HAAS SERVICE AND OPERATOR MANUAL ARCHIVE

Lathe Operators Manual 96-8700U 9 RevU English June 2008

- 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.

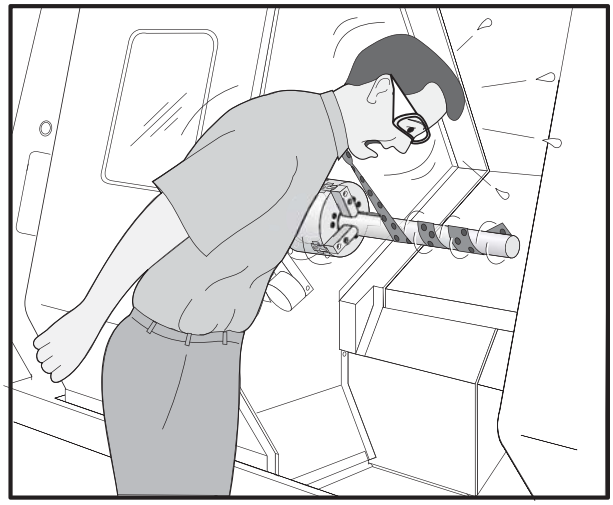
WARNING: Some mechanical and electrical service procedures can be extremely dangerous or life-threatening. Know your skill level and abilities.

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 , safety decals, safety procedures and instructions for safe machine operation.

Safety Contents

USES AND GUIDELINES FOR PROPER MACHINE OPERATION 4
- Modifications to the Machine 4
- Safety Placards 4
- Mill Warning Decals 6
- Lathe Warning Decals 7
- Declaration of Warnings, Cautions, and Notes 7

General Product Use Specifications and Limits

Environmental (Indoor use only)*		
	Minimum	Maximum
Operating Temperature	41° F (5° C)	122° F (50° C)
Storage Temperature	-4° F (-20° C)	158° F (70° C)
Ambient Humidity	20% relative, non-condensing	90% relative, non-condensing
Altitude	Sea Level	6000 ft. (1829 m)

Noise		
	Minimum	Maximum**
Emitted from all areas of machine during use at a typical operator's position	Greater than 70 dB	Greater than 85dB

*Do not operate the machine in explosive atmospheres (explosive vapors and / or particulate matter)

**Take precautions to prevent hearing damage from machine/machining noise. Wear ear protection, alter cutting application (tooling, spindle speed, axis feed, fixturing, programmed path) to reduce noise, and / or restrict access to machine area during cutting.

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 key and 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.
- Consult your local safety codes and regulations before operating the machine. Contact you dealer anytime safety issues need to be addressed.
- 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.**

Unattended Operation

Fully enclosed Haas CNC machines are designed to operate unattended; however, your machining process may not be safe to operate unmonitored.

As it is the shop owner's responsibility to set up the machines safely and use best practice machining techniques, it is also their responsibility to manage the progress of these methods. The machining process must be monitored to prevent damage if a hazardous condition occurs.

For example, if there is the risk of fire due to the material machined, then an appropriate fire suppression system must be installed to reduce the risk of harm to personnel, equipment and the building. A suitable specialist must be contacted to install monitoring tools before machines are allowed to run unattended.

It is especially important to select monitoring equipment that can immediately perform an appropriate action without human intervention to prevent an accident, should a problem be detected.

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.**

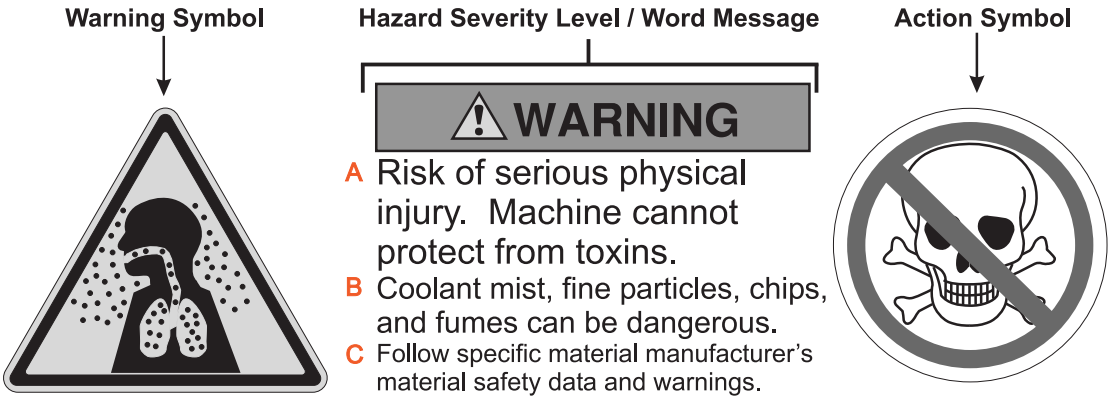
Modifications to the Machine

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 Decals

To help ensure that CNC tool dangers are quickly communicated and understood, hazard symbol decals are placed on Haas Machines in locations where hazards exist. If decals become damaged or worn, or if additional decals are needed to emphasize a particular safety point, contact your dealer or the Haas factory. **Never allow anyone to alter or remove any safety decal or symbol.**

Each hazard is defined and explained on the general safety decal, located at the front of the machine. Particular locations of hazards are marked with warning symbols. Review and understand the four parts of each safety warning, explained below, and familiarize yourself with the symbols on the following pages.



Warning Symbol - Identifies the potential hazard and reinforces the word message.

Word Message - Clarifies or reinforces the intent of the warning symbol.
A: Hazard.
B: Consequence if warning is ignored.
C: Action to prevent injury. Also refer to Action Symbol.

Hazard Severity Level - Color-coded to indicate risk in ignoring a hazard.
Red + "DANGER" = Hazard WILL cause death or serious injury if ignored.
Orange + "WARNING" = Hazard COULD cause death or serious injury if ignored.
Yellow + "CAUTION" = Hazard MAY cause minor to moderate injury if ignored.
Blue + "NOTICE" = Indicates an action to prevent damage to the machine.
Green + "INFORMATION" = Details about machine components.

Action Symbol: Indicates actions to prevent injury. Blue circles indicate mandatory actions to avoid harm, red circles with diagonal slashes indicate prohibited actions to avoid harm.

DANGER



Electrocution hazard.
Death by electric shock can occur.
Turn off and lock out system power before servicing.





Machine may start at any time.
Injury or death could be caused by untrained operator.
Read and understand operator's manual and safety signs before using this machine.





Risk of serious physical injury. Machine cannot protect from toxins.
Coolant mist, fine particles, chips, and fumes can be dangerous.
Follow specific material manufacturer's material safety data and warnings.





Risk of eye and ear injury.
Flying debris into unprotected eyes can cause loss of sight.
Noise levels can exceed 70 dBA.
Must wear safety glasses and hearing protection when operating or in the area of machine.





Risk of fire and explosion.
Machine is not designed to resist or contain blasts or fire.
Do not machine explosive or flammable materials or coolants.
Refer to specific material manufacturer's material safety data and warnings.





Risk of bodily injury.
Serious cuts, abrasions, and physical injury may result from slips and falls.
Avoid using the machine in wet, damp, or poorly lit areas.



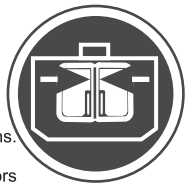


Severe injury can occur.
Moving parts can entangle, trap, and cut. Sharp tools or chips can cut skin easily.
Ensure the machine is not in automatic operation before reaching inside.





Risk of serious bodily injury.
The enclosure may not stop every type of projectile.
Double-check job set up before beginning any machining operations.
Always follow safe machining practices. Do not operate with doors or windows open or guards removed.



Safety windows may become brittle and lose effectiveness when exposed to machine coolants and oils over time.
If signs of discoloration, crazing, or cracking are found, replace immediately.
Safety windows should be replaced every two years.

For additional information, visit www.haascnc.com 29-0764 Rev B

WARNING

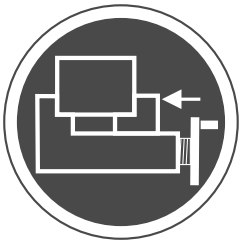


Severe injury can occur.
Moving parts can entangle and trap.
Always secure loose clothing and long hair.





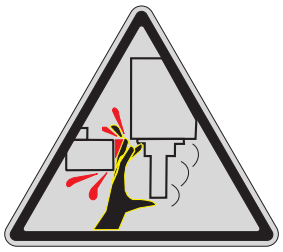
Risk of serious bodily injury.
Follow safe clamping practices. Inadequately clamped parts can be thrown with deadly force.
Securely clamp workpieces and fixtures.





Impact hazard.
Machine components can crush and cut.
Do not handle any part of the machine during automatic operation.
Always keep clear of moving parts.





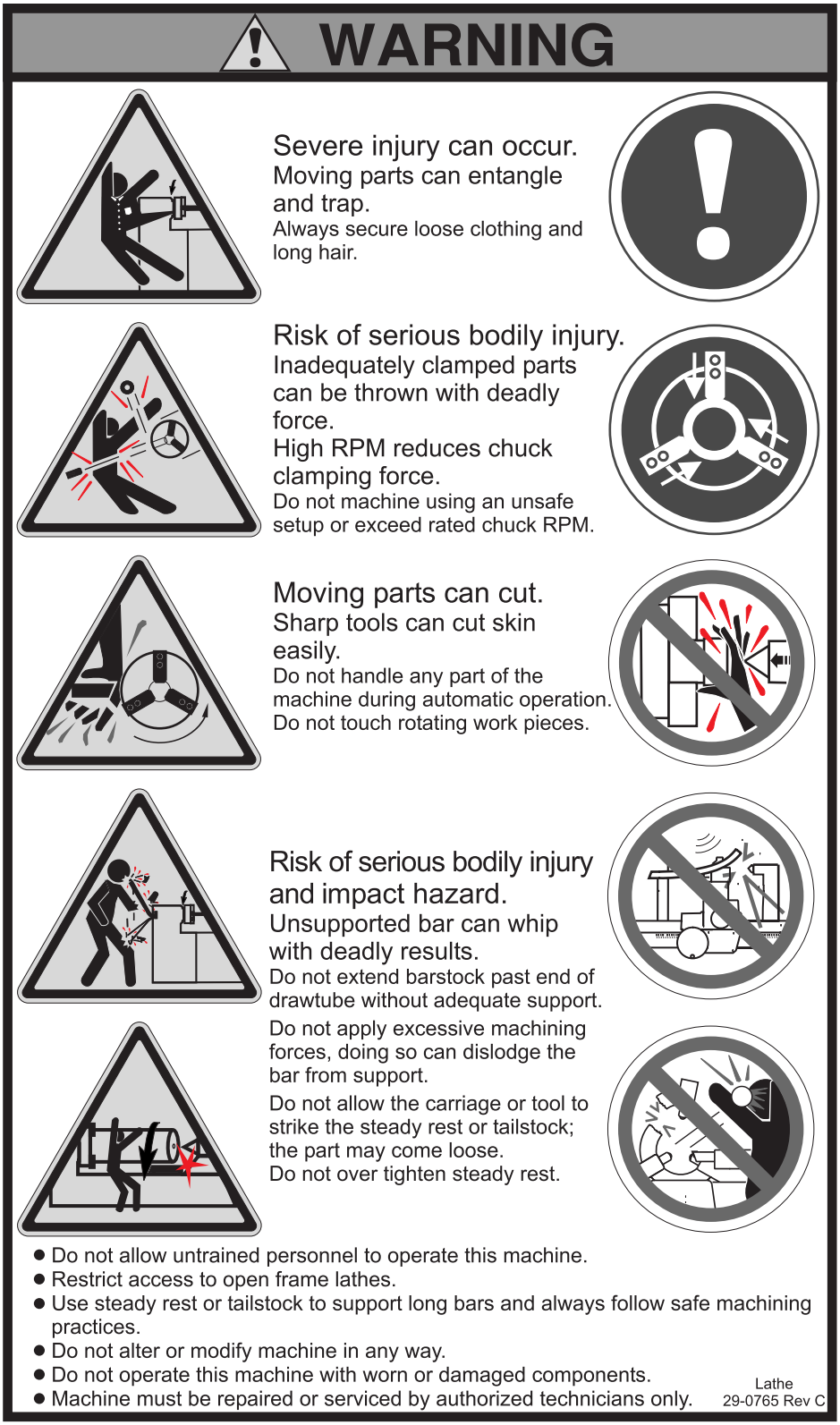
Moving parts can crush.
The tool changer will move in and crush your hand.
Never place your hand on the spindle and press ATC FWD, ATC REV, NEXT TOOL, or cause a tool change cycle.



- Do not allow untrained personnel to operate this machine.
- Do not alter or modify machine in any way.
- Do not operate this machine with worn or damaged components.
- No user serviceable parts inside. Machine must be repaired or serviced by authorized service technicians only.

Mill
29-0769 Rev C

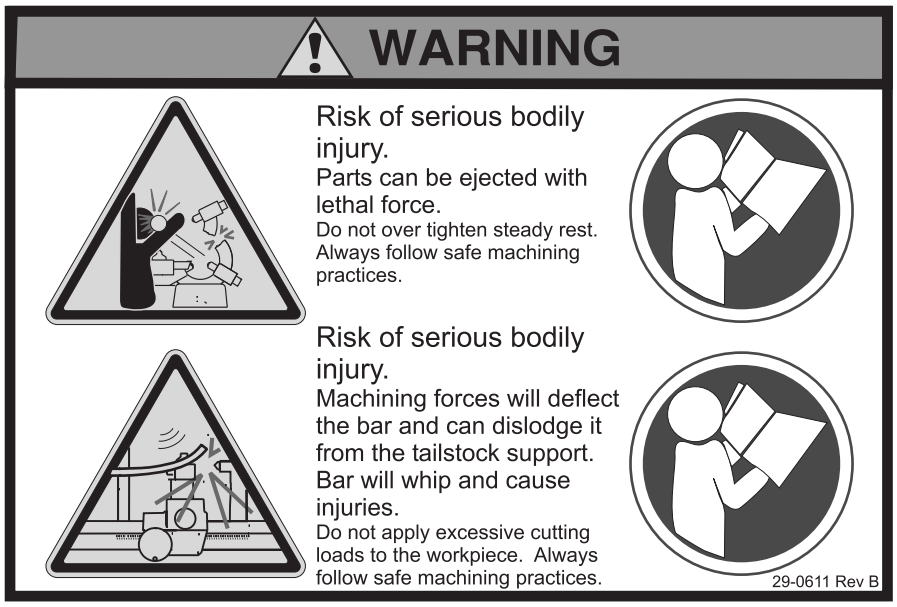
Lathe Warning Decals

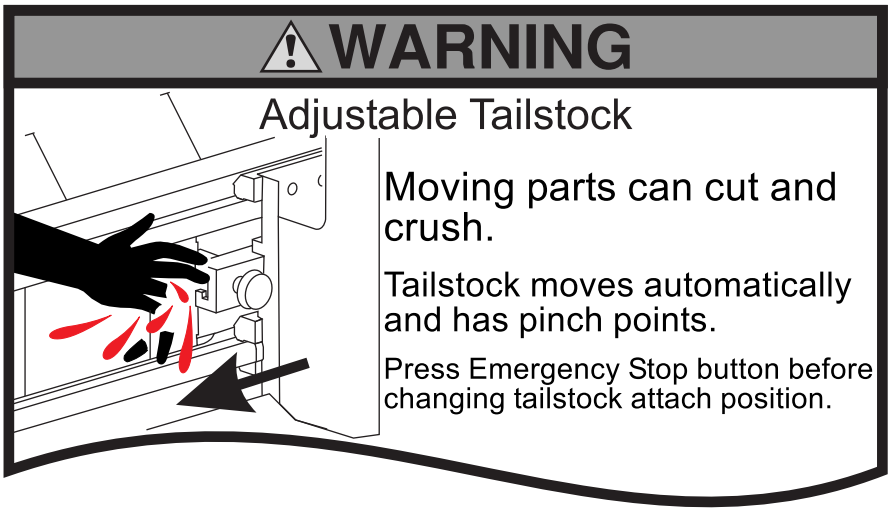


Other Safety Decals

Other decals may be found on your machine, depending on the model and options installed:







Refer to the tailstock section for further explanation.



Declaration of Warnings, Cautions, and Notes

Throughout this manual, important and critical information is prefaced with the word “Warning”, “Caution” and “Note”

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!

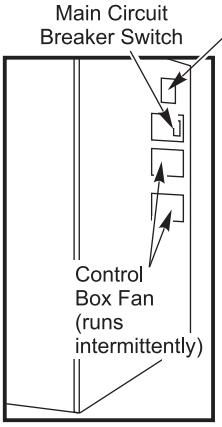
Never put hands between tool changer and spindle head.

Cautions are used when there is the potential for minor personal injury or mechanical damage, for example: **CAUTION!** Power down the machine before performing any maintenance tasks.

Notes give additional information to the operator about a particular step or procedure. This information should be taken into consideration by the operator as the step is performed to ensure there is no confusion, for example:

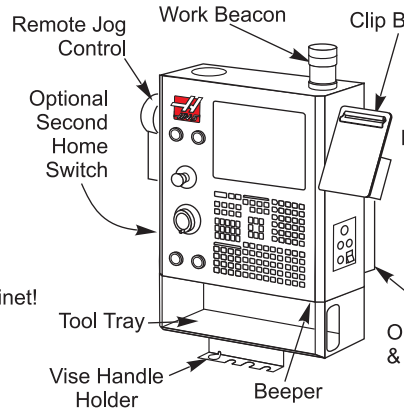
NOTE: If machine is equipped with the optional extended Z-clearance table, follow these guidelines:
• Avoid extreme loads on the center of table or far end of table. The weight of the part should be distributed evenly over the table or one pad.
• Flatness of part should be within 0.002”.

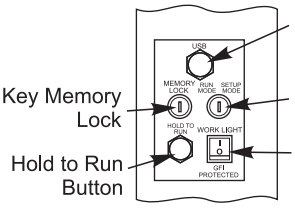
The following is a visual introduction to a Haas turning center. Some of the features shown will be highlighted in their appropriate sections.



Data Plate
Model
Serial Number
Date of Manufacture
Voltage
Phase
Hertz
Full Load
Largest Load
Short Circuit Interrupting Capacity
Wiring Diagram
Over current protection provided at machine supply terminals)
Warning Lethal voltages inside cabinet! Disconnect from power source before opening cabinet!
Trained service personnel only!
Made in USA

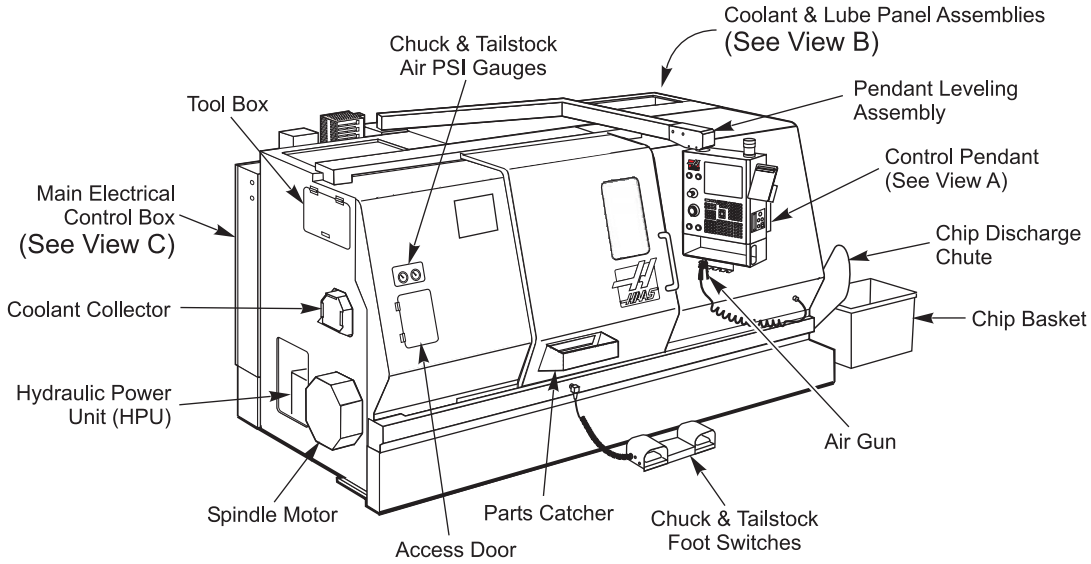
View C
(View Rotated for Clarity)

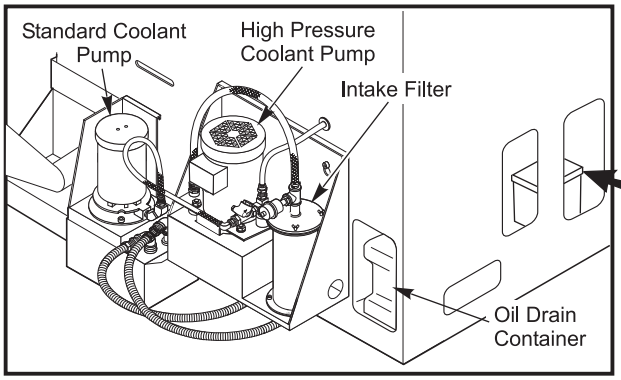




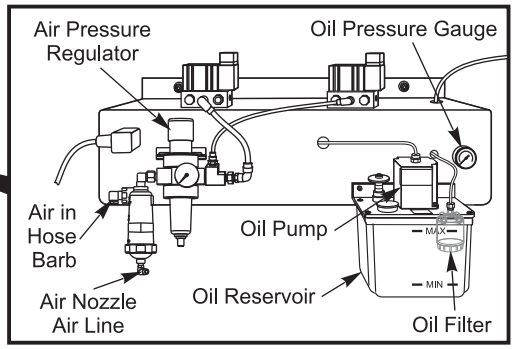
(Some Features are Optional)

View A

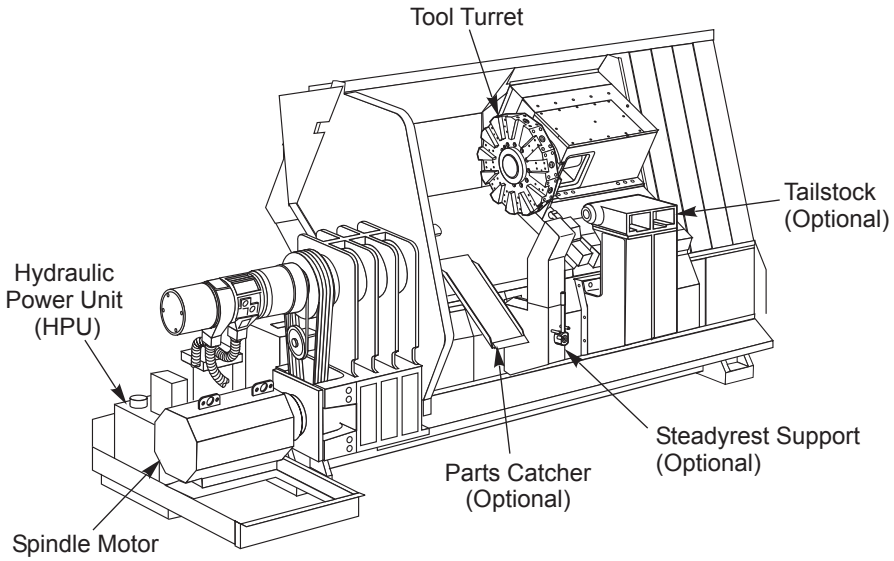


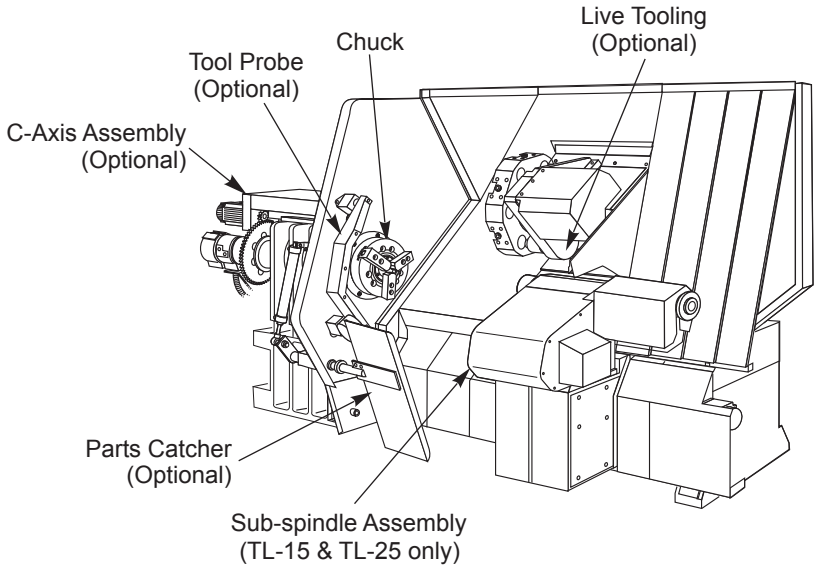


View B
(View Rotated 180°)



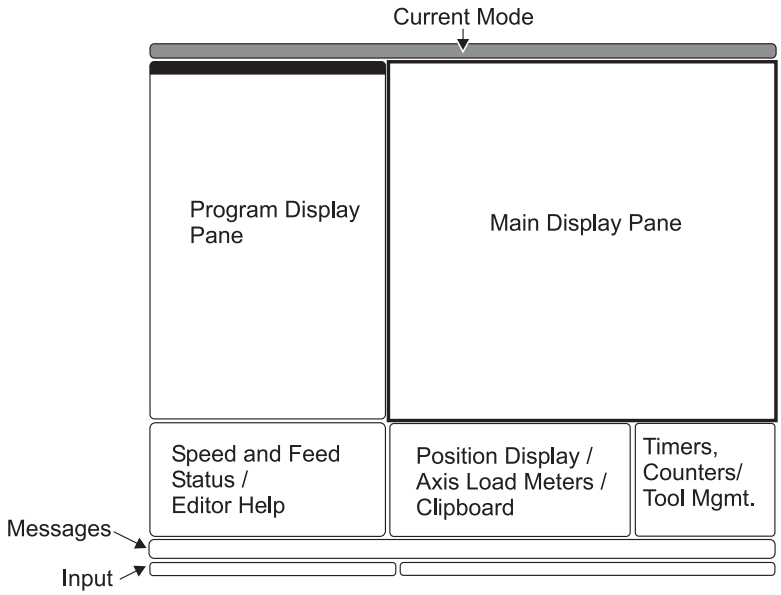
Lube Panel Assembly
(Left Rear Enclosure Panel Removed)


Tool Turret
Tailstock
(Optional)
Hydraulic
Power Unit
(HPU)
Steadyrest Support
(Optional)
Parts Catcher
(Optional)
Spindle Motor


Live Tooling
(Optional)
Chuck
Tool Probe
(Optional)
C-Axis Assembly
(Optional)
Parts Catcher
(Optional)
Sub-spindle Assembly
(TL-15 & TL-25 only)

Control Display and Modes

The control display is organized into panes that vary depending on the current control mode, and on what display keys are used. The following illustration shows the basic display layout:



Interaction with data can be carried out only within the currently active pane. Only one pane is active at any given time, and it is indicated with a white background. For example, to work with the Tool Offsets table, you must first make the table active by pressing the Offset key until it displays with a white background. Then you can make changes to the data. Changing the active pane within a control mode is typically done with the display keys.

Control functions are organized into three modes: **Setup**, **Edit**, and **Operation**. Each mode provides all of the necessary information to perform tasks that fall under the mode, organized to fit in one screen. For example, Setup mode displays both the work and tool offset tables, and position information. Edit mode provides two program editing panes and access to the VQCP and IPS/WIPS systems (if installed).

Access modes using the mode keys as follows:

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

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

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

The current mode is shown in the title bar at the top of the display.

Note that functions from other modes can still be accessed from within the active mode by using the display keys. For example, while in Operation mode, pressing OFFSET will display the offsets tables as the active pane; toggle the offset display using the OFFSET key. pressing PROGRM CONVRS in most modes will shift to the edit pane for the current active program.

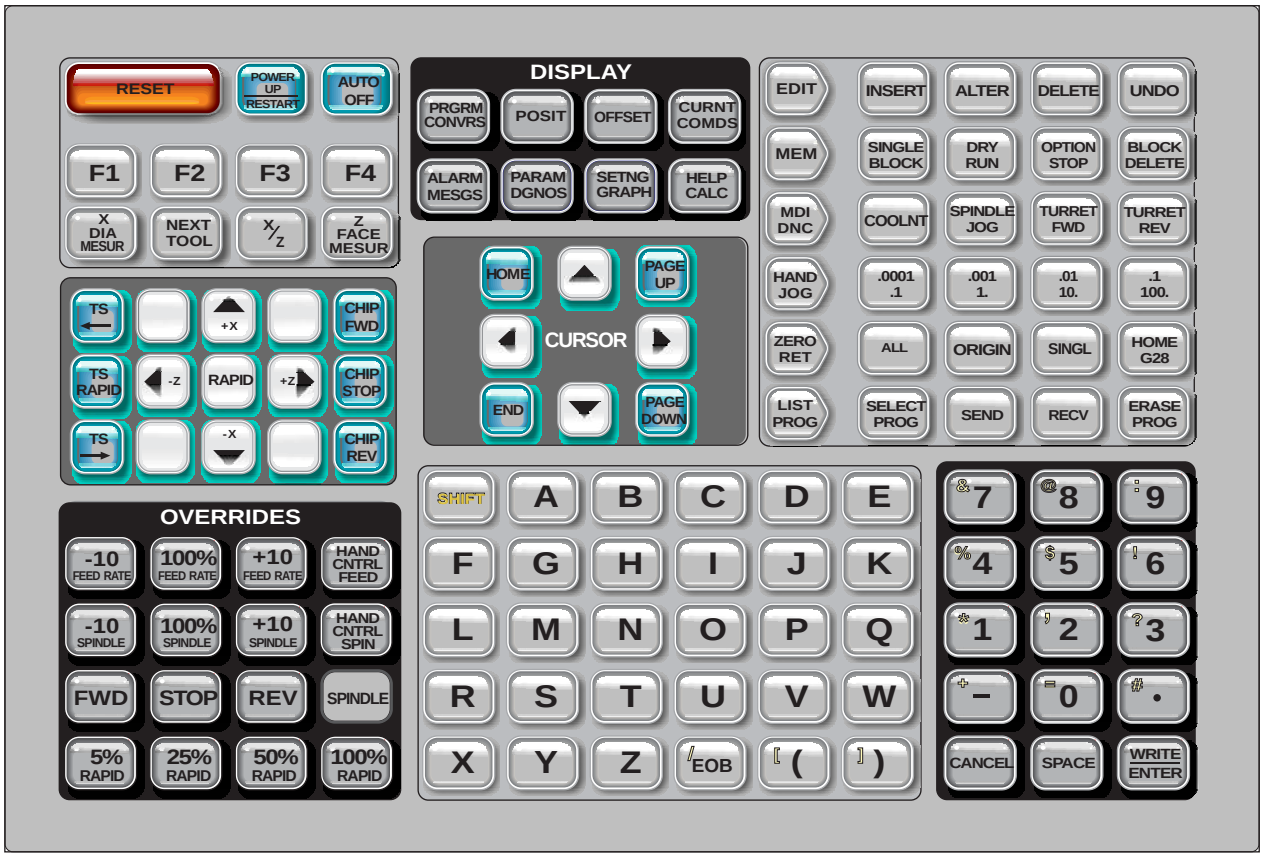
Navigating Tabbed Menus

Tabbed menus are used in several control functions, such as Parameters, Settings, Help, List Prog, and IPS. To navigate these menus, use the arrow keys to select a tab, then press Enter to open the tab. If the selected tab contains sub-tabs, use the arrow keys and Enter to select the appropriate one.

To go up one tab level, press Cancel.

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. This will not work for toolroom lathes, subspindle lathes, or automatic parts loader (APL).

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 G154 P20. 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.

XZ (2-Axis) Jogging

The lathe X and Z axes can be jogged simultaneously using the X and Z jog buttons. Holding any combination of +/-X and +/-Z jog buttons will cause two axis jogging. Releasing both jog buttons will result in the control reverting to X-axis jog mode. If only a single button is released, the control will continue jogging the single axis of the button still held. Note: Normal tailstock restricted zone rules are active while engaged in XZ jogging.

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 overriden 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 ±1% increments.

Hand Cntrl Spin (Handle Control Spindle) - Pressing this button allows the jog handle to be used to control spindle speed in ±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 ±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 M06, M30 and/or pressing RESET (See Settings 83, 87,88).

Display Keys

Display keys provide access to the machine displays, operational information and help pages. They are often used to switch active panes within a function mode. Some of these keys will display additional screens when pressed more than once.

Prgrm/Convrs - Selects the active program pane in most modes. In EDIT:MDI mode, press to access VQC and IPS (if installed).

Posit (Position) - Selects the positions pane, located in the lower center of most screens. Displays the current axis positions. Toggle between relative positions by pressing the POSIT key. To filter the axes displayed in the pane, type the letter for each axis you want to display and press WRITE/ENTER. Each axis position is displayed in the order you indicate.

Offset - Press to toggle between the two offsets tables. Select the Tool Offsets table to display and edit tool length geometry, radius offsets, wear offsets, and coolant position. Select the Work Offsets table to display and edit the G-code specified work offset locations used in programs.

Currnt Comds (Current Commands) - Press PAGE UP / PAGE DOWN to cycle through menus for Maintenance, Tool Life, Tool Load, Advanced Tool Management (ATM), Barfeeder, System Variables, azClock settings and timer / counter settings.

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). Press the Right Arrow key to view Alarm History. Use the Up and Down Arrow keys to scroll through alarm history entries, and press F2 to write to a disk.

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. You can then scroll through the alarms by pressing the Up and Down Arrow buttons. You can also enter an alarm number and press Enter or the Up/Down arrow keys to display the alarm name and description.

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

Param / Dgnos (Parameters / Diagnostics) - Displays parameters that define the machine's operation. Parameters are organized by category in a tabbed menu, or 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 except by authorized Haas personnel.

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 additional pages of diagnostic data.

Setng / Graph (Settings / Graphics) - Displays and allows changing of user settings. Like Parameters, Settings are organized by category in a tabbed menu. 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, you 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 help topics in a tabbed menu. Available help includes brief descriptions of G and M codes, definitions of control features, troubleshooting and maintenance issues. The help menu also includes several calculators.

Pressing the HELP/CALC key within some modes will call a pop-up help window. Use this window to access help topics relevant to the current mode, and also to execute certain functions as noted in the menu. To access the tabbed menu described above from a pop-up help window, press HELP/CALC a second time.

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 "B" and then Handle Jog.

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 text 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 mode is used to edit programs in the control's memory. Edit mode provides two editing panes: one for the currently active program, and another for background editing. Switch between the two panes by pressing the EDIT key. **Note:** While using this mode in an active program, press F1 to access help pop-up menus.

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.

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. Programs are run from this mode, and the MEM row contains keys that control the manner in which a program is executed.

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 Input” 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 CoolInt 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 zero position. This will not work on toolroom lathes, subspindle lathes, or automatic parts loader (APL).

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 move a single axis to the 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 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 that the current program will have an “A” 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 cursor selected program in List Prog 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

Positions Pane

Located at the lower center of the screen, the positions pane displays current axis positions relative to four reference points (Operator, Work, Machine and Distance-to-go). Press the POSIT key to activate the positions pane, and press it again to cycle through the available position displays. When the pane is active, you can change the axes displayed by typing the axis letters in the desired order and then pressing WRITE/ENTER. For example, typing "X" will display only the X-axis. Typing "ZX" will display those axes in the order indicated. A larger positions display is available by pressing CURNT COMDS, then Page Up or Page Down until the positions display is shown.

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, except when the machine is first powered on. Axes can be zeroed by typing the axis letter and pressing the Origin key.

Work Display

This displays the position of X, Y and Z in relation to the part; not machine zero. On power-up, it will display the value in work offset G54 automatically. The position can only be changed by entering values in work offsets G55 through G59, G110 through G129, or by commanding a G92 in a program.

Machine Display

This displays the axes' positions relative to machine zero.

Distance To Go

This display shows the distance remaining before the axes reach their commanded position. When in hand jog mode, this position display can be used to show a distance moved. You can zero this display by changing modes (EDIT, MEM, MDI) and then switching back to hand jog.

Offsets Display

There are two offsets tables. The first is the Tool Geometry/Wear table. The second is the Work Zero offset table. Depending on the mode, these tables may appear in two separate display panes, or they may share a pane. In either case, toggle between the two tables using the OFFSET key.

Tool Geometry/Wear

This table displays tool numbers and tool length geometry. Press the left cursor arrow when the cursor is in the first column of the tool geometry table to access the tool wear table.

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 "Zero All (Y/N)" press Y to zero all or press N to leave all the values unchanged.

Work Zero Offset

This table displays the values entered so that each tool knows where the part is located. 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
Current command information persists throughout most modes. Spindle information such as speed, load, direction, surface feet per minute (SFM), chip load and current transmission gear (if equipped) is shown in the lower-left display pane in all modes except Edit.

Axis positions are shown in the lower-center display pane. Cycle through the coordinate system (operator, work, machine, or distance to go) using the POSIT key. This pane also displays load data for each axis in some displays.

The coolant level is displayed near the upper-right of the screen.

Current Display Command This read-only display lists active program codes in the upper center of the screen.

Access the following screens by pressing CURNT COMDS, then PAGE UP or PAGE DOWN to cycle among displays.

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.

Further, two macro variables can be monitored within 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.

Active Codes Lists active program codes. It is an expanded display of the program code display described above.

Positions Display Provides a larger view of current machine positions, with all reference points (operator, machine, work, distance to go) displayed at once. You can also handle jog axes from this screen.

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

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. 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.

Alarms/Messages Display

Alarms

Select the Alarms display 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. You can then scroll through all the alarms by pressing the Up and Down Arrow keys. To view Alarm details for a known alarm number, type the number while the alarm viewer is active, then press WRITE/ENTER or the left/right cursor key.

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 new 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, MDI or Edit mode. From MEM or MDI, press the Setng/Graph key twice to select the Graphics Mode. From Edit mode, press Cycle Start while the active program edit pane is selected to start a simulation.

The Graphics display has a number of available features.

Key Help Area The lower left of the graphics display pane 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 pane displays the whole table area and indicates where the tool is currently located during simulation.

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

Adjusting Zoom Press F2 to display a rectangle (zoom window) indicating the area to be magnified. Use the Page Down key to decrease the size of the zoom window (zooming in), and use the Page Up key to increase the size of the zoom window (zooming out). Use the Cursor Arrow keys to move the zoom window to the desired location and press WRITE/ENTER to complete the zoom and rescale the tool path window. The locator window (small view at the bottom right) shows the entire table with an outline of where the Tool Path window is zoomed. Tool Path window is cleared when zoomed, and the program must be re-run to view 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.

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

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

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

Position Pane The position pane displays axes locations just as it would during a live part run.

F3 / F4 Use these keys to control simulation speed. F3 decrements speed, F4 increments speed.

Date and Time

The control contains a clock and date function. To view the time and date, press the CURNT COMDS button, then Page Up/Down until the date and time appears.

To make adjustments, press Emergency Stop, type the current date (in MM-DD-YYYY format) or current time (in HH:MM format), and press WRITE/ENTER. Reset Emergency Stop when finished.

Tabbed Help / Calculator Function

Press the HELP/CALC key to display the tabbed help menu. If pressing HELP/CALC calls a pop-up help menu, press HELP/CALC again to access the tabbed menu. Navigate tabs using the Cursor Arrow keys. Press WRITE/ENTER to select tabs, and press CANCEL to back up one tab level. The main tab categories and their sub-tabs are described here:

Help

G Codes: View a list of G Codes.

M Codes: View a list of M Codes.

Features: View a list of new software features.

Index: Select this tab to view various help topics. Press WRITE/ENTER to display information about the topic.

Drill Table

Displays a drill size table featuring decimal equivalents and tap sizes.

Calculator

Calculator functions are available under the third Help tab. Select the calculator mode you want to use from the lower tabs and press WRITE/ENTER to use it.

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, evaluating it (doing multiplication and division first) and placing the result, 89.8 in this case, in the window.

Note that data cannot be entered in any field where the label is highlighted. You must clear data in other fields until the label is no longer highlighted in order to change the field directly.

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

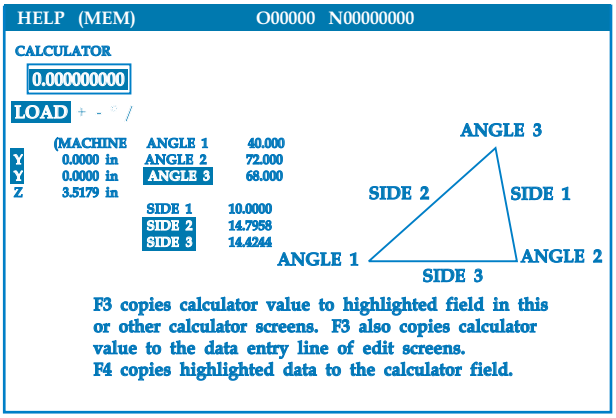
F3 In EDIT and MDI modes the F3 key will copy the highlighted triangle/circular milling/tapping value into the data entry line at the bottom of the screen. This is useful when the calculated solution will be used in a program.

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

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

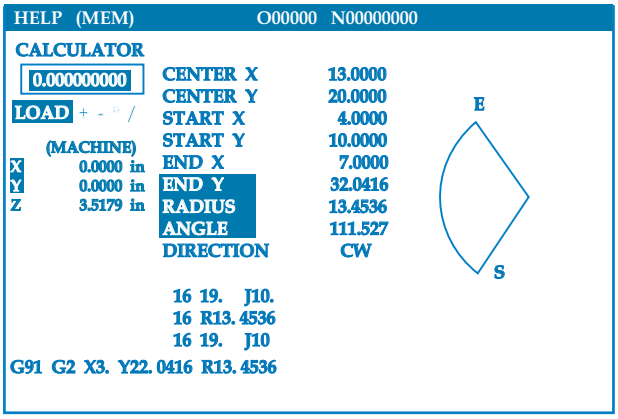
Trigonometry Help Function

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



Circular Interpolation Help

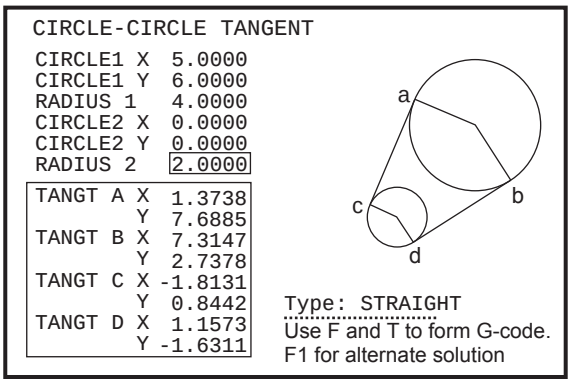
The Circular calculator page will help solve a circle problem. You enter the center, radius, angles, start and end points; when enough data has been entered, the control will solve for the circular motion and display the rest of the values. Use the Cursor Up/Down buttons to select the value to be entered 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.

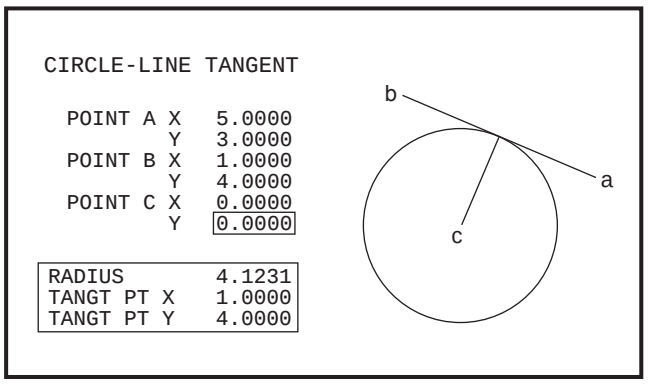


For inputs that have more than one solution, entering the last data value a second time will cause the next possible solution to be displayed. To change the CW value to the CCW value, highlight the CW/CCW column and press the Write/Enter button .

Circle-Line Tangent Calculator

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





Circle-Circle Tangent 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 that 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
A Drill and Tap Chart is available in the tabbed help menu.

Coolant Level Gauge

The coolant level is displayed in the upper right of the screen in MEM mode, or when you press CURNT COMDS. A vertical bar shows the status of the coolant. The display will flash when the coolant reaches a point that could cause intermittent 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 grippers.

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.

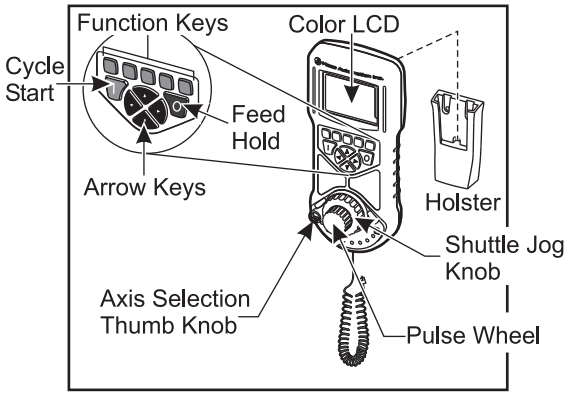
C-Axis
The C-axis provides high-precision, bi-directional spindle motion that is fully interpolated with X and/or Z motion. Cartesian-to-polar interpolation allows programming of face contouring operations using traditional X and Y coordinates.

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.

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

Enhanced Remote Jog Handle (RJH-C)
The Enhanced Color Remote Jog Handle (RJH-C) features a color liquid crystal display (LCD) and controls for increased functionality. The RJH-E is configured to control a number of machine functions and settings while providing real-time data from the control via the LCD screen. It also features a magnifier and a high-intensity LED inspection light. Note that the RJH-C is not compatible with lathe software earlier than version 9.



Refer to the section on offsets and machine operation for more information on those topics.

LCD: Displays machine data and the RJH-C interface.

Function Keys (F1-F5): Variable-function keys. Each key corresponds to a label at the bottom of the LCD screen. Pressing a function key will perform or toggle the corresponding menu. Toggled functions are highlighted when on.

Cycle Start: Starts programmed axis motion.

Feed Hold: Stops programmed axis motion.

Arrow Keys: Used to navigate between menu fields (up/down) and to select pulse jog rates (left/right).

Pulse Wheel: Jogs a selected axis by the selected increment. Works like the jog handle on the control.

Shuttle Jog: Rotates up to 45 degrees CW or CCW from center, and returns to center when released. Used to jog axes at variable speeds. The farther the shuttle jog is rotated from the center position, the faster the axis moves. Allow the shuttle knob to return to center to stop motion.

Axis Select: Used to select any of the available axes for jogging. The selected axis is displayed at the bottom of the screen. The far right position of this selector is used to access the auxiliary menu.

Removing the unit from the cradle powers it on. In Hand Jog, jogging control is turned over from the pendant to the RJH-C (The hand wheel on the pendant is disabled).

Place the RJH-C back in its cradle to power it off and return jogging control to the pendant.

The pulse knob and shuttle knob function as scrollers to change the value of a user-definable field such as tool offset, length, wear, etc.

Built-in "Panic" Function — Press any key during axis motion to instantly stop the spindle and all axis motion. Pressing Feed Hold while the spindle is in motion and the control is in Handle Jog mode will stop the spindle. The message **"Button pressed while axis was moving—Reselect Axis"** appears on the display. Move the axis selection knob to a different axis to clear.

If the axis selection knob is moved while the shuttle jog is turned, the message **"Axis selection changed while axis was moving—Reselect Axis"** appears on the display, and all axis motion stops. Move the axis selection knob to a different axis to clear the error.

If the shuttle jog knob is turned from its centered position when the RJH-C is removed from its cradle, or when the control mode is changed to a mode with motion (for instance, from MDI to Handle Jog mode), the message **"Shuttle off center—No Axis selected"** appears on the display and no axis motion will occur. Move the axis selection knob to clear the error.

If the pulse jog knob is rotated while the shuttle jog knob is in use, the message **"Conflicting jog commands— Reselect Axis"** appears on the display, and all axis motion stops. Move the axis selection knob to a different axis to clear the error, then back to reselect the previously selected axis.

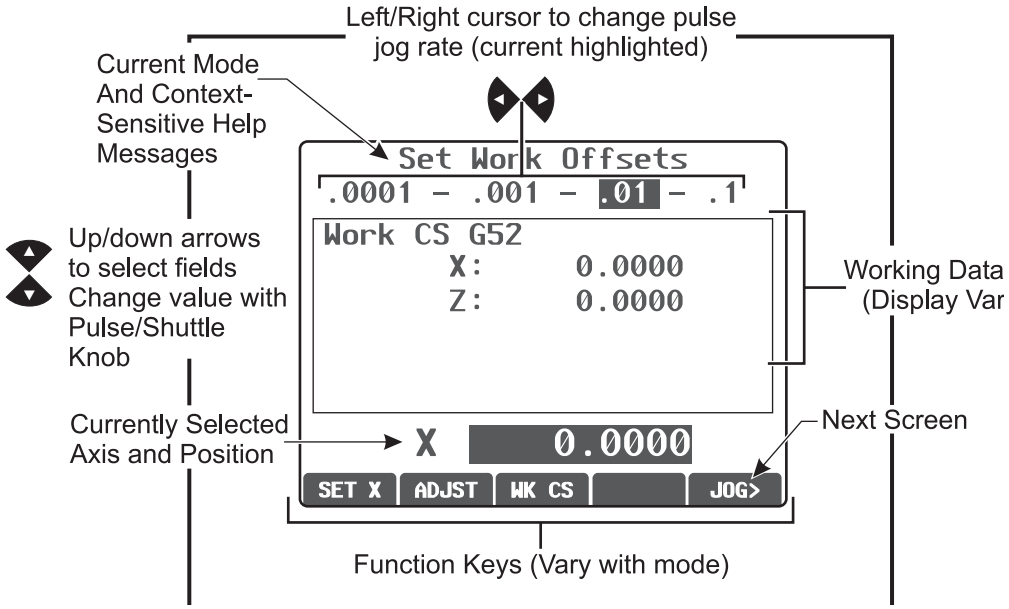
NOTE: If any of the above errors fail to clear when the axis selection knob is moved, there may be a problem with the shuttle jog knob. Contact Haas service for repair/replacement.

If contact between the RJH-C and the control is broken for any reason (cable cut or disconnected, etc.), all axis motion stops. When reconnected, the message **"RJH / Control Communication Fault—Reselect Axis"** appears on the RJH-C display. Move the axis selection knob to clear the error. If the error does not clear, place the RJH-C in its cradle, wait for it to power off, and then remove from the cradle.

NOTE: This error can also indicate a fault in the SKBIF, the RJH-E, or the wiring. If the error persists, further diagnosis and repair may be necessary.

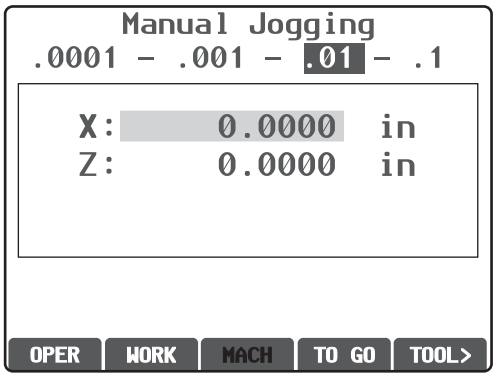
RJH-C Menus

The RJH-C uses four program menus to control manual jogging, set tool length offsets, set work coordinates, and display the current program. The four screens display information differently, but navigating and changing options are always controlled in the same way, as noted in this illustration.



Manual Jogging (Remote Jog Handle)

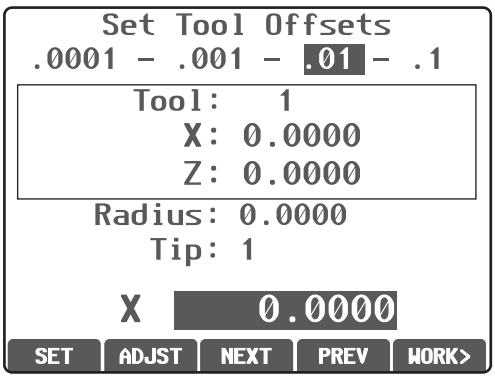
This menu contains a large display of the current machine position.Turning the shuttle jog or pulse knob will move the currently selected axis. Select the jog increment using the left/right arrow keys. The current position coordinate system is highlighted in the function key area of the screen and can be changed by pressing a different function key. To zero the operator position, press the function key under OPER to select the position, then press the function key again (it now reads ZERO).



Tool Offsets (Remote Jog Handle)

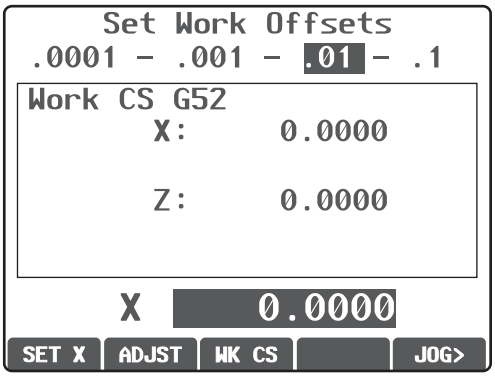
Use this menu to set and check tool offsets. Select fields using the function keys and change values using the pulse knob. Select axes using the thumb knob. The axis line (at the bottom of the display) must be highlighted to jog that axis. Press SET to record the current axis position into the offset table, and use the arrow keys to select the Radius and Tip settings. To make adjustments to table values, select ADJST, use the pulse or shuttle knob to select the amount to increase or decrease the value (use the left and right arrows to change the increment), then press ENTER to apply the adjustment.

CAUTION: Stay clear of the spindle when changing tools.



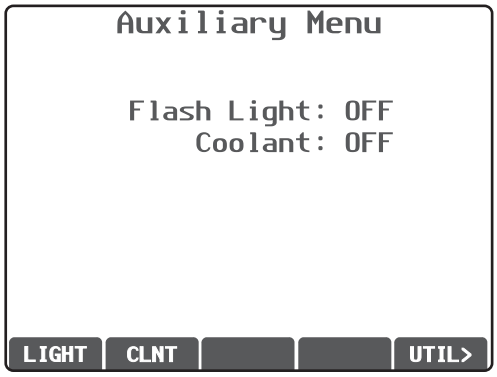
Work Offsets (Remote Jog Handle)

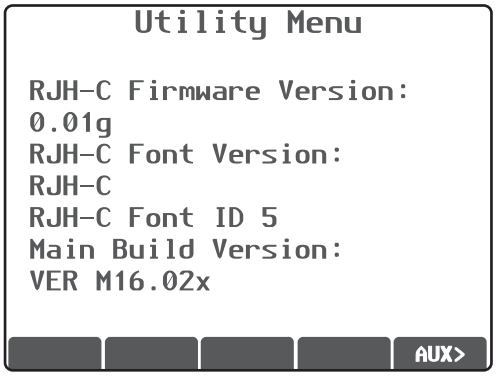
Select WK CS to change the work offset G code. Manually jog the selected axis with the shuttle or pulse knob when the axis field at the bottom of the screen is highlighted. Press SET to record the current position of the current axis into the work offset table. Move the axis selector to the next axis and repeat the process to set that axis. To make adjustments to a set value, move the axis selector to the desired axis. Press ADJST and use the pulse knob to increase or decrease the adjustment value, then press ENTER to apply the adjustment.



Auxiliary Menu

The RJH-C auxiliary menu features controls for machine coolant and the RJH flashlight. Access the menu by moving the axis selector to the far right position (indicated by a page icon molded into the RJH-C case). Toggle the available features by pressing the corresponding function key. Press UTIL to access the utility menu for technician diagnosis, and press AUX to return to the Auxiliary Menu.





Program Display (Run Mode)

This mode displays the currently running program. Enter run mode by pressing MEM or MDI on the control pendant. The tab options at the bottom of the screen provide controls for coolant on/off, single block, optional stop, and block delete. Toggled commands such as COOL will appear highlighted when turned on. The CYCLE START and FEED HOLD buttons function just as the buttons on the pendant. Return to jogging by pressing HAND JOG on the control pendant, or place the RJH-E back in the radle to continue running the program from the pendant.

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 display either the Messages screen, if a message was left, or the Alarms screen. In either case, the lathe 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 from which to start all operations. This point is called “Home”. To home the machine, press the Power-Up Restart button. Note that this will not work for toolroom lathes, subspindle lathes, or automatic parts loader (APL). For these machines, each axis must be homed separately.

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

```
G97 S1000 M03 ;
G00 X2. Z0.1 ;
G01 X1.8 Z-1. F12 ;
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, type 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.

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.

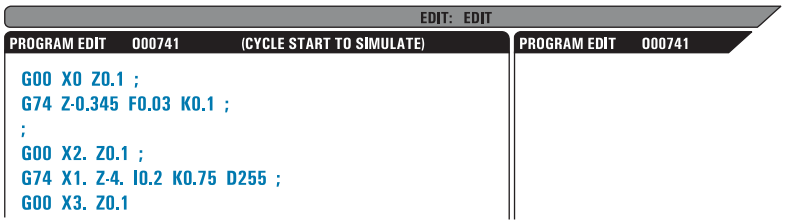
Numbered Programs

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.



To add program code to an existing program, highlight the area the additional code will go in front of, type in the data, and press Insert key. More than one code; such as **X** 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 can 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 enter spaces between commands. Spaces are automatically entered between elements for ease of reading and 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 or commands, highlight them 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. 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 (toward the start of the program) and the cursor down key will search forward (toward 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. To delete multiple programs, highlight each program to be deleted and press Write to select them. Press the Erase Prog key to delte the files.

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 before erasing all programs. Note that 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.

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, the letter “A” appears next to the program name. This program is now active and will be run when the mode is changed to Mem and Cycle Start is pressed. It is also the program that you will see on the Edit display.

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

CNC Data Transfer

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

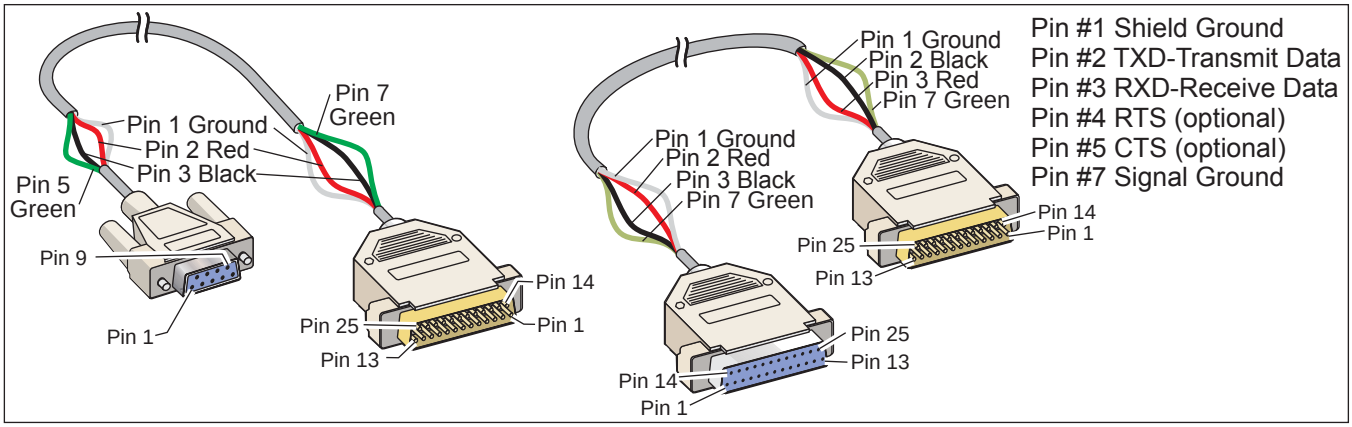
Unknown G-Code, if received by the CNC, is converted into a comment, stored in the program, and an alarm is generated. However, the data will still be loaded in to the control. This will also occur when trying to write Macros to the machine without the Macro option installed.

RS-232

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

Programs are sent or received through the RS-232 port (Serial Port 1) located on side of the control box (not the operator’s 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 lathe 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. Open the file on a PC. If an abort message is received, check the set-up between the lathe and the PC and the cable.

File Numeric Control (FNC)

A program can be run from its place on the network or from a storage device (USB memory device, floppy disk, hard drive). To run a program from such a location, go to the Device Manager screen (Press List Prog), highlight a program on the selected device and press “Select Prog”.The program will be displayed in the active program pane, and an “FNC” next to the program name in List Prog indicates that it is the currently active FNC program. Sub-programs can be called using an M98, provided that the subprogram is in the same directory as the main program. Additionally, the subprogram must be named using the Haas naming convention with case sensitivity, e.g. O12345.nc.

CAUTION: The program can be modified remotely, and the change will take affect the next time the program is run. Subprograms may be changed while the CNC program is running.

Program editing is not allowed in FNC. The program is displayed and can be browsed through, but not edited. Editing can be done from a networked computer or by loading the program into memory.

To run a program in FNC:

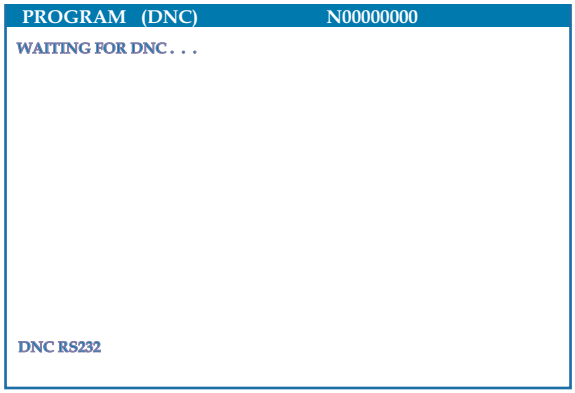
1. Press List Prog, then navigate to the tabbed menu for the appropriate device (USB, hard drive, Net Share).

2. Cursor down to the desired program and press Select Prog. The program will appear in the Active Program pane and can be run directly from the memory device.

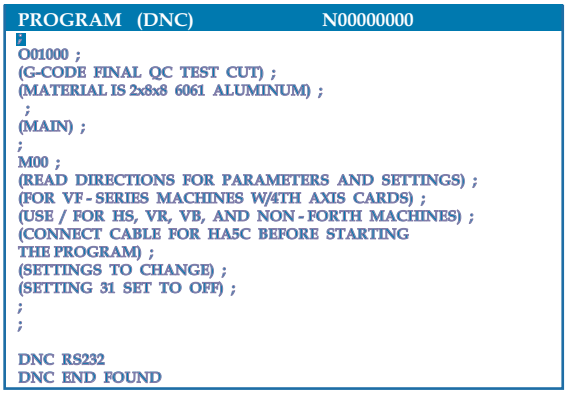
To Exit FNC, highlight the program again and press Select Prog, or select a program in CNC memory.

Direct Numeric Control (DNC)

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



Program Received from DNC

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

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

DNC is selected by pressing MDI twice at the top of the page. DNC requires a minimum of 8k bytes of user memory available. Checking the amount of free memory on the bottom of the List Programs 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.

USB / Hard Drive / Ethernet Device Manager

The Haas control incorporates a device manager which shows the available memory devices on the machine in a tabbed menu.

Enter the Device Manager by pressing “List Prog”. Navigate the tabbed menu using the arrow keys to select the appropriate device tab and press Enter.

When browsing a list of programs within a device tab, use the up/down arrow keys to highlight programs and press Enter to add the highlighted program to the selection.

The following example shows the directory for the USB device. The selected program in memory is shown with an “A”. The selected file will also show in the active program display.

Tabbed Menu Navigation

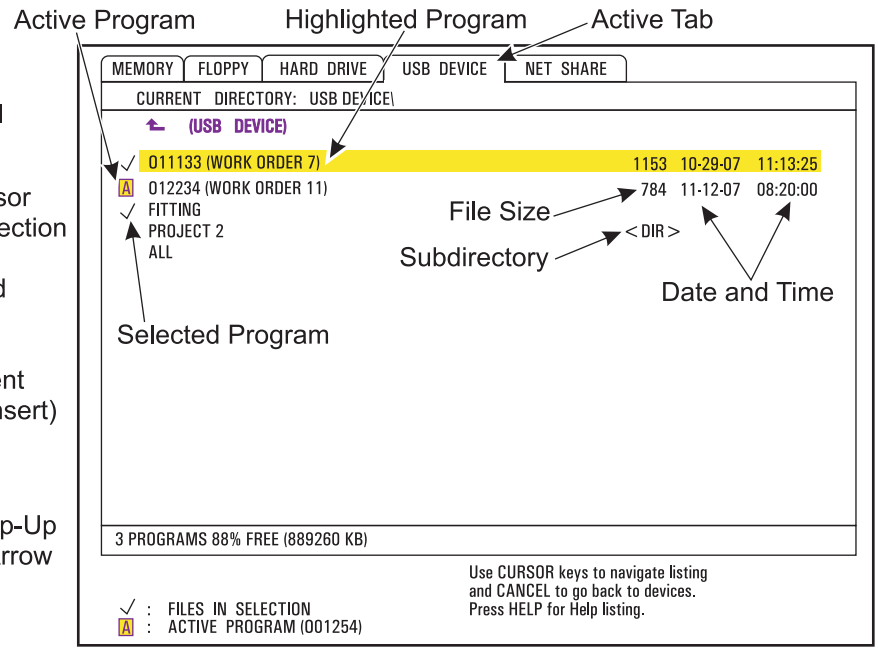
Cursor Arrows: Navigate Tabs
WRITE/ENTER: Select a Tab
CANCEL: Move Back One Tab Level

Program Selection

Cursor Arrows: Move Selection cursor
WRITE/ENTER: Add Program to Selection (A Check Mark is Placed)
SELECT PROG: Makes the Selected Program the Active Program ("**A**") or Selects Program for FNC
INSERT: Create New Folder in Current Directory (Type Folder Name, then Insert)
ALTER: Rename Folder or Program

Help Mode

Press **HELP/CALC** to Access the Pop-Up Help Menu. Navigate Using Cursor Arrow Keys. Choose Options for Selected Programs (Copy, Delete, Etc.)



Navigating Directories

To enter a sub-directory, scroll to the sub-directory and press Enter.

To leave a sub-directory, go to the top of the sub-directory and press Enter or press Cancel. Both options will return to the device manager.

Create Directories

Create a new folder by entering a name and pressing "Insert".

To create a new Sub-directory, go to the directory where the new sub-directory will be located, enter a name and press "Insert". Sub-directories are displayed with their name followed by <DIR>".

Copying Files

Highlight a file and press "Enter" to select it. A check mark appears next to the file name. Choose a destination and press F2 to copy the file.

Note that files copied from the control's memory to a device will have the extension ".NC" added to the end of the file name. However the name can be changed by entering a new name, at the destination directory, and then pressing F2.

Duplicating a File

An existing file can be duplicated using the Device manager. Select a file by pressing "Enter", then press "Cancel" to return to the top level tabbed menu. Select the destination device tab, press "Enter", then select the destination directory on the device if applicable. Press F2 to duplicate the selected file, or type a new name and then press F2 to rename the file in the destination directory.

File Naming Convention

File names should be kept to a typical eight-dot-three format. For example: program1.txt. However, some CAD/CAM programs use ".NC" as a file type recognition, which is acceptable.

Files developed in the control will be named with the letter "O" followed by 5 digits. For example O12345.NC.

Renaming

To change the name of a file, highlight the file, type a new name and press "Alter".

Deleting
To delete a program file from a device, highlight the file and press “Erase Prog”.

On-Screen Help
On-Screen help is available by pressing “HELP/CALC”. Select functions from the popup menu and press “Enter” to execute, or use the hot key listed. To exit the Help screen, press the “Cancel” button to return to the device manager.

Machine Data Collection

Machine Data Collection is enabled by Setting 143, which allows the user to extract data from the control using a Q command sent through the RS-232 port (or by using an optional hardware package). This feature is software-based and requires an additional computer to request, interpret and store data from the control. Certain Macro variables can also be set by the remote computer.

Data Collection Using the RS-232 Port
The control only responds to a Q command when Setting 143 is ON. The following output format is used:

<STX> <CSV response> <ETB> <CR/LF> <0x3E>

STX (0x02) marks the start of data. This control character is for the remote computer.
CSV is the Comma Separated Variables, one or more data variables separated by commas.
ETB (0x17) is the end of the data. This control character is for the remote computer.
CR/LF tells the remote computer data segment is complete and to move to the next line.
0x3E Displays the “ > “ prompt.

If the control is busy, it outputs “Status, Busy”. If a request is not recognized, the control outputs “Unknown” and a new prompt “ > “. The following commands can be used:

Q100 - Machine Serial Number
>Q100
S/N, 12345678

Q101 - Control Software Version
>Q101
SOFTWARE, VER M16.01

Q102 - Machine Model Number
>Q102
MODEL, VF2D

Q104 - Mode (LIST PROG, MDI, etc.)
>Q104
MODE, (MEM)

Q200 - Tool Changes (total)
>Q200
TOOL CHANGES, 23

Q201 - Tool Number in use
>Q201
USING TOOL, 1

Q300 - Power-on Time (total)
>Q300
P.O. TIME, 00027:50:59

Q301 - Motion Time (total)
>Q301
C.S. TIME, 00003:02:57

Q303 - Last Cycle Time
>Q303
LAST CYCLE, 000:00:00

Q304 - Previous Cycle Time
>Q304
PREV CYCLE, 000:00:00

Q402 - M30 Parts Counter #1 (resettable at control)
>Q402
M30 #1, 553

Q403 - M30 Parts Counter #2 (resettable at control)
>Q403
M30 #2, 553

Q500 - Three-in-one (PROGRAM, Oxxxxx, STATUS, PARTS, xxxxx)
>Q500
STATUS, BUSY

Q600 Macro or system variable
>Q600 801
MACRO, 801, 333.339996

The user has the ability to request the contents of any macro or system variable by using the Q600 command, for example, “Q600 xxxx”. This will display the contents of macro variable xxxx on the remote computer. In addition, macro variables #1-33, 100-199, 500-699, 800-999 and #2001 thru #2800 can be “written to” using an “E” command, for example, “Exxxx yyyyyy.yyyyyy” where xxxx is the macro variable and yyyyyy. yyyyyy is the new value. Note that this command should only be used when there are no alarms present.

Data Collection Using Optional Hardware

This method is used to provide machine status to a remote computer, and is enabled with the installation of an 8 Spare M-code relay board (all 8 become dedicated to below functions and cannot be used for normal M-code operation), a power-on relay, an extra set of Emergency Stop contacts, and a set of special cables. Contact your dealer for pricing 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. Parameter 315 bit 26, Status Relays, must be enabled. Standard spare M-codes are still available for use.

The following machine statuses will be available:

* E-STOP contacts. This will be closed when the E-STOP button is pushed.
* Power ON - 115 VAC. Indicates 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 (Feed Rate is NOT 100%)
* Spare Output Relay 46 Spindle Speed Override active (Spindle Speed is NOT 100%)
* Spare Output Relay 47 Control is in EDIT mode

Alphabetical Address Codes

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

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

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

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

F Feedrate – Used to enter a feedrate; the value is entered in inches per minute or mm per minute. See Setting 77 for feedrate settings.

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 -7740.0000 and 7740.0000 for inches or between -196596.000 and 196596.000 for millimeters.

S Spindle speed command – Used to specify the spindle 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 manufacturer'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 jog handle, Cycle Start button, Feed Hold button, Axes selection dial and an increment dial.

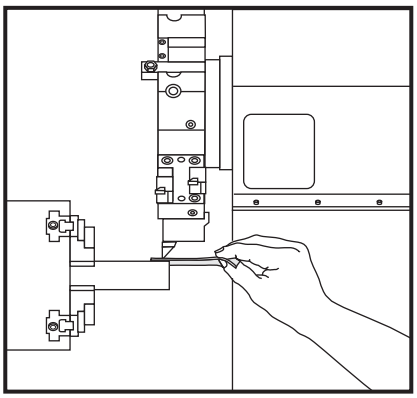
To jog the axes with the optional Remote Jog Handle (RJH) select the desired axes (X, U, Z, W, B, C or V) using the dial, choose the increment value (X1, X10 or X100) and use the jog handle to move the axes.

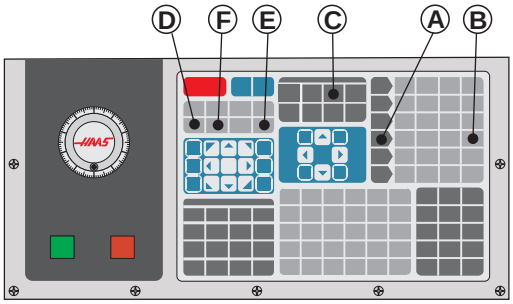
Setting the Tool Offset

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. If the diameter is known, enter the value. You can also touch off the diameter of the part and press Z Face Meas. The control will add the diameter of the part to the location of the X-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 with the position of the tool.
8. Back the tool from the part and position the tool tip so that it is touching the face of the stock.
9. Press Z Face Meas. **(E)**. This will take the current Z position and write it to the tool offset.
10. The cursor will move to the Z-axis location for the tool.
11. Press Next Tool **(F)**.

Repeat all of the previous steps for each tool in the program.

Hybrid Turret VDI to BOT Centerline Offset

Select Handle Jog and enter the Tool Geometry offset page. Select the centerline value row of and press F2.

SL-20/30 - Type in a value of 4.705 and press Write/Enter to offset the BOT tool position the correct amount from the VDI positions. 4.705 is a rough centerline. Physically measure for the correct centerline, then adjust accordingly (within a range of 4.697 - 4.713).

SL-40 - Type in a value of 5.520 and press Write/Enter to offset the BOT tool position the correct amount from the VDI positions. 5.520 is a rough centerline. Physically measure for the correct centerline, then adjust accordingly (within a range of 5.512 - 5.528).

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 just touching the face of the part.
3. Press Z Face Meas. to set part zero.
4. Repeat steps 1-3 for each tool used in the program.

Graphics Mode

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

Graphics mode can be run from Memory, MDI, DNC or Edit modes. To run a program press the SETNG/GRAPH button until the Graphics page is displayed. In Edit mode, press Cycle Start from the active program pane to enter Graphics mode. To run DNC in graphics, you must select DNC first, then go to graphics display and send your program to the machine’s 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, Page Up and Page Down to control the zoom level, and pressing the Write button. F3 and F4 are used to control the simulation speed. Note that 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 allows editing of one program while another program is running.

To activate Background Edit, press Edit until the background edit pane (on the right side of the screen) is active. Select a program to background edit (it must be a program loaded in Memory) from the list and press Write/Enter to begin editing. To select a different program to background edit, press Select Prog from the background edit pane and choose a new program from the list.

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.

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 no program name is entered, the Program Review is displayed, which allows the program currently running to be edited. Selecting any other display or pressing F4 exits from Background Edit. While a program is running, press List Prog 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.

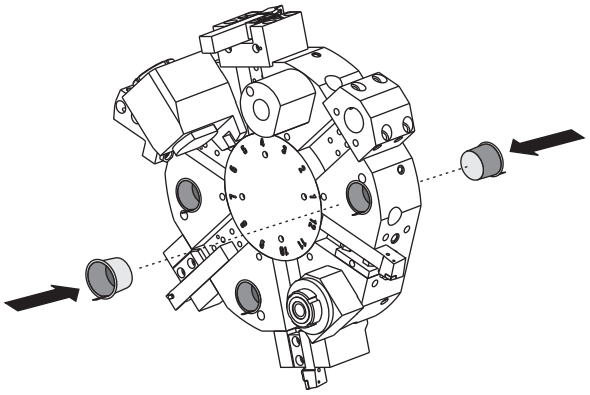
The subroutines can include an **L** or repeat count. If there is an **L**, the subroutine call is repeated that number of times before the main program continues with the next block.

Tool Turret 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.



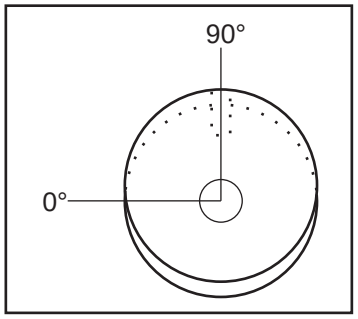
Eccentric Cam Locating Button (SL-Series only)

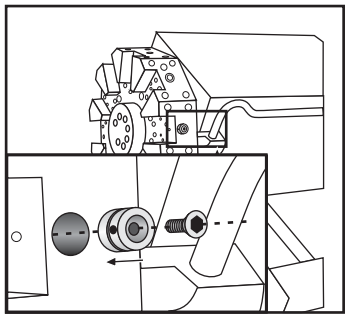
Bolt on turrets are equipped with eccentric locating buttons that allow for fine alignment of ID tool holders to spindle center line.

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.

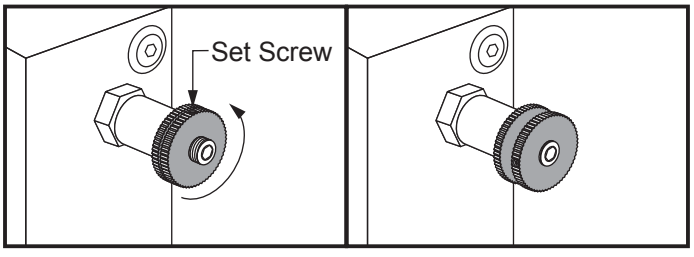
After Power Up/Restart and Zero Ret, the control will insure that the tool turret is in a normal position. To load or change tools, select MDI mode, and then press Turret Fwd or Turret Rev and the machine will index the turret on tool position. The Curnt Comds display will show what tool is currently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

Warning! Check the workpiece in the chuck or collet after any power loss. A power outage can reduce the clamping pressure on the workpiece, which may shift in the chuck or collet.

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.

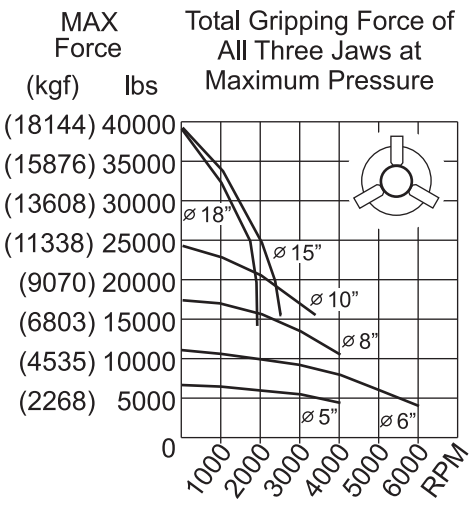
See the “Hydraulic System Information” on the machine for safe operation. Setting a pressure beyond the recommendations will damage the machine and/or inadequately hold the workpiece.

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. See the following chart.



	Maximum Operating Pressures	
	PSI	(kgf/cm²)
5” Chuck	330	(23)
6” Chuck	330	(23)
8” Chuck	330	(23)
10” Chuck	330	(23)
12” Chuck	400	(28)
15” Chuck	300	(21)
18” Chuck	300	(21)
Tailstock	400	(28)

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

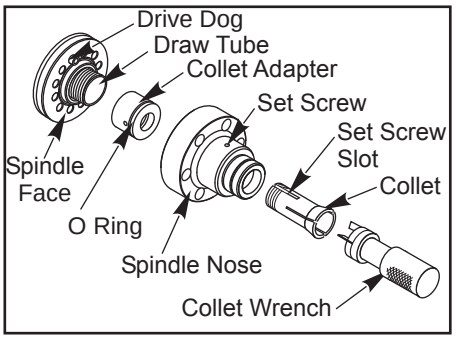
Chuck and Collet Replacement

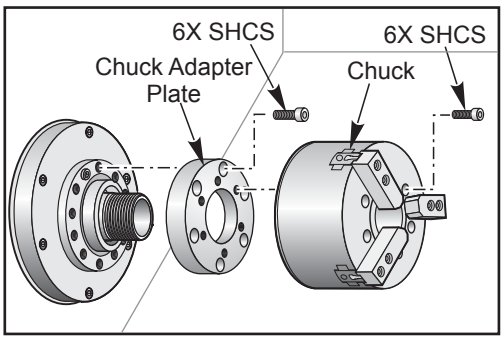
Chuck Removal

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 adapter plate.

Collet Removal

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.





Chuck Installation

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

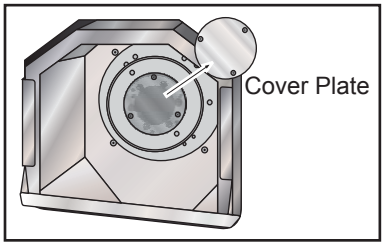
1. Clean the face of the spindle and the back face of the chuck. Position the drive dog at the top of the spindle.
2. Remove the jaws from the chuck. Remove the center cup or coverplate from the front of the chuck. If available, install a mounting guide into the drawtube and slide the chuck over it.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

Collet Installation

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 on the collet with the set screw on the spindle nose. Tighten the setscrew on the side of the spindle nose.

Draw Tube Cover Plate

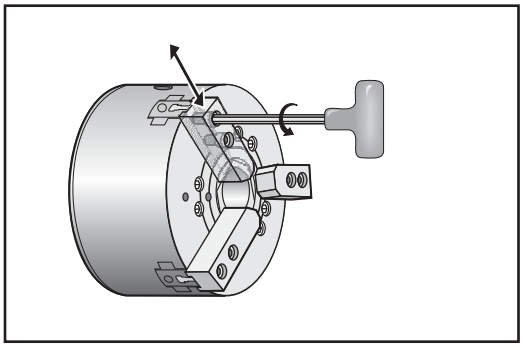
It is necessary to remove the cover plate at the far end of the drawbar when using a barfeeder. Replace the cover plate anytime bar stock is not being fed automatically.



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.



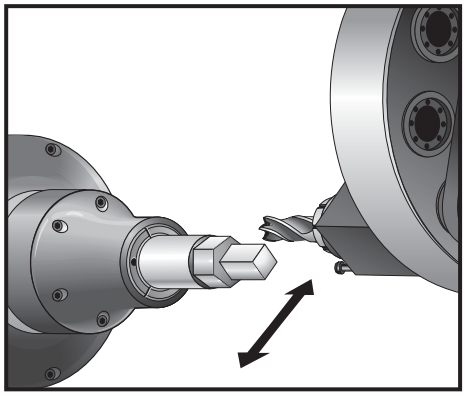
1. Use a hex key to loosen the two SHCS attaching the jaw to the chuck.
2. Slide jaw to new position and re-tighten the two SHCS.
3. Repeat procedure for remaining two jaws. Jaws must remain concentric.

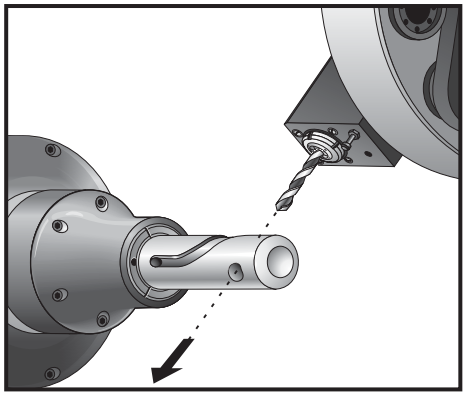
Taper Compensation

Deflection of the part occurs if it is not supported precisely in the center, or if 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. The range of this value is betweeen 0 and .005; this value represents a slope.

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.
- 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.

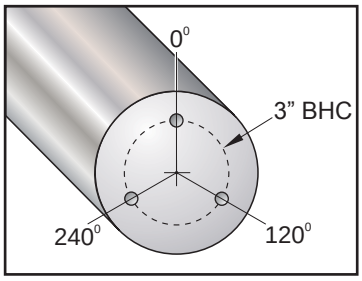
M19 Angle CMD (Optional)

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 rounds 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
M13
```



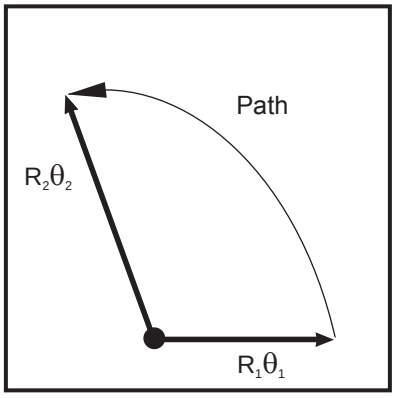
Synchronous Milling

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.

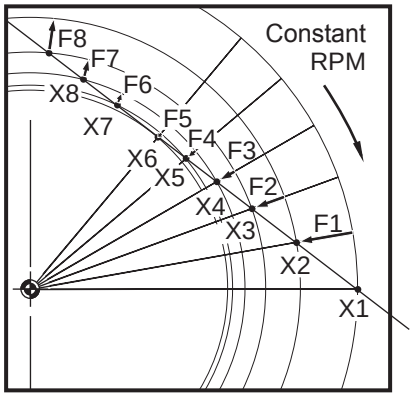
G32 is commonly used to create threads, the spindle rotates at a constant rpm and constant Z-axis motion begins at the same reference Z-axis mark for each stroke. Many strokes can be repeated because the reference mark sets the location of the start thread.

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 simplify the creation of simple geometric shapes. G77, Flatting Cycle, automates the motions of 1 or more sided uniform shapes.

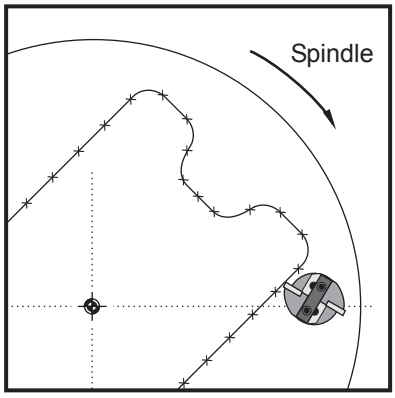
In addition to synchronous motions, G5 is a motion mode that accepts point to point commands and controls the spindle like a rotary device, similar to a rotary table motion. It is commanded in angle and distance point to point motion.



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

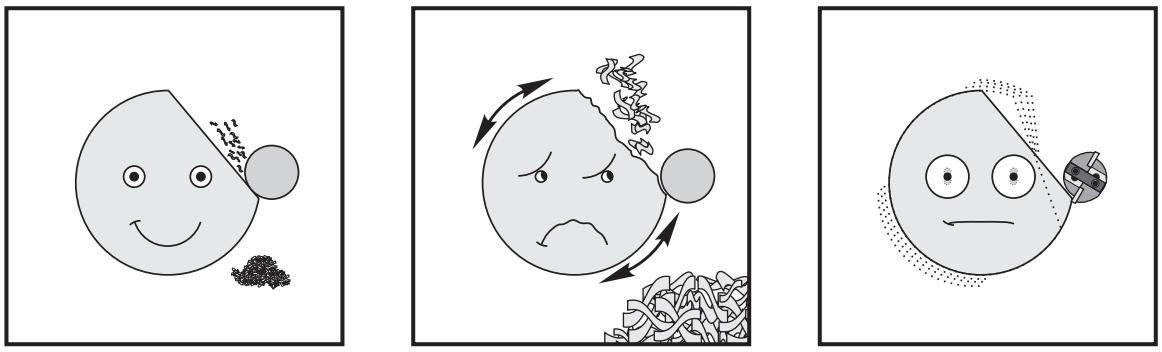
Introduction

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.

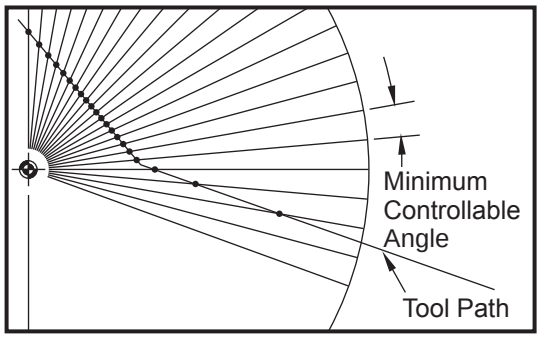


Light Cut *Heavy Cut* *Large Cutter*

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

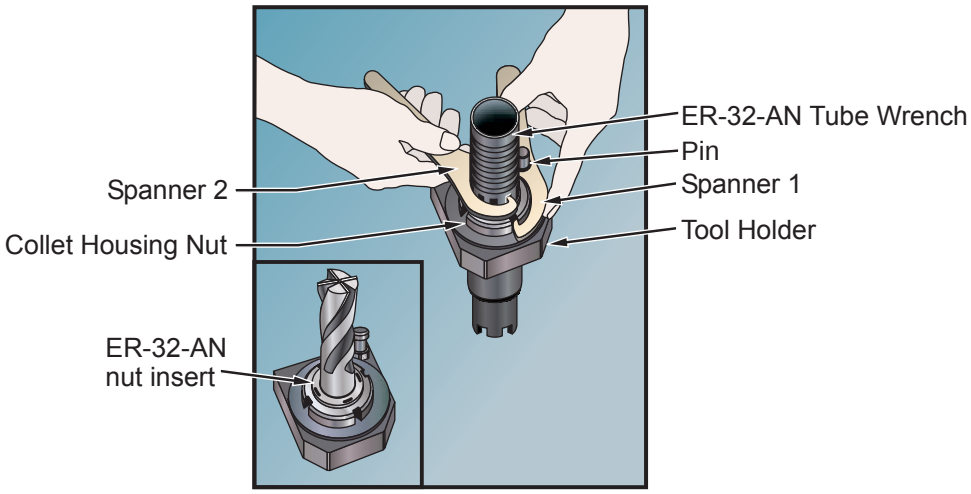
The limit also applies to positioning the spindle in general. This has an effect when trying to perform cuts that are close to centerline.

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.



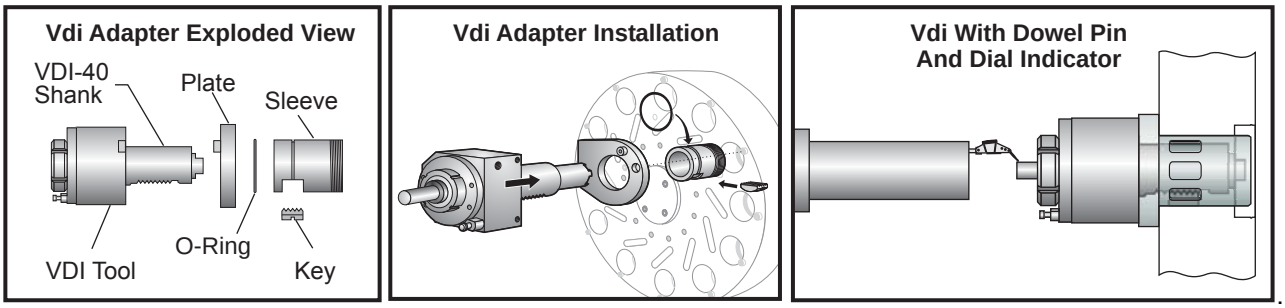
Live Tooling Installation

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 turret.



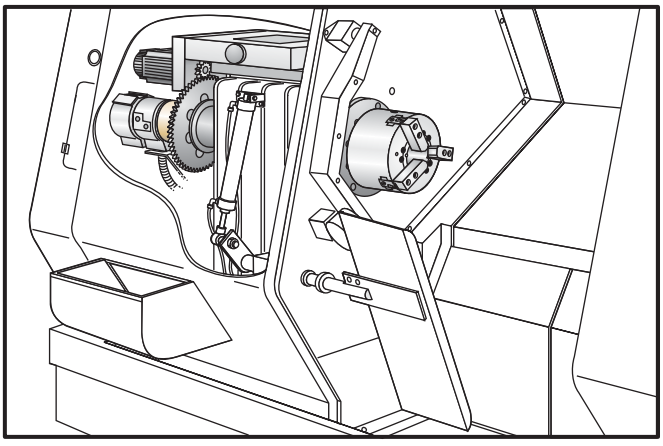
Installation Procedure:

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.

C-Axis operation is dependant on the mass, diameter and length of the workpiece and/or the workholding (chuck). Contact the Haas Applications Department if any unusually heavy, large diameter, or long configuration is used.



Operation

M154 C-axis engage
M155 C-axis disengage

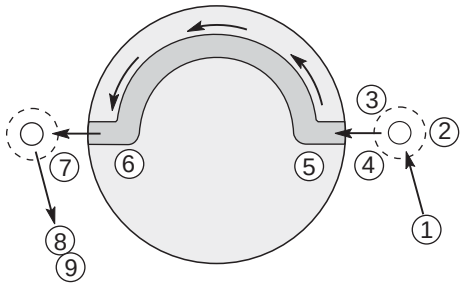
Setting 101, Diameter, is used to calculate the feed rate.

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.)
```

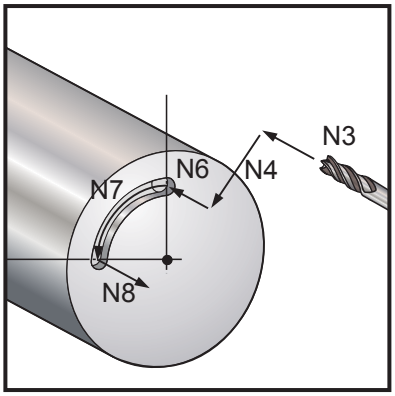
Sample Programs



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, however, in Cartesian, only end points are necessary. This feature allows face machining programming in the Cartesian coordinate system.

Programming notes:
Programmed moves should always position the tool centerline.

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.

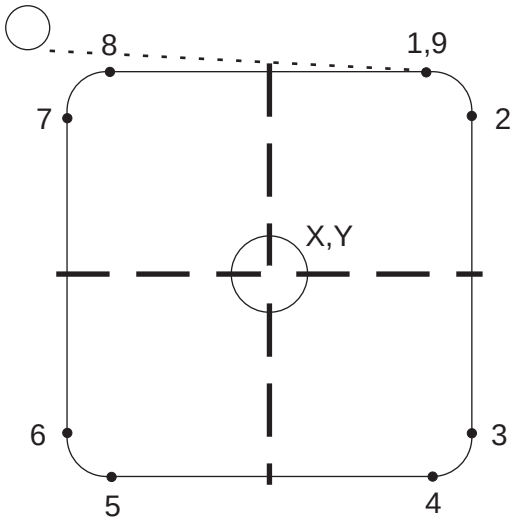
Cartesian to Polar conversion is a modal command (see the G-code section).

Cartesian Interpolation

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

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 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 page is used to enter the amount that the tool path is shifted in the radius column. The offset is entered as a radius value for both the geometry values. The compensated value is calculated by the control from the values entered in the Radius. Cutter radius compensation is only available using G17 within G112.

- G41 will select cutter compensation left.
- G42 will select cutter compensation right.
- G40 cancels cutter compensation.

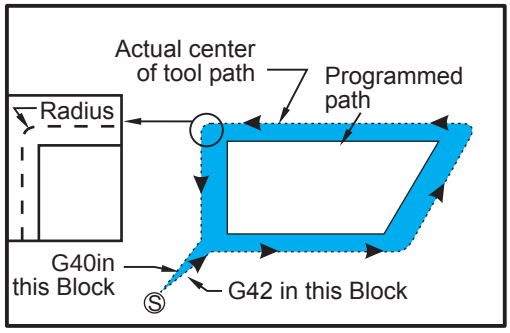
Offset values entered for the radius should be in positive numbers. If the offset contains a negative value, cutter compensation will operate as though the opposite G code was specified. For example, a negative value entered for a G41 will behave as if a positive value was entered for G42.

When 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.

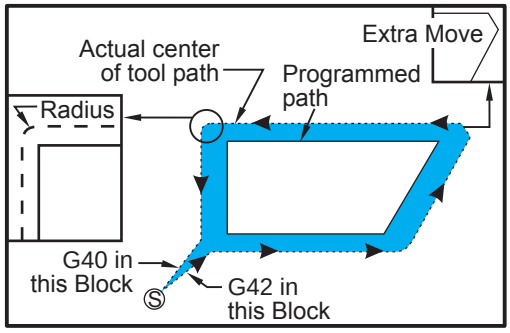
When selecting Fanuc for Setting 58, the control does not require that the tool cutting edge be placed along all edges of the programmed contour, preventing over-cutting. Outside angles less than or equal to 270° are joined by a sharp corner and outside angles of more than 270° are joined by an extra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

G42 Cutter Compensation - (YASNAC)



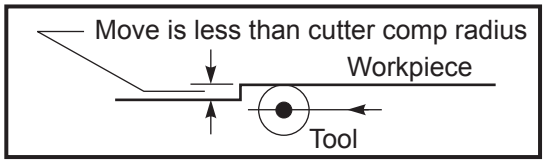
G42 Cutter Compensation - (FANUC)



Entry and Exit

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 offset column. In the block that turns off cutter compensation, compensation will be turned off as the tool reaches the end of block position. 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 the tool position until the next **X** or **Y** move is encountered.

When turning on cutter compensation in a move that is followed by a second move at an angle of less than 90°, 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 diagrams on the following pages illustrate the differences between type A and type B for both Fanuc and Yasnac settings (Setting 58).



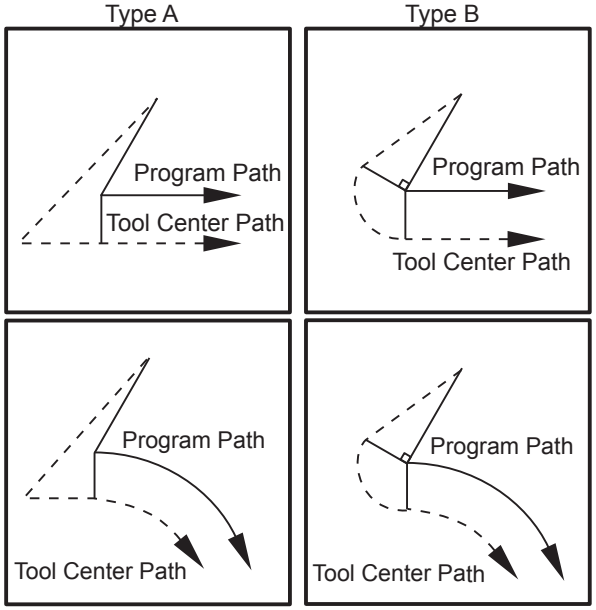
Improper Cutter Compensation Application

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.

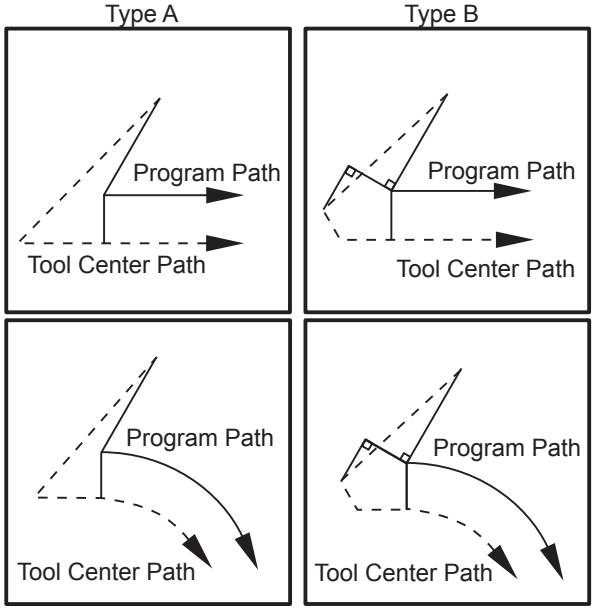
Feed Adjustments in Cutter Compensation

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)

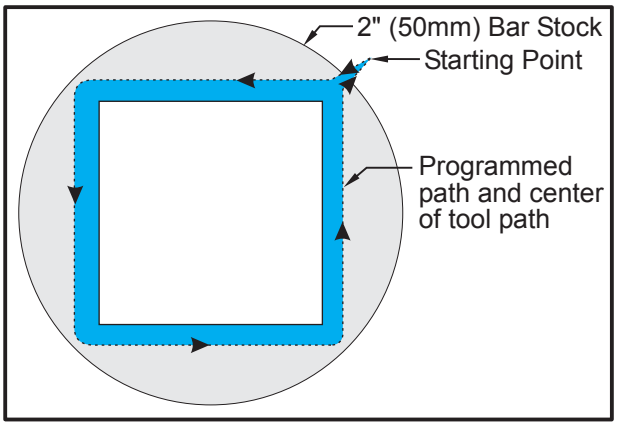


Cutter Compensation Entry (Fanuc style)



Cutter Compensaiton Example

```
T0101 (Tool .500" 4-Flute Endmill)
G54
G17
G112
M154
G0G98Z.3
G0X1.4571Y1.4571
M8
G97P3000M133
Z.15
G01Z-.25F20.
G01G42X1.1036Y1.1036F10.
G01X.75Y.75
G01X-.5
G03X-.75Y.5R.25
G01Y-.5
G03X-.5Y-.75R.25
G01X.5
G03X.75Y-.5R.25
G01Y.75
G01X1.1036Y1.1036
G0G40X1.4571Y1.4571
G0Z0.
G113
G18
M9
M155
M135
G28U0.
G28W0.H0.
M30
%
```



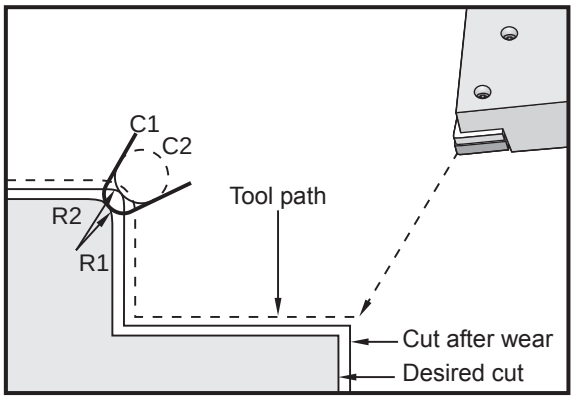
Tool Nose Compensation

Introduction

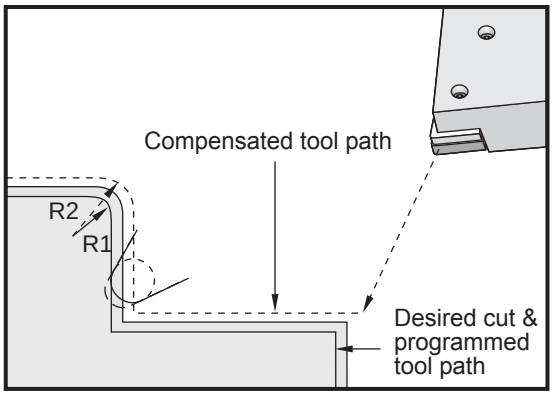
Tool Nose Compensation is a feature that allows the user to adjust a programmed tool path in response to different cutter sizes or for normal cutter wear. The user can do this by entering minimal offset data at run-time without any additional programming effort.

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 Radii

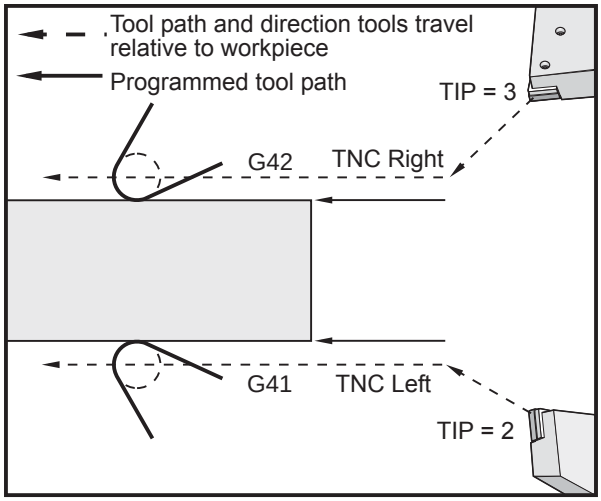


Path Generated when Tool Nose Compensation is used

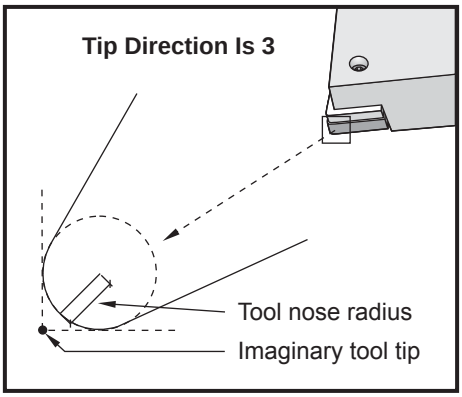
Note that the second programmed path coincides with the final part dimension. Although parts do not have to be programmed using tool nose compensation, it is the preferred method because it makes program problems easier to detect and resolve.

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 a 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 required. In a depart move, the control moves 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) to 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.

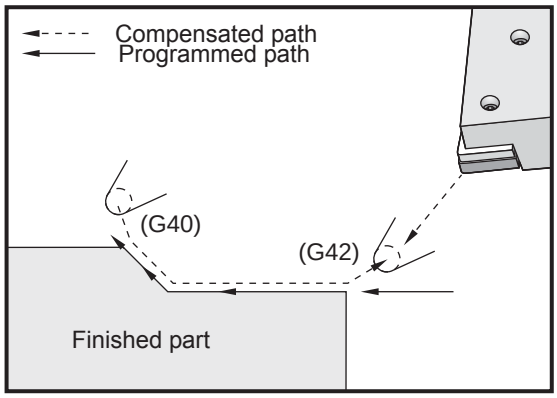
Tool Geometry Offset – Set the tool length geometry and clear the length wear offsets of each tool.

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: an alarm will be generated indicating compensation interference, or the incorrect geometry will be seen generated in graphics mode.

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

Approach and Departure Moves For Tool Nose Compensation

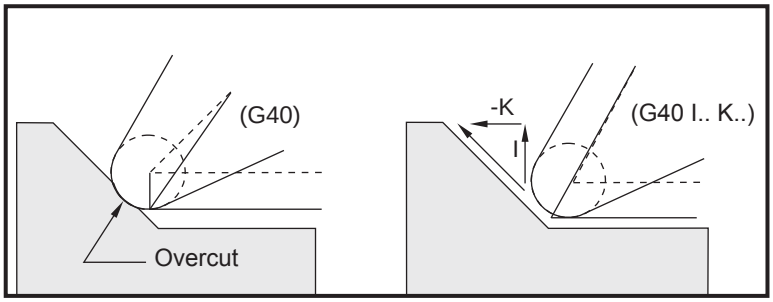
The first X or Z motion in the same line that contains a G41 or G42 is called the “Approach” move. The approach must be a linear move, that is a G01 or G00. The first move is not compensated, yet at the end of the approach move the machine position will be fully compensated. See the following figure.



Approach and Departure Moves.

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.

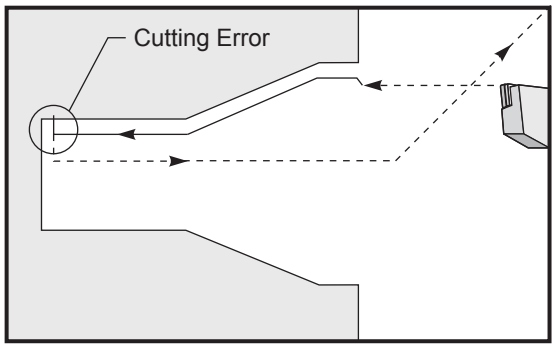
Small adjustments (positive values) to the radius offset during production runs should be placed in the wear offset page. This allows the operator to easily track the wear for a given tool. As a tool is used, the insert will generally wear so that there is a larger radius at the end of the tool. When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



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, 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).

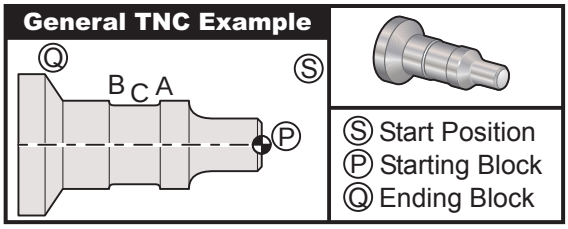
The following canned cycles will ignore tool nose radius compensation. Cancel tool nose compensation before any of these canned cycles.

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

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	Description
%	
O0811 (G42 Test BCA)	(Example1)
N1 G54 S1000	
T101	(Select tool 1 and offset 1 - tip direction for offset 1 is 3)
G97 S500 M03	
G54 G00 X2.1 Z0.1	(Move to point S)
G96 S200	
G71 P10 Q20 U0.02 W0.005 D.1 F0.015	(Rough P to Q with T1 Using G71 and TNC)
	(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	(A)
G02 X1. Z-1.625 R0.125	
G01 Z-2.5	
G02 X1.25 Z-2.625 R0.125	(B)
G01 Z-3.5	
X2. Z-3.75	
N20 G00 G40 X2.1	(TNC Cancel)
G97 S500	
G28	(Zero for tool change clearance)
M01	
N2 G50 S1000	
T202	
G97 S750 M03	(Select tool 2 and offset 2 Tip direction is 3)
G00 X2.1 Z0.1	(move to point S)
G96 S400	
G70 P10 Q20	(Finish P to Q with T2 using G70 and TNC)
G97 S750	
G28	(Zero for tool change clearance)
M01	
N3 G50 S1000	
T303	(Select tool 3, offset 3 Tip direction is 3)
G97 S500 M03	(Groove to point B Using Offset 3)
G54 G42 X1.5 Z-2.0	(Move to point C TNC right
G96 S200	
G01 X1. F0.003	
G01 Z-2.5	
G02 X1.25 Z-2.625 R0.125	(B)
G40 G01 X1.5	(TNC cancel - Groove to point A using offset 4)
T313	(Change offset to other side of tool)
G00 G41 X1.5 Z-2.125	(Move to point C - TNC approach)
G01 X1. F0.003	
G01 Z-1.625	
G03 X1.25 Z-1.5 R0.125	(A)
G40 G01 X1.6	(TNC cancel)
G97 S500	
G28	
M30	
%	

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

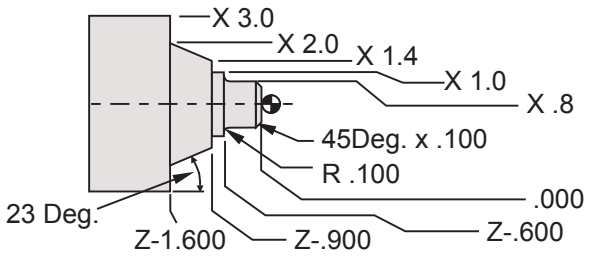
Program Example	Description
%	
O0813	*(Example 3)*
G50 S1000	
T101	*(Select tool 1)*
G00 X1.5 Z.1	*(Rapid to start point)*
G96 S100 M03	
G71 P80 Q180 U.01 W.005 D.08 F.012	*(Rough P to Q with T1 using G71 and TNC)*
	(Define Part Path PQ sequence)
N80 G42 G00 X0.6	(P) (G71 Type I, TNC right)
G01 Z0 F0.01	(Start of finish part path)
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	(Q) (End of part path)
N180 G40 G00 X3.0 M05	(TNC cancel)
G28	(Zero X for tool change clearance)
M30	
%	

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.

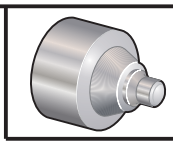
Example 3

TNC with a G72 roughing canned cycle

TNC with G72 Example



Preparation



Setting 33 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**
%	
O0814	(Example 3)
G50 S1000	
T101	(Select tool 1)
G00 X3.5 Z0.1	(Move to start point)
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	(P) (G72 Type I, TNC left)
G01 X2. F0.005	
X1.4 Z-0.9	
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

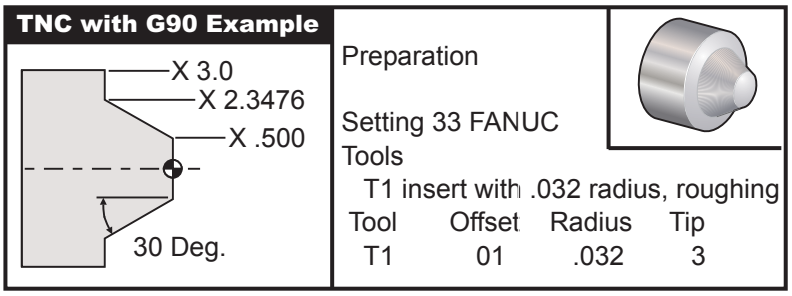
T1 Insert with .032 radius, roughing
T2 Insert with .016 radius, finishing

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

Program Example	Description
%	
O0815	(Example 4)
T101	(Select Tool 1)
G50 S1000	
G00 X3.5 Z.1	(Move to point S)
G96 S100 M03	
G73 P80 Q180 U.01 W0.005 I0.3 K0.15 D4 F.012	(Rough P to Q with T1 using G73 and TNC)
N80 G42 G00 X0.6	(Part path PQ sequence, G72 Type I, TNC right)
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	(Q)
G00 Z0.1 M05	(TNC Cancel)
(******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.

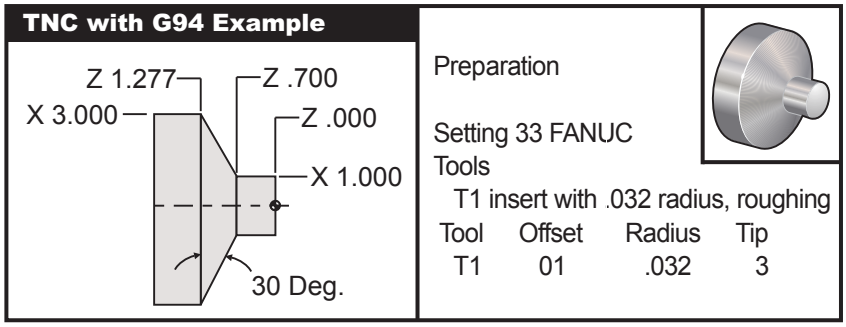
Example 5
TNC with a G90 modal rough turning cycle



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
X2.45                                          (Optional Additional Passes)
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 **Description**

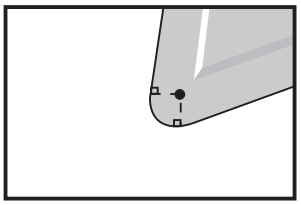
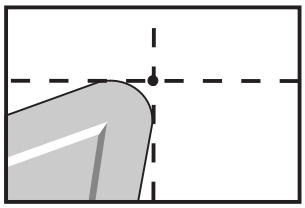
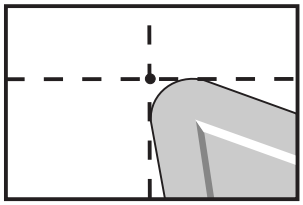
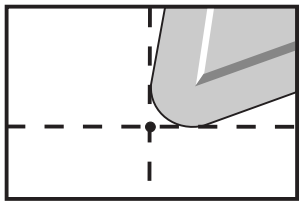
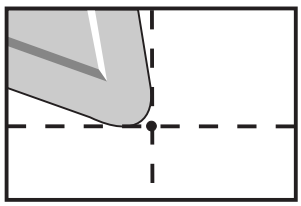
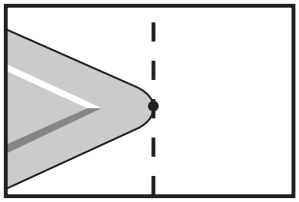
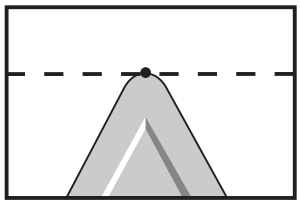
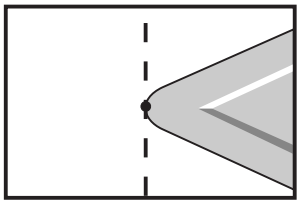
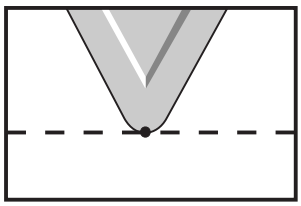
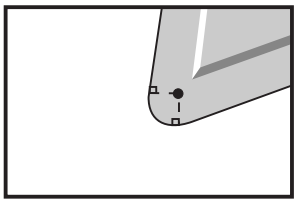
```
%
O0817                                          (Example 6)
G50 S1000
T101                                           (Select tool 1)
G00 X3.0 Z0.1                                  (Move to start point)
G96 S100 M03
G94 G41 X1.0 Z-0.5 K-0.577 F.03                (Rough 30° angle to X1. and Z-0.7 using G94 and TNC)
Z-0.6                                          (Optional additional passes)
Z-0.7
G00 G40 X3. Z0.1 M05                           (TNC Cancel)
G28                                            (Zero for tool change clearance)
M30
%
```

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

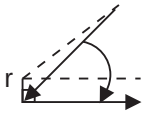
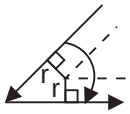
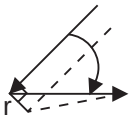
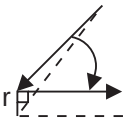
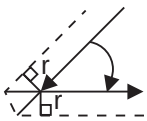
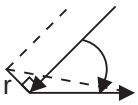
Use the values found on the charts (angle and radius size) to calculate the correct tool path position for the program.

Tool Nose Compensation Geometry

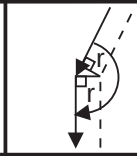
The following figure shows the various geometries of tool nose compensation. It is organized into four categories of intersection. The intersections can be: 1) linear to linear, 2) linear to circular, 3) circular to linear, or 4) circular to circular. Beyond these categories the intersections are classified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

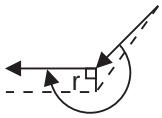
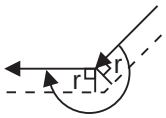
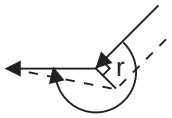
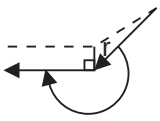
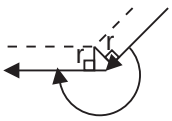
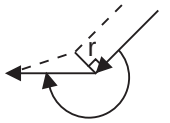
Angle: <90

	Approach	Mode to Mode	Departure
G41			
G42			

Angle: >=90, <180

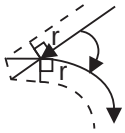
	Approach	Mode to Mode	Departure
G41			
G42			

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)

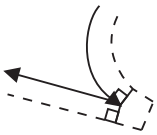
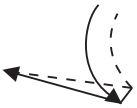
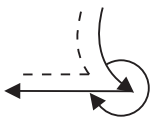
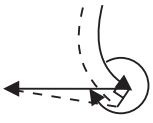
Angle: <90

	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

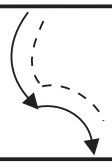
	Approach	Mode to Mode	Departure
G41	Not Permitted		
G42	Not Permitted		

Circular-to-Circular (Type A)

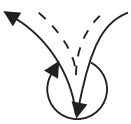
Angle: <90

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

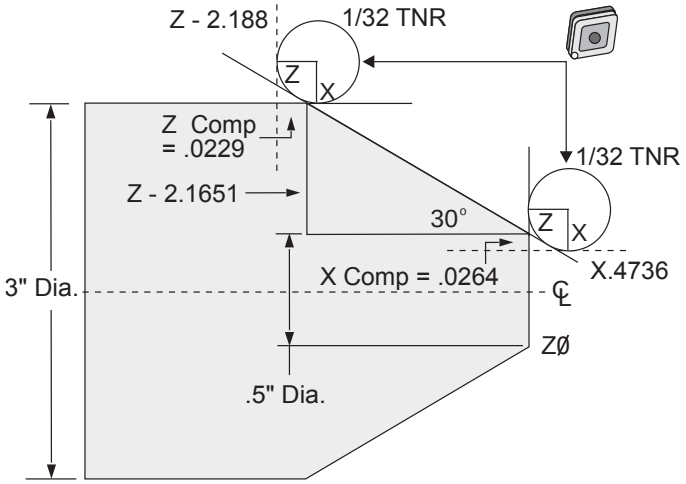
Angle: >=90, <180

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

Angle: >180

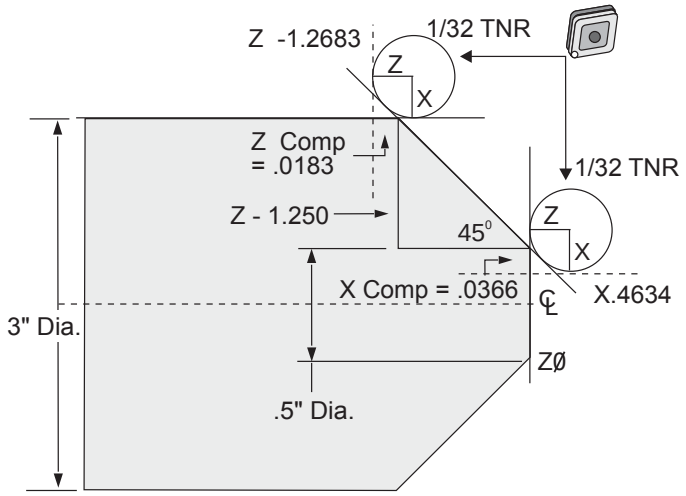
	Approach	Mode to Mode	Departure
G41	Not Permitted		Not Permitted
G42	Not Permitted		Not Permitted

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)

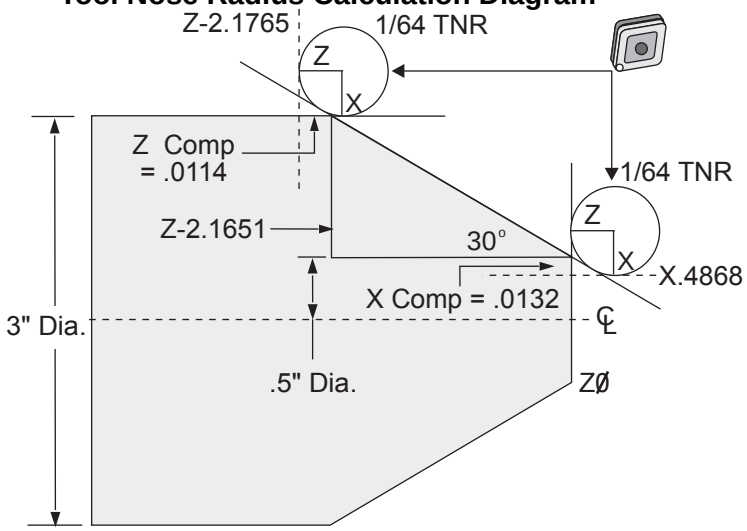
Note: Compensation Value For 45° Angle

Tool Radius And Angle Chart (1/32 RADIUS)

The X measurement calculated is based on part diameter.

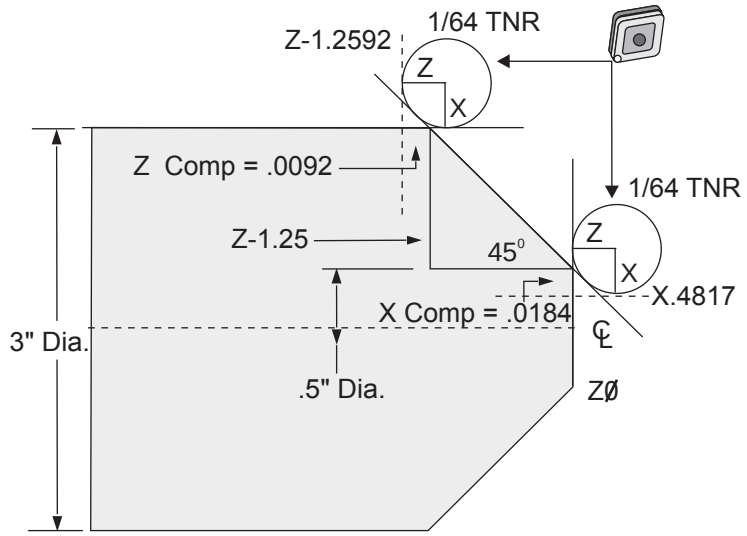
ANGLE	Xc CROSS	Zc LONGITUDINAL	ANGLE	Xc CROSS	Zc LONGITUDINAL
1.	.0010	.0310	46.	.0372	.0180
2.	.0022	.0307	47.	.0378	.0177
3.	.0032	.0304	48.	.0386	.0173
4.	.0042	.0302	49.	.0392	.0170
5.	.0052	.0299	50.	.0398	.0167
6.	.0062	.0296	51.	.0404	.0163
7.	.0072	.0293	52.	.0410	.0160
8.	.0082	.0291	53.	.0416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

Tool Nose Radius Calculation Diagram



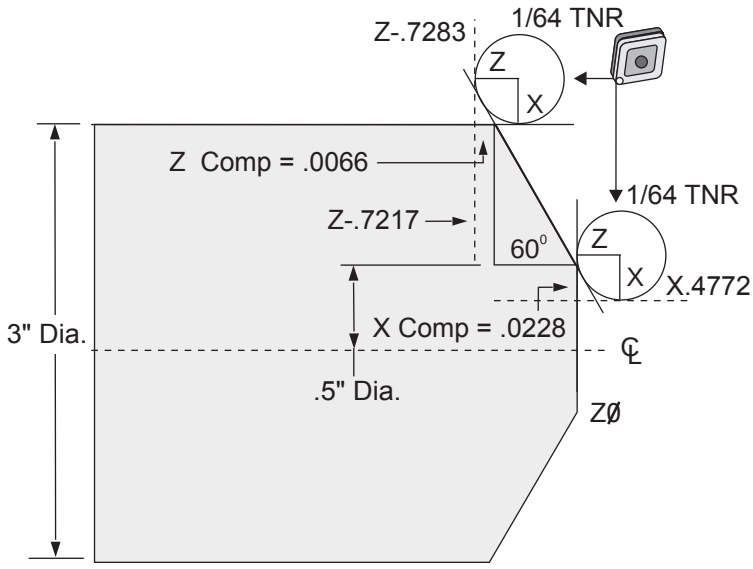
Program	
Code	Compensation (1/64 TNR)
G0 X0 Z.1	
G1 Z0	
X.4868	(X.5 - 0.0132 Comp)
X 3.0 Z-2.1765	(Z-2.1651 + 0.0114 Comp)

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			

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

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 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.

FANUC 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.

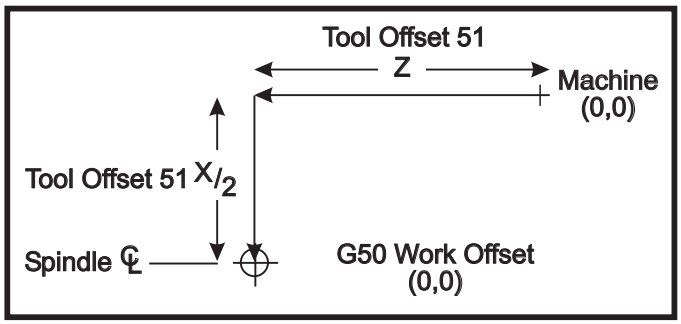
FANUC Common Coordinate System - The common (Comm) coordinate system is found on the second work coordinate offsets display page just below the global coordinate system (G50). The common coordinate system is retained in memory when power is turned off. The common coordinate system can be changed manually 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.

YASNAC 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.

YASNAC Tool Offsets - There are two offsets available: geometry offsets and wear offsets. Geometry offsets adjusts 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.

Geometry offsets are not available; they are replaced with tool shift offsets (50 tool shift offsets numbered 51 - 100). YASNAC tool shift offsets modify the global coordinate to allow for varying tool lengths. Tool shift offsets must be used prior to calling for the use of a tool with 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.




```
000101
N1 G51 (Return to Machine Zero)
N2 G50 T5100;  (Offset for Tool 1)
.
.
.
%
```

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 screen displays the spindle and axis loads, the current feed and speed, positions, and the currently active codes while a program runs. Changing display modes will change the information displayed.

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), and press Alter. This will transfer the MDI program into your list of programs under that program number.

Program Review - Program Review allows you to cursor through and review the active program on the right side of the display screen, while also viewing the same program as it is running on the left side of the screen. To enter Program Review, press F4 while the edit pane containing the program is active.

Background Edit - This feature allows you to edit a program while a program is running. Press EDIT until the background edit pane (on the right side of the screen) is active. Select a program to edit from the list and press Enter. Press Select Prog from this pane to select another program. Edits are possible as the program runs, however, edits to the running program will not take effect until the program ends with an M30 or RESET.

Graphics Zoom Window - F2 will activate the zoom window when in graphics mode. Page down zooms in and page up expands 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.

Copying Programs - In the Edit Mode a program can be copied (Insert) into another program, a line, or a block of lines in a program. Start defining a block with the F2 key, 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 Files - Load multiple files by selecting them in the device manager, then press F2 to select a destination.

Editing Programs - Pressing the F4 key while in Edit mode will display another version of the current program to in the right-hand pane. 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 F2. This can also be done through the popup help menu. Press F1, then select the option from the list. Type the new program name and press Enter.

Several programs can be sent to the serial port. Select the desired programs from the program list by highlighting them and pressing Enter. Press Send RS232 to transfer the files.

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 the offset.

Pressing Offset will toggle back and forth between the Tool Length Offsets and Work Zero Offset panes.

Settings and Parameters

Jog handle can be used to scroll through settings and parameters, when not in jog mode. You can also locate a known parameter or setting quickly by typing the number and pressing the up or down arrow key.

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 **On**, memory edit functions are locked out. When **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 program beginning.

Scale Integer F (Setting 77) 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 slow at corners **only** when needed.

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

Cycle Start/Feed hold (Setting 103) when 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) allows the jog handle to be used to step through a program. Reversing the jog handle generates 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 - Returns 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 stops 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 Handle Jog.

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.

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, will be evaluated when the Write key is pressed and the result (89.8 in this case) displayed in the calculator box.

Intuitive Programming System (IPS)

Introduction

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

To enter the IPS menu press MDI/DNC, then PROGRM CONVRS. 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.

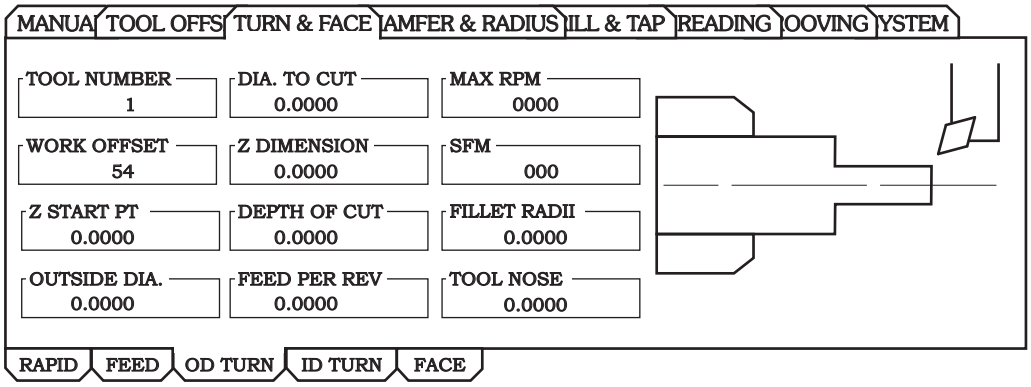
To exit the IPS menus press any of the Display keys, except Offset. Press MDI/DNC, then PROGRM CONVRS to return to the IPS menus.

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

Automatic Mode

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

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



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.

Turning the Option On and Off

The IPS option is toggled off and on using parameter 315 bit 31 (Intuitive Prog Sys). Lathes with the option can be returned to the traditional Haas program displays by turning this parameter bit to 0.

To do this, Press the PARAM/DGNOS button. Enter “315” and press the down arrow. Use the left and right arrow, or jog handle to scroll to the last parameter bit (Intuitive Prog Sys). Press the Emergency Stop Button, type “0” (zero) and press enter.

To re-activate the IPS option, scroll to the parameter bit as previously described, press the Emergency Stop button, type “1” and press Enter. You will need the activation code; see the parameter list that shipped with the machine or contact your dealer.

Auxiliary Axis Control

An additional external positioning axis may be added to the directly controlled axes. This axis, V, may be commanded directly from the program, but 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 pins 1, 2, 3, and 7 directly from the second (lower) serial port of the CNC to the servo control.

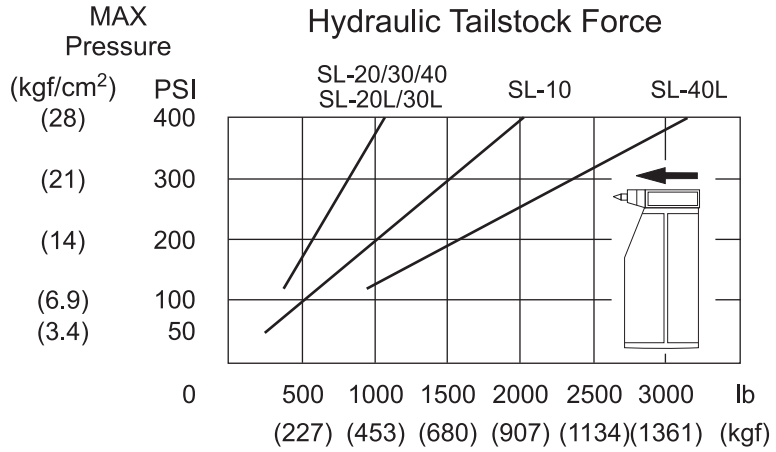
TAILSTOCK

The optional Tailstock (not field installable) 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 through program code, in jog mode, or 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. High pressure is called "rapid" and can be programmed with G00. 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.

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 the machine or serious damage may occur. Adjust Settings 93 and 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, and 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:

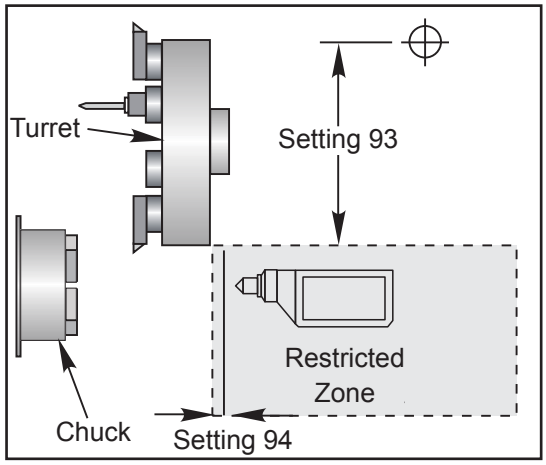
1. Place the control in Zero Ret and Home G28 all axes.
2. Select the X-axis and move the turret in front of the tailstock center tip.
3. Move the Z-axis so that the rear of the tool turret is within about 0.25 inches of the tailstock tip.
4. Find the Z-axis “machine” position on the display, enter this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

A properly defined restricted zone will handle most, but not all, conflicts between the turret and 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 Settings 94, 94 105, 106, and 107; see the Settings chapter for more information.



Tailstock Restricted Zone

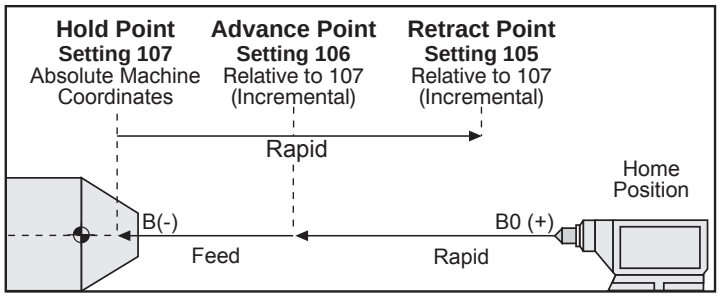


Diagram of Settings 105, 106, and 107

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; 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 toward the retract point (M22). If tailstock is right of the retract point, the foot pedal will also move the tailstock toward the retract point (M22). If tailstock is at retract point, pressing the foot pedal will move the tailstock toward 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 through program code, in jog mode, or by the foot switch.

The SL-10 tailstock consists of a fixed head, and a moveable center rod. Therefore, the only moving part is the tailstock center. Pressing PowerUp/Restart or AutoAll Axes 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.

SL-10 Tailstock Foot pedal Operation

Pressing the foot pedal will advance or retract the foot pedal. However, pressing and holding the pedal for 5 seconds, will retract the tailstock all the way and maintain retract pressure to ensure the tailstock does not creep forward. Use this method to stow the tailstock anytime it is not in use.

The tailstock position can change over time if it is left at a position that is not fully retracted or not in contact with a part. This is due to normal hydraulic system leakage.

WARNING

• 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 Clearance as necessary.

• Feed Hold will not stop the hydraulic tailstock. The Emergency Stop button is the only way to stop the tailstock

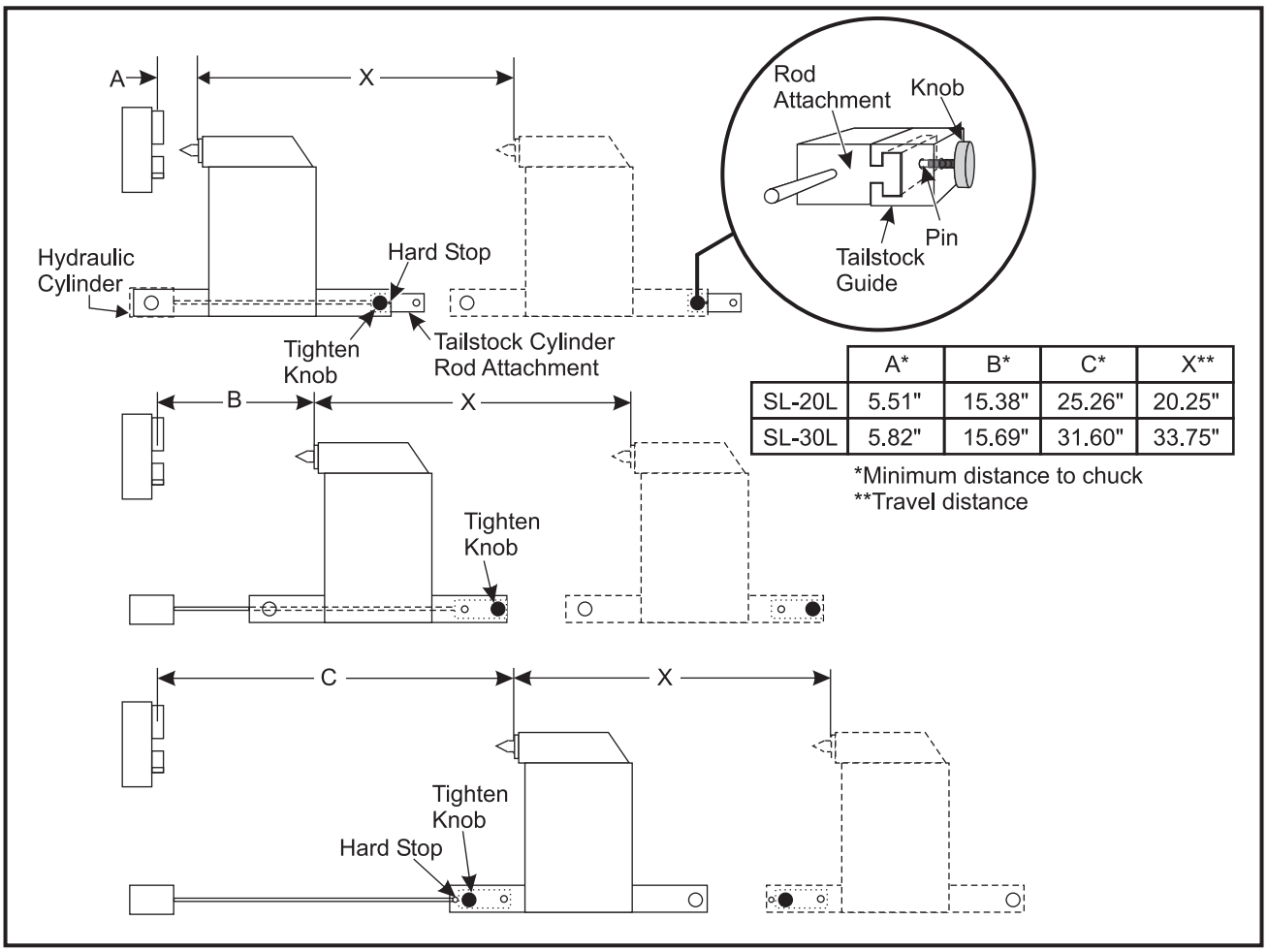
Tailstock Programming

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

CAUTION! Do not use an 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.

Adjustable Tailstock

The adjustable tailstock for the SL-20L and SL-30L uses two pins to attach the tailstock to the hydraulic cylinder in three different ways, creating 20.25” (SL-20L) and 30.75” (SL-30L) travels from short, medium, and long distances to the chuck.



	A*	B*	C*	X**
SL-20L	5.51"	15.38"	25.26"	20.25"
SL-30L	5.82"	15.69"	31.60"	33.75"

*Minimum distance to chuck
**Travel distance

Tailstock Adjustment

1. Position the tailstock at full travel to the left.

2. Press Emergency Stop to turn off hydraulics and prevent accidental machine motion.

3. Disengage the hydraulic cylinder attachment from the tailstock by unscrewing the knob.

4. Manually slide the tailstock to align the cylinder rod attachment with the tailstock in the correct position for the desired travel range. For the shortest minimum distance to the chuck, slide the tailstock left until it hits the hard stop. For the middle minimum distance to the chuck, slide the tailstock until the right side of the tailstock guide is flush with the right side of the cylinder rod attachment. For the longest minimum distance to the chuck, slide the tailstock right until it hits the hard stop. See Illustration.

5. When the cylinder attachment and pin are aligned in the desired position, engage the pin by screwing in the knob.

6. Reset Emergency Stop and re-zero the tailstock to resume machine operation.

Parts Catcher

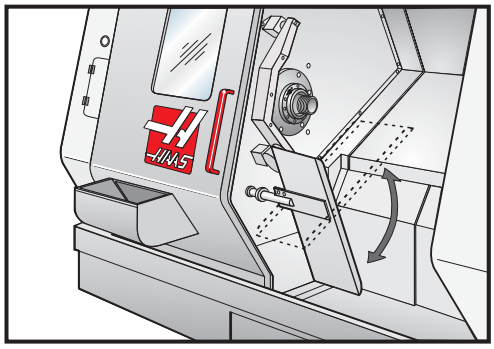
The parts catcher is an automatic part retrieval system designed to work with bar feed applications. The parts catcher, is commanded using M Codes (M36 to activate and 37 to deactivate). The parts catcher rotates to catch finished parts and directs them into the bin mounted on the front door.

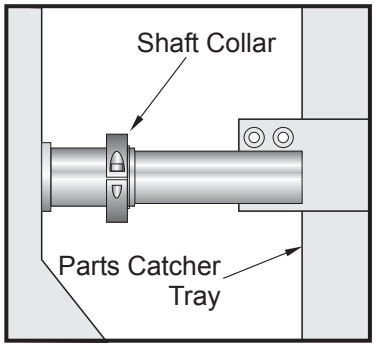
Operation

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

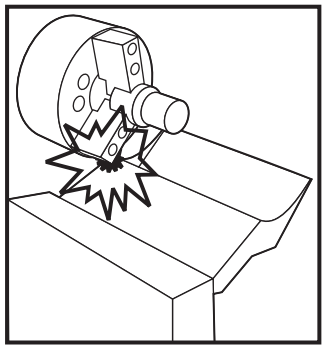
Check the Z-axis, X-axis, tool and turret position during part catcher actuation to avoid possible collisions during operation.

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.

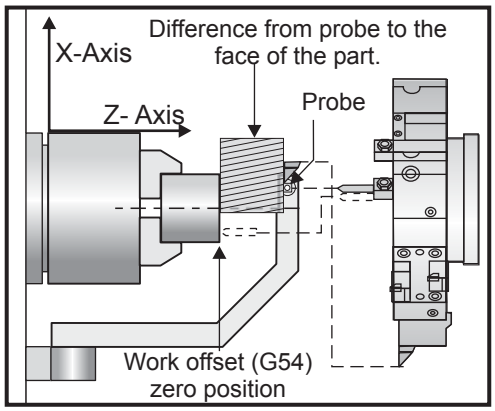


Tool Pre-Setter

The Tool Pre-Setter allows an operator to quickly set up a machine with the necessary tool and work offsets rather than having to manually record the tool offsets. Each tool has to be “touched-off” on the probe (a known point in space) to record the tool’s location. When the locations are recorded, the tool’s location relative to the part has to be recorded. At this point the user only has to touch-off one tool on the parts zero position, and the machine adjusts the work offsets for every tool. These tool and work offsets are used in order to tell the machine where the part is in relation to the “home” position, and how far a tool has to travel to get to the part.

When the probe is down, the machine will not allow any programs to be run and the axes can only be moved using the handle jog feature. The “Tool Offset” dimension will be recorded in the offset page under the corresponding offset number G52-G59 (G54 is typically used unless otherwise specified).

NOTE: There are up to 200 offset values available so that multiple offsets can be recorded for a single tool. In a program, an example would look like: “T417” which would select tool number 4 with offset number 17, etc.



Operation

IMPORTANT: Automatic storing of the machine’s position can only be performed when the jog buttons are used. Once the probe is touched, the control will beep, the turret will halt, and the position of the tool will be stored. The operator will not be able to continue jogging towards the probe. This prevents the operator from damaging the probe and ensures greater accuracy.

NOTE: If the operator jogs the tool tip into the probe too fast, the probe may slip off the tool tip.

The axis last jogged will be disabled; use the other axis to jog the turret away from the probe. All axes will then be re-enabled. If this does not work, raise the probe arm to its home position. If this is not possible, the prox switch that senses that the arm in the upright position can be activated, which will re-enable all axes, and the tool can be jogged away.

WARNING

When changing tools, always back the tool a safe distance away from the probe or you could crash the tool into the arm!

Setting Tool Geometry and Tool Shift Offsets Using the Probe

1. Setting 33, Coordinate System, controls whether the current tool offsets obtained while using the tool setter are stored in Tool Geometry (FANUC) or Tool Shift (YASNAC).

2. Index the turret to the tool to probe.

3. Jog the tool to a safe position and lower the arm.

4. Jog the turret in the X direction until the tool tip is close to the probe (use .001” jog speed). Press the X-axis button until the tool 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. When retouching a tool, Setting 64 needs to be off to ignore the value on G54.

IMPORTANT! The jog buttons must be used to automatically store the position of the tool. The jog handle can also be used; however, the values will have to be manually entered into the control.

5. Next, 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. Index the turret to the tool to probe.

7. Jog the tool in the Z direction until it touches the probe (Use .001” jog speed). 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. Index the turret to the desired tool and touch off the face of the part.

3. Press Z Face Mesur to reference the rest of the tools to the face of the part.

Lathe Tool Pre-setter Alignment

1. Install a turning tool in the tool 1 station of the tool turret and clamp a piece of material in the spindle to be able to turn a diameter on the material.

2. Use the turning tool in station 1 to take a small cut on the diameter of the material clamped in the spindle.

3. Jog the turning tool away from the part in the Z-axis only - do not jog the X-axis away from the diameter. The tool position is needed to set the geometry offset for the tool in station 1 using the X Diameter Measure Button.

4. Measure the diameter of the cut made on the work piece with a micrometer and press the X DIA MEASURE button – enter the diameter that was measured.

5. Write down the Geometry Offset for tool number 1. Go to the settings page and change setting 59 and 63 to 0 (zero).

6. Pull down the Tool Pre-setter and touch tool #1 to the probe. Subtract the new Geometry Offset value for tool 1 from the Offset value you had written down previously. Enter this value into setting 59.

7. Measure the tool probe width and multiply it by two. Subtract that value from Setting #59, and enter this new value into setting 60 (X- probe offset).

8. Enter 0 (zero) for setting 61. The value for setting 62 is the probe width as a negative number and setting 63 is the probe width as a positive number.

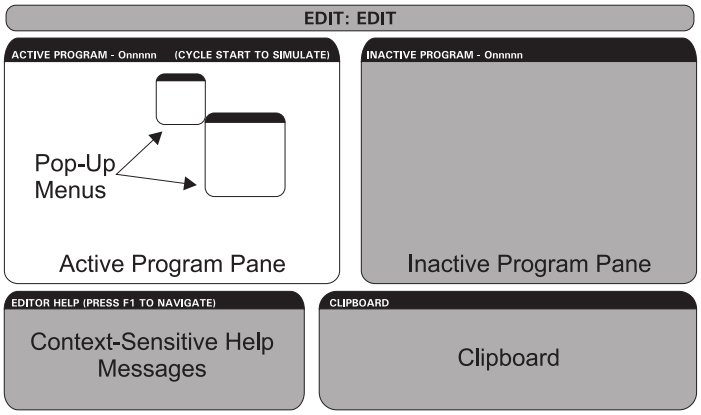
Once the tool probe is properly aligned, the values from X Dia Measure and the value from the probe will be the same.

Edit Mode

Edit gives the user the ability to edit programs using popup menus.

Press the EDIT key to enter edit mode. Two editing panes are available; an active program pane and an inactive program pane. Switch between the two by pressing the EDIT key.

To edit a program, enter the program name (Onnnnn) from the active program pane and press SELECT PROG; the program will open in the active window. Pressing the F4 button will open another copy of that program in the inactive program pane if there is not a program there already. You can also select a different program in the inactive progam pane by pressing SELECT PROG from the inactive program pane and selecting the program from the list. Press F4 to exchange the programs between the two panes (make the active program inactive and vice versa). Use the jog handle or the down/up arrows buttons to scroll through the program code.



Basic Edit Mode Layout

Press F1 to access the Popup Menu. Use the Left and Right cursor arrow keys to select from the topic menu (HELP, MODIFY, SEARCH, EDIT, PROGRAM), and use the Up and Down arrow keys or handle jog to select a function. Press Enter to execute from the menu. A context-sensitive help pane in the lower left provides information on the currently selected function. Use Page Up/Down to scroll through the help message. This message also lists hotkeys that can be used for some functions.

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. *Hot Key - Select Prog*

Select Program From List
Choose this menu item to edit a program that exists in the memory.

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. *Hot key - Select Prog*

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 memory. *Hot Key - Erase Prog*

Swap Editor Programs

Puts the active program in the inactive program pane and the inactive program in the active program pane. *Hot Key-F4*

Switch To Left Or Right Side

This will switch between the active and inactive program for editing. Inactive and active programs remain in their respective panes. *Hot Key - Edit*

The Edit Menu

Undo

The last edit operation will be undone, up to the last 9 editing operations. *Hot Key - Undo*

Select Text

This menu item will select lines of program code to set the start point of the text selection. Then use the cursor keys, home, end, page up/down, or the jog handle to scroll to the last line of code to be selected and Press F2 or Write/Enter. The selected text will be highlighted. To deselect the block, press Undo. *Hot Key - F2 to begin selection, F2 or Write to end selection*

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 (>). *Hot Key - Select Text, Position Cursor and Press Write*

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 renumbers the N code (line numbers) in the program code.

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

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

Other Keys



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



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



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



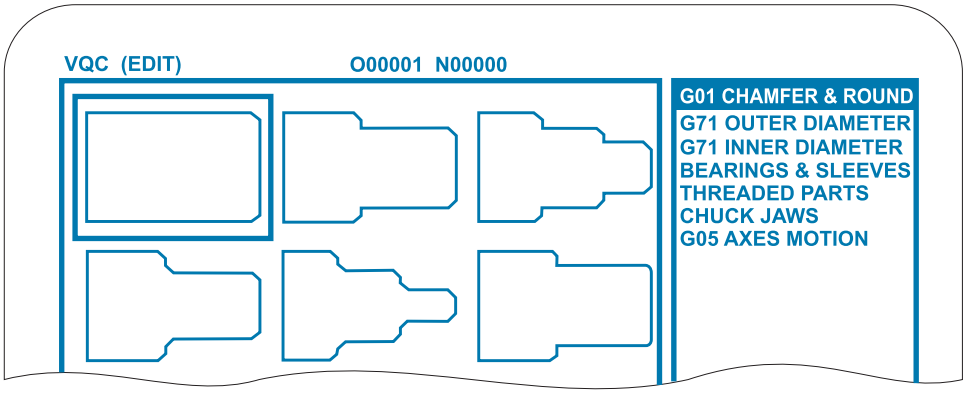
If a block has been selected, pressing undo will simply exit a block definition.

Visual Quick Code

To start Visual Quick Code (VQC), press MDI/DNC, then PROGRM CONVRS. Select VQC from the tabbed menu.

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, to 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 that anything in MDI will be overwritten.

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

NOTE: The program will also be available for editing in Edit mode. It is a good idea to check the program by running it in Graphics mode.

Macros (Optional)

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

Introduction

Macros add capabilities and flexibly 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.

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 for which Macros may be used.

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:

- Family of parts
- Soft jaw machining
- 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.

- Profiling of a part to determine unknown dimensions for later machining.
- Tool calibration for offset and wear values.
- 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 block. 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, USB or the optional floppy DNC, much like settings and offsets. Refer to the Control Data Transfer 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 a program is easier when inserting a G103 at the beginning of a program to limit block buffering, then removing it 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:	A	B	C	D	E	F	G	H	I	J	K	L	M
Variable:	1	2	3	7	8	9	-	11	4	5	6	-	13
Address:	N	O	P	Q	R	S	T	U	V	W	X	Y	Z
Variable:	-	-	-	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 the Example 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: 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.

Variable:	1	2	3	4	5	6	7	8	9	10	11
Address:	A	B	C	I	J	K	D	E	F		H
Alternate:							I	J	K	I	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							
Alternate:	J	K	I	J	K	I	J	K	I	J	K

Note that variables 10, 12, 14-16 and 27-33 do not have corresponding address arguments. They can be set if a sufficient number of I, J and K arguments are used as indicated above in the section about arguments. Once in the macro subroutine, the local variables can be read and modified by referencing the variable numbers 1-33.

When the **L** argument is used to do multiple repetitions of a macro subroutine, the arguments are set only on the first repetition. This means that if local variables 1-33 are modified in the first repetition, 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 for X, Y, Z, A, and B-axes on MOCON1
#1080-#1087	Raw analog to digital inputs (read only)
#1090-#1098	Filtered analog to digital inputs (read only)
#1094	Coolant Level
#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 for U, V, W, SS, and TT on MOCON2
#2001-#2050	X axis tool shift offsets
#2101-#2150	Z axis tool shift offsets
#2201-#2250	Tool nose radius offsets
#2301-#2350	Tool tip direction
#2401-#2450	Tool diameter/radius offsets
#2601-#2650	Tool diameter/radius wear
#2701-#2750	X axis tool wear offsets
#2801-#2850	Z axis tool wear offsets
#2901-#2950	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
#3022	Feed timer
#3023	Present cycle time
#3024	Last cycle time
#3025	Previous cycle time
#3026	Tool in spindle (read only)
#3027	Spindle RPM (read only)
#3030	Single Block
#3031	Dry Run
#3032	Block Delete
#3033	Opt Stop
#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-#5069	Present skip signal position - X, Z, Y, A, B, C, U, V, W
#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)	G114 (G154 P3) additional work offsets
#7061-#7066 (#14061-#14066)	G115 (G154 P4)additional work offsets
#7081-#7086 (#14081-#14086)	G116 (G154 P5) additional work offsets
#7101-#7106 (#14101-#14106)	G117 (G154 P6) additional work offsets
#7121-#7126 (#14121-#14126)	G118 (G154 P7) additional work offsets
#7141-#7146 (#14141-#14146)	G119 (G154 P8) additional work offsets
#7161-#7166 (#14161-#14166)	G120 (G154 P9) additional work offsets
#7181-#7186 (#14181-#14186)	G121 (G154 P10) additional work offsets
#7201-#7206 (#14201-#14206)	G122 (G154 P11) additional work offsets
#7221-#7226 (#14221-#14221)	G123 (G154 P12) additional work offsets
#7241-#7246 (#14241-#14246)	G124 (G154 P13) additional work offsets
#7261-#7266 (#14261-#14266)	G125 (G154 P14) additional work offsets
#7281-#7286 (#14281-#14286)	G126 (G154 P15) additional work offsets
#7301-#7306 (#14301-#14306)	G127 (G154 P16) additional work offsets
#7321-#7326 (#14321-#14326)	G128 (G154 P17) additional work offsets
#7341-#7346 (#14341-#14346)	G129 (G154 P18) additional work offsets
#7361-#7366 (#14361-#14366)	G154 P19 additional work offsets
#7381-#7386 (#14381-#14386)	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
#14561-#14566	G154 P29 additional work offsets
#14581-#14586	G154 P30 additional work offsets
⋮	
#14781-#14786	G154 P40 additional work offsets
⋮	
#14981-#14986	G154 P50 additional work offsets
⋮	
#15181-#15186	G154 P60 additional work offsets
⋮	
#15381-#15386	G154 P70 additional work offsets
⋮	
#15581-#15586	G154 P80 additional work offsets
⋮	
#15781-#15786	G154 P90 additional work offsets
⋮	
15881-15886	G154 P95 additional work offsets
15901-15906	G154 P96 additional work offsets
15921-15926	G154 P97 additional work offsets
15941-15946	G154 P98 additional work offsets
15961-15966	G154 P99 additional work offsets

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 informs the programmer if data is 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; therefore, 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, 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 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, 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 are 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 can be set independent of each other.

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, 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 act 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

#5021 X-axis	#5024 A-axis	#5027 U-axis
#5022 Y-axis	#5025 B-axis	#5028 V-axis (used for the Haas Bar Feeder)
#5023 Z-axis	#5026 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-#5069 Current Skip Signal Position

The position where the last skip signal was triggered can be obtained through #5061-#5069, X, Y, Z, A, B, C, U, V, and W 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.

#6996-#6999 Parameter access using macro variables

It is possible for a program to access parameters 1 to 1000 and any of the parameter bits, as follows:

#6996: Parameter Number
#6997: Bit Number (optional)
#6998: Contains value of parameter number in variable 6996
#6999: Contains bit value (0 or 1) of parameter bit specified in variable 6997.

Note: Variables 6998 and 6999 are read-only.

Usage

To access the value of a parameter, the number of that parameter is copied into variable 6996, after which, the value of that parameter is available using macro variable 6998, as shown:

```
#6996=601 (Specify parameter 601)
#100=#6998 (Copy the value of parameter 601 to variable #100)
```

To access a specific parameter bit, the number of that parameter is copied into variable 6996 and the bit number is copied to macro variable 6997. The value of that parameter bit is available using macro variable 6999, as shown:

```
#6996=57 (Specify parameter 57)
#6997=0 (Specify bit zero)
#100=#6999 (Copy parameter 57 bit 0 to variable #100)
```

Note: Parameter bits are numbered 0 through 31. 32-bit parameters are formatted, on-screen, with bit 0 at the top-left, and bit 31 at the bottom-right.

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; such as: #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, 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. 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, 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: Arithmetic operators, Logical operators, and Boolean 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 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

#1=1.0; 0000 0001 #2=2.0; 0000 0010 #3=#1 OR #20000 0011	Here the variable #3 will contain 3.0 after the OR operation.
#1=5.0; #2=3.0; IF [[#1 GT 3.0] AND [#2 LT 10]] GOTO1	Here control will transfer to block 1 because #1 GT 3.0 evaluates to 1.0 and #2 LT 10 evaluates to 1.0, thus 1.0 AND 1.0 is 1.0 (true) and the GOTO occurs.

Note that care must be taken when using logical operators so that the desired result is achieved.

Boolean operators 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	Explanation
IF [#1 EQ 0.0] GOTO100;	Jump to block 100 if value in variable #1 equals 0.0.
WHILE [#101 LT 10] DO1;	While variable #101 is less than 10 repeat loop DO1..END1.
#1=[1.0 LT 5.0];	Variable #1 is set to 1.0 (TRUE).
IF [#1 AND #2 EQ #3] GOTO1	If variable #1 logically ANDed with variable #2 is equal to the value in #3 then control jumps to block 1.

Expressions

Expressions are defined as any sequence of variables and operators surrounded by the square brackets “[“ and “]”. There are two uses for expressions: conditional expressions or arithmetic expressions. Conditional expressions return False (0.0) or True (any non zero) values. Arithmetic expressions use arithmetic operators along with functions to determine a value.

Conditional Expressions

In the Haas control, All expressions set a conditional value. The value is either 0.0 (False) or the value is non-zero (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. 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, and 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 variab le, whether directly or indirectly. The following macro initializes a sequence of variables to any value. Here bot h 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 preceding 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 may 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 ads serial numbers to parts:

```
O9200                                          (Engrave digit at current location.)
;
(D=Decimal digit to engrave);
IF [[#7 NE #0] AND [#7 GE O] 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.,...)
```

The previous subroutine will 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>] GOTOn

As discussed, <conditional expression> is any expression that uses any of the six Boolean operators EQ, NE, GT, LT, GE, or LE. The brackets surrounding the expression are mandatory. In the Haas control, it is not necessary to include these operators. For example: IF [#1 NE 0.0] GOTO5; could also be: IF [#1] GOTO5;.

In this statement, if the variable #1 contains anything but 0.0, or the undefined value #0, 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, 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, the segment containing the three lowest levels of nesting can be made into a subroutine, thereby 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 9 times, which means, program 1 can call program 2, program 2 and 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 (Parameter 91). 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.

G-Code Aliasing		M-Code Aliasing	
Haas Parameter	**O Code**	**Haas Parameter**	**M Macro Call**
91	9010	81	9000
92	9011	82	9001
93	9012	83	9002
94	9013	84	9003
95	9014	85	9004
96	9015	86	9005
97	9016	87	9006
98	9017	88	9007
99	9018	89	9008
100	9019	90	9009

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.

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

POPEN 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 that eight. Thus the following formats are illegal:

```
[00]  [54]  [45]  [36]  /* not legal formats */
```

A decimal point is printed out between the whole part and the fractional part. The fractional part is rounded to the least significant place. When zero places are reserved for the fractional part, then no decimal point is printed out. Trailing zeros are printed if there is a fractional part. At least one place is reserved for the whole part, even when a zero is used. If the value of the whole part has fewer digits than have been reserved, then leading spaces are output. If the value of the whole part has more digits than has been reserved, then the field is expanded so that these numbers are printed.

A carriage return is sent out after every DPRNT block.

DPRNT[ ] Examples

	Code	Output
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. This 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.

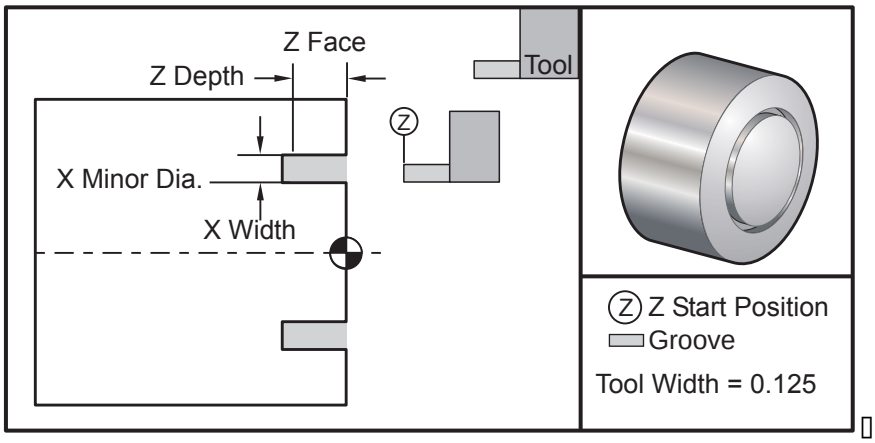
GOTO-nnnn

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 are used to command specific actions for the machine: such as simple machine moves or drilling functions. They also command more complex features which can involve optional live tooling and the 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.)

Default G-codes for each group are shown on the Current Commands screen. If another G code from the group is commanded (active), that G-code is 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 effect 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 line's 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 tijme 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

G00 Rapid Motion Positioning (Group 01)....116
G01 Linear Interpolation Motion (Group 01)....116
G02 CW Circular Interpolation Motion / G03 CCW Circular Interpolation Motion (Group 01)....120
G04 Dwell (Group 00)....122
G05 Fine Spindle Control Motion (Group 00)....122
G09 Exact Stop (Group 00)....123
G10 Set Offsets (Group 00)....123
G14 Sub-Spindle Swap / G15 Cancel (Group 17)....123
G17 XY Plane....123
G18 ZX Plane Selection (Group 02)....124
G19 YZ Plane (Group 02)....124
G20 Select Inches / G21 Select Metric (Group 06)....124
G28 Return To Machine Zero, set optional G29 Reference point (Group 00)....124
G29 Return from Reference Point (Group 00)....124
G31 Skip Function (This G-code is optional and requires a probe) (Group 00)....124
G32 Thread Cutting (Group 01)....124
G40 Tool Nose Compensation Cancel (Group 07)....126
G41 Tool Nose Compensation (TNC) Left / G42 TNC Right (Group 07)....126
G50 Set Global coordinate Offset FANUC, YASNAC (Group 00)....127
G50 Spindle Speed Clamp....128
G51 Cancel Offset (YASNAC) (Group 00)....128
G52 Set Local Coordinate System FANUC (Group 00)....128
G53 Machine Coordinate Selection (Group 00)....128
G54-59 Select Coordinate System #1 - #6 FANUC (Group 12)....128
G61 Exact Stop Modal (Group 15)....128
G64 Exact Stop Cancel G61 (Group 15)....128
G70 Finishing Cycle (Group 00)....128
G71 O.D./I.D. Stock Removal Cycle (Group 00)....129
G72 End Face Stock Removal Cycle (Group 00)....137
G73 Irregular Path Stock Removal Cycle (Group 00)....140
G74 End Face Grooving Cycle, Peck Drilling (Group 00)....141
G75 O.D./I.D. Grooving Cycle (Group 00)....143
G76 Threading Cycle, Multiple Pass (Group 00)....145
G77 Flatting Cycle (This G-code is optional and is used for live tooling) (Group 00)....148
G80 Canned Cycle Cancel (Group 09*)....150
G81 Drill Canned Cycle (Group 09)....150
G82 Spot Drill Canned Cycle (Group 09)....150
G83 Normal Peck Drilling Canned Cycle (Group 09)....151
G84 Tapping Canned Cycle (Group 09)....152
G85 Boring Canned Cycle (Group 09)....152
G86 Bore and Stop Canned Cycle (Group 09)....153
G87 Bore and Manual Retract Canned Cycle (Group 09)....153
G88 Bore and Dwell and Manual Retract Canned Cycle (Group 09)....154
G89 Bore and Dwell Canned Cycle (Group 09)....154
G90 O.D./I.D. Turning Cycle (Group 01)....155
G92 Threading Cycle (Group 01)....155
G94 End Facing Cycle (Group 01)....157
G95 Live Tooling Rigid Tap (Face) (Group 09)....158

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

G65, the Macro Subroutine Call, is described in the Macro chapter.

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 machines 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.

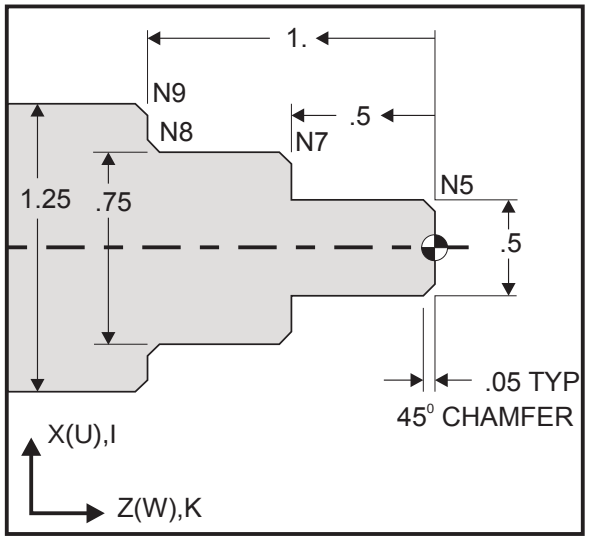
G01 Linear Interpolation Motion (Group 01)

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 (except when U,V and W are used with the Advanced Parts Loader ,APL).

Corner Rounding and Chamfering

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
%
```

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.

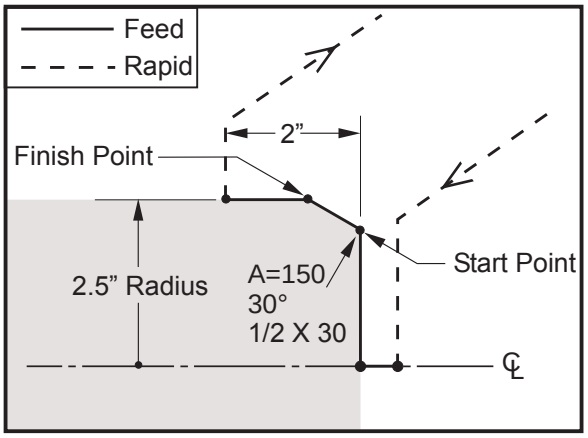
Chamfering Syntax	Corner Rounding Syntax
G01 X(U) x Kk	G01 X(U) x Rr
G01 Z(W) z Ii	G01 Z(W) z Rr

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

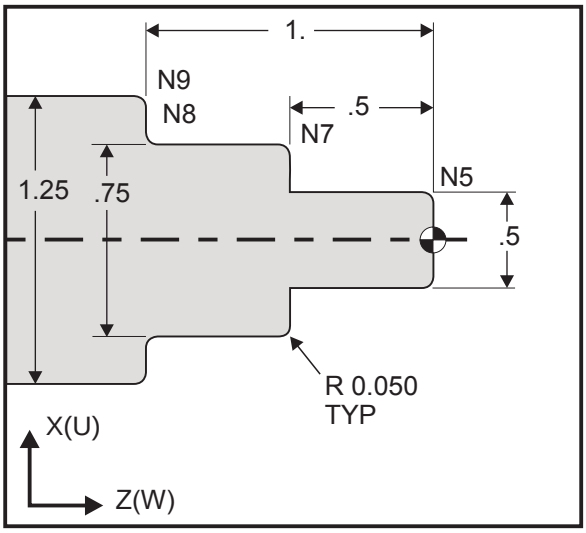
G01 Chamfering with A

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;
```

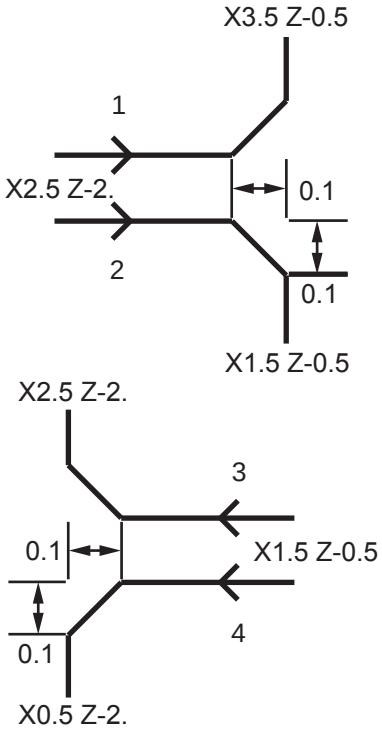
Corner Chamfering



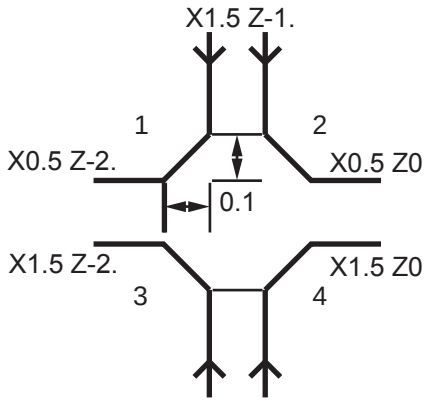

```
%
O0005 (Automatic Corner Rounding)
T101;
N1 G50 S1500;
N2 G00 G97 S500 M03;
N3 X0 Z0.25;
N4 G01 Z0 F0.005;
N5 G01 X0.5 R-0.050;
N6 G01 Z-0.50;
N7 G01 X0.75 R-0.050;
N8 G01 Z-1.0 R0.050;
N9 G01 X1.25 R-0.050;
N10 G01 Z-1.5;
N11 G00 X1.5 Z0.25;
G28;
M30;
%
```

NOTES: 1) Incremental programming is possible if Ub or Wb is specified in place of Xb or Zb, respectively. So its actions will be as follows: X($POS_{current}$ + i) = Ui, Z($POS_{current}$+k) = Wk, X($POS_{current}$+r)=Ur, Z(POScurrent+r)=Wr.
2) **$POS_{current}$** indicates current position of X or Z axis. 3) I, K and R always specify a radius value (radius programming value)

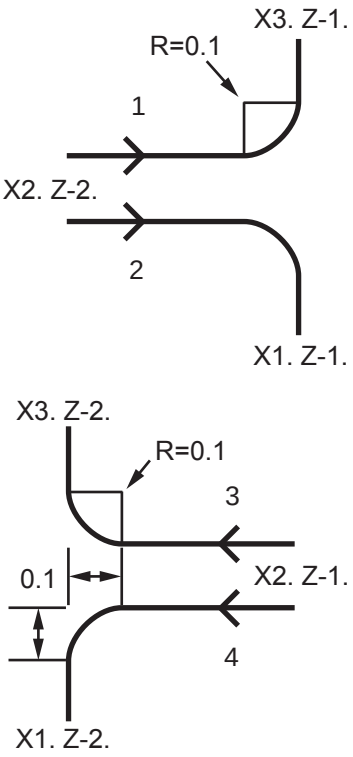
Chamfering	Code/Example	Movement
1. Z+ to X+	X2.5 Z-2; G01 Z-0.5 I0.1; X3.5;	X2.5 Z-2; G01 Z-0.6; X2.7 Z-0.5; X3.5;
2. Z+ to X-	X2.5 Z-2.; G01 Z-0.5 I-0.1; X1.5;	X2.5 Z-2.; G01 Z-0.6; X2.3 Z-0.5; X1.5;
3. Z- to X+	X1.5 Z-0.5.; G01 Z-2. I0.1; X2.5;	X1.5 Z-0.5 G01 Z-1.9; X1.7 Z-2.; X2.5;
4. Z- to X-	X1.5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



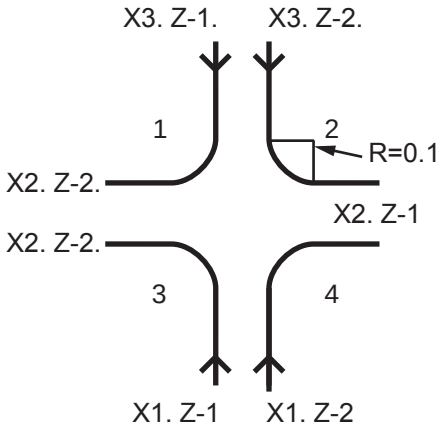
Chamfering	Code Example	Movement
1. X- to Z-	X1.5 Z-1.; G01 X0.5 K-0.1; Z-2.;	X1.5 Z-1.; G01 X0.7; X0.5 Z-1.1; Z-2.
2. X- to Z+	X1.5 Z-1.; G01 X0.5 K0.1; Z0.;	X1.5 Z-1.; G01 X0.7; X0.5 Z-0.9; Z0.;
3. X+ to Z-	X0.5 Z-1.; G01 X1.5 K-0.1; Z-2.;	X0.5 Z-1.; G01 X1.3; X1.5 Z-1.1; Z-2.
4. X+ to Z+	X0.5 Z-1.;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

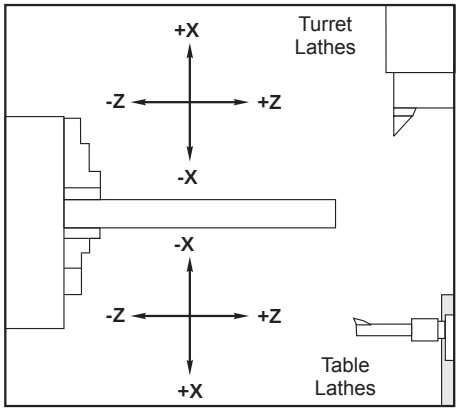
1) Use K address only with X(U) address. Use I address only with Z(W) address.
2) Use R address with either X(U) or Z(W), but not both in the same block.
3) Do not use I and K together on the same block. When using R address, do not use I or K.
4) The next block must be another single linear move that 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)

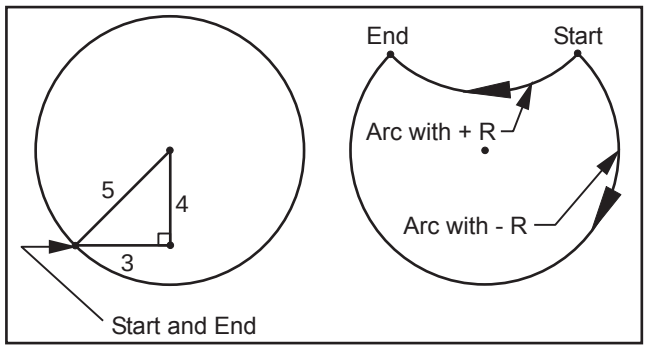
F	Feed rate
*I	Distance along X-axis to center of circle
*K	Distance along Z-axis to center of circle
*R	Radius of arc
*U	X-axis incremental motion command
*W	Z-axis incremental motion command
*X	X-axis absolute motion command
*Z	Z-axis absolute motion command
,C	Distance from center of intersection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

* indicates optional

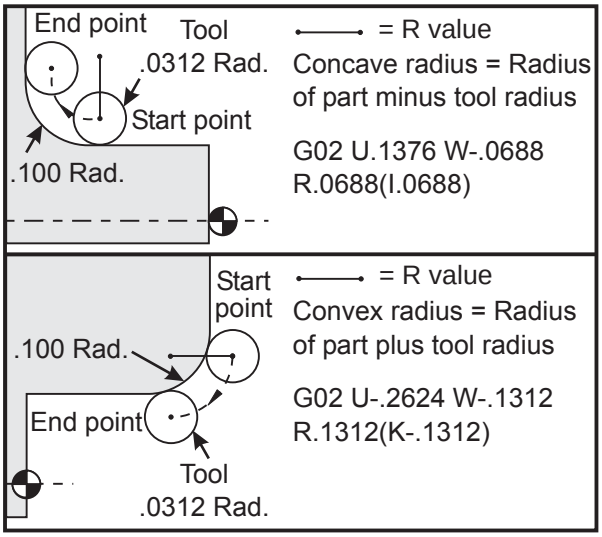
These G codes are used to specify a circular motion (CW or CCW) of the linear axes (Circular motion is possible in the X and Z axes as selected by G18). The X and Z values are used to specify the end point of the motion and can use either absolute (U and W) or incremental motion (X and Z). If either 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).



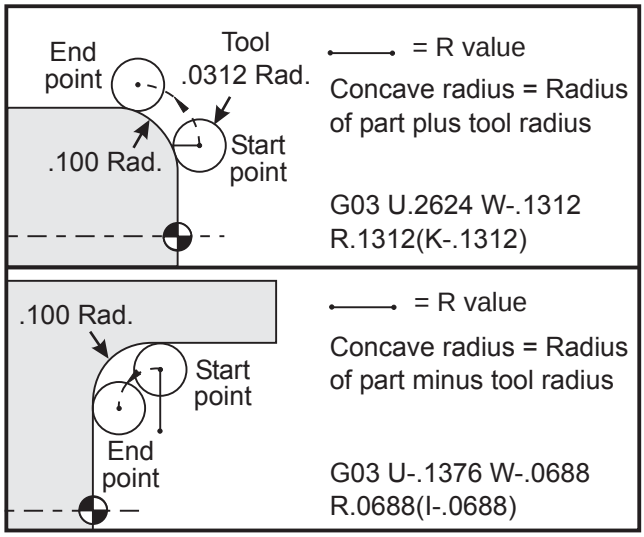
Axis Definitions



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 milliseconds.

G05 Fine Spindle Control Motion (Group 00) (Also see the C-axis section)

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.

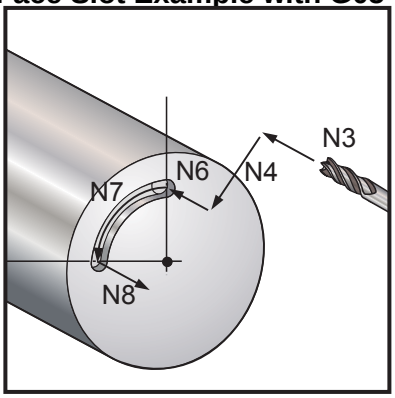
* indicates optional

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.

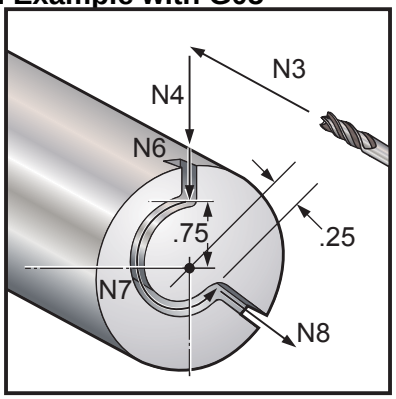
The largest feed per revolution value that can be specified is aproximately 14.77. This means that G5 motions with small R motions relative to X or Z motions will not work. For example, an R motion of 1.5 degrees, the largest X or Z motion that can be specified is 14.77 * 1.5 / 360 = .0615 inches. Conversely,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

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

This code indicates to the controller that programmed circular motion G02 and G03 will be performed in the XY plane. The G17 plane is parallel to X and Y-axes.

G17 code supports G112 Cartesian to Polar transformation. Plane selection codes are modal and remain in effect until another plane is selected.

Programming tool nose radius compensation G41 or G42 will work while using G112 and in the G17 plane.

G18 ZX Plane Selection (Group 02)

This code indicates to the controller that programmed circular motion G02 and G03 will be performed in the ZX plane. The G18 plane is parallel to Z and X-axes.

G18 is the power on default plane for the HAAS lathe. Plane selection codes are modal and remain in effect until another plane is selected.

G19 YZ Plane (Group 02)

This code indicates to the controller that programmed circular motion G02 and G03 will be performed in the YZ plane. The G19 plane is parallel to Y and Z-axes. Plane selection codes are modal and remain in effect until another plane is selected.

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
A	A-axis absolute motion command
B	B-axis absolute motion command
C	C-axis absolute motion command

* indicates optional

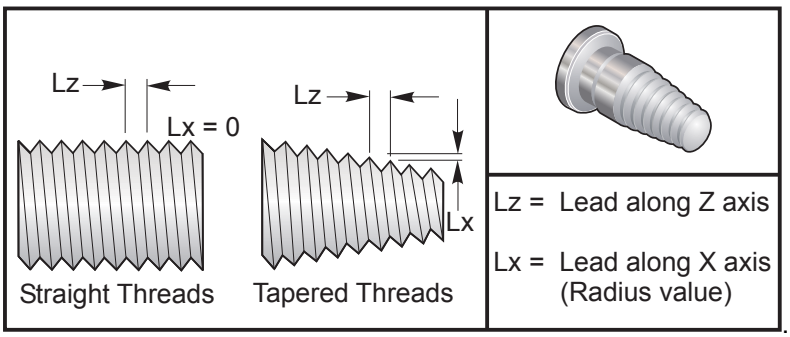
This G-code moves the axes to the programmed position. It applies only to the block in which G31 is specified. The specified move is started and continues until the position is reached or the probe receives a signal (skip signal). The control will beep when the end of travel is reached.

Do not use Cutter Compensation with a G31. Also see M78 and M79.

G32 Thread Cutting (Group 01)

F	Feed rate
Q	Thread Start Angle (optional). See example on the following page.
U/W	X/Z-axis incremental positioning command. (Incremental thread depth values have to be user specified)
X/Z	X/Z-axis absolute positioning command. (Thread depth values have to be user specified)

NOTE: 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 activ



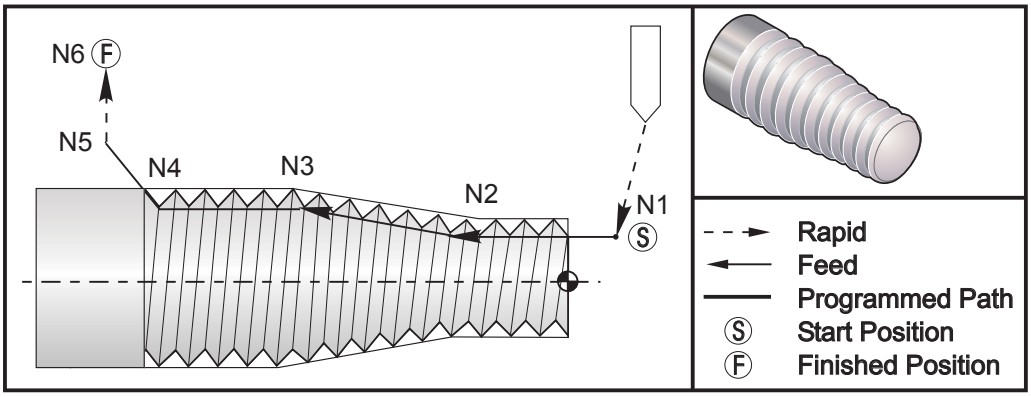
G32 Definition of Lead (Feed Rate) for Straight and Tapered Threads

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. 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 affect 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 G9



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 **Comments**

```
...
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. Do not use a decimal point. 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.

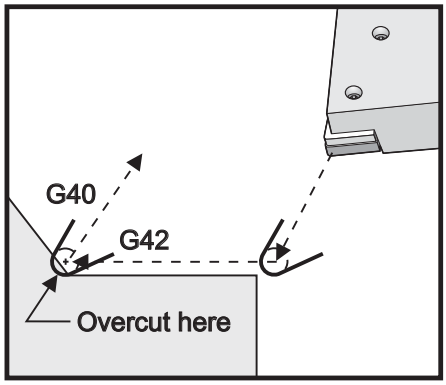
G40 Tool Nose Compensation Cancel (Group 07)

*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

* 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.

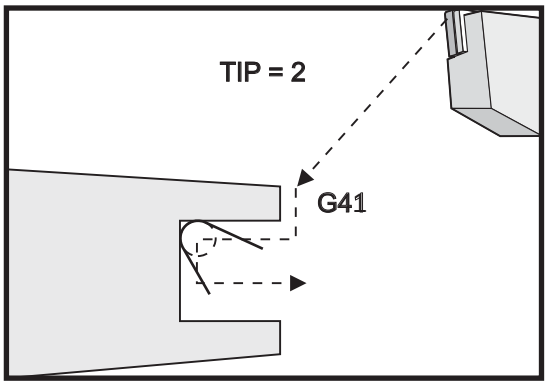
The tool departure usually does not correspond with a point on the part. In many cases overcutting or undercutting can occur.

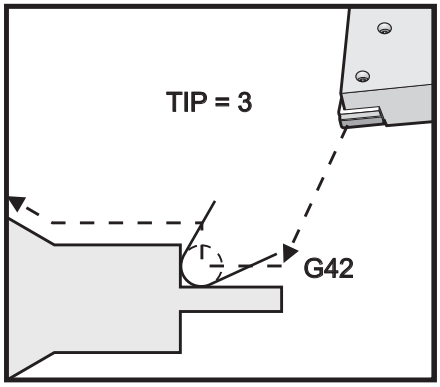


G40

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);

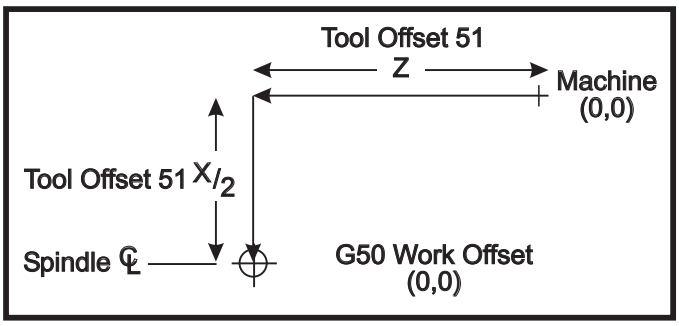
To set a YASNAC style work coordinate shift, specify G50 with a T value (Setting 33 must be set to YASNAC). The global coordinate is set to the X and Z values in the Tool Shift Offset page. Values for the T address code are, Txxyy where xx is between 51 and 100 and yy is between 00 and 50. For example, 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
```




```
000101
N1 G51 (Return to Machine Zero)
N2 G50 T5100;  (Offset for Tool 1)
.
.
.
%
```

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.

Work Coordinate Systems

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.

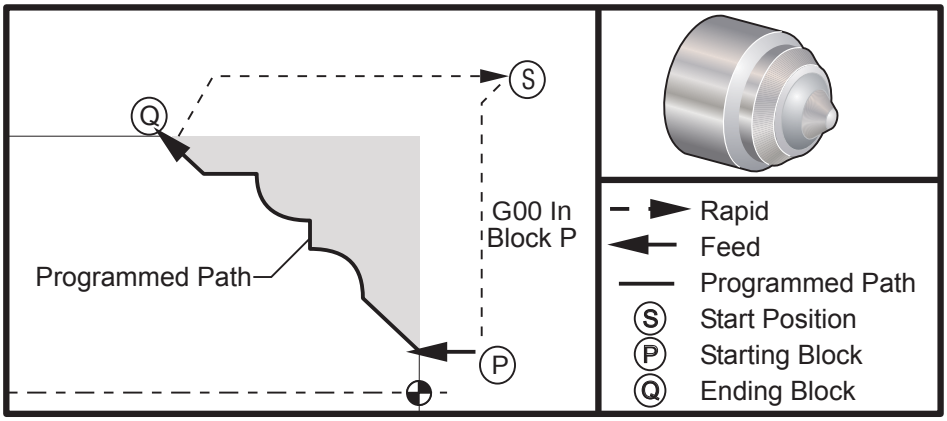
G64 Exact Stop Cancel G61 (Group 15)

The G64 code is used to cancel exact stop. Selects normal cutting mode.

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)
...
```

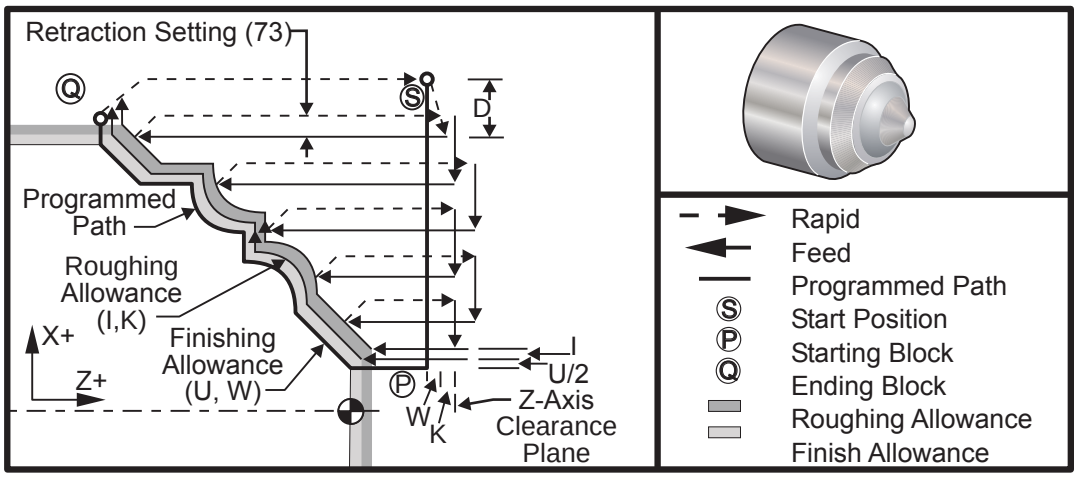
The G70 cycle is similar to a local subprogram call. However, the G70 requires that a beginning block number (P code) and an ending block number (Q code) be specified.

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



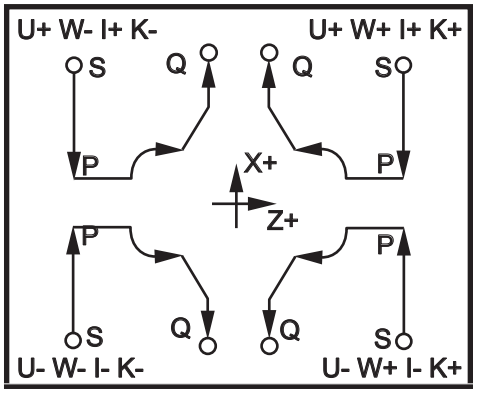
G71

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 allowance.



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).

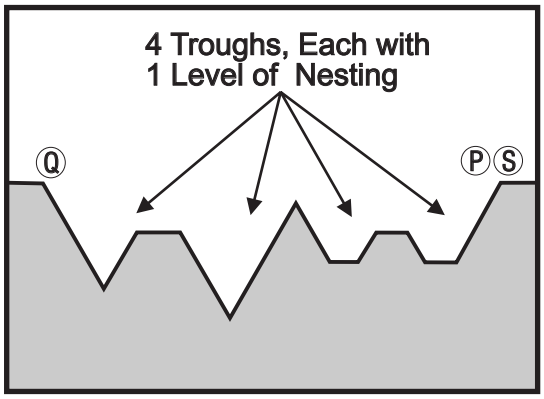
The X axis PQ path must not exceed the original starting location. The only exception is the ending Q block.

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

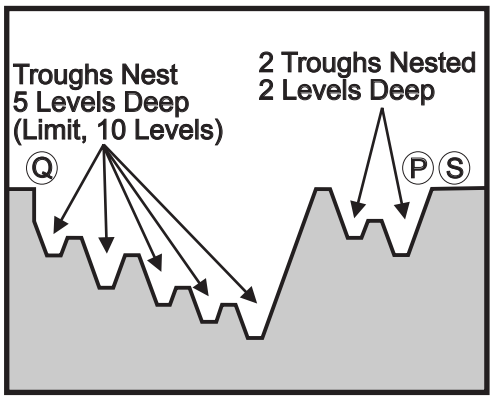
Type II, when Setting 33 is set to FANUC, must have a reference move, in both the X and Z axis, in the block specified by P.

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

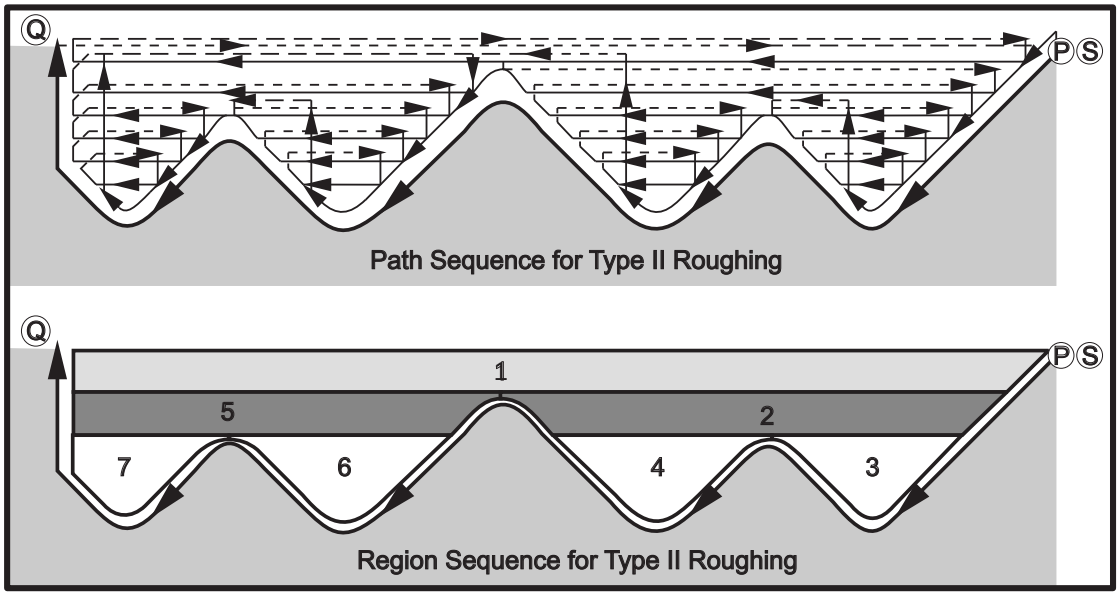


Troughs on the Same Level

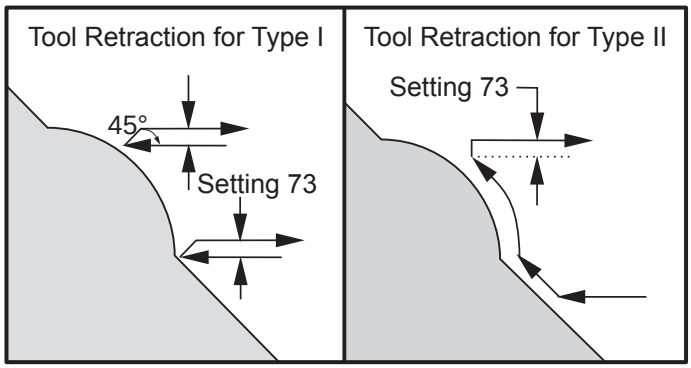


Nested Troughs

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.

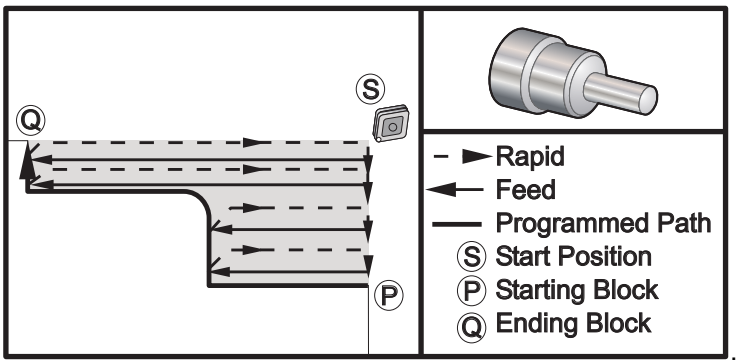
For example, if G71 Type 2 path contains the following:

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

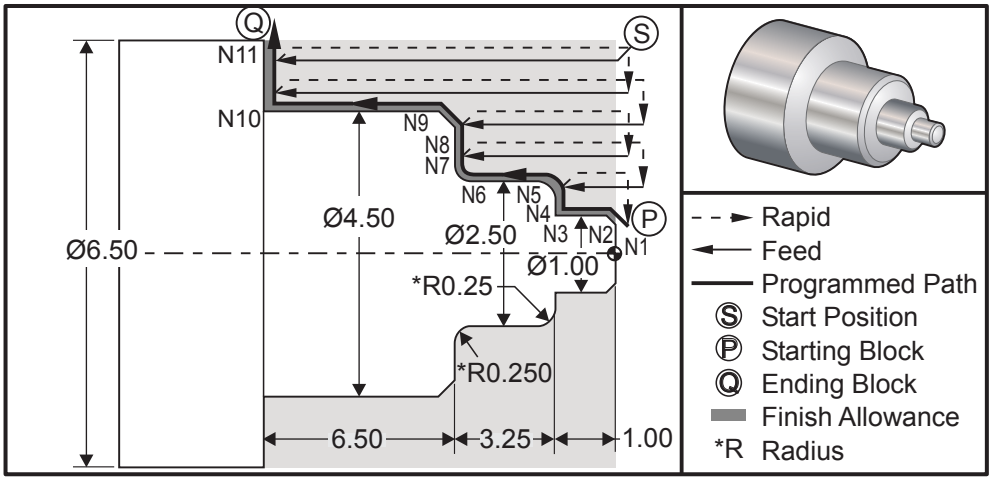
NOTE: If the last cut in the P-Q path is a non-monotonic curve (using a finish allowance) add a short retraction cut; do not use W.



G71 Basic G Code Example

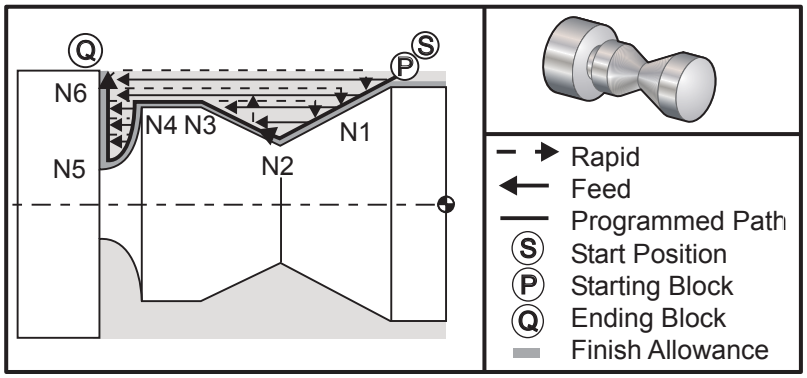
Program Example **Description**

```
%
O0070                                          (G71 Roughing Cycle)
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.
G01 Z-3. F0.006
X3.5
G03 X4. Z-3.25 R0.25
G01 Z-6.
N2 X6.
G70 P1 Q2                                      (FINISH PASS)
M09
G28 M05
M30
%
```



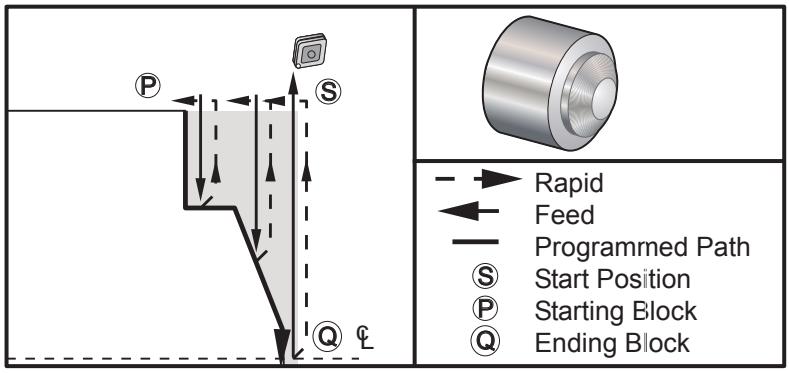
G71 Type 1 O.D./I.D. Stock Removal Example

Program Example	**Description**
%	
O0071	**(FANUC G71 TYPE I EXAMPLE)**
T101 (CNMG 432)	
(Tool change & apply Offsets)	
G00 G54 X6.6 Z.05 M08	*(Rapid to Home Position)*
G50 S2000	*(Set Max RPM 2000)*
G97 S636 M03	*(Spindle On)*
G96 S750	*(Constant surface speed On)*
G71 P1 Q11 D0.15 U0.01 W0.005 F0.012	*(Define rough cycle)*
N1 G00 X0.6634	P (Begin definition)
N2 G01 X1. Z-0.1183 F0.004	*(Finish pass .004" Feed)*
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	Q (End definition)
G00 X0 Z0 T100	*(Rapid to tool change position)*
T202	*(Finish tool)*
G50 S2500	
G97 S955 M03	
G00 X6. Z0.05 M08	
G96 S1500	
G70 P1 Q11	
G00 X0 Z0 T200	
M30	
%	



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

Program Example	**Description**
%	
O0001	(YASNAC G71 Type II Example)
T101	(Roughing tool)
G97 S1200 M03	
;	
G00 X2. Z0	**S** (Start position)
G71 P1 Q6 D.035 U.03 W0.01 F0.01	
;	
N1 G01 X1.5 Z-0.5 F0.004	**P** (PQ Path definition)
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.	**Q** (PQ Path end)
;	
T202	(Finishing tool)
G97 S1500 M03	
;	
G70 P1 Q6	(Finish pass))
;	
G28 M30	
%	

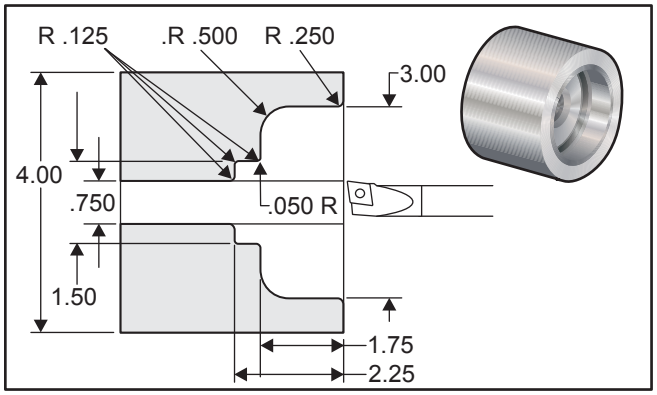


G72 Basic G Code Example

Program Example	**Description**
%	
O0069	*(G72 Roughing Cycle)*
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	*(Finish Pass)*
M05	
G28	
M30	
%	

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.

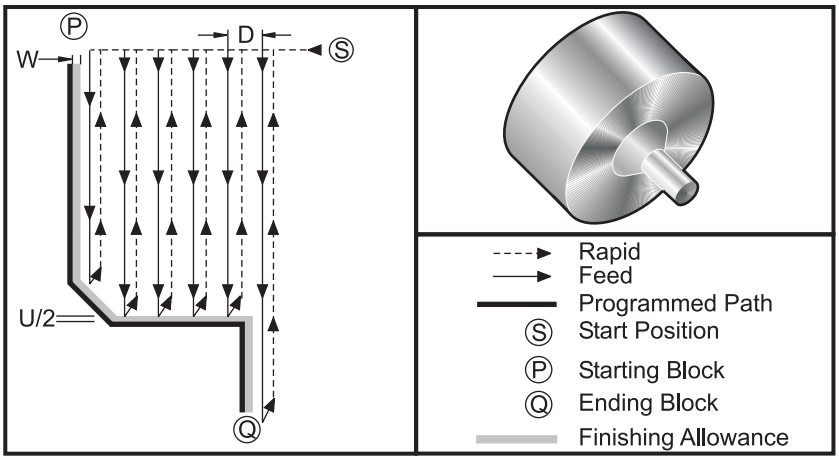


TOOL	**OFFSET**	**RADIUS**	**TIP**
4	04	.0	0

```
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)

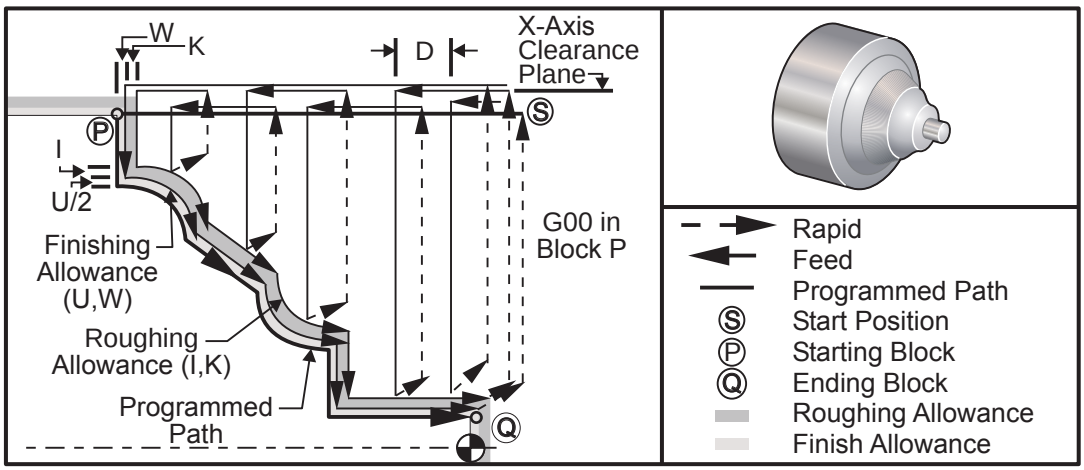
- * D Depth of cut for each pass of stock removal, positive
- * F Feed rate to use throughout G72 PQ block
- * I X-axis size and direction of G72 rough pass allowance, radius
- * K Z-axis size and direction of G72 rough pass allowance
- P Starting Block number of path to rough
- Q Ending Block number 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




```
O00100 (END FACE)
T101
G54
G50 S2500
G96 S500 M03
G00 X4.05 Z0.2
G72 P101 Q102 U0.03 W0.03 D0.2 F0.01
N101 G00 Z-1.
G01 X1.5 X1. Z-0.75
G01 Z0
N102 X0
G70 P101 Q102
G00 X4.05 Z0.2
M30
```

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).

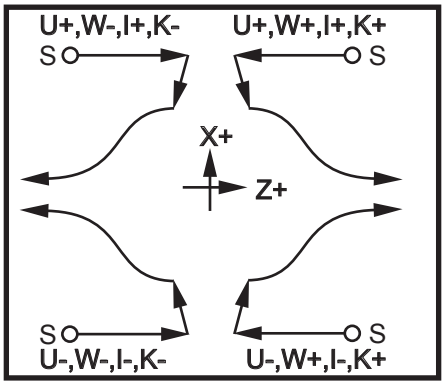


G72 End Face Stock Removal Cycle

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.

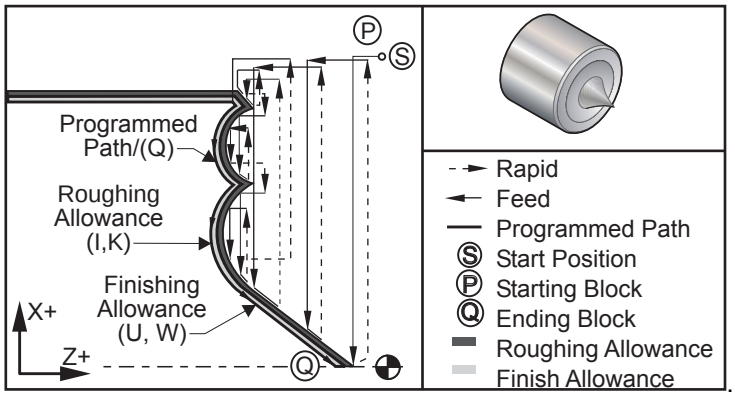
Roughing is similar to Type I except after each pass, along the X axis, the tool will follow the path defined by PQ. The tool will then retract parallel to the Z axis by the distance defined in Setting 73 (Can Cycle Retraction). The Type II roughing method does not leave steps in the part prior to finish 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-5.1 Z-5.1
X-8.1 Z-3.1
...
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

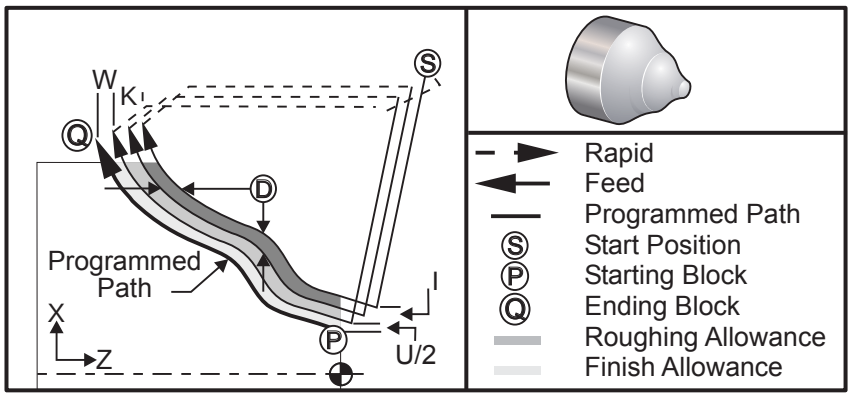
Program Example **Description**

```
%
00722                                                    (G72 Roughing Cycle)
T101
S1000 M03
G00 G54 X2.1 Z0.1
G72 P1 Q2 D0.06 I0.02 K0.01 U0.0 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-.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                                                (Finish Pass)
M05
G28
M30
%
```

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.

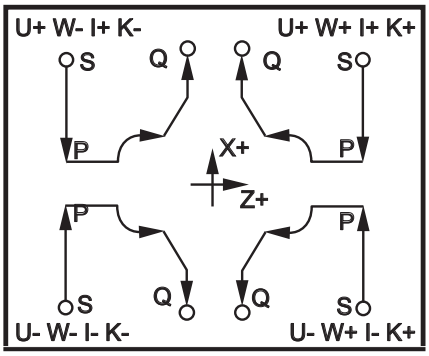
Only F, S and T prior to or in the G73 block are in effect. Any feed (F), spindle speed (S) or tool change (T) codes on the lines from P to Q are ignored.

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.

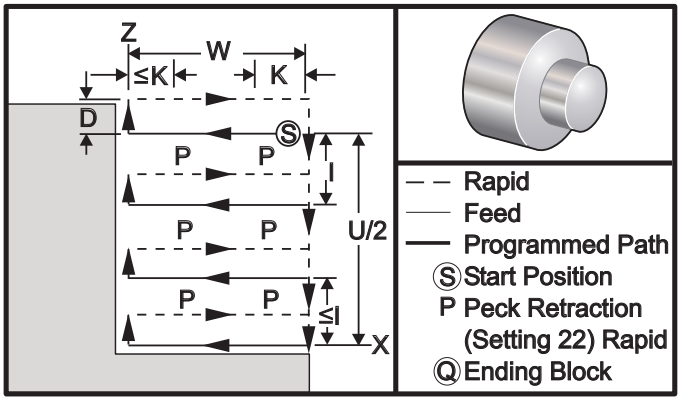
The value of D must be a positive integral number. If the D value includes a decimal, an alarm will be generated.

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

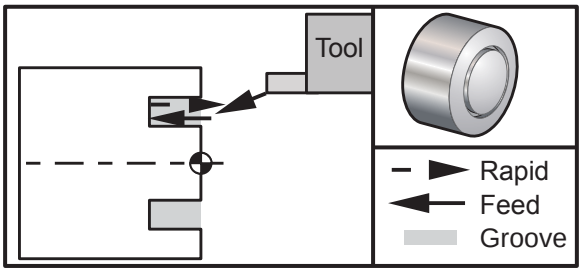


G74 End Face Grooving Cycle, Peck Drilling

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

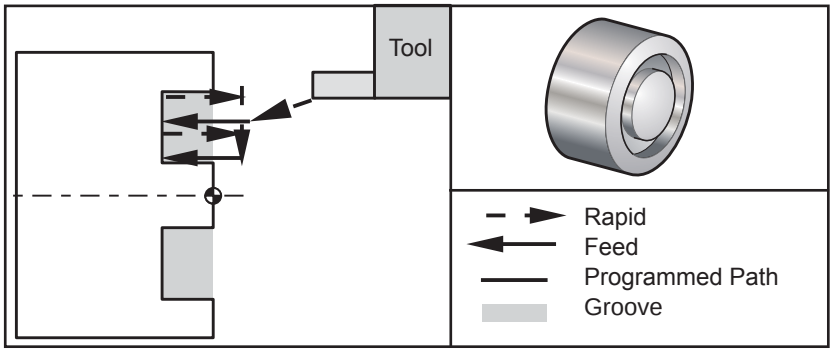
When an X, or U, code is added to a G74 block and X is not the current position, then a minimum of two pecking cycles will occur. One at the current location and another at the X location. The I code is the incremental distance between X axis pecking cycles. Adding an I will perform multiple, evenly 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	**Description**
%	
O0071	
T101	
G97 S750 M03	
G00 X3. Z0.05	(Rapid to Start position)
G74 Z-0.5 K0.1 F0.01	(Feed Z-.5 with a .100" peck)
G28	
M30	
%	



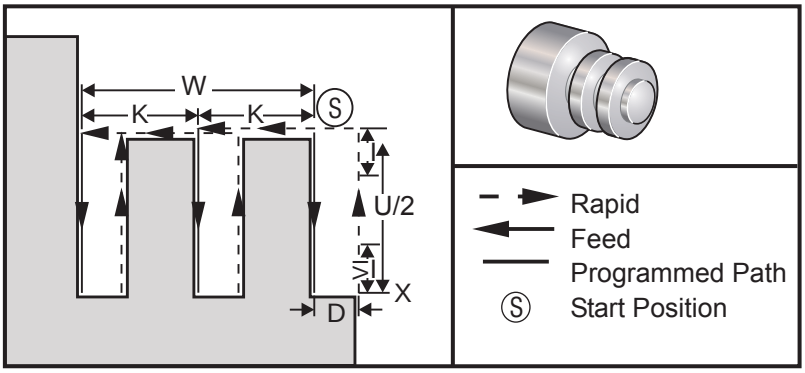
G74 End Face Grooving Cycle (Multiple Pass)

Program Example	**Description**
%	
O0074	
T101	
G97 S750 M03	
G00 X3. Z0.05	(Rapid to Start position)
G74 X1.75 Z-0.5 I0.2 K0.1 F0.01	(Face grooving cycle multiple pass)
G28	
M30	
%	

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

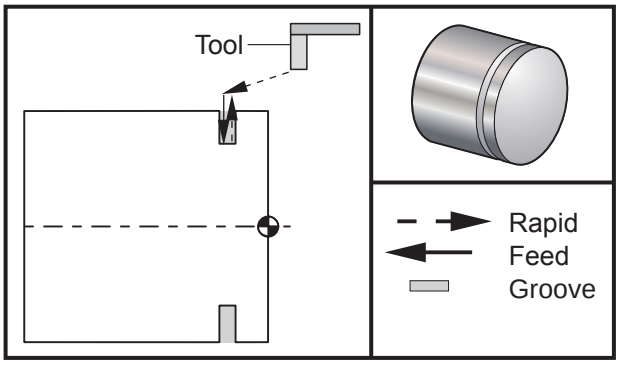
* * D Tool clearance when returning to starting plane, positive
* * F Feed rate
* * I X-axis size of increment between pecks in a cycle (radius measure)
* * K Z-axis size of increment between peck cycles
* * U X-axis incremental distance to total pecking depth
  W Z-axis incremental distance to furthest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



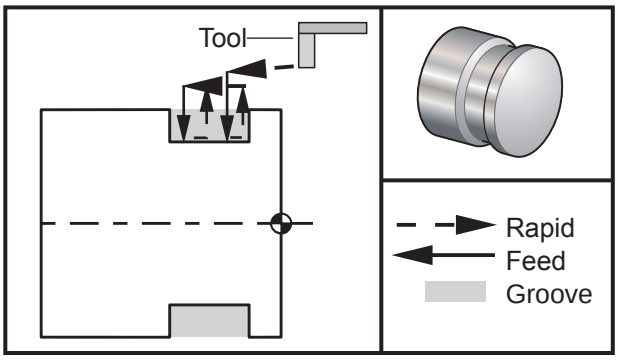
G75O.D. / I.D. Grooving Cycle

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	Description
%	
O0075	
T101	
G97 S750 M03	
G00 X4.1 Z0.05	(Rapid to Clear position)
G01 Z-0.75 F0.05	(Feed to Groove location)
G75 X3.25 I0.1 F0.01	(O.D./I.D. Peck grooving single pass)
G00 X5. Z0.1	
G28	
M30	
%	

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

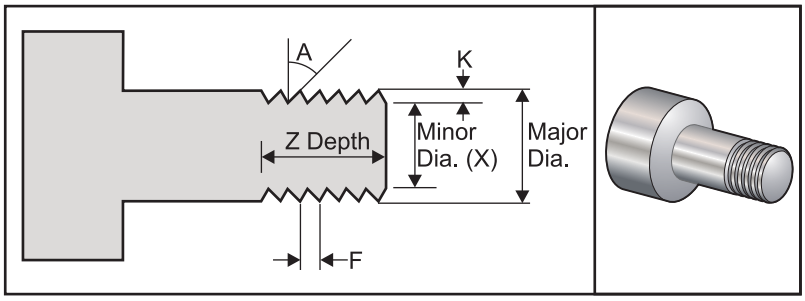


Program Example	Description
%	
O0075	
T101	
G97 S750 M03	
G00 X4.1 Z0.05	(Rapid to Clear position)
G01 Z-0.75 F0.05	(Feed to Groove location)
G75 X3.25 Z-1.75 I0.1 K0.2 F0.01	(O.D./I.D. Peck groove multiplepass)
G00 X5. Z0.1	
G28	
M30	
%	

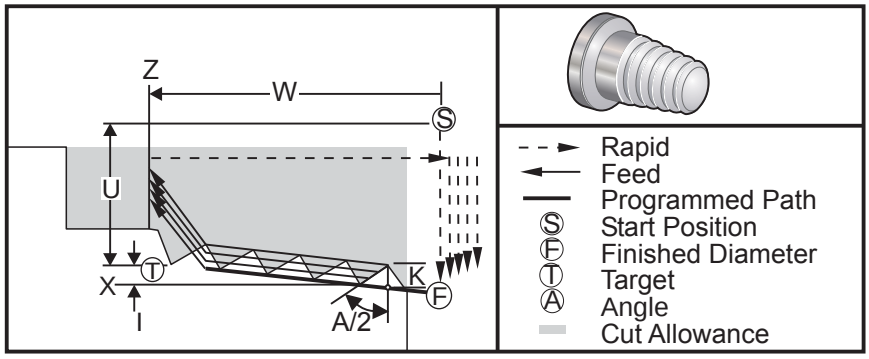
G76 Threading Cycle, Multiple Pass (Group 00)

* 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.

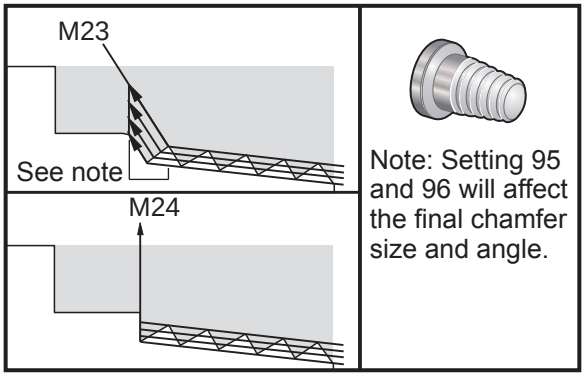
The thread taper amount is specified in I. Thread taper is measured from the target position X, Z at point **T** to position **F**. Note that a conventional O.D. taper thread will have a negative I value.

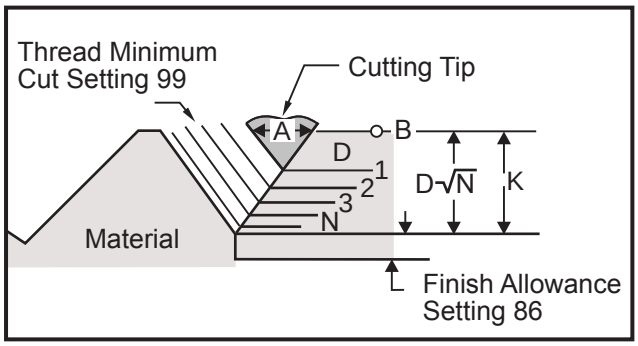
The depth of the first cut through the thread is specified in D. The depth of the last cut through the thread can be controlled with Setting 86.

The tool nose angle for the thread is specified in A. The value can range from 0 to 120 degrees. If A is not used, 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, or using M24. The default value for Setting 95 is 1.000 and the default angle for the thread (Setting 96) is 45 degrees.





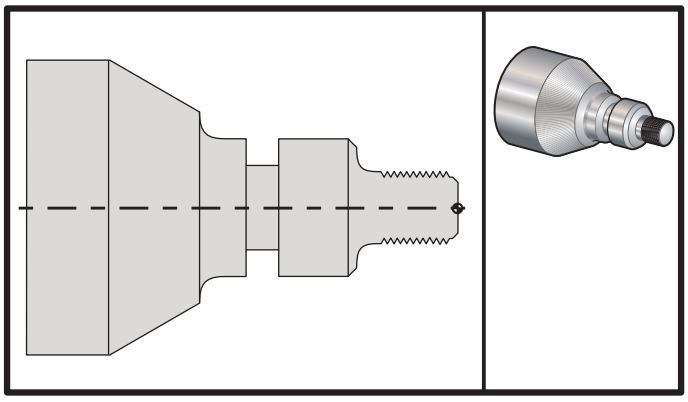
G76 using an "A" Value

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.To calculate the X position of each pass you have to take the sum of all the previous passes, measured from the start point the X value of each pass



G76 Thread Cutting Cycle, Multiple Pass

Program Example	Description
%	
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 X1.2 Z0.3 M08	(Rapid to position)
G76 X0.913 Z-0.85 K0.042 D0.0115 F0.0714	(Threading cycle)
G00X1.5 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.

Multiple Start Threading Example

Multiple 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).

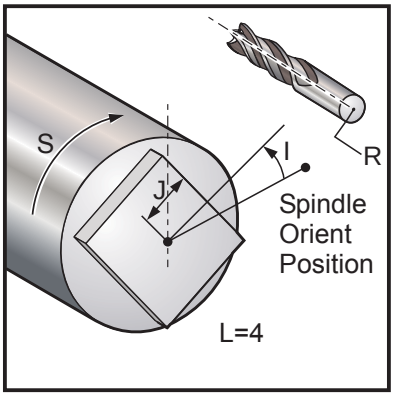
```
(1) M08
(2) G00 X1.1 Z0.5                                  (Initial Start Point)
(3) G76 X0.913 Z-0.85 K0.042 D0.0115 F0.0714       (Threading cycle)
(4) G00 X1.1 Z0.5238                               (Next Start Point [.5 + .0238 = 5.238])
(5) G76 X0.913 Z-0.85 K0.042 D0.0115 F0.0714       (Threading cycle)
(6) G00 X1.1 Z0.5476                               (Last Start Point [.5238 + .0238 = 5.476])
(7) G76 X0.913 Z-0.85 K0.042 D0.0115 F0.0714       (Threading cycle)
```

G77 Flatting Cycle (This G-code is optional and is used for live tooling) (Group 00) (Also see the C-axis section)

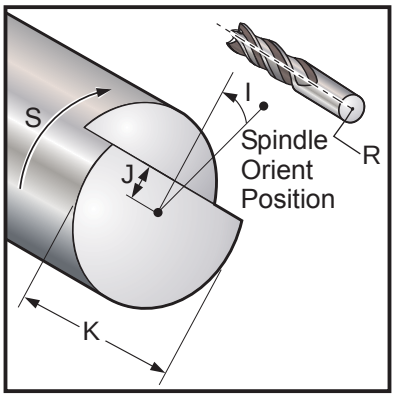
NOTE: This cycle is only available on lathes with the live tooling option.

* 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.

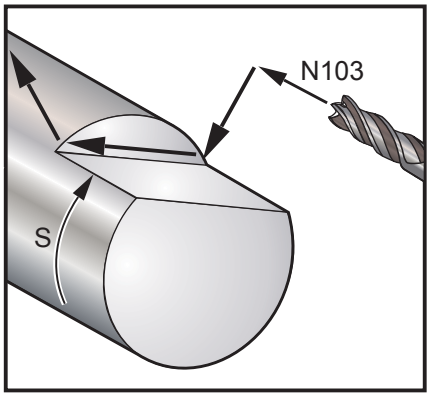
The S value specifies the rpm speed that the spindle will maintain during the flatting cycle. The default value is 6. Higher values will not affect the flatness, but will affect the position of the flats. To calculate the maximum error in degrees, use RPM * .006.

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.

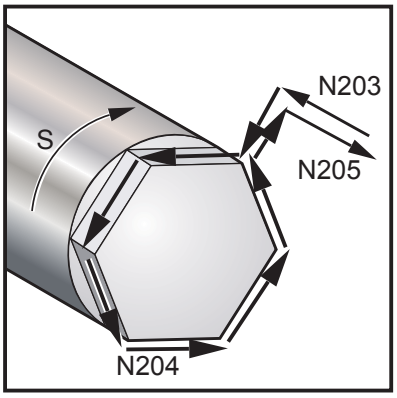
Flatting Examples with G77:

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:



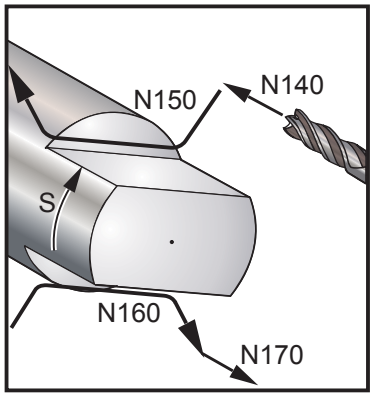

```
...
N100 S10 M03(Start spindle)
N101 M133 P1000 (Turn live tool)
N102 G00 X6.1
N103 Z-1.
N104 G77 J1.5 K4. R0.5
N105 Z1.
N106 M135 (Stop live tool)
N107 M05 (Stop spindle)
...
```

Cut a hexagon into the top half inch of a part that is three inches in diameter, using a tool 1/2” 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
%
```

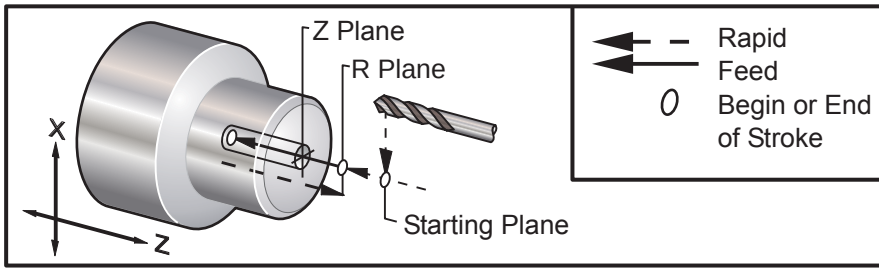
G80 Canned Cycle Cancel (Group 09*)

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



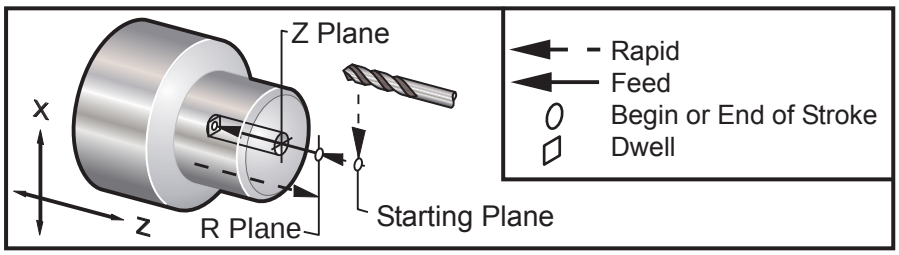
G81 Drill Canned Cycle

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.

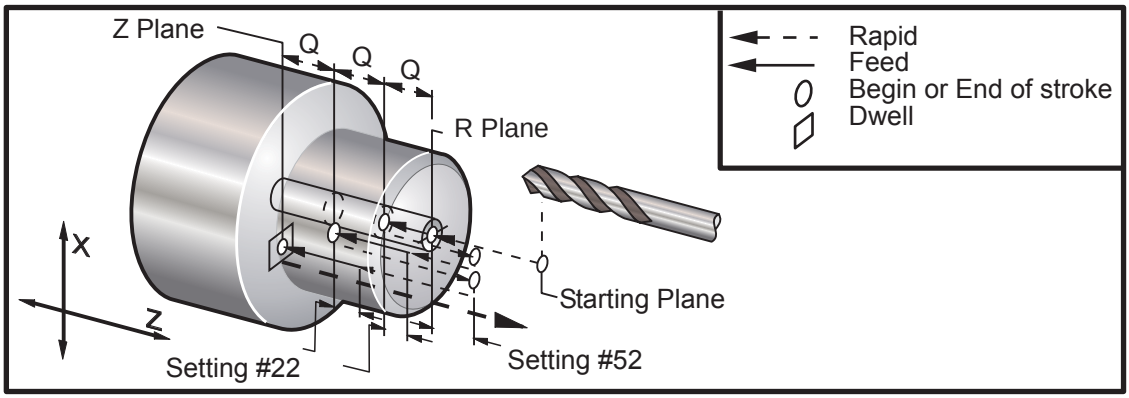


G82 Spot Drill Canned Cycle

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. Do not use a **Q** value when programming with **I**,**J**,**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

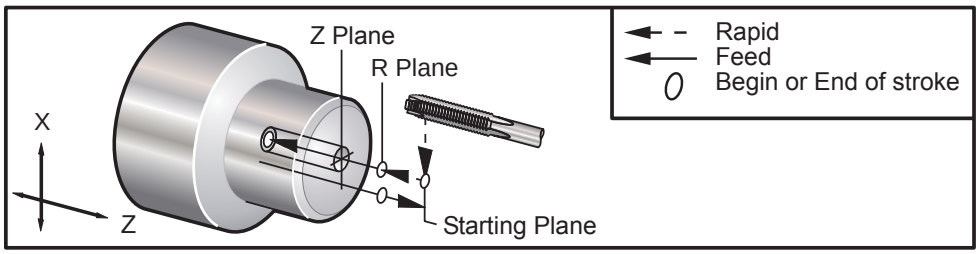
Programming Notes: It is not necessary to start the spindle CW before this canned cycle. The control does this automatically.

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

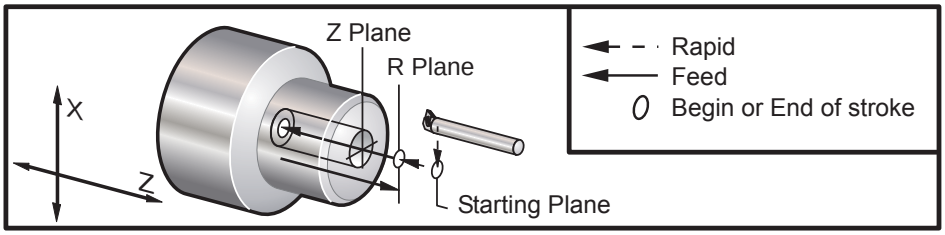


G84 Tapping Canned Cycle

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



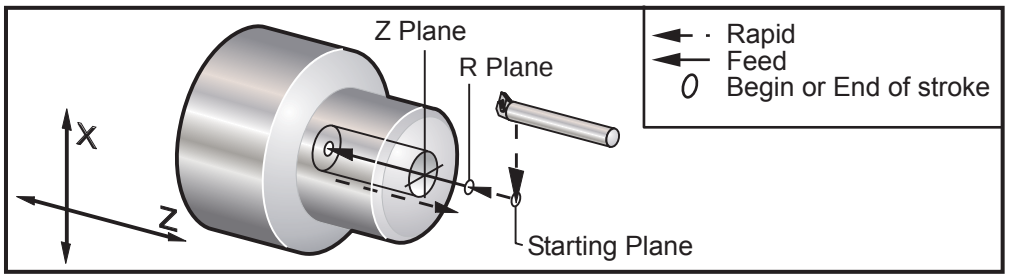
G85 Boring Canned Cycle

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 tool will be retracted once the spindle has stopped.

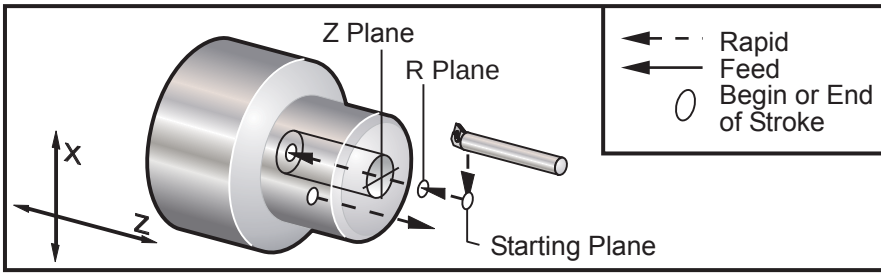


G86 Bore & Stop Canned Cycle

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



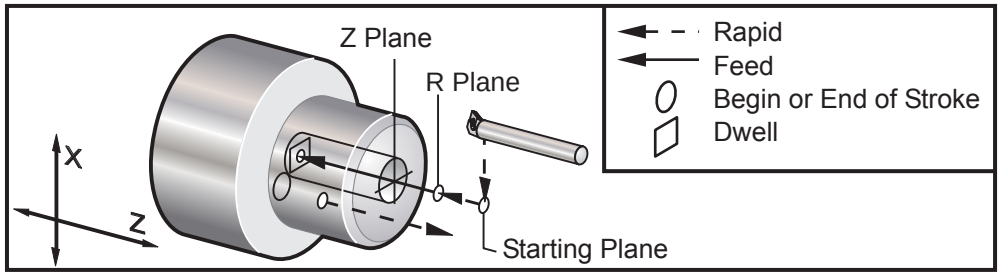
G87 Bore & Manual Retract Canned Cycle

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.

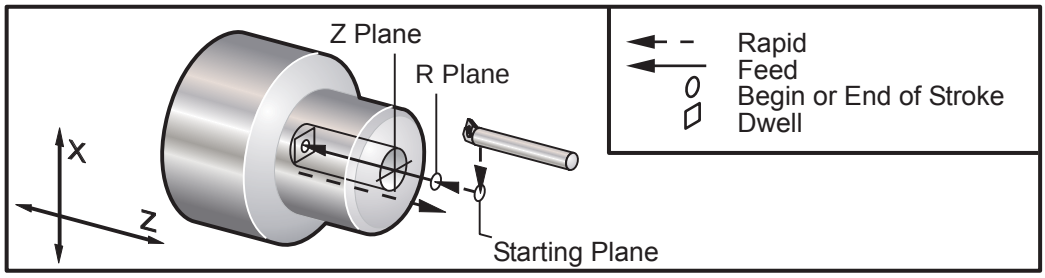


G88 Bore, Dwell & Manual Retract Canned Cycle

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

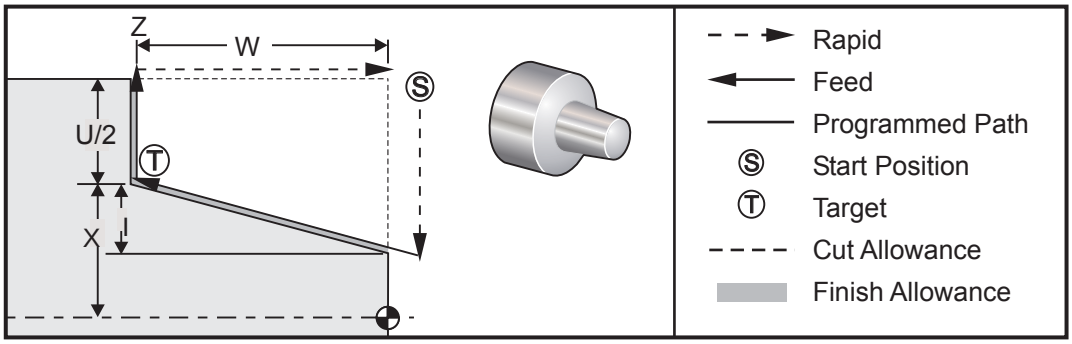
* indicates optional



G89 Bore & Dwell Canned Cycle

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

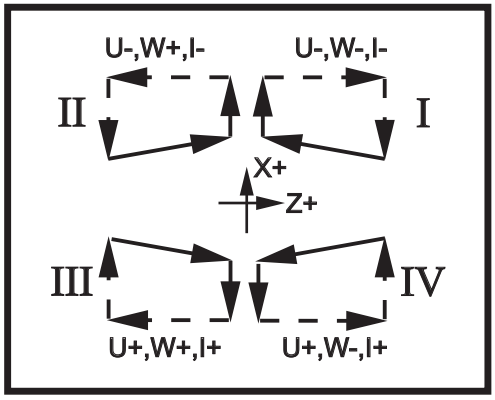
F(E) Feed rate
* I Optional distance and direction of X axis taper, radius
* U X-axis incremental distance to target, diameter
* W Z-axis incremental distance to target
X X-axis absolute location of target
Z Z-axis absolute location of target
* indicates optional



G90 is used for simple turning, however, multiple passes are possible by specifiing 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

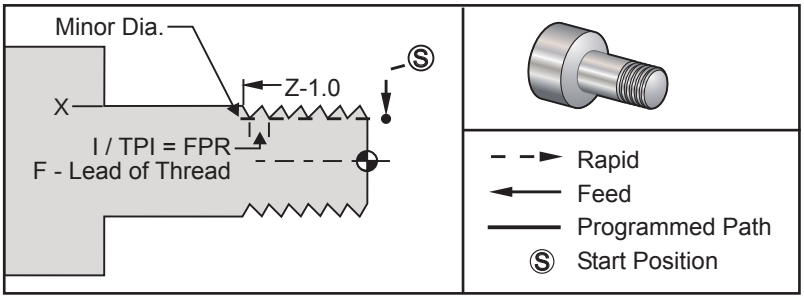
G92 Threading Cycle (Group 01)

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 specifiing 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		**Description**
%		(1”-12 Thread Cutting Program)
T101		
G50 S3000 M3		
G97 S1000		
X1.2 Z.2		(Rapid to Clear position)
G92 X.980 Z-1.0 F.0833		(Set up Threading cycle)
2nd pass	X.965	(Subsequent Passes)
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	(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. Q270123 F0.2; (270.123 degree cut)

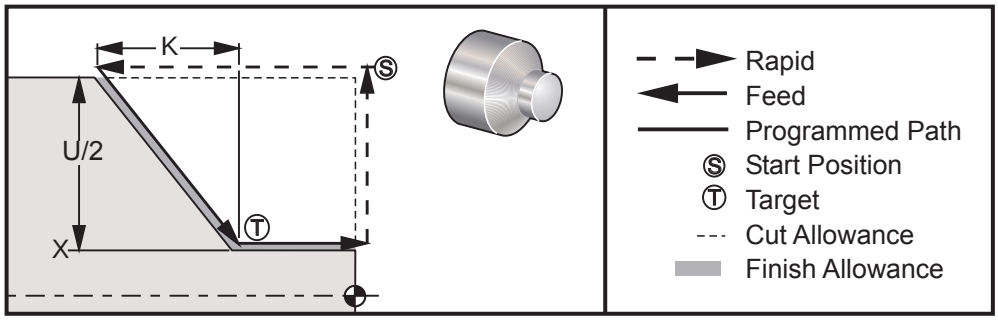
The following rules apply to the usage of Q:

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. Do not use a decimal point in the entry; 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

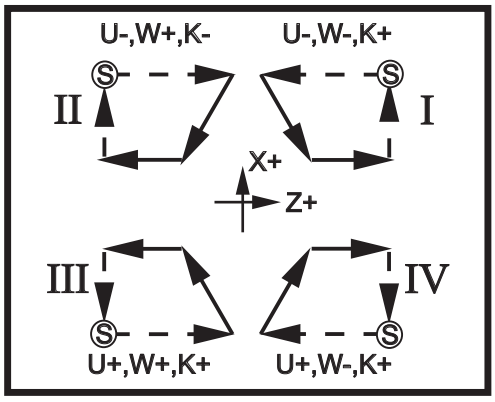


G94 End Face Turning Cycle

Straight end facing cuts can be made by just specifiing 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. 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:

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

```
%
O00800
N1 T101 (Axial 1/4-20 Tap)
G99 (Neccesary 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 commands the control to maintain a constant cutting speed. This means the spindle speed increases as the workpiece gets smaller. 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 commands 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. Use this G code 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”, and describes 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.

G105 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, G17 XY plane is used for G01 linear XY strokes and G02 and G03 for circular motion. X, Y position commands are converted into rotary C-axis and linear X-axis moves.

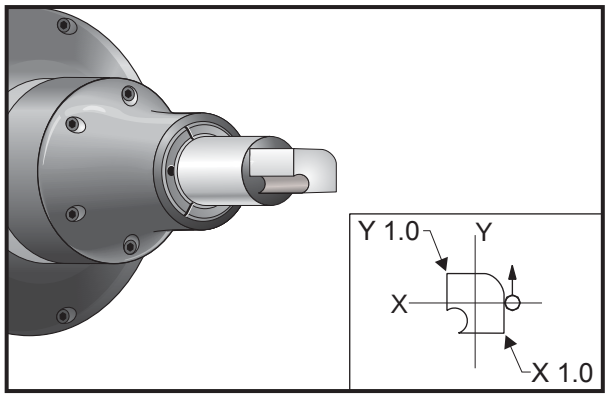
Note that mill-style Cutter Compensation becomes active when G112 is used. Cutter Compensation (G41, G42) must be canceled (G40) before exiting G112.

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.

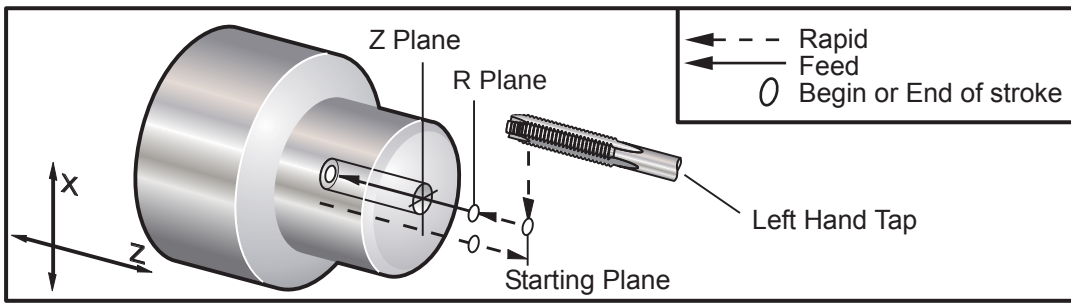
G161 APL Axis Command Mode Off

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 (optional)
*X	X-axis motion command (optional)
*Z	Position of bottom of hole (optional)

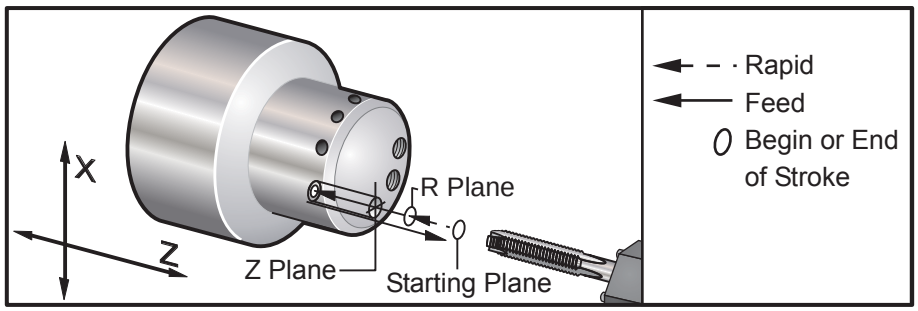
Programming Notes: When tapping, the feedrate is the lead of the thread. See example of G84.
It is not necessary to start the spindle CCW before this canned cycle; the control does this automatically.



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.

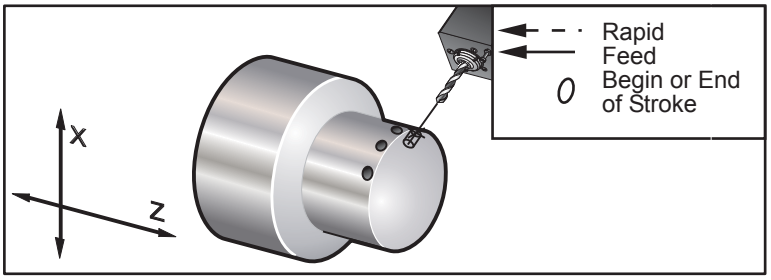
G195 Live Tool Radial Tapping (Diameter) (Group 00)

F Feed Rate per revolution (G99)
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 per revolution (G99)
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 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.

M Code List

M00 Stop Program
M01 Optional Program Stop
M02 Program End
M03 Spindle Forward
M04 Spindle Reverse
M05 Spindle Stop
M08 Coolant On
M09 Coolant Off
M10 Clamp Chuck
M11 Unclamp Chuck
M12 Auto Air Jet On (Optional)
M13 Auto Air Jet Off (Optional)
M14 Spindle Brake On
M15 Spindle Brake Off
M17 Turret Rotation Always Forward
M18 Turret Rotation Always Reverse
M19 Orient Spindle (Optional)
M21-M28 Optional User M Function with M-Fin
M21 Tailstock Forward
M22 Tailstock Reverse
M23 Thread Chamfer ON
M24 Thread Chamfer OFF
M30 Prog End and Rewind
M31 Chip Conveyor Forward
M33 Chip Conveyor Stop
M36 Parts Catcher Up (Optional)
M37 Parts Catcher Down (Optional)
M38 Spindle Speed Variation On
M39 Spindle Speed Variation Off
M41 Low Gear (if equipped with transmission)
M42 High Gear (if equipped with transmission)
M43 Turret Unlock (Service Use Only)
M44 Turret Lock (Service Use Only)
M51-M58 Optional User M turn ON
M59 Set Output Relay
M61-M68 Optional User M turn OFF
M69 Clear Output Relay
M76 Disable Displays
M77 Enable Displays
M78 Alarm if skip signal found
M79 Alarm if skip signal not found
M85 Open Automatic Door (optional)
M86 Close Automatic Door (optional)
M88 Turns On High Pressure Coolant (optional)
M89 Turns Off High Pressure Coolant (optional)
M93 Start Axis Pos Capture
M94 Stop Axis Pos Capture
M95 Sleep Mode
M96 Jump if no Input
M97 Local Sub-Program Call
M98 Sub Program Call
M99 Sub Program Return Or Loop
M109 Interactive User Input
M119 Subspindle Orient (Optional)
M121-128 Optional User M
M133 Live Tooling Drive Forward (Optional)
M134 Live Tool Drive Reverse (Optional)
M135 Live Tool Drive Stop (Optional)
M143 Subspindle Forward (Optional)
M144 Subspindle Reverse (Optional)
M145 Subspindle Stop (Optional)
M154 C-axis engage (Optional)
M155 C-axis disengage (Optional)

M00 Stop Program

M00 stops a program. It stops the axes, spindle, turns off the coolant (including Through Spindle Coolant). The next block (block after 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

M02 ends 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 1500 RPM.

M08 Coolant On / M09 Coolant Off

M08 turns on the optional coolant supply and M09 turns it off (also see M88/89 for High Pressure Coolant).

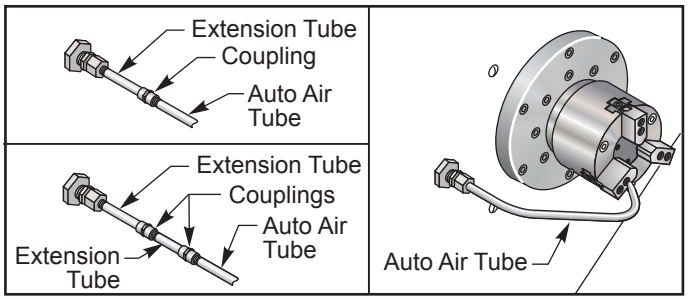
NOTE: Coolant status is checked only at the start of a program, so a low coolant condition will not stop a program that is already running.

M10 Clamp Chuck / M11 Unclamp Chuck

M10 clamps the chuck and M11 unclamps it. 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)

M12 and M13 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 Main Spindle Brake On / M15 Main Spindle Brake Off

These M Codes are used for machines equipped with the optional C-Axis. M14 applies a caliper-style brake to hold the main spindle, while M15 releases the brake.

M17 Turret Rotation Always Forward / M18 Turret Rotation Always Reverse

M17 and M18 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)

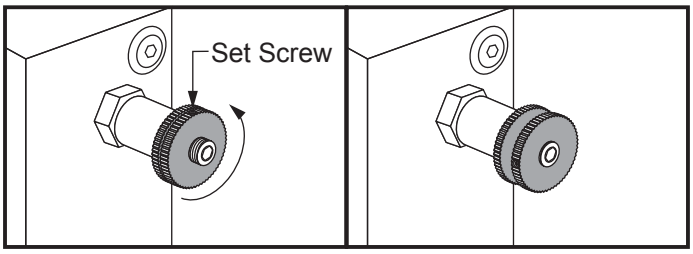
M19 adjusts the spindle to a fixed position. The spindle will only orient to the zero position without the optional M19 spindle orient feature.

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.

Spindle orientation is dependant on the mass, diameter and length of the workpiece and/or the workholding (chuck). Contact the Haas Applications Department if any unusually heavy, large diameter, or long configuration is used.

M21 Tailstock Forward / M22 Tailstock Reverse

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

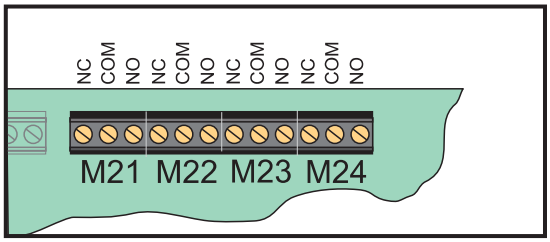


M21-M28 Optional User M Function with M-Fin

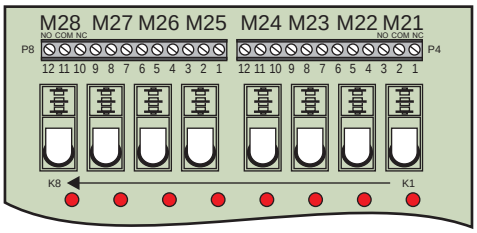
The M codes M21 through M28 are optional for user relays; each M code activating one of the optional relays. The Reset button will terminate any operation that is waiting for a relay-activated accessory to finish (also see M51-58 and M61-68).

Some or all of the M21-25 (M21-M22 on Toolroom and Office lathes) on the I/O PCB may be used for factory-installed options. Inspect the relays for existing wires to determine which have been used. Contact the Haas factory for more details.

M-Code Relays - These outputs can be used to activate probes, auxiliary pumps or clamping devices etc. The auxiliary devices are electrically connected to the terminal strip for the individual relay. The terminal strip has a position for, Normally Open (NO), Normally Closed (NC) and Common (CO).



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, 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 I/O PCB. 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 or M59/69. Parameter 352 is shipped set to “1” as standard.

NOTE: With any probing option (wit the exception of LTP), Parameter 352 must be set to ‘1’. When the 8M option is installed, access its relays using M59/69.

M23 Thread Chamfer ON / M24 Thread Chamfer OFF
M23 commands the control to execute a chamfer at the end of a thread executed by G76 or G92. M24 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 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 Reset
M30 stops 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)
M36 activates the optional parts catcher. M37 deactivates the optional parts catcher. M36 rotates the parts catcher into position to catch a part. M37 rotates 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 to 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, set Setting 165 to 50 and Setting 166 to 30. Using 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 the Reset button also turns SSV Off. If the RPM swing is larger than the commanded speed value, any negative RPM values (below zero) will translate into an equivalent positive value. The spindle, however, will not be allowed to go below 10 RPM when SSV mode is active.

Constant Surface Speed: When Constant Surface Speed (G96) is activated (which will calculate spindle speed) the M38 command will alter that value using Settings 165 and 166.

Threading Operations: G92, G76 and G32 will allow the spindle speed to vary in SSV mode. This is **not** recommended due to possible thread lead errors caused by mismatched acceleration of the spindle and the Z-axis.

Tapping cycles: G84, G184, G194, G195 and G196 will be executed at their commanded speed and SSV will not be applied.

M41 Low Gear / M42 High Gear
On machines with a transmission, M41 selects low gear and 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.

NOTE: 8M #1 uses addresses 1140-1147.

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

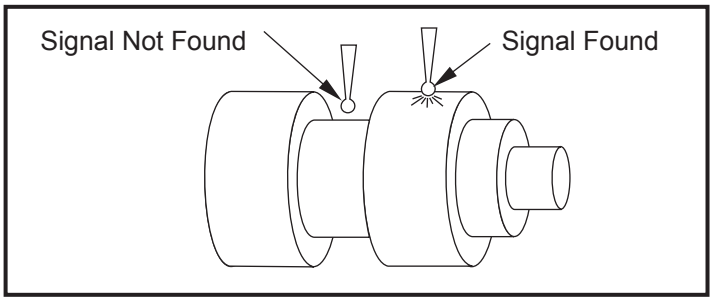
This M code turns off a relay. An example of its usage is **M69 Pnn**, where “nn” is the number of the relay being turned off. An M69 command can be used to turn off any of the output relays in the range from 1100 to 1155. When using Macros, M69 P1103 does the same thing as using the optional macro command #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. M78 generates 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. M79 generates 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)

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

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

M88 turns 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.

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 M95 must contain the hours and minutesfor which the machine is to sleep. For example, if the current time were 6pm and the user wanted the machine to sleep until 6:30am the next day, the following command would be used; M95 (12:30). The line(s) following M95 should be axis moves and spindle warm-up commands.

M96 Jump If No Input

P Program block to go to when conditional test is met
Q Discrete input variable to test (0 to 63)

This code is used to test a discrete input for 0 (off) status. This is useful for checking the status of automatic work holding or other accessories that will generate a signal for the control. The Q value must be in the range 0 to 63, which corresponds to the inputs found on the diagnostic display (The upper left input is 0 and the lower right is input 63. When this program block is executed and the input signal specified by Q has a value of 0, the program block Pnnnn is performed (the Pnnnn line must be in the same program). M96 Example:

```
N05 M96 P10 Q8    (Test input #8, Door Switch, until closed);
N10               (Start of program loop);
.
.                 (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 L**nn** code in the M97 block will repeat the subroutine call that **nn** times. M97 Example:

```
O0001
M97 P1000 L2                (L2 command will cause the program to run the N1000 line twice)
M30
N1000 G00 G90 G55 X0 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 M99 to return to the main program. An **Lnn** count can be put on the line containing M98 and will cause the subroutine to be called **nn** times before continuing to the next block.

```
O0001                   (Main Program number)
M98 P100 L4;            (Call sub-program, sub -program number, loop 4 times)
M30                     (End of program)

O0100                   (Sub-progam Number)
G00 G90 G55 X0 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
```

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.

M104 Extend Probe Arm / M105 Retract Probe Arm - The optional tool setting probe arm is extended and retracted using M104 and M105.

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, 3, 4 or a 5 to be entered; all other characters will be ignored.

```
%
O01234 (M109 Program)

N1 #501= 0 (Clear Variable #501)
(Variable #501 will be checked)
(Operator enters one of the following selections)
M109 P501 (1,2,3,4,5)
N5 IF [ #501 EQ 0 ] GOTO5
(Wait for keyboard entry loop until entry)

(Decimal equivalent from 49-53 represent 1-5)

IF [ #501 EQ 49 ] GOTO10 (1 was entered go to N10)
IF [ #501 EQ 50 ] GOTO20 (2 was entered go to N20)
IF [ #501 EQ 51 ] GOTO30 (3 was entered go to N30)
IF [ #501 EQ 52 ] GOTO40 (4 was entered go to N40)
IF [ #501 EQ 53 ] GOTO50 (5 was entered go to N50)

GOTO1 (Keep checking for user input loop until found)

N10
(If 1 was entered run this sub-routine)
(Go to sleep for 10 minutes)
#3006= 25 (Cycle start sleeps for 10 minutes)
M95 (00:10)
GOTO100

N20
(If 2 was entered run this sub routine)
(Programmed message)
#3006= 25 (Programmed message cycle start)
GOTO100

N30
(If 3 was entered run this sub routine)
(Run sub program 20)
#3006= 25 (Cycle start program 20 will run)
G65 P20 (Call sub-program 20)
GOTO100
```

```
N40
(If 4 was entered run this sub routine)
(Run sub program 22)
#3006= 25 (Cycle start program 22 will be run)
M98 P22 (Call sub program 22)
GOTO100

N50
(If 5 was entered run this sub-routine)
(Programmed message)
#3006= 25 (Reset or cycle start will turn power off)
#1106= 1

N100
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 Rxx.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 a P address code. For example, P1200 would command a spindle speed of 1200 RPM.

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 1200 RPM.

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

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

The setting pages contain values that control machine operation and that the user may need to change. Most settings can be changed by the operator. They 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

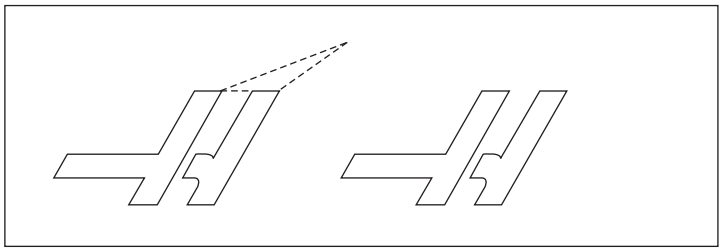
This setting is used to power-down the machine when it has not been used for some time. The value entered in this setting is the number of minutes the machine will remain idle until it is powered down. The machine will not be powered down while a program is running, and the time (number of minutes) will start back at zero anytime buttons is pressed or the jog handle is used. The auto-off sequence gives the operator a 15-second warning before power down, at which time pressing any button will stop the power down.

2 - Power Off at M30

Powers down the machine at the end of a program (M30) if this setting is set to “On”. The machine will give the operator a 30-second warning once an M30 is reached; pressing any button will interrupt the sequence.

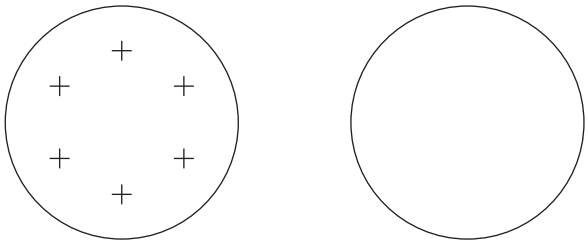
4 - Graphics Rapid Path

This setting changes the way a program is viewed in the Graphics mode. When it is Off, rapid (non-cutting tool motions do not leave a path. When it is On, rapid tool motions leave a dashed line on the screen.



5 - Graphics Drill Point

This setting changes the way a program is viewed in Graphics mode. When it is ON, motion in the Z-axis will leave an **X** mark on the screen. When it is OFF, no additional marks are shown on the graphics display.



6 - Front Panel Lock

This Setting disables the Spindle CW and CCW buttons when it is set to “On”.

7 - Parameter Lock
Turning this setting On will stop the parameters from being changed, except for parameters 81-100. Note that 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 350 ipm 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 that 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. 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 RS-232 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-xis
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.

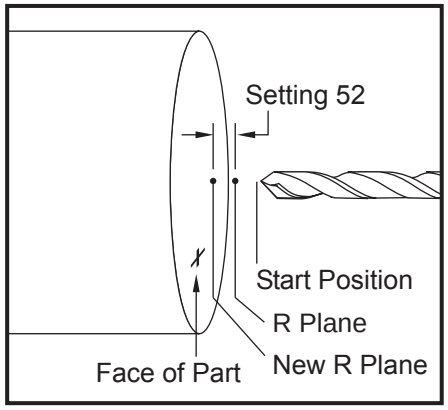
The “XMODEM” selection is receiver-driven, which sends data in blocks of 128 bytes. XMODEM gives the RS-232 communication added reliability because each block is checked for integrity.

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 30.00 inches or 0-761mm). 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 the 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”.

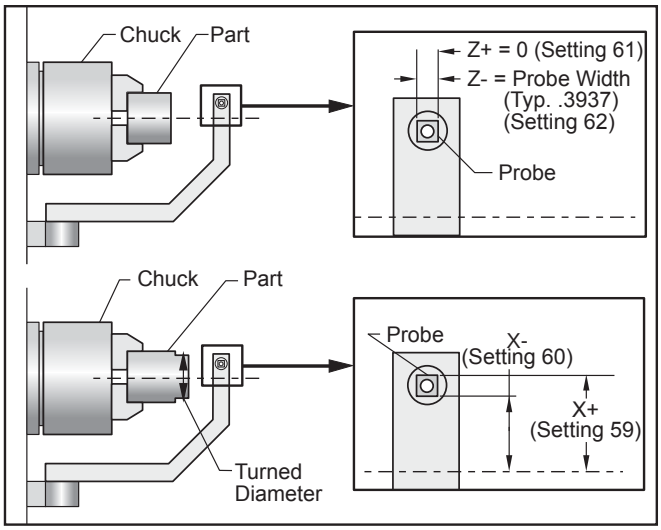
56 - M30 Restore Default G
When this setting is On, ending a program with M30 or pressing Reset returns all modal G codes to their defaults.

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.

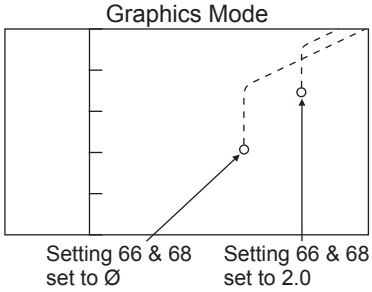
64 - T. OFS Meas Uses Work
This setting changes the way the Tool Ofset Mesur (Tool Offset Measure) button works. When this is On, the entered tool offset will be the measured tool offset plus the work coordinate offset (Z-axis). When it is Off, the tool offset equals the Z machine position.

65 - Graph Scale (Height)
This setting specifies the height of the work area that is displayed on the Graphics mode screen. The default value for this setting is the maximum height, which is the entire machine work area. Using the following formula can set a specific scale:

Total Y travel = Parameter 20/Parameter 19
Scale = Total Y travel/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.

	OUTPUT	
#1 = 3.0 ;		
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 are 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

There are 5 feedrate settings:

INCH		**MILLIMETER**	
DEFAULT	(.0001)	DEFAULT	(.001)
INTEGER	F1 = F1	INTEGER	F1 = F1
.1	F1 = F.0001	.1	F1 = F.001
.01	F10 = F.001	.01	F10 = F.01
.001	F100 = F.01	.001	F100 = F.1
.0001	F1000 = F.1	.0001	F1000 = F1

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 restores 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.

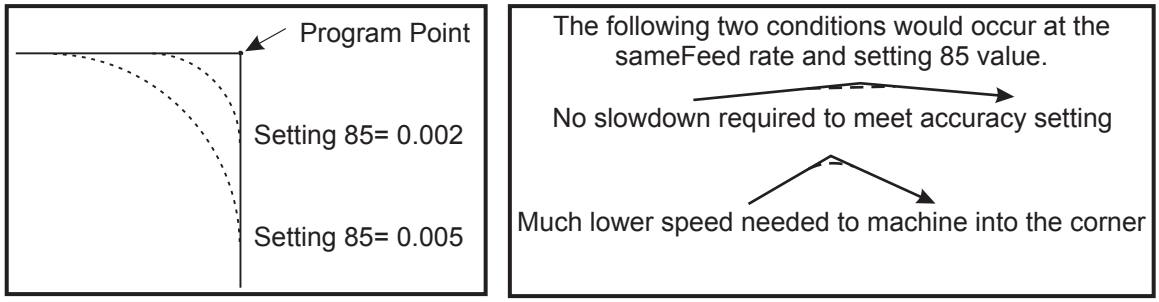
When set to “Autofeed”, the lathe automatically limits the feedrate based on the tool load.

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

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

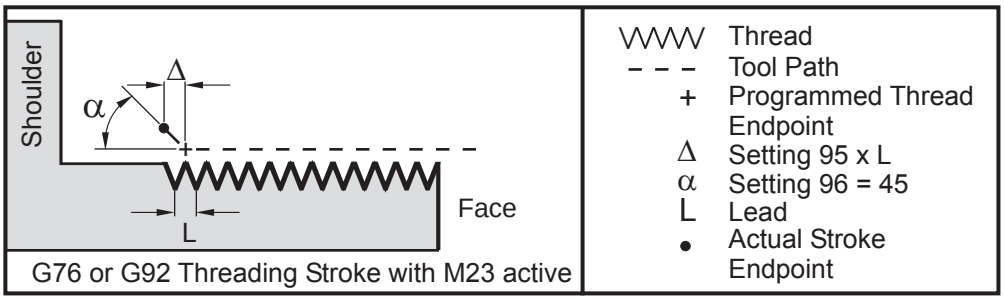
This setting adjusts for extreme values in tool geometry or shift values. In graphics, tool offsets are ignored so that the cutting paths of different tools are displayed in the same location. Setting this to an approximate value of machine coordinates for the programmed part zero will void any x Over 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 that 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 scren 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 affect 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 Setting 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 (max = ± 0.0020" or ± 0.051 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 (Conveyor Cycle Time) is the interval that the conveyor will turn on automatically. Setting 115 (Conveyor On-Time) is the amount of time 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 on every half an hour, run for 2 minutes, then turn off.

On-time should be set no greater than 80% of cycle time. Note the following:

The CHIP FWD button (or M31) will start the conveyor in the forward direction and activate the cycle.

The CHIP REV button (or M32) will start the conveyor in the reverse direction and activate the cycle

The CHIP STOP button (or M33) will stop the conveyor and cancel the cycle.

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 an 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.

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, and to set Macro variables by using an E command. This feature is software-based and requires an additional computer to request, interpret and store data from the control. A hardware option also allows the reading of machine status. See CNC Data Transfer in the Operation Programming section for detailed information.

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.

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 the setting is Off, values following address codes that do not include decimal points are taken as machinists notation (i.e., thousandths or ten-thousandths.) This setting will exclude the A value (tool angle) in a G76 block. Thus, the feature applies to the following address codes:

	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
A (except with G76)If a G76 A value containing a decimal point is found during program execution, alarm 605 Invalid Tool Nose Angle is generated.
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 is used to set the maximum spindle RPM each time the machine is powered on. It will essentially cause a G50 Snnn command to be executed at power on time, where nnn is the value from the setting. If the setting contains zero, or a value equal to or greater than parameter 131 MAX SPINDLE RPM, Setting 164 will have no effect.

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.

166 - SSV CYCLE (0.1) SECS

Specifies the duty cycle, or the rate of change of Spindle Speed. Positive value only.

167-186 - Periodic Maintenance

There are 14 items that can be monitored, as well as six spare items, in the periodic maintenance settings. These settings will allow the user to change the default number of hours for each item when it is initialized during use. If the number of hours is set to zero, the item will not appear in the list of items shown in the maintenance page of current commands.

187 - Machine Data Echo

Turning this setting on will display the data collection Q commands on the PC screen.

201 - Show Only Work and Tool Offsets In Use

Turning this setting on will display only the Work and Tool offsets used by the running program in Graphics or MEM mode.

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	High Voltage Requirements
SL-10	(195-260V)	(354-488V)
Power Supply	50 AMP	25 AMP
Haas Circuit Breaker	40 AMP	20 AMP
If service run from elec. panel is less than 100' use:	8 GA. WIRE	12 GA. WIRE
If service run from elec. panel is more than 100' use:	6 GA. WIRE	10 GA. WIRE

20 HP System	Voltage Requirements	High Voltage Requirements
[1]SL-20, TL-15	(195-260V)	(354-488V)
Power Supply	50 AMP	25 AMP
Haas Circuit Breaker	40 AMP	20 AMP
If service run from elec. panel is less than 100' use:	8 GA. WIRE	12 GA. WIRE
If service run from elec. panel is more than 100' use:	6 GA. WIRE	10 GA. WIRE

30-40 HP System	Voltage Requirements	High Voltage Requirements[2]
TL-15BB, SL-20BB, SL-30, SL-30BB, [1]SL-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	High Voltage Requirements
[1]SL-40, SL-40BB, SL-40L	(195-260V)	(354-488V)
Power Supply	150 AMP	**Must use an external transformer**
Haas Circuit Breaker	125 AMP	
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 un-grounded 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 260V, or 504V for high-voltage machines with the Internal High Voltage Option.

[1] The 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.

[2] The 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 loose 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

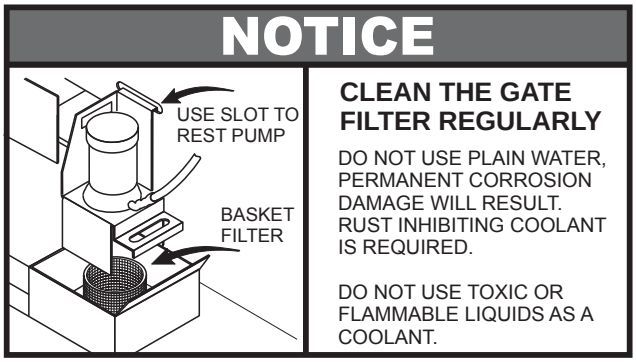
The following is a list of required regular maintenance for the Haas SL-Series Turning Centers. Listed are the frequency of service, capacities, and type of fluids required. These required specifications must be followed in order to keep your machine in good working order and protect your warranty.

Interval	Maintenance Performed
Daily	• 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 (10 gallons for SL-30B and up).
Weekly	• 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	• 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. • Check for dust buildup on electrical cabinet vector drive vents (beneath power switch). If buildup exists, open cabinet and wipe vents with a clean cloth rag. Apply com pressed air as necessary to remove dust accumulation.
Six Months	• Replace coolant and thoroughly clean the coolant tank. • Replace hydraulic unit oil filter. • Check all hoses and lubrication lines for cracking.
Annually	• Replace gearbox oil. • Clean oil filter inside Lube Air Panel Oil Reservoir and clean residue from bottom of filter.

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	Mobil Vactra #2	2-2.5 qts
Transmission	Mobil SHC 625	2.25 liters

Periodic Maintenance

A periodic maintenance page is found within the Current Commands screens titled “Maintenance”. Access the screen by pressing CURNT COMDS and using Page Up or Page Down to scroll to the page.

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 “Maintenance” screen, press the Origin button to deactivate it, then press Origin again to reactivate it with the default number of hours remaining.

Refer to settings 167-186 for additional maintenance defaults. Note that settings 181-186 are used as spare maintenance alerts by keying-in a number. The maintenance number will display on the Current Commands page once a value (time) is added to the setting.

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 manufactures’ 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 and void the warranty**.

Coolant must have rust inhibitors. Do not use pure water as a coolant; machine components will rust.

Do not use flammable liquids as coolant.

Acidic and high alkaline fluids will damage components throughout the machine.

See safety section and labeling concerning flammable and explosive fluids and materials.

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

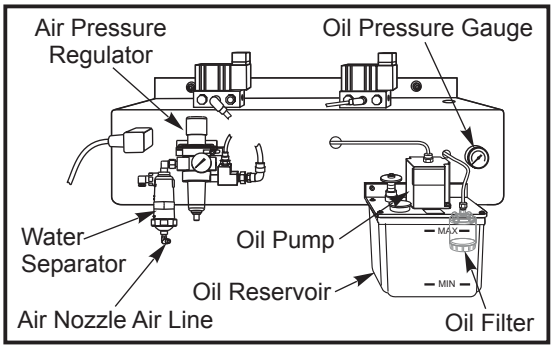
When machining castings, sand from the casting process and the abrasive properties of cast aluminum and cast iron will shorten coolant pump life unless a special filter is used in addition to the standard filter. Contact Haas Automation for recommendations.

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).

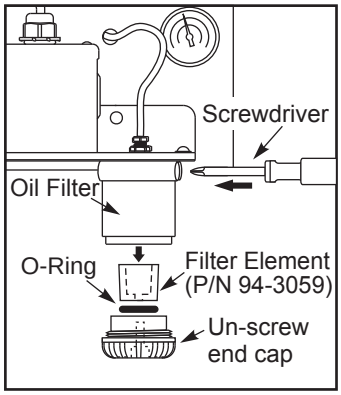
To change the filter element follow these steps:

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 and clean the inside of the filter housing and the filter end cap as required.
4. 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.

5. Replace the oil reservoir; ensure the gasket seats properly between the reservoir and the top flange.



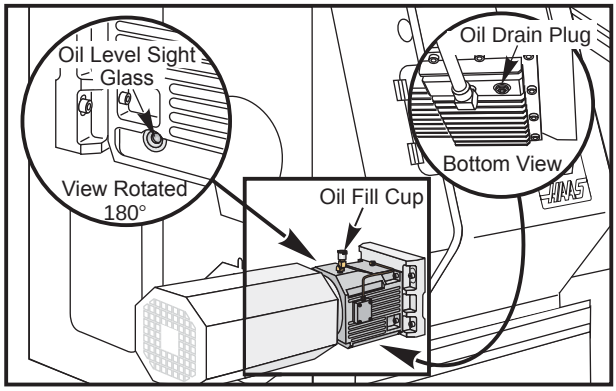
Transmission Oil

Oil Check

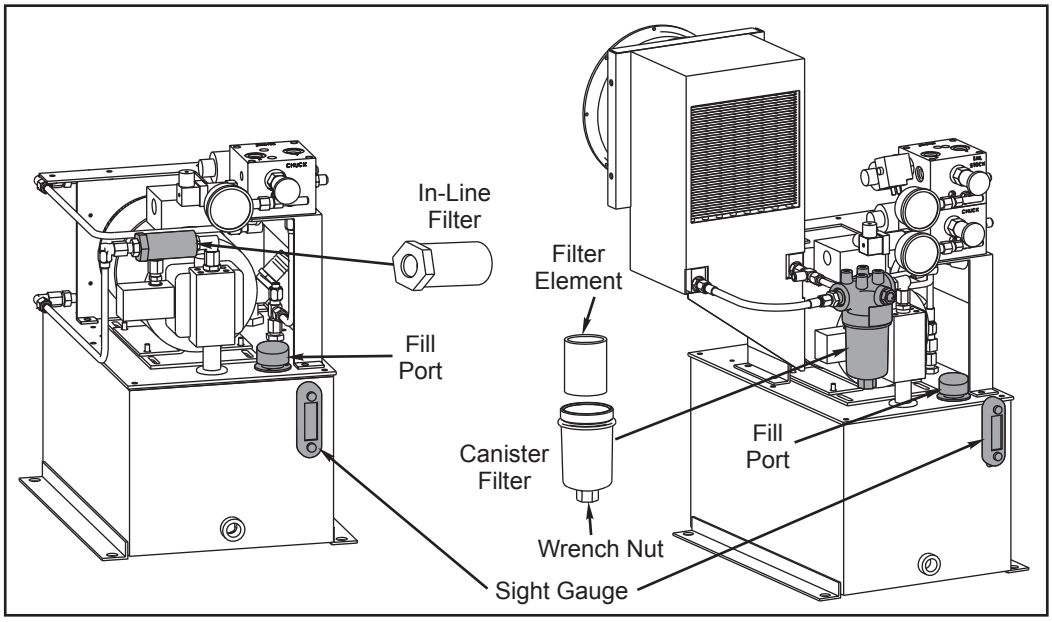
Check the oil level at the sight glass through the opening in the side of the machine as shown in the illustration. Fill as needed through the fill port on top of the gear box.

Oil Change

1. Remove the sheet metal necessary to gain access to the transmission, and remove the fourteen (14) SHCS from the oil pan and remove it. Inspect the magnetic drain-plug for signs of metal particles.
2. Wipe down the oil pan and reinstall it with a new gasket. 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.
3. Fill the gear case with 2¼ liters of Mobil SHC-625 gear oil. Check the sight glass. The level should be 3/4 of the way up when full. Fill as needed.
4. Install access plate with new gasket, and run a spindle warm-up and check for leaks.



Hydraulic Power Unit (HPU)



Checking the Oil Level

Observe that the oil level is above the fill line in the sight gauge of the HPU. If not, use the fill port to supply DTE-25 oil to the unit. Fill the unit until oil is observed at the top of the sight glass.

Oil Filter Replacement

In-Line: Unscrew the filter at both ends, remove it from the unit and replace it with a new in-line filter. Discard the old filter.

Canister: Unscrew the canister using the wrench nut at the bottom, remove the filter element and replace it with a new one. Tighten the canister using the wrench nut. Discard the old filter element.

NOTE: If a barfeeder or automatic parts loader is connected to the lathe, remove it to gain access to the Hydraulic Power Unit.

SL-30B \ SL-40 Filters and Replacement Elements

Filter Manufacturer	Oil Filter Part Number	Replacement Element Part Number
Pall	58-1064	58-1065
Hydac	58-1064	58-6034
Flow Ezy	58-1064	58-1067

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 the same way 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).

Loosen the clamps and open the 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 the clamps.

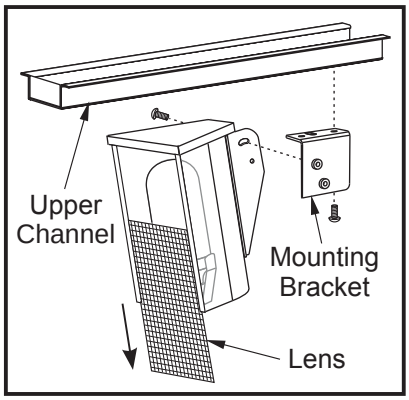
1000 psi HPC Maintenance

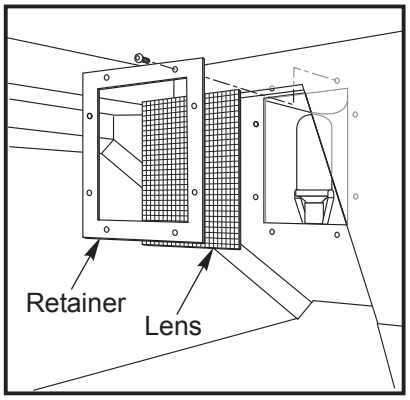
Before doing any maintenance to the 1000psi system, disconnect the power source; unplug it from the power supply.

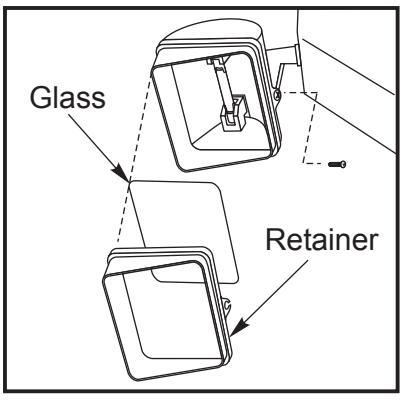
Check the oil level on a daily basis. If the oil is low, add oil through the fill cap on the reservoir. Fill the reservoir about 25% full with 5-30W synthetic oil.

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.

Index

A

Accuracy Control 162
Air Requirements 188
Alarms 17
Auto Door 168
Axis Load Monitor 23

B

Background Edit 42

C

Calculator 17, 24
Cartesian Interpolation 54
Cartesian to Polar Transformation 53
C-Axis 52, 172
Chip Auger 15, 167, 194
Chuck Maintenance 190
Coolant 165, 191
Coolant Tank 191
Corner Rounding/Chamfering 116, 120
Current Commands 56

D

Date 24
Direct Numeric Control 36
DIRECTORY LISTING 36
DNC 36

E

Electricity Requirements 187
End-Of-Block 18
Enhanced Remote Jog Handle 28
EOB 18
Ethernet 36

G

G65 Macro Call 115
G65 Macro Subroutine Call 115

H

Hard Drive 36
Help 17
Home 18
HOME G28 20

I

Index on the Fly 163

J

Jogging, 2-Axis 15

K

Keyboard 14
Key Switch 15

L

Live Tooling 48, 172
Lubrication System 191

M

Machine Data Collect 185
Macro Subroutine Call 115
Macro Subroutine Call (G65) 23
Maintenance Schedule 189
Maximum Corner Rounding 17
Memory Lock Key Switch 164

O

Option Try-Out 27
Orient Spindle 165

P

Parameters 17
Parentheses 18
Periodic Maintenance 190
Power Up 14

R

Remote Jog Handle, Enhanced 28
right slash 18
RS-232 38, 159, 176, 185

S

Second Home 15
Settings 17
shift key 18
Sleep Mode 169
Spindle Orientation 165
Square brackets
Sub-Program 169
subroutine 169

T

Tailstock 15, 166, 182
Taper Compensation 48
Time 24
Tool Geometry 21
Tool Life Display 22

Tool Load Monitor and Display 22
Tool Nose Compensation 58
TOOL OFSET MESUR 179
Transmission Oil 192

U

USB 36
User M Function 164, 166

V

VDI Installation 52
Visual Quick Code 91

W

Warm-up Compensation 183
Worklight 194
Work Zero Offset 21

X

XZ (2-Axis) Jogging 15