



**Haas Factory Outlet**

**A Division of Productivity Inc**

*Mill Series  
Training Manual*

*Haas TM Series  
Tool Room Mill Operator*



*This Manual is the Property of Productivity Inc*

*The document may not be reproduced without the express written permission of  
Productivity Inc.*

*The content must not be altered, nor may the Productivity Inc name be removed  
from the materials.*

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

*To obtain permission, please contact [trainingmn@productivity.com](mailto:trainingmn@productivity.com).*


# ***Tool Room Mill Operator Training Manual***

## ***Table of Contents***

INTRODUCTION .....	3
INTRODUCTION TO BASIC TM SERIES OPERATION.....	4
CARTESIAN COORDINATES .....	4
MACHINE HOME POSITION .....	8
THE HAAS CNC CONTROL .....	10
CONTROL DISPLAY .....	11
KEYBOARD INTRODUCTION.....	12
1 – FUNCTION KEYS .....	13
2 – JOG KEYS .....	13
3 – OVERRIDE KEYS .....	14
4 – DISPLAY KEYS .....	15
5 – CURSOR KEYS .....	19
6 AND 7 – ALPHA KEYS AND NUMERIC KEYS .....	19
8 – MODE KEYS .....	21
SETTINGS .....	24
TOOLROOM MILL ORIENTATION AND WALK AROUND .....	27
TOOLROOM MILL SAFETY.....	28
EMERGENCY STOP SWITCH.....	28
MAINTENANCE OF THE TM SERIES MILL .....	29
GREASE POINTS .....	30
CAT 40 TOOLING: .....	31
MILL WORK-HOLDING OPTIONS.....	32
INTUITIVE PROGRAMMING SYSTEM.....	33
ENTERING THE IPS.....	33
NAVIGATING TABULATED SCREENS .....	36
DRILLING AND TAPPING USING IPS .....	42
DRILL APPLICATION IS MULTIPLE HOLES.....	46
ENGRAVING .....	53



For more information on Additional Training Opportunities  
or our Classroom Schedule  
Contact the Productivity Inc Applications Department in Minneapolis:

 763.476.8600

Visit us on the Web: [www.productivity.com](http://www.productivity.com)

Click on the Training Registration Button

 [trainingmn@productivity.com](mailto:trainingmn@productivity.com)

# ***Introduction***

Welcome to Productivity, Inc., the Haas CNC Machine Tool Distributor for your local area.

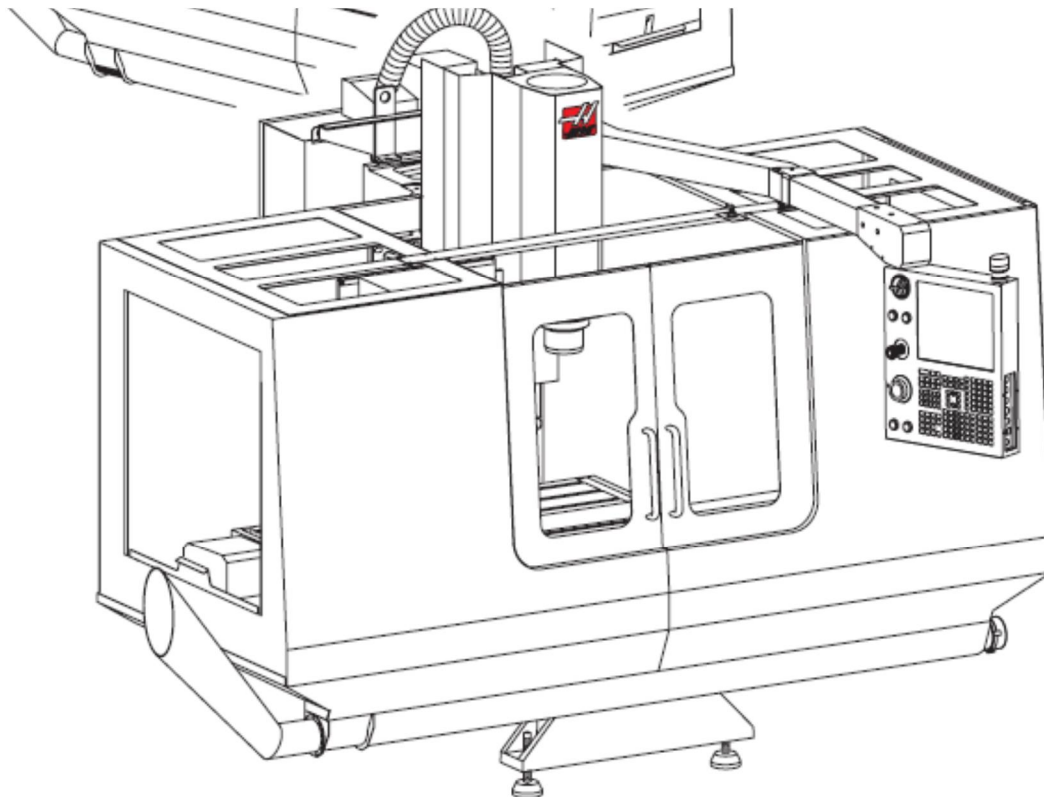
As part of your companies Haas CNC purchase is standard lifetime training as long as your company owns the machine.

What we plan on covering in this one-day class is the operation and programming of the unique features of the Haas TM- Series Tool Room Mills.

These mills are unique in their own way as they are designed for manual, semi-manual, and full CNC G&M code operation. Even though the TM series can be run from a G&M code program, Haas has equipped these unique machines with a unique control. The Haas IPS (Intuitive Programming System) allows for quick and easy set-up and programming of standard tool-room style parts.

Since the TM series is so unique, Productivity, Inc. has decided to generate a specific class just for the TM series to best suit its unique features.

If you or your company would like to learn more advanced G&M code programming to take advantage of the Haas control equipped on the TM series of mills, we suggest the more advanced class Mill Programming after attending this one.



*Rev 05/2012-CK; 10/2014 CK; 1/2015 CK*



# Introduction to Basic TM Series Operation

## Cartesian Coordinates

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

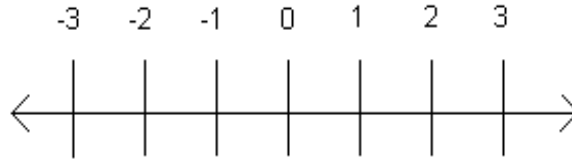


Fig-1-1 Horizontal number line – X Axis

The number line also has numbered increments on either side of absolute zero. Moving away from zero to the right are positive increments. Moving away from zero to the left are negative increments. The "+", or positive increments, are understood, therefore no sign is needed.

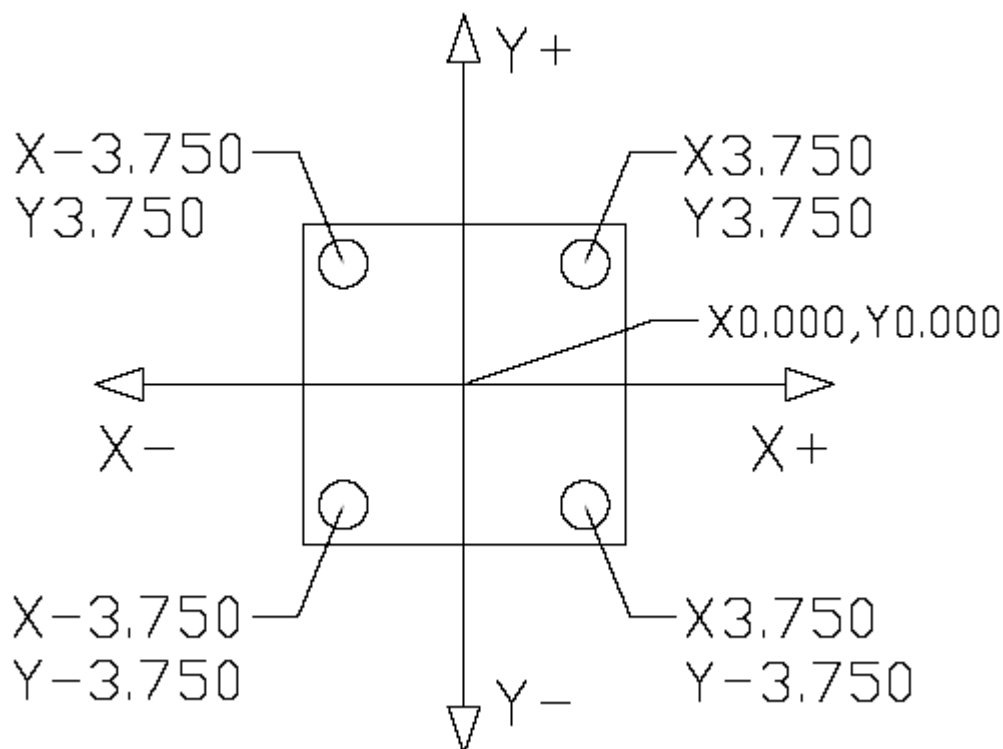
We use positive and negative along with the increment's value to indicate its relationship to zero on the line. In the case of the previous line, if we choose to move to the third increment on the minus (-) side of zero, we would call for -3. If we choose the second increment in the plus range, we would call for 2. Our concern is with distance and direction from zero.

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.



Fig.1-2 Vertical Number Line – Y Axis

The next illustration (Fig. 1-3) shows the X & Y directions of travel on a TM Series Mill. It shows the positive and negative movement of our spindle in both axis. Also note where these two axis meet, they create a common point of “zero” where they both are at the value of “0.000” at the same point.



The first number line is easy to conceive as belonging to the left-to-right, or “X”, axis of the machine. If we place a similar number line along the front-to-back, or “Y”, axis, the increments away from the operator are the positive increments, and the increments on the other side of zero towards the operator are the negative increments.

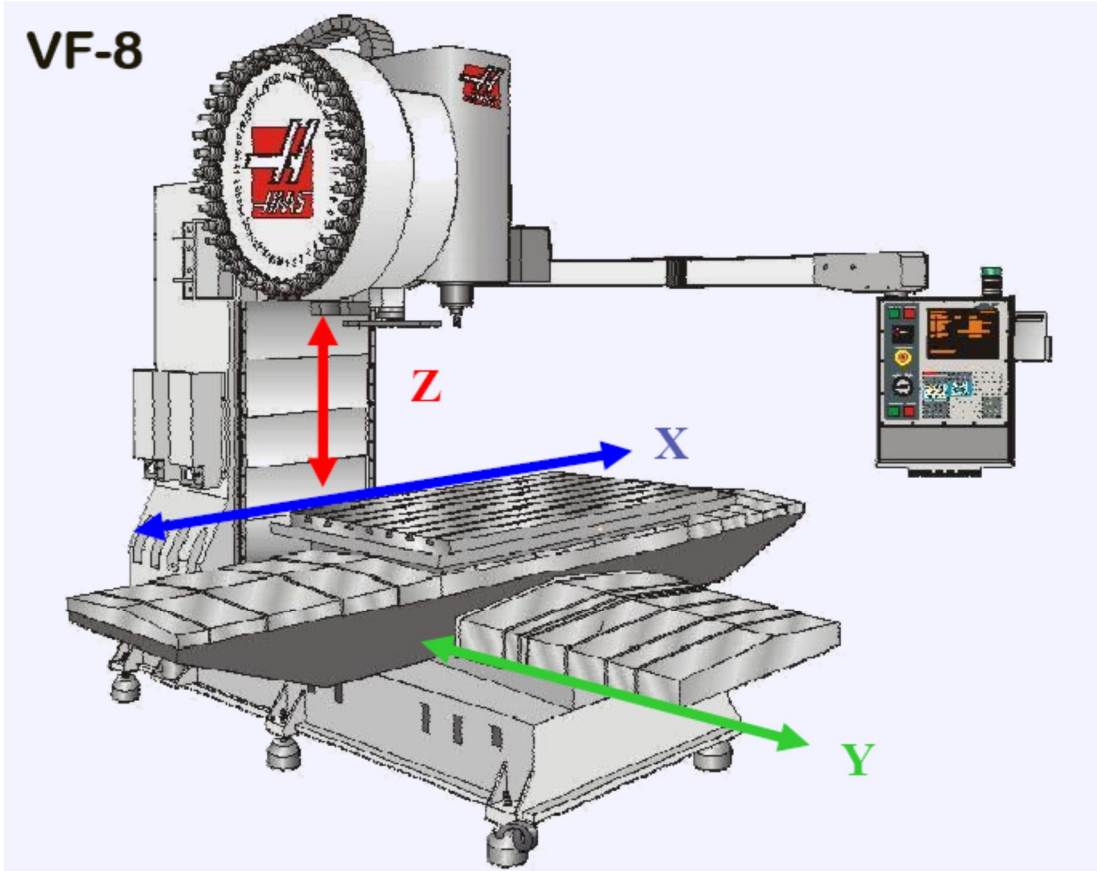
The zero position may be placed at any point along each of the two number lines, and in fact will probably be different for each setup of the machine.

As with any VMC (Vertical Machining Center) the TM series has 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.

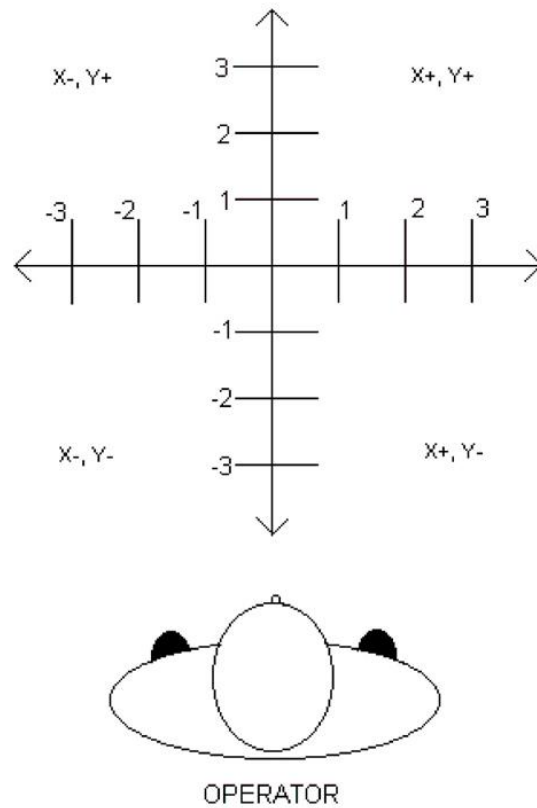
Now that we have discussed all 3 of the axis, we can take a look at this picture of a Haas VF-8 VMC. Even though it is much larger than a Tool Room Mill, the operation and orientation are exactly the same for both machines.



VF-8



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.*

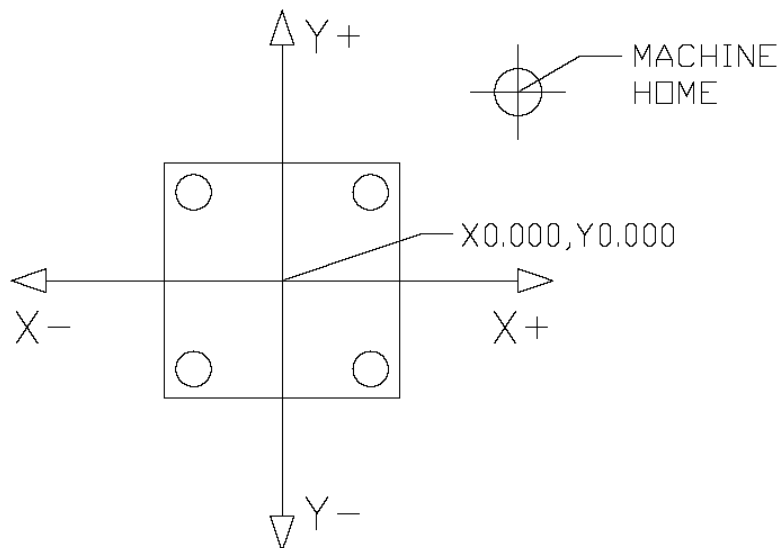
Whenever we set a zero somewhere on the X axis and somewhere on the Y axis, we have automatically caused an intersection of the two lines. This is known as "zero", where our X and Y lines would both be at a value of 0.0000. We will move this point to a "Zero" position on our part so that we can steer the machine to locations that relate to our print.

## ***Machine Home Position***

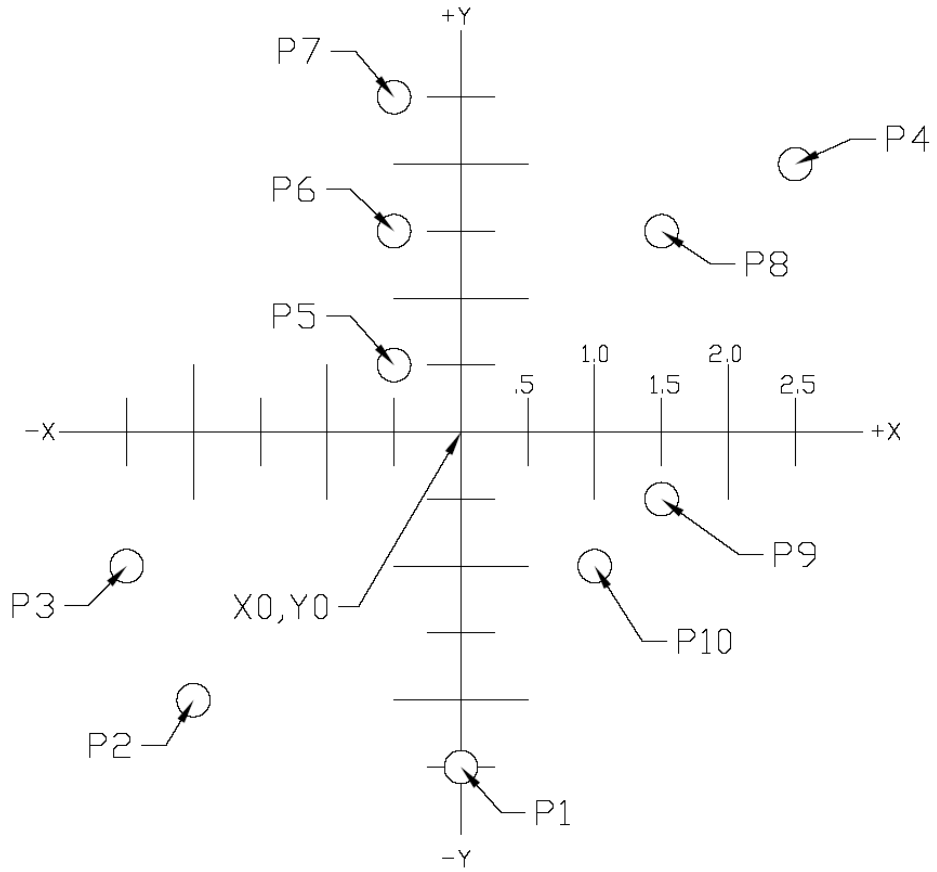
The principal of machine home may be seen when doing a manual reference return of all machine axes. When a zero return (ZERO RET) is performed at machine start up, all axes are moved to the furthest positive direction until limit switches are reached. The machine needs to do this to double check itself to a known point on the machine. This point is at the far right hand side of the table, and as far away from the operator as possible.

Like double checking a micrometer or caliper when you take it out of the box, this ensures the machine repeats accurately.

Sometimes this point is referred to as “Machine Zero” as pointed out below:



## Cartesian Coordinate Exercise



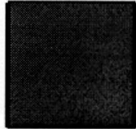
POINT #	X Position	Y Position
P1	X 0.0	Y -2.5
P2		
P3		
P4		
P5		
P6		
P7		
P8		
P9		
P10		

# The Haas CNC Control

## Powering On the Machine

To power up a Haas machine, regardless of where the machine table was when it was turned off, press **POWER ON**. The machine must first find its fixed machine zero reference point before any operations can occur. After it's powered on, pressing **POWER UP/RESTART** will send the machine to its machine zero reference location. The machine doors must be cycled and closed to return to machine zero. Also the machine needs to see the Emergency Stop cycled. Haas provides directions on the screen on what needs to be done to start the machine up in the morning.

### POWER ON



#### 1. POWER ON

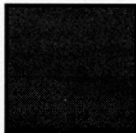
When powering on the machine, if there is a message in the **MESGS** display, it will be the first display seen on your control screen.



#### 2. POWER UP/RESTART

Will move all axis to machine zero and then changes tools to put tool #1 in the spindle. Machine will move up in Z first then the X and Y move to machine zero.

### CYCLE START



#### 3. CYCLE START

If the correct program has been selected and the part program is proven to be good and it's ready to run, press cycle start.

## 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.

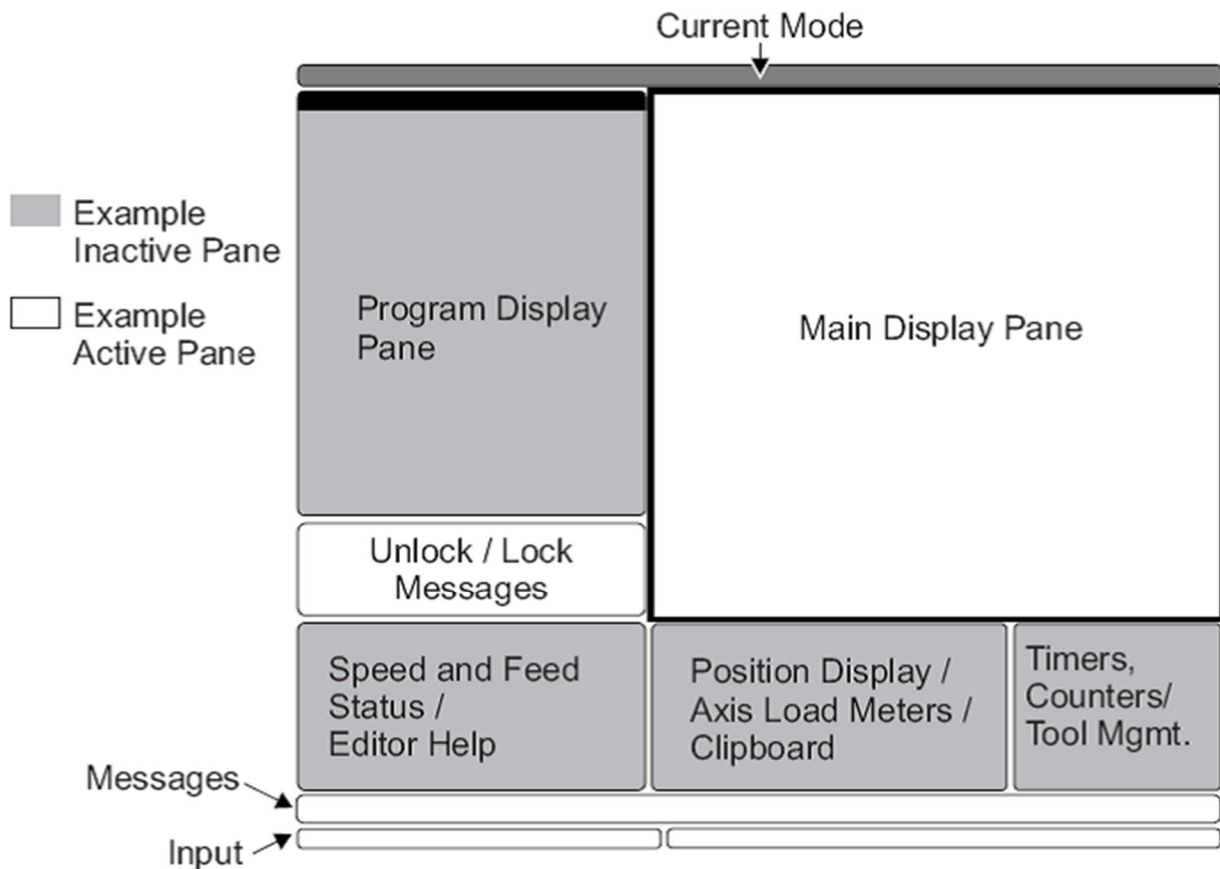
**Feed Hold** – Stops all axis motion. Spindle will continue to turn.

**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.

## 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.

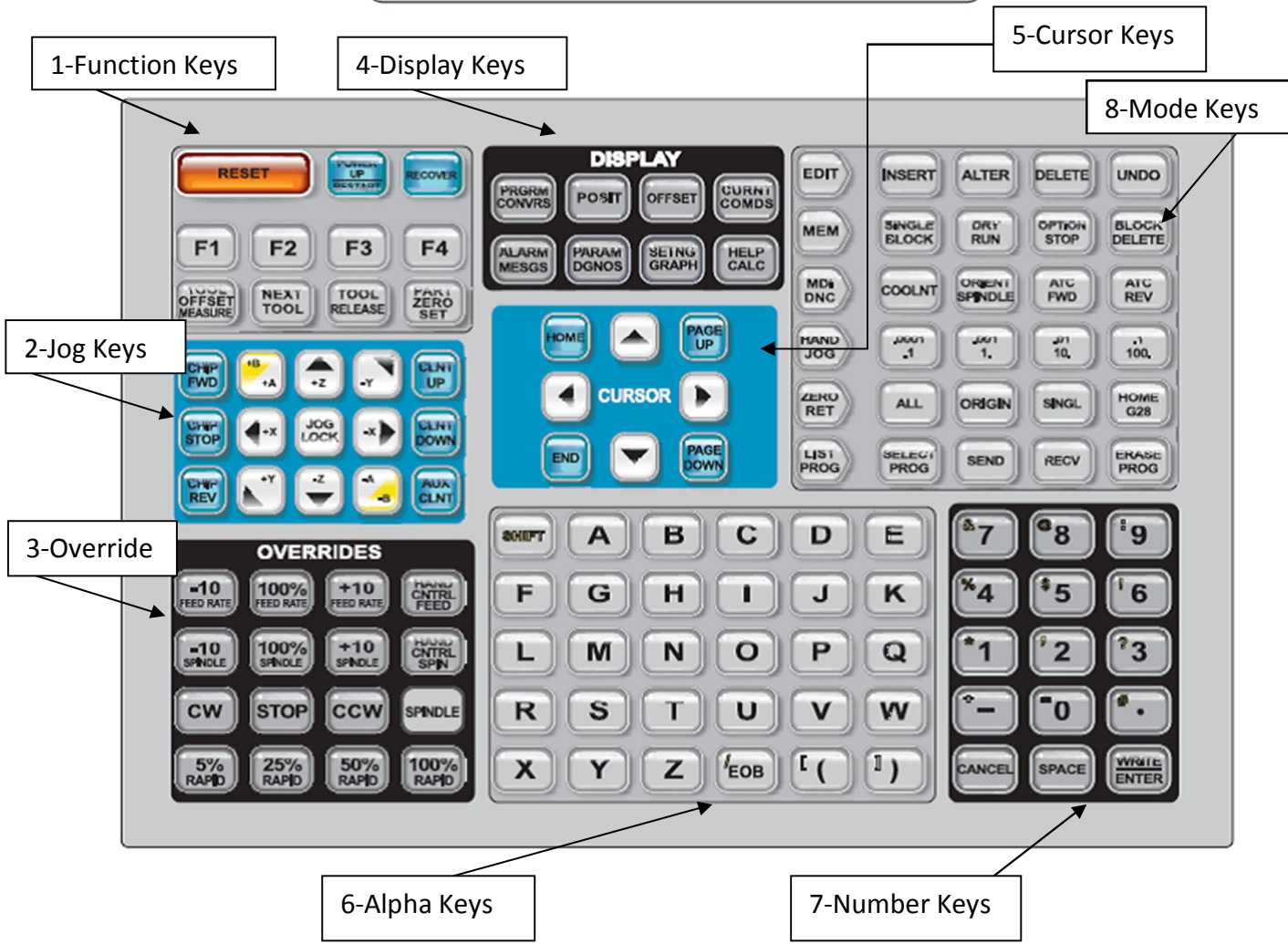
**Operation:** MEM key. Provides all control features necessary to make a part.

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.

# 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.



## 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.

**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.

## 4 – Display Keys

**PRGM/CONVRS** – Selects the active program pane (highlights in white). In MDI/DNC mode pressing a second time will allow access to VQC (Visual Quick Code) and IPS (Intuitive Programming System)

**POSIT** (*Position*) – Selects the positions display window (lower middle). Repeated pressing of the **POSIT** key will toggle through relative positions in the Memory Mode. In Handle Jog mode all four are listed together.

1. **POS-OPER** digital display. This is a reference display only. Each axis can be zeroed out independently; then the display shows the axis position relative to where you decided to zero it. In the Handle Jog mode, you can press the X, Y or Z JOG keys and ORIGIN key to zero that selected axis. On this display page, you can also enter in an axis letter and number (X-1.25) and press ORIGIN to have that value entered in that axis display.
2. **POS-WORK** digital display. This position display tells how far away the tools are in X, Y and Z from the presently selected work offset zero point.
3. **POS-MACH** digital display. This is in reference to machine zero, the location that the machine moves to automatically when you press POWER UP/RESTART. This display will show the current distance from machine zero.
4. **POS-TO-GO** digital display. When you're running the machine, or when you have the machine in a Feed Hold, this incrementally displays the travel distance remaining in the active program move being run. This is useful information when you are stepping a program through during a set up.

When the position pane is active one can change which axis is displayed simply by typing X or Y or Z or any combination and pressing write. Then only that particular axis or combination will be displayed.

**OFFSET** – Selects one of two offsets tables: Tool Geometry/Wear and Work Zero Offset. Depressing the **OFFSET** button toggles between the two tables. Tool Geometry/Wear table displays 200 tool length offsets (100 tool length offsets on older machines) - labeled (LENGTH) GEOMETRY - along with wear offsets. It also displays cutter compensation radius or diameter offsets with wear offsets. You can select how you would like a tool size to be specified using cutter compensation (G41 and G42) with Setting 40 (Radius or Diameter).

The tool-offset display has a CLNT POS column for setting the coolant spigot position.

1. Programmable coolant has 34 possible positions. These may be entered into the column under CLNT POS for each tool in the offset display. This will move the coolant spigot to the position entered when that tool length offset is read.
2. When the cursor is positioned in the CLNT POS column, the actual coolant position will be shown at the lower left corner of the screen.

The Work Zero Offset table has G54-G59 plus G154 P1-G154 P99 offsets available.

The **WRITE/ENTER** key will add the number in the input buffer to the selected offset, and the **F1** key will replace the selected offset with the number entered into the input buffer. Offsets can also be entered using **TOOL OFFSET MEASUR** and **PART ZERO SET**

## **CURNT COMDS** – Ten different pages; use **PAGE UP** and **PAGE DOWN**

1. Operation Timers – displays Power-On Time, Cycle Start Time, Feed Cutting Time. Hitting **ORIGIN** will clear any display that is highlighted by the cursor.
2. Real time clock and date.
3. System Variables, for machines with Macro Programming.
4. All Active Codes, displays current and modal command values.
5. Position information: Machine, Distance to Go, Operator, Work Coordinate.
6. Tool life, displays the usage of each tool. An alarm can be set for the number of times you want that tool to be used, and when that condition has been met (that is, the tool has been used the set number of times), the machine will stop, with an alarm for you to check the condition of that tool. Pressing **ORIGIN** will clear the cursor-selected display, and pressing **ORIGIN** when the cursor is at the top of a column will clear the whole column.
7. Tool Load displays the Tool Load Max % of each tool being used. You can use the Limit% column to set the maximum spindle load for a particular tool. When that condition has been met (the tool has reached maximum load), the machine will stop for you to check the condition of that tool. Pressing **ORIGIN** will clear the cursor-selected display, and pressing **ORIGIN** when the cursor is at the top of a column will clear the whole column. Setting 84 determines the Overload Action when this limit is met.  
Also vibration loads may be entered.
8. Maintenance times for various items may be loaded.
9. Advanced Tool Management (Optional)
10. Tool Pot Table: Gives information on which tool is in which Pot. Refer to Automatic Tool Change section on information on how to use this table.

**ALARM/MESGS** – Displays messages and current active alarms. Press right arrow key gives alarm history. Press right arrow key again goes to the Alarm Viewer Page. Enter alarm number and press write will give detailed information on a particular alarm code.

**PARAM/DGNOS** – Lists machine parameters that are seldom-modified values which change the operation of the machine. These include servo motor types, gear ratios, speeds, stored stroke limits, lead screw compensations, motor control delays and macro call selections. All of these are rarely changed by the user and should be protected by Setting 7, PARAMETER LOCK. A second press of **PARAM/DGNOS** will show the diagnostics display. The **PAGE UP** and **PAGE DOWN** keys are then used to select one of two different pages. This display is for service diagnostic purposes, and the user will not normally need them.

**SETNG/GRAPH** – Displays settings - machine parameters and control functions that the user may need to turn on and off or change to suit specific needs. A list of settings is found on page 30.

- Settings are organized into functionally similar page groups with a title.
- Settings are listed with a number and a short description, and a value or choice on the right.
- To find a particular setting, enter the setting number and then press either the up or down cursor arrow key to move to the desired setting.
- You can change a setting using the left or right cursor arrows to display the choices, or, if the setting contains a value, by typing in a new number. A message at the top of the screen will tell you how to change the selected setting. When you changed, it will flash on and off.
- A setting change is not active until it stops flashing. To activate, press **WRITE/ENTER**.

**SETNG/GRAPH (2<sup>nd</sup> part)** - The second press of SETNG/GRAPH will bring up the **graphics display** in the Main Display Pane. In this screen you can dry-run a program without moving the axes or risking tool damage from any programming errors. This function is far more powerful than using **DRY RUN**, because all of your offsets and travel limits can be checked before any attempt is made to move the axes. The risk of a crash during setup is greatly reduced. The **Graphics Screen** will display the programmed tool path and generate an alarm if there are any problems. Some of the features of the Graphics display are controlled by selections made in the Settings display, on the page titled GRAPHICS.

1. Press either **MEM** or **MDI** and select the program that you want to run in Graphics. Graphics will also run in the Edit Mode.
2. Press **SETNG/GRAPH** twice.
  - The top left line of the screen will list the GRAPHICS title. Above that line will list the mode you are in (MEM or MDI). The bottom lists explanations for use of function keys **F1** through **F4**.
  - The small window on the lower right side of the screen displays the whole table area during the simulation run, indicating the location of the tool and any zoom window. The center window of the display is a large window that represents a top-down perspective of the X and Y axes. This is where the tool path is displayed during graphic simulation of a CNC program.
3. Press **CYCLE START** to see all the X and Y-axis moves demonstrated.  
Note machine axis and spindle will not when graphic window is up.
4. To step through a program one block at a time in Graphics, press **SINGLE BLOCK**.
5. **F1** is a help key.
6. Press **F2** to zoom in on the Graphics view screen.
  - Use **PAGE DOWN** to zoom in further and **PAGE UP** to expand the view.
  - Use the **Cursor Keys** to position the new zoom window over the area you wish to zoom in on using the small window in the bottom right hand corner. Pressing **HOME** will display the whole table.
  - After positioning the desired zoom window, press **WRITE/ENTER** to accept the view and **CYCLE START** to see the new view.
  - **F3** slows the execution speed of the graphic simulation
  - **F4** speeds up the execution speed of simulation.

Use **SINGLE BLOCK** to step through a program in graphics to find any mistakes. During single block you can re-zoom your window to look at tool paths in tight corners etc. Also use position display to see find any discrepant values.

**HELP/CALC** – Will bring up a help POP UP relevant to the screen you are in. This provides information only pertaining to that screen. Pressing the **HELP/CALC** button again brings up a tabbed menu. With tabulated screens highlighting tab and pressing **WRITE/ENTER** key will open up respective tab. Pressing the **CANCEL** key will close the tab.

**Help** Opening up the Help tab brings you to the table of contents of the entire Mill Operators Manual. High light the topic of interest and press **WRITE/ENTER** will bring up subtopics on the area of interest. Select subtopic in similar fashion will bring up the relevant page in the manual.

**Search** The search tab will do a search of the manuals content for relevant information on a keyword. Type in the search term and press **F1**. Topics relevant to the keyword will appear. Highlight the topic and press **WRITE/ENTER** key to open.

**Drill Table** Displays a common drill sizes, decimal information and tap drill sizes.

**Calculator** Different calculator functions are available under this tab. The calculator gives ordinary calculations like addition, subtraction, multiplication and division in all tabs. It also will solve trig problems with information about triangles, circles, circle line tangent and circle- circle tangent. A milling and tapping tab will give you suggested cutting speeds and feeds per different materials and sized tools.

**Simple Calculator** It will calculate simple addition, subtraction, multiplication and division operations. Operations are listed as: LOAD + - \* /. These are selected using the left or right cursor arrow.

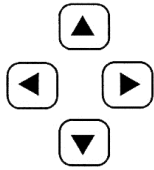
- To enter a number cursor on to LOAD; type the number you want to load and press **WRITE/ENTER**.
- To perform one of the arithmetic functions, enter the first number into the calculator window. Select the operation you want ( + - \* / ). Finally, enter the second number into the input buffer, press **WRITE/ENTER** to perform the calculation.

**Milling and Tapping** Help you solve values for feed rates SFM, RPM, and chip load under different conditions. It uses the three equations related to milling and tapping on page 67. The first one includes cutter diameter with SFM and RPM. The second one includes RPM, number of flutes, feed rate and chip load. The third one includes thread pitch, RPM and feed rate.

### The Milling & Tapping Tab

MILLING:	Cutter Diameter	1.2500 IN	(entered)
	Surface Speed	210.0000 FT/MIN	(entered)
	<b>RPM</b>	<b><u>642</u></b>	<b>(calculated)</b>
	Flutes	4	(entered)
	<b>Feed</b>	<b><u>12.8343 FT/MIN</u></b>	<b>(calculated)</b>
	Chip Load	0.0005 IN	(entered)
TAPPING:	Threads	16.0/IN	(entered)
	RPM	500	(entered)
	<b>FEED</b>	<b><u>31.2500 IN/MIN</u></b>	<b>(calculated)</b>

## 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.

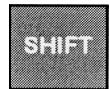
**Page Up/Down** – Used to change displays or move up/down one page when viewing a program.

**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.



**EOB** key enters the end-of-block character, which is displayed as a semicolon on the screen 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)

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.

## ***6 and 7 – Alpha Keys and Numeric Keys (continued)***

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 a program stored in memory. When you press **EDIT** two panes appear at the top of the screen. In the left pane the active program appears. In the right an inactive program appears or the select program screen appears. On the bottom left a editor help pane appears and on the right a clipboard pane. Editing may be performed in either the active or inactive panes. Pressing **EDIT** toggles between the two panes, (changes background to white). To call up a program from memory and put it in one of the edit panes press **SELCT/PROG**. Highlight the program desired by using the up or down cursor buttons and press **WRITE/ENTER**.

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 edit changes.

**F1 KEY** While in the edit mode pressing **F1** will bring up an edit pop up window. Using the sideways cursor buttons will toggle thru **HELP**, **MODIFY**, **SEARCH**, **EDIT AND PROGRAM MENUS**. The up and down buttons will cursor thru the different options in each of the above.

**MODIFY** Gives options on changing line numbers.

**SEARCH** Will perform a search and gives the option of replacing text.

**EDIT** Gives option of cutting or copying and pasting to a clipboard and to another program.

**PROGRAM** Gives options of creating new program, selecting a program from list to edit, duplication of programs, switching from left to right side of window panes.

### Background

**Edit** When a program is being run pushing the edit will bring up the Background Edit pane in the Main Display Pane. Simple edits may be performed on the program that is being run or another program. The edits on the running program will not take place until after the current cycle has completed.



<b><u>MEM</u></b>	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.
<b>SINGLE BLOCK</b>	When depressed <b>SINGLE BLOCK</b> is highlighted in black and will appear on the bottom of the CRT. 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.
<b>DRY RUN</b>	Used to check machine movement without cutting a part. In dry run the machine runs at one feed rate. With the availability of graphics which show visually what the machine tool path is this mode is rarely used.
<b>OPTION STOP</b>	When <b>OPTION STOP</b> is depressed program will stop at any M01 which is in the program. Normally M01s are placed after a tool is run 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.
<b>BLOCK DELETE</b>	When this button is depressed any block with a slash (/) in it is ignored or skipped.
<b><u>MDI</u> DNC</b>	(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.
<b>COOLNT</b>	Turns coolant on and off manually
<b>ORIENT SPINDLE</b>	Rotates and locks spindle to specific angle. Used when lining up tools where spindle orientation may be an issue such as boring heads.
<b>ATC FWD</b>	Rotates turret to next tool and performs tool change - also used to call up specific tools or pots. Enter tool number (T1) and press <b>ATC FWD</b> .
<b>ATC REV</b>	Rotates turret to previous tool and performs tool change - also used to call up specific tools or pots. Enter tool number (T1) and press <b>ATC REV</b> .
<b>HAND JOG</b>	Puts machine in jog mode for set ups. 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.
- 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** Will bring up list of programs in a tab format. Pressing **Cancel** will return you to tab at top usually MEM or USB. Cursor to left or right for which list one wants. Pressing **Enter** will open a list of programs. Cursor UP (^) or DOWN (v) to program desired. Select the desired programs to be moved by pressing **WRITE/ENTER**. This will put a check mark beside it. **F2** will copy selected program or programs to be moved. A pop up menu will ask where you want the selected programs to be copied.
- SELECT PROG** After highlighting a program from List Program with up or down cursor pressing this button will place the program in the Active Program Pane. This is the program that will run the CNC machine in the Memory mode. Use in the Edit mode in the Main Display will enter selected program in the Main Display pane for editing.
- SEND** Will send a selected program or programs out thru RS-232 serial port
- RECV** Will get machine ready to receive program from RS-232 serial port.
- ERASE PROG** Will erase highlighted program or programs. A prompt will appear asking if you want to delete selected program asking for Y/N.

## Settings

**Scrolling through Settings with Jog Handle** - The jog handle can now be used to scroll through the settings. In previous versions, the jog handle could only be used to scroll through (cursor-highlight) the parameters, but not the settings. This has been corrected. (Any Mill Control Ver. 10.15 and above; any Lathe Control Ver. 3.05 and above.)

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.

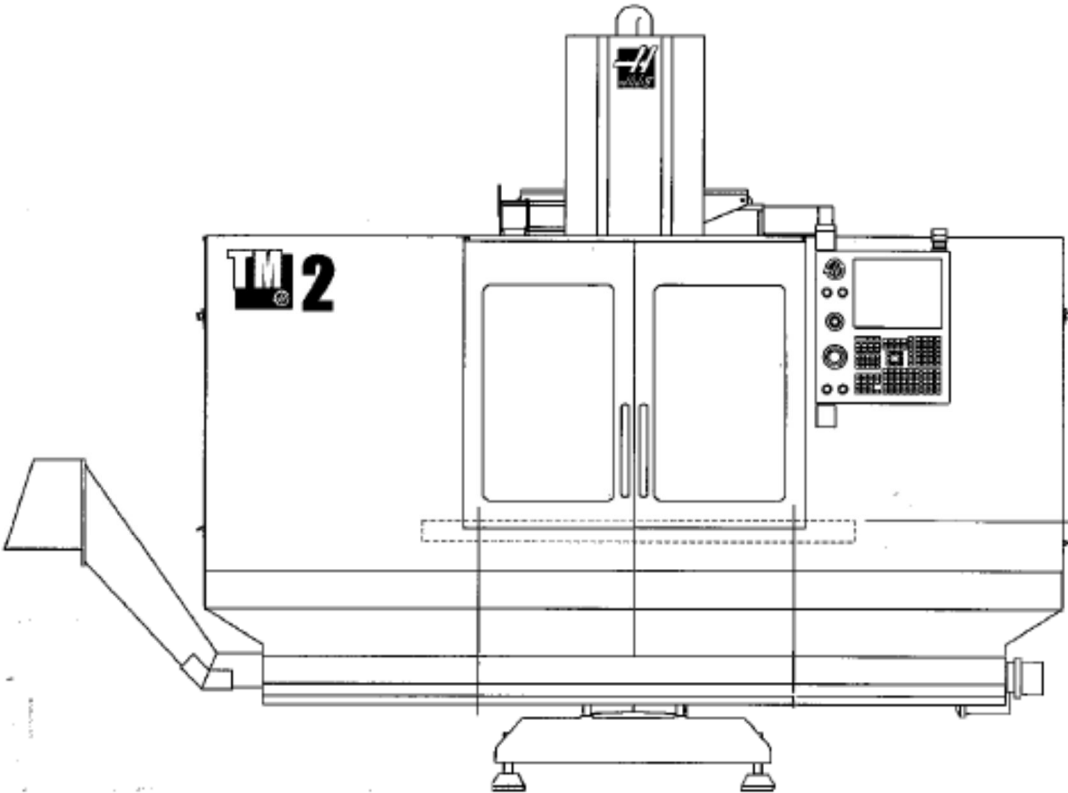
- Setting 33**     **COORDINATE SYSTEM** – This setting changes the way the G92/G52 offset system works. It can be set to Fanuc, Haas, or Yasnac. Normally it is set to Haas.
- Setting 36**     **PROGRAM RESTART** – When it is OFF, starting a program from anywhere other than the beginning of a program 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)*
- Setting 201** **SHOW ONLY WORK and Tool Offsets in Use** – With this feature turned on only the Work and Tool Offsets used within a program will be shown on the respective pages. To activate the program first must be run in graphics or memory regular mode.



# ***Toolroom Mill Orientation and Walk Around***

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

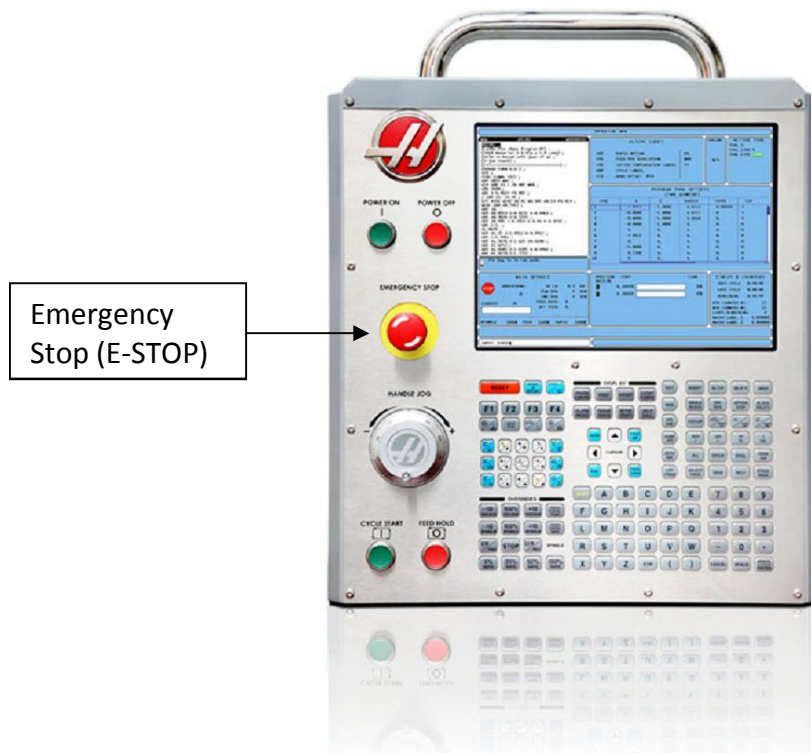


FRONT VIEW OF TM – SERIES MILL

# TOOLROOM MILL SAFETY

## Emergency Stop Switch

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





## ***Maintenance of the TM Series Mill***

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

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

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

Productivity, Inc. - Service and Parts



[www.productivity.com](http://www.productivity.com)

MN – 15150 – 25<sup>th</sup> Avenue North, Plymouth, MN 55447

Ph 763-476-8600 Fax 763-476-4092

IA – 9440 Atlantic Drive SW, Cedar Rapids, IA 52404

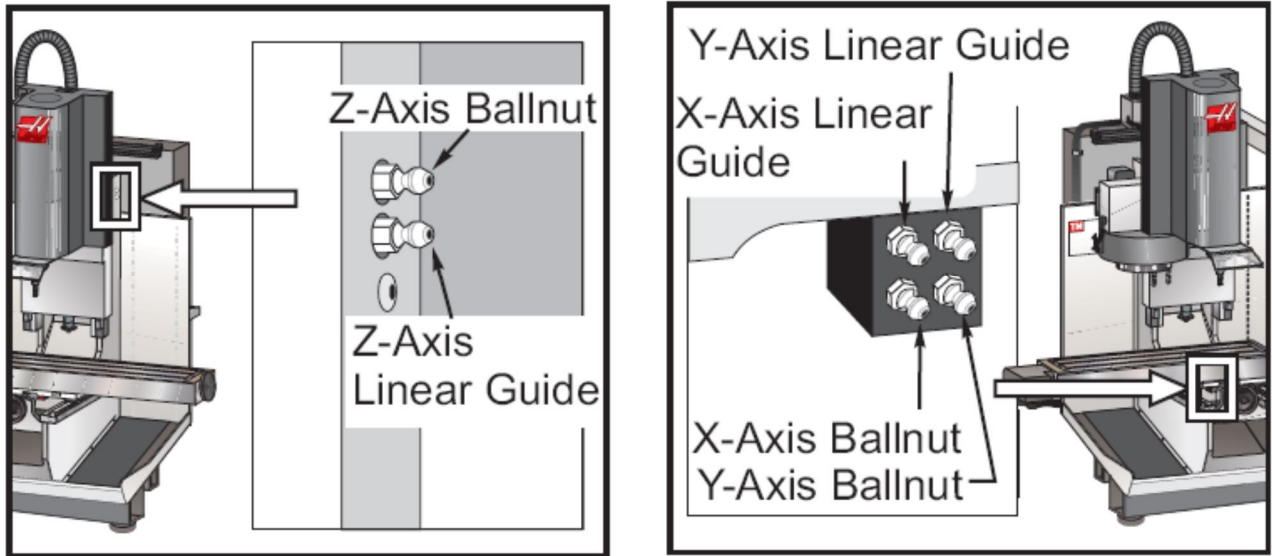
Ph 319-734-3403 Fax 319-632-4289

NE – 8402 South 117<sup>th</sup> Street, Omaha, NE 68128

Ph 402-330-2323 Fax 402-330-6901

## Grease Points

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

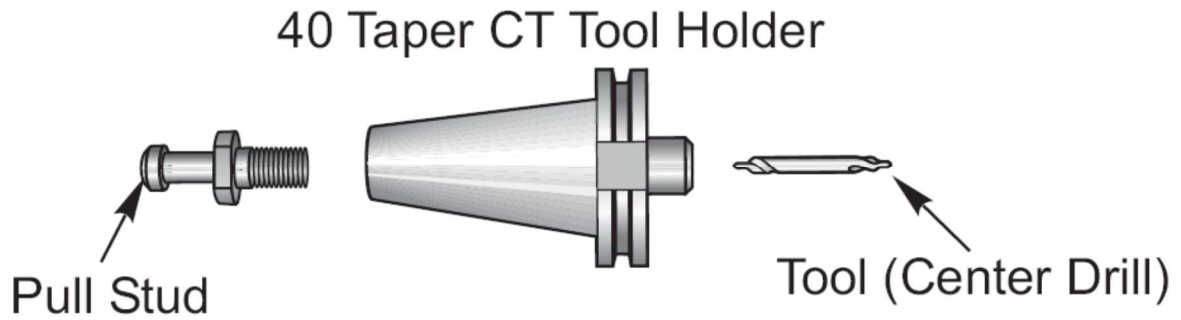


**TM Series Lubrication Points**

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

### ***CAT 40 Tooling:***

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



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

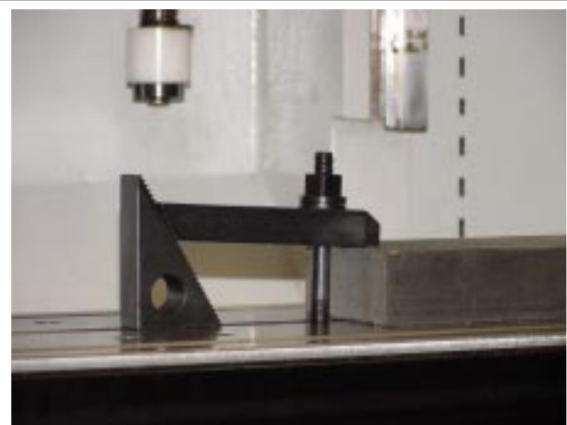
## ***Mill Work-Holding Options***

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

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



**Vise**



**Toe Clamp**



**3-Jaw Chuck**

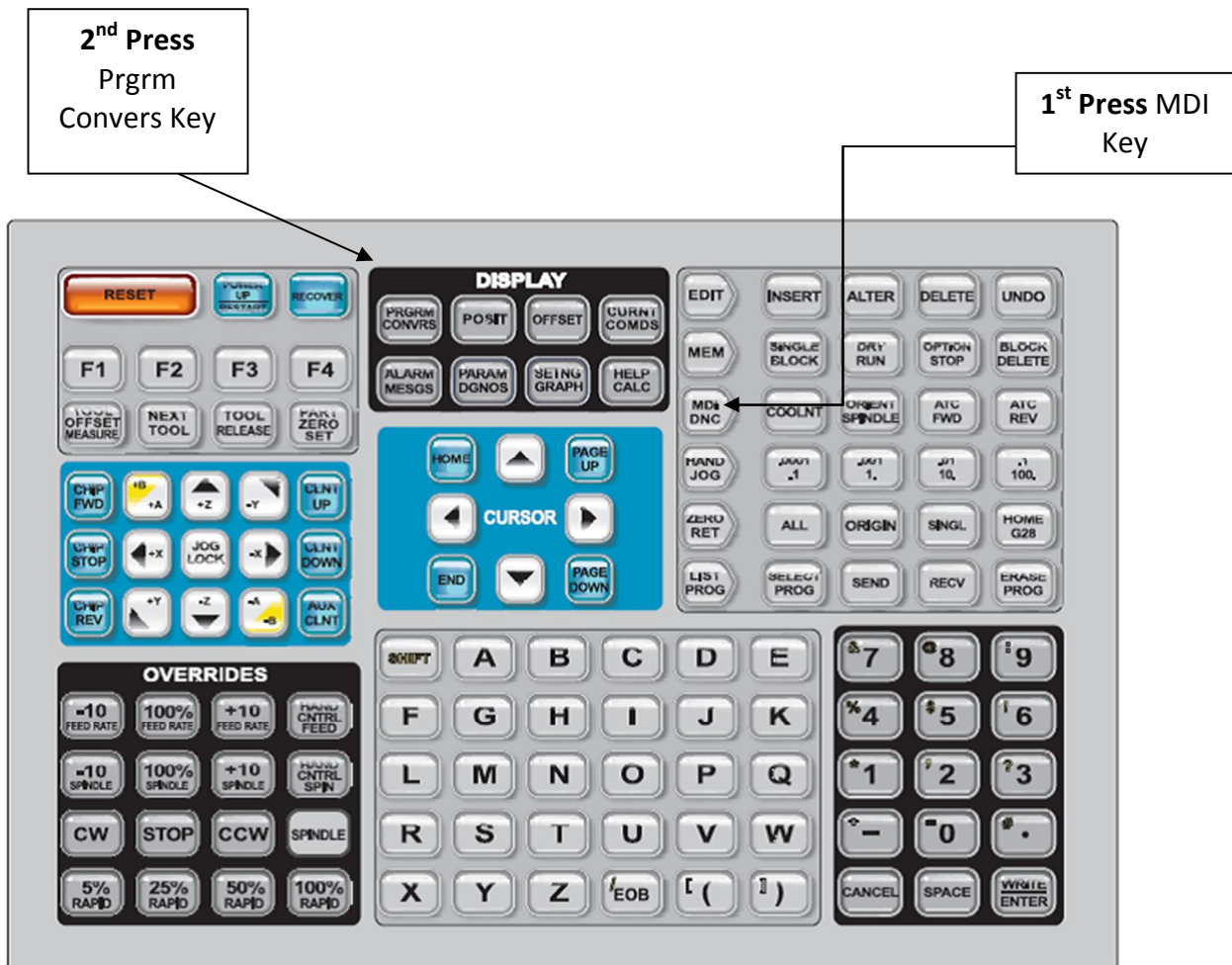


# Intuitive Programming System

## Entering the IPS

As shown below first press MDI key. Machine must be in **MDI mode** to access IPS

Then press Program Conversational key (**PRGM/CONVRS**). This highlights and makes the IPS screen active.



Below is the IPS Screen:

EDIT: IPS JOG

MDI N00000999

```

(PROBE - OUTSIDE CORNER) ;
G65 P9995 W54. A18. D3. E0.5 H2. I2. ;
;
M30 ;
;
        
```


MANUAL **SETUP** FACE DRILL POCKET MILLING ENGRAVING VQC

SETUP

WORK TOOL TOOL PROBE CALIBRATION WORK PROBE CALIBRATION

The key is in setup mode.  
 Max spindle RPM with the door open is 750  
 Max rapid is 25%

MAIN SPINDLE



SPEED(RPM) 0

LOAD(%) 0%

SP LD: 0.0 KW

SURF SPD: 0 FPM

CHIP LOAD: 0.00000

FEED RATE: 80.

ACT FEED: 80.

SPINDLE 100% FEED 100% RAPID 25%

POSITION: (IN)	JOG RATE 0.0100	LOAD
MACHINE		
X <span style="background-color: yellow;">-23.4637</span>	<input type="text"/>	4%
Y 0.0000	<input type="text"/>	5%
Z 0.0000	<input style="background-color: green;" type="text"/>	53%

ACTIVE TOOL

T20 - P5 T25

LOAD 0 LIFE 100%

TOOL TYPE NOT SET

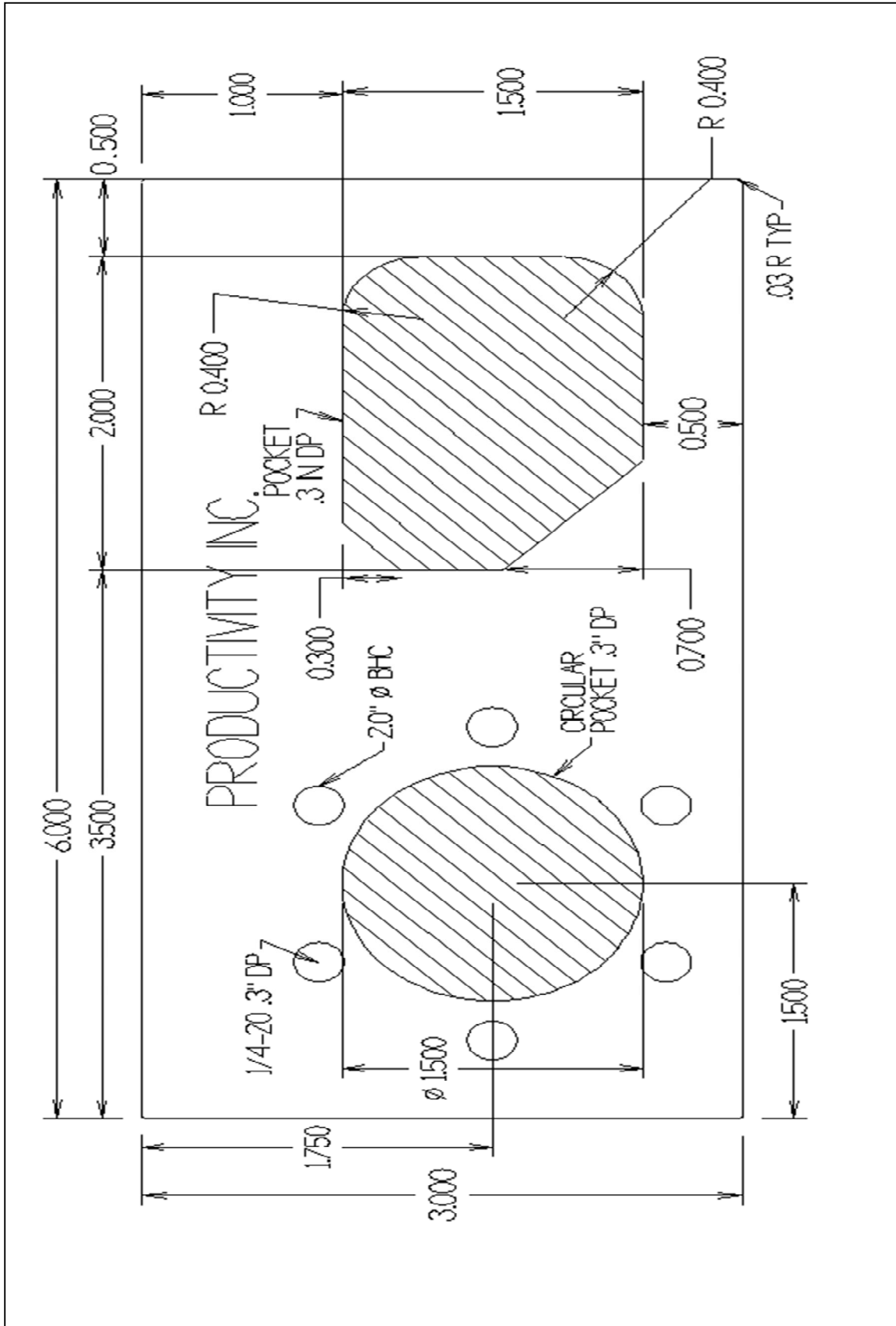
Use the [LEFT / RIGHT] cursor keys to navigate between tabs. Press [WRITE / ENTER] to activate mode. Press CANCEL to exit current node

INPUT:





Create G-code using the IPS to machine the top of a 3.06"x7"x3.0" block in a Haas Tool Room Mill.

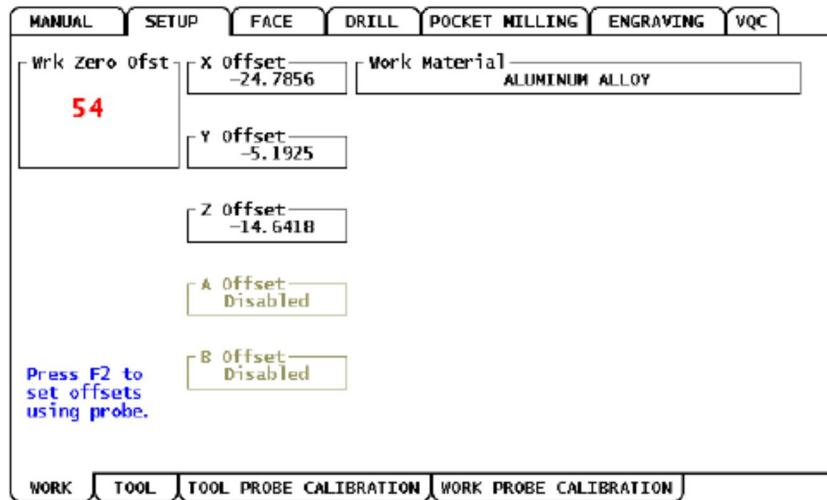




## Navigating Tabulated Screens

Select tab by moving the cursor key either right or left and highlight in red. Then press ENTER key to open that tab or move from top tabs to lower tabs as shown below. When the tab highlighted in red jumps to the bottom list of tabs, cursor to the one that is desired then press enter key. To close screen press cancel key. Depressing cancel key will make top tabs active.

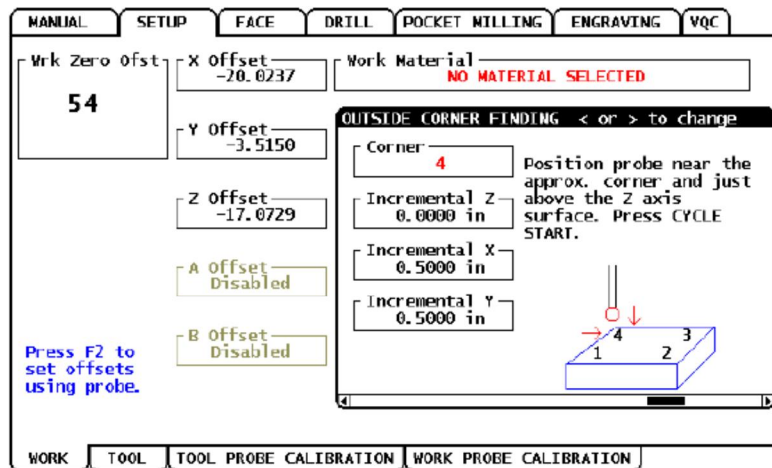
Below the top Setup tab on top was selected and the bottom Work tab selected and opened. On this screen different work offsets may be selected, by pressing the up and down arrow keys when the work offset box is highlighted in red. In this instance G54 was selected.



Press the UP and DOWN arrows to change displayed Work Zero Offset.

Press CANCEL to exit current mode

To set the work offset with a probe press F2.

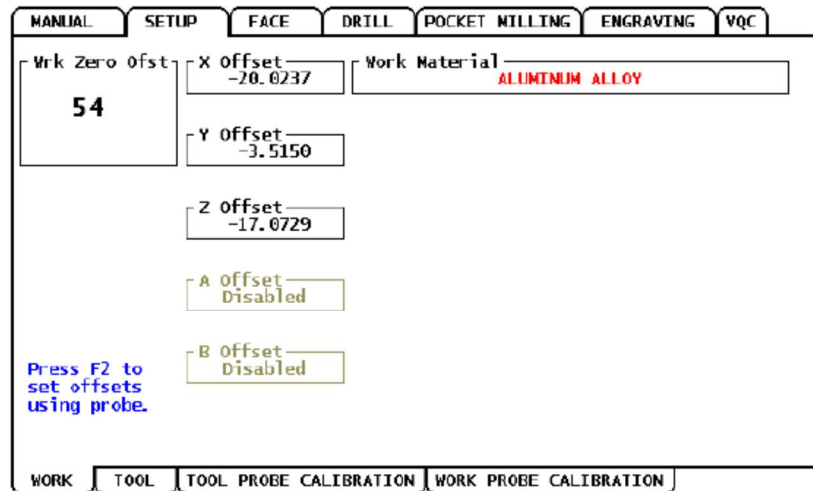


Corner to be probed. 1=bottom left, 2=bottom right, 3=top right, 4=top left - default

Press CANCEL to exit current mode

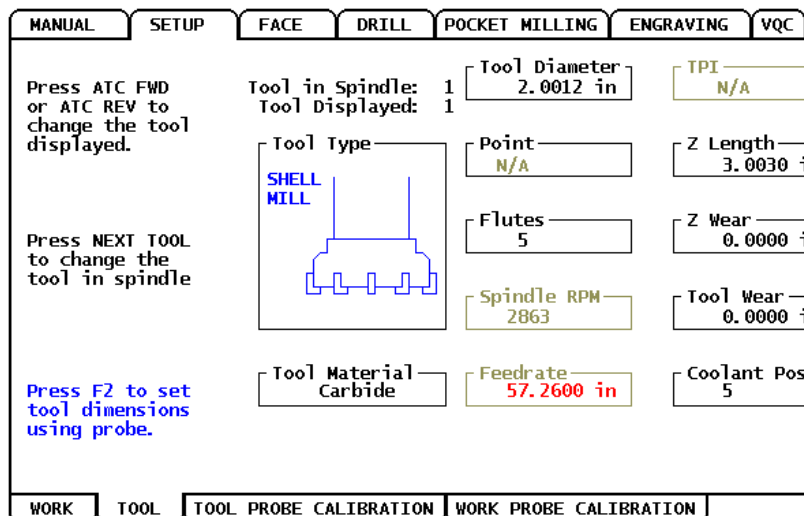
First select which method you want to probe by pressing the left or right arrow keys. Then follow directions on positioning probe, enter any relevant information and press cycle start to probe. Haas provides 12 different ways to probe for different types of work offsets.

Haas has a library of recommended feeds and speeds which may be accessed for different work materials. If you want the software in the machine to select feeds and speeds highlight the Work Material box in red and use the up and down cursor keys to select the material you are working on. In the instance below an Aluminum Alloy has been selected. If you want to determine the feeds and speeds yourself select “no materials selected”.



Use the UP and DOWN cursor arrows to change material type. Press WRITE to select.  
Press CANCEL to exit current mode

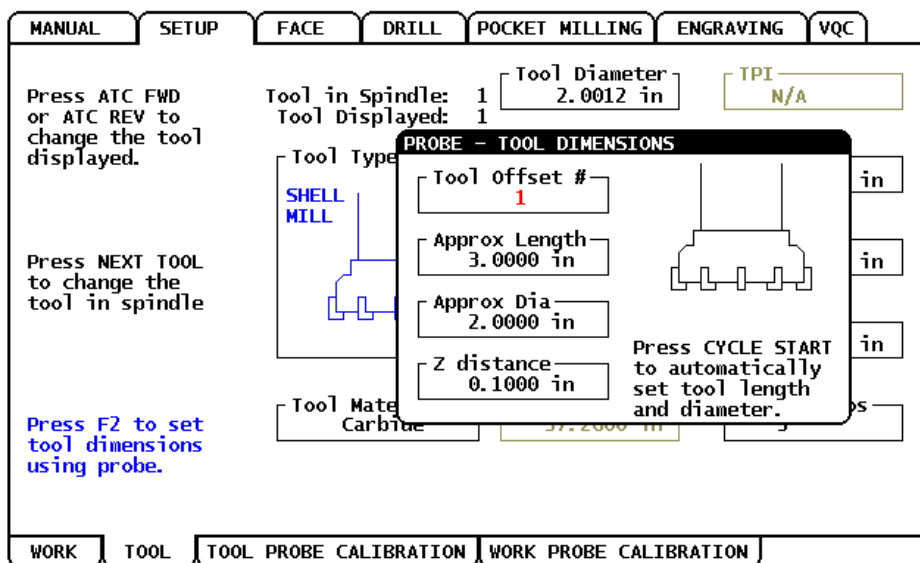
After setting up the work offset the tool offset page must be accessed. This is accomplished by first closing the work offset page by pressing the **Cancel** key. Cursor to the **TOOL** tab and open it up by pressing the **ENTER** key. Use up and down keys to cursor to the Tool Type. Use the right and left arrow keys to select the tool type and press enter. Below a Shell Mill has been selected. Next cursor down to the Tool Material and select tool material type with the right and left key and press **ENTER**. Below Carbide has been selected. Next cursor to Tool Diameter and Flutes and enter appropriate information. As shown below Spindle RPM and Feed rate will be determined. If user wants to calculate feed rate and RPM choose Tool Material and select **User** then feed rate and RPM for the particular tool can be entered in Setup window.



In the same page tool offsets may be set. Tools may be manually touched off by using **Tool Offset Measure** key. They also may be set using a probe by pressing **F2**. Enter appropriate information in the pop up and press **Cycle Start**. The tool in the spindle will then go thru probe routine to touch off tool using the table probe.

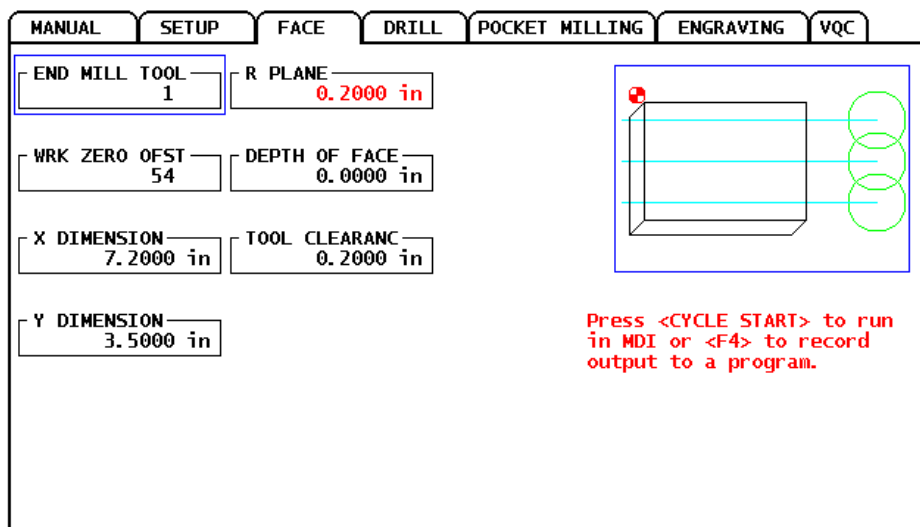
After the tool length offset is set pressing the **NEXT TOOL** key will index to the next tool. Pressing **ATC FWD** or **ATC REV** will change to the tool displayed in the window.

Both the Work tab page and the Tool tab page must be set for all the tools used on a part before using the rest of the IPS functions. Particularly the Tool Tab page must have information on RPM and feed rate if the IPS is utilized. Below is Tool #1 set up page for a 2 inch slab mill.



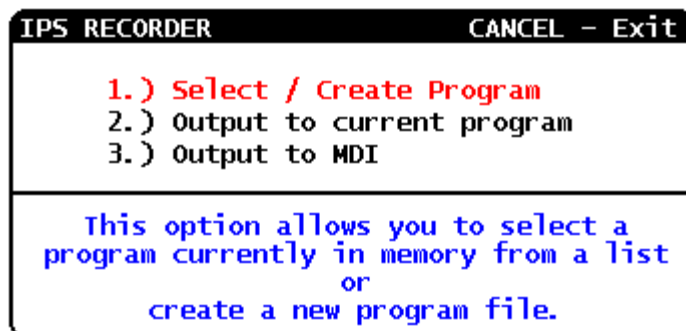
### Facing the part off using IPS:

After setting Work and Tool offsets using IPS press the **Cancel** key twice to get to the top of the tabulated IPS screen. Then right arrow cursor key to the FACE tab and press the ENTER key. Note using this function the **upper left hand corner must** be designated the part zero.



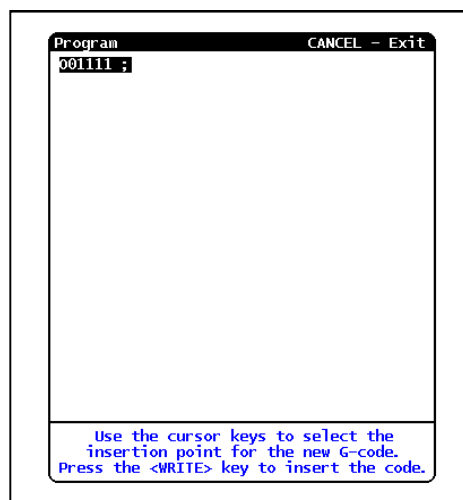
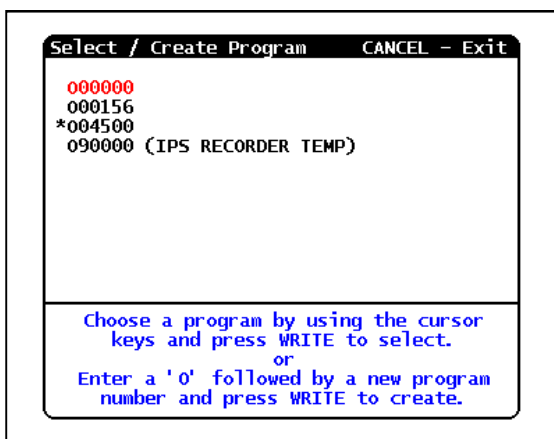
Using the up and down cursor keys enter all information required in the boxes. The X and Y dimensions refer to the width in X and the width in Y of the part to be faced off. The depth of face asks how far below part Z 0 you want to face off. It is a positive number. Normally you would make this 0 and subtract the depth you want to face off in your work offset Z. Otherwise all the other functions used in IPS will be off .01 as in the screen shot in the opposite page.

With all information correctly input, the program created may be run in MDI by pressing CYCLE START as highlighted in red above. The preferred method would be to press **F4**. The following pop appears. The options below may be selected by either keying in, 1, 2 or 3. They also may be selected by using the up and down arrow key, highlighting the desired one in red and pressing the **ENTER** key.



If only one part is going to be run than option 3 will work where the program created may be run in MDI mode and deleted.

If more than one part is going to be run select 1 or 2. For demonstration 1 was selected. See below left. Instead of inserting code in a specific program a new one was created by keying in O1111 and pressing the **ENTER** key. A new program appears in the pop up, below right. In this screen the proper location for inserting the code is selected. The up and down cursor key selects the location for insertion of the code. If satisfied with correct insertion press **ENTER**.



The screen below shows the code created in the new program O1111. In this screen one can verify if the code has been inserted in the right location. Using the up and down cursor key the entire program may be checked to see if the code has been inserted in the right location. If the codes has been put in the right location press the **ENTER** key.

```

Program                                CANCEL - Exit
001111 ;
;
(FACING) ;
;
T1 M06 ;
G00 G90 G54 X7.2 Y-3.3333 ;
S2865 M03 ;
G43 H01 Z0.2 M08 ;
;
(PASS 1) ;
G90 ;
G00 Z0. ;
G01 X-1.2 F57.3 ;
G00 Z0.2 ;
G00 G54 X7.2 ;
G91 ;
G00 Y1.3333 ;
;
(PASS 2) ;
G90 ;
G00 Z0. ;

```

Press <WRITE> to continue...

```

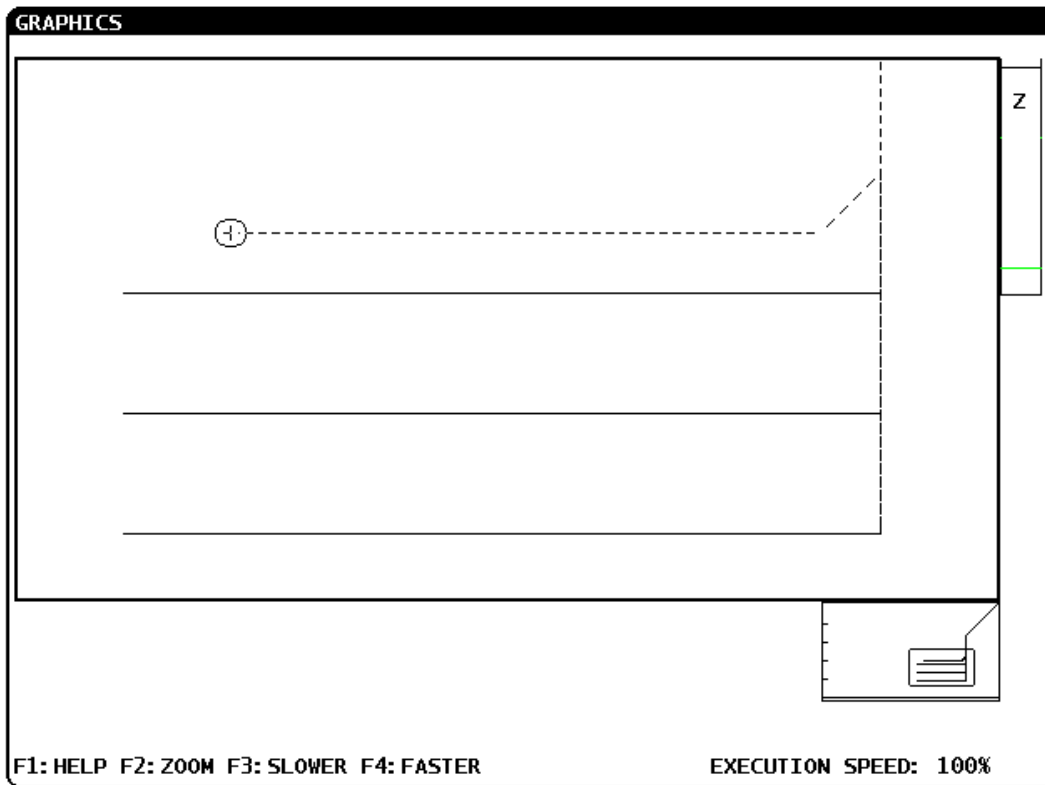
MEM                                001111                                N00000004
001111 ;
;
(FACING) ;
;
T1 M06 ;
G00 G90 G54 X7.2 Y-3.3333 ;
S2865 M03 ;
G43 H01 Z0.2 M08 ;
;
(PASS 1) ;
G90 ;
G00 Z0. ;
G01 X-1.2 F57.3 ;
G00 Z0.2 ;
G00 G54 X7.2 ;
G91 ;
G00 Y1.3333 ;
;
(PASS 2) ;
G90 ;
G00 Z0. ;
G01 X-1.2 F57.3 ;
G00 Z0.2 ;
G00 G54 X7.2 ;
G91 ;
G00 Y1.3333 ;
;
(PASS 3) ;

```

The key is in run mode.

This places the new program in **active memory** above in the upper left window.

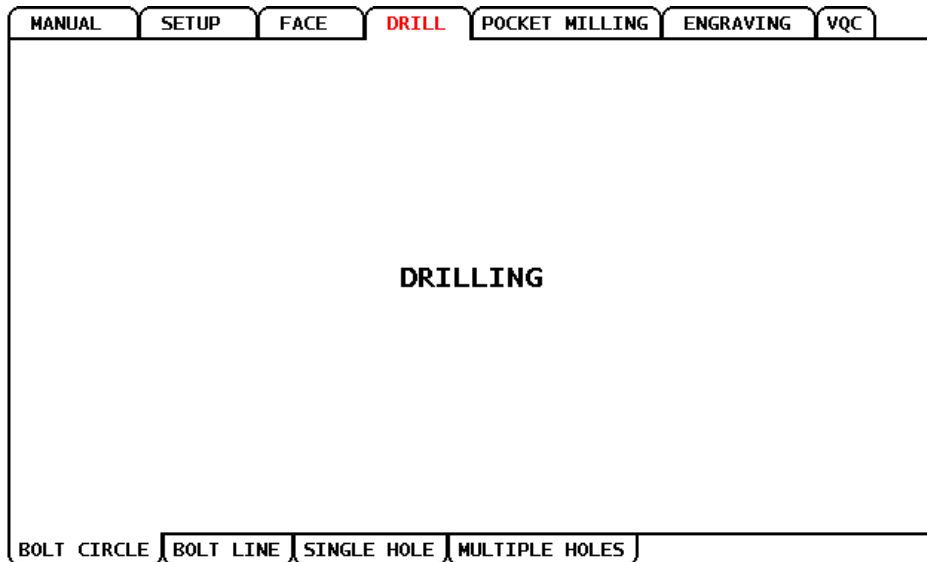
The program may then be verified in Graphics below



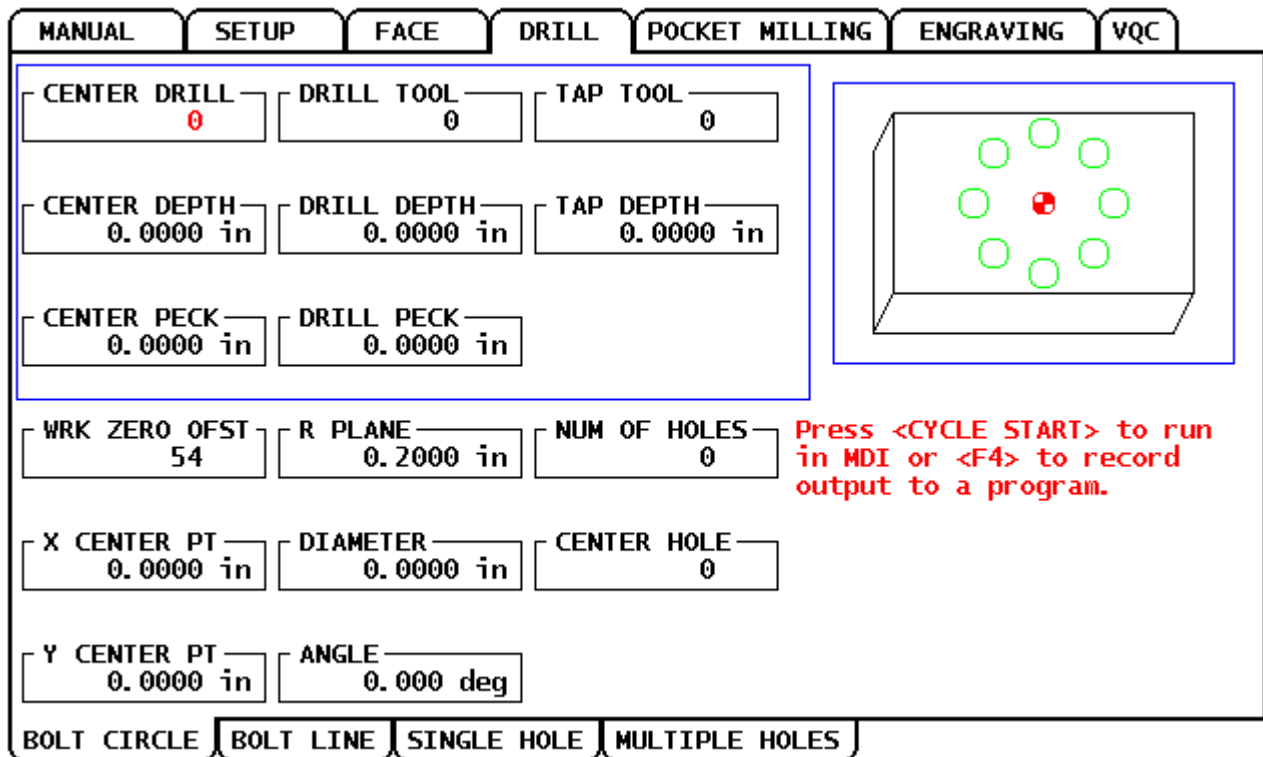


## Drilling and Tapping using IPS

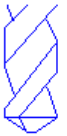
With the IPS screen up cursor to highlight in red the Drilling Tab.



IPS provides four different options for Drilling and Tapping: Bolt Hole Circle, Bolt Line, Single Hole and Multiple Holes. Press **ENTER** key to get to the bottom tabs, cursor to highlight Bolt Circle in red and press **ENTER**.

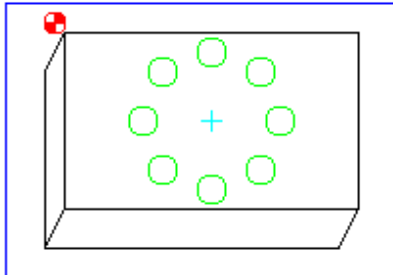


The above screen appears with options to Center Drill, Drill and Tap for a Bolt Hole Circle. Note before using this screen all tools used must be described and touched off in the Tool tab under the SET UP tab.

MANUAL	SETUP	FACE	DRILL	POCKET MILLING	ENGRAVING	VQC
Press ATC FWD or ATC REV to change the tool displayed.  Press NEXT TOOL to change the tool in spindle		Tool in Spindle: 4 Tool Displayed: 4	Tool Diameter 0.0890 in	TPI N/A		
		Tool Type DRILL 	Point 118 deg	Z Length 5.0000 in		
			Flutes 2	Z Wear 0.0000 in		
			Spindle RPM 7500	Tool Wear 0.0000 in		
		Tool Material High Speed Steel	Feedrate 49.9950 in	Coolant Pos 0		
WORK	TOOL					

Enter the included angle of the tool. Enter 0 or 180 to cancel. Press ENTER to record the value.

Above Tool 4 was touched off and set up as a high speed #43 drill with a 118 angle degree point. Each tool used in an IPS cycle must be set before in the tool set up tab. Speed and Feed have been calculated by the control.

MANUAL		SETUP		FACE		DRILL		POCKET MILLING		ENGRAVING		VQC	
CENTER DRILL 3		DRILL TOOL 4		TAP TOOL 5									
CENTER DEPTH 0.0300 in		DRILL DEPTH 0.5000 in		TAP DEPTH 0.3000 in									
CENTER PECK 0.0300 in		DRILL PECK 0.0900 in											
WRK ZERO OFST 54		R PLANE 0.2000 in		NUM OF HOLES 6		<p>Press &lt;CYCLE START&gt; to run in MDI or &lt;F4&gt; to record output to a program.</p>							
X CENTER PT 1.5000 in		DIAMETER 2.0000 in		CENTER HOLE 0									
Y CENTER PT -2.0000 in		ANGLE 0.000 deg											
BOLT CIRCLE		BOLT LINE		SINGLE HOLE		MULTIPLE HOLES							

Do you want a hole in the center of the pattern? Enter '0' for NO and '1' for YES.

In the Bolt Hole Circle Tab above Tool #3 is designated as the Center Drill, #4 the Drill, #5 the 4-40 Tap. The depths and pecking depths described and Work Offset G54 selected. Information describing the Bolt Hole Circle is entered in the bottom half: Bolt Hole Circle Diameter, starting location, and number of holes. Note that if the center of the circle is not drilled a 0 is placed in the Center Hole box. After all information has been entered press **F4**. Cursor to the 2<sup>nd</sup> option **Output to current program** and press **ENTER**.

```

Program          CANCEL - Exit
001111 ;
;
(FACING) ;
;
T1 M06 ;
G00 G90 G54 X7.2 Y-3.3333 ;
S2865 M03 ;
G43 H01 Z0.2 M08 ;
;
(PASS 1) ;
G90 ;
G00 Z0. ;
G01 X-1.2 F57.3 ;
G00 Z0.2 ;
G00 G54 X7.2 ;
G91 ;
G00 Y1.3333 ;
;
(PASS 2) ;
G90 ;
G00 Z0. ;

```

Use the cursor keys to select the insertion point for the new G-code. Press the <WRITE> key to insert the code.

```

Program          CANCEL - Exit
G00 Z0.2 ;
G00 G54 X7.2 ;
G91 ;
G00 Y1.3333 ;
;
(PASS 3) ;
G90 ;
G00 Z0. ;
G01 X-1.2 F57.3 ;
G00 Z0.2 ;
G00 G54 X7.2 ;
G91 ;
G00 Y1.3333 ;
;
M09 ;
M05 ;
G28 G91 Z0 ;
G00 G90 G54 X0 Y0 ;
M01 ;
3

```

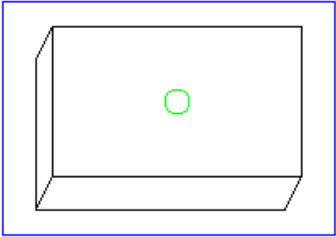
Use the cursor keys to select the insertion point for the new G-code. Press the <WRITE> key to insert the code.

Press **End** in the Cursor Panel to insert New Code at the end of the program.

Pressing **ENTER** two times adds new code at the end of program O1111 in the active memory:

```
MEM          001111          N000000000
(CIRCLE BOLT PATTERN) ;
;
(SPOT DRILL) ;
T3 M06 ;
G00 G90 G54 X1.5 Y-2. ;
S7500 M03 ;
G43 H03 Z0.2 M08 ;
G83 G98 Z-0.03 F49.995 Q0.03 L0 ;
G70 I1. J0. L6 ;
G00 G80 Z0.2 M09 ;
M05 ;
G28 G91 Z0 ;
G00 G90 G54 X0 Y0 ;
M01 ;
;
(DRILL) ;
T4 M06 ;
G00 G90 G54 X1.5 Y-2. ;
S7500 M03 ;
G43 H04 Z0.2 M08 ;
G83 G98 Z-0.5267 F49.995 Q0.09 L0 ;
G70 I1. J0. L6 ;
G00 G80 Z0.2 M09 ;
M05 ;
G28 G91 Z0 ;
G00 G90 G54 X0 Y0 ;
M01 ;
;
```

## Drill Application is Multiple Holes

MANUAL	SETUP	FACE	DRILL	POCKET MILLING	ENGRAVING	VQC														
CENTER DRILL 3	DRILL TOOL 4	TAP TOOL 5																		
CENTER DEPTH 0.0300 in	DRILL DEPTH 0.5000 in	TAP DEPTH 0.3000 in																		
CENTER PECK 0.0300 in	DRILL PECK 0.1000 in																			
WRK ZERO OFST 54	R PLANE 0.2000 in																			
<table border="1"> <thead> <tr> <th>HOLE</th> <th>X POINT</th> <th>Y POINT</th> </tr> </thead> <tbody> <tr> <td>1</td> <td>4.0000</td> <td>-1.0000</td> </tr> <tr> <td>2</td> <td>5.0000</td> <td>-1.0000</td> </tr> <tr> <td>3</td> <td>4.0000</td> <td>-2.0000</td> </tr> <tr> <td>4</td> <td>5.0000</td> <td>-2.0000</td> </tr> </tbody> </table>	HOLE	X POINT	Y POINT	1	4.0000	-1.0000	2	5.0000	-1.0000	3	4.0000	-2.0000	4	5.0000	-2.0000	<p>Press &lt;CYCLE START&gt; to run in MDI or &lt;F4&gt; to record output to a program.</p> <p>Press F1 to exit drill table.</p>				
HOLE	X POINT	Y POINT																		
1	4.0000	-1.0000																		
2	5.0000	-1.0000																		
3	4.0000	-2.0000																		
4	5.0000	-2.0000																		
BOLT CIRCLE	BOLT LINE	SINGLE HOLE	MULTIPLE HOLES																	

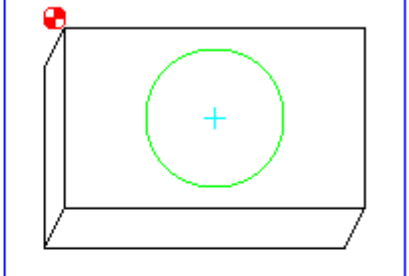
TOOL OFFSET MEAS - Records X & Y  
pts. INSERT - Adds a hole to table.  
DELETE - Removes selected hole.

This application is designed to incorporate coordinates of several holes at one time. Depth information is entered similar to Bolt Hole Circle application. The table at lower left is accessed by pressing **F1**. Hole locations are entered and new holes created by pressing the **INSERT** key. Code is created and inserted into programs as in the Bolt Hole Circle example.

## Circular Pocket Milling example

Highlighting in red the POCKET MILLING tab and pressing **ENTER** gives three different options. For example below is the **Circular Pocket application** screen:

MANUAL	SETUP	FACE	DRILL	POCKET MILLING	ENGRAVING	VQC		
CENTER DRILL	END MILL TOOL							
0	2							
HOLE DEPTH								
0.2500 in								
WRK ZERO OFST	R PLANE							
54	0.2000 in							
X START PT	DIA OF POCKET							
1.5000 in	1.5000 in							
Y START PT	TOTAL DEPTH	PASSES						
-2.0000 in	0.2500 in	3						
CIRCULAR	RECTANGLE	IRREGULAR						



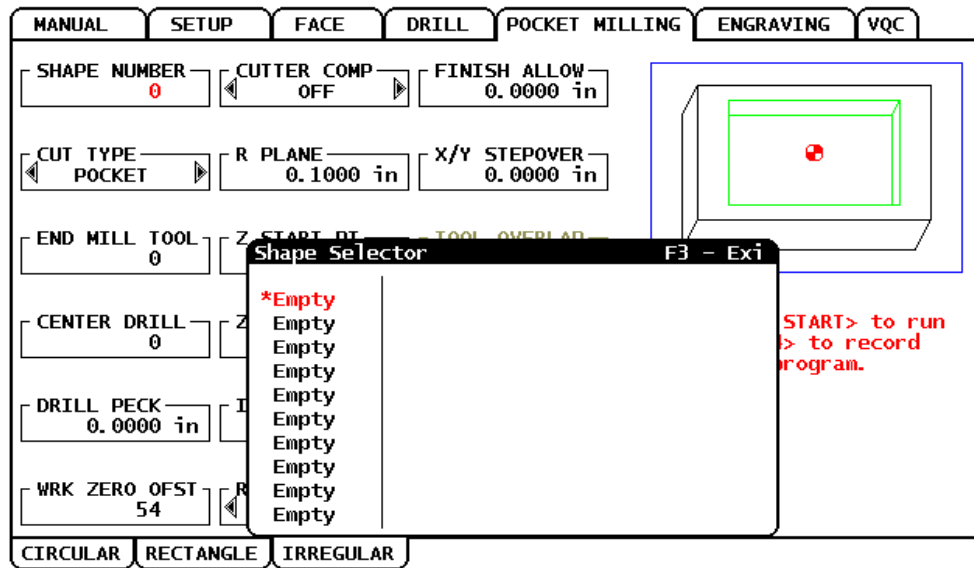
Press <CYCLE START> to run in MDI or <F4> to record output to a program.

Enter the center drill tool number.  
Enter '0' to skip center drilling cycle.

In the example above a 1.5 inch diameter pocket is milled. Note a center drill option is left out by putting "0" in the Center Drill box. The center of the pocket has the coordinates of X1.50 Y-2.0. Three passes of equal depth will be made to create the pocket using a G13 code.

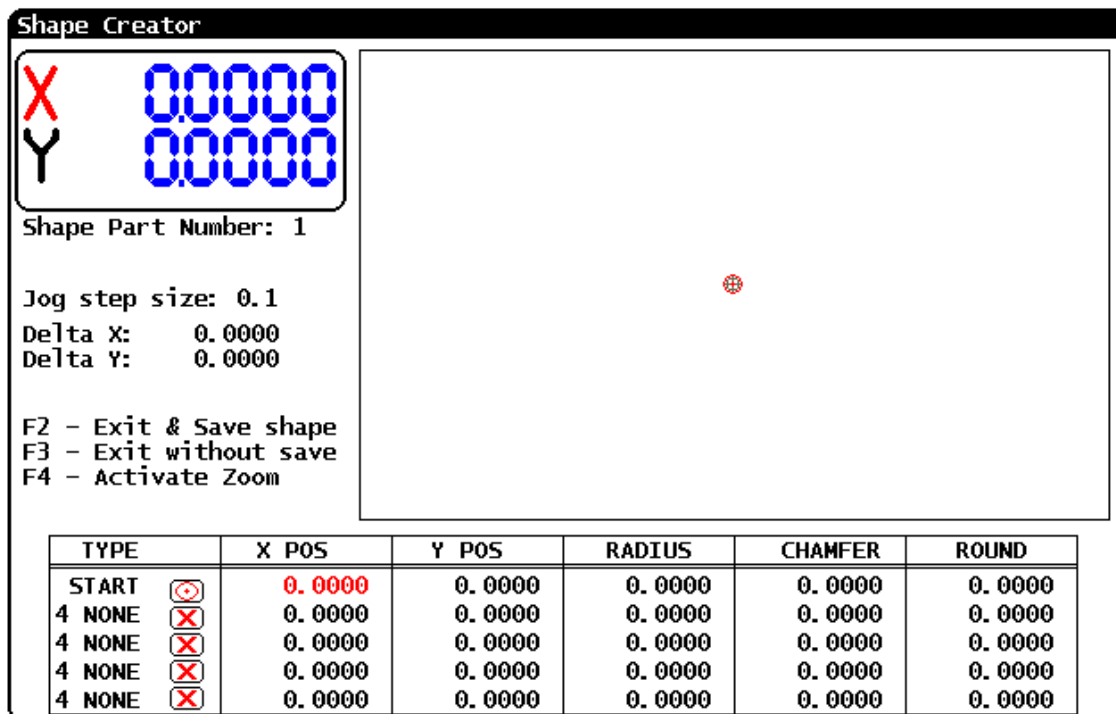
## Irregular Pocket Milling Example

Enter Irregular by highlighting in red **IRREGULAR** and pressing enter. Highlight **Shape Number** in red and press **ENTER**. This will get you access to the shape selector box. See below:



WRITE - Select/Create shape file.  
 ALTER - Edit existing shape file.  
 DELETE - Removes shape file.  
 Press **CANCEL** to exit current mode

Pressing **ENTER** will get you to the Shape Creator window.



WRITE - Select/Create shape file.  
 ALTER - Edit existing shape file.  
 DELETE - Removes shape file.  
 Press **CANCEL** to exit current mode

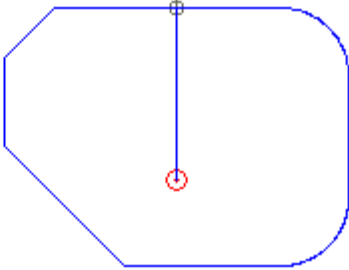
The start position is the first thing that is entered in the shape creator. Normally a hole will be drilled to depth before the pocket milling cycle is run. Enter the start position in X and Y. If no red circle appears it may be necessary to activate the zoom by pressing **F4**. With the zoom function center the circle inside the box.

Press **F4** to exit zoom. Next, cursor to the next line to the far left, and select feed by entering 1. Cursor to the Y position column and either use the hand wheel or enter the Y first position. Repeat this process dropping down to the next row first noting either a feed rate move **1**, a CCW (counter clockwise radial move) **2**, a CW (clockwise move)**3** in the left column.

Then indicate the next location in either X or Y in that row. Continue until the pocket becomes closed by going to the location at the first move. Corner chamfers or corner rounds may be entered at the corners after the closed pocket has been created. See below.

**Shape Creator**






X 4.5000  
Y -1.0000



Shape Part Number: 1

Jog step size: 0.1  
Delta X: 0.0000  
Delta Y: 1.0000

F2 - Exit & Save shape  
F3 - Exit without save  
F4 - Activate Zoom

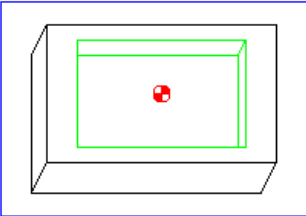
TYPE		X POS	Y POS	RADIUS	CHAMFER	ROUND
START		4.5000	-2.0000	0.0000	0.0000	0.0000
1 FEED		4.5000	-1.0000	0.0000	0.0000	0.0000
1 FEED		3.5000	-1.0000	0.0000	0.3000	0.0000
1 FEED		3.5000	-2.5000	0.0000	0.7000	0.0000
1 FEED		5.5000	-2.5000	0.0000	0.0000	0.4000

Use the Jog handle to select move type and press enter, or 1-FEED, 2-CW, 3-CCW or 4 for no move.

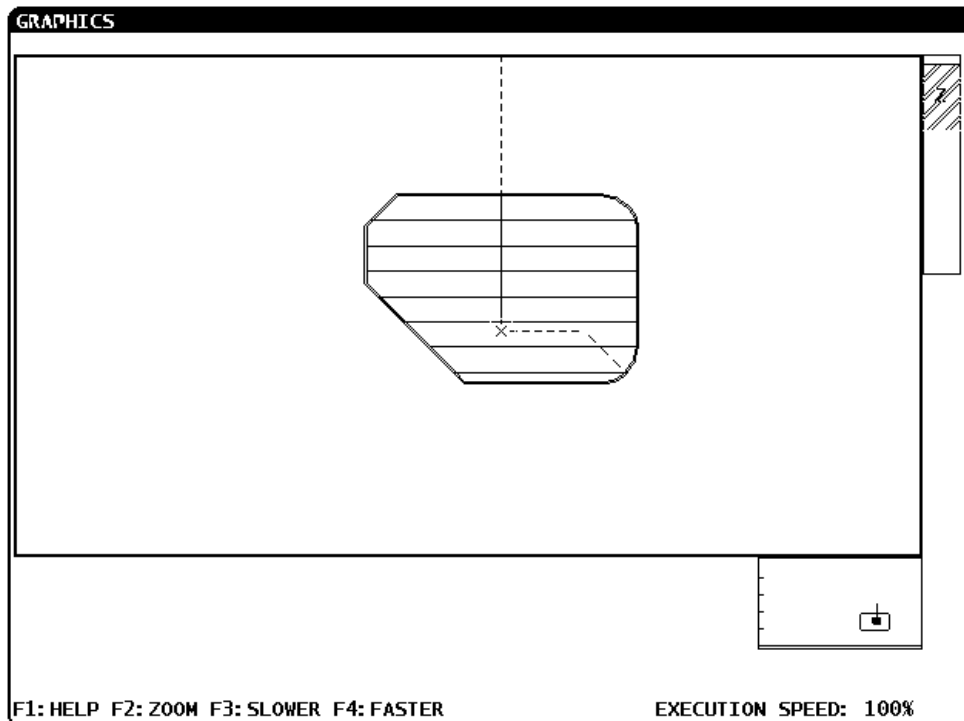
Then save the shape number 1 by pressing **F3**.



This brings you back to the irregular pocket screen. Then by using the up and down arrow cursor keys enter or select correct information. The first is to enter the shape number that was created. The next is to select (**POCKET**) in the Cut Type box. The rest of the information entered is straight forward. The **Rough Cut Dir** box asks which direction to cut along the x axis or the y axis. The **X/Y Steover** is the radial cut width of each pass. Normally this value should be under ½ the diameter of the end mill used. To check the code in the graphics box select **ON** and press **cycle start**.

MANUAL	SETUP	FACE	DRILL	POCKET MILLING	ENGRAVING	VQC
SHAPE NUMBER 1	CUTTER COMP LEFT	FINISH ALLOW 0.0100 in				
CUT TYPE POCKET	R PLANE 0.1000 in	X/Y STEPOVER 0.1500 in				
END MILL TOOL 2	Z START PT 0.0000 in	TOOL OVERLAP N/A				
CENTER DRILL 3	Z DIMENSION -0.3000 in	GRAPHICS ON				
DRILL PECK 0.3000 in	INC. DEPTH 0.1000 in		Press <CYCLE START> to run in MDI or <F4> to record output to a program.			
WRK ZERO OFST 54	ROUGH CUT DIR X AXIS					
CIRCULAR    RECTANGLE    IRREGULAR						

Enter shape program number, or press WRITE/ENTER without a number to bring up the Shape Selector window.  
Press **CANCEL** to exit current mode



Above: IPS code verified in graphics. This code may then be output to the current program using **F4** in the IPS window.

## Irregular Pocket Milling Application used for Contour Milling

The irregular pocket milling function may also be used for contour milling. The part created so far needs to be contour milled on the outside to form a 4" x 6" block. In the Irregular Pocket Milling screen the Cut Type needs to be changed to (**Countour**).

Next cursor to the Shape Number and open the Shape Selector Screen and press enter to create a new shape part number 2. With contour milling the start position is on the outside. The start position has coordinates of X3.0 Y.5. The start position must be at greater than the radius of the tool from the edge of the part. The contour is also going around the outside part in a clockwise motion. The contour of the part will then be climb milled. Also the corners have a .03 radius break by adding .03 in the Round column. See below.

**Shape Creator**

X 3.0000  
Y 0.5000

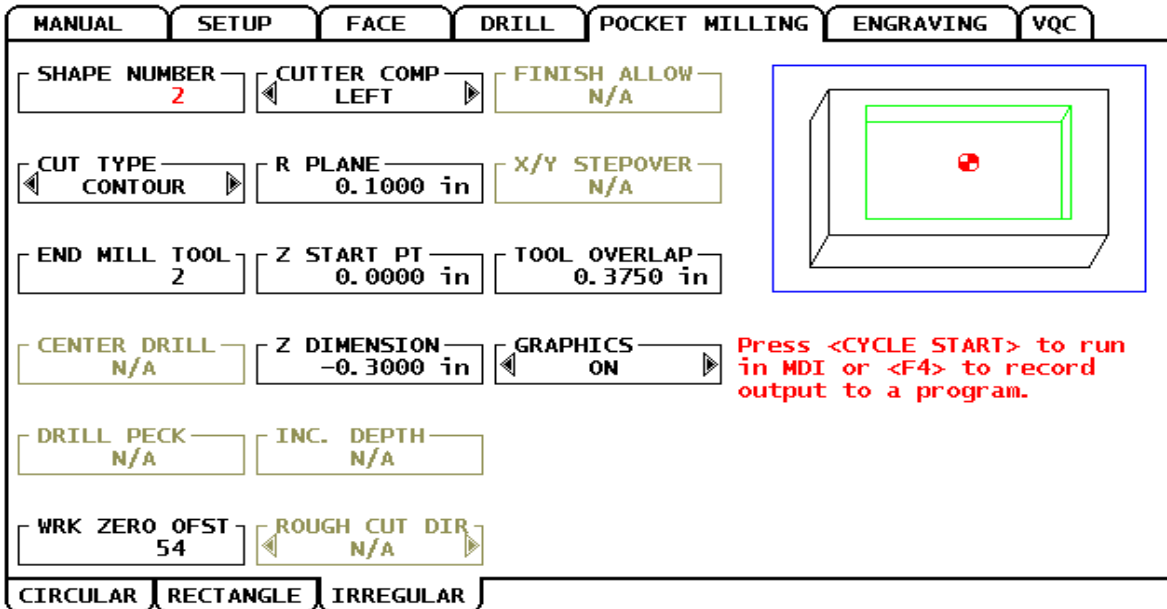
Shape Part Number: 2

Jog step size: 0.1  
Delta X: 3.0000  
Delta Y: 0.5000

F2 - Exit & Save shape  
F3 - Exit without save  
F4 - Activate Zoom

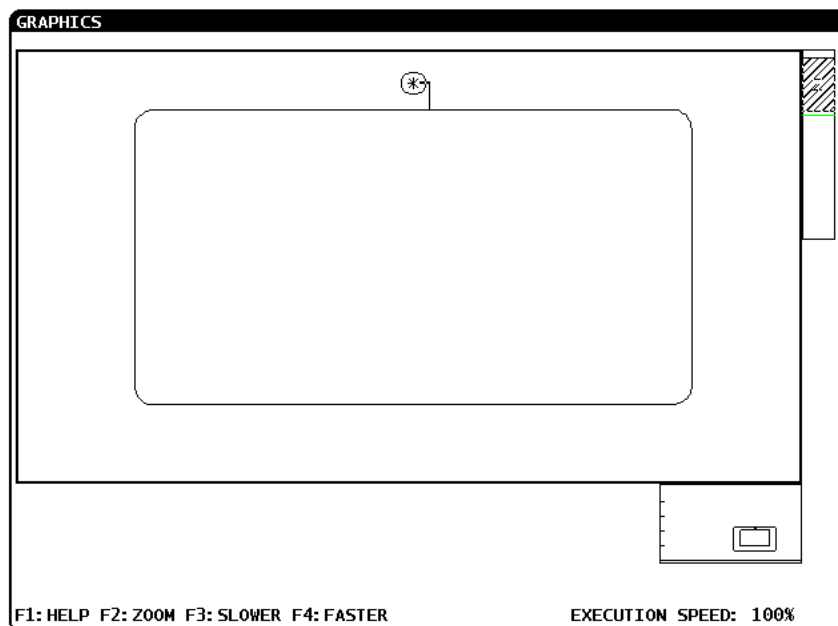
TYPE		X POS	Y POS	RADIUS	CHAMFER	ROUND
START	⏪	3.0000	0.5000	0.0000	0.0000	0.0000
1 FEED	⏩	3.0000	0.0000	0.0000	0.0000	0.0000
1 FEED	⏩	6.0000	0.0000	0.0000	0.0000	0.0300
1 FEED	⏩	6.0000	-3.0000	0.0000	0.0000	0.0300
1 FEED	⏩	0.0000	-3.0000	0.0000	0.0000	0.0300

Exit and Save the Shape Creator by pressing the **F2** key.



In the Irregular screen above note the cut type has been changed from Pocket to **Contour**. As we are climb cutting the Cutter Comp is **Left**. The depth of cut is the **Z dimension -0.300 in**. The **Tool Overlap** must be **greater** or equal to the cutter diameter **.375 in**.

With Graphics **On** pressing **Cycle Start** will verify the code. See below.

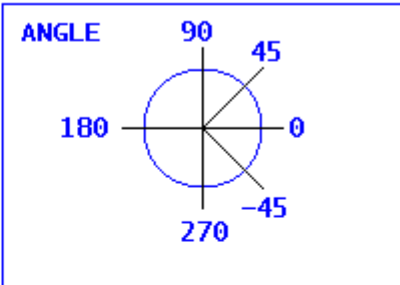




## Engraving

Using the Engraving function is relatively easy. For the engraving function to work the tool must be classified an **end mill** in the tool set up in IPS. For this example a 1/8 inch ball end mill has been placed in Tool #6.

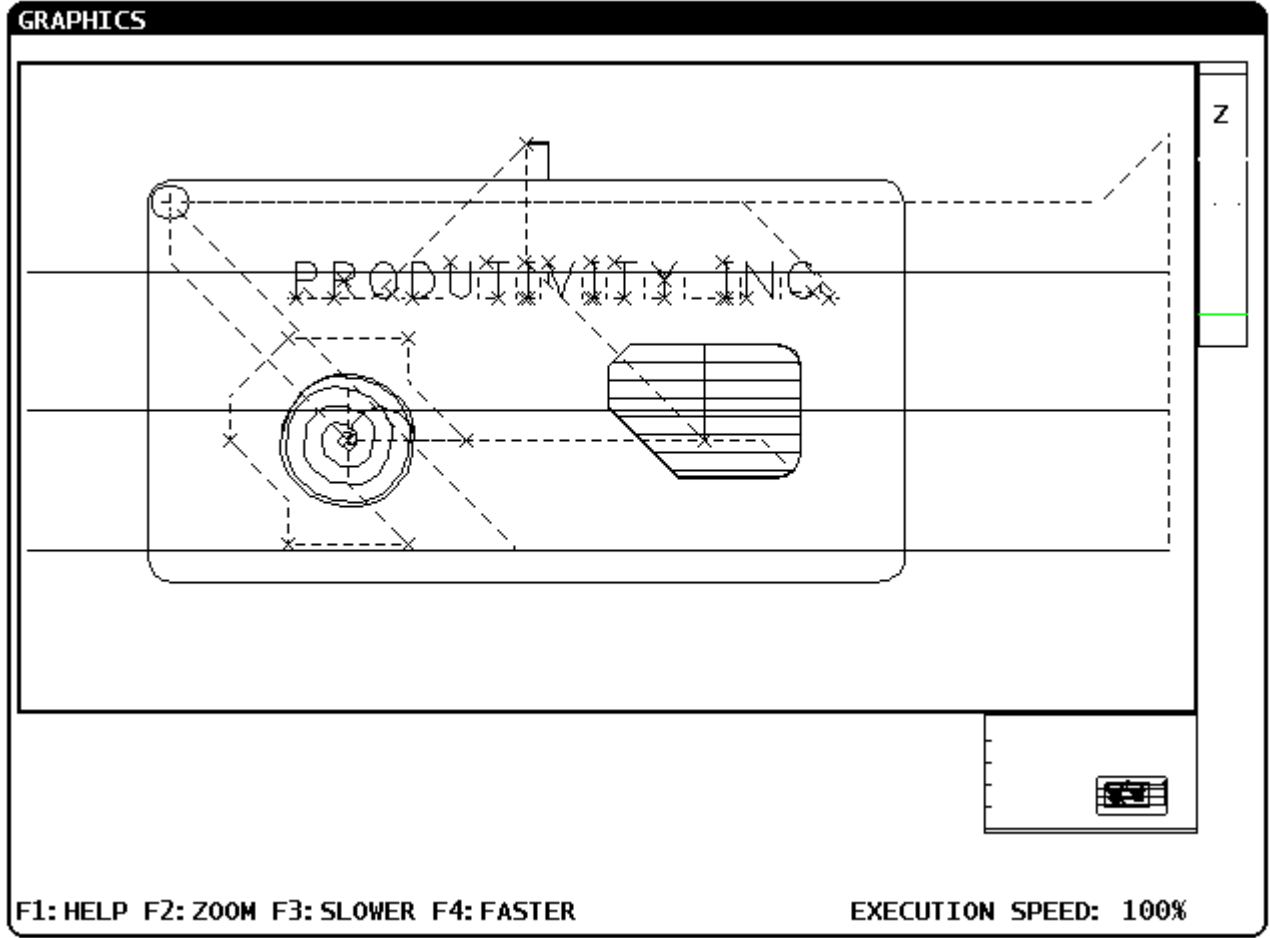
In the IPS screen select **Engraving tab** on top and open the **Straight Line** tab on the bottom. See below.

MANUAL	SETUP	FACE	DRILL	POCKET MILLING	ENGRAVING	VQC
TOOL <input type="text" value="6"/>		ANGLE <input type="text" value="0.000 deg"/>				
WRK ZERO OFST <input type="text" value="54"/>		TEXT HEIGHT <input type="text" value="0.3000 in"/>				
X START PT <input type="text" value="1.0000 in"/>		R PLANE <input type="text" value="0.0300 in"/>				
Y START PT <input type="text" value="-0.8000 in"/>		DEPTH <input type="text" value="0.0050 in"/>				
TEXT TO BE ENGRAVED <input type="text" value="PRODUCTIVITY INC."/>						
STRAIGHT LINE		SERIAL NUMBER		Press <CYCLE START> to run in MDI or <F4> to record output to a program.		

Enter text to be engraved.

The X and Y Start Pt is coordinates of the bottom of the first letter in your text. Above the bottom left of the P in Productivity would be X and Y position. If angle is desired the angle will be positive degrees from 0 at the three o'clock position. The **Text Height** must be entered and the depth of cut entered. Normally in engraving a relatively small **R Plane** number is used .03 in. Code may then be recorded in the current program. Text to be Engraved is entered in the box.

Program O1111 created using IPS verified by running in Graphics.



## ***Text of Program O1111***

%  
O01111  
(FACING)  
T1 M06  
G00 G90 G54 X8.4 Y-2.9167  
S2865 M03  
G43 H01 Z0.2 M08

(PASS 1)  
G90  
G00 Z0.  
G01 X-1.2 F57.3  
G00 Z0.2  
G00 G54 X8.4  
G91  
G00 Y1.1667

(PASS 2)  
G90  
G00 Z0.  
G01 X-1.2 F57.3  
G00 Z0.2  
G00 G54 X8.4  
G91  
G00 Y1.1667

(PASS 3)  
G90  
G00 Z0.  
G01 X-1.2 F57.3  
G00 Z0.2  
G00 G54 X8.4  
G91  
G00 Y1.1667  
M09  
M05  
G28 G91 Z0  
G00 G90 G54 X0 Y0  
M01

(CIRCLE BOLT PATTERN)  
(SPOT DRILL)  
T3 M06  
G00 G90 G54 X1.5 Y-2.  
S7500 M03  
G43 H03 Z0.2 M08  
G83 G98 Z-0.03 F49.995 Q0.03 L0  
G70 I1. J0. L6  
G00 G80 Z0.2 M09  
M05  
G28 G91 Z0  
G00 G90 G54 X0 Y0  
M01

(DRILL)  
T4 M06  
G00 G90 G54 X1.5 Y-2.  
S7500 M03  
G43 H04 Z0.2 M08  
G83 G98 Z-0.5267 F49.995 Q0.09 L0  
G70 I1. J0. L6  
G00 G80 Z0.2 M09  
M05  
G28 G91 Z0  
G00 G90 G54 X0 Y0  
M01

(TAP)  
T5 M06  
G00 G90 G54 X1.5 Y-2.  
S400  
G43 H05 Z0.2 M08  
G84 G98 Z-0.3 R0.2 F10. L0  
G70 I1. J0. L6  
G00 G80 Z0.2 M09  
M05  
G28 G91 Z0  
G00 G90 G54 X0 Y0  
M01

(COUNTER-CLOCKWISE CIRCULAR POCKET)

(END MILL)  
T2 M06  
G00 G90 G54 X1.5 Y-2.  
S6112 M03  
G43 H02 Z0.2 M08  
G01 Z0. F61.12  
N100 G13 G91 G01 Z-0.0833 I0.2625 K0.75 Q0.15 F61.12 L3 D02  
G00 G90 Z0.2 M09  
G28 G91 Z0 M05  
G00 G90 G54 X0 Y0  
M01

(POCKET MILLING)  
G00 G17 G40 G49 G80 G90

(DRILL START FOR POCKET)  
T3 M06  
G00 G54 G90 X4.5 Y-2. S7500 M03  
G43 H03 Z0.1 M08  
G83 G98 Z-0.3 X4.5 Y-2. R0.1 Q0.3 F49.995  
G80 M09  
G00 G53 Z0.



(G150 POCKET ROUTINE)

T2 M06

G00 G54 G90 X4.5 Y-2. S6112 M03

G43 H02 Z0.1 M08

G150 X4.5 Y-2. Z-0.3 G41 J0.15 K0.01 Q0.1 R0.1 P47297 D02 F61.12

G40 X4.5 Y-2.

G00 G53 Z0.

M01

(CONTOUR MILLING)

G00 G17 G40 G49 G80 G90

(CONTOUR ROUTINE)

T2 M06

G00 G54 G90 X3. Y0.5 S6112 M03

G43 H02 D02 Z0.1 M08

G01 Z0.3 G41 F61.12

M97 P12270

G40 X3. Y0.5

G00 G53 Z0.

M01

(TEXT ENGRAVING)

T6 M06

G00 G90 G98 G54 X1. Y-0.8

S7500 M03

G43 H06 Z0.03 M08

G47 P0 X1. Y-0.8 I0. J0.3 R0.03 Z-0.005 F75. E25. (PRODUCTIVITY INC.)

G00 G80 Z0.03 M09

M05

G28 G91 Z0

G00 G90 G54 X0 Y0

M01

M30

(MILLING ROUTINE SUB-PROGRAM)

N12270

G01 X3. Y0.5

G01 X3. Y0.

G01 X5.97 Y0.

G02 X6. Y-0.03 R0.03

G01 X6. Y-2.97

G02 X5.97 Y-3. R0.03

G01 X0.03 Y-3.

G02 X0. Y-2.97 R0.03

G01 X0. Y-0.03

G02 X0.03 Y0. R0.03

G01 X3. Y0.

G01 X3.375 Y0.0037

G01 X3.375 Y0.5037

M99

(MILLING ROUTINE SUB-PROGRAM)

N47297

G01 X4.5 Y-2.

G01 X4.5 Y-1.

G01 X3.8 Y-1.

G01 X3.5 Y-1.3

G01 X3.5 Y-1.8

G01 X4.2 Y-2.5

G01 X5.1 Y-2.5

G03 X5.5 Y-2.1 R0.4

G01 X5.5 Y-1.4

G03 X5.1 Y-1. R0.4

G01 X4.5 Y-1.

M99

M30