1. MAINTENANCE

1.1 GENERAL REQUIREMENTS

Operating Temperature Range   41°F to 104°F (5 to 40°C)
Storage Temperature Range   -4°F to 158°F (-20 to 70°C)
Ambient Humidity: 20% – 95% relative humidity, non-condensing
Altitude: 0-7000 ft.

ELECTRICITY REQUIREMENTS

IMPORTANT! REFER TO LOCAL CODE REQUIREMENTS BEFORE WIRING MACHINES.

ALL MACHINES REQUIRE:
Three phase 50 or 60Hz power supply.
Line voltage that does not fluctuate more than +/-10%

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

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

30-40 HP System	Voltage Requirements	High Voltage Requirements[2]
TL-15BB, SL-20BB, SL-30, SL-30BB,		
[1]SL-40, SL-40BB	(195-260V)	(354-488V)
Power Supply	100 AMP	50 AMP
Haas Circuit Breaker	80 AMP	40 AMP
If service run from elec. panel is less than 100' use:	4 GA. WIRE	8 GA. WIRE
If service run from elec. panel is more than 100' use:	2 GA. WIRE	6 GA. WIRE

55HP System	Voltage Requirements	High Voltage Requirements
[1]SL-40, SL-40BB, SL-40L	(195-260V)	(354-488V)
Power Supply	150 AMP	**Must use an external transformer**
Haas Circuit Breaker	125 AMP	
If service run from elec. panel is less than 100' use:	1 GA. WIRE	
If service run from elec. panel is more than 100' use:	0 GA. WIRE	

WARNING!

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

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

The rated horsepower of the machine may not be achieved if the imbalance of the incoming voltage is beyond an acceptable limit. The machine may function properly, yet may not deliver the advertised power. This is noticed more often when using phase converters. A phase converter should only be used if all other methods cannot be used.

The maximum leg-to-leg or leg-to-ground voltage should not exceed 260 volts, or 504 volts for high-voltage machines with the Internal High Voltage Option.

[1] The current requirements shown in the table reflect the circuit breaker size internal to the machine. This breaker has an extremely slow trip time. It may be necessary to size the external service breaker up by 20-25%, as indicated by "power supply", for proper operation.

[2] The high-voltage requirements shown reflect the Internal 400V configuration which is standard on European machines. Domestic and all other users must use the External 480V option.

AIR REQUIREMENTS

The CNC Lathe requires a minimum of 100 PSI at 4 scfm at the input to the pressure regulator on the back of the machine. This should be supplied by at least a two horsepower compressor, with a minimum 20-gallon tank, that turns on when the pressure drops to 100 PSI.

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".

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.

1.2 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, housing, rotating union and extension tube. Make sure the drawtube cover plate is installed either on the rotating union or on the chuck opening. ✓ Check hydraulic unit oil level (DTE-25 ONLY). Capacity-8 gallons.
Weekly	✓ 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 Gearbox oil level (if applicable). If oil is not visible at the bottom edge of the sight gauge, remove the end panel and add DTE-25 through the top filler hole until it is visible in the sight gauge.
Six Months	✓ Replace coolant and thoroughly clean the coolant tank. ✓ Replace hydraulic unit oil filter. ✓ Check all hoses and lubrication lines for cracking.
Annually	✓ Replace gearbox oil. ✓ Clean oil filter 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

Do not use a wash-down hose on the Haas lathe; doing so may cause damage to the spindle.
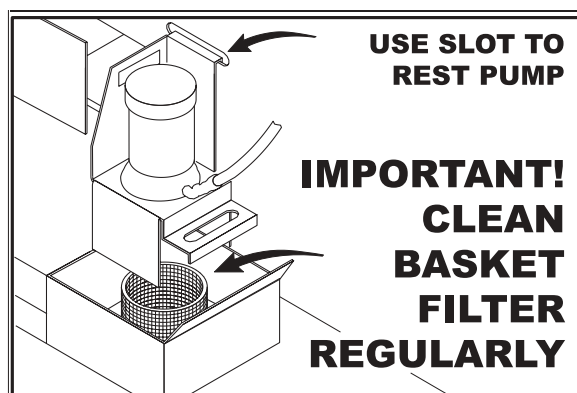
Poor Coolant flow can be caused by a dirty filter. To clean the filter, turn off the coolant pump, lift the coolant tank lid and remove the filter. Clean and reinstall filter.

1.3 LUBRICATION CHART

Item	Capacity	Fluid Type
Coolant	15 Gallons SL-10 35 Gallons for SL-20 50 for Gallons SL-30 77 Gallons for SL-40	Water soluble or synthetic
Way Lube	2-2.5 Qt. depending on pump style	Vactra #2
Transmission	54 oz.	Mobil DTE 25

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

Do not use pure water as a coolant; machine components will rust.

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 mainte-nance 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)

1.4 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

1.5 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.
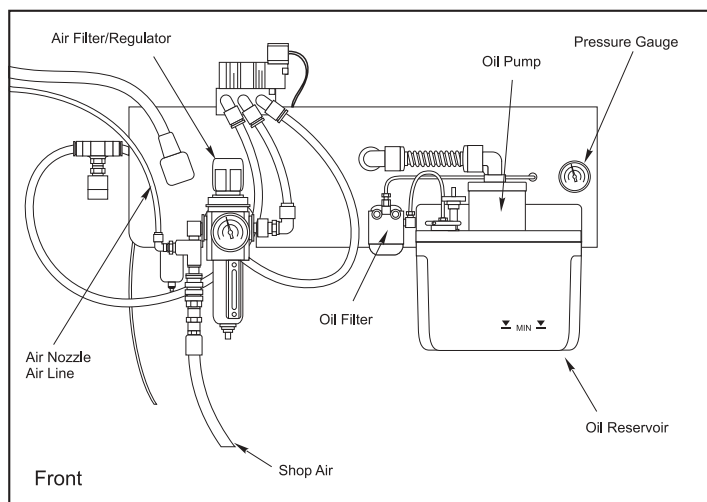
External Lubrication System

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.


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

Oil Check

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

Oil Change

1. Remove the sheet metal necessary to gain access to the transmission.

2. Remove the fourteen (14) SHCS from the oil pan and remove it. Inspect the magnetic drainplug for signs of metal particles.

3. Wipe down the oil pan and reinstall it with a new gasket.

4. Blow downward with an air hose in the vicinity of the access plate to prevent dirt and metal particles from entering the gear case. Remove the access plate.

5. Fill the gear case with 2¼ liters of Mobil DTE-25 gear oil. Check the sight glass. The level should be 3/4 of the way up when full. Fill as needed.

6. Install access plate with new gasket.

7. Run a spindle warm-up and check for leaks.

Oil Level Sight Glass

View Rotated 180°

Oil Fill Cup

Oil Drain Plug

Bottom View
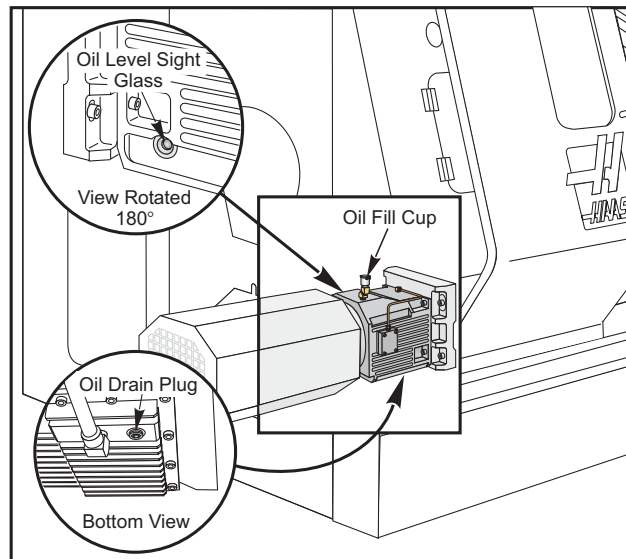
1.7 CHIP AUGER

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.

1.8 PERIODIC MAINTENANCE

A periodic maintenance page has been added to the Current Commands screens (titled SCHEDULED MAINTE-NANCE and accessed by pressing PAGE UP or PAGE DOWN) which 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. The maintenance item can have its time adjusted by using the left and right arrows. One hour is added or subtracted for each keypress, up to a maximum of 10,000 hours, and a minimum of 1 hour. Pressing the Origin key will reinstate the default time. 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

1.9 WINDOWS / GUARDING

Polycarbonate windows and guarding can be weakened by exposure to cutting liquids and chemicals that contain amines. It is possible to loose up to 10% of the remaining strength annually. If degradation is suspected, window replacement should occur at no more than a two year interval.

Windows and guarding should be replaced if damaged or severely scratched - Replace damaged windows immediately

1.10 INTERIOR WORKLIGHT

1. TURN OFF power to the machine at the main breaker.

2. Remove the light lens by bowing it over the lower light tabs and pulling it down.

3. Remove the light bulb and replace.

4. Replace the light lens and restore power to the machine.
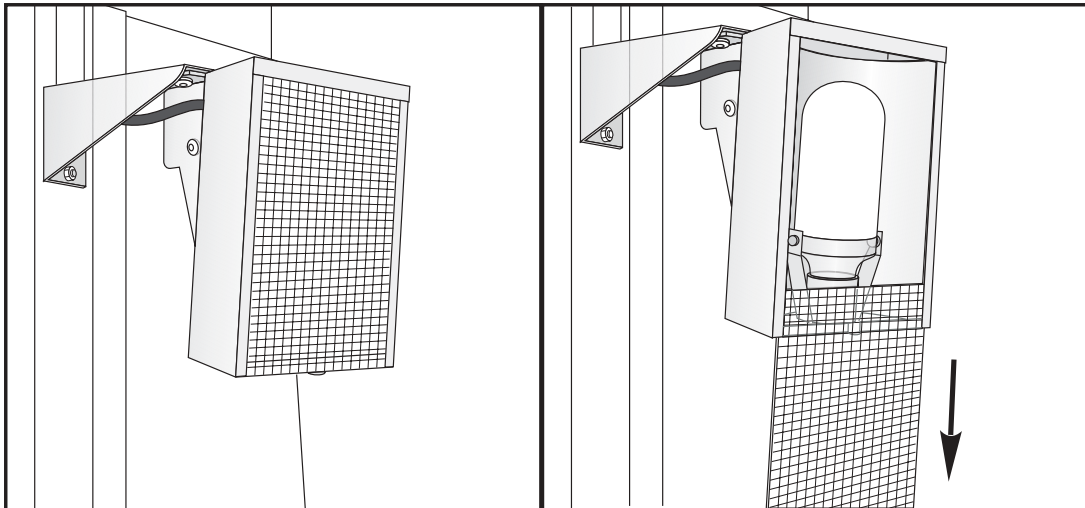
Interior worklight assembly.

2. OPERATION

The Information contained in this manual is constantly being updated. The latest updates and other helpful information can be downloaded in .pdf format from the Haas website. Go to www.HaasCNC.com and click on "Manual Updates" under the "Customer Services" drop-down menu on the navigation bar.

Additional technical information is available in the Haas Reference Manual. The Reference Manual contains definitions of parameters and alarms, machine installation instructions, and explanations of the machine's mechanical and electrical subassemblies.

The Haas Reference Manual is available online as a free download in .pdf format (go to www.HaasCNC.com and click on "Manual Updates" under the "customer services" drop-down menu in the navigation bar), or may be purchased through the Haas Service Department.

2.1 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.

2.2 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.
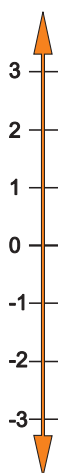
-3 -2 -1 0 1 2 3
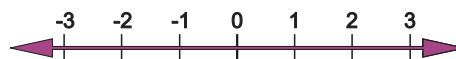
*Horizontal number line.*

Vertical number line

3
2
1
0
-1
-2
-3

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.
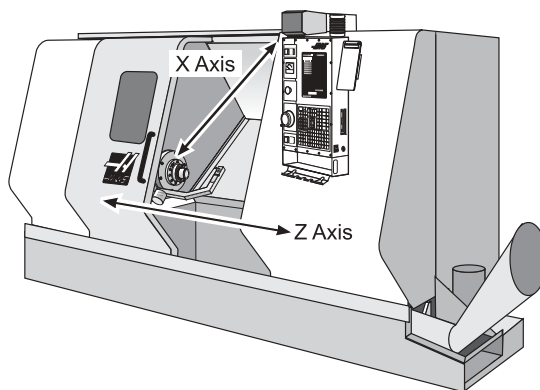
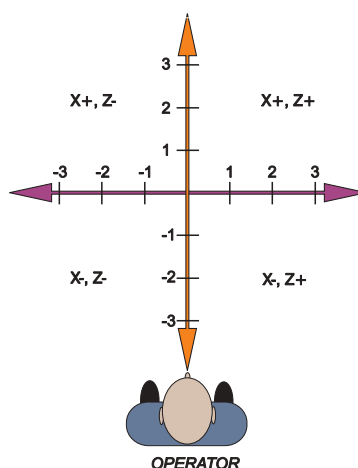X Axis

Z Axis

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.

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.

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 a 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.

2.3 MACHINE HOME

The principal of machine home may be seen when doing a manual reference return of all machine axes. When a zero return (ZERO RET) is performed at machine start up, all axes are moved to the furthest positive direction until 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.

2.4 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.
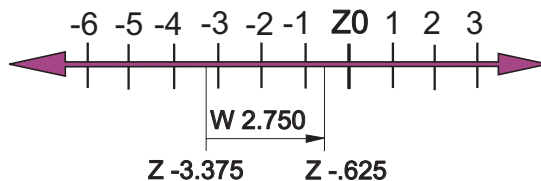
An example of an incremental move.

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.

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, select the rpmand the direction.
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 G-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.

2.6 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.

2.7 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 Txxyy 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:

G12345;   (Program name here will be displayed under List Prog)
N1 G28
N2 T101 (O.D.Tool x .031 TNR)
N3 G50 S2400
N4 G97 S800 M03
N5 G54 G00 X2. Z0.1 M08
N6 G96 S425

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

2.8 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.

2.9 OPERATORS CONTROL PANEL
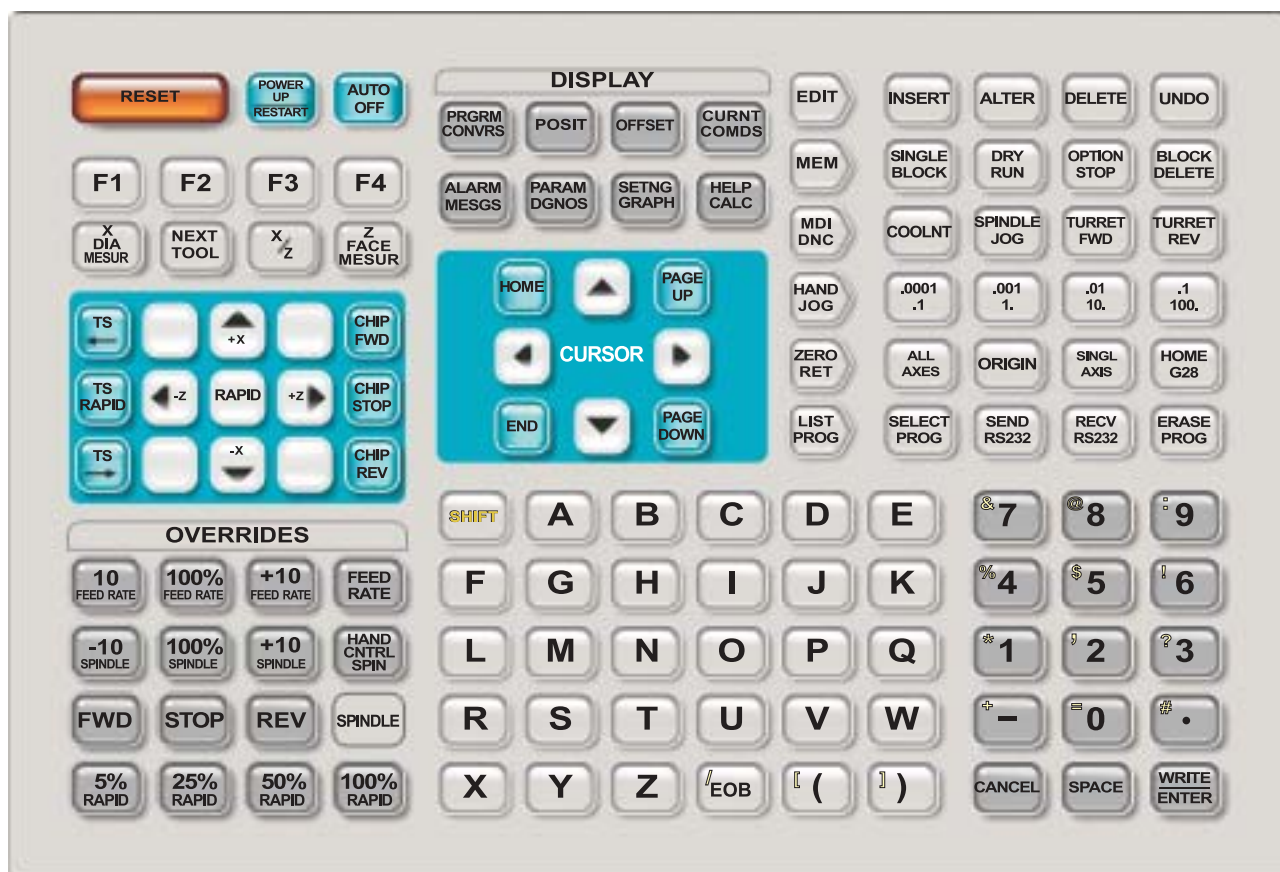
Control panel keypad with operating and display keys highlighted.

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

Key switch locks Settings and all programs.
Setting 7 locks parameters; parameters 57, 209 and 278 lock other features.
Setting 8 locks all programs.
Setting 23 locks 9xxx programs.
Setting 119 locks offsets.
Setting 120 locks macro variables.

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.

Function 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.

2.10 REAL TIME CLOCK

Displays current Date and Time. The date and time as supplied by the real- time clock are displayed on the diagnostics screen.

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.

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).

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.

2.11 KEYBOARD

The control panel keyboard 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

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. The key must be held down for operation.

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.

<u>OVERRIDES:</u>   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 0 to 200%).
100%   Sets control feed rate to programmed feed rate.
+10   Increases current feed rate by 10% (from 0 to 200%).
SPINDLE   Not a key.
-10   Decreases current spindle speed by 10% (from 0 to 200%).
100%   Sets spindle speed to programmed speed.
+10   Increases current spindle speed by 10% (from 0 to 200%).
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. There are three types of Alarms screens. The first shows the currently active alarms. Pressing the RIGHT ARROW key switches to the Alarm History screen. Pressing RIGHT ARROW again switches to the alarm viewer screen. This screen simply shows one alarm at a time with its description (if any). The default will be the last alarm in the alarm history. The user can then scroll through all the alarms by pressing the UP and DOWN ARROW keys. Also, the user can enter an alarm number and press WRITE and it will be displayed. Pressing RIGHT ARROW again returns the user to the home page of the current alarms screen. 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 high-lighted 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.

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 blockdeleted, 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.   0.1 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 SINGL AXIS   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 high-lighted, 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 / ENTER	This acts as the general purpose enter key. Any time that user needs to change 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.

2.12 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.

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 that the outside.

2.13 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← or TS→ 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 ½ second. After ½ second, that axis is moved in the selected direction and at the selected feed rate.

2.14 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 (motion is stopped by pressing Feed hold or Reset). 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.

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.

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-STP. 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 overide 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.

2.15 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 parentheses; maximum of 80 characters) 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.

EDITING PROGRAMS

The EDIT mode is used to make changes to a program already in memory. Or use the LIST PROG mode to create a new one. 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

With BACKGROUND EDIT, you may edit a program in memory while any other program is being run. BACK-GROUND 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 **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 OFSET 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 BACK GROUND EDIT or PROGRAM REVIEW. BACKGROUND EDIT is selected by entering the pro gram 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 cur- rently-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.

2.16 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. 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.

25-25 Pin
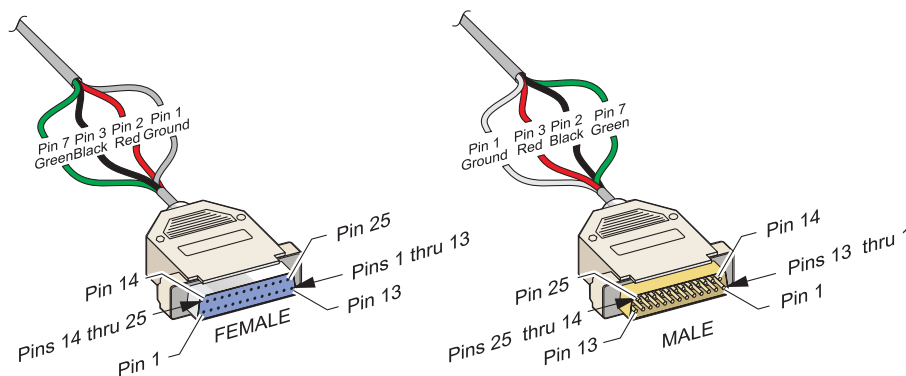
9-25 Pin
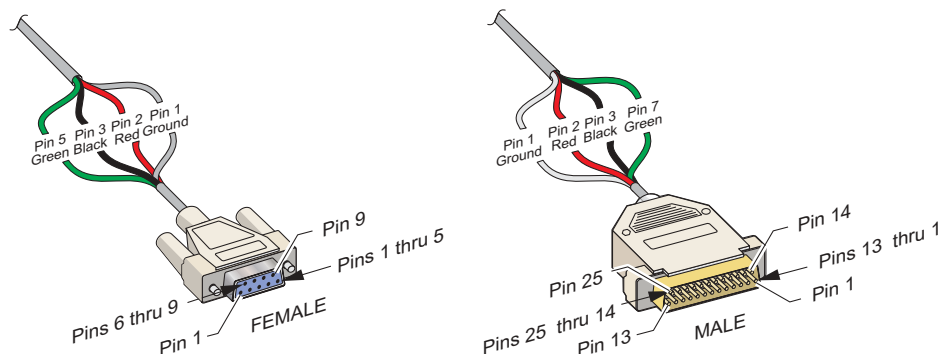
All other pins are optional and are not usually used. The RS-232 connector is a DB-25 and is wired as a DTE. This means that we send data on the TXD wire and receive data on the RXD wire. If you do not understand this, your dealer will be glad to help.

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 a 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.

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 following table shows the use of the N numbers in the output file when offsets are saved to RS-232 or disk.

Tool Geometry:

Tool/function	X Offset	Z Offset	Radius	Taper	Tip
1	101	001	201	301	501
2	102	002	202	302	502
.	.	.	.	.	.
.	.	.	.	.	.
50	150	050	250	350	550

The nnn numbers for coordinate offsets are as follows

Offset/axis	X	Z	B
G52	351	353	355
G54	357	359	361
G55	363	365	367
.			
.			
G59	387	389	391
G110	393	395	397
G111	399	401	403
.			
.			
G128	489	491	493
G129	495	497	499

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.

<div align="center">

WARNING!

One of the biggest causes of electronic damage is a lack of a good earth ground on both the CNC <u>and</u> 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.

</div>

Port #2 on the side cabinet is dedicated to auxiliary axes communication.

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 8k 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.
DISK ABORT   When anything causes abnormal stop.

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:** 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.

PRINTING FROM HAAS MACHINES

Important! You must have a serial cable and serial printer that is compatible with the Haas controller.

Setting up the Printer

- Plug in the printer power cord.
- Plug the printer into the control in the RS232 port.
- Turn on the printer.
- Make sure the following settings are correct.

Note: Write down the old settings in case you to need change them back later.

The Machine Settings for Printing

- Setting #11   9600 (Baud)
- Setting #12   None (Parity)
- Setting #13   1 (Stop Bit)
- Setting #14   Xon/Xoff (synch)
- Setting #37   8 (Data Bits)

Printing Rules

- The program must be in memory. A program cannot be printed in MDI.
- A program cannot be printed while it is running.

Printing a Program

- Make sure the printer is turned on.
- Make sure there is paper in the printer.
- Go to "List Programs".
- Highlight the program to be printed.
- Push "Send RS232.
- To get the last page of the program out, push the paper feed button on the printer.

2.17 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.
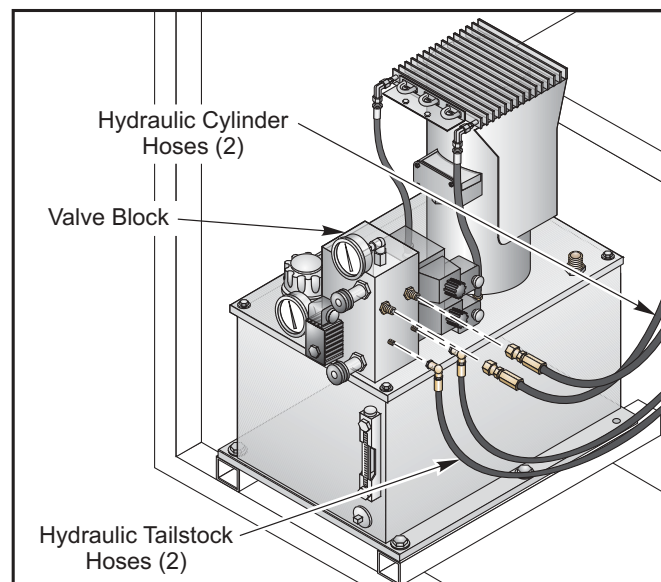
Hydraulic unit pump.

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
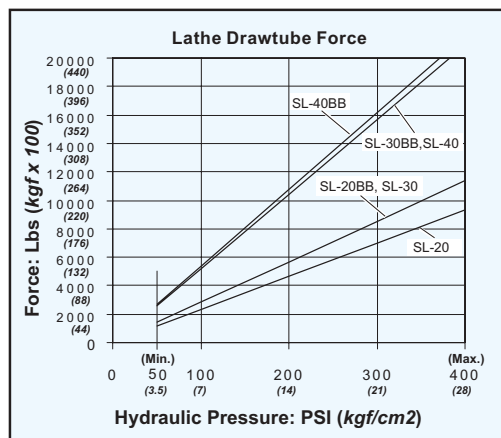
Chart of hydraulic pressure vs. hydraulic drawtube force

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.

LATHE DRAWTUBE SPECIFICATIONS

MINI-LATHE

24.13

Spindle ID = 1.38

1.24-20 UNC2B I.D. Thread
(5C COLLET)

1.18

M34x1.5 O.D. Thread

3.32 Extended
2.93 Retracted

22.37
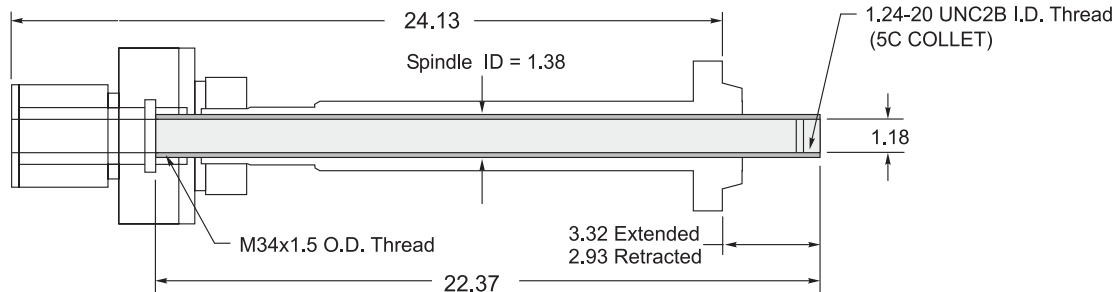
DRAWTUBE STROKE = .37

SL-10 MAIN SPINDLE

33.76

8.99

SPINDLE ID = 2.31

1.81 ID

DRAWTUBE STROKE = .51

RETRACTED
DRAWTUBE
(1.13 Kitagawa)
(1.36 LMC)

EXTENDED DRAWTUBE
(1.72 Kitagawa)
(1.79 LMC)

1.81 ID   2.50 ID

THREAD M55XP2.0
.70" LONG

.62

A2-5 SPINDLE NOSE
11X7/16 ON 12 EQ SP
ON 4.125 DBC

CHUCK MOUNTING
SURFACE

.50

SL-20/TL-15 MAIN SPINDLE

33.00

8.60

2.047 ID

SPINDLE ID = 3.00

DRAWTUBE STROKE = .87

2.01
EXTENDED
DRAWTUBE

1.14
RETRACTED
DRAWTUBE

CHUCK
MOUNTING
SURFACE

2.06 ID   3.34 ID

THREAD M60 X 1.5
1.10" LONG

.56

A2-6 SPINDLE NOSE
11X1/2-13 ON 12 EQ SP
ON 5.250 DBC

.75

SL-20 7K SPINDLE (5" HYDRAULIC CLOSER)
LMC PN ZKP100/34-10

30.31

8.56

1.34 ID

SPINDLE ID = 1.90

DRAWTUBE STROKE = .394

CHUCK MOUNTING SURFACE

1.21 END OF DRAWTUBE

.50

.81 RETRACTED POSITION

1.34 ID   (1.90 ID)

THREAD M40 X 1.5 X
.78" LONG

A2-5 SPINDLE NOSE
11X7/16-14 ON 12 EQ SP
ON 4.125 DBC
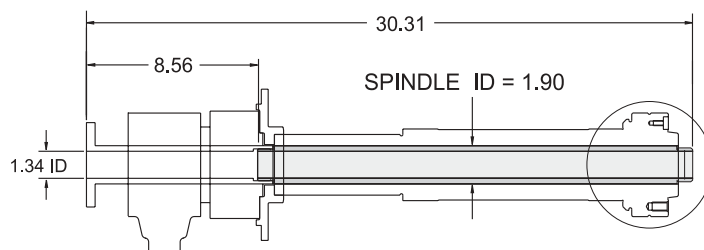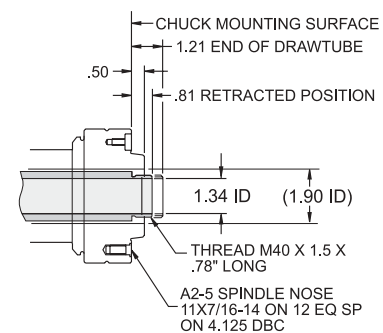
DIMENSIONS IN INCHES

LATHE DRAWTUBE SPECIFICATIONS

SL-20 7K SPINDLE (6"HYDRAULIC CLOSER)
LMC PN ZKP125/46-13

31.85

8.99

SPINDLE ID = 2.31

1.81 ID

DRAWTUBE STROKE = .51

.90 RETRACTED DRAWTUBE

1.41 EXTENDED DRAWTUBE

1.34 ID   2.50

THREAD M40XP1.5 .77" LONG

CHUCK MOUNTING SURFACE

.62

.50

A2-5 SPINDLE NOSE 11X7/16 ON 12 EQ SP ON 4.125 DBC
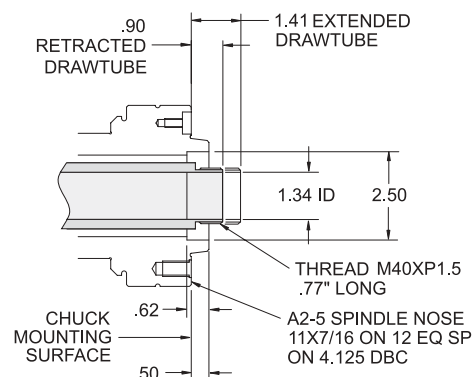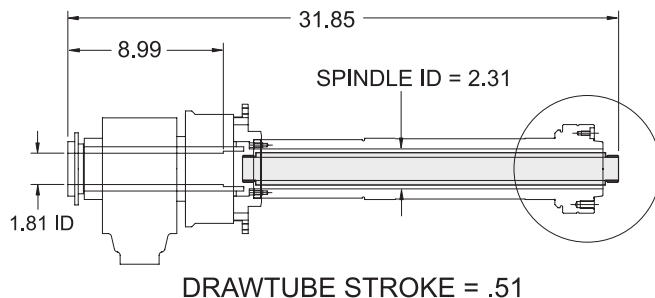
SL-20 BB, TL-15 BB MAIN SPINDLE

33.50

9.84

2.96 ID

SPINDLE ID = 3.00

DRAWTUBE STROKE = .98

2.05 EXTENDED DRAWTUBE

1.07 RETRACTED DRAWTUBE

CHUCK MOUNTING SURFACE

2.56 ID   3.34 ID

THREAD M75 X 2.0 1.30" LONG

.56

.75

A2-6 SPINDLE NOSE 11X1/2-13 ON 12 EQ SP ON 5.250 DBC

TL SERIES SUB-SPINDLE

28.60

8.35

SPINDLE ID = 1.28

1.34 ID

DRAWTUBE STROKE = .39

1.38 EXTENDED DRAWTUBE

.99 RETRACTED DRAWTUBE

.88 ID   2.00

THREAD M40XP1.5 .75" LONG

CHUCK MOUNTING SURFACE

.50

A2-5 SPINDLE NOSE 11X7/16 ON 12 EQ SP ON 4.125 DBC

TL SERIES SUB-SPINDLE
(CURRENT PRODUCTION)

28.60

8.35

SPINDLE ID = 1.38

1.34 ID

DRAWTUBE STROKE = .39

1.38 EXTENDED DRAWTUBE

.99 RETRACTED DRAWTUBE

M34 X 1.5 (After Apr 02) M28 X 1.5 (Before Apr 02)

1.18 ID   2.00

THREAD M40XP1.5 .75" LONG

CHUCK MOUNTING SURFACE

.50

A2-5 SPINDLE NOSE 11X7/16 ON 12 EQ SP ON 4.125 DBC
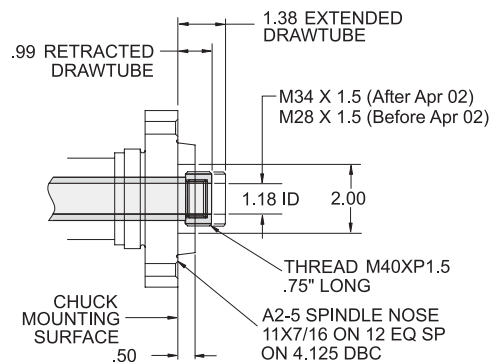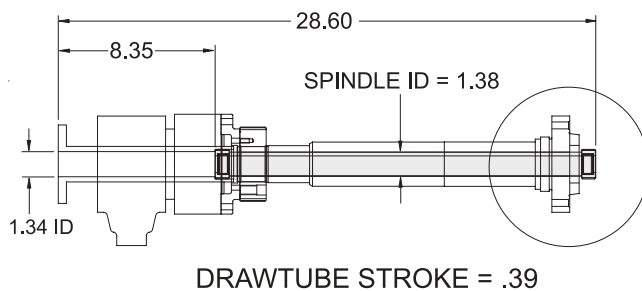
DIMENSIONS IN INCHES

LATHE DRAWTUBE SPECIFICATIONS

SL-30 MAIN SPINDLE*

33.79

9.72

3.03

SPINDLE ID = 3.50

DRAWTUBE STROKE = .75

*S/N 64804 and later

CHUCK MOUNTING SURFACE

2.13 EXTENDED DRAWTUBE

1.38 RETRACTED DRAWTUBE

3.07 ID   3.56

THREAD M85 X 2.0 1.18 LONG

.56

A2-6 SPINDLE NOSE 11 X 1/2-13 ON 12 EQ SP ON 5.250 DBC

.75

SL-30 MAIN SPINDLE**

33.50

9.84

2.96 ID

SPINDLE ID = 3.00

DRAWTUBE STROKE = .98

**Prior to S/N 64804

CHUCK MOUNTING SURFACE

2.05 EXTENDED DRAWTUBE

1.07 RETRACTED DRAWTUBE

2.56 ID   3.34 ID

THREAD M75 X 2.0 1.30" LONG

.56

A2-6 SPINDLE NOSE 11X1/2-13 ON 12 EQ SP ON 5.250 DBC

.75

SL-40, SL-30 BB, SL-30 BF MAIN SPINDLE

48.39

14.19

SPINDLE ID = 4.625

4.626 ID

DRAWTUBE STROKE = .91

CHUCK MOUNTING SURFACE

2.00 RETRACTED DRAWTUBE

2.91 EXTENDED DRAWTUBE

4.06 ID

THREAD M115 X 2.0 1.13" LONG

.69

A2-8 SPINDLE NOSE 11X5/8-11 ON 12 EQ SP ON 6.750 DBC

SL-40 BB MAIN SPINDLE

50.10

3.55

15.61

6.38 ID

SPINDLE ID =6.75

DRAWTUBE STROKE = .99

CHUCK MOUNTING SURFACE

2.87 EXTENDED DRAWTUBE

1.88 RETRACTED DRAWTUBE

6.38 ID   (6.75 ID)

THREAD M170 X 3.0 1.73" LONG

A2-11 SPINDLE NOSE 11X3/4-10 ON 12 EQ SP ON 9.250 DBC

.75
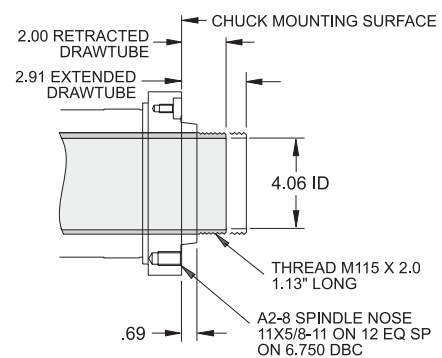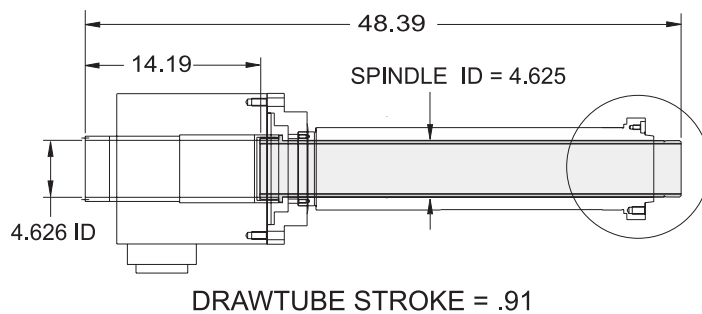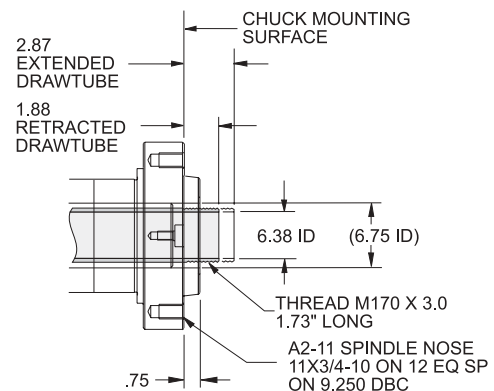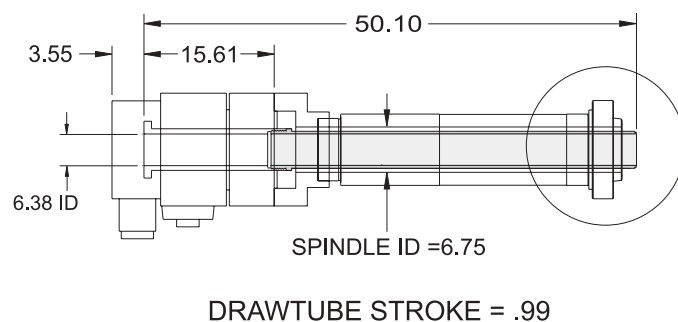
DIMENSIONS IN INCHES

Removal - Removal of Chuck

1. Move both axes to their zero positions. Remove the chuck jaws.
2. Remove the three (3) screws that mount the center cup (or plate) from the center of the chuck and remove the cup.
3. Clamp the chuck and remove the six (6) SHCS that mount the chuck to the collet. See figure 1.
4. Place a chuck wrench inside the center bore of the chuck and unscrew the chuck from the drawtube.
5. **SL-30, SL-40:** Remove the adapter plate. See figure 2.

Removal of Collet

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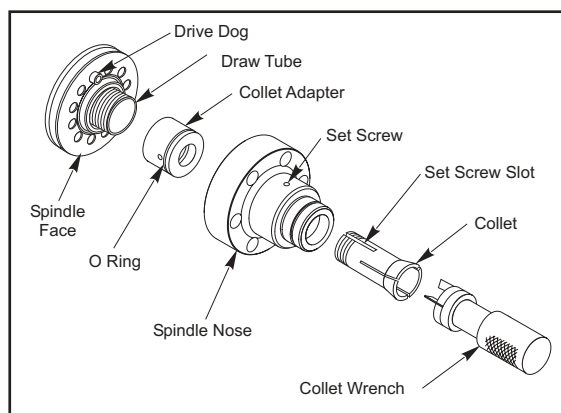
Figure 1
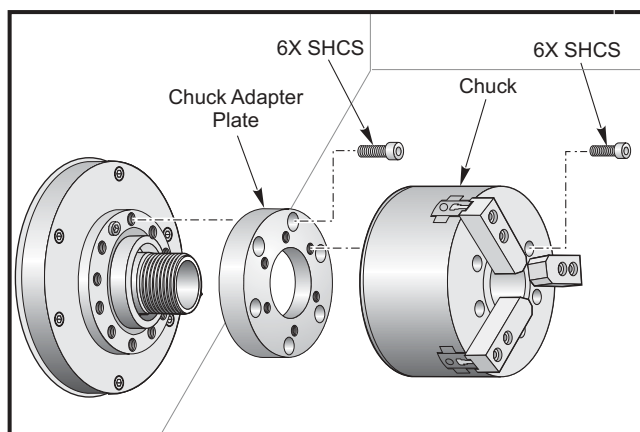
Figure 2

Installation - Installation of Collet

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

Installation of Chuck

On SL-30, SL-40 install an adapter plate before installing the chuck

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

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).
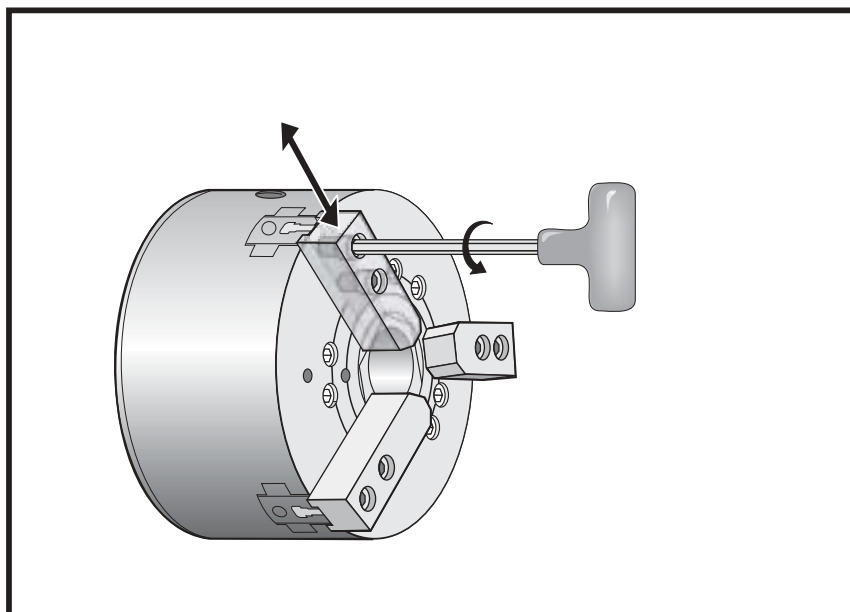
1. Use a hex key to loosen the two SHCS attaching the jaw to the chuck.

2. Slide jaw to new position and re-tighten the two SHCS.

3. Repeat procedure for remaining two jaws. Jaws must remain concentric.

2.18 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 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.
AUX AXIS 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.

DISK ABORT   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!**
SETTING 104 ON	The commanded function is unavailable as Setting 104 is ON.
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 the messages display. Entering the Alarms display will show the full text of an alarm when the alarm message is flashing. There are three types of Alarms screens. The first shows the currently active alarms. Pressing the RIGHT ARROW key switches to the Alarm History screen. Pressing RIGHT ARROW again switches to the alarm viewer screen. This screen simply shows one alarm at a time with its description (if any). The default will be the last alarm in the alarm history. The user can then scroll through all the alarms by pressing the UP and DOWN ARROW keys. Also, the user can enter an alarm number and press WRITE and it will be displayed. Pressing RIGHT ARROW again returns the user to the home page of the current alarms screen. Pressing PAGE DOWN will display a page for user messages and notes.

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/GRAPH 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 24 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.

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

> **Program Command Check (Home Page)**,
> **Current Display Command**,
> **Operation Timers**,
> **Macro Variables**,
> **Maintenance,**
> **Tool Life Timers**,
> **Tool 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 (**CMDnnnnn**), and if Parameter 278 bit DISPLAY ACT is set to 1, the actual encoder spindle speed (**ACTnnnnn**). 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.

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 the Maintenance section).

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. The maximum value is 32767, once this value is reached, and not reset, the control will start back at zero.

This display may also be used to generate an alarm when a tool has been used a specific number of times. The last column is labeled ALARM, 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%, for an extended period of time, can lead to an axis overload alarm.

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. There are three types of Alarms screens. The first shows the currently active alarms. Pressing the RIGHT ARROW key switches to the Alarm History screen. Pressing RIGHT ARROW again switches to the alarm viewer screen. This screen simply shows one alarm at a time with its description (if any). The default will be the last alarm in the alarm history. The user can then scroll through all the alarms by pressing the UP and DOWN ARROW keys. Also, the user can enter an alarm number and press WRITE and it will be displayed. Pressing RIGHT ARROW again returns the user to the home page of the current alarms screen. Pressing PAGE DOWN will display a page for user messages and notes.

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 be 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.

The HOME key displays the first parameter page. Pressing the PAGE DOWN key will display the next page of parameters in the above list. The END key displays the last parameter page. 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 high-lighted.

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/GRAPH 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/GRAPH 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 or the up and down arrows 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	N	SET UP PROCEDURES
B	PROG. REVIEW / DNC / BGEDIT / POWER DOWN	O	OVERRIDES: FEED/SPIN/COOLANT
C	G/M/S/T COMMAND CODES	P	PARAMETERS / DIAGNOSTICS
D	RETURN TO THIS DIRECTORY	Q	POSITION DISPLAYS
E	EDITING PROGRAMS	R	RECV / SEND PROGRAMS
F	SETTING PAGE	S	SAMPLE PROGRAM
G	SPECIAL G CODES	T	TOOL OFS/TOOL LIFE/LOAD
H	TROUBLE SHOOTING	U	GRAPHIC FUNCTION
I	MDI / MANUAL DATA INPUT	V	TOOL TURRET
J	JOGGING / HANDLE FUNCTION	W	WORK COORDINATES
K	CRT DISPLAY / KEYBOARD	X	CREATING PROGRAMS
L	ALARMS / MESSAGES	Y	SPECIAL FUNCTIONS
M	MAINTENANCE REQUIREMENTS	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 (Surface Feet per Minute) = (TURNING DIAMETER IN.) * RPM * 3.14159 / 12

2. (FEED IN./MIN.) = RPM / (THREAD PITCH)

3. (CHIP LOAD IN.) = (FEED IN./MIN.) / RPM / # FLUTES

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, threads per mm, mm per minute, and meters, respectively.

Circle-Line Tangent Calculator
This feature provides the ability to determine points of intersection where a circle and a line meet as tangent. The user provides two points, A and B, on a line and a third point, C, away from that line. The control then calculates the point of intersection where a normal line from point C will intersect with the line AB, as well as, the perpendicular distance to that line. This is equivalent to calculating the radius and the tangent point of a circle, which has its center located at point C, and has a radius equal to the distance from point C to the line AB. Note that until all information is entered, the message NO SOLUTION YET is displayed. Note also that if the center of the circle lies on the line, no solution can be calculated.

```
        CIRCLE-LINE TANGENT


    POINT A X      5.0000
          Y      3.0000
    POINT B X      1.0000
          Y      4.0000
    POINT C X      0.0000
          Y     [0.0000]


    RADIUS         4.1231
    TANGT PT X     1.0000
    TANGT PT Y     4.0000
```
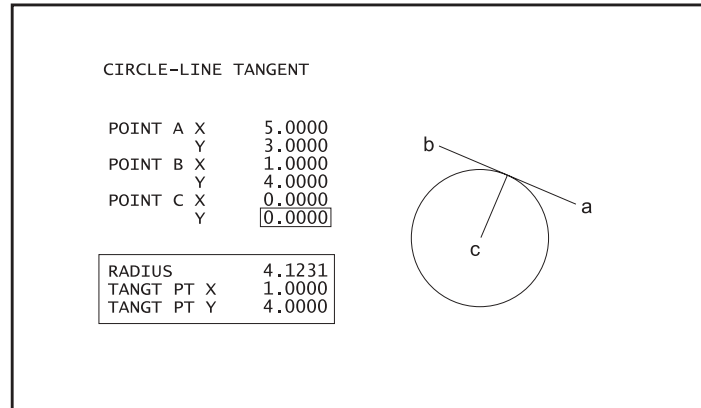
Circle-Circle Tangent Calculator

This feature provides the ability to determine points of intersection between two circles or points. The user provides the location of two circles and their radii. (A zero radius specifies a point instead of a circle.) The control then calculates all the intersection points that are formed by lines tangent to both circles/points. It is important to note that for every input condition where there are two disjointed circles specified, there are up to eight intersection points. Four points are obtained from drawing straight tangents and four points by forming cross tangents. The F1 key is used to toggle between the two diagrams. When F is pressed, the control will prompt for the 'from' and 'to' points (A, B, C, etc.) that specify a segment of the diagram. If the segment is an arc, the control will also prompt for C or W (CW or CCW). Sample G code is then displayed on the input line at the bottom of the screen. When T is entered, the previous 'to' point becomes the new 'from' point and the control prompts for a new 'to' point. When M is pressed, the G code will toggle between G90 and G91 modes. The mode remains active until changed by the user. Because the G code is already on the input line, the user can switch to MDI or EDIT mode and press ENTER to insert the G code into a program.

```
        CIRCLE-CIRCLE TANGENT


    CIRCLE1 X      5.0000
    CIRCLE1 Y      6.0000
    RADIUS 1       4.0000
    CIRCLE2 X      0.0000
    CIRCLE2 Y      0.0000
    RADIUS 2      [2.0000]


    TANGT A X      1.3738
          Y      7.6885
    TANGT B X      7.3147
          Y      2.7378
    TANGT C X     -1.8131
          Y      0.8442
    TANGT D X      1.1573
          Y     -1.6311


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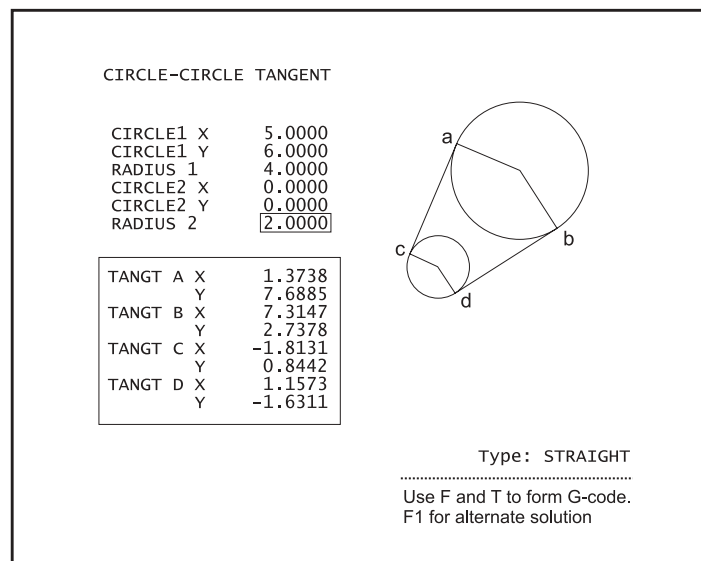
2.19 TOOLCHANGER OPERATIONS

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.
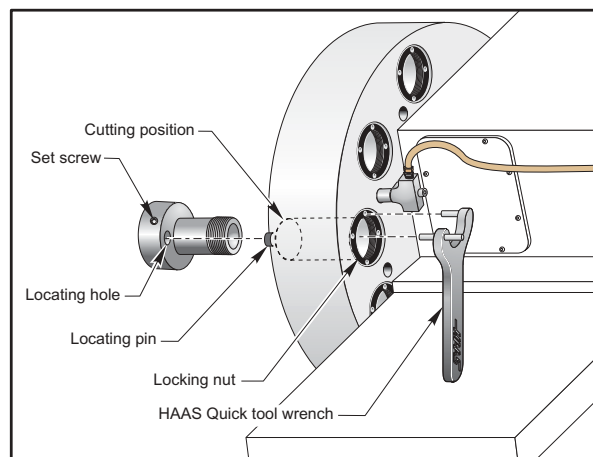
Tool change operation

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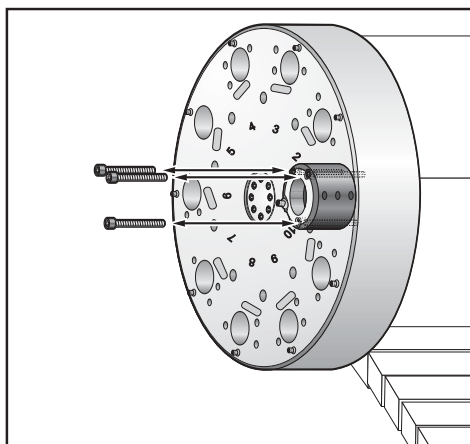
In addition to the retaining nut, the 2 inch I.D. tool holder has three face mounting bolts for extreme rigidity.

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.

IMPORTANT Insert protective caps into any empty turret pockets to protect the pocket from accumulating debris.
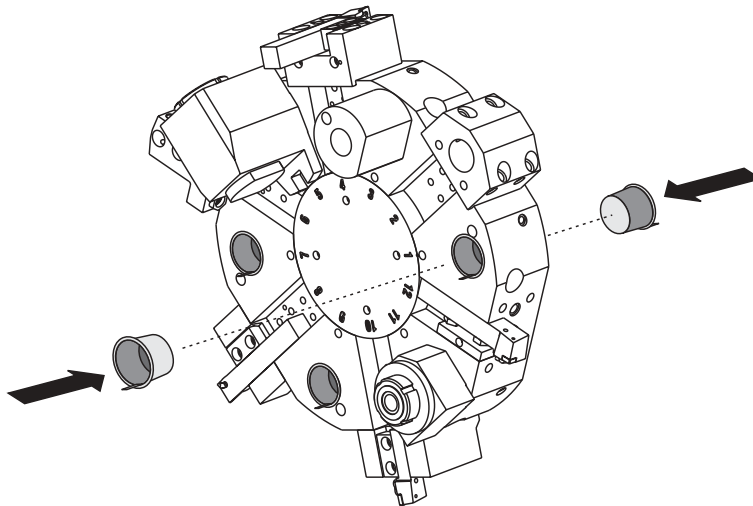
TOOL HOLDER SPECIFICATIONS
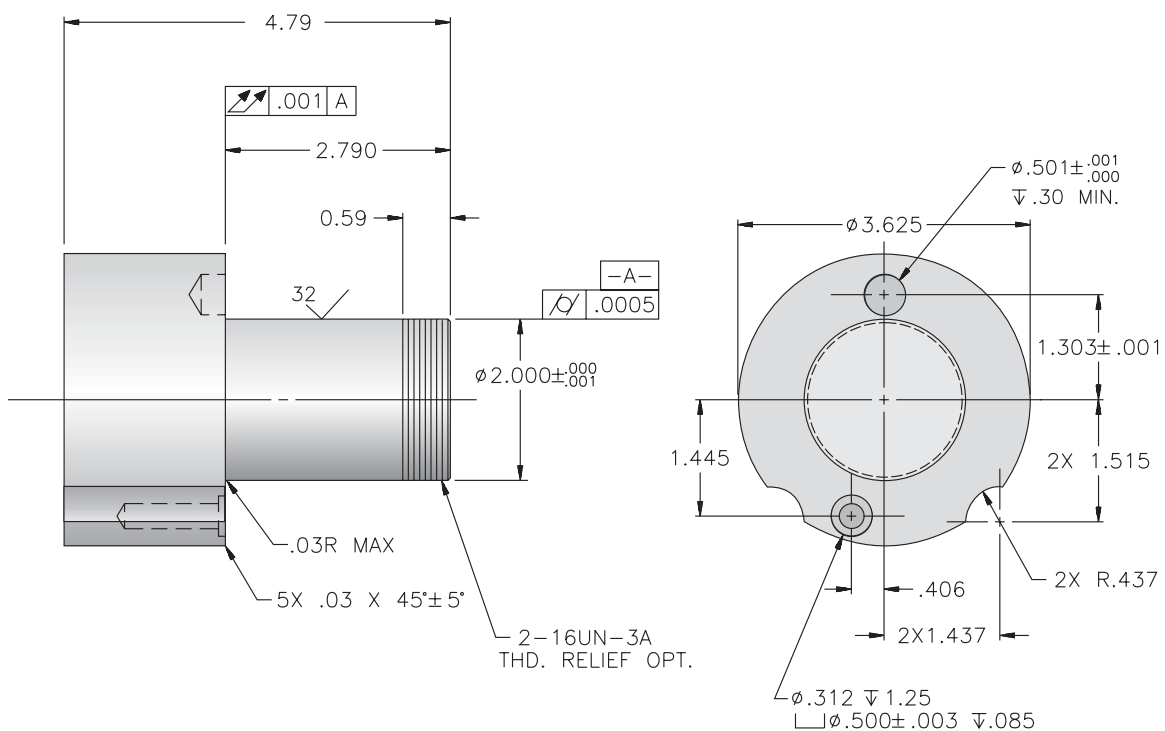
2.20 TRAVEL LIMITS

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.

2.21 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.

2.22 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.

2.23 HOW THE CONTROL MOVES THE MACHINE

Acceleration and deceleration is what the machine does when it is changing speed. Acceleration is what the machine does when the speed is increasing and deceleration is what the machine does when it is slowing down. The machine cannot change speed instantly so that a change of speed occurs over some amount of time and a distance.

Changes of speed affect how the control moves for both rapid motion and feed motion. Rapid motion occurs independently for each axis in motion and uses acceleration set for each axis. Feed motion coordinates one or more axes to accelerate in unison, move in unison, and decelerate in unison. This type of feed motion is called acceleration before interpolation and uses a fixed acceleration rate for all axes.

RAPID MOTION

Rapid motion uses constant acceleration and deceleration, with maximum acceleration and maximum speed set as parameters per axis. End-point arrival in rapid motion occurs with S-curve velocity to prevent shock vibration to the machine. A rapid motion followed by another rapid motion is blended with a rounded corner, controlled by a parameter called "In Position Limit." It is usually about 0.06 inch. A rapid motion followed by a feed motion or a rapid motion in "Exact Stop" mode will always decelerate to an exact stop before the next motion.

S-curve velocity control refers to the rate of change of acceleration or deceleration. Without S-curve, there may be abrupt changes in deceleration, resulting in machine vibration. With S-curve at the end of a rapid move, there are only gradual changes to deceleration and thus machine vibration is reduced.

FEED MOTION

A feed motion coordinates or "interpolates" the motion of multiple axes. Feed motions always use constant acceleration before interpolation. With the Haas control, up to five axes can be in motion in a feed. These are X, Y, Z, A and B. Maximum feedrate is 500 inches per minute for the linear (XYZ) axes and 300 degrees per minute for the rotary (AB) axes.

The blending of axis motion is controlled by Setting 85 (Max Corner Rounding) and the G187 command. Setting 85 and G187 provide for a continuously adjustable range of corner rounding between an exact stop and inexact stop. The value of the setting is the maximum deviation allowed from the exact programmed path.

A linear move (G01) started at an exact stop and ending at an exact stop will have zero positioning error. That is, it will follow exactly that programmed path. It is only at the start or end at speed that blending or corner rounding can occur. This blending action should not be confused with overshoot. Overshoot is where a control would go past a corner and then reverse back onto the required path. The Haas control does not overshoot under any circumstances.

Circular moves (G02 or G03) are not treated any differently than a linear feed motion. In a circular motion, if a feed starts at an exact stop and ends at an exact stop, there is no positional error introduced no matter what the feedrate. Feed in a circular motion is limited to 300 inches per minute. Corner rounding can still occur when a circular motion is blended with a linear or a circular motion, but is also controlled by Setting 85 and G187.

FEED PATH LOOK-AHEAD

Block look-ahead is something that is needed when the distance which the control requires to get up to speed is more than half the length of the programmed linear strokes. Without look-ahead, the control will simply overlap the deceleration of one stroke with the acceleration of the next stroke. This would limit the speed of the motion based on the length of those strokes. The Haas control has block look-ahead standard, but without the High-Speed Machining option it is limited. With look-ahead, blending of one stroke to another can occur at full speed, but the angle of blending is small without the High-Speed Machining option.

2.24 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.
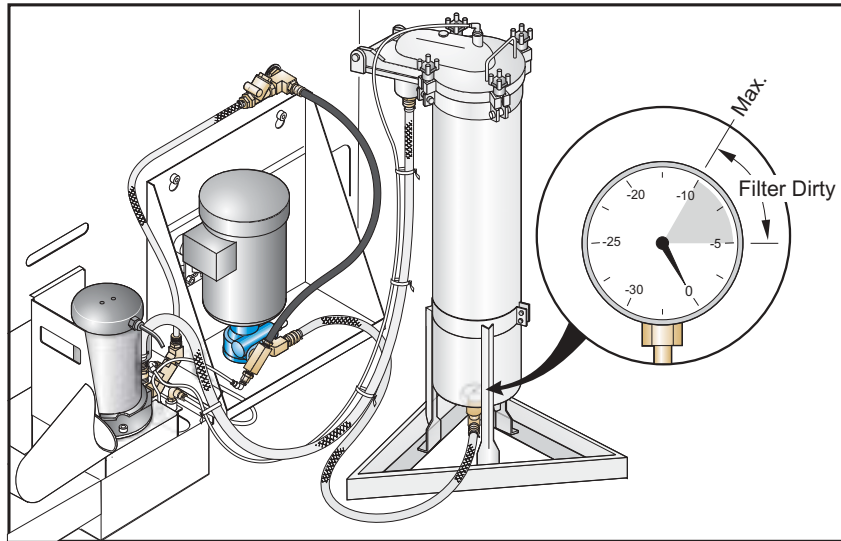
Filter Gauge Indicator

2.25 HIGH INTENSITY LIGHTS

The High Intesity 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 lighing system can be turned on manually using a switch located on the side of the light brackets.

2.26 200 HOUR TRY-OUT

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 theoption. 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.

3. 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.
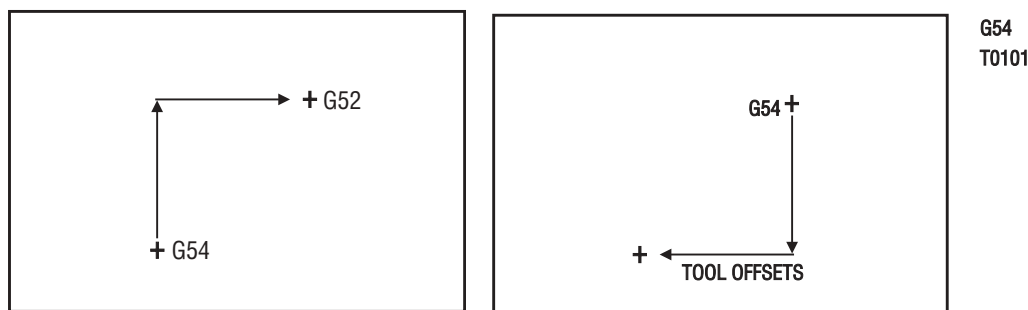
Effective coordinate system and its constituents.

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

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 machine is powered off. 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, M30, reset or power down.

Common Coordinate System (FANUC)

The common (Comm) 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 (G50). The common coordinate system is retained in memory when power is turned off. The common coordinate system can be changed manually 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.

Machine Coordinate System

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

Tool Offsets (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.


```
000101   PROGRAM TO SET UP FIG 4-2
N1       G51 (RETURN TO MACHINE ZERO)
N2       G50 T5100; (SHIFT FOR TOOL 1)
.
.
.
%        TOOL X DIAMETER AND Z SHIFT
```
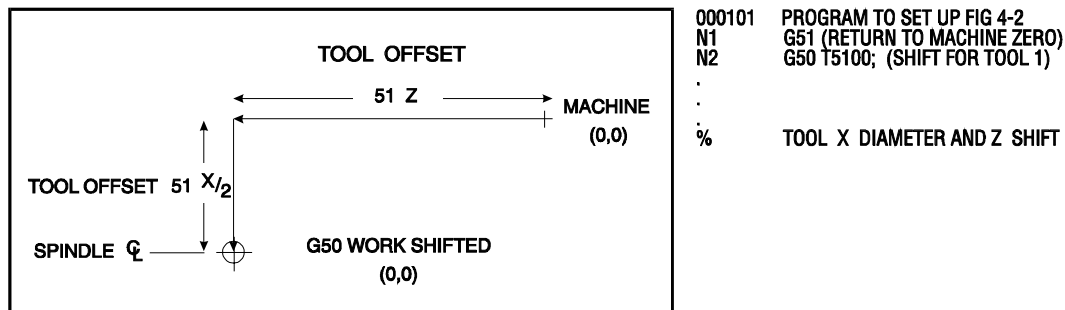
G50 YASNAC Tool Shift.

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 X, Z cordinate reference point as a tool change location. All tools should come to the same point.

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.

3.1 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 may include the optional " / " 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 % SIGN
O12345	:5 DIGIT PROGRAM NUMBER STARTING WITH LETTER ⬚O⬚
(SPINDLE PART)	:PROGRAM NAME HERE WILL SHOW UP IN LIST PROG
N101 G28	:RAPID TO MACHINE ZERO
N102 T101 (ROUGH TURN TOOL)	:TOOL1 OFFSET 1, TOOL INFO BETWEEN PARENTHESIS
N103 G50 S2000	:DEFINES A MAXIMUM SPINDLE SPEED OF 2000 RPM
N104 G97 S554 M03	:CANCEL CSS WITH A CONSTANT SPINDLE SPEED AT 2200
N105 G54 G00 X3.1 Z0.1 M08	:WORK OFFSET G54, RAPID TO X–Z LOCATION, COOLANT ON
N106 G96 S450	:TURN ON CONSTANT SURFACE SPEED AT450 SFM
N107 G71 P108 Q117 D0.095 U.010 W0.005 F0.010	:ROUGH TURN G71 CANNED CYCLE USING PQ PATH
N108 G00 X1.648	:P BLOCK #, FIRST GEOMETRY BLOCK OF G71 COMMAND
N109 G01 Z0. F0.01	:DEFINED BETWEEN BLOCKS N108 TO N117, WITH THE
N110 G03 X1.748 Z–0.05 R0.05 F0.003	:ROUGH CYCLE FEED OF .010 INCH PER REV. IN THE G71
N111 / G01 Z–2.5	:LEAVING .010 INCH FINISH ALLOWANCE IN X DIAMETER
N112 / X2.25	:AND .005 IN THE Z AXIS, WITH A .095 DEPTH OF CUT.
N113 G01 Z–3.25	:N111 AND N112 HAVE A / SLASH SYMBOL TO BE ABLE TO
N114 X2.742 F0.005	:IGNORE THESE LINES ON THE PART, IF YOU TURN ON
N115 X2.942 Z–3.35	:BLOCK DELETE.
N116 Z–4.1	:
N117 X3.1	:Q BLOCK #, LAST GEOMETRY BLOCK OF G71 COMMAND
N118 G97 S554 M09	:CANCEL CONSTANT SURFACE SPEED, COOLANT OFF
N119 G28	:RAPID TO MACHINE HOME FOR A TOOL CHANGE
N120 M01	:OPTIONAL STOP COMMAND
N201 G28	:RAPID TO MACHINE ZERO
N202 T202 (FINISH TURN TOOL)	:CALL TOOL 2 OFFSET 2, TOOL INFO IN PARENTHESIS
N203 G50 S2000	:DEFINES A MAXIMUM SPINDLE SPEED OF 2000 RPM
N204 G97 S554 M03	:CANCEL CONSTANT SURFACE SPEED AT 554
N205 G54 G00 X3.1 Z0.1 M08	:WORK OFFSET G54, RAPID TO X–Z LOCATION, COOLANT ON
N206 G96 S450	:TURN ON CONSTANT SURFACE SPEED AT450 SFM
N207 G71 P108 Q117	:FINISH TURN G70 PART GEOMETRY BETWEEN P AND Q
N208 G97 S554 M09	:CANCEL CONSTANT SURFACE SPEED, COOLANT OFF
N209 G28	:RAPID TO MACHINE HOME FOR A TOOL CHANGE
N210 M01	:OPTIONAL STOP COMMAND
N301 G28	:RAPID TO MACHINE ZERO
N302 T303 (1/2 DIA. SPOT DRILL)	:CALL TOOL 3 OFFSET 3, TOOL INFO IN PARENTHESIS
N303 G97 S2200 M03	:CANCEL CSS WITH A CONSTANT SPINDLE SPEED AT 2200
N304 G54 G00 X0 Z1. M08	:WORK OFFSET G54, RAPID TO X–Z LOCATION, COOLANT ON
N305 G82 Z–0.2 P0.3 R0.1 F.0025	:SPOT DRILL WITH A .3 SECONDS DWELL AT THE BOTTOM

N306 G80 G00 Z1. M09	:CANCEL CANNDED CYCLE, RAPID TO 1.0, COOLANT OFF
N307 G28	:RAPID TO MACHINE HOME FOR A TOOL CHANGE
N308 M01	:OPTIONAL STOP COMMAND
N401 G28	:RAPID TO MACHINE ZERO
N402 T404 (5/16 DIA. DRILL)	:CALL TOOL 4 OFFSET 4, TOOL INFO IN PARENTHESIS
N403 G97 S2750 M03	:CANCEL CSS WITH A CONSTANT SPINDLE SPEED AT 2750
N404 G54 G00 X0 Z1. M08	:WORK OFFSET G54, RAPID TO X–Z LOCATION, COOLANT ON
N405 G83 Z–1.5 Q0.2 R0.1 F0.003	:PECK DRILL TO A 1.5 DEPTH PECKING EVERY 0.200
N406 G80 G00 Z1. M09	:CANCEL CANNDED CYCLE, RAPID TO 1.0, COOLANT OFF
N407 G28	:RAPID TO MACHINE HOME FOR A TOOL CHANGE
N408 M01	:OPTIONAL STOP COMMAND
N401 G28	:RAPID TO MACHINE ZERO
N402 T505 (3/8–16 TAP)	:CALL TOOL 5 OFFSET 5, TOOL INFO IN PARENTHESIS
N403 G97 S650 M03	:CANCEL CSS WITH A CONSTANT SPINDLE SPEED AT 650
N404 G54 G00 X0 Z1. M08	:WORK OFFSET G54, RAPID TO X–Z LOCATION, COOLANT ON
N405 G84 Z–.825 R0.2 F0.0625	:RIGHT HAND TAPPING TO A .825 DEPTH
N406 G80 G00 Z1. M09	:CANCEL CANNDED CYCLE, RAPID TO 1.0, COOLANT OFF
N407 G28	:RAPID TO MACHINE HOME FOR A TOOL CHANGE
N408 M01	:OPTIONAL STOP COMMAND
%	:PROGRAMS MUST END WITH A % SIGN

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 in the middle of a program, 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.
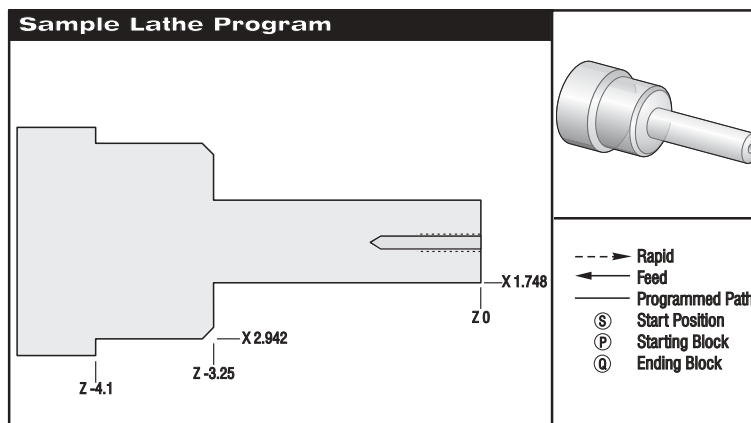
Sample Lathe program

3.2 TOOLING DIMENSIONS SL-20, SL-30, SL-40

HAAS/VDI OD TOOL

HAAS/VDI ID TOOL

BOT OD TOOL

BOT ID TOOL

LIVE RADIAL TOOL

LIVE AXIAL TOOL

* Subtract 1.00" on SL-20 and SL-30 models with Live Tooling
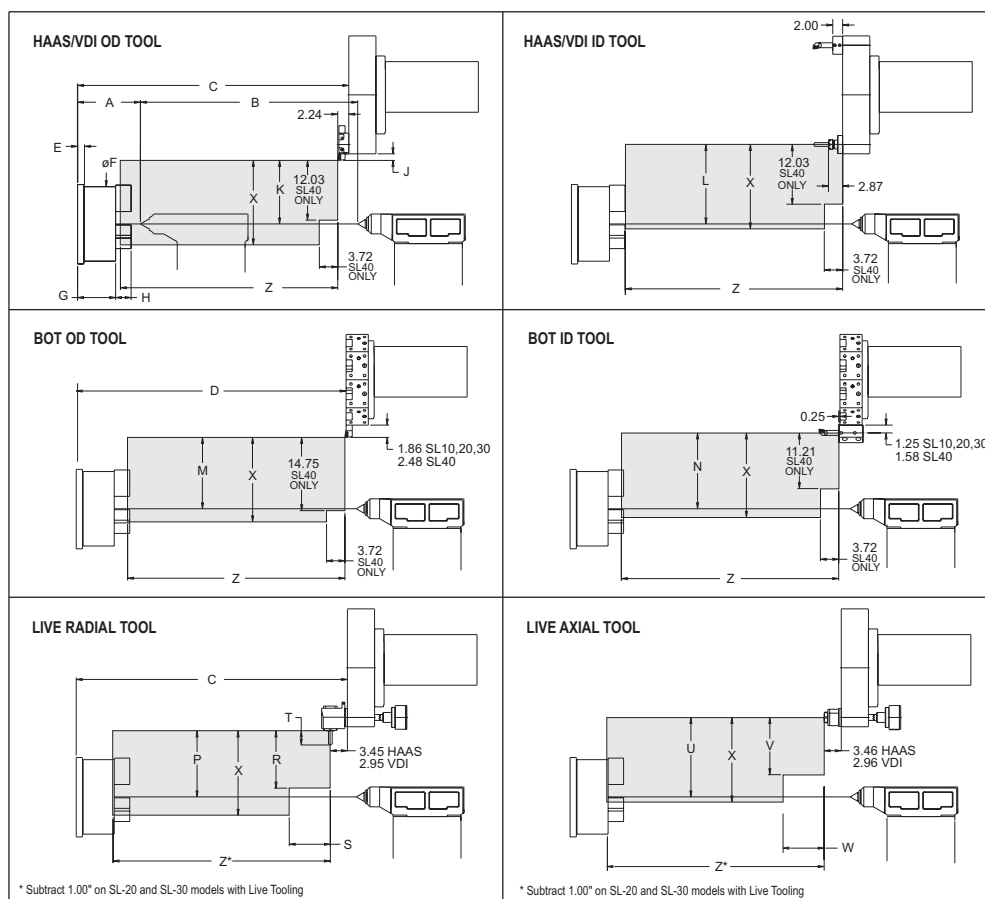
Dimensions	SL-10	SL-20	SL-30	SL-40		SL-10	SL-20	SL-30	SL-40
A – Tailstock at Full Travel	9.35" 237 mm	10.96" 278 mm	12.03" 306 mm	12.72" 323 mm	M – BOT OD Tool Home	5.19" 132 mm	5.14" 131 mm	8.05" 205 mm	14.37" 365 mm
B – Tailstock Travel	13.55" 344 mm	20.00" 508 mm	34.00" 867 mm	44.00" 1118 mm	N – BOT ID Tool Home	5.80" 147 mm	5.75" 146 mm	8.66" 220 mm	15.28" 388 mm
C – Turret Home Haas/VDI	N/A	25.73" 654 mm	39.90" 1013 mm	54.91" 1395 mm	P – Radial Live Tool Home	N/A	4.74" 120 mm	7.59" 193 mm	13.29" 338 mm
D – Turret Home BOT	18.96" 482 mm	25.73" 654 mm	39.90" 1013 mm	54.64" 1388 mm	R – Radial Live Tool X T/S Zone	N/A	3.52" 89 mm	5.83" 148 mm	11.53" 293 mm
E – Spindle Face	2.12" 54 mm	1.39" 35 mm	1.39" 35 mm	1.40" 36 mm	S – Radial Live Tool Z T/S Zone	N/A	4.40" 112 mm	4.01" 102 mm	8.32" 211 mm
F – Chuck Diameter	6.50" 165 mm	8.27" 210 mm	10.00" 254 mm	15.00" 381 mm	T – Radial Live Tool Stickout	N/A	2.83" 72 mm	4.12" 105 mm	4.86" 123 mm
G – Chuck Face	5.66" 144 mm	5.45" 139 mm	6.12" 155 mm	7.70" 196 mm	U – Axial Live Tool Home	N/A	7.45" 189 mm	10.30" 262 mm	16.00" 406 mm
H – Chuck Jaws	1.26" 32 mm	2.00" 51 mm	1.81" 46 mm	3.14" 80 mm	V – Axial Live Tool X T/S Zone	N/A	3.52" 89 mm	5.83" 148 mm	11.53" 293 mm
J – Haas OD Tool Stickout	N/A	1.26" 32 mm	1.15" 29 mm	1.33" 34 mm	W – Axial Live Tool Z T/S Zone	N/A	4.40" 112 mm	4.01" 102 mm	8.32" 211 mm
K – Haas OD Tool Home	N/A 113 mm	4.44" 185 mm	7.29" 325 mm	12.79"	X – X-Axis Travel	6.25" 159 mm	8.45" 215 mm	11.30" 287 mm	17.00" 432 mm
L – Haas ID Tool Home	N/A	7.45" "190 mm	10.30" 262 mm	16.00" 406 mm	Z – Z-Axis Travel	14.00" 356 mm	20.00" 508 mm	34.00" 864 mm	44.00" 1118 mm

3.3 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.

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

SETTINGS AND PARAMETERS

Jog handle can be used to scroll through settings and parameters

This control can turn itself off 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 is 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 then 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.

3.5 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.

3.6 HYDRAULIC TAILSTOCK OPERATION *

*This option is not field installable

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

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.
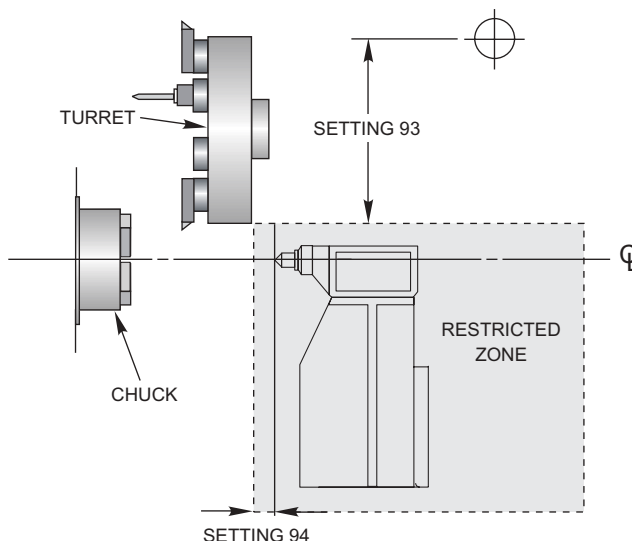
Tailstock Restricted Zone

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.
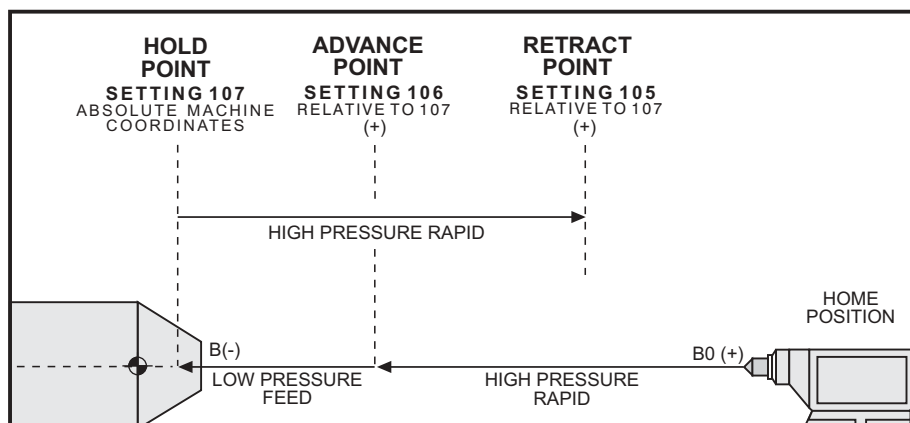
Diagram of Settings 105, 106, and 107.

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 is 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.

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 unreliably.
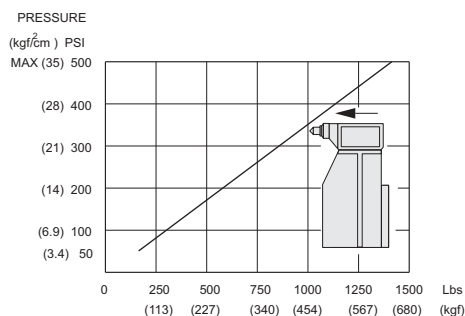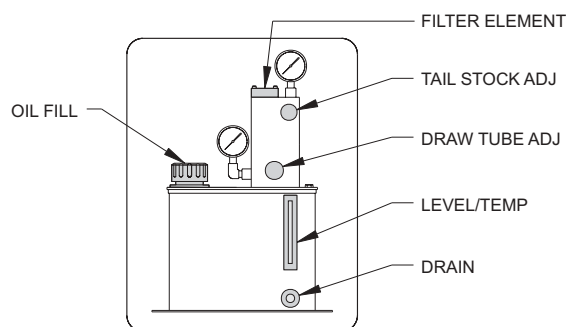
HYDRAULIC TAILSTOCK FORCE
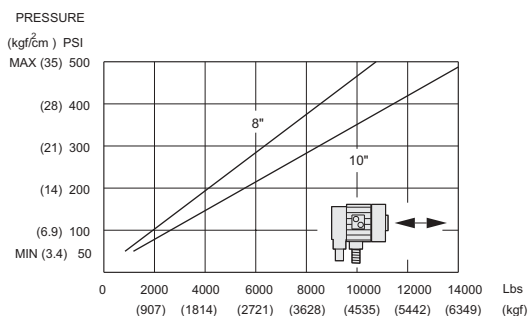
HYDRAULIC DRAWTUBE FORCE
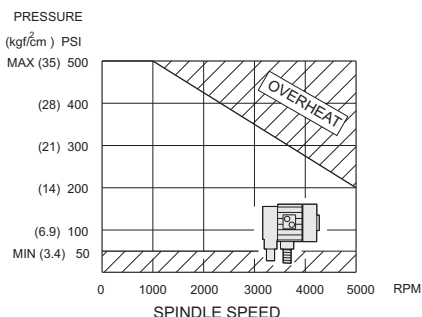
MAXIMUM RECOMMENDED CYLINDER PRESSURE
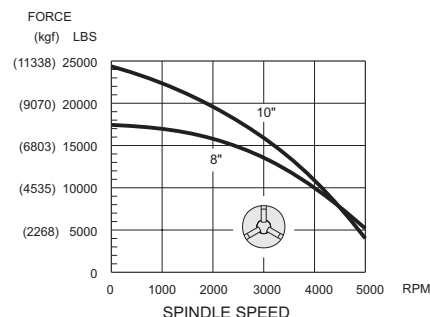
CHUCK GRIPPING FORCE (STANDARD JAWS, 330 PSI)

SL 20 and 30

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.

STEADY REST ADJ

OIL FILL

FILTER ELEMENT

TAIL STOCK ADJ

DRAW TUBE ADJ

LEVEL/TEMP

DRAIN

PRESSURE
(kgf/cm^2) PSI
MAX (35) 500
(28) 400
(21) 300
(14) 200
(6.9) 100
(3.4) 50

0   250   500   750   1000   1250   1500   Lbs
   (113) (227) (340) (454) (567) (680) (kgf)

HYDRAULIC TAILSTOCK FORCE

PRESSURE
(kgf/cm^2) PSI
MAX (35) 500
(28) 400   15"
(21) 300
(14) 200
(6.9) 100
MIN (3.4) 50

0   5000 10000 15000 20000 25000 30000 Lbs
  (2268) (4536) (6803) (9070)(11338)(13605) (kgf)

HYDRAULIC DRAWTUBE FORCE

PRESSURE
(kgf/cm^2) PSI
MAX (35) 500
(28) 400   OVERHEAT
(21) 300
(14) 200
(6.9) 100
MIN (3.4) 50

0   500   1000   1500   2000   2500   RPM
SPINDLE SPEED

**MAXIMUM RECOMMENDED
CYLINDER PRESSURE**

FORCE
(kgf) LBS
(18141) 40000
(14512) 32000   15"
(10884) 24000
(7256) 16000
(3628) 8000
0

0   500   1000   1500   2000   2500   RPM
SPINDLE SPEED
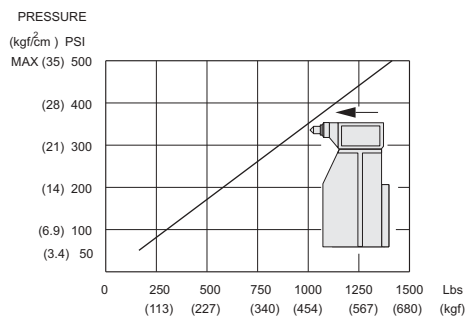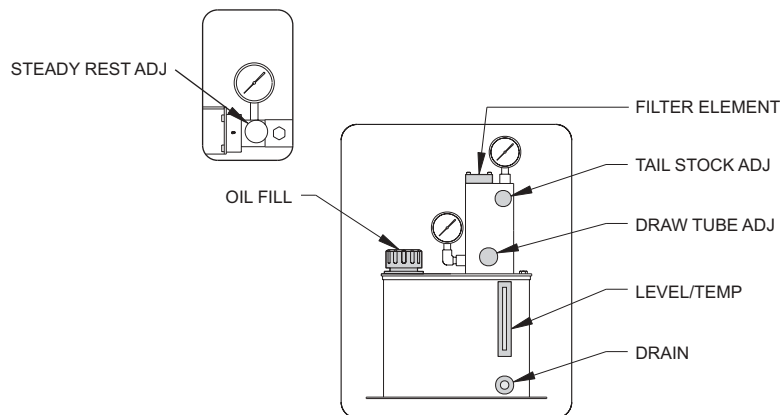
CHUCK GRIPPING FORCE
(STANDARD JAWS, 330 PSI)

SL30BB, SL-40 and SL-40L

SL-10 TAILSTOCK OPERATION

The optional HAAS Tailstock for the SL-10 is a hydraulically actuated quill which moves within a positionable cast iron headstock. The tailstock is manually positioned and held in place with the lock-lever. Tailstock motion is controlled in one of 3 ways:

1. Through program code,
2. In jog mode,
3. By a foot switch.

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

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

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

CAUTION!!

1. It is important to verify tailstock and turret clearance before operating machine or serious damage could occur. Adjust setting 93 and 94 as necessary

2. FEED HOLD will _not_ stop the hydraulic tailstock.

3. Retract tailstock fully and set pressure to zero when not in use. The tailstock may drift out from its retracted position when the solenoid valve is not activated.

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
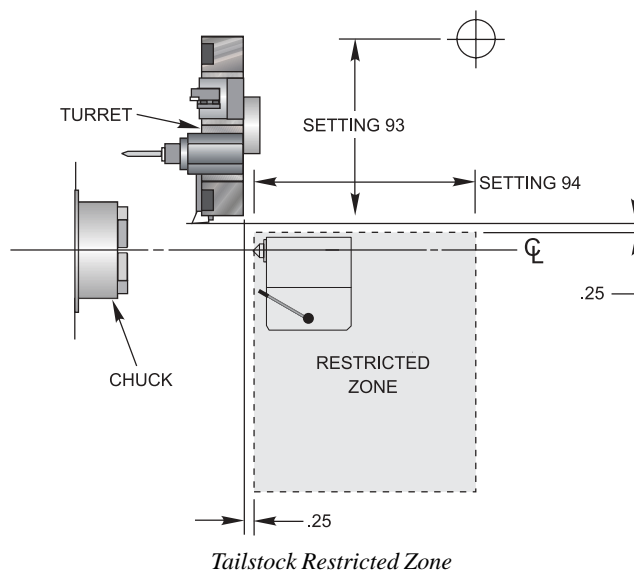
Setting A Restricted Zone For The Tailstock

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. Setting 93 specifies the X axis clearance plane and Setting 94 specifies the Z. 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.

Note: This provides 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.

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.
- Position the tailstock in the machining location
- 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 The Z-Zone Below The X Clearance Plane:

- Place the control in Jog Mode.
- Select the Z-Axis and move the turret towards the spindle as far as it will go.
- 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.

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.

TAILSTOCK PROGRAMING

M21 will cause the tailstock quill to extend toward the spindle.
M22 will cause the tailstock quill to retract away from the spindle.

When an M21 is commanded, the tailstock center will be commanded to move towards the spindle and maintain continuous pressure. Note that the program will not wait while this is completed. Instead, the next block will be executed immediately. A dwell should be commanded 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.

Note:	The tailstock may creep out when not in use. If the tailstock is not used during an operation, either turn the tailstock pressure valve to zero, or adjust the time the retract pressure is applied. This time is adjusted using Parameter 580 "TS HYD RETRACT TIME" which controls the time interval retract pressure is applied. It is measured in milliseconds. A value of 60,000 will cause the pressure to remain on for 1 minute, 900,000 for 15 minutes, etc…. The factory setting is 5000 (5 seconds).

TAILSTOCK FOOT PEDAL OPERATION

- Invokes M21 or M22, depending on current position.

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.

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.

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

HYDRAULIC POWER SYSTEM

1. SWITCH POWER OFF BEFORE INSPECTING, LUBRICATING OR CLEANING ANY HYDRAULIC COMPONENTS.

2. BLOW ACCUMULATED DUST FROM HYDRAULIC HEAT EXCHANGER COILS LOCATED ON TOP OF MOTOR.

HYDRAULIC DRAW TUBE CYLINDER

3. REMOVE CHIPS FROM COOLANT RETURN SUMP ON HYDRAULIC DRAW BAR CYLINDER PERIODICALLY.

   WHEN NOT USING THROUGH HOLE YOU MUST INSTALL A PLUG IN THE DRAW TUBE TO PREVENT CHIPS AND COOLANT FROM ENTERING HYDRAULIC DRAW BAR CYLINDER.
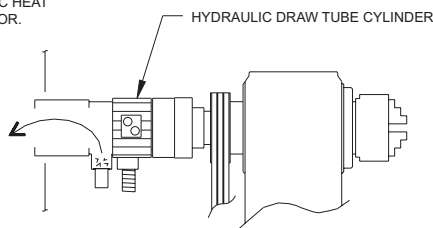
4. NEVER ATTACH DEAD LENGTH STOPS TO THE HYDRAULIC CYLINDER, DAMAGE WILL RESULT.

5. NORMAL OPERATING PRESSURES

		PSI	(kgf/cm^2)
	MAXIMUM CHUCK HYDRAULIC PRESSURE	300	(21)
	MAXIMUM TAILSTOCK HYDRAULIC PRESSURE	200	(14)
DRAW TUBE	NORMAL PRESSURE FOR 5C COLLETS	160	(11)
	NORMAL PRESSURE FOR CHUCKS	200	(14)
TAILSTOCK	NORMAL PRESSURE FOR TAILSTOCK	50	(3.5)

- AVOID HIGH DRAW TUBE PRESSURES AT HIGH RPM, OVERHEATING OF HYDRAULIC CYLINDER WILL OCCUR. NORMAL OPERATING TEMPERATURE OF HYDRAULIC CYLINDER 100 TO 140 °F.

6. USE DTE 24/25 OR EQUIVALENT HYDRAULIC OIL. CHANGE OIL EVERY 24 MONTHS.
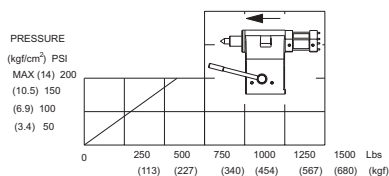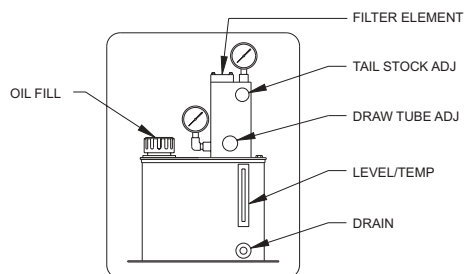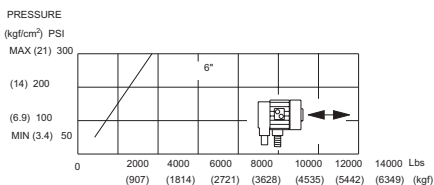
FILTER ELEMENT
TAIL STOCK ADJ
OIL FILL
DRAW TUBE ADJ
LEVEL/TEMP
DRAIN

HYDRAULIC TAILSTOCK FORCE

HYDRAULIC DRAWTUBE FORCE
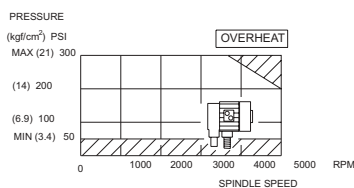
MAXIMUM RECOMMENDED CYLINDER PRESSURE
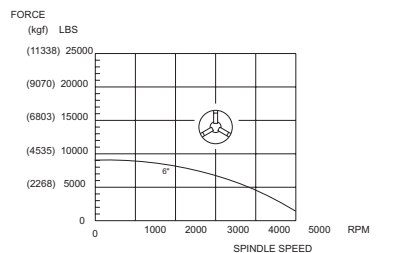
CHUCK GRIPPING FORCE
(STANDARD JAWS, 300 PSI)

SL-10

3.7 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. The following M Codes are displayed in their factory settings. They are used for Parts Catcher operation:

M36 to activate
M37 to deactivate
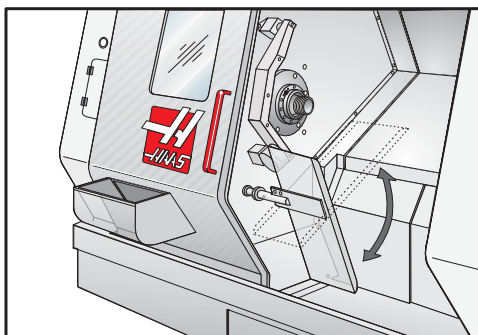
Max. Part Length: 5.5" (all machines)
Max. Bar Stock Dia.:   2.0" SL-20
                        2.5" SL-30

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 (M36), 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.

Shaft Collar
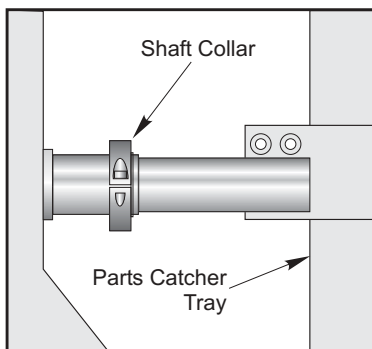
Parts Catcher
Tray

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.

    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.

<div align="center">

WARNING!!

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

</div>

SL-10 Turning Centers

Large chuck jaws may interfere with the operation of the parts catcher. Be sure to check the clearances before operating the parts catcher.
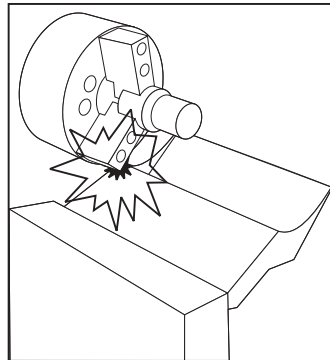
3.8 TOOL PRE-SETTER

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

OPERATION

The Tool Pre-Setter is 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 (set at the factory). 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 in the upright position can be pressed. This will re-enable all axes, and the tool can be carefully backed off.

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**).

SETTINGS

The following settings pertain to probe offsets and have been factory set. Changes to these settings may cause damage to the machine:

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.
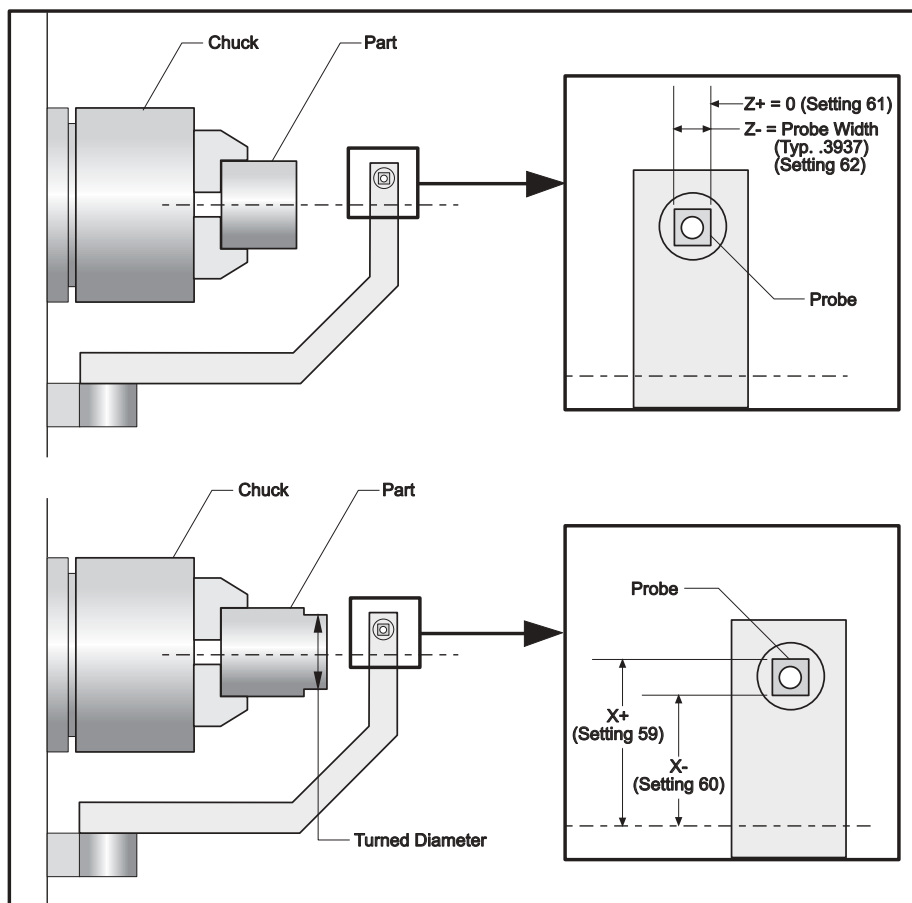
Probe Settings Diagram.

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 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 tool.

7. Press the F2 key. This function converts the value in Parameter 254 (Spindle Center) into programmed units (inches/millimeters) and stores it in the selected X axis tool offset.

8. In .001 inch mode, jog the tool in the Z-direction until it touches the probe. That value is then stored in the selected Z axis tool offset.

SETTING WORK ZERO OFFSETS

Before running your program, the machines WORK ZERO OFFSETS (G52-129) must be entered.

1. In the OFFSETS page select the desired work offset.

2. Call up the desired tool and touch off the face of the part.

3. Press Z FACE MESUR; this will reference the rest of the tools to the face of the part.

3.9 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.

AUGER KEYBOARD COMMANDS

The auger can be started at any time from the keyboard. The auger can be enabled in either direction by pressing the CHIP FWD or CHIP REV and stopped by pressing the CHIP STOP key. The reverse direction of the auger will stop when the CHIP REV key is released. The auger will also stop by pressing the RESET key.

AUGER PROGRAM COMMANDS

Use M codes M31and M33 to control the auger from within a program or in MDI. M31 commands the auger forward and M33 stops the auger. Refer to the "M Codes" section for a more detailed description.

AUGER PARAMETERS

The following parameters control the conveyor:

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

3.10 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.

3.11 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 50. The tool geometry X and Z values are added to the work offsets. If tool nose compensation is used, yy specifies the tool geometry index for radius, taper, and tip. If yy=00 no tool geometry or wear is applied.

YASNAC Coordinate System

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

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

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

NOTE: If Tnn is used, it works like Tnnnn, i.e.,oo=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.

3.12 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. Rigid tapping eliminates the pullout and distortion of the first thread that occurs on all spring compression/tension devices and tapping heads. 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.

3.13 CONTOURING ACCURACY

When machining 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.

3.14 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.

3.15 HAAS BOLT-ON TURRET

The Bolt-On Turret 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.
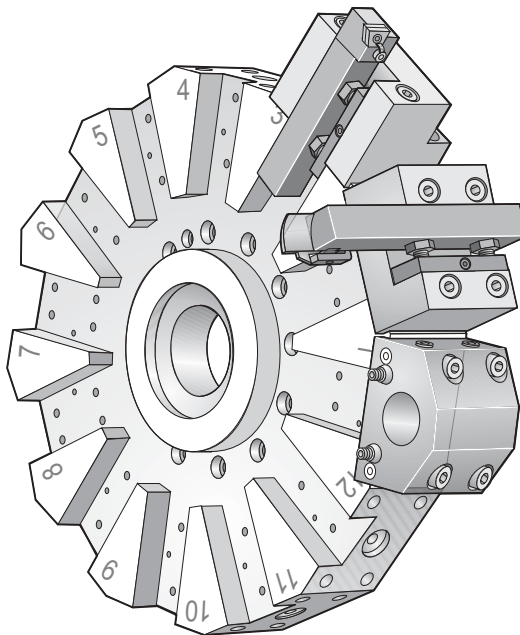
SL10	12 Tool Turret
SL20	10 Tool Turret
SL30	12 Tool Turret
SL40	10 Tool Turret
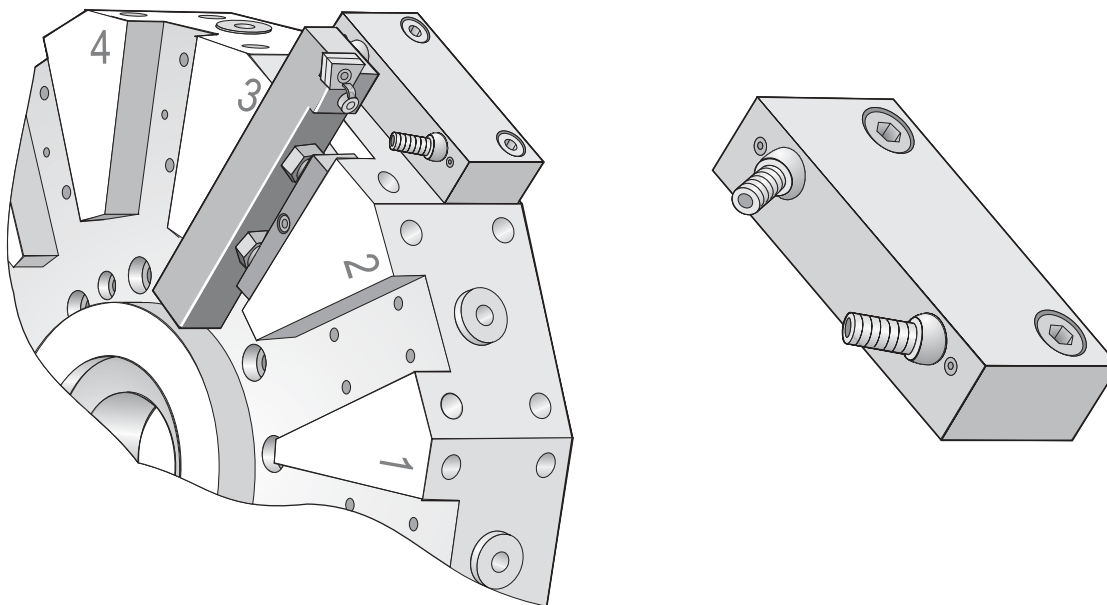
Coolant Nozzle Block

BOLT ON TURRET SETUP

SPINDLE CENTER VALUE USED IN TOOL OFFSET TABLES HAS
BEEN PRE-SET BY THE FACTORY BASED ON THE FOLLOWING VALUES:

MACHINE	TOOL HOLDER CENTER HEIGHT
SL-10	1.000" (25.4mm)
SL-20	1.250" (32mm)
SL-30	1.250" (32mm)
SL-40	1.575" (40mm)

*CHANGING SPINDLE CENTER VALUE IS NOT RECOMMENDED
UNLESS ALL OTHER BORING HOLDERS HAVE THE SAME
CENTER HEIGHT. SPINDLE CENTER PARAMETER EFFECTS
TOOL PRE-SETTER ARM AND OTHER MACHINE FUNCTIONS.

TOOL HOLDER
CENTER HEIGHT

TOOL HOLDER OFFSET VALUE
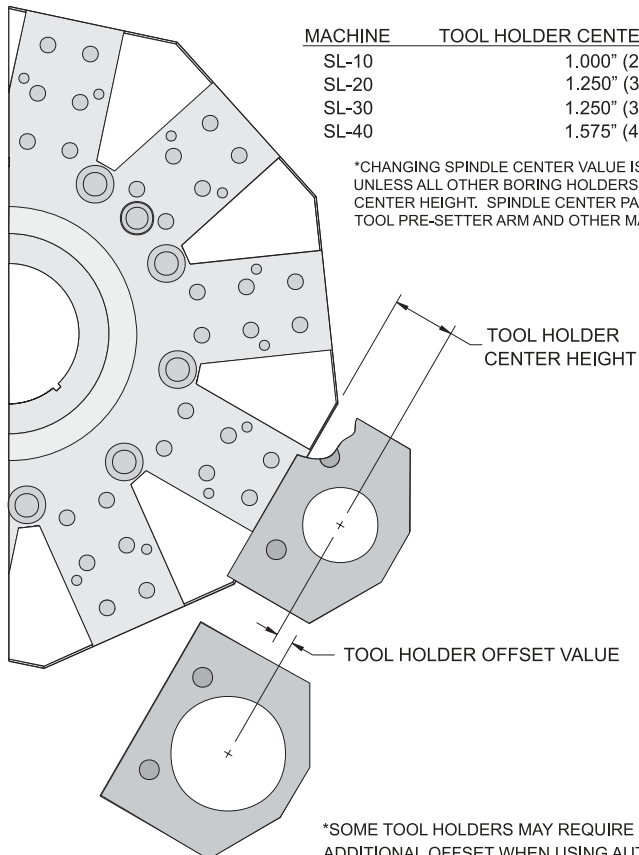
*SOME TOOL HOLDERS MAY REQUIRE
ADDITIONAL OFFSET WHEN USING AUTOMATIC
SPINDLE CENTER COMPENSATION.
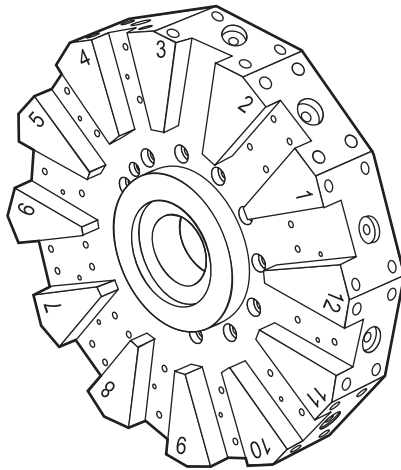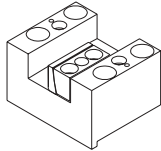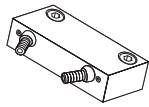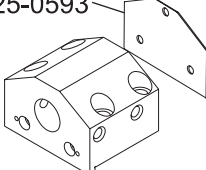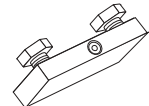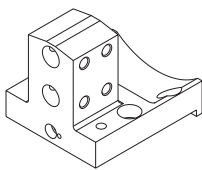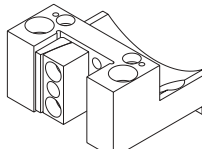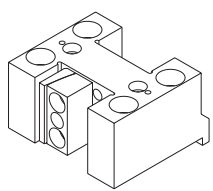
Bolt-on Tool Holders

• SL-20/30 10 and 12 station turret
• Sl-10 10 station turret (up to Aug 1, 2001)
• TL-15 12 station 6 BOT + 6 VDI

Basic BOT ID Tool Specifications

.250"   1.250"
1.580 (SL-40)

BOLT ON TOOLS PARTS LIST					
Face Grooving	Part No.	Size	**Coolant Block**	Part No	Size
	BOT20F/G BOT20MFG-25	1" 25mm		CB20, SL 10 **/20/30 CB40, SL 40 CB20M CB40M	
Boring Bar 25-0593	BOT20ID-1" BOT20ID-1 1/4" BOT20ID-1 1/2" BOT20ID-2" BOT20MID-32 BOT20MID-40	1" 1 1/4" 1 1/2" 2" 32mm 40mm	**Turret Face Clamps**	BOT20OD/Clamp (Clamp Block SL 10 **/20/30) BOT40OD/Clamp (Clamp Block SL 40) BOT 20MOD/Clamp BOT 40MOD/Clamp	1" 1 1/4" 25mm 32mm
Twin Bore	BOT 20ID-3/4* BOT 20MID-20*	3/4" (2x) 20mm (2x)	**Nozzles**	20-0562 90° 20-0689 Str long 20-2077 Plug 20-8730 Str 20-8739 45°	
Parting Tool	BOT 20OD-1* BOT20MOD-25*	1" 25mm	**Twin Turn**	BOT 20OD-1T* BOT20MOD-25T*	1" (2x) 20mm(2x)
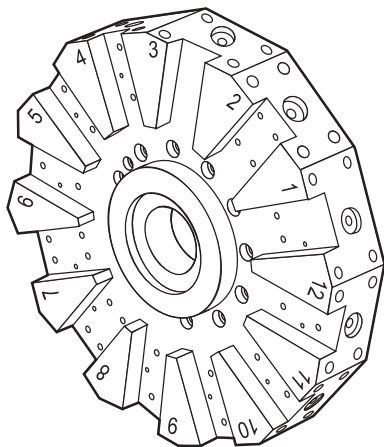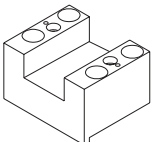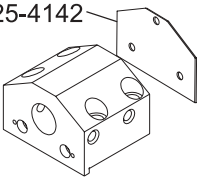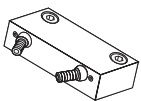
SL-10
12 Station Turret

Basic BOT ID Tool Specifications

.250"   1.000"

BOLT ON TOOLS PARTS LIST

Face Grooving	Part No.	Size	Kits
	BOT10F/G BOT10MF/G	3/4" 20mm	BOTKIT10 (Option) 2ea. BOT10ID-1_1/4 2ea. BOT10ID-1 1ea. CB 10 9ea Turret Face Wedge/Clamps
Boring Bar 25-4142	BOT10ID-1 BOT10ID-1_1/4 BOT10MID-32	1" 1 1/4" 32mm	BOTKIT10M (metric option) 4ea. BOT10MID-32 1ea CB10M 9ea. Turret face Wedge/Clamps
Cooling Block	CB10 CB10M	Inch Metric	SI-10 Standard tooling includes: 1 BOT10 ID-1 1 BOT10 ID-1-1/4 1 BOT 10 F/G 3 BOT 10 OD/Clamp
Turret Face Wedge Clamps	BOT 10OD/clamp BOT 10MOD /clamp	INCH METRIC	
Nozzles	20-0562 90° 20-0689 Str long 20-2077 Plug 20-8730 Str 20-8739 45°		

Axis Travel Diagram
SL-10 BOT 12 Pocket

4.96" (126mm)
14.00" (355.60mm)
2.12" (53.85mm)
1.00" (25.4mm)
0.25" (6.35mm)
0.25" (6.35mm)
1.25" (31.75mm)
5.55" (141mm)
0.25" (6.35mm)
6.00" (152.40mm)
.45" (11.43mm)
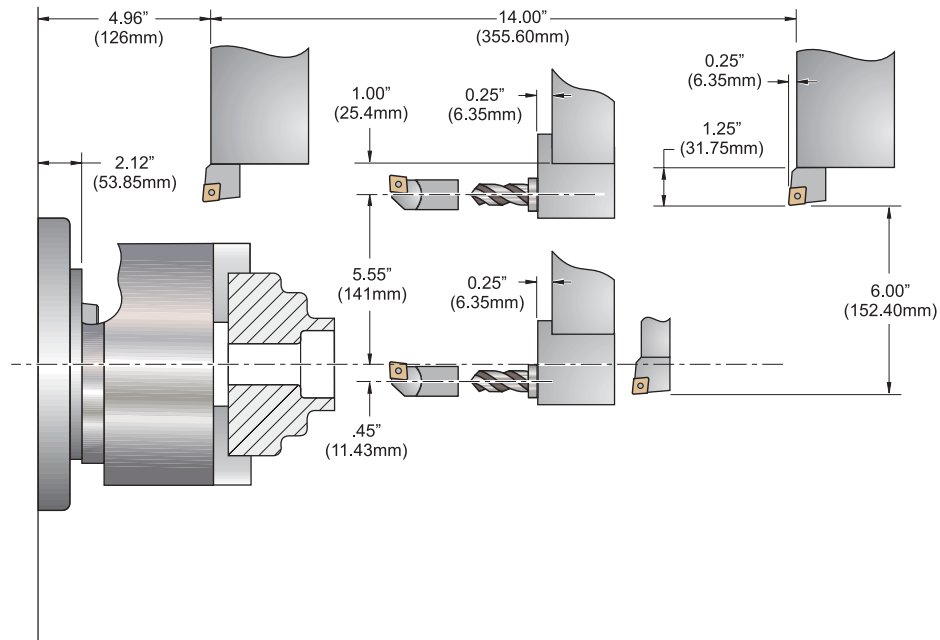
Turret Tooling Clearance
SL-10 12 Pocket

9.26" (235.20mm)
2.47" (62.74mm)
Ø1.25" (31.75mm)
Ø15.28" (388.11mm)
Ø6.80" (172.72mm)
.75" (20mm metric option)
3.95" (100.33mm)
Ø11.00 (279.40mm)

Axis Travel Diagram
SL-20 BOT

5.90"
(149.86mm)

20.00"
(508mm)

0.25
(6.35mm)

0.25
(6.35mm)

1.25
(31.75mm)

1.49"
(37.85mm)

0.25
(6.35mm)

5.80"
(147.32mm)

0.25
(6.35mm)

2.65
(67.31mm)
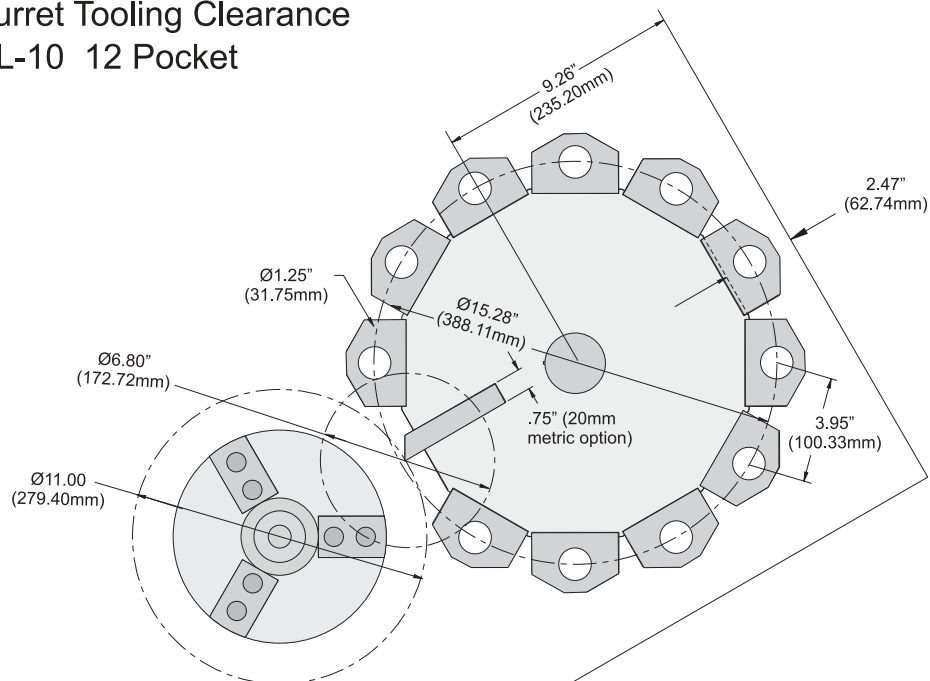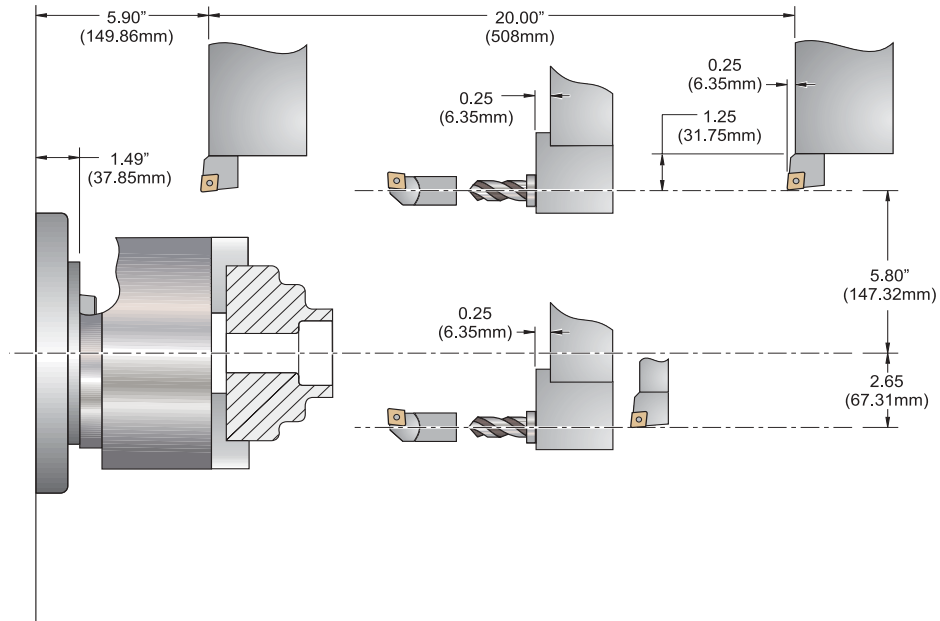
Turret Tooling Clearance
SL-20 10 Pocket

*Note: Obstructing coolant nozzles
must be removed
to achieve this diameter.

*Ø8.44" (214.38mm)
Maximum Part Diameter
With Tool at
Centerline of Spindle
and Tools Mounted
on Each side.

Before July 2002
1.86
(47mm)

Ø10.28" (261.11mm)
Maximum Part Diameter
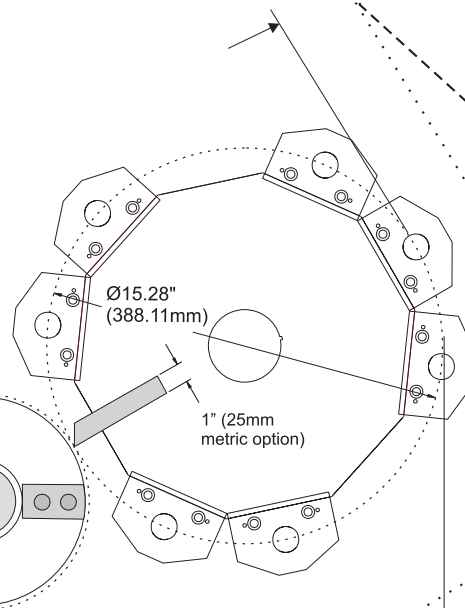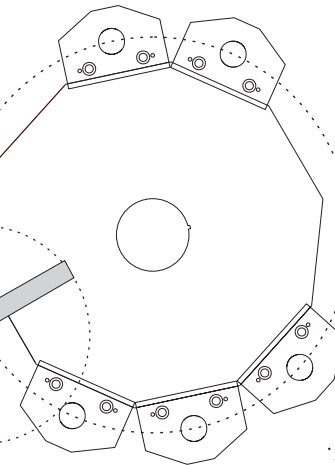
Ø15.28"
(388.11mm)

1" (25mm
metric option)

Manufactured Before July 2002

Manufactured July 2002 and After

July 2002 and After
3.86
(98mm)

Axis Travel Diagram
SL-30 BOT
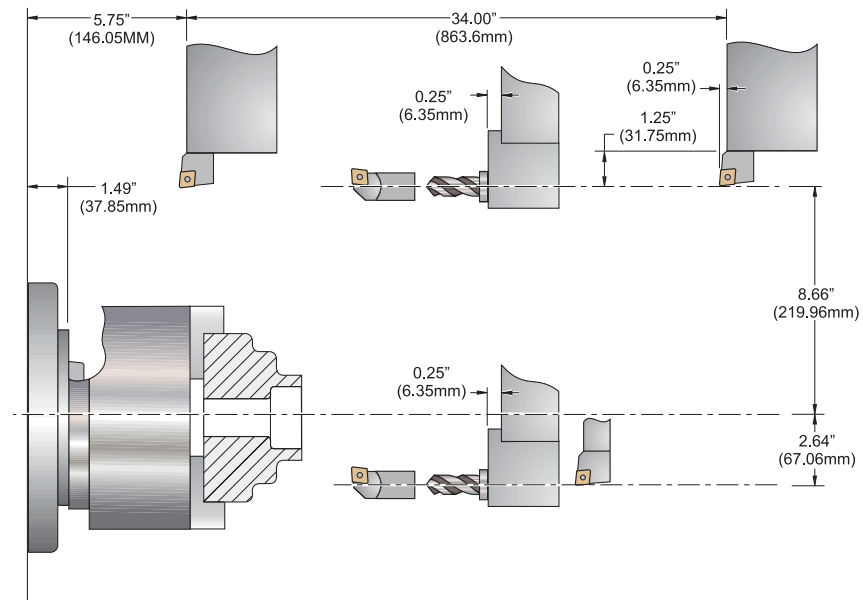
Turret Tooling Clearance
SL-30 12 Pocket

*Note: Obstructing coolant nozzles
must be removed
to achieve this diameter.

*Ø8.61 (218.69mm)
Maximum Part Diameter
With Tool at Centerline
of Spindle
and Tools mounted
on each side.
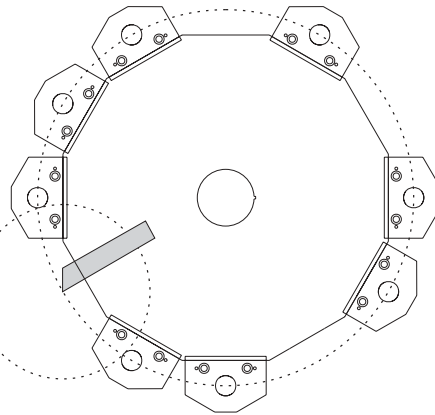
Before July 2002
1.86
(47mm)

Ø17.0 (431.8mm)
Maximum Part Diameter
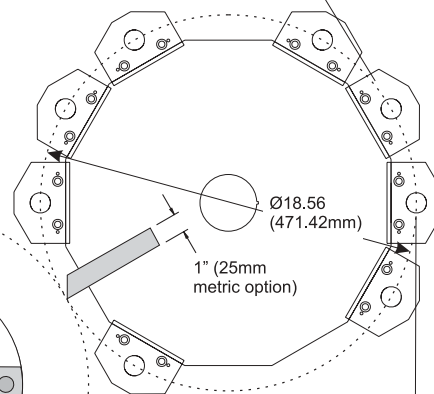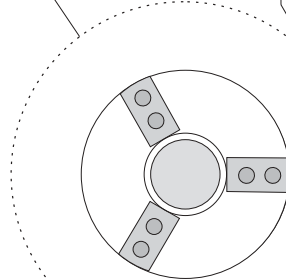
Ø18.56
(471.42mm)

1" (25mm
metric option)

Before July 2002

July 2002 and After

July 2002 and After
3.86
(98mm)

Axis Travel Diagram
SL-40 BOT

10.44"
(265.18mm)

44.00"
(1117.60mm)

0.25"
(6.35mm)

0.25"
(6.35mm)

1.58"
(40.13mm)

1.40"
(35.56mm)

15.29"
(388.37mm)

1.35"
(34.29mm)
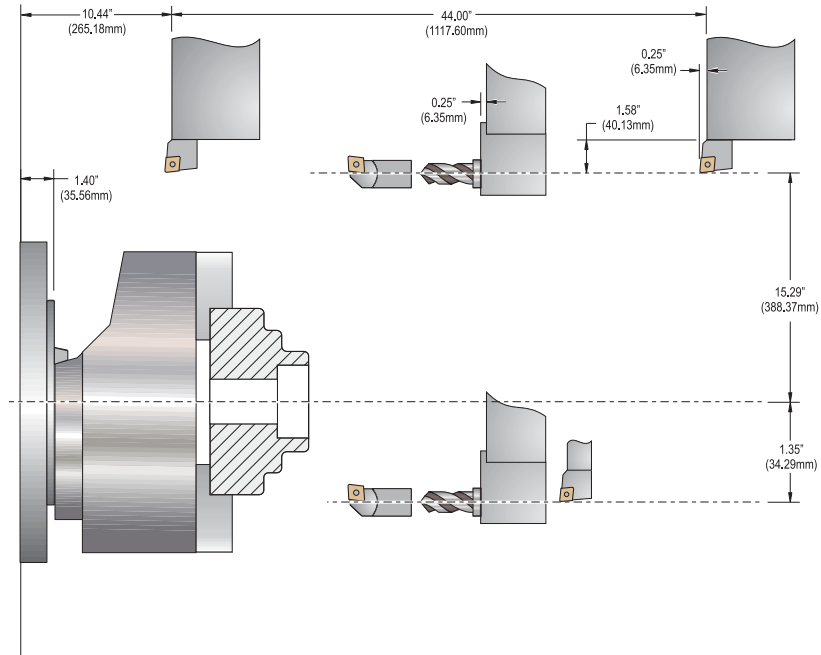
Turret Tooling Clearance
SL-40 10 Pocket

2.23" Pocket Depth
(56.64mm)
(Live tool option only)

15.74"
(400mm)

Ø2.50"
(63.50mm)

Ø21.42"
(544.07.mm)

Ø10.74"
(426mm)

Ø25.5"
(647.1mm)

1.25" (32mm
metric option)
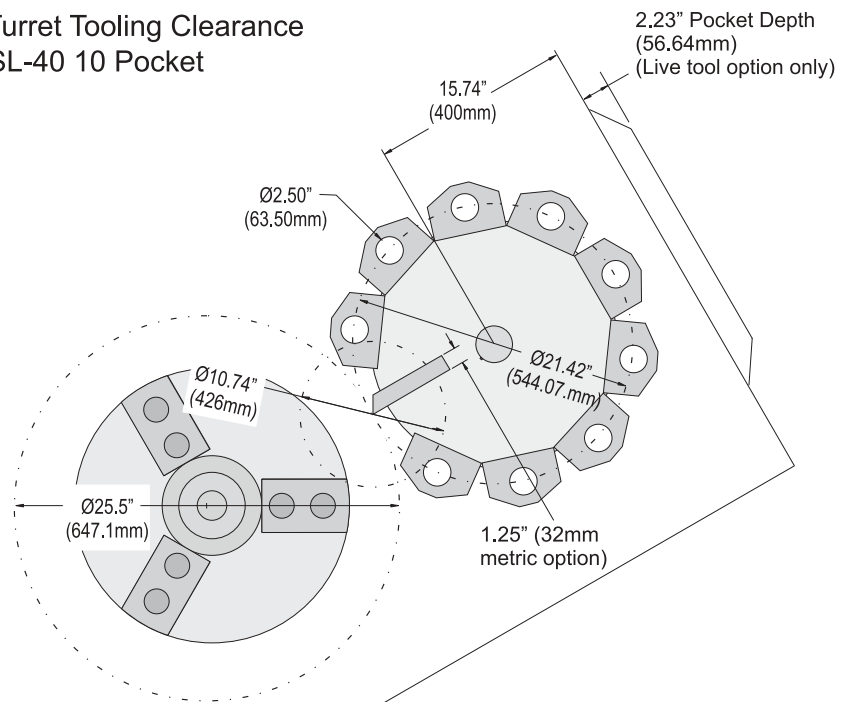
3.16 VDI30 TURRET

Axis Travel Diagram
SL-10 VDI-30 12 Pocket (6 I.D., 6 O.D.)

3.30
(8.4 cm)

14.00 (35.6cm)

1.57
(4 cm)

.25 (.6 cm)

2.36 (6 cm)

2.12
(5.4 cm)

.80 (2 cm)

1.33 (3.4 cm)

5.80
(14.7 cm)

5.13
(13 cm)

6.25
(9.5 cm)

.45 (1.1 cm)

.366 (.93 cm)

∅17.96 (45.6 cm)
Turret Diameter

∅10.26 (26.1 cm)
Max Cutting
Diameter

6.00 (15.2 cm)

7.64 (19.4 cm)

∗∗∅15.28
(38.8 cm)

∗∅12.00
(30.5 cm)

∗ O.D. Turning tools only

∗∗ I.D. Drill/Bore tools only
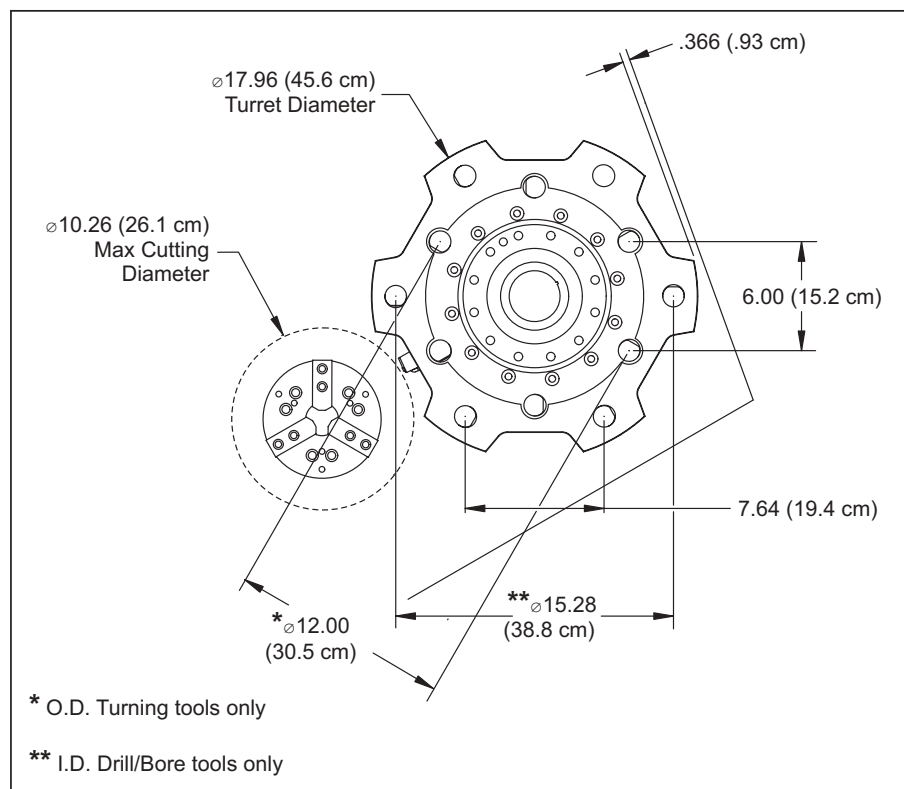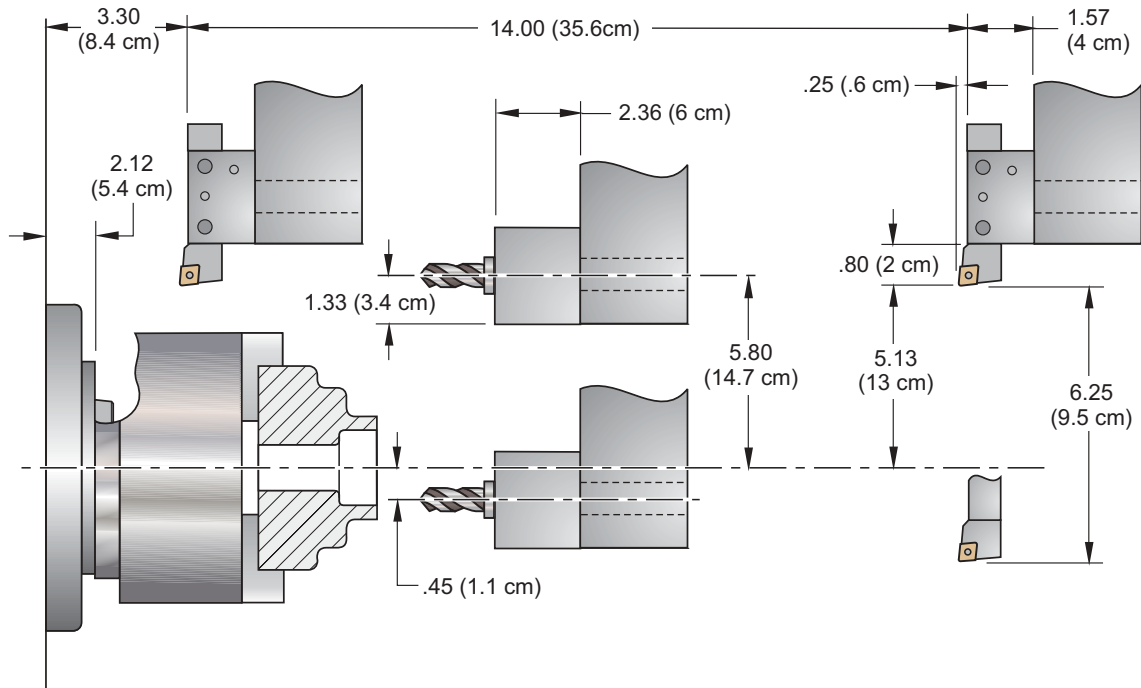
3.17 BAR FEEDER WITH THE HAAS LATHE

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:

```
M21 _____|‾‾‾‾‾‾‾‾‾‾‾‾‾|_____

M-FIN
Discrete                    _____
Input 1009 _____|              |_____

CNC is:    Running    | Waiting  |.05 ms delay| Waiting  | Running
                        for M-fin           for end M-fin

Barfeed:   Waiting    |      Running       |      Waiting
```
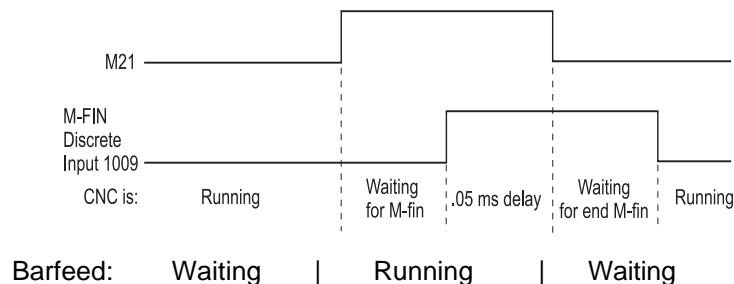
The high state in this diagram indicates an active circuit or closed circuit.

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.

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.

3.18 LIVE TOOLING*

*This option is not field installable
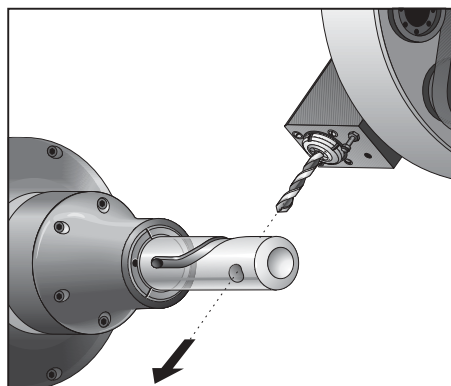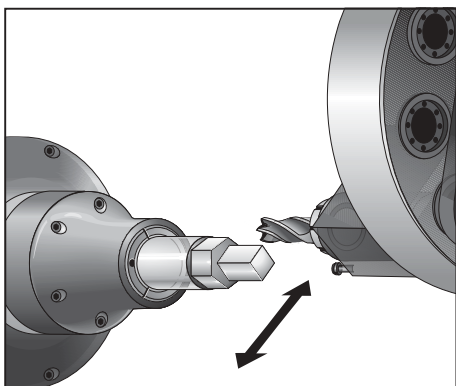
The live tooling option allows the user to drive VDI axial or radial tools to perform such secondary operations as milling, drilling or slotting. The main spindle of the lathe is indexable in one degree increments for precise part positioning and repeatability. A hydraulic brake locks the spindle during 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.

Notes: 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.

LIVE TOOLING 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.

Parameter 298   Y AX 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.

LIVE TOOLING M CODES

M19 ANGLE CMD (Optional)

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.

> Degree of accuracy
> > P rounds to the nearest whole degree.
> > R round to the nearest hundredth of a degree (x.xx)

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.

CURRENT COMMANDS DISPLAYS

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

PROGRAM EXAMPLE:

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

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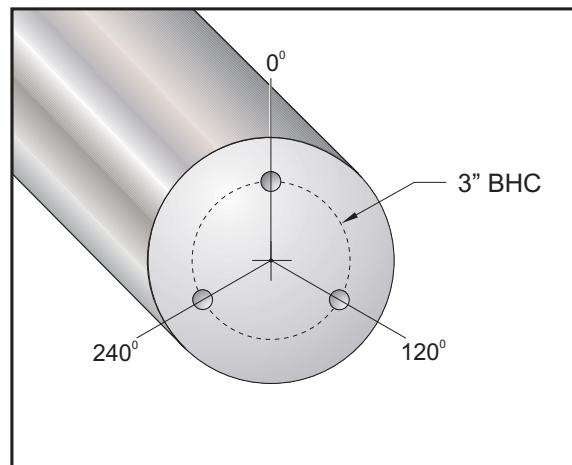
SYNCHRONOUS MILLING

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

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 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 simplify the creation of simple geometric shapes. G77 is a general flatting algorithm that automates the motions of 1, 3 and more sided uniform shapes. In 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.
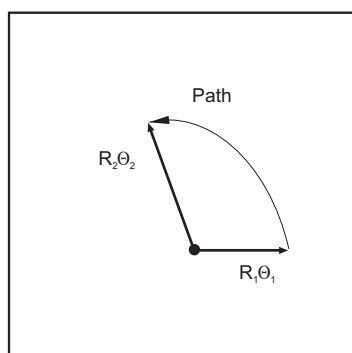
G32 paths between commanded points are curves
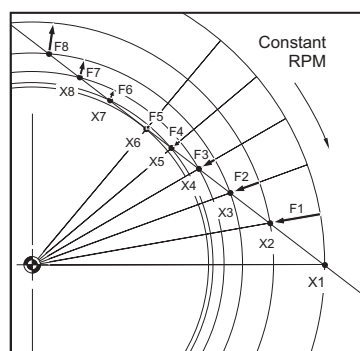
G32 motion includes both X feed rate and position commands at a constant RPM.
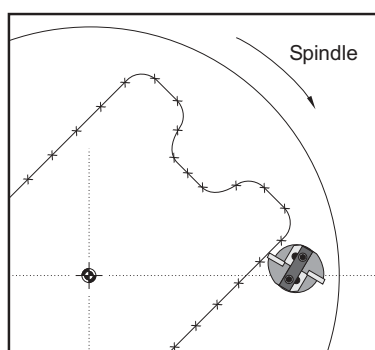
Using G32 many small motion commands can result in geometric shapes

FINE SPINDLE CONTROL CODES AND LIVE TOOLING G-CODES

Introduction

Many uses of live tooling involve holding the spindle still while performing a cut with the live tool. For certain types of operations, 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.
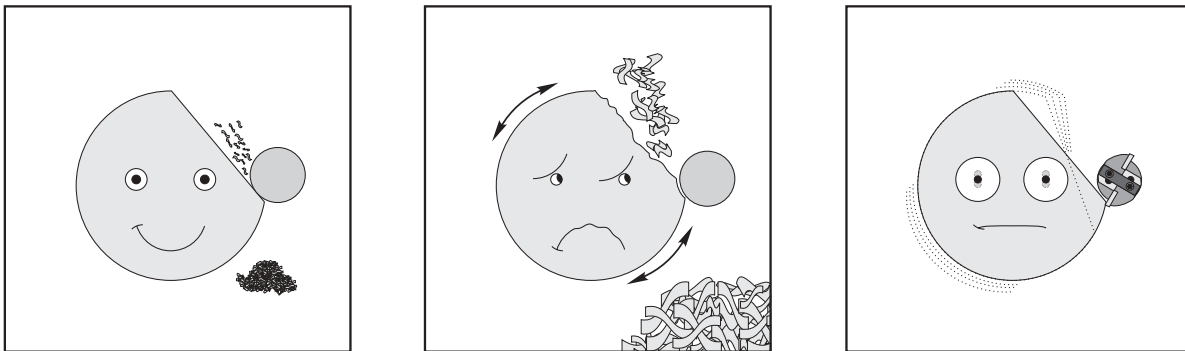
In many cases you will want to "track" the motion of the spindle with motion in the X axis. The spindle was designed to turn rapidly, rather than precisely. Because of this, the accuracy with which the position of the spindle is known is .045 degrees. This limit also applies to positioning the spindle 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.

Minimum Controllable Angle

Path A
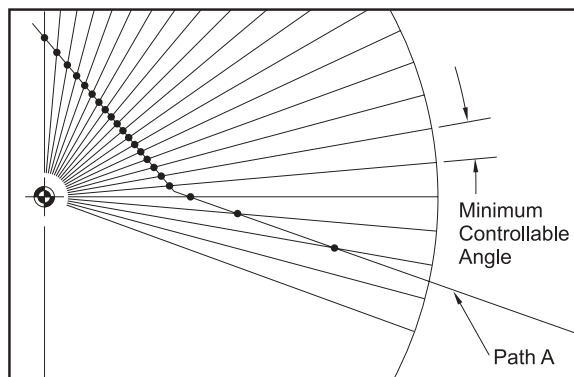
RADIAL AND AXIAL TOOLING INSTALLATION

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.
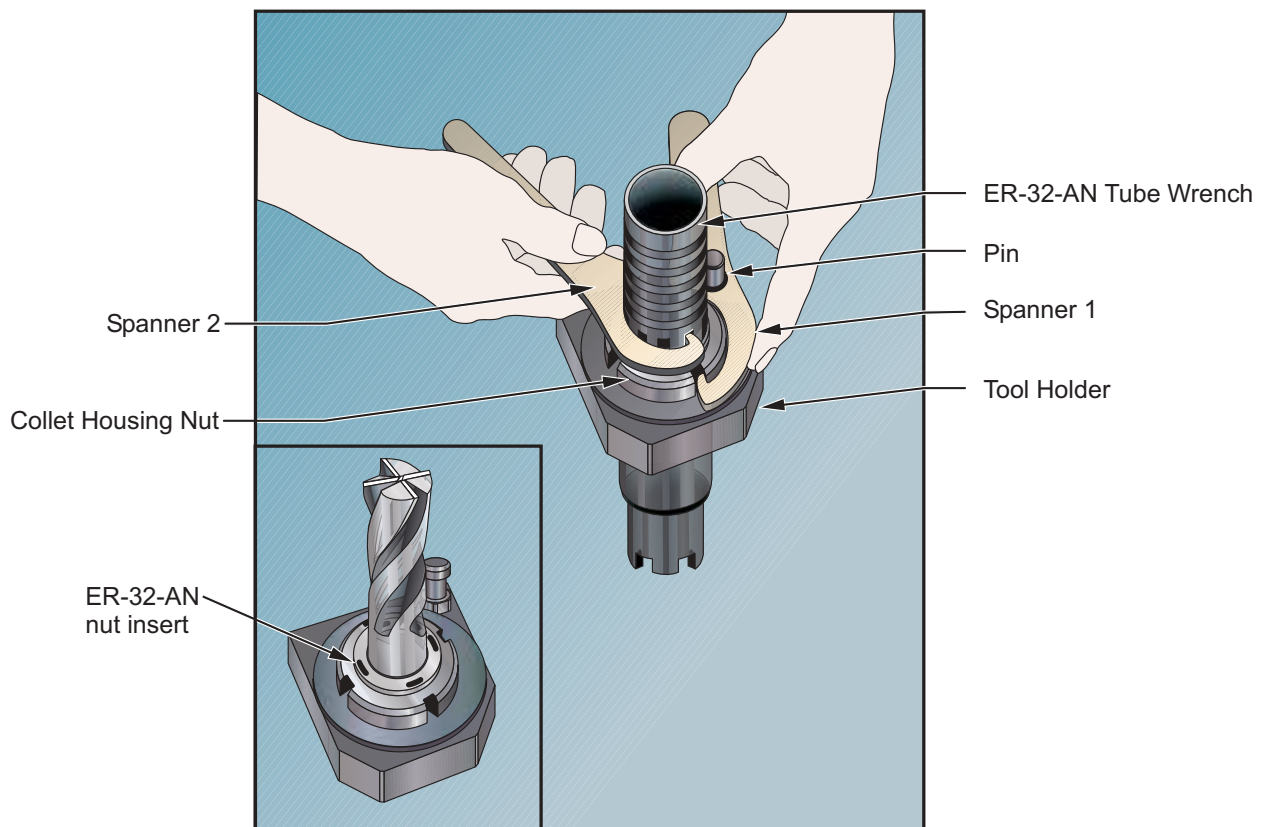
Live Tooling Tool Holders and Torque Chart

AXIAL TOOLHOLDER   RADIAL TOOLHOLDER

	in/(mm)	HAAS TURRET * SL-20/30 TCE-550/550T/860/860T	VDI TURRET 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 (130.25) P/N 59-6352	4.628 (117.55) P/N 59-0082
RADIAL TOOLHOLDER	X-DIM P/N	5.128 (130.25) P/N 59-6353	4.628 (117.55) P/N 59-0084

* VDI ADAPTER KIT (HAAS P/N VDIA) REQUIRED.

SPINDLE DRIVE PERFORMANCE

TORQUE (IN-LBS): 132, 72, 48, 24 — PEAK TORQUE, CONTINUOUS — RPM: 1500, 2700, 3000
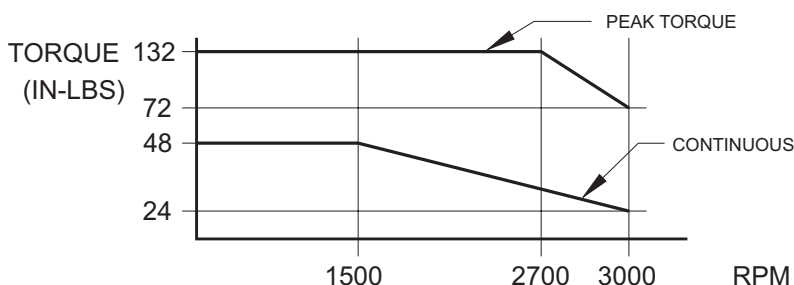
- Haas Live tooling is designed for medium duty milling; e.g. 3/4" diameter end mill in mild steel maximum.
- Large tool diameters may require reduction tool holders.
- Tool Length should be kept to a minimum for toolholder rigidity.
- Typical life of toolholder bearing is 1000 hours with medium duty operation.
- Worn toolholder brearings 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		TURRET TYPE		TURRET TYPE	
		HAAS	VDI	HAAS	VDI	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	
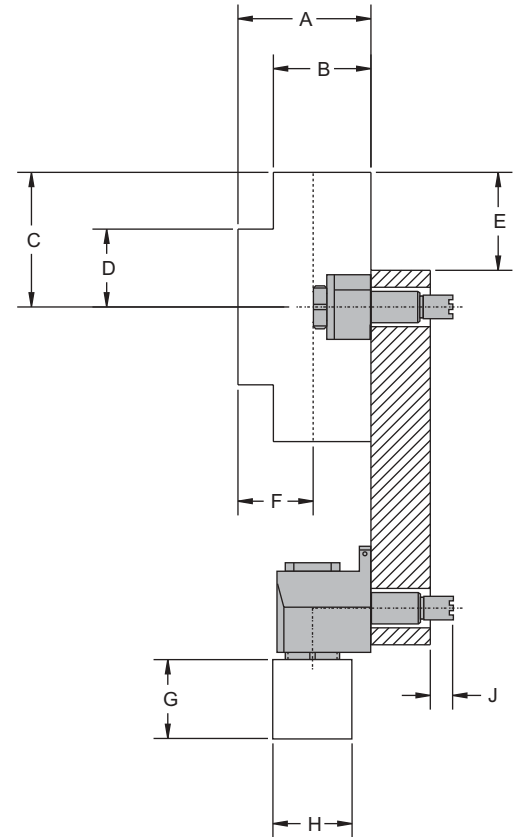
* LIVETOOLING IN HAAS TURRETS REQUIRES 1/2 INCH ADAPTER PLATE.
  (REF: ES-0122 VDI-40 TO HAAS ADAPTER KIT. HAAS P/N: VDIA)

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.

V*DI* A*PAPTER* I*NSTALLATION*

IMPORTANT!
VDI ADAPTERS MAKE IT POSSIBLE TO USE VDI-40 TOOLS IN HAAS TURRETS

VDI ADAPTER EXPLODED VIEW
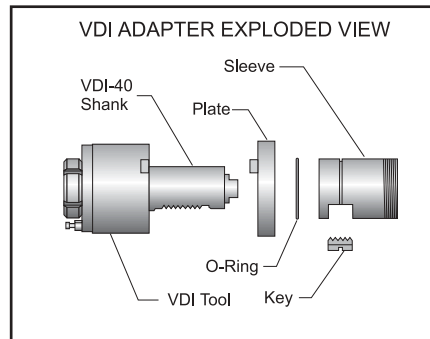
INSTALLATION PROCEDURE:

1. Install plate over VDI-40 tool shank. Orient plate boss to VDI tool counterbore.

VDI ADAPTER INSTALLATION
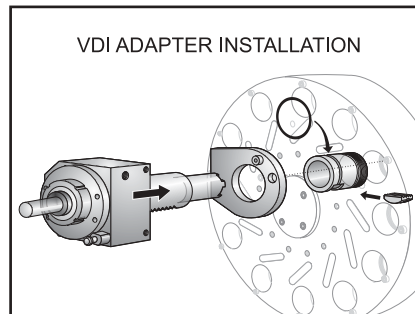
2. Slide adapter sleeve onto tool shank with cut-out facing towards base of tool shank. Align cut-out with tooth profile of shank.

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.

VDI WITH DOWEL PIN
AND DIAL INDICATOR
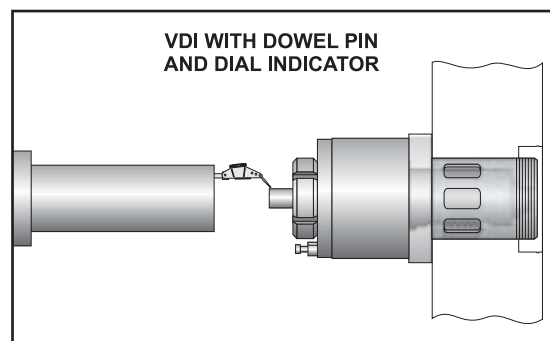
6. Tighten the draw nut to lock assembly in place.

HAAS SL LIVE TOOLING CUTTING CAPACITY

90° SPOT DRILL

Material: 86L20 (mild steel)

F .004/REV
F .006/REV

LOAD — DEPTH

STUB DRILL

Material: 86L20 (mild steel)

F.004/REV
F.006/REV

LOAD — DIAMETER

END MILL DEPTH OF CUT TO STALL

Material: 86L20 (mild steel)

3/4 4FL CARBIDE EM
5/8 4FL CARBIDE EM

DEPTH — SFPM @ 0.01/REV
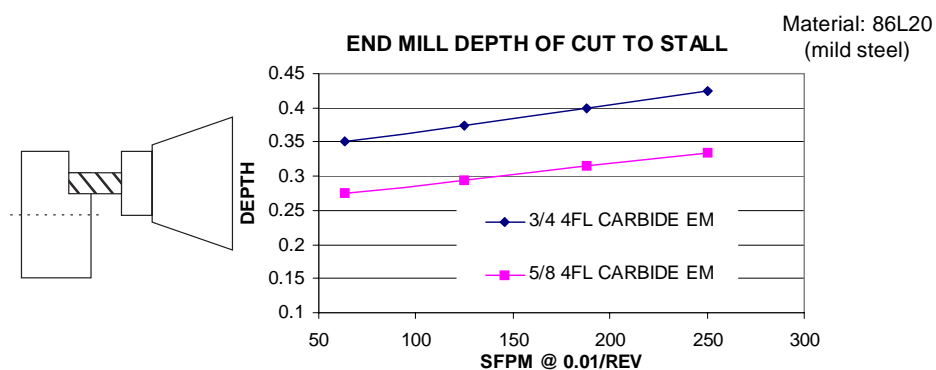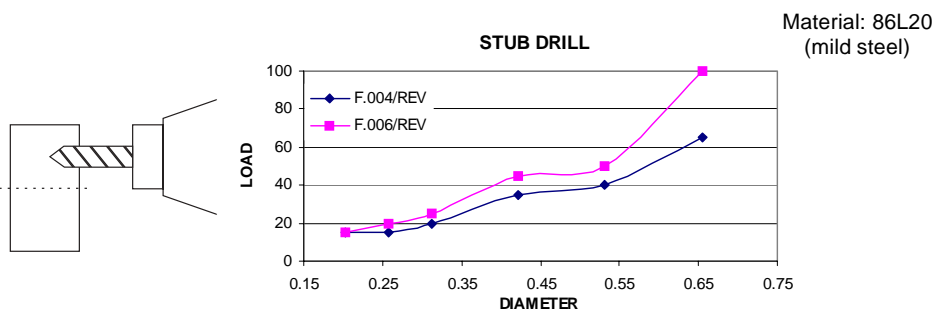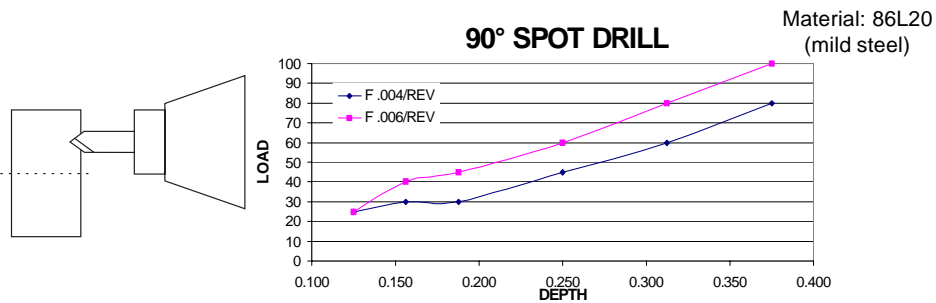
Driven Axial And Radial Tool Holder Warranty Conditions

Each driven tool is warrantied for 1000 (one thousand) hours or 6 (six) months, whichever comes first. Call Exsys directly for warranty claims, service and repairs at: 1-800-EXSYS-4U or www.exsys-tool.com All other live tooling option issues should be handled by an authorized Haas distributor.

Limits and Exclusions

Factory-specified maintenance procedures must be adhered to and recorded in order to maintain this warranty.

This warranty is void if the unit is subjected to misuse, neglect, accident, improper installation or application, or if the unit was improperly repaired by the customer or by an unauthorized service technician. Warranty and general service is available from Exsys/Eppinger GmbH.

Haas Automation and Exsys Tool are not responsible for any additional or incidental damage to parts, fixtures, machines, loss of time, or any other incidental damages that may be caused by a malfunction.

This warranty is transferrable from the original end-user to another party if the machine is sold via private sale before the end of the warranty period.

NOTE: Should you have a problem with your machine, please review the instructions in the owner's manual first. If this does not resolve the problem, call your authorized Haas distributor. As a final solution, call Exsys at the number given previously, or Haas Automation directly. In Europe, call Eppinger GmbH, Telephone 011 49 711 934 934 0.

3.19 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

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.

GENERAL WARNINGS

• The High Pressure Coolant pump is a precision gear pump and will wear out faster and lose pressure if abrasive particles are present in the coolant.

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

• 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 for recommendations.

• Machining of ceramics and the like voids all warranty claims for wear and is done entirely at customer's risk. Increased maintenance schedules are absolutely required with abrasive swarf. The coolant must be changed more often and the tank thoroughly cleaned of sediment on the bottom. An auxiliary coolant filter is recommended.

    • Coolant will be used more quickly when the High Pressure Coolant system is in use. Make sure to keep the coolant level up and to check the level more frequently (check after every eight hour shift). **Premature wear of the pump can result from running with a low coolant level in the tank**. The spindle will shut off automatically if the coolant pressure gets too low.

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.
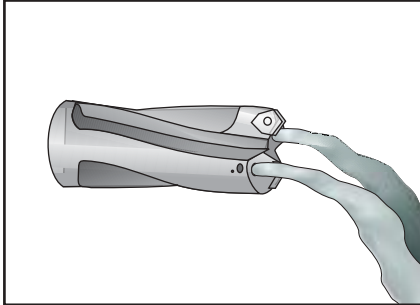
Larger tooling has larger diameter coolant passages. Coolant flow is higher at lower pressures.
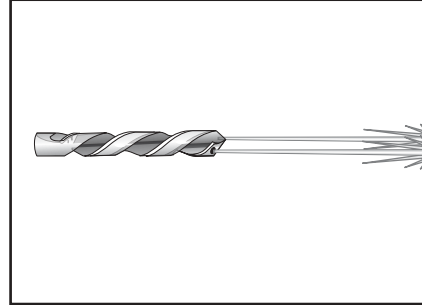
Smaller tooling has smaller diameter coolant passages. This produces higher pressures at lower flow.

TSC, HPC Pump Output
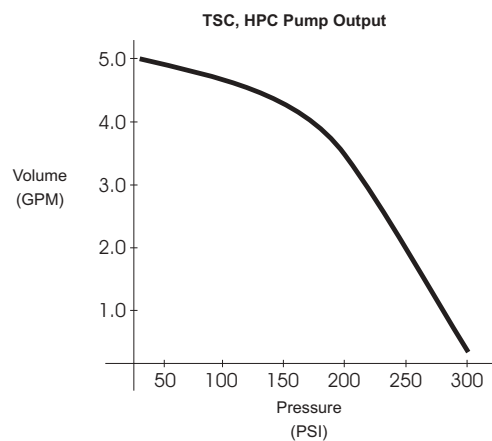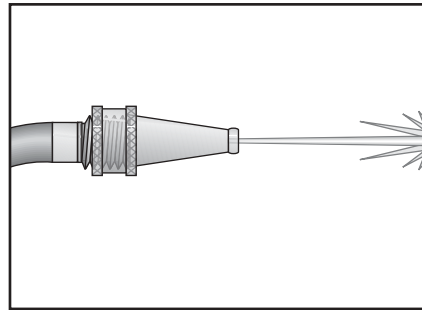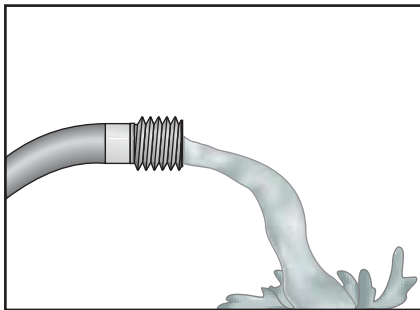
3.20 AUTO AIR JET*

*This option is not field installable
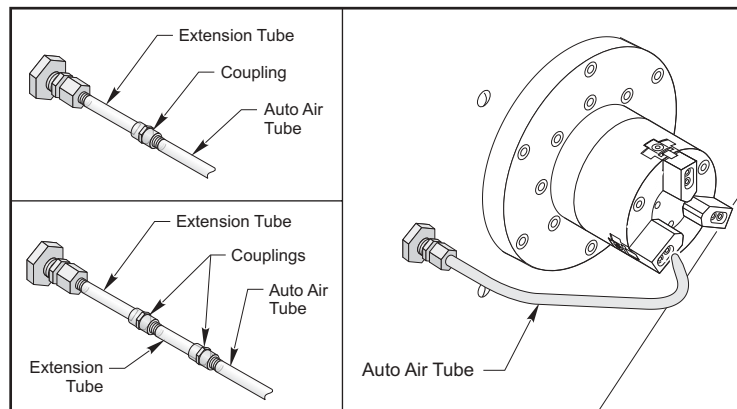
The Auto Air Jet can be programmed to blow air at the chuck in order to clear chips and coolant between operations.

OPERATION

M12 turns on the Auto Air Jet.
M13 turns off the Auto Air Jet.

3.21 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 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

3.22 C-AXIS

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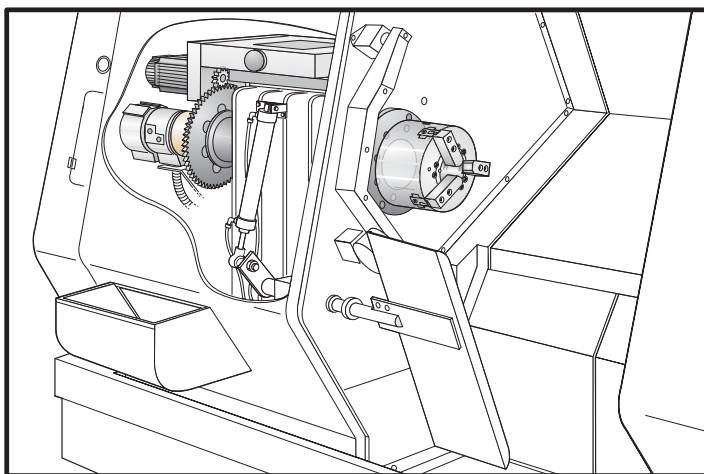
OPERATION

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)

Spindle brake automatically engaged/disengaged with C-axis motion

The lathe will automatically disengage the spindle brake when the C-axis is commanded to move and to reengage it afterwards (if it has previously been engaged.) Note that in order for this feature to operate properly, Parameter 498 NO I IN BRAK must be set to 1.

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.)

SAMPLE PROGRAMS

Example #1
1. M154
2. G00 G98 (feed/min) X2.0 Z0.5
3. C90
4. G01 Z–0.1 F6.0
5. X1.0
6. C180. F10.0
7. X2.0
8. G00 Z0.5
9. M155
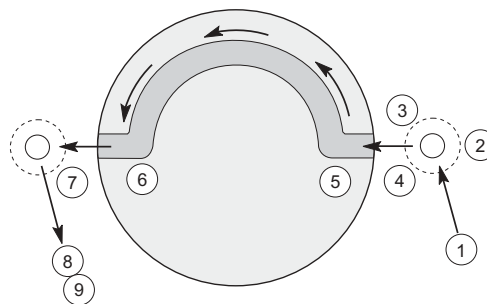
Example #1

N1 M154
N2 G00 Z2.
N3 C0.
N4 G00 X1.
N5 M133 P1500
N6 G98 G01 Z–0.25 F10.   (Plunge into pre drilled hole)
N7 C90. F40.   (Make slot)
N8 G01 Z0.5 F10.   (Retract)
N9 M135
N10 G99 G28 U0. W0.
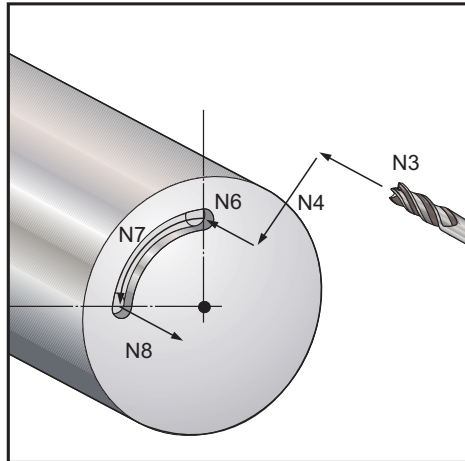
3.23 CARTESIAN TO POLAR TRANSFORMATION

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 reference 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.

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
%
```
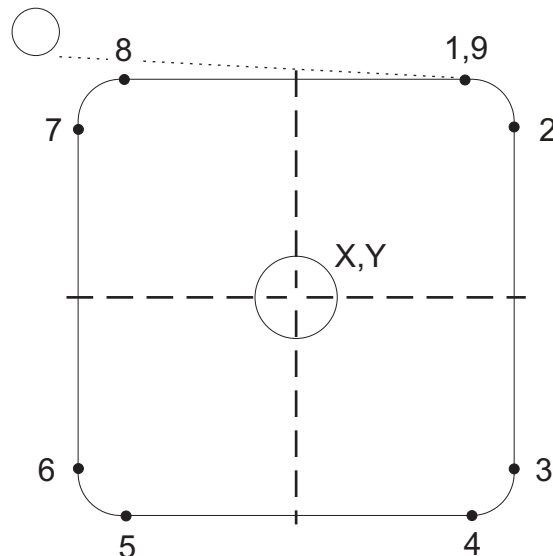
3.24 8 "M" FUNCTIONS

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.
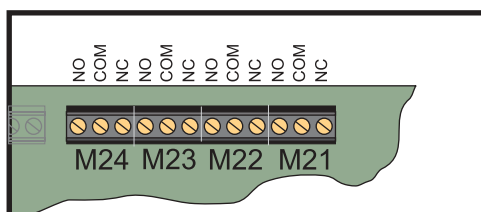
8M Relay Board

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 Connectiors:

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 IOPCB (M21- M28) (540) (INPUT)
P3 Unused
P2 12 VDC From Power Supply Board (860A)
P6 16-Pin Relay Output To 2nd 8M Relay Board
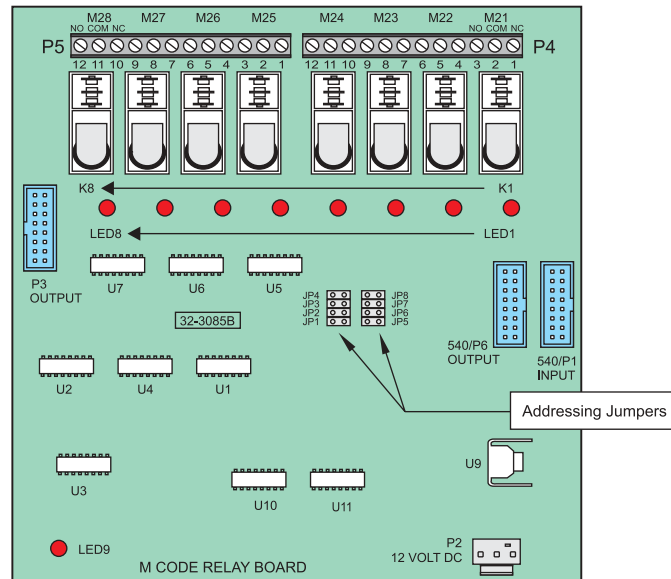
M-Code relay board.

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.

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 3.25-1, 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 3.25-2, 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 3.25-3.
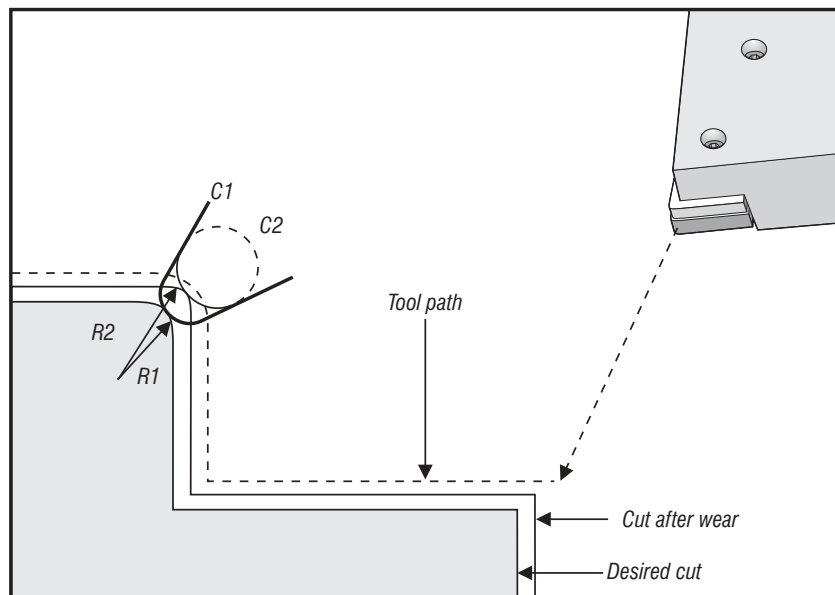
Figure 3.25-1. Tool path for 2 cutter radiuses.
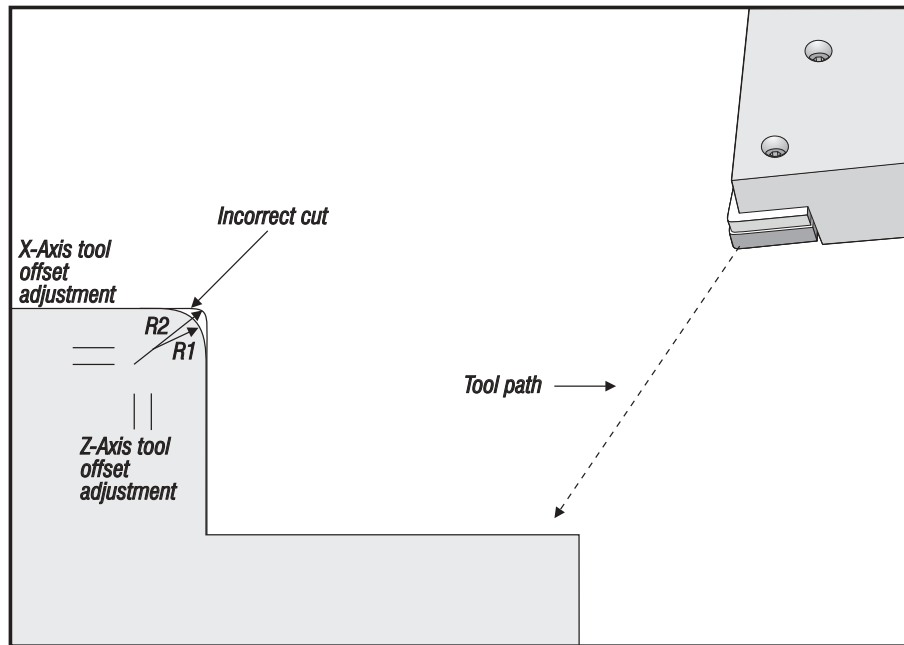
Figure 3.25-2. Two cuts overlaid to show cutting error.
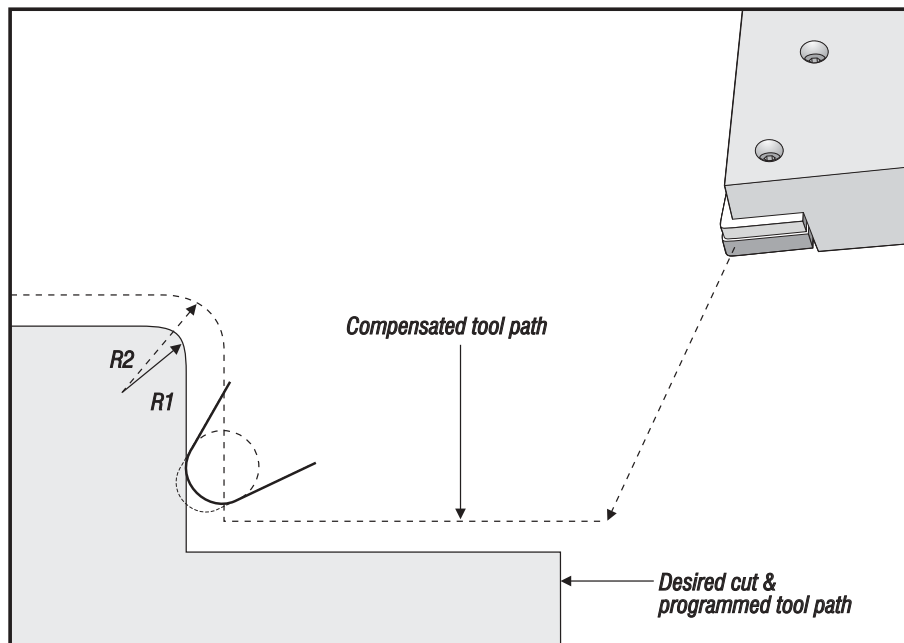
Figure 3.25-3 Path generated when tool nose compensation is used.

Note that in Figure 3.25-3 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.
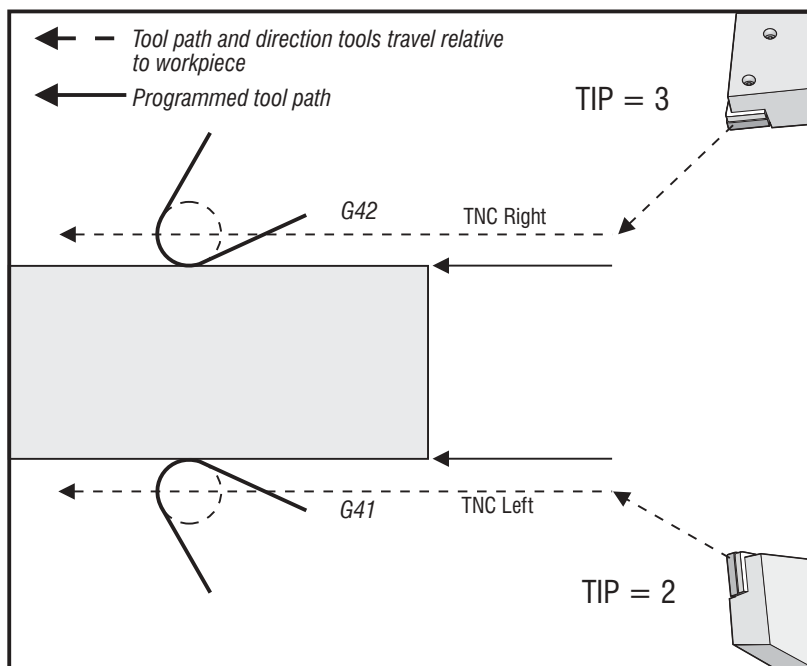
Figure 3.25-4. Shift direction.

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.

TIP DIRECTION IS 3

Tool nose radius
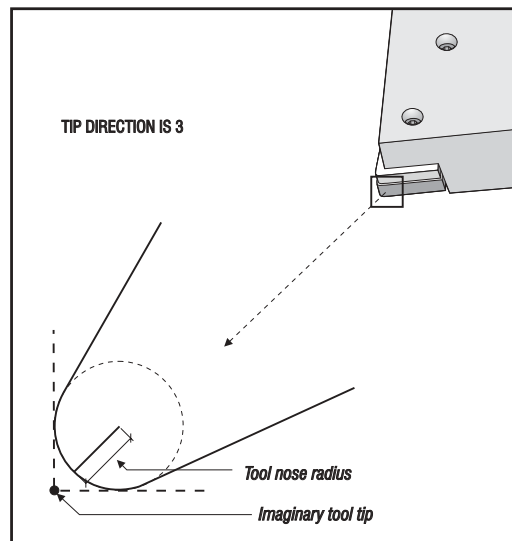
Imaginary tool tip

Figure 3.25-5. 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 3.25-5 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 PRO-CESSING.

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 3.25-6
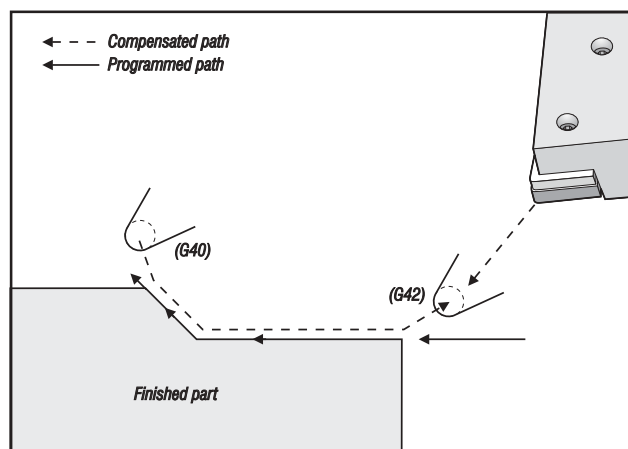
Figure 3.25-6. Approach and Departure moves.

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 3.25-6.

Figure 3.25-7 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 3.25-7 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.
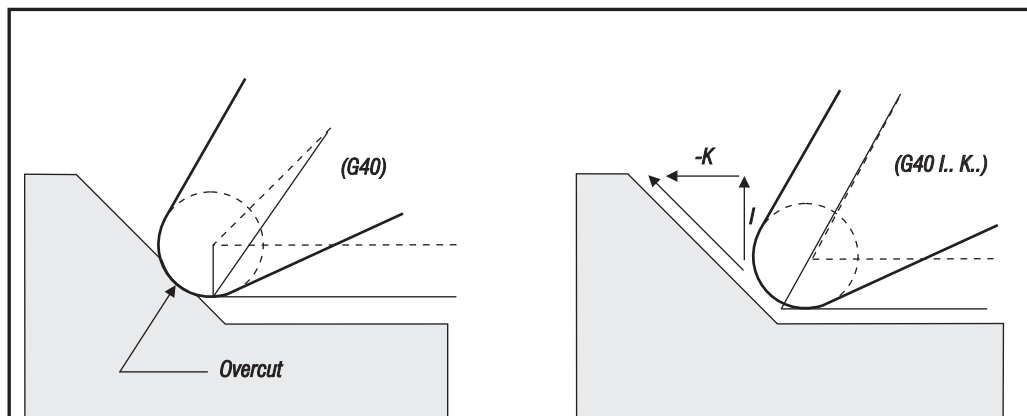
Figure 3.25-7. Use of I and K in a G40 block.

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 ;
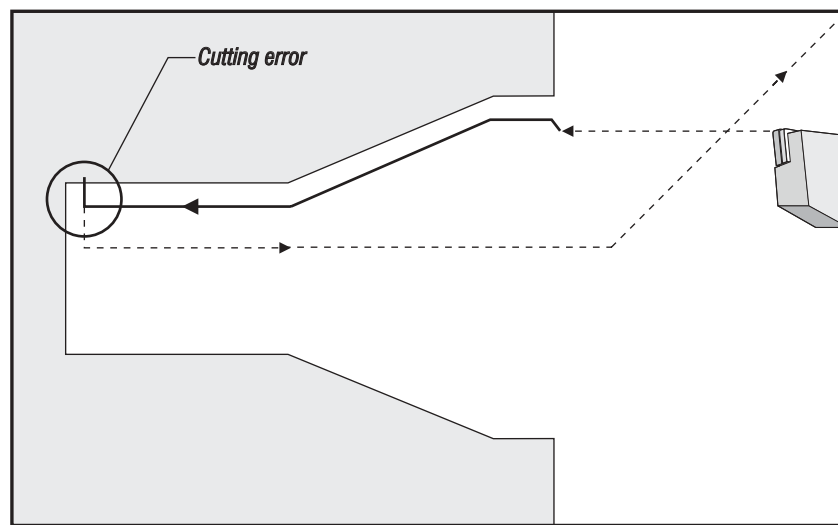
Figure 3.25-8. An invalid program using TNC and G70.

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 3.25-5.
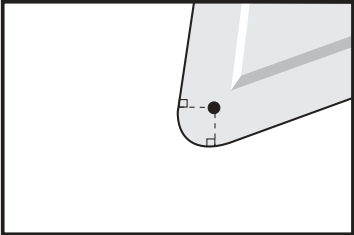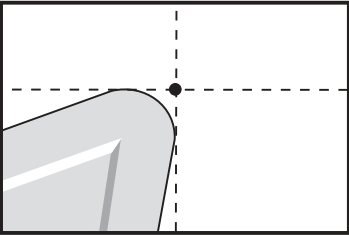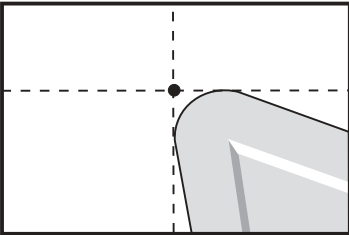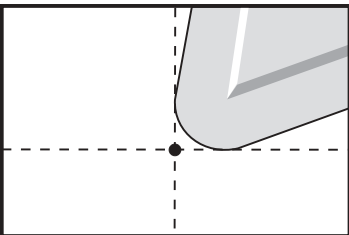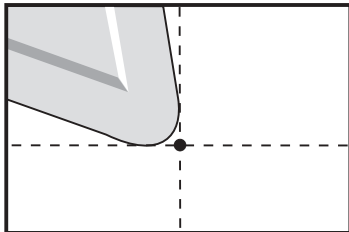
The tool tip direction of each tool is coded as a single integer from 0 to 9. The tip direction code is found next to the radius offset on the geometry offsets page. It is recommended that a tip direction be specified for all tools using tool nose compensation. Figure 3.25-9 is a summary of the tip coding 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
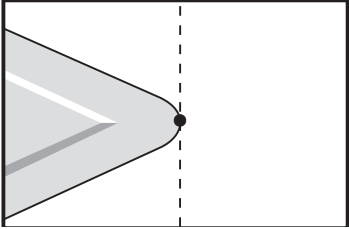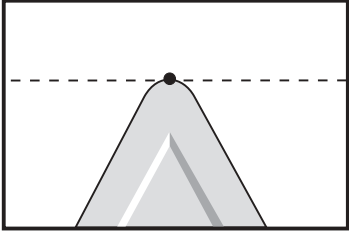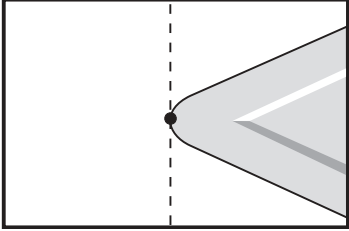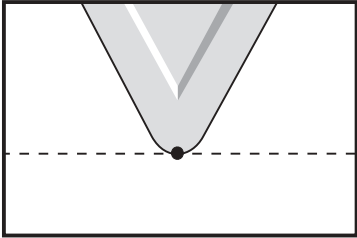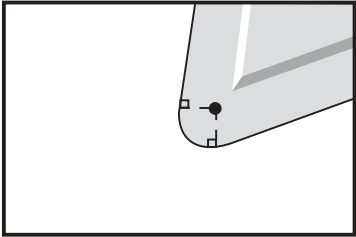
The length geometries of tools that use tool nose compensation are set up in the same manner as tools not using compensation. Refer to the "Setup Procedures" section of this manual for details on touching off tools and recording tool length geometries. When a new tool is set up, the geometry wear should 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 SPECI FIED DIRECTION. IT IS USU ALLY 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 3.25-9a. 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 3.25-9b. Imaginary tool tip.

TOOL NOSE COMPENSATION GEOMETRY

Figure 3.25-10 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**

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.

LINEAR TO LINEAR - TYPE A

ANGLE: <90

	APPROACH	MODE TO MODE	DEPARTURE
G41			
G42			

ANGLE: >=90, <180

	APPROACH	MODE TO MODE	DEPARTURE
G41			
G42			
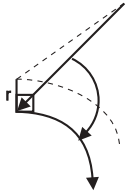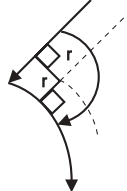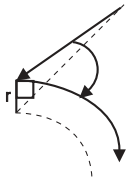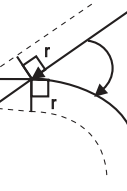
ANGLE: >180

	APPROACH	MODE TO MODE	DEPARTURE
G41			
G42			
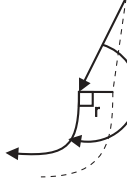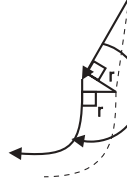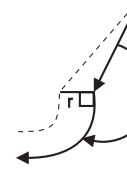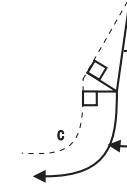
Figures 3.25-10a shows Type A compensation.

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 3.25-10b 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
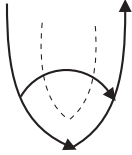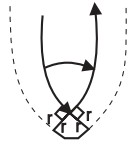
	APPROACH	MODE TO MODE	DEPARTURE
G41	NOT PERMITTED		
G42	NOT PERMITTED		

ANGLE: >180
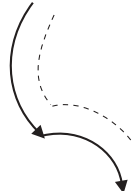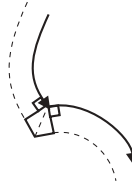
	APPROACH	MODE TO MODE	DEPARTURE
G41	NOT PERMITTED		
G42	NOT PERMITTED		

Figures 3.25-10c 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 3.25-10d shows Type A 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.
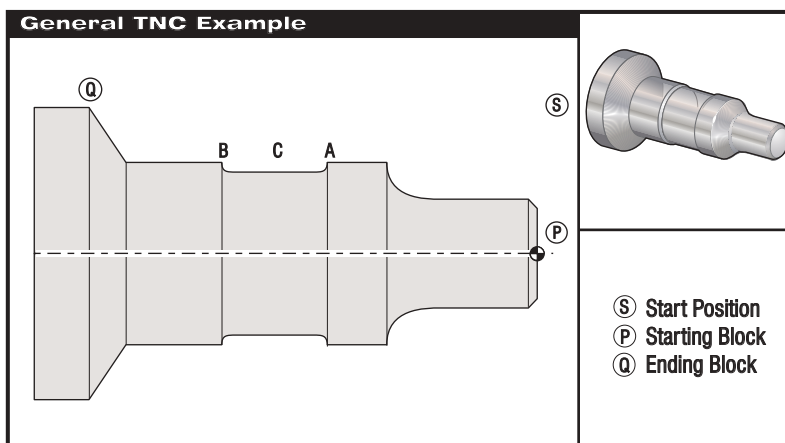
Figure 3.25-11. General TNC

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   DESCRIPTION
```
%
O0811 (G42 TEST BCA)                        (EXAMPLE1)
N1 G54 S1000
T101                                        (SELECT TOOL 1 AND OFFSET 1
                                            (TIP DIRECTION FOR OFFSET 1 IS 3)

G97 S500 M03
G54 G00 X2.1 Z0.1                           (MOVE TO POINT S)
G96 S200
```

```
(ROUGH P TO Q WITH T1 USING G71 AND TNC)
G71 P10 Q20 U0.02 W0.005 D.1 F0.015

(DEFINE PART PATH PQ SEQUENCE)
N10 G42 G00 X0. Z0.1 F.01                    (P)(G71 TYPE II, TNC RIGHT)
G01 Z0 F.005
X0.65
X0.75 Z−0.05
Z−0.75
G02 X1.25 Z−1. R0.25
G01 Z−1.5                                     (A)
G02 X1. Z−1.625 R0.125
G01 Z−2.5
G02 X1.25 Z−2.625 R0.125                      (B)
G01 Z−3.5
X2. Z−3.75
N20 G00 G40 X2.1                             (TNC CANCEL)
G97 S500
G28                                          (ZERO FOR TOOL CHANGE
                                             CLEARANCE)

M01
N2 G50 S1000
T202
G97 S750 M03                                 (SELECT TOOL 2 AND OFFSET 2)
                                             (TIP DIRECTION FOR OFFSET 2 IS 3)
G00 X2.1 Z0.1                                (MOVE TO POINT S)
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)
G54 G42 X1.5 Z−2.0                           (MOVE TO POINT C)(TNC RIGHT)
G96 S200
G01 X1. F0.003
G01 Z−2.5
G02 X1.25 Z−2.625 R0.125                     (B)
G40 G01 X1.5                                 (TNC CANCEL)
(GROOVE TO POINT A USING OFFSET 4)
T313                                         (CHANGE OFFSET TO OTHER SIDE
                                             OF TOOL)
G00 G41 X1.5 Z−2.125                         (MOVE TO POINT C)(TNC APPROACH)
G01 X1. F0.003
G01 Z−1.625
G03 X1.25 Z−1.5 R0.125                       (A)
G40 G01 X1.6                                 (TNC CANCEL)
G97 S500
```

        G28

        M30
        %

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

EXAMPLE 3   TNC WITH G71

This example uses tool nose compensation with a G71 roughing canned cycle.

PREPARATION

        SETTING 33   FANUC

        TOOLS
         T1   Insert with .032 radius, roughing

TOOL	OFFSET	RADIUS	TIP
T1	01	.032	3

PROGRAM EXAMPLE	DESCRIPTION
%	
O0813	*(EXAMPLE 3)*
G50 S1000	
T101	*(SELECT TOOL 1)*
G00 X1.5 Z.1	*(RAPID TO START POINT)*
G96 S100 M03	
G71 P80 Q180 U.01 W.005 D.08 F.012	*(ROUGH P TO Q WITH T1 USING G71 AND TNC)*
	(DEFINE PART PATH PQ SEQUENCE)
N80 G42 G00 X0.6	*(P) (G71 TYPE I, TNC RIGHT)*
G01 Z0 F0.01	*(START OF FINISH PART PATH)*
X0.8 Z–0.1 Fx0.005	
Z–0.5	
G02 X1.0 Z–0.6 I0.1	
G01 X1.5	
X2.0 Z–0.85	
Z–1.6	
X2.3	
G03 X2.8 Z–1.85 K–0.25	
G01 Z–2.1	*(Q) (END OF PART PATH)*
N180 G40 G00 X3.0 M05	*(TNC CANCEL)*
G28	*(ZERO X FOR TOOL CHANGE CLEARANCE)*
M30	
%	

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

EXAMPLE 4   **TNC WITH G72**

This example uses tool nose compensation with a G72 roughing canned cycle.
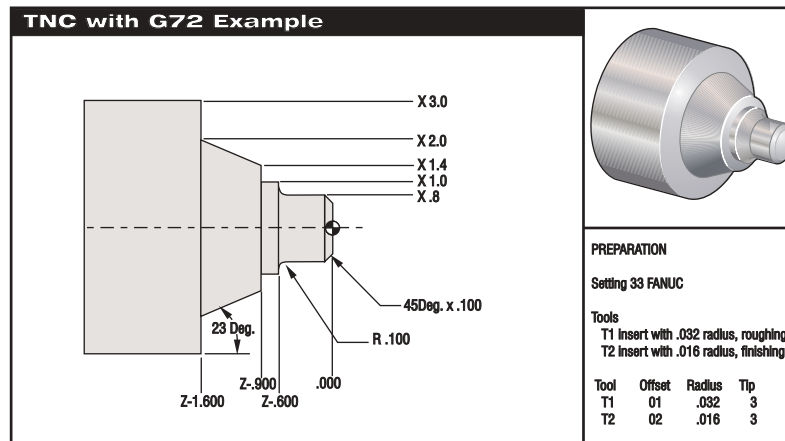
Figure 3.25-12. Part drawing for TNC with G72 example.

PROGRAM EXAMPLE   **DESCRIPTION**

%
O0814   *(EXAMPLE 4)*
G50 S1000
T101   *(SELECT TOOL 1)*
G00 X3.5 Z0.1   *(MOVE TO START POINT)*
G96 S190 M03

(ROUGH P TO Q WITH T1 USING G72 AND TNC)
G72 P80 Q180 U0.005 W0.01 D0.05 F.010
(DEFINE PART PATH PQ SEQUENCE)
N80 G41 G00 Z–1.6   *(P) (G72 TYPE I, TNC LEFT)*
G01 X2. F0.005
X1.4 Z–0.9
X1.0
Z–0.6
G03 X0.8 Z–0.5 K0.1

G01 Z–0.1
X0.6 Z0.
X0.

N180 G40 G00 Z0.01   *(TNC CANCEL)*
*(******OPTIONAL FINISHING SEQUENCE*****)*
G28   *(ZERO FOR TOOL CHANGE CLEARANCE)*
M01
T202   *(SELECT TOOL 2)*

N2 G50 S1000
G00 X3.5 Z0.1   *(MOVE TO START POINT)*
G96 S325 M03

                                             (FINISH P TO Q WITH T2 USING G70 AND TNC)

G70 P80 Q180
G00 Z0.5 M05
G28   *(ZERO FOR TOOL CHANGE CLEARANCE)*
M30
%

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

EXAMPLE 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   *(EXAMPLE 5)*
T101   *(SELECT TOOL 1)*
G50 S1000
G00 X3.5 Z.1   *(MOVE TO POINT S)*
G96 S100 M03
(ROUGH P TO Q WITH T1 USING G73 AND TNC)
G73 P80 Q180 U.01 W0.005 I0.3 K0.15 D4 F.012
(DEFINE PART PATH PQ SEQUENCE)
N80 G42 G00 X0.6   *(P)   (G72 TYPE I, TNC RIGHT)*
G01 Z0 F0.1
X0.8 Z–0.1 F.005
Z–0.5
G02 X1.0 Z–0.6 I0.1
G01 X1.4
X2.0 Z–0.9
Z–1.6
X2.3
G03 X2.8 Z–1.85 K–0.25
G01 Z–2.1
N180 G40 X3.1   *(Q)*
G00 Z0.1 M05   *(TNC CANCEL)*

```
(******OPTIONAL FINISHING SEQUENCE*****)
G28                                        (ZERO FOR TOOL CHANGE CLEARANCE)
M01
T202                                       (SELECT TOOL 2)
N2 G50 S1000
G00 X3.0 Z0.1                              (MOVE TO START POINT)
G96 S100 M03
(FINISH P TO Q WITH T2 USING G70 AND TNC)
G70 P80 Q180
G00 Z0.5 M05
G28                                        (ZERO FOR TOOL CHANGE CLEARANCE)
M30
%
```

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

EXAMPLE 6   TNC WITH G90

This example uses tool nose compensation with a G90 modal rough turning cycle.
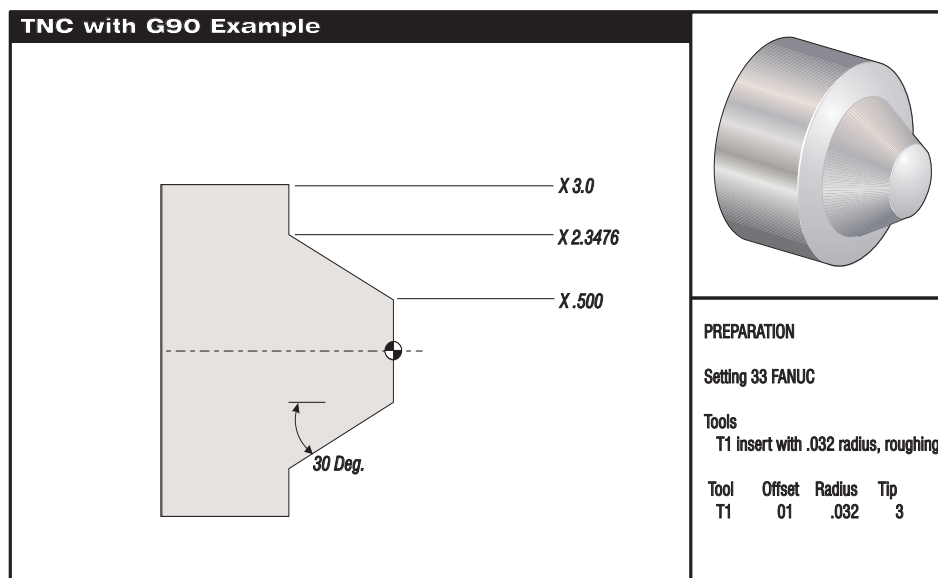
Figure 3.25-13. Part drawing for TNC with G90 example.

PROGRAM EXAMPLE

DESCRIPTION

```
%
O0816                    (EXAMPLE 6)
T101                     (SELECT TOOL 1)
G50 S1000
G00 X4.0 Z0.1            (MOVE TO START POINT)
G96 S100 M03
```

(ROUGH 30 DEG. ANGLE TO X2. AND Z–1.5 USING G90 AND TNC)
G90 G42 X2.55 Z–1.5 I–0.9238 F0.012
X2.45   *(OPTIONAL ADDITIONAL PASSES)*
X2.3476
G00 G40 X3.0 Z0.1 M05   *(TNC CANCEL)*
G28   *(ZERO FOR TOOL CHANGE
                                                                     CLEARANCE)*

M30
%

EXAMPLE 7   TNC WITH G94

This example uses tool nose compensation with a G94 modal rough turning cycle.
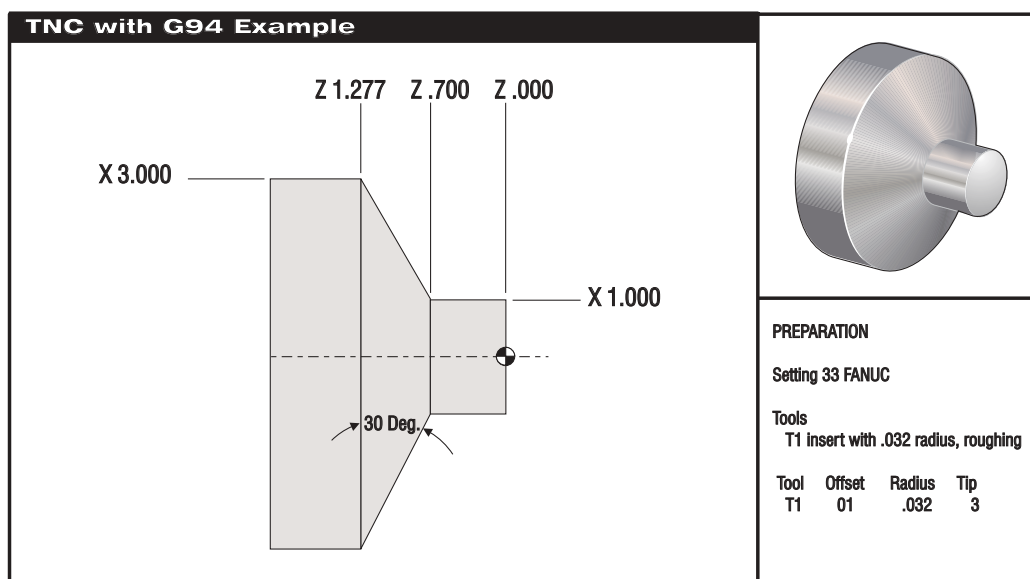
Figure 3.25-14. Part drawing for TNC with G94 example.

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)

3.26 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.
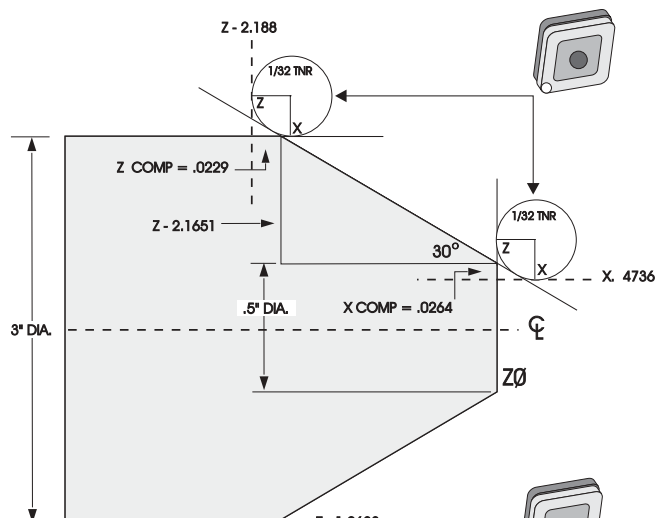
PROGRAM

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 30⁰ ANGLE
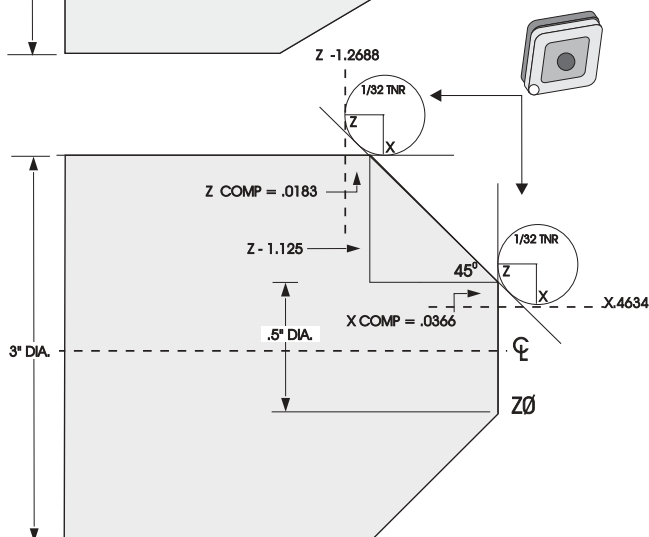
PROGRAM

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 45⁰ ANGLE
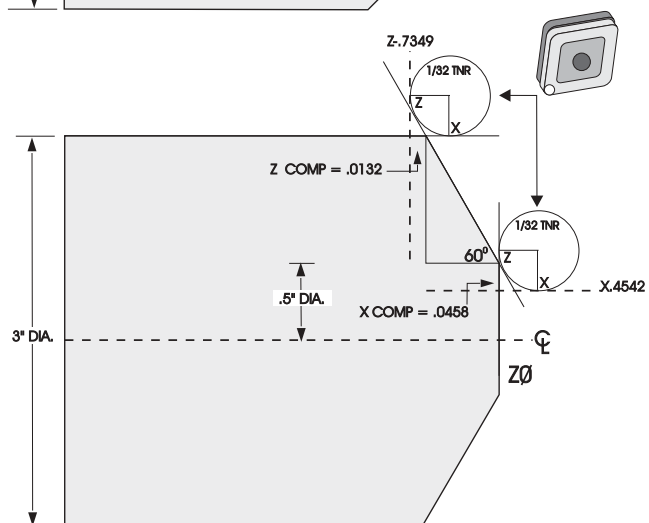
PROGRAM

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 60⁰ ANGLE

TOOL RADIUS AND ANGLE CHART

1/32 RADIUS

NOTE: THE X MEASUREMENT CALCULATED IS BASED ON PART DIAMETER

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

TOOL NOSE RADIUS CALCULATION DIAGRAM
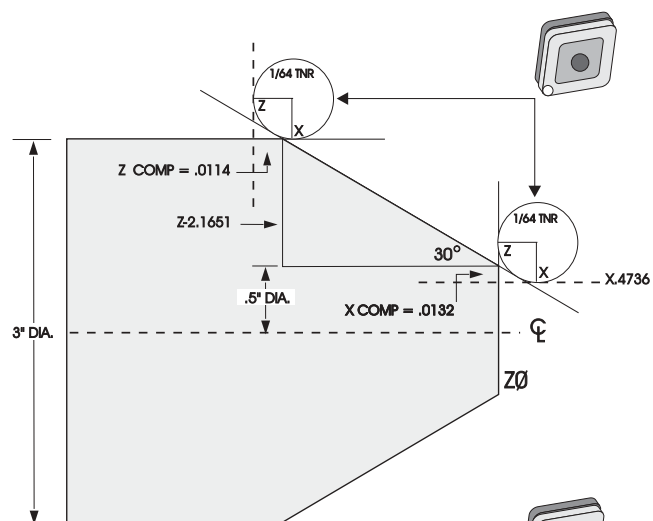
PROGRAM	
CODE	COMPENSATION (1/64 TNR)
G0 X0 Z.1	
G1 Z0	
X.4868	(X.5 - 0.0132 COMP)
X 3.0 Z-2.1765	(Z-2.1651 + 0.0114 COMP)

NOTE: USING COMPENSATION VALUES FOR 30°
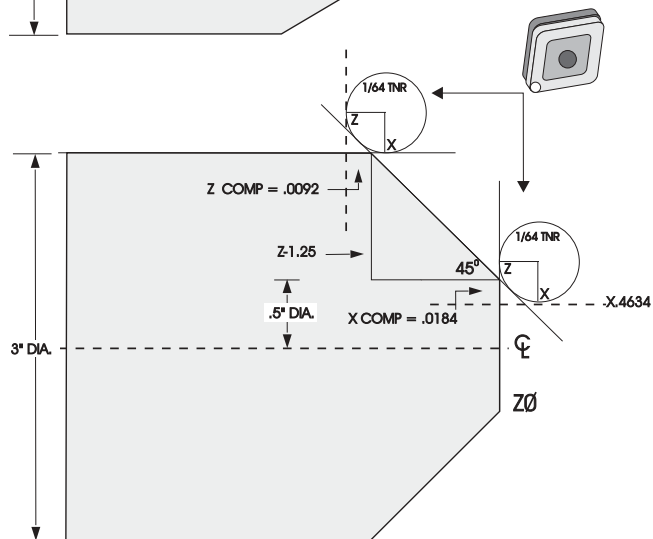
PROGRAM	
CODE	COMPENSATION (1/64 TNR)
G0 X0 Z.1	
G1 Z0	
X.4816	(X.5 - 0.0184 COMP)
X 3.0 Z-1.2592	(Z-1.25 + 0.0092 COMP)

NOTE: USING COMPENSATION VALUES FOR 45°
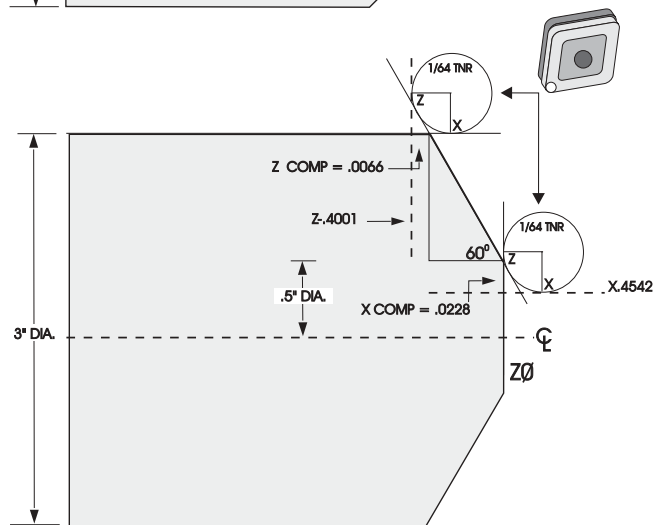
PROGRAM	
CODE	COMPENSATION (1/64 TNR)
G0 X0 Z.1	
G1 Z0	
X.4772	(X.5 - 0.0228 COMP)
X 3.0 Z-.467	(Z-.7217 + 0.066 COMP)

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			

3.27 TOOL NOSE CUTTER COMPENSATION USING G112

Tool Nose cutter compensation is a method of shifting the tool path so that the actual finished cut is moved to either the left or right of the programmed path. Normally cutter compensation is programmed to shift by exactly the radius of the tool so that the finished cut matches the programmed path. The Offset display is used to enter the amount for the tool to be shifted. The offset can be entered as the radius for both the geometry and wear values. The effective value is the sum of the geometry and wear values. Cutter radius compensation is only available in the X and Y planes (G17) within cartesian mode (G112).

GENERAL DESCRIPTION OF THE NOSE CUTTER COMPENSATION

G41 will select cutter compensation left; that is, the tool is moved to the left of the programmed path to compensate for the size of the tool. A tool must be programmed to select the correct tool size from compensation memory. If compensation memory contains a negative value for cutter size, cutter compensation will operate as though G42 was specified. Cutter path compensation in this machine applies only to motion in the **X** and **Y** coordinate planes.

G42 will select cutter compensation right; that is, the tool is moved to the right of the programmed path to compensate for the size of the tool. A tool must also be programmed to select the correct tool size from compensation memory. If compensation memory contains a negative value for cutter size, cutter compensation will operate as though G41 was specified.

The code G40 will cancel cutter compensation and is the default condition when a machine is powered on. When canceled, the programmed path is the same as the center of the cutter path. You may not end a program (M30, M00, M01, or M02) with cutter compensation active.

If cutter radius compensation is selected (G41 or G42), you may only use the X-Y plane for circular motions (G17). Cutter radius compensation is only available in the **X** and **Y** coordinate planes (G17).

There is a simple rule about cutter compensation which helps to make clear the motions the control uses to compensate for tool size. The control operates on one motion block at a time. It will look ahead, however, to check the next two blocks containing **X** or **Y** motions. The interference checks are performed on these three motions. Setting 58 controls how this part of cutter compensation works. It can be set to Fanuc or Yasnac.

When Setting 58 is set to Yasnac, the control must be able to position the tool edge along all of the programmed cuts without overcutting the next two motions. All outside angles are joined by a circular motion.

When Setting 58 is set to Fanuc, the control does not require that the tool cutting edge be placed along all programmed cuts. Overcutting, however, is still prevented, and if overcutting cannot be prevented, an alarm will occur. Outside angles less than or equal to 270 degrees are joined by a square corner and outside angles of more than 270 degrees are joined by an extra linear motion.

The following four diagrams show how cutter compensation works for the two possible values of Setting 58. Note that a small cut of less than tool radius and at a right angle to the previous motion will only work with the Fanuc setting.

When entering and exiting cutter compensation or when changing from left side to right side compensation, there are special considerations to be aware of. Cutting should not be performed during any of these three moves. In a block that turns on cutter compensation, the starting position of the move is the same as the programmed position, but the ending position will be offset by the cutter compensation size. In a block that turns off cutter compensation, the starting point is offset and the ending point is not offset. Similarly, when a block changes from left to right compensation, the starting point is shifted in one direction and the ending point is shifted in the other direction. The result of all this is that the tool is moved through a path that may not be the same as the intended path or direction.

If cutter compensation is turned on or off in a block without any X-Y move, there is no change made to cutter compensation until the next **X** or **Y** move is encountered. To enter cutter compensation, a nonzero **D** code must be specified and either G41 or G42 specified. To exit from cutter compensation, you may specify either D0 or G40, or both.

You should always turn off cutter compensation in a move which clears the tool away from the part being cut. If a program is terminated with cutter compensation still active, an alarm is generated. In addition, you cannot turn cutter compensation on or off during a circular move (G02 or G03); otherwise an alarm will be generated.

An offset selection of D0 will use zero as the offset size and have the effect of turning off cutter compensation. If a new value from offset memory is selected while cutter compensation is active, the starting point of a move will reflect the old value and the ending point will reflect the new value. This will have the effect of shifting the motion to something other than what was intended by the programmer. You cannot change the offset code or side during a circular motion block.

When turning on cutter compensation in a move that is followed by a second move at an angle of less than 90 degrees, there are two ways of computing the first motion: cutter compensation type A and type B (Setting 43). Type A is the default in Setting 43 and is what is normally needed; the tool moves directly to the programmed start point for the second cut. Type B is used when clearance around a fixture of clamp is needed, or in rare cases when part geometry demands it. The diagrams on the following pages illustrate the differences between type A and type B for both Fanus and Yasnac settings (Setting 58).

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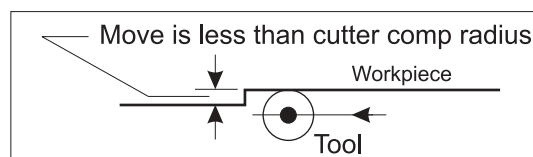
Note that a small cut of less than tool radius and at a right angle to the previous motion will only work with the Fanuc setting. A cutter compensation alarm will be generated if the machine is set to the Yasnac setting.

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

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

Cutter Compensation - (YASNAC Style)

G41 with Positive Tool Diameter or G42 with Negative Tool Diameter

Radius

Actual center of tool path

Programmed path

G41 or G42 in this Block

G40 in this Block
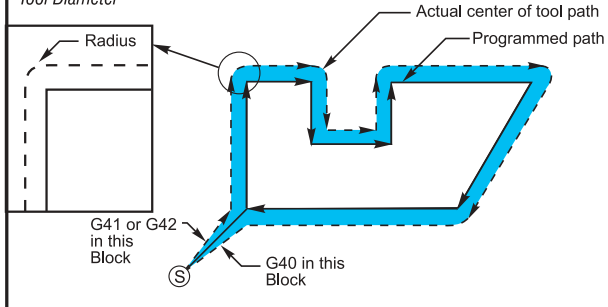
Cutter Compensation - (YASNAC Style)

G42 with Positive Tool Diameter or G41 with Negative Tool Diameter

Radius

Actual center of tool path

Programmed path

G40 in this Block

G41 or 42 in this Block
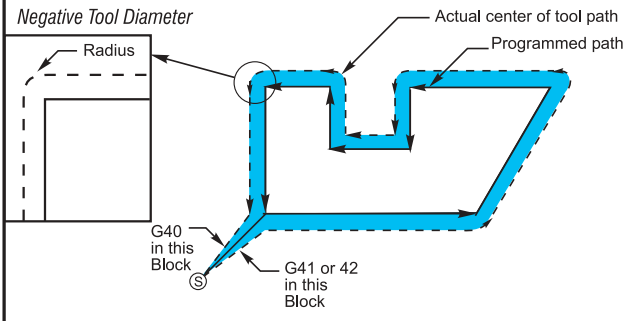
Cutter Compensation - (FANUC Style)

G42 with Positive Tool Diameter or G41 with Negative Tool Diameter

Actual center of tool path

Programmed path

G40 in this Block

G41 or 42 in this Block

Extra Move
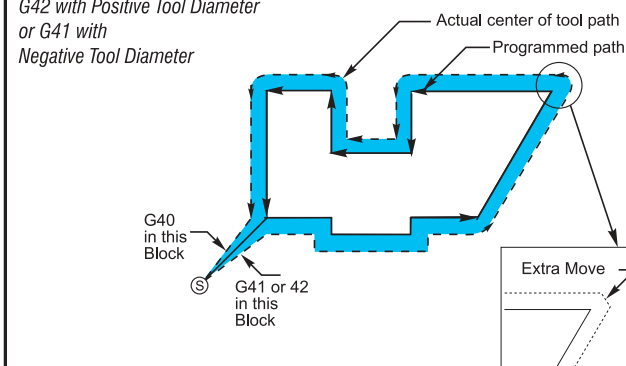
Cutter Compensation - (FANUC Style)

G41 with Positive Tool Diameter or G42 with Negative Tool Diameter

Actual center of tool path

Programmed path

G41 or G42 in this Block

G40 or G42 in this Block

Extra Move
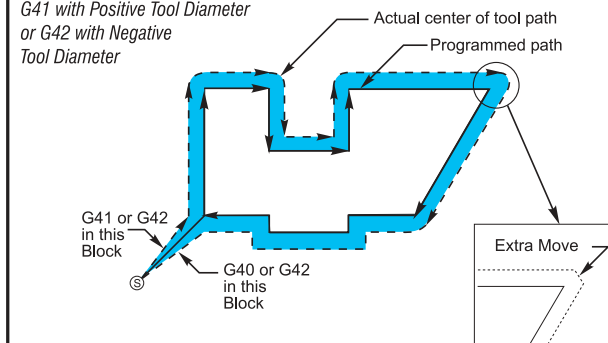
CUTTER COMPENSATION ENTRY
(Yasnac)

TYPE A   TYPE B

G42

Program Path

Tool Center Path

r   Tool Radius
Program Path
Tool Center Path

G42

Program Path

Tool Center Path

G42

r

Program Path

Tool Center Path

G42

r

Program Path

Tool Center Path

G42

r
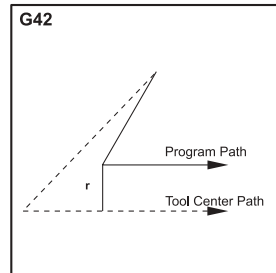
Program Path
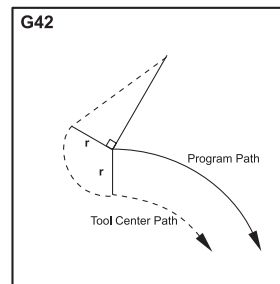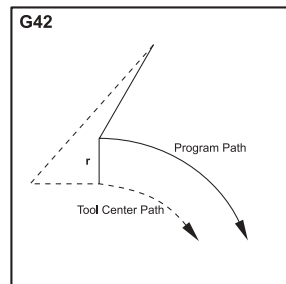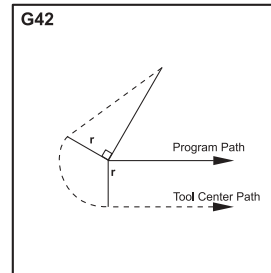
Tool Center Path

CUTTER COMPENSATION ENTRY
(Fanuc style)

TYPE A   TYPE B

G42

Program Path

Tool Center Path

r   Tool Radius
Program Path
Tool Center Path

G42

r

Program Path

Tool Center Path

G42

r

Program Path

Tool Center Path

G42

r   r

Program Path

Tool Center Path

r

G42

r   r

Program Path

Tool Center Path

r
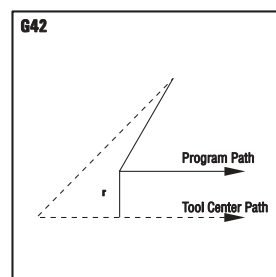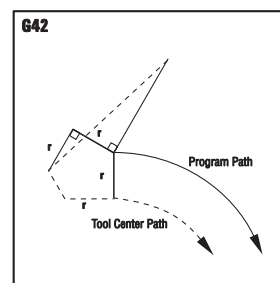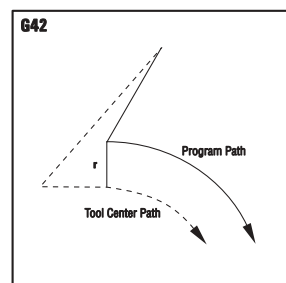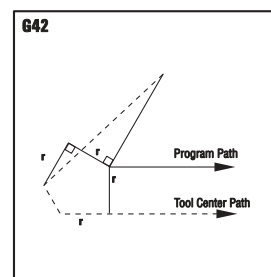
3.28 QUICK CODE

INTRODUCTION

This programming option can be activated by contacting your local HAAS dealer.

QUICK CODE is an innovative 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. This is possible because with one menu selection you can replace a large number of individual keystrokes, with just a few.

An Open System

Quick Code 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.

QUICK CODE TERMINOLOGY

Before describing the Quick Code environment you need to know the terms listed below. The following is a list of the Quick Code terms and an illustration of the display.

USAGE AND FEATURES

ACCESSING QUICK CODE

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 Quick Code screen. Each additional press of this key will switch between the Visual Quick Code, 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 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.

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. 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.
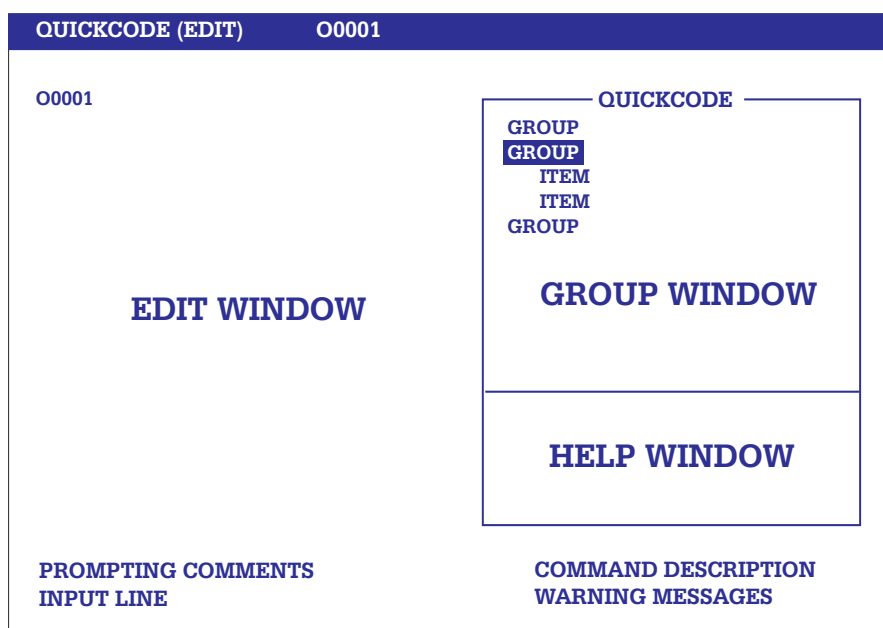
THE HELP WINDOW

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.

SPECIAL KEYS

Quick Code makes use of the jog handle to select from the group list and group items. 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.

```
QUICKCODE (EDIT)         O0001


O0001                           ┌──── QUICKCODE ────┐
                                │  GROUP             │
                                │  GROUP             │
                                │     ITEM           │
                                │     ITEM           │
                                │  GROUP             │
                                │                    │
                                │   GROUP WINDOW     │
        EDIT WINDOW             │                    │
                                ├────────────────────┤
                                │                    │
                                │   HELP WINDOW      │
                                │                    │
                                │                    │
                                └────────────────────┘

PROMPTING COMMENTS               COMMAND DESCRIPTION
INPUT LINE                       WARNING MESSAGES
```

The Quick Code display.

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 your at in the program.

Pressing RESET will exit Quick Code and send the cursor back to the beginning.

> G84 Z-0.65 R0.3 F0.05   (G84 Z-.65 TAPPING CYCLE USING A 1/4-20 TAP)

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.

3.29 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. Before starting VQC, if you wish to insert G-code to an existing program, open your program in the Advanced Editor and place the cursor at the ";" (End of Block) where the new program will be added.

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 open a window diplaying 4 options.

   A) SELECT/CREATE A PROGRAM
   If selected, another window will open prompting the user to select a program name. Simply highlight the desired name and press WRITE. This will cause VQC to output to the selected program. If the program already contains G-code, VQC will output to the beginning before the existing G-code and will not pass any M30s. The user also has the option to CREATE a new program by simply entering a program name using 'O'. Then by simply pressing WRITE again will cause VQC to output to the newly created program.

   B) ADD TO CURRENT PROGRAM
   If selected, the G-code generated by VQC will be inserted at the last position of the cursor within a program before entering VQC. If the program was empty, then any M30s. in the G-CODE section of the VQC template will be passed into the program. If the program contained anything, then NO M30s. will be passed.

   C) MDI
   If selected, the newly created VQC G-code will output to MDI. Note: Anything that may be in MDI will be overwritten.

   D) CANCEL
   If selected, the window will close and the focus returns to the parameter list. Note: the UNDO and CANCEL keys also perform this operation.

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.

Note: Tool Nose Compensation may be applied, if so all Tool Nose Compensation rules will be used, refer to the Tool Nose Compensation section for complete details.

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.

The ability to load different templates is useful for users wanting to try templates without the need of making the template name 9997 or deleting an older template. By pressing the F2 key, a window will open prompting the user to SELECT a template. The default template name which VQC will attempt to load after power up is: O9997. If O9997 is not found, a window will open prompting the user to load or select a different template.

After a different template has been selected, VQC will remember the selected template while the machine is powered up. After the power has been cycled, VQC will switch back to the default template file name: O9997. If the selected program is not a valid VQC template, the message "No Category" will be shown on the lower left corner of the screen and the Category List will be empty.

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.
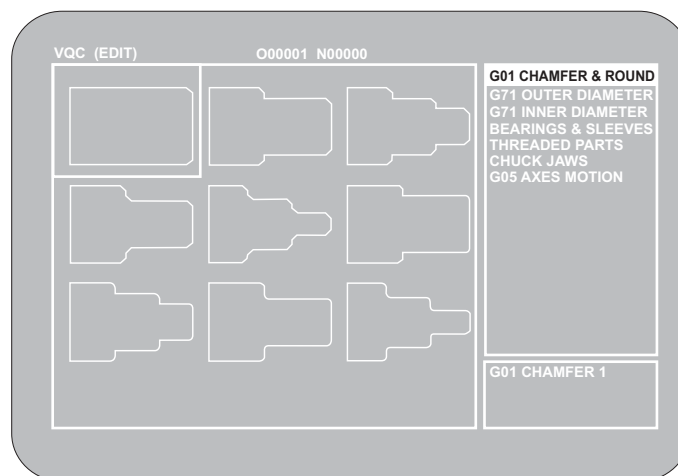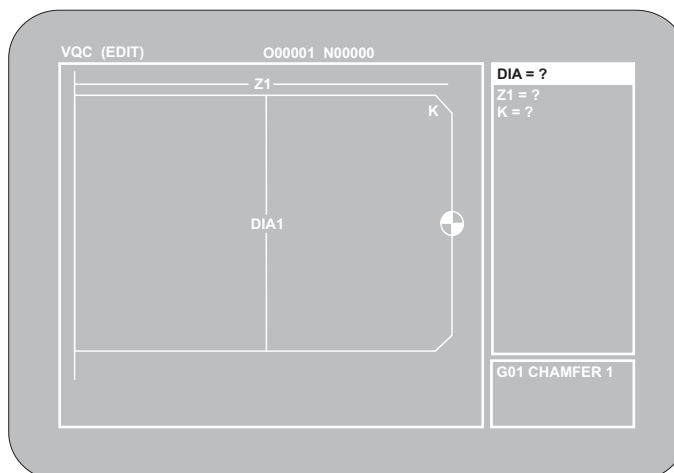
Selecting a Part Template

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, another prompt displays asking the user where the G-code is to be placed. There are 4 options within this prompt:

1) SELECT/CREATE A PROGRAM
If selected, another window will open prompting the user to select a program name. Simply highlight the desired name and press WRITE. This will cause VQC to output to the selected program. If the program already contains G-code, VQC will output to the beginning before the existing G-code and will not pass any M30s. The user also has the option to CREATE a new program by simply entering a program name using 'O'. Then by simply pressing WRITE again will cause VQC to output to the newly created program.

2) ADD TO CURRENT PROGRAM
If selected, the G-code generated by VQC will be inserted at the last position of the cursor within a program before entering VQC. If the program was empty, then any M30s. in the G-CODE section of the VQC template will be passed into the program. If the program contained anything, then NO M30s. will be passed.

3) MDI
If selected, the newly created VQC G-code will output to MDI. Note: Anything that may be in MDI will be overwritten.

4) CANCEL
If selected, the window will close and the focus returns to the parameter list. Note: the UNDO and CANCEL keys also perform this operation.
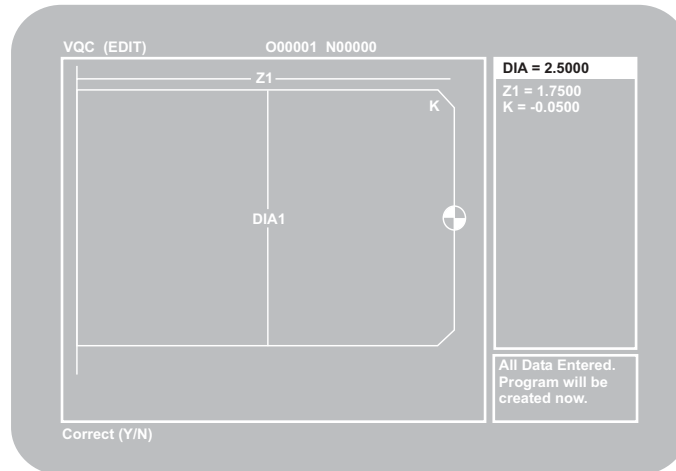
Additionally the program will be put into the Advanced Editor in order to double check the program. Verify the program by first running it in Graphics mode.

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.

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
CIRCLE	Specifies a Circle	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.

PROGRAMMIMG 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 catagory 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, becomming more and more specific. This is the way that Visual Quick Code is used. First the user sees a list of catagories. 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)
(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 circle is:

   (CIRCLE [X,Y] r)

Where X and Y are the center of the circle, and r is the radius.

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.

3.30 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 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 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 Advanced Editor is entered by pressing the EDIT key. The user can alternate between the Advanced Editor, the 40 column editor, and Quick Code with successive presses of the PRGRM/CONVRS key.

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.

The UNDO key is used to deactivate the pull-down menu system. If UNDO is pressed after invoking an executing function from a pull-down menu, it will abort that function.

The EDIT key can be used to "switch", left or right, between the 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.

Mode —— **PROGRAM (EDIT)   Onnnnn   F1 KEY TURNS MENU ON / OFF** —— Key Help

Menu bar —— **PROGRAM EDIT SEARCH MODIFY I/O HELP**

Onnnnn

Active Edit Display

Inactive Edit Display

Context sensitive help

Alarm Messages
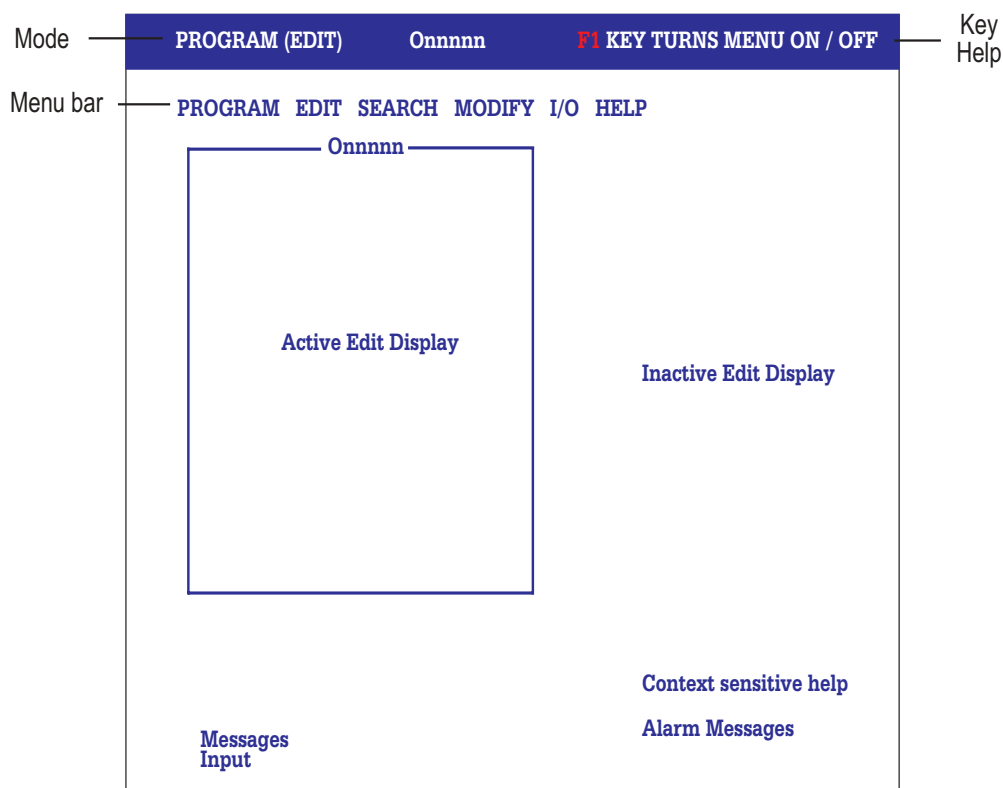
Messages
Input

Figure 3.30-1. 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 alerm 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.

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

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.

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.

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.

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 MENU

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.

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.

COPY SELECTED TEXT
All selected text will be copied to the line following the current cursor arrow position.

DELETE SELECTED TEXT
This item deletes any selected block. If no block is selected, the currently highlighted item is deleted. 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.

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

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.

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 MODIFY MENU

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.

RENUMBER BY TOOL
Searches selected text, or the entire program, for T codes and renumbers program blocks grouped by T code.

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 I/O MENU

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

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 it's 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 Quick Code section of 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

SELECT PROG	Will quickly bring up the program list on the inactive side of edit display to SELECT PROGRAM FROM LIST.
F2	This key will begin to SELECT TEXT and define the starting line of a block to be edited. Scroll down to the last line in the block definition, and press the F2 or ENTER key. The selected block of text will then be highlighted.
EDIT	This key can be used to SWITCH TO LEFT OR RIGHT SIDE between two programs that have been selected to edit.
F4	Pressing F4 will open another copy of the same program on the other side of the Advanced Editor display. The user can quickly edit two different locations in the same program. The edit key will switch you back and forth and update between the two programs. 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	INSERT can be used to COPY SELECTED TEXT in a program to the line after where you place the cursor arrow point.
ALTER	ALTER can be used to MOVE SELECTED TEXT in a program to the line after where you place the cursor arrow point.
DELETE	DELETE can be used to DELETE SELECTED TEXT in a program.
UNDO	If a block has been selected, pressing UNDO will simply exit a block definition.
SEND RS232	Pressing the SEND RS-232 key will activate that I/O menu selection.
RECV RS232	Pressing RECV RS-232 key will activate that I/O menu selection.
ERASE PROG	Pressing the ERASE PROG key will activate that I/O menu selection. This will bring up program list on the inactive side of edit display for you to cursor to a program and delete it.

3.31 MACROS (OPTIONAL)

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 <u>not</u> 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		H
Alternate:							I	J	K	I	J

Variable:	12	13	14	15	16	17	18	19	20	21	22
Address:		M				Q	R	S	T	U	V
Alternate:	K	I	J	K	I	J	K	I	J	K	I

Variable:	23	24	25	26	27	28	29	30	31	32	33
Address:	W	X	Y	Z							
Alternate:	J	K	I	J	K	I	J	K	I	J	K

Note that variables 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)
#1064-#1068	Maximum axis loads
#1080-#1087	Raw analog to digital inputs (read only)
#1090-#1098	Filtered analog to digital inputs (read only)
#1094	Spindle load with OEM spindle drive (read only)
#1098	Spindle load with Haas vector drive (read only)
#1100-#1139	40 discrete outputs
#1140-#1155	16 extra relay outputs via multiplexed output
#1264-#1268	Maximum axis loads
#2001-#2099	X axis tool shift offsets
#2101-#2199	Z axis tool shift offsets
#2201-#2299	Tool nose radius offsets
#2301-#2399	Tool tip direction
#2401-#2499	Tool diameter/radius offsets
#2601-#2699	Tool diameter/radius wear
#2701-#2799	X axis tool wear offsets
#2801-#2899	Z axis tool wear offsets
#2901-#2999	Tool nose radius wear offsets
#3000	Programmable alarm
#3001	Millisecond timer
#3002	Hour timer
#3003	Single block suppression
#3004	Override control
#3006	Programmable stop with message
#3011	Year, month, day
#3012	Hour, minute, second
#3020	Power on timer (read only)
#3021	Cycle start timer (read only)
#3022	Feed timer (read only)
#3023	Present part timer (read only)
#3024	Last complete part timer (read only)
#3025	Previous part timer (read only)
#3026	Tool in spindle (read only)
#3027	Spindle RPM (read only)

#3030	Single Block (read only)
#3031	Dry Run (read only)
#3032	Block Delete (read only)
#3033	Opt Stop (read only)
#3901	M30 count 1
#3902	M30 count 2
#4001-#4020	Previous block group codes
#4101-#4126	Previous block address codes

NOTE: Mapping of 4101 to 4126 is the same as the alphabetic addressing of "Macro Arguments" section; e.g. the statement x1.3 sets variable #4124 to 1.3.

#5000-#5006	Previous block end position
#5020-#5027	Present machine coordinate position
#5041-#5046	Present work coordinate position
#5061-#5065	Present skip signal position - X,Y,Z,A,B
#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	G114 additional work offsets
#7061-#7066	G115 additional work offsets
#7081-#7086	G116 additional work offsets
#7101-#7106	G117 additional work offsets
#7121-#7126	G118 additional work offsets
#7141-#7146	G119 additional work offsets
#7161-#7166	G120 additional work offsets
#7181-#7186	G121 additional work offsets
#7201-#7206	G122 additional work offsets
#7221-#7226	G123 additional work offsets
#7241-#7246	G124 additional work offsets
#7261-#7266	G125 additional work offsets
#7281-#7286	G126 additional work offsets
#7301-#7306	G127 additional work offsets
#7321-#7326	G128 additional work offsets
#7341-#7346	G129 additional work offsets

1-Bit Discrete Inputs

Inputs designated as "Spare" can be connected to external devices and used by the programmer.

Maximum Axis Loads

The following variables are now used to contain the maximum load values for each axis. They can be cleared by cycling power to the machine or by setting the macro to zero in a program (for example, #1064=0;).

1064 = X axis	1264 = C axis
1065 = Y axis	1265 = U axis
1066 = Z axis	1266 = V axis
1067 = A axis	1267 = W axis
1068 = B axis	1268 = T axis

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. The first 15 characters of the comment will be displayed on the lower left corner of the screen.

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–stoppable code	(FEED HOLD not allowed)
#3004=0;	(Enables FEED HOLD button)
Depart code	(FEED HOLD allowed)

The following is a map of variable #3004 bits and the associated overrides.　　　E=Enabled D=Disabled

#3004	FEED HOLD	FEED RATE OVERRIDE	EXACT STOP CHECK
0	E	E	E
1	D	E	E
2	E	D	E
3	D	D	E
4	E	E	D
5	D	E	D
6	E	D	D
7	D	D	D

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

Expressions

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

Conditional Expressions

In the HAAS control, ALL expressions set a conditional value. The value is either 0.0 (FALSE) or the value is 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 O] AND [#7 LE 9]] GOTO99;
#3000=1 (INVALID DIGIT)
;
N99
#7=FIX[#7]             (TRUNCATE ANY FRACTIONAL PART) ;
;
GOTO#7                 (NOW ENGRAVE THE DIGIT)   ;
;
N0                     (DO DIGIT ZERO)
...
M99
;
N1                     (DO DIGIT ONE)
;
M99
;
N2                     (DO DIGIT TWO)
;
...
;
(etc.,...)
```

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[ ]

Macros allow additional capabilities to communicate with peripheral devices. One can do digitizing of parts, provide runtime inspection reports, or synchronize controls with user provided devices. The commands provided for this are POPEN, DPRNT[ ] and PCLOS.

Communication preparatory commands

POPEN and PCLOS are not required on the HAAS lathe. It has been included so that programs from different controls can be ported to the HAAS. On some controls POPEN is required prior to using a DPRNT statement. POPEN 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 that eight. Thus the following formats are illegal:

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

A decimal point is printed out between the whole part and the fractional part. The fractional part is rounded to the least significant place. When zero places are reserved for the fractional part, then no decimal point is printed out. Trailing zeros are printed 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]] ;	X1.5436 Z 1.544 T 1
N3	DPRNT[***MEASURED*INSIDE*DIAMETER***] ;	MEASURED INSIDE DIAMETER
N4	DPRNT[] ;	(no text, only a carriage return)
N5	#1=123.456789 ;	
N6	DPRNT[X−#1[25]] ;	X−123.45679 ;

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 **N**OTES

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 ] I [ #20 * 0.9 ] F#9
G00 X0 Z0 T100
M30
%
```

Face Groove Macro Example

Z Face
Z Depth
Tool
X Width
X Minor Dia.

Z Start Position
Groove

Tool Width = 0.125
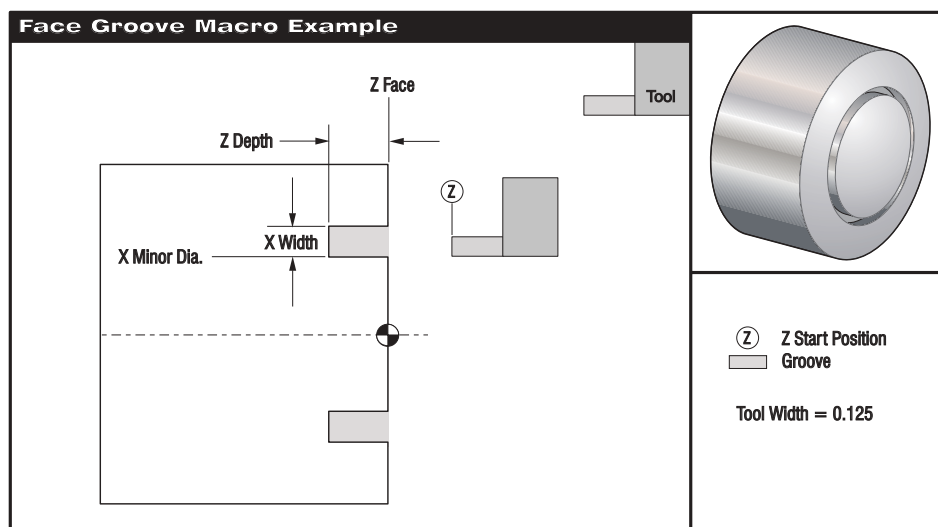
4. G CODES - PREPARATORY FUNCTIONS

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	221
G01	01	Linear Interpolation Motion	221
G01	01	Chamfering and Corner Rounding	222
G02	01	CW Interpolation Motion	227
G03	01	CCW Interpolation Motion	228
G04	00	Dwell	229
G05	00	Fine Spindle Control Motion	288
G09	00	Exact Stop	229
G10	00	Set Offsets	229
G14	17	Sub-Spindle Swap	230
G15	17	Sub Spindle Mode Cancel	230
G17	02	XY Plane Selection	230
G18	02*	ZX Plane Selection	232
G19	02	YZ Plane Selection	230
G20	06*	Selection Inch	232
G21	06	Select Metric	232
G28	00	Return To Reference Point	232
G29	00	Return From Reference Point	233
G31	00	Feed Until Skip (optional)	233
G32	01	Threading	233
G40	07*	Tool Nose Compensation Cancel	236
G41	07	Tool Nose Compensation Left	237
G42	07	Tool Nose Compensation Right	237
G50	00	Spindle Speed Clamp / Set Global Coor. Offset	238
G51	00	Cancel Offset (Yasnac)	240
G52	00	Set Local Coordinate System (Fanuc)	240
G53	00	Non Modal Machine Coordinate Selection	240
G54	12*	Select Work Coordinate System 1	240
G55	12	Select Work Coordinate System 2	240
G56	12	Select Work Coordinate System 3	240
G57	12	Select Work Coordinate System 4	240
G58	12	Select Work Coordinate System 5	240
G59	12	Select Work Coordinate System 6	240
G61	15	Exact Stop Modal	240
G64	15*	G61 Cancel	240
G65	00	Macro Subroutine Call (optional)	197
G70	00	Finishing Cycle	241
G71	00	O.D./I.D. Stock Removal Cycle	243
G72	00	End Face Stock Removal Cycle	252
G73	00	Irregular Path Stock Removal Cycle	256
G74	00	End Face Grooving Cycle, Peck Drilling	257
G75	00	O.D./I.D. Grooving Cycle, Peck Drilling	260
G76	00	Threading Cycle, Multiple pass	263
G77	00	Flatting Cycle (optional)	290
G80	09*	Canned Cycle Cancel	270
G81	09	Drill Canned Cycle	270
G82	09	Spot Drill Canned Cycle	270
G83	09	Normal Peck Drill Canned Cycle	271

G84	09	Tapping Canned Cycle	272
G85	09	Boring Canned Cycle	274
G86	09	Bore and Stop Canned Cycle	275
G87	09	Bore/Manual Retract Canned Cycle	276
G88	09	Bore/Dwell/Manual Retract Canned Cycle	277
G89	09	Bore/Dwell Canned Cycle	278
G90	01	O.D./I.D. Turning Cycle, Modal	279
G92	01	Threading Cycle, Modal	281
G94	01	End Facing Cycle, Modal	283
G95	09	Live Tool Rigid Tap (optional)	292
G96	13	Constant Surface Speed On	284
G97	13*	Constant Surface Speed Cancel	284
G98	05	Feed per Minute	285
G99	05*	Feed per Revolution	285
G100	00	Disable Mirror Image	285
G101	00	Enable Mirror Image	285
G102	00	Programmable Output To RS-232	285
G103	00	Limit Block Lookahead	286
G110	12	Select Work Coordinate System 7	286
G111	12	Select Work Coordinate System 8	286
G112	04	XY to XC Interpretation	230
G113	04	G112 Cancel	231
G114	12	Select Work Coordinate System 9	286
G115	12	Select Work Coordinate System 10	286
G116	12	Select Work Coordinate System 11	286
G117	12	Select Work Coordinate System 12	286
G118	12	Select Work Coordinate System 13	286
G119	12	Select Work Coordinate System 14	286
G120	12	Select Work Coordinate System 15	286
G121	12	Select Work Coordinate System 16	286
G122	12	Select Work Coordinate System 17	286
G123	12	Select Work Coordinate System 18	286
G124	12	Select Work Coordinate System 19	286
G125	12	Select Work Coordinate System 20	286
G126	12	Select Work Coordinate System 21	286
G127	12	Select Work Coordinate System 22	286
G128	12	Select Work Coordinate System 23	286
G129	12	Select Work Coordinate System 24	286
G154	12	Select Work Coordinates P1-P99	286
G184	09	Reverse Tap Canned Cycle	185
G186	09	Reverse Live Tool Rigid Tap (optional)	294
G187	00	Accuracy Control for High Speed Machining	287
G195	00	Live Tool Radial Tapping	295
G196	00	Reverse Live Tool Vector Tapping	295
G200	00	Index on the Fly	296

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.

G00   **Rapid Motion Positioning**   **Group 01**

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

This G code is used to 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.

G01   **Linear Interpolation Motion**   **Group 01**
    F   Feed rate
    *B   B–axis motion command
    *U   X–axis incremental motion command
    *W   Z–axis incremental motion command
    *X   X–axis absolute motion command
    *Z   Z–axis absolute motion command

This **G** code provides for straight line (linear) motion from point to point. Motion can occur in 1, 2 or 3 axes. All axes will start and finish motion at the same time. The C-axis may also be commanded and this will provide a helical motion. The speed of all axes is controlled so that the feed rate specified is achieved along the actual path. C-axis feed rate is dependent on the C-axis diameter setting (Setting 102) and will provide a helical motion. The **F** command is modal and may be specified in a previous block. Only the axes specified are moved. The auxiliary axes **B**, **U**, **V**, and **W** can also be moved with a G01 but only one axis is moved at a time.

A chamfer block or a corner rounding block can be automatically inserted between two linear interpolation blocks by specifying K (chamfering) or R (corner rounding). There must be a terminating linear interpolation block following the beginning block (a G04 pause may intervene). These two linear interpolation blocks specify a theoretical corner of intersection. If the beginning block specifies a K the value following the K is the distance from the corner of intersection where the chamfer begins and also the distance from that same corner where the chamfer ends. If the beginning block specifies an R the value following the R is the radius of a circle tangent to the corner at two points: the beginning of the corner rounding arc block that is inserted and the endpoint of that arc. There can be consecutive blocks with chamfer or corner rounding specified. There must be movement on the two axes specified by the selected plane (whichever plane that is active X-Y (G17) X-Z (G18) or Y-Z (G19).
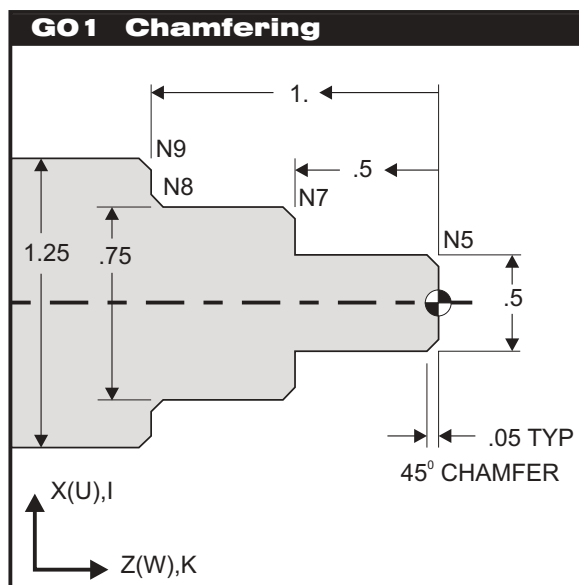
GO1 Chamfering

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 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 Ii

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

Addresses
I = chamfering, Z to X (X axis direction, +/-, "Radius" value)
K = chamfering, X to Z (Z axis direction, +/-)
R = corner rounding (X or Z axis direction, +/-, "Radius" value)

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

G01 Chamfering with A   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 or G72.

G01 Chamfering with A
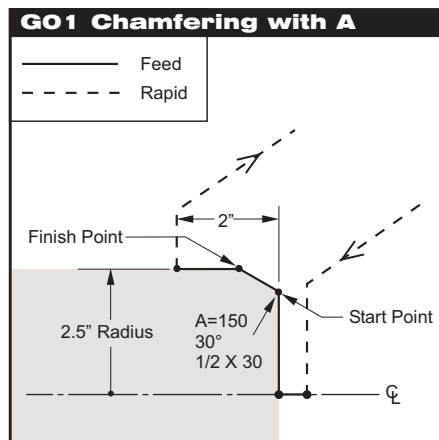
PROGRAM EXAMPLE

T606

G54;
M03 S1500;
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;

G01 Corner Rounding

G01 Corner Rounding
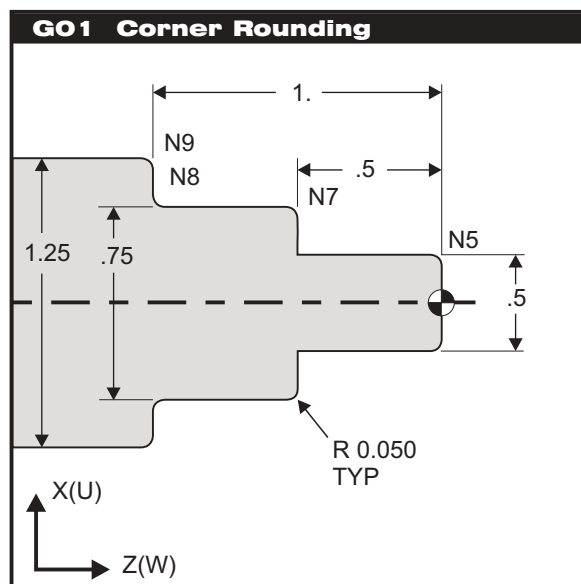
PROGRAM EXAMPLE

Automatic Corner Rounding

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

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

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

X3.5 Z-0.5

1

X2.5 Z-2. 0.1

0.1

2

X1.5 Z-0.5

X2.5 Z-2.

3

0.1

X1.5 Z-0.5

0.1

4

X0.5 Z-2.

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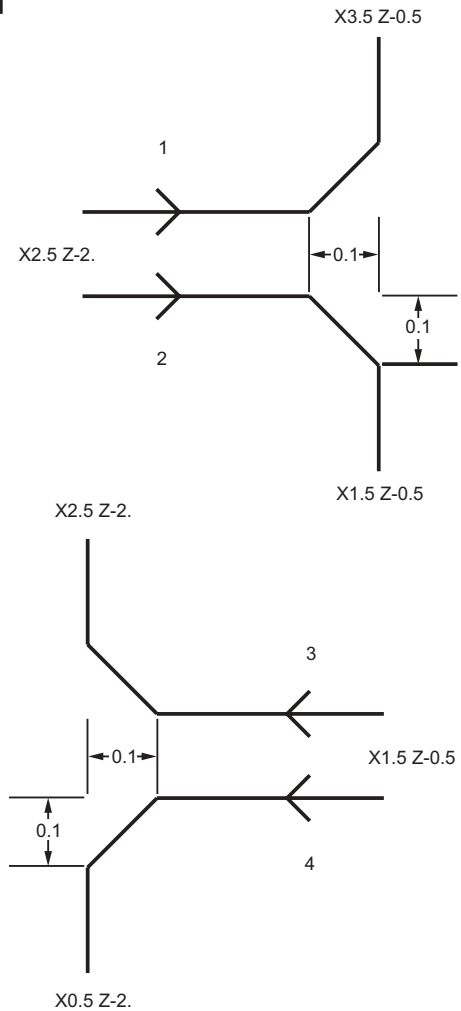
1. X- to Z-	X1.5 Z-1.; G01 X0.5 K-0.1; Z-2.;	X1.5 Z-1.; G01 X0.7; X0.5 Z-1.1; Z-2.
2. X- to Z+	X1.5 Z-1.; G01 X0.5 K0.1; Z0.;	X1.5 Z-1.; G01 X0.7; X0.5 Z-0.9; Z0.;
3. X+ to Z-	X0.5 Z-1.; G01 X1.5 K-0.1; Z-2.;	X0.5 Z-1.; G01 X1.3; X1.5 Z-1.1; Z-2.
4. X+ to Z+	X0.5 Z-1.; G01 X1.5 K0.1; Z0.;	X0.5 Z-1.;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

X1.5 Z-1.

1   2

0.1

X0.5 Z-2.   X0.5 Z0

0.1

X1.5 Z-2.

X1.5 Z0

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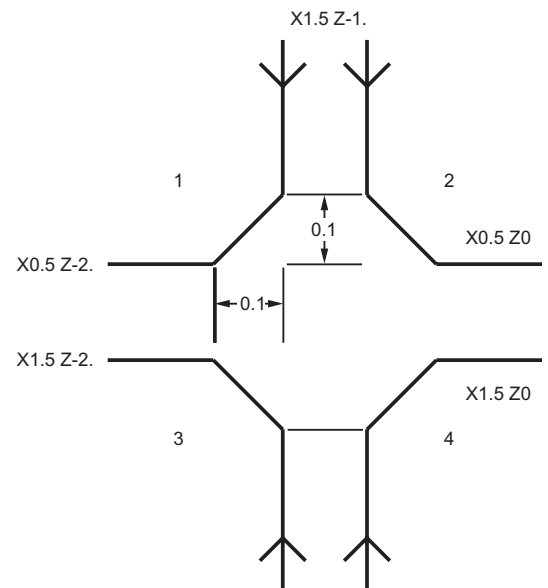
Corner Rounding	Code Example	Movement
1. Z+ to X+	X2. Z-2.; G01 Z-1 R.1; X3.;	X2. Z-2.; G01 Z-1.1; G03 X2.2 Z-1. R0.1; G01 X3.;
2. Z+ to X-	X2. Z-2.; G01 Z-1. R-0.1; X1.;	X2. Z-2.; G01 Z-1.1; G02 X1.8 Z-1 R0.1; G01 X1.;
3. Z- to X+	X2. Z-1.; G01 Z-2. R0.1; X3.;	X2. Z-1.; G01 Z-1.9; G02 X2.2 Z-2. R0.1;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

X3. Z-1.

1   R=0.1

X2. Z-2.

2

X1. Z-1.

X3. Z-2.

R=0.1

3

X2. Z-1.

4
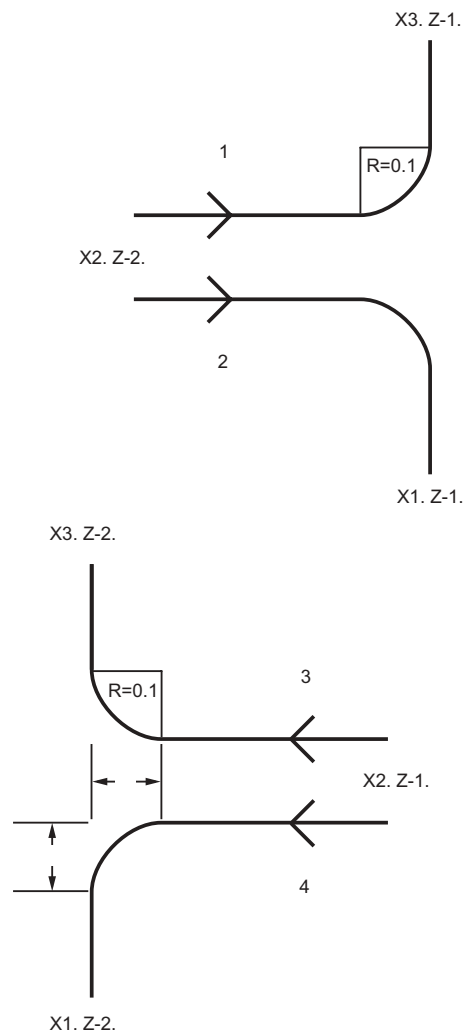
X1. Z-2.

1. X- TO Z-	X3. Z-1.; G01 X2. R-0.1; Z-2.	X3. Z-1.; G01 X2.2; G02 X2. Z-1.1 R0.1; G01 Z-2.;
2. X- TO Z+	X3. Z-2.; G01 X2. R0.1; Z-1.	X3 Z-2; G01 X2.2; G03 X2. Z-1.9 R0.1; G01 Z-1.;
3. X+ TO Z-	X1. Z-1.; G01 X2. R-0.1; Z-2	X1. Z-1.; G01 X1.8; G03 X2. Z-1.1 R0.1; G01 Z-2.;
4. X+ TO Z+	X1. Z-2.; G01 X2. R0.1; Z-1.;	X1. Z-2.; G01 X1.8; G02 X2. Z-1.9 R0.1; G01 Z-1.;

X3. Z-1.   X3. Z-2.

1   2   R=0.1

X2. Z-2.

X2. Z-1

X2. Z-2.

3   4

X1. Z-1   X1. Z-2
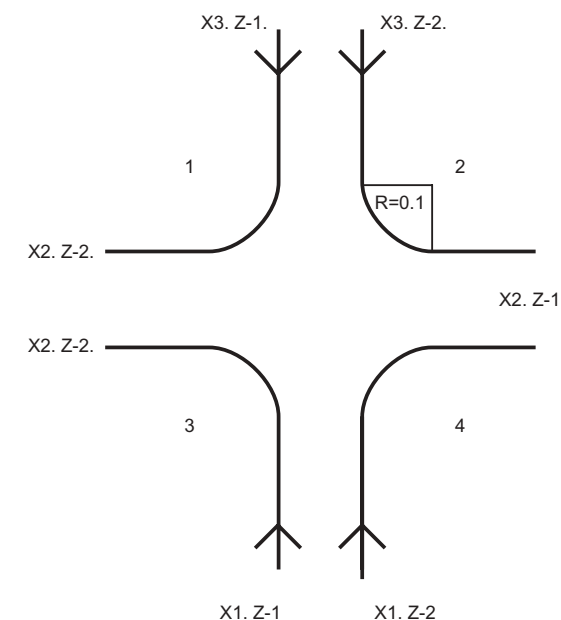
Rules:
1) Use K address only with X(U) address. Use I address only with Z(W) address.
2) Use R address with either X(U) or Z(W), but not both in the same block.
3) Do not use I and K together on the same block. When using R address, do not use I or K.
4) The next block must be another single linear move 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

<div align="center">

WARNING!

All rules must be conformed to, otherwise the outcome will be unpredictable.

</div>

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 (maximum 7740 inches). These are further described, and shown on the following pages:
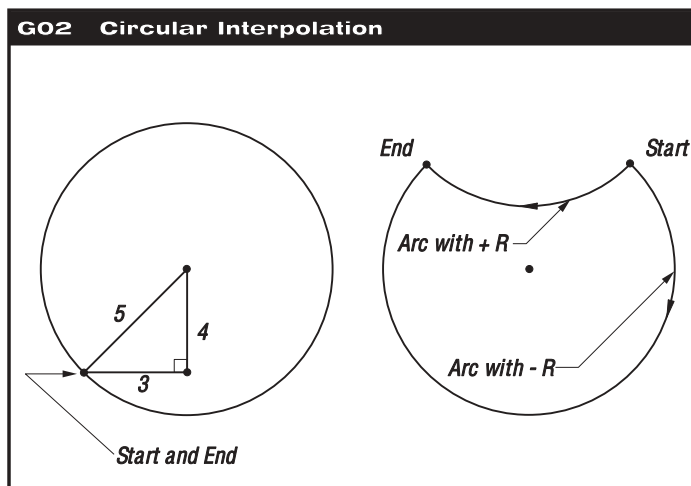
Figure 4.0-1

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.
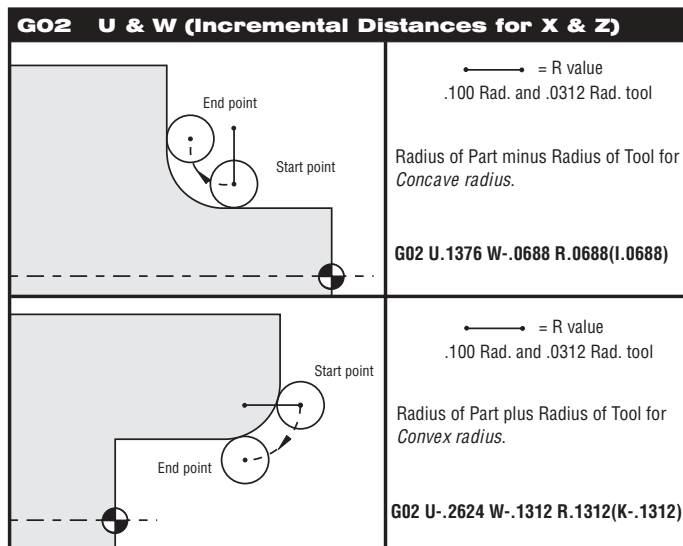
Fig. 4.0-2 G02

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.
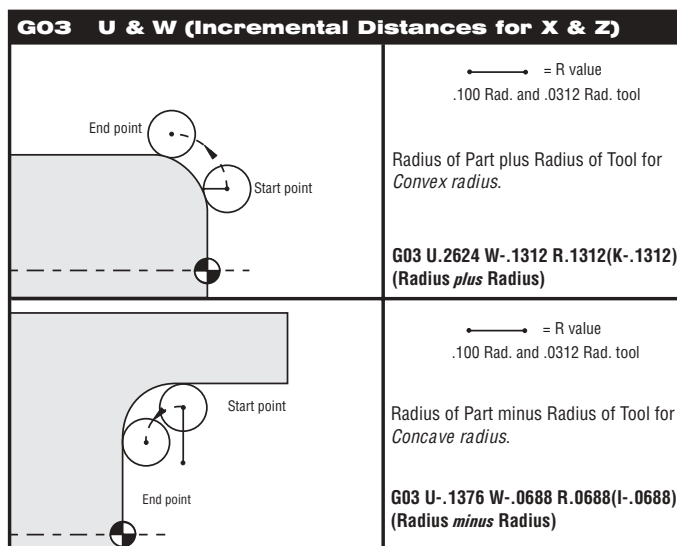
Fig. 4.0-3 G03

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 W6.0	(Move coordinate G54 6.0 units to the right);
G10 L20 P2 X–10.Z–8.	(Set work coordinate G111 to X–10.0, Z–8.0);
G10 L10 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.

G41/G42 Cutter Compensation for use with G112
Mill-style Cutter Compensation becomes active when G112 is used. Note that G41/G42 must be canceled before the next G113 (Cancel Cartesian to Polar transformation) or alarm 455 will be generated.

G113   G112 Cancel   Group 04

G113 cancels the Cartesian to Polar coordinate conversion.

G112 / G113 Program Example

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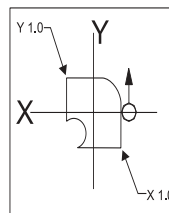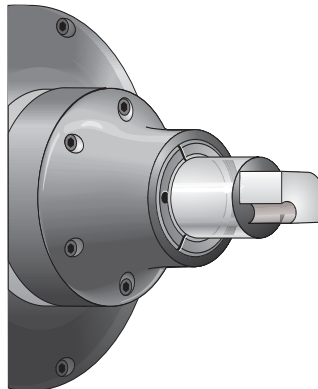
CIRCULAR PLANE SELECTION (G18)

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

G 20   Select Inch   Group 06
G 21   Select Metric   Group 06

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 ignores tool length offsets in this move.

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
Q   Optional Thread Angle
U   X-axis incremental positioning command
W   Z-axis incremental positioning command
X   X-axis absolute positioning command
Z   Z-axis absolute positioning command

NOTES: • 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.

G32 Thread Cutting

Lz = Lead along Z axis

Lx = Lead along X axis
(Radius value)

Straight Threads   Tapered Threads
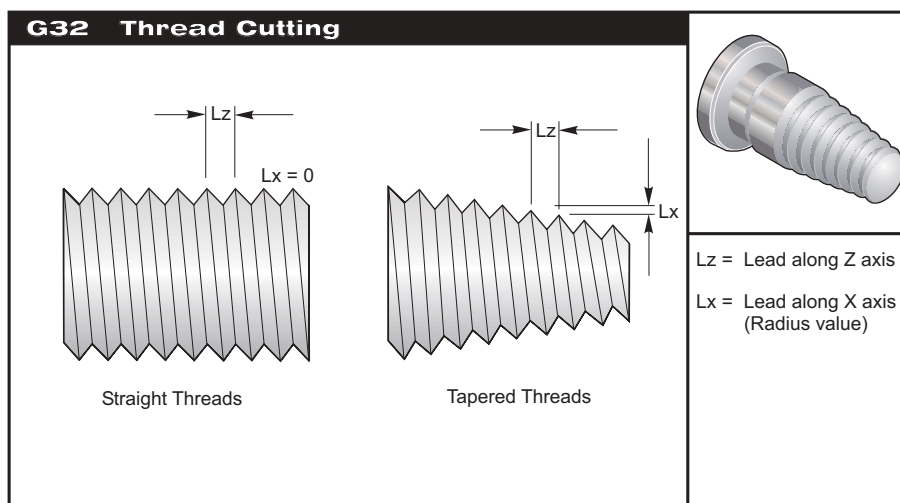
Figure 4.0-4. Definition of Lead (Feed Rate) for Straight and Tapered Threads

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 willalways 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)
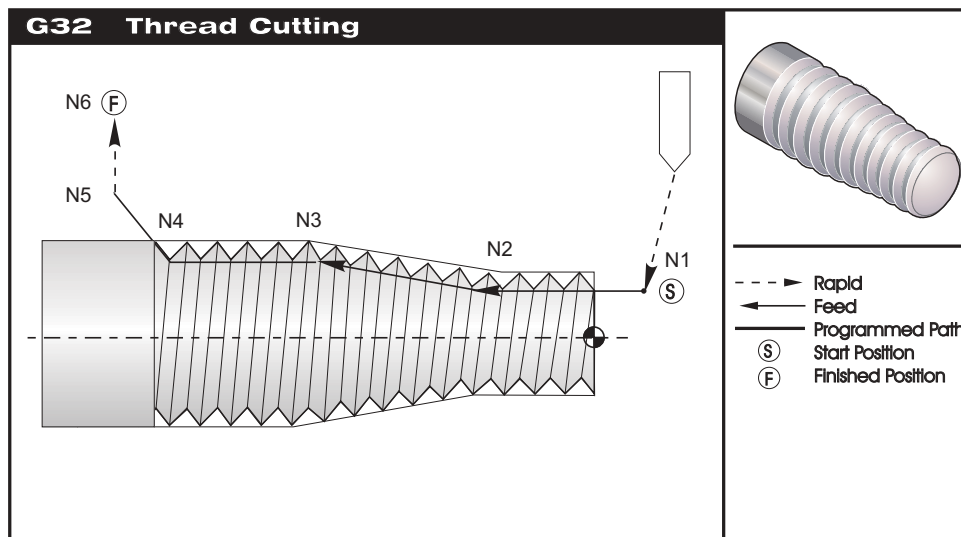
G32   Thread Cutting

Figure 4.0-5. Straight to Taper, Taper to Straight Thread Cutting Cycle

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

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

Q option example:

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

The following rules apply to the usage of Q:

1. The start angle Q is not a modal value. It must be specified every time it is used. If no value is specified then a zero (0) angle is assumed.

2. The angle of threading increment is 0.001 degrees and it cannot have any decimal point. So a 180 degree must be specified as Q180000 and an angle of 35 degrees as Q35000.

3. The Q angle must be entered as a positive value from 0 to 360000. If a value greater than 360 degrees is entered then a modulus of 360 is applied to that value.

G40   Tool Nose Compensation Cancel   Group 07

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

* indicates optional

G40 cancels G41 or G42. Programming Txx00 will also cancel tool nose compensation. 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.
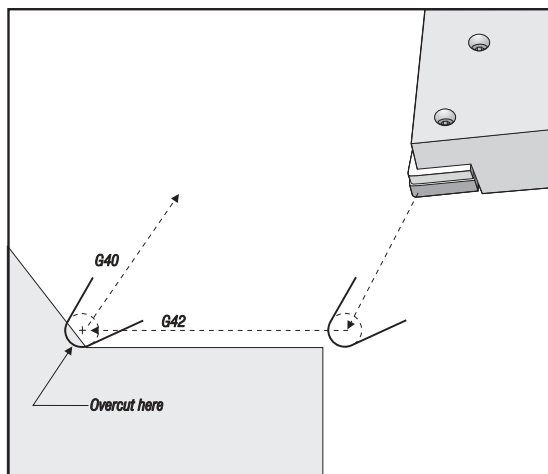
G40

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. The following figure illustrates where I and K lie in relation to the departure stroke. Usually I and K lie along a face of the machined part.
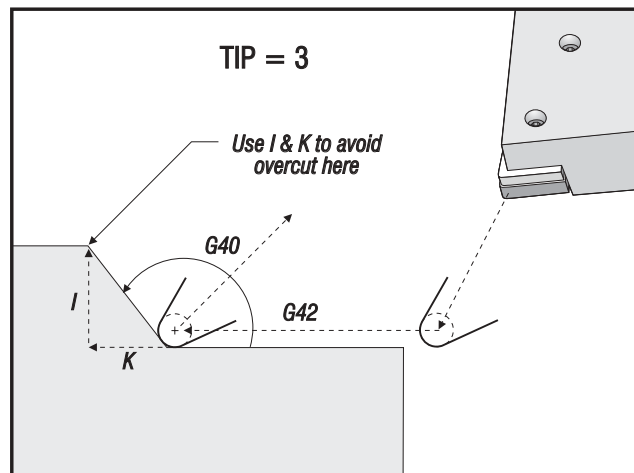
Use of I and K.

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.

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. A tool offset must be selected with a Tnnxx code, where xx corresponds to the offsets that are to be used with the tool.
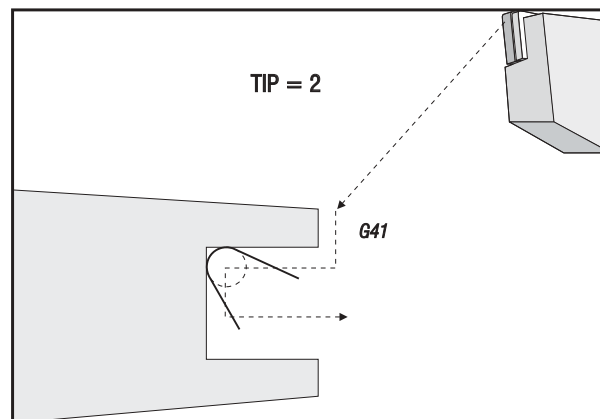
G41

G42 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. A tool offset must be selected with a Tnnxx code, where xx corresponds to the offsets that are to be used with the tool.

TIP = 3

G42

G42
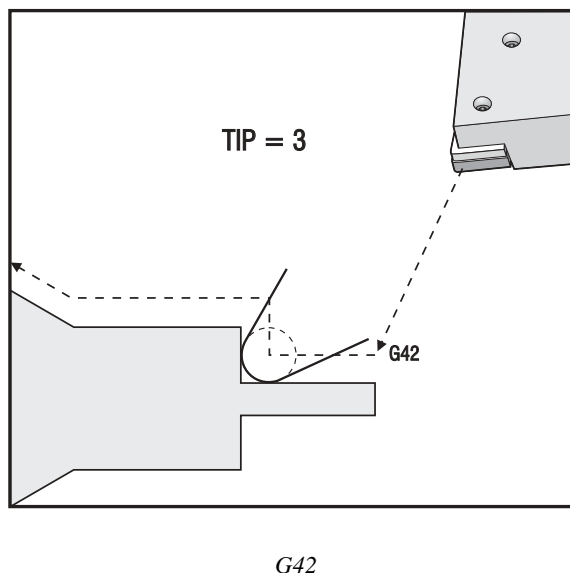
SETTING A GLOBAL COORDINATE (G50)

G50   SET Global coordinate Offset FANUC, YASNAC   Group 00

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

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

To set the global coordinate, 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.
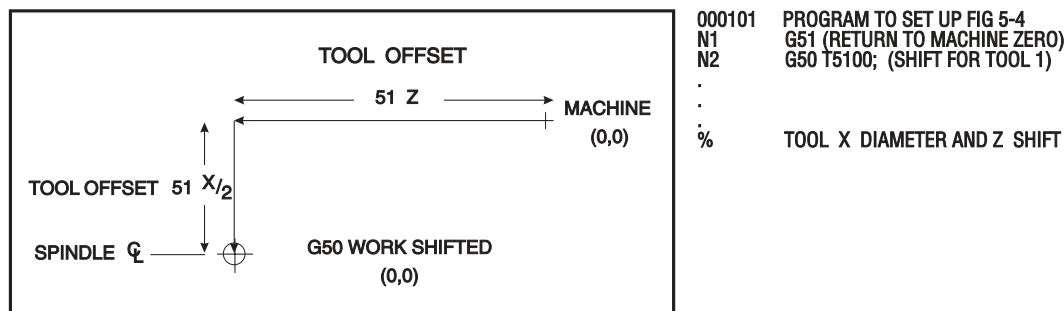
Fig. 4.0-6 G50 YASNAC Tool Shift

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) |

Note: To undo this command, use another G50 and specify the maximum allowed spindle RPM for the macine (see parameter 131).

G51   Cancel Offset (YASNAC)   Group 00

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 15

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 15

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 (modal)	Group 01
G92 Thread Cutting Cycle (modal)	Group 01
G94 End Face Cutting Cycle (modal)	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
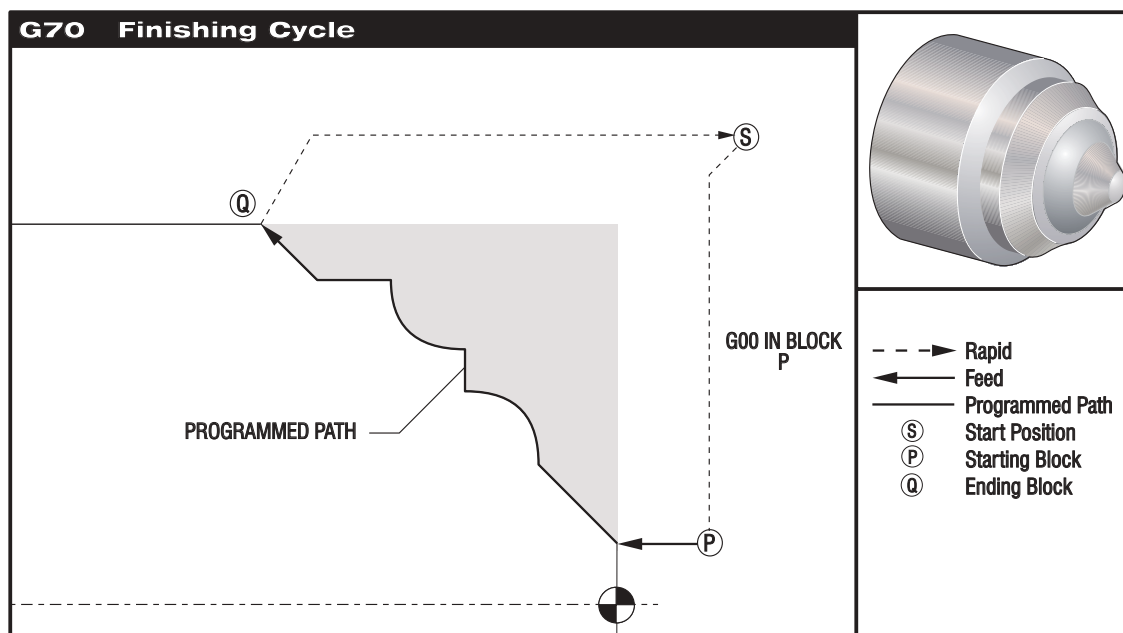
Fig. 4.0-7 G70.

A traditional calling sequence using G70 appears as below.

       G71 P10 Q50 F.012 (rough out N10 to N50 the path)
       N10
       F0.014
       …
       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 routing simply by specifiing 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
     Q   Ending Block number of path to rough
  * S   Spindle speed to use throughout G71 PQ block
  * T   Tool and offset to use throughout G71 PQ block
  * U   X-axis size and direction of G71 finish allowance, diameter
  * W   Z-axis size and direction of G71 finish allowance
  * R1   YASNAC select Type II roughing
* indicates optional
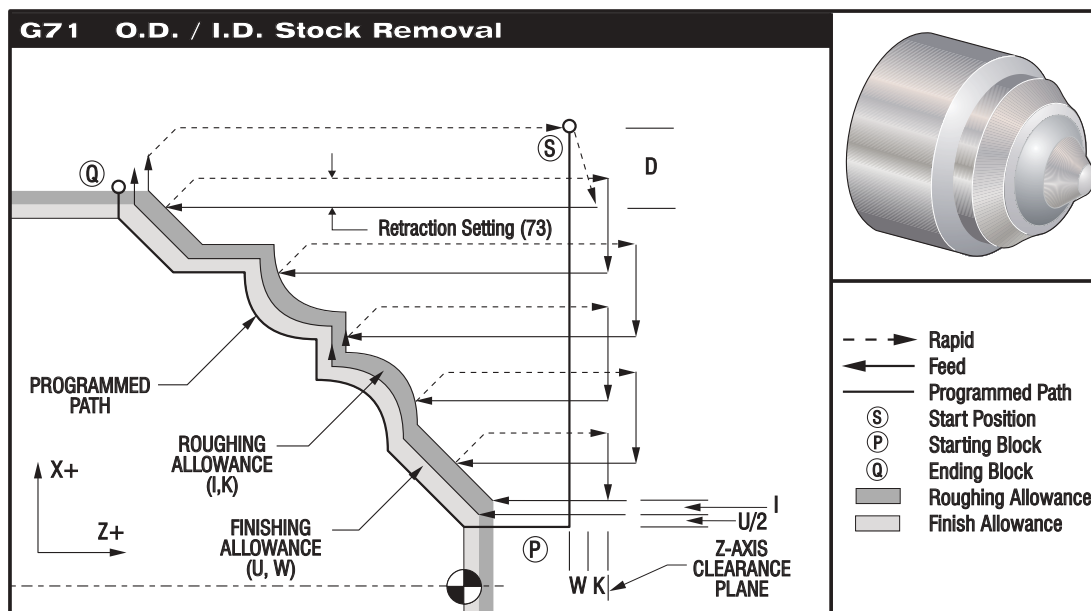
Fig. 4.0-8 G71

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 4.0-9. 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 4.0-14. 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 specifiing address codes D, I, K, U, and W properly. Figure 4.0-8 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.
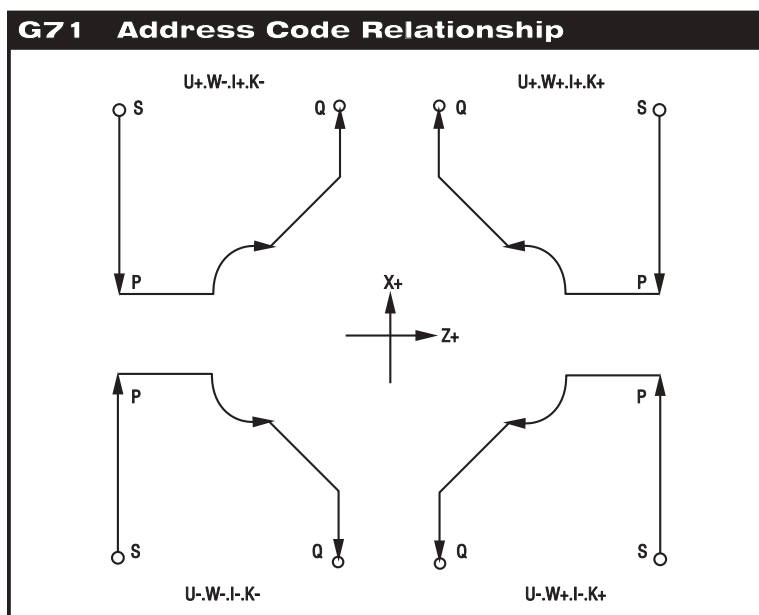
Fig. 4.0-9 G71 Address relationships

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.

G71 Type II Troughs on the Same Level

4 TROUGHS, EACH WITH
1 LEVEL OF NESTING
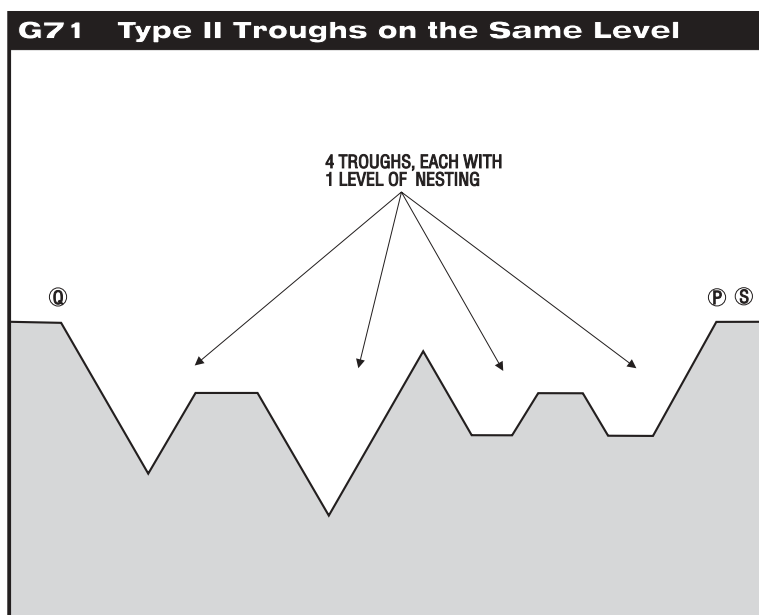
Fig.4.0-10 Troughs on the same level.

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 4.0-10 and 4.0-11 illustrate the troughing concept.

G71   Type II Nested Troughs

TROUGHS NEST 5 LEVELS DEEP
(LIMIT, 10 LEVELS)

2 TROUGHS NESTED
2 LEVELS DEEP
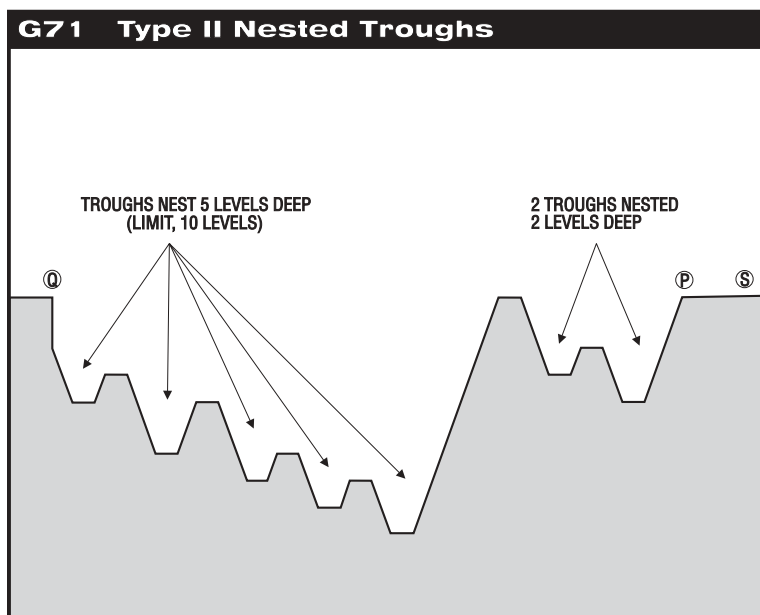
Fig. 4.0-11 Nested Troughs

Figure 4.0-12 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:

Sequences of Roughing Cuts for PQ Paths

Path Sequence for Type II Roughing

Region Sequence for Type II Roughing
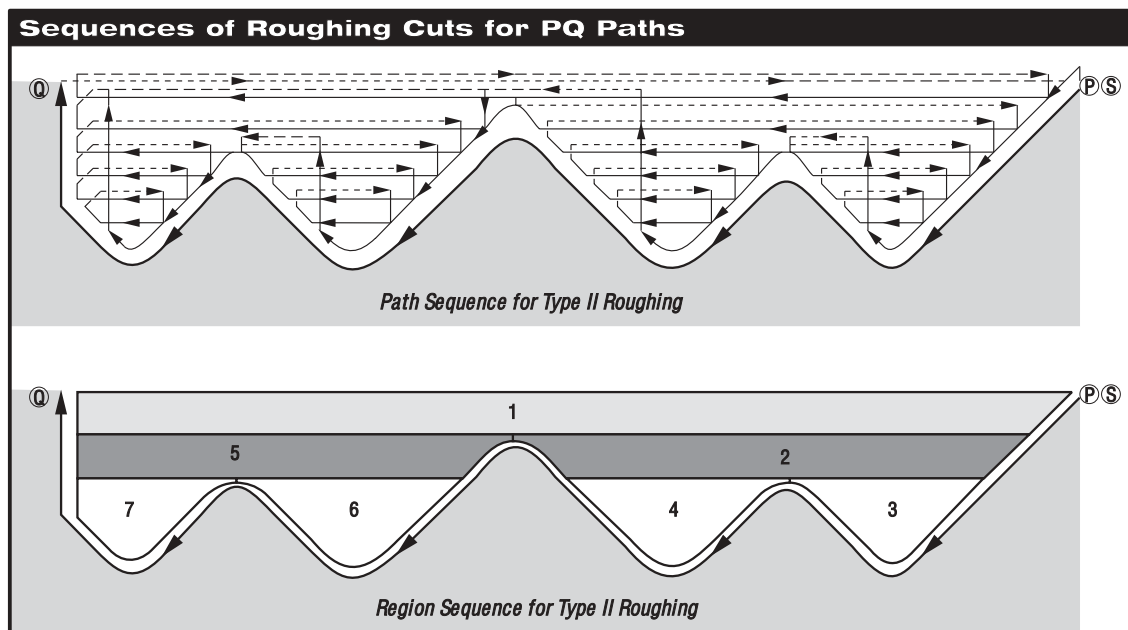
Fig. 4.0-12 Path sequence for Type II roughing

Tool Retraction

Tool Retraction for Type I

Tool Retraction for Type II

45°

SETTING 73

SETTING 73
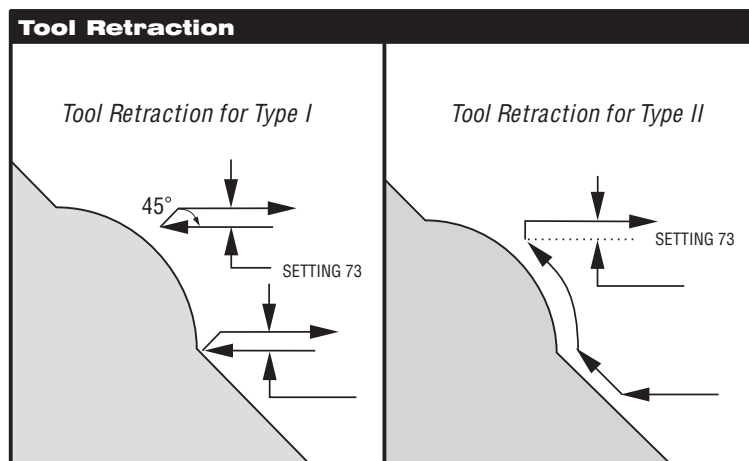
Fig. 4.0-13 Type I and II tool retraction

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.

G71 Basic G Code Example

- - - - →	Rapid
←	Feed
——	Programmed Path
Ⓢ	Start Position
Ⓟ	Starting Block
Ⓠ	Ending Block
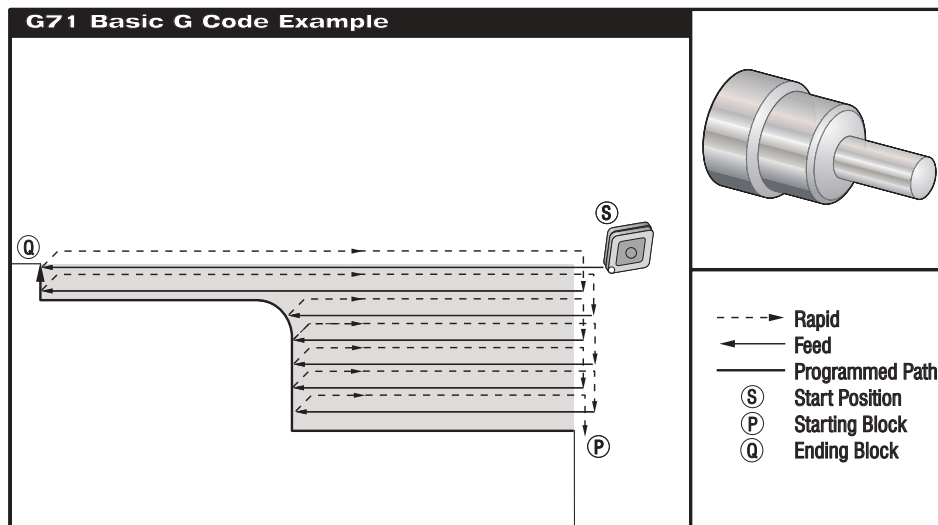
PROGRAM EXAMPLE

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

DESCRIPTION

(G71 Roughing Cycle)

(FINISH PASS)

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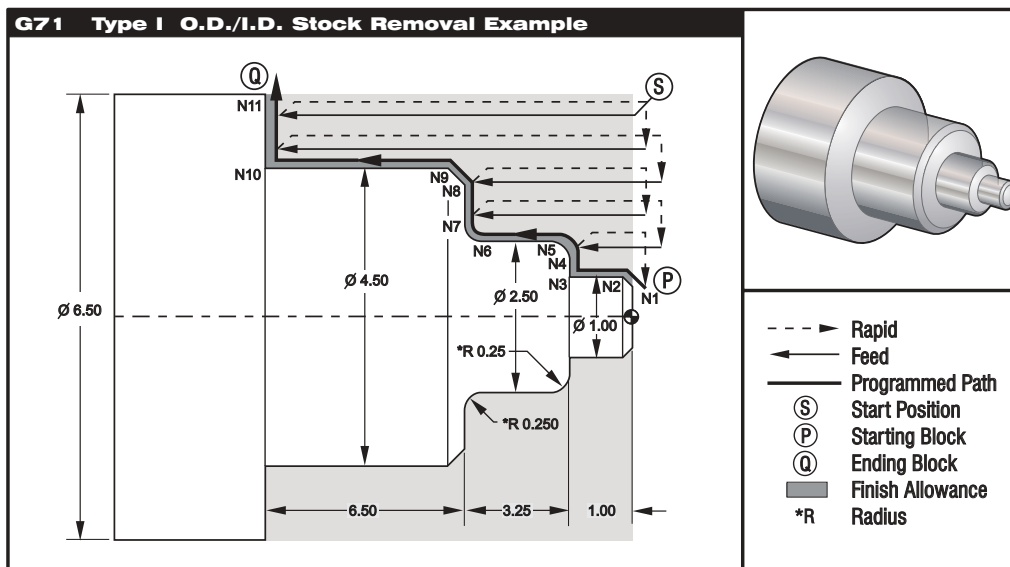
PROGRAM EXAMPLE

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

DESCRIPTION

(FANUC G71 TYPE I EXAMPLE)
(Tool change & apply Offsets)
(Rapid to Home Position)
(Set Max RPM 2000)
(Spindle On)
(Constant surface speed On)
(Define rough cycle)
P (Begin definition)
(Finish pass .004" Feed)

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

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

- – – → Rapid
- ← Feed
- — Programmed Path
- Ⓢ Start Position
- Ⓟ Starting Block
- Ⓠ Ending Block
- ▬ Finish Allowance
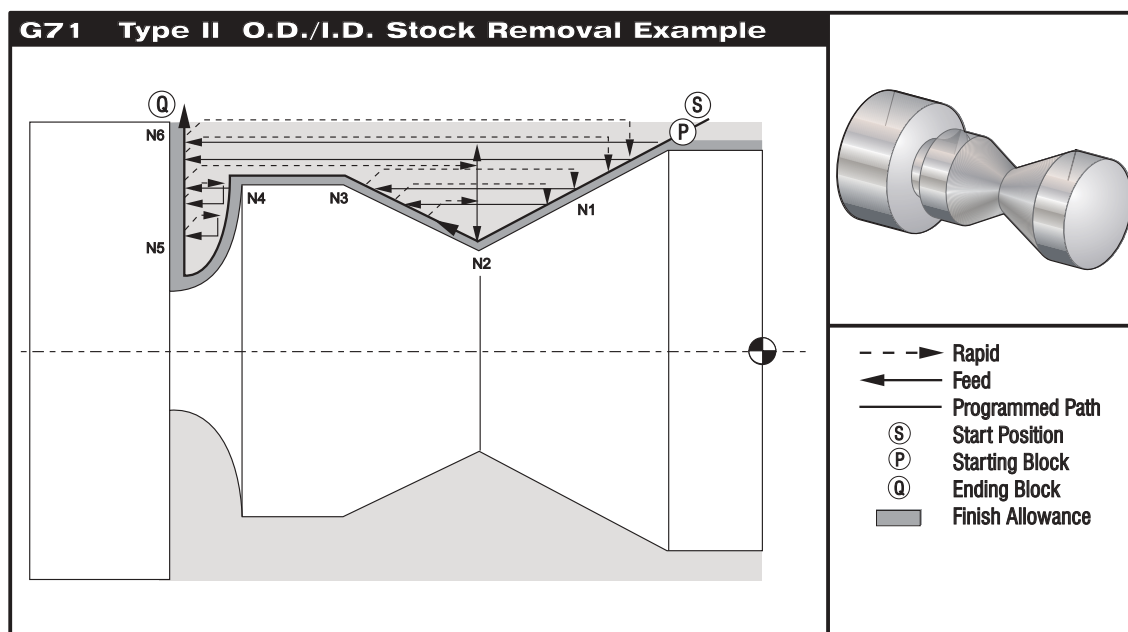
PROGRAM EXAMPLE

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

DESCRIPTION

(YASNAC G71 TYPE II EXAMPLE)
(Roughing tool)

S *(Start position)*

P (PQ Path definition)

Q *(PQ Path end)*

(Finishing tool)

(Finish pass))

G72 Basic G Code Example

Legend:
- - - ► Rapid
◄——— Feed
——— Programmed Path
Ⓢ Start Position
Ⓟ Starting Block
Ⓠ Ending Block

PROGRAM EXAMPLE
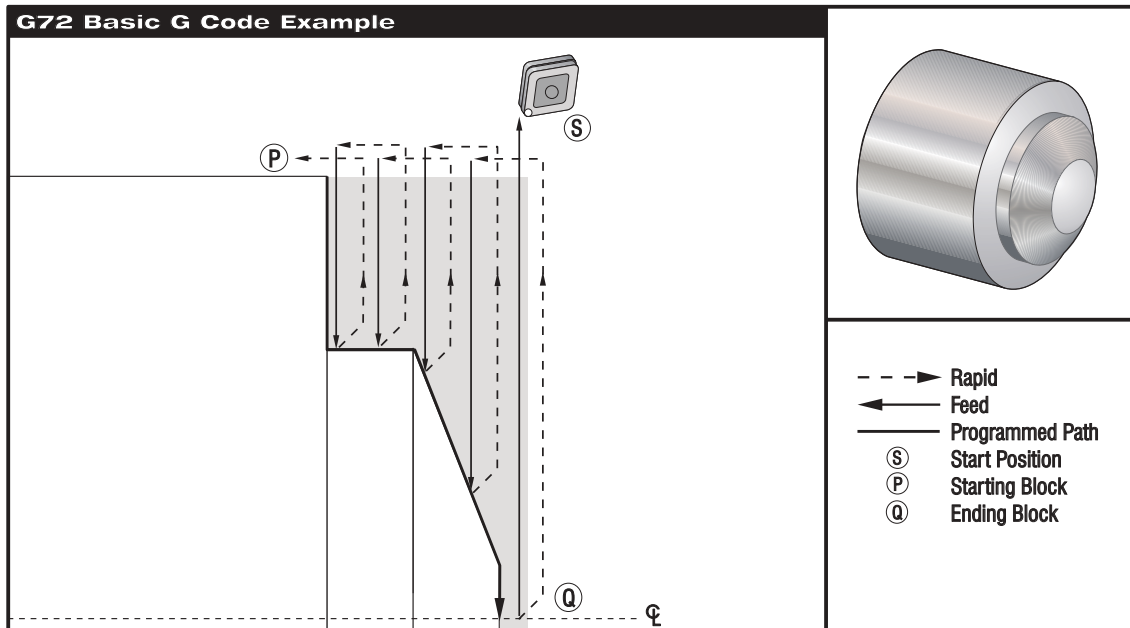
DESCRIPTION

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

G72   End Face Stock Removal Cycle   Group 00

* D   Depth of cut for each pass of stock removal, positive
* F   Feed rate to use throughout G72 PQ block
* I   X-axis size and direction of G72 rough pass allowance, radius
* K   Z-axis size and direction of G72 rough pass allowance
   P   Starting Block number of path to rough
   Q   Ending Block number of path to rough
* S   Spindle speed to use throughout G72 PQ block
* T   Tool and offset to use throughout G72 PQ block
* U   X-axis size and direction of G72 finish allowance, diameter
* W   Z-axis size and direction of G72 finish allowance
* indicates optional
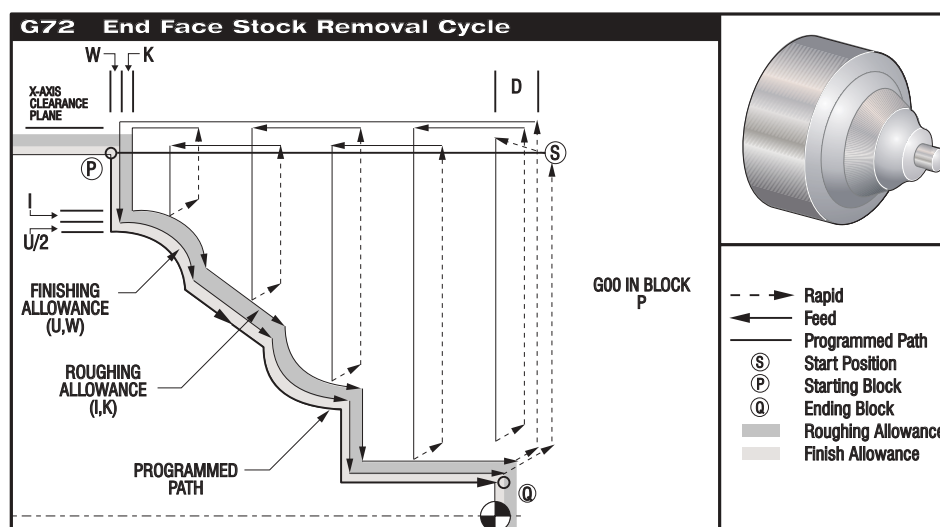
Fig. 4.0-14 G72

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 4.0-15. 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 4.0-16 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.

G72 Address Code Relationship

U+,W-,I+,K-   U+,W+,I+,K+

S   S

X+

Z+

S   S

U-,W-,I-,K-   U-,W+,I-,K+
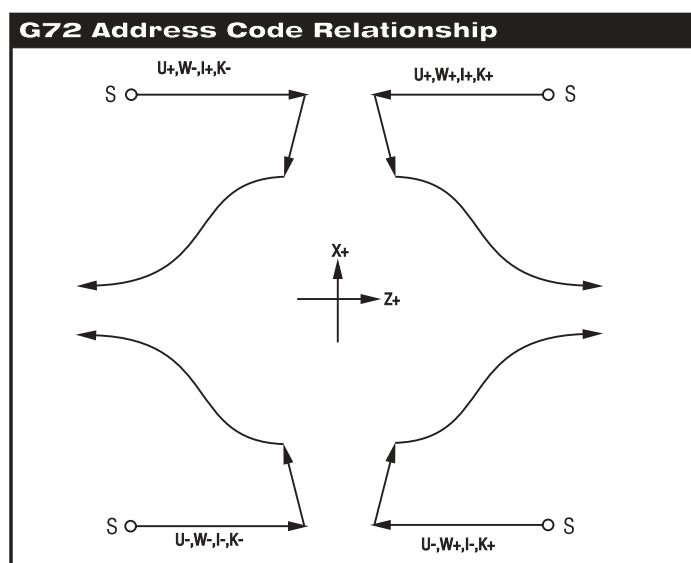
Fig. 4.0-15 G72 Address relationships

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.

G72 End Face Removal

PROGRAMMED PATH/(Q)

ROUGHING ALLOWANCE (I,K)

FINISHING ALLOWANCE (U, W)

X+

Z+

- - → Rapid
— Feed
— Programmed Path
(S) Start Position
(P) Starting Block
(Q) Ending Block
■ Roughing Allowance
▨ Finish Allowance
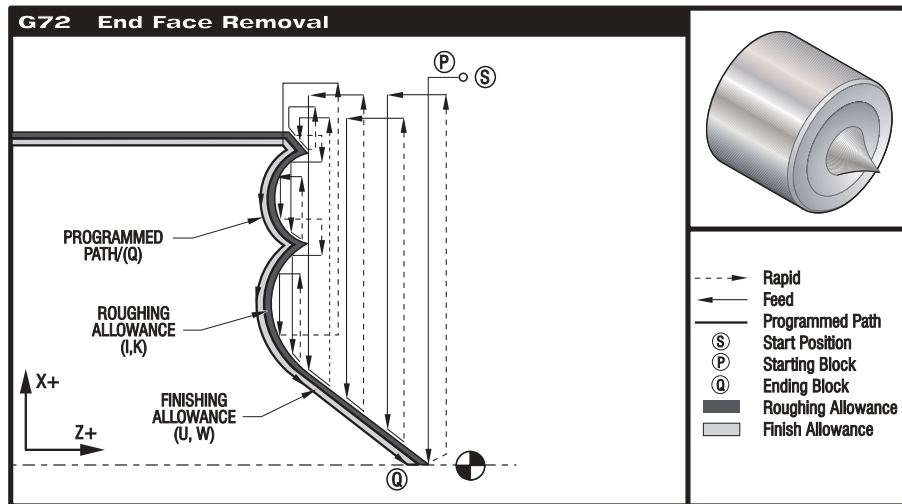
PROGRAM EXAMPLE

```
%
00722
G28
T101
S1000 M03
G00 G54 X2.1 Z0.1
G72 P1 Q2 D0.06 I0.02 K0.01 U0.02 W0.01 S1100 F0.015
N1 G01 Z−0.46 X2.1 F0.005
X2.
G03 X1.9 Z−0.45 R0.2
G01 X1.75 Z−0.4
G02 X1.65 Z−.4 R0.06
G01 X1.5 Z−0.45
G03 X1.3 Z−0.45 R0.12
G01 X1.17 Z−0.41
G02 X1.03 Z−0.41 R0.1
G01 X0.9 Z−0.45
G03 X0.42 Z−0.45 R0.19
G03 X0.2 Z−0.3 R0.38
N2 G01 X0.01 Z0
G70 P1 Q2
M05
G28
M30
%
```

DESCRIPTION

(*G72 ROUGHING CYCLE*)

(*FINISH PASS*)

G73   Irregular Path Stock Removal Cycle   Group 00

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

* indicates optional
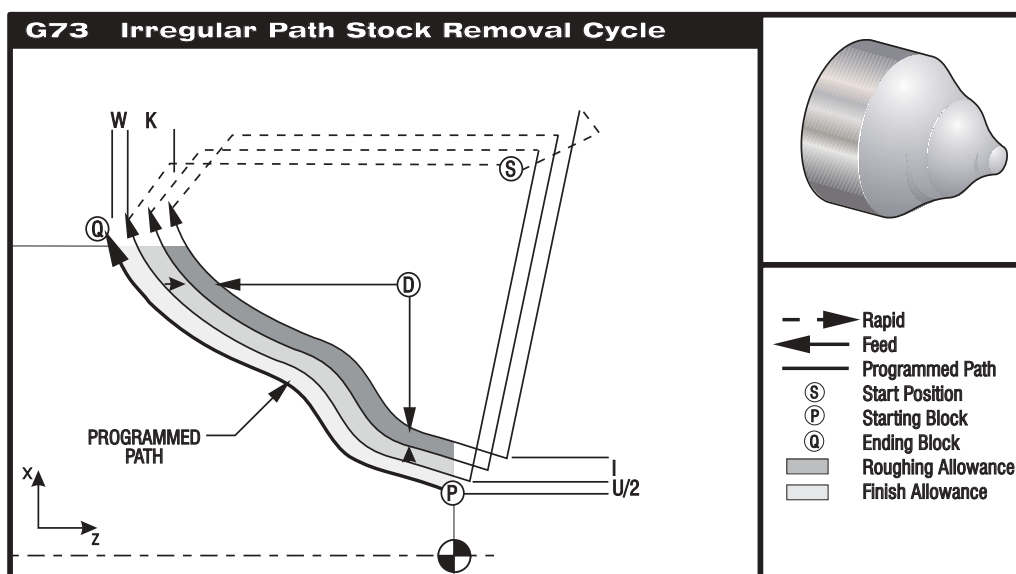
Fig. 4.0-16 G73

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 4.0-16.

```
XAXIS   ZAXIS   TOOL APPROACHES FROM THIS QUADRANT.

U I     W K
+ +     + +      I
+ +     - -      II
- -     - -      III
- -     + +      IV
```

G74   End Face Grooving Cycle, Peck Drilling   Group 00

* D   Tool clearance when returning to starting plane, positive
* F   Feed rate
* I   X-axis size of increment between peck cycles, positive radius
  K   Z-axis size of increment between pecks in a cycle
* U   X-axis incremental distance to furthest peck, 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
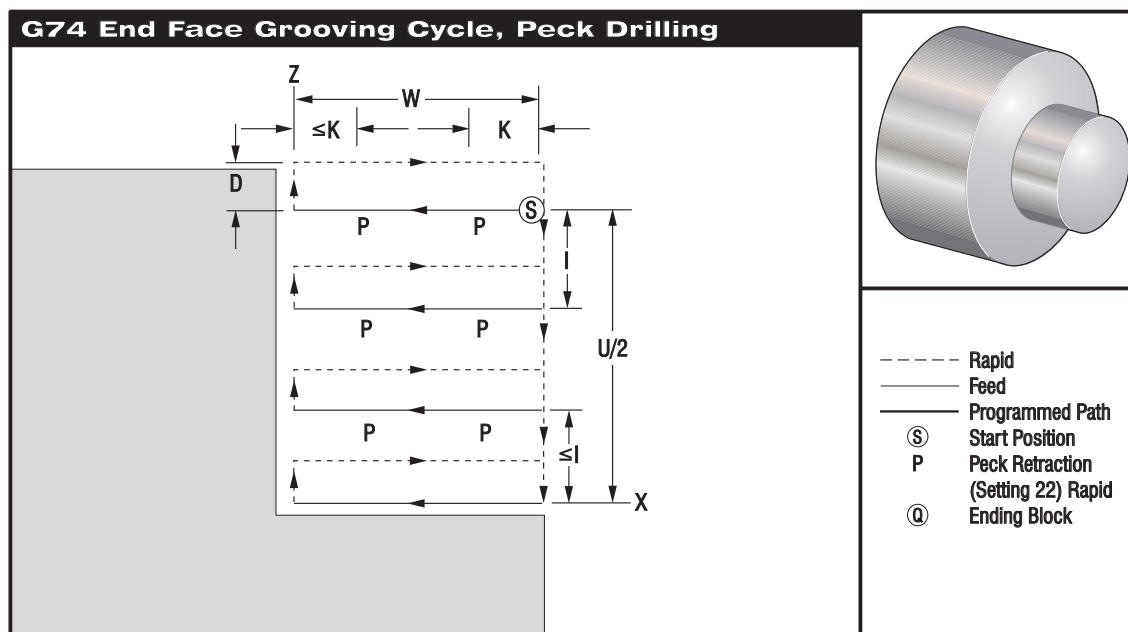
Fig. 4.0-17 G74

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.

G74 End Face Grooving Cycle (Single Pass) Example
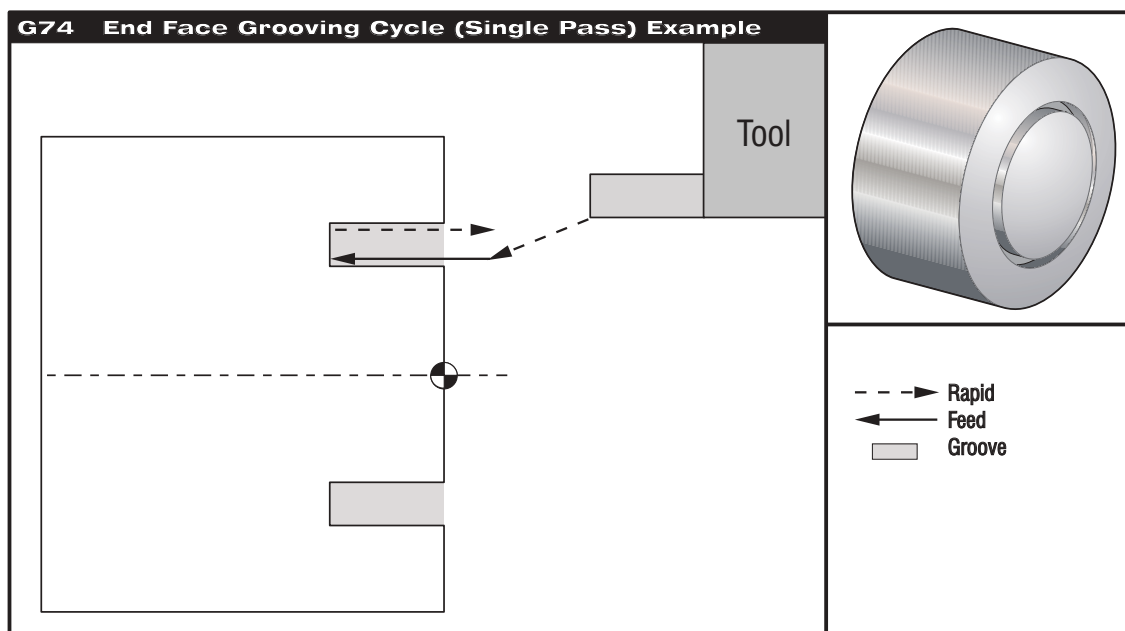
- - - → Rapid
— ← Feed
▭ Groove

PROGRAM EXAMPLE

%
O0071
T101
G97 S750 M03
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   End Face Grooving Cycle (Multiple Pass) Example

Rapid
Feed
Programmed Path
Groove
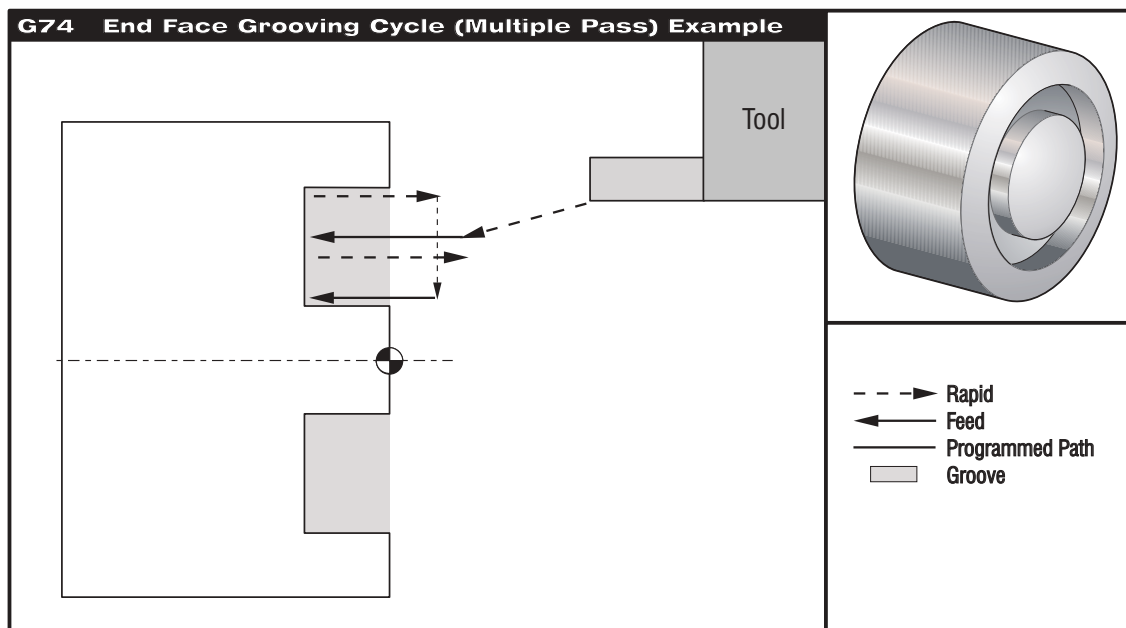
PROGRAM EXAMPLE

```
%
O0074
T101
G97 S750 M03
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   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.
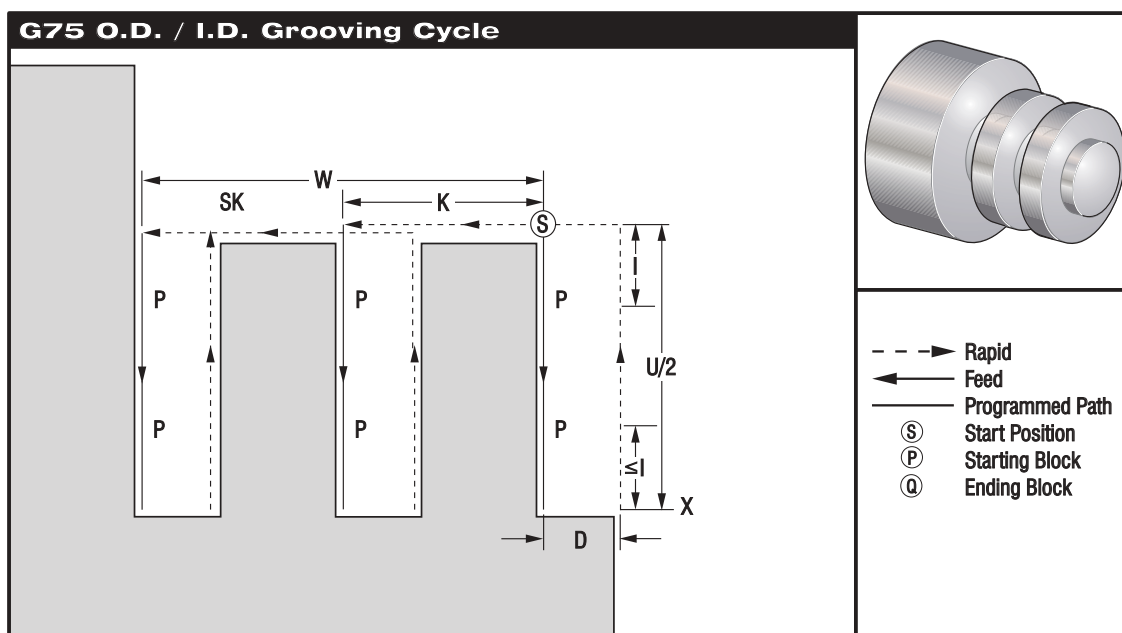
Fig. 4.0-18 G75

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.

G75   O.D. / I.D. Grooving Cycle (Single Pass) Example

Tool
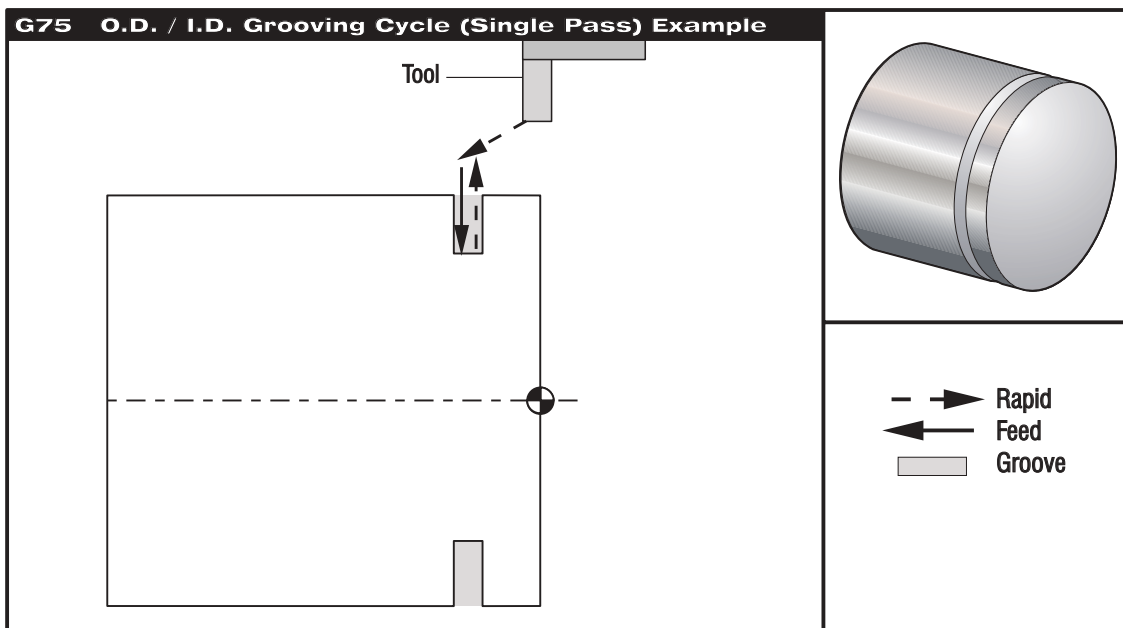
Rapid
Feed
Groove

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

DESCRIPTION

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

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

G75   O.D. / I.D. Grooving Cycle (Multiple Pass) Example

Tool
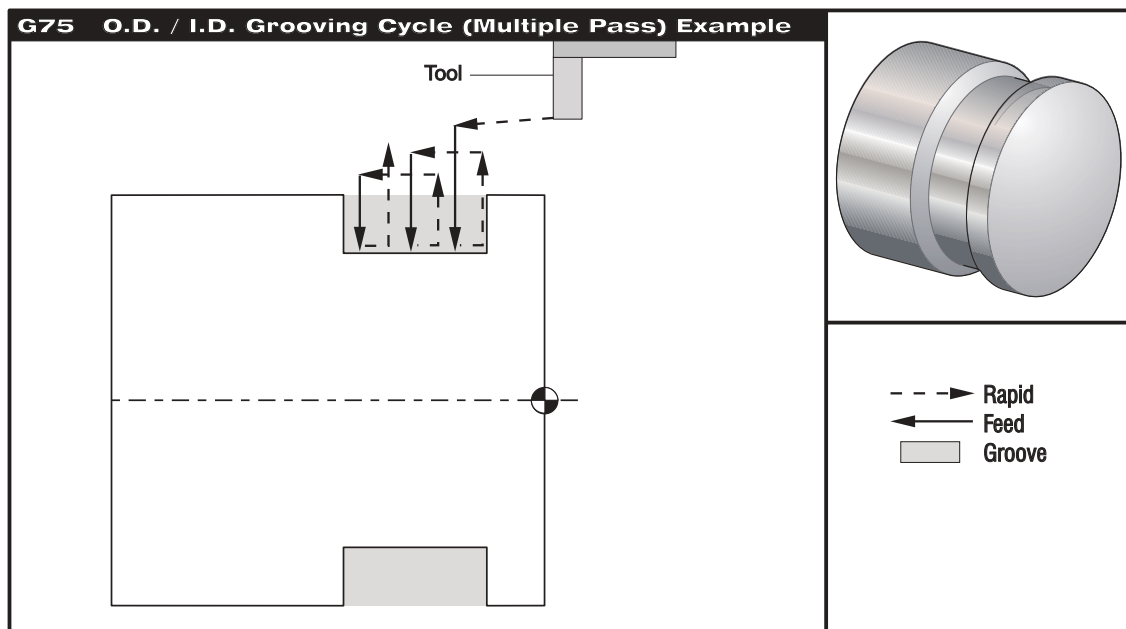
Rapid
Feed
Groove

PROGRAM EXAMPLE

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

DESCRIPTION

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

G76　　Threading Cycle, Multiple Pass　　　　　　　　　　Group 00

* A　Tool nose angle
  D　First pass cutting depth
  F(E)　Feed rate, the lead of the thread
* I　Thread taper amount, radius measure
  K　Thread height, defines limit of multiple passes, radius measure
  Q　Optional Thread Angle
* U　X-axis incremental distance, start to maximum thread Depth Diameter
* W　Z-axis incremental distance, start to maximum thread length
* X　X-axis absolute location, maximum thread Depth Diameter
* Z　Z-axis absolute location, maximum thread length
* indicates optional

Settings 95 / 96 determine chamfer size / angle; M23 / 24 turn chamfering on / off.
See figure 4.0-19.5
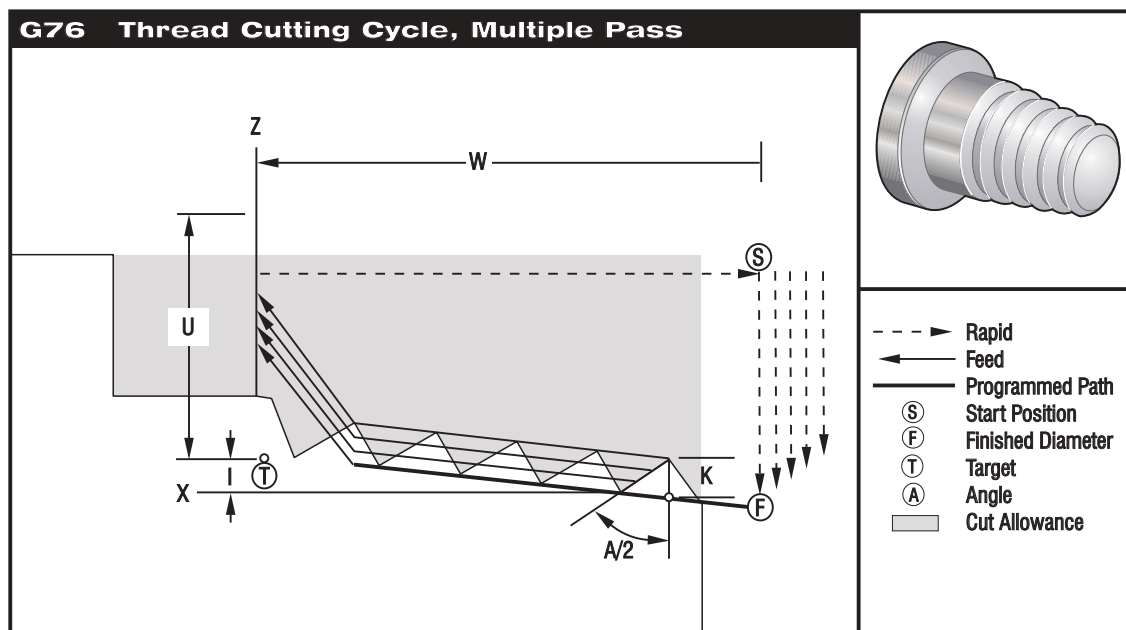
Fig. 4.0-19 G76

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 (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. See figure 4.0-19.5.

Figure 4.0-19.5

G76   Cutting Methods

P1: SINGLE EDGE CUTTING CONSTANT

THREAD MINIMUM
CUT SETTING 99

CUTTING TIP

A   B

D

1
2
3
N

$D_S \overline{N}$ K

MATERIAL

FINISH ALLOWANCE
SETTING 86

Fig. 4.0-20 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: The default is P1. Future implementations are:
P2 TWO EDGES CUTTING, CUTTING LOAD CONSTANT
P3 SINGLE EDGE CUTTING, CUTTING DEPTH CONSTANT
P4 TWO EDGES CUTTING, CUTTING DEPTH CONSTANT

G76 Thread Cutting Cycle, Multiple Pass

- - - ▶ Rapid
◀──── Feed
───── Programmed Path
Start Position
Starting Block
Ending Block
Finish Allowance
*R   Note: Radius was calculated using Tool Nose Radius Compensation

PROGRAM EXAMPLE

```
%
T101
G50 S2500
G97 S1480 M03
G54 G00 X3.1 Z0.5  M08

G96 S1200
G01 Z0 F0.01
X–0.04
G00 X3.1 Z0.5
G71P1 Q10 U0.035 W0.005 D0.125 F0.015
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 M09
G28
N20


T505
G50 S2000
G97 S1200 M03
G00 X1.5 Z0.5
```

DESCRIPTION

(Set max RPM select tool geometry)
(Spindle on select tool one offset one)
(Select work coord. and 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 Z0.3
G76 X0.913 Z−0.85 K0.042 D0.0115 F0.0714          (Threading cycle)
G00X1.5 Z0.5 G28 M09
N30                                               (HAAS SL−Series FANUC System)
T404
G50 S2500
G97 S1200 M03                                     (Groove tool)
G54 G00 X1.625 Z0.5 M08
G96 S800
G01 Z−1.906 F0.012
X1.47 F0.006
X1.51
W0.035
G01 W−0.035 U−0.07
G00 X1.51
W−0.035
G01 W0.035 U−0.07
X1.125
G01 X1.51
G00 X3. Z0.5 M09
G28
M30
%
```

Example using the Q

G76 X1.92 Z-2. Q60000 F0.2 D0.01 K0.04 (60 degree cut)
G76 X1.92 Z-2. Q120000 F0.2 D0.01 K0.04 (120 degree cut)
G76 X1.92 Z-2. Q127123 F0.2 D0.01 K0.04 (270.123 degree cut)
The following rules apply to the usage of Q:
1. The start angle Q is not a modal value. It must be specified every time it is used. If no value is specified then a zero (0) angle is assumed.
2. The angle of threading increment is 0.001 degrees and it cannot have any decimal point. So a 180 degree must be specified as Q180000 and an angle of 35 degrees as Q35000.
3. The Q angle must be entered as a positive value from 0 to 360000. If a value greater than 360 degrees is entered then a modulus of 360 is applied to that value.

Mutiple Start Threading Example

Mutliple threads can be cut using the CNC lathe. In order to do so the start point for each threading cycle must be changed.

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

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

Introduction

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

1. Positioning of the X and Z axes.
2. Rapid traverse to the reference R-plane
3. Drilling, boring or tapping action.
4. Operation at the bottom of the hole
5. Retraction to the reference R-plane, or starting plane.

Canned Cycle Planes
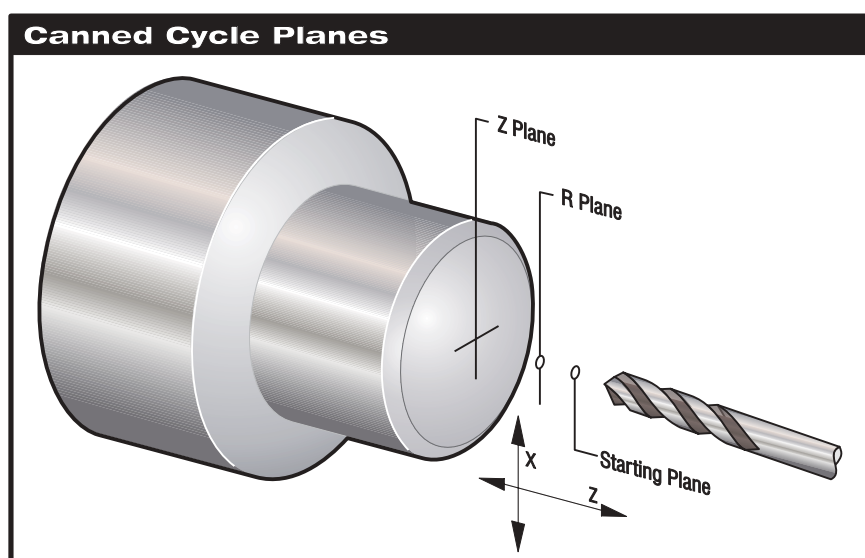
Canned Cycle Summary

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	dwell	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
G184	CCW	feed	spindle CW	feed	left-hand tapping cycle

Using Canned Cycles

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

Some of the canned cycle command values can also be changed after the canned cycle is defined. The command often changed during a canned cycle is the Z-depth value.

The operation of a canned cycle will vary according to whether incremental (U,W) or absolute (X,Z) is specified.

If a loop count (Lnn code number) is defined with in the block, the canned cycle will repeat that many times with an incremental U or W move between each cycle.

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

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.

Canned cycles with Live Tooling

The canned cycles G81, G82, G83, G85, G89 can be used with the live tooling provided parameter 315 bit 1 NO SPIN CAN is set to 1. This parameter prevents the main spindle from turning during one of the above listed canned cycles. If this bit is set to 1, it is the user's responsibility to activate the appropriate spindle prior to performing the canned cycle, that is, older G-code programs which use these canned cycles must be checked to be sure they EXPLICITLY turn on the main spindle before running the canned cycles. Note that G86, G87 and G88 are not usable with live tooling. If the parameter bit, NO SPIN CAN, is set to zero, the canned cycles operate in the usual way by turning the main spindle.

Canceling a Canned Cycle

The G80 code is used to cancel a canned cycle; in addition, a G00 or G01 code will also cancel any active canned cycle.

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
* indicated optional
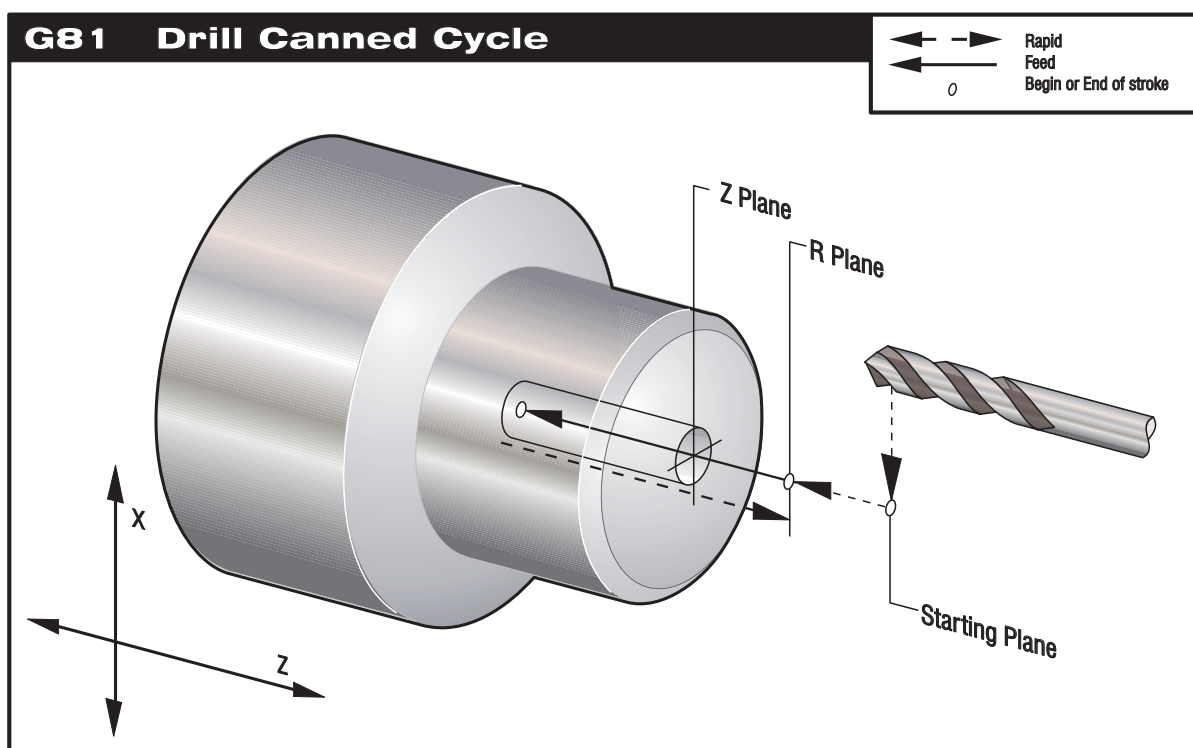
G81   Drill Canned Cycle

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.

G82   Spot Drill Canned Cycle

Rapid
Feed
Begin or End of stroke
Dwell

Z Plane
R Plane
X
Z
Starting Plane

G83   Normal Peck Drilling Canned Cycle   Group 09

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

* indicates optional

G83   Peck Drilling Canned Cycle
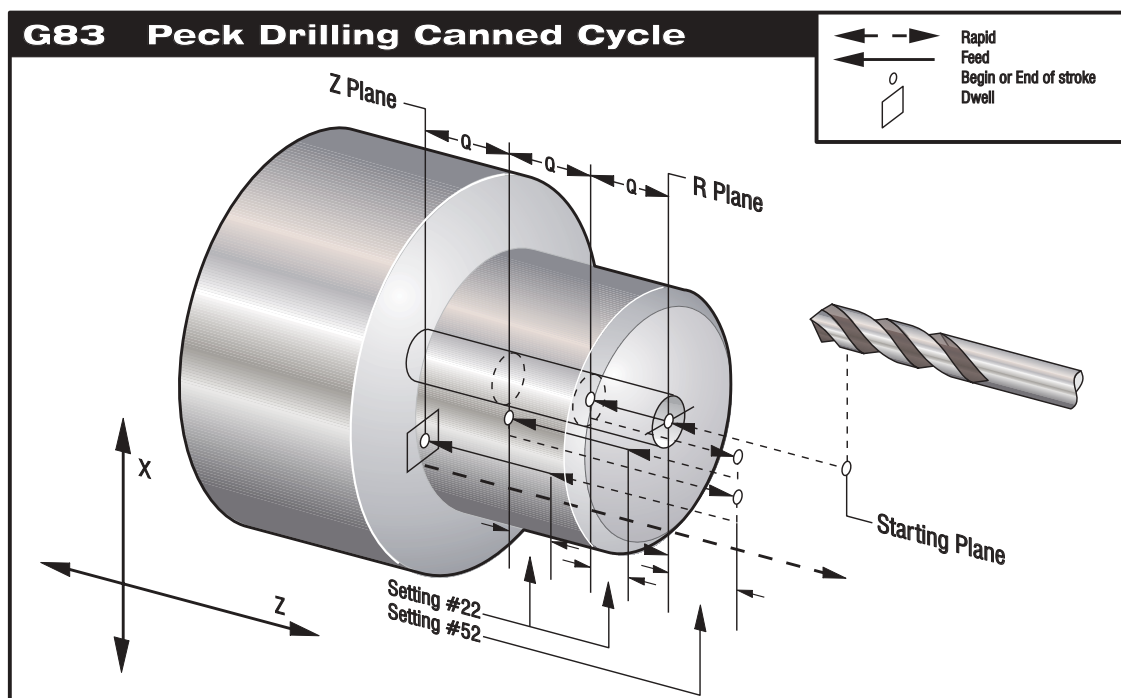
This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X will cause this canned cycle to be executed.

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

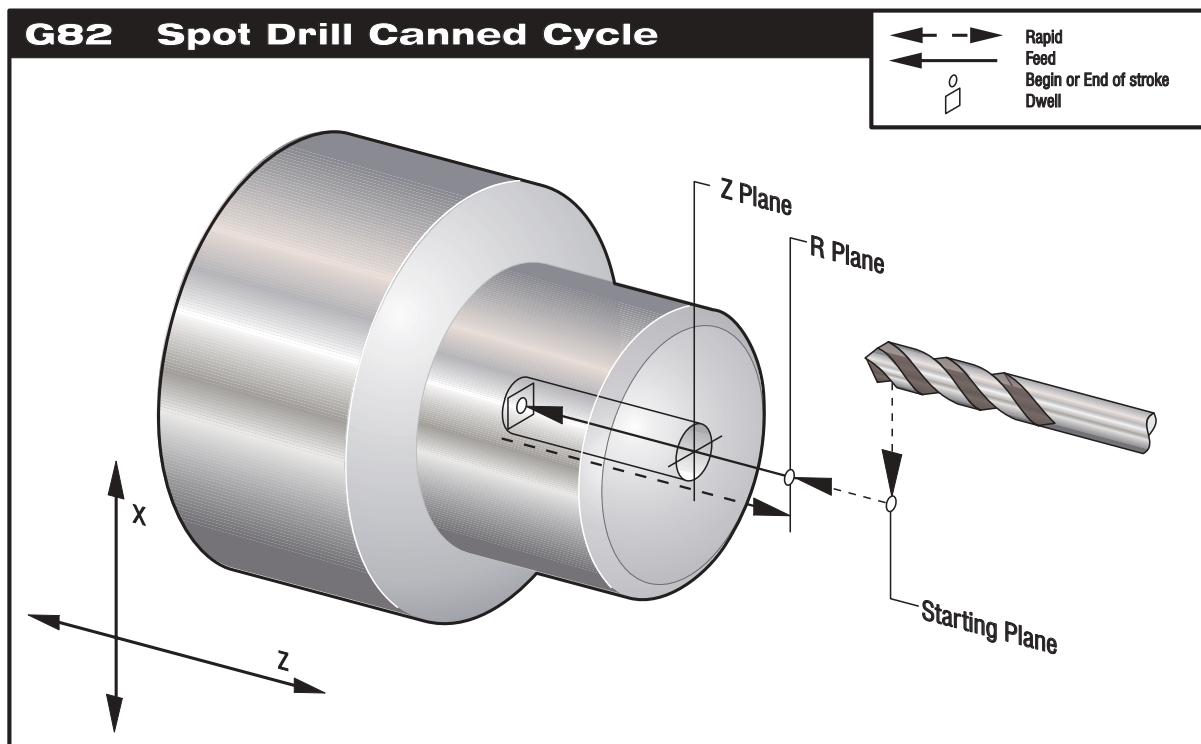|		
F	Feed Rate	
R	Position of the R plane	
*W	Z-axis incremental distance	
*X	X-axis motion command	
*Z	Position of bottom of hole	

* 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 |

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.
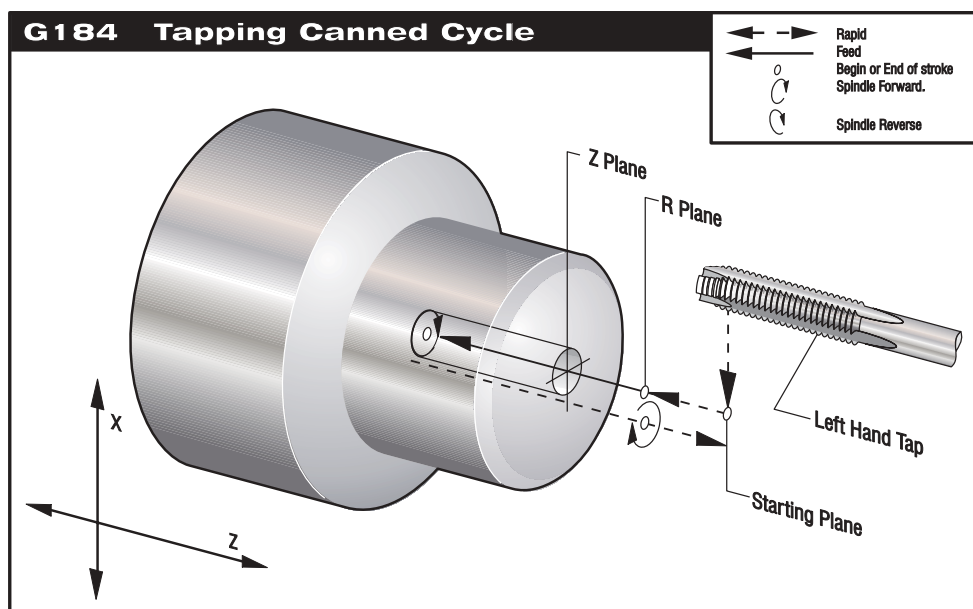
G184   Tapping Canned Cycle

G85   Boring Canned Cycle   Group 09

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

* indicates optional

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.

G85   Boring Canned Cycle

	Rapid
	Feed
0	Begin or End of stroke

Z Plane

R Plane

X

Z

Starting Plane
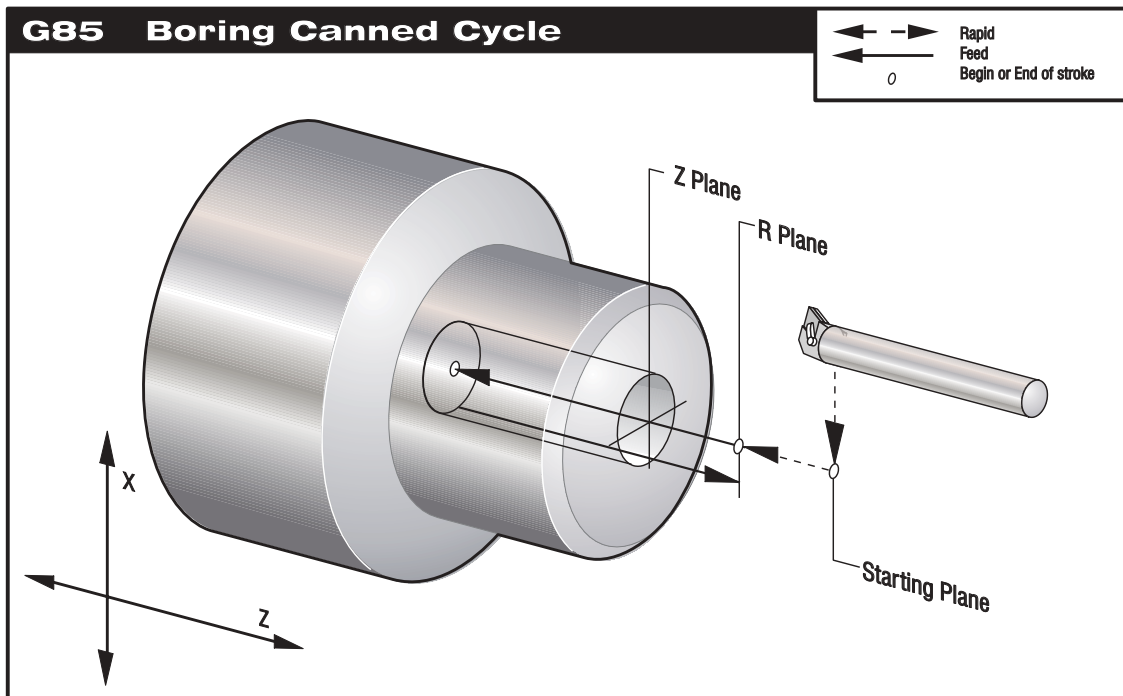
G86   Bore and Stop Canned Cycle   Group 09

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.

G86 Bore & Stop Canned Cycle

Rapid
Feed
Begin or End of stroke
Spindle Forward.
Spindle Stop

Z Plane

R Plane

X

Z

Starting Plane

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.
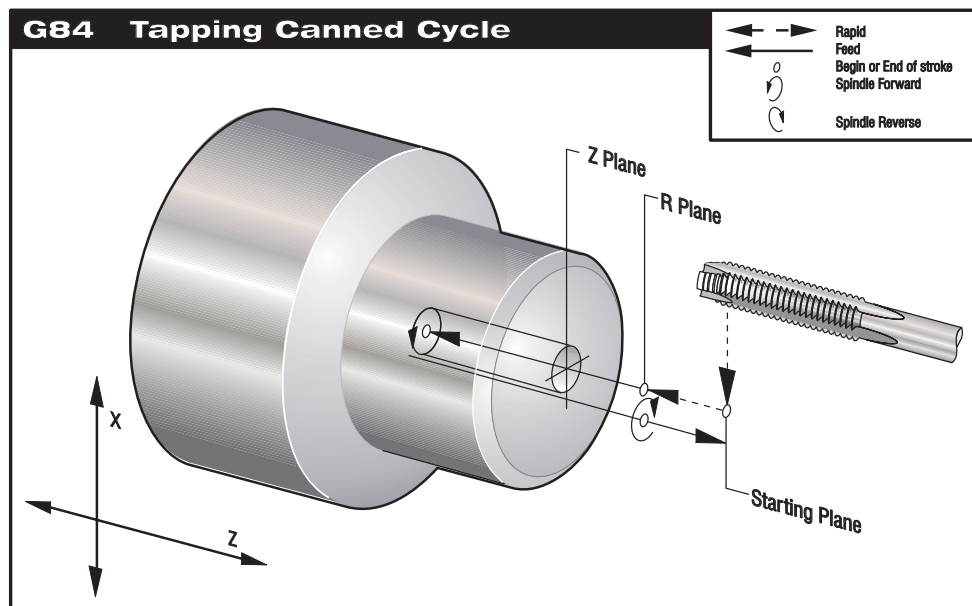
G87   Bore & Manual Canned Cycle

Rapid
Feed
Begin or End of stroke
Spindle Stop
Handle Jog

Z Plane

R Plane

X

Z

Starting Plane

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.

G88 Bore, Dwell & Manual Canned Cycle

Rapid
Feed
Begin or End of stroke
Dwell
Spindle Stop
Handle Jog

Z Plane

R Plane

X

Z

Starting Plane
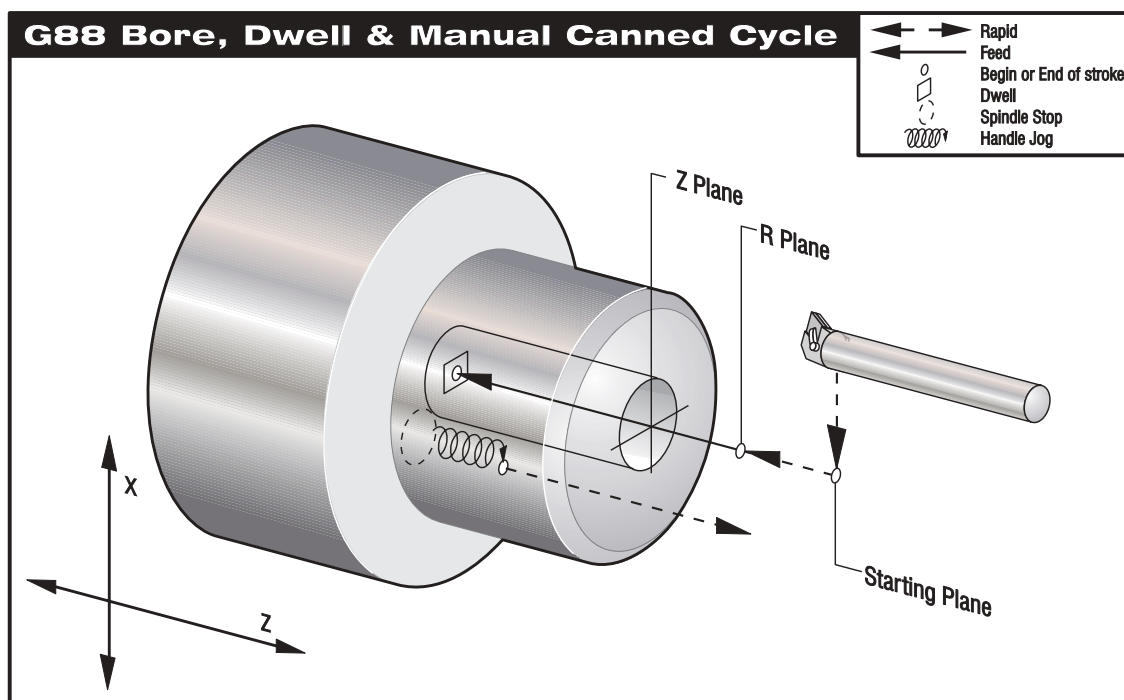
G89   Bore and Dwell Canned Cycle   Group 09

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.
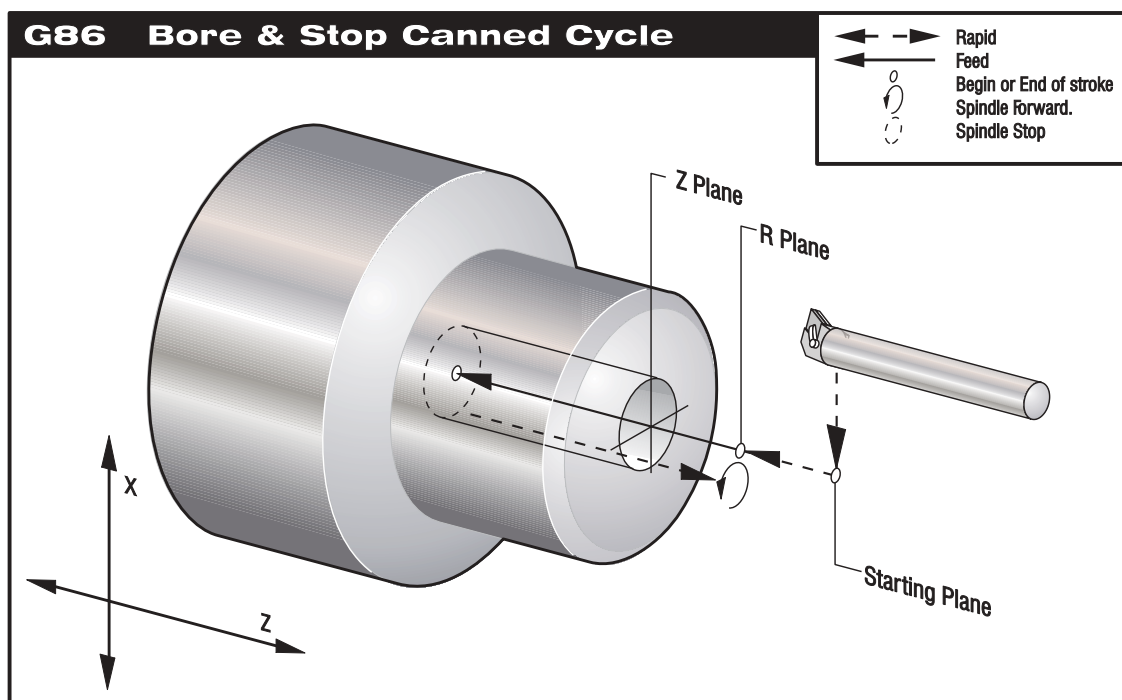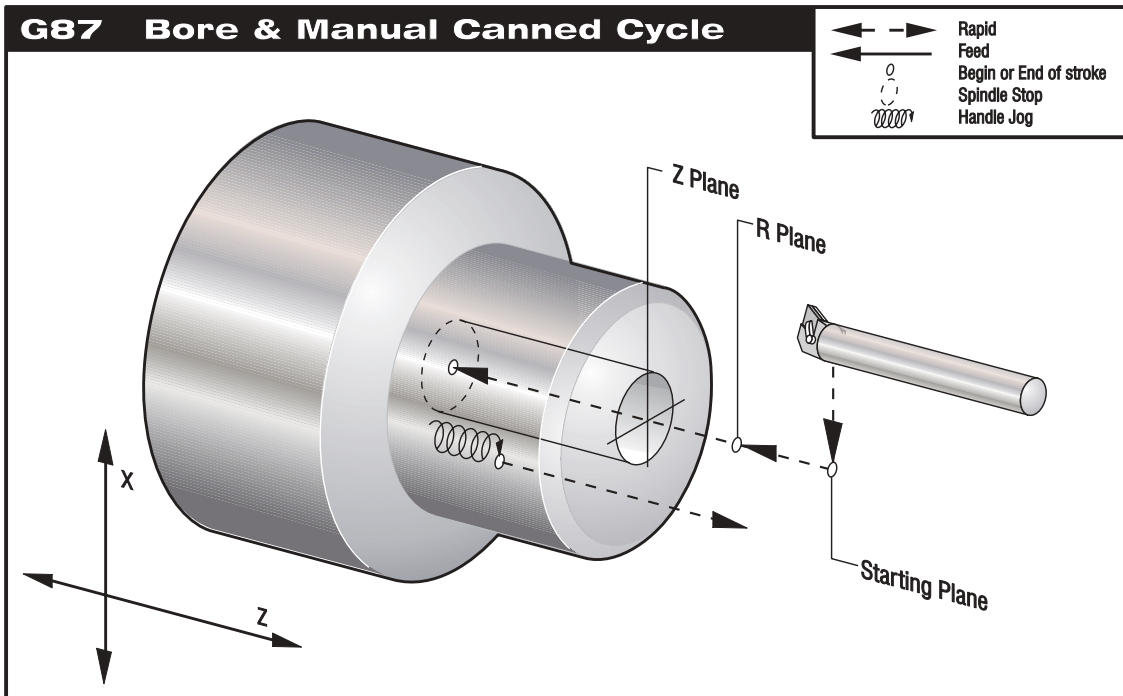
G89   Bore & Dwell Canned Cycle

Rapid
Feed
Begin or End of stroke
Dwell

Z Plane
R Plane
X
Z
Starting Plane
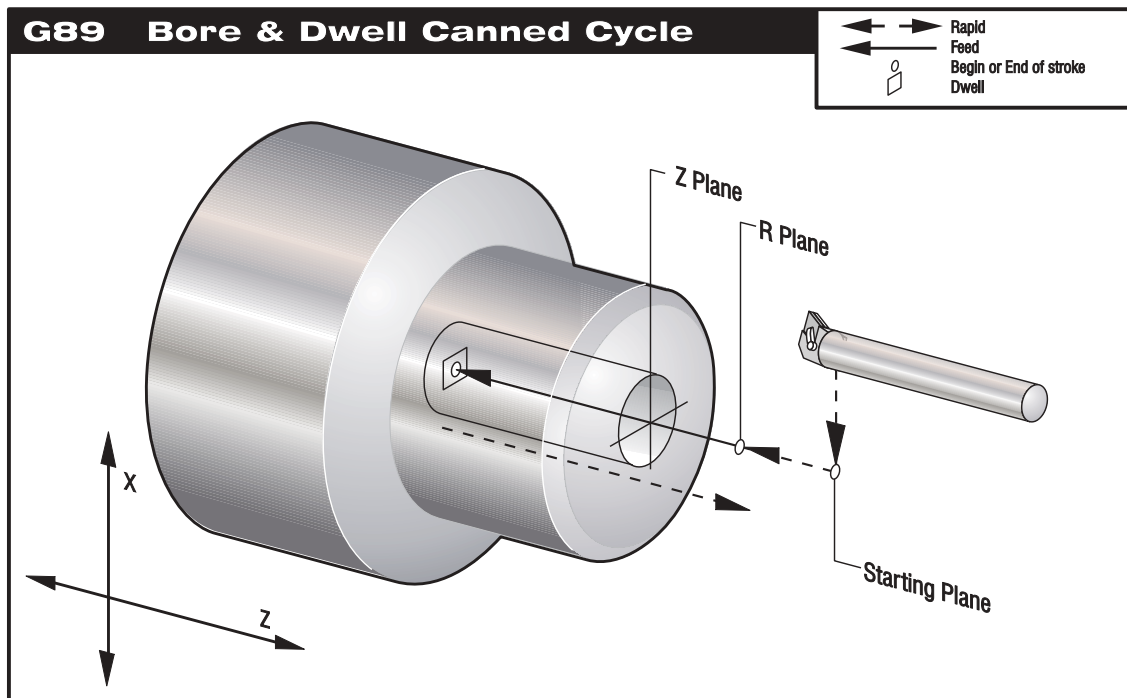
MODAL CANNED CYCLES (G90, G92, G94)

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
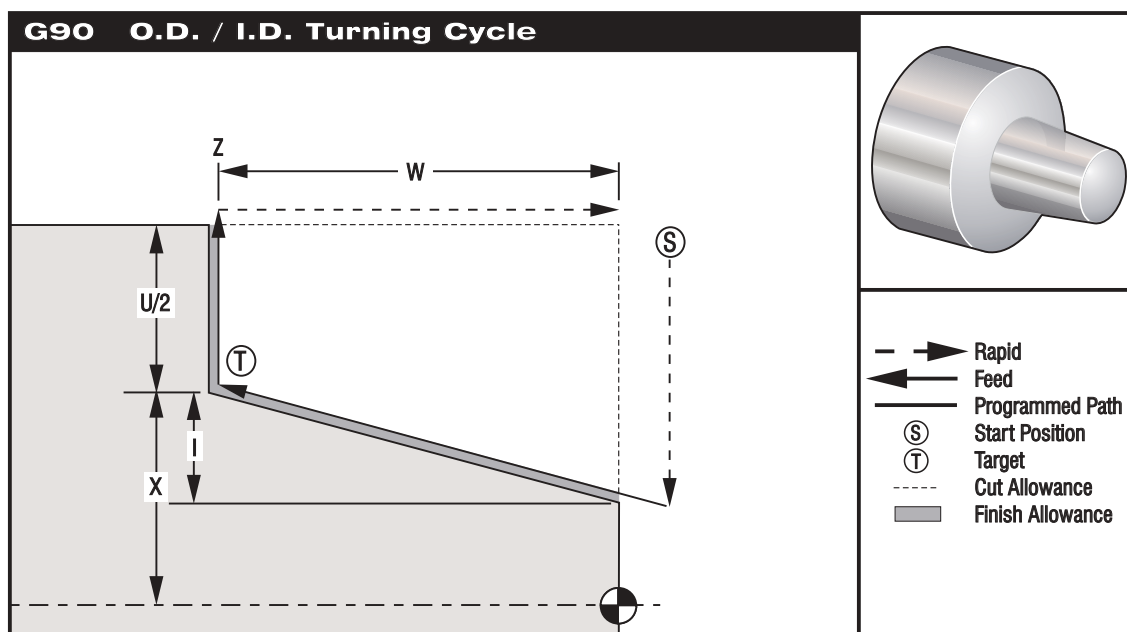
Fig. 4.0-21 G90

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 specifiing 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 4.0-20.

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 4.0-22 gives a few examples of the values required for machining in each of the four quadrants.
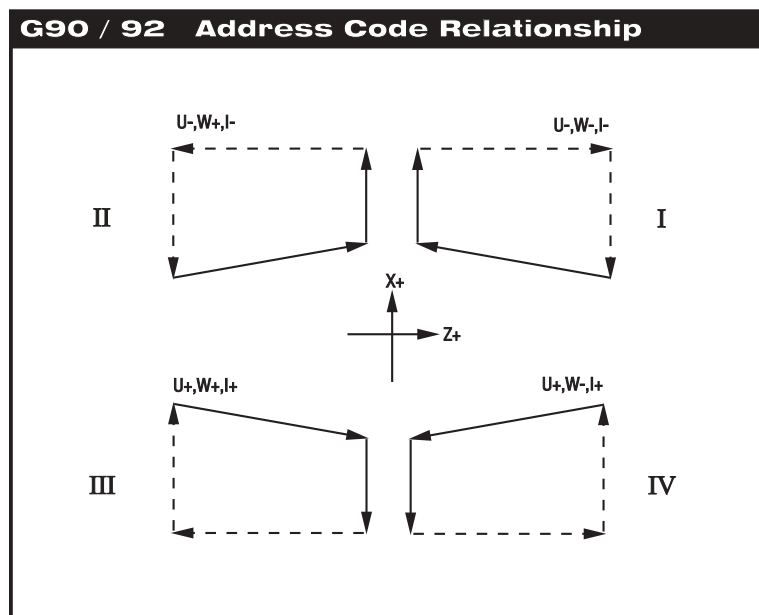
Fig. 4.0-22 G90-92 Address Relationships

G92   Threading Cycle   Group 01

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

Setting 95 / 96 determine chamfer size / angle M23 / 24 turn chamfering on / off.
See figure 4.0-19.5

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 specifiing 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 4.0-22.

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.

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

Note: The range of thread leads,limitation of spindle speed, etc. are the same as
             G32 Threading.
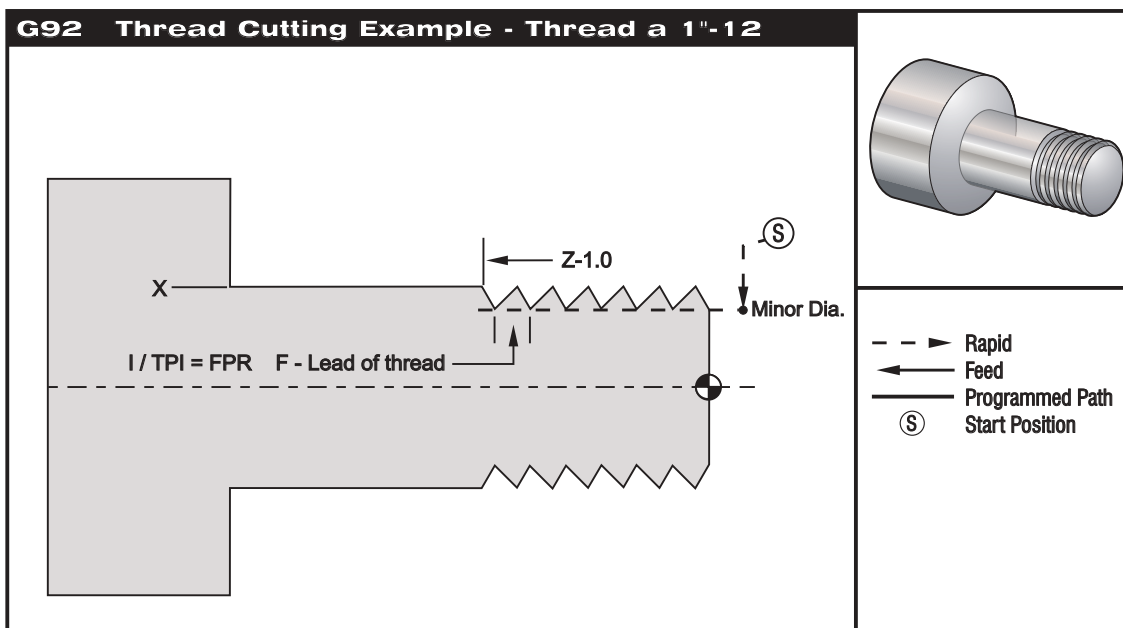
G92 Thread Cutting

PROGRAM EXAMPLE
```
%
T101
G50 S3000 M3
G97 S1000
X1.2 Z.2
G92 X.980 Z−1.0 F.0833
2nd pass    X.965
3rd pass    X.955
4th pass    X.945
5th pass    X.935
6th pass    X.925
7th pass    X.917
8th pass    X.910
9th pass    X.905
10th pass   X.901
11th pass   X.899
%
```

DESCRIPTION

(Rapid to Clear position)
(Set up Threading cycle)
(Subsequent Passes)
"
"
"
"
"
"
"
"
(Subsequent Passes)

Q Option example

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

The following rules apply to the usage of Q:

1. The start angle Q is not a modal value. It must be specified every time it is used. If no value is specified then a zero (0) angle is assumed.
2. The angle of threading increment is 0.001 degrees and it cannot have any decimal point. So a 180 degree must be specified as Q180000 and an angle of 35 degrees as Q35000.
3. The Q angle must be entered as a positive value from 0 to 360000. If a value greater than 360 degrees is entered then a modulus of 360 is applied to that value.

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

G94   End Faceing 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
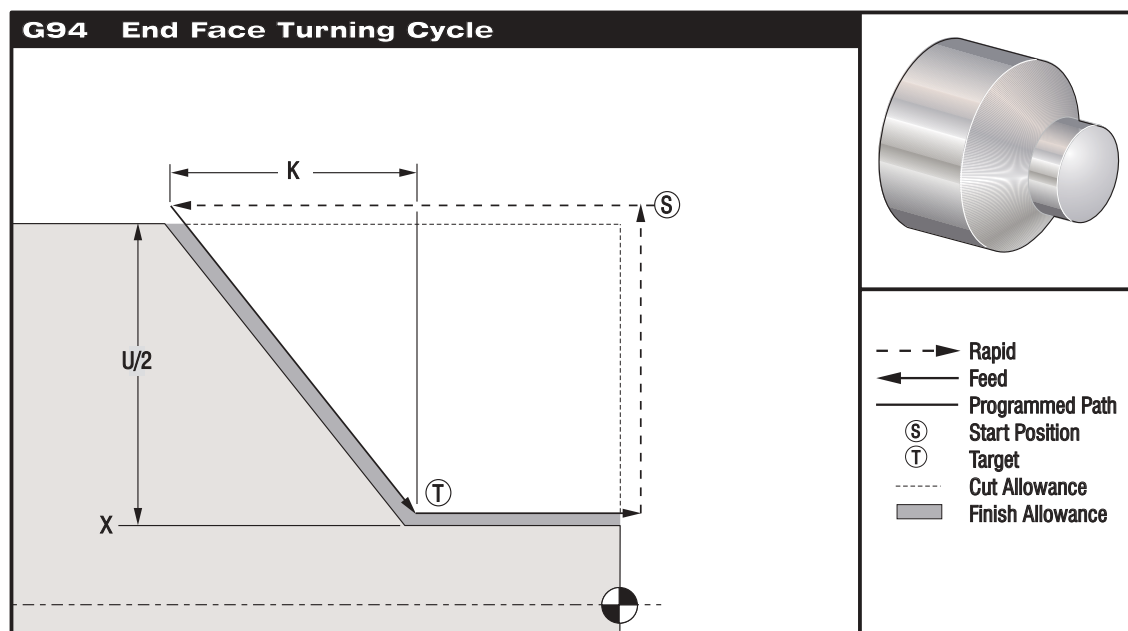
Fig. 4.0-23 G94

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 specifiing 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 4.0-22.

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 4.0-24 gives a few examples of the values required for machining in each of the four quadrants.

During incremental programming, the sign of the number following the U and W variables depends on the direction of the tool path. For example, if the direction of a path along the X-axis is negative, the value of U is negative.
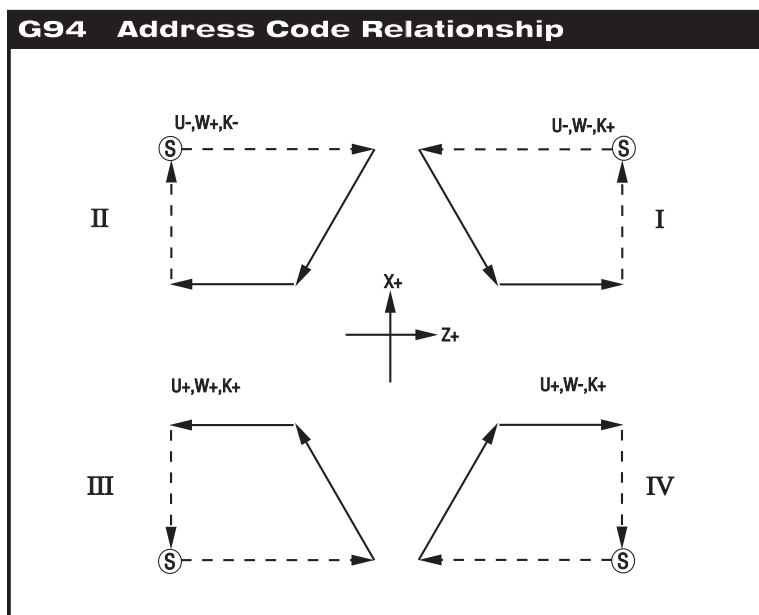
Fig. 4.0-24 G94 Address Relationships

SPINDLE SPEED COMMANDS (G96, G97)

G96   Constant Surface Speed ON   Group 13

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. The value of S indicates inches per revolution of the spindle when Setting 9 is set to INCH, while S indicates millimeters per revolution of the spindle when Setting 9 is set to METRIC. This code is modal, and is the default feed mode.

G97   Constant Surface Speed OFF   Group 13

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 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 18 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.

G154 Select Work Coordinates P1-99   Group 12

This feature provides 99 additional work offsets. G154 with a P value from 1 to 99 will activate the additional work offsets. For example G154 P10 will select work offset 10 from the list of additional work offset. Note that G110 to G129 refer to the same work offsets as the G154 P1 through P20; they can be selected by using either method. The Work Offset display screens have been adjusted accordingly. When a G154 work offset is active, the heading in the upper right work offset will show the G154 P value.

G154 work offsets format
#14001-#14006 G154 P1 (also #7001-#7006 and G110)
#14021-#14026 G154 P2 (also #7021-#7026 and G111)
#14041-#14046 G154 P3 (also #7041-#7046 and G112)
#14061-#14066 G154 P4 (also #7061-#7066 and G113)
#14081-#14086 G154 P5 (also #7081-#7086 and G114)
#14101-#14106 G154 P6 (also #7101-#7106 and G115)
#14121-#14126 G154 P7 (also #7121-#7126 and G116)
#14141-#14146 G154 P8 (also #7141-#7146 and G117)
#14161-#14166 G154 P9 (also #7161-#7166 and G118)
#14181-#14186 G154 P10 (also #7181-#7186 and G119)
#14201-#14206 G154 P11 (also #7201-#7206 and G120)
#14221-#14221 G154 P12 (also #7221-#7226 and G121)
#14241-#14246 G154 P13 (also #7241-#7246 and G122)
#14261-#14266 G154 P14 (also #7261-#7266 and G123)
#14281-#14286 G154 P15 (also #7281-#7286 and G124)
#14301-#14306 G154 P16 (also #7301-#7306 and G125)
#14321-#14326 G154 P17 (also #7321-#7326 and G126)
#14341-#14346 G154 P18 (also #7341-#7346 and G127)
#14361-#14366 G154 P19 (also #7361-#7366 and G128)
#14381-#14386 G154 P20 (also #7381-#7386 and G129)
#14401-#14406 G154 P21
#14421-#14426 G154 P22
#14441-#14446 G154 P23
#14461-#14466 G154 P24
#14481-#14486 G154 P25
#14501-#14506 G154 P26
#14521-#14526 G154 P27
#14541-#14546 G154 P28
#14561-#14566 G154 P29
#14581-#14586 G154 P30
#14781-#14786 G154 P40
#14981-#14986 G154 P50
#15181-#15186 G154 P60
#15381-#15386 G154 P70
#15581-#15586 G154 P80
#15781-#15786 G154 P90
#15881-#15886 G154 P95
#15901-#15906 G154 P96
#15921-#15926 G154 P97
#15941-#15946 G154 P98
#15961-#15966 G154 P99

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.

G05 Fine Spindle Control Motion   Group 00

R   Angular motion of the spindle, in degrees.
F   Feed Rate of the center of the tool, in inches per minute.
*U   X-axis incremental motion command.
*W   Z-axis incremental motion command.
*X   X-axis absolute motion command.
*Z   Z-axis absolute motion command.
* indicates optional

This G code is used to specify a precise motion of the spindle, 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 aproximately 14.77. This means that G5 motions with small R motions relative to X or Z motions will not work. For example, an R motion of 1.5 degrees, the largest X or Z motion that can be specified is 14.77 * 1.5 / 360 = .0615 inches. Conversely, an X or Z motion of .5 inches must have an R travel of at least .5 * 360 / 14.77 = 12.195 degrees.

Simple Face Slot Example with G05
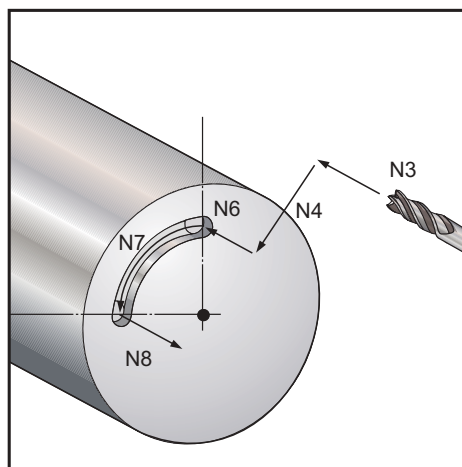
(Assume pilot hole is already drilled.)
N1   T303   (Small End Mill)
N2   M19   (Orient Spindle)
N3   G00 Z0.5
N4   G00 X1.
N5   M133 P1500
N6   G98 G1 F10. Z−.25   (Plunge into pre−drilled hole)
N7   G05 R90. F40.   (Make slot)
N8   G01 F10. Z0.5   (Retract)
N9   M135
N10 G99 G28 U0 W0

Simple Cam Example with G05
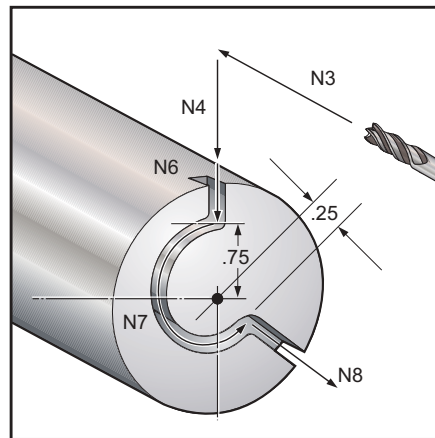
N1   T303   (Small End Mill)
N2   M19
N3   G00 Z−.25
N4   G00 X2.5   (Approach 2" diam stock)
N5   M133 P1500
N6   G98 G01 X1.5 F40.   (Cut to top of cam)
N7   G05 R215. X.5 F40.   (Cut Cam)
N8   G01 X2.5 F40.   (Cut out of cam)
N9   M135
N10 G99 G28 U0 W0

Flatting Example with G05

$$\#104 = \frac{\#101}{COS(\varnothing)}$$
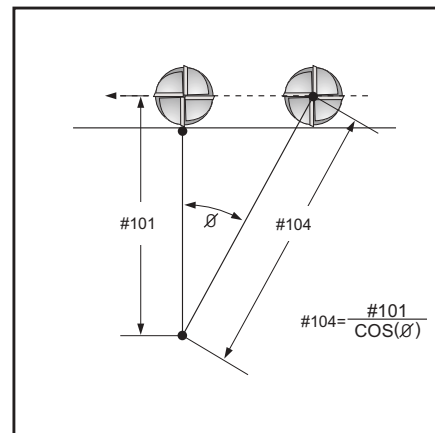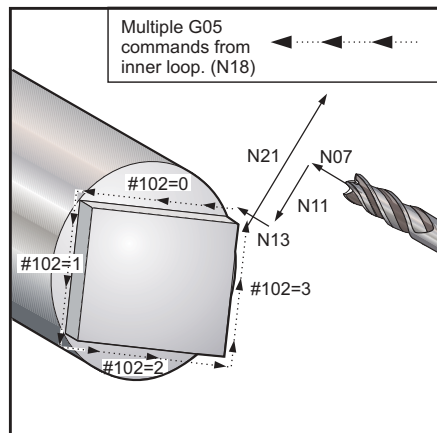
```
O01484                                    (Cut a square with G05)
N1  G28 X0.
N2  G28 Z0.
N3  G54 G00 G40 G97
N4  G103 P3
N5  T707                                   (.75 dia high–speed end mill)
N6  M19
N7  G00 Z0.5
        ()
        (101 = Closest approach. Center to)
        (side plus half of 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 Flatting 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.

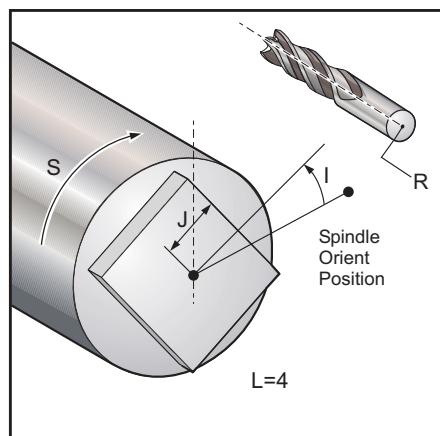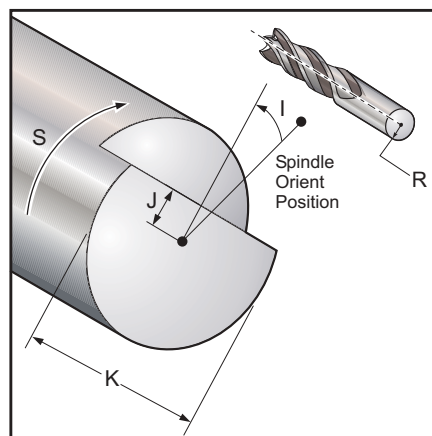G77 operates in one of two modes, depending on whether a K code or an L code is specified. If a K code is specified, one flat surface will be cut. If an L code is specified, L flat surfaces will be cut, equally spaced all the way around the part. L must be greater than or equal to 3. L2 is not supported, 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:

G77 Motion
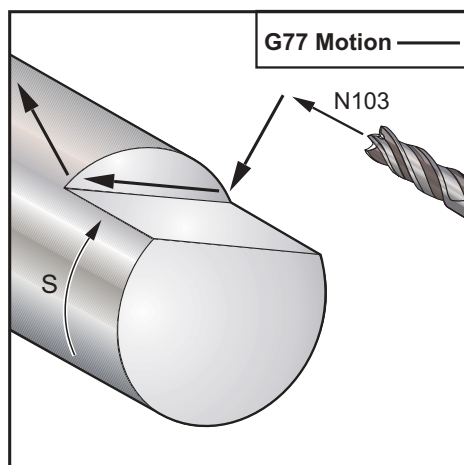
N103

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

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:

G77 Motion
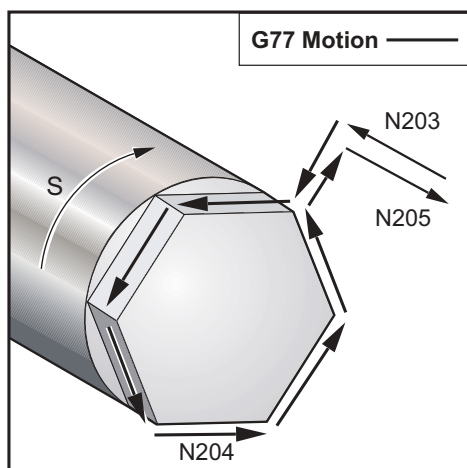
N203

N205

N204

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

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

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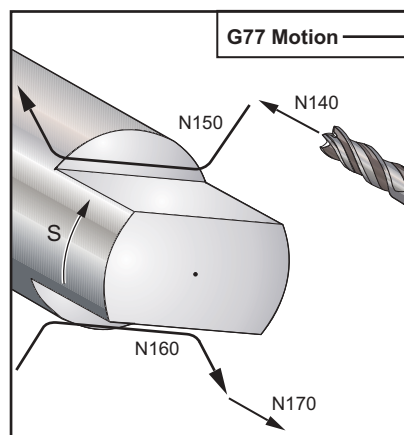

Also see the C-axis section for examples using the C-axis

G95   Live Tooling Rigid Tap (Face)   Group 09

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

G186   Rev Live Tool Rig Tap   Group 09

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

G95 / 186   Live Tooling Rigid Tapping

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.

Below is a brief program example of G95

```
    %
    O00800
    N1 T101 (AXIAL 1/4–20 TAP)
    G99 (Neccesary for this cycle)
    G00 Z0.5
    X2.5
    S500 (rpm should look like this, cw direction)**
    M19PXX (Orient spindle at desired location)
    M14(Lock spindle up)
    G95 Z–.500 R.25 F0.05 (thread down .50 deep)
    G28 U0
    G28 W0
    M135 (Stop Live tooling spindle)
    M15 (Unlock Spindle brake)
    M30
    %
```

G195 Live Tool Radial Tapping (Diameter)   Group 00

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

G196 Reverse Live Tool Vector Tapping (Diameter)   Group 00

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

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.

G195 / G196 Live Tooling Rigid Tapping

Rapid
Feed
Begin or End of stroke
Spindle forward.
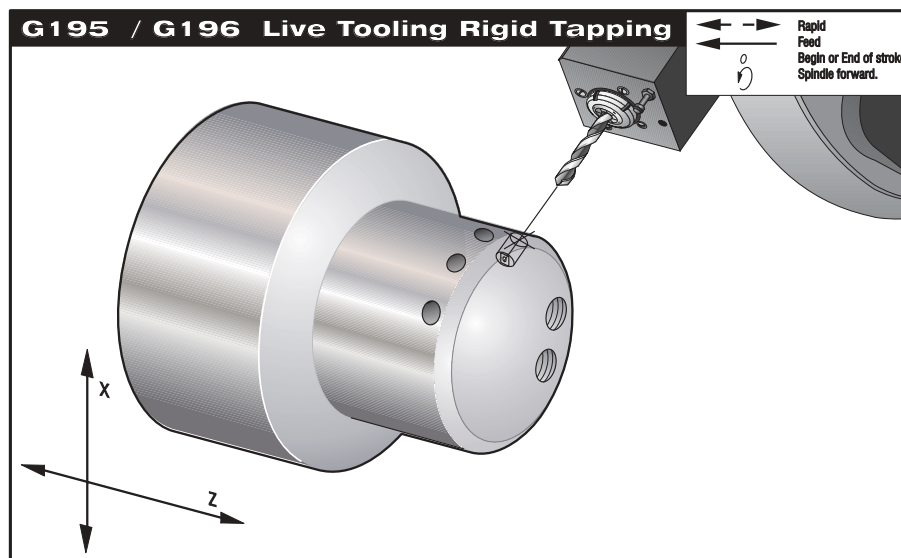
Below is a brief program example of G195

```
   %
 O00800
 N1 T101 (RADIAL 1/4–20 TAP)
 G99 (Neccesary for this cycle)
 G00 Z0.5
  X2.5
 Z–0.7
  S500 (rpm should look like this, cw direction)**
 M19PXX (Orient spindle at desired location)
 M14(Lock spindle up)
 G195 X1.7 F0.05 (thread down to X1.7)
  G28 U0
  G28 W0
  M135 (Stop Live tooling spindle)
  M15 (Unlock Spindle brake)
   M30
    %
```

G200 Index on the Fly   Group 00

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

5. 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 (Optional)
M13	Auto Air Jet Off (Optional)
M14	Clamp Main Spindle
M15	Unclamp Main Spindle
M17	Turret Rotation Always Forward
M18	Turret Rotation Always Reverse
M19	Orient Spindle (Optional)
M21	Tailstock Forward
M22	Tailstock Reverse
M23	Thread Chamfer ON
M24	Thread Chamfer OFF
M30	Prog End and Rewind
M31	Chip Conveyor Forward
M33	Chip Conveyor Stop
M36	Parts Catcher Up (Optional)
M37	Parts Catcher Down (Optional)
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
M59	Set Output Relay
M61-M68	Optional User M turn OFF
M69	Clear Output Relay
M76	Disable Displays
M77	Enable Displays
M78	Alarm if skip signal found
M79	Alarm if skip signal not found
M85	Open Automatic Door (optional)
M86	Close Automatic Door (optional)
M88	Turns On High Pressure Coolant (optional)
M89	Turns Off High Pressure Coolant (optional)
M93	Start Axis Pos Capture
M94	Stop Axis Pos Capture
M95	Sleep Mode

M96	Jump if no Input
M97	Local Sub-Program Call
M98	Sub Program Call
M99	Sub Program Return Or Loop
M109	Interactive User Input
M119	Subspindle Orient (Optional)
M121-128	Optional User M
M133	Live Tooling Drive Forward (Optional)
M134	Live Tool Drive Reverse (Optional)
M135	Live Tool Drive Stop (Optional)
M143	Subspindle Forward (Optional)
M144	Subspindle Reverse (Optional)
M145	Subspindle Stop (Optional)
M154	C-axis engage (Optional)
M155	C-axis disengage (Optional)

* SL-30 and 40 only.

Note: The following M codes will be processed when setting 36 PROGRAM RESTART is enabled:

M08 COOLANT ON	M37 PARTS CATCHER OFF
M09 COOLANT OFF	M41 LOW GEAR
M14 CLMP MAIN SPNDL	M42 HIGH GEAR
M15 UNCLMP MAIN SPNDL	M51-58 SET USER M
M36 PARTS CATCHER ON	M61-68 CLEAR USER M

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 of 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 of 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 (Optional)

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 (Optional)

The M13 code is used to activate 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 two places to the right of the decimal point (round off to the nearest hundredth of a degree). Note: the actual position accuracy of the spindle is further limited by the servo-encoder system resolution. An M19 R123.45 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 movethe 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. M23 is the default at power-up and when reset.

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 incremented. M30 cancels 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

M33   Chip Conveyor Stop

M33 Stops Conveyor motion.

M36   Parts Catcher Up (Optional)

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 (Optional)

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

M59 SET OUTPUT RELAY

This M-code directly enables a discrete output relay. The syntax for its usage is M59 Pnn where "nn" specifies the number of the relay being turned ON. An M59 command can be used to turn ON any of the discrete output relays in the range of 1100 and 1155. For example, M59 P1103 does the same thing as #1103=1 except that it is processed as an M-code (processed when coming out of the queue instead of going into the queue.) To turn off a relay, use M69. This M-code **is not** a general-purpose M-code. It is for special functions, and its use should be strictly monitored. Any misuse of this function will cause serious damage to the machine and will void any warranty.

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.

M69 CLEAR OUTPUT RELAY

This M-code directly disables a discrete output relay. The syntax for its usage is M69 Pnn where "nn" specifies the number of the relay being turned OFF. An M69 command can be used to turn OFF any of the discrete output relays in the range of 1100 and 1155. For example, M69 P1103 does the same thing as #1103=0 except that it is processed as an M-code (processed when coming out of the queue instead of going into the queue.) To turn on a relay, use M59. This M-code **is not** a general-purpose M-code. It is for special functions, and its use should be strictly monitored. Any misuse of this function will cause serious damage to the machine and will void any warranty.

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);
.	
N85 M21	(EXECUTE AN EXTERNAL USER FUNCTION)
N90 M96 P10 Q27	(LOOP TO N10 IF SPARE INPUT IS 0);
M95 M30	(IF SPARE INPUT IS 1 THEN END PROGRAM);

M97   Local Sub-Program Call

This code is used to call a subroutine referenced by a line 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.

M109   Interactive User Input

This M code allows a G-code program to place a short prompt on the screen, get a single character input from the user and store it in a macro variable. The first 15 characters of the M109 block will display as a prompt in the lower left corner of the screen. A macro variable in the range 500 through 599 must be specified by a P- code. Note also that due to the look-ahead feature, it is necessary to include a loop in the program following the M109 to check for a non-zero response before continuing. The program can check for any character that can be entered from the keyboard by comparing with the decimal equivalent of the ASCII character. Here are a few common characters:

A - 65	a - 97		N - 78	n - 110		1 - 49	-	45
B - 66	b - 98		Y - 89	y - 121		2 - 50	*	42
C - 67	c - 99		0 - 48	+ - 43		3 - 51	/	47

The following sample program will ask the user a Yes or No question then wait for either a Y or an N to be entered. All other characters will be ignored.

```
N1 #501= 0. (CLEAR THE VARIABLE)
M109 P501 (Sleep 1 min?)
N5 IF [ #501 EQ 0. ] GOTO5 (WAIT FOR A KEY)
IF [ #501 EQ 89. ] GOTO10 (Y)
IF [ #501 EQ 78. ] GOTO20 (N)
GOTO1 (KEEP CHECKING)
N10 (A Y WAS ENTERED)
M95 (00:01)
GOTO30
N20 (AN N WAS ENTERED)
G04 P1. (DO NOTHING FOR 1 SECOND)
N30 (STOP)
M30
```

The following sample program will ask the user to select a number then wait for a 1, 2 or, 3 to be entered. All other characters will be ignored.

```
O00234 (SAMPLE PROGRAM)
N1 #501= 0. (CLEAR THE VARIABLE)
M109 P501 (Pick 1, 2 or 3:)
N5 IF [ #501 EQ 0. ] GOTO5 (WAIT FOR A KEY)
IF [ #501 EQ 49. ] GOTO10 (1)
IF [ #501 EQ 50. ] GOTO20 (2)
IF [ #501 EQ 51. ] GOTO30 (3)
GOTO1 (KEEP CHECKING)
N10 (A 1 WAS ENTERED)
M95 (00:01)
GOTO30
N20 (A 2 WAS ENTERED)
G04 P5. (DO NOTHING FOR 5 SECONDS)
N30 (A 3 WAS ENTERED)
M30
```

M119   Subspindle Orient

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

    M119 Pxxx
    M119 Rxx.x

M121-M128   Optional User M

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

M133   Live Tool Drive Forward (Optional)

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 (Optional)

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 (Optional)

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** (Optional)

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** (Optional)

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** (Optional)

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** (Optional)

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** (Optional)

This M code is used to disengage the optional C-axis motor. Parameter 278 bit 26 C Axis Drive must be set to 1.

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

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

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

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

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

1 AUTO POWER OFF TIMER

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 in changed from INCH to MM, all offset values will be converted accordingly. Tool Tip Values are unaffected.

	INCH	METRIC
Feed Max Travel Min. Programmable Dimension Feed Range	inches/min. +/- 15400.0000 .0001 .0001 to 300.000 in/min.	mm/min. +/-39300.000 .001 .001 to 1000.000

Axis Jog Keys		
.0001 Key .001 .01 .1 Key	.0001 in/jog click .001 in/jog click .01 in/jog click .1 in/jog click	.001 mm/jog click .01 mm/jog click .1 mm/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 settings 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.

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.

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.

Note:	The following M codes will be processed when setting 36 PROGRAM RESTART is enabled:	
	M08 COOLANT ON	M37 PARTS CATCHER OFF
	M09 COOLANT OFF	M41 LOW GEAR
	M14 CLMP MAIN SPNDL	M42 HIGH GEAR
	M15 UNCLMP MAIN SPNDL	M51-58 SET USER M
	M36 PARTS CATCHER ON	M61-68 CLEAR USER M

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.

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 Z FACE MEASURE 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 6 / Parameter 5
    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 display-ing 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 UP/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 enables languages other than English to be loaded from disk files. Instructions for loading the files are displayed on the screen.

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. As some machines use various pitch ball screws, the control will take the lesser of the allowed error in encoder counts from all axes in order to maximize machining accuracy.
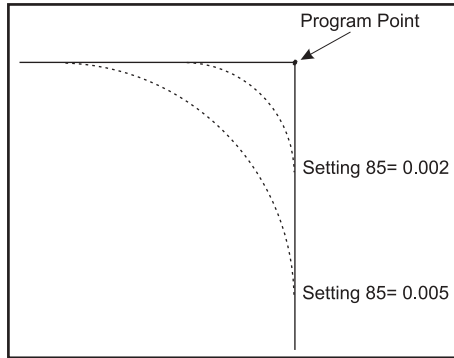
The following two conditions would occur at the same feed rate and setting 85 value.

No slowdown required to meet accuracy setting

Much lower speed needed to machine into the corner

Setting 85 Examples

86 THREAD FINISH ALLOWANCE

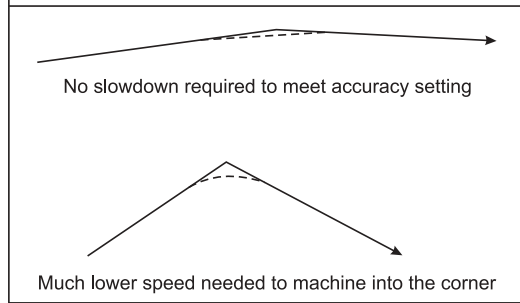Used in G76 canned threading cycle, this setting specifies how much material will be left on the thread for finishing after all passes of the cycle. Values range from 0 to .9999 inches. The default value is 0.

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.

G76 or G92 Threading Stroke with M23 active

Legend:
- Thread
- Tool Path
- + Programmed Thread Endpoint
- Setting 95 x L
- Setting 96 = 45
- L Lead
- ● Actual Stroke Endpoint
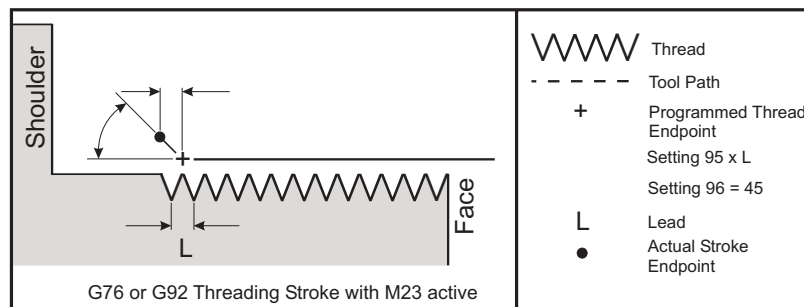
96 THREAD CHAMFER ANGLE

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 OVERIDE -> 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.

The autodoor can be used when the safety circuit is enabled provided the operator keeps their finger on the CYCLE START button until the door has completed its motion. If the operator lifts the button before the door has fully closed, the control will stop all motion and display DOOR NOT CLOSED. The AUTODOOR BUTTON functions in a similar manner. It will close an open door as long as the user holds the button. If the door was closed, then pressing the button will open the door as long as it is held. When the safety circuit is enabled, the autodoor operations associated with M00, M01, M30, M80, M81, M85 and M86 will be disabled.

For machines not equipped with Safety Circuit, this setting will enable and disable the autodoor operation via CYCLESTART, M00, M01, M30, and the autodoor button (if installed). **WARNING!** With this change, M85 and M86 cannot be disabled with this setting. Unexpected movement may occur. To prevent M85 and M86 codes from causing the autodoor to move, parameter 251 A DOOR OPEN ERRTIME must be changed to zero. Note: If the autodoor is disabled in this manner, the user must make a note of the value of parameter 251 so that it can be properly re-enabled later. If an incorrect value is entered into parameter 251, the autodoor may malfunction.

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.

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.

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.

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.

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.)

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.)

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.)

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.)

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.)

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

142 OFFSET CHNG TOLERANCE

This setting to generates a warning message if an offset is changed more than the specified amount. It is intended to help prevent operator errors. The user can set it to any number from 0 to 99.9999. When the setting contains zero, the feature is inactive and the control behaves as before. When it contains a non-zero number and an attempt is made to change an offset by more that this amount (either positive or negative) the following prompt is displayed:

XX changes the offset by more than Setting 142! Accept (Y/N)?"

If "Y" is entered, the control updates the offset as usual, otherwise, the change is rejected.

143   MACHINE DATA COLLECT

This setting enables the user to extract data from the control using a Q command sent via the RS-232 port. This feature is software-based and requires a computer to request, interpret and store data from the control. Note that the control will only respond to a Q command when this setting is ON. The following output format is used:

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

Note: STX = 0x02 (ctrl-B); ETB = 0x17 (ctrl-W)

The following commands can be used:
Q100 - Machine Serial Number
Q101 - Control Software Version
Q102 - Machine Model Number
Q104 - Mode (LIST PROG, MDI, MEM, JOG, etc.)
Q200 - Tool Changes (total)
Q201 - Tool Number in use
Q300 - Power-on Time (total)
Q301 - Motion Time (total)
Q303 - Last Cycle Time
Q304 - Previous Cycle Time
Q400 - not currently used
Q401 - not currently used
Q402 - M30 Parts Counter #1 (resettable at control)
Q403 - M30 Parts Counter #2 (resettable at control)
Q500 - Three-in-one (PROGRAM, Oxxxxx, STATUS, PARTS, xxxxx)

If the control is busy, the control will output "STATUS, BUSY." If a request is not recognized, the control will output "UNKNOWN."

In additional, when Setting 143 MACHINE DATA COLLECT is ON, the user has the ability to request the contents of any macro or system variable by using the Q600 command. Additionally, variables 1 thru 999 and 2001 thru 2800 can be written to using the new E command. Syntax is .Q600 xxxx. where xxxx is the macro variable. To write to a macro variable, use .Exxxx yyyyyy.yyyyyy. where xxxx is the macro variable and yyyyyy.yyyyyy is the new, floating point value. Note that this command should only be used when there are no alarms present.

This feature can be enhanced with the installation of the following optional parts, M-code relay board, a power ON relay and a new E-STOP switch (set parameter 209 bit 23 MCD RLY BRD and parameter 315 bit 26 STATUS RELYS to 1). The Haas Service personnel will install these optional parts. Once installed output relays 40 thru 47, a power ON relay and the E-STOP switch are used to communicate the status of the control.

The following communications will be received when used with the optional parts.
* E-STOP contacts. This will be closed when the E-STOP button is pushed.
* Power ON - 115 VAC. Indicates that the control is turned ON. It should be wired to a 115 VAC coil relay for interface.
* Spare Output Relay 40. Indicates that the control is In-Cycle (running.)
* Spare Output Relay 41 and 42:
  11 = MEM mode & no alarms (AUTO mode.)
  10 = MDI mode & no alarms (Manual mode.)
  01 = Single Block mode (Single mode)
  00 = other modes (zero, DNC, jog, list prog, etc.)
* Spare Output Relay 43 and 44:
  11 = Feed Hold stop (Feed Hold.)
  10 = M00 or M01 stop
  01 = M02 or M30 stop (Program Stop)
  00 = none of the above (could be single block stop or RESET.)
* Spare Output Relay 45 (Feed Rate Override is active and Feed Rate is NOT 100%)
* Spare Output Relay 46 (Spindle Speed Override active and Spindle Speed is NOT 100%)
* Spare Output Relay 47 (Control is in EDIT mode)

144 FEED OVERIDE->SPINDLE

This 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.

145 TS AT PART FOR CS

This setting that will requires the tailstock to be holding the part when a program is started. When it is OFF, the machine behaves as before. When this setting is ON, parameter 278 HYDRAULIC TS is ON and parameter 315 SIMPLE TS is OFF, the tailstock must be pressing against the part at the moment Cycle Start is pressed or the message TS NOT AT PART will be displayed and the program will not be started.

156 SAVE OFFSET WITH PROG

This ON/OFF setting controls whether offsets are automatically saved with programs to floppy disk and RS-232. The offsets are saved (to floppy disk or RS-232) in the same file as the programs but under the heading O999999. The offsets will appear in the file before the final % sign.

157 OFFSET FORMAT TYPE

This setting controls the format in which offsets are saved with programs to floppy disk or RS-232. When it is set to B, the old-style format will be used, that is, each offset is saved on a separate line with an N value and a V value. When it is set to A, a new format is used that is easier to read and edit. The new format resembles the display screen format and contain decimal points and column headings. Offsets saved in this format can be more easily edited on a PC and later reloaded into the control.

158,159,160 XYZ SCREW THERMAL COMP%

These settings can be set from -30 to +30 and will adjust the existing screw thermal compensation by -30% to +30% accordingly.

162 DEFAULT TO FLOAT

This feature that is compatible with Fanuc. When this setting is ON, the control will append a decimal point to values entered wiithout a decimal point (for certain address codes). When this setting is Off, the machine behaves as before, that is, values following address codes that do not include decimal points are treated as machinist's notation (i.e., thousandths or ten-thousandths.) For example, in INCH mode:

    Value entered   With Setting OFF   With Setting ON
        X-2   X-.0002   X-2.
 In MM mode:
        X-2   X-.002   X-2.

This feature applies to the following address codes:

X,Y,Z,A,B,C,E,F,I,J,K,U,W

            D (except with G73)

            R (except with G71 in YASNAC mode)

Note that this setting affects the interpretation of all programs entered either manually or from disk or via RS-232. It does not alter the effect of settin 77 Scale Integer F.

163 DISABLE .1 JOG RATE

This setting disables the highest jog rate. If the highest jog rate is selected, the next lower rate is automatically selected instead. During times when it would be dangerous to jog at the highest rate, the operator can turn this setting on and it will help to prevent accidental damage to tools and parts.