



HAAS SERVICE AND OPERATOR MANUAL ARCHIVE

Mill Operators Manual 96-8000N RevN English June 2006

- This content is for illustrative purposes.
- Historic machine Service Manuals are posted here to provide information for Haas machine owners.
- Publications are intended for use only with machines built at the time of original publication.
- As machine designs change the content of these publications can become obsolete.
- You should not do mechanical or electrical machine repairs or service procedures unless you are qualified and knowledgeable about the processes.
- Only authorized personnel with the proper training and certification should do many repair procedures.

**WARNING: Some mechanical and electrical service procedures can be extremely dangerous or life-threatening.
Know your skill level and abilities.**

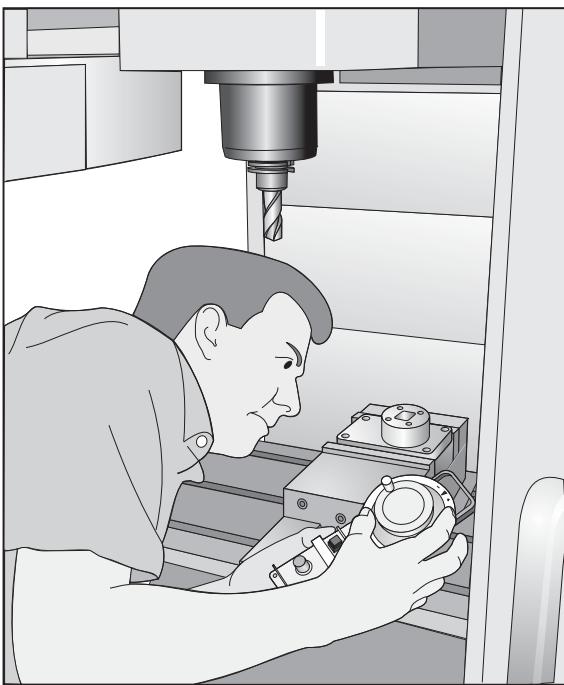
All information herein is provided as a courtesy for Haas machine owners for reference and illustrative purposes only. Haas Automation cannot be held responsible for repairs you perform. Only those services and repairs that are provided by authorized Haas Factory Outlet distributors are guaranteed.

Only an authorized Haas Factory Outlet distributor should service or repair a Haas machine that is protected by the original factory warranty. Servicing by any other party automatically voids the factory warranty.



HAAS SAFETY PROCEDURES

THINK SAFETY!



DON'T GET CAUGHT UP IN YOUR WORK

All milling machines contain hazards from rotating parts, belts and pulleys, high voltage electricity, noise, and compressed air. When using CNC machines and their components, basic safety precautions must always be followed to reduce the risk of personal injury and mechanical damage.

Important – This machine to be operated only by trained personnel in accordance with the Operator's manual and instructions for safe operation of machine.

Safety Contents

USES AND GUIDELINES FOR PROPER MACHINE OPERATION	4
MODIFICATIONS TO THE MACHINE	4
SAFETY PLACARDS	4
AUTOMATIC EQUIPMENT	4
MACHINE GUARDING	4
ROTATING PARTS	4
CHIP AUGERS	5
DANGEROUS VOLTAGES	5
JOB SET-UP	5
HOT AREAS	5
MACHINE AWARENESS	5
PROPER ATTIRE	5
TOOLING	6
EYE-SAFETY	6
ELECTRICAL SERVICE	6
SPILLS	6
FIXTURES	6
KEEP CLEAR	6
DECLARATION OF WARNINGS, CAUTIONS, AND NOTES	7
SAFETY WINDOWS	7



READ BEFORE OPERATING THIS MACHINE:

- ◆ Only authorized personnel should work on this machine. Untrained personnel present a hazard to themselves and the machine, and improper operation will void the warranty.
- ◆ Check for damaged parts and tools before operating the machine. Any part or tool that is damaged should be properly repaired or replaced by authorized personnel. Do not operate the machine if any component does not appear to be functioning correctly. Contact your shop supervisor.
- ◆ Use appropriate eye and ear protection while operating the machine. ANSI-approved impact safety goggles and OSHA-approved ear protection are recommended to reduce the risks of sight damage and hearing loss.
- ◆ Do not operate the machine unless the doors are closed and the door interlocks are functioning properly. Rotating cutting tools can cause severe injury. When a program is running, the mill table and spindle head can move rapidly at any time in any direction.
- ◆ The Emergency Stop button (also known as an Emergency Power Off button) is the large, circular red switch located on the Control Panel. Pressing the Emergency Stop button will instantly stop all motion of the machine, the servo motors, the tool changer, and the coolant pump. Use the Emergency Stop button only in emergencies to avoid crashing the machine.
- ◆ The electrical panel should be closed and the three latches on the control cabinet should be secured at all times except during installation and service. At those times, only qualified electricians should have access to the panel. When the main circuit breaker is on, there is high voltage throughout the electrical panel (including the circuit boards and logic circuits) and some components operate at high temperatures. Therefore, extreme caution is required. Once the machine is installed, the control cabinet must be locked and the key available only to qualified service personnel.
- ◆ DO NOT modify or alter this equipment in any way. If modifications are necessary, all such requests must be handled by Haas Automation, Inc. Any modification or alteration of any Haas Milling or Turning Center could lead to personal injury and/or mechanical damage and will void your warranty.
- ◆ It is the shop owner's responsibility to make sure that everyone who is involved in installing and operating the machine is thoroughly acquainted with the installation, operation, and safety instructions provided with the machine BEFORE they perform any actual work. The ultimate responsibility for safety rests with the shop owner and the individuals who work with the machine.



OBSERVE ALL OF THE WARNINGS AND CAUTIONS BELOW:

- ◆ This machine is automatically controlled and may start at any time.
- ◆ This machine can cause severe bodily injury.
- ◆ Do not operate with the doors open.
- ◆ Avoid entering the machine enclosure.
- ◆ Do not operate without proper training.
- ◆ Always wear safety goggles.
- ◆ Never place your hand on the tool in the spindle and press ATC FWD, ATC REV, NEXT TOOL, or cause a tool change cycle. The tool changer will move in and crush your hand.
- ◆ To avoid tool changer damage, ensure that tools are properly aligned with the spindle drive lugs when loading tools.
- ◆ The electrical power must meet the specifications in this manual. Attempting to run the machine from any other source can cause severe damage and will void the warranty.
- ◆ DO NOT press POWER UP/RESTART on the control panel until after the installation is complete.
- ◆ DO NOT attempt to operate the machine before all of the installation instructions have been completed.
- ◆ NEVER service the machine with the power connected.
- ◆ Improperly clamped parts machined at high speeds/feeds may be ejected and puncture the safety door. Machining oversized or marginally clamped parts is not safe.
- ◆ Windows must be replaced if damaged or severely scratched - Replace damaged windows immediately.
- ◆ Do not process toxic or flammable material. Deadly fumes can be present. Consult material manufacturer for safe handling of material by-products before processing.



USES AND GUIDELINES FOR PROPER MACHINE OPERATION

All milling machines contain hazards from rotating cutting tools, belts and pulleys, high voltage electricity, noise, and compressed air. When using milling machines and their components, basic safety precautions should always be followed to reduce the risk of personal injury and mechanical damage. **READ ALL APPROPRIATE WARNINGS, CAUTIONS, AND INSTRUCTIONS BEFORE OPERATING THIS MACHINE.**

MODIFICATIONS TO THE MACHINE

DO NOT modify or alter this equipment in any way. If modifications are necessary, all such requests must be handled by Haas Automation, Inc. Any modification or alteration of any Haas machining center could lead to personal injury and/or mechanical damage and will void your warranty.

SAFETY PLACARDS

Some or all of these warnings may be on your machine. Be aware of the possible dangers.

If a sticker is missing from the machine, or an extra one is needed to remind employees of the need to work safely, contact the Haas factory



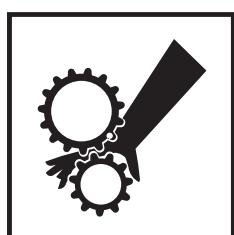
Automatic Equipment

As this machine equipment is programmable it may start unexpectedly. Operators and other shop personnel need to be aware of this. Keep the doors closed at all times other than loading or unloading parts and tools.



Machine Guarding

Some guarding and access panels may close unexpectedly due to machine operation. Ensure all guarding and panels are closed and if possible, secured. Personal injury will result if the guarding falls or the access panels swing closed.



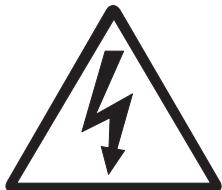
Rotating Parts

If it is necessary to work on the machine, ensure the power is disconnected before servicing the machine. Rotating parts within the machine can cause serious or fatal injuries.



Chip Augers

The chip augers can be programmed to start and stop automatically. Keep all body parts out of these as they may rotate unexpectedly. Keep out of the discharge tubes as well. The collected chips will cut you.



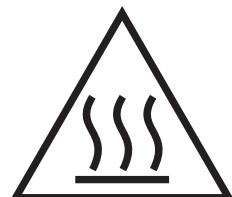
Dangerous Voltages

Potential lethal voltages and currents are present within the system. Service should be done only by qualified personnel.



Job Set-up

Incorrect tooling, machining practices, or improperly clamped workpieces or fixtures will create devastating results. Double check your set-up before beginning any machining operations.



Hot Areas

This warning is on areas of the machine that have a lot of heat associated with it. Serious burns will result if personnel come in contact with these areas.



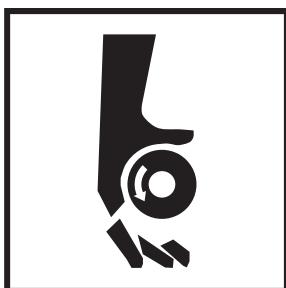
Machine Awareness

Most areas of the machine are not designed to support the weight of the operator.



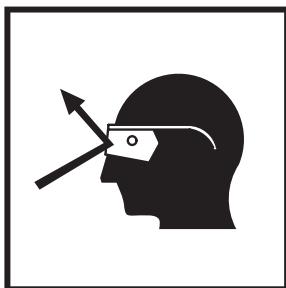
Proper Attire

Watch for loose clothing and other personal belongings. These may become entangled in the machine and cause personal injury.



Tooling

Do not slow, or try to stop the rotating tools. These are extremely sharp and will cut or cause extensive bodily harm.



Eye-Safety

Always wear safety glasses or a face mask during machine operation. Flying hot chips will cause injury.



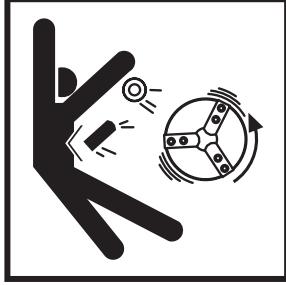
Electrical Service

Electrical shock is possible with any machine. Only authorized certified personnel should service the machine. Do not open the control cabinet or any other electrical guarding.



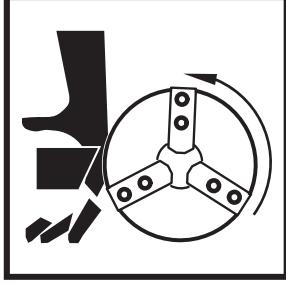
Spills

Clean up spills immediately. Liquids, such as machine coolant will be slippery and will cause a hazardous work environment.



Fixtures

Improperly clamped workpieces can be thrown from the fixture with deadly force.



Keep Clear

Do not put anything between the tool and the workpiece. Severe personal injury can occur.



WARNING

- THE SAFETY WINDOWS MAY NOT STOP EVERY TYPE OF PROJECTILE.
- SAFETY WINDOWS MAY BECOME BRITTLE AND LOSE EFFECTIVENESS WHEN EXPOSED TO MACHINE COOLANTS AND OIL OVER TIME.
- IF SIGNS OF DISCOLORATION, CRAZING OR CRACKING ARE FOUND REPLACE IMMEDIATELY.
- SAFETY WINDOWS SHOULD BE REPLACED EVERY TWO YEARS.

Safety Windows

Improperly clamped work pieces may be thrown through the windows; serious personal injury may occur. Use care and proper machining techniques during all operations. Replace windows immediately if they are damaged or show signs of wear. Contact the Haas Service Department for Replacement parts.

DECLARATION OF WARNINGS, CAUTIONS, AND NOTES

Throughout this manual, important and critical information is contained in the following styles/formats; Warnings, Cautions and Notes:

Warnings are used when there is an extreme danger to the operator and/or to the machine. Take all steps necessary to heed the warning given. Do not continue if you cannot follow the warning instructions. An example warning is:

WARNING!

NEVER PUT HANDS BETWEEN TOOL CHANGER AND SPINDLE HEAD.

Cautions are used when there is the potential for minor personal injury or mechanical damage. An example of a caution is:

CAUTION! Power down the machine before performing any maintenance tasks.

Notes are used to give additional information to the operator about a particular step or procedure. This information should be taken into consideration by the operator as he/she performs the step or procedure to ensure there is no confusion. An example of a note is:

NOTE: If machine is equipped with the optional extended Z-clearance table, follow these guidelines:

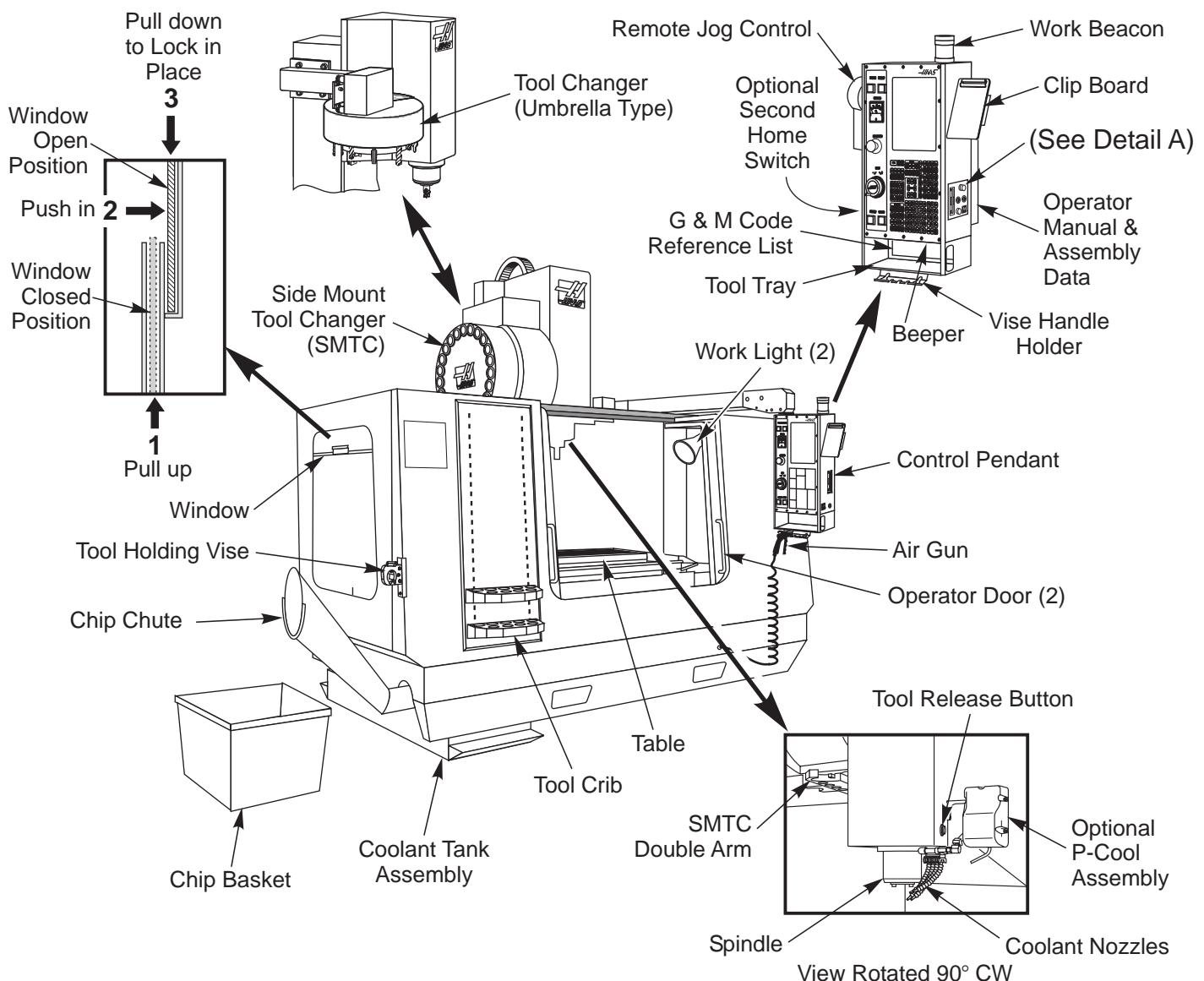
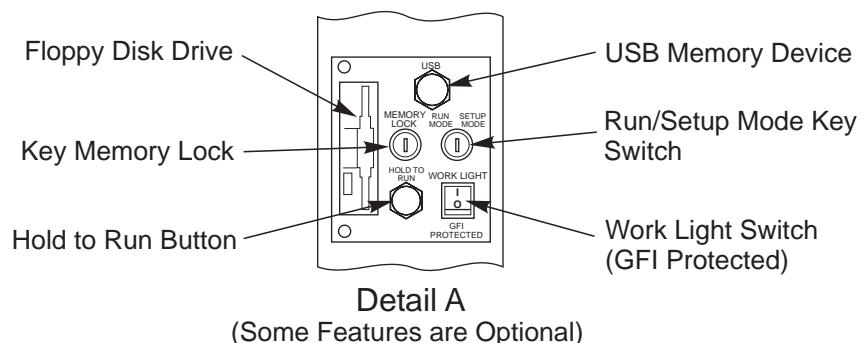
- Avoid extreme loads on the center of table or far end of table. The weight of the part should be distributed evenly over the table or one pad.
- Flatness of part should be within 0.002".

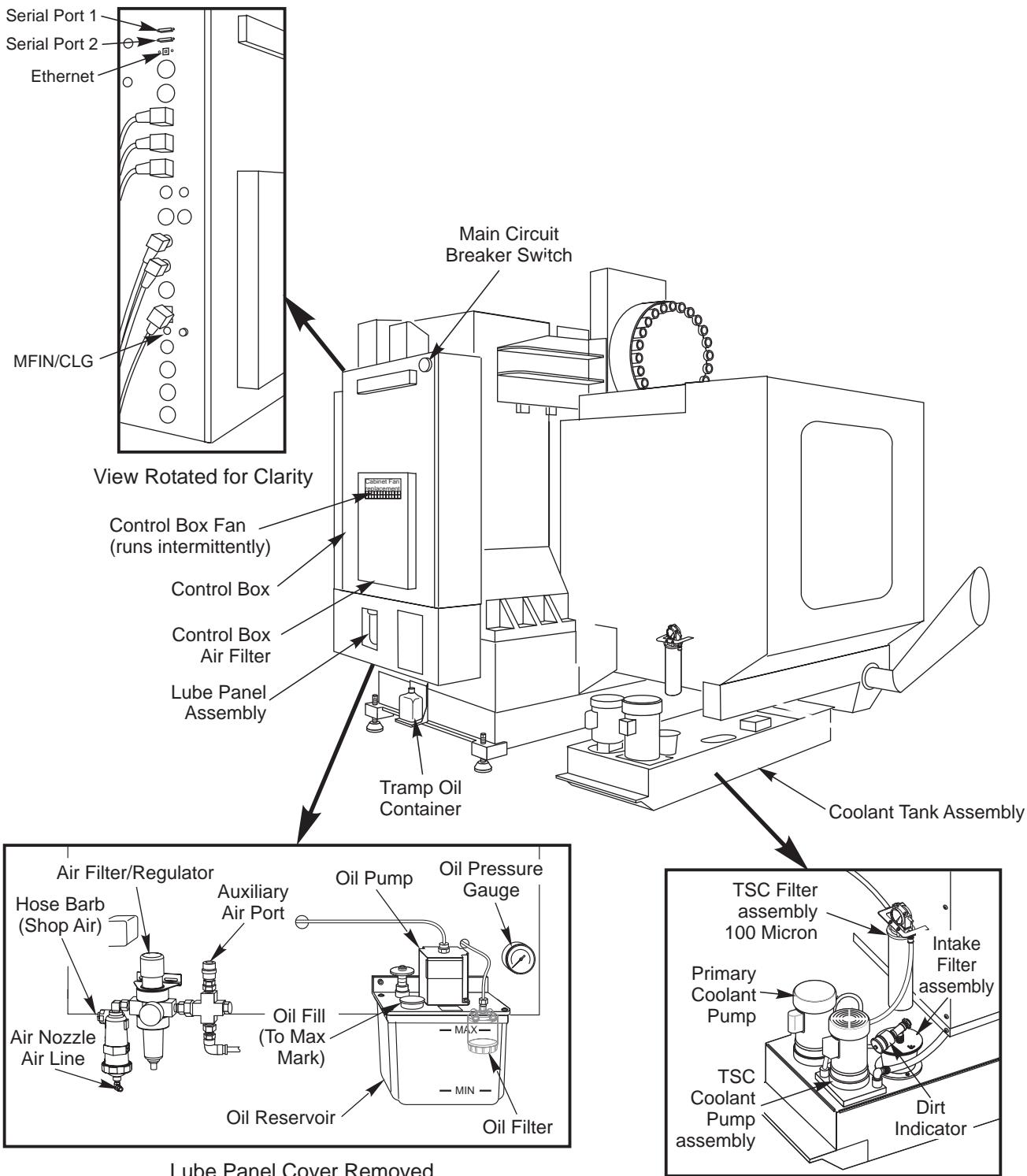




OPERATION

The following is a visual introduction to a HAAS mill. Some of the features shown will be highlighted in their appropriate sections.

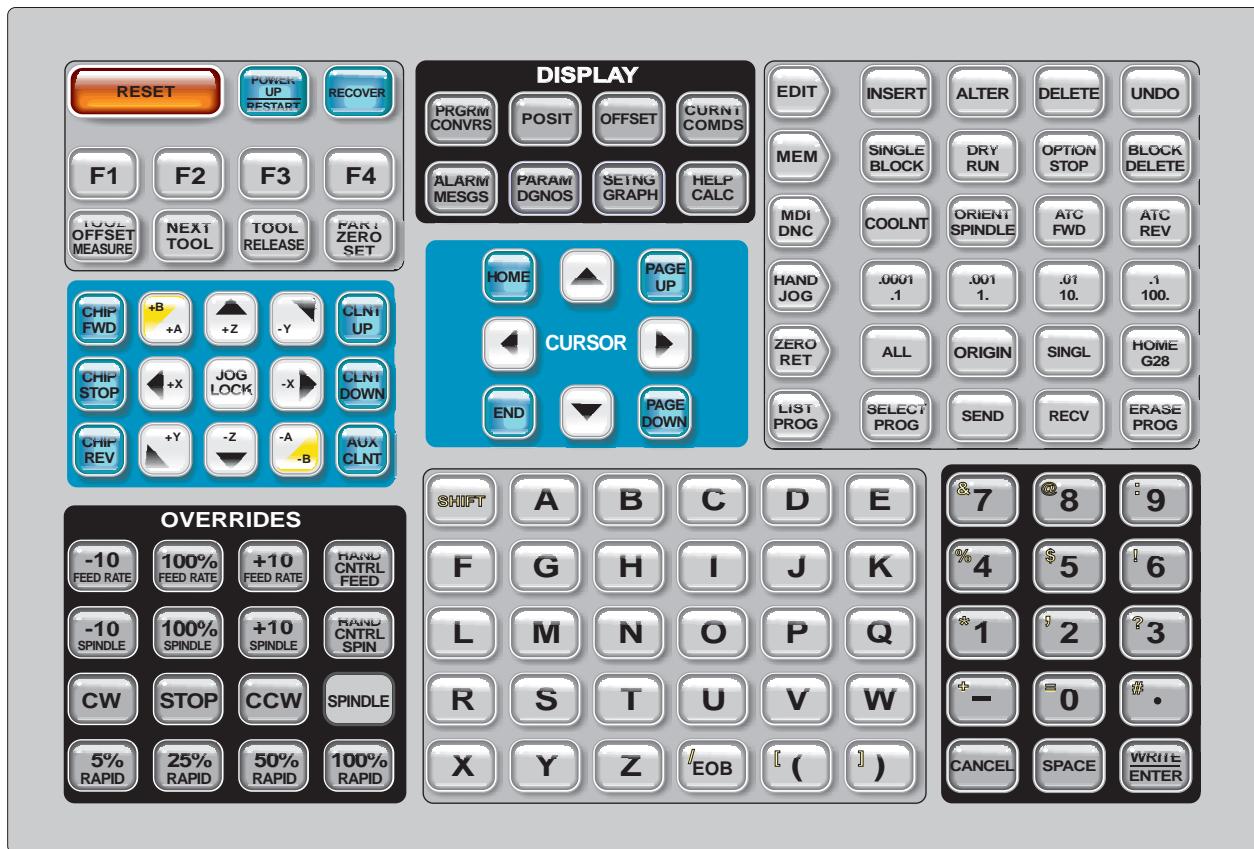






PENDANT KEYBOARD INTRODUCTION

The keyboard is broken up into eight sections: Function Keys, Jog Keys, Override Keys, Display Keys, Cursor Keys, Alpha Keys, Mode Keys and Number Keys. In addition there are miscellaneous keys and features located on the pendant and keyboard which are described briefly.



Power On- Turns the machine on.

Power Off- Turns the machine off.

Spindle Load Meter - Displays the spindle load, in percent.

Emergency Stop - This stops all axes motion, stops the spindle and tool changer, and turns off the coolant pump.

Jog Handle - This is used to jog all axes. Can also be used to scroll through program code or menu items while editing.

Cycle Start - Starts a program. This button is also used to start a program in Graphics mode.

Feed Hold - Will stop all axis motion. Note: Spindle will continue to turn during cutting.

Reset - Will stop the machine (axes, spindle, coolant pump, and tool changer are stopped). This is **not** a recommended method to stop the machine, as it may be difficult to continue from that point.

Power Up / Restart - When this key is pressed, the axes will return to the machine zero position and a tool change may occur. See Setting 81 in the Settings chapter for more information.

Restore - This button aids the operator in recovering the tool changer from an abnormal stop. See the tool changer section for more information.



Memory Lock Key Switch - This switch prevent the operator from editing programs and from altering settings when turned to the locked position. The following describes the hierarchy of locks:

- Key switch locks Settings and all programs.
- Setting 7 locks parameters.
- Setting 8 locks all programs.
- Setting 23 locks 9xxx programs.
- Setting 119 locks offsets.
- Setting 120 locks macro variables.

Second Home Button - This button will rapid all axes to the coordinates specified in work Offset G129. The sequence is as follows: First, the Z axis is returned to machine zero, then the X and Y axes are moved, then the Z axis is moved to its second home position. This feature will work in any mode except DNC.

Work Light Switch - This switch will turn on the work light inside of the machine.

Keyboard Beeper - Located at the top of the parts tray. The volume can be adjusted by turning the cover.

FUNCTION KEYS

F1- F4 Keys - These buttons have different functions depending upon which mode of operation you are in. For example, F1-F4 will cause a different action in Editing mode, than Program mode, than Offset mode. See the specific mode section for further descriptions and examples.

Tool Offset Meas (Tool Offset Measure) - Used to record tool length offsets during part setup.

Next Tool - Used to select the next tool from the tool changer (usually used during part setup).

Tool Release - Releases the tool from the spindle when in MDI mode, zero return mode, or handle jog mode.

Part Zero Set - Used to automatically set work coordinate offsets during part setup (see Setting Offsets in the Operation section).

JOG KEYS

Chip FWD (Chip Auger Forward) - Starts optional chip auger in the "Forward" direction, moving chips out of the machine.

Chip Stop (Chip Auger Stop) - Stops auger movement.

Chip REV (Chip Auger Reverse) - Starts the optional chip auger in the "Reverse" direction, which is useful in clearing jams and debris from auger.

X/-X, Y/-Y, Z/-Z, A/-A and B/-B (axis keys)- Allows the operator to manually jog axis by holding down the individual button or pressing the desired axes and using the jog handle.

Jog Lock - Works with the axes buttons. Press jog lock and then an axis button and the axis will move to maximum travel or until jog lock is pressed again.

CLNT Up (Coolant Up) - Moves the optional Programmable Coolant (P-Cool) nozzle up.

CLNT Down (Coolant Down) - Moves the optional P-Cool nozzle down.

AUX CLNT (Auxiliary Coolant) - Pressing this key while in MDI mode will turn on the optional Through the Spindle Coolant (TSC) system; pressing it a second time will turn off TSC.



OVERRIDE KEYS

These keys give the user the ability to override the speed of non-cutting (rapid) axes motion, programmed feeds and spindle speeds.

-10 - Decreases current feedrate by 10%.

100% - Sets overridden feedrate to programmed feedrate.

+10 - Increases current feedrate by 10%.

-10 - Decreases current spindle speed by 10%.

100% - Sets overridden spindle speed to programmed speed.

+10 - Increases current spindle speed by 10%.

Hand Cntrl Feed (Handle Control Feedrate) - Pressing this button allows the jog handle to be used to control the feedrate in $\pm 1\%$ increments.

Hand Cntrl Spin (Handle Control Spindle) - Pressing this button allows the jog handle to be used to control spindle speed in $\pm 1\%$ increments.

CW - Starts the spindle in the clockwise direction. This button is disabled on CE (export) machines.

CCW - Starts the spindle in the counterclockwise direction. This button is disabled on CE (export) machines.

The spindle can be started or stopped with the CW or CCW buttons any time the machine is at a Single Block stop or the Feed Hold button has been pressed. When the program is restarted with Cycle Start, the spindle will be turned back on to the previously defined speed.

STOP - Stops the spindle.

5% / 25% / 50% / 100% Rapid - Limits machine rapids to the value on the key. The 100% Rapid button allows maximum rapid.

Override Usage

The feedrate can be varied from 0% to 999% of the programmed value while in operation. This is done with the feedrate +10%, -10% and 100% buttons. The feedrate override is ineffective during G74 and G84 tapping cycles. Feedrate override does not change the speed of any auxiliary axes. During manual jogging, the feedrate override will adjust the rates selected from the keypad. This allows for fine control of the jog speed.

The spindle speed can also be varied, from 0% to 999%, using the spindle overrides. It is also ineffective for G74 and G84. In the Single Block mode, the spindle may be stopped. It will automatically start up upon continuing the program (pressing Cycle Start).

By pressing the Handle Control Feedrate key, the jog handle can be used to control feedrate from 0% to 999% in $\pm 1\%$ increments. By pressing the Handle Control Spindle key, the jog handle can be used to control spindle speed in $\pm 1\%$ increments (from 0% to 999%).

Rapid moves (G00) may be limited to 5%, 25%, or 50% of maximum using the keypad. If the 100% rapid is too fast, it may be set to 50% of maximum by Setting 10.

In the Settings page, it is possible to disable the override keys so that the operator cannot select them. These are Settings 19, 20 and 21.



The Feed Hold button acts as an override button as it sets the rapid and feed rates to zero when it is pressed. The Cycle Start button must be pressed to proceed after a Feed Hold. The door switch on the enclosure also has a similar result but will display "Door Hold" when the door is opened. When the door is closed, the control will be in Feed Hold and Cycle Start must be pressed to continue. Door Hold and Feed Hold do not stop any auxiliary axes.

The operator can override the coolant setting by pressing the COOLNT button. The pump will remain either on or off until the next M-code or operator action (see Setting 32).

Overrides can be reset to defaults with an M30 and/or pressing Reset (See Setting 83).

DISPLAY KEYS

Displays keys provide access to the machine displays, operational information and help pages. Some of these keys will display additional screens when pressed more than once.

Prgrm/Convs - Displays the currently selected program. In Edit mode, pressing the button twice will enter the Quick Code (See Quick Code section) feature and pressing it three times will enter the Visual Quick Code feature (See Visual Quick Code section in Quick Code).

Posit (Position) - Displays the position of the machine axes. Pressing the Page Up/Down buttons scroll through operator, machine, work, and distance-to-go formats and displays them in larger formats.

Offset - Displays the tool length geometry, radius offsets, wear offsets, and coolant position. Pressing Offset button twice or pressing the Page Up button will access the work offsets page.

Curnt Comds (Current Commands) - Displays the current program details (for example G, M, H and T codes), Spindle load information and machine axes positions while the program runs. Press Page Up/ Down to view tool load/vibration (See the tool load/vibration section), tool life (See the tool life section), maintenance, macro variables, program timers and program code details.

Alarm / Mesgs (Alarms / Messages) - Displays the alarm viewer and message screens. There are three alarm screens, the first shows the currently active alarms (first press of the Alarm/Mesgs button). Pressing the Right Arrow button switches to the Alarm History screen, which keeps a history of recent alarms.

Pressing Right Arrow again switches to the alarm viewer screen. This screen shows one alarm at a time with its description. The default will be the last alarm in the alarm history. The user can then scroll through the alarms by pressing the Up and Down Arrow buttons. Also, the user can enter an alarm number and press Enter/Write and the name and description will be displayed.

Pressing Alarm/mesgs a second time will display a page for user messages and notes. Use the keypad to enter messages for other operators/programmer or write notes for a current project. If there is a message, every time the machine is powered on the messages page will display. Messages are displayed at power up until they are erased. See Message section for more details.

Param / Dgnos (Parameters / Diagnostics) - Displays parameters that define the machine's operation. To find a known parameter, type in the number and press the up or down arrow. Parameters are set at the factory and should not be modified by the user.

A second press of the Param / Dgnos key will display the first page of diagnostic data. This information is mainly used for troubleshooting by a certified Haas service technician. The first page of diagnostic data is discrete inputs and outputs. Pressing Page Down will display the additional pages of diagnostic data.

Setng / Graph (Settings / Graphics) - Displays and allows changing of user settings. (Note that the settings are grouped; these groups are for a specific subject). To find a known setting, type in the number and press the up or down arrow.



Pressing the Setng / Graph key a second time enables Graphics mode. In Graphics mode the user can view the generated tool path of the program and, if necessary, debug the program before running it (See Graphics Mode in the Operation section)

Help / Calc (Help / Calculator) - Displays an abbreviated manual. In this on-screen manual there are brief descriptions of G and M codes, definitions of control features, troubleshooting and maintenance issues.

Pressing Help/ Calc a second time will display the help calculator. Press the Page Down button to scroll through the calculator pages (see the calculator section).

CURSOR KEYS

Cursor Keys give the user the ability to move to various screens and fields in the control and are used in the editing of CNC programs.

Home - This button will move the cursor to the top-most item on the screen; in editing, this is the top left block of the program.

Up / Down Arrows - 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.

Left Arrow - Used to select individually editable items when viewing a program; moves cursor to the left. It is used to scroll through setting selections.

Right Arrow - Used to select individually editable items when viewing a program; moves cursor to the right. It is used to scroll through setting selections and moves the zoom window right when in graphics mode.

End - This button generally moves the cursor to the bottom-most item on the screen. In editing, this is the last block of the program.

ALPHA KEYS

The alpha keys allow the user to enter the letters of the alphabet along with some special characters. Some of the special characters are entered in by first pressing the "Shift" key.

Shift - The shift key provides access to additional characters on the keyboard. The additional characters are seen in the upper left of some of the alpha and number keys. Pressing Shift and then the character will enter that character on the data entry line. When entering text, UPPER CASE is the default, to enter lower case characters, press and hold the Shift key.

When a control has a fifth axis installed, the B axis is selected for jogging by pressing the Shift button and then the +/-A jog keys.

EOB - This is the End-Of-Block character. It is displayed as a semicolon (;) on the screen and it signifies the end of a program line.

() - Parentheses are used to separate CNC program commands from user comments. They must always be entered as a pair. Note: Any time an invalid line of code is received through the RS-232 port while receiving a program, it is added to the program between parenthesis.

/ - The right slash is used in the Block Delete feature and in Macro expressions. If this symbol is the first symbol in a block and a Block Delete is enabled, then that block is ignored at run time. The symbol is also used for division (divide by) in macro expressions (see the Macro section).

[] - Square brackets are used in macro functions. Macros are an optional software feature (see the Macro section).



MODE KEYS

Mode keys change the operational state of the CNC machine tool. Once a mode button is pressed, the buttons in the same row are made available to the user. The current mode is always displayed on the top line just to the right of the current display in between parenthesis.

Edit - Selects edit mode. This mode is used to edit programs in control's memory.

Insert - Pressing this button will enter commands into the program in front of the cursor. This button will also insert the text from the clipboard to the current cursor location, and is also used to copy blocks of code in a program (See Advanced Editor Section)

Alter - Pressing this button will change the highlighted command or text to the newly entered commands or text. This button will also change the highlighted variables to the text stored in the clipboard, or move a selected block to another location.

Delete - Deletes the item that the cursor is on, or deletes a selected program block.

Undo - Undoes up to the last 9 edit changes, and deselects a highlighted block.

MEM (Memory) - Selects the memory mode. This page displays the current program that is selected in the list of programs.

Single Block - Turns single block on or off. When single block is on, only one block of the program is executed, for every press of Cycle Start.

Dry Run - This is used to check actual machine movement without cutting a part. (See the Dry Run section in the Operation Chapter)

Opt Stop (Optional Stop) - Turns on and off optional stops. Also see G103 in the G-Code chapter.

When this feature is ON and an M01 (optional stop) code is programmed, the machine will stop when it reaches the M01. The machine will continue once Cycle Start is pressed. However, depending on the look-ahead function (G103), it may not stop immediately (See block look ahead section). In other words, the block look-ahead feature may cause the Optional Stop command to ignore the nearest M01.

If the Optional Stop button is pressed during a program it will take effect on the line after the highlighted line when the Opt Stop button is pressed.

Block Delete - Turns On/Off block delete function. Blocks with a slash ("/") as the first item are ignored (not executed) when this option is enabled. If a slash is within a line of code, the commands after the slash will be ignored if this feature is enabled. Block Delete will take effect two lines after Block Delete is pressed, except when cutter compensation is used, in this case, block delete will not take effect until at least four lines after the highlighted line. Processing will slow down for paths containing block deletes during high-speed machining. Block Delete will stay active when power is cycled.

MDI/DNC - MDI mode is the "Manual Data Entry" mode where a program can be written but it is not entered into memory. DNC mode "Direct Numeric Control", allows large programs to be "drip fed" into the control so it can be executed (See DNC mode section)

CoolInt (Coolant) - Turns the optional coolant on and off.

Orient Spindle - Rotates the spindle to a given position and then locks the spindle. Can be used during setup to indicate parts.

ATC FWD - Rotates the tool turret to the next tool. To load a specific tool into the spindle, enter MDI mode, type a tool number (T8) and press ATC FWD.

ATC REV - Rotates the tool turret to the previous tool. To load a specific tool into the spindle, enter MDI mode, type a tool number (T10) and press ATC REV.



Handle Jog - Selects axis jogging mode .0001, .1 - 0.0001 inches (metric 0.001mm) for each division on the jog handle. For dry run, .1 inches/min.

.0001/.1, .001/1., .01/10., .1/100. - The first number (top number), when in inch mode, selects that amount to be jogged for each click of the jog handle. When the mill is in MM mode the first number is multiplied by ten when jogging the axis (e.g. .0001 becomes 0.001mm). The second number (bottom number) is used for dry run mode and is used to select the speed feedrate and axis motions.

Zero Ret (Zero Return) - Selects Zero Return mode, which displays axis location in four different categories, they are; Operator, Work G54, Machine and Dist (distance) to go. You can page up or down to view each category in a larger format by itself.

All - Returns all axes to machine zero. This is similar to Power Up/Restart except a tool change will not occur. This can be used to establish the initial axes zero position.

Origin - Sets selected displays and timers to zero.

Singl (Single) - Returns one axis to machine zero. Press the desired axis letter and then press the Singl Axis button. This can be used to establish the initial axis zero position.

HOME G28 - Returns all axes to machine zero in rapid motion. Home G28 will also home a single axis in the same manner if you enter an axis letter and press the home G28 button. **CAUTION!** There is no warning message to alert the operator of any possible collision. For example, if the Z-axis is down in amongst parts on the table when X or Y is zeroed, a crash can result.

List Prog (List Programs) - Displays the programs stored in the control.

Select Prog - Makes the highlighted program, in the program list, the current program. Note: The current program will have an “*” preceding it in the program list.

Send - Transmits programs out the RS-232 serial port. (See RS-232 section)

Recv - Receives programs from the RS-232 serial port. (See RS-232 section)

Erase Prog - Erases the highlighted program in Memory mode or the entire program when in MDI mode.

NUMERIC KEYS

The numeric keys give the user the ability to enter numbers and a few special characters into the control.

Cancel - The Cancel key is used to delete the last character entered.

Space - Used to format comments placed into programs or in the message area.

Write / Enter - General purpose enter key.

- (Minus sign)- Used to enter negative numbers.

. (Decimal Point)- Used for decimal precision.



POSITION DISPLAYS

The following are the position displays.

Home Page

This display shows the four displays (Operator, Work, Machine and Distance-to-go) simultaneously. Use the Page Up/Down keys or the UP/Down Arrows to scroll through these pages.

Operator Display

This display is used to show the distance the operator has jogged any of the axes. This does not represent the actual distance the axis is from machine zero. Axes can be zeroed by selecting an axis (X, Y or Z) and pressing the Origin key.

Work Display

This displays the position of X, Y and Z in relation to the part; not machine zero. On power-up, it will display the value in work offset G54 automatically. The position can only be changed by entering values in work offsets G54 through G59, G110 through G129, or by commanding a G92 in a program.

Machine Display

This display is the actual axes position away from machine zero.

Distance To Go

This display shows the distance remaining before the axes reaches its commanded position.

OFFSETS DISPLAY

There are two offsets pages. The first offset page is the Tool Geometry/Wear page. The second is the Work Zero offset page.

Tool Geometry/Wear

The Tool/Geometry page is displayed by pressing the Offset button. This page displays tool numbers, tool length geometry and wear values, tool radius geometry and wear values, flutes values and actual diameter values. If the mill has the optional programmable coolant unit, this page will also display the coolant position entered for each tool. To enter values into these fields key in a number and press F1. Keying in a number and pressing F2 will set the negative of the entered value into the offsets. Entering a value and pressing Write/Enter will add the value to what is currently entered. To clear all the values on the page press Origin, the mill will prompt the operator with "Zero All (Y/N) press Y to zero all or press N to leave all the values unchanged.

Work Zero Offset

The Work Zero Offset page is displayed by pressing the Offset button twice. This page displays the values entered so that each tool knows where the part is located on the table. A value can be set for each axis (X, Y and Z). The operator can scroll to each column by using the arrow key or by pressing the Page Up or Down buttons to access the other offsets in the Work Zero section. In order for each tool to locate the part, the tools used in a program must be "Touched off" the part (See the Operations section) . A value can also be entered by typing in a number and pressing F1, or the value can be added to an existing value by pressing Enter/Write. Keying in a number and pressing F2 will set the negative of the entered value into the offsets. To clear all the values on the page press Origin, the mill will prompt the operator with "Zero All (Y/N) press Y to zero all or press N to leave all the values unchanged.



CURRENT COMMANDS DISPLAY

The following are several Current Command pages in the control. Press the Current Commands button and use the Page Up/Down buttons to navigate through the pages.

Program Command Check Display This display shows a current overview of the important commands. It shows the programmed spindle speed (PGM), the commanded spindle speed commanded by the program (CMD), and the actual spindle speed (ACT). In addition, this display shows the CW, CCW, or stopped command being sent to the spindle and the current transmission gear (if equipped) position.

This display also shows the position of the axes. There are four coordinates displayed (operator, work, machine, or distance to go) (See Positions Display section above for description) and are selected using the cursor Up/Down or Page Up/Down keys.

If the machine has the optional Programmable Coolant (P-Cool), the current spigot position will be displayed as well.

Current Display Command This display shows the current program codes and their current value. These values may not be changed in this display.

Macro Variables Display This display shows a list of the macro variables and their present values. As the control runs the program, the variables will be updated. In addition the variables may be modified in this display; refer to the "Macros" section for more information.

Operation Timers Display This display shows the current power-on time, cycle start time (the amount of total time the machine has been running a program), and the feed time (total amount of time the machine has been feeding). These times may be reset to zero by using the cursor up and down keys to highlight the desired title and pressing the ORIGIN button.

Listed below these times are two M30 counters, these counters are used for counting completed parts. They may be set to zero independently to provide for the number of parts per shift and total parts.

Tool Life display This display shows the time the tool is used in a feed (Feed-Time), the time the tool is in the spindle (Total-Time), and the number of times the tool has been selected (Usage). This information is used to assist in predicting tool life. The values in this display can be reset to zero by highlighting the value and pressing the Origin button. Note: The maximum value is 32767, once this value is reached, the control will start back at zero.

This display may also be used to generate an alarm when a tool has been used a specific number of times. The last column is labeled "Alarm," entering a number in that column will cause the machine to generate an alarm (#362 Tool Usage Alarm) when that count is reached.

Tool Load Monitor and Display The operator can enter the maximum amount of tool load, in %, that is expected for each tool. The operator can select the appropriate action to be taken when this load is exceeded. This display provides for the entry of this alarm point and also displays the largest load that tool has seen in a previous feed.

NOTE: Surface Feet per Minute (SFM) and Chip Load are displayed on the Current Commands page. SFM is displayed as fpm (feet per minute) or mpm (meters per minute).

The tool load monitor function operates whenever the machine is in a feed operation (G01, G02, or G03). If the limit is exceeded the action specified in Setting 84 will occur (See settings section for a description).

Axis Load Monitor Axis load is 100% to represent the maximum continuous load. Up to 250% can be shown, however an axis load above 100%, for an extended period of time, can lead to an overload alarm.

Maintenance This page allows the operator to activate and deactivate a series of checks (see Maintenance section).



ALARMS / MESSAGES DISPLAY

Alarms

The Alarms display is selected by pressing the ALARM / MESGS button. There are three types of Alarms screens. The first shows any current alarms. Pressing the Right Arrow key switches to the Alarm History screen, which shows the previously received alarms. Pressing Right Arrow again switches to the alarm viewer screen. This screen shows one alarm at a time with its description. The user can then scroll through all the alarms by pressing the Up and Down Arrow keys. Also, the user can enter an alarm number and press Write/Enter and the name and description will be displayed. Pressing Page Down will display a page for user messages and notes.

Note: The Cursor and Page Up and Page Down buttons can be used to move through a large number of alarms.

Messages

The Message Display can be selected by pressing the Alarm/Mesgs button twice. This is an operator message display and has no other effect on operation of the control. Use the keypad to enter the messages. The cancel and space keys can be used to remove existing messages and the Delete button can be used to remove an entire line. Data is automatically stored and maintained even in a power-off state. The message display page will come up during power-up if there are no alarms present.

SETTING / GRAPHIC DISPLAY FUNCTION

The Settings are selected by pressing the Setng/Graph button. There are some special functions in the settings which change the way the mill behaves; refer to the "Settings" section for a more detailed description.

The Graphics function is selected by pressing the Setng/Graph button twice. Graphics is a visual dry run of your part program without the need to move the axes and risk tool or part damage from programming errors. This function may be considered more useful than the Dry Run mode, because all of your work offsets, tool offsets, and travel limits can be checked before running the machine. The risk of a crash during setup is greatly reduced.

Graphics Mode Operation

To run a program in Graphics, a program must be loaded and the control must be in either MEM or MDI mode. Press the Setng/Graph key twice to select the Graphics Mode.

The Graphics display has a number of features within it.

Key Help Area The right side of the top line is the function key help area. Function keys that are currently available are displayed here with a brief description of their usage.

Locator Window The lower right part of the screen has two modes. It can display the whole table area and indicate where the tool is currently located during simulation. Or it can be used to display four lines of the program that is being executed. The F4 key is used to toggle between these two modes.

Tool Path Window In the center of the display is a large window that represents a top view of the X and Y axes. It displays tool paths during a graphics simulation of the program. Rapid moves are displayed as dotted lines, while feed motion is displayed as fine continuous lines. (Note: Setting 4 can disable the rapid path.) The places where a drilling canned cycle is used are marked with an X. Note: Setting 5 can disable the drill mark.

Scaling the Tool Path Window The tool path window can be scaled. After running a program, pressing F2 will scale the tool path. Use the Page Down key and the arrow keys to select the portion of the tool path to be magnified. Pressing F2 will display a rectangle (zoom window) indicating the magnified area. Note: The Help area will flash, indicating the view rescaling process. The locator window (small view at the bottom right) shows the entire table with an outline of where the Tool Path window is zoomed. The Page Up key reduces the zoom (unzooms) the rectangle one step. After sizing and/or moving the zoom window, pressing the Write/Enter key will complete the zoom process and re-scale the Tool Path window. After the Tool Path window is re-scaled, the Tool Path window is cleared and the program must be re-run to see the tool path.



The scale and position of the Tool Path window is saved in Settings 65 through 68. Leaving graphics to edit the program and then returning to Graphics will keep the previous scaling in effect.

Pressing F2 and then the Home key will expand the Tool Path window to cover the entire table.

Z Axis Part Zero Line This feature consists of a horizontal line displayed on the Z-axis bar at the top-right corner of the graphics screen to indicate the position of the current Z-axis work offset plus the length of the current tool. While a program is running, the shaded portion of the bar indicates the depth of Z-axis motion. The user will be able to watch the position of the tool tip relative to the Z-axis part zero position as the program runs.

Control Status The lower left portion of the screen displays control status. It is the same as the last four lines of all other displays.

Position Window The location of all enabled axes can be viewed in this window. Press F3 to open this window. Additional presses of the F3 key or the up and down arrows will display the various position formats. This window also displays the current scale of the tool path window and the current simulated tool number.

DATE AND TIME

The control contains a clock and date function. To view the time and date, press the Help button until the date and time appears.

Setting the date and time, use the Up and Down arrow keys to select the date or time. Use the Right and Left arrow keys or the jog handle to make adjustments.

SCREEN SAVER

The Haas machine includes a screen saver for the operator's pendant. After a period of time, with no keyboard activity the screen saver will start. When the screen saver is active, the words "Screen saver" will be displayed and will change positions every 2 seconds. The screen saver is cancelled by any key press, jog handle motion or machine alarm. The screen saver will not activate if the control is in Sleep, Jog, Edit, Graphics, MEM or MDI mode with the PRGRM screen displayed.

HELP / CALCULATOR FUNCTION

Help

The Help Feature is selected by pressing the Help button, which will display a mini-manual.

Use the alphanumeric keys to select the topics, or use the Page Up/Down button or Up/Down arrows to navigate through the help topics and descriptions.

Calculator

The Calculator function is selected by pressing the Help key twice. Use the Page Up / Down keys to move through the pages.

All of the Calculator functions will do simple add, subtract, multiply, and divide operations. When one of the functions is selected, a calculator window appears with the possible operations (LOAD, +, -, *, and /). LOAD is initially highlighted, and the other options can be selected with the left and right cursor arrows. Numbers are entered by typing them in and pressing the Write/Enter key. When a number is entered and LOAD is selected, that number will be entered into the calculator window directly. When a number is entered when one of the other functions (+ - * /) is selected, that calculation will be performed with the number just entered and any number that was already in the calculator window. The calculator will also accept a mathematical expression such as $23^*4-5.2+6/2$. It will evaluate it (doing multiplication and division first) and place the result, 89.8 in this case, in the window.



Function Keys: The function keys can be used to copy and paste the calculated results into a section of a program or into another area of the Calculator feature.

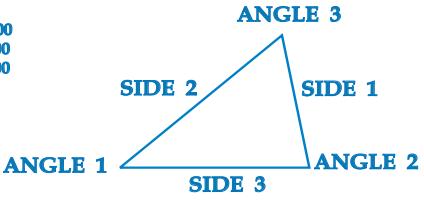
F3 In EDIT and MDI modes the F3 key will copy the highlighted triangle/circular milling/tapping value into the data entry line at the bottom of the screen. This is useful when the calculated solution will be used in a program.

In the Calculator function, pressing F3 copies the value in the calculator window to the highlighted data entry for Trig, Circular or Milling/Tapping calculations.

F4 In the Calculator function, this button uses the highlighted Trig, Circular or Milling/Tapping data value to load, add, subtract, multiply, or divide with the calculator.

Trigonometry Help Function

The Trigonometry calculator page will help solve a triangular problem. Enter the lengths and the angles of a triangle and when enough data has been entered, the control will solve for the triangle and display the rest of the values. Use the Cursor Up/Down buttons to select the value to be entered with WRITE. For inputs that have more than one solution, entering the last data value a second time will cause the next possible solution to be displayed.

HELP (MEM)		O00000 N00000000	
CALCULATOR			
0.000000000			
LOAD + - × /			
(MACHINE)	ANGLE 1	40.000	
X	ANGLE 2	72.000	
Y	ANGLE 3	68.000	
Z	SIDE 1	10.0000	
	SIDE 2	14.7958	
	SIDE 3	14.4244	
ANGLE 1 SIDE 2 SIDE 1 SIDE 3 ANGLE 2 ANGLE 3			
F3 copies calculator value to highlighted field in this or other calculator screens. F3 also copies calculator value to the data entry line of edit screens. F4 copies highlighted data to the calculator field.			

Circular Interpolation Help

The Circular calculator page will help solve a circle problem. You enter the center, radius, angles, start and end points; when enough data has been entered, the control will solve for the circular motion and display the rest of the values. Use the Cursor Up/Down buttons to select the value to be entered with Write. In addition, it will list alternate formats that such a move could be programmed with a G02 or G03. The formats can be selected using the Cursor Up/Down buttons, and the F3 button will import the highlighted line into a program you are editing.

HELP (MEM)		O00000 N00000000	
CALCULATOR			
0.000000000			
LOAD + - × /			
(MACHINE)	CENTER X	13.0000	
X	CENTER Y	20.0000	
Y	START X	4.0000	
Z	START Y	10.0000	
	END X	7.0000	
	END Y	32.0416	
	RADIUS	13.4536	
	ANGLE	111.527	
	DIRECTION	CW	
16 19. J10. 16 R13.4536 16 19. J10 G91 G2 X3. Y22.0416 R13.4536			



For inputs that have more than one solution, entering the last data value a second time will cause the next possible solution to be displayed. To change the CW value to the CCW value, highlight the CW/CCW column and press the Write/Enter button .

Circle-Line Tangent Calculator

This feature provides the ability to determine points of intersection where a circle and a line meet as tangent. Enter two points, A and B, on a line and a third point, C, away from that line, the control will calculate the point of intersection. The point is where a normal line from point C will intersect with the line AB, as well as, the perpendicular distance to that line.

CIRCLE- CIRCLE TANGENT	
CIRCLE1 X	5.0000
CIRCLE1 Y	6.0000
RADIUS 1	4.0000
CIRCLE2 X	0.0000
CIRCLE2 Y	0.0000
RADIUS 2	2.0000
TANGT A X	1.3738
	Y 7.6885
TANGT B X	7.3147
	Y 2.7378
TANGT C X	-1.8131
	Y 0.8442
TANGT D X	1.1573
	Y -1.6311

Type: STRAIGHT
Use F and T to form G-code.
F1 for alternate solution

CIRCLE- LINE TANGENT	
POINT A X	5.0000
	Y 3.0000
POINT B X	1.0000
	Y 4.0000
POINT C X	0.0000
	Y 0.0000
RADIUS	4.1231
TANGT PT X	1.0000
TANGT PT Y	4.0000

Circle-Circle Tangent Calculator

This feature provides the ability to determine points of intersection between two circles or points. The user provides the location of two circles and their radii. The control then calculates all the intersection points that are formed by lines tangent to both circles. Note: For every input condition where there are two disjointed circles, there are up to eight intersection points. Four points are obtained from drawing straight tangents and four points by forming cross tangents. The F1 key is used to toggle between the two diagrams. When "F" is pressed, the control will prompt for the from and to points (A, B, C, etc.) that specify a segment of the diagram. If the segment is an arc, the control will also prompt for C or W (CW or CCW). Sample G code is then displayed at the bottom of the screen. When "T" is entered, the previous to point becomes the new from point and the control prompts for a new to point. To enter the solution (line of code), switch to MDI or Edit and press F3, as the G-code is already on the input line.

Milling/Tapping Help

The Milling/Tapping Help page will help you solve equations relating to milling and tapping. They are:

1. SFM (Surface Feet per Minute) = (CUTTER DIAMETER IN.) * RPM * 3.14159 / 12
2. (FEED IN./MIN.) = RPM / (THREAD PITCH)
3. (CHIP LOAD IN.) = (FEED IN./MIN.) / RPM / #FLUTES

Enter all but one of the values and the control will compute the remaining value. Note that the RPM value for the first two equations are the same entry.



HELP (MEM)	O00000 N00000000
CALCULATOR	
0.00000000	MILLING
LOAD + - * /	CUTTER DIA 0.5000 IN
(MACHINE)	SURFACE SPEED 5.0000 FT/MIN RECOMMENDED
X 0.000 in	RPM 38
Y 0.000 in	FLUTES 4
Z 3.5179 in	FEED 6.0000 IN/MIN
	CHIP LOAD 0.0365 IN RECOMMENDED
	MATERIAL LOW CARBON UNALLOYED STEEL
	TOOL TYPE CARBIDE
	CUT WIDTH 2.0000 IN
	CUT DEPTH 0.0500 IN REQUIRED POWER
PGM F0.	TAPPING
ACT F0.	THREADS 3.0000 /IN
PGM S0	RPM 40
CMD S0	FEED 13.3333 IN/MIN
ACT S0	

Note: When metric units are selected, the units displayed change to millimeters, threads per mm, mm per minute, and meters, respectively.

Materials

The Milling calculator includes a field called MATERIAL, which, when highlighted, allows the operator to select a type of material from the list using the left and right arrow keys.

A recommended surface speed and chip load will be displayed based on the material chosen, as shown.

SURFACE SPEED *.*.* FT/MIN RECOMMENDED *.*.* TO *.*.*
CHIP LOAD *.*.* IN RECOMMENDED *.*.* TO *.*.*

Also, the required horsepower will be calculated and displayed as shown below on the right.

CUT DEPTH *.*.* IN REQUIRED POWER *.* HP

When in metric mode, the required power is displayed as KW.

Drill/Tap Chart

Pressing the Help/Calc button a third time displays a Drill and Tap chart

SPINDLE WARM-UP PROGRAM

All spindles, which have been idle for more than 4 days, must be thermally cycled prior to operation. This warm-up will prevent possible overheating of the spindle due to the settling of lubrication. A 20-minute warm-up program (number O02020) has been supplied with the machine which will bring the spindle up to speed slowly, allowing the spindle to thermally stabilize. This program may also be used daily for spindle warm-up prior to high-speed use.

RUN-STOP-JOG-CONTINUE

This feature allows the operator to stop a running program, jog away from the part, and then resume program execution. The following is an operation procedure:

1. Press Feed Hold to stop the running program
2. Press X, Y or Z followed by the Handle Jog button. The control will store the current X, Y, and Z positions.
Note: Axes other than X, Y, and Z cannot be jogged.



3. The control will display the message "Jog Away". Use the jog handle, remote jog handle, jog and jog-lock buttons to move the tool away from part. Control buttons such as AUX CLNT (optional Through Spindle Coolant - TSC), or Coolnt to turn on/off the coolant (AUX CLNT requires that the spindle is rotating and the door is closed). The spindle can be controlled by pressing CW, CCW, Stop, Tool Release. Tools can be swapped out and the associated length and diameter offsets adjusted, however, when the program is continued, the old offsets will be used for the return position. Therefore, it is unsafe and not recommended to change tools and offsets when the program is interrupted.
4. Jog to a position as close as possible to the stored position, or to a position where there will be an unobstructed rapid path back to the stored position.
5. Return to the previous mode by pressing MEM, MDI, or DNC. The control will only continue if the mode that was in effect when stopped is re-entered.
6. Press Cycle Start. The control will display the message Jog Return and rapid X and Y at 5% to the position where Feed Hold was pressed, then return the Z-axis. **Caution:** The control will not follow the path used to jog away. If Feed Hold is pressed during this motion, the mill will enter a feed hold and display the message "Jog Return Hold". Pressing Cycle Start will cause the control to resume the Jog Return motion. When the motion is completed, the control will again go into a feed hold state.
7. Press Cycle Start again and the program will resume normal operation.

COOLANT LEVEL GAUGE

The coolant level is displayed on the control on the Current Commands page. A vertical bar on the left of the screen shows the status of the coolant. The display will flash once the coolant reaches a point that could cause intermittently coolant flow.

OPTIONS

200 Hour Control Option Try-Out

Options that normally require a unlock code to activate (Rigid Tap, Macros, etc.) can now be activated and deactivated as desired simply by entering the number "1" instead of the unlock code to turn it on. Enter a "0" to turn off the option. An option activated in this manner will be automatically deactivated after a total of 200 power-on hours. Note that the deactivation only occurs when power to the machine is turned off, not while it is running. An option can be activated permanently by entering the unlock code. Note that the letter "T" will be displayed to the right of the option on the parameter screen during the 200 hour period. Note that the safety circuit option is an exception; it can be turned on and off only by unlock codes.

To enter a 1 or 0 into the option you must have setting 7 (Parameter Lock) turned off and the Emergency Stop button pressed in.

When the option reaches 100 hours the machine will give an alarm warning that the try out time is almost up.

To permanently activate an option, contact your dealer.

Rigid Tapping

Synchronized tapping eliminates the need for expensive, floating tap holders, and prevents lead-thread distortion and start-thread pullout.

Auto Door

The auto door option opens the machine doors automatically via the part program. This reduces operator fatigue, or allows for unattended operation when used with a robotic loader.



Hard Disk Drive, USB and Ethernet

Store and transfer data between your Haas machine(s) and a network. Program files are easily transferred to and from memory, and allows DNC of large files at up to 800 blocks per second.

Auto Air Gun

Provides a constant air blast to the cutting tool to clear chips safely and automatically during dry machining, or blow chips and coolant out of deep pockets. Activated by M code, the air blast can be programmed to turn on while the spindle is turning, or at the end of a cycle (air consumption 6 scfm).

High Intensity Lighting

Halogen lights provide bright, even illumination of the work area for part inspection, job setup and changeovers — ideal for jobs like mold making. Lights turn on and off automatically when doors open and close, or can be activated manually via a switch on the lights.

Macros

Create subroutines for custom canned cycles, probing routines, operator prompting, math equations or functions, and family-of-parts machining with variables.

Rotation and Scaling

Use rotation in conjunction with work offset probing to speed workpiece setup, or to rotate a pattern to another location or around a circumference, etc. Use scaling to reduce or enlarge a toolpath or pattern.

Part Probing

Renishaw - This tool probe option incorporates an easy-to-use Renishaw tool probe with macros. This optional package enables all Haas VMCs to automatically define tool offsets; and when combined with macro programming, it allows operators to automatically update tool offsets (length and diameter) and check for broken tools while in process.

Marposs - Marposs ML75 laser/non-contact tool check system and T25 high precision, contact, work setting and inspection probe for Haas 40 taper VMCs. ML75 measures tools up to 6.88" (175mm) diameter. Customer must provide own tool holder to accommodate 1/2" cal bar.

M-Function Relays

Adds extra relays to increase productivity. These additional M code outputs may be used for activating probes, auxiliary pumps, part loaders, etc.

Remote jog Handle

The versatile Haas Remote Jog Handle can be used to move all axes, as well as edit programs, override spindle speeds and feedrates, single-block scroll through programs — and much more.

Programmable Coolant Spigot

The optional programmable coolant spigot allows the user to direct the coolant stream to the most optimum location in order to flush out chips from the cutting area. The direction of the coolant can be changed by the CNC program.

Automatic Chip Auger

The automatic chip auger assists the user in removal of chips for jobs with heavy material removal.

Through the Spindle Coolant (TSC)

The Haas Through The Spindle Coolant (TSC) option uses an auxiliary coolant pump to supply high-pressure coolant to the cutting tool.

Extra Rapid Traverse

With the XRT option, high-pitch ballscrews combined with powerful brushless servomotors provide faster rapids and cutting feeds. Reducing non-cutting time during machining, especially on repetitive parts and multiple fixtures, means faster cycle times and lower cost per part.



Linear Scales

When absolute positioning accuracy is necessary, linear scales are the only way to ensure exact axis positioning under the fluctuating conditions that exist in today's machine shops. Linear scales ensure positioning accuracy along X, Y and Z axes by compensating for the thermal variations inherent in all machine tools. These scales deliver positioning accuracy of 2 microns to guarantee the type of precision that mold making and other high-tolerance machining applications demand.

Memory Lock Key Switch

Locks memory to prevent accidental or unapproved program editing by unauthorized personnel. It can also be used to lock settings, parameters, offsets and macro variables.

Spindle Orientation

The Spindle Orientation option allows spindle positioning to a specific, programmed angle, using the standard spindle motor and the standard spindle encoder for feedback. This option provides inexpensive, accurate (0.1 degree) positioning.

Second Home Button

This option allows the operator to quickly send the axes to a second, user-defined Home position in addition to the machine's standard Home position.

Programmable Coolant

P-Cool is a multi-position programmable nozzle that automatically directs coolant precisely at the part. The position of the nozzle is controlled via the program, saving operator time by eliminating constant adjustments

Remote Jog Handle

The versatile Haas Remote Jog Handle can be used to move all axes, as well as edit programs, override spindle speeds and feedrates, single-block scroll through programs etc.

High Speed Machining

High speed machining makes it possible for an increase in the removal rate of material, improve surface finish, and reduce cutting forces which will reduce machining costs and extend the life of the tools.

High Speed Machining is most often required for the machining of smoothly sculpted shapes as is typical of mold making. The Haas High Speed Machining option increases the amount of lookahead to 80 blocks and allows full speed (500 inches per minute) blending of feed strokes.

It is important to understand that high speed machining works best with smoothly blended shapes where the feed rate can remain high through the blend of one stroke to the next. If there are sharp corners, the control will always need to slow down or corner rounding will occur.

The affect that blending of strokes can have on feed rate is always to slow down motion. The programmed feed rate (F) is thus a maximum and the control will sometimes go slower than that in order to achieve the required accuracy.

Too short of a stroke length can result in too many data points. Check how the CAD/CAM system generates data points to insure that it does not exceed 1000 blocks per second.

Too few data points can result in either "facetting" or blending angles which are so great that the control must slow down the feed rate. Facetting is where the desired smooth path is actually made up of short, flat, strokes that are not close enough to the desired smoothness of the path.



High Speed Tooling

The tool holders should be an AT-3 or better with a nylon back-up screw. The tolerances maintained in the AT-3 design are the minimum that would be recommended for a high speed process. The nylon back-up screw increases collet grip on the tool and creates a better seal to aid in coolant transfer.

Use single angle collet chucks and collets for best grip and concentricity. These collet systems are made up of a long single angle located in the holder. The angle per side should be eight degrees or less for best results. Avoid double angle collet systems when maximum rigidity and close tolerance are dictated. It is recommended that minimum engagement of 2/3 of the full length of the bore in the double split single angle collet. However for better results 3/4 to full engagement is preferred if possible.

Auxiliary Filter

This 25-micron #2 bag-type filter system removes contamination and minute particles from the coolant before they can be recirculated through the coolant pump. The filter is mandatory for machines equipped with through-spindle coolant when machining cast iron, cast aluminum and other abrasive materials, and may be used on non-TSC machines as well.



PROGRAMMING

MACHINE POWER-UP

Turn the machine on by pressing the Power-On button on the pendant.

The machine will go through a self test and then display either the Messages screen, if a message was left, or the Alarms screen. In either case the mill will have one alarm present (102 SERVOS OFF). Pressing the Reset button a couple times will clear the alarms. If an alarm cannot be cleared the machine may need servicing, if this is the case call your dealer.

Once the alarms are cleared the machine needs a reference point to start all operations from; this point is called "Home". To home the machine, press the Power-Up Restart button. Caution: Automatic motion will begin once this button is pressed. Keep away from the inside of the machine and the tool changer. Note that pressing the Power-Up button will automatically clear alarm 102 if it was present.

After home is found the Current Commands page is displayed, and the machine is now ready to run.

PROGRAMMING INTRODUCTION

Manual Data Input (MDI)

Manual Data Input (MDI) is a means to command automatic CNC moves without using a formal program.

Press the MDI button to enter this mode. Programming code is entered by typing in the commands and pressing Enter at the end of each line. Note that an End of Block (EOB) will be automatically inserted at the end of each line.

PROGRAM (MDI)	N00000000
G97 S1000 ; G00 X2. Z0.1 ; G92 X1.8 Z-1. F0.05 ; X1.78 ; X1.76 ; X1.75 ↴	

To edit the MDI program use the keys to the right of the Edit button. Cursor to the point that is changing, then the different edit functions can be used.

To enter an additional command to the line, enter the command and press Enter.

To change a value use the arrow buttons or the jog handle to highlight the command, enter the new command and press Alter.

To delete a command, highlight the command and press Delete.

The Undo key will reverse changes (up to 9 times) that have been made to the MDI program.

An MDI program can be saved to the memory of the control. To do so cursor to the beginning of the program (or press Home), enter a program name (programs need to be named using the format **O**nnnnn; the letter "O" followed by up to 5 numbers) and press Alter. This will add the program to the list of programs and clear MDI. To re-access the program, press List Prog and select it.

The data in MDI is retained after exiting MDI mode and when the machine is turned off.

To clear the current MDI commands press the Erase Prog button.



Numbered Programs

To create a new program, press LIST PROG to enter the program display and the list of programs mode. Enter a program number (**Onnnnn**) and press Select Prog key or Enter. If the program exists, it will be selected. If it does not yet exist, it will be created. Press Edit to show the new program. A new program will consist of only the program name and an End of Block (;).

Numbered programs are retained when the machine is turned off.

Basic Editing of MDI and Numbered Programs

The only difference between an MDI program and a numbered program is the O code. To edit an MDI program, simply press MDI. To edit a numbered program, select it, then press Edit.

Type in the program data and press enter. Program data falls into three categories, addresses, comments or EOBs.

```
PROGRAM (EDIT)          O00741          F1 KEY TURNS MENU ON/OFF
PROGRAM   EDIT   SEARCH   MODIFY   I/O   HELP
O00741
G00 X0 Z0.1 ;
G74 Z-0.345 F0.03 K0.1 ;
;
G00 X2. Z0.1 ;
G74 X1. Z-4. I0.2 K0.75 D255 ;
G00 X3. Z0.1 ;
```

To add program code to the existing program, highlight the code that the additional code will go in front of, type in the data and press the Insert key. More than one code, such as **X**, **Y**, and **Z**, can be entered before pressing Insert.

Address data is a letter followed by a numeric value. For example: G04 P1.0. The G04 commands a dwell (pause) and the P1.0 is the length (1 second) of the dwell.

Comments can be either alpha or numeric characters, but must be prefaced with parentheses. For example: (1 second dwell). Comments can be a maximum of 80 characters.

Lower-case text is to be entered between parentheses (comments). To type lower case text, press the Shift key first (or hold it in) and then the letter or letters.

End of Blocks are entered by pressing the EOB button and are displayed as a semicolon (;). These are used like a carriage return at the end of a paragraph. In CNC programming an EOB is entered at the end of a string of program code.

An example of a line of code using the three types of commands would be:

G04 P1. (1 second dwell);

There is no need to put any symbols or spaces between the commands. Usually a space is put between elements for ease of reading (Editing).

To alter characters, highlight a portion of the program using the arrow keys or the jog handle, enter the replacement code, and press Alter.

To remove characters, highlight the characters and press Delete.

To remove commands, highlight the command and press Delete.

Use the Undo button to reverse any changes. The Undo button will work for the last nine entries.

There is no save command, as the program is saved as each line is entered.



Converting an MDI program to a numbered program

An MDI program can be converted to a numbered program. To do so cursor to the beginning of the program (or press Home), enter a program name (programs need to be named using the format **O**nnnnn; the letter "O" followed by up to 5 numbers) and press Alter. This will add the program to the list of programs and clear MDI. To re-access the program, press List Prog and select it.

Searching the program

While in MDI, EDIT or MEM mode, the cursor up and down keys can be used to search the program for specific codes or text. To search for particular character(s), enter the character(s) on the data entry line (i.e. G40) and press the cursor up or down keys. The cursor up key will search for the entered item backwards (towards the start of the program) and the cursor down key will search forward (towards the end of the program).

Deleting Programs

To delete a program, press LIST PROG. Use the cursor up or down keys to highlight the program number (or type in the program number) and press the ERASE PROG key.

Highlighting ALL at the end of the list and pressing the ERASE PROG key will delete all the programs in the list. There are some important programs that you will receive with your machine; they are O02020 (spindle warm up) and O09997, O09999 (Visual Quick Code). You will want to save these programs to a floppy disk or to your PC before erasing all programs.

Note: The UNDO key will not recover programs that are deleted.

Renaming Programs

After creating a program, the program number can be renamed by changing the name (Onnnnn), in Edit mode, on the first line and pressing the Alter key. Another method of changing the program name is to enter the list of programs (press List Prog) cursor to the program, type in the letter "O" followed by a five-digit number, such as O12345, and press the Alter key.

Maximum Number of Programs

If the maximum number of programs (500) is in the control's memory, the message "DIR FULL" will display and the program cannot be created.

Program Selection

Enter the program directory by pressing "List Prog"; this will display the stored programs. Scroll to the desired program and press "Select Prog" to select the program. Entering the program name and pressing "Select Prog" will also select a program.

Once "Select Prog" is pressed, an asterisk ("*") appears next to the program name. This is the program that will be run when the mode is changed to MEM and CYCLE START is pressed. It is also the one that you will see on the EDIT display.

When in MEM mode, another program can be selected and displayed quickly by entering the program number (Onnnnn) and pressing the Up/Down arrow, or **F4**.

The selected program will remain selected after the machine is turned off.

Loading Programs to the CNC Control

Numbered programs can be copied from the CNC control to a personal computer (PC) and back again. It is best if the programs are saved to a file that ends in ".txt". That way, they will be recognized by any PC as a simple text file. Programs can be transferred by many different methods such as RS-232 and floppy disk. Settings, offsets and macro variables can be transferred between the CNC and a PC in a similar manner.

Corrupted program data, if received by the CNC, is converted into a comment, stored in the program, and an alarm is generated. However, the data will still be loaded into the control.

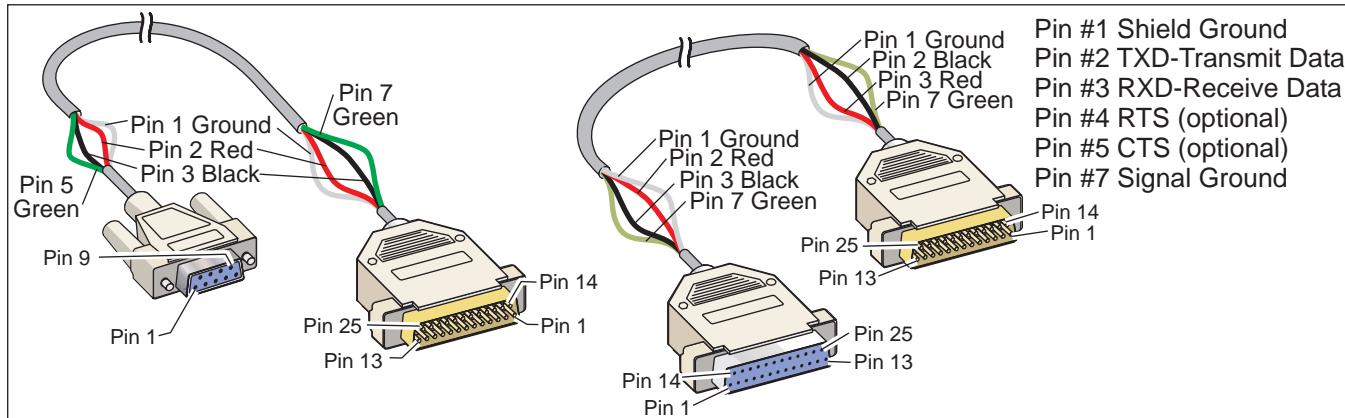


RS-232

RS-232 is one way of connecting the Haas CNC control to another computer. This feature enables the programmer to up and download programs, settings and tool offsets from a PC.

Programs are sent or received through the RS-232 port (Serial Port 1) located on side of the control box (Not the operators pendant).

A cable (not included) is necessary to link the CNC control with the PC. There are two styles of RS-232 connections: the 25-pin connector and the 9-pin connector. The 9-pin connector is more commonly used on PCs.



WARNING!

One of the biggest causes of electronic damage is a lack of a good ground on both the CNC mill and the computer. The lack of grounding will damage the CNC, or the computer, or both.

Cable length

The following lists baud rate and the respective maximum cable length.

9,600 baud rate: 100 feet (30 m) RS-232

38,400 baud rate: 25 feet (8 m) RS-232

115,200 baud rate: 6 feet (2 m) RS-232

The settings between the CNC control and the other computer must match. To change the setting in the CNC control, enter the Settings page (press Setng/Graph) and scroll to the RS-232 settings (or enter “11” and press the up or down arrow). Use the up/down arrows to highlight the settings and the left and right arrows to change the values. Press Enter when the proper selection is highlighted.

The settings (and defaults) that control the RS-232 port are:

- | | |
|-----------------------------|---------------------------|
| 11 Baud Rate (9600) | 24 Leader to Punch (None) |
| 12 Parity (Even) | 25 EOB Pattern (CR LF) |
| 13 Stop Bits (1) | 37 Number Data Bits (7) |
| 14 Synchronization Xon/Xoff | |

There are a number of different programs that can link with the Haas control. An example is the Hyper Terminal program that is installed with most Microsoft Windows applications. To change the settings on this program go to the “File” drop down menu at the top left. Choose the “Properties” selection from the menu and then press the “Configure” button. This will open up the port settings; change these to match what is in the CNC control.

To receive a program from the PC, push the LIST PROG key. Move the cursor to the word ALL and push the RECV RS-232 key and the control will receive all main and sub programs until it receives a "%" indicating end of input. All programs sent to the control from the PC must begin with a line containing a single "%" and must end with a line containing a single "%". Note that when using "ALL", your programs must have a Haas formatted program number (Onnnnn). If you do not have a program number, type in a program number before you push RECV RS-232 and the program will be stored under that number. You can also select an existing program for input and it will be replaced.



To send a program to the PC, use the cursor to select the program and push the SEND RS-232 key. You can select “ALL” to send all of the programs in the memory of the control. A setting (Setting 41) can be turned on to add spaces to the RS-232 output and improve the readability of your programs.

Parameters, settings, offsets, and macro variables pages may also be sent individually via RS-232 by selecting the “LIST PROG” mode, selecting the desired display screen, and pushing the SEND key. They can be received by pushing the RECV key and selecting the file on the PC that is to be received.

The file can be viewed on a PC by adding “.txt” to the file name from CNC control. Then open the file on a PC using a program such as Windows Notepad.

If an abort message is received, check the set-up between the mill and the PC and the cable.

Optional Floppy Drive

All files must be on MS-DOS formatted 1.44M floppy disks and must reside in the root directory. This means that the desired file cannot be in a folder on the disk.

All programs must begin with a line starting with a single “%” and end with a line ending in a single “%”. Note: Files saved by the control will have these lines.

To load a program from a floppy disk, press the LIST PROG button, enter the floppy disk file name and press F3 button. The control will receive all main and sub programs.

Please note that when using the “ALL” command to receive every program on the disk, all the programs must have a Haas-convention file name (**Onnnnn**).

To save a program to a floppy disk, press the PRGRM button and then the LIST PROG button. Enter a file name and press the F2 key. You can select “ALL” to send all of the programs in memory. Note: Setting 23 (Program Edit Lock) must be turned off.

To erase a file from the floppy drive, press the List Prog button, type “DEL <file name>” where “<file name>” is the name of the file on the floppy disk. Do not use the program number, unless it is also the file name. Press Write/Enter.

Parameters, Settings, Macro Variables, and Offsets are sent to the floppy disk by pressing LIST PROG, selecting the desired display screen (PARAM, SETNG, OFSET, or the Macro Variables page of CRNT CMDS), entering a file name and pressing the F2 key. They can be received by pressing the F3 key.

Note: It is not recommended that parameters be loaded into the machine. If parameters need to be loaded contact the Haas service department.

Disk Directory

To get a program directory listing, select the PRGM/LIST PROG mode, and press F4. This will generate a disk directory listing that will be saved in a program (the default program is O08999). Press Edit to view the contents of the disk.

File Delete

On the List Prog page, type “DEL <filename>” where <filename> is the name of a file on the floppy disk. Press WRITE. The message “DISK DELETE” will appear, and the file will be deleted from the floppy disk.

Direct Numeric Control (DNC)

Direct Numeric Control (DNC) is another method of loading a program into the control.

Direct Numeric Control (DNC) is the ability to run a program as it is received through the RS-232 port. This feature differs from a program loaded through the RS-232 port in that there is no limit to the size of the CNC program. The program is run by the control as it is sent to the control; the program is not stored in the control.



PROGRAM (DNC)	N00000000
WAITING FOR DNC . . .	
DNC RS232	O01000 ; (G-CODE FINAL QC TEST CUT) ; (MATERIAL IS 2x8x8 6061 ALUMINUM) ; ; (MAIN) ; ; M00 ; (READ DIRECTIONS FOR PARAMETERS AND SETTINGS) ; (FOR VF-SERIES MACHINES W/4TH AXIS CARDS) ; (USE / FOR HS, VR, VB, AND NON-FORTH MACHINES) ; (CONNECT CABLE FOR HA5C BEFORE STARTING THE PROGRAM) ; (SETTINGS TO CHANGE) ; (SETTING 31 SET TO OFF) ; ; ;

DNC Waiting for program

Program received from DNC

DNC is enabled using Parameter 57 bit 18 and Setting 55. Turn the parameter bit on (1) and change Setting 55 to On. It is recommended that DNC be run with Xmodem or parity selected because an error in transmission will then be detected and will stop the DNC program without crashing. The settings between the CNC control and the other computer must match. To change the setting in the CNC control, enter the Settings page (press Setng/Graph) and scroll to the RS-232 settings (or enter 11 and press the up or down arrow). Use the up/down arrows to highlight the variables and the left and right arrows to change the values. Press Enter when the proper selection is highlighted.

The recommended RS-232 settings for DNC are:

Settings: 11 Baud Rate Select: 19200
12 Parity Select: NONE
13 Stop Bits:1
14 Synchronization: XMODEM
37 RS-232 Date Bits: 8

DNC is selected by pressing MDI twice (The DNC page "Program DNC") at the top of the page. Note: DNC needs a minimum of 8k bytes of user memory available. This can be done by going to the List Programs page and checking the amount of free memory on the bottom of the page.

The program sent to the control must begin and end with a %. The data rate selected (Setting 11) for the RS-232 port must be fast enough to keep up with the rate of block execution of your program. If the data rate is too slow, the tool may stop in a cut.

Start sending the program to the control before the Cycle Start button is pushed. Once the message "DNC Prog Found" is displayed, press Cycle Start.

Floppy Disk DNC

Floppy disk DNC is selected by entering the floppy disk file name and pressing MDI a second time when (must be in MDI mode). Note: Pressing MDI a third time will stop the DNC and the message "DISK ABORT" will be received.

DNC Notes:

While a program is running in DNC, you cannot change modes. Therefore, editing features such as Background Edit is not available.

DNC supports Drip Mode. The control will perform one block (command) at a time. Each block will be performed immediately with no block look-ahead. The exception is when Cutter Compensation is commanded. Cutter Compensation requires three blocks of motion commands to be read prior to a compensated block being performed.

Full duplex communication during DNC is possible by using the G102 command or DPRNT to output axes coordinates back to the controlling computer.



ALPHABETICAL ADDRESS CODES

The following is a list of the address codes used in programming the CNC.

A, B, C, U, V, W, X, Y, Z Axis motion – Specifies axis motion (distance or angle).

D Tool diameter selection – Selects the tool diameter or radius used for cutter compensation. See the Cutter Compensation section.

E Contouring accuracy – Used, with G187, to select the accuracy required when cutting a corner during high-speed machining operations. See the section on Contouring Accuracy.

F Feed rate – Used to enter a feed rate; the value is entered in inches per minute or mm per minute.

G, M Preparatory functions – See the chapters on G or M codes.

H Tool length offset selection – Selects the tool length offset. The H is followed by a number between 0 and 200.

I, J, K Canned cycle and circular optional data – These address characters are used to specify data for some canned cycles and circular motions. They are entered either as inches or mm.

L Loop count for repeated cycles – Specifies a repetition count for some canned cycles and auxiliary functions.

N Number of block – Identifies or numbers each block of a program (optional).

O Program number/name – Used to identify a program. It is followed by a number between 0 and 99999.

P Delay time or program number – Used to enter either a time in seconds or a program number for a subroutine call.

Q Canned cycle optional data – Used in canned cycles and is followed by a signed number, between 0 and 8380.000 for inches or between 0 and 8380.00 for metric.

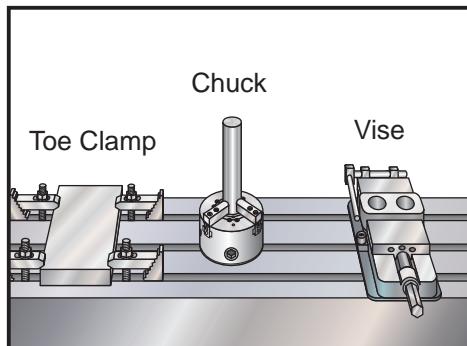
R Canned cycle and circular optional data – Defines the reference plane for canned cycles and circular interpolation. R is followed by a signed number between -15400.0000 and 15400.0000 for inches or between -39300.000 and 39300.000 for millimeters.

S Spindle speed command – Used to specify the spindle speed.

T Tool selection code – Used to select the tool for the next tool change.

PART SETUP

It is necessary to properly secure the part to the table. This can be done a number of ways, using vises, chucks or using T-bolts and toe clamps.





TOOLING

Tool Functions (Tnn)

The **Tnn** code is used to select the next tool to be placed in the spindle from the tool changer. The **T** address does not start the tool change operation; it only selects which tool will be used next. M06 and will start a tool change operation, for example T1M06 will put tool 1 in the spindle.

NOTE: There is no **X** or **Y** motion required prior to performing a tool change and, however, if your work piece or fixture is large, you may need to position **X** or **Y** prior to a tool change in order to prevent a crash between the tools and the part or fixture.

You may command a tool change with the **X**, **Y**, and **Z** axes in any position, and the control will bring the **Z** axis up to the machine zero position prior to starting the tool change. The control will move the **Z** axis to a position above machine zero during a tool change but will never move below machine zero. At the end of a tool change, the **Z** axis will be at machine zero.

Tool Holders

There are number of different spindle options for the Haas mills. Each of these types requires a specific tool holder. The most common spindles are the #40 and #50 taper. 40 taper spindles are divided into two types, BT and CT; these are referred to as BT40 and CT40. The tool changer is only capable of holding one type. The decal on the front of the machine will tell what type of tooling the machine is set up for.

The 50-taper spindle option use CT #50 taper tool holders; referred to as "CT 50."

Pull Studs

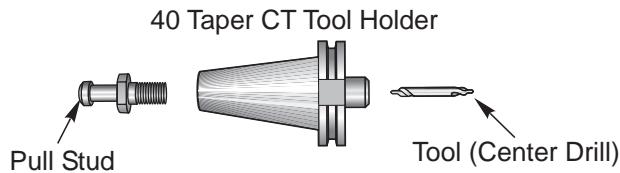
A pull stud or retention knob is required to secure the tool holder into the spindle. Pull studs are threaded into the top of the tool holder and are specific to the type of spindle. The following chart describes the pull studs used on the Hass mill. Do not use the short shaft or pull studs with a sharp right angle (90-degree) head; they will **not** work and will cause serious damage to the spindle.

CT CAT V-Flange					40T CT	50T CT	40T BT	50T BT
40T	2.69	2.50	.44	5/8"-11	1.75	• TPS24CT (TSC) • PS24CT (Non-TSC)	5/8-11 Inch Threads JMTBA Standard MAS 403 P40T-1	5/8-11 Inch Threads JMTBA Standard MAS 403 P40T-1
50T	4.00	3.87	.44	1"-8	2.75	• TPS24CT50 (TSC) • PS24CT50 (Non-TSC)	1"-8 Inch Threads JMTBA Standard MAS 403 P50T-1	1"-8 Inch Threads JMTBA Standard MAS 403 P50T-1
BT MAS 403					• TPS24BT (TSC) • PS24BT (non-TSC)	M16 X 2 Threads JMTBA Standard MAS 403 P40T-1	M16 X 2 Threads JMTBA Standard MAS 403 P40T-1	M24 X 3 Threads JMTBA Standard MAS 403 P50T-1
40T	2.57	2.48	.65	M16X2	1.75	• TPS24E50 (TSC) • PS24E50 (Non-TSC)	1.104 TSC only Ø 0.172 Thru. 45°	1.386 TSC only Ø 0.31 Thru. 45°
50T	4.00	3.94	.91	M24X3	2.75		1.386 TSC only Ø 0.31 Thru. 45°	1.780 TSC only Ø 0.31 Thru. 45°



Tool Holder Assembly

Tool holders and pull studs must be in good condition and tightened together with wrenches or they may stick in the spindle. Clean the tool holder body (the part that goes into the spindle) with a lightly oiled rag to leave a film, which will prevent rusting.



Install a tool into the tool holder as instructed by the tool manufacturer.

TOOL CHANGER

There are two types of tool changers available for the Haas mills; these are the side mount tool changer and the umbrella style. Both types are commanded in the same manner, but each one is set up differently.

Before loading tools the mill must be zero returned (power up restart button), which should have been done at machine power up.

The tool changer is manually operated using the tool release button and the ATC FWD and ATC REV buttons. There are two tool release buttons; one on the side of the spindle head cover and the second on the keypad.

Loading the Tool Changer

Specifications – Do not exceed the maximum specifications

Side-Mount

	40-Taper VF1-4, (EC300-500)	40-Taper (MDC 500), VF 5-11	50-Taper VF 5	50-Taper VF 6-11 (HS 3-7), [EC-1600]	Shuttle 40-Taper VF1-5
Max.Tool Diameter with all pockets full	3"	3"	4"	4"(4.9") [4"]	8"
Max.Tool Diameter if tool is declared oversized	5" (6")	6"	7"	10" (9.8") [20"]	N/A
Max.Tool Length from gauge line	11"	16" (12")	16"	16" (23.6") [30"]	N/A
Max.Tool Weight	12 lb.	12 lb.	30 lb.	30 lb.	12 lb. 120 lb Max. Total Tool Weight

CAUTION! Extremely heavy tool weights should be distributed evenly. This means heavy tools should be located across from one another, not next to each other. Ensure there is adequate clearance between tools in the tool changer; this distance is 3.6" for a 20-pocket.

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

CAUTION: Keep clear of the tool changer during power up, power down, and any tool changer operations.

Tools are always loaded into the tool changer by first installing the tool into the spindle. **Never load a tool directly into the tool changer.**

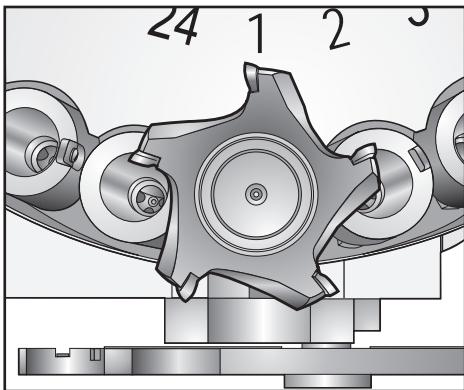
Note: Tools that make a loud bang when being released indicate a problem and should be checked before serious damage to the tool changer occurs.



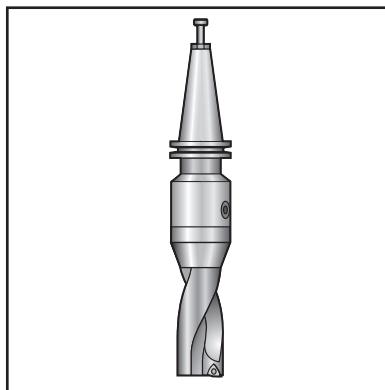
Tool Loading for a Side Mount Tool Changer

NOTE: A normal size tool has a diameter of less than 3" for 40-taper machines, or less than 4" for 50-taper machines. Tools larger than these measurements are considered large size.

1. Ensure the tools you will be loading are of the correct pull stud type for your mill.
2. Enter Handle Jog mode.
3. Press the OFFSET button. From a fresh Power Up/Restart, press the End key, then the Page Down key once to reach the Tool Pocket Table display. From a normal operating state, press Page Up/Down until you reach the Tool Pocket Table.
4. Clear any current "Large" or "Heavy" tool designations. Use the cursor keys to scroll to any tool pocket that has an "L" or "H" next to it. Press Space, then Write/Enter to clear the "Large" or "Heavy" tool designations.



Large (and heavy) tool with surrounding pockets empty



Heavy Tool (not large)

5. Press Origin to reset the Tool Pocket Table to default values. This will place tool 1 in the spindle, tool 2 in pocket 1, tool 3 in pocket 2, etc. This is done to clear the previous Tool Pocket Table settings as well as renumber the Tool Pocket Table for the next program. Another way to reset the Tool Pocket Table is to enter 0 (zero) and press Origin, this will reset all the values to zero.

NOTE: You cannot have two different tool pockets holding the same tool number. Trying to enter a tool number already displayed in the Tool Pocket Table will result in an "Invalid Number" error.

6. Determine if your next program will need any large tools. A large tool has a diameter of greater than 3" for 40-taper machines and greater than 4" for 50-taper machines. If you will not be using any large tools, proceed to Step 10. If you will be using large tools, proceed to the next step.
7. Organize your tools to match to your CNC program. Determine the numerical positions of any large tools and designate those pockets as Large in the Tool Pocket Table. To designate a tool pocket as "Large", scroll to that pocket and press L, then Write/Enter.

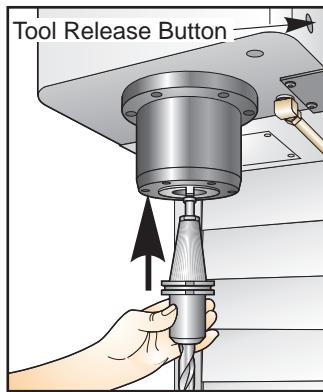
NOTE: When setting up the tooling for your CNC program, large tools need to have the surrounding pockets empty to prevent a tool changer crash. However, large tools can share adjoining empty pockets. Placing large tools next to each other will help conserve space in the tool changer.

CAUTION! A large tool cannot be placed in the tool changer if one or both of the surrounding pockets already contain tools. Proceeding with this action will cause the tool changer to crash.

8. When you have designated all necessary Large and Heavy tool pockets, press the Origin key to renumber the Tool Pocket Table. At this point, the machine is ready to accept tool 1 into the spindle.



9. Take tool 1 in your hand and insert the tool (pull stud first) into the spindle. Turn the tool so that the two cutouts in the tool holder line up with the tabs of the spindle. Push the tool upward while pressing the Tool Release button. When the tool is fitted into the spindle, release the Tool Release button.



10. Press the "Next tool" button.

11. Repeat Steps 9 and 10 until all the tools are loaded.

High-Speed Side Mount Tool Changer

The high-speed tool changer has an additional tool assignment, which is "Heavy". Heavy tools are defined as tools weighing more than 4 pounds. If a tool, heavier than 4 pounds is used, the tool must be entered in the table with an "H" (Note: All large tools are considered heavy). During operation an "h" in the tool table denotes a heavy tool in a large pocket.

As a safety precaution, the tool changer will run at a maximum of 25% of the normal speed if changing a heavy tool. The pocket up/down speed is not slowed down. The control restores the speed to the current rapid, once the tool change is complete. If problems are encountered changing unusual or extreme tooling, contact the Haas Service Department for assistance.

Note: The high-speed side-mount tool changer has the added ability to designate a tool as "Heavy" in the Pocket-tool table. A "Heavy Tool" is defined as a tool weighing 4 pounds or heavier.

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

L - Empty pockets are required on both sides (large tools are assumed heavy).

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

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

Large tools are assumed to be heavy.

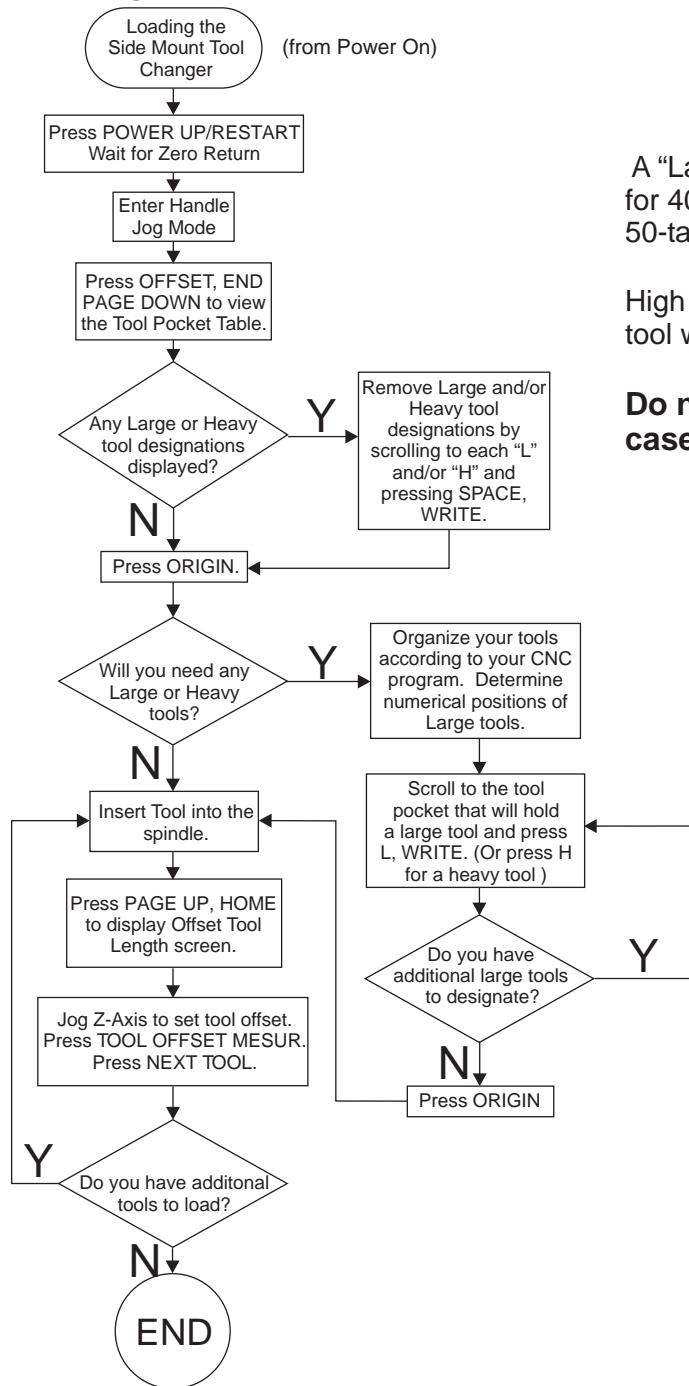
Heavy tools are not assumed to be large.

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

This means that pockets designated as normal size are not dedicated to any particular tool; during tool change operations a normal size tool can be taken from one pocket and put back into another. Tool pockets designated as large size are dedicated only to large tools; large tools will not migrate to an empty normal pocket during a tool change.



Tool Loading Flowchart



A "Large Tool" has a diameter greater than 3" for 40-taper mills, or greater than 4" for 50-taper mills.

High -speed Toolchangers: A "Heavy Tool" is a tool weighing 4 or more pounds.

Do not enter a lower case "h" or lower case "l"(L) in the tool table.

Using 0 for a Tool Designation

A 0 (number zero) can be inserted in the tool table in place of a tool number. If this is done the tool changer does not "see" this pocket and will never try to install or retrieve a tool from pockets with a "0" designation.



Enter 0 to zero all pockets, enter 1 to sequence the pockets, and enter a 3 to clear all H,h,L,I entries.

A 0 **cannot be used** to designate the tool inserted into the spindle. The spindle must always have a tool number designation.

To designate a pocket as an “always empty” pocket: Use the arrow keys to move to and highlight the pocket to be empty, press the **0** button on the numeric keypad and then press

Moving Tools in the Carousel

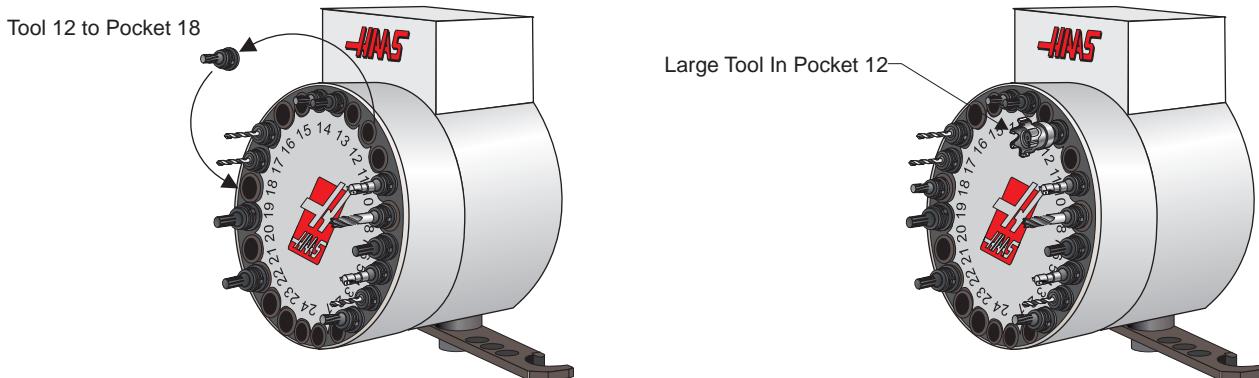
Should you need to move tools around in the carousel, follow the steps below.

CAUTION: Plan out your reorganization of the tools in the carousel ahead of time. To reduce the potential for tool changer crashes, keep your tool movement to a minimum. If you have any large or heavy tools currently in the tool changer, ensure that you only move them between tool pockets designated as such.

Creating Room for a Large Size Tool

The tool changer pictured has an assortment of normal size tools. For the purposes of this example, tool 12 will be moved to pocket 18 to create room for a large size tool to be placed in pocket 12.

1. Select MDI mode. Press the OFSET button. Press Page Up/Down (if necessary) until you reach the Tool Pocket Table display. Verify which tool number is in pocket 12.
2. Enter **Tnn** into the control (where **Tnn** is the tool number from step 1). Press ATC FWD. This will place the tool from pocket 12 into the spindle.
3. Enter P18 into the control, then press ATC FWD. This will place the tool currently in the spindle into pocket 18.



4. Scroll to pocket 12 in the Tool Pocket Table and press L, Write/Enter to designate that pocket as Large.
5. Enter the tool number into SPNDL (spindle) on the Tool Pocket Table. Insert the tool into the spindle.

NOTE: You cannot have two different tool pockets holding the same tool number. Trying to enter a tool number already displayed in the Tool Pocket table will result in an “Invalid Number” error.



6. Enter P12 into the control and press ATC FWD. The tool will be placed into pocket 12.

Umbrella Tool Changer

Tool Loading Tools are loaded into the umbrella tool changer by first loading the tool into the spindle. To load a tool into the spindle, prepare the tool

1. Ensure the tools you will be loading are of the correct pull stud type for your mill.
2. Enter Handle Jog mode.
3. Organize your tools to match to your CNC program.
4. Take tool 1 in your hand and insert the tool (pull stud first) into the spindle. Turn the tool so that the two cutouts in the tool holder line up with the tabs of the spindle. Push the tool upward while pressing the Tool Release button. When the tool is fitted into the spindle, release the Tool Release button.
5. Press the "Next Tool" button.
6. Repeat Steps 8 and 9 until all the tools are loaded.

Umbrella Tool Change Recovery

If the tool changer jams, the control will automatically come to an alarm state. To correct this, push the Emergency Stop button and remove the cause of the jam. Push the RESET button to clear the alarms. Push the Tool Changer Restore button to reset the tool changer.

CAUTION! Never put your hands near the tool changer when powered unless the EMERGENCY STOP button is pressed first.

Tool Changer Recovery

If a problem occurred during a tool change, a tool changer recovery needs to be performed. Enter the tool changer recovery mode by pressing Tool Changer Restore. Once in tool changer recovery mode, the operator is given instructions and questions to perform a proper tool changer recovery. The entire tool changer recovery process must be completed before exiting. If an exit was done prematurely the tool changer recovery must be started from the beginning.

Side Mount Tool Changer Recovery

If a problem occurred during a tool change, a tool changer recovery needs to be performed. Enter the tool changer recovery mode by pressing Tool Changer Restore. Once in tool changer recovery mode, the operator is given instructions and questions to perform a proper tool changer recovery. The entire tool changer recovery process must be completed before exiting. If an exit was done prematurely the tool changer recovery must be started from the beginning.

EC-300 / EC-400 / MDC Tool Changer Door and Switch Panel

The Manual/Auto switch must be set to "Auto" for automatic tool changer operation. If the switch is set to "Manual", the other two buttons, labeled CW and CCW, are enabled and automatic tool changes are disabled. The CW and CCW buttons rotate the tool changer in the clockwise and counterclockwise directions. The door has a switch which detects when the door is open.

Operation

If the cage door is opened while a tool change is in progress, the tool change will stop and not resume until the cage door is closed. However, any machining operations that are in progress will continue.

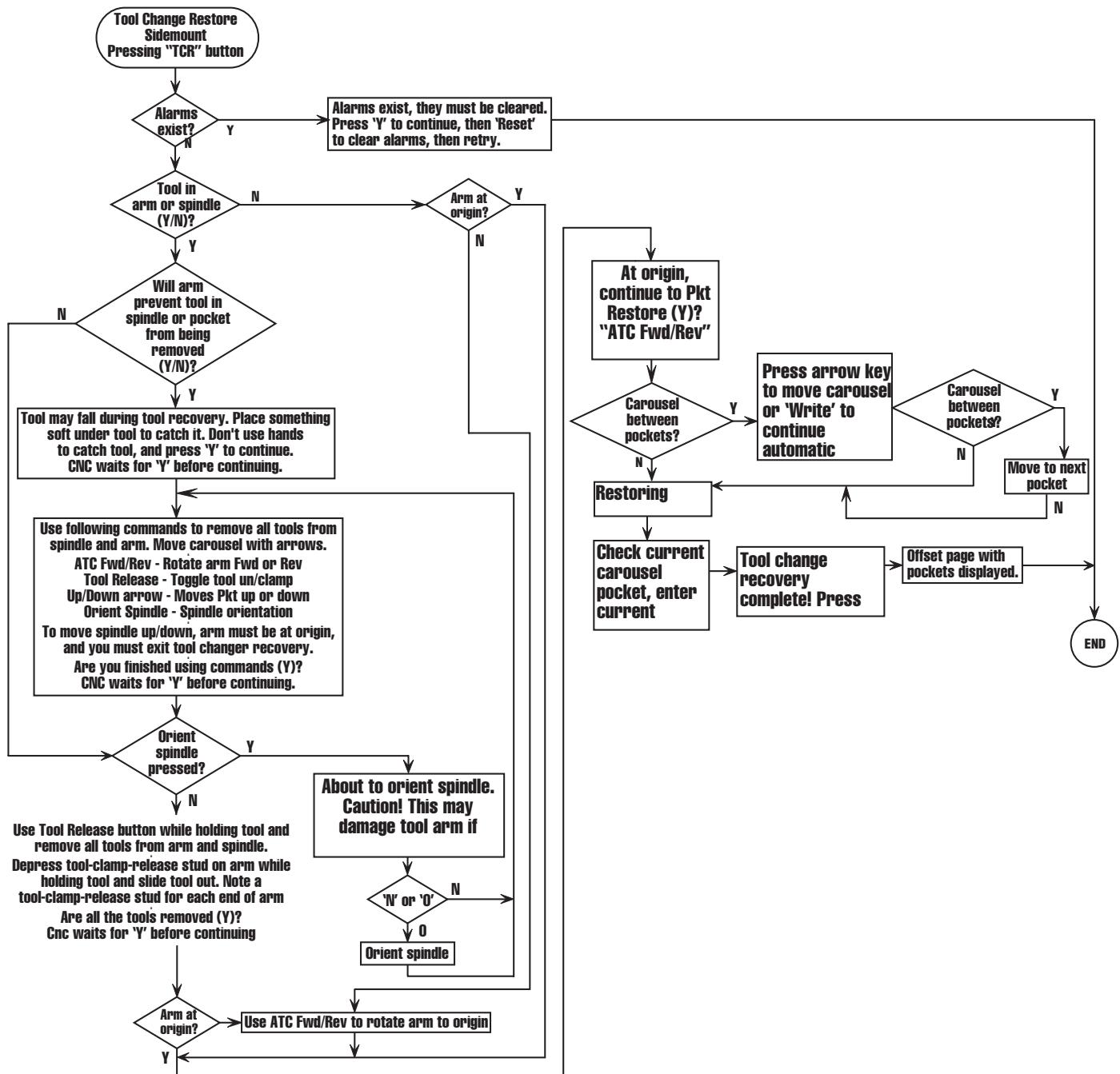
If the switch is turned to "Manual" while a tool change is in progress, current tool changer motion will be completed. The next tool change will not execute until the switch is turned back to "Auto". Any machining operations that are in progress will continue.

The carousel will rotate one position whenever the CW or CCW button is pressed once, while the switch is set to "Manual"



During tool changer recovery, if the cage door is open or the switch is in the "Manual" position and Tool Changer Restore is pressed, a message is displayed telling the operator the door is open or is in manual mode. The operator must close the door and set the switch to the automatic position in order to continue.

Side Mount Tool Changer Recovery Flow Chart





Hydraulic Tool Changer

Tool Pocket Setup

The Tool pocket table is accessed by pressing the Offset key and then press Page Up/Down, until the page titled "Pockets with Tool Numbers" is displayed. Enter the values for each tool used. This table must be properly setup by the operator to avoid the possibility of damaging tools, the spindle or the tool changer.

Creating a New Tool Table

During the course of operating the machining center it will become necessary to completely reprogram the tool table. There are two useful functions that can be used as an aid in creating a new tool table:

Pressing the '**ORIGIN**' button on the keypad at any time while on the tool table screen will set all the tool pockets to their default values. For example, tool 1 in the spindle, tool 1 in pocket 1, tool 2 in pocket 2, and so on.

Pressing '0' and then the '**ORIGIN**' button on the keypad at any time while on the tool table screen will set all the tool pockets to '0'.

Tool Numbering System

Powering up the machine for the first time sets the default tool-pocket table. The table is setup such that each pocket contains a tool with the same number as the pocket number. The spindle is initialized to contain Tool 1 (T1) . For example, Pocket 1 is dedicated to contain Tool 1 (T1), Pocket 2 is dedicated to Tool 2 (T2), etc. Therefore, the ATC 38 side-mount tool changer has a default of 38 tools, T1 through T38 with T1 being in the spindle.

The tool numbers in the tool table assigns a dedicated tool number to each pocket of the tool changer. This number remains in the tool table regardless of where the tool actually is. For example, if tool 5 (T5) is removed from the tool pocket 5 and placed in the spindle, the tool table will indicate that T5 is in the spindle and that pocket 5 is dedicated to tool T5.

A tool called for from the machine's program commands the control to search the tool table for the **tool number** and indexes the tool changer to the **pocket** containing the tool number.

CAUTION: Damage to the machine and/or can occur if the tool called for, from the program, does not match the tool listed in the tool table and/or installed in the corresponding pocket.

Acceptable Tool Numbers

In general the tool numbers are from T1 to the number of pockets in the tool changer chain (T38 on a 30-pocket tool changer, however, it is possible to use all the numbers in the tool table. This is done to allow for a situation where the operator requires more than available tool pockets to complete the machining job. For example, say 55 different tools are required to perform a complete machining job. The first 38 tools would be used and then the machine would be paused while the operator removes 17 of the original tools and replaces them with the 17 tools required to complete the job. After updating the tool table to reflect the new tools in their respective pockets using tool numbers T39 through T55 the machining job is ready to continue.

Using '0' for a Tool Designation

A tool pocket can be labeled as an "always empty" pocket, by entering "0" (zero) for the tool number in the tool table. If this is done the tool changer does not "see" this pocket and will never try to install or retrieve a tool from pockets with a '0' designation.

A 0 **cannot be used** to designate the tool inserted into the spindle. The spindle must always have a tool number designation.

Use of "Large" Tool Designations

The tool changer can accept oversized tools as long as the adjacent pockets are left empty and a designator is added to the tool table. **Oversize (large) tools are tools with a diameter of more than 4.9" (125mm).**



To designate a large tool, scroll to highlight the specific pocket (the one that holds the large tool) and press "L" and then the 'WRITE/ENTER' button. The letter "L" will appear on the screen next to this pocket and the tool numbers on either side will be changed to "-" indicating that tools cannot be inserted into these pockets.

To remove an 'L' designation, highlight the 'L' pocket and press the 'SPACE' button and then the 'WRITE/ENTER' button.

Note: Large tools cannot be bigger than 9.8" (250mm).

Heavy tools

Designating a tool as "Heavy" will have no affect on tool changer speed or actions.

Installing/Removing Tools (hydraulic tool changer)

Tools can be installed into the tool changer by inserting them directly into the pockets or by installing them into the spindle and using the ATC FWD/REV buttons on the keypad to stow the tools in the tool changer. During initial setup, using the ATC FWD/REV buttons, the first tool inserted into the spindle will be tool T1 and will be placed into pocket 1.

To install them directly in the tool changer chain, turn switch on the operator's sub-panel to "Manual", open the operator door (it is at the back of the tool changer cage), and install the tool in the chain. Use the CW/CCW buttons to move the chain to the next empty pocket. Remember to leave an empty pocket on either side of a large tool.

Update the tool changer tool table once all the tools are installed.

Tools can be removed, by calling each tool to the spindle and removing them from it, or by taking them directly from the chain at the operator station in the tool cage. To remove tools from the chain, turn the switch to "Manual", open the tool-cage operator door, grasp the tool and press the foot pedal.

ATC FWD, ATC REV Operation (hydraulic tool changer)

Using ATC FWD/REV will cause the tool changer to move to the next tool pocket with respect to the tool in the spindle. For example, if tool T15 is in the spindle and it is dedicated to pocket 20, the tool changer will put tool T15 into pocket 20 and then ATC FORWARD to pocket 21. **It will not retrieve tool 16 (T16).**

The ATC FWD/REV buttons will change the tool, in the spindle, to the next or previous tool. However, if the next, or previous, tool is a pocket designated with a zero (an empty pocket) the tool changer will skip that pocket and get a tool from a non-zero pocket.

Tool Changer Recovery (hydraulic tool changer)

The tool changer recovery mode is used to manually move the tool changer's arm and carriage into its HOME position.

Press the "Recover" button and follow the on-screen prompts to return the tool changer to the home position.

Jog Mode

Jog Mode allows you to jog each of the axes to a desired location. Before jogging the axes it is necessary to home (beginning axes reference point) the axes (See the Machine Power-up Section).

To enter jog mode press the handle jog button, then press one of the desired axes (e.g. X, Y, Z, A or B etc.) and either use the handle jog buttons or the jog handle to move the axes. There are different increment speeds that can be used while in jog mode they are; .0001, .001, .01 and .1.

An optional Remote Jog Handle (RJH) can also be used to jog the axes. The RJH consists of a the jog handle, Cycle Start button, Feed Hold button, Axes selection dial and an increment dial.

To jog the axes with the optional Remote Jog Handle (RJH) select the desired axes (X, Y, Z, A, B, C or V) using the dial, choose the increment value (X1, X10 or X100) and use the jog handle to move the axes.



Setting Offsets

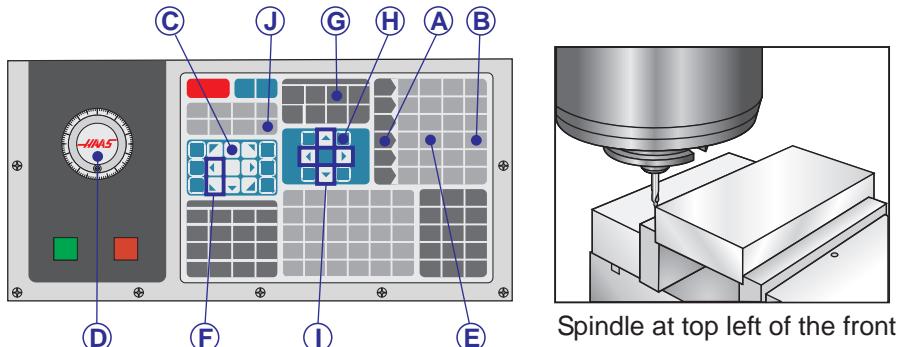
In order for the mill to accurately machine a work piece, the mill needs to know where the part is located on the table. Jog the mill with a pointer tool in the spindle, until it reaches the top left corner of the part (see the following illustration) this position is part zero. The values will be entered into G54 on the Work Offset page.

Offsets can also be entered manually by choosing one of the offsets pages, moving the cursor to the desired column, typing a number and pressing Write or F1. Pressing F1 will enter the number in the selected column. Entering a value and pressing Write will add the amount entered to the number in the selected column.

Typical Work Offset Set-up

1. Place the material in the vise and tighten
2. Load a pointer tool in the spindle
3. Press Handle Jog (A)
4. Press .1/100. (B) (The mill will move at a fast speed when the handle is turned)
5. Press +Z (C)
6. Handle jog (D) the Z-axis approximately. 1" above the part
7. Press .001/1. (E) (The mill will move at a slow speed when the handle is turned)
8. Handle jog (D) the Z-axis approximately. 0.2" above the part
9. Select between the X and Y axes (F) and handle jog (D) the tool to the upper left corner of the part (See the following illustration)
10. Press Ofset (G)
11. Press Page Up (H) repeatedly until you see the Work Zero Offset Page
12. Cursor (I) to G54 Column X
13. Press Part Zero Set (J) to load the value into the X-axis column. the second press of Part Zero Set (J) will load the value into the Y-axis column.

CAUTION! Do Not Press Part Zero Set a third time; doing so will load a value into the Z-axis. This will cause a crash or Z-axis alarm when the program is run.



Setting the Tool Offset

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

1. Load the tool in the spindle (See 5 Cutting Tools in this guide)
2. Press Handle Jog (A)

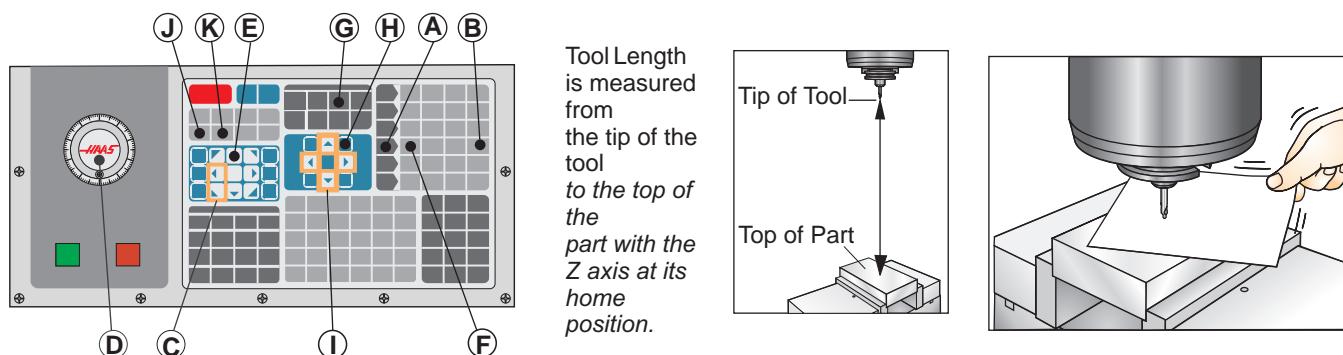


3. Press .1/100. (B) (The mill will move at a fast rate when the handle is turned)
4. Select between the X and Y axes (C) and handle jog (D) the tool near the center of the part
5. Press +Z (E)
6. Handle jog (D) the Z axis approximately 1" above the part
7. Press .0001/.1 (F) (The mill will move at a slow rate when the handle is turned)
8. Place a sheet of paper between the tool and the work piece. Carefully move the tool down to the top of the part, as close as possible, and still be able to move the paper
9. Press Offset (G)
10. Press Page Up (H) until the page with "Coolant - Length - Radius at the top and scroll to tool #1.
11. Cursor (I) to Geometry for position #1
12. Press Tool Offset Mesur (J)

This will take the Z position located in the bottom left of the screen and put it the tool number position

CAUTION! The next step will cause the spindle to move rapidly in the Z axis

13. Press Next Tool (K)



Additional Tooling Set-up

There are other tool set-up pages within the Current Commands. Press Curnt Comds and then use the Page Up/Down buttons to scroll to these pages.

The first is the page with "Spindle Load" and "Vibration" at the top of the page. The programmer can add a tool load limit for spindle load and vibration. The control will reference these values and can be set to do a specific action should the limitations be reached (See setting 84).

The second page is the Tool Life page. On this page there is a column called "Alarm". The programmer can put a value in this column which will cause the machine to stop once the tool has been used that amount of times.

Setting Part (Work piece) Zero

Part Zero is a user defined reference point that the CNC control will use to program all moves from.

1. Select tool #1 by pressing MDI, enter "T1" and press ATCFWD.
2. Jog X and Y until the part is approximately under the spindle.
3. Jog the Z-axis down, first by using the .1 jog increment
4. Press .0001/.1
5. Place a sheet of paper between the tool and the work piece. Carefully move the tool down to the top of the part, as close as possible, and still be able to move the paper. Do not jog any further or damage to the tool will result.



6. Set the G54 work offsets by highlighting G54 X-axis on the Work Offsets screen and press Part Zero Set.
7. The control will automatically switch to the Y-axis. Press Part Zero Set again the Y position will be copied to the G54 Y offset.
8. Repeat steps 1-7 for all the tools used in the program.

Advanced Tool Management

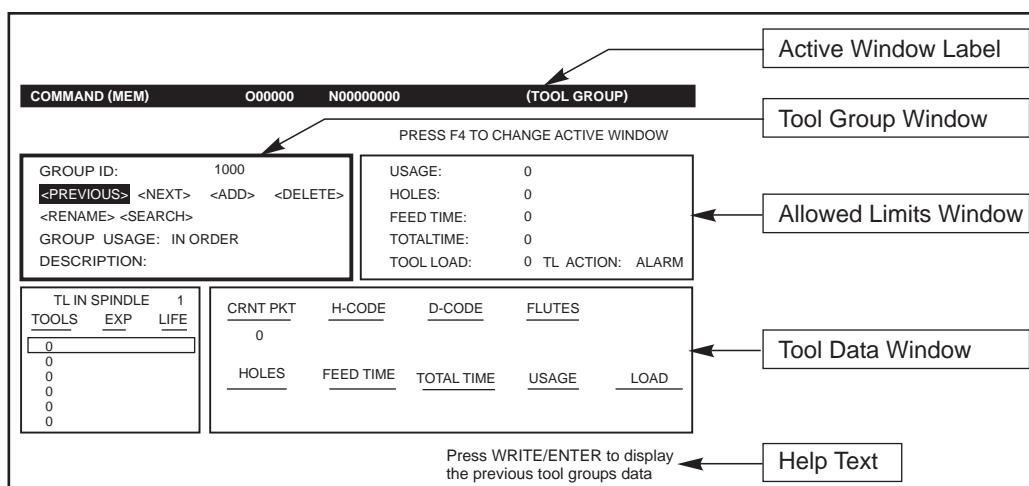
Introduction

Advanced Tool Management (ATM) allows the programmer to setup and access duplicate tools for the same or a series of jobs. The ATM page is located in the Current Commands mode (Press Current Commands button and page up once). A sample of the ATM screen follows; the screen has "(TOOL GROUP)" in the title.

Duplicate or backup tools are classified into specific groups. The programmer specifies a group of tools instead of a single tool in the G-code program. ATM will track the usage of individual tools in each tool group and compare it to user defined limits. Once a limit (e.g. number of times used, or tool load) has been reached the mill will automatically choose one of the other tools in the group the next time that tool is needed.

To activate ATM, make sure that Setting 7 (Parameter lock) is off and press E-stop. Under Parameter 315, bit 28, change the value from "0" to "1" and press F4 to switch between windows. Use the cursor keys (left, right, up, down) to move to different items inside the active window. Pressing the Enter key will select, modify or clear values in each item depending on the selection.

The bottom right corner displays simple help information for selected items.





Operation

Tool Group

In the Tool Group Window the operator defines the tool groups used in the programs.

PREVIOUS – Highlighting <PREVIOUS> and pressing Enter changes the display to the previous group.

NEXT – Highlighting <NEXT> and pressing Enter changes the display to next group.

ADD – Highlight <ADD>, enter a number between 1000 and 2999, and press Enter to add a tool group.

DELETE – Use <PREVIOUS> or <NEXT> to scroll to the group to delete. Highlight <DELETE> and press Enter.

You will be asked to confirm the deletion; answering 'Y' will complete the deletion; answering 'N' will cancel the deletion.

RENAME - Highlight <RENAME>, enter a number 1000 and 2999 and press enter to renumber the group ID.

SEARCH - To search for a group, highlight <SEARCH>, enter a group number and press enter.

GROUP ID – Displays the group ID number.

GROUP USAGE – Enter the order in which the tools in the group are called. Use the left and right cursor keys to select how the tools are used.

DESCRIPTION – Enter a descriptive name of the tool group.

Allowed Limits

The Allowed Limits window contains the user defined limits to determine when a tool is worn out. These variables affect every tool in the group. Leaving any variable set to zero causes them to be ignored.

FEED TIME – Enter the total amount of time, in minutes, a tool is used in a feed.

TOTAL TIME – Enter the total time, in minutes, a tool is used.

TOOL USAGE – Enter the total times a tool is used (number of tool changes).

HOLES – Enter the total number of holes a tool is allowed to drill.

TOOL LOAD – Enter the maximum tool load (in percent) for the tools in the group.

TL ACTION* – Enter the automatic action to be taken when the maximum tool load percentage is reached. Use the left and right cursor keys to select the automatic action.

Tool Data

TL IN SPINDLE – Tool in the spindle.

TOOL – Used to add or remove a tool from a group. To add a tool press F4 until the Tool Data window is outlined. Use the cursor keys to highlight any of the areas under the "Tool" heading and enter a tool number. Entering zero will clear the tool or highlighting the tool number and pressing ORIGIN will reset the H-Code, D-Code and Flutes data to the default values.

EXP (Expire) – Used to manually obsolete a tool in the group. To obsolete a tool, enter a '*' or to clear a '*', press enter.

LIFE – The percentage of life left in a tool. This is calculated by the CNC control, using actual tool data and the limits the operator entered for the group.

CRNT PKT – The tool changer pocket the highlighted tool is in.

H-CODE* – The H-code (tool length) that will be used for the tool. H-code cannot be edited unless Setting 15 H & T Code Agreement is set to Off. The operator can change the H-code by entering a number and pressing Enter. The number entered will correspond to the tool number in the tool offsets display.

D-CODE* – The D-code that will be used for that tool. D-code can be changed by entering a number and pressing Enter.

*Note: By default the H and D-codes in Advanced Tool Management are set to equal the tool number that is added to the group.

FLUTES – The number of flutes on the tool. This can be edited by selecting it, entering a new number, and pressing Enter. This is the same as the "Flutes" column listed on the tool offsets page.



Highlighting any of the following sections (Holes through Load) and pressing ORIGIN will clear their values. To change the values, highlight the value in the specific category, enter a new number and press enter.

LOAD – The maximum load, in percent, exerted on the tool.

HOLES – The number of holes the tool has drilled/ tapped/ bored using Group 9 canned cycles .

FEED TIME – The amount of time, in minutes , the tool has been in a feed.

TOTAL TIME – The total amount of time, in minutes, the tool has been used.

USAGE – The number of times the tool has been used.

Tool Group Setup

To add a tool group press F4 until the Tool Group window is outlined. Use the cursor keys until <ADD> is highlighted. Enter a number between 1000 and 2999 (this will be your group ID number). To change a group ID number, highlight the <RENAME> feature, enter a new number and press enter.

Tool Group Usage

A tool group must be setup prior to using a program. To use a tool group in a program first set up a tool group.

Next, substitute the tool group ID number for the tool number and for the H-codes and D-codes in the program. See the following program for an example of the new programming format.

Example:

T1000 M06 (tool group 1000)

G00 G90 G55 X0.565 Y-1.875 S2500 M03

G43 H1000 Z0.1 (H-code 1000 same as group ID number)

G83 Z-0.62 F15. R0.1 Q0.175

X1.115 Y-2.75

X3.365 Y-2.875

G00 G80 Z1.0

T1000 M06 (this will check if the tool in that group is still good to use)

G00 G90 G56 X0.565 Y-1.875 S2500 M03

G43 H00 Z0.1 (H00 cancels H Code)

G83 Z-0.62 F15. R0.1 Q0.175

X1.115 Y-2.75

X3.365 Y-2.875

G00 G80 Z1.0

M30

Macros

Tool Management can use macros to obsolete a tool within a tool group. Macros 8001 to 8200 represents tool 1 through 200. By setting one of these macros to 1, the operator can expire a tool.

Example:

#8001 = 1 (this will expire tool 1 and it will no longer be used)

#8001 = 0 (if tool 1 was expired manually or with a macro, then setting macro 8001 to 0 will make tool 1 available again for use)

See the Operator manual for further information on macros.

Save and Restore Advanced Tool Management tables

The control can save and restore the variables associated with the Advanced Tool Management (ATM) feature to floppy disk and RS-232. These variables hold the data that is entered on the ATM screen. The information can be saved, either as part of an overall backup by using the LIST PROG/POSIT page, or save only the ATM information by bringing up the ATM display screen and pressing F2. When the Advanced Tool Management data is saved as part of an overall backup, the system creates a separate file with a .ATM extension. The ATM data can be saved and restored via the RS232 port by pressing the SENDRS232 and RECV232 buttons while the Advanced Tool Management screen is displayed.

Optional Programmable Coolant Spigot

The optional programmable coolant (P-cool) spigot allows the user to direct the coolant stream at the work piece in order to flush chips from the cutting area. The direction of the coolant can be changed by the CNC program.

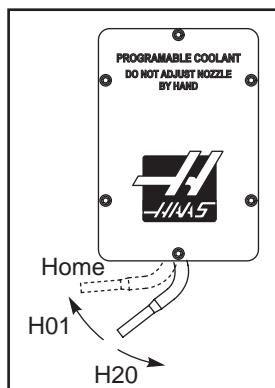


An additional column on the tool offset page titled CLNT POS will be displayed. The spigot will be moved to the position entered into the specific field when the associated H code and M08 is called.

Programmable Coolant (P-Cool) Set-up

1. Press the OFFSET button to enter the offsets page, press the CLNT UP or CLNT DOWN button to move the P-cool nozzle into the desired position. Press the COOLNT button to turn on the coolant in order to check the P-cool position. Note: The P-cool position is displayed at the bottom left corner of the screen.
2. Enter the coolant position number for the tool in the Coolant Position column.
3. Repeat steps 1 and 2 for each tool.
4. Enter the coolant position as an H code (for example H2 will command the coolant to the position entered in the Tool 2 Coolant Position column) in the program.

If setting 15 (H & T Agreement) is on the H code and T code commanded in the program must be the same (e.g. T1 H1 must be commanded). If setting 15 is turned off the H code and T code commanded does not need to be the same (e.g. T1 H2 can be commanded).



Warning

Do not rotate the coolant spigot by hand. Serious motor damage will occur.

Minimum Oil Machining (MOM)

Minimum Oil Machining (MOM) is an automated process, which can operate in two modes: lubricating a tool during a canned cycle operation (Canned Mode), or as an alternative to flood coolant (MOM Mode). MOM Mode is similar to Near Dry Machining (NDM) and Minimum Quantity Lubrication (MQL), and can produce dry or near dry chips. Canned Mode is used to lubricate taps, drills, or boring bars during Canned Cycles. Parameters 696-699 and M codes 101-103 are used with this option, refer to these sections for further information. MOM can be setup and tested by using the MOM page of the CURNT COMDS display. The MOM page displays the following information:

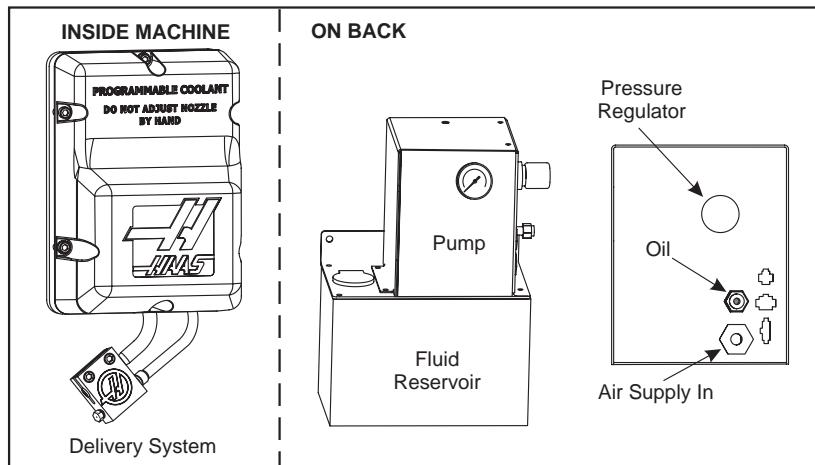
MOM Override:	None -	Use M-Codes to operate MOM.
	Ignore -	Ignore MOM M-Codes.
	Canned Cycle -	Act as if M101 is always active (squirt per G-Code).
	Manual -	Turns MOM Mode on (squirt every time between squirts)
Squirt On Time Canned Cycle (M101 Ix.xxx):	0.100 sec	(Tapping)
Squirt On Time MOM (M102 Ix.xxx):	0.050 sec	
Time Between Squirts MOM (M102 Jx.xxx):	2.000 sec	(Time between squirts)

Pressing display page function keys will perform specific operations useful in setting up the MOM system:

- F1** - Squirt once
F2 - Toggle MOM On and Off



F3 - Toggle Purge On and Off



Graphics Mode

A safe way to trouble shoot a program is by running it in Graphics mode. No movement will occur on the machine, instead the movement will be illustrated on the screen.

Graphics mode can be run from Memory, MDI or DNC modes. To run a program press the SETNG/GRAFH button until the Graphics page is displayed. To run DNC in graphics, you must select DNC first, then go to graphics display and send your program to the machines control (See the DNC section). There are three helpful display features in Graphics mode that can be accessed by pressing one of the function keys (F1, F2, F3 and F4). F1 is the help button, which will give a short description of each of the functions possible in Graphics mode. F2 is the zoom button, which will zoom in on an area of the graphics screen by using the arrow buttons and pressing the Write button. F3 is the positions button, which will allow you to view the different Positions pages while in graphics mode. F4 is the program button, which will display the current program.

NOTE: Not all machine functions or motions are simulated in graphics.

Dry Run Operation

The Dry Run function is used to check a program quickly without actually cutting parts. Dry Run is selected by pressing the Dry Run button while in MEM or MDI mode. When in Dry Run, all rapids and feeds are run at the speed selected with the jog speed buttons.

Dry Run can only be turned on or off when a program has completely finished or the Reset button is pressed. Dry Run will still make all of the requested tool changes. The override keys can be used to adjust the Spindle speeds in Dry Run. Note: Graphics mode is just as useful and may be safer as it does not move the axes of the machine before the program is checked (see the previous section on the Graphics function).

Running Programs

To run a program one must be loaded on the machine. Once a program is entered and the offsets are set, run the program by pressing the Cycle Start button. It is suggested that the program is run in Graphics mode before doing any cutting.

Background Edit

Background Edit will allows editing of one program while another program is running.

Background Edit is activated in Mem mode by pressing the Prgrm/Convrs button, typing the name (Onnnnn) of a program to edit and pressing F4. If the program name is not entered the Program Review is displayed, which will allow the program currently running to be edited. Selecting any other display or pressing F4 will exit from Background Edit. While a program is running, press the List Prog button to display a list of programs in the mill.

All of the changes made during Background Edit will not affect the running program, or its subprograms. The changes will go into affect the next time the program is run.



The Cycle Start button may not be used while in Background Edit. If the program contains a programmed stop (M00) exit Background Edit (press F4) and then press Cycle Start to resume the program.

PALLET CHANGER (EC-SERIES AND MILL DRILL CENTER)

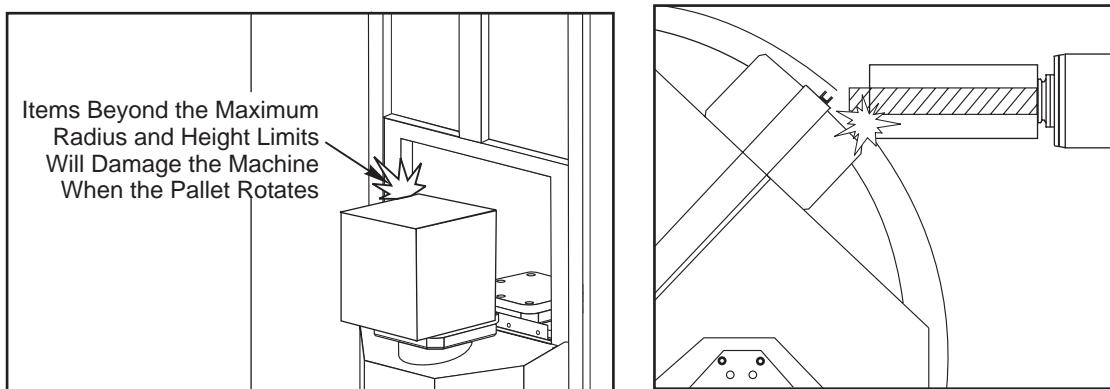
Introduction

The pallet changer is commanded through a program in CNC memory. The M50 (execute pallet change) function consists of unlocking, lifting and rotating the pallets, then lowering and locking the pallets again. The pallet changer rotates the pallets 180°, then back; it does not continuously rotate in the same direction.

The pallet changer is provided with an audible signaling device to alert nearby personnel when a pallet change is taking place. However, do not depend on the signal to avoid mishaps.

Pallet Changer Warnings and Cautions

- Large workpieces may collide into frame during a pallet change.
- Verify tool length clearance during pallet changes. Long tools may collide with workpiece.



EC-300 Shown

Maximum Pallet Loads

EC-300 550lb (249kg) per station, balanced within 20%

MDC 700lb (318kg) per station, balanced within 20%

EC-400 1 and 45 degree indexer – 1000 lb per pallet

Full 4th Axis – 660 lb per pallet

Pallet Changer Operation

The Pallet Changer is commanded using M Codes.

M50 determines if a pallet has been scheduled. The pallets will change if a pallet is scheduled or the program will pause and prompt the operator the pallet is not scheduled.

G188 uses the pallet schedule table to load and run the program scheduled for the current pallet. Once the part program completes, the M99 command loops back to the M50 (pallet change) in order to load the next pallet.

M36 is not used in this method, as M50 monitors the pallet scheduling. M36 is provided for backward compatibility, and to program pallet changes without using PST.



Messages aid the operator with loading/unloading and pallet changes. For example, at the beginning of the M50 pallet change, if the load station is not ready, a message is displayed on the screen. The message will not stop flashing and the pallet change will not continue until the load station is ready and the Part Ready button is pressed. At the beginning of a pallet change, if the load station is ready, no messages appear, and no button press is required, and the pallet change starts directly.

M46 – Qn Pmm

Jump to line mm in the current program if pallet n is loaded, otherwise go to the next block.

M48 – Validate that current program is appropriate for loaded pallet

Checks in the Pallet Schedule Table that the current program is assigned to the loaded pallet . If the current program is not in the list or the loaded pallet is incorrect for the program, an alarm is generated. M48 can be in a program listed in the PST, but never in a subroutine of the PST program. An alarm will occur if M48 is incorrectly nested.

M49Pnn Qmm – Sets the status of pallet nn to a value of mm.

Without a P-code, this command sets the status of the currently loaded pallet. The status of each pallet are defined in the pull-down menu of the (PST).

Operator Load Station (EC-300, EC-400, MDC)

To ease loading/unloading parts and to speed production pallet changing mills have an additional load area. The load station is guarded by a door, and a sub-panel includes a few buttons to control the pallet changer. As a safety precaution the load station door must be closed before a pallet change can occur.

Note: The EC-400 must have the pallet in the load station at home to do a pallet change.

Sub-Panel Controls

Emergency Stop: The button behaves just like the one on the operator's pendant.

Rotary Index (EC-300 only): Rotates the load station pallet (see Setting 164).

Part Ready: Used to indicate the pallet is ready. It also contains a light that 1) **blinks** when the control is waiting for the operator or 2) is **on** when the operator is ready for a pallet change.

Pallet Changer G-Code

G188 Get Program From PST

Calls the parts program for the loaded pallet based on the PST entry for the pallet

Pallet Changer Programming

The Pallet Changer can be programmed to run the same parts program on both pallets, or a different program on each pallet. See "Sample Programs" for some of the options available for pallet change programming.

Method 1

The following method is the preferable to accomplish a pallet change.

To perform automatic pallet sequencing and part program selection, each pallet must be "scheduled" and must have a part program assigned to it. Scheduling is done in two ways, the first is a pallet can be scheduled with the Part Ready button on the operator's panel. Pushing the button schedules the pallet that is outside of the machining area.



The second is the pallets can be scheduled from the Pallet Schedule Table (PST). This display can be found by pressing the CURNT COMDS key and then pressing the Page Up or Page Down keys until the Pallet Schedule Table page is reached. Use the arrow keys to highlight the "Load Order" box for the pallet. Enter a pallet number and then the Write/Enter key. If there is already a priority number for that pallet, the "Load Order" numbers of the other pallets will be updated as necessary. A pallet that is on the receiver (in the work area) will have an asterisk in the "Load Order" column; this pallet cannot be scheduled.

Assigning a part program is also done in the PST display. The arrow keys are used to highlight the "Program Number" box for the pallet. The program number is entered by keying the number and then pressing the Write/Enter key. For example, keying "O123", then Write/Enter will put the program number O00123 into the table.

If a parts program encounters an M50 (without a P code) and the Part Ready button has not been pressed, the control will pause operation, the beacon will blink green, and the message "None Scheduled" is displayed. The mill will wait until the Part Ready button has been pressed, or the PST has been updated, before performing the pallet change. This feature prevents a pallet change from occurring before the operator is ready. The Part Ready button can be pressed at any time and will be recognized when the next pallet change is required.

Method 2

Although the previous method is recommended, the pallet changer can also be operated without automatic sequencing or PST inputs. This is done using M50 with a P code. For proper operation M50 must be preceded by M36. M36 P1 before the M50 P1 will check that pallet #1 is ready.

Pallets can be changed without automatic sequencing or PST inputs. This is done using M50 with a P code. M50 P1 will load pallet #1 without checking to see if it is scheduled. If the PART READY button has been pressed, then pallet #1 will be loaded. If the PART READY button for pallet #1 has not been pressed, the indicator light on the button will flash and the message, "Schedule Pal#1" is displayed.

Pallet Schedule Table

The pallet schedule table incorporates a number of features to help the user with their routine.

Load Order and Pallet Status These two features work together to show which pallet is currently in the machining area.

Pallet Usage This feature gives the number of times the specific pallet has been loaded into the machining area. The counter will turn over to 0 after 32767 pallet changes.

Program Number This detail shows which program number has been assigned to the pallet.

Program Comment This area displays the comments that are written in the part program.

There are 30 different pallet status values to use. The first four: Unscheduled, Scheduled, Loaded, and Completed, are fixed and cannot be changed. The remaining 26 can be modified and used as needed.

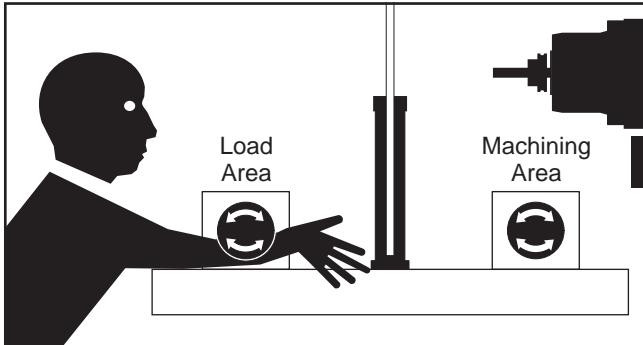
Changing or adding status text can be done in the PST. Use the arrow keys to move the cursor to the "Pallet Status" column, and press the F1 key. A selection menu will appear over the "Pallet Status" column (Pressing F1 again or Reset will close the menu.) The number to the left of the text is the status number. This number is used with the M49 command to set the status from the part program. The items in the menu can be selected with the Up and Down arrow keys, or the jog handle. Enter the text, then press F3. Note: All pallets use the same list of status items. Pressing F1 closes the menu without changing the status of any of the pallets.

Resetting a status item to "User" is done by selecting an item from the menu and pressing F4. All status items can be reset at the same time with the Origin key.

Changing the status of an individual pallet can be done from the PST or with the M49 command. In the PST, move the table cursor to the "Pallet Status" column for the pallet desired. Press F1 for the status item menu. Use the arrow keys to select the status, then press F2 or Write/Enter. See the previous description of M49 and in the following examples for setting the pallet status from a program.



Caution: The following commands can cause either rotary products to move:
Zero Return
Handle Jog



M48 must be placed at the beginning of the program (or section of program) that is run for the current pallet. This will check each time that a program is run, that the program matches the pallet. For example:

```
Oxxxx (Users program)
M48
;
; (Users part program for pallet 1)
;
M30
Oxxxx (Users program)
M48
;
; (Users part program for pallet 2)
;
M30
```

An alarm will be generated, "A (or B) not in Position", if the pallet in the machine is not the one associated with the part program. If this alarm is generated, verify the correct program is running for the loaded pallet.

Important: Verify that the rotary table on pallet one is plugged into "Connector 1", and that the rotary table on pallet two is plugged into "Connector 2".

Sample Programs

Example #1

A basic pallet change program that loads the next scheduled pallet and runs the parts program. The following is a sample of the PST, which indicates that pallet #1 is loaded and pallet #2 is scheduled. Pallet #2 will be loaded next (see column 2, "Load Order") and program O06012 will be used to cut parts on that pallet (see column 5, "Program Number"). The program comment is captured from the program.

Pallet Schedule Sample Table 1

Pallet Number	Load Order	Pallet Status	Pallet Usage	Program Number	Program Comment
1	*	Loaded	23	O04990	(Rough and Finish)
2	1	Scheduled	8	O06012	(Cut Slot)



O00001 (Program Number)
M50 (Perform pallet change to next scheduled pallet, after Part Ready button is pressed)
G188: (Calls the parts program for the loaded pallet based on the PST entry for the pallet.)
M99 (Loop to top of main program)

O04990
Part Program (User's part program)
M99 (Return from subroutine)

O06012
Part program (User's part program)
M49Q12 Set current pallet status to 12 actual string defined by operator. In this case, status number 12 is defined as "Last Pallet".
M99 (Return from subroutine.)

Description: The first loop through program O00001 will load pallet #2 (M50) and run program O06012 (G188 selects program from PST for pallet #2). The PST will then resemble Sample Table 2. The asterisk for pallet #2 in the "Load Order" column indicates that this pallet is in the mill.

Pallet Schedule Sample Table 2

Pallet Number	Load Order	Pallet Status	Pallet Usage	Program Number	Program Comment
1	0	COMPLETED	23	O04990	(Rough and Finish Tog)
2	*	LOADED	9	O06012	(Cut Slot)

Description: In the next loop through program O00001, the M50 will detect that no pallets are scheduled. The beacon light will flash green and program O00001 will pause until the operator schedules a pallet or presses Reset. A pallet can be scheduled by pressing the Part Ready button.

Example #2

Basic pallet change program that tracks which part is to be machined on each pallet. Each pallet has a different machining operation. Note that the P code for M46 is a line number in the current program, not a subroutine number.

Oxxxxx Program number
M50 (Perform pallet change after the Part Ready button is pressed or PST is updated)
M46 Q1 Pxx1 This line will check to see if pallet #1 is on the machine. If it is then it will jump to line xx1. If the pallet is not on the machine, then it will continue to the next line. (See description of M46.)

M46 Q2 Pxx2 (If pallet #2 is loaded, program will jump to line xx2, otherwise it will go to the next line.)
M99 Pxxxx (Jump to line Nxxxx: see the "Mcode" section for a more detailed description of M99)
Nxx1 (Line number)
Part program for Pallet #1 (User's part program for Pallet #1.)

M99 Pxxxx (Jump to line Nxxxx: see the "Mcode" section for a more detailed description of M99)
Nxx2 (Line number)
Part program for Pallet #2 (User's part program for Pallet #2)

M99 Pxxxx (Jump to line Nxxxx)

Nxxxx (Line number)
M99 (Repeat program: see the "Mcode" section for a more detailed description of M99)

Example #3

This is an alternate method to Example #2 that uses subroutine calls, but does not jump if the pallet is unscheduled.



Note: For proper operation M50 with a P code must be preceded by M36.

M36 P1	(Flash "No Pallet Scheduled" on display, flash green beacon, on Schedule Pallet button for pallet #1 until button is pushed or pallet is scheduled in PST)
M50 P1	(Load Pallet #1)
M98 Pxxx1	(Control jumps to program Oxxx1 and runs this program. See the "Mcode" section for a more detailed description of M99)
M36 P2	(Wait for pallet to be scheduled)
M50 P2	(Load Pallet #2)
M98 Pxxx2	(Control jumps to program Oxxx2 and runs this program.)
M99	(Repeat program: see the M-codes section for a more detailed description of M99)

Note: M99 at the end of a program will cause continuous operation. M30 at the end of a program will cause the control to wait for the operator to press Cycle Start.

Pallet Changer Recovery (EC-300, EC-400, MDC)

EC-300 or MDC

If the pallet change is interrupted another M50 must be run; use M50P1 or M50P2. If this puts the wrong pallet into the mill, then an additional M50 must be run.

EC-400

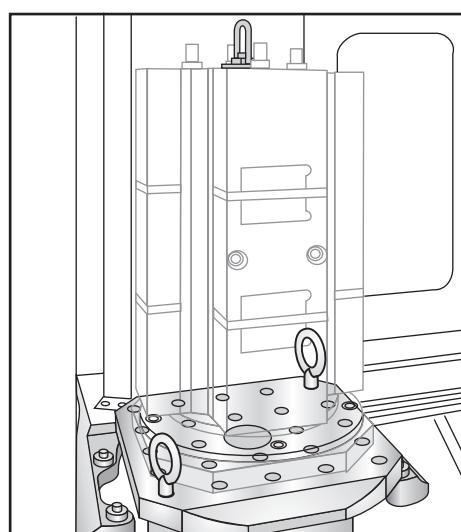
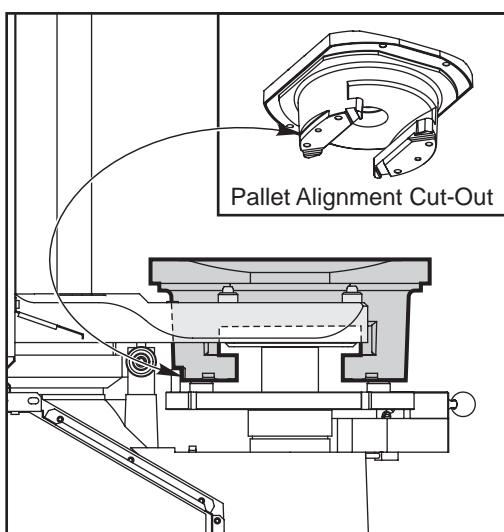
The control has a pallet changer recovery mode to assist the operator if the pallet changer fails to complete a pallet change. To enter the pallet changer recovery mode press "Tool Changer Restore" and then press the specific function key (F2) for pallet changer recovery mode. Note that is the pallet is in the proper position, the pallet changer restore function is not available.

The most convenient way to recover from a failed pallet change attempt is to press "Y" and follow the on-screen help text. A message will prompt the operator to command an individual step of the pallet change sequence. More than one step may need to be completed, as each step is done, press "Y" for the next step. The control will exit the recovery screen once the pallet changer has recovered.

Pallet Replacement (EC-400)

The pallets can only be loaded into the mill through the load station. Note the orientation of the pallet; the pallet can only be loaded one way. A cut-out is machined into the pallet to insure the proper orientation of the pallet.

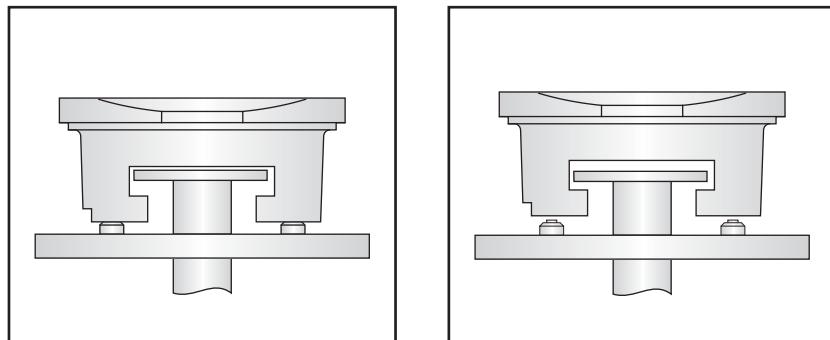
1. Orient the pallet 90 degrees from home in either direction.



2. Attach a suitable lifting device to the top of the fixture(s) or use eye bolts screwed into the pallet holes.



3. Raise the pallet approximately .25" (6.35mm) to position it above the load station pins, but below the load station lock plate.



4. Pull the pallet towards you until it has cleared the load station.

Pallet Storage (EC-400)

When removing the pallet, be sure to set it on a soft surface such as a wooden pallet. The bottom side of the pallet has machined surfaces that must be protected.

TIPS AND TRICKS

General Tips

Cursor Searching for a Program. When in EDIT or MEM mode, you can select and display another program quickly by entering the program number (Onnnnn) and pressing the Up/Down arrow, or **F4**.

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 and value (A1.23), and press the Up/Down arrow. If the address code and no value is entered, the search will stop at the next use of that address code, 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.

Saving an MDI Program. You can save a program from MDI to the list of programs. To do so, make sure the cursor is at the beginning of the MDI program, enter a program number (Onnnnn) and press **ALTER**.

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 to machine zero in rapid motion. Enter the axis letter (e.g. X), then press **HOME G28. CAUTION!** There is no warning to alert you of any possible collision.

Program /Conversation

Program Review. Program Review allows you to cursor through and review the active program on the right side of the display screen, while also viewing the same program as it is running on the left side of the screen. To enter Program Review, press **F4** while running a program in MEM mode and PRGRM display.

Background Edit. This feature allows you to edit a program while a program is running. Press **MEM** and then **PRGRM**, enter the program number to edit and press **F4**. Edits are possible as the program runs, however, edits to the running program will not take effect until the program ends with an **M30** or **RESET**.

Position

DIST-TO-GO Display. To zero the 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 all axes on the DIST-TO-GO display.

POS-OPER Display. This display is used for reference only. Each axis can be zeroed independently. To zero a specific axis, Page up/down in the POSIT display to the large-digit POS-OPER page. Pressing Origin will zero the selected axis. Or press the X, Y, or Z button and then Origin to zero that axis display. To enter a value for an axis, enter the axis letter and a value (e.g. X2.125), and press Origin.



Offset

Entering Offset Values. Pressing **OFSET** again will toggle between the Tool Length Offsets and Work Zero Offsets pages. Pressing Write/Enter will add the number entered to the selected offset value. Pressing F1 will replace the selected offset with the entered number. Pressing F2 will enter the negative value into the offsets.

Coolant Spigot Position. The coolant nozzle position is displayed in the lower left corner of the Tool Geometry Offset page.

Clearing All Offsets and Macro Variables. When you are in the Tool Length Offset display, you can clear all the offsets by pressing the Origin key. This also works on the Work Zero Offset and Macro Variables page.

Current Commands

Programmed (PGM) and Actual (ACT feedrate and spindle speed are displayed on the first page of the CURNT COMDS, as PGM Fn, PGM Sn, ACT Fn and CMD Sn.

Calculator

Transferring Simple Calculations. The number in the simple calculator box (in the upper left corner) can be transferred to any cursor-selected data line by moving the cursor to the line and pressing **F3**.

Transferring to EDIT or MDI. Pressing **F3** will transfer the number in the calculator box (when the cursor is on the number in the box) to either the EDIT or MD mode. Enter the letter (X, Y or Z) you wish to use with the number from the calculator.

Circular Calculator. The Circular Calculator will list four different ways that a circular move can be programmed using the values entered. One of the solutions can be transferred to either EDIT or MDI modes. To do so, cursor onto the program line you wish to use. Press either EDIT or MDI. Press the **F3** key, which will transfer the circular move to the data entry line at the bottom of the display. Press Insert to add that circular command line into your program.

One-Line Expressions. The calculator allows a simple, one-line expression without parentheses, such as $23*45.2+6/2$, to be solved. It will be evaluated when the Write/Enter button. Note: Multiplication and division are performed before addition and subtraction.

Edit

Lower-Case Text. Lower-case text can be used if it is between parentheses (comments); use the Shift key.

Exiting Block Edit. Block Edit highlighting is cancelled by pressing the Undo; the cursor will remain where you are at in the program. Undo **will not** change back an edit done in Block Edit. **RESET** will also turn off the block highlight, but the cursor will return to the beginning of the program.

Advanced Editor Searching. When "Find Text" is used and the text is found, pressing F1 will automatically select the Find Again option. Also, when "Select Text" is used, "Copy Selected Text" will be the next menu selection.

Editing Two Versions of the Same Program. In Edit mode, **F4** is the hot key that displays another version of the active program for editing. The same program will be displayed on both halves of the screen, and each program can be edited by using the **EDIT** key to switch from one side to the other. Both programs will be updated as the edits are done. This is useful for editing a long program; you can view and edit one section of the program on one side of the screen and another section on the other side.

A Quick Cursor Arrow in the Advanced Editor. A cursor arrow can be called up within the program. To do this, Enter the Advanced Editor, press **F2** and use the jog handle to scroll through the program. To exit from this mode, and remain at that position in the program, press the **UNDO** key.

Programming

Quick Reverse Out of a G84 Rigid Tapping Cycle. This rigid tapping feature has the tap exit faster than it entered. A J code on the G84 line commands this, for example, J2 retracts twice as fast, J3 retracts three times as fast, etc. up to J9. The J code must be specified in each block.



Duplicating a Program in LIST PROG. In the List Prog mode, a program can be duplicated by selecting the program number, typing in a new program number (Onnnnn), and pressing **F1**. The Advanced Editor can also be used to duplicate a program, using the Program menu and the Duplicate Active Program selection.

Communications

Receiving Program Files from a Floppy Disk. Program files can be loaded from a floppy disc using the I/O menu and the Floppy Directory selection of the Advanced Editor. Pressing Write/Enter in the Floppy Directory will display a list of the programs. Use the arrow keys or the jog handle to select the file to load, then press Write/Enter. The floppy directory will remain displayed to the selection of more files. Use Reset or Undo to exit.

I/O Menu Send RS232 or Send Floppy Commands. Programs can be sent to the RS232 port or a floppy disk from the Advanced Editor. Select the menu item (Send RS232 or Send Floppy), and a program list is displayed. Choose the program to save, or select "All" to send all programs under one file name. Use the up and down arrow keys or the jog handle, and the Insert button to mark the programs to send. If programs are not selected using the Insert key, the currently highlighted program will be sent.

Sending Multiple Programs Using Program Numbers. The Send Floppy under the I/O menu of the Advanced Editor allows programs to be saved to floppy disk. It will prompt, "Enter Floppy File Name"; enter a file name and press Write/Enter. If a file name is not entered, the control will save each program using the five-digit program number as the file name.

Sending a Program File from LIST PROG Display. Files can be sent to a floppy disk or through the RS-232 port from the LIST PROG display. Use the cursor arrows and the Insert key to select the program(s), or "ALL" to send all of them under one file name. When F2 is pressed (to send the selected programs), the control will ask for a file name (8CHRCTRS.3XT). Press F2 a second time to send it. You can also use the I/O menu in the Advanced Editor to send and receive program files.

Sending Multiple Programs from LIST PROG Using SEND RS232. Several programs can be sent to the serial port by typing all the program names together on the input line without spaces (e.g., O12345O98765) and pressing SEND RS232.

Send and Receive Offsets, Settings, Parameters and Macro Variables to/from Disk. Offsets, settings, parameters and macro variables can be saved to a floppy disk. Press List Prog, and select a display page (e.g. OFSET, SETNG). Type in a file name and press F2 (or F3 to read that file from a disk).

Send and Receive Offsets, Settings, Parameters and Macro Variables to/from RS232. Offsets, settings, parameters and macro variables can be saved to the RS-232 port. Press List Prog, and select a display page (e.g. OFSET, SETNG). Enter a file name and press Send RS232 to send that display page to the RS-232 port. Press RECV RS232 to read the file via RS-232.

Deleting a Program File from a Floppy Disk. A file can be erased from the floppy drive from the LIST PROG display. Type "DEL <file name>" and press Write Enter.



SUBROUTINES

Subroutines (subprograms) are usually a series of commands that are repeated several times in a program. Instead of repeating the commands many times in the main program, subroutines are written in a separate program. The main program then has a single command that "calls" the subroutine program. A subroutine is called using M97 or M98 and a P address. The P code is the same as the program number (Onnnnn) number of the subroutine to be called.

Canned Cycles are the most common use of subroutines. The X and Y locations of the holes are placed in a separate program and then called. Instead of writing the X, Y locations once for each tool, the X, Y locations are written once for any number of tools.

The subroutines can include an L or repeat count. If there is an L, the subroutine call is repeated that number of times before the main program continues with the next block.

LOCAL SUBROUTINES

A local subroutine is a block of code in the main program that is referenced several times by the main program. Local subroutines are commanded (called) using an M97 and a Pnnnnn which refers it to the N line number of the local subroutine.

The local subroutine format is to end the main program with an M30 then enter the local subroutines after the M30. Each subroutine must have an N line number at the start and a M99 at the end that will send the program back to the next line in the main program.

Subroutine Example

O00104 (sub program with an M98)

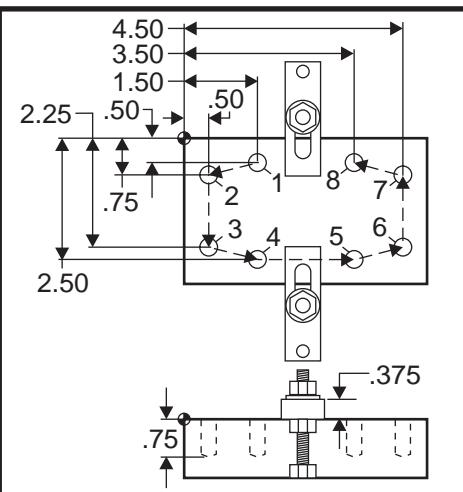
T1 M06
G90 G54 G00 X1.5 Y-0.5
S1406 M03
G43 H01 Z1. M08
G81 G99 Z-0.26 R0.1 F7.
M98 P105 (Call Sub-Program O00105)

T2 M06
G90 G54 G00 X1.5 Y-0.5
S2082 M03
G43 H02 Z1. M08
G83 G99 Z-.75 Q0.2 R0.1 F12.5
M98 P105 (Call Sub-Program O00105)

T3 M06
G90 G54 G00 X1.5 Y-0.5
S750
G43 H03 Z1. M08
G84 G99 Z-.6 R0.1 F37.5
M98 P105 (Call Sub-Program O00105)
G53 G49 Y0.
M30 (End Program)

Sub Program

O00105
X.5 Y-.75
Y-2.25
G98 X1.5 Y-2.5
G99 X3.5
X4.5 Y-2.25
Y-.75
X3.5 Y-.5
G80 G00 Z1.0 M09
G53 G49 Z0. M05
M99





SUBROUTINE CANNED CYCLE EXAMPLE

```
O1234 (Canned Cycle Example Program)
T1 M06
G90 G54 G00 X.565 Y-1.875 S1275 M03
G43 H01 Z.1 M08
G82 Z-.175 P.03 R.1 F10.
M98 P1000
G80 G00 Z1.0 M09
T2 M06
G00 G90 G54 X.565 Y-1.875 S2500 M03
G43 H02 Z.1 M08
G83 Z-.720 Q.175 R.1 F15.
M98 P1000
G00 G80 Z1.0 M09
T3 M06
G00 G90 G54 X.565 Y-1.875 S900 M03
G43 H03 Z.2 M08
G84 Z-.600 R.2 F56.25
M98 P1000
G80 G00 Z1.0 M09
G28 G91 Y0 Z0
M30
```

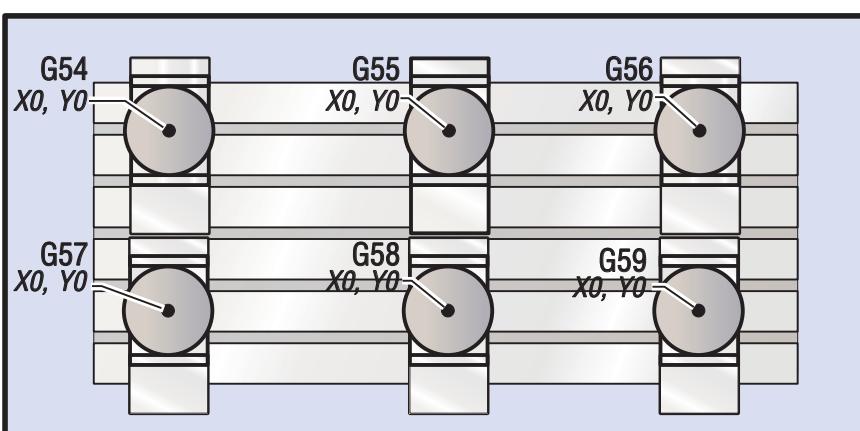
Sub Program
O1000 (X,Y Locations)
X 1.115 Y-2.750
X 3.365 Y-2.875
X 4.188 Y-3.313
X 5.0 Y-4.0
M99

SUBROUTINES WITH MULTIPLE FIXTURES

Subroutines can also be useful when cutting the same part in different X and Y locations within the machine. For example, there are six vises mounted on the table. Each of these vises will use a new X, Y zero. They will be referenced in the program using the G54 through G59 work offsets. Use an edge finder or an indicator to establish the zero point on each part. Use the part zero set key in the work coordinate offset page to record each X, Y location. Once the X, Y zero position for each workpiece is in the offset page, the programming can begin.

The figure shows what this setup would look like on the machine table. For an example, each of these six parts will need to be drilled at the center, X and Y zero.

Main Program	Subroutine
O2000	O3000
T1 M06	X0 Y0
G00 G90 G54 X0 Y0 S1500 M03	G83 Z-1.0 Q.2 R.1 F15.
G43 H01 Z.1 M08	G00 G80 Z.2
M98 P3000	M99
G55	
M98 P3000	
G56	
M98 P3000	
G57	
M98 P3000	
G58	
M98 P3000	
G59	
M98 P3000	
G00 Z1.0 M09	
G28 G91 Y0 Z0	
M30	





ADVANCED EDITOR

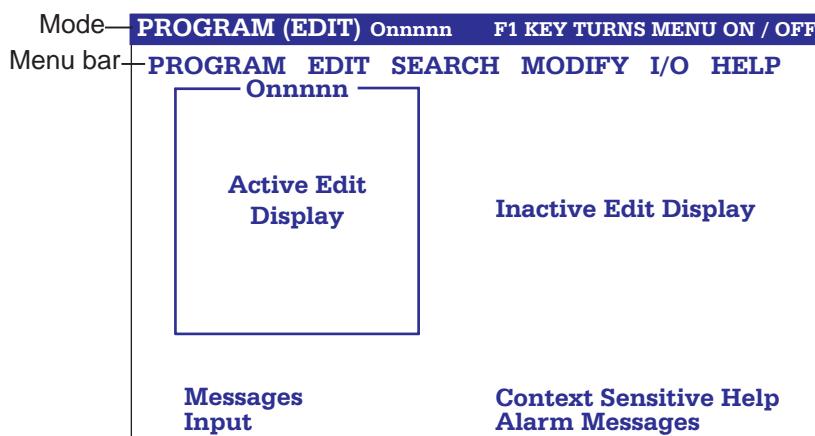
The HAAS Advanced Editor gives the user the ability to edit programs using pull-down menus.

Pressing the EDIT button enters the Advanced Editor. The user can switch between the Advanced Editor, the standard editor, and Quick Code with multiple presses of the PRGRM/CONVRS button.

To edit a program, enter the program name (Onnnnn) and press F4, the program will open in the active window. Pressing the F4 button will open another copy of that program. Use the jog handle or the down/up arrows buttons to scroll through the program code.

Use the EDIT button to select between the two programs.

The following figure illustrates the layout of the advanced editor.



The Advanced Editor Screen Layout.

THE PROGRAM MENU

Create New Program

This menu item will create a new program. To do so, enter a program name (Onnnnn) (that is not already in the program directory) and press Enter to create the program.

Select Program From List

Choose this menu item to edit a program that exists in the directory.

When this menu item is selected, the programs in the control are displayed. Scroll through the list by using the cursor buttons or the jog handle. Pressing Enter or Select Prog will select the highlighted program to replace the program list with the selected program.

Duplicate Active Program

This selection will copy the current program. The user will be prompted to enter a program (Onnnnn) number for the duplicate program.

Delete Program From List

This menu item will delete a program from the program directory.

Switch To Left Or Right Side

This will switch the active window between the two programs, to make the active program inactive and the inactive program active.



THE EDIT MENU

Undo

The last edit operation will be undone, up to the last 9 editing operations.

Select Text

This menu item will select lines of program code to set the start point of the text selection. Then use the arrow buttons or the jog handle to scroll to the last line of code to select and Press F2 or Write/Enter button. The selected text will be highlighted. To deselect the block, press Undo.

Move Selected Text

This feature works with the "Select Text" feature. Scroll the cursor arrow to desired part of code and press the Write/Enter button to move the selected text to its new location. The selected text will be moved to the point following the cursor (>).

Copy Selected Text

To select text, scroll the cursor arrow (>) to a portion of text and press the Write/Enter button. The text copied will be highlighted. Scroll the cursor arrow to the part of text you want to insert the copied text into. Press F2 or Write/Enter to insert the copied text to the point following the cursor (>).

Delete Selected Text

To select text, scroll the cursor arrow (>) to a portion of text and press the Write/Enter button. The text copied will be highlighted. Once highlighted, press the Write/Enter button to delete the text. If no block is selected, the currently highlighted item is deleted.

Cut Selection To Clipboard

All selected text will be moved from the current program to a new program called the clipboard. Any previous contents of the clipboard are deleted

Copy Selection To Clipboard

All selected text will be copied from the current program to a new program called the clipboard. Any previous contents of the clipboard are deleted.

Paste From Clipboard

The contents of the clipboard are copied into the current program at the line following the current cursor position.

THE SEARCH MENU

Find Text

This menu item will search for text or program code in the current program.

Find Again

This menu item will search again for the same program code or text.

Find And Replace Text

This menu item will search the current program for specific text or program and optionally replace each (or all) with another G-Code item.

THE MODIFY MENU

Remove All Line Numbers

This menu item will automatically remove all unreferenced N-Codes (line numbers) from the edited program. If a group of lines is selected only these lines will be affected.

Renumber All Lines

This menu item will either renumber all selected blocks in the program or, if a group of lines is selected the renumber feature will affect those lines only.



Renumber By Tool

Searches for T (tool) codes, highlights all the program code up to the next T code and renames the N code (line numbers) in the program code.

Reverse + & - Signs

This menu item will reverse the signs of the numeric values. Press the enter key to start the process and then enter the axes (e.g. X, Y, Z etc.) that is to change. When using this feature be cautious if your program contains a G10 or G92 (See the G Code section for a description).

Reverse X & Y

This feature will change X address codes in the program to Y address codes and Ys to Xs.

THE I/O MENU

Send RS-232

This menu item will send program(s) to the RS-232 port. When this menu item is selected, the program list is displayed.

To select a program, cursor to the program number and press the Insert button. A highlighted space will appear before the program to indicate it has been selected. (Press Insert again to deselect the program). The Delete button can be used to deselect all selected programs.

To send the selected program(s), press the Write/Enter button. If more than one program or "All" is selected, the data will be sent with a "%" at the beginning of the stream and a "%" at the end.

Receive RS-232

This menu item will receive program(s) from the RS-232 serial port.

On the List Prog display "ALL" must first be highlighted before using this menu item. Note, "ALL" must be reselected on the List Prog screen after each file transfer.

Send Disk

This menu item will send program(s) to the floppy disk. When this menu item is selected the program list is displayed.

To select a program, cursor to the program number and press the Insert button (or enter a file name, Onnnnn, and press the Write/Enter button). A highlighted space will appear before the program to indicate it has been selected (Press Insert again to deselect the program). The Delete button can be used to deselect all selected programs.

Receive Disk

This menu item will receive programs from the floppy disk. Type in the filename (Text, e.g. JOB5.NC, or Onnnnn) of the disk file being received, and press the Enter button.

Disk Directory

This menu item will display the directory of the floppy disk. To select a file, press the up and down arrow buttons or use the jog handle to scroll through the directory list and press Write/Enter to load a file.

THE F1: HELP MENU

How To Use The Editor

The help is displayed whenever a menu is accessed. The help menu gives a brief description of the editor and its features. The up and down arrows and the jog handle control the menus, and the Page Up, Page Down Home and End buttons are used to scroll through the help display. In addition, if the F1 button is pressed during the use of one of the menu options, the help is likewise displayed. Pressing F1 again will exit the help display. Pressing the Undo button returns to the active program.



ADVANCED EDITOR SHORTCUTS

Pressing these buttons, when in the Advanced Editor display, will quickly get you to these menu items without having to press the F1 button and cursoring to that selection.

Hot Keys



Description

Will quickly bring up the program list on the inactive side of edit display to select program from list.



This key will begin to select text and define the starting line of a block to be edited. Scroll down to the last line in the block definition, and press the F2 or WRITE key. The selected block of text will then be highlighted.

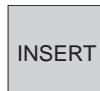


This key can be used to switch to the left or right side between two programs that have been selected to edit.



Pressing F4 will open another copy of the same program on the other side of the Advanced Editor display. The user can quickly edit two different locations in the same program. The edit key will switch you back and forth and update between the two programs.

If you enter the program number (Onnnn) and then press F4 or the arrow down key, that program will be brought up on the other side of the Advanced Editor.



INSERT can be used to copy selected text in a program to the line after where you place the cursor arrow point.



ALTER can be used move selected text in a program to the line after where you place the cursor arrow point.



DELETE can be used to delete selected text in a program.



If a block has been selected, pressing undo will simply exit a block definition.



Pressing the Send key will activate that I/O menu selection.



Pressing Recv key will activate that I/O menu selection.

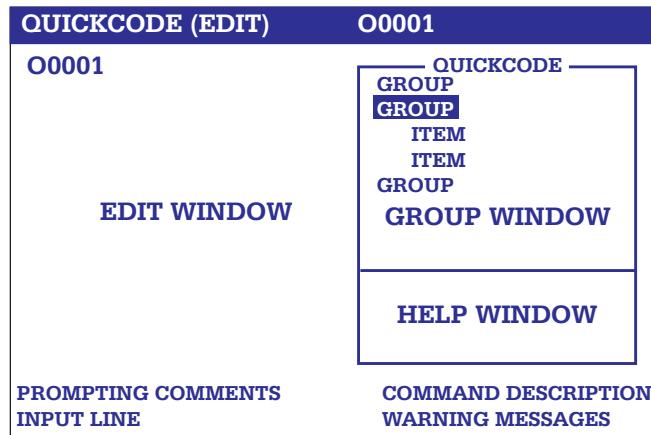


Pressing the Erase Prog key will activate that I/O menu selection. This will bring up program list on the inactive side of edit display for you to cursor to a program and delete it.



QUICK CODE

Quick Code simplifies writing a program by explaining G code commands in plain English commands. On the right side of the screen you have English commands that describe the operation to perform. By selecting the operation from the Group Window feature and with one button push, the code is inserted in your program on the left side of the screen. The groups can be selected by turning the jog handle in the clockwise direction. To view and cursor through items within a group, turn the jog handle in the counter clockwise direction. Another feature is the ability to cursor through a program and Quick Code will describe what the G and M codes mean, shown at the bottom of the screen.



The Quick Code Display.

Accessing Quick Code

Enter Quick Code by selecting Edit mode and then pressing the Prgrm/Convrs key twice. The first press of the Prgrm/Convrs key enters the standard editor, the second press of this key will enter the Quick Code feature. Each additional press of this key will switch between Visual Quick Code, Advanced Editor, the standard editor, and Quick Code modes.

The Edit Window

Each time that you select a group item, as described in the next section, the edit window will update to show you what code has been added to the currently edited program. All the edit functions with the exception of the jog handle and the block copy function keys. In Quick Code, the jog handle is used to maneuver through the group list. You can cursor through the program text by using the cursor keys provided on the center of the keypad. Switch to standard edit mode by pressing the Prgrm/Convrs key to access the jog handle (for long comments) and the block copy functions. Quick Code is not available while in BACKGROUND EDIT MODE (See the Background Edit Section in Operation Chapter).

The Group Window

The group window displays a list of groups that are available in Quick Code.

The Help Window

The help window is just below the group window. It is used to display Quick Code help messages and warnings and program examples.

A Sample Quick Code Session

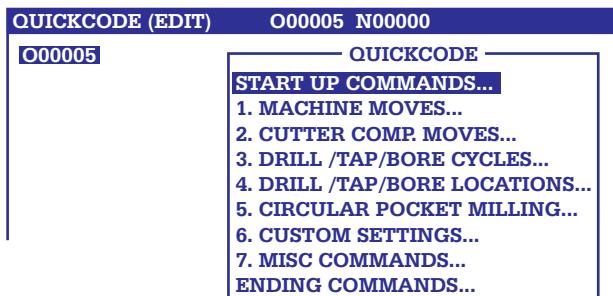
The following illustrates how Quick Code can be used to build a program. A program will be built to spot, drill, and tap 5 holes on a circular bolhole pattern. We will assume that tool 1 is a spot drill, tool 2 is a drill for a 10-32 tap, and tool 3 is the tap. Verify that Quick Code source program, O9999, is one of the programs in the control, before starting.

The jog handle is an integral part of using Quick Code and is used quite often.



Create A Program

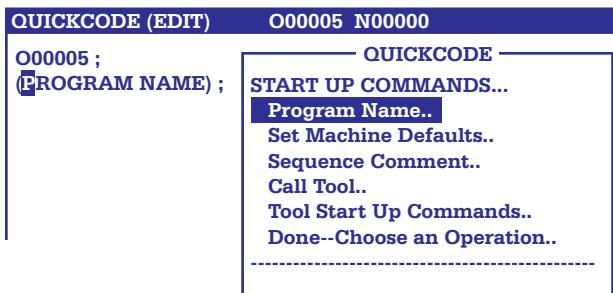
Quick Code will not generate the new program number for you. To create a program, press List Prog, enter a program number (e.g. O00005) and press Write/Enter. Or select a program to edit. In order to edit the program press the Edit button and then press the Prgrm/Convrs key twice to enter Quick Code (**Note that in the advanced edit menus under the HELP menu is a sub menu selection for Quick Code.**



Select The Start Up Commands

1. Turn Jog Handle clockwise (CW) until the group titled "Start Up Commands" in the group window is highlighted.
2. Turn the Jog Handle counter clockwise (CCW) one click. The items belonging to Start Up Commands will appear and the item, "Program Name" is the one highlighted.
3. Press the Write key. This will enter in a (T), for you to cursor arrow left twice onto the "T" in-between the parenthesis, then type in a program name and press Alter.

The following figure shows what the screen with a program name entered looks like.



Start a Program Using Quick Code.



Call Tool 1

1. While on the “Start Up Commands” menu, turn the jog handle CCW to highlight the group item titled “Call Tool.”
2. Press the Write button to have the control ask you for a tool number for your program, and the control will be flashing with a 1 in the lower left corner as the default value. Press Write to accept the number 1.
3. Highlight the group item titled “Tool Start Up Commands”.
4. Press the Write key to have the control ask you for the commands to define the start up of tool 1, and enter into your program.

```
QUICKCODE (EDIT)      O00005 N00000
O00005;
(PROGRAM NAME);
T1 M06 (T);
G90 G54 G00 X0 Y0;
S750 M03;
G43 H01 Z1. M08;
```

```
QUICKCODE
START UP COMMANDS...
Program Name..
Set Machine Defaults..
Sequence Comment..
Call Tool..
Tool Start Up Commands..
Done--Choose an Operation..

-----
```

1. MACHINE MOVES...

2. CUTTER COMP. MOVES...

3. DRILL /TAP/BORE CYCLES...

Programmed with the Start Up Command Selections Entered in with Quick Code for Tool 1.

For this example, the material is aluminum and that the work coordinate zero for G54 is at the center of the bolthole pattern.

Invoke the Spot Drilling Canned Cycle G82

1. Scroll and highlight the group titled “4. Drill/Tap/Bore Cycles”.
2. Scroll CCW two clicks. “Drill with Dwell G82” will be highlighted.
3. Press the Write button to start the prompts.

Note that Quick Code defined a block of code to execute a spot drill cycle at that present location. More X and Y drill cycle locations can be added, if needed, by selecting “6. Drill/Tap/Bore Locations”.

Note: The part does not have a hole drilled at X0 Y0, which is the center of the bolthole circle; manually enter an L0 on the end of the G82 command line. This will ignore the G82 canned cycle until the next location.

The program will look like this:

```
QUICKCODE (EDIT)      O00005 N00000
O00005;
(PROGRAM NAME);
T1 M06 (T);
G90 G54 G00 X0 Y0;
S750 M03;
G43 H01 Z1. M08;
G82 G99 Z-0.109 P0.2 R0.1 F5;
```

```
QUICKCODE
3. DRILL/TAP/BORE CYCLES...
Drill G81..
Drill with Dwell G82..
Deep Hole Peck Drill G83..
High Speed Peck Drill G73..
H.S.P.D. W/Return R plane G73..
Bore IN Bore OUT G85..
Bore IN Rapid OUT G86..
Bore IN Shift Rapid OUT G76..
Right Hand Tapping G84..
G80 CANCEL Canned Cycle..
```

Program with Spot Drilling Invoked.



Execute a Circular Bolt Hole Pattern

Scroll CW and highlight the group titled “6. Drill/Tap/Bore Locations”. Scroll CCW and highlight the group titled “Bolt Hole Circle Locations”. Press the Write button and the control will ask for the code on positioning around a bolthole circle. Enter in the numbers to answer all the questions.

Example:

QUICKCODE (EDIT)	O00005 N00000
O00005 ; (PROGRAM NAME) ; T1 M06 (T) ; G90 G54 G00 X0 Y0 ; S750 M03 ; G43 H01 Z1. M08 ; G82 G99 Z-0.109 P0.2 R0.1 F5. ; G70 X0 Y0 I2.5 J30. L5 ;	4. DRILL/TAP/BORE LOCATIONS... X Location.. Y Location.. X & Y Location.. A Location.. Initial Point or R Plane Return.. Bolt Hole Circle Locations.. Bolt Hole Arc Locations.. Bolt Holes At Angle Locations.. More Bolt Hole Pattern Help.. G80 CANCEL Canned Cycle..

Program with a Bolt-hole Circle Added.

The remaining selections are similar to what has been done.

Call Tool 2

1. Scroll CW to the group titled “Start Up Commands” then scroll CCW and highlight the group item titled “Call Tool”. Press the Write button and the control will ask for a tool; enter in the number 2.

2. Scroll CCW and highlight the group item titled “Tool Start Up Commands”. Press the Write key and the control will ask for information necessary to use tool 2 in the program.

Call the Spot Drilling Canned Cycle G83

Scroll CW and highlight the group titled “4. Drill/Tap Cycles”. Scroll CCW until “Deep Hole Peck Drill G83” is highlighted. Press the Write button and the control will ask for the information to drill with G83.

Note that Quick Code defined a block to execute a spot drill cycle at that present location. More X and Y drill cycle locations can be added, if needed, by selecting “6. Drill/Tap/Bore Locations”.

Note: Manually enter an L0 on the end of the G82 command line to stop the mill from drilling a hole at the center of the bolt hole circle. This will ignore the G82 canned cycle until the next location.

Execute a Circular Bolt Hole Pattern

Scroll CW and highlight the group titled “6. Drill/Tap/Bore Locations”. Scroll CCW and highlight the group titled “Bolt Hole Circle Locations”. Press the Write button and the control will ask for the information to position around a bolt hole circle. Enter in the numbers to answer all the questions in the lower left corner of the control screen, to define all the commands necessary to position around to drill a Bolt Hole Circle with a G83 canned cycle.

Call Tool 3

1. Scroll CW to the group titled “ Start Up Commands” then scroll CCW and highlight the group item titled Call Tool. Press the Write button and the control will ask for a tool to enter into number 3.

2. Scroll CCW and highlight the group item titled “Tool Start Up Commands”. Press the Write button and the control will ask for the information necessary to use tool 3 in the program.



Call the Tapping Canned Cycle G84

Scroll CW and highlight the group titled 4. Drill/Tap Cycles. Scroll CCW until Right Hand Tapping G84 is highlighted. Press the Write key and the control will ask for the information to set up tapping with G84.

Note that Quick Code defined a block to execute a tapping cycle at that present location. Enter the appropriate speed and feed for this tapping cycle. More X and Y tap cycle locations can be added, if needed, by selecting "6. Drill/Tap/Bore Locations".

Note: Manually enter an L0 at the end of the G84 command line so the mill will not tap a hole at X0 Y0. This will ignore the G84 canned cycle until the next location.

Execute a Circular Bolt Hole Pattern

1. Scroll CW and highlight the group titled "6. Drill/Tap/Bore Locations". Scroll CCW and highlight the group titled "Bolt Hole Circle Locations". Press the Write button and the control will ask for the information to position around a bolthole circle.
2. Answer the questions in the lower left corner of the control screen, this will define the commands necessary to tap a bolt hole circle with a G84 canned cycle.

To move the table forward to ease the removal of the part use the following instructions:

Rapid Z-axis / Coolant Off

Scroll CW and highlight the group titled "Ending Commands". Scroll CCW until the "Rapid Z Axis Coolant Off" is highlighted and press Write.

The control will ask for the location to rapid the Z axis to..

Sending the Machine Home and Ending the Program

1. Scroll CW and highlight the group titled "Ending Commands". Scroll CCW until "Home Y and Z Axes" is highlighted and press Write.
2. Scroll CCW and highlight the group titled "End Program, Reset M30" and press Write.

The program is ready to run. Run the program in graphics to make certain that all the steps are entered.

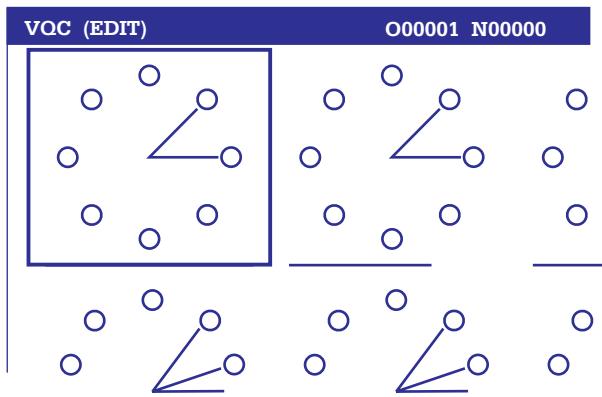


VISUAL QUICK CODE

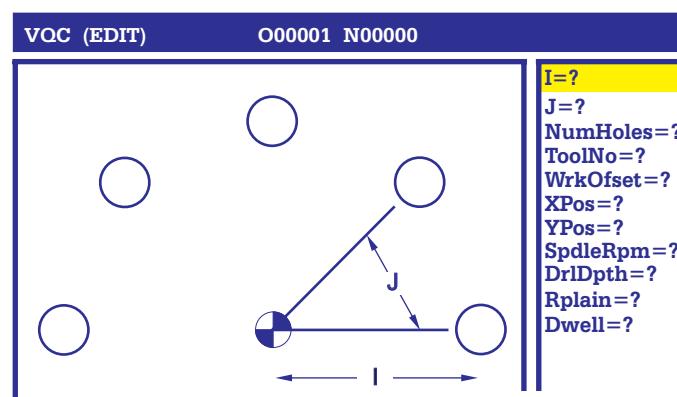
To start Visual Quick Code (VQC), enter Edit mode then press the PRGRM/CONVRS key three times. Another way is to enter VQC is to use the pull down menus in the Advanced Editor under HELP.

Selecting a Category

Use the arrow keys to select the parts category whose description closely matches the desired part and press Write. A set of illustrations of the parts in that category will appear.



VQC Bolt-Hole Circle Category



Bolt-Hole Circle Data Entry Screen

Selecting a Part Template

Use the arrow keys to select a template on the page. Pressing Write will display an outline of the part and wait for the programmer to enter values to make the selected part.

Entering the Data

The control will prompt the programmer for information about the selected part. Once the information is entered, the control another asks the user where the G-code is to be placed:

1) Select/Create a Program

A window will open prompting the user to select a program name. Highlight the desired name and press Write. This will add the new lines of code to the selected program. If the program already contains code, VQC enter the lines of code to the beginning of the program, before the existing code. The user also has the option to Create a new program by entering a program name and by pressing Write, this will add the lines of code to the new program.

2) Add to Current Program – The code generated by VQC will be added after the cursor.

3) MDI – The code will output to MDI. Note: Anything in MDI will be overwritten.

4) Cancel – The window will close and the program values will be displayed.

Note: The program is also available for editing and viewing in the Advanced Editor. It is a good idea to check the program by running it in Graphics mode.



CUTTER COMPENSATION

Cutter compensation shifts the programmed tool path so that the centerline of the tool is moved to the left or right of the programmed path. The OFFSET (Length and Radius) page is used to enter the amount that the tool is shifted. The offset is entered as either a diameter/radius value (see setting 40) for both the geometry and wear values. Note that if diameter is specified, the cutter compensation shift amount is half of the value entered. The compensated value is calculated by the control from the values entered in the Radius (radius of the tool) and the Wear (wear of the tool) values in the offsets page. In 2D machining cutter compensation is used in the X-axis and the Y-axis (G17) and for 3D machining, cutter compensation is used in the X-axis, Y-axis and Z-axis (G141).

G41 will select cutter compensation left; that is, the tool is moved to the left of the programmed path.

G42 will select cutter compensation right.

G40 will cancel cutter compensation.

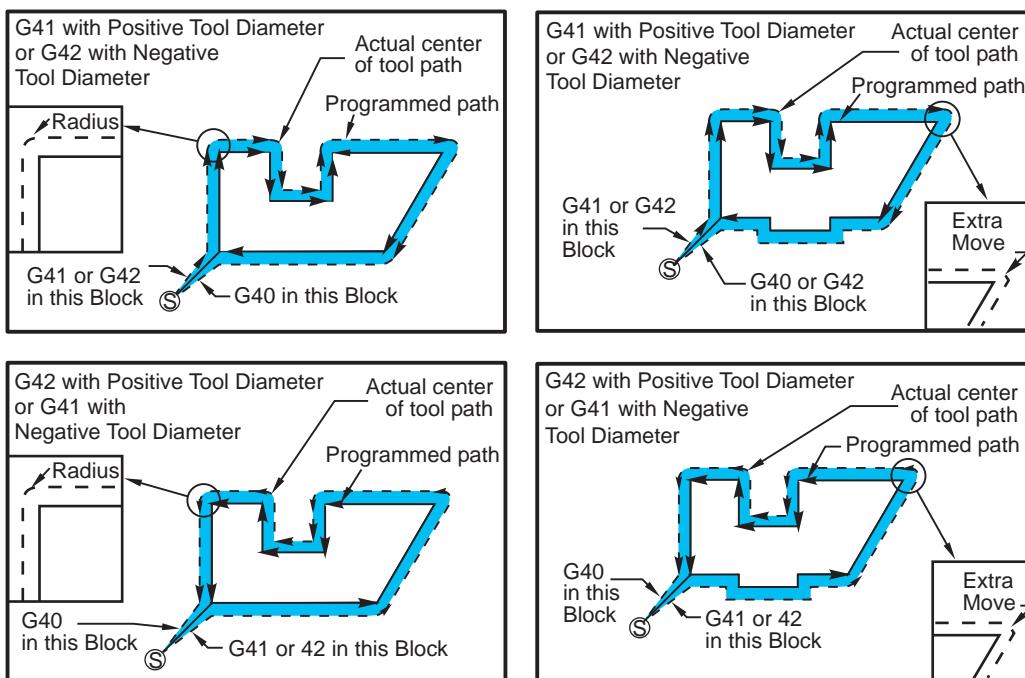
A **Dnnn** must also be programmed with G41 or G42 to select the correct offset number from the radius/diameter offset column. Offset values entered for the radius/diameter should be in positive numbers. If the offset contains a negative value, cutter compensation will operate as though the opposite G code was specified. For example, a negative value entered for a G41 will behave as if a positive value was entered for G42.

Selecting Yasnac for Setting 58, the control must be able to position the side of the tool along all of the edges of the programmed contour without over cutting the next two motions. A circular motion joins all of the outside angles.

Selecting Fanuc for Setting 58, the control does not require that the tool cutting edge be placed along all edges of the programmed contour, preventing over-cutting. Outside angles less than or equal to 270 degrees are joined by a sharp corner and outside angles of more than 270 degrees are joined by an extra linear motion (See the following diagrams).

The following diagrams show how cutter compensation works for the two values of Setting 58.

Note: When canceled, the programmed path returns to being the same as the center of the cutter path. Cancel cutter comp (G40) before ending a program.



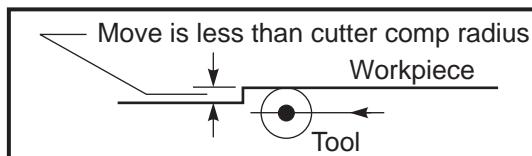


ENTRY AND EXIT FROM CUTTER COMPENSATION

Cutting should not be performed when entering and exiting cutter compensation or when changing from left side to right side compensation. When cutter compensation is turned on, the starting position of the move is the same as the programmed position, but the ending position will be offset, to either the left or right of the programmed path, by the amount entered in the, radius/diameter offset column. In the block that turns off cutter compensation, the starting point is offset and the ending point is not offset. Similarly, when changing from left to right or right to left side compensation, the starting point of the move needed to change cutter compensation direction will be offset to one side of the programmed path and end at a point that is offset to the opposite side of the programmed path. The result of all this is that the tool moves through a path that may not be the same as the intended path or direction. If cutter compensation is turned on or off in a block without any X-Y move, there is no change made to cutter compensation until the next X or Y move is encountered.

An offset selection of D0 will use zero as the offset value and have the same effect as not using cutter compensation. If a new D value is selected with cutter compensation active, the new value will take effect at the end of the proceeding move. You cannot change the D value or change sides during a circular motion block (G02 or G03).

When turning on cutter compensation in a move that is followed by a second move at an angle of less than 90 degrees, there are two ways of computing the first motion these are: cutter compensation type A and type B (Setting 43). The first, type A, moves the tool directly to the offset start point for the second cut. The second, type B, is used to clear fixtures or clamps or in cases when part geometry demands it. The diagrams on the following pages illustrate the differences between type A and type B for both Fanuc and Yasnac settings (Setting 58).



Improper Cutter Compensation Application

Note that a small cut of less than tool radius and at a right angle to the previous motion will only work with the Fanuc setting. A cutter compensation alarm will be generated if the machine is set to the Yasnac setting.

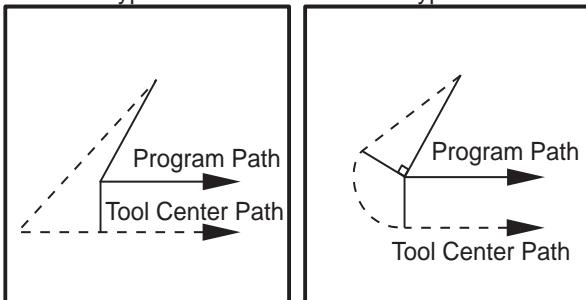
FEED ADJUSTMENTS IN CUTTER COMPENSATION

When using cutter compensation in circular moves, there is the possibility of speed adjustments to what has been programmed. If the intended finish cut is on the inside of a circular motion, the tool should be slowed down to ensure that the surface feed does not exceed what was intended. There are problems when the speed is slowed by too much, therefore, setting 44 is used to limit the amount of feed adjustment. It can be set between 1% and 100%. If set to 100%, there will be no speed changes. If set to 1% the speed can be slowed to 1% of the programmed feed.

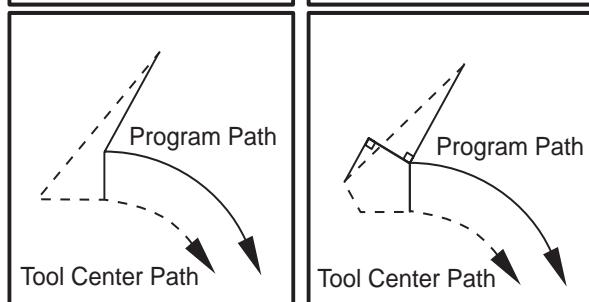
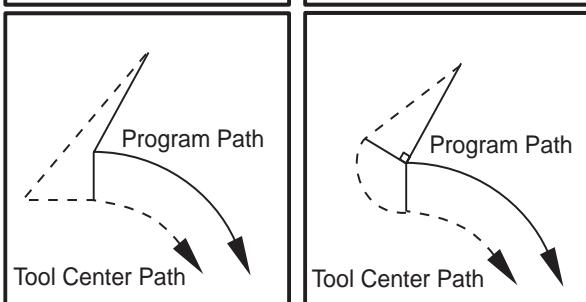
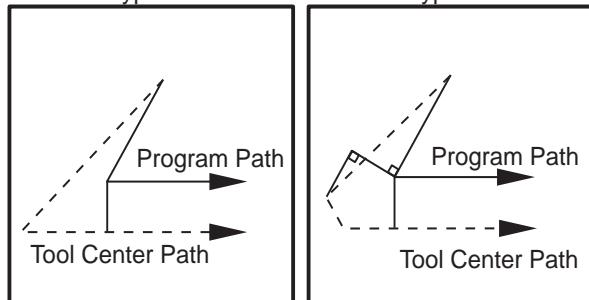
When the cut is on the outside of a circular motion, there is no speed-up adjustment made to the rate feed.



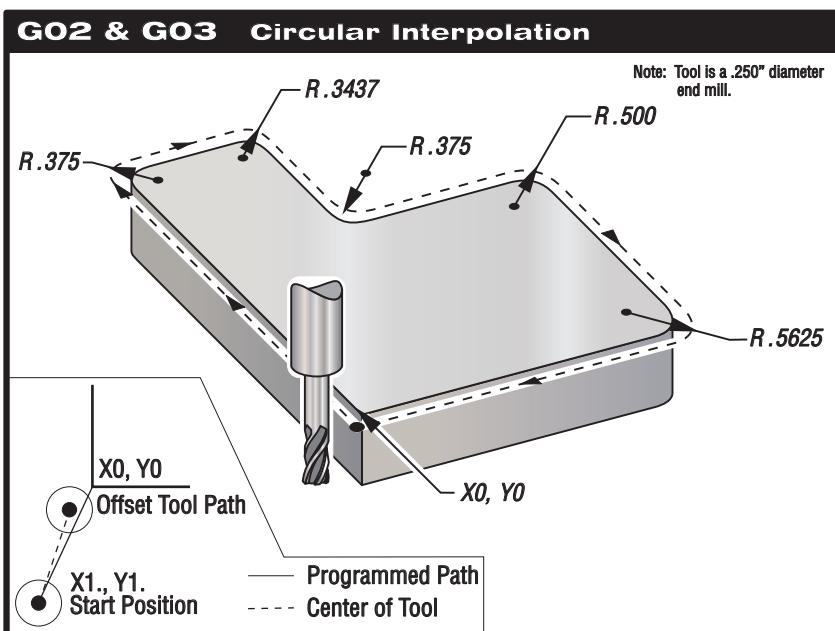
Cutter Compensation Entry (YASNAC) Type A Type B



Cutter Compensation Entry (Fanuc style) Type A Type B



The following illustration shows how the tool path is calculated for the cutter compensation. The detail section shows the tool in the starting position and then in the offset position as the cutter reaches the workpiece.



%
O6100
T1 M06
G00 G90 G54 X-1. Y-1. S5000 M03
G43 H01 Z.1 M08
G01 Z-1.0 F50.
G41 G01 X0 Y0 D1. F50.
Y4.125
G02 X.250 Y4.375 R.375
G01 X1.6562
G02 X2.0 Y4.0313 R.3437
G01 Y3.125
G03 X2.375 Y2.750 R.375
G01 X3.5
G02 X4.0 Y2.25 R.5
G01 Y.4375
G02 X3.4375 Y-.125 R.5625
G01 X-.125
G40 X-1. Y-1.
G00 Z1.0 M09
G28 G91 Y0 Z0
M30
%

Programming exercise showing tool path.

The following program uses cutter compensation. Tool path is programmed to centerline of the cutter. This is also the way the control calculates for cutter compensation.



INTUITIVE PROGRAMMING SYSTEM (IPS)

INTRODUCTION

The optional Intuitive Programming System (IPS) software simplifies the development of full CNC programs.

To enter the IPS menu press handle jog. Navigate through the menus, using the left and right arrow keys. To select the menu press Write/Enter. Some menus have sub-menus, which again use the left and right arrow keys and Enter to select a sub-menu. Use the arrow keys to navigate through the variables. Key in a variable using the number pad and press Write/Enter. To exit the menu press Cancel.

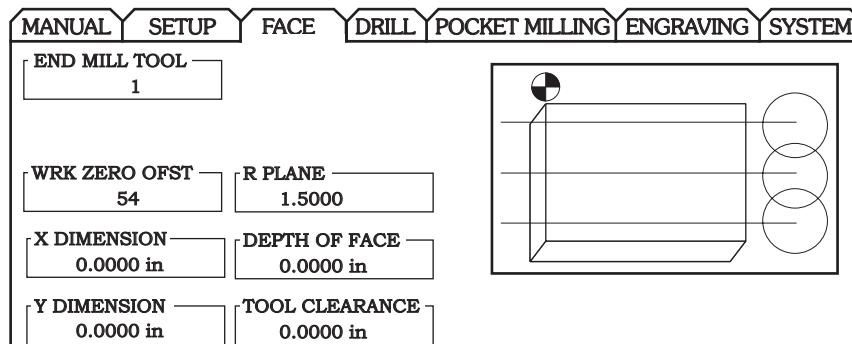
To exit the IPS menus press any of the Display keys, except Offset. Press Handle Jog to return to the IPS menus.

Note that a program entered through the IPS menus is also accessible in MDI mode.

AUTOMATIC MODE

Tool and Work offsets must be set before an automatic operation can be run. Enter the values for each tool used on the Setup screen. The tool offsets will be referenced when that tool is called in the automatic operation.

On each of the following interactive screens the user will be asked to enter data needed to complete common machining tasks. When all the data has been entered, pressing "Cycle Start" will begin the machining process.



Sample IPS Screen

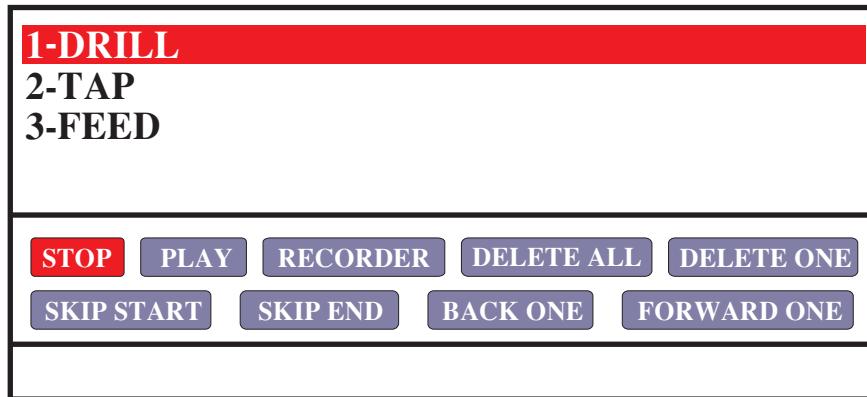
SYSTEM MODE

The System Mode screens are set up to show the user current alarms, an alarm history, an alarm viewer and write display messages. In addition the "Recorder" feature is in this group.

IPS Recorder

The Player/Recorder is used to create programs for multi-feature parts by combining part programs created using single feature automatic part programming

The recorder screen has a number of commands that are highlighted using the left and right arrow keys.



These commands are:

Stop – Stops the recording.

Play – Starts the playback, starting with the highlighted process. The mill will not start until Cycle Start is pressed.

Record – Begins recording the functions and entered data.

Delete All – Deletes all the recorded processes.

Delete One – Delete a single queued process. **Note that it will only delete the last process within the play list.** If a process is not last, you will have to delete all the processes that occur after it until you get to the one you need. Pressing the “Delete” key will delete the last process too.

Skip Start – Rewinds program to the first process. Pressing “Home” will return to the first process in the list as well.

Skip End – Forwards the program to the last process. Pressing “End” will skip to the end of the processes list as well.

Back One – Steps backwards through the processes. Pressing the up arrow on the keypad will do the same.

Forward One – Steps forwards through the processes. Pressing the down arrow on the keypad will do the same.

Operation

Enter the IPS feature, by pressing “Handle Jog”. Use the left and right arrow keys to highlight the “System” tab and Press “Enter”. Use the left and right arrow keys to highlight the “Recorder” tab and press “Enter”.

Additionally, the Recorder mode can be entered quickly by pressing F4 from any IPS screen. F4 will toggle the Recorder mode on and off.

Creating a Part Program

To develop a part program, set the Recorder/Player to “Record”, exit the System mode and enter the mode for the first process. Notice that when the control is in “Record” mode a blinking red “Recording” will display in the upper left.

Set up the machining process, enter the values and press “Cycle Start”. The mill will run the program and cut the first feature. Once finished, repeat the previous steps for the remaining part features. Note: Once Cycle Start is pushed, the operation is recorded, even if the operation is not completed.

Once all the machining processes are entered, return to the Recorder/Player (or press F4 to bring up the Recorder), highlight “Stop” and press “Enter” (or press F4), this stops the recording session.

You will notice that there is now a list of processes in the recorder window. These can be edited from this page using the other Recorder/Player buttons. An alternative for editing the operations is to execute the operation, then enter MDI mode. MDI mode reveals the machine code and can be edited there.

Note: If the machine is equipped with a tool changer the mill will automatically change tools as needed in the program.



Running a Part

Load a part, go to the Recorder/Player mode, highlight “Play” and press “Enter”; this initiates the Player. Pressing the Cycle Start button will start the recorded machining process. The Player will start on the highlighted line in the queue. **Warning: The mill will start, once Cycle Start is pressed.** The consecutive operations will execute, if desired, but Cycle Start must be pressed for each operation to continue. **Warning:** The operator must change tools, if required, prior to pressing Cycle Start for the next operation.

The program features can be verified by opening the Recorder/Player menu, highlight “Play”, and press “Enter”. Enter MDI mode, then press the “Graphics” key. This graphics display allows the user to view each operation prior to live cutting.

From the MDI mode, it is also possible to create a new G-code program in memory by placing the cursor at the top line, type Onnnnn (the letter “O” followed by a program number), then press the “Alter” button. This will create a new program in memory. This is useful for editing, backup etc.

Editing the Processes

You can remove processes by deleting them one at a time, by highlighting “Delete One” and pressing enter, or removing the whole list, by highlighting “Delete All” and pressing “Enter”. Use the arrows keys to scroll within the list.

The other editing buttons, Skip Start, Skip End, Back One and Forward One, can be used to start the part program at a specific place. The operator can add operations anytime by pressing the F4 key (to enter the Recorder/Player mode and executing a new operation.



MACROS

INTRODUCTION

This control feature is optional; call your dealer for information.

Macros add capabilities and flexibility to the control that are not possible with standard G-code. Some possible uses are, families of parts, custom canned cycles, complex motions, and driving optional devices. The possibilities are almost endless.

A Macro is any routine/subprogram that may be run multiple times. A macro statement can assign a value to a variable or read a value from a variable, evaluate an expression, conditionally or unconditionally branch to another point within a program, or conditionally repeat some section of program.

Here are a few examples of the applications for Macros. Rather than give macro code here, we will outline the general applications that Macros can be used for.

- **Tools For Immediate, On-Table Fixturing** Many setup procedures can be semi-automated to assist the machinist. Tools can be reserved for immediate situations that were not anticipated during tool design. For instance, suppose a company uses a standard clamp with a standard bolthole pattern. If it is discovered, after setup, that a fixture will need an additional clamp and if macro subroutine 2000 has been programmed for drilling the bolt pattern of the clamp, then the following two-step procedure is all that is needed for adding the clamp to the fixture.

1. Determine X, Y, and Z coordinates and angle where the clamp is to be placed by jogging the machine to the proposed clamp position and reading the position coordinates from the machine display.

2. Execute the following command in MDI mode:

G65 P2000 X??? Y??? Z??? A??? ;
Where “???” Are the values determined in Step 1.

Here, macro 2000 takes care of all the work since it was designed to drill the clamp bolthole pattern at the specified angle of **A**. Essentially; the machinist has created a custom canned cycle.

- **Simple Patterns That Are Repeated Over And Over Again In The Shop** Patterns that recur over and over again can be defined using macros and stored. For example:

- 1) Bolt hole patterns
- 2) Slotting
- 3) Angular patterns, any number of holes, at any angle, with any spacing
- 4) Specialty milling such as soft jaws
- 5) Matrix Patterns, (e.g. 12 across and 15 down)
- 6) Fly cutting a surface, (e.g. 12 inches by 5 inches using a 3 inch fly cutter)

- **Automatic Offset Setting Based On The Program** With macros, coordinate offsets can be set in each program so that setup procedures become easier and less error-prone (macro variables #2001-2800).

- **Probing** Probing enhances the capabilities of the machine in many ways, some examples are:

- 1) Profiling of a part to determine unknown dimensions for later machining.
- 2) Tool calibration for offset and wear values.
- 3) Inspection prior to machining to determine material allowance on castings.
- 4) Inspection after machining to determine parallelism and flatness values as well as location.



Useful G and M Codes

M00, M01, M30 - Stop Program

G04 - Dwell

G65 Pxx - Macro subprogram call. Allows passing of variables.

M96 Pxx Qxx - Conditional Local Branch when Discrete Input Signal is 0

M97 Pxx - Local Sub Routine Call

M98 Pxx - Sub Program Call

M99 - Sub Program Return or Loop

G103 - Block Lookahead Limit. No cutter comp allowed

M109 - Interactive User Input (see "M Codes" section)

Settings

There are 3 settings that can affect macro programs (9000 series programs), these are 9xxxx progs Lock (#23), 9xxx Progs Trace (#74) and 9xxx Progs Single BLK (#75).

Lookahead

Lookahead is an issue of great importance to the macro programmer. The control will attempt to process as many lines as possible ahead of time in order to speed up processing. This includes the interpretation of macro variables. For example,

```
#1101=1  
G04 P1.  
#1101=0
```

This is intended to turn an output ON, wait 1 second, and then turn it off. However, lookahead will cause the output to turn on then immediately back off while the dwell is being processed. G103 P1 can be used to limit lookahead to 1 blocks. To make this example work properly, it must be modified as follows:

```
G103 P1 (See the G-code section of the manual for a further explanation of G103)  
;  
#1101=1  
G04 P1.  
;  
;  
;  
#1101=0
```

Round Off

The control stores decimal numbers as binary values. As a result, numbers stored in variables can be off by 1 least significant digit. For example, the number 7 stored in macro variable #100 may later be read as 7.000001, 7.000000, or 6.999999. If your statement was, "IF [#100 EQ 7]...", it may give a false reading. A safer way of programming this would be, "IF [ROUND [#100] EQ 7]...". This issue is usually only a problem when storing integers in macro variables where you do not expect to see a fractional part later.

OPERATION NOTES

Macro variables can be saved or loaded through the RS-232 or the optional floppy drive, much like settings, and offsets. Refer to the "Loading Programs to the Control" section.

Variable Display Page

The macro variables are displayed and can be modified through the current commands display. To get to the pages, press CURNT COMDS and use the page up/down key.

As the control interprets a program, the variable changes are displayed on the variable display page and results can be viewed.

The macro variable is set by entering a value and then pressing the Write/Enter button. Macro Variables can be cleared by pressing the Origin key, this will clear all variables.



Entering the macro variable number and pressing the up/down arrow will search for that variable.

The variables displayed represent the values of the variables during the running of the program. At times, this may be up to 15 blocks ahead of the actual machine actions. Debugging the programs are easier when inserting a G103 at the beginning of a program to limit block buffering and then removing the G103 after debugging is completed.

Macro Arguments

The arguments in a G65 statement are a means of sending values to and setting the local variables of a macro subroutine.

In the previous example 2 , the arguments (values of) X and Y are passed to the macro subroutine local variables. Local variable #24 is associated with X and is set to 0.5. Similarly, Local variable #25 is associated with Y and is set to 0.25.

The following two tables indicate the mapping of the alphabetic address variables to the numeric variables used in a macro subroutine.

Alphabetic Addressing

Address: Variable:	A 1	B 2	C 3	D 7	E 8	F 9	G -	H 11	I 4	J 5	K 6	L -	M 13
Address: Variable:	N -	O -	P -	Q 17	R 18	S 19	T 20	U 21	V 22	W 23	X 24	Y 25	Z 26

Alternate Alphabetic Addressing

Address: Variable:	A 1	B 2	C 3	I 4	J 5	K 6	I 7	J 8	K 9	I 10	J 11		
Address: Variable:	K 12	I 13	J 14	K 15	I 16	J 17	K 18	I 19	J 20	K 21	I 22		
Address: Variable:	J 23	K 24	I 25	J 26	K 27	I 28	J 29	K 30	I 31	J 32	K 33		

Arguments accept any floating-point value to four decimal places. If the control is in metric, it will assume thousandths (.000). In Example 3, local variable #7 will receive .0004.

If a decimal is not included in an argument value, such as:

G65 P9910 A1 B2 C3

The values are passed to macro subroutines according to the following table:

Integer Argument Passing (no decimal point)

Address: Variable:	A .001	B .001	C .001	D 1.	E 1.	F 1.	G -	H 1.	I .0001	J .0001	K .0001	L 1.	M 1.
Address: Variable:	N -	O -	P -	Q .0001	R .0001	S 1.	T 1.	U .0001	V .0001	W .0001	X .0001	Y .0001	Z .0001

All 33 local macro variables can be assigned values with arguments by using the alternate addressing method. The following example shows how one could send two sets of coordinate locations to a macro subroutine. Local variables #4 through #9 would be set to .0001 through .0006 respectively.



Example 3: G65 P2000 I1 J2 K3 I4 J5 K6;

The following letters cannot be used to pass parameters to a macro subroutine: G, L, N, O or P.

Macro Variables

There are three categories of macro variables: *system* variables, *global* variables, and *local* variables.

Macro Constants are floating point values placed in a macro expression. They can be combined with addresses A-Z or they can stand alone when used within an expression. Examples of constants are .0001, 5.3 or -10.

Local Variables

Local variables range between #1 and #33. A set of local variables is available at all times. When a call to a subroutine with a G65 command is executed, the local variables are saved and a new set is available for use. This is called "nesting" of the local variables. During a G65 call, all of the new local variables are cleared to undefined values and any local variables that have corresponding address variables in the G65 line are set to the G65 line values. Below is a table of the local variables along with the address variable arguments that change them.

Variable:	1	2	3	4	5	6	7	8	9	10	11
Address:	A	B	C	I	J	K	D	E	F	I	H
Alternate:					I		J	K		J	
Variable:	12	13	14	15	16	17	18	19	20	21	22
Address:	M	R	S	T	Q	K	R	S	T	U	V
Alternate:	K	I	J	K	I	J	K	I	J	K	I
Variable:	23	24	25	26	27	28	29	30	31	32	33
Address:	W	X	Y	Z	K	I	J	K	I	J	K
Alternate:	J	K	I	J	K	I	J	K	I	J	K

Note that variables 10, 12, 14-16 and 27-33 do not have corresponding address arguments. They can be set if a sufficient number of I, J and K arguments are used as indicated above in the section about arguments.

Once in the macro subroutine, the local variables can be read and modified by referencing the variable numbers 1-33.

When the **L** argument is used to do multiple repetitions of a macro subroutine, the arguments are set only on the first repetition. This means that if local variables 1-33 are modified in the first repetition, then the next repetition will have access only to the modified values. Local values are retained from repetition to repetition when the **L** address is greater than 1.

Calling a subroutine via an M97 or M98 does not nest the local variables. Any local variables referenced in a subroutine called by an M98 are the same variables and values that existed prior to the M97 or M98 call.

Global Variables

Global variables are variables that are accessible at all times. There is only one copy of each global variable. Global variables occur in three ranges: 100-199, 500-699 and 800-999. The global variables remain in memory when power is turned off.

Occasionally, there have been some macros written for factory installed options that use the global variables. For example, probing, pallet changers, etc. When using global variables, be sure they are not in use by another program on the machine.

System Variables

System variables give the programmer the ability to interact with a variety of control conditions. By setting a system variable, the function of the control can be modified. By reading a system variable, a program can modify its behavior based on the value in the variable. Some system variables have a Read Only status; this means that the programmer cannot modify them. A brief table of currently implemented system variables follows with an explanation of their use.



VARIABLES	USAGE
#0	Not a number (read only)
#1-#33	Macro call arguments
#100-#199	General-purpose variables saved on power off
#500-#699	General-purpose variables saved on power off
#700-#749	Hidden variables for internal use only.
#800-#999	General-purpose variables saved on power off
#1000-#1063	64 discrete inputs (read only)
#1064-#1068	Maximum axis loads.
#1080-#1087	Raw analog to digital inputs (read only)
#1090-#1098	Filtered analog to digital inputs (read only)
#1094	Spindle load with OEM spindle drive (read only)
#1098	Spindle load with Haas vector drive (read only)
#1100-#1139	40 discrete outputs
#1140-#1155	16 extra relay outputs via multiplexed output
#1264-#1268	Maximum axis loads.
#1601-#1800	Number of Flutes of tools #1 through 200
#1801-#2000	Maximum recorded vibrations of tools 1 through 200
#2002-#2200	Tool length offsets
#2201-#2400	Tool length wear
#2401-#2600	Tool diameter/radius offsets
#2601-#2800	Tool diameter/radius wear
#3000	Programmable alarm
#3001	Millisecond timer
#3002	Hour timer
#3003	Single block suppression
#3004	Override control
#3006	Programmable stop with message
#3011	Year, month, day
#3012	Hour, minute, second
#3020	Power on timer (read only)
#3021	Cycle start timer
#3022	Feed timer
#3023	Present part timer
#3024	Last complete part timer
#3025	Previous part timer
#3026	Tool in spindle (read only)
#3027	Spindle RPM (read only)
#3028	Number of pallet loaded on receiver
#3030	Single Block
#3031	Dry Run
#3032	Block Delete
#3033	Opt Stop
#3201-#3400	Actual Diameter for tools 1 through 200
#3401-#3600	Programmable coolant positions for tools 1 through 200
#3901	M30 count 1
#3902	M30 count 2
#4000-#4021	Previous block G-Code group codes
#4101-#4126	Previous block address codes

NOTE: Mapping of 4101 to 4126 is the same as the alphabetic addressing of "Macro Arguments" section; e.g. the statement x1.3 sets variable #4124 to 1.3. The mapping of the axes are x=1, y=2, ... b=5, for example, the Z machine coordinate system variable would be #5023.



#5001-#5005	Previous block end position
#5021-#5025	Present machine coordinate position
#5041-#5045	Present work coordinate position
#5061-#5065	Present skip signal position - X,Y,Z,A,B
#5081-#5085	Present tool offset
#5201-#5205	Common offset
#5221-#5225	G54 work offsets
#5241-#5245	G55 work offsets
#5261-#5265	G56 work offsets
#5281-#5285	G57 work offsets
#5301-#5305	G58 work offsets
#5321-#5325	G59 work offsets
#5401-#5500	Tool feed timers (seconds)
#5501-#5600	Total tool timers (seconds)
#5601-#5699	Tool life monitor limit
#5701-#5800	Tool life monitor counter
#5801-#5900	Tool load monitor maximum load sensed so far
#5901-#6000	Tool load monitor limit
#6001-#6277	Settings (read only)
#6501-#6999	Parameters (read only)

NOTE: The low order bits of large values will not appear in the macro variables for settings and parameters

#7001-#7006 (#14001-#14006)	G110 (G154 P1) additional work offsets
#7021-#7026 (#14021-#14026)	G111 (G154 P2) additional work offsets
#7041-#7046 (#14041-#14046)	G112 (G154 P3) additional work offsets
#7061-#7066 (#14061-#14066)	G113 (G154 P4) additional work offsets
#7081-#7086 (#14081-#14086)	G114 (G154 P5) additional work offsets
#7101-#7106 (#14101-#14106)	G115 (G154 P6) additional work offsets
#7121-#7126 (#14121-#14126)	G116 (G154 P7) additional work offsets
#7141-#7146 (#14141-#14146)	G117 (G154 P8) additional work offsets
#7161-#7166 (#14161-#14166)	G118 (G154 P9) additional work offsets
#7181-#7186 (#14181-#14186)	G119 (G154 P10) additional work offsets
#7201-#7206 (#14201-#14206)	G120 (G154 P11) additional work offsets
#7221-#7226 (#14221-#14221)	G121 (G154 P12) additional work offsets
#7241-#7246 (#14241-#14246)	G122 (G154 P13) additional work offsets
#7261-#7266 (#14261-#14266)	G123 (G154 P14) additional work offsets
#7281-#7286 (#14281-#14286)	G124 (G154 P15) additional work offsets
#7301-#7306 (#14301-#14306)	G125 (G154 P16) additional work offsets
#7321-#7326 (#14321-#14326)	G126 (G154 P17) additional work offsets
#7341-#7346 (#14341-#14346)	G127 (G154 P18) additional work offsets
#7361-#7366 (#14361-#14366)	G128 (G154 P19) additional work offsets
#7381-#7386 (#14381-#14386)	G129 (G154 P20) additional work offsets
#7501-#7506	Pallet priority
#7601-#7606	Pallet status
#7701-#7706	Part program numbers assigned to pallets
#7801-#7806	Pallet usage count
#8500	Advanced Tool Management (ATM). Group ID
#8501	ATM. Percent of total available tool life of all tools in the group.
#8502	ATM. Total available tool usage count in the group.
#8503	ATM. Total available tool hole count in the group.
#8504	ATM. Total available tool feed time (in seconds) in the group.
#8505	ATM. Total available tool total time (in seconds) in the group.
#8510	ATM. Next tool number to be used.
#8511	ATM. Percent of available tool life of the next tool.



#8512	ATM. Available usage count of the next tool.
#8513	ATM. Available hole count of the next tool.
#8514	ATM. Available feed time of the next tool (in seconds).
#8515	ATM. Available total time of the next tool (in seconds).
#14401-#14406	G154 P21 additional work offsets
#14421-#14426	G154 P22 additional work offsets
#14441-#14446	G154 P23 additional work offsets
#14461-#14466	G154 P24 additional work offsets
#14481-#14486	G154 P25 additional work offsets
#14501-#14506	G154 P26 additional work offsets
#14521-#14526	G154 P27 additional work offsets
#14541-#14546	G154 P28 additional work offsets
#14561-#14566	G154 P29 additional work offsets
#14581-#14586	G154 P30 additional work offsets
•	
•	
#14781-#14786	G154 P40 additional work offsets
•	
•	
#14981-#14986	G154 P50 additional work offsets
•	
•	
#15181-#15186	G154 P60 additional work offsets
•	
•	
#15381-#15386	G154 P70 additional work offsets
•	
•	
#15581-#15586	G154 P80 additional work offsets
•	
•	
#15781-#15786	G154 P90 additional work offsets
•	
•	
15881-15886	G154 P95 additional work offsets
15901-15906	G154 P96 additional work offsets
15921-15926	G154 P97 additional work offsets
15941-15946	G154 P98 additional work offsets
15961-15966	G154 P99 additional work offsets

SYSTEM VARIABLES IN-DEPTH

Variables #750 and #751

These macro variables collect the input from serial port 2. The programmer can test for data queued in the serial port 2 buffer, and collect the data for processing. Macro variable #750 will inform the programmer if there is data waiting in the RS232 port 2. A value of 1 means that the receive buffer has data queued, otherwise a value of 0 is returned. Macro variable 751 collects the first character from the input buffer, when data is queued. This means that the buffer content is first checked to see if it is empty; if not the value of the next character queued is returned.



1-Bit Discrete Inputs

Inputs designated as "spare" can be connected to external devices and used by the programmer.

1-Bit Discrete Outputs

The Haas control is capable of controlling up to 56 discrete outputs. However, a number of these outputs are already reserved for use by the Haas controller.

CAUTION! Do not use outputs that are reserved by the system. Using these outputs may result in injury or damage to your equipment.

The user can change the state of these outputs by writing to variables designated as "spare". If the outputs are connected to relays, then an assignment of "1" sets the relay. An assignment of "0" clears the relay.

Referencing these outputs will return the current state of the output and this may be the last assigned value or it may be the last state of the output as set by some user M code. For example, after verifying that output #1108 is "spare":

```
#1108=1; (Turns #1108 relay on)
#101=#3001+1000; (101 is 1 second from now)
WHILE [[#101 GT #3001] AND [#1109 EQ 0]] D01
END1 (Wait here 1 second or until relay #1109 goes high)
#1108=0; (Turns #1108 relay off)
```

If the control is not equipped with the M-code relay board, then M21 through M28 will be mapped from #1132-#1139. If the M-code relay board is installed, see the 8M-option section for information and instructions.

Note: Always test or dry run programs that have been developed for macros that is using new hardware.

Maximum Axis Loads

The following variables are used to contain the maximum load values for each axis. They can be cleared by cycling power to the machine or by setting the macro to zero in a program (for example, #1064=0;).

1064 = X axis	1264 = C axis
1065 = Y axis	1265 = U axis
1066 = Z axis	1266 = V axis
1067 = A axis	1267 = W axis
1068 = B axis	1268 = T axis

Tool Offsets

Each tool offset has a length (H) and radius (D) along with associated wear values.

#2001-#2200 H geometry offsets (1-200) for length.
#2200-#2400 H geometry wear (1-200) for length.
#2401-#2600 D geometry offsets (1-200) for diameter.
#2601-#2800 D geometry wear (1-200) for diameter.

Programmable Messages

#3000 Alarms can be programmed. A programmable alarm will act just like Haas internal alarms. An alarm is generated by setting the macro variable #3000 to a number between 1 and 999.

#3000= 15 (MESSAGE PLACED INTO ALARM LIST);

When this is done, "Alarm" flashes at the bottom of the display and the text in the next comment is placed into the alarm list. The alarm number (in this example, 15) is added to 1000 and used as an alarm number. If an alarm is generated in this manner all motion stops and the program must be reset to continue. Programmable alarms are always numbered between 1000 and 1999.

The first 34 characters of the comment will be used for the alarm message.



Timers

Haas macros can access two timers. These timers can be set to a value by assigning a number to the respective variable. A program can then later read the variable and determine the time passed since the timer was set. Timers can be used to imitate dwell cycles, determine part-to-part time or wherever time-dependent behavior is desired.

#3001 Millisecond Timer - The millisecond timer is updated every 20 milliseconds and thus activities can be timed with an accuracy of only 20 milliseconds. At Power On, the millisecond timer is reset. The timer has a limit of 497 days. The whole number returned after accessing #3001 represents the number of milliseconds.

#3002 Hour Timer - The hour timer is similar to the millisecond timer except that the number returned after accessing #3002 is in hours. The hour and millisecond timers are independent of each other and can be set separately.

System Overrides

#3003 Variable 3003 is the Single Block Suppression parameter. It overrides the Single Block function in G-code. In the following example Single Block is ignored when #3003 is set equal to 1. After M3003 is set =1, each G-code command (lines 2-4) are executed continuously even though the Single Block function is ON. When #3003 is set equal to zero, Single Block will operate as normal. That is, the user must press Cycle Start start each line of code (lines 6-8).

```
#3003=1;  
G54 G00 G90 X0 Y0;  
G81 R0.2 Z-0.1 F20 L0;  
S2000 M03;  
#3003=0;  
T02 M06;  
G83 R0.2 Z-1 F10. L0;  
X0. Y0.;
```

Variable #3004

Variable #3004 is a variable that overrides specific control features while running.

The first bit disables the Feed Hold button. If feed hold is not to be used during a section of code, then put variable #3004, assigned to 1, before the specific lines of code. After that section of code set #3004 to 0 to restore the function of the Feed Hold button. For example:

Approach code	(Feed Hold allowed)
#3004=1;	(Disables Feed Hold button)
Non-stoppable code	(Feed Hold not allowed)
#3004=0;	(Enables Feed Hold button)
Depart code	(Feed Hold allowed)

The following is a map of variable #3004 bits and the associated overrides. E – Enabled D – Disabled

#3004	Feed Hold	Feed Rate Override	Exact Top Check
0	E	E	E
1	D	E	E
2	E	D	E
3	D	D	E
4	E	E	D
5	D	E	D
6	E	D	D
7	D	D	D

#3006 Programmable Stop

Stops can be programmed which acts like an M00 - The control stops and waits until Cycle Start is pressed. Once Cycle Start is pressed, the program continues with the block after the #3006. In the following example, the first 15 characters of the comment are displayed on the lower left part of the screen.



IF [#1 EQ #0] THEN #3006=101 (comment here);

#4001-#4021 Last Block (Modal) Group Codes

The grouping of G codes permits more efficient processing. G codes with similar functions are usually under the same group. For example, G90 and G91 are under group 3. These variables store the last or default G code for any of 21 groups. By reading the group code, a macro program can change the behavior of the G-code. If 4003 contains 91, then a macro program could determine that all moves should be incremental rather than absolute. There is no associated variable for group zero; group zero G codes are Non-modal.

#4101-#4126 Last Block (Modal) Address Data

Address codes A-Z (excluding G) are maintained as modal values. The information represented by the last line of code interpreted by the lookahead process is contained in variables 4101 through 4126. The numeric mapping of variable numbers to alphabetic addresses corresponds to the mapping under alphabetic addresses. For example, the value of the previously interpreted **D** address is found in #4107 and the last interpreted **I** value is #4104. When aliasing a macro to an M-code, you may not pass variables to the macro using variables 1-33; instead, use the values from 4101-4126 in the macro.

#5001-#5005 Last Target Position

The final programmed point for the last motion block can be accessed through variables #5001-#5005, X, Y, Z, A, and B, respectively. Values are given in the current work coordinate system and can be used while the machine is in motion.

Axis Position Variables

#5020 X-axis #5021 Y-axis #5022 Z-axis
#5023 A-axis #5024 B-axis #5025 C-axis

#5021-#5025 Current Machine Coordinate Position

The current position in machine coordinates can be obtained through #5021-#5025, X, Y, Z, A, and B, respectively. The values CANNOT be read while the machine is in motion. The value of #5023 (Z) has tool length compensation applied to it.

#5041-#5045 Current Work Coordinate Position

The current position in the current work coordinates can be obtained through #5041-5045, X, Y, Z, A, and B, respectively. The values CANNOT be read while the machine is in motion. The value of #5043 (Z) has tool length compensation applied to it.

#5061-#5065 Current Skip Signal Position

The position where the last skip signal was triggered can be obtained through #5061-#5065, X, Y, Z, A, and B, respectively. Values are given in the current work coordinate system and can be used while the machine is in motion. The value of #5063 (Z) has tool length compensation applied to it.

#5081-#5085 Tool Length Compensation

The current total tool length compensation that is being applied to the tool. This includes tool length offset referenced by the current value set in H (#4008) plus the wear value.

Note: The mapping of the axes are x=1, y=2, ... b=5. So as an example, the Z machine coordinate system variable would be #5023.

Pallet Changer

The status of the pallets, from the Automatic Pallet Changer are checked using the following variables:

#7501-#7506	Pallet priority
#7601-#7606	Pallet status
#7701-#7706	Part program numbers assigned to pallets
#7801-#7806	Pallet usage count
#3028	Number of pallet loaded on receiver



Offsets

All tool work offsets can be read and set within a macro expression. This allows the programmer to preset coordinates to approximate locations, or to set coordinates to values based upon the results of skip signal locations and calculations. When any of the offsets are read, the interpretation lookahead queue is stopped until that block is executed.

```
#5201-#5205 G52 X, Y, Z, A, B OFFSET VALUES  
#5221-#5225 G54 " " " " "  
#5241-#5245 G55 " " " " "  
#5261-#5265 G56 " " " " "  
#5281-#5285 G57 " " " " "  
#5301-#5305 G58 " " " " "  
#5321-#5325 G59 " " " " "  
#7001-#7005 G110 X, Y, Z, A, B OFFSET VALUES  
" " " " " "  
#7381-#7385 G129 X, Y, Z, A, B OFFSET VALUES
```

VARIABLE USAGE

All variables are referenced with a pound sign (#) followed by a positive number. Examples are: #1, #101, and #501.

Variables are decimal values that are represented as floating point numbers. If a variable has never been used, it can take on a special “undefined” value. This indicates that it has not been used. A variable can be set to undefined with the special variable #0. #0 has the value of undefined or 0.0 depending on the context it is used in. Indirect references to variables can be accomplished by enclosing the variable number in brackets.

#[<Expression>]

The expression is evaluated and the result becomes the variable accessed. For example:

#1=3;

#[#1]=3.5 + #1;

This sets the variable #3 to the value 6.5.

Variables can be used in place of G-code address where “address” refers to the letters A-Z.
In the block: N1 G0 G90 X1.0 Y0;

the variables can be set to the following values:

```
#7=0;  
#11=90;  
#1=1.0;  
#2=0.0;
```

and replaced by: N1 G#7 G#11 X#1 Y#2;

The values in the variables at runtime are used as the address values.

ADDRESS SUBSTITUTION

The usual method of setting control addresses A-Z is the address followed by a number. For example:

G01 X1.5 Y3.7 F20. ;

Sets addresses G, X, Y and F to 1, 1.5, 3.7 and 20.0 respectively and thus instructs the control to move linearly, G01, to position X=1.5 Y=3.7 at a feed rate of 20 inches per minute. Macro syntax allows the address values to be replaced with any variable or expression.



The previous statement can be replaced by the following code:

```
#1=1;  
#2=.5;  
#3=3.7;  
#4=20;  
G#1 X[#1+#2] Y#3 F#4 ;
```

The permissible syntax on addresses A-Z (exclude N or O) is as follows:

<address><-><variable>	A-#101
<address>[<expression>]	Y[#5041+3.5]
<address><->[<expression>]	Z-[SIN[#1]]

If the value of the variable does not agree with the range of the address, the control will generate an alarm. For example, the following code would result in a range error alarm because tool diameter numbers range from 0-50.

```
#1=75;  
D#1;
```

When a variable or expression is used in place of an address value, the value is rounded to the least significant digit. If #1=.123456, then G1X#1 would move the machine tool to .1235 on the X axis. If the control is in the metric mode, the machine would be moved to .123 on the X axis.

When an undefined variable is used to replace an address value, then that address reference is ignored. For example, if #1 is undefined then the block

G00 X1.0 Y#1 ;

Becomes

G00 X1.0.

No Y movement takes place.

Macro Statements

Macro statements are lines of code that allow the programmer to manipulate the control with features similar to any standard programming language. Included are functions, operators, conditional and arithmetic expressions, assignment statements, and control statements.

Functions and operators are used in expressions to modify variables or values. The operators are essential to expressions while functions make the programmer's job easier.

Functions

Functions are built-in routines that the programmer has available to use. All functions have the form

<function_name>[argument] and return floating-point decimal values. The functions provided in the Haas control are as follows:



Function	Argument	Returns	Notes
SIN[]	Degrees	Decimal	Sine
COS[]	Degrees	Decimal	Cosine
TAN[]	Degrees	Decimal	Tangent
ATAN[]	Decimal	Degrees	Arctangent Same as FANUC ATAN[]/[1]
SQRT[]	Decimal	Decimal	Square root
ABS[]	Decimal	Decimal	Absolute value
ROUND[]	Decimal	Decimal	Round off a decimal
FIX[]	Decimal	Integer	Truncate fraction
ACOS[]	Decimal	Degrees	Arccosine
ASIN[]	Decimal	Degrees	Arcsine
#[]	Integer	Integer	Variable Indirection
DPRNT[]	ASCII text		External Output

Notes on Functions

The function "Round" works differently depending on the context that it is used. When used in arithmetic expressions, any number with a fractional part greater than or equal to .5 is rounded up to the next whole integer; otherwise, the fractional part is truncated from the number.

```
#1= 1.714 ;
#2= ROUND[#1] ; (#2 is set to 2.0)
#1= 3.1416 ;
#2= ROUND[#1] ; (#2 is set to 3.0)
```

When round is used in an address expression, then the argument "Round" is rounded to the significant precision. For *metric* and *angle* dimensions, three-place precision is the default. For *inch*, four-place precision is the default.

```
#1= 1.00333 ;
G0 X[ #1 + #1 ] ;
        (Table moves to 2.0067) ;
G0 X[ ROUND[ #1 ] + ROUND[ #1 ] ] ;
        (Table moves to 2.0066) ;
G0 A[ #1 + #1 ] ;
        (Axis moves to 2.007) ;
G0 A[ ROUND[ #1 ] + ROUND[ #1 ] ] ;
        (Axis moves to 2.006) ;
D[1.67] (Diameter 2 is made current) ;
```

Fix vs. Round

#1=3.54; #2=ROUND[#1]; #3=FIX[#1]. #2 will be set to 4. #3 will be set to 3.

Operators

Operators can be classified into three categories: Arithmetic operators, Logical operators and Boolean operators.

Arithmetic Operators

Arithmetic operators consist of unary and binary operators. They are:

+	- Unary plus	+1.23
-	- Unary minus	-[COS[30]]
+	- Binary addition	#1=#1+5
-	- Binary subtraction	#1=#1-1
*	- Multiplication	#1=#2*#3
/	- Division	#1=#2/4
MOD	- Remainder	#1=27 MOD 20 (#1 contains 7)



Logical Operators

Logical operators are operators that work on binary bit values. Macro variables are floating point numbers. When logical operators are used on macro variables, only the integer portion of the floating point number is used. The logical operators are:

OR - logically OR two values together
XOR - Exclusively OR two values together
AND - Logically AND two values together

Examples:

#1=1.0; 0000 0001 #2=2.0; 0000 0010 #3=#1 OR #2 0000 0011	Here the variable #3 will contain 3.0 after the OR operation.
#1=5.0; #2=3.0; IF [#1 GT 3.0] AND [#2 LT 10]] GOTO1	Here control will transfer to block 1 because #1 GT 3.0 evaluates to 1.0 and #2 LT 10 evaluates to 1.0, thus 1.0 AND 1.0 is 1.0 (TRUE) and the GOTO occurs.

Note that care must be taken when using logical operators so that the desired result is achieved.

Boolean Operators

Boolean operators always evaluate to 1.0 (TRUE) or 0.0 (FALSE). There are six Boolean operators. These operators are not restricted to conditional expressions, but they most often are used in conditional expressions. They are:

EQ - Equal to
NE - Not Equal to
GT - Greater Than
LT - Less Than
GE - Greater than or Equal to
LE - Less Than or Equal to

The following are four examples of how Boolean and Logical operators can be used:

<u>Example</u>	<u>Explanation</u>
IF [#1 EQ 0.0] GOTO100;	Jump to block 100 if value in variable #1 equals 0.0.
WHILE [#101 LT 10] DO1;	While variable #101 is less than 10 repeat loop DO1..END1.
#1=[1.0 LT 5.0];	Variable #1 is set to 1.0 (TRUE).
IF [#1 AND #2 EQ #3] GOTO1	If variable #1 logically ANDed with variable #2 is equal to the value in #3 then control jumps to block 1.

Expressions

Expressions are defined as any sequence of variables and operators surrounded by the square brackets "[" and "]". There are two uses for expressions: conditional expressions or arithmetic expressions. Conditional expressions return FALSE (0.0) or TRUE (any non zero) values. Arithmetic expressions use arithmetic operators along with functions to determine a value.

Conditional Expressions

In the HAAS control, ALL expressions set a conditional value. The value is either 0.0 (FALSE) or the value is nonzero (TRUE). The context in which the expression is used determines if the expression is a conditional expression. Conditional expressions are used in the IF and WHILE statements and in the M99 command. Conditional expressions can make use of Boolean operators to help evaluate a TRUE or FALSE condition.



The M99 conditional construct is unique to the HAAS control. Without macros, M99 in the HAAS control has the ability to branch unconditionally to any line in the current subroutine by placing a P code on the same line. For example:

N50 M99 P10;

branches to line N10. It does not return control to the calling subroutine. With macros enabled, M99 can be used with a conditional expression to branch conditionally. To branch when variable #100 is less than 10 we could code the above line as follows.

N50 [#100 LT 10] M99 P10;

In this case, the branch occurs only when #100 is less than 10, otherwise processing continues with the next program line in sequence. In the above, the conditional M99 can be replaced with

N50 IF [#100 LT 10] GOTO10;

Arithmetic Expressions

An arithmetic expression is any expression using variables, operators, or functions. An arithmetic expression returns a value. Arithmetic expressions are usually used in assignment statements, but are not restricted to them.

Examples of Arithmetic expressions: #101=#145*#30;

```
#1=#1+1;  
X[#105+COS[#101]];  
#[#2000+#13]=0;
```

Assignment Statements

Assignment statements allow the programmer to modify variables. The format of the assignment statement is:

<expression>=<expression>

The expression on the left of the equal sign must always refer to a macro variable, whether directly or indirectly. The following macro initializes a sequence of variables to any value. Here both direct and indirect assignments are used.

```
O0300  
N1 IF [#2 NE #0] GOTO2  
#3000=1  
N2 IF [#19 NE #0] GOTO3  
#3000=2  
N3 WHILE [#19 GT 0] DO1 ;  
#19=#19-1  
#[#2+#19]=#22  
END1 ;  
M99 ;
```

(Initialize an array of variables) ;
(B=base variable) ;
(Base variable not given) ;
(S=size of array);
(Size of array not given) ;
(Decrement count) ;
(V=value to set array to) ;

The above macro could be used to initialize three sets of variables as follows:

```
G65 P300 B101. S20 (INIT 101..120 TO #0) ;  
G65 P300 B501. S5 V1 (INIT 501..505 TO 1.0) ;  
G65 P300 B550. S5 V0 (INIT 550..554 TO 0.0) ;
```

The decimal point in B101., etc. would be required.

Control Statements

Control statements allow the programmer to branch, both conditionally and unconditionally. They also provide the ability to iterate a section of code based on a condition.



Unconditional Branch (GOTOnnn and M99 Pnnnn)

In the Haas control, there are two methods of branching unconditionally. An unconditional branch will always branch to a specified block. M99 P15 will branch unconditionally to block number 15. The M99 can be used whether or not macros is installed and is the traditional method for branching unconditionally in the Haas control. GOTO15 does the same as M99 P15. In the Haas control, a GOTO command can be used on the same line as other G-codes. The GOTO is executed after any other commands like M codes.

Computed Branch (GOTO#n and GOTO [expression])

Computed branching allows the program to transfer control to another line of code in the same subprogram. The block can be computed as the program is running, using the GOTO [expression] form. Or the block can be passed in through a local variable, as in the GOTO#n form.

The GOTO will round the variable or expression result that is associated with the Computed branch. For instance, if #1 contains 4.49 and GOTO#1 is executed, the control will attempt to transfer to a block containing N4. If #1 contains 4.5, then execution will transfer to a block containing N5.

The following code skeleton could be developed to make a program that adds serial numbers to parts:

```
O9200                                (Engrave digit at current location.)  
;  
(D=Decimal digit to engrave);  
IF [[#7 NE #0] AND [#7 GE 0] AND [#7 LE 9]] GOTO99;  
#3000=1                               (Invalid digit)  
;  
N99  
#7=FIX[#7]                            (Truncate any fractional part) ;  
;  
GOTO#7                                (Now engrave the digit) ;  
;  
N0                                     (Do digit zero)  
...  
M99  
;  
N1                                     (Do digit one)  
;  
M99  
;  
N2                                     (Do digit two)  
;  
...  
;  
(etc,...)
```

With the above subroutine, you would engrave digit five with the following call:

G65 P9200 D5;

Computed GOTOS using expression could be used to branch processing based on the results of reading hardware inputs. An example might look like the following:



```
GOTO [[#1030*2]+#1031];
NO    (1030=0, 1031=0);
...
M99;
N1    (1030=0, 1031=1);
...
M99;
N2    (1030=1, 1031=0);
...
M99;
N3    (1030=1, 1031=1);
...
M99;
```

The discrete inputs always return either 0 or 1 when read. The GOTO [expression] will branch to the appropriate line of code based on the state of the two discrete inputs #1030 and #1031.

Conditional Branch (IF and M99 Pnnnn)

Conditional branching allows the program to transfer control to another section of code within the same subroutine. Conditional branching can only be used when macros are enabled. The Haas control allows two similar methods for accomplishing conditional branching.

IF [<conditional expression>] GOTON

As discussed, <conditional expression> is any expression that uses any of the six Boolean operators EQ, NE, GT, LT, GE, or LE. The brackets surrounding the expression are mandatory. In the Haas control, it is not necessary to include these operators. For example:

```
IF [#1 NE 0.0] GOTON5;
```

could also be:

```
IF [#1] GOTON5;
```

In this statement, if the variable #1 contains anything but 0.0, or the undefined value #0, then branching to block 5 will occur; otherwise, the next block will be executed.

In the Haas control, a conditional expression can also be used with the M99 Pnnnn format. For example:

```
G0 X0 Y0 [#1EQ#2] M99 P5;
```

Here, the conditional is for the M99 portion of the statement only. The machine tool is instructed to X0, Y0 whether or not the expression evaluates to True or False. Only the branch, M99, is executed based on the value of the expression. It is recommended that the IF GOTON version is used if portability is desired.

Conditional Execution (IF THEN)

Execution of control statements can also be achieved by using the IF THEN construct. The format is:

IF [<conditional expression>] THEN <statement>;

NOTE: To preserve compatibility with FANUC syntax "THEN" may not be used with GOTON.

This format is traditionally used for conditional assignment statements such as:

```
IF [#590 GT 100] THEN #590=0.0 ;
```

Variable #590 is set to zero when the value of #590 exceeds 100.0. In the Haas control, if a conditional evaluates to FALSE (0.0), then the remainder of the IF block is ignored. This means that control statements can also be conditioned so that we could write something like:

```
IF [#1 NE #0] THEN G1 X#24 Y#26 F#9 ;
```

This executes a linear motion only if variable #1 has been assigned a value. Another example is:



IF [#1 GE 180] THEN #101=0.0 M99 ;

This says that if variable #1 (address A) is greater than or equal to 180, then set variable #101 to zero and return from the subroutine.

Here is an example of an IF statement that branches if a variable has been initialized to contain any value. Otherwise, processing will continue and an alarm will be generated. Remember, when an alarm is generated, program execution is halted.

```
N1 IF [#9NE#0] GOTO3 (TEST FOR VALUE IN F) ;
N2 #3000=11(NO FEED RATE) ;
N3 (CONTINUE) ;
```

Iteration/Looping (WHILE DO END)

Essential to all programming languages is the ability to execute a sequence of statements a given number of times or to loop through a sequence of statements until a condition is met. Traditional G coding allows this with the use of the L address. A subroutine can be executed any number of times by using the L address.

M98 P2000 L5;

This is limited since you cannot terminate execution of the subroutine on condition. Macros allow flexibility with the WHILE-DO-END construct. For example:

```
WHILE [<conditional expression>] DOn;
<statements>;
ENDn;
```

This executes the statements between DOn and ENDn as long as the conditional expression evaluates to True. The brackets in the expression are necessary. If the expression evaluates to False, then the block after ENDn is executed next. WHILE can be abbreviated to WH. The DOn-ENDn portion of the statement is a matched pair. The value of n is 1-3. This means that there can be no more than three nested loops per subroutine. A good example of how nesting of WHILE loops can be used is in defining a matrix.

```
#101= 3;
#102= 4;
G0 X#101 Y4. ;
F2.5;
WH [#101 GT 0] DO1;
#102= 4;
WH [#102 GT 0] DO2;
G81 X#101 Y#102 Z-0.5;
#102= #102 - 1;
END2;
#101= #101 - 1;
END1;
;
M30;
```

This program drills a 3 x 4 matrix hole pattern.

Although nesting of WHILE statements can only be up to three levels, there really is no limit since each subroutine can have up to three levels of nesting. If there is a need to nest to a level greater than 3, then the segment containing the three lowest levels of nesting can be made into a subroutine thus overcoming the limitation.

If two separate WHILE loops are in a subroutine, they can use the same nesting index. For example:

```
#3001=0 (WAIT 500 MILLISECONDS);
WH [#3001 LT 500] DO1;
END1;
```



<Other statements>

```
#3001=0 (WAIT 300 MILLISECONDS) ;  
WH [#3001 LT 300] DO1 ;  
END1 ;
```

You can use GOTO to jump out of a region encompassed by a DO-END, but you cannot use a GOTO to jump into it. Jumping around inside a DO-END region using a GOTO is allowed.

An infinite loop can be executed by eliminating the WHILE and expression. Thus,

```
DO1;  
<statements>  
END1;
```

Executes until the RESET key is pressed.

CAUTION! The following code can be confusing: WH [#1] D01;
END1;

In the this example, an alarm results indicating no "Then" was found; "Then" refers to the D01. Change D01 (zero) to DO1 (letter O).

G65 MACRO SUBROUTINE CALL

G65 is the command that calls a subroutine with the ability to pass arguments to it. The format follows.

G65 Pnnnn [Lnnnn] [*arguments*];

Anything italicized in square brackets is optional. The G65 command requires a P address corresponding to a program number currently in the control's memory. When the L address is used the macro call is repeated the specified number of times.

In Example 1, subroutine 1000 is called once without conditions passed to the subroutine. G65 calls are similar to, but not the same as, M98 calls. G65 calls can be nested up to 9 times, which means, program 1 can call program 2, program 2a nd call program 3 and program 3 can call program 4.

Example 1:	G65 P1000;	(Call subroutine 1000 as a macro)
	M30;	(Program stop)
	O1000;	(Macro Subroutine)
	...	
	M99;	(Return from Macro Subroutine)

In Example 2, subroutine 9010 is designed to drill a sequence of holes along a line whose slope is determined by the X and Y arguments that are passed to it in the G65 command line. The Z drill depth is passed as Z, the feed rate is passed as F, and the number of holes to be drilled is passed as T. The line of holes is drilled starting from the current tool position when the macro subroutine is called.



Example 2:

G00 G90 X1.0 Y1.0 Z.05 S1000 M03;	(Position tool)
G65 P9010 X.5 Y.25 Z.05 F10. T10;	(Call 9010)
G28;	
M30;	
O9010;	(Diagonal hole pattern)
F#9;	(F=Feed rate)
WHILE [#20 GT 0] DO1;	(Repeat T times)
G91 G81 Z#26;	(Drill To Z depth)
#20=#20-1;	(Decrement counter)
IF [#20 EQ 0] GOTO5;	(All holes drilled)
G00 X#24 Y#25;	(Move along slope)
N5 END1;	
M99;	(Return to caller)

Aliasing

Aliasing is a means of assigning a G-code or M-code to a G65 P##### sequence. For instance, in Example 2 it would be easier to write:

G06 X.5 Y.25 Z.05 F10. T10;

When aliasing, if you alias to a G-code you may pass variables with it. If you alias to an M-code, you may NOT pass variables with it.

Here, an unused G code has been substituted, G06 for G65 P9010. In order for the previous block to work the parameter associated with subroutine 9010 must be set to 06 (Parameter 91). Note that G00, G65, G66, and G67 cannot be aliased. All other codes between 1 and 255 can be used for aliasing.

Program numbers 9010 through 9019 are reserved for G code aliasing. The following table lists which HAAS parameters are reserved for macro subroutine aliasing.

Haas Parameter	O Code
91	9010
92	9011
93	9012
94	9013
95	9014
96	9015
97	9016
98	9017
99	9018
100	9019

G-Code Aliasing

Haas Parameter	M Macro Call
81	9000
82	9001
83	9002
84	9003
85	9004
86	9005
87	9006
88	9007
89	9008
90	9009

M-Code Aliasing

Setting an aliasing parameter to 0 disables aliasing for the associated subroutine. If an aliasing parameter is set to a G-code and the associated subroutine is not in memory, then an alarm will be given.

COMMUNICATION WITH EXTERNAL DEVICES - DPRNT[]

Macros allow additional capabilities to communicate with peripheral devices. One can do digitizing of parts, provide runtime inspection reports, or synchronize controls with user provided devices. The commands provided for this are POPEN, DPRNT[] and PCLOS.



Communication preparatory commands

POOPEN and PCLOS are not required on the Haas mill. It has been included so that programs from different controls can be sent to the Haas control.

Formatted output

The DPRNT statement allows the programmer to send formatted text to the serial port. Any text and any variable can be printed to the serial port. The form of the DPRNT statement is as follows:

```
DPRNT [<text> <#nnnn[wf]>... ] ;
```

DPRNT must be the only command in the block. In the previous example, <text> is any character from A to Z or the letters (+,-,/,* , and the space). When an asterisk is output, it is converted to a space. The <#nnnn[wf]> is a variable followed by a format. The variable number can be any macro variable. The format [wf] is required and consists of two digits within square brackets. Remember that macro variables are real numbers with a whole part and a fractional part. The first digit in the format designates the total places reserved in the output for the whole part. The second digit designates the total places reserved for the fractional part. The total places reserved for output cannot be equal to zero or greater than eight. Thus the following formats are illegal:

```
[00] [54] [45] [36] /* not legal formats */
```

A decimal point is printed out between the whole part and the fractional part. The fractional part is rounded to the least significant place. When zero places are reserved for the fractional part, then no decimal point is printed out. Trailing zeros are printed if there is a fractional part. At least one place is reserved for the whole part, even when a zero is used. If the value of the whole part has fewer digits than have been reserved, then leading spaces are output. If the value of the whole part has more digits than has been reserved, then the field is expanded so that these numbers are printed.

A carriage return is sent out after every DPRNT block.

DPRNT[] Examples

	<u>Code</u>	<u>Output</u>
N1	#1= 1.5436 ;	
N2	DPRNT[X#1[44]*Z#1[03]*T#1[40]] ;	X1.5436 Z 1.544 T 1
N3	DPRNT[***MEASURED*INSIDE*DIAMETER***] ;	MEASURED INSIDE DIAMETER
N4	DPRNT[] ;	(no text, only a carriage return)
N5	#1=123.456789 ;	
N6	DPRNT[X-#1[25]] ;	X-123.45679 ;

Execution

DPRNT statements are executed at block interpretation time. This means that the programmer must be careful about where the DPRNT statements appear in the program, particularly if the intent is to print out.

G103 is useful for limiting lookahead. If you wanted to limit lookahead interpretation to one block, you would include the following command at the beginning of your program: (This actually results in a two block lookahead.)

```
G103 P1;
```

To cancel the lookahead limit, change the command to G103 P0. G103 cannot be used when cutter compensation is active.



Editing

Improperly structured or improperly placed macro statements will generate an alarm. Be careful when editing expressions; brackets must be balanced.

The DPRNT[] function can be edited much like a comment. It can be deleted, moved as a whole item, or individual items within the bracket can be edited. Variable references and format expressions must be altered as a whole entity. If you wanted to change [24] to [44], place the cursor so that [24] is highlighted, enter [44] and press the write key. Remember, you can use the jog handle to maneuver through long DPRNT[] expressions.

Addresses with expressions can be somewhat confusing. In this case, the alphabetic address stands alone. For instance, the following block contains an address expression in X:

G1 G90 X [COS [90]] Y3.0; CORRECT

Here, the X and brackets stand-alone and are individually editable items. It is possible, through editing, to delete the entire expression and replace it with a floating-point constant.

G1 G90 X 0 Y3.0 ; WRONG

The above block will result in an alarm at runtime. The correct form looks as follows:

G1 G90 X0 Y3.0; CORRECT

Note that there is no space between the X and the Zero (0). REMEMBER when you see an alpha character standing alone it is an address expression.

FANUC-STYLE MACRO FEATURES NOT INCLUDED IN HAAS CNC CONTROL

This section lists the FANUC macro features that are not available on the Haas control.

M Aliasing Replace G65 Pnnnn with Mnn PROGS 9020-9029.	
G66	Modal call in every motion block
G66.1	Modal call in every motion block
G67	Modal cancel
M98	Aliasing, T code PROG 9000, VAR #149, enable bit
M98	Aliasing, S Code PROG 9029, VAR #147, enable bit
M98	Aliasing, B Code PROG 9028, VAR #146, enable bit
SKIP/N	N=1..9
#3007	Mirror image on flag each axis
#4201-#4320	Current block modal data
#5101-#5106	Current servo deviation

Names for Variables for Display Purposes

ATAN []/[]	Arctangent, FANUC version
BIN []	Conversion from BCD TO BIN
BCD []	Conversion from BIN TO BCD
FUP []	Truncate fraction ceiling
LN []	Natural logarithm
EXP []	Bade E Exponentiation
ADP []	Re-Scale variable to whole number
BPRNT []	

The following can be used as alternative methods for achieving the same results for a few of the unavailable FANUC macro features.

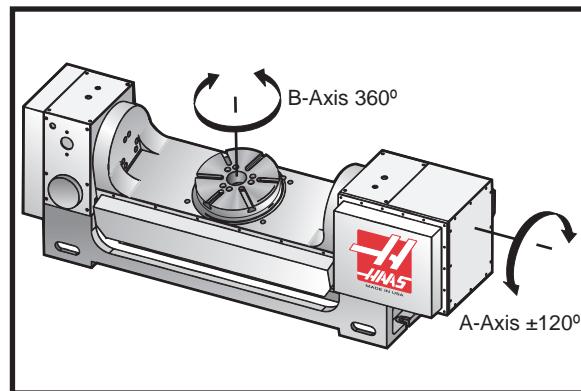
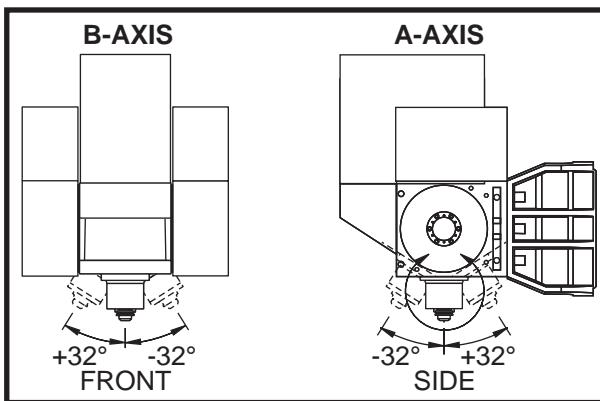
GOTO-nnnn

Searching for a block to jump in the negative direction, i.e. backwards through a program, is not necessary if you use unique N address codes.

A block search is made starting from the current block being interpreted. When the end of the program is reached, searching continues from the top of the program until the current block is encountered.



4TH AND 5TH AXIS PROGRAMMING

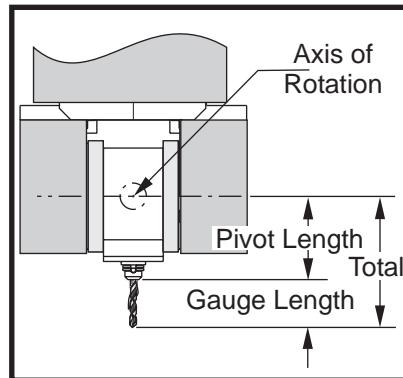


Axis motion on the VR-11 Mill and the Haas TRT 210

CREATING FIVE-AXIS PROGRAMS

Most five-axis programs are rather complex and should be written using a CAD/CAM package. It is necessary to determine the pivot length and gauge length of the machine, and input them into these programs.

Each machine has a specific pivot length. This is the distance from the spindle head's center of rotation to the bottom surface of the master tool holder. The pivot length can be found in Setting 116, and is also engraved into the master tool holder that is shipped with a 5-axes machine.



When setting up a program, it will be necessary to determine the gauge length for each tool. The gauge length is the distance from the bottom flange of the master tool holder to the tip of the tool. This distance can be calculated by setting a magnetic base indicator on the table, indicating the bottom surface of the master tool holder, and setting this point as Z0 in the control. Then, insert each tool, and calculate the distance from the tool tip to the Z0; this is the gauge length.

The total length is the distance from the spindle head center of rotation to the tip of the tool. It can be calculated by adding the gauge length and pivot length. This number is entered into the CAD/CAM program, which will use the value for its calculations.

Offsets

The work-offset display is found on the offset display by pushing the Page Up button. You can display and manually enter work offsets from here. The G54 through G59 or G110 through G129 offsets can be set by using the Part Zero Set button. Position the axes to the work zero point of your part. Using the cursor, select the proper axis and work number. Press the Part Zero Set button and the current machine position will be automatically stored in that address. This will work with only the work zero offsets display selected. Note that entering a nonzero Z work offset will interfere with the operation of an automatically entered tool length offset.



Work coordinate numbers are usually entered as positive numbers.

Work coordinates are entered into the table as a number only. To enter an X value of X2.00 into G54, cursor to the X column and enter 2.0.

Five-axis Programming Notes

Use a tight synchronization cut across resolution of geometry in the CAD/CAM system will allow smooth flowing contours and a more accurate part.

Positioning the machine to an approach vector should only be done at a safe distance above or to the side of the workpiece. When in the rapid mode, the axes will arrive at the programmed position at different times; the axis with shortest distance from target will arrive first, and longest distance last. A high feed rate will force the axes to arrive at the commanded position at the same time avoiding the possibility of a crash.

G Codes

Fifth-axis programming is not affected by the selection of inch (G20) or metric (G21); the **A** and **B** axes are always programmed in degrees.

G93 inverse time must be in effect for simultaneous 5-axis motion. In G93 mode, the maximum feed rate will include the combination of all axis motion in one block of code. The limit is set by the control and looks at encoder steps programmed for all axes in a block of code.

Limit your post processor (CAD/CAM software) if possible; the maximum speed in G93 mode is 32 degrees per minute. This will result in smoother motion, which may be necessary when fanning around tilted walls.

M Codes

Important! It is highly recommended that the A/B brakes be engaged when doing any non 5-axis motion. Cutting with the brakes off can cause excessive wear in the gear sets.

M10/M11 engages/disengages the A-axis brake

M12/M13 engages/disengages the B-axis brake

When in a 4 or 5 axis cut, the machine will pause between blocks. This pause is due to the A and/or B axis brakes releasing. To avoid this dwell and allow for smoother program execution, program an M11 and/or M13 just before the G93. The M-codes will disengage the brakes, resulting in a smoother motion and an uninterrupted flow of motion. Remember that if the brakes are never re-engaged, they will remain off indefinitely.

Settings

A number of settings are used to program the 4th and 5th axis. See settings 30, 34 and 48 for the 4th axis and 78, 79 and 80 for the 5th axis.

Setting 85 should be set to .0500 for 5-axis cutting. Settings lower than .0500 will move the machine closer to an exact stop and cause uneven motion.

G187 can also be used in the program to slow the axes down.

CAUTION! When cutting in 5-axis mode poor positioning and over-travel can occur if the tool length offset (H-code) is not canceled. To avoid this problem use G90 G40, H00 and G49 in your first blocks after a tool change. This problem can occur when mixing 3-axis and 5-axis programming; restarting a program or when starting a new job and the tool length offset is still in effect.

Feed Rates

A feed-rate must be commanded for each line of 4 and/or 5 axis code. Limit the feed-rate to less than 75IPM when drilling. The recommended feeds for finish machining in 3-axis work should not exceed 50 to 60 IPM with at least .0500" to .0750" stock remaining for the finish operation.

Rapid moves are not allowed; rapid motions, entering and exiting holes (full retract peck-drill cycle) are not supported.

When programming simultaneous 5-axis motion, less material allowance is required and higher feedrates may be permitted. Depending on finish allowance, length of cutter and type of profile being cut, higher feed rates may be possible. For example, when cutting mold lines or long flowing contours, feedrates may exceed 100 IPM.



Jogging the 4th and 5th Axis

All aspects of handle jogging for the fifth axis work as they do for the other axes. The exception is the method of selecting jog between axis A and axis B.

By default the '+A' and '-A' keys, when pressed, will select the **A** axis for jogging. The **B** axis can be selected for jogging by pressing the Shift button, and then pressing either the '+A' or '-A' key.

EC-300: Jog mode shows A1 and A2, use "A" to jog A1 and Shift "A" to jog A2.

EC-300 Pallet and 4th axis Operation

The rotary table in the machining area will always appear, and operate, as the A axis. The rotary axis on pallet 1 is referred to by "A1" and the other axis, on pallet 2, by "A2".

Operation examples:

To jog axis A1, enter "A1" and press "HAND JOG".

To key jog, use the +/-A jog buttons to jog the A1 axis and +/-B buttons to jog the A2 axis.

To zero return the A axis on pallet #2, enter "A2" press ZERO SINGL AXIS.

Mirroring Feature: If G101 is used to mirror the A axis, then mirroring is turned on for both A axes. When pallet #1 is in the machining area, A1-MIR will be displayed at the bottom of the screen. When pallet #2 is in the machine, A2-MIR will be displayed. The behavior of the Mirror Settings is different, if Setting 48 Mirror Image A-Axis is ON, only the A axis on pallet #1 is mirrored and the message A1-MIR is displayed.

If setting 80 (parameter 315, bit 20 MAP 4TH AXIS is a 1, the name for setting 80 is the same as that for Setting 48, i.e. Mirror Image A-Axis) is ON mirroring will be turned on for the A-axis on pallet #2. When pallet #2 is inside of the mill, A2-MIR will be displayed.

Crash Recovery Procedure

If the machine crashes while cutting a five-axis part, it can often be difficult to clear the tool away from the part due to the angles involved. **Do Not** immediately press the Tool Changer Restore key or turn the power off. To recover from a crash in which the spindle is stopped while the tool is still in a cut, retract the spindle using the Vector Jog feature. To do this press the letter "V" on the keypad, press "Handle Jog", and use the jog handle to move along that axis. This feature will allow motion along any axes determined by A and/or B axis.

The Vector Jog feature is intended to allow the operator to clear the cutting tool from the part in an extreme situation as a result of a crash or an alarm condition.

G28 is not available in the vector jog mode; it is only available for the X, Y, Z A and B when selecting single axis.

If there was a loss of power during a cut vector jog will not work as the control requires a reference position. Other means of clearing the tool from the part will be necessary.

If the tool is not in a cut when it is crashed, press the Tool Changer Restore button and answer the questions that appear on the screen. When Tool Changer Restore is pressed, the spindle head will move the A, B and Z-axes simultaneously, in order to retract the tool. If the tool is in a cut at an angle, it will crash when this key is pressed.

INSTALLING AN OPTIONAL FOURTH AXIS

When adding a rotary table to the Haas mill change settings 30 and 34 to the specific rotary table and part diameter currently used. **Warning:** Failure to match the correct brush or brushless rotary setting to the actual product being installed on the mill may cause motor damage. "B" in the settings denotes a brushless rotary product. Brushless indexers have two cables from table and two connectors at the mill control.

Parameters

In rare cases you may be required to modify some parameters to get a specific performance out of your indexer. Do not do this without a list of parameters to change. (If you did not get a list of special parameters with your indexer, then you do not need to change the default parameters—DO NOT CHANGE THEM. This will void your warranty.)



To change parameters for a forth or fifth axis indexer, proceed as follows: Press the E-stop switch and leave it in. You must then turn off the parameter lock (Setting 7). Go to the settings page by pressing the Setting button. Enter "7" and press the down arrow; this will jump to setting 7. With the cursor on Setting 7, use the right or left arrow button to select "Off" and press the Write button. The parameter lock is now off. Go to the parameter page and enter the parameter number of the parameter to change and press the down arrow button. Enter the new value for the new parameter and press the Write button. If necessary change the other parameters. Return to the Setting 7 and turn this setting back on. Reset the E-stop button. Home the indexer and verify that it operates properly by pressing Handle Jog and the "A" button. Jog the A axis using the jog handle, the indexer should move. Check for proper ratio by marking the table, rotate 360 degrees as viewed in the position page and check mark to be in the same location. If it is close (within 10 degrees) then the ratio is correct.

Initial Start-up

Turn on the mill (and servo control, if applicable) and home the indexer. All Haas indexers home in the clockwise direction as viewed from the front. If the indexer homes counter-clockwise, press E-stop and call the Haas service department.

INSTALLING AN OPTIONAL FIFTH AXIS

The fifth axis is installed in the same manner as the forth axis. Settings 78 and 79 govern the 5th axis, and the axis is jogged and commanded using the B address.

AUXILIARY AXIS

Besides the five directly controlled axes in this control, up to four additional external positioning axes may be added. These axes may be commanded directly from the program using the C, U, V, and W axis codes. Commands to these axes are only allowed in a G00 or G01 block. Connection of these axes is done through the second RS-232 port to one or more HAAS single axis controls. In the Setting page, Setting 38 is used to select the number of auxiliary axes. The machine position display will show the present position of these axes.

Only one auxiliary axis is moved at a time. If a feed (G01) is programmed, the feed rate programmed in the CNC is sent to the auxiliary control. A G00 motion of an auxiliary axis will move at its maximum feed rate. When the CNC control is waiting for an auxiliary axis motion to complete, the bottom of the screen will display "C FIN". A failure in RS-232 communication with the auxiliary axes may cause this display to pause indefinitely. The Reset button will terminate any "hung-up" auxiliary axis communication.

Emergency Stop or turning on Single Block are the only way to stop an auxiliary axis; Feed Hold or Reset will not stop the axis.

If you add one auxiliary axis, Setting 38, then the auxiliary axis will be designated C. If you add two auxiliary axes they will be designated C and U; etc.

Parameters

When interfacing to an auxiliary axis the Haas single axis servo control must have Parameter 21 set according to the following table.

Name in CNC:	Parameter 21:	Axis select:
C	6	Z
U	1	U
V	2	V
W	3	W

Multiple auxiliary axes must be daisy chained through the second RS-232 port as described in the auxiliary axis operator's manual.

Auxiliary axes can be jogged from the CNC front panel using the jog handle.



There are no work offsets for these axes so all commands are in the machine coordinate system. But if a displaced zero position has been entered into the HAAS servo control, that position will be used as zero. On power-up of the CNC, the auxiliary axes control will also be initialized and zero will be shifted by the value set in the single axis control. To set a displaced zero, you must jog the single axis control to a new zero position and then press and hold the Clear button on the single axis control.

Auxiliary axes communication is always seven data bits, even parity, two stop bits. The data rate is CNC Setting 54 and should be set to 4800. CNC Setting 50 must be set to XON/XOFF. Parameter 26 in the single axis control must be set to 5 for 4800 bit per second and Parameter 33 must be set to 1 for XON/XOFF. Parameter 12 in the single axis control should always be set to 3 or 4 to prevent circular wraparound. The cable connecting the CNC to the single-axis control must be a DB-25 cable (male lead on both ends) and must wire at least pins 1, 2, 3, and 7 directly from the second (lower) serial port of the Haas CNC to the upper connector of the Haas servo control.

DISABLEING THE AXES

If you have a fourth-axis rotary table or 5C indexer, or a fifth axis, it should be disabled from the Settings display when it is removed from the machine. Do not connect or disconnect any cables with the control on. If you do not disable the rotary axis setting when it is disconnected, you will get an alarm.



G CODES (PREPARATORY FUNCTIONS)

G codes are used to command specific actions for the machine, for example simple machine moves or drilling functions. They will also command more complex features from bolt hole circles to non-vertical machining.

G-codes are divided into groups. Each group of codes is commands for a specific subject. For example, Group 1 G-codes command point-to point moves of the machine axes, Group 7 are specific to the Cutter Compensation feature.

Each group has a dominant G-code; referred to as the **default** G-code. A default G-code means they are the one in each group the machine uses unless another G-code from the group is specified. For example programming an X, Y, Z move like this, X-2. Y-3. Z-4. will position the machine using G00. (Note that proper programming technique is to preface all moves with a G-code).

The default G-codes for each group are shown on the Current Commands screen in the upper right corner. If another G code from the group is commanded (active) then that G-code will be displayed on the Current Commands screen.

G-codes commands can be modal or non-modal. A **modal** G-code means that once commanded the G-code will stay in affect until the end of the program or until another G-code from the same group is commanded. A **non-modal** G-code is only affects the line it is in; the program line after will not be affected by the previous lines non-modal G-code.

Programming notes

The Group 00 codes are non-modal; the other groups are modal.

Group 01 G codes will cancel the Group 09 (canned cycles) codes, for example, if a canned cycle (G73 through G89) is active, the use of G00 or G01 will cancel the canned cycle.

Canned Cycles

A canned cycle is used to simplify programming of a part, for example common Z-axis repetitive operations, such as drilling, tapping, and boring. Canned Cycles will not work with Group 00 G codes. See the specific canned cycles G-code for more information and programming example.



G-CODE TABLE OF CONTENTS

G00 Rapid Motion Positioning (Group 01)	110
G01 Linear Interpolation Motion (Group 01).....	110
G02 CW / G03 CCW Circular Interpolation Motion (Group 01)	111
G04 Dwell (Group 00)	115
G09 Exact Stop (Group 00)	115
G10 Set Offsets (Group 00)	115
G12 Circular Pocket Milling CW / G13 Circular Pocket Milling CCW (Group 00)	115
G17 XY / G18 XZ / G19 YZ plane selection (Group 02)	117
G20 Select Inches / G21 Select Metric (Group 06)	118
G28 Return to Machine Zero Thru Optional G29 Reference Point (Group 00)	118
G29 Return From Reference Point (Group 00)	119
G31 Feed Until Skip (Group 00)	119
G35 Automatic Tool Diameter Measurement (Group 00)	119
G36 Automatic Work Offset Measurement (Group 00)	120
G37 Automatic Tool Offset Measurement (Group 00)	121
G40 Cutter Comp Cancel (Group 07)	122
G41 2D Cutter Compensation Left / G42 2D Cutter Comp. Right (Group 07)	122
G43 Tool Length Compensation + (Add) / G44 Tool Length Comp - (Subtract) (Group 08)	122
G47 Text Engraving (Group 00)	122
G49 G43/G44/G143 Cancel (Group 08)	124
G50 Cancel Scaling (Group 11)	124
G51 Scaling (Group 11)	124
G52 Set Work Coordinate System YASNAC (Group 00 or 12)	126
G53 Non-Modal Machine Coordinate Selection (Group 00)	126
G54-59 Select Work Coordinate System #1 - #6 (Group 12)	126
G60 Uni-Directional Positioning (Group 00)	126
G61 Exact Stop Mode (Group 15)	126
G64 G61 Cancel (Group 15)	126
G68 Rotation (Group 16)	126
G69 Cancel G68 Rotation (Group 16)	128
G70 Bolt Hole Circle (Group 00)	129
G71 Bolt Hole Arc (Group 00)	129
G72 Bolt Holes Along an Angle (Group 00)	129
G73 High-Speed Peck Drilling Canned Cycle (Group 09)	133
G74 Reverse Tap Canned Cycle (Group 09)	134
G76 Fine Boring Canned Cycle (Group 09)	134
G77 Back Bore Canned Cycle (Group 09).....	135
G80 Canned Cycle Cancel (Group 09).....	135
G81 Drill Canned Cycle (Group 09)	136
G82 Spot Drill Canned Cycle (Group 09)	136
G83 Normal Peck Drilling Canned Cycle (Group 09)	137
G84 Tapping Canned Cycle (Group 09)	139
G85 Boring Canned Cycle (Group 09)	140
G86 Bore and Stop Canned Cycle (Group 09)	140
G87 Bore In and Manual Retract Canned Cycle (Group 09)	140
G88 Bore In, Dwell, Manual Retract Canned Cycle (Group 09)	141



G89 Bore In, Dwell, Bore Out Canned Cycle (Group 09)	141
G90 Absolute Position Commands (Group 03)	142
G91 Incremental Position Commands (Group 03)	142
G92 Set Work Coordinate Systems Shift Value (Group 00)	142
G93 Inverse Time Feed Mode (Group 05)	143
G94 Feed Per Minute Mode (Group 05)	143
G95 Feed per Revolution (Group 05)	143
G98 Canned Cycle Initial Point Return (Group 10)	143
G99 Canned Cycle R Plane Return (Group 10)	143
G100 Cancel Mirror Image (Group 00).....	143
G101 Enable Mirror Image (Group 00).....	143
G102 Programmable Output to RS-232 (Group 00)	145
G103 Limit Block Buffering (Group 00)	146
G107 Cylindrical Mapping (Group 00)	146
G110-G129 Coordinate System #7-26 (Group 12)	147
G136 Automatic Work Offset Center Measurement (Group 00).....	148
G141 3D+ Cutter Compensation (Group 07)	149
G143 5-Axis Tool Length Compensation + (Group 08)	150
G150 General Purpose Pocket Milling (Group 00).....	150
G153 5-Axis High Speed Peck Drilling Canned Cycle (Group 09)	154
G154 Select Work Coordinates P1-P99 (Group 12).....	155
G155 5-Axis Reverse Tap Canned Cycle (Group 09)	156
G161 5-Axis Drill Canned Cycle (Group 09)	157
G162 5-Axis Spot Drill Canned Cycle (Group 09)	157
G163 5-Axis Normal Peck Drilling Canned Cycle (Group 09)	158
G164 5-Axis Tapping Canned Cycle (Group 09)	159
G165 5-Axis Boring Canned Cycle (Group 09)	160
G166 5-Axis Bore and Stop Canned Cycle (Group 09)	160
G169 5-Axis Bore and Dwell Canned Cycle (Group 09)	161
G174 CCW Non-Vertical Rigid Tap (Group 00)	161
G184 CW Non-Vertical Rigid Tap (Group 00)	161
G187 Setting the Smoothness Level (Group 00)	162
G188 Get Program From PST (Group 00)	162



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 move the machine's axis at the maximum speed. It is primarily used to quickly position the machine to a given point before each feed (cutting) command (All moves are done at full rapid speed). This G code is modal, so a block with G00 causes all following blocks to be rapid motion until another Group 01 code is specified.

Programming note: Generally, rapid motion will not be in a straight line. Each axis specified is moved at the same speed, but all axes will not necessarily complete their motions at the same time. The machine will wait until all motions are complete before starting the next command.

Incremental or absolute position commands (G90 or G91) will change how those axis motion values are interpreted. Setting 57 (Exact Stop Canned X-Y) can change how closely the machine waits for a precise stop before and after a rapid move.

G01 Linear Interpolation Motion (Group 01)

- F Feedrate 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
- ,R Radius of the circle
- ,C Distance from center of intersection where the chamfer begins

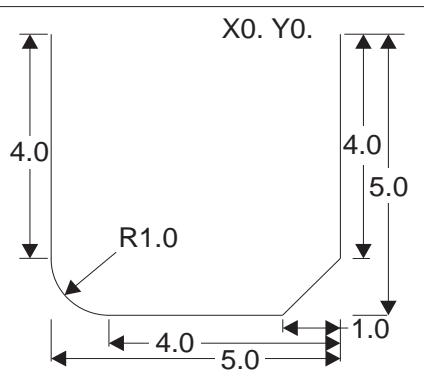
This G code moves the axes at a commanded feed rate. It is primarily used to cut the workpiece. A G01 feed can be a single axis move or a combination of the axes. The rate of axes movement is controlled by feedrate (F) value. This F value can be in units (inch or metric) per minute (G94) or per spindle revolution (95), or time to complete the motion (G93). The feedrate value (F) can be on the current program line, or a previous line (the control will always use the most recent F value until another F value is commanded).

G01 is a modal command, which means that it will stay in effect until canceled by a rapid command such as G00 or a circular motion command like G02 or G03.

Once a G01 is started all programmed axes will move and reach the destination at the same time. If an axis is not capable of the programmed feedrate the control will not proceed with the G01 command and an alarm (max feedrate exceeded) will be generated.

Corner Rounding and Chamfering Example

```
O1234 (Corner Rounding and Chamfering Example);
T1 M6;
G00 G90 G54 X0. Y0. S3000 M3;
G43 H01 Z0.1 M08;
G01 Z-0.5 F32.;
X0. Y-5. ,C1.;
X-5. Y-5. ,R1.;
X-5. Y0.;
G0 Z0.1 M09;
G53 Z0.;
G53 Y0.;
M30;
```





A chamfer block or a corner-rounding block can be automatically inserted between two linear interpolation blocks by specifying ,C (chamfering) or ,R (corner rounding). There must be a terminating linear interpolation block following the beginning block (a G04 pause may intervene). These two linear interpolation blocks specify a corner of intersection. If the beginning block specifies a C, the value following the C is the distance from the intersection to where the chamfer begins, and also the distance from the intersection to where the chamfer ends. If the beginning block specifies an R, the value following the R is the radius of a circle tangent to the corner at two points: the beginning of the corner-rounding arc and the endpoint of that arc. There can be consecutive blocks with chamfering or corner rounding specified. There must be movement on the two axes specified by the selected plane, whether the active plane is XY (G17), XZ (G18) or YZ (G19).

G02 CW / G03 CCW Circular Interpolation Motion (Group 01)

F	Feedrate 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
,R	Radius of corner rounding circle
,C	Distance from the center of intersection where chamfer begins.

These G codes are used to specify circular motion. Two axes are necessary to complete circular motion and the correct plane, G17-19, must be used. There are two methods of commanding a G02 or G03, the first is using the I, J, K addresses and the second is using the R address.

A chamfer or corner-rounding feature can be added to the program, by specifying ,C (chamfering) or ,R (corner rounding), as described in the G01 definition.

Using I, J, K addresses

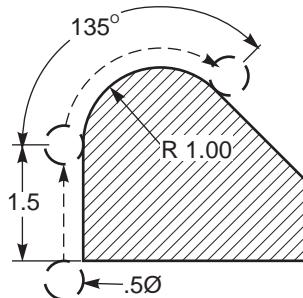
I, J and K address are used to locate the arc center in relation to the start point. In other words, the I, J, K addresses are the distances from the starting point to the center of the circle. Only the I, J or K specific to the selected plane are allowed (G17 uses IJ, G18 uses IK and G19 uses JK). The X, Y, and Z commands specify the end point of the arc. If the X, Y, or Z location for the selected plane is not specified, the endpoint of the arc is the same as the starting point for that axis.

To cut a full circle the I, J, K addresses must be used; using an R address will not work. To cut a full circle, do not specify an ending point (X, Y and Z); program I, J or K to define the center of the circle. For example: G02 I3.0 J4.0 (Assumes G17; XY plane)

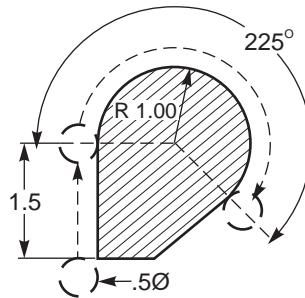
Using the R address

The R-value defines the distance from the starting point to the center of the circle. Use a positive R-value for radii of 180° or less, and a negative R-value for radii more than 180°.

Programming Examples



G90 G54 G00 X-0.25 Y-.25
G01 Y1.5 F12.
G02 X1.884 Y2.384 R1.25



G90 G54 G00 X-0.25 Y-0.25
G01 Y1.5 F12.
G02 X1.884 Y0.616 R-1.25



Corner Rounding and Chamfering example:

```
G00 X1. Y1.  
G01 X5. F10. ,C0.75  
Y2.5 ,R0.4  
G03 X8. Y5. R3. ,R0.8  
G01 X5. ,C0.8  
Y7. ,R1.  
X1. ,R1.  
Y1.  
G00 X0 Y0  
M30
```

Thread Milling

Thread milling uses a standard G02 or G03 move to create the circular move in X-Y and then adds a Z move on the same block to create the thread pitch. This will generate one turn of the thread; the multiple teeth of the cutter will generate the rest. A typical line of code follows:

```
N100 G02 I-1.0 Z-.05 F5. (generates a 1-inch radius for 20-pitch thread)
```

Thread Milling notes: Internal holes smaller than 3/8 inch may not be possible or practical. Always climb cut the cutter.

Use a G03 to cut I.D. threads or a G02 to cut O.D. threads. An **I.D.** right hand thread will move **up** in the Z-axis by the amount of one thread pitch. An **O.D.** right hand thread will move **down** in the Z-axis by the amount of one thread pitch. PITCH = 1/Threads per inch (**Example - 1.0 divided by 8 TPI = .125**)

Thread Milling Example:

This program will I.D. thread mill a 1.5 x 8 TPI hole using a .750 diameter x 1.0 thread hob.

To start, take the hole diameter (1.500). Subtract the cutter diameter .750 and then divide by 2.

$$(1.500 - .75) / 2 = .375$$

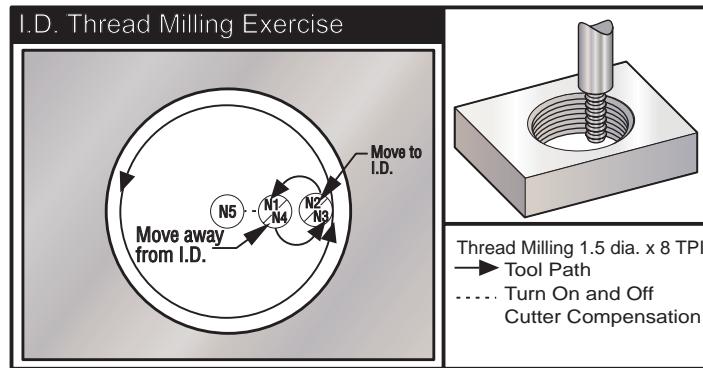
The result (.375) is the distance the cutter starts from the I.D. of the part.

After the initial positioning, the next step of the program is to turn on cutter compensation and move to the I.D. of the circle.

The next step is to program a complete circle (G02 or G03) with a Z-axis command of the amount of one full pitch of the thread (this is called "helical interpolation")

The last step is to move away from the I.D. of the circle and turn off cutter compensation

Cutter compensation cannot be turned off or on during an arc movement. A linear move must be made, either in the X or Y axis to move the tool to and from the diameter to cut. This move will be the maximum compensation amount that can be adjusted.



Thread Milling Example

Program Example

%
O2300

G00 G90 G54 X0 Y0 S400 M03
G43 H01 Z.1 M08
Z-.6
N1 G01 G41 D01 X.175 F25.
N2 G03 X.375 R.100 F7.
N3 G03 I-.375 Z-.475
N4 G03 X.175 R.100
N5 G01 G40 X0 Y0
G00 Z1.0 M09
G28 G91 Y0 Z0
M30
%

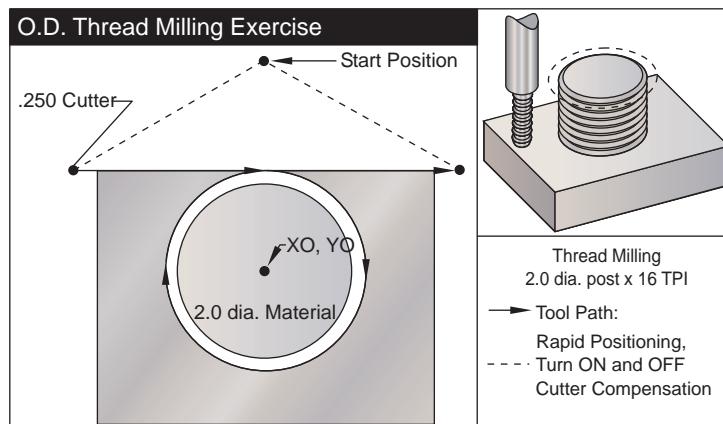
Description

(*Thread milling 1.5 diameter x 8 TPI*)
(*X0, Y0 is at the center of the hole*)
(*Z0 is at the top of the part – Using .5 thick material*)

(*Turn on Cutter Comp*)
(*Move to I.D. of bored hole*)
(*One full revolution with Z moving up .125*)
(*Move away from the new threads*)
(*Cancel Cutter Comp*)

NOTE: Maximum cutter compensation adjustability is .175.

O.D. Thread Milling



O.D. Thread Milling Example



Program Example

%
O2400

G00 G90 G54 X0 Y2.0 S2000 M03
G43 H01 Z.1 M08
Z-1.0
G41 D01 X-1.5 Y1.125
G01 X0. F15.
G02 J-1.125 Z-1.0625
G01 X1.5
G00 G40 X0 Y2.0
Z1.0 M09
G28 G91 Y0 Z0
M30
%

Description

(Thread milling a 2.0 diameter post x 16 TPI)
(X0,Y0 is at the center of the post)
(Z0 is at the top of the part – Post height is 1.125 inch)

(Turn on cutter compensation.)
(Linear move to the post.)
(Circular move; negative Z move.)
(Linear move away from the post.)
(Turn off cutter compensation.)

NOTE: A cutter compensation move can consist of any X or Y move from any position just as long as the move is greater than the amount being compensated for.

Single-Point Thread Milling Example

The program is for a 2.500 diameter hole, with a cutter diameter of .750" a radial value of .875 and a thread pitch of .0833 (12 TPI) and a part thickness of 1.0.

Program Example

%
O1000
T1 M06
G00 G90 G54 X0 Y0 S2500 M03
G43 H01 Z.1 M08
G01 Z-1.083 F35.
G41 X.275 DI
G3 X.875 I3 F15.
G91 G3 I-.875 Z.0833 L14
G90 G3 X.275 I-.300
G00 G90 Z1.0 M09
G1 G40 X0 Y0
G28 G91 Y0 Z0
M30
%

Description

(X0,Y0 is at the center of the hole, Z0 is at the top of the part
(Tool #1 is a .750 diameter single-point thread tool)

(Radial value)
(Multiply .0833 pitch x 14 passes = 1.1662 Z-axis move)

Helical Motion

Helical (spiral) motion is possible with G02 or G03 by programming the linear axis that is not in the selected plane. This third axis will be moved 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 feedrate.



G04 Dwell (Group 00)

P The 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 by the P code. For example G04 P10.0. This will delay the program for 10 seconds. Note the use of the decimal point G04 P10. is a dwell of 10 seconds; G04 P10 is a dwell of 10 microseconds.

G09 Exact Stop (Group 00)

The G09 code is used to specify a controlled axes stop. It only affects the block in which it is commanded; it is non-modal, it does not affect the following blocks. Machine moves will decelerate to the programmed point before another command is processed.

G10 Set Offsets (Group 00)

G10 allows the programmer to set offsets within the program. Using G10 replaces the manual entry of offsets (i.e. Tool length and diameter, and work coordinate offsets).

L □ Selects offset category.

L2 Work coordinate origin for G52 and G54–G59

L10 Length offset amount (for H code)

L1 or L11 Tool wear offset amount (for H code)

L12 Diameter offset amount (for D code)

L13 Diameter wear offset amount (for D code)

L20 Auxiliary work coordinate origin for G110–G129

P □ Selects a specific offset.

P1–P100 Used to reference D or H code offsets (L10–L13)

P0 G52 references work coordinate (L2)

P1–P6 G54–G59 references work coordinates (L2)

P1–P20 G110–G129 references auxiliary coordinates (L20)

P1–P99 G154 P1–P99 reference auxiliary coordinate (L20)

R Offset value or increment for length and diameter.

X Optional X-axis zero location.

Y Optional Y-axis zero location.

Z Optional Z-axis zero location.

A Optional A-axis zero location.

Programming Examples

G10 L2 P1 G91 X6.0 {Move coordinate G54 6.0 to the right};

G10 L20 P2 G90 X10. Y8. {Set work coordinate G111 to X10.0 ,Y8.0};

G10 L10 G90 P5 R2.5 {Set offset for Tool #5 to 2.5};

G10 L12 G90 P5 R.375 {Set diameter for Tool #5 to .375};

G10 L20 P50 G90 X10. Y20. {Set work coordinate G154 P50 to X10. Y20.}

G12 Circular Pocket Milling CW / G13 Circular Pocket Milling CCW (Group 00)

These two G codes are used to mill circular shapes. They are different only in which direction of rotation is used. Both G codes use the default XY circular plane (G17) and imply the use of G42 (cutter compensation) for G12 and G41 for G13. These two G-codes are non-modal.



*D Tool radius or diameter selection

I Radius of first circle (or finish if no K) The I value must be greater than the Tool Radius, but less than the K value.

K Radius of finished circle (if specified)

L Loop count for repeating deeper cuts

Q Radius increment, or stepover (must be used with K)

F Feedrate in inches (mm) per minute

Z Depth of cut or increment

*In order to get the programmed circle diameter, the control uses the selected D code tool size. If you want to program tool centerline select D0.

NOTE: If no cutter compensation is desired, a D00 must be specified. If no D is specified in the G12/G13 block, the last commanded D value will be used, even if it was previously canceled with a G40.

The tool must be positioned at the center of the circle using X and Y. To remove all the material within the circle, use I and Q values less than the tool diameter and a K value equal to the circle radius. To cut a circle radius only, use an I value set to the radius and no K or Q value.

%

O00098 (SAMPLE G12 AND G13)

(OFFSET D01 SET TO APPROX. TOOL SIZE)

(TOOL MUST BE MORE THAN Q IN DIAM.)

T1M06

G54G00G90X0Y0

(Move to center of G54)

G43Z0.1H01

S2000M03

G12I1.5F10.Z-1.2D01

(Finish pocket clockwise)

G00Z0.1

G55X0Y0

(Move to center of G55)

G12I0.3K1.5Q0.3F10.Z-1.2D01

(Rough and finish clockwise)

G00Z0.1

G56X0Y0

(Move to center of G56)

G13I1.5F10.Z-1.2D01

(Finish pocket counterclockwise)

G00Z0.1

G57X0Y0

(Move to center of G57)

G13I0.3K1.5Q0.3F10.Z-1.2D01

(Rough and finish counterclockwise)

G00Z0.1

G28

M30

%



These G codes assume the use of cutter compensation, so a G41 or G42 is not required in the program line. However, a D offset number, for cutter radius or diameter, is required to adjust the circle diameter.



The following programming examples show the G12 and G13 format, as well as the different ways these programs can be written.

Single Pass: Use **I** only.

Applications: One-pass counter boring; rough and finish pocketing of smaller holes, ID cutting of O-ring grooves.

Multiple Pass: Use **I**, **K**, and **Q**.

Applications: Multiple-pass counter boring; rough and finish pocketing of large holes with cutter overlap.

Multiple Z-Depth Pass: Using **I** only, or **I**, **K**, and **Q** (G91 and **L** may also be used).

Applications: Deep rough and finish pocketing.

The previous figures show the tool path during the pocket milling G-codes.

Example G13 multiple-pass using **I**, **K**, **Q**, **L**, and G91:

This program uses G91 and an **L** count of **4**, so this cycle will execute a total of four times. The **Z** depth increment is 0.500. This is multiplied by the **L** count, making the total depth of this hole 2.000.

The G91 and **L** count can also be used in a G13 “**I** only” line.

NOTE: If the geometry column of the control Offsets display has a value inserted, the G12/G13 will read the data, whether a D0 is present or not. To cancel cutter compensation insert a D00 in the program line, this will bypass the value in the Offsets geometry column.

Program Example

```
%  
O4000  
T1 M06  
G00 G90 G54 X0 Y0 S4000 M03  
G43 H01 Z.1 M08  
G01 Z0 F10.  
G13 G91 Z-.5 I.400 K2.0 Q.400 L4 D01 F20.  
G00 G90 Z1.0 M09  
G28 G91 Y0 Z0  
M30  
%
```

Description

(0.500 entered in the Radius/Diameter offset column)
(Tool #1 is a 0.500" diameter endmill)

G17 XY / G18 XZ / G19 YZ plane selection (Group 02)

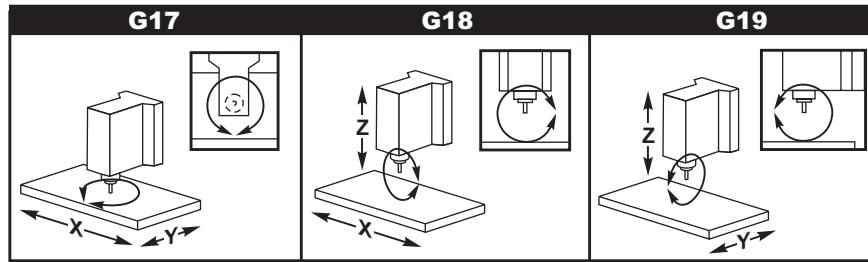
The face of the workpiece that will have a circular milling operation (G02, G03, G12, G13) done to it must have two of the three main axes (X, Y and Z) selected. One of three G codes is used to select the plane, G17 for XY, G18 for XZ, and G19 for YZ. Each is modal and will apply to all subsequent circular motions. The default plane selection is G17, which means that a circular motion in the XY plane can be programmed without selecting G17. Plane selection also applies to G12 and G13, circular pocket milling, which must always be in the XY plane.

If cutter radius compensation is selected (G41 or G42), you may only use the XY plane (G17) for circular motion.

G17 Defined - Circular motion with the operator looking down on the XY table from above. This defines the motion of the tool relative to the table.

G18 Defined - Circular motion is defined as the motion for the operator looking from the rear of the machine toward the front control panel.

G19 Defined - Circular motion is defined as the motion for the operator looking across the table from the side of the machine where the control panel is mounted.

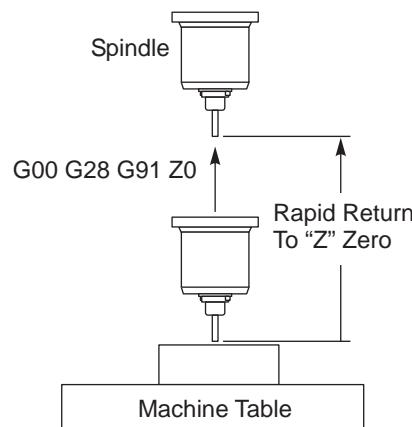


G20 Select Inches / G21 Select Metric (Group 06)

The G codes G20 (inch) and G21 (mm) codes are used to ensure that the inch/metric selection is set correctly for the program. Selection between inch and metric programming should be done using Setting 9.

G28 Return to Machine Zero Thru Optional G29 Reference Point (Group 00)

The G28 code is used to return all axes to machine zero, unless an axis (or axes) is specified, in which case only that axis (or axes) is returned to machine zero. G28 cancels tool length offsets for the following lines of code.



Example 1

Work Offset G54: Z = 2.0

Tool 2 Length: 12.0

Program segment:

```
G90 G54;  
G43 H02;  
G28 Z0.;  
G00 Z1.
```

The G28 block will move to machine coordinate Z = 14.0 before moving to Z = 0. The following block (G00 Z1.) will move to machine coordinate Z = 1.

Example 2

(same work and tool offsets as Example 1)

Program segment:

```
G54;  
G43 H02;  
G00 G91 G28 Z0
```

The G28 block will move directly to machine coordinate Z = 0 since incremental positioning is in effect.



G29 Return From Reference Point (Group 00)

The G29 code is used to move the axes to a specific position. The axes selected in this block are moved to the G29 reference point saved in G28, and then moved to the location specified in the G29 command.

G31 Feed Until Skip (Group 00)

(This G-code is optional and requires a probe)

F	Feedrate 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
B	Optional B-axis motion command

This G-code moves the axes to the programmed position. It applies only to the block in which G31 is specified. The specified move is started and continues until the position is reached or the probe receives a signal (skip signal). The control will beep when the end of travel is reached.

Do not use Cutter Compensation with a G31.

Use the assigned M-codes (for example M52 and M62), with a dwell, to turn the table probe on and off

For example:

M53
G04 P100
M63

Also see M75, M78 and M79.

G35 Automatic Tool Diameter Measurement (Group 00)

(This G-code is optional and requires a probe)

F	Feedrate in inches (mm) per minute
D	Tool diameter offset number
X	Optional X-axis command
Y	Optional Y-axis command

Automatic Tool Diameter Offset Measurement function (G35) is used to set the tool diameter (or radius) using two passes of the probe; one on each side of the tool. The first point is set with a G31 block using an M75, and the second point is set with the G35 block. The distance between these two points is set into the selected (non-zero) Dnnn offset. Setting 63 (Tool Probe Width) is used to reduce the measurement of the tool by the width of the tool probe.

This G-code moves the axes to the programmed position. The specified move is started and continues until the position is reached or the probe sends a signal (skip signal).

Notes:

Also see G31.

Use the assigned M-code (M52) to turn the table probe on.

Use the assigned M-code (M62) to turn the table probe off.

Also see M75, M78, and M79.

Do not use Cutter Compensation with a G35.

Turn on the spindle in reverse (M04), for a right handed cutter.



O1234 (G35)
M52
T1 M06
G00 G90 G54 X0 Y1.
G43 H01 Z0
G01 Z-1. F10.
M04 S200
G31 Y0.49 F5. M75
G01 Y1. F20.
Z0
Y-1.
Z-1.
G35 Y-0.49 D1 F5.
G01 Y-1. F20.
M62
G00 G53 Z0 M05
M30

G36 Automatic Work Offset Measurement (Group 00)

(This G-code is optional and requires a probe)

F	Feedrate in inches (mm) per minute
I	Optional offset distance along X-axis
J	Optional offset distance along Y-axis
K	Optional offset distance along Z-axis
X	Optional X-axis motion command
Y	Optional Y-axis motion command
Z	Optional Z-axis motion command

Automatic Work Offset Measurement (G36) is used to command a probe to set work fixture offsets. A G36 will feed the axes of the machine in an effort to probe the workpiece with a spindle mounted probe. The axis (axes) will move until a signal from the probe is received, or the travel limit is reached.

Tool offsets (G41, G42, G43, or G44) must not be active this function is preformed. The currently active work coordinate system is set for each axis programmed. The point where the skip signal is received becomes the zero position.

If an **I**, **J**, or **K** is specified, the appropriate axis work offset is shifted by the amount in the **I**, **J**, or **K** command. This allows the work offset to be shifted away from where the probe actually contacts the part.

Notes:

The points probed are offset by the values in Settings 59 through 62.

Use G91 incremental moves when using a G36.

Use the assigned M-codes (for example M53 and M63) with a dwell to turn the spindle probe on or off.

Example:

M53
G04 P100
M63



Program Example

```
O1234 (G36)
M53
G04 P100
M63
G00 G90 G54 X1. Y0
Z-18.
G91 G01 Z-1. F20.
G36 X-1. F10.
G90 G01 X1.
M53
G04 P100
M63
G00 G90 G53 Z0
M30
```

G37 Automatic Tool Offset Measurement (Group 00)

(This G-code is optional and requires a probe)

F Feedrate in inches (mm) per minute
H Tool offset number
Z Required Z-axis offset

Automatic Tool Length Offset Measurement (G37) is used to command a probe to set tool length offsets. A G37 will feed the Z-axis in an effort to probe a tool with a table mounted probe. The Z-axis will move until a signal from the probe is received, or the travel limit is reached. A non-zero H code and either G43 or G44 must be active. When the signal from the probe is received (skip signal) the Z position is used to set the specified tool offset (Hnnn). The resulting tool offset is the offset between the work zero point and the point where the probe is touched.

The coordinate system (G54-G59, G110-G129) and the tool length offsets (H01-H200) may be selected in this block or the previous block.

Notes:

Use the assigned M-code (M52) to turn the table probe on.
Use the assigned M-code (M62) to turn the table probe off.
Cutter Compensation may not be active during a skip function.
Also see M78 and M79.
Specify Z0 for no offset.

```
O1234 (G37)
T1 M06
M52
G00 G90 G110 X0 Y0
G00 G43 H1 Z5.
G37 H1 Z0. F30.
G00 G53 Z0
M62
M30
```



G40 Cutter Comp Cancel (Group 07)

G40 will cancel G41 or G42 cutter compensation.

G41 2D Cutter Compensation Left / G42 2D Cutter Comp. Right (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 D address must be programmed to select the correct tool radius or diameter offset. If the value in the selected offset is negative, cutter compensation will operate as though G42 (Cutter Comp Right.) was specified.

The right or left side of the programmed path is determined by looking at the tool as it moves away from you. If the tool needs to be on the left of the programmed path as it moves away from you, Use G41. If it needs to be on the right of the programmed path as it moves away from you, use G42.

Refer to the "Cutter Compensation" section for more information.

G43 Tool Length Compensation + (Add) / G44 Tool Length Comp - (Subtract) (Group 08)

A G43 code selects tool length compensation in the positive direction; the tool length in the offsets page is added to the commanded axis position. A G44 code selects tool length compensation in the negative direction; the tool length in the offsets page is subtracted from the commanded axis position. A non-zero H address must be entered to select the correct entry from the offsets page.

G47 Text Engraving (Group 00)

In order to use G47, the program must be using G90 (absolute) mode, and setting 29 (G91 non-modal) must be OFF.

E	Plunge feed rate (units/min)
F	Engraving feedrate (units/min)
I	Angle of rotation (-360. to +360.); default is 0
J	Height of text in inches (minimum = 0.001 inch); default is 1.0 inch
P	0 for literal string engraving 1 for sequential serial number engraving 32-126 for ASCII characters
R	Return plane
X	X start of engraving
Y	Y start of engraving
Z	Depth of cut

Sequential Serial Number Engraving

This method is used to engrave numbers on a series of parts, with the number being increased 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 (P1 (##) will limit the serial number to two digits etc.).

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

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 variable #599 is used to hold the initial serial number to be engraved. For example, when macro variable #599 is set to "1234,"

G47 P1 (####)

will produce:

1234

See the Macros section for more information.



Literal String Engraving

This method is used to engrave desired text on a part. The text should be in the form of a comment on the same line as the P0 statement. For example:

G47 P0 (ENGRAVE THIS)

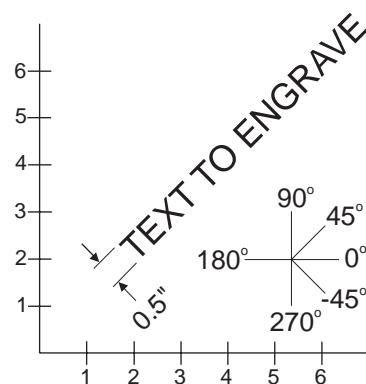
will produce

ENGRAVE THIS

Example

This example will create the figure shown.

G47 P0 X2.0 Y2.0 I45. J5 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° angle
J5	sets the text height to 0.5 inch
R.05	commands the cutter to retract to 0.05 inch (mm) above the cutting plane after engraving
Z-.005	selects a 0.005 inch (mm) deep cut
F15.0	selects an engraving feedrate of 15 units/min
E10.0	commands the cutter to plunge at a rate of 10 units/min

P values to engrave specific characters:

32	blank	41)	59	;	93]
33	!	42	*	60	<	94	^
34	"	43	+	61	=	95	-
35	#	44	,	62	>	96	\
36	\$	45	-	63	?	97-122	a-z
37	%	46	.	64	@	123	{
38	&	47	/	65-90	A-Z	124	
39	'	48-57	0-9	91	[125	}
40	(58	:	92	\	126	~



Example

To engrave "\$2.00" two lines of code are necessary. The first will be using a P36 to engrave the dollar sign (\$), and the second will use P0 (2.00). Note that the axes will need to be shifted between the first and second line of code in order to make a space between the dollar sign and the 2.

G49 G43/G44/G143 Cancel (Group 08)

This G code cancels tool length compensation.

Programming note: An H0, G28, M30, and Reset will also cancel tool length compensation.

G50 Cancel Scaling (Group 11)

G50 cancels the optional scaling feature. Any axis scaled by a previous G51 command is no longer in effect.

G51 Scaling (Group 11)

(This G-code is optional and requires Rotation and Scaling)

- X optional center of scaling for the X axis
- Y optional center of scaling for the Y axis
- Z optional center of scaling for the Z axis
- P optional scaling factor for all axes; three-place decimal from 0.001 to 8383.000.

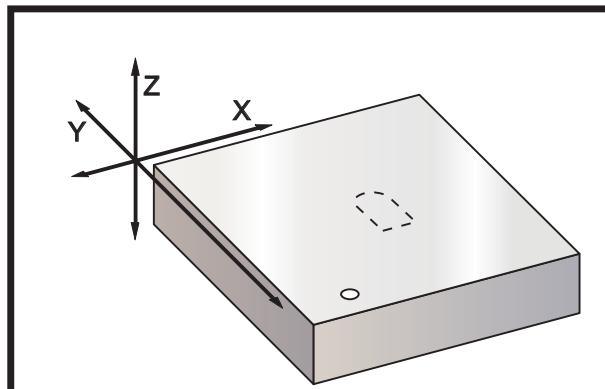
G51 [X...] [Y...] [Z...] [P...]

A scaling center is always used by the control in determining the scaled position. If any scaling center is not specified in the G51 command block, then the last commanded position is used as the scaling center.

When scaling (G51) is commanded, all X, Y, Z, I, J, K, or R values addressing machine motion are multiplied by a scaling factor and are offset relative to a scaling center.

G51 will affect all appropriate positioning values in the blocks following the G51 command. The X, Y and Z axes can be scaled using a P address, if a P address is not entered the Setting 71 scaling factor is used.

The following programs illustrate how scaling is performed when different scaling centers are used.

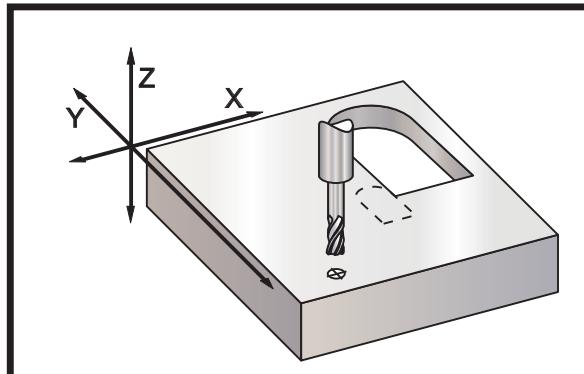


```
0001 (GOTHIC WINDOW) ;  
F20. S500 ;  
G00 X1. Y1. ;  
G01 X2. ;  
Y2. ;  
G03 X1. R0.5;  
G01 Y1. ;  
G00 X0 Y0 ;  
M99 ;
```

○ = Work coordinate origin
No Scaling

G51

The first example illustrates how the control uses the current work coordinate location as a scaling center. Here, it is X0 Y0 Z0.

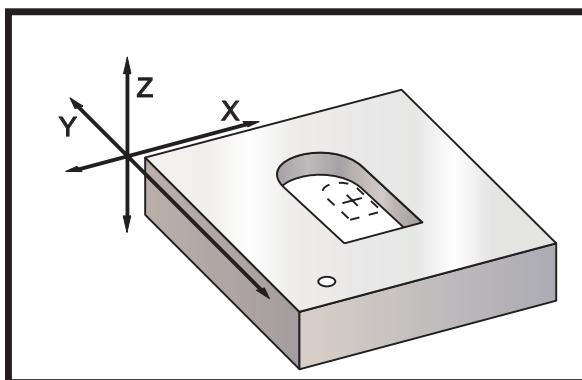


```
00010 ;  
G59 ;  
G00 G90 X0 Y0 Z0 ;  
G51 P2. (scaling center is X0 Y0 Z0) ;  
M98 P1 ;  
M30 ;
```

- = Work coordinate origin
- + = Center of scaling

G51 Scaling

The next example specifies the center of the window as the scaling center.

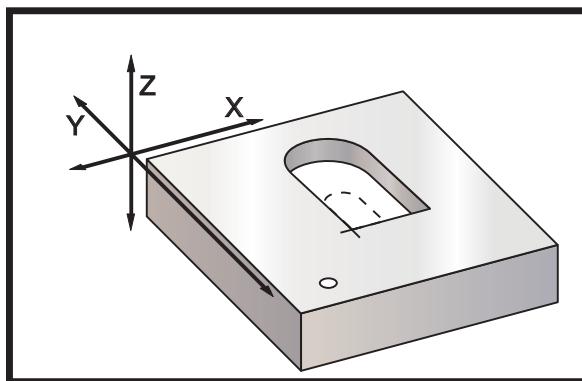


```
00011 ;  
G59 ;  
G00 G90 X0 Y0 Z0 ;  
G51 X1.5 Y1.5 P2. ;  
M98 P1 ;  
M30 ;
```

- = Work coordinate origin
- = Center of scaling
- + = Center of scaling

G51 Scaling

The last example illustrates how scaling can be placed at the edge of tool paths as if the part was being set against locating pins.



```
00011 ;  
G59 ;  
G00 G90 X0 Y0 Z0 ;  
G51 X1.0 Y1.0 P2 ;  
M98 P1 ;  
M30 ;
```

- = Work coordinate origin
- = Center of scaling
- + = Center of scaling

G51 Scaling

Programming notes:

Tool offsets and cutter compensation values are not affected by scaling.

Scaling does not affect canned cycle Z-axis movements such as clearance planes and incremental values.

The final results of scaling are rounded to the lowest fractional value of the variable being scaled.



G52 Set Work Coordinate System YASNAC (Group 00 or 12)

The G52 command works differently depending on the value of setting 33. Setting 33 selects the Fanuc, Haas, or Yasnac style of coordinates.

If Yasnac is selected, G52 is a group 12 G-code. G52 works the same as G54, G55, etc. All of the G52 values will not be set to zero (0) when powered on, reset is pressed, at the end of the program, or by an M30. When using a G92 (Set Work Coordinate Systems Shift Value), in Yasnac format, the X, Y, Z, A, and B values are subtracted from the current work position, and automatically entered into the G52 work offset.

If Fanuc is selected, G52 is a group 00 G-code. This is a global work coordinate shift. The values entered into the G52 line of the work offset page are added to all work offsets. All of the G52 values in the work offset page will be set to zero (0) when powered on, reset is pressed, changing modes, at the end of the program, by an M30, G92 or a G52 X0 Y0 Z0 A0 B0. When using a G92 (Set Work Coordinate Systems Shift Value), in Fanuc format, the current position in the current work coordinate system is shifted by the values of G92 (X, Y, Z, A, and B). The values of the G92 work offset are the difference between the current work offset and the shifted amount commanded by G92.

If Haas is selected, G52 is a group 00 G-code. This is a global work coordinate shift. The values entered into the G52 line of the work offset page are added to all work offsets. All of the G52 values will be set to zero (0) by a G92. When using a G92 (Set Work Coordinate Systems Shift Value), in Haas format, the current position in the current work coordinate system is shifted by the values of G92 (X, Y, Z, A, and B). The values of the G92 work offset are the difference between the current work offset and the shifted amount commanded by G92 (Set Work Coordinate Systems Shift Value).

G53 Non-Modal Machine Coordinate Selection (Group 00)

This code temporarily cancels work coordinate offsets and uses the machine coordinate system. In the machine coordinate system, the zero point for each axis is the position where the machine goes when a Zero Return is performed. G53 will revert to this system for the block it is commanded in.

G54-59 Select Work Coordinate System #1 - #6 (Group 12)

These codes select one of the six user coordinate system. All future references to axes positions will be interpreted using the new (G54 G59) coordinate system.

G60 Uni-Directional Positioning (Group 00)

This G code is used to provide positioning only from the positive direction. It is not recommended for use with this control. It is provided only for compatibility with older systems. It is non-modal, so does not affect the blocks that follow it. Also see Setting 35.

G61 Exact Stop Mode (Group 15)

The G61 code is used to specify an exact stop. It is modal; therefore, it affects the blocks that follow it. The machine axes will come to an exact stop at the end of each commanded move.

G64 G61 Cancel (Group 15)

The G64 code is used to cancel exact stop (G61).

G68 Rotation (Group 16)

(This G-code is optional and requires Rotation and Scaling.)

G17, G18, G19 optional plane of rotation, default is current

- A optional center of rotation for the first axis of the selected plane
- B optional center of rotation for the second axis of the selected plane
- R optional angle of rotation specified in degrees
Three-place decimal -360.000 to 360.000.

A G17, 18 or 19 must be used before the G68 command to establish the axis plane being rotated. For example:

G17 G68 Annn Bnnn Rnnn;

A and B correspond to the axes of the current plane; for the G17 example A is the X-axis and B is the Y-axis.



A center of rotation is always used by the control to determine the positional values passed to the control after rotation. If any axis center of rotation is not specified, then the current location is used as the center of rotation.

When rotation (G68) is commanded, all X, Y, Z, I, J, and K values are rotated through a specified angle **R** using a center of rotation.

G68 will affect all appropriate positional values in the blocks following the G68 command. Values in the line containing G68 are not rotated. Only the values in the plane of rotation are rotated, therefore, if G17 is the current plane of rotation, only X and Y values are affected.

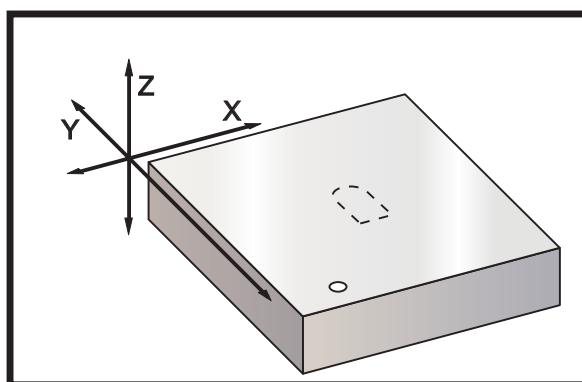
Entering a positive number (angle) for the R address will rotate the feature counterclockwise.

If the angle of rotation (R) is not entered, then the angle of rotation is taken from Setting 72.

In G91 mode (incremental) with Setting 73 ON, the rotation angle is changed by the value in R. In other words, each G68 command will change the rotation angle by the value specified in R.

The rotational angle is set to zero at the beginning of the program, or it can be set to a specific angle using a G68 in G90 mode.

The following examples illustrate rotation using G68.

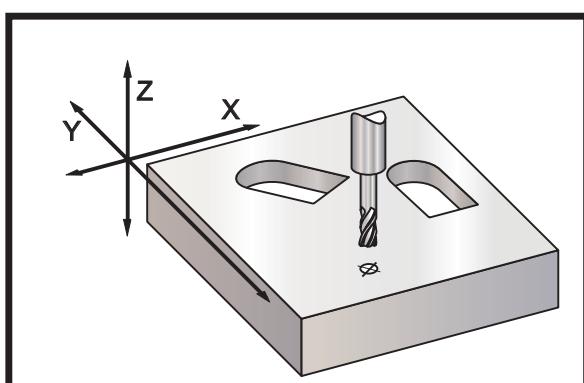


```
0001 (GOTHIC WINDOW) ;  
F20, S500 ;  
G00 X1. Y1. ;  
G01 X2. ;  
Y2. ;  
G03 X1. R0.5  
G01 Y1. ;  
M99 ;
```

○ = Work coordinate origin
No Rotation

G68

The first example illustrates how the control uses the current work coordinate location as a rotation center (X0 Y0 Z0).

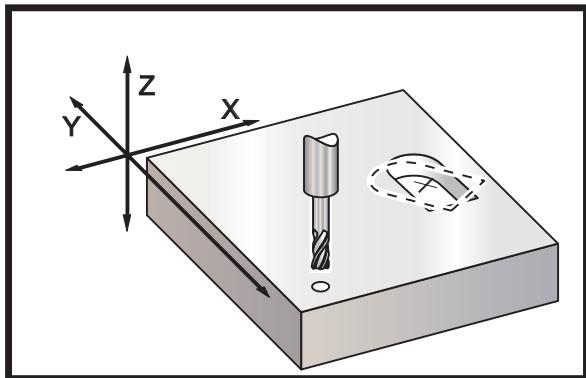


```
00002 ;  
G59 ;  
G00 G90 X0 Y0 Z0 ;  
M98 P1 ;  
G90 G00 X0 Y0 ; (Last Commanded Position)  
G68 R60. ;  
M98 P1 ;  
G69 G90 G00 X0 Y0 ;  
M30 ;
```

○ = Work coordinate origin
+ = Center of rotation

G68 Rotation

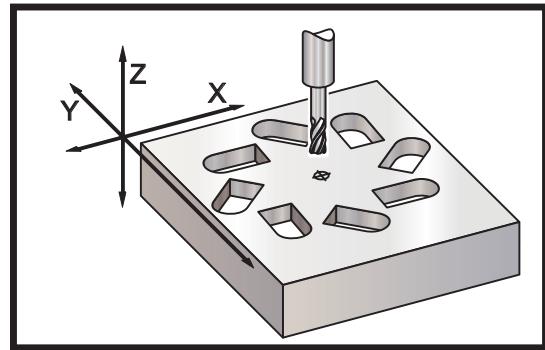
The next example specifies the center of the window as the rotation center.



G68 Rotation

```
00003 ;  
G59 ;  
G00 G90 X0 Y0 Z0 ;  
M98 P1 ;  
G00 G90 X0 Y0 Z0 ;  
G68 X1.5 Y1.5 R60. ;  
M98 P1 ;  
G69 G90 G00 X0 Y0 ;  
M30 ;
```

○ = Work coordinate origin
+ = Center of rotation



G68 Rotation

```
00004 ;  
G59 ;  
G00 G90 X0 Y0 Z0 ;  
M98 P10 L8 (SUBROUTINE 00010) ;  
M30 ;
```

```
00010 ;  
G91 G68 R45. ;  
G90 M98 P1 ;  
G90 G00 X0 Y0 ;  
M99 ;
```

○ = Work coordinate origin
+ = Center of rotation

Do not change the plane of rotation while G68 is in effect.

Rotation with Scaling

If scaling and rotation are used simultaneously, it is recommended that scaling be turned on prior to rotation, and that separate blocks be used. Use the following template when doing this.

```
G51 ..... (SCALING) ;  
...  
G68 ..... (ROTATION) ;  
.  
. program  
. .  
G69 ..... (ROTATION OFF) ;  
...  
G50 ..... (SCALING OFF) ;
```

Rotation with Cutter Compensation

Cutter compensation should be turned on after the rotation command is issued. Compensation should also be turned off prior to turning rotation off.

G69 Cancel G68 Rotation (Group 16)

(This G-code is optional and requires Rotation and Scaling.)

G69 cancels any rotation specified previously. Work Coordinate System Selection (G52, G53, G54-59).



G70 Bolt Hole Circle (Group 00)

- I Radius (+CCW / -CW)
- J Starting angle (0 to 360.0 degrees CCW from horizontal; or 3 o'clock position)
- L Number of holes evenly spaced around the circle

This non-modal G code must be used with one of the canned cycles G73, G74, G76, G77, or G81-G89. A canned cycle must be active so that at each position, a drill or tap function is performed.

G71 Bolt Hole Arc (Group 00)

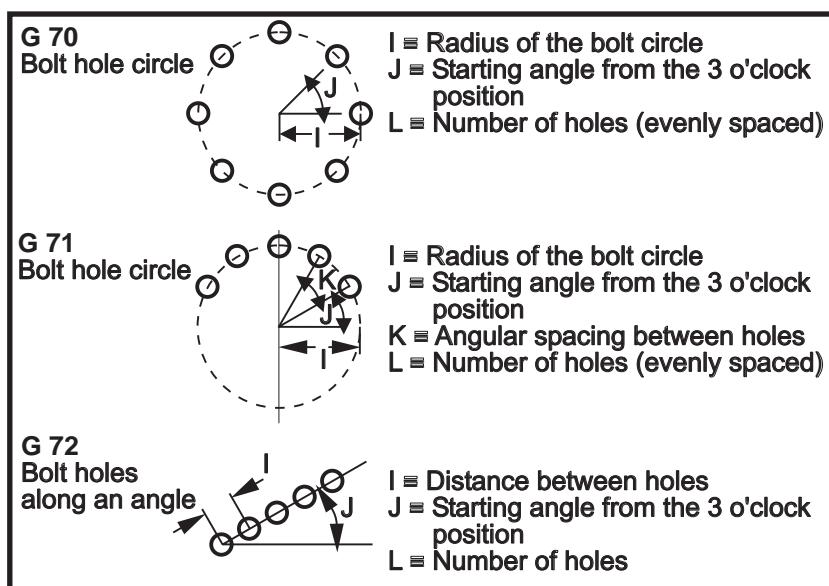
- I Radius (+CCW / -CW)
- J Starting angle (degrees CCW from horizontal)
- K Angular spacing of holes (+ or -)
- L Number of holes
- ,R Radius of the circle
- ,C Distance from center of intersection where the chamfer begins

This non-modal G code is similar to G70 except that it is not limited to a complete circle. G71 belongs to Group 00 and thus is non-modal. A canned cycle must be active so that at each position, a drill or tap function is performed.

G72 Bolt Holes Along an Angle (Group 00)

- I Distance between holes (+CCW / -CW)
- J Angle of line (degrees CCW from horizontal)
- L Number of holes
- ,R Radius of the circle
- ,C Distance from center of intersection where the chamfer begins

This non-modal G code drills "L" number of holes in a straight line at the specified angle. It operates similarly to G70. For a G72 to work correctly, a canned cycle must be active so that at each position, a drill or tap function is performed.



Rules For Bolt Pattern Canned Cycles:

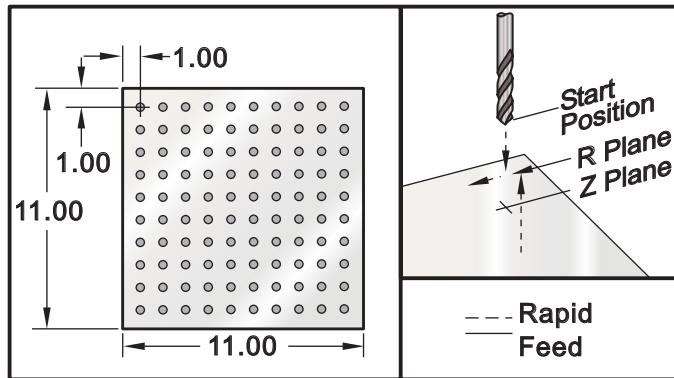
1. The tool must be placed at the center of the bolt pattern before the canned cycle execution. The center is usually X0, Y0.
2. The **J** code is the angular starting position and is always 0 to 360 degrees counterclockwise from the three o'clock position.

Looping Canned Cycles

The following is an example of a program using a drilling canned cycle that is incrementally looped.



NOTE: The sequence of drilling used here is designed to save time and to follow the shortest path from hole to hole.



G81 Drilling Canned Cycle (Incremental) and Grid Plate for Multiple-Fixture Subprogram

Program Example

Description

```
%  
O3400  
T1 M06  
G00 G90 G54 X1.0 Y-1.0 S2500 M03  
(Drilling grid plate)  
G43 H01 Z.1 M08  
G81 Z-1.5 F15. R.1  
G91 X1.0 L9  
G90 Y-2.0  
(Or stay in G91 and repeat Y-1.0)  
G91 X-1.0 L9  
G90 Y-3.0  
G91 X1.0 L9  
G90 Y-4.0  
G91 X-1.0 L9  
G90 Y-5.0  
G91 X1.0 L9  
G90 Y-6.0  
G91 X-1.0 L9  
G90 Y-7.0  
G91 X1.0 L9  
G90 Y-8.0  
G91 X-1.0 L9  
G90 Y-9.0  
G91 X1.0 L9  
G90 Y-10.0  
G91 X-1.0 L9  
G00 G90 G80 Z1.0 M09  
G28 G91 Y0 Z0  
M30  
%
```

Modifying Canned Cycles

In this section we will cover canned cycles that have to be customized in order to make the programming of difficult parts easier.

Using G98 and G99 to clear clamps – For example, A square part being held to the table with one-inch tall table clamps. A program needs to be written to clear the table clamps.



<u>Program Example</u>	<u>Description</u>
%	
O4500	
T1 M06	
G00 G90 G54 X1.0 Y-1.0 S3500 M03	
G43 H01 Z1.125 M08	
G81 G99 Z-1.500 R.05 F20.	
X2.0 G98	(Will return to starting point after executing cycle)
X6.0 G99	(Will return to reference plane after executing cycle)
X8.0	
X10.0	
X12.0 G98	
X16.0 G99	
X18.0 G98	
G00 G80 Z2.0 M09	
G28 G91 Y0 Z0	
M30	
%	

X, Y Plane Obstacle Avoidance In A Canned Cycle:

There is also a way to avoid an obstacle in the X, Y plane during a canned cycle by placing an L0 in a canned cycle line, we can tell the control to make an X, Y move without executing the Z-axis canned operation.

For example, we have a six-inch square aluminum block, with a one-inch by one-inch deep flange on each side. The print calls for two holes centered on each side of the flange. We need to write a program to avoid each of the corners on the block.

<u>Program Example</u>	<u>Description</u>
%	
O4600	(X0,Y0 is at the top left corner, Z0 is at the top of the part)
T1 M06	
G00 G90 G54 X2.0 Y-.5 S3500 M03	
G43 H01 Z-.9 M08	
G81 Z-2.0 R-.9 F15.	
X4.0	
X5.5 L0	(angular corner avoidance)
Y-2.0	
Y-4.0	
Y-5.5 L0	
X4.0	
X2.0	
X.5 L0	
Y-4.0	
Y-2.0	
G00 G80 Z1.0 M09	
G28 G91 Y0 Z0	
M30	
%	



CANNED CYCLES

Introduction

Canned cycles are used to simplify programming. They are used for repetitive operations, such as drilling, tapping, and boring. The canned cycle is executed every time an X and/or Y-axis motion is programmed.

Using Canned Cycles

The positioning of a canned cycle in the X and/or Y-axes can be done in either absolute (G90) or incremental (G91). Incremental (G91) motion in a canned cycle is often useful with a loop count (Lnn) which will repeat the canned cycle operation that many times with each incremental X or Y move for the canned cycle.

Example:

```
G81 G99 Z-0.5 R0.1 F6.5 (This will drill one hole at the present location)  
G91 X-0.5625 L9 (This will drill 9 more holes .5625 equally spaced in the minus direction)
```

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 the canned cycle will vary according to whether incremental (G91) or absolute (G90) positioning is active. Incremental motion in a canned cycle is often useful as a loop (**L**) count as it can be used to repeat the operation with an incremental **X** or **Y** move between each cycle.

Example:

```
X1.25 Y-0.75 (center location of bolt hole pattern)  
G81 G99 Z-0.5 R0.1 F6.5 L0 (L0 on the G81 line will not drill a hole in the bolt hole circle)  
G70 I0.75 J10. L6 (6-hole bolt hole circle)
```

Once a canned cycle is commanded, that operation is done at every X-Y position listed in a block. Some of the canned cycle numerical values can 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 XY commands, the XY move is done and all subsequent canned cycles are performed with the new **R** or **Z** value.

The positioning of the X and Y-axes prior to a canned cycle is done with rapid moves.

G98 and G99 change the way the canned cycles operate. When G98 is active, the Z-axis will return to the initial start plane at the completion of each hole in the canned cycle. This allows for positioning up and around areas of the part and/or clamps and fixtures.

When G99 is active, the Z-axis returns to the **R** (rapid) plane after each hole in the canned cycle for clearance to the next XY location. Changes to the G98/G99 selection can also be made after the canned cycle is commanded, which will affect all later canned cycles.

A **P** address is an optional command for some of the canned cycles. This is a programmed pause at the bottom hole to help break chips, provide a smoother finish and relieve any tool pressure to hold closer tolerance. Note that if a value for **P** is entered for one canned cycle it will be used in others unless cancelled (G00, G01, G80 or the Reset button).

An **S** (spindle speed) command must be defined in, or before the G-code line of code.

Tapping in a canned cycle needs a feed rate calculated. The Feed Formula is:
Spindle speed divided by Threads per inch of the tap = Feedrate in inches per minute

Canned Cycles also benefit from the use of Setting 57. Turning this setting ON will perform an exact stop between rapids. This is useful to avoid nicking the part at the bottom of the hole.

NOTE: The **Z**, **R**, and **F** addresses are required data for all canned cycles.

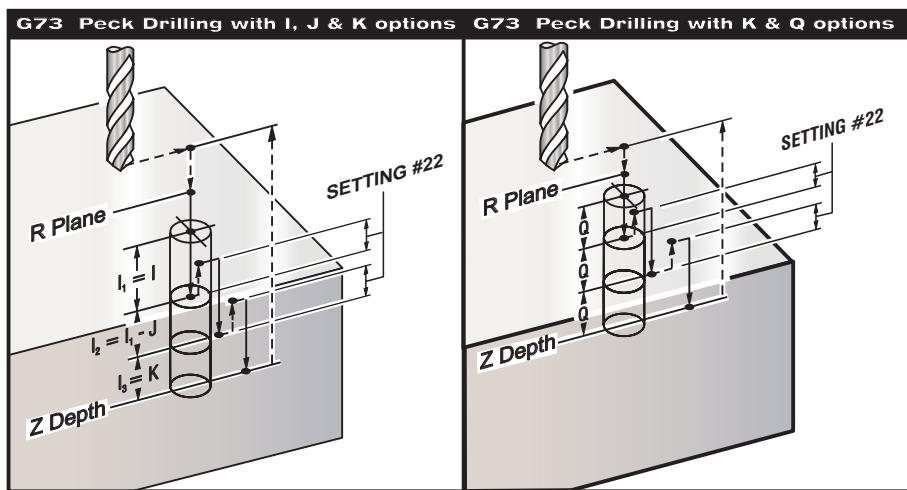
Cancelling a Canned Cycle

The G80 code is used to cancel all canned cycles; note that a G00 or G01 code will also cancel a canned cycle. Once selected, a canned cycle is active until canceled with G80, G00 or G01.



G73 High-Speed Peck Drilling Canned Cycle (Group 09)

- F Feedrate in inches (mm) per minute
- I First cut depth
- J Amount to reduce cutting depth for pass
- K Minimum depth of cut (The control will calculate the number of pecks)
- L Number of repeats (Number of holes to drill) if G91 (Incremental Mode) is used
- P Pause at the bottom of the hole (in seconds)
- Q Cut Depth (always incremental)
- R Position of the R plane (Distance above part surface)
- X X-axis location of hole
- Y Y-axis location of hole
- Z Position of the Z-axis at the bottom of hole
- ,R Radius of the circle
- ,C Distance from center of intersection where the chamfer begins

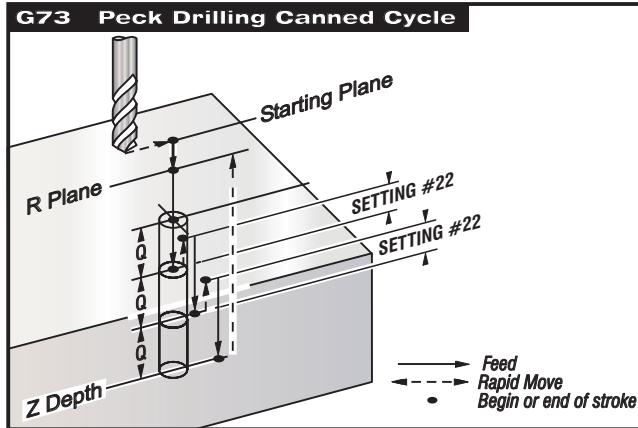


I, J, K, and Q are always positive numbers.

There are two methods to program a G73; first using the I, J, K addresses and the second using the K and Q addresses.

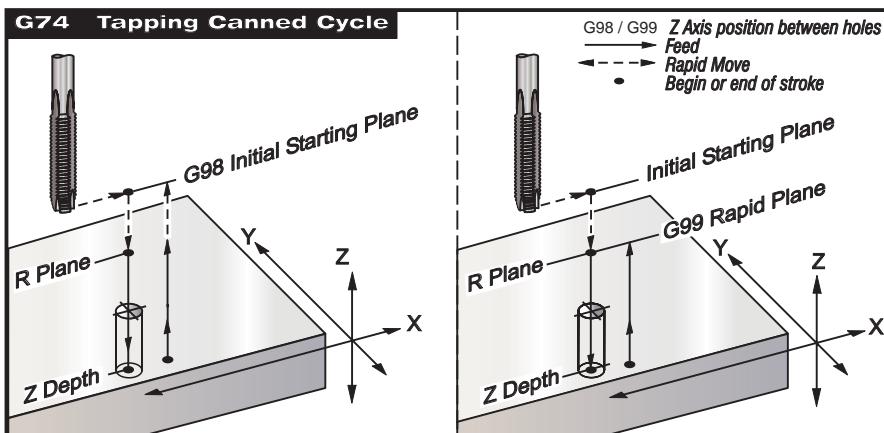
If **I, J, and K** are specified, The first pass will cut in by the value **I**, each succeeding cut will be reduced by the value of **J**, and the minimum cutting depth is **K**. If **P** is specified, the tool will pause at the bottom of the hole for that amount of time.

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 the number of passes totals up to the **K** amount.



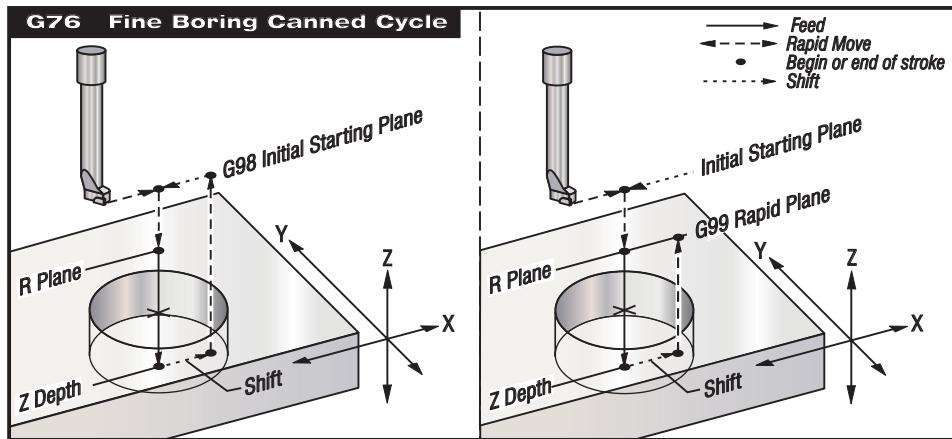
G74 Reverse Tap Canned Cycle (Group 09)

- F Feedrate in inches (or mm) per minute (use the formula, described in the canned cycle introduction, to calculate feed rate and spindle speed)
- J Retract Multiple (How fast to retract - see Setting 130)
- L Number of repeats (How many holes to tap) if G91 (Incremental Mode) is used
- R Position of the R plane (position above the part) where tapping starts
- X X-axis location of hole
- Y Y-axis location of hole
- Z Position of the Z-axis at the bottom of hole



G76 Fine Boring Canned Cycle (Group 09)

- F Feedrate in inches (or mm) per minute
- I Shift value along the X-axis before retracting, if Q is not specified
- J Shift value along the Y-axis before retracting, if Q is not specified
- L Number of holes to bore if G91 (Incremental Mode) is used
- P The dwell time at the bottom of the hole
- Q The shift value, always incremental
- R Position of the R plane (position above the part)
- X X-axis location of hole
- Y Y-axis location of hole
- Z Position of the Z-axis at the bottom of hole

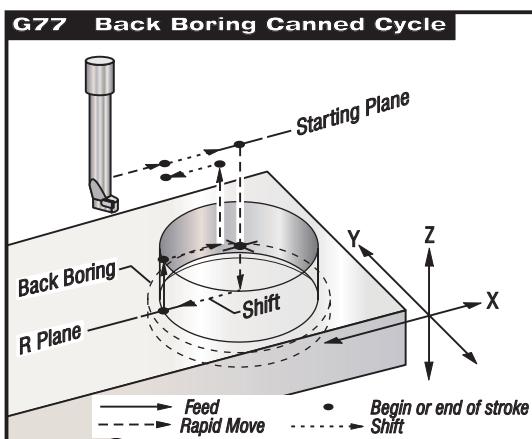


In addition to boring the hole, this cycle will shift the **X** and/or **Y** axis prior to retracting in order to clear the tool while exiting the part. If Q is used Setting 27 determines the shift direction. If Q is not specified, the optional I and J values are used to determine the shift direction and distance.

G77 Back Bore Canned Cycle (Group 09)

- F Feedrate in inches (or mm) per minute
- I Shift value along the X-axis before retracting, if Q is not specified
- J Shift value along the Y-axis before retracting, if Q is not specified
- L Number of holes to bore if G91 (Incremental Mode) is used
- Q The shift value, always incremental
- R Position of the R plane (position above the part)
- X X-axis location of hole
- Y Y-axis location of hole
- Z Position of the Z-axis at the bottom of hole

In addition to boring the hole, this cycle will shift the **X** and/or **Y** axis prior to and after cutting in order to clear the tool while entering and exiting the part (see G76 for an example of a shift move). Setting 27 determines the shift direction. 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)

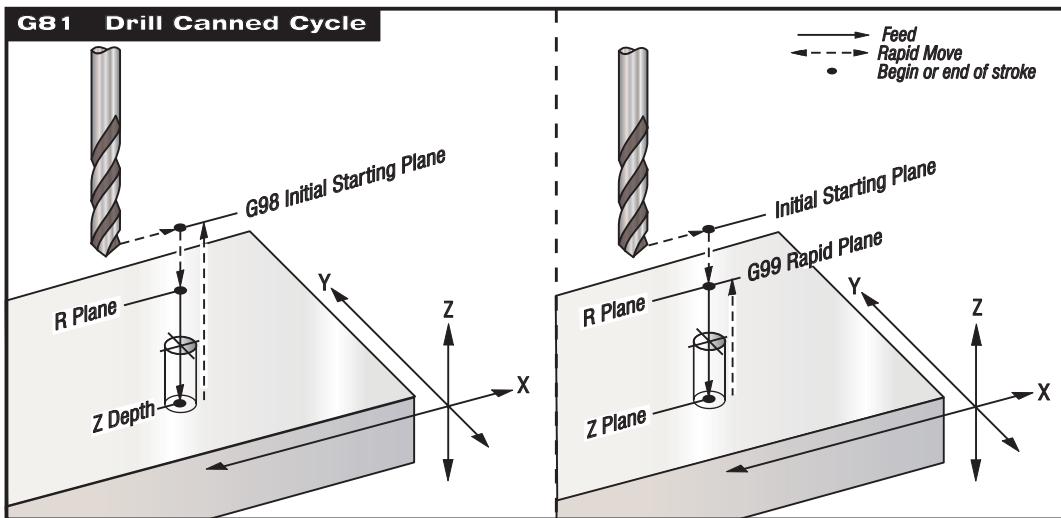
This G code deactivates all canned cycles until a new one is selected. Note that use of G00 or G01 will also cancel a canned cycle.



G81 Drill Canned Cycle (Group 09)

- F Feedrate in inches (or mm) per minute
- L Number of holes to drill if G91 (Incremental Mode) is used
- R Position of the R plane (position above the part)
- X X-axis motion command
- Y Y-axis motion command
- Z Position of the Z-axis at the bottom of hole

NOTE: The X and Y addresses, in most cases, are the location of the first hole to be drilled.



Program Example

The following is a program to drill through an aluminum plate:

```
T1 M06  
G00 G90 G54 X1.125 Y-1.875 S4500 M03  
G43 H01 Z0.1  
G81 G99 Z-0.35 R0.1 F27.  
X2.0  
X3.0 Y-3.0  
X4.0 Y-5.625  
X5.250 Y-1.375  
G80 G00 Z1.0  
G28  
M30
```

G82 Spot Drill Canned Cycle (Group 09)

- F Feedrate in inches (or mm) per minute
- L Number of holes if G91 (Incremental Mode) is used
- P The dwell time at the bottom of the hole
- R Position of the R plane (position above the part)
- X X-axis location of hole
- Y Y-axis location of hole
- Z Position of bottom of hole



Programming Note: G82 is similar to G81 except that you have the option to program a dwell (P)

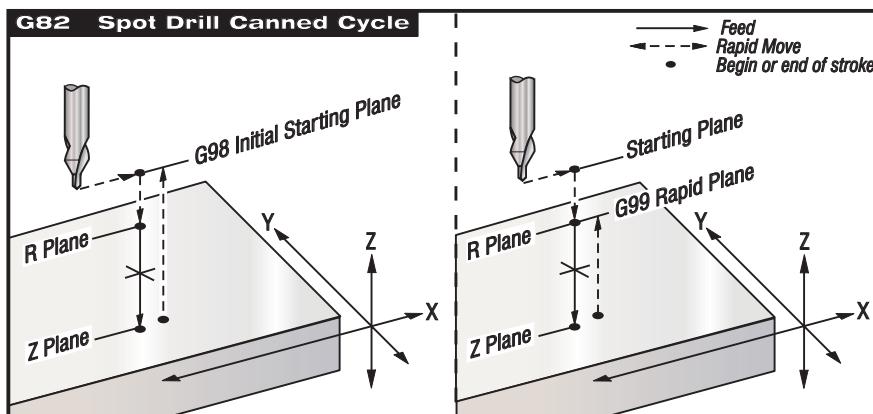
Program Example

```
%  
O1234  
T1 M06  
G90 G54 G00 X.565 Y-1.875 S1275 M03  
G43 H01 Z0.1 M08  
G82 Z-0.175 P.3 R0.1 F10.  
X1.115 Y-2.750  
X3.365 Y-2.875  
X4.188 Y-3.313  
X5.0 Y-4.0  
G80 G00 Z1.0 M09
```

(Sample program)
(Tool #1 is a 0.5" x 90-degree spot drill)

(90-degree spot drill; the depth is)
(half of the chamfer diameter)

Description



G82 Spot drilling example

G83 Normal Peck Drilling Canned Cycle (Group 09)

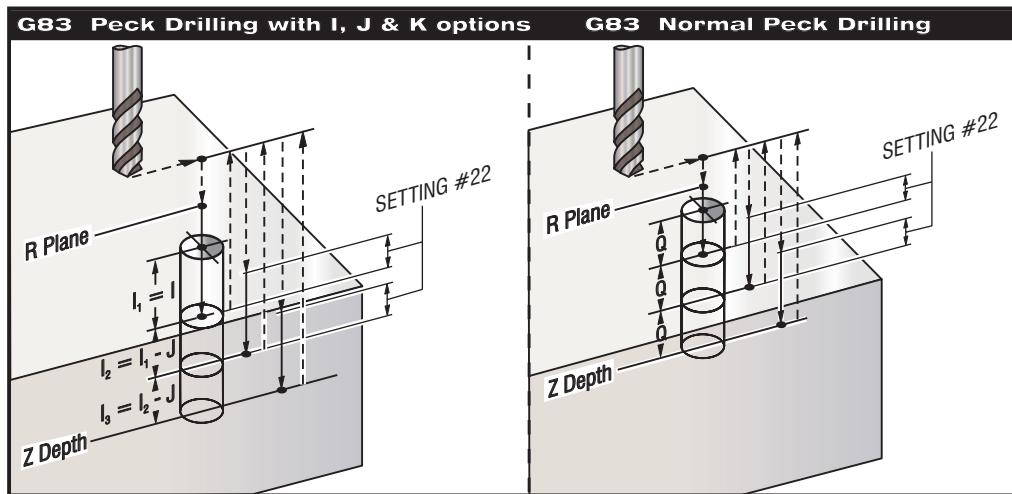
F	Feedrate in inches (or mm) per minute
I	Size of first cutting depth
J	Amount to reduce cutting depth each pass
K	Minimum depth of cut
L	Number of holes if G91 (Incremental Mode) is used
P	Pause at end of last peck, in seconds (Dwell)
Q	Cut depth, always incremental
R	Position of the R plane (position above the part)
X	X-axis location of hole
Y	Y-axis location of hole
Z	Position of the Z-axis at the bottom of hole

If I, J, and K are specified, the first pass will cut in by the amount of I, each succeeding cut will be reduced by amount J, and the minimum cutting depth is K.

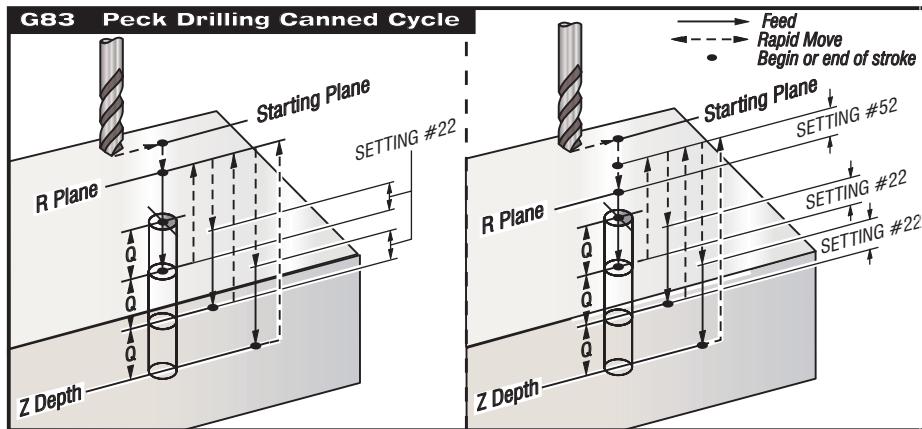
If P is specified, the tool will pause at the bottom of the hole for that amount of time. The following example will peck several times and dwell for 1.5 seconds:

G83 Z-0.62 F15. R0.1 Q0.175 P1.5.

The same dwell time will apply to all subsequent blocks that do not specify a dwell time.



Setting 52 changes the way G83 works when it returns to the R plane. Usually the R plane is set well above the cut to ensure that the peck motion allows the chips to get out of the hole. This wastes time as the drill starts by drilling "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 chip-clearing move to R occurs, the Z axis distance above R is determined by this setting.



Program Example

```
T2 M06  
G90 G54 G00 X0.565 Y-1.875 S2500 M03  
G43 H02 Z0.1 M08  
G83 Z-0.720 Q0.175 R0.1 F15.  
X1.115 Y-2.750  
X3.365 Y-2.875  
X4.188 Y-3.313  
X5.0 Y-4.0  
G80 G00 Z1.0 M09
```

Description

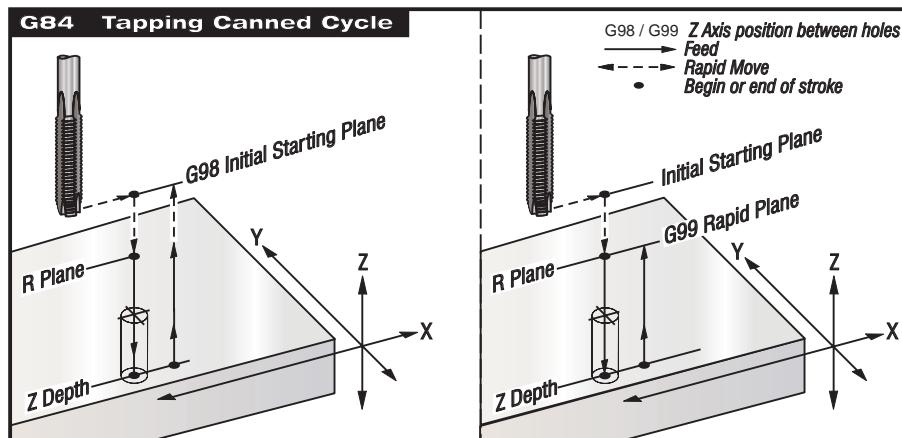
(Tool #2 is a 0.3125" stub drill)

(Drill point is 1/3 of the drill diameter)



G84 Tapping Canned Cycle (Group 09)

- F Feedrate in inches (or mm) per minute
J Retract Multiple (Example: J2 will retract twice as fast as the cutting speed, also see Setting 130)
L Number of holes if G91 (Incremental Mode) is used
R Position of the R plane (Position above the part)
X X-axis location of hole
Y Y-axis location of hole
Z Position of the Z-axis at the bottom of hole



G 84 Tapping Canned Cycle Example

Program Example

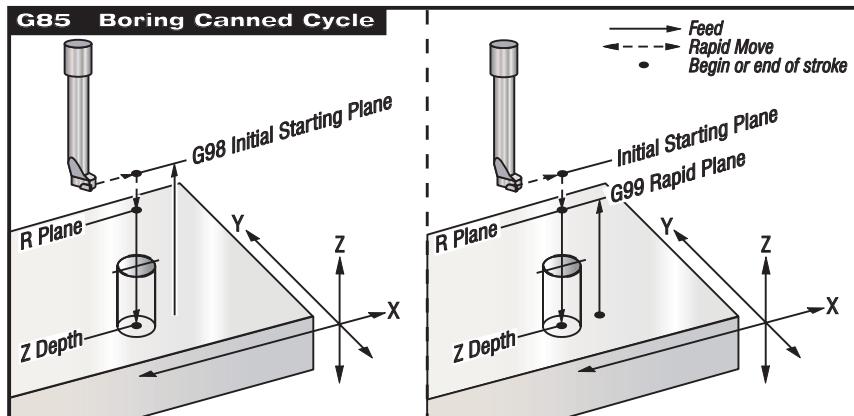
Helpful notes are listed in parentheses ().

T3 M06
G90 G54 G00 X0.565 Y-1.875 S900 M03
(Tool #3 is a 3/8-16 tap)
G43 H03 Z0.2 M08
G84 Z-0.600 R0.2 F56.25
(900 rpm divided by 16 tpi = 56.25 ipm)
X1.115 Y-2.750
X3.365 Y-2.875
X4.188 Y-3.313
X5.0 Y-4.0
G80 G00 Z1.0 M09
G28 G91 Y0 Z0
M30
%



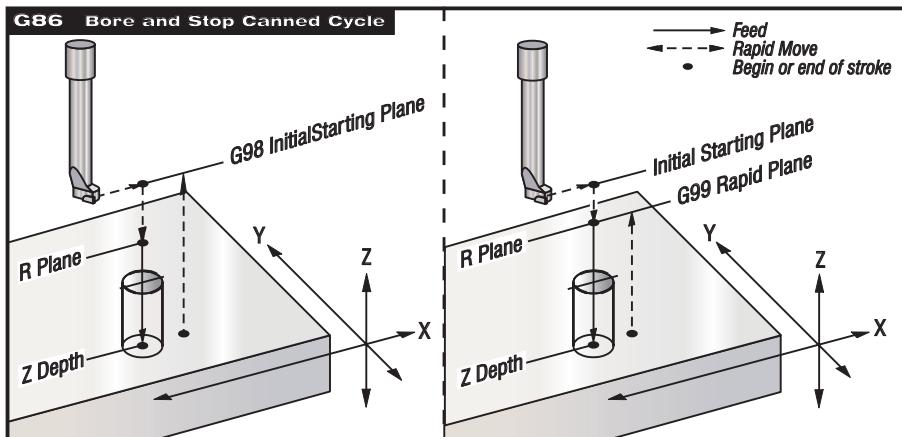
G85 Boring Canned Cycle (Group 09)

- F Feedrate in inches (or mm) per minute
L Number of holes if G91 (Incremental Mode) is used
R Position of the R plane (position above the part)
X X-axis location of holes
Y Y-axis location of holes
Z Position of the Z-axis at the bottom of hole



G86 Bore and Stop Canned Cycle (Group 09)

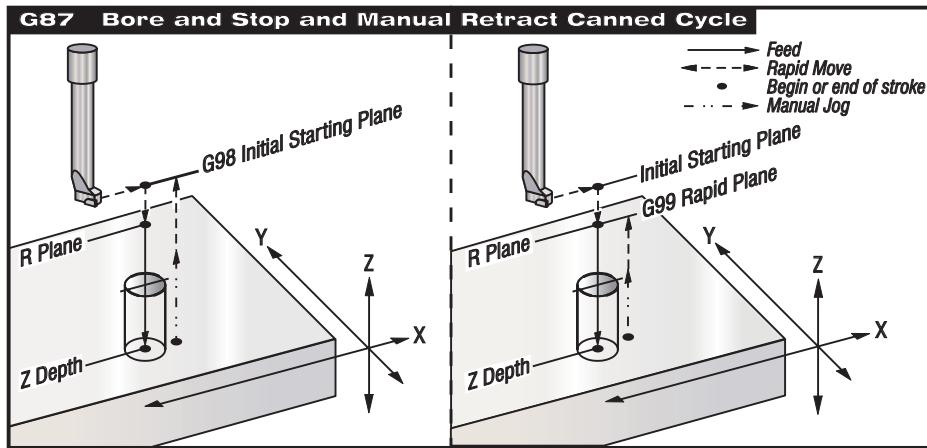
- F Feedrate in inches (or mm) per minute
L Number of holes if G91 (Incremental Mode) is used
R Position of the R plane (position above the part)
X X-axis location of hole
Y Y-axis location of hole
Z Position of the Z-axis at the bottom of hole



G87 Bore In and Manual Retract Canned Cycle (Group 09)

- F Feedrate in inches (or mm) per minute
L Number of holes if G91 (Incremental Mode) is used
R Position of the R plane (position above the part)
X X-axis location of hole
Y Y-axis location of hole
Z Position of the Z-axis at the bottom of hole

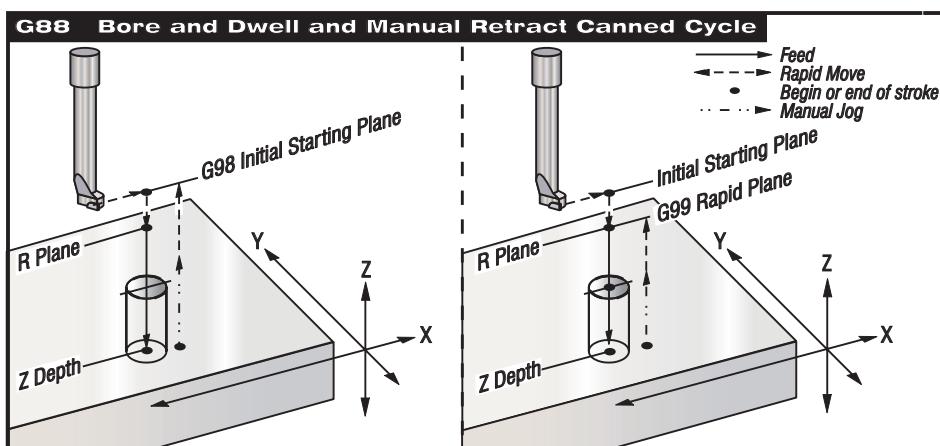
This G code will stop once the hole is bored. At this point the tool is manually jogged out of the hole. The program will continue when Cycle Start is pressed.



G88 Bore In, Dwell, Manual Retract Canned Cycle (Group 09)

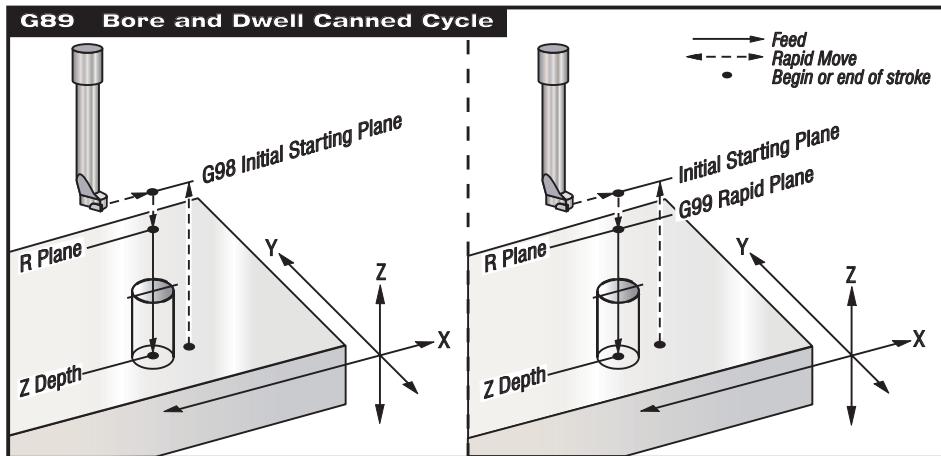
- F Feedrate in inches (or mm) per minute
- L Number of holes if G91 (Incremental Mode) is used
- P The dwell time at the bottom of the hole
- R Position of the R plane (position above the part)
- X X-axis location of hole
- Y Y-axis location of hole
- Z Position of the Z-axis at the bottom of hole

This G code will stop once the hole is bored. At this point the tool is manually jogged out of the hole. The program will continue when Cycle Start is pressed.



G89 Bore In, Dwell, Bore Out Canned Cycle (Group 09)

- F Feedrate in inches (or mm) per minute
- L Number of holes if G91 (Incremental Mode) is used
- P The dwell time at the bottom of the hole
- R Position of the R plane (position above the part)
- X X-axis location of holes
- Y Y-axis location of holes
- Z Position of the Z-axis at the bottom of hole

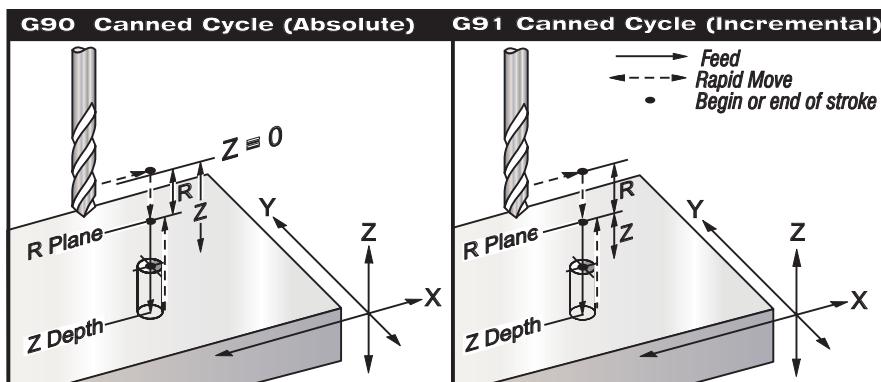


G90 Absolute Position Commands (Group 03)

G91 Incremental Position Commands (Group 03)

These G codes change the way the axis commands are interpreted. Axes commands following a G90 will move the axes to the machine coordinate. Axes commands following a G91 will move the axis that distance from the current point.

G91 is not compatible with G143 (5-Axis Tool Length Compensation).



G92 Set Work Coordinate Systems Shift Value (Group 00)

This G-code does not move any of the axes; it only changes the values stored as user work offsets. G92 works differently depending on Setting 33, which selects a FANUC, HAAS, or YASNAC coordinate system.

FANUC or HAAS

If setting 33 is set to Fanuc or Haas, a G92 command shifts all work coordinate systems (G54-59, G110-129) so that the commanded position becomes the current position in the active work system. G92 is a non-modal.

A G92 command cancels any G52 in effect for the commanded axes. Example: G92 X1.4 cancels the G52 for the X-axis. The other axes are not affected.

The G92 shift value is displayed at the bottom of the Work Offsets page and may be cleared there if necessary. It is also cleared automatically after power-up, and any time ZERO RET and AUTO ALL AXES or ZERO SINGLE AXIS are used.

YASNAC

If setting 33 is set to Yasnac, a G92 command sets the G52 work coordinate system so that the commanded position becomes the current position in the active work system. The G52 work system then automatically becomes active until another work system is selected.



G93 Inverse Time Feed Mode (Group 05)

F Feed Rate (strokes per minute)

This G code specifies that all F (feedrate) values are interpreted as **strokes per minute**. In other words the F code value, when divided into 60, is the number of seconds that the motion takes to complete.

G93 is generally used in 4 and 5-axis work. It is a way of translating the linear feedrate (inches/min) into a value that takes rotary motion into account. In G93 mode, the F value will tell you how many times per minute the tool move can be repeated.

When G93 is active, the feedrate specification is mandatory for all interpolated motion blocks; i.e., each non-rapid motion block must have its own feedrate specification.

* Pressing RESET will reset the machine to G94 (Feed per Minute) mode.

* Settings 34 and 79 (4th & 5th axis diameter) not necessary when using 93.

G94 Feed Per Minute Mode (Group 05)

This code deactivates G93 (Inverse Time Feed Mode) and returns the control to Feed Per Minute mode.

G95 Feed per Revolution (Group 05)

When G95 is active; a spindle revolution will result in a travel distance specified by the Feed value. If the Setting 9 Dimensioning is set to Inch, then the feed value F will be taken as inches/rev (set to MM, then the feed will be taken as mm/Rev). Feed Override and Spindle override will affect the behavior of the machine while G95 is active. When a spindle override is selected, any change in the spindle speed will result in a corresponding change in feed in order to keep the chip load uniform. However, if a feed override is selected, then any change in the feed override will only affect the feed rate and not the spindle.

G98 Canned Cycle Initial Point Return (Group 10)

Using G98, the Z-axis returns to its initial starting point (the Z position in the block before the canned cycle was commanded) between each X and/or Y location. This allows for positioning up and around areas of the part and/or clamps and fixtures.

G99 Canned Cycle R Plane Return (Group 10)

Using G99, the Z-axis will stay at the R plane between each X and/or Y location. When obstructions are not in the path of the tool G99 saves machining time.

G100 Cancel Mirror Image (Group 00)

G101 Enable Mirror Image (Group 00)

X X-axis command

Y Y-axis command

Z Z-axis command

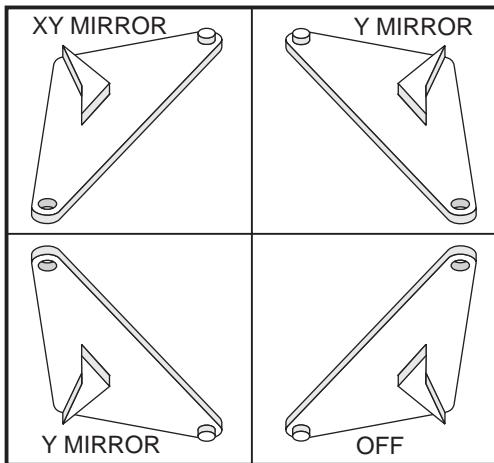
A A-axis command

Programmable mirror imaging is used to turn on or off any the axes. When one is ON, axis motion may be mirrored (or reversed) around the work zero point. These G codes should be used in a command block without any other G codes. They do not cause any axis motion. The bottom of the screen will indicate when an axis is mirrored. Also see Settings 45 through 48 for mirror imaging.

The format for turning Mirror Image on and off is:

G101 X0 = Will turn on mirror imaging for the X axis.

G100 X0 = Will turn off mirror imaging for the X axis.



Mirror Image and Cutter Compensation

When using cutter compensation with mirror imaging, follow this guideline: After turning mirror imaging on or off with G100 or G101, the next motion block should be to a different work coordinate position than the first one. The following code is an example:

Correct:

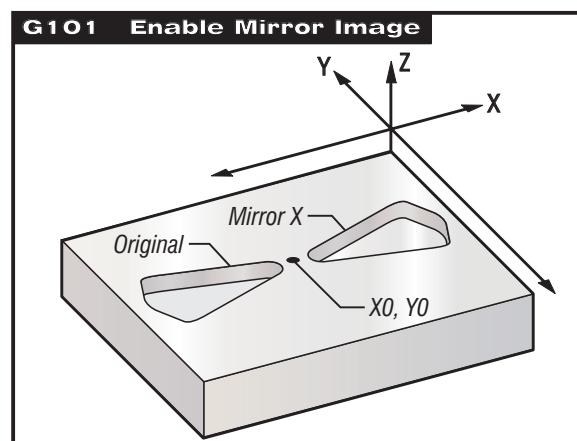
```
G41 X1.0 Y1.0  
G01 X2.0 Y2.0  
G101 X0  
G00 Z1.0  
G00 X1.0  
G00 X2.0 Y2.0  
G40
```

Incorrect:

```
G41 X1.0 Y1.0  
G01 X2.0 Y2.0  
G101 X0  
G00 Z1.0  
G00 X2.0 Y2.0  
G40.
```

Note: mirroring only one of the X or Y axes will cause the cutter to move along the opposite side of a cut. In addition, if mirror imaging is selected for only one axis of a circular motion plane (G02, G03) then they are reversed, and left and right cutter compensation commands (G41, G42) are reversed.

NOTE: When milling a shape with XY motions, turning **on** MIRROR IMAGE for just one of the X or Y axes will change climb milling to conventional milling and/or conventional milling to climb milling. As a result, you may not get the type of cut or finish that was desired. Mirror imaging of both X and Y will eliminate this problem.



Mirror Image and Pocket Milling



Program Code for Mirror Imaging in the X-Axis:

Program Example

Program Example	Description
%	
O3600	(<i>Mirror image X axis</i>)
T1 M06	(<i>Tool #1 is a 0.250" diameter endmill</i>)
G00 G90 G54 X-.4653 Y.052 S5000 M03	
G43 H01 Z.1 M08	
G01 Z-.25 F5.	
F20.	
M98 P3601	
G00 Z.1	
G101 X0.	
X-.4653 Y.052	
G01 Z-.25 F5.	
F20.	
M98 P3601	
G00 Z.1	
G100 X0.	
G28 G91 Y0 Z0	
M30	
%	
%	
O3601	(<i>Contour subprogram</i>)
G01 X-1.2153 Y.552	
G03 X-1.3059 Y.528 R.0625	
G01 X-1.5559 Y.028	
G03 X-1.5559 Y-.028 R.0625	
G01 X-1.3059 Y-.528	
G03 X-1.2153 Y-.552 R.0625	
G01 X-.4653 Y-.052	
G03 X-.4653 Y.052 R.0625	
M99	
%	

G102 Programmable Output to RS-232 (Group 00)

X	X-axis command
Y	Y-axis command
Z	Z-axis command
A	A-axis command

Commanding a G102 will send the current work coordinates of the axes to the first RS-232 port, from there a computer is used to record the values sent. Each axis listed in the G102 command block is output to the RS-232 port in the same format as values displayed in a program. A G102 should be used in a command block without any other G-codes. It will not cause any axis motion, the value for the axes have no effect.

Also see Setting 41 and Setting 25. The values sent out are always the current axis positions referenced to the current work coordinate system.

This G-code is useful in order to probe a part (also see G31). When the probe touches the part, the next line of code could be a G102 to send the axes position to a computer in order to store the coordinates. This is referred to as digitizing a part. Digitizing is taking a tangible part and making an electronic copy of it. Additional software for the personal computer is required to complete this function.



G103 Limit Block Buffering (Group 00)

Maximum number of blocks the control will look ahead (Range 0-15), for example:

G103 [P..]

This is commonly referred to, as "Block Look-ahead" which is a term used to describe what the control is doing in the background during machine motions. The control prepares future blocks (lines of code) ahead of time. While the current block is executing, the next block has already been interpreted and prepared for continuous motion.

When G103 P0 is programmed, block limiting is disabled. Block limiting is also disabled if G103 appears in a block without a P address code.

When G103 Pn is programmed, look-ahead is limited to n blocks.

G103 is also useful for debugging macro programs. Macro expressions are done during look-ahead time. For example, by inserting a G103 P1 into the program, macro expressions will be performed one block ahead of the currently executing block.

G107 Cylindrical Mapping (Group 00)

X	X-axis command
Y	Y-axis command
Z	Z-axis command
A	A-axis command
Q	Diameter of cylindrical surface
R	Radius of rotary axis

This G code translates all programmed motion occurring in a specified linear axis into the equivalent motion along the surface of a cylinder (as attached to a rotary axis) See the following figure). It is a Group 0 G code, but its default operation is subject to Setting 56 (M30 Restores Default G). The G107 command is used to either activate or deactivate cylindrical mapping.

- Any linear-axis program can be cylindrically mapped to any rotary axis (one at a time).
- An existing linear-axis G-code program can be cylindrically mapped by inserting a G107 command at the beginning of the program.
- The radius (or diameter) of the cylindrical surface can be redefined, allowing cylindrical mapping to occur along surfaces of different diameters without having to change the program.
- The radius (or diameter) of the cylindrical surface can either be synchronized with or be independent of the rotary axis diameter(s) specified in the Settings 34 and 79.
- G107 can also be used to set the default diameter of a cylindrical surface, independently of any cylindrical mapping that may be in effect.

G107 Description

1) Three address codes can follow a G107: **X**, **Y** or **Z**; **A** or **B**; and **Q** or **R**.

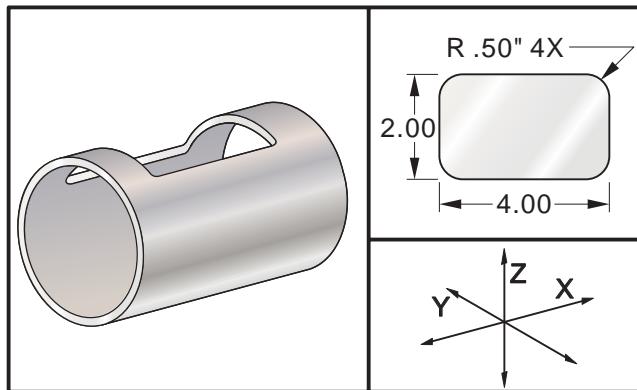
X, Y, or Z: An X, Y, or Z address specifies the linear axis that will be mapped to the specified rotary axis (A or B). When one of these linear axes is specified, a rotary axis must also be specified.

A or B: An A or B address identifies which rotary axis holds the cylindrical surface.

Q or R: Q defines the diameter of the cylindrical surface, while R defines the radius. When Q or R is used, a rotary axis must also be specified. If neither Q nor R is used, then the last G107 diameter is used. If no G107 command has been issued since power-up, or if the last value specified was zero, then the diameter will be the value in Setting 34 and/or 79 for this rotary axis. When Q or R is specified, that value will become the new G107 value for the specified rotary axis.



Cylindrical mapping will also be turned off automatically whenever the G-code program ends, but only if Setting 56 is ON. Pressing the RESET key will turn off any cylindrical mapping that is currently in effect, regardless of the status of Setting 56.



Example

```
%  
O0079 (G107 TEST)  
G00 G40 G49 G80 G90  
G28 G91 A0  
G90  
G00 G54 X1.75 Y0 S5000 M03  
G107 A0 Y0 R2. (IF NO R OR Q VALUE, MACHINE WILL USE VALUE IN SETTING 34)  
G43 H01 Z0.25  
G01 Z-0.25 F25.  
G41 D01 X2. Y0.5  
G03 X1.5 Y1. R0.5  
G01 X-1.5  
G03 X-2. Y0.5 R0.5  
G01 Y-0.5  
G03 X-1.5 Y-1. R0.5  
G01 X1.5  
G03 X2. Y-0.5 R0.5  
G01 Y0.  
G40 X1.75  
G00 Z0.25  
M09  
M05  
G91 G28 Z0.  
G28 Y0.  
G90  
G107  
M30  
%
```

G110-G129 Coordinate System #7-26 (Group 12)

These codes select one of the additional work coordinate systems. All subsequent references to axis positions will be interpreted in the new coordinate system. Operation of G110 to G129 is the same as G54 to G59.



G136 Automatic Work Offset Center Measurement (Group 00)

(This G-code is optional and requires a probe)

F	Feedrate in inches (mm) per minute
I	Optional offset distance along X-axis
J	Optional offset distance along Y-axis
K	Optional offset distance along Z-axis
X	Optional X-axis motion command
Y	Optional Y-axis motion command
Z	Optional Z-axis motion command

Automatic Work Offset Center Measurement (G136) is used to command a probe to set work offsets. A G136 will feed the axes of the machine in an effort to probe the workpiece with a spindle mounted probe. The axis (axes) will move until a signal from the probe is received, or the travel limit is reached.

Tool offsets (G41, G42, G43, or G44) must not be active when this function is preformed. The currently active work coordinate system is set for each axis programmed. Use a G31 cycle with an M75 to set the first point. A G136 will set the work coordinates to a point at the center of a line between the probed point and the point set with an M75. This allows the center of the part to be found using two separate probed points.

If an **I**, **J**, or **K** is specified, the appropriate axis work offset is shifted by the amount in the **I**, **J**, or **K** command. This allows the work offset to be shifted away from where the probe actually contacts the part.

Notes:

Also see G31.

The points probed are offset by the values in Settings 59 through 62.

Use G91 incremental moves when using a G36.

Use the assigned M-codes (M53 & M63) with a dwell to turn the spindle probe on or off.

Example:

M53
G04 P100
M63

Programming example to probe the center of a bore:

```
O1234 (G136)
M53
G04 P100
M63
G00 G90 G54 X0 Y0
Z-17.
G91 G01 Z-1. F20.
G31 X1. F10. M75
G01 X-1.
G136 X-1. F10.
G01 X1.
M53
G04 P100
M63
G00 G90 G53 Z0
M30
```



Programming example to probe the center of a part:

```
O1234 (G136)
M53
G04 P100
M63
G00 G90 G54 X0 Y5.
Z-17.
G91 G01 Z-1. F20.
G31 Y-1. F10. M75
G01 Y1. F20.
G00 Z2.
Y-10.
G01 Z-2. F20.
G136 Y1. F10.
G01 Y-1.
M53
G04 P100
M63
G00 G90 G53 Z0
M30
```

G141 3D+ Cutter Compensation (Group 07)

X	X-axis command
Y	Y-axis command
Z	Z-axis command
D	Cutter Size Selection
I	X-axis vector direction for application of cutter compensation
J	Y-axis vector direction for application of cutter compensation
K	Z-axis vector direction for application of cutter compensation

This feature performs 3D+ cutter diameter compensation. The form is:

G141 Xnnn Ynnn Znnn Dnnn Innn Jnnn Knnc

Subsequent lines can be of the form:

G01 Fnnc Xnnn Ynnn Znnn Innn Jnnn Knnc Or

G00 Xnnn Ynnn Znnn Innn Jnnn Knnc

The 3D+ G141 cutter compensation command is not just for 5-axis work. Any CAD system can output the I, J, K values to shift the tool by the amount in the Offset, even if the motions are only in 2 or 3 axes. Only G00 and G01 can use G141 cutter compensation. G93 Inverse time is usually used with G141. The Dnn code selects which radius or diameter offset to use. The I, J, and K values give the direction in which cutter compensation is applied.

G141 Example:

```
T1 M06
G00 G90 G54 X0 Y0 Z0 A0 B0
G141 D01 X0.Y0.Z0. (RAPID POSIT WITH 3 AX C COMP)
G01 G93 X.01 Y.01 Z.01 I.1 J.2 K.9747 F300. (FEED INV TIME)
X.02 Y.03 Z.04 I.15 J.25 K.9566 F300.
X.02 Y.055 Z.064 I.2 J.3 K.9327 F300
.
.
.
X2.345 Y.1234 Z-1.234 I.25 J.35 K.9028 F200. (LAST MOTION)
G94 F50. (CANCEL G93)
G0 G90 G40 Z0 (Rapid to Zero, Cancel Cutter Comp)
X0 Y0
M30
```



G143 5-Axis Tool Length Compensation + (Group 08)

(This G-code is optional; it only applies to machines on which all rotary motion is movement of the cutting tool.) This G code allows the user to correct for variations in the length of cutting tools without the need for a CAD/CAM processor. An H code is required to select the tool length from the existing length compensation tables. A G49 or H00 command will cancel 5-axis compensation. For G143 to work correctly there must be two rotary axes, A and B. G90, absolute positioning mode must be active (G91 cannot be used). Work position 0,0 for the A and B axes must be so the tool is parallel with Z-axis motion.

The intention behind G143 is to compensate for the difference in tool length between the originally posted tool and a substitute tool. Using G143 allows you to run the program without having to repost a new tool length.

G143 tool length compensation works only with rapid (G00) and linear feed (G01) motions; no other feed functions (G02 or G03) or canned cycles (drilling, tapping, etc.) can be used. For a positive tool length, the Z-axis would move upward (in the + direction). If one of X, Y or Z is not programmed, there will be no motion of that axis, even if the motion of A or B produces a new tool length vector. Thus a typical program would use all 5 axes on one block of data. G143 may affect commanded motion of all axes in order to compensate for the A and B axes.

Inverse feed mode (G93) is recommended, when using G143. An example follows:

```
T1 M06  
G00 G90 G54 X0 Y0 Z0 A0 B0  
G143 H01 X0. Y0. Z0. A-20. B-20. (RAPID POSIT W. 5AX COMP)  
G01 G93 X.01 Y.01 Z.01 A-19.9 B-19.9 F300. (FEED INV TIME)  
X0.02 Y0.03 Z0.04 A-19.7 B-19.7 F300.  
X0.02 Y0.055 Z0.064 A-19.5 B-19.6 F300  
X2.345 Y.1234 Z-1.234 A-4.127 B-12.32 F200. (LAST MOTION)  
G94 F50. (CANCEL G93)  
G0 G90 G49 Z0 (RAPID TO ZERO, CANCEL 5 AXS COMP)  
X0 Y0  
M30
```

G150 General Purpose Pocket Milling (Group 00)

D	Tool radius/diameter offset selection
F	Feedrate
I	X-axis cut increment (positive value)
J	Y-axis cut increment (positive value)
K	Finishing pass amount (positive value)
P	Subprogram number that defines pocket geometry
Q	Incremental Z-axis cut depth per pass (positive value)
R	Position of the rapid R-plane location
S	Optional spindle speed
X	X start position
Y	Y start position
Z	Final depth of pocket

The G150 starts by positioning the cutter to a start point inside the pocket. The end mill will plunge straight down in the Z-axis. A subprogram P### is then called that defines the pocket geometry of a closed area using G01, G02, and G03 motions in the X and Y axes on the pocket. Note: When defining the G150 pocket geometry in the subprogram, do not move back to the starting hole after the pocket shape is closed.

An I or J value defines the roughing pass amount the cutter moves over for each cut increment. If I is used, the pocket is roughed out from a series of increment cuts in the X-axis. If J is used, the increment cuts are in the Y-axis.

The K command defines a finish pass amount on the pocket. If a K value is specified, a finish pass is performed by K amount, around the inside of pocket geometry for the last pass and is done at the final Z depth. There is no finishing pass command for the Z depth.



The R value needs to be specified, even if it is zero (R0), or the last R value that was specified will be used.

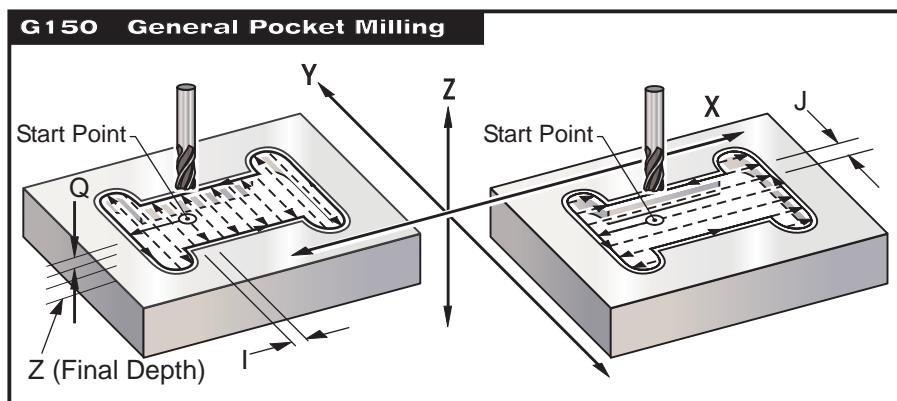
Multiple passes in the pocket area are done, starting from the R plane, with each Q (Z-axis depth) pass to the final depth. The G150 command will first make a pass around pocket geometry, leaving stock with K, then doing passes of I or J roughing out inside of pocket after feeding down by the value in Q until the Z depth is reached.

The Q command must be in the G150 line, even if only one pass to the Z depth is desired. The Q command starts from the R plane.

Notes: The subprogram (P) must not consist of more than 40 pocket geometry moves.

The Q command must be in the G150 line, even if only one pass to the Z depth is desired. The Q command starts from the R plane.

It may be necessary to drill a starting point, for the G150 cutter, to the final depth (Z). Then position the end mill to the start location in the XY axes within the pocket for the G150 command.



Example

O01001

(G150 Pocket example)

T1 M06

G90 G54 G00 X3.25 Y4.5 S1200 M03

(T1 Drills clearance hole for endmill)

G43 H01 Z1.0 M08

(Pocket start point)

G83 Z-1.5 Q0.25 R0.1 F20.

(Tool length offset, rapid to a Z start point, turn coolant on)

G53 G49 Z0

(Peck drill cycle)

(Returns Z to home position)

T2 M06 (.5" Endmill)

(T2 Cuts pocket in two passes to Z depth)

G54 G90 G00 X3.25 Y4.5 S1450 M03

(Pocket start point)

G43 H02 Z1.0 M08

(Tool length offset, rapid to a Z start point, turn coolant on)

G150 X3.25 Y4.5 Z-1.5 G41 J0.35 K.01 Q0.8 R.1 P2001 D02 F15. (K does a 0.01" finish pass on sides)

G40 X3.25 Y4.5

(Cancel cutter comp. and position back to start point of pocket)

G53 G49 Y0 Z0

(Returns Z to home position)

M30

(End of main program)

O02001

(Separate program as a subprogram for G150 pocket geometry)

G01 Y7.

(The first move onto pocket geometry with a G01 move)

X1.5

(The following lines define pocket geometry)

G03 Y5.25 R0.875

G01 Y2.25

G03 Y0.5 R0.875

G01 X5.

G03 Y2.25 R0.875

G01 Y5.25

G03 Y7. R0.875

G01 X3.25

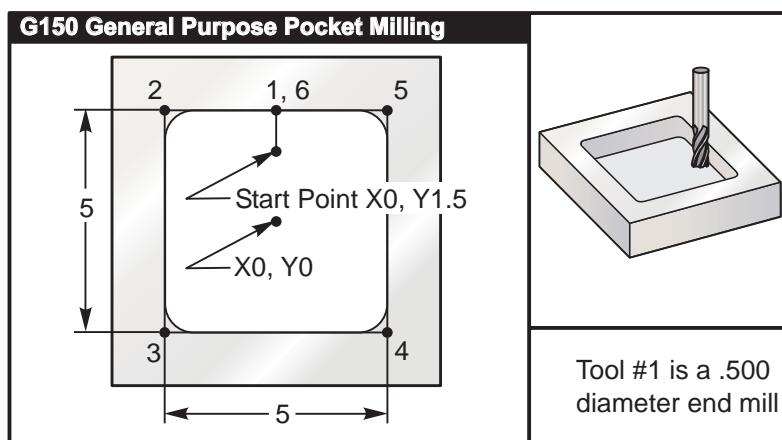
(Close pocket geometry loop. Do not position back to start hole.)

M99

(Return to Main Program)



Square Pocket



Pocket milling for G150 operation.

5.0 x 5.0 x 0.500 DP. Square Pocket

Main Program

```
%  
O01001  
T1 M06 (Tool #1 is a 0.500" diameter endmill)  
G90 G54 G00 X0. Y1.5 (XY Start Point)  
S2000 M03  
G43 H01 Z0.1 M08  
G01 Z0.01 F30.  
G150 P1002 Z-0.5 Q0.25 R0.01 J0.3 K0.01 G41 D01 F10.  
G40 G01 X0. Y1.5  
G00 Z1. M09  
G53 G49 Y0. Z0.  
M30  
%
```

Subprogram

```
%  
O01002 (Subprogram for G150 in O01001)  
G01 Y2. (1)  
X-2.5 (2)  
Y-2.5 (3)  
X2.5 (4)  
Y2.5 (5)  
X0. (6) (Close Pocket Loop)  
M99 (Return to Main Program)  
%
```

Absolute and Incremental examples of a subprogram called up by the P### command in the G150 line:

Absolute Subprogram

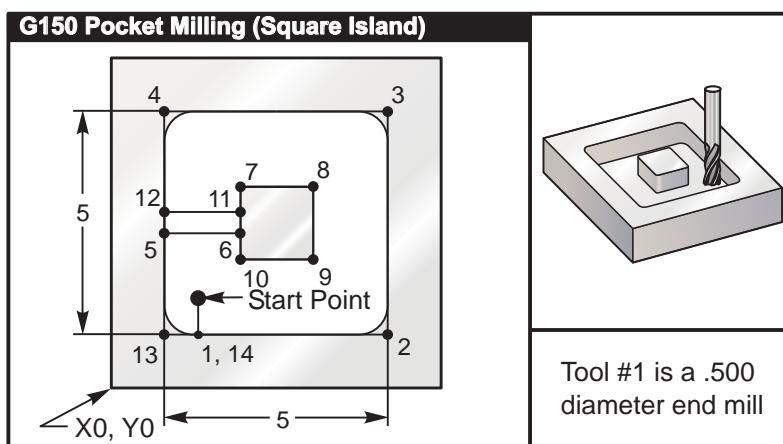
```
%  
O01002 (G90 Subprogram for G150)  
G90 G01 Y2. (1)  
X-2.5 (2)  
Y-2.5 (3)  
X2.5 (4)  
Y2.5 (5)  
X0. (6)  
M99  
%
```

Incremental Subprogram

```
%  
O01002 (G91 Subprogram for G150)  
G91 G01 Y0.5 (1)  
X-2.5 (2)  
Y-5. (3)  
X5. (4)  
Y5. (5)  
X-2.5 (6)  
G90  
M99  
%
```



Square Island



G150 Pocket milling program with a Square island.

5.0 x 5.0 x 0.500 DP. Square Pocket with Square Island

Main Program

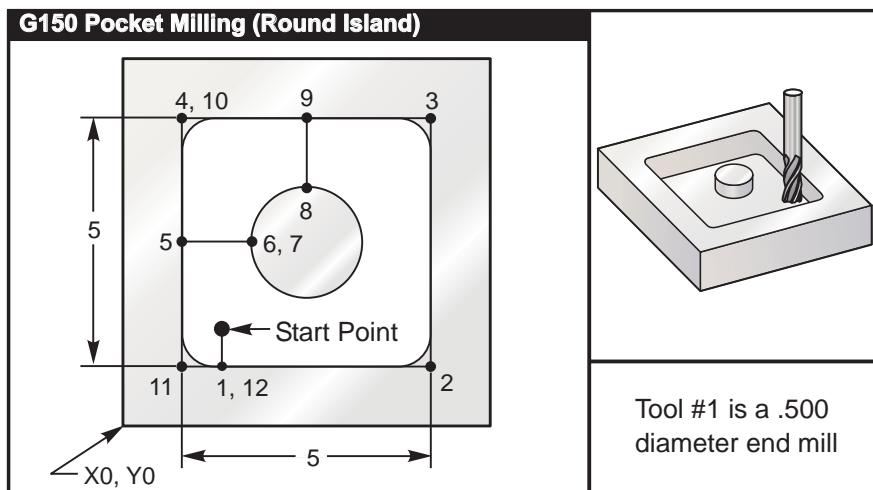
```
%  
O02010  
T1 M06 (Tool is a 0.500" diameter endmill)  
G90 G54 G00 X2. Y2. (XY Start Point)  
S2500 M03  
G43 H01 Z0.1 M08  
G01 Z0.01 F30.  
G150 P2020 X2. Y2. Z-0.5 Q0.5 R0.01 I0.3 K0.01 G41 D01 F10.  
G40 G01 X2. Y2.  
G00 Z1.0 M09  
G53 G49 Y0. Z0.  
M30  
%
```

Subprogram

```
%  
O02020 (Subprogram for G150 in O02010)  
G01 Y1. (1)  
X6. (2)  
Y6. (3)  
X1. (4)  
Y3.2 (5)  
X2.75 (6)  
Y4.25 (7)  
X4.25 (8)  
Y2.75 (9)  
X2.75 (10)  
Y3.8 (11)  
X1. (12)  
Y1. (13)  
X2. (14) (Close Pocket Loop)  
M99 (Return to Main Program)  
%
```



Round Island



G150 Pocket milling program with a Round island.

5.0 x 5.0 x 0.500 DP. Square Pocket with Round Island

Main Program

```
%  
O03010  
T1 M06 (Tool is a 0.500" diameter endmill)  
G90 G54 G00 X2. Y2. (XY Start Point)  
S2500 M03  
G43 H01 Z0.1 M08  
G01 Z0. F30.  
G150 P3020 X2. Y2. Z-0.5 Q0.5 R0.01 J0.3 K0.01 G41 D01 F10.  
G40 G01 X2. Y2.  
G00 Z1. M09  
G53 G49 Y0. Z0.  
M30  
%
```

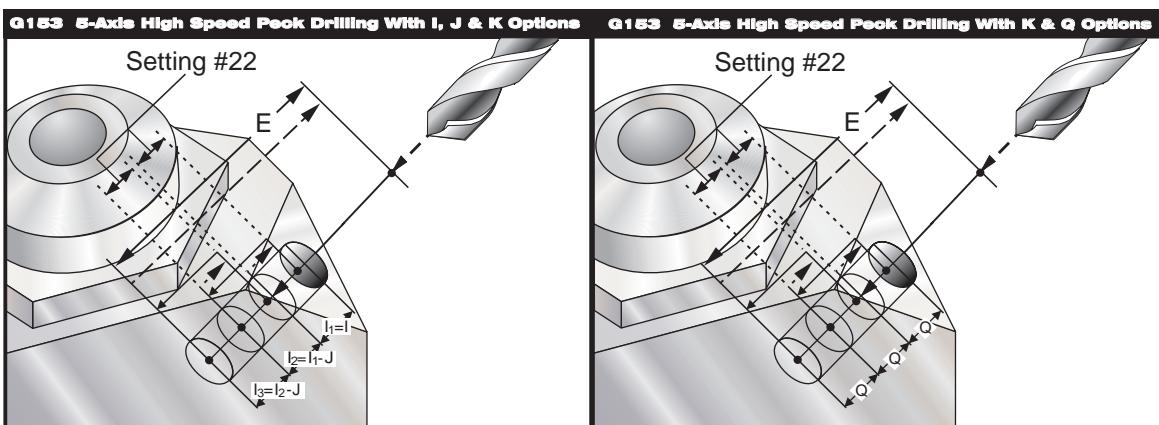
Subprogram

```
%  
O03020 (Subprogram for G150 in O03010)  
G01 Y1. (1)  
X6. (2)  
Y6. (3)  
X1. (4)  
Y3.5 (5)  
X2.5 (6)  
G02 I1. (7)  
G02 X3.5 Y4.5 R1. (8)  
G01 Y6. (9)  
X1. (10)  
Y1. (11)  
X2. (12) (Close Pocket Loop)  
M99 (Return to Main Program)  
%
```



G153 5-Axis High Speed Peck Drilling Canned Cycle (Group 09)

- E Specifies the distance from the start position to the bottom of the hole
- F Feedrate in inches (mm) per minute
- I Size of first cutting depth (must be a positive value)
- J Amount to reduce cutting depth each pass (must be a positive value)
- K Minimum depth of cut (must be a positive value)
- L Number of repeats
- P Pause at end of last peck, in seconds
- Q The cut-in value (must be a positive value)
- A A-axis tool starting position
- B B-axis tool starting position
- X X-axis tool starting position
- Y Y-axis tool starting position
- Z Z-axis tool starting position



This 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 amount **I**, each succeeding cut will be reduced by amount **J**, and the minimum cutting depth is **K**. If **P** is used, the tool will pause at the bottom of the hole for that amount of time.

Note that the same dwell time applies to all subsequent blocks that do not specify a dwell time.

G154 Select Work Coordinates P1-P99 (Group 12)

This feature provides 99 additional work offsets. G154 with a P value from 1 to 99 will activate the additional work offsets. For example G154 P10 will select work offset 10 from the list of additional work offset. Note that G110 to G129 refer to the same work offsets as the G154 P1 through P20; they can be selected by using either method. When a G154 work offset is active, the heading in the upper right work offset will show the G154 P value.

G154 work offsets format

- #14001-#14006 G154 P1 (also #7001-#7006 and G110)
- #14021-#14026 G154 P2 (also #7021-#7026 and G111)
- #14041-#14046 G154 P3 (also #7041-#7046 and G112)
- #14061-#14066 G154 P4 (also #7061-#7066 and G113)
- #14081-#14086 G154 P5 (also #7081-#7086 and G114)
- #14101-#14106 G154 P6 (also #7101-#7106 and G115)
- #14121-#14126 G154 P7 (also #7121-#7126 and G116)
- #14141-#14146 G154 P8 (also #7141-#7146 and G117)
- #14161-#14166 G154 P9 (also #7161-#7166 and G118)
- #14181-#14186 G154 P10 (also #7181-#7186 and G119)
- #14201-#14206 G154 P11 (also #7201-#7206 and G120)
- #14221-#14221 G154 P12 (also #7221-#7226 and G121)



#14241-#14246 G154 P13 (also #7241-#7246 and G122)
#14261-#14266 G154 P14 (also #7261-#7266 and G123)
#14281-#14286 G154 P15 (also #7281-#7286 and G124)
#14301-#14306 G154 P16 (also #7301-#7306 and G125)
#14321-#14326 G154 P17 (also #7321-#7326 and G126)
#14341-#14346 G154 P18 (also #7341-#7346 and G127)
#14361-#14366 G154 P19 (also #7361-#7366 and G128)
#14381-#14386 G154 P20 (also #7381-#7386 and G129)
#14401-#14406 G154 P21
#14421-#14426 G154 P22
#14441-#14446 G154 P23
#14461-#14466 G154 P24
#14481-#14486 G154 P25
#14501-#14506 G154 P26
#14521-#14526 G154 P27
#14541-#14546 G154 P28
#14561-#14566 G154 P29
#14581-#14586 G154 P30
#14781-#14786 G154 P40
#14981-#14986 G154 P50
#15181-#15186 G154 P60
#15381-#15386 G154 P70
#15581-#15586 G154 P80
#15781-#15786 G154 P90
#15881-#15886 G154 P95
#15901-#15906 G154 P96
#15921-#15926 G154 P97
#15941-#15946 G154 P98
#15961-#15966 G154 P99

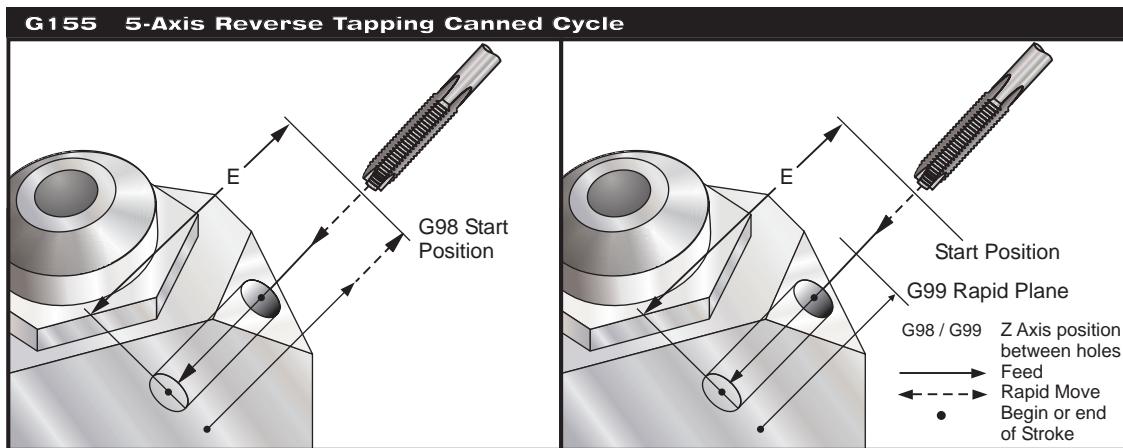
G155 5-Axis Reverse Tap Canned Cycle (Group 09)

G155 only performs floating taps. G174 is available for 5-axis reverse rigid tapping.

E	Specifies the distance from the start position to the bottom of the hole
F	Feedrate in inches (mm) per minute
L	Number of repeats
A	A-axis tool starting position
B	B-axis tool starting position
X	X-axis tool starting position
Y	Y-axis tool starting position
Z	Z-axis tool starting position
S	Spindle Speed

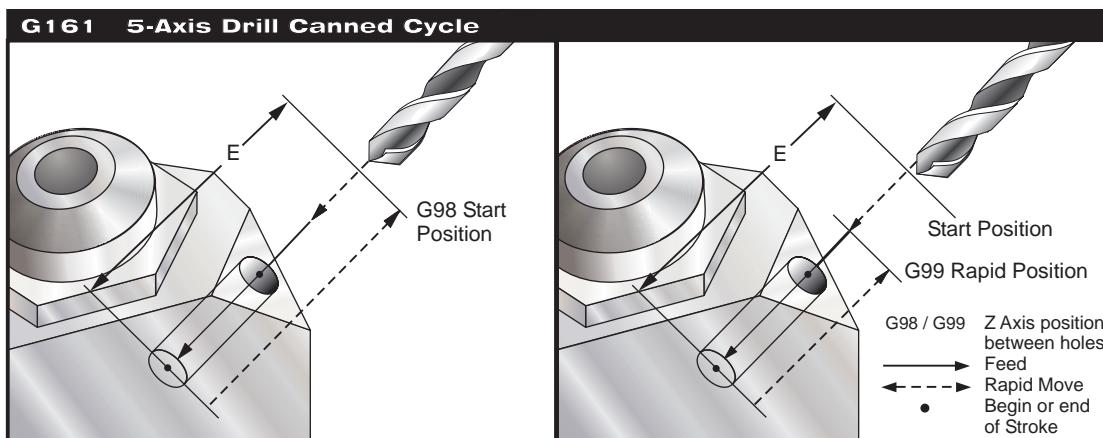
A specific X, Y, Z, A, B position must be programmed before the canned cycle is commanded. This position is used as the "Initial Start position".

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



G161 5-Axis Drill Canned Cycle (Group 09)

- E Specifies the distance from the start position to the bottom of the hole
- F Feedrate in inches (mm) per minute
- L Number of repeats
- A A-axis tool starting position
- B B-axis tool starting position
- X X-axis tool starting position
- Y Y-axis tool starting position
- Z Z-axis tool starting position

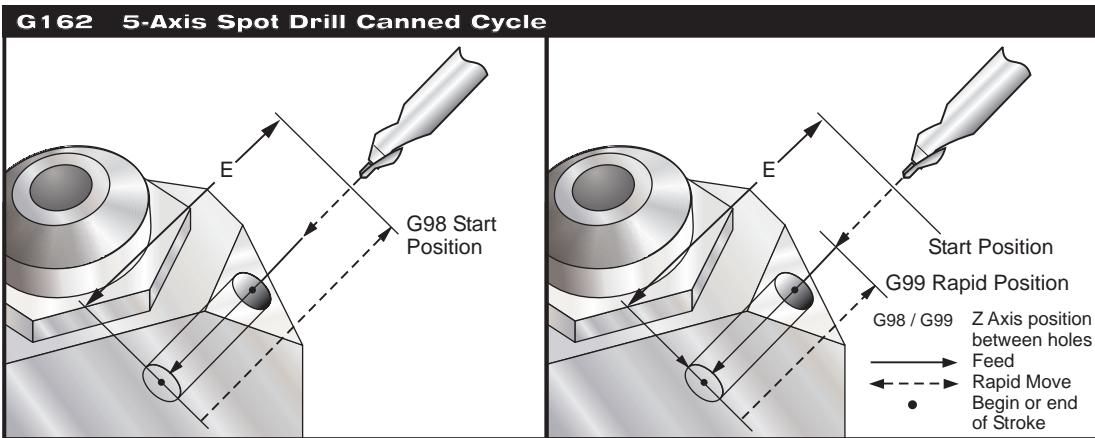


A specific X, Y, Z, A, B position must be programmed before the canned cycle is commanded. This position is used as the "Initial Start position".

G162 5-Axis Spot Drill Canned Cycle (Group 09)

- E Specifies the distance from the start position to the bottom of the hole
- F Feedrate in inches (mm) per minute
- L Number of repeats
- P The dwell time at the bottom of the hole
- A A-axis tool starting position
- B B-axis tool starting position
- X X-axis tool starting position
- Y Y-axis tool starting position
- Z Z-axis tool starting position

A specific X, Y, Z, A, B position must be programmed before the canned cycle is commanded. This position is used as the "Initial Start position".



G163 5-Axis Normal Peck Drilling Canned Cycle (Group 09)

- E Specifies the distance from the start position to the bottom of the hole
- F Feedrate 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
- P Optional pause at end of last peck, in seconds
- Q The cut-in value, always incremental
- A A-axis tool starting position
- B B-axis tool starting position
- X X-axis tool starting position
- Y Y-axis tool starting position
- Z Z-axis tool starting position

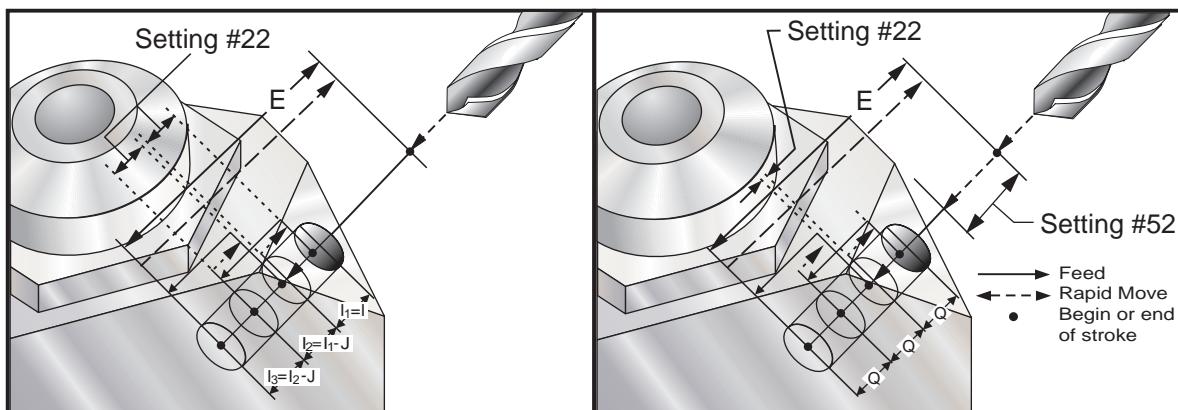
A specific X, Y, Z, A, B position must be programmed before the canned cycle is commanded. This position is used as the "Initial Start position".

If I, J, and K are specified the first pass will cut in by amount I, each succeeding cut will be reduced by amount J, and the minimum cutting depth is K.

A P value is used the tool will pause at the bottom of the hole after the last peck for that amount of time. The following example will peck several times and dwell for one and a half seconds at the end:

G163 Z-0.62 F15. R0.1 Q0.175 P1.5.

Note that the same dwell time applies to all subsequent blocks that do not specify a dwell time.



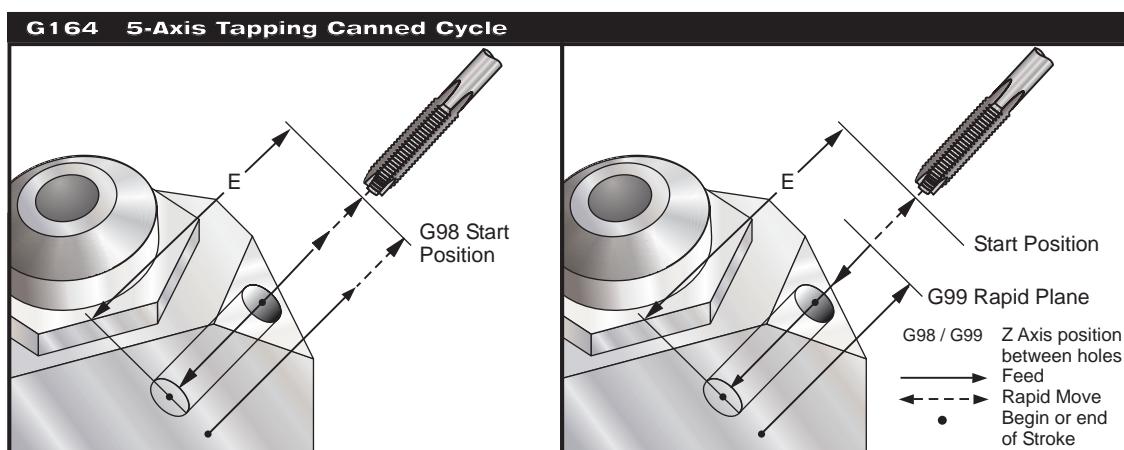


Setting 52 also changes the way G163 works when it returns to the **start position**. Usually the **R** plane is set well above the cut to ensure that the peck motion allows the chips to get out of the hole. This wastes time as the drill starts by drilling "empty" space. If Setting 52 is set to the distance required to clear chips, the **start position** can be put much closer to the part being drilled. When the chip-clearing move to the **start position** occurs, the **Z** axis will be moved above the **start position** by the amount given in this setting.

G164 5-Axis Tapping Canned Cycle (Group 09)

G164 only performs floating taps. G174/184 is available for 5-axis rigid tapping.

- E Specifies the distance from the start position to the bottom of the hole
- F Feedrate in inches (mm) per minute
- L Number of repeats
- A A-axis tool starting position
- B B-axis tool starting position
- X X-axis tool starting position
- Y Y-axis tool starting position
- Z Z-axis tool starting position
- S Spindle Speed



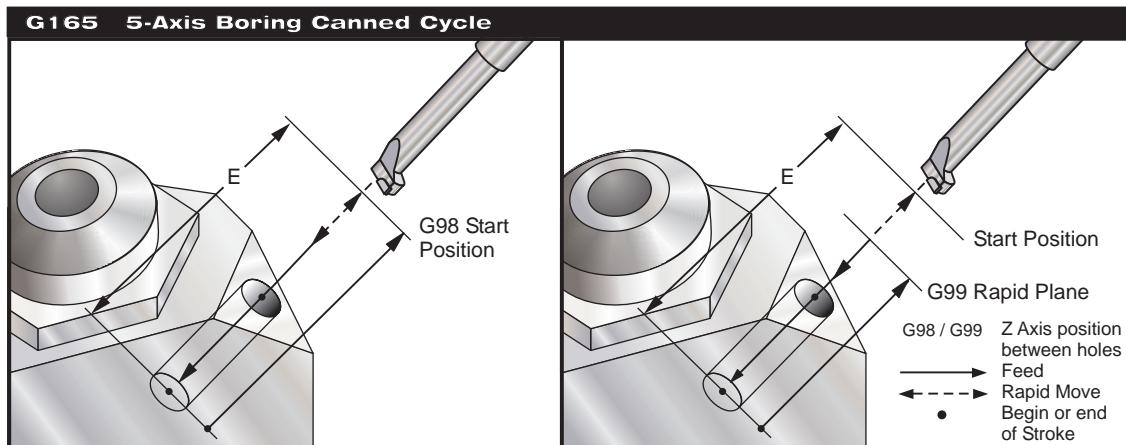
A specific X, Y, Z, A, B position must be programmed before the canned cycle is commanded. This position is used as the "Initial Start position".

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



G165 5-Axis Boring Canned Cycle (Group 09)

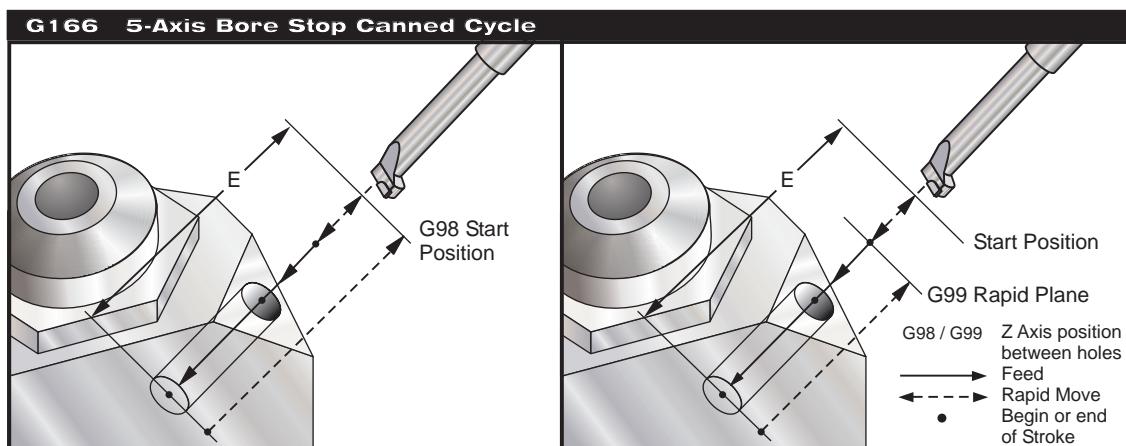
- E Specifies the distance from the start position to the bottom of the hole
- F Feedrate in inches (mm) per minute
- L Number of repeats
- A A-axis tool starting position
- B B-axis tool starting position
- X X-axis tool starting position
- Y Y-axis tool starting position
- Z Z-axis tool starting position



A specific X, Y, Z, A, B position must be programmed before the canned cycle is commanded. This position is used as the "Initial Start position".

G166 5-Axis Bore and Stop Canned Cycle (Group 09)

- E Specifies the distance from the start position to the bottom of the hole
- F Feedrate in inches (mm) per minute
- L Number of repeats
- A A-axis tool starting position
- B B-axis tool starting position
- X X-axis tool starting position
- Y Y-axis tool starting position
- Z Z-axis tool starting position

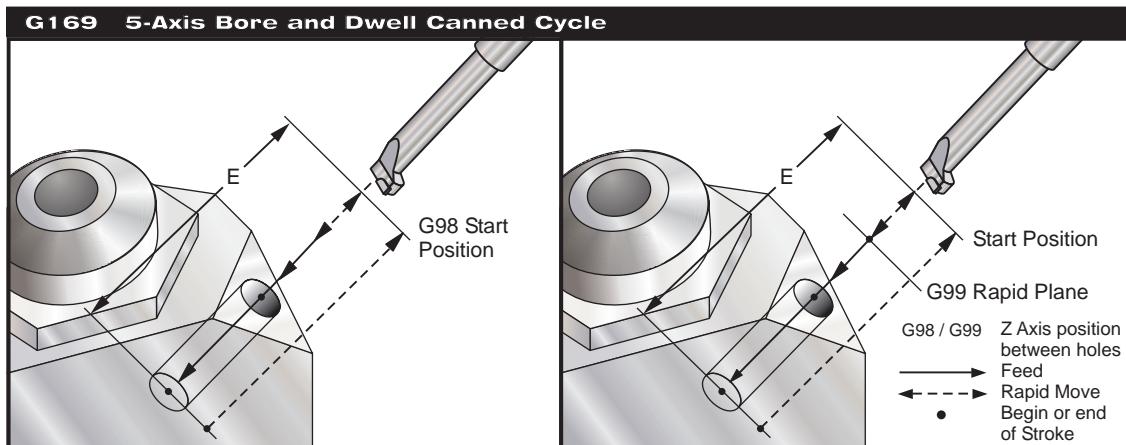


A specific X, Y, Z, A, B position must be programmed before the canned cycle is commanded. This position is used as the "Initial Start position".



G169 5-Axis Bore and Dwell Canned Cycle (Group 09)

- E Specifies the distance from the start position to the bottom of the hole
- F Feedrate in inches (mm) per minute
- L Number of repeats
- P The dwell time at the bottom of the hole
- A A-axis tool starting position
- B B-axis tool starting position
- X X-axis tool starting position
- Y Y-axis tool starting position
- Z Z-axis tool starting position



A specific X, Y, Z, A, B position must be programmed before the canned cycle is commanded. This position is used as the "Initial Start position".

G174 CCW Non-Vertical Rigid Tap (Group 00)

G184 CW Non-Vertical Rigid Tap (Group 00)

- F Feedrate in inches per minute
- X X position at bottom of hole
- Y Y position at bottom of hole
- Z Z position at bottom of hole
- S Spindle Speed

A specific X, Y, Z, A, B position must be programmed before the canned cycle is commanded. This position is used as the "Start position".

This G code is used to perform rigid tapping for non-vertical holes. It may be used with a right-angle head to perform rigid tapping in the **X** or **Y** axis on a three-axis mill, or to perform rigid tapping along an arbitrary angle with a five-axis mill. The ratio between the feedrate and spindle speed must be precisely the thread pitch being cut.

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



G187 Setting the Smoothness Level (Group 00)

G-187 is an accuracy command that can set and control both the smoothness and max corner rounding value when cutting a part. The format for using G187 is **G187 Pn Ennnn**.

P Controls the smoothness level, P1(rough), P2(medium), or P3(finish).

E Sets the max corner rounding value, temporarily overriding Setting 85 "Max Corner Rounding". Setting 191 sets the default smoothness to the user specified "rough," "medium," or "finish" when G187 is not active. The "medium" setting is the factory default setting. **NOTE:** Changing setting 191 to "Finish" will take longer to machine a part. Use this setting only when needed for the best finish.

G187 Pm Ennnn sets both the smoothness and max corner rounding value. **G187 Pm** sets the smoothness but leaves max corner rounding value at its current value. **G187 Ennnn** sets the max corner rounding but leaves smoothness at its current value. **G187** by itself cancels the E value and sets smoothness to the default smoothness specified by Setting 191. G187 will be cancelled whenever "Reset" is pressed, M30 or M02 is executed, the end of program is reached, or E-stop is pressed.

G188 Get Program From PST (Group 00)

Calls the parts program for the loaded pallet based on the Pallet Schedule Table entry for the pallet.



M CODES (MISCELLANEOUS FUNCTIONS)

M Code Introduction

M-Codes are non axes moving commands for the machine. The format for an M code is the letter "M" followed by two numbers, for example M03.

Only one M code may be programmed per line of code. All M codes take effect at the end of the block.

M00 Stop Program

The M00 code is used to stop a program. It stops the axes, spindle, turns off the coolant (including Through Spindle Coolant). The next block (block after the M00) will be highlighted when viewed in the program editor. Pressing Cycle Start will continue program operation from the highlighted block.

M01 Optional Program Stop

M01 works the same as M00, except the Optional Stop feature must be on.

M02 Program End

The M02 code is used to end a program. Note that the most common way of ending a program is with an M30.

M03 / M04 / M05 Spindle Commands

M03 turns spindle on in the forward direction

M04 turns spindle on in the reverse direction

M05 Stops the spindle

Spindle speed is controlled with an S address code, for example, S5000 will command a spindle speed of 5000RPM.

Note: It is not recommended to run an M04 command with Through-Spindle Coolant (TSC).

M06 Tool Change

The M06 code is used to change tools, for example M06 T12 this will put tool 12 into the spindle. If the spindle is running, the spindle and coolant (including TSC) will be stopped by the M06 command.

M08 Coolant On / M09 Coolant Off

The M08 code will turn on the optional coolant supply and an M09 code will turn it off.

Also see M34/M35 for optional P-Cool and M88/89 for optional Through-the-spindle coolant.

Note: Coolant status is checked only at the start of a program, so a low coolant condition will not stop a program that is already running.

M10 Engage 4th Axis Brake/ M11 Release 4th Axis Brake

These codes will apply and release the brake to the optional 4th axis. The brake is normally engaged, so the M10 command is only required when an M11 has been used to release the brake.

M12 Engage 5th Axis Brake / M13 Release 5th Axis Brake

These codes will apply and release the brake to the optional 5th axis. The brake is normally engaged, so the M12 command is only required when an M13 has been used to release the brake.

M16 Tool Change

This M code behaves the same as M06. However M06 is the preferred method for commanding tool changes.

M17 Unclamp APC Pallet and Open APC Door/ M18 Clamp Pallet and Close Door

This M-code is used on vertical machining centers with pallet changers. It is used as a maintenance/test function only. Pallet changes should be commanded with an M50 command only.



M19 Orient Spindle (P and R values are an optional feature)

The M19 code is used to adjust the spindle to a fixed position.

The spindle will only orient to the zero position without the optional M19 spindle orient feature

The optional Orient Spindle function allows P and R address codes. For example, M19 P270 will orient the spindle to 270 degrees. The R-value allows the programmer to specify up to four decimal places, for example, M19 R123.4567.

M21-M28 Optional User M Function with M-Fin

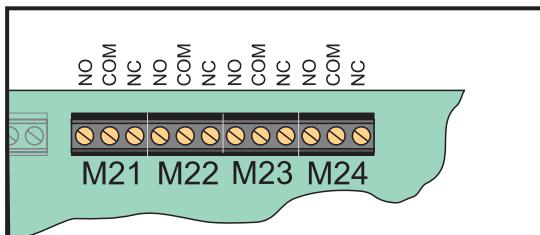
The M codes M21 through M28 are optional for user relays; each M code will activate one of the optional relays.

The Reset button will terminate any operation that is waiting for a relay-activated accessory to finish. Also see M51-58 and M61-68.

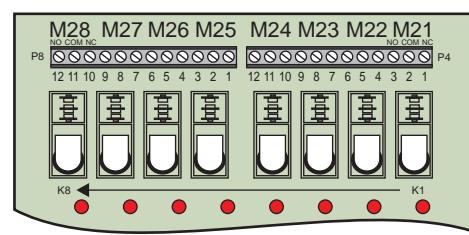
Some or all of the M21-25 (M21-M22 on Toolroom, Office and Mini mills) on the I/O PCB may be used for factory-installed options. Inspect the relays for existing wires to determine which have been used. Contact the Haas factory for more details.

M-Code Relays

These outputs can be used to activate probes, auxiliary pumps or clamping devices etc. The auxiliary devices are electrically connected to the terminal strip for the individual relay. The terminal strip has a position for, Normally Open (NO), Normally Closed (NC) and Common (COM).



Main I/O PCB M-Code Relays



Optional M-Code Relay Board
(Mounted above main I/O PCB)

Optional 8M-Code Relays

Additional M-Code relay functions can be purchased in banks of 8. A maximum of two 8M-code relay boards can be installed in the machine, for a total of 16 additional outputs.

A total of 4 banks of 8 relays are possible in the Haas system these are numbered from 0-3. Banks 0 and 1 are internal to the main I/O PCB. Bank 1 includes the M21-25 relays at the top of the IOPCB. Bank 2 addresses the first 8M option PCB. Bank 3 addresses the second 8M option PCB.

NOTE: Bank 3 may be used for some Haas installed options and may not be available. Contact the Haas factory for more details.

Only one bank of outputs may be addressable with M-codes at a time. This is controlled by parameter 352 "Relay Bank Select". Relays in the non-activated banks are only accessible with macro variables. Parameter 352 is shipped set to "1" as standard. The M-fin and probe cables (if equipped) are moved to the first 8M option PCB and parameter 352 is set to "2" when an 8M option is installed.

M30 Program End and Rewind

The M30 code is used to stop a program. It stops the spindle and turns off the coolant (including TSC) and the program cursor will return to the start of the program. M30 cancels tool length offsets.

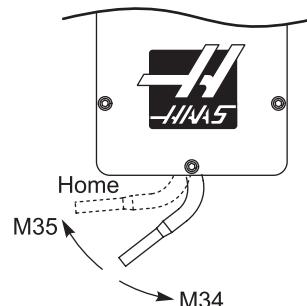


M31 Chip Conveyor Forward / M33 Chip Conveyor Stop

M31 starts the optional chip conveyor motor in the forward direction; the direction that moves the chips out of the machine. The conveyor will not turn if the door is open. It is recommended that the chip auger be used intermittently. Continuous operation will cause the motor to overheat.

M33 Stops Conveyor motion.

M34 Coolant Increment / M35 Coolant Decrement



M34 moves the optional P-Cool spigot one position away from the current position (farther from home).

M35 moves the coolant spigot one position towards the home position.

Do not rotate the coolant spigot by hand. Serious motor damage will occur.

M36 Pallet Part Ready

Used on machines with pallet changers. This M code delays the pallet change until the Part Ready button is depressed. A pallet change will occur after the pallet ready button is pressed (and the doors are closed). For example:

Onnnnn (program number)

M36 (Flash "Part Ready" light, wait until the button is pressed)

M50 (Perform pallet change after Part Ready button is pressed)

(Part Program)

M30

M39 Rotate Tool Turret

Tool changes should be commanded using M06. M39 is not normally required but is useful for diagnostic purposes or to recover from a tool changer crash.

The M39 code is used to rotate the side mount tool changer without performing a tool change. The desired tool pocket number (**Tn**) must be programmed previous to the M39.

M41 / M42 Low / High Gear Override

On machines with a transmission the M41 command is used to hold the machine in low gear and an M42 will hold the machine in high gear. Normally the spindle speed (Snnn) will determine which gear the transmission should be in.

M50 Execute Pallet Change

Used with a P value or pallet schedule table to perform a pallet change. Also see the pallet changer section.

M51-M58 Set Optional User M Codes

The M51 through M58 codes are optional for user interfaces. They will activate one of the relays and leave it active. Use M61-M68 to turn these off. The RESET key will turn off all of these relays.

See M21-M28 for details on the M-Code relays.



M59 Set Output Relay

This M code turns on a relay. An example of its usage is **M59 Pnn**, where “nn” is the number of the relay being turned on. An M59 command can be used to turn on any of the discrete output relays in the range from 1100 to 1155. When using Macros, M59 P1103 does the same thing as using the optional macro command #1103=1, except that it is processed at the end of the line of code.

M61-M68 Clear Optional User M Codes

The M61 through M68 codes are optional for user interfaces. They will turn off one of the relays. Use M51-M58 to turn these on. The Reset key will turn off all of these relays.

See M21-M28 for details on the M-Code relays.

M69 Clear Output Relay

This M code turns off a relay. An example of its usage is **M69 Pnn**, where “nn” is the number of the relay being turned off. An M69 command can be used to turn off any of the output relays in the range from 1100 to 1155. When using Macros, M69 P1103 does the same thing as using the optional macro command #1103=0, except that it is processed at the end of the line of code.

M75 Set G35 or G136 Reference Point

This code is used to set the reference point for G35 and G136 commands. It must be used after a probing function.

M76 / M77 Control Display Inactive / Control Display Active

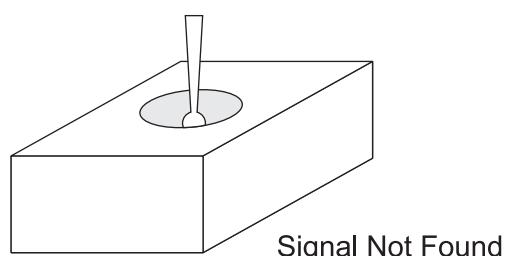
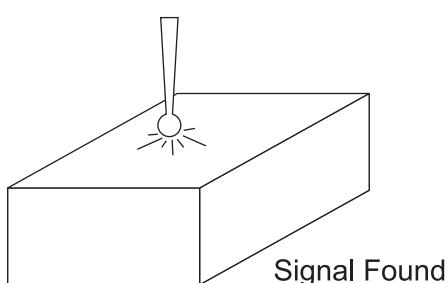
These codes are used to disable and enable the screen display. This M-Code is useful during the running of a large complicated program as refreshing the screen takes processing power that otherwise may be necessary to command the moves of the machine.

M78 Alarm if Skip Signal Found

This M-code is used with a probe. An M78 will generate an alarm if a programmed skip function (G31, G36 or G37) receives a signal from the probe. This is used when a skip signal is not expected, and may indicate a probe crash. This code can be placed on the same line as the skip G-code or in any block after.

M79 Alarm if Skip Signal Not Found

This M-code is used with a probe. An M79 will generate an alarm if a programmed skip function (G31, G36, or G37) did not receive a signal from the probe. This is used when the lack of the skip signal means a probe positioning error. This code can be placed on the same line as the skip G-code or in any block after.



M80 / M81 Auto Door Open / Close

The M80 opens the Auto Door and the M81 closes the close the Auto Door. The control pendant will beep when the door is in motion.

M82 Tool Unclamp

This code is used to release the tool from the spindle. It is used as a maintenance/test function only. Tool changes should be done using an M06.

M83 / M84 Auto Air Gun On / Off

An M83 will turn the Air Gun on, and an M84 will turn it off. Additionally, an M83 Pnnn (where nnn is in milliseconds) will turn it on for the specified time, then off automatically. The Auto Air Gun is also manual toggled on and off by pressing “Shift” followed by the “Coolant” button.



M86 Tool Clamp

This code will clamp a tool into the spindle. It is used as a maintenance/test function only. Tool changes should be done using an M06.

M88 Through-Spindle Coolant On / M89 Through-Spindle Coolant Off

The M88 code is used to turn on the through-spindle coolant (TSC) option, an M89 turns the coolant off.

Proper tooling, with a through-hole, must be in place before using the TSC system. Failure to use proper tooling will flood the spindle head with coolant and void the warranty.

Running an M04 (Spindle Reverse) command with TSC on is not recommended.

Sample Program

Note: The M88 command should be before the spindle speed command.

```
T1 M6; (TSC Coolant Through Drill)
G90 G54 G00 X0 Y0;
G43 H06 Z.5;
M88; (Turn TSC on)
S4400 M3;
G81 Z-2.25 F44. R.03;
M89 G80; (Turn TSC off)
G91 G28 Z0;
M30;
```

M95 Sleep Mode Sleep

Sleep mode is essentially a long dwell (pause). Sleep mode can be used when the user wants the machine to begin warming itself up so it can be ready for use upon the operators arrival. The format of the M95 command is:

M95 (hh:mm)

The comment immediately following the M95 must contain the hours and minutes that the machine is to sleep for. For example, if the current time were 6 p.m. and the user wanted the machine to sleep until 6:30 a.m. the next day, the following command would be used:

M95 (12:30)

The line(s) following the M95 should be axis moves and spindle warm-up commands.

M96 Jump If No Input

P Program block to go to when conditional test is met
Q Discrete input variable to test (0 to 63)

This code is used to test a discrete input for 0 (off) status. This is useful for checking the status of automatic work holding or other accessories that will generate a signal for the control. The Q value must be in the range 0 to 63, which corresponds to the inputs found on the diagnostic display (The upper left input is 0 and the lower right is input 63. When this program block is executed and the input signal specified by Q has a value of 0, the program block Pnnnn is performed (the Pnnnn line must be in the same program)).

M96 Example:

```
N05 M96 P10 Q8      (Test input #8, Door Switch, until closed);
N10                  (Start of program loop);
.
.
.
(Normal part of program);

N85 M21              (Execute an external user function)
N90 M96 P10 Q27      (Loop to N10 if spare input [#27] is 0);
N95 M30              (If spare input is 1 then end program);
```



M97 Local Sub-Program Call

This code is used to call a subroutine referenced by a line number (N) within the same program. A code is required and must match a line number within the same program. This is useful for simple subroutines within a program; does not require a separate program. The subroutine must end with an M99. An **Lnn** code in the M97 block will repeat the subroutine call that **nn** times.

M97 Example:

```
O0001
M97 P1000 L2      (L2 command will cause the program to run the N1000 line twice)
M30
N1000 G00 G90 G55 X0 Y0      (N line that will run after M97 P1000 is run)
S500 M03
G43 H01 Z1.
Z-.5
G01 G41 X.5 F100.
G03 YI-.5
G01 X0 G40
Z1. F50.
G91 G28 Z0
G90
M99
```

M98 Sub Program Call

This code is used to call a subroutine, the format is M98 Pnnnn (Pnnnn is the number of the program being called). The subprogram must be in the program list, and it must contain an M99 to return to the main program. An **Lnn** count can be put on the line containing the M98 and will cause the subroutine to be called **nn** times before continuing to the next block.

```
O0001      (Main Program number)
M98 P100 L4; (Call Sub-program, Sub-program Number, Loop 4 Times)
M30        (End of program)

O0100      (SUB-PROGAM NUMBER)
G00 G90 G55 X0 Y0
S500 M03
G43 H01 Z1.
Z-.5
G01 G41 X.5 F100.
G03 YI-.5
G01 X0 G40
Z1. F50.
G91 G28 Z0
G90
M99
```



M99 Sub-Program Return or Loop

This code is used to return to the main program from a subroutine or macro, the format is M99 Pnnnn (Pnnnn is the line in the main program to return to). It will cause the main program to loop back to the beginning without stopping when used in the main program.

Programming Notes - You can simulate Fanuc behavior by using the following code:

calling program:	Haas	Fanuc
	O0001	O0001

	N50 M98 P2	N50 M98 P2
	N51 M99 P100	...
	...	N100 (continue here)
	N100 (continue here)	...
	...	M30
subroutine:	O0002	O0002
	M99	M99 P100

M99 With Macros - If the machine is equipped with the optional macros, you can use a global variable and specify a block to jump to by adding #nnn=ddd in the subroutine and then using M99 P#nnn after the subroutine call.

M101 MOM (Minimum Oil Machining) Canned Cycle Mode

M101 tells the system to start MOM whenever the appropriate G-Code Canned Cycle is encountered (G73, G74, G76, G77, and G81 thru G89). Oil is dispensed for the **on time** duration whenever the tool is at the R-Plane.

I(on time): Canned Cycle Mode

On Time: Squirt duration in seconds (0.050 is 50 msec).

M102 MOM Mode

M102 tells the system to ignore the G-Code Canned Cycles and dispense oil whenever M102 is encountered in the program. Oil is dispensed for the **on time** duration at a periodicity dictated by the Time Between Squirts MOM.

I(on time) J(cycle time): MOM Mode

On Time: Squirt duration in seconds (0.050 is 50 msec).

Cycle Time: Time (in seconds) between squirts.

M103 Cancels MOM Mode.

M103 tells the system to cancel both Canned Mode and MOM Mode (no oil will be dispensed via MOM).

M103: Cancels MOM Mode.

M109 Interactive User Input

This M code allows a G-code program to place a short prompt (message) on the screen. A macro variable in the range 500 through 599 must be specified by a P code. The program can check for any character that can be entered from the keyboard by comparing with the decimal equivalent of the ASCII character (G47, Text Engraving, has a list of ASCII characters).

The following sample program will ask the user a Yes or No question, then wait for either a "Y" or an "N" to be entered. All other characters will be ignored.

```
N1 #501= 0.      (Clear the variable)
M109 P501        (Sleep 1 min?)
N5 IF [ #501 EQ 0. ] GOTO5      (Wait for a key)
IF [ #501 EQ 89. ] GOTO10      (Y)
IF [ #501 EQ 78. ] GOTO20      (N)
GOTO1           (Keep checking)
N10            (A Y was entered)
M95 (00:01)
GOTO30
N20            (An N was entered)
G04 P1.         (Do nothing for 1 second)
N30            (Stop)
M30
```



The following sample program will ask the user to select a number, then wait for a 1, 2 or a 3 to be entered. All other characters will be ignored.

```
O00234 (Sample program)
N1 #501= 0.      (Clear the variable)
M109 P501        (Pick 1, 2 or 3:)
N5 IF [ #501 EQ 0. ] GOTO5      (Wait for a key)
IF [ #501 EQ 49. ] GOTO10     (1)
IF [ #501 EQ 50. ] GOTO20     (2)
IF [ #501 EQ 51. ] GOTO30     (3)
GOTO1 (Keep checking)
N10    (A 1 was entered)
M95 (00:01)
GOTO30
N20    (A 2 was entered)
G04 P5. (Do nothing for 5 seconds)
N30    (A 3 was entered)
M30
```



SETTINGS

Introduction

Settings are machine conditions that have an affect on machine operation. In general, settings allow the operator or setup person to lock out, turn on or specify functions.

The settings are organized into pages of similar groups rather than numeric order. This puts associated settings on the same displayed page.

Changing Settings

Press the "SETNG/GRAFH" button to enter the setting pages. Use the page up/down buttons scroll through the setting pages, use the vertical cursor keys to move to the desired setting. Or enter a setting number and press the down arrow button to go directly to that setting.

There are two types of Settings, the first has built-in selections that can be scrolled through and then selected by pressing Write (Use the left/right arrow keys to scroll through the selections). The second type expect to have a value entered. There is help text at the top of the screen that advises of the type of setting.

In some cases the Emergency Stop button must be pressed in order to change a setting. The message "Servo is On" will display as a reminder that the Emergency stop button is not pressed.

Setting List

1 Auto Power Off Timer

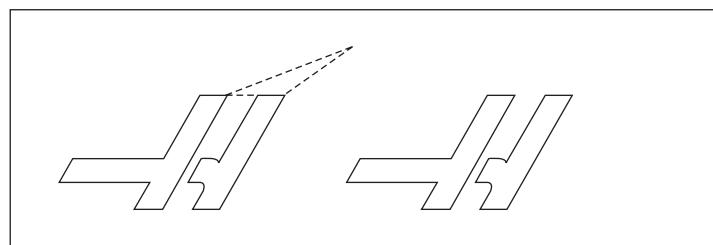
This setting is used to power-down the machine when it has not been used for some time. The value entered in this setting is the number of minutes the machine will remain idle until it is powered down. The machine will not be powered down while a program is running, and the time (number of minutes) will start back at zero anytime buttons is pressed or the jog handle is used. The auto-off sequence gives the operator a 15-second warning before power down, at which time pressing any button will stop the power down.

2 Power Off At M30

Powers down the machine at the end of a program (M30) if this setting is set to "On". The machine will give the operator a 30-second warning once an M30 is reached; pressing any button will interrupt the sequence.

4 Graphics Rapid Path

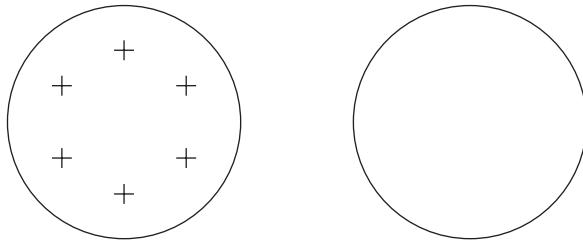
This setting changes the way a program is viewed in the Graphics mode. When it is OFF, rapid (non-cutting tool motions do not leave a path. When it is ON, rapid tool motions leave a dashed line on the screen.





5 Graphics Drill Point

This setting changes the way a program is viewed in Graphics mode. When it is ON, motion in the Z-axis will leave an X mark on the screen. When it is OFF, no additional marks are shown on the graphics display.



6 Front Panel Lock

This Setting disables the Spindle CW and CCW buttons when it is set to "ON".

7 Parameter Lock

Turning this Setting "ON" will stop the parameters from being changed, except for parameters 81-100. Note: When the control is powered up, this setting is on.

8 Prog Memory Lock

This setting locks out the memory editing functions (Alter, Insert, etc.) when it is set to "ON".

9 Dimensioning

This setting selects between inch and metric mode. When it is set to "INCH", the programmed units for **X**, **Y**, and **Z** are inches to 0.0001". When it is set to "METRIC", programmed units are millimeters to 0.001mm. All offset values are converted when this setting is changed from inches to metric, or vice versa. However, changing this setting will not automatically translate a program stored in memory; you must change the programmed axis values for the new units.

When set to INCH, the default G code is G20, when set to METRIC, the default G code is G21.

	INCH	METRIC
Feed	inches/min.	mm/min.
Max Travel	+/- 15400.0000	+/-39300.000
Min. Programmable Dimension	.0001	.001
Feed Range	.0001 to 300.000 in/min.	.001 to 1000.000

Axis Jog Keys		
.0001 Key	.0001 in/jog click	.001 mm/jog click
.001	.001 in/jog click	.01 mm/jog click
.01	.01 in/jog click	.1 mm/jog click
.1 Key	.1 in/jog click	1 mm/jog click

10 Limit Rapid At 50%

Turning this setting "ON" will limit the machine to 50% of its fastest non-cutting axis motion (rapids). This means, if the machine can position the axes at 700 inches per minute (ipm), it will be limited to 350ipm when this setting is On. The control will display a 50% rapid override message, when this setting is on. When it is OFF, the highest rapid speed of 100% is available.

11 Baud Rate Select

This setting allows the operator to change the rate at which data is transferred to/from the first serial port (RS-232). This applies to the upload/download of programs etc. and to DNC functions. This setting must match the transfer rate from the personal computer.



12 Parity Select

This setting defines parity for the first serial port (RS-232). When set to None, no parity bit is added to the serial data. When set to Zero, a 0 bit is added. Even and Odd work like normal parity functions. Make sure you know what your system needs, for example, XMODEM must use 8 data bits and no parity (set to "None"). This setting must match the transfer rate from the personal computer.

13 Stop Bit

This setting designates the number of stop bits for the first serial port (RS-232). It can be 1 or 2. This setting must match the transfer rate from the personal computer.

14 Synchronization

This changes the synchronization protocol between sender and receiver for the first serial port (RS-232). This setting must match the transfer rate from the personal computer.

When set to RTS/CTS, the signal wires in the serial data cable are used to tell the sender to temporarily stop sending data while the receiver catches up.

When set to **XON/XOFF, the most common setting**, ASCII character codes are used by the receiver to tell the sender to temporarily stop.

The selection DC CODES is like XON/XOFF, except that paper tape punch or reader start/stop codes are sent.

XMODEM is a receiver-driven communications protocol that sends data in blocks of 128 bytes. XMODEM has added reliability as each block is checked for integrity. XMODEM must use 8 data bits and no parity.

15 H & T Code Agreement

Turning this setting ON has the machine check to ensure that the **H** offset code matches the tool in the spindle. This check can help to prevent crashes.

Settings 16-21

These settings can be turned on in order to stop unfamiliar operators from altering the machine's functions and causing damage to the machine or workpiece.

16 Dry Run Lock Out

The Dry Run feature will not be available when this setting is turned ON.

17 Opt Stop Lock Out

The Optional Stop feature will not be available when this setting is on.

18 Block Delete Lock Out

The Block Delete feature will not be available when this setting is ON.

19 Feedrate Override Lock

The feedrate override buttons will be disabled when this setting is turned ON.

20 Spindle Override Lock

The spindle speed override buttons will be disabled when this setting is turned ON.

21 Rapid Override Lock

The axis rapid override buttons are disabled when this setting is turned ON.

22 Can Cycle Delta Z

This setting specifies the distance the Z-axis is retracted to clear chips during a G73 canned cycle. The range is 0.0 to 29.9999 inches (0-760 mm).

23 9xxx Progs Edit Lock

Turning this setting on will stop the 9000 series of programs from being viewed, edited or deleted. 9000 series programs cannot be uploaded or downloaded with this setting on. Note: 9000 series programs are usually macro programs.



24 Leader To Punch

This setting is used to control the leader (the blank tape at the beginning of a program) sent to a paper tape punch device connected to the first RS-232 port.

25 EOB Pattern

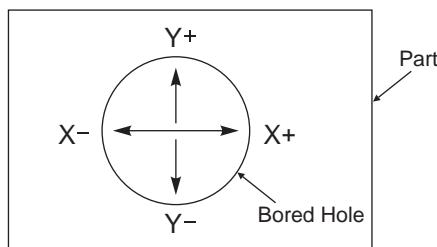
This setting controls the EOB (End of Block) pattern when data is sent and received to/from serial port 1 (RS-232). This setting must match the transfer rate from the personal computer.

26 Serial Number

This is the serial number of your machine. **It cannot be changed.**

27 G76/G77 Shift Dir.

This setting controls the direction the tool is shifted (moved) to clear a boring tool during a G76 or G77 canned cycle. Selections are: X+, X-, Y+, or Y-.



28 Can Cycle Act w/o X/Y

Turning this setting "ON" will cause the commanded canned cycle to complete without an X or Y command. The preferred method of operation is with this setting ON.

When this setting is OFF, the control will stop if a canned cycle is programmed without an X or Y axis move.

29 G91 Non-modal

Turning this setting ON will use the G91 command only in the program block it is in (non-modal). When it is OFF, and a G91 is commanded, the machine will use incremental moves for all axis positions.

30 4th Axis Enable

This setting initializes the control for a specific 4th axis. When this setting is OFF, the fourth axis is disabled; no commands can be sent to that axis. See setting 78 for 5th axis. Note that there are two selections "USER1" and "USER2" that can be used to set-up a unique rotary table.

31 Reset Program Pointer

When this setting is OFF, the Reset button will not change the position of the program pointer. When it is ON, the Reset button will move the program pointer to the beginning of the program.

32 Coolant Override

This setting controls how the coolant pump operates. The "Normal" selection allows the operator to turn the pump on and off manually or with M-codes. The "Off" selection will generate an alarm if an attempt is made to turn the coolant on manually or from a program. The "Ignore" selection will ignore all programmed coolant commands, but the pump can be turned on manually.

33 Coordinate System

This setting changes the way the Haas control recognizes the work offset system when a G52 or G92 is programmed. It can be set to FANUC, HAAS, or YASNAC.

Set to YASNAC

G52 becomes another work offset, just like using a G55.



Set to FANUC with G52

Any values in the G52 register will be added to all work offsets (global coordinate shift). This G52 value can be entered either manually or through a program. When FANUC is selected, pressing RESET, commanding an M30, or machine power down will clear out the value in G52.

Set to HAAS with G52

Any values in the G52 register will be added to all work offsets. This G52 value can be entered either manually or through a program. The G52 coordinate shift value is set to zero (zeroed) by manually entering zero, or by programming it with G52 X0, Y0, and/or Z0.

Set to YASNAC with G92:

Selecting YASNAC and programming a G92 X0 Y0, the control will enter the current machine location as a new zero point (Work Zero Offset), and that location will be entered into and viewed in the G52 list.

Set to FANUC or HAAS with G92:

Selecting FANUC or HAAS with a G92, it will work like the YASNAC setting, except that the new Work Zero location value will be loaded as new G92. This new value in the G92 list will be used, in addition to, the presently recognized work offset to define the new work zero location.

34 4th Axis Diameter

This is used to set the diameter of the A-axis (0.0 to 50 inches), which the control will use to determine the angular feedrate. The feedrate in a program is always inches per minute (or mm per minute), therefore, the control must know the diameter of the part being machined in the A-axis in order to compute angular feedrate. See setting 79 for 5th axis diameter.

35 G60 Offset

This is a numeric entry in the range 0.0 to 0.9999 inches. It is used to specify the distance an axis will travel past the target point prior to reversing. Also see G60.

36 Program Restart

When this setting is ONN, restarting a program from a point other than the beginning will direct the control to scan the entire program to ensure that the tools, offsets, G and M codes, and axis positions are set correctly before the program starts at the block where the cursor is positioned.

Note: The following M codes will be processed when Setting 36 is enabled:

M08 Coolant On	M41 Low Gear Override
M09 Coolant Off	M42 High Gear Override
M10 Engage 4th Axis Brake	M51-M58 Set Optional M Code
M11 Release 4th Axis Brake	M61-M68 Clear Optional M Code
M12 Engage 5th Axis Brake	M83 Air Gun On
M13 Release 5th Axis Brake	M84 Air Gun Off
M34 Increment Coolant Spigot Position	M88 Thru-Spindle Coolant ON
M35 Decrement Coolant Spigot Position	M89 Thru-Spindle Coolant OFF

When it is Off the program will start without checking the conditions of the machine. Having this setting Off may save time when running a proven program.

37 RS-232 Data Bits

This setting is used to change the number of data bits for serial port 1 (RS-232). This setting must match the transfer rate from the personal computer. Normally 7 data bits should be used but some computers require 8. XMODEM must use 8 data bits and no parity.

38 Aux Axis Number

This setting is used to select the number of external auxiliary axes added to the system. If it is set to 0, there are no auxiliary axes. If it is set to 1, there is a C axis. If it is set to 2, there are **C** and **U** axes. (Range 0-4).

39 Beep at M00, M01, M02, M30

Turning this setting ON will cause the keyboard beeper to sound when an M00, M01 (with Optional Stop active), M02 or an M30 is found. The beeper will continue until a button is pressed.



40 Tool Offset Measure

This setting selects how tool size is specified for cutter compensation. It can be set to either Radius or Diameter.

41 Add Spaces RS232 Out

When this setting is ON spaces are added between address codes when a program is sent out via RS-232 serial port 1. This can make a program much easier to read/edit on a personal computer (PC). When it is set to OFF, programs sent out the serial port have no spaces and are more difficult to read.

42 M00 After M06

Turning this setting ON will stop the program after a tool change and a message will be displayed stating this. The Cycle Start button must be pressed to continue the program.

43 Cutter Comp Type

This setting controls how the first stroke of a compensated cut begins and the way the tool is cleared from the part being cut. The selections can be A or B; see the cutter compensation section for examples.

44 Min F in Radius CC %

(Minimum feedrate in radius cutter compensation percent) This setting affects the feedrate when cutter compensation moves the tool toward the inside of a circular cut. This type of cut will slow down to maintain a constant surface feedrate. This setting specifies the slowest feedrate as a percentage of the programmed feedrate (range 1-100). The default is 50 (50%).

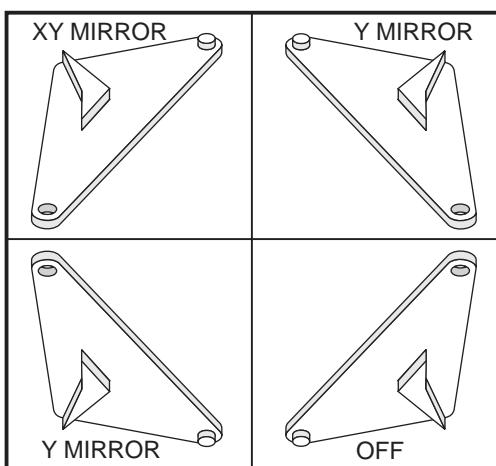
45 Mirror Image X-axis

46 Mirror Image Y-axis

47 Mirror Image Z-axis

48 Mirror Image A-axis

When one or more of these settings is ON, axis motion will be mirrored (reversed) around the work zero point. See also G101 Enable Mirror Image in G-codes section.



49 Skip Same Tool Change

In some program, the same tool may be called in the next section of a program or a subroutine. The control will do two changes and finish with the same tool in the spindle. Turning this setting ON will skip same tool, tool changes; a tool change will only occur if a different tool will be placed in the spindle.



50 Aux Axis Sync

This changes the synchronization between sender and receiver for the second serial port. The second serial port is used for auxiliary axes. The settings between the CNC control and the auxiliary axes must be the same.

Selecting "RTS/CTS" will tell the sender to temporarily stop sending data while the receiver catches up.

Selecting "XON/XOFF" uses ASCII character codes from the receiver to tell the sender to temporarily stop. **XON/XOFF is the most common setting.**

The "DC CODES" selection is like XON/XOFF, with the exception that start/stop codes are sent.

The "XMODEM" selection is receiver-driven, which sends data in blocks of 128 bytes. XMODEM gives the RS-232 communication added reliability because each block is checked for integrity.

51 Door Hold Switch Override (Safety Switch Override)

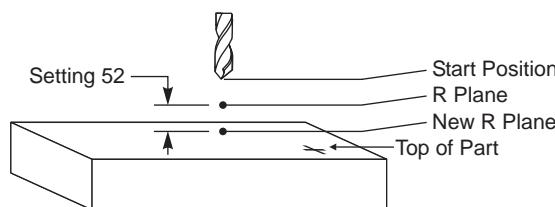
Selecting "OFF" will not allow a program to start when the doors are open, and opening a door will cause the running program to stop (the same as pressing Feed Hold).

Machines equipped with a hand held safety switch will go into feed hold if the safety switch is released.

When the control is powered on, this setting automatically returns to OFF.

52 G83 Retract Above R

(Range 0.0 to 30.00 inches or 0-761mm). This setting changes the way G83 (peck drilling cycle) behaves. Most programmers set the reference (R) plane well above the cut to ensure that the chip clearing motion actually allows the chips to get out of the hole. However this wastes time as the machine will "drill" through this empty distance. 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.



53 Jog w/o Zero Return

Turning this setting ON allows the axes to be jogged without zero returning the machine (finding machine home). This is a dangerous condition as the axis can be run into the mechanical stops and possibly damage the machine.

When the control is powered up, this setting automatically returns to OFF.

54 Aux Axis Baud Rate

This setting allows the operator to change the data rate for the second serial port (Auxiliary axis). This setting needs to match the value in the auxiliary axis control.

55 Enable DNC From MDI

Turning this setting "ON" will make the DNC feature available. DNC is selected in the control by pressing the MDI/DNC button twice.

The DNC Direct Numeric Control feature is not available when set to "OFF".

56 M30 Restore Default G

When this setting is ON, ending a program with M30 or pressing Reset will return all modal G codes to their defaults and will also reset the Feed Per Rev back to its default.

57 Exact Stop Canned X-Y

The rapid XY motion associated with a canned cycle may not achieve an exact stop when this setting is OFF. Turning this setting ON will ensure that the XY motion comes to an exact stop.



58 Cutter Compensation

This setting selects the type of cutter compensation used (FANUC or YASNAC). See the cutter compensation section.

59 Probe Offset X+

60 Probe Offset X-

61 Probe Offset Y+

62 Probe Offset Y-

These settings are used to define the displacement and size of the spindle probe. These four settings specify the travel distance and direction from where the probe is triggered to where the actual sensed surface is located. These settings are used by G31, G36, G136, and M75 codes. The values entered for each setting can be either positive or negative numbers.

Macros can be used to access these settings, see the Macro section for more information.

63 Tool Probe Width

This setting is used to specify the width of the probe used to test tool diameter. This setting only applies to the probing option; it is used by G35.

64 T. OFS Meas Uses Work

This setting changes the way the Tool Ofset Mesur (Tool Offset Measure) button works. When this is ON, the entered tool offset will be the measured tool offset plus the work coordinate offset (Z-axis). When it is OFF, the tool offset equals the Z machine position.

65 Graph Scale (Height)

This setting specifies the height of the work area that is displayed on the Graphics mode screen. The default value for this setting is the maximum height, which is the entire machine work area. Using the following formula can set a specific scale:

$$\text{Total Y travel} = \text{Parameter 20} / \text{Parameter 19}$$

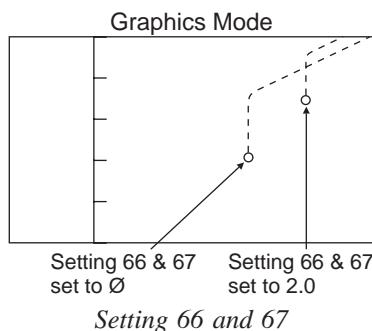
$$\text{Scale} = \text{Total Y travel} / \text{Setting 65}$$

66 Graphics X Offset

This setting locates the right side of the scaling window relative to the machine X zero position (see the Graphics section). Its default is zero.

67 Graphics Y Offset

This setting locates the top of the zoom window relative to the machine Y zero position (see the Graphics section). Its default is zero.





69 DPRNT Leading Spaces

This is an On/Off setting. When set to OFF the control will not use leading spaces generated by a macro DPRNT format statement. Conversely when set to ON the control will use leading spaces. The following example illustrates control behavior when this setting is OFF or ON.

#1= 3.0 ;	OUTPUT
G0 G90 X#1 ;	OFF ON
DPRNT[X#1[44]] ;	X3.0000 X 3.0000

Notice the space between the "X" and the 3 when the setting is ON. Information can be easier to read when this setting is ON.

70 DPRNT Open/Clos Dcode

This setting is available for compatibility with other systems. When the setting is ON, the macro statements will send DC control codes. When it is OFF, the DC codes will not be sent. See the Macros section.

71 Default G51 Scaling

This specifies the scaling for a G51 (See G-Code Section, G51) command when the P address is not used. The default is 1.000 (Range 0.001 to 8380.000).

72 Default G68 Rotation

This specifies the rotation, in degrees, for a G68 command when the R address is not used. It must be in the range 0.0000 to 360.0000°.

73 G68 Incremental Angle

This setting allows the G68 rotation angle to be changed for each commanded G68. When this switch is ON and a G68 command is executed in the Incremental mode (G91), then the value specified in the R address is added to the previous rotation angle. For example an R value of 10 will cause the rotation of the feature to be 10° the first time commanded, 20° the next time etc.

74 9xxxx PROGS Trace

This setting, along with Setting 75, is useful for debugging CNC programs. When Setting 74 is ON, the control will display the code in the macro programs (O9xxxx). When the setting is OFF, the control will not display the 9000 series code. The default setting is ON.

75 9xxx PROGS Single BLK

When Setting 75 is ON and the control is operating in Single Block mode, then the control will stop at each block of code in a macro program (O9xxxx) and wait for the operator to press Cycle Start. When Setting 75 is OFF, the macro program is run continuously, the control will not pause at each block, even if Single Block is on. The default setting is ON.

When Setting 74 and Setting 75 are both ON, the control acts normally. That is, all blocks executed are highlighted and displayed, and when in Single Block mode there is a pause before each block is executed.

When Setting 74 and Setting 75 are both OFF, the control will execute 9000 series programs without displaying the program code. If the control is in Single Block mode, no single-block pause will occur during the running of the 9000 series program.

When Setting 75 is ON and Setting 74 is OFF, then 9000 series programs will be displayed as they are executed.

76 Tool Release Lock Out

When this setting is ON, the tool release key on the keyboard is disabled.

77 Scale Integer F

This setting allows the operator to select how the control interprets an F value (feedrate) that does not contain a decimal point. (It is recommended that programmers always use a decimal point.) This setting helps operators run programs developed on a control other than HAAS. For Example F12:

Setting 77 OFF 0.0012units/minute
Setting 77 ON 12.0 units/minute



78 Fifth-axis Enable

When this setting is OFF, the fifth axis is disabled and no commands can be sent to that axis. In order to change this setting the Emergency Stop button must be pressed. See setting 30 for 4th axis. Note that there are two selections "USER1" and "USER2" that can be used to set-up a unique rotary table.

79 Fifth-axis Diameter

This is used to set the diameter of the B-axis (0.0 to 50 inches), which the control will use to determine the angular feedrate. The feedrate in a program is always inches per minute (or mm per minute), therefore, the control must know the diameter of the part being machined in the B-axis in order to compute angular feedrate. See setting 34 for 4th axis diameter.

80 Mirror Image B-axis

This is an On/Off setting. When it is OFF, axis motions occur normally. When it is on, B axis motion may be mirrored (or reversed) around the work zero point. Also see settings 45-48 and G101.

81 Tool At Power Up

When the Power Up/Restart key is pressed, the control will change to the tool specified in this setting. If zero (0) is specified, no tool change occurs at power up. The default setting is 1.

Setting 81, will cause one of the following actions to occur during a Power-up/Restart:

- A. If Setting 81 is set to zero, the carousel will be rotated to pocket #1. No tool change will be performed.
- B. If Setting 81 contains the tool number of the tool currently in the spindle, the carousel will remain at the same pocket and no tool change will be performed.
- C. If Setting 81 contains the tool number of a tool not currently in the spindle, the carousel will be rotated to pocket #1 and then to the pocket containing the tool specified by Setting 81. A tool change will be performed to change the specified tool into the spindle.

82 Language

Languages other than English are available in the Haas control. To change to another language, choose a language and press Enter.

83 M30 Resets Override

When this setting is ON, an M30 will restore any overrides (feedrate, spindle, rapid) to their default values (100%).

84 Tool Overload Action

This setting causes the specified action (Alarm, Feedhold, Beep, Autofeed) to occur anytime a tool becomes overloaded. (See the Tool Set up section)

Choosing "Alarm" will cause the machine to stop when the tool is overloaded.

When set to "Feedhold", the message "Tool Overload" will be displayed and the machine will stop in a feedhold situation when this condition occurs. Pressing any key will clear the message.

Selecting "Beep" will cause an audible noise (beep) from the control when the tool is overloaded.

When set to "Autofeed", the mill automatically limits the feedrate based on the tool load.

Autofeed Notes:

When tapping (rigid or floating), the feed and spindle overrides will be locked out, so the Autofeed feature will be ineffective (the control will appear to respond to the override buttons, by displaying the override messages).

The Autofeed feature should not be used when thread milling or auto reversing tapping heads, as it may cause unpredictable results or even a crash.

The last commanded feedrate would be restored at the end of the program execution, or when the operator presses Reset or turns off the Autofeed feature.

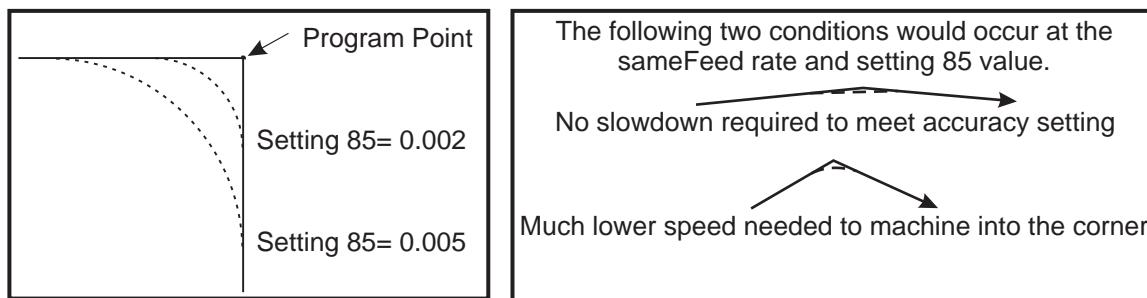


The operator may use the keyboard feedrate override buttons while the Autofeed feature is selected. These buttons will be recognized by the Autofeed feature as the new commanded feedrate as long as the tool load limit is not exceeded. However, if the tool load limit has already been exceeded, the control will ignore the feedrate override buttons.

85 Max Corner Rounding

This setting defines the machining accuracy of rounded corners within a selected tolerance. The initial default value is 0.025 inch (except on VF-5 trunion). If this setting is zero (0), the control acts as if an exact stop is commanded in each motion block.

G187 can be used in the program to alter the effective value of Setting 85, without changing the setting.



Setting 85 Examples

86 M39 Lockout

This is an On/Off setting. When it is ON, M39 commands are ignored.

87 M06 Resets Override

This is an On/Off setting. When this setting is ON and an M06 is commanded, any overrides are canceled and set to their programmed values or defaults.

88 Reset Resets Overrides

This is an On/Off setting. When it is ON and the Reset key is pressed, any overrides are canceled and set to their programmed values or defaults.

90 Max Tools To Display

This setting limits the number of tools displayed on the Tool Geometry screen. The range of this setting is 1 to 200.

91 Advanced Jog

Turning this setting ON enables the Index Jog and Jog Travel Limits features. This setting is used on the Tool Room series of machines. See the Toolroom Mill addendum for more information on these features.

100 Screen Saver Delay

When the setting is zero, the screen saver is disabled. If setting is set to some number of minutes, then after that time with no keyboard activity, the screen saver will activate the Haas logo that will change position every 2 seconds (deactivate with any key press, handle jog or alarm). The screen saver will not activate if the control is in Sleep, Jog, Edit, or Graphics mode.

101 Feed Overide -> Rapid

Turning this setting ON and pressing Handle Control Feedrate will cause, the jog handle to effect both the feedrate and the rapid rate overrides. Setting 10 affects the maximum rapid rate.

103 CYC Start/FH Same Key

The Cycle Start button must be pressed and held to run a program when this setting is ON. When Cycle Start is released, a feed hold is generated.

This setting cannot be turned on while Setting 104 is on. When one of them is set to ON, the other will automatically turn off.



104 Jog Handl to SNGL BLK

The jog handle can be used to single-step through a program when this setting is ON. Reversing the jog handle direction will generate a feed hold.

This setting cannot be turned ON while Setting 103 is on. When one of them is set to ON, the other will automatically turn off.

108 Quick Rotary G28

Turning this setting ON will return the rotary unit back to zero using the shortest distance.

For example if the rotary unit is at 10° and a zero return is commanded, the rotary table will rotate 350° if this setting is OFF. If the setting is ON the table will rotate -10°.

In order to use Setting 108, the parameter bit CIRC. WRAP. (10) must be set to 1 on parameter 43 for the A axis and Parameter 151 for the B-axis. If the parameter bit(s) are not set to 1 the control will ignore setting 108.

109 Warmup Time in Min.

This is the number of minutes (up to 300 minutes from power-up) during which the compensations specified in Settings 110-112 are applied.

Overview – When the machine is powered on, if Setting 109, and at least one of Settings 110, 111 or 112, are set to a nonzero value, the following warning will be displayed:

CAUTION! Warm up Compensation is specified!
Do you wish to activate
Warm up Compensation (Y/N)?

If a 'Y' is entered, the control immediately applies the total compensation (setting 110,111, 112), and the compensation begins to decrease as the time elapses. For instance, after 50% of the time in Setting 109 has elapsed, the compensation distance, will be 50%.

To "restart" the time period, it is necessary to power the machine off and on, and then answer "yes" to the compensation query at start-up. **Caution:** Changing settings 110, 111 or 112 while compensation is in progress can cause a sudden movement of up to 0.0044 inch.

The amount of remaining warmup time is displayed on the bottom right hand corner of the Diagnostics Inputs 2 screen using the standard hh:mm:ss format.

110 Warmup X Distance

111 Warmup Y Distance

112 Warmup Z Distance

Settings 110, 111 and 112 specify the amount of compensation (max = $\pm 0.0020"$ or ± 0.051 mm) applied to the axes. Setting 109 must have a value entered for settings 110-112 to have an affect.

114 Conveyor Cycle Time (Minutes)

115 Conveyor On-time (Minutes)

These two settings control the optional chip conveyor. Setting 114 controls how often the cycle is to be repeated, and Setting 115 specifies the number of minutes the conveyor will run. For example, if Setting 114 is set to 30 and Setting 115 is set to 2, the chip conveyor will turn itself on every half hour (30 minutes), run for two minutes, then turn itself off.

116 Pivot Length

Setting 116 is set when the machine is first built and never changed. Only a qualified service technician should modify this setting.

117 G143 Global Offset

This setting is provided for customers who have several 5-axis Haas mills and want to transfer the programs and tools from one to another. The pivot length difference (difference between Setting 116 for each machine) can be entered into this setting, and it will be applied to the G143 tool length compensation.



118 M99 Bumps M30 CNTRS

When this setting is ON, an M99 will add one to the M30 counters (these are visible on the CURNT COMNDS displays). Note that an M99 will only increase the counters as it occurs in a main program, not a sub-program.

119 Offset Lock

Turning the setting ON will not allow the values in the Offset display to be altered. However, programs that alter offsets will still be able to do so.

120 Macro VAR Lock

Turning this setting ON will not allow the macro variables to be altered. However, programs that alter macro variables will still be able to do so.

121 APC Pal. One Load X

122 APC Pal. One Load Y

123 APC Pal. One Unload X

124 APC Pal. One Unload Y

125 APC Pal. Two Load X

126 APC Pal. Two Load Y

127 APC Pal. Two Unload X

128 APC Pal. Two Unload Y

129 APC Pal. 1 and 2 Safe X Pos

Settings 121-129 are the pallet positions for the X and Y axis. They are set at the time the APC is installed and should not need altering.

130 Tap Retract Speed

This setting affects the retract speed during a tapping cycle. Entering a value, such as 2, will command the mill to retract the tap twice as fast as it went in. If the value is 3, it will retract three times as fast. A value of 0 or 1 will have no affect on the retract speed. (Range 0-4)

Entering a value of 2, is the equivalent of using a J code of 2 for G84 (Tapping canned cycle). However, specifying a J code for a rigid tap will override setting 130.

Note: If the machine does not have the Rigid Tap option, this setting has no effect.

131 Auto Door

This setting supports the Auto Door option. It should be set to ON for machines with an autodoor. Also see M80/81 (Autodoor Open/Close M-codes).

The door will close when Cycle Start is pressed and will open when the program reaches an M00, M01 (with Optional Stop turned on) or M30 and the spindle has stopped turning.

133 Repeat Rigid Tap

This setting ensures that the spindle is oriented during tapping so that the threads will line up when a second tapping pass, in the same hole, is programmed.

134 Connection Type

This setting can be FLOPPY, NET, or ZIP. When it is set to FLOPPY, loading and saving programs is performed using the floppy disk drive.

When it is set to NET, loading and saving occur through the user-supplied network connection.

When it is set to ZIP, program loading and saving occurs through the user-supplied Zip™ drive.

135 Network Type

This setting can be NONE, NOVELL, NT/IPX, NT/TCP or ADV/TCP. It specifies the user-supplied network connection type. When it is set to NONE, only a floppy disk or a user-supplied Zip™ drive is accessible. If ADV/TCP is selected, a setting page will appear in place of setting 135. To revert back, change "Net Type" to "None".



136 Server

This setting contains the user-entered server name (up to 8 characters long.) Enter a semicolon (EOB symbol) if the machine is not linked to a server.

137 Username

This setting contains the user-entered account name (up to 8 characters long.) Enter a semicolon (EOB symbol) if the machine is not linked to a server.

138 Password

This setting contains the user-entered password (up to 8 characters long.) Enter a semicolon (EOB symbol) if the machine is not linked to a server.

139 Path

This setting contains the user-entered Novell-Path or Windows NT root directory name (up to 18 characters long). For a Novell network, this is the users path name; for example, U:\USERS\JOHNDOE. For a Microsoft network (i.e. Windows NT), this is the root directory\desired directory name; for example, USERS\JOHNDOE. Enter a semicolon (EOB symbol) if the machine is not linked to a server.

140 TCP ADDR

This setting is used only for TCP networks and contains the user-entered TCP/IP address in the server domain (up to 15 characters long). For example: 192.168.1.2. Enter a semicolon (EOB symbol) if the machine is not linked to a server.

141 Subnet

This setting is used only for TCP networks and contains the user-entered subnet mask (up to 15 characters long; for example, 255.255.255.0). Enter a semicolon (EOB symbol) if the machine is not linked to a server.

142 Offset CHNG Tolerance

This setting generates a warning message if an offset is changed by more than the amount entered for this setting. The following prompt will be displayed: "XX changes the offset by more than Setting 142! Accept (Y/N)?" if an attempt is made to change an offset by more than the entered amount (either positive or negative). If "Y" is entered, the control updates the offset as usual; otherwise, the change is rejected.

If "Y" is entered, the control updates the offset as usual, otherwise, the change is rejected.

143 Machine Data Collect

This setting enables the user to extract data from the control using a Q command sent through the RS-232 port. This feature is software-based and requires an additional computer to request, interpret and store data from the control. Note that the control will only respond to a Q command when this setting is ON. The following output format is used:

<STX> <CSV response> <ETB> <CR/LF> <0x3E>

("CSV" means Comma Separated Variable that is a line of information in which each piece of information is separated from the next with a comma).

Note: STX = 0x02 (ctrl-B); ETB = 0x17 (ctrl-W)

The following commands can be used:

Q100 - Machine Serial Number

Q101 - Control Software Version

Q102 - Machine Model Number

Q104 - Mode (LIST PROG, MDI, MEM, JOG, etc.)

Q200 - Tool Changes (total)

Q201 - Tool Number in use

Q300 - Power-on Time (total)

Q301 - Motion Time (total)

Q303 - Last Cycle Time

Q304 - Previous Cycle Time

Q400 - not currently used

Q401 - not currently used



Q402 - M30 Parts Counter #1 (resettable at control)

Q403 - M30 Parts Counter #2 (resettable at control)

Q500 - Three-in-one (PROGRAM, Oxxxxx, STATUS, PARTS, xxxx)

If the control is busy, the control will output "STATUS, BUSY." If a request is not recognized, the control will output "UNKNOWN."

In addition, when Setting 143 is ON, the user has the ability to request the contents of any macro or system variable by using the Q600 command, for example, "Q600 xxxx" where xxxx is the macro variable. Macro variables 1 thru 999 and 2001 thru 2800 can be written to using an "E" command, for example, "Exxxx yyyy.yyyyy" where xxxx is the macro variable and yyyy.yyyyy is the new value. Note that this command should only be used when there are no alarms present.

This feature can be enhanced with the installation of the following optional parts, M-code relay board, a power-on relay and a new Emergency Stop switch; contact your dealer for information on these parts. Once installed output relays 40 thru 47, a power-on relay and the Emergency Stop switch are used to communicate the status of the control.

The following communications will be received (* are only available when used with the optional parts):

* E-STOP contacts. This will be closed when the E-STOP button is pushed.

* Power ON - 115 VAC. Indicates that the control is turned ON. It should be wired to a 115 VAC coil relay for interface.

* Spare Output Relay 40. Indicates that the control is In-Cycle (running.)

* Spare Output Relay 41 and 42:

11 = MEM mode & no alarms (AUTO mode.)

10 = MDI mode & no alarms (Manual mode.)

01 = Single Block mode (Single mode)

00 = other modes (zero, DNC, jog, list prog, etc.)

* Spare Output Relay 43 and 44:

11 = Feed Hold stop (Feed Hold.)

10 = M00 or M01 stop

01 = M02 or M30 stop (Program Stop)

00 = none of the above (could be single block stop or RESET.)

* Spare Output Relay 45 (Feed Rate Override is active and Feed Rate is NOT 100%)

* Spare Output Relay 46 (Spindle Speed Override active and Spindle Speed is NOT 100%)

* Spare Output Relay 47 (Control is in EDIT mode)

144 Feed Overide->Spindle

This setting is intended to keep the chip load constant when an override is applied. When this setting is ON, any feedrate override will also be applied to the spindle speed, and the spindle overrides will be disabled.

146 APC Pallet 3 Load X

147 APC Pallet 3 Load Y

148 APC Pallet 3 Unload X

149 APC Pallet 3 Unload Y

150 APC Pallet 4 Load X

151 APC Pallet 4 Load Y

152 APC Pallet 4 Unload X

153 APC Pallet 4 Unload Y

154 APC Pallet 3 & 4 Safe X

See Settings 121-129



155 Load Pocket Tables

This setting should only be used when a software upgrade is performed and/or memory has been cleared and/or the control is re-initialized. In order to replace the contents of the side-mount tool changer pocket tool table with the data from the file, the setting must be ON.

If this setting is OFF when loading an Offset file from a floppy disk or RS-232, the contents of the Pocket Tool table will be unaltered. Setting 155 automatically defaults to OFF when the machine is turned on.

156 Save Offset with PROG

Turning this setting ON will have the control save the offsets in the same file as the programs, but under the heading O999999. The offsets will appear in the file before the final % sign.

157 Offset Format Type

This setting controls the format in which offsets are saved with programs.

When it is set to A the format looks like what is displayed on the control, and contains decimal points and column headings. Offsets saved in this format can be more easily edited on a PC and later reloaded into the control.

When it is set to B, each offset is saved on a separate line with an N value and a V value.

158,159,160 XYZ Screw Thermal COMP%

These settings can be set from -30 to +30 and will adjust the existing screw thermal compensation by -30% to +30% accordingly.

162 Default to Float

When this setting is ON, the control will add a decimal point to values entered without a decimal point (for certain address codes). When this setting is Off, the values following the address codes are treated as machinist's notation (i.e., thousandths or ten-thousandths.) For example:

	Value entered	With Setting OFF	With Setting ON
In Inch mode	X-2	X-.0002	X-2.
In MM mode	X-2	X-.002	X-2.

This feature applies to the following address codes:

X,Y,Z,A,B,C,E,F,I,K,U,W

D (except with G73)

R (except with G71 in YASNAC mode)

Note that this setting affects the interpretation of all programs entered either manually or from disk or via RS-232. It does not alter the effect of setting 77 Scale Integer F.

163 Disable .1 Jog Rate

This setting disables the highest jog rate. If the highest jog rate is selected, the next lower rate is automatically selected instead.

164 Rotary Increment

This setting applies to the Rotary Index button on the EC300. It specifies the rotation for the rotary table in the load station. It should be set to a value from -360 to 360 degrees. For example, entering "90" will rotate the pallet 90° each time the rotary index button is pressed. If it is set to zero, the rotary table will not rotate.



167-186 Periodic Maintenance

There are 14 items that can be monitored, as well as six spare items, in the periodic maintenance settings. These settings will allow the user to change the default number of hours for each item when it is initialized during use. If the number of hours is set to zero, the item will not appear in the list of items shown in the maintenance page of current commands.

- 167 Coolant Replacement default in power-on hours
- 168 Control Air Filter Replacement default in power-on hours
- 169 Oil Filter Replacement default in power-on hours
- 170 Gearbox Oil Replacement default in power-on hours
- 171 Coolant Tank Level Check default in power-on hours
- 172 Way Lube Level Check default in motion-time hours
- 173 Gearbox Oil Level Check default in power-on hours
- 174 Seals/Wipers Inspection default in motion-time hours
- 175 Air Supply Filter Check default in power-on hours
- 176 Hydraulic Oil Level Check default in power-on hours
- 177 Hydraulic Filter Replacement default in motion-time hours
- 178 Grease Fittings default in motion-time hours
- 179 Grease Chuck default in motion-time hours
- 180 Grease Tool Changer Cams default in tool-changes
- 181 Spare Maintenance Setting #1 default in power-on hours
- 182 Spare Maintenance Setting #2 default in power-on hours
- 183 Spare Maintenance Setting #3 default in motion-time hours
- 184 Spare Maintenance Setting #4 default in motion-time hours
- 185 Spare Maintenance Setting #5 default in tool-changes
- 186 Spare Maintenance Setting #6 default in tool-changes

187 Machine Data Echo

This setting can be set ON or OFF. When set to ON, the data collection Q commands issued from the user's PC will be displayed on the PC screen. When set to OFF, these commands will not be displayed.

191 Default Smoothness

This setting may be set to ROUGH, MEDIUM, or FINISH and uses parameters 302, 303, 314, 749, and 750-754 and G187 to set the smoothness and a maximum corner rounding factor. The default values are used when not overridden by a G187 command.





MAINTENANCE

GENERAL REQUIREMENTS

Operating Temperature Range: 41°F to 104°F (5 to 40°C)
Storage Temperature Range: -4°F to 158°F (-20 to 70°C)
Ambient Humidity: 20% – 95% relative humidity, non-condensing
Altitude: 0-7000 ft.

ELECTRICITY REQUIREMENTS

All Machines Require:

AC input power is three phase Delta or Wye power, except that the power source must be grounded (e.g. leg or center leg for delta, neutral for Wye)
Frequency range of 47-66 Hz
Line voltage that does not fluctuate more than ± 10%
Harmonic distortion not to exceed 10% of the total RMS voltage

20-15 HP System (Standard VF and 10K, EC300, EC400)

	195-260V Voltage Requirements	354-488V High-Voltage Requirements
Power Supply ¹	50 AMP	25 AMP
Haas Circuit Breaker	40 AMP	20 AMP
If service run from elec. panel is less than 100' use:	8 GA. WIRE	12 GA. WIRE
If service run from elec. panel is more than 100' use:	6 GA. WIRE	10 GA. WIRE

40-30 HP System (50 Taper, 40 Taper HT 10K, VF Super Speed, EC-300, EC-400 12K, VM)

	195-260V Voltage Requirements	354-488V High-Voltage Requirements ²
Power Supply ¹	100 AMP	50 AMP
Haas Circuit Breaker	80 AMP	40 AMP
If service run from elec. panel is less than 100' use:	4 GA. WIRE	8 GA. WIRE
If service run from elec. panel is more than 100' use:	2 GA. WIRE	6 GA. WIRE

40-30 HP System (VS 1/3, HS 3-7 incl R models)

	195-260V Voltage Requirements
Power Supply	125 AMP
Haas Circuit Breaker	100 AMP
If service run from ele. panel is less than 100' use:	2 GA. WIRE
If service run from ele. panel is more than 100' use:	0 GA. WIRE

WARNING!

A separate earth ground wire of the same conductor size as the input power is required to be connected to the chassis of the machine. This ground wire is required for operator safety and for proper operation. This ground must be supplied from the main plant ground at the service entrance, and should be routed in the same conduit as the input power to the machine. A local cold water pipe or ground rod adjacent to the machine cannot be used for this purpose.



Input power to the machine must be grounded. For wye power, the neutral must be grounded. For delta power, a central leg ground or one leg ground should be used. The machine will not function properly on ungrounded power. (This is not a factor with the External 480V Option.)

The rated horsepower of the machine may not be achieved if the imbalance of the incoming voltage is beyond an acceptable limit. The machine may function properly, yet may not deliver the advertised power. This is noticed more often when using phase converters. A phase converter should only be used if all other methods cannot be used.

The maximum leg-to-leg or leg-to-ground voltage should not exceed 260 volts, or 504 volts for high-voltage machines with the Internal High Voltage Option.

¹The current requirements shown in the table reflect the circuit breaker size internal to the machine. This breaker has an extremely slow trip time. It may be necessary to size the external service breaker up by 20-25%, as indicated by "power supply", for proper operation.

²The high-voltage requirements shown reflect the Internal 400V configuration which is standard on European machines. Domestic and all other users must use the External 480V option.

AIR REQUIREMENTS

The mill requires a minimum of 100 psi at the input to the pressure regulator on the back of the machine. A volume of 4 scfm (9scfm for EC and HS mills) is also necessary. This should be supplied by at least a two-horsepower compressor, with a minimum 20-gallon tank, that turns on when the pressure drops to 100 psi.

NOTE: Add 2 scfm to the minimum air requirements (below) if the operator will be using the air nozzle during pneumatic operations.

Machine Type	Main Air Regulator	Input Airline Hose Size
EC-300	85 psi	1/2" I.D.
EC-400	85psi	1/2" I.D.
EC-1600	85psi	1/2" I.D.
HS 3/4/6/7 incl R models	85psi.	1/2" I.D.
VF-1 - VF-11 (40Taper), VM	85psi	3/8" I.D.
VF-5 - VF-11 (50 Taper)	85psi	1/2" I.D.
VR Series	85psi	1/2" I.D.
VS 1/3	85psi	1/2" I.D.

The recommended method of attaching the air hose is to the barb fitting at the back of the machine with a hose clamp. If a quick coupler is desired, use a 1/2" coupler.

NOTE: Excessive oil and water in the air supply will cause the machine to malfunction. The air filter/regulator has an automatic bowl dump that should be empty before starting the machine. This must be checked for proper operation monthly. Also, excessive contaminants in the air line may clog the dump valve and cause oil and/or water to pass into the machine.

NOTE: Auxiliary air connections should be made on the unregulated side of the air filter/regulator.

WARNING!

When the machine is operating and the pressure gauge (on the machine regulator) drops by more than 10 psi during tool changes or pallet changes, insufficient air is being supplied to the machine.



MAINTENANCE SCHEDULE

The following is a list of required regular maintenance for the machining center. These required specifications must be followed in order to keep your machine in good working order and protect your warranty.

Interval	Maintenance Performed
Daily	<ul style="list-style-type: none">• Check coolant level each eight-hour shift (especially during heavy TSC usage).• Check way lube lubrication tank level.• Clean chips from way covers and bottom pan.• Clean chips from tool changer.• Wipe spindle taper with a clean cloth rag and apply light oil.
Weekly	<ul style="list-style-type: none">• Check Through the Spindle Coolant (TSC) filters. Clean or replace element if needed.• Check for proper operation of auto drain on filter regulator.• On machines with the TSC option, clean the chip basket on the coolant tank. Remove the tank cover and remove any sediment inside the tank. Disconnect the coolant pump from the cabinet and power off the machine before working on the coolant tank. <p>Do this MONTHLY for machines without the TSC option.</p> <ul style="list-style-type: none">• Check air gauge/regulator for 85 psi. Check the spindle air pressure regulator for 17 psi. For 15K-spindle machines, check spindle air pressure regulator for 20 psi.• For machines with the TSC option, place a dab of grease on the V-flange of tools. <p>Do this MONTHLY for machines without the TSC option.</p> <ul style="list-style-type: none">• Clean exterior surfaces with mild cleaner. DO NOT use solvents.• Check the hydraulic counterbalance pressure according to the machine's specifications.
Monthly	<ul style="list-style-type: none">• Check oil level in gear box. For 40 taper spindles: Remove inspection cover beneath spindle head. Add oil slowly from top until oil begins dripping from overflow tube at bottom of sump tank. For 50 taper spindles: Check oil level in sight glass. Add from side of gearbox if necessary.• Inspect way covers for proper operation and lubricate with light oil, if necessary.• Place a dab of grease on the outside edge of the guide rails of the tool changer and run through all tools.• EC-400 Clean the locating pads on the A-axis and the load station. This requires removing the pallet
Six Months	<ul style="list-style-type: none">• Replace coolant and thoroughly clean the coolant tank.• Check all hoses and lubrication lines for cracking.• Check the rotary A-axis. If necessary add oil (Mobil SHC-630). The correct oil level is halfway on the sight glass.• Check SMTC oil level in sight glass, (see Side Mount Tool Changer Oil Level in this section).
Annually	<ul style="list-style-type: none">• Replace the gearbox oil. Drain the oil from the bottom of the gearbox. Remove inspection cover beneath spindle head. Add oil slowly from top until oil begins dripping from overflow tube at bottom of sump tank. For 50 taper spindles, add oil from the side of the transmission.• Check oil filter and clean out residue at bottom of filter.
2 years	<ul style="list-style-type: none">• Replace air filter on control box.• EC-400 Replace the Rotary A-axis oil



PERIODIC MAINTENANCE

A periodic maintenance page is found on the Current Commands screens titled "Scheduled Maintenance" and accessed by pressing Page Up or Page Down to activate and deactivate a series of tests.

An item on the list can be selected by pressing the up and down arrow keys. The selected item is then activated or deactivated by pressing Origin. If an item is active, the remaining hours will be displayed, a deactivated item will display, “—” instead.

The maintenance item time is adjusted by using the left and right arrows. Pressing the Origin key will reinstate the default time.

Items are tracked either by the time accumulated while power is on (ON-TIME) or by cycle-start time (CS-TIME). When the time reaches zero the message “Maintenance Due” is displayed at the bottom of the screen (a negative number of hours indicates the hours past due).

This message is not an alarm and does not interfere with machine operation in any way. After the necessary maintenance has been performed, the operator can select that item on the “Scheduled Maintenance” screen, press the Origin button to deactivate it, then press Origin again to reactivate it with the default number of hours remaining.

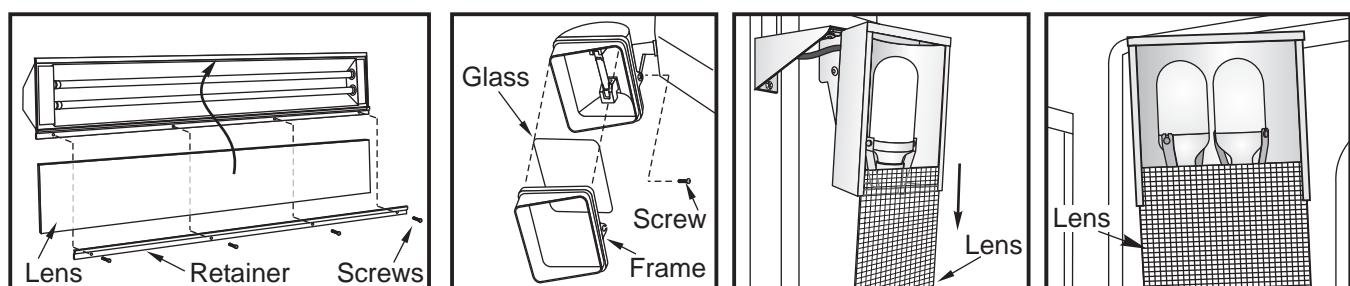
WINDOWS / GUARDING

Polycarbonate windows and guarding can be weakened by exposure to cutting liquids and chemicals that contain amines. It is possible to lose up to 10% of the remaining strength annually. If degradation is suspected, replace the window. It is recommended that the windows be replaced every two years.

Windows and guarding should be replaced if damaged or severely scratched. Replace damaged windows immediately.

WORKLIGHT

There are four types of worklights for the Haas mills. Turn off power to the machine at the main breaker before doing any work on the mill.



Note: The power for the worklight comes from the GFI circuit. If the worklight does not turn on, check this first, it can be reset at the side of the control panel.

CHIP AUGER

During normal operation, most chips are discharged from the machine at the discharge tube. However, very small chips may flow through the drain and collect in the coolant tank strainer. To prevent drain blockage, clean this strainer regularly. Should the drain become clogged and cause coolant to collect in the pan, stop the machine, loosen the chips blocking the drain, and allow the coolant to drain. Empty the coolant tank strainer, then resume operation.



SPINDLE AIR PRESSURE

Verify Spindle air pressure using the gauge located behind the main air regulator. VF, VR, and VS mills should be set to 17 psi. EC-series and HS Series should be set to 25psi. Adjust if necessary.

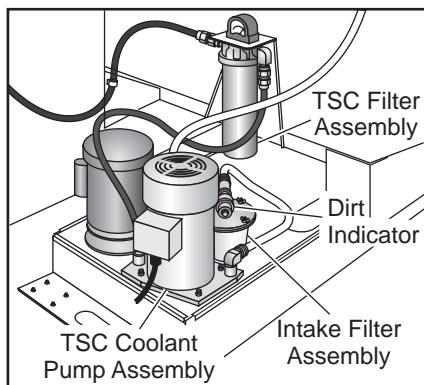
12K & 15K Spindle

The air pressure for 12K &15K Spindles is 20 psi. The 12K &15K Spindles require higher pressure to slightly reduce the delivery speed and amount of oil on the bearings.

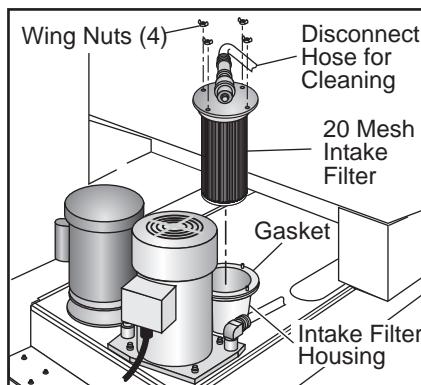
TSC MAINTENANCE

The TSC pump is a precision gear pump and will wear out faster and lose pressure if abrasive particles are present in the coolant.

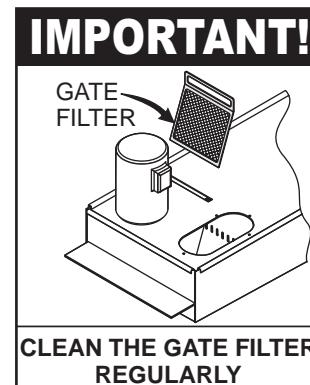
- Check the dirt indicator on the 20-micron mesh filter with the TSC system running and no tool in the spindle. Change the element when the indicator reaches the red zone.
- Clean the pump intake filter when indicator is in the red zone. Reset indicator with button. Gate filters can be cleaned with a wire brush.
- After changing or cleaning filter elements, run the TSC system with no tool in spindle for at least one minute to prime system.
- Coolant will be used more quickly when the TSC system is in use. Make sure to keep the coolant level up and check the level more frequently (check after every eight hour shift). **Premature wear of the pump can result from running with a low coolant level in the tank.**



TSC Coolant Pump Assembly



Cleaning the Intake Filter



Gate Filter

Warnings

Use of coolants with extremely low lubricity can damage the TSC coolant tip and pump.

Shortened pump life, reduction of pressure and increased maintenance are normal and to be expected in abrasive environments and are not covered by warranty. A special filter, in addition to the standard filter should be used; contact Haas for recommendations.

Machining of ceramics and the like voids all warranty claims for wear and is done entirely at customer's risk. Increased maintenance schedules are absolutely required with abrasive swarf. The coolant must be changed more often and the tank thoroughly cleaned of sediment on the bottom. An auxiliary coolant filter is recommended.

When machining castings, sand from the casting process and the abrasive properties of cast aluminum and cast iron will shorten pump life unless a special filter is used in addition to the 20 mesh intake filter. Contact Haas for recommendations.



LUBRICATION CHART

System	Lubricant	Quantity
Vertical Mills		
Way lube and pneumatics	Mobil Vactra #2*	2-2.5 qts
Transmission	Mobil DTE 25	40Taper 34 oz, 50 Taper 51 oz
A and B axis (VR-Series)	Mobil SHC 625	A-axis 5qts, B-axis 4qts
EC-Series		
Way lube and pneumatics	Mobile Vactra #2*	2-2.5 qts
Transmission	Mobil DTE 25	34 oz
Rotary Table	Mobil SHC-625	Cover sight glass
HS 3/4/6/7 incl R		
Way lube and pneumatics	Mobil DTE-25	2-2.5 qts
Transmission	Mobil DTE 25	34 oz
Rotary Table	Mobil SHC-630	Cover sight glass

*All 50-Taper Machines, 30,000 RPM 40-taper spindles and 15,000 RPM spindles in GR-Series mills use DTE 25.

COOLANT AND COOLANT TANK

Machine coolant must be water-soluble, synthetic oil based or synthetic based coolant/lubricant. **Using mineral cutting oils will damage rubber components throughout the machine and void the warranty.**

Do not use pure water as a coolant; machine components will rust. Do not use flammable liquids as coolant.

If the mill is equipped with Through-the-Spindle Coolant (TSC) do not use coolants with extremely low lubricity; these types of coolant can damage the TSC Coolant tip and pump.

The coolant tank must be thoroughly cleaned periodically, especially for mills equipped with TSC.

Coolant Overview

As the machine runs the water will evaporate which will change the concentration of the coolant. Coolant is also carried out with the parts.

A proper coolant mixture is between 6% and 7%. To top-off coolant only more coolant or deionized water should be used. Be sure that the concentration is still within the range. A refractometer can be used to check the concentration.

Coolant should be replaced at regular intervals. A schedule should be set and held to. This will avoid a build up of machine oil and will ensure that coolant is within the proper concentration and lubricity.

AUXILIARY FILTER ELEMENT REPLACEMENT

Change the filter bag when the filter gauge displays a vacuum level of -5 in. Hg or more. Do not allow the suction to exceed -10 in. Hg or pump damage may occur. Replace with a 25-micron rated filter bag (Haas P/N 93-9130).

Slowly loosen the eye-bolts. This will relieve the pressure at the top of the filter housing and open lid. Use the handle to remove the basket (the filter element will be removed with the basket). Remove the filter element from the basket and discard. Clean the basket. Install a new filter element and replace the basket (with element). Close the lid and secure by tightening the eye-bolts.

1000 PSI TSC MAINTENANCE

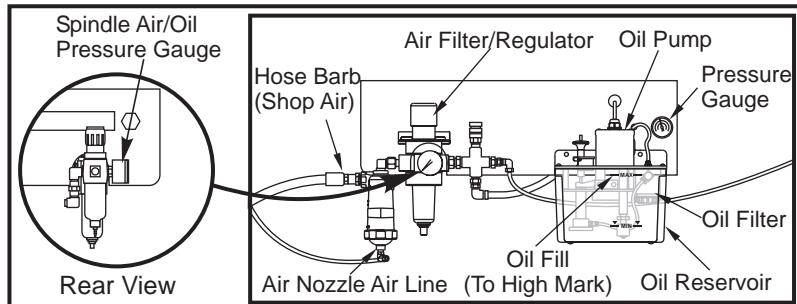
Before doing any maintenance to the 1000psi system, disconnect the power source; unplug it from the power supply.

Check the oil level on a daily basis. If the oil is low, add oil through the fill cap on the reservoir. Fill the reservoir about 25% full with 10-30W oil.



AIR/LUBRICATION SYSTEM

All machine lubrication is supplied by the external lubrication system. Current lube level is visible in the reservoir; Add oil as necessary to maintain proper oil level. **Warning!** Do not add lube above the "high" line marked on the reservoir. Do not allow the lube level to go below the "low" line marked on the reservoir as machine damage could result.



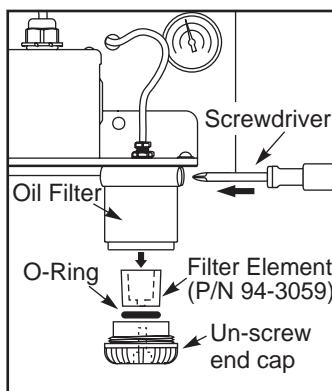
External Lubrication System

Lube Oil Filter

The way lube oil filter element is a 25-micron porous metal filter (94-3059). It is recommended that the filter should be replaced annually or every 2000 hours of machine operation. The filter element is housed in the filter body, which is located **in** the oil pump reservoir (internal filters).

To change the filter element follow these steps:

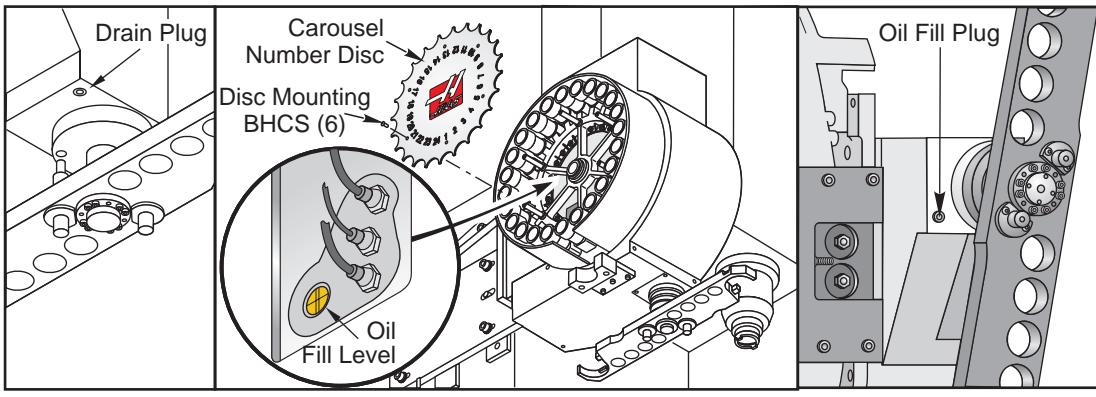
1. Remove the screws that hold the oil reservoir to the pump body, carefully lower the reservoir and set aside.
2. Use a strap wrench, pipe wrench or adjustable pliers to unscrew the end cap (see the figure). **Caution:** Use a screwdriver or similar tool to stop the filter from turning while the end cap is removed.
3. Remove the oil filter element from the filter body once the end cap is removed.
4. Clean the inside of the filter housing and the filter end cap as required.
5. Install the new oil filter element (p/n 94-3059), O-ring and the end cap. Use the same tools that were used to remove the filter end cap, to tighten it - Do Not Over Tighten.
6. Replace the oil reservoir; ensure the gasket seats properly between the reservoir and the top flange.





SIDE MOUNT TOOL CHANGER GEARBOX OIL

Checking the oil level



Vertical Mills

EC-Series Mills

Vertical Mills: View the oil level sight glass in the location shown. The proper level is half way on the sight glass. If more oil is needed, fill the gear box to the proper level at the oil fill/breather where shown.

EC-Series: Remove the plug and feel for oil with your finger. If no oil is felt, add oil until it starts to come out of the hole. Replace plug.

SMTA Oil Types

Mobilgear 632 or equivalent, for standard tool changers

Mobil SHC 630 or equivalent, for high speed tool changers

HS 3/4/6/7 38-Tool Tool Changer Maintenance

Six Months • Lubricate the following parts using red grease:

Magazine Drive Gear,
Tool Pot,
Changer Slide Rack.

• Lubricate the Arm Shaft using Moly grease.

Annually • Lubricate the Changer Slide Linear Guide with red grease.

Tool Pot Chain Tension

The tool pot chain tension should be checked regularly as a preventive maintenance procedure. Chain tension adjustment is performed in the lower left area of the magazine. Loosen the four 12x50 SHCS from the front of the magazine. This will allow the plate to move. Loosen the hex lock nut on the shaft and tighten the shaft using the hex bolt. Lock in the adjustment with the hex lock nut, and retighten the four 12x50 SHCS. Tensioning will not change the indexed pot location, but check the alignment between the manual tool push cylinder and the tool pot.

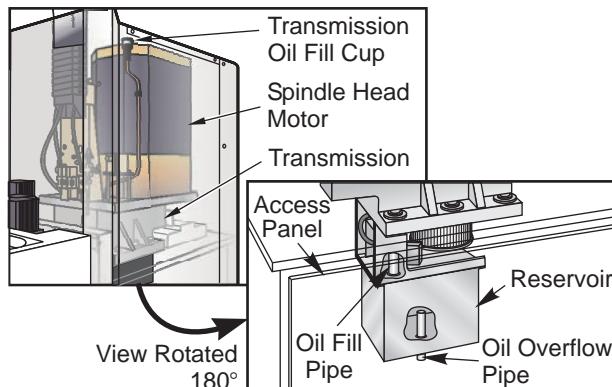


TRANSMISSION

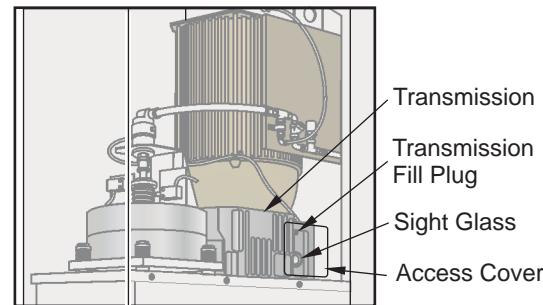
Vertical Mill 40-Taper Transmission Oil Replacement

There is no visible indicator for the level of transmission oil in the VF 1-6/40T models.

To add transmission oil, remove the access panel located directly behind the spindle head. This will expose the Transmission Oil Overflow Pipe. Place a container on the table, beneath this outlet. Manually jog the Z axis to its full -Z travel. Power down the machine. Locate the transmission oil fill cup, accessed from the top of the motor housing. There is a cut-out provided in the top of the motor housing sheetmetal for filling. Slowly pour in Mobil DTE 25 oil until oil starts to come out of the overflow pipe; this overflow indicates the reservoir is full. Close the transmission oil fill cup, wipe off the overflow pipe and replace the access cover. Consider any overflow oil to be used and dispose of properly.



VF 1-6 40 Taper



VF 6 through 11 50T

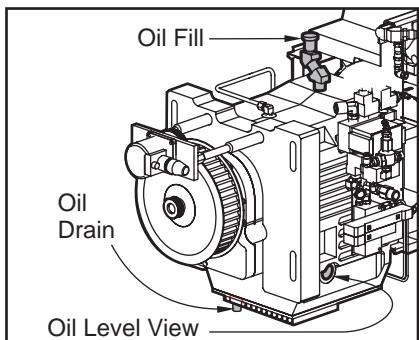
NOTE: The VF-5 50 taper, does not have a sight glass; the oil is circulated directly onto the gears by a hose.

Oil Change

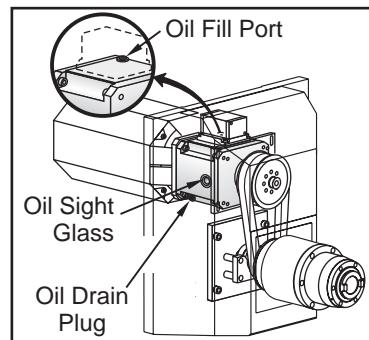
1. Remove the sheet metal from the spindle head.
2. Remove the encoder and the encoder mounting plate.
3. Remove the oil drain plug. When the oil drains, check the magnet for metal particles.
4. Replace the oil drain plug and fill the gear case with 1½ liters of Mobil DTE 25 gear oil at the fill cup on top.
5. Replace the oil overflow plug, put a small amount of thread sealant on the threads. (Do not use thread locking compound) install encoder and insure that the spindle orientation is correct.
6. Install the sheet metal and run a spindle warm up and check for leaks.



EC-1600 AND HS 3/4/6/7 TRANSMISSION OIL



EC-1600



HS-3/4/6/7

Oil Check

Remove the sheet metal necessary to gain access to the transmission. View the sight glass on the side of the transmission box as shown. The oil level should be half way on the sight glass. Fill as needed.

Oil Change

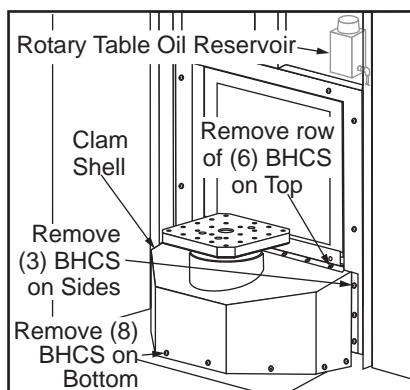
1. Remove the sheet metal from the spindle head.
2. Remove the drain plug as shown. Inspect the magnetic drainplug for signs of metal particles.
3. Blow downward with an air hose in the vicinity of the fill hole to prevent dirt and metal particles from entering the gear case. Remove the fill plug.
4. Add Mobil DTE-25 gear oil until the oil level is half way up the sightglass.
5. Run a spindle warm-up and check for leaks.

EC-SERIES PALLET CHANGER ROTARY TABLE

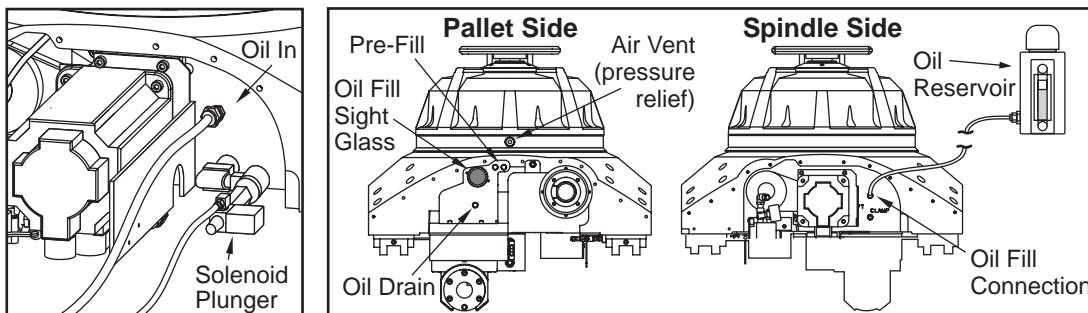
Oil Replacement

EC-300

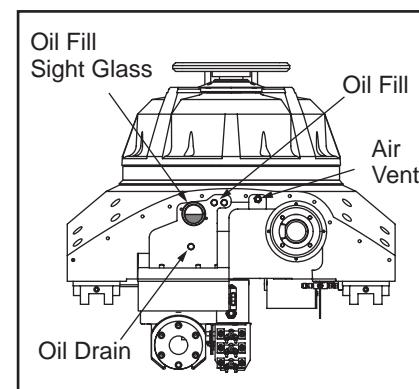
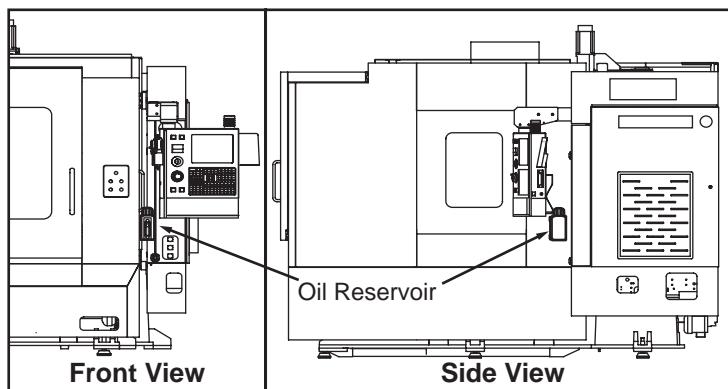
Periodically check the oil level in the reservoir and keep it filled. It is not necessary to replace the oil.



EC-400 Full Fourth Axis Rotary Table (Perform Every 2 years)



1. Remove the fourteen (14) BHCS on the right Z-axis way cover at the receiver end and slide it toward the column.
2. Remove the left Z-axis way cover: Jog the Z-axis all the way toward the column and rotate the H-frame 45° counter clockwise. Remove the thirteen (13) BHCS that fasten the waycover to the receiver and remove it through the door at the control pendant.
3. Disconnect the reservoir at the rotary indexer end and plug the end of the hose.
4. Remove the drain plug on the opposite side of the rotary indexer. Replug the hole when the oil has drained.
5. Remove the air escape hole plug on the side of the platter.
6. Fill the rotary table until oil begins to escape from the air escape hole and plug it.
7. Replace the reservoir hose and the way covers. Command the receiver 180° to 0° repeatedly for fifteen minutes. The reservoir will drop in level as it continues to replace the oil. Add oil as needed to the reservoir to just below the full line.



EC-400 Rotary Indexer

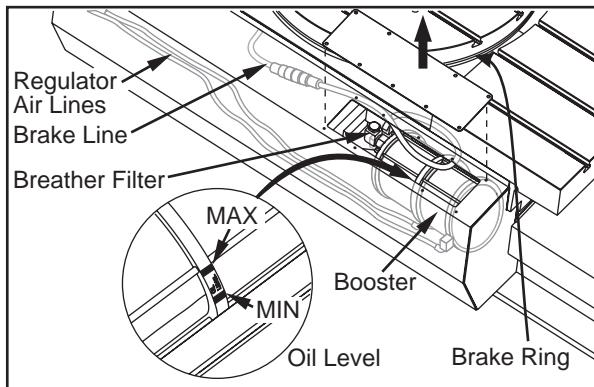
Rotary Indexer (1° or 45°)

1. Remove the drain plug located on the left side of the rotary indexer. Replace the plug when the oil has drained.
2. Remove the air vent plug located to the upper right of the drain hole.
3. Fill the rotary indexer at the oil fill hole shown in the illustration. Replace the plug when oil begins to seep from the air vent.
4. Command a 180° to 0° rotation for fifteen minutes. This will remove air remaining in the system. The rotary indexer is full when the oil level is half way on the site glass. Fill as needed.
5. Replace the waycover.



HYDRAULIC BRAKE (EC-1600-3000, HS3-7R)

Check the brake fluid level by viewing the fluid level in the booster. To check the EC 1600-3000 remove the brake booster cover. The cover/booster is located at the right, front of the machine. The HS 3-7R brake booster is located on the operator pendant side of the machine. Remove the way cover from the table and slide the way cover away from the table.



Adding Oil

Use only Mobil DTE 25. Remove the breather filter from the brake booster assembly and add oil. The proper oil level is between the minimum and maximum marks on the brake booster.

VR-SERIES

The following items must be performed in addition to the regular maintenance items.

Interval

Monthly

Maintenance Performed

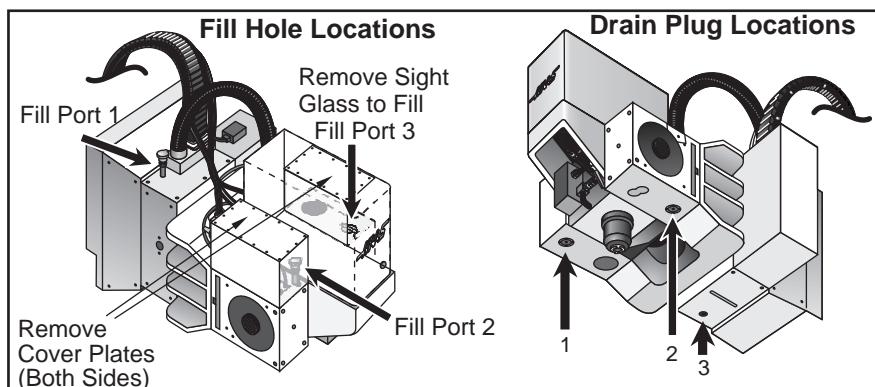
- Grease all pivot points on the tool changer assembly.
- Check the oil in the three (3) areas of the head. The A-axis covers need to be removed to access the filler cap and the sight glass. The B-axis filler is on the outside of the casting. Add Mobil SHC-630 to the fill port at the top of the casting.

Annually

- Replace the oil in the three (3) areas of the head:

For the areas on either side of the spindle head (A-axis), remove the drain plug (4 BHCS) and drain the oil. **Note:** Remove the plug closest to the front on the left side of the head, and the plug towards the rear of the right side of the head. Fill the two areas with Mobil SHC-630 as described in the "Monthly" section above.

B-Axis For the area at the rear of the spindle head, remove the 1/4" NPT pipeplug with an Allen wrench and drain the oil. **Note:** The plug is near the center of this rear area. Fill with Mobil SHC-630 as described in the "Monthly" section above.

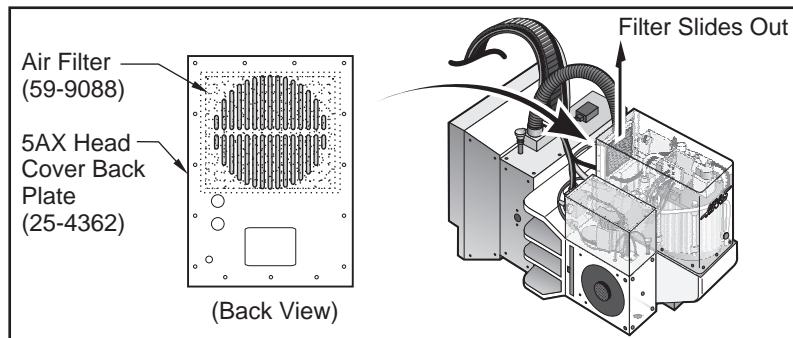




VR-SERIES AIR FILTER

The VR mills are equipped with an air filter (P/N 59-9088) for the motor housing. The recommended replacement interval is monthly, or sooner depending on your machining environment.

The air filter is located on the rear of the head cover. To remove the air filter, simply pull up on the filter; the filter will slide upward out of its bracket. To replace the filter, slide in the new air filter, oriented properly to filter air into the motor housing. Filter air-flow direction is determined by a sticker on the replacement filter.

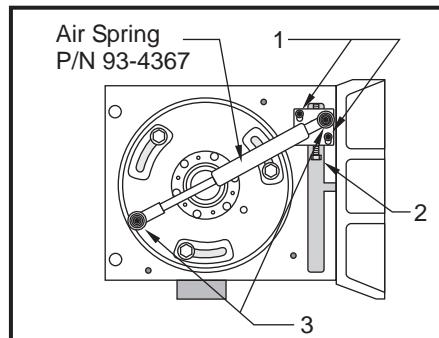


VR-II Air Filter Location

VR-SERIES AIR SPRING COUNTERBALANCE REPLACEMENT

The counterbalance air spring and rod ends should be replaced every two (2) years.

1. Verify that the Axis is at 0 degrees before beginning. Press E-stop before doing any disassembly.
2. Remove sheet metal cover and loosen the two 3/8-16 SHCS (1).
3. Back out the 1/4-20 SHCS (2), and tighten the two 3/8-16 SHCS (1) this will keep the pre-load cam secure while the next step is accomplished.
4. Remove 3/8-16 SHCS that mount Air Spring and Rod Ends (3).
5. Tighten the rod ends onto the Air Spring and secure the Air Spring using the two 3/8-16 SHCS removed in step 4.



6. Slightly loosen the 3/8-16 SHCS (1). Screw in the 1/4-20 SHCS to force pre-load cam counterbalance down (this will push the air spring inward). Tighten this adjusting bolt until the slots in the cam have contacted the tops of the mounting bolts. Tightening the two 3/8-16 SHCS (1), these will hold the pre-load cam in place.

7. Replace the sheet metal, reset the E-stop and clear the alarms.