



Lathe Operator's Manual

J U N E 2 0 0 1

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



Haas Technical Publications

HAAS SERVICE AND OPERATOR MANUAL ARCHIVE

SL-Series Operators Manual 96-8700 RevC English June 2001

- 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.

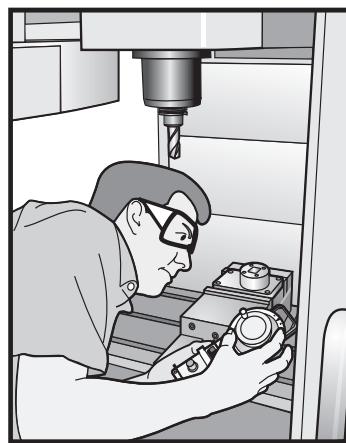
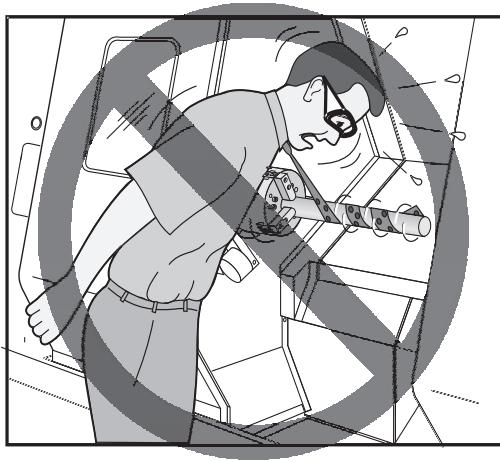
Only an authorized Haas Factory Outlet distributor should service or repair a Haas machine that is protected by the original factory warranty. Servicing by any other party automatically voids the factory warranty.



Haas Technical Publications

HAAS SAFETY PROCEDURES

THINK SAFETY!



DON'T GET CAUGHT UP IN YOUR WORK

All milling and 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.



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.
- ◆ Use appropriate eye and ear protection while operating the machine. ANSI approved impact safety goggles and OSHA approved ear protection are recommended to reduce the risks of sight damage and hearing loss.
- ◆ Do not operate the machine unless the doors are closed and the door interlocks are functioning properly. Rotating cutting tools can cause severe injury. When a program is running, the mill table and spindle head can move rapidly at any time in any direction.
- ◆ The Emergency Stop button 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 your 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.
- ◆ **This machine can cause bodily injury.**
- ◆ **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.**
- ◆ **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 machine at high feeds/feed may be ejected and puncture the safety door. Machining oversized or marginally clamped parts is not safe.**
- ◆ **Windows must be replaced if damaged or severely scratched - Replace damaged windows immediately.**
- ◆ **The spindle head can drop without notice. Personnel must avoid the area directly under the spindle head.**
- ◆ **Do not reset a circuit breaker until the reason for the fault is investigated. Only Haas-trained service personnel should troubleshoot and repair the equipment.**



♦ **Follow these guidelines while performing jobs on the machine:**

Normal operation - Keep the door closed and guards in place, while machine is operating.

Part loading and unloading – An operator opens the door or guard, completes task, closes door or guard before pressing cycle start (starting automatic motion).

Tool loading or unloading – A machinist enters the machining area to load or unload tools. Exit the area completely before automatic movement is commanded (for example, next tool, ATC/Turret FWD/REV).

Machining job set-up – Press emergency stop before adding or removing machine fixtures.

Maintenance / Machine Cleaner– Press emergency stop or power off the machine before entering enclosure.

Do not enter the machining area anytime the machine is in motion; severe injury or death may result.

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.

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 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.

NEVER OPERATE THIS MACHINE WITH THE DOORS OPEN





MILL WARNING DECALS

DANGER



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



Automatic 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 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.



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 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.

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.

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.

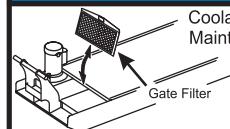


Moving parts can crush.
The tool changer will move in and crush your hand.
Never place your hand on the spindle and press ATC FWD, ATC REV, NEXT TOOL, or cause a tool change cycle.

- 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.

©2009 Haas Automation, Inc.
25-0769 Rev E

NOTICE



Coolant Tank Maintenance

Clean the filter screen weekly.

Remove the coolant tank cover and clean out any sediment inside the tank weekly.

Do not use plain water, permanent corrosion damage will result. Rust inhibiting coolant is required.

Do not use toxic or flammable liquids as a coolant.



LATHE WARNING DECALS

DANGER



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.

WARNING



NOTICE



Clean the filter screen weekly.

Remove the coolant tank cover and clean out any sediment inside the tank weekly.

Do not use plain water, permanent corrosion damage will result. Rust inhibiting coolant is required.

Do not use toxic or flammable liquids as a coolant.

29-0765 Rev F
© 2009 Haas Automation, Inc.



Haas Technical Publications

OTHER SAFETY DECALS

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

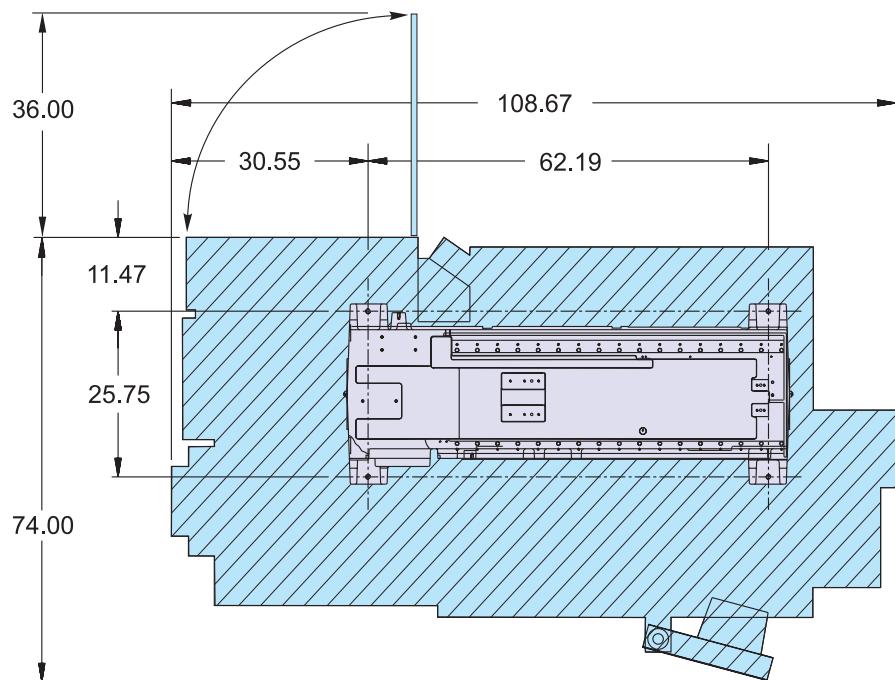




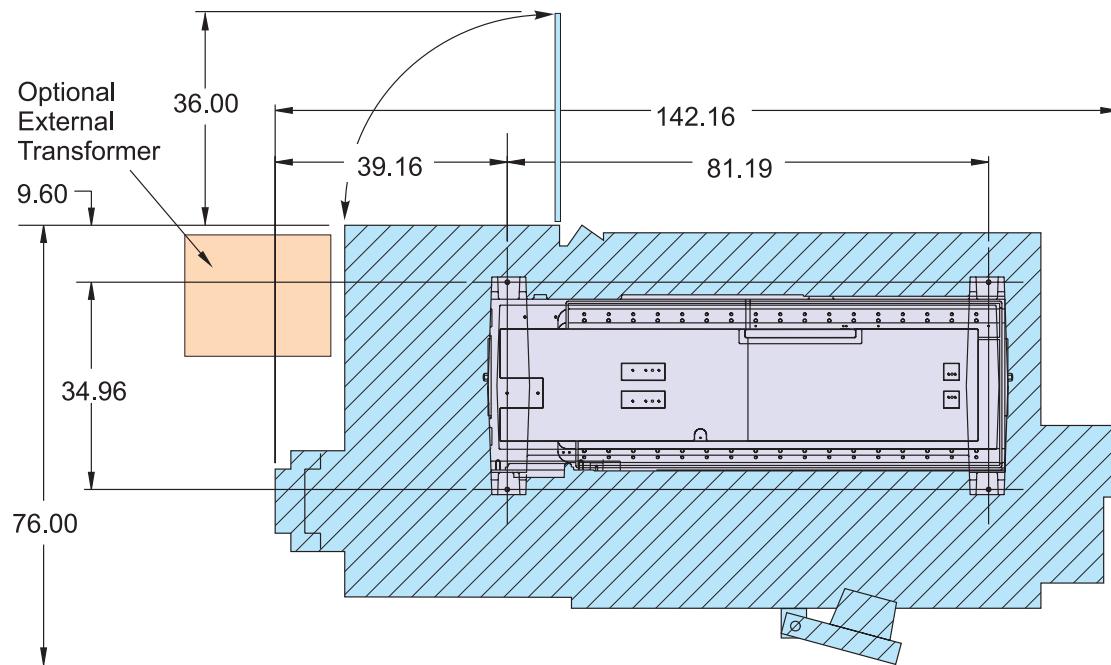
1. INSTALLATION FOR SL-SERIES

Operating Dimensions	SL-20	SL-30	SL-40
Machine Height (in.)	73	76	89
Machine Width (in.)	96	123	162
with coolant tank	105	132	169
with auger chute	120	147	183
with chip conveyor	137	165	202
Machine Depth (in.)	70	77	97
Machine Weight (lb.)	9,000	14,000	23,000

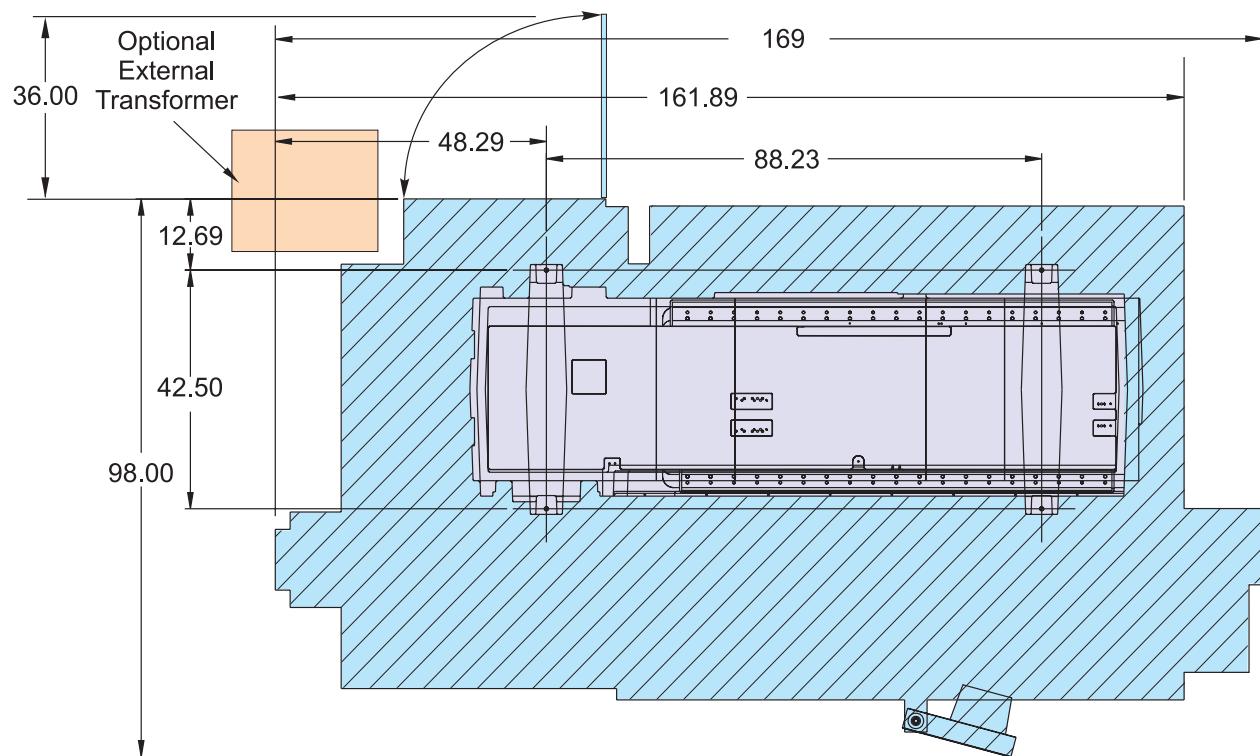
1.1 MACHINE FOOTPRINTS



SL-20



SL-30



SL-40



1.2 SERVICE REQUIREMENTS

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: less than 90% 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 +/-5%

20 HP System	Voltage Requirements	High Voltage Requirements
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
------------------------	-----------------------------	----------------------------------

TL-15BB, SL-20BB, SL-30, SL-30BB, 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

WARNING!

A separate earth ground wire of the same conductor size as the input power is required to be connected to the chassis of the machine. This ground wire is required for operator safety and for proper operation. This ground must be supplied from the main plant ground at the service entrance, and should be routed in the same conduit as the input power to the machine. A local cold water pipe, or ground rod adjacent to the machine cannot be used for this purpose.

Input power to the machine must be grounded. For wye power, the neutral must be grounded. For delta power, a central leg ground or one leg ground should be used. The machine will not function properly on ungrounded power. (This is not a factor with the External 480V Option)

The maximum voltage leg-to-leg or leg-to-ground should not exceed 260 volts or 504 volts for high voltage machines with the internal 400V option.



The high voltage requirements shown reflect the Internal 400V option which is available only in Europe. Domestic and all other users must use the External 480V option.

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.

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.

Machine Type	Main Air Regulator	Input Airline Hose Size
SL-Series	85 psi	3/8" I.D.

The recommended method of attaching the air hose is to the barb fitting at the back of the machine with a hose clamp. If a quick coupler is desired, use at least a 3/8".

WARNING

AN INADEQUATE AIR SUPPLY
WILL CAUSE TOOL CHANGER FAULTS

FOLLOW THESE GUIDELINES:

MINIMUM AIR SUPPLY PRESSURE TO MACHINE IS 100 PSI.
OBSERVE GAGE DURING TOOL CHANGE - 10 PSI MAX. DROP.
USING THE AIR GUN DURING TOOL CHANGES MAY CAUSE
FAULTS IF THE AIR SUPPLY TO THE MACHINE IS MARGINAL.
ALLOW 2 HP OF AIR COMPRESSOR PER MACHINE,
(I.E., 5 MACHINES REQUIRE A 10 HP AIR COMPRESSOR).
USE MINIMUM 3/8 ID HOSE FOR 40 TAPER MACHINES,
MINIMUM 1/2 ID HOSE FOR 50 TAPER & HS MACHINES.
AVOID QUICK DISCONNECTS IN SUPPLY LINES - THEY ARE
RESTRICTIVE.

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: The nipple between the air filter/regulator and the Bijur oil lubricator (See illustration in "Air Connection" section) reservoir tank below the control box on the back of the machine is for the optional rotary table. DO NOT use this as a connection for an auxiliary air line. Auxiliary connections should be made on the left side of the air filter/regulator.



1.3 MOVING THE CRATE

TOOLS REQUIRED

Precision bubble level (0.0005 inch per 10")
 1 1/8" hex wrench or ratchet
 1 1/2" wrench

Test indicator (0.0005)
 3/4" wrench
 Claw hammer

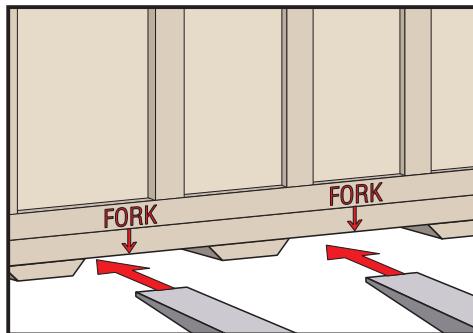
A forklift capable of lifting more than 9,000 pounds (14,000 pounds for the SL-30, 23,000 pounds for SL-40), with forks at least 5' long by 6" wide (6' by 6" wide for SL-30 and 8' by 8" wide for SL-40).

MATERIALS REQUIRED

Wire and air hose or piping as specified in the Service Requirements section
 A small amount of grease
 Way lube for the lubricator (Vactra #2)

WARNING!

THE LATHE CRATE CAN ONLY BE MOVED WITH A FORKLIFT.



CAUTION! The fork positions are marked on the crate. (Also, note that there are three skids at each side of the pallet. The heavy part of the machine [the back] is positioned over the two skids that are closest together.) If the fork positions are ignored, there is a good chance that the retaining bolts will be sheared off by the forks and also that the machine will tip over when it is picked up.

**1.4 UNPACKING THE LATHE****UNCRATING**

NOTE: Unless you are certain that you will not be shipping the machine, the crate and packing materials should be stored for reuse. Be careful not to damage the crate and the other packing materials.

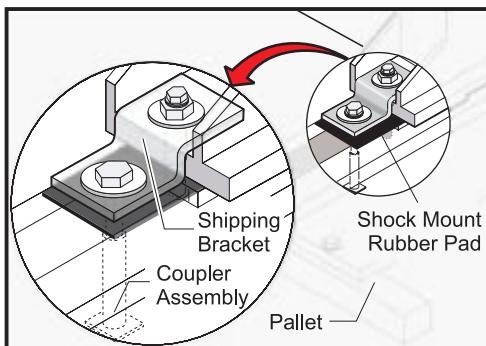
1. Pry off the clips around the top of the crate with a claw hammer and remove the top panel.
2. Pry off all but one clip at each corner of the crate.
3. Remove the plastic cover.

CAUTION! Do not put undue pressure on the top of the machine as you remove the plastic.

4. Pry off the last clip at each corner and remove the side panels.

CAUTION! The side panels are heavy — be careful that they do not drop on your feet or tip over on you.

5. Lift out the coolant tank. Remove the cleats that held them in place.



6. Remove the $\frac{3}{4}$ " bolts holding the base to the pallet and the plastic thread protecting sleeve from the base.



7. Remove the nuts, on the leveling screws, holding the shipping bracket to the base casting. Remove the shipping brackets.
8. Lift the machine off the pallet.



SETTING IN PLACE

CAUTION!

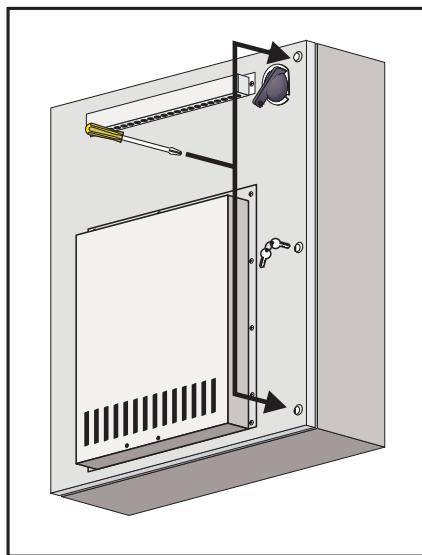
Do not lift the machine any farther than necessary off the floor when moving it, and move as slowly and cautiously as possible. Dropping the machine, even from a height of a few inches, can cause injury, result in expensive repairs, and void the warranty.

1. Unbolt clamp plates. Lift the machine until the bolts clear the pallet.
2. Thread the leveling screws through the casting until thread is approximately 3/8" above the top of the casting. If a screw is excessively hard to turn, remove it, dress the threads in the hole with a 1-14 tap, and inspect the screw. If the screw has dings, dress the threads with a 60° V file. Install the lock nuts on the leveling screws, but do not tighten. **SL-20 machines require the lock nuts to be installed under the leveling foot.**
3. Move the machine to where it will be located. Take leveling pads out of the tote kit, grease the dimple in each pad, and locate them under the leveling screws at the four corners. Lower the machine.
4. Remove all banding and packing material around the control panel, monitor and doors.
5. Remove foot switches from inside machine and attach cable to socket located at left end of front support beam, with cable facing downward.

**1.5 INITIAL SETUP****WARNING!**

At this point, there should be NO electrical connection to the machine. The electrical panel should be closed and the three latches on the door 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 switch 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, you must exercise extreme caution when you are working in the panel.

1. Set the main switch at the upper right of the electrical panel on the back of the machine to OFF.

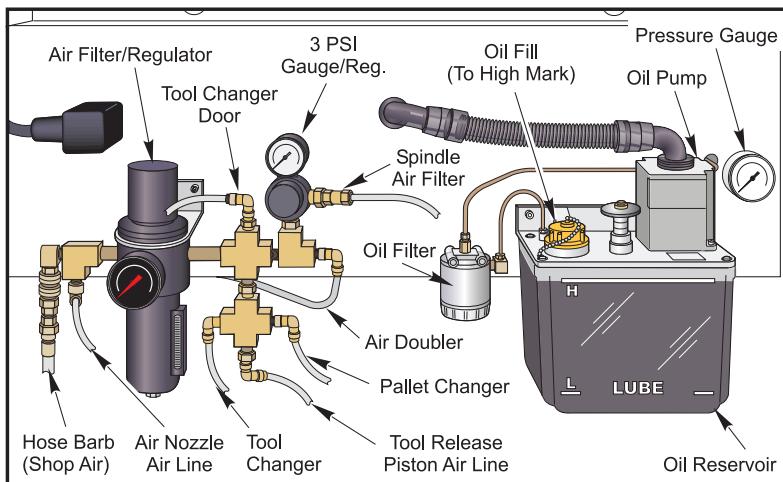


2. Using a screwdriver to unlock the two latches on the panel door, unlock the cabinet with the key, and open the door.
3. Take sufficient time to check all the components and connectors associated with the circuit boards. With the power off, push on them gently to make sure that they are seated in their sockets. Look for any cables that have become disconnected, look for any signs of damage and loose parts in the bottom of the panel box. If there are any signs that the machine was mishandled, be extremely careful in powering up the machine (be ready to shut it off IMMEDIATELY). Or if there are obvious problems, call the factory BEFORE proceeding.


AIR CONNECTION
CAUTION!

Working with the air service required for LATHE can be hazardous. Make sure that pressure has been removed from the air line before you connect it to the machine, disconnect it from the machine, or service parts of the air system on the machine.

- When the machine leaves the factory, the air filter is empty, and air lubricator and the lubricator reservoir tank are full. However, they should be checked and serviced if required before compressed air is supplied to the machine.



- With the pressure off in the air line, connect the air supply to the hose barb next to the air filter/regulator (below the electrical panel). If the fitting supplied is not compatible, simply replace it.
- Start the compressor, set it between 100 and 150 PSI. Set the regulator on the machine to 85 to 90 PSI.
- Prime the lubricator to make sure it is working. To prime the lubrication system, pull up on the handle on top of the reservoir tank.

CAUTION!

NEVER push down on the primer handle! It gradually returns to the down position by itself, and the corresponding pressure increase can be seen on the pressure gauge.

NOTE: Depending on the position of the cam that drives it, the lubrication system may not activate until a few minutes after the machine is started. However, if there is a problem with the system, an alarm will stop the machine.

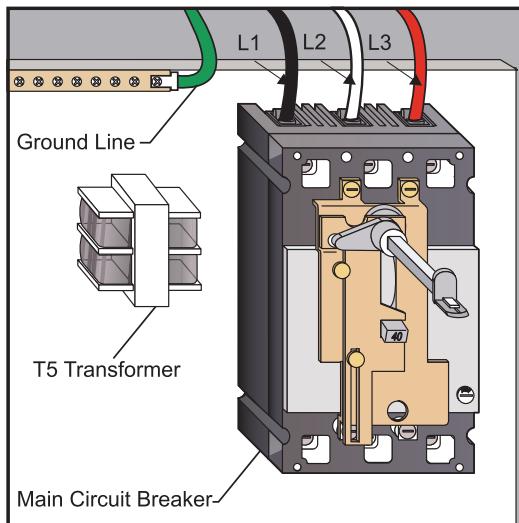
**ELECTRICAL CONNECTIONS**

NOTE: The machine must have air pressure at the air gauge, or a "Low Air Pressure" alarm will be present on power up.

CAUTION! Working with the electrical services required for the lathe can be extremely hazardous. The electrical power must be off and steps must be taken to ensure that it will not be turned on while you are working with it. In most cases this means turning off a circuit breaker in a panel and then locking the panel door. However, if your connection is different or you are not sure how to do this, check with the appropriate personnel in your organization or otherwise obtain the necessary help BEFORE you continue.

WARNING!

The electrical panel should be closed and the three latches on the door 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.



1. Hook up the three power lines to the terminals on top of the main switch at upper right of electrical panel and the separate ground line to the ground bus to the left of the terminals.

NOTE: Make sure that the service wires actually go into the terminal-block clamps. (It is easy to miss the clamp and tighten the screw. The connection looks fine but the machine runs intermittently or has other problems, such as servo overloads.) To check, simply pull on the wires after the screws are tightened.

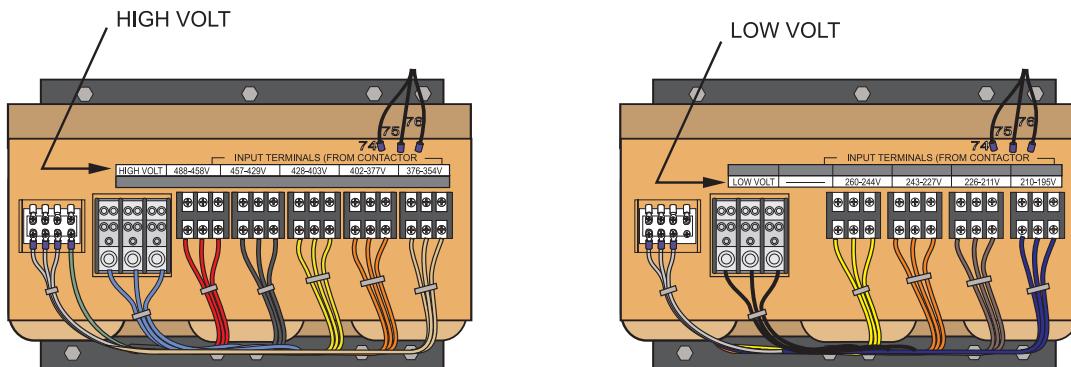


2. After the line voltage is connected to the machine, make sure that main circuit breaker (at top-right of rear cabinet) is OFF (rotate the shaft that connects to the breaker counterclockwise until it snaps OFF). Turn ON the power at the source. Using an accurate digital voltmeter and appropriate safety procedures, measure the voltage between all three pair phases at the main circuit breaker and write down the readings. The voltage must be between 195 and 260 volts (360 and 480 volts for high voltage option).

NOTE: Wide voltage fluctuations are common in many industrial areas; you need to know the minimum and maximum voltage which will be supplied to the machine while it is in operation. U.S. National Electrical Code specifies that machines should operate with a variation of +5% to -5% around an average supply voltage. If problems with the line voltage occur, or low line voltage is suspected, an external transformer may be required. If you suspect voltage problems, the voltage should be checked every hour or two during a typical day to make sure that it does not fluctuate more than +5% or -5% from an average.

CAUTION! Make sure that the main breaker is set to OFF and the power is off at your supply panel BEFORE you change the transformer connections. Make sure that all three black wires are moved to the correct terminal block and that they are tight.

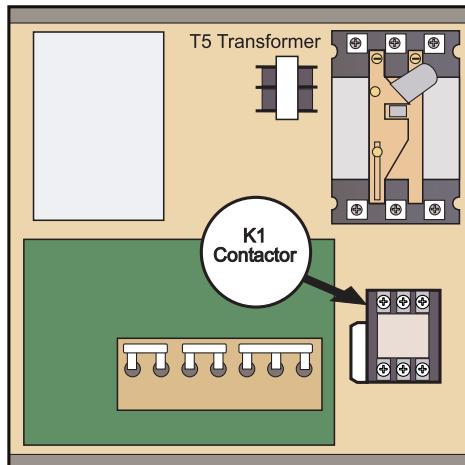
3. Check the connections on the transformer at the bottom-right corner of the rear cabinet. The three black wires labeled **74**, **75**, and **76** must be moved to the terminal block triple which corresponds to the average voltage measured in **Step 2** above. There are four positions for the input power for the 260 volt transformer and five positions for the 480 volt transformer. The labels showing the input voltage range for each terminal position is shown in the following illustration:



4. Transformer T5 supplies 24VAC used to power the main contactor. There are two versions of this transformer for use on 240 and 400V machines (32-0964B and 32-0965B, respectively). The 240V transformer has two input connectors located about two inches from the transformer, which allow it to be connected to either 240V or 200V. Users that have 220V-240V RMS input power should use the connector labeled 200V. Users with the External High Voltage Option should use the 240V connector if they have 420V-510V 60Hz power or the 200V connector if they have 50Hz power. Failure to use the correct input connector will result in either overheating of the main contactor or failure to reliably engage the main contactor.

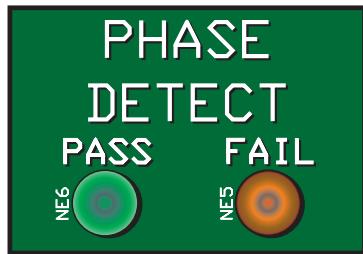


5. Set the main switch to ON (rotate the shaft that engages the handle on the panel door clockwise until it snaps into the ON position). Check for evidence of problems, such as the smell of overheating components or smoke. If such problems are indicated, set the main switch to OFF immediately and call the factory before proceeding.

**WARNING!**

High Pressure Coolant (HPC) pump is a three phase pump and must be phased correctly! Improper phasing will cause damage to the HPC pump and void the warranty. Refer to the HPC start up section IF YOUR MACHINE IS EQUIPPED WITH HPC.

6. After the power is on, measure the voltage across the upper terminals on the contactor K1 (located below the main circuit breaker). It should be the same as the measurements where the input power connects to the main breaker. If there are any problems, check the wiring.
7. Apply power to the control by pressing the Power-On switch on the front panel. Check the high voltage buss on the Vector Drive (pin 2 with respect to pin 3 on the terminal bus at the bottom of the drive). It must be between 310 and 360 volts. If the voltage is outside these limits, turn off the power and recheck steps 2 and 3. If the voltage is still outside these limits, call the factory. Next, check the DC voltage displayed in the second page of the Diagnostic data on the CRT. It is labeled DC BUS. Verify that the displayed voltage matches the voltage measured at pins 2 and 3 of the Vector Drive +/- 7 VDC.
8. Electrical power must be phased properly to avoid damage to your equipment. The Power Supply Assembly PC board incorporates a "Phase Detect" circuit with neon indicators, shown below. When the orange neon is lit (NE5), the phasing is incorrect. If the green neon is lit (NE6), the phasing is correct. If both neon indicators are lit, then you have a loose wire. Adjust phasing by swapping L1 and L2 of the incoming power lines at the main circuit breaker.


WARNING!

ALL POWER MUST BE TURNED OFF AT THE SOURCE PRIOR TO ADJUSTING PHASING.

9. Turn off the power (rotate the shaft that engages the handle on the panel door counterclockwise until it snaps into the OFF position). Also, set the main switch handle on the panel door to OFF. (Both the handle and the switch must be set to OFF before the door can be closed). Close the door, lock the latches, and turn the power back on.
10. Remove the key from the control cabinet and give it to the shop manager.

INSTALLATION PROCEDURE FOR EXTERNAL 480V TRANSFORMER

Introduction

The external transformer adds to overall machine reliability and performance, however it does require extra wiring and a place to locate it. The external transformer provides electrostatically shielded isolation. This type of transformer acts to isolate all common mode line transients and improve EMI conducted emissions.

The external transformer has a 45 KVA rating.

Installation

The transformer should be located as close to the machine as possible. The input and output wiring of the transformer should conform to the local electrical codes and should be performed by a licensed electrician. The following is for guidance only, and should not be construed to alter the requirements of local regulations.

The input wire should not be smaller than the 6AWG for the 45KVA transformer. Cable runs longer than 100" will require at least one size larger wire. The output wire size should be 4 AWG.

The transformer is 480V to 240V isolation transformers with delta wound primary and secondary windings. The primary windings offer 7 tap positions, 2 above and 4 below the nominal input voltage of 480V.



For domestic installations and all others using 60Hz power, the primary side should be wired as follows:

Input Voltage Range	Tap
493-510	1 (504)
481-492	2 (492)
469-480	3 (480)
457-468	4 (468)
445-456	5 (456)
433-444	6 (444)
420-432	7 (432)

This should produce a voltage on the secondary side of 234-243 V RMS L-L. Verify this and readjust the taps as required. At the machine, connect the cables at the input of the internal 230V transformer to the 227-243V taps. Apply power to the machine and verify that the DC voltage between pins 2 and 3 of the Vector Drive (2nd and 3rd pins from the left) is 329-345VDC. If not, return to the 480V isolation transformer and readjust the taps as required. Do not use the taps on the internal 230V transformer to adjust the voltage.

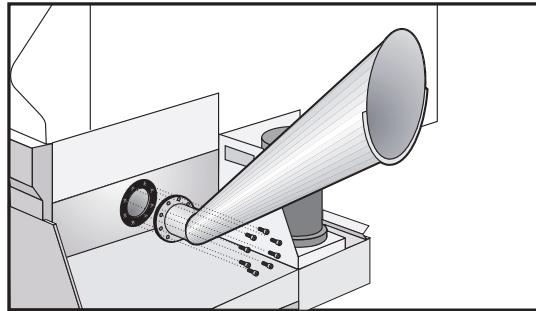
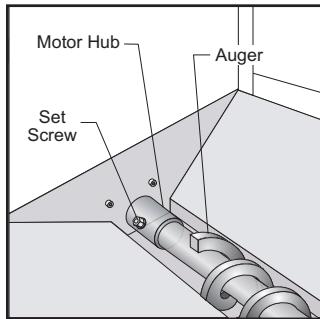
50Hz Installations

The external transformers are 60Hz rated, and cannot be used at 50Hz without derating the input voltage. For these applications, the internal 230V transformer should be tapped on the lowest setting (195-210V RMS). The external transformer should be tapped according to the table shown below. If these tap setting do not produce a DC bus voltage between pins 2 and 3 on the Vector Drive between 320 and 345VDC, readjust the taps on the external transformer as required. DO NOT move the taps on the internal transformer from the lowest position.

Input Voltage Range	Tap
423-440	1 (504)
412-422	2 (492)
401-411	3 (480)
391-400	4 (468)
381-390	5 (456)
371-380	6 (444)
355-370	7 (432)


OPTIONAL CHIP AUGER INSTALLATION

1. Unpack the auger and discharge tube.
2. Slide the auger into the discharge tube opening and then slip opposite end onto motor hub. Fasten to motor hub with the 5/16-18 x 2½" bolt.



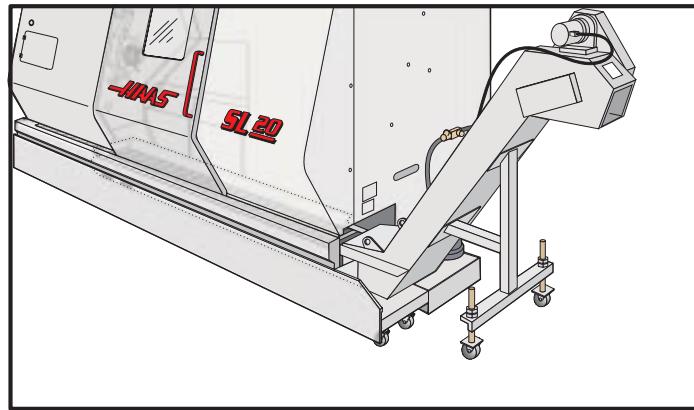
3. Install gasket and slide the discharge tube into the opening. Attach the discharge tube with bolts and locking washers and tighten uniformly.
4. After machine start-up, check the operation of the auger to ensure the direction of rotation will move the chips toward the discharge tube. If the auger is turning so that the chips are not being moved toward the discharge tube, change PARAM 209 bit 12 from 1 to 0 or 0 to 1 to establish a new forward direction.

MAINTENANCE

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 and pan coolant drain (under the pan). 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.

NOTE: Auger and discharge tube are subject to wear. Abrasive swarf, hard steel chips and continuous use will accelerate this wear.

NOTE: On a machine with a safety circuit, the chip auger will only run with the door closed regardless of the Conveyor Door Override bit.

**OPTIONAL CHIP CONVEYOR**

1. Unpack the chip conveyor and locate the conveyor discharge cover.
2. Remove the side and nose wings from the conveyor pickup area.
3. Use the hoist loops at the incline start point to raise the conveyor high enough to remove the caster wheels and reinstall them in the operating position.
4. Slide the nose of the conveyor into the opening on the right side of the machine until the incline start point is near the machine enclosure.
5. With the nose resting in the enclosure adjust the caster wheels to support the conveyor $1/8"$ to $1/4"$ above the lip of the enclosure pan.
6. Install the side, nose wings and discharge cover.
7. Plug the conveyor motor in to the side of the control panel and check for operation.

NOTE: On a machine with a safety circuit, the chip conveyor will only run with the door closed regardless of the Conveyor Door Override bit.



1.6 COOLANT SYSTEM

COOLANT TANK

1. Position the coolant tank under the front of the machine.
2. Connect the coolant pump and the auger power lines to the connectors located on the side of the control cabinet.
3. Attach the coolant hose to the pump fitting located at the base of the coolant pump
4. Fill the tank with the approximately 35 gallons of coolant (50 gallons for SL-30, 75 gallons for the SL-40). Fill with water based coolant only.*

***Mineral cutting oils will damage rubber based components throughout the machine.**

COOLANT TANK MAINTENANCE

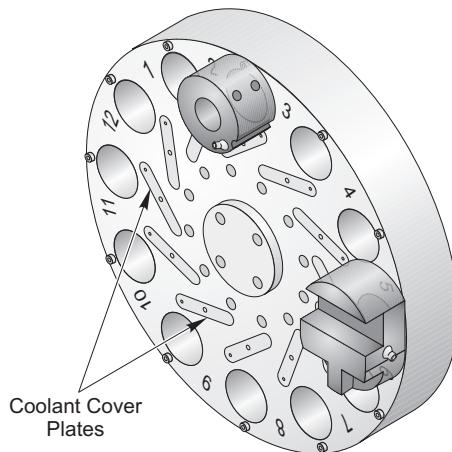
Clean the chips out that collect in the holes of the auger trough.

Clean the chips out of the baffles of the coolant tank. The baffles are accessible by lifting the lid of the coolant tank.

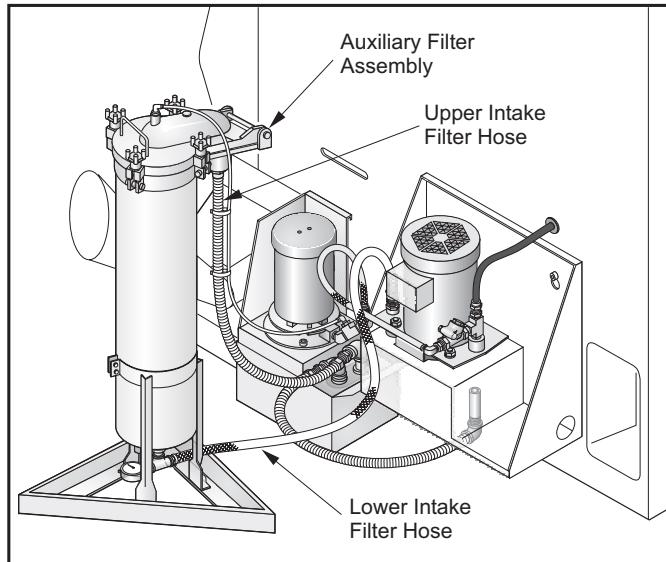
COOLANT FLOW

The volume of coolant flow may be controlled manually by using the valve handle to the right of the turret. This handle can be turned to shut the coolant flow off completely, or to select any volume of flow desired.

1. There are 10 or 12 coolant plate covers on the turret; one for each tool holder. To allow coolant to flow through a tool holder, the coolant plate cover must be in place. Otherwise, the plate covers must be removed.



12 Pocket Turret

**AUXILIARY FILTER FOR STANDARD COOLANT SYSTEMS**

1. Place the Auxiliary Filter system next to the coolant tank of the machine.
2. Connect the output of the Standard Coolant pump to the input of the Auxiliary Filter.
3. Connect the Auxiliary Filter output hose to the coolant hose of the machine.
4. The Auxiliary Filter tank must be filled with coolant before use.
5. To fill the Auxiliary Filter tank from the Standard Coolant tank, perform the following steps:
 - Turn on the Standard Coolant Pump.
 - Open the ball valve, located on the top of the Auxiliary Filter tank.
 - Wait for coolant to appear in the drain-back hose.
 - Close the ball valve; the Auxiliary Filter tank is full.

AUXILIARY FILTER REPLACEMENT (STANDARD COOLANT)

The condition of the filter element should be inspected regularly to ensure proper operation. With a clean filter, the two pressure gauges will read equally. A pressure difference of 10 psi indicates the filter is dirty and needs to be replaced. The pressure difference between the two gauges should not exceed 15 psi. Pressure should be checked with the coolant pump running and the coolant ball valves open.

NOTE: The bottom pressure gauge will drop in pressure as the filter becomes dirty.

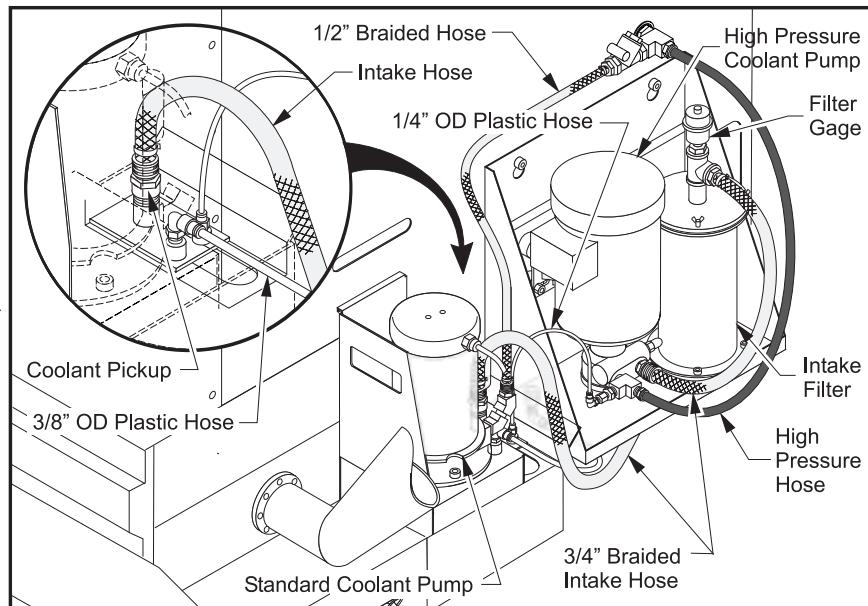
HAAS recommends using 25-micron rated filter bags; one is provided with the unit. Replacement bags can be purchased from your local filter supplier or from HAAS (P/N 93-9130). Finer micron ratings can be used if desired.


OPTIONAL HIGH PRESSURE COOLANT SYSTEM
APPLICATION

The High Pressure Coolant System for Haas CNC lathes is a dual coolant system, having both standard and high pressure coolant pumps. Both pumps are controlled by M codes. The standard pump delivers coolant to the tool at low pressure. The high pressure coolant pump delivers coolant to the tool at up to 300 psi, with a maximum volume of 5 gallons per minute.

INSTALLATION

1. Connect the intake filter hose to the coolant pickup connection next to the coolant pump on the coolant tank.
2. Route the 1/4" OD plastic hose attached to the high pressure coolant pump down into the coolant tank. Insert it in to the 1/4" OD connector next to the coolant pickup.
3. Route the 3/8" OD plastic hose from the bottom of the HPC unit to the 3/8" OD push-in elbow next to the coolant pump.
4. Attach the 1/2" braided hose to the standard coolant pump.
5. Prime the high pressure coolant system.
6. Run the standard coolant pump and check all connections for leaks.


Installed High Pressure Coolant System

**OPTIONAL AUXILIARY FILTER FOR HIGH PRESSURE COOLANT****Installation:**

1. Attach the hose from the top of the auxiliary filter to the hose connector on the coolant pickup.

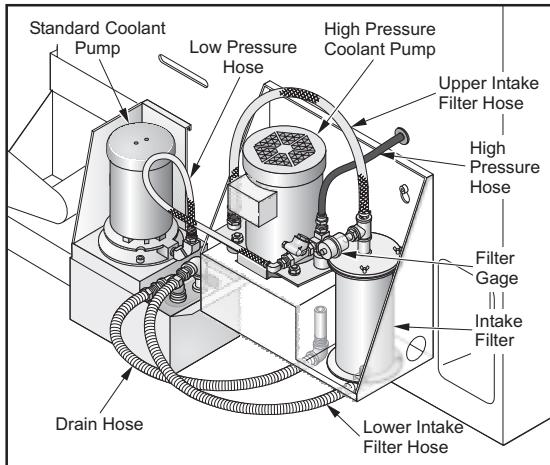


Figure 1.0 Standard Filtration Setup

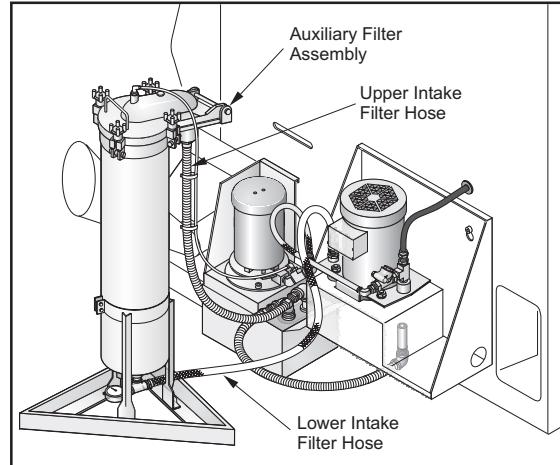


Figure 2.0 Optional Auxiliary Filter setup

2. Install the 1/2" street-tee from the kit between the two 45° elbows on the primary coolant pump. Install the pipe reducer from the kit into the side outlet of the street-tee. Install the 1/4" push-in elbow into the pipe reducer. (See figure 3.0).

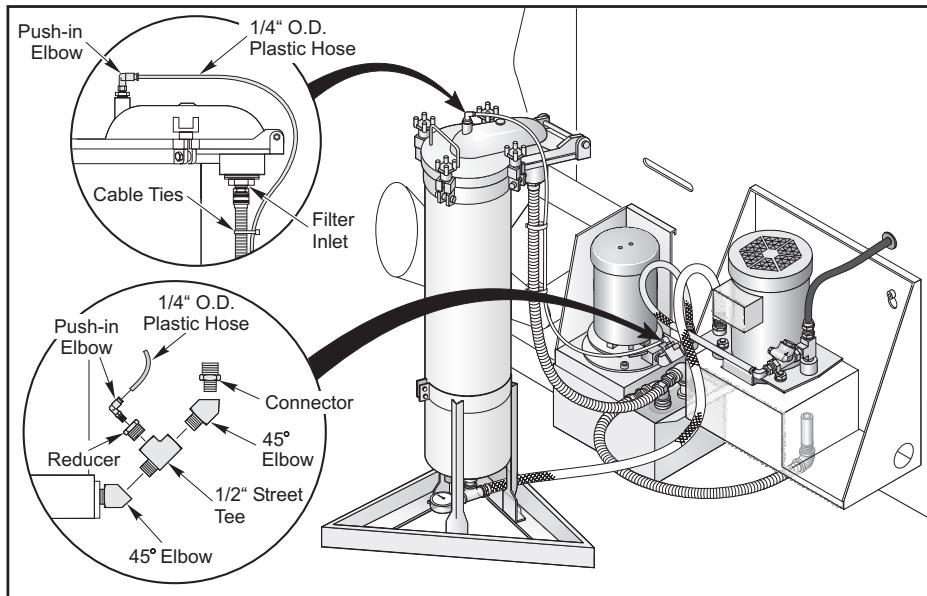


Figure 3.0 Auxiliary Filter Priming System



3. Insert the 1/4" OD plastic hose into the push-in elbow on the pump. Route the hose along the intake filter hose and around the hinge of the auxiliary filter. Trim the plastic hose to length and insert it into the push-in elbow at the top of the filter. Secure the plastic hose to the inlet hose with the supplied cable ties. (See figure 3.0).
4. Attach the hose from the bottom of the auxiliary filter to the inlet of the high pressure pump.
5. Check that the filter lid is securely closed. Using a wrench handle or metal bar, tighten the two rear bar nuts first and then the front pair. Torque the bar nuts according to the manufacturer's recommendations. (approximately 30-50 ft-lbs)
6. Run the primary coolant system for four minutes to prime the bag filter housing before using the high pressure system.

**1.7 MACHINE POWER ON****WARNING!**

DO NOT press POWER UP/RESTART on the control panel while the shipping bracket is in place.

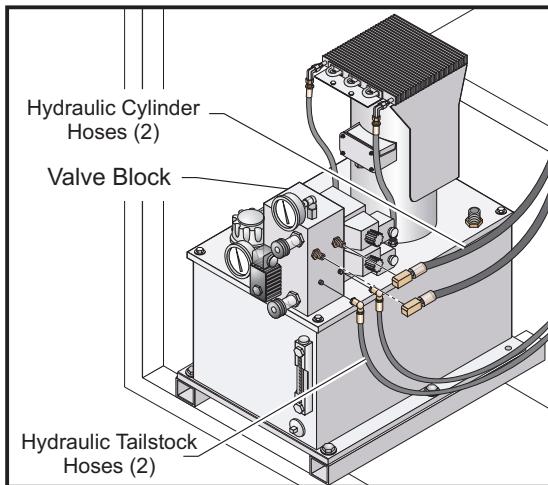
- With the main switch on the electrical panel set to ON, press and release POWER ON at the upper left of the control panel. You will hear a click in the back of the machine and the fans will energize. (If you don't hear these sounds, the machine is not getting power and, with all necessary safety precautions, you should check the connections to the electrical panel.) After a few seconds, the display will appear on the screen.

HYDRAULIC UNIT PHASING

MACHINE MUST BE PHASED PROPERLY!! Improper phasing will cause damage to the hydraulic unit and void the warranty.

- Press and release the RESET button twice, or until you have no alarms, to turn the servos on. (The message "ALARM" appears at the lower right of the screen if one or more alarms are in effect.)

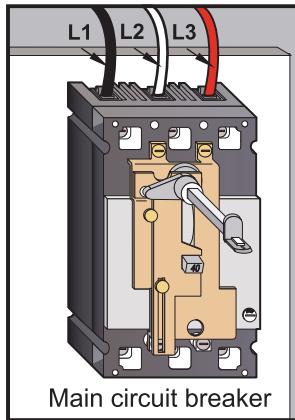
NOTE: The hydraulic pump runs whenever the servo is on.



- Check the pump pressure gauge on the hydraulic unit (see figure). If the pressure reads zero, IMMEDIATELY POWER OFF THE MACHINE.

CAUTION!If the hydraulic pump is allowed to run for more than 30 seconds in this condition, serious damage will occur.

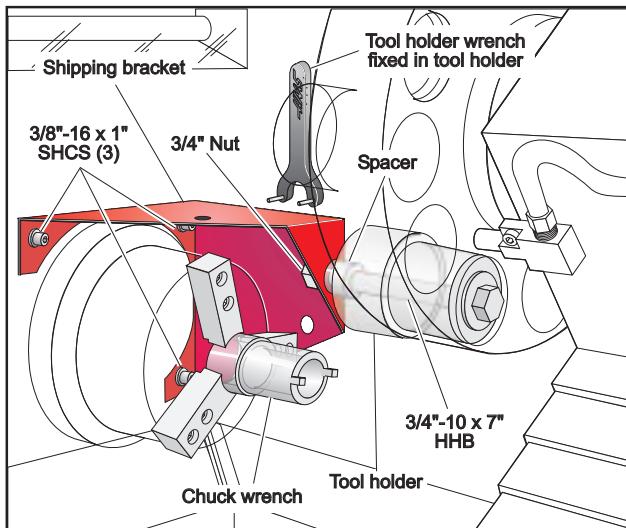
NOTE: When the pressure reads zero, it means the machine is not properly phased, and the pump is rotating backwards. If the pressure gauge shows a proper pressure, the phasing is correct, and no further action is required.



4. To properly phase the machine:
 - Make sure there is no power at the input side (top) of the main circuit breaker. MEASURE THE VOLTAGE!
 - Exchange any two wires at the input side (top) of the main circuit breaker
 - Close the control box.
 - Return to Step 1 and retest for proper phasing.

REMOVING SHIPPING BRACKET

5. With a 1 1/4" box wrench, remove the 3/4"-10" x 7" HHB, nut and spacer.
6. Press ZERO RET key, Z key, then ZERO SINGLAXIS key (the turret will move away from the spindle). **DO NOT PRESS POWER UP/RESTART !!**



7. Remove the three 3/8"-16 x 1" SHCS then remove the shipping bracket (see figure).
8. Replace the three 3/8"-16 x 1" SHCS with the three 3/8"- 16 x 1/2" BHCS that are in bag attached to the turret.



9. Unclamp the chuck, to release the chuck wrench. Loosen and release the tool holder wrench from the tool holder on the turret.
10. Press and release the SETNG / GRAPH key. Then page down to the last page (press and release PAGE DOWN several times). Cursor to Setting 53, JOG W/O ZERO RETURN (with the cursor **down** key). To turn this setting on, press and release the **right** cursor key and then press and release the WRITE key at the extreme lower right of the control panel. Turning on JOG W/O ZERO RETURN bypasses the zero return interlock.

NOTE: This setting, like many others, resets to OFF when the machine is powered up. This prevents the machine from operating until a zero return has been executed — the machine control cannot determine position until it has been set by a zero return routine. For this reason, it is important that you execute a zero return immediately each time the machine is started for normal operation BUT NOT FOR THIS START-UP ROUTINE.

11. Press and release the RESET button twice, or until you have no alarms, to turn the servos on. (The message "ALARM" appears at the lower right of the screen if one or more alarms are in effect.)

NOTE: If any alarms are present and cannot be cleared with the RESET button, press and release the ALARM / MESGS key for more information on the alarms. If you are unable to clear the alarms, write down the alarm numbers and call the factory.

12. Press and release the HANDLE JOG button and check the screen for the "JOGGING Z AXIS HANDLE .001" message. Verify that the carriage will travel SLOWLY (not more than 0.001 inch per impulse — the ".001" part of the Z-axis message). If the message does not read .001, press and release the .001 button next to the HANDLE JOG button.

NOTE: The upper numbers on the buttons next to HANDLE JOG are for jog handle use, and the lower numbers are the jog speed in inches per minute when using the JOG buttons on the keypad.

13. Once you are certain that the Z-axis is working correctly (that it operates smoothly and there are no strange noises, etc.), make sure that all alarms are clear — check for the "ALARM" message at the lower right of the screen. Next, close the doors and press and release the ZERO RETURN button followed by the AUTO ALL AXES button. The Z-axis moves to the right slowly. Then, after it has reached its home position, the X- axis moves to its home position.

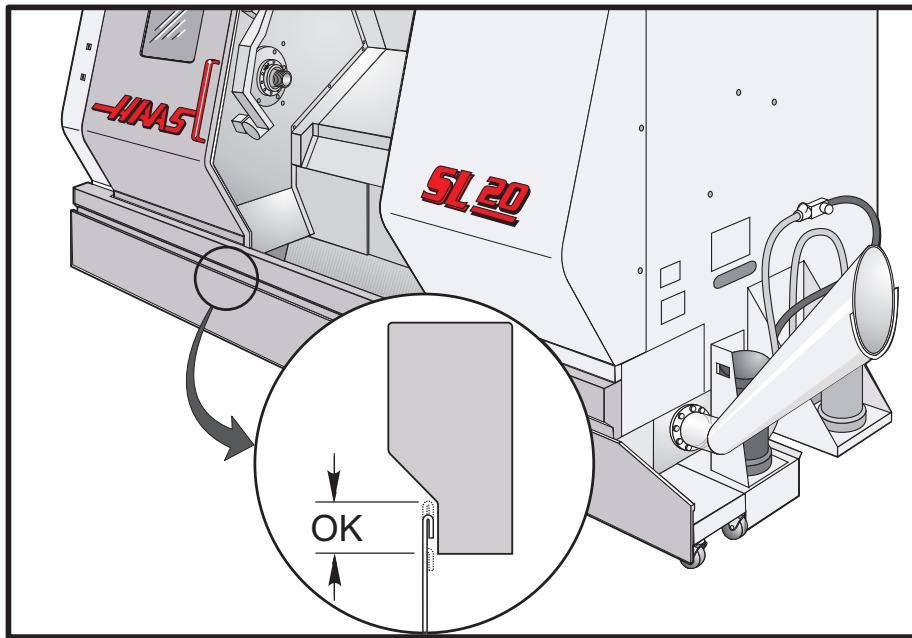
CAUTION! If you hear any strange noises, hit the EMERGENCY STOP button immediately and call the factory.

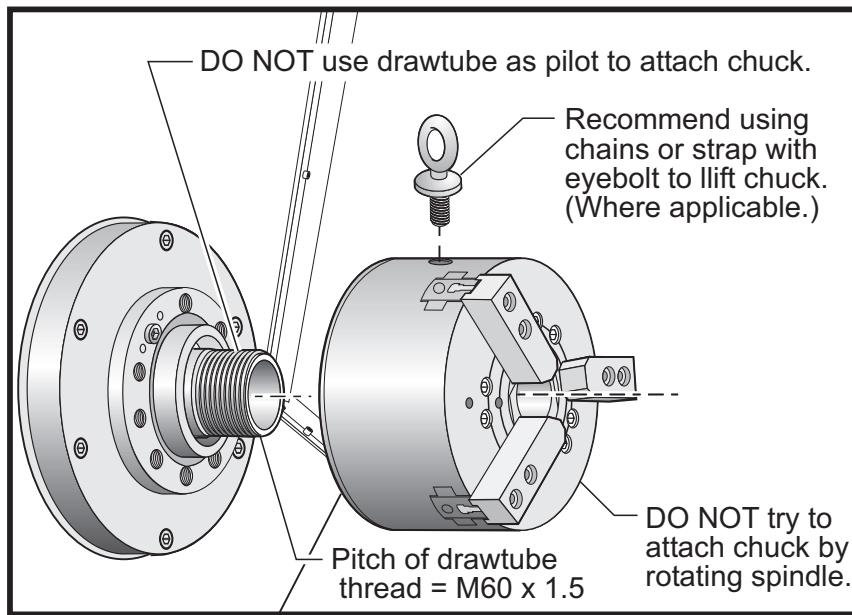
14. The machine is now ready for leveling.

**1.8 LEVELING THE LATHE**

Leveling the machine is required to provide proper coolant and lubrication drainage and to ensure equal loading on all four of the casting feet for consistent cutting performance. Please read through entire sequence before starting.

1. Position the turret close to the chuck (this is how the machine was shipped). Remove right-end rear panel to access the Z-axis linear guide rails.
2. Place a machinist's level **across** linear guides to level front-to-back. Place level **along** linear guides to level machine left-to-right. **Take care to avoid damage to linear guide rails.**
3. Level machine by rotating leveling screws. Adjust adjacent screws alternately to maintain proper loading.
4. Adjust machine height (see figure).
5. Verify that each leveling screw requires approximately the same torque to turn. This will ensure proper loading.
6. Tighten lock nuts.



**1.9 CHUCK INSTALLATION****HYDRAULIC DRAWTUBE**

The following instructions are to be used in accordance with the manufacturer's installation guide.

1. In order to mount the chuck, the drawtube must be in the forward position (unclamped position).
2. Mount the chuck to the spindle face. (Recommend using a hoist and eyebolt for SL-30,40)

NOTE: Use Starter bolts to help guide the chuck onto the drawtube. The SL-20 has a keyway on the chuck that must be aligned with the drawtube.

3. Thread the chuck onto the drawtube using the chuck wrench.

NOTE: The chuck must be fully threaded onto the drawtube. Look inside the chuck bore to see if there are any exposed drawtube threads. Retracting the chuck with exposed threads could damage drawtube.

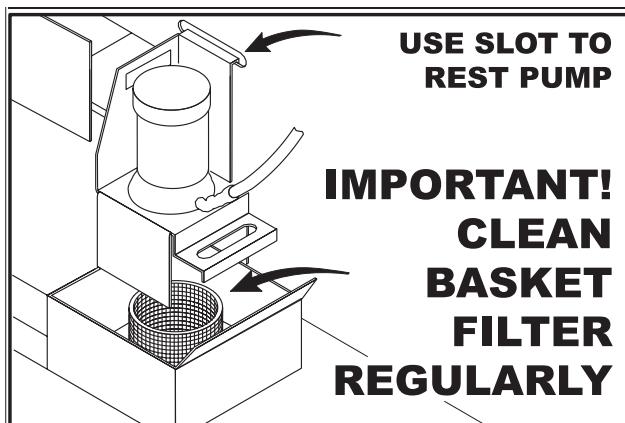
4. Tighten all chuck mounting bolts.
5. Install drawtube collar.



2. 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 and housing. Check hydraulic unit oil level (DTE-25 ONLY). Capacity-8 gallons.
Weekly	Check for proper operation of auto drain on filter regulator. Check air guage / 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! Be careful to disconnect the coolant pump from the controller and to POWER OFF the control before working on the coolant tank. Dump the oil drain bucket. Check transmission oil level (if applicable). If oil is not visible at the bottom edge of the sight gauge, remove the end panel and add DTE-25 through the top filler hole until it is visible in the sight gauge.
Six Months	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. With the air pressure OFF, disassemble and clean the small filter at end of lubricator (right side of machine). Clean oil filter and remove residue from the bottom of filter. Replace air filter on control box every (2) years. The filter box must be removed on the SL-20 lathes in order to replace the air filter.



Poor Coolant flow can be caused by a dirty filter.

To clean the filter:

- Turn off the coolant pump.
- Lift the coolant tank LID.
- Remove the filter.
- Clean and reinstall filter.

**2.1 LUBRICATION CHART**

ITEM	CAPACITY	FLUID TYPE
COOLANT	40 gallons (50 for SL-30, 75 gallons for SL-40)	Water based coolant only*
WAY LUBE	2-2.5 Qt. depending on pump style	Vactra #2
TRANSMISSION	54 oz.	Mobile DTE25

*Mineral cutting oils will damage rubber based components throughout the machine.

WARNING!

When machining castings, sand from the casting process and the abrasive properties of cast aluminum and cast iron will shorten pump life unless a special filter is used in addition to the 100 mesh suction 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. A larger coolant tank is recommended.

Shortened pump life, reduction of pressure and increased maintenance are normal and to be expected in abrasive environments and is not covered by warranty.

Lubrication Requirements:

Each jaw requires two strokes of grease:

- Every 1000 clamp / unclamp cycles
- or at least once a week

Use provided grease gun for chuck lubrication

Lubrication type: Molybdenum Disulfide Grease (20% to 25% moly content)



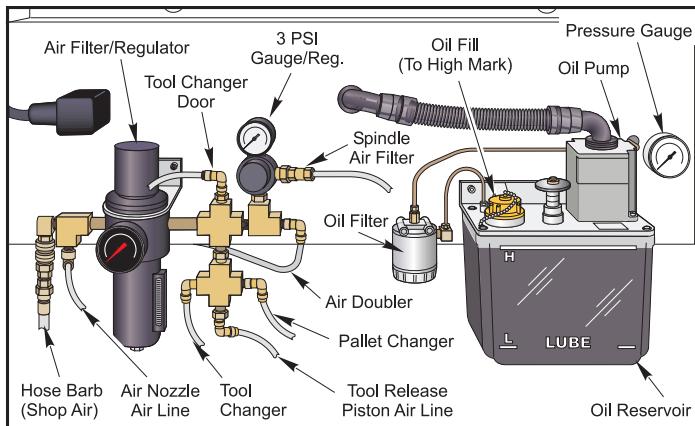
2.2 CHUCK MAINTENANCE

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.
- Caution: Lack of grease significantly reduces clamping force and can result in chatter, improper clamping, or thrown parts.
- 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

2.3 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.

To lubricate the system, pull up on the primer pull-tab located next to the fill port. The primer will automatically send 3cc of lube through the system.

**2.4 CHIP AUGER****MAINTENANCE**

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.

**This section contains the following:**

- **Introduction / basic machine overview**
- **Control panel overview**
- **Set up and operation**

3. OPERATION**3.1 BASIC INTRODUCTION**

This section provides the basic programming and operation principles necessary to begin operating the machine.

In an "NC" (Numerically Controlled) machine, the tool is controlled by a code system that enables it to be operated with minimal supervision and with a great deal of repeatability. "CNC" (Computerized Numerical Control) is the same type of operating system, with the exception that the machine tool is monitored by a computer.

The same principles used in operating a manual machine are used in programming an NC or CNC machine. The main difference is that instead of cranking handles to position a slide to a certain point, the dimension is stored in the memory of the machine control once. The control will then move the machine to these positions each time the program is run.

The operation of the SL-Series Lathe requires that a part program be designed, written, and entered into the memory of the control. The most common way of writing part programs is off-line, that is, away from the CNC in a facility that can save the program and send it to the CNC control. The most common way of sending a part program to the CNC is via an RS-232 interface. The HAAS Lathe has an RS-232 interface that is compatible with most existing computers and CNC's.

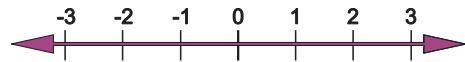
In order to operate and program a CNC controlled machine, a basic understanding of machining practices and a working knowledge of math is necessary. It is also important to become familiar with the control console and the placement of the keys, switches, displays, etc., that are pertinent to the operation of the machine.

This manual can be used as both an operator's manual and as a programmer's manual. It is intended to give a basic understanding of CNC programming and its applications. It is not intended as an in-depth study of all ranges of machine use, but as an overview of common and potential situations facing CNC programmers. Much more training and information is necessary before attempting to program on the machine.

The programming section of this manual is meant as a supplementary teaching aid to users of the HAAS Lathes. The information in this section may apply in whole or in part to the operation of other CNC machines. Its use is intended only as an aid in the operation of the HAAS Lathe.

**3.2 THE COORDINATE SYSTEM**

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

*Horizontal number line.*

V
e
r
t
i
c
a
l
n
u
m
b
e
r
l
i
n
e

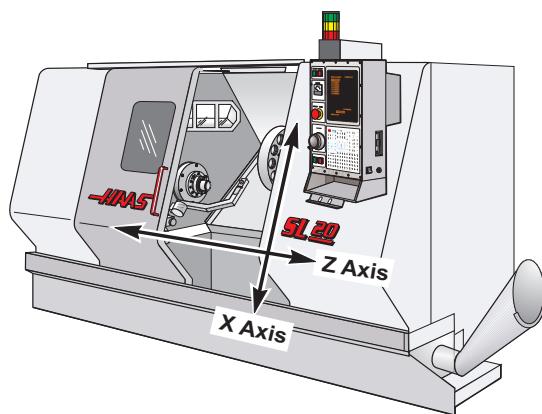
The number line also has numbered increments on either side of absolute zero. Moving away from zero to the right are positive increments. Moving away from zero to the left are negative increments. The "+" or positive increments, are understood, therefore no sign is needed.

We use positive and negative along with the increment's value to indicate its relationship to zero on the line. In the case of the previous line, if we choose to move to the third increment on the minus

(-) side of zero, we would call for -3. If we choose the second increment in the plus range, we would call for 2. Our concern is with distance and direction from zero.

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

The illustration below shows the two directions of travel on a lathe. To carry the number line idea a little further, imagine such a line placed along each axis of the machine.

*SL-20 showing X and Z axis lines.*



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

The increments on each number line on the HAAS lathe equals .0001 inches. While a line theoretically has infinite length in either direction, the two lines placed along the X and Z axes of the machine do not have unlimited accessibility. That is to say, we are limited by the range of travel on the machine. For the HAAS SL-20, for example, we have access to 8.45 inches in the X axis and 20 inches in the Z axis.

The zero position may be placed at any point along each of the two number lines, and in fact will probably be different for each setup of the machine. It is noteworthy to mention here that the X-axis is usually set with the work zero position on the center line of the spindle, while the Z axis zero is usually set at the finished right end of the part being machined. This will place all the X axis cutting in a positive range of travel, whereas the Z axis cutting will be in the negative range of travel.

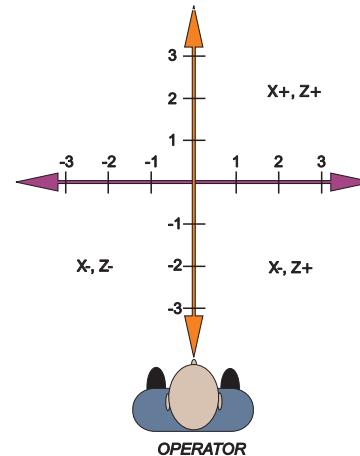
The diagram at right shows a front view of the grid as it would appear on the lathe. This view shows the X and Z axes as the operator faces the lathe. Note that at the intersection of the two lines, a common zero point is established. The four areas to the sides and above and below the lines are called "QUADRANTS" and make up the basis for what is known as rectangular coordinate programming.

THE BOTTOM RIGHT QUADRANT IS $X-, Z+$

THE TOP LEFT QUADRANT IS $X+, Z-$

THE BOTTOM LEFT QUADRANT IS $X-, Z-$

THE TOP RIGHT QUADRANT IS $X+, Z+$



Operator's working grid.

Whenever we set a zero somewhere on the X axis and somewhere on the Z axis, we have automatically caused an intersection of the two lines. This intersection where the two zeros come together will automatically have the four quadrants to its sides, above, and below it. How much of each quadrant we will be able to access is determined by where we placed the zeros on the travel axes of the lathe.

For example, if we set zero exactly in the middle of the Z axis and if we set the X axis zero on the spindle center line, we have created four quadrants. For an SL-20, for example, the upper two quadrants each have Z travel of 10 inches and X travel of 7.6 inches. The lower two quadrants will have Z travel of 10 inches and X travel of 1 inch. The HAAS lathes have 1 inch of negative travel beyond the center line of the spindle.

**3.3 MACHINE HOME**

The principle of machine home may be seen when doing a manual reference return of all machine axes. When a zero return (ZERO RET) is performed at machine start up, all axes are moved to the furthest positive direction until the limit switch is reached. When this condition is satisfied, the only way to move any of the axes is in the negative direction. This is because a new zero was set for each of the axes automatically when the machine was brought Home. Machine home is placed at the edge of each axes travel. In effect, now the positive quadrants cannot be reached, and all the X and Z moves will be found to be in the X-, Z- quadrant. It is only by setting a new part zero somewhere within the travel of each axis that other quadrants are able to be reached.

Sometimes it is useful in the machining of a part to utilize more than one of the X, Z quadrants. A good example of this is a part that needs to be faced off.

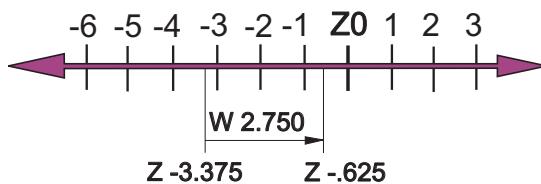
These are just some examples of how to make use of the quadrants of the X and Z axes on the machine. As more experience is gained in the techniques of machine tool programming and setup, each programmer and setup person develops their own methods and style. Some methods will be faster than others, but each individual will have to determine the needs of each job in question, and reflect back on notes and the previous jobs completed.

3.4 ABSOLUTE AND INCREMENTAL POSITIONING

Up to this point, we have dealt with a system of positioning the tool that is known as absolute programming. In absolute, all coordinate points are given with regard to their relationship to the origin, a fixed zero point, or considered as part zero. This is the most common type of positioning.

Another type of positioning is called incremental. Incremental positioning is defined using U and W. The "U" character is used to specify incremental motion in the X-axis, and the "W" character is used to specify incremental motion in the Z-axis and concerns itself with the distance and direction. An incremental coordinate position is entered, using U and W, in terms of its relationship to the previous position, and not from a fixed zero or origin point. In other words, after a block of information has been executed, the position that the tool is now at is the new zero point for the next move to be made. The "U" address is used to specify an incremental move along the X-axis and the "W" address is used to specify an incremental move along the Z-axis.

An example of the use of the incremental system is shown below. Note that to move from Z -3.375 to Z -.625 on the scale, a positive incremental move of W 2.750 was made, even though the move W still places the tool on the minus side of the scale. Therefore the move was determined from the start point position, with no regard for the last point, with no regard for the fixed zero reference point. The + and - signs are used in terms of direction, and not in regard to the position of part zero.



An example of an incremental move.

Keep in mind that when positioning in **absolute**, we are concerned with distance and direction from a fixed zero reference point, and when positioning in **incremental** we are concerned with distance and direction from the last position.



3.5 AUTOMATIC ACCELERATION/DECELERATION

This machine is not capable of instantly changing speed; it takes some nonzero time to accelerate and decelerate. Acceleration and deceleration in this machine have both a constant accel/decel mode and an exponential mode. Constant acceleration is used at the beginning of a rapid move and at the end of any move whose speed exceeds the exponential accel/decel time constant.

CONSTANT ACCELERATION

Constant acceleration is a type of motion when the amount of speed change over time is constant. This constant is set by Parameters 7, 21, 35, and 49. It has units of encoder increments per second per second.

Constant acceleration applies to the beginning of a rapid move so that the minimum time is spent getting up to rapid speed. It also applies to the end of rapid moves until the speed drops below the exponential accel/decel time constant. That change occurs at about 129.7 inches per minute in the following example.

EXPONENTIAL ACCELERATION

Exponential acceleration/deceleration is a type of motion where the speed is proportional to the distance remaining in a programmed travel. The exponential accel/decel time constant is set by Parameters 113, 114, 115, and 116. It has units of 0.0001 seconds. The speed limit at which exponential accel/decel is not available is defined by the relationship between Parameters 7 and 113 (for the X-axis). Thus, if Parameter 7 is 8000000 steps/sec/sec and Parameter 113 is 375 (0.0375 seconds); the maximum velocity for exponential accel/decel would be:

$$8000000 \times 0.0375 = 300000 \text{ steps/second}$$

For a 8192 line encoder and 6 mm screw, this would be:

$$60 \times 300000 / 138718 = 129.7 \text{ inches/minute}$$

ACCELERATION IN FEED MOTIONS

In the normal feed cutting mode, with G64 active, giving continuous cutter motion, deceleration of the axes in motion begins at some distance away from the end point. If lookahead has buffered another motion, the acceleration for that motion will begin at the same instant. This means that two motions, at right angles to each other, will not produce a perfectly square corner. The corner will be rounded. In addition, two motions which smoothly blend one into the other will not cause the tool to pause.

If you use tool nose compensation to cut an outside corner, there will be no rounding if the tool nose compensation amount is close to the actual tool size. This is because the tool is moved beyond the end of the first programmed stroke before it is moved to the beginning of the second stroke. Note that in this machine, using the default parameter settings, rapid and feed moves will both be blended to provide continuous cutter path and rounded corners. Unless you specify exact stop, the following rapid or feed block will be started slightly before the completion of the previous block.

The end of a feed move is delayed until the following error is below an amount set in Setting 85. If Setting 85 is set to 0.1, with Parameter 113 set to 375 (0.0375 seconds), this means that the highest feed rate which will give continuous cutter motion is:

$$(0.1) * 60 / 0.0375 = 160 \text{ inches per minute}$$

**ACCELERATION IN RAPID MOVES**

Rapid moves have a slightly different operation when continuous cutter mode is active. Acceleration for the next motion is started when the axes being moved are all within the "In Position Limit" Parameters 101, 102, 103, and 104. These parameters have units of encoder steps. Rapid moves will also decelerate at the constant accel/decel limit until the speed drops below that for exponential accel/decel. An example of the "In Position Limit" values follows. If Parameter 101 (for X) is 8000 and Parameter 5 is 138718, a rapid move will proceed to the next block when the X-axis is within a distance of:

$$8000 / 138718 = 0.0577 \text{ inches}$$

To prevent the rounding of corners, you can specify exact stop either with G09 (non-modal) or with G61 (modal). When either of these is active in a motion, all of the axes are brought to an exact stop, at zero speed, before the next motion is started.

Note that in this machine, using the default parameter settings, rapid and feed moves will both be blended to provide continuous cutter path and rounded corners. Unless you specify exact stop, the following rapid or feed block will be started slightly before the completion of the previous block.

ACCELERATION/DECELERATION IN CIRCULAR MOVES

The tool path in a circular move (G02 or G03) is not changed by the exponential acceleration/deceleration so that there is no error introduced in the radius of the cut unless the speed exceeds that for exponential accel/decel (see example above giving 129.7 inches per minute). However, the actual radius of a circular move will always be slightly smaller than the programmed value. The amount of change can be computed by the following equation:

$$Ra = \text{SQRT}(R^2 - L^2)$$

Where **Ra** is the actual radius,
R is the programmed radius, and
L is the accel/decel lag in feed motion.

The lag amount is computed by:

$$L = (\text{Par. 113}) * (\text{feed in/min}) / 600000$$

As an example; if Par 113 is 375 (0.0375 sec) and the feed is 30 inches per minute and the programmed diameter is two inches, the actual radius will be:

$$L = 375 * 30 / 600000 = 0.0187 \text{ inches}$$

and

$$Ra = \text{SQRT}(1 - 0.000351) = 0.999824$$

or an error of 176 millionth's of an inch. This is an upper bound on the accuracy of this cut and many other factors could contribute additional errors.



FANUC 6M, 10M, AND 15M COMPATIBILITY

Parameter 57 may be used to change the rapid accel/decel mode to one closer to that of the 10M and 15M controls. This is done with the flag called "EX ST MD CHG". This means "exact stop in mode change" and, if this flag is set to 1, will cause an exact stop at both the beginning and end of any rapid move. Thus continuous cutter motion is provided only for a feed motion followed by another feed motion. When this flag is set, the exact stop codes G09 and G61 will still provide an exact stop between two feed motions.

Setting 33 controls how the G52 and G92 codes work. These are different between Fanuc class controls and Yasnac class controls. To operate like a Fanuc control, Setting 33 should be set to FANUC.

Setting 58 controls how cutter compensation goes around outside corners. This motion is different between Fanuc class controls and Yasnac class controls. To operate like a Fanuc control, Setting 58 should be set to FANUC.

NOTE: The Haas CNC Control is **compatible** with many other controls; it is not **identical** in performance to any single control.

3.6 PROGRAMMING WITH CODES

A program is written as a set of instructions given in the order they are to be performed. The instructions, if given in English, might look like this:

LINE #1 =	SELECT CUTTING TOOL.
LINE #2 =	TURN THE SPINDLE ON AND SELECT THE RPM.
LINE #3 =	TURN THE COOLANT ON.
LINE #4 =	RAPID TO THE STARTING POSITION OF THE PART.
LINE #5 =	CHOOSE THE PROPER FEED RATE AND MAKE THE CUT(S).
LINE #6 =	TURN OFF THE SPINDLE AND THE COOLANT.
LINE #7 =	RETURN TOOL TO HOLDING POSITION AND SELECT NEXT TOOL.

and so on. But our machine control understands only these messages when given in machine code.

Before considering the meaning and the use of codes, it is helpful to lay down a few guidelines:

1. Codes come in groups. Each group has an alphabetical address. The rule is, with the exception of G codes and macro calls, codes with the same alphabetical address cannot be used more than once on the same line.
2. A **G** code come in groups. Each G code group has a specific group number. G codes from the same group cannot be used more than once on the same line.
3. There are modal **G** codes which, once established, remain effective until replaced with another code from the same group.
4. There are non-modal **G** codes which, once called, are effective only in the calling block, and are immediately forgotten by the control.



The rules above govern the use of all codes for programming the Haas (and other) controls. The concept of grouping codes and rules that apply will have to be remembered if we are to effectively program the machine tool. The following is a discussion of the codes most basic to the operation of the machine.

G CODES:

- G00 Rapid traverse motion; Used for positioning and during non-cutting moves.
- G01 Linear interpolation motion; Used for actual machining and metal removal. Governed by programmed feed rate in inches (or mm) per revolution.
- G02 Circular interpolation - Clockwise.
- G03 Circular interpolation - Counterclockwise.
- G18 Z,X Plane selection.
- G28 Machine home (Rapid traverse).
- G40 Cancel Cutter Comp.
- G70 Finishing cycle.
- G71 O.D./I.D. stock removal cycle.
- G72 End Face stock removal cycle.
- G76 Thread cutting cycle.
- G80 Canned cycle cancel.
- G81 Drill canned cycle.
- G82 Spot drill canned cycle.
- G83 Peck drill canned cycle.
- G84 Tapping canned cycle.
- G90 O.D./I.D Turning cycle.
- G96 Constant surface speed on.
- G97 Constant surface speed cancel.



G98 Feed per minute.

G99 Feed per revolution.

M CODES:

M00 Program stop. Press CYCLE START button to continue.

M01 Optional program stop. Press OPT STOP key on control panel to execute an M01 code.

M02 End of program. Cannot continue.

M03 Start spindle forward (Clockwise). Must be accompanied by a spindle speed.

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

M05 Spindle stop.

M08 Coolant ON command.

M09 Coolant OFF command.

M10 Clamp spindle chuck.

M11 Unclamp spindle chuck.

M30 Program end and rewind to beginning of program.

M97 Local subroutine call.

M98 Subprogram call.

M99 Subprogram return, or loop.

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

**3.7 MACHINE DEFAULTS**

A default is an automatic function of the machine tool control. When powering up the machine, the control looks for the home position of all axes, then will read the default values or the preset "G" codes.

The defaults for the Haas lathe are listed in this manual, and are indicated by an asterisk (*) next to the specific G codes.

The control automatically reads these G codes when power is turned on.

G00	Rapid traverse
G18	X-Z Circular plane selection
G20	Select inches
G40	Cutter Compensation cancel
G64	Exact stop cancel
G80	Canned cycle cancel
G97	Constant surface speed cancel
G99	Feed per revolution

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



3.8 PROGRAM FORMAT

Program format, or program style is an important part of CNC machining. Each individual will format their programs differently and, in most cases, a programmer could not identify a program written by themselves. The point is that a programmer needs to be consistent and efficient, writing code in the way it is listed and in the order it appears in the program. For example:

Program X, Z in alphabetical order on any block. The machine will read X or Z in any order, but we want to be consistent. Write X first, Z second.

The first line or block in a program, (using active G codes), should be a return to machine zero. Any tool change should be preceded by a return to machine zero. Although this is not necessary it is a good safety measure.

The second line of code should apply to any appropriate tool shifts, work coordinates, or spindle speed maximums for the tool being used.

The third line or block should place the control in rapid positioning mode (G00). It should cancel any constant surface speed mode (G97). It should select a tool and apply any additional offsets with the T_{xxxy} command. And it should specify a spindle speed command (S____) along with a spindle ON clockwise command (M03).

The fourth line should contain a preparatory X, Z positioning command.

The fifth line should optionally specify constant surface speed with (G96) and surface feet per minute (sfm) with (S____) and optionally turn on the coolant with (M08).

An example program's first five lines will look like this:

```
G51 ;
G50 S2000 ;
G00 G97 T101 S500 M3 ;
X2.0 Z.01 ;
G96 S500 M8 ;
```

All the necessary codes for each operation are listed above. This format is a good practice and will separate your style from other programmers.

QUESTION:

If G00 is a default, why do we list it in the third line of a program and for each different tool?

ANSWER:

G00 is listed to assist the operator in determining if the machine will rapid position. It is also important for tool shift. The tool offset is always different between setups, and multiple tool offsets are very common.

QUESTION:

Can we turn on the coolant at the same time that we turn on the spindle?

ANSWER:

No. Only one M code per program line is allowed.

Although it is not necessary, the tool number should always remain numerically matched with the tool offset number. Setting 15 (the Offset & Tool agreement) will ensure the tool number and tool offset will match. (eg. T01 in line #3 should have 01 as an offset, similarly T02 should have 02 as an offset.)

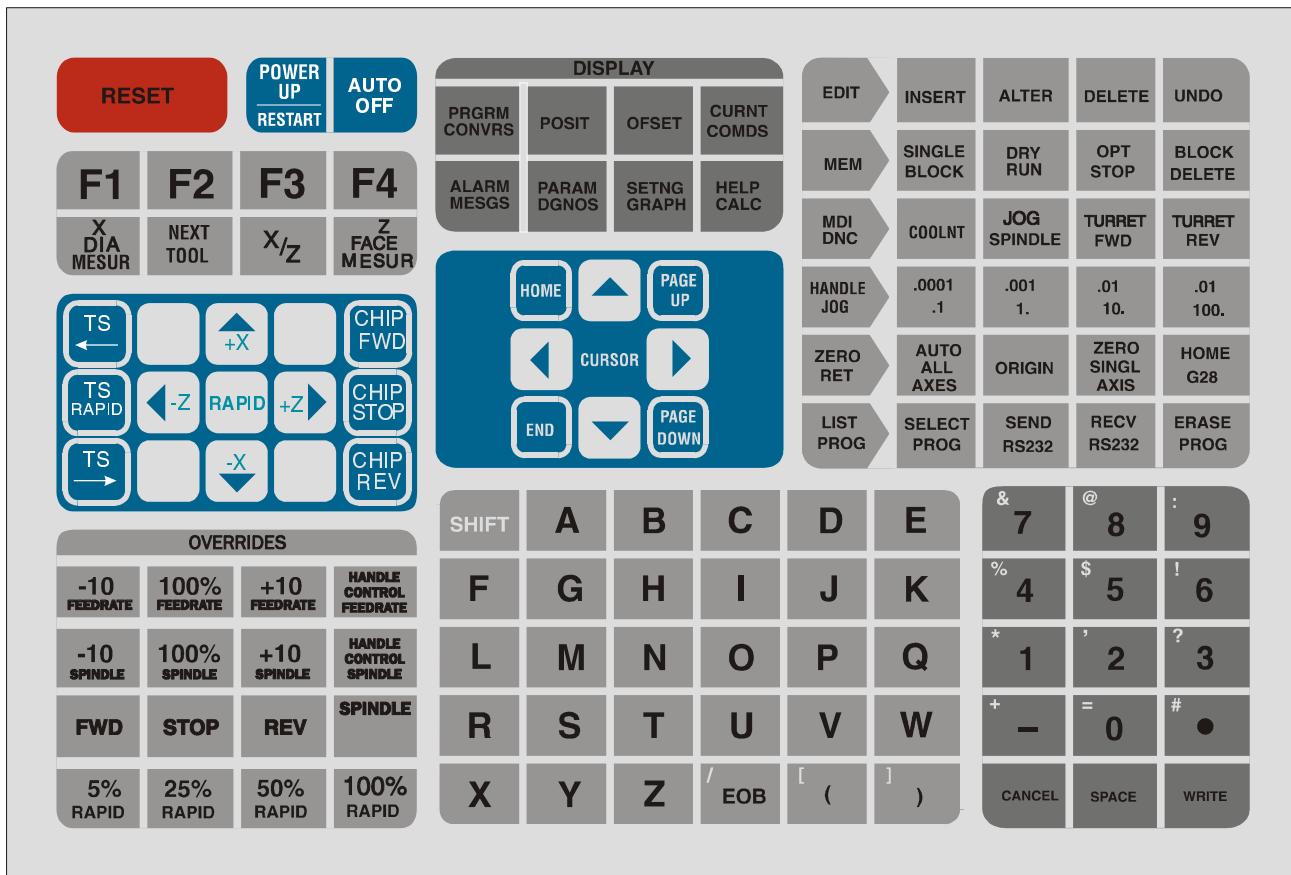
**3.9 CANNED CYCLES**

A canned cycle is used to simplify programming of a part. Canned cycles are defined for most common repetitive operations such as turning, threading, grooving, tapping, drilling, and boring. There are both modal and non-modal canned cycles. Modal canned cycles such as the turning cycle G90, remain in effect after they are defined. After any subsequent X or Z positioning, the canned cycle is executed. Modal canned cycles remain in effect until canceled by a G80, G00, end of program or reset. Non-modal canned cycles are effective only for the block that contains them.

TAPPING WITH THE LATHE

Making tapped holes with the Haas Lathe can be made with several devices. Threads may be generated using a tap held in a rigid tool holder (rigid tapping), a floating tap holder, a reversing tapping head, or helical thread milling. Each method has distinct advantages.

Tapping is done using canned cycles. You must select the tapping RPM and, using the pitch (threads per inch), calculate the feed rate that is entered in the F command. The HELP/CALC page will help compute these numbers for you.


3.10 OPERATORS CONTROL PANEL


Control panel keypad with operating and display keys highlighted.

In operation, it is important to be aware of the operating mode selected for the CNC. There are six operating modes and one simulation mode in this control. The operating mode is selected with the six buttons labeled:

- | | |
|------------|--|
| EDIT | To edit a program already in memory |
| MEM | To run a program stored in memory |
| MDI / DNC | To directly run manually entered program or to select DNC mode |
| HANDLE JOG | To use jog keys or jog handle |
| ZERO RET | To establish machine zero |
| LIST PROG | To list, send, or receive programs |



The Graphics simulation mode is entered with the DISPLAY select buttons.

In MEM or MDI mode, a program can be started with the CYCLE START button. While a program is running, you cannot change to another mode; you must wait until it finishes or press RESET to stop the program.

When already in MDI, a second press of the MDI button will select DNC if the DNC mode is enabled by settings and parameters in your machine.

In any of the above modes, you can select any of the following displays using the eight DISPLAY buttons:

PRGRM / CONVRS	To show the program selected
POSIT	To show the axes positions
OFSET	To show or enter working Offsets
CURNT COMDS	To show Current Commands and times
ALARM / MESGS	To show Alarms and user messages
PARAM / DGNOS	To show Parameters or Diagnostic data
SETNG / GRAPH	To show or enter Settings or to select

Graphics simulation mode is entered with the DISPLAY select button

HELP / CALC	To show the Help data and calculator
-------------	--------------------------------------

In addition to the above displays, when a program is already running, you may press LIST PROG to select a list of the programs in memory. This is useful to determine what programs can be edited in BACKGROUND EDIT. BACKGROUND EDIT is selected from the PROGRAM DISPLAY.

All operation of the CNC is controlled from the operator's panel. The control panel is composed of the CRT display, the keypad, On/Off switch, Load meter, Handle, EMERGENCY STOP, CYCLE START, and FEED HOLD buttons.

The **keypad** is a flat membrane type that requires approximately eight (8) ounces of pressure. The **SHIFT** button replaces the function of the numeric buttons with the white characters in the upper left corner. The **SHIFT** button must be pressed once before each shifted character. Pressing the **SHIFT** button twice will turn off shift.

The **load meter** measures the power to the spindle motor. At 100%, the spindle motor can be operated continuously. The 150% level can be sustained for no more than ten (10) minutes, and at 200% level no more than three (3) minutes. After the specified time, the spindle may begin to slow and even stall. A 200% load should be reduced to 150% by reducing spindle speed or decreasing the feed rate. Spindle load may increase temporarily during speed changes.



The **jog handle** is used to jog one of the axis. Each step of the crank can be 0.0001, 0.001, 0.01 or 0.1 inch. The handle has 100 steps per rotation. When using metric, the smallest handle step is 0.001 mm and the largest is 1.0 mm. As an option, the handle can also be used to move the screen cursor while in EDIT mode.

The **EMERGENCY STOP** button will instantly stop all motion of the machine including the servo motors, the spindle, the tool changer, and the coolant pump. It will also stop any auxiliary axes.

CYCLE START will start a program running in MEM or MDI mode, continue motion after a FEED HOLD, or continue after a SINGLE BLOCK stop.

FEED HOLD will stop all axis motion until the CYCLE START is pressed.

WARNING!

FEED HOLD will not stop motion of the spindle, turret, coolant pump, or the tailstock.

The **SINGLE BLOCK** button on the keypad will turn on and off the SINGLE BLOCK condition. When in SINGLE BLOCK, the control will operate one block and stop. Every press of the START button will then operate one more block.

The **RESET** button on the keypad will always stop motion of the servos, the spindle, the coolant pump, and tool changer. It will also stop the operation of a running program. This is not, however, a recommended method to stop the machine as it may be difficult to continue from that point. SINGLE BLOCK and FEED HOLD provide for continuation of the program. RESET will not stop motion of any auxiliary axes but they will stop at the end of any motion in progress.

The CRT is the only display or readout device in the control. All status and position data is shown on the CRT.

The F1, F2, F3, and F4 buttons perform different functions depending on what display and mode is selected. The following is a quick summary of the **Fn** buttons:

F1 In EDIT mode and PROGRAM DISPLAY, this will start a block definition.

In LIST PROG mode, F1 will duplicate a program already stored and give it a new name from the command line.

In offsets display, F1 will set the entered value into the offsets.

F2 In EDIT mode, PROGRAM DISPLAY, this will end a block definition.

F3 In EDIT and MDI modes, the F3 key will copy the highlighted circular help line into the data entry line at the bottom of the screen. This is useful when you want to use the solution developed for a circular motion. Press INSERT to add that circular motion command line to your program.



In the calculator Help function, this button copies the value in the calculator window to the highlighted data entry for Trig, Circular, or Milling Help.

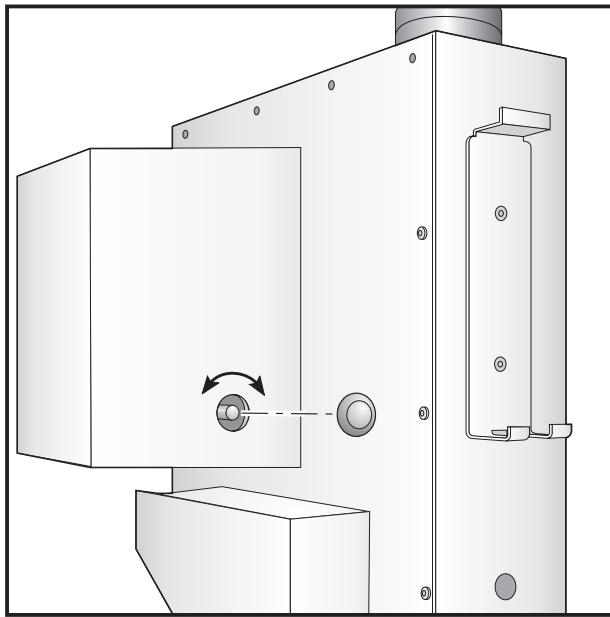
F4 In MEM mode and PROGRAM DISPLAY, this will select either BACKGROUND EDIT or PROGRAM REVIEW. BACKGROUND EDIT is selected by entering **Onnnnn** with the program number to edit. Program review is selected with just F4. Program review shows the running program on the left half screen and allows the operator to review the program on the right half screen.

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

3.11 REAL TIME CLOCK

Real Time Clock places a date time stamp on the following:

1. Displays current Date and Time. The date and time as supplied by the real-time clock are displayed on the diagnostics screen.
2. Alarm history tagged with date and time. The alarm history is now displayed with the date and time that each alarm occurred. See the Alarms section on how to access alarm history.
3. Macro variables. Macro variable #3011 contains the date in the format yymmdd (two digit year* 10000+ month* 100+ day). Macro variable #3012 contains the time in the format hhmmss (hours* 10000 + minutes* 100 + seconds).
4. Parameter output contains date and time. When outputting a parameter file to a floppy disk or the serial port, it will contain two new comments near the top containing the current date and time.
5. Floppy disk directory contains file creation date and time. When files are saved to the floppy disk, the directory will now show the file creation date and time.

**DISPLAY BRIGHTNESS ADJUSTMENT**

Remove plug to access the brightness adjustment knob. Be sure to replace plug.

**3.12 KEYBOARD**

The control panel keyboard consists of 130 keys and is divided into nine separate regions. They are:

RESET keys	Three (3) keys
FUNCTION keys	Eight (8) keys
JOG keys	Fifteen (15) keys
OVERRIDES	Fifteen (15) keys
DISPLAYS	Eight (8) keys
CURSOR keys	Eight (8) keys
ALPHA keys	Thirty (30) keys
MODE keys	Thirty (30) keys
NUMERIC keys	Fifteen (15) keys

A detailed description of how and where these keys are used can be found through use of the index. The following are short descriptions of the control panel keys' usage.

RESET KEYS: The RESET keys are in the upper left corner of the control panel.

RESET	Stops all machine motion and places the program pointer to the top of the current program.
POWER UP/ RESTART	Automatically initializes the machine at power up. After power up, the axes zero return and tool one is put into the cutting position.
AUTO OFF	Automatically positions axes to machine zero and prepares the machine for power down.

FUNCTION KEYS: Below the reset keys are the function keys. There are eight function keys. They are used to execute special functions implemented throughout the control software.

F1-F4	Used in editing, graphics, background edit, and the help/calculator to execute special functions.
-------	---

X DIAM MESUR Used to record X axis tool shift offsets on the offset page during part setup.



NEXT TOOL	Used to select the next tool during part setup.
X/Z	Used to toggle between X axis and Z axis jog modes during part setup.
Z FACE MESUR	Used to record Z axis tool shift offsets on the offset page during part setup.

JOG KEYS: The jog keys are on the left below the function keys. These keys select which axes the jog handle sends signals to and provides for continuous jogging. When a key is pressed briefly, that axis is selected for use by the jogging handle. When a key is pressed and held down, that axis is moved as long as the key is held down.

If a "+" key is pressed and held, the axis is moved so that the tool position is changed in a positive direction relative to the work coordinates. If a "-" key is pressed and held, the axis is moved so that the tool position is changed in a negative direction relative to the work coordinates. The jog keys are locked out if the machine is running.

+Z, -Z	Selects the Z axis.
+X, -X	Selects the X axis.
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.

To the right side of the jog keys are three keys to control the optional chip conveyor:

CHIP FWD	Turns the chip conveyor in a direction that removes chips from the work cell.
CHIP STOP	Stops chip conveyor movement.
CHIP REV	Turns the chip conveyor in the reverse direction.

To the left side of the jog keys are three keys to control the optional tailstock:

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.



OVERIDES: The overrides are at the lower left of the control panel. They give the user the ability to override the speed of rapid traverse motion, as well as programmed feeds and spindle speeds.

FEED RATE	Not a key.
-10	Decreases current feed rate by 10% (from 10 to 200%).
100%	Sets control feed rate to programmed feed rate.
+10	Increases current feed rate by 10% (from 10 to 200%).
SPINDLE	Not a key.
-10	Decreases current spindle speed by 10% (from 10 to 150%).
100%	Sets spindle speed to programmed speed.
+10	Increases current spindle speed by 10% (from 10 to 150%).
FWD	Starts the spindle in the forward (clockwise) direction. Except on CE machines.
STOP	Stops the spindle.
REV	Starts the spindle in the reverse (counterclockwise) direction. Except on CE machines.
5% RAPID	Limits rapid traverse to 5 percent of maximum.
25% RAPID	Limits rapid traverse to 25 percent of maximum.
50% RAPID	Limits rapid traverse to 50 percent of maximum.
100% RAPID	Allows rapid traverse to feed at its maximum.

DISPLAYS: The display keys are in the center at the top. These eight keys provide access to the different displays and operational information and help routines available to the user. Some of these keys are multi-action keys in that they will display different screens when pressed multiple times. The current display is always displayed on the top left line of the video screen.

PRGRM / CONVRS	Displays the currently selected program. Also used in QuickCode applications.
POSIT	Displays the position of the machine axes. Pressing PAGE UP and PAGE DOWN will show operator, machine, work, and distance-to-go formats in large letter format.
OFSET	Displays the tool length and radius offsets. PAGE UP will display the values of the axes' work offsets.
CURNT COMDS	Displays the current program, modal program values, and position during run time. Succeeding presses of the PAGE DOWN key will display modal values, system timers, macro variables, tool life and tool load information.
ALARM / MESGS	Shows the full text of an alarm when the alarm message is flashing. Pressing the left or right arrow keys will display an alarm history. Pressing PAGE DOWN will display a page for user messages and notes.



PARAM / DGNOS	Displays and allows changing of parameters that define machine character. Pressing PAGE UP will display lead screw compensation values. Successive PAGE DOWN presses will display general parameters as well as the X, Y, Z, A and B parameters. A second press of the PARAM key will display the first page of diagnostic data. The first page of diagnostic data is discrete inputs and outputs. Pressing PAGE DOWN will display the second page of diagnostic data that consists of additional inputs and analog data.
SETNG / GRAPH	Displays and allows changing of user settings. Pressing the SETNG key twice enables graphics mode where the user can debug the current program and view the program's generated tool path.
HELP / CALC	Displays a brief, on-line manual. Pressing HELP a second time will display the help calculator. There are three pages of calculator help. Pressing the PAGE DOWN key will display turning and tapping help, trigonometry help, or circle help.
CURSOR KEYS:	The cursor keys are in the center of the control panel. They give the user the ability to move to various screens and fields in the control. They are used extensively for editing of CNC programs.
HOME	Context-sensitive key that generally moves the cursor to the topmost item on the screen. In editing, this is the top block of the program. In graphics, it will display the entire table in the view window after F2 is pressed.
 (UP ARROW)	The up arrow moves up one item, block, or field. In graphics, the zoom window is moved up.
PAGE UP	Used to change displays, move up one page in the editor, or zoom out when in graphics.
 (LEFT ARROW)	Used to select individually editable items within the editor, moves cursor to the left. It selects optional data in fields of the settings page and moves the zoom window left when in graphics.
 (RIGHT ARROW)	Used to select individually editable items within the editor, moves cursor to the right. It selects optional data in fields of the settings page and moves the zoom window right when in graphics.
END	Context-sensitive key that generally moves the cursor to the bottom most item on the screen. In editing, this is the last block of the program.



(DOWN ARROW) The down arrow moves down one item, block, or field. In graphics, the zoom window is moved down.

PAGE DOWN Used to change displays, move down one page in the editor, or zoom closer when in graphics.

ALPHA KEYS: The alpha keys allow the user to enter the 26 letters of the alphabet along with some special characters.

SHIFT The shift key provides access to the white characters on the keyboard. Pressing SHIFT and then the white character will cause that character to be sent to the control. When entering text, UPPER CASE is the default. To access lower case characters, press and hold the SHIFT key while pressing the appropriate characters. The SHIFT key can also be continuously held down while a number of other keys are pressed.

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

EOB This is the END-OF-BLOCK character. It is displayed as a semicolon on the screen and it signifies the end of a programming block. It is the same as a carriage return and then a line feed.

() The parenthetical brackets are used to separate CNC program commands from user comments. They must always be entered as a pair and may or may not have additional characters separating them. 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 these two brackets.

/ The right slash is used as a block delete flag. 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 in macro expressions.

In some FANUC compatible controls, the block delete symbol can be used to choose between two options when the "/" symbol is not at the beginning of the line. For instance, in the following line, T2 is executed when the block delete option is off, and when the block delete option is on, T1 is executed.

T1 / T2;
N1 G54

This cannot be done on a HAAS control.



A coding method for achieving the same results on a HAAS control is given below:

/ T2 M99	(T2 executed when block delete is off)
T1	(T1 executed when block delete is on)
N1 G54	

[and]

Square brackets are used in macro expressions and functions.

MODE KEYS:

The mode keys are in the upper right part of the control panel. These keys change the operational state of the CNC machine tool. There are six major operation modes. The user can enter a specific mode by pressing the desired "arrow" shaped key on the left. The keys in the same row as the pressed mode key are then made available to the user. Otherwise, these keys are not available. The current mode is always displayed on the top line just to the right of the current display on the video screen.

EDIT**Selects edit mode.**

INSERT	Inserts the text in the input buffer after the current cursor location. Also used to copy blocks of code in a program.
ALTER	Changes the item that the cursor is on to the text in the input buffer. Places an MDI program in the program list.
DELETE	Deletes the item that the cursor is on.
UNDO	Backs out or undoes up to the last 9 edit changes.

MEM**Selects MEM mode.**

SINGLE BLOCK	Turns single block on so that when the cycle start button is pressed, only one block of the program running is executed.
DRY RUN	Used to check actual machine movement without cutting a part. Programmed feeds are replaced by the speed keys in the handle jog row.
OPT STOP	Turns on optional stops. If an M01 code is encountered in the program and OPT STOP is on, then a stop is executed. Depending on the lookahead function, it may not stop immediately. If the program has been interpreted many blocks ahead, and the OPT STOP is pressed, then the nearest M01 may not be commanded. See G103.

1. OPT STOP will take effect on the line after the highlighted line when OPT STOP is pressed.
2. M01 is not allowed during cutter compensation. Alarm 349 will be generated in this case, as for M02, M30, and M00.



BLOCK DELETE	Blocks with a slash ("/") as the first item are ignored or not executed when this option is enabled. If a slash is within a block, address codes after the slash will be ignored until after the block, if this option is enabled.
	<ol style="list-style-type: none">1. When not in cutter compensation, block delete will take effect two lines after BLOCK DELETE is pressed.2. When in cutter compensation, blocks must be processed earlier. Therefore, block delete will not take effect until at least four lines after the highlighted line when BLOCK DELETE is pressed.3. If BLOCK DELETE changes state during the processing of the first block of a chamfering/rounding pair, and at least one of the pair is block deleted, the behavior is undefined.
MDI/DNC	Selects MDI or DNC mode.
COOLNT	Turns the coolant on and off.
JOG SPINDLE	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 the turret is positioned at the highest numbered tool, the turret will advance in the forward direction to tool #1. If Tnn is entered on the input line, the turret will advance in the forward direction to tool nn. Except on CE machines.
TURRET REV	Rotates the tool turret backward to the previous tool. If the turret is positioned at tool number #1, the turret will advance in the reverse direction to the highest numbered tool. If Tnn is entered on the input line, the turret will advance in the reverse direction to tool nn. Except on CE machines.
HANDLE JOG	Selects Jogging mode.
.0001, .1	.0001 inches or .001 mm for each division on the jog handle. For dry run .1 inches/min.
.001, 1.	.001 inches or .01 mm for each division on the jog handle. For dry run 1. inches/min.
.01, 10.	.01 inches or .1 mm for each division on the jog handle. For dry run 10. inches/min.
.1, 100.	.1 inches or 1.0 mm for each division on the jog handle. For dry run 100. inches/min.

**ZERO RET Selects Zero Return mode.**

- | | |
|----------------|---|
| AUTO ALL AXES | Searches for all axes' machine zero. |
| ORIGIN | Zeros out various displays and timers. |
| ZERO SINGLAXIS | Searches for machine zero on the axis that is specified in the input buffer. |
| HOME G28 | Returns all axes to machine zero in rapid motion. Does not search. |
| SINGLE AXIS | Either the X, Z, or B (tailstock) axis can be returned to zero alone. |
| HOME G28 | To use this feature the operator enters "X", "Z", or "B", then presses the HOME G28 key. Pressing HOME G28 without first entering an axis letter will cause all axes to be returned to zero. If the chosen axis is disabled, the message DISABLED AXIS will be generated. If moving the chosen axis would cause the tool turret to enter the tailstock restricted zone, the message X IN THE WAY will be generated and no motion will occur. |
| SECOND HOME | The control will rapid all axes (which have the 2ND HOME BTN bit=1) to the coordinates specified in Work Offset G129. This feature was added to enable the operator to quickly move the turret to the window so that he can manually install tools without having to enter the machine. In order for this feature to work properly, the operator must first set the G129 Work Offsets to the desired values. This feature works from any mode except DNC. |

LIST PROG Selects Program List mode and displays a list of the programs in the control.

- | | |
|-------------|---|
| SELECT PROG | Makes the highlighted program on the program list the current program. The current program will have an asterisk preceding it in the program list. |
| SEND RS232 | Transmits programs out the RS-232 serial port. If ALL is highlighted, all the programs will be sent with one "%" at the beginning and one at the end of the stream. |
| RECV RS232 | Receives programs from the RS-232 serial port. Unless ALL is highlighted, enter a valid program name in the form Onnnnn before pressing RECV RS232. If ALL is highlighted, do not enter a program name. The program names will be entered automatically from the input stream data. |
| ERASE PROG | Erases the highlighted program or the program specified in the input buffer. |



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 during editing or field input.

SPACE

This is a space and can be used to format comments placed into programs.

WRITE /

This acts as the general purpose enter key. Any time that user needs to change ENTER any information in the control, this key is pressed.

-, .

Used to negate numbers, or provide decimal precision.

**+, =,
#, and ***

These symbols are accessed by first pressing the SHIFT key and then the key with these symbols. They are used in macro expressions.

**? , % , \$,
! , & , @ ,
and :**

These are additional symbols, accessed by pressing the SHIFT key. They can be used in program comments.



3.13 POWER ON/OFF

POWER ON

There is only one way to turn on this CNC. This is by pressing the green "On" button at the top left of the control panel. The main breaker at the rear of the mill must be on before this button will turn on the lathe. Any interruption to power will turn the lathe off and this button must be used to turn power back on again.

Upon power up, the machine must find its fixed reference point before any operations can occur. After power on, pressing the POWER UP/RESTART will establish this point. The ZERO RET mode and AUTO ALL AXES button may also be used to initialize the system after all alarms are cleared. A single axis can be selected by first pressing the **X** or **Z** key and then the ZERO SINGL AXIS key. The position thus found is used as machine zero.

CAUTION!

After power on, the machine does not know its home position or stored stroke limits until it has been zero returned by the POWER UP/RESTART key or the ZERO RET/AUTO ALL AXES key. It is possible to jog the machine with the handle or jog keys at the lower feeds. If it is jogged unchecked in either direction, you may damage the sheet metal covers or overload the ball screws. To avoid this, always properly ZERO RET the machine immediately after power on before doing anything else.

NOTE: Turret moves to tool #1 first, then to tool designated in Setting 81.

After initializing, all machine Position displays are reset to zero.

The HOME G28 key should be used any time after the initial power up. This will return both the Z-axis and the **X** axis at a rapid rate. If the Z-axis is positioned above the machine zero, the **X** axis is moved first. This key will work in any of the operating modes. The manual G28 button does not use any intermediate return point the way the programmed G28 does.

NOTE:

Repairs to the motor, ball screw, or home switch will affect the zero return point and must be done only by a factory trained technician. Serious damage to the ball screw, way covers, linear guides, or tool turret may occur if the zero return point is not properly set.

POWER OFF

Pressing the red POWER OFF button will remove power to the machine instantly. The machine can also be programmed to turn off at an end of cycle (M30) or after a preset amount of time that the machine sees no activity. These are Settings 1 and 2.

A sustained overvoltage condition or sustained overheat condition will also shut this machine off automatically. If either of these conditions exists for 4.5 minutes, the machine will start the 30 second auto-shutdown. Alarm 176 is displayed when an overheat shutdown begins and Alarm 177 is displayed when an overvoltage shutdown begins.

Any power interruption, including the rear cabinet main circuit breaker, will also turn this machine off. Power must be restored and the POWER ON button pressed to restore operation.

**SETUP PROCEDURES**

The following sequence of operations is an example of how one might set up tools for a bar stock job on this machine:

1. Load a program into memory. This is either manually entered at the keyboard or loaded to the control via the RS-232 interface.
2. Determine tools needed and get them ready.
3. Prepare the chuck, collet or fixture for holding the part.
4. Change to the JOG mode and the OFFSET display. This will enable the setup keys found below the function keys on the keyboard. The setup keys are X DIA MESUR, NEXT TOOL, X/ Z, and Z FACE MESUR.

To set up O.D. cutting tools do the following.

5. Load the setup part or material.
6. Provide clearance for a tool change. You can do this by jogging the X and Z axis to a place where no tool will interfere with the part, chuck or fixturing. Alternately, you can press HOME G28 which will return all axes to machine zero.
7. Select a turning tool, this may be the first tool, or a general purpose turning and facing setup tool. Select the tool by entering the number of the tool station that the tool resides in on the input line and pressing NEXT TOOL. This will make the entered tool the current tool. It also highlights the tool shift offset for that tool and places the control in X axis jog mode.

To measure the X tool shift offset do the following.

8. Start the spindle for a skim cut on the O.D. Perform the skim cut using the jog handle. You can use the X/Z key to change from X to Z axis jog as necessary. Generally you may want to start at a point nearest the spindle and work away from the spindle. Clean up the part 75% so that you can measure the O.D. with micrometers.
9. Back the Z axis away from the turned part, turn the spindle off with the spindle STOP key, and measure the O.D.
10. Press X DIA MESUR to record the offset from machine zero to the turned surface. Enter the measured diameter on the control input line and press WRITE. This will add the diameter to the offset so that the X offset for the tool tip is at the spindle center.

To measure the Z tool shift offset do the following.

11. Position the tool to do a cleanup cut on the face of the part. Press X/Z to select X jogging.
12. Turn on the spindle with the CW key, face off the part using the jog handle, back the X axis away from the part and turn the spindle off.
13. Press Z FACE MESUR to record the offset from machine zero to the face of the part. If the part program is not referenced from the face of the part you can add an additional offset amount by entering the amount on the control input line and then pressing WRITE.

**Measure additional O.D. tool offsets by selecting other O.D. tools and touching off the X and Z axes on the turned part.**

14. Provide clearance for a tool change as in 6 above.
15. Press NEXT TOOL to obtain the next tool as in 7 above.
16. Jog the tool to a point on the outside of the part and touch off the tool on the part. With the tip touching the part, press X DIA MESUR to record the X shift offset. Enter the part diameter and press WRITE.
17. Jog the tool to a point on the face of the part and touch off the tool on the part. With the tip touching the part, press Z FACE MESUR to record the Z shift offset. Enter any additional offset amount and press WRITE.
18. REPEAT steps 14 thru 17 for all remaining O.D. tools.

To set up I.D. cutting tools do the following.

19. Select a drill from the tool turret and jog to the spindle center.
20. Turn on the spindle and drill a hole in the setup stock sufficiently deep enough to turn a surface for measurement. Back out the drill for a tool change.
21. Select an I.D. turning tool by pressing the NEXT TOOL key as in 7 above.
22. Position the X axis to a measurable inside diameter, select the Z axis with the X/Z key and turn the I.D. Move the tool back out to clear for I.D. measurement. Record the X tool shift offset as in 10 above with the X DIA MESUR key.
23. Repeat steps 14 through 17 for all remaining I.D tools, with the exception that you will touch off the inside diameter of the setup part rather than the outside.

**3.14 MANUAL OPERATION****MANUAL DATA INPUT (MDI)**

Manual data input allows you to enter data that can be executed on a line by line basis instantly without having to use the EDIT and MEM modes. In this control, MDI is actually a scratch pad memory that can execute many lines of instruction without having to disturb your main program in memory. The data in MDI will be retained even when switching modes or in power off.

Editing with MDI is the same as memory editing.

The MDI mode also allows for manual operation of coolant, spindle, and tool changer.

A program in MDI can be saved as a normal named program in memory by placing the cursor at the beginning of the first line (HOME), typing **Onnnnn** (new program number), then pressing ALTER. This will add that name to the program list and clear MDI.

The entire MDI program may be cleared by pressing the ERASE PROG key while in MDI.

A fast way to select a tool is to type **Tnn** and, instead of INSERT, press either TURRET FWD or TURRET REV. This will directly select that tool.

When DNC is enabled with Setting 55, a second press of the MDI button will put the control into DNC mode.

When the Parameter 57 flag DOOR STOP is set to 1, manual tool change operations are not allowed with the doors open. In addition to this, the maximum spindle speed is 500 RPM.

HANDLE/JOG

Manually moving the axes is accomplished by pressing the mode button labeled HANDLE JOG and then by using the JOG keys or the handle to move the axis. Both the JOG buttons and the Handle are enabled simultaneously without needing to select between them. The display is changed to the Position Display.

In order to jog the tailstock the operator can press "B" , "handle jog", and then use the jog handle to jog the tailstock. Alternately, if the operator uses the TS \leftarrow or TS \rightarrow buttons, when the button is released the control will stay in tailstock jog mode and he can likewise use the jog handle.

Jog feed rate or handle resolution is selected by the four keys to the right of the HANDLE JOG key. Jog feeds from .1 inch per minute to 100 inch per minute or handle divisions from .0001 inch to .1 inch are selectable.

During jogging, the FEED RATE override buttons will adjust the rates selected from the keypad. This allows for very fine control of the jog speed. It does not change the handle step size.

This feature is handy, for example, when you are slow milling the soft jaws of a vise.

In order to select another axis for jogging while using the Handle, use +/- **X** or **Z** buttons. When one of these buttons is pressed, that axis is selected for HANDLE JOG but does not move unless the button is held down for more than $\frac{1}{2}$ second. After $\frac{1}{2}$ second, that axis is moved in the selected direction and at the selected feed rate.



3.15 AUTOMATIC OPERATION

OPERATION MODE

There are six modes of operation of the HAAS Lathe. They are:

EDIT	Used to make manual changes to a part program.
MEM	Used to run a users part program stored in memory.
MDI/DNC	Used to quickly manually enter and run a program.
HANDLE/JOG	Used to move the axes with the handle or JOG buttons.
ZERO RET	Used to search for machine zero and to return to machine zero automatically.
LIST PROG	Used to list, send, receive and delete programs.

Changes to the mode are made by pressing of the buttons on the top right quadrant of the keypad that have the above labels. If an operation is started, such as running a program, you cannot change modes until the operation is stopped. The six mode selection buttons are arranged vertically and, generally, the keys to their right apply only in that selected mode.

PROGRAM SELECTION

Program selection is done from the LIST PROG mode. This mode will list all of the programs stored in memory and allow you to select one as the current program. This is the program that will be run when you press START in MEM mode. It is the program with the “*” on the LIST PROG display. The selected program will also be seen on the EDIT display.

To select an existing program, press the CURSOR **up** or **down** buttons until the program you want is highlighted and then press the SELECT PROG button. The “*” will move to that program.

To select a new program (create a new program) or to select an existing program, you may also enter **Onnnnn** from the keyboard and then press SELECT PROG button.

There is a maximum of 500 programs stored in this control at a time.

**STARTING AUTOMATIC OPERATION**

Before you can run a program, it must be selected in the current memory. To select a program, press the LIST PROG mode key. Use the cursor to find the desired program and then press SELECT PROG. The program list includes the program name and the first comment. The last program selected when the machine is turned off will still be selected when the machine is turned back on.

If the machine has just powered up, you need to first press the POWER UP/RESTART key. This will initialize all axes and the tool changer, display the Current Commands, and go to MEM mode with the control ready to run. Pressing the CYCLE START button in the lower left of the control panel will begin execution.

To start a program at other than the beginning, scan to the block number using the keypad and the **down** arrow or PAGE DOWN until you reach the desired start place. Press the MEM key and CYCLE START to begin. The Program Restart function, selected from the Setting page 36, will change the way a program operates if you start from other than the first block. The setting called Program restart "ON" will ensure that the correct tool and axis positions are selected when you start from part way through a program.

Any errors in your program will cause an alarm and stop the running of the program. Typical alarms are travel limits and missing I, J, and Q codes. Attempts to move outside of the limits of travel will also cause an alarm.

When cutting materials that produce hot chips, it is important to use coolant.

At any time that a program is running, the bottom left corner of the CRT will show RUNNING. If it does not show this, the program has completed, has been stopped by the operator, or has been stopped by a fault condition.

PROGRAM RESTART

The Program Restart function may be selected from the Setting page. It allows a program to be restarted from other than the first block. You do this by using the CURSOR **up** and **down** keys in MEM mode to select the block to start operation and pressing CYCLE START. If Program-Restart is on, program interpretation will begin with the first block but no motion of the machine will occur until execution gets to the selected restart block. When it gets to the restart block, the axis and tools will be moved to the correct position and normal operation will proceed from there.

STOPPING AUTOMATIC OPERATION

There are several ways a program can be stopped. They include both normal stops and abnormal, or alarm caused, stops. The normal stops are:

1. Normal completion at M00, M01, M02, or M30
2. A FEED HOLD stop by the operator. This is continued by pressing CYCLE START again.
3. A SINGLE BLOCK stop when operator selected. The program is continued by pressing CYCLE START again.
4. Door Hold stop caused by operator opening the enclosure doors. The program continues when the door is closed.



The abnormal stops are:

1) Operator Reset

This stops all axes' motion, stops the tool changer, turns off the spindle, and turns off the coolant pump. Program operation cannot be continued from the stopping point. If Setting 31 is On, the program pointer is reset to the beginning.

2) Emergency Stop

This stops all axes' motion, disables the servos, stops the tool changer, turns off the spindle, and turns off the coolant pump. Program operation cannot be continued from the stopping point. This will also stop any auxiliary axes' motion. RESET must be used at least twice to remove the alarms and Start again.

3) Alarm Condition

This can occur any time an alarm comes on during program operation. Since a program cannot be restarted until RESET is pressed, a program execution cannot be continued from the stopping point. Alarms can be caused by programming errors or machine faults. Use the Graphics simulation mode to test your program first for errors.

4) Power-Off

This will stop all motors within one second but does not guarantee any conditions when the machine is powered-on again.

EMERGENCY STOP SWITCH

The EMERGENCY STOP switch is normally closed. If the switch opens or is broken, power to the servos will be removed instantly. This will also shut off the tool changer, spindle drive, and coolant pump.

NOTE: Be careful of the fact that Parameter 57 contains a status switch POF AT E-STOP. If this switch is set it will cause the control to be powered down when EMERGENCY STOP is pressed.

You should not normally stop a tool change with EMERGENCY STOP as this will leave the tool changer in an abnormal position that takes special action to correct.

Note the tool changer alarms can be easily corrected by first correcting any mechanical problem, pressing RESET until the alarms are clear, selecting ZERO RETURN mode, and selecting "AUTO ALL AXES".

If the tool changer should become jammed, the control will automatically come to an alarm state. To correct this, press the EMERGENCY STOP button and remove the cause of the jam. Press the RESET key to clear any alarms. Press the ZERO RETURN and the AUTO ALL AXES keys to reset the Z-axis and tool changer. Never put your hands near the tool changer when powered unless the EMERGENCY STOP button is pressed.

**WORK BEACONS**

The red and green work beacons located directly on top of the control arm indicate the machine status.

While a program is running normally, the GREEN beacon will be on.

The beacon will *flash* GREEN if:

- the operator selects FEEDHOLD or SINGLE BLOCK stop.

- the control is in a M00, M01, M02, M30. It will stop flashing when RESET is pressed. If the control is in an M02 or M30, and door hold override is not on, the beacon will stop flashing when the door is opened.

The beacon will *flash* RED if:

- the control encounters an alarm, such as when EMERGENCY STOP is pressed. It will stop flashing when RESET is pressed to clear all alarms.



3.16 PART PROGRAM STORAGE AND EDIT

When using anything other than HELP or Messages function, alphanumeric key entries are displayed along the bottom line of the CRT. This is called the data entry line. When the line contains what you want to enter, press the WRITE, ALTER, or INSERT key as appropriate.

When the HELP display is selected, the alphanumeric keys are used to select one of the topics; so they are not displayed on the data entry line of the CRT.

When the Message function is selected, the cursor is positioned on the screen and you type directly into the display.

CREATING PROGRAMS

To create a new program, you must be in the PRGRM/CONVRS display and LIST PROG mode. Enter **O** (letter, not number) and a four digit program number and press SELECT PROG. The selected program is the "Main" program and is the one you will see on the MEM and EDIT modes. Press EDIT to show the new program. A new program will consist of only the **Onnnnn** and an EOB (;). All further entries are made by typing a letter followed by a numeric value and pressing INSERT, ALTER, or WRITE. All items entered into a program are either addressed data (a letter of the alphabet followed by a number), a comment (text surrounded by parenthesis), or the End-Of-Block (EOB or ;).

The CURSOR **up** and **down** keys can be used to search for the entered value. Simply enter the value to search for on the bottom line and press the CURSOR **up** or **down** keys. The CURSOR **up** key will search for the entered item backwards to the start of the program. The CURSOR **down** key will search forward to the end of the program. Searching also works in MEM mode. If you enter a letter without a number, the search will stop on the first use of that letter with any value.

NOTE: When INSERT is pressed, the new data is put in after the highlighted (reverse video) data. The CURSOR **up**, **down**, **left**, and **right** keys are used to select the highlighted item. The PAGE UP and PAGE DOWN keys move farther distances and the HOME and END keys go to the start or end of the program. All of these keys work in EDIT, MEM, and MDI modes.

A comment can be edited without entering the entire comment again. Simply highlight the characters you wish to change, enter the new characters, and press ALTER. To add characters move the cursor to where the text is to be added, enter the new characters, and press INSERT. To remove characters highlight the characters and press DELETE. Use the UNDO button to reverse any changes. The UNDO button will work for the last nine entries.

After creating a program, the name can be very easily changed by simply altering the **Onnnnn** on the first line. If the maximum number of programs are already present, the message "DIR FULL" will be displayed and the program cannot be created. The maximum number of programs in memory is 500.

**EDITING PROGRAMS**

The EDIT mode is used to make changes to a program already in memory. If a program does not exist yet, the LIST PROG mode is used to create it. A newly created program contains only the program **Onnnnn** name and an EOB.

To enter the EDIT mode, press the EDIT mode key. The screen will display the current program. If no program file exists, program O00000 will be displayed. To change a program name, move the cursor to the existing **Onnnnn**, type in the letter "O" followed by a five digit number, such as O12345, and press the ALTER key. The upper right hand screen will display the new program number. Your data will first appear in the lower left screen and will be input to the upper screen upon pressing INSERT, ALTER, or WRITE.

To enter a program from the keypad, type in the data you wish and press the INSERT key. More than one code, such as **X**, **Y**, and **Z**, can be entered before you press INSERT. After a program is entered, you may wish to change the data. Highlight the characters you wish to change, enter the new characters, and press ALTER. To add characters move the cursor to where the text is to be added, enter the new characters, and press INSERT. To remove characters highlight the characters and press DELETE. Use the UNDO button to reverse any changes. The UNDO button will work for the last nine entries.

The CURSOR **up** and **down** keys can be used to search for the entered value. Simply key in the value being searched for on the bottom line and press the CURSOR **up** or **down** keys. The CURSOR **up** key will search for the entered item backwards to the start of the program. The CURSOR **down** key will search forward to the end of the program. Searching also works in MEM mode. If you key in a letter without a number, the search will stop on the first use of that letter with any value.

You can change to a different program while in the EDIT mode by using the CURSOR **up** and **down** keys, enter **Onnnnn** on the input line and then press the CURSOR **up** and **down** keys or the **F4** key. **Onnnnn** is the program you wish to change to.

As an option, the jog handle can be used to move the cursor during editing. Parameter 57 is used to enable this function. If enabled, the handle will act like the CURSOR **left** and **right** buttons.

Editing error messages:

Guarded Code	You tried to remove the Onnnnn from start of a program.
Bad Code	A line contained invalid data or comment over 80 characters.
Editing Error	Some previous edit was not completed; fix the problem or press UNDO.
Bad Name	Program name Onnnnn is invalid or missing.
Invalid Number	The number with an alphabet code was invalid.
Block Too Long	A block may only contain 256 characters.
No Code	An insert was done without any data to insert.
Can't Undo	May only use undo for previous nine changes.
End Of Prg	End of prog EOB cannot be deleted.

**BACKGROUND EDIT**

As a standard feature, this machine is shipped with a BACKGROUND EDIT capability. With BACKGROUND EDIT, you may edit a program in memory while any other program is being run. BACKGROUND EDIT can be enabled and disabled by Parameter 57.

BACKGROUND EDIT is selected from MEM mode when in PROGRAM DISPLAY by typing **Onnnnn** for the program you want to edit and pressing F4. If you do not enter the **Onnnnn**, you will instead get the PROGRAM REVIEW display.

While in BACKGROUND EDIT, you may perform any of the operations available in the EDIT mode. The last five lines of the CRT will, however, display the status of the running program and the top line will show the name and line number of the running program.

Selecting any other display or pressing F4 will exit from BACKGROUND EDIT. In order to list the programs that are in memory, a display function is available to view the program memory list while a program is running. This display is called LIST. It is selected by pressing the LIST PROG button while a program is running. The display is just like the LIST PROG mode display but it does not allow any send, receive, copy, select, or erase functions.

The CYCLE START button may not be used while in BACKGROUND EDIT. If the program contains an M00 stop, you must exit BACKGROUND EDIT and then press CYCLE START to resume the program.

All of the changes made during BACKGROUND EDIT are saved in a different memory area until the running program stops. This means that you can even edit the program that is running, or any of its subprograms, and those changes will not affect the running program.

The first time you select a program for BACKGROUND EDIT, you will get the message PROG EXISTS if the program is already in memory or NEW PROG if it is not. The NEW PROG message means that the program is being created and will be initially empty. In either case, you will then be able to edit that program. The second time you select a program for BACKGROUND EDIT without stopping the running program, you will get the message SECOND EDIT.

When you are in BACKGROUND EDIT and the running program finishes, the display will automatically change to the PROGRAM DISPLAY and will show the program that just finished running. To continue editing your program, you must select it with LIST PROG and then display it in EDIT mode.

BACKGROUND EDIT is not available from MDI or from DNC operating modes.

**DELETING PROGRAMS**

To delete an existing program, you must be in LIST PROG mode. The programs will be listed here by program number. Use the CURSOR **up** or **down** keys to highlight the program number, or type in the program number at the blinking cursor, then press the ERASE PROG key.

All the programs in the list may be deleted by selecting ALL at the end of the list and pressing the ERASE PROG key. Use caution when deleting single programs, and read all prompts, to ensure that ALL programs are not selected. The UNDO key will not recover programs that are deleted.

SPECIAL FUNCTION KEYS

The F1, F2, F3, and F4 buttons perform different functions depending on what display and mode is selected. The following is a quick summary of the **F_n** buttons:

- F1 In EDIT mode and PROGRAM DISPLAY, this will start a block definition.
In LIST PROG mode, F1 will duplicate a program already stored and give it a new name from the command line.

In OFFSET display, F1 will set the entered value into the offsets.
- F2 In EDIT mode, PROGRAM DISPLAY, this will end a block definition.
- F3 a. In EDIT and MDI modes, the F3 key will copy the highlighted G-code solution generated by Circular Help in calculator, into the data entry line at the bottom of the screen. This is useful when you want to use the solution developed for a circular motion. Press INSERT to add that circular motion command line to your program.

b. In EDIT and MDI modes, if a G-code solution is not highlighted, the F3 key will copy the value in the boxed calculator window into the data entry line.

c. In the calculator HELP function this key copies the value in the boxed calculator window to the highlighted data entry Trig or Circular Help.
- F4 When in EDIT mode with no program running, entering **Onnnnn** in the input line and pressing F4 will change the program being edited to **Onnnnn**.

When in MEM mode and PROGRAM DISPLAY, F4 can be pressed to select either BACKGROUND EDIT or PROGRAM REVIEW. BACKGROUND EDIT is selected by entering the program number at the input line and pressing F4. BACKGROUND EDIT can only be selected when a program is running. PROGRAM REVIEW can be selected whether or not a program is running, simply by pressing F4. If a program is running, PROGRAM REVIEW will show the running program on the left half of the screen, and allows the operator to review the program on the right half of the screen.

In the calculator HELP function, F4 uses the highlighted Trig, Circular, or Tapping data value to load, add, subtract, multiply or divide with the calculator.

**THE UNDO KEY**

A very powerful keyboard button available in this control is the UNDO button. When editing, this button will allow you to basically undo any changes or edits you have made but wish you had not. Any time you use the INSERT, ALTER, or DELETE buttons, the condition of the original block is saved and can be restored with the UNDO button. In fact, the previous nine changes can be undone in the opposite order that they were entered by pressing the UNDO button for each change that is to be backed out.

The UNDO button can be used in EDIT, BACKGROUND EDIT, and MDI. But if you change operating modes between EDIT and MDI, you cannot use the UNDO button as the list of saved data is cleared.

BLOCK OPERATIONS

Block operations can be performed on a group of one or more blocks of the program. These operations include BLOCK DUPLICATE, BLOCK MOVE, and BLOCK DELETE. Prior to a block being defined, the bottom right of the screen shows how to define a block; the F1 key is pressed when the cursor is on the first line of the block and the F2 key is pressed when the cursor is on the last line of the block.

Once a block is defined, the lower right of the screen shows how to manipulate the block; the INSERT key is used to duplicate the defined block wherever the cursor is positioned, the DELETE key is used to delete the block, the ALTER key is used to move the block, and the UNDO key cancels the block definition.

When a block is defined, the cursor is indicated by the “>” symbol and is always at the beginning of a line. When a block is copied or moved, the lines are added after the block with the cursor. Only whole command lines may be moved with the block functions.

Parts of programs can be copied from one program to another with the block copy feature. This is done by highlighting the section of code that is to be copied using the F1 and F2 keys. Once a section of code is highlighted, you then change to another program by selecting an existing program or create a new one. Cursor to the location that the previously defined block is to be inserted and press the INSERT or write key. A copy of the defined block will be inserted into the current program and the copied code segment becomes the currently-defined block. Press the UNDO key to exit the BLOCK COPY mode.

Blocks of code can be copied into an MDI program, but blocks of code cannot be copied from an MDI program into another program. You can always rename the MDI program and then copy its text to any other program in the manner described above.

**3.17 PART PROGRAM INPUT / OUTPUT****RS232 DATA INPUT / OUTPUT**

Programs are sent or received through the first RS-232 port located on the rear control box pendant side. Note that this is the top connector. All data sent or received is ASCII. In order to use this port, you will need to obtain a cable and connectors with the following wiring:

Pin #1 Shield Ground Pin #2 TXD-Transmit Data

Pin #3 RXD-Receive Data Pin #4 RTS (optional)

Pin #5 CTS (optional) Pin #7 Signal Ground

Cables for the RS-232 must be shielded

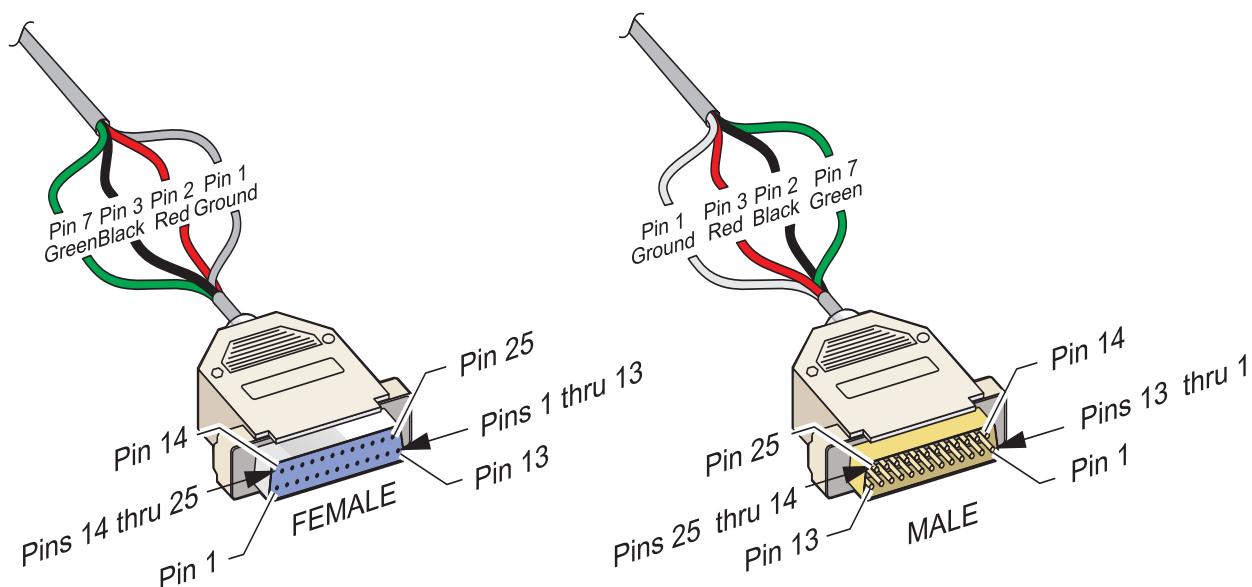
The following lists baud rate and the respective maximum cable length. This list assumes proper cable shielding and no signal boost.

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

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

115200 baud rate: 6 feet (2m) RS-232

It is possible to use RS-232 to RS-422 converters on each end of the cable to accomplish longer cables at up to 115,200 baud. Proper twisted pair wire greatly improves reliability and increases maximum distance. A pentium processor should be used for baud speed of 115,200.





All other pins are optional and are not usually used. The RS-232 connector is a DB-25 and is wired as a DTE. This means that we send data on the TXD wire and receive data on the RXD wire. If you do not understand this, your dealer will be glad to help. The simplest connection would be to an IBM PC that can be done with a standard cable made up of a DB-25 male on one end and a DB-25 female on the other. Pin 2 at one end is wired to pin 3 at the other end, pin 3 to pin 2 and pin 7 is wired to pin 7.

All RS-232 data is ASCII but the number of bits, parity and speed can be changed from settings. The number of data bits is selected with Setting 37 for either 7 or 8. Parity is selected with Setting 12 and is none, even, odd, or zero. Zero parity will always set the parity bit to 0. The data speed is selected with Setting 11.

Once the connection to your computer has been made and verified, go to the Setting page and set the baud rate, parity, number of stop bits, end of block (EOB) format, and leader parameters to match your requirements.

All programs sent to the control must begin with a line containing a single % and must end with a line containing a single %. All programs sent by the control will have these % symbols.

To receive a program, press the LIST PROG key. Move the cursor to the word ALL and press the RECV RS-232 key and the control will receive all main and sub programs until it receives a % sign indicating end of input. Please note that when using "ALL", all your programs must have an address **Oxxxxx** to be filed. If you do not have a program number, type in the program number before you press 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. An ASCII EOF character (code 04) will also terminate input. The colon character (:) may be used in place of the **O** for a program name, but it is always displayed as **O**.

When receiving RS-232 data, there is a status message at the bottom of the screen. It will update as follows:

WAITING	When you first press RECV RS-232.
LOADING XXX	When first % is received; if in XMODEM, XXX is the current block being loaded.
LOADING Onnnn	When program name is received.
RS232 DONE	When complete and last % is received.
RS232 ABORT	When anything causes abnormal stop.

There is a maximum of 500 programs stored in this control at a time.

To send a program, use the cursor as above to select the program and press the SEND RS-232 key. You can select "ALL" to send all of the programs in memory. A setting can be turned on to add spaces to the RS-232 output and improve the readability of your programs.

The synchronization protocol used to send data to slower computers is selected from the Setting 14. Setting 14 may be set to XON/XOFF, RTS/CTS, DC CODES, or XMODEM. Transmission can be stopped with either the XON/XOFF characters or the RTS/CTS wires.

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 pressing the SEND key. They can be received by pressing the RECV key.



The settings that control RS-232 are:

- | | |
|----|------------------|
| 11 | BAUD RATE |
| 12 | PARITY |
| 13 | STOP BITS |
| 14 | SYNCHRONIZATION |
| 24 | LEADER TO PUNCH |
| 25 | EOB PATTERN |
| 37 | NUMBER DATA BITS |

The EOB (semicolon) character is not normally sent by the RS-232 port. If it is received by the input port, it will cause a blank line in the program.

The format of data sent and received for parameters, settings, and offsets is the following:

```
%  
N0 Vnnnnnn  
N1 Vnnnnnn  
N2 Vnnnnnn  
. . .  
%
```

The format of data sent and received for macro variables is the same as above except there is a line N9999, and no line NO. The **N** number is the data number and **V** is the value. N0 is a CRC code that is computed by the control prior to sending the data. The N0 value is mandatory with parameters but is optional with settings and offsets. If you make a change to some saved data value and leave the old CRC, you will get an alarm when you try to load that data. With settings and offsets, you should delete the N0 line if you make changes to the saved data.

NOTE: Data will be loaded even though an alarm has been generated.

Data that is received garbled is usually converted into a comment and stored into your program while an alarm is generated. In addition, any parity errors or framing errors will generate an alarm and they will also stop the receive operation.

At the end of a send or receive function, the bottom left corner of the display will show either: "RS232 DONE" for normal completion or "RS232 ABORT" if any errors cause it to stop. The actual errors are listed in the ALARM display.

The Haas CNC serial ports optionally support the full DC1, DC2, DC3, DC4 code sequence that is compatible with paper tape readers and punches. Setting 14 is used to select this mode of operation. Setting 14 can be set to "DC CODES". When this setting is selected, the following occurs:

- 1) When sending out of the serial port, a DC2 (0x12) will precede all other data. This code is used to turn on a paper punch.
- 2) When sending out of the serial port, a DC4 (0x14) will follow all other data. This code is used to turn off a paper tape punch.



- 3) When receiving from the serial port, a DC1 (0x11 Xon) is sent first. This code is used to turn on a paper reader.

- 4) When receiving from the serial port, a DC3 (0x13 Xoff) is sent after the last % is received. This code is used to turn off a paper tape reader.

Note that the Setting 14 selection XON/XOFF is similar to the "DC CODES" selection. Both of these settings use the DC1/DC3 XON/XOFF codes to start/stop the sender when data is received too fast. When DC CODES is selected for Setting 14 (synchronization), serial port #1 will transmit an XON (DC1) if a character has not been received for five (5) seconds.

XMODEM may also be selected in setting 14. It is a receiver-driven communications protocol that sends data in blocks of 128 bytes. Setting **synchronization** to XMODEM gives your RS-232 communication an added level of reliability because each block is checked for integrity. If the receiver determines that the most recently sent block is in error, it will request that the sender try to send the block again.

In order to use XMODEM, parity must be none, and RS-232 data bits must be set to 8. Also, the computer that is sending the data must be equipped with a communications package that supports the XMODEM protocol. It must be set to XMODEM to operate.

This version of XMODEM supports **checksum** verification only. Also, 512 bytes of memory must be available before using XMODEM with DNC.

WARNING!

One of the biggest causes of electronic damage is a lack of a good earth ground on both the CNC and the computer that is connected by RS-232. A ground fault condition (i.e., a lack of good ground on both) will damage the CNC or the computer, or both.

Port #2 on the side cabinet is dedicated to auxiliary axes communication. See Auxiliary Axis Control for more information.

DIRECT NUMERICAL CONTROL (DNC)

As a standard feature, this machine is shipped with a DNC capability. With DNC, there is no limit to the size of your CNC programs. The programs are directly executed by the control as they are sent over the RS-232 interface. Note, that this is the first serial port or the top connector. Do not confuse DNC with RS-232 uploading and downloading.

If you wish to use DNC, it is enabled by Parameter 57 and Setting 55.

NOTE: Floppy disk DNC is selected by entering the floppy file name and pressing MDI a second time when already in MDI mode. Do not press MDI **three** consecutive times or a "DISK ABORT" will result.



When enabled, DNC is selected by pressing MDI a second time when already in MDI. DNC mode will not be enabled unless there is a minimum of 512 bytes of user memory available. When DNC is selected, the PROGRAM DISPLAY will show:

WAITING FOR DNC...

This means that no DNC data has been received yet and you may begin sending data. You must start sending the program to the control before the CYCLE START button can be pushed. After the beginning of the program is seen by the control, the display will show part of the program and a message at the bottom, left of the CRT will show DNC PROG FOUND. After the program is found, you may push CYCLE START just like running any other program from Memory.

If you try to press CYCLE START before receiving a program, you will get the message: NO DNC PROG YET. The reason for not allowing the command of CYCLE START before receiving the DNC program is for safety. If the operation is allowed to start from a remote location, the operator may not be present to ensure that the machine is operating safely.

While a DNC program is executing, you are not allowed to change modes. You must first press RESET to stop the program.

When the end of the DNC program is received, the message DNC END FOUND is displayed. When the DNC program is finished running, the PROGRAM DISPLAY will show the last few lines of the program. You must press RESET or exit the DNC mode before you can run any other programs. If you try to press CYCLE START before RESET of the previous DNC, you will get the message: RESET FIRST.

DNC supports Dripmode. The control will execute one block at a time from the RS-232 port. Each block entered will be executed immediately with no block lookahead buffering. The exception is that Cutter Compensation requires three blocks of motion commands to be buffered prior to a compensated block being executed.

There are several restrictions on what can be in a DNC program. An M30 is not allowed as it is not possible to start over at the beginning. Canned cycles G70, G71, G72, and G73 cannot be programmed while in DNC, since they require the control to look ahead.

The program must begin with a % just like any other program sent over RS-232 and the program must end with a %. The data rate selected for the RS-232 port by settings 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 be stopped in a cut when you might otherwise expect continuous cutter motion. The highest standard RS-232 data rate available is 115,200 bits per second.

It is recommended that DNC be run with Xmodem or parity selected because an error in transmission will then be detected and will stop operation of the DNC program without crashing. The settings page is used to select parity. The recommended RS-232 settings for DNC are:

9600 or 19200 BITS PER SECOND
EVEN PARITY
1 STOP BIT
XON/XOFF



Full duplex communication during DNC is possible by using the G102 command or DPRNT to output axes coordinates back to the controlling computer. When DNC is running, BACKGROUND EDIT is not available.

To run DNC in graphics, you must select DNC first, then go to graphics display and send your program to the CNC.

FLOPPY DISK OPERATION

All files must be on MS-DOS formatted 1.44M floppy disks and must reside in the root directory. Parameter 209 DISK ENABLE must be 1.

NOTE: In order to enable the floppy disk drive, an unlock code must be entered. If necessary contact the Service Department for more information.

NOTE: Use an empty (containing no other files) floppy disk for faster operation.

All programs must begin with a line containing a single % and must end with a line containing a single %. All programs saved by the control will have these % symbols.

Programs may be loaded and saved from the floppy disk. To **LOAD** a program, press the LIST PROG key with PRGM selected. Enter the floppy disk file name and press F3 and the control will receive all main and sub programs until it receives a % sign indicating end of input. Please note that when using "ALL", all your programs must have an address **Oxxxx** to be filed. An ASCII EOF character (code 04) will also terminate input. The colon character (:) may be used in place of the O for a program name, but it is always displayed as O.

When loading floppy disk data, there is a status message at the bottom of the screen. It will update as follows:

LOADING Onnnnn	When program name is received.
DISK DONE	When complete and last % is received.
DISKABORT	When anything causes abnormal stop.

There is a maximum of 500 programs stored in this control at a time.

To **SAVE** a program to floppy disk, press the LIST PROG key with PRGM selected. Enter the floppy disk file name, use the cursor as above to select the program, and press the F2 key. You can select "ALL" to send all of the programs in memory.

NOTE: To load or save a program numbered greater than 9000, Setting 23 must be off.

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

If an EOB (semicolon) is loaded, it will cause a blank line in the program.



The format of data sent and received for settings, offsets, and parameters is the following:

```
%  
N0 Vnnnnnn  
N1 Vnnnnnn  
N2 Vnnnnnn  
. . .  
%  
%
```

The **N** number is the data number and **V** is the value. N0 is a CRC code that is computed by the control prior to sending the data. The N0 value is mandatory with parameters but is optional with settings and offsets. If you make a change to some saved data value and leave the old CRC, you will get an alarm when you load that data. With settings and offsets, you should delete the N0 line if you make changes to the saved data.

Data that is received garbled is usually converted into a comment and stored into your program while an alarm is generated. Errors generating an alarm may also stop the receive operation.

To get a **DIRECTORY LISTING**, select the PRGM/LIST PROG mode, and press F4. This will generate a Disk Directory Listing that will be saved in program 0xxxx where xxxx is defined in parameter 227. The default value is 8999.

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

At the end of a save or load function, the bottom left corner of the display will show either: "DISK DONE" for normal completion or "DISK ABORT" if any errors cause it to stop. The actual errors are listed in the ALARM display.



3.18 DRAWTUBE OPERATION

HYDRAULIC DRAWTUBE

The hydraulic unit provides the pressure necessary to chuck or clamp a part. The hydraulic pump pressure is proportional to the clamping force exerted.

Clamping Force Adjustment Procedure

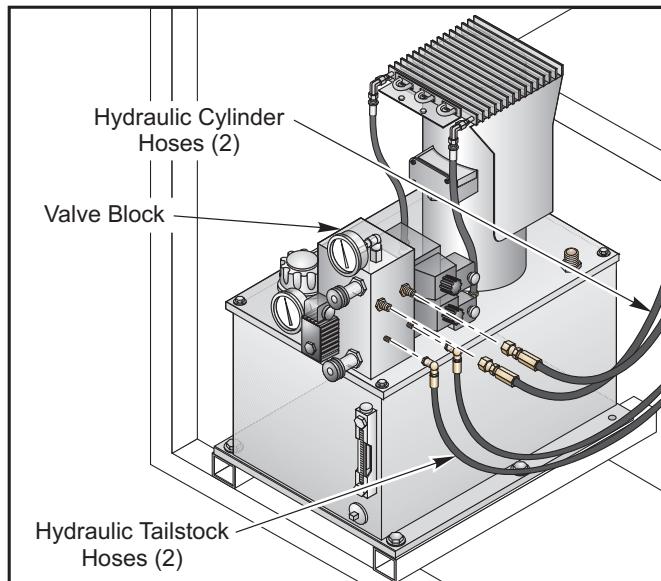
1. Go to Setting 92 on the Settings page and choose either 'I.D.' or 'O.D. Clamping'.

NOTE: The control will not allow Setting 92 to be changed while a program is running.

2. If it is necessary to change the clamping force on the part, refer to Figure below and the Drawtube Force Chart on the next page to adjust the hydraulic pump pressure.

NOTE: The maximum and minimum hydraulic pressures are 500 psi and 50 psi, respectively.

- Loosen the locking knob at the base of the adjustment knob
 - Turn the adjustment knob until the gauge reads the desired pressure
 - Tighten the locking knob
3. Clamp the part by placing it in the chuck and depressing the foot pedal. To unclamp, depress the foot pedal again.



Hydraulic unit pump.

**HYDRAULIC DRAWTUBE FORCE CHART**

The following chart shows the relationship between hydraulic pressure and the amount of force applied to the drawtube.

DRAWTUBE FORCE (LBS)	HYDRAULIC PRESSURE (PSI)			
	SL-20	SL-20BB, SL-30	SL-30BB, SL-40	SL-40BB
1800 ¹	77	63		
2700	116	95	52	50
3600 ²	155	126	69	67
4500	194	158	86	84
5400 ³	232	190	103	100
6300	271	222	121	117
7200	310	253	138	134
8100	348	285	155	151

¹ Normal for 5C collets

² Recommended max. for 5C collets

³ Normal for chucks

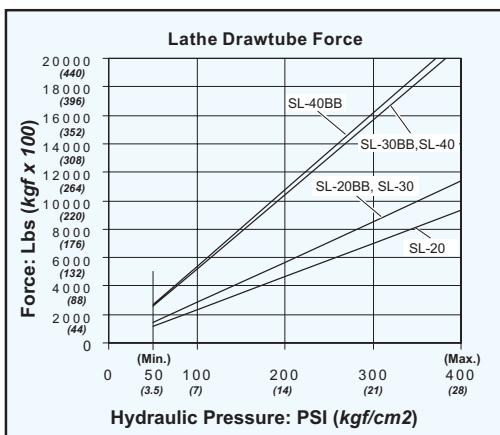


Chart of hydraulic pressure vs. hydraulic drawtube force

WARNING

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 regarding the chuck and workholding procedures.

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

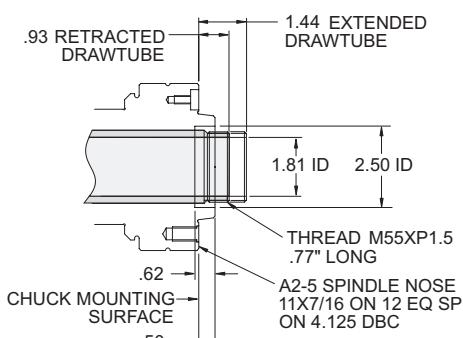
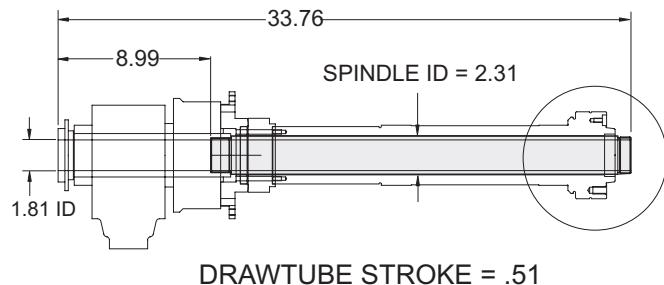
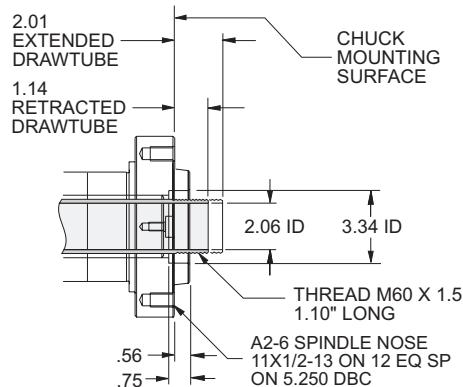
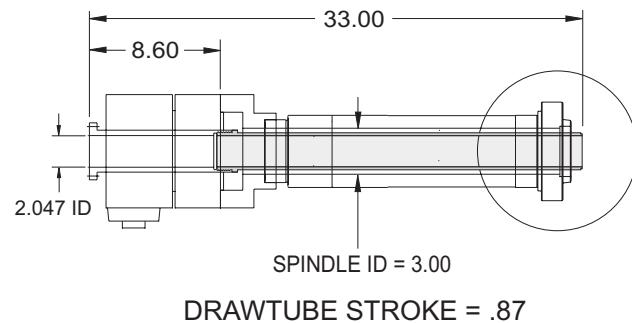
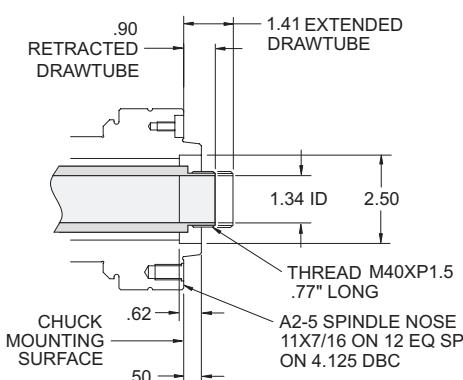
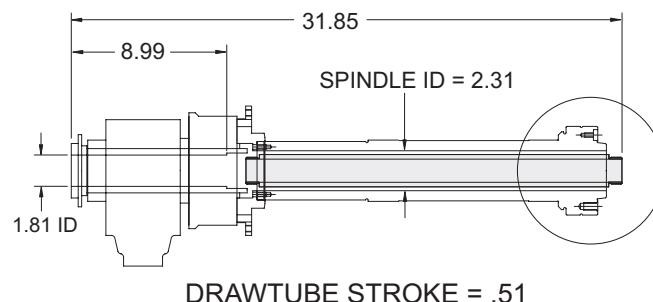
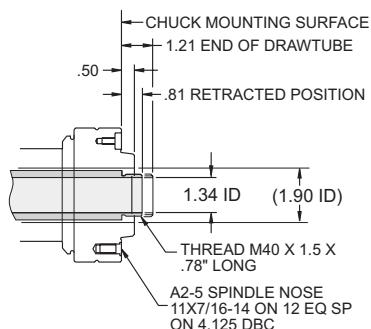
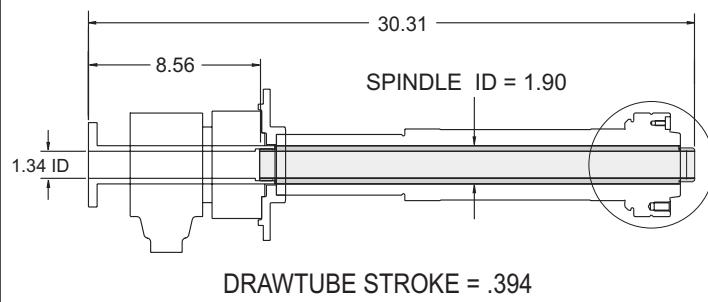
Chuck jaws must not protrude beyond the diameter of the chuck.

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

Do not exceed rated chuck RPM

Higher RPM reduces chuck clamping force

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

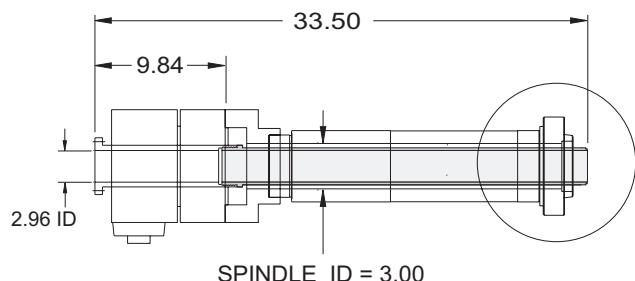
**DRAWTUBE SPECIFICATIONS****LATHE DRAWTUBE SPECIFICATIONS****SL-10 MAIN SPINDLE****SL-20/TL-15 MAIN SPINDLE****SL-20 7K SPINDLE (6" HYDRAULIC CLOSER)
LMC PN ZKP125/46-13****SL-20 7K SPINDLE (5" HYDRAULIC CLOSER)
LMC PN ZKP100/34-10**

DIMENSIONS IN INCHES

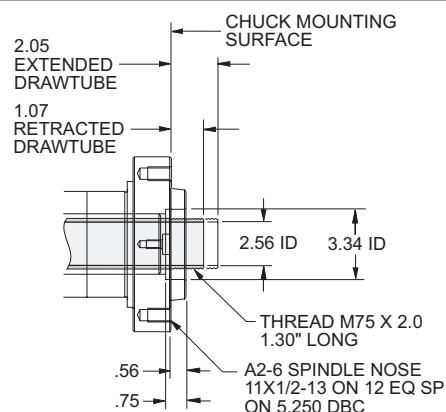


LATHE DRAWTUBE SPECIFICATIONS

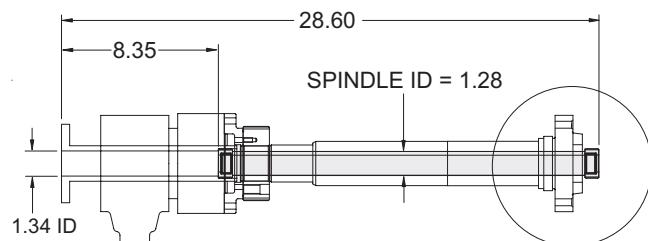
SL-30, SL-20 BB, TL-15 BB MAIN SPINDLE



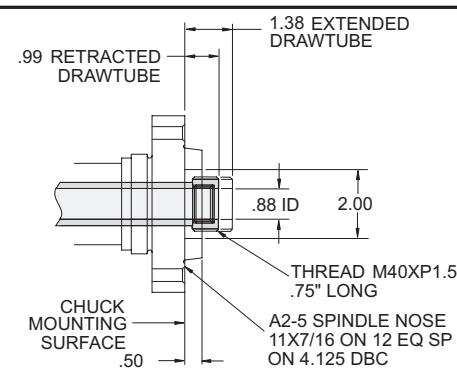
DRAWTUBE STROKE = .98



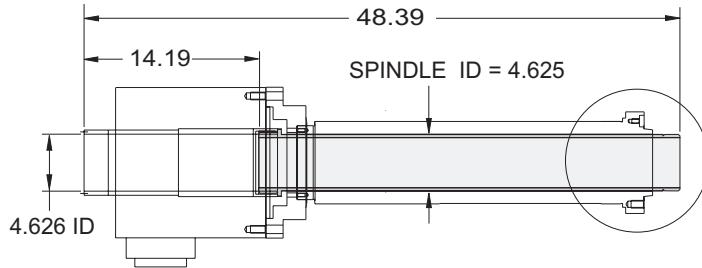
TL-15 SUB-SPINDLE



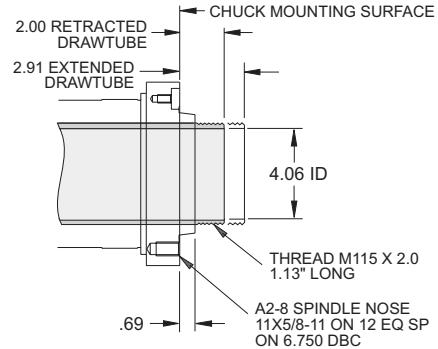
DRAWTUBE STROKE = .39



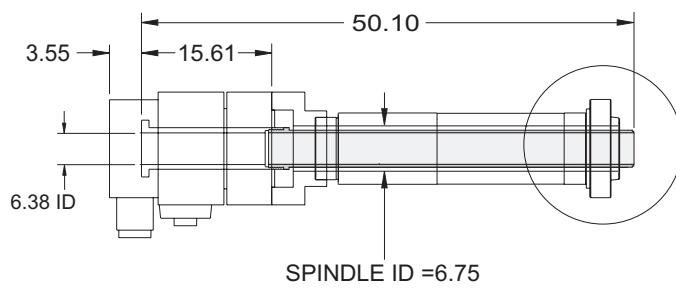
SL-40, SL-30 BB MAIN SPINDLE



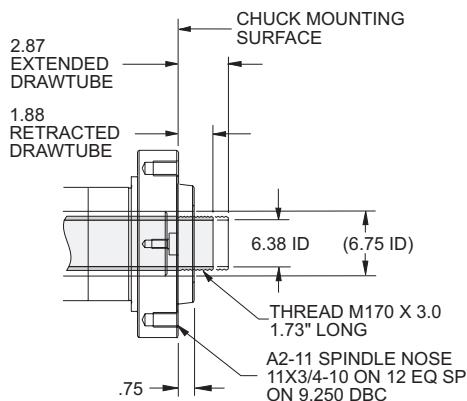
DRAWTUBE STROKE = 1.18



SL-40 BB MAIN SPINDLE



DRAWTUBE STROKE = .90



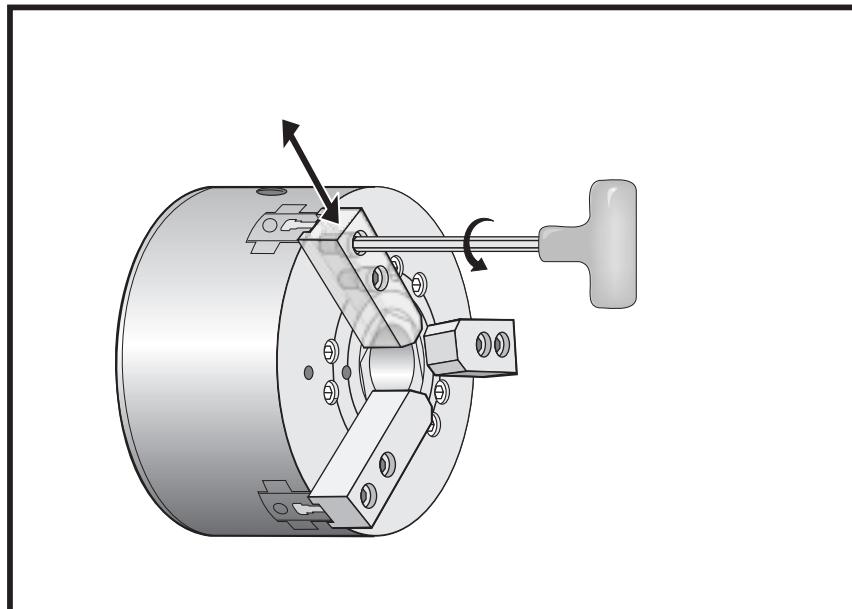
DIMENSIONS IN INCHES

**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.

Consult the Lathe Operator's Manual for specific clamp pressures at a specific stroke (and RPM).



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.

**3.19 MACHINE OPERATION****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 DRY RUN speed selected from the JOG speed buttons. The bottom of the screen will display the rate as 100, 10, 1.0 or 0.1 inches per minute.

DRY RUN cannot be turned on while a program is running. It can only be turned on or off when a program has completely finished or is reset. The first push of the DRY RUN button turns on this function and the second push will turn it off again. DRY RUN will still make all of the requested tool changes. The speed used in DRY RUN can be changed at any time and the operator can then check that the motions that are programmed are exactly what were intended. Note that Graphics mode is just as useful and may be even safer since it does not begin moving the machine before the program is checked.

DISPLAYS

You can select any of the following displays using the eight DISPLAY select buttons:

PRGRM / CONVRS To show or edit the program selected. Also used in Quick Code applications.

POSIT To show the axes positions.

OFSET To show or enter working offsets.

CURNT COMDS To show current commands and times.

ALARM / MESGS To show alarms and user messages.

PARAM / DGNOS To show parameters and diagnostic data.

SETNG / GRAPH To show or enter settings OR to select graphics simulation mode.

HELP / CALC To show the help data and calculator.

In addition, when a program is running, you may press LIST PROG to select a list of the programs in memory. This is helpful in determining which programs can be edited in BACKGROUND EDIT, which is selected from the PROGRAM DISPLAY.

**MESSAGES**

The CRT will ALWAYS show some of the current conditions selected in the control. These are fixed status displays that describe the condition of the machine. The following conditions are displayed on the screen:

- The selected display in the top left corner,
- The selected mode in parentheses,
- The presently selected program in the top right corner,
- The most recent line number in the top right corner,
- Up to 18 lines of variable display data,
- Any of the following conditions that apply:

ALARM	Blinking in lower right corner when alarm occurs.
BLKDEL	BLOCK DELETE is turned on.
BUF	When next block is ready in continuous path.
C CLAMPED	Spindle has been clamped by an M14 command (Live Tooling option)
DOOR HOLD	An open door has stopped the program.
DRYRUN	DRY RUN is selected.
DWELL	When a G04 is being performed.
FEED	When a feed motion in progress.
FEED %	Feed rate override is active.
FEED HOLD	FEED HOLD is active.
OPTSTP	OPTIONAL STOP is turned on.
RAPID %	Rapid override is active.
RUNNING	When a program is running.
SINGBK	SINGLE BLOCK is turned on.
SINGBK STOP	When a program is stopped in SINGLE BLOCK.
SPIND %	Spindle speed override is active.

The following error messages are received when the wrong button is pressed:

ALARM ON	Cannot start an operation until alarms are reset.
ALTER	The selected text can now be altered.
AUXAXIS BUSY	One or more auxiliary axes are busy in an operation.
BAD CODE	Code entered is not understood.
BAD NAME	Program name entered is not Onnnnn .
BLOCK TOO LONG	Block being edited would be too long.
CAN NOT COPY	The selected program can not be copied.
CAN'T RENAME	The selected program cannot be renamed.
CAN'T UNDO!	The last function can not be undone.
C CLAMPED	The spindle has been clamped by an M14 command.
CHUCK UNCLAMPED	The hydraulic chuck is unclamped.
CNVEYR DISABLED	Conveyor has been disabled by parameters, or conveyor motion was commanded while disabled.
COOLANT OFF	Coolant pump is off.
COOLANT ON	Coolant pump is on.
DEL ALL (Y/N) ?	Do you want to delete all, yes or no ?
DELETE	Deleting text as requested.
DIR FULL	Maximum number of programs exceeded.
DIR NOT FOUND	Directory of floppy disk not found.
DISABLED AXIS	Requested axis has been disabled, and cannot be jogged.



DISKABORT	Something caused an abnormal stop.
DISK DIR	Directory of programs on floppy disk.
DISK DONE	When complete, and last % is received.
DISK FOUND	Floppy disk drive is present.
DISK NAME REQ	Program name required for floppy disk file.
DISK NOT ENBLED	Floppy disk drive has not been enabled by parameters.
DISK NOT IN DRV	No disk in floppy disk drive.
DISK NOT RDABLE	Floppy disk cannot be read by control.
DISK READ	Reading from floppy disk drive.
DISK WRITE	Writing to floppy disk drive.
DISK WRT PROTECT	Cannot save to floppy disk, it is write protected.
DISPLAYS OFF	Indicates that M76 was used to turn off displays.
DIVIDE BY ZERO	An attempt was made to divide by zero in calculator mode, or system error exists.
DNC END FOUND	The end of a DNC program has been found.
DNC PROG READY	The DNC program is ready to run.
DOOR IS OPEN	The door is open; some functions not allowed.
DRY RUN OVERRIDE	Dry Run mode has been overridden.
EMPTY PROG	No program found between the % and %.
END FOUND	End of program has been received.
END OF PROG	The program being run has completed.
ENTER DIAM	Enter the desired diameter to be cut.
EOF FOUND	End of file has been found.
EXIT BG EDIT	Exiting Background Edit mode.
FILE NOT FOUND	The requested file was not found.
FUNCTION ABORT	Requested function has been aborted.
FUNCTION LOCKED	Function attempted is locked from settings.
GUARDED CODE	Cannot remove Onnnnn at start of program.
INSERT	Selected text now being inserted.
INSUF DSK SPACE	Insufficient disk space to save the selected file.
INVALID AXIS	Selected axis is invalid.
INVALID NUMBER	Number entered is invalid.
JOG COMD	An axis jog has been commanded.
LOADING...	Reading programs or data from RS-232.
LOW COOLANT	Coolant tank level is low.
M00 AFTER TC	Cannot have an M00 after an TC.
MACHINE LOCKED	Front panel has been locked by setting.
MACRO LOCKED	Macros 9000 to 9099 are locked by setting.
MEMORY FULL	Memory space is full.
MEMORY LOCKED	Memory lock is set in settings.
NEW PROGRAM	A new program may be entered.
NO DNC PROG YET	Attempted to start program before it was completely received.
NO DISK FOUND	Cannot find the floppy disk drive.
NO INPUT	Cannot alter until something has been entered.
NO NAME ENTRY	No file name has been entered.
NO PROG YET	Cannot press Cycle Start until a program is received.
NO ZERO X	Cannot run machine until search for zero is complete on X-axis.
NO ZERO Z	Cannot run machine until search for zero is complete on Z-axis.
NOT AVAILABLE	Function requested is not available at the present time.
NOT FOUND	Item not found during search in editor.
NOT IN DRYRUN	The function requested applies to DRY RUN, but control is not presently in that mode.
O CODE ONLY	An O code (program name) must be entered.
ONE BIT ONLY	Only 0 or 1 is accepted to alter a parameter switch.



ONE PROG ONLY	Program name being selected cannot be ALL.
OVERWRITE (Y/N)	Do you want to overwrite the file, yes or no ?
PLEASE WAIT	Wait until spindle is stopped.
PROG EXISTS	Cannot rename to an existing program.
PROG NOT FOUND	Requested program not in memory.
PROG READY	Program has been received and is ready to run.
PROGRAM END	Cannot remove last EOB in program.
PROGRAM IN USE	Program is already in use.
RANGE ERROR	Data being entered is outside of the valid range.
RESET FIRST	Must press RESET before performing this function.
RIGID TAP	Rigid tapping is being performed.
RS-232 ABORT	RS-232 was aborted by operator action.
RS-232 DONE	RS-232 operation is complete.
RS-232 ERROR	RS-232 error (shown in alarms).
SEARCHING...	Searching program for requested text or G code.
SEL HI GEAR	High gear selected in program.
SEL LOW GEAR	Low gear selected in program.
SENDING OFFSET	Sending offsets via RS-232.
SENDING PARS	Sending parameters via RS-232.
SENDING SETTING	Sending settings via RS-232.
SENDING VARS	Sending variables via RS-232.
SENDING...	RS-232 output is in process.
SERVO IS OFF	When servos are off , you cannot start a program.
SERVO IS ON!	Parameter change was attempted with servo on. This is dangerous!
SPEED COMD	A spindle speed must be commanded.
SPINDLE CCW	Spindle is turning counterclockwise.
SPINDLE CW	Spindle is turning clockwise.
SPINDLE HIGH	Spindle is in high-gear.
SPINDLE IN USE	Spindle is being controlled by program - manual controls not available at this time.
SPINDLE LOCKED	Spindle orientation is complete, and it is locked in place.
SPINDLE LOW	Spindle is in low gear.
SPINDLE ORI	Spindle orientation is in progress.
SPINDLE STOP	Spindle is not turning. STRING TOO LONG. The text being entered is too long.
SYSTEM ERROR	Call your dealer.
TOOL OVERLOAD	Cutting tool is overloaded.
TURRET IN USE	The turret has already been commanded.
WAIT OR RESET	Cannot perform requested function until program finishes or RESET is pressed.
WAIT...	Wait for the function to be performed.
WAITING...	Waiting for RS-232 input.
WRONG MODE	Function requested is available only in another mode.
X IN THE WAY	Entering a HOME G28 causing the tool turret to enter the tailstock restricted zone

And the following responses, only when in graphics mode:

CIRCULAR	A circular motion is being performed.
LINEAR	A linear motion is being performed.
M30 FOUND	End of program found and execution stopped.
NO ZOOM IN 3D	Zoom not allowed in 3D graphics mode.
RAPID	A rapid motion is being performed.



In addition, the CRT display can show one of the following eight types of data in the 18 lines of variable display:

Program Displays:

The PROGRAM DISPLAY is used to show your program while in either MEM, EDIT, or MDI modes.

Position Display:

The POSITION DISPLAY is used to select the **X** and **Z** axes positions in any of several coordinate systems. The PAGE UP and PAGE DOWN keys select between these.

Offsets Display:

The OFFSETS DISPLAY is used to enter and display tool wear offsets, tool shift offsets, and work offsets. The PAGE UP and PAGE DOWN keys select between these.

Current Commands Display:

The CURRENT COMMANDS DISPLAY is used to display the Program Command Check, the Current Commands, Running Timers, Tool Life Timers, Tool Load Monitor, and Axis Load Monitor. The PAGE UP and PAGE DOWN keys select between these.

Alarms / Messages Displays:

The ALARMS/MESSAGES DISPLAY is used to display alarms and to enter and display user messages. The second push of the ALARM button will select messages display. The CURSOR **up** and **down** buttons will display additional alarms if there is more than will fit on one page.

Parameters / Diagnostics Displays:

The PARAMETERS DISPLAY shows all of the machine dependent control parameters and the Diagnostic data. The second push of the PARAM DGNOS button will select the diagnostic display. The PAGE UP and PAGE DOWN buttons will select additional data for display.

Settings / Graphics Displays:

The SETTINGS DISPLAY is used to display and change user controlled parameters. The second push of the SETNG GRAPH button will select the Graphics display. The cursor and PAGE UP and PAGE DOWN buttons will select additional settings.

Help / Calculator Displays:

The HELP DISPLAY shows a mini-manual on the CRT along with a directory of available help information. Each alphabet button will select a different topic within the HELP display. The second push of the HELP button will select the Calculator display. The PAGE UP and PAGE DOWN buttons will select different calculator functions.



PROGRAM DISPLAYS

The PROGRAM DISPLAY is used to show a program being edited in EDIT mode or a program being run in MEM. In MEM mode, there is also a PROGRAM REVIEW display available.

The PROGRAM DISPLAY uses 18 lines of the text display area of the CRT to show the command blocks of a CNC program. The display is 40 positions wide and blocks that are longer than 40 positions are continued on the next line of the display.

The PROGRAM REVIEW function is available whenever a program is being run. This allows you to review the program that is running. This is selected by pressing F4 while in MEM mode and PROGRAM DISPLAY. The screen is changed to an 80 column display with the normal MEM display on the left and PROGRAM REVIEW on the right. The CURSOR and PAGE UP and PAGE DOWN keys can be used to change the display on the right to a different part of the program. The display on the left will show the progress of the running program. To exit PROGRAM REVIEW, select any other display.

While you are running a program, the BACKGROUND EDIT function is available as a standard feature. BACKGROUND EDIT allows you to edit any named program in memory while any program is being run in memory. BACKGROUND EDIT is selected from MEM mode in PROG display by entering **Onnnnn** with the program number and pressing F4. The display will change to the selected program while still running the first program. BACKGROUND EDIT is enabled by parameters if it is available in your machine.

POSITION DISPLAYS

The following are the five position displays in this control:

Home Page

This display shows the four displays simultaneously in small characters. The PAGE UP and PAGE DOWN keys will change displays. The other displays are shown in large characters. The last display selected will be shown in CURNT COMDS and SETNG/GRAFH displays when they are selected. In this display, any axis that is at the zero position will be highlighted.

Operator Display

This display is for the operator/setup person to use as desired, and is not used by the control for any positioning functions. In JOG mode, and with this display selected, the ORIGIN button can be used to set the zero position. This display will then show position relative to the selected zero position.

Work Display

This display shows how far the tool is away from the **X** and **Z** zero of the programmed part. On power up, it will display the value in work offset G54 automatically. It can only be changed by G54 through G59, G110 through G129, or by a G92 command. The machine uses this coordinate system to run the part.

Machine Display

This display is the machine coordinate system that is automatically set on power up and the first ZERO RET. It cannot be changed by the operator or any work coordinate systems, and will always show the distance from machine zero. It can be used by a non-modal G53 command.

Distance To Go

This display is an incremental display that shows the travel distance remaining before the axes stop. When in the ZERO RET mode, this display shows a diagnostic value. When in JOG mode, this display shows the total distance jogged. In rigid tapping, this number decreases to zero at the bottom of the hole and then increases again as the reverse stroke occurs.

**TOOL OFFSETS DISPLAY**

Tool geometry, tool wear, and work coordinates are entered in the offsets display. Both FANUC and YASNAC style offsets are supported. The appearance of the tool offset display will change slightly depending on whether Setting 33, COORDINATE SYSTEM, is set to YASNAC or FANUC. Tool offsets can be viewed or set from within the OFFSETS display.

There are three major sections of the OFFSETS display. If Setting 33 is set to YASNAC, TOOL SHIFT, TOOL WEAR, AND WORK ZERO OFFSET pages are available. If Setting 33 is set to FANUC, TOOL GEOMETRY, TOOL WEAR, and WORK ZERO OFFSET pages are available. There are 50 geometry (shift) offsets available, 50 wear offsets available, and 27 work offsets available.

TOOL GEOMETRY values are FANUC's method of compensating for various tool lengths. The tool geometry offsets are numbered from 1 to 50. Each offset contains values for the distance from machine zero to the tip of a tool when the tool is positioned at (X0, Z0) in the current work coordinate. The values placed in the TOOL GEOMETRY page are values that will not change for the extent of a job. There is a geometry for the X and Z axes, along with tip radius and direction. Each tool is typically assigned a different offset. Refer to the "Setup Procedures" section for instructions on the automatic setting of TOOL GEOMETRYs. When specifying a tool in a T address code, the third and fourth integers of the T code are associated with tool geometry. For FANUC, the third and fourth integer of the T address code simultaneously select both tool geometry and wear for the tool indicated by the first and second integer.

TOOL SHIFT values are the YASNAC version of FANUC's TOOL GEOMETRY. The tool shift offsets are numbered from 51 to 100. The tool shift page is intended for data that will not change for the extent of a job run. It contains shift offset values for X and Z along with tool radius and tip data that are used for tool radius compensation. The G50 is used to apply TOOL SHIFT offsets. Each tool should be assigned a different TOOL SHIFT offset, and the TOOL SHIFT offset should be applied with a G50 command prior to changing to that tool. Refer to the "Setup Procedures" section for instructions on the automatic setting of TOOL SHIFTS.

TOOL WEAR values are intended for recording minute adjustments to offset that are required to compensate for normal wear during the course of a job. Wear offset values are provided for X and Z and tool radius. Tool wear values are numbered 1 through 50. Tool wear offsets are applied by placing a number from 1 to 50 in the third and fourth integer of a T address code. Using 00 for a wear offset in a T address code cancels wear offsets in both YASNAC and FANUC coordinates.

Tool wear is added to the current work coordinate or tool shift such that the tool position is shifted the additional amount specified by the tool wear. Tool wear is set and used as a diameter value. The initial value is entered onto the shift page by the setup person. During operation, the operator makes minor wear changes on the wear page. This method allows the operator to see actual tool wear by limiting it to the wear page. The initial offset values can be entered on the shift page automatically when using the X DIA MESUR or Z FACE MESUR keys during setup procedures.

WORK ZERO OFFSET values are used only when Setting 33 is set to FANUC. The work coordinates are G54 through G59, and G110 through G129. Work zero offsets can be accessed by pressing PAGE UP in the OFFSETS display until the WORK ZERO OFFSETS become visible. Fanuc style work coordinates can be used to shift all tools by the same amount regardless of their individual geometries. G54 is the default work coordinate when the control is powered on. If you do not intend to use work coordinates in a job, you should set all axes of the G54 work coordinate to zero and ensure that the control is using G54.



PAGE DOWN in the OFSET display will go through all 50 possible tool shift or geometry offsets and then change to the work zero offsets. After all 27 work offsets are displayed PAGE DOWN will display all 50 possible tool wear offsets.

Searching for and highlighting a specific offset can be accomplished by entering the desired offset number and then pressing the DOWN arrow key. Entering 1 through 50 will search for WEAR offsets. Entering 51 through 100 will search for SHIFT and GEOMETRY offsets. For example, when in FANUC coordinate mode you would search for geometry offset number 3 by entering 53 followed by a down arrow key press.

When entering offsets, pressing WRITE after keying in a numeric value will cause the new value to be added to the old value. Pressing F1 will set the offset to that value. This allows small adjustments to the offsets. Entering a negative value and pressing WRITE will decrease the value of the offset.

Cutter compensation is controlled by G41 and G42 and the selected tool radius offset. Positive values for cutter compensation work normally. Negative values for cutter compensation cause the opposite side cutter compensation to be used. This means that a negative G41 will work the same as a positive G42 with the same number.

Offsets may be sent and received with the RS-232 port. Refer to the "Data Input/Output" section for more information on how to do this.

CURRENT COMMANDS DISPLAY

The following are the seven current command displays in this control:

- Program Command Check (Home Page),**
- Current Display Command,**
- Operation Timers,**
- Macro Variables,**
- Tool Life Timers,**
- Tool Load Monitor,**
- and Axis Load Monitor.**

The PAGE UP and PAGE DOWN keys are used to select among these displays.

Program Command Check Display

This display, which is the Home Page for the Current Commands Display, shows a current overview of the important commands. It shows the programmed spindle speed (**Snnnnn**), the spindle speed commanded to the spindle drive (**CMDnnnnnn**), and if Parameter 278 bit DISPLAY ACT is set to 1, the actual encoder spindle speed (**ACTnnnnnn**). In addition, this display shows the CW, CCW, or stopped command being sent to the spindle. The current gear is also displayed, as either HIGH GEAR, LOW GEAR, or NO GEAR.

This display also shows the position of the axes. The coordinates displayed (operator, work, machine, or distance to go) are selected using the cursor **up** and **down** keys.



Current Display command

This display shows all of the alphabetical address codes (i.e. G, M, S) and their current value. These values may not be changed in this display. The default value is shown for the address codes that are not being used in the current program.

Macro Variables Display

This display shows a list of the macro variables and their present values. As the control interprets a program, the variable changes are displayed on this page and the results can be viewed. The variables may be modified in this display. For more information on this display, refer to the "Macros" section of this manual.

Operation Timers Display

This display shows the current power-on time, cycle start time, and the feed time. These times may be reset to zero by using the Cursor **up** and **down** buttons to highlight the desired title and pressing the ORIGIN button.

Listed below these times are two M30 counters that 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. Both counters are increased when an M30 is operated.

Tool Life display

This display shows the time the tool is in feed, the time the tool is selected, and the number of times the tool has been used. This information can be used to assist in predicting tool life. The values in this display can be reset to zero using the Cursor and ORIGIN buttons. This is done by putting the cursor on the title line, and pressing ORIGIN to zero all of the data in that column.

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, and if the number for a tool is not zero, an alarm will be generated when that count is reached. This number can be changed by the operator. Alarm 362 is generated when the count is reached, and may be cleared with RESET.

Tool Load Monitor and Display

With the tool load display, the operator can enter the maximum load that is expected for each tool, and when this load is exceeded in a feed, a certain action will be taken. This display provides for the entry of this alarm point and also displays the largest load that tool has seen in any previous feed.

The tool load monitor function operates whenever the machine is in a feed operation (G01, G02, or G03) and the machine is not in constant surface speed mode (G96). The values entered into the tool load display are checked against the actual spindle motor load. If the limit is exceeded, the tool overload action specified in Setting 84 (alarm, feedhold, beep, or Autofeed) will be taken. If "alarm" is selected and the limit is exceeded, Alarm 174, "Tool Load Exceeded", will be generated. This alarm will stop the axis motors and the spindle motor, turn off the coolant, and disable the servos.

If, during a feed the load exceeds the tool limit and the AUTOFEED feature, is selected, it will automatically override the feed rate (reduce it) down to the percentage specified by parameter 301 (i.e. 1%) at the rate specified by parameter 300 (i.e. 20% per second). If the tool load later falls below 95% of the tool load limit percentage, the AUTOFEED feature will automatically override the feed rate (increase it) back to the feed rate that was in effect at the start of the feed at the rate specified by parameter 299 (i.e. 10% per second). These adjustments will be made in 0.1 second increments.

**M19 ORIENT SPINDLE**

Viewed in the Current Commands Tool Load screen. An M19 will orient the spindle to the zero position. A P value can be added that will cause the spindle to be oriented to a particular position (in degrees.) The position specified by the last M19 will be displayed when parameter 278 bit 24 LIVE TOOLING is set to 1.

SUB SP RPM CMD (Live Tooling only)

Viewed in the Current Commands Tool Load screen. The last commanded Live Tooling Drive RPM specified by an M133 or M134 is displayed when parameter 278 bit 24 LIVE TOOLING is set to 1.

BALL SCREW TEMPERATURES

When equipped with the Temp-Track option, the X and Z ball screw temperatures are now displayed below the axis load displays when parameter 266 or 268 (respectively) bit 9 TEMP SENSOR is set to 1.

Axis Load Monitor

Axis load is 100% to represent the maximum continuous load. Up to 250% can be shown, and above 100% can lead to an axis overload alarm.

Periodic Maintenance Screen

The periodic maintenance page (titled SCHEDULED MAINTENANCE and accessed by pressing PAGE UP or PAGE DOWN from the Current Commands screen) allows the operator to activate and deactivate a series of checks (see list below). 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 to the right. If an item is deactivated, “—” will be displayed instead. Items are tracked either by the time accumulated while power is on (ON-TIME) or by cycle-start time (CS-TIME). When power is applied, and every hour thereafter, the remaining time for each item is decremented. When it reaches zero (or has gone negative) the message MAINTENANCE DUE is displayed at the bottom of the screen. A negative number of hours indicates the hours past expiration. This message is not an alarm and does not interfere with machine operation in any way. The intent is to warn the operator that one of the items on the list requires attention. After the necessary maintenance has been performed, the operator can select that item on the SCHEDULED MAINTENANCE screen, press ORIGIN to deactivate it, then press ORIGIN again to reactivate it, and the countdown begins again with a default number of hours remaining (this value is determined by the software and cannot be altered by the operator.) Items available for checking are:

COOLANT - needs replacement	100 ON-TIME
AIR FILTER in control enclosure - replace	250 ON-TIME
OIL FILTER - replace	250 ON-TIME
GEARBOX OIL - replace	1800 ON-TIME
COOLANT TANK - check level, leakage, oil in coolant	10 ON-TIME
WAY LUBE SYSTEM - check level	50 CS-TIME
GEARBOX OIL - check level	250 ON-TIME
SEALS/WIPERS missing, torn, leaking - check	50 CS-TIME
AIR SUPPLY FILTER - check for water	10 ON-TIME
HYDRAULIC OIL - check level	250 ON-TIME

**ALARMS / MESSAGES DISPLAY**

The **ALARMS DISPLAY** can be selected at any time by pressing the ALARM MESGS button. When there are no alarms, the display will show NO ALARM. If there are any alarms, they will be listed with the most recent alarm at the bottom of the list. The CURSOR and PAGE UP and PAGE DOWN buttons can be used to move through a large number of alarms. The CURSOR **right** and **left** buttons can be used to turn on and off the ALARM history display.

The **MESSAGE DISPLAY** can be selected at any time by pressing the ALARM MESGS button a second time. This is an operator message display and has no other effect on operation of the control. Any message can be typed into the message display and called up later.

You may leave an electronic note to yourself or anyone else by using this feature. The note may be for the operator to change tools after running a number of parts or it may be a diary for machine maintenance intervals that are performed. Data is automatically stored and maintained even in a power off state. The message display page will come up during power up if there are no alarms present.

To enter messages, press the ALARM MESGS button twice. You may now enter data by simply typing directly onto the screen. The cancel and space keys can be used to remove existing messages. The DELETE button can be used to remove an entire line.

PARAMETER / DIAGNOSTIC DISPLAY

The **PARAMETER DISPLAY** can be selected at any time by pressing the PARAM DGNOS button. Changes to parameters can be made when in any mode except when running a program. The CURSOR **up** and **down** buttons move to different parameters and the PAGE UP and PAGE DOWN buttons move through groups of parameters. Parameters 1, 15, 29, 43, and 57 are displayed as a single page of discrete flags. Selecting among the flags is done with the CURSOR **left** and **right** buttons. It is recommended that parameters not be changed with the servos on. Parameters cannot be changed with the servos on. The EMERGENCY STOP button can be used to turn off the servos.

The parameters have been organized so that logically-associated parameters are grouped together. These logical groupings are placed together into contiguous screens called pages. The most commonly changed parameters have been placed at the beginning of the page list. A list of the parameter pages and the order of succession in the control are given below.



PAGE TITLE	DATA DESCRIPTION
COMMON SWTCH	Non-axis bit switches.
COMMON PAGE1	First page of non-axis parameters.
COMMON PAGE2	Second page of non-axis parameters.
COMMON PAGE3	Third page of non-axis parameters.
MACRO M CALL	Parameters that alias M codes to subroutines.
MACRO G CALL	Parameters that alias G codes to macros.
X BIT SWITCH	Bit switches for the X axis.
X PARAMETERA	First page of X axis parameters.
X PARAMETERB	Second page of X axis parameters.
Z BIT SWITCH	Bit switches for the Z axis.
Z PARAMETERA	First page of Z axis parameters.
Z PARAMETERB	Second page of Z axis parameters.
B BIT SWITCH	Bit switches for the B axis.
B PARAMETERA	First page of B axis parameters.
B PARAMETERB	Second page of B axis parameters.
X SCREW COMP	X axis screw compensation value.
Z SCREW COMP	Z axis screw compensation value.

The HOME key displays the first parameter page "COMMON SWTCH". Pressing the PAGE DOWN key will display the next page of parameters in the above list. The END key displays the last parameter page "B PARAMETERB". Pressing the PAGE UP key will display the preceding page of parameters in the above list. All other features on the parameters display have remained the same. So, if you are unfamiliar with the new format of the parameters, you can still search by parameter number. Enter the number of the parameter you want to see or view and press the **up** or **down** arrow key. The page that the parameter is on will be displayed and the parameter being searched for will be highlighted. Refer to the "Parameters" section for more information.

The **DIAGNOSTIC DATA DISPLAY** can be selected at any time by pressing the PARAM DGNOS button a second time. There are two pages of diagnostic data and the PAGE UP and PAGE DOWN buttons are used to select between them. After this, the current run time and the number of tool changes is displayed.

SETTING / GRAPHIC DISPLAY FUNCTION

The **SETTINGS DISPLAY** can be selected at any time by pressing the SETNG/GRAFH key. When the settings are displayed, changes can be made to any of the settings. There are some special functions in the settings; refer to the "Settings" section for a more detailed description.

The **GRAPHICS FUNCTION** is a visual dry run of your part program without the need to move the axes and risk tool damage from programming errors. This function is far more powerful than using the DRY RUN mode because all of your work offsets, tool offsets, and travel limits can be checked before any attempt is made to move the machine. The risk of a crash during setup is greatly reduced.

To run a program in Graphics, you must be in either MEM or MDI mode.

After loading the program into memory, select MEM (or MDI) and press the SETNG/GRAFH key twice to select the Graphics Simulation mode. This function operates the same as if running a program on the machine except no physical machine action occurs.



The graphics screen is composed of the following areas:

DISPLAY TITLE AREA

The title area is on the top left line of the screen and indicates the display (GRAPHICS), the mode you are in (MEM or MDI), the program number, and the current program line being executed. It is the same as the top line of all displays.

KEY HELP AREA

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

LOCATOR WINDOW

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

TOOL PATH WINDOW

In the center of the display is a large window that represents a look down perspective of the X-Z axis. It displays tool paths during a graphics simulation of a CNC program. Rapid moves are displayed as coarse dotted lines, while feed motion is displayed as fine continuous lines. The rapid path can be disabled by Setting 4.

The tool path window can be scaled. After running a program, you can scale any portion of the tool path by pressing F2 and then using the PAGE DOWN key and the ARROW keys to select the portion of the tool path that you want to see enlarged. During this process, a rectangle will appear within the TOOL PATH window and the Locator window indicating what the TOOL PATH window will represent when the zoom process is complete. The locator window always portrays the entire table with an outline of where the TOOL PATH window is zoomed to. The PAGE UP key unzooms the rectangle one step. After sizing or moving the rectangle, pressing the WRITE key will complete the zoom process and re-scale the TOOL PATH window. Pressing F2 and then the HOME key will expand the TOOL PATH window to cover the entire table. After the TOOL PATH window is re-scaled, the TOOL PATH window is cleared and you must rerun the program, or a portion of it, to see the tool path. The tool path is not retained in the control.

The scale and position of the TOOL PATH window is saved in Settings 65 through 68. Any scaling performed on the TOOL PATH window is retained. You can leave graphics to edit your program and when you return, your previous scaling is still in effect.

CONTROL STATUS

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

POSITION WINDOW

The location of all enabled axes can be viewed in this window. By default it is OFF. This window can be opened by pressing the F3 key. Additional presses of the F3 key will display the various position formats that the control keeps track of. This window also displays the current scale of the tool path window and the current simulated tool number. The value represented by the vertical dimension of the Tool Path window is labeled X-SIZE. At power-on, this will be the full X-axis work cell travel. When you zoom into a work cell area, this value will become smaller, indicating that you are viewing a smaller portion of the work cell. In addition to the above, a perspective 3D graphics view is also selected by Setting 3.

To exit the Graphic mode, select any other display or mode. When you exit Graphics, the graphics image is lost and must be built again by running the program.


HELP / CALCULATOR FUNCTION

The **HELP FUNCTION** is selected by pressing the HELP display button. This will bring a mini-manual up on the CRT. There are 26 topic areas selectable with the A-Z keys. Pressing the D key will display a directory of the topics. The topics covered are:

- A START UP AND RUNNING
- B PROG. REVIEW / DNC / BGEDIT / POWER DOWN
- C G/M/S/T COMMAND CODES
- D RETURN TO THIS DIRECTORY
- E EDITING PROGRAMS
- F SETTING PAGE
- G SPECIAL G CODES
- H TROUBLE SHOOTING
- I MDI / MANUAL DATA INPUT
- J JOGGING / HANDLE FUNCTION
- K CRT DISPLAY / KEYBOARD
- L ALARMS / MESSAGES
- M MAINTENANCE REQUIREMENTS
- N SET UP PROCEDURES
- O OVERRIDES: FEED/SPIN/COOLANT
- P PARAMETERS / DIAGNOSTICS
- Q POSITION DISPLAYS
- R RECV / SEND PROGRAMS
- S SAMPLE PROGRAM
- T TOOL OFS/TOOL LIFE/LOAD
- U GRAPHIC FUNCTION
- V TOOL TURRET
- W WORK COORDINATES
- X CREATING PROGRAMS
- Y SPECIAL FUNCTIONS
- Z ZERO RETURN

The PAGE UP and PAGE DOWN buttons move to the adjacent topic. The CURSOR up and down buttons move through the text of each topic. When the HELP display is selected, the alphanumeric keys cannot be used to input data on the data entry line of the screen.

The **CALCULATOR FUNCTION** is selected by pressing the HELP key a second time. There are three calculator pages: Trig Help, Circular Interpolation Help, and Turning/Tapping Help. All of these have a simple calculator and an equation solver. Trig, Circular, and Turning Help are selected using the PAGE UP and PAGE DOWN keys. The **Fn** keys also allow data to be moved from other displays to/from the calculator.

All of the Calculator Help functions have a calculator for simple add, subtract, multiply, and divide operations. When one of these functions (Trig, Circular, or Turning) is selected, a calculator window appears in the upper left corner of the screen, and below it 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 at the cursor in the lower left corner of the screen and pressing the WRITE 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.



- 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 you want to use the solution developed for a circular motion. Push INSERT to add that circular motion command line to your program.

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

- F4** In the Calculator Help 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 Trig Help page will help you solve a triangular problem. You 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** and **down** buttons to select the value to be entered with WRITE. For inputs that have more than one solution, entering the last data value a second time will cause the next possible solution to be displayed. The F3 and F4 buttons perform special data import and export functions.

Circular Interpolation Help

The Circular Help page will help you solve a circle problem. You enter the center, radius, angles, start and end points and when enough data has been entered, the control will solve for the circular motion and display the rest of the values. In addition, it will list the four ways that such a move could be programmed with a G02 or G03. Those four lines can be selected using the CURSOR **up** or **down** buttons, and the F3 button will import the highlighted line into a program you are editing. Use the CURSOR **up** and **down** buttons to select the value to be entered with WRITE.

For inputs that have more than one solution, entering the last data value a second time will cause the next possible solution to be displayed. The CW/CCW entry is changed to the other value by pressing WRITE.

Turning/Tapping Help

The Turning/Tapping Help page will help you solve three equations relating to turning and tapping. They are:

1. $SFM = (\text{TURNING DIAMETER IN.}) * \text{RPM} * 3.14159 / 12$
2. $(\text{CHIP LOAD IN.}) = (\text{FEED IN./MIN.}) / \text{RPM} / \# \text{ FLUTES}$
3. $(\text{FEED IN./MIN.}) = \text{RPM} / (\text{THREAD PITCH})$

With all three equations, you may enter all but one of the values and the control will compute the remaining value and display it. Note that the RPM value for equations 1 and 2 are the same entry.

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

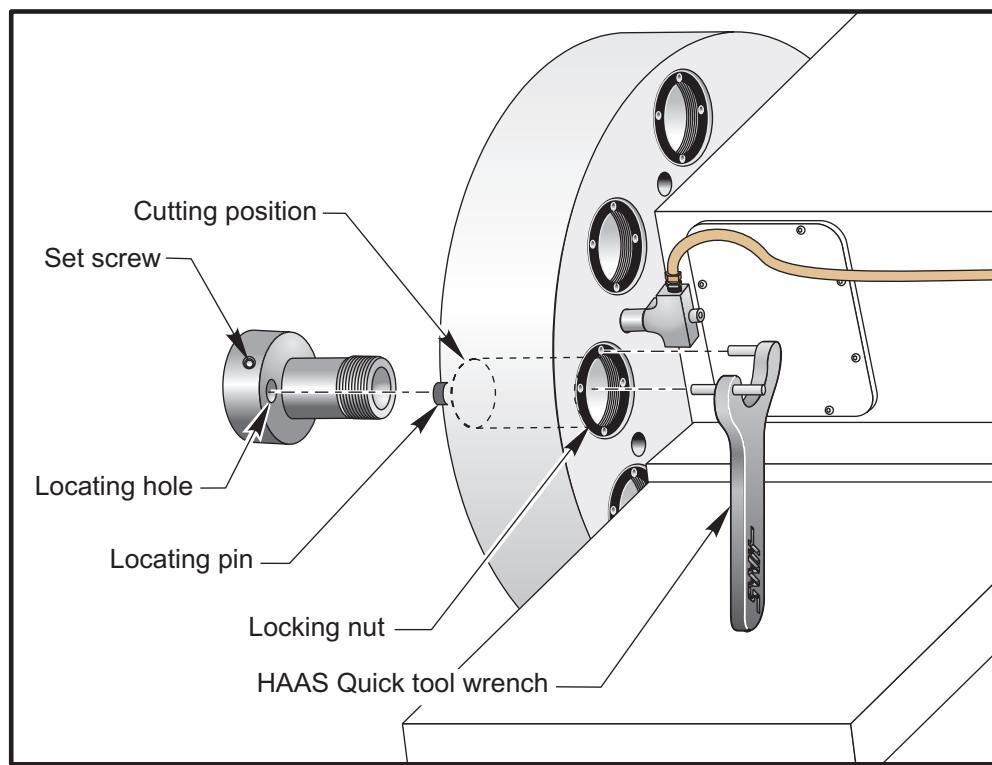


3.20 TOOLCHANGER OPERATIONS

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.

After POWER UP/RESTART and ZERO RET, the control will ensure 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. If you enter Tnn prior to pressing TURRET FWD or TURRET REV the turret will bring the entered tool around to the cutting position. Use the CURNT COMDS display to see what tool is currently in the cutting position.

Tools can only be changed out of the turret one tool at a time and the selected tool must be in the cutting position.

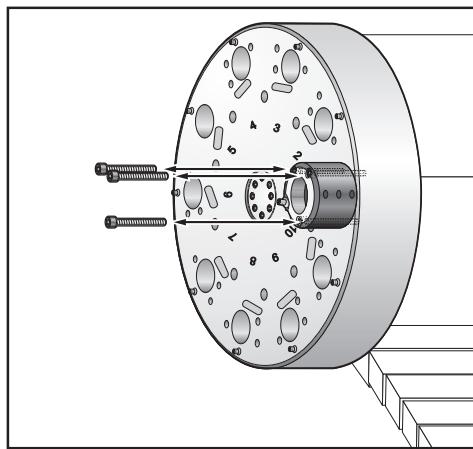


Tool change operation

1. Insert cutting tool into holder and tighten down set screw with 3/8" Allen wrench.
2. Place the tool holder into the empty turret pocket ensuring the locating pin and hole are aligned properly.
3. Screw the nut on the back side of the turret onto the tool holder using the single pin side of the Haas Quick Tool Wrench.



4. Tighten the nut completely with the double pin side of the wrench.

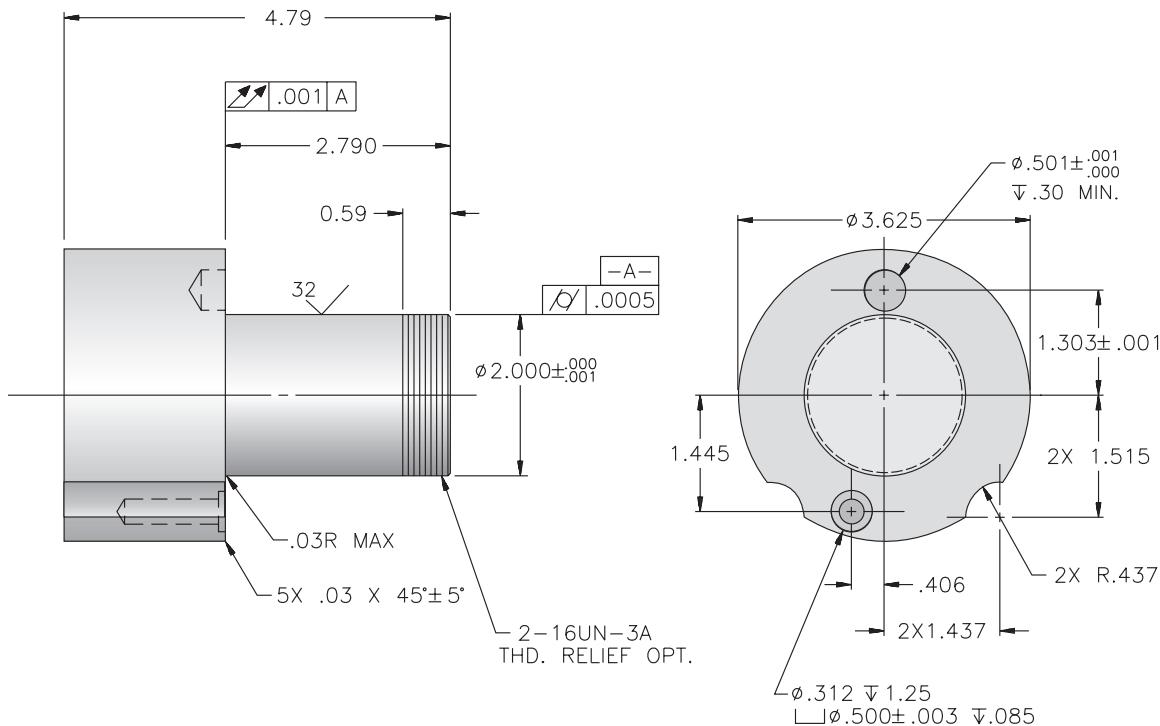


In addition to the retaining nut, the 2 inch I.D. tool holder has three face mounting bolts for extreme rigidity.

IMPORTANT!! Be sure to remove the wrench before rotating the turret!

5. In MDI mode use TURRET FWD or TURRET REV to move to the next tool position.

TOOL HOLDER SPECIFICATIONS



**3.21 TRAVEL LIMITS**

Travel limits in this machine are defined by a limit switch in the positive direction and by stroke limits set by parameter in the negative direction. Prior to establishing the home positions with the POWER UP/RESTART or AUTO ALL AXES buttons, there are no travel limits and the user must be careful not to run the turret into the stops and damage the screws or way covers.

Prior to establishing the home positions (POWER UP/RESTART or AUTO ALL AXES), jogging is normally not allowed. Setting 53 can be turned on to allow jogging prior to zero return but this defeats the travel limits and you may damage the machine running the axes into the stops.

Note that all motion is in a negative direction from machine zero. Travel limits for any auxiliary axes are set into those single axis controls.

When jogging, an attempt to move past the travel limits will not cause an alarm but the axis will stop at the limit. The JOG handle inputs may be ignored in this case.

When running a program, an attempt to move outside of the travel limits will cause an alarm prior to starting the motion and the program will stop. An exception is a circular motion which starts and ends inside of the travel limits but moves outside of the limits during the motion. This will cause an alarm to occur part way through the motion.

Travel limits apply even when running a program in Graphics mode. An alarm is generated and the program will stop.

**3.22 FEED/RAPID/SPINDLE OVERRIDES**

The feed rate can be varied from 10% to 200% of the programmed value while in operation. This is done with the feed rate +10%, -10% and 100% buttons. The FEED RATE override is ineffective during G76 and G92 threading cycles. FEED RATE override does not change the speed of any auxiliary axes.

During manual jogging, the feed rate 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 10% to 150%, using the SPINDLE overrides as above. It is also ineffective for G76 and G92. In the SINGLE BLOCK mode, the spindle may be stopped. It will automatically start up upon continuing the program. When spindle speed is varied in G99 (Feed Per Rev) mode, the feedrate will automatically be adjusted to keep the feed per revolutions constant.

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

In the Setting page, it is possible to disable the override keys so that the operator cannot select them. This is Setting 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 must be pressed to proceed after a FEED HOLD. When in a FEED HOLD, the bottom left of the screen will indicate this. The door switch on the enclosure also has a similar result but it will display "Door Hold" when the door is opened. When the door is closed, machine operation will continue normally. Door hold can be prevented with Setting 51. Door Hold and FEED HOLD do not stop any auxiliary axes.

When Parameter 57 flag DOOR STOP SP is set to 1, the door switch will stop the servos and the spindle. In addition, the override setting does not work, and you will not be able to start a program. However, Door Hold will not stop a tool change operation or a tapping operation, and will not turn off the coolant pump. Also, if the doors are open, the spindle speed will be limited to 500 RPM.

There is also an override function for the coolant supply. This is done from the Setting 32. The "NORMAL" setting checks the low coolant alarm and turns the pump on and off with **M** codes. The "OFF" setting ignores the coolant alarm but will alarm if an attempt is made to turn the coolant on. The "IGNORE" setting is used to ignore all coolant commands and the low coolant alarm.

At any time a program is running, the operator may override the coolant setting by pressing the MDI Coolant button. The pump will remain either on or off until the next **M** command or operator action.

Overrides can now be reset to defaults upon processing M30 and/or RESET. This feature is selected by Setting #83.

**3.23 WARMUP COMPENSATION**

When the machine is powered on, if setting 109, and at least one of Settings 110 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 the operator responds 'Y', the control immediately applies the total compensation, (setting 110 and/ or 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 for the X-axis, in Setting 110, will be 50%.

As with other settings, the Warm-up Compensation Settings can be changed at any time. Updating the Warm-up Compensation Time may activate compensation, but changes to the X or Z distance settings will not activate compensation. To "restart" the time period, it is necessary to power the machine off and on and answer "yes", to the compensation query at start-up.

WARNING!

Changing settings 110 or 112 while compensation is in progress can cause a sudden movement of up to 0.0044 inches.

The amount of remaining warmup time is displayed on the bottom right hand corner of the DIAGNOSTICS INPUTS2 screen using the standard hh:mm:ss format. The initial amount of warmup time to be used starting when power is applied is specified in Setting 109 WARMUP TIME IN MIN.



OPERATION

SL Series

OPERATOR'S MANUAL

June 2001


This section contains the following:

- Alphabetical Address Codes
- Tool Nose Compensation
- Subroutines
- Functions

4. PROGRAMMING

CNC controls use a variety of coordinate systems and offsets that allow the user to control the location of the tooling point to the part in an efficient and precise manner. The HAAS lathe support both YASNAC and FANUC style coordinate systems and offsets. This section describes the interaction between various coordinate systems and tooling offsets. The differences and similarities between FANUC and YASNAC systems are detailed.

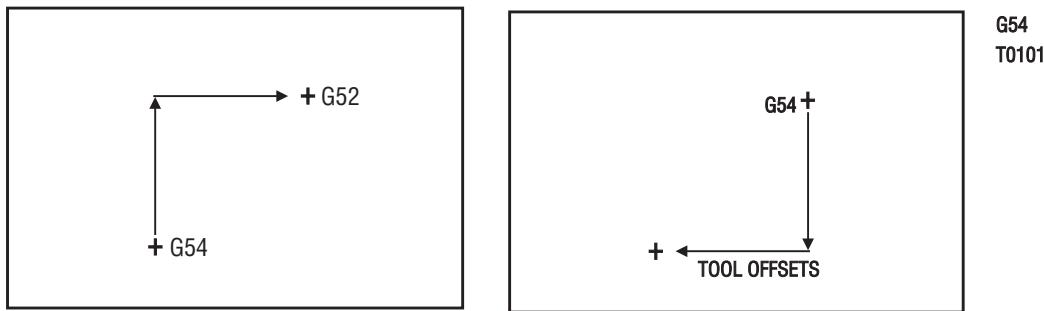
Refer to the "Setup Procedures" section for a detailed description of setting up offsets. Refer to the "G Codes" section for more information on the codes that affect work coordinates and offsets.

Effective Coordinate System

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

effective coordinate	=	global coordinate
+		common coordinate
+		work coordinate
+		child coordinate
+		tool offsets

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



Effective coordinate system and its constituents.

NOTE: All of these coordinates and offsets are available for use by the part programmer. It is the rare case that more than two or three are used to create any part.



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 has the advantage that it is calculated at run time by the CNC control so that the current machine location becomes the effective coordinates specified by a G50 command. The calculated global coordinate system values can be seen on the second work coordinate offsets display page 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.

Common Coordinate System (FANUC)

The common coordinate system is also available to the programmer. It is found on the second work coordinate offsets display page just below the global coordinate system. The common coordinate system is retained in memory when power is turned off. The common coordinate system can be changed at runtime with the G10 command or by using macro variables.

Work Coordinate Shift (YASNAC)

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.

Work Coordinate Systems (FANUC)

Work coordinates are an additional optional coordinate shift relative to the global coordinate system. There are 26 work coordinate systems available on a HAAS control, designated G54 through G59 and as G110 through G129. Work coordinates are retained in memory at power down. The last used work coordinate stays in effect until another work coordinate is invoked or until another power up. Setting 56, if turned on, will cause an M30 to select G54. G54 is the work coordinate in effect when the control is powered on. G54 can be deselected by ensuring that the X and Z values on the work offset page for G54 are set to zero. The FANUC style work coordinates are also available when Setting 33 is set to YASNAC.

Child Coordinate System (FANUC)

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. Each program must explicitly set a child coordinate system. There is no default child coordinate and any G52 set during program execution is removed upon program termination.

Machine Coordinate System

The effective coordinates take on the value of machine coordinates when all other coordinates are zero. Machine coordinates can be referenced by specifying G53 in a motion block.

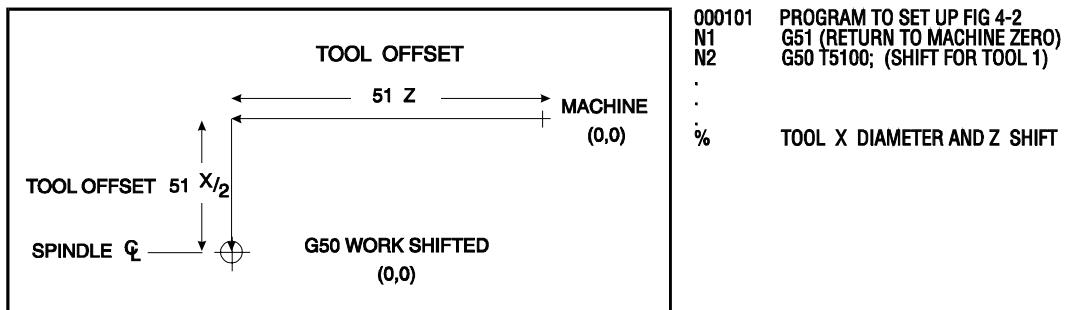
Tool Offsets (FANUC)

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



Tool Offsets (YASNAC)

When Setting 33 is set to YASNAC mode, tool offsets operate differently than described above. Wear offsets are applied in the same manner. In YASNAC mode, no geometry offsets are available but are replaced with tool shift offsets. YASNAC tool shift offsets modify the global coordinate to allow for varying tool lengths. Tool shift offsets must be invoked prior to the use of a tool with G50 Txx00 command format. The tool shift offset replaces any previously calculated global shift offset when invoked. Any G50 command overrides a previously selected tool shift. There are 50 tool shift offsets numbered 51 through 100.



G50 YASNAC Tool Shift.

Automatic Setting of Tool Offsets

Tool offsets can be recorded automatically by using the X DIA MESUR or the Z FACE MESUR key. Care must be taken when using these keys. If the common, global, or currently selected work offset has 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 programmed reference point using the runtime coordinate values. All tools should come to the same point.

**4.1 PROGRAM STRUCTURE**

A CNC part program consists of one or more blocks of commands. When viewing the program, a block is the same as a line of text. Blocks shown on the CRT are always terminated by the ";" symbol which is called an EOB (End Of Block). Blocks are made up of alphabetical address codes and the " / " symbol. Address codes are always an alphabetical character followed by a numeric value. For instance, the specification of the position to move the X-axis would be a number preceded by the X symbol.

The " / " symbol, sometimes called a slash, is used to define an optional block. A block that starts with this symbol can be optionally ignored with the BLKDEL button when running a program.

There is no positional requirement for address codes. They may be placed in any order within the block. The following is a sample lathe program as it would appear on the CRT. The words following the ":" are not part of the program but are put here as further explanation.

%	:PROGRAMS MUST BEGIN AND END WITH %
O1234	:PROGRAM NUMBER
(OP1 SAMPLE LATHE PART)	:PROGRAM COMMENTS
N1(ROUGH TURN TOOL)	:FIRST OPERATION
N100 G28	:MOVE TO MACHINE ZERO,CANCEL OFFSETS
N101 G50 S2000	:SPINDLE SPEED MAXIMUM 2000 RPM
N102 G00 G97 T101 S500 M03	:RAPID MODE, SELECT ROUGHING TOOL1 WITH OFFSET 1, SPINDLE SPEED 500
N103 G00 X3.1 Z.03	:MOVE TO X-Z LOCATION
N104 G96 S450 M08	:ENABLE CONSTANT SURFACE SPEED AT 450 SURFACE FEET PER MINUTE, COOLANT ON
N105 G71 P106 Q111 D.095 U.020 W.005 F.010	:ROUGH TURN CANNED CYCLE USING PATH DE FINED IN BLOCKS N106 TO N111 USING FEED OF .010 INCH PER REVOLUTION FOR ROUGHING WHILE LEAVING .020 INCH FINISH ALLOWANCE IN THE X DIAMETER AND .005 INCH IN Z. REMOVE .095 INCH EACH PASS. :PATH USED BY G71
N106 G01 X1.748 F.005	
N107 Z-3.25	
N108 X2.87	
N109 X2.942 Z-3.286	
N110 Z-4.1	
N111 X3.1	
N114 M09	:1ST BLOCK EXECUTED AFTER G71, COOLANT OFF
N115 G00 G97 S500	:CANCEL CONSTANT SURFACE SPEED
N116 G28	:RETURN TO TOOL CHANGE POSITION
N2(FINISHING TOOL)	
N201 G50 S2000	:SPINDLE SPEED MAXIMUM 2000 RPM
N202 G00 G97 T202 S500 M03	:RAPID MODE, SELECT FINISH TOOL 2 WITH OFFSET2, SPINDLE SPEED 500
N203 G00 X3.1 Z.03	:MOVE TO X-Z LOCATION
N204 G96 S450 M08	:ENABLE CONSTANT SURFACE SPEED AT 450 SURFACE FEET PER MINUTE, COOLANT ON
N206 G70 P106 Q111 TO N208 M09	:FINISHING CYCLE USING PATH DEFINED BY BLOCKS N106 :COOLANT OFF AFTER CANNED CYCLE
N209 G00 G97 S700	:CANCEL CONSTANT SURFACE SPEED
N210 G28	:RETURN TO TOOL CHANGE POSITION



N3(1/2 DIA. 90 DEG. SPOT DRILL)

N301 G50 S2000

N302 G00 G97 T303 S700 M03

:SPINDLE SPEED MAXIMUM 2000 RPM

:RAPID MODE, SELECT SPOT DRILL 3 WITH OFFSET 3,
SPINDLE SPEED 700

N303 G00 X0 Z1.

N304 G82 Z-.2 R0.02 F.003 P.3

N305 G28

N4(5/16 DIA. DRILL)

N401 G50 S2000

N402 G00 G97 T404 S1200 M03

:SPOT DRILL, DWELL .3 SECONDS AT BOTTOM OF SPOT.
:RETURN TO TOOL CHANGE POSITION

N403 G00 X0 Z.2

N404 G83 Z-1.5 R0.02 Q.35 F.008

:SPINDLE SPEED MAXIMUM 2000 RPM
:RAPID MODE, SELECT TOOL 4 WITH OFFSET 4, SPINDLE
SPEED 1200:MOVE TO CENTERLINE OF PART, APPLY TOOL OFFSET
:PECK DRILL FACE TO 1.5 INCH DEPTH, CLEAR CHIP AT .35
INCH INTERVAL, .02 CLEARANCE PLANE.

N405 G28

N5(ROUGHING TOOL)

N501 G50 S2000

N502 G00 G97 T505 S600 M03

:RETURN TO TOOL CHANGE POSITION

:SPINDLE SPEED MAXIMUM 2000 RPM

:RAPID MODE, SELECT ROUGHING TOOL 5 WITH OFFSET
5, SPINDLE SPEED 600

N503 G00 X0 Z1.

N504 G84 Z-.85 R0.2 F.0625

N505 G28

M30

%

:TAP TO DEPTH OF Z-.85

:RETURN TO TOOL CHANGE POSITION

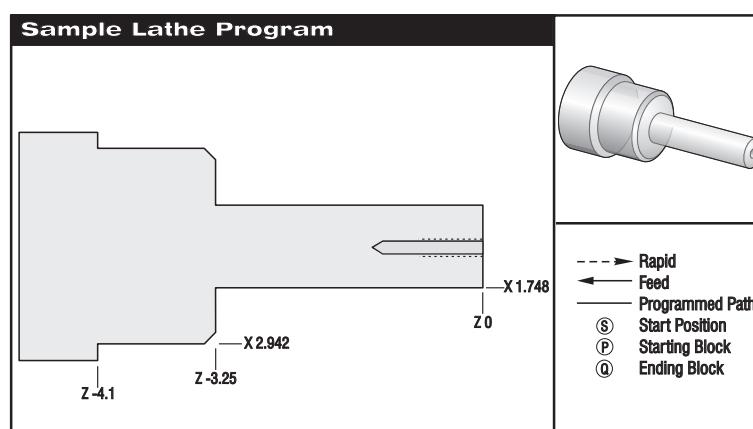
:RESET PROGRAM TO BEGINNING

:END OF TAPE

This program roughs and finishes a two step cylinder and then drills and taps a thread on one end.

Please note that each tool begins by setting the spindle speed and ends by returning to the tool change position. It is common to precede each machining operation with preparatory code and to end each operation by returning to machine zero for the tool change. This is done both for safety and to insure that the proper environment is attained in case the operator has to begin a program in the middle in the event of tool breakage. This is common programming practice.

More than one program can be stored in the memory of the CNC. Every program stored has an Onnnn program name address code to define the number of that program. This number is used to identify the program for selection as the main program to be run, or as a subprogram called from a main program.



Sample Lathe program

**4.2 ALPHABETICAL ADDRESS CODES**

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

A Fourth axis rotary motion

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

The A axis is currently reserved for the tool turret and is hidden to the programmer. The units of rotation indicate tool positions such that 1.000 represents tool #1.

B Linear B-axis motion

The B address character is used to specify absolute motion for the B axis. It specifies a position or distance along the B axis. It is either in inches with four fractional positions or millimeters with three positions. It is followed by a signed number between -8380.00 and 8380.00. If no decimal point is entered, the last digit is assumed to be 1/10000 inch or 1/1000 millimeters.

The B axis is currently reserved for the tailstock.

C Fifth axis rotary motion

The C address character is used to specify motion for the optional external fifth, C, axis. It specifies an angle in degrees for the rotary axis. It is always followed by a signed number and up to three decimal positions. If no decimal point is entered, the last digit is assumed to be 1/1000 degree. The smallest magnitude is 0.001 degree, the most negative value is -8380.00, and the largest value is 8380.00 degrees.

D Depth of cut

The D address character is used to select the depth of cut for each pass of a stock-removal cycle. It is either in inches with four fractional positions, or mm with three fractional positions.

E Feed rate, 6 place precision (same as F)

The E address character is used to select feed rate applied to any interpolating G codes or canned cycles. The unit is in inches per revolution or mm per revolution. Up to six fractional positions can be specified. The default of units/revolution (G99) can be changed to units/minute with G98. For YASNAC and FANUC control compatibility use the E code when 5 or 6 place precision is desired.

F Feed rate

The F address character is used to select feed rate applied to any interpolating G codes or canned cycles. The unit is in inches per revolution or mm per revolution. The default of units/revolution (G99) can be changed to units/minute with G98. Traditionally, the F code was capable of only 4 fractional position accuracy; but on this control you can specify F to six place accuracy with a maximum of 154.000000 inches (393.000000 millimeters). Code E and F are equivalent.



G Preparatory Functions (G codes)

The G address character is used to specify the type of operation to occur in a block. The G is followed by a two or three-digit number between 00 and 255. Each G code is part of a numbered group. The Group 0 codes are non-modal; that is, they specify a function applicable to this block only and do not affect other blocks. The other groups are modal and the specification of one code in the group cancels the previous code applicable from that group. A modal G code applies to all subsequent blocks so those blocks do not need to re-specify the same G code. Multiple G codes can be placed in a block in order to specify all of the setup conditions for an operation, provided no two are from the same numbered group. See the following section (Preparatory Functions (G Codes)) for a detailed list of G codes.

H Not Used, optional macro parameter

I Canned cycle and circular optional data

The I address character is used to specify data used for some canned cycles and circular motions, either in inches with four fractional positions or mm with three fractional positions. I is followed by a signed number between -15400.0000 and 15400.0000 in inches or between -39300.000 and 39300.000 mm for metric.

J Canned cycle and circular optional data

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

K Canned cycle and circular optional data

The K address character is used to specify data used for some canned cycles and circular motions. It is formatted just like the I data. K is followed by a signed number between -15400.0000 and 15400.0000 in inches or between -39300.000 and 39300.000 mm for metric.

L Loop count for repeated cycles

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

M M Code Miscellaneous Functions

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

N Number of block

The N address character is entirely optional. It can be used to identify or number each block of a program. It is followed by a number between 0 and 99999. The M97 and M99 functions may reference an N line number.



O Program number/name

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

P Delay time or program number

The **P** address character is used to enter a time in seconds, a program number for a subroutine call, or a line number for a stock-removal cycle. If it is used as a time (for a G04 dwell), it may be a positive decimal between 0.001 and 1000.0.

If it is used as a line number (for an M97), or a program name (for an M98), the value must be a positive number without a decimal point, up to 9999.

Q Canned cycle optional data

The **Q** address character is 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

The **R** address character is used in canned cycles and circular interpolation. It is either in inches with four fractional positions or mm with three fractional positions. **R** is followed by a signed number between 15400.0000 and -15400.0000 for inches or between 39300.000 and -39300.000 for metric. It is usually used to define the reference plane for canned cycles.

S Spindle speed command

The **S** address character is used to specify the spindle speed or surface speed. The **S** is followed by an unsigned number between 1 and 99999. The **S** command does not turn the spindle on or off; it only sets the desired speed. This command will not cause a gear change to occur.

T Tool selection code

T-codes are used to both select the tool number and to specify the tool shift or geometry and tool wear values to be applied. See section 4.4 for a detailed description of this code.

U Incremental X axis motion

The **U** address character is used to specify motion for the X-axis. It specifies an incremental position or distance along the X axis relative to the current machine position. It is either in inches with four fractional positions or mm with three fractional positions. **U** is followed by a signed number between -15400.0000 and 15400.0000 in inches or between -39300.000 and 39300.000 mm for metric.

**V Optional macro parameter****W Incremental Z-axis motion**

The **W** address character is used to specify motion for Z axis. It specifies an incremental position or distance along the Z axis relative to the current machine position. It is formatted the same as address **U**. **W** is followed by a signed number between -15400.0000 and 15400.0000 in inches or between -39300.000 and 39300.000 mm for metric.

X Linear X-axis motion

The **X** address character is used to specify absolute motion for the X axis. It specifies a position or distance along the X axis. It is either in inches with four fractional positions or mm with three fractional positions. **X** is followed by a signed number in 15400.0000 and -15400.0000 for inches or between 39300.000 and -39300.000 for metric. If no decimal point is entered, the last digit is assumed to be 1/10,000 inches or 1/1000 mm.

Y Not used, optional macro parameter**Z Linear Z-axis motion**

The **Z** address character is used to specify absolute motion for the Z axis. It specifies a position or distance along the Z axis. It is either in inches with four fractional positions or mm with three fractional positions. It is followed by a signed number between 15400.0000 and -15400.0000 for inches or between 39300.000 and -39300.000 for metric. If no decimal point is entered, the last digit is assumed to be 1/10,000 inches or 1/1000 mm.

**4.3 TIPS AND TRICKS****PROGRAMMING**

Floppy Disk file delete - Go to the List Prog page, type “DEL {filename}” Press Write.

The message “DISK DELETE” will appear, and the file will be deleted from the floppy disk.

Requirement: floppy disk driver chip version 2.11.

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

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

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

Offsets, parameters, and settings can be saved to a floppy disk or RS-232. To save to a disk, press LIST PROG, then select the OFSET, SETNG, or PARAM display page. Type in a file name and press F2 to write that display file to disk or press F3 to read the file from disk. To save to RS-232, press LIST PROG first, and then select OFSET, SETNG, or PARAM display page. Press SEND RS-232. Press RECV RS-232 to read the file from RS-232.

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.

The chip conveyor can be turned on or off manually with control keys, or in the program with M-codes, using M31 (Chip FWD), or M32 (Chip REV), or M33 (Chip STOP), when a program is running. Conveyor cycle time can be set with Settings 114 and 115.

The spindle can be stopped or started at a single-block stop or a Feed Hold. Cycle Start is used to restart the program at the original spindle speed.

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

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



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

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

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

Loading multiple files is possible from the Advanced Editor. From the I/O main menu, go to the Floppy Disk Directory menu. The control will load the selected file when you press Enter. The cursor will still remain to allow further files to be selected and loaded. Reset or Undo will exit the screen.

Editing programs Pressing the F4 key while in the Advanced Editor is the hot key to display another version of the current program to edit. The same program will be displayed on both halves of the screen. 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 when switching to the other side.

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

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

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

It is not necessary to turn off coolant, stop the spindle, or move the Z-axis prior to a tool change. The control handles those tasks and, in fact, it will be faster because the control will overlap some of these operations (do them all at the same time).

The coolant can be turned on or off manually any time a program is running. This will override what the program commands until the program commands "on" or "off." This also applies to the chip conveyor.

When tapping, you do not need to turn the spindle on with M03 or M04. The control starts the spindle itself prior to each cycle and it will, in fact, be faster if you do not turn on the spindle, as the control must stop it again anyway.

Taper compensation - Parts that are not supported precisely in the center or if unsupported and too long can suffer from deflection. This feature allows the user to enter a calculated value to the X movement based on the position of the Z cut. 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.



ALARMS

In the alarm display, pressing the right or left cursor arrow reviews the previous alarms, (up to 100 alarms) press either one again to go back to the normal alarm display.

The last 100 alarms can be saved to a floppy disk by entering a file name and pressing F2, while on the Alarm History page. Alternately, the alarm history can be sent to a PC using RS-232 by pressing SEND RS-232.

OFFSETS

The lathe offers up to 200 tool offsets.

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

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

SETTINGS AND PARAMETERS

Jog handle can be used to scroll through settings and parameters

This control can turn itself off in ways controlled by 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. In addition, for safety reasons, the control will turn itself off if an overvoltage or overheat condition is detected for longer than four minutes.

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

Setting 9 Dimensioning changes from INCH to MM. This will change all offset values accordingly.

Setting 31 Reset Program Pointer. Turns on and off the ability of sending the program pointer back to the beginning of the program.

Setting 77 Scale Integer F. This is used to change how the control interprets a feed rate. A feed rate that is entered in your program 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. Or have it assume Integer for an inch value feed rate, or to recognize a feed rate for a selected decimalposition, for a feed rate that does not have a decimal.

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

Setting 88 Reset Resets Override. Turns on and off having the Reset key set all the overrides back to 100%.



Setting 103 Cycle Start / Feed hold - Same key

When this setting is turned ON, Cycle Start must be pressed and held to run a program. Releasing Cycle Start, generates a Feed Hold condition. This setting cannot be ON while Setting 104 is ON. When one of them is ON, the other will automatically turn OFF. This setting can be changed while running a program.

Setting 104 Jog Handle to Single Block

When this setting is turned ON, and single block is selected, the jog handle can be used to single step through a program. Reversing the jog handle will generate a Feed Hold condition. This can be useful when an unexpected long motion block is encountered.

Cycle Start must be used to begin running a program.

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. This setting can be changed while running a program.

Offset Lock (Setting 119) When this setting is turned **ON**, the user is prevented from altering any of the offsets.

Macro Variable Lock (Setting 120) When this setting is turned **ON**, the user is prevented from altering any of the macro variables.

OPERATION

Optional Stop takes effect on the line after the highlighted line when pressed.

Block Delete takes effect four lines after pressed when in cutter compensation; two lines when cutter compensation or tool nose compensation is not used.

Block Look Ahead- The control looks ahead for block interpretation, up to 20 blocks. This is not needed for high speed operation. It is used to insure that DNC program input is never starved, and to allow cutter compensation to have non-XY moves inserted while cutter compensation is on.

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

HOME G28 key Pressing the HOME G28 button alone will return **all axes** to machine zero. To send just one axis to machine home, in rapid, enter the axis letter and press Home G28. **Caution!** There is no warning of any possible collisions

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. The distance from this new zero point is now displayed.

Each axis can be zeroed out independently to show a position relative to the selected zero. To do this go to the POS-OPER page (page up or down), the large position display. Enter handle jog mode, position the axis, X, Y, Z, etc. 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 is added to everytime a tool is used. The tool life monitor will stop the machine if the usage number for the tool is the same or above the number in the alarms column. This helps in avoiding tool breakage and scrapped parts.



Tool overload - Tool load can be defined by the "Tool Load monitor", this will stop normal machine operation if it reaches the tool load defined for that tool. When a tool overload condition is encountered four actions of the control can be taken and are 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.

When you receive (input) a program from the floppy disk, it is always a "receive all". That is, there must be an Onnnnn program name in the floppy disk file. The name you enter on the input line is the file name.

Program files on a floppy disk still must start and end with a %, like RS-232.

You can select an axis for jogging by entering that axis name on the input line and pressing the Handle Jog button. This works for the normal X, Y, Z, and A axes, and the B, C, U, and V auxiliary axes.

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.

If the tool probe is in the down position the tail stock foot pedal is deactivated. In addition, running program and tool changes are stopped if the probe is down.

Live tooling spindle axis RPM is displayed on the Current Commands tool load page.

Spindle/Chuck/Door Rules

The rules for how the lathe spindle, chuck and door operate are as follows:

- 1) A program can be started or continued (Cycle Start) with the door open.
- 2) A program can be started with the chuck unclamped provided parameter 278 bit 8 CHUCK OPN CS is set to 1.
- 3) No spindle commands will be allowed when the door is open. When ever the CNC comes to an 'S' command it will check if the door is open and generate an alarm if it is. The spindle speed 'S' command will not be executed if the door is open. If a program alarms out and the spindles keep turning or the spindles are turning and a program is not running they will be stopped whenever the door is opened.
- 4) No spindle command greater than the value of parameter 248 CHUCK UNCLAMP RPM will be allowed when the chuck is unclamped. If the chuck is commanded to unclamp and the spindle is turning at a speed above that of parameter 248, or the chuck is unclamped and the spindle is commanded to turn at a rate higher then parameter 248, then alarm 171 UNCLAMP RPM TO HIGH will be generated.



5) No chuck unclamp will be allowed with spindle turning (manual operation) and the door open. An alarm will be generated if the CW or CCW buttons are pushed and the door is open and the chuck is unclamped or the spindle speed is greater than that of parameter 248 and the chuck is unclamped.

6) Chuck unclamp will be allowed with the spindle at less than parameter 248 CHUCK UNCLAMP RPM.

7) A door open M-code (M85) will be allowed with a program running but the spindle must be stopped. If the spindle is turning when an M85 is commanded, the control will generate an alarm and will not open the door. This is intended to keep the APL arm (if present) from crashing into the door.

ADVANCED EDITOR

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

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

CALCULATOR

The number in the calculator box in the upper right corner 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 (you will need to first enter the letter (X,Y,Z, etc. for the command you wish 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.

The circular calculator will list four different ways that a circular move could be programmed with all the values entered in for a calculated solution. Four different program lines will be listed at the bottom of that display for executing a circular move. One of the four program lines can be transferred to either EDIT or MDI. To do this first cursor onto the circular move you want to use. Press either EDIT or MDI mode to display where you want to insert the circular move. Press the F3 key, which will transfer the circular move into the input buffer line at the bottom of either the EDIT or MDI display. Pressing INSERT will add that circular command line into your program.

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

**4.3 SUBROUTINES**

One of the more important programming features of a CNC is called subroutines. Subroutines allow the CNC programmer to define a series of commands which might be repeated several times in a program and, instead of repeating them many times, they can be "called". A subroutine call is done with M97 or M98 and a **Pnnnn**. The **P** code is the same as the **O** number of the subroutine to be called.

It is important to note that there is little difference between the main program and the subroutines. In the LIST PROG display, they all appear as numbered programs. When starting execution of a program, the LIST PROG display is used to select the MAIN program and any subroutines used are called from within the main program.

Local subroutines can be used with the M97. This can be even easier to use than the M98 because the subroutine is part of a single main program without the need to define a different **Onnnnn** program. With local subroutines, you can code an M30 for the end of your main program followed by a line number and a subroutine that ends with an M99.

The subroutine call causes the blocks in the subroutine to be executed just as if they were included in the main program. In order to return control to the main program, subroutines must end with an M99.

Another very important feature of subroutines is that the M98 "call" block may also 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.

4.4 Tool Functions (Tnnoo**)**

The **Tnnoo** code is used to select the next tool and offset. The use of this code differs slightly depending on Setting 33 FANUC or YASNAC coordinate system, as described below.

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

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

Inside a G50 block, **xx** 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).

NOTE: If **Txx** is used, it works like **Txxxx**, i.e., **yy=xx**.

**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. If you enter Tnn prior to pressing TURRET FWD or TURRET REV the turret will bring the entered tool around to the front. Use the CURNT COMDS display to see what tool is currently in position.

**4.5 SPINDLE FUNCTIONS****SPINDLE SPEED COMMANDS**

Spindle speed functions are controlled primarily by the **S** address code. The **S** address specifies RPM in integer values from 1 to maximum spindle speed (Parameter 131, NOT TO BE CHANGED BY USER!).

High gear and low gear are selected by programming an M41 (Low Gear) or M42 (High Gear). **The spindle will not change gears automatically.** The spindle will come to a complete stop when changing gears.

If there is no gear box in your machine the gear box is disabled by parameters, it is always in high gear, and M41 and M42 commands are ignored.

Three **M** codes are used to start and stop the spindle. M03 starts the spindle clockwise, M04 starts the spindle counterclockwise, and M05 stops the spindle.

Note that only one **M** code is allowed in a block. This means that if you wish to override the gear with M41 or M42, you must put the **Snnnn** and M41 (or M42) in one block and the M03 (or M04) in the next block. The **Snnnn** should always be in the same block as the M41 or M42, as an unneeded double gear change might otherwise be performed.

RIGID TAPPING CONTROL OF SPINDLE

Rigid tapping eliminates the cost of special tap holders since taps can be held in drill collet holders. The spindle is accurately synchronized with the Z-axis feed, thereby producing threads as accurately as a lead screw tapper. No side forces are generated on the flanks of the threads and tighter thread tolerances are produced. Rigid tapping also eliminates the pullout and distortion of the first thread that occurs on all spring compression/tension devices and tapping heads. While this is not usually a problem on medium to coarse threads, small diameter, fine pitch or soft material tapped holes can have their last thread damaged when the tap pops out of the hole. You can also re-tap a hole without cross-threading, provided the tap and **Z** depths have not been changed. Rigid tapping is used with canned cycle G84 and G184.

It is enabled with the Parameter 57 "Rigid Tap" flag. When enabled, it changes the way G84 and G184 work and a floating tap holder is not needed for these **G** codes.

Rigid tapping allows the use of a tap without a floating tap holder. Pitch control is within 0.0005 inch. Bottom depth control is +/-0.020 inch and repeatability is +/-0.005 inch. Rigid tapping will operate from 100 to 2000 RPM and up to 100 inches per minute feed. Bottom depth control is better at lower speeds and low gear. Thread pitch is limited from 4 to 100 TPI.

The pitch of a tapped hole is defined by the ratio between the feed rate and spindle speed. When rigid tapping is selected, these two must be set exactly. An encoder mounted with the spindle tracks the position of the spindle and the Z-axis is moved precisely to match the pitch of the thread. If the repeatable option is selected, a position pulse from the encoder is used to synchronize the starting of the **Z** motion with the position of the spindle.

Note that with G84/G184, you do not need to use M03, M04, or M05. These canned cycles start and stop the spindle automatically.



4.6 CONTOURING ACCURACY

When milling a part it is desirable to machine it at the fastest feedrate possible and yet obtain the highest accuracy. No machine can instantly accelerate or decelerate and this leads to positioning errors when axis directions change. The higher the feedrate and the sharper the turn the more pronounced the effect. A high feedrate selected for straight line milling will cause corner rounding at intersections when you actually want a sharp corner. Similarly a high feed milling cut blending into a radius will cause the radius to be smaller than programmed because of axis acceleration distance. To alleviate this the HAAS control uses a special function called Contouring Accuracy (G187) so the operator can select the required accuracy.

When using this function the control will move up to the programmed feedrate in straight line moves and slow down at intersections or radii to obtain the required accuracy you want. This accuracy is measured in true three dimensional motion. It uses setting 85 to define a default value and G187 to program a new value directly from your program. The amount of slow down depends on the accuracy specified and on how well one stroke blends into the next. If two strokes blend into each other exactly (in one line), there is never any slow down.

Programming G187 is as follows:

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

The first line will set the required accuracy to 0.01 inches. G187 must be programmed on a line by itself. If there is no E code, the accuracy reverts to setting 85. If MM programming is active, the units are millimeters. The range of values possible are 0.0001 to 0.25 inches and 0.001 to 2.5 mm.

The most important thing to remember with Contouring Accuracy is that normal, well blended strokes should not get a slowdown in feedrate. Only the sharp corners need this in order to achieve the programmed accuracy. However, if you set the accuracy to extremely small values (0.0005 and smaller) you may effect such a slow feedrate that the machine appears to pause when you don't think it should. Even well blended strokes can have a slight error.

When roughing out a pocket the default setting (85) should be used for maximum speed then on the final clean up passes you can specify a higher accuracy. A simple test part of milling a rectangle and varying the values for Contouring Accuracy will demonstrate the principles outlined above.

4.7 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. This feature 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 user can modify the taper at any time.

**4.8 Tool Nose Compensation Programming****OVERVIEW**

Tool nose compensation is a feature that allows the user to adjust a programmed tool path in response to differing cutter sizes or for normal cutter wear. The user can do this by entering minimal offset data at runtime without any additional programming effort.

WHEN To USE TOOL NOSE COMPENSATION

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 Figure 4.7-1, 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, as shown in Figure 4.7-2, 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, as shown in Figure 4.7-3.

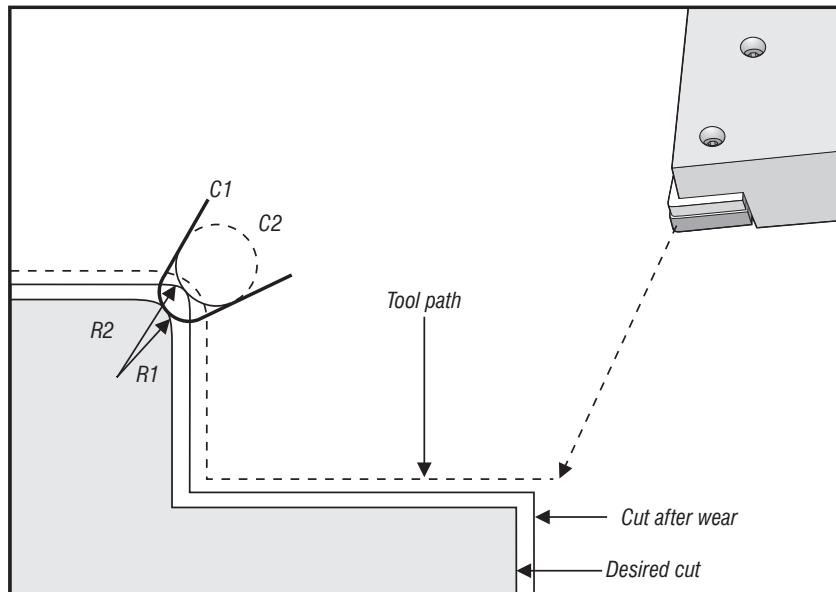


Figure 4.7-1. Tool path for 2 cutter radii.

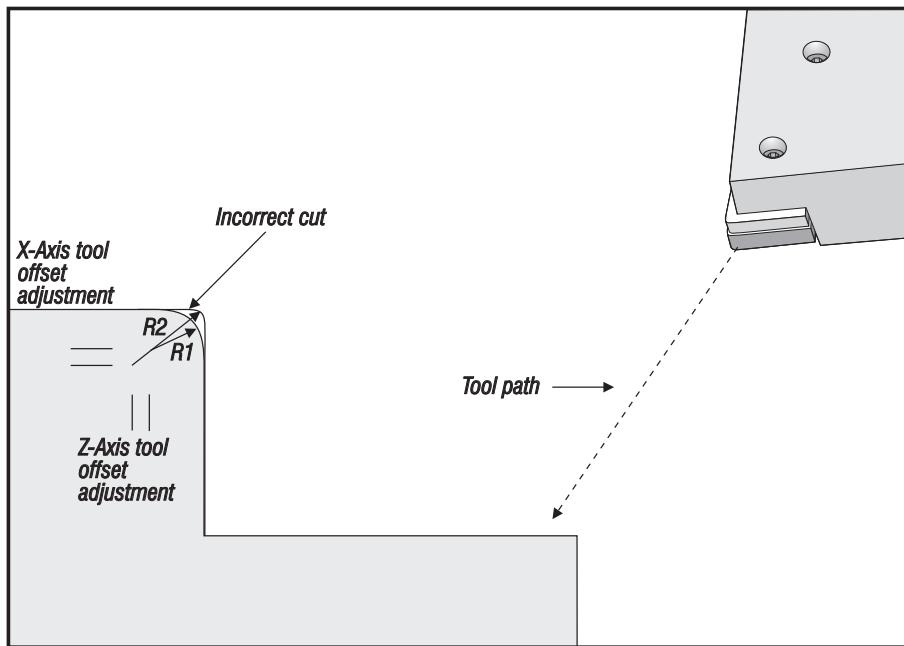


Figure 4.7-2. Two cuts overlaid to show cutting error.

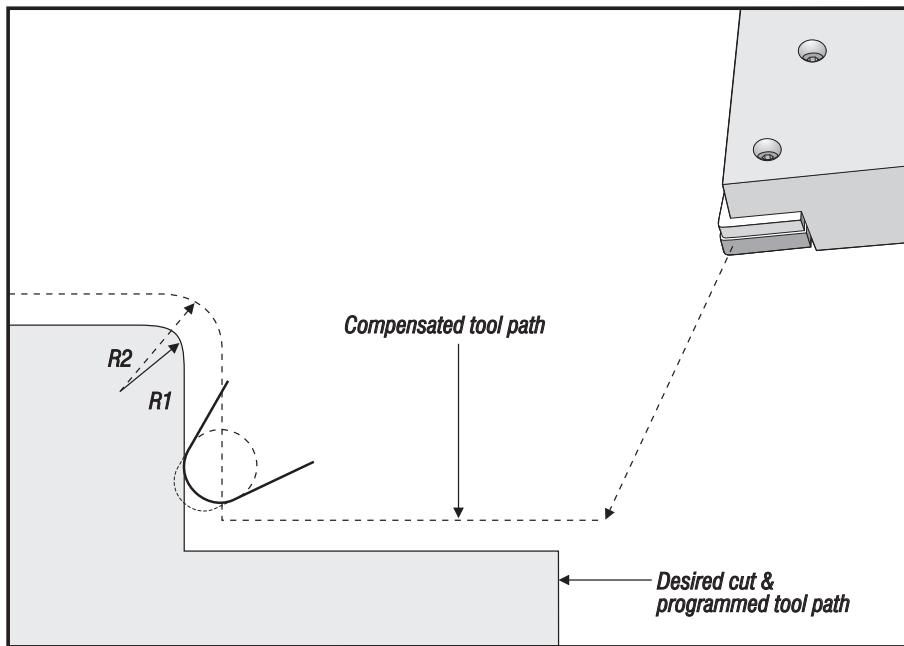


Figure 4.7-3. Path generated when tool nose compensation is used.

Note that in Figure 4.7-3 the 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 current manufacturing operation finish size. When tool nose compensation is used, the control will compensate for a tool's diameter 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. G40, G41, and G42 are described in detail later in the "Tool Nose Compensation G Codes" section.

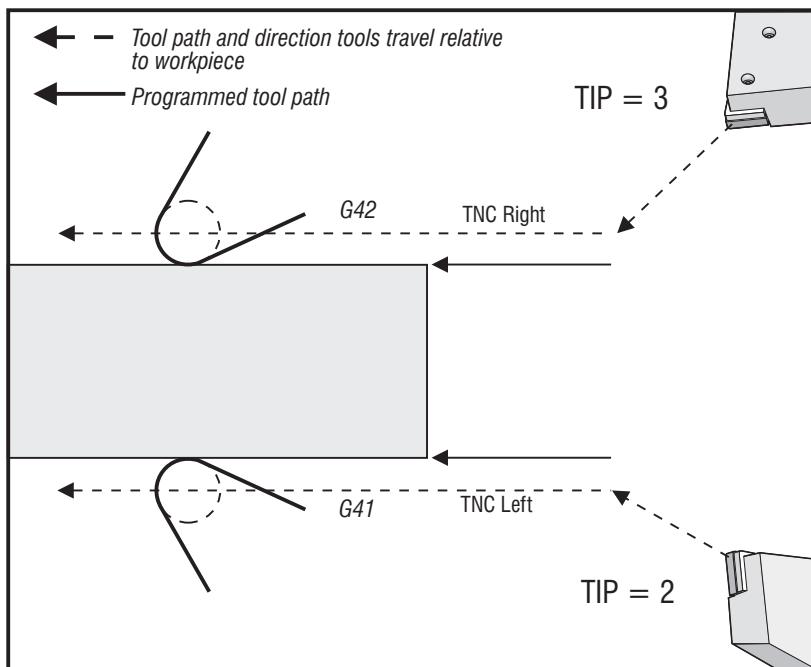


Figure 4.7-4. Shift direction.

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 yourself sitting on the tool tip and steering the tool as if it were a car. Commanding G41 will move the tool tip to the left and a G42 will move the tool tip to the right. For a lathe, this means that normal O.D. turning will require a G42 for correct tool compensation, while normal I.D. turning will require a G41.

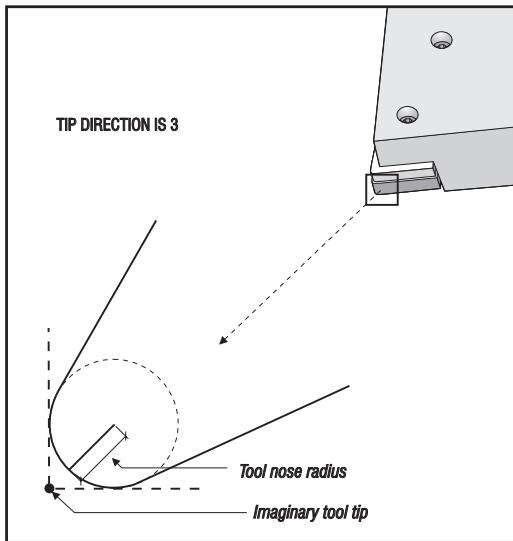


Figure 4.7-5. Imaginary Tool Tip.

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. Figure 4.7-5 shows these features for a common tool configuration.

Tool nose compensation accomplishes its task by reading ahead one or two blocks to determine how it must modify the current block of code. This is referred to as BLOCK LOOKAHEAD or LOOKAHEAD TIME PROCESSING.

When the control is first powered on or in the reset condition, tool nose compensation is cancelled. Tool nose compensation is turned on in a program by programming a G41 or G42 command. When this command is executed, the control will look ahead to determine where the first compensated move will be. The first compensated move is generally a move from a non-compensated position to a compensated position and is therefore unusual. This first move is called the APPROACH move and is required when using tool nose compensation. Similarly, a DEPART move is also required. In a depart move, the control will move from a compensated position to a non-compensated position. A depart move occurs when tool nose compensation is cancelled with a G40 command or Txx00 command. Although approach and depart moves can be precisely planned, they are generally uncontrolled moves and the tool should not be in contact with the part when they occur.

**Using Tool Nose Compensation**

Does it sound complicated? It is not if you follow the steps below when using tool nose compensation.

PROGRAM the part to finished dimensions.

APPROACH AND DEPARTURE

Ensure that there is an approach move for each compensated path and determine if G41 or G42 is to be used.
Ensure that there is also a departure move for each compensated path.

TOOL NOSE RADIUS AND WEAR

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

TOOL TIP DIRECTION

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

TOOL GEOMETRY OFFSET

Set the tool length geometry and clear the length wear offsets of each tool.

CHECK COMPENSATION GEOMETRY

Debug the program in graphics mode and correct any tool nose compensation geometry problems that may occur. A problem can be detected in two ways: either 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.

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


APPROACH AND DEPARTURE Moves For Tool Nose Compensation

The first X or Z motion in the same line that contains a G41 or G42 is called the APPROACH move. The first move must be a linear move, that is a G01 or G00. At the start of an approach move the current position is not compensated. At the end of the approach move the machine position will be fully compensated. This is shown in Figure 4.7-6.

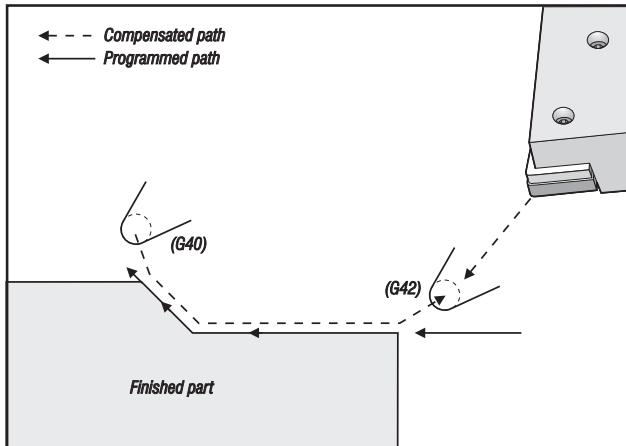


Figure 4.7-6. Approach and Departure moves.

Any line that contains a G40 will cancel tool nose compensation and is called the DEPARTURE move. The last move must be a linear move, that is a G01 or G00. At the start of a departure move, the current position is fully compensated. The compensated position at this point will be normal (right angle) to the last programmed block. At the end of the departure move the machine position is not compensated. Refer to Figure 4.6-6.

Figure 4.7-7 shows the normal condition just prior to cancellation of tool nose compensation. Some geometries will result in over or undercutting of the part. The programmer can control this by including an I and K in the G40 cancellation block. The I and K address codes in a G40 block define a vector that the control will use in determining the compensated target position of the previous block. The vector is usually aligned with an edge or wall of the completed part. Figure 4.7-7 shows how I and J can correct undesired cutting in a departure move. Refer to the description of the G40 command for instructions on calculating the values of I and K.

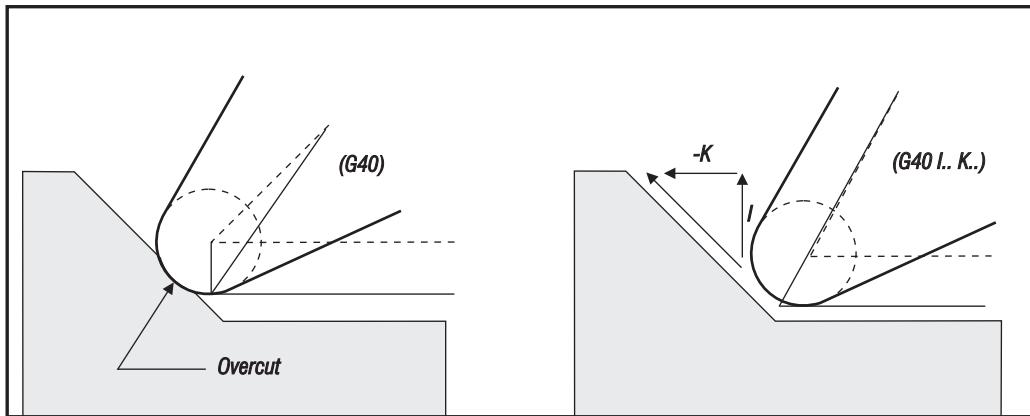


Figure 4.7-7. Use of I and K in a G40 block.

**Tool Nose Radius Offset and Wear Offset**

Each turning tool that uses tool nose compensation requires a TOOL NOSE RADIUS. The tool nose radius specifies how much the control is to compensate for a given tool. It is determined by the geometry of the tool tip. 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 where the tool nose radius of each tool is placed. 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. It is 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 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. This should place positive values in the wear column. When replacing a worn tool with a new one, the wear offset should be cleared to zero.

NOTE: It is important to remember that tool nose compensation values are in terms of radius rather than diameter. This is important in blocks where tool nose compensation is cancelled. If the incremental distance of a departure move in a compensated path is not twice the radius of the cutting tool's radius, 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 is less than .032, which is the value required to avoid cut-in with a departure move
before TNC is cancelled.)
G40 G00 X.25 ;
Z.05 ;
G28 ;
M30 ;
```

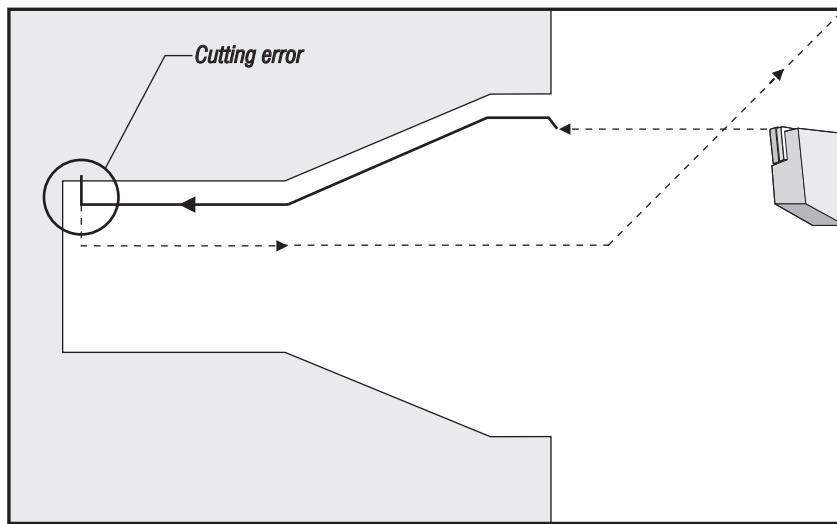


Figure 4.7-8. An invalid program using TNC and G70.

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. Refer to Figure 4.7-5.

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. Figure 4.7-9 is a summary of the tip coding scheme along with cutter orientation examples.

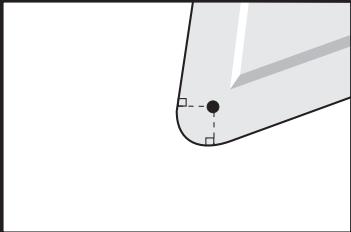
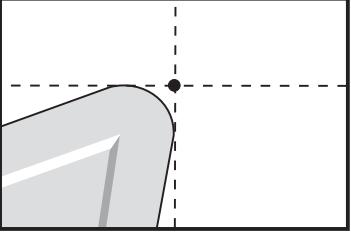
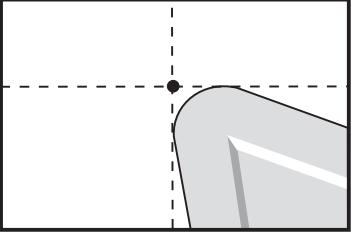
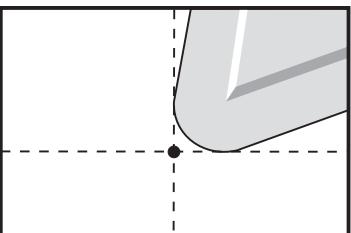
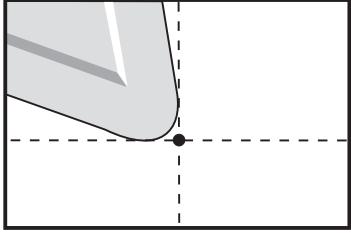
Note that the tip indicates to the setup person how the programmer intends the tool offset geometry to be measured. For instance, if the setup sheet shows tip direction 8, the programmer intends the tool geometry to be at the edge of and on the centerline of the tool insert.

**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 "Setup Procedures" 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.

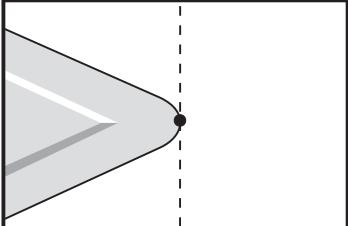
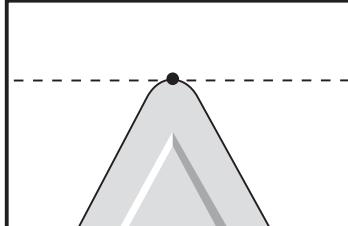
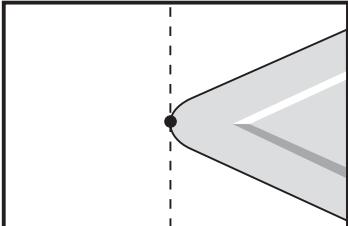
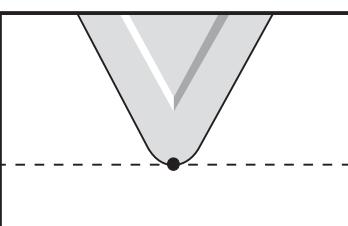
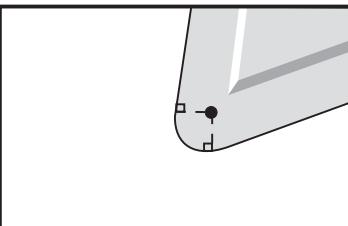
Often a tool will exhibit uneven wear. This occurs when particularly heavy cuts occur on one edge of the tool. In this case it may be desirable to adjust the X or Z GEOMETRY WEAR rather than the RADIUS WEAR. By adjusting X or Z length geometry wear, the operator can often compensate for uneven tool nose 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. One can determine which wear to adjust by checking several X and Z dimensions on a finished part. Wear that is even will result in similar dimensional changes on the X and Z axes, and suggests that the radius wear offset should be increased. Wear that affects the dimensions on one axis only suggests length geometry wear. Good program design based on the geometry of the part being cut should eliminate uneven wear problems. Generally, it is a good idea to rely on finishing tools that use the entire radius of the cutter for tool nose compensation.



TIP CODE	IMAGINARY TOOL TIP ORIENTATION	COMMENT 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

FIGURES 4.7-9a. Imaginary tool tip.



TIP CODE	IMAGINARY TOOL TIP ORIENTATION	COMMENT TOOL CENTER LOCATION
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

FIGURES 4.7-9b. Imaginary tool tip.

**TOOL NOSE COMPENSATION GEOMETRY**

Figure 4.7-10 shows the various geometries encountered in tool nose compensation. It is organized into four categories of intersection based on block to block motion. 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.

TOOL NOSE COMPENSATION IN CANNED CYCLES

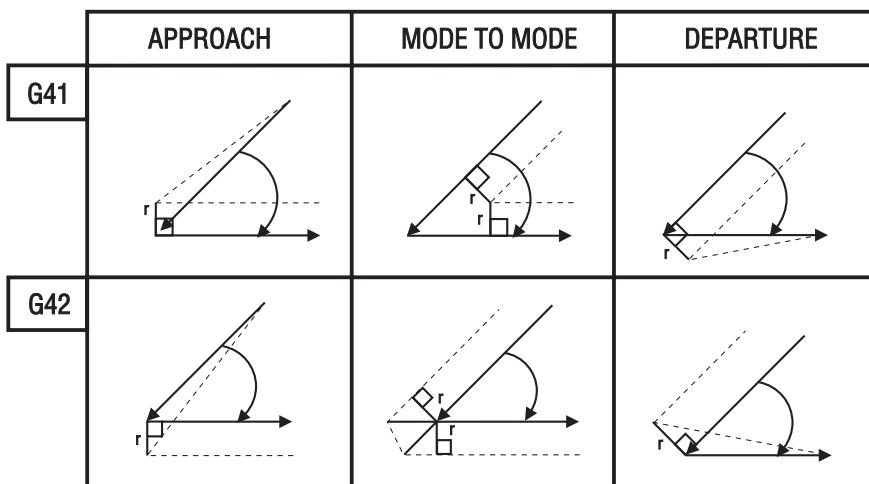
This section describes how tool nose compensation works when a canned cycle is used. Refer to the "Canned Cycles" section of this manual for a detailed description of canned cycles. Some canned cycles ignore tool nose compensation, some canned cycles expect a specific coding structure, while other canned cycles perform their own specific canned cycle activity.

The following canned cycles will ignore tool nose radius compensation. It is recommended that tool nose compensation be cancelled prior to executing any of these canned cycles.

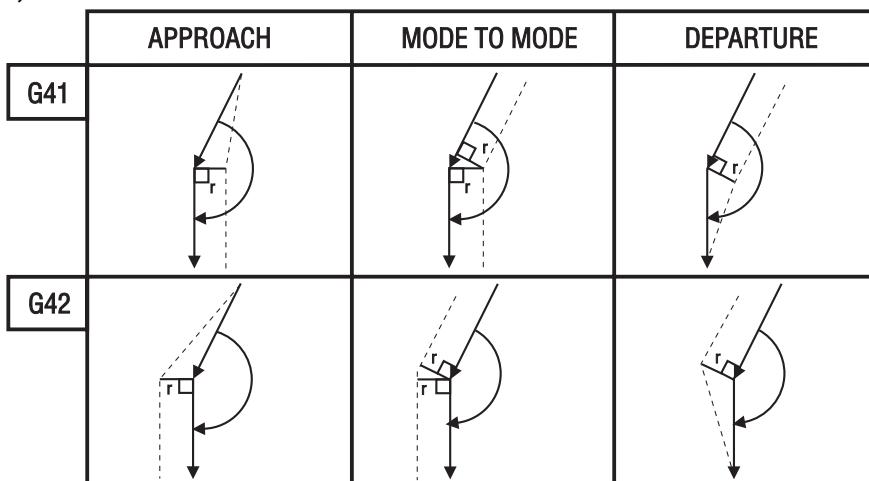
G74	End face grooving cycle, peck drilling
G75	O.D./I.D. grooving cycle, peck drilling
G76	Thread cutting cycle, multiple pass
G92	Thread cutting cycle, modal

**LINEAR TO LINEAR - TYPE A**

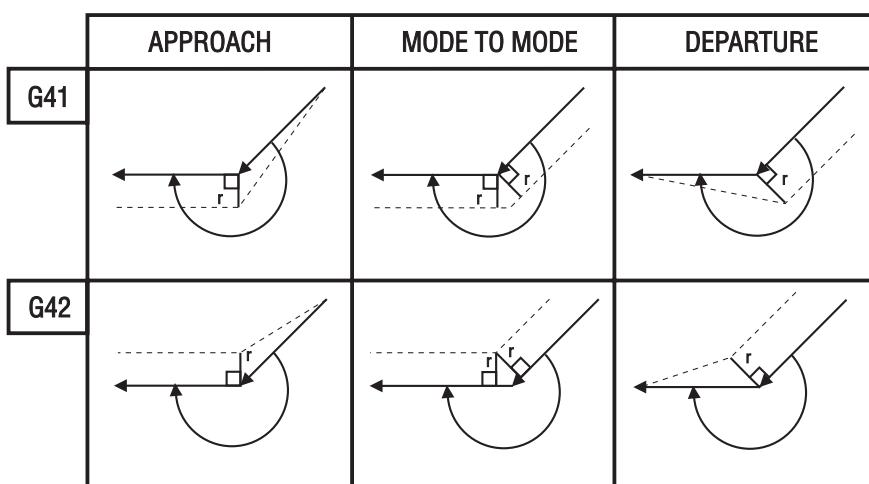
ANGLE: <90



ANGLE: >=90, <180

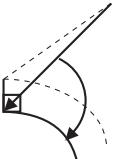
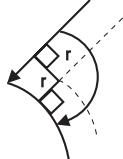
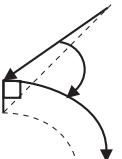
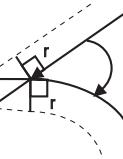


ANGLE: >180

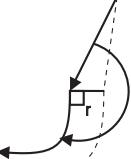
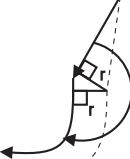
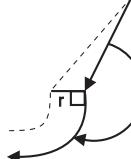
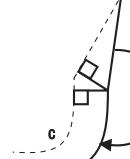
*Figures 4.7-10a shows Type A compensation.*

**LINEAR TO CIRCULAR - TYPE A**

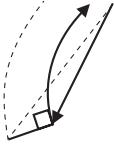
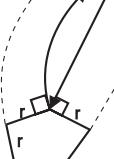
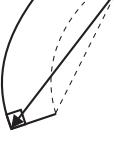
ANGLE: <90

	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

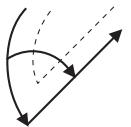
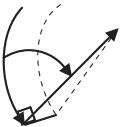
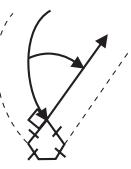
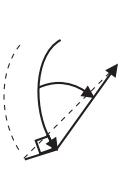
ANGLE: >180

	APPROACH	MODE TO MODE	DEPARTURE
G41			NOT PERMITTED
G42			NOT PERMITTED

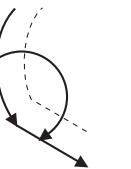
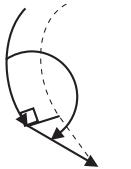
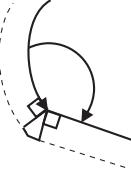
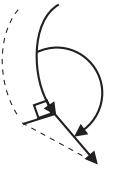
Figures 4.7-10b shows Type A compensation.

**CIRCULAR TO LINEAR - TYPE A**

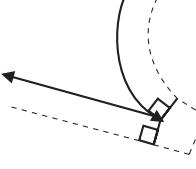
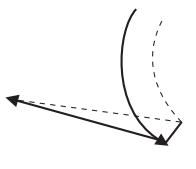
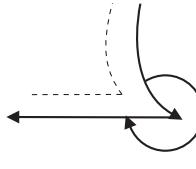
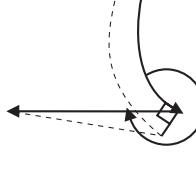
ANGLE: <90

	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		

Figures 4.6-10c shows Type A compensation.

**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

Figures 4.6-10d shows Type A compensation.



The following canned cycles work well when a specific programming sequence is used. This programming sequence is called a TEMPLATE. By using the suggested template the programmer should have no problem using these canned cycles with tool nose compensation. These canned cycles make use of P and Q to identify a path that the canned cycle is to work with.

G70 Finishing Cycle

Usually G70 is used following the use of a G71, G72 or G73, but it can be used alone. Below is the template for using tool nose compensation with G70 alone. Note that TNC approach is part of the PQ path definition sequence, whereas TNC departure is after the execution of G70.

G70 TEMPLATE

```
G28
T101
G97 S_M03
G54 G00 X_Z_
G71...          (ROUGHING CYCLE)
N1 G42 G00 X_Z_ (P)(TNC APPROACH)
G01 X_Z_F_     (DEFINE PATH GEOMETRY)
```

```
...
N2 G40 G00 X_ (Q)(END OF PART PATH) (TNC DEPARTURE)
G70 P_Q_        (EXECUTE G70 FINISHING CYCLE)
G00 Z1.0 M09
G28
M30
```

G71 O.D./I.D. Stock Removal Cycle G72 End Face Stock Removal Cycle

G71 and G72 are similar canned cycles with regard to tool nose compensation. **The finishing and rough finishing passes of G71 and G72 recognize tool nose compensation; however, the roughing pass of these two G codes does not.** The template below can be applied to either G71 or G72.

G71/G72 TEMPLATE

```
G00 X_Z_          (POSITION TO G71/G72 START)
G96 S_M03         (EXECUTE STOCK REMOVAL AND ALLOWANCE)
G71 P1 Q2 U_W_D_F (DEFINE PART PATH PQ SEQUENCE)
N1 G42 G00 X_Z_   (P)(TNC APPROACH)
G_X_Z_F_          (DEFINE PATH GEOMETRY)

...
N2 G40 X_Z_       (Q)(END OF PART PATH)
G00 X_Z_          (TNC DEPARTURE)
G28
```



G73 IRREGULAR PATH STOCK REMOVAL CYCLE

G73 is similar to G71 and G72, except that G73 recognizes TNC on all roughing passes.

G73 TEMPLATE

G00 X__Z__
G96 S__ M03

(POSITION TO G73 START)

(EXECUTE STOCK REMOVAL AND ALLOWANCE)

G73 P1 Q2 U__ W__ I__ K__ D__ F__
(DEFINE PART PATH PQ SEQUENCE)

N1 G42 G00 X__Z__
G__Z__ F__

(P)(TNC APPROACH)
(DEFINE PATH GEOMETRY)

⋮
N2 G40 X__Z__
G40 G00 X__Z__
G28

(Q)(END OF PART PATH)
(TNC DEPARTURE)

The following canned cycles perform their own special compensation.

G90

O.D./I.D. MODAL TURNING CYCLE

G90 performs tool nose compensation independently. Since G90 executes only linear moves, some of the compensation overhead can be eliminated to produce more accurate results on tapered cuts. Figure 4.7-11 shows the four strokes generated for each block G90 is active in. Figure 4.7-11 also shows the various approach and departure moves based on tool tip direction. Figure 4.7-12 indicates which TNC command is active based on the work quadrant being machined.

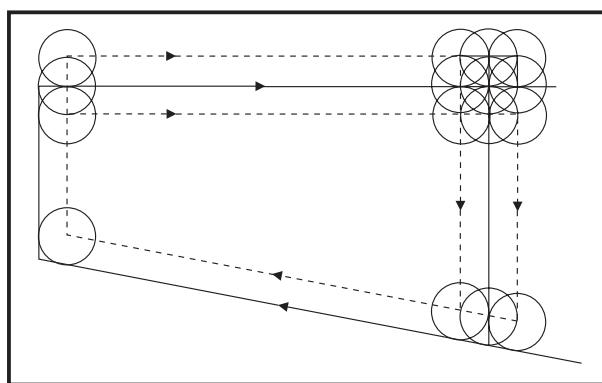


Figure 4.7-11. Tool nose compensation with G90.

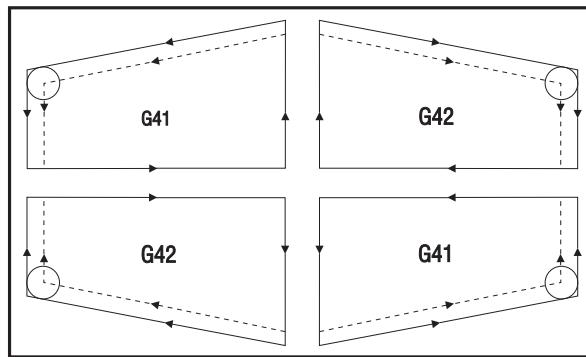


Figure 4.7-12. Offset direction for G90.

G90 TEMPLATE

G00 X__ Z__	(POSITION TO G90 START)
G96 S__ M03	
(ROUGH USING G90 AND TNC)	
G90 G42 X__ Z__ I__ F__	(MODAL G CODE SETUP AND EXECUTION)
X__ Z__	(OPTIONAL ADDITIONAL ROUGH OR FINISH)
X__ Z__	
G0 G40 X__ Z__ M05	(TNC DEPARTURE)
G28	

G94 End Face Cutting Cycle

G94 performs tool nose compensation independently. Since G94 executes only linear moves, some of the compensation overhead can be eliminated to produce more accurate results on tapered cuts. Figure 4.7-13 shows the four strokes generated for each block G94 is active in. Figure 4.7-13 also shows the various approach and departure moves based on tool tip direction. Figure 4.7-14 indicates which TNC command is active based on the work quadrant being machined.

G94 TEMPLATE

G0 X__ Z__	(POSITION TO G94 START)
G96 S__ M03	
(ROUGH USING G94 AND TNC)	
G94 G41 X__ Z__ I__ F__	(MODAL G CODE SETUP AND EXECUTION)
X__ Z__	(OPTIONAL ADDITIONAL ROUGH OR FINISH)
X__ Z__	
G00 G40 X__ Z__ M05	(TNC DEPARTURE)
G28	

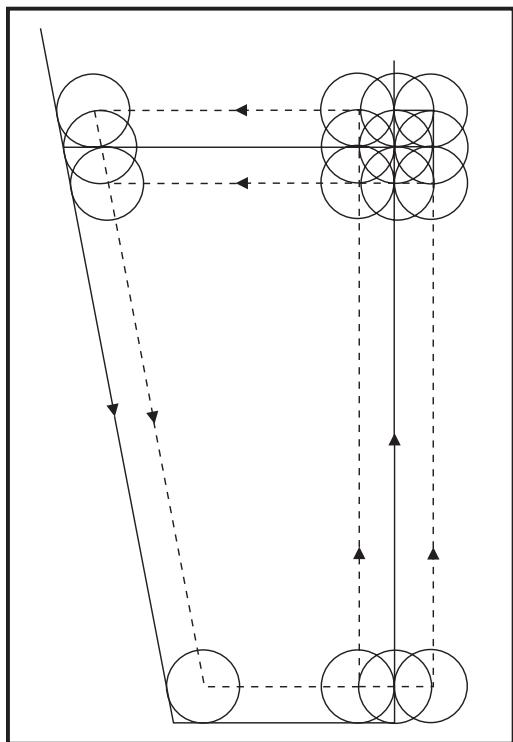


Figure 4.7-13. Tool nose compensation with G94.

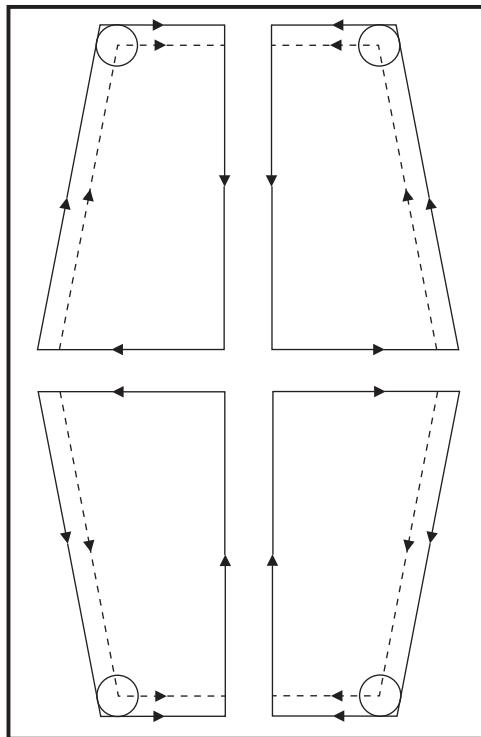


Figure 4.7-14. Offset direction for G94.

TOOL NOSE COMPENSATION G CODES**G40 Tool Nose Compensation Cancel****Group 7**

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

* indicates optional

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



The departure target usually does not correspond with a point on the part. In many cases overcutting or undercutting can occur. Figure 4.7-15 illustrates this.

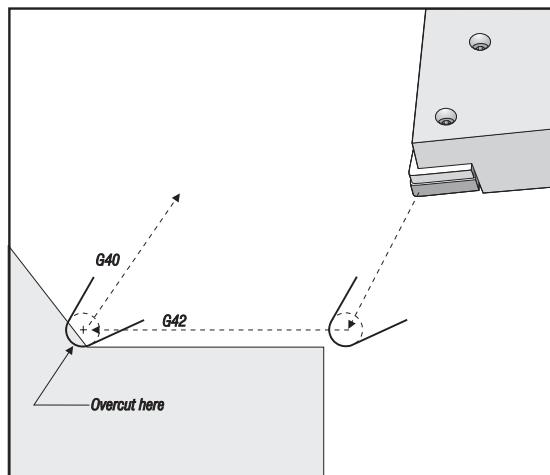


Figure 4.7-15. G40

When address codes I and K are used in a G40 departure block, the control will use these values as an intersection vector for the end point of the last completely compensated motion stroke. Figure 4.7-15 illustrates where I and K lie in relation to the departure stroke. Usually I and K lie along a face of the machined part.

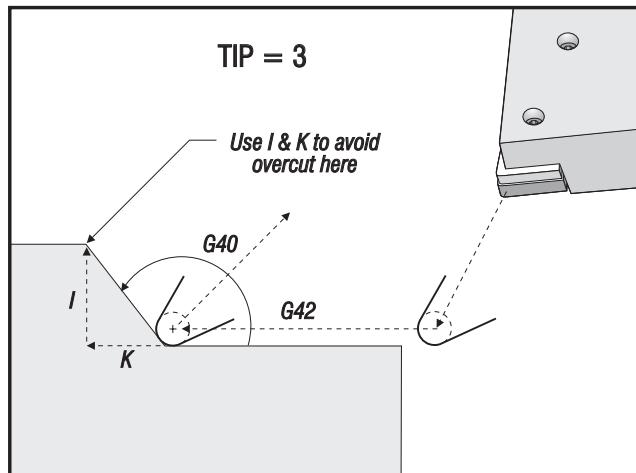
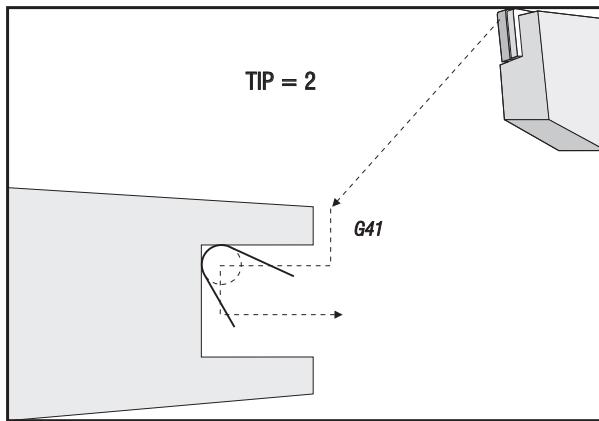


Figure 4.6-16. Use of I and K.

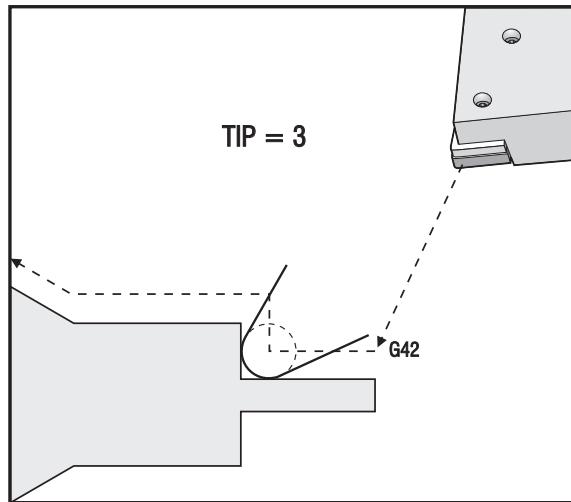
The values of I and K can be determined by calculating the sine of the angle for I and the cosine of the angle for K, where the angle is taken to the face of the part with respect to the Z axis. This is shown in Figure 4.7-16.

**G41 Tool Nose Compensation Left****Group 7**

G41 will select tool nose compensation left; that is, the tool is moved to the left of the programmed path to compensate for the size of a tool. A tool offset must be selected with a Tnxxx code, where xx corresponds to the offsets that are to be used with the tool.

*Figure 4.7-17. G41***G42 Tool Nose Compensation Right****Group 7**

G42 will select tool nose compensation right; that is, the tool is moved to the right of the programmed path to compensate for the size of a tool. A tool offset must be selected with a Tnxxx code, where xx corresponds to the offsets that are to be used with the tool.

*Figure 4.7-18. G42*

**EXAMPLE PROGRAMS USING TOOL NOSE COMPENSATION**

This section is comprised of example programs that use tool nose compensation. At least one example of each G code is given. All of these programs have been proven to work on both HAAS and FANUC compatible controls.

EXAMPLE 1**GENERAL TNC**

This example illustrates tool nose compensation using standard interpolation modes G01/G02/G03.

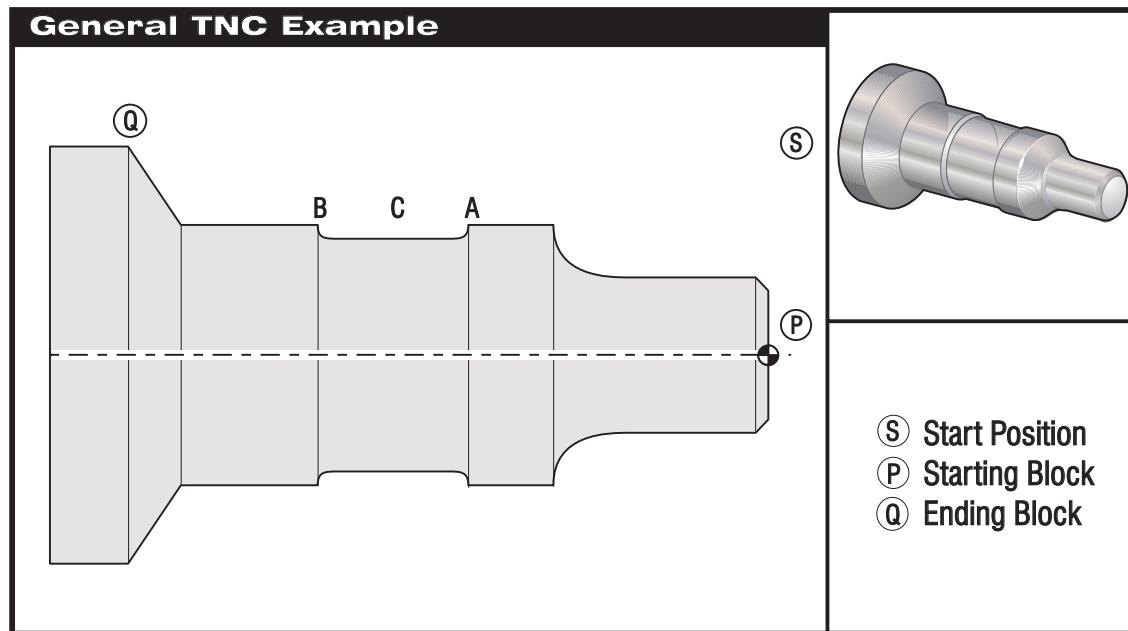


Figure 4.7-19. General TNC

PREPARATION**SETTING 33 FANUC****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
G54		0.0000	0.0000		

**PROGRAM EXAMPLE**

%
O0811 (G42 TEST BCA)
N1 G54 S1000
T101

G97 S500 M03
G54 G00 X2.1 Z0.1
G96 S200

(ROUGH P TO Q WITH T1 USING G71 AND TNC)
G71 P10 Q20 U0.02 W0.005 D.1 F0.015

(DEFINE PART PATH PQ SEQUENCE)

N10 G42 G00 X0. Z0.1 F.01

(P)(G71 TYPE II, TNC RIGHT)

G01 Z0 F.005

X0.65

X0.75 Z-0.05

Z-0.75

G02 X1.25 Z-1. R0.25

(A)

G01 Z-1.5

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 FOR OFFSET 2 IS 3)
(MOVE TO POINT S)

G00 X2.1 Z0.1

G96 S400

(FINISH P TO Q WITH T2 USING G70 AND TNC)

G70 P10 Q20

G97 S750

G28

(ZERO FOR TOOL CHANGE
CLEARANCE)

M01

N3 G50 S1000

T303

(SELECT TOOL 3, OFFSET 3)
(TIP DIRECTION FOR OFFSET 3 IS 3)

G97 S500 M03

(GROOVE TO POINT B USING OFFSET 3)

DESCRIPTION

(EXAMPLE1)

(SELECT TOOL 1 AND OFFSET 1
(TIP DIRECTION FOR OFFSET 1 IS 3))

(MOVE TO POINT S)



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)
(MOVE TO POINT C)(TNC APPROACH)
G00 G41 X1.5 Z-2.125
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 3**TNC WITH G71**

This example uses tool nose compensation with a G71 roughing canned cycle. Refer to Figure 4.7-20 for the part drawing for this example.

PREPARATION

SETTING 33 FANUC

TOOLS

T1 Insert with .032 radius, roughing

TOOL	OFFSET	RADIUS	TIP
T1	01	.032	3

PROGRAM EXAMPLEDESCRIPTION

%	
O0813	(EXAMPLE 3)
G50 S1000	
T101	(SELECT TOOL 1)
G00 X1.5 Z.1	(RAPID TO START POINT)
G96 S100 M03	
(ROUGH P TO Q WITH T1 USING G71 AND TNC)	
G71 P80 Q180 U.01 W.005 D.08 F.012	
(DEFINE PART PATH PQ SEQUENCE)	
N80 G42 G00 X.6	(P) (G71 TYPE I, TNC RIGHT)
G01 Z0 F.01	(START OF FINISH PART PATH)
X0.8 Z-0.1 F.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	
G28	(TNC CANCEL)
M30	(ZERO X FOR TOOL CHANGE CLEARANCE)
%	

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

**EXAMPLE 4****TNC WITH G72**

This example uses tool nose compensation with a G72 roughing canned cycle.

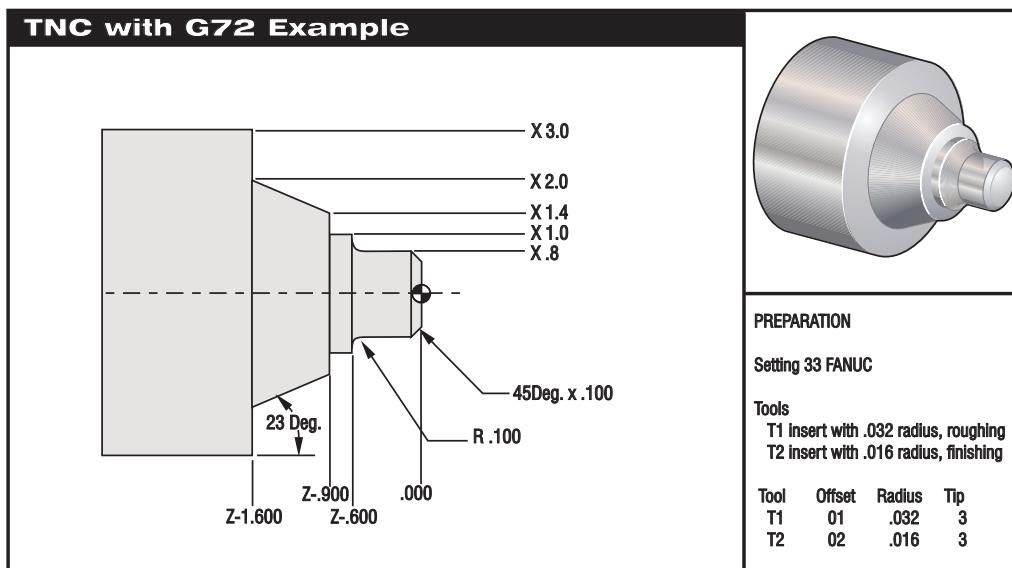


Figure 4.7-21. Part drawing for TNC with G72 example.

PROGRAM EXAMPLE**DESCRIPTION**

%
O0814 (EXAMPLE 4)
G50 S1000
T101 (SELECT TOOL 1)
G00 X3.5 Z.1 (MOVE TO START POINT)
G96 S100 M03

(ROUGH P TO Q WITH T1 USING G72 AND TNC)

G72 P80 Q180 U.005 W.01 D.05 F.010

(DEFINE PART PATH PQ SEQUENCE)

N80 G41 G00 Z-1.6

(P) (G72 TYPE I, TNC LEFT)

G01 X2.0 F0.005

X1.4 Z-0.9

X1.0

Z-.60

G03 X0.8 Z-0.5 K0.1

G01 Z-0.1

X0.6 Z0.0

X0.0

N180 G40 G00 Z.01

(TNC CANCEL)

(*****OPTIONAL FINISHING SEQUENCE*****)

G28

(ZERO FOR TOOL CHANGE CLEARANCE)

M01

T202

(SELECT TOOL 2)



N2 G50 S1000

G00 X3.5 Z.1

G96 S325 M03

(MOVE TO START POINT)

G70 P80 Q180

G00 Z.5 M5

G28

M30

%

(FINISH P TO Q WITH T2 USING G70 AND
TNC)

(ZERO FOR TOOL CHANGE CLEARANCE)

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 5**TNC WITH G73**

This example uses tool nose compensation with a G73 roughing canned cycle. Refer to Figure 4.7-21 for the part drawing for this 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**

%
O0815
T101
G50 S1000
G00 X3.5 Z.1
G96 S100 M03
(ROUGH P TO Q WITH T1 USING G73 AND TNC)
G73 P80 Q180 U.01 W.005 I0.3 K0.15 D4 F.012
(DEFINE PART PATH PQ SEQUENCE)
N80 G42 G00 X0.6
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
G00 Z0.1 M05
(*****OPTIONAL FINISHING SEQUENCE*****)
G28
M01
T202
N2 G50 S1000
G00 X3.0 Z.1
G96 S100 M03
(FINISH P TO Q WITH T2 USING G70 AND TNC)
G70 P80 Q180
G00 Z.5 M05
G28
M30
%

(EXAMPLE 5)
(SELECT TOOL 1)
(MOVE TO POINT S)
(P) (G72 TYPE I, TNC RIGHT)
(Q)
(TNC CANCEL)
(ZERO FOR TOOL CHANGE CLEAR
ANCE)
(SELECT TOOL 2)
(MOVE TO START POINT)
(ZERO FOR TOOL CHANGE
CLEARANCE)

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

**EXAMPLE 6****TNC WITH G90**

This example uses tool nose compensation with a G90 modal rough turning cycle.

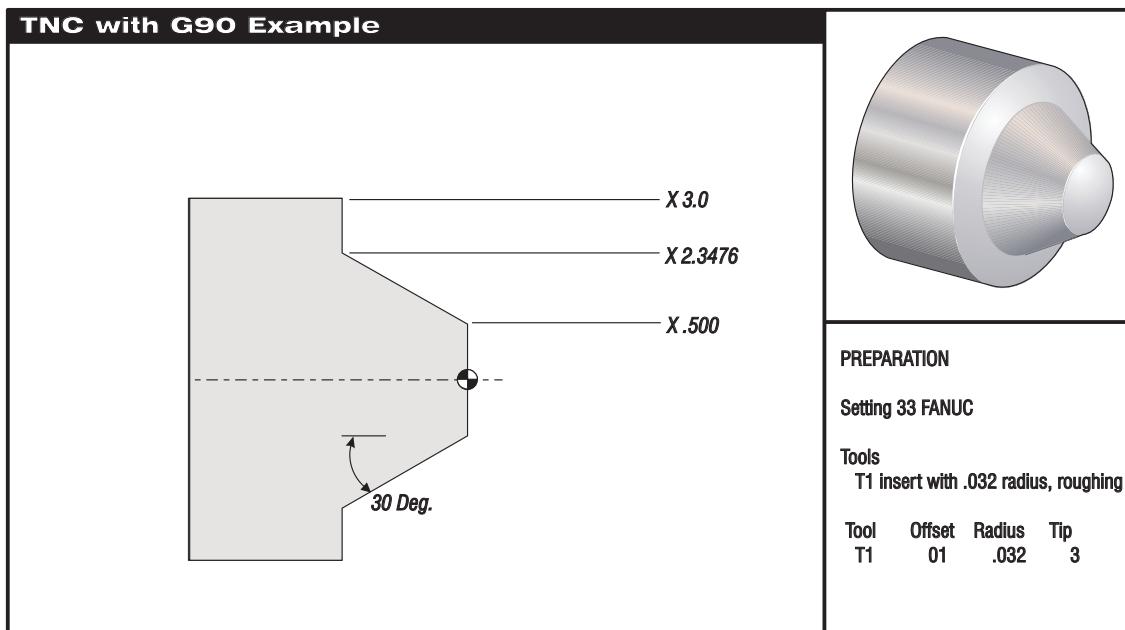


Figure 4.7-22. Part drawing for TNC with G90 example.

PROGRAM EXAMPLE

```
%  
O0816  
T101  
G50 S1000  
G00 X4.0 Z0.1  
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

G28

M30

%

DESCRIPTION

(EXAMPLE 6)
(SELECT TOOL 1)

(MOVE TO START POINT)

(TNC CANCEL)
(ZERO FOR TOOL CHANGE
CLEARANCE)

**EXAMPLE 7****TNC WITH G94**

This example uses tool nose compensation with a G94 modal rough turning cycle.

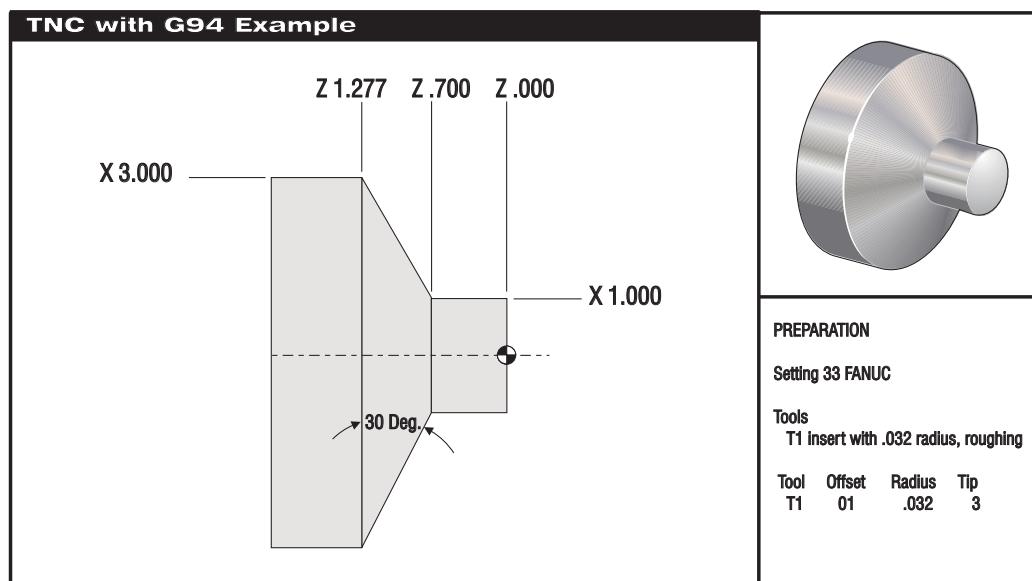


Figure 4.7-23. Part drawing for TNC with G94 example.

PROGRAM EXAMPLE

%
O0817
G50 S1000
T101
G00 X3.0 Z0.1
G96 S100 M03

G94 G41 X1.0 Z-0.5 K-0.577 F.03
Z-0.6
Z-0.7
G00 G40 X3. Z0.1 M05
G28
M30
%

DESCRIPTION

(EXAMPLE 7)

(SELECT TOOL 1)
(MOVE TO START POINT)

(ROUGH 30 DEG. ANGLE TO X1. AND Z-0.7
USING G94 AND TNC)

(OPTIONAL ADDITIONAL PASSES)

(TNC CANCEL)
(ZERO FOR TOOL CHANGE CLEARANCE)

**4.9 PROGRAMMING WITHOUT TOOL NOSE COMPENSATION**

This section is to help those who have not had the time or patience to learn how to use TNC. Our suggestion is if you want to save time and money in the long run, TNC can help.

MANUALLY CALCULATING COMPENSATION

When you program a straight line in either X or Z the tool tip touches the part at the same point where you touched your original tool offsets in X and Z. 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 insert you are using. If you were to try and program your part without using any compensation you would see over and undercutting in your part.

The following pages contain tables and illustrations demonstrating how to calculate the compensation in order to program your part accurately. There are 2 tables with 1/32 and 1/64 insert radius values in both the X and Z axes.

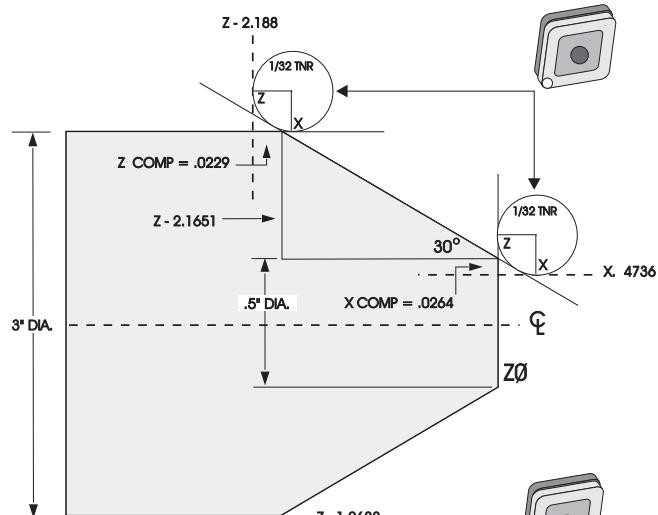
Along with each chart are 3 illustrated examples of compensation using both types of inserts and cutting along 3 different angles. Next to each illustration is a sample program and explanation of how the compensation was calculated.

Refer to the illustrations on the following pages while reading the text below.

1. The tool tip is shown as a circle with X and Z points called out. These points designate where you would normally touch off the X diameter and Z face offsets.
2. Each illustration is a 3" diameter part with lines extending off the part and intersecting at 30°, 45° and 60° angles.
3. The point at which the tool tip comes to rest on both intersection lines is where we measure the compensation value.
4. This compensation value is the distance from the face of the tool tip measured 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 avoid any over or undercutting.
5. Use the values found on the charts (angle and radius size) to calculate the correct tool path position for your program.



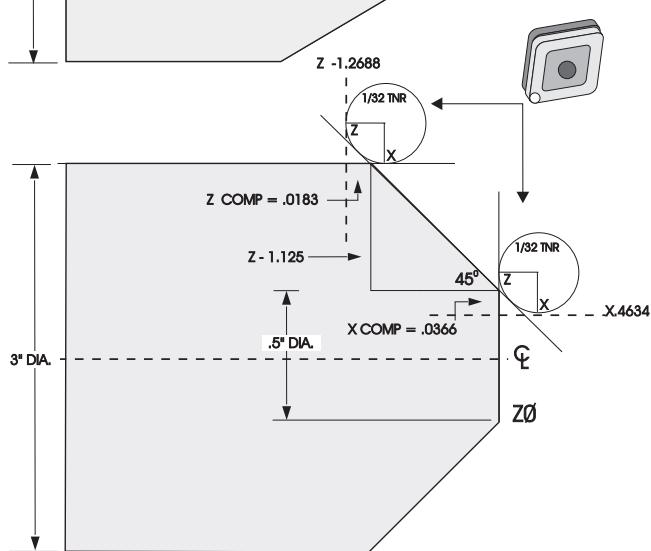
TOOL NOSE RADIUS CALCULATION DIAGRAM



PROGRAM

CODE	COMPENSATION (1/32 TNR)
G0 X0 Z.1	(1/32 TNR)
G1 Z0	
X.4736	(X.5 - 0.0264 COMP)
X 3.0 Z-2.188	(Z-2.1651 + 0.0229 COMP)

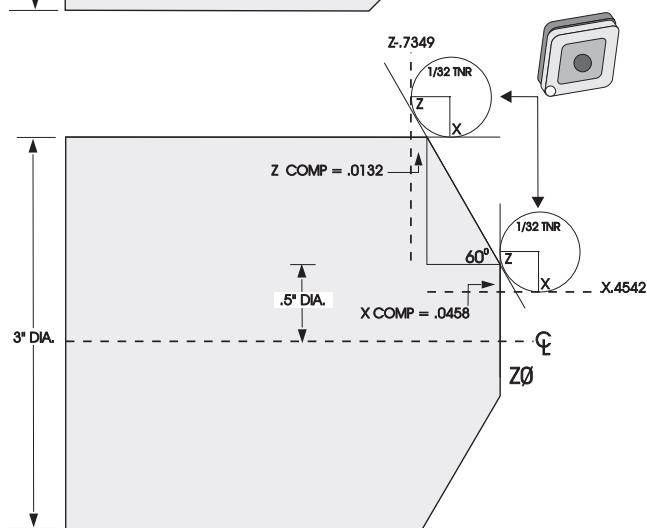
NOTE: COMPENSATION VALUE FOR 30° ANGLE



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-1.25 +

NOTE: COMPENSATION VALUE FOR 45° ANGLE



PROGRAM

CODE	COMPENSATION (1/32 TNR)
G0 X0 Z.1	
G1 Z0	
X.4542	(X.5 - 0.0458 COMP)
X 3.0 Z-.4322	(Z-.7217 + 0.0132)

NOTE: COMPENSATION VALUE FOR 60° ANGLE



TOOL RADIUS AND ANGLE CHART

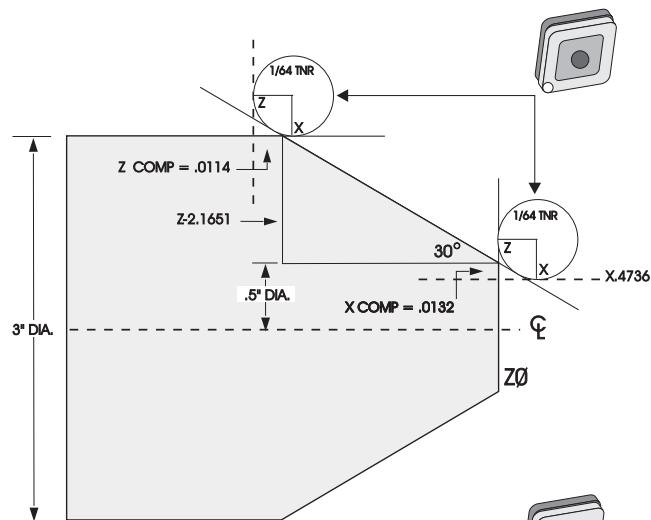
1/32 RADIUS

NOTE: THE X MEASUREMENT CALCULATED IS BASED ON PART DIAMETER

ANGLE	X _c CROSS	Z _c LONGITUDINAL	ANGLE	X _c CROSS	Z _c 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			

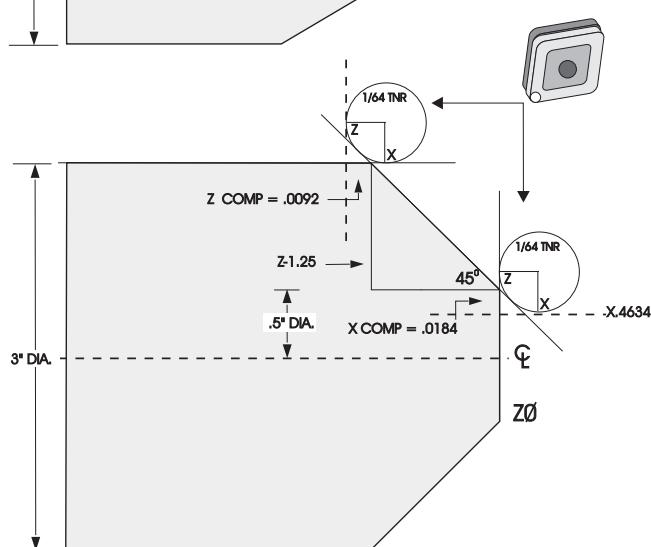


TOOL NOSE RADIUS CALCULATION DIAGRAM



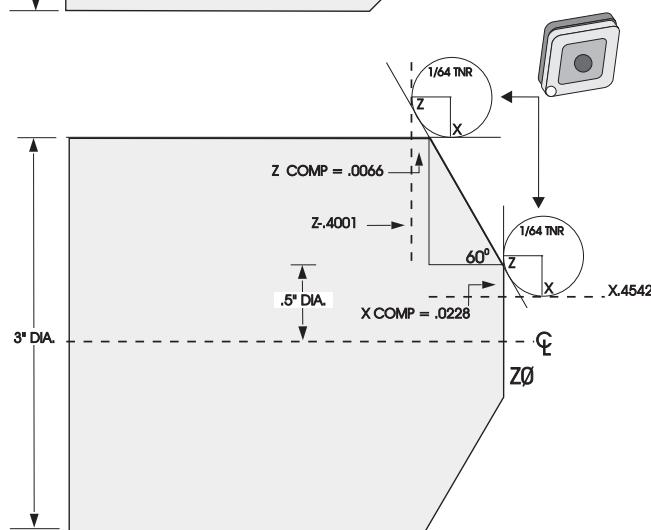
PROGRAM	
CODE	COMPENSATION (1/64 TNR)
G0 X0 Z1 G1 Z0 X.4868 X 3.0 Z-2.1765	(X.5 - 0.0132 COMP) (Z-2.1651 + 0.0114 COMP)

NOTE: USING COMPENSATION VALUES FOR 30°



PROGRAM	
CODE	COMPENSATION (1/64 TNR)
G0 X0 Z.1 G1 Z0 X.4816 X 3.0 Z-1.2592	(X.5 - 0.0184 COMP) (Z-1.25 + 0.0092 COMP)

NOTE: USING COMPENSATION VALUES FOR 45°



PROGRAM	
CODE	COMPENSATION (1/64 TNR)
G0 X0 Z1 G1 Z0 X.4772 X 3.0 Z-.467	(X.5 - 0.0228 COMP) (Z-.4001 + 0.0066 COMP)

NOTE: USING COMPENSATION VALUES FOR 60°



TOOL RADIUS AND ANGLE CHART

1/64 RADIUS

NOTE: 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			



TOOL NOSE COMPENSATION

SL OPERATOR'S MANUAL
Series

June 2001



5. G CODES - PREPARATORY FUNCTIONS

The following is a G codes summary. A “*” indicates the default within each group, if there is one.

Code:	Group:	Function:	Description On Page:
G00	01*	Rapid Motion	161
G01	01	Linear Interpolation Motion	161
G01	01	Chamfering and Corner Rounding	190
G02	01	CW Interpolation Motion	162
G03	01	CCW Interpolation Motion	163
G04	00	Dwell	164
G05	00	Fine Spindle Control Motion (optional)	207
G09	00	Exact Stop	164
G10	00	Set Offsets	164
G14	17	Sub-Spindle Swap	165
G15	17	Sub Spindle Mode Cancel	165
G17	02	XY Plane	166
G18	02*	ZX Plane Selection	167
G19	02	YZ Plane Selection	166
G20	06*	Selection Inch	167
G21	06	Select Metric	167
G28	00	Return To Reference Point	167
G29	00	Return From Reference Point	167
G31	00	Feed Until Skip (optional)	168
G32	01	Threading	168
G40	07*	Tool Nose Compensation Cancel	170
G41	07	Tool Nose Compensation Left	170
G42	07	Tool Nose Compensation Right	170
G50	11	Spindle Speed Clamp / Set Global Coor. Offset	170
G51	11	Cancel Offset (Yasnac)	172
G52	00	Set Local Coordinate System (Fanuc)	172
G53	00	Machine Coord. Selection	172
G54	12*	Select Work Coordinate System 1	172
G55	12	Select Work Coordinate System 2	172
G56	12	Select Work Coordinate System 3	172
G57	12	Select Work Coordinate System 4	172
G58	12	Select Work Coordinate System 5	172
G59	12	Select Work Coordinate System 6	172
G61	13	Exact Stop Modal	173
G64	13*	G61 Cancel	173
G65	00	Macro Subroutine Call (optional)	385
G70	00	Finishing Cycle	174
G71	00	O.D./I.D. Stock Removal Cycle	175
G72	00	Face Stock Removal Cycle	181
G73	00	Irregular Path Stock Removal Cycle	184
G74	00	Face Grooving Cycle, Peck Drilling	186
G75	00	O.D./I.D. Grooving Cycle, Peck Drilling	187
G76	00	Threading Cycle, Multiple pass	188
G77	00	Flattening Cycle (optional)	210
G80	09*	Canned Cycle Cancel	192
G81	09	Drill Canned Cycle	192
G82	09	Spot Drill Canned Cycle	192
G83	09	Normal Peck Drill Canned Cycle	194



G84	09	Tapping Canned Cycle	195
G85	09	Boring Canned Cycle	196
G86	09	Bore/Stop Canned Cycle	197
G87	09	Bore/Manual Retract Canned Cycle	198
G88	09	Bore/Dwell/Manual Retract Canned Cycle	199
G89	09	Bore/Dwell Cannxed Cycle	200
G90	01	O.D./I.D. Turning Cycle, Modal	201
G92	01	Threading Cycle, Modal	202
G94	01	End Facing Cycle, Modal	203
G95	09	Live Tool Rigid Tap (optional)	213
G96	12	Constant Surface Speed On	204
G97	12*	Constant Surface Speed Cancel	205
G98	05	Feed per Minute	205
G99	05*	Feed per Revolution	205
G100	00	Disable Mirror Image	205
G101	00	Enable Mirror Image	205
G102	00	Programmable Output To RS-232	206
G103	00	Limit Block Lookahead	206
G110	12	Select Work Coordinate System 7	207
G111	12	Select Work Coordinate System 8	207
G112	12	Cartesian to Polar Transformation Enable	166
G113	12	Cartesian to Polar Transformation Disable	166
G114	12	Select Work Coordinate System 11	207
G115	12	Select Work Coordinate System 12	207
G116	12	Select Work Coordinate System 13	207
G117	12	Select Work Coordinate System 14	207
G118	12	Select Work Coordinate System 15	207
G119	12	Select Work Coordinate System 16	207
G120	12	Select Work Coordinate System 17	207
G121	12	Select Work Coordinate System 18	207
G122	12	Select Work Coordinate System 19	207
G123	12	Select Work Coordinate System 20	207
G124	12	Select Work Coordinate System 21	207
G125	12	Select Work Coordinate System 22	207
G126	12	Select Work Coordinate System 23	207
G127	12	Select Work Coordinate System 24	207
G128	12	Select Work Coordinate System 25	207
G129	12	Select Work Coordinate System 26	207
G184	09	Reverse Tap Canned Cycle	215
G186	09	Reverse Live Tool Rigid Tap (optional)	213
G187	00	Accuracy Control for High Speed Machining	207
G195	00	Live Tool Vector Tapping	216
G196		Reverse Live Tool Vector Tapping	216
G200		Index on the Fly	216

The G Code System is selected using Setting 34. The HAAS lathe currently implements only system A codes.

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



There is also one case where the Group 01 G codes will cancel the Group 9 (canned cycles) codes. If a canned cycle is active (G81 through G89), the use of G00 or G01 will cancel the canned cycle.

The control will store up to 500 G-code programs in memory.

RAPID POSITION COMMANDS (G00)

G00 Rapid Motion Positioning

Group 01

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

Generally, rapid motions will not be in straight lines. All of the axes specified are moved at the same time but will not necessarily complete their motions at the same time. The control will wait until all motions are complete. Only the axes specified are moved and the incremental or absolute commands will change how those values are interpreted.

INTERPOLATION COMMANDS (G01, G02, G03)

G01 Linear Interpolation Motion

Group 01

- F Feed rate
- *B B-axis incremental 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 provides for straight line (linear) motion from point to point. Motion can occur in 1 or 2 dimensions. Both axes will start and finish motion at the same time. The speeds of all axes are controlled so that the feed rate specified is achieved along the actual path. The F command is modal and may be specified in a previous block. Only the axes specified are moved and the incremental or absolute commands will change how those values are interpreted. The auxiliary axes B, C and V can also be moved with a G01 but only one axis is moved at a time.

**G02 CW Circular Interpolation Motion****Group 01**

- F Feed rate
- *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

* indicates optional

This G code is used to specify a clockwise circular motion of two of the linear axes. Circular motion is possible in the X and Z axes as selected by G18. The X and Z are used to specify the end point of the motion that can use either absolute or incremental motion. 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. These are further described, and shown on the following pages:

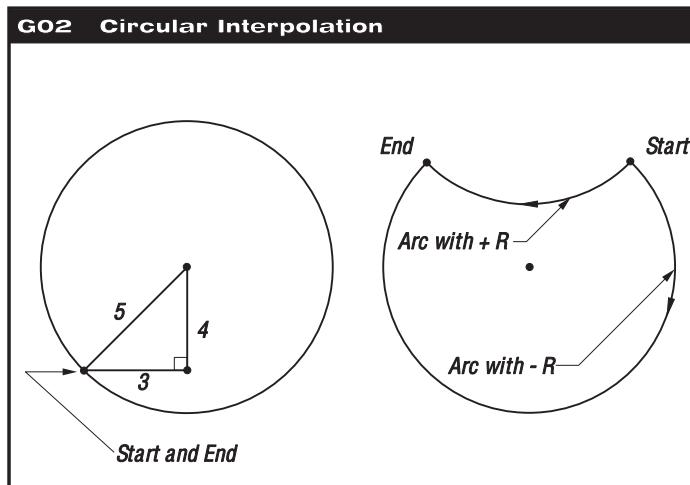


Figure 5.0-1

I,K: When I or K are used to specify the center of the arc, R may not be used. If only one of I or K is specified, the other is assumed to be zero. The I or K is the signed distance from the starting point to the center of the circle. Small errors in these values are tolerated up to 0.0010 inches.

In most cases it is much easier to use R instead of I and K.

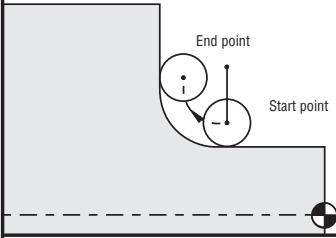
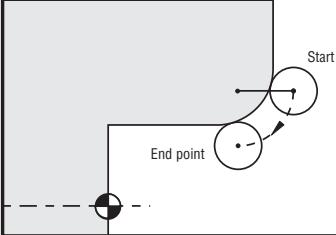
G02 U & W (Incremental Distances for X & Z)	
 <p>End point Start point</p>	<p>= R value .100 Rad. and .0312 Rad. tool</p> <p>Radius of Part minus Radius of Tool for <i>Concave radius</i>.</p> <p>G02 U.1376 W-.0688 R.0688(I.0688)</p>
 <p>Start point End point</p>	<p>= R value .100 Rad. and .0312 Rad. tool</p> <p>Radius of Part plus Radius of Tool for <i>Convex radius</i>.</p> <p>G02 U.-2624 W-.1312 R.1312(K-.1312)</p>

Fig. 5.0-2 G02

R: When R is used to specify the center of the circle, X or Z is required to specify an endpoint different from the starting point. R is the distance from the starting point to the center of the circle. With a positive R, the control will generate a path of 180 degrees or less; to generate a radius of over 180 degrees, specify a negative R. Small errors in this value are tolerated up to 0.0010 inches.

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

G01 X3.0 Z4.0
G02 Z-3.0 R5.0

G03 CCW Circular Interpolation Motion

Group 01

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

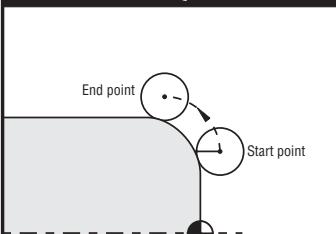
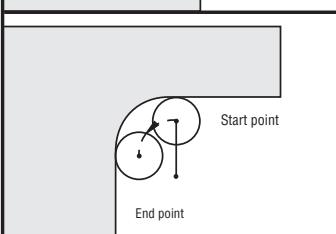
G03 U & W (Incremental Distances for X & Z)	
 <p>End point Start point</p>	<p>= R value .100 Rad. and .0312 Rad. tool</p> <p>Radius of Part plus Radius of Tool for <i>Convex radius</i>.</p> <p>G03 U.2624 W-.1312 R.1312(K-.1312) (Radius plus Radius)</p>
 <p>Start point End point</p>	<p>= R value .100 Rad. and .0312 Rad. tool</p> <p>Radius of Part minus Radius of Tool for <i>Concave radius</i>.</p> <p>G03 U.-1376 W-.0688 R.0688(I-.0688) (Radius minus Radius)</p>

Fig. 5.0-3 G03

**MISCELLANEOUS G CODES (G04, G09)****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 in the P code. If the P does not have a decimal the delay is in milliseconds (0.001 seconds); otherwise the delay is in seconds.

G09 Exact Stop**Group 00**

The G09 code is used to specify exact stop. It is not modal and does not affect the following blocks. 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.

PROGRAMMABLE OFFSET SETTING (G10)**G10 Set Offsets****Group 00**

- L Selection of geometry, wear, shift or work coordinates
- P Selection of offset number
- 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

This G code can be used to change the tool geometry/shift offsets, wear offsets, or the work offsets from a running program. The following codes are used for selection of offsets:

- 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

The following P codes are used to index the appropriate offsets:

- | | | |
|----------|--|---------|
| 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 |



The R,X, and Z codes are signed numbers with fractions in inches (or millimeters). The U and W are incremental. If Setting 33 is FANUC, using P51 to P100 will set tool geometries for tool 1 to 50. It is recommended that L10 and P1 through P50 be used to set tool geometries if FANUC compatibility is desired.

G10 examples:

G10 L2 P1 U6.0	(Move coordinate G54 6.0 units to the right);
G10 L20 P2 X-10.0 Z-8.	(Set work coordinate G111 to X-10.0, Z-8.0);
G10 L10 P51 X10.	(Set X tool shift of Tool #1 to 10);
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");

Sub-Spindle G Codes

G14 and G15 Main-Spindle/Sub-Spindle Swap

Group 17

Commanding a G14 causes the sub-spindle to become the primary spindle and to receive certain commands which would normally cause motion of the main spindle. After a G14 has been commanded:

M03, M04, M05 and M19 will affect the sub-spindle.

M143, M144, M145 and M119 will generate alarm 329 UNDEFINED M CODE.

G50 will limit the sub-spindle speed. G96 will set the sub-spindle surface feed value. The sub-spindle will adjust its speed according to G50 and G96 when there is motion in X.

G01 Feed Per Rev will feed based on the sub-spindle.

Note: G14 causes mirroring to be turned on so that Z motion in the negative direction is toward the sub-spindle, and Z motion in the positive direction is away from the sub-spindle. If Z is already mirrored due to setting 47 or G101, Z will be unmirrored. G14 is canceled by a G15, and M30, reaching the end of a program, and by pressing RESET.

If G14 is commanded on a machine without a sub-spindle, alarm 310 INVALID G-CODE will be generated. When a G14 has been commanded:

The area currently displaying ACT (actual spindle speed) will be replaced with SSC (commanded sub-spindle speed) and the line below will display SSA (actual sub-spindle speed.) The area of the screen that normally displays mirroring will alternate between displaying mirroring status and SUBSPIN. The Z axis mirroring status displayed will reflect only the status caused by setting 47 and G101.

**CARTESIAN TO POLAR TRANSFORMATION****G17 XY Plane and G19 YZ Plane**

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

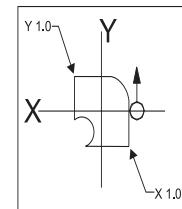
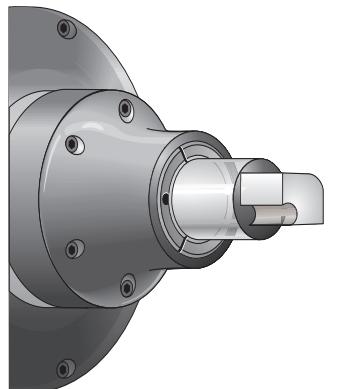
G112 XY to XC interpretation**Group 04**

The G112/G113 Cartesian to Polar coordinate transformation feature allows the user to program subsequent blocks in Cartesian XY coordinates, which the control will automatically convert to polar XC coordinates. G112 activates this feature. While it is active, G18 ZX plane is used for G01 linear XY strokes and G17 YZ plane is used for G02 and G03 XY interpolated circular motion.

G113 G112 Cancel

G113 cancels the Cartesian to Polar coordinate conversion.

T101;
G54;
G98;
GZ;
X40.M154;
C;
M133
P3000;
G112;
G17;
G1ZF300.;
Y10.Z-2.;
G3X10.Y20.Z-4.R10.;
G1X-20.Z-6.;
Y-5.Z-8.;
G2X-5.Y-20.Z-6.I15.J;
G1X20.Z-4.;
YZ-2.;
G113;
M135;
M155;
G54;
G99;
M30;




CIRCULAR PLANE SELECTION (G18)
G18 ZX Plane Selection
Group 02

The G18 code is used to select the ZX plane for circular motion. It is modal and applies to all following circular motions until another Group 02 is found. This is the default for the HAAS lathe. This means that a circular motion in the plane of the X-Z work space may be programmed without first selecting G18. In this plane, circular motion is defined as clockwise for the operator looking down onto the X-Z work space from the front. This is the motion of the tool relative to the part.

SELECT INCH / METRIC (G20, G21)

The standard G codes G20 and G21 are sometimes used to select between inch and metric BUT, in this control, the G20 (inch) and G21 (mm) codes can only be used to insure that the inch/metric setting is set correctly for that program.

Selection between inch and metric programming can only be done from the Setting 9.

REFERENCE POINT DEFINITION AND RETURN (G28, G29)
G28 Return To MACHINE ZERO, set optional G29 REFERENCE point Group 00

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

G29 Return from REFERENCE Point
Group 00

The G29 code is used to move the axes to a commanded **X** or **Z** position. The axes that are selected in this block are moved to the G29 reference point saved or recorded in G28 and then moved to **X** or **Z** specified in the G29 command point. The positions are interpreted in the current coordinate system.

**FEED UNTIL SKIP (G31)****G31 Skip Function** (This G-code is optional and requires a probe)**Group 00**

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

* indicates optional

The skip function is a non-modal operation that causes a linear move to the specified X or Z position. It applies only to the block in which G31 is specified. A feed rate must be defined previously or in this block. The specified move is started and continues until the end point or the skip signal. The skip signal is a discrete input that usually indicates that the end of travel has been reached; this is usually a probe. The keypad will beep when the end of travel is reached. Cutter compensation may not be active during a skip function. M78 or M79 may be used to test if the skip signal was received.

THREAD CUTTING (G32)**G32 Thread Cutting****Group 01**

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

NOTES:

- All the above commands are optional.
- Feed rate is equivalent to thread lead.
- Movement on at least one axis must be specified.
- Tapered threads have lead in both X and Z. In this case set the feed rate to the larger of the two leads. G99 (Feed per Revolution) must be active.
- This G code is modal and will be cancelled by another Group 01 G code, such as G00,G01,G02,G03, or RESET.

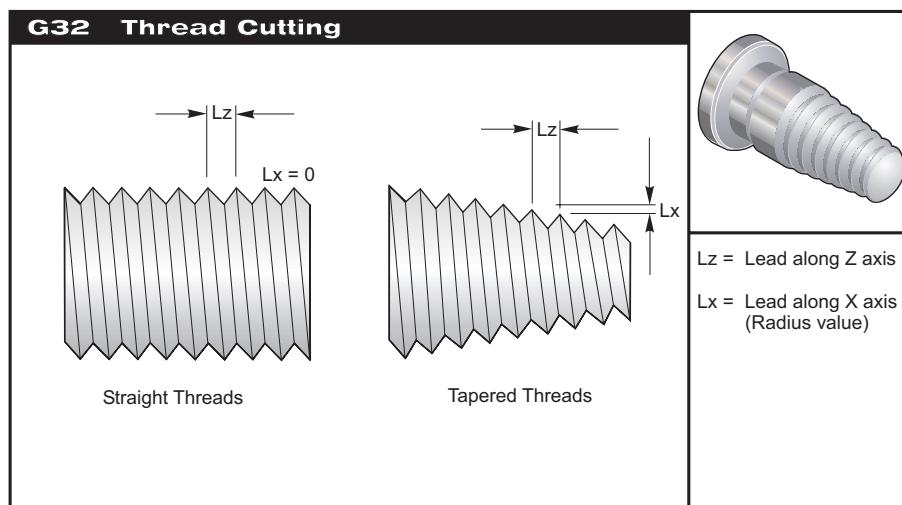


Figure 5.0-4. Definition of Lead (Feed Rate) for Straight and Tapered Threads



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. G32 is modal and is meant to be used with one or more consecutive blocks. The control considers a series of consecutive G32 blocks to be one block (G32 sequence). This means a Single Block Stop can not be performed until the last G32 line is executed. Since G32 is modal, it is not necessary to include "G32" on every line. G32 remains active until another Group 01 G code is encountered or RESET is pressed.

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).

Always start and end threading operations far enough from work piece to avoid deformed threads due to servo acceleration/deceleration delays.

NOTE: SINGLE BLOCK STOP and FEED HOLD are deferred until last line of a G32 sequence.

NOTE: FEED RATE OVERRIDE is ignored while G32 is active. Actual Feed Rate will always be 100% of programmed feed rate.
 M23 and M24 have no effect on a G32 operation, the user must program chamfering if needed.
 G32 must not be used inside any G-Code Cycles (i.e.: G71).
 Changing spindle RPM during threading may cause unexpected results.

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

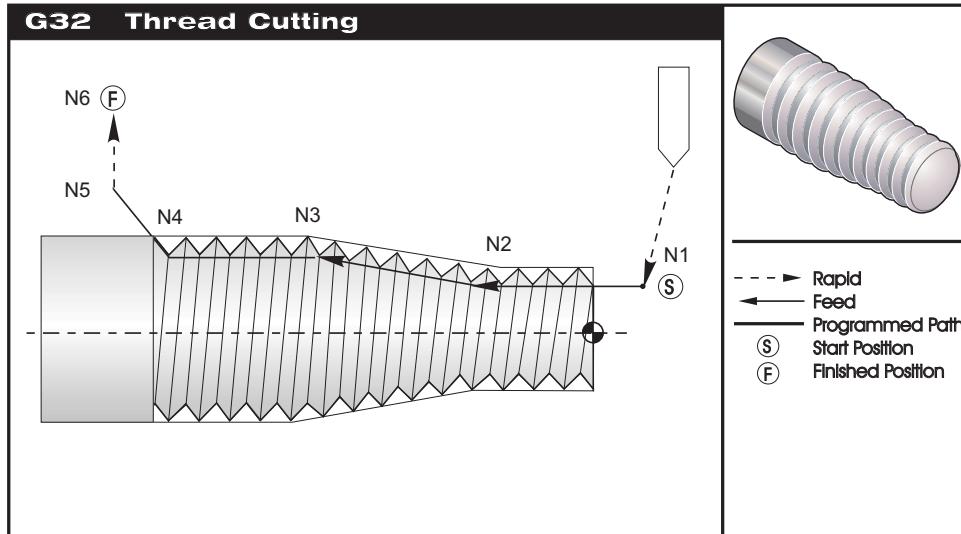


Figure 5.0-5. Straight to Taper, Taper to Straight Thread Cutting Cycle

**G32 PROGRAM EXAMPLE****COMMENTS**

...
G97 S400 M03
N1 G00 X0.25 Z0.1
N2 G32 Z-0.26 F0.065
N3 X0.455 Z-0.585
N4 Z-0.9425
N5 X0.655 Z-1.0425
G00 X1.2
G00 Z0.1
...

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

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

TOOL NOSE COMPENSATION (G40, G41, G42)

Refer to the "Tool Nose Compensation" section for the detailed description of these G codes.

G40 Tool Nose Comp Cancel**Group 07**

G40 cancels G41 or G42.

G41 Tool Nose Compensation Left**Group 07**

G41 will select tool nose compensation left; that is, the tool is moved to the left of the programmed path to compensate for the size of a tool tip radius

G42 Tool Nose Compensation Right**Group 07**

G42 will select tool nose compensation right; that is, the tool is moved to the right of the programmed path to compensate for the size of a tool tip radius.

SETTING A GLOBAL COORDINATE (G50)**G50 SET Global coordinate Offset FANUC, YASNAC****Group 11**

- 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, specify 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. This method can be used only when Setting 33 is 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 of the form Txxyy where xx is from 51 to 100 and yy is from 00 to 50. 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 Tool7, 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)

To set a maximum spindle speed, specify G50 with an S address code. After execution, the spindle will not exceed the RPM value indicated with S.

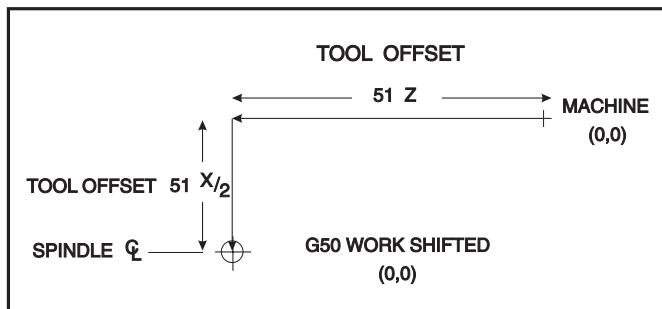
Example G50 S1800 (Spindle speed will not exceed 1800 RPM);

Work Coordinate Shift (YASNAC)

When the control has setting 33 set to YASNAC there are 50 tool shift offsets that can be used. These are found on the offsets page and are numbered 51 to 100. Tool shifts work differently than work coordinates. Typically each tool is assigned a unique tool shift. Before a tool change the appropriate tool shift is applied by using G50. Tool shifts can be set using the setup keys and setup procedures as outlined in the "Setup Procedures" section of this manual. Below is a typical tool change sequence using G50 to apply a tool shift offset.

N1 G51 ;	(RETURN TO MACHINE ZERO, CANCEL OFFSETS)
N2 G50 T5700 ;	(APPLY TOOL SHIFT - OFFSET TO TOOL 7)
N3 T700 M3 ;	(CHANGE TO TOOL 7, TURN ON SPINDLE)
N4 G0 X1.9428 T707 ;	(RAPID TO X, APPLY WEAR FOR TOOL 7 FIRST)

X Value tool shifts 51 through 100 are represented in diameter values when the X axis is configured for diameter programming.



000101 PROGRAM TO SET UP FIG 5-4
N1 G51 (RETURN TO MACHINE ZERO)
N2 G50 T5100; (SHIFT FOR TOOL 1)
:
% TOOL X DIAMETER AND Z SHIFT

Fig. 5.0-6 G50 YASNAC Tool Shift

G50 Spindle Speed Clamp

G50 can be used to clamp the maximum spindle speed. The control will not allow the spindle to exceed the S address value specified in the most recent G50 command. This is most often used when the control is 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)

G51 Cancel Offset (YASNAC)

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

N1 G51 T0000 ; (CANCEL WEAR AND WORK SHIFT)
N2 G00 X0 Z0 ; (RAPID TO MACHINE ZERO)

WORK COORDINATE SYSTEM SELECTION (G52, G53, G54-59, G61, G64)

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 cell with great flexibility. Refer to "Programmable Offset Setting" and "Work Coordinates and Offsets" section for details on selecting tool offsets.

G52 Set Local Coordinate System FANUC

Group 00

This code selects the user coordinate system. It is non-modal. FANUC compatible.

G53 Machine Coordinate Selection

Group 00

This code temporarily cancels work coordinates offset and uses the machine coordinate system. It is non-modal; so the next block will revert to whatever conditions were previously selected.

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 13**

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

G64 G61 Cancel (Select normal cutting mode)**Group 13**

The G64 code is used to cancel exact stop. It is modal and thus affects the following blocks. Rapid and interpolated moves will not decelerate to an exact stop before another block is processed. Rapid blocks will decelerate to within the distance specified in Parameters 101-104 before another block is processed and interpolated motion will not decelerate at all before the next block is processed.

CANNED CYCLES (G70, 71, 72, 73, 74, 75, 76, 81, 82, 83, 84, 85, 86, 87, 88, 89, 90, 92, 94)

A canned cycle is used to simplify the programming of a part. Canned cycles are defined for the most common machining operations. They can be divided into two types. There are canned cycles for turning and grooving and there are canned cycles for drilling and tapping. Canned cycles can be either single block canned cycles or modal canned cycles.

Modal canned cycles remain in effect after they are defined and are executed for each positioning of the axes. Once a canned cycle is defined, that operation is performed at every X-Z position subsequently listed in a block. Some of the canned cycle numerical values can also be changed after the canned cycle is defined. The most important of these are the R plane value and the X or Z plunge value. Modal canned cycles can be canceled with the G80, G01 or G00 command. Positioning between canned cycle execution for modal cycles is performed as a rapid motion (G00).

If a modal canned cycle is defined in a block without an X or Z motion, there are two common actions taken by other controls; some will execute the canned cycle at that time and some will not. With the HAAS control, these two options are selectable from Setting 28. Regardless of the value in Setting 28, the programmer can force the control to not execute a canned cycle on the defining block by placing a loop count of zero (L0) within the defining block.

The operation of a canned cycle will vary according to whether incremental (U,W) or absolute (X,Z) is specified. Incremental motion is often useful in a canned cycle. If a loop count is defined within the block (L code), the canned cycle will repeat with an incremental U or W move between each cycle.

The positioning of the X-Z axis prior to a canned cycle is normally a rapid move (G00). The positioning move of a canned cycle does NOT exact stop prior to the plunging move of the X or Z axis to the R depth. This may cause a crash if a close tolerance fixture is being used. Setting 57 can be used to select exact stop after positioning motions for modal canned cycles.



Canned cycles G70, G71, G72, and G73 cannot be programmed while in DNC, since they require the control to look ahead. These codes do not force interpolated motion within the PQ block. To prevent rapid motion from occurring when feeds are intended, a G01 should be included near the beginning of the PQ block. Canned cycles G70, G71, G72 and G73 cannot be executed while tool nose compensation is in effect. If tool nose compensation is desired, it should be applied in the P block specified by the canned cycle.

Canned Cycles for Turning and Grooving

The following is a list of the canned cycles that can be used for turning and grooving on HAAS lathe controls.

G70 Finishing Cycle	Group 00
G71 O.D./I.D. Stock Removal Cycle	Group 00
G72 End Face Stock Removal Cycle	Group 00
G73 Irregular Path Stock Removal Cycle	Group 00
G74 End Face Grooving Cycle, Peck Drilling	Group 00
G75 O.D./I.D. Grooving Cycle	Group 00
G76 Thread Cutting Cycle, Multiple Pass	Group 00
G90 O.D./I.D. Turning Cycle	Group 01
G92 Thread Cutting Cycle	Group 01
G94 End Face Cutting Cycle	Group 01

G70 FINISHING CYCLE

GROUP 00

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

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

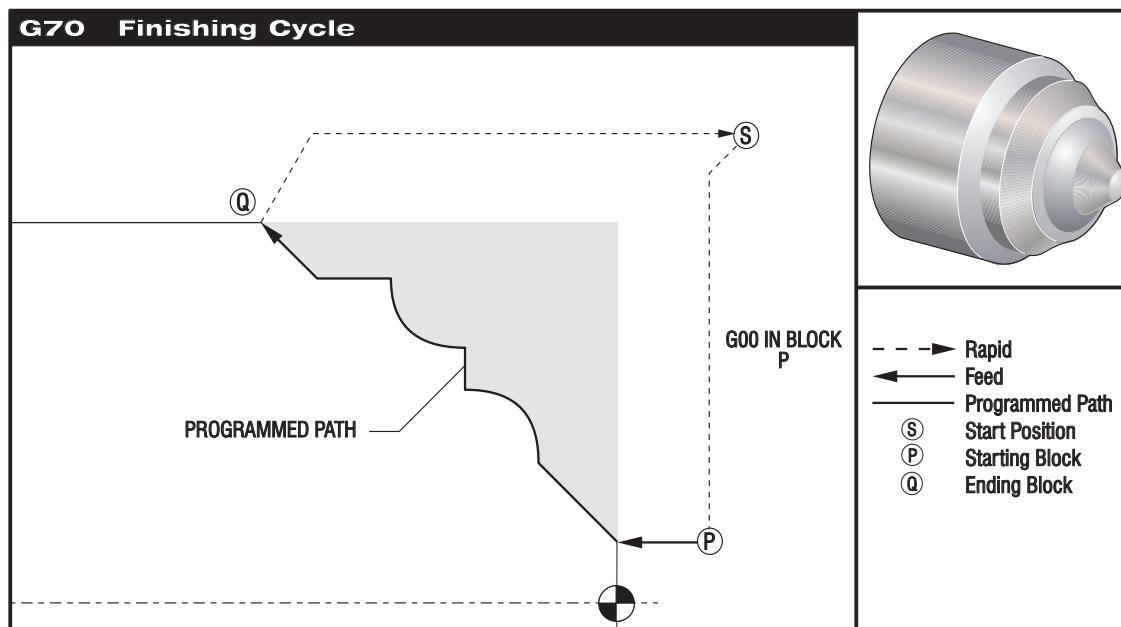


Fig. 5.0-7 G70.



A traditional calling sequence using G70 appears as below.

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

N10

F.014

...

N50

...

...

...

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

...

It is not necessary to use G71, G72 or G73 to use G70. G70 is a stand-alone G code.

The G70 cycle is similar to a local subprogram call. Instead of calling the routine simply by specifying the block number as with an M97 call, the G70 requires that a beginning block number (P code) and an ending block number (Q code) be specified. In addition, the block sequence that is defined by P and Q does not have an M99 indicating its end.

The G70 cycle is usually used after a G71, G72 or G73 has been performed using the blocks specified by P and Q. All codes in the block defined by P and Q are executed. Any F, S or T codes with the PQ block are effective. The PQ sequence is searched for in the current program starting at the beginning of the program. If the sequence is not found an alarm is generated. If the sequence is found, the location of the block following the G70 call is placed on the subroutine return stack. The current machine position is saved and remembered as the start position. Then the block starting at P is executed. Processing continues in a normal fashion with blocks following P until a block is found and executed that contains an N code that matches the Q code in the G70 calling block. After execution of the Q block, a rapid (G00) is executed returning the machine to the start position that was saved earlier during G70 initialization. The program then returns to the block following the G70 call. You can have a subroutine in the PQ sequence 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. In the HAAS controls the PQ sequence for a G70 does not have to precede the G70 call.

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
- * S Spindle speed to use throughout G71 PQ block
- * T Tool and offset to use throughout G71 PQ block
- Q Ending Block number of path to rough
- 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

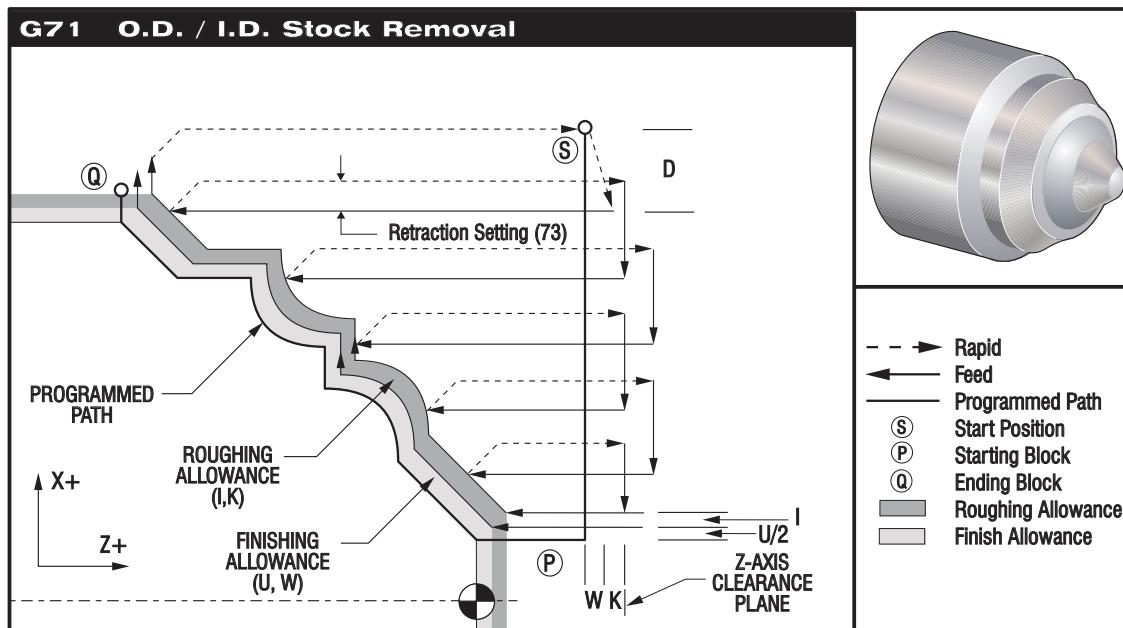


Fig. 5.0-8 G71

This canned cycle will rough out material on a part given the finished part shape. All a programmer needs to do is to define the shape of a part by programming the finished tool path and then submitting the path definition to the G71 call by means of a PQ block designation. Any feeds, spindle speeds or tools within the block defining the path are ignored by the G71 call. 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 using the programmed feeds, speeds, tools and offsets listed in those blocks.

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. This is depicted in Figure 5.0-9. This type of path is called a monotonic path. The second type of path (TYPE II) allows the X-axis to change direction, and is shown in Figure 5.0-14. For both the first type and the second type of programmed path the Z-axis must be monotonous, that is it 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. Figure 5.0-8 indicates the proper signs for these address codes to obtain the desired path in the associated quadrants.

G71 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 Z-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 types is a return to the starting position S.



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

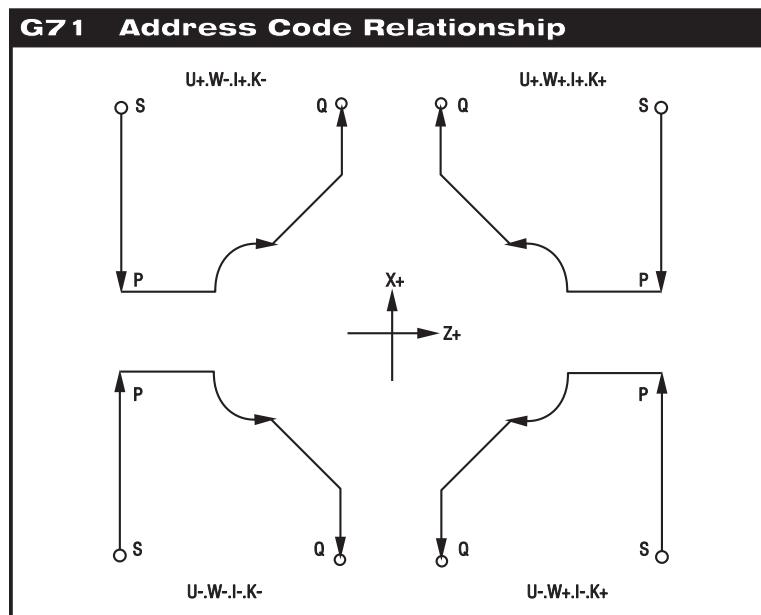


Fig. 5.0-9 G71 Address relationships

TYPE I DETAILS

When Type I is specified by the programmer it is assumed that the tool path is monotonic in the X-axis. Prior to any roughing motion the tool path designated by PQ is checked for monotonicity and G code compliance. An alarm is generated if a problem is found.

Roughing begins by advancing from the start position S and moving to the first roughing pass. All roughing passes start and end at the Z clearance plane. Each roughing pass X-axis location is determined by applying the value specified in D to the current X location. The direction that D is applied is determined by the signs of U and W. 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. Roughing continues until the X-axis position in block P is exceeded.

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 non-monotonically. In other words, the X axis can change direction throughout the PQ path. Z must still continue along in the same direction as the initial Z direction. The PQ path is checked prior to the start of any cutting and an alarm is generated if a problem exists.

The X axis PQ path must not exceed the original starting location. If it does, alarm 619 STROKE EXCEEDS START POSITION will be generated. The only exception is on the Q block.

Specify Type II roughing when Setting 33 is set to YASNAC by including R1 on the G71 command block. R1 must be specified with no decimal.

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

Roughing is similar to Type I except while roughing, 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.

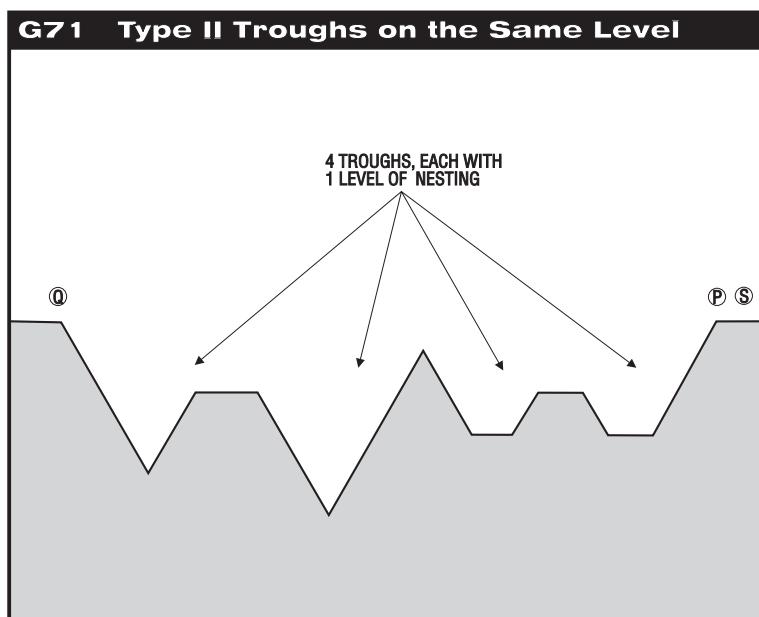


Fig. 5.0-10 Troughs on the same level.

There is virtually no limit to the number of blocks used to define a Type II PQ path. There is a limit to the number of troughs that can be included in a PQ path definition. 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. An alarm is generated when this limit is exceeded. Figures 5.0-10 and 5.0-11 illustrate the trouching concept.

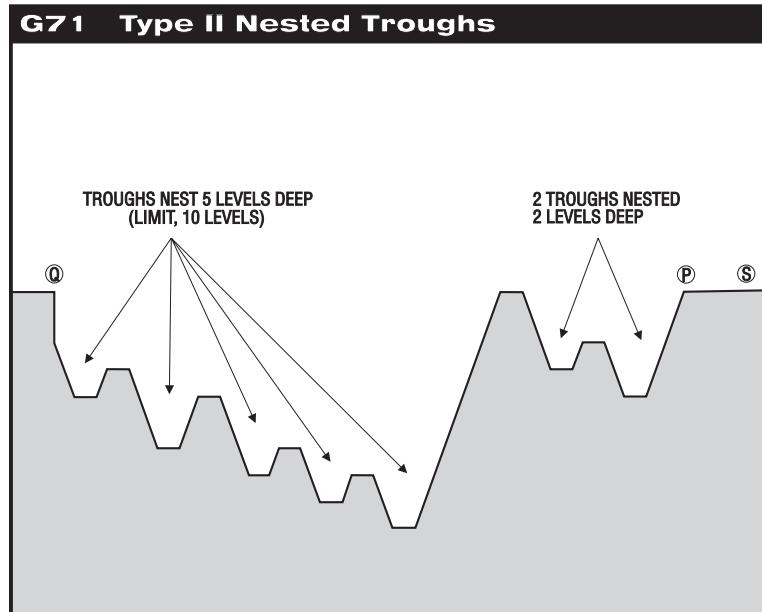


Fig. 5.0-11 Nested Troughs

Figure 5.0-12 illustrates the sequences of roughing cuts for PQ paths with multiple troughs. All material above troughs is roughed first, followed by the troughs themselves in the direction of Z. Tool retraction for Types I and II is illustrated below:

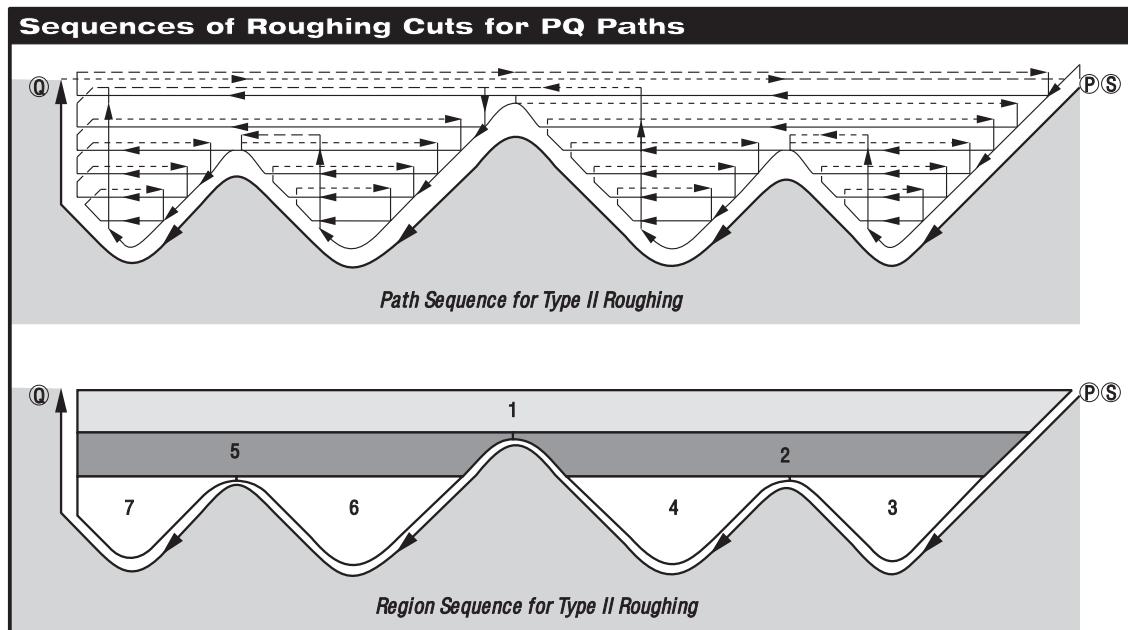


Fig. 5.0-12 Path sequence for Type II roughing

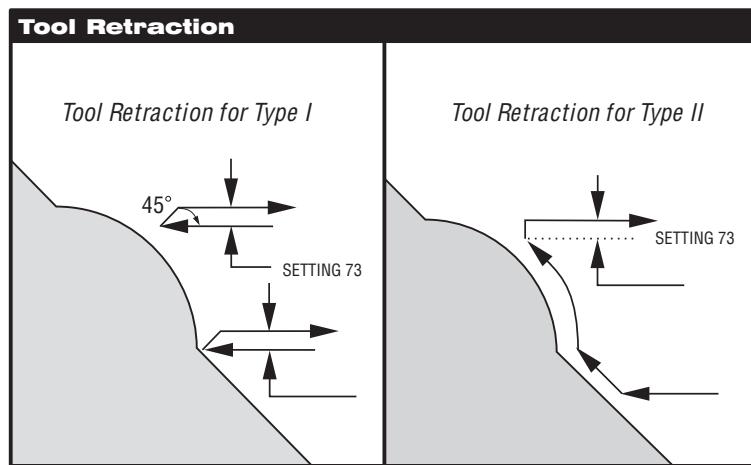


Fig. 5.0-13 Type I and II tool retraction

The sum of the roughing and finish allowance will be referred to simply as the 'allowance.' A side effect of using a Z finish or roughing allowance is a limit on the minimum horizontal distance between the intersection of 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
...

then the greatest allowance that can be specified is 0.999, since the horizontal distance from the start of cut 2 to the corresponding point on cut 3 is 0.2. If a larger allowance is specified, overcutting will occur. Since allowance is specified, overcutting will occur. Since allowances are typically small, this should only be a problem with complex curves made up of small segments.

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.

Known Problems: If the last cut in the P-Q path is a non-monotonic curve, using a finish allowance (parameter W) can cause overcutting. A workaround is to add a short retraction cut.

G72 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
- * S Spindle speed to use throughout G72 PQ block
- * T Tool and offset to use throughout G72 PQ block
- Q Ending Block number of path to rough
- U X-axis size and direction of G72 finish allowance, diameter
- W Z-axis size and direction of G72 finish allowance
- * indicates optional

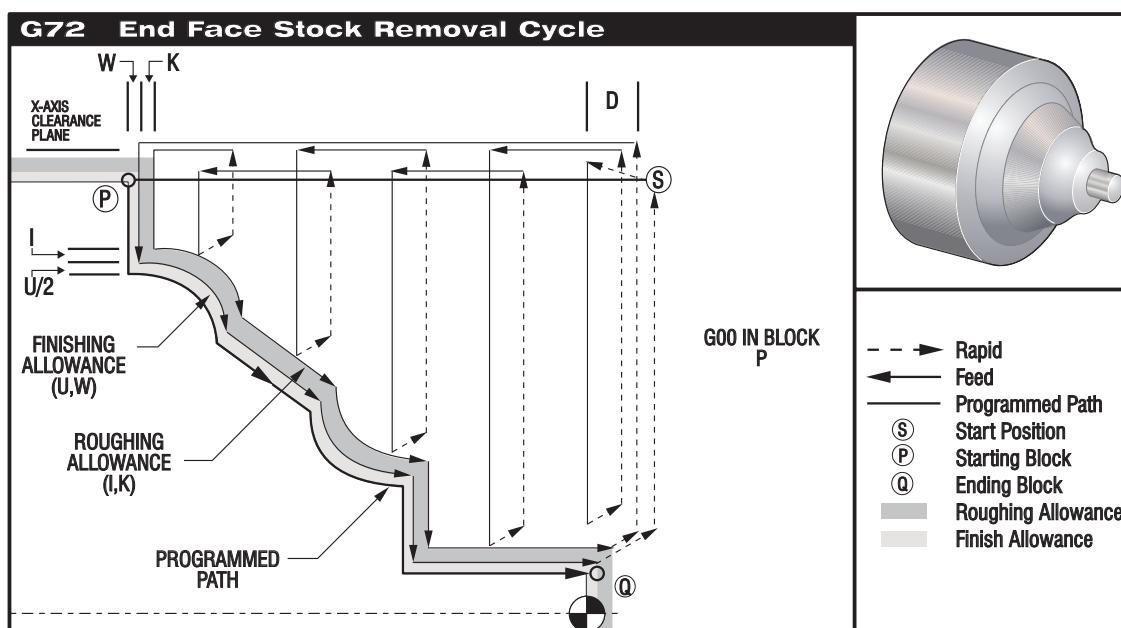


Fig. 5.0-14 G72

This canned cycle will rough out material on a part given the finished part shape. It is similar to G71 but roughs out material along the face of a part. All a programmer needs to do is to define the shape of a part by programming the finished tool path and then submitting the path definition to the G72 call by means of a PQ block designation. Any feeds, spindle speeds or tools within the block defining the path are ignored by the G72 call. 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 using the programmed feeds, speeds, tools and offsets.



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. This is depicted in Figure 5.0-15. This type of path is called a monotonic path. 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 must be monotonic, that is it 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)

Any one of the four quadrants of the X-Z plane can be cut by specifying address codes I, K, U, and W properly. Figure 5.0-16 indicates the proper signs for these address codes to obtain the desired performance in the associated quadrants.

The G72 consists of a roughing phase and a finishing phase. The roughing and finishing phase are handled slightly differently for types I and types 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 types is a return to the starting position S.

In the following 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.

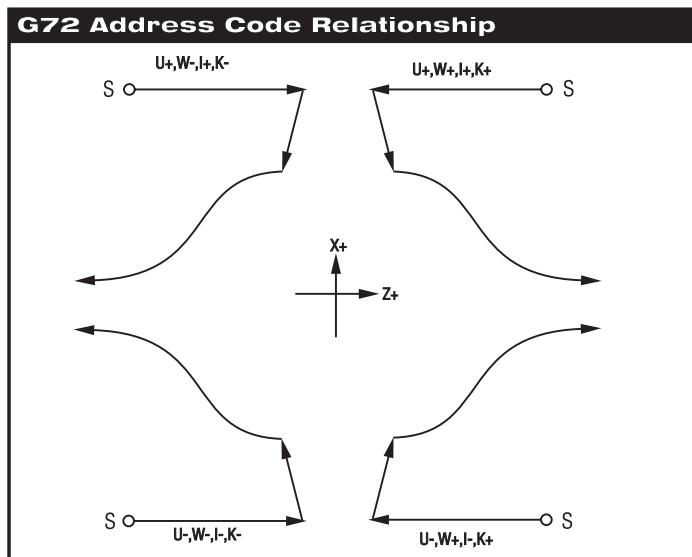


Fig. 5.0-15 G72 Address relationships



TYPE I DETAILS

When Type I is specified by the programmer it is assumed that the tool path is monotonic in the Z-axis. Prior to any roughing motion the tool path is checked for monotonicity and G code compliance. An alarm is generated if a problem is found.

Roughing begins by advancing from the start position S and moving to the first roughing pass. All roughing passes start and end at the X clearance plane. Each roughing pass Z-axis location is determined by applying the value specified in D to the current Z location. The direction that D is applied is determined by the signs of U and W. 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. Roughing continues until the Z-axis position in block P is exceeded.

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 non-monotonically. In other words, the Z axis can change direction throughout the PQ path. X must still continue along in the same direction as the initial X direction. The PQ path is checked prior to the start of any cutting and an alarm is generated if a problem exists.

The Z axis PQ path must not exceed the original starting location. If it does, alarm 619 STROKE EXCEEDS START POSITION will be generated. The only exception is on the Q block.

Specify Type II roughing when Setting 33 is set to YASNAC by including R1 on the G71 command block. R1 must be specified with no decimal.

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

Roughing is similar to Type I except while roughing, 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 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.

A side effect of using a X finish or roughing allowance is a limit on the minimum horizontal distance between the intersection of 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
...

then the greatest allowance that can be specified is 0.999, since the horizontal distance from the start of cut 2 to the corresponding point on cut 3 is 0.2. If a larger allowance is specified, overcutting will occur. Since allowances are typically small, this should only be a problem with complex curves made up of small segments.

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 (parameter U) can cause overcutting. To resolve this problem add a short retraction cut.

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
* S Spindle speed to use throughout G73 PQ block
* T Tool and offset to use throughout G73 PQ block
Q Ending Block number of path to rough
U X-axis size and direction of G73 finish allowance, diameter
W Z-axis size and direction of G73 finish allowance

* indicates optional

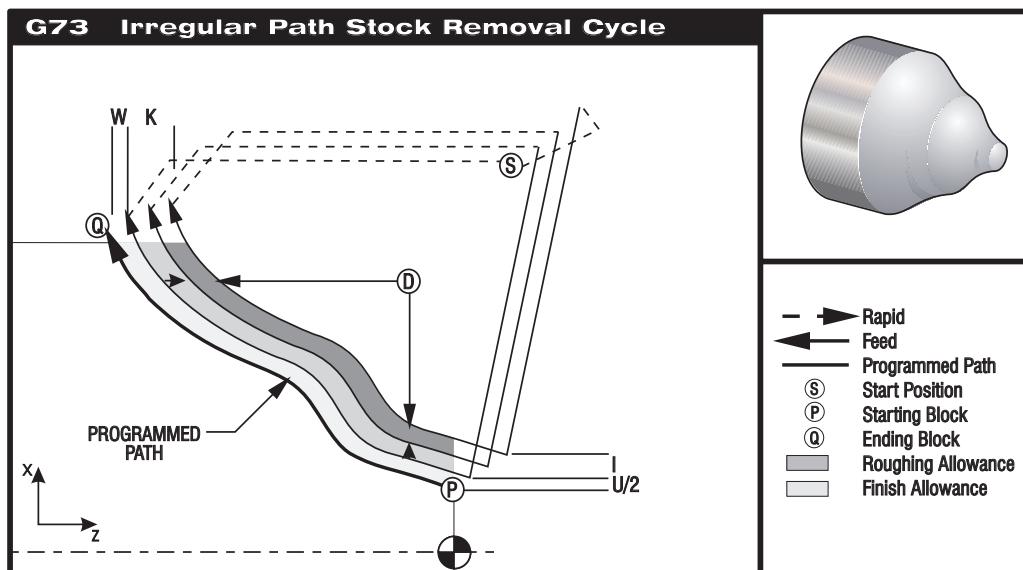


Fig. 5.0-16 G73



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, the start 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.

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

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. See Figure 5.0-16.

XAXIS ZAXIS TOOL APPROACHES FROM THIS QUADRANT.

U I	W K	
++	++	I
++	--	II
--	--	III
--	++	IV

**G74 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, signed diameter
- W Z-axis incremental distance to total pecking depth, signed
- * X X-axis absolute location of furthest peck cycle, signed diameter
- Z Z-axis absolute location total pecking depth, signed
- * indicates optional

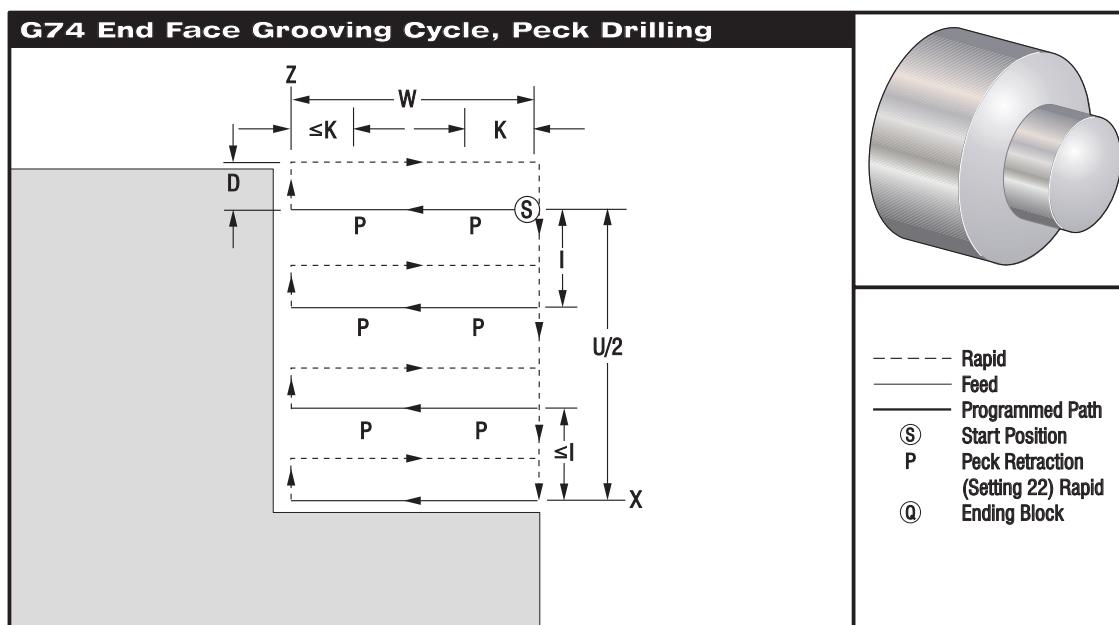


Fig. 5.0-17 G74

The G74 canned cycle can be used for grooving on the face of a part for peck drilling, or for turning with a chip break. With this canned cycle either a single pecking cycle can be executed, as for drilling on the spindle centerline, or a series of pecking cycles can be performed.

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 along X 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 and the peck distance is obtained from Setting 22. The D code can be used for grooving and turning to provide material clearance when returning to starting plane S.

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
- * 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

Settings 95 / 96 determine chamfer size / angle. M22 / 23 turns chamfering on / off.

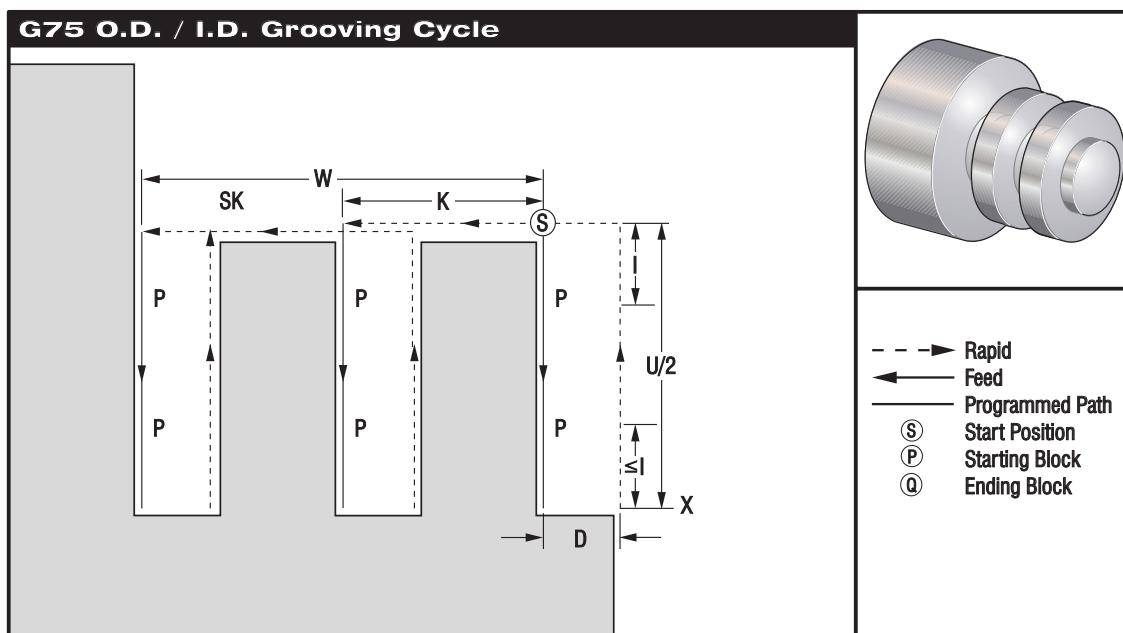


Fig. 5.0-18 G75

The G75 canned cycle can be used for grooving an outside diameter with a chip break. With this canned cycle either a single pecking cycle can be executed, as for a single groove, or a series of pecking cycles can be performed, as for multiple grooves.

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, pecking cycles between the starting position S and Z. If the distance between S and Z is not evenly divisible by K then the last interval along Z will be less than K.

When I is added to a G75 block, then pecking will be performed at each interval specified by I, the peck is a rapid move opposite the direction of feed and the peck distance is obtained from Setting 22.

The D code can be used for grooving to provide material clearance when returning to starting plane S.

**G76 Threading Cycle, Multiple Pass****Group 00**

- * A Tool nose angle
 - D First pass cutting depth
 - F(E) Feed rate
 - * I Thread taper amount, radius measure
 - K Thread height, defines limit of multiple passes, radius measure
 - * U X-axis incremental distance, start to maximum thread I.D.
 - * W Z-axis incremental distance, start to maximum thread length
 - X X-axis absolute location, maximum thread I.D.
 - Z Z-axis absolute location, maximum thread length
- * indicates optional

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

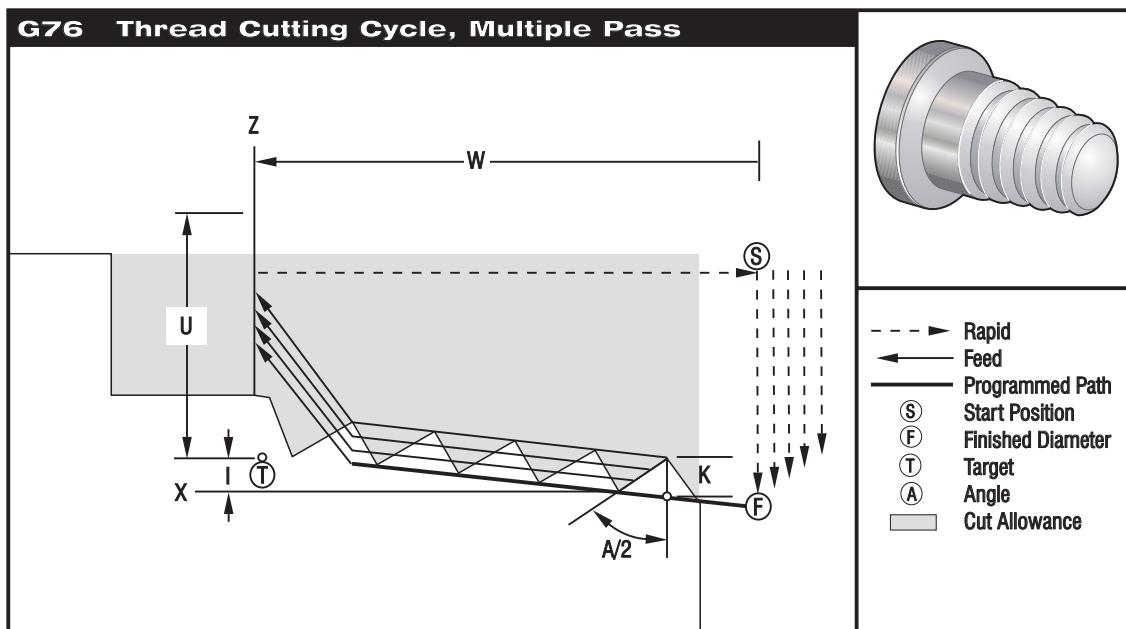


Fig. 5.0-19 G76

The G76 canned cycle can be used for threading both straight or tapered (pipe) threads. With G76 a programmer can easily command multiple cutting passes along the length of the thread. The nature of tool load and wear can be controlled by using the P code. The P code can specify which side the tool cuts on and it can specify how much material will be cut.

The height of the thread is specified in K and must be positive. 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 will be K less the finish allowance. Setting 86 (THREAD FINISH ALLOWANCE) is this amount and is defaulted to 0.

The thread taper amount is specified in I. It is measured from the target position X, Z at point T to position F. 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. This also determines the number of passes over the thread based on the value of K and the cutting method used.

The depth of the last cut through the thread can be controlled with Setting 86 (THREAD FINISH ALLOWANCE). For any of the methods specified in P, the last cut will never be less than this value. The default value is .001 inches/.01 mm.

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 then 0 degrees is assumed.

The F code specifies the feed rate for threading. It is always good programming practice to specify G99/G95 (feed per revolution) prior to a threading canned cycle. The F code also indicates the thread pitch or lead.

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

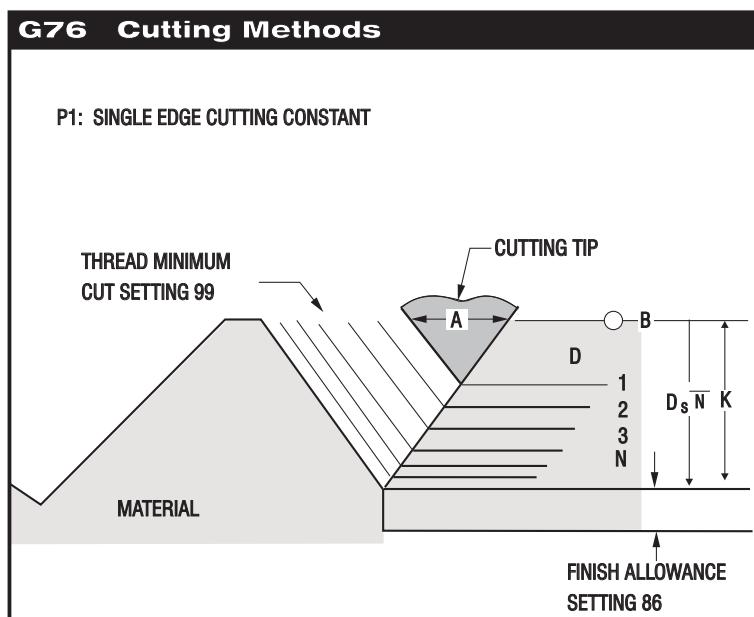


Fig. 5.0-20 G76 Cutting Methods



The following describes the four FANUC compatible cutting methods that are available with this control.

P1 SINGLE EDGE CUTTING, CUTTING LOAD CONSTANT

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.

NOTE: P2 TWO EDGES CUTTING, CUTTING LOAD CONSTANT
P3 SINGLE EDGE CUTTING, CUTTING DEPTH CONSTANT
P4 TWO EDGES CUTTING, CUTTING DEPTH CONSTANT
Future implementations. Defaults to P1.

G01 Chamfering and Corner Rounding

Group 01

AUTOMATIC CHAMFERING OR CORNER ROUNDING CAN NOT BE USED IN A THREADING CYCLE OR A CANNED CYCLE. REFER TO RULES AT THE END OF THIS SECTION FOR MORE DETAILS.

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

Chamfering Syntax

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

Corner Rounding Syntax

G01 X(U) x Rr
G01 Z(W) z Rr

Addresses

I = chamfering, Z to X
K = chamfering, X to Z
R = corner rounding

Values

k = chamfer dimension (Z axis direction, +/-)
i = chamfer dimension (X axis direction, +/-, "Radius" value)
r = corner rounding radius (X or Z axis direction, +/-, "Radius" value)

Canned cycles for Drilling and Tapping

The following is a summary of the canned cycles that can be used on HAAS lathe controls.

G Code	Spindle at Start	Z Drilling Operation	Operation at Bottom of Hole	Retraction Z Direction	Application
G81	—	feed	none	rapid	spot drilling
G82	—	feed	dwell	rapid	counter boring
G83	—	intermittent	none	rapid	peck drilling feed
G84	CW	feed	spindle CCW	feed	tapping cycle
G85	—	feed	none	feed	boring cycle
G86	CW	feed	spindle stop	rapid	boring cycle
G87	CW	feed	spindle stop	manual/rapid	back boring
G88	CW	feed	dwell, then	manual/rapid	boring cycle spindle stop
G89	—	feed	dwell	feed	boring cycle

A canned cycle is presently limited to operations in the Z-axis. That is, only the G18 plane is allowed. This means that the canned cycle will be executed in the Z-axis whenever a new position is selected in the X axis.

There are five operations involved in every canned cycle:

1. Positioning of X axis
2. Rapid traverse to R plane
3. Drilling operation at the bottom of hole
4. Retraction to R plane
5. Rapid traverse up to initial start point

NOTE: Whenever a repeated operation is needed, the number of repeats (L) must be specified. The number of repeats (L) is not retained for the next canned cycle.

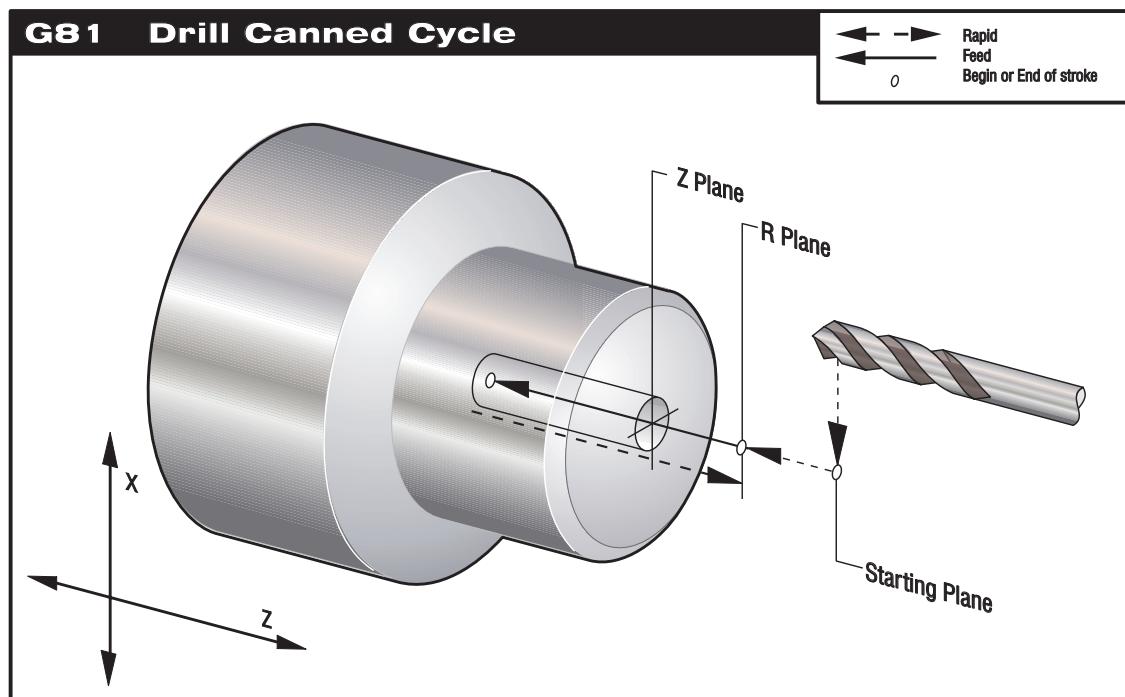
Spindle control M codes should not be used while a canned cycle is active.

**G80 Canned Cycle Cancel****Group 09***

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

G81 Drill Canned Cycle**Group 09**

- F Feed Rate
- 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



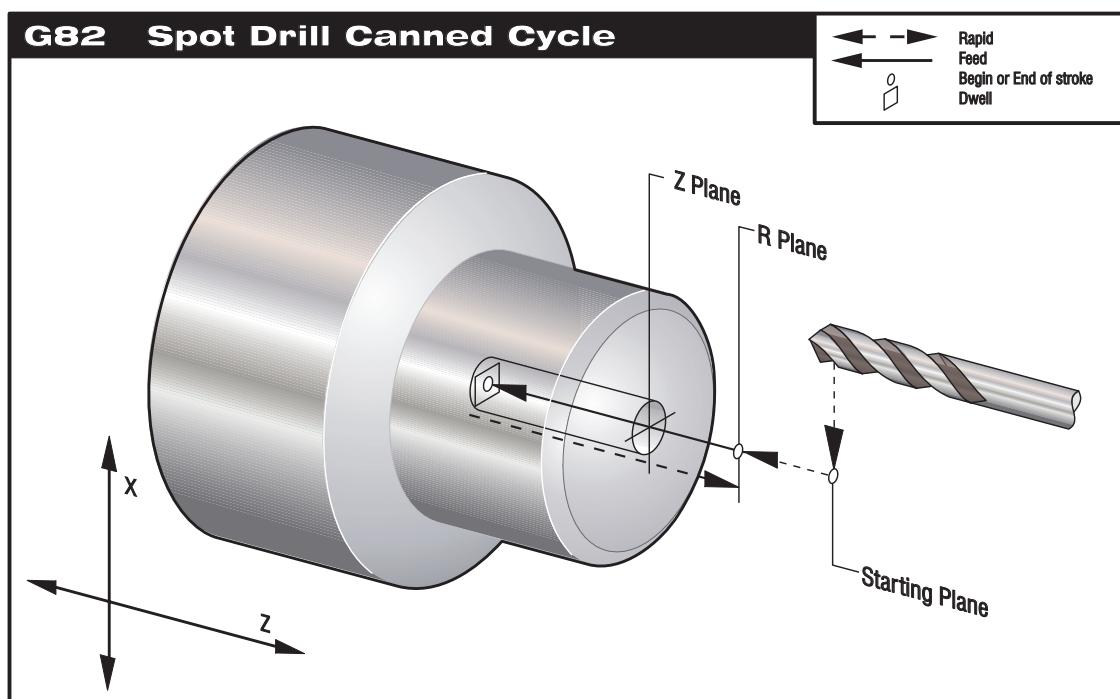
This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X will cause this canned cycle to be executed.

G82 Spot Drill Canned Cycle**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

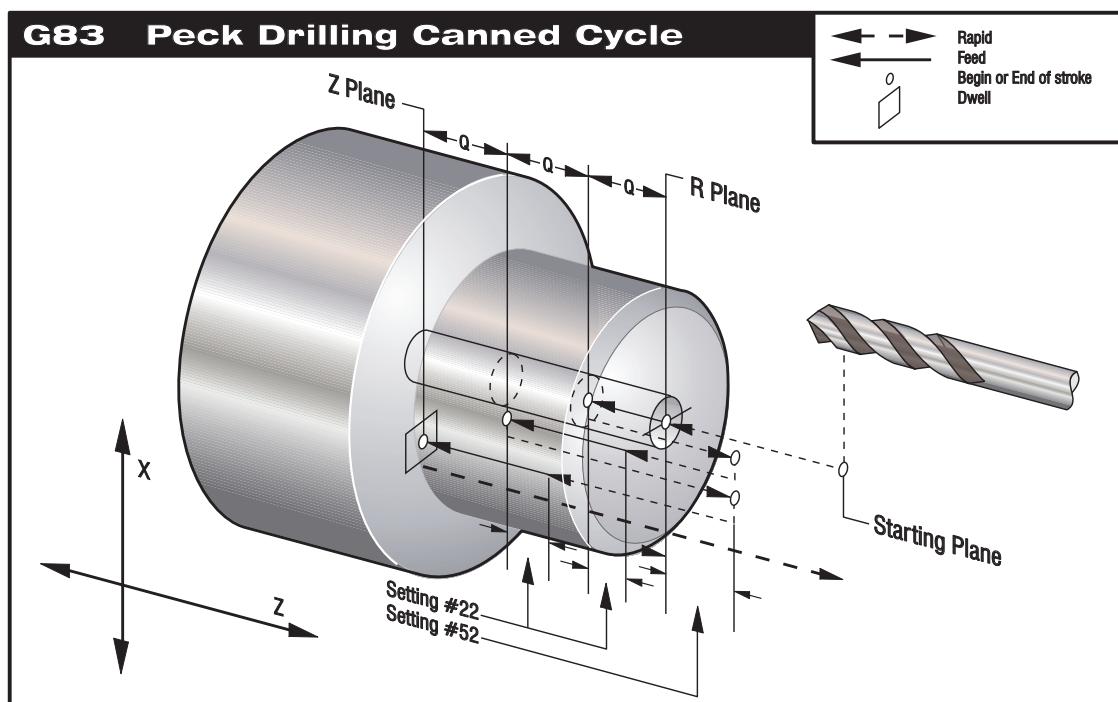
This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X will cause this canned cycle to be executed.



**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



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.

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

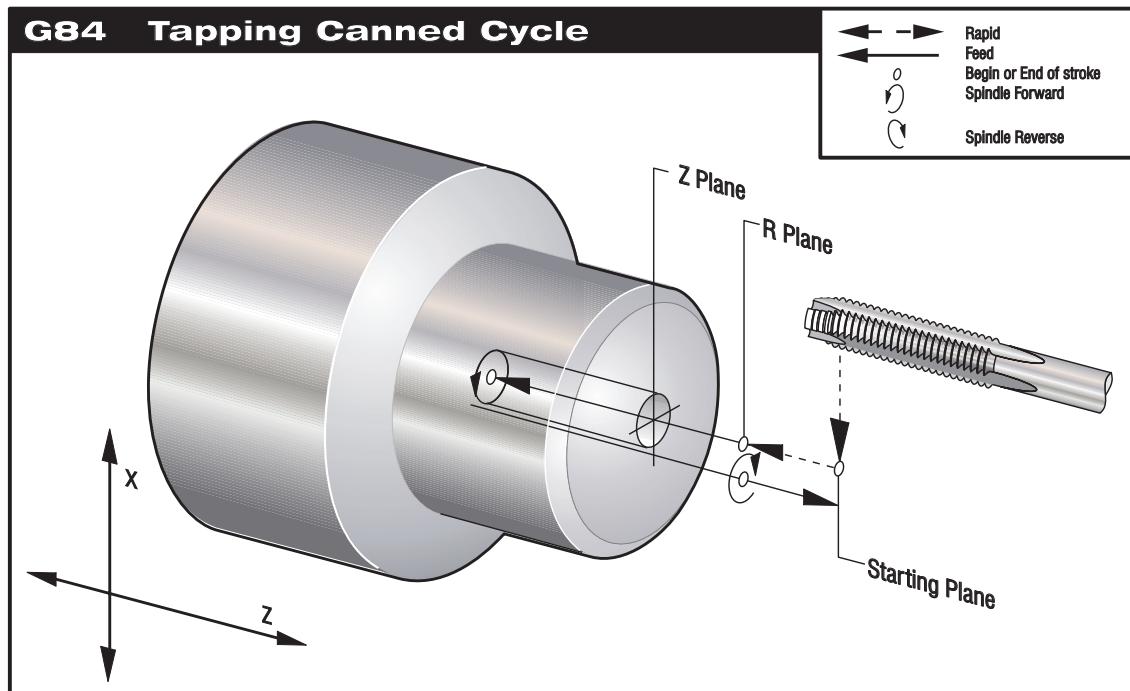
Setting 52 also changes the way G83 works when it returns to the R plane. Most programmers set the R plane well above the cut to insure that the chip clear motion actually allows the chips to get out of the hole but this causes a wasted motion when first drilling through this "empty" space. If Setting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

Setting 22 is the amount to feed in Z after a peck 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



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.

You do not need 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

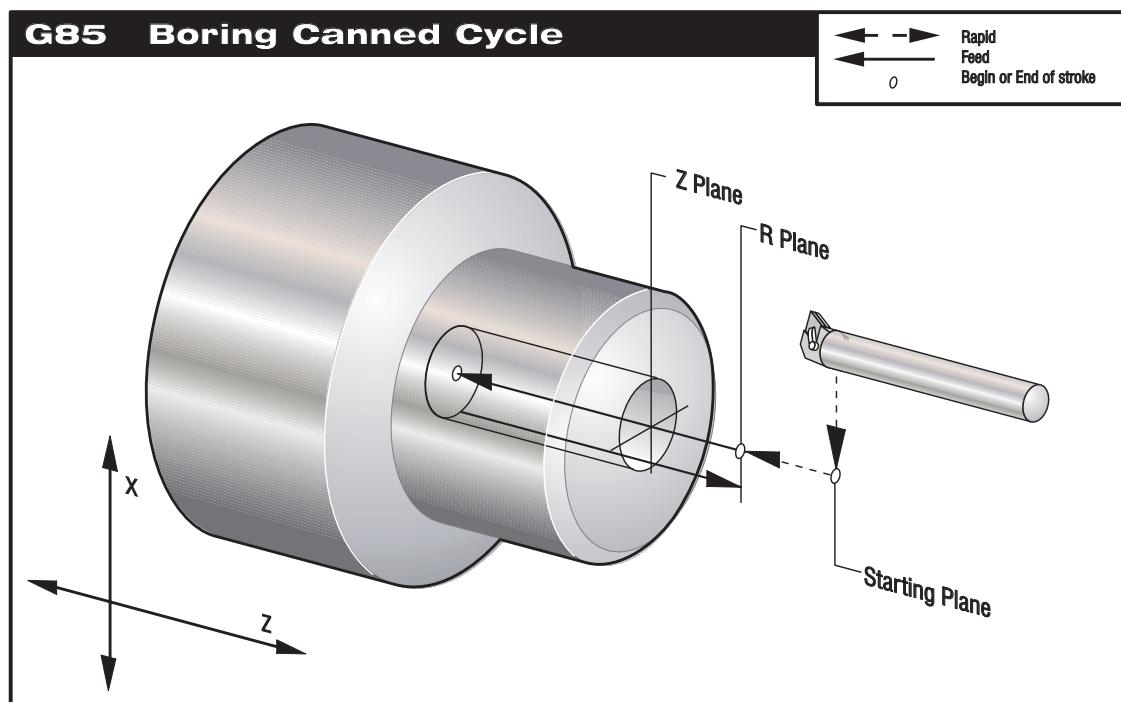
For Metric taps, divide the pitch by 25.4

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

**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

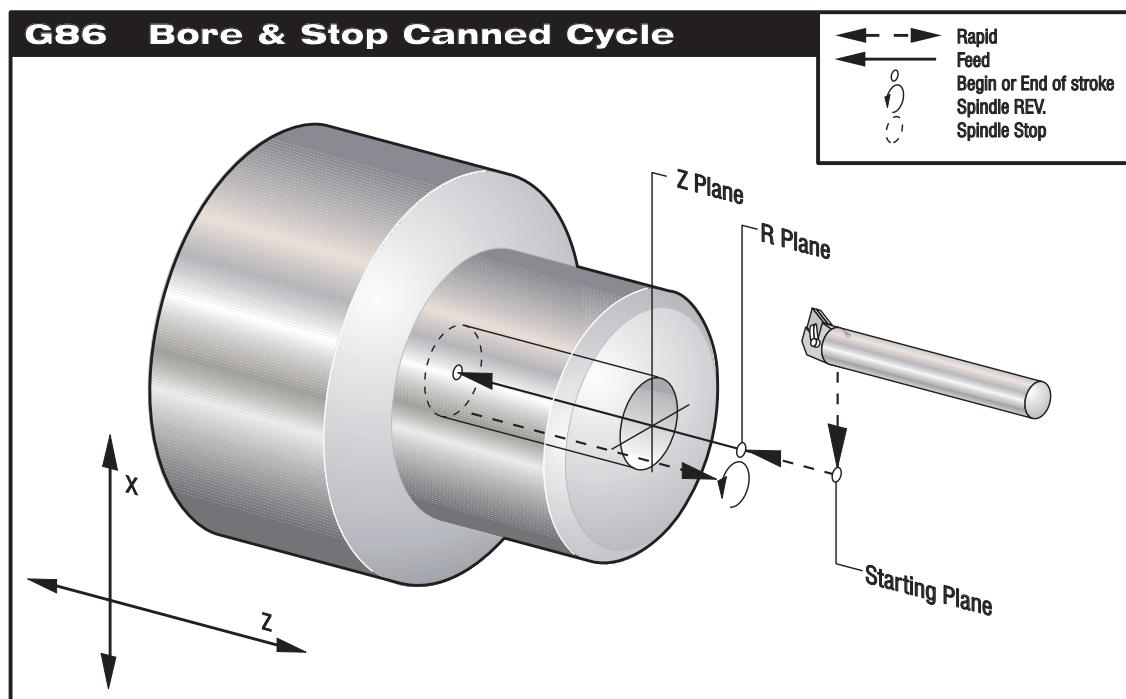


This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X will cause this canned cycle to be executed.

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

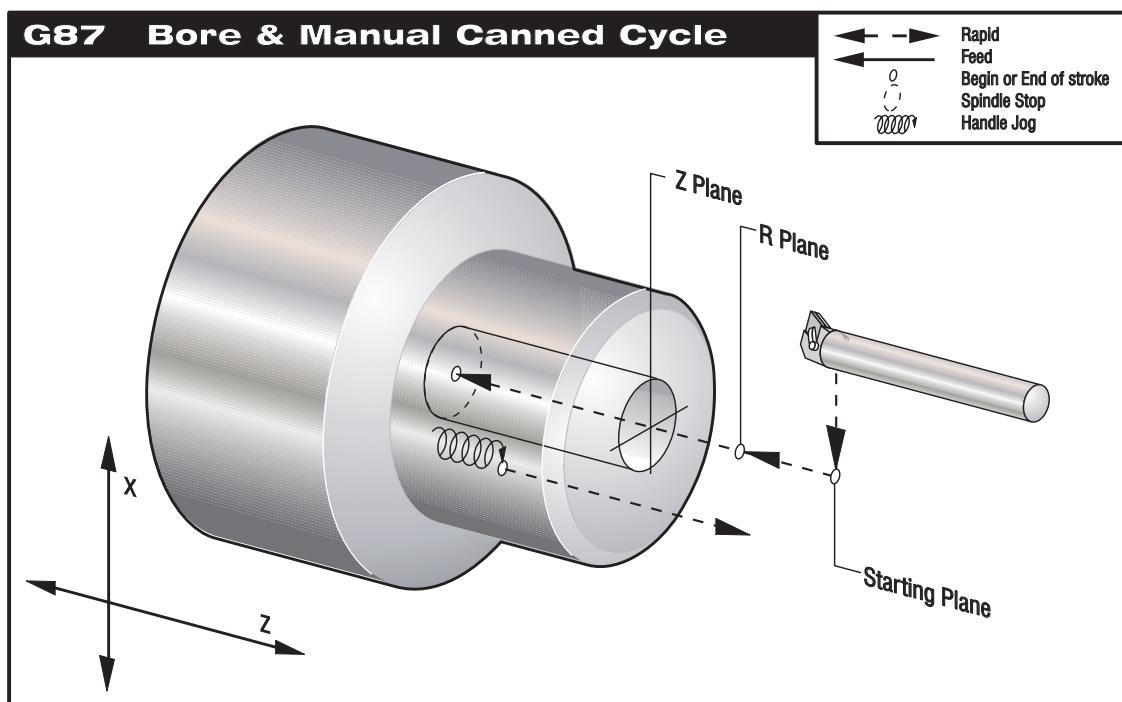


This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X will cause this canned cycle to be executed.

**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

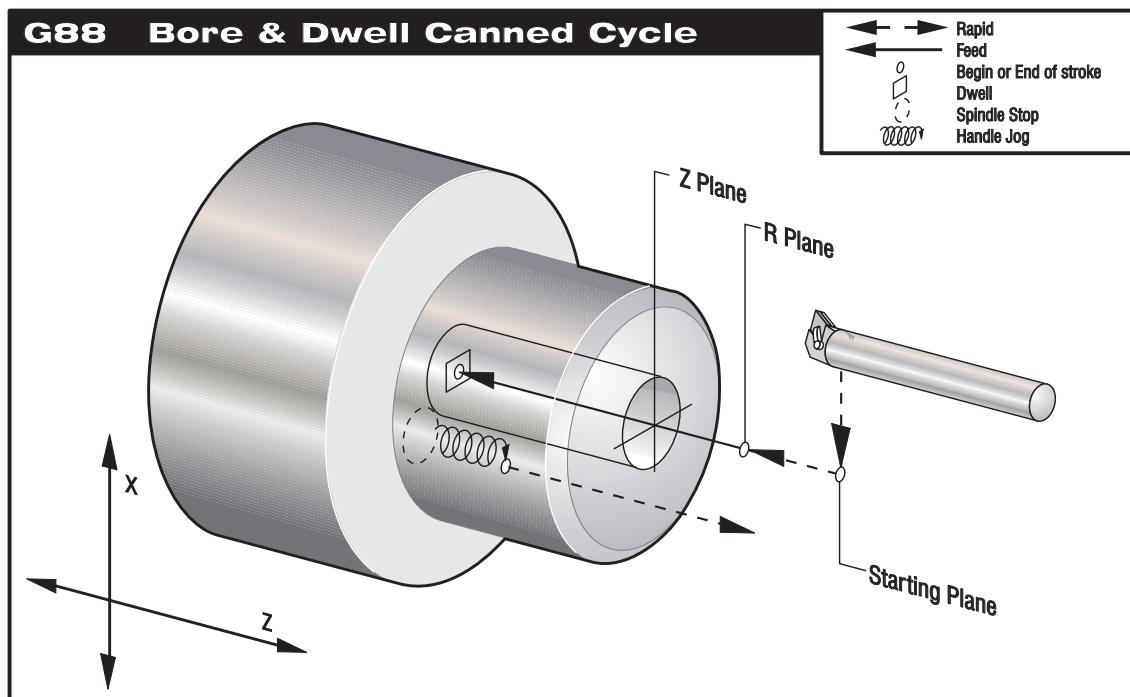


This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X will cause this canned cycle to be executed.

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

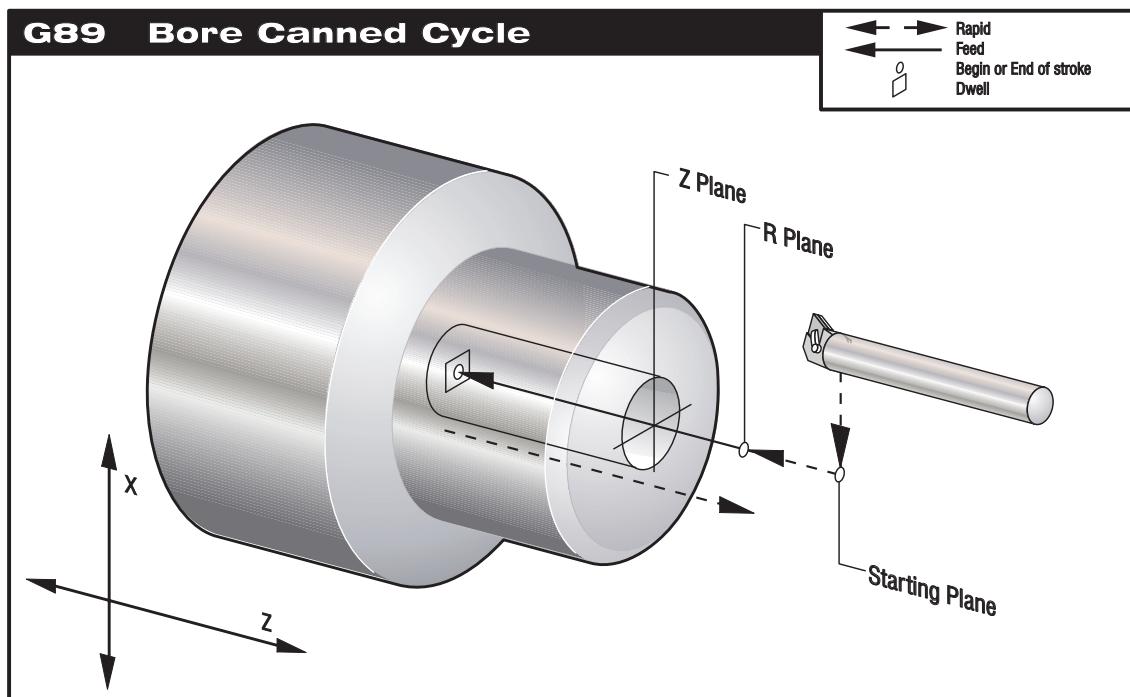


This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X will cause this canned cycle to be executed.

**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



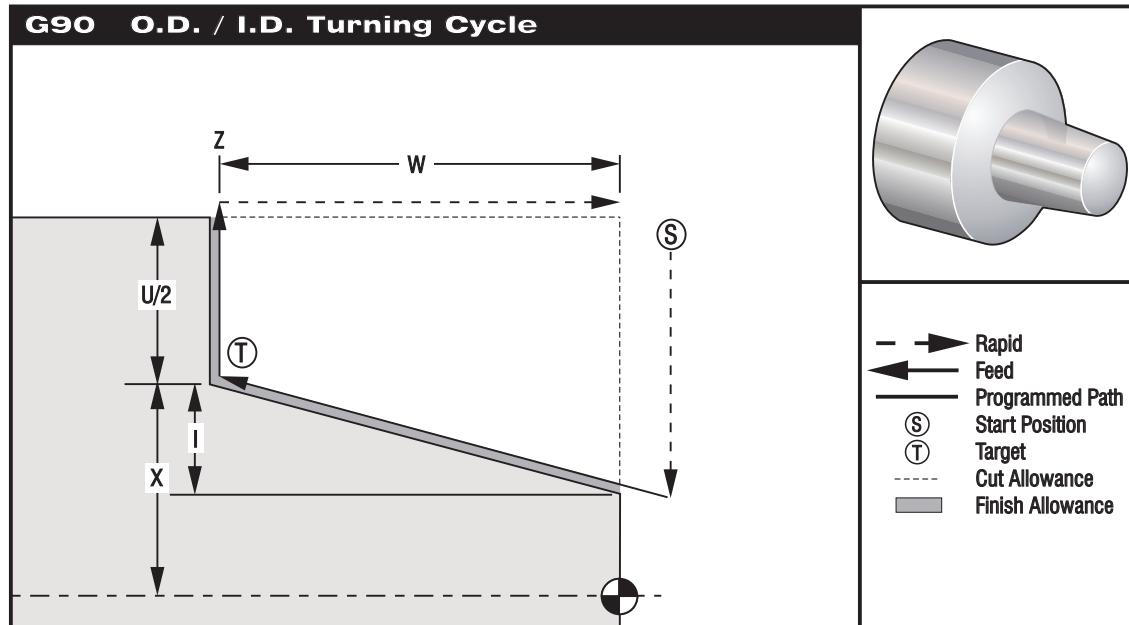
This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X will cause this canned cycle to be executed.


MODAL CANNED CYCLES (G90, G92, G94)

Modal canned cycles are much like G00 and G01 in that they only need to be specified once and then they remain in effect until they are changed by another G code from the same group. While a modal canned cycle is in effect it will be executed for each occurrence of one of the canned cycle parameters in a block, specifically I, K, U, W, X or Z. If L0 is included in the block the canned cycle will not be executed. Modal canned cycles can be used for simple turning or where other more complex canned cycles cannot be used.

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


Fig. 5.0-21 G90



G90 is a modal canned cycle. It can be used for simple turning. Since it is modal, you can do multiple passes for turning by just specifying the X locations of successive passes.

Straight turning cuts can be made by just specifying X, Z and F. By adding I 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, see Figure 5.0-20.

Any of the four ZX quadrants can be programmed by varying U, W, X, and Z. The taper can be positive or negative. Selecting the sign direction is not intuitive. Figure 5.0-22 gives a few examples of the values required for machining in each of the four quadrants.

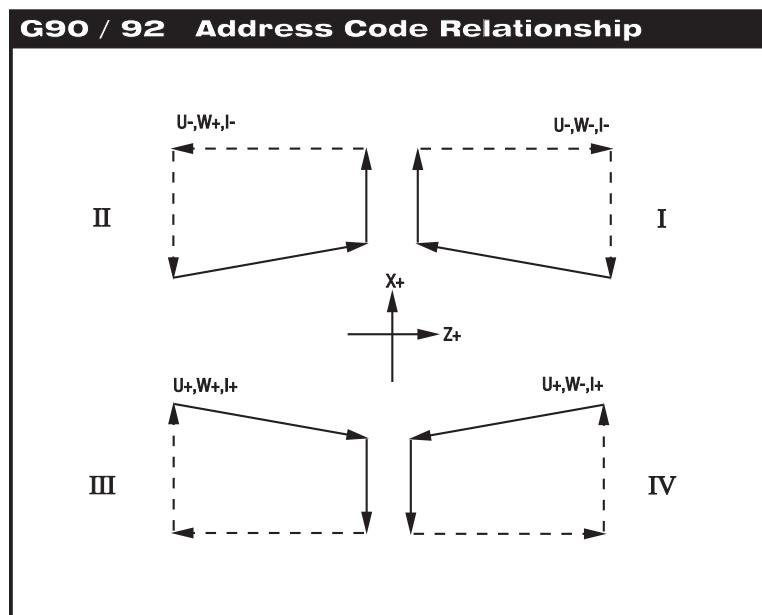


Fig. 5.0-22 G90-92 Address Relationships

G92 Threading Cycle

Group 01

G92 is a modal canned cycle. It can be used for simple threading. Since it is modal, you can do multiple passes for threading by just specifying the X locations of successive passes.

Straight threads can be made by just specifying X, Z and F. By adding I 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, see Figure 5.0-22.



At the end of the thread, an automatic chamfer is executed 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.

- F(E) Feed rate, the lead of the thread
- * 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

Setting 95 / 96 determine chamfer size / angle M22 / 23 turn chamfering on / off.

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

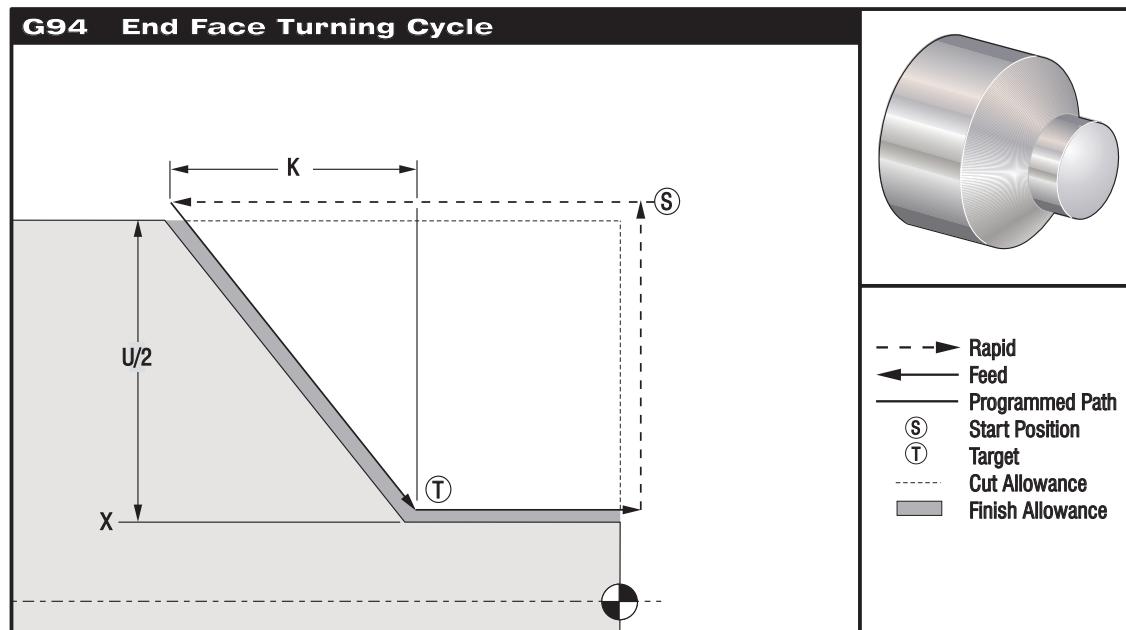


Fig. 5.0-23 G94

G94 is a modal canned cycle. It can be used for simple end facing. Since it is modal, you can do multiple passes for facing by just specifying the Z locations of successive passes.

Straight end facing cuts can be made by just specifying X, Z and F. By adding K a conical 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, see Figure 5.0-22.



Any of the four ZX quadrants can be programmed by varying U, W, X, and Z. The coning can be positive or negative. Selecting the sign direction is not intuitive. Figure 5.0-24 gives a few examples of the values required for machining in each of the four quadrants.

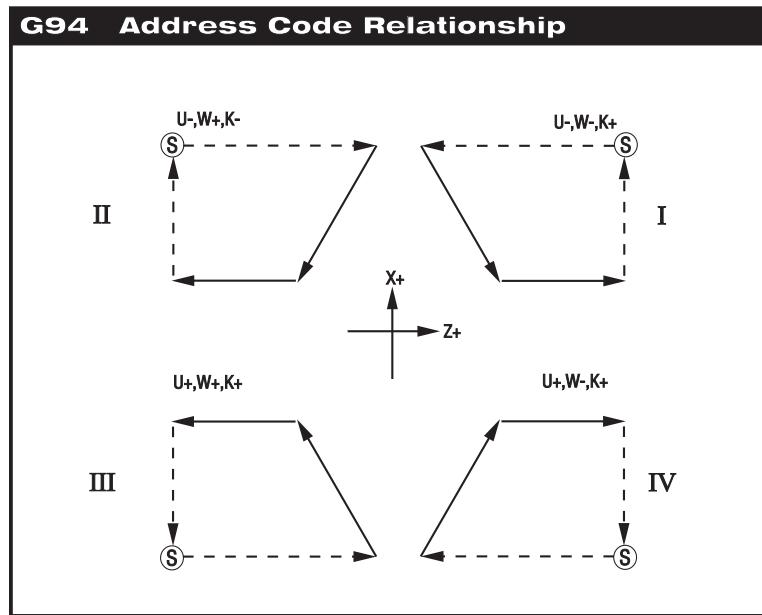


Fig. 5.0-24 G94 Address Relationships

SPINDLE SPEED COMMANDS (G96, G97)

G96 Constant Surface Speed ON

Group 12

This commands the control to maintain a constant surface speed of the part relative to the tool tip. Surface speed is based on the distance of the tool tip to the spindle center. This is the radius of cut. Surface speed is maintained by adjusting the spindle speed based on the radius of cut. G 96 is modal. The current S code is used to determine the surface speed.

G97 Constant Surface Speed OFF

Group 12

This commands the control to NOT adjust the spindle speed based on the radius of cut. It is used to cancel any current G96 command. G97 is the default Group 12 command at POWER UP and RESET. G97 is modal. When G97 is in effect, any S command is interpreted in units of revolution per minute (RPM).


CANNED CYCLE AUXILIARY FUNCTIONS (G98, G99)
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. This code is modal.

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. This code is modal, and is the default feed mode.

PROGRAMMABLE MIRROR IMAGE (G100, G101)
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 or Z axis. The two **G** codes (G100 and G101) are non-modal but the mirror image status of each axis is modal. 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. Note that G100 or G101 by itself will have no effect.

Mirror image may also be used to convert programs that were written for some older controls, in which the normally positive values will be negative.

Settings 45 through 48 may be used to manually select mirror image.

**PROGRAMMABLE OUTPUT TO RS-232 (G102)****G102 Programmable Output to RS-232****Group 00**

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

* indicates optional

Programmable output to the RS-232 port allows the current work coordinates of the two axes to be output. This G code (G102) is non-modal so only affects the block in which it is programmed. This G code should be used in a command block without any other G codes and will not cause any axis motion. The actual value given for the X or Z code has no effect. One complete line of text is sent to the first RS-232 port (same one used for upload, download, and DNC). Each axis listed in the G102 command block is output to the RS-232 port in the same format as values are displayed in a program.

Optional spaces (Setting 41) and EOB control (Setting 25) are applied. The values sent out are always the current axes positions referenced to the current work coordinate system.

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.

LIMIT BLOCK LOOKAHEAD FUNCTION (G103)**G103 Limit Block Lookahead****Group 00**

P = 0-15 Max. number of blocks the control will look ahead.
G103 [P..]

"Block Lookahead" is a term used to describe what the control is doing in the background during machine motion. A motion block may take several seconds to execute. The control can take advantage of this by preparing additional blocks of the program ahead of time. Time is saved while the current block is executing and the next block has already been interpreted and prepared by the continuous, uninterrupted motion between consecutive blocks. Block lookahead is also important for obtaining information necessary for predicting compensated positions for cutter compensation.

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, lookahead is limited to n blocks.

At this time G103 cannot be used if cutter compensation, G41 or G42, is in effect. Alarm 387 is generated if you attempt to do so.

G103 is also useful for debugging programs using macros. Macro expressions are executed at lookahead time. By inserting a G103 P1 into the program, macro expressions will be performed one block ahead of the current executing block.

G103 is not a FANUC compatible command.

**MORE WORK COORDINATE SELECTION****G110,G111 and G114-G129 Coordinate System****Group 12**

These codes select one of the additional 20 user coordinate systems stored within the offsets memory. 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.

IN POSITION ACCURACY (G187)**G187 Accuracy Control****Group 00**

Programming G187 is as follows:

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

This G code is only available on machines equipped with brushless servos. 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.

LIVE TOOLING G CODES**G05 Fine Spindle Control motion (This G-code is optional and is used for live tooling) Group 00**

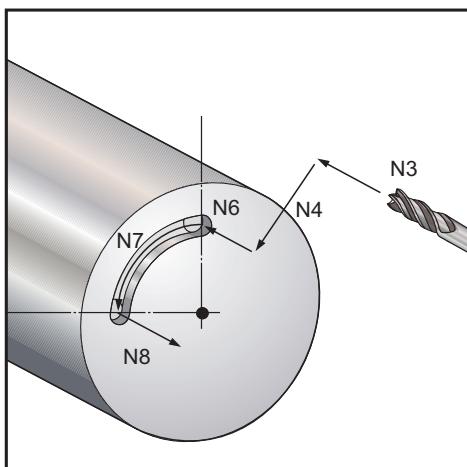
- R Angular motion of the spindle, in degrees.
- F Feed Rate of the center of the tool, in inches per minute.
- *U X-axis incremental motion command.
- *W Z-axis incremental motion command.
- *X X-axis absolute motion command.
- *Z Z-axis absolute motion command.

* indicates optional

This G code is used to specify a precise motion of the spindle, and is intended to be used for slotting. Any motion specified along the X and Z axes tracks the spindle motion. Currently, the resolution of the R code value is .045 degrees.

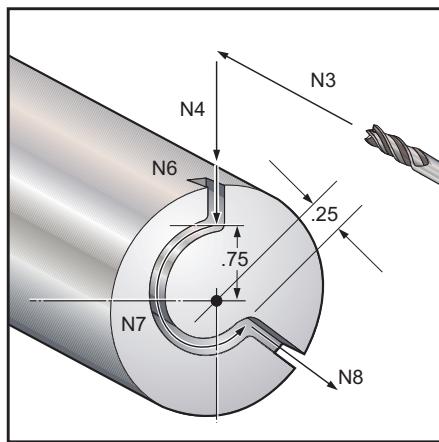
The rotational speed of the spindle will remain constant throughout each G5 cut. If there is motion along the X axis during the G05, the actual feed rate will vary. The spindle speed is determined by looking at the greatest X value encountered during the cut. Therefore the specified feed rate will not be exceeded at any point along the cut.

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

**Simple Face Slot Example with G05**

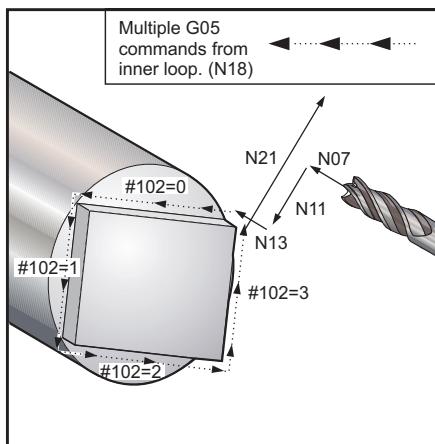
(Assume pilot hole is already drilled.)

- | | |
|----------------------|--------------------------------|
| N1 T303 | (Small End Mill) |
| N2 M19 | (Orient Spindle) |
| N3 G0 Z.5 | |
| N4 G0 X1. | |
| N5 M133 P1500 | |
| N6 G98 G1 F10. Z-.25 | (Plunge into pre-drilled hole) |
| N7 G5 R90. F40. | (Make slot) |
| N8 G1 F10. Z.5 | (Retract) |
| N9 M135 | |
| N10 G99 G28 U0 W0 | |

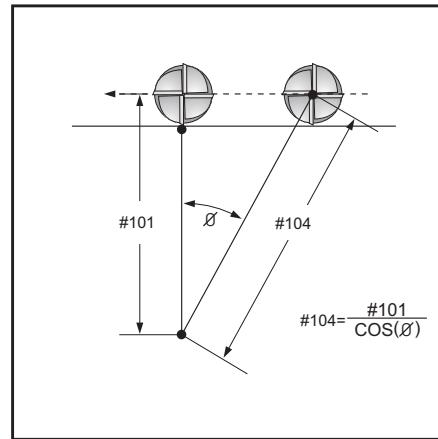
Simple Cam Example with G05

- | | |
|----------------------|--------------------------|
| N1 T303 | (Small End Mill) |
| N2 M19 | |
| N3 G0 Z-.25 | |
| N4 G0 X2.5 | (Approach 2" diam stock) |
| N5 M133 P1500 | |
| N6 G98 G1 X1.5 F40. | (Cut to top of cam) |
| N7 G5 R215. X.5 F40. | (Cut Cam) |
| N8 G1 X2.5 F40. | (Cut out of cam) |
| N9 M135 | |
| N10 G99 G28 U0 W0 | |

Flattening Example with G05



O1484



(Cut a square with G05)

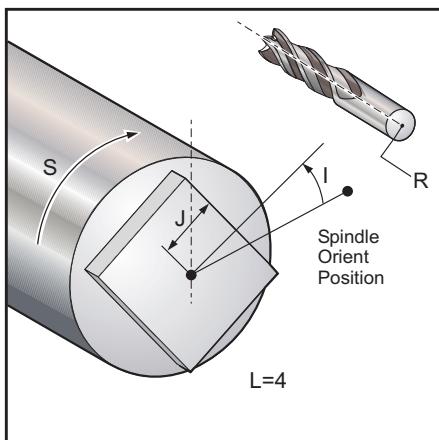
N1 G28 X0.
 N2 G28 Z0.
 N3 G54 G00 G40 G97
 N4 G103 P3
 N5 T707 (.75 dia high-speed end mill)
 N6 M19
 N7 G00 Z0.5
 ()
 (101 = Closest approach. Center to)
 (side plus half of tool diameter)
 N8 #101=[0.707 + 0.75 / 2.]
 (Multiply by 2 for diam.)
 N9 #101=#101 * 2
 (104 = Distance at corner.)
 N10#104=[#101 / COS[45.]]
 N11 G98 G01 X#104 F100.
 N12 M133 P1500
 N13 Z-0.1 (Feed into pre-drilled hole)
 N14#102=0
 WHILE [#102 LT 4] DO1 (Four sided shape)
 N15#103=-45. (Angle from center of flat)
 ()
 WHILE [#103 LT 45.] DO2
 N16#103=[#103 + 5.]
 N17#104=[#101 / COS[#103]]
 N18 G05 X#104 R5. F20.
 END2
 ()
 N19#102=[#102 + 1]
 END1
 ()
 N20 M135
 N21 G28 U0
 N22 G28 W0
 N23 M30

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

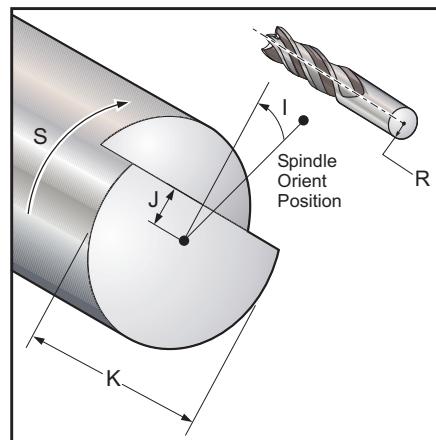
Note: This cycle is only available on lathes with the live tooling option.

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

G77 with L specified



G77 with K specified



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

G77 operates in one of two modes, depending on whether a K code or an L code is specified. If a K code is specified, one flat surface will be cut. If an L code is specified, L flat surfaces will be cut, equally spaced all the way around the part. L must be greater than or equal to 3. L2 is not supported, if two sides are desired, perform two K cuts at I angle spacing. If L and K are both specified, alarm 339 MULTIPLE CODES is generated.



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 K value specifies the diameter of the part. Specifying a smaller diameter than the diameter of the acutal part may cause the tool to crash into the part during its approach.

The R value specifies the radius of the live cutting tool. It is important that this value is correct, as it is used for automatic tool compensation and the entry and exit motions.

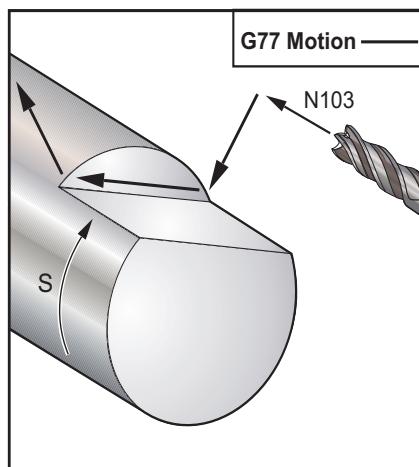
The S value specifies the rpm speed that the spindle will maintain during the flatting cycle. The default value is 6. This value can be increased for small parts. Higher values will not affect the flatness, but will affect the position of the flats. To calculate the maximum error in degrees, use $RPM * 360 / 60000$, or $RPM * .006$.

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:

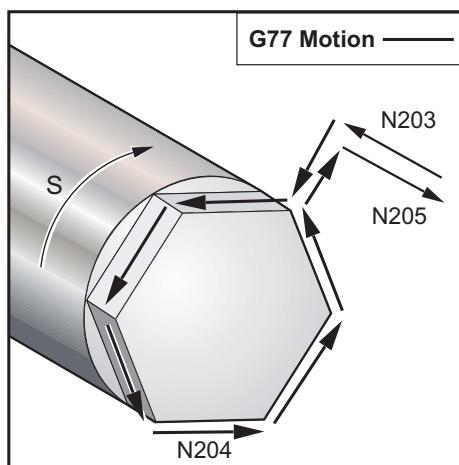
To cut a half-inch deep flat into the top inch of a part that is four inches in diameter, using a tool one inch in diameter:





...
N100 S10 M3
N101 M133 P1000
N102 G0 X6.1
N103 Z-1.
N104 G77 J1.5 K4. R.5
N105 Z1.
N106 M135
N107 M5
...
(Start spindle)
(Turn live tool)
(Stop live tool)
(Stop spindle)

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

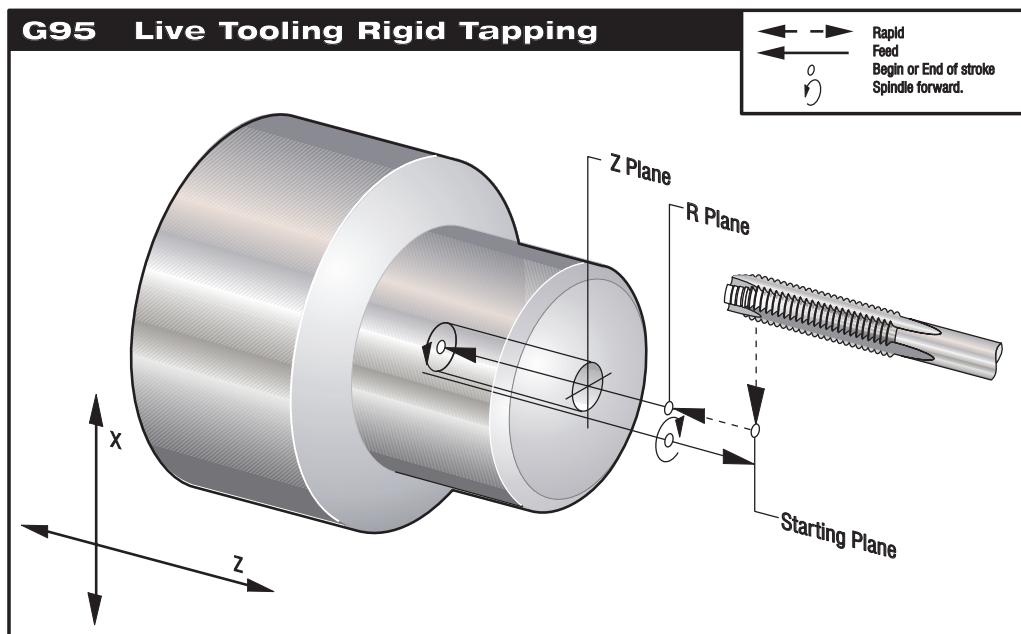


...
N200 S10 M3
N201 M133 P1000
N202 G0 X4.5
N203 Z-.5
N204 G77 J1.299 L6 R.25
N205 Z1.
N206 M135
N207 M5
...
(Start spindle)
(Turn live tool)
(Stop live tool)
(Stop spindle)

G95 Live Tooling Rigid Tap
G186 Reverse Live Tooling Rigid Tap

Group 09
Group 09

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



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

You do not need 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

Currently tapping is supported in the Z-axis. G95 Live Tooling Rigid Tapping is similar to G84 Rigid Tapping in that it uses the F, R, X and Z parameters, however, it has the following differences:

1. The main spindle must be clamped (use M14) before G95 is commanded or an alarm will be generated.
2. The control must be in G99 FEED PER REVOLUTION mode in order for tapping to work properly.
3. An S (spindle speed) command must have been issued prior to the G95 because the specified spindle speed will be used to control the Live Tool speed.
4. The X axis can be positioned between zero and the center of the main spindle. If it is positioned beyond the center of the main spindle, an alarm will be generated.

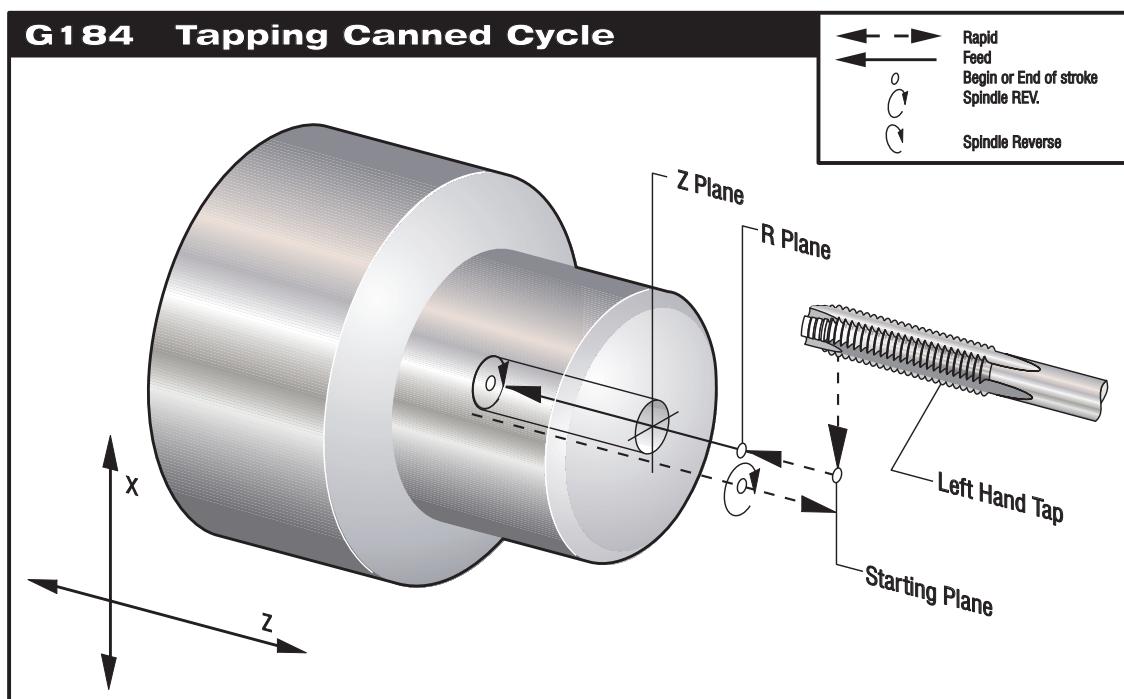
G184 Reverse Tapping Canned Cycle For Left Hand Threads
Group 09

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

* indicates optional

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

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





G CODES

SL Series OPERATOR'S MANUAL

June 2001

G195 Live Tool Vector Tapping

Group 00

G196 Reverse Live Tool Vector Tapping

Group 00

NOTE: G95 is for axial tapping only and allows the user to specify an R plane.

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

X (Optional) End Point For Tapping Cycle
F Inch Per Rev (G99)
Z Optional Z end point

Below is a brief program example of G195

```
%  
O00800  
N1 T101 (RADIAL 1/4-20 TAP)  
G99 (Necessary 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

This G code will cause the lathe to change tools while performing a rapid move away from and back to the part, thus saving 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 is moving into the "rotate" position (popping out). X and Z specify the position to move to as the tool turret moves back into the normal position (reseats). Both motions are rapid. More concisely:

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



6. M CODES (MISCELLANEOUS FUNCTIONS)

M Code Summary

Only one **M** code may be programmed per block of a program. All **M** codes are effective or cause an action to occur at the end of the block. However, when Parameter 278 bit "CNCR SPINDLE" is set to 1, an M03 (spindle start) will occur at the beginning of a block.

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
M13	Auto Air Jet Off
M14	Clamp Main Spindle
M15	Unclamp Main Spindle
M17	Turret Rotation Always Forward
M18	Turret Rotation Always Reverse
M19	Orient Spindle. P and R values optional.
M21	Tailstock Forward
M22	Tailstock Reverse
M23	Thread Chamfer ON
M24	Thread Chamfer OFF
M30	Prog End and Rewind
M31	Chip Conveyor Forward
M32	Chip Conveyor Reverse
M33	Chip Conveyor Stop
M36	Parts Catcher Up
M37	Parts Catcher Down
M41	Low Gear*
M42	High Gear*
M43	Turret Unlock (Service Use Only)
M44	Turret Lock (Service Use Only)
M51-M58	Optional User M turn ON
M61-M68	Optional User M turn OFF
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
M119	Subspindle Orient
M121-128	Optional User M
M133	Live Tooling Drive Forward
M134	Live Tool Drive Reverse
M135	Live Tool Drive Stop
M143	Subspindle Forward
M144	Subspindle Reverse
M145	Subspindle Stop
M154	C-axis engage
M155	C-axis disengage

* SL-30 and 40 only.

M CODE DETAILED DESCRIPTION

M00 Stop Program

The M00 code is used to stop a program. It also stops the spindle and turns off the coolant and stops interpretation look-ahead processing. The program pointer will advance to the next block and stop. A cycle start will continue program operation from the next block. Additionally M00 will turn off the live tooling motor if the option is present and active. If equipped with an Auto Door, M00 will open the door, provided parameter 57 Safety Circuit is set to 0, and parameters 235, 236, and 251 are set appropriately.

M01 Optional Program Stop

The M01 code is identical to M00 except that it only stops if OPTIONAL STOP is turned on from the front panel. A cycle start will continue program operation from the next block. Additionally M00 will turn off the live tooling motor if the option is present and active. If equipped with an Auto Door, M01 will open the door, provided parameter 57 Safety Circuit is set to 0, and parameters 235, 236, and 251 are set appropriately.

M02 Program End

The M02 code will stop program operation the same as M00 but does not advance the program pointer to the next block. It will not reset the program pointer to the beginning of the program as an M30 does. M03 Spindle Forward

The M03 code will start the spindle moving in a clockwise direction at whatever speed was previously set. The block will delay until the spindle reaches about 90% of commanded speed. If bit 10 of parameter 278 (CNCR SPINDLE) is set to 1, then this command is executed at the beginning of block execution rather than the end as most M codes are.



M04 Spindle Reverse

The M04 code will start the spindle moving in a counterclockwise direction at whatever speed was previously set. The block will delay until the spindle reaches about 90% of commanded speed. If bit 10 of parameter 278 (CNCR SPINDLE) is set to 1, then this command is executed at the beginning of block execution rather than the end as most M codes are.

M05 Spindle Stop

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

M08 Coolant On

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

M09 Coolant Off

The M09 code will turn off the coolant supply.

M10 Clamp Chuck

The M10 code is used to Clamp the chuck. It is only used when M11 is used to unclamp the chuck. A delay is provided to allow the chuck time to clamp prior to the next block executing. This is parameter 249 CHUCK CLAMP DELAY and is specified in milliseconds. The default is 500 for .5 seconds.

M11 Unclamp Chuck

The M11 code will unclamp the chuck. A delay is provided to allow the chuck time to unclamp prior to the next block executing. This is parameter 250 CHUCK UNCLAMP DELAY and is specified in milliseconds. The default is 500 for .5 seconds. If the spindle is turning, it will be stopped before the chuck is unclamped.

M12 Auto Air Jet On

The M12 code is used to activate the Auto Air Jet. The optional air blast is turned on with this M code. Additionally, M12 Pnnn (where nnn is in milliseconds) will turn it on for the specified time, then turn off automatically.

M13 Auto Air Jet Off

The M13 code is used to deactivate the Auto Air Jet. The optional air blast is turned off with this M code.

M14 CLAMP MAIN SPINDLE

This M code will clamp the main spindle.

M15 UNCLAMP MAIN SPINDLE

This M code will unclamp the main spindle.

**M17 Turret Rotation Always Forward**

The M17 code is a modal M code that forces the turret to rotate in the forward direction when a tool change is made. Whereas most M codes are invoked as the last action in a block, M17 works concurrently with other commands in the same block. This means that the following command will cause the tool turret to advance in the forward direction to tool #1.

N1 T0101 M17;

Because M17 is modal, any subsequent T command will cause the turret to rotate in the forward direction to the commanded tool. Refer to the description of Setting 97, TOOL CHANGE DIRECTION, for more information.

M18 Turret Rotation Always Reverse

The M18 code is a modal M code that forces the tool turret to rotate in the reverse direction when a tool change is made. Whereas most M codes are invoked as the last action in a block, M18 works concurrently with other commands in the same block. This means that the following command will cause the tool turret to advance in the reverse direction to tool #10.

N1 T1010 M18;

Because M18 is modal, any subsequent T command will cause the turret to rotate in the reverse direction to the commanded tool. Refer to the description of Setting 97, TOOL CHANGE DIRECTION, for more information.

M19 Orient Spindle (P and R values optional)

The M19 code is used to orient the spindle to a fixed position. A P value can be added as an option that will cause the spindle to be oriented to a particular angle (in degrees). For example, M19 P270 will orient the spindle to 270 degrees. An R value will recognize up to four places to the right of the decimal point. R value precision is limited to 4 decimal places. Note: the actual position accuracy of the spindle is further limited by the servo-encoder system resolution. An M19 R123.4567 will position the spindle to the angle specified by the R value.

M21 Tailstock Forward

The M21 code uses Settings 105, 106 and 107 to move to the tailstock HOLD POINT.

M22 Tailstock Reverse

The M22 code uses Setting 107 to move the tailstock to the RETRACT POINT.

M23 Thread Chamfer ON

The M23 code commands the control to execute a chamfer at the end of a thread executed by G76 or G92. This M code is modal. It remains in effect until changed by M24. Refer to Settings 95 and 96 to control the chamfer size and angle.



M24 Thread Chamer OFF

The M24 code commands the control to perform no chamfering at the end of a G76 or G92 threading cycle. This M code is modal. M24 may be cancelled with an M23, program reset, or a POWER ON condition.

M30 Prog End and Reset

The M30 code is used to stop a program. It also stops the spindle and turns off the coolant. The program pointer will be reset to the first block of the program and stop. The parts counters displayed on the current commands display are also incremented. M30 will also cancel tool length offsets.

M31 Chip Conveyor Forward

M31 starts the chip conveyor motor in the forward direction. The forward direction is defined as the direction that the conveyor must move to transport chips out of the work cell. The conveyor will not turn if the door is open. This may be overridden by setting bit 17 of parameter 209 (CNVY DR OVRD). See also Setting 114 and 115

M32 Chip Conveyor Reverse

M32 starts the chip conveyor motor in the reverse direction. The reverse direction is defined as the direction opposite of forward. The conveyor will not turn if the door is open. This may be overridden by setting bit 17 of parameter 209 (CNVY DR OVRD). See also Setting 114 and 115

M33 Chip Conveyor Stop

M33 Stops Conveyor motion.

M36 Parts Catcher Up

This code is used to activate the optional parts catcher. It will rotate the parts catcher counterclockwise, into position to catch a part.

M37 Parts Catcher Down

This code is used to deactivate the optional parts catcher. It will rotate the parts catcher clockwise, out of the work envelope.

M41 Low Gear

The M41 code is used to select low gear. The spindle will come to a stop when changing gears. M41 is ignored if there is no gear box.

The machine will remain in its current gear even after the machine is powered off. When the machine is powered up, it will be in the same gear (or between gears) as when it was powered off.

M42 High Gear

The M42 code is used to select high gear. The spindle will come to a stop when changing gears. M42 is ignored if there is no gear box.



The machine will remain in its current gear even after the machine is powered off. When the machine is powered up, it will be in the same gear (or between gears) as when it was powered off.

M43 Turret Unlock

For Service use only.

M44 Turret Lock

For Service use only.

M51-M58 Optional User M ON

The M51 through M58 codes are optional for user interfaces. They will activate one of relays 1132 through 1139 and leave it active. These are the same relays used for M121-M128. Use M61-M68 to turn these off. The RESET key will turn off all of these relays. See 8M section for more information on additional user outputs

M61-M68 Optional User M OFF

The M61 through M68 codes are optional for user interfaces. They will deactivate one of relays 25 through 28. These are the same relays used for M121-M128.

M76 Disable Displays

This code is used to disable the updating of the screen displays. It is not necessary for machine performance.

M77 Enable Displays

This code is used to enable the updating of the screen displays. It is only used when M76 has been used to disable the displays.

M78 Alarm if Skip Signal Found

This code is used to generate an alarm if the previous skip function actually got the skip signal. This is usually used when a skip signal is not expected and may indicate a probe crash. This code can be placed in a block with the skip function or in any subsequent block. The skip functions are G31, G36, and G37.

M79 Alarm If Skip Signal Not Found

This code is used to generate an alarm if the previous skip function did not actually get the skip signal. This is usually done when the absence of the skip signal means a positioning error of a probe. This code can be placed in a block with the skip function or in any subsequent block. The skip functions are G31, G36, and G37.

M85 Open Automatic Door (optional)

On lathes fitted with an auto door, an M85 will cause the door to open and an M86 will cause it to close. To use this feature, the following settings must be made:

- Setting 51 DOOR HOLD OVERRIDE set to ON,
- Parameter 57 bit 31 DOOR STOP SP set to zero,
- Setting 131 AUTO DOOR set to ON.

The control will beep while the door is in motion.

M86 Close Automatic Door (optional)

M88 Turns On High Pressure Coolant (optional)

Parameter 209 bit 24 must be set to 1 for this command to function.

M89 Turns Off High Pressure Coolant (optional)

Parameter 209 bit 24 must be set to 1 for these commands to function.

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 goes high. The format is:

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

M93 causes the control to watch the discrete input specified by the Q value, and when it goes high, captures the position of the axis specified by the P value. The position is then copied to hidden macro variables 749. M94 stop 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. Sleep mode can be used when the user wishes his machine to begin warming itself up early in the morning and be ready for use upon his arrival. The format of the M95 command is:

M95 (hh:mm)

The comment immediately following the M95 must contain the hours and minutes that the machine is to sleep for. For example, if the current time were 6pm and the user wanted the machine to sleep until 6:30am the next morning, the following command would be used:

M95 (12:30)

Up to 99 hours can be specified thus allowing the machine to sleep for over 4 days. If the time is specified using an incorrect format, alarm 324 DELAY TIME RANGE ERROR will be generated. When the machine enters sleep mode, and display the following message:

HAAS
SLEEP MODE
REMAINING TIME nnn MIN.



The message will be re-displayed in a different position on the screen each second so that the user can see at a glance that the machine is sleeping. This has the advantage of preventing the message from being "burned" into one spot on the screen.

When less than one minute of sleep time remains, the message will change to:

REMAINING TIME nn SEC.

If the user presses any key or opens the door, sleep mode will be cancelled, and the active program will wait at the block following the M95 until the user presses the Cycle Start key.

For the last 30 seconds of the sleep time, the machine will beep and display an additional message:

WAKE UP IN nn SECONDS

When the sleep time has elapsed, and the active program will continue at the block following M95.

M96 Jump If No input

- P Block to branch to when conditional test succeeds
Q Discrete input to test, 0..31

This code is used to test a discrete input for 0 status. When this block is executed and the input signal specified by Q is 0, a branch to the block specified by P is performed. A Pnnnn code is required and must match a line number within the same program. The Q value must be in the range of 0 to 31. These correspond to the discrete inputs found on the diagnostic display page with the upper left being input 0 and the lower right being 31. Q is not required within the M96 block. The last specified Q will be used. This command stops the look-ahead queue until the test is made at runtime. Since the look-ahead queue is exhausted, M96 cannot be executed when cutter compensation is invoked. M96 cannot be executed from a main DNC program. If you wish to use M96 in DNC, it must be in a resident subroutine called from the DNC program.

The following is an M96 example:

```
N05 M96 P5 Q8      (TEST INPUT DOOR S, UNTIL CLOSED);  
N10                 (START OF SOME PROGRAM LOOP);  
.                  (PROGRAM THAT MACHINES PART);  
.                (EXECUTE AN EXTERNAL USER FUNCTION)  
N85 M21             (LOOP TO N10 IF SPARE INPUT IS 0);  
N90 M96 P10 Q27     (IF SPARE INPUT IS 1 THEN END PROGRAM);  
M95 M30
```

M97 Local Sub-Program Call

This code is used to call a subroutine referenced by a line N number within the same program and listed after lathe M30. A Pnnnn code is required and must match a line number within the same program. This is useful for simple subroutines within a program and does not require the complication of a separate program. The subroutine must still be ended with an M99. An L count on the M97 block will repeat the subroutine call that number of times.



M98 Sub Program Call

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

M99
```

M99 Sub Program Return Or Loop

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

M99 Pnnnn in the HAAS control varies from that seen in FANUC compatible controls. In FANUC compatible controls, M99 Pnnnn will return to the calling program and resume execution at block N specified in Pnnnn. For the HAAS control, M99 will NOT return to the calling program, but instead will jump to block N specified in Pnnnn in the current program.

FANUC behavior can be simulated 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

If you have 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. There are many ways to jump conditionally after a M99 return when using macros.

M119 SUBSPINDLE ORIENT

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



M119 Pxxx
M119 Rx.x

M121-M128 Optional User M

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

M133 LIVE TOOL DRIVE FORWARD

The live tooling motor is mounted on the Z axis. The format for this command is:
M133 Pxxx where xxx is the commanded spindle speed in RPM. M133 will cause the motor to rotate forward.

M134 LIVE TOOL DRIVE REVERSE

The live tooling motor is mounted on the Z axis. The format for this command is:
M134 Pxxx where xxx is the commanded spindle speed in RPM. M134 will cause the motor to rotate in reverse.

M135 LIVE TOOL DRIVE STOP

The live tooling motor is mounted on the Z axis. The format for this command is:
M135 Pxxx where xxx is the commanded spindle speed in RPM. M135 will cause the motor to decelerate to zero.

M143 SUB SPINDLE FWD

The subspindle is mounted on the tailstock. The format for this command is:
M143 Pxxx where xxx is the commanded spindle speed in RPM. M143 will cause the subspindle to rotate forward, M144 will cause the subspindle to rotate in reverse, M145 will cause the subspindle to decelerate to zero.

M144 SUB SPINDLE REV

The subspindle is mounted on the tailstock. The format for this command is:
M144 Pxxx where xxx is the commanded spindle speed in RPM. M144 will cause the subspindle to rotate in reverse.

M145 SUBSPINDLE STOP

The subspindle is mounted on the tailstock. The format for this command is:
M145 Pxxx where xxx is the commanded spindle speed in RPM. M145 will cause the subspindle to decelerate to zero.

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

M154 C-AXIS ENGAGE

This M code is used to engage the optional C-axis motor. Parameter 278 bit 26 C Axis Drive must be set to 1.

M155 C-AXIS DISENGAGE

This M code is used to disengage the optional C-axis motor. Parameter 278 bit 26 C Axis Drive must be set to 1.



M CODES

SL Series OPERATOR'S MANUAL

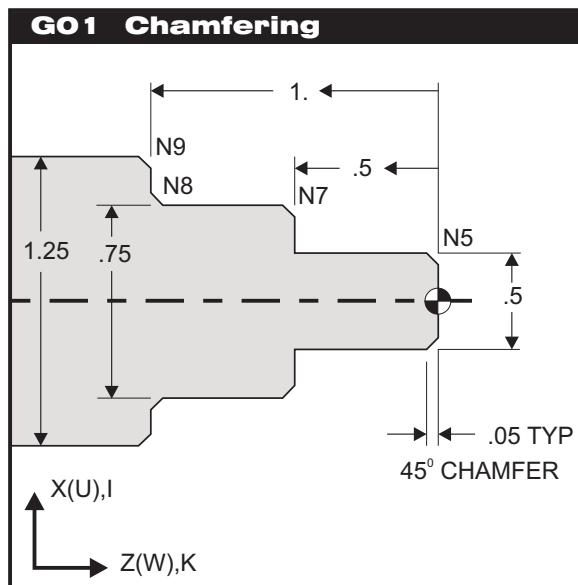
June 2001



7. PROGRAMMING EXAMPLES

G01 CHAMFERING AND CORNER ROUNDING

G01 Chamfering



PROGRAM EXAMPLE

```

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
G51
M30
%
```

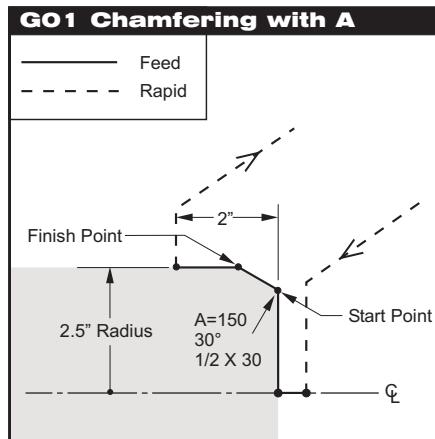
**G01 Chamfering with A****Group 01**

F	Feed rate
U	Optional X-axis incremental motion command
W	Optional Z-axis incremental motion command
X	Optional X-axis absolute motion command
Z	Optional Z-axis absolute motion command
A	Optional angle of movement (used with only one of X, Z, U, W)

This G code provides for straight line (linear) motion from point to point. Motion can occur in 1 or 2 dimensions. Both axes will start and finish motion at the same time. The speeds of all axes are controlled so that the feed rate specified is achieved along the actual path. The F command is modal and may be specified in a previous block. Only the axes specified are moved and the incremental or absolute commands will change how those values are interpreted. The auxiliary axes B and V can also be moved with a G01 but only one axis is moved at a time.

When specifying an angle use only one of the other axes, the corresponding X or Z destination is calculated based on the angle.

CAUTION: This G Code is not supported in roughing canned cycles G71, G72 or G73.

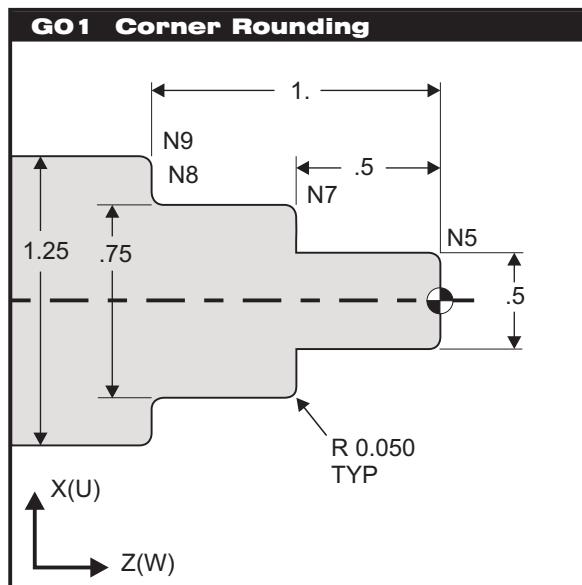
**PROGRAM EXAMPLE**

```
G54;  
M03 S1500;  
T606  
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;
```

NOTE: A -30 = A 150; A -45 = A 135



G01 Corner Rounding



PROGRAM EXAMPLE

```

Automatic Corner Rounding
%
O0005 (AUTOMATIC CORNER ROUNDING)
N1 G50 S1500
N2 G00 T101 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
G51
M30
%
```



Type	Syntax	Action / Movement
Chamfering $Z+ \rightarrow X+/-$	N1 G01 Zb li N2 Xc	G01 Z(b - i) G01 X(POS current + i) Zb G01 Xc
Chamfering $Z- \rightarrow X+/-$		G01 Z(b + i) G01 X(POS current + i) Zb G01 Xc
Chamfering $X- \rightarrow Z+/-$	N1 G01 Xb Kk N2 Zc	G01 X(b + k) G01 Z(POS current + k) Xb G01 Zc
Chamfering $X+ \rightarrow Z+/-$		G01 X(b - k) G01 Z(POS current + k) Xb G01 Zc



Corner rounding $Z+ \rightarrow X+/-$ $Z- \rightarrow X+/-$	N1 G01 Zb Rr N2 Xc $G01 Z(b - r)$ r: $G03 X(POS_{current} + r) Zb R r $ -r: $G02 X(POS_{current} + r) Zb R r $ $G01 Xc$ $G01 Z(b + r)$ r: $G02 X(POS_{current} + r) Zb R r $ -r: $G03 X(POS_{current} + r) Zb R r $ $G01 Xc$	
Corner rounding $X- \rightarrow Z+/-$ $X+ \rightarrow Z+/-$	N1 G01 Xb Rr N2 Zc $G01 X(b + r)$ r: $G03 Z(POS_{current} + r) Xb R r $ -r: $G02 Z(POS_{current} + r) Xb R r $ $G01 Zc$ $G01 X(b - r)$ r: $G02 Z(POS_{current} + r) Xb R r $ -r: $G03 Z(POS_{current} + r) Xb R r $ $G01 Zc$	

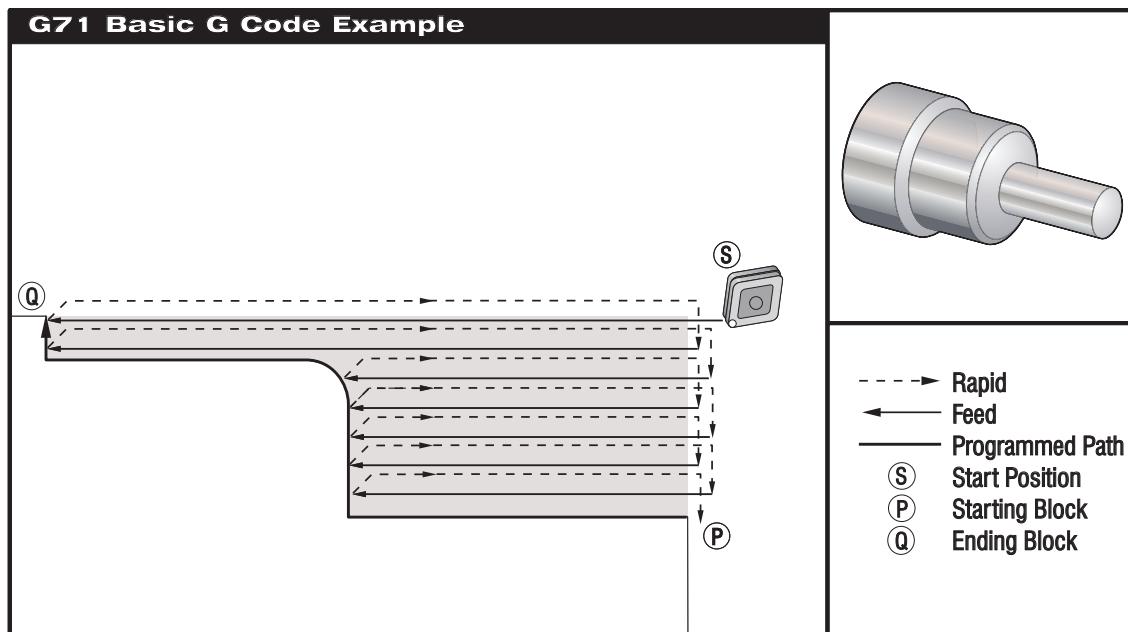
NOTES: 1) Incremental programming is possible if Ub or Wb is specified in place of Xb or Zb, respectively. So its actions will be as follows: $X(POS_{current} + i) = U_i$, $Z(POS_{current} + k) = W_k$, $X(POS_{current} + r) = U_r$, $Z(POS_{current} + r) = W_r$.
 2) **POS_{current}** indicates current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

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 can not be used in a threading cycle **or 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

WARNING!

All rules must be conformed to, otherwise the outcome will be unpredictable.

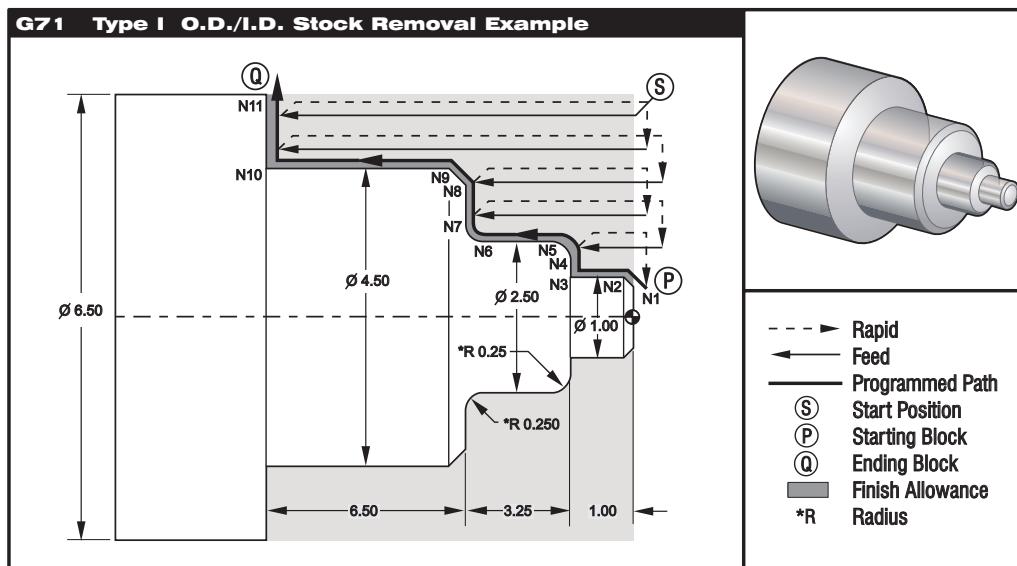
**G71 O.D. STOCK REMOVAL CYCLE****PROGRAM EXAMPLE**

```
%  
O0070  
G00 T101  
G50 S2500  
G97 S509 M03  
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  
X3.4376  
G03 X4.0002 Z-3.2813 R0.2813  
G01 Z-6.  
N2 X6.  
G70 P1 Q2  
M05  
G28  
M30  
%
```

DESCRIPTION

(G71 Roughing Cycle)

(FINISH PASS)

**G71 TYPE 1 O.D. / I.D. Stock Removal****PROGRAM EXAMPLE**

```

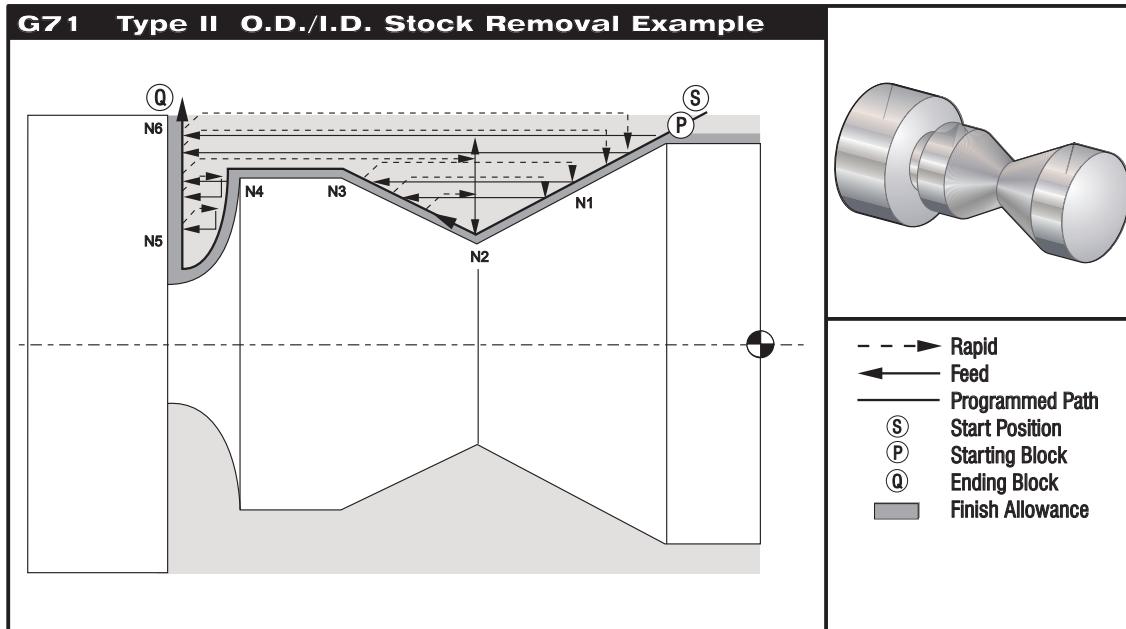
%
O0071
G20
G00 G54 X6.6 Z.05 M08
T101 (CNMG 432)
G50 S2000
G97 S636 M03
G96 S750
G71 P1 Q11 D0.15 U0.01 W0.005 F0.012
N1 G00 X0.6634
N2 G01 X1. Z-0.1183 F0.004
N3 Z-1.
N4 X1.9376
N5 G03 X2.5 Z-1.2812 R0.2812
N6 G01 Z-3.0312
N7 G02 X2.9376 Z-3.25 R0.2188
N8 G01 X3.9634
N9 X4.5 Z-3.5183
N10 Z-6.5
N11 X6.0
G00 X0 Z0 T100
T202
G50 S2500
G97 S955 M03
G00 X6. Z0.05 M08
G96 S1500
G70 P1 Q11
G00 X0 Z0 T200
M30
%

```

DESCRIPTION**(FANUC G71 TYPE I EXAMPLE)**

(Check for Inch Setting)
(Rapid to Home Position)
(Tool change & apply Offsets)
(Set Max RPM 2000)
(Spindle On)
(Constant surface speed On)
(Define rough cycle)
P (Begin definition)
(Finish pass .004" Feed)

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

**G71 TYPE 2 O.D. / I.D. Stock Removal****PROGRAM EXAMPLE**

```
%  
O0001  
G50 T5100  
T101  
S1200 M03  
;  
G0 X2.Z0  
G71 P1 Q6 D.035 U.03 W.01 R1 F.01  
;  
N1 G01 X1.5 Z-.5 F.004  
N2 X1.Z-1.  
N3 X1.5 Z-1.5  
N4 Z-2.  
N5 G02 X.5 Z-2.5 R.5  
N6 G01 X2.  
;  
G50 T5200  
T202  
S1500 M03  
;  
G70 P1 Q6  
;  
G28 M30  
%
```

DESCRIPTION

(YASNAC G71 TYPE II EXAMPLE)
(Set YASNAC style tool shift)
(Roughing tool)

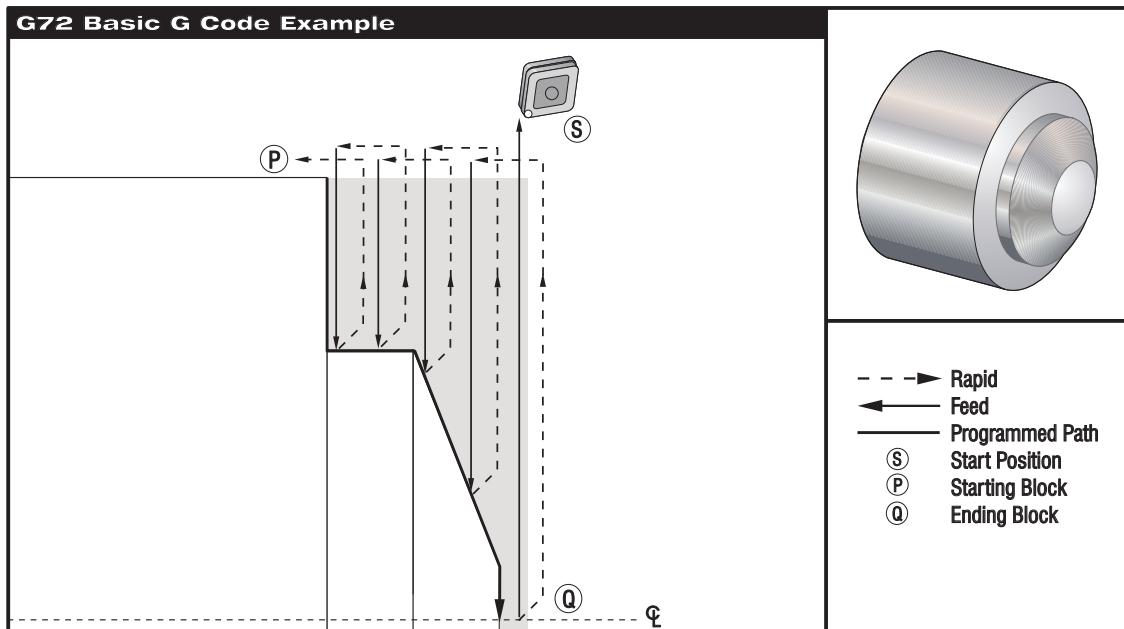
S (*Start position*)

P (*PQ Path definition*)

Q (*PQ Path end*)

(*Finishing tool*)

(*Finish pass*)

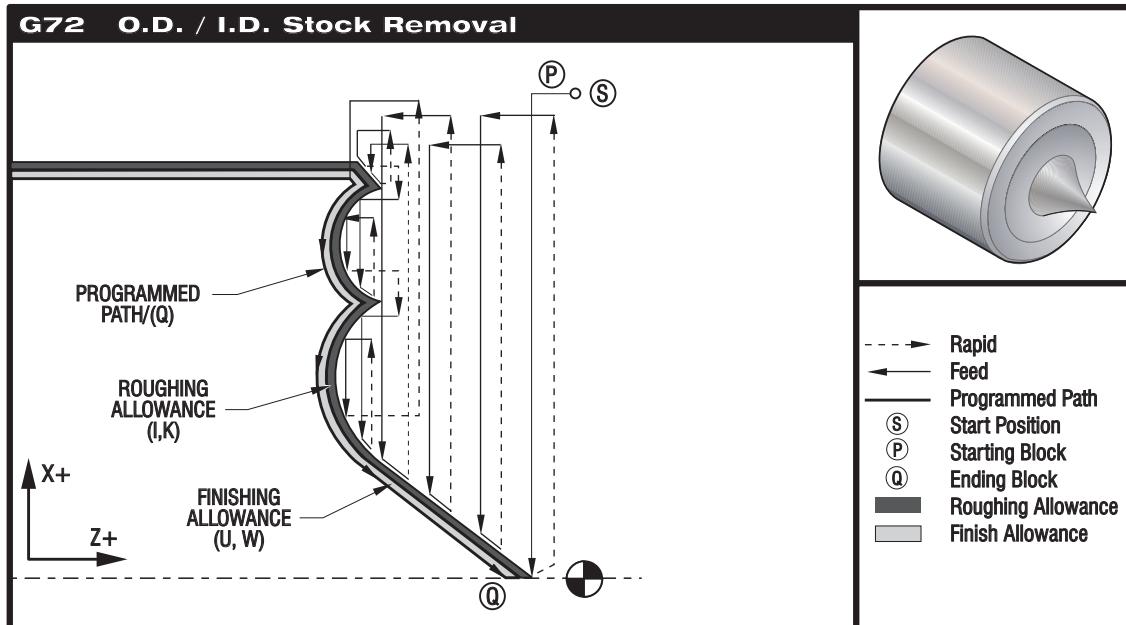
**G72 Face Stock Removal Type 1****PROGRAM EXAMPLE**

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

DESCRIPTION

(G72 Roughing Cycle)

(FINISH PASS)

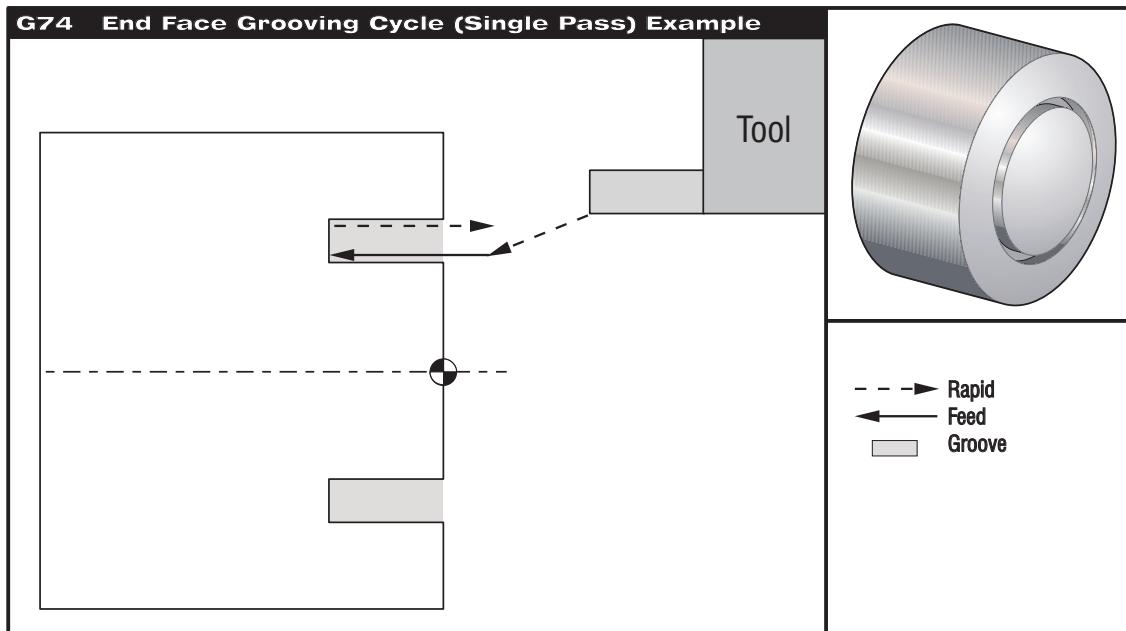
**G72 Face Stock Removal Type 2****PROGRAM EXAMPLE**

```
%  
00722  
G28  
T101  
S1000 M03  
G00 G54 X2.1 Z0.1  
G72 R1 P1 Q2 D0.06 I0.02 K0.01 U0.02 W0.01 S1100 F0.015  
N1 G01 Z-0.46 X2.1 F0.005  
X2.  
G03 X1.9 Z-0.45 R0.2  
G01 X1.75 Z-0.4  
G02 X1.65 Z-0.4 R0.06  
G01 X1.5 Z-0.45  
G03 X1.3 Z-0.45 R0.12  
G01 X1.17 Z-0.41  
G02 X1.03 Z-0.41 R0.1  
G01 X0.9 Z-0.45  
G03 X0.42 Z-0.45 R0.19  
G03 X0.2 Z-0.3 R0.38  
N2 G01 X0.01 Z0  
G70 P1 Q2  
M05  
G28  
M30  
%
```

DESCRIPTION

(G72 ROUGHING CYCLE)

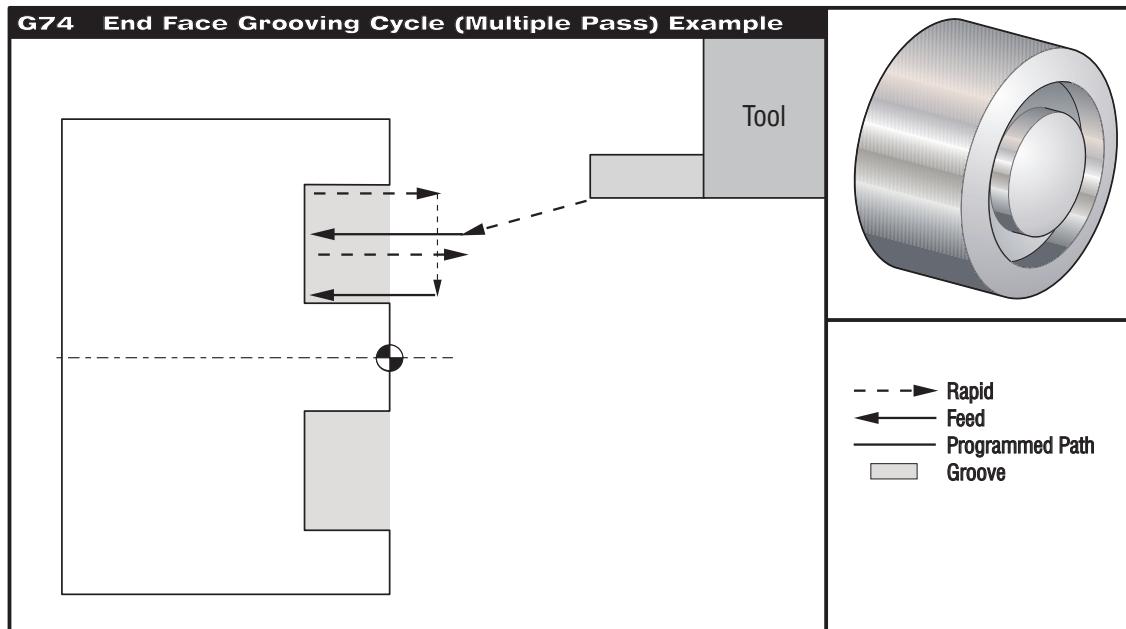
(FINISH PASS)


G74 Face Grooving Cycle (Single Pass)

PROGRAM EXAMPLE

```
%  
O0071  
T101  
M03 S750  
G00 X3. Z0.05  
G74 Z-0.5 K0.1 F0.01  
G28  
M30  
%
```

DESCRIPTION

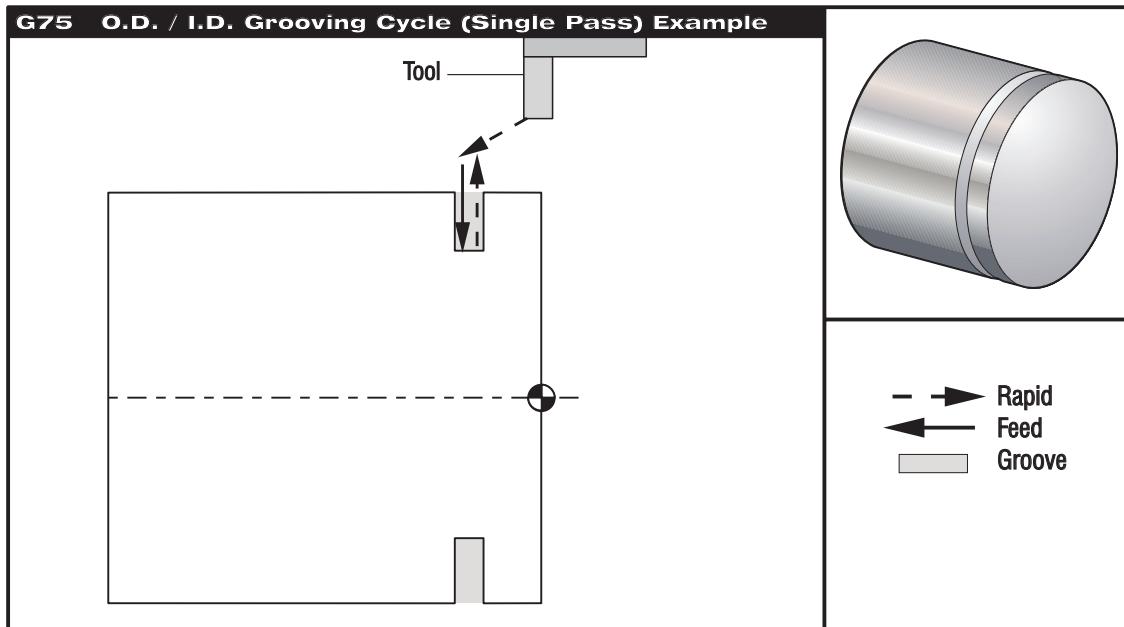
(Rapid to Start position)
(Feed Z-.5 with a .100" peck)

**G74 FACE GROOVING (MULTIPLE PASS)****PROGRAM EXAMPLE**

```
%  
O0074  
T101M03 S750  
G00 X3. Z0.05  
G74 X1.75 Z-0.5 I0.2 K0.1 F0.01  
G28  
M30  
%
```

DESCRIPTION

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


G75 GROOVING CYCLE (SINGLE PASS)

PROGRAM EXAMPLE

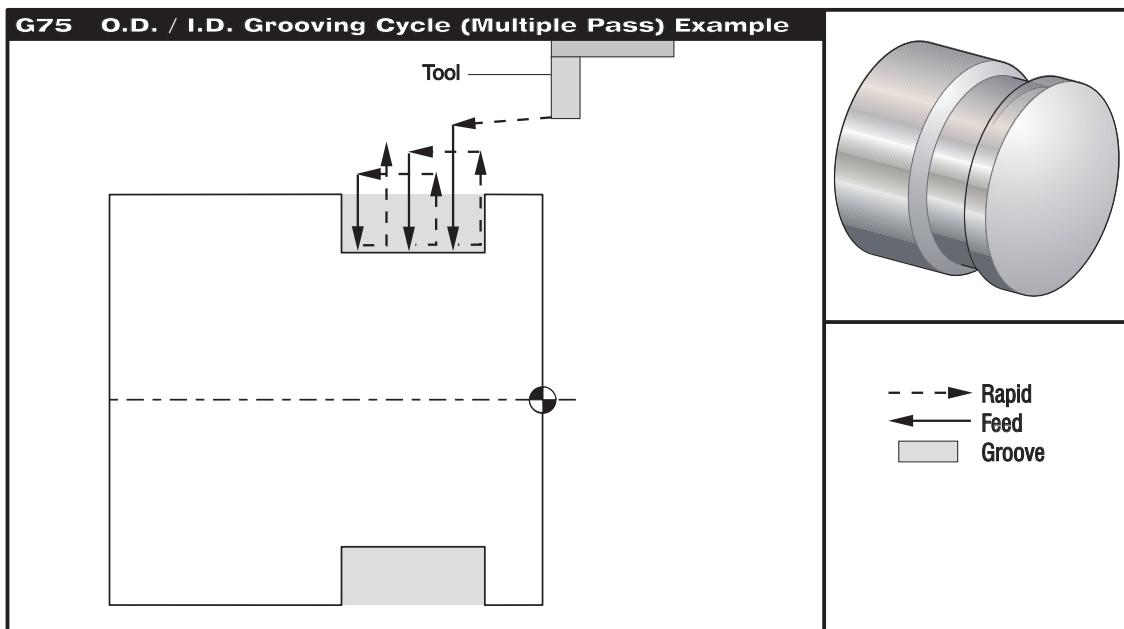
```
%  
O0075  
T101M03 S750  
G00 X4.1 Z0.05  
G01 Z-0.75 F0.05  
G75 X3.25 I0.1 F0.01  
G00 X5. Z0.1  
G28  
M30  
%
```

DESCRIPTION

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

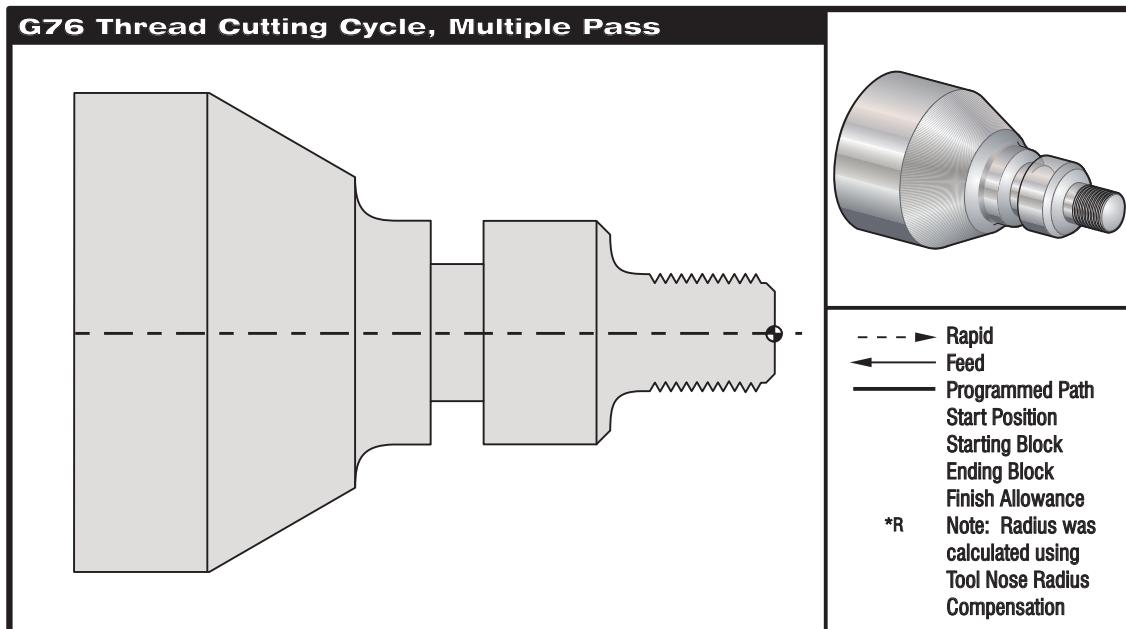
**G75 GROOVING CYCLE (MULTIPLE PASS)**

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

**PROGRAM EXAMPLE**

```
%  
O0075  
T101M03 S750  
G00 X4.1 Z0.05  
G01 Z-0.75 F0.05  
G75 K0.2 X3.25 I0.1 Z-1.75 F0.01  
(Rapid to Clear position)  
(Feed to Groove location)  
(O.D./I.D. Peck groove multiplepass)  
G00 X5. Z0.1  
G28  
M30  
%
```

DESCRIPTION


G76 THREAD CUTTING CYCLE (MULTIPLE PASS)

PROGRAM EXAMPLE

```

G54
G50 S2000
G97 S2000 T101 M03
G00 X3.1 Z0.5
M08
G96 S1200
G01 Z0 F0.01
X-0.04
G00 Z0.025 X3.
G71P1 Q10 F0.015 U0.035 W0.005 D0.125
N1 X0.875 Z0
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
G00 Z0.1
G54 T100 Z0
N20

G54
G50 S2000
G97 S1200 T505 M03
G00 X1.5 Z0.5
  
```

DESCRIPTION

(Select work coordinate)
 (Set max RPM select tool geometry)
 (Spindle on select tool one offset one)
 (Rapid to reference point)
 (Coolant ON)
 (Constant surface speed ON)
 (Position to part Z0)

(Define roughing cycle)
 (Begin tool path)

(End tool path)

(Thread sample program HAAS SL-Series FANUC System)

(Threading tool)
 (Rapid to position)



M08

G00 X1.1 Z0.1

G76 D0.0115 X0.913 K0.042 Z-0.85 F0.0714 (

(Threading cycle)

G00 X1.5 Z0.5 M09 G54 T500 Z0

N30*(HAAS SL-Series FANUC System)*

G54

G50 S2000

G97 S1200 T404 M03

(Groove tool)

G00 X1.625 Z0.5

M08

G96 S800

G01 Z-1.906 F0.025

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

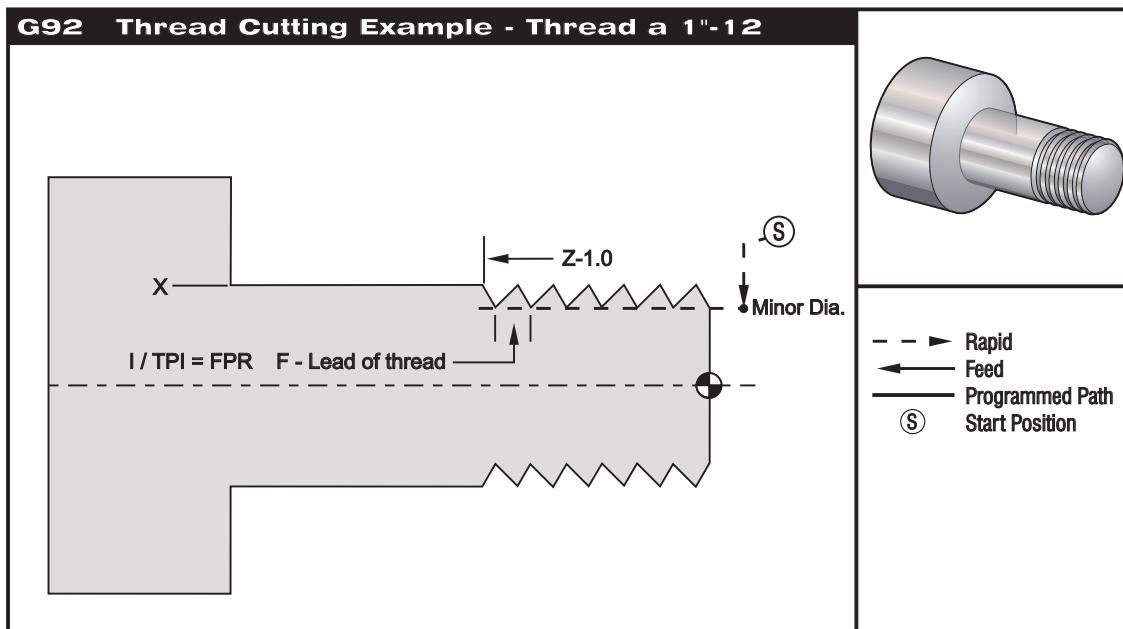
G01 X1.51

G00 X3. Z0.5 M09

G54 T400 Z0

M30

%

**MODAL CANNED CYCLE****G92 Thread Cutting***G92 Thread Cutting***PROGRAM EXAMPLE**

```
%  
X1.2 Z.2  
G92 X.980 Z-1.0 F.0833  
2nd pass X.965  
3rd pass X.955  
4th pass X.945  
5th pass X.935  
6th pass X.925  
7th pass X.917  
8th pass X.910  
9th pass X.905  
10th pass X.901  
11th pass X.899  
%
```

DESCRIPTION

(Rapid to Clear position)
 (Set up Threading cycle)
 (Subsequent Passes)
 "
 "
 "
 "
 "
 "
 "
 "
 "
 (Subsequent Passes)



PROGRAMMING EXAMPLES

SL OPERATOR'S MANUAL
Series

June 2001



8. SETTINGS

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

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

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

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

The settings are listed here with a description of each. The page title will precede each page of settings and the settings will appear in order as shown on the screen.

Setting No:	Description:	Range of value:
GENERAL		
26	SERIAL NUMBER	0 to 8380000
82	LANGUAGE	English, German, French, Spanish, Italian.
1	AUTO POWER OFF TIMER	0 to 9999 minutes
81	TOOL AT POWER DOWN	1 to 10
9	DIMENSIONING	INCH or METRIC
77	SCALE INTEGER F	ON or OFF
33	COORDINATE SYSTEM	YASNAC or FANUC
34	G CODE SYSTEM	A, B, or C
53	JOG W/O ZERO RETURN	ON or OFF
98	SPINDLE JOG RPM	0 to 10000 rpm
92	CHUCK CLAMPING	O.D Clamping/I.D. Clamping
64	T. OFS MEAS USES WORK	ON or OFF
93	TAIL ST. X CLEARANCE	0 to -8.2500 inches
94	ZB DIFF @X CLEARANCE	0 to 19 inches



Setting No:	Description:	Range of value:
-------------	--------------	-----------------

PROGRAM 1

2	POWER OFF AT M30	ON or OFF
31	RESET PROGRAM POINTER	ON or OFF
36	PROGRAM RESTART	ON or OFF
39	BEEP AT M30	ON or OFF
51	DOOR HOLD OVERRIDE	ON or OFF
56	M30 RESTORE DEFAULT G	ON or OFF
118	M99 BUMPS M30 CNTRS	ON or OFF
59	PROBE OFFSET X+	-30.0000 to +30.0000 inches
60	PROBE OFFSET X-	-30.0000 to +30.0000 inches
61	PROBE OFFSET Z+	-30.0000 to +30.0000 inches
62	PROBE OFFSET Z-	-30.0000 to +30.0000 inches
63	TOOL PROBE WIDTH	-30.0000 to +30.0000 inches
15	TOOL OFFSET AGREEMENT	ON or OFF
97	TOOL CHANGE DIRECTION	SHORTEST or M17/M18

PROGRAM 2

38	AUX AXIS NUMBER	0 to 1
43	CUTTER COMP TYPE	A or B
44	MIN F IN RADIUS CC%	1 to 100
58	CUTTER COMPENSATION	FANUC or YASNAC
85	MAX CORNER ROUNDING	0 to 0.25

PROGRAM 3

32	COOLANT OVERRIDE	NORMAL, OFF, or IGNORE
42	M00 AFTER TOOL CHANGE	ON or OFF
49	SKIP SAME TOOL CHANGE	ON or OFF
45	MIRROR IMAGE X-AXIS	ON or OFF
47	MIRROR IMAGE Z-AXIS	ON or OFF

PROGRAM 4

105	TS RETRACT DISTANCE	0 to 33.5
106	TS ADVANCE DISTANCE	0 to 33.5
107	TS HOLD POINT	0 to -33.5
121	FOOT PEDAL TS ALARM	
109	WARM-UP TIME IN MIN.	
110	WARMUP X DISTANCE	
112	WARMUP Z DISTANCE	



Setting No:	Description:	Range of value:
-------------	--------------	-----------------

CANNED CYCLE

95	THREAD CHAMFER SIZE	0 to 30.0000 inches
96	THREAD CHAMFER ANGLE	0 to 89 degrees
86	THREAD FINISH ALLOWANCE	0 to 1.0000 inch
99	THREAD MINIMUM CUT	0 to 0.9999 inch
72	CAN CYCLE CUT DEPTH	0 to 30.0000 inches
73	CAN CYCLE RETRACTION	0 to 30.0000 inches
22	CAN CYCLE DELTAZ	0 to 30.0000 inches
28	CAN CYCLE ACT W/O X/Z	ON or OFF
52	G83 RETRACT ABOVE R	0 to 30.0000 inches
57	EXACT STOP CANNED X-Z	ON or OFF

RS-232 PORTS

11	BAUD RATE SELECT	50, 110, 200, 300, 600, 1200, 2400, 4800, 7200, 9600, 19200, or 38400
12	PARITY SELECT	NONE, ODD, EVEN, ZERO
13	STOP BIT	1 or 2
14	SYNCHRONIZATION	XON/XOFF, RTS/CTS, DC CODES, or XMODEM
37	RS-232 DATA BITS	7 or 8
24	LEADER TO PUNCH	NONE, BLANK, or NULL
25	EOB PATTERN	CR LF, LF ONLY, CR ONLY, or LF CR CR
41	ADD SPACES RS232 OUT	ON or OFF
50	AUX AXIS SYNC	XON/XOFF or RTS/CTS
54	AUX AXIS BAUD RATE	50, 110, 200, 300, 600, 1200, 2400, 4800, 7200, 9600, 19200, 38400, 115200
69	DPRNT LEADING SPACES	ON or OFF
70	DPRNT OPEN/CLOS DCODE	ON or OFF
133	NETWORK/ZIP OFF/ON	ON or OFF
134	CONNECTION TYPE	FLOPPY, NET or ZIP
135	NETWORK TYPE	NONE, NOVELL, NT/IPX, or NT/TCP
136	SERVER	(server name)
137	USERNAME	(account name)
138	PASSWORD	(password)
139	PATH	(user specified network path)
140	TCPADDR	(TCP/IP address)
141	SUBNET	(user-specified subnet mask)

**CONTROL PANEL**

6	FRONT PANEL LOCK	ON or OFF
55	ENABLE DNC FROM MDI	ON or OFF
76	FOOT PEDAL LOCK OUT	ON or OFF
16	DRY RUN LOCK OUT	ON or OFF
17	OPT STOP LOCK OUT	ON or OFF
18	BLOCK DELETE LOCK OUT	ON or OFF
10	LIMIT RAPID AT 50%	ON or OFF
84	TOOL OVERLOAD ACTION	ALARM, FEEDHOLD, BEEP, PAUSE, AUTOFEED
103	CYC START/FH SAME KEY	ON or OFF
104	JOG HANDL TO SNGL BLK	ON or OFF

MISCELLANEOUS

114	CONVEYOR CYCLE	0 - 1440
115	CONVEYOR ON-TIME	0 - 1440

Setting No: **Description:** **Range of value:****EDITING**

7	PARAMETER LOCK	ON or OFF
119	OFFSET LOCK	ON or OFF
120	MACRO VAR LOCK	ON or OFF
8	MEMORY PROTECT	ON or OFF
23	9xxx PROGS EDIT LOCK	ON or OFF
74	9xxx PROGS TRACE	ON or OFF
75	9xxx PROGS SINGLE BLK	ON or OFF

GRAPHICS

3	RESERVED	N/A
4	GRAPHICS RAPID PATH	ON or OFF
5	GRAPHICS DRILL POINT	ON or OFF
65	GRAPH SCALE (HEIGHT)	0 to 16.250
66	GRAPHICS X OFFSET	0 to 16.0
68	GRAPHICS Z OFFSET	0 to 16.0
90	GRAPHICS Z ZERO LOCATION	
91	GRAPHICS X ZERO LOCATION	

OVERRIDES

19	FEED RATE OVERRIDE LOCK	ON or OFF
20	SPINDLE OVERRIDE LOCK	ON or OFF
21	RAPID OVERRIDE LOCK	ON or OFF
87	M06 RESETS OVERRIDES	ON or OFF
83	M30/RESETS OVERRIDES	ON or OFF
88	RESET RESETS OVERRIDES	ON or OFF
144	FEED OVERRIDE->SPINDLE	ON or OFF



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

1 AUTO POWER OFF TIMER

Specifies the number of minutes with no activity before the machine will turn itself off. The process requires 15 seconds. The sequence is:

- 15 seconds: Start of count down
- 10 seconds: Coolant off
- 5 seconds: Servo motors off
- 0 seconds: Power off

2 POWER OFF AT M30

This is an On/Off setting. If it is set to ON, the machine will begin an automatic power down when an M30 ends a program. The auto off sequence gives the operator a 30 second warning and pressing any key will interrupt the sequence.

4 GRAPHICS RAPID PATH

This is an On/Off setting. It changes what is displayed in graphics. When it is off, the rapid motions do not leave a trail. When it is on, rapid motions leave a dashed line on the screen.

5 GRAPHICS DRILL POINT

This is an On/Off setting. It changes what is displayed in graphics. When it is off, nothing is added to the graphics display. When it is on, any motion in the Z-axis will leave an X mark on the screen.

6 FRONT PANEL LOCK

This is an On/Off setting. When it is off, the machine operates normally. When it is on, the spindle CW and CCW buttons are disabled.

7 PARAMETER LOCK

This is an On/Off setting. When it is off, parameters can be changed. When it is on, parameter changes are locked out except for parameters 81 through 100. When the control is turned on, this setting is set to On.

8 PROG MEMORY LOCK

This is an On/Off setting. When it is off, and the key switch (if installed) is unlocked memory can be edited. When it is on, memory edit functions are locked out.

9 DIMENSIONING

This is an Inch/Metric setting. When it is set to Inch, the programmed units for X and Z are inches to 0.0001. When it is set to Metric, programmed units are millimeters to 0.001. If Setting 9 is changed from INCH to MM, all offset values will be converted accordingly. Tool Tip Values are unaffected.



	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

NOTE: Changing this setting will not automatically translate a program already stored in memory. You must reload your programs for the new units. When set to Inch, the Group 6 default G Code is G20. When set to Metric, the default G Code is G21.

10 LIMIT RAPID AT 50%

This is an On/Off setting. When it is off, the highest rapid speed of 100% is available normally. When it is on, the highest rapid rate is limited to 50% of maximum. When you press the 100% button, the display will indicate a 50% rapid override. When this setting is turned on, the rapid override will not automatically change from 100% to 50%; you must press the 100% override buttons to get 50%. If the machine is turned on after this setting is turned on, the maximum override will automatically be limited to 50%.

11 BAUD RATE SELECT

This setting allows the operator to change the serial data rate for the first serial port. This applies to program, settings, offsets, and parameters upload and download and to DNC functions.

12 PARITY SELECT

This setting allows the setting of parity for the first serial port. The possible values are: NONE, ODD, EVEN, ZERO. When set to none, no parity bit is added to the serial data. When set to zero, a 0 bit is added in the place of parity. Even and odd work like normal parity functions. Make sure you know what your system needs. XMODEM must use 8 data bits and no parity.

13 STOP BIT

This setting changes the number of stop bits for the first serial port. It can be selected to be 1 or 2.

14 SYNCHRONIZATION

This changes the synchronization protocol between sender and receiver for the first serial port. It can be RTS/CTS or XON/XOFF. 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 it is set to XON/XOFF, those ASCII character codes are used by the receiver to tell the sender to temporarily stop. XON/XOFF is the most common setting. DC CODES is like XON/XOFF but the paper tape punch or reader start/stop codes are sent.



15 XMODEM

Is a receiver-driven communications protocol that sends data in blocks of 128 bytes. Setting synchronization to XMODEM gives your RS-232 communication an added level of reliability because each block is checked for integrity. If the receiver determines that the most recently sent block is in error, it will request that the sender try to send the block again.

In order to use XMODEM, parity must be none, and RS-232 data bits must be set to 8. Also, the computer that is sending the data must be equipped with a communications package that supports the XMODEM protocol. It must be set to XMODEM to operate.

This version of XMODEM supports checksum verification only. Also, 512 bytes of memory must be available before using XMODEM with DNC.

15 TOOL OFFSET AGREEMENT

This is an On/Off setting. When it is Off, no special functions occur. When it is set to On, a check is made to ensure that the tool offset code matches the tool in the cutting position. This check can help to prevent crashes. In program restart, this check is not done until motion begins.

16 DRY RUN LOCK OUT

This is an On/Off setting. When it is off, the machine operates normally. When it is on, the DRY RUN function cannot be turned on.

17 OPT STOP LOCK OUT

This is an On/Off setting. When it is off, the machine operates normally. When it is on, the OPTIONAL STOP function cannot be turned on.

18 BLOCK DELETE LOCK OUT

This is an On/Off setting. When it is off, the machine operates normally. When it is on, the BLOCK DELETE function cannot be turned on.

19 FEED RATE OVERRIDE LOCK

This is an On/Off setting. When it is off, the machine operates normally. When it is on, the feed rate override buttons are locked out.

20 SPINDLE OVERRIDE LOCK

This is an On/Off setting. When it is off, the machine operates normally. When it is on, the spindle speed override buttons are locked out.

21 RAPID OVERRIDE LOCK

This is an On/Off setting. When it is off, the machine operates normally. When it is on, the rapid speed override buttons are locked out.

22 CAN CYCLE DELTA Z

This is a decimal numeric entry. It must be in the range of 0.0 to 29.9999 inches. This setting specifies the amount of axis movement that occurs to clear chips in the pecking canned cycles. For G83 and G74 it is the amount that the Z axis moves. For G75 it is the amount that the X axis moves represented in diameter programming.

**23 9xxx PROGS EDIT LOCK**

This is an On/Off setting. When it is off, the machine operates normally. When it is on, the 9000 series programs (usually macro programs) are invisible to the operator and cannot be uploaded or download. They also cannot be listed, edited, or deleted.

24 LEADER TO PUNCH

This setting is used to control the leader sent to a paper tape punch device connected to the first RS-232 port. The values that can be selected are: NONE, BLANK, or NULL. None causes no extra data to be sent as a leader. Blank causes two feet of blanks to be punched at the start of a program and one foot of blanks at the end. Null causes the same thing as blanks but uses the ASCII code null which is all zero.

25 EOB PATTERN

This setting controls what is sent out and expected as input to represent the EOB (end of block) on serial port one. The possible selections are: CR LF, LF only, CR only, or LF CR CR.

26 SERIAL NUMBER

This is a numeric entry. It is the serial number of your machine. It cannot be changed.

28 CAN CYCLE ACT W/O X/Z

This is an On/Off setting. When it is off, an initial canned cycle definition without an X or Z motion will not cause the canned cycle to be executed. When it is on, the initial canned cycle definition will cause one cycle to be executed even if there is no X or Z motion in that command block. Note that if an L0 is in that block, it will never execute the canned cycle on the definition line.

31 RESET PROGRAM POINTER

This is an On/Off setting. When it is off, the RESET button will not change the execution program pointer. When it is on, a RESET will change the program execution pointer to the beginning of the program.

32 COOLANT OVERRIDE

This setting controls how the coolant pump operates. The possible selections are: NORMAL, OFF, or IGNORE. The "NORMAL" setting allows the operator to turn the pump on and off manually or with **M** codes. The "OFF" setting will generate an alarm if an attempt is made to turn the coolant on manually or from a program. The "IGNORE" setting will ignore all 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. The default is YASNAC.

34 G CODE SYSTEM

This setting determines which G code system (A, B, or C) the lathe will use to interpret user programs. The HAAS lathe currently implements only system A codes.



36 PROGRAM RESTART

This is an On/Off setting. When it is off, starting a program from anywhere other than the beginning may produce inconsistent results. When it is on, starting a program from the middle causes the entire program to be scanned to ensure that the correct tools, offsets, G codes, and axes positions are set correctly before starting at the block where the cursor is positioned. Some alarm conditions are not detected prior to motion starting.

37 RS-232 DATA BITS

This setting can be selected to be either 7 or 8. It is used to change the number of data bits for serial port one. Normally, seven data bits should be used. Some computers require eight. Note that parity is added to this count. 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 AT M30

This is an On/Off setting. When it is off, nothing is changed. When it is on, a program ending in an M30 will cause the keyboard beeper to sound until another keyboard key is pressed.

41 ADD SPACES RS232 OUT

This is an On/Off setting. When it is off, programs sent out the serial port have no spaces and are difficult to read. When it is on, spaces are added between address codes when a program is sent out RS-232 serial port one. This can make program much easier to read.

42 M00 AFTER TOOL CHANGE

This is an On/Off setting. When it is off, tool changes occur normally. When it is on, a program stop will occur after a tool change and M00 AFTER TOOL CHANGE is displayed as a message at the bottom left. This affects only programmed tool changes.

43 CUTTER COMP TYPE

This setting controls how an entry to cutter compensation occurs. It can be selected to be A or B. It affects only the first stroke that begins cutter compensation and changes the way the tool is cleared from the part being cut.

44 MIN F IN RADIUS TNC %

This setting is a numeric entry between 1 and 100. It affects the feed rate when cutter compensation moves the tool towards the inside of a circular cut. In order to maintain a constant surface feed rate, such a cut will be slowed down. This setting specifies the minimum feed rate as a percentage of the programmed feed rate.

45 MIRROR IMAGE X-AXIS

47 MIRROR IMAGE Z-AXIS

These are On/Off settings. When it is off, axes motions occur normally. When it is on, the specific axis motion is mirrored (or reversed) around the work zero point.



49 SKIP SAME TOOL CHANGE

This is an On/Off setting. When it is off, an Tnn will always cause a turret rotation sequence to occur; even if the same tool is selected. When it is on, a tool change to the same tool will cause no action.

50 AUX AXIS SYNC

This changes the synchronization protocol between sender and receiver for the second serial port. 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 it is set to XON/XOFF, those ASCII character codes are used by the receiver to tell the sender to temporarily stop. XON/XOFF is the most common setting. Make sure that the Haas servo control is set to the same condition.

DC CODES is like XON/XOFF but the 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 gives the RS-232 communication added reliability because each block is checked for integrity. Refer to "Data Input/Output" section for more information.

51 DOOR HOLD OVERRIDE

This is an On/Off setting. When it is **off**, a program cannot be started when the doors are open and opening the doors will cause a running program to stop just like in FEED HOLD. When it is **on**, and Parameter 57 bits DOOR STOP SP and SAFETY CIRC are set to zero, the door condition is ignored. When the control is turned on, this setting is set to Off.

52 G83 RETRACT ABOVE R

This is a numeric entry in the range of 0.0 to 9.9999 inches. This setting changes the way G83 (peck drilling) works when it returns to the R plane. Most programmers set the R plane well above the cut to ensure that the chip clear motion actually allows the chips to get out of the hole but this causes a wasted motion when first drilling through this "empty" space. If Setting 52 is set to the distance required to clear chips, the R plane can be put much closer to the part being drilled. When the clear move to R occurs, the Z will be moved above R by this setting.

53 JOG W/O ZERO RETURN

This is an On/Off setting. When it is off, jogging of an axis is inhibited until the zero return operation is completed. When it is ON, jogging of an axis is allowed prior to the zero return. The ON condition can be dangerous in that an axis can be run into the stops, however, the maximum speed allowed is one inch per minute or 0.0010 inches per handle increment. When the control is turned on, this setting is set to OFF.

54 AUX AXIS BAUD RATE

This setting allows the operator to change the serial data rate for the second serial port. This applies to the interface with the optional V axes. The possible values include: 50, 110, 200, 300, 600, 1200, 2400, 4800, 7200, 9600, 19200, 38400. Note that 4800 is standard in Haas servo controls and this should be set to the same value.

55 ENABLE DNC FROM MDI

This is an On/Off setting. When it is off, DNC cannot be selected. When it is turned on, DNC is selected by pressing MDI while already in MDI. The DNC option must be enabled in the control.

56 M30 RESTORE DEFAULT G

This is an On/Off setting. When it is off, no change to the modal G codes occurs at the end of a program (normally M30). When it is on, an M30 will reset all of the modal G codes to their defaults. When this setting is on, RESET will also reset defaults.



57 EXACT STOP CANNED X-Z

This is an On/Off setting. When it is off, the rapid X-Z motion associated with a canned cycle may not get exact stop; according to other conditions. When it is on, the X-Z motion always gets exact stop. This will make canned cycles slower but less likely to run into a close tolerance fixture. The default is OFF.

58 CUTTER COMPENSATION

This setting controls the type of cutter compensation used in the control. The types are similar to the method of cutter compensation available in other classes of controls.

59 PROBE OFFSET X+

60 PROBE OFFSET X-

61 PROBE OFFSET Z+

62 PROBE OFFSET Z-

Settings 59 through 62 are used to define the displacement and size of the tool probe. These numbers only apply to the probing option. These four numbers specify the travel distance in four directions from where the probe is triggered to where the actual sensed surface is located. They are used by G31, G36, G136, and M75. They can be both positive and negative numbers. If the probe width were 0.23 inches in diameter and the probe was set exactly at the center of the spindle, these four settings would all be 0.115 inches.

63 TOOL PROBE WIDTH

This setting is used to specify the width of the probe that is used to test tool offset. This setting only applies to the probing option. It is used by G35.

64 T. OFS MEAS USES WORK

This is an on/off setting. It changes the way the TOOL OFSET MESUR button works. When this is ON, the entered tool offset will be relative to the currently selected work coordinate Z offset. 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 screen. The maximum size is automatically limited to default height. The default shows the machine's entire work area. A specific scale can be set by using the following formula.

Total X travel = Parameter 7 / Parameter 6

Scale = Total X travel / Setting 65

66 GRAPHICS X OFFSET

This setting locates the top of the scaling window relative to the machine X zero position. Its default is zero.

68 GRAPHICS Z OFFSET

This setting locates the right side of the scaling window relative to the machine Z zero position. Its default is zero.

69 DPRNT LEADING SPACES

This setting suppresses leading spaces that are generated by a macro DPRNT format statement. In a DPRNT statement the format specifies the number of characters printed to the serial port for the whole portion of a variable. If the number is smaller than the space allowed for, then leading spaces are sent to the serial port. When this setting is OFF, then no leading spaces are generated.



The following example illustrates control behavior when this setting is OFF or ON.

#1= 3.0 ;	Setting 69: OFF	ON
G0 G90 X#1 ;	OUTPUT: X3.0000	X 3.0000
DPRNT[X#1[44]] ;		

The default value is OFF.

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 default incremental cutting depth for each succeeding pass of the cutting tool 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 default retraction amount of the tool after a roughing cut. It represents the tool to material clearance as the tool is returning over the material to position 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 set to ON, the control will display all blocks that are executed in programs that have an O number of 9000 or above. When the setting is OFF, the control will not display 9000 series blocks. The default setting is ON.

75 9xxxx PROGS SINGLE BLK

When Setting 75 is set to ON and the control is operating in SINGLE BLOCK mode, then the control will stop at each block in a 9000 series program and wait for the operator to press CYCLE START. When Setting 75 is set to OFF, then all blocks in a 9000 series program are executed in a continuous manner 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 subroutines without displaying the blocks contained in that subroutine. If the control is in single block mode, no single block pause will occur within a 9000 series subroutine.

When Setting 75 is ON and Setting 74 is OFF, then 9000 series subroutines will be displayed as they are executed.

76 FOOT PEDAL LOCK OUT

This is an ON/OFF setting. When it is OFF, the foot pedal operates normally. When it is ON, any action at the foot pedal is ignored by the control.



77 SCALE INTEGER F

This setting aids those wishing to run programs developed on a control other than HAAS. It allows the operator to select how the control interprets an F address code that does not contain a decimal point, (It is recommended that the programmer always use a decimal point). The setting can be set to the following values:

DEFAULT	- F12 is interpreted as .0012 units/minute.
INTEGER	- " " " " 12.0 "
.1	- " " " " 1.2 "
.01	- " " " " .12 "
.001	- " " " " .012 "
.0001	- " " " " .0012 "

The default setting is DEFAULT.

81 TOOL AT POWER DOWN

When the POWER UP 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. 1 is the default.

82 LANGUAGE

This setting allows the user to change between available languages. If the language selected does not reside in the control, NOT AVAILABLE will be displayed in the message area when that language is selected.

83 M30/RESETS OVERRIDES

When turned on, an M30 causes feed rate override, rapid override, and spindle override to be reset to default values.

84 TOOL OVERLOAD ACTION

Causes the specified action to occur anytime a tool becomes overloaded (ALARM, FEEDHOLD, BEEP, AUTOFEED). When set to FEEDHOLD, the message "Tool Overload" will be displayed whenever this condition occurs. Pressing any key will clear the message. When set to AUTOFEED, the mill automatically limits the feed rate based on the tool load (see tool load monitor display).

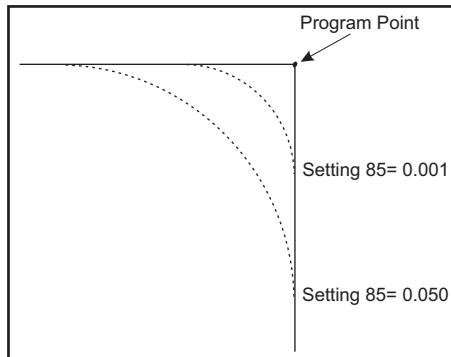
NOTES: When tapping (rigid and floating), the feed and spindle overrides will be locked out, so the AUTOFEED feature will be ineffective (although the display will appear to respond to the override buttons.)

The last commanded feed rate will 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 feed rate override buttons while the AUTOFEED feature is active. As long as tool load limit is not exceeded, these buttons will have the expected effect and the overridden feed rate will be recognized as the new commanded feed rate by the AUTOFEED feature. However, if the tool load limit has already been exceeded, the control will ignore the feed rate override buttons and the commanded feed rate will remain unchanged.

85 MAXIMUM CORNER ROUNDING

Defines the accuracy of corners within a selected tolerance. Initial default value is set to .05 inch. If this setting is zero, the control acts as if exact stop is commanded on each motion block. Parameter 134 is used as a floor so the machine will not slow down to extremely slow speeds. Alternatively, a G187 can be used in the program to alter the effective value of Setting 85 without permanently changing that setting. This method likewise takes advantage of the floor, but does not require that the machine be rebooted.

*Setting 85 Example*

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

When the reset key is pressed and this is on, any overrides are cancelled and set to their programmed values.

90 GRAPH Z ZERO LOCATION

This setting allows the user to adjust 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 allows the user to adjust 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 what direction the chuck is considered to be clamped. The default value is O.D. for the Outside Diameter. With this value, the chuck is considered clamped when the jaws are moved to the spindle center. When the setting is set to I.D., the chuck is considered clamped when the jaws are moved away from the spindle center.

93 TAIL ST. X CLEARANCE

This setting is effective only if the B axis is enabled. It works with Setting 94 to define a travel forbidden zone that limits interaction between the tailstock and the tool turret. This setting determines the X axis travel limit in machine coordinates when the difference between the Z axis location and the B axis location falls below the value in Setting 94. If this condition occurs and a program is running then Alarm 609 is generated. When jogging, no alarm is generated, but travel will be limited. Units are in inches. The suggested default value for this setting is -8.2500 inches.

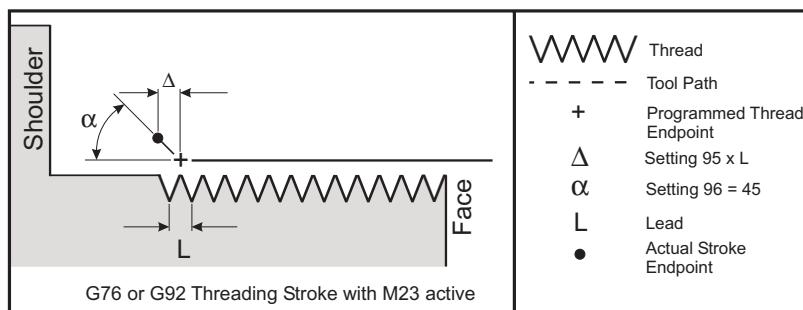


94 Z/TS DIFF @ X CLEARNCE

This setting is effective only if the B axis is enabled. It represents the minimum allowable difference between the Z and B axes at, or below, the tailstock X clearance plane (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 tailstock position in the Z axis negative direction. The default value for this setting is zero.

95 THREAD CHAMFER SIZE

See figure below. Valid range: 0 to 29.999. Default: 1.000. Typical: 0.5 to 1.5 Units: Multiple of current thread lead (F or E). Used in G76 and G92 threading cycles. Command M23 must be active. When command M23 is active, threading strokes are terminated with an angled retraction over a preset distance (as opposed to pulling straight out). This distance is stored in Setting 95. The angle of retraction is stored in Setting 96. Settings 95 and 96 interact with each other and care must be taken to ensure that the tool retracts completely from the material and does not collide with a shoulder or chuck.



96 THREAD CHAMFER ANGLE

See Setting 95.

Valid range: 0 to 89 degrees (No decimal point allowed) Default: 45 Typical: 20 to 70 Units: Degrees

97 TOOL CHANGE DIRECTION

This setting determines the default tool change direction. It may be set to either SHORTEST or M17/M18. When the control is powered on, RESET, or encounters an M30/M02, the control uses this setting to determine the initial tool change direction.

When SHORTEST is selected, the control will move the tool turret along the shortest path for all programmed tool changes. The program can still use M17 and M18 to fix the tool change direction, but once this is done there is no means to revert back to the SHORTEST tool direction other than RESET or M30/M02. This usually is not a problem since most programs will use one method or the other for all tool changes.

When M17/M18 is selected, the control will move the tool turret either always forward or always reverse based on the most recently specified 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.

This setting does not affect the keyboard keys TURRET FWD or TURRET REV. TURRET FWD is always forward and TURRET REV is always reverse. The default for this setting is SHORTEST.

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 increment for 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

This feature that is intended to activate when the machine is idle or unattended to prevent the monitor screen from becoming etched (burned-in) after displaying the same information for many hours. When this setting is zero, the machine behaves normally. If it is set to some number of minutes, then after that amount of time with no key presses, no jog handle motion and no alarms, the Screen Saver will be activated. It will be deactivated by any key press, jog handle motion or alarm. When it is active, the words "SCREEN SAVER, Press CANCEL to exit" will be displayed in random places on an otherwise blank screen and will be changed every two seconds. Any key will cancel the screen saver, however, if the operator was in the middle of operating the machine when the screen saver activated, the CANCEL key will have the least effect on anything he was doing. Note that the screen saver will activate when the number of minutes specified by setting 100 have elapsed except as follows:

- Not while any alarms are present
- Not while a program is running
- Not while the machine is in MDI, Jog or Sleep Mode
- Not while a graphics screen is displayed

101 FEED OVERRIDE -> RAPID

When this setting is OFF, the machine will behave normally. When it is ON and HANDLE CONTROL FEED RATE is pressed, the jog handle will affect both the feed rate override and the rapid rate override simultaneously. That is, changing the feed rate override will cause a proportional change to the rapid rate. The maximum rapid rate will be maintained at 100% or 50% according to setting 10.

102 C AXIS DIAMETER

This setting supports the C axis. The default value is 1.0 inches and the maximum allowable value is 29.999 inches.

Setting 103, CYC START/FH SAME KEY

This is an ON/OFF setting. When it is OFF, the machine operates normally. When it is ON, CYCLE START must be pressed and held to run a program. When CYCLE START is released, a FEED HOLD is generated. This setting cannot be ON when Setting 104 is ON. When one of them is set to ON, the other will automatically turn OFF. This setting can be changed while running a program.

Setting 104, JOG HANDL TO SNGL BLK

This is an ON/OFF setting. When it is OFF, the machine operates normally. When it is ON, and SINGLE BLOCK is selected, the jog handle can be used to single step through a program. Reversing the jog handle direction will generate a FEED HOLD. This can be useful when an unexpected long motion block is encountered. CYCLE START must be used to begin running a program. This setting cannot be ON when Setting 103 is ON. When one of them is set to ON, the other will automatically turn OFF. This setting can be changed while running a program.

105 TS RETRACT DISTANCE

The distance from the HOLD POINT (Setting 107) the tailstock will retract when commanded. This setting should be a positive value. 3.0 is a good starting value.



106 TS ADVANCE DISTANCE

When the tailstock 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. 2.0 is a good starting 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 invoked. 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.

108 QUICK ROTARY G28

This is an ON/OFF setting. When it is ON, the G28 command will return a rotary axis only back to zero when it has been commanded to greater than 360 degrees. For example, if a rotary axis has been commanded 361 degrees, and this setting is on, a G28 command will move it back only one degree instead of "unwinding" it all 361 degrees. Note that other G codes, such as G00 and G01, are unaffected by this setting.

109 WARM-UP TIME IN MIN.

This is the number of minutes (maximum= 300 from the time the machine is powered on) to apply the compensation. If it is set to zero no compensation will be applied.

110 WARMUP X DISTANCE

112 WARMUP Z DISTANCE

Setting 110 and 112 specify the amount of compensation (maximum=+/- .0020" or +/- .050mm) applied to the X, and Z axis respectively. No compensation is applied if the setting is zero.

114 CONVEYOR CYCLE (minutes)

115 CONVEYOR ON-TIME (minutes)

The above two settings control the intermittent chip conveyor function. If setting 114 is zero, the chip conveyor will behave normally. If it is set to some number of minutes, the chip conveyor will automatically turn itself off after the number of minutes specified by setting 115, then turn itself back on later.

Setting 114 controls how often the cycle is to be repeated, that is, if setting 114 is set to 30 and setting 115 is set to 2, the chip conveyor will turn itself on every half hour, run for two minutes, then turn itself off. On-time should be set no greater than 80% of cycle time.

NOTES: 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.

If Parameter 209 CNVY DR OVRD bit 16 is set to 0 and the chip conveyor is cycling, opening the door will cause the conveyor to stop and suspend cycling. When the door is closed, the cycle will resume.

NOTE: If Parameter 209 CNVY DR OVRD is set to 1 and the chip conveyor is cycling, the conveyor will continue to run when the door is open but will stop at the end of the cycle and cancel the cycle. When the door is closed, the cycle will resume.

Under no circumstances will the chip conveyor automatically start running when the door is open.

**118 M99 BUMxPS M30 CNTRS**

When this setting is turned ON, an M99 will increment the M30 counters that are visible by pressing CURNT COMNDS and PAGE DOWN twice. Note that an M99 will only increment the counters in loop mode in a main program, not a sub program. An M99 used as a subprogram return or used with a P value to jump to another part of the program will not increment the M30 counters.

119 OFFSET LOCK

This is an ON / OFF setting. When it is OFF, no special functions occur. When it is ON, the user is prevented from altering any of the offsets. However, programs which alter offsets will still be able to do so.

120 MACRO VAR LOCK

This is an ON / OFF setting. When it is OFF, no special functions occur. When it is ON, the user is prevented from altering any of the macro variables. However, programs which alter macro variables will still be able to do so.

121 FOOT PEDAL TS ALARM

When M21 is used to move the tailstock to the hold point and hold a part, the control will generate alarm 439 TS FOUND NO PART if no part is encountered and the hold point is actually reached. Setting 121 can be switched to ON and the alarm 439 TS FOUND NO PART will be generated when the foot pedal is used to move the tailstock to the hold point and no part is encountered.

122 TS 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

It should be set to ON when the automatic door hardware is installed and the operator wants it to function. Otherwise it should be set to OFF. When it is set to ON, and setting 51 DOOR HOLD OVERRIDE is set to OFF, parameter 57 SAFETY CIRC is set to zero, parameters 235, 236, and 251 are set appropriately, the automatic door feature will function. That is, the door will close when cycle start is pressed and will open when the program has reached an M30 and the spindle has stopped turning. Note that if any of the aforementioned parameters and settings are set incorrectly, the Auto-Door feature will not function.

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, the control assumes that a crash is likely and displays the message CHK TOOL CLEAR instead of doing the requested tool change. If, however, the operator had pressed HANDLE JOG immediately prior to requesting a tool change, the control will assume that he had just jogged the axes to a safe position and will perform the tool change.

Setting 133 NETWORK/ZIP OFF/ON

This is an ON/OFF setting that is used to activate the internal Zip/Enet PC104 board at power-on time. When it is set to OFF, the CNC will not access the board. When it is set to ON, the CNC will access it at power-on time and display the message "LOADING" on the Zip/Enet settings page just below setting 139. After some time (2 minutes maximum,) the control will instead display the message "DISK DONE" indicating that communications have been established with the internal PC104 board and the user can now use the control.



Setting 134 CONNECTION TYPE

This setting can be FLOPPY, NET, or ZIP. When it is set to FLOPPY, program loading and saving will be performed in the usual way via the floppy disk drive installed in the control. When it is set to NET, program loading and saving will be performed via the user-supplied network connection provided that connection was successfully established at power-on time.) When it is set to ZIP, program loading and saving will be performed via the user-supplied ZIP drive (assume such a device is connected.) When setting 133 is set to ON, the value of this setting will appear on the LISTPROG screen as follows: F4 DIR-FLOPPY, F4 DIR-NET, or F4 DIR-ZIP.

Zip/Enet settings page

In previous versions, the method of accessing the Zip/Enet feature was by pressing LISTPROG, entering the word "NET" or "ZIP", and pressing F4. The control would search the directory of the selected device and create program O8999 containing a listing of that devices directory. To return to the floppy disk drive, the process was repeated but using the word "FLOPPY". The following settings have been added to simplify access to the Zip/Enet feature. Note: After changing any of the following text settings, the user must press F1 or cycle the power before the new information will take effect. Refer to the Zip/Enet Options Release Notes for more information.

Setting 135 NETWORK TYPE

This setting can be NONE, NOVELL, NT/IPX, or NT/TCP and specifies the user-supplied network connection type. When it is set to NONE, only a floppy disk or a user-supplied Zip drive are accessible.

Setting 136 SERVER

This setting is used to contain the user-supplied server name (up to 8 characters long.) If this setting is not to be used, the user should enter a semicolon (EOB.)

Setting 137 USERNAME

This setting is used to contain the user-specified account name (up to 8 characters long.) If this setting is not to be used, the user should enter a semicolon (EOB.)

Setting 138 PASSWORD

This setting is used to contain the user-specified password (up to 8 characters long.) If this setting is not to be used, the user should enter a semicolon (EOB.)

Setting 139 PATH

This setting is used to contain the user-specified Novell-Path or NT Root Directory name depending on the network server used (up to 18 characters long.) For a Novell network, this is the user's path name, for example: U:\USERS\JOHNDOE. For a Microsoft network, this is the root directory\desired directory name, for example: USERS\JOHNDOE. If this setting is not to be used, the user should enter a semicolon (EOB.)

Setting 140 TCP ADDR

This setting is used only for TCP networks and contains the static user-specified TCP/IP address in the server domain (up to 15 characters long.) For example: 192.168.1.2. If this setting is not to be used, the user should enter a semicolon (EOB.)

Setting 141 SUBNET

This setting is used only for TCP networks and contains the user-specified subnet mask (up to 15 characters long.) For example: 255.255.255.0. If this setting is not to be used, the user should enter a semicolon (EOB.)



SETTINGS

Setting 144 FEED OVERRIDE->SPINDLE

This feature that is intended to keep the chip load constant when an override is applied. When this setting is OFF, the control behaves normally. When it is ON, any feed rate override that is applied will be applied to the spindle speed also, and the spindle overrides will be disabled.



9. PARAMETERS

Parameters are seldom-modified values that change the operation of the machine. These include servo motor types, gear ratios, speeds, stored stroke limits, lead screw compensations, motor control delays and macro call selections. These are all rarely changed by the user and should be protected from being changed by the parameter lock setting. If you need to change parameters, contact HAAS or your dealer. Parameters are protected from being changed by Setting 7.

The Settings page lists some parameters that the user may need to change during normal operation and these are simply called "Settings". Under normal conditions, the parameter displays should not be modified. A complete list of the parameters is provided here.

The PAGE UP, PAGE DOWN, up and down cursor keys , and the jog handle can be used to scroll through the parameter display screens in the control. The left and right cursor keys are used to scroll through the bits in a single parameter.

PARAMETER LIST

Parameter	1 X SWITCHES	
	Parameter 1 is a collection of single-bit flags used to turn servo related functions on and off. The left and right cursor arrows are used to select the function being changed. All values are 0 or 1 only. The function names are:	
0	REV ENCODER	Used to reverse the direction of encoder data.
1	REV POWER	Used to reverse direction of power to motor.
2	REV PHASING	Used to reverse motor phasing.
3	DISABLED	Used to disable the X-axis.
4	Z CH ONLY	With A only, indicates that no home switch.
5	AIR BRAKE	With A only, indicates that air brake is used.
6	DISABLE Z T	Disables encoder Z test (for testing only).
7	SERVO HIST	Graph of servo error (for diagnostics only).
8	INV HOME SW	Inverted home switch (N.C. switch).
9	INV Z CH	Inverted Z channel (normally high).
10	CIRC. WRAP.	With A only, causes 360 wrap to return to 0. Note for parameter 498 bit 10: When the bit is set to 1, the lathe will automatically unwind the C-axis no more than half a rotation. When the bit is set to zero, it behaves as if the C axis had been rotated many times then disengaged, when it is engaged again, the control will zero it by unwinding as many times as it had been wound.
11	NO I IN BRAK	With A only, removes I feedback when brake is active.
12	LOW PASS +1X	Adds 1 term to low pass filter.
13	LOW PASS +2X	Adds two terms to low pass filter.



14	OVER TEMP NC	Selects a normally closed overheat sensor in motor.
15	CABLE TEST	Enables test of encoder signals and cabling.
16	Z TEST HIST	History plot of Z channel test data.
17	SCALE FACT/X	If set to 1, the scale ratio is interpreted as divided by X; where X depends on bits SCALE/X LO and SCALE/X HI.
18	INVIS AXIS	Used to create an invisible axis.
19	DIAMETER PRG	Used to set diameter programming. When set to 1, it will interpret inputs as diameters instead of radii.
20	TRAVL LIMITS	Travel limits are used.
21	NO LIMSW ALM	Alarms are not generated at the limit switches.
22	D FILTER X8	Enables the 8 tap FIR filter. Used to eliminate high frequency vibrations, depending on the axis motor.
23	D FILTER X4	Enables the 4 tap FIR filter. Used to eliminate high frequency vibrations, depending on the axis motor.
24	TORQUE ONLY	For HAAS only.
25	3 EREV/MREV	For HAAS only.
26	2 EREV/MREV	For HAAS only.
27	NON MUX PHAS	For HAAS only.
28	BRUSH MOTOR	Enables the brush motor option.
29	ROTARY AXIS	When set to 1, the axis is treated as a rotary axis. Position will be displayed in degrees, and inputs will be interpreted as angles.
30	SCALE/X LO	With SCALE/X HI bit, determines the scale factor used in bit SCALE FACT/X,
31	SCALE/X HI	With SCALE/X LO bit, determines the scale factor used in bit SCALE FACT/X. See below

HI	LO	
0	0	3
0	1	5
1	0	7
1	1	9

Parameter 2 X P GAIN
Proportional gain in servo loop.

Parameter 3 X D GAIN
Derivative gain in servo loop.



Parameter	4 X I GAIN	Integral gain in servo loop.
Parameter	5 X RATIO (STEPS/UNIT)	The number of steps of the encoder per unit of travel. Encoder steps supply four (4) times their line count per revolution. Thus, an 8192 line encoder and 6mm pitch screw give: $8192 \times 4 \times 25.4 / 6 = 138718$
Parameter	6 X MAX TRAVEL (STEPS)	Max negative direction of travel from machine zero in encoder steps. Does not apply to A-axis. Thus, a 20 inch travel, 8192 line encoder and 6 mm pitch screw give: $20.0 \times 138718 = 2774360$
Parameter	7 X ACCELERATION	Maximum acceleration of axis in steps per second per second.
Parameter	8 X MAX SPEED	Max speed for this axis in steps per second.
Parameter	9 X MAX ERROR	Max error allowed in servo loop before alarm is generated. Units are encoder steps.
Parameter	10 X FUSE LEVEL	Used to limit average power to motor. If not set correctly, this parameter can cause an "overload" alarm.
Parameter	11 X BACK EMF	Back EMF of motor in volts per 1000 RPM times 10. Thus a 63 volt/KRPM motor gives 630.
Parameter	12 X STEPS/REVOLUTION	Encoder steps per revolution of motor. Thus, an 8192 line encoder gives: $8192 \times 4 = 32768$
Parameter	13 X BACKLASH	Backlash correction in encoder steps.
Parameter	14 X DEAD ZONE	Dead zone correction for driver electronics. Units are 0.0000001 seconds.
Parameter	15 Y SWITCHES	See Parameter 1 for description.
Parameter	16 Y P GAIN	See Parameter 2 for description.
Parameter	17 Y D GAIN	See Parameter 3 for description.



- Parameter 18 Y I GAIN
See Parameter 4 for description.
- Parameter 19 Y RATIO(STEPS/UNIT)
See Parameter 5 for description.
- Parameter 20 Y MAX TRAVEL(STEPS)
See Parameter 6 for description.
- Parameter 21 Y ACCELERATION
See Parameter 7 for description.
- Parameter 22 Y MAX SPEED
See Parameter 8 for description.
- Parameter 23 Y MAX ERROR
See Parameter 9 for description.
- Parameter 24 Y FUSE LEVEL
See Parameter 10 for description.
- Parameter 25 Y BACK EMF
See Parameter 11 for description.
- Parameter 26 Y STEPS/REVOLUTION
See Parameter 12 for description.
- Parameter 27 Y BACKLASH
See Parameter 13 for description.
- Parameter 28 Y DEAD ZONE
See Parameter 14 for description.
- Parameter 29 Z SWITCHES
See Parameter 1 for description.
- Parameter 30 Z P GAIN
See Parameter 2 for description.
- Parameter 31 Z D GAIN
See Parameter 3 for description.
- Parameter 32 Z I GAIN
See Parameter 4 for description.
- Parameter 33 Z RATIO(STEPS/UNIT)
See Parameter 5 for description.



- Parameter 34 Z MAX TRAVEL (STEPS)
See Parameter 6 for description.
- Parameter 35 Z ACCELERATION
See Parameter 7 for description.
- Parameter 36 Z MAX SPEED
See Parameter 8 for description.
- Parameter 37 Z MAX ERROR
See Parameter 9 for description.
- Parameter 38 Z FUSE LEVEL
See Parameter 10 for description.
- Parameter 39 Z BACK EMF
See Parameter 11 for description.
- Parameter 40 Z STEPS/REVOLUTION
See Parameter 12 for description.
- Parameter 41 Z BACKLASH
See Parameter 13 for description.
- Parameter 42 Z DEAD ZONE
See Parameter 14 for description.
- Parameter 43 A SWITCHES
See Parameter 1 for description.
- Parameter 44 TURRET P GAIN
See Parameter 2 for description.
- Parameter 45 TURRET D GAIN
See Parameter 3 for description.
- Parameter 46 TURRET I GAIN
See Parameter 4 for description.
- Parameter 47 TURRET RATIO (STEPS/UNIT)
See Parameter 5 for description.
- Parameter 48 TURRET MAX TRAVEL (STEPS)
See Parameter 6 for description.



Parameter 49 TURRET ACCELERATION
See Parameter 7 for description.

Parameter 50 TURRET MAX SPEED
See Parameter 8 for description.

Parameter 51 TURRET MAX ERROR
See Parameter 9 for description.

Parameter 52 TURRET FUSE LEVEL
See Parameter 10 for description.

Parameter 53 TURRET BACK EMF
See Parameter 11 for description.

Parameter 54 TURRET STEPS/REVOLUTION
See Parameter 12 for description

Parameter 55 TURRET BACKLASH
See Parameter 13 for description.

Parameter 56 TURRET DEAD ZONE
See Parameter 14 for description.

Parameters 57 through 128 are used to control other machine dependent functions. They are:

Parameter 57 COMMON SWITCH 1

Parameter 57 is a collection of general purpose single bit flags used to turn some functions on and off. The left and right cursor arrows are used to select the function being changed. All values are 0 or 1 only. The function names are:

- | | | |
|----|---------------|--|
| 0 | REV CRANK | Reverses direction of jog handle. |
| 1 | DISABLE T.C. | Disables tool changer operations. |
| 2 | DISABLE G.B. | Disables gear box functions. |
| 3 | POF AT E-STOP | Stops spindle then turns the power off at EMERGENCY STOP. |
| 4 | RIGID TAP | Indicates hardware option for rigid tap. |
| 5 | REV SPIN ENC | Reverses sense direction of spindle encoder. |
| 6 | SYNC THREADS | Threads will repeat between passes. |
| 7 | EX ST MD CHG | Selects exact stop in moves when mode changes. |
| 8 | SAFETY CIRC | This enables safety hardware, if machine is so equipped. |
| 9 | SP DR LIN AC | Selects linear deceleration for rigid tapping. 0 is quadratic. |
| 10 | PH LOSS DET | When enabled, will detect a phase loss. |



12	OVER T IS NC	Selects Regen over temp sensor as N.C.
13	SKIP OVERSHT	Causes Skip (G31) to act like Fanuc and overshoot sense point.
14	NONINV SP ST	Non-inverted spindle stopped status.
15	SP LOAD MONI	Spindle load monitor option is enabled.
16	SP TEMP MONI	Spindle temperature monitor option is enabled.
18	ENABLE DNC	Enables DNC selection from MDI.
19	ENABLE BGEDT	Enables BACKGROUND EDIT mode.
20	ENA GRND FLT	Enables ground fault detector.
21	M19 SPND ORT	This bit makes the P and R codes a protected feature which can only be enabled with an unlock code. The unlock code will be printed on the parameter listing of all new machines. If this bit is set to 0, an M19 will orient the spindle to 0 degrees regardless of the value of any P or R code in the same block. If this is set to 1, a P code in the block will cause the spindle to be oriented to the specified angle such as P180. Alternately, a decimal R code can be used, such as R180.53. Note that the P and R codes only work on a vector drive machine.
22	ENABLE MACRO	Enables macro functions.
23	INVERT SKIP	Invert sense of skip to active low=closed.
24	HANDLE CURSR	Enable use of jog handle to move cursor.
25	NEG WORK OFS	Selects use of work offsets in negative direction.
27	ENA QUIKCODE	Enables conversational programming.
28	OILER ON/OFF	Enables oiler power when servos or spindle is in motion.
29	NC OVER VOLT	Inverts sense of over voltage signal.
30	VEC DRV ENC	Second spindle encoder
31	DOOR STOP SP	Enables functions to stop spindle and manual operations at door switch.
Parameter	58	LEAD COMPENS SHIFT Shift factor when applying lead screw compensation. Lead screw compensation is based on a table of 256 offsets; each +/-127 encoder steps. A single entry in the table applies over a distance equal to two raised to this parameter power encoder steps.
Parameter	59	MAX FEED RATE (INCH) Maximum feed rate in inches per minute.
Parameter	60	TURRET IN POS DELAY Amount of time to delay after the turret rotates to the tool position. This delay allows the turret to settle.



Parameter	61 TURRET LOCK DELAY	Amount of time to delay after the turret is sensed to be locked. This delay allows for mechanical settling.
Parameter	62 TURRET UNLOCK ERROR TIME	Maximum delay allowed for tool turret to unlock. Units are milliseconds. After this time, an alarm is generated.
Parameter	63 TURRET LOCK ERRTIME	Maximum delay allowed for tool turret to lock. Units are milliseconds. After this time, an alarm is generated.
Parameter	64 Z TOOL CHANGE OFFSET	For turret, displacement from home switch to tool 0.
Parameter	65 NUMBER OF TOOLS	Number of tool positions in tool changer. This number must be set to the lathe's configuration.
Parameter	66 SPINDLE ORI DELAY	Maximum delay allowed when orienting spindle. Units are in milliseconds. After this time, an alarm is generated.
Parameter	67 GEAR CHANGE DELAY	Maximum delay allowed when changing gears. Units are milliseconds. After this time, an alarm is generated.
Parameter	68 DRAWBAR MAX DELAY	Maximum delay allowed when clamping and unclamping tool. Units are milliseconds. After this time, an alarm is generated.
Parameter	69 AIR BRAKE DELAY	Delay provided for air to release from brake prior to moving. Units are milliseconds.
Parameter	70 MIN SPIN DELAY TIME	Minimum delay time in program after commanding new spindle speed and before proceeding. Units are milliseconds.
Parameter	71 SPIN STALL DET DLAY	Time to delay after spindle is started before spindle stall checking is started. Each unit represents 1/50 of a second.
Parameter	72 LIVE TOOL CHNG DLAY	This parameter specifies the amount of time (in milli seconds) to wait after commanding the Live Tooling Drive motor to turn at the velocity specified by parameter 143. This process is required to engage the live tooling motor and tool and is only performed prior to the first M133 or M134 after a tool change.



Parameter	73 SP HIGH G/MIN SPEED	Command speed used to rotate spindle motor when orienting spindle in high gear. Units are maximum spindle RPM divided by 4096.
Parameter	74 SP LOW G/MIN SPEED	Command speed used to rotate spindle motor when orienting spindle in low gear. Units are maximum spindle RPM divided by 4096.
Parameter	75 GEAR CHANGE SPEED	Command speed used to rotate spindle motor when changing gears. Units are maximum spindle RPM divided by 4096.
Parameter	76 LOW AIR DELAY	Delay allowed after sensing low air pressure before alarm is generated. Alarm skipped if air pressure returns before delay. Units are 1/50 seconds.
Parameter	77 SP LOCK SETTLE TIME	Required time in milliseconds that the spindle lock must be in place and stable before spindle orientation is considered complete.
Parameter	78 GEAR CH REV TIME	Time in milliseconds before motor direction is reversed while in a gear change.
Parameter	79 SPINDLE STEPS/REV	Sets the number of encoder steps per revolution of the spindle. Applies only to hard tapping option.
Parameter	80 MAX SPIN DELAY TIME	The maximum delay time control will wait for spindle to get to commanded speed or to get to zero speed. Units are milliseconds.
Parameter	81 M MACRO CALL O9000	<p>M code that will call O9000. This parameter can contain a value from 1 through 98, inclusive, zero causes no call. However it is best to use a value that is not already in use (see current M code list). Using M37 the value 37 would be entered in parameter 81 (for example). A program would be written to include the M37, such as:</p> <pre> G X0... M37 . . . M30 </pre> <p>The control would run the program until it got to the M37, It would call program O9000, run that, and then return to the point that it left, and continue the main program.</p> <p>Be aware that, if program O9000 contains another M37, it will call itself, and keep calling until it fills the stack (9 times) and then alarm out with 307 SUBROUTINE NESTING TOO DEEP.</p> <p>Note that if M33 (for example) is used, it would override the normal M33 Conveyor Stop function.</p>



Parameter	82	M MACRO CALL O9001 Same as 81.
Parameter	83	M MACRO CALL O9002 Same as 81.
Parameter	84	M MACRO CALL O9003 Same as 81.
Parameter	85	M MACRO CALL O9004 Same as 81.
Parameter	86	M MACRO CALL O9005 Same as 81.
Parameter	87	M MACRO CALL O9006 Same as 81.
Parameter	88	M MACRO CALL O9007 Same as 81.
Parameter	89	M MACRO CALL O9008 Same as 81.
Parameter	90	M MACRO CALL O9009 Same as 81.
Parameter	91	G MACRO CALL O9010 G code that will call O9010. This parameter can contain a value from 1 through 98, inclusive, zero causes no call. However it is best to use a value that is not already in use (see current G code list). Using G45 the value 45 would be entered in parameter 91 (for example). A program would be written to include the G45, such as: G X0... G45 . M30 The control would run the program until it got to the G45, It would call program O9010, run that, and then return to the point that it left, and continue the main program. Be aware that, if program O9010 contains another G45, it will call itself, and keep calling until it fills the stack (4 times) and then alarm out with 531 MACRO NESTING TOO DEEP. Note that if G84 (for example) is used, it would override the normal G84 Tapping Canned Cycle.
Parameter	92	G MACRO CALL O9011 Same as 91.
Parameter	93	G MACRO CALL O9012 Same as 91.



Parameter	94	G MACRO CALL O9013 Same as 91.
Parameter	95	G MACRO CALL O9014 Same as 91.
Parameter	96	G MACRO CALL O9015 Same as 91.
Parameter	97	G MACRO CALL O9016 Same as 91.
Parameter	98	G MACRO CALL O9017 Same as 91.
Parameter	99	G MACRO CALL O9018 Same as 91.
Parameter	100	G MACRO CALL O9019 Same as 91.
Parameter	101	IN POSITION LIMIT X How close motor must be to endpoint before any move is considered complete when not in exact stop (G09 or G61). Units are encoder steps.
Parameter	102	IN POSITION LIMIT Y Same definition as Parameter 101.
Parameter	103	IN POSITION LIMIT Z Same definition as Parameter 101.
Parameter	104	IN POSITION LIMIT A Same definition as Parameter 101.
Parameter	105	X MAX CURRENT Fuse level in % of max power to motor. Applies only when motor is stopped.
Parameter	106	Y MAX CURRENT Same definition as Parameter 105.
Parameter	107	Z MAX CURRENT Same definition as Parameter 105.
Parameter	108	A MAX CURRENT Same definition as Parameter 105.



Parameter	109 D*D GAIN FOR X	Second derivative gain in servo loop.
Parameter	110 D*D GAIN FOR Y	Second derivative gain in servo loop.
Parameter	111 D*D GAIN FOR Z	Second derivative gain in servo loop.
Parameter	112 D*D GAIN FOR A	Second derivative gain in servo loop.
Parameter	113 X ACC/DEC T CONST	Exponential acceleration time constant. Units are 1/10000 seconds. This parameter provides for a constant ratio between profiling lag and servo velocity. It is also the ratio between velocity and acceleration.
Parameter	114 Y ACC/DEC T CONST	Same definition as Parameter 113.
Parameter	115 Z ACC/DEC T CONST	Same definition as Parameter 113.
Parameter	116 A ACC/DEC T CONST	Same definition as Parameter 113.
Parameter	117 LUB CYCLE TIME	If this is set nonzero, it is the cycle time for the lube pump and the lube pressure switch option is checked for cycling in this time. It is in units of 1/50 seconds.
Parameter	118 SPINDLE REV TIME	Time in milliseconds to reverse spindle motor.
Parameter	119 SPINDLE DECEL DELAY	Time in milliseconds to decelerate spindle motor.
Parameter	120 SPINDLE ACC/DECEL	Accel/decel time constant in 200ths of a step/ms/ms for spindle motor.
Parameter	121 X PHASE OFFSET	The motor phase offset for X motor. This is arbitrary units.
Parameter	122 Y PHASE OFFSET	See Parameter 121 for description.
Parameter	123 Z PHASE OFFSET	See Parameter 121 for description.



- Parameter 124 A PHASE OFFSET
See Parameter 121 for description.
- Parameter 125 X GRID OFFSET
This parameter shifts the effective position of the encoder Z pulse. It can correct for a positioning error of the motor or home switch.
- Parameter 126 Y GRID OFFSET
See Parameter 125 for description.
- Parameter 127 Z GRID OFFSET
See Parameter 125 for description.
- Parameter 128 A GRID OFFSET
See Parameter 125 for description.
- Parameter 129 GEAR CH SETTLE TIME
Gear change settle time. This is the number of one millisecond samples that the gear status must be stable before considered in gear.
- Parameter 130 GEAR STROKE DELAY
This parameter controls the delay time to the gear change solenoids when performing a gear change.
- Parameter 131 MAX SPINDLE RPM
This is the maximum RPM available to the spindle. When this speed is programmed, the D-to-A output will be +10V and the spindle drive must be calibrated to provide this.
- Parameter 132 Y SCREW COMP. COEF.
This parameter is used to hold the thermal compensation coefficient. This is the coefficient of heating of the lead screw. This parameter should be set to zero.
- Parameter 133 Z SCREW COMP. COEF.
This parameter is used to hold the thermal compensation coefficient. This is the coefficient of heating of the lead screw. The value entered for this parameter is always negative as it is used to shorten the screw length. It should be set to -6000000.
- Parameter 134 X EXACT STOP DIST.
- Parameter 135 Y EXACT STOP DIST.
- Parameter 136 Z EXACT STOP DIST.
- Parameter 137 A EXACT STOP DIST.
These parameters control how close each axis must be to its end point when exact stop is programmed. They apply only in G09 and G64. They are in units of encoder steps. A value of 34 would give $34/138718 = 0.00025$ inch.



NOTE: To change the values of parameters 134-137 permanently the machine must be rebooted.

Parameter	138 X	FRICTION COMPENSATION
Parameter	139 Y	FRICTION COMPENSATION
Parameter	140 Z	FRICTION COMPENSATION
Parameter	141 A	FRICTION COMPENSATION
		These parameters compensate for friction on each of the four axes. The units are in 0.004V.
Parameter	142	HIGH/LOW GEAR CHANG
		This parameter sets the spindle speed at which an automatic gear change is performed. Below this parameter, low gear is the default; above this, high gear is the default.
Parameter	143	LIVE TOOL CHNG VEL
		This parameter specifies the velocity to command the Live Tooling Drive motor for the period specified by parameter 72. This process is required to engage the live tooling motor and tool, and is only performed prior to the first M133 or M134 after a tool change.
Parameter	144	RIG TAP FINISH DIST
		This parameter sets the finish tolerance for determining the end point of a hard tapping operation. Units are encoder counts.
Parameter	145 X	ACCEL FEED FORWARD
		This parameter sets the feed forward gain for the X-axis servo. It has no units.
Parameter	146 Y	ACCEL FEED FORWARD
		Same as Parameter 145.
Parameter	147 Z	ACCEL FEED FORWARD
		Same as Parameter 145.
Parameter	148 A	ACCEL FEED FORWARD
		Same as Parameter 145.
Parameter	150	MAX SP RPM LOW GEAR
		Maximum spindle RPM in low gear.
Parameter	151 B	SWITCHES
		See Parameter 1 for description.
Parameter	152 B	P GAIN
		See Parameter 2 for description.
Parameter	153 B	D GAIN
		See Parameter 3 for description.



- Parameter 154 B I GAIN
See Parameter 4 for description.
- Parameter 155 B RATIO (STEPS/UNIT)
See Parameter 5 for description.
- Parameter 156 B MAX TRAVEL (STEPS)
See Parameter 6 for description.
- Parameter 157 B ACCELERATION
See Parameter 7 for description.
- Parameter 158 B MAX SPEED
See Parameter 8 for description.
- Parameter 159 B MAX ERROR
See Parameter 9 for description.
- Parameter 160 B FUSE LEVEL
See Parameter 10 for description.
- Parameter 161 B BACK EMF
See Parameter 11 for description.
- Parameter 162 B STEPS/REVOLUTION
See Parameter 12 for description.
- Parameter 163 B BACKLASH
See Parameter 13 for description.
- Parameter 164 B DEAD ZONE
See Parameter 14 for description.
- Parameter 165 IN POSITION LIMIT B
See Parameter 101 for description.
- Parameter 166 B MAX CURRENT
See Parameter 105 for description.
- Parameter 167 B D*D GAIN
See Parameter 109 for description.
- Parameter 168 B ACC/DEC T CONST
See Parameter 113 for description.
- Parameter 169 B PHASE OFFSET
See Parameter 121 for description.



- Parameter 170 B GRID OFFSET
See Parameter 125 for description.
- Parameter 171 B EXACT STOP DIST.
See Parameter 134 for description.
- Parameter 172 B FRICTION COMPENSATION
See Parameter 138 for description.
- Parameter 173 B ACCEL FEED FORWARD
See Parameter 145 for description.
- Parameter 174 B SCREW COMP. COEF.
This parameter is used to hold the thermal compensation coefficient. This is the coefficient of heating of the lead screw. This parameter should be set to zero.
- Parameter 175 B AIR BRAKE DELAY
See Parameter 69 for description.
- Parameter 176 Sp SWITCHES
See Parameter 1 for description.
- Parameter 177 C P GAIN
See Parameter 2 for description.
- Parameter 178 C D GAIN
See Parameter 3 for description.
- Parameter 179 C I GAIN
This parameter is used when a Vector Drive is installed, see Parameter 4 for description. If Vector Drive is not installed this parameter is not used
- Parameter 180 SLIP GAIN
This name is used when a Vector Drive is installed. The slip rate calculated depends on two other variables: speed and current.

Slip rate = slip gain x (speed/max speed) x (current/max current)

The slip gain value is the value that slip rate would assume at maximum speed, and maximum current (16.384=1 Hz). If a Vector Drive is not installed, this parameter is called: C AXIS RATIO (STEPS/UNIT) and is not used.
- Parameter 181 MIN SLIP
This name is used when a Vector Drive is installed. The minimum value allowed from the slip rate. From the equation:

Slip rate = slip gain x (speed/max speed) x (current/max current)



it can be seen that at a zero speed, the slip rate would become zero. Therefore a minimum value for slip rate is required. (16.384 =1Hz). If a Vector Drive is not installed, this parameter is called: C AXIS MAX TRAVEL (STEPS) and is not used.

- | | |
|-----------|--|
| Parameter | 182 C ACCELERATION
This name is used when a Vector Drive is installed. See Parameter 7 for description. If a Vector Drive is not installed this parameter is not used. |
| Parameter | 183 C MAX SPEED
This name is used when a Vector Drive is installed. See Parameter 8 for description. If a Vector Drive is not installed this parameter is not used. |
| Parameter | 184 C MAX ERROR
See Parameter 9 for description. |
| Parameter | 185 C FUSE LEVEL
See Parameter 10 for description. |
| Parameter | 186 C BACK EMF
This name is used when a Vector Drive is installed. See Parameter 11 for description. If a Vector Drive is not installed this parameter is not used. |
| Parameter | 187 C HIGH GEAR STEPS/REV
This name is used when a Vector Drive is installed. The number of encoder steps per revolution of the motor when the transmission is in high gear. If the machine does not have a transmission, this is simply the number of encoder steps per revolution of the motor. If a Vector Drive is not installed this parameter is not used. |
| Parameter | 188 C ORIENT GAIN
This name is used when a Vector Drive is installed. The proportional gain is used in the position control loop when performing a spindle orientation. If a Vector Drive is not installed this parameter is called, C axis BACKLASH, and is not used. |
| Parameter | 189 C BASE FREQ
This name is used when a Vector Drive is installed. This is the rated frequency of the motor. If a Vector Drive is not installed this parameter is called, C axis DEAD ZONE, and is not used. |
| Parameter | 190 C HI SP CURR LIM
This name is used when a Vector Drive is installed. At speeds higher than the base frequency, the maximum current that is applied to the motor must be reduced. This is done linearly from base to maximum frequency. The value set in this parameter is the maximum current at the maximum frequency. If a Vector Drive is not installed this parameter is called, C axis IN POSITION LIMIT, and is not used. |
| Parameter | 191 C MAX CURRENT
See Parameter 105 for description. |



- Parameter 192 C MAG CURRENT
This name is used when a Vector Drive is installed. This is the magnetization component of the current in the motor, also called the flux or the field current. If a Vector Drive is not installed this parameter is called, C axis D*D GAIN, and is not used.
- Parameter 193 C SPIN ORIENT MARGIN
This name is used when a Vector Drive is installed. When a spindle orientation is done, if the actual position of the spindle is within this value (plus or minus), the spindle will be considered locked. Otherwise, the spindle will not be locked. If a Vector Drive is not installed this parameter is called, C axis ACC / DEC T CONST, and is not used.
- Parameter 194 C SP STOP SPEED
This name is used when a Vector Drive is installed. The spindle is considered to be stopped (discrete input SP ST*=0) when the speed drops below this value. Units are encoder steps/millisecond. If a Vector Drive is not installed this parameter is called, C axis PHASE OFFSET, and is not used.
- Parameter 195 C START / STOP DELAY
This name is used when a Vector Drive is installed. This delay is used at the start of motion to magnetize the rotor before acceleration starts. Also when the motor comes to a stop, it remains energized for this amount of time. Units are milliseconds. If a Vector Drive is not installed this parameter is called, C axis GRID OFFSET, and is not used.
- Parameter 196 ACCEL LIMIT LOAD
This name is used when a Vector Drive is installed. This is the percent of load limit during acceleration. If the load reaches this limit during acceleration, the control slows the acceleration. If a Vector Drive is not installed this parameter is called, C axis EXACT STOP DIST, and is not used.
- Parameter 197 SWITCH FREQUENCY
This name is used when a Vector Drive is installed. This is the frequency at which the spindle motor windings are switched. Note that there is a hysteresis band around this point, defined by parameter 198. If a Vector Drive is not installed this parameter is called, C axis FRICTION FACTOR, and is not used.
- Parameter 198 SWITCH HYSTERESIS
This name is used when a Vector Drive is installed. This defines the \pm hysteresis band around parameter 197. For example if par. 197 is 85Hz, and par. 198 is 5Hz, switching will take place at 90Hz when the spindle is speeding up, and at 80Hz when the spindle is slowing down. If a Vector Drive is not installed this parameter is called, C axis FEED FORWARD, and is not used.
- Parameter 199 PRE-SWITCH DELAY
This name is used when a Vector Drive is installed. This is the amount of time allowed for the current in the motor to drop before the winding change contactors are switched. Units are in microseconds. If a Vector Drive is not installed this parameter is called, C axis THERMAL COMP. COEF., and is not used.



Parameter	200	POST SWITCH DELAY
		This name is used when a Vector Drive is installed. This is the amount of time allowed for the contactors to stabilize after a switch is commanded, before current is applied to the motor. Units are in microseconds. If a Vector Drive is not installed this parameter is called, C axis AIR BRAKE DELAY, and is not used.
Parameter	201	X SCREW COMP. COEF.
		This parameter is used to hold the thermal compensation coefficient. This is the coefficient of heating of the lead screw. The value entered for this parameter is always negative as it is used to shorten the screw length. It should be set to -12000000.
Parameter	205	A SCREW COMP. COEF.
		This parameter is used to hold the thermal compensation coefficient. This is the coefficient of heating of the lead screw. This parameter should be set to zero.
Parameter	206	Reserved
Parameter	207	Reserved
Parameter	208	SPIN. FAN OFF DELAY
		Delay for turning the spindle fan off after the spindle has been turned off.
Parameter	209	COMMON SWITCH 2
		This is a collection of general purpose single bit flags used to turn some functions on and off. The left and right cursor arrows are used to select the function being changed. All values are 0 or 1 only. The function names are:
	0	LATHE T.C. Designates control as a lathe.
	1	RST STOPS T.C. Tool changer can be stopped with RESET button.
	2	BRIDGE Not Used
	3	ENA CONVEYOR Enables chip conveyor, if machine is so equipped.
	4	50% RPD KBD When (1) the control will support the new style keyboards with the 50% rapid traverse key. For controls without a 50% rapid keypad set this bit to (0).
	5	FRONT DOOR When enabled the control will look for an additional door switch and will generate an operator message.
	10	T SUBROUTINE Not Used
	11	RESERVED
	12	REV CONVEYOR Reverses the direction of the chip conveyor.
	13	M27-M28 CONVYR Usually the chip conveyor motor and direction relays are attached to the user relays M21 M22. When this bit is set, the control expects to see the conveyor hooked up to M27 and M28.



15 GREEN BEACON	When (1) user relay M25 is used to flash a beacon. If the control is in a reset state, the beacon will be off. If the control is running normally, the beacon will be steadily on. If the control is in a M00, M01, M02, M30 feedhold, or single block state, then the beacon will flash.
16 RED BEACON	When (1) user relay M26 is used to flash a beacon. The beacon flashes if the control is experiencing an alarm or emergency stop condition.
17 CONVY DR OVRD	When (1) the conveyor will continue to run with the door open. When (0) the conveyor will stop when the door is open, but will resume when the door is closed. For safety it is recommended that the bit be set to (0).
18 RESERVED	
19 TC FWD CW	Determines the direction that the turret moves as viewed from the spindle, when the turret is commanded forward. When (1), the turret will rotate clockwise for a forward command, and when (0), it will rotate counterclockwise. The default is 1.
21 DISK ENABL	Enables an installed floppy disk drive.
23 MCD RLY BRD	If set to 1, adds 16 additional relays, for a total of 56.
24 HPC ENABLE	When this parameter bit is set to zero the machine will behave normally. When it is set to 1, the High Pressure Coolant pump can be turned on with M88 (this will first turn off the regular coolant if it was on, just like an M9). High Pressure Coolant can be turned off with M89. Note also that if a tool change is commanded when the HPC pump is running, it will be turned off, followed by a pause of the length specified by parameter 237. HPC must then be turned back on by the user's program.
25 AUX JOG NACC	Does not allow accumulation on auxiliary axis jog. If the jog handle is moved rapidly the auxiliary axis will not develop extremely large lags.
27 RAPID EXSTOP	Default is 1. When this bit is set to 1, the control will execute an exact stop after all rapid motions, regardless of the next motion. When set to zero, the control will exact stop after a rapid only if the next motion is not a rapid move.
29 HYDRAULICS	This bit must be set to 1 if a lathe has the hydraulic chuck clamping option.
30 STALL DETECT	Enables detection of spindle stall. If spindle stalls, the spindle motor is stopped and an alarm is generated.
31 SPNDL NOWAIT	When (1), the machine will not wait for the spindle to come up to speed immediately after an M03 or M04 command. Instead, it will check and/or wait for the spindle to come up to speed immediately before the next interpolated motion is initiated. This bit does not affect rigid tapping.



Parameter	214 D:Y CURRENT RATIO%	This name is used when a Vector Drive is installed. This defines the ratio between the two winding configurations. This default winding is Y, and the parameters are set for the Y winding. This number is used to adjust the parameters for the delta winding when the windings are switched. If a Vector Drive is not installed, this parameter is called C axis TOOL CHANGE OFFSET, and is not used.
Parameter	215 CAROUSEL OFFSET	Parameter used to align tool 1 of tool changing carousel precisely. Units are encoder steps.
Parameter	216 CNVYR RELAY DELAY	Delay time in 1/50 seconds required on conveyor relays before another action can be commanded. Default is 5.
Parameter	217 CNVYR IGNORE OC TIM	Amount of time in 1/50 seconds before overcurrent is checked after conveyor motor is turned on. Default is 50.
Parameter	218 CONVYR RETRY REV TIM	Amount of time that the conveyor is reversed in 1/50 seconds after overcurrent is sensed. Default is 200.
Parameter	219 CONVYR RETRY LIMIT	Number of times that the conveyor will cycle through the reverse/forward sequencing when an overcurrent is sensed before the conveyor will shut down. An overcurrent is sensed when chips jam the conveyor. By reversing and then forwarding the conveyor, the chip jam may be broken. Default is 3.
Parameter	220 CONVYR RETRY TIMEOUT	Amount of time in 1/50 seconds between consecutive overcurrents in which the overcurrents is considered another retry. If this amount of time passes between overcurrents then the retry count is set to (0). Default is 1500, 30 seconds.
Parameter	221 MAX TIME NO DISPLAY	The maximum time (in 1/50 sec.) between screen updates. When executing short blocks at a high feed rate, the control will use the resources available for interpreting G-code and generation of motion blocks. The display may not update until this time is exceeded. For high speed operation, updating of the display may cause the motion queue to become exhausted. This will manifest itself as a pause in motion. See M76 and M77 to disable the display completely.
Parameter	222 LOW HYD. IGNORE	The amount of time that the control ignores the LO HYD input bit after servos have been engaged. The hydraulic unit requires a short period of time to come up to pressure. The default value is 50, which is equal to 1 second.
Parameter	226 EDITOR CLIPBOARD	This parameter assigns a program number (nnnnn) to the contents of the clipboard (for the advanced editor).



Parameter	227 DISK DIR NAME	When the floppy disk drive is enabled and a floppy disk directory is read. The directory listing is placed into a program as comments. The program is then made the current program so the user can read the contents of the floppy disk drive. This parameter designates what program is used to write the directory listing to. Program O8999 is the default value.
Parameter	228 QUICKCODE FILE	This parameter set the program numbers to store in the Quick Code definition.
Parameter	229 X LEAD COMP 10E9	This parameter sets the X-axis lead screw compensation signed parts per billion.
Parameter	230 Y LEAD COMP 10E9	This parameter sets the Y-axis lead screw compensation signed parts per billion.
Parameter	231 Z LEAD COMP 10E9	This parameter sets the Z-axis lead screw compensation signed parts per billion.
Parameter	232 A LEAD COMP 10E9	This parameter sets the A-axis lead screw compensation signed parts per billion.
Parameter	233 B LEAD COMP 10E9	This parameter sets the B-axis lead screw compensation signed parts per billion.
Parameter	234 C BELT COMPENSATION	This parameter sets the belt compensation.
Parameter	235 AUTO DOOR PAUSE	This parameter that supports the Auto-Door feature. It specifies the length of a pause (in 50ths of a second) that occurs during the door close sequence. As the door closes and the switch is activated, the motor is turned off for this amount of time and the door coasts. This allows the door to close smoothly. This parameter should be set to 3 (0.06 seconds) nominally. It works in conjunction with parameter 236.
Parameter	236 AUTO DOOR BUMP	This parameter that supports the Auto-Door feature. It specifies the length of time (in 50ths of a second) that the motor should be reactivated after the pause specified by parameter 235. This causes the motor to close the door fully and smoothly. This parameter should be set to 15 (0.3 seconds) nominally.
Parameter	237 HPC PRESSURE BLEED	This parameter is for the HPC (High Pressure Coolant) feature. It is the amount of time given for the coolant to purge when the HPC system is shut off. This should be set to 250 on all lathes.



Parameter	238 SPINDLE AT SPEED %	This parameter is used to allow a program to command the spindle to a certain speed and then continue to the next block before the spindle has actually reached that speed. This is intended to make G-code programs run faster because the spindle can usually finish accelerating while approaching the part. It is recommended that this parameter be set to 20. The result will be that the lathe will act as though the spindle is at speed when it is within +/- 20% of the commanded speed.
Parameter	239 SPNDL ENC STEPS/REV	This parameter sets the number of encoder steps per revolution of the spindle encoder.
Parameter	240 1ST AUX MAX TRAVEL	This parameter sets the maximum travel of the first auxiliary axis in the positive direction.
Parameter	241 2ND AUX MAX TRAVEL	This parameter sets the maximum travel of the second auxiliary axis in the positive direction.
Parameter	242 3RD AUX MAX TRAVEL	This parameter sets the maximum travel of the third auxiliary axis in the positive direction.
Parameter	243 4TH AUX MAX TRAVEL	This parameter sets the maximum travel of the fourth auxiliary axis in the positive direction.
Parameter	244 1ST AUX MIN TRAVEL	This parameter sets the maximum travel of the first auxiliary axis in the negative direction.
Parameter	245 2ND AUX MIN TRAVEL	This parameter sets the maximum travel of the second auxiliary axis in the negative direction.
Parameter	246 3RD AUX MIN TRAVEL	This parameter sets the maximum travel of the third auxiliary axis in the negative direction.
Parameter	247 4TH AUX AXIS MIN TRAVEL	This parameter sets the maximum travel of the fourth auxiliary axis in the negative direction.
Parameter	248 MAX SPINDLE SPEED ALLOWED	The RPM above which the chuck will not operate. If the spindle is spinning faster than this value the chuck will not open, and if it is spinning slower than this value the chuck will open. The default is 0, for safety.
Parameter	249 DLY AFTER CHUCK IS CLMPED	The dwell time that is allowed after clamping the chuck (an M10 command). Program execution will not continue until this time has expired. Units are in milliseconds.



Parameter	250 DLY AFTER CHUCK IS UNCLMP	The dwell time that is allowed after unclamping the chuck (an M11 command). Program execution will not continue until this time has expired. Units are in milliseconds.
Parameter	251 A DOOR OPEN ERRTIME	This parameter specifies the number of milliseconds allowed for the door to open (move away from the door-closed switch). If the door is commanded to open, and does not open within the allowed time, alarm 127 DOOR FAULT is generated. Also, the value of this parameter plus one second specifies the number of milliseconds allowed for the door to close (activate the door-closed switch). If the door is commanded to close, and does not close within the allowed time, alarm 127 DOOR FAULT is generated. If an automatic door is installed, this parameter should be set to 2400 (2.4 seconds) nominally, otherwise it should be set to zero.
Parameter	252 TAILSTOCK OVERLOAD -DIR	Determines the overload limit when the tailstock is traveling in the minus direction, toward the spindle. This is an arbitrary value based on the effective voltage being sent to the tailstock servo motor. If this value is too low, you may not be able to move the tailstock. Increase the value until you are able to move the tailstock. The value for Parameter 252 should be approximately 1/2 the value of Parameter 253. This parameter is used for leadscrew tailstock or TL-15.
Parameter	253 TAIL STOCK OVERLOAD +DIR	Determines the overload limit when the tailstock is traveling in the positive direction, away from the spindle. The value for Parameter 253 should be approximately twice the value of Parameter 252. This parameter is used for leadscrew tailstock or TL-15.
Parameter	254 SPINDLE CENTER	Reserved for service use only.
Parameter	255 CONVEYOR TIMEOUT	The amount of time the conveyor will operate without any motion or keyboard action. After this time, the conveyor will automatically shut off. Note that this parameter value will cause the conveyor to shut off even if the intermittent feature is functioning. Note also that if this parameter is set to zero, the chip conveyor will shut off immediately, i.e., pressing CHIP FWD or CHIP REV will not turn it on.
Parameter	257 SPINDLE ORIENT OFFSET	This is used for the Vector Drive and the value is determined at the time of assembly.
Parameter	266 X SWITCHES	Parameter 266 is a collection of single-bit flags used to turn servo related functions on and off. The left and right cursor arrows are used to select the function being changed. All values are 0 or 1 only. The function names are:
0	X LIN SCALE EN	Used to enable linear scales for the X axis.
1	X INVRT LN SCL	Used to invert the X axis linear scale.
2	X DSBL LS ZTST	Used to disable the linear scale Z test.



3	TH SNSR COMP	This parameter is used for Lead Screw Thermal Compensation via a temperature sensor attached to the ball nut. When this bit is set to 1, the feature is activated for that axis. Note that the feature can only be used when temperature sensors are installed. The following parameters must be set appropriately: 201, 133 XZ SCREW COMP. COEF. =-190000000 272, 274 XZ SCREW COMP T. CONST =-27000000 351 TEMP PROBE OFFSET =450000
4	X 2ND HOME BTN	Used to move axis to coordinate specified in Work Ofset G129
5	X NEG COMP DIR	Used to negate the direction of thermal compensation
7	MAX TRAV INP	
8	NO ZERO/NOHOME	This feature is intended for lathes that have extra tools mounted on the outside of the turret. If this bit is set to zero, it will have no effect. If it is set to 1, the associated axis will not move when POWER UP/RESTART, HOME G28 or AUTO ALL AXES is pressed. The reason for this feature is to help prevent collisions between tools mounted on the outside of the turret and a sub-spindle mounted on the tailstock. It is important to note that a single axis HOME G28 (e.g., press Z then HOME G28) and any G28 specified in a program will still cause the axis to move regardless of the value of this parameter bit. The operator must exercise care when commanding any axis move.

Parameter	267 Y SWITCHES	Parameter 267 is a collection of single-bit flags used to turn servo related functions on and off. The left and right cursor arrows are used to select the function being changed. All values are 0 or 1 only. The function names are
0	Y LIN SCALE EN	Used to enable linear scales for the Y axis.
1	Y INVRT LN SCL	Used to invert the Y axis linear scale.
2	Y DSBL LS ZTST	Used to disable the linear scale Z test.
3	TH SNSR COMP	This parameter is used for Lead Screw Thermal Compensation via a temperature sensor attached to the ball nut. When this bit is set to 1, the feature is activated for that axis. Note that the feature can only be used when temperature sensors are installed. The following parameters must be set appropriately: 201, 133 XZ SCREW COMP. COEF. =-190000000 272, 274 XZ SCREW COMP T. CONST =-27000000 351 TEMP PROBE OFFSET =450000
4	Y 2ND HOME BTN	Used to move axis to coordinate specified in Work Ofset G129
5	Y NEG COMP DIR	Used to negate the direction of thermal compensation
7	MAX TRAV INP	



- 8 NO ZERO/NOHOME This feature is intended for lathes that have extra tools mounted on the outside of the turret. If this bit is set to zero, it will have no effect. If it is set to 1, the associated axis will not move when POWER UP/RESTART, HOME G28 or AUTO ALL AXES is pressed. The reason for this feature is to help prevent collisions between tools mounted on the outside of the turret and a sub-spindle mounted on the tailstock. It is important to note that a single axis HOME G28 (e.g., press Z then HOME G28) and any G28 specified in a program will still cause the axis to move regardless of the value of this parameter bit. The operator must exercise care when commanding any axis move.

Parameter	268 Z SWITCHES	Parameter 268 is a collection of single-bit flags used to turn servo related functions on and off. The left and right cursor arrows are used to select the function being changed. All values are 0 or 1 only. The function names are:
0	Z LIN SCALE EN	Used to enable linear scales for the Z axis.
1	Z INVRT LN SCL	Used to invert the Z axis linear scale.
2	Z DSBL LS ZTST	Used to disable the linear scale Z test.
3	TH SNSR COMP	This parameter is used for Lead Screw Thermal Compensation via a temperature sensor attached to the ball nut. When this bit is set to 1, the feature is activated for that axis. Note that the feature can only be used when temperature sensors are installed. The following parameters must be set appropriately: 201, 133 XZ SCREW COMP. COEF. =-190000000 272, 274 XZ SCREW COMP T. CONST =-27000000 351 TEMP PROBE OFFSET =450000
4	Z 2ND HOME BTN	Used to move axis to coordinate specified in Work Ofset G129
5	Z NEG COMP DIR	Used to negate the direction of thermal compensation
7	MAX TRAV INP	
8	NO ZERO/NOHOME	This feature is intended for lathes that have extra tools mounted on the outside of the turret. If this bit is set to zero, it will have no effect. If it is set to 1, the associated axis will not move when POWER UP/RESTART, HOME G28 or AUTO ALL AXES is pressed. The reason for this feature is to help prevent collisions between tools mounted on the outside of the turret and a sub-spindle mounted on the tailstock. It is important to note that a single axis HOME G28 (e.g., press Z then HOME G28) and any G28 specified in a program will still cause the axis to move regardless of the value of this parameter bit. The operator must exercise care when commanding any axis move.
Parameter	269 A SWITCHES	Parameter 269 is a collection of single-bit flags used to turn servo related functions on and off. The left and right cursor arrows are used to select the function being changed. All values are 0 or 1 only. The function names are:
0	A LIN SCALE EN	Used to enable linear scales for the A axis.



1	A INVRT LN SCL	Used to invert the A axis linear scale.
2	A DSBL LS ZTST	Used to disable the linear scale Z test.
3	TH SNSR COMP	This parameter is used for Lead Screw Thermal Compensation via a temperature sensor attached to the ball nut. When this bit is set to 1, the feature is activated for that axis. Note that the feature can only be used when temperature sensors are installed. The following parameters must be set appropriately: 201, 133 XZ SCREW COMP. COEF. =-190000000 272, 274 XZ SCREW COMPT. CONST =-27000000 351 TEMP PROBE OFFSET =450000
4	A 2ND HOME BTN	Used to move axis to coordinate specified in Work Ofset G129
5	A NEG COMP DIR	Used to negate the direction of thermal compensation
7	MAX TRAV INP	
8	NO ZERO/NOHOME	This feature is intended for lathes that have extra tools mounted on the outside of the turret. If this bit is set to zero, it will have no effect. If it is set to 1, the associated axis will not move when POWER UP/RESTART, HOME G28 or AUTO ALL AXES is pressed. The reason for this feature is to help prevent collisions between tools mounted on the outside of the turret and a sub-spindle mounted on the tailstock. It is important to note that a single axis HOME G28 (e.g., press Z then HOME G28) and any G28 specified in a program will still cause the axis to move regardless of the value of this parameter bit. The operator must exercise care when commanding any axis move.

Parameter	270 B SWITCHES	Parameter 270 is a collection of single-bit flags used to turn servo related functions on and off. The left and right cursor arrows are used to select the function being changed. All values are 0 or 1 only. The function names are:
0	B LIN SCALE EN	Used to enable linear scales for the B axis.
1	B INVRT LN SCL	Used to invert the B axis linear scale.
2	B DSBL LS ZTST	Used to disable the linear scale Z test.
3	TH SNSR COMP	This parameter is used for Lead Screw Thermal Compensation via a temperature sensor attached to the ball nut. When this bit is set to 1, the feature is activated for that axis. Note that the feature can only be used when temperature sensors are installed. The following parameters must be set appropriately: 201, 133 XZ SCREW COMP. COEF. =-190000000 272, 274 XZ SCREW COMPT. CONST =-27000000 351 TEMP PROBE OFFSET =450000
4	B 2ND HOME BTN	Used to move axis to coordinate specified in Work Ofset G129
5	B NEG COMP DIR	Used to negate the direction of thermal compensation
7	MAX TRAV INP	



- 8 NO ZERO/NOHOME This feature is intended for lathes that have extra tools mounted on the outside of the turret. If this bit is set to zero, it will have no effect. If it is set to 1, the associated axis will not move when POWER UP/RESTART, HOME G28 or AUTO ALL AXES is pressed. The reason for this feature is to help prevent collisions between tools mounted on the outside of the turret and a sub-spindle mounted on the tailstock. It is important to note that a single axis HOME G28 (e.g., press Z then HOME G28) and any G28 specified in a program will still cause the axis to move regardless of the value of this parameter bit. The operator must exercise care when commanding any axis move.

Parameter	271 C SWITCHES	Parameter 271 is a collection of single-bit flags used to turn servo related functions on and off. The left and right cursor arrows are used to select the function being changed. All values are 0 or 1 only. The function names are:
0	C LIN SCALE EN	Used to enable linear scales for the C axis.
1	C INVRT LN SCL	Used to invert the C axis linear scale.
2	C DSBL LS ZTST	Used to disable the linear scale Z test.
3	TH SNSR COMP	This parameter is used for Lead Screw Thermal Compensation via a temperature sensor attached to the ball nut. When this bit is set to 1, the feature is activated for that axis. Note that the feature can only be used when temperature sensors are installed. The following parameters must be set appropriately: 201, 133 XZ SCREW COMP. COEF. =-190000000 272, 274 XZ SCREW COMP T. CONST =-27000000 351 TEMP PROBE OFFSET =450000
4	C 2ND HOME BTN	Used to move axis to coordinate specified in Work Ofset G129
5	C NEG COMP DIR	Used to negate the direction of thermal compensation
7	MAX TRAV INP	
8	NO ZERO/NOHOME	This feature is intended for lathes that have extra tools mounted on the outside of the turret. If this bit is set to zero, it will have no effect. If it is set to 1, the associated axis will not move when POWER UP/RESTART, HOME G28 or AUTO ALL AXES is pressed. The reason for this feature is to help prevent collisions between tools mounted on the outside of the turret and a sub-spindle mounted on the tailstock. It is important to note that a single axis HOME G28 (e.g., press Z then HOME G28) and any G28 specified in a program will still cause the axis to move regardless of the value of this parameter bit. The operator must exercise care when commanding any axis move.
Parameter	272 X THERM COMP T. CONST	This parameter supports Lead Screw Thermal Compensation. The value is the time constant that govern the rate of cool down of the screw. This parameter should be set to -5000.



Parameter	273 Y THERM COMPT. CONST	This parameter supports Lead Screw Thermal Compensation. The value is the time constant that govern the rate of cool down of the screw. This parameter should be set to 0.
Parameter	274 Z THERM COMPT. CONST	This parameter supports Lead Screw Thermal Compensation. The value is the time constant that govern the rate of cool down of the screw. This parameter should be set to -3000.
Parameter	275 A THERM COMPT. CONST	This parameter supports Lead Screw thermal Compensation. The value is the time constant that govern the rate of cool down of the screw. This parameter should be set to 0.
Parameter	276 B THERM COMPT. CONST	This parameter supports Lead Screw thermal compensation. The value is the time constant that govern the rate of cool down of the screw. This parameter should be set to zero.
Parameter	278 COMMON SWITCH 3	Parameter 278 is a collection of general purpose single bit flags used to turn some functions on and off. The left and right cursor arrows are used to select the function being changed. All values are 0 or 1 only. The function names are:
0	INVERT G.B.	Default is 0. When this bit is set to 1, the sense of the discrete inputs for SP HIGH and SP LOW (high and low gear) are inverted.
1	DPR SERIAL	Causes the main serial inputs/outputs to go through the floppy disk video board.
2	CK PALLET IN	
3	CK HIDDN VAR	
4	DISPLAY ACT	When set to 1, displays the actual spindle speed on the Current Commands display page.
6	HYDRAULIC TS	This bit enables the hydraulic tailstock
7	SPND DRV LCK	This bit must be set to 0 if machine is equipped with a Haas vector spindle drive.
8	CHUCK OPN CS	When set to 1, the user can press CYCLE START and run a program with the chuck unclamped. If the spindle is commanded with this bit set to 1, the spindle will not exceed the CHUCK UNCLAMP RPM (Parameter 248). The default for this bit is 0. This feature is ineffective when the CE safety circuit is enabled.
9	CNCR SPINDLE	When set to 0, spindle start occurs at the end of a block, as in normal M code operation. When set to 1, spindle start occurs at the beginning of a block and concurrent with axis motion.
10	TL SET PROBE	This bit must be set to 1 in order to enable the Tool Pre-Setter.
11	HAAS VECT DR	(Haas Vector Drive) This bit must be set to 1 if machine is equipped with a HAAS vector spindle drive. When set to 1, voltage to the Haas vector drive is displayed in the diagnostics display as DC BUSS.



12 uP ENCL TEMP	(Microprocessor enclosure temperature) When set to 1, the enclosure temperature will be displayed on INPUTS2 screen of the diagnostics display.
13 HAAS RJH	Haas remote jog handle. This bit must be set to 1 if the machine is equipped with a Haas 5-axis Remote jog handle.
14 SPIN TEMP NC	Spindle temperature normally closed. This bit specifies the type (normally open normally closed) of the spindle temperature sensor. This bit should be set to 1 for machines with a Haas Vector Drive, and 0 for machines without a Vector Drive.
15 SUBSP TMP NC	(Subspindle Temperature Sensor Normally Closed) This bit specifies the type, normally open or normally closed, of the subspindle temperature sensor.
17 NO MFIN CKPU	When it is set, it will prevent checking of MFIN at power-up. It should be set to 1 for all machines that have the new Haas Automatic Pallet Changer attached, and 0 for all other machines.
18 D:Y SW ENABL	Delta Wye switch enable, this is used for machine with a Vector Drive. If this switch is set, but bit 19 is not, then winding switching will only be done when the spindle is at rest, depending on the target speed of the spindle
19 DY SW ON FLY	Delta Wye switch enable, this is used for machine with a Vector Drive. This parameter enables switching on the fly, as the spindle motor is accelerating or decelerating through the switch point.
20 CK BF STATUS	This bit has been added for the improved Bar Feeder interface. When this bit is set to 1, the control will constantly check the Bar Feeder Status on discrete input 1027. If this input goes high, alarm 450 BAR FEEDER FAULT will be generated and the servos and spindle will be turned off. Note that the spindle will simply coast to a stop.
21 CK BF SP ILK	This bit has been added for the improved Bar Feeder interface. When this bit is set to 1, the control will constantly check the Bar Feeder Spindle Interlock on discrete input 1030. If this input goes high, and the spindle is being commanded to turn, or coasting or being manually turned at 10rpm or more, alarm 451 BAR FEEDER SPINDLE INTERLOCK will be generated and the servos and spindle will be turned off. Note that the spindle will simply coast to a stop.
24 LIVE TOOLING	Lathes fitted with the Live Tooling drive this bit must be set to 1. For all other lathes, this bit is set to 0.
25 SUBSPINDLE	This bit enables G14, G15, M143, M144, M145. It must be set to 1 for all lathes with the subspindle. When this bit is set to 1, the control will display FUNCTION LOCKED when the AUTO ALL AXES, HOME G28, or POWER UP/RESTART buttons are pressed.
26 CAXIS DRIVE	This bit enables M154 and M155. It must be set to 1 for all lathes with the C axis.



29	SAFETY INVERT	This bit supports the CE door interlock that locks when power is turned off. For machines that have the regular door lock that locks when power is applied, this bit must be set to 0. For machines that have the inverted door lock, this bit must be set to 1.
31	INV SPD DCEL	Inverse spindle speed deceleration. When this parameter is set to 1, the spindle decelerates faster at lower speeds, resulting in a shorter deceleration time.
Parameter	291	HYDRAULIC TAIL STK NO MOTION DETEC TIME The number in milliseconds that must pass with no B-axis encoder change before the control decides that the tailstock has stopped. The parameter affects homing and alarm situations on the tailstock. If the tailstock pressure is set low and the tailstock does not home properly then increase this parameter.
Parameter	292	HYD TS RTRACT MARGN (Hydraulic Tailstock Retract Margin) This parameter sets the acceptable range, in encoder steps, for the retract point. When the tailstock stops anywhere within this range, the control assumes it is at the retract point. The default is 5 encoder steps. This means that a 10 encoder step range is set around the retract point.
Parameter	293	HYD TS SLOW DISTNCE (Hydraulic Tailstock Slow Distance) This parameter sets the distance, prior to a target point, where the tailstock will transition from a rapid movement to a feed. For example, if this parameter is set to 30 (the default), this means the tailstock will slow to a feed 30 encoder steps before reaching the target point. Units are in encoder steps.
Parameter	294	MIN BUSS VOLTAGE This parameter specifies the minimum Haas Vector Drive buss voltage. If the machine has a Haas Vector Drive, the parameter should be set to 270 (volts). Machines without a Vector Drive should be set to 0. Alarm 160 LOW VOLTAGE will be generated if the voltage falls below the minimum specified.
Parameter	296	MAX OVER VOLT TIME Specifies the amount of time (in 50ths of a second) that an overvoltage condition (alarm 119 OVER VOLTAGE) will be tolerated before the automatic shut down process is started.
Parameter	297	MAX OVERHEAT TIME Specifies the amount of time (in 50ths of a second) that an overheat condition (alarm 122 REGEN OVERHEAT) will be tolerated before the automatic shut down process is started.
Parameter	298	YAX RTAP BACKLASH This parameter is normally set to zero, but can be adjusted by the user (to a number typically between 0 and 1000) to compensate for play in the center of the main spindle. It takes effect during G95 SUBSPIDLE RIGID TAP when the tool has reached the bottom of the hole and must reverse direction to back out.
Parameter	299	AUTOFEED STEP-UP This parameter works with the AUTOFEED feature. It specifies the feed rate step-up percentage per second and should initially be set to 10.



Parameter 300 AUTOFEED-STEP-DOWN
This parameter works with the AUTOFEED feature. It specifies the feed rate step-down percentage per second and should initially be set to 20.

Parameter 301 AUTOFEED-MIN-LIMIT
This parameter works with the AUTOFEED feature. It specifies the minimum allowable feed rate override percentage that the AUTOFEED feature can use and should initially be set to 1. For more information see AUTOFEED under the new features section.

NOTE: When tapping, the feed and spindle overrides will be locked out, so the AUTOFEED feature will be ineffective (although the display will appear to respond to the override buttons.)

NOTE: The last commanded feed rate will be restored at the end of the program execution, or when the operator presses RESET or turns off the AUTOFEED feature.

NOTE: The operator may use the feed rate override buttons while the AUTOFEED feature is active. As long as tool load limit is not exceeded, these buttons will have the expected effect and the overridden feed rate will be recognized as the new commanded feed rate by the AUTOFEED feature. However, if the tool load limit has already been exceeded, the control will ignore the feed rate override buttons and the commanded feed rate will remain unchanged.

Parameter 304 SPINDLE BRAKE DELAY
This parameter specifies the amount of time (in milliseconds) to wait for the main spindle brake to unclamp when spindle speed has been commanded, and also the amount of time to wait after the main spindle has been commanded to stop before clamping it.

Parameter 305 SERVO PO BRK DLY
Specifies the time (in milliseconds) that the control should wait after turning off the Hyd Pump Enable relay (which will activate the brake) before turning off power to the servo motors via the MOCON. This is intended to allow time for the brake to engage. This parameter should be set to 200.

Parameter 315

0 ALIS M GRPHC	All user defined M codes (such as M50) will be ignored when a program is run in graphics mode if this bit is set to 0. If it is necessary to have graphics recognize such M codes, this bit should be set to 1.
5 DOOR OPEN SW	This ensures that when the door is opened automatically, it opens all the way. It is intended to be used in conjunction with an automatic parts loader. If this bit is set to zero, the control behaves as before. If this bit is set to 1, the control will look for a second door switch when the door is opened automatically. If the switch is not found, alarm 127 DOOR FAULT will be generated. This bit should be set to 1 on all machines fitted with the second door switch.
16 SS REV SPN E	Reverses sense direction of subspindle encoder



17 SS VEC D ENC	Enables a second encoder that is mounted on the subspindle motor and wired into the "C" axis input of the Mocon. It is required to control the vector algorithm when the lathe's belts might slip at high load.
18 SS VEC DRIVE	This bit must be set to 1 if the machine is equipped with a HAAS vector subspindle drive. When set to 1, voltage to the Haas vector drive is displayed in the diagnostics display as DC BUSS. For the TL-15 and VTC-48, this bit must be set to 1. For all others, it must be set to 0.
19 SS D:Y SW EN	Delta Wye switch enable. This is used for the Vector Drive. If this switch is set, but bit 19 is not, then winding switching will only be done when the subspindle is at rest, depending on the target speed of the subspindle.
20 SS DY SW FLY	Delta Wye switch on the fly. This is used for the Vector Drive. Enables switching on the fly, as the subspindle motor is accelerating or decelerating through the switch point. If bit 18 (SS VEC DRIVE) is not set, this switch will be ignored.
21 SS IN SPD DC	Subspindle Inverse Speed Deceleration. When this parameter is set to 1, the subspindle decelerates faster at lower speeds, resulting in a shorter deceleration time.
22 SS DISBLE GB	Disables gear box functions. For the TL-15 and VTC-48, this bit must be set to 1. For all others, it must be set to 0.
23 VERT TRN CTR	This is a new parameter for the VTC-48.
24 SS INVERT GB	This bit allows an alternate gearbox configuration. It inverts the sense of the gearbox inputs. The default is 0. When this bit is set to 1, the sense of the discrete inputs for SP HIG and SP LOW (high and low gear) are inverted.



Parameter	315 SIMPLE T.S.
	This parameter supports the SL-10 tailstock, which has no encoder. It should be set to 1 only on an SL-10 with a hydraulic tailstock. It should be set to zero on all other machines. When this bit is set to 1 the following differences will be observed: 1. First, note that the SL-10 tailstock consists of a fixed head, and a moveable center rod. Therefore, the only moving part is called the tailstock center. 2. The tailstock center is always considered to be at zero; as there is no encoder, the control cannot know where the tailstock center is. 3. Pressing POWERUP/RESTART or AUTOALLAXES will not cause the tailstock center to physically move. It is the operator's responsibility to move it out of the way to avoid a collision. 4. Tailstock center movement using the jog handle and remote jog handle is disabled. 5. M21 TAIL STOCK FORWD and M22 TAIL STOCK REVERS. When an M21 is commanded, the tailstock center will be commanded to move towards the spindle and maintain continuous pressure. Note that the program will not wait while this is completed. Instead, the next block will be executed immediately. Because of this, a dwell should be commanded of sufficient length 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 for the time specified by parameter 580, and then stop. The running program will wait during this time. 6. Setting 94 Z/TS DIFF @X CLEARNCE. Normally, this setting (in conjunction with setting 93) specifies a moveable restricted zone, that is, the minimum allowable difference between the Z and B axes at, or below, the tailstock center X clearance plane specified by Setting 93. When SIMPLE T.S. is set to 1, however, this setting is instead used to specify the distance from the Z axis home position. This is because, lacking an encoder, the control cannot know the exact position of the tailstock center. Thus, instead of providing a dynamic restricted zone which moves with the tailstock, it provides only a fixed restricted zone based at the home position. The software will alert the operator if the X or Z axes enter this restricted zone, but will not alert the operator if the tailstock center leaves the zone. Care should be taken to ensure that the tailstock center operates only within this zone. 7. Tailstock center Foot Pedal movement. Because there is no encoder, the concepts of Retract Point, Advance Point and Hold Point do not apply. The tailstock center foot pedal has the following effects. a) If the tailstock center was previously commanded to retract, pressing the pedal will cause it to advance at low pressure and maintain pressure on anything it encounters until commanded otherwise. b) If the tailstock center was previously commanded to advance, pressing and holding the pedal will cause it to retract at high pressure until the pedal is released.
Parameter	316 MEASURE BAR RATE
	This parameter supports the Haas Servo Bar 300 barfeeder. It is the rate at which the bars are measured. Units are inches*1000.
Parameter	317 MEASURE BAR INC
	This parameter supports the Haas Servo Bar 300 barfeeder. This is the increment used for bar measurement. Units are inches*10,000
Parameter	318 GEAR MOTOR TIMEOUT
	This parameter supports the Haas Servo Bar 300 barfeeder. This is the timeout value for gearmotor operations. Units are in milliseconds.



Parameter	319 MAX RETRACT POS This parameter supports the Haas Servo Bar 300 barfeeder. This is the maximum V axis position when retracted. Units are inches * 10000.
Parameter	320 MIN RETRACT POS This parameter supports the Haas Servo Bar 300 barfeeder. This is the minimum space between bar and push rod when retracted. Units are inches*10,000
Parameter	321 PUSH ROD ZERO POS This parameter supports the Haas Servo Bar 300 barfeeder. This is the V axis position for loading and unloading a bar. Units are in inches*10,000.
Parameter	322 GEARMOTOR BUMP TIME This parameter supports the Haas Servo Bar 300 barfeeder. Gear motor run time for bump and internal functions. Units are in milliseconds.
Parameter	323 PUSH RATE This parameter supports the Haas Servo Bar 300 barfeeder. This is the rate at which the last 1/4 inch of feed is done. Units are inches per minute*1000.
Parameter	324 GEAR MOTOR SETTLE This parameter supports the Haas Servo Bar 300 barfeeder. This is the minimum dwell time for reversing the gear motor direction. Units are in milliseconds.
Parameter	325 STANDARD BAR LEN This parameter supports the Haas Servo Bar 300 barfeeder. This is the length of bar for G105 Q5. Units are in inches per minute*1000.
Parameter	326 G5 DECELERATION This parameter supports the G05 FINE SPINDLE CTRL feature. This is the rate at which to decelerate the spindle during G5. Units are in encoder steps per second. It should be set to 15000.
Parameter	327 X LS PER INCH This parameter is used on machines equipped with linear scales. It should be set to zero.
Parameter	328 Y LS PER INCH Same as parameter 327.
Parameter	329 Z LS PER INCH Same as parameter 327.
Parameter	330 A LS PER INCH Same as parameter 327.
Parameter	331 B LS PER INCH Same as parameter 327.



Parameter 333 X LS PER REV
This parameter is used on machines equipped with linear scales. It should be set to zero.

Parameter 334 Y LS PER REV
Same as parameter 333.

Parameter 335 Z LS PER REV
Same as parameter 333.

Parameter 336 A LS PER REV
Same as parameter 333.

Parameter 337 B LS PER REV
Same as parameter 333.

Parameter 339 X SPINDLE THERM COEF.
This parameter supports the Spindle Head Thermal Compensation feature. It should be set to 8000.

Parameter 340 Y SPINDLE THERM COEF.
This parameter supports the Spindle Head Thermal Compensation feature. It should be set to 0.

Parameter 341 Z SPINDLE THERM COEF.
This parameter supports the Spindle Head Thermal Compensation feature. It should be set to 3692.

Parameter 342 A SPINDLE THERM COEF.
This parameter supports the Spindle Head Thermal Compensation feature. It should be set to 0.

Parameter 343 B SPINDLE THERM COEF.
This parameter supports the Spindle Head Thermal Compensation feature. It should be set to 0.

Parameter 345 X SPINDLE THERM T.C.
This parameter supports the Spindle Head Thermal Compensation feature. It should be set to -12561.

Parameter 346 Y SPINDLE THERM T.C.
This parameter supports the Spindle Head Thermal Compensation feature. It should be set to 0.

Parameter 347 Z SPINDLE THERM T.C.
This parameter supports the Spindle Head Thermal Compensation feature. It should be set to -20000.



- Parameter 348 A SPINDLE THERM T.C.
This parameter supports the Spindle Head Thermal Compensation feature. It should be set to 0.
- Parameter 349 B SPINDLE THERM T.C.
This parameter supports the Spindle Head Thermal Compensation feature. It should be set to 0.
- Parameter 351 THRML SENSOR OFFSET
This parameter is used for Lead Screw Thermal Compensation via a temperature sensor attached to the ball nut.
- Parameter 352 RELAY BANK SELECT
In all previous versions, parameter 209 bit 23 MCD RLY BRD assumes that relay bank zero is to be used. This parameter allows the user to change which bank is to be used. It may be set to a number from 0 to 3 (inclusive). M codes M21 through M28 will be switched to the selected bank. Note that this feature requires the I/O-S board. If a previous board is installed without the additional banks of relays, this parameter should be set to zero.
- Parameter 353 MAX SUBSPINDLE RPM
This is the maximum RPM available to the subspindle. This parameter works in conjunction with parameters 570 and 571
The following 6 parameters are reserved for future use:
- 354 U Axis Switches
 - 390 V Axis Switches
 - 426 W Axis Switches
 - 462 Tt Axis Switches
 - 498 C Axis Switches
 - 534 Ss Axis Switches
- Parameter 354 U SWITCH A
See Parameter 1 for description.
- Parameter 390 V SWITCH A
See Parameter 1 for description.
- Parameter 426 W SWITCH A
See Parameter 1 for description.
- Parameter 498 C SWITCH A
See Parameter 1 for description.
- Parameter 570 SUBSPIN ENC ST/REV
This parameter sets the number of encoder steps per revolution of the subspindle encoder.



Parameter	571 SUBSPINDLE ST/REV	This parameter sets the number of encoder steps per revolution of the subspindle. This parameter only applies to the subspindle rigid tapping option.
Parameter	572 C AXIS ENG TIMEOUT	Specifies the C axis timeout value for seeing the engaged switch on engagement or the disengaged switch on disengage. The units are in milliseconds and it should be set to 1000 for all lathes.
Parameter	573 C AXIS ENG DELAY 1	Specifies the C axis delay after spindle orientation and before engagement. Its purpose is to let the spindle orientation settle. The units are milliseconds and it should be set to 250 for all lathes.
Parameter	574 C AXIS ENG DELAY 2	Specifies the C axis delay after engagement before the motion completes. Its purpose is to allow the C axis engagement to come up to pressure. The units are milliseconds and it should be set to 250 for all lathes.
Parameter	575 THRD PTCH FACT PPM	This allows the customer to factor the feed rate on G32, G76 and G92 threading as necessary for particular applications. The units are ppm (parts per million.) This parameter can be adjusted as necessary, for example, increasing the value by 100 will advance the lead of the thread by 1 ten-thousandth of an inch per inch. Note that this parameter is internally limited to 1000. All lathes should be shipped with this parameter set to 200.
Parameter	576 MAX SS RPM LOW GEAR	Max subspindle RPM in low gear. This is the maximum RPM available to the subspindle. When this speed is programmed, the D-to-A output will be +10V and the subspindle drive must be calibrated to provide this. Gear ratio low to high is 4.1:1.
Parameter	577 SS ORIENT OFFSET	Subspindle Orientation Offset. It is used to orient the subspindle properly anytime it needs to be locked such as prior to a tool change, or orient subspindle command. This is used for the vector drive and the value is determined at assembly time. The Subspindle position is displayed on the POS-RAW DAT screen just to the right of SYSTEM TIME.
Parameter	578 SS HIGH GR MIN SPD	Command speed used to rotate subspindle motor when orienting subspindle in high gear. Units are maximum subspindle RPM divided by 4096.
Parameter	579 SS LOW GR MIN SPD	Command speed used to rotate subspindle motor when orienting subspindle in low gear. Units are maximum subspindle RPM divided by 4096.
Parameter	580 TS HYD RETRACT TIME	This parameter has been added for the SL-10 hydraulic no-encoder tailstock. It specifies the amount of time (in ms) that the tailstock center will be commanded to retract as a result of commanding an M22 and only takes effect when SIMPLE TS is set to 1.



Parameter 587 EXTENDED PUSH TIME
 This parameter supports the barfeeder pusher rod which is mounted on the barfeeder trolley (for barfeeders with the 1-foot extension option.) The units are 50ths of a second. It causes a delay of the amount of time specified to enable the pusher rod to full extend before the trolley begins to travel back to the home position. This parameter should be set to 150 (3 seconds) on the SL-30 Big Bore and SL-40 only. For all other lathes, it should be set to zero. On older lathes without the pusher rod, this parameter will have no effect. Note also that with this change, the I/O board discrete output has been changed from #23 to #1.

Parameter 588 X ENC. SCALE FACTOR
 These are new axis parameters that work in place of the axis parameters called SCALE/X LO and SCALE/X HI. If SCALE FACT/X is set to 1, the scale ratio is determined by SCALE/X LO and SCALE/X HI as follows:

HI	LO
0	0
0	1
1	0
1	1

If, however, SCALE FACT/X is set to zero, the value of ENC. SCALE FACTOR will be used for the scale ratio instead. Note that any value outside the range of 1 to 100 will be ignored and the scale ratio will remain unaffected. Note also that currently, these parameters are intended for use only on rotary axes (A and B).

Parameter 589 Y ENC. SCALE FACTOR
 See parameter 588 for description

Parameter 590 Z ENC. SCALE FACTOR
 See parameter 588 for description

Parameter 591 A ENC. SCALE FACTOR
 See parameter 588 for description

Parameter 592 B ENC. SCALE FACTOR
 See parameter 588 for description

Parameter 593 Sp ENC. SCALE FACTOR
 See parameter 588 for description

Parameter 594 U ENC. SCALE FACTOR
 See parameter 588 for description

Parameter 595 V ENC. SCALE FACTOR
 See parameter 588 for description

Parameter 596 W ENC. SCALE FACTOR
 See parameter 588 for description



- Parameter 597 C ENC. SCALE FACTOR
See parameter 588 for description
- Parameter 598 Tt ENC. SCALE FACTOR
See parameter 588 for description
- Parameter 599 Ss ENC. SCALE FACTOR
See parameter 588 for description
- Parameter 600 PEAK SPIN. PWR - KW
This parameter supports the spindle kilowatt (KW) load display which appears on the current commands page, next to the spindle load percentage. This parameter should be set to the peak power output in KW for the spindle motor.

ELECTRONIC THERMAL COMPENSATION

When ballscrews rotate they generate heat. Heat causes the ballscrews to expand. In constant duty cycles as in mold making the resultant ball screw growth can lead to cutting errors on the next morning start up. Haas' ETC algorithm can accurately model this heating and cooling effect and electronically expand and contract the screw to give near glass scale accuracy and consistency.

This compensation is based on a model of the lead screw which calculates heating based on the distance traveled and the torque applied to the motor. This compensation does not correct for thermal growth due to changes in ambient temperature or due to part expansion.

Electronic thermal compensation works by estimating the heating of the screw based on the total amount of travel over its length and including the amount of torque applied to the screw. This heat is then turned into a thermal coefficient of expansion and the position of the axis is multiplied by the coefficient to get a correction amount.

If the machine is turned off when there is some compensation applied (due to motion and heating of screw), when the machine is turned back on, the compensation will be adjusted by the clock indicated elapsed time.

SPINDLE HEAD THERMAL COMPENSATION

This feature integrates spindle speed over time and builds a model of thermal growth. As the model shows the spindle head warming up, the control adjusts the axes to compensate for thermal growth.

X-AXIS THERMAL COMPENSATION

During machining, the heating of the ballscrews transfers heat by conduction to the thermal sensor body. This causes the resistance of the sensor to vary according to the temperature. The resistance value is read by the software which compensates for the change in temperature by adjusting the accuracy of the program accordingly.

The thermal sensor is connected to the ballscrew and compensates program accuracy for changes in ballscrew temperature.



10. ALARMS

Any time an alarm is present, the lower right hand corner of the screen will have a blinking "ALARM". Push the ALARM display key to view the current alarm. All alarms are displayed with a reference number and a complete description. If the RESET key is pressed, one alarm will be removed from the list of alarms. If there are more than 18 alarms, only the last 18 are displayed and the RESET must be used to see the rest. The presence of any alarm will prevent the operator from starting a program.

The **ALARMS DISPLAY** can be selected at any time by pressing the ALARM MESGS button. When there are no alarms, the display will show NO ALARM. If there are any alarms, they will be listed with the most recent alarm at the bottom of the list. The CURSOR and PAGE UP and PAGE DOWN buttons can be used to move through a large number of alarms. The CURSOR **right** and **left** buttons can be used to turn on and off the ALARM history display.

Note that tool changer alarms can be easily corrected by first correcting any mechanical problem, pressing RESET until the alarms are clear, selecting ZERO RET mode, and selecting AUTO ALL AXES. Some messages are displayed while editing to tell the operator what is wrong but these are not alarms. See the editing topic for those errors.

The following alarm list shows the alarm numbers, the text displayed along with the alarm, and a detailed description of the alarm, what can cause it, when it can happen, and how to correct it.

Alarm number and text:	Possible causes:
101 MOCON Comm. Failure	During a self-test of communications between the MOCON and main processor, the main processor does not respond, and is suspected to be dead. Check cable connections and grounding.
102 Servos Off	Indicates that the servo motors are off, the tool changer is disabled, the coolant pump is off, and the spindle motor is stopped. Caused by EMERGENCY STOP, motor faults, tool changer problems, or power fail.
103 X Servo Error Too Large	Too much load or speed on X-axis motor. The difference between the motor position and the commanded position has exceeded a parameter. The motor may also be stalled, disconnected, or the driver failed. The servos will be turned off and a RESET must be done to restart. This alarm can be caused by problems with the driver, motor, or the slide being run into the mechanical stops.
104 Y Servo Error Too Large	Same as alarm 103.
105 Z Servo Error Too Large	Same as alarm 103.
106 A Servo Error Too Large	Same as alarm 103.



107 Emergency Off

EMERGENCY STOP button was pressed. Servos are also turned off. After the E-STOP is released, the RESET button must be pressed at least twice to correct this; once to clear the E-STOP alarm and once to clear the Servos Off alarm.

108 X Servo Overload

Excessive load on X-axis motor. This can occur if the load on the motor over a period of several seconds or even minutes is large enough to exceed the continuous rating of the motor. The servos will be turned off when this occurs. This can be caused by running into the mechanical stops but not much past them. It can also be caused by anything that causes a very high load on the motors.

109 Y Servo Overload

Same as alarm 108.

110 Z Servo Overload

Same as alarm 108.

111 A Servo Overload

Same as alarm 108.

112 No Interrupt

Electronics fault. Call your dealer.

113 Turret Unlock Fault

The turret took longer to unlock and come to rotation position than allowed for in Parameter 62. The value in Parameter 62 is in milliseconds. This may occur if the air pressure is too low, the tool turret clamp switch is faulty or needs adjustment, or there is a mechanical problem.

114 Turret Lock Fault

The turret took longer to lock and seat than allowed for in Parameter 63. The value in Parameter 63 is in milliseconds. This may occur if the air pressure is too low, the tool turret clamp switch is faulty or needs adjustment, or there is a mechanical problem.

115 Turret Rotate Fault

Tool motor not in position. During a tool changer operation the tool turret failed to start moving or failed to stop at the right position. Parameters 62 and 63 can adjust the time-out times. This alarm can be caused by anything that jams the rotation of the turret. A loss of power to the tool changer can also cause this, so check CB5 and relays 1-8, 2-3, and 2-4.

116 Spindle Orientation Fault

Spindle did not orient correctly. During a spindle orientation function, the spindle is rotated until the lock pin drops in; but the lock pin never dropped. Parameters 66, 70, 73, and 74 can adjust the time-out times. This can be caused by a trip of circuit breaker CB4, a lack of air pressure, or too much friction with the orientation pin.

117 Spindle High Gear Fault

Gearbox did not shift into high gear. During a change to high gear, the high gear sensor was not detected in time. Parameters 67, 70 and 75 can adjust the time-out times. Check circuit breaker CB4, the circuit breaker for the air pressure solenoids and the spindle drive.



- 118 Spindle Low Gear Fault Gearbox did not shift into low gear. During a change to low gear, the low gear sensor was not detected in time. Parameters 67, 70 and 75 can adjust the time-out times. Check the solenoid's circuit breaker CB4, and the spindle drive.
- 119 Over Voltage Incoming line voltage is above maximum. The tool changer, and coolant pump will stop. If this condition persists, an automatic shutdown will begin after the interval specified by parameter 296.
- 120 Low Air Pressure Air pressure dropped below 80 PSI for a period of time defined by Parameter 76. Check your incoming air pressure for at least 100 PSI and ensure that the regulator is set at 85 PSI.
- 121 Low Lub or Low Pressure Way lube is low or empty or there is no lube pressure or too high a pressure. Check tank at rear of machine and below control cabinet. Also check connector on the side of the control cabinet. Check that the lube lines are not blocked.
- 122 Regen Overheat The regenerative load temperature is above a safe limit. This alarm will turn off the spindle drive, coolant pump, and tool changer. One common cause of this overheat condition is an input line voltage too high. If this condition persists, an automatic shutdown will begin after the interval specified by parameter 297. It can also be caused by a high start/stop duty cycle of the spindle.
- 123 Spindle Drive Fault Overheat or failure of spindle drive or motor. The exact cause is indicated in the LED window of the spindle drive inside the control cabinet. This can be caused by a stalled motor, shorted motor, overvoltage, undervoltage, overcurrent, overheat of motor, or drive failure.
- 124 Low Battery Memory batteries need replacing within 30 days. This alarm is only generated at power on and indicates that the 3.3 volt Lithium battery is below 2.5 volts. If this is not corrected within about 30 days, you may lose your stored programs, parameters, offsets, and settings.
- 125 Tool Turret Fault Turret has not seated itself properly. There may be something obstructing the turret between the housing and the turret itself.
- 126 Gear Fault GGearshifter is out of position when a command is given to start a program or rotate the spindle. This means that the two speed gear box is not in either high or low gear but is somewhere in between. Check the air pressure, the solenoid's circuit breaker CB4, and the spindle drive. Use the POWER UP/RESTART button to correct the problem.
- 127 Door Fault The control failed to detect a low signal at the Door Switch input after the door was commanded and the Door Switch input was not received after the door was commanded to close and the time set in parameter #251 has elapsed.



129	M Fin Fault	M-Fin was active at power on. Check the wiring to your M code interfaces. This test is only performed at power-on.
130	Chuck Unclamped	The control detected that the chuck is unclamped. This is a possible fault in the air solenoids, relays on the I/O Assembly, or wiring.
131	Tool Not Clamped	When clamping or powering up the machine, the Tool Release Piston is not Home. "This is a possible fault in the air solenoids, relays on the I/O Assembly, the drawbar assembly, or wiring.
132	Power Down Failure	Machine did not turn off when an automatic power-down was commanded. Check wiring to POWIF card on power supply assembly, relays on the IO assembly, and the main contactor K1.
133	Spindle Brake Engaged	The brake is engaged. It must be released before the spindle can turn.
134	Low Hydraulic	Hydraulic pressure is sensed to be low. Check pump pressure and Pressure hydraulic tank oil level. Verify proper pump and machine phasing.
135	X Motor Over Heat	Servo motor overheat. The temperature sensor in the motor indicates over 150 degrees F. This can be caused by an extended overload of the motor such as leaving the slide at the stops for several minutes.
136	Y Motor Over Heat	Same as alarm 135.
137	Z Motor Over Heat	Same as alarm 135.
138	A Motor Over Heat	Same as alarm 135.
139	X Motor Z Fault	Encoder marker pulse count failure. This alarm usually indicates that the encoder has been damaged and encoder position data is unreliable. This can also be caused by loose connectors at P1-P4.
140	Y Motor Z Fault	Same as alarm 139.
141	Z Motor Z Fault	Same as alarm 139.
142	A Motor Z Fault	Same as alarm 139.
143	Spindle Not Locked	Shot pin not fully engaged when a tool change operation is being performed. Check air pressure and solenoid circuit breaker CB4. This can also be caused by a fault in the sense switch that detects the position of the lock pin.
144	Time-out-Call Your Dealer	Time allocated for use prior to payment exceeded. Call your dealer.



145	X Limit Switch	Axis hit limit switch or switch disconnected. This is not normally possible as the stored stroke limits will stop the slides before they hit the limit switches. Check the wiring to the limit switches and connector P5 at the side of the main cabinet. Can also be caused by a loose encoder shaft at the back of the motor or coupling of motor to the screw.
146	Y Limit Switch	Same as alarm 145.
147	Z Limit Switch	Same as alarm 145.
148	A Limit Switch	Normally disabled for rotary axis.
149	Spindle Turning	Spindle not at zero speed for tool change. A signal from the spindle drive indicating that the spindle drive is stopped is not present while a tool change operation is going on.
150	I Mode Out Of Range	Internal software error; call your dealer.
151	Low TSC	
152	Self Test Fail	Control has detected an electronics fault. All motors and solenoids are shut down. This is most likely caused by a fault of the processor board stack at the top left of the control. Call your dealer.
153	X-axis Z Ch Missing	Broken wires or encoder contamination. All servos are turned off. This can also be caused by loose connectors at P1-P4.
154	Y-axis Z Ch Missing	Same as alarm 153.
155	Z-axis Z Ch Missing	Same as alarm 153.
156	A-axis Z Ch Missing	Same as alarm 153.
157	MOCON Watchdog Fault	The self-test of the MOCON has failed. Replace the MOCON.
158	Video/Keyboard PCB Failure	Internal circuit board problem. The VIDEO PCB in the processor stack is tested at power-on. This could also be caused by a short in the front panel membrane keypad. Call your dealer.
159	Keyboard Failure	Keyboard shorted or button pressed at power on. A power-on test of the membrane keypad has found a shorted button. It can also be caused by a short in the cable from the main cabinet or by holding a switch down during power-on.
160	Low Voltage	The line voltage to control is too low. This alarm occurs when the AC line voltage drops below 190 when wired for 230 volts or drops below 165 when wired for 208 volts.



161	X-Axis Drive Fault	Current in X servo motor beyond limit. Possibly caused by a stalled or overloaded motor. The servos are turned off. This can be caused by running a short distance into a mechanical stop. It can also be caused by a short in the motor or a short of one motor leads to ground.
162	Y-Axis Drive Fault	Same as alarm 161.
163	Z-Axis Drive Fault	Same as alarm 161.
164	A-Axis Drive Fault	Same as alarm 161.
165	X Zero Ret Margin Too Small	This alarm will occur if the home/limit switches move or are misadjusted. This alarm indicates that the zero return position may not be consistent from one zero return to the next. The encoder Z channel signal must occur between 1/8 and 7/8 revolution of where the home switch releases. This will not turn the servos off but will stop the zero return operation.
166	Y Zero Ret Margin Too Small	Same as alarm 165.
167	Z Zero Ret Margin Too Small	Same as alarm 165.
168	A Zero Ret Margin Too Small	Not normally enabled for A-axis.
169	Spindle Direction Fault	Problem with rigid tapping hardware. The spindle started turning in the wrong direction.
170	Phase Loss	Problem with incoming line voltage between legs L1 and L2. This usually indicates that there was a transient loss of input power to the machine.
171	Rpm Too High To Unclamp	The spindle speed exceeded the max speed allowed in parameter 248 to unclamp.
173	Spindle Ref Signal Missing	The Z channel pulse from the spindle encoder is missing for hard tapping synchronization.
174	Tool Load Exceeded	The tool load monitor option is selected and the maximum load for a tool was exceeded in a feed. This alarm can only occur if the tool load monitor function is installed in your machine.
175	Ground Fault Detected	A ground fault condition was detected in the 115V AC supply. This can be caused by a short to ground in any of the servo motors, the tool change motors, the fans, or the oil pump.
176	Overheat Shutdown	An overheat condition persisted longer than the interval specified by parameter 297 and caused an automatic shutdown.



177	Over Voltage Shutdown	An overvoltage condition persisted longer than the interval specified by parameter 296 and caused an automatic shutdown.
178	Divide by Zero	Software error, or parameters are incorrect. Call your dealer.
179	Low Trans Oil Pressure	
181	Macro not completed-spindle disabled	Macro code operating Haas optional equipment (bar feeder, etc.) was not completed for some reason (ESTOP, RESET, Power Down, etc.). Check optional equipment and run recovery procedure.
182	X Cable Fault	Cable from X-axis encoder does not have valid differential signals.
183	Y Cable Fault	Same as alarm 182.
184	Z Cable Fault	Same as alarm 82.
185	A Cable Fault	Same as alarm 182.
186	Spindle Not Turning	Trying to feed while spindle is in the stopped position.
187	B Servo Error Too Large	Same as alarm 103.
188	B Servo Overload	Same as alarm 108.
189	B Motor Overheat	Same as alarm 135.
190	B Motor Z Fault	Same as alarm 139.
191	B Limit Switch	Same as alarm 145.
192	B Axis Z Ch Missing	Same as alarm 153.
193	B Axis Drive Fault	Same as alarm 161.
194	B Zero Ret Margin Too Small	Same as alarm 165.
195	B Cable Fault	Same as 182.
197	100 Hours Unpaid Bill	Call your dealer.
198	Spindle Stalled	Control senses that no spindle fault has occurred, the spindle is at speed, yet the spindle is not turning. Possibly the belt between the spindle drive motor and spindle has slipped or is broken.
199	Negative RPM	Internal software error; call your dealer.



201	Parameter CRC Error	Parameters lost maybe by low battery. Check for a low battery and low battery alarm.
202	Setting CRC Error	Settings lost maybe by low battery. Check for a low battery and low battery alarm.
203	Lead Screw CRC Error	Lead screw compensation tables lost maybe by low battery. Check for CRC Error low battery and low battery alarm.
204	Offset CRC Error	Offsets lost maybe by low battery. Check for a low battery and low battery alarm.
205	Programs CRC Error	Users program lost maybe by low battery. Check for a low battery and low battery alarm.
206	Internal Program Error	Possible corrupted program. Save all programs to disk, delete all, then reload. Check for a low battery and low battery alarm.
207	Queue Advance Error	Software Error; Call your dealer.
208	Queue Allocation Error	Software Error; Call your dealer.
209	Queue Cutter Comp Error	Software Error; Call your dealer.
210	Insufficient Memory	Not enough memory to store users program. Check the space available in the LIST PROG mode and possibly delete some programs.
211	Odd Prog Block	Possible corrupted program. Save all programs to disk, delete all, then reload.
212	Program Integrity Error	Possible corrupted program. Save all programs to disk, delete all, then reload. Check for a low battery and low battery alarm.
213	Program RAM CRC Error	Electronics fault; Call your dealer.
214	No. of Programs Changed	Indicates that the number of programs disagrees with the internal variable that keeps count of the loaded programs. Call your dealer.
215	Free Memory PTR Changed	Indicates the amount of memory used by the programs counted in the system disagrees with the variable that points to free memory. Call your dealer.
216	Probe Arm Down While Running	Indicates that the probe arm was pulled down while a program was running.
217	X Axis Phasing Error	Error occurred in phasing initialization of brushless motor. This can be caused by a bad encoder, or a cabling error.



218	Y Axis Phasing Error	Same as alarm 217.
219	Z Axis Phasing Error	Same as alarm 217.
220	A Axis Phasing Error	Same as alarm 217.
221	B Axis Phasing Error	Same as alarm 217.
222	C Axis Phasing Error	Same as alarm 217.
223	Door Lock Failure	In machines equipped with safety interlocks, this alarm occurs when the control senses the door is open but it is locked. Check the door lock circuit.
224	X Transition Fault	Illegal transition of count pulses in X axis. This alarm usually indicates that the encoder has been damaged and encoder position data is unreliable. This can also be caused by loose connectors at the MOCON and MOTIF PCB.
225	Y Transition Fault	Same as alarm 224.
226	Z Transition Fault	Same as alarm 224.
227	A Transition Fault	Same as alarm 224.
228	B Transition Fault	Same as alarm 224.
229	C Transition Fault	Same as alarm 224.
231	Jog Handle Transition Fault	Illegal transition of count pulses in jog handle encoder. This alarm usually indicates that the encoder has been damaged and encoder position data is unreliable. This can also be caused by loose connectors.
232	Spindle Transition Fault	Illegal transition of count pulses in spindle encoder. This alarm usually indicates that the encoder has been damaged and encoder position data is unreliable. This can also be caused by loose connectors at the MOCON.
233	Jog Handle Cable Fault	Cable from jog handle encoder does not have valid differential signals.
234	Spindle Enc. Cable Fault	Cable from spindle encoder does not have valid differential signals.
235	Spindle Z Fault	Same as alarm 139.
236	Spindle Motor Overload	This alarm is generated in machines equipped with a Haas vector drive, if the spindle motor becomes overloaded.



237	Spindle Following Error	The error between the commanded spindle speed and the actual speed has exceeded the maximum allowable (as set in Parameter 184).
240	Empty Prog or No EOB	DNC program not found, or no end of program found.
241	Invalid Code	RS-232 load bad. Data was stored as comment. Check the program being received.
242	No End	Check input file for a number that has too many digits.
243	Bad Number	Data entered is not a number.
244	Missing)	Comment must end with a ") ".
245	Unknown Code	Check input line or data from RS-232. This alarm can occur while editing data into a program or loading from RS-232.
246	String Too Long	Input line is too long. The data entry line must be shortened.
247	Cursor Data Base Error	Software Error; Call your dealer.
248	Number Range Error	Number entry is out of range.
249	Prog Data Begins Odd	Possible corrupted program. Save all programs to disk, delete all, then reload.
250	Program Data Error	Same as alarm 249.
251	Prog Data Struct Error	Same as alarm 249.
252	Memory Overflow	Same as alarm 249.
253	Electronics Overheat	This alarm is generated if the control cabinet temperature exceeds 135°F. This can be caused by an electronics problem, high room temperature, or clogged air filter.
254	Spindle Motor Overheat	Motor driving spindle is too hot. This alarm is only generated in machines with a Haas vector drive. The spindle motor temperature sensor sensed a high temperature for greater than 1.5 seconds.
257	Program Data Error	Same as alarm 249.
258	Invalid DPRNT Format	Macro DPRNT statement not structured properly.
259	Bad Language Version	Call your dealer.
260	Bad Language CRC	Indicates FLASH memory has been corrupted or damaged.



261	Rotary CRC Error	Rotary table saved parameters (used by Settings 30, 78) have a CRC error.
262	Parameter CRC Missing	RS-232 or disk read of parameter had no CRC when loading from disk or RS-232.
263	Lead Screw CRC Missing	Lead screw compensation tables have no CRC when loading from disk or RS-232.
264	Rotary CRC Missing	Rotary table parameters have no CRC when loading from disk or RS-232.
265	Macro Variable File CRC Error	Macro variables lost maybe by low battery. Check for a low battery and low battery alarm. Reload the macro variable file.
268	DOOR OPEN @ M95 START	Generated whenever an M95 (Sleep Mode) is encountered and the door is open. The door must be closed in order to start sleep mode.
270	C Servo Error Too Large	Same as alarm 103.
271	C Servo Overload	Same as alarm 108.
272	C Motor Overheat	Same as alarm 135.
273	C Motor Z Fault	Same as alarm 139.
274	C Limit Switch	Same as alarm 145.
275	C Axis Z Ch Missing	Same as alarm 153.
276	C Axis Drive Fault	Same as alarm 161.
277	C Zero Ret Margin Too Small	Same as alarm 165.
278	C Cable Fault	Same as alarm 182.
292	Mismatch Axis with I, K Chamfering	I, (K) was commanded as X axis (Z axis) in the block with chamfering.
293	Invalid I,K or R in G01	The move distance in the block commanded with chamfering, corner R is less than the chamfering, corner R amount.
294	Not G01 after	The command after the block commanded with chamfering, corner R is not Chamfering, Corner R G01.
295	Invalid Move After Chamfering	The command after the block commanded with chamfering, corner R is either missing or wrong. There must be a move perpendicular to that of the chamfering block.



296	Not One Axis Move	Consecutive blocks commanded with chamfering, corner R (i.e., G01 Xb Kk; with Chamfering G01 Zb li). After each chamfering block, there must be a single move perpendicular to the one with chamfering, corner R amount.
302	Invalid R in G02 or G03	Check your geometry. R must be greater than or equal to half the distance from start to end.
303	Invalid X, B, or Z in G02 or G03	Check your geometry.
304	Invalid I, J, or K in G02 or G03	Check your geometry. Radius at start must match radius at end of arc within 0.001 inches (0.01 mm.)
305	Invalid Q in Canned Cycle	Q in a canned cycle must be greater than zero and must be a valid N number.
306	Invalid I, J, K, or Q in Canned Cycle	I , J , K , and Q in a canned cycle must be greater than zero.
307	Subroutine Nesting Too Deep	Subprogram nesting is limited to nine levels. Simplify your program.
308	Invalid Tool Offset	A tool offset not within the range of the control was used.
309	Exceeded Max Feed Rate	Use a lower feed rate.
310	Invalid G Code	G code not defined and is not a macro call.
311	Unknown Code	Possible corruption of memory by low battery. Call your dealer.
312	Program End	End of subroutine reached before M99. Need an M99 to return from sub-routine.
313	No P Code In M97, M98, or G65	Must put subprogram number in P code.
314	Subprogram or Macro Not In Memory	Check that a subroutine is in memory or that a macro is defined.
315	Invalid P Code In M97, M98 or M99	The P code must be the name of a program stored in memory without a decimal point for M98 and must be a valid N number for M99, G70, 71, 72, and 73.
316	X Over Travel Range	X-axis will exceed stored stroke limits. This is a parameter in negative direction and is machine zero in the positive direction. This will only occur during the operation of a user's program.
317	Y Over Travel Range	Same as alarm 316.
318	Z Over Travel Range	Same as alarm 316.
319	A Over Travel Range	Not normally possible with A-axis.



320	No Feed Rate Specified	Must have a valid F code for interpolation functions.
321	Auto Off Alarm	Occurs in debug mode only.
322	Sub Prog Without M99	Add an M99 code to the end of program called as a subroutine.
324	Delay Time Range Error	P code in G04 is greater than or equal to 1000 seconds (over 999999 milliseconds).
325	Queue Full	Control problem; call your dealer.
326	G04 Without P Code	Put a Pn.n for seconds or a Pn for milliseconds.
327	No Loop For M Code Except M97, M98	L code not used here. Remove L Code.
328	Invalid Tool Number	Tool number must be between 1 and the value in Parameter 65.
329	Undefined M Code	That M code is not defined and is not a macro call.
330	Undefined Macro Call	Macro name O90nn not in memory. A macro call definition is in parameters and was accessed by user program but that macro was not loaded into memory.
331	Range Error	Number too large.
332	H and T Not Matched	This alarm is generated when Setting 15 is turned ON and an H code number in a running program does not match the tool number in the spindle. Correct the Hn codes, select the right tool, or turn off Setting 15.
333	X-Axis Disabled	Parameters have disabled this axis. Not normally possible.
334	Y-Axis Disabled	Same as alarm 333.
335	Z-Axis Disabled	Same as alarm 333.
336	A-Axis Disabled	An attempt was made to program the A-axis while it was disabled (DISABLED bit in Parameter 43 set to 1).
337	GOTO or P line Not Found	Subprogram is not in memory, or P code is incorrect. P not found
338	Invalid IJK and XYZ in G02 or G03	There is a problem with circle definition; check your geometry.
339	Multiple Codes	Only one M , X , Y , Z , A , Q , etc. allowed in any block or two G codes in the same group. Two or more I,K , R are commanded in the same block with chamfering, corner rounding



340	Cutter Comp Begin With G02 or G03	Select cutter compensation earlier. Cutter comp. must begin on a linear move.
341	Cutter Comp End With G02 or G03	Disable cutter comp later.
342	Cutter Comp Path Too Small	Geometry not possible. Check your geometry.
343	Display Queue Record Full	A block exists that is too long for displaying queue. Shorten title block.
344	Cutter Comp With G18 and G19	Cutter comp only allowed in XY plane (G17).
345	Invalid R Value in M19 or G105	R value must be positive.
346	Illegal M Code	There was an M85 or M86 commanded. These commands are not allowed while Setting 51 DOOR HOLD OVERRIDE is OFF.
348	Illegal Spiral Motion	Linear axis path is too long. For helical motions, the linear path must not be more than the length of the circular component.
349	Prog Stop W/O Cancel Cutter Comp	Cutter compensation has been cancelled without an exit move. Potential damage to part.
350	Cutter Comp Look Ahead Error	There are too many non-movement blocks between motions when cutter comp is being used. Remove some intervening blocks.
351	Invalid P Code	In a block with G103 (Block Lookahead Limit), a value between 0 and 15 must be used for the P code.
352	Aux Axis Power Off	Aux B , C , U , V , or W axis indicate servo off. Check auxiliary axes. Status from control was OFF.
353	Aux Axis No Home	A ZERO RET has not been done yet on the aux axes. Check auxiliary axes. Status from control was LOSS.
354	Aux Axis Disconnected	Aux axes not responding. Check auxiliary axes and RS-232 connections.
355	Aux Axis Position Mismatch	Mismatch between machine and aux axes position. Check aux axes and interfaces. Make sure no manual inputs occur to aux axes.
356	Aux Axis Travel Limit	Aux axes are attempting to travel past their limits.
357	Aux Axis Disabled	Aux axes are disabled.
358	Multiple Aux Axis	Can only move one auxiliary axis at a time.
359	Invalid I, J Or K In G12 Or G13	Check your geometry.



360	Tool Changer Disabled	Check Parameter 57. Not a normal condition for the Lathe.
361	Gear Change Disabled	Not used.
362	Tool Usage Alarm	Tool life limit was reached. To continue, reset the usage count in the Current Commands display and press RESET.
363	Coolant Locked Off	Override is off and program tried to turn on coolant.
364	No Circ Interp Aux Axis	Only rapid or feed is allowed with aux axes.
366	Cutter Comp Interference	G01 cannot be done with tool size.
367	Cutter Comp Interference	G01 cannot be done with tool size.
368	Groove Too Small	Tool too big to enter cut.
369	Tool Too Big	Use a smaller tool for cut.
372	Tool Change in Canned Cycle	Tool change not allowed while canned cycle is active.
373	Invalid Code in DNC	A code found in a DNC program could not be interpreted because of DNC restrictions.
374	Missing XBZA in G31 or G36	G31 skip function requires an X, B, Z, or A move.
376	No Cutter Comp In Skip	Skip G31 function cannot be used with cutter compensation.
377	No Skip in Graph/Sim	Graphics mode cannot simulate skip function.
378	Skip Signal Found	Skip signal check code was included but skip was found when it was not expected.
379	Skip Signal Not Found	Skip signal check code was included but skip was not found when it was expected.
380	X,B,A Or G49 Not Allowed In G37	G37 may only specify Z-axis and must have tool offset defined.
381	G43,G44 Not Allowed In G36 Or G136	Auto work offset probing must be done without tool offset.
382	D Code Required In G35	A Dnn code is required in G35 in order to store the measured tool diameter.
383	Inch Is Not Selected	G20 was specified but settings have selected metric input.
384	Metric Is Not Selected	G21 was specified but settings have selected inches.



385	Invalid L, P, or R Code in G10	G10 was used to changes offsets but L , P , or R code is missing or invalid.
386	Invalid Address Format	An address A..Z was used improperly.
387	Cutter Comp Not Allowed With G103	If block buffering has been limited, Cutter comp cannot be used
388	Cutter Comp Not Allowed With G10	Coordinates cannot be altered while cutter comp is active. Move G10 outside of cutter comp enablement.
389	G17, G18, G19 Illegal in G68	Planes of rotation cannot be changed while rotation is enabled.
390	No Spindle Speed	S code has not been encountered. Add an S code.
391	Feature Disabled	An attempt was made to use a control feature not enabled by a parameter bit. Set the parameter bit to 1.
392	B Axis Disabled	Same as alarm 333.
393	Invalid Motion in G84 or G184	Rigid Tapping can only be in the Z minus G74 or G84 direction. Make sure that the distance from the initial position to the commanded Z depth is in the minus direction.
394	B Over Travel Range	The tailstock (B-axis) has exceeded it's maximum range of travel.
395	Invalid Code in Canned Cycle	Any canned cycle requiring a PQ path sequence may not have an M code in the same block. That is G70, G71, G72, and G73.
396	Conflicting Axes	An Incremental and Absolute command can not be used in the same block of code. For example, X and U cannot be used in the same block.
397	Invalid D Code	In the context that the D code was used it had an invalid value. Was it positive?
398	Aux Axis Servo Off	Aux. axis servo shut off due to a fault.
399	Invalid U Code	In the context that the U code was used it had an invalid value. Was it positive?
403	RS-232 Too Many Progs	Cannot have more than 200 programs in memory.
404	RS-232 No Program Name	Need name in programs when receiving ALL; otherwise has no way to store them.
405	RS-232 Illegal Prog Name	Check files being loaded. Program name must be Onnnnn and must be at beginning of a block.



406	RS-232 Missing Code	A receive found bad data. Check your program. The program will be stored but the bad data is turned into a comment.
407	RS-232 Invalid Code	Check your program. The program will be stored but the bad data is turned into a comment.
408	RS-232 Number Range Error	Check your program. The program will be stored but the bad data is turned into a comment.
409	RS-232 Invalid N Code	Bad Parameter or Setting data. User was loading settings or parameters and something was wrong with the data.
410	RS-232 Invalid V Code	Bad parameter or setting data. User was loading settings or parameters and something was wrong with the data.
411	RS-232 Empty Program	Check your program. Between % and % there was no program found.
412	RS-232 Unexpected End of Input	Check Your Program. An ASCII EOF code was found in the input data before the complete program was completely received. This is a decimal code 26.
413	RS-232 Load Insufficient Memory	Program received doesn't fit. Check the space available in the LIST PROG mode and possibly delete some programs.
414	RS-232 Buffer Overflow	Data sent too fast to CNC. This alarm is not normally possible as this control can keep up with even 38400 bits per second.
415	RS-232 Overrun	Data sent too fast to CNC. This alarm is not normally possible as this control can keep up with as much as 38400 bits per second.
416	RS-232 Parity Error	Data received by CNC has bad parity. Check parity settings, number of data bits and speed. Also check your wiring.
417	RS-232 Framing Error	Data received was garbled and proper framing bits were not found. One or more characters of the data will be lost. Check parity settings, number of data bits and speed.
418	RS-232 Break	Break condition while receiving. The sending device set the line to a break condition. This might also be caused by a simple break in the cable.
419	Invalid Function For DNC	A code found on input of a DNC program could not be interpreted.
420	Program Number Mismatch	The O code in the program being loaded did not match the O code entered at the keyboard. Warning only.
423	Servo Bar Eob Switch Position Unknown	Place 12 inch standard bar in charging position and run G105 Q5 to set End of Bar Switch Position.



424	Servo Bar Metric Unsupported	Metric mode is currently unsupported. Change setting (9) to inch.
425	Servo Bar Length Unknown	Both the bar length and reference position are unknown. Unload bar, Run G105 Q4 followed by G105 Q2 or Q3.
426	Servo Bar Illegal Code	G105 (feed bar) commanded with an illegal code on block. Legal codes are I,J,K,P,Q,R
428	Servo Bar Switch Failure	One of the switches controlling the Servo Bar failed.
429	Disk Dir Insufficient Memory	Disk memory was almost full when an attempt was made to read the disk directory.
430	Disk Unexpected	Check your program. An ASCII EOF code was found in the input data End of Input before the complete program was received. This is a decimal code 26.
431	Disk No Prog	Need name in programs when receiving ALL; otherwise has no way to store them.
432	Disk Illegal Prog Name	Check files being loaded. Program must be Onnnnn and must be at the beginning of a block.
433	Disk Empty Prog Name	Check your program. Between % and % there was no program found.
434	Disk Load Insufficient Memory	Program received doesn't fit. Check the space available in the LIST PROG mode and possibly delete some programs.
435	Disk Abort	Could not read disk.
436	Disk File Not Found	Could not find disk file.
437	TS Under Shoot	The tailstock did not reach its intended destination point.
438	TS Moved While Holding Part	The tailstock moved more than a preset amount while holding a part (e.g., the part slips in the chuck).
439	TS Found No Part	During an M21 or G01, the tailstock reached the hold point without encountering the part.
440	Servo Bar Max Parts Reached	Job Complete. Reset Current # Parts Run on Servo Bar current commands page.
441	Servo Bar Max Bars Reached	Job Complete. Reset Current # Bars Run on Servo Bar current commands page.



442	Servo Bar Max Length Reached	Job Complete. Reset Current Length Run on Servo Bar current commands page.
443	Servo Bar Already Nested	An Illegal G105 Pnnn was found in cutoff subprogram.
445	Servo Bar Fault	SERVO BAR program error.
446	Servo Bar Bar Too Long	The Bar that was just loaded is longer than the Length of Longest Bar as displayed on the Servo Bar current commands page. The system was unable to accurately measure it.
447	Servo Bar Bar In Way	The end of bar switch was depressed and a load or unload bar was commanded. Remove the bar.
448	Servo Bar Out Of Bars	Add more Bars.
449	Servo Bar Cutter Comp Not Allowed	G105 cannot be executed while cutter compensation is invoked.
450	Bar Feeder Fault	This means that discrete input 1027 (BFSPLK) is too high. See parameter 278 bit 20 CK BF status.
451	Bar Feeder Spindle Interlock	This means that discrete input 1030 (BF FLT) is high. See parameter 278 bit 21 CK BF SP ILK.
452	Servo Bar Gearmotor Timeout	The motor which loads bars and the Push rod did not complete its motion in the allowed time. Check for jammed bars.
453	C Axis Engaged	A spindle command (M14, M41, M42, G05 or G77) was given with the C axis drive engaged. The C axis motormust be disengaged with M155 before a spindle brake or gear change.
454	C-Axis Not Engaged	A command was given to the C-axis without the C-axis engaged. The C-axis drive must be engaged with M154 before commanding the C-axis.
501	Too Many Assignments In One Block	Only one assignment “=” is allowed per block. Divide block in error into multiple blocks.
502	[Or = Not First Term In Expressn	An expression element was found where it was not preceded by “[“ or “=”, that start expressions.
503	Illegal Macro Variable Reference	A macro variable number was used that is not supported by this control, use another variable.
504	Unbalanced Brackets In Expression	Unbalanced brackets, “[“ or ”]”, were found in an expression. Add or delete a bracket.
505	Value Stack Error	The macro expression value stack pointer is in error. Call your dealer.



506	Operand Stack Error	The macro expression operand stack pointer is in error. Call your dealer.
507	Too Few Operands On Stack	An expression operand found too few operands on the expression stack. Call your dealer.
508	Division By Zero	A division in a macro expression attempted to divide by zero. Re-configure expression.
509	Illegal Macro Variable Use	See "MACROS" section for valid variables.
510	Illegal Operator or Function Use	See "MACROS" section for valid operators.
511	Unbalanced Right Brackets	Number of right brackets not equal to the number of left brackets.
512	Illegal Assignment Use	Attempted to write to a read-only macro variable.
513	Var. Ref. Not Allowed With N Or O	Alphabetic addresses N and O cannot be combined with macro variables. Do not declare N#1, etc.
514	Illegal Macro Address Reference	A macro variable was used incorrectly with an alpha address. Same as 513.
515	Too Many Conditionals In a Block	Only one conditional expression is allowed in any WHILE or IF-THEN block.
516	Illegal Conditional Or No Then	A conditional expression was found outside of an IF-THEN, WHILE, or M99 block.
517	Exprsn. Not Allowed With N Or O	A macro expression cannot be concatenated to N or O. Do not declare O[#1], etc.
518	Illegal Macro Exprsn Reference	An alpha address with expression, such as A[#1+#2], evaluated incorrectly. Same as 517.
519	Term Expected	In the evaluation of a macro expression an operand was expected and not found.
520	Operator Expected	In the evaluation of a macro expression an operator was expected and not found.
521	Illegal Functional Parameter	An illegal value was passed to a function, such as SQRT[or ASIN[.



522	Illegal Assignment Var Or Value	A variable was referenced for writing. The variable referenced is read only.
523	Conditional Reqd Prior To THEN	THEN was encountered and a conditional statement was not processed in the same block.
524	END Found With No Matching DO	An END was encountered without encountering a previous matching DO. DO-END numbers must agree.
525	Var. Ref. Illegal During Movement	Variable cannot be read during axis movement.
526	Command Found On DO/END Line	A G-code command was found on a WHILE-DO or END macro block. Move the G-code to a separate block.
527	= Not Expected Or THEN Required	Only one Assignment is allowed per block, or a THEN statement is missing.
528	Parameter Precedes G65	On G65 lines all parameters must follow the G65 G-code. Place parameters after G65.
529	Illegal G65 Parameter	The addresses G, L, N, O, and P cannot be used to pass parameters.
530	Too Many I, J, or K's in G65	Only 10 occurrences of I, J, or K can occur in a G65 subroutine call. Reduce the I, J, or K count.
531	Macro Nesting Too Deep	Only four levels of macro nesting can occur. Reduce the amount of nested G65 calls.
532	Unknown Code In Pocket Pattern	Macro syntax is not allowed in a pocket pattern subroutine.
533	Macro Variable Undefined	A conditional expression evaluated to an UNDEFINED value, i.e. #0. Return True or False.
534	DO Or END Already In Use	Multiple use of a DO that has not been closed by and END in the same subroutine. Use another DO number.
535	Illegal DPRNT Statement	A DPRNT statement has been formatted improperly, or DPRNT does not begin block.
536	Command Found On DPRNT Line	A G-code was included on a DPRNT block. Make two separate blocks.
537	RS-232 Abort On DPRNT	While a DPRNT statement was executing, the RS-232 communications failed.
538	Matching END Not	A WHILE-DO statement does not contain a matching END statement. Add the proper END statement.



539 Illegal Goto

Expression after GOTO not valid.

540 Macro Syntax Not Allowed

A section of code was interpreted by the control where macro statement syntax is not permitted. In lathe controls, PQ sequences describing part geometry cannot use macro statements in the part path description.

541 Macro Alarm

This alarm was generated by a macro command in a program.

600 Code Not Expected In This Context

During program interpretation, the control found code out of context. This may indicate an invalid address code found in a PQ sequence. It may also indicate faulty memory hardware or lost memory. Look at the highlighted line for improper G-code.

601 Maximum PQ Blocks Exceeded

The maximum number of blocks making up a PQ sequence was exceeded. Currently, no more than 65535 blocks can be between P and Q.

602 Non Monotonous PQ Blocks in X

The path defined by PQ was not monotonic in the X axis. A monotonic path is one which does not change direction starting from the first motion block.

603 Non Monotonous PQ Blocks in Z

The path defined by PQ was not monotonic in the Z axis. A monotonic path is one which does not change direction starting from the first motion block.

604 Non Monotonous Arc In PQ Block

A non-monotonic arc was found in a PQ block. This will occur in PQ blocks within a G71 or G72 if the arc changes it's X or Z direction. Increasing the arc radius will often correct this problem.

605 Invalid Tool Nose Angle

An invalid angle for the cutting tool tip was specified. This will occur in a G76 block if the A address has a value that is not from 0 to 120 degrees.

606 Invalid A Code

An invalid angle for linear interpolation was specified. This will occur in a G01 block if the A address was congruent to 0 or 180 degrees.

607 Invalid W Code

In the context that the W code was used it had an invalid value. Was it positive?



609	Tailstock Restricted Zone	This alarm is caused by an axis moving into the tailstock restricted zone during program execution. To eliminate the problem, change the program to avoid the restricted zone or change Setting 93 or Setting 94 to adjust the restricted zone. To recover, go to jog mode, press RESET twice to clear the alarm, then jog away from the restricted zone.
610	G71/G72 Domain Nesting Exceeded	The number of troughs nested has exceeded the control limit. Currently, no more than 10 levels of trough can be nested. Refer to the explanation of G71 for a description of trough nesting.
611	G71/G72 Type I Alarm	When G71 or G72 is executing and the control detects a problem in the defined PQ path. It is used to indicate which method of roughing has been selected by the control. It is generated to help the programmer when debugging G71 or G72 commands.
		The control often selects Type I roughing when the programmer has intended to use Type II roughing. To select Type II, add R1 to the G71/G72 command block (in YASNAC mode), or add a Z axis reference to the P block (in FANUC mode).
612	G71/G72 Type II Alarm	This alarm is similar to Alarm 611, but indicates that the control has selected Type II roughing.
613	Command Not Allowed In Cutter Comp.	A command (M96, for example) in the highlighted block cannot be executed while cutter comp. is invoked.
614	Invalid Q Code	A Q address code used a numeric value that was incorrect in the context used. Q used to reference tip codes in G10 can be 0...9. In M96 Q can reference only bits 0 to 31. Use an appropriate value for Q
615	No Intersection to	While cutter comp was in effect, a geometry was encountered whose Offsets in CC compensated paths had no solution given the tool offset used. This can occur when solving circular geometries. Correct the geometry or change the tool radius.
616	Canned Cycle Using P & Q is Active	A canned cycle using P & Q is already executing. A canned cycle can not be executed by another PQ canned cycle.
617	Missing Address	This alarm is generated if an address code is missing. This alarm supports G77.
618	INVALID ADDRESS	This alarm is generated if an address code is being used incorrectly. For example, a negative value is being used for an address code that should be positive.
619	Stroke Exceeds Start Position	This alarm is generated by an incorrect G71 or G72 type 2 command. It refers to a stroke in the PQ path of a G71 or G72 type 2 canned cycle has passed the starting point. Try adjusting the starting point in the block before the G71 or G72.



620	C Axis Disabled	Same as alarm 333.
621	C Over Travel Range	Same as alarm 316.
623	Invalid Code In G112	Only G1, G2, G3 and G17 are allowed. G113 cancels G112. Axes X and Y Cartesian coordinate are used for G1,G2, and G3.
629	Exceeded Max Feed Per Rev	This alarm supports G77 and G5. If the alarm is received during a G77, reduce diameter of part or change geometry. If the alarm is received during a G5, reduce X or Z travel.
701	U Servo Error Too Large MOCON2	Same as alarm 103.
702	V Servo Error Too Large Mocon2	Same as alarm 103.
703	W Servo Error Too Large Mocon2	Same as alarm 103.
704	C Servo Error Too Large Mocon2	Same as alarm 103.
705	Tt Servo Error Too Large Mocon2	Same as alarm 103.
706	Ss Servo Error Too Large Mocon2	Same as alarm 103.
707	J Servo Error Too Large Mocon2	Same as alarm 103.
708	S Servo Error Too Large Mocon2	Same as alarm 103.
711	U Servo Overload Mocon2	Same as alarm 108.
712	V Servo Overload Mocon2	Same as alarm 108.
713	W Servo Overload Mocon2	Same as alarm 108.
714	A Servo Overload Mocon2	Same as alarm 108.
715	B Servo Overload Mocon2	Same as alarm 108.
716	C Servo Overload Mocon2	Same as alarm 108.
717	J Servo Overload Mocon2	Same as alarm 108.
718	S Servo Overload Mocon2	Same as alarm 108.
721	U Motor Over Heat Mocon2	Same as alarm 135.
722	V Motor Over Heat Mocon2	Same as alarm 135.



723	W Motor Over Heat Mocon2	Same as alarm 135.
724	A Motor Over Heat Mocon2	Same as alarm 135.
725	B Motor Over Heat Mocon2	Same as alarm 135.
726	C Motor Over Heat Mocon2	Same as alarm 135.
727	J Motor Over Heat Mocon2	Same as alarm 135.
728	S Motor Over Heat Mocon2	Same as alarm 135.
731	U Motor Z Fault Mocon2	Same as alarm 139.
732	V Motor Z Fault Mocon2	Same as alarm 139.
733	W Motor Z Fault Mocon2	Same as alarm 139.
734	A Motor Z Fault Mocon2	Same as alarm 139.
735	B Motor Z Fault Mocon2	Same as alarm 139.
736	C Motor Z Fault Mocon2	Same as alarm 139.
737	J Motor Z Fault Mocon2	Same as alarm 139.
738	S Motor Z Fault Mocon2	Same as alarm 139.
741	U Axis Z Ch Missing Mocon2	Same as alarm 153.
742	V Axis Z Ch Missing Mocon2	Same as alarm 153.
743	W Axis Z Ch Missing Mocon2	Same as alarm 153.
744	A Axis Z Ch Missing Mocon2	Same as alarm 153.
745	B Axis Z Ch Missing Mocon2	Same as alarm 153.
746	C Axis Z Ch Missing Mocon2	Same as alarm 153.
747	J Axis Z Ch Missing Mocon2	Same as alarm 153.
748	S Axis Z Ch Missing Mocon2	Same as alarm 153.
751	U Axis Drive Fault Mocon2	Same as alarm 161.



752	V Axis Drive Fault Mocon2	Same as alarm 161.
753	W Axis Drive Fault Mocon2	Same as alarm 161.
754	A Axis Drive Fault Mocon2	Same as alarm 161.
755	B Axis Drive Fault Mocon2	Same as alarm 161.
756	C Axis Drive Fault Mocon2	Same as alarm 161.
757	J Axis Drive Fault Mocon2	Same as alarm 161.
758	S Axis Drive Fault Mocon2	Same as alarm 161.
761	U Cable Fault Mocon2	Same as alarm 182.
762	V Cable Fault Mocon2	Same as alarm 182.
763	W Cable Fault Mocon2	Same as alarm 182.
764	A Cable Fault Mocon2	Same as alarm 182.
765	B Cable Fault Mocon2	Same as alarm 182.
766	C Cable Fault Mocon2	Same as alarm 182.
767	J Cable Fault Mocon2	Same as alarm 182.
768	S Cable Fault Mocon2	Same as alarm 182.
771	U Phasing Error Mocon2	Same as alarm 217.
772	V Phasing Error Mocon2	Same as alarm 217.
773	W Phasing Error Mocon2	Same as alarm 217.
774	A Phasing Error Mocon2	Same as alarm 217.
775	B Phasing Error Mocon2	Same as alarm 217.
776	C Phasing Error Mocon2	Same as alarm 217.
777	J Phasing Error Mocon2	Same as alarm 217.
778	S Phasing Error Mocon2	Same as alarm 217.



781	U Transition Fault Mocon2	Same as alarm 224.
782	V Transition Fault Mocon2	Same as alarm 224.
783	W Transition Fault Mocon2	Same as alarm 224.
784	A Transition Fault Mocon2	Same as alarm 224.
785	B Transition Fault Mocon2	Same as alarm 224.
786	C Transition Fault Mocon2	Same as alarm 224.
787	J Transition Fault Mocon2	Same as alarm 224.
788	S Transition Fault Mocon2	Same as alarm 224.
791	Comm. Failure With Mocon2	Same as alarm 101.
792	MOCON2 Watchdog Fault	Same as alarm 157.
796	Sub Spindle Not Turning	Same as alarm 186.
797	Sub Spindle Orientation Fault	Spindle did not orient correctly. During a spindle orientation function, the spindle is rotated until the lock pin drops in; but the lock pin never dropped. This can be caused by a trip of circuit breaker CB4, a lack of air pressure, or too much friction with the orientation pin.
900	Manual Parameter Changes	When the operator alters the value of a parameter, alarm 900 "PAR NO xxx HAS CHANGED. OLD VALUE WAS xxx." will be added to the alarm history. When the alarm history is displayed, the operator will be able to see the parameter number and the old value along with the date and time the change was made. Note that this is not a re-settable alarm, it is for information purposes only.
901	Parameter Changes Via Disk Load	This is a new feature. When a parameter file has been loaded from disk, alarm 901 PARAMETERS HAVE BEEN LOADED BY DISK will be added to the alarm history along with the date and time. Note that this alarm is not a re-settable alarm, it is for information purposes only.
902	Parameter Changes Via RS-232 Load	When a parameter file has been loaded via RS-232, alarm 902 PARAMETERS HAVE BEEN LOADED BY RS-232 will be added to the alarm history along with the date and time. Note that this alarm is not a re-settable alarm, it is for information purposes only.
903	Machine Power Up	When the machine is powered up, alarm 903 CNC MACHINE POWERED UP will be added to the alarm history along with the date and time. Note that this alarm is not a re-settable alarm, it is for information purposes only.

End Of List

NOTE: Alarms 1000-1999 are user defined.



ALARMS

SI OPERATOR'S MANUAL
Series

June 2001



 11. OPTIONS

11.1 QUICK CODE	336
11.2 ADVANCED EDITOR	372
11.4 MACROS	383
11.5 AUXILIARY AXIS CONTROL	409
11.6 HYDRAULIC TAILSTOCK OPERATION*	410
11.7 PARTS CATCHER	418
11.8 TOOL PRE-SETTER	420
11.9 AUTOMATIC CHIP CONVEYOR/AUGER	424
11.10 BAR FEEDER WITH THE HAAS LATHE	426
11.11 LIVE TOOLING*	430
11.12 HAAS BOLT-ON TURRET	449
11.13 HIGH PRESSURE COOLANT SYSTEM*	458
11.14 REMOTE JOG HANDLE	460
11.15 LATHE 7,000 RPM SPINDLE*	460
11.16 AUTO AIR JET*	461
11.17 LATHE AUXILIARY FILTER SYSTEM	462
11.18 AUTO DOOR OPTION	463
11.19 C-AXIS	464
11.20 CARTESIAN TO POLAR TRANSFORMATION	466
11.21 8 "M" FUNCTIONS	468
11.22 200 HOUR TRY-OUT FEATURE	469
11.23 ETHERNET	470
11.24 ZIP DRIVE	470
11.25 HIGH INTENSITY LIGHTS	470
11.26 MEMORY LOCK KEY SWITCH	470
11.27 SPINDLE ORIENTATION	470
11.28 SECOND HOME	470

* NOT FIELD INSTALLABLE

**11.1 Quick Code****INTRODUCTION**

This programming option can be activated by contacting your local HAAS dealer.

QUICK CODE is an innovative new way to program CNC machines. It combines the simplicity and flexibility of G code programming with English descriptive sentences to enable even beginning programmers to construct most 2 dimensional parts. Experienced programmers will also love the speed they can now enter programs manually. This is possible because with one menu selection you can replace a large number of individual keystrokes, with just a few. And what if you don't like the way Quick Code is programmed? Simple! You can change it to suit your needs or programming tastes. Make it as complex or simple as you like.

Background

When NC machines were first introduced they had very limited or no memory at all. They were often run from tapes and instructions needed to be as concise as possible. In order to accomplish this a sort of encryptive language evolved which we called G code programming. A command to "TURN OFF COOLANT" which requires 16 letters and spaces is reduced to "M09" which takes only 3 characters. This made tape lengths and memory requirements manageable to say the least. As it evolved, hundreds of instructions and canned cycles were encrypted into G and M code programming. For an experienced programmer, the G codes are actually very easy to use but the learning process requires constant referring back to the manual to figure out which code to use to accomplish the task. And even the most experienced programmers have to admit that every once in a while you forget to put the right "I,K,Q or P's" into say a G83 drilling cycle. Quick Code eliminates this tedium. Simply handle cursor over to the cycle you want and press the write button and all the code you need to complete the cycle is inserted with default values for all necessary "I,K,Q,P's". And you can edit those values to suit your individual needs.

How It Works

Quick Code reverses the G code encryption confusion. On the right side of the screen you have English commands that describe the operation to perform. By selecting the operation and with one button push, the code is inserted in your program on the left side of the screen. A program is constructed by selecting English commands that are then changed over to machine language or G codes. In doing this you will learn quickly the G code format without studying any manual. Another feature is the ability to scan through a program and Quick Code will tell you what all the G and M codes mean, at the bottom of the screen, a great help in learning the code.

An Open System

One of the neatest features of Quick Code is that it is adaptable to the way you program. Everybody programs a little differently and have special preferences, such as, do you rapid to machine home for a tool change or do you define a tool change location that is closer to the part. With Quick Code you can edit the program so that any English command you desire can be matched with any G code to be inserted. Because of this open format we are letting you define innovative new ways to program complex parts using Quick Code.



What it is Not

Quick Code is not a CAD/CAM package for generating complex moves on 3 dimensional parts. With most CAM packages you have to draw a drawing much like you would in AUTO CAD and then indicate the moves around the drawing and finally generate the code through the post processor. Not a simple task. The difference with these packages is that they require training and much like learning a second language you have to have the time and determination to learn them. They have a tremendous amount of power but you do not always need it. Quick Code is a bridge between high end CAD/CAM and slow and cumbersome G code programming. It is our expectation that it can be used by anyone with very minimal training. For most simple parts we believe that Quick Code is an ideal choice.

Conversational Quick code

Conversational Quick Code makes programming with Quick Code even easier. This feature can be used to "prompt" the operator for the information necessary to create a program. Refer to the "Conversational Quick Code" section for a description of how to use this feature.

QUICK CODE TERMINOLOGY

Before describing the Quick Code environment you need to know the terms listed below. Following this brief list is an illustration of the Quick Code display and how the terms are related to the display.

EDIT WINDOW	Portion of the display that shows the currently edited program.
GROUP WINDOW	Portion of the display which presents a list of groups and items.
GROUP	A list of items that usually have something in common so that they can be grouped together.
ITEM	A line of text representing code that can be added to the edit window when it is selected.
HELP WINDOW	Portion of the display which presents user created help, address code help, and warning messages.

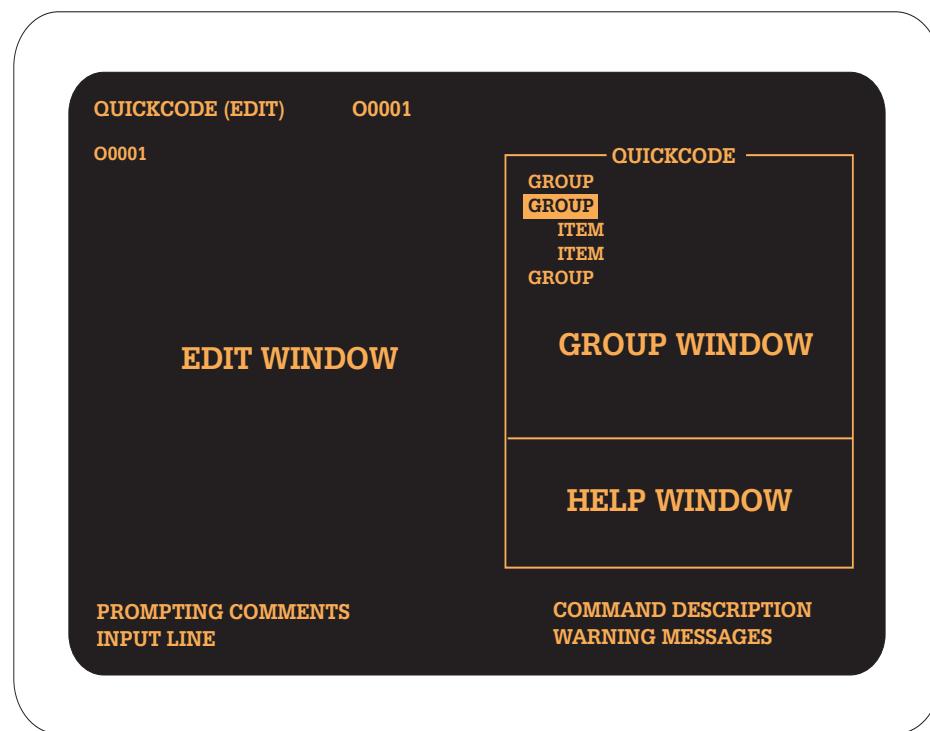


Figure 11.1-1 The Quick Code display.

USAGE AND FEATURES

ACCESSING QUICK CODE

Before Quick Code can be used, the bit labeled ENA Quick Code in parameter 57 must be set to 1. When this bit is set to 0, you will not be able to access the Quick Code screen. Enter Quick Code by selecting edit mode and then pressing the PRGRM/CONVRS key twice. When you press the EDIT key, the 80 column advanced editor is displayed. Then the first press of the PRGRM/CONVRS key will enter you into the 40 column standard editor, whereas the second press of this key will enter the 80 column format of the Quick Code screen. Each additional press of this key will switch between the Advanced Editor, the Standard Editor and Quick Code modes.

THE EDIT WINDOW

The Quick Code edit window is exactly the same as the standard editor that you are used to on the HAAS control. Each time that you select a group item, the edit window will be updated to show you what code has been added to the currently edited program. You have access to all of the edit functions with the exception of the jog handle and the block copy function keys. In the standard editor, you can use the jog handle to traverse program text quickly. While in Quick Code, the jog handle is reserved to maneuver through the group list. You can still cursor through the program text by using the cursor keys provided on the center of the keypad. You are also restricted from using the block copy keys while in Quick Code. For this, you can always switch back to standard edit mode by pressing the PROGRM/CONVRS key. At this point you have access to the jog handle, for long comments, and the block copy functions. Quick Code is not available while in BACKGROUND EDIT MODE.



THE GROUP WINDOW

The group window displays a list of groups that are defined in the Quick Code source file. The groups can be moved through for selection by turning the jog handle in the plus, clockwise, direction. For each jog handle click in the plus direction, the group window cursor will advance to the next group. In this manner you can move through every group in the list. When the last group is highlighted, the next plus click will move the cursor to the first group in the list. To view and cursor through items within a group, turn the jog handle in the minus, counter clockwise, direction. As long as you turn the jog handle in the minus direction the cursor will advance through, display and highlight items in the current group. By turning the jog handle one click clockwise, the group item list will be closed and additional plus clicks will continue to traverse the group list.

THE HELP WINDOW

The help window is just below the group window. It is used to display Quick Code source file help, address code help, and warning messages to the user.

The Quick Code source file can contain comments that will not be placed into the edit window. These comments will be displayed on the first five lines of the help window. These comments are typically used for explaining item code and usage.

As the user cursors through a program, each address code that is highlighted will be interpreted and a short description of its usage is displayed below the help window. This address code help is as accurate as possible. Since the program is not being interpreted sequentially as it is when a program is run, full interpretation cannot take place. When the context of an address code cannot be fully determined, the most likely usage is displayed.

Sometimes during editing we can determine if a run time error will occur without actually running the program. For instance we can tell if multiple codes from one G code group are on a line. In this case Quick Code will display a highlighted warning message to the user indicating that there is a problem. This is found below the help window.

SPECIAL KEYS

Quick Code makes use of the jog handle to select from the group list and group items. This is described in the group window section above. Quick Code action takes place when the WRITE key is pressed. If there is text on the input line, normal text insertion takes place when the WRITE key is pressed. When the input line is blank, pressing the WRITE key will cause Quick Code to take the following action:

- If the currently highlighted Quick Code item is designed as a text help item only, the edit window is not modified.

- If numeric program code associated with the highlighted Quick Code item, the edit window cursor is moved to the end of the current edit block and the associated code is inserted after that block. The edit cursor is left at the end of the last Quick Code block that was inserted.



OPTIONS

SL Series OPERATOR'S MANUAL

June 2001

CONVERSATIONAL QUICK CODE

Quick Code is used to "prompt" the operator for the information necessary to create a program. The "prompting comments" are created by placing a '?' as the first character of a (?comment) in the Quick Code source file (O9999). A comment is any text, up to 34 characters, that is contained in parentheses. When a program is written using Quick Code, the prompting comments will appear on the screen, requiring a response from the operator. The numeric value entered by the operator will be assigned to the G-code item that immediately precedes the prompting comment in the source file. The Quick Code source file program is O09999.

For example, defining an X axis feed move, the following line of code would be in the source file:

G01 X2. (?WHAT IS THE X LOCATION) F.005 (?WHAT IS THE FEED RATE);

This will produce the following prompt when creating a new program, under a different program number (O1234) using Quick Code. And the default X location value will be displayed below with the prompt, as shown:

WHAT IS THE X LOCATION

X2.

and you ENTER a new X location value of 1.25

X1.25

and then the next command prompt comes up with the default feed rate value displayed below the prompt, as shown:

WHAT IS THE FEED RATE

F.005

and you decide to keep this default feed rate value by pressing ENTER

The operator can enter a numeric value and press the WRITE key to change the default feed rate, or simply press the WRITE/ENTER key to accept the default feed rate. The control will wait for an operator response before entering the block to the edit window. Unacceptable responses, such as those containing too many digits or an unnecessary decimal, will cause the control to flash an error message and wait for another response.

Once the operator has input a value for all of the 'variable' G-code items in a block of code, the entire (revised) block is displayed on the input line, as shown:

CORRECT (Y/N) ?

G01 X1.25 F.005 ;



If the block of code is too long to fit on the screen, the operator can scroll to view the entire line using the right or left arrow keys, the HOME key, or the END key. The operator then must enter 'Y' or ENTER to accept the block, or 'N' to cancel it. If it is accepted, the block is written to the edit file, and the Quick Code processing resumes with the next block (if there is one). If it's not accepted, the prompting process is repeated again for the same block.

Pressing the UNDO key while in Quick Code, will exit the current block at any time and remain where you are in the program.

Pressing RESET will exit Quick Code and send the cursor back to the beginning.

A SAMPLE QUICK CODE SESSION

Quick Code program display versions (ver.2.5, ver.2.6 etc.) may vary slightly from control to control.

The following illustrates how Quick Code can be used to build a program. A program will be built to face part and then rough and finish the contour along with spot drill, drill and tap end of part. We will assume that all the appropriate tools are installed. Before you proceed, make sure that Quick Code is enabled in parameter 57. ENA QUIKCODE should be set to 1. You will also need the quick code source program O09999 in the control.

The jog handle is an integral part of using Quick Code and is used quite often. For brevity we use JHCW to mean jog handle clockwise and JHCCW to mean jog handle counter clockwise. For instance, seeing JHCW means that you should turn the jog handle in a clockwise direction.

ENTER A NEW PROGRAM NUMBER

Quick Code will not generate the new program number for you. So the first thing you must do is to select or create the program number you wish to edit with quick code. To create a new program number:

- 1.) Press LIST PROG.
- 2.) Type O00012 (The letter "O" and then any convenient program number)
- 3.) Press WRITE.

This creates a new program in the usual manner. Proceed to edit the program by pressing EDIT. The control will switch to the PROGRAM display and you will see the program number and semicolon at the top left of the screen. You can Select the QUICK CODE menu in the advanced editor menus by pressing the F1 key, and cursor over to the HELP menu and down to the QUICK CODE menu item and press ENTER. Or you can also get to Quick Code by pressing the PRGRM/CONVRS key twice to enter Quick Code.



The following screen is presented.

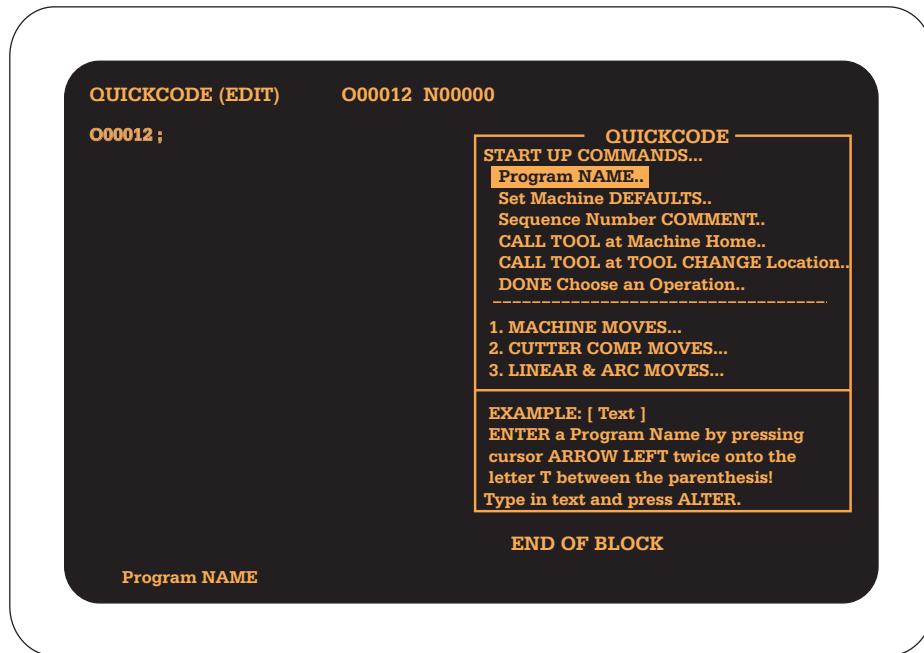


Figure 11.1-2 Empty Program.

STARTUP COMMANDS

- 1.) JHCW until the group titled **STARTUP COMMANDS** in the group window is highlighted.
- 2.) JHCCW one click. The items belonging to STARTUP COMMANDS will appear and the first item, **Program NAME**, is highlighted.
- 3.) Press the WRITE key. This will enter in a (T), for you to cursor arrow left twice onto the letter T in-between the parenthesis, then type in a program name and press ALTER.

The following figure shows what the screen should look like. Note that the cursor will always move to the end of the block that the cursor is sitting on by pressing ENTER to select a menu item. This is where the next block of code will be entered in after. Also note that, for the code that will be entered for each menu selection there is text help information displayed just below the group window. When the Quick Code source file is constructed properly, you will see the actual code and/or text help information that will be helpful in determining which menu item in a group you want.

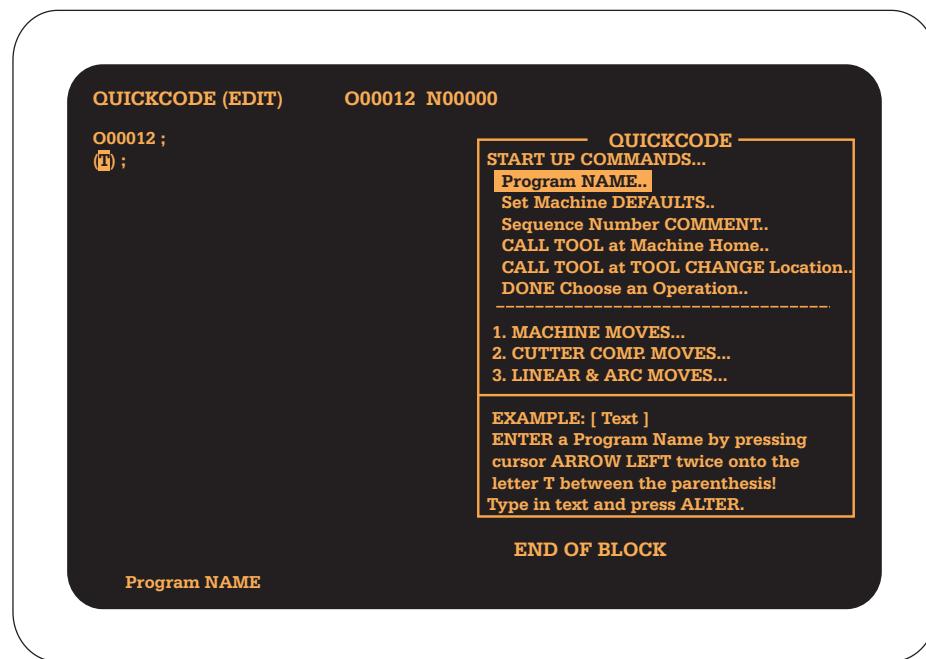


Figure 11.1-3 Program Name

- CALL TOOL 1

- 1.) While on the START UP COMMANDS menu JHCCW and highlight the group item titled **CALL TOOL at Machine Home** then press ENTER.
- 2.) The control querys you for a tool number in your program. The control will be flashing a line of text with a 101 in the lower left corner as the default value. ENTER a new number or use the default number and then press WRITE (or Y for yes) to accept the block listed in the lower left corner. Or N for no, to start the questioning over again.
- 3.) The control will then query you for a Maximum spindle speed. ENTER a new value or except the default value by pressing WRITE (or Y for yes) to accept the block listed in the lower left corner. Or N for no, to start the questioning over again.
- 4.) This same type of querys will happen for the spindle speed, spindle direction, work offset, X axis start position (Enter in 3.1 for X) and Z axis start position (Press ENTER for the default of 1. for Z), and the surface/spindle speed that are used for defining the startup commands for a tool.
- 5.) All of this happens with selecting the **CALL TOOL at Machine Home** menu item to start a tool sequence.

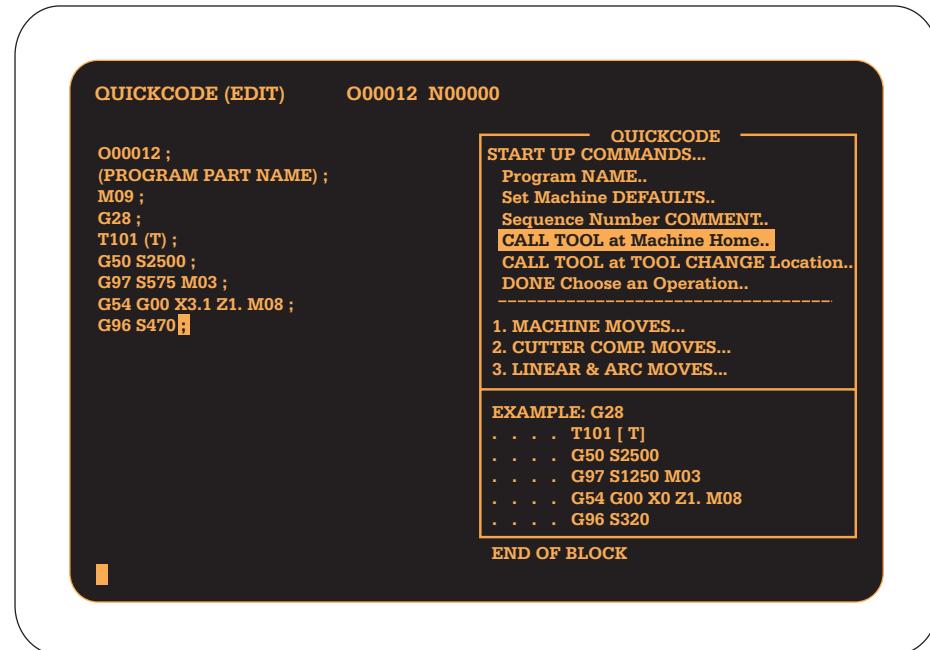


Figure 11.1-4 Call tool at machine home

After pressing WRITE/ENTER you can always edit the program, to make minor adjustments to the code that Quick Code inserts into your program. You do not have to leave the Quick Code display to do this. But you must remember to reposition the cursor back to the block where you want to add the next item. Quick Code will automatically seek the end of the current block that the cursor is on, so there is no need to cursor to the end of the block.

We will assume that the material is a 3.0" dia. stock of aluminum, and we will use the Quick Code menus to create our program. We will use the work coordinate offset G54. The X part zero location is the center of the spindle and Z zero is the face of the part. And you will need to ENTER in the spindle speeds you want for the program being built.

You could have different Quick Code source files for different defaults or menus to select for different materials or part menu selection formats. By changing Parameter 228 (to O09998 or O09997) you can quickly change the source file that Quick Code works with, if you took the time to create another one by duplicating and modifying the O0999 source file (to O09998 or O09997).



QUICKCODE (EDIT) O00012 N00000

```
O00012 ;
(PROGRAM PART NAME) ;
M09 ;
G28 ;
T101 (T) ;
G50 S2500 ;
G97 S575 M03 ;
G54 G00 X3.1 Z1. M08 ;
G96 S470 ;
G00 X3.1 Z0.05 ;
Z0 ;
G01 X-0.062 F0.005 ;
G00 X3.1 Z0.05 ;
```

QUICKCODE

8. MISC. COMMANDS...

Face End of Part..

X Move..

Z Move..

X & Z Move..

FEED or RAPID Command, X Axis..

FEED or RAPID Command, Z Axis..

FEED or RAPID Command, X & Z Axes..

ENDING COMMANDS...

EXAMPLE: G00 X1.6 Z.02

. . . . G00 Z0

. . . . G01 X-.062 F0.005

. . . . G00 X1.6 Z.02

ENTER X & Z Locations to FACE Part.

END OF BLOCK

Figure 11.1-5 Face end of part

- FACING END OF PART

- 1.) JHCW and highlight the group titled **8. MISC. COMMANDS...**
- 2.) JHCCW and highlight the group item titled **Face End of Part..**
- 3.) Press the WRITE key to select the menu to start defining the commands to face end of a part.
- 4.) The control will be flashing with a line of text and 1.6 in the lower left corner as the default X start position above the part. ENTER a new number (Enter in 3.1 for X) and press WRITE (or Y for yes) to except the block listed in the lower left corner. Or N for no, to start the questioning over again.
- 5.) The control will then flash a line of text with 0.05 in the lower left corner as the default Z clearance starting position above the part. Enter a new number or use the default value (Press WRITE for the default of 0.05).
- 6.) The control will then flash with text, Correct (Y/N), in the lower left corner and below that will be G00 X3.1 Z0.05. Press WRITE (or Y for yes) to except, Or N for no, to start the questioning over again.
- 7.) The control will then query you for the Z rapid point to face part. ENTER a new value or press WRITE (or Y for yes) to accept the default (Press WRITE to accept the default of 0.).
- 8.) The control will then flash with text, Correct (Y/N), in the lower left corner and below that will be G00 Z0. and press WRITE (or Y for yes) to accept, Or N for no, to start the questioning over again. (ENTER)
- 9.) The control will then query you for the X point to face down part. Enter a new value or press WRITE (or Y for yes) to accept the default (Press ENTER to accept the default of -.062).
- 10.) The control will then query you for the feed rate to face down the part. Enter a new value or press WRITE (or Y for yes) to accept the default (Press WRITE to accept the default of 0.005).
- 11.) The control will then flash with text, Correct (Y/N), in the lower left corner and below that will be G01 X-0.062 F0.005. Press WRITE (or Y for yes) to except, Or N for no, to start the questioning over again.
- 12.) The control will be flashing with a line of text and 1.6 in the lower left corner as the default X clearance point above the part. ENTER a new number (Enter in 3.1 for X) and press WRITE (or Y for yes) to except the block listed in the lower left corner. Or N for no, to start the questioning over again.
- 13.) The control will then flash a line of text with 0.05 in the lower left corner as the default Z clearance starting position above part. Enter a new number or use the default value (Press WRITE for the default of 0.05).
- 14.) The control will then flash with text, Correct (Y/N), in the lower left corner and below that will be G00 X3.1 Z0.05, and press WRITE (or Y for yes) to accept, Or N for no, to start the questioning over again.

Remember that the cursor will move to the end of the block, when you start selecting Quick Code menu items for your program. This is where the next block of code will be entered in after.



QUICKCODE (EDIT) O00012 N00000

```
O00012 ;
(PROGRAM PART NAME) ;
M09 ;
G28 ;
T101 (T) ;
G50 S2500 ;
G97 S1250 M03 ;
G54 G00 X3.1 Z1. M08 ;
G96 S320 ;
G00 X3.1 Z0.05 ;
Z0 ;
G01 X-0.062 F0.005 ;
G00 X3.1 Z0.05 ;
G71 P10 Q20 U0.01 W0.005 D0.1 F0.012 ;
N10 ;
```

QUICKCODE
START UP COMMANDS...
1. MACHINE MOVES...
2. CUTTER COMP. MOVES...
3. LINEAR & ARC MOVES...
4. MACHINING CYCLES...
FINISHING CYCLE G70..
O.D./I.D. Stock Removal Cycle G71..
END FACE Stock Removal Cycle G72..
IRREGULAR Stock Removal Cycle G73..
ENDING Block # G71, G72 & G73..
O.D/I.D. Turning Cycle G90..

G71 O.D./I.D. Stock Removal Cycle.
Define the Cycle Commands Along
with the Starting Block Number.
Define a Toolpath Start to End with
Menu #3. LINEAR & ARC MOVES.

END OF BLOCK

Figure 11.1-6 OD/ID stock removal cycle G71

By now you should have a good idea of how your program changes after selecting a group item and pressing WRITE. To save space we will not show you each display as a selection is made. Instead we will list the remaining actions needed to finish. The remaining selections are very similar to what we have already done.

CALL G71 TYPE 1 ROUGH CONTOUR

- 1.) JHCW and highlight the group titled **4. MACHINING CYCLES...**
- 2.) JHCCW and highlight the group item titled **O.D./I.D. Stock Removal Cycle G71..** (ENTER)
- 3.) ENTER starting block number (10)
- 4.) ENTER ending block number (20)
- 5.) ENTER stock, plus O.D. minus I.D. (0.01)
- 6.) ENTER face stock allowance (0.005)
- 7.) ENTER depth of cut each pass (0.1)
- 8.) ENTER feed rate (change to 0.012)
- 9.) G71 P10 Q20 U0.01 W0.005 D0.1 F0.012 (ENTER)
- 10.) ENTER starting block number (10) (ENTER)



QUICKCODE (EDIT) O00012 N00000

```

O00012 ;
(PROGRAM PART NAME) ;
M09 ;
G28 ;
T101 (T) ;
G50 S2500 ;
G97 S1250 M03 ;
G54 G00 X3.1 Z1. M08 ;
G96 S320 ;
G00 X3.1 Z0.05 ;
Z0 ;
G01 X-0.062 F0.005 ;
G00 X3.1 Z0.05 ;
G71 P10 Q20 U0.01 W0.005 D0.1 F0.012 ;
N10 ;
G42 G00 X1.8;

```

QUICKCODE

2. CUTTER COMP. MOVES...

C.C. ON X Axis Move with FEED..
C.C. ON Z Axis Move with FEED..
C.C. X & Z Axes Move with FEED..
Cutter Comp. ON X Axis..
Cutter Comp. ON Z Axis..
Cutter Comp. ON X & Z Axes..
Cutter Comp. OFF X Axis..
Cutter Comp. OFF Z Axis..
Cutter Comp. OFF X & Z Axes..

EXAMPLE: G41 G00 X2.5
. . . . or G42 G01 X2.5
ENTER 41=CC Left, or 42=CC Right.
ENTER 0=Rapid or 1=Feed and X Axis Location Move.

END OF BLOCK

Figure 11.1-7 Cutter compensation **ON X-axis**

- 1.) JHCW and highlight the group titled **2. CUTTER COMP. MOVES...**
- 2.) JHCCW and highlight the group item titled **Cutter Comp ON X axis..** (ENTER).
- 3.) ENTER 41=C.C. Left, G42=C.C. Right (42)
- 4.) Enter 0=Rapid or 1=Feed (change to 0)
- 5.) ENTER X point (change to 0.8)
- 6.) G42 G00 X0.8 (ENTER)



QUICKCODE (EDIT) O00012 N00020

```
G00 X3.1 Z0.05 ;
Z0 ;
G01 X-0.062 F0.005 ;
G00 X3.1 Z0.05 ;
G71 P10 Q20 U0.01 W0.005 D0.1 F0.012 ;
N10 ;
G42 G00 X.8 ;
G01 Z0 F0.005 ;
X.9 ;
X1.0 Z-.05 ;
Z-2 ;
X2. Z-2.5 ;
X3. Z-3. ;
```

QUICKCODE
START UP COMMANDS...
1. MACHINE MOVES...
RAPID or FEED X Axis Move..
RAPID or FEED Z Axis Move..
RAPID or FEED X & Z Axes Move..
X Move..
Z Move..
X & Z Move..
LINEAR FEED Command, X Axis..
LINEAR FEED Command, Z Axis..
LINEAR FEED Command, X & Z Axes..

EXAMPLE: X2.125 Z-1.75
ENTER X and Z Axes Location Move.

Figure 11.1-8 Machine moves

- 1.) JHCW and highlight the group titled **1. MACHINE MOVES...**
 - 2.) JHCCW and highlight the group item titled **LINEAR FEED command, Z axis..** is highlighted (ENTER)
 - 3.) ENTER Z point (0.)
 - 4.) ENTER feed rate (0.005)
 - 5.) G01 Z0 F0.005 (ENTER)
-
- 1.) JHCCW and highlight the group item titled **X move..** (ENTER)
 - 2.) ENTER X point (change to .9)
 - 3.) X0.9 (ENTER)
-
- 1.) JHCCW and highlight the group item titled **X & Z move..** (ENTER)
 - 2.) ENTER X point (change to 1.0)
 - 3.) ENTER Z point (change to -0.05)
 - 4.) X1. Z-0.05 (ENTER)
-
- 1.) JHCCW and highlight the group item titled **Z move..** (ENTER)
 - 2.) ENTER Z point (change to -2.0)
 - 3.) Z-2. (ENTER)
-
- 1.) JHCCW and highlight the group item titled **X & Z move..** (ENTER)
 - 2.) ENTER X point (change to 2.0)
 - 3.) ENTER Z point (change to -2.5)
 - 4.) X2. Z-2.5 (ENTER)
-
- 1.) JHCCW and highlight the group item titled **Z move..** (ENTER)
 - 2.) ENTER Z point (change to -3.0)
 - 3.) Z-3. (ENTER)
-
- 1.) JHCCW and highlight the group item titled **X move..** (ENTER)
 - 2.) ENTER X point (change to 3.0)\
 - 3.) X3. (ENTER)



QUICKCODE (EDIT) O00012 N00020

```

G00 X3.1 Z0.05 ;
Z0 ;
G01 X-0.062 F0.005 ;
G00 X3.1 Z0.05 ;
G71 P10 Q20 U0.01 W0.005 D0.1 F0.012 ;
N10 ;
G42 G00 X.8 ;
G01 Z0 F0.005 ;
X.9 ;
X1.0 Z-0.5 ;
Z-.2. ;
X2. Z-2.5 ;
Z-.3. X3. ;
G40 G00 X3.1 ;
N20 ;

```

QUICKCODE
2. CUTTER COMP. MOVES...

- C.C. ON X Axis Move with FEED..
- C.C. ON Z Axis Move with FEED..
- C.C. X & Z Axes Move with FEED..
- Cutter Comp. ON X Axis..
- Cutter Comp. ON Z Axis..
- Cutter Comp. ON, X & Z Axes..
- Cutter Comp. OFF X Axis..**
- Cutter Comp. OFF Z Axis..
- Cutter Comp. OFF X & Z Axes..

EXAMPLE: G40 G00 X2.1
. . . or G40 G01 X2.1
G40 Cancel Cutter Comp. with X Move.
ENTER 0=Rapid or 1=Feed

END OF BLOCK



Figure 11.1-9 Cutter compensation moves

- 1.) JHCW and highlight the group titled **2. CUTTER COMP. MOVES...**
- 2.) JHCCW and highlight the group item titled **Cutter Comp OFF X axis..** (ENTER)
- 3.) Enter 0=Rapid or 1=Feed (change to 0)
- 5.) ENTER X point (change to 3.1)
- 6.) G40 G00 X3.1 (ENTER)

- 1.) JHCW and highlight the group titled **4. MACHINING CYCLES...**
- 2.) JHCCW and highlight the group item titled **ENDING Block # G71, G72 & G73..** (ENTER)
- 3.) ENTER ending block number (20) (ENTER)



QUICKCODE (EDIT) O00012 N00020

```
G00 X3.1 Z0.05 ;
Z0 ;
G01 X-0.062 F0.005 ;
G00 X3.1 Z0.05 ;
G71 P10 Q20 U0.01 W0.005 D0.1 F0.012 ;
N10 ;
G42 G00 X.8 ;
G01 Z0 F0.005 ;
X.9 ;
Z1.01 Z-.05 ;
Z-.2 ;
X2. Z-2.5 ;
Z-3. X3. ;
G40 G00 X3.1 ;
N20 ;
G70 P10 Q20 ;
```

QUICKCODE
4. MACHINING CYCLES...
FINISHING Cycle G70..
O.D./I.D. Stock Removal Cycle G71..
END FACE Stock Removal Cycle G72..
IRREGULAR Stock Removal Cycle G73..
ENDING Block # G71, G72 & G73...
O.D./I.D. Turning Cycle G90..
THREAD CUTTING Cycle G92..
ANOTHER X Dia. Pass, G90 or G92..
END FACE Cutting Cycle G94..
ANOTHER Z Axis Pass, G94

EXAMPLE: G70 P10 Q20
G70 Finishes blocks P thru Q
on part done with G71, G72 or G73.
P### is for starting block N###
Q### is for ending block N###

END OF BLOCK

Figure 11.1-10 Machining cycles

- 1.) JHCW and highlight the group titled **4. MACHINING CYCLES...**
- 2.) JHCCW and highlight the group item titled **FINISHING Cycle G70..** (ENTER)
- 3.) ENTER starting block number (10)
- 4.) ENTER ending block number (20)
- 5.) G70 P10 Q20 (ENTER)

Ending Tool Sequence

- 1.) JHCW and highlight the group titled **ENDING COMMANDS...**
- 2.) JHCCW and highlight the group item titled **END A TOOL Sequence, Rapid HOME..** (ENTER)
- 3.) ENTER Incr. X axis RAPID move (0)
- 4.) ENTER absolute Z axis RAPID move (1.0)
- 5.) G00 U0 Z1. M09 (ENTER)
- 6.) CSS Cancel, ENTER a Spindle Speed (500)
- 7.) G97 S500 (ENTER)



QUICKCODE (EDIT) **O00012 N00020**

```
G70 P10 Q20 ;
M09 ;
G28 ;
T202 (T) ;
G50 S2500 ;
G97 S750 M03 ;
G54 G00 X1.1 Z1. M08 ;
G97 S750 ;
```

QUICKCODE

START UP COMMANDS...

Program NAME..

Set Machine DEFAULTS..

Sequence Number COMMENT..

CALL TOOL at Machine Home..

CALL TOOL at TOOL CHANGE Location..

DONE Choose an Operation..

1. MACHINE MOVES...

2. CUTTER COMP. MOVES...

3. LINEAR & ARC MOVES...

EXAMPLE: G28

```
. . . . T101 [ T ]
. . . . G50 S2500
. . . . G97 S1250 M03
. . . . G54 G00 X0 Z1. M08
. . . . G96 S320
```

Figure 11.1-11 Start up commands

CALL TOOL 2

INVOKE 1.0-12 G76 O.D. THREAD

- 1.) JHCW and highlight the group titled **START UP COMMANDS...**
- 2.) JHCCW and highlight the group item titled **CALL TOOL AT Machine Home..** (ENTER)
- 2.) ENTER tool and offset number (change to 202) (ENTER)
- 3.) ENTER Spindle Speed MAXIMUM (2500) (ENTER)
- 4.) ENTER Spindle Speed (change to 750)
- 5.) ENTER 3=FWD, 4=REV (3) (ENTER)
- 6.) ENTER Work Offset number (54)
- 7.) ENTER START position X axis (change to 1.1)
- 8.) ENTER Z axis START position (1.0) (ENTER)
- 9.) ENTER 96 CSS ON or 97 CSS OFF (change to 97)
- 10.) ENTER Surface or Spindle Speed (change to 750 for a spindle speed) (ENTER)

- 1.) JHCW and highlight the group titled **THREADING CYCLES...**
- 2.) JHCCW and highlight the group item titled **1.0-12 O.D. G76 THREADING CYCLE..** (ENTER)
- 3.) ENTER start position X AXIS (1.1)
- 4.) ENTER Z AXIS start position (0.4)
- 5.) ENTER 23 to angle off, 24 straight off (23)
- 6.) G00 X1.1 Z0.4 M23 (ENTER)

- 1.) ENTER Z thread length (-1.)
- 2.) G76 X0.8978 Z-1. K0.0511 D0.0148 F0.08333 (ENTER)



QUICKCODE (EDIT) O00012 N00020

```
M09 ;  
G28 ;  
T303 (SPOT DRILL) ;  
G50 S2500 ;  
G97 S1000 M03 ;  
G54 G00 X0 Z1. M08 ;  
G97 S1000 ;  
G81 Z-0.15 R0.1 F0.0025 ;
```

QUICKCODE

5. DRILL AND BORE CYCLES...

DRILL G81...

DRILL with Dwell G82..
Deep Hole Peck DRILL G83..
High Speed Peck DRILL G74..
BORE in BORE out G85..
BORE in Rapid out G86..
BORE in Dwell BORE out G89..
CANCEL Canned Cycle G80..

6. TAPPING CYCLES...

EXAMPLE: G81 Z-0.5 R0.1 F0.004
Define X and Z starting location
in the CALL UP A TOOL menu.
ENTER Drill depth and rapid plane.

END OF BLOCK

Figure 11.1-12 Drill and bore cycles

CALL TOOL 3

INVOKE A SPOT DRILL

- 1.) JHCW and highlight the group titled **START UP COMMANDS...**
- 2.) JHCCW and highlight the group item titled **CALL TOOL AT Machine Home..** (ENTER)
- 2.) ENTER tool and offset number (change to 303) (ENTER)
- 3.) ENTER Spindle Speed MAXIMUM (2500) (ENTER)
- 4.) ENTER Spindle Speed (change to 1000)
- 5.) ENTER 3=FWD, 4=REV (3) (ENTER)
- 6.) ENTER Work Offset number (54)
- 7.) ENTER START position X axis (0.)
- 8.) ENTER Z axis START position (1.0) (ENTER)
- 9.) ENTER 96 CSS ON or 97 CSS OFF (change to 97)
- 10.) ENTER Surface or Spindle Speed (change to 1000 for a spindle speed) (ENTER)

- 1.) JHCW and highlight the group titled **DRILL AND BORE CYCLES...**
- 2.) JHCCW and highlight the group item titled **DRILL G81** (ENTER)
- 3.) ENTER the Z drill depth (change to -.15)
- 4.) ENTER Z rapid position before feed (0.1)
- 5.) ENTER feed rate (change to .0025)
- 6.) G81 Z-0.15 R0.1 F0.0025



CALL TOOL 4

INVOKE A DEEP HOLE PECK DRILL

- 1.) JHCW and highlight the group titled **START UP COMMANDS...**
- 2.) JHCCW and highlight the group item titled **CALL TOOL AT Machine Home..** (ENTER)
- 3.) ENTER tool and offset number (change to 404) (ENTER)
- 4.) ENTER Spindle Speed MAXIMUM (2500) (ENTER)
- 5.) ENTER Spindle Speed (change to 1000)
- 6.) ENTER 3=FWD, 4=REV (3) (ENTER)
- 7.) ENTER Work Offset number (54)
- 8.) ENTER START position X axis (0.)
- 9.) ENTER Z axis START position (1.0) (ENTER)
- 10.) ENTER 96 CSS ON or 97 CSS OFF (change to 97)
- 11.) ENTER Surface or Spindle Speed (change to 1000 for a spindle speed) (ENTER)

- 1.) JHCW and highlight the group titled **5. DRILL AND BORE CYCLES...**
- 2.) JHCCW and highlight the group item titled **Deep Hole Peck DRILL G83..** (ENTER)
- 3.) ENTER the Z pecking drill depth (change to -1.15)
- 4.) ENTER peck amount (0.2)
- 5.) ENTER Z rapid position before feed (0.1)
- 6.) ENTER feed rate (change to .003)
- 7.) G81 Z-1.4 Q0.2 R0.1 F0.003 (ENTER)

CALL TOOL 5

INVOKE A 1/4-20 TAP

- 1.) JHCW and highlight the group titled **START UP COMMANDS...**
- 2.) JHCCW and highlight the group item titled **CALL TOOL AT Machine Home..** (ENTER)
- 3.) ENTER tool and offset number (change to 505) (ENTER)
- 4.) ENTER Spindle Speed MAXIMUM (2500) (ENTER)
- 5.) ENTER Spindle Speed (change to 600)
- 6.) ENTER 3=FWD, 4=REV (3) (ENTER)
- 7.) ENTER Work Offset number (54)
- 8.) ENTER START position X axis (0.)
- 9.) ENTER Z axis START position (1.0) (ENTER)
- 10.) ENTER 96 CSS ON or 97 CSS OFF (change to 97)
- 11.) ENTER Surface or Spindle Speed (change to 600 for a spindle speed) (ENTER)

- 1.) JHCW and highlight the group titled **6. TAPPING CYCLES...**
- 2.) JHCCW and highlight the group item titled **20 Pitch TAPPING Cycle G84..** (ENTER)
- 3.) ENTER the Z depth of tapped hole (change to -.65)
- 4.) ENTER Z rapid position before feed (0.3)
- 5.) G84 Z-0.65 R0.3 F0.05 (ENTER)

- 1.) JHCW and highlight the group titled **ENDING COMMANDS...**
- 2.) JHCCW and highlight the group item titled **Rapid HOME change TOOL program END...** (ENTER)
- 3.) ENTER a tool (100) (ENTER)

Done



You now have a ready to run program. You should always verify everything in Graphics Mode to make certain that you have not forgotten any steps. Although this looks like a lot of steps, it is actually very easy once you become familiar with the Quick Code environment.

QUICK CODE EXAMPLE PROGRAM

SUMMARY: G54 OFFSET

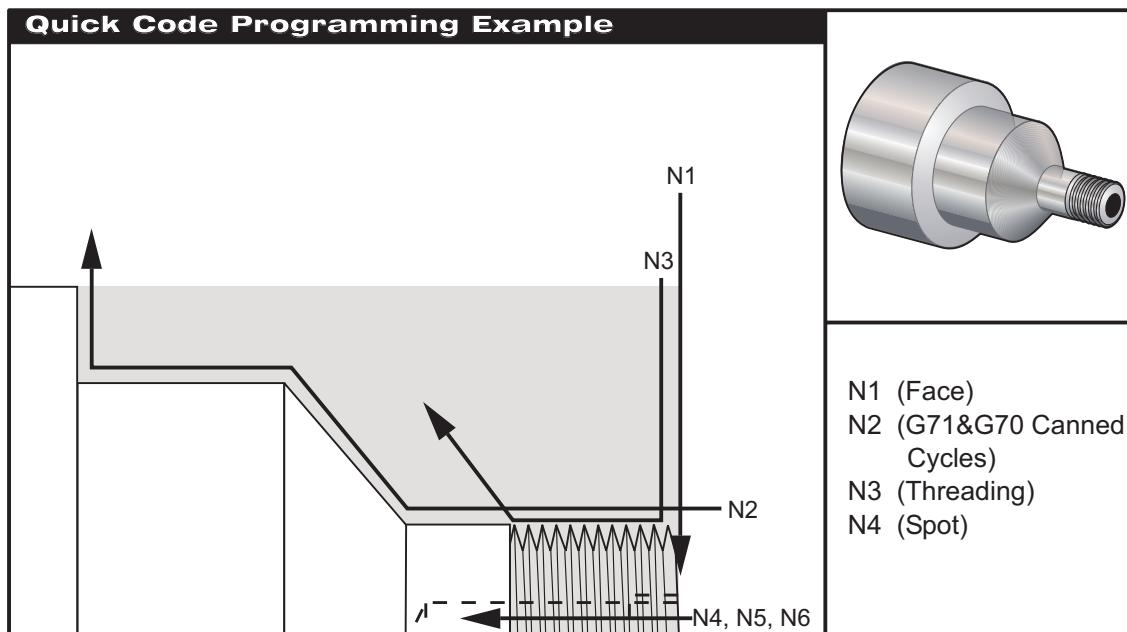
TOOL 1 FACE A 3.0 INCH DIA. AND A G71 TYPE 1 ROUGH CONTOUR (N1) (N2)
TOOL 2 1.0-12 G76 O.D. THREAD (N3)

TOOL 4 SPOT DRILL (N4)

TOOL 5 DEEP HOLE PECK DRILL (N5)

TOOL 6 1/4-20 TAP (N6)

END PROGRAM


EXAMPLE PROGRAM


```

%
O00012
(TYPE IN PROGRAM NAME)
M09
G28
T101 (O.D. TURNING TOOL)          (TOOL 1) (O.D. FACING AND TURNING)
G50 S2500                          (MAX RPM)
G97 S1250 M03
G54 G00 X3.1 Z1. M08
G96 S320
G00 X3.1 Z0.05                    (POSITION ABOVE 3.0 IN. DIA. STOCK TO FACE)
Z0                                 (FACE AT Z0)
G01 X-0.062 F0.005
G00 X3.1 Z0.05
G71 P10 Q20 U0.01 W0.005 D0.1 F0.012 (G71 ROUGH O.D. TURNING CYCLE)
N10                               (N10) (G71 START OF FINISH PART GEOMETRY)
G42 G00 X0.8
G01 Z0 F0.005
X0.9
X1. Z-0.05
Z-2.
X2. Z-2.5
Z-3.
X3.
G40 G00 X3.1
N20                               (N20) (G71 END OF FINISH PATH PART GEOMETRY)
G70 P10 Q20
G00 U0 Z1. M09
G97 S500
G28

```



M01
M09
G28
T202 (O.D. TREADING TOOL) (TOOL 2) (O.D. THREADING)
G50 S2500
G97 S750 M03
G54 G00 X1.1 Z1. M08
G97 S750
(1.0-12 O.D. THREAD)
G00 X1.1 Z0.4 M23
G76 X0.8978 Z-1. K0.0511 D0.0148 F0.08333 (76 THREADING CYCLE TO DO A 1.0-12 O.D. THREAD)
M09
G28
T303 (T1/2 DIA. SPOT DRILL) (TOOL 3)
G50 S2500
G97 S1000 M03
G54 G00 X0 Z1. M08
G97 S1000
G81 Z-0.145 R0.1 F0.0025 (G81 Z-.145 DEPTH DRILLING CANNED CYCLE)
M09
G28
T404 (#7 .201 DIA. DRILL) (TOOL 4)
G50 S2500
G97 S1000 M03
G54 G00 X0 Z1. M08
G97 S1000
G83 Z-1.15 Q0.2 R0.1 F0.003 (G83 Z-1.15 DEEP HOLE PECK DRILLING CYCLE)
M09
G28
T505 (1/4-20 SPIRAL FLUTED TAP) (TOOL 5)
G50 S2500
G97 S600 M03
G54 G00 X0 Z1. M08
G97 S600
G84 Z-0.65 R0.3 F0.05 (G84 Z-.65 TAPPING CYCLE USING A 1/4-20 TAP)
M09
G28
T100
M30
%



THE QUICK CODE SOURCE FILE

All of the text seen in the group window, all of the code associated with items of groups, and much of the help text observed in the help window is contained in a G code program. This program is called the Quick Code source file. With this design, the user can modify Quick Code and tailor it to his specific needs. You can add or change groups and items. The user can develop his own Quick Code file, or program, by editing this file. Dealers can develop new applications and distribute them to their customers. The ability to edit the source file makes Quick Code an extremely flexible tool.

SOURCE FILE PROGRAM DESIGNATION

Program number 9999 is the default Quick Code source file. Every HAAS control equipped with Quick Code comes with a sample O9999 program installed. The default program number can be changed by changing Parameter 228. If the file number in Parameter 228 is not found in the control, the message FILE NOT FOUND is displayed and you will not be able to enter the Quick Code screen. The source file must be formatted as defined below. If program O9999 is not formatted in the appropriate manner then you may not see all, or any, of the defined Quick Code groups. You can use the following skeleton as a start for defining a Quick Code source file.

```
%  
09999 (QUICK CODE - HAAS AUTOMATION INC)  
()  
(ADD ANY COMMENTS HERE THAT PERTAIN TO)  
(THE ENTIRE SOURCE FILE. FOR INSTANCE)  
(YOU CAN RECORD WHO MADE THE FILE, THE)  
(DATE AND TIME OF THE LAST CHANGE, A )  
(VERSION NUMBER, OR ANYTHING ELSE YOU )  
(WANT. ALL COMMENTS PRIOR TO THE FIRST)  
(GROUP ARE NOT SEEN BY THE USER.)  
()  
(QUICK CODE GROUP DEFINITIONS FOLLOW)  
  
.  
  
.  
()  
(END OF QUICK CODE)  
%
```

DEFINING A GROUP IN THE GROUP LIST (*)

To define a group that will show up in the group window, simply enter a comment where the first character is an asterisk. For instance if you want five groups to show up in the group window, then you would include the following five lines in the Quick Code source file.

```
(*GROUP1)  
(*GROUP2)  
(*GROUP3)  
(*GROUP4)  
(*GROUP5)
```

Of course, you can use any descriptive title for the group that is appropriate to what the group will contain. Group titles can be up to 35 characters long. Any additional characters beyond 35 will not be displayed.



GROUP HELP

The first five comments after the group definition will be displayed in the help window. These comments can be used to explain what is contained in the group. For example:

```
(*HELP)
(THIS GROUP CONTAINS HELP ON HOW TO)
(USE QUICK CODE. WHEN THIS GROUP IS)
(HIGHLIGHTED, TURN THE JOG HANDLE IN)
(THE MINUS DIRECTION FOR MORE HELP.)
```

Additional comments beyond five lines are not displayed by Quick Code. This is a method of documenting the source file for the developer of the Quick Code file. Documenting comments can also be hidden in the source file by placing an empty comment after group help comments. In the following example only the first two comments are displayed in the

Help Window.

```
(*HELP)
(ONLY THE FIRST TWO COMMENTS ARE)
(DISPLAYED IN THE HELP WINDOW.)
()
(THIS COMMENT IS NOT DISPLAYED)
```

If more than five lines are required to comment on a group, then you can use several groups to display 5, 10 or 15 lines of help. With this method you can add any amount of information you want about the group that is desired.

GROUP CODE

What happens when a group definition is highlighted and the user presses the WRITE key? If there is a G code after the group definition and before any other group or item definitions, then that G code will be inserted into the program that is being developed. Groups do not have to contain items for generating G code. A group title can stand alone as a code generating entity. The following group definition would add a G28 M30 to the program being developed when WRITE is pressed.

```
(*END OF PROGRAM)
(THIS RETURNS ALL AXES TO MACHINE)
(ZERO AND ENDS PROGRAM EXECUTION)
(G28 M30)
G28 M30
```

Note that the user will not see what G code is generated until the WRITE key is pressed and the code is inserted into the program. For this reason you may want to place the code that is to be generated in a help comment as is done above.



Quick Code can also generate comments in the program being generated. Any comments following an empty comment will be added to the currently edited program. In fact all code following an empty comment is inserted into the program until another empty comment is encountered or until a group or item definition is encountered. The empty comment must be the first code in the block. Any code in the same block as the empty comment is not entered into the program. In the following example, only the code in blocks between the empty comment blocks are added to the program being generated.

```
(*GENERATES COMMENTS AND CODE)
(THIS IS NOT ADDED TO PROGRAM)
()(THIS IS NOT ADDED TO PROGRAM)
(THESE COMMENTS WILL BE ADDED TO THE)
(PROGRAM WHEN THIS GROUP IS)
(HIGHLIGHTED AND WRITE IS PRESSED)
G00 G90 G54 (THIS CODE IS ADDED)
()
(THESE COMMENTS ARE NOT ADDED TO THE)
(PROGRAM BEING GENERATED)
```

DEFINING AN ITEM BELONGING TO A GROUP ()**

To define an item belonging to a group simply enter a comment after a group definition where the first two characters of the comment are asterisks. For instance, the following code generates a group with four subordinate items.

```
(*GROUP)
(**ITEM1)
(**ITEM2)
(**ITEM3)
(**ITEM4)
```

With the above Quick Code source file, only one group is displayed in the group window when the jog handle is turned clockwise. When the jog handle is turned counter clockwise, the five items are displayed and traversed. The item titles are indented one space so that you can differentiate items from groups. Only 34 characters of the item definition comment are displayed in the group window. Additional characters are ignored. The group that items belong to will always be displayed on the screen. The only limit to the number of items in a group is the amount of control memory available.

ITEM HELP

Item help works the same way as group help. The first four comments after the item definition are displayed in the help window. If more than four lines are required, it is recommended that prior items contain the desired comments. In this case instructions would have to be added to indicate which item generates the



G code. For example:

```
(*GROUP)
(**HELP FOR THE FOLLOWING ITEM)
(THESE LINES OF CODE ARE HELP)
(COMMENTS THAT REQUIRE MORE THAN)
(FIVE LINES OF COMMENTARY)
(THIS IS THE LAST LINE OF THIS ITEM)
(**ITEM THAT GENERATES CODE)
(AND HERE WE FINISH THE COMMENTARY)
(FOR CODE GENERATED BY THIS ITEM)
G0 G90 G01 F30
()(******)
(*NEXT GROUP)
```

Although the above example is somewhat awkward, it does provide a method that will satisfy unusual cases. The line with all of the asterisks is legal. It is not inserted into the current program when the WRITE key is pressed. It is used to visually separate groups.

ITEM CODE

Code generated by group items is formatted in the same manner as group code is formatted. Refer to the section on group code for an explanation of how code is generated.

A SAMPLE QUICK CODE SOURCE FILE

After developing or modifying a Quick Code file, it is recommended that you save an off-line copy in a computer. You can keep comments in the Quick Code source file prior to the first group indicating what version the file is and how it differs from other versions. Maintain this program as you would any other G code program in your control with a proper backup scheme. Remember! This file operates the Quick Code feature in your HAAS machine.

A sample Quick Code source file can be found on the floppy disk that comes with the control. It contains many examples of how Quick Code can be used.



11.2 VISUAL QUICK CODE

Visual Quick Code (VQC) is a graphical editor made to help simplify programming for commonly made, simple parts. Given a standard part template and a set of dimensions, a program is created.

Quick Start Guide

1. Either create a new, empty program, or place the cursor at the ";" (End of Block) where the new program will be added. Note: You must be in Advanced Editor.
2. In Edit mode, press the PRGRM/CONVRS key three times to enter VQC. You can also enter VQC by using the pull-down menus in the Advanced Editor under HELP. After entering you will see a mostly empty screen with a list of words or short phrases on the right. These are the part categories.
3. Using the up and down arrow keys, select the part category you want, then press WRITE. Part templates will be seen in the large square area.
4. Using the up, down, left and right arrow keys, select a part template and press WRITE or press CANCEL to return to the category selection screen (step 3). Pressing WRITE (on the part template) will display an enlarged image of the selected part in the large square area including variables identifying the part dimensions.
5. Enter the data for the part. NOTE: Z0 will typically be 0, and the other Z values will typically be negative. R and C values are used to specify the radius or chamfer of a corner.
6. When the last value is entered, the control will ask if all data is correct. Press Y or N. If Y is pressed, the new program will be generated and sent to the Advanced Editor. Check the program that was created, for example, run the program in graphics mode and check the tool paths. Verify the tool offsets, and run the preliminary part using reduced feeds.

VISUAL QUICK CODE INTRODUCTION

Starting

You have the choice of either starting from scratch by creating a new empty program; or use VQC to insert code into an existing program. To insert into an existing program, select the program, enter Advanced Editor and position the cursor at the ";" (end of block) where you want the new code to be inserted **after**.

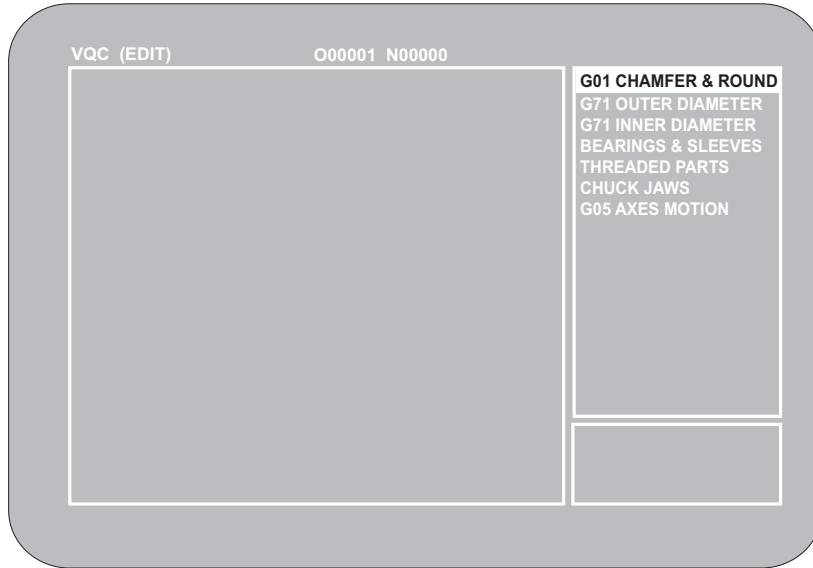
If you choose to start with a new program, VQC will end the program with an M30 (program end and rewind), if it exists in the template.

VQC will not end the code with an M30 if it is inserted into an existing program. Regardless if there is an M30 in the template (this is to prevent unwanted or multiple M30s).

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

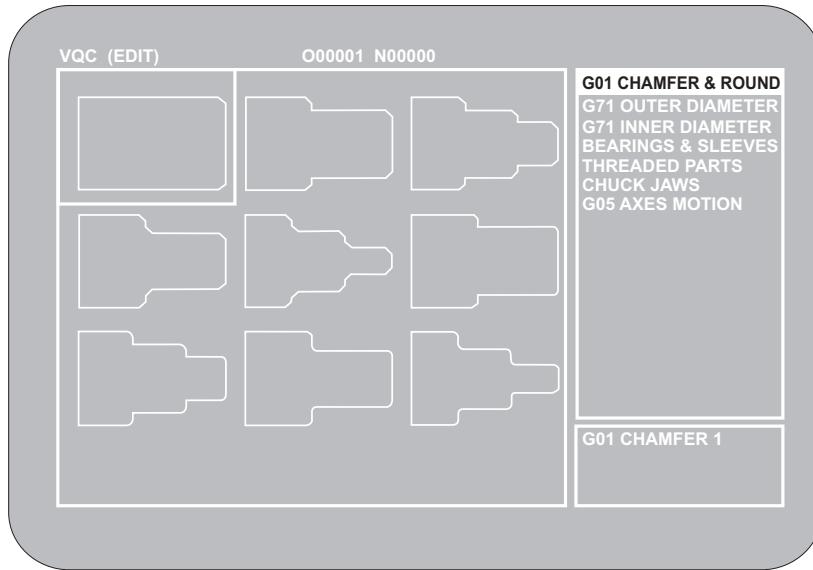


When creating a program with Visual Quick Code an empty program should be selected. Visual Quick Code will add its output code to the selected program.



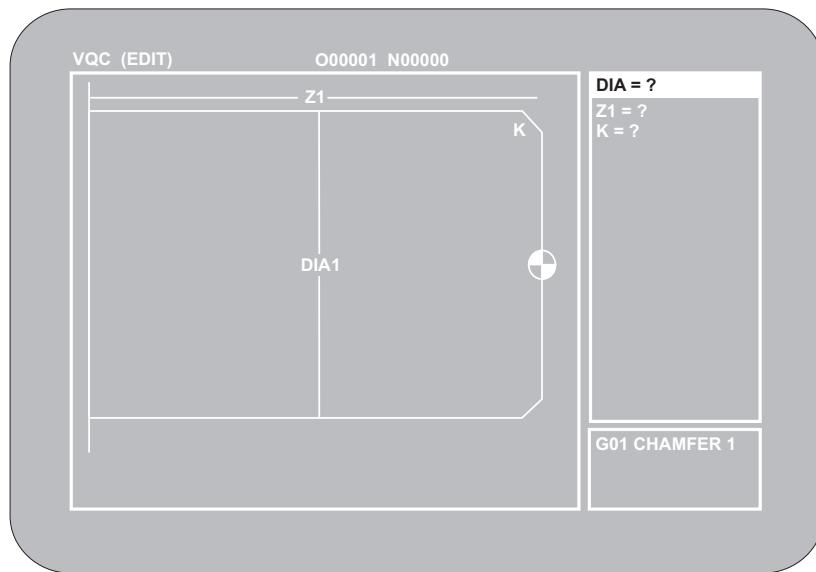
Selecting a Category

Use the arrow keys to select the parts category that most closely matches the desired part and press WRITE. A set of thumbnail illustrations of the parts in that category will appear. These are the part templates for that category.



Selecting a Part Template

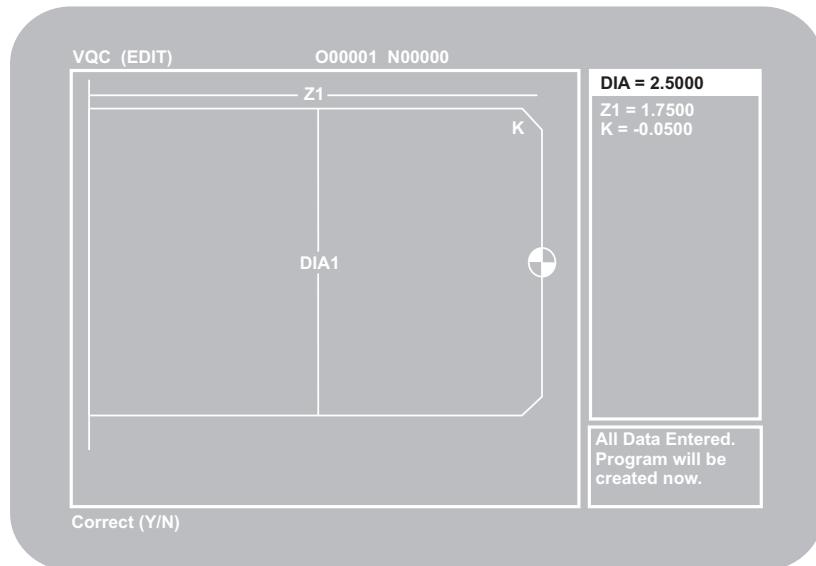
Use the arrow keys to select a template on the page. Pressing WRITE will display an outline of the part and allow the programmer to enter dimensions and other information to make the selected part. Press CANCEL to return to the list of categories.

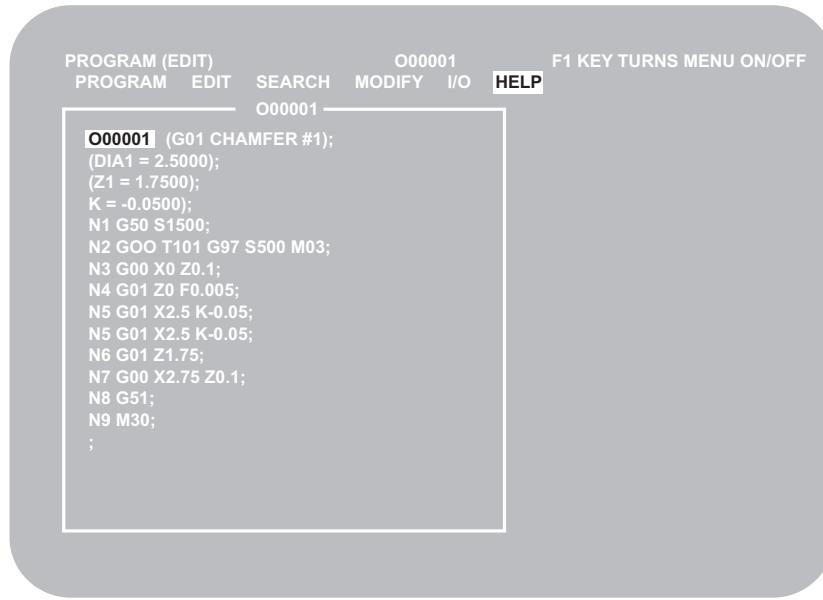


Entering the Data

The control will prompt the programmer for information about the selected part. The data is not checked for consistency, so be careful to enter the numbers correctly. Once the information is entered, the final prompt is: Correct (Y/N). Press Y if the information is correct, or N to go back and edit the data.

After pressing Y, the G-code necessary to produce the part specified will be written to the selected program number. Additionally the program will be put into the Advanced Editor in order to double check the program. Verify the program by first running it in Graphics mode.





Extending Visual Quick Code

The Visual Quick Code system uses program O09997 to generate the icons and questions that the user sees, and the G code that is produced. Program O09997 consists entirely of comments. The comments contain keywords that the Visual Quick Code system understands. Some of the keywords are used to divide program O09997 into sections. The sections are hierarchical, that is the whole program is divided into CATEGORY sections, a CATEGORY section is divided into part TEMPLATE sections, and a TEMPLATE is divided into DIAGRAM, PARAMETER, and CODE sections.

Other keywords are used within sections to set the attributes of the object defined in that section. For example, within the PARAMETER section, we might see the following lines:

```
(LABEL G71 O.D. ROUGHING)
(POSITION [20,6])
```

The first line defines the label, to put on the visual portion of the screen. The label tells the Visual Quick Code system to print anything following the keyword "LABEL" on the screen. The second line with the key word "POSITION" tells the Visual Quick Code where the label will be put onto the screen. The following is a complete list of the keywords used by Visual Quick Code.

**LIST OF KEYWORDS**

Keyword	Definition	Section
CATEGORY	The beginning of a CATEGORY section	
END CATEGORY	The end of a CATEGORY section	
TEMPLATE	The beginning of a TEMPLATE section	Category
END TEMPLATE	The end of a TEMPLATE section	Category
DIAGRAM	The beginning of a DIAGRAM section	Template
END DIAGRAM	The end of a DIAGRAM section	Template
LINE	Specifies a LINE in the DIAGRAM	Diagram
DATUM	Specifies a DATUM in the DIAGRAM	Diagram
ARROW	Specifies an ARROW in the DIAGRAM	Diagram
CW	Specifies a clockwise arc	Diagram
CCW	Specifies a counterclockwise arc	Diagram
THREAD	Specifies a thread in the DIAGRAM	Diagram
PARAMETER	The beginning of a PARAMETER section	Template
END PARAMETER	The end of a PARAMETER section	Template
LABEL	The LABEL attribute of a Parameter	Parameters
NO DECIMAL	Sets the NO DECIMAL attribute	Parameters
ONE PLACE	Sets the ONE PLACE attribute	Parameters
TWO PLACE	Sets the TWO PLACE attribute	Parameters
THREE PLACE	Sets the THREE PLACE attribute	Parameters
FOUR PLACE	Sets the FOUR PLACE attribute	Parameters
GCODE	The beginning of a G-code section	Template
END GCODE	The end of a Template section	Template

The Visual Quick Code system expects the keyword to appear in the Section column. If the keyword appears in a different area, Visual Quick Code will ignore it, or produce an error message because it mistook the keyword for one that it expected, and the text following the keyword did not fit into the Visual Quick Code pattern.

PROGRAMMING VISUAL QUICK CODE

The selections fall into two groups, categories and parts. The programmer first selects, from a list, the type of machining that will be used, for example, drill/tap, boring, threading. This is the category group. Selecting one of these categories displays an illustrated group of parts. The programmer chooses from these illustrations the one that most resembles the desired part. Once chosen the control now prompts the programmer for the dimensions of the parts. Programming code is generated after the programmer enter the dimensions.

Program O09997 is the Visual Quick Code model. The program consists of several Category sections which define the categories available to the programmer.

The following is a basic outline of program O9997 using a top-down approach, becoming more and more specific. This is the way that Visual Quick Code is used. First the user sees a list of categories. After selecting a category, the user sees a list of parts. After selecting a part, the user sees what dimensions he or she can specify, and then the G-code is produced.



%
O09997
(CATEGORY)
...
(END CATEGORY)
(CATEGORY)
...
(END CATEGORY)
(CATEGORY)
...
(END CATEGORY)
%

Each CATEGORY section in turn consists of several TEMPLATE sections. These sections define which parts are available to the user once a category has been selected.

%
O09997
(CATEGORY)
...
(TEMPLATE)
...
(END TEMPLATE)
(TEMPLATE)
...
(END TEMPLATE)
(TEMPLATE)
...
(END TEMPLATE)
(END CATEGORY)
%

Each TEMPLATE section consists of a DIAGRAM section, a PARAMETER section, and a GCODE section. The GCODE section is where the programming code is stored, but is missing some values that are entered by the programmer, via variables.



```
%  
O09997  
(CATEGORY)  
...  
(TEMPLATE)  
...  
(DIAGRAM)  
...  
(END DIAGRAM)  
(PARAMETER)  
...  
(END PARAMETER)  
(GCODE)  
...  
(END GCODE)  
(END TEMPLATE)  
(END CATEGORY)  
%
```

CATEGORY Section

The Category section is a collection of Part Templates. The necessary items are a beginning, a name, templates, and an end. CATEGORY marks the beginning of a Category section, and END CATEGORY marks the end. All of the templates that appear between the beginning and the end of a category belong to that category. NAME (your category name) should be the first line in the category section. The entered name will appear in the list of Visual Quick Code categories (this list appears when Visual Quick Code is first started).

Example:

```
%  
(CATEGORY)  
(NAME: Parts With Threads)  
(TEMPLATE)  
...  
(DIAGRAM)  
...  
(END DIAGRAM)  
(PARAMETER)  
...  
(END PARAMETER)  
(GCODE)  
...  
(END GCODE)  
(END TEMPLATE)  
(END CATEGORY)  
%
```

Part TEMPLATE Section

The Part TEMPLATE section specifies all the information about a typical part. This includes an illustration of the part and what variables can be entered to machine the part.



DIAGRAM Section

The DIAGRAM section is the part of the program that creates the part illustration on the screen. The illustration can be drawn with lines, arcs, and jagged lines that represent threads. This drawing is both for the thumbnail sketch and the full-sized illustration. The labels that appear on the full-sized version of the part are specified in the PARAMETERS section (see Parameters description).

DIAGRAM Coordinates

Each of the elements of the diagram must have a starting point and an ending point. The notation is [X,Y] where X is the horizontal coordinate and Y is the vertical coordinate. The best way to find out what the starting and ending points should be is to use graph paper. First sketch, on the graph paper, what is to appear on the screen. Then pick a point on the graph paper to be the origin, [0,0] (Any point will work, as the diagram will be scaled to fit wherever it is drawn). You can then determine the starting and ending points of all the lines, arcs (CW or CCW) and threads.

DIAGRAM Elements

The elements that make up a diagrams are lines, arcs (CW or CCW) and Threads. For each element, the starting point is specified first, then the ending point.

The format for a line is:

(LINE [X1,Y1] [X2,Y2])

The format for a clockwise arc is:

(CW [X1,Y1] [X2,Y2] r)

where r is the radius of the arc.

The format for a counterclockwise arc is:

(CCW [X1,Y1] [X2,Y2] r)

where r is the radius of the arc.

The format for a jagged line to represent a thread is;

(THREAD [X1,Y1] [X2,Y2])

NOTE: Arcs (CW or CCW) may only cover 180 degrees, or half a circle. If an arc of more than 180 degrees is needed, another arc must be used.



PARAMETERS Section

The PARAMETERS section lists all of the parameters that can be used to customize the standard part. Some of these would be the physical dimensions of the raw material and the part. Others would be tool and offset information, feed rates, and spindle speeds.

Each parameter begins with “#”, which tells Visual Quick Code that value followed by the “#” will be the name for a specific variable. The format is:

```
(PARAMETERS)
(#your variable name)
(END PARAMETERS)
```

After a variable (parameter) has been specified, then any attributes of that particular parameter can be specified.

The POSITION Attribute

If you wish the parameter to appear in the diagram, you must supply a position. The format is (POSITION [X,Y]), where X is the horizontal coordinate and Y is the vertical coordinate. These coordinates are relative to the coordinates specified in the DIAGRAM section. Typically, only physical dimensions of the part will have a POSITION attribute.

Formatting Attributes

Several attributes are used to modify the value entered by the user. This is so that when the PARAMETER is used in the GCODE section, it will appear correctly. The Format column in the following table shows what would result from the G code template X#A, if the user enters 1 when asked for the value of parameter A. If none of the formatting attributes are used, the resulting G code would be X1.

Attribute	Format	Description
(NO DECIMAL)	X1	The value will appear in the final G-code output without a decimal point. Can be used for spindle speeds, tool numbers and offsets.
(ONE PLACE)	X.1	Numbers entered without a decimal point are automatically scaled to tenths.
(TWO PLACE)	X.01	Numbers entered without a decimal point are automatically scaled to hundredths.
(THREE PLACE)	X.001	Numbers entered without a decimal point are automatically scaled to thousandths.
(FOUR PLACE)	X.0001	Numbers entered without a decimal point are automatically scaled to ten-thousandths.

If more than one of these attributes are used with a single parameter, the results are not defined.

NOTE: Do not use more than one formatting attribute for a single parameter.



G CODE Section

The GCODE section is responsible for producing the G code necessary to cut the specified part. Similar to the previous sections of program O09997, the GCODE section consists only of comments. The comments contain standard programming code, just as a user would type it into the editor, except that the end-of-block marker (;) is not used within the comments. The other difference is an extension similar to macro variables: in place of a numeric value, "#" followed by a letter may be entered. The letter represents the variable name of a parameter in the PARAMETERS section.

For example, one of the lines might be:

(X#A)

Which means, "X followed by the number entered for parameter A." For example, if the user entered 3.5 for parameter A, the resulting G-code would be

X3.5;

Remember, the "#letter" combination can be used anywhere a number would be used; this means in expressions, as well as with simple codes. For example, (X [#A - #B]) is valid, as long as both A and B exist in the PARAMETER section.

NOTE: Be sure to use the parameter formatting attributes to make sure the G-code that is produced is valid. For example, "T101.;" is not a valid G-code, because of the decimal point. So if a line in the G-code section reads (T#E), then parameter E must have the NO_DECIMAL attribute set.

EXAMPLE PROGRAM

Below is an example of a simple G71 template for the lathe. This example is a complete O09997 template and is provided to help complete what you have just read.

```
%  
O09997  
(CATEGORY)  
(TEMPLATE)  
(NAME G71 ID W/Radius #1)  
(DIAGRAM)  
(LINE [0,20] [40,20]) (CENTER LINE)  
(DATUM [37,20])  
(LINE [2,27] [2,13])  
(LINE [2,13] [19,13])  
(LINE [19,13] [19,5])  
(LINE [19,5] [37,5])  
(LINE [37,5] [37,9])  
(LINE [37,9] [25,9])  
(CW [25,9] [23,11] 2)  
(LINE [23,11] [23,14])
```



(CCW [23,14] [21,16] 2)

(LINE [21,16] [10,16])

(CW [10,16] [9,17] 1)

(LINE [9,17] [9,23])

(CW [9,23] [10,24] 1)

(LINE [10,24] [21,24])

(CCW [21,24] [23,26] 2)

(LINE [23,26] [23,29])

(CW [23,29] [25,31] 2)

(LINE [25,31] [37,31])

(LINE [37,31] [37,35])

(LINE [37,35] [19,35])

(LINE [19,35] [19,27])

(LINE [19,27] [2,27])

(ARROW [32,21] [32,9]) (DIST1)

(ARROW [32,24] [32,31])

(ARROW [17,21] [17,16]) (DIST2)

(ARROW [17,23] [17,24])

(ARROW [25,27] [23,27]) (Z1)

(ARROW [11,19] [9,19]) (Z2)

(ARROW [21,33] [23,31]) (R1)

(ARROW [21,28] [22,25]) (R2)

(PRINT [3,33] USING TOOL #4)

(END DIAGRAM)

(PARAMETER)

(#DIST1)

(POSITION [30,22])

(#DIST2)

(POSITION [15,22])

(#Z1)

(POSITION [26,28])

(#Z2)

(POSITION [12,19])

(#R1)

(POSITION [20,34])

(#R2)

(POSITION [20,30])

(#IDDEPTH)

(LABEL G71 ID DOC)

(#DRILL)

(LABEL DRILL HOLE DIA)

(END PARAMETER)



OPTIONS

SL Series OPERATOR'S MANUAL

June 2001

(GCODE)
(N1 G28)
(N2 T404)
(N3 G50 S2500)
(N4 G97 S2000 M03)
(N5 G54 G00 X[#DRILL-.1] Z0.1 M08)
(N6 G96 S450)
(N7 G71 P8 Q15 U-0.01 W0.005 D#IDDEPTH F0.012)
(N8 G41 G00 X#DIST1)
(N9 G01 Z0. F0.02)
(N10 Z[#Z1+#R1] F0.005)
(N11 G03 X[#DIST1-#R1*2] Z#Z1 R#R1)
(N12 G01 X[#DIST2+#R2*2])
(N13 G02 X#DIST2 Z[#Z1-#R2] R#R2)
(N14 G01 Z#Z2)
(N15 G40 G00 X.9)
(N16 G70 P8 Q16)
(N17 G97 S2000 M03)
(N18 M09)
(N19 G28)
(N20 M30)
(END GCODE)
(END TEMPLATE)
(END CATEGORY)
%



11.3 ADVANCED EDITOR

The HAAS Advanced Editor gives the user the ability to view and edit two CNC programs at a time. This makes it easier to modify existing programs and to create new ones. The editor has an 80 column display, and includes pull-down menus that allow the user to access the features of the editor. Additionally, a context-sensitive help function is available to provide information on all of the editor's features.

The following terms are used throughout this addendum to describe the advanced editor:

CURRENT PROGRAM	The program that is expected to be run from MEM mode.
ACTIVE PROGRAM	The program that is altered by user input.
INACTIVE PROGRAM	The program opposite the active program in the editor.
CONTEXT-SENSITIVE HELP	A help function that provides information based on what the user is currently doing.
PULL-DOWN MENUS	Menus accessed, or "pulled down", via the "menu bar" at the top left side of the screen. Only one menu can be accessed at any one time. When a menu is pulled down, by pressing F1, menu items appear that can be scrolled through and selected.
HOT KEY	A key that, when pressed, will immediately execute an editor menu item.

The 80 column advanced editor is entered by pressing the EDIT key. The old 40 column editor can be accessed by pressing the PRGRM/CONVRS key. Another press of PRGRM/CONVRS will place the user into the Quick Code display. The Quick Code application can also be accessed from within the F1:HELP pull-down menu. A third press of the PRGRM/CONVRS key will access the advanced editor. The user can alternate between the advanced editor, the 40 column editor, and Quick Code with successive presses of the PRGRM/CONVRS key.

The 80 column advanced editor and the 40 column editor use the same simple edit (Insert, Alter, Delete, Undo) functions. Pressing the F1 key activates a pull-down menu in the advanced editor, which appears descending from the menu bar.

Whenever the pull-down menu system is active, the current menu is pulled down and one item is highlighted. The user can then use the up and down arrow keys to scroll through the items of that menu, or use the left and right arrow keys to open other menus.

When a menu item is highlighted, the user can see a brief description of it in the lower right hand corner. This shows context-sensitive help, which describes what that menu item does. The context-sensitive help explains any prompts that may appear, what keys are available for action, and what the "hot key" (if one exists) is for that menu item.

The UNDO key is used to deactivate the pull-down menu system. Pressing RESET will also deactivate the pull-down menu, but UNDO is preferred. If UNDO is pressed after invoking an executing function from a pull-down menu, it will abort that function. A context-sensitive help session can also be ended by pressing the UNDO key. This will return the user to the active program.



The EDIT key can be used to "switch", left or right, between two programs that have been selected to edit.

Pressing the F4 key will open another copy of the current program in the advanced editor. The user can quickly edit two different locations in the same program by pressing F4, and then using the EDIT key to move back and forth between the two locations. If the user enters 'Onnnnn' and then presses F4, program Onnnnn is opened in the inactive window.

Figure 11.2-1 illustrates the layout of the advanced editor.

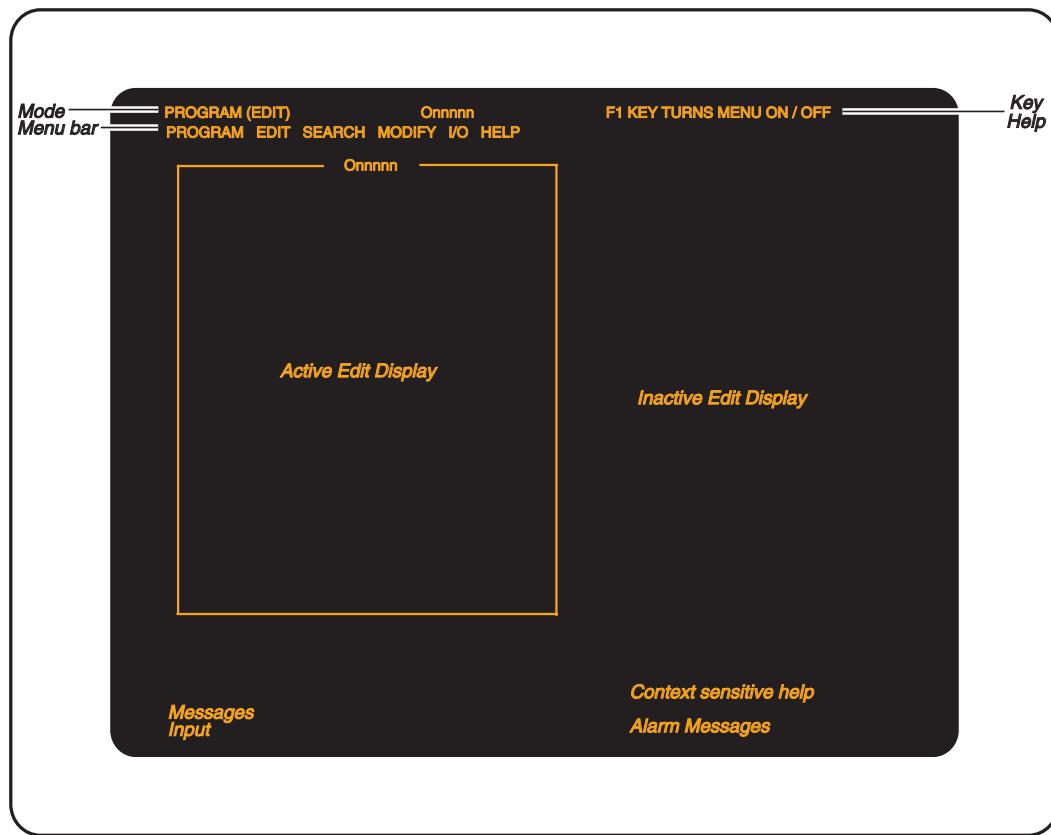


Figure 11.2-1. The advanced editor screen layout.

The advanced editor screen is divided into the following areas:

- **Mode and Control Status** - contain the Current Display page, Operating mode and Control status.
- **Messages** - where control status messages are displayed. This area is used to display prompts for user input in the advanced editor and any alarm messages.



- **Input** - where the user's input is displayed.

- **Menu Bar** - contains the pull-down menu banner.

- **Key Help** - contains short immediate help messages. These are meant to show the user the most important keys that can be used in the current context or operation.

- **Left/Right Side Display** - shows the active and inactive programs. When first entering the editor, the current program will be displayed on the left side and the right side will be blank. The blank area can show another program or can display program lists and help text when the appropriate menu item is selected from the pull-down menu.

- **Context-Sensitive Help** - where context-sensitive help is displayed when you cursor onto a menu item.

FEATURES

This section briefly describes each feature of the advanced editor, each item found in the pull-down menus, and any prompts that might appear.

CONTEXT SENSITIVE HELP

While in the advanced editor, press the F1 key to get into the menus, use the arrow keys or the jog handle to highlight the menu item. The help text for that item will appear in the lower right corner of the screen. PAGE UP and PAGE DOWN keys are used to view the help text. Press UNDO or RESET to exit the menus entirely.

THE PROGRAM MENU

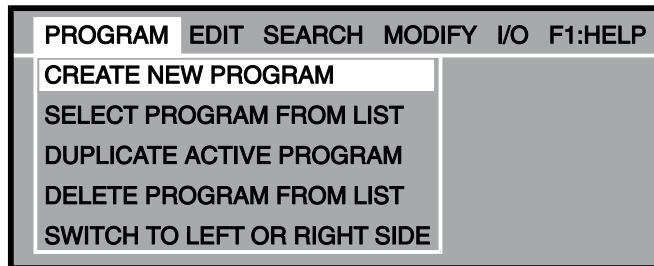


Figure 11.2-2. The PROGRAM menu items.

CREATE NEW PROGRAM

This menu item will create a new program, providing there is room in the program directory and enough memory is available. Enter a program name (Onnnnn) in the range of 0 through 99999 that is not already in the program directory.



SELECT PROGRAM FROM LIST

The HAAS control maintains a directory of programs that the user can select. Select this menu item to edit a program that exists in the directory.

When this menu item is selected, a list of programs is presented for viewing. Scroll through the list by using the cursor keys or the jog handle. Pressing the ENTER key will select the program that is highlighted and will replace the selection list with the selected program. The selected program is now active, and the previously active program will appear on the inactive edit screen.

Program size and memory usage appear at the bottom of the display.

SELECT PROG is the hot key for this item.

DUPLICATE ACTIVE PROGRAM

This menu item will create a new program, copy the contents of the current program into it, rename it as specified, and make it the active program.

ENTER NEW PROGRAM NUMBER: If no program code is present on the input line when this menu item is invoked, this prompt will appear. Enter a valid program number (Onnnnn), then press the ENTER key. Only numeric inputs will be accepted.

DELETE PROGRAM FROM LIST

This menu item will delete a program from the program directory. A list of all programs is presented, with 'ALL' at the end.

To delete a single program, cursor to the program number and press the ENTER key. A prompt will ask for a confirmation of the deletion operation. Enter 'Y' to delete the highlighted program. If any other key is pressed, the program will not be deleted. After a program is deleted, the list of programs will remain in the display.

To delete all programs, cursor to 'ALL' and press ENTER. Confirm deletion of all programs by pressing 'Y'. When all programs are deleted, program O0000 is created, and it is made the active program.

Program size and memory usage are displayed at the bottom of the screen. Press UNDO to exit this menu item and return to the active program.

ERASE PROG is the hot key for this menu item.

SWITCH TO LEFT OR RIGHT SIDE

This will make the active program inactive and the inactive program active. If there is no inactive program, then nothing happens. There are only two possible displays: one on the left and one on the right. The inactive display is used when a second program is selected or created.

The EDIT key is the hot key for this item.



THE EDIT MENU

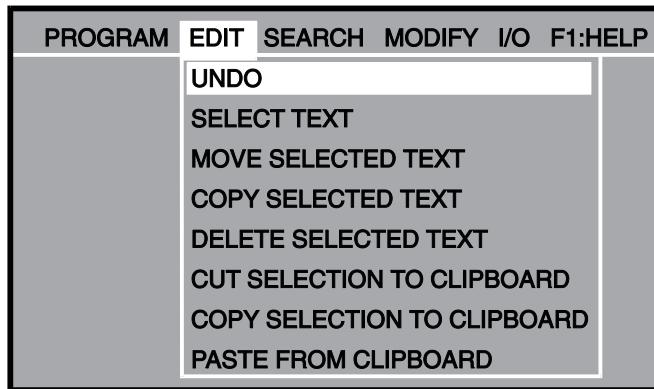


Figure 11.2-3. The EDIT menu items.

UNDO

The last insert, delete, or alter (simple edit) operation will be undone. Pressing UNDO again will restore the previous editing operation, up to the last 9 editing operations. If a block has been selected, choosing this item will simply exit block select mode without undoing anything.

UNDO is the hot key for this item.

SELECT TEXT

This item will set the start point of the block selection. To set the end point, scroll up or down to the desired place, and press the F2 or ENTER key. The selected block will then be highlighted. To deselect the block, press UNDO. This function works the same as in the 40 column editor, except this menu option is used to start selecting text, instead of the F1 key. Either the ENTER or F2 key can be used to end the selection. The following prompt will appear when this item is selected:

SCROLL UP/DOWN, PRESS ENTER OR F2 (to complete the text selection)

MOVE SELECTED TEXT

All selected text will be moved to the line following the current cursor arrow position.

ALTER is the hot key for this menu item

COPY SELECTED TEXT

All selected text will be copied to the line following the current cursor arrow position.

INSERT is the hot key for this menu item.



DELETE SELECTED TEXT

This item deletes any selected block. If no block is selected, the currently highlighted item is deleted. This function works the same as in the 40 column editor, except that if the cursor is in the middle of a comment (between parentheses), it will delete the entire comment instead of just the highlighted character. The UNDO key will restore any deleted comment, or individual commands, but will not restore any blocks of code that were deleted. The DELETE key deletes individual characters from comments.

DELETE is the hot key for this menu item.

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. The program number (8998) for the clipboard is specified by Parameter 226 and can be altered, if necessary.

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. The program number (8998) for the clipboard is specified by Parameter 226 and can be altered, if necessary.

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

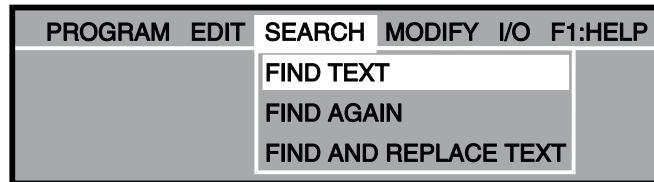


Figure 11.2-4. The SEARCH menu items.

FIND TEXT

This menu item will search for one or more G-Code items in the current program. The search can be performed in either the forward or backward direction from the current cursor location. If the item is found, the cursor will be positioned on it. The following prompts will appear when this menu item is selected:

ENTER TEXT/ITEM TO SEARCH FOR: Type in one or more G-code items, a single address code character, or a comment to be searched for. Press the ENTER key to enter this input. If a full G-Code item is specified, only items that exactly match will be found. If a single address code character is specified, the next matching address code will be found, regardless of the numeric value associated with it. And finally, if a comment is specified, the next comment that contain the block of code specified will be found.



FORWARD OR BACKWARD (F/B) ? Type in either 'F' or 'B'; all other input will be ignored. Entering 'F' will commence the search for the specified G-code item in the forward direction. Entering 'B' will commence the search in the backward direction; i.e. it will find the previous occurrence of the specified code.

FIND AGAIN

This menu item will search the current program for the last block of code that was searched for. It will begin to search at the current cursor location, in the direction that was specified in the previous search. This function will search both selected and unselected blocks.

FIND AND REPLACE TEXT

This menu item will search the current program for one or more occurrences of a specified G-Code item and optionally replace each (or all) with another G-Code item. The search can be performed in either the forward or backward direction from the current cursor location. As each G-Code item is found, the cursor will be positioned on it, and a prompt will ask whether to replace the item, continue the search, both, or neither. This function affects both selected and unselected blocks. The following prompts will appear when this item is selected:

ENTER TEXT/ITEM TO SEARCH FOR: Enter either one or more G-code items, a single letter address code character, or a comment. Press the ENTER key to enter the input. If one or more G-code items are specified, only G-code items that exactly match will be found. If a single letter address code character is specified, all matching address codes will be found, regardless of their associated numeric value. And finally, if a comment is specified, all comments that contain the specified text will be found.

ENTER REPLACEMENT TEXT/ITEM: Enter one or more G-code items that will replace each occurrence of the G-Code item(s) found. If nothing is entered at this prompt, all occurrences of the G-code items found will be deleted, upon verification.

FORWARD OR BACKWARD (F/B) ? Enter either an 'F' or a 'B' ; all other inputs will be ignored. Entering 'F' will commence the search for the specified G-code item in the forward direction. Entering 'B' will commence the search in the backward direction.

REPLACE (YES/NO/ALL/CANCEL) ? As each G-Code item is found, the cursor will be positioned on it, and this prompt will appear. Typing in a 'Y' (for 'Yes') will replace the item, and continue the search in the specified direction. Typing an 'N' (for 'No') will not replace the item, but will continue the search in the specified direction. Typing an 'A' (for 'All') will replace all occurrences of the item with the replacement text, and end the search process. Pressing any key while the editor is "replacing all" will abort the process. Typing a 'C' (for 'Cancel') will abort the search process.

THE MODIFY MENU

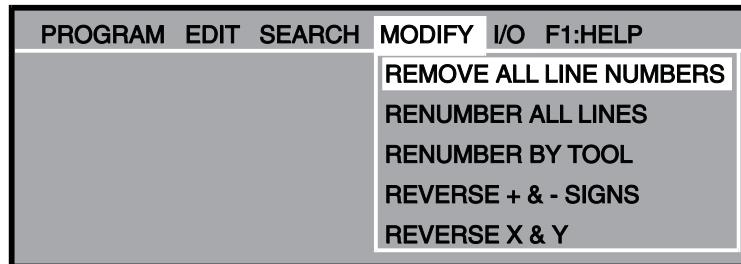


Figure 11.2-5 The MODIFY menu items.



REMOVE ALL LINE NUMBERS

This menu item will automatically remove all unreferenced N-Codes from the edited program. If a block is selected, only the G-Code blocks contained within it will be affected.

RENUMBER ALL LINES

This menu item will either renumber all selected G-Code blocks in the program or, if a block is selected, renumber only those G-Code blocks contained in that block. Below are the prompts that may be encountered while in this item, with a brief explanation of each:

ENTER STARTING N-CODE NUMBER: Type in the starting N-Code number, then press the ENTER key to enter the number. The maximum value accepted is 99999. Any non-numeric input will be ignored.

ENTER N-CODE INCREMENT: Type in the desired numeric difference between consecutive N-Codes, then press the ENTER key to enter the number.

RESOLVE OUTSIDE REFERENCES (Y/N) ? This prompt will appear only if a selected block was defined prior to execution of this menu item. Entering a 'Y' (for Yes) will cause G-Code items outside of the selected block that refer to N-Codes inside the block (such as a GOTO) to be changed to reference the new N-Codes correctly. If an 'N' (for No) is entered, G-Code references that exist outside the selected block will not be changed.

RENUMBER BY TOOL

Searches selected text, or the entire program, for T codes and renames program blocks grouped by T code. The following prompts will appear when this item is selected:

ENTER STARTING N-CODE NUMBER: This prompt will appear when a T code is found. Blocks of code will be renamed, starting with the code entered here, until the next T code is found.

ENTER N-CODE INCREMENT: Each block that is renamed is incremented by the amount entered here.

REVERSE + & - SIGNS

This menu item will reverse the signs of all numeric values associated with one or more address codes in the program. If a certain block is selected, only the address codes in the selected block will be affected. The following prompt will appear when this item is selected:

ENTER ADDRESS CODE(S) TO CHANGE: Type in the valid address code characters whose associated values in the program are to have their signs reversed. They can be entered in any order, but duplicate entries will be ignored. Entries that don't make sense in the context of reversing signs, such as the 'G' character, will also be ignored.



THE I/O MENU

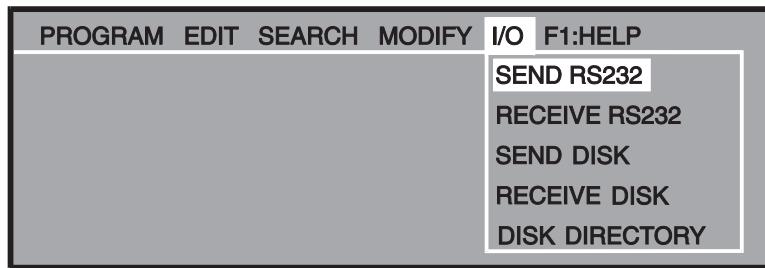


Figure 11.2-6 The I/O menu items.

SEND RS232

This menu will send program(s) that are selected from the program directory to the RS-232 port. When this menu item is selected, a list of all the programs in memory is presented, with 'ALL' at the end.

To select a program, cursor to the program number and press the INSERT key. A highlighted space will appear before the program to indicate it has been selected. Pressing INSERT again will deselect the program, and the highlighted space will disappear. The DELETE key can be used to deselect all selected programs. When the cursor is on "ALL", all programs are selected regardless of highlighting.

To send the selected program(s), press the ENTER key. If more than one program or "ALL" is selected, the data will be sent with one "%" at the beginning of the stream and one at the end.

RECEIVE RS-232

This menu item will receive program(s) from the RS-232 serial port. The program(s) will then be stored in the CNC memory with the corresponding Onnnnn program number(s).

On LISTPROG "ALL" must be first highlighted before using this menu item. The Onnnnn program numbers will be entered automatically from the input stream data. Note, "ALL" must be reselected on the LISTPROG screen after each file transfer.

SEND DISK

This menu item will send program(s) to the floppy. When this menu item is selected, a list of all the programs in memory is presented, with 'ALL' at the end.

To select a program, cursor to the program number and press the INSERT key. A highlighted space will appear before the program to indicate it has been selected. Pressing INSERT again will deselect the program, and the highlighted space will disappear. The DELETE key can be used to deselect all selected programs. When the cursor is on "ALL", all programs are selected regardless of highlighting.



ENTER FLOPPY FILENAME: Type in the desired floppy filename (in standard PC DOS format) for the floppy file being sent, then press the ENTER key. If more than one program or "ALL" is selected, the data will be sent with one "%" at the beginning of the stream and one at the end. If a filename is not entered, the controller will send each selected file separately using the Onnnnn program number as the filename.

RECEIVE FLOPPY

This menu item will receive programs from the floppy. The program(s) will then be stored in the CNC memory with the corresponding Onnnnn program number(s).

ENTER FLOPPY FILENAME: Type in the filename (in standard PC DOS format) of the floppy file being received, then press the ENTER key.

FLOPPY DIRECTORY

This menu item will display the directory of the floppy, with the first entry highlighted. To select a file, press the up and down arrow buttons or use the jog handle. To load a file, select it and press the ENTER key. The G-code programs in the file will be loaded into memory.

If there is insufficient memory for the entire file, Alarm 429 will be generated, and only a partial directory will be displayed.

THE F1:HELP MENU

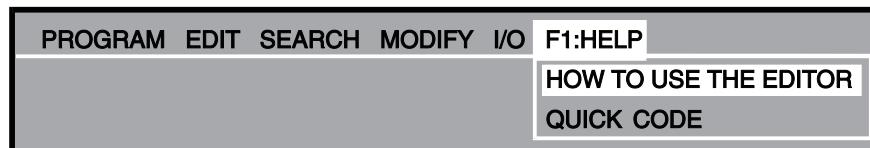


Figure 11.2-7 The F1:HELP menu items.

HOW TO USE THE EDITOR

Selecting this menu item will place an on-line help manual on the inactive screen. This help manual gives a brief description of the editor and its features. The arrow keys, PAGE UP and PAGE DOWN keys, and the jog handle can be used to maneuver through this help manual. Pressing the UNDO key returns the user to the active program.

QUICK CODE

Selecting this menu item will place Quick Code on the inactive side of the editor. All Quick Code functions are now available to the user. Refer to the Operator's Manual for a full description of Quick Code. Pressing the EDIT key will exit Quick Code.


ADVANCED EDITOR SHORTCUTS

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

HOT KEYS**DESCRIPTION OF HOT KEY**

Will quickly bring up the program list on the inactive side of edit display to SELECT PROGRAM FROM LIST.



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



This key can be used to SWITCH TO LEFT OR RIGHT SIDE between two programs that have been selected to edit.



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



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

INSERT can be used to COPY SELECTED TEXT in a program to the line after where you place the cursor arrow point.



ALTER can be used to MOVE SELECTED TEXT in a program to the line after where you place the cursor arrow point.



DELETE can be used to DELETE SELECTED TEXT in a program.



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



Pressing the SEND RS-232 key will activate that I/O menu selection.



Pressing RECV RS-232 key will activate that I/O menu selection.



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

11.4 MACROS**INTRODUCTION**

This control function is optional. If you would like further information on installing this feature please call Haas Automation or your dealer for more information.

This is an introduction to macros as implemented on the HAAS CNC controls. MACROS adds capabilities and flexibility to standard G-code programming that allow the programmer to better define a tool path in a quicker and more natural way. With few exceptions, MACROS, as implemented on the HAAS controls, is compatible with FANUC 10M and 15M controls. Macro features not included in the current release are listed at the end of this section. Programmers already familiar with macro programming will want to review this section in order to avoid unnecessary work.

In traditional CNC programming, a program consists of subroutines that CANNOT be changed or altered except by editing individual values with an editor. MACROS allows the capability to program subroutines where the tool path or location of the tool path is changed, depending on the values contained within variables set by the programmer. These variables can be passed to the subroutine as parameters, or the values can reside in what are called *global variables*.

What this all means is that a programmer can create a collection of subroutines that have been fully debugged. These programs can be used as high level tools that can enhance programmer and machinist productivity. MACROS is not intended to replace modern CAD/CAM software, but it can and has improved machine productivity for those who use it.

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

Tools For Immediate Fixturing

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

1. Determine X and Z coordinates and angle where the clamp is to be placed by jogging the machine to the proposed clamp position and reading the position coordinates from the machine display.
2. Execute the following command in MDI mode:

G65 P2000 X??? Z??? A??? ;

Where ??? are the values determined in Step 1.

Here, macro 2000 (not shown) takes care of all the work since it was designed to drill the clamp bolt hole pattern at the specified angle of A. Essentially, the machinist has created his own custom canned cycle.

Parametric Macro Programming

Parameterized macros can be used to write a single program for a family of parts (parts that are similar but with various dimensions). Only the parameters (dimensions) used when calling the program are changed while the macro program itself remains exactly the same.

This method of programming is best suited for operations that are used often since extra effort is required to write the initial program.

Parametric programming example operations:

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

Automatic Offset Setting Based On The Program

With macros, coordinate offsets can be set in each program so that setup procedures become easier and less error-prone.

Probing

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

1. Profiling of a part to determine unknown dimensions for later machining.
2. Tool calibration for offset and wear values.
3. Inspection prior to machining to determine material allowance on castings.

Macros allow less experienced personnel to operate the machine. Conditions can be detected and custom operator messages or alarms can be displayed on the console to notify the operator.

MACRO SUBROUTINE CALL (G65)

G65 is the command that calls a subroutine with the ability to pass arguments to it. The format follows.

[N#####] G65 P##### [L#####] [arguments] ;

Anything enclosed in brackets is optional. This should not be confused with expression brackets that are explained below. The G65 command requires a **P** address parameter corresponding to any program number currently in memory. When the optional **L** address is used the macro call is repeated the specified number of times.

In Example 1, subroutine 1000 is called once with no parameters passed to the routine. G65 calls are similar to, but not the same as, M98 calls. Up to four G65 calls can be made at the same time (Nesting four deep).

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. Note that G00 and G65 can not be aliased. All other codes between 1 and 255 can be used for aliasing.

Program numbers 9010 through 9019 are reserved for G code aliasing. The following table lists which HAAS parameters are reserved for macro subroutine aliasing.

Haas Parameter	O Code
91	9010
92	9011
93	9012
94	9013
95	9014
96	9015
97	9016
98	9017
99	9018
100	9019

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.

M-Code Aliasing

Haas Parameter	M Macro Call
81	9000
82	9001
83	9002
84	9003
85	9004
86	9005
87	9006
88	9007
89	9008
90	9009



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 you are in metric, the control will assume thousandths (.000). In Example 2 below, local variable #7 will receive .0004.

If a decimal is not included in an argument value, such as:

G65 P9910 A1 B2 C3

The values are passed to macro subroutines according to the following table:

Integer Argument Passing (no decimal point)

Address:	A	B	C	D	E	F	G	H	I	J	K	L	M
Variable:	.001	.001	.001	1.	1.	1.	-	1.	.0001	.0001	.0001	1.	1.
Address:	N	O	P	Q	R	S	T	U	V	W	X	Y	Z
Variable:	-	-	-	.0001	.0001	1.	1.	.0001	.0001	.0001	.0001	.0001	.0001

All 33 local macro variables can be assigned values with arguments by using the alternate addressing method. The following example shows how one could send two sets of coordinate locations to a macro subroutine. Local variables #4 through #9 would be set to .0001 through .0006 respectively.

Example 2: G65 P2000 I1 J2 K3 I4 J5 K6 ;

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

MACRO CONSTANTS

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.

MACRO VARIABLES

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

Variable Usage

All variables are referenced with a number sign (#) followed by a positive number. Examples are: #1, #101, and #501.

Variables are decimal values that are represented as floating point numbers. If a variable has never been used, it can take on a special “undefined” value. This indicates that it has not been used. A variable can be set to undefined with the special variable #0. #0 has the value of undefined or 0.0 depending on the context it is used in. More about this later. 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 constants 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.

Local Variables

Local variables range between #1 and #33. A set of local variables is available at all times. When a call to a subroutine with a G65 command is executed, the local variables are saved and a new set is available for use. This is called “nesting” of the local variables. During a G65 call, all of the new local variables are cleared to undefined values and any local variables that have corresponding address variables in the G65 line are set to the G65 line values. Below is a table of the local variables along with the address variable arguments that change them.



Local Variables and Corresponding Address

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

Note that variables 11, 12, 14..16 and 27..33 do not have corresponding address arguments. They can be set if a sufficient number of I, J and K arguments are used as indicated above in the section about arguments.

Once in the macro subroutine, the local variables can be read and modified by referencing the variable numbers 1..33.

When the L argument is used to do multiple repetitions of a macro subroutine, the arguments are set only on the first repetition. This means that if local variables 1..33 are modified in the first repetition, then the next repetition will have access only to the modified values. Local values are retained from repetition to repetition when the L address is greater than 1.

Calling a subroutine via an 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 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 two ranges: 100..199 and 500..599. The global variables remain in memory when power is turned off. They are not cleared as in the FANUC controls.

System Variables

System variables give the programmer the ability to interact with a variety of control parameters and settings. By setting a system variable, the function of the control can be modified or altered. 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 they can not be modified by the programmer. 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
#800-#999	General purpose variables saved on power off
#1000-#1063	64 discrete inputs (read only)
#1080-#1087	Raw analog to digital inputs (read only)
#1090-#1098	Filtered analog to digital inputs (read only)
#1094	Spindle load with OEM spindle drive (read only)
#1098	Spindle load with Haas vector drive (read only)
#1100-#1139	40 discrete outputs
#1140-#1155	16 extra relay outputs via multiplexed output
#2001-#2099	X axis tool shift offsets
#2101-#2199	Z axis tool shift offsets
#2201-#2299	Tool nose radius offsets
#2301-#2399	Tool tip direction
#2401-#2499	Tool diameter/radius offsets
#2601-#2699	Tool diameter/radius wear
#2701-#2799	X axis tool wear offsets
#2801-#2899	Z axis tool wear offsets
#2901-#2999	Tool nose radius wear offsets
#3000	Programmable alarm
#3001	Millisecond timer
#3002	Hour timer
#3003	Single block suppression
#3004	Override control
#3006	Programmable stop with message
#3011	Year, month, day
#3012	Hour, minute, second
#3020	Power on timer (read only)
#3021	Cycle start timer (read only)
#3022	Feed timer (read only)
#3023	Present part timer (read only)
#3024	Last complete part timer (read only)
#3025	Previous part timer (read only)
#3026	Tool in spindle (read only)
#3027	Spindle RPM (read only)
#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-#5064	Present skip signal position
#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	G110 additional work offsets
#7021-#7026	G111 additional work offsets
#7041-#7046	G112 additional work offsets
#7061-#7066	G113 additional work offsets
#7081-#7086	G114 additional work offsets
#7101-#7106	G115 additional work offsets
#7121-#7126	G116 additional work offsets
#7141-#7146	G117 additional work offsets
#7161-#7166	G118 additional work offsets
#7181-#7186	G119 additional work offsets
#7201-#7206	G120 additional work offsets
#7221-#7226	G121 additional work offsets
#7241-#7246	G122 additional work offsets
#7261-#7266	G123 additional work offsets
#7281-#7286	G124 additional work offsets
#7301-#7306	G125 additional work offsets
#7321-#7326	G126 additional work offsets
#7341-#7346	G127 additional work offsets
#7361-#7366	G128 additional work offsets
#7381-#7386	G129 additional work offsets

SYSTEM VARIABLES IN-DEPTH**1-Bit Discrete Inputs**

For a complete description of discrete inputs, refer to the "Technical Reference" section. Inputs designated as "Spare" can be connected to external devices and used by the programmer.

Haas Lathe Bar Feeder

#709

This variable can be set by a G code program. When the FWD or REV buttons are pressed, or when M03, M04, M133 or M134 is commanded, the control will check the value of macro 749 first. If it is non-zero and parameter 278 bit 3 CK HIDDEN VAR is set to 1, alarm 181 will be generated.

#1032-#1063

These Macro variables support the Haas Bar Feeder. They are available when parameter 209 bit 23 MCD RLY BRD is set to 1.

1-Bit Discrete Outputs

The HAAS control is capable of controlling up to 56 discrete outputs. However, a number of these outputs are already reserved for use by the HAAS controller.

CAUTION! Do not use outputs that are reserved by the system. Using these outputs may result in injury or damage to your equipment.

The user can change the state of these outputs by writing to variables designated as "spare". If the outputs are connected to relays, then an assignment of "1" sets the relay. An assignment of "0" clears the relay.

Referencing these outputs will return the current state of the output and this may be the last assigned value or it may be the last state of the output as set by some user M code. For example, after verifying that output #1108 is "spare":

```
#1108=1;           (Turns #1108 relay on)
#101=#3001+1000; (101 is 1 second from now)
WHILE [[#101 GT #3001] AND [#1109 EQ 0]] D01
END1             (Wait here 1 second or until relay #1109 goes high)
#1108=0;          (Turns #1108 relay off)
```

The number of outputs available to the user and where user M codes are mapped is model dependent. If your control is not equipped with the new M-code relay board, then M21 through M24 will be mapped from #1124-#1127. If you have equipment with the M-code relay board installed. See the 8M option section for information and instructions.

You should always test or dry run programs that have been developed for macros that is running with new hardware.

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 in the lower right hand corner 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 can always be identified in alarm history because the alarm numbers range between 1000 and 1999.

The first 34 characters of the comment will be used for the alarm message.

Timers

HAAS macros supports access to 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 emulate dwell cycles, determine part to part time or wherever time dependent behavior is desired.

- | | |
|-------|--|
| #3001 | MILLISECOND TIMER - The millisecond timer is updated every 20 milliseconds and thus activities can be timed with an accuracy of only 20 milliseconds. At POWER ON, the millisecond timer is reset. The timer has a limit of 497 days. The whole number returned after accessing #3001 represents the number of milliseconds. |
| #3002 | HOUR TIMER - The hour timer is similar to the millisecond timer except that the number returned after accessing #3002 is in hours. The hour and millisecond timers are independent of each other and can be set separately. |

System Overrides

#3003 Variable 3003 is the Single Block Suppression parameter. It overrides the Single Block function in G-code. In the example below, suppression of Single Block is initiated when #3003 is set equal to 1. After M3003 is set =1, each G-code instruction block (lines 2-4) are executed continuously even though the Single Block function is enabled. When #3003 is set equal to zero, the operator of Single Block will resume as normal. That is, the user must press Cycle Start to initiate each new code block (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.;
```

#3004 Variable #3004 is a bitmapped variable that overrides specific control features during runtime.

The first bit disallows FEED HOLD from the keypad. If you do not want feed hold to be executed during any section of code, then bracket that code with assignments to variable #3004. Assigning "1" to #3004 disables the console's feed hold button. Assigning "0" to #3004 re-enables the FEED HOLD button. For example:

Approach code	(FEED HOLD allowed)
#3004=1;	(Disables FEED HOLD button)
Non-stopable 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



Programmable Stop

- #3006 Stops can be programmed. A programmable stop acts like an M00. In the following example, when the assignment statement is executed, the first 15 characters of the comment are displayed in the messaging area on the lower left part of the screen above the command input line. The control stops and waits for a cycle start from the operator. Upon cycle start, operation continues with the next block after the assignment statement.

IF [#1 EQ #0] THEN #3006=101 (ARG.A REQUIRED);

Last Block (MODAL) Group Codes

- #4001-#4020 The grouping of G codes permits more efficient processing. G codes with similar functions are usually under the same group. For instance, G20 and G21 are under group 6. Variables have been set aside to store the last or default G code issued for any of 21 groups. By reading the group code, a macro program can change its behavior based on the contents of the group code. If 4006 contains 21, then a macro program could determine that all moves should be in Metric dimensions rather than Inch dimensions. There is no associated variable for group zero, group zero G codes are NON-modal.

Last Block (MODAL) Address Data

- #4101-#4126 Address codes A..Z (excluding G) are also maintained as modal values. The modal information represented by the last block 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 instance, the value of the previously interpreted **D** address is found in #4107 and the last interpreted **J** value is #4104.

Last Target Position

- #5001-#5006 The final programmed point, target position, for the most recent motion block can be accessed through variables #5001-#5006, X, Z, Y, A, B, and C, respectively. Values are given in the current work coordinate system and can be used while the machine is in motion.

Axis Position Variables

#5020 X-axis	#5023 A-axis	#5026 U-axis
#5021 Y-axis	#5024 B-axis	#5027 V-axis (used for the Haas Bar Feeder)
#5022 Z-axis	#5025 C-axis	

Current Machine Coord Position

- #5021-#5026 The current position in machine coordinates can be obtained through #5021-#5026, X, Z, Y, A, B, and C, respectively. The values CANNOT be read while the machine is in motion. #5022 (Z) represents the value after the tool offset has been applied.

Current Work Coord Position

- #5041-#5046 The current position in the current work coordinates can be obtained through #5041-5046, X, Z, Y, A, B, and C, respectively. The values can NOT be read while the machine is in motion. #5042 (Z) represents the value after the tool offset has been applied.

Current Skip Signal Position

- #5061-#5064 The position where the last skip signal was triggered can be obtained through #5061-#5064, X, Z, Y, and A, respectively. Values are given in the current work coordinate system and can be used while the machine is in motion. #5062 (Z) represents the value after the tool offset has been applied.

Tool Length Compensation

- #5081-#5086 The current total tool length compensation that is being applied to the tool is returned. This includes tool geometry referenced by the current modal value set in the T code plus the wear value.

Offsets

All tool work offsets can be read and set within a macro expression. This allows the programmer to preset coordinates to approximate locations, or to set coordinates to values based upon the results of skip signal locations and calculations. When any of the offsets are read, the interpretation lookahead queue is stopped until that block is executed.

#5201-#5206	G52 X, Z, Y, A, B , C OFFSET VALUES
#5221-#5226	G54 " " " " " "
#5241-#5246	G55 " " " " " "
#5261-#5266	G56 " " " " " "
#5281-#5286	G57 " " " " " "
#5301-#5306	G58 " " " " " "
#5321-#5326	G59 " " " " " "
#7001-#7006	G110 X, Z, Y, A, B, C OFFSET VALUES
#7021-#7026	" " " " " "
#7381-#7386	G129 X, Z, Y, A, B, C OFFSET VALUES

ADDRESS CONSTANT SUBSTITUTION

The usual method of setting control addresses A..Z is by appending a constant to the address. For instance,

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 constants to be replaced with any variable or expression in any section of code (i.e., you do not have to be in a macro subroutine).

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 constant, then the floating point 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 constant, then that address reference is ignored. For example, if #1 is undefined then the block

G00 X1.0 Z#1 ;

becomes

G00 X1.0.

No Z movement takes place.

MACRO STATEMENTS

Macro statements are lines of code that allow the programmer to manipulate the control with features similar to any standard programming language. Included are functions, operators, conditional and arithmetic expressions, assignment statements, and control statements.

Functions and operators are used in expressions to modify variables or values. The operators are essential to expressions while functions make the programmer's job easier.

Functions

Functions are built-in routines that the programmer has available to use. All functions have the form **<function_name> [argument]**. Functions can be passed any expression as arguments. Functions return floating point decimal values. The function provided with the HAAS control are as follows:

FUNCTION	ARGUMENT	RETURNS	NOTES
SIN[]	Degrees	Decimal	Sine
COS[]	Degrees	Decimal	Cosine
TAN[]	Degrees	Decimal	Tangent
ATAN[]	Decimal	Degrees	Arctangent Same as FANUC ATAN[]/[1]
SQRT[]	Decimal	Decimal	Square root
ABS[]	Decimal	Decimal	Absolute value
ROUND[]	Decimal	Decimal	Round off a decimal
FIX[]	Decimal	Integer	Truncate fraction
ACOS[]	Decimal	Degrees	Arccosine
ASIN[]	Decimal	Degrees	Arcsine
#[]	Integer	Integer	Variable Indirection
DPRNT[]	ASCII text		External Output

Notes on Functions

The function ROUND works differently depending on the context that it is used. When used in arithmetic expressions, the round function works as one would expect. That is, any number with a fractional part greater than or equal to .5 is rounded up to the next whole integer; otherwise, the fractional part is truncated from the number.

```
#1= 1.714 ;
#2= ROUND[#1] ; (#2 is set to 2.0)
#1= 3.1416 ;
#2= ROUND[#1] ; (#2 is set to 3.0)
```

When round is used in an address expression, then the argument of 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) ;
```

Operators

Operators can be classified into three categories: Arithmetic operators, Logical operators and Boolean operators.

Arithmetic Operators

Arithmetic operators consist of the usual unary and binary operators. They are:

+	- Unary plus	+1.23
-	- Unary minus	-[COS[30]]
+	- Binary addition	#1=#1+5
-	- Binary subtraction	#1=#1-1
*	- Multiplication	#1=#2*#3
/	- Division	#1=#2/4
MOD	- Remainder	#1=27 MOD 20 (#1 contains 7)

Logical Operators

Logical operators are operators that work on binary bit values. Macro variables are floating point numbers. When logical operators are used on macro variables, only the integer portion of the floating point number is used. The logical operators are:

OR - logically OR two values together
 XOR - Exclusively OR two values together
 AND - Logically AND two values together

Examples:

#1=1.0; 0000 0001	Here the variable #3
#2=2.0; 0000 0010	will contain 3.0 after
#3=#1 OR #20000 0011	the OR operation.
#1=5.0;	Here control will
#2=3.0;	transfer to block 1
IF [#1 GT 3.0] AND [#2 LT 10] GOTO1	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.

As can be seen from the previous examples, CARE must be taken when using logical operators so that the desired result is achieved.

Boolean Operators

Boolean operators always evaluate to 1.0 (TRUE) or 0.0 (FALSE). There are six Boolean operators. These operators are not restricted to conditional expressions, but they most often are used in conditional expressions. They are:

EQ - Equal to

NE - Not Equal to

GT - Greater Than

LT - Less Than

GE - Greater than or Equal to

LE - Less Than or Equal to

The following are four examples of how Boolean and Logical operators can be used:

Example

IF [#1 EQ 0.0] GOTO100;

WHILE [#101 LT 10] DO1;

#1=[1.0 LT 5.0];

IF [#1 AND #2 EQ #3] GOTO1

Explanation

Jump to block 100 if value in variable #1 equals 0.0.

While variable #101 is less than 10 repeat loop DO1..END1.

Variable #1 is set to 1.0 (TRUE).

If variable #1 logically ANDed with variable #2 is equal to the value in #3 then control jumps to block 1.

Expressions

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

Conditional Expressions

In the HAAS control, ALL expressions set a conditional value. The value is either 0.0 (FALSE) or the value is nonzero (TRUE). The context in which the expression is used determines if the expression is a conditional expression. Conditional expressions are used in the IF and WHILE statements and in the M99 command. Conditional expressions can make use of Boolean operators to help evaluate a TRUE or FALSE condition.

The M99 conditional construct is unique to the HAAS control. Without macros, M99 in the HAAS control has the ability to branch unconditionally to any line in the current subroutine by placing a P code on the same line. For example:

N50 M99 P10 ;



branches to line N10. It does not return control to the calling subroutine. With macros enabled, M99 can be used with a conditional expression to branch conditionally. To branch when variable #100 is less than 10 we could code the above line as follows:

N50 [#100 LT 10] M99 P10 ;

In this case, the branch occurs only when #100 is less than 10, otherwise processing continues with the next program line in sequence. In the above, the conditional M99 can be replaced with:

N50 IF [#100 LT 10] GOTO10 ;

Arithmetic Expressions

An arithmetic expression is any expression using constants, variables, operators, or functions. An arithmetic expression returns a value. Arithmetic expressions are usually used in assignment statements, but are not restricted to them.

Examples of arithmetic expressions: #101=#145*#30;

```
#1=#1+1;
X[#105+COS[#101]];
#[#2000+#13]=0;
```

Assignment Statements

Assignment statements allow the programmer to modify variables. The format of an assignment statement is:

<expression>=<expression>

The expression on the left of the equal sign must always refer to a macro variable, whether directly or indirectly. The following macro initializes a sequence of variables to any value. Here both direct and indirect assignments are used.

```
O0300          (Initialize an array of variables) ;
N1 IF [#2 NE #0] GOTO2 (B=base variable) ;
#3000=1         (BASE VARIABLE NOT GIVEN) ;
N2 IF [#19 NE #0] GOTO3 (S=size of array);
#3000=2         (SIZE OF ARRAY NOT GIVEN) ;
N3 WHILE [#19 GT 0] DO1 ;
#19=#19-1       (DECREMENT COUNT) ;
#[#2+#19]=#22   (V=value to set array to) ;
END1 ;
M99 ;
```

The above macro could be used to initialize three sets of variables as follows:

```
G65 P300 B101. S20      (INIT 101..120 TO #0) ;
G65 P300 B501. S5 V1    (INIT 501..505 TO 1.0) ;
G65 P300 B550. S5 V0    (INIT 550..554 TO 0.0) ;
```

The decimal point in B101., etc. would be required.

Control Statements

Control statements allow the programmer to branch, both conditionally and unconditionally. They also provide the ability to iterate a section of code based on a condition.

Unconditional Branch (GOTOnnn and M99 Pnnnn)

In the HAAS control, there are two methods of branching unconditionally. An unconditional branch will always branch to a specified block. M99 P15 will branch unconditionally to block number 15. The M99 can be used whether or not macros is installed and is the traditional method for branching unconditionally in the HAAS control. GOTO15 does the same as M99 P15. In the HAAS control, a GOTO command can be used on the same line as other G coding. The GOTO is executed after any other control commands as are traditional M codes.

Computed Branch (GOTO#n and GOTO[expression])

Computed branching allows the program to transfer control to another block in the same subprogram. The block can be computed on the fly, as in the case of the GOTO[expression] form, or the block can be passed in through a local variable, as in the GOTO#n form.

The GOTO will round the variable or expression result that is associated with the Computed branch. For instance, if #1 contains 4.49 and GOTO#1 is executed, the control will attempt to transfer to a block containing N4. If #1 contains 4.5, then execution will transfer to a block containing N5.

The following code skeleton could be developed to make a program that serializes parts:

```
O9200          (Engrave digit at current location.)  
;  
(D=DECIMAL DIGIT TO ENGRAVE);  
IF [[#7 NE #0] AND [#7 GE 0] AND [#7 LE 9]] GOTO99;  
#3000=1 (INVALID DIGIT)  
;  
N99  
#7=FIX[#7]      (TRUNCATE ANY FRACTIONAL PART) ;  
;  
GOTO#7         (NOW ENGRAVE THE DIGIT)  ;  
;  
N0             (DO DIGIT ZERO)  
...  
M99  
;  
N1             (DO DIGIT ONE)  
;  
M99  
;  
N2             (DO DIGIT TWO)  
;  
...  
;(etc.,...)
```



With the above subroutine, you would engrave digit five with the following call:

G65 P9200 D5;

Computed GOTOs using expression could be used to branch processing based on the results of reading hardware inputs. An example might look like the following:

```
GOTO[[#1030*2]+#1031]      ;
NO (1030=0, 1031=0)        ;
...
M99;
N1 (1030=0, 1031=1)      ;
...
M99;
N2 (1030=1, 1031=0)      ;
...
M99;
N3 (1030=1, 1031=1)      ;
...
M99;
```

The discrete inputs always return either 0 or 1 when read. The GOTO[expression] will branch to the appropriate G-code based on the state of the two discrete inputs #1030 and #1031.

Conditional Branch (IF and GOTOn)

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

Here, as discussed above, <conditional expression> is any expression that uses the six Boolean operators EQ, NE, GT, LT, GE, or LE. The brackets surrounding the expression are mandatory. In the HAAS control, it is not necessary to include these operators. For example:

IF [#1 NE 0.0] GOTO5;

could also be:

IF [#1] GOTO5 ;

In this statement, if the variable #1 contains anything but 0.0, or the undefined value #0, then branching to block 5 will occur; otherwise, the next block will be executed. If portability to a control other than HAAS is desired, then it is recommended that the conditional operators be used.

In the HAAS control, a conditional expression can also be used with the M99 Pnnnn format, providing that macros have been enabled. An example is as follows:

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, Z0 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 ;
```

Here, 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. You might try something like this:

```
IF [#1 GE 180] THEN #101=0.0 M99 ;
```

This says that if variable #1 (address A) is greater than or equal to 180, then set variable #101 to zero and return from the subroutine.

Here is an example of an IF statement that branches if a variable has been initialized to contain any value. Otherwise, processing will continue and an alarm will be generated. Remember, when an alarm is generated, program execution is halted.

```
N1 IF [#9NE#0] GOTO3 (TEST FOR VALUE IN F) ;
N2 #3000=11(NO FEED RATE) ;
N3 (CONTINUE) ;
```

Iteration/Looping (WHILE DO END)

Essential to all programming languages is the ability to execute a sequence of statements a given number of times or to loop through a sequence of statements until a condition is met. Traditional G coding allows this with the use of the L address. A subroutine can be executed any number of times by using the L address.

```
M98 P2000 L5 ;
```

This is limited since you can not terminate execution of the subroutine on condition. Macros allows more flexibility with the WHILE-DO-END construct. The syntax is as follows:

```
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 basically a loop within a loop.



Although nesting of WHILE statements can only be nested to three levels, there really is no limit since each subroutine can have up to three levels of nesting. If there ever is a need to nest to a level greater than 3, then the segment containing the three lowest levels of nesting can be made into a subroutine thus overcoming the limitation.

If two separate WHILE loops are in a subroutine, they can use the same nesting index. For example:

```
#3001=0 (WAIT 500 MILLISECONDS) ;
WH [#3001 LT 500] DO1 ;
END1 ;
<other statements>
#3001=0 (WAIT 300 MILLISECONDS) ;
WH [#3001 LT 300] DO1 ;
END1 ;
```

This is valid code.

You can use GOTO to jump out of a region encompassed by a DO-END, but you can not use a GOTO to jump into it. Jumping around inside a DO-END region using a GOTO is allowed.

An infinite loop can be executed by eliminating the WHILE and expression. Thus,

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

Executes until the RESET key is pressed.

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

In the above, an alarm results indicating no "then" was found; here "then" refers to the D01. Change D01 (zero) to DO1 (letter O).

COMMUNICATION WITH EXTERNAL DEVICES - DPRNT[1]

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[1] and PCLOS.

Communication preparatory commands

POOPEN and PCLOS are not required on the HAAS lathe. It has been included so that programs from different controls can be ported to the HAAS. On some controls POOPEN is required prior to using a DPRNT statement. POOPEN prepares the device on the serial port by sending it a DC2 code. PCLOS terminates communication with external devices by sending it a DC4 code.

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. Variables can be formatted. The form of the DPRNT statement is as follows:

DPRNT[<text> <#nnnn[wf]>...] ;

DPRNT must be the only command in the block. In the above, <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 legal macro variable. The format [wf] is required and consists of two digits within square brackets. Remember that macro variables are real numbers with a whole part and a fractional part. The first digit in the format designates the total places reserved in the output for the whole part. The second digit designates the total places reserved for the fractional part. The total places reserved for output cannot be equal to zero or greater than eight. Thus the following formats are illegal:

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

A decimal point is printed out between the whole part and the fractional part. The fractional part is rounded to the least significant place. When zero places are reserved for the fractional part, then no decimal point is printed out. Trailing zeros are printed as necessary if there is a fractional part. At least one place is reserved for the whole part, even when a zero is used there. 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]] ;	X.1.5436 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 OF DPRNT

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 positional information. Generally, a program is interpreted many blocks ahead in order to prevent the machine from pausing between movements.

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, then issue a G103 P0 ;. G103 can not be used when cutter compensation is active.

OPERATION NOTES

This section explains the additional screens and operator actions that come with macros.

Macro variables can be saved and restored to RS-232 or the optional floppy disk, much like parameters, settings, and offsets. Refer to the "Part Program Input / Output" section for RS-232 sending and receiving of macro variables, or the "Disk Operation" section for sending and receiving them with this method.

Variable Display Page

The macro variables are displayed and can be modified through the current commands display. The variable display is located after the operation timers display. To get to this page, 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.

Pages contain up to 32 variables and the display can be “paged” by pressing the left/right arrow keys.

Setting of a variable is accomplished by entering a value and then pressing the WRITE key. The variable that is highlighted on the screen is the variable that is affected.

Searching for a variable can be done by entering the variable number and pressing the up/down arrow. The page will change to the one that contains that variable and the entered variable will become the highlighted item.

The variables displayed represent the values of the variables at program interpretation time. At times, this may be up to 15 blocks ahead of the actual machine activity. Debugging of programs can be made easier by inserting a G103 at the beginning of a program to limit block buffering and then removing the G103 block after debugging is completed.

Editing

For the most part, the editing of macro programs from the control is the same as before. There are a few peculiarities to be aware of.

Editing macro statements is more open than previously. For instance, it is possible to place a floating point constant within a standard G-code block, but it doesn't make much sense, and the control will raise an alarm at runtime. For all instances of improperly structured or improperly placed macro statements, the control will raise an appropriate alarm. Most of these alarms have been put off until runtime so that operator editing can be more flexible. Be careful when editing expressions. Brackets must be balanced and you will not receive an alarm until runtime.

The DPRNT[] function can be edited much like a comment. You can delete it or move it as a whole item, or you can edit individual items within the brackets. 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 crank 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 floating point constant.

G1 X 0 Z3.0 (!!! WRONG !!!) ;

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

G1 X0 Z3.0 (CORRECT) ;

Note that the zero is attached to **X**. 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 have not yet been implemented.

M ALIASING	REPLACE G65 Pnnnn WITH Mnn PROGS 9020-9029.
G66 MODAL	CALL IN EVERY MOTION BLOCK
G66.1 MODAL	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
ADDITIONAL OFFSETS	G54.1P## FORMAT
NAMES FOR VARIABLES	FOR DISPLAY PURPOSES
ATAN []/[]	ARCTANGENT, FANUC VERSION
BIN []	CONVERSION FROM BCD TO BIN
BCD []	CONVERSION FROM BIN TO BCD
FUP []	TRUNCATE FRACTION CEILING
LN []	NATURAL LOGARITHM
EXP []	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

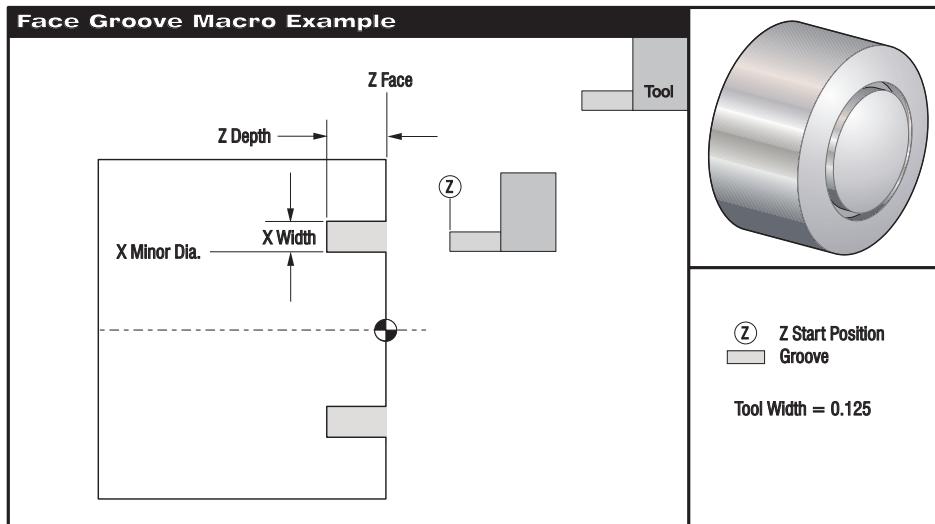
Searching for a block to jump in the negative direction (i.e. backwards through a program) is not necessary if you use unique N address codes.

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 ] | [ #20 * 0.9 ] F#9  
G00 X0 Z0 T100  
M30  
%
```



**11.5 AUXILIARY AXIS CONTROL**

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

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 SINGL 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". A failure in RS-232 communication with the auxiliary axis may cause this display to pause indefinitely. The RESET button will terminate any "hung-up" auxiliary axis communication.

The auxiliary axis cannot be jogged from the CNC front panel. The single axis control jog button should be used for this. When the auxiliary axis is idle, the front panel JOG button for the auxiliary axis can be used to jog that 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 and zero will be shifted by the value set into the single axis control. To set a displaced zero, you must jog the single axis control to a new zero position and then press and hold the CLEAR key on the single axis control. This must only be done when the single axis control is otherwise idle.

Auxiliary axes communication is always seven data bits, even parity, two stop bits. The data rate is CNC Setting 54 and should be set to 4800. CNC Setting 50 must be set to XON/XOFF. Parameter 26 in the single axis control must be set to 5 for 4800 bit per second and Parameter 33 must be set to 1 for XON/XOFF. Parameter 12 in the single axis control should always be set to 3 or 4 to prevent circular wraparound.

The cable connecting the CNC to the single-axis control must be a DB-25 cable (male lead on both ends) and must wire at least pins 1, 2, 3, and 7 directly from the second (lower) serial port of the CNC to the upper connector of the servo control.

11.6 HYDRAULIC TAILSTOCK OPERATION*

*This option is not field installable

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

1. Through program code,
2. In jog mode,
3. By a foot switch.

The tailstock is designed to travel to position at 2 rates:

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

CAUTION!!

1. If tailstock hydraulic pressure is set lower than 120 psi, it may not function reliably.
2. It is important to verify tailstock and turret clearance before operating machine or serious damage could occur. Adjust setting 93 and 94 as necessary
3. 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) can be used to ensure that the tailstock does not collide with the turret or any tools in the turret. The restriction zone is limited to a rectangular area in the lower right hand part of the lathe's work space. The restriction zone changes dynamically, assuring 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 separation to maintain. If a programmed motion crosses the tailstock's protected area, alarm 609 (TAILSTOCK CONFLICT) will be generated. Keep in mind that a restricted zone is not always desired (e.g. when setting up). To set a proper restricted zone it will be necessary to reduce the restricted zone to nothing by placing 0 in Setting 94 and maximum X machine travel in setting 93. Refer to the setting section for further information about Settings 93 and 94.

DETERMINING A VALUE FOR THE X CLEARANCE PLANE:

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

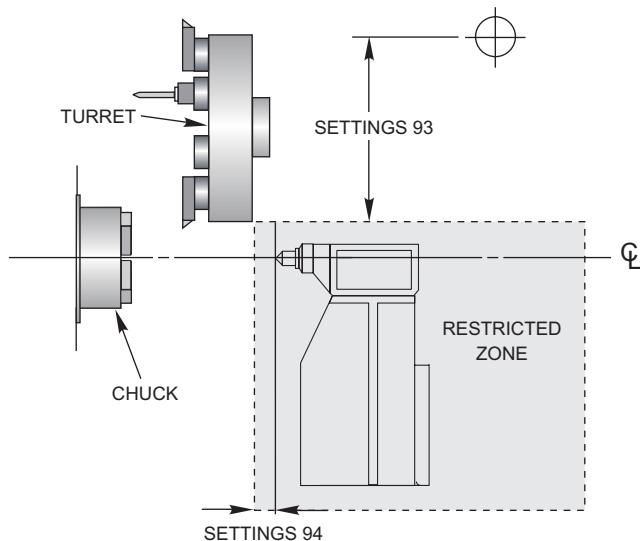
DETERMINING A SEPARATION FOR Z AND B AXIS BELOW THE X CLEARANCE PLANE:

- Place the control in ZERO RET and HOME G28 all axes.
- Select the X axis and move the turret in front of the tailstock center tip.
- Move the Z axis so that the rear of the tool turret is within about 0.25 inches of the tailstock tip.
- Find the Z axis "machine" position on the display, this becomes the value for Setting 94.

The above method will handle most, but not all, conflicts between the tool turret and the tailstock. After making these settings, it may still be necessary to adjust your program to avoid the restricted zone. Or the restricted zone may have to be expanded, depending on tooling and part size.

TAILSTOCK SETTINGS

Two settings are provided for protecting the tailstock. Setting 93 (TAIL ST. X CLEARANCE) and Setting 94 (Z/TS DIFF @X CLEARANCE) interact together to disallow any motion that would crash the tailstock into the tool turret. The figure below illustrates the relationship between these two settings and how they can be used to set up a protected area for the tailstock.



Tailstock Restricted Zone

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

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.

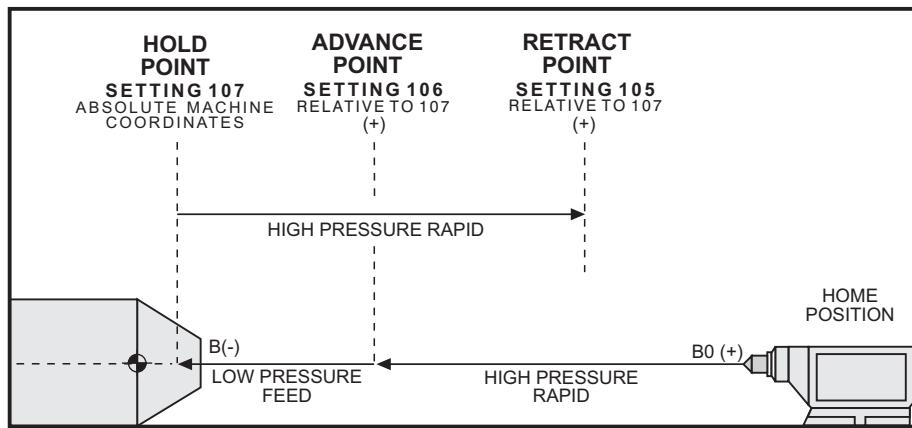


Diagram of Settings 105, 106, and 107.

93 TAIL ST. X CLEARANCE

This setting is effective only if the B axis is enabled. It works together with Setting 94 to define a travel forbidden zone that limits interaction between the tailstock and the tool turret. This setting determines the X axis travel limit in machine coordinates when the difference between the Z axis location and the B axis location falls below the value in Setting 94. If this condition occurs and a program is running then alarm 609 is generated. When jogging, no alarm is generated, but travel will be limited. Units are in inches. The default value for this setting is lathe model dependent. Suggested values are:

SL-20 -6.0"
SL-30 -6.0"
SL-40 -6.0"

94 ZB DIFF @X CLEARANCE

This setting is effective only if the B axis is enabled. It represents the minimum allowable difference between the Z and B axes at, or below, the tail stock X clearance plane (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 tailstock position in the Z axis negative direction. The default value for this setting is 0.0"

105 TS RETRACT DISTANCE

The distance from the HOLD POINT (Setting 107) the tailstock will retract when commanded. This setting should be a positive value. 3.0 is a good starting value.

106 TS ADVANCE DISTANCE

When the tailstock 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. 2.0 is a good starting value.

NOTE: If the values in Settings 105 and 106 are zero, the tailstock may not function as desired.

107 TS HOLD POINT (absolute machine coordinates)

Point to advance to for holding when M21 is invoked. 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. This setting should be a negative value. The tailstock should never actually reach the hold point.

TAILSTOCK M CODES

M21 Tailstock Advance (Standard M code)

Uses Settings 105, 106 and 107 to advance to the HOLD POINT.

M22 Tailstock Retract (Standard M code)

Uses Setting 107 to withdraw to the RETRACT POINT.

TAILSTOCK FOOT PEDAL OPERATION

- Invokes M21 or M22, depending on current position.
- If tailstock is to the left of the retract point, pressing the foot pedal will invoke M22 (move to RETRACT POINT).
- If tailstock is to the right of the retract point, pressing the foot pedal will also invoke M22 (move to RETRACT POINT).
- If tailstock is at retract point, pressing the foot pedal will invoke M21 (move to HOLD POINT).
- If the foot pedal is pressed while tailstock is moving, the tailstock will stop and a new sequence must begin anew.

JOGGING THE TAILSTOCK

- In JOG mode the keys, "TS <—" and TS "—>" are used to jog the tailstock at low pressure (feed).
- By pressing TS <— or TS —> keys together with TS RAPID high pressure (rapid) is selected.
- When tailstock keys are released. The control reverts to the last jogged axis, X or Z.

RUN TIME ALARMS

If a part is being held and tailstock motion is detected, Alarm (394/317) B AXIS OVERTRAVEL is generated. This will stop the program and turn off the spindle.

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



OPTIONS

SL Series OPERATOR'S MANUAL

June 2001

TAILSTOCK DIAGNOSTIC DISPLAYS

OUTPUTS:	TSFAST 0 = low pressure 1 = high pressure
	TS → 1 = tailstock away from chuck, + direction
	TS ← 1 = tailstock toward chuck, - direction
INPUTS	TS FSW foot pedal input

The following information can also be found on the side panel of the machine.

RECOMMENDED HYDRAULIC TAILSTOCK OPERATING PRESSURE IS 120 PSI.

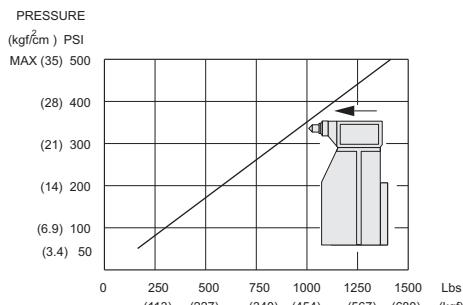
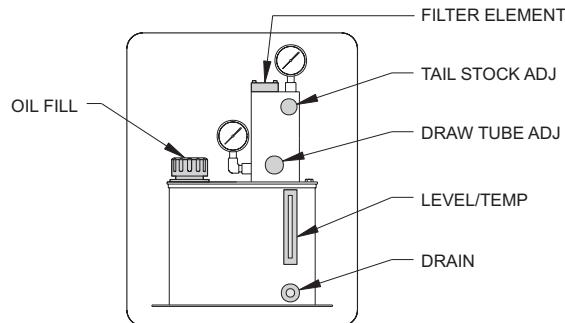
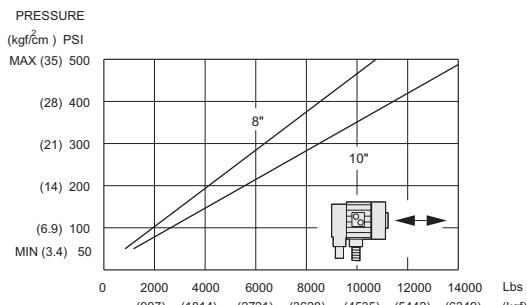
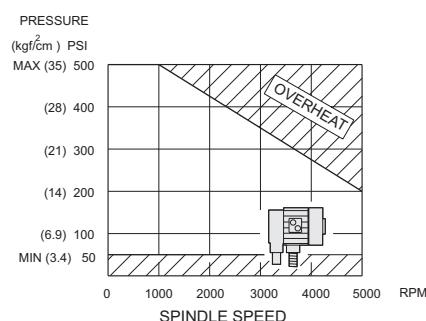
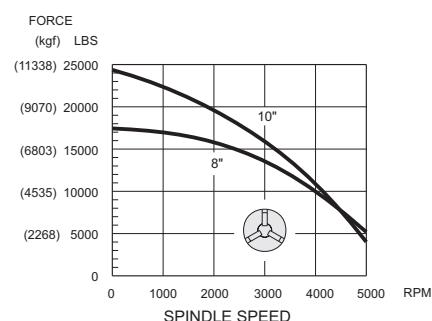
CAUTION: Operating pressures below 120 psi may cause the tailstock to function unreliably.

TAILSTOCK HYDRAULIC PRESSURE

The following information can also be found on the side panel of the machine.

RECOMMENDED HYDRAULIC TAILSTOCK OPERATING PRESSURE IS 120 PSI.

CAUTION: Operating pressures below 120 psi may cause the tailstock to function unreliable.


HYDRAULIC TAILSTOCK FORCE

HYDRAULIC DRAWTUBE FORCE

MAXIMUM RECOMMENDED CYLINDER PRESSURE

**CHUCK GRIPPING FORCE
(STANDARD JAWS, 330 PSI)**
SL 20 and 30



OPTIONS

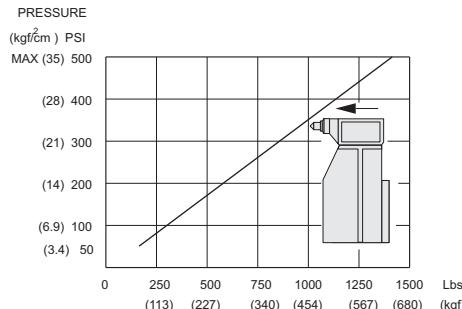
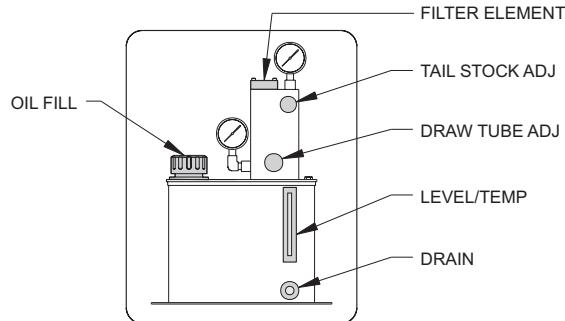
SL Series OPERATOR'S MANUAL

June 2001

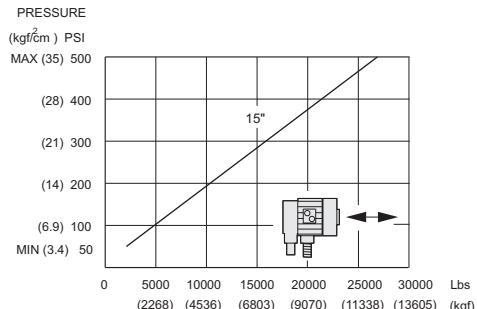
The following information can also be found on the side panel of the machine.

RECOMMENDED HYDRAULIC TAILSTOCK OPERATING PRESSURE IS 120 PSI.

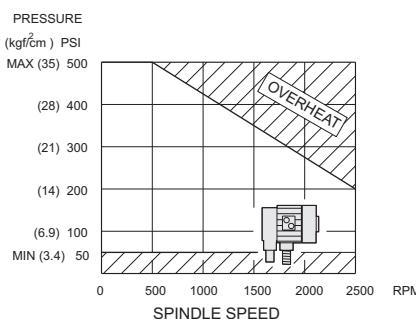
CAUTION: Operating pressures below 120 psi may cause the tailstock to function unreliable.



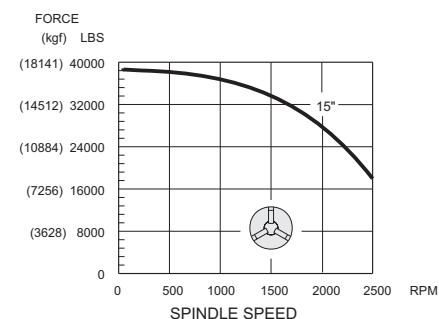
HYDRAULIC TAILSTOCK FORCE



HYDRAULIC DRAWTUBE FORCE



MAXIMUM RECOMMENDED CYLINDER PRESSURE

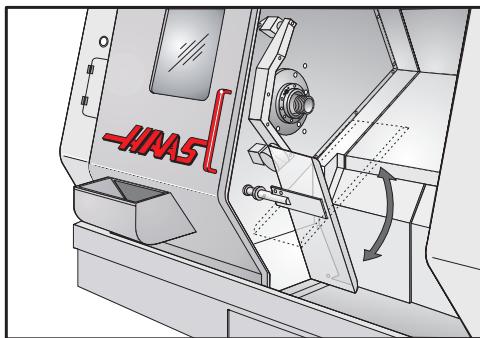
CHUCK GRIPPING FORCE
(STANDARD JAWS, 330 PSI)**SL-40**



11.7 PARTS CATCHER

The parts catcher is an automatic part retrieval system designed to work with bar feed applications. The parts chute, which is actuated using auxiliary M Codes, rotates to catch finished parts and directs them into the bin mounted on the front door. Use the following M Codes for Parts Catcher operation:

M36 to activate
M37 to deactivate

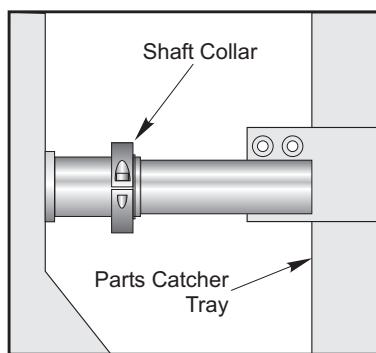


Max. Part Length: 5.5" (all machines)
Max. Bar Stock Dia.: 2.0" SL-20
2.5" SL-30

OPERATION

The parts catcher must be properly aligned before operation.

1. Power on the machine. In MDI mode enter a program that will activate the parts catcher (M53), deactivate the parts catcher (M63), then loop back to the beginning (M99).
2. Step through the program using single block mode. Check the rotation of the part catcher shaft at each step. It should rotate counter clockwise when activated and rotate clockwise when deactivated. If it is functioning properly, stop the program with the shaft rotated counterclockwise.
3. Loosen the screw in the shaft collar on the outer parts catcher shaft as shown.



SL-20 Shaft Collar shown.

4. 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.



OPTIONS

SL Series OPERATOR'S MANUAL

June 2001

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

5. Continue stepping through the program to check tray operation.
6. When using 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.

WARNING!!

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

**11.8 Tool Pre-Setter**

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

OPERATOR

The Tool Pre-Setter is located next to the chuck and is raised and lowered manually. In order to activate the Tool Pre-Setter, Parameter 278 bit TL SET PROBE must be set to 1. When the arm is in the upright HOME position, discrete input #23 PROBNH will display 0.

Once the arm is lowered, the turret can be jogged using the jog handle or jog buttons; discrete input #23 PROBNH will be 1 and the CYCLE START button will be disabled. If the CYCLE START button is pressed, the message "PROBE IS DOWN" will be displayed. This prevents the operator from accidentally running a program and crashing into the arm. As a safety feature the jog rate defaults to a maximum of .01 and the RAPID key is disabled.

IMPORTANT! Automatic storing of the machine's position can only be performed when the jog buttons are used. The jog handle can be used to jog the tool tip into contact with the probe, however, once the control beeps and the axis has stopped moving, the value must then be manually entered into the control.

Once the probe is touched, the control will beep and not allow the operator to continue jogging in that direction. This prevents the operator from inadvertently damaging the probe and also ensures greater accuracy.

WARNING

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

The probe can be pressed momentarily by hand to test the audible beep and verify the PROBNH discrete input. However, if the operator deliberately holds the probe aside and jogs the tool tip within the space normally occupied by the probe, he may discover that the axis he needs to jog away from has become disabled.

The same thing can occur if the operator jogs the tool tip into the probe too fast. The probe may actually slip off the tool tip, and the wrong axis becomes disabled.

If this should happen, use the other axis to slide the tool tip off the probe. All axes will then be re-enabled. If this does not work, then the probe arm should be raised back to its home position. If this is not possible, the switch that senses that the arm is in the upright position can be pressed. This will re-enable all axes, and the tool can be carefully backed off.

**SETTINGS**

Before using the arm to calculate tool geometry or tool shift offsets, the following settings must be adjusted:

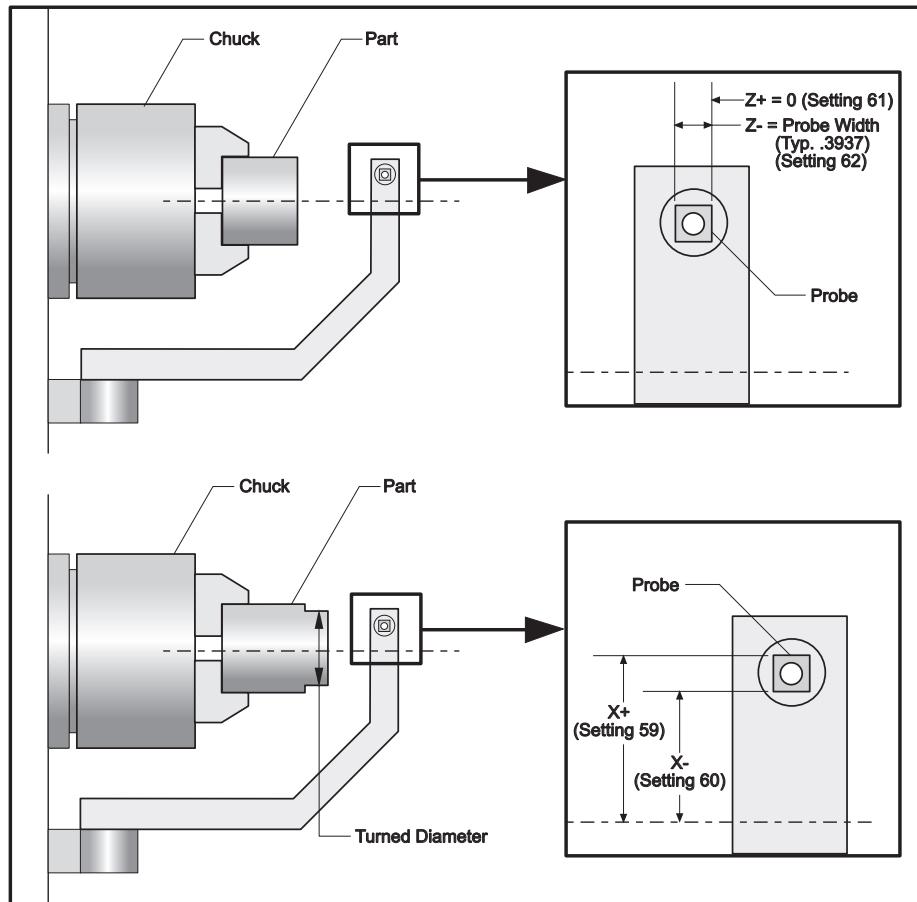
Setting 59 PROBE OFFSET X+

Setting 60 PROBE OFFSET X-

Setting 61 PROBE OFFSET Z+

Setting 62 PROBE OFFSET Z-

*Note that Setting 63 TOOL PROBE WIDTH is not used.



Probe Settings Diagram.

GETTING STARTED**Setting 'X' offsets.**

1. Clamp a piece of material in the chuck and cut the outside diameter.
2. Measure the diameter of the part using a pair of micrometers and record the measurement on a piece of paper.
3. Touch the tool tip to the turned section of the part and using the **origin** key, zero the 'X' register of the **operator position** display.
4. Using the **operator position** display as a guide move the tool in the 'X' direction until the display reads the same value as the measured diameter and using the **origin** key, zero the 'X' register of the display.
5. Move the tool to a safe position and lower the tool setter arm and touch the tool tip using the jog handle in the .001 mode. **Note:** While jogging, when the tool comes in contact with the probe the control will beep and jogging in the current direction will stop.
6. Record the value shown in the 'X' operator position display into **Setting 59 PROBE OFFSET X+**.
7. Subtract 2 times the probe width from the 'X' operator position display and store this value into **Setting 60 PROBE OFFSET X-**.

Setting 'Z' offsets.

1. The value of **Setting 61 PROBE OFFSET Z+** should be Zero.
2. The value of **Setting 62 PROBE OFFSET Z-** should be the width of the probe (i.e. if the probe measures .3937 **Setting 62 PROBE OFFSET Z-** would be **.3937**).

SETTING TOOL GEOMETRY AND TOOL SHIFT OFFSETS USING THE PROBE

NOTE: Settings 59 through 62 (probe offsets) must be set according to the previous instructions before proceeding.

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



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

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

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

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

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

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

SETTING WORK ZERO OFFSETS

Before running your program, the machine's WORK ZERO OFFSETS (G52-129) must be entered.

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

11.9 AUTOMATIC CHIP CONVEYOR/AUGER

The automatic chip conveyor assists the user in removal of chips for jobs with heavy material removal. When running, the chip conveyor will sense conveyor motor overcurrent and reverse direction momentarily, thus attempting to free up chip jams. This procedure will be repeated until chips are cleared or conveyor retry limit (Parameter 219) is reached. If the chip conveyor is running and the door is opened, the chip conveyor will stop, thus adding a degree of safety to conveyor operation. If there is no axis motion or keyboard action within the time set in Parameter 255, the conveyor will automatically shut off.

NOTE: It is recommended that the chip conveyor be used intermittently. Continuous operation will cause the motor to overheat.

NOTE: On a machine with a safety circuit, the chip auger will only run with the door closed regardless of the Conveyor Door Override bit.

CONVEYOR KEYBOARD COMMANDS

The conveyor can be started at anytime from the keyboard. The conveyor can be enabled in either direction by pressing the CHIP FWD or CHIP REV and stopped by pressing the CHIP STOP key. The conveyor will also stop by pressing the RESET key.

CONVEYOR PROGRAM COMMANDS

Use M codes M31, M32 and M33 to control the conveyor from within a program or in MDI. M31 commands the conveyor forward, M32 commands the conveyor in reverse and M33 stops the conveyor. Refer to the "M Codes" section for a more detailed description.

CONVEYOR PARAMETERS

The parameters that control the conveyor are below.

CNVYR RELAY DELAY	Parameter 216
CNVYR IGNORE OC TIM	Parameter 217
CNVYR RETRY REV TIM	Parameter 218
CNVYR RETRY LIMIT	Parameter 219
CNVYR RETRY TIMEOUT	Parameter 220
CONVEYOR TIMEOUT	Parameter 255

A complete description of conveyor parameters is given in the "Parameters" Section.

**CONVEYOR M CODES****M31 Chip Conveyor Forward**

M31 starts the chip conveyor motor in the forward direction. The forward direction is defined as the direction that the conveyor must move to transport chips out of the work cell. If the conveyor motor is on, then the conveyor will be stopped and restarted in the forward direction.

M32 Chip Conveyor Backward

M32 starts the chip conveyor motor in the reverse direction. The reverse direction is defined as the direction opposite of forward. If the conveyor motor is on, then the conveyor will be stopped and restarted in the reverse direction.

M33 Chip Conveyor Stop

M33 Stops conveyor motion.

11.10 BAR FEEDER WITH THE HAAS LATHE

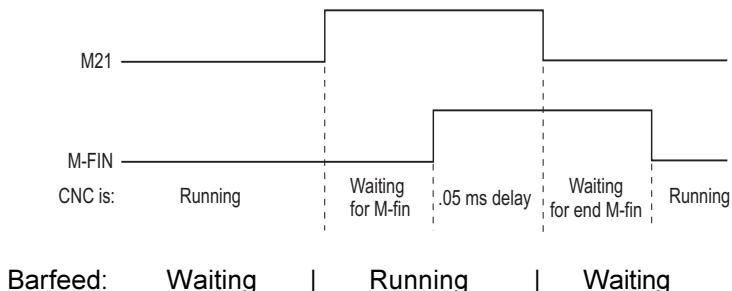
This section describes how the HAAS CNC Lathe control can be interfaced to a bar feed. It provides a generalized interface to cover most of the common bar feeder types. However, due to the number of bar feeders available, this description cannot cover all possibilities.

NOTE: Refer to drawtube specifications in the features section.

M-CODE / M-FIN INTERFACE

The M-code/M-fin interface from the HAAS control is identical to those in most other CNC's. This interface is usually required in a bar feeder interface to initiate a bar feed or bar change. An amphenol connector at the side of the control cabinet provides this interface using M121 and M124 as the programming code.

When an M121 or M124 code is programmed, the CNC will pause executing blocks until an M-fin response signal is received. The following diagram shows the timing sequence for M21 output:



The high state in this diagram indicates an active circuit or closed circuit.

**END-OF-BAR INPUT SIGNAL AND TESTING**

Many bar feeder interfaces provide a signal to the CNC control to indicate that the end of the bar has been reached. This usually means that another bar feed operation cannot be performed and that the CNC control must either initiate a bar change or stop for the operator to change the bar. There are several spare input signals to the Haas CNC which can be used to determine the status of this signal prior to initiating a bar feed.

Two general purpose input signals are provided in the Haas Bar Feed Interface. They are called End-of-bar and Load-Ok. They are wired to pins H and J respectively with a signal common on pin K.

The most easily used signal for End-Of-Bar is labeled "A161" on the IOPCB PCB connector P52. It can be tested by the following lines of G-code programming:

N2000 ... (PROGRAM CONTINUES AT THIS LINE)

.

M96 Q26 P2000 (GOTO LINE 2000 IF INPUT 26 IS CLOSED CIRCUIT)
... (ELSE FALL INTO THIS LINE OF PROGRAM)

Note that "A161" is discrete input number 21 of 31.

In order to test the second input signal, Loader-Ok, the Q 26 code should be changed to Q27. If macro variables are used for this, they would be #1026 and #1027 respectively.

PROGRAMMING SEQUENCE FOR A BAR FEED OPERATION

A typical bar feed operation requires the following programming steps in the user's G-code program:

1. Stop spindle.
2. Test for end-of-bar; if End-Of-Bar, either use M-code to change bar or stop machine (see the previous section on M-Code/M-fin interface).
3. If required, position a selected tool as a stop in front of the spindle.
4. Unclamp the spindle.
5. If required, turn on an M-code to apply bar pressure (see the following section).
6. If a tool is used as a bar stop, feed in +Z to allow bar to come out.
7. Clamp the spindle.
8. If required, turn off an M-code to remove bar pressure (see the following section).

Note that this interface does not require the macros option. A G-Code program to perform all of the above steps would look like:

M54	(TURN ON AUTO MODE)
M96 P200 Q27	(JUMP TO N200 ON LOADER NOT OK, SPARE 5)
M96 P100 Q26	(JUMP TO N100 ON END OF BAR, SPARE 4)
G00 X0 Z.01	(MOVE STOP INTO PLACE)
M11	(OPEN CHUCK)
M122	(FEED PART)
M10	(CLOSE CHUCK)
M96 P200 Q27	(JUMP TO N200 ON LOADER NOT OK, SPARE5)
G4 P4.	(INSERT MAIN PROGRAM HERE)
M99	(RETURN TO TOP OF PROGRAM)
N100 G00 X5 Z7	(START OF RELOAD SECTION MOVE STOP AWAY)
M11	(OPEN CHUCK)
M122	(LOAD NEW BAR)
G00 X0 Z.1	(MOVE STOP INTO PLACE)
M122	(LOAD NEW BAR)
M10	(CLOSE CHUCK)
M96 P200 Q27	(JUMP TO N200 ON LOADER NOT OK, SPARE5)
G4 P3.5	(INSERT TOP CUT PROGRAM HERE)
M99	(RETURN TO TOP OF PROGRAM)
N200	(LINE NUMBER FOR LOADER NOT OK)
M2	(END OF PROGRAM)

M CODES TO TURN A SPARE RELAY ON OR OFF

If an output from the CNC is required to turn the bar pressure on and off, there are two relays which can be used for this. The relays are labeled M21 and M24. These relays can be turned on and left in that state by using other M-Codes built into the control. They can also be turned off with other M-Codes. Thus the following G-Code program will turn on the M22 relay for 1 second:

M51	(turn M21 on)
G04 P1.0	(dwell for 1 second)
M61	(turn M21 off)

The following can be used to control all four spare M-Code relays:

M21 is turn on by M51 and off by M61.

M24 is turn on by M54 and off by M64.

M21 is pre-wired to a connector at the side of the cabinet so is not assumed to be available.

Note that M51-54 and M61-64 do not wait for any M-fin signal. They simply turn the appropriate relay on or off. M121-M124 will turn a relay on and wait for the M-fin signal.

EMERGENCY STOP INPUT TO HAAS CONTROL

If your interface requires that the Haas CNC go to emergency stop condition based on an external input signal, there is a second E-Stop input to the IOPCB. This input is connector P40 and is labeled 770A. It requires a normally-closed isolated contact. The circuit is 12 volts DC, must close to a drop of less than 1 volt, and must carry 10ma. It must open to more than 200k ohms and close to less than 200k ohms. The status of this switch is labeled E-stp on the diagnostic display page.

The emergency stop input pins are S and T.

**EMERGENCY STOP OUTPUT FROM HAAS CONTROL**

If your interface requires that the Haas CNC provide an E-Stop input signal to the bar feeder, a second switch contact pair can be added to the back of the Haas front panel E-Stop switch. This can then be wired to the bar feeder. If a normally-open switch is required, the Haas part number is 61-0020C; if you require a normally-closed switch, the Haas part number is 61-0030C. One input from this switch is already wired to the control and is labeled E-STP on the Discretes page.

The emergency stop output pins from the Haas Control are L and M.

SPINDLE CLAMPED/UNCLAMPED STATUS SWITCHES FROM HAAS LATHE

If a positive chuck unclamped signal is required for your bar feeder this is provided with a bar feeder interface. It is provided on pins A and B. When the chuck is unclamped, this circuit is closed.

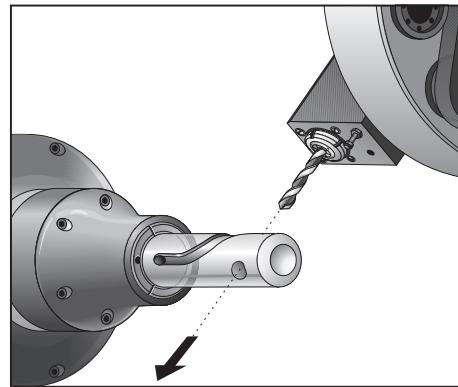
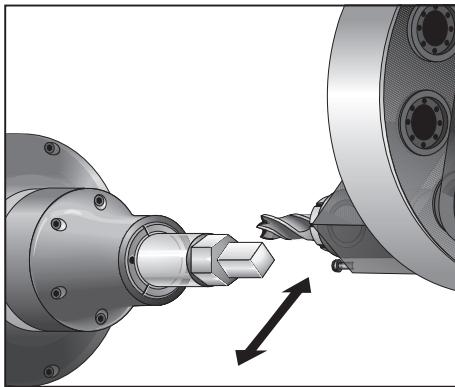
If you have the hydraulic actuator, two switches can be added to the back of the actuator assembly to provide clamped/unclamped status. These switches are provided by Kitigawa, and are part number PSW-CSS18.

RUN MODE / MANUAL MODE / DOOR INTERLOCK FUNCTIONS

There is no output signal from the Haas control to indicate that the machine is in a Run mode versus a manual mode. There also is no signal from the control to indicate the status of the door switches on the enclosure. If your bar feeder requires these signals, contact the bar feeder manufacturer for other options.

11.11 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 secondary operations as milling, drilling or slotting. The main spindle of the lathe is indexable in one degree increments for precise part positioning and repeatability. A hydraulic brake locks the spindle during heavy cutting. Milling shapes is possible using spindle motion G codes and canned cycles.

The standard Haas turret is used in conjunction with Haas VDI adaptors. These adaptors locate, orient and lock standard 40 mm VDI tools in any existing Haas turret pocket. In addition a VDI turret can be ordered with the live tooling option. The VDI turret is configured for standard 40 mm shank VDI tools.

PARAMETERS**Parameter 72 LIVE TOOL CHNG DLAY**

This parameter specifies the amount of time (in milli seconds) to wait after commanding the Live Tooling Drive motor to turn at the velocity specified by parameter 143. This process is required to engage the live tooling motor and tool and is only performed prior to the first M133 or M134 after a tool change.

Parameter 143 LIVE TOOL CHNG VEL

This parameter specifies the velocity to command the Live Tooling Drive motor for the period specified by parameter 72. This process is required to engage the live tooling motor and tool and is only performed prior to the first M133 or M134 after a tool change.

Parameter 278 LIVE TOOLING Bit 24

For lathes fitted with the Live Tooling drive, this bit must be set to 1. For all other lathes, this bit must be set to 0.

**Notes on Live Tooling:**

- a) The live tool drive will automatically turn itself off when a tool change is commanded.
- b) 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.
- c) Maximum live tooling drive speed is 3000 rpm.
- d) Canned cycles are not supported.
- e) The Y axis parameters are used for the Live Tooling drive, however the Y-axis should remain disabled.

Parameter 298 YAX RTAP BACKLASH

This parameter is normally 0 but can be adjusted by the user (to a number typically between 0 and 1000) to compensate for play in the coupling between the motor and the live tooling. It takes effect during G95 LIVE TOOLING RIGID TAP when the tool has reached the bottom of the hole and must reverse direction to back out.

Parameter 304 SPINDLE BRAKE DELAY

This parameter specifies the amount of time (in milliseconds) to wait for the main spindle brake to unclamp when spindle speed has been commanded, and also the amount of time to wait after the main spindle has been commanded to stop before clamping it.

M CODES**M14 CLAMP MAIN SPINDLE**

This M code will clamp the main spindle.

M15 UNCLAMP MAIN SPINDLE

This M code will unclamp the main spindle.

M133 LIVE TOOL DRIVE FORWARD

This M code commands the Live Tool Drive to turn in the forward direction and requires a P code specifying the Live Tooling Drive RPM, for example M133 P1000.

M134 LIVE TOOL DRIVE REVERSE

This M code commands the Live Tool Drive to turn in the reverse direction and requires a P code specifying the Live Tooling Drive ,RPM, for example M134 P1000.

M135 LIVE TOOL DRIVE STOP

This M code commands the Live Tooling Drive to stop.

MESSAGES
C CLAMPED

This message indicates that the spindle has been clamped by an M14 command.

CURRENT COMMANDS DISPLAYS
M19 ANGLE CMD (Optional)

Viewed in the Current Commands Tool Load screen.

An M19 (introduced in version 2.21 for vector drives only) will orient the spindle to the zero position. A P value can be added that will cause the spindle to be oriented to a particular position (in degrees.) The position specified by the last M19 will be displayed when parameter 278 bit 24 LIVE TOOLING is set to 1.

Degree of accuracy

P rounds to the nearest whole degree.

R round to the nearest hundredth of a degree (x.xx)

SUB SP RPM CMD (LIVE TOOL COMMANDED RPM)

Viewed in the Current Commands Tool Load screen.

The last commanded Live Tooling Drive RPM specified by an M133 or M134 is displayed when parameter 278 bit 24 LIVE TOOLING is set to 1.

TOOL SPINDLE MOTOR:

M133 Pxxxx	=	Spindle Forward @	xxxxrpm
M134 Pxxxx	=	Spindle Reverse @	xxxxrpm
M135	=	Spindle Stop @	

SPINDLE INDEXING:

M19 Pxxx	=	Index to Angle xxx from Spindle Mark
M14	=	Spindle Brake Clamp
M15	=	Spindle Brake Unclamp

PARAMETER CHECKLIST:

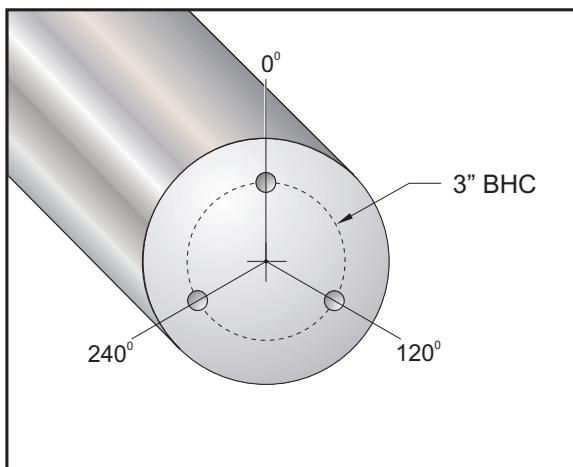
PARAM 72	LIVE TOOL CHNG DLY
PARAM 143	LIVE TOOL CHNG VEL
PARAM 278	LIVE TOOLING
PARAM 298	Y AX RTAP BACKLASH
PARAM 304	SPINDLE BRAKE DELAY

NOTES:

- a) Live Tool spindle will turn itself off at tool change.
- b) Main spindle clamp will unclamp at spindle command or reset.
- c) Live Tool spindle max speed = 3000rpm.
- d) Canned cycles not supported for live tooling.

**PROGRAM EXAMPLE:****Bolt Hole Circle 3 holes @ 120° on 3" BHC**

```
G0 X3.0 Z0.1  
G98  
M19 P0  
M14  
M133 P2000  
G1 Z-0.5 F40.0  
G0 Z0.1  
M19 P120  
G4 P3          (Dwell for servo stabilization)  
M14  
G1 Z-0.5  
G0 Z0.1  
M19 P240  
M14  
G1 Z-0.5  
G0 Z0.1  
M15  
M135
```

**SYNCHRONOUS MILLING**

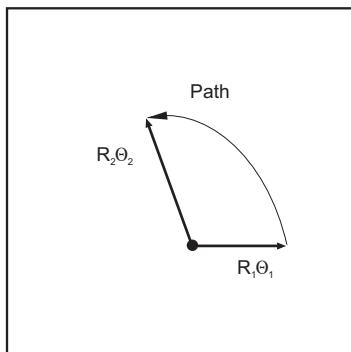
G32 synchronous motion is a control mode where the X, Z axes are commanded to move distances at constant feed rates and spindle is commanded to rotate at constant speeds. The motions of the X, Z, spindle start at the same time and at the same spindle reference mark each time thus the term synchronous, they are not interpolated motions only coordinated.

G32 is commonly used to create thread, the spindle rotates at a constant rpm and constant z motion begins at the same reference z mark for each stroke. Many strokes can be repeated because the reference mark sets the location of the start thread. Variable pitch threads can be created using G33.

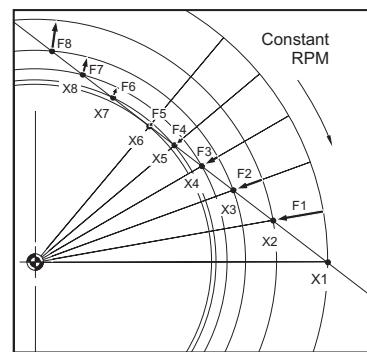


Geometric shapes can be created using G32. By varying the feed and distances of motion travel geometric paths can be approximated. Because each motion must be accelerated, timing the moves is critical to control the final path. The main spindle has a large mass it cannot accelerate as fast as the X or Z axes so common practice is to command the main spindle to a constant rpm and the X, Z motions are timed to start at the spindle reference mark. Each X, Z move must be pre-programmed to result in the desired path. A contour or flat may require many incremental motions, the combined effects of feed rate limit and control update time results in a limit to the number of points that can be processed.

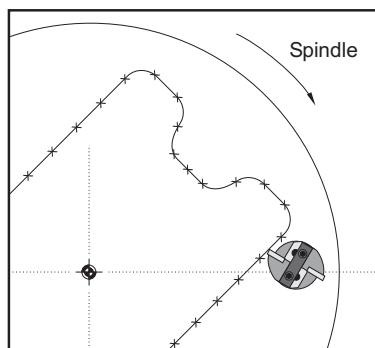
G32 motions can be cumbersome to create and difficult to adjust in the final program. The math required to calculate the point to point distance and feed commands involves trigonometry. Typically a spreadsheet table is generated by the programmer or a macro G code is written to obtain the commands. To relieve the user of this burden, the HAAS CNC control has a canned cycle G code available which simplifies the creation of simple geometric shapes. G77 is a general flattening algorithm that automates the motions of 1, 3 and more sided uniform shapes. It uses G32 motions and input variables for the angle offset, number of sides, diameter, etc... 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 Xfeed 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**

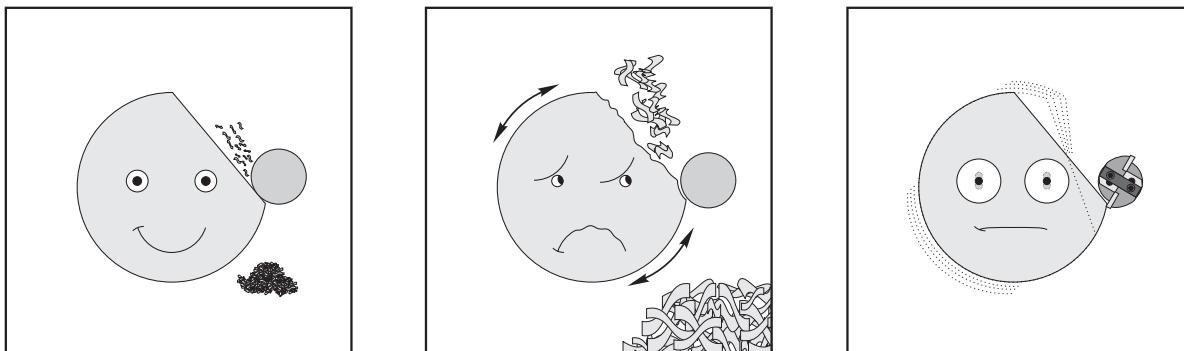
Many uses of live tooling involve holding the spindle still while performing a cut with the live tool. For certain types of operations, however, it is necessary to move this spindle in a controlled manner while cutting with the live tool. This section of the manual is a guide to the G-codes that are available to perform Fine spindle Control.

Uses for Fine Spindle Control

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.

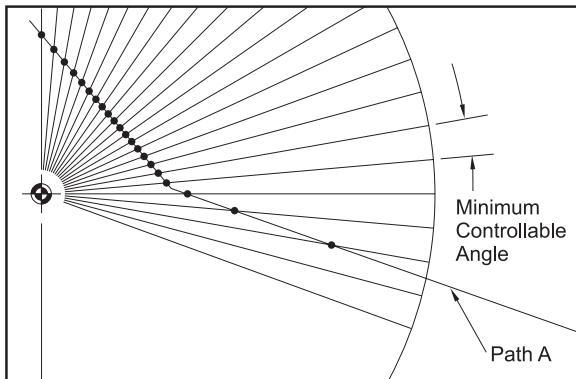
Limitations of Fine Spindle Control

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. The spindle was designed to turn rapidly, rather than precisely. Because of this, the accuracy with which the position of the spindle is known is .045 degrees. This limit also applies to positioning the spindle in general. This also 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. See Path A below.


PROGRAMMING EXAMPLES
G05 Fine Spindle Control motion (This G-code is optional and is used for live tooling)

Group 00

- R Angular motion of the spindle, in degrees.
- F Feed Rate of the center of the tool, in inches per minute.
- U Optional X-axis incremental motion command.
- W Optional Z-axis incremental motion command.
- X Optional X-axis absolute motion command.
- Z Optional Z-axis absolute motion command.

This G code is used to specify a precise motion of the spindle, and is intended to be used for slotting. Any motion specified along the X and Z axes tracks the spindle motion. Currently, the resolution of the R code value is .045 degrees.

The rotational speed of the spindle will remain constant throughout each G5 cut. If there is motion along the X axis during the G05, the actual feed rate will vary. The spindle speed is determined by looking at the greatest X value encountered during the cut. Therefore the specified feed rate will not be exceeded at any point along the cut.

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

Example 1

Simple Face Slot with G05

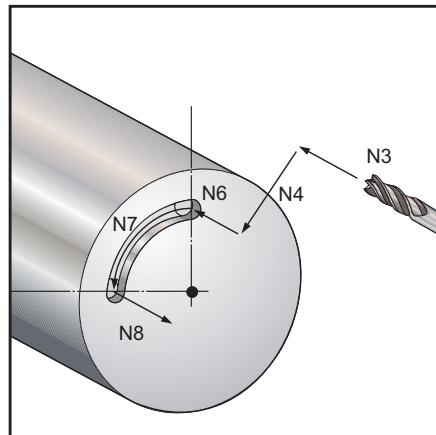
(Assume pilot hole is already drilled.)

N1 T303	(Small End Mill)
N2 M19	(Orient Spindle)
N3 G0 Z.5	



N4 G0 X1.
N5 G133 P1500
N6 G98 G1 F10. Z-.25
N7 G5 R90. F40.
N8 G1 F10. Z.5
N9 G135
N10G99 G28 U0 W0

(Plunge into pre-drilled hole)
(Make slot)
(Retract)

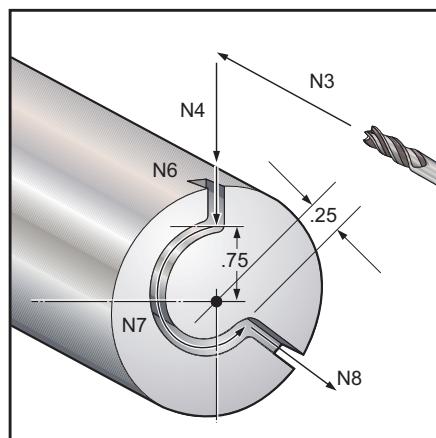


Example 2

Simple Cam with G05

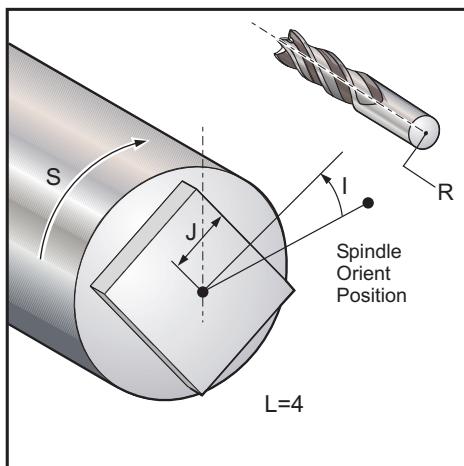
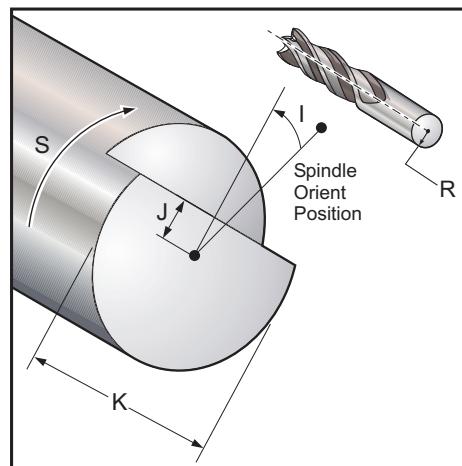
N1 T303
N2 M19
N3 G0 Z-.25
N4 G0 X2.5
N5 M133 P1500
N6 G98 G1 X1.5 F40.
N7 G5 R215. X.5 F40.
N8 G1 X2.5 F40.
N9 M135
N10G99 G28 U0 W0

(Small End Mill)
(Approach 2" dia. stock)
(Cut to top of cam)
(Cut Cam)
(Cut out of cam)



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

- * I Angle of first flat, in degrees.
- J Distance from center to flat.
- * L Number of flat surfaces to cut ($L \geq 3$)
- R Tool Radius
- * S Spindle Speed
- * K Part Diameter (not required)

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. The K value specifies the diameter of the part. Specifying a smaller diameter than the diameter of the actual part may cause the tool to crash into the part during its approach.

The L value allows a part with multiple flat surfaces to be specified. For example, L4 specifies a square, and L6 specifies a hex. 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. L2 is not supported, if two sides are desired perform two K cuts at I angle spacing. If L and K are both specified, alarm 339 MULTIPLE CODES is generated.

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 R value specifies the radius of the live cutting tool. It is important that this value is correct, as it is used for automatic tool compensation and the entry and exit motions.

The S value specifies the rpm speed that the spindle will maintain during the flatting cycle. If an S value is not specified on the G77 line, the part will be cut at 6 RPM. This value can be increased for small parts. Higher values will not affect the flatness, but will affect the position of the flats. To calculate the maximum error in degrees, use $RPM * 360 / 60000$, or $RPM * .006$.

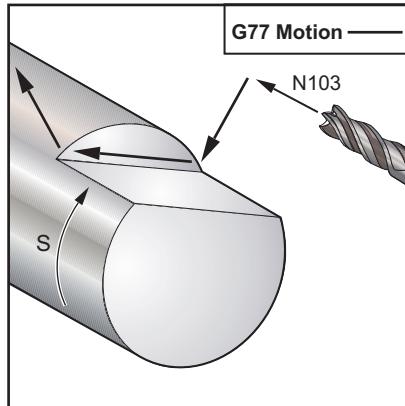
The I value specifies the offset of the center of the first flat surface from the zero position, in degrees. If the I value is not used, the first flat surface will start at the zero position. This is equivalent to specifying an I equal to half the number of degrees covered by the flat surface. For example,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

FLATTING EXAMPLES WITH G77

Example 1

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 M3          (Start spindle)
N101 M133 P1000      (Turn live tool)
N102 G0 X6.1
N103 Z-1.
N104 G77 J1.5 K4. R.5 I0
N105 Z1.
N106 M135 G77 I180. (Stop live tool)
N107 M5              (Stop spindle)
...
```



Example 2

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 You need
opposing flats)

N160 G77 J0.625 I180. R0.25 K2.

(J=1.25 flat dia, I180.=flat center, R.25=.5

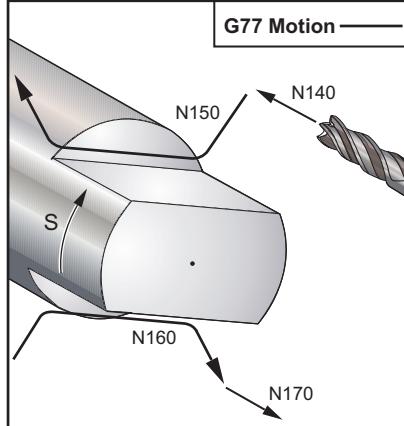
dia end-mill, K=part stock dia You need to
opposing flats)

N170 G0 Z1.
N180 M135
N190 M05
N200 G00 X10. Z12.
N210 M30

%

to run both for

run both for

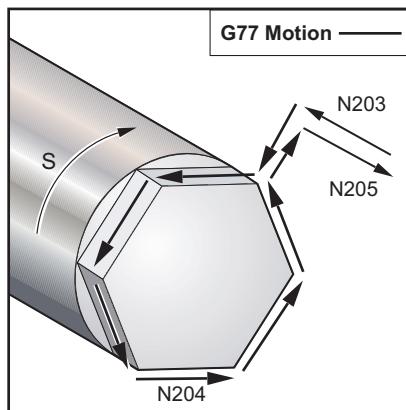




Example 3

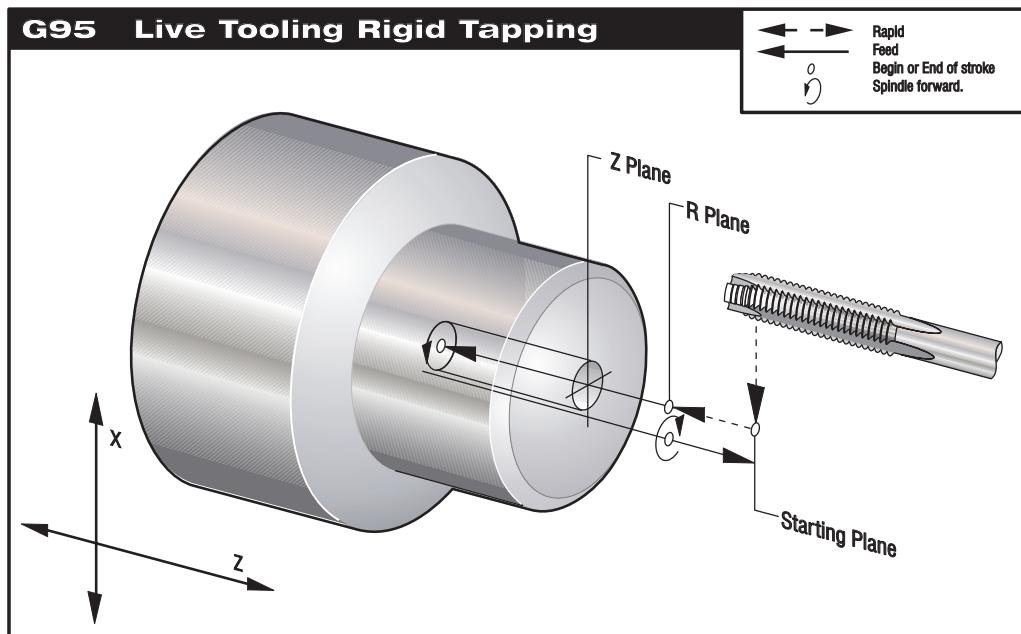
To cut a hexagon into the top half inch of a part that is three inches in diameter, using a tool half an inch in diameter:

...
N200 S10 M3 (Start spindle)
N201 M133 P1000 (Turn live tool)
N202 G0 X4.5
N203 Z-.5
N204 G77 J1.299 L6 R.25
N205 Z1.
N206 M135 (Stop live tool)
N207 M5 (Stop spindle)
...



G95 Live Tooling Rigid Tapping

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



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

You do not need 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

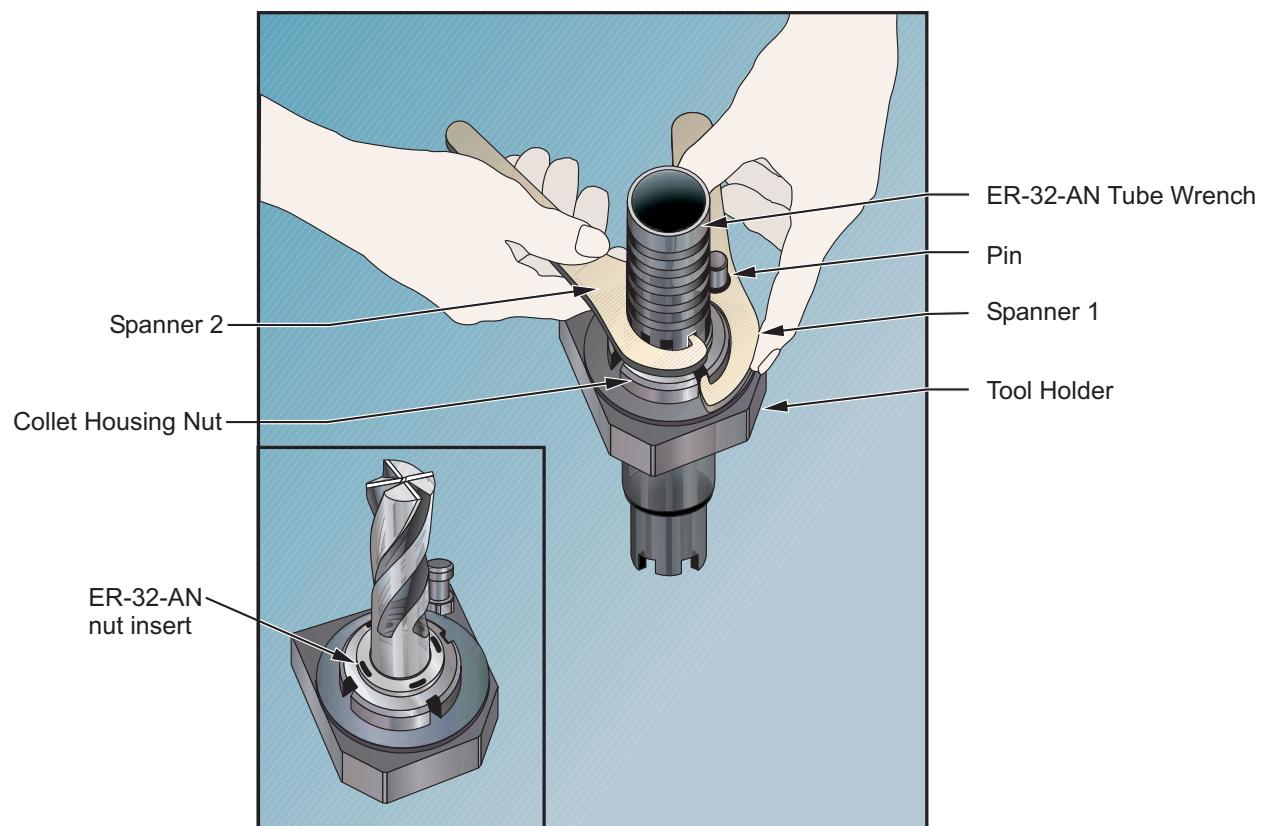


Currently tapping is supported in the Z-axis. G95 Live Tooling Rigid Tapping is similar to G84 Rigid Tapping in that it uses the F, R, X and Z parameters, however, it has the following differences:

1. The main spindle must be clamped (use M14) before G95 is commanded or an alarm will be generated.
2. The control must be in G99 FEED PER REVOLUTION mode in order for tapping to work properly.
3. An S (spindle speed) command must have been issued prior to the G95 because the specified spindle speed will be used to control the Live Tool speed.
4. The X axis can be positioned between zero and the center of the main spindle. If it is positioned beyond the center of the main spindle, an alarm will be generated.

RADIAL AND AXIAL TOOLING INSTALLATION

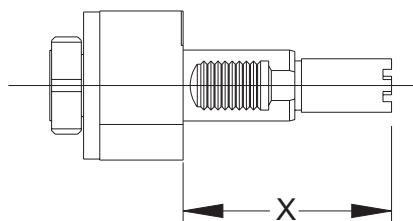
1. Insert the tool bit into the ER-AN nut insert. Thread the nut insert into the collet housing nut.
2. Place the ER-32-AN tube wrench over the tool bit and engage the teeth of the ER-AN nut insert. Snug the ER-AN nut insert by hand using the tube wrench.
3. Place Spanner 1 over the pin and lock it against 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.



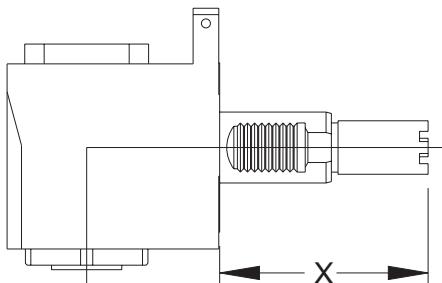


Live Tooling Tool Holders and Torque Chart

AXIAL TOOLHOLDER



RADIAL TOOLHOLDER

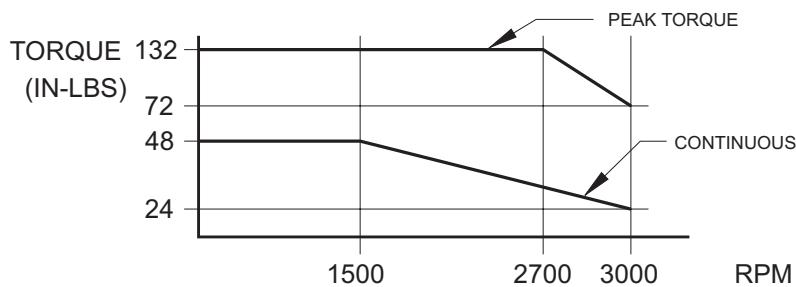


		HAAS TURRET *	VDI TURRET
in/(mm)		SL-20/30 TCE-550/550T/860/860T	SL-20/30 TCE-550/550T/860/860T
AXIAL TOOLHOLDER	X-DIM P/N	4.628 (117.55) P/N 59-0082	4.128 (104.85) P/N 59-0083
RADIAL TOOLHOLDER	X-DIM P/N	4.628 (117.55) P/N 59-0084	4.128 (104.85) P/N 59-0085

		SL-40	TCE-1100/T	SL-40/TL-15	TCE-1100/T
AXIAL TOOLHOLDER	X-DIM P/N	5.128 P/N 59-6352	(130.25)	4.628 P/N 59-0082	(117.55)
RADIAL TOOLHOLDER	X-DIM P/N	5.128 P/N 59-6353	(130.25)	4.628 P/N 59-0084	(117.55)

* VDI ADAPTER KIT (HAAS P/N VDIA) REQUIRED.

SPINDLE DRIVE PERFORMANCE



- Haas Live tooling is designed for medium duty milling; e.g. 3/4" diameter end mill in mild steel maximum.
- Large tool diameters may require reduction tool holders.
- Tool Length should be kept to a minimum for toolholder rigidity.
- Typical life of toolholder bearing is 1000 hours with medium duty operation.
- Worn toolholder bearings will not machine properly

SL TOOL ENVELOPE

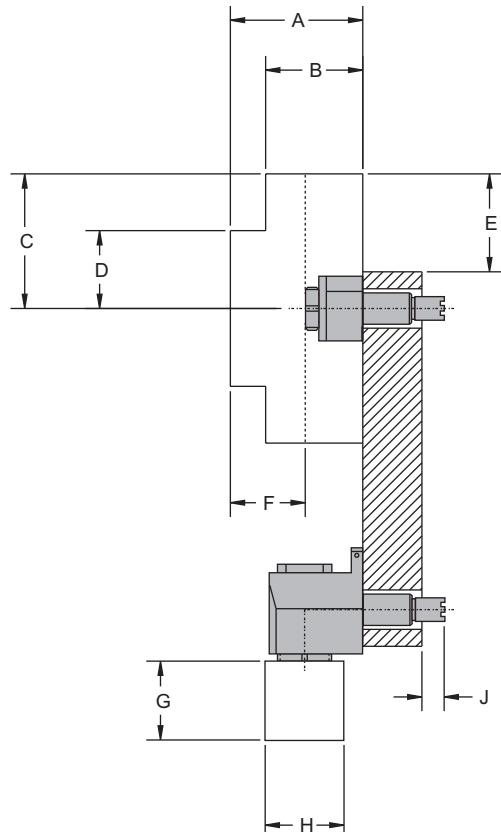
SL TOOL ENVELOPE

MAXIMUM TOOL SIZES BASED ON MACHINE FULL TRAVELS.
 TAILSTOCK AND WORK PIECE RESTRICTIONS NOT INCLUDED.
 LONGER TOOLS REDUCE Z-TRAVEL.
 ALL DIMENSIONS IN INCHES.

	SL-20/BB20	SL-30/BB30	SL-40/BB40			
	TCE 550/T	TCE 860/T	TCE 1600/T			
A	6.90	6.72	10.72			
B	4.75	4.93	5.50			
C	5.54	6.83	7.57			
D	3.55	4.01	5.53			
E	3.86	4.97	5.89			
*	TURRET TYPE HAAS VDI	TURRET TYPE HAAS VDI	TURRET TYPE HAAS VDI			
F	3.4	3.9	3.3	3.8	7.3	7.8
G	2.89	2.83	4.12	4.12	4.86	4.86
H	2.5	3.5	3.0	4.0	4.0	5.0
BEHIND TURRET						
J	1.125		1.125		1.125	

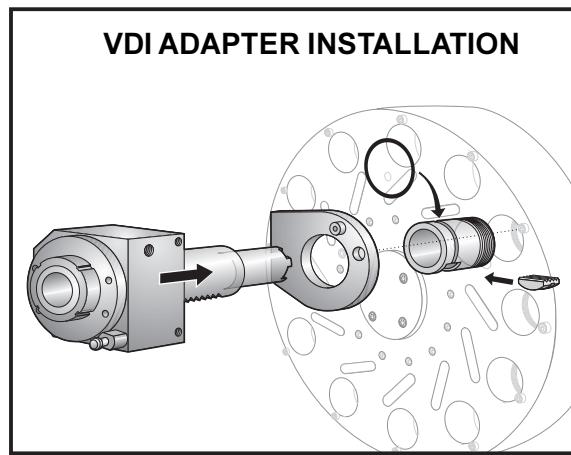
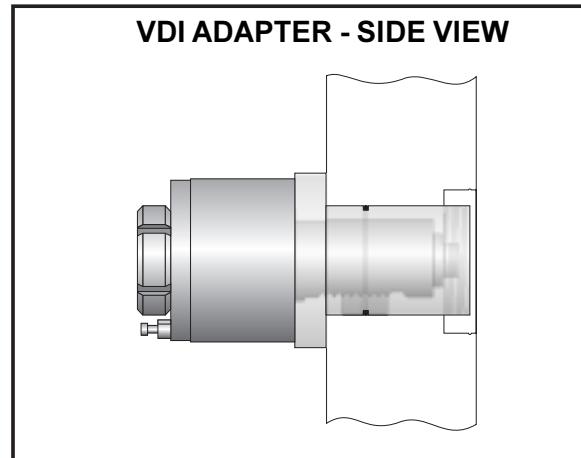
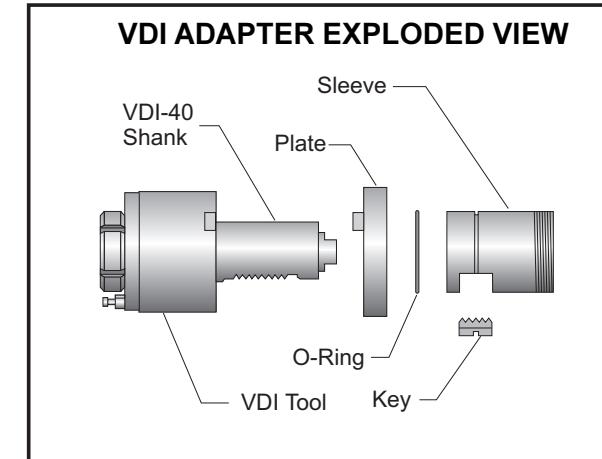
* LIVETOOLING IN HAAS TURRETS REQUIRES 1/2 INCH ADAPTER PLATE.
 (REF: ES-0122 VDI-40 TO HAAS ADAPTER KIT. HAAS P/N: VDIA)

NOTE: DOCUMENT ONLY APPLIES TO MACHINES WITH POCKETED Z-WAYCOVERS AND THE
 FOLLOWING Z-AXIS TRAVELS: SL-20/BB20=19", SL-30/BB30=33", SL-40/BB40=44".
 SPECIFICATIONS SUBJECT TO CHANGE WITHOUT NOTICE.



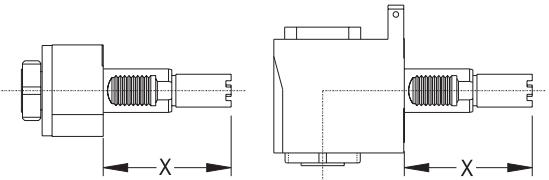
**VDI ADAPTER INSTALLATION****IMPORTANT!**
VDI ADAPTERS MAKE IT POSSIBLE TO USE VDI-40 TOOLS IN HAAS TURRETS**INSTALLATION PROCEDURE:**

1. Install plate over VDI-40 tool shank. Orient plate boss to VDI tool counterbore
2. Slide adapter sleeve onto tool shank with cut-out facing towards base of tool shank. Align cut-out with tooth profile of shank.
3. Insert key into sleeve cut-out. Ensure tooth profile of key fits into tool shank properly.
4. Place o-ring over O-ring 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.



HAAS LIVE TOOLING

AXIAL TOOLHOLDER **RADIAL TOOLHOLDER**



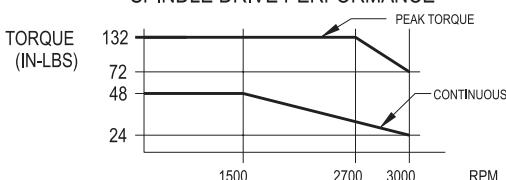
in/(mm)

HAAS TURRET *		VDI TURRET	
SL-20/30 TCE-550/550T/860/860T		SL-20/30 TCE-550/550T/860/860T	
AXIAL TOOLHOLDER	X-DIM P/N	4.628 (117.55) P/N 59-0082	4.128 (104.85) P/N 59-0083
RADIAL TOOLHOLDER	X-DIM P/N	4.628 (117.55) P/N 59-0084	4.128 (104.85) P/N 59-0085

SL-40		TCE-1100/T		SL-40		TCE-1100/T	
AXIAL TOOLHOLDER	X-DIM P/N	5.128 (130.25) P/N 59-6352		4.628 (117.55) P/N 59-0082		4.628 (117.55) P/N 59-0084	
RADIAL TOOLHOLDER	X-DIM P/N	5.128 (130.25) P/N 59-6353		4.628 (117.55) P/N 59-0084		4.628 (117.55) P/N 59-0084	

* VDI ADAPTER KIT (HAAS P/N VDIA) REQUIRED.

SPINDLE DRIVE PERFORMANCE



PEAK TORQUE

TORQUE (IN-LBS)

132

72

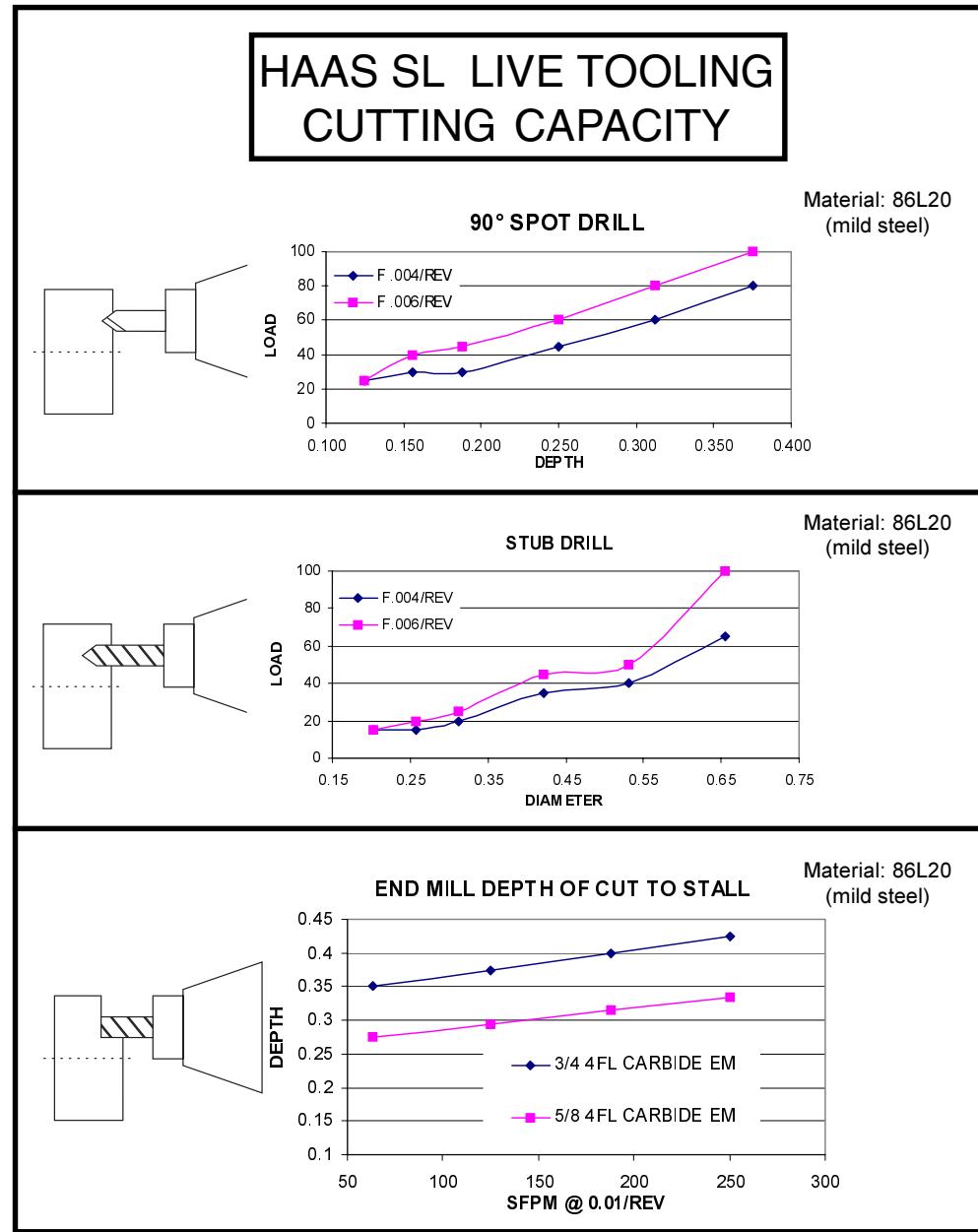
48

24

1500 2700 3000 RPM

CONTINUOUS

- 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 TOOLHOLDERS.
- TOOL LENGTH SHOULD BE KEPT TO A MINIMUM FOR TOOLHOLDER RIGIDITY.
- TYPICAL LIFE OF TOOLHOLDER BEARING IS 1000 HOURS WITH MEDIUM DUTY OPERATION.
- WORN TOOLHOLDER BEARINGS WILL NOT MACHINE PROPERLY.



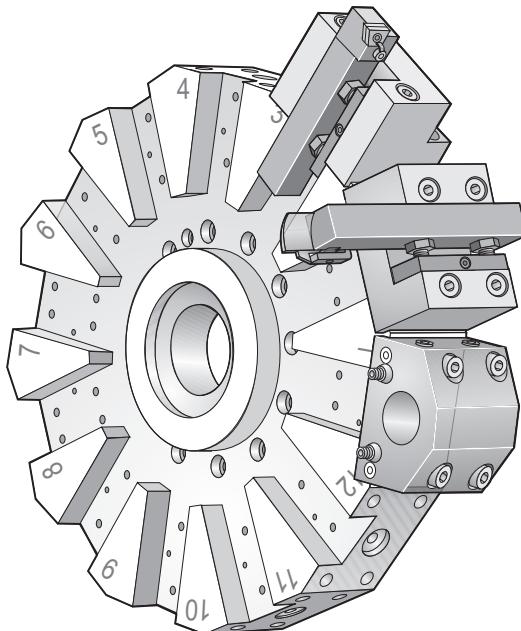


OPTIONS

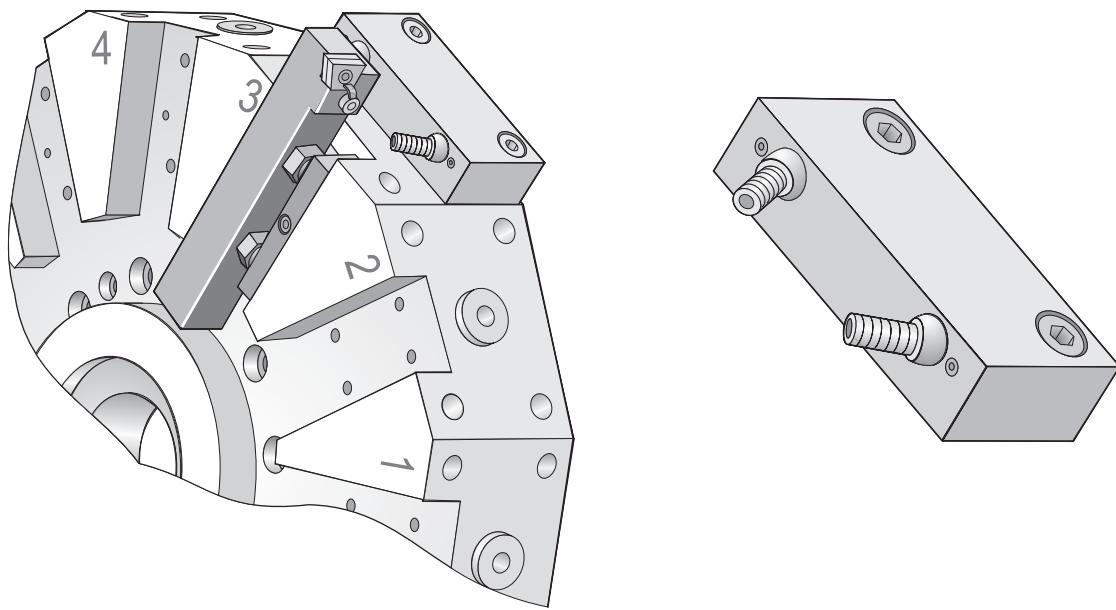
11.12 HAAS BOLT-ON TURRET

The Bolt-On Turret option consists of a special turret disk that accepts bolt-on tools around the perimeter. This turret also features an equal number of standard radial slots on the face for mounting tuning tools in either right- or left-hand directions.

The SL-20 and SL-30 turrets are compatible with LB-25 bolt-on tooling, and they accommodate boring bars up to 2" and accept standard 1" square turning tools. The SL-40 turret is compatible with LB-45 bolt-on tooling and it accommodates boring bars up to 2.5" and accepts standard 1.25" square turning tools.



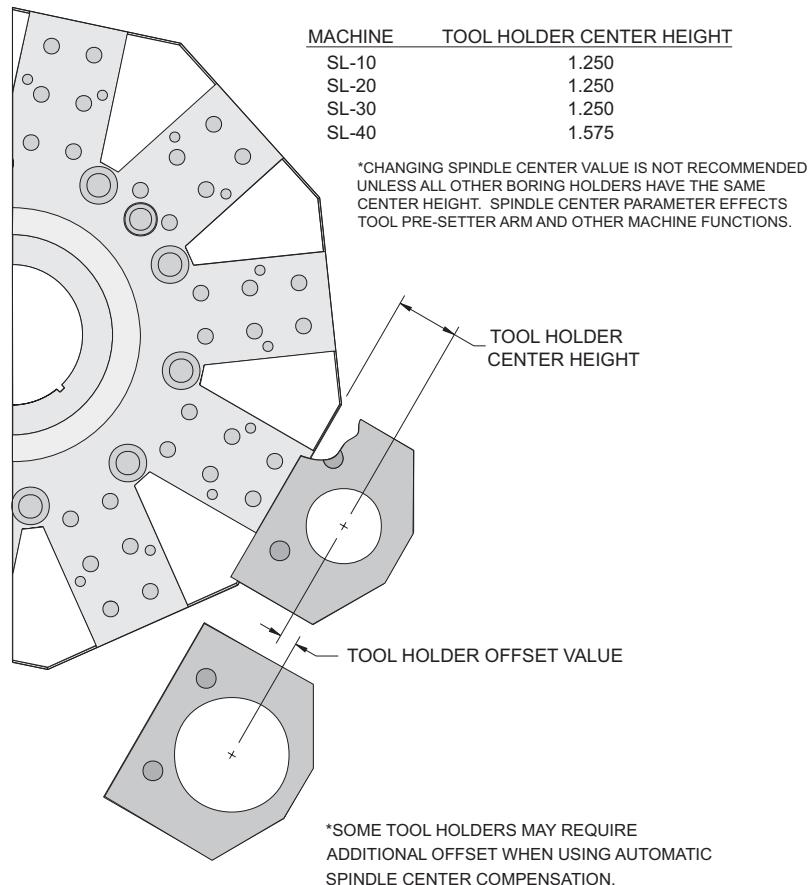
SL20	10 Tool Turret
SL30	12 Tool Turret
SL40	10 Tool Turret



Coolant Nozzle Block

BOLT ON TURRET SETUP

SPINDLE CENTER VALUE USED IN TOOL OFFSET TABLES HAS
BEEN PRE-SET BY THE FACTORY BASED ON THE FOLLOWING VALUES:

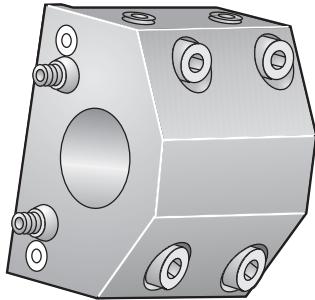
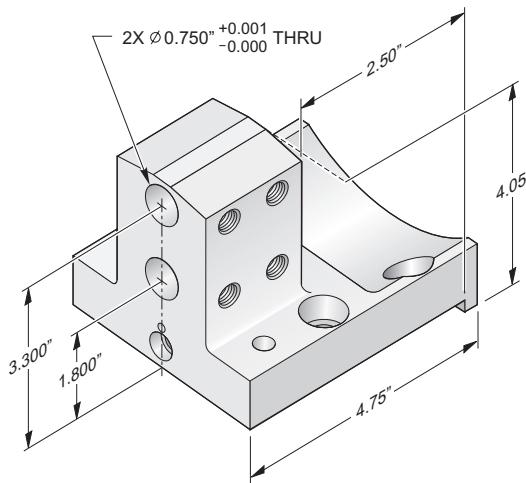
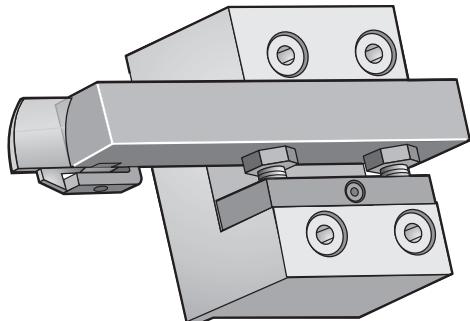
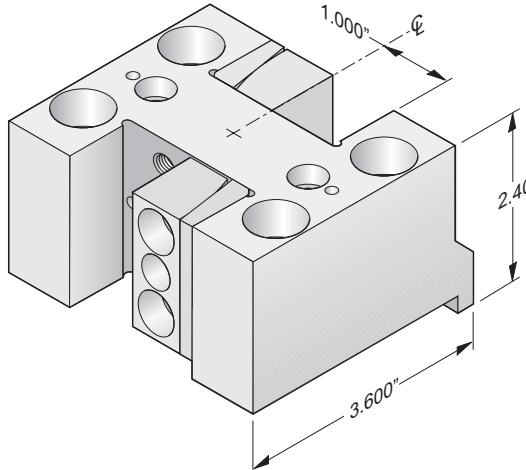
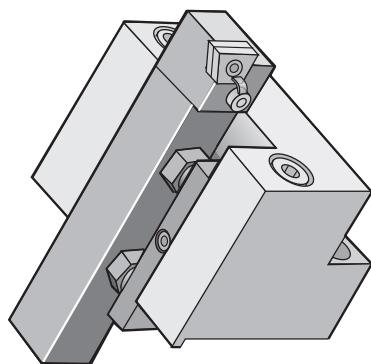
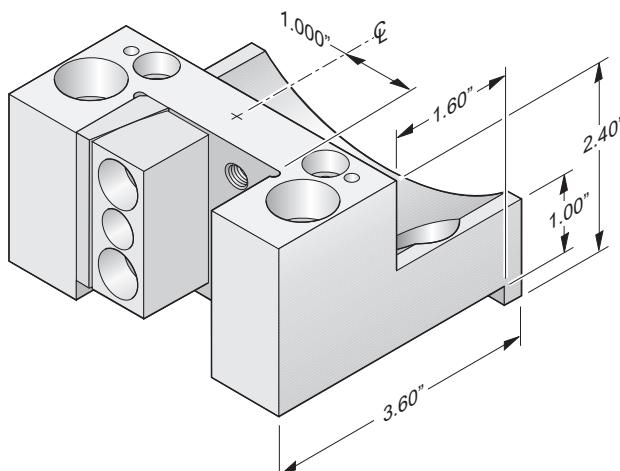




OPTIONS

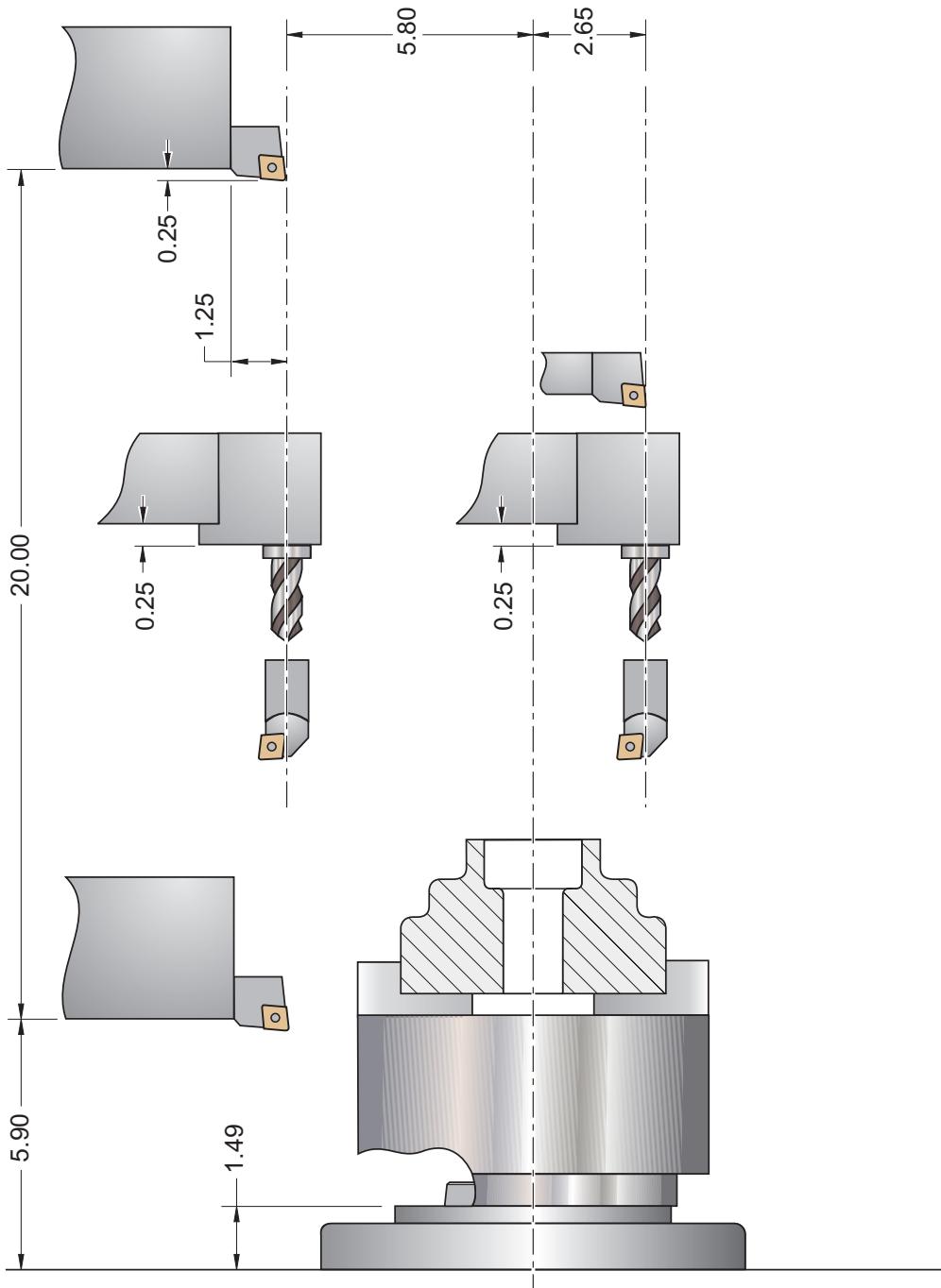
SL Series OPERATOR'S MANUAL

June 2001

BOLT ON TURRET TOOL HOLDERS**Boring Bar Holder**Available Sizes for the SL 20 and SL30
1.000, 1.250, 1.500, 2.000**Twin Boring Tool**
3/4 INCH**Face Grooving Holder**Available Size for the SL20 and SL30
1.000**Twin Turning Tool**
1 INCH**O.D. Turn Extension Block**Available size for the SL20 and SL30
1.00**Parting Tool**
1INCH

SL40 tool holders sold in Tooling Package only

Axis Travel Diagram SL-20 BOT

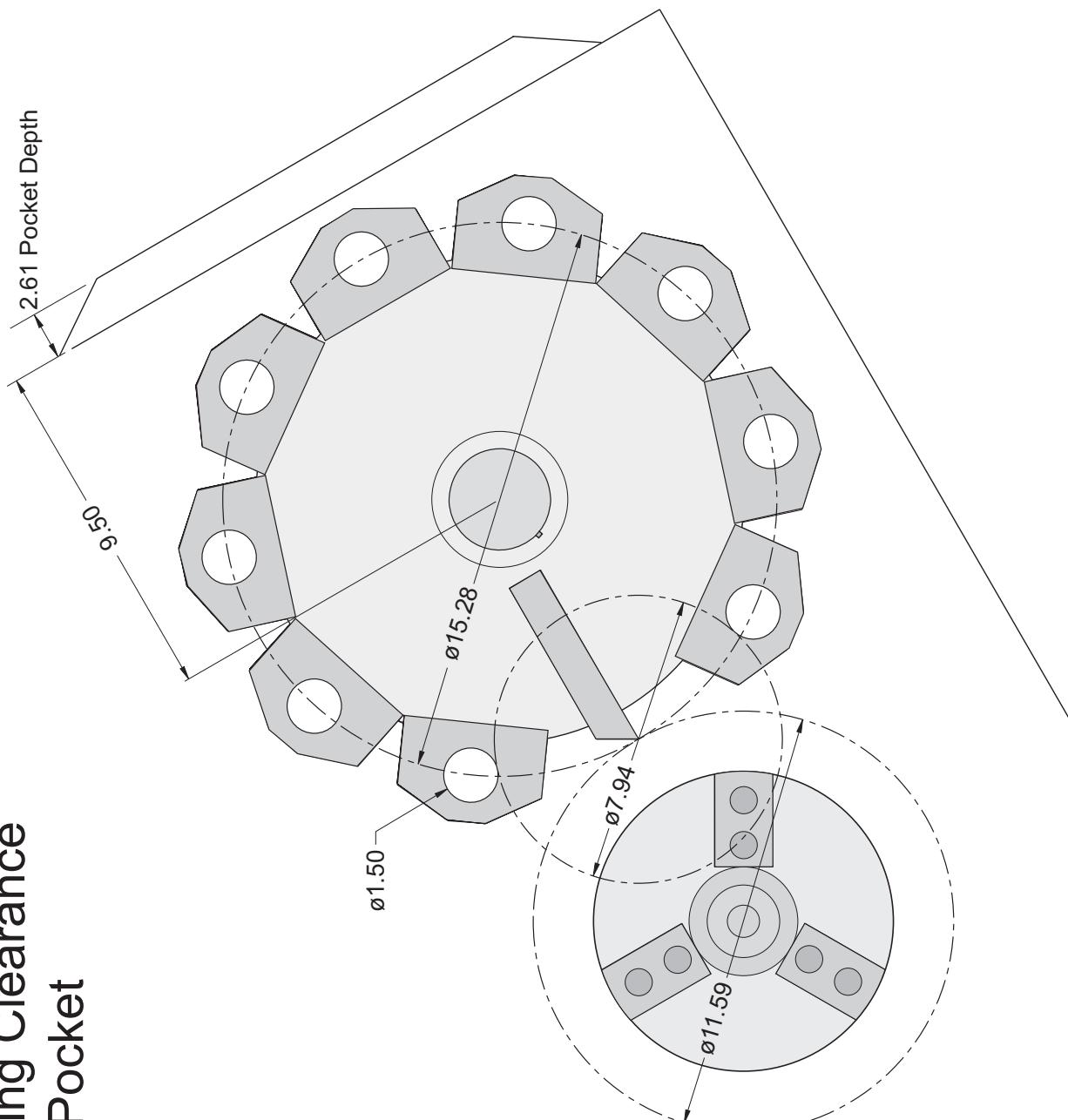




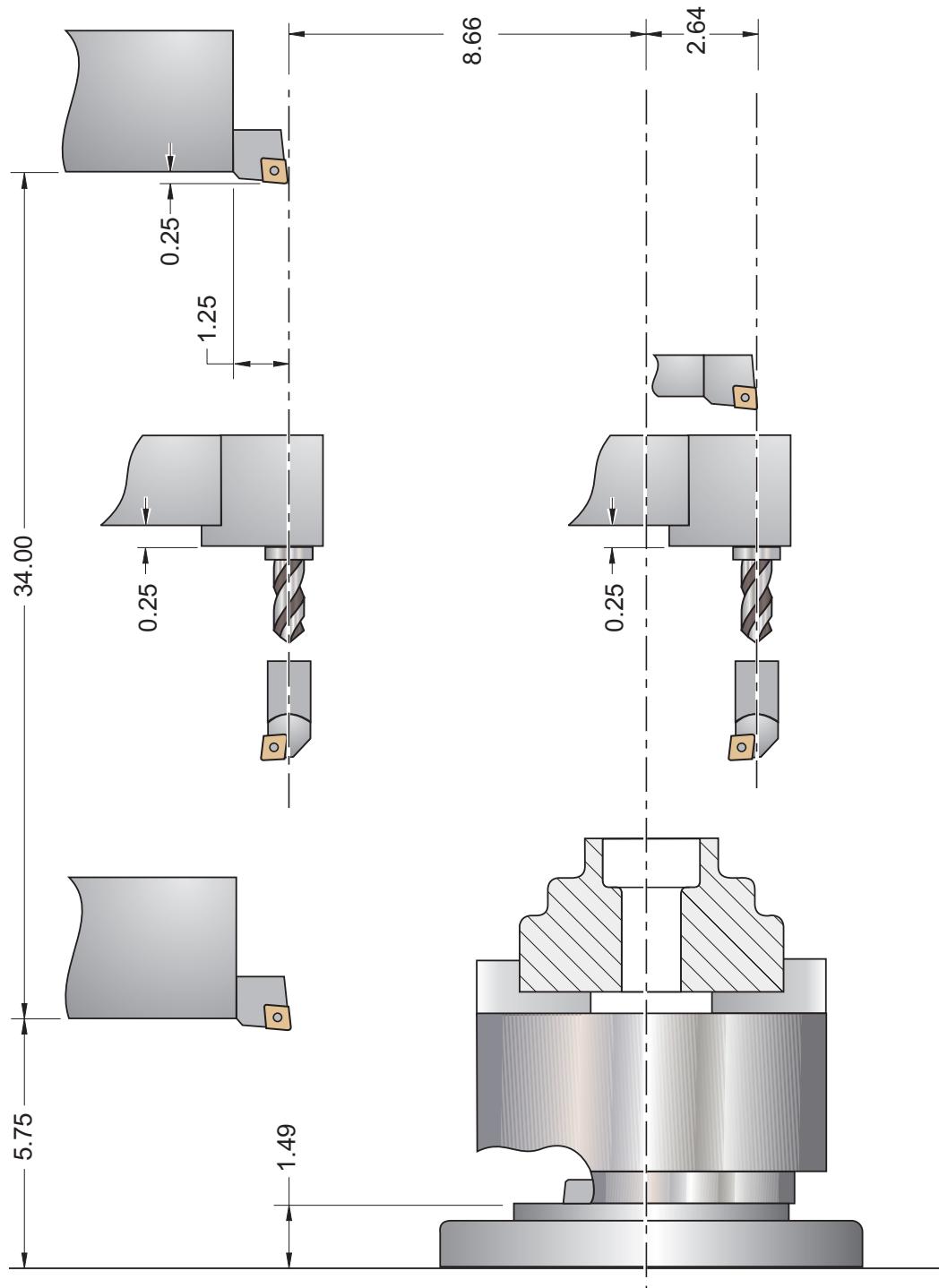
OPTIONS

SL OPERATOR'S MANUAL
Series

June 2001

**Turret Tooling Clearance
SL-20 10 Pocket**

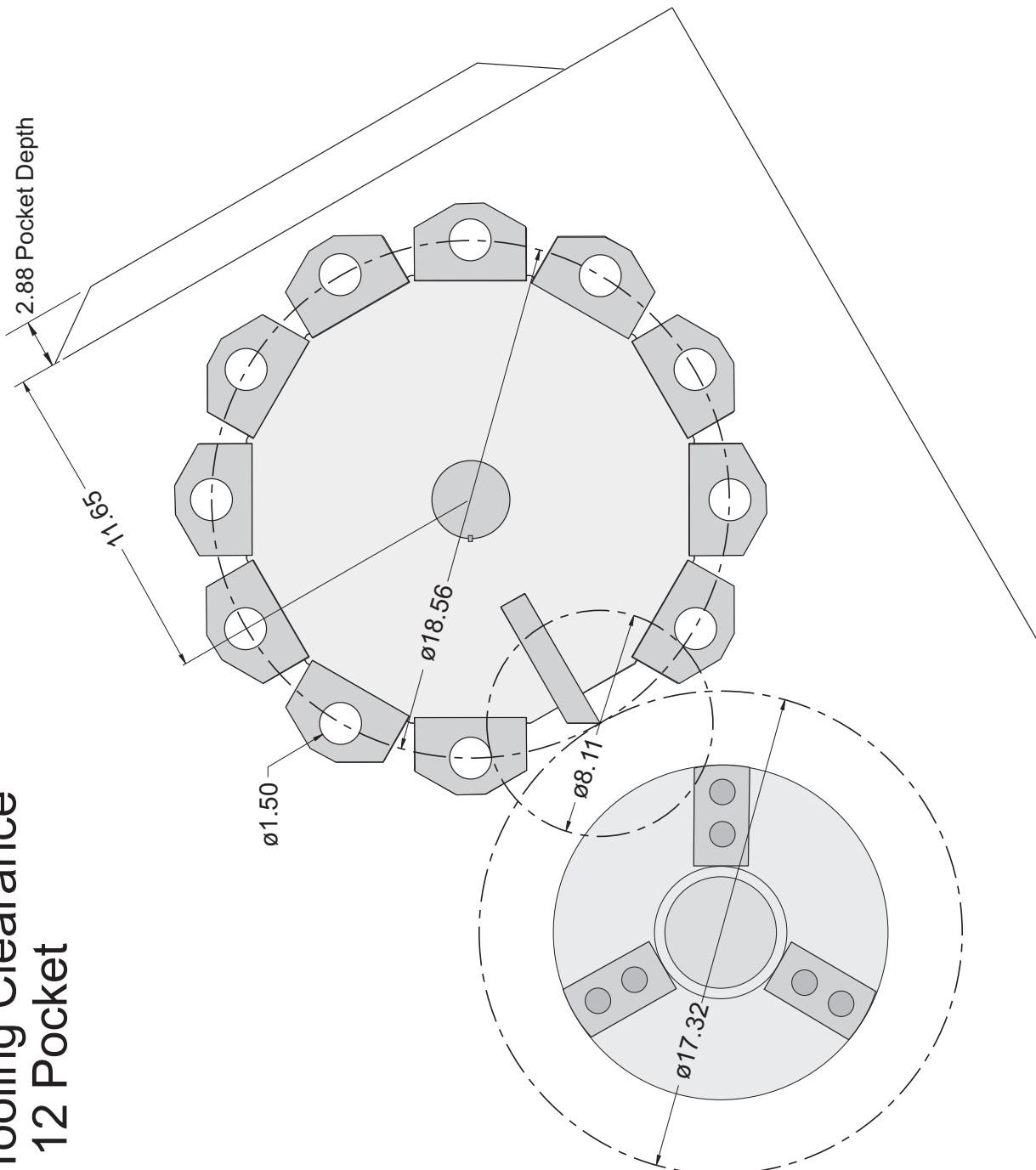
Axis Travel Diagram SL-30 BOT



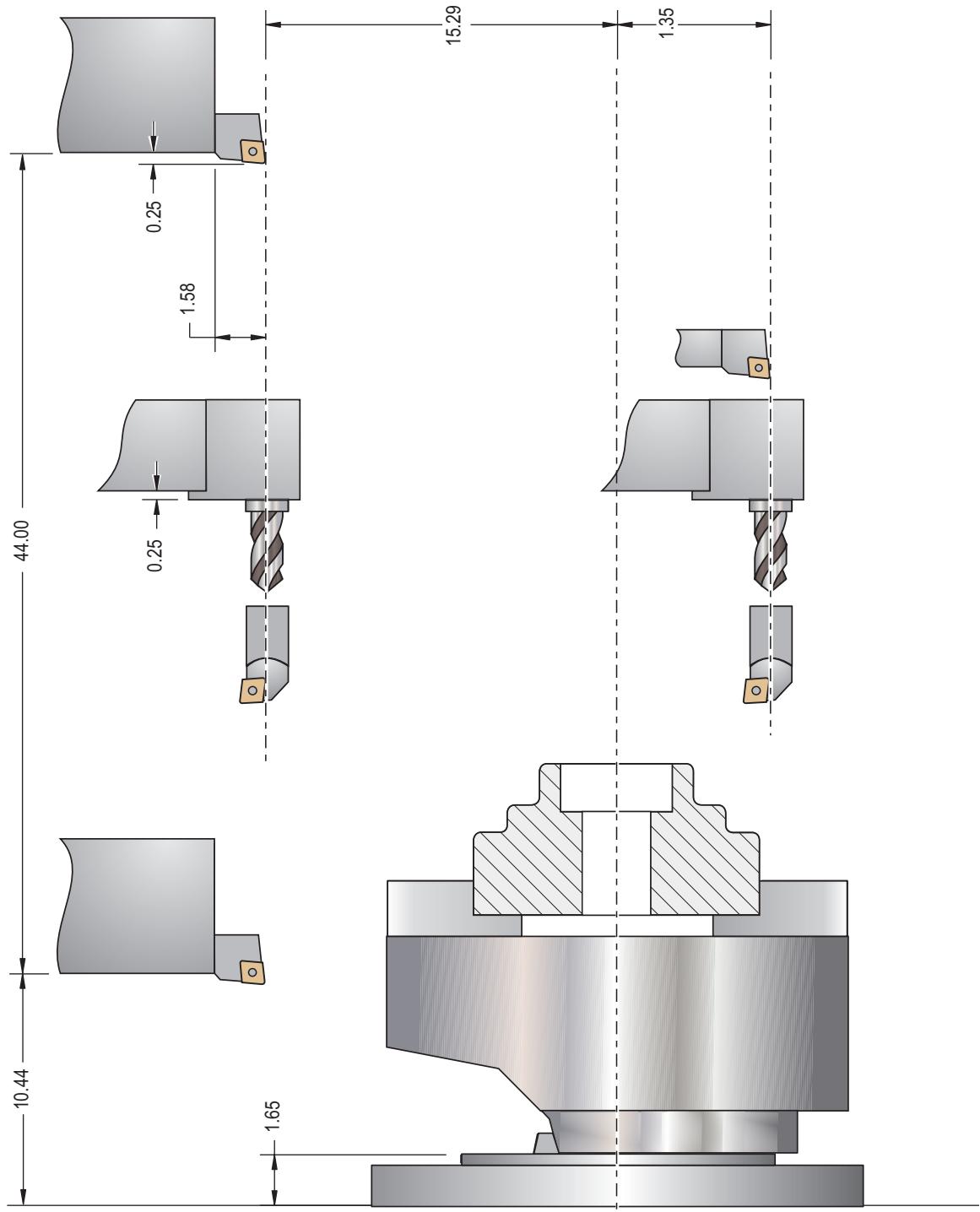


OPTIONS

Turret Tooling Clearance SL-30 12 Pocket



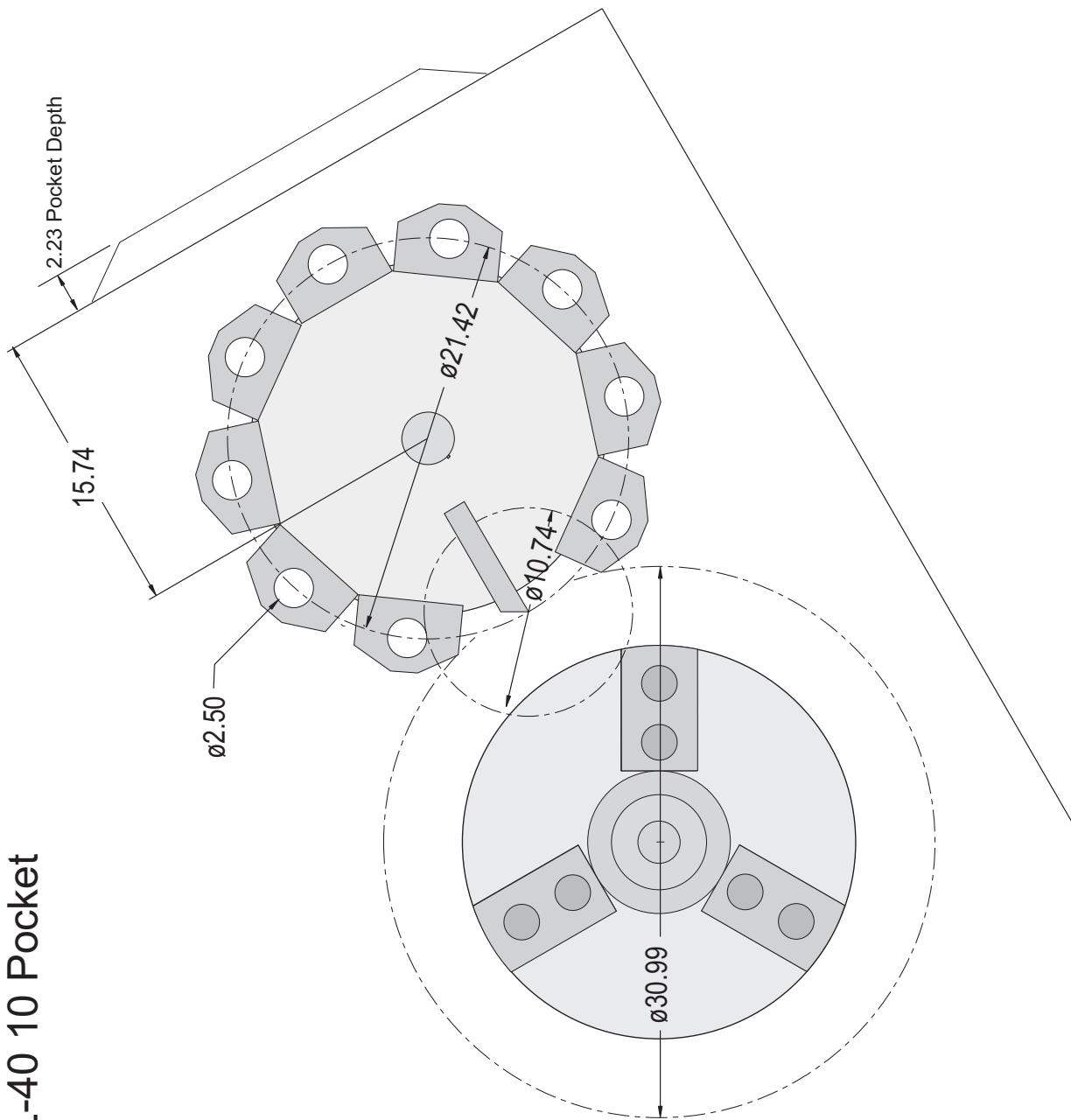
Axis Travel Diagram SL-40 BOT





OPTIONS

Turret Tooling Clearance SL-40 10 Pocket



11.13 HIGH PRESSURE COOLANT SYSTEM*

*This option is not field installable

Priming the high pressure coolant system

Rotate the turret to an open pocket (no tool holder in it) and open the coolant ball valve on the front of the tool changer. Close the door. Switch to MDI mode. Program M88, and press Cycle Start. Run the pump for one minute. Press Reset to turn coolant off.

Operation

Use Macro Command M88 to turn **ON** and M89 to turn **OFF** High pressure coolant.

- Prime the HPC system when starting up the machine for the first time or after cleaning the coolant tank or the filter. Priming may have to be done if the machine has been idle for more than a day.
- Keep the coolant tank full. High pressure coolant will use more than normal. Top off the coolant tank after every 8 hour shift.
- **Do Not** turn on the normal coolant during HPC operation (No M08 command).

SHIFT/COOLNT Button

NOTE: This function does not work when the control is in MEM mode.

HPC (High Pressure Coolant) can be activated by pressing the SHIFT button followed by the COOLNT button. Note that as HPC and regular coolant share a common orifice, they cannot both be on at the same time. This is what happens when the following key sequence is pressed:

COOLNT (turns on regular coolant)
 COOLNT (turns off regular coolant)
 SHIFT/COOLNT (turns on HPC)
 SHIFT/COOLNT (turns off HPC)
 COOLNT (turns on regular coolant)
 SHIFT/COOLNT (turns off regular coolant, turns on HPC)
 SHIFT/COOLNT (turns off HPC, leaves regular coolant off)
 SHIFT/COOLNT (turns on HPC)
 COOLNT (turns off HPC, turns on regular coolant)
 COOLNT (turns off regular coolant, leaves HPC off)

WARNING!

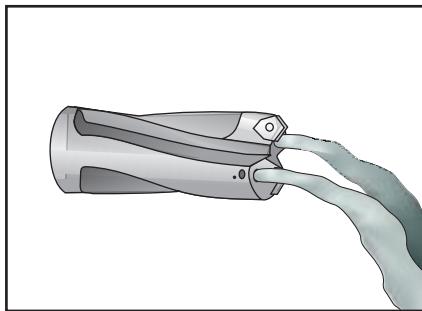
Turn off High Pressure Coolant before performing a tool change. Use M89 to turn off High Pressure coolant during program execution before rotating the turret.



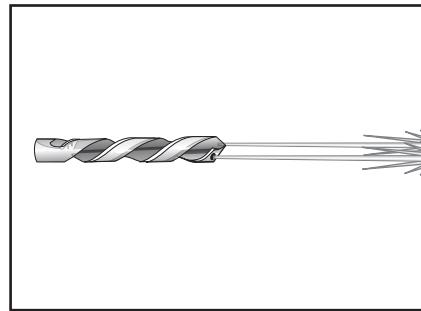
OPTIONS

HPC PRESSURE EFFECTS

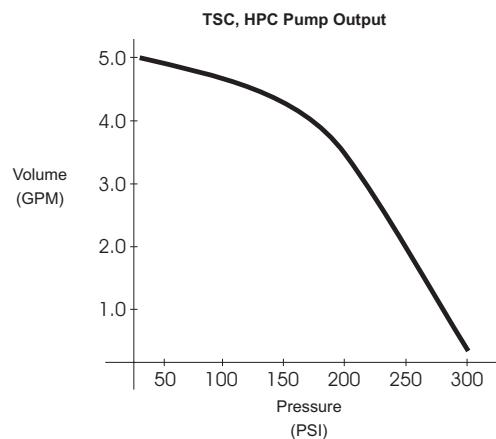
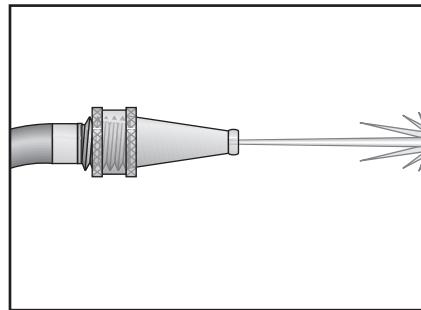
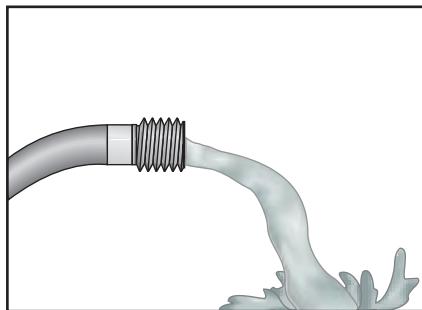
On machines using TSC and/or HPC during cutting operations, tooling size will have to be taken into consideration. As shown below, proper TSC and/or HPC system operation will deliver different pressures at the orifice of the tool; this depends on the diameter and number of coolant passages in the tool.



Larger tooling has larger diameter coolant passages. Coolant flow is higher at lower pressures.



Smaller tooling has smaller diameter coolant passages. This produces higher pressures at lower flow.



11.14 REMOTE JOG HANDLE

An optional remote jog handle is available. Its operation is exactly the same as the standard jog handle, except that the desired axis and jog increments can be selected by switches on the remote handle.

The jog axis switch on the remote handle may be switched to OFF, X, Z, or B. When it is set to OFF, the standard jog handle on the control works normally. When X, Z, or B is selected, that axis is selected for jogging by the remote handle. The jog increments switch may be switched to X1, X10, or X100. These correspond to the .0001/.1, .001/1., and .01/10 buttons, respectively, on the keypad.

The CYCLE START and FEED HOLD buttons on the remote jog handle perform the same exact functions as the same buttons on the control. They cannot be turned off, and can be used at any time.

11.15 LATHE 7,000 RPM SPINDLE*

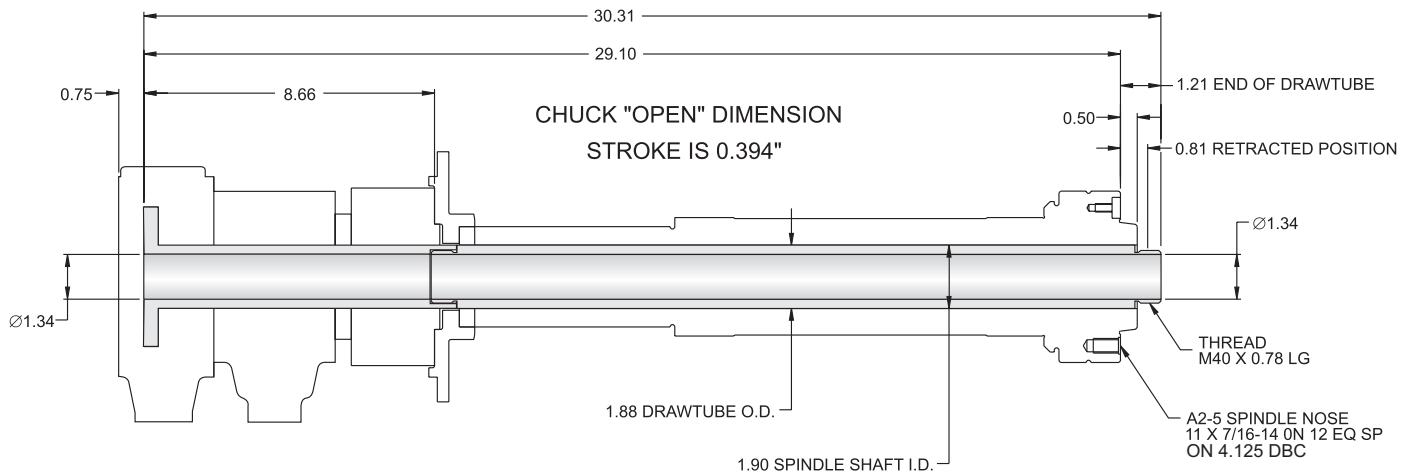
*This option is not field installable

For very high speed turning of smaller parts, the Haas SL-20 can be configured with a 7,000 rpm / 20 HP spindle. The 7K Spindle option for the SL-20 dramatically reduces cycle times while also providing an improved surface finish.

The 7K Spindle Features:

- 10" x 20" turning capacity,
- An A2-5 Spindle Nose,
- A cartridge-style spindle,
- A 1.34" through-hole with 1.31" bar capacity,
- A 5.31" dynamically balanced 3-jaw power chuck with hydraulic closer.

7000 RPM Drawtube Specifications





OPTIONS

11.16 Auto Air Jet*

*This option is not field installable

The Auto Air Jet can be programmed to blow air at the chuck in order to clear chips and coolant between operations.

OPERATION

M12 turns on the Auto Air Jet.
M13 turns off the Auto Air Jet.



11.17 LATHE AUXILIARY FILTER SYSTEM

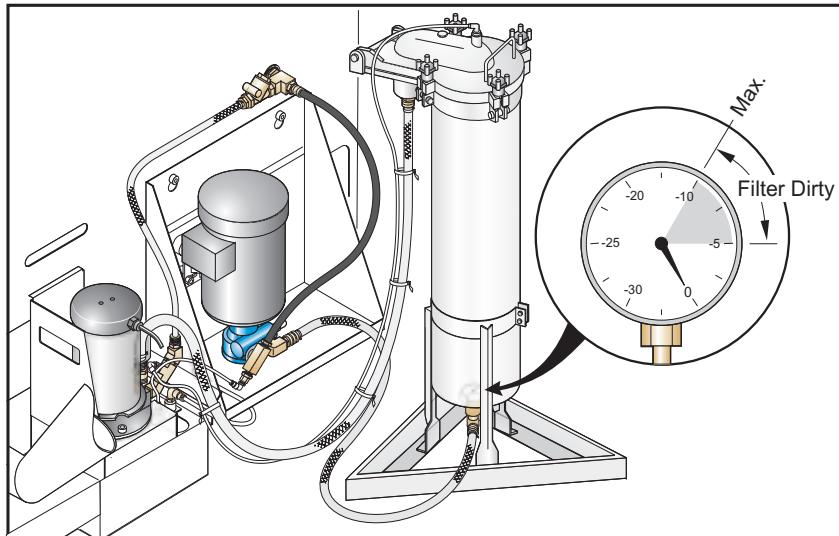
This Auxiliary Filter is used to protect the high pressure pump from particle damage. It is recommended for customers doing medium to high production machining of cast aluminum, cast iron or titanium. It may also be useful for customers who perform high speed operations and produce small powder-like chips.

The filter unit consists of a #2 filter bag, filter housing, pipe fittings and hoses. The large size of the filter bag will extend the time interval between bag changes. To reduce wear and prolong the filter system, a small plastic hose is connected between the filter housing and the primary coolant pump to prime the system with coolant. The filter is automatically topped off whenever coolant is used. The amount of flow through the priming hose has an unnoticeable effect on normal coolant operation.

REPLACEMENT FILTER BAGS

Change the filter bag when the filter gauge indicator displays a vacuum level of -5 inHg or more. Do not allow the suction to exceed -10 inHg or pump damage may occur (refer to the figure below). HAAS recommends using 25 micron rated filter bags (one is provided with the unit). Replacement bags can be purchased from local filter suppliers (#2 trade size filter bag) or from HAAS (Part No. 93-9130). Finer micron ratings can be used if desired.

***Important** - Run the primary coolant system for four minutes to prime the bag filter on initial startup or after changing filter bags.



Filter Gauge Indicator

**11.18 Auto Door Option**

The door is commanded to open and close automatically in concert with cutting operations. As an additional incorporated safety feature, when the Cycle Start button is pressed, the CNC control will automatically poll the door switch to see if the door is open or closed. Should the door still be open, the CNC control will close the door before any operations are begun.

The Auto Door is opened and closed by way of a motor-driven link chain. The mechanism can be seen in the illustration above.

The main functionality of the Auto Door is to ease the burden of the operator. The Auto Door can be set to open after the completion of a program, facilitating part changing. When a new cycle or program is begun, the door will automatically close.

OPERATION

Operation of the Auto Door feature has been automated. No special programming is required. To activate the Auto Door, set Parameter 251 to a value between 2400 and 4500. This is the amount of time in milliseconds that the Auto Door has to open. During normal operation the door will close when the Cycle Start button is pressed and open when an M30 code is encountered in the program.

Additionally, with Setting 131 AUTO DOOR ENABLE set to **ON**, an M00 and an M01 will open the Auto Door. Parameter 57 SAFETY CIRC must be set to **0** and Parameters 235, 236, and 251 must be set appropriately.

M Codes**M85 Open Automatic Door**

On lathes fitted with an auto door, an M85 will cause the door to open and an M86 will cause it to close. To use this feature, the following settings must be made:

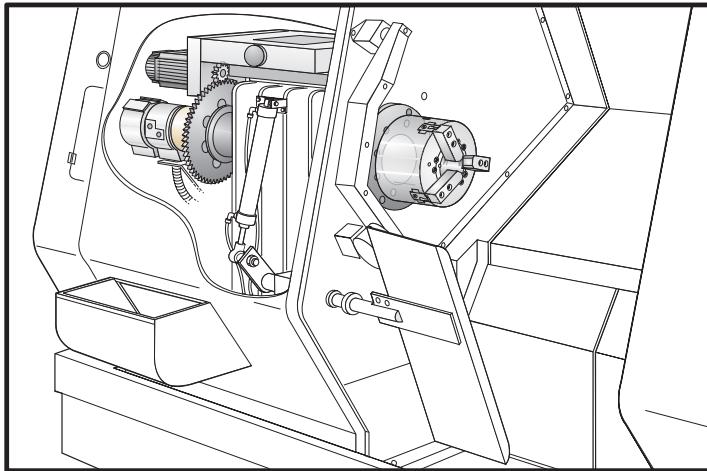
- Setting 51 DOOR HOLD OVERRIDE set to ON,
- Parameter 57 bit 31 DOOR STOP SP set to zero,
- Setting 131 AUTO DOOR set to ON.

The control will beep while the door is in motion.

M86 Close Automatic Door

11.19 C-AXIS

This option provides high-precision bi-directional spindle motion that is fully interpolated with X and / or Z motion. The spindle positioning is accurate to +/- 0.01 degree, and speeds from .01 to 60 RPM can be commanded.


OPERATION

M154 C-axis engage
 M155 C-axis disengage
 RPM range .01-60
 Accuracy +/- .01 degree
 Torque 100 ft/lbs.

Setting 101 Diameter used to calculate the feed rate

Parameter 373 Grid Offset (This parameter is set at the factory and is used to match the gear mesh of the two gears)

C-Axis Quick Rewind

When the C-axis is engaged with M154, the control first zeros the spindle and C-axis motor so the gears will mesh. When an M154 is commanded, the control will unwind the C-axis no more than half a rotation. The quick unwind will also be used when the C-axis is zeroed or homed such as with G28.

C-Axis Incremental Move

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.)



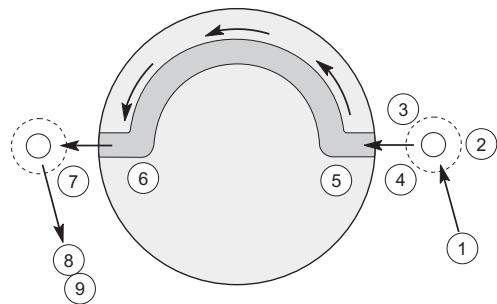
OPTIONS

SL Series OPERATOR'S MANUAL

June 2001

SAMPLE PROGRAM

1. M154
2. G0 G98 (feed/min) X2.0 Z0.5
3. C90
4. G1 Z-0.1 F6.0
5. X1.0
6. C180. F10.0
7. X2.0
8. G0 Z0.5
9. M155



**11.20 CARTESIAN TO POLAR TRANSFORMATION****INTRODUCTION**

Polar coordinates are specific to rotary applications. Position is calculated based on C angular degrees from a fixed reference line and a given radius defined by X. The C component can be either a positive or negative degree of rotation, while X starts at the origin and is always positive. If looked at like the concentric circles of a bull's-eye target, the origin is in the center. Angular degrees are measured between a horizontal line to the right of the origin and a line from the origin to the point being located. X is the length of the line from the origin to the point.

Cartesian coordinates are the common programming format for milling applications. The X-Y Cartesian plane forms a grid of unit measurements. From the origin coordinates of X zero and Y zero a position can be measured in units in a positive or negative distance along each axis. If looked at like a sheet of graph paper, the origin is in the middle, right is positive X, left is negative X, top is positive Y and bottom is negative Y.

Both Cartesian and Polar Coordinate systems share a common Z-axis passing through their respective origins. Z units are either positive or negative depending on which side of the Cartesian or Polar plane the point in question resides, up or away being positive and down/into labeled negative.

Any point in space can be referenced from a common origin by either coordinate system. For example: A polar point of X 1.414 and C of 45 degrees, converts as a Cartesian equivalent of X 1, Y1.

Cartesian to Polar coordinate programming is a feature that converts user inputs of X,Y position commands into rotary C-axis and linear X moves. Cartesian to Polar coordinate programming greatly reduces the amount of code required to command complex moves. A straight line machined in Polar coordinates would require many programmed points to define the path, while in Cartesian, only the end points are necessary. Utilizing both the Live Tool and C-axis Lathe options, linear moves that previously required many lines of code can be completed in a single line. This feature dramatically reduces the length of a program. It creates a program that is compact and easier to read. Live tooling lathes can machine the face of a part with the same commands used in a program for a mill.

This lathe feature allows face machining programming in the Cartesian coordinate system. The software converts Cartesian coordinate code to Polar coordinate machine moves.

Programmed moves should always position the tool center line. Do not use G41 or G42 Cutter Compensation.

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

Cartesian to Polar conversion is a modal command. G112 enables conversion G113 disables conversion. At this time Z axis moves are not allowed while this mode is enabled.



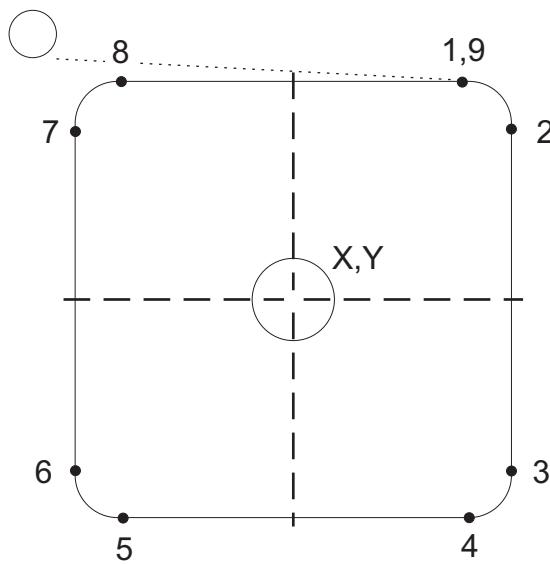
OPTIONS

CARTESIAN INTERPOLATION

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

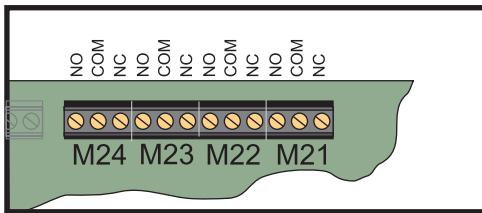
Example Program

```
%  
O00069  
N6 (SQUARE)  
G59  
( TOOL 11, .75 DIA. ENDMILL )  
(CUTTING ON CENTER)  
T1111  
M154  
G00 C0.  
/G97 M133 P1500  
G00 Z1.  
G00 G98 X2.35 Z0.1 (POSITION)  
G01 Z-0.05 F25.  
G112  
G17  
G0 X-.75 Y.5  
G01 X0.45 F10. (POINT 1)  
G02 X0.5 Y0.45 R0.05 (POINT 2)  
G01 Y-0.45 (POINT 3)  
G02 X0.45 Y-0.5 R0.05 (POINT 4)  
G01 X-0.45 (POINT 5)  
G02 X-0.5 Y-0.45 R0.05 (POINT 6)  
G01 Y0.45 (POINT 7)  
G02 X-0.45 Y0.5 R0.05 (POINT 8)  
G01 X0.45 (POINT 9) Y.6  
G113  
G00 Z3.  
M30  
%
```



11.21 8 "M" FUNCTIONS

This option adds 8 additional outputs for each 8M option. The machine can be fitted with two 8M options for a total of 16 additional outputs. These outputs can be used to activate probes, auxiliary pumps or clamping devices etc. The 8M relay board contains 8 relay outputs (M21- M28) and 2 terminal strips P4 and P5. Each terminal strip has 12 positions which are Normally Open, Normally Closed and Common.



8M Relay Board

A total of 4 banks of 8 relays are possible in the Haas system. Banks 0 and 1 are internal to the I/O PCB. Bank 1 includes the M21-25 relays at the top of the IOPCB. Bank 2 addresses the first 8M option PCB. Bank 3 addresses the second 8M option PCB.

NOTE: Bank 3 may be used for some Haas installed options and may not be available. Contact the Haas factory for more details.

Only one bank of outputs may be addressable with M-codes at a time. This is controlled by parameter 352 "Relay Bank Select". Relays in the non-activated banks are only accessible with macro variables. Parameter 352 is shipped set to "1" as standard. When either one or two 8M options are installed, the M-fin and probe cables are moved to the first 8M option PCB and parameter 352 is set to "2". With the 8M option, M-codes M121-128 correspond to relays labeled M21-28.

Bank addressing on the 8M PCB itself is done through selectable jumpers. Only one address should be selected at a time. The MCD jumper should be set to JP1 for bank 1 (first 8M option). The MCD jumper should be set to JP2 for bank 2 (second 8M option). The other positions are used by service only for installation in older controls. See the following figure.

M51-M58 will turn on the relays and M61-M68 will turn off the relays. M51 and M61 correspond to M21, etc. on the 8M relay board.

NOTE: Some or all of the M21-25 on the I/O PCB may be used for factory installed options. Inspect the relays for existing wires to determine which have been used. Contact the Haas factory for more details.

Terminals normally closed: 1, 4, 7, 10

Terminals normally open: 3, 6, 9, 12

COMMON TERMINALS: 2, 5, 8, 11

8M RELAY BOARD CONNECTORS:
P4 CONTAINS:

M21	M FUNCTION
M22	PROBE OPTION
M23	SPARE
M24	SPARE



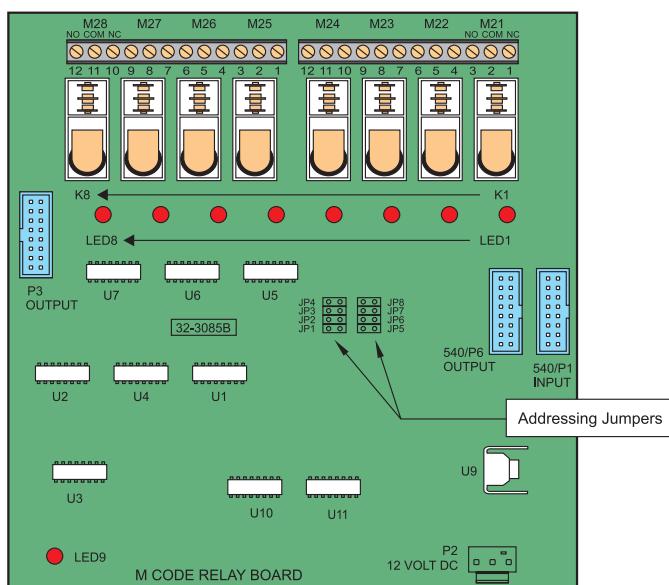
P5 CONTAINS:

M25 SPARE
M26 SPARE
M27 SPARE
M28 SPARE

P1 16-PIN RELAY DRIVERS FROM IO PCB (M21-M28) (540) (INPUT)
P3 UNUSED

P2 12 VDC FROM POWER SUPPLY BOARD (860A)

P6 16-PIN RELAY OUTPUT TO 2ND 8M RELAY BOARD



M-Code relay board.

NOTE: If the 8M option is installed, relays M21-28 become available on the secondary board. These relays will be controlled by outputs M21-28.

11.22 200 HOUR TRY-OUT FEATURE

Options that normally require a unlock code to activate (Quick Code, Macros, etc.) can now be activated and deactivated as desired simply by entering the letter **1** instead of the unlock code. 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.

**11.23 ETHERNET**

The Ethernet option is a great way to store and transfer data between your CNC and the server. In a typical shop environment, being able to easily transfer or download large files has always been a problem. Now with connection capabilities between the Haas CNC and a network, or Zip drive, large program files are easily downloaded/uploaded to/from memory or executed in DNC mode. For additional information, see Haas sales document ES-0115 (included with the machine).

11.24 ZIP DRIVE

As the Zip drive is portable, it increases the operator flexibility in how and where it can be used. Disks can be transferred from one drive to another (desktop to CNC) or the Zip drive can be moved from the PC to the mill or vice versa. For additional information, see Haas sales document ES-0168 (included with the machine).

11.25 HIGH INTENSITY LIGHTS

The High Intensity Lighting option for the Haas Lathes is a door activated system, providing bright, even illumination of the work area for part inspection. The dual top-mounted halogen bulbs also provide ample lighting for job setup.

The High Intensity Lighting system is operated automatically. When the doors are fully opened, a switch turns on the lights. In addition, the lighting system can be turned on manually using a switch located on the side of the light brackets.

11.26 MEMORY LOCK KEY SWITCH

The optional Memory Lock Key Switch will prevent the operator from editing programs and from altering settings when turned to the locked position.

11.27 SPINDLE ORIENTATION

The M19 code is used to orient the spindle to a fixed position. A P value is used to orient the spindle to a particular angle (in degrees). An R value will recognize up to four places to the right of the decimal point.

11.28 SECOND HOME

An additional button on the side of the control commands the machine to rapid all axes to the coordinates specified in Work Offset G129. This is helpful for tool setups.

**12. TECHNICAL REFERENCE****12.1 SPINDLE**

Spindle speed functions are controlled primarily by the **S** address code. The **S** address specifies RPM in integer values from 1 to maximum spindle speed (Parameter 131). NOT TO BE CHANGED BY USER!

Two **M** codes, M41 (Low Gear) and M42 (High Gear), can be used for gear selection. Spindle speed accuracy is best at the higher speeds and in low gear.

The spindle is hardened and ground with a A2-6, A2-8, A2-11 spindle nose.

12.2 Two-SPEED GEAR TRANSMISSION (SL-30 AND 40)

The spindle motor is directly coupled to the transmission, which is between the motor and the spindle casting. The transmission is V belt-coupled to the spindle pulley. An electric motor drives the gearbox shifter into high or low gear.

LUBRICATION

The gearbox is lubricated and cooled with Mobil DTE 25 oil.

OPERATION

High gear and low gear are selected by programming an M41 (Low Gear) or M42 (High Gear). **The spindle will not change gears automatically.** The spindle will come to a complete stop when changing gears.

The machine will remain in its current gear (until changed with an M41 or M42) even after the machine is powered off. When the machine is powered up, it will be in the same gear (or between gears) as when it was powered off.

The current gear status is monitored by discrete outputs SP HIG (Spindle High) and SP LOW (Spindle Low). A "0" (zero) in either of these outputs indicates it is the current gear. If the outputs are the same, neither gear is selected. If the gearbox remains in this condition (between gears) for a certain amount of time, Alarm 126, "Gear Fault", is generated. The only way to reset this alarm is to press the POWER UP/RESTART key. The current gear can also be monitored by pressing the CURNT COMDS key. This display will show whether the machine is currently in "HIGH GEAR", "LOW GEAR", or "NO GEAR".

There are a number of parameters related to the gearbox. Their values should not be changed by the operator.

**12.3 SERVOS (BRUSHLESS)****SERVO ENCODERS (BRUSHLESS)**

Haas machines are equipped with brushless motors, which provide for better performance, and no maintenance. In addition to the performance differences, these machines differ from brush type machines in the following areas:

- The brushless motors have 8192 line encoders built in, which result in a resolution of 32768 parts per revolution.
- "In Position" parameters 101, 102, 103, 104 and 165 also affect brushless motors.
- The motor controller board has a dedicated processor which does all the servo control algorithm.
- There is no servo distribution board anymore, therefore there is no CHARGE light present. Care should still be taken however, since there are high voltages present on the amplifiers, even when power is shut off. The high voltage comes from the vector drive, which does have a CHARGE light.
- The servo drive cards are replaced by Brushless Servo Amplifiers, and are controlled differently.
- A low voltage power supply card is added to the servo drive assembly to supply the low voltage requirement to the amplifiers.
- The user interface and motion profiling have not changed however, and the user should not see any functional differences between a brush type machine and a brushless machine.

**SERVO AMPLIFIERS (BRUSHLESS)**

The brushless servo amplifier is a PWM based current source. The PWM outputs control the current to a three phase brushless motor. The PWM frequency is either 12.5 KHz or 16 KHz. The amplifiers are current limited to 30 amps peak (45A peak for a medium amplifier). However there are fuse limits both in hardware and software to protect the amplifiers and motors from over current. The nominal voltage for these amplifiers is 320 volts. Therefore the peak power is about 9600 watts or 13 H.P. The amplifiers also have short circuit, over temperature and over voltage protection.

There is a 15 amp (20A for a medium amplifier) supply fuse for failure protection. This fuse is relatively slow, therefore it can handle the 30 amp peak. Continuous current limit to the motor is controlled by software.

The user should never attempt to replace these fuses.

Commands to the amplifier are +/-5 volts current in two legs of the motor and a digital enable signal. A signal from the amplifier indicates drive fault or sustained high current in a stalled motor.

The connectors on the amplifiers are:

+H.V.	+ 320 volts DC
-H.V.	320 volts return
A	motor lead phase A
B	motor lead phase B
C	motor lead phase C
J1	Three pin Molex connector used for +/-12 and GND.
J2	Eight pin Molex connector used for input signals.



12.4 INPUT/OUTPUT ASSEMBLY

The IOPCB contains a circuit for sensing a ground fault condition of the servo power supply. If more than 0.5 amps is detected flowing through the grounding connection of the 160V DC buss, a ground fault alarm is generated and the control will turn off servos and stop.

Relay K6 is for the coolant pump 230V AC. It is a plug-in type and is double-pole. Relays K9 and K10 are used for the Barfeeder (when equipped).

The Input/Output Assembly consists of a single printed circuit board called the IOPCB.

The connectors on the IOPCB are:

- P1 16-pin relay drivers from MOCON 1 to 8 (510)
- P2 16-pin relay drivers from MOCON 9 to 16 (520)
- P3 16-pin relay drivers from MOCON 17 to 24 (M21-M24) (540)
- P4 34-pin inputs to MOCON (550)
- P5 Servo power on relay 1-1 (110)
- P6 230V AC from CB3 (930)
- P7 230V AC to coolant pump (940)
- P8 Auto-off relay 1-7 (170)
- P9 Spindle drive commands (710)
- P10 Spindle fan and oil pump 115V AC (300)
- P12 115V AC to spindle head solenoids (880A)
- P13 Turret status inputs (820)
- P14 Low TSC (900)
- P15 Spindle head status inputs (890)
- P16 Emergency stop input (770)
- P17 Low Lube input (960)
- P18 Over Voltage Input (970)
- P19 Low Air Input (950)
- P20 Overheat input (830)
- P21 Spindle drive status inputs (780)
- P22 M-FIN input (100)
- P23 Footswitch (190)
- P24 Spare 2
- P25 Spare 3
- P26 Spare terminals for M21 to M24
- P27 Door lock (1040)
- P28 115V AC from CB4 (910)
- P29 A-axis brake solenoid output (390)
- P30 Tool changer shuttle motor output (810A)
- P31 230 VAC for Chip Conveyor (160)
- P33 115V AC three-phase input from power supply assembly (90)
- P34 115V AC to CRT (90A)
- P35 115V AC to heat exchanger (90B)



- P36 115V AC to CB4 (90C)
- P37 115V AC spare (870)
- P38 Door open (1050)
- P39 Tool changer turret motor output (810)
- P40 (770A) A/B
- P43 Ground fault sense signal input (1060) Axis Brake
- P44 5TH axis brake (319)
- P45 HTC Shuttle
- P46 Chip Conveyor (140)
- P47 Skip input signal (1070)
- P48 spare 1
- P49 spare 2
- P50 Spigot Motor (200)
- P51 16 PIN Relay drivers 17-24 (530)
- P52 spare 1
- P53 Spigot Sense (180)
- P54 Servo Brake (350)
- P55 Red/green lights (280)
- P56 Thru spindle coolant pump (940A)
- P57 115V spare
- P58 115V spare
- P59 Gear Box (370B)
- P60 TSC 230 IN 930A



12.5 CONTROL PENDANT

JOG HANDLE

The JOG handle is actually a 100-line-per-revolution encoder. We use 100 steps per revolution to move one of the servo axes. If no axis is selected for jogging, turning of the crank has no effect. When the axis being moved reaches its travel limits, the handle inputs will be ignored in the direction that would exceed the travel limits.

Parameter 57 can be used to reverse the direction of operation of the handle.

POWER ON/OFF SWITCHES

The POWER ON switch engages the main contactor. The on switch applies power to the contactor coil and the contactor thereafter maintains power to its coil. The POWER OFF switch interrupts power to the contactor coil and will always turn power off. POWER ON is a normally open switch and POWER OFF is normally closed. The maximum voltage on the POWER ON and POWER OFF switches is 24V AC and this voltage is present any time the main circuit breaker is on.

SPINDLE LOAD METER

The Load meter measures the load on the spindle motor as a percentage of the rated continuous power of the motor. There is a slight delay between a load and the actual reflection of the meter. The eighth A-to-D input also provides a measure of the spindle load for cutter wear detection. The second page of diagnostic data will display % of spindle load. The meter should agree with this display within 5%. The spindle drive display #7 should also agree with the load meter within 5%.

There are different types of spindle drive that are used in the control. They are all adjusted differently.

EMERGENCY STOP SWITCH

The EMERGENCY STOP switch is normally closed. If the switch opens or is broken, power to the servos will be removed instantly. This will also shut off the turret, spindle drive, and coolant pump. The EMERGENCY STOP switch will shut down motion even if the switch opens for as little 0.005 seconds.

Be careful of the fact that Parameter 57 contains a status switch that, if set, will cause the control to be powered down when EMERGENCY STOP is pressed.

You should not normally stop a tool change with EMERGENCY STOP as this will leave the tool changer in an abnormal position that takes special action to correct.

NOTE Tool changer alarms can be easily corrected by first correcting any mechanical problem, pressing RESET until the alarms are clear, selecting ZERO RETURN mode, and selecting AUTO ALL AXES.



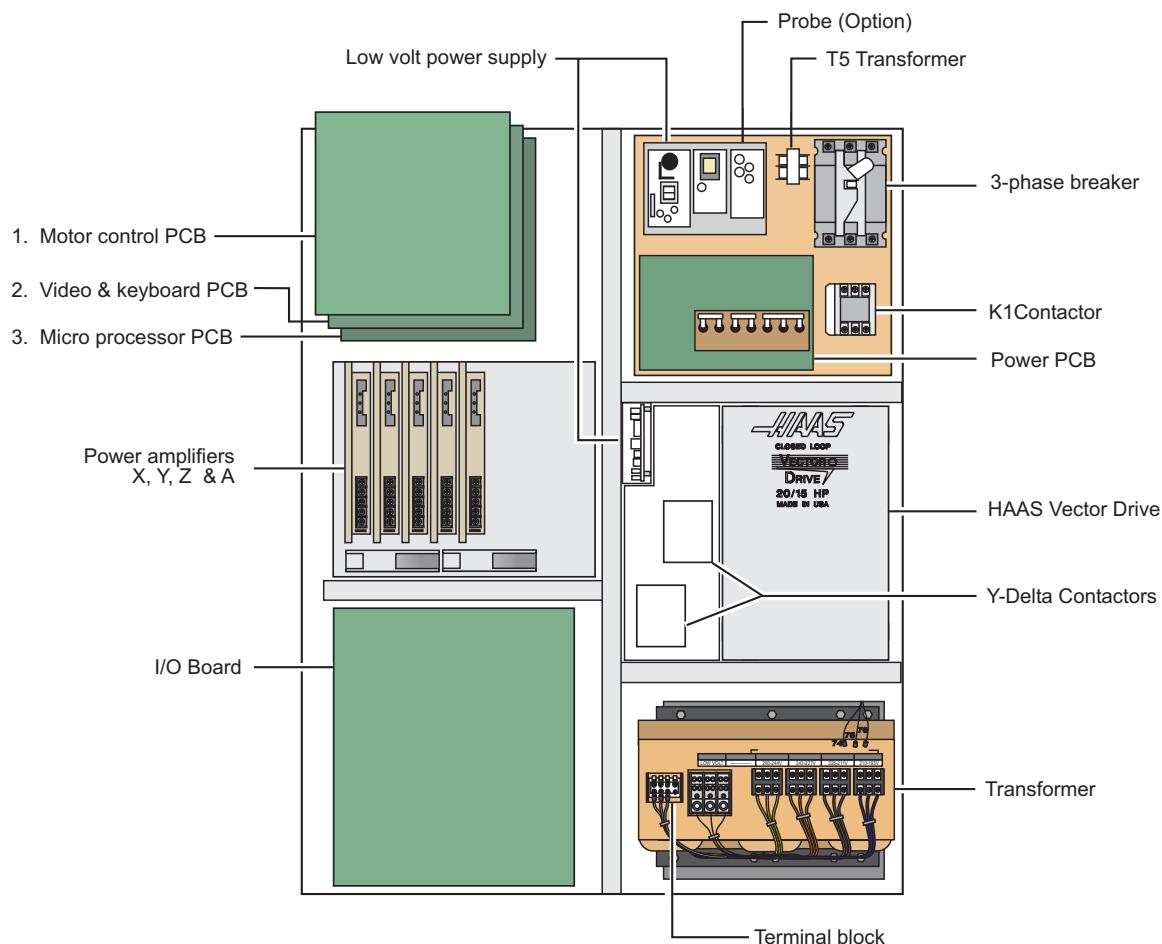
If the turret should become jammed, the control will automatically come to an alarm state. To correct this, push the EMERGENCY STOP button and remove the cause of the jam. Push the RESET key to clear any alarms. Push the ZERO RETURN and the AUTO ALL AXES keys to reset the Z-axis and turret. Never put your hands near the turret when powered unless the EMERGENCY STOP button is pressed.

KEYBOARD BEEPER

There is a beeper under the control panel that is used as an audible response to pressing keyboard buttons and as a warning beeper. The beeper is a one kHz signal that sounds for about 0.1 seconds when any keypad key, CYCLE START, or FEED HOLD is pressed. The beeper also sounds for longer periods when an auto-shutdown is about to occur and when the "BEEP AT M30" setting is selected.

If the beeper is not audible when buttons are pressed, the problem could be in the keypad, keyboard interface PCB or in the speaker. Check that the problem occurs with more than one button and check that the beeper volume is not turned down.

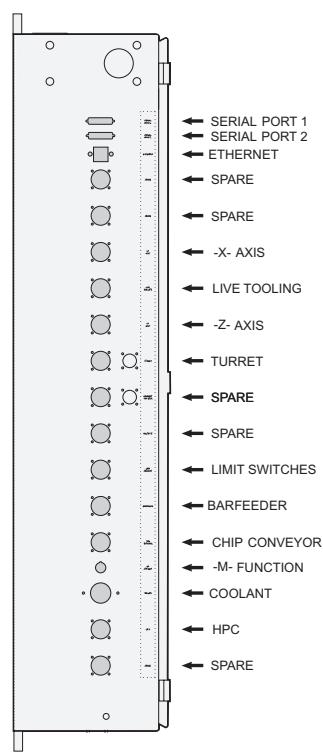
CONTROL CABINET



Control cabinet general overview.



The following illustration shows the connectors on the side of the control cabinet.



Side of control cabinet.

**12.6 MICROPROCESSOR ASSEMBLY**

The microprocessor assembly is in the rear cabinet at the top left position. It contains three large boards. They are: microprocessor, the video and the MOCON. All three boards of the processor assembly receive power from the low voltage power supply. The three PCB's are interconnected by a local buss on dual 50-pin connectors. At power-on of the control, some diagnostic tests are performed on the processor assembly and any problems found will generate alarms 157 or 158. In addition, while the control is operating, it continually tests itself and a self test failure will generate Alarm 152.

MICROPROCESSOR PCB (68ECO30)

The Microprocessor PCB contains the 68ECO30 processor running at 40 MHz, one 128K EPROM; between 1MB and 16MB of CMOS RAM and between 512K and 1.5MB of FAST STATIC RAM. It also contains a dual serial port, a five year battery to backup RAM, buffering to the system buss, and eight system status LED's.

Two ports on this board are used to set the point at which an NMI* is generated during power down and the point at which RESET* is generated during power down.

The eight LED's are used to diagnose internal processor problems. As the system completes power up testing, the lights are turned on sequentially to indicate the completion of a step. The lights and meanings are:

- | | |
|-------------|---|
| +5V | +5V logic power supply is present. (Normally On)
If this light does not come on, check the low voltage power supply and check that all three phases of 230V input power are present. |
| HALT | Processor halted in catastrophic fault. (Normally Off)
If this light comes on, there is a serious problem with the processor PCB. Check that the EPROM is plugged in. Test the card with the buss connectors off. |
| POR | Power-on-reset complete. (Normally On)
If this light does not come on, there is a serious problem with the processor PCB. Check that the EPROM is plugged in. Test the card with the buss connectors off. |
| SIO | Serial I/O initialization complete. (Normally On)
If this light does not come on, there is a problem with the serial ports. Disconnect anything on the external RS-232 and test again. |
| MSG | Power-on serial I/O message output complete. (Normally On)
If this light does not come on, there is a problem with serial I/O or interrupts. Disconnect anything on the external RS-232 and test again. |
| CRT | CRT/VIDEO initialization complete. (Normally On)
If this light does not come on, there is a problem communicating with the VIDEO PCB. Check the buss connectors and ensure the VIDEO PCB is getting power. |

**PGM****Program signature found in memory. (Normally On)**

If this light does not come on, it means that the main CNC program package was not found in memory or that the auto-start switch was not set. Check that switch S1-1 is on and the EPROM is plugged in.

RUN**Program running without fault exception. (Normally On)**

If this light does not come on or goes out after coming on, there is a problem with the microprocessor or the software running in it. Check all of the buss connectors to the other two PCB's and ensure all three cards are getting power.

There is a two-position DIP switch on the processor PCB labeled S1. Switch S1-1 must be ON to auto-start the CNC operational program. If S1-1 is OFF, the PGM light will remain off.

Switch S2-1 is used to enable FLASH. If it is disabled it will not be possible to write to FLASH.

The processor connectors are:

- | | |
|----|--|
| J1 | Address buss |
| J2 | Data buss |
| J4 | Serial port #1 (for upload/download/DNC) (850) |
| J5 | Serial port #2 (for auxiliary 5th axis) (850A) |
| J3 | Power connector |
| J6 | Battery |

MEMORY RETENTION BATTERY

The memory retention battery is soldered into the process board. This is a 3.3V Lithium battery that maintains the contents of CMOS RAM during power off periods. Prior to this battery being unusable, an alarm will be generated indicating low battery. If the battery is replaced within 30 days, no data will be lost. The battery is not needed when the machine is powered on. Connector J6 on the processor PCB can be used to connect an external battery.

VIDEO KEYBOARD FLOPPY DISK PCB

The VIDEO and KB PCB generates the video data signals for the monitor and the scanning signals for the keyboard. In addition, the keyboard beeper is generated on this board. There is a single jumper on this board used to select inverse video. The video PCB connectors are:

- | | |
|-----|------------------------------------|
| P1 | Low Voltage Power Supply PCB (860) |
| P3* | Keyboard info. (700) |
| P4 | Address Buss |
| P5 | Data Buss |
| P10 | Disk Dr. Power |
| P11 | Spare |
| P12 | Disk Dr. Signal |
| P13 | Video Signal (760) |
| J9 | RS422 B |
| J13 | Serial Data (850) |

**MOTOR CONTROLLER (MOCON) BRUSHLESS**

The brushless machining centers are equipped with a microprocessor based brushless motor controller board (MOCON) that replaces the motor interface in the brush type controls. It runs in parallel with the main processor, receiving servo commands and closing the servo loop around the servo motors.

In addition to controlling the servos and detecting servo faults, the motor controller board, (MOCON), is also in charge of processing discrete inputs, driving the I/O board relays, commanding the spindle and processing the jog handle input. Another significant feature is that it controls 6 axes, so there is no need for an additional board for a 5 axis machine.

**12.7 HAAS VECTOR DRIVE**

The Haas vector drive is a current amplifier controlled by the MOCON software, using the C axis output. The vector drive parameters are a part of the machine parameters and are accessible through the Haas front panel. The spindle encoder is used for the closed loop control and spindle orientation, as well as rigid tapping if the option is available. Spindle speed is very accurate since this is a closed loop control, and the torque output at low speeds is superior to non vector drive spindles.

Never work on the spindle drive until the small red CHARGE light goes out. Until this light goes out, there are dangerous voltages inside the drive, even when power is shut off.

12.8 RESISTOR ASSEMBLY

The Resistor Assembly is located on top of the control cabinet. It contains the servo and spindle drive regen load resistors.

SPINDLE DRIVE REGEN RESISTOR

A 5.6-ohm (8.6-ohm (6-ohm for SL-30 and 40) for older machines), 300-watt resistor bank is used by the vector drive to dissipate excess power caused by the regenerative effects of decelerating the spindle motor. If the spindle motor is accelerated and decelerated again in rapid succession repeatedly, this resistor will get hot. In addition, if the line voltage into the control is above 255V, this resistor will begin to heat. This resistor is overtemp protected at 100° C. At that temperature, an alarm is generated and the control will begin an automatic shutdown. If the resistor is removed from the circuit, an alarm may subsequently occur because of an overvoltage condition inside the spindle drive.

OVERHEAT SENSE SWITCH (OLDER MACHINES)

There is an overtemperature sense switch mounted near the above-mentioned regen resistors. This sensor is a normally-closed switch that opens at about 100° C. It will generate an alarm and all motion will stop. After the time period, specified by parameter 297, of an overheat condition, an automatic shutdown will occur in the control.

**12.9 POWER SUPPLY ASSEMBLY**

All power to the control passes through the power supply assembly. It is located on the upper right corner of the control cabinet.

MAIN CIRCUIT BREAKER CB1

Circuit breaker CB1 is rated at 40 amps (20 amps for High Voltage option, 80 amps for SL-30 and 40) and is used to protect the vector drive and to shut off all power to the control. The locking On/Off handle on the outside of the control cabinet will shut this breaker off when it is unlocked. A trip of this breaker indicates a SERIOUS overload problem and should not be reset without investigating the cause of the trip. The full circuit breaker rating corresponds to as much as 15 horsepower.

MAIN CONTACTOR K1

Main contactor K1 is used to turn the control on and off. The POWER ON switch applies power to the coil of K1 and after it is energized, an auxiliary switch on K1 continues to apply power to the coil. The POWER OFF switch on the front panel will always remove power from this contactor.

When the main contactor is off, the only power used by the control is supplied through two $\frac{1}{2}$ amp fuses to the circuit that activates the contactor. An overvoltage or lightning strike will blow these fuses and shut off the main contactor.

The power to operate the main contactor is supplied from a 24V AC control transformer that is primary fused at $\frac{1}{2}$ amp. This ensures that the only circuit powered when the machine is turned off is this transformer and only low voltage is present at the front panel on/off switches.

LOW VOLTAGE POWER SUPPLY

The low voltage power supply provides +5V DC, +12V DC, and -12V DC to all of the logic sections of the control. It operates from 115V AC nominal input power. It will continue to operate correctly over a 90VAC to 133V AC range.

**POWER PCB (POWER)**

The low voltage power distribution and high voltage fuses and circuit breakers are mounted on a circuit board called the POWER PCB.

POWER-UP LOW VOLTAGE CONTROL TRANSFORMER (T5)

The low voltage control transformer, T5, supplies power to the coil of the main contactor K1. It guarantees that the maximum voltage leaving the Power Supply assembly when power is off is 12V AC to earth ground. It is connected via P5 to the POWER PCB.

SECONDARY CIRCUIT BREAKERS

The following circuit breakers are located on the Power supply assembly.

CB2 controls the 115 volt power from the main transformer to the servo transformers and, if tripped, will turn off the servo motors and air solenoids. CB2 could be blown by a severe servo overload.

CB3 controls the power to coolant pump only. It can be blown by an overload of the coolant pump motor or a short in the wiring to the motor.

CB4 controls the 115V AC to the air solenoids and the oiler. It is never expected to trip. If it does trip, it is likely caused by a short circuit in the wiring on the I/O assembly or the wiring to the solenoids on the spindle head.

OPERATOR'S LAMP

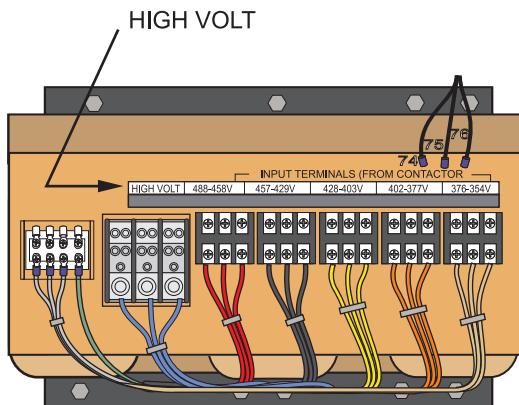
The operator's lamp is using 115 VAC taken from P19 on the main power distribution.

**12.10 POWER TRANSFORMER ASSEMBLY (T1)**

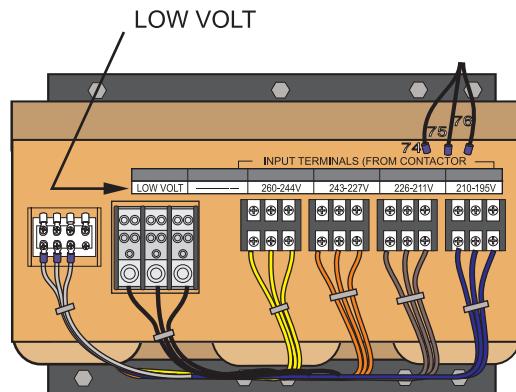
The power transformer assembly is used to convert three-phase input power (50/60Hz) to three phase 230V and 115V power. Two different transformers are used depending on the input voltage range. The low voltage transformer has four different input connections to allow for a range of voltages from 195 V RMS to 260 V RMS. The high voltage transformer has five different input connections and will accept a range of voltages from 354V RMS to 488 V RMS.

The 230 V is used to power the spindle drive, which also develops the 325 VDC power for the axis servo amplifiers. The 115 V is used by the video monitor, solenoids, fans and pumps, in addition to supplying power to the main LVPS used by the control electronics.

The transformer assembly is located in the lower right hand corner of the main cabinet. Besides the high/low voltage variations, two different power levels are available depending on the spindle motor used. The small and large transformers have power ratings of 14 KVA and 28 KVA, respectively. They are protected by the main circuit breaker to the levels shown in the preceding table.



Transformer with 354-488V range



Transformer with 195-260V range

PRIMARY CONNECTION TO T1

Input power to T1 is supplied through CB1, the 40 amp or 80 amp three-phase main circuit breaker. Three-phase 230 to T1 is connected to the first three terminals of TB10.

VOLTAGE SELECTION TAPS

There are four labeled plastic terminal blocks for . Each block has three connections for wires labeled 74, 75, and 76. Follow the instructions printed on the transformer.

SECONDARY CONNECTION TO T1

The secondary output from T1 is 115V AC three-phase CB2 protects the secondary of transformer T1 and is rated at 25 amps.


OPTIONAL 480V 60Hz TRANSFORMER

The external transformers have either 30 or 45 KVA ratings depending on the size of the machine to which they will be attached. SL-20 5K, SL-20 BB, SL-30 and SL-40 machines will get the 45KVA transformer while the smaller machines will get the 30KVA transformers.

For domestic installations and all others using 60Hz power, the primary side should be wired as follows:

Input Voltage Range	Tap
493-510	1 (504)
481-492	2 (492)
469-480	3 (480)
457-468	4 (468)
445-456	5 (456)
433-444	6 (444)
420-432	7 (432)

OPTIONAL 480V 50Hz TRANSFORMER

Input Voltage Range	Tap
423-440	1 (504)
412-422	2 (492)
401-411	3 (480)
391-400	4 (468)
381-390	5 (456)
371-380	6 (444)
355-370	7 (432)

**12.11 FUSES**

The brushless amplifier has one fuse, F1 15 amps. This fuse protects the amplifier itself from drastic damage. If this fuse is ever blown, the associated motor will stop. This will only happen if there is a failure of the amplifier card. **The user should never attempt to replace these fuses.**

The POWER PCB contains three ½-amp fuses located at the top right (FU1, FU2, FU3). If the machine is subject to a severe overvoltage or a lightning strike, these fuses will blow and turn off all of the power. Replace these fuses only with the same type and ratings. FU 4,5 and 5A protect the chip conveyor (FU6 is only used with 3 phase motors). FU7-12 are ultra fast 20A fuses. They will only blow in the case of a cable short for either the TSC or coolant pump. Spare fuses for the power card are located above the breakers on the spare fuse PCB.

SIZE	FUSE NAME	TYPE	RATING (amps)	VOLTAGE	LOCATION
5mm	FU1	Slo-Blo	½	250V	PSUP pcb, upper right
5mm	FU2	AGC	½	250V	" "
5mm	FU3	AGC	½	250V	" "
1/4	FU1	Ultra fast	10	250V	I/O PCB
1/4	F1	Ultra fast	15	250V	Amplifier (X,Y,Z,A,B)
5mm	FU4,5	Fast blow	5A	250V	PSUP, bottom right corner
1/4	FU7-12	Ultra fast	20A	250V	PSUP, bottom

FU2 on the IOPCB is a spare.

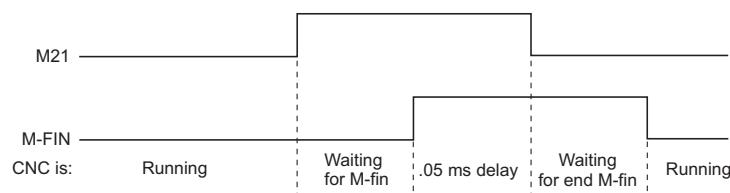


12.12 SPARE USER M CODE INTERFACE

The M code interface uses outputs M21-25 and one discrete input circuit. M codes M21 through M25 will activate relays labeled M21-25. These relay contacts are isolated from all other circuits and may switch up to 120V AC at three amps. The relays are SPDT. **WARNING!** Power circuits and inductive loads must have snubber protection.

The M-FIN circuit is a normally open circuit that is made active by bringing it to ground. The one M-FIN applies to all of the user M codes.

The timing of a user M function must begin with all circuits inactive, that is, all circuits open. The timing is as follows:



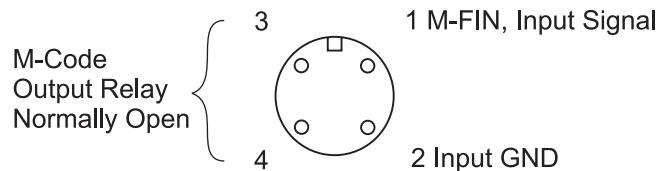
The Diagnostic Data display page may be used to observe the state of these signals.

NOTE: See the 8M option section for more details.

M FUNCTION RELAYS

The M code relay board has five relays (M21-25) that may be available to the user. M21 is already wired out to P12 at the side of the control cabinet. This is a four-pin DIN connector and includes the M-FIN signal.

NOTE: Refer to the Diagnostic section in the manual for specific machine Inputs and Outputs.



NOTE: Some or all of the M21-25 on the I/O PCB may be used for factory installed options. Inspect the relays for existing wires to determine which have been used. Contact the Haas factory for more details.

**M-FIN DISCRETE INPUT**

The M-FIN discrete input is a low voltage circuit. When the circuit is open, there is +12V DC at this signal. When this line is brought to ground, there will be about 10 millamps of current. M-FIN is discrete input #10 and is wired from input #10 on the I/O PCB. The return line for grounding the circuit should also be picked up from that PCB. For reliability, these two wires should be routed in a shielded cable where the shield is grounded at one end only. The diagnostic display will show this signal a "1" when the circuit is open and a "0" when this circuit is grounded.

TURNING M FUNCTIONS ON AND OFF

The M code relays can also be separately turned on and off using M codes M51-M55 and M61-M65. M51 to M55 will turn on one of the eight relays and M61 to M65 will turn the relays off. M51 and M61 correspond to M21, etc.

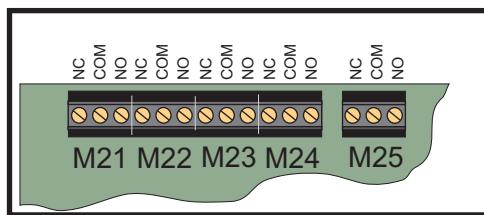
NOTE: Refer to the Diagnostic section in the manual for specific machine Inputs and Outputs.

WIRING THE RELAYS

The relays are marked on the IOPCB, with their respective terminals forward of them. If the optional 8M relay board is installed then the connections on the IOPCB are to be left unused as they are replaced by the relays on the optional board. Refer to the figure, and the Probe Option figure in the Electrical Diagrams section for the terminal labeling.

WARNING!

Power circuits and inductive loads must have snubber protection.



IOPCB Relays

CAUTION! If a screw terminal is already in use DO NOT connect anything else to it. Call your dealer.

**12.13 LUBRICATION PUMP**

The lubrication system is a resistance type system which forces oil through metering units at each of the 16 lubricating points within the machine. The system uses one metering unit at each of the lubricating points: one for each linear guide pad, one for each lead screw and one for spindle lubrication. A single oil pump is used to lubricate the system. The pump is powered only when the spindle and/or an axis moves. Once powered the pump cycles approximately 3.0 cc of oil every 30 minutes throughout the oil lines to the lube points. Every lube point receives approximately 1/16 of oil. The control monitors this system through an internal level switch in the reservoir and external pressure switch on the lube panel.

LOW LUBRICATION AND LOW PRESSURE SENSE SWITCHES

There is a low lube sense switch in the oil tank. When the oil is low, an alarm will be generated. This alarm will not occur until the end of a program is reached. There is also a lube pressure switch that senses the lube pressure. Parameter 117 controls the lube pressure check. If Parameter 117 is not zero, the lube pressure is checked for cycling high within that period. Parameter 117 has units of, 1/50 seconds; so 30 minutes gives a value of 90000. Parameter 57, bit "Oiler on/off", indicates the lube pump is only powered when the spindle fan is powered. The lube pressure is only checked when the pump is on.

**12.14 SWITCHES****LAMP ON/OFF SWITCH**

An on/off switch is supplied for the operator's lamp. It is located on the front panel.

DOOR OPEN SENSE SWITCH

The DOOR OPEN switch is in the open position when the door is open and closed when the door is fully closed.

When the doors open, the switch will open and the machine will stop with a "Door Hold" function. When the door is closed again, operation will continue normally.

If the doors are open, you will not be able to start a program. Door Hold will not stop a tool change operation or a tapping operation, and will not turn off the coolant pump. Also, if the doors are open, the spindle speed will be limited to 500 RPM.

The Door Hold function can be temporarily disabled by turning Setting 51 **on**, if Parameter 57 bits DOOR STOP SP and SAFETY CIRC are set to zero, but this setting will return to OFF when the control is turned off.

LIMIT SWITCHES**TURRET CLAMP/UNCLAMP SWITCHES**

There are two switches used to sense the position of the turret. They are both normally closed and one will activate at the end of travel during unclamping and the other during clamping. When both switches are closed, it indicates that the turret is between positions.

The diagnostic display can be used to display the status of the relay outputs and the switch inputs.

DOOR HOLD SWITCH

The switch is normally closed. When the door opens, the switch will open and the machine will stop with a "Door Hold" function. When the door is closed again, operation will continue normally.

If the door is open, you will not be able to start a program. Door hold will not stop a tool change operation, will not turn off the spindle, and will not turn off the coolant pump.

The door hold function can be temporarily disabled with Setting 51, but this setting will return to OFF when the control is turned off.

X AND Z LIMIT SWITCHES

Prior to performing a POWER UP/RESTART or an AUTO ALL AXES operation, there are no travel limits. Thus, you can jog into the hard stops in either direction for X and Z. After a ZERO RETURN has been performed, the travel limits will operate unless an axis hits the limit switch. When the limit switch is hit, the zero returned condition is reset and an AUTO ALL AXES must be done again. This is to ensure that if you hit the limit switch, you can still move the servo back away from it.



The limit switches are normally closed. When a search for zero operation is being performed, the X and Z axes will move towards the limit switch unless it is already active (open); then they will move away from the switch until it closes again; then they will continue to move until the encoder Z channel is found. This position is machine zero.

TURRET HOME SWITCH

The tool rotation turret has a switch that is activated when tool #1 is in the cutting position. At POWER ON this switch indicates that tool #1 is in the cutting position. If this switch is not active at power-on, the first tool change will rotate the turret until the switch engages and then move to the selected tool. The diagnostic display will show this status of this input switch as "TOOL #1". A "1" indicates that tool #1 is in position.

What Can Go Wrong With Limit Switches?

If the machine is operated without connector P5, a LOW LUBE and DOOR OPEN alarm will be generated. In addition, the Home search will not stop at the limit switch and will instead run into the physical stops on each axis.

If the switch is damaged and permanently open, the zero search for that axis will move in the negative direction at about 0.5 in/min until it reaches the physical travel stops at the opposite end of travel.

If the switch is damaged and permanently closed, the zero search for that axis will move at about 10 in/min in the positive direction until it reaches the physical stops.

If the switch opens or a wire breaks after the zero search completes, an alarm is generated, the servos are turned off, and all motion stops. The control will operate as though the zero search was never performed. The RESET can be used to turn servos on but you can jog that axis only slowly.

**12.15 DIAGNOSTIC DATA**

The ALARM MSGS display is the most important source of diagnostic data. At any time after the machine completes its power-up sequence, it will either perform a requested function or stop with an alarm. Refer to the alarms list for their possible causes, and some corrective action.

If there is an electronics problem, the controller may not complete the power-up sequence and the CRT will remain blank. In this case, there are two sources of diagnostic data; these are the audible beeper and the LED's on the processor PCB. If the audible beeper is alternating a ½ second beep, there is a problem with the main control program stored in EPROM's on the processor PCB. If any of the processor electronics cannot be accessed correctly, the LED's on the processor PCB will or will not be lit.

If the machine powers up but has a fault in one of its power supplies, it may not be possible to flag an alarm condition. If this happens, all motors will be kept off and the top left corner of the CRT will have the message:

POWER FAILURE ALARM

and all other functions of the control will be locked out.

When the machine is operating normally, a second push of the PARAM/DGNOS key will select the diagnostics display page. The PAGE UP and PAGE DOWN keys are then used to select one of two different displays. These are for diagnostic purposes only and the user will not normally need them. The diagnostic data consists of 32 discrete input signals, 32 discrete output relays and several internal control signals. Each can have the value of 0 or 1. In addition, there are up to three analog data displays and an optional spindle RPM display. Their number and functions are:

DISCRETE INPUTS / OUTPUTS**DISCRETE INPUTS**

#	Name	#	Name
1000	Tool Turret Unlock	1016	Spare
1001	Tool Turret Lock	1017	Spare
1002	Spare	1018	Spare
1003	Low Coolant	1019	Spare
1004	Automatic Door	1020	Low hyd pressure
1005	Spindle In Hi Gear	1021	T.S. Foot Switch
1006	Spindle In Low Gear	1022	Probe Not Home
1007	Emergency Stop	1023	Spare 2b
1008	Door Switch	1024	Tool Unclamp Rmt*
1009	M Code Finish	1025	Low Phasing 115V
1010	Over Voltage	1026	B F End of Bar
1011	Low Air Pressure	1027	Bar Feeder Fault
1012	Low Lube Press.	1028	Ground Fault
1013	Regen Overheat	1029	G31 Block Skip
1014	Spare	1030	B F Spindle Intlk
1015	Spare	1031	Conveyr Overcrnts



#	Name	#	Name
1100	Hyd Pump Enable	1116	Move Spigot CW
1101	Spare	1117	Move Spigot CCW
1102	Spare	1118	Pal Ready Light
1103	Spare	1119	T.S. High Pressure
1104	Spindle Brake	1120	Tool Turret Out
1105	Coolant Pump on	1121	T.S. Reverse
1106	Power Off	1122	T.S. Forward
1107	Way Lube Pump	1123	(CE) Door Locked
1108	SB Motor Load PR	1124	M21 (Auto Door Clutch)
1109	SB Motor Load Bar	1125	M22 (Parts Catcher)
1110	Auto Door Open	1126	M23 (C Axis Engage)
1111	Auto Door Close	1127	HPC Coolant
1112	Spindle Hi Gear	1128	Green Beacon On
1113	Spindle Low Gear	1129	Red Beacon On
1114	Unclamp Chuck	1130	Enable Conveyor
1115	Lock Spindle	1131	Reverse Conveyor

The names of discrete outputs **1124**, **1125** and **1126** will change if options are installed. The options and associated Discrete Outputs are:

1124 Auto Door Clutch

1125 Parts Catcher

1126 C axis Engage

If the machine does not have these options the discrete outputs will remain M21, M22 and M23.

The 32 inputs are numbered the same as the 32 connections on the inputs printed circuit board. The last eight outputs are reserved for expansion by HAAS.

The second page of diagnostic data is displayed using the PAGE UP and PAGE DOWN keys. It contains:

INPUTS 2

Name	Name
X-axis Z Channel	X Motor Over Heat
Y-Axis Z Channel	Y Motor Over Heat
Z-axis Z Channel	Z Motor Over Heat
A-axis Z Channel	A Motor Over Heat
B-axis Z Channel	B Motor Over Heat
C-axis Z Channel	C Motor Over Heat
X Home Switch	X drive fault
Y Home Switch	Y drive fault
Z Home Switch	Z drive fault
A Home Switch	A drive fault
B Home Switch	B drive fault
C Home Switch	C drive fault



X Cable Input
Y Cable Input
Z Cable Input
A Cable Input
B Cable Input
C Cable Input

S Z CH Spindle Z Channel

When equipped with the Temp-Track option, the X and Z ball screw temperatures are now displayed on the INPUTS2 diagnostics screen just above SP LOAD when parameter 266 or 268 (respectively) bit 9 TEMP SENSOR is set to 1.

The following inputs and outputs pertain to the Haas Vector Drive. If it is not enabled, these will display a value of *. Otherwise, it will display a 1 or 0.

HAAS VECTOR DRIVE

Name	Name
Spindle Forward	Spindle Fault
Spindle Reverse	Spindle Locked
Spindle Lock	Spindle Cable Fault
Spindle At Speed	Spindle Overheat
Spindle Stopped	

ANALOG DATA

Name	Description
SP LOAD	Spindle load in %
SP SPEED	Spindle RPM CW or CCW
RUN TIME	Total machine run time
TOOL CHANGES	Number of tool changes
VERX.XXX	Software version number
YY/MM/DD	Today's date
MDL SL__	Model number
DC BUSS	Mocon II



12.16 LIVE TOOLING

Live Tooling provides the ability to utilize standard 40mm VDI-driven tools, operated by a 5-HP motor. This auxiliary motor is capable of 0-3,000 RPM, controllable in 1 RPM increments.

BRAKE

13.25" (348mm) diameter disc, 500 psi (34 bar), with 1,000 lbs. (4450 N) clamp force.

A solenoid actuates a hydraulically operated brake. The brake is located on the main spindle and can be CLAMPED with an M14 command and UNCLAMPED with an M15 command.

A clamped brake will unclamp at any spindle speed command or while the spindle is at rest.

12.17 FORMULAS

TO FIND:

S.F.M.

TO FIND THE SFM OF A CUTTER OR WORKPIECE

EXAMPLE: To find the SFM of a cutter rotating at 600 RPM with a diameter of 10 inches.

$$\text{SFM} = \frac{3.1416 \times d \times \text{RPM}}{12} = .262 \times d \times \text{RPM}$$

R.P.M.

TO FIND THE RPM OF A CUTTER OR WORKPIECE

EXAMPLE: To find the RPM of a cutter rotating at 150 SFM with a diameter of 8 inches.

$$\text{SFM} = \frac{12 \times \text{SFM}}{3.1416 \times d} = \frac{3.82 \times \text{SFM}}{d}$$

I.P.M.

TO FIND THE FEED (table travel in inches per minute)

EXAMPLE: To find the feed of a 10 tooth cutter rotating at 200 RPM with a feed per tooth of 0.012".

$$\text{IPM} = \text{F.P.T.} \times T \times \text{RPM}$$

TO FIND:

F.P.R.

TO FIND THE FEED PER REVOLUTION (in inches) OF A CUTTER.

EXAMPLE: To find the feed per revolution of a cutter rotating at 200 RPM with a table travel of 22 inches per minute.

$$\text{F.P.R.} = \frac{\text{I.P.M.}}{\text{R.P.M.}}$$

F.P.T.

TO FIND THE FEED PER TOOTH OF A CUTTER.

EXAMPLE: To find the feed per tooth of a cutter rotating at 200 RPM with a table travel of 22 inches per minute.

$$\text{F.P.T.} = \frac{\text{I.P.M.}}{T \times \text{R.P.M.}}$$

D = Depth of cut

d = diameter of cutter

I.P.M. = Feed (table travel in inches per minute)

K = Constant (cubic inches per minute per HPc). Power required to remove 1 cubic inch per minute.

HPc = Horsepower at the cutter

F.P.R. = Feed per revolution

R.P.M. = Revolutions per minute

T = Number of teeth in cutter

W = Width of cut (in inches)



TECHNICAL REFERENCE

SL OPERATOR'S MANUAL
Series

June 2001