtures or parts are out of the way before initiating this rapid move to home.

LIST PROG   Brings up lists of programs in a tab format. Cursor up will return you to tabs at

top. Cursor to left or right to highlight which directory is desired. Below Memory is highlighted and gives a list of programs that are stored in the memory. Cursor UP (∧) or DOWN (∨) to the program desired to run in the machine.

Highlight the desired programs to be moved to the active memory and press **[SELECT PROGRAM].** This will call up that program and make it the **active program**. This program is the one that will show up in the program display pane. In the memory directory it has an asterisk behind it denoting that as the active program. If the cycle start button is pushed this is the program the machine will run.

List Prog

Memory	User Data	USB

Search (TEXT) [F1], or [F1] to clear.

Current Directory: Memory/

	O #	Comment	File Name	Size	Last Modified				
☐			09000	<DIR>	2016/08/31 03:57	>		New	[INSERT]
☐		(PDF-1.7.PDF)	AE_flyer.pdf	29 B	2016/12/12 10:11			Load	[PROG]
☐	00012	(CLASS)	O00012.nc	1015 B	2016/09/12 15:48				
☐	00023	(2X2X1 CLASS …	O00023.nc	3 KB	2016/09/30 21:03			Edit	[ALTER]
☐	00024	(2X2X1 CLASS …	O00024.nc	7 KB	2016/09/12 15:48				
☐	00450	(INGERSOL 41…	O00450.nc	15 KB	2016/09/12 15:48			Mark	[ENTER]
☐	00666	(KEY WRITER F…	KEY_WRITE…	309 B	2016/12/12 09:58				
☐	01111	(TEST)	1ST OP.nc	51 B	2017/01/05 17:18	E		Copy	[F2]
☐	04040	(ATM)	O04040.nc	170 B	2016/11/04 15:49				
☐	09019	(G128 CTF V2.7)	G128_CTF…	5 KB	2016/12/21 17:07			File	[F3]
☐	09999	(G128 CTF V2….	O09999.nc	5 KB	2016/12/13 14:51				
☐	12345	(TEST G128)	G128_test	83 B	2016/12/13 16:23			System	[F4]
☐	56565	(XXX)	XYXY.nc	13 B	2017/01/03 13:53				

File Name: O00023.nc

File comment: (2X2X1 CLASS VF-2)

Folder Has: 13 Items   Disk Space: 906 MB Free (88%)   Selected Items: 0

F2 will copy selected program or programs to be moved. A pop up menu will ask where you want the programs to be copied. First the programs to be copied are marked by highlighting the program and

pressing the enter key. Below two programs have been selected and check marks indicate the programs to be moved. USB0 has been highlighted as the recipient location. Pressing enter copies files O0012 and O0023 to the USB device. The reverse may be done. Files created on a computer and put on a USB device can be copied to the memory of the machine to be run. If the machine is connected by Ethernet to a computer network a Net Share column appears and programs can be loaded and downloaded directly from a computer network.
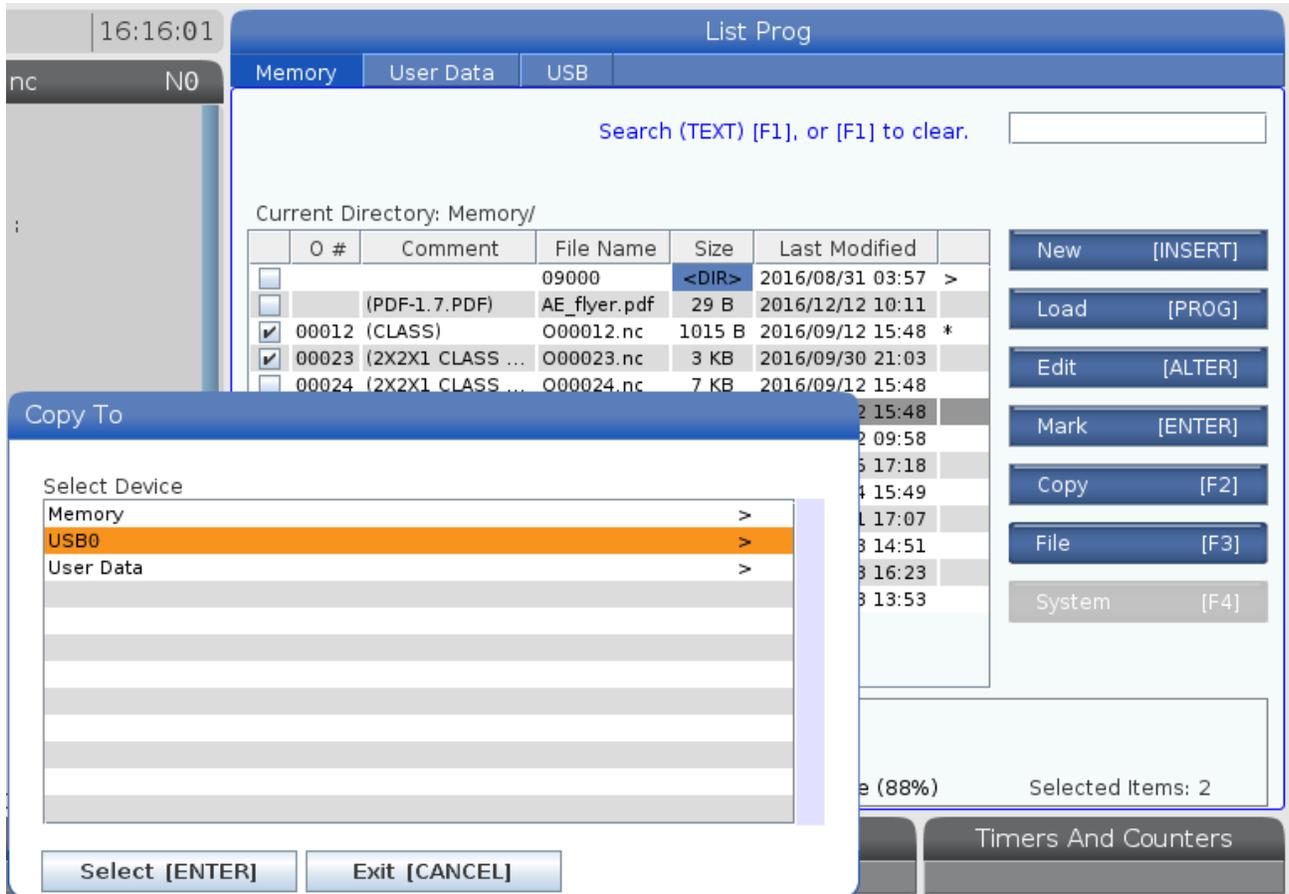
F3 key gives various options such as renaming, deleting, duplication, and others in the List Program page.

ATC (Automatic Tool Change)
SIDE MOUNT TOOL CHANGER

The side mount tool changer allows you to select a tool and put it in the waiting position while machining a part. During a tool change the tool in the spindle is put in the same pot in the carousel that the current tool came from. This allows a speedier process and also keeps the tools out of the working envelope. With this system the tools are in random access mode, in other words tool 1 isn't assigned to the pot 1 location. The machine however knows which pocket each tool is in. In the tool offsets display cursor to the right several times to the tool pocket column. It will show which pot the tool is located in. Press **CURRENT COMMANDS** and cursor over to Tool Table. In this table tool pockets are labeled with the tools that are located in them.
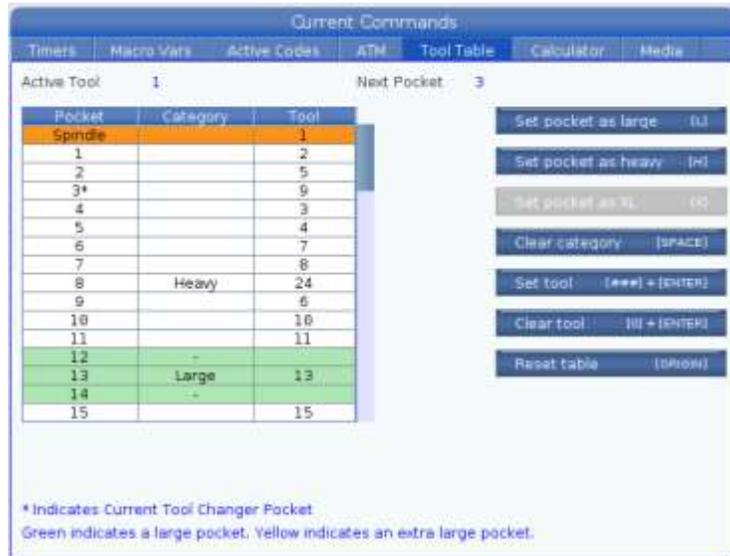
Note that in the Spindle (under the Pocket column) is Tool 1.   Note pocket 3 has an asterisk in it. This designates the pocket in the carousal that is in the tool change position.   If just a tool change is called Tool 9 in the carousal will be exchanged for Tool 1 which is in the spindle.   Note in the category column that Tool 24 is designated as a Heavy tool. When a tool is designated a heavy tool is indexes at 25%. Tool 24 is the spindle probe. Because of the violent nature of the tool change process in VF-2 and above Cat 40 machines the calibration of the spindle probe may be compromised. If a Cat 40 tool is greater than 4 lbs. it should also designated as a heavy tool.   This is because it may not index properly and fall out of the spindle.

Tool 13 is designated a Large Tool. This is any tool that is 3" or larger. Because of clearance issues when the tool is in the carousal it will not let any tools be put on each side of it. Note the – in the category column.   Also when tool 13 is placed back in the carousal it will always be put back in pocket position 13.

Changing Tools – Tool changes may be manually operated by using the tool release button on the spindle housing and using the **ATC FWD** and **ATC REV** buttons. Tools may also be changed in MDI with (Tnn M6).   After touching off a tool and using **Tool Offset Meas, Next Tool** will initiate a tool change to the next tool. Tools will automatically go to the tool change position in Z.   Tools should always be loaded through the spindle. ***Never load directly into the tool changer.***

Setting Tool Length & Work Zero Offsets

To set Tool Length Offsets:

1. Press **HANDLE JOG**.
2. Press **OFSET** and cursor to the tool number you want to set in the LENGTH display.
3. In Handle Jog mode, use the **.1**, **.01**, **.001** and **.0001** keys to jog the tip of the tool (the Z-axis) to the part zero surface. Touch-off the tool tip to the part surface using a feeler gauge.
4. Press **TOOL OFSET MESUR**. This will take the number in the machine Z position readout (listed at bottom of screen) and load it into the cursor-selected offset register.
5. Press **NEXT TOOL**. This will automatically change to the next tool and the next offset number, and the Handle Jog function switches back to the Z-axis .01 increment setting.
6. **TOOL OFSET MESUR** the next tool in the same way.
7. Repeat steps 3 through 5 for all tools needed for part setup.

To set Work Zero Offsets:

1. **HANDLE JOG** the machine axes to the work zero point on your part.
2. Press **OFSET** and **PAGE UP**, and cursor to the correct **WORK ZERO OFFSET** number in the X-axis column. Repeat pressing the **F4** key toggles between Work Offset page and Tool Length Offset page.
3. Press **PART ZERO SET** to enter the X offset position. The cursor will then automatically move over to the Y-axis column.
4. Position the Y-axis and then press **PART ZERO SET** to enter the Y offset position.
5. The Z-axis is normally entered separately if any adjustment is needed. This setting is usually zero but may need to be adjusted.
6. When the A axis is being used, find the A axis zero position and press **PART ZERO SET**.

SET UP PROCEDURE

WORK OFFSETS (X AND Y PART ZEROS)

1. TURN POWER ON
2. RESET ALARMS
3. PRESS POWER UP RESTART
4. PRESS SETTINGS DISPLAY (SETTING 51 DOOR HOLD OVERRIDE - ON)
5. PRESS M.D.I. MODE (ORIENT SPINDLE/ S1000, ENTER, CYCLE START)
6. PRESS HANDLE JOG MODE
7. PRESS POSITION DISPLAY (PAGE DOWN TO OPERATOR POSITION DISPLAY)
8. LOCATE X PART ZERO (REMEMBER TO SHIFT HALF THE DIA. WHEN USING AND EDGE FINDER)
9. PRESS OFFSET DISPLAY (PAGE UP TO WORK OFFSETS, CURSOR TO DESIRED OFFSET IN X AXIS COLUMN)
10. PRESS PART ZERO SET
11. REPEAT STEPS 7 THROUGH 10 FOR Y AXIS ZERO

TOOL LENGTH OFFSETS

1. PRESS M.D.I. MODE
2. TYPE IN DESIRED TOOL NUMBER TO BE PLACED IN THE SPINDLE AND PRESS A.T.C. BUTTON
3. PLACE APPROPRIATE TOOL IN SPINDLE
4. PRESS HANDLE JOG MODE
5. PRESS OFFSET DISPLAY (PAGE UP TO LENGTH OFFSETS)
6. HANDLE Z AXIS DOWN TO Z PART ZERO
7. PRESS TOOL OFFSET MEASURE (ENTER IN THE RADIUS AMOUNT OF CUTTER IF APPLICABLE)
8. PRESS NEXT TOOL
9. REPEAT STEPS 6 THROUGH 8 FOR REMAINING TOOLS

PROGRAM PROOFING AND RUNNING IN MEMORY

1. PRESS MEM MODE
2. PRESS SETTING/GRAPH TWICE
3. RUN PROGRAM IN GRAPHICS
4. IF RUNNING IN GRAPHICS IS SUCCESSFUL PRESS SETTING/GRAPH TO GET OUT
5. TURN RAPID OVERIDE TO 25%
6. PRESS SINGLE BLK
7. PAGE DOWN IN CURRENT COMMANDS UNTIL DISTANCE TO GO, MACH COORD,G54 ETC SCREEN APPEARS
8. RUN PROGRAM UP TO 1" IN FRONT OF PART AND PUSH FEED HOLD
9. IF DISTANCE TO GO LOOKS CLOSE TO 1" CONTINUE RAPID BLOCK TO TOP FACE OF PART
10. WHEN PROGRAM GOES TO FEED OR CANNED CYCLE, TAKE OFF SINGLE BLOCK
11. RUN PROGRAM TO NEXT TOOL, COMPLETE 6 TO 10 ABOVE.
12. AFTER RUNNING THRU WHOLE PROGRAM SUCCESSFULLY TURN RAPID TO 100%

HAAS MILL CONTROL TIPS

GENERAL TIPS

> **Searching for a Program Command** - Searching for a specific command in a program can be done in either MEM or EDIT mode. Enter the address letter code (A, B, C, etc.) or address letter code with the value (A1.23), and press the up or down cursor arrow. If you enter just the address code and no value, the search will stop at the next use of that letter, regardless of the value.

> **Spindle Command** - You can stop or start the spindle with **CW** or **CCW** any time you are at a Single Block stop or a Feed Hold. When you restart the program with **CYCLE START**, the spindle will be turned back on to the previously defined speed.

> **Coolant Pump** - The coolant pump can be turned on or off manually while a program is running, by pressing the **COOLNT** button. This will override what the program is doing until another M08 or M09 coolant command is executed. This also applies to the chip conveyor.

CONTROL TIPS

> **Coolant Spigot Position** - The coolant spigot position can be changed manually when a program is running, with the **CLNT UP** or **CLNT DOWN** key. This will override what the program commands until another spigot position is commanded with an Hnn or M08 command.

> **Optional Stop** - Takes effect on the line after the highlighted line when pressed.

> **A Block Delete** - Takes effect four lines after that key is pressed when cutter compensation is in use, or two lines later when cutter compensation is not in use.

> **Block Look-Ahead** - This control actually does look ahead for block interpretation, up to 20 blocks. This is not needed for high-speed operation. It is instead used to ensure that DNC program input is never starved, and to allow Cutter Compensation to have non-XY moves inserted while Cutter Compensation is On.

> **Memory Lock Key Switch** - This is a customer machine option that prevents the operator from editing or deleting programs, and from altering settings when in the locked position. Since the Key switch locks out the Settings, it also allows you to lock out other areas within the settings: Setting 7 locks parameters: Parameter 57, 209, and 278 lock other features. Setting 8 locks all programs. Setting 23 locks 9xxx programs. Setting 119 locks offsets. Setting 120 locks macro variables.

> **Chip Conveyor** - The chip conveyor can be turned on or off when a program is running, either manually using the control keys or in the program using M codes. The M code equivalent to **CHIP FWD** is M31, **CHIP REV** is M32, and **CHIP STOP** is M33. You can set the Conveyor Cycle time (in minutes) with Setting 114, and the Conveyor On-Time (in minutes) with Setting 115.

➢ ***Transferring an MDI Program*** - You can transfer and save a program in MDI to your list of programs. When in the MDI display, *make sure that the cursor is at the beginning* of the MDI program. Enter a program number (Onnnnn) that's not being used. Then press **ALTER** and this will transfer the MDI data into your list of programs under that program number.

➢ ***To Rapid an Axis Home*** - You can rapid *all* axes to machine zero by pressing the **HOME G28** key. You can also send just one axis (X, Y, Z, A or B) to machine zero in rapid motion. Enter the letter X, Y, Z, A or B, then press **HOME G28** and that axis alone will rapid home. ***CAUTION***! There is no warning to alert you of any possible collision! For example, if the Z axis is down near the part or fixture on the table, and then the X or Y axis is sent home using HOME G28, a crash may result. Care must be exercised.
(Any Mill Control Ver. 9.49 and above; any Lathe Control Ver. 2.24 and above.)

➢ ***Rotate Tool Carousel without Changing Tools (M39)*** - You can rotate the tool carousel without changing tools by using M39 and Tnn. This can be used to recover from some unusual conditions. However, it will also tell the control you have a different tool in the spindle, which may not be the case.
(Any Mill Control Ver. 2.3 and above.)

POSIT (Position)

➢ ***Quick Zero on DIST-TO-GO Display*** - To clear out and get a quick zero position display, for a distance reference move, use the **DIST-TO-GO** position display. When you are in the POSIT display and in HANDLE JOG mode, press any other operation mode (EDIT, MEM, MDI, etc.) and then go back to HANDLE JOG. This will zero out all axes on the DIST-TO-GO display and begin showing the distance moved.

➢ ***To Origin the POS-OPER Display*** - This display is used for reference only. Each axis can be zeroed out independently, to then show its position relative to where you selected to zero that axis. To zero out a specific axis, **PAGE UP** or P**AGE DOWN** in the POSIT display to the POS-OPER large-digit display page. When you Handle Jog the X, Y or Z axis and then press **ORIGIN**, the axis that is selected will be zeroed. Or, you can press an **X**, **Y** or **Z** letter key and then **ORIGIN** to zero that axis display. You can also press the **X**, **Y** or **Z** key and enter a number (X2.125), then press **ORIGIN** to enter the number in that axis display.

➢ ***Jog Keys***. The **JOG** keys (+X, -X, +Y, -Y, +Z, -Z, +A, -A, +B, -B) use the jog speeds of 100., 10., 1. and .1 inches per minute listed next to the **HANDLE JOG** key (jogging with the handwheel uses the .1, .01, .001 and .0001 inch increments). You can also adjust feed rate using the **FEED RATE OVERRIDE** buttons, which allow you to increase or decrease feed rate in 10% increments, up to 200% or using the **HANDLE CONTROL FEED RATE** or **HANDLE CONTROL SPINDLE** keys to adjust the programmed feed or speed 1% up or down with every increment of the Handle.

➢ ***Jog Keys*** - You can also select an axis for jogging by entering the axis letter on the input line and then pressing the **HANDLE JOG** button. This works for the X, Y, Z, and A axes as well as the B, C, U, and V auxiliary axes.

PROGRAMMING

➢ ***Program Format at the Beginning and End*** - Programs written on a PC and sent to the control from a floppy disk or through the RS-232 port must start and end with a % sign, on a line by itself. The second line in a program received via floppy or RS-232 (which will be the first line the operator sees) must be **O**nnnnn, a six-character program number that starts with the letter **O** followed by five digits. When you create a program on the Haas control the percent (%) signs will be entered automatically, though you won't see them displayed.

➢ ***Program Format with M06*** - It is not necessary to turn off the coolant (M09), stop the spindle (M05), or move the Z axis home (G28) prior to a tool change. The control will do these tasks for you during a tool change M06 command. However, you may decide to program these commands anyway for convenience and timely execution of a tool change sequence. If you are using Single Block to step through a program you will be able to see the commands when you stop on that line.

➢ ***Tapping with G84 or G74*** - When tapping, you don't need to start the spindle with an M03 or M04 command. The control starts the spindle for you automatically with each G84 or G74 cycle, and it will in fact be faster if you don't turn on the spindle with an M03 or M04. The control stops the spindle and turns it back on again in the G84 or G74 tapping cycle to get the feed and speed in sync. The operator just needs to define the spindle speed.

➢ ***Quick Reverse Out of a G84 Rigid Tapping Cycle*** - This feature for rigid tapping has the spindle back out faster than it went into a tapped hole. The way to specify this is with a J code on the G84 command line. J2 retracts twice as fast as the entry motion; J3 retracts three times as fast, and so on, up to J9. A J code of zero will be ignored. If a J code less than 0 or greater than 9 is specified, Alarm 306 - "INVALID I, J, K or Q" - is generated. The J code is not modal and must be specified in each block where this effect is wanted. The J value should not contain a decimal point. (Any Mill Control Ver. 10.13 and above.)

➢ ***M19 (Orient Spindle) with a P Value*** - This feature works on any vector drive mill. Previously, the M19 command would simply orient the spindle to only one position ó that suitable for a tool change. Now, a P value can be added that will cause the spindle to be oriented to a particular position (in degrees). If a whole number is used for the P value, no decimal point is needed. For example, M19 P270 will orient the spindle to 270 degrees. Note that P270.001 (or any other fraction) will be truncated to P270, and P365 will be treated as P5. (Any Mill Control Ver. 9.49 and above. Any Lathe Control Ver. 2.21 and above.)

➢ ***M19 (Orient Spindle) with a Fractional R Value*** - This feature works on any vector drive mill. An M19 R123.4567 command will position the spindle to the angle specified by the R fractional value; up to 4 decimal places will be recognized. This R command now *needs a decimal point*: if you program M19 R60, the spindle will orient to 0.060 of a degree. Previously, R commands were not used for this purpose and only integer P values could be used. (Any Mill Control Ver. 9.49 and above; any Lathe Control Ver. 2.29 and above.)

➢ ***Duplicating a Program in LIST PROG*** - In the LIST PROG mode, you can duplicate an existing program by cursor-selecting the program number you wish to duplicate, typing in a new program number (**O**nnnnn), and then pressing **F1**. You can also go to the Advanced Editor menu to duplicate a program, using the PROGRAM menu and the SAVE AS PROGRAM item.

Formulas

Tapping

STANDARD thread formula:

Feed rate in inches per minute = Revolutions per minute (RPM) divided by threads per inch (TPI)

$$F = RPM/TPI$$

METRIC thread formula:

Feed rate in inches per minute = Pitch (P) multiplied by .03937 multiplied by RPM

$$F = (P \times .03937) \times RPM$$

Speeds and Feeds

S.F.M. (Surface Feet per Minute):

SFM = .262 multiplied by the cutter diameter multiplied by the RPM

$$SFM = .262 \times Cutter\ Diameter \times RPM$$

R.P.M. (Revolutions per Minute):

RPM = 3.82 multiplied by the recommended SFM divided by the cutter diameter

$$RPM = 3.82 \times SFM / Cutter\ Diameter$$

Feed (Inch per Minute) for **<u>twist drills</u>**:

$$F(inch/min) = F(inch\ /rev) \times RPM$$

Feed (Inch per Minute) for **<u>end mills</u>**:

Feed rate in inches per minute = Feed per tooth (Inch/rev) multiplied by the number of cutter teeth (N) multiplied by the RPM =

$$F\ (inch/min) = (Feed/tooth \times N) \times RPM$$

CUBIC INCH PER MINUTE:

Cubic inch per minute = Effective diameter of cut multiplied by the depth of cut multiplied by the inch per minute feed rate.

$$CIPM = (E\ Diameter \times d) \times IPM$$

Notes

| |
| |
| |
| |
| |
| |
| |
| |
| |
| |
| |
| |
| |
| |
| |
| |
| |

Toolroom Mill Orientation and Walk Around:

The agenda for this section of the training manual is to familiarize everyone with the physical layout of the TM series machine, the functions of the mechanical features of the machine, and general maintenance of the TM series mill.
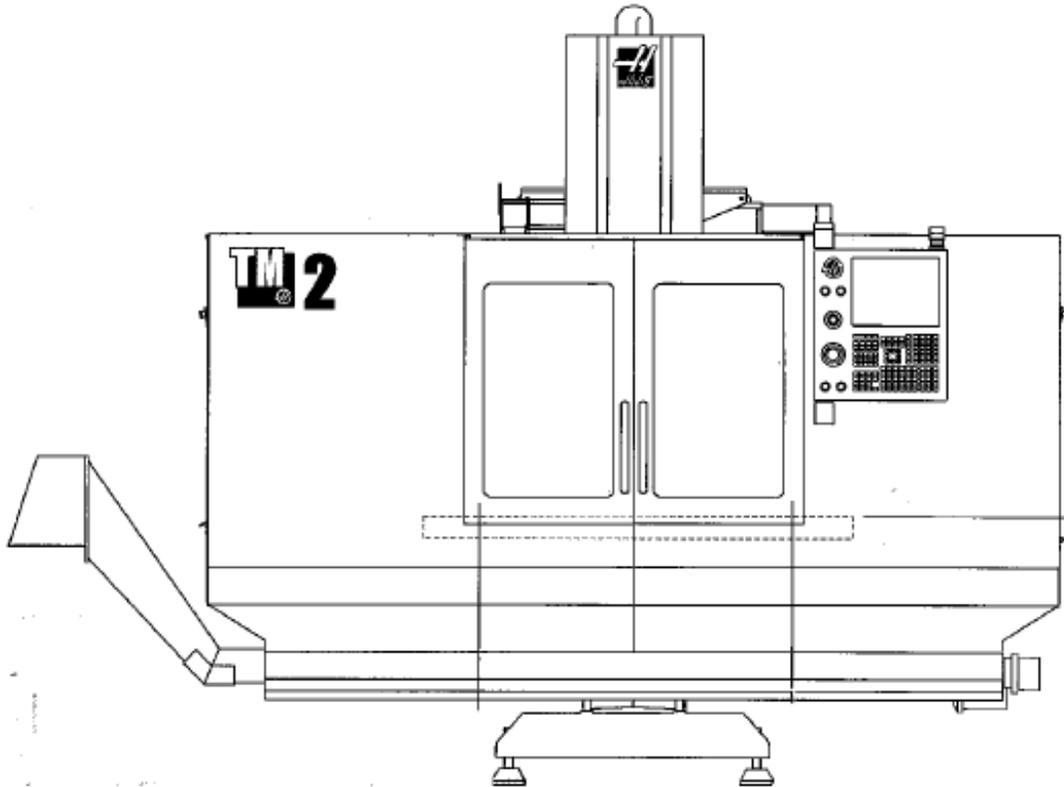
FRONT VIEW OF TM – SERIES MILL

Power-Up Procedure

The TM series of mills follows the same standard start-up procedures as with any other Haas   machine, and should be followed in the procedure demonstrated below:

1.a.i – Main Power

Locate the main power switch on the back of the machine's control cabinet, and switch it to the "ON" position.

1.a.ii – Power-Up Procedure

After switching the main power supply on, we need to "power-up" the Haas control. That procedure is as follows:

1. Visually verify that all of the machine's ways are clean and free of dirt, and any moving parts of the machine are clear of chips, coolant, etc. and verify that there is sufficient air pressure (80 psi) to the machine

2. Press the Green POWER ON button located on the upper left corner of the Haas control. The machine will take a couple of minutes to load its software.

3. The machine is now "Powered-On"

1.a.iii – Machine Restart / Zero Return Procedure

After the control is powered on, we still need to "reference" the machine. Upon start-up, the control does not know the location of it's axis X, Y, or Z, so the machine needs to "find" these axis thru a procedure known as a "Zero Return". This will travel the machine to its axis farthest most positive limits so that the table can be detected by the limit switches.

1. Locate the orange RESET key and push it to clear any alarms that are present, and to power on the servos of the machine.
2. Next to the RESET button is the POWER UP / RESTART button. Press it once and the machine will travel the Z axis first, then the X & Y axis to their ut-most farthest limits, which is all the way to the left in the X axis, Y axis all the way towards the operator, and all the way up for the Z axis / spindle. The machine will stop at machine home, as described in section-4.a.iii

TOOLROOM MILL SAFETY

Emergency Stop Switch

Located on the left side of the control panel, there is a large, round, red button that is circled with a yellow ring. This is the EMERGENCY STOP or "E-STOP" button. In the event of an un-controlled machine condition, a crash of the machine, or any situation that requires an immediate shut down of all axis functions, movement, spindle functions and power to the servos of the machine, the Emergency Stop button should be used. This button is used to halt everything on the machine immediately!

Maintenance of the TM Series Mill

Even though your TM Series machine has been built to the highest standards, using the highest available techniques and materials, general maintenance is required on both the machine and on the control to insure long life of its components.

Haas has gone thru extensive testing to verify the specifications on the required lubricants and filters that went into your new machine before they ever entered Haas's production, please use parts, filters, and lubricants that are as specified to insure longevity of the life of your or your boss's investment. This will maintain the highest levels of performance and accuracies for the longest possible time.

If there is any question on what exactly to use, please refer to the Operator's Manual for these specifications or contact the Productivity Service and Parts Department for the proper fluids and parts required for maintenance.

Productivity, Inc.
Service and Parts
763-476-8600

Grease Points

The TM series mills are equipped with grease fittings rather than an oil lube system to provide surface to surface lubrication for all contact points on the machine. The linear ways and the ballscrews equipped with grease fittings, and they should be lubricated weekly to insure consistent performance and accuracy of your machine.
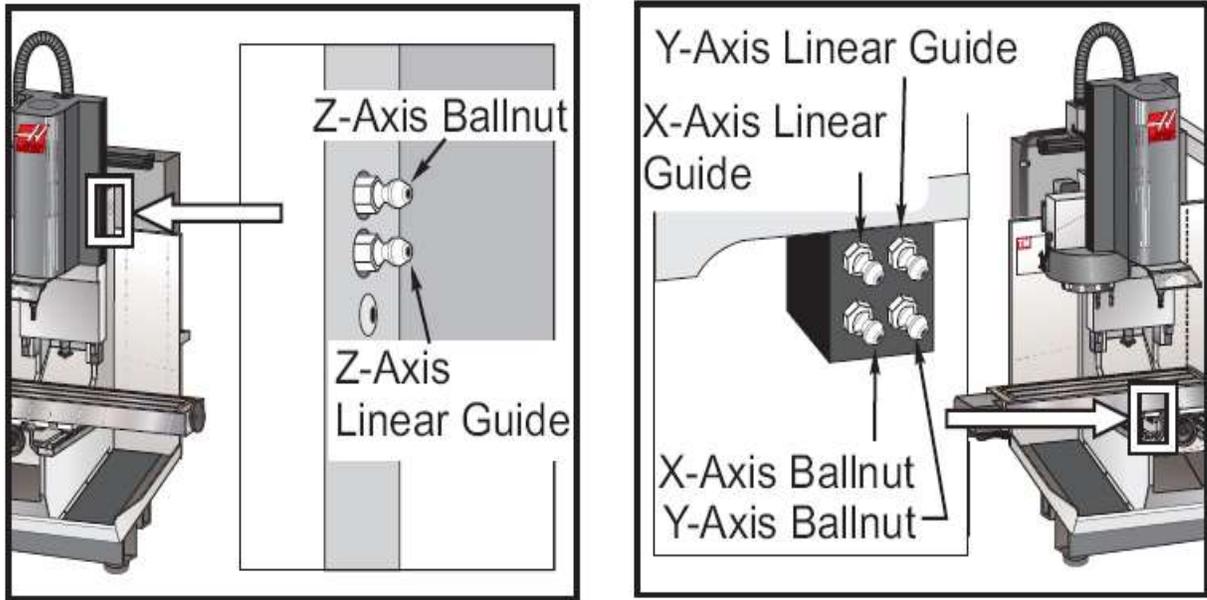
TM Series Lubrication Points

For the lubrication points shown above, a general purpose lithium grease is required with enough volume to push grease out from the front and back of the linear ways, and from around the ball screw bearing housing

CAT 40 Tooling

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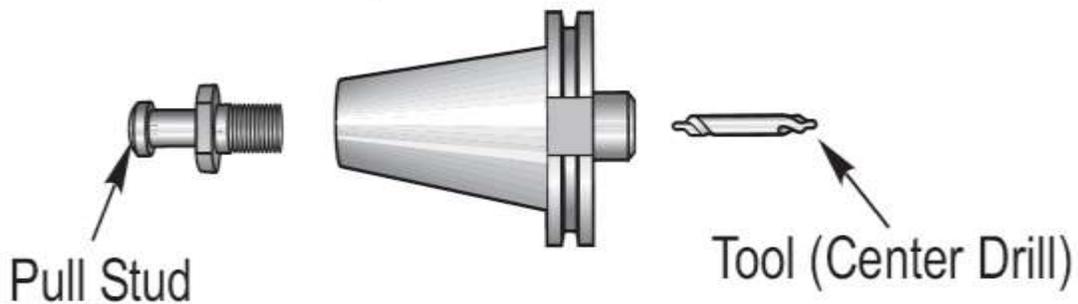
There is a wide variety of tooling available for CT-40, as it is the most popular tooling option for machining centers.

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
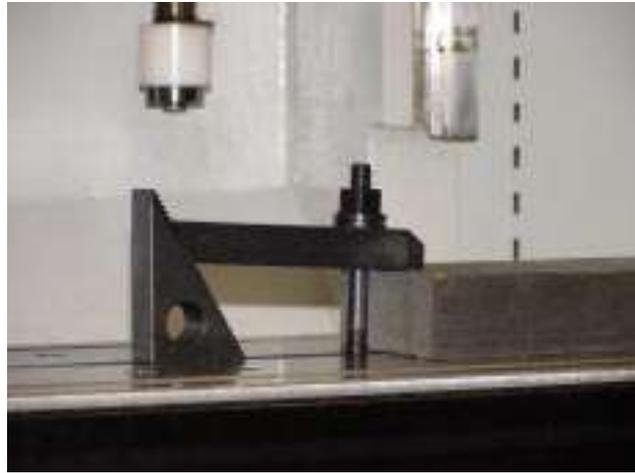
Toe Clamp

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

The following set up will use VPS to create G-code to machine a part to this print

Virtual Programming System

Machine must be in **EDIT mode** to access **VPS**. As shown below press EDIT key.

1st Press EDIT Key

Use the right arrow cursor key to highlight VPS tab if the Editor tab is highlighted.

Use the down arrow cursor key to highlight the VPS directory.
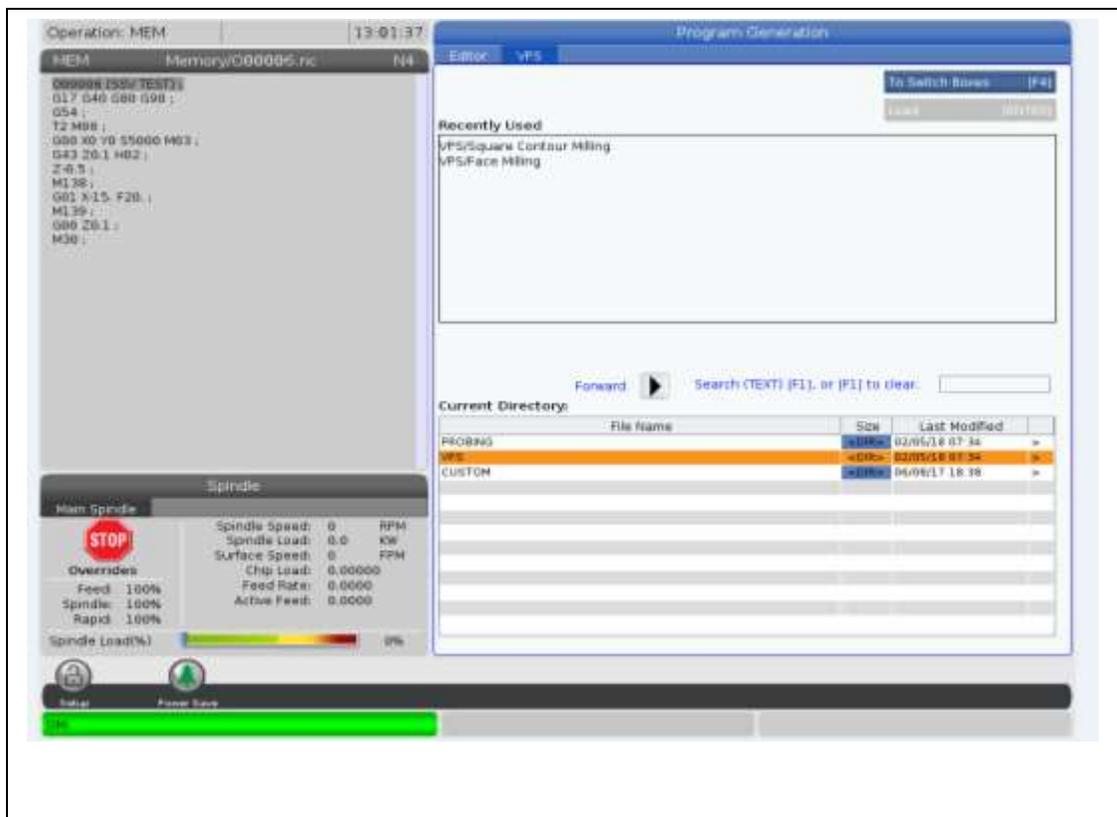
Face Milling

Then use the right arrow cursor key to open the VPS directory as shown below. Then use the down arrow cursor key to highlight Face Milling for the first operation. Then press enter to get open as in the bottom.

.
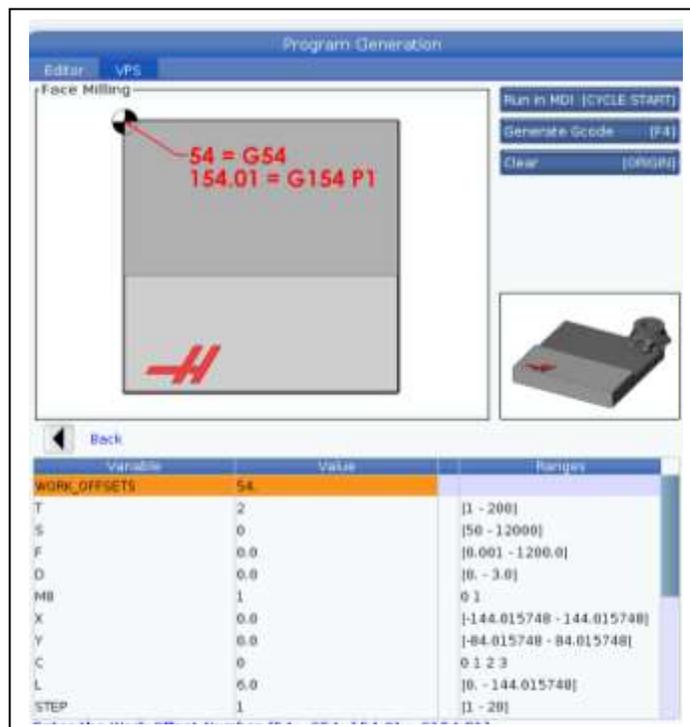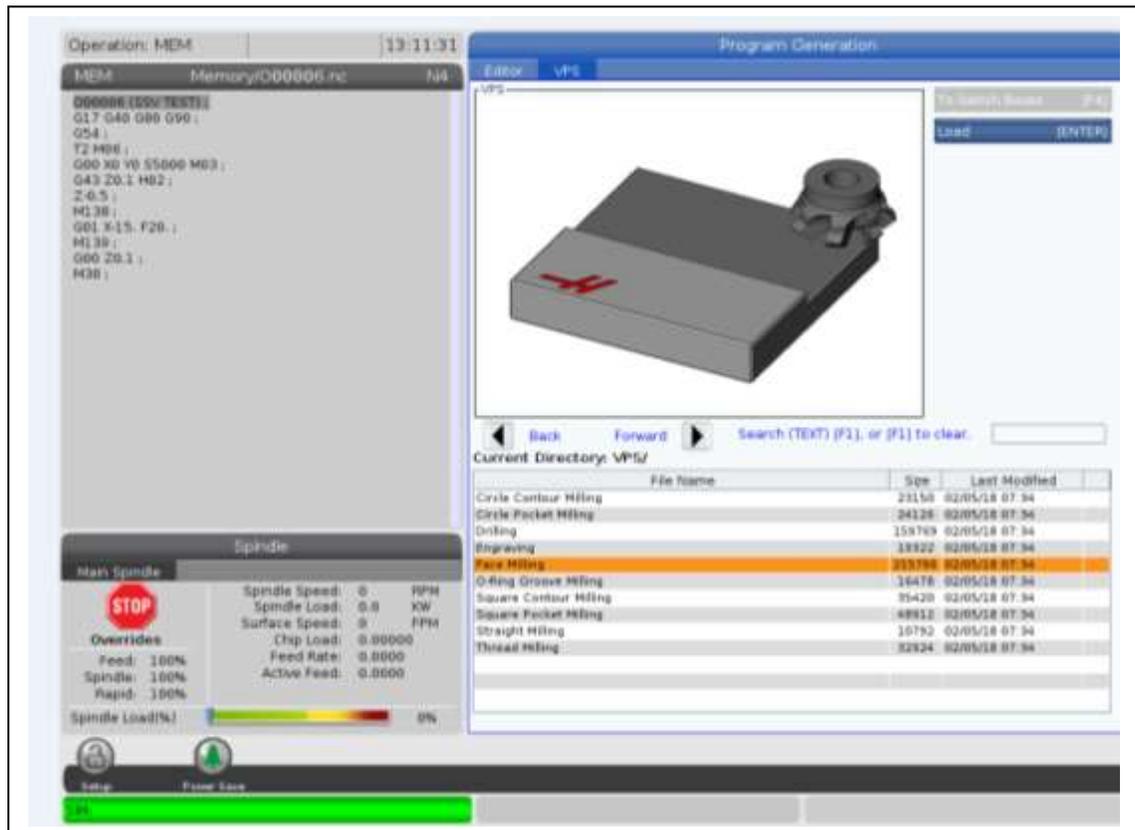
Key in the appropriate information such as work offset, tool, feed and speed, hitting enter. Pictures will pop up to give you information on what is needed to create the code. When you get to variable C it gives the different tool paths that may be created. When you pick the tool path desired then the X and Y values are the distances from your part zero to the corner one is starting from. For 0 the starting corner is the upper left hand corner.

If you were using the upper left hand corner of the part to set your work offset than X and Y are 0. If your part is 3" x 6" wide if you use method 1 than the X value will be 6" and the Y value 0. That is how far from home the start corner is in X and Y. Method 2 will be the lower right hand corner, method 3 the lower left hand corner. The L value is the length of the face and direction it is going from in X or Y. For 0 it will the 6" in X, 1 it will be 3" in Y. For 3 it will be 3" in Y.
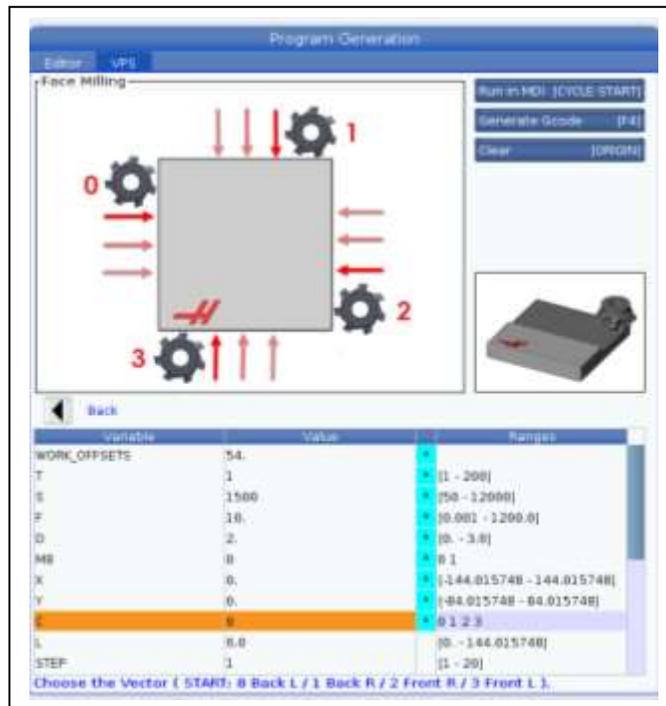
Below the step over width was calculated at 75% of the tool width. Also note R set to .1. Z in work coordinates set to zero. D is set to 1 for only one pass.

Enter the Width of the Step Over Passes.
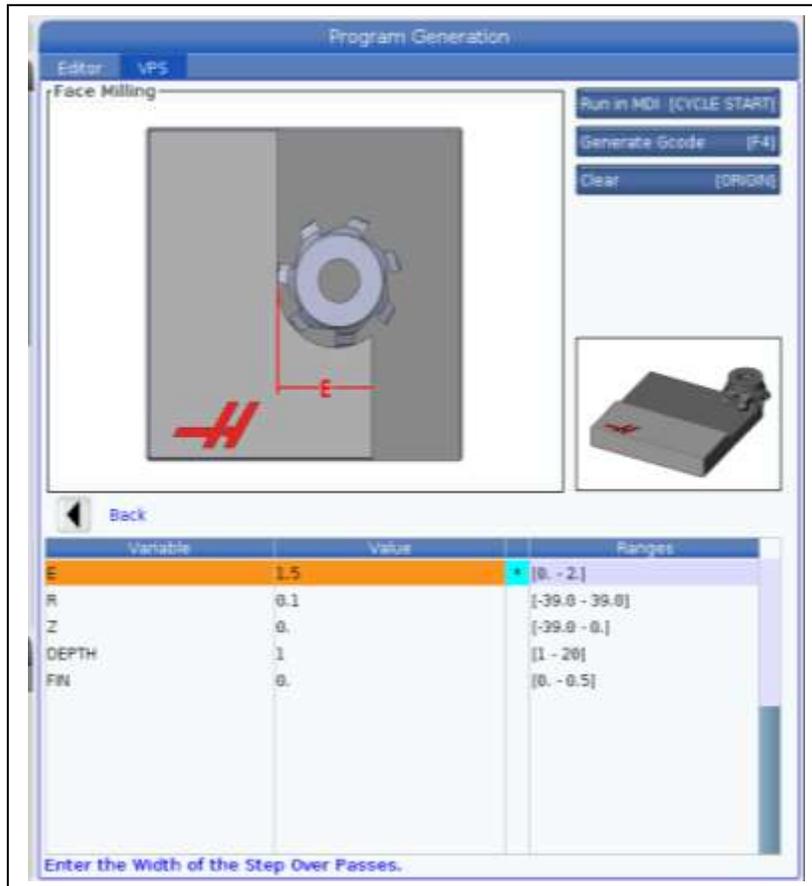
After appropriate data is entered then press the F4 key to generate the code. The pop up below asks where to code is to be generated.
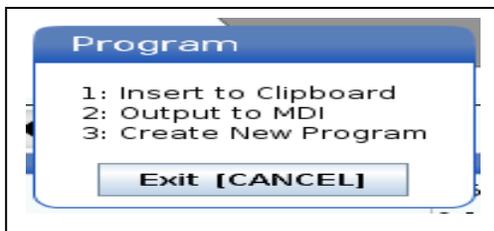
Pressing 3 gives the pop up for creating a new program at right.
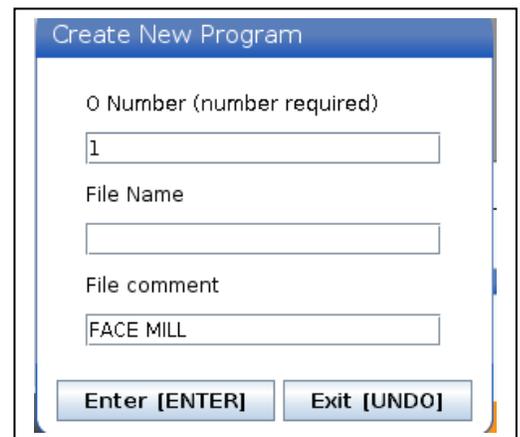
The code on the next page was created in program O00001. Open the memory directory, highlight O0001 and press Select Key

Square Contouring

Next go to the VPS directory and select the next operation which will be square contour milling.

Fill in the appropriate information such as center of the square or rectangle such as X3.0, Y-1.5 as seen below.
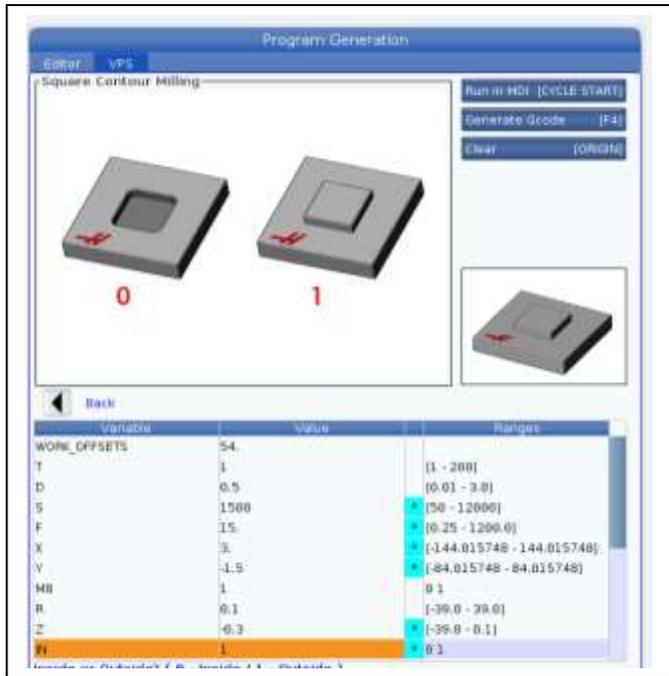
On the second page the 6"x3" width and length are given along with.06 radius.
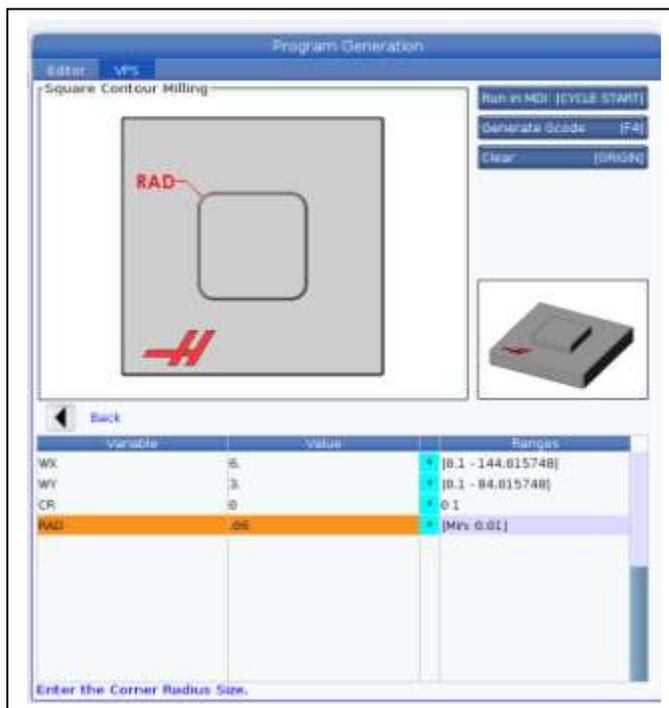
Instead of creating a new Program, key in 1 (Insert to Clipboard)

This will put the contour milling code in the clipboard which than can be pasted at the end of the original program using the F4 key.
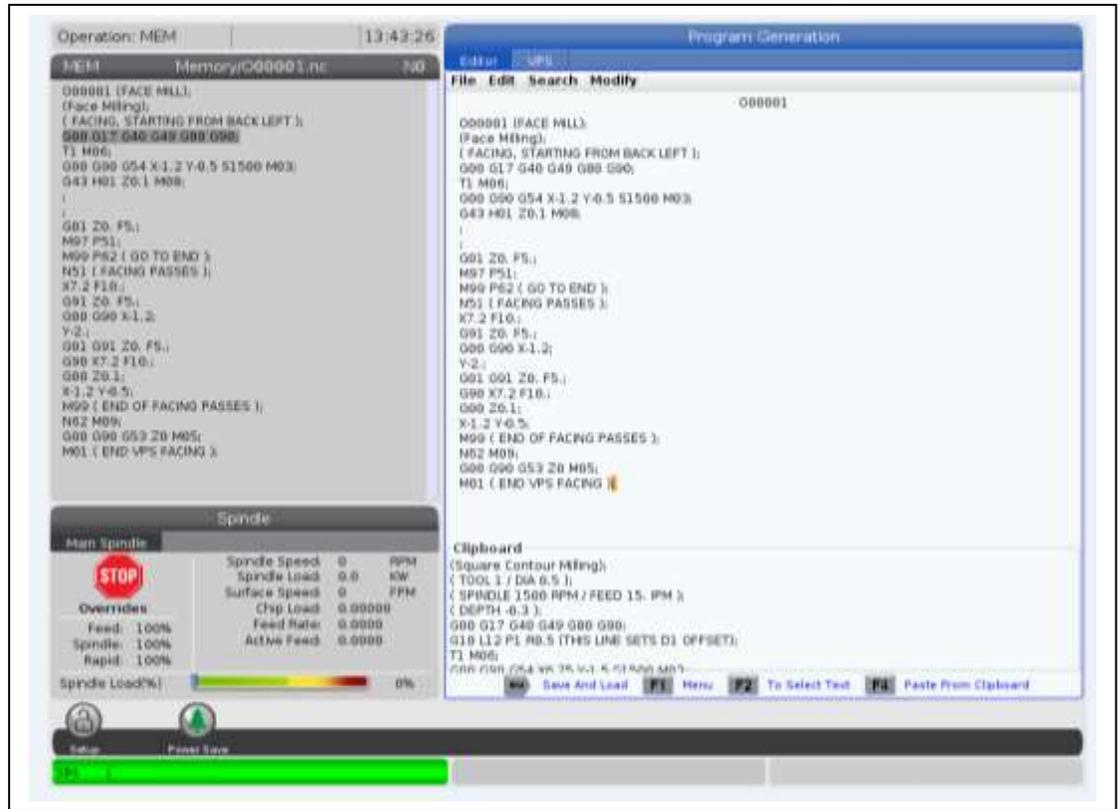
Code created in the other options in the VPS directory may be similarly added. This may be pocket milling, engraving or drilling.

The code created then may be checked for accuracy by running the program in graphics as below.

Circular Pocket Milling:

Below information has been keyed in for a 1.5" diameter, .3" deep circular pocket using the VPS.

The upper left hand corner is G54. Center of the pocket is X1.5, Y-1.5. For low carbon steel with a four flute high speed end the recommended speed is 175 ft./min chip load of .002"/rev-tooth. This gives a 1330 RPM, feed of 11 in/min.

Program Generation

Editor | **VPS**

Circle Pocket Milling
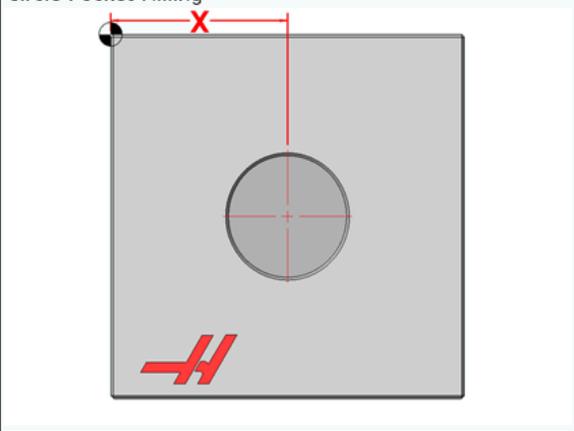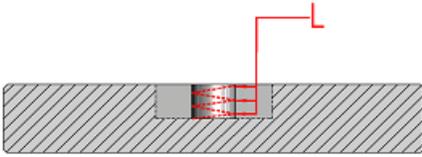
Run in MDI [CYCLE START]

Generate Gcode [F4]

Clear [ORIGIN]
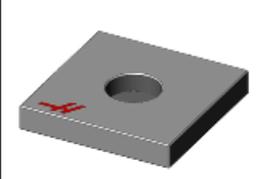
◀ Back

Variable	Value		Ranges
B	1.0	*	[0.51 - 1.0]
K	1.5	*	[1.0 - 84.015748]
L	6	*	[1 - 50]
J	.1	*	[0.01 - 0.5]
I	0.		[0. - .1]
ZFIN	0.		[0. - 0.05]

Number of Passes to Depth. (Program will make 1 additional pass at depth)

With the VPS a start drill hole is not required. The end mill circular ramps (helical interpolates) down to create a starting hole. The largest diameter an end mill can circular ramp down is 2 times the diameter. For a .5 inch end mill that gives B value of 1. K is the final diameter of 1.5" if the pocket.

L is the number of circular passes to get to the bottom. The center of the end mill will make a .5" diameter circular pass. This gives a circumference .5 x Π which equals 1.57. The recommended ramp angle is 2° for steel. Tan 2° x 1.57 equals .055 depth per turn or pitch. The total depth of .3/.055=5.45 passes. Rounding up to 6 gives the L value of number of circular interpolation passes to depth. A step over of .1" is given for the J value. No finish pass is given for the Z depth or ID of the pocket. The code for pocket created is put in the clipboard and pasted at the end of the program.

Square Pocket Milling:

Next a 2" x 2" square pocket is the created with the center location at X4.5 and Y-1.5.
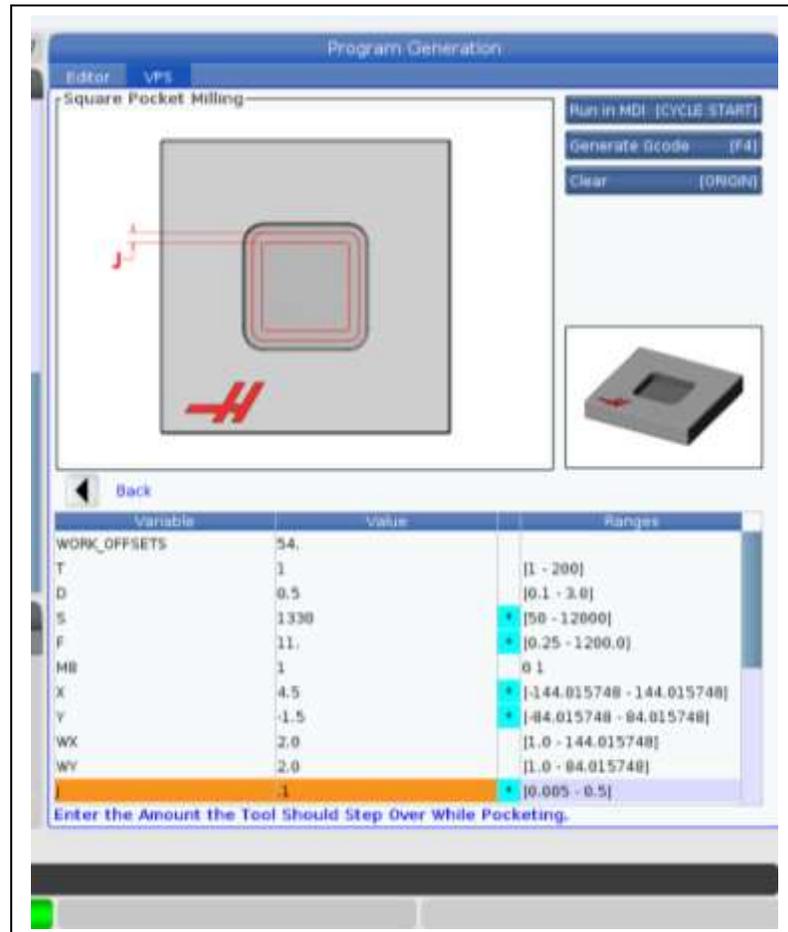
Going back to the VPS directory the square pocket option is highlighted and entered to give the above menu to be filled in. The same feeds and speeds as the circular pocket are entered along with the location of the middle of the pocket and the size of each side. A step over of .1 is entered for the J value.
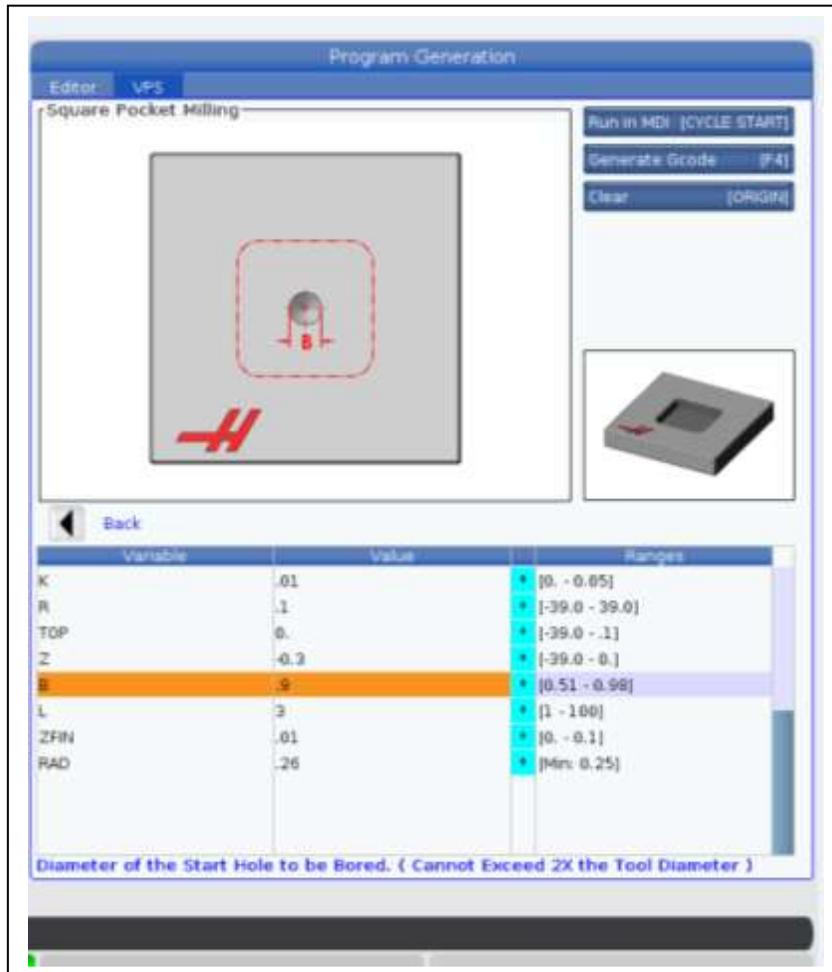
On the second page a finish pass of .01" is inserted for K. Depth information is given for the Z. The square pocket also circular interpolates to give a starting hole similar to the circular pocket. The size is indicated in B. The number of step downs is given in B.   Radii will be created in the corners of .26.

Drill Routines:

Below shows all the options with respect to drill type operations. These include CB counter bore operation, tapping boring and reaming. Once a tool# is indicated information about speeds, feeds,
and depths must be given.

Below information on feeds and speed for spot drill T4 and drill T5 are indicated.
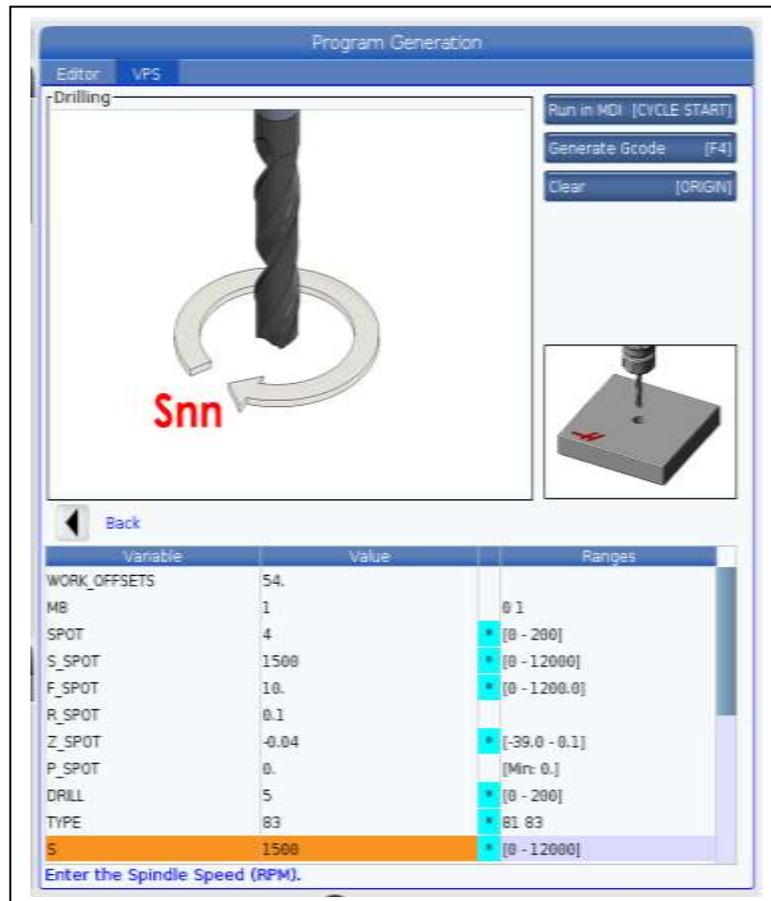
Below the depth, peck length, feed and thru the spindle coolant (M88) are indicated.
M gives the different hole placement options that may be used. In option 1 the X and Y
locations of all the holes may be given. Option 2 bolt circle with holes equally spaced may be

indicated. Option 3 holes equally spaced along an angle may be given. Option 4 bolt hole circles equally spaced by an angle may be created. Option 5 holes along a grid may be given.

Below information on bolt hole location, radial size, starting angle, and number of holes is indicated.
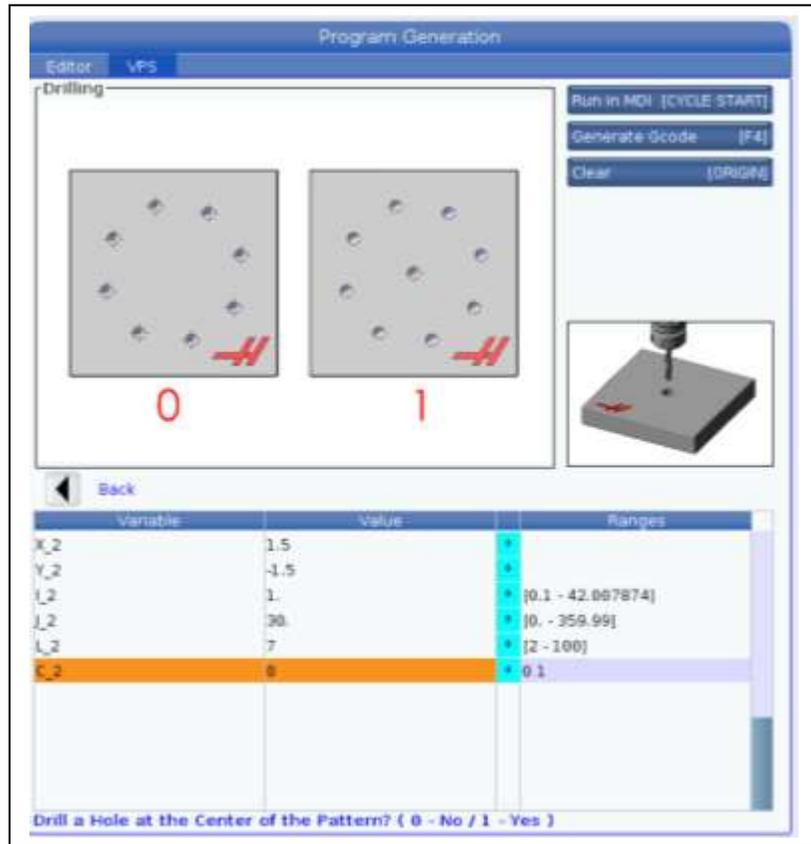
Below gives the graphics for the complete part.
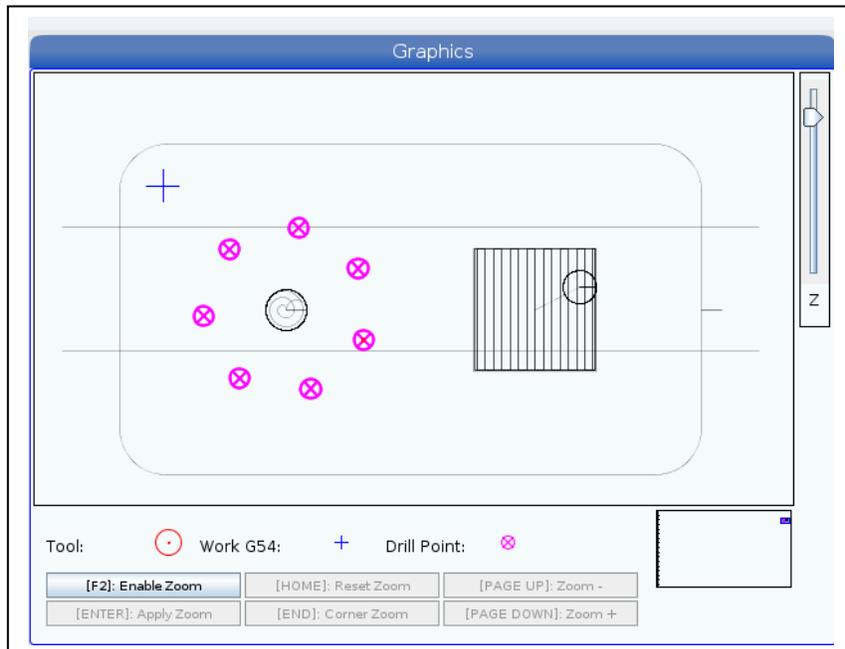
Engraving:

Below gives the template for engraving. Special engraving tools, spot drills, and ball end mills are used for engraving. The tool that is being used here is the spot drill T5.

High rpm's are used for engraving because just the tip is engaged with the material. The X and Y coordinates describe the location of the lower left hand corner of the first letter to be engraved. The P value indicates if engraving text (0) is desired or serial number (1). J is the height in inches of the letters to be engraved. I is the angle in degrees desired to text. The degree is the counterclockwise angle from 3 o'clock. In the text line just type in the text desired. With texting the alphabet letters will be upper case. If lower case is desired key in yellow shift key first. The shift key will also get you to special characters like # or $.

Running the program in graphics gives below
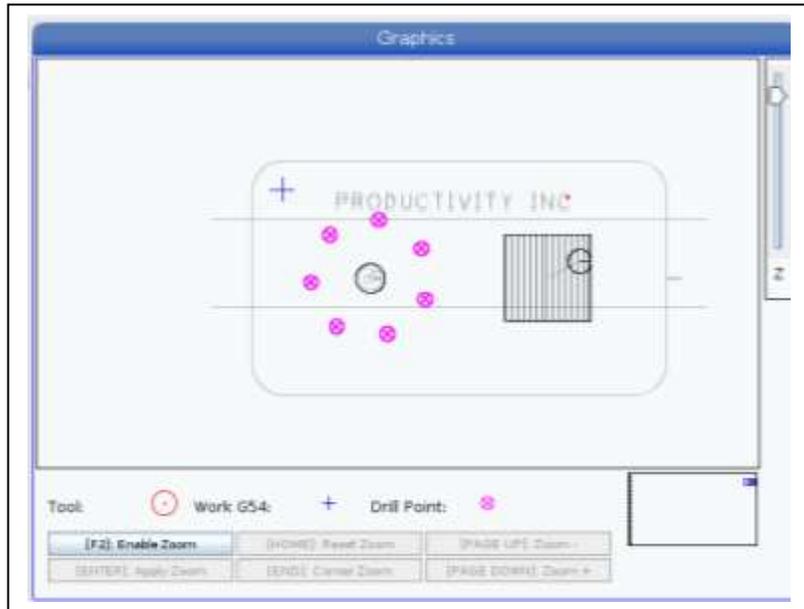
Final program created using VPS.

```
%
O00001 (FACE MILL)
(Face Milling)
( FACING, STARTING FROM BACK LEFT )
G00 G17 G40 G49 G80 G90
T1 M06
G00 G90 G54 X-1.2 Y-0.5 S1500 M03
G43 H01 Z0.1 M08
G01 Z0. F5.
M97 P51
M99 P62 ( GO TO END )
N51 ( FACING PASSES )
X7.2 F10.
G91 Z0. F5.
G00 G90 X-1.2
Y-2.
G01 G91 Z0. F5.
G90 X7.2 F10.
G00 Z0.1
X-1.2 Y-0.5
M99 ( END OF FACING PASSES )
N62 M09
G00 G90 G53 Z0 M05
M01 ( END VPS FACING )

(Square Contour Milling)
( TOOL 2 / DIA 0.5 )
( SPINDLE 1500 RPM / FEED 15. IPM )
( DEPTH -0.3 )
G00 G17 G40 G49 G80 G90
```

```
G10 L12 P1 R0.5 (THIS LINE SETS D1 OFFSET)
T2 M06
G00 G90 G54 X6.75 Y-1.5 S1500 M03
G43 H02 Z0.1 M08
G01 Z-0.3 F7.5
G01 G41 D02 X6.5 Y-1. F15.
G03 X6. Y-1.5 R0.5
( RADIUS CORNERS - OUTSIDE )
G01 G90 Y-3. ,R0.0600
G01 X0. ,R0.0600
G01 Y0. ,R0.0600
G01 X6. ,R0.0600
G01 Y-1.5
G03 X6.5 Y-2. R0.5
G01 G40 X6.75 Y-1.5
G00 Z0.1 M09
G00 G90 G53 Z0 M05
M01 ( END VPS SQUARE CONTOUR )

(Circle Pocket Milling)
( TOOL = 2 / DIA. 0.5 )
( SPINDLE 1330 RPM / FEED 11. IPM )
( DEPTH -0.3 / STEPOVER .1 / FINISH 0. )
G00 G17 G40 G49 G80 G90
G10 L12 P2 R0.5 (THIS LINE SETS D2 OFFSET)
T2 M06
G00 G54 X1.5 Y-1.5
S1330 M03
G43 H02 Z0.1
M08
G01 X1.75 Y-1.5 F11.
G01 Z0. F5.5
(BORE DOWN TO DEPTH)
G03 G91 Z-0.05 I-0.25 F11. L6
(FINAL BORE PASS AT DEPTH)
G03 G90 I-0.25 Z-0.3 F11.
G01 G90 X1.5 Y-1.5 Z0.1 F5.5
( ROUGH AND FINISH POCKET )
G13 G90 D02 I0.5 K0.75 Q0.1 F11. Z-0.3
G00 Z0.1 M09
G00 G90 G53 Z0 M05
M01 ( END VPS CIRCLE POCKET )

(Square Pocket Milling)
( POCKET - 2.0 X 2.0 INCH )
( TOOL 2 / DIA 0.5 )
( SPINDLE 1330 RPM / FEED 11. IPM )
( DEPTH -0.3 / STEPOVER .1 )
G00 G17 G40 G49 G80 G90
G10 L12 P2 R0.5 (THIS LINE SETS D1 OFFSET)
T2 M06
G00 G90 G54 X5.04 Y-1.22 S1330 M03
G43 H02 Z0.1 M08
G01 G91 X0.2 F5.5
G01 Z0. F5.5
G03 G91 Z-0.05 I-0.2 F5.5 L6
```

```
G03 G90 Z-0.3 I-0.2 F5.5
G01 G90 X5.04 Y-1.22 Z0. F11.
G150 P125 Z-0.3 G41 I0.1 K0.01 R0. D02 Q0.0967 F11.
G40 X4.5 Y-1.5 Z0.1
M09
G00 G90 G53 Z0 M05
M99 P135
N125 ( SUBROUTINE FOR VPS SQUARE POCKET )
G01 X5.5
Y-0.5
X3.5
Y-2.5
X5.5
G01 Y-1.22
M99
N135 M01 ( END VPS SQUARE POCKET )

(Drilling)
( BOLT HOLE CIRCLE )
( 1. RADIUS / 30. START ANGLE / 7 HOLES )
( *** TOOL LIST *** )
( SPOT - T4 / 1500 RPM / F10. )
( R PLANE 0.1 / Z DEPTH -0.04 )
( DRILL - T5 / 1500 RPM / F15. )
( G83 / R PLANE 0.1 / Z DEPTH -1.1 )
( *** )
( SPOT - T4 / 1500 RPM / F10. )
( R PLANE ::R_SPOT / Z DEPTH -0.04 )
G00 G17 G40 G49 G80 G90
T4 M06
M01
G00 G90 G54 X1.5 Y-1.5
S1500 M03
G43 H04 Z0.1
M08
G82 G98 Z-0.04 F10. R0.1 P0. L0
G70 I1. J30. L7
G80 Z0.1 M09
G00 G90 G53 Z0 M05

( DRILL - T5 / 1500 RPM / F15. )
( G83 / R PLANE 0.1 / Z DEPTH -1.1 )
G00 G17 G40 G49 G80 G90
T5 M06
M01
G00 G90 G54 X1.5 Y-1.5
M88
S1500 M03
G43 H05 Z0.1
M08
G83 G98 Z-1.1 Q0.25 F15. R0.1 P0. L0
G70 I1. J30. L7
G80 Z0.1 M09
M89
G00 G90 G53 Z0 M05
G00 G90 G53 Z0 M05
```

M01 (END VPS DRILLING PROGRAM)

(Engraving)
(TEXT ENGRAVING: PRODUCTIVITY INC)
(TOOL 4 / DEPTH -0.008)
(SPINDLE 2000 RPM / FEED 15. IPM)
G00 G17 G40 G49 G50 G69 G80 G90
T4 M06
G00 G90 G54 X0.9 Y-0.3 S2000 M03
G43 Z0.05 H04
G47 E7.5 F15. I0. J0.25 P0 R0.05 Z-0.008 (PRODUCTIVITY INC)
G00 Z0.05 M09
G00 G90 G53 Z0 M05
M01 (END VPS ENGRAVING)
(Engraving)
(TEXT ENGRAVING: PRODUCTIVITY INC)
(TOOL 5 / DEPTH -0.008)
(SPINDLE 2000 RPM / FEED 15. IPM)
G00 G17 G40 G49 G50 G69 G80 G90
T4 M06
G00 G90 G54 X0.9 Y-0.35 S2000 M03
G43 Z0.05 H04
G47 E7.5 F15. I0. J0.25 P0 R0.05 Z-0.008 (PRODUCTIVITY INC)
G00 Z0.05 M09
G00 G90 G53 Z0 M05
M01 (END VPS ENGRAVING)

%

.

Thread Milling:

Thread Milling may be selected in the VPS directory as shown below

Note internal threads with major diameters of less than an inch are not possible with this application. Also when the code turns on the cutter compensation the thread mill tool must smaller than ½ the diameter of the major diameter in ID threads. 1.5"-12 thread using VPS is
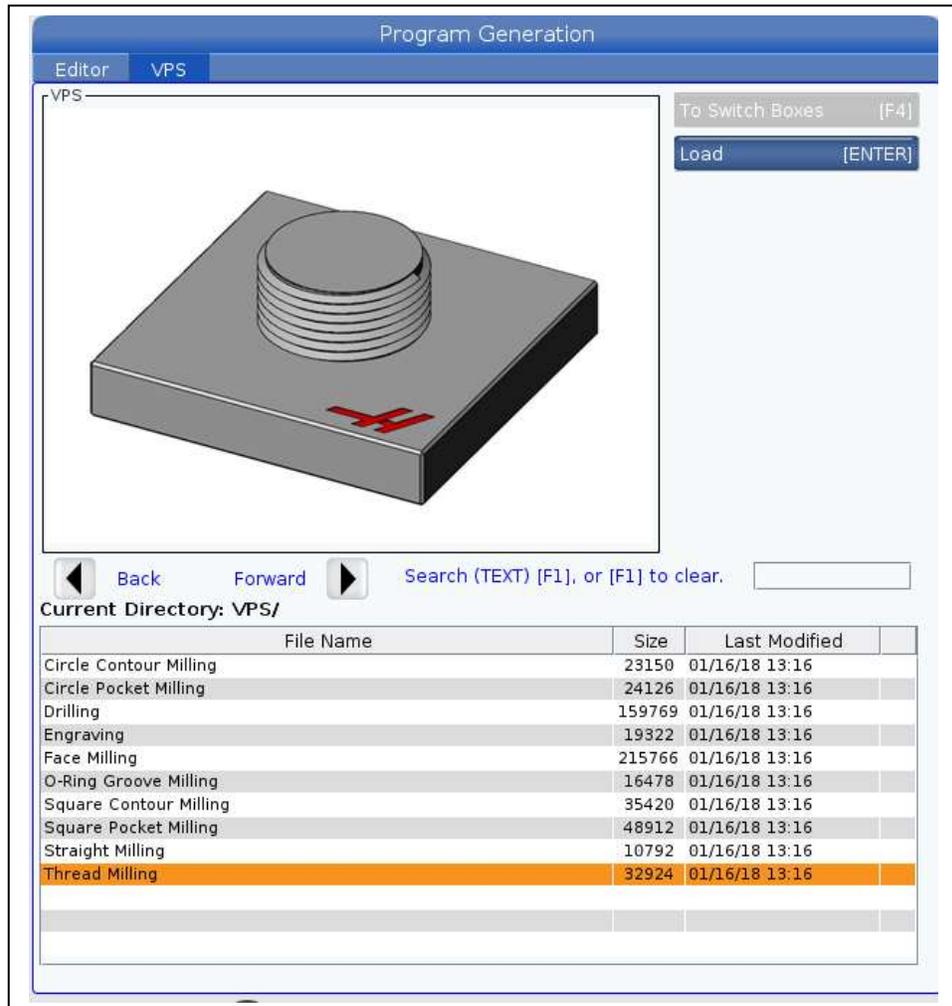
The following example shows using VPS to create the code for a 1.5"-12 internal thread. Thread minor 1.439".

For mild steel a recommended surface feed is 450 ft/mn and feed rate of .002"/rev-tooth. The tool being used is a Carmex single flute thread mill with a diameter of .538. This gives a **RPM=3195** and a feed rate of

6.4 in/min. When calculating feed rate for thread mills the difference in size of the thread mill and the thread
major being cut must be considered. If D1= diameter of the thread mill and D2 = the diameter of the thread major to be cut the feed rate must be taken times the ratio of (D2-D1)/D2. In the example (1.5-.538)/1.5 which equals .64. 6.4 in/min x .64 = **4.1 in/min**
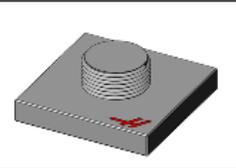

The pitch of the thread must be entered as below. The pitch for 12 threads/in =1/12**=.0833**.

To get the number of passes the depth of the thread must divided by the pitch.
For a .5" deep thread .5/.0833=**6.0.** The control will put in the **Z depth**. I is the major for a 1.5"-12 thread and it is 1.5.
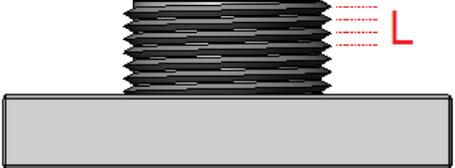
The code for circular pocket milling the minor and thread milling created is using VPS
Note with the VPS thread milling the cutting starts at top using G2 (clockwise radius) which is conventional milling. To use climb milling the thread milling starts at the bottom and G3 or counter clockwise radius is thrown going in the positive Z direction.

Thread Milling Program

```
%
O00012 (THREAD MILL 1.5-12 INT)

(Circle Pocket Milling)
( TOOL = 2 / DIA. 0.5 )
( SPINDLE 1330 RPM / FEED 11. IPM )
( DEPTH -0.6 / STEPOVER .1 / FINISH 0. )
```

```
(MILL POCKET MINOR 1.419)
G00 G17 G40 G49 G80 G90
G10 L12 P2 R0.5 (THIS LINE SETS D2 OFFSET)
T2 M06
G00 G54 X0. Y0.
S1330 M03
G43 H02 Z0.1
G01 X0.25 Y0. F11.
G01 Z0. F5.5
(BORE DOWN TO DEPTH)
G03 G91 Z-0.0545 I-0.25 F11. L11
(FINAL BORE PASS AT DEPTH)
G03 G90 I-0.25 Z-0.6 F11.
G01 G90 X0. Y0. Z0.1 F5.5
( ROUGH AND FINISH POCKET )
G13 G90 D02 I0.5 K0.7095 Q0.1 F11. Z-0.6
G00 Z0.1 M09
G00 G90 G53 Z0 M05
M01 ( END VPS CIRCLE POCKET )

(Thread Milling)
( INTERNAL RIGHT HAND THREAD MILLING )
( TOOL 9 / DIA .538 )
( SPINDLE 3195 RPM / FEED 4.1 IPM )
( TOP OF THREAD 0. / PITCH .0833 )
G00 G17 G40 G49 G50 G69 G80 G90
G10 L12 P9 R0.538 (THIS LINE SETS D9 OFFSET)
T9 M06
G00 G90 G54 X0. Y0. S3195 M03
G43 H09 Z0.1
G01 Z0. F2.05
G01 G42 X0.375 Y0.375 D09 F4.1
G02 X0.75 Y0. R0.375
G02 G91 I-0.75 Z-0.0833 L6
G02 G90 X0.375 Y-0.375 R0.375
G01 G40 X0. Y0.
G01 Z0.1 F2.05 M09
G00 G90 G53 Z0 M05
M01 ( END VPS THREAD MILLING )

%
```