



Lathe Operator's Manual

J U N E 2 0 0 5

HAAS AUTOMATION INC. • 2800 STURGIS ROAD • OXNARD, CA 93030
TEL. 888-817-4227 FAX. 805-278-8561
www.HaasCNC.com



Haas Technical Publications

HAAS SERVICE AND OPERATOR MANUAL ARCHIVE

Lathe Operator Manual 96-8700 RevL English June 2005

- 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 turning 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 is 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 tool turret 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:

- ◆ Do not operate with the door open.
- ◆ Do not operate without proper training.
- ◆ Always wear safety goggles.
- ◆ The machine is automatically controlled and may start at any time.
- ◆ Improperly or inadequately clamped parts may be ejected with deadly force.
- ◆ Do not exceed rated chuck rpm.
- ◆ Higher rpm reduces chuck clamping force.
- ◆ Unsupported bar stock must not extend past draw tube end.
- ◆ Chucks must be greased weekly and regularly serviced.
- ◆ Chuck jaws must not protrude beyond the diameter of the chuck.
- ◆ Do not machine parts larger than the chuck.
- ◆ Follow all of the warnings of the chuck manufacturer regarding the chuck and work holding procedures.
- ◆ Hydraulic pressure must be set correctly to securely hold the work piece without distortion.
- ◆ 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 at high velocity may puncture the safety door. Reduced rpm is required to protect the operator when performing dangerous operations (e.g. turning oversized or marginally clamped parts). Turning 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 turning machines contain hazards from rotating cutting tools, belts and pulleys, high voltage electricity, noise, and compressed air. When using turning 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



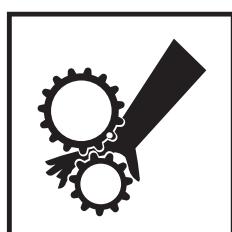
Automatic Equipment

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.



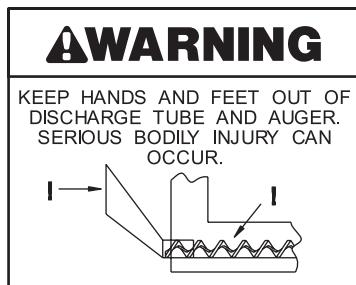
Machine Guarding

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.



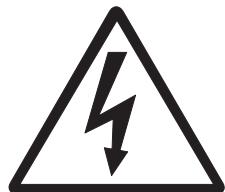
Rotating Parts

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.



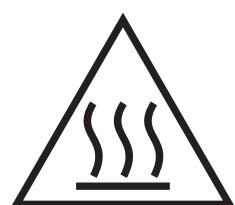
Dangerous Voltages

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.



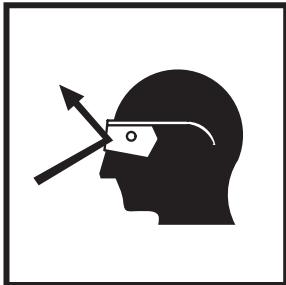
Proper Attire

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.



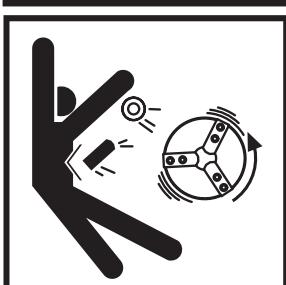
Electrical Service

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.



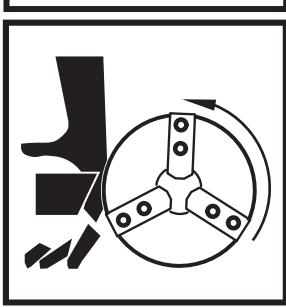
Spills

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/format; 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:

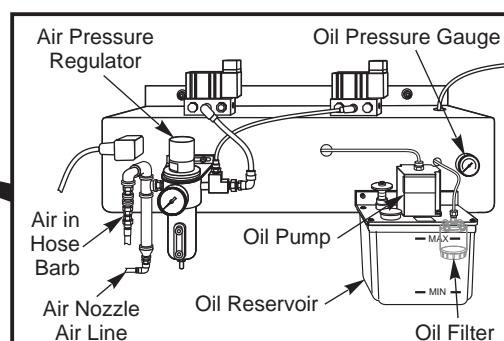
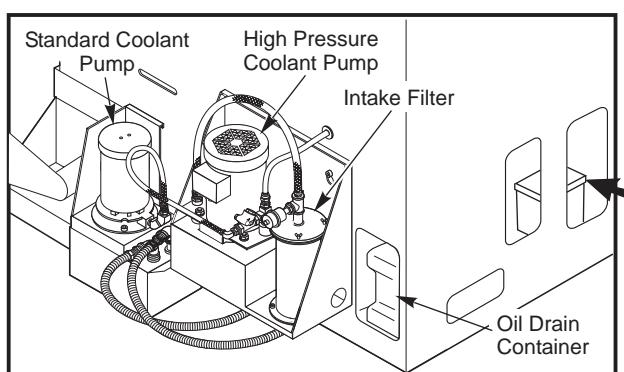
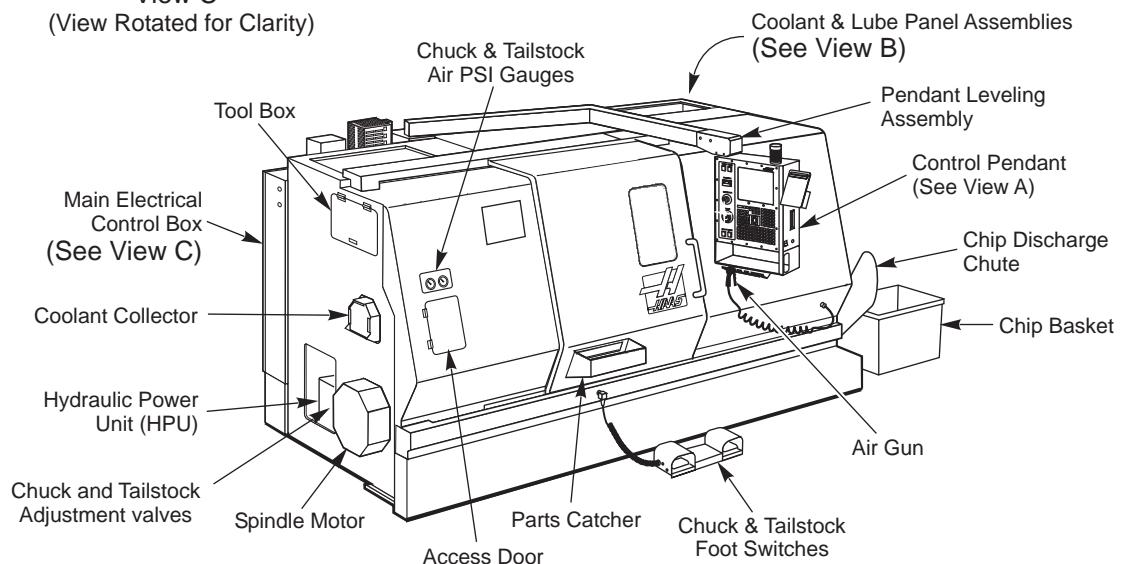
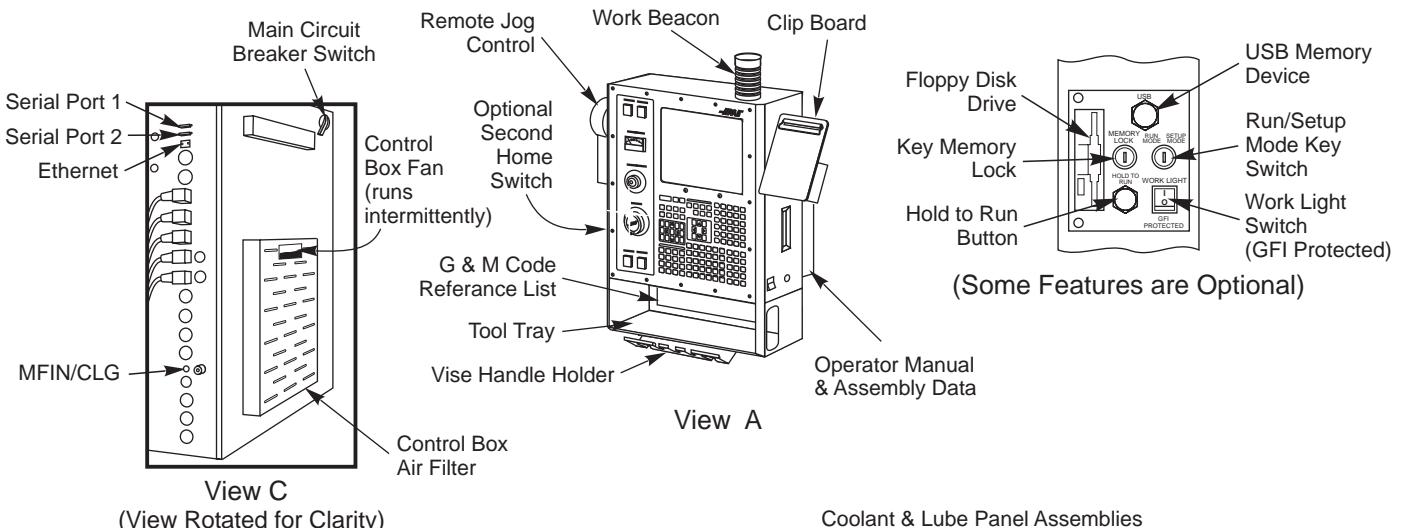
- Avoid extreme loads on the center of table or far end of table. The weight of the part should be distributed evenly over the table or one pad.
- Flatness of part should be within 0.002".





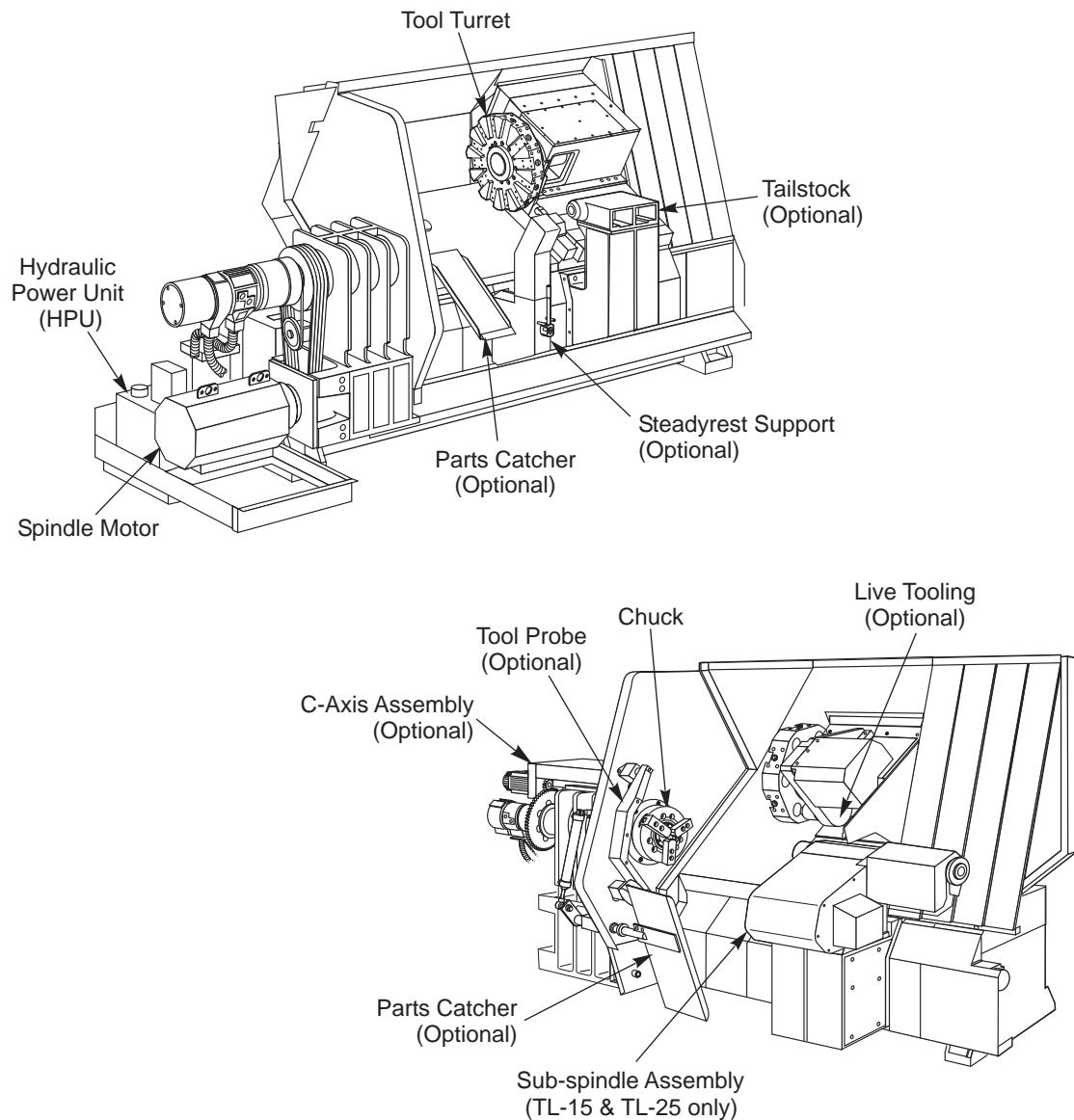
INTRODUCTION

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



View B
(View Rotated 180°)

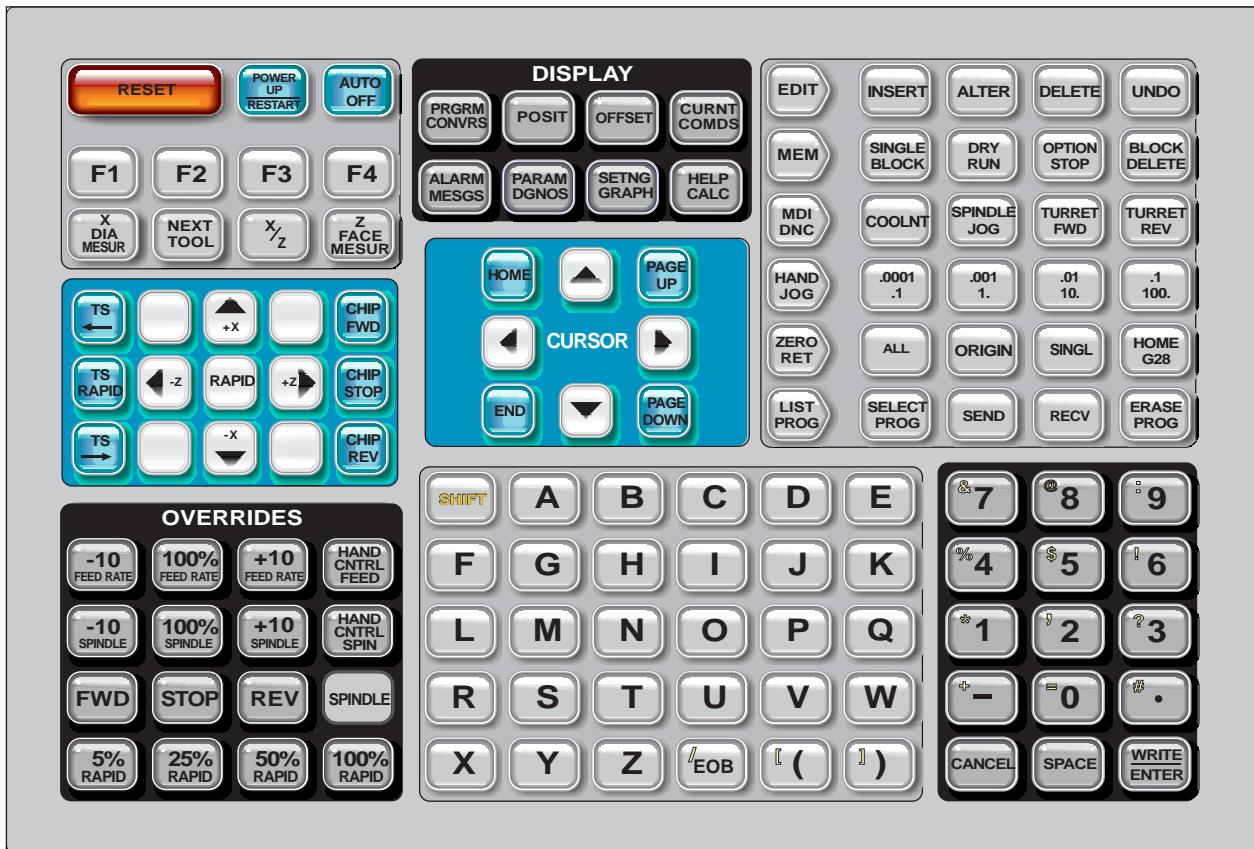
Lube Panel Assembly
(Left Rear Enclosure Panel Removed)





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, turret, 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 turret 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.

Auto Off - Automatically positions axes to machine zero and prepares the machine for power down.



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. 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.

X Dia Mesur (X Diameter Measure) – Used to record X axis tool shift offsets on the offset page during part setup.

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

X/Z – Used to toggle between X axis and Z axis jog modes during part setup.

Z Face Mesur (Z Face Measure) – Used to record Z axis tool shift offsets on the offset page during part setup.

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 and Z-Z (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.

Rapid - When pressed simultaneously with one of the above keys (X+, X-, Z+, Z-), that axis will move in the selected direction at maximum jog speed.

<- TS – Pressing this key moves the tailstock towards the spindle.

TS Rapid – Increases the speed of the tailstock when pressed simultaneously with one of the other tailstock keys.

-> TS – Pressing this key moves the tailstock away from the spindle.



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 control overidden 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.

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

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

The spindle can be started or stopped with the FWD or REV 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 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 tapping cycles. In the Single Block mode, the spindle may be stopped. It will automatically start up upon continuing the program with the Cycle Start button.

By pressing the Handle Control Feedrate key, the jog handle can be used to control feedrate from in $\pm 1\%$ increments.

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. If you press the button twice you will enter the Quick Code (See Quick Code section) feature and if you push it three times you 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 machines 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 and moves the zoom window left when in graphics mode.

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 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.

Edit - Selects edit mode. This page is used to edit programs in controls memory.

Insert - Pressing this button will enter commands into the program at 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 delete a selected program block.

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

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

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. The optional HPC (High Pressure Coolant) is activated by pressing the Shift button followed by the COOLNT button. Note that as HPC and regular coolant share a common orifice, they cannot both be on at the same time.

Spindle Jog - Rotates the spindle at the speed selected in Setting 98 (Spindle Jog RPM).

Turret FWD - Rotates the tool turret forward to the next sequential tool. If Tnn is entered on the input line, the turret will advance in the forward direction to tool nn.



Turret REV - Rotates the tool turret backward to the previous tool. If Tnn is entered on the input line, the turret will advance in the reverse direction to tool nn.

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 lathe 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.

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. Each axis can be zeroed by selecting an axis (X or Z) and pressing the Origin key.

Work Display

This displays the position of X 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 three offsets pages: Tool Geometry, Tool and Work Zero Offset.

Tool Geometry/Wear

The Tool/Geometry page is displayed by pressing the Offset button. This page displays tool numbers and tool length geometry. Pressing the Offset button again will display the tool wear screen.

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 lathe 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. Use the arrow keys to scroll to each column or 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 lathe 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 cutting position (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.

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)

It is not recommended to use tool load monitoring while in G96 Constant Surface Speed Mode. It is not possible for the system to distinguish the load due to spindle acceleration from the load on the tool. Tool overload conditions may be generated during X-axis feeds due to spindle acceleration while in G96 constant surface speed mode.

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 lathe 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 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 ("un-zooms") 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.

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 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 calculations.

F4 In the Calculator function, this button uses the highlighted 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.

The screenshot shows a software interface for solving triangles. At the top, it says "HELP (MEM) O00000 N00000000". Below that is a "CALCULATOR" section with a highlighted value "0.00000000". A "LOAD" button is followed by a row of operators: + - × / . To the left, there's a "(MACHINE)" coordinate section with X, Y, and Z values: X=0.0000 in, Y=0.0000 in, Z=3.5179 in. To the right, there's a triangle diagram with vertices labeled "ANGLE 1", "ANGLE 2", and "ANGLE 3" at the top, and "SIDE 1", "SIDE 2", and "SIDE 3" along the bottom. Data for the triangle is listed: ANGLE 1 = 40.000, ANGLE 2 = 72.000, ANGLE 3 = 68.000; SIDE 1 = 10.0000, SIDE 2 = 14.7958, SIDE 3 = 14.4244. A note at the bottom states: "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		
0.00000000		
LOAD	+ - × /	
(MACHINE)		
X	0.0000 in	CENTER X 13.0000
Y	0.0000 in	CENTER Y 20.0000
Z	3.5179 in	START X 4.0000
		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.

CI RCLE-CI RCLE TANGENT
CI RCLE1 X 5.0000
CI RCLE1 Y 6.0000
RADI US 1 4.0000
CI RCLE2 X 0.0000
CI RCLE2 Y 0.0000
RADI US 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

CI RCLE-LI NE 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
RADI US 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.

Drill/Tap Chart

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



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.

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.

Macros

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

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

Auto Jet Blast

The Auto Jet Blast keeps your workpiece clean: With the doors closed, an M-code activated air blast clears chips and coolant from the chuck and workpiece.

Tool Presetter

The manual tool-probe arm swings down for fast tool setting. Touch the tool tip to the probe and offsets are automatically entered.

High Intensity Lighting

Halogen lights provide bright, even illumination of the work area for part inspection, job setup and change-overs — 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.

Steady Rest Provision

The steady rest mounting platform provides increased support for long-shaft or narrow-shaft operations. Industry-standard mounting holes accept most aftermarket steady-rest arms.



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.

Tailstock

The fully programmable hydraulic tailstock can be activated via the part program or controlled directly by the operator with the standard foot switch.

Parts Catcher

The optional part chute rotates into position to catch the finished part and directs it into a bin located on the front door. There's no need to stop the machine and open the door to retrieve parts.

Barfeeder

Designed to boost productivity and streamline turning operations, this servo driven bar feeder is exclusively for Haas CNC lathes. Unique features make setup and operation simple, like a large access door for spindle liner change-out, and a single adjustment for setting bar diameter.

Live Tooling

The live tooling option allows you to drive standard VDI axial or radial driven tools to perform such secondary operations as drilling or tapping, both on the face of the part and around the diameter. The main spindle provides indexing in precise increments for part positioning and repeatability.

C-Axis

The C axis provides high-precision, bi-directional spindle motion that is fully interpolated with X and/or Z motion. Cartesian-to-polar interpolation allows programming of face contouring operations using traditional X and Y coordinates.

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.



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. 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.

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 ; G74 X4. Z-4. I0.2 K0.75 D255 ; G00 X2. Z0.9 ; G74 X1. Z4. I0.2 K0.75 D255 ; G00 X3. Z0.9 ; G74 X4. Z4. I0.2 K0.75 D255 ; ; G00 X0 Z0.5 T101 ; G83 Z-1.5 Q0.25 R0.03 ; G51 ;			

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 delete characters , 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 **Oⁿnnnn**; 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 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 Program Numbers

If the maximum number of programs (500) is in the controls 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 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

Files loaded on a CNC control can also be viewed on a personal computer (PC) by adding (rename) ".txt" to the file name from the CNC control.



The format of data sent and received for macro variables is the same, except there is a line N9999 and no line N0. The **N** number is the data number and **V** is the value. N0 is a code computed by the control prior to sending the data. The N0 value is mandatory with parameters but is optional with settings and offsets. If changes are made to the Settings and Offsets, delete the N0 line before sending the information back to the CNC control.

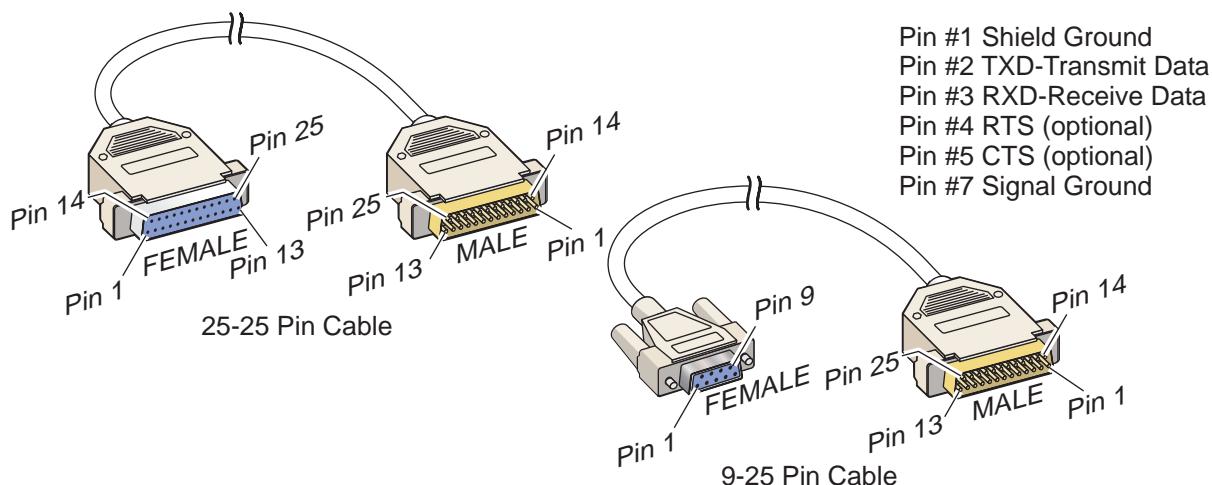
Corrupted data, if received, is usually converted into a comment, stored in the program, and an alarm is generated. However, the data will still be loaded in to the control

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, setting 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)
- 12 Parity (Even)
- 13 Stop Bits (1)
- 14 Synchronization Xon/Xoff

- 24 Leader to Punch (None)
- 25 EOB Pattern (CR LF)
- 37 Number Data Bits (7)



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 beginning an ending with 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 erased 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.

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 Baude 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, W, X, Z Axis Motion – Specifies axis motion (distance or angle)

D Depth of cut – Selects the depth of cut for each pass of a stock-removal cycle.

E Feed rate, 6 place precision (same as F) – Selects feed rate applied to any interpolating G codes or canned cycles.

F Feed rate – Selects feed rate applied to any interpolating G codes or canned cycles.

G, M Preparatory Functions – See the chapters on G or M codes

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 as a positive number in inches/mm between 0 and 100.0, or to identify the final block of a stock-removal path.

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 metric.

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

T Tool selection code – Used to select the tool number and to specify the tool shift, tool geometry or tool wear values.

PART SETUP

It is necessary to properly secure the part in the chuck. See the chuck or collet manufacture's manual for the proper procedure for fixturing a workpiece.

TOOLING

The **Tnn** code is used to select the tool to be used in a program.

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.

To enter jog mode press the handle jog button, then press one of the desired axes (e.g. X, Z, 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, U, Z, W, B, C or V) using the dial, choose the increment value (X1, X10 or X100) and use the jog handle to move the axes.

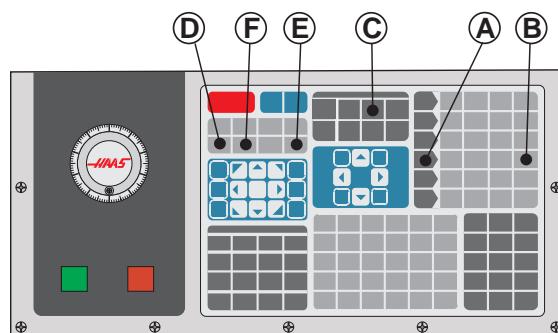
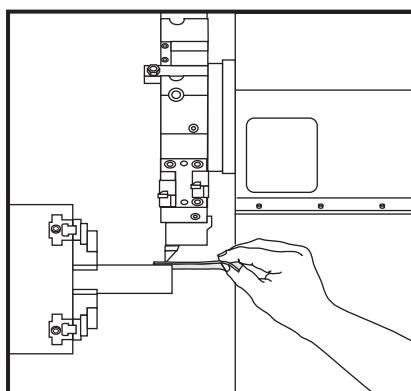
Setting the Tool Offset

The next step is to touch off the tools. Doing this defines the distance from the tip of the tool to the side of the part. Enter the Tool Geometry offset page, this should be the first page in the offsets screens if not use the page up button until the Tool Geometry page is selected, and press X DIA. MEAS., the control will send a prompt to enter the diameter of the part. The control will add the diameter of the part to the location of the X axis. Next, Press the Z FACE MEAS. button, the control will send a prompt to enter the diameter of the part. The control will add the diameter of the part to the location of the Z axis.



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.

1. Load a tool into the tool turret.
2. Press the Handle Jog button **(A)**
3. Press .1/100. **(B)** (The lathe will move at a fast rate when the handle is turned).
4. Toggle between the X and Z jog buttons until the tool is touching the side of the part at about a 1/8 of an inch from the front edge.
5. Place a sheet of paper between the tool and the part. Carefully move the tool as close as possible, and still be able to move the paper.



6. Press Offset **(C)**, the Tool Geometry page should be displayed. If not page up until the Tool geometry page is displayed.
7. Press X DIAM. Mesur **(D)** the operator will then be prompted to add the diameter of the part. This will take the X position located in the bottom left of the screen and the diameter of the part and put it the tool number position.
8. The cursor will move to the Z axis location for the tool.
9. Press Z FACE MEAS. **(E)** the operator will then be prompted to add the diameter for the part. This will take the Z position located in the bottom left of the screen and the diameter of the part and put it the tool number position.
10. The cursor will move to the Z axis location for the tool.
11. Press Next Tool **(F)**

Additional Tooling Setup

There are other tool setup 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" at the top of the page. The programmer can add a tool load limit. The control will reference these values and they can be set to do a specific action should the limitation 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 Turret Fwd.



2. Jog X and Z until the tool is nearly touching the part approximately an 1/8 of an inch from the front of the part.
3. Place a sheet of paper between the tool and the part. Carefully move the tool down as close as possible and still be able to move the paper. Do not jog any further or damage to the tool will result.
4. Set the G54 work offsets by highlighting G54 X-axis on the Work Offsets screen. Enter the x position, located on the bottom left of the screen, plus the diameter of the part into the X column using the F1 button.
5. Cursor over to the Z-axis. Enter the z position, located on the bottom left of the screen, plus the diameter of the part into the Z column using the F1 button.
6. Repeat steps 1-5 for all the tools used in the program.

Graphics Mode

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.



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.

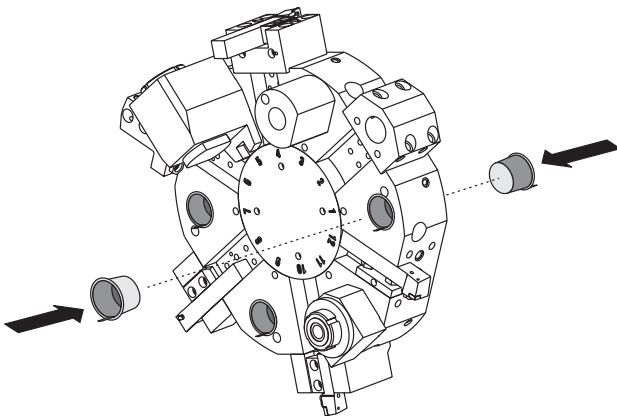
The subroutines can include an L or repeat count. If there is an L, the subroutine call is repeated that number of times before the main program continues with the next block.

TOOL CHANGER OPERATIONS

Low air pressure or insufficient volume will reduce the pressure applied to the turret clamp/unclamp piston and will slow down the turret index time or will not unclamp the turret.

To load or change tools, select MDI mode, and then press TURRET FWD or TURRET REV and the machine will index the turret on tool position. If you enter Tnn prior to pressing TURRET FWD or TURRET REV the turret will bring the entered tool around to the cutting position.

IMPORTANT Insert protective caps into any empty turret pockets to protect the pocket from accumulating debris.



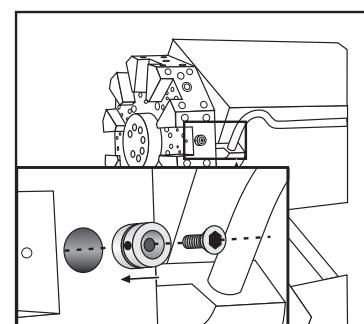
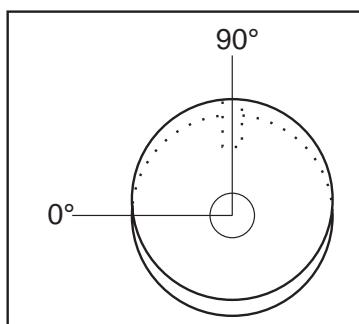
Eccentric Cam Locating Button

Bolt on turrets are equipped with eccentric locating buttons that allow for fine alignment of ID tool holders to spindle center line.

Mount the tool holder to the turret and align the tool holder to the spindle in the X-axis. Measure the alignment in the Y-axis. If necessary remove the tool holder and use a narrow tool in the cam button hole, to rotate the eccentric to correct misalignment.

The following table gives the result for specific positions of the cam button.

Rotation	Result
0°	No change
15°	.0018"
30°	.0035"
45°	.0050"
60°	.0060"
75°	.0067"
90°	.0070"





TOOL FUNCTIONS

The Tnnoo code is used to select the next tool (nn) and offset (oo). The use of this code differs slightly depending on Setting 33 FANUC or YASNAC coordinate system.

FANUC Coordinate System

T-codes have the format Txxyy where xx specifies the tool number from 1 to the value in Parameter 65; and yy specifies the tool geometry and tool wear indices from 1 to 50. The tool geometry X and Z values are added to the work offsets. If tool nose compensation is used, yy specifies the tool geometry index for radius, taper, and tip. If yy=00 no tool geometry or wear is applied.

YASNAC Coordinate System

T-codes have the format Tnnoo, nn has different meanings depending on whether the T-code is inside or outside a G50 block. The oo value specifies the tool wear from 1 to 50. If tool nose compensation is used, 50+oo specifies the tool shift index for radius, taper, and tip. If oo+00, no tool wear or tool nose compensations are applied.

Outside a G50 block, nn specifies the tool number from 1 to the value in Parameter 65.

Inside a G50 block, nn specifies the tool shift index from 51 to 100. The tool shift X and Z values are subtracted from the work offsets (and thus are of opposite sign than the tool geometries used in the FANUC coordinate system).

Tool Offsets Applied by T0101, FANUC vs YASNAC

Setting a negative tool wear in the tool wear offsets will move the tool further in the negative direction of the axis. Thus, for O.D. turning and facing, setting a negative offset in the X axis will result in a smaller diameter part and setting a negative value in the Z axis will result in more material being taken off the face.

NOTE: There is no X or Z motion required prior to performing a tool change and it would waste time in most cases to return X or Z to the home position. However, if your work piece or fixture is quite large, you may need to position X or Z prior to a tool change in order to prevent a crash between the tools and your fixture or part.

Low air pressure or insufficient volume will reduce the pressure applied to the turret clamp/unclamp piston and will slow down the turret index time or will not unclamp the turret.

After Power Up/Restart and Zero RET, the control will insure that the tool turret is in a normal position. To load or change tools, select MDI mode, and then press Turret FWD or Turret REV and the machine will index the turret on tool position. The CURNT COMDS display will show what tool is currently in position.

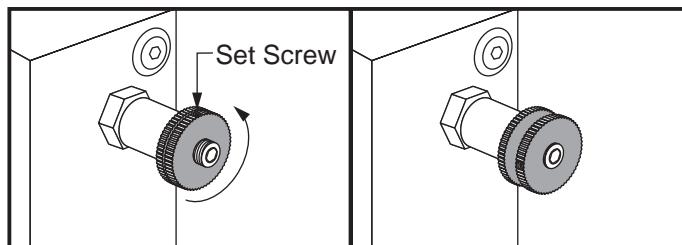


DRAWTUBE OPERATION

The hydraulic unit provides the pressure necessary to clamp a part.

Clamping Force Adjustment Procedure

1. Go to Setting 92 on the Settings page and choose either 'I.D.' or 'O.D. Clamping'. Do not do this with a program running.
2. Loosen the locking knob at the base of the adjustment knob
3. Turn the adjustment knob until the gauge reads the desired pressure
4. Tighten the locking knob



Drawtube Warnings

Never attach dead length stops to the hydraulic cylinder, damage will result.

DO NOT machine parts larger than the chuck.

Follow all of the warnings of the chuck manufacturer.

Hydraulic pressure must be set correctly to securely hold the workpiece without distortion.

Chuck jaws must not protrude beyond the diameter of the chuck.

Improperly or inadequately clamped parts will be ejected with deadly force.

Do not exceed rated chuck RPM

Higher RPM reduces chuck clamping force

NOTE: Chucks must be greased weekly and kept free from debris.

CHUCK AND COLLET REPLACEMENT

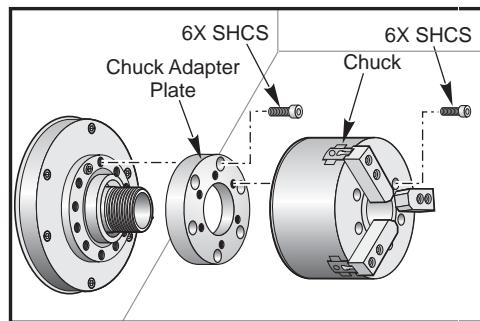
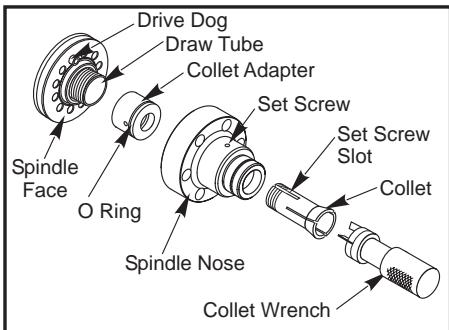
Removal

Chuck

1. Move both axes to their zero positions. Remove the chuck jaws.
2. Remove the three (3) screws that mount the center cup (or plate) from the center of the chuck and remove the cup.
3. Clamp the chuck and remove the six (6) SHCS that mount the chuck to the collet.
4. Place a chuck wrench inside the center bore of the chuck and unscrew the chuck from the drawtube. If equipped, removed adaptor plate.

Collet

1. Loosen the set screw on the side of the spindle nose. Using the collet wrench, unscrew the collet from the spindle nose.
2. Remove the six (6) SHCS from the spindle nose and remove it.
3. Remove the collet adapter from the drawtube.



Installation

Chuck

Note: If necessary, install an adapter plate before installing the chuck

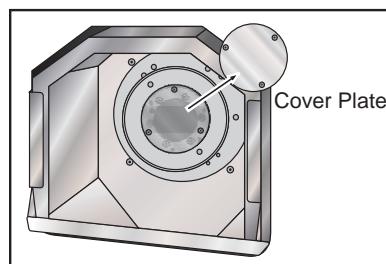
1. Clean the face of the spindle and the back face of the chuck.
2. Position the drive dog at the top of the spindle.
3. Remove the jaws from the chuck.
4. Remove the center cup or coverplate from the front of the chuck.
5. If available, install a mounting guide into the drawtube and slide the chuck over it.
6. Orient the chuck so that one of the guide holes are aligned with the drive dog.
7. Use the chuck wrench to thread the chuck onto the drawtube.
8. Screw the chuck all the way onto the drawtube and back it off 1/4 turn.
9. Align the drive dog with one of the holes in the chuck.
10. Tighten the six (6) SHCS.
11. Install the center cup or plate with three (3) SHCS .
12. Install the jaws. If necessary replace the rear cover plate. This is located on the left side of the machine.

Collet

1. Thread the collet adapter into the drawtube.
2. Place the spindle nose on the spindle and align one of the holes on the back of the spindle nose with the drive dog.
3. Fasten the spindle nose to the spindle with six (6) SHCS.
4. Thread the collet onto the spindle nose and align the slot on the collet with the set screw on the spindle nose. Tighten the setscrew on the side of the spindle nose.

DRAWTUBE COVER PLATE

It is necessary to remove the cover plate at the far end of the drawbar when using a barfeeder. Replace the cover plate anytime bar stock is not being fed automatically.

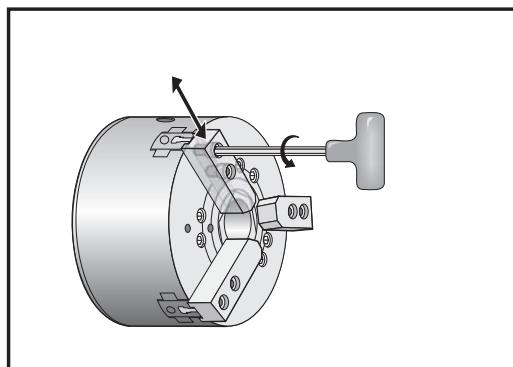




RE-POSITIONING CHUCK JAWS

Re-position chuck jaws when the jaw stroke travel cannot generate sufficient clamp force to hold the material, e.g., when changing to a smaller diameter stock.

The part will not be sufficiently clamped if there is not extra stroke before bottoming out the jaws.



1. Use a hex key to loosen the two SHCS attaching the jaw to the chuck.
2. Slide jaw to new position and re-tighten the two SHCS.
3. Repeat procedure for remaining two jaws. Jaws must remain concentric.

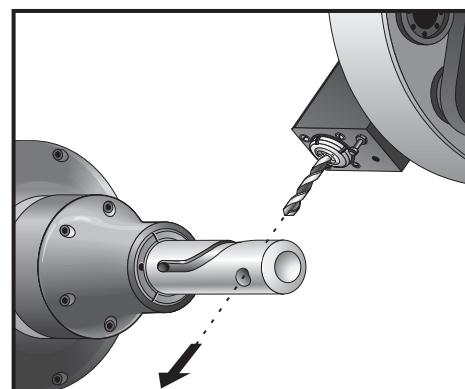
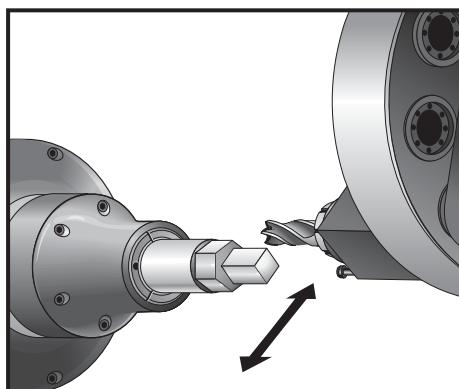
TAPER COMPENSATION

Deflection of the part occurs if it is not supported precisely in the center, or if it is too long and unsupported. This causes the cut to be too shallow so the resultant part is under-cut. This can apply to O.D and I.D cutting. Taper Compensation provides the ability to compensate by adding in a calculated value to the X movement based on the position of the Z cut. The zero point of the taper is defined to be the 0.0 of the work-zero coordinate of Z.

The taper is entered on the tool shift page as a 5 place number and stored in an array indexed by tool, which is called "Taper" on the Tool Shift / Geometry page. The value entered should be the deflection in the X-axis divided by the length in the Z-axis, over which the deflection occurs.

LIVE TOOLING

This option is not field installable





INTRODUCTION

The live tooling option allows the user to drive VDI axial or radial tools to perform such operations as milling, drilling or slotting. The main spindle of the lathe is indexable in one degree increments for precise part positioning and repeatability. Milling shapes is possible using spindle motion G codes.

Programming Notes

- The live tool drive will automatically turn itself off when a tool change is commanded.
- The main spindle can be clamped (M14 and M15) for using the live tooling. It will automatically unclamp when a new main spindle speed is commanded or Reset is pressed.
- Maximum live tooling drive speed is 3000 rpm.
- Canned cycles are not supported.
- Haas live tooling is designed for medium duty milling, e.g.: 3/4" diameter end mill in mild steel max.
- Large Tool diameters may require reduction tool holders

LIVE TOOLING M CODES

Also see the M-Code Chapter

M19 Angle CMD (Optional)

An M19 will orient the spindle to the zero position. A P value can be added that will cause the spindle to orient to a specific position (in degrees.) Degrees of accuracy - P rounds to the nearest whole degree, and R round to the nearest hundredth of a degree (x.xx) The angle is viewed in the Current Commands Tool Load screen.

M133 Live Tool Drive Forward

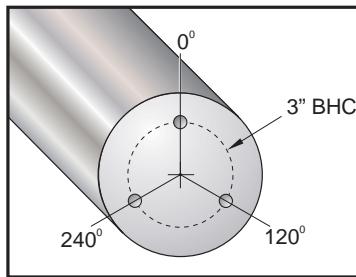
M134 Live Tool Drive Reverse

M135 Live Tool Drive Stop

Program Example:

Bolt Hole Circle 3 holes @ 120° on 3" BHC

```
G00 X3.0 Z0.1
G98
M19 P0
G04 P2. (Dwell for motor stabilization; length of time depends on set-up)
M14
M133 P2000
G01 Z-0.5 F40.0
G00 Z0.1
M19 P120
G04 P2. (Dwell for motor stabilization; length of time depends on set-up)
M14
G01 Z-0.5
G00 Z0.1
M19 P240
G04 P2. (Dwell for motor stabilization; length of time depends on set-up)
M14
G01 Z-0.5
G00 Z0.1
M15
M135
```



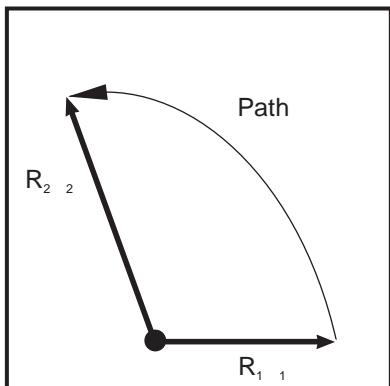
SYNCHRONOUS MILLING

G32 synchronous motion is a control mode where the X, Z axes are commanded to move distances at constant feed rates and spindle is commanded to rotate at constant speeds.

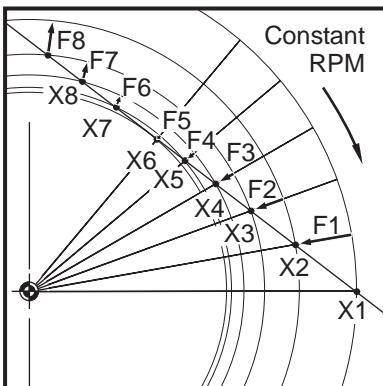
G32 is commonly used to create threads, the spindle rotates at a constant rpm and constant Z-axis motion begins at the same reference Z-axis mark for each stroke. Many strokes can be repeated because the reference mark sets the location of the start thread.

Geometric shapes can be machined using G32, however, G32 motions can be cumbersome to create and difficult to adjust in the final program. To relieve the user of this burden, the Haas CNC control has a canned cycle G code which simplifies the creation of simple geometric shapes. G77 Flatting Cycle automates the motions of 1 or more sided uniform shapes.

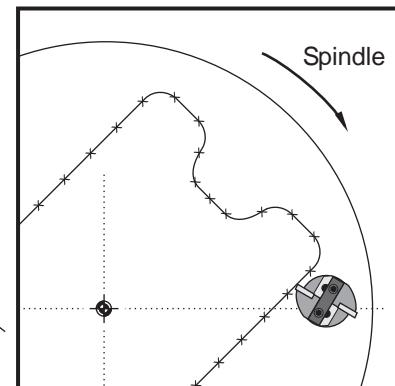
In addition to synchronous motions G5 is a motion mode that accepts point to point commands and controls the spindle like a rotary device, similar to a rotary table motion. It is commanded in angle and distance point to point motion.



G32 paths between commanded points are curves



G32 motion includes both X feed rate and position commands at a constant RPM



Using G32, many small motion commands can result in geometric shapes

FINE SPINDLE CONTROL CODES AND LIVE TOOLING G-CODES

Introduction

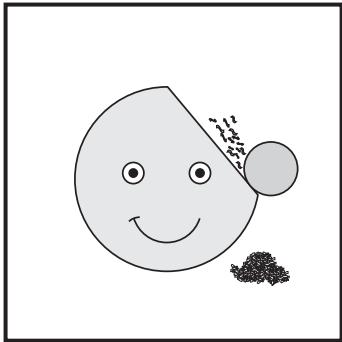
Many uses of live tooling involve holding the spindle still while performing a cut with the live tool. For certain types of operations, it is necessary to move this spindle in a controlled manner while cutting with the live tool.

Fine Spindle Control (FSC) is most commonly used to create features on or near the face of a part, such as grooves, slots, and flat surfaces. Typically an end mill pointing along the Z axis is used to perform the cutting, after pilot holes are drilled. Live tooling is almost always required in order to use FSC. Single point turning is not recommended as the surface feet per minute required is too high for the FSC function.



Limitations

The primary function of the spindle is to turn rapidly. The introduction of G codes for FSC does not change the mechanical design of the spindle motor. Therefore, you should be aware of certain factors that apply when the spindle is turning at very low torque. This limits the depth of cut that can practically be performed with the live tool while the spindle is not locked. In many cases you will want to "track" the motion of the spindle with motion in the X axis.

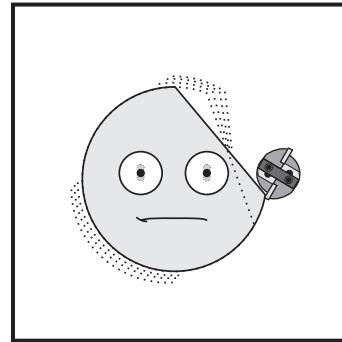


Light Cut



Heavy Cut

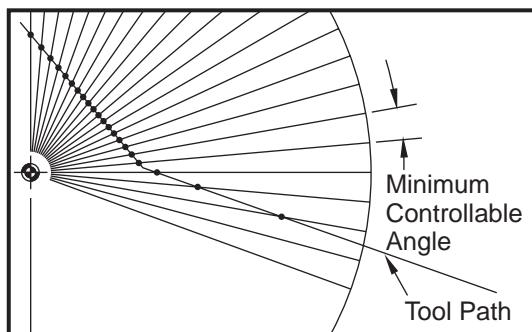
The type of cut and the cutter will have an impact on part finish.



Large Cutter

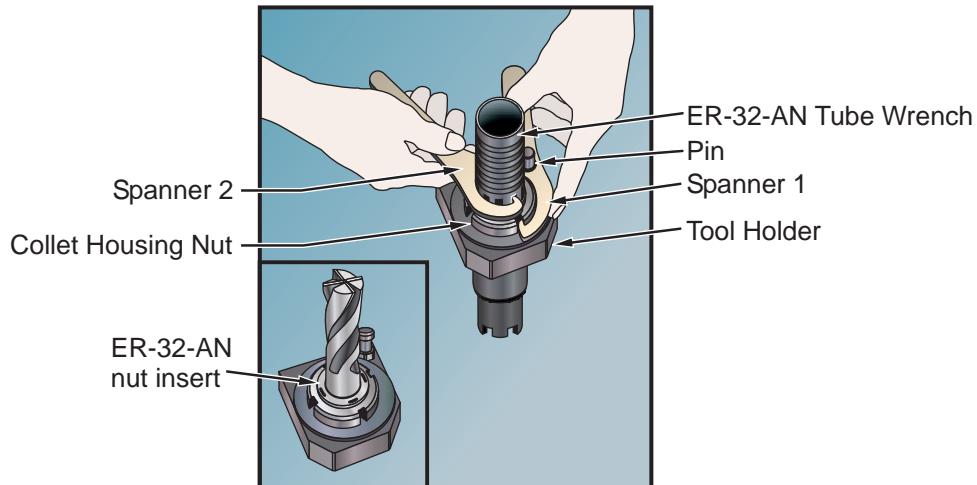
The limit also applies to positioning the spindle in general. This has an effect when trying to perform cuts that are close to centerline.

The number of control points depends on radius and direction of cutter path. Cutter paths with a large radius and a shallow angle towards the center will result in few control points.



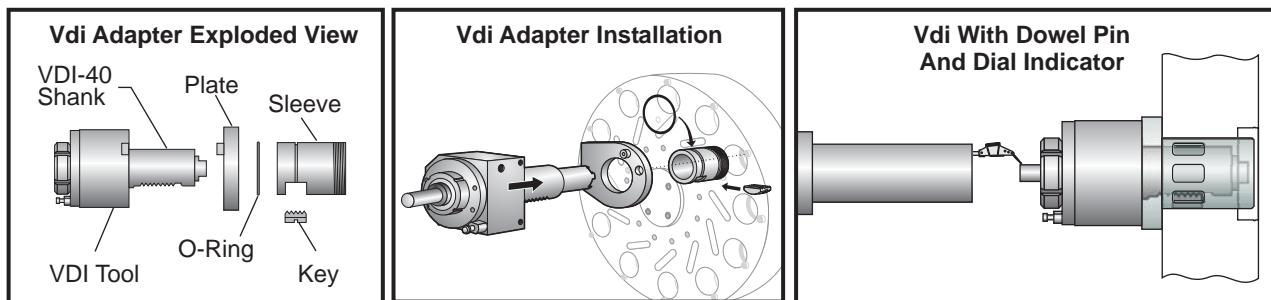
LIVE TOOLING INSTALLATION

1. Insert the tool bit into the ER-AN nut insert. Thread the nut insert into the collet housing nut.
2. Place the ER-32-AN tube wrench over the tool bit and engage the teeth of the ER-AN nut insert. Snug the ER-AN nut insert by hand using the tube wrench.
3. Place Spanner 1 over the pin and lock it against the collet housing nut. It may be necessary to turn the collet housing nut to engage the spanner.
4. Engage the teeth of the tube wrench with Spanner 2 and tighten.



VDI ADAPTER INSTALLATION

VDI adapters make it possible to use VDI-40 tools in haas turrets



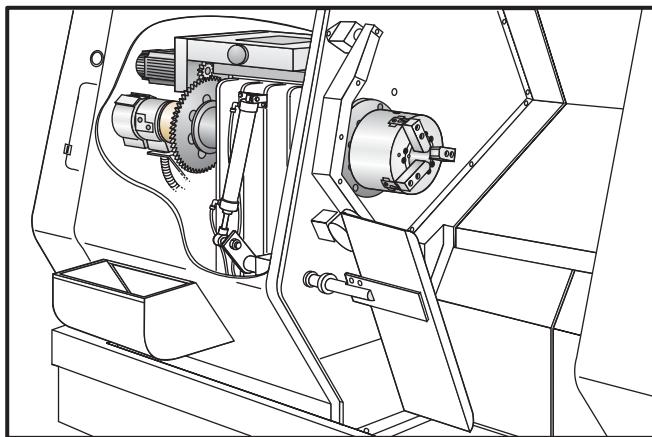
Installation Procedure:

1. Install plate over VDI-40 tool shank. Orient plate boss to VDI tool counterbore.
2. Slide adapter sleeve onto tool shank with cut-out facing towards base of tool shank. Align cut-out with tooth profile of shank.
3. Insert key into sleeve cut-out. Ensure tooth profile of key fits into tool shank properly.
4. Place O-ring in the groove as shown. The O-ring will keep the key from falling out.
5. Install VDI tool with adapter into turret. Ensure the turret locating pin and plate hole are properly aligned.
6. Tighten the draw nut to lock assembly in place.



C-Axis

This option provides high-precision bi-directional spindle motion that is fully interpolated with X and /or Z motion. Spindle speeds from .01 to 60 RPM can be commanded.



OPERATION

M154 C-axis engage

M155 C-axis disengage

Setting 101 Diameter used to calculate the feed rate

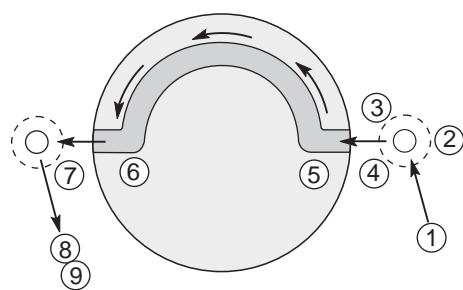
The lathe will automatically disengage the spindle brake when the C-axis is commanded to move and to reengage it afterwards (if it has previously been engaged.)

C-Axis incremental moves are possible using the "H" address code as shown in the following example.

G0 C90.; (C axis moves to 90. deg.)

H-10.; (C axis moves to 80. deg.)

SAMPLE PROGRAMS

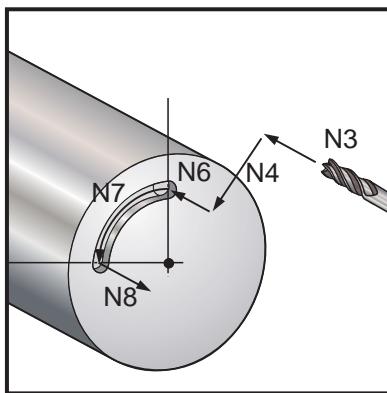


Example #1

M154
G00 G98 (feed/min) X2.0 Z0
C90
G01 Z-0.1 F6.0
X1.0
C180. F10.0
X2.0
G00 Z0.5
M155



Example #2



(Assume pilot hole is already drilled.)
 N1T303 (Small End Mill)
 N2M19 (Orient Spindle)
 N3G00 Z0.5
 N4G00 X1.
 N5M133 P1500
 N6G98 G1 F10. Z-.25 (Plunge into pre-drilled hole)
 N7G05 R90. F40.(Make slot)
 N8G01 F10. Z0.5 (Retract)
 N9M135
 N10 G99 G28 U0 W0

CARTESIAN TO POLAR TRANSFORMATION

Cartesian to Polar coordinate programming that converts X,Y position commands into rotary C-axis and linear X-axis moves. Cartesian to Polar coordinate programming greatly reduces the amount of code required to command complex moves. normally a straight line would require many points to define the path, while in Cartesian, only the end points are necessary. This feature allows face machining programming in the Cartesian coordinate system.

Programming notes:

Programmed moves should always position the tool center line.

Do not use G41 or G42 Cutter Compensation.

Tool paths should never cross the spindle centerline. Cuts that must cross spindle center can be accomplished with two parallel passes on either side of spindle center.

Cartesian to Polar conversion is a modal command (see the G-code section).

Z-axis moves are not allowed while this mode is enabled.

CARTESIAN INTERPOLATION

Cartesian coordinate commands are interpreted into movements of the linear axis (turret movements) and spindle movements (rotation of the workpiece).

Example Program

```
%  

O00069  

N6 (Square)  

G59  

( TOOL 11, .75 DIA. Endmill )  

(Cutting on Center)  

T1111  

M154  

G00 C0.  

G97 M133 P1500  

G00 Z1.  

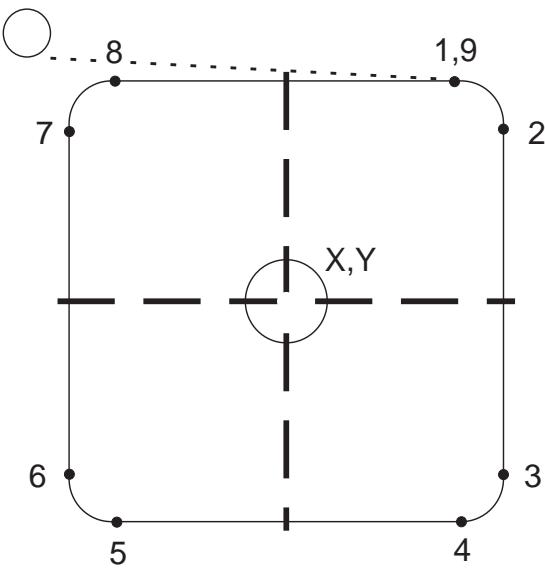
G00 G98 X2.35 Z0.1 (Position)  

G01 Z-0.05 F25.  

G112
```



G17
G0 X-.75 Y.5
G01 X0.45 F10. (Point 1)
G02 X0.5 Y0.45 R0.05 (Point 2)
G01 Y-0.45 (Point 3)
G02 X0.45 Y-0.5 R0.05 (Point 4)
G01 X-0.45 (Point 5)
G02 X-0.5 Y-0.45 R0.05 (Point 6)
G01 Y0.45 (Point 7)
G02 X-0.45 Y0.5 R0.05 (Point 8)
G01 X0.45 (Point 9) Y.6
G113
G00 Z3.
M30
%





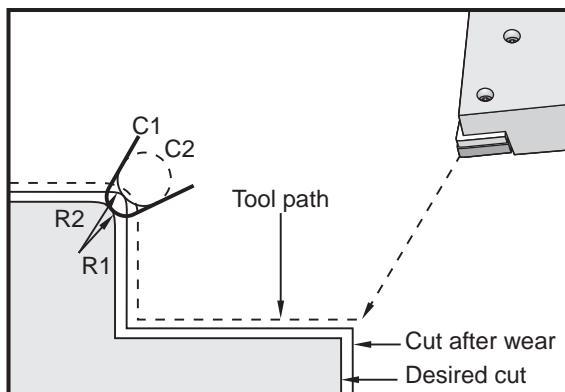
TOOL NOSE COMPENSATION

Introduction

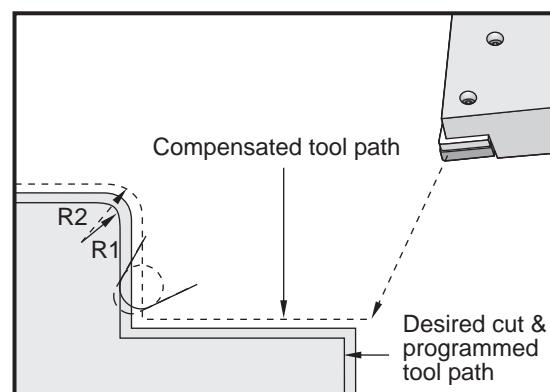
Tool Nose Compensation is a feature that allows the user to adjust a programmed tool path in response to different cutter sizes or for normal cutter wear. The user can do this by entering minimal offset data at run-time without any additional programming effort.

Programming

Tool Nose Compensation is used when the tool nose radius changes, and cutter wear is to be accounted for with curved surfaces or tapered cuts. Tool Nose Compensation generally does not need to be used when programmed cuts are solely along the X or Z axis. For taper and circular cuts, as the tool nose radius changes, under or overcutting can occur. In the figure, suppose that immediately after setup, C1 is the radius of the cutter that cuts the programmed tool path. As the cutter wears to C2, the operator might adjust the tool geometry offset to bring the part length and diameter to dimension. If this were done, a smaller radius would occur. If tool nose compensation is used, a correct cut is achieved. The control will automatically adjust the programmed path based on the offset for tool nose radius as set up in the control. The control will alter or generate code to cut the proper part geometry.



Tool path for 2 cutter radiiuses.



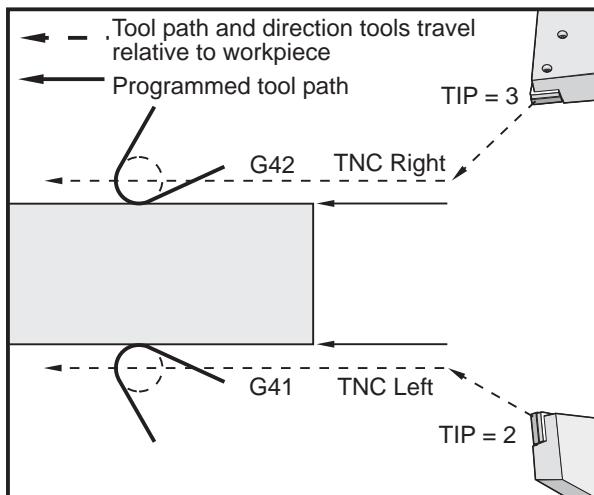
Path generated when tool nose compensation is used.

Note that the second programmed path coincides with the final part dimension. Although parts do not have to be programmed using tool nose compensation, it is the preferred method because it makes program problems easier to detect and resolve.

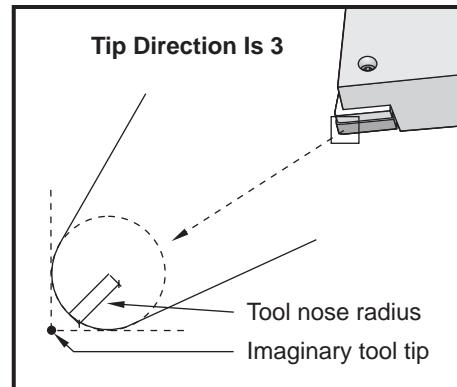


TOOL NOSE COMPENSATION CONCEPTS

Tool nose compensation works by shifting the Programmed Tool Path to the right or to the left. The programmer will usually program the tool path to the finished size. When tool nose compensation is used, the control will compensate for the diameter of the tool based on special instructions written into the program. Two G-code commands are used to do this for compensation within a two-dimensional plane. G41 commands the control to shift to the left of the programmed tool path, and G42 commands the control to shift to the right of the programmed tool path. Another command, G40, is provided to cancel any shift made by tool nose compensation.



Shift Direction



Imaginary Tool Tip

The shift direction is based on the direction of the tool movement relative to the tool, and which side of the part it is on. When thinking about which direction the compensated shift will occur in tool nose compensation, imagine looking down the tool tip and steering the tool. Commanding G41 will move the tool tip to the left and a G42 will move the tool tip to the right. This means that normal O.D. turning will require a G42 for correct tool compensation, while normal I.D. turning will require a G41.

Tool nose compensation assumes that a compensated tool has a radius at the tool tip that it must compensate for. This is called the Tool Nose Radius. Since it is difficult to determine exactly where the center of this radius is, a tool is usually set up using what is called the Imaginary Tool Tip. The control also needs to know which direction the tool tip is relative to the center of the tool nose radius, or the Tip Direction. The tip direction should be specified for each tool.

The first compensated move is generally a move from a non-compensated position to a compensated position and is therefore unusual. This first move is called the "Approach" move and is required when using tool nose compensation. Similarly, a "Depart" move is also required. In a depart move, the control will move from a compensated position to a non-compensated position. A depart move occurs when tool nose compensation is cancelled with a G40 command or Txx00 command. Although approach and depart moves can be precisely planned, they are generally uncontrolled moves and the tool should not be in contact with the part when they occur.



USING TOOL NOSE COMPENSATION

The following are the steps used to program a part using TNC:

Program the part to finished dimensions.

Approach and Departure – Ensure that there is an approach move for each compensated path and determine which direction (G41 or G42) is used. Ensure that there is also a departure move for each compensated path.

Tool Nose Radius and Wear – Select a standard insert (tool with radius) that will be used for each tool. Set the tool nose radius of each compensated tool. Clear the corresponding tool nose wear offset to zero for each tool.

Tool Tip Direction – Input the tool tip direction for each tool that is using compensation, G41 or G42.

Tool Geometry Offset – Set the tool length geometry and clear the length wear offsets of each tool.

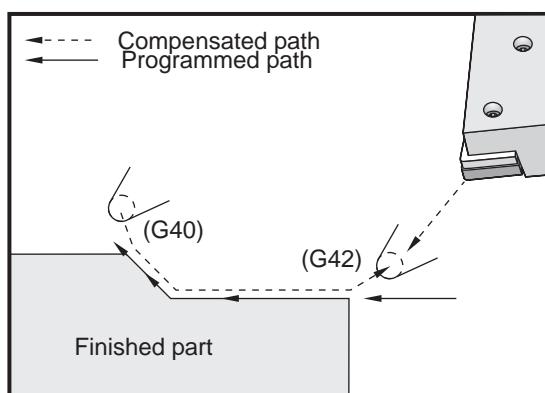
Check Compensation Geometry – Debug the program in graphics mode and correct any tool nose compensation geometry problems that may occur. A problem can be detected in two ways: 1) either an alarm will be generated indicating compensation interference, or 2) the incorrect geometry will be seen generated in graphics mode.

Run and Inspect First Article – Adjust compensated wear for the setup part.

Each of the above steps is described in detail in the following sections.

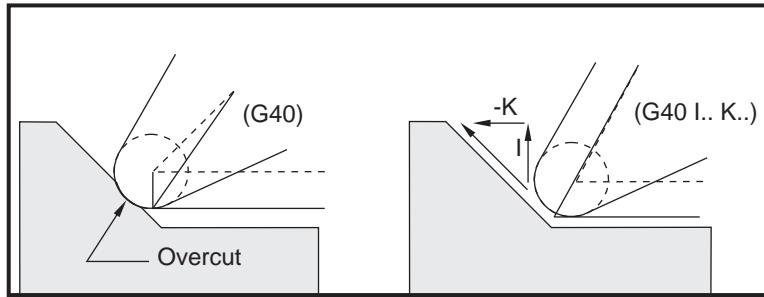
APPROACH AND DEPARTURE MOVES FOR TOOL NOSE COMPENSATION

The first X or Z motion in the same line that contains a G41 or G42 is called the “Approach” move. The approach must be a linear move, that is a G01 or G00. The first move is not compensated, yet at the end of the approach move the machine position will be fully compensated. See the following figure.



Any line of code with a G40 will cancel tool nose compensation and is called the “Departure” move. The departure must be a linear move, that is a G01 or G00. The start of a departure move is fully compensated; the position at this point will be at a right angle to the last programmed block. At the end of the departure move the machine position is not compensated. See the previous figure.

The following figure shows the condition just prior to cancelling tool nose compensation. Some geometries will result in over or undercutting of the part. This is controlled by including an I and K address code in the G40 cancellation block. The I and K in a G40 block define a vector that is used to determine the compensated target position of the previous block. The vector is usually aligned with an edge or wall of the completed part. The following figure shows how I and J can correct undesired cutting in a departure move.



Use of *I* and *K* in a G40 block.

TOOL NOSE RADIUS OFFSET AND WEAR OFFSET

Each turning tool that uses tool nose compensation requires a Tool Nose Radius. The tool tip (tool nose radius) specifies how much the control is to compensate for a given tool. If standard inserts are being used for the tool, then the tool nose radius is simply the tool tip radius of the insert.

Associated with each tool on the geometry offsets page is a Tool Nose Radius Offset. The column labeled "Radius" is the value for the tool nose radius of each tool. If the value of any tool nose radius offset is set to zero, no compensation will be generated for that tool.

Associated with each radius offset is a Radius Wear Offset, located on the wear offset page. The control adds the wear offset to the radius offset to obtain an effective radius that will be used for generating compensated values.

Small adjustments (positive values) to the radius offset during production runs should be placed in the wear offset page. This allows the operator to easily track the wear for a given tool. As a tool is used, the insert will generally wear so that there is a larger radius at the end of the tool. When replacing a worn tool with a new one, the wear offset should be cleared to zero.

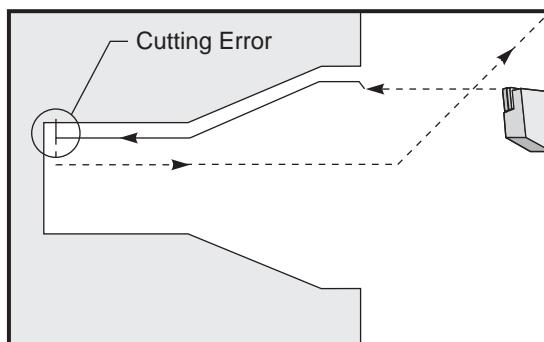
It is important to remember that tool nose compensation values are in terms of radius rather than diameter. This is important when tool nose compensation is cancelled. If the incremental distance of a compensated departure move is not twice the radius of the cutting tool; overcutting will occur. Always remember that programmed paths are in terms of diameter and allow for twice the tool radius on departure moves. The Q block of canned cycles that require a PQ sequence can often be a departure move. The following example illustrates how incorrect programming will result in overcutting.



Example

Setting 33 is FANUC: X	Z	Radius	Tip
Tool Geometry 8: -8.0000	-8.0000	0.0160	2

```
O0010 ;
G28 ;
T808 ;(Boring bar)
G97 2400 M03 ;
G54 G00 X.49 Z.05;
G41 G01 X.5156 F.004 ;
Z-.05 ;
X.3438 Z-.25
Z-.5 ;
X.33; (Move less than .032; the value required to avoid cut-in with a departure move before TNC is cancelled.)
G40 G00 X.25 ;
Z.05 ;
G28 ;
M30 ;
```



An invalid program using TNC and G70.

TOOL NOSE COMPENSATION AND TOOL LENGTH GEOMETRY

The length geometries of tools that use tool nose compensation are set up in the same manner as tools not using compensation. Refer to the "Tooling" section of this manual for details on touching off tools and recording tool length geometries. When a new tool is set up, the geometry wear should be cleared to zero.

Often a tool will exhibit uneven wear. This occurs when particularly heavy cuts occur on one edge of the tool. In this case it may be desirable to adjust the X or Z Geometry Wear rather than the Radius Wear. By adjusting X or Z length geometry wear, the operator can often compensate for uneven tool nose wear. Length geometry wear will shift all dimensions for a single axis.

The program design may not allow the operator to compensate for wear by using length geometry shift. Which wear to adjust can be determined by checking several X and Z dimensions on a finished part. Wear that is even will result in similar dimensional changes on the X and Z axes, and suggests that the radius wear offset should be increased. Wear that affects the dimensions on one axis only suggests length geometry wear.

Good program design based on the geometry of the part being cut should eliminate uneven wear problems. Generally, it is a good idea to rely on finishing tools that use the entire radius of the cutter for tool nose compensation.



TOOL NOSE COMPENSATION IN CANNED CYCLES

Some canned cycles ignore tool nose compensation, expect a specific coding structure, or perform their own specific canned cycle activity (Also see the "Canned Cycles" section).

The following canned cycles will ignore tool nose radius compensation. Cancel tool nose compensation before any of these canned cycles.

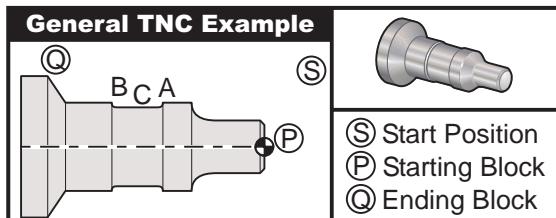
- G74 End face grooving cycle, peck drilling
- G75 O.D./I.D. grooving cycle, peck drilling
- G76 Thread cutting cycle, multiple pass
- G92 Thread cutting cycle, modal

EXAMPLE PROGRAMS USING TOOL NOSE COMPENSATION

This section is comprised of example programs that use tool nose compensation.

Example 1

General Tool Nose Compensation using standard interpolation modes G01/G02/G03.



Preparation

Turn Setting 33 to FANUC.

Set up the following tools

- T1 Insert with .0312 radius, roughing
- T2 Insert with .0312 radius, finishing
- T3 .250 wide grooving tool with .016 radius/same tool for offsets 3 and 13

Tool	Offset	X	Z	RADIUS	TIP
T1	01	-8.9650	-12.8470	.0312	3
T2	02	-8.9010	-12.8450	.0312	3
T3	03	-8.8400	-12.8380	.016	3
T3	13	"	-12.588	.016	4

Program Example

%

O0811 (G42 Test BCA)

Description

(Example 1)

N1 G54 S1000

(Select tool 1 and offset 1 – tip direction for offset 1 is 3)

T101

G97 S500 M03

(Move to point S)

G54 G00 X2.1 Z0.1

G96 S200

(Rough P to Q with T1 Using G71 and TNC)

G71 P10 Q20 U0.02 W0.005 D.1 F0.015

(Define part path PQ sequence)

N10 G42 G00 X0. Z0.1 F.01

(P)(G71 Type II, TNC right)

G01 Z0 F.005

X0.65

X0.75 Z-0.05



Z-0.75
G02 X1.25 Z-1. R0.25
G01 Z-1.5
G02 X1. Z-1.625 R0.125
G01 Z-2.5
G02 X1.25 Z-2.625 R0.125
G01 Z-3.5
X2. Z-3.75
N20 G00 G40 X2.1
G97 S500
G28
M01
N2 G50 S1000
T202
G97 S750 M03
G00 X2.1 Z0.1
G96 S400
(Finish P to Q with T2 using G70 and TNC)
G70 P10 Q20
G97 S750
G28
M01
N3 G50 S1000
T303
G97 S500 M03
G54 G42 X1.5 Z-2.0
G96 S200
G01 X1. F0.003
G01 Z-2.5
G02 X1.25 Z-2.625 R0.125
G40 G01 X1.5
T313
G00 G41 X1.5 Z-2.125
G01 X1. F0.003
G01 Z-1.625
G03 X1.25 Z-1.5 R0.125
G40 G01 X1.6
G97 S500
G28
M30
%

(A)

(B)

(TNC Cancel)

(Zero for tool change clearance)

(Select tool 2 and offset 2 Tip direction is 3)
(move to point S)

(Zero for tool change clearance)

(Select tool 3, offset 3 Tip direction is 3)
(Groove to point B Using Offset 3)
(Move to point C TNC right)

(B)
(TNC cancel – Groove to point A using offset 4)
(Change offset to other side of tool)
(Move to point C – TNC approach)

(A)
(TNC cancel)

Note that the suggested template of the previous section for G70 is used. Also note that compensation is enabled in the PQ sequence but is cancelled after G70 is completed.

Example 2

TNC with a G71 roughing canned cycle

Preparation

Turn Setting 33 to FANUC.

Tools

T1 Insert with .032 radius, roughing

Tool	Offset	Radius	Tip
T1	01	.032	3



Program Example

%
O0813
G50 S1000
T101
G00 X1.5 Z.1
G96 S100 M03
G71 P80 Q180 U.01 W.005 D.08 F.012

N80 G42 G00 X0.6
G01 Z0 F0.01
X0.8 Z-0.1 Fx0.005
Z-0.5
G02 X1.0 Z-0.6 I0.1
G01 X1.5
X2.0 Z-0.85
Z-1.6
X2.3
G03 X2.8 Z-1.85 K-0.25
G01 Z-2.1
N180 G40 G00 X3.0 M05
G28
M30
%

Description

(Example 3)

(Select tool 1)
(Rapid to start point)

(Rough P to Q with T1 using G71 and TNC)

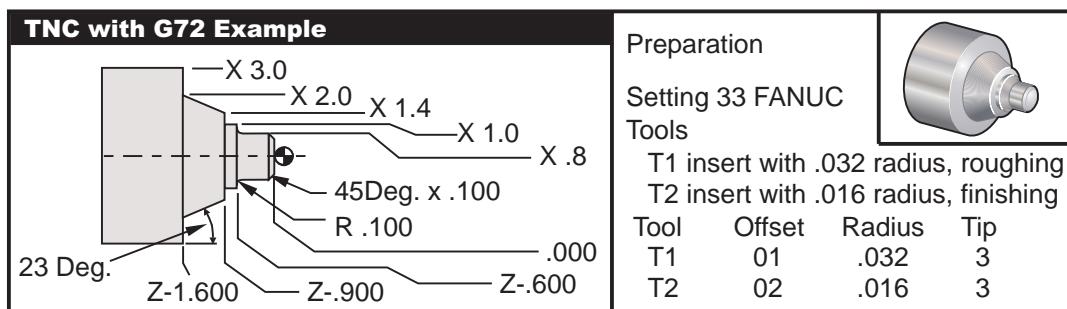
(Define Part Path PQ sequence)
(P) (G71 Type I, TNC right)
(Start of finish part path)

(Q) (End of part path)
(TNC cancel)
(Zero X for tool change clearance)

Note that this part is a G71 Type I path. When using TNC it is very unusual to have a Type II path. This is because the current compensation methods can only compensate the tool tip in one direction.

Example 3

TNC with a G72 roughing canned cycle



Program Example

%
O0814
G50 S1000
T101
G00 X3.5 Z0.1
G96 S190 M03
(Rough P to Q with T1 using G72 and TNC)
G72 P80 Q180 U0.005 W0.01 D0.05 F.010
(Define part path PQ sequence)
N80 G41 G00 Z-1.6
G01 X2. F0.005
X1.4 Z-0.9

(P) (G72 Type I, TNC left)

Description

(Example 3)

(Select tool 1)
(Move to start point)



X1.0
Z-0.6
G03 X0.8 Z-0.5 K0.1
G01 Z-0.1
X0.6 Z0.
X0.
N180 G40 G00 Z0.01 *(TNC Cancel)*
*(*****Optional Finishing Sequence*****)*
G28 *(Zero for tool change clearance)*
M01
T202 *(Select tool 2)*
N2 G50 S1000
G00 X3.5 Z0.1 *(Move to start point)*
G96 S325 M03 *(Finish P to Q with T2 using G70 and TNC)*
G70 P80 Q180
G00 Z0.5 M05
G28 *(Zero for tool change clearance)*
M30
%

G72 is used instead of G71 because the roughing strokes in X are longer than the Z roughing strokes of a G71. It is therefore more efficient to use G72.

Example 4

TNC with a G73 roughing canned cycle

Preparation

Turn Setting 33 to FANUC

Tools

T1 Insert with .032 radius, roughing
T2 Insert with .016 radius, finishing

Tool	Offset	Radius	Tip
T1	01	.032	3
T2	02	.016	3

Program Example

	Description
%	
O0815	<i>(Example 4)</i>
T101	<i>(Select Tool 1)</i>
G50 S1000	
G00 X3.5 Z.1	<i>(Move to point S)</i>
G96 S100 M03	
G73 P80 Q180 U.01 W0.005 I0.3 K0.15 D4 F.012	<i>(Rough P to Q with T1 using G73 and TNC)</i>
N80 G42 G00 X0.6	<i>(Part path PQ sequence, G72 Type I, TNC right)</i>
G01 Z0 F0.1	
X0.8 Z-0.1 F.005	
Z-0.5	
G02 X1.0 Z-0.6 I0.1	
G01 X1.4	
X2.0 Z-0.9	
Z-1.6	
X2.3	
G03 X2.8 Z-1.85 K-0.25	
G01 Z-2.1	
N180 G40 X3.1	<i>(Q)</i>
G00 Z0.1 M05	<i>(TNC Cancel)</i>



(*****Optional Finishing Sequence*****)

G28

(Zero for tool change clearance)

M01

T202

(Select tool 2)

N2 G50 S1000

G00 X3.0 Z0.1

(Move to start point)

G96 S100 M03

(Finish P to Q with T2 using G70 and TNC)

G70 P80 Q180

G00 Z0.5 M05

G28

(Zero for tool change clearance)

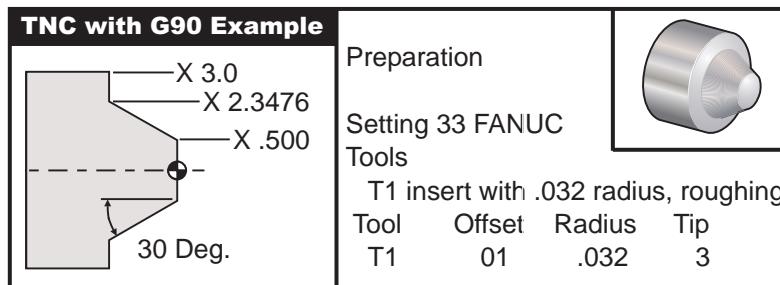
M30

%

G73 is best used when you want to remove a consistent amount of material in both the X and Z axes.

Example 5

TNC with a G90 modal rough turning cycle



Program Example

Description

%

O0816

(Example 5)

T101

(Select tool 1)

G50 S1000

G00 X4.0 Z0.1

(Move to start point)

G96 S100 M03

(ROUGH 30 DEG. ANGLE TO X2. AND Z-1.5 USING G90 AND TNC)

G90 G42 X2.55 Z-1.5 I-0.9238 F0.012

(Optional Additional Passes)

X2.45

X2.3476

G00 G40 X3.0 Z0.1 M05

(TNC Cancel)

G28

(Zero for tool change clearance)

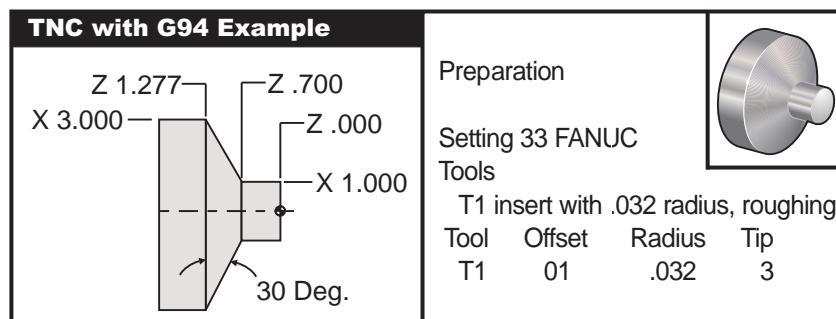
M30

%



Example 6

TNC with a G94 modal rough turning cycle



Program Example

```
%  
O0817  
G50 S1000  
T101  
G00 X3.0 Z0.1  
G96 S100 M03  
G94 G41 X1.0 Z-0.5 L-0.577 F.03  
Z-0.6  
Z-0.7  
G00 G40 X3.0 Z0.1 M05  
G28  
M30  
%
```

Description

(Example 6)

(Select tool 1)
(Move to start point)

(Rough 30° angle to X1. and Z-0.7 using G94 and TNC)
(Optional additional passes)

(TNC Cancel)
(Zero for tool change clearance)

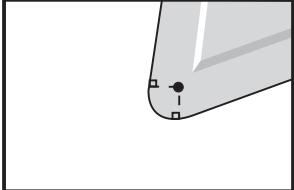
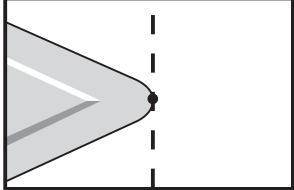
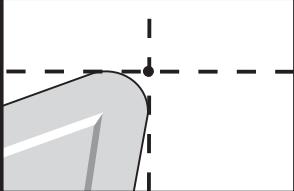
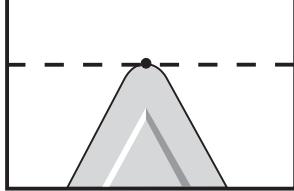
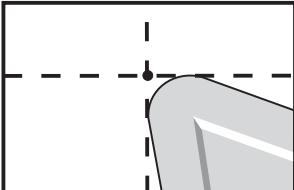
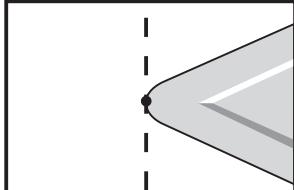
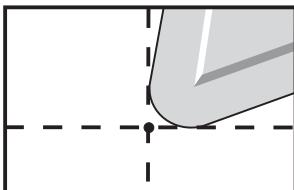
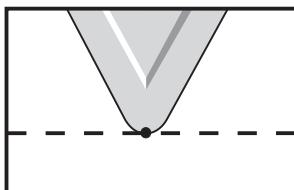
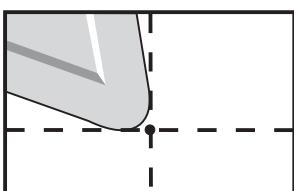
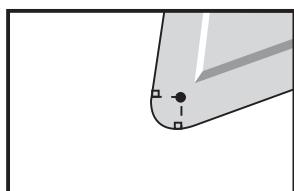


IMAGINARY TOOL TIP AND DIRECTION

For a lathe it is not easy to determine the center of a tool radius. The cutting edges are set when a tool is touched off to record tool geometry. The control can calculate where the center of the tool radius is by using the edge information, the tool radius, and the direction the cutter is expected to cut in. The X and Z axis geometry offsets intersect at a point, called the Imaginary Tool Tip, that aids in determining the tool tip direction. The Tool Tip Direction is determined by a vector originating from the center of the tool radius and extending to the imaginary tool tip. See the following figures.

The tool tip direction of each tool is coded as a single integer from 0 to 9. The tip direction code is found next to the radius offset on the geometry offsets page. It is recommended that a tip direction be specified for all tools using tool nose compensation. The following figure is a summary of the tip coding scheme along with cutter orientation examples.

Note that the tip indicates to the setup person how the programmer intends to measure the tool offset geometry. For example, if the setup sheet shows tip direction 8, the programmer intends the tool geometry to be at the edge of and on the centerline of the tool insert.

Tip Code	Imaginary Tool Tip Orientation	Tool Center Location	Tip Code	Imaginary Tool Tip Orientation	Tool Center Location
0		Zero (0) indicates no specified direction. It is usually not used when Tool Nose Compensation is desired.	5		Direction Z+: Tool edge
1		Direction X+, Z+: Off tool	6		Direction X+: Tool edge
2		Direction X+, Z-: Off tool	7		Direction Z-: Tool edge
3		Direction X-, Z-: Off tool	8		Direction X-: Tool edge
4		Direction X-, Z+: Off tool	9		Same as Tip 0



PROGRAMMING WITHOUT TOOL NOSE COMPENSATION

Manually Calculating Compensation

When you program a straight line in either X or Z axes the tool tip touches the part at the same point where you touched your original tool offsets in X and Z axes. However, when you program a chamfer or an angle, the tip does not touch the part at those same points. Where the tip actually touches the part is dependent upon the degree of angle being cut and also the size of the tool insert. Programming a part without using any compensation, overcutting and undercutting will occur.

The following pages contain tables and illustrations demonstrating how to calculate the compensation in order to program the part accurately.

Along with each chart are three examples of compensation using both types of inserts and cutting along three different angles. Next to each illustration is a sample program and explanation of how the compensation was calculated.

Refer to the illustrations on the following pages

The tool tip is shown as a circle with X and Z points called out. These points designate where the X diameter and Z face offsets are touched off.

Each illustration is a 3" diameter part with lines extending from the part and intersecting at 30°, 45° and 60° angles.

The point at which the tool tip intersects the lines is where the compensation value is measured.

The compensation value is the distance from the face of the tool tip to the corner of the part. Notice that the tool tip is slightly offset from the actual corner of the part; this is so the tool tip is in the correct position to make the next move and to avoid any overcutting or undercutting.

Use the values found on the charts (angle and radius size) to calculate the correct tool path position for the program.

TOOL NOSE COMPENSATION GEOMETRY

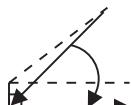
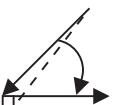
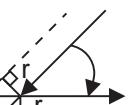
The following figure shows the various geometries of tool nose compensation. It is organized into four categories of intersection. The intersections can be: 1) linear to linear, 2) linear to circular, 3) circular to linear, or 4) circular to circular. Beyond these categories the intersections are classified into angle of intersection and approach, mode to mode, or departure motions.

Two FANUC compensation types are supported, Type A and Type B. The default compensation is Type A.

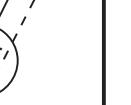


Linear-to-Linear (Type A)

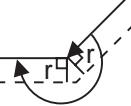
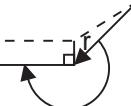
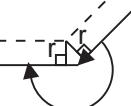
Angle: <90

	Approach	Mode to Mode	Departure
G41			
G42			

Angle: >=90, <180

	Approach	Mode to Mode	Departure
G41			
G42			

Angle: >180

	Approach	Mode to Mode	Departure
G41			
G42			

Linear-to-Circular (Type A)

Angle: <90

	Approach	Mode to Mode	Departure
G41			Not Permitted
G42			Not Permitted

Angle: >=90, <180

	Approach	Mode to Mode	Departure
G41			Not Permitted
G42			Not Permitted

Angle: >180

	Approach	Mode to Mode	Departure
G41			Not Permitted
G42			Not Permitted



Circular-to-Linear (Type A)

Angle: <90

	Approach	Mode to Mode	Departure
G41	Not Permitted		
G42	Not Permitted		

Angle: >=90, <180

	Approach	Mode to Mode	Departure
G41	Not Permitted		
G42	Not Permitted		

Angle: >180

	Approach	Mode to Mode	Departure
G41	Not Permitted		
G42	Not Permitted		

Circular-to-Circular (Type A)

Angle: <90

	Approach	Mode to Mode	Departure
G41	Not Permitted		Not Permitted
G42	Not Permitted		Not Permitted

Angle: >=90, <180

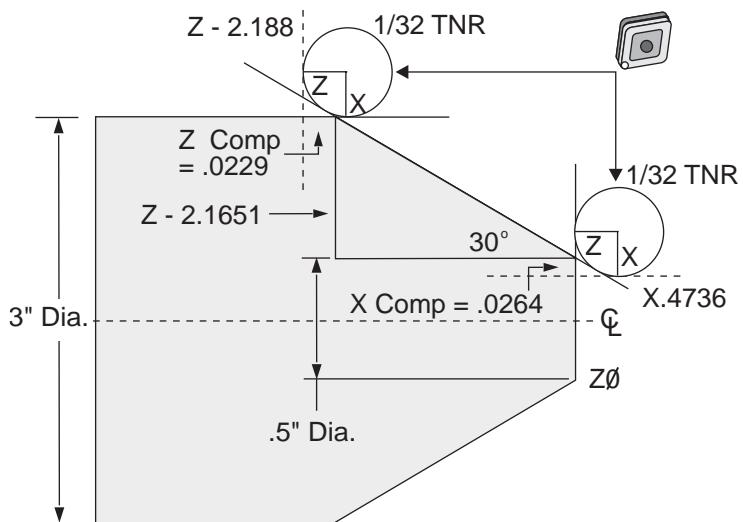
	Approach	Mode to Mode	Departure
G41	Not Permitted		Not Permitted
G42	Not Permitted		Not Permitted

Angle: >180

	Approach	Mode to Mode	Departure
G41	Not Permitted		Not Permitted
G42	Not Permitted		Not Permitted

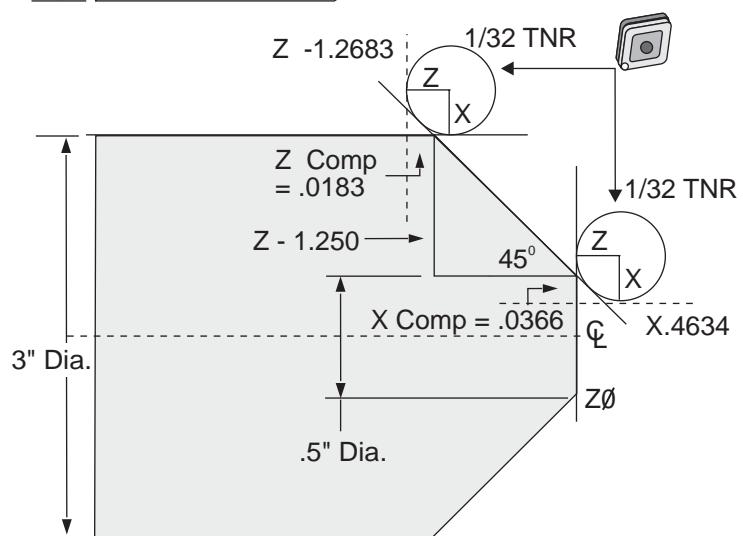


Tool Nose Radius Calculation Diagram



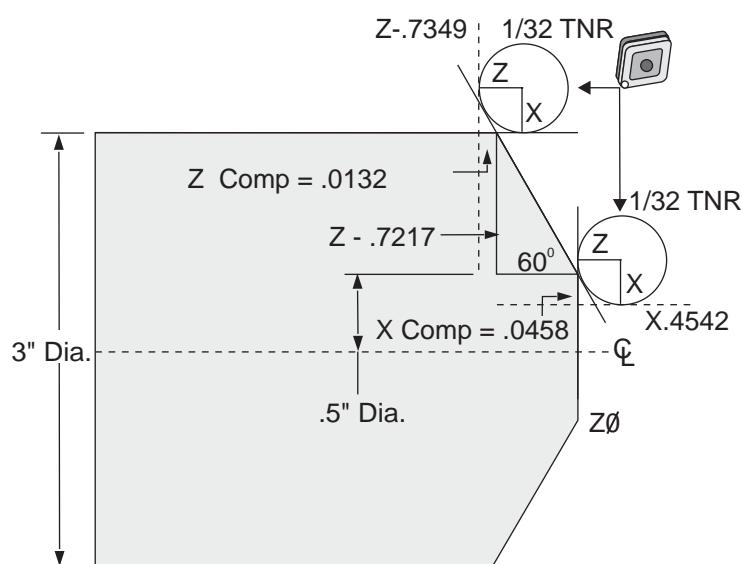
Program	
Code	Compensation (1/32 TNR)
G0 X0 Z.1	
G1 Z0	
X.4736	(X.5 - 0.0264 Comp)
X 3.0 Z-2.188	(Z-2.1651 + 0.0229 Comp)

Note: Compensation Value For 30° Angle



Program	
Code	Compensation (1/32 TNR)
G0 X0 Z.1	
G1 Z0	
X.4634	(X.5 - 0.0366 Comp)
X 3.0 Z-1.2683	(Z-1.250+ 0.0183 Comp)

Note: Compensation Value For 45° Angle



Program	
Code	Compensation (1/32 TNR)
G0 X0 Z.1	
G1 Z0	
X.4542	(X.5 - 0.0458 Comp)
X 3.0 Z-.7349	(Z-.7217+ 0.0132 Comp)

Note: Compensation Value For 60° Angle



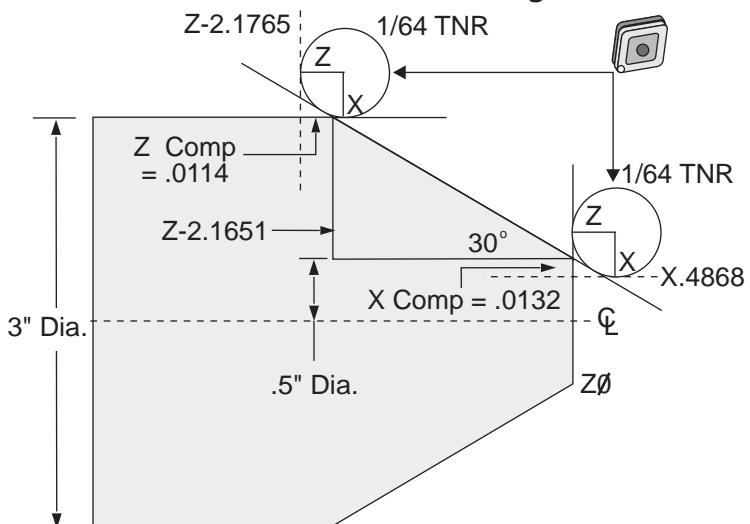
Tool Radius And Angle Chart (1/32 RADIUS)

The X measurement calculated is based on part diameter.

ANGLE	Xc CROSS	Zc LONGITUDINAL	ANGLE	Xc CROSS	Zc LONGITUDINAL
1.					
2.	.0010	.0310	46.	.0372	.0180
3.	.0022	.0307	47.	.0378	.0177
4.	.0032	.0304	48.	.0386	.0173
5.	.0042	.0302	49.	.0392	.0170
6.	.0052	.0299	50.	.0398	.0167
7.	.0062	.0296	51.	.0404	.0163
8.	.0072	.0293	52.	.0410	.0160
9.	.0082	.0291	53.	.0416	.0157
10.	.0092	.0288	54.	.0422	.0153
11.	.01	.0285	55.	.0428	.0150
12.	.0011	.0282	56.	.0434	.0146
13.	.0118	.0280	57.	.0440	.0143
14.	.0128	.0277	58.	.0446	.0139
15.	.0136	.0274	59.	.0452	.0136
16.	.0146	.0271	60.	.0458	.0132
17.	.0154	.0269	61.	.0464	.0128
18.	.0162	.0266	62.	.047	.0125
19.	.017	.0263	63.	.0474	.0121
20.	.018	.0260	64.	.0480	.0117
21.	.0188	.0257	65.	.0486	.0113
22.	.0196	.0255	66.	.0492	.0110
23.	.0204	.0252	67.	.0498	.0106
24.	.0212	.0249	68.	.0504	.0102
25.	.022	.0246	69.	.051	.0098
26.	.0226	.0243	70.	.0514	.0094
27.	.0234	.0240	71.	.052	.0090
28.	.0242	.0237	72.	.0526	.0085
29.	.025	.0235	73.	.0532	.0081
30.	.0256	.0232	74.	.0538	.0077
31.	.0264	.0229	75.	.0542	.0073
32.	.0272	.0226	76.	.0548	.0068
33.	.0278	.0223	77.	.0554	.0064
34.	.0286	.0220	78.	.056	.0059
35.	.0252	.0217	79.	.0564	.0055
36.	.03	.0214	80.	.057	.0050
37.	.0306	.0211	81.	.0576	.0046
38.	.0314	.0208	82.	.0582	.0041
39.	.032	.0205	83.	.0586	.0036
40.	.0326	.0202	84.	.0592	.0031
41.	.0334	.0199	85.	.0598	.0026
42.	.034	.0196	86.	.0604	.0021
43.	.0346	.0193	87.	.0608	.0016
44.	.0354	.0189	88.	.0614	.0011
45.	.036	.0186	89.	.062	.0005
	.0366	.0183			

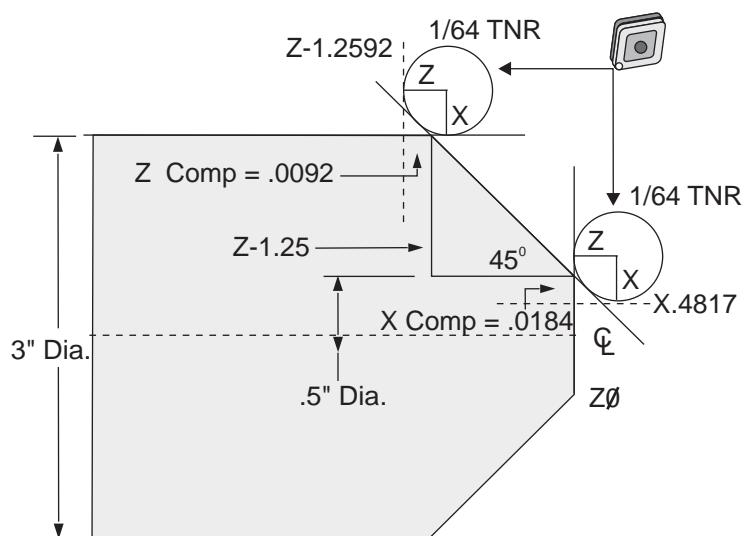


Tool Nose Radius Calculation Diagram



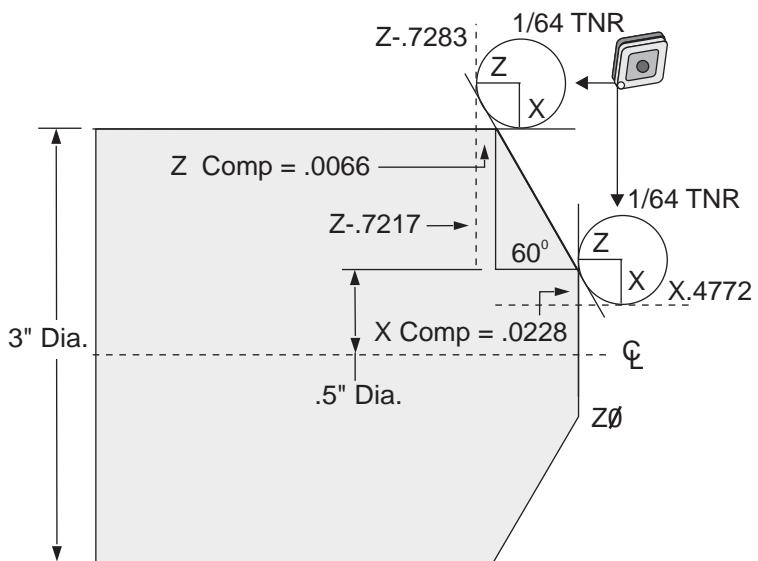
Program	
Code	Compensation (1/64 TNR)
G0 X0 Z.1	
G1 Z0	
X.4868	(X.5 - 0.0132 Comp)
X 3.0 Z-2.1765	(Z-2.1651 + 0.0114 Comp)

Note: Using Compensation Values For 30°



Program	
Code	Compensation(1/64 TNR)
G0 X0 Z.1	
G1 Z0	
X.4816	(X.5 - 0.0184 Comp)
X 3.0 Z-1.2592	(Z-1.25 + 0.0092 Comp)

Note: Using Compensation Values For 45°



Program	
Code	Compensation (1/64 TNR)
G0 X0 Z.1	
G1 Z0	
X.4772	(X.5 - 0.0228 Comp)
X 3.0 Z-.467	(Z-.7217 + 0.0066 Comp)

Note: Using Compensation Values For 60°



Tool Radius And Angle Chart (1/64 Radius)

The X measurement calculated is based on part diameter.

ANGLE	Xc CROSS	Zc LONGITUDINAL	ANGLE	Xc CROSS	Zc LONGITUDINAL
1.	.0006	.0155	46.	.00186	.0090
2.	.0001	.0154	47.	.0019	.0088
3.	.0016	.0152	48.	.0192	.0087
4.	.0022	.0151	49.	.0196	.0085
5.	.0026	.0149	50.	.0198	.0083
6.	.0032	.0148	51.	.0202	.0082
7.	.0036	.0147	52.	.0204	.0080
8.	.0040	.0145	53.	.0208	.0078
9.	.0046	.0144	54.	.021	.0077
10.	.0050	.0143	55.	.0214	.0075
11.	.0054	.0141	56.	.0216	.0073
12.	.0060	.0140	57.	.022	.0071
13.	.0064	.0138	58.	.0222	.0070
14.	.0068	.0137	59.	.0226	.0068
15.	.0072	.0136	60.	.0228	.0066
16.	.0078	.0134	61.	.0232	.0064
17.	.0082	.0133	62.	.0234	.0062
18.	.0086	.0132	63.	.0238	.0060
19.	.0090	.0130	64.	.024	.0059
20.	.0094	.0129	65.	.0244	.0057
21.	.0098	.0127	66.	.0246	.0055
22.	.0102	.0126	67.	.0248	.0053
23.	.0106	.0124	68.	.0252	.0051
24.	.011	.0123	69.	.0254	.0049
25.	.0014	.0122	70.	.0258	.0047
26.	.0118	.0120	71.	.0260	.0045
27.	.012	.0119	72.	.0264	.0043
28.	.0124	.0117	73.	.0266	.0041
29.	.0128	.0116	74.	.0268	.0039
30.	.0132	.0114	75.	.0272	.0036
31.	.0136	.0113	76.	.0274	.0034
32.	.014	.0111	77.	.0276	.0032
33.	.0142	.0110	78.	.0280	.0030
34.	.0146	.0108	79.	.0282	.0027
35.	.015	.0107	80.	.0286	.0025
36.	.0154	.0103	81.	.0288	.0023
37.	.0156	.0104	82.	.029	.0020
38.	.016	.0102	83.	.0294	.0018
39.	.0164	.0101	84.	.0296	.0016
40.	.0166	.0099	85.	.0298	.0013
41.	.017	.0098	86.	.0302	.0011
42.	.0174	.0096	87.	.0304	.0008
43.	.0176	.0095	88.	.0308	.0005
44.	.018	.0093	89.	.031	.0003
45.	.0184	.0092			



PROGRAMMING

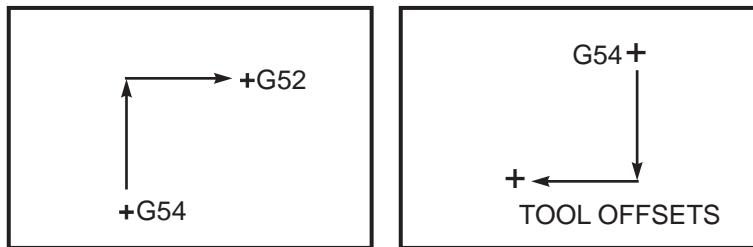
CNC controls use a variety of coordinate systems and offsets that allow the user to control the location of the tooling point to the part. This section describes the interaction between various coordinate systems and tooling offsets.

Effective Coordinate System

The effective coordinate system is the sum total of all coordinate systems and offsets in effect. It is the system that is displayed under the label "Work" on the positions display. It is also the same as the programmed values in a G code program assuming no Tool Nose Compensation is being performed.

Effective Coordinate = global coordinate
+ common coordinate
+ work coordinate
+ child coordinate
+ tool offsets

The figure below indicates how these coordinate systems are related to each other to make up the effective coordinate system.



FANUC

Work Coordinate Systems

Work coordinates are an additional optional coordinate shift relative to the global coordinate system. There are 26 work coordinate systems available on a HAAS control, designated G54 through G59 and as G110 through G129. G54 is the work coordinate in effect when the control is powered on. The last used work coordinate stays in effect until another work coordinate is used or the machine is powered off. G54 can be deselected by ensuring that the X and Z values on the work offset page for G54 are set to zero.

Child Coordinate System

A child coordinate is a coordinate system within a work coordinate. Only one child coordinate system is available and it is set through the G52 command. Any G52 set during the program is removed once the program finishes at an M30, Reset or power down.

Common Coordinate System

The common (Comm) coordinate system is found on the second work coordinate offsets display page just below the global coordinate system (G50). The common coordinate system is retained in memory when power is turned off. The common coordinate system can be changed manually with G10 command or by using macro variables.

YASNAC

Work Coordinate Shift

YASNAC controls discuss a work coordinate shift. It serves the same function as the common coordinate system. When Setting 33 is set to YASNAC, it is found on the work offsets display page as T00.



Machine Coordinate System

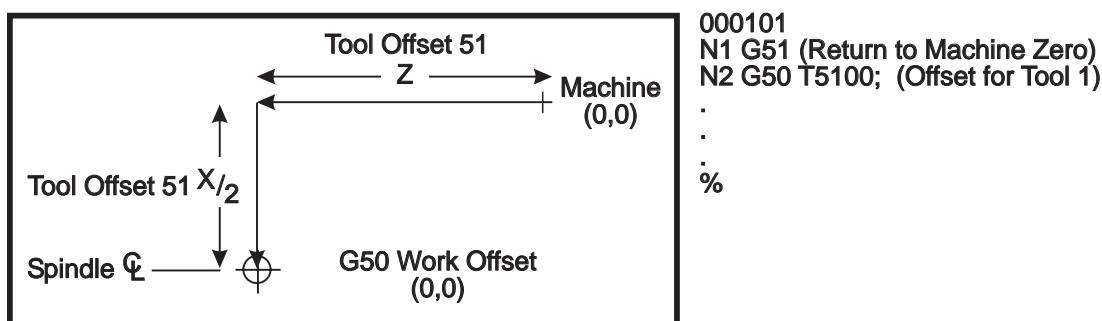
The effective coordinates take the value from machine zero coordinates. Machine coordinates can be referenced by specifying G53 with X and Z in a motion block.

Tool Offsets

There are two offsets available: geometry offsets and wear offsets. Geometry offsets adjust for different lengths and widths of tools, so that every tool comes to the same reference plane. Geometry offsets are usually done at setup time and remain fixed. Wear offsets allow the operator to make minor adjustments to the geometry offsets to compensate for normal tool wear. Wear offsets are usually zero at the beginning of a production run and may change as time progresses. In a FANUC compatible system, both geometry and wear offsets are used in the calculation of the effective coordinate system.

Tool Offsets

Geometry offsets are not available; they are replaced with tool shift offsets. YASNAC tool shift offsets modify the global coordinate to allow for varying tool lengths. Tool shift offsets must be used prior to calling the use of a tool with using a G50 Txx00 command. The tool shift offset replaces any previously calculated global shift offset and a G50 command overrides a previously selected tool shift. There are 50 tool shift offsets numbered 51 through 100.



G50 YASNAC Tool Shift.

Automatic Setting of Tool Offsets

Tool offsets are recorded automatically using the X DIA MESUR or the Z FACE MESUR key. If the common, global, or currently selected work offset have values assigned to them, the recorded tool offset will differ from actual machine coordinates by these values. After setting up tools for a job, all tools should be commanded to a safe X, Z coordinate reference point as a tool change location.

Global Coordinate System (G50)

The global coordinate system is a single coordinate system that shifts all work coordinates and tool offsets away from machine zero. The global coordinate system is calculated by the control so the current machine location becomes the effective coordinates specified by a G50 command. The calculated global coordinate system values can be seen on the work coordinate offsets display just below auxiliary work offset 129. The global coordinate system is cleared to zero automatically when the CNC control is powered on. The global coordinate is not changed when RESET is pressed.



TIPS AND TRICKS

Programming

Short programs looped many times will not reset the chip conveyor if the intermittent feature is activated. The conveyor will continue to start and stop at the commanded times. See settings 114 and 115.

The Current Commands screen displays the spindle and axis loads, the current feed and speed, and 15 lines of the current program.

The Origin button can be used to clear offsets and macro variables. This is accomplished by going to the Offsets (Macros) screen and pressing the Origin button. The control will display the prompt: ZERO ALL (Y/N). If "Y" is entered all the Offsets (Macros), in the area being displayed will be set to zero. The values in the Current Commands display pages can be cleared as well. The Tool Life, Tool Load, and Timer registers can be cleared by selecting the one to clear and press ORIGIN. To clear everything in a column, cursor to the top of the column, onto the title, and press ORIGIN.

Selecting another program quickly can be accomplished simply by entering the program number (Onnnnn) and pressing arrow up or down. The machine must be in either MEM or EDIT mode. Searching for a specific command in a program can be done as well in either MEM or EDIT. Enter the address code (A, B, C etc.), or the address code and the value. (A1.23), and press the up or down arrow button. If the address code is entered without a value, the search will stop at the next use of that letter.

You can transfer and save a program in MDI to your list of programs. When on the MDI program display, position the cursor at the beginning of the MDI program. Enter a program number (Onnnnn), Then press ALTER and this will transfer the MDI program into your list of programs under that program number.

Program Review - Pressing F4 while in PRGRM display or MEM mode, displays a split screen, which shows the program running on the left and displays the program on the right for the operator to cursor through and review.

Background Edit Type in a program number (Onnnnn) of the program you want to edit and then press F4, while in the PRGRM display (the program can be running in MEM operating mode). Simple edits, Insert, Alter, Delete and Undo can be done to either an existing program, a new program, or even the program which is running. However, the running program will not update until the program ends with an M30 or Reset.

Graphics Zoom Window F2 will activate the zoom window. Page down will zoom in and page up will expand the view. Use the arrow keys to move the window over the desired area of the part and press Enter. Press F2 and Home to see full table view.

In the Edit Mode a program can be copied (Insert) into another program, a line, or a block of lines in a program. Start by defining a block with the Edit: Select Text menu, then cursor to the last program line to define, press F2 or Write to highlight the block. Select another program to copy the selection to. Cursor to the point where the copied block will be placed and press Insert.

Loading multiple files is possible from the Advanced Editor. Go to the Floppy Disk Directory menu. The control will load the selected file when you press Enter. The cursor will still remain to allow further files to be selected and loaded.

Editing programs Pressing the F4 key while in the Advanced Editor will display another version of the current program to edit. Different portions of the programs can be edited alternately by pressing the Edit key to switch from one side to the other. The program will be updated once switched to the other program.

Duplicating a Program Using the List Prog mode, an existing program can be duplicated. To do this select the program number you wish to duplicate, type in a new program number (Onnnnn) and press F1. Duplication can also be done by selecting the Program menu, then Duplicate Active Program menu in Edit.

Several programs can be sent to the serial port by typing all the program names together on the input line without spaces (e.g. O12345O98765O45678) and pressing Send RS232.



When you send files to the floppy disk, you must put the highlighted cursor on the program you are saving or on the "ALL". The name entered on the input line is the floppy disk file name.

Offsets

Entering offsets: Pressing WRITE will add the entered number to the cursor-selected value. Pressing F1 will take the entered number and overwrite the cursor selected offset register. Pressing F2 will enter the negative value into as the offset.

Pressing OFSET will toggle back and forth between the Tool Length Offsets and Work Zero Offset pages

Settings and Parameters

Jog handle can be used to scroll through settings and parameters

This control can turn itself off using settings. These settings are: Setting 1 to turn off after machine is idle for nn minutes, and Setting 2 to turn off when M30 is executed.

Memory Lock (Setting 8) When this setting is turned **ON**, memory edit functions are locked out. When it is **OFF**, memory can be modified.

Dimensioning (Setting 9) changes from Inch to MM; This will change all offset values too.

Reset Program Pointer (Setting 31), turns on and off the program pointer returning to the beginning of the program.

Scale Integer F (Setting 77), this is used to change the interpretation of a feed rate. A feed rate can be misinterpreted if there is not a decimal point in the Fnn command. The selections for this setting can be "Default", to recognize a 4 place decimal. Another selection is "Integer" which will recognize a feed rate for a selected decimal position, for a feed rate that does not have a decimal.

Max Corner Rounding (Setting 85) is used to set the corner rounding accuracy required by the user. Any feed rate up to the maximum can be programmed, without the errors ever getting above that setting. The control will **only** slow at corners when needed.

Reset Resets Override (Setting 88) turns on and off the Reset key setting the overrides back to 100%.

Cycle Start / Feed hold - Same key (Setting 103) When this setting is turned ON, Cycle Start must be pressed and held to run a program. Releasing Cycle Start, generates a Feed Hold condition.

Jog Handle to Single Block (Setting 104) The jog handle can be used to step through a program. Reversing the jog handle will generate a Feed Hold condition.

Offset Lock (Setting 119) Prevents the operator from altering any of the offsets.

Macro Variable Lock (Setting 120) Prevents the operator from altering any of the macro variables.

Operation

Memory Lock Key Switch prevents the operator from editing programs and from altering settings when in the locked position.

HOME G28 button – Pressing the HOME G28 button will return all axes to machine zero. To send just one axis to machine home, enter the axis letter and press Home G28.

To zero out all axes on the POS-TO-GO display, while in handle jog, press any other operation mode (Edit, Mem, MDI, etc.) then back to Handle Jog.

Each axis can be zeroed independently to show a position relative to the selected zero. To do this go to the POS-OPER page, enter handle jog mode, position the axes to the desired position and press Origin to zero that display. In addition a number can be entered for the axis position display. To do this, enter an axis and number, for example, X2.125 then Origin.



Tool Life – Within the Current Commands page there is a tool life (usage) monitor. This register counts each time the tool is used. The tool life monitor will stop the machine once the tool reaches the value in the alarms column.

Tool overload – Tool load can be defined by the Tool Load monitor, this will change normal machine operation if it reaches the tool load defined for that tool. When a tool overload condition is encountered four actions can be set by Setting 84.

Alarm – Generate an alarm

Feedhold – Stop the feed

Beep – Sounds an audible alarm

Autofeed – Automatically increase or decrease the feed rate

You can verify the exact spindle speed by checking the Curnt Comds “Act” display. Live tooling spindle axis RPM is also displayed on this page.

You can select an axis for jogging by entering that axis name on the input line and pressing the Handle Jog button.

The Help display has all the G and M codes listed. To get to them quickly, press the Help button and then the C button.

The Jogging speeds of 100, 10, 1.0 and 0.1 inches per second can be adjusted by the Feed Rate Override buttons. This gives an additional 10% to 200% control.

Advanced Editor

The advanced editor allows the operator to select several programs (using the INSERT key) and will send them all to the RS-232 port.

To scroll line by line through the program press F2 key, then use the jog handle . To stop the handle jog scrolling and remain at the position in the program press Undo.

Calculator

The number in the calculator box can be transferred to the data entry line by pressing F3 in Edit or MDI mode. This will transfer the number from the calculator box to the Edit or MDI input buffer (Enter a letter, X,Z etc. for the command to use with the number from the calculator).

The highlighted Trig, Circular, or Milling data can be transferred to load, add, subtract, multiply or divide in the calculator by selecting the value and pressing F4.

Simple expressions can be entered into the calculator. For example $23*4-5.2+6/2$, it will be evaluated when the WRITE key is pressed and the result (89.8) in this case displayed in the calculator box.



AUXILIARY AXIS CONTROL

Besides the directly controlled axes in this control, an additional external positioning axis may be added. This axis, V, may be commanded directly from the program. Commands to this axis are only allowed in a G00 or G01 block. Connection of these axes is done through the second RS-232 port to a HAAS single axis control. Setting 38 is used to select the number of auxiliary axes (0 or 1). The machine position display will show the present position of this axis.

There are no work offsets for this axis, 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 axis control will also be initialized. To set a displaced zero, you must jog the single axis control to a new zero position and then press and hold the Clear key on the single axis control.

If a feed (G01) is programmed, the feed rate programmed in the CNC is sent to the auxiliary control without any changes. For a V-axis feed at F30.0, this means that the V-axis will move at 30 degrees per second. A G00 motion will move the axis at its maximum feed rate.

The Feed Hold and Reset buttons will not stop the auxiliary axis. Emergency Stop and Single Block will stop an auxiliary axis. When the CNC control is waiting for an auxiliary axis motion to complete, the bottom of the screen will display "V FIN". The Reset button will terminate any "hung-up" auxiliary axis communication.

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 CNC to the servo control.



TAILSTOCK

This option is not field installable

The optional Tailstock is a hydraulically actuated cast iron member which runs along two linear guides. The 20 inches (33 1/2 inches on SL-30, 44 inches on the SL-40) of travel allows a long part to be machined. Tailstock motion is controlled in one of 3 ways: 1) Through program code, 2) In jog mode, 3) By a foot switch.

Also see "SL-10 Tailstock Operation" at the end of this section

The tailstock is designed to travel to position at 2 rates: 1) High pressure is called "rapid" and can be programmed with G00. 2) Low pressure is called "feed" and can be programmed with G01. It is used to hold the part. An F code is required for feed mode (even if previously invoked) but it does not affect the actual feed rate.

Recommended hydraulic tailstock operating pressure is 120 psi. Operating pressures below 120 psi may cause the tailstock to function unreliably.

CAUTION

- If tailstock hydraulic pressure is set lower than 120 psi, it may not function reliably.
- It is important to verify tailstock and turret clearance before operating machine or serious damage could occur. Adjust Setting 93 and Setting 94 as necessary.
- Feed Hold will NOT stop the hydraulic tailstock.

Setting a Restricted Zone for the Tailstock

Setting 93 (TAIL ST. X Clearance) and Setting 94 (Z/TS DIFF @X Clearance) are used to ensure that the tailstock does not collide with the turret or any tools in the turret. The restriction zone is an rectangular area in the lower right hand area of the lathe's work space. The restriction zone will change so that the Z axis and tailstock maintain a proper distance from each other when below a specified X axis clearance plane. Setting 93 specifies the clearance plane and Setting 94 specifies the Z and B axis (tailstock axis) separation to maintain. If a programmed motion crosses the tailstock protected area, an alarm is generated. Keep in mind that a restricted zone is not always desired (e.g. when setting up). To cancel a enter a 0 in Setting 94 and maximum X machine travel in setting 93.

Setting a Value for the X Clearance Plane:

1. Place the control in MDI mode.
2. Select the longest tool in the turret (the tool that protrudes furthest on the X-axis plane).
3. Place the control in jog mode.
4. Select the X axis for jogging and move the X axis clear of the tailstock.
5. Select the tailstock (B axis) for jogging and move the tailstock beneath the selected tool.
6. Select the X axis and approach the tailstock until the tool and tailstock are about 0.25 inches apart.
7. Find the X axis "machine" position on the display, enter this value for Setting 93. Back away from the tool in X a small amount before entering the value in Setting 93.

Setting a Separation for Z and B Axis Below the X Clearance Plane:

1. Place the control in Zero RET and Home G28 all axes.
2. Select the X axis and move the turret in front of the tailstock center tip.
3. Move the Z axis so that the rear of the tool turret is within about 0.25 inches of the tailstock tip.
4. Find the Z axis "machine" position on the display, enter this value for Setting 94.

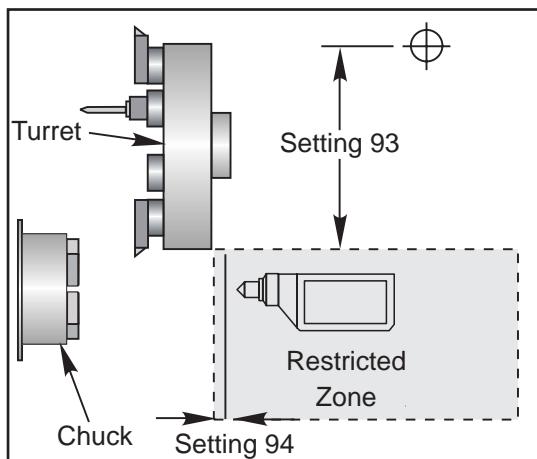
A properly defined restricted zone will handle most, but not all, conflicts between the tool turret and the tailstock.



TAILSTOCK SETTINGS

Default values for these settings, as shipped from the factory, will prevent the tailstock from running into the tool turret, provided the tool turret is empty. You will need to change the protection settings for any job you perform to avoid turret collisions based on tooling and part size. It is recommended that you test the limits after changing these settings.

Two settings are provided for protecting the tailstock. Setting 93 (Tail St. X Clearance) and Setting 94 (Z/TS Diff @X Clearance). These settings, when set correctly, will stop any motion that would cause the tailstock to hit the tool turret. The following figures illustrates these settings 94, 94 105, 106, and 107; see the Settings chapter for more information.



Tailstock Restricted Zone

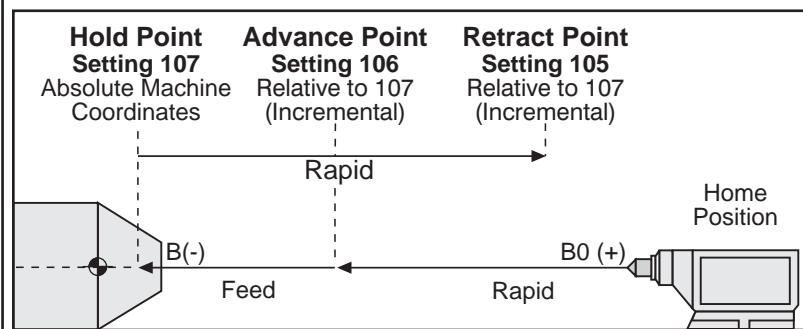


Diagram of Settings 105, 106, and 107.

Setting 93 is the X axis machine clearance plane that the X axis can not move below when the difference between the Z and B axes positions is less than setting 94. When the Z and B axis location difference is greater than Setting 94, the X axis is allowed to move to its travel limit. As long as the proper Z and B axis distance is maintained the X axis can move to its full travel. Likewise, if the X axis is at its full travel, or below the clearance plane designated by setting 93; then it is not possible to reduce the Z and B axis difference below Setting 94.

TAILSTOCK FOOT PEDAL OPERATION

Pressing the tailstock foot pedal will command either an M21 or M22, depending on current position. In other words, if the tailstock is left of the retract point, pressing the foot pedal will move the tailstock towards the retract point (M22). If tailstock is right of the retract point, the foot pedal will also move the tailstock towards the retract point (M22). If tailstock is at retract point, pressing the foot pedal will move the tailstock towards the hold point (M21).

If the foot pedal is pressed while tailstock is moving, the tailstock will stop and a new sequence must begin.

JOGGING THE TAILSTOCK

In Jog mode the keys, "TS <—" and TS "—>" are used to jog the tailstock at low pressure (feed). By selecting TS Rapid and pressing TS <— or TS —> buttons the tailstock will move at the rapid speed. The control reverts to the last jogged axis when the buttons are released



ALARMS / MESSAGES

If a part is being held and tailstock motion is detected an alarm is generated. This will stop the program and turn off the spindle.

This alarm is also generated if the tailstock reaches the hold point during a low pressure feed, indicating that the part has fallen out.

SL-10 TAILSTOCK OPERATION

The optional Haas tailstock for the SL-10 is a hydraulically actuated quill which moves within a manually positioned headstock. The tailstock is manually positioned and held in place with the lock-lever. Tailstock motion is controlled in one of 3 ways: 1)Through program code, 2) In jog mode, 3) By a foot switch.

Notes: The SL-10 tailstock consists of a fixed head, and a moveable center rod. Therefore, the only moving part is called the tailstock center.

Pressing POWERUP/RESTART or AUTOALLAXES will **not** cause the tailstock center to physically move. It is the operator's responsibility to move it out of the way to avoid a collision. Tailstock center movement using the jog handle and remote jog handle is unavailable.

The tailstock center is always considered to be at zero; as the control will not know where the tailstock center is.

WARNING

- Retract tailstock fully and set pressure to zero when not in use. The SL-10 tailstock quill does not have a feedback device; therefore the control does not track the position of the tailstock body and quill. The tailstock can drift from its retracted position even when the solenoid valve is not activated.
- It is important to verify tailstock and turret clearance before operating machine or serious damage could occur. Adjust setting 93 TAIL ST. X CLEARANCE and setting 94 Z/TS DIFF @ X CLEARNCE as necessary.
- **Feed Hold will NOT stop the hydraulic tailstock.** The Emergency Stop button is the only way to stop the tailstock

TAILSTOCK PROGRAMMING

M21 will cause the tailstock quill to extend toward the spindle.

M22 will cause the tailstock quill to retract away from the spindle.

When an M21 is commanded, the tailstock center will be commanded to move towards the spindle and maintain continuous pressure. **Note that the program will not wait while this is completed;** instead, the next block will be executed immediately. A dwell should be commanded to allow the tailstock center movement to complete, or the program should be run in Single Block mode. When an M22 is commanded, the tailstock center will move away from the spindle, and then stop.

Note: The tailstock may creep out when not in use. If the tailstock is not used during an operation, either turn the tailstock pressure valve to zero, or adjust the time the retract pressure is applied.

Caution: Do not use a M21 in the program if the tailstock is positioned manually. If this is done the tailstock will back away from the part and then reposition against the part, which may cause the workpiece to drop.



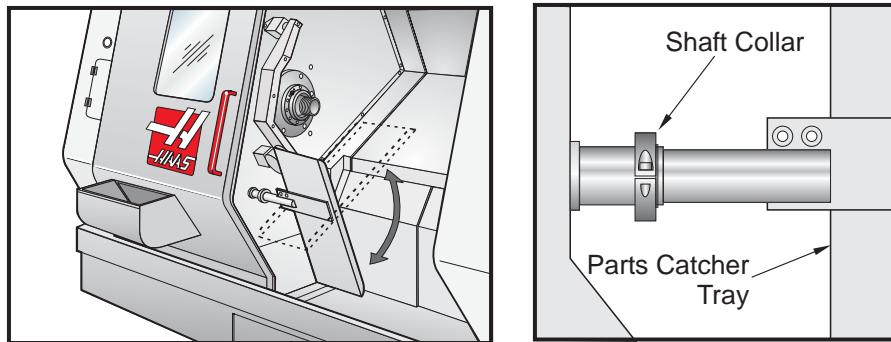
PARTS CATCHER

The parts catcher is an automatic part retrieval system designed to work with bar feed applications. The parts catcher, is commanded using M Codes (M36 to activate and 37 to deactivate). The parts catcher rotates to catch finished parts and directs them into the bin mounted on the front door.

OPERATION

The parts catcher must be properly aligned before operation.

1. Power on the machine. In MDI mode, activate the parts catcher (M36).
2. Loosen the screw in the shaft collar on the outer parts catcher shaft.



SL-20 Shaft Collar Shown

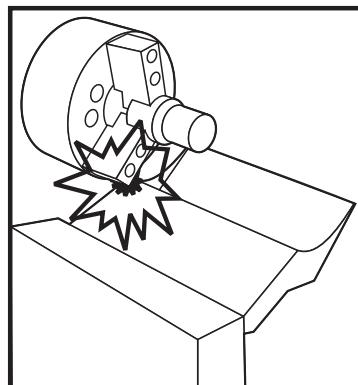
3. Slide parts catcher tray into the shaft far enough to catch the part and clear the chuck. Rotate the tray to open the sliding cover of the parts collector mounted in the door and tighten the shaft collar on the part catcher shaft. **WARNING:** Check the "Z" axis, "X" axis, tool and turret position during part catcher actuation to avoid possible collisions during operation.

NOTE: The operator door must be closed when actuating the part catcher.

4. When programming the part catcher in a program you must use a G04 code between M53 and M63 to pause the catcher pan in the open position long enough to cut off the part and allow it to fall into the collector.

SL-10 Warning

Large chuck jaws may interfere with the operation of the parts catcher. Be sure to check the clearances before operating the parts catcher.





TOOL PRE-SETTER

The Tool Pre-Setter allows an operator to quickly set up his machine with the necessary tool and work offsets.

OPERATION

The Tool Pre-Setter is raised and lowered manually. Once the arm is lowered, the turret can be jogged using the jog handle or jog buttons. The Cycle Start button is disabled to prevent the operator from accidentally running a program and crashing into the arm. As a safety feature the jog rate defaults to a maximum of .01 and the Rapid button is disabled.

Once the probe is touched, the control will beep and not allow the operator to continue jogging in that direction. This prevents the operator from inadvertently damaging the probe and ensures greater accuracy (The probe can be pressed momentarily by hand to test the audible beep).

Important! Automatic storing of the machine's position can only be performed when the jog buttons are used.

WARNING

When changing tools, always back the tool a safe distance away from the probe to avoid crashing the tool into the arm

SETTING TOOL GEOMETRY AND TOOL SHIFT OFFSETS USING THE PROBE

1. Setting 33 controls whether the current tool offsets obtained while using the tool setter are stored in Tool Geometry (Fanuc) or Tool Shift (Yasnac).
2. Call the tool to probe.
3. Jog the tool to a safe position and lower the arm.

Touching off I.D. or O.D. Tools

4. In .001 inch mode, jog the turret in the X direction until the tool tip touches the probe.

NOTE: Once the tool tip touches the probe, the control will beep and not allow the operator to continue jogging in that direction.

NOTE: When retouching a tool, setting 64 needs to be off to ignore the value on G54.

Important! Automatic storing of the machine's position can only be performed when the jog buttons are used. The jog handle can also be used, however, those values will have to be manually entered into the control.

5. Jog the tool in the Z- direction until it touches the probe. That value is then stored in the Offsets page.

Touching off Drills, Taps or Center Cutting Tools

6. Call the tool.
7. Press the F2 key. This function converts the value in Parameter 254 (Spindle Center) into programmed units (inches/millimeters) and stores it in the selected X axis tool offset.
8. In .001 inch mode, jog the tool in the Z-direction until it touches the probe. That value is then stored in the selected Z axis tool offset.

SETTING WORK ZERO OFFSETS

Before running your program, the machines Work Zero Offsets (G52-129) must be entered.

1. In the Offsets page select the desired work offset.
2. Call up the desired tool and touch off the face of the part.
3. Press Z FACE MESUR; this will reference the rest of the tools to the face of the part.



INTUITIVE PROGRAMMING SYSTEM (IPS)

This optional software feature 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.

MACHINE POSITION

The upper left hand display, shows the coordinates of the machine. There are four coordinate modes, which can be scrolled through using the Page Up and Page Down buttons. These are:

Operator Display – This display is for the operator/setup person to use as desired, and is not used by the control for any positioning functions. In JOG mode, and with this display selected (Operator), select an axis (press X- or X+ for the X-axis or Z- or Z+ for the Z-axis). Then press the ORIGIN button to set the display to zero. This display will then show position relative to this newly reset zero position.

Work Display – This display shows how far the tool is away from the **X** and **Z** zero of the programmed part. On power up, it will display the value in work offset G54 automatically. The machine uses this coordinate system to run the part.

Machine Display – This display is the machine coordinate system that is automatically set on power up and the first ZERO RET. It cannot be changed by the operator or any work coordinate systems, and will always show the distance from machine zero.

Distance To Go – This display is an incremental display that shows the travel distance remaining before the axes stop, during a programmed movement.

TOOL OFFSETS

Tool Offset

Tool offsets are described in detail in this manual within the “Tool Nose Compensation” section. This includes specific instructions on Radius, Radius Wear, Taper, and Tip.

Tool – The current tool number.

X Offset – The X axis offset for the current tool.

X Wear – The amount of tool wear, in the X axis for the current tool.

Z Offset – The Z axis offset for the current tool.

Z Wear – The amount of tool wear, in the Z axis for the current tool.

Radius** – The tip radius of the current tool.

Radius Wear – The amount of wear in the radius.

Taper – Compensation value for part deflection.

Tip** – Tool tip direction will be a value of 0-9.

Note: Tool offsets are required for running full G-code programs; they are not required by any of the single feature part programs.

**Must be entered to use Cutter Compensation; See the Cutter Compensation section.



AUTOMATIC MODE

Tool offsets must be set before an automatic operation can be run. Enter the values for each tool used on the Tool Offsets 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.

Values entered must be measured from the centerline of the spindle.

The following is an example of an Automatic Mode screen and the definitions of the variables that will need to be entered.

TOOL NUMBER 1	DIA. TO CUT 0.0000	MAX RPM 0000
WORK OFFSET 54	Z DIMENSION 0.0000	SFM 000
Z START POINT 0.0000	DEPTH OF CUT 0.0000	
OUTSIDE DIA. 0.0000	FEED PER REV 0.0000	

RAPID FEED OD TURN ID TURN FACE

Position (0,0)

Turn and Face - OD Turn

Tool Number – Tool to use. Note: Tool offsets must be set before an automatic operation is started.

Work Offset – Work offset to use.

Z Start Pt – Enter the Z-axis starting point. New starting points can be generated by entering a positive or negative value. This value will shift the starting point the amount in Z Start Point from the work offset.

Outside Dia. – Enter the current diameter of the workpiece. Manually measure the diameter.

Diameter to Cut – Enter the finished diameter.

Z Dimension – Enter the Z-axis dimension of the part from the Z start point.

Depth of Cut – Enter the depth of cut for each pass of the stock removal.

Feed Per Rev – Enter the feed per revolution. This is the distance the tool will move for each revolution of the spindle.

Max RPM – Enter the maximum spindle turning speed.

SFM – Enter the surface feed per minute.

Advanced Users

Some operations (for example, Threading) may need additional settings modified to create the required groove. These setting numbers are: 22, 28, 72, 73, 86, 95, 96, 99. See the Settings chapter.



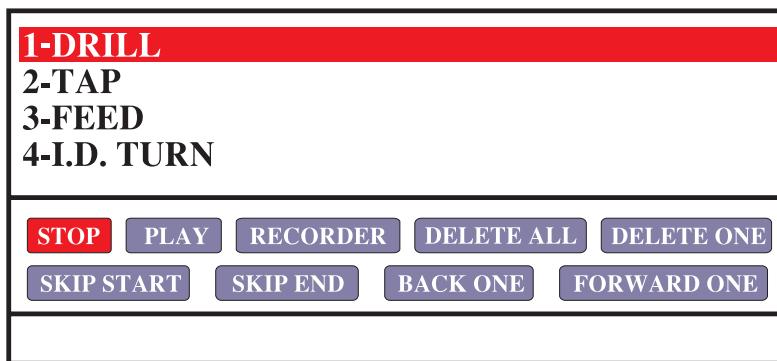
SYSTEM MODE

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.



Sample after features have been recorded

These commands are:

Stop – Stops the recording.

Play – Starts the playback, starting with the highlighted process. The lathe 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.

The operator can scroll though the commands using the up and down arrow keys.

Operation – Enter the IPS menus, by pressing “Handle Jog”. Use the left and right arrows key 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.

Note: This feature will not record Semi-Automatic moves.



Set up the machining process, enter the values and press "Cycle Start". The lathe 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.

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 lathe 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.

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 part program in memory by placing the cursor at the top line, type Onnnnnn (the letter "O" followed by a program number), then press the "Alter" button.

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, are 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.

OTHER SYSTEM TABS

The "Alarms" tab, displays any current alarms. If an alarm is displayed, correct the problem press Reset and the lathe will continue.

The "Alarm History" tab will display the previous alarms. The arrow keys are used to scroll through the alarms.

The "Alarm Viewer" tab allows the user to enter an alarm number and once "Enter" is pressed the control will give the definition.

The "Messages" tab allows the user to leave a message to the next operator, or reminders for themselves. Messages are entered at the lower left of the screen. Press Enter to set the text in the message window. Use the arrow keys to navigate between the lines. The Delete key will delete the entire line the cursor is on.



ADVANCED EDITOR

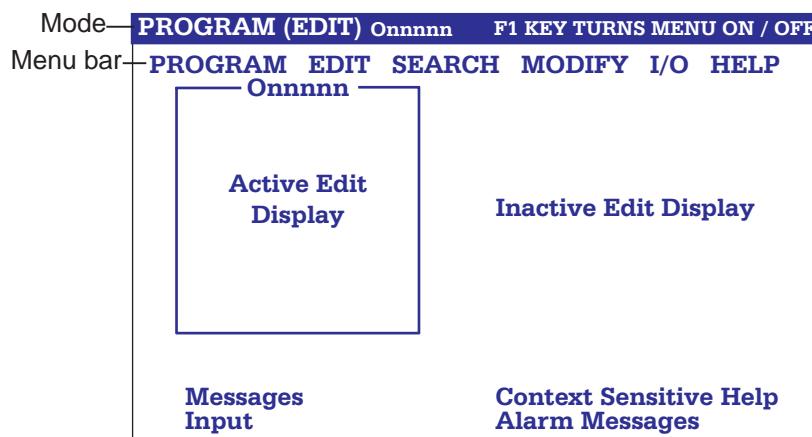
The HAAS Advanced Editor gives the user the ability to edit programs using pull-down menus.

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 rennumbers 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, 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).

THE I/O MENU

Send RS-232

This menu 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 cursor to that selection.

Hot Keys

Description

SELECT PROG

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

F2

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.

EDIT

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

F4

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.

INSERT

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

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

DELETE

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

UNDO

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

SEND

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

RECV

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

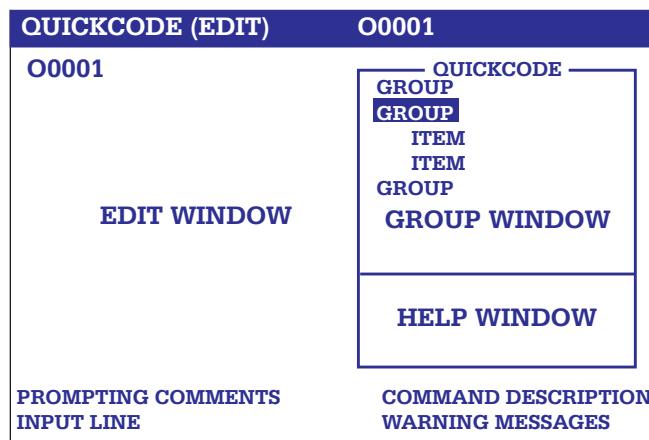
ERASE PROG

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, commands 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

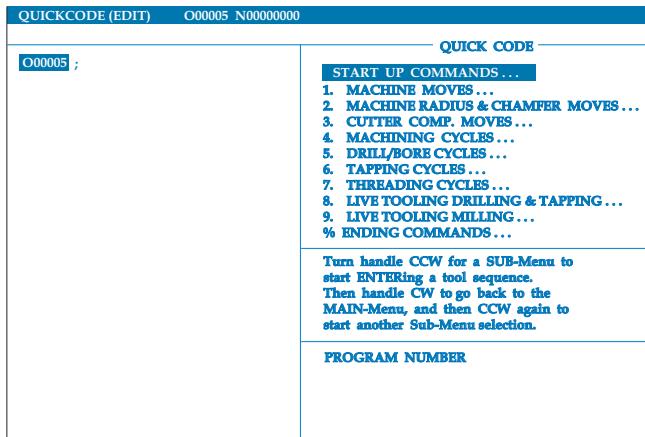
The following illustrates how Quick Code can be used to build a program. A program will be built to remove stock and thread the end of a part. We will assume that tool 101 is a stock removal tool and tool 202 is a threading tool. Verify that Quick Code source program, O9999, is one of the programs in the control, before starting.

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



Create A Program

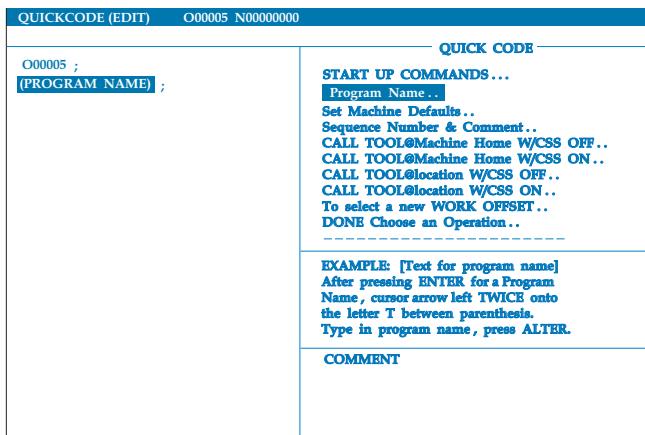
Quick Code will not generate the new program number for you. To create a program, press List Prog, enter a program number (e.g. O00005) and press Write/Enter. Or select a program to edit. In order to edit the program press the Edit button and then press the Prgrm/Convrsl key twice to enter Quick Code (**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 Start Up Commands will appear and the first, "Program Name" is highlighted.
3. Press the Write key. This will enter in a (T), for you to cursor arrow left twice onto the "T" 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 101

- While on the “Start Up Commands” menu, turn the jog handle CCW to highlight the group item titled “Call Tool@Location W/CSS On”.
- Press the Write button to have the control ask you for a tool number for your program, and the control asks for information for the program.

QUICKCODE (EDIT)	O00005 N00000000	QUICK CODE
(PROGRAM NAME ; M09 ; G53 G00 X-2. Z-3. T0 ; T101 (T) ; G50 S2500 ; G97 S1250 M03 ; G54 G00 X0 Z1. M08 ; G96 S320 ;	START UP COMMANDS ... Program Name .. Set Machine Defaults .. Sequence Number & Comment .. CALL TOOL@Machine Home W/CSS OFF .. CALL TOOL@Machine Home W/CSS ON .. CALL TOOL@location W/CSS OFF .. CALL TOOL@location W/CSS ON .. To select a new WORK OFFSET .. DONE Choose an Operation ..	

Programmed with the Start Up Command Selections Entered in with Quick Code for Tool 101.

Use the G71 Stock Removal Cycle Program

- Scroll and highlight the group titled “4. Machining Cycles”.
- Scroll CCW two clicks. “G71 Stock Removal Cycle Program” will be highlighted.
- Press the Write button to start the prompts.

The program will look like this:

QUICKCODE (EDIT)	O00005 N00000000	QUICK CODE
M09 ; G28 ; T101 (T) ; G97 S1250 M03 ; G54 G00 X0 Z1. M08 ; G82 Z-0.2 P0.2 R0.1 F0.003 ;	START UP COMMANDS ... 1. MACHINE MOVES ... 2. MACHINE RADIUS & CHAMFER MOVES ... 3. CUTTER COMP. MOVES ... 4. MACHINING CYCLES ... Face End of Part Program. G71 Stock Removal Cycle Program. FINISHING Cycle G70 command.. O.D./I.D. Cycle G71 command.. END FACE Cycle G72 command.. IRREGULAR Cycle G73 command.	

G71 Stock Removal Program

Call Tool 202

- Scroll CW to the group titled “Start Up Commands” then scroll CCW and highlight the group item titled “Call Tool@Location W/CSS On”. Press the Write button and the control will ask for the information for tool 202; enter in the number 202 when prompted for tool number.



Call the Threading Cycle 3/4-16 O.D. G76 Threading Cycle

Scroll CW and highlight the group titled "7. Threading Cycle". Scroll CCW until "3/4-16 O.D. G76 Threading Cycle" is highlighted. Press the Write button and the control will ask for the information to thread with G76.

Scroll CW and highlight the group titled "Ending Commands". Scroll CCW until the "Coolant Off M09" is highlighted and press Write.

Sending the Machine Home and Ending the Program

1. Scroll CW and highlight the group titled "Ending Commands". Scroll CCW until "Home X, Z and Tailstock with G28" is highlighted and press Write.

2. Scroll CCW and highlight the group titled "End Program, Program End Command M30" and press Write.

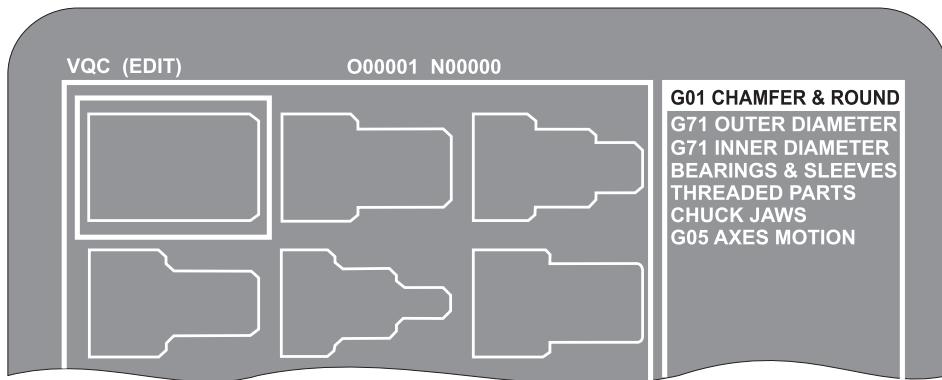
The program is ready to run. Run the program in Graphics mode to make certain that all the steps are entered.

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.



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 asks 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 program 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.



TOOL NOSE CUTTER COMPENSATION USING G112

Tool Nose 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. Cutter radius compensation is only available in the X and Y planes (G17) within Cartesian mode (G112).

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.

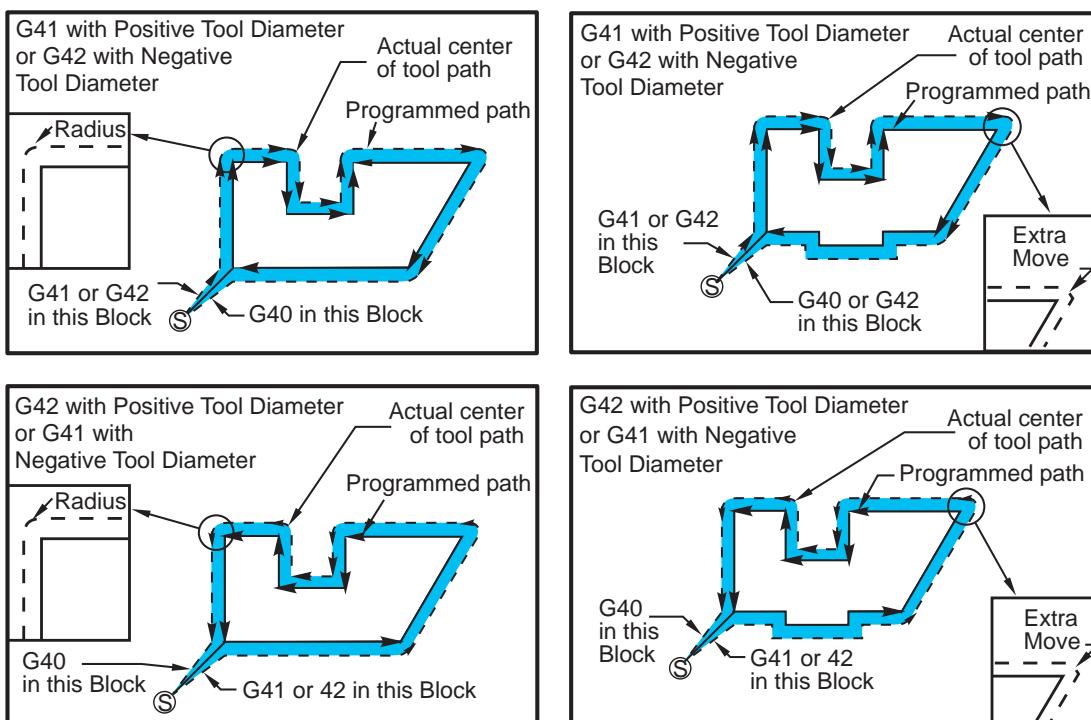
A **Dnnn** must also be programmed with G41 or G42 to select the correct offset number from the radius/diameter offset column. Offset values entered for the radius/diameter should be in positive numbers. If the offset contains a negative value, cutter compensation will operate as though the opposite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



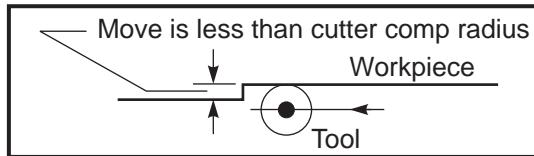


ENTRY AND EXIT FROM CUTTER COMPENSATION

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, type A or 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

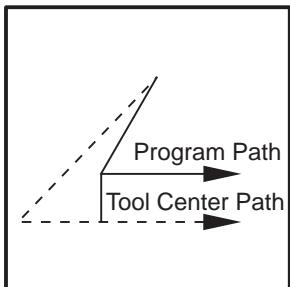
Note that a small cut of less than tool radius and at a right angle to the previous motion will only work with the Fanuc setting. A cutter compensation alarm will be generated if the machine is set to the Yasnac setting.

FEED ADJUSTMENTS IN CUTTER COMPENSATION

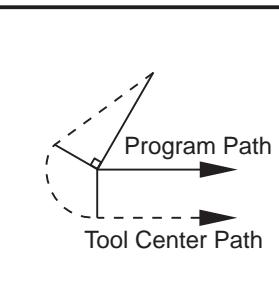
When using cutter compensation in circular moves, there is the possibility of speed adjustments to what has been programmed. If the intended finish cut is on the inside of a circular motion, the tool should be slowed down to ensure that the surface feed does not exceed what was intended.

Cutter Compensation Entry (YASNAC)

Type A

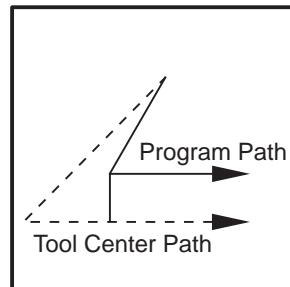


Type B

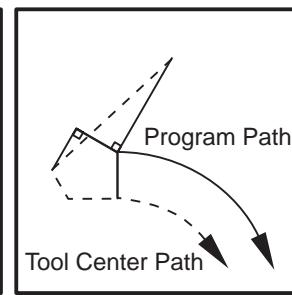
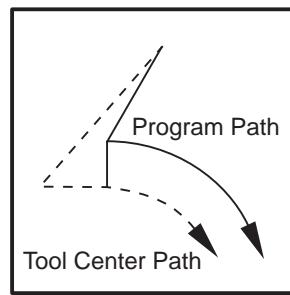
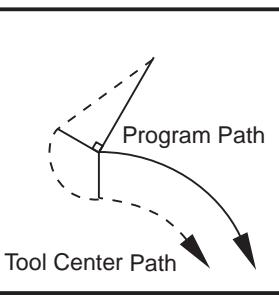
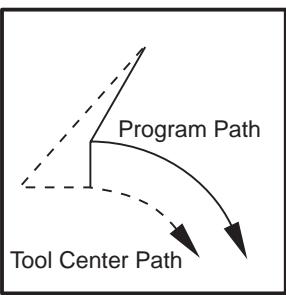
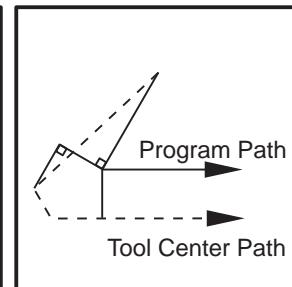


Cutter Compensation Entry (Fanuc style)

Type A



Type B





MACROS (OPTIONAL)

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.

- **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. Family of parts
2. Soft jaw machining
3. User defined "canned" cycles (such as custom grooving cycles)

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.

Probing Probing enhances the capabilities of the machine in many ways. Below is just a hint of the possibilities.

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.

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

Look ahead 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.

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 called macro subroutine.

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	B	C	D	E	F	G	H	I	J	K	L	M
	1	2	3	7	8	9	-	11	4	5	6	-	13
Address: Variable:	N	O	P	Q	R	S	T	U	V	W	X	Y	Z
	-	-	-	17	18	19	20	21	22	23	24	25	26



Alternate Alphabetic Addressing

Address:	A	B	C	I	J	K	I	J	K	I	J
Variable:	1	2	3	4	5	6	7	8	9	10	11
Address:	K	I	J	K	I	J	K	I	J	K	I
Variable:	12	13	14	15	16	17	18	19	20	21	22
Address:	J	K	I	J	K	I	J	K	I	J	K
Variable:	23	24	25	26	27	28	29	30	31	32	33

Arguments accept any floating point value to four decimal places. If the control is in metric, it will assume thousandths (.000). In Example 2 below, 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:	A	B	C	D	E	F	G	H	I	J	K	L	M
Variable:	.001	.001	.001	1.	1.	1.	-	1.	.0001	.0001	.0001	1.	1.
Address:	N	O	P	Q	R	S	T	U	V	W	X	Y	Z
Variable:	-	-	-	.0001	.0001	1.	1.	.0001	.0001	.0001	.0001	.0001	.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 2: G65 P2000 I1 J2 K3 I4 J5 K6 ;

The letters G, L, N, O and P cannot be used to pass parameters to a macro subroutine.

Macro Variables

There are three categories of macro variables: *system* variables, *global* variables, and *local* variables.

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.



Local Variables and Corresponding Address

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	I	J	J
Variable:	12	13	14	15	16	17	18	19	20	21	22
Address:	M					Q	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							

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-#599	General purpose variables saved on power off
#600-#699	General purpose variables saved on power off
#700-#749	Hidden variables for internal use only
#750-#751	Serial port #2 data collection
#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
#2001-#2099	X axis tool shift offsets
#2101-#2199	Z axis tool shift offsets
#2201-#2299	Tool nose radius offsets
#2301-#2399	Tool tip direction
#2401-#2499	Tool diameter/radius offsets
#2601-#2699	Tool diameter/radius wear
#2701-#2799	X axis tool wear offsets
#2801-#2899	Z axis tool wear offsets
#2901-#2999	Tool nose radius wear offsets
#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 (read only)
#3022	Feed timer (read only)
#3023	Present part timer (read only)
#3024	Last complete part timer (read only)
#3025	Previous part timer (read only)
#3026	Tool in spindle (read only)
#3027	Spindle RPM (read only)
#3030	Single Block (read only)
#3031	Dry Run (read only)
#3032	Block Delete (read only)
#3033	Opt Stop (read only)
#3901	M30 count 1
#3902	M30 count 2
#4001-#4020	Previous block 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.

#5000-#5006	Previous block end position
#5020-#5027	Present machine coordinate position
#5041-#5046	Present work coordinate position
#5061-#5065	Present skip signal position - X,Z,C
#5081-#5086	Present tool offset
#5201-#5206	Common offset
#5221-#5226	G54 work offsets
#5241-#5246	G55 work offsets
#5261-#5266	G56 work offsets
#5281-#5286	G57 work offsets
#5301-#5306	G58 work offsets
#5321-#5326	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
#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
#14661-#14666	G154 P29 additional work offsets
#14581-#14686	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 now 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

Use the following macro variables to read or set the following geometry, shift or wear offset values:

#2001-#2050	X-axis geometry/shift offset
#2101-#2150	Z-axis geometry/shift offset
#2201-#2250	Tool nose radius geometry
#2301-#2350	Tool tip direction
#2701-#2750	X-axis tool wear
#2801-#2850	Z-axis tool wear
#2901-#2950	Tool nose radius wear

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 Z0;  
G81 R0.2 Z-0.1 F20 L0;  
S2000 M03;  
#3003=0;  
T02 M06;  
G83 R0.2 Z-1 F10. L0;  
X0. Z0.;
```

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 STOP 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	#5023 A-axis	#5026 U-axis
#5021 Y-axis	#5024 B-axis	#5027 V-axis (used for the Haas Bar Feeder)
#5022 Z-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-#5086 Tool Length Compensation

The current total tool length compensation that is being applied to the tool is returned. This includes tool geometry referenced by the current modal value set in the T code plus the wear value.

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-#5206	G52 X, Z, Y, A, B , C OFFSET VALUES
#5221-#5226	G54 " " " " " "
#5241-#5246	G55 " " " " " "
#5261-#5266	G56 " " " " " "
#5281-#5286	G57 " " " " " "
#5301-#5306	G58 " " " " " "
#5321-#5326	G59 " " " " " "
#7001-#7006	G110 X, Z, Y, A, B, C OFFSET VALUES
#7021-#7026	" " " " " "
#7381-#7386	G129 X, Z, Y, A, B, C OFFSET VALUES



Variable Usage

All variables are referenced with a number 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 X1.0;

the variables can be set to the following values:

#7=0;

#1=1.0;

and the block replaced by:

N1 G#7 X#1;

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 Z3.7 F.02 ;

sets addresses G, X, Z and F to 1, 1.5, 3.7 and 0.02 respectively and thus instructs the control to move linearly, G01, to position X=1.5 Z=3.7 at a feed rate of 0.02 inches per revolution. Macro syntax allows the address value 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=0.02;

G#1 X[#1+#2] Z#3 F#4 ;

The permissible syntax on addresses A..Z (exclude N or O) is as follows:

<address><-><variable>	A-#101
<address>[<expression>]	Z[#5041+3.5]
<address><->[<expression>]	Z-[SIN[#1]]

If the value of the variable does not agree with the range of the address, then the usual control alarm will result. For instance, the following code would result in an invalid G code alarm because there is no G143 code:

#1=143;

G#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 G1 X#1 would move the machine tool to .1235 on the X axis. If the control is in the metric mode, the tool 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 Z#1 ;

becomes

G00 X1.0.

No Z 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]. Functions can be passed any expression as arguments. Functions return floating point decimal values. The function provided with 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 addresses significant precision. For *metric* and *angle* dimensions, three-place precision is the default. For *inch*, four-place precision is the default. Integral addresses such as T are rounded normally.



```
#1= 1.00333 ;
G0 X[ #1 + #1 ] ;
(X moves to 2.0067) ;
G0 X[ ROUND[ #1 ] + ROUND[ #1 ] ] ;
(X moves to 2.0066) ;
G0 C[ #1 + #1 ] ;
(Axis moves to 2.007) ;
G0 C[ ROUND[ #1 ] + ROUND[ #1 ] ] ;
(Axis moves to 2.006) ;
```

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 the usual 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 #20000 0011
#1=5.0;
#2=3.0;
IF [[#1 GT 3.0] AND [#2 LT 10]] GOTO1
```

Here the variable #3 will contain 3.0 after the OR operation.
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:

Example

```
IF [#1 EQ 0.0] GOTO100;  
WHILE [#101 LT 10] DO1;  
#1=[1.0 LT 5.0];  
IF [#1 AND #2 EQ #3] GOTO1
```

Explanation

Jump to block 100 if value in variable #1 equals 0.0.
While variable #101 is less than 10 repeat loop DO1..END1.
Variable #1 is set to 1.0 (TRUE).
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 an 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          (Initialize an array of variables) ;
N1 IF [#2 NE #0] GOTO2   (B=base variable) ;
#3000=1           (Base variable not given) ;
N2 IF [#19 NE #0] GOTO3  (S=size of array);
#3000=2           (Size of array not given) ;
N3 WHILE [#19 GT 0] DO1 ;
#19=#19-1         (Decrement count) ;
#[#2+#19]=#22      (V=value to set array to) ;
END1 ;
M99 ;
```

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 G-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>] GOTO



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] GOTO5;
```

could also be:

```
IF [#1] GOTO5;
```

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 Z0 [#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 GOTO 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 Z#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 nest is a loop within a loop

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 therefore 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, for example:

```
DO1;  
<statements>  
END1;
```

Executes until the RESET key is pressed.

CAUTION! The following code can be confusing: WH [#1] D01;
END1;

In the above 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 4 times, which means, program 1 can call program 2, program 2 can 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)

Aliasing

Aliasing is a means of assigning a G code to a G65 P##### sequence. For example:

G65 P9010 X.5 Z.05 F.01 T1;

Can be written as:

G06 X.5 Z.05 F.01 T1;

Here, we have substituted an unused G code, G06, for G65 P9010. In order for the above block to work we must set the parameter associated with subroutine 9010 to 06 (parameter91). Note that G00 and G65 can not 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 (zero) 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 have 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 X [COS[90]] Z3.0; 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 number.

G1 X 0 Z3.0; WRONG

The above block will result in an alarm at runtime. The correct form looks as follows:

G1 X0 Z3.0; 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 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 call in every 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 to ceiling
LN []	Natural Logarithm
EXP []	Base E exponentiation
ADP []	Re-scale VAR to whole number
BPRNT []	

The following can be used as an alternative methods for achieving the same results for a few unimplemented 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.

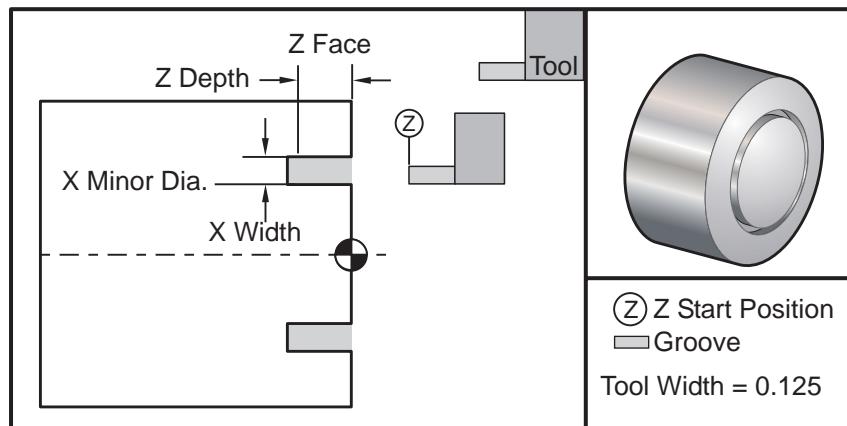
A block search is made starting from the current block being interpreted. When the end of the program is reached, searching continues from the top of the program until the current block is encountered.



EXAMPLE PROGRAM USING MACROS

The following example will cut a face groove in a part using easily-edited variables.

```
%  
O0010                               (MACRO G74)  
G50 S2000  
G97 S1000 M03 T100  
G00 T101  
#24 = 1.3                         (X MINOR DIAMETER)  
#26 = 0.14                          (Z DEPTH)  
#23 = 0.275                         (X GROOVE WIDTH)  
#20 = 0.125                         (TOOL WIDTH)  
#22 = -0.95                         (Z START POSITION)  
#6 = -1.                            (ACTUAL Z FACE)  
#9 = 0.003                          (FEED RATE IPR)  
G00 X [ #24 + [ #23 * 2 ] - [ #20 * 2 ] ] Z#126  
G74 U - [ [ #23 - #20 ] * 2 ] W - [ #26 + ABS [ #6 - #22 ] ] K [ #20 * 0.75 ] I [ #20 * 0.9 ] F#9  
G00 X0 Z0 T100  
M30  
%
```







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 which can involve the optional live tooling and C-axis.

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, Z move like this, X-2. 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 **nonmodal** G-code is only affects the line it is in; the program line after will not be affected by the previous lines nonmodal G-code. **The Group 00 codes are non-modal; the other groups are modal.**

Programming Notes

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. Canned cycles are defined for most common Z-axis repetitive operations, such as drilling, tapping, and boring. Once selected, a canned cycle is active until canceled with G80. When active, the canned cycle is executed every time an X-axis motion is programmed. X-axis motions are executed as rapid commands (G00) and the canned cycle operation is performed after the X-axis motion.

Using Canned Cycles

Modal canned cycles remain in effect after they are defined and are executed in the Z-axis, for each position of the X-axis. Note that X-axis positioning moves, during a canned cycle will be rapid moves.

The operation of a canned cycle will vary according to whether incremental (U,W) or absolute (X,Z) axis moves are used.

If a loop count (Lnn code number) is defined within the block, the canned cycle will repeat that many times with an incremental (U or W) move between each cycle. Enter the number of repeats (L) each time a repeated operation is needed; the number of repeats (L) is not remembered for the next canned cycle.

Spindle control M codes should not be used while a canned cycle is active.

Canned Cycles with Live Tooling

The canned cycles G81, G82, G83, G85, G89 can be used with the live tooling. This parameter prevents the main spindle from turning during one of the above listed canned cycles. If this bit is set to 1, it is the user's responsibility to activate the appropriate spindle prior to performing the canned cycle, that is, some programs must be checked to be sure they **explicitly** turn on the main spindle before running the canned cycles. Note that G86, G87 and G88 are not usable with live tooling.



G CODE TABLE OF CONTENTS

G CODES - PREPARATORY FUNCTIONS	115
G00 Rapid Motion Positioning (Group 01)	118
G01 Linear Interpolation Motion (Group 01)	118
G02 CW Circular Interpolation Motion / G03 CCW Circular Interpolation Motion (Group 01)	123
G04 Dwell (Group 00)	124
G05 Fine Spindle Control Motion (Group 00)	124
G09 Exact Stop (Group 00)	126
G10 Set Offsets (Group 00)	126
G14 Sub-Spindle Swap / G15 Cancel (Group 17)	127
G17 XY Plane, G18 ZX Plane and G19 YZ Plane (Group 02)	127
G18 ZX Plane Selection (Group 02)	127
G18 ZX Plane Selection (Group 02)	127
G20 Select Inches / G21 Select Metric (Group 06)	127
G28 Return To Machine Zero, set optional G29 Reference point (Group 00)	127
G29 Return from Reference Point (Group 00)	127
G31 Skip Function (This G-code is optional and requires a probe) (Group 00)	127
G32 Thread Cutting (Group 01)	128
G40 Tool Nose Compensation Cancel (Group 07)	129
G41 Tool Nose Compensation (TNC) Left / G42 TNC Right (Group 07)	130
G50 Set Global coordinate Offset FANUC, YASNAC (Group 00)	130
G50 Spindle Speed Clamp	131
G51 Cancel Offset (YASNAC) (Group 00)	131
G52-G64 Work Coordinate System	131
G52 Set Local Coordinate System FANUC (Group 00)	131
G53 Machine Coordinate Selection (Group 00)	132
G54-59 Select Coordinate System #1 - #6 FANUC (Group 12)	132
G61 Exact Stop Modal (Group 15)	132
G64 G61 Cancel (Select normal cutting mode) (Group 15)	132
G70 Finishing Cycle (Group 00)	132
G71 O.D./I.D. Stock Removal Cycle (Group 00)	133
G72 End Face Stock Removal Cycle (Group 00)	140
G73 Irregular Path Stock Removal Cycle (Group 00)	143
G74 End Face Grooving Cycle, Peck Drilling (Group 00)	145
G75 O.D./I.D. Grooving Cycle (Group 00)	147
G76 Threading Cycle, Multiple Pass (Group 00)	148
G77 Flatting Cycle (This G-code is optional and is used for live tooling) (Group 00)	152
G80 Canned Cycle Cancel (Group 09*)	154
G81 Drill Canned Cycle (Group 09)	154
G82 Spot Drill Canned Cycle (Group 09)	154
G83 Normal Peck Drilling Canned Cycle (Group 09)	155
G84 Tapping Canned Cycle (Group 09)	155
G85 Boring Canned Cycle (Group 09)	156
G86 Bore and Stop Canned Cycle (Group 09)	156
G87 Bore and Manual Retract Canned Cycle (Group 09)	157
G88 Bore and Dwell and Manual Retract Canned Cycle (Group 09)	157



G89 Bore and Dwell Canned Cycle (Group 09)	158
G90 O.D./I.D. Turning Cycle (Group 01)	158
G92 Threading Cycle (Group 01)	159
G94 End Facing Cycle (Group 01)	160
G95 Live Tooling Rigid Tap (Face) (Group 09)	161
G96 Constant Surface Speed ON (Group 13)	162
G97 Constant Surface Speed OFF (Group 13)	162
G98 Feed Per Minute (Group 10)	162
G99 Feed Per Revolution (Group 10)	162
G100 Disable Mirror Image (Group 00)	162
G101 Enable Mirror Image (Group 00)	162
G102 Programmable Output to RS-232 (Group 00)	162
G103 Limit Block Lookahead (Group 00)	163
G105 Servo Bar Command	163
G110,G111 and G114-G129 Coordinate System (Group 12)	163
G112 XY to XC interpretation (Group 04)	163
G113 G112 Cancel (Group 04)	163
G154 Select Work Coordinates P1-99 (Group 12)	164
G159 Backround Pickup / Part Return	165
G160 APL Axis Command Mode On	165
G161 APL Axis Command Mode Off	165
G184 Reverse Tapping Canned Cycle For Left Hand Threads (Group 09)	165
G186 Rev Live Tool Rig Tap (Group 09)	166
G187 Accuracy Control (Group 00)	166
G195 Live Tool Radial Tapping (Diameter) (Group 00)	167
G196 Reverse Live Tool Vector Tapping (Diameter) Group 00)	167
G200 Index on the Fly (Group 00)	167



G00 Rapid Motion Positioning (Group 01)

- *B B-axis motion command
- *U X-axis incremental motion command
- *W Z-axis incremental motion command
- *X X-axis absolute motion command
- *Z Z-axis absolute motion command

* indicates optional

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.

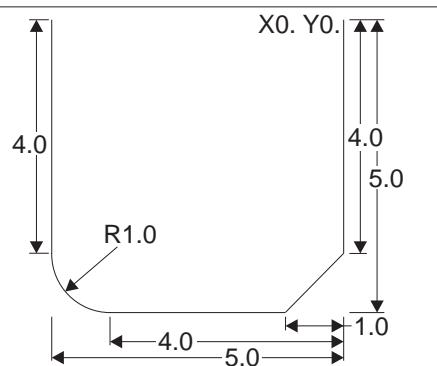
G01 Linear Interpolation Motion (Group 01)

- F Feed rate
- *B B-axis motion command
- *U X-axis incremental motion command
- *W Z-axis incremental motion command
- *X X-axis absolute motion command
- *Z Z-axis absolute motion command
- A Optional angle of movement (used with only one of X, Z, U, W)
- ,C Distance from center of intersection where the chamfer begins
- ,R Radius of the circle

This G code provides for straight line (linear) motion from point to point. Motion can occur in 1 or 2 axes. All axes will start and finish motion at the same time. The speed of all axes is controlled so that the feed rate specified is achieved along the actual path. The C-axis may also be commanded and this will provide a helical (spiral) motion. C-axis feed rate is dependent on the C-axis diameter setting (Setting 102) to create a helical motion. The F address (feedrate) command is modal and may be specified in a previous block. Only the axes specified are moved. The auxiliary axes B, U, V, and W can also be moved with a G01 but only one axis is moved at a time (accept when U,V and W are used with the Advanced Parts Loader ,APL).

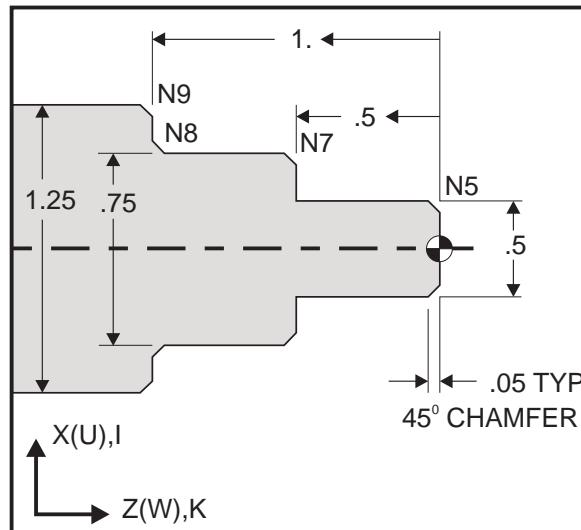
Corner Rounding and Chamfering Example

```
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 theoretical corner of intersection. If the beginning block specifies a ,C the value following the C is the distance from the corner of intersection where the chamfer begins and also the distance from that same corner 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 block that is inserted and the endpoint of that arc. There can be consecutive blocks with chamfer or corner rounding specified. There must be movement on the two axes specified by the selected plane (whichever plane that is active X-Y (G17) X-Z (G18) or Y-Z (G19)). For chamfering a **90° angle only**, a K value can be substituted where ,C is used.



Automatic Chamfering
%
O0001 (Automatic Chamfering)
N1 G50 S1500
N2 G00 T101 G97 S500 M03
N3 G00 X0 Z0.25
N4 G01 Z0 F0.005
N5 G01 X0.50 K-0.050
N6 G01 Z-0.50
N7 G01 X0.75 K-0.050
N8 G01 Z-1.0 I0.050
N9 G01 X1.25 K-0.050
N10 G01 Z-1.5
N11 G00 X1.5 Z0.25
M30
%

Chamfering and Corner Rounding

The following G-code syntax automatically includes a 45° chamfer or corner radius between two blocks of linear interpolation which intersect a right (90 degree) angle.

Chamfering Syntax
G01 X(U) x Kk
G01 Z(W) z li

Corner Rounding Syntax
G01 X(U) x Rr
G01 Z(W) z Rr

Addresses

I = chamfering, Z to X (X axis direction, +/-, "Radius" value)

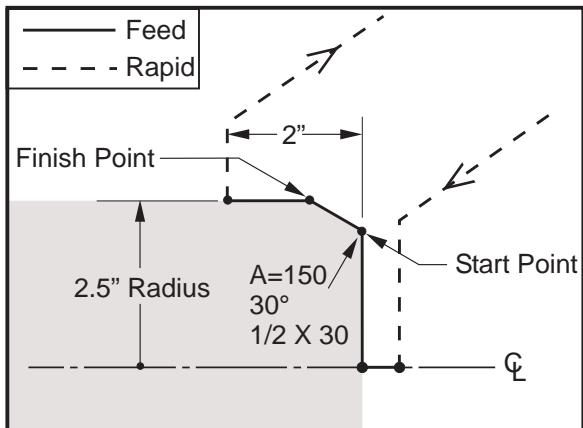
K = chamfering, X to Z (Z axis direction, +/-)

R = corner rounding (X or Z axis direction, +/-, "Radius" value)

NOTE: A -30 = A150; A -45 = A135

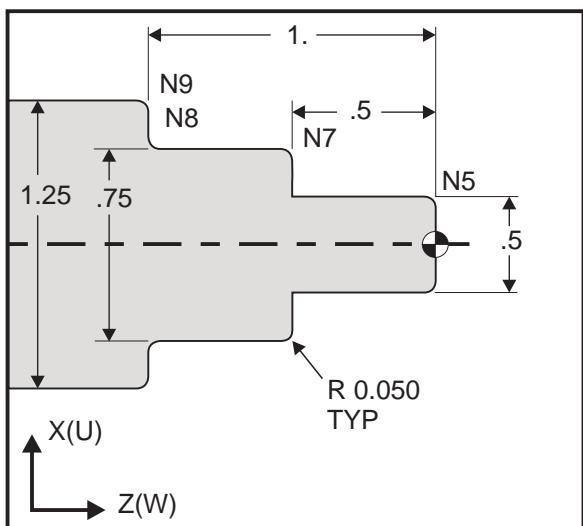
G01 Chamfering with A

When specifying an angle (A), command motion in only one of the other axes (X or Z), the other axis is calculated based on the angle.



```
T606  
G54;  
M03 S1500 G97;  
G00 X5. Z0.1;  
X0;  
G01 Z0 F0.01  
G01 X4. Z0 F0.012;(start point)  
X5. (finish point) A150. (angle to finish point);  
Z-2.;  
X6.;  
G28;  
M30;
```

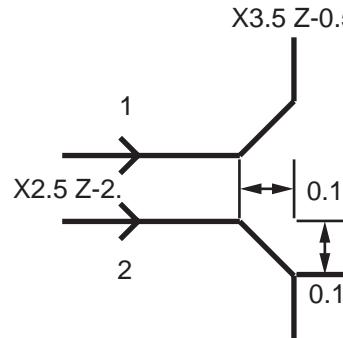
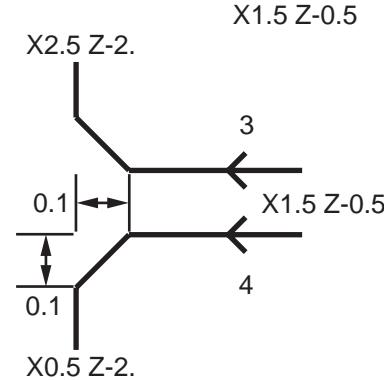
Corner Chamfering

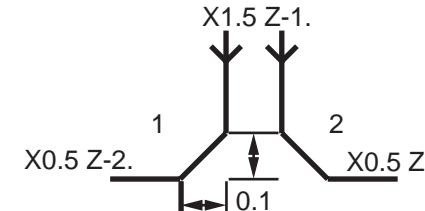
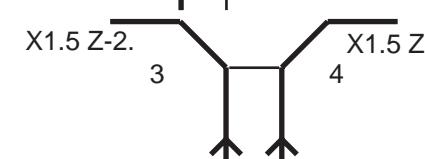


```
%  
O0005 (Automatic Corner Rounding)  
T101  
N1 G50 S1500  
N2 G00 G97 S500 M03  
N3 X0 Z0.25  
N4 G01 Z0 F0.005  
N5 G01 X0.5 R-0.050  
N6 G01 Z-0.50  
N7 G01 X0.75 R-0.050  
N8 G01 Z-1.0 R0.050  
N9 G01 X1.25 R-0.050  
N10 G01 Z-1.5  
N11 G00 X1.5 Z0.25  
G28  
M30  
%
```

NOTES: 1) Incremental programming is possible if U_b or W_b is specified in place of X_b or Z_b , respectively. So its actions will be as follows: $X(POS_{current} + i) = U_i$, $Z(POS_{current} + k) = W_k$, $X(POS_{current} + r) = U_r$, $Z(POS_{current} + r) = W_r$.
2) $POS_{current}$ indicates current position of X or Z axis. 3) I, K and R always specify a radius value (radius programming value).



Chamfering	Code/Example	Movement	
1. Z+ to X+	X2.5 Z-2; G01 Z-0.5 I0.1; X3.5;	X2.5 Z-2; G01 Z-0.6; X2.7 Z-0.5; X3.5;	
2. Z+ to X-	X2.5 Z-2.; G01 Z-0.5 I-0.1; X1.5;	X2.5 Z-2.; G01 Z-0.6; X2.3 Z-0.5; X1.5;	
3. Z- to X+	X1.5 Z-0.5.; G01 Z-2. I0.1; X2.5;	X1.5 Z-0.5 G01 Z-1.9; X1.7 Z-2.; X2.5;	
4. Z- to X-	X1.5 Z-0.5.; G01 Z-2. I-0.1; X0.5;	X1.5 Z-0.5; G01 Z-1.9; X1.3 Z-2.; X0.5;	

Corner Rounding	Code Example	Movement	
1. X- to Z-	X1.5 Z-1.; G01 X0.5 K-0.1; Z-2.;	X1.5 Z-1.; G01 X0.7; X0.5 Z-1.1; Z-2.	
2. X- to Z+	X1.5 Z-1.; G01 X0.5 K0.1; Z0.;	X1.5 Z-1.; G01 X0.7; X0.5 Z-0.9; Z0.;	
3. X+ to Z-	X0.5 Z-1.; G01 X1.5 K-0.1; Z-2.;	X0.5 Z-1.; G01 X1.3; X1.5 Z-1.1; Z-2.	
4. X+ to Z+	X0.5 Z-1.; G01 X1.5 K0.1; Z0.;	X0.5 Z-1.; G01 X1.3; X1.5 Z-0.9; Z0.;	



Corner Rounding	Code Example	Movement
1. Z+ to X+	X2. Z-2.; G01 Z-1 R.1; X3.;	X2. Z-2.; G01 Z-1.1; G03 X2.2 Z-1. R0.1; G01 X3.;
2. Z+ to X-	X2. Z-2.; G01 Z-1. R-0.1; X1.;	X2. Z-2.; G01 Z-1.1; G02 X1.8 Z-1 R0.1; G01 X1.;
3. Z- to X+	X2. Z-1.; G01 Z-2. R0.1; X3.;	X2. Z-1.; G01 Z-1.9; G02 X2.2 Z-2. R0.1; G01 X3.;
4. Z- to X-	X2. Z-1.; G01 Z-2. R-0.1; X1.;	X2. Z-1.; G01 Z-1.9. ; G03 X1.8 Z-2.; G01 X1.;

Corner Rounding	Code Example	Movement
1. X- to Z-	X1.5 Z-1.; G01 X0.5 K-0.1; Z-2.;	X1.5 Z-1.; G01 X0.7; X0.5 Z-1.1; Z-2.
2. X- to Z+	X1.5 Z-1.; G01 X0.5 K0.1; Z0.;	X1.5 Z-1.; G01 X0.7; X0.5 Z-0.9; Z0.;
3. X+ to Z-	X0.5 Z-1.; G01 X1.5 K-0.1; Z-2.;	X0.5 Z-1.; G01 X1.3; X1.5 Z-1.1; Z-2.
4. X+ to Z+	X0.5 Z-1.; G01 X1.5 K0.1; Z0.;	X0.5 Z-1.; G01 X1.3; X1.5 Z-0.9; Z0.;



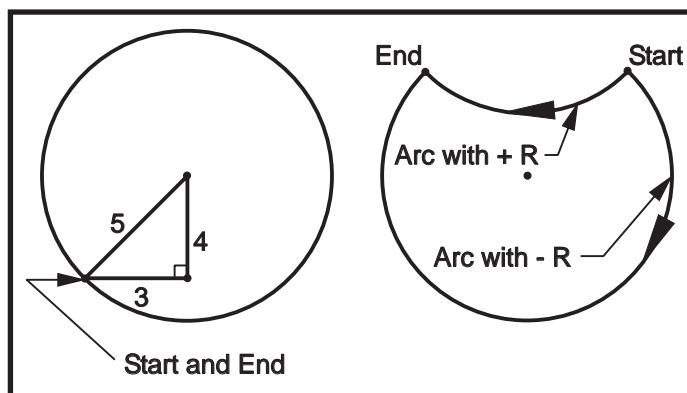
- Rules:**
- 1) Use K address only with X(U) address. Use I address only with Z(W) address.
 - 2) Use R address with either X(U) or Z(W), but not both in the same block.
 - 3) Do not use I and K together on the same block. When using R address, do not use I or K.
 - 4) The next block must be another single linear move that is perpendicular to the previous one.
 - 5) Automatic chamfering or corner rounding cannot be used in a threading cycle **or in a canned cycle.**
 - 6) Chamfer or corner radius must be small enough to fit between the intersecting lines.
 - 7) There should be only a single move along the X or Z in linear mode (G01) for chamfering or corner rounding.

G02 CW Circular Interpolation Motion / G03 CCW Circular Interpolation Motion (Group 01)

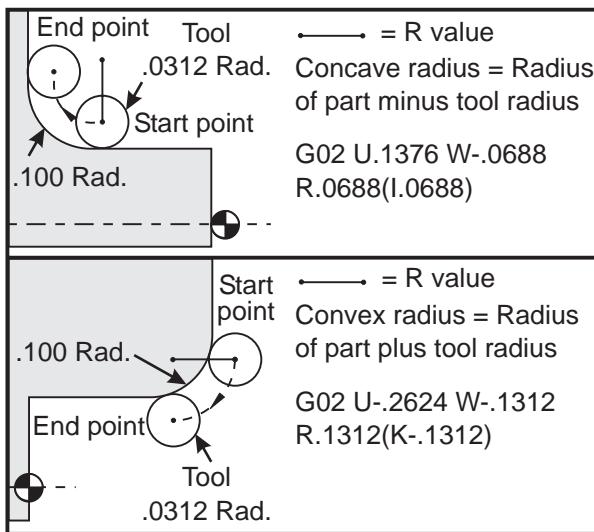
F	Feed rate
*I	Distance along X-axis to center of circle
*K	Distance along Z-axis to center of circle
*R	Radius of arc
*U	X-axis incremental motion command
*W	Z-axis incremental motion command
*X	X-axis absolute motion command
*Z	Z-axis absolute motion command
,C	Distance from center of intersection where the chamfer begins
,R	Radius of the circle

* indicates optional

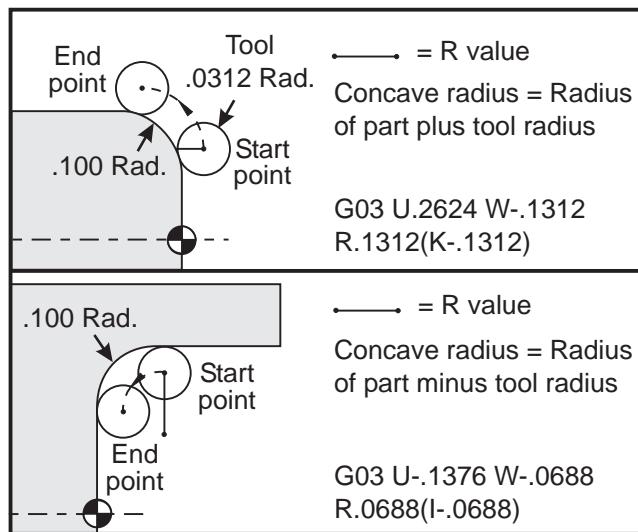
These G codes are used to specify a circular motion (CW or CCW) of the linear axes (Circular motion is possible in the X and Z axes as selected by G18). The X and Z values are used to specify the end point of the motion and can use either absolute (U and W) or incremental motion (X and Z). If either the X or Z is not specified, the endpoint of the arc is the same as the starting point for that axis. There are two ways to specify the center of the circular motion; the first uses I or K to specify the distance from the starting point to the center of the arc; the second uses R to specify the radius of the arc (maximum 7740 inches).



G02



G02



G03

R is used to specify the center of the circle. R is the distance from the starting point to the center of the circle. With a positive R, the control will generate a path of 180 degrees or less; to generate a radius of over 180 degrees, specify a negative R. X or Z is required to specify an endpoint if different from the starting point.

The following line will cut an arc of less than 180 degrees:

G01 X3.0 Z4.0

G02 Z-3.0 R5.0

I and K are used to specify the center of the arc. When I and K are used, R may not be used. The I or K is the signed distance from the starting point to the center of the circle. If only one of I or K is specified, the other is assumed to be zero.

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.

G05 Fine Spindle Control Motion (Group 00)

R Angular motion of the spindle, in degrees.

F Feed Rate of the center of the tool, in inches per minute.

*U X-axis incremental motion command.

*W Z-axis incremental motion command.

*X X-axis absolute motion command.

*Z Z-axis absolute motion command.

* indicates optional

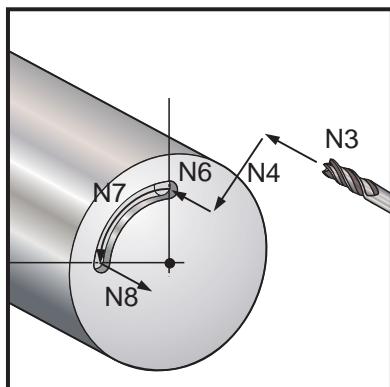
This G code is used to specify a precise motion of the spindle.

The spindle speed is determined by looking at the greatest X-axis value encountered during the cut.

The largest feed per revolution value that can be specified is approximately 14.77. This means that G5 motions with small R motions relative to X or Z motions will not work. For example, an R motion of 1.5 degrees, the largest X or Z motion that can be specified is $14.77 * 1.5 / 360 = .0615$ inches. Conversely, an X or Z motion of .5 inches must have an R travel of at least $.5 * 360 / 14.77 = 12.195$ degrees.

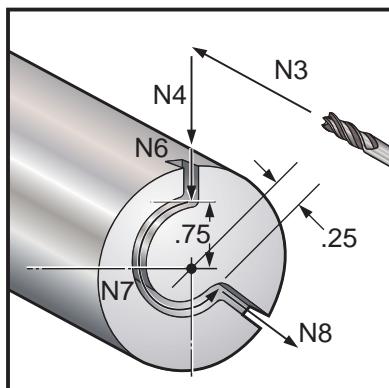


Simple Face Slot Example with G05



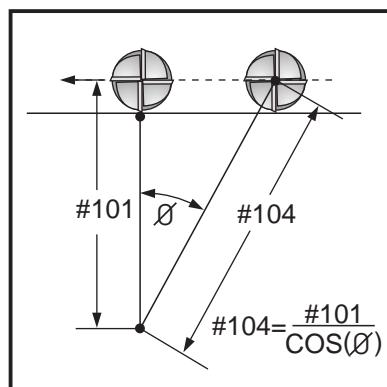
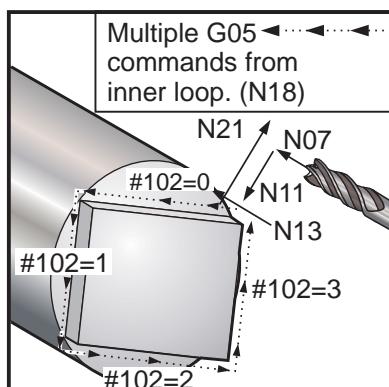
(Assume pilot hole is already drilled.)
 N1T303 (Small End Mill)
 N2M19 (Orient Spindle)
 N3G00 Z0.5
 N4G00 X1.
 N5M133 P1500
 N6G98 G1 F10. Z-.25 (Plunge into pre-drilled hole)
 N7G05 R90. F40.(Make slot)
 N8G01 F10. Z0.5 (Retract)
 N9M135
 N10 G99 G28 U0 W0

Simple Cam Example with G05



N1 T303 (Small End Mill)
 N2M19
 N3G00 Z-.25
 N4G00 X2.5 (Approach 2" diam stock)
 N5M133 P1500
 N6G98 G01 X1.5 F40. (Cut to top of cam)
 N7G05 R215. X.5 F40. (Cut Cam)
 N8G01 X2.5 F40. (Cut out of cam)
 N9M135
 N10G99 G28 U0 W0

Flattening Example with G05



O01484 (Cut a square with G05)

N1 G28 X0.
 N2 G28 Z0.
 N3 G54 G00 G40 G97
 N4 G103 P3
 N5 T707 (.75 dia high-speed end mill)
 N6 M19
 N7 G00 Z0.5
 ()
 N8 #101= [0.707 + 0.75 / 2.] (101 = Closest approach. Center to side plus half of tool diameter)



```
N9 #101= #101 * 2 (Multiply by 2 for diam.)  
N10#104= [ #101 / COS[ 45. ] ] (104 = Distance at corner. )  
N11G98 G01 X#104 F100.  
N12M133 P1500  
N13Z-0.1 (Feed into pre-drilled hole)  
N14#102=0  
    WHILE [ #102 LT 4 ] DO1 (Four sided shape)  
N15#103= -45. (Angle from center of flat)  
    ()  
    WHILE [ #103 LT 45. ] DO2  
N16#103= [ #103 + 5. ]  
N17#104= [ #101 / COS[ #103 ] ]  
N18G05 X#104 R5. F20.  
    END2  
    ()  
N19#102= [ #102 + 1 ]  
    END1  
    ()  
N20M135  
N21G28 U0  
N22G28 W0  
N23M30
```

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 COMMON and G54-G59
- L10 Geometry or shift offset
- L1 or L11 Tool wear
- L20 Auxiliary work coordinate origin for G110-G129

P Selects a specific offset.

- P1-P50 References geometry, wear or work offsets (L10-L11)
- P51-P100 References shift offsets (YASNAC) (L10-L11)
- P0 References COMMON work coordinate offset (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)

Q Imaginary tool nose tip direction

R Tool nose radius

*U Incremental amount to be added to X-axis offset

*W Incremental amount to be added to Z-axis offset

*X X-axis offset

*Z Z-axis offset

* indicates optional

Programming Examples

G10 L2 P1 W6.0 (Move coordinate G54 6.0 units to the right);



G10 L20 P2 X-10.Z-8. (Set work coordinate G111 to X-10.0, Z-8.0);
G10 L10 P5 R.032 (Set geometry offset of Tool #5 to .032);
G10 L10 P5 R.0625 (Set radius of Tool #5 to 1/16");

G14 Sub-Spindle Swap / G15 Cancel (Group 17)

G14 causes the subspindle to become the primary spindle and will react to commands normally used for the main spindle. For example, M03, M04, M05 and M19 will affect the sub-spindle, and M143, M144, M145 and M119 will cause an alarm. Note that G50 will limit the sub-spindle speed, and G96 will set the sub-spindle surface feed value. These G-codes will adjust the subspindle speed when there is motion in the X-axis. G01 Feed Per Rev will feed based on the sub-spindle.

The G14 command will automatically activate Z-axis mirroring. If the Z-axis is already mirrored (Setting 47 or G101) the mirror function will be canceled. G14 is canceled by a G15, an M30, reaching the end of a program, and by pressing Reset.

G17 XY Plane, G18 ZX Plane and G19 YZ Plane (Group 02)

The G18 code is used to select the ZX plane for circular motion. This is the default for the HAAS lathe. This means that a circular motion in the plane of the X-Z work space may be programmed without first selecting G18. In this plane, circular motion is defined as clockwise for the operator looking down onto the X-Z work space from the front. This is the motion of the tool relative to the part.

G17/G18 – These G code supports the Cartesian to Polar transformation feature (G112/G113). Commanding a G18 will cause the control to return to the normal ZX plane.

G18 ZX Plane Selection (Group 02)

The G18 code is used to select the ZX plane for circular motion. In this plane, circular motion is defined as clockwise for the operator looking down onto the X-Z work space from the front. This is the motion of the tool relative to the part.

G18 ZX Plane Selection (Group 02)

The G18 code is used to select the ZX plane for circular motion. In this plane, circular motion is defined as clockwise for the operator looking down onto the X-Z work space from the front. This is the motion of the tool relative to the part.

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, set 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.

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 Skip Function (This G-code is optional and requires a probe) (Group 00)

F	Feed rate
*U	X-axis incremental motion command
*W	Z-axis incremental motion command
*X	X-axis absolute motion command
*Z	Z-axis absolute motion command

* indicates optional

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.



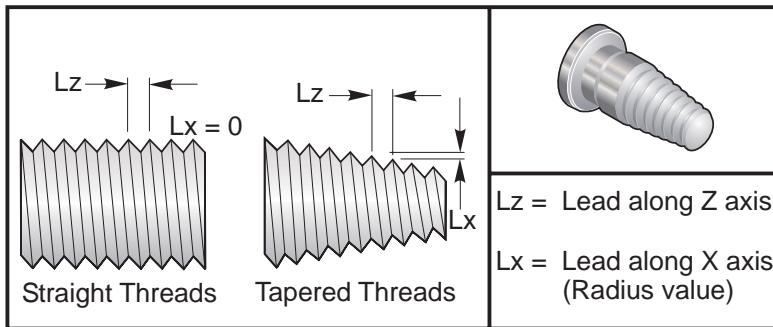
Do not use Cutter Compensation with a G31.

Also see M78 and M79.

G32 Thread Cutting (Group 01)

F	Feed rate
Q	Thread Angle
U	X-axis incremental positioning command
W	Z-axis incremental positioning command
X	X-axis absolute positioning command
Z	Z-axis absolute positioning command

-
- NOTES:**
- Feed rate is equivalent to thread lead.
 - Movement on at least one axis must be specified.
 - Tapered threads have lead in both X and Z. In this case set the feed rate to the larger of the two leads. G99 (Feed per Revolution) must be active.



G32 Definition of Lead (Feed Rate) for Straight and Tapered Threads

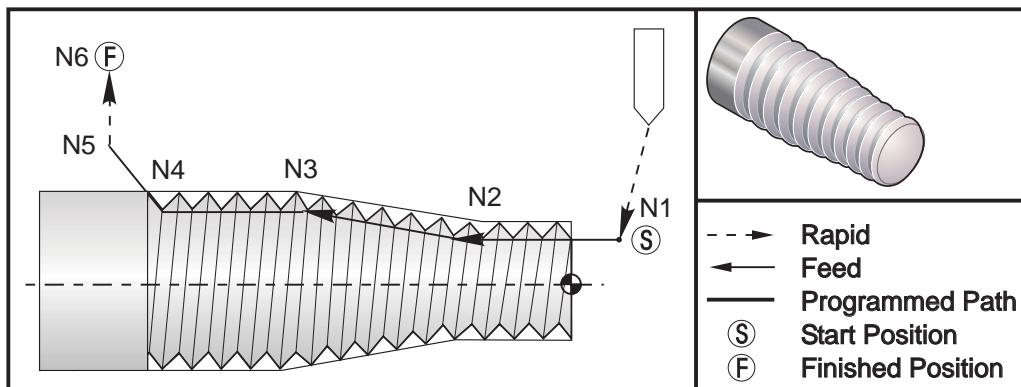
G32 differs from other thread cutting cycles in that taper and/or lead can vary continuously throughout the entire thread. In addition, no automatic position return is performed at the end of the threading operation.

At the first line of a G32 block of code, axis feed is synchronized with the rotation signal of the spindle encoder. This synchronization remains in effect for each line in a G32 sequence. It is possible to cancel G32 and recall it without losing the original synchronization. This means multiple passes will exactly follow the previous tool path (The actual spindle RPM must be exactly the same between passes).

NOTE: Single Block Stop and Feed Hold are deferred until last line of a G32 sequence.

NOTE: Feed Rate Override is ignored while G32 is active, Actual Feed Rate will always be 100% of programmed feed rate.
M23 and M24 have no effect on a G32 operation, the user must program chamfering if needed.
G32 must not be used with any G-Code Canned Cycles (i.e.: G71).
Do Not change spindle RPM during threading.

CAUTION! G32 is Modal. Always cancel G32 with another Group 01 G-code at the end of a threading operation. (Group 01 G-Codes: G00, G01, G02, G03, G32, G90, G92, and G94)



Straight-to-Taper-to-Straight Thread Cutting Cycle

NOTE: Example is for reference only, multiple passes are usually required to cut actual threads.

G32 Program Example

```
...
G97 S400 M03          (Constant Surface Speed Cancel)
N1 G00 X0.25 Z0.1     (Rapid to Start Position)
N2 G32 Z-0.26 F0.065   (Straight thread, Lead(Lz) = 0.065)
N3 X0.455 Z-0.585     (Straight thread blends to tapered thread)
N4 Z-0.9425            (Taper thread blends back to straight thread)
N5 X0.655 Z-1.0425    (Escape at 45 degrees)
G00 X1.2               (Rapid to Finish Position, cancel G32)
G00 Z0.1
```

...

Q option example:

```
G32 X-1.99 Z-2. Q60000 F0.2;      (60 degree cut)
G32 X-1.99 Z-2. Q120000 F0.2;    (120 degree cut)
G32 X-1.99 Z-2. Q270123 F0.2;   (270.123 degree cut)
```

The following rules apply to the use of Q:

1. The start angle (Q) is not a modal value. It must be specified every time it is used. If no value is specified then a zero (0) angle is assumed.
2. The angle of threading increment is 0.001 degrees and it cannot have a decimal point. Therefore a 180° angle must be specified as Q180000 and a 35° angle as Q35000.
3. The Q angle must be entered as a positive value from 0 to 360000.

Comments

(Constant Surface Speed Cancel)
(Rapid to Start Position)
(Straight thread, Lead(Lz) = 0.065)
(Straight thread blends to tapered thread)
(Taper thread blends back to straight thread)
(Escape at 45 degrees)
(Rapid to Finish Position, cancel G32)

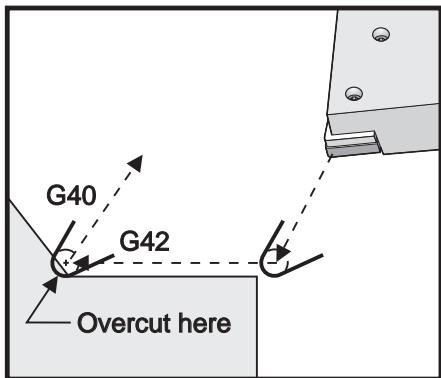
G40 Tool Nose Compensation Cancel (Group 07)

*X	X axis absolute location of departure target
*Z	Z axis absolute location of departure target
*U	X axis incremental distance to departure target
*W	Z axis incremental distance to departure target
*I	X axis intersection vector direction, (radius)
*K	Z axis intersection vector direction

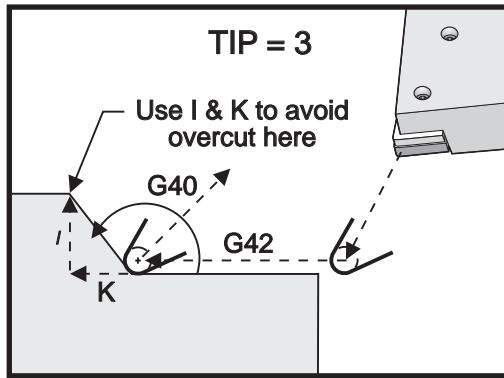
* indicates optional

G40 cancels G41 or G42. Programming Txx00 will also cancel tool nose compensation. Cancel tool nose compensation before the end of a program.

The tool departure usually does not correspond with a point on the part. In many cases overcutting or undercutting can occur.



G40



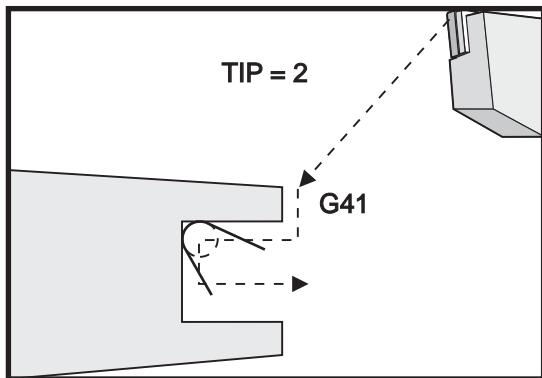
G40 Using I and K.

When address codes I and K are used in a G40 departure block, the control will use these values as an intersection for the end point of the last compensated cut. The following figure illustrates where I and K lie in relation to the departure stroke. Usually I and K lie along a face of the machined part.

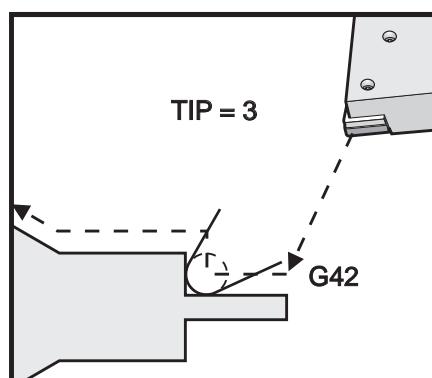
The values of I and K can be determined by calculating the sine of the angle for I and the cosine of the angle for K, where the angle is taken to the face of the part with respect to the Z axis.

G41 Tool Nose Compensation (TNC) Left / G42 TNC Right (Group 07)

G41 or G42 will select tool nose compensation. G41 moves the tool to the left of the programmed path to compensate for the size of a tool and vice versa for G42. A tool offset must be selected with a Tnnxx code, where xx corresponds to the offsets that are to be used with the tool.



G41



G42

G50 Set Global coordinate Offset FANUC, YASNAC (Group 00)

- U Incremental amount and direction to shift global X coordinate.
- X Absolute global coordinate shift.
- W Incremental amount and direction to shift global Z coordinate.
- Z Absolute global coordinate shift.
- S Clamp spindle speed to specified value
- T Apply tool shift offset (YASNAC)

G50 can perform several functions. It can set the global coordinate, it can shift the global coordinate, and it can limit the spindle speed to a maximum value. Refer to the "Coordinate Systems and Offsets" section for a discussion of these.

To set the global coordinate, command G50 with an X or Z value. The effective coordinate will become the value specified in address code X or Z. Current machine location, work offsets and tool offsets are taken into account. The global coordinate is calculated and set.



Example: G50 X0 Z0 (Effective coordinates are now zero);

To shift the global coordinate system, specify G50 with a U or W value. The global coordinate system will be shifted by the amount and direction specified in U or W. The current effective coordinate displayed will change by this amount in the opposite direction. This method is often used to place the part zero outside of the work cell.

Example: G50 W-1.0 (Effective coordinates will be shifted left 1.0);

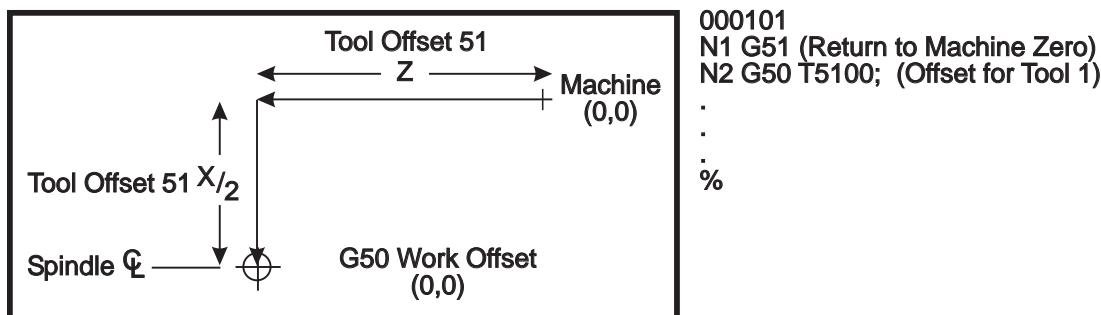
To set a YASNAC style work coordinate shift, specify G50 with a T value (Setting 33 must be set to YASNAC). The global coordinate is set to the X and Z values in the Tool Shift Offset page. Values for the T address code are, Txxyy where xx is between 51 and 100 and yy is between 00 and 50. For example, T5101 specifies tool shift index 51 and tool wear index 01; it does not cause tool number 1 to be selected. To select another Txxyy code must be used outside the G50 block. The following two examples illustrate this method to select Tool 7 using Tool Shift 57 and Tool Wear 07.

Example 1

G51; (Cancel Offsets)
T700 M3; (Change to Tool 7, Turn on Spindle)
G50 T5707; (Apply Tool Shift and Tool Wear to Tool 7)

Example 2

G51; (Cancel Offsets)
G50 T5700; (Apply Tool Shift)
T707 M3; (Change to Tool 7 and apply Tool Wear)



G50 YASNAC Tool Shift

G50 Spindle Speed Clamp

G50 can be used to limit the maximum spindle speed. The control will not allow the spindle to exceed the S address value specified in the G50 command. This is used in constant surface feed mode (G96).

N1 G50 S3000 ; (Spindle rpm will not exceed 3000 rpm)
N2 G97 M3 ; (Enter constant surface speed cancel, spindle on)

Note: To cancel this command, use another G50 and specify the maximum spindle RPM for the machine.

G51 Cancel Offset (YASNAC) (Group 00)

G51 is used to cancel any existing tool wear and work coordinate shift and return to the machine zero position.

G52-G64 Work Coordinate System

The HAAS CNC lathe control supports both YASNAC and FANUC coordinate systems. Work coordinates together with tool offsets can be used to position a part program anywhere within the work area. Also see the Tool Offsets section.

G52 Set Local Coordinate System FANUC (Group 00)

This code selects the user coordinate system.



G53 Machine Coordinate Selection (Group 00)

This code temporarily cancels work coordinates offsets and uses the machine coordinate system.

G54-59 Select Coordinate System #1 - #6 FANUC (Group 12)

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

G61 Exact Stop Modal (Group 15)

The G61 code is used to specify exact stop. Rapid and interpolated moves will decelerate to an exact stop before another block is processed. In exact stop, moves will take a longer time and continuous cutter motion will not occur. This may cause deeper cutting where the tool stops.

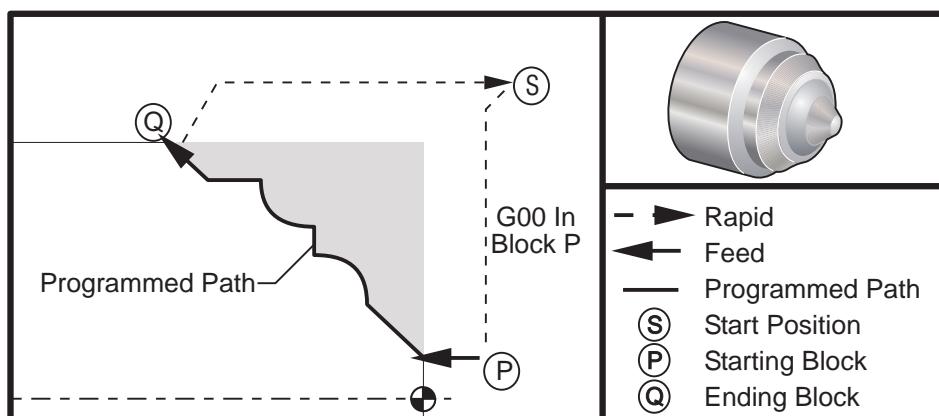
G64 G61 Cancel (Select normal cutting mode) (Group 15)

The G64 code is used to cancel exact stop.

G70 Finishing Cycle (Group 00)

The G70 Finishing cycle can be used to finish cut paths that are rough cut with stock removal cycles such as G71, G72 and G73.

- P Starting Block number of routine to execute
Q Ending Block number of routine to execute



G70

Programming Example

G71 P10 Q50 F.012 (rough out N10 to N50 the path)

N10
F0.014
...

N50
...

...

G70 P10 Q50 (finish path defined by N10 to N50)

...

The G70 cycle is similar to a local subprogram call. However, the G70 requires that a beginning block number (P code) and an ending block number (Q code) be specified.

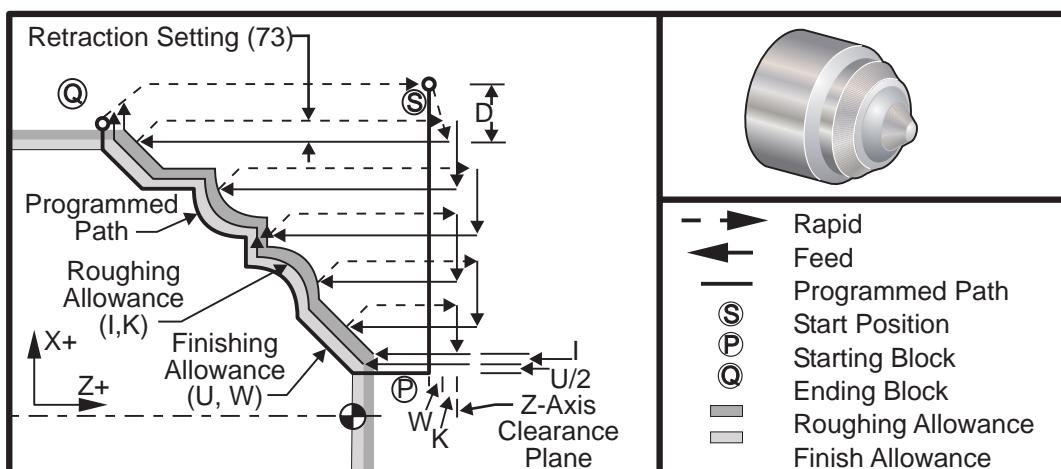


The G70 cycle is usually used after a G71, G72 or G73 has been performed using the blocks specified by P and Q. Any F, S or T codes with the PQ block are effective. After execution of the Q block, a rapid (G00) is executed returning the machine to the start position that was saved before the starting of the G70. The program then returns to the block following the G70 call. A subroutine in the PQ sequence is acceptable providing that the subroutine does not contain a block with an N code matching the Q specified by the G70 call. This feature is not compatible with FANUC or YASNAC controls.

G71 O.D./I.D. Stock Removal Cycle (Group 00)

- * D Depth of cut for each pass of stock removal, positive radius
- * F Feed rate to use throughout G71 PQ block
- * I X-axis size and direction of G71 rough pass allowance, radius
- * K Z-axis size and direction of G71 rough pass allowance
- P Starting Block number of path to rough
- Q Ending Block number of path to rough
- * S Spindle speed to use throughout G71 PQ block
- * T Tool and offset to use throughout G71 PQ block
- * U X-axis size and direction of G71 finish allowance, diameter
- * W Z-axis size and direction of G71 finish allowance
- * R1 YASNAC select Type II roughing

* indicates optional



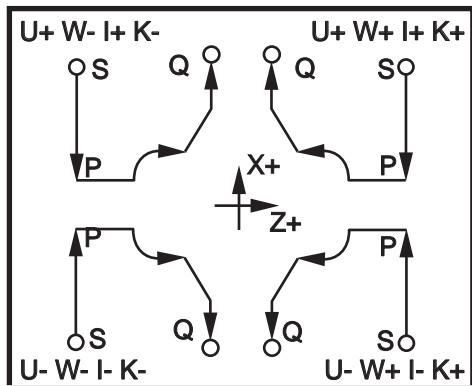
G71

This canned cycle roughs material on a part given the finished part shape. Define the shape of a part by programming the finished tool path and then use the G71 PQ block. Any F,S or T commands on the G71 line or in effect at the time of the G71 is used throughout the G71 roughing cycle. Usually a G70 call to the same PQ block definition is used to finish the shape.

Two types of machining paths are addressed with a G71 command. The first type of path (Type I) is when the X-axis of the programmed path does not change direction. The second type of path (Type II) allows the X-axis to change direction. For both Type I and Type II the programmed path the Z-axis cannot change direction. Type I is selected by having only an X-axis motion in the block specified by P in the G71 call. When both an X-axis and Z-axis motion are in the P block then Type II roughing is assumed. When in YASNAC mode, Type II roughing is selected by including R1 on the G71 command block.

Any one of the four quadrants of the X-Z plane can be cut by specifying address codes D, I, K, U, and W properly.

In the figures, the start position S is the position of the tool at the time of the G71 call. The Z clearance plane is derived from the Z-axis start position and the sum of W and optional K finish allowances.



G71 Address relationships

Type I Details

When Type I is specified by the programmer it is assumed that the X-axis tool path does not reverse during a cut. Each roughing pass X-axis location is determined by applying the value specified in D to the current X location. The nature of the movement along the Z clearance plane for each roughing pass is determined by the G code in block P. If block P contains a G00 code, then movement along the Z clearance plane is a rapid mode. If block P contains a G01 then movement will be at the G71 feed rate.

Each roughing pass is stopped before it intersects the programmed tool path allowing for both roughing and finishing allowances. The tool is then retracted from the material, at a 45 degree angle by the distance specified in setting 73. The tool then moves in rapid mode to the Z-axis clearance plane.

When roughing is completed the tool is moved along the tool path to clean up the rough cut. If I and K are specified an additional rough finish cut parallel to the tool path is performed.

Type II Details

When Type II is specified by the programmer the X axis PQ path is allowed to vary (for example, the X-axis tool path can reverse direction).

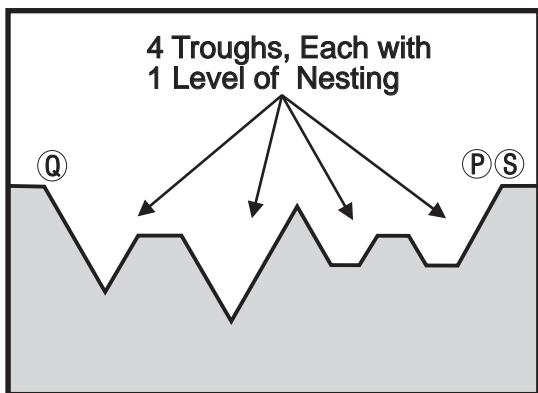
The X axis PQ path must not exceed the original starting location. The only exception is the ending Q block.

Type II roughing, when Setting 33 is set to YASNAC, must include R1 (with no decimal) on the G71 command block.

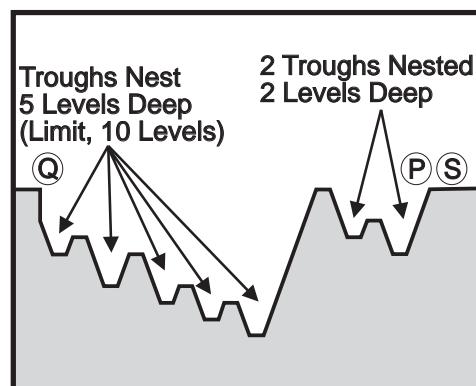
Type II, when Setting 33 is set to FANUC, must have a reference move, in both the X and Z axis, in the block specified by P.

Roughing is similar to Type I except after each pass along the Z axis, the tool will follow the path defined by PQ. The tool will then retract parallel to the X axis by a distance defined in Setting 73 (Can Cycle Retraction). The Type II roughing method does not leave steps in the part prior to finish cutting and typically results in a better finish.

Troughs



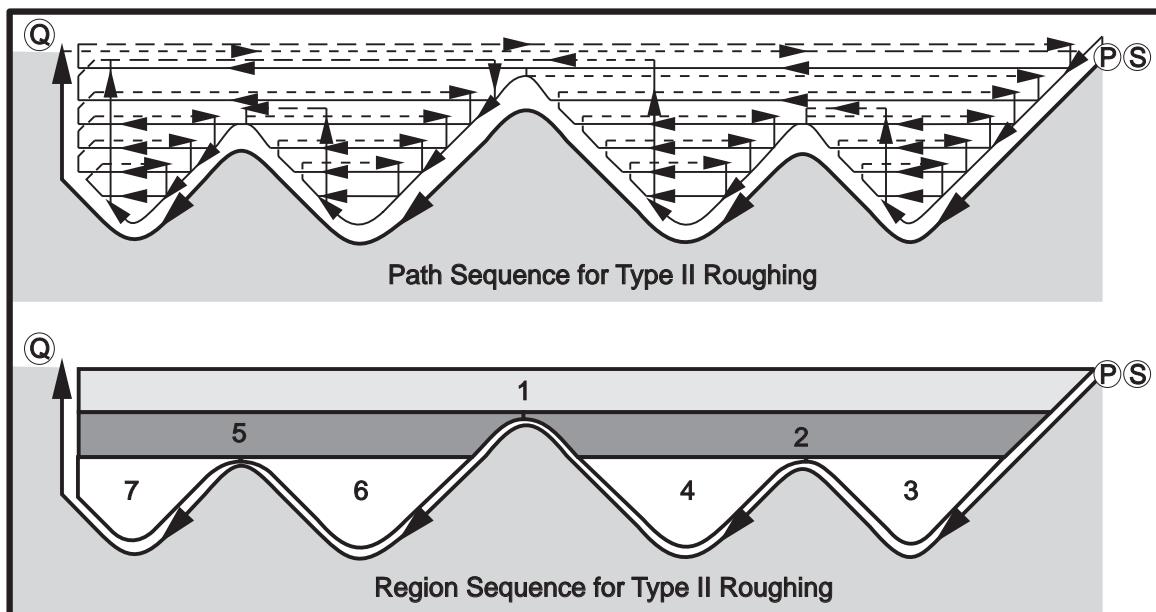
Troughs on the Same Level



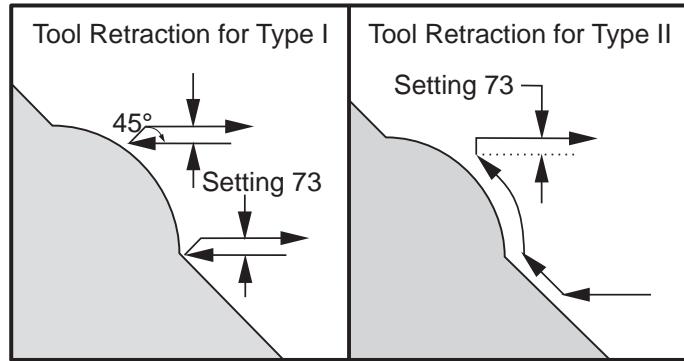
Nested Troughs

A trough can be defined as a change in direction which creates a concave surface in the material being cut. If successive troughs are on the same level, there can be an unlimited number of troughs. When troughs are within troughs (nested), there can be no more than 10 levels of trough nesting.

The following figures illustrate the sequence of roughing cuts (Type I and II) for PQ paths with multiple troughs. All material above troughs is roughed first, followed by the troughs themselves in the direction of Z.



Path Sequence for Type II Roughing



Type I and II Tool Retraction

Note: An effect of using a Z finish or roughing allowance is the limit between the two cuts on one side of a trough and the corresponding point on the other side of the trough. This distance must be greater than double the sum of the roughing and finish allowances.

For example, if G71 Type 2 path contains the following:

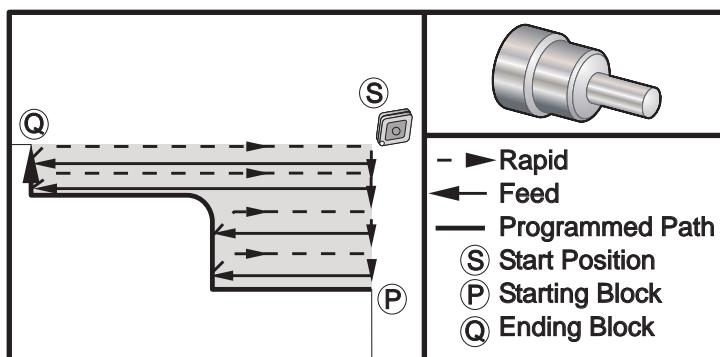
...
X-5.Z-5.
X-5.1Z-5.1
X-3.1Z-8.1

...

The greatest allowance that can be specified is 0.999, since the horizontal distance from the start of cut 2 to the same point on cut 3 is 0.2. If a larger allowance is specified, overcutting will occur.

Cutter compensation is approximated by adjusting the roughing allowance according to the radius and tip type of the tool. Therefore, the limitations that apply to the allowance also apply to the sum of the allowance and the tool radius.

Note: If the last cut in the P-Q path is a non-monotonic curve (using a finish allowance) add a short retraction cut; do not use W.



G71 Basic G Code Example

Program Example

```

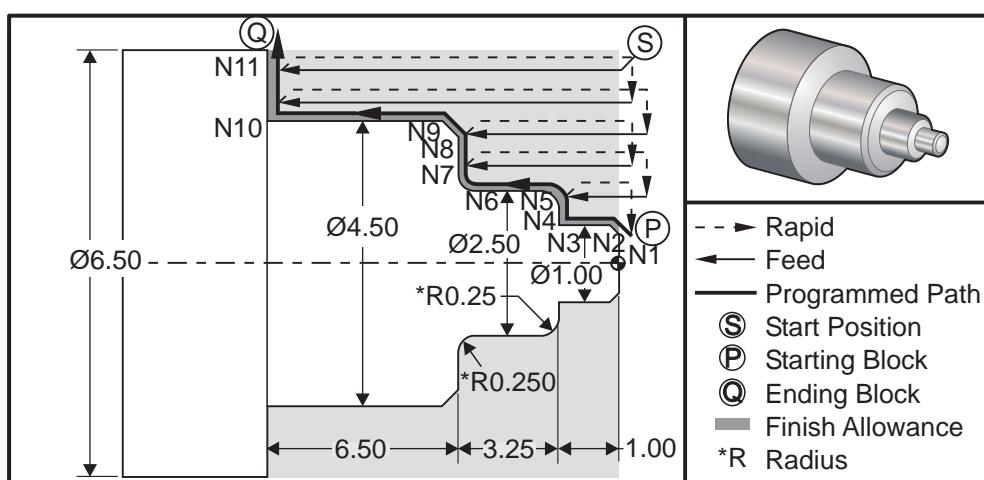
%
O0070
T101
G50 S2500
G97 S509 M03
G00 G54 X6. Z0.05
G96 S800
G71 P1 Q2 D0.15 U0.01 W0.005 F0.014
N1 G00 X2.
N1 G01 Z-3. F0.006
X3.5
G03 X4. Z-3.25 R0.25
G01 Z-6.
N2 X6.
G70 P1 Q2
M09
G28 M05
M30
%

```

Description

(G71 Roughing Cycle)

(FINISH PASS)



G71 Type I O.D./I.D. Stock Removal Example



Program Example

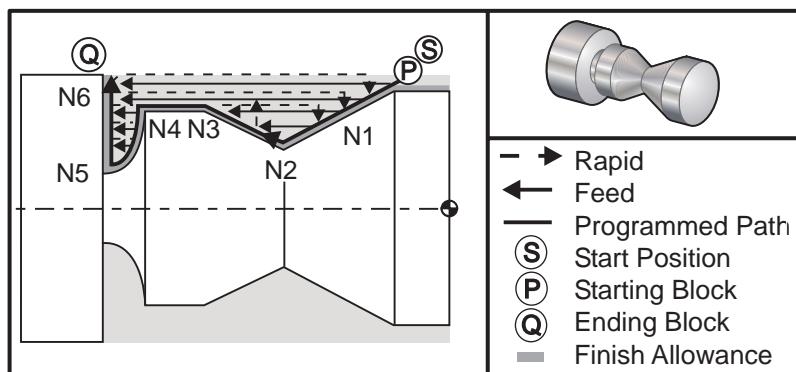
```
%  
O0071  
T101 (CNMG 432)  
(Tool change & apply Offsets)  
G00 G54 X6.6 Z.05 M08  
G50 S2000  
G97 S636 M03  
G96 S750  
G71 P1 Q11 D0.15 U0.01 W0.005 F0.012  
N1   G00 X0.6634  
N2   G01 X1. Z-0.1183 F0.004  
N3   Z-1.  
N4   X1.9376  
N5   G03 X2.5 Z-1.2812 R0.2812  
N6   G01 Z-3.0312  
N7   G02 X2.9376 Z-3.25 R0.2188  
N8   G01 X3.9634  
N9   X4.5 Z-3.5183  
N10  Z-6.5  
N11  X6.0  
G00 X0 Z0 T100  
T202  
G50 S2500  
G97 S955 M03  
G00 X6. Z0.05 M08  
G96 S1500  
G70 P1 Q11  
G00 X0 Z0 T200  
M30  
%
```

Description

(FANUC G71 TYPE I EXAMPLE)

(Rapid to Home Position)
(Set Max RPM 2000)
(Spindle On)
(Constant surface speed On)
(Define rough cycle)
P (Begin definition)
(Finish pass .004" Feed)

Q (End definition)
(Rapid to tool change position)
(Finish tool)



G71 Type II O.D./I.D. Stock Removal Example



Program Example

```

%
O0001
T101
G97 S1200 M03
;
G00 X2. Z0
G71 P1 Q6 D.035 U.03 W0.01 F0.01
;
N1 G01 X1.5 Z-0.5 F0.004
N2 X1. Z-1.
N3 X1.5 Z-1.5
N4 Z-2.
N5 G02 X0.5 Z-2.5 R0.5
N6 G01 X2.
;
T202
G97 S1500 M03
;
G70 P1 Q6
;
G28 M30
%

```

Description

(YASNAC G71 Type II Example)
(*Roughing tool*)

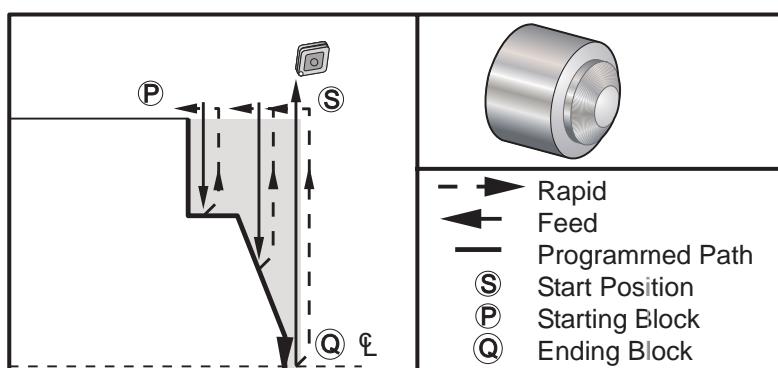
S (*Start position*)

P (PQ Path definition)

Q (PQ Path end)

(*Finishing tool*)

(*Finish pass*)



G72 Basic G Code Example

Program Example

```

%
O0069
T101
G50 S2500
G97 S509 M03
G54 G00 X6. Z0.05
G96 S800
G72 P1 Q2 D0.075 U0.01 W0.005 F0.012
N1 G00 Z-0.65
G01 X3. F0.006
Z-0.3633
X1.7544 Z0.
X-0.0624
N2 G00 Z0.02
G70 P1 Q2
M05
G28
M30
%

```

Description

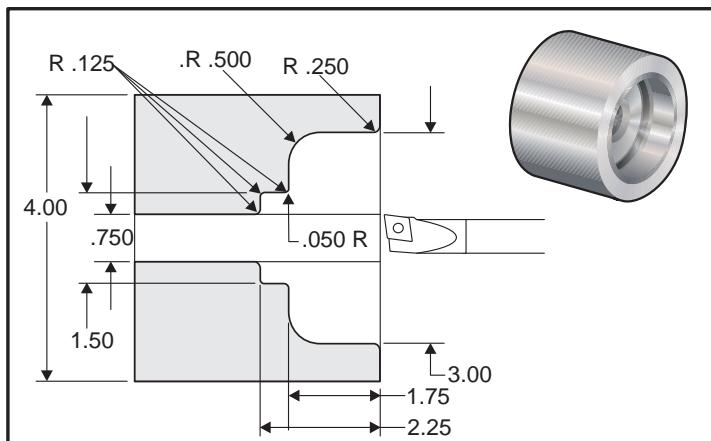
(*G72 Roughing Cycle*)

(*Finish Pass*)



G71 I.D. Stock Removal Example

NOTE: Be sure the start position of the tool is positioned below the diameter of the part you wish to start roughing out, before defining a G71 on an I.D. with this cycle.



TOOL
4

OFFSET
04

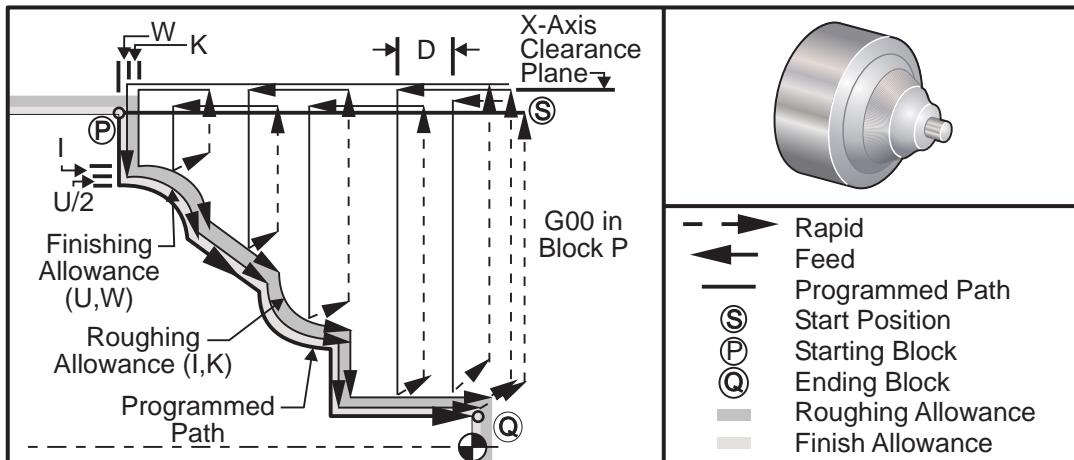
RADIUS
.0

TIP
0

O00088	(Example of using a G71 on an I.D.)
N2 T404	(Tool 4 Offset 4)
N3 G97 S2000 M03	
N4 G54 G00 X0.7 Z0.1 M08	(Rapid to start position below the I.D. stock diameter)
N5 G71 P6 Q10 U-0.01 W0.005 D0.08 F0.01	(U is minus for G71 I.D. Roughing)
N6 G00 X3.	(N6 is start of part path geometry defined by P6 in G71 line)
N7 G01 Z-1.75 F0.005	
N8 X1.5	
N9 Z-2.25 F0.003	
N10 X0.73	(N10 is end of part path geometry defined by Q10 in G71 line)
N11 G70 P6 Q10	(G70 Defines a finish pass for lines P6 through P10)
N12 M09	
N13 G28	(To send machine home for a tool change)

G72 End Face Stock Removal Cycle (Group 00)

- * D Depth of cut for each pass of stock removal, positive
- * F Feed rate to use throughout G72 PQ block
- * I X-axis size and direction of G72 rough pass allowance, radius
- * K Z-axis size and direction of G72 rough pass allowance
- P Starting Block number of path to rough
- Q Ending Block number of path to rough
- * S Spindle speed to use throughout G72 PQ block
- * T Tool and offset to use throughout G72 PQ block
- * U X-axis size and direction of G72 finish allowance, diameter
- * W Z-axis size and direction of G72 finish allowance
- * indicates optional



G72 End Face Stock Removal Cycle

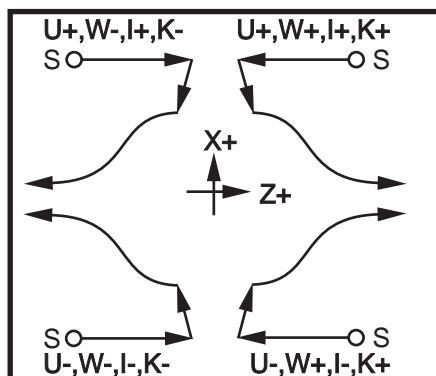
This canned cycle will remove material on a part given the finished part shape. It is similar to G71 but removes material along the face of a part. Define the shape of a part by programming the finished tool path and then use the G72 PQ block. Any F,S or T commands on the G72 line or in effect at the time of the G72 is used throughout the G72 roughing cycle. Usually a G70 call to the same PQ block definition is used to finish the shape.

Two types of machining paths are addressed with a G72 command. The first type of path (Type I) is when the Z-axis of the programmed path does not change direction. The second type of path (Type II) allows the Z-axis to change direction. For both the first type and the second type of programmed path the X-axis cannot change direction. If Setting 33 is set to FANUC, Type I is selected by having only an X-axis motion in the block specified by P in the G72 call. When both an X-axis and Z-axis motion are in the P block then Type II roughing is assumed. If Setting 33 is set to YASNAC, Type II is specified by including R1 on the G72 command block (Refer to Type II details).

The G72 consists of a roughing phase and a finishing phase. The roughing and finishing phase are handled slightly differently for Type I and Type II. Generally the roughing phase consists of repeated passes along the X-axis at the specified feed rate. The finishing phase consists of a pass along the programmed tool path to remove excess material left by the roughing phase but to leave finish material for a G70 block with perhaps a finishing tool. The final motion in either type is a return to the starting position S.

In the previous figure the start position S is the position of the tool at the time of the G72 call. The X clearance plane is derived from the X-axis start position and the sum of U and optional I finish allowances.

Any one of the four quadrants of the X-Z plane can be cut by specifying address codes I, K, U, and W properly. The following figure indicates the proper signs for these address codes to obtain the desired performance in the associated quadrants.



G72 Address Relationships



Type I Details

When Type I is specified by the programmer it is assumed that the Z-axis tool path does not reverse during a cut.

Each roughing pass Z-axis location is determined by applying the value specified in D to the current Z location. The nature of the movement along the X clearance plane for each roughing pass is determined by the G code in block P. If block P contains a G00 code, then movement along the X clearance plane is a rapid mode. If block P contains a G01 then movement will be at the G72 feed rate.

Each roughing pass is stopped before it intersects the programmed tool path allowing for both roughing and finishing allowances. The tool is then retracted from the material, at a 45 degree angle by the distance specified in Setting 73. The tool then moves in rapid mode to the X-axis clearance plane.

When roughing is completed the tool is moved parallel to the tool path to clean up the rough cut. If I and K are specified an additional rough finish cut parallel to the tool path is performed.

Type II Details

When Type II is specified by the programmer the Z axis PQ path is allowed to vary (for example, the Z-axis tool path can reverse direction).

The Z axis PQ path must not exceed the original starting location. The only exception is on the Q block.

Type II roughing when Setting 33 is set to YASNAC, must include R1 (with no decimal) on the G71 command block.

Type II, when Setting 33 is set to FANUC, must have a reference move, in both the X and Z axis, in the block specified by P.

Roughing is similar to Type I except after each pass, along the X axis, the tool will follow the path defined by PQ. The tool will then retract parallel to the Z axis by the distance defined in Setting 73 (Can Cycle Retraction). The Type II roughing method does not leave steps in the part prior to finish cutting and typically results in a better finish.

Note: a side effect of using a X finish or roughing allowance is the limit between the two cuts on one side of a trough and the corresponding point on the other side of the trough. This distance must be greater than double the sum of the roughing and finish allowances.

For example, if G72 type 2 path contains the following:

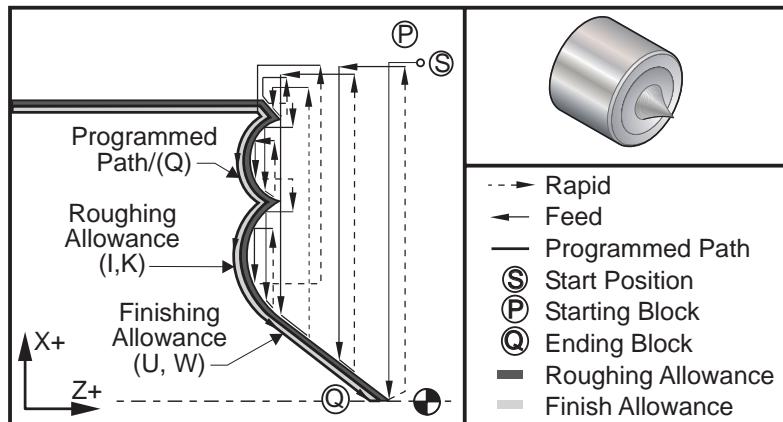
...
X-5.Z-5.
X-5.1Z-5.1
X-8.1Z-3.1

...

then the greatest allowance that can be specified is 0.999, since the horizontal distance from the start of cut 2 to the start point on cut 3 is 0.2. If a larger allowance is specified, overcutting will occur.

Cutter compensation is approximated by adjusting the roughing allowance according to the radius and tip type of the tool. Thus, the limitations that apply to the allowance also apply to the sum of the allowance and the tool radius.

CAUTION! If the last cut in the P-Q path is a non-monotonic curve, using a finish allowance, add a short retraction cut (do not use U).



G72 End Face Removal

Program Example

```
%  
00722  
T101  
S1000 M03  
G00 G54 X2.1 Z0.1  
G72 P1 Q2 D0.06 I0.02 K0.01 U0.02 W0.01 S1100 F0.015  
N1 G01 Z-0.46 X2.1 F0.005  
X2.  
G03 X1.9 Z-0.45 R0.2  
G01 X1.75 Z-0.4  
G02 X1.65 Z-0.4 R0.06  
G01 X1.5 Z-0.45  
G03 X1.3 Z-0.45 R0.12  
G01 X1.17 Z-0.41  
G02 X1.03 Z-0.41 R0.1  
G01 X0.9 Z-0.45  
G03 X0.42 Z-0.45 R0.19  
G03 X0.2 Z-0.3 R0.38  
N2 G01 X0.01 Z0  
G70 P1 Q2  
M05  
G28  
M30  
%
```

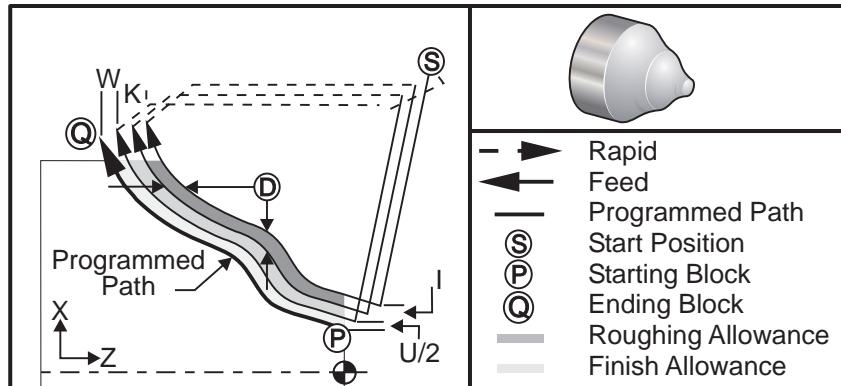
Description

(*G72 ROUGHING CYCLE*)

(*FINISH PASS*)

G73 Irregular Path Stock Removal Cycle (Group 00)

- | | |
|-----|--|
| D | Number of cutting passes, positive number |
| * F | Feed rate to use throughout G73 PQ block |
| I | X-axis distance and direction from first cut to last, radius |
| K | Z-axis distance and direction from first cut to last |
| P | Starting Block number of path to rough |
| Q | Ending Block number of path to rough |
| * S | Spindle speed to use throughout G73 PQ block |
| * T | Tool and offset to use throughout G73 PQ block |
| * U | X-axis size and direction of G73 finish allowance, diameter |
| * W | Z-axis size and direction of G73 finish allowance |
- * indicates optional



G73 Irregular Path Stock Removal Cycle

The G73 canned cycle can be used for rough cutting of preformed material such as castings. The canned cycle assumes that material has been relieved or is missing a certain known distance from the programmed tool path PQ.

Machining starts from the current position (S), and either rapids or feeds to the first rough cut. The nature of the approach move is based on whether a G00 or G01 is programmed in block P. Machining continues parallel to the programmed tool path. When block Q is reached a rapid departure move is executed to the Start position plus the offset for the second roughing pass. Roughing passes continue in this manner for the number of rough passes specified in D. After the last rough is completed, the tool returns to the starting position S.

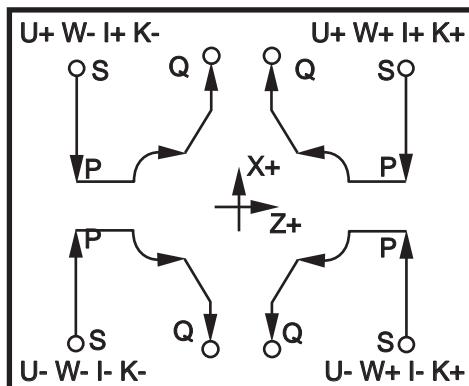
Only F, S and T prior to or in the G73 block are in effect. Any feed (F), spindle speed (S) or tool change (T) codes on the lines from P to Q are ignored.

The offset of the first rough cut is determined by $(U/2 + I)$ for the X axis, and by $(W + K)$ for the Z axis. Each successive roughing pass moves incrementally closer to the final roughing finish pass by an amount of $(I/(D-1))$ in the X axis, and by an amount of $(K/(D-1))$ in the Z axis. The last rough cut always leaves finish material allowance specified by U/2 for the X axis and W for the Z axis. This canned cycle is intended for use with the G70 finishing canned cycle.

The programmed tool path PQ does not have to be monotonic in X or Z, but care has to be taken to insure that existing material does not interfere with tool movement during approach and departure moves.

The value of D must be a positive integral number. If the D value includes a decimal, an alarm will be generated.

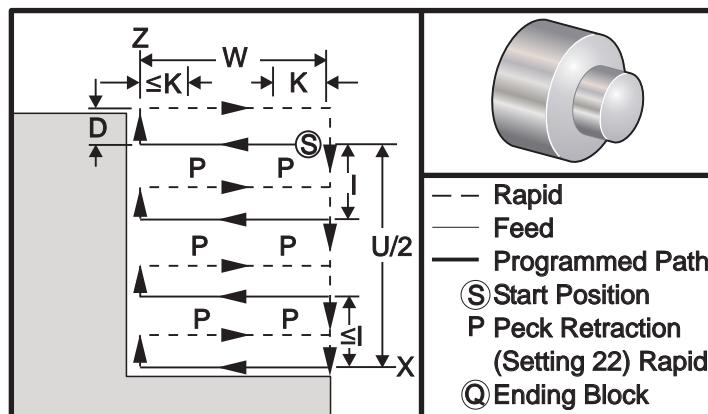
The four quadrants of the ZX plane can be machined if the following signs for U, I, W, and K are used.





G74 End Face Grooving Cycle, Peck Drilling (Group 00)

- * D Tool clearance when returning to starting plane, positive
- * F Feed rate
- * I X-axis size of increment between peck cycles, positive radius
- K Z-axis size of increment between pecks in a cycle
- * U X-axis incremental distance to furthest peck (diameter)
- W Z-axis incremental distance to total pecking depth
- * X X-axis absolute location of furthest peck cycle (diameter)
- Z Z-axis absolute location total pecking depth
- * indicates optional

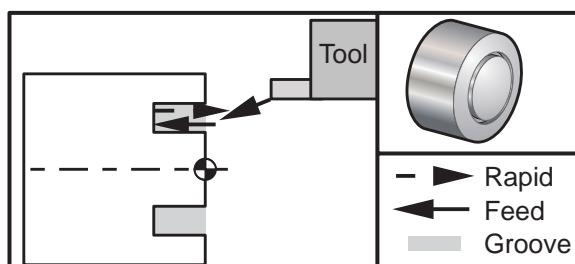


G74 End Face Grooving Cycle, Peck Drilling

The G74 canned cycle can be used for grooving on the face of a part for peck drilling, or for turning.

When an X, or U, code is added to a G74 block and X is not the current position, then a minimum of two pecking cycles will occur. One at the current location and another at the X location. The I code is the incremental distance between X axis pecking cycles. Adding an I will perform multiple, evenly spaced, pecking cycles between the starting position S and X. If the distance between S and X is not evenly divisible by I then the last interval will be less than I.

When K is added to a G74 block, then pecking will be performed at each interval specified by K, the peck is a rapid move opposite the direction of feed with a distance as defined by Setting 22. The D code can be used for grooving and turning to provide material clearance when returning to starting plane S.



G74 End Face Grooving Cycle

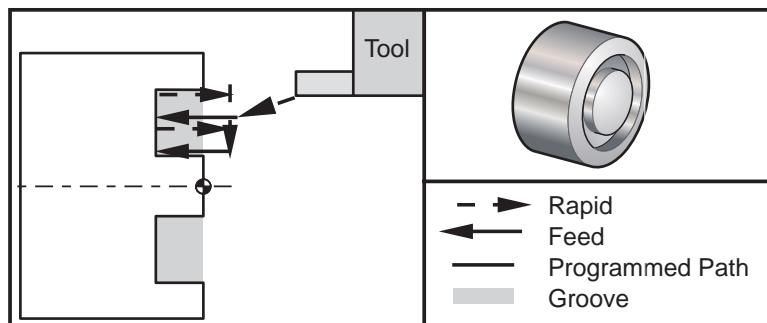


Program Example

```
%  
O0071  
T101  
G97 S750 M03  
G00 X3. Z0.05  
G74 Z-.5 K0.1 F0.01  
G28  
M30  
%
```

Description

(Rapid to Start position)
(Feed Z-.5 with a .100" peck)



G74 End Face Grooving Cycle (Multiple Pass)

Program Example

```
%  
O0074  
T101  
G97 S750 M03  
G00 X3. Z0.05  
G74 X1.75 Z-.5 I0.2 K0.1 F0.01  
G28  
M30  
%
```

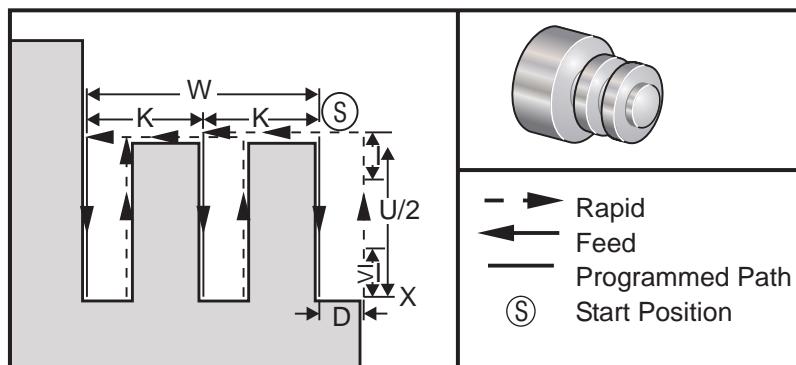
Description

(Rapid to Start position)
(Face grooving cycle multiple pass)



G75 O.D./I.D. Grooving Cycle (Group 00)

- * D Tool clearance when returning to starting plane, positive
 - * F Feed rate
 - * I X-axis size of increment between pecks in a cycle (radius measure)
 - * K Z-axis size of increment between peck cycles
 - * U X-axis incremental distance to total pecking depth
 - W Z-axis incremental distance to furthest peck cycle, signed
 - * X X-axis absolute location total pecking depth, signed diameter
 - Z Z-axis absolute location to furthest peck cycle, signed
- * indicates optional

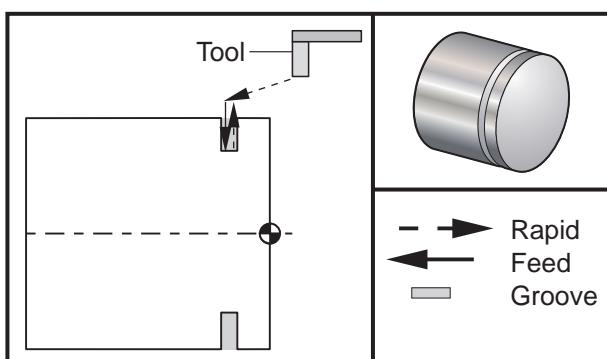


G75 O.D. / I.D. Grooving Cycle

The G75 canned cycle can be used for grooving an outside diameter.

When an Z, or W, code is added to a G75 block and Z is not the current position, then a minimum of two pecking cycles will occur. One at the current location and another at the Z location. The K code is the incremental distance between Z axis pecking cycles. Adding a K will perform multiple, evenly spaced, grooves. If the distance between the starting position and the total depth (Z) is not evenly divisible by K then the last interval along Z will be less than K.

Note that chip clearance is defined by Setting 22.





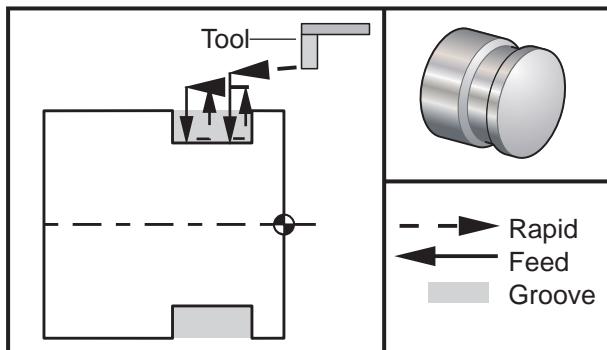
Program Example

```
%  
O0075  
T101  
G97 S750 M03  
G00 X4.1 Z0.05  
G01 Z-0.75 F0.05  
G75 X3.25 I0.1 F0.01  
G00 X5. Z0.1  
G28  
M30  
%
```

Description

(Rapid to Clear position)
(Feed to Groove location)
(O.D./I.D. Peck grooving single pass)

The following program is an example of a G75 program (Multiple Pass):



Program Example

```
%  
O0075  
T101  
G97 S750 M03  
G00 X4.1 Z0.05  
G01 Z-0.75 F0.05  
G75 X3.25 Z-1.75 I0.1 K0.2 F0.01  
G00 X5. Z0.1  
G28  
M30  
%
```

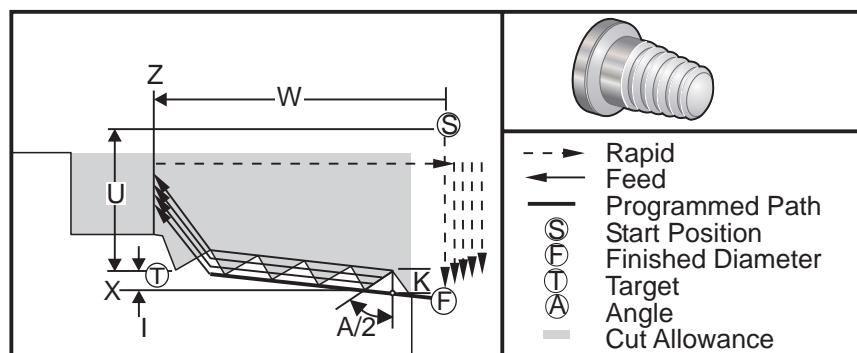
Description

(Rapid to Clear position)
(Feed to Groove location)
(O.D./I.D. Peck groove multiplepass)

G76 Threading Cycle, Multiple Pass (Group 00)

- * A Tool nose angle (value: 0 to 120 degrees) Do not use a decimal point
 - D First pass cutting depth
 - F(E) Feed rate, the lead of the thread
 - * I Thread taper amount, radius measure
 - K Thread height, defines thread depth, radius measure
 - * P Single Edge Cutting (load constant)
 - * Q Thread Start Angle (Do not use a decimal point)
 - * U X-axis incremental distance, start to maximum thread Depth Diameter
 - * W Z-axis incremental distance, start to maximum thread length
 - * X X-axis absolute location, maximum thread Depth Diameter
 - * Z Z-axis absolute location, maximum thread length
- * indicates optional

Settings 95 / 96 determine chamfer size / angle; M23 / 24 turn chamfering on / off.



The G76 canned cycle can be used for threading both straight or tapered (pipe) threads.

The height of the thread is defined as the distance from the crest of thread to the root of the thread. The calculated depth of thread (K) will be the value of K less the finish allowance (Setting 86 Thread Finish Allowance) is this amount.

The thread taper amount is specified in I. Thread taper is measured from the target position X, Z at point **T** to position **F**. Note: A conventional O.D. taper thread will have a negative I value.

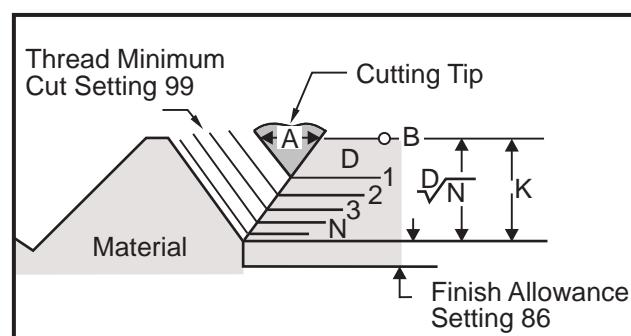
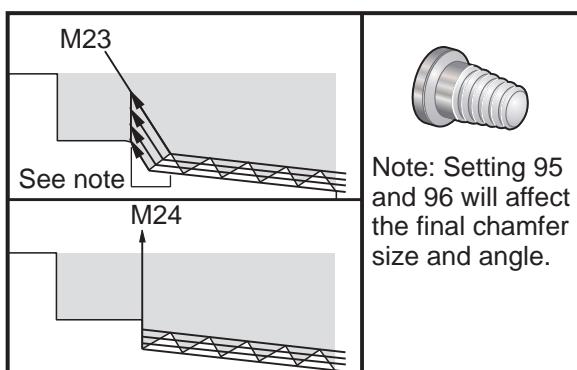
The depth of the first cut through the thread is specified in D.

The depth of the last cut through the thread can be controlled with Setting 86.

The tool nose angle for the thread is specified in A. The value can range from 0 to 30 degrees. If A is not used then 0 degrees is assumed.

The F code specifies the feed rate for threading. It is always good programming practice to specify G99 (feed per revolution) prior to a threading canned cycle. The F code also indicates the thread pitch or lead.

At the end of the thread an optional chamfer is performed. The size and angle of the chamfer is controlled with Setting 95 (Thread Chamfer Size) and Setting 96 (Thread Chamfer Angle). The chamfer size is designated in number of threads, so that if 1.000 is recorded in Setting 95 and the feed rate is .05, then the chamfer will be .05. A chamfer can improve the appearance and functionality of threads that must be machined up to a shoulder. If relief is provided for at the end of the thread then the chamfer can be eliminated by specifying 0.000 for the chamfer size in Setting 95. The default value for Setting 95 is 1.000 and the default angle for the thread (Setting 96) is 45 degrees.



G76 using an "A" Value



Notes:

P value must be included

Four options for G76 Multiple Thread Cutting is available

P1: Single edge cutting, cutting amount constant

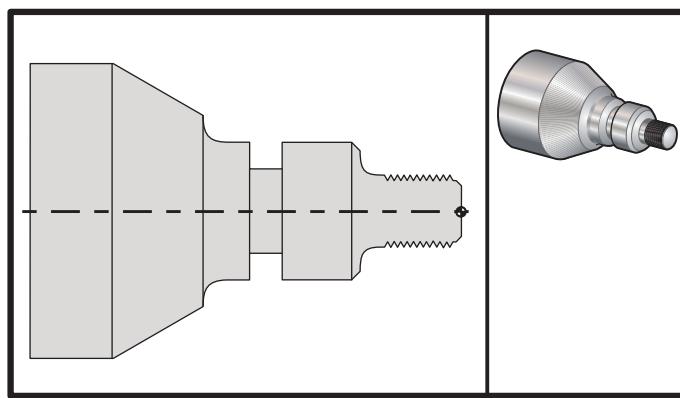
P2: Double edge cutting, cutting amount constant

P3: Single edge cutting, cutting depth constant

P4: Double edge cutting, cutting depth constant

P1 and P3 both allow for single edge threading, but the difference is that with P3 a constant depth cut is done with every pass. Similarly, P2 and P4 options allow for double edge cutting with P4 giving constant depth cut with every pass. Based on industry experience, double edge cutting option P2 may give superior threading results.

D specifies the depth of the first cut. Each successive cut is determined by the equation $D * \sqrt{N}$ where N is the Nth pass along the thread. The leading edge of the cutter does all of the cutting.



G76 Thread Cutting Cycle, Multiple Pass

Program Example

<u>Program Example</u>	<u>Description</u>
%	
T101	
G50 S2500	(Set max RPM select tool geometry)
G97 S1480 M03	(Spindle on select tool one offset one)
G54 G00 X3.1 Z0.5 M08	(Select work coord. and rapid to reference point, coolant on)
G96 S1200	(Constant surface speed ON)
G01 Z0 F0.01	(Position to part Z0)
X-0.04	
G00 X3.1 Z0.5	
G71P1 Q10 U0.035 W0.005 D0.125 F0.015	(Define roughing cycle)
N1 X0.875 Z0	(Begin tool path)
N2 G01 X1. Z-0.075 F0.006	
N3 Z-1.125	
N4 G02 X1.25 Z-1.25 R0.125	
N5 G01 X1.4	
N6 X1.5 Z-1.3	
N7 Z-2.25	
N8 G02 X1.9638 Z-2.4993 R0.25	
N9 G03X2.0172 Z-2.5172 R0.0325	



N10 G01 X3. Z-3.5
(End tool path)
G00 Z0.1 M09
G28
N20
(Thread sample program HAAS SL-Series FANUC System)
T505
G50 S2000
G97 S1200 M03
(Threading tool)
G00 X1.5 Z0.5
(Rapid to position)
M08
G00 X1.1 Z0.3
G76 X0.913 Z-0.85 K0.042 D0.0115 F0.0714 (Threading cycle)
G00 X1.5 Z0.5 G28 M09
N30
(HAAS SL-Series FANUC System)
T404
G50 S2500
G97 S1200 M03
(Groove tool)
G54 G00 X1.625 Z0.5 M08
G96 S800
G01 Z-1.906 F0.012
X1.47 F0.006
X1.51
W0.035
G01 W-0.035 U-0.07
G00 X1.51
W-0.035
G01 W0.035 U-0.07
X1.125
G01 X1.51
G00 X3. Z0.5 M09
G28
M30
%

Example Using Start Thread Angle (Q)

G76 X1.92 Z-2. Q60000 F0.2 D0.01 K0.04 (60 degree cut)
G76 X1.92 Z-2. Q120000 F0.2 D0.01 K0.04 (120 degree cut)
G76 X1.92 Z-2. Q270123 F0.2 D0.01 K0.04 (270.123 degree cut)

The following rules apply to the usage of Q:

1. The start angle, Q, must be specified every time it is used. If no value is specified then a zero (0) angle is assumed.
2. The angle of threading increment is 0.001 degrees and it cannot have any decimal point, for example a 180° angle must be specified as Q180000 and an angle of 35° as Q35000.
3. The Q angle must be entered as a positive value from 0 to 360000.

Mutiple Start Threading Example

Mutliple threads can be cut by changing the start point for each threading cycle.



The previous example has been modified to now create a multiple start thread. To calculate the additional start points the feed (F0.0714) is divided by the number of start points (3). $.0714 / 3 = .0238$. This value is then added to the initial Z-axis start point (line 2) in order to calculate the next start point (line 4). Add the same amount again to the previous start point (line 4) to calculate the next start point (line 6).

- (1) M08
- (2) G00 X1.1 Z0.5
- (3) G76 X0.913 Z-0.85 K0.042 D0.0115 F0.0714
- (4) G00 X1.1 Z0.5238
- (5) G76 X0.913 Z-0.85 K0.042 D0.0115 F0.0714
- (6) G00 X1.1 Z0.5476
- (7) G76 X0.913 Z-0.85 K0.042 D0.0115 F0.0714

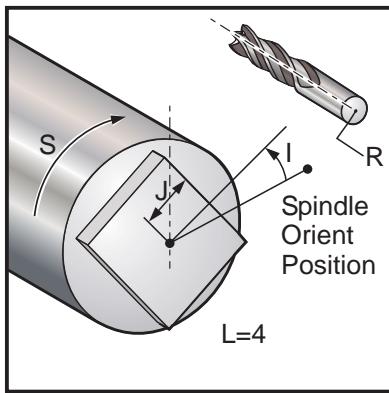
(Initial Start Point)
(Threading cycle)
(Next Start Point [.5 + .0238 = 5.238])
(Threading cycle)
(Last Start Point [.5238 + .0238 = 5.476])
(Threading cycle)

G77 Flattening Cycle (This G-code is optional and is used for live tooling) (Group 00)

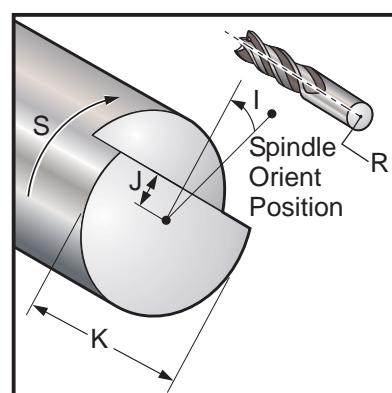
Note: This cycle is only available on lathes with the live tooling option.

- * I Angle of first flat, in degrees.
- J Distance from center to flat.
- * L Number of flat surfaces to cut
- R Tool Radius
- * S Spindle Speed
- * K Part Diameter

* indicates optional



G77 with L specified



G77 with K specified

The G77 canned cycle can be used to create one or more flat surfaces on a round part.

G77 operates in one of two modes, depending on whether a K code or an L code is specified. If a K code is specified, one flat surface will be cut. If an L code is specified, L flat surfaces will be cut, equally spaced all the way around the part. L must be greater than or equal to 3. If two sides are desired, perform two K cuts at I angle spacing.

The J value specifies the distance from the center of the part to the center of a flat surface. Specifying a larger distance will result in a shallower cut. This may be used to perform separate roughing and finishing passes. When using an L code, care should be taken to verify that the corner to corner size of the resulting part is not smaller than the diameter of the original part, or the tool may crash into the part during its approach.

The S value specifies the rpm speed that the spindle will maintain during the flattening cycle. The default value is 6. Higher values will not affect the flatness, but will affect the position of the flats. To calculate the maximum error in degrees, use $\text{RPM} * .006$.

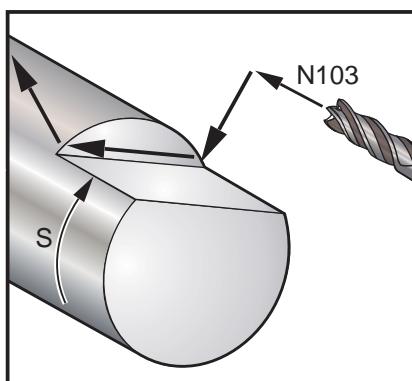
The L value allows a part with multiple flat surfaces to be specified. For example, L4 specifies a square, and L6 specifies a hex.



The I value specifies the offset of the center of the first flat surface from the zero position, in degrees. If the I value is not used, the first flat surface will start at the zero position. This is equivalent to specifying an I equal to half the number of degrees covered by the flat surface. For example, a square cut without an I value would be the same as a square cut with I set to 45.

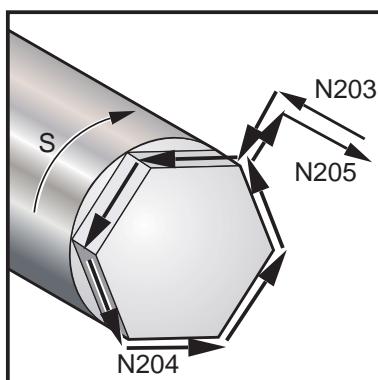
Flattening Examples with G77:

Cut a half-inch deep flat into the top inch of a part that is four inches in diameter, using a tool one inch in diameter:



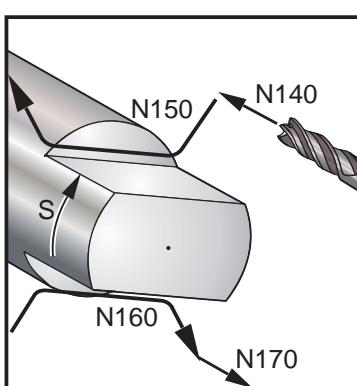
...
N100 S10 M03(Start spindle)
N101 M133 P1000 (Turn live tool)
N102 G00 X6.1
N103 Z-1.
N104 G77 J1.5 K4. R0.5
N105 Z1.
N106 M135 (Stop live tool)
N107 M05 (Stop spindle)
...

Cut a hexagon into the top half inch of a part that is three inches in diameter, using a tool half an inch in diameter:



...
N200 S10 M03(Start spindle)
N201 M133 P1000 (Turn live tool)
N202 G00 X4.5
N203 Z-0.05
N204 G77 J1.299 L6 R.25
N205 Z1.
N206 M135 (Stop live tool)
N207 M05 (Stop spindle)
...

Cut a 3/8" flat into the top and bottom of a part that is two inches in diameter, using a half inch diameter tool:



%
O00015 (Sample 2 Sided Flat Program)
N100 T606
N110 G97 S3 M03
N120 M133 P2000
N130 G00 X4. Z0.05
N140 Z-1.849
N150 G77 J0.625 I0 R0.25 K2.
(J=1.25 Flat Dia, I0=flat center, R.25=.5 dia end-mill, K=part stock dia)
N160 G77 J0.625 I180. R0.25 K2.
(J=1.25 flat dia, I180.=flat center, R.25=.5 dia end-mill, K=part stock dia)
N170 G00 Z1.
N180 M135
N190 M05
N200 G00 X10. Z12.
N210 M30
%



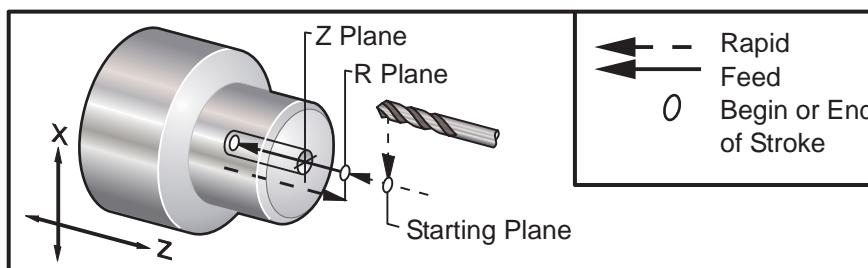
G80 Canned Cycle Cancel (Group 09*)

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

G81 Drill Canned Cycle (Group 09)

- F Feed Rate
- *L Number of repeats
- R Position of the R plane
- *W Z-axis incremental distance
- *X Optional X-axis motion command
- *Z Position of bottom of hole

* indicated optional



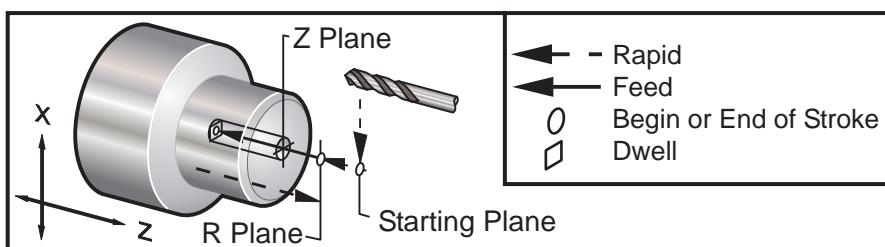
G81 Drill Canned Cycle

G82 Spot Drill Canned Cycle (Group 09)

- F Feed Rate
- *L Number of repeats
- P The dwell time at the bottom of the hole
- R Position of the R plane
- W Z-axis incremental distance
- *X X-axis motion command
- *Z Position of bottom of hole

* indicates optional

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



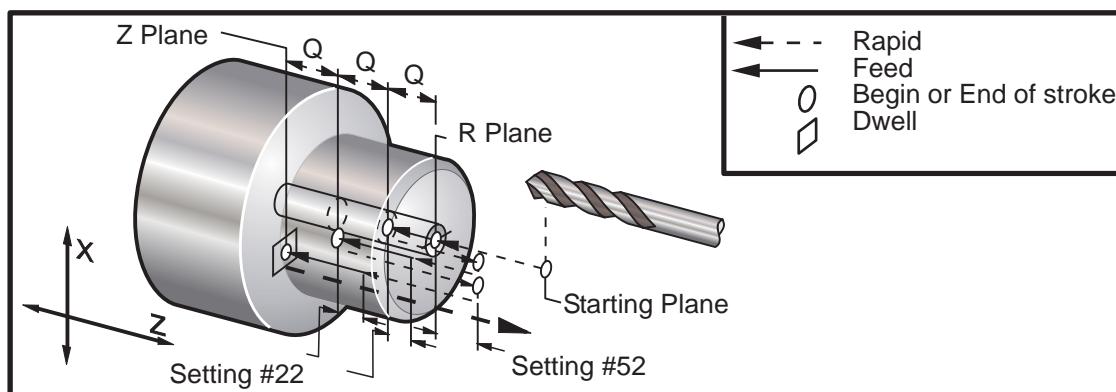
G82 Spot Drill Canned Cycle



G83 Normal Peck Drilling Canned Cycle (Group 09)

- F Feed Rate
- *I size of first cutting depth
- *J amount to reduce cutting depth each pass
- *K minimum depth of cut
- *L Number of repeats
- *P The dwell time at the bottom of the hole
- *Q The cut-in value, always incremental
- R Position of the R plane
- *W Z-axis incremental distance
- *X X-axis motion command
- *Z Position of bottom of hole

* indicates optional



G83 Peck Drilling Canned Cycle

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

Setting 52 changes the way G83 works when it returns to the R plane. Usually the R plane is set well outside the cut to insure that the chip clearing motion allows the chips to clear the hole, but this causes a wasted motion when first drilling through this "empty" space. If Setting 52 is set to the distance required to clear chips, the R plane can be put much closer to the part being drilled. When the clear move to R occurs, the Z will be moved past R by this value in setting 52.

Setting 22 is the amount to feed in Z to get back the same point at which the retraction occurred.

G84 Tapping Canned Cycle (Group 09)

- F Feed Rate
- R Position of the R plane
- *W Z-axis incremental distance
- *X X-axis motion command
- *Z Position of bottom of hole

* indicates optional

Programming Notes: It is not necessary to start the spindle CW before this canned cycle. The control does this automatically.

The feed rate for tapping is the lead of the thread. This is found by dividing 1 by the number of threads.

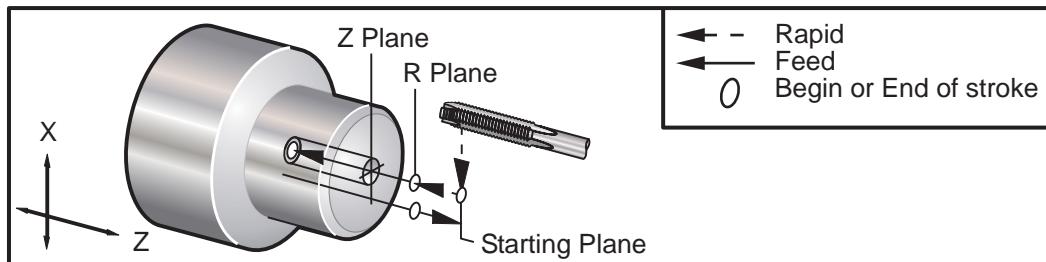
Example:

20 pitch 1/20 = .05 Feedrate
18 pitch 1/18 = .0555 Feedrate
16 pitch 1/16 = .0625 Feedrate



For Metric taps, divide the pitch by 25.4

Example: M6 x 1 = F.03937
 M8 x 1.25 = F.0492

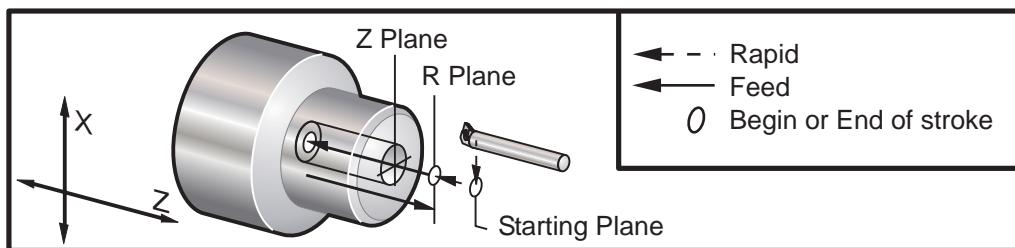


G84 Tapping Canned Cycle

G85 Boring Canned Cycle (Group 09)

F Feed Rate
*L Number of repeats
R Position of the R plane
*U X-axis incremental distance
*W Z-axis incremental distance
*X X-axis motion command
*Z Position of bottom of hole

* indicates optional



G85 Boring Canned Cycle

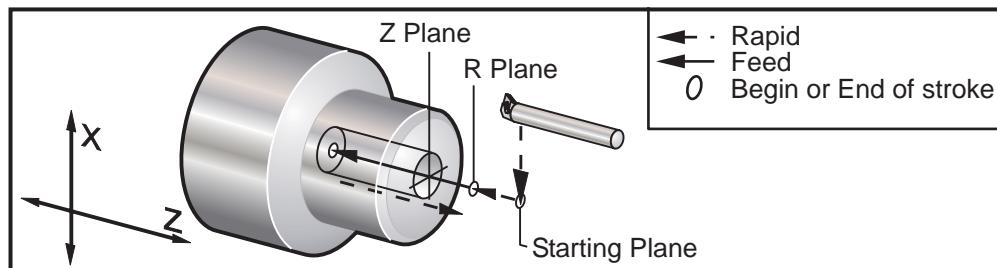
G86 Bore and Stop Canned Cycle (Group 09)

F Feed Rate
*L Number of repeats
R Position of the R plane
*U X-axis incremental distance
*W Z-axis incremental distance
*X X-axis motion command
*Z Position of bottom of hole

* indicates optional



Programming Note: The spindle will stop once the tool reaches the bottom of the hole. The will be retracted once the spindle has stopped.

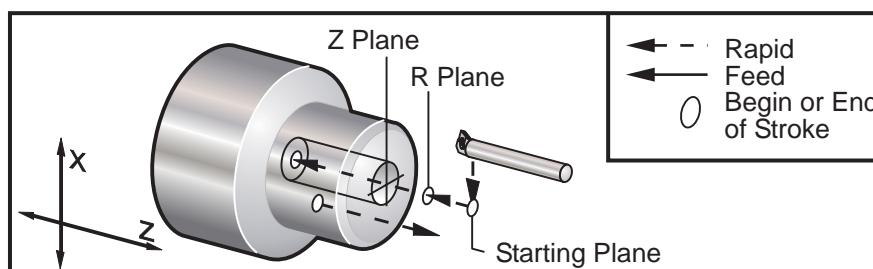


G86 Bore & Stop Canned Cycle

G87 Bore and Manual Retract Canned Cycle (Group 09)

- F Feed Rate
- *L Number of repeats
- R Position of the R plane
- *U X-axis incremental distance
- *W Z-axis incremental distance
- *X X-axis motion command
- *Z Position of bottom of hole

* indicates optional



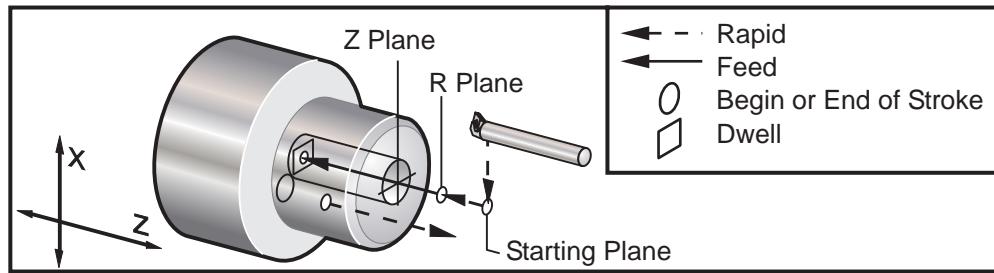
G87 Bore & Manual Retract Canned Cycle

G88 Bore and Dwell and Manual Retract Canned Cycle (Group 09)

- F Feed Rate
- *L Number of repeats
- P The dwell time at the bottom of the hole
- R Position of the R plane
- *U X-axis incremental distance
- *W Z-axis incremental distance
- *X X-axis motion command
- *Z Position of bottom of hole

* indicates optional

Programming Note: The tool will dwell at the bottom of the hole for the value of P, then the spindle stop. The tool will need to be retracted manually.

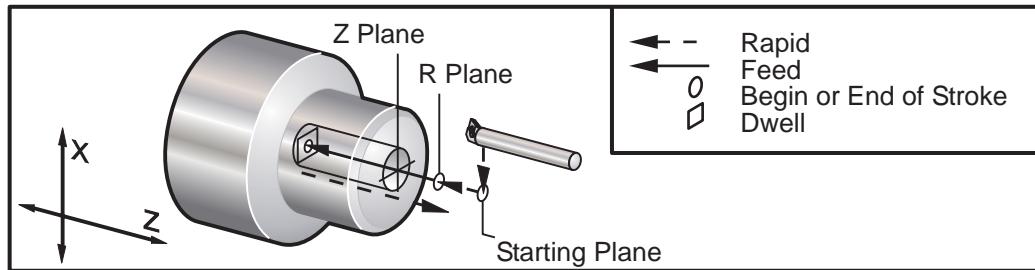


G88 Bore, Dwell & Manual Retract Canned Cycle

G89 Bore and Dwell Canned Cycle (Group 09)

- F Feed Rate
*L Number of repeats
P The dwell time at the bottom of the hole
R Position of the R plane
*U X-axis incremental distance
*W Z-axis incremental distance
*X X-axis motion command
*Z Position of bottom of hole

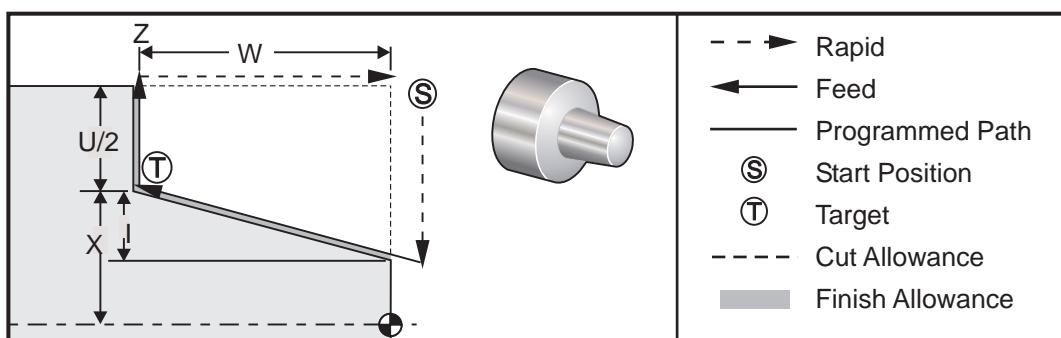
* indicates optional



G89 Bore & Dwell Canned Cycle

G90 O.D./I.D. Turning Cycle (Group 01)

- F(E) Feed rate
* I Optional distance and direction of X axis taper, radius
* U X-axis incremental distance to target, diameter
* W Z-axis incremental distance to target
X X-axis absolute location of target
Z Z-axis absolute location of target
* indicates optional

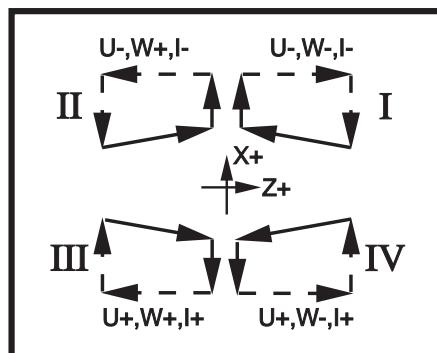


G90 is used for simple turning, however, multiple passes are possible by specifying the X locations of additional passes.



Straight turning cuts can be made by specifying X, Z and F. By adding an I value, a taper cut can be made. The amount of taper is referenced from the target. That is, I is added to the value of X at the target.

Any of the four ZX quadrants can be programmed using U, W, X, and Z; the taper can be positive or negative. The following figure gives a few examples of the values required for machining in each of the four quadrants.



G90-92 Address Relationships

G92 Threading Cycle (Group 01)

- F(E) Feed rate, the lead of the thread
- * I Optional distance and direction of X axis taper, radius
- * Q Start Thread Angle
- * U X-axis incremental distance to target, diameter
- * W Z-axis incremental distance to target
- X X-axis absolute location of target
- Z Z-axis absolute location of target
- * indicates optional

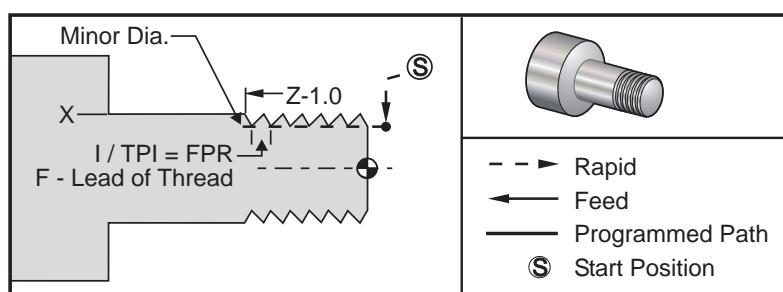
Programming Notes: Setting 95 / 96 determine chamfer size / angle M23 / 24 turn chamfering on / off.

G92 is used for simple threading, however, multiple passes for threading are possible by specifying the X locations of additional passes.

Straight threads can be made by specifying X, Z and F. By adding an I value, a pipe or taper thread can be cut. The amount of taper is referenced from the target. That is, I is added to the value of X at the target.

At the end of the thread, an automatic chamfer is cut before reaching the target; default for this chamfer is one thread at 45 degrees. These values can be changed with Setting 95 and Setting 96.

During incremental programming, the sign of the number following the U and W variables depends on the direction of the tool path. For example, if the direction of a path along the X-axis is negative, the value of U is negative.



G92 Threading Cycle



Program Example

```
%  
T101  
G50 S3000 M3  
G97 S1000  
X1.2 Z.2  
G92 X.980 Z-1.0 F.0833  
2nd pass X.965  
3rd pass X.955  
4th pass X.945  
5th pass X.935  
6th pass X.925  
7th pass X.917  
8th pass X.910  
9th pass X.905  
10th pass X.901  
11th pass X.899  
%  
(Rapid to Clear position)  
(Set up Threading cycle)  
(Subsequent Passes)  
"  
"  
"  
"  
"  
"  
"  
"  
"  
"  
(Subsequent Passes)
```

Example Using Start Thread Angle Q

G92 X-1.99 Z-2. Q60000 F0.2; (60 degree cut)
 G92 X-1.99 Z-2. Q120000 F0.2; (120 degree cut)
 G92 X-1.99 Z-2. Q270.123 F0.2; (270.123 degree cut)

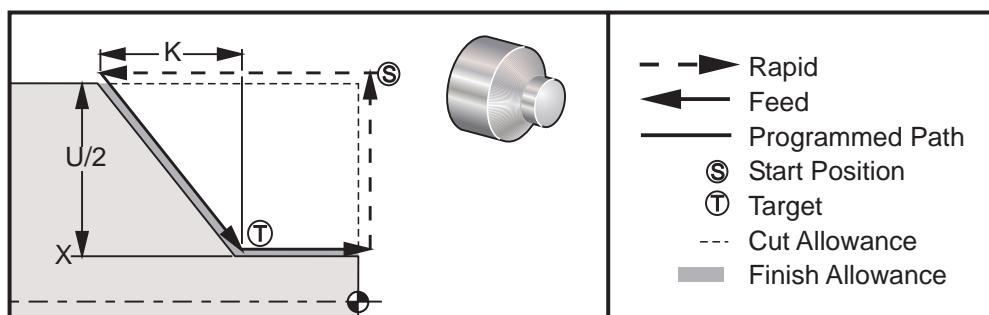
The following rules apply to the usage of Q:

1. The start angle, Q, must be specified every time it is used. If no value is specified then a zero (0) angle is assumed.
2. The angle of threading increment is 0.001 degrees and it cannot have any decimal point, for example, a 180° angle must be specified as Q180000 and an angle of 35° as Q35000.
3. The Q angle must be entered as a positive value from 0 to 360000.

In general, when multi-threads are being performed it is a good practice to achieve the depth of the threads at a uniform level across all the threading angles. One way to achieve this is to make a sub-program that will only cause the Z-axis to move for the different angles of threading. After the sub-program has finished change the X-axis depth and call the sub-program again.

G94 End Facing Cycle (Group 01)

F(E) Feed rate
 * K Optional distance and direction of Z axis coning
 * U X-axis incremental distance to target, diameter
 * W Z-axis incremental distance to target
 X X-axis absolute location of target
 Z Z-axis absolute location of target
 * indicates optional



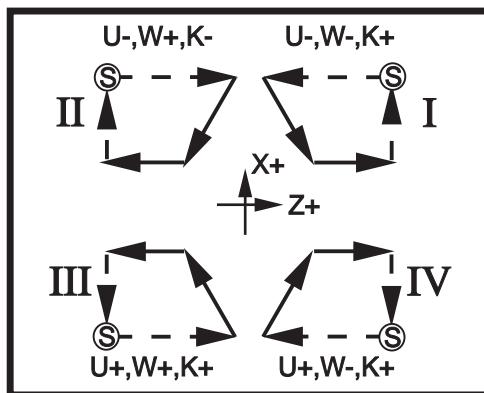
G94 End Face Turning Cycle



Straight end facing cuts can be made by just specifying X, Z and F. By adding K a cone-shaped face can be cut. The amount of coning is referenced from the target. That is K is added to the value of X at the target.

Any of the four ZX quadrants can be programmed by varying U, W, X, and Z. The coning can be positive or negative. The following figure gives a few examples of the values required for machining in each of the four quadrants.

During incremental programming, the sign of the number following the U and W variables depends on the direction of the tool path. For example, if the direction of a path along the X-axis is negative, the value of U is negative.



G94 Address Relationships

G95 Live Tooling Rigid Tap (Face) (Group 09)

F	Feed Rate
R	Position of the R plane
W	Z-axis incremental distance
X	Optional Part Diameter X-axis motion command
Z	Position of bottom of hole

G95 Live Tooling Rigid Tapping is similar to G84 Rigid Tapping in that it uses the F, R, X and Z addresses, however, it has the following differences:

1. The main spindle must be clamped (use M14) before G95 is commanded.
2. The control must be in G99 Feed per Revolution mode in order for tapping to work properly.
3. An S (spindle speed) command must have been issued prior to the G95.
4. The X axis must be positioned between zero and the center of the main spindle, do not position beyond spindle center.

Below is a brief program example of G95:

```
%  
O00800  
N1 T101 (Axial 1/4-20 Tap)  
G99 (Necessary or This Cycle)  
G00 Z0.5  
X2.5  
S500 (RPM Should Look Like This, CW Direction)  
M19PXX (Orient Spindle at Desired Location)  
M14(Clamp Spindle)  
G95 Z-.500 R.25 F0.05 (Thread Down .50 Deep)  
G28 U0
```



G28 W0
M135 (Stop Live Tooling Spindle)
M15 (Unlock Spindle)
M30
%

G96 Constant Surface Speed ON (Group 13)

This command tells the control to maintain a constant cutting speed. This means, as the workpiece gets smaller the spindle speed increases. Surface speed is based on the distance of the tool tip to the spindle center (radius of cut). The current S code is used to determine the surface speed. The value of S indicates inches per revolution of the spindle when Setting 9 is set to Inch, while S indicates millimeters per revolution of the spindle when Setting 9 is set to Metric.

G97 Constant Surface Speed OFF (Group 13)

This command tells the control to NOT adjust the spindle speed based on the radius of cut and used to cancel any G96 command. When G97 is in effect, any S command is revolution per minute (RPM).

G98 Feed Per Minute (Group 10)

This command changes how the F address code is interpreted. The value of F indicates inches per minute when Setting 9 is set to Inch, and F indicates millimeters per minute when Setting 9 is set to Metric.

G99 Feed Per Revolution (Group 10)

This command changes how the F address is interpreted. The value of F indicates inches per revolution of the spindle when Setting 9 is set to Inch, while F indicates millimeters per revolution of the spindle when Setting 9 is set to Metric.

G100 Disable Mirror Image (Group 00)

G101 Enable Mirror Image (Group 00)

X Optional X-axis command
Z Optional Z-axis command

At least one is required.

Programmable mirror image can be turned on or off individually for the X and/or Z axis. The bottom of the screen will indicate when an axis is mirrored. These **G** codes should be used in a command block without any other **G** codes and will not cause any axis motion. G101 will turn on mirror image for any axis listed in that block. G100 will turn off mirror image for any axis listed in the block. The actual value given for the **X** or **Z** code has no effect; G100 or G101 by itself will have no effect. For example G101 X 0 turns on X-axis mirror

Note that settings 45 through 48 may be used to manually select mirror image.

G102 Programmable Output to RS-232 (Group 00)

*X X-axis command
*Z Z-axis command

* indicates optional

Programmable output to the first RS-232 port send the current work coordinates of the axes to another computer. This G code should be used in a command block without any other G codes; it will not cause any axis motion.

Programming note: Optional spaces (Setting 41) and EOB control (Setting 25) are applied.

Digitizing of a part is possible using this G code and a program which steps over a part in X-Z and probes across in Z with a G31. When the probe hits, the next block could be a G102 to send the X and Z position out to a computer which could store the coordinates as a digitized part. Additional software for the personal computer is required to complete this function.



G103 Limit Block Lookahead (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.

G 105 Servo Bar Command

Bar Feed Command. See the Haas bar feeder manual.

G110,G111 and G114-G129 Coordinate System (Group 12)

These codes select one of the additional user coordinate systems. All subsequent references to axes positions will be interpreted in the new coordinate system. Operation of G110 to G129 are the same as G54 to G59.

G112 XY to XC interpretation (Group 04)

The G112 Cartesian to Polar coordinate transformation feature allows the user to program subsequent blocks in Cartesian XY coordinates, which the control automatically converts to polar XC coordinates. While it is active, G18 ZX plane is used for G01 linear XY strokes and G17 YZ plane is used for G02 and G03 XY interpolated circular motion.

Note that mill-style Cutter Compensation becomes active when G112 is used. Cutter Compensation (G41, G42) must be canceled (G40) before the next G113.

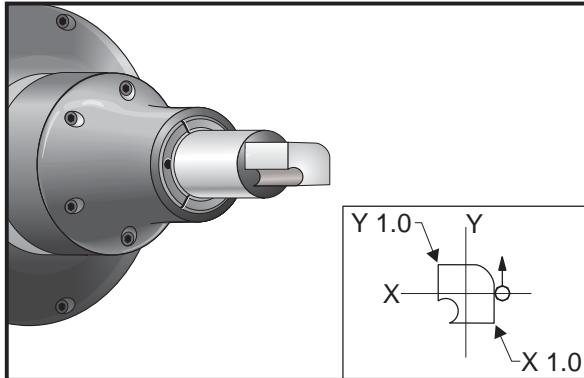
G113 G112 Cancel (Group 04)

G113 cancels the Cartesian to Polar coordinate conversion.



G112 Program Example

```
%  
T0101  
G54  
G17  
G112  
M154  
G0G98Z.1  
G0X.875Y0.  
M8  
G97P2500M133  
G1Z0.F15.  
Y.5F5.  
G3X.25Y1.125R.625  
G1X-.75  
G3X-.875Y1.R.125  
G1Y-.25  
G3X-.75Y-.375R.125  
G2X-.375Y-.75R.375  
G1Y-1.  
G3X-.25Y-1.125R.125  
G1X.75  
G3X.875Y-1.R.125  
G1Y0.  
G0Z.1  
G113  
G18  
M9  
M155  
M135  
G28U0.  
G28W0.H0.  
M30  
%
```



G154 Select Work Coordinates P1-99 (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

G159 Background Pickup / Part Return

Automatic Parts Loader (APL) Command. See the Haas APL manual.

G160 APL Axis Command Mode On

Automatic Parts Loader Command. See the Haas APL manual.

G161 APL Axis Command Mode Off

Automatic Parts Loader Command. See the Haas APL manual.

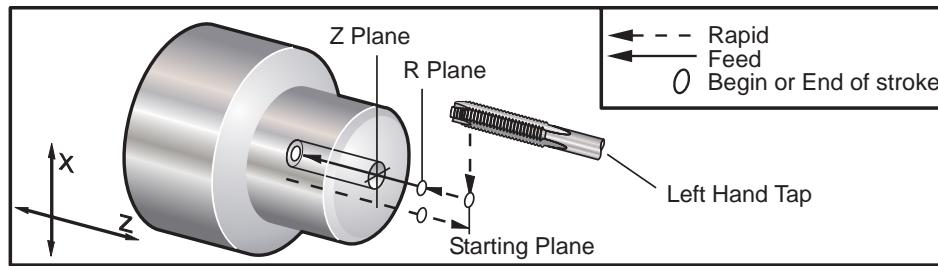
G184 Reverse Tapping Canned Cycle For Left Hand Threads (Group 09)

F Feed Rate in inches (mm) per minute
R Position of the R plane
*W Z-axis incremental distance
*X X-axis motion command
*Z Position of bottom of hole

* indicates optional

Programming Notes: When tapping, the feedrate is the lead of the thread. See example of G84.

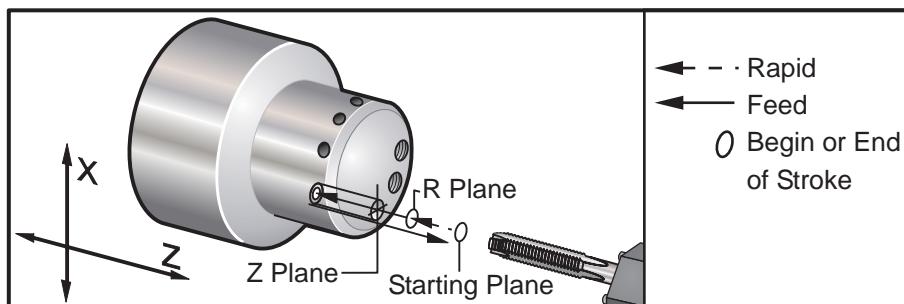
It is not necessary to start the spindle CCW before this canned cycle; the control does this automatically.



G184 Tapping Canned Cycle

G186 Rev Live Tool Rig Tap (Group 09)

F Feed Rate
R Position of the R plane
W Z-axis incremental distance
X Optional Part Diameter X-axis motion command
Z Position of bottom of hole



G95 / G186 Live Tooling Rigid Tapping (Face)

it is not necessary to start the spindle CW before this canned cycle; the control does this automatically.

The feed rate for tapping is the lead of the thread. This is found by dividing 1 by the number of threads.

Example: 20 pitch 1/20 = .05 Feedrate
 18 pitch 1/18 = .0555 Feedrate
 16 pitch 1/16 = .0625 Feedrate

For Metric taps, divide the pitch by 25.4

Example: M6 x 1 = F.03937
 M8 x 1.25 = F.0492

G187 Accuracy Control (Group 00)

Programming G187 is as follows:

G187 E0.01 (to set value)
G187 (to revert to setting 85 value)

The G187 code is used to select the accuracy with which corners are machined. The form for using G 187 is G187 Ennnn, where nnnn is the desired accuracy.



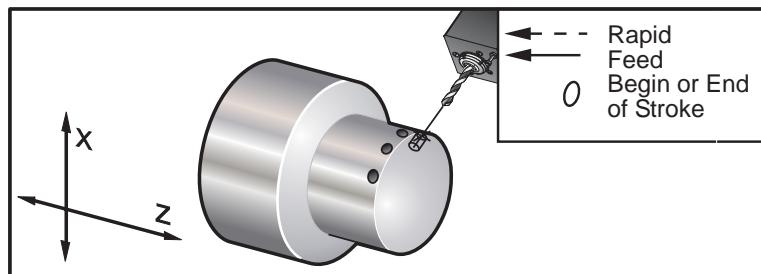
G195 Live Tool Radial Tapping (Diameter) (Group 00)

F Feed Rate in inches (mm) per minute
 R Position of the R plane
 *U X-axis incremental distance
 *X X-axis motion command
 *Z Position of bottom of hole

G196 Reverse Live Tool Vector Tapping (Diameter) Group 00

F Feed Rate in inches (mm) per minute
 R Position of the R plane
 *U X-axis incremental distance
 *X X-axis motion command
 *Z Position of bottom of hole

These G codes perform live tooling radial or vector tapping on a lathe; they do not permit an "R" plane.



G195 / G196 Live Tooling Rigid Tapping (Diameter)

Below is a brief program example of G195

```
%  

O00800  

N1 T101 (RADIAL 1/4-20 TAP)  

G99 (Neccesary for this cycle)  

G00 Z0.5  

X2.5  

Z-0.7  

S500 (rpm should look like this, cw direction)**  

M19PXX (Orient spindle at desired location)  

M14(Lock spindle up)  

G195 X1.7 F0.05 (thread down to X1.7)  

G28 U0  

G28 W0  

M135 (Stop Live tooling spindle)  

M15 (Unlock Spindle brake)  

M30  

%
```

G200 Index on the Fly (Group 00)

U Optional relative move in X to tool change position
 W Optional relative move in Z to tool change position
 X Optional final X position
 Z Optional final Z position
 T Required tool number and offset number in standard form

This G code will cause the lathe to change tools while performing a rapid move away from and back to the part, to save time.

Example: G200 T202 U0.5 W0.5 X8. Z2.

U and W specify a relative motion in X and Z, which is performed as the tool turret unseats. X and Z specify the position to move to as the tool turret reseats. Both motions are rapid.



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, S1500 will command a spindle speed of 1500RPM.

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 M88/89 for High Pressure Coolant.

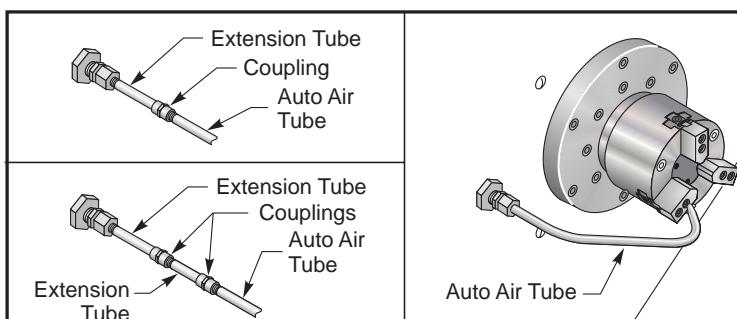
Note: Coolant status is checked only at the start of a program, so a low coolant condition will not stop a program that is already running.

M10 Clamp Chuck / M11 Unclamp Chuck

The M10 is used to clamp the chuck and M11 will unclamp the chuck. If the spindle is turning, it will be stopped before the chuck is unclamped.

M12 Auto Air Jet On (Optional) / M13 Auto Air Jet Off (Optional)

The M12 and M13 codes are used to activate the optional Auto Air Jet. M12 turns the air blast on and M13 turns the air blast off. Additionally, M12 Pnnn (nnn is in milliseconds) will turn it on for the specified time, then turn off automatically.



M14 Clamp Main Spindle / M15 Unclamp Main Spindle

M14 will clamp the main spindle. M15 will unclamp the main spindle.



M17 Turret Rotation Always Forward / M18 Turret Rotation Always Reverse

The M17 and M18 codes rotate the turret in the forward (M17) or reverse (M18) direction when a tool change is made. M17 and M18 work with other M-codes in the same block. The following M17 program code will cause the tool turret to move forward to tool 1 or reverse to tool 1 if an M18 is commanded.

Forward: N1 T0101 M17; Reverse: N1 T0101 M18;

An M17 or M18 will stay in affect for the remainder of the program. Note that Setting 97, Tool Change Direction, must be set to M17/M18.

M19 Orient Spindle (P and R values are an optional feature)

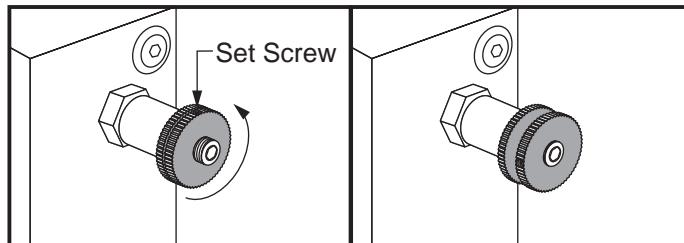
The M19 code is used to adjust the spindle to a fixed position.

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 Tailstock Forward / M22 Tailstock Reverse

M21 and M22 are used to position the tailstock. M21 uses Settings 105, 106 and 107 to move to the tailstock Hold Point. M22 uses Setting 107 to move the tailstock to the Retract Point. Adjust pressure using the valves on the HPU.

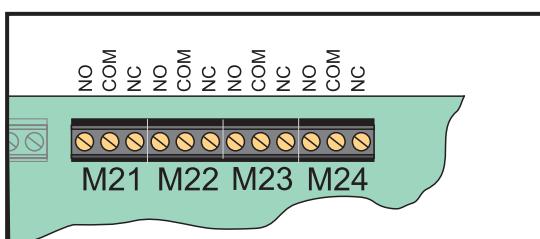


M21-M28 Optional User M Function with M-Fin

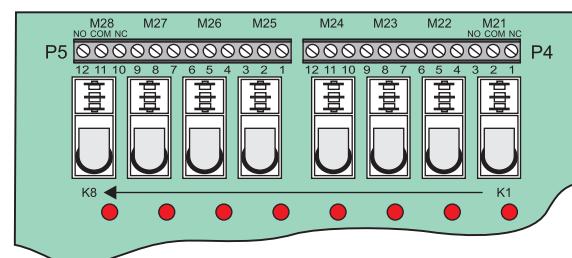
The M codes M21 through M28 are optional for user relays; each M code will activate one of the optional relays. The Reset button will terminate any operation that is waiting for a relay-activated accessory to finish.

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.

NOTE: Some or all of the M21-25 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.

M23 Thread Chamfer ON/M24 Thread Chamfer OFF

The M23 code commands the control to execute a chamfer at the end of a thread executed by G76 or G92. The M24 code commands the control not to perform chamfering at the end of the threading cycles (G76 or G92). An M23 remains in effect until changed by M24, likewise for an M24. Refer to Settings 95 and 96 to control the chamfer size and angle. M23 is the default at power-up and when the control is reset.

M30 Program End and Rewind

The M30 code is used to stop a program. It stops the spindle and turns off the coolant 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.

M36 Parts Catcher Up (Optional) / M37 Parts Catcher Down (Optional)

The M36 code is used to activate the optional parts catcher. The M37 code is used to deactivate the optional parts catcher. The M36 will rotate the parts catcher into position to catch a part. The M37 will rotate the parts catcher out of the work envelope.

M38 Spindle Speed Variation On / M39 Spindle Speed Variation Off

Spindle Speed Variation (SSV) allows the operator to specify a range within which the spindle speed will continuously vary. This is helpful in suppressing tool chatter, which can lead to undesirable part finish and/or damage the cutting tool. The control will vary the spindle speed based on Settings 165 and 166. For example: In order to vary spindle speed +/- 50 RPM from its current commanded speed with a duty cycle of 3 seconds, Setting 165 should be set to 50 and Setting 166 should be set to 30. With these settings, the following program will vary the spindle speed between 950 and 1050 RPM after the M38 command.

```
O0010;  
S1000 M3  
G4 P3.  
M38    (SSV ON)  
G4 P60.  
M39    (SSV OFF)  
G4 P5.  
M30
```

The spindle speed will continuously vary with a duty cycle of 3 seconds until an M39 command is found. At that point the machine will come back to its commanded speed and the SSV mode will be turned off. A program stop command such as M30 or pressing Reset also turns SSV Off. If the RPM swing is larger than the commanded speed value, then any negative swing (below zero) will translate into an equivalent value in the plus region. The spindle, however, will not be allowed to go below 10 RPM when SSV mode is active. Constant Surface Speed, Threading & Tapping: When Constant Surface Speed (G96) is activated (which will calculate spindle speed) the M38 command will alter that value using settings 165 and 166. Similarly, threading operations such as G92, G76 and G32 will allow the spindle speed to vary in SSV mode. Tapping cycles (G84) however will be executed at their com-



manded speed and SSV will NOT be applied.

M41 Low Gear/ M42 High Gear

On machines with a transmission the M41 command is used to select low gear and an M42 will select high gear.

M43 Turret Unlock / M44 Turret Lock

For Service use only.

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 M121-M128 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 M121-M128 for details on the M-Code relays.

M69 Clear Output Relay

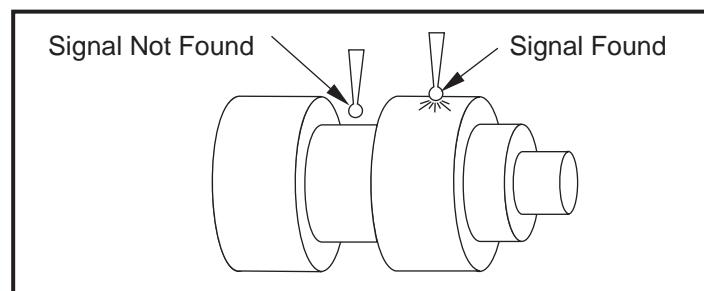
This M code turns off a relay. An example of its usage is **M69 Pnn**, where "nn" is the number of the relay being turned off. An M69 command can be used to turn off any of the output relays in the range from 1100 to 1155. When using Macros, M69 P1103 does the same thing as using the optional macro command #1103=0, except that it is processed at the end of the line of code.

M76 Disable Display / M77 Enable Display

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 / M79 Alarm if Skip Signal Not Found

This M-code is used with a probe. An M78 will generate an alarm if a programmed skip function (G31) receives a signal from the probe. This is used when a skip signal is not expected, and may indicate a probe crash. An M79 will generate an alarm if a programmed skip function (G31) did not receive a signal from the probe. This is used when the lack of the skip signal means a probe positioning error. These codes can be placed on the same line as the skip G-code or in any block after.



M85 Open Automatic Door (Optional) / M86 Close Automatic Door (Optional)



The M85 opens the Auto Door and the M86 closes the Auto Door. The control pendant will beep when the door is in motion.

M88 High Pressure Coolant On (Optional) / M89 High Pressure Coolant Off (Optional)

The M88 code is used to turn on the high pressure coolant option, an M89 turns the coolant off. Warning! Turn off High Pressure Coolant before performing a tool change. Use M89 to turn off High Pressure coolant during program execution before rotating the tool turret.

M93 Start Axis Pos Capture / M94 Stop Axis Pos Capture

These M codes permit the control to capture the position of an auxiliary axis when a discrete input changes to a 1. The format is:

M93 Px Qx P is the axis number. Q is a discrete input number from 0 to 63.
M94

M93 causes the control to watch the discrete input specified by the Q value, and when it goes to a 1, it captures the position of the axis specified by the P value. The position is then copied to hidden macro variables 749. M94 stops the capture. M93 and M94 were introduced to support the Haas Bar Feeder, which uses a single axis controller to the V auxiliary axis. P5 (V axis) and Q2 must be used for the bar feeder.

M95 Sleep Mode

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 P5 Q8 (Test input #5, Door Switch, until closed);
N10 (Start of program loop);
.
. (Program that machines part);

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:

```
D0  
M97 P1000 L2      (L2 command will cause the program to run the N1000 line twice)  
D1  
M97 P1000  
M30  
N1000 G00 G90 G55 X0 Z0      (N line that will run after M97 P1000 is run)  
S500 M03  
G00 Z-.5  
G01 X.5 F100.  
G03 ZI-.5  
G01 X0  
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-program Number)  
. . .  
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
	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=dddd in the subroutine and then using **M99 P#nnn** after the subroutine call.

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
```

M119 Subspindle Orient

This command will cause the subspindle to be oriented to the position specified by the P or R command. The format is:

```
M119 Pxxx
M119 Rx.x
```

M121-M128 Optional User M

The M121 through M128 codes are optional for user interfaces. They will activate one of relays 1132 through 1139, wait for the M-fin signal, release the relay, and wait for the M-fin signal to cease. The RESET button will terminate any operation that is hung-up waiting for M-fin.

M133 / M134 / M135 Live Tool Drive Commands



M133 turns the live tool spindle in the forward direction

M134 turns the live tool spindle in the reverse direction

M135 Stops the live tool spindle

Spindle speed is controlled with an P address code, for example, P1200 will command a spindle speed of 1200RPM.

M143 / M144 / M145 Sub Spindle Commands (Optional)

M143 turns the sub spindle in the forward direction

M144 turns the sub spindle in the reverse direction

M145 Stops the sub spindle

The sub spindle speed is controlled with an P address code, for example, P1200 will command a spindle speed of 1200RPM.

*Main spindle M codes that can be used to command the sub-spindle while the control is in G14 mode.

M154 C-Axis Engage / M155 C-Axis Disengage (Optional)

This M code is used to engage or disengage the optional C-axis motor.





SETTINGS

The setting pages contain values that the user may need to change and that control machine operation. Most settings can be changed by the operator. The settings are preceded by a short description on the left and the value on the right. In general, settings allow the operator or setup person to lock out or turn on specific functions.

The settings are organized into pages of functionally similar groupings. This will make it easier for the user to remember where the settings are located and reduces the amount of time spent maneuvering through the settings display. The list below is separated into page groups with the page title as the heading.

Use the vertical cursor keys to move to the desired setting. Depending on the setting, you may change it by entering a new number or, if the setting has specific values, press the horizontal cursor keys to display the choices. Press the WRITE key to enter or change the value. The message near the top of the screen tells you how to change the selected setting.

The serial number is Setting 26 on this page and is protected from user change. If you need to change this setting, contact HAAS or your dealer.

The following is a detailed description of each of the settings:

1 Auto Power Off Timer

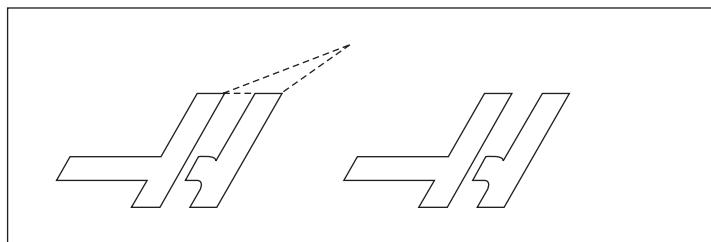
This setting is used to power-down the machine when it has not been used for some time. The value entered in this setting is the number of minutes the machine will remain idle until it is powered down. The machine will not be powered down while a program is running, and the time (number of minutes) will start back at zero anytime a button 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.

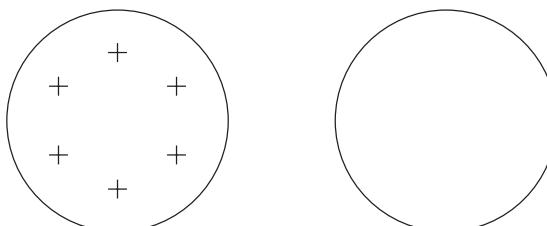
4 Graphics Rapid Path

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.

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.**

28 Can Cycle Act w/o X/Z

Turning this setting "ON" will cause the commanded canned cycle to complete without an X or Z 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 Z axis move.

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 tool shift offsets work. It can be set to either Yasnac or Fanuc. This setting changes the way a Txxxx command is interpreted and the way the coordinate system is specified. If it is Yasnac, tool shifts 51 to 100 are available on the offsets display and G50 T5100 is allowed. If it is FANUC, tool geometry for tools 1 to 50 is available on the offsets display and G54 style work coordinates are available.

36 Program Restart

When this setting is ON, 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	M37 Parts Catcher Off
M09 Coolant Off	M41 Low Gear
M14 Clmp Main Spndl	M42 High Gear
M15 Unclmp Main Spndl	M51-58 Set User M
M36 Parts Catcher On	M61-68 Clear User M

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 is a numeric entry between 0 and 1. It 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 V-axis.

39 Beep @ 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.



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 Tool Change

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 TNC %

(Minimum feedrate in radius tool nose compensation percent) This setting affects the feed rate when cutter compensation moves the tool towards the inside of a circular cut. This type of cut will slow down to maintain a constant surface feed rate. This setting specifies the slowest feed rate as a percentage of the programmed feed rate (range 1-100).

45 Mirror Image X-axis

47 Mirror Image Z-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.

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)

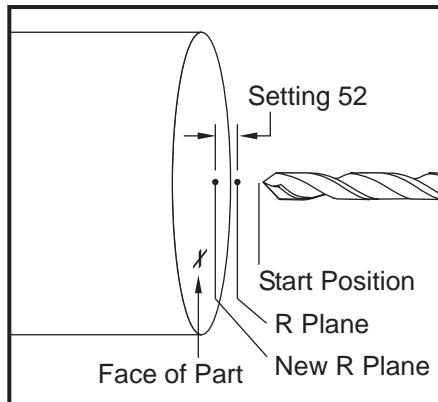
Selecting "Off" will not allow a program to start when the doors are open, and opening a door will cause the running program to stop (the same as pressing Feed Hold).

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 9.9999 inches or 0-254mm). This setting changes the way G83 (peck drilling cycle) behave. 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.

57 Exact Stop Canned X-Z

The rapid XZ 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 XZ 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+

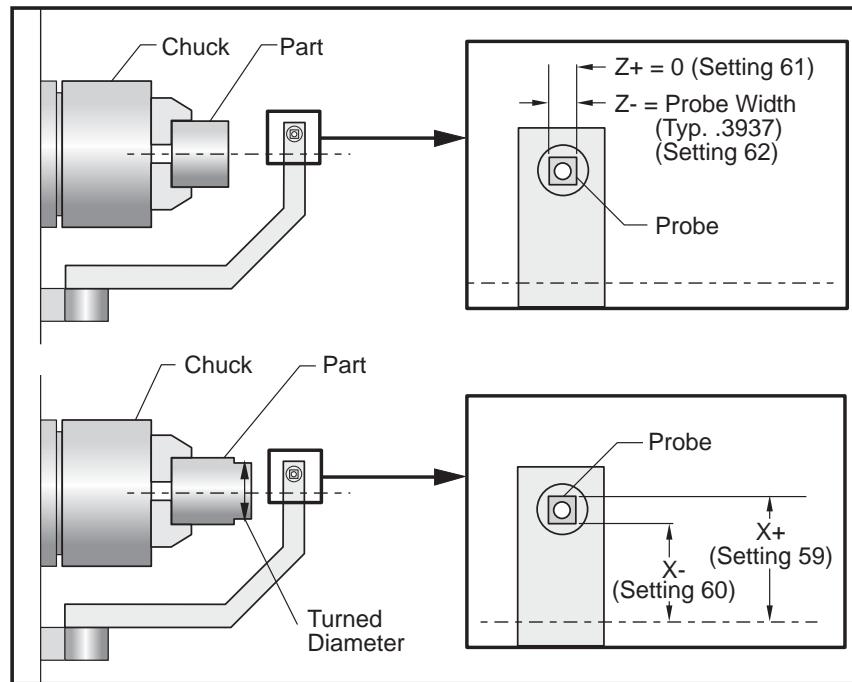
60 Probe Offset X-

61 Probe Offset Z+

62 Probe Offset Z-

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 Offset 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:

$$\text{Total Y travel} = \text{Parameter 20} / \text{Parameter 19}$$

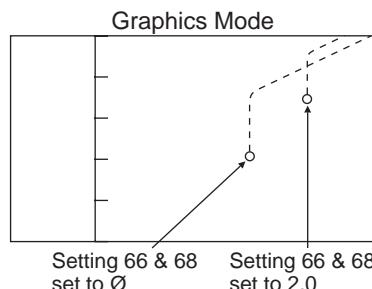
$$\text{Scale} = \text{Total Y travel} / \text{Setting 65}$$

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.

68 Graphics Z Offset

This setting locates the top of the zoom window relative to the machine Z 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 controls whether the POPEN and PCLOS statements in macros send DC control codes to the serial port. When the setting is ON, these statements will send DC control codes. When it is OFF, the control codes are suppressed. Its default value is ON.

72 Can Cycle Cut Depth

Used with canned cycles G71 and G72, this setting specifies the incremental depth for each pass during rough cutting. It is used if the programmer does not specify a D code. Valid values range from 0 to 29.9999 inches or 299.999 mm. The default value is .1000 inches.

73 Can Cycle Retraction

Used with canned cycles G71 and G72, this setting specifies the retraction amount after a roughing cut. It represents the tool to material clearance as the tool returns for another pass. Valid values range from 0 to 29.9999 inches or 299.999 mm. The default value is .0500 inches.

74 9xxx 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.

75 9xxxx Progs Singls 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 Foot Pedal Lock Out

This is an On/Off setting. When it is Off, the foot pedal operates normally. When it is On, any action at the foot pedal is ignored by the control.

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



81 Tool at Auto Off

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.

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 Overrides

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 Tooling 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 lathe 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).

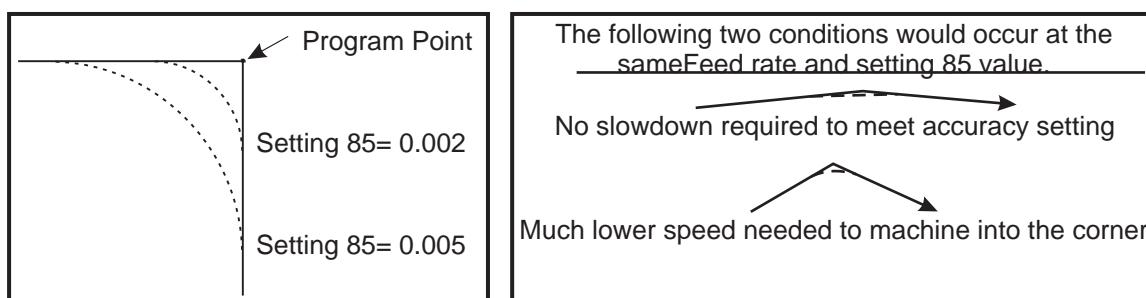
The Autofeed feature should not be used when thread milling or auto reversing tapping heads, as it may cause unpredictable results or even a crash.

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 Maximum Corner Rounding

This setting defines the machining accuracy of rounded corners within a selected tolerance. The initial default value is 0.05 inch. If this setting is zero (0), the control acts as if an exact stop is commanded in each motion block.





86 Thread Finish Allowance

Used in G76 canned threading cycle, this setting specifies how much material will be left on the thread for finishing after all passes of the cycle. Values range from 0 to .9999 inches. The default value is 0.

87 TNN Resets Override

This is an on/off setting. When M06 is executed and this setting is on, any overrides are canceled and set to their programmed values.

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 Graph Z Zero Location

This setting adjusts for extreme values in tool geometry or shift values. In graphics, tool offsets are ignored so that the cutting paths of different tools are displayed in the same location. Setting this to an approximate value of machine coordinates for the programmed part zero will void any Z Over Travel Range alarms that you may encounter in graphics. The default is -8.0000.

91 Graph X Zero Location

This setting adjusts for extreme values in tool geometry or shift values. In graphics, tool offsets are ignored so that the cutting paths of different tools are displayed in the same location. Setting this to an approximate value of machine coordinates for the programmed part zero will void any x Over Travel Range alarms that you may encounter in graphics. The default is -8.0000.

92 Chuck Clamping

This setting determines chuck clamping direction. Set to O.D., the chuck is considered clamped when the jaws are moved to the spindle center. Set to I.D., the chuck is considered clamped when the jaws are moved away from the spindle center.

93 Tailstock X Clearance

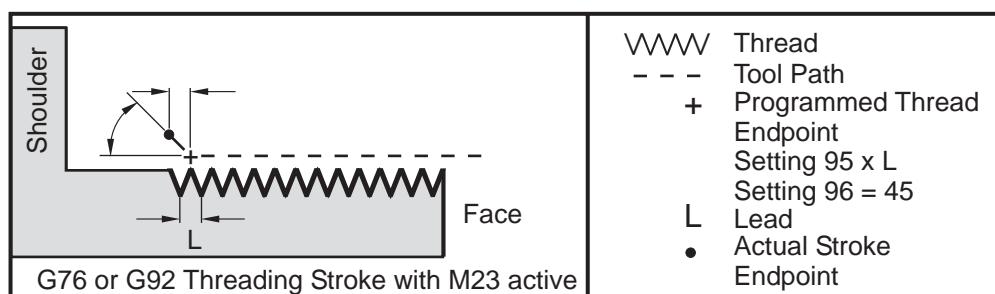
This setting works with Setting 94 to define a tail stock travel restriction zone that limits interaction between the tail stock and the tool turret. This setting determines the X axis travel limit when the difference between the Z axis location and the tail stock location falls below the value in Setting 94. If this condition occurs and a program is running then an alarm is generated. When jogging, no alarm is generated, but travel will be limited. Units are in inches.

94 Tailstock Z Clearance

This setting is the minimum allowable difference between the Z-axis and the tail stock (see Setting 93). Units are in inches. A value of -1.0000 means that when the X axis is below the X clearance plane (Setting 93), the Z axis must be more than 1 inch away from the tail stock position in the Z axis negative direction. The default value for this setting is zero. Units are in inches.

95 Thread Chamfer Size

This setting is used in G76 and G92 threading cycles when an M23 is commanded. When command M23 is active, threading strokes end with an angled retraction, as opposed to pulling straight out. The value in Setting 95 is equal to the number of turns (chamfered threads) desired. Note: Settings 95 and 96 interact with each other. Valid range: 0 to 29.999 (Multiple of current thread lead, F or E).





96 Thread Chamfer Angle

See Setting 95.

Valid range: 0 to 89 degrees (No decimal point allowed)

97 Tool Change Direction

This setting determines the default tool change direction. It may be set to either Shortest or M17/M18.

When "Shortest" is selected, the control will turn the direction necessary to reach the next tool with the least movement. The program can still use M17 and M18 to fix the tool change direction, but once this is done it is not possible to revert back to the shortest tool direction other than Reset or M30/M02.

Selecting M17/M18, the control will move the tool turret either always forward or always reverse based on the most recent M17 or M18. When Reset, power on, or M30/M02 is executed, the control will assume M17 as the tool turret direction during tool changes, always forward. This option is useful when a program must avoid certain areas of the tool turret due to odd-sized tools.

98 Spindle Jog RPM

This setting determines the spindle rpm for the Spindle Jog key. The default value is 100 rpm.

99 Thread Minimum Cut

Used in G76 canned threading cycle, this setting sets a minimum amount of successive passes of the thread cut. Succeeding passes cannot be less than the value in this setting. Values can range from 0 through .9999 inch. The default value is .0010 inches.

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 IPS screen will be displayed. After the second screen saver delay, the Haas logo will be displayed 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.

102 C Axis Diameter

This setting supports the C axis. See the C-axis Section. The default value is 1.0 inches and the maximum allowable value is 29.999 inches.

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 Handle 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.

105 TS Retract Distance

The distance from the Hold Point (Setting 107) the tail stock will retract when commanded. This setting should be a positive value.

106 TS Advance Distance

When the tail stock is moving toward the Hold Point (Setting 107), this is the point where it will stop its rapid movement and begin a feed. This setting should be a positive value.



107 TS Hold Point

This setting is in absolute machine coordinates and should be a negative value. It is the point to advance to for holding when M21 is commanded. Usually this is inside of a part being held. It is determined by jogging to the part and adding some amount to the absolute position.

109 Warm-Up 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

112 Warmup Z Distance

Settings 110 and 112 specify the amount of compensation ($\text{max} = \pm 0.0020"$ or $\pm 0.051 \text{ mm}$) applied to the axes. Setting 109 must have a value entered for settings 110 and 112 to have an affect.

113 Tool Change Method

This setting is used for the TL-1 and TL-2 lathes. See the Toolroom Lathe manual.

114 Conveyor Cycle (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.

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 Foot Pedal TS Alarm

When M21 is used to move the tail stock to the hold point and hold a part, the control will generate an alarm if a part is not found and the hold point is reached. Setting 121 can be switched to On and a alarm will be generated when the foot pedal is used to move the tail stock to the hold point and no part is found.

122 SS Chuck Clamping

This feature supports Subspindle lathes. Its value can be either O.D. or I.D; similar to setting 92 for the main spindle.

131 Auto Door

This setting supports the Auto Door option. It should be set to ON for machines with an autodoor. Also see M85/86 (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.

132 Jog or Home Before TC

When this setting is OFF, the machine behaves normally. When it is ON and Turret FWD, Turret REV or Next Tool is pressed while one or more axes is away from zero, it is assumed that a crash is likely and a message is displayed instead of doing the tool change. However, if the operator had pressed Handle Jog prior to the tool change, it is assumed that the axis had just been jogged to a safe position and will perform the tool change.

133 REPT 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/IP 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 Mask

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.

145 TS at Part for CS

(Tail stock at part for Cycle Start) When it is OFF, the machine behaves as before. When this setting is ON, the tail stock must be pressing against the part at the moment Cycle Start is pressed or a message is displayed and the program will not start.

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,J,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 Powerup SP Max RPM

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.

Setting 165 SSV Variation

Specifies the amount by which to allow the RPM to vary above and below its commanded value during the use of the Spindle Speed Variation feature. Positive value only.

Setting 166 SSV CYCLE (0.1) SECS

Specifies the duty cycle, or the rate of change of Spindle Speed. Positive value only.



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

IMPORTANT! REFER TO LOCAL CODE REQUIREMENTS BEFORE WIRING MACHINES.

ALL MACHINES REQUIRE:

Three phase 50 or 60Hz power supply.
Line voltage that does not fluctuate more than +/-10%

15 HP System	Voltage Requirements (195-260V)	High Voltage Requirements (354-488V)
SL-10	50 AMP	25 AMP
Power Supply	40 AMP	20 AMP
Haas Circuit Breaker		
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
20 HP System	Voltage Requirements (195-260V)	High Voltage Requirements (354-488V)
1SL-20, TL-15	50 AMP	25 AMP
Power Supply	40 AMP	20 AMP
Haas Circuit Breaker		
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
30-40 HP System	Voltage Requirements	High Voltage Requirements ²
TL-15BB, SL-20BB, SL-30, SL-30BB,		
1SL-40, SL-40BB	(195-260V)	(354-488V)
Power Supply	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
55HP System	Voltage Requirements (195-260V)	High Voltage Requirements (354-488V)
1SL-40, SL-40BB, SL-40L	150 AMP	<u>Must use an external transformer</u>
Power Supply	125 AMP	
Haas Circuit Breaker		
If service run from elec. panel is less than 100' use:	1 GA. WIRE	
If service run from elec. 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 CNC Lathe requires a minimum of 100 PSI at 4 scfm at the input to the pressure regulator on the back of the machine. 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. At least a 3/8" I.D. hose is recommended. Set the main air regulator to 85 psi.

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 at least a 3/8".

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.

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, window replacement should occur at no more than a two year interval.

Windows and guarding should be replaced if damaged or severely scratched - Replace damaged windows immediately

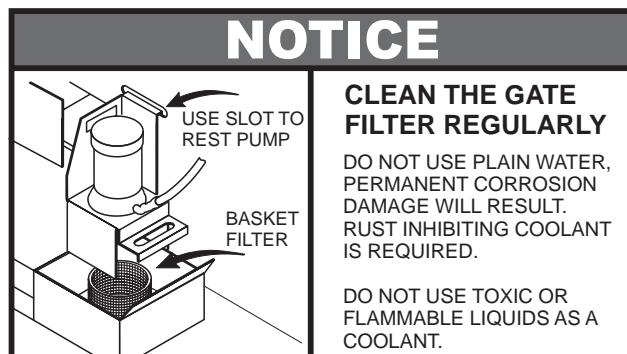


MAINTENANCE SCHEDULE

The following is a list of required regular maintenance for the Haas SL-Series Turning Centers. Listed are the frequency of service, capacities, and type of fluids required. 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. Check way lube lubrication tank level.• Clean chips from way covers and bottom pan.• Clean chips from turret, housing, rotating union and extension tube. Make sure the drawtube cover plate is installed either on the rotating union or on the chuck opening.• Check hydraulic unit oil level (DTE-25 ONLY). Capacity: 8 gallons.
Weekly	<ul style="list-style-type: none">• Check for proper operation of auto drain on filter regulator.• Check air gauge / regulator for 85 psi.• Clean exterior surfaces with mild cleaner. DO NOT use solvents.• Clean out small chip catch pan in coolant tank.
Monthly	<ul style="list-style-type: none">• Inspect way covers for proper operation and lubricate with light oil, if necessary.• Remove pump from the coolant tank. Clean sediment from inside the tank. Reinstall pump. Caution! Disconnect the coolant pump from the controller and Power Off the control before working on the coolant tank.• Dump the oil drain bucket. Check Gearbox oil level (if applicable). If oil is not visible at the bottom edge of the sight gauge, remove the end panel and add DTE-25 through the top filler hole until it is visible in the sight gauge.
Six Months	<ul style="list-style-type: none">• Replace coolant and thoroughly clean the coolant tank.• Replace hydraulic unit oil filter.• Check all hoses and lubrication lines for cracking.
Annually	<ul style="list-style-type: none">• Replace gearbox oil.• Clean oil filter and remove residue from the bottom of filter.• Replace air filter on control box every (2) years.

Caution! Do not use a wash-down hose on the Haas lathe; doing so may cause damage to the spindle.



Poor Coolant flow can be caused by a dirty filter. To clean the filter, turn off the coolant pump, lift the coolant tank lid and remove the filter. Clean and reinstall filter.



LUBRICATION

System	Lubricant	Quantity
Way lube and pneumatics	Mobile Vactra #2	2-2.5 qts
Transmission	Mobil DTE 25	34 oz

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 checks.

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.

CHUCK MAINTENANCE

Ensure all moving part are thoroughly greased
Check for excessive wear on jaws.
Check T-nuts for excessive wear.
Check front retaining bolts for damage.
Chucks should be broken in according to the manufacturers' specifications.
Disassemble and inspect chuck once a year.

Refer to chuck manual for disassembly procedures.

- Check for excessive wear.
- Check for galling or burnishing.
- Clean guide ways of contamination, chips and coolant
- Lubricate chuck before reassembly.

Caution: Lack of grease significantly reduces clamping force and can result in chatter, improper clamping, or thrown parts.

Chuck Jaws

Each chuck jaw requires two strokes of grease every 1000 clamp/unclamp cycles, or at least once a week. Use the provided grease gun for chuck lubrication. The Lubrication type is Molybdenum Disulfide Grease (20%-25% moly content).



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,**

Do not use pure water as a coolant; machine components will rust. Do not use flammable liquids as coolant.

The coolant tank must be thoroughly cleaned periodically, especially for mills equipped with High Pressure Coolant.

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. It will also ensure that coolant with the proper concentration and lubricity will be replaced.

WARNING!

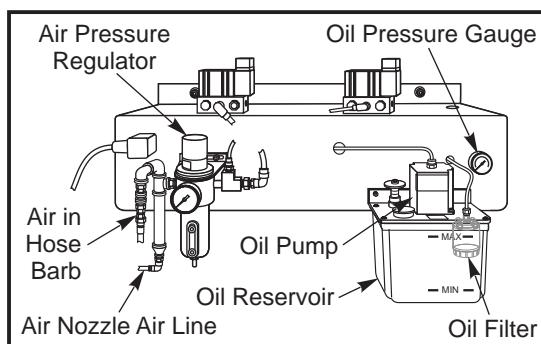
When machining castings, sand from the casting process and the abrasive properties of cast aluminum and cast iron will shorten coolant pump life unless a special filter is used in addition to the standard filter. Contact Haas Automation for recommendations.

Machining of ceramics and the like voids all warranty claims for wear and is done entirely at the 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.

Shortened pump life, reduction of coolant pressure and increased maintenance are normal and to be expected in abrasive environments and is not covered by warranty.

LUBRICATION SYSTEM

All machine lubrication is supplied by the external lubrication system. The reservoir is located on the lower rear of the machine (see Figure below). Current lube level is visible in the reservoir. If additional lube needs to be added, remove the cap from the fill port and add lube to proper level.



External Lubrication System

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.

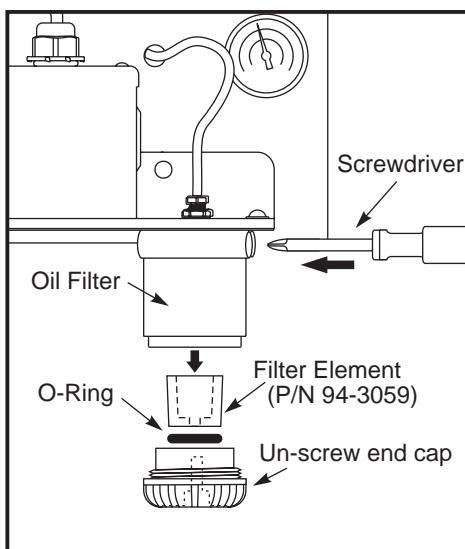


Lube Oil Filter

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.



TRANSMISSION OIL

Oil Check

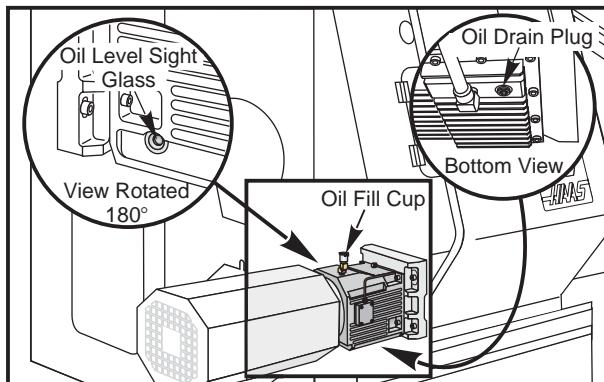
Check the oil level at the sight glass through the opening in the side of the machine as shown in the illustration. Fill as needed through the fill port on top of the gear box.

Oil Change

1. Remove the sheet metal necessary to gain access to the transmission.
2. Remove the fourteen (14) SHCS from the oil pan and remove it. Inspect the magnetic drain-plug for signs of metal particles.
3. Wipe down the oil pan and reinstall it with a new gasket.
4. Blow downward with an air hose in the vicinity of the access plate to prevent dirt and metal particles from entering the gear case. Remove the access plate.



5. Fill the gear case with 2½ liters of Mobil DTE-25 gear oil. Check the sight glass. The level should be 3/4 of the way up when full. Fill as needed.
6. Install access plate with new gasket.
7. Run a spindle warm-up and check for leaks.



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 trap regularly. Should the drain become clogged and cause coolant to collect in the machine's pan, stop the machine, loosen the chips blocking the drain, and allow the coolant to drain. Empty the coolant tank strainer, then resume operation.

Machining Remnants

The end of bar remnant must be collected to same way the parts are when using a bar feeder. Remove the remnant by hand or if using the parts catcher, program it to collect the remnant. Discharge tubes or auger pans that have remnants pushed through them will not be covered under the warranty.

AUXILIARY FILTER ELEMENT REPLACEMENT

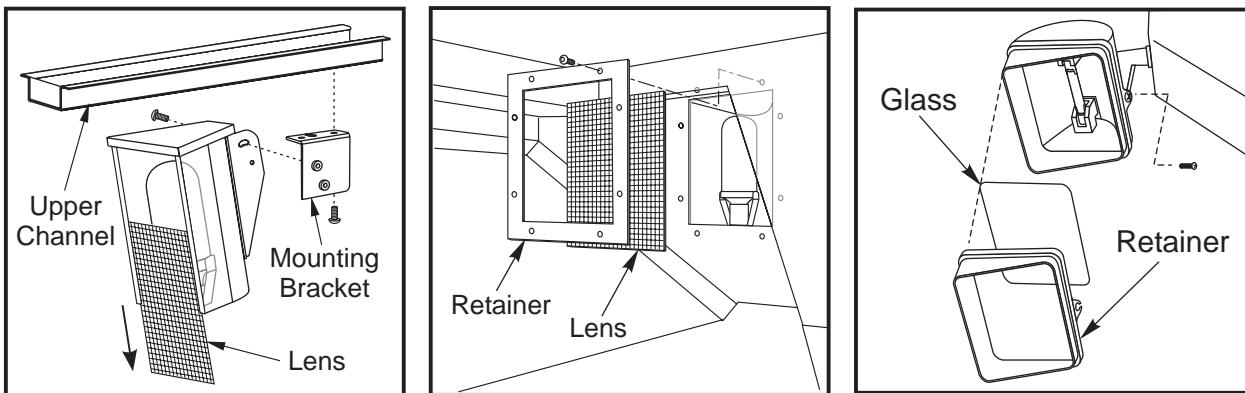
Change the filter bag when the filter gauge displays a vacuum level of -5 in. Hg or more. Do not allow the suction to exceed -10 in. Hg or pump damage may occur. Replace with a 25-micron rated filter bag (Haas P/N 93-9130).

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.



WORKLIGHT

Before doing any work on the lathe, turn off power to the machine at the main breaker.



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.