Haas Factory Outlet

A Division of Productivity Inc

HFOMN/Haas CNC Series

Mill Operator Training Manual with NG2 Next Generation Control &

VPS (Visual Programming System)

Rev 2/2017

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
For Minnesota Classes, Click on the Training Registration Button
✉ trainingmn@productivity.com

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.

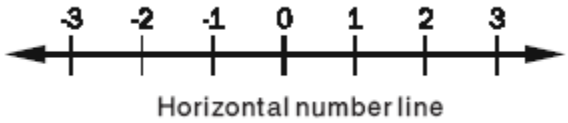
This manual can be used as both an operator's manual and as a programmer's manual. It is intended to give a ***basic*** understanding of CNC programming and its applications. It is not intended as an in-depth study of all ranges of machine use, but as an overview of common and potential situations facing 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 3/25/12; Added NG2 02/2017

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.



"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 value's 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"

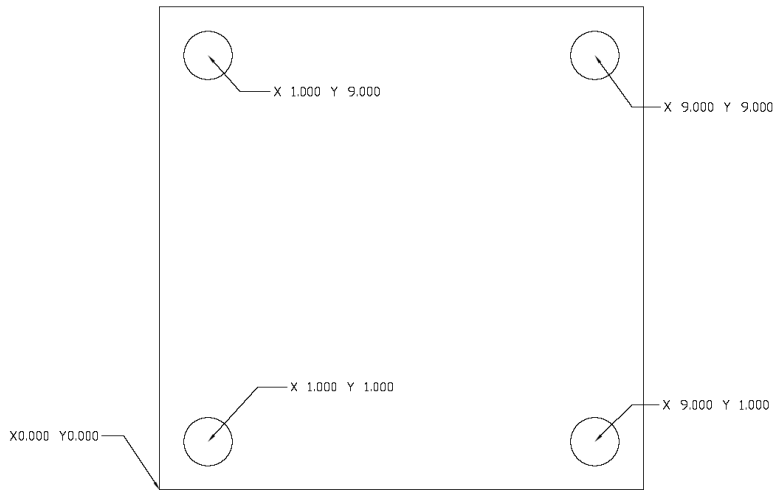
Remember that zero may be placed at any point along the line, and that once placed, one side of zero has negative increments and the other side has positive increments.

Vertical Number Line known as the "Y" axis

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.)

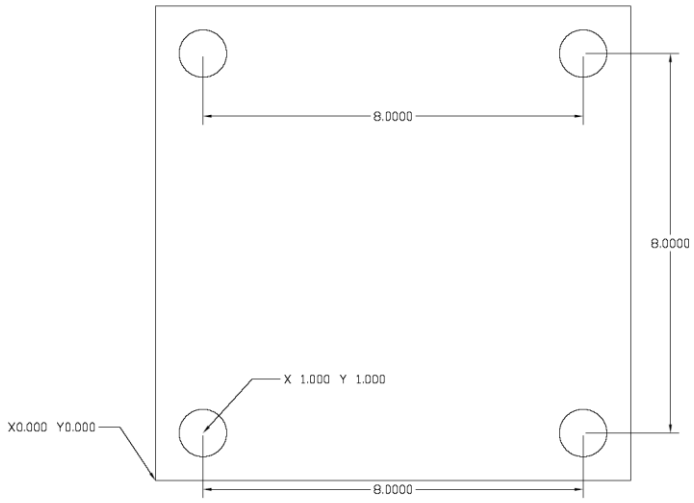
X 1.0000 Y 1.0000
X 9.0000 Y 1.0000
X 9.0000 Y 9.0000
X 1.0000 Y 9.0000

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

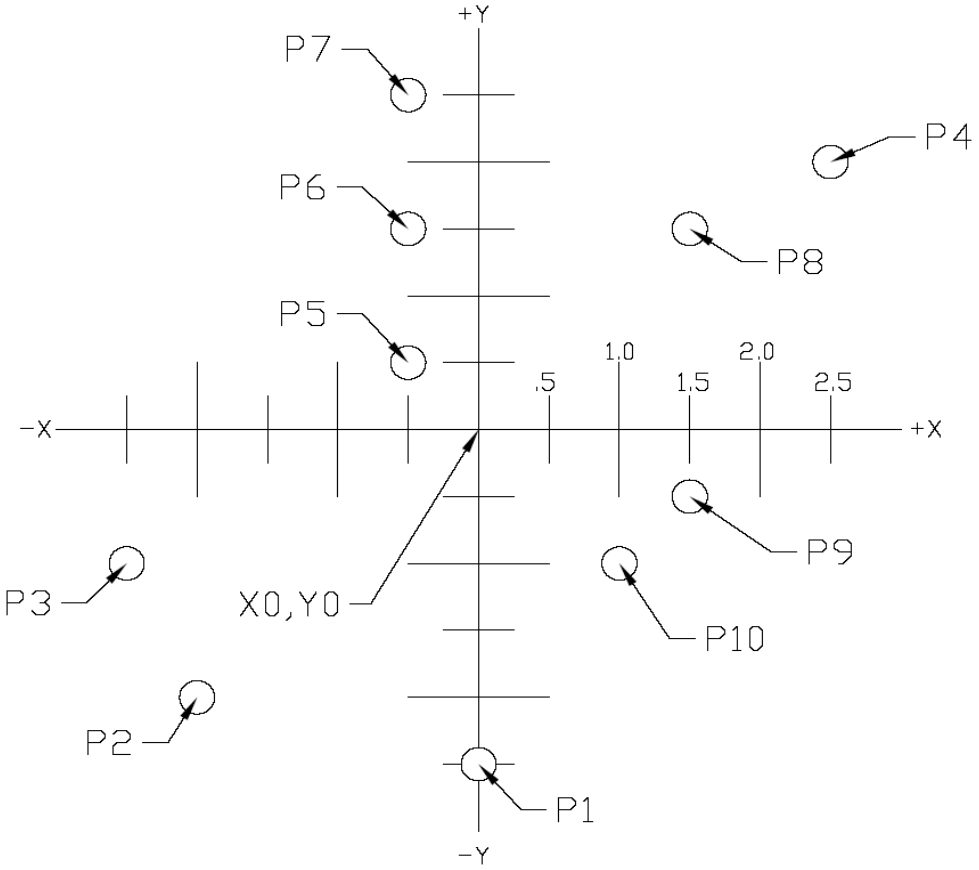
Program to move the machine to the same 4 hole locations using G91 (Incr.)

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		

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.

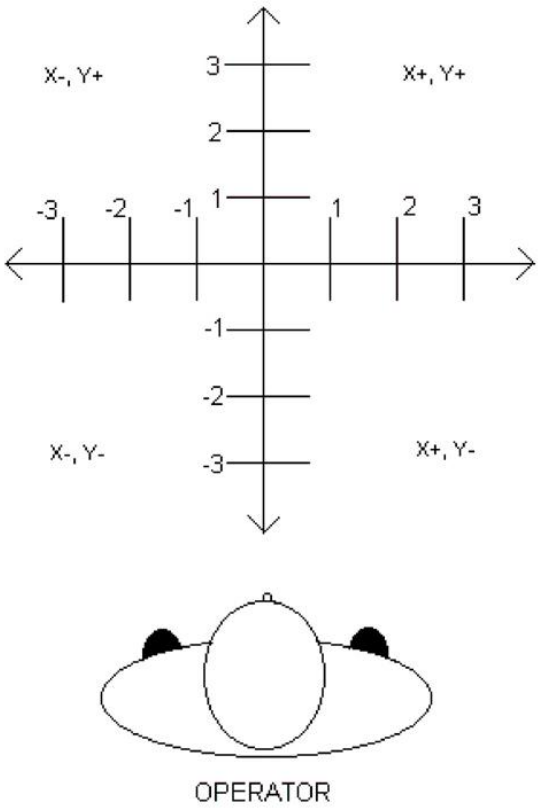
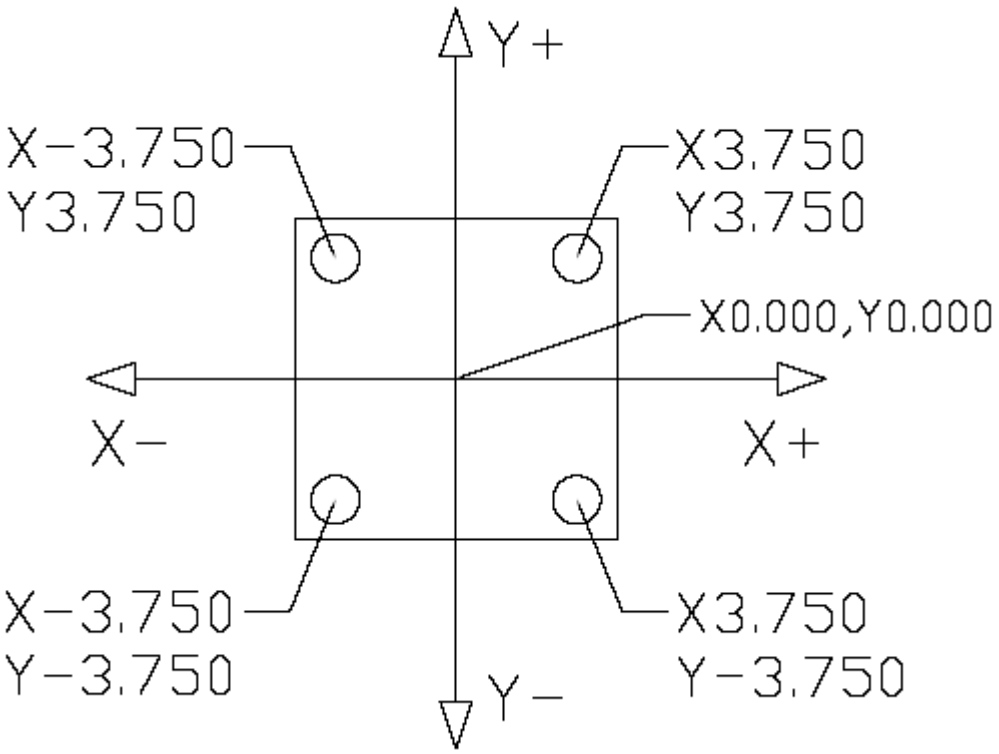


Fig. 1-4: Operator's working grid.

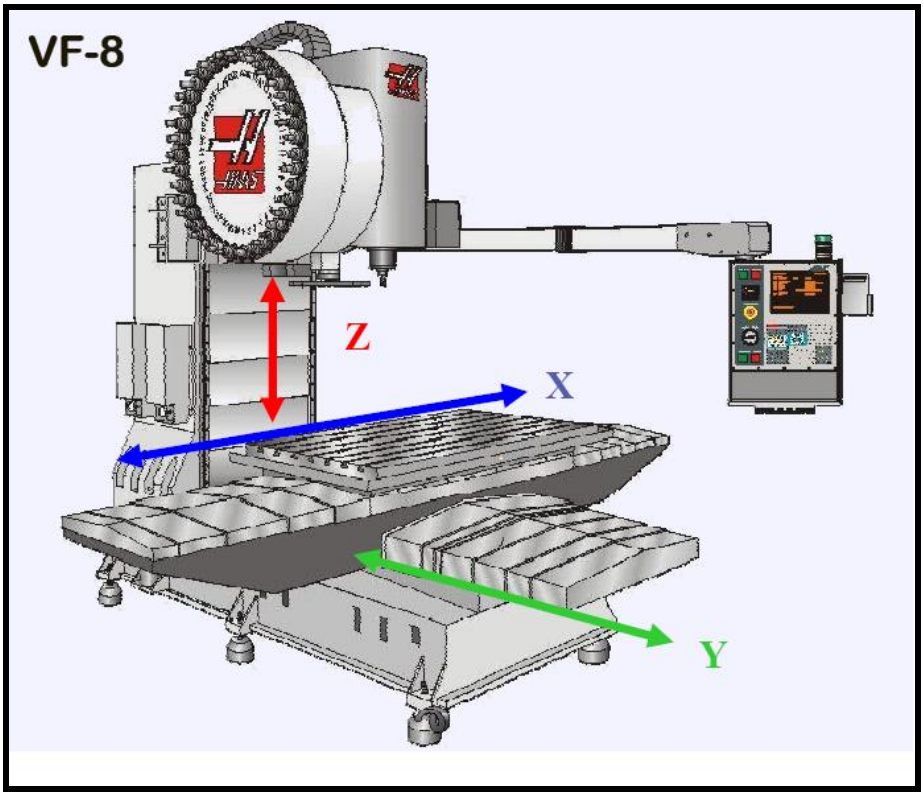
Whenever we set a zero somewhere on the X axis and somewhere on the Y axis, we have automatically caused an intersection of the two lines. This is known as "zero", where our X and Y lines would both be at a value of 0. We will move this point to a "Zero" position on our part so that we can steer the 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 axes of Haas VMCs have a resolution of .0001” inches (or .001mm).

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

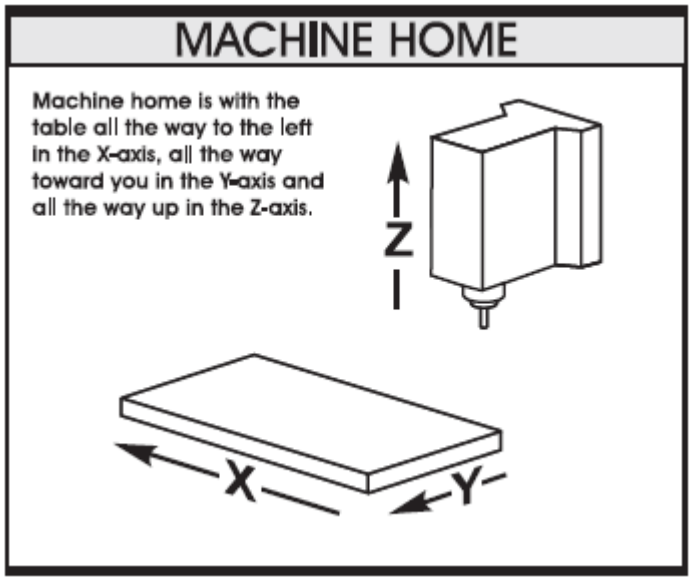
Remember, when we are moving the machine, we are concerned with positioning the spindle. Although the machine table is the moving part, our programs are written as if the spindle was moving.

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.

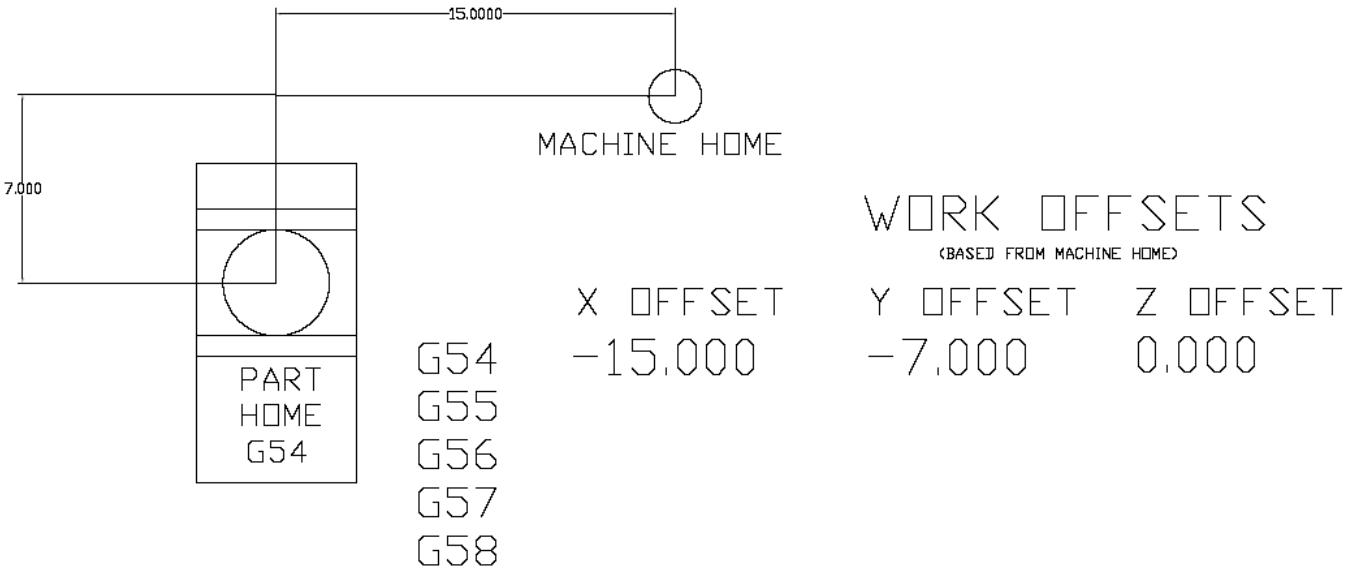


This is crucial to the operation and function of a CNC machine as all of our programs, fixtures, and tooling are based off of the **machine home position described as X0, Y0, Z0**.

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.*

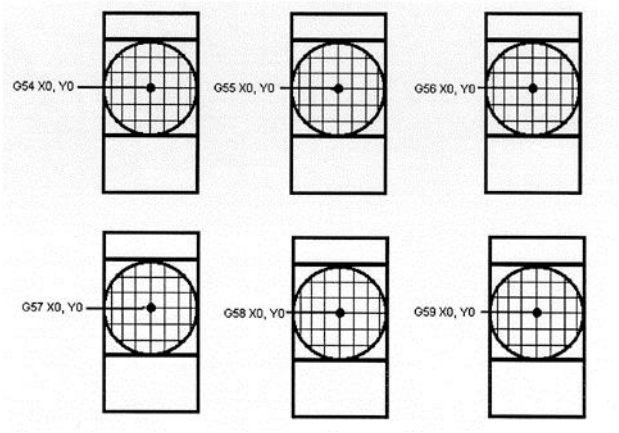
G53 Positioning In Reference to Machine Home

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

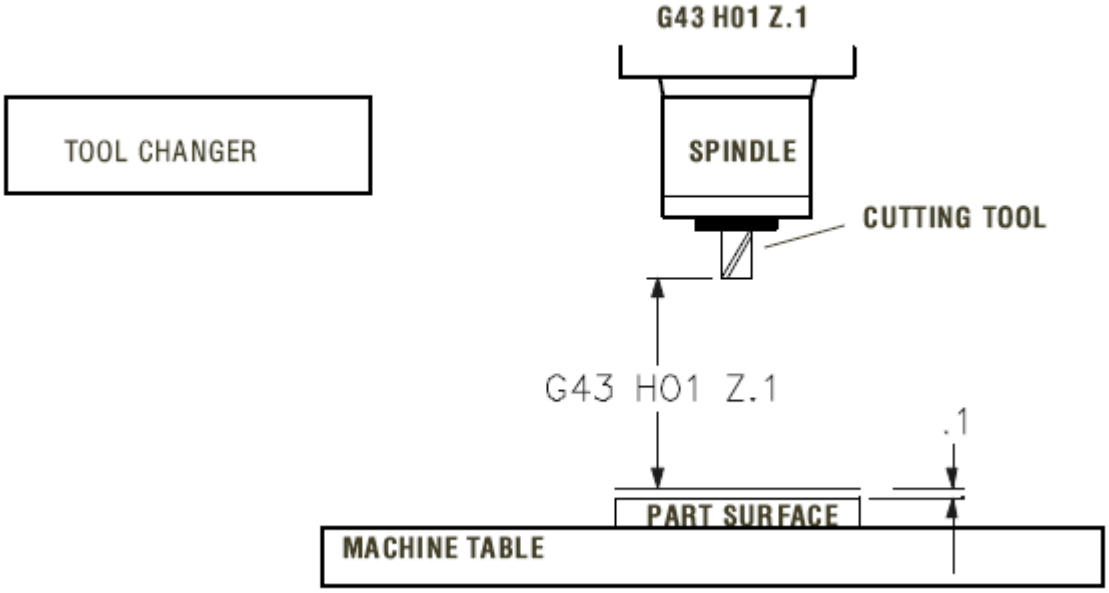
The figure above represents a multiple-fixture setup. Each vise will have an absolute zero once it is specified in the program. This is done by using G54 through G59. G54 through G59 and G154 P1 through G154 P99 gives a total of 105 different work offsets which may be used.

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.

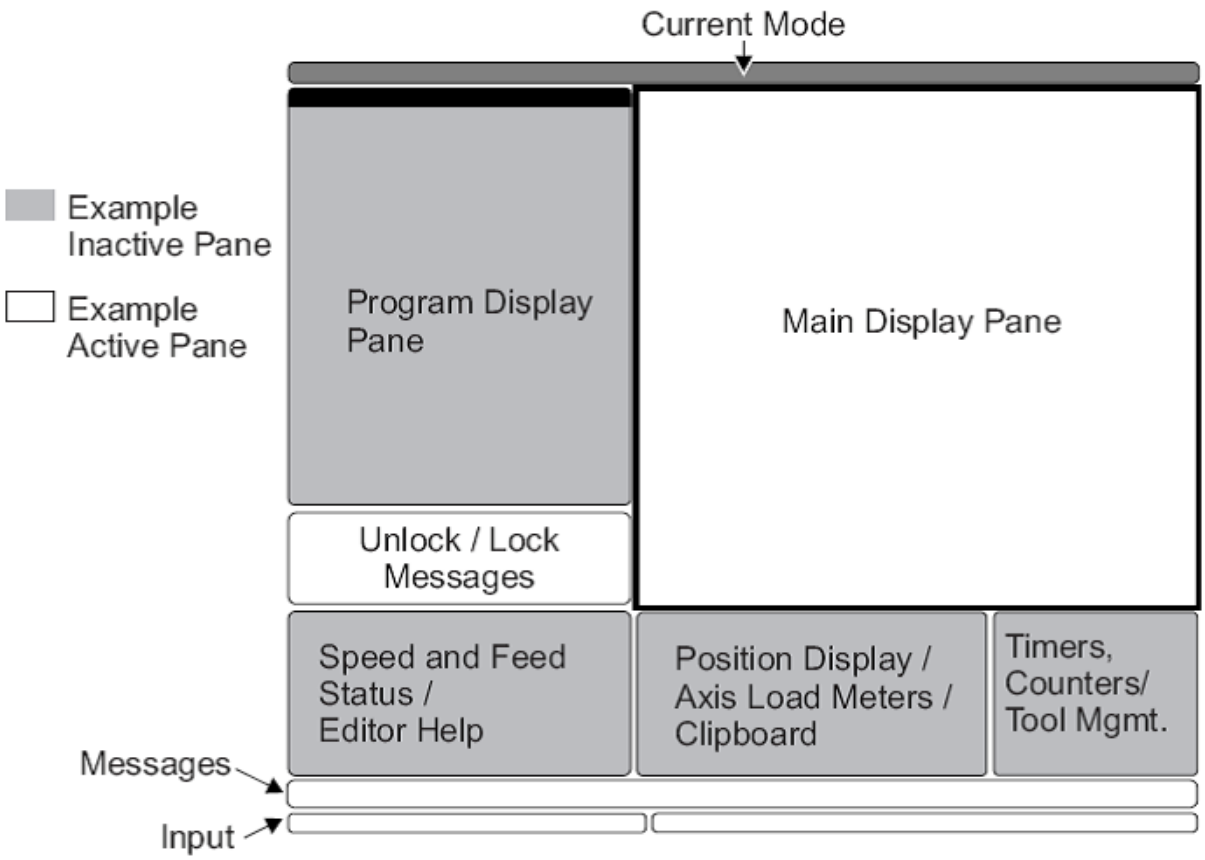
Cycle Start – Starts program in run mode or graphics mode.

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.

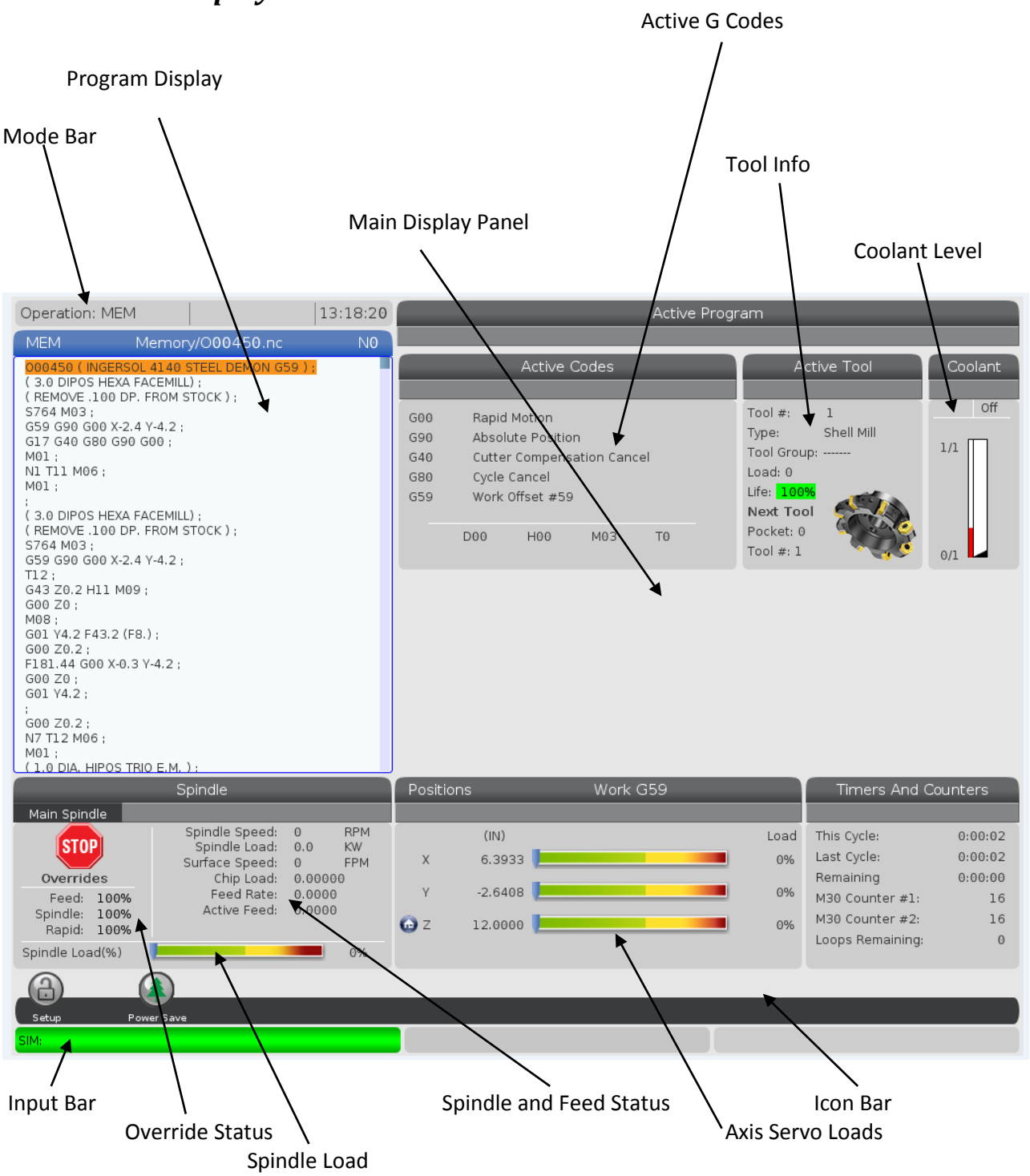
Edit: EDIT, MDI/DNC, LIST PROG keys. Provides all program editing, management, and transfer functions.

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.

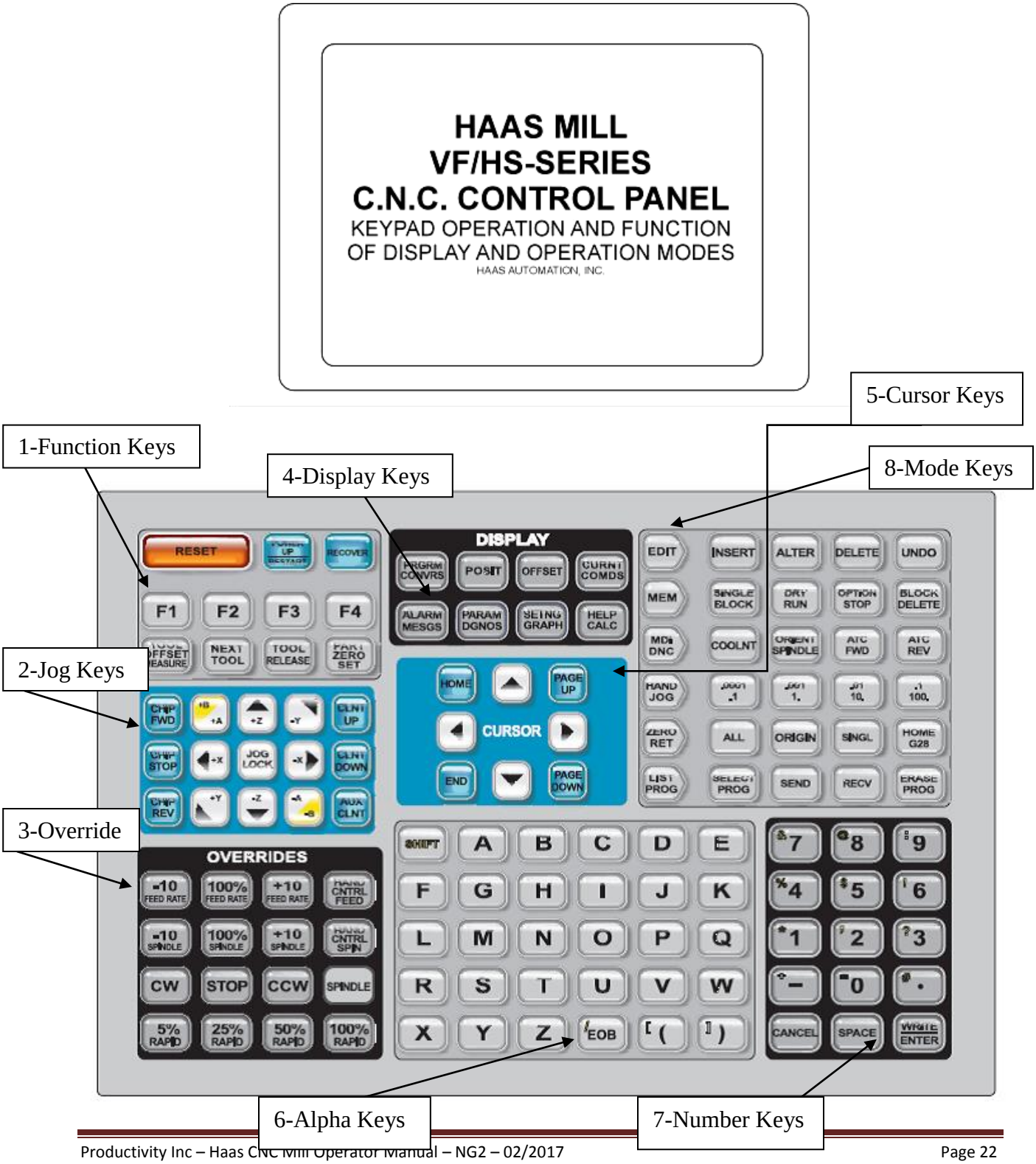
NGA Control Display



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.

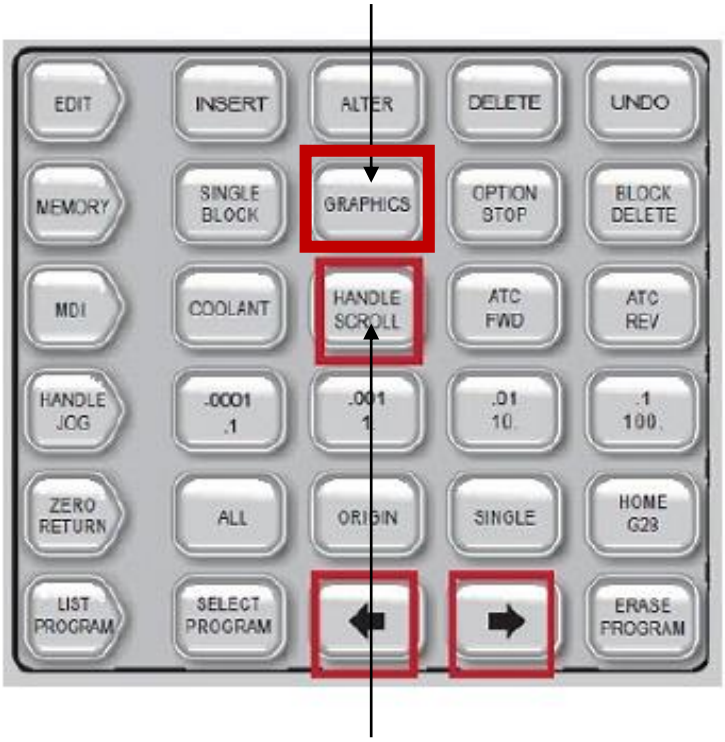


Next Generation Keyboard Changes

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.

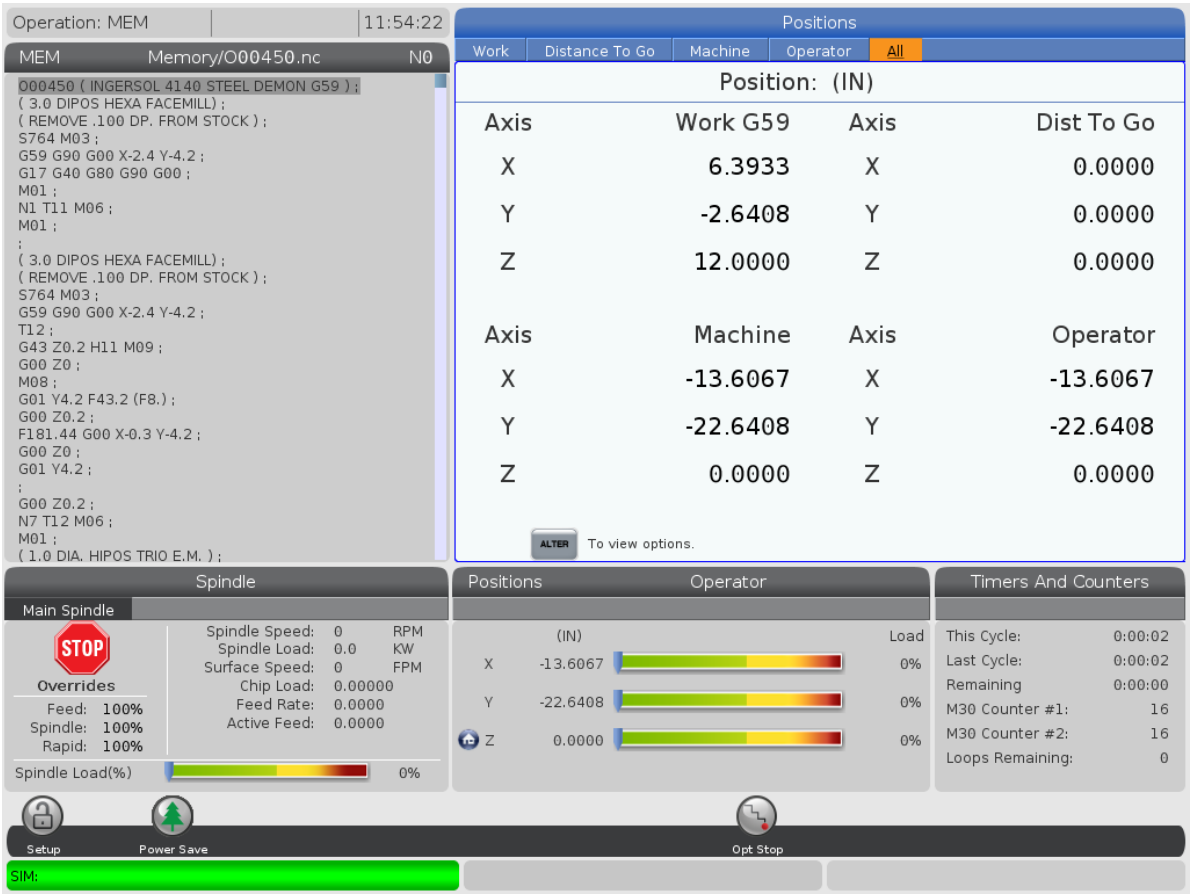
These keys allow navigation forward and backward thru page history.

NGC does not provide for a RS232 port.

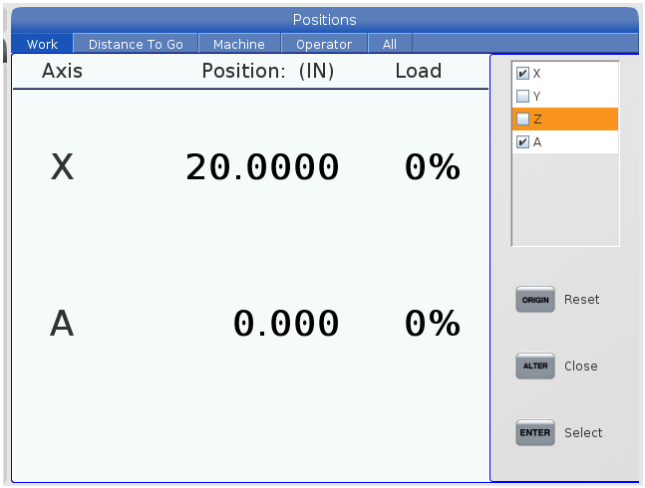
Only Ethernet connections are available or an upgrade to WIFI.

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.

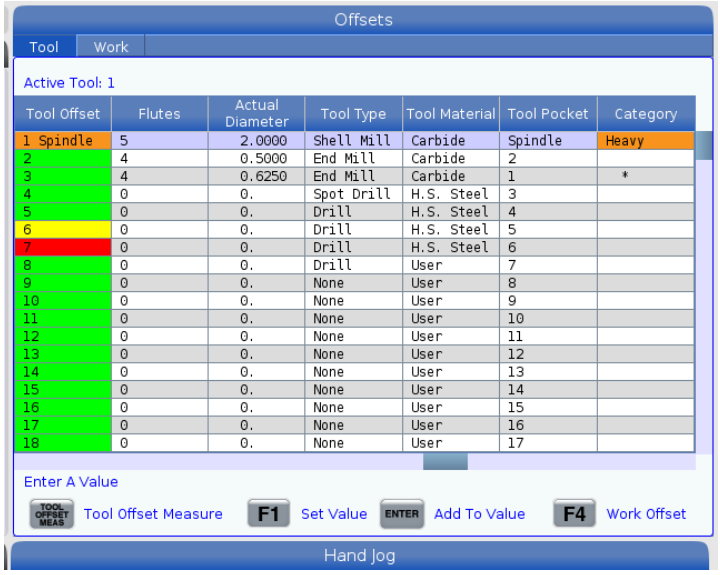


By depressing the [ALTER] key the display may be customized as shown below:



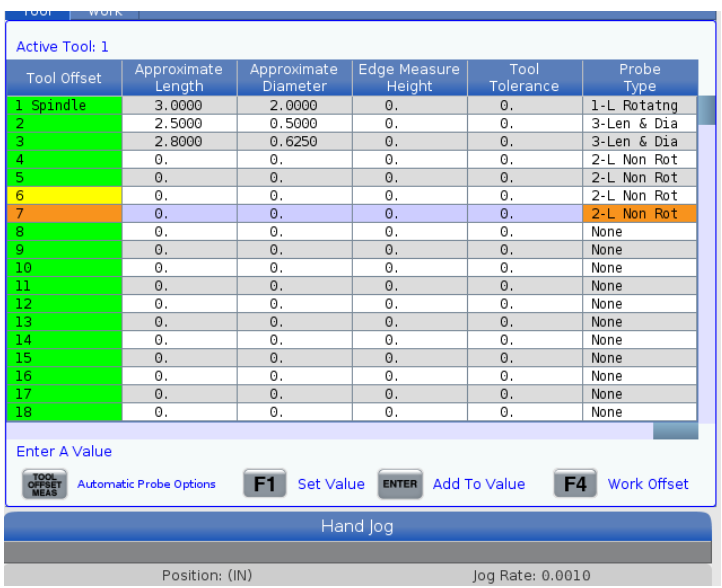
The **[Offset Key]** is active below.
Tool Length Geometry values are shown along with Tool Diameter. If tool life management is being used red will indicate the tool life has expired, yellow is a warning. Also Wear values may be entered for H(Length) and D(Tool Diameter).

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



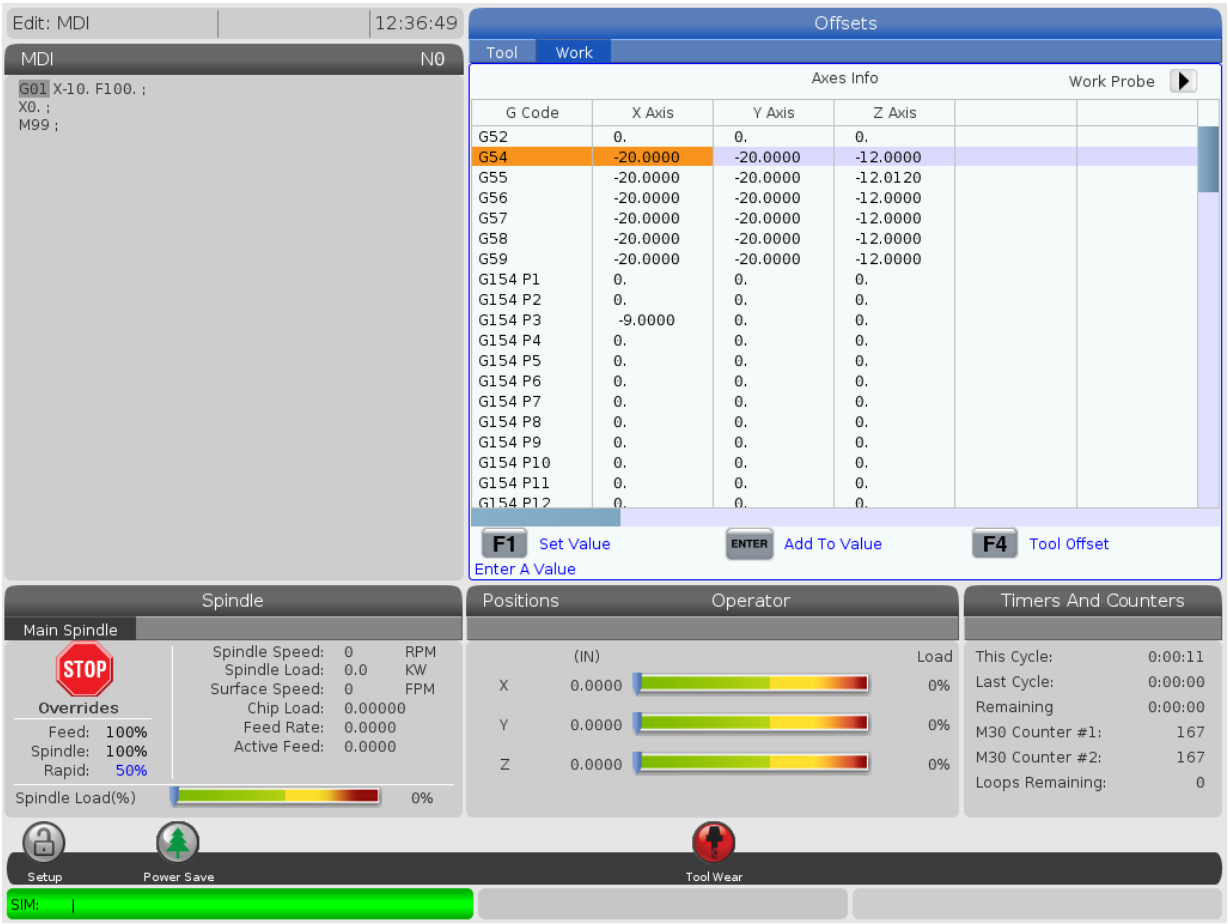
Tool Offset	Flutes	Actual Diameter	Tool Type	Tool Material	Tool Pocket	Category
1 Spindle	5	2.0000	Shell Mill	Carbide	Spindle	Heavy
2	4	0.5000	End Mill	Carbide	2	
3	4	0.6250	End Mill	Carbide	1	*
4	0	0.	Spot Drill	H.S. Steel	3	
5	0	0.	Drill	H.S. Steel	4	
6	0	0.	Drill	H.S. Steel	5	
7	0	0.	Drill	H.S. Steel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

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.

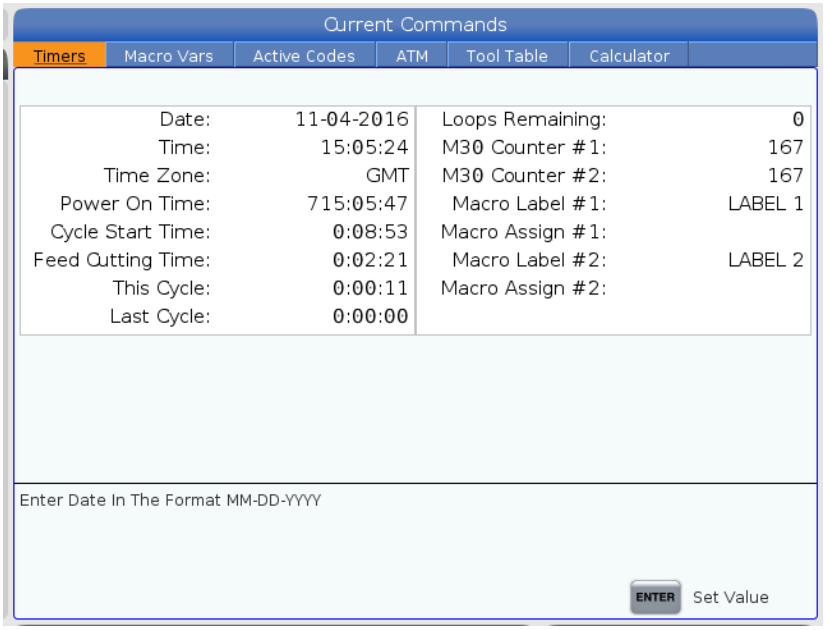


Tool Offset	Approximate Length	Approximate Diameter	Edge Measure Height	Tool Tolerance	Probe Type
1 Spindle	3.0000	2.0000	0.	0.	1-L Rotatng
2	2.5000	0.5000	0.	0.	3-Len & Dia
3	2.8000	0.6250	0.	0.	3-Len & Dia
4	0.	0.	0.	0.	2-L Non Rot
5	0.	0.	0.	0.	2-L Non Rot
6	0.	0.	0.	0.	2-L Non Rot
7	0.	0.	0.	0.	2-L 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

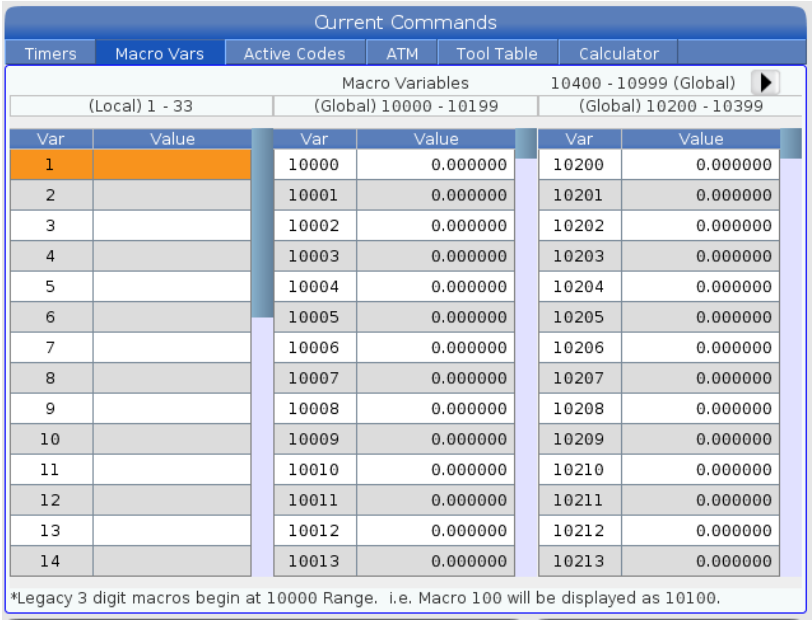
Going back to the first offset page curser to the top to highlight **Tool.** Then highlight **Work.** This opens the work offsets page as below.
One may also toggle between work and tool offsets pages by using the **[F4]** key.



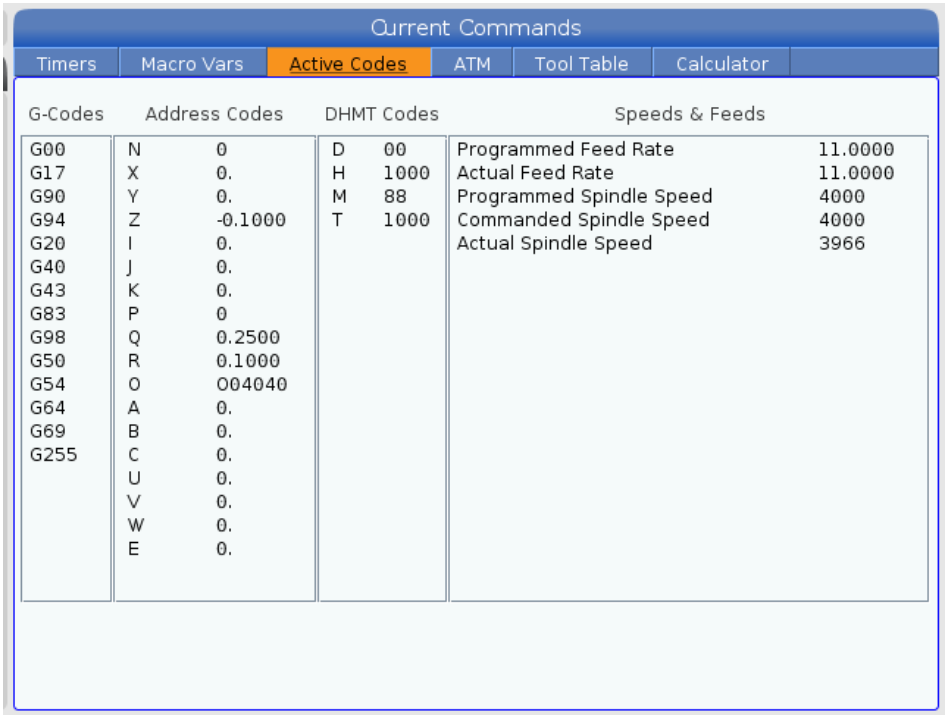
[Current Commands] key gives many different options. Note the **Timers** tab is highlighted. On this page current date and time, cycle time, M30 counters for keeping track of the number of parts and Macro values may be indicated. One may cursor down into the page to reset the M30 counter by pressing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



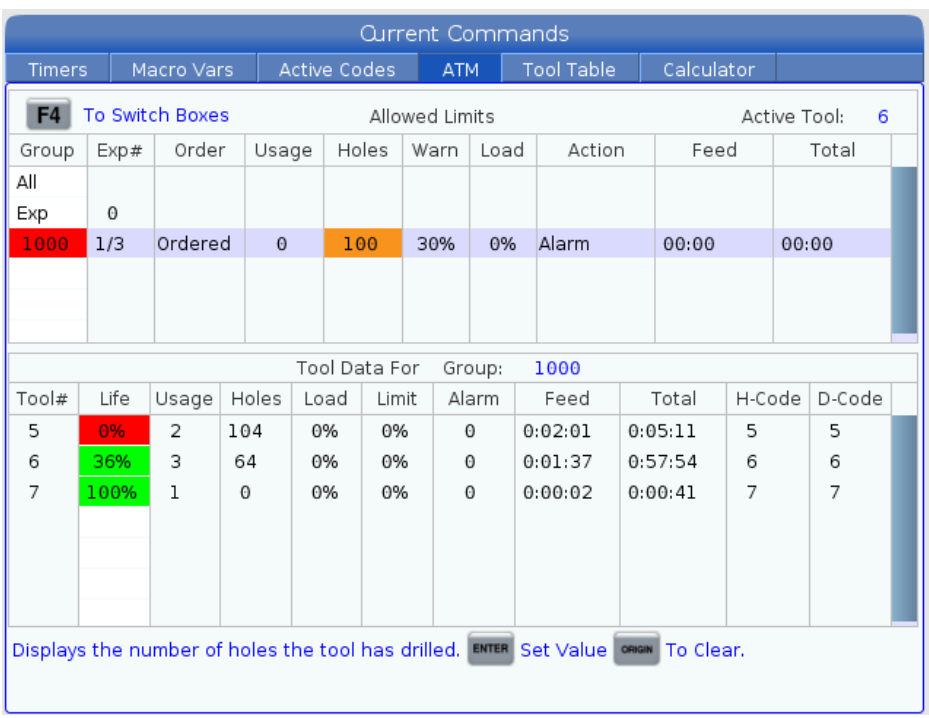
[Macro Vars] key gives the values of different variables # used in Macro Programming. Note 10000 must be added to the old legacy numbers to find the equivalent variable. Variable #188 becomes #10188. Once inside the variable table the variable # needed can be keyed in and the down cursor key will do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



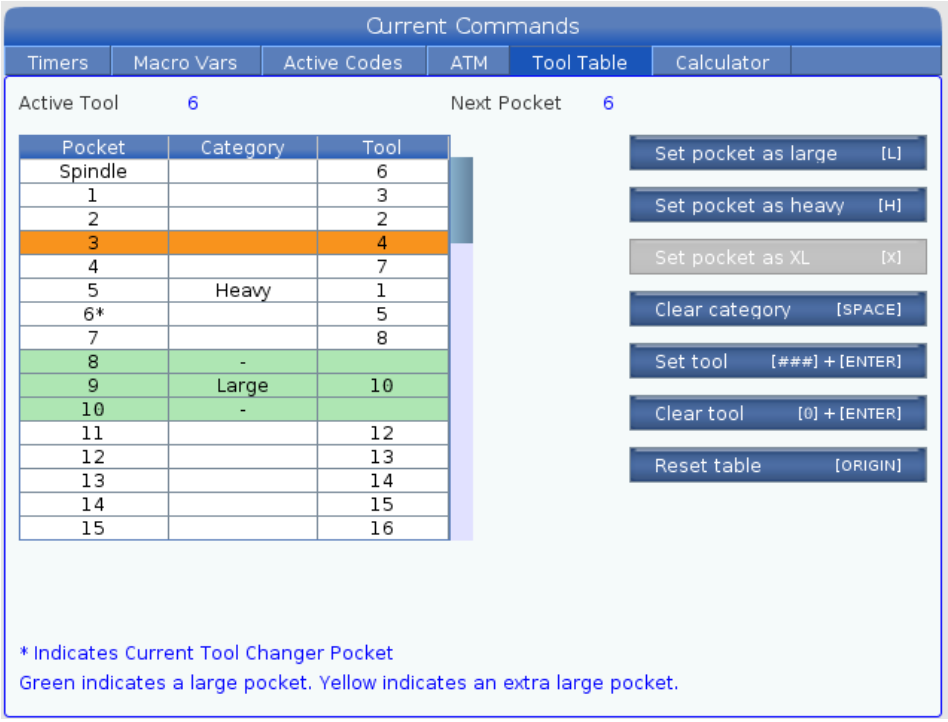
Highlighting the **Active Codes** tab gives the below panel. It may be used for trouble shooting a program. All current alphabet address codes are shown with their values. Below the T and M values are 1000.
This indicates that the machine is using ATM (Advanced Tool Management).



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.



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

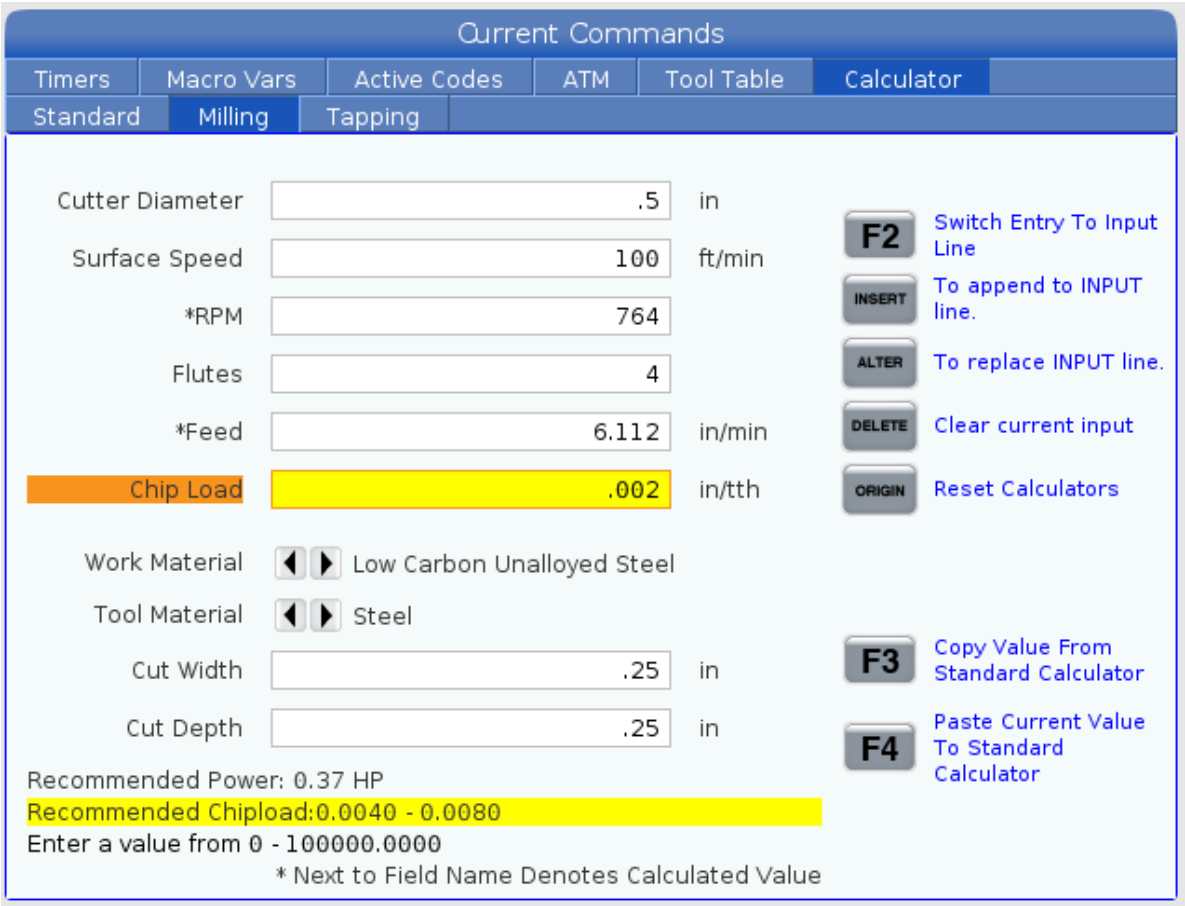


With side mount tool changing machines the pot position of each tool is saved. Above tool number 6 is in the spindle. The asterisk indicates which tool and pocket are staged for the next tool index. In this example it is tool 5 in pocket 6.

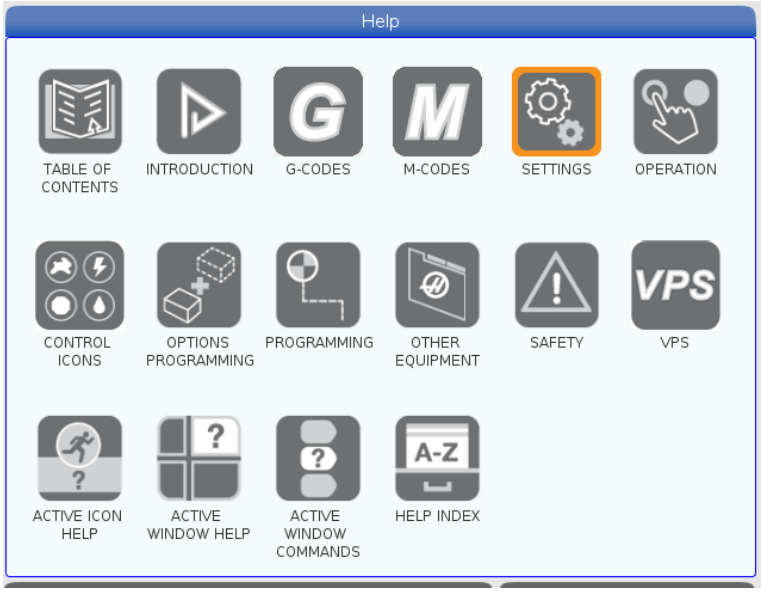
With Cat 40 tools any tool greater than 4 lbs. must be denoted as **Heavy** tool. With the **Heavy** tool designation the machine will only index that particular tool at 25% rapid. This will prevent the tool from inadvertently being thrown. Also the **spindle probe** must be denoted as a heavy tool because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



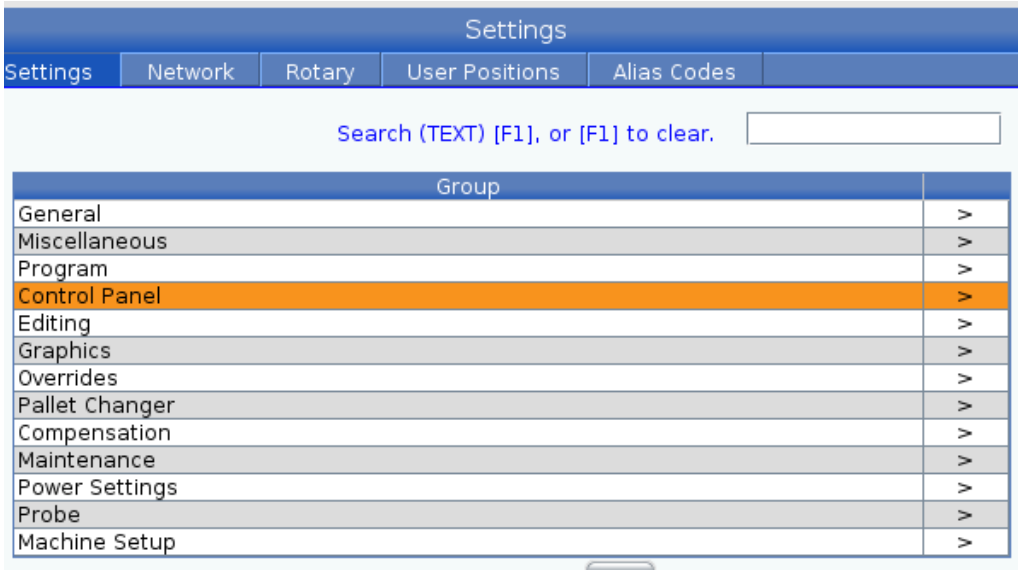
[HELP] key basically leads you to the Operators Manual. It is stored in the control. Use the cursor key to highlight the topic you want more information from. Below SETTINGS is highlighted. Pressing the [ENTER] key will bring up a list of settings and a description. The total number of settings in a Haas is 277.



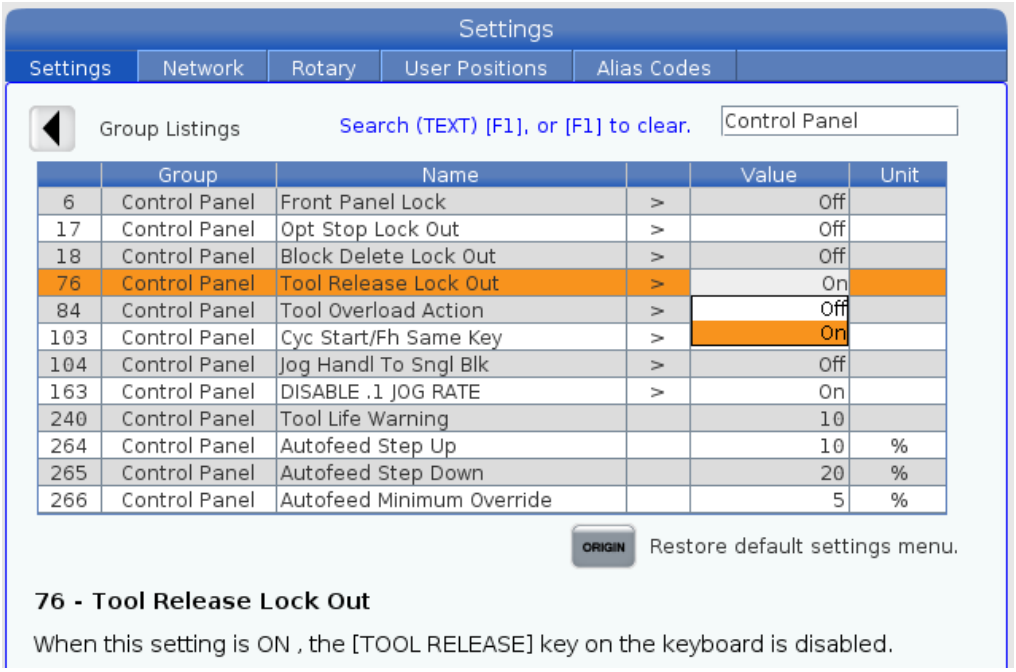
Highlighting Control Icons will give all the different icons which indicate the status of your machine. Below the open lock on the Icon Bar indicates the machine is in the Set Up Mode.

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.



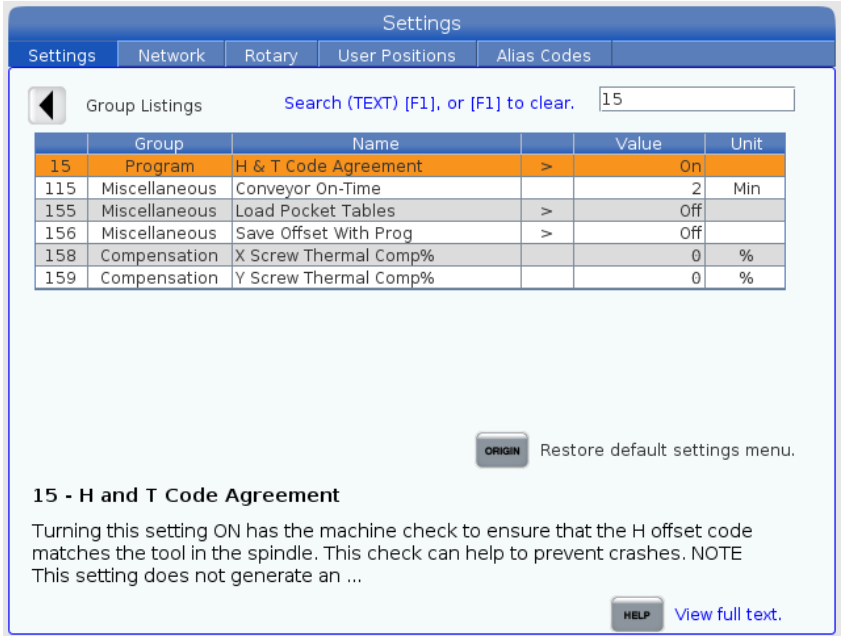
Below setting 76 is highlighted. Pressing the right cursor key [>] gives a drop down. Select one of the options by highlighting it and press [ENTER]. It is recommended that the Tool Release key is locked out if it is not used. Inadvertent pressing this key and the tool in the spindle will fall out.
Also 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.



	Group	Name		Value	Unit
15	Program	H & T Code Agreement	>	On	
115	Miscellaneous	Conveyor On-Time		2	Min
155	Miscellaneous	Load Pocket Tables	>	Off	
156	Miscellaneous	Save Offset With Prog	>	Off	
158	Compensation	X Screw Thermal Comp%		0	%
159	Compensation	Y Screw Thermal Comp%		0	%

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 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. If it is set at 0 the machine will operate in the exact stop mode, slowing speed of machine.

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.)

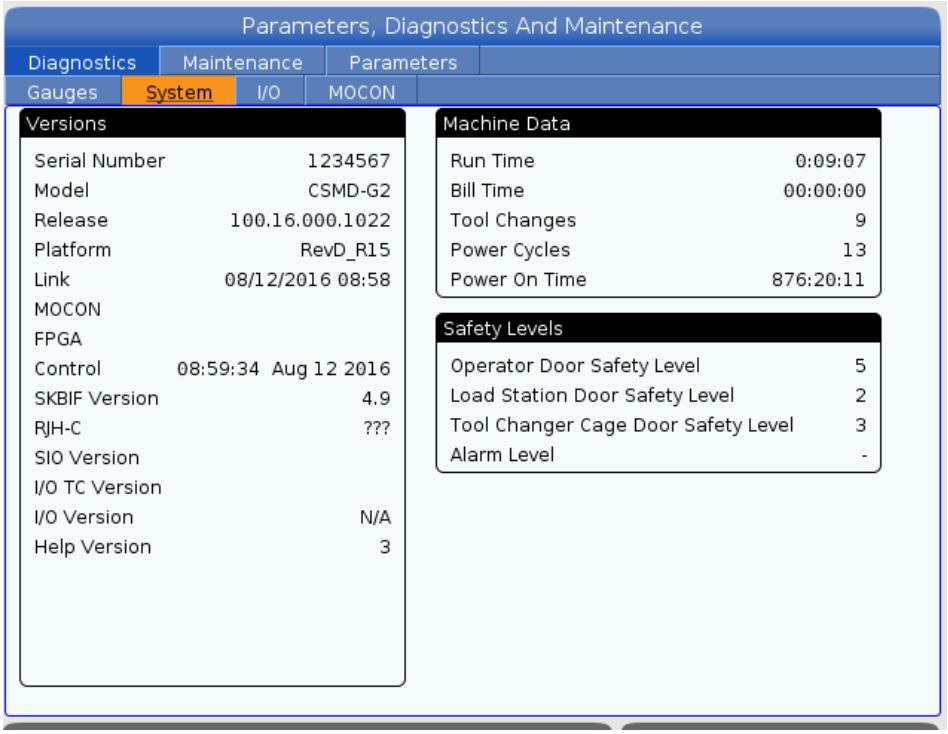
Setting 104 **JOG HANDL TO SNGL BLK** – When running a program in MEM mode in the Program or Graphics display, you can use the SINGLE BLOCK key to cycle through your program one line at a time with each press of the CYCLE START button, when the machine is running or you are in Graphics. If you turn Setting 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.

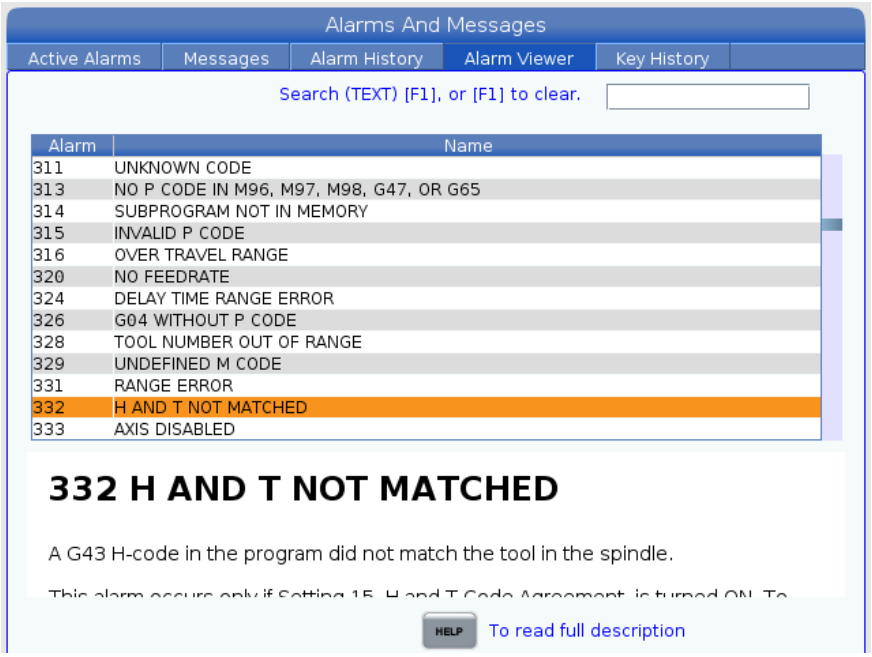
Setting 130 **RIG. TAP RETRACT SPEED** – This feature augments one introduced in version 10.13, the Quick reversal out of a G84 rigid-tapped hole. If it is set to 0 or 1, the machine behaves normally. Setting it to 2 is the equivalent of a G84 command with a J value of 2. That is, the spindle will retract 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)

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%.

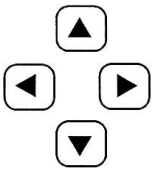
Settings 19, 20, 21 make it possible to disable override keys.

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

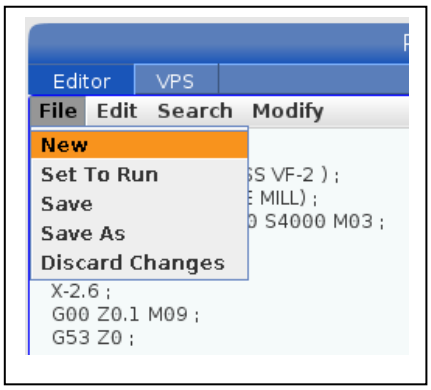
Mode keys set the operational state of the machine tool. Once a mode is set the keys to the right may be used. The current operation mode of the machine is displayed at the top thin pane of the CRT.

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**.**

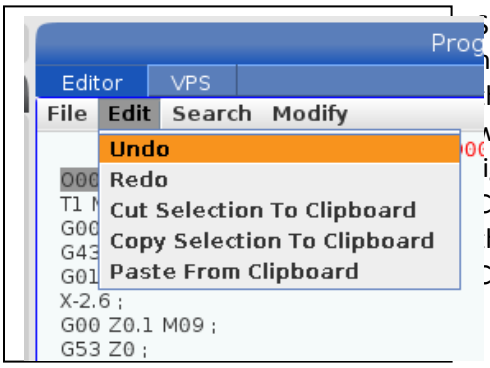
Above is the screen in the edit mode. To edit a word highlight it and then press DELETE, INSERT, ALTER or UNDO keys to change the program. Note on the bottom of the screen the program is not saved until the [MEMORY] key is pressed. Text may be selected to put in the Clipboard by using the [F2] key. [F4]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

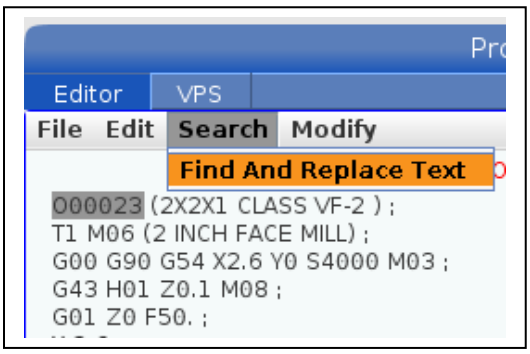


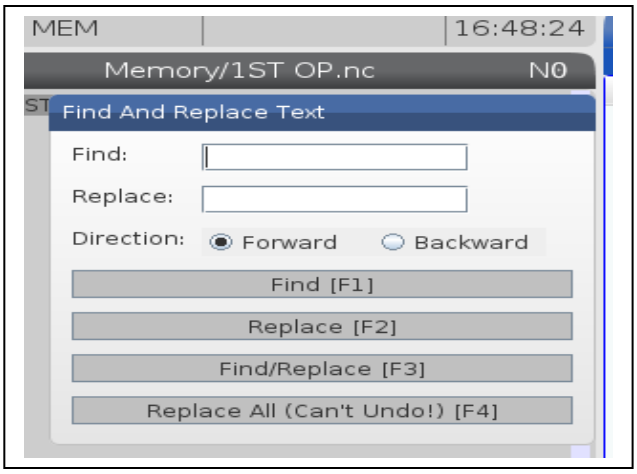
In the File drop down a **New** program may be created by high lighting New and pressing enter. Any editing changes can be saved by lighting **Save** and pressing enter. If a program needs to be duplicated the **Save As** function can be used. The Discard Changes function will undo all the current edits.

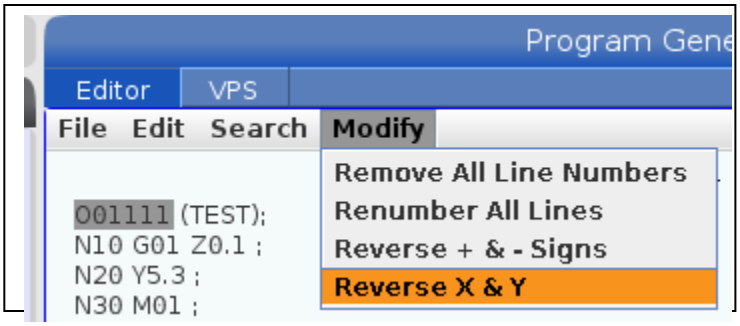


Several lines of code can be selected by using the F2 key. First highlight the first line of code and press F2. Then cursor down to the last line of code that needs to selected and press Enter. This will select several lines of code and they will be highlighted in a light blue. This selected text can 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.







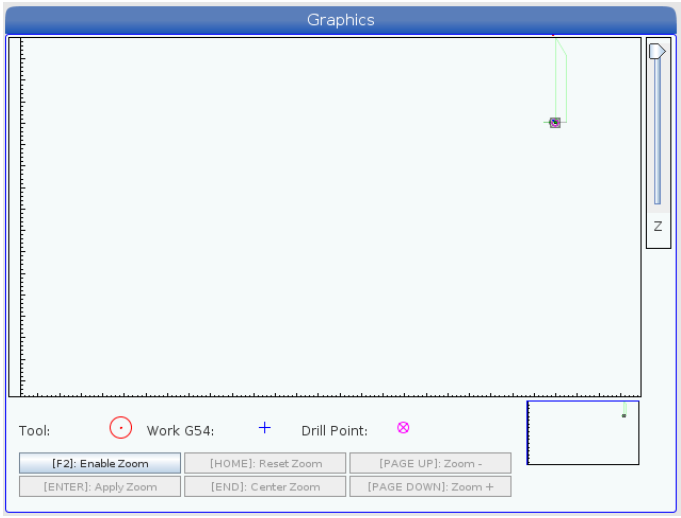
The Modify tab gives the option of putting in block numbers in a program. Also +/- signs and the X and Y may be reversed.

MEM

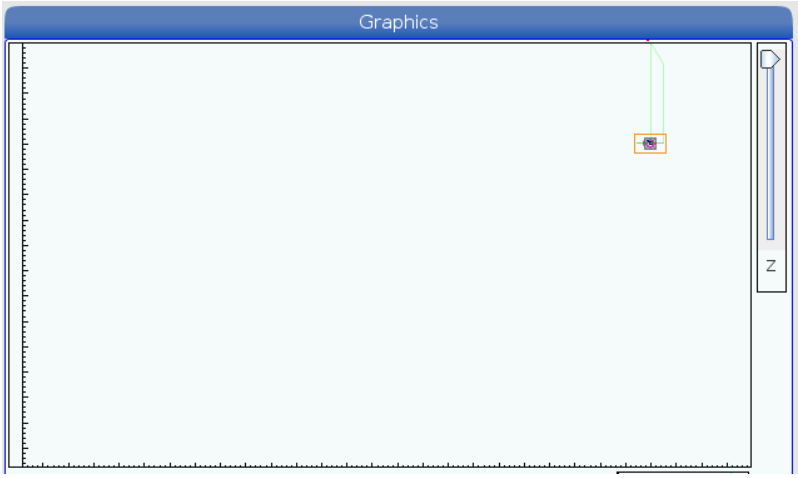
The memory mode is the mode used when running the machine and making a part. The active program is shown in the Program Display Pane. Keys in the memory mode line reflect different ways of running a part in memory. When the keys to the right are depressed they will show up highlighted in black on the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

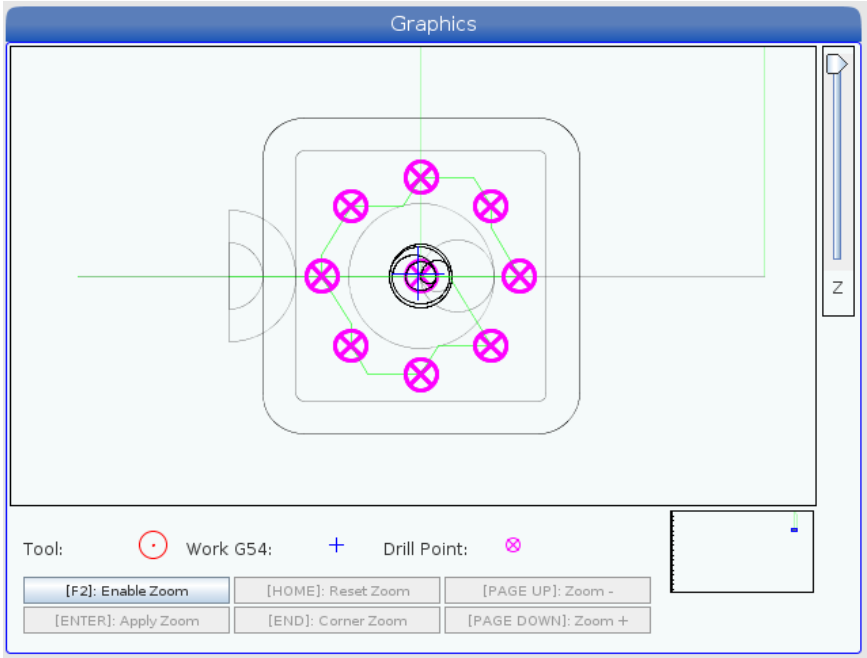
GRAPHICS On older controls the graphics were opened by the Settings/Graphics display key. In NGA controls it is found on the Memory line beneath the [Alter] key. On the NGA controls a separate Graphics key is designated and replaces the DRY RUN key. NGA controls no longer have the Dry Run feature. Graphics 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.



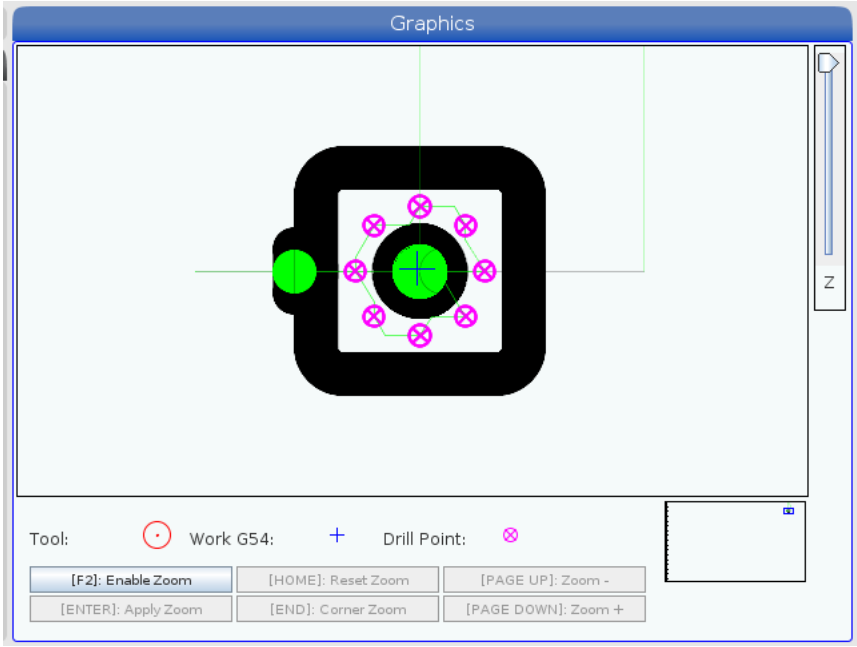
Note the graphics shown is the XY Plane, as if one was looking at the whole table from above. Zoom into the part by pressing [F2]
Then [END] to center and [PAGE DOWN] several times until the box is just barely outside of the part as below.



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 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.

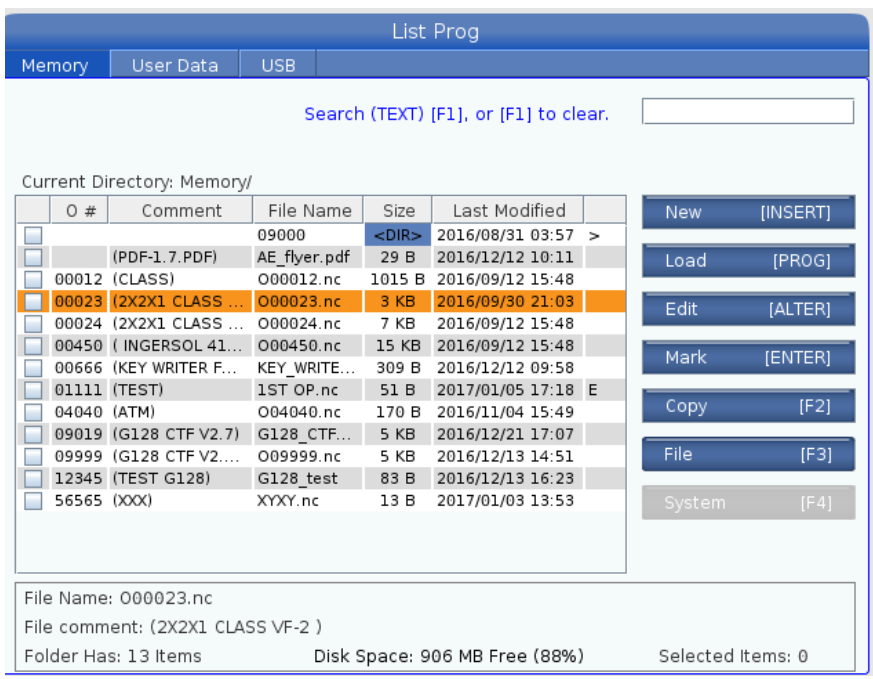
ALL Returns all axes to machine home similar in similar fashion as a Power Up/Restart. First the Z axis then the X and Y axis.

ORIGIN Sets selected displays to zero or other functions.

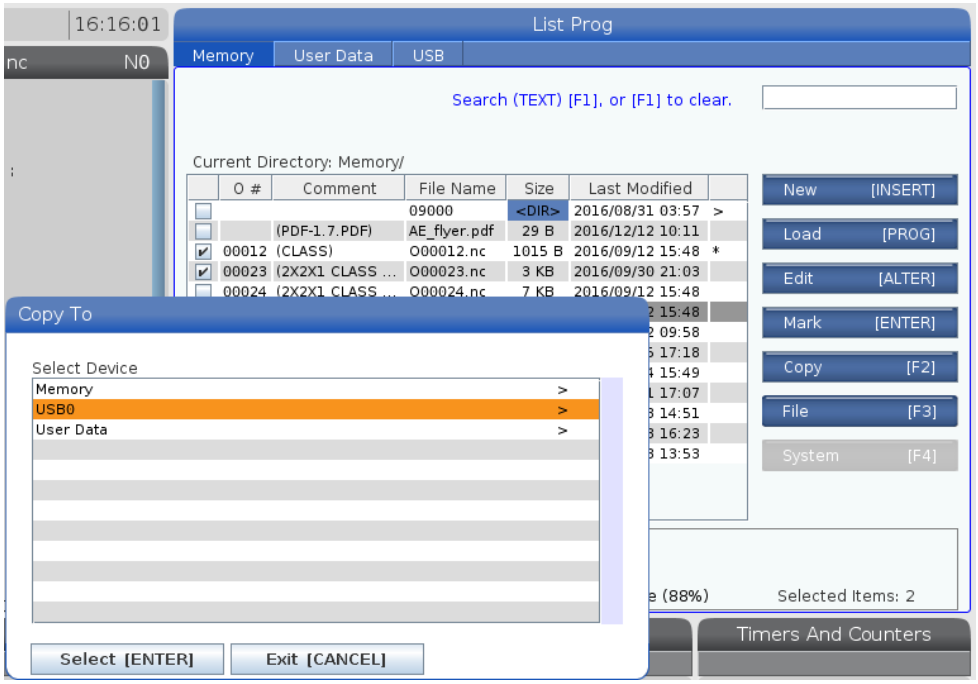
SINGL Returns a single axis to machine home. Select desired axis (X, Y, or Z) then press **Singl** axis button.

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.



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.

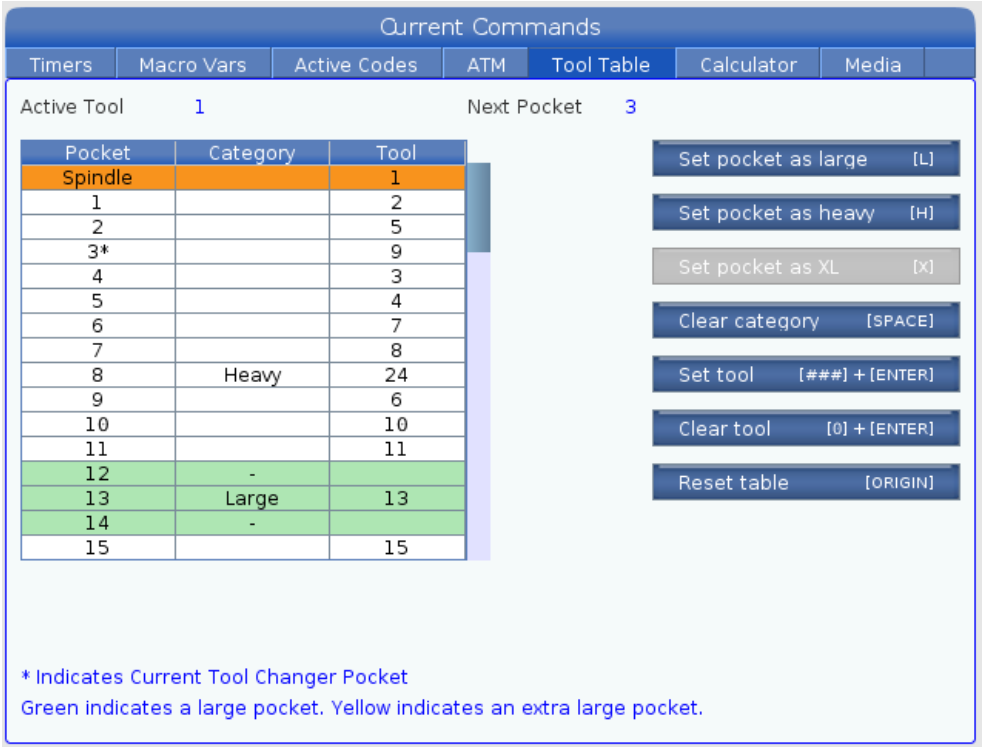


F3 key gives various options such as renaming, deleting, duplication, and others in the List Program page.

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.



Pocket	Category	Tool
Spindle		1
1		2
2		5
3*		9
4		3
5		4
6		7
7		8
8	Heavy	24
9		6
10		10
11		11
12	-	
13	Large	13
14	-	
15		15

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 **OFSET** 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.

Programming

Typical Haas G-Codes:

The definition of "G" code is typically referred to as a "Preparatory Function". They establish the mode of operation that the machine needs to be in to accomplish what the programmer intends. Imagine a rotary switch like that on an older TV; we are just turning the switch to different "modes".

Let's discuss the G Codes that are most commonly used during the setup of a Haas VMC

G00 Rapid traverse motion; Used for positioning and during non-cutting moves.
NOTE: Machine rapids can be up to 1400+ inches per minute (IPM).

G01 Linear interpolation motion; used for actual machining and metal removal. Governed by programmed feed rate

G02 Circular interpolation – Clockwise

G03 Circular interpolation – Counterclockwise

G28 Machine home (Rapid traverse)

G40 Cutter compensation cancel

G41 Cutter compensation to **left** of path.

G42 Cutter compensation to **right** of path.

G43 Read tool length compensation.

G54 Work coordinate #1 (Part zero)

G80 Drilling Cycle Cancel

G81 Drill canned cycle

G82 Spot drill canned cycle.

G83 Peck drill canned cycle.

G84 Tapping canned cycle.

G90 Absolute Positioning

G91 Incremental Positioning

G98 Make all drilling X&Y moves at the Initial Plane (H Line Height)

G99 Make all drilling X&Y moves at the Rapid Plane (R Value in Canned Cycle)

Typical Haas M Codes:

M Codes are used by the programmer to turn on and off certain functions of the machine. Think of M codes as codes that turn on and off different systems of the machine.

M00 Program Stop. The machine will wait for the operator to push Cycle Start

M01 Optional program stop. The machine will stop if the OPT STOP mode is turned on.

M03 Start spindle forward (Clockwise). Must have a spindle speed

M04 Start spindle reverse (Counterclockwise). Must have a spindle speed

M05 Spindle stop

M06 Tool change command. Must have a T value on the same line (Tool Number)

M08 Coolant **ON** command

M09 Coolant **OFF** command

M88 High Pressure Coolant On (Typically Thru Spindle Coolant)

M89 High Pressure Coolant Off

M30 Program end and rewind to beginning of program

M97 Program Jump – Jumps to a specific line number (M97 P100, Jump to Line N100)

M98 Subprogram call – Jump to another program

M99 Subprogram return, or loop. Used to tell the machine to go back from a Sub Prog.

NOTE: Only one "M" code can be used per line. The "M" code will be the last item of code to be performed, regardless of where it is located in the line.

Alphabetical Address Codes

The following is a list of the Address Codes used in programming the CNC.

A FOURTH AXIS ROTARY MOTION

The **A** address character is used to specify motion for the optional fourth, **A,** axis. It specifies an angle in degrees for the rotary axis. It is always followed by a signed number and up to three fractional decimal positions. If no decimal point is entered, the last digit is assumed to be 1/1000 degrees. The smallest magnitude is 0.001 degrees, the most negative value is -8380.000 degrees, and the largest number is 8380.000 degrees.

B FIFTH AXIS ROTARY MOTION

The **B** address character is used to specify motion for the optional fifth, **B,** axis. It specifies an angle in degrees for the rotary axis. It is always followed by a signed number and up to three fractional decimal positions. If no decimal point is entered, the last digit is assumed to be 1/1000 degrees. The smallest magnitude is 0.001 degrees, the most negative value is -8380.000 degrees, and the largest number is 8380.000 degrees.

C AUXILIARY EXTERNAL ROTARY AXIS

The **C** address character is used to specify motion for the optional external sixth, C, axis. It specifies an angle in degrees for the rotary axis. It is always followed by a signed number and up to three fractional decimal positions. If no decimal point is entered, the last digit is assumed to be 1/1000 degrees. The smallest magnitude is 0.001 degrees, the most negative value is -8380.000 degrees, and the largest number is 8380.000 degrees.

D TOOL DIAMETER SELECTION

The **D** address character is used to select the tool diameter or radius used for cutter compensation. The number following must be between 0 and 100. D0 specifies that the tool size is zero and serves to cancel a previous On. Any other value of B selects the numbered entry from the tool diameter/radius list under the Offsets display.

E CONTOURING ACCURACY

The **E** address character is used, with G187, to select the accuracy required when cutting a corner during high speed machining operations. The range of values possible for the E code is 0.0001 to 0.25. Refer to the "Contouring Accuracy" section for more information.

F FEED RATE

The **F** address character is used to select the feed rate applied to any interpolation functions, including pocket milling and canned cycles. It is either in inches per minute with four fractional positions or mm per minute with three fractional positions.

G PREPARATORY FUNCTIONS (G CODES)

The **G** address character is used to specify the type of operation to occur in the block containing the G code. The G is followed by a two or three digit number between 0 and 187. Each G code defined in this control is part of a group of G codes. The Group 0 codes are non-modal; that is, they specify a function applicable to this block only and do not affect other blocks. The other groups are modal and the specification of one code in the group cancels the previous code applicable from that group. A modal G code applies to all subsequent blocks so those blocks do not need to re-specify the same G code. More than one G code can be placed in a block in order to specify all of the setup conditions for an operation. See "G Codes" section.

H TOOL LENGTH OFFSET SELECTION

The H address character is used to select the tool length offset entry from the offsets memory. The H is followed by a two digit number, between 0 and 100. H0 will cause no offset to be used and Hn will use the tool length entry n from the Offsets display. Note that G49 is the default condition disabling tool length offsets; so you must also select either G43 or G44 for tool offsets to work. The TOOL OFSET MESUR button will enter a value into the offsets to correspond to the use of G43.

I CANNED CYCLE AND CIRCULAR OPTIONAL DATA

The **I** address character is used to specify data used for some canned cycles and circular motions. It is either in inches with four fractional positions or mm with three fractional positions. It is followed by a signed number in inches between -8380.000 and 8380.000 for inches or between -83800.00 and 83800.00 for metric.

J CANNED CYCLE AND CIRCULAR OPTIONAL DATA

The **J** address character is used to specify data used for some canned cycles and circular motions. It is formatted just like the **I** data.

K CANNED CYCLE AND CIRCULAR OPTIONAL DATA

The **K** address character is used to specify data used for some canned cycles and circular motions. It is formatted just like the **I** data.

L LOOP COUNT FOR REPEATED CYCLES

The **L** address character is used to specify a repetition count for some canned cycles and auxiliary functions. It is followed by an unsigned number between 0 and 32767.

M M CODE MISCELLANEOUS FUNCTIONS

The M address character is used to specify an M code for a block. These codes are used to control miscellaneous machine functions. Note that only one M code is allowed per block of the CNC program and all M codes are performed at the end of the block. See the “M Codes" section for a detailed list of M codes.

N NUMBER OF BLOCK

The **N** address character is entirely optional. It can be used to identify or number each block of a program. It is followed by a number between 0 and 99999. The M97 functions may reference an **N** line number.

O PROGRAM NUMBER/NAME

The O address character is used to identify a program. It is followed by a number between 0 and 9999. A program saved in memory always has a **Onnnn** identification in the first block; it cannot be deleted. Altering the O in the first block causes the program to be renamed. An **Onnnn** can be placed in other blocks of a program but will have no effect and can be confusing to the reader. A colon (:) may be used in the place of **O,** but is always displayed as "**O**".

P DELAY TIME OR PROGRAM NUMBER

The **P** address character is used to enter either a time in seconds or a program number for a subroutine call. If it is used as a time (for a G04 dwell) or a program name (for a M98), the value may be either a positive number without decimal point up to 9999. If it is used as a time, it may be a positive decimal with fraction between 0.001 and 1000.0.

Q CANNED CYCLE OPTIONAL DATA

The **Q** address character is used in canned cycles and is followed by a signed number in inches between -8380.000 and 8380.000 for inches or between -83800.00 and 83800.00 for metric.

R CANNED CYCLE AND CIRCULAR OPTIONAL DATA

The **R** address character is used in canned cycles and circular interpolation. It is either in inches with tour fractional positions or mm with three fractional positions. It is followed by a signed number in inches between
-8380.000 and 8380.000 for inches or between -83800.00 and 83800.00 for metric. It is usually used to define the reference plane for canned cycles.

S SPINDLE SPEED COMMAND

The **S** address character is used to specify the spindle speed in conjunction with M41 and M42. The **S** is followed by an unsigned number between 1 and 99999. The **S** command does not turn the spindle on or off; it only sets the desired speed. If a gear change is required in order to set the commanded speed, this command will cause a gear change to occur even if the spindle is stopped. If the spindle is running, a gear change operation will occur and the spindle will continue running at the new speed.

T TOOL SELECTION CODE

The **T** address character is used to select the tool for the next tool change. The number following must be a positive number between 1 and the number in Parameter 65. It does not cause the tool change operation to occur. The **Tn** may be placed in the same block that starts the tool change (M6 or M16) or in any previous block.

U AUXILIARY EXTERNAL LINEAR AXIS

The **U** address character is used to specify motion for the optional external linear, **U,** axis. It specifies a position of motion in inches. It is always followed by a signed number and up to four fractional decimal positions. If no decimal point is entered, the last digit is assumed to be 1/10000 inches. The smallest magnitude is 0.0001 inches, the most negative value is -838.0000 inches, and the largest number is 838.0000 inches.

V AUXILIARY EXTERNAL LINEAR AXIS

The **V** address character is used to specify motion for the optional external linear, V, axis. It specifies a position of motion in inches. It is always followed by a signed number and up to four fractional decimal positions. If no decimal point is entered, the last digit is assumed to be 1/10000 inches. The smallest magnitude is 0.0001 inches, the most negative value is -838.0000 inches, and the largest number is 838.0000 inches.

W AUXILIARY EXTERNAL LINEAR AXIS

The **W** address character is used to specify motion for the optional external linear, W, axis. It specifies a position of motion in inches. It is always followed by a signed number and up to four fractional decimal positions. If no decimal point is entered, the last digit is assumed to be 1/10000 inches. The smallest magnitude is 0.0001 inches, the most negative value is -838.0000 inches, and the largest number is 838.0000 inches.

X LINEAR X-AXIS MOTION

The **X** address character is used to specify motion for the X-axis. It specifies a position or distance along the X-axis. It is either in inches with four fractional positions or mm with three fractional positions. It is followed by a signed number in inches between -8380.000 and 8380.000 for inches or between -83800.00 and 83800.00 for metric. If no decimal point is entered, the last digit is assumed to be 1/10000 inches or 1/1000 mm.

Y LINEAR Y-AXIS MOTION

The **Y** address character is used to specify motion for the Y-axis. It specifies a position or distance along the Y-axis. It is either in **inches** with four fractional positions or **mm** with three fractional positions. It is followed by a signed number in inches between -8380.000 and 8380.000 for inches or between **-** 83800.00 and 83800.00 for metric. If no decimal point is entered, the last digit is assumed to be 1/10000 inches or 1/1000 mm.

Z LINEAR Z-AXIS MOTION

The **Z** address character is used to specify motion for the Z-axis. It specifies a position or distance along the Z-axis. It is either in **inches** with four fractional positions or **mm** with three fractional positions. It is followed by a signed number in inches between -8380.000 and 8380.000 for inches or between -83800.00 and 83800.00 for metric. If no decimal point is entered, the last digit is assumed to be 1/10000 inches or 1/1000 mm.

Machine Defaults

A default is an automatic function of the machine tool control. When powering up the machine, the machine needs to assume some sort of “mode”. Upon following a power-up procedure, the machine will be sitting in the following modes:

G00	Rapid traverse
G17	X,Y Circular plane selection
G40	Cutter Compensation cancel
G49	Tool length compensation cancel
G54	Work Coordinate zero #1 (1 of 26 available)
G64	Exact stop cancel
G80	Canned cycle cancel
G90	Absolute programming
G98	Initial point return

There is no default FEED RATE (F code), but once an F code is programmed, it will apply until another is entered or the machine is turned off.

Preparatory Functions (G Codes)

The following is a summary of the G codes, A " * " indicates the default within each group, if there is one:

Code:	Group:	Function:
G00	*01	Rapid Motion
G01	01	Linear Interpolation Motion
G02	01	CW Interpolation Motion
G03	01	CCW Interpolation Motion
G04	00	Dwell
G09	00	Exact Stop
G10	00	Programmable Offset Setting
G12	00	CW Circular Pock Milling (Yasnac)
G13	00	CCW Circular Pock Milling (Yasnac)
G17	*02	XY Plane Selection
G18	02	ZX Plane Selection
G19	02	YZ Plane Selection
G20	06	Inch Programming Selection
G21	06	Metric Programming Selection
G28	00	Return to Machine Zero
G29	00	Move to Location Through G29 Reference
G31	00	Skip Function
G35	00	Automatic Tool Diameter Measurement
G36	00	Automatic Work Offset Measurement
G37	00	Automatic Tool Length Measurement
G40	*07	Cutter Comp Cancel
G41	07	Cutter Compensation Left
G42	07	Cutter Compensation Right
G43	08	Tool Length Compensation +
G44	08	Tool Length Compensation -
G47	00	Engraving
G49	*08	G43/G44 Cancel
G50	11	G51 Cancel
G51	11	Scaling
G52	12	Select Work Coordinate System G52 (Yasnac)
G52	00	Set Local Coordinate System (Fanuc)
G52	00	Set Local Coordinate System (HAAS)
G53	00	Non-Modal Machine Coordinate Selection
G54	*12	Select Work Coordinate System l
G55	12	Select Work Coordinate System 2
G56	12	Select Work Coordinate System 3
G57	12	Select Work Coordinate System 4

Preparatory G-Codes (continued)

G58	12	Select Work Coordinate System 5
G59	12	Select Work Coordinate System 6
G60	00	Unidirectional Positioning
G61	13	Exact Stop Modal
G64	*13	G61 Cancel
G65	00	Macro Subroutine Call
G68	16	Rotation
G69	16	G68 Cancel
G70	00	Bolt Hole Circle (Yasnac)
G71	00	Bolt Hole Arc (Yasnac)
G72	00	Bolt Holes Along an Angle (Yasnac)
G73	09	High Speed Peck Drill Canned Cycle
G74	09	Reverse Tap Canned Cycle
G76	09	Fine Boring Canned Cycle
G77	09	Back Bore Canned Cycle
G80	*09	Canned Cycle Cancel
G81	09	Drill Canned Cycle
G82	09	Spot Drill Canned Cycle
G83	09	Peck Drill Canned Cycle
G84	09	Tapping Canned Cycle
G85	09	Boring Canned Cycle
G86	09	Bore/Stop Canned Cycle
G87	09	Bore/Manual Retract Canned Cycle
G88	09	Bore/Dwell Canned Cycle
G89	09	Bore Canned Cycle
G90	*03	Absolute
G91	03	Incremental
G92	00	Set Work Coordinates - FANUC or HAAS
G92	00	Set Work Coordinates - YASNAC
G98	*10	Initial Point Return
G99	10	R Plane Return
G100	00	Disable Mirror Image
G101	00	Enable Mirror Image
G102	00	Programmable Output To RS-232
G103	00	Block Look ahead Limit

Preparatory G-Codes (continued)

G110	12	Select Coordinate System 7
G111	12	Select Coordinate System 8
G112	12	Select Coordinate System 9
G113	12	Select Coordinate System 10
G114	12	Select Coordinate System 11
G115	12	Select Coordinate System 12
G116	12	Select Coordinate System 13
G117	12	Select Coordinate System 14
G118	12	Select Coordinate System 15
G119	12	Select Coordinate System 16
G120	12	Select Coordinate System 17
G121	12	Select Coordinate System 18
G122	12	Select Coordinate System 19
G123	12	Select Coordinate System 20
G124	12	Select Coordinate System 21
G125	12	Select Coordinate System 22
G126	12	Select Coordinate System 23
G127	12	Select Coordinate System 24
G128	12	Select Coordinate System 25
G129	12	Select Coordinate System 26
G136	00	Automatic Work Offset Center Measurement
G150	00	General Purpose Pocket Milling
G154 P1-P99	12	Replaces G110-G129 on newer machines
G187	00	Accuracy Control for High Speed Machining

Each **G** code defined in this control is part of a group of **G** codes. The Group 0 codes are non-modal; that is, they specify a function applicable to this block only and do not affect other blocks. The other groups are modal and the specification of one code in the group cancels the previous code applicable from that group. A modal **G** code applies to all subsequent blocks so those blocks do not need to recall the same **G** code.

There is also one case where the Group 01 **G** codes will cancel the Group 9 (canned cycles) codes. If a canned cycle is active (G73 through G89), the use of G00 or G01 will cancel the canned cycle.

RAPID POSITION COMMANDS (G00)

G00 **RAPID MOTION POSITIONING** **GROUP 01**

X Optional X-axis motion command
Y Optional Y-axis motion command
Z Optional Z-axis motion command
A Optional A-axis motion command

This **G** code is used to cause a rapid traverse of the three or four axes of the machine. The auxiliary axes **B, C, U, V,** and **W** can also be moved with a G00. This **G** code is modal so that a previous block with G00 causes all following blocks to be rapid motions until another Group 01 code is specified. The rapid traverse rate is dependent on the maximum speed possible for each axis independently as modified by the RAPID override keys.

Generally, rapid motions will not be in straight lines. All of the axes specified are moved at the same time but will not necessarily complete their motions at the same time. The control will wait until all motions are complete. Only the axes specified are moved and the incremental or absolute modal conditions (G90 or G91) will change how those values are interpreted. Parameter 57 can change how closely the machine waits for a precise stop before and after a rapid move.

LINEAR & CIRCULAR INTERPOLATION COMMANDS (G01, G02)

G01 LINEAR INTERPOLATION MOTION GROUP 01

F	Feed rate in inches (mm) per minute
X	Optional X-axis motion command
Y	Optional Y-axis motion command
Z	Optional Z-axis motion command
A	Optional A-axis motion command

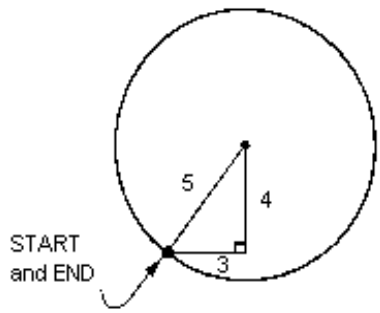
This **G** code provides for straight line (linear) motion from point to point. Motion can occur in 1, 2 or 3 dimensions. All axes will start and finish motion at the same time. The rotary axis may also be commanded and this will provide a helical motion. The speed of all axes is controlled so that the feed rate specified is achieved along the actual path. Rotary axis speed is dependent on the rotary axis diameter setting (Setting 34) and will provide a helical motion. The **F** command is modal and may be specified in a previous block. Only the axes specified are moved and the incremental or absolute modal conditions (G90 or G91) will change how those values are interpreted. The auxiliary axes **B, C, U, V,** and **W** can also be moved with a G01 but only one axis is moved at a time.

G02 CW CIRCULAR INTERPOLATION MOTION GROUP 01

F	Feed rate in inches (mm) per minute
I	Optional distance along X-axis to center of circle
J	Optional distance along Y-axis to center of circle
K	Optional distance along Z-axis to center of circle
R	Optional radius of circle
X	Optional X-axis motion command
Y	Optional Y-axis motion command
Z	Optional Z-axis motion command
A	Optional A-axis motion command

This **G** code is used to specify a clockwise circular motion of two of the linear axes. Circular motion is possible in any two of **X, Y,** and **Z** axes as selected by G17, G18, and G19. The **X, Y,** and **Z** are used to specify the end point of the motion that can use either absolute (G90) or incremental (G91) motion. If any of the **X, Y,** or **Z** for the selected plane is not specified, the endpoint of the arc is the same as the starting point for that axis. There are two ways to specify the center of the circular motion; the first uses **I**, **J,** or **K** to specify the distance from the starting point to the center of the arc; the second uses **R** to specify the radius of the arc. These are further described below:

I, J, K: When **I, J,** or **K** are used to specify the center of the arc, **R** may not be used. Only the **I, J,** or **K** specific to the selected plane (IJ for G17, IK for G18, JK for G19) are allowed. If only one of the **I**, **J, K** is specified, the others are assumed to be zero. The **I**, **J,** or **K** is the signed distance from the starting point to the center of the circle. Small errors in these values are tolerated up to 0.0010 inches. Use of **I**, **J,** or **K** is the only way to cut a complete 360 degree arc; in this case, the starting point is the same as the ending point and no **X, Y,** or **Z** is needed.

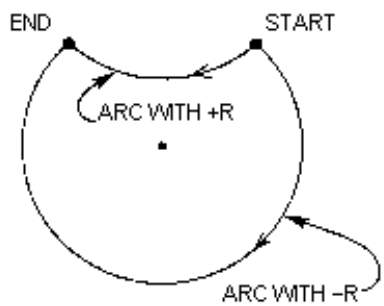


To cut a complete circle of 360 degrees (360°), you do not need to specify an ending point **X, Y,** or **Z**; just program **I, J,** or **K** to define the center of the circle. The following line will cut a complete circle:

G02 I3.0 J4.0 (Assumes G17; XY plane)

In cases where you are cutting less than a complete circle, it is much easier to use **R** instead of **I**, **J, K.**

R: When **R** is used to specify the center of the circle, a complete 360 degree arc is not possible. **X, Y,** or **Z** is required to specify an endpoint different from the starting point. **R** is the distance from the starting point to the center of the circle. With a positive **R,** the control will generate a path of 180 degrees or less; to generate an angle of over 180 degrees, specify a negative **R.** Small errors in this-value are tolerated up to 0.0010 inches.



The following line will cut an arc less than 180 degrees (180°):

G01 X3.0 Y4.0
G02 X-3.0 R5.0

and the following line will cut an arc of more than 180 degrees (180°):

G01 X3.0 Y4.0
G02 X-3.0 R-5.0

MISCELLANEOUS G-CODES (G04, G03)

DWELL (G04)

P: Dwell time in seconds or milliseconds.

G04 Is used to cause a delay or dwell in the program. The block containing **G04** will delay for the time specified in the **P** code. When programmed on a line following some motion such as G00, G01, G02 and G03 all motion will be stopped for the amount of time specified in the **P** command in seconds. If the **P** has no fraction part, the delay is in milliseconds (0.001 seconds); otherwise the delay is in seconds.

The slide motion is stopped, but the spindle will continue to rotate at the requested RPM and the coolant stays on.

G04 P____ Minimum value – **P.001** of a second. Maximum value – **P.1000.0** seconds

G03 CCW CIRCULAR INTERPOLATION MOTION GROUP 01

G03 will generate counterclockwise circular motion but is otherwise the same as G02.

HELICAL INTERPOLATION

A helical motion is possible with G02 or G03 by programming the linear axis that is not in the selected plane. This third axis will be interpolated along the specified axis in a linear manner while the other two axes will be moved in the circular motion. The speed of each axis will be controlled so that the helical rate matches the programmed feed rate.

The length of the third axis motion may not be greater than the length of the motion of the two axes for the circular motion. This means that for a complete revolution around a one-inch diameter, the circumference will be 3.1416 and the third axis motion may not be more than 3.1416 inches.

CIRCULAR POCKET MILLING (G12, G13)

There are two G codes that will provide for pocket milling of a circular shape. They are different only in which direction of rotation is used.

G12 CIRCULAR POCKET MILLING CLOCKWISE GROUP 00

*D	Tool Radius or Diameter Selection
I	Radius of First Circle (Or Finish If No K)
K	Radius of Finished Circle (if Specified)
L	Loop count for repeating deeper cuts
Q	Radius Increment (Must Be Used With K)
F	Feed Rate in inches (mm) per minute
Z	Z depth of cut or increment

***In order to get the exact programmed circle diameter, the control uses the selected D code tool size. If this compensation is not desired, program D0.**

This **G** Code implies the use of G42.

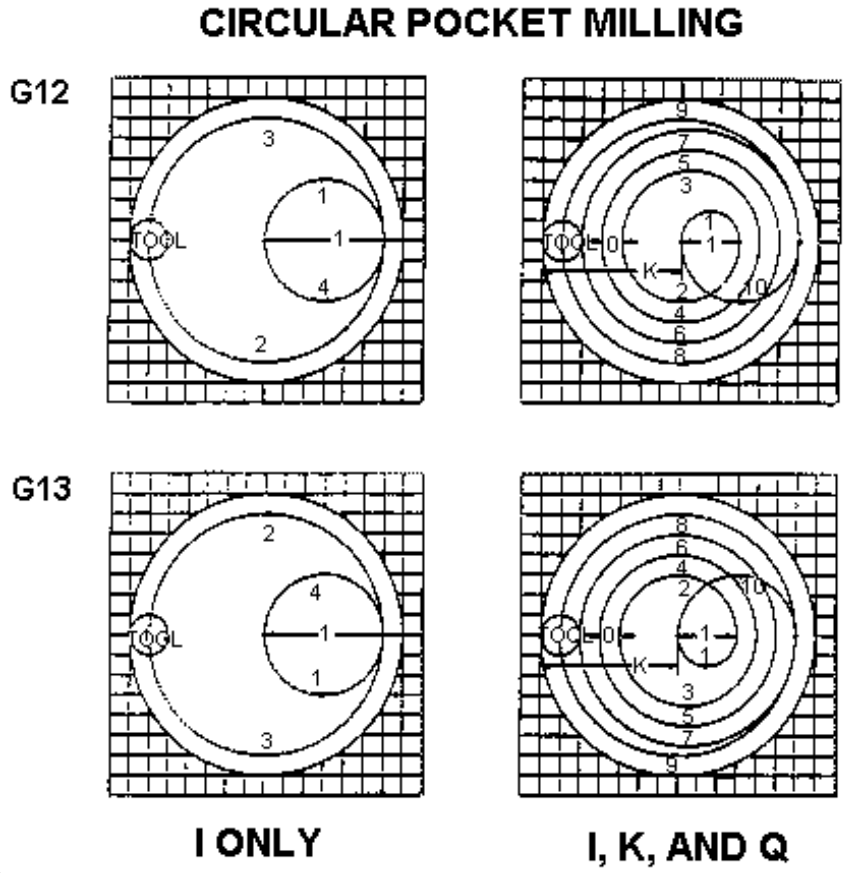
The tool must be positioned at the center of the circle either in a previous block or in this block using **X** and **Y.** The cut is performed entirely with circular motions of varying radius. G12 belongs to Group zero and thus is non-modal. If G91 (incremental) is specified and an **L** count is included, the **Z** increment is repeated **L** times at the **F** feed rate. If no **K** is specified, the center of the cut is removed completely.

G13 CIRCULAR POCKET MILLING COUNTERCLOCKWISE GROUP 00

This **G** Code implies the use of G41 and is otherwise similar to G12. G13 belongs to Group 00 and thus is non-modal.

```
%
00100 (SAMPLE G12 AND G13)
(OFFSET D01 SET TO APPROX. TOOL SIZE)
(TOOL MUST BE MORE THAN 0.3 IN DIAM.)
G54 G00 G90 Z-1. X0. Y0.
S2000 M03
G12 I1.5 F10. Z-1.2 D01
G28
G55 Z-1. X0. Y0.
G12 I0.3 Kl.5 QO.3 F10. Z-1.2 D01
G28
G56 Z-1. X0. Y0.
G13 I1.5 F10. Z-1.2 D01
G28
G57 Z-1. X0. Y0.
G13 I0.3 Kl.5 QO.3 F10. Z-1.2 D01
G28 M30
```

Example of Circular Pocket Milling



%

REFERENCE POINT DEFINITION AND RETURN (G28)

G28 RETURN TO MACHINE ZERO, SET OPTIONAL G29 REFERENCE POINT GROUP 00

The G28 code is used to return to the machine zero position on all axes. If an **X, Y, Z,** or **A** axis is specified on the same **block,** only those axes will move and return to the machines' zero point. If **X, Y, Z,** or **A** specifies a different location than the current position, then the movement to machine zero will be through the specified point. This point is called the G29 reference point and is saved for use in G29. If no **X, Y, Z,** or **A** is specified, all axes will be moved directly to machine zero. Any auxiliary axes **(B, C,...)** are returned to home after the **X, Y, Z,** and **A** axes. G28 also cancels tool length offsets.

CUTTER COMPENSATION (G40, G41, G42) G43

Cutter Compensation - This feature allows the programmer to forget about the cutting tools radius or diameter during programming. Like all forms of compensation, it makes programming easier, since the programmer need not be concerned with the exact cutter diameter while the program is being prepared. Cutter compensation also allows the radius of the cutting tool to be adjusted without any modification to the program.

G40 CUTTER COMP CANCEL GROUP 07

G40 will cancel the G41 or G42 cutter compensation. Programming a D00 will also cancel cutter compensation.

G41 CUTTER COMPENSATION LEFT GROUP 07

G41 will select cutter compensation left; that is the tool is moved to the left of the programmed path to compensate for the size of the tool. A Dnn must also be programmed to select the correct tool size from compensation memory. If compensation memory contains a negative value for cutter size, cutter compensation will operate as though G42 was specified.

G42 CUTTER COMPENSATION RIGHT GROUP 07

G42 will select cutter compensation right; that is the tool is moved to the right of the programmed path to compensate for the size of the tool. A 0nn must also be programmed to select the correct tool size from compensation memory. If compensation memory contains a negative value for cutter size, cutter compensation will operate as though G41 was specified.

TOOL LENGTH COMPENSATION (G43)

G43 TOOL LENGTH COMPENSATION + (PLUS) GROUP 08

This code selects tool length compensation in a positive direction. That is; the tool length offsets are added to the commanded axis positions. A nonzero Hnn must be programmed to select the correct entry from offsets memory. The automatically entered offsets using the TOOL OFSET MESUR key assume that G43 is being used.

ENGRAVING (G47)

G47 ENGRAVING GROUP 00

E	=	Plunge rate (units/min)
F	=	Engraving feed rate (units/min)
I	=	Angle of rotation (-360. to +360.), default is 0.
J	=	Scaling factor in inches (minimum = 0.001 inches), default is 1.0 inch
P	=	l for Sequential Serial Number Engraving
	=	0 for Literal String Engraving
R	=	Return plane
X	=	X start of engraving
Y	=	Y start of engraving
Z	=	Depth of cut

The text to engrave should be in the form of a comment on the same line as G47, with either a P1 or P0 before it. P1 selects Sequential Serial Number Engraving and P0 selects Literal String Engraving.

SEQUENTIAL SERIAL NUMBER ENGRAVING

This method is used to engrave numbers on a series of parts, with the number being incremented by one each time. The '#' symbol is used to select the number of digits in the serial number. For example:

G47 P1 (####) Will limit the serial number to four digits.

The initial serial number can be either programmed or set manually. If it is programmed, for example:

G47 P1 (1234) Will set the initial serial number to "1234".

The initial serial number can also be set manually into a macro variable. The "MACROS" option does not have to be enabled to do this. Macro variables are temporary storage locations for numbers. Macro variable #599 is used to hold the initial serial number to be engraved. To set this variable, go to the CURNT COMDS page and press the PAGE DOWN key until the "Macro Variables" page appears. Then type in "599" and press the (DOWN ARROW). Now enter the desired initial serial number at the cursor and press the WRITE key. For example, when macro variable #599 is set to "1234",

G47 P1 (####) Will produce this:

1234

If the number in macro variable #599 has more characters than specified in the format string, only the quantity specified will be printed. For example, if #599 is set to"12345" and only four places are specified in the format string, only "2345" will be engraved.

LITERAL STRING ENGRAVING (G54-G59)

This method is used to engrave desired text on a part. The characters available for engraving are:

A..Z
a..z
0..9
! “ # & ‘ ( ) * + , / : ; < = > ? [\ ] ^ { }

However, programs downloaded through the serial port or the floppy drive can take advantage of characters not available on the mill keypad.

For Literal String Engraving, the text should be in the form of a comment on the same line as the P0 statement.

For example:

```
G47    P0    (ENGRAVE THIS)
```

will produce

ENGRAVE THIS

Note that P0 is used, instead of P1, for Literal String Engraving.

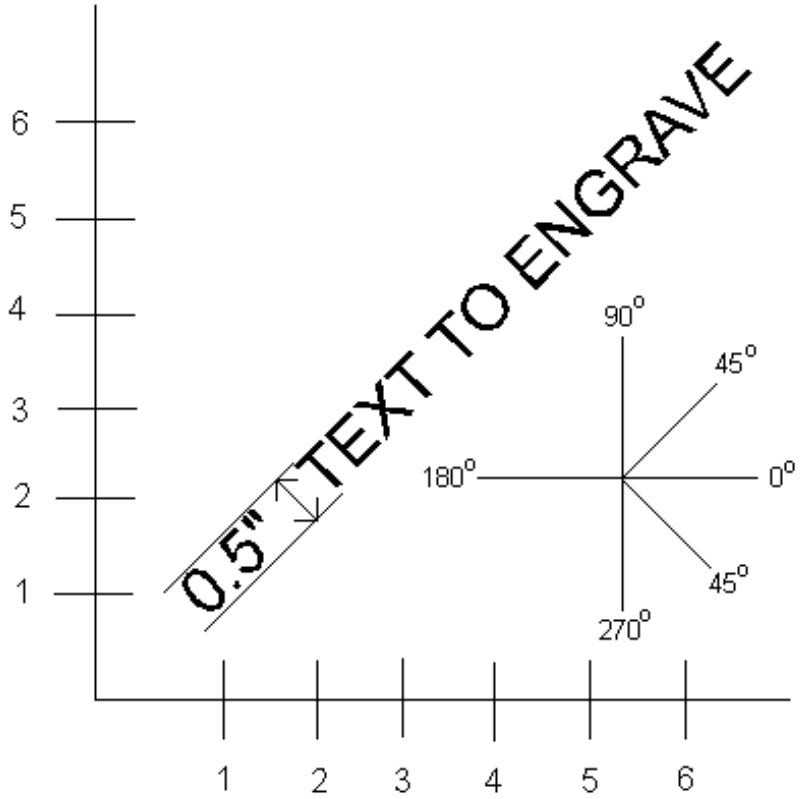
EXAMPLE

This example will create the figure below.

```
G47    P0    X2.0    Y2.0    I45.    J.5    R.05    Z-.005    F15.0    E10.0    (TEXT TO ENGRAVE)
```

In this example:

G47 P0	Select Literal String Engraving
X2.0 Y2.0	Select 2.0**,** 2.0 as the starting point for the text.
I45.	Places the text at a positive 45 degree angle
J.5	Sets the text height to 0.5 inch
R.05	Commands the cutter to retract to .05 inches (mm) above the cutting plane after engraving
Z-.005	Selects a .005 inch (mm) deep cut
F15.0	Selects an engraving feed rate of 15 units/min
E10.0	Commands the cutter to plunge at a rate of 10 units/min



G54-59 ***SELECT COORDINATE SYSTEM #1 - #6*** ***GROUP 12***

These codes select one of the six user coordinate systems stored within the offsets memory. All subsequent references to axes' positions will be interpreted in the new coordinate system. Work coordinate system offsets are entered from the Offsets display page.

BOLT HOLE PATTERNS (G70, G71, G72)

There are three **G** codes that provide patterns usually used for bolt holes. These are G70**,** G71, and G72. They are normally used with one of the Group 09 canned cycles.

G70 ***BOLT HOLE CIRCLE*** ***GROUP 00***

I	Radius (Minus Reverses Direction)
J	Starting angle (0 to 360.0 degrees CCW from horizontal)
L	Number of holes evenly spaced around the circle

This G code must be used with one of the canned cycles G73, G74, G76, G77, or G81-G89. The tool must be positioned at the center of the circle either in a previous block or in the G70 block. G70 belongs to Group zero and thus is non-modal. For a G70 to work correctly, a canned cycle should be active so that at each of the positions, some type of drill or tap cycle is performed.

G71 *BOLT HOLE ARC* *GROUP 00*

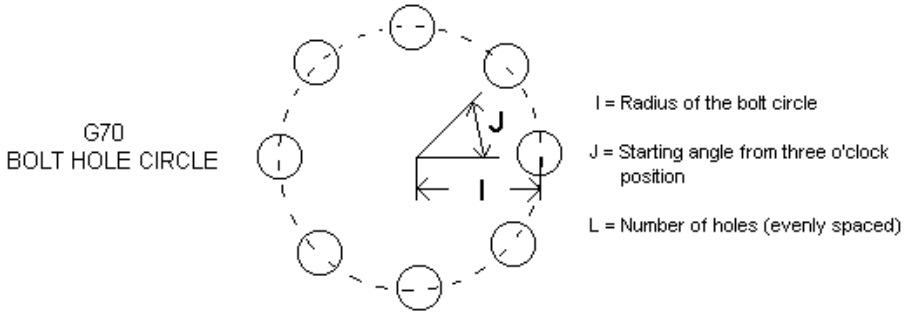
I	Radius
J	Starting angle (Degrees CCW from horizontal)
K	Angular spacing of holes (+ or -)
L	Number of holes

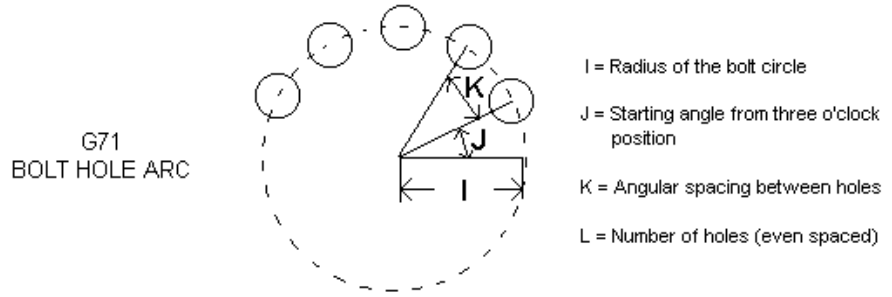
This G code is similar to G70 except that it is not limited at one complete circle. G71 belongs to Group zero and thus is non-modal. For a G71 to work correctly, a canned cycle should be active so that at each of the positions, some type of drill or tap cycle is performed.

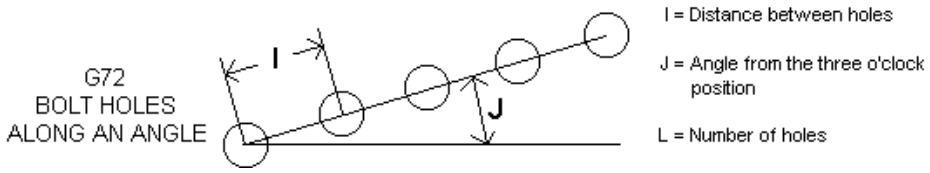
G72 *BOLT HOLES ALONG AN ANGLE* *GROUP 00*

I	Distance between holes (Minus will reverse direction)
J	Angle of line (Degrees CCW from horizontal)
L	Number of holes

This G code drills L holes in a straight line at the specified angle. It operates similarly to G70 and G71. G72 belongs to Group zero and thus is non-modal. For a G72 to work correctly, a canned cycle should be active so that at each of the positions, some type of drill or tap cycle is performed.







CANNED CYCLES (G73-G89)

A canned cycle is used to simplify programming of a part. Canned cycles are defined for most common Z axis repetitive operation such as drilling, tapping, and boring. Once selected a canned cycle is active until canceled with G80. When active, the canned cycle is executed every time an X or Y-axis motion is programmed. Those X-Y motions are executed as rapid commands (G00) and the canned cycle operation is performed after the X-Y mode. There are six operations involved in every canned cycle:

1) Positioning of **X** and **Y** axes (and optional **A),**
2) Rapid traverse to **R** plane,
3) Drilling,
4) Operation at bottom of hole,
5) Retraction to **R** plane,
6) Rapid traverse up to initial point.

A canned cycle is presently limited to operations in the Z-axis. That is, only the G17 plane is allowed. This means that the canned cycle will be executed in the Z-axis whenever a new position is selected in the X or Y axes.

The following is a summary of the canned cycles defined for the VF Series Mill:

G Code	Z Drilling Direction	Operation at Bottom of Hole	Retraction Z Direction	Application
G73	Intermittent Feed	None	Rapid	High Speed Peck Drilling
G74	Feed	Spindle CW	Feed	Left Hand Tapping
G76	Feed Then Stop	Orient Spindle	Rapid	Fine Boring
G81	Feed	None	Rapid	Spot Drilling
G82	Feed	Dwell	Rapid	Counter Boring
G83	Intermittent Feed	None	Rapid	Peck Drilling Full Retraction
G84	Feed	Spindle CCW	Feed	Tapping Cycle
G85	Feed	None	Feed	Boring Cycle
G86	Feed	Spindle Stop	Rapid	Boring Cycle
G87	Feed	Spindle Stop	Manual/Rapid	Back Cycle
G88	Feed	Dwell, then Spindle Stop	Manual/Rapid	Boring Cycle
G89	Feed	Dwell	Feed	Boring Cycle

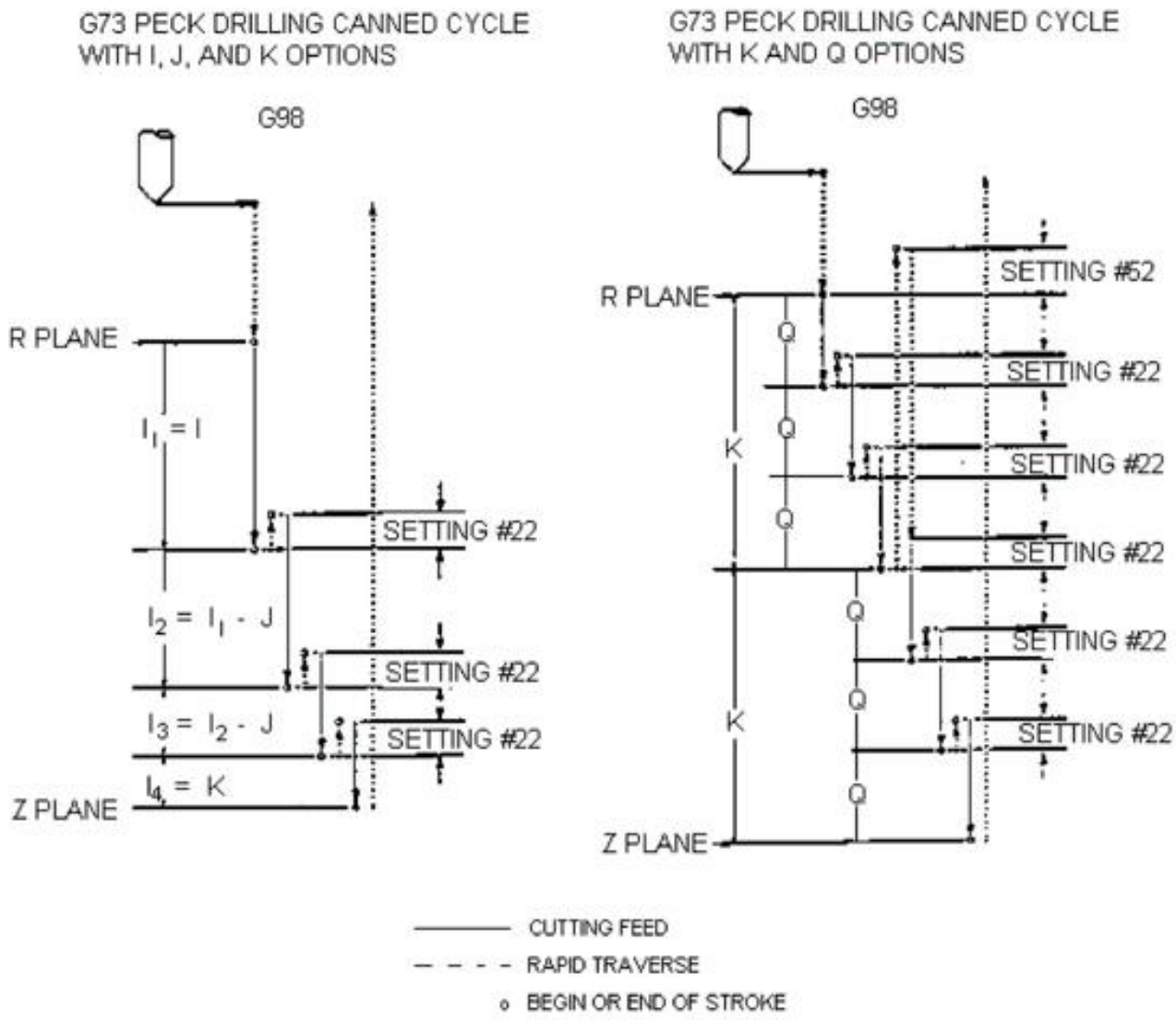
G98 versus G99

G98 and G99 are modal commands that change the way the canned cycles operate. When G98 is active, the Z-axis will be returned to the same position as at the start of the canned cycle when it completes. When G99 is active, the Z-axis will be returned to the R point when the canned cycle completes.

If a canned cycle is defined in a block without an **X** or **Y** motion, there are two common actions taken by other controls; some will execute the canned cycle at that time and some will not. With the VF Series Mill, these two options are selectable from Setting 28. In addition to this, if a canned cycle is defined without an **X** or **Y** and a loop count of 0 (L0), the cycle will not be performed initially. The operation of a canned cycle will vary according to whether incremental (G91) or absolute (G90) is active.

Incremental motion in a canned cycle is often useful as a loop **(L)** count can be used to repeat the operation with an incremental **X** or **Y** move between each cycle.

The positioning of the X-Y axis prior to a canned cycle is normally a rapid move and that move does not exact stop prior to plunging the Z-axis to the R depth. This may cause a crash with a close tolerance fixture. Setting 57 can be used to select exact stop of these X-Y moves.



The **G80** code is used to cancel a canned cycle. In addition to this**,** a G00 or G01 code will also cancel any active canned cycle. Once a canned cycle is defined, that operation is performed at every X-Y position subsequently listed in a block. Some of the canned cycle numerical values can also be changed after the canned cycle is defined. The most important of these are the **R** plane value and the **Z** depth value.

If these are listed in a block with an X-Y, the X-Y move is done and all subsequent canned cycles are performed with the new **R** or **Z** value.

Changes to the G98/G99 selection can also be made after the canned cycle is active. If changed, the new G98/G99 value will change all subsequent canned cycle.

F	Feed Rate in inches (mm) per minute
I	Optional size of first cutting depth
J	Optional amount to reduce cutting depth each pass
K	Optional minimum depth of cut
L	Number of repeats
Q	The cut-in value, always incremental
R	Position of the R plane
X	Optional X-axis motion command
Y	Optional Y-axis motion command
Z	Position of bottom of hole

This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X or Y will cause this canned cycle to be executed. This cycle is a high speed peck cycle where the retract distance is set by Setting 22.

If **I**, **J,** and **K** are specified, a different operating mode is selected. The first pass will cut in by **I**, each succeeding cut will be reduced by amount **J,** and the minimum cutting depth is **K.**

If **K** and **Q** are both specified, a different operating mode is selected for this canned cycle. In this mode, the tool is returned to the **R** plane after a number of passes totals up to the **K** amount. This allows much faster drilling than G83 but still returns to the **R** plane occasionally to clear chips.

I, J, K, and **Q** are always positive numbers.

Setting 52 also changes the way G73 works when it returns to the **R** plane. Most programmers set the **R** plane well above the cut to ensure that the chip clear motion actually allows the chips to get out of the hole but this causes a wasted motion when first drilling through this "empty" space. If Setting 52 is set to the distance required to clear chips, the **R** plane can be put much closer to the part being drilled. When the clear move to **R** occurs, the **Z** will be moved above **R** by this setting.

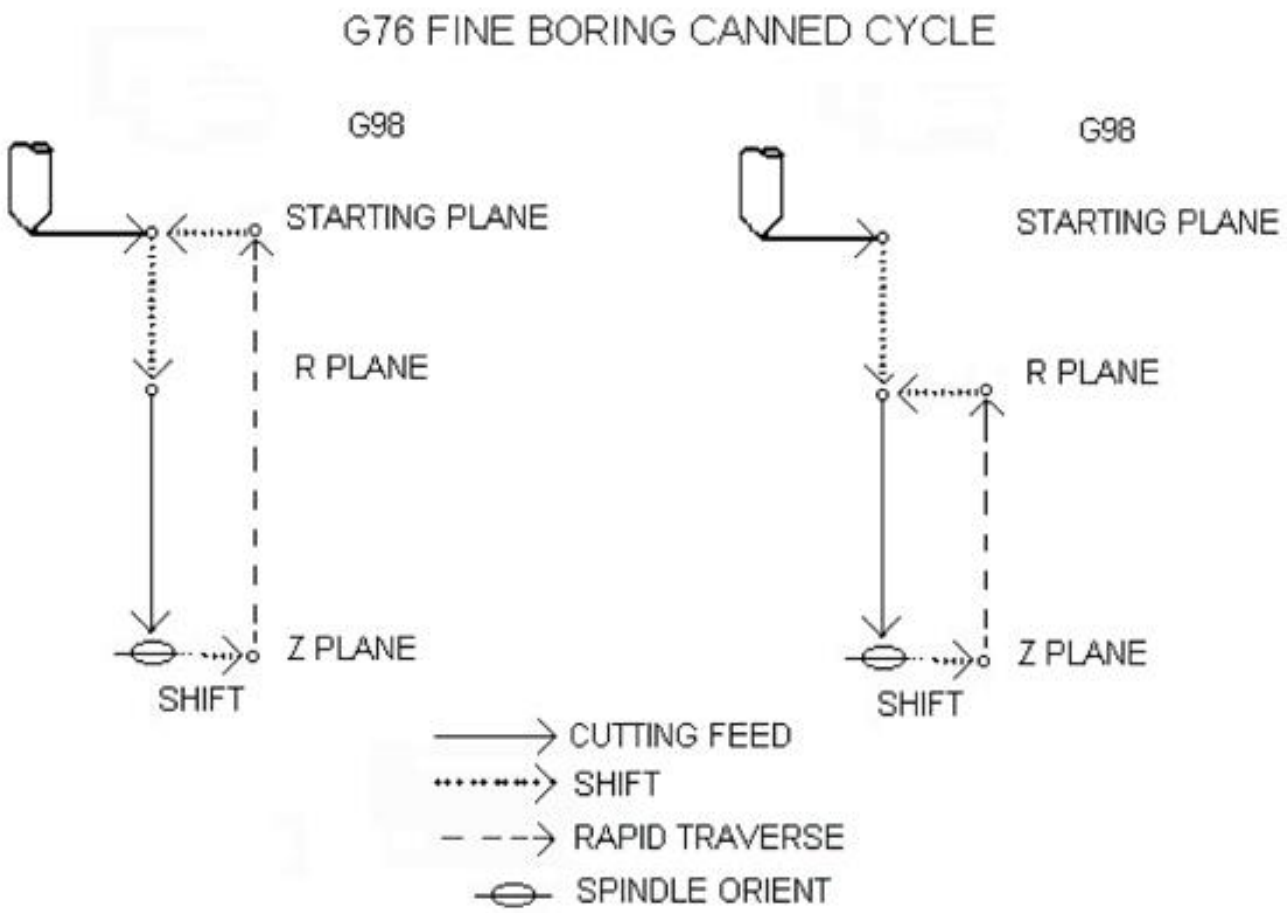
G74 REVERSE TAP CANNED CYCLE GROUP 09

F	Feed Rate in inches (mm) per minute
L	Number of repeats
R	Position of the R plane
X	Optional X-axis motion command
Y	Optional Y-axis motion command
Z	Position of bottom of hole

This **G** code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of **X** or **Y** will cause this canned cycle to be executed. Note that operation of this cycle is different if the rigid tapping option is installed and selected (See Section 7.2).

When rigid tapping is used, the ratio between the feed rate and spindle speed must be precisely the thread pitch being cut.

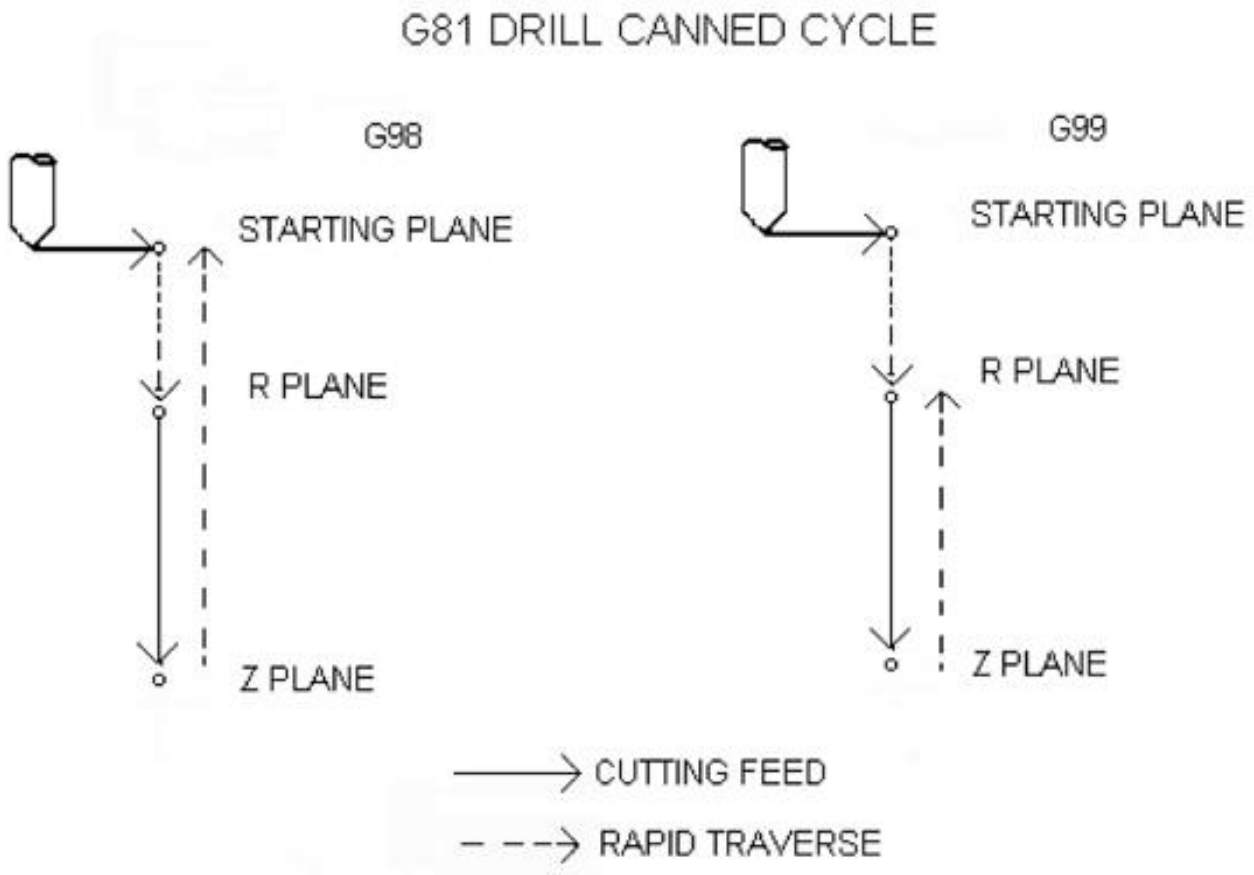
You do not need to start the spindle CCW before this canned cycle. The control does this automatically.



F	Feed Rate in inches (mm) per minute
I	Optional shift value, if Q is not specified.
J	Optional shift value**,** if Q is not specified.
L	Number of repeats
P	The dwell time at the bottom of the hole
Q	The shift value, always incremental
R	Position of the R plane
X	Optional X-axis motion command
Y	Optional Y-axis motion command
Z	Position of bottom of hole

This **G** code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of **X** and/or **Y** will cause this canned cycle to be executed. This cycle will shift the **X** and/or Y-axis prior to retracting in order to clear the tool while exiting the part. This shift direction is set by Setting 27.

The Q value shift direction is set by setting 27. If Q is not specified, the optional I and J values are used to determine the shift direction and distance.



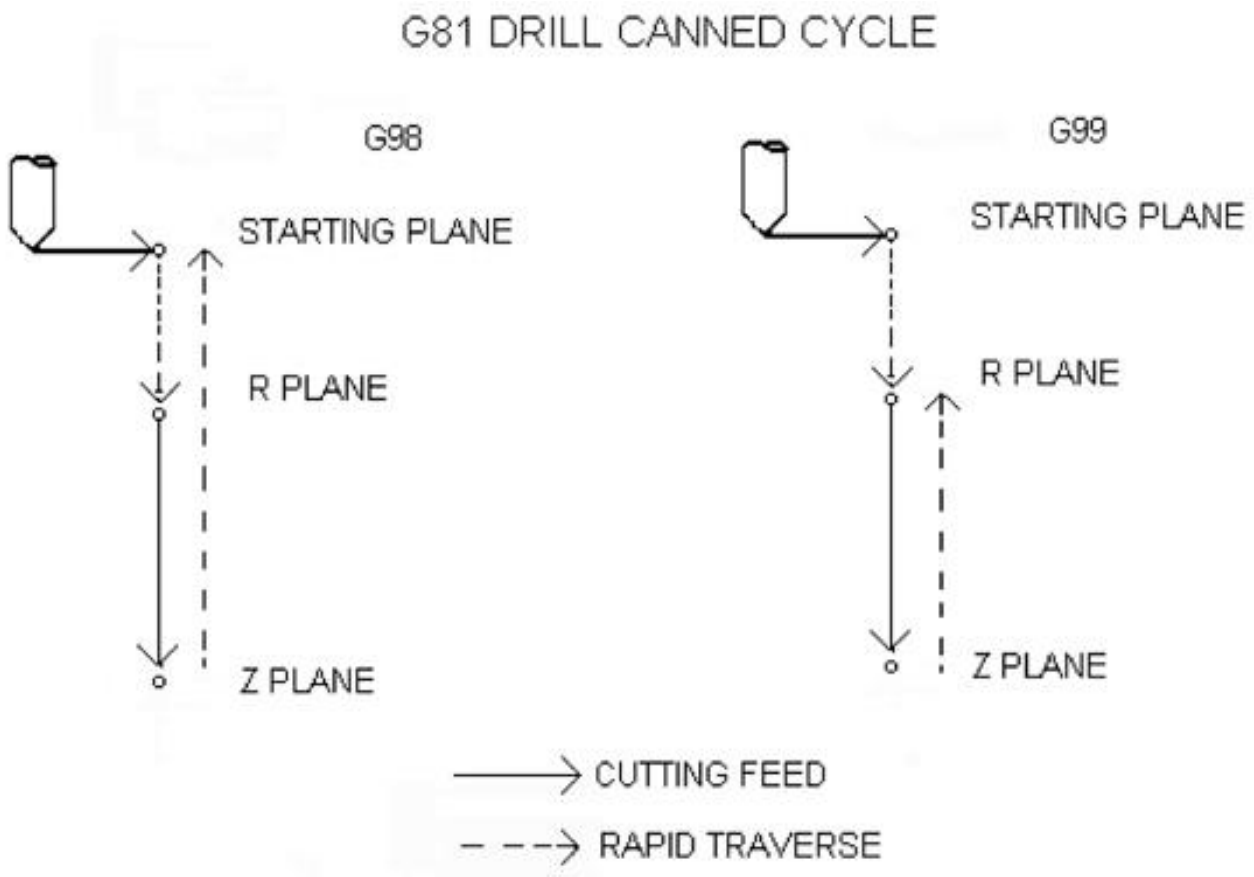
G80 ***CANNED CYCLE CANCEL*** ***GROUP 09***

This **G** code is modal in that it deactivates all canned cycles until a new one is selected. Note that use of G00 or G01 will also cancel a canned cycle.

G81 ***DRILL CANNED CYCLE*** ***GROUP 09***

F	Feed Rate in inches (mm) per minute
L	Number of repeats
R	Position of the R plane
X	Optional X-axis motion command
Y	Optional Y-axis motion command
Z	Position of bottom of hole

This **G** code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of **X** or **Y** will cause this canned cycle to be executed.



G82 SPOT DRILL CANNED CYCLE GROUP 09

F	Feed Rate in inches (mm) per minute
L	Number of repeats
P	The dwell time at the bottom of the hole
R	Position of the R plane
X	Optional X-axis motion command
Y	Optional Y-axis motion command
Z	Position of bottom of hole

This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X or Y will cause this canned cycle to be executed.

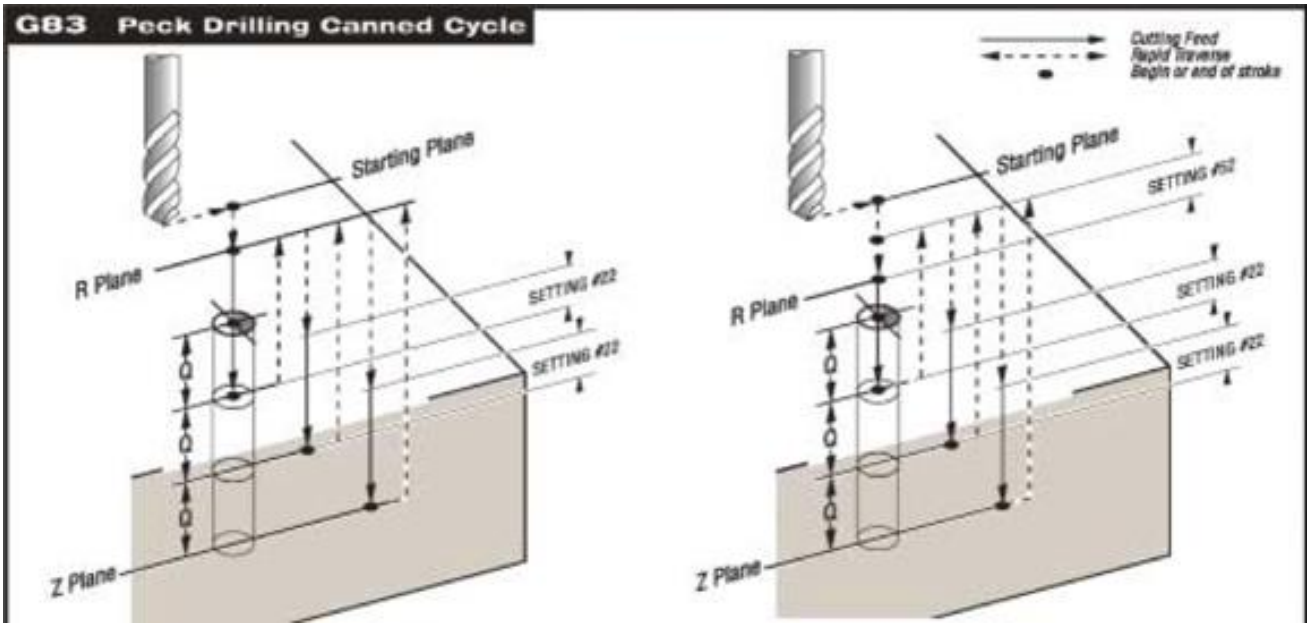
G83 PECK DRILLING CANNED CYCLE GROUP 09

F	Feed Rate in inches (mm) per minute
I	Optional size of first cutting depth
J	Optional amount to reduce cutting depth each pass
K	Optional minimum depth of cut
L	Number of repeats
Q	The cut-in value, always incremental
R	Position of the R plane
X	Optional X-axis motion command
Y	Optional Y-axis motion command
Z	Position of bottom of hole

This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X or Y will cause this canned cycle to be executed.

If **I, J,** and **K** are specified, a different operating mode is selected. The first pass will cut in by **I**, each succeeding cut will be reduced by amount **J,** and the minimum cutting depth is **K.**

Setting 52 also changes the way G83 works when it returns to the **R** plane. Most programmers set the **R** plane well above the cut to insure that the chip clear motion actually allows the chips to get out of the hole but this causes a wasted motion when first drilling through this "empty" space. If Setting 52 is set to the distance required to clear chips, the **R** plane can be put much closer to the part being drilled. When the clear move to **R** occurs, the **Z** will be moved above **R** by this setting.



G84 ***TAPPING CANNED CYCLE*** ***GROUP 09***

F	Feed Rate in inches (mm) per minute
L	Number of repeats
R	Position of the R plane
X	Optional X-axis motion command
Y	Optional Y-axis motion command
Z	Position of bottom of hole

This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X or Y will cause this canned cycle to be executed. Note that operation of this cycle is different if the rigid tapping option is installed and selected (See Section 7.2). When rigid tapping is used, the ratio between the feed rate and spindle speed must be precisely the thread pitch being cut. You do not need to start the spindle CW before this canned cycle. The control does this automatically.

G85 ***BORING CANNED CYCLE*** ***GROUP 09***

F	Feed Rate in inches (mm) per minute
L	Number of repeats
R	Position of the R plane
X	Optional X-axis motion command
Y	Optional Y-axis motion command
Z	Position of bottom of hole

This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X or Y will cause this canned cycle to be executed.

Absolute/Incremental Selection

G90 ABSOLUTE POSITION COMMANDS GROUP 03

This code is modal and changes the way axis motion commands are interpreted. G90 makes all subsequent commands absolute positions within the selected user coordinate system. Each axis which is moved will be placed at the position coded in the command block.

G91 INCREMENTAL POSITION COMMANDS GROUP 03

This code is modal and changes the way axis motion commands are interpreted. G91 makes all subsequent commands incremental. Each axis which is moved will be moved by the amount coded in the command block.

Canned Cycle Auxiliary Functions

G98 CANNED CYCLE INITIAL POINT RETURN GROUP 10

This **G** code is modal and changes the way canned cycles operate. With 98, the canned cycle will return to the initial starting point of the canned cycle when it completes.

G99 CANNED CYCLE R PLANE RETURN GROUP 10

This **G** code is modal and changes the way canned cycles operate. With G99, the canned cycle will return to the **R** plane when the canned cycle completes.

Miscellaneous Functions (M Functions)

M CODE SUMMARY

Only one **M** code may be programmed per block of a program. All **M** codes are effective or cause an action to occur at the end of the block and only one **M** code is allowed in each block.

M00	Stop Program
M01	Optional Program Stop
M02	Program End
M03	Spindle Forward
M04	Spindle Reverse
M05	Spindle Stop
M06	Tool Change
M08	Coolant On
M09	Coolant Off
M10	Engage 4th Axis Brake
M11	Release 4th Axis Brake
M12	Engage 5th Axis Brake
M13	Release 5th Axis Brake
M16	Tool Change (same as M06)
M19	Orient Spindle
M21-M24	Optional Pulsed User M Function with Fin
M30	Prog End and Rewind
M31	Chip Conveyor Forward
M32	Chip Conveyor Reverse
M33	Chip Conveyor Stop
M34	Increment Coolant Spigot Position
M35	Decrement Coolant Spigot Position
M36	Pallet Rotate
M39	Rotate Tool Turret
M41	Low Gear Override
M42	High Gear Override
M51-M54	Optional User M turn ON
M61-M64	Optional User M turn OFF
M75	Set G35 or G136 reference point
M76	Disable Displays
M77	Enable Displays
M78	Alarm if skip signal found
M79	Alarm if skip signal not found
M82	Tool Unclamp
M86	Tool Clamp
M88	Thru Spindle Coolant ON
M89	Thru Spindle Coolant OFF
M96	Conditional Local Branch when Discrete Input Signal is 0.
M97	Local Sub-Program Call
M98	Sub Program Call
M99	Sub Program Return or Loop

M Code Detailed Description

M00 STOP PROGRAM

The M00 code is used to stop a program. It also stops the spindle and turns off the coolant and stops interpretation look-ahead processing. The program pointer will advance to the next block and stop. A cycle start will continue program operation from the next block. If the Through the Spindle Coolant option is ON, M00 will shut it off.

M01 OPTIONAL PROGRAM STOP

The M01 code is identical to M00 except that it only stops if OPTIONAL STOP is turned on from the front panel. A cycle start will continue program operation from the next block. If the Through the Spindle Coolant option is **ON,** M01 will shut it off.

M02 PROGRAM END

The M02 code will stop program operation the same as M00 but does not advance the program pointer to the next block.

M03 SPINDLE FORWARD

The M03 code will start the spindle moving is a clockwise direction at whatever speed was previously set. The block will delay until the spindle reaches about 90% of commanded speed.

M04 SPINDLE REVERSE

The M04 code will start the spindle moving is a counterclockwise direction at whatever speed was previously set. The block will delay until the spindle reaches about 90% of commanded speed.

M05 SPINDLE STOP

The M05 code is used to stop the spindle. The block is delayed until the spindle slows below 10 RPM.

M06 TOOL CHANGE

The M06 code is used to initiate a tool change. The previously selected tool (Tn) is put into the spindle. If the spindle was running, it will be stopped. No previous axis commands are required before the tool change unless there is a problem with tool/part/fixture clearance. The Z-axis will automatically move up to the machine zero position and the selected tool will be put into the spindle. The Z-axis is left at machine zero. The spindle will not be started again after the tool change but the Snnnn speed and gear will be unchanged. The Tnn must be in the same block or in a previous block. The coolant pump will be turned off during a tool change.

When the Through the Spindle Coolant (TSC) is ON, M06 will orient the spindle and move the Z-axis to tool change position, turn off the TSC pump, purge the coolant from the drawbar, then perform a tool change. TSC will remain OFF until an M88 is called.

M08 COOLANT ON

The M08 code will turn on the coolant supply. Note that the M code is performed at the end of a block; so that if a motion is commanded in the same block, the coolant is turned on after the motion. The low coolant status is only checked at the start of a program so a low coolant condition will not stop a program which is already running.

M09 COOLANT OFF

The M09 code will turn off the coolant supply.

M30 PROG END AND REWIND

The M30 code is used to stop a program. It also stops the spindle and turns off the coolant. The program pointer will be reset to the first block of the program and stop. The parts counters displayed on the Current Commands display are also incremented. M30 will also cancel tool length offsets. When the Through the Spindle Coolant (TSC) option is ON, M30 will shut it OFF, and then perform an M30 operation.

M97 LOCAL SUB-PROGRAM CALL

This code is used to call a subroutine referenced by a line N number within the same program. A Pnnnn code is required and must match a line number within the same program. This is useful for simple subroutines within a program and does not require the complication of a separate program. The subroutine must still be ended with an M99. An L count on the M97 block will repeat the subroutine call that number of times.

M98 SUB PROGRAM CALL

This code is used to call a subroutine. The Pnnnn code is the number of the program being called. The Pnnnn code must be in the same block. The program by the same number must already be loaded into memory and it must contain an M99 to return to the main program. An L count can be put on the line containing the M98 and will ocause the subroutine to be called L times before continuing to the next block.

M99 SUB PROGRAM RETURN OR LOOP

This code is used to return to the main program from a subroutine or macro. It will also cause the main program to loop back to the beginning without stopping if it is used in other than a subprogram without a **P** code. If an M99 Pnnnn is used, it will cause a jump to the line containing Nnnnn of the same number.

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 x .03937) x RPM

Speeds and Feeds

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

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

SFM = .262 x Cutter Diameter x RPM

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

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

RPM = 3.82 x SFM / Cutter Diameter

Feed (Inch per Minute) for **twist drills**:

F(inch/min) = F(inch /rev) x RPM

Feed (Inch per Minute) for **end mills**:

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 x N) x 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 x d) x IPM