Haas Factory Outlet

A Division of Productivity Inc

Lathe Series Training Manual

Haas CNC Lathe Operator

Revised 10/2014

This material is to be used as a guide to the subject content. The Operator is responsible for following Safety Procedures as outlined by their instructor or manufacturer's specifications.

Downloading and/or other use of this manual does not indicate completion of the Training course. This manual is for reference only.

Haas Lathe Operator Training Manual
Table of Contents

Productivity Inc®

For more information on Additional Training Opportunities
or our Classroom Schedule
Contact the Productivity Inc Applications Department in Minneapolis:
☏ 763.476.8600
Visit us on the Web: www.productivity.com
Click on the Training Registration Button
✉ trainingmn@productivity.com

Introduction to Basic Lathe Operation

Welcome to Productivity, Inc., your local Haas Factory Outlet (H.F.O.) for the Haas Lathe Operator Class. This class is intended to give a basic understanding of the set-up and operation of a Haas Turning Center.

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 a computer monitors the machine tool.

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 Vertical Turning Center requires that a part program be designed, written, and entered into the memory of the control. There are several options for getting these programs to the control. RS-232 (serial port with a computer), 3.5” Floppy Disk, Ethernet / Networking/ and USB are all viable ways to transmit and receive programs.

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.

At Productivity, we have two classes that pertain to Haas Turning Centers. The one we are in right now is the Operator Class.   We also have a Haas Lathe Programming class.

We have two classes to fill the different needs of our customers as not all people that require training require programming training.

We do include the entire Programming manual in order to give everyone the opportunity to study G&M code lathe programming, but we also go over SOME programming basics so that the operator can understand the programmer’s intensions.

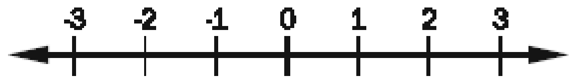
This manual can be used as both an operator's manual and as a programmer's manual. It is intended to give a ***basic*** understanding of CNC programming and its applications. It is not intended as an in-depth study of all ranges of machine use, but as an overview of common and potential situations facing 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 Turning Center. 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 Turning Center.

(Updated CK 5/1/12) Rev 10/2014

The Cartesian Coordinate System

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



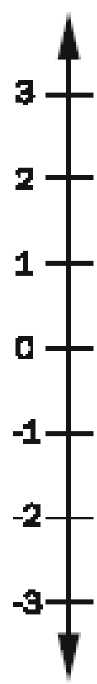
Horizontal number line

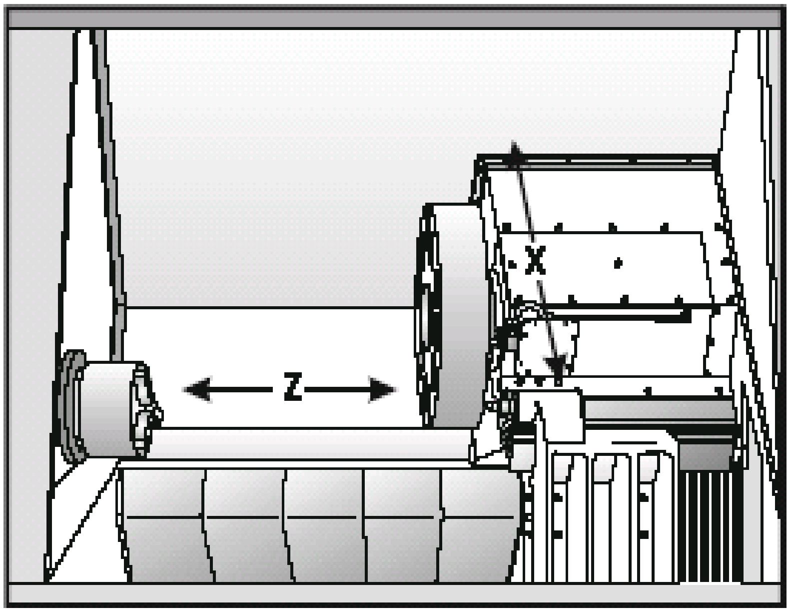
The number line also has numbered increments on either side of absolute zero.
Moving away from zero to the right are positive increments. Moving away from zero to the left are negative increments. The “+”, or positive increments, are understood, therefore no sign is needed. We use positive and negative signs along with increment value's to indicate its relationship to zero on the line.

Our concern is the distance and the direction from zero and is labeled as “Absolute Programming”

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.

Vertical Number Line known as the “X” axis on a Lathe





Haas Lathe X and Z axis lines

Haas VMC (SL-10) showing the X and Z axis

The machine illustration shows two directions of travel available on a turning center. Now to carry the number line idea a little further, imagine such a line placed along each set of travels (or axis) of the machine. The first number line would be the left-to-right, or "Z", axis of the machine. Positive Z values would move the turret away from the chuck, negative values towards the chuck.

If we place a similar number line along the front-to-back movement, or "X" axis, this moves the turret to and from the centerline of the spindle of the machine. The X-Axis is programmed for diametrical values. That means when we tell the machine to move an X-dimension, it will place the tool at the position in which it will generate that diameter. Since this is a lathe, to make a 1.0000 diameter part, the tool needs to be .5000 above the center of the spindle.

All axes of Haas Turning Centers have a resolution of .0001" inches (or .001mm).

Now theoretically all of our number lines for each axis are infinite in length but we are limited to the travels of the particular machine we are using. Below is an example of the travels of different Haas Turning Centers showing how much movement we have on each particular model.

See "Turning Center Max Cutting" chart on the next page.

Turning Centers Max Cutting

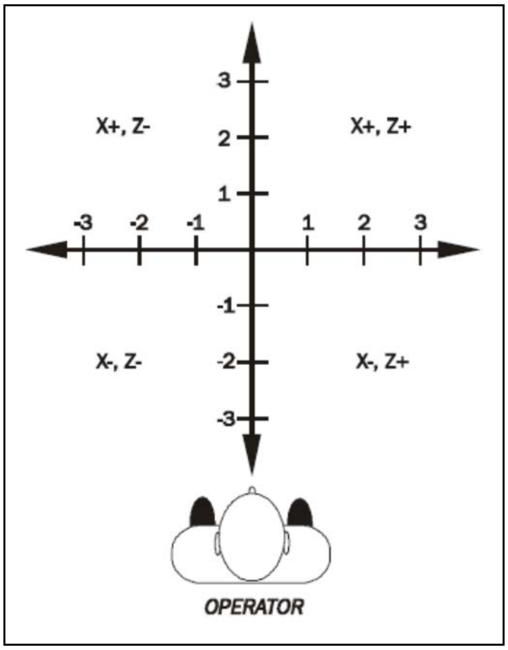
Maximum Cutting Diameter and Lengths of Various Haas Lathes

Model	Max Cutting Diameter	Max Cutting Length	Chuck Size	Bar Capacity
OL-1	1.06”	8.0”	4”	1.06”
ST-10	14.0"	14.0"	6”	1.75”
ST-15	14.0”	14.0”	8”	2.5”
ST-20	15.0"	21.0"	8”	2”
ST-20SSY	10.0”	21.0”	8”	2”
ST-25	15.0”	21.0”	10”	3”
ST-30	21.0"	26.0"	10”	3”
ST-35Y	18.0”	23.0”	12”	4”
ST-40	25.0"	44.0"	15”	4”
TL-1	16.0”	29.0”	8”	2.03”
TL-2	16.0”	48.0”	10”	3.0”
ST-20SSY	10.0”	21.0”	8”	2”
DSL-30Y	18.0”	23.0”	8”	2”

Note ST15 is big bore variant of ST10, ST25 a big bore of ST20 and so forth. Note the on these lathes a larger chuck is put on the lathes with a 5 addition.

The diagram below shows a front view of the grid as it would appear on the lathe. This view shows the X and Z axis' as the operator faces the lathe.

Note that at the intersection of the two lines, a common zero point is established.



"QUADRANTS" are the four areas to the sides, above, and below the lines and make up the basis for what is known as rectangular coordinate programming.

QUADRANT 1 - ON THE TOP RIGHT X+, Z+
QUADRANT 2 - ON THE TOP LEFT X+, Z-
QUADRANT 3 - ON THE BOTTOM LEFT X-, Z-
QUADRANT 4 - ON THE BOTTOM RIGHTX-, Z+

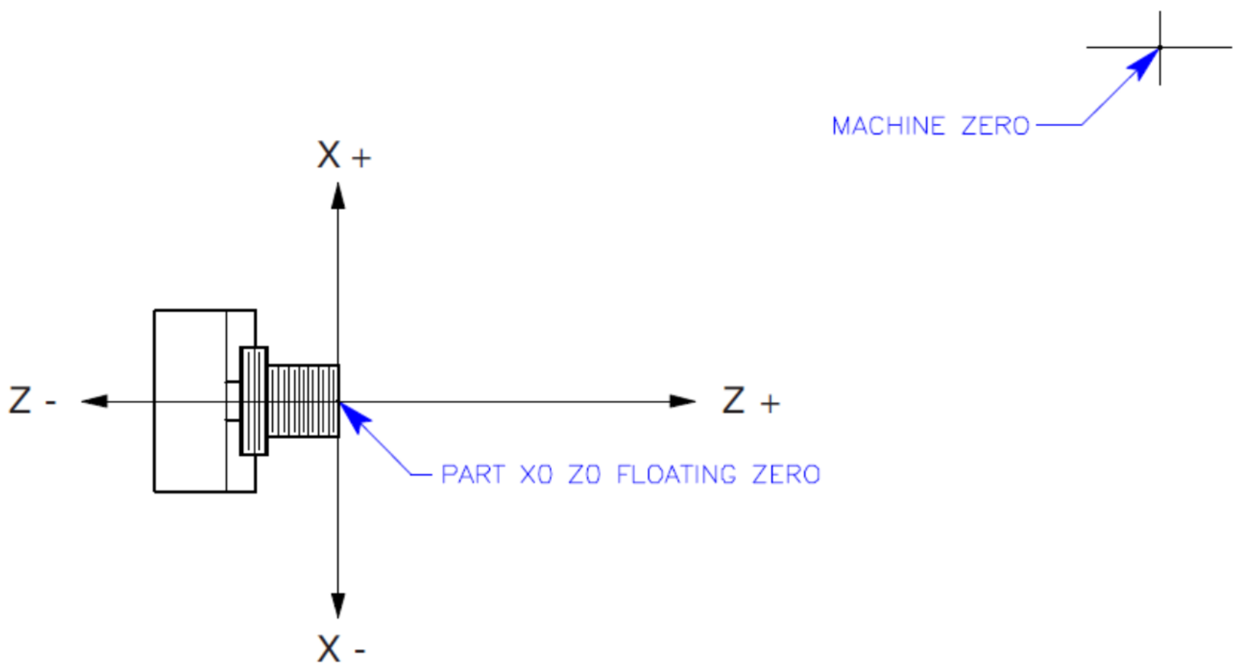
Whenever we set a zero somewhere on the X-axis and somewhere on the Z-axis, we have automatically caused an intersection of the two lines. The intersection where the two zeros come together will automatically have the four quadrants to its sides, above, and below it. How much of each quadrant that is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

For example, if we set zero exactly in the middle of the Z-axis and set the X-axis zero on the spindle centerline, we have created four quadrants. For an ST30, for example, the upper two quadrants of the Z travel is 26 inches and the X travel diametrically is 21.00 inches. The lower two quadrants will have Z travel of 26 inches and diametrical X travel of 5 inch. The HAAS SL30 lathe has a radial 2 ½" of negative travel beyond the centerline of the spindle.

Machine Home Position

The principle of machine home may be seen when doing a manual reference return of all the machine axes. When a zero return (**POWER UP/RESTART**) is performed when the machine is first powered up, all axes are moved to the furthest positive direction until the limit switches are reached. First the X moves in a positive direction until it reaches a limit switch. Then the Z moves in a positive direction until its limit switch is reached. After the HOME position is reached the turret indexes to tool #1. The machine then sets this position as machine home where X = 0 and Z = 0. When this condition is satisfied, the only way to move any of the two axes is in the negative direction. The **machine coordinates at machine home** are **X0 Z0**.

Machine Home is placed at the edge of each axis' travel. In effect, the positive quadrants cannot be reached, and all the X and Z moves will be found in the -X – Z quadrant. Other quadrants are reached only by setting a part zero somewhere within the travel of each axis.



At the power-on or start-up of the Haas lathe, a Machine Zero Return operation must be performed using the key: **(POWER UP/RESTART).** The machine may also be sent home by 1st pressing the **ZERO/RET** then the **AUTO ALL/AXES** or **HOME /G28.**

It would not be convenient to program our parts from the machine zero, so a secondary zero is established. This zero is referred to by one of two names: **<u>FLOATING ZERO</u>** OR **<u>PART ZERO,</u>** both having the same meaning. See the illustration above.

Tool Geometry

To create the floating or part zero, each tool is manually moved to the part being machined and touched to the diameter and length of the part before machining starts. Then, through a series of control keystrokes, the distance from machine zero to the part zero in X and Z is stored and activated later from the part program when that tool is needed for cutting the part.

The ability to establish a "floating zero" means that the "Z" zero will be allowed to "float" to any face on the part that reflects most of the length dimensions or is a primary datum. Normally, the **front face** is used because it provides easy access for the touch-off procedure. Programming in this manner ensures continuity between the part print dimensions and the HAAS part program dimension moves.

Manual Touch-Off Method:

First select and index to an 80 degree cutting tool with standard CNMG insert configuration. This is the tool that normally would be used for rough facing and turning operations. Normally it is in tool holder #1 in the turret. First take machine turret to a safe index position utilizing buttons **ZERO/RET AND AUTO/ALL AXES OR HOME/G28.** Index to tool #1 in MDI mode (pressing **MDI/DNC). Key** in **T1** and press **TURRET/FWD** or **TURRET/REV** key**.**

Start spindle while in the Jog Mode (press **HANDLE/JOG),** key in 800 and press spindle **FWD key**. This will start the spindle rotating at 800 rpm.

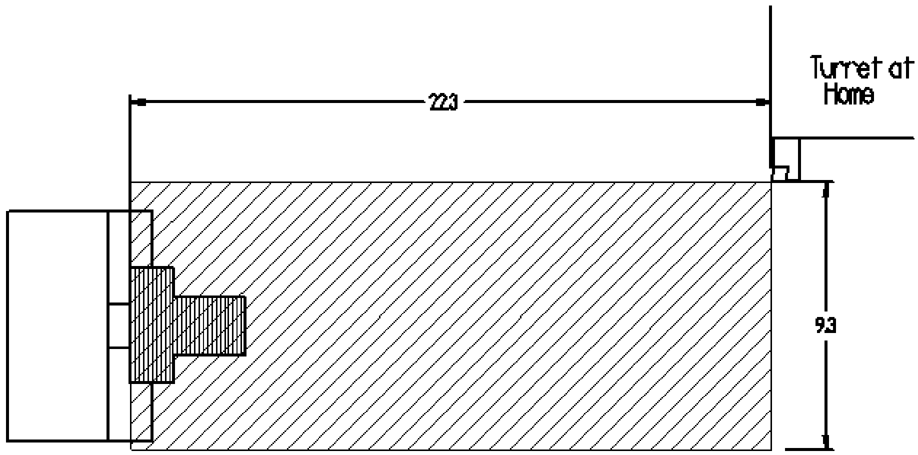
Manually position the tool in front of the part to take a skim cut on the outside diameter using the hand wheel in the **HANDLE/JOG** mode. Take a skim cut by moving turret in the Z direction, then off the part in Z so an outside diameter measurement may be made with a micrometer. Go to Tool Geometry page by pressing **OFSET.** Cursor to tool number 1 and press **X DIA MESUR.** The control will prompt for a diameter. Key in the value of the outside diameter measured and press **WRITE/ENTER.** The control will then calculate the position from home to the centerline of the part (Tool Geometry for X) by adding the current machine coordinate position of the turret to the negative value of the diameter which was measured.

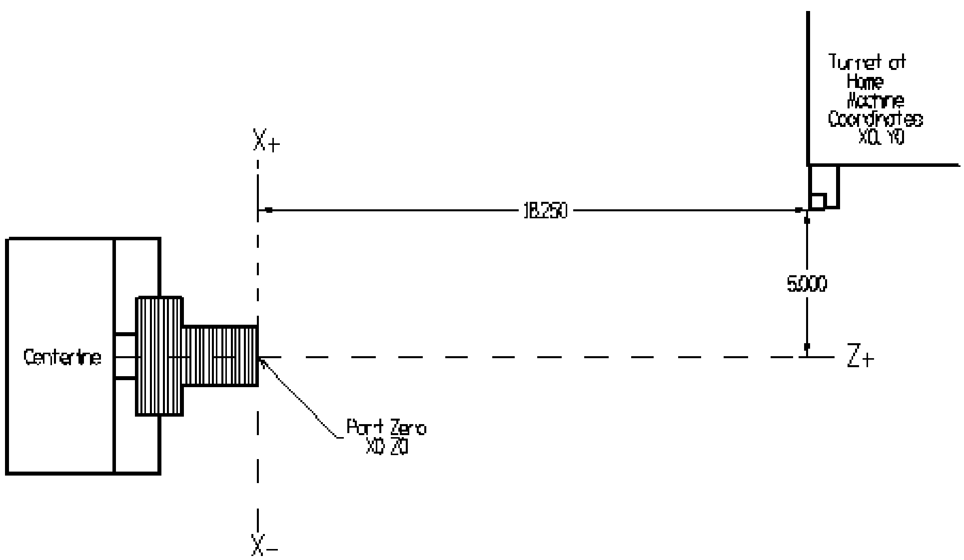
Next manually position turret to take a skin face cut and face off the front of the part. Move tool off the part in X then press **Z FACE MESUR.** The control will record the current machine coordinate in Z into the Z register for tool # 1. The rest of the OD tools may be determined in a similar fashion.

Drills need not be touched off as centerline has been established when the machine was initialized. To enter the established centerline, cursor to the **X** column for the drill in the offset page. Press **F2 in Handle Jog mode.** Z geometry for drills needs to be determined by touching off on the face of the part. ID tools need a drilled hole to be determined. Measuring the inside diameter by taking a cut or touching off and using **X DIA MESUR** will determine the X geometry. Z geometry for ID tools is determined in similar fashion as OD tools.

Below shows the total travel capability of a ST20YSS lathe.   Note the tool shown below can go past the centerline of the part to the other side of the chuck. The reason for this is so the live tooling on the face of the lathe can get to and past the centerline of the part.

Total travel of ST20YSS Lathe shown in crossed hatch





Above shows the dimensions in X and Z from home position to the face and centerline of the part in the chuck. Manually touching off on the face and a known diameter will give the following values using the Z Face Measure key and the X Diameter Measure key respectfully in the Tool Geometry Page for Tool #1. Note the 5.0 dimension from above has been taken times 2. This is because all offset values in X are in **Diameters.** Also all X's in the program are in diameters.

(Tool Geometry)
T 1   X-10.   Z-18.25

On the Work Zero Offset page the **Z** in whatever work offset you are using **must be set to 0** if you are **manually touching off**. Normally the default is used G54. On the **work offset page the X must always be set to zero.**

Work Zero Offset
G54 X0 Z0

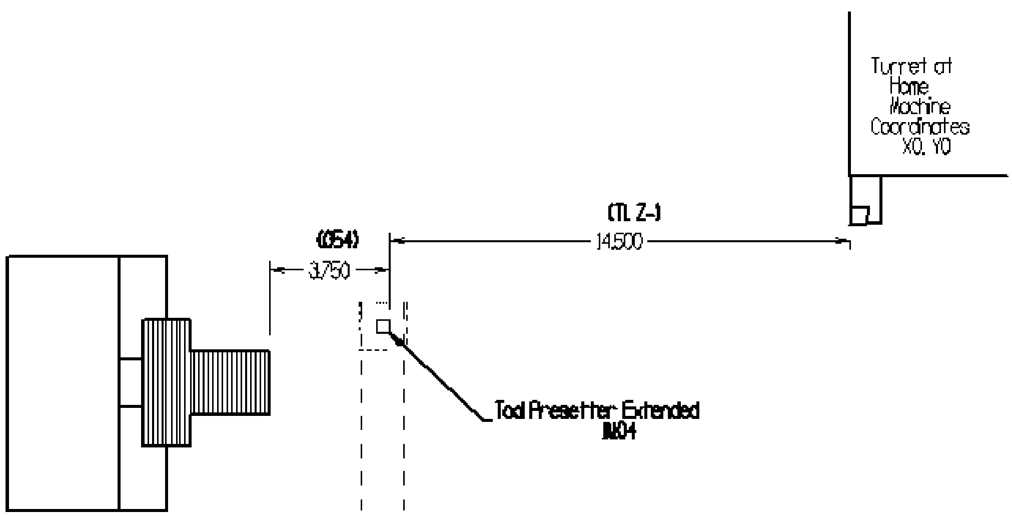
Below shows the same part but the tool has been touched off using the probe. Note the tool geometry in X is the same as using the manual method. The Z however is much smaller. The difference is made up in the work zero offset in Z. (-14.25 plus -3.75 = -18.25).

The advantage of using the tool presetter is that is faster than manually touching off on the face or od of a part. Also once the tools are set using the probe for a different length part all that needs to be done is reset the Work Zero Offset in Z.

Using the **manual** method all tools must be **retouched off in the Z** for a different length part.

(Tool Geometry)
T 1 X-10. Z-14.25

Work Zero Offset
G54 X0 Z-3.75



Probe Touch-Off Method using Work Offset:

Swing down probe by using **M104** in MDI (**M105 retracts)**. Manually bring tool close to center of probe in the X. Switch Handle/Jog to .001 increment and press **--X** jog key until a beep is heard. Reverse directions by pressing the **+X** key. Switch Handle/Jog increment to .0001. Then press and hold **–X** jog key until a beep is heard. A value will automatically be entered into X register for the particular tool.

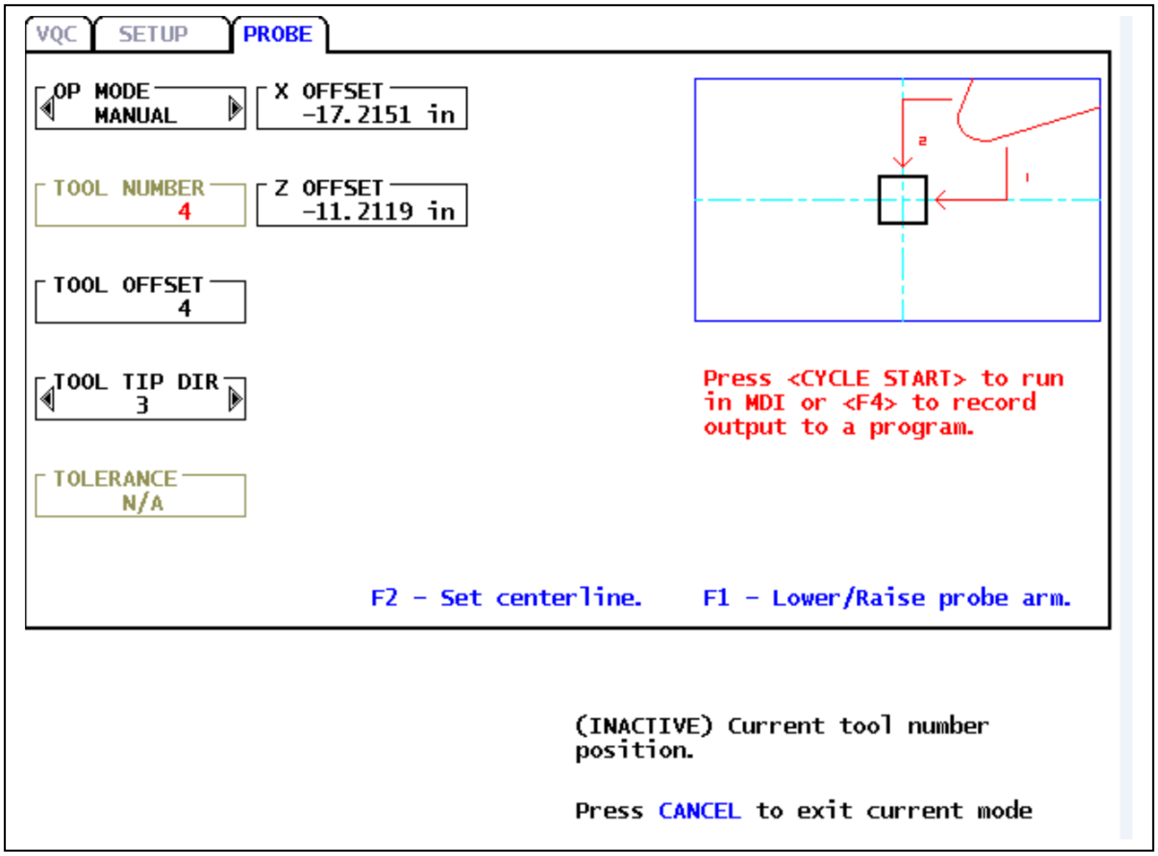
Z geometry values are determined in a similar fashion by manual movement and tough off in the Z direction. A similar procedure is used with ID tools. Only the ID side of the probe is touched off on. Drills are touched off in a similar fashion in the Z. Use **F2** to determine the X geometry for drills in handle jog mode. Drill geometry in **X** may also be set by sweeping in the drill pocket and entering the X machine coordinate into the X geometry column of the offset page for the drill.

Correct values for the probe are determined when the machine is installed. Over time and after a crash the probe may need to be recalibrated. A calibration procedure may be found on page ## of this manual.

When **using the probe** for touching off all the tools in the lathe **a work offset must be determined**. When the face of the part is used to touch off the tools all the Zs the work zero offset will be 0. However, when using the probe method the distance from machine home to the probe and the face of the part are different. To determine this distance press **OFSET** until the Work Zero Offset register appears. Highlight the Z in Work Offset to be used (G54 to G59) by using the up and down arrow cursor. Manually move any active tool up to the face of the part to be machined. Using a piece of paper, touch off the face. Press **Z FACE MESUR.** The correct work offset will be placed in the Z column. Normally the X column is always left at zero.

Automatic Tool Setting Probe

To access the automatic tool setting probe page first press the MDI key, then depress the Program Conversational key. Then cursor to the PROBE tab and press the Enter key. The folowing panel will then be displayed on your control. To change to a box use the cursor up and down keys until the text appears red.



OP MODE: Gives selection of Manual, Automatic and Break Det. Modes using the right and left cursor key. All tool setting must be first set up in **Manual** mode.

TOOL NUMBER: Automatically comes up in the Manual mode as the current tool number. May be changed in automatic or break detection modes.

TOOL OFFSET: Automatically comes up as the current tool. This value may be changed.

TOOL TIP DIR: Use right and left cursor to change. It gives up to 8 different options. Above tool tip 3 is selected. A picture of the tool tip type shows up in the upper right hand corner. Above it is set to normal OD turning option, tool type 3.

TOLERANCE: Above in gray only accessible when break detection mode is active.

X OFFSET, Z OFFSET: Read only. Displays the current offset values.

F1: Pressing lower or raises the probe from its resting location.

Manual Mode Operation: <u>This is the mode that **must performed 1st**. It is also the one most used.</u>

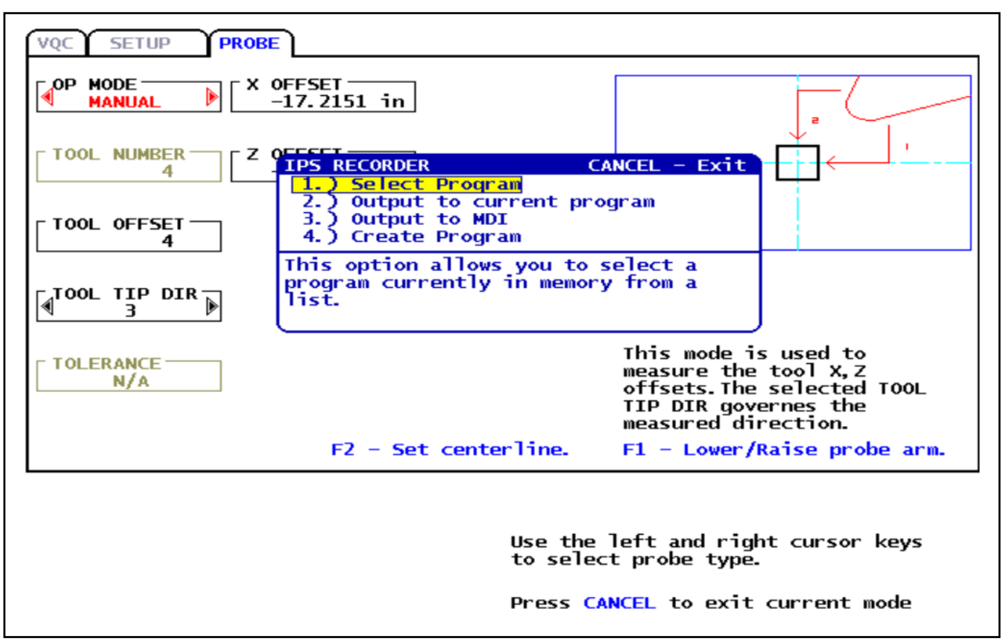
- **I)** Select tool to be touched off by indexing using Turret Forward of Turret Reverse key.
- **II)** Lower the probe using the F1 key
- **III)** Select the **Manual Mode** in the OP Mode box
- **IV)** Key in Tool Offset if different than the one used in your program
- **V)** Using cursor keys set the correct Tool type.
- **VI)** Put machine in the Hand Jog mode and move the tool to within .25" of the probe. Look at the picture to locate it in the proper location.
- **VII)** Press Cycle Start key. The tool will be touched off and the offset recorded.
- **VIII)** Return the probe to its home position by depressing the F1 key.

Automatic Mode: This mode can be used only <u>after the initial manual mode cycle</u> has been run for a particular tool.

To run the automatic mode, go the probe page. Enter the tool and tool offset number. Then press cycle start. The machine will automatically index to the particular tool and rapid to the start position of the probing operation and run the routine.

The cycle may be used to update tool offsets after tool wear or after an insert is replaced.

All the above operations are generated and run in MDI mode. If the code needs to be loaded inside a program or a new program created pressing F4 will bring up the IPS recorder. Here you can select a program to insert it into, output into current program or create a new program. After the recipient program has been called up, cursor to the location inside the program and press the Insert key. The code then will be copied to that location.



Break Detection Mode: This mode can be used <u>only after the initial manual mode cycle has been run for a particular tool.</u>

In the break detection mode all values should be already set as in the Auto mode. The only thing that needs to be set is the desired tolerance to be checked. As in the automatic mode the code created may be copied into a program stored in memory using the F4 key to call up the IPS recorder.

Absolute and Incremental Positioning

By using our WORK OFFSETS we can establish a common point on our part as a "ZERO". This is some point on our part that we can physically find. The programmer uses this point as a base to write the intended movement of our tooling.

The most common method is to use the front end of our finish machined part (**Z Zero**) and the centerline of our part (**X Zero**).

There are two methods used by the programmer to "Steer" our machine. The first is "**ABSOLUTE POSITIONING**". Absolute means that we input code that is based on this **ZERO POINT** on our part. If we want a diameter of 1.0000 inches, it is input as X1.0000. If we need to face a shoulder that is 3 inches back from the front of the part, we input Z-3.0000

The programmer has another tool available to him called "**INCREMENTAL POSITIONING**". This is movement based on where the machine is currently sitting. If we wanted to change the diameter of the machine from where it is currently sitting a half a inch smaller, we would input U-.5000. If we had a grooving tool making a groove that is located ¾" behind a groove we already finished, we can input W-.7500

The letters X&Z represent ABSOLUTE POSTIONING

The letters U&W represent INCREMENTAL POSTIONING

If you are familiar with the mill programming language, absolute and incremental are handled differently for Mills and Lathes. A mill uses G codes (G90 and G91) to go back and forth between the two. Where as a lathe uses the different letters to differentiate them.

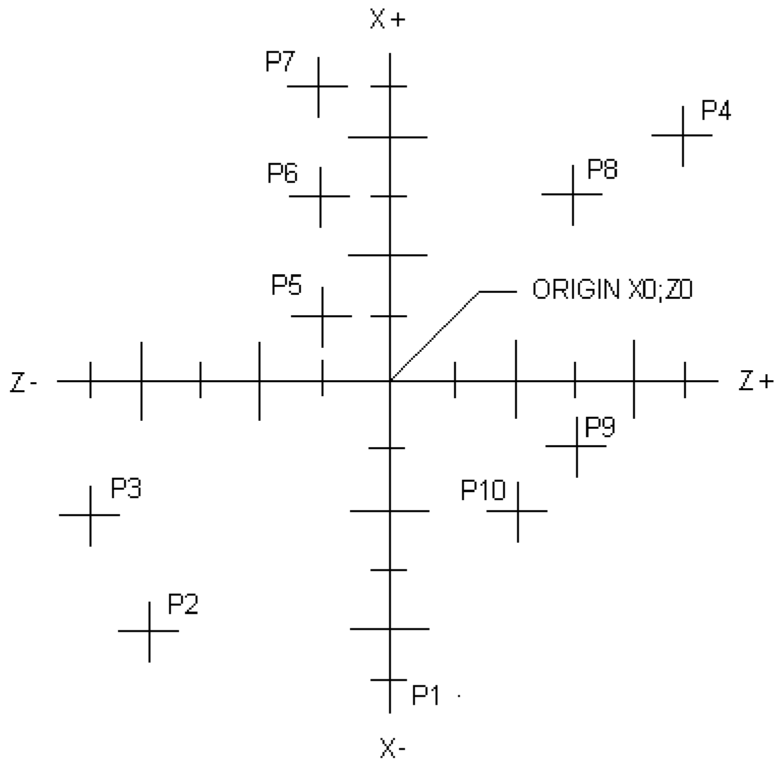
QUESTION:
Why doesn't a lathe take G90 and G91 like a mill?

ANSWER:
A lathe has the unique possibility to do Absolute AND Incremental moves AT THE SAME TIME.

The programmer can place an ABS. letter and an INC. letter on a line of code together, and is most commonly used in making tapers and radius moves.
G01 X2.000 W-.25 (Move X in Absolute, Z in Incremental)
or
G01 U.5000 Z-.5000 (Move X in Incremental, Z in Absolute)

All X and U dimensions are diametric!

THE CARTESIAN COORDINATE SYSTEM



Coordinate exercise: Give X and U values as diameters.

ABSOLUTE

P1	***X-5.0***	***Z0***
P2		
P3		
P4		
P5		
P6		
P7		
P8		
P9		
P10		

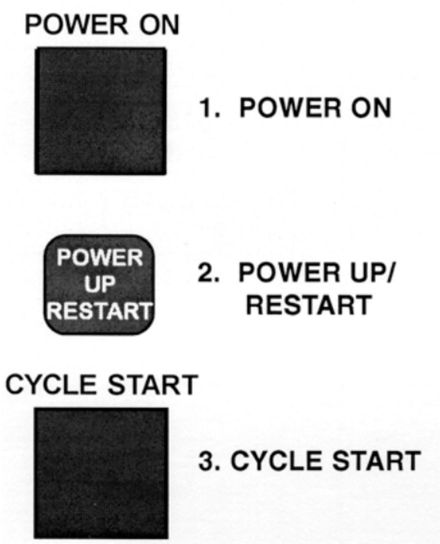
INCREMENTAL

P1	***U-5.0***	***W0***
P2		
P3		
P4		
P5		
P6		
P7		
P8		
P9		
P10		

The Haas CNC Control

Powering On the Machine

To power up a Haas machine, regardless of where the machine turret was when it was turned off, press **POWER ON**. The machine must first find its fixed machine zero reference point before any operations can occur. After it's powered on, pressing **POWER UP/RESTART** will send the machine to its machine zero reference location. The machine doors must be cycled and closed to return to machine zero. Also the machine needs to see the Emergency Stop cycled. Haas provides directions on the screen on what needs to be done to start the machine up in the morning.



When powering on the machine, if there is a message in the **MESGS** display, it will be the first display seen on your control screen.

Will move all axis to machine zero and then indexes the turret to tool #! Machine will move up in X first to machine zero and then the Z move to machine zero.

If the correct program has been selected and the part program is proven to be good and it's ready to run, press cycle start.

General Machine Keys

Power On - Turns CNC machine on.

Power Off - Turns CNC machine tool off.

Emergency Stop - Stops all axis motion, stops spindle, tool changer and turns off coolant pump.

Jog Handle – Jogs axis selected, also may be used to scroll through programs, menu items while editing and also altering feeds and speeds.

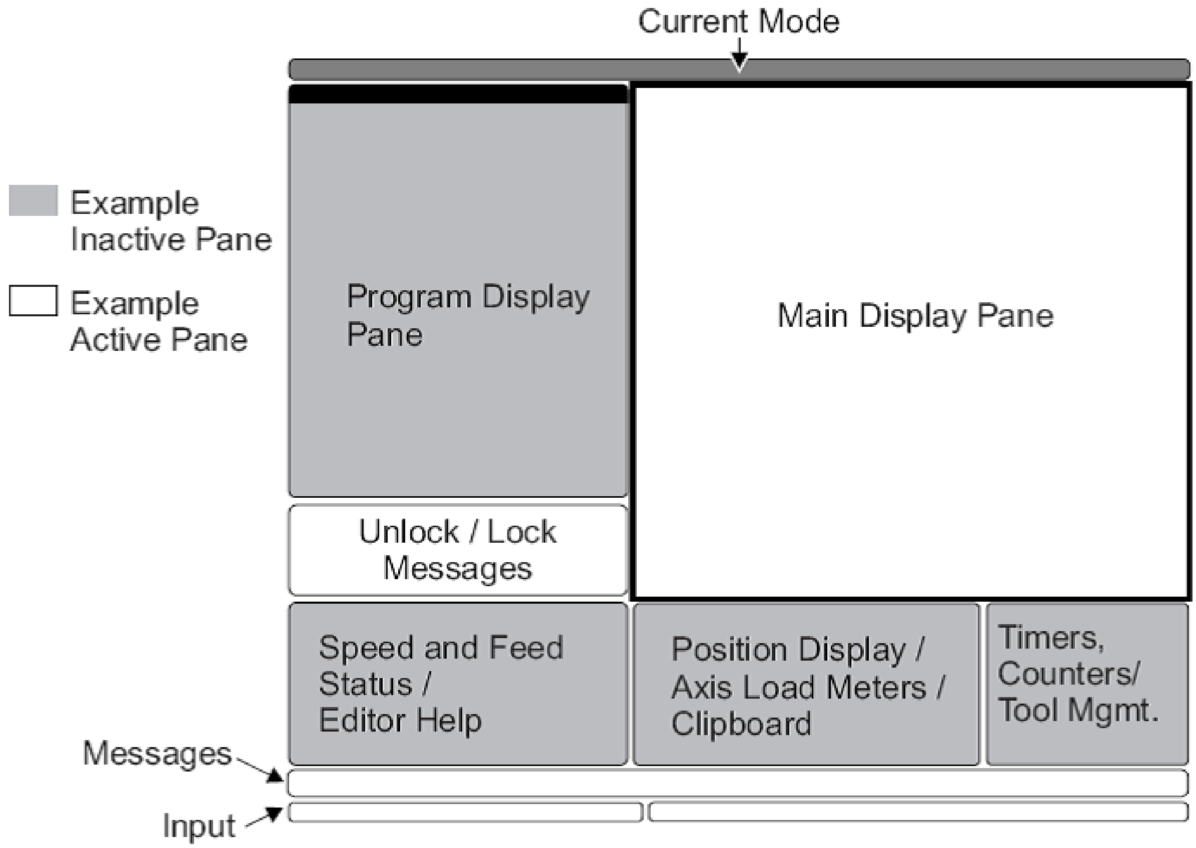
Cycle Start – Starts program in run mode or graphics mode.

Feed Hold – Stops all axis motion. Spindle will continue to turn**.**

Reset – Stops machine, will rewind program.

Power Up/Restart – Axis will return to machine zero and tool change will occur per Setting 81
Recover – If a tool change is stopped in middle of a cycle an alarm will come up. Push the **Recover** button and follow the instructions to bring the tool change cycle to the beginning.

Control Display



The new 16 software has a larger display and more panes than older versions. Above is the basic display layout. What is displayed depends on which display keys have been used. The only pane active is the one with the white background. Only when a pane is active may changes be made to data.

Control functions in Haas machine tools are organized in three modes: **Setup, Edit** and **Operation.**

Access Modes using the mode keys as follows:

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

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

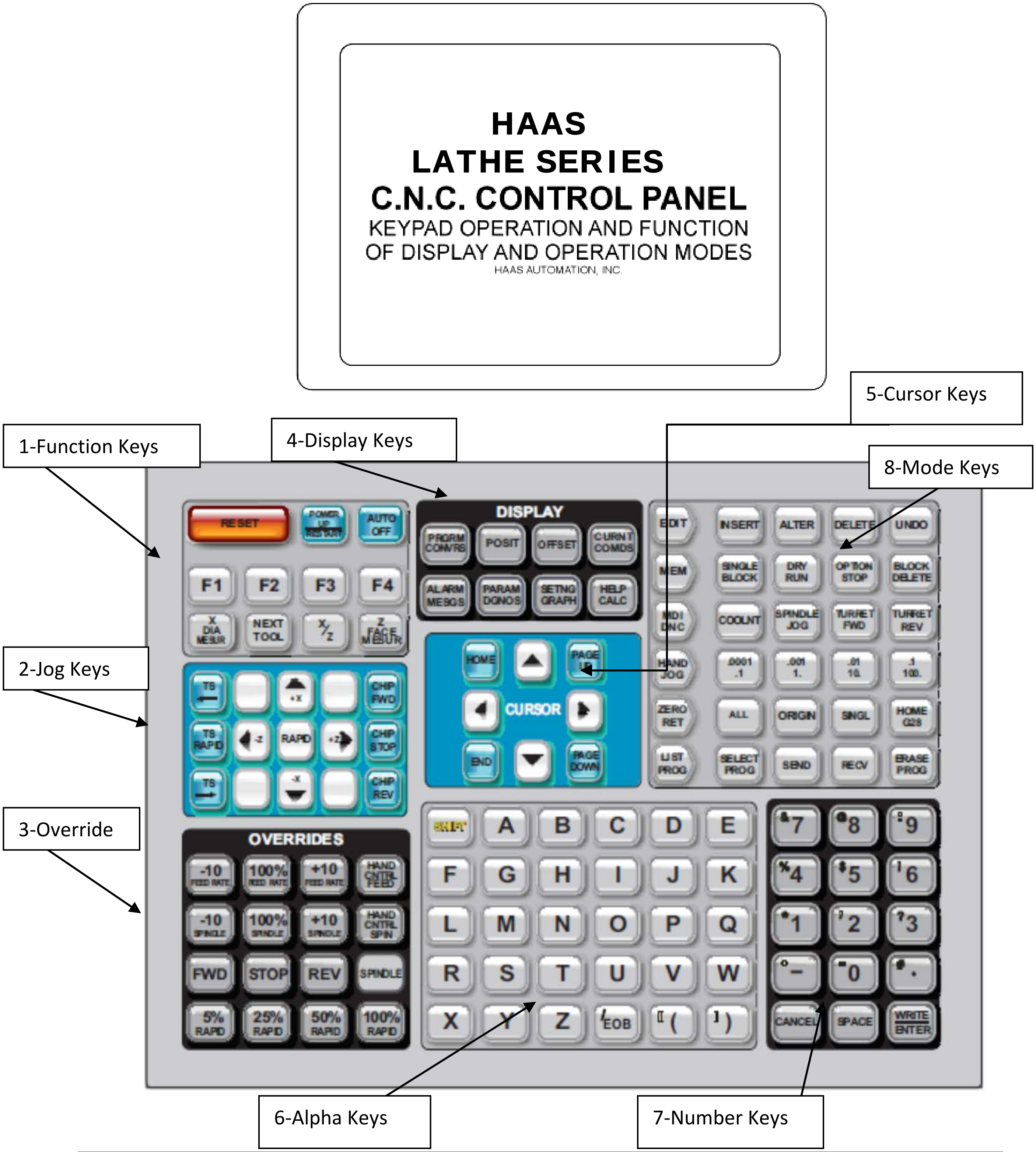
Operation: MEM key. Provides all control features necessary to make a part.

Current mode is displayed at top of display.

Functions from another mode can still be accessed within the active mode. For example, while in the Operation mode, pressing OFFSET will display the offset tables as the active pane in the Main Display Pane and offsets may be altered; press OFFSET to toggle the offset display. While running a part in operation mode another program may be edited in the Main Display Pane. Press PROGRM CONVRS in most modes to shift to the edit pane for the current active program.

Keyboard Introduction

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



1 – Function Keys

F1 – F4 – Perform different functions depending on which mode the machine is in. Example in offsets mode **F1** will directly enter value given it into offset geometry.

X DIAMETER MEASURE – Will take machine X position ask for a diameter measurement on the part which tool turned and put correct X Geometry in Tool Offsets page.

NEXT TOOL – In set up this will select the next tool and make a tool index.

X/Z - Toggles between X-axis and Z-axis jog modes during a set up.

Z FACE MEASURE – Used to record Z tool offsets and Z work offsets**.**

2 – Jog Keys

Chip FWD *(Chip Conveyer Forward)* – Turns the chip conveyer in a direction that removes chips from the work cell.

Chip STOP *(Chip Auger Stop)* – Stops chip conveyer movement.

Chip REV *(Chip Auger Reverse)* – Turns the chip conveyer in reverse.

<-TS – Moves tailstock toward the spindle.

TS Rapid – Increases speed of tailstock movement when used concurrently with the other **TS** keys.

->TS - Moves tailstock away from spindle.

+X, -X *(Axis)* Selects the X axis for continuous motion when depressed.

+Z, -Z *(Axis)* Selects the Z axis for continuous motion when depressed.

Rapid – When pressed simultaneously with X or Z keys will move at maximum jog speed.

3 – Override Keys

The overrides are at the lower right 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.

-10 FEED RATE Decreases current feed rate in increments of 10 percent.
100% FEED RATE Resets the control feed rate to the programmed feed rate.
+10 FEED RATE Increases current feed rate in increments of 10 percent.
HANDLE CONTROL FEED RATE Hand wheel will control feed rate at 1% increments.

-10 SPINDLE Decreases current spindle speed in increments of 10 percent.
100% SPINDLE Sets the control spindle speed at the programmed spindle speed
+10 SPINDLE Increases current spindle speed in increments of 10 percent.
HANDLE CONTROL FEED Hand wheel will control feed rate at 1% increments.

CW Starts the spindle in the clockwise direction.
STOP Stops the spindle.
CCW Starts the spindle in the counterclockwise direction.

5% RAPID Limits rapid moves to 5 percent of maximum.
25% RAPID Limits rapid moves to 25 percent of maximum.
50% RAPID Limits rapid moves to 50 percent of maximum.
100% RAPID Allows rapid traverse to feed at its maximum.

Override Usage

Feed rates may be varied from 0% to 999%. Feed rate override is ineffective during G74 and G84 tapping cycles. Spindle speeds may be varied from 0% to 999%. Depressing Handle Control Feed rate or Handle Control Spindle keys, the jog handle movement varies by +/-1% increments.

Setting 10 will limit rapid movement to 50%.

Settings 19, 20, 21 make it possible to disable override keys.

Coolant may be over rode by depressing **COOLNT** button.

Feed Hold - Stops rapid and feed moves. **Cycle Start** button must be depressed to resume machine feeds. Similar situation applies when Door Hold appears. Door must be closed and **Cycle Start** pressed to continue running program.

Overrides may be reset to defaults with a M06, M30 or pressing **RESET** by changing Settings 83, 87 and 88 respectively.

4 – Display Keys

PRGM/CONVRS – Selects the active program pane (highlights in white). In MDI/DNC mode pressing a second time will allow access to VQC (Visual Quick Code) and IPS (Intuitive Programming System)

POSIT *(Position)* **–** Selects the positions display window (lower middle). Repeated pressing of the **POSIT** key will toggle through relative positions in the Memory Mode. In Handle Jog mode all four are listed together.

1. **POS-OPER** digital display. This is a reference display only. Each axis can be zeroed out independently; then the display shows the axis position relative to where you decided to zero it. In the Handle Jog mode, you can press the X, Y or Z JOG keys and ORIGIN key to zero that selected axis. On this display page, you can also enter in an axis letter and number (X-1.25) and press ORIGIN to have that value entered in that axis display.

2. **POS-WORK** digital display. This position display tells how far away the tools are in X, Y and Z from the presently selected work offset zero point.

3. **POS-MACH** digital display. This is in reference to machine zero, the location that the machine moves to automatically when you press POWER UP/RESTART. This display will show the current distance from machine zero.

4. **POS-TO-GO** digital display. When you're running the machine, or when you have the machine in a Feed Hold, this incrementally displays the travel distance remaining in the active program block being run. This is useful information when you are stepping a program through during a set up.

When the position pane is active one can change which axis is displayed simply by typing X or Y or Z or any combination and pressing write. Then only that particular axis or combination will be displayed.

OFFSET – Selects one of two offsets tables: Tool Geometry/Wear and Work Zero Offset. Depressing the **OFFSET** button toggles between the two tables Tool Geometry/Wear table displays 50 tool length offsets (100 tool length offsets on older machines) - labeled (LENGTH) GEOMETRY - along with wear offsets. It also displays radius and tool tip type.

The Work Zero Offset table has G54-G59 plus G154 P1 to G154 P99 offsets available.

The **WRITE/ENTER** key will add the number in the input buffer to the selected offset, and the **F1** key will replace the selected offset with the number entered into the input buffer. Offsets can also be entered using **TOOL OFSET MEASUR** and **PART ZERO SET**

CURNT COMDS – Ten different pages; use **PAGE UP** and **PAGE DOWN**

1. Operation Timers – displays Power-On Time, Cycle Start Time, Feed Cutting Time. Hitting **ORIGIN** will clear any display that is highlighted by the cursor.

2. Real time clock and date.

3. System Variables, for machines with Macro Programming.

4. All Active Codes, displays current and modal command values.

5. Position information: Machine, Distance to Go, Operator, Work Coordinate.

6. Tool life, displays the usage of each tool. An alarm can be set for the number of times you want that tool to be used, and when that condition has been met (that is, the tool has been used the set number of times), the machine will stop, with an alarm for you to check the condition of that tool. Pressing **ORIGIN** will clear the cursor-selected display, and pressing **ORIGIN** when the cursor is at the top of a column will clear the whole column.

7. Tool Load displays the Tool Load Max % of each tool being used. You can use the Limit % column to set the maximum spindle load for a particular tool. When that condition has been met (the tool has reached maximum load), the machine will stop and alarm out for you to check the condition of that tool. Pressing **ORIGIN** will clear the cursor-selected display, and pressing **ORIGIN** when the cursor is at the top of a column will clear the whole column. Setting 84 determines the Overload Action when this limit is met.
   Also vibration loads may be entered.

8. Maintenance times for various items may be loaded.

9. Bar Feeder 300 – Haas servo bar system variables displayed

ALARM/MESGS – Displays messages and current active alarms. Press right arrow key gives alarm history. Press right arrow key again goes to the Alarm Viewer Page. Enter alarm number and press write will give detailed information on a particular alarm code.

PARAM/DGNOS – Lists machine parameters that are seldom-modified values which change the operation of the machine. These include servo motor types, gear ratios, speeds, stored stroke limits, lead screw compensations, motor control delays and macro call selections. All of these are rarely changed by the user and should be protected by Setting 7, PARAMETER LOCK. A second press of **PARAM/DGNOS** will show the diagnostics display. The **PAGE UP** and **PAGE DOWN** keys are then used to select one of two different pages. This display is for service diagnostic purposes, and the user will not normally need them.

SETNG/GRAPH – Displays settings - machine parameters and control functions that the user may need to turn on and off or change to suit specific needs. A list of settings is found on page 30.

- Settings are organized into functionally similar page groups with a title.
- Settings are listed with a number and a short description, and a value or choice on the right.
- To find a particular setting, enter the setting number and then press either the up or down cursor arrow key to move to the desired setting.
- You can change a setting using the left or right cursor arrows to display the choices, or, if the setting contains a value, by typing in a new number. A message at the top of the screen will tell you how to change the selected setting. When you changed, it will flash on and off.
- A setting change is not active until it stops flashing. To activate, press **WRITE/ENTER**.

SETNG/GRAPH (2nd part) - The second press of SETNG/GRAPH will bring up the **graphics display** in the Main Display Pane. In this screen you can dry-run a program without moving the axes or risking tool damage from any programming errors. This function is far more powerful than using **DRY RUN**, because all of your offsets and travel limits can be checked before any attempt is made to move the axes. The risk of a crash during setup is greatly reduced. The **Graphics Screen** will display the programmed tool path and generate an alarm if there are any problems. Some of the features of the Graphics display are controlled by selections made in the Settings display, on the page titled GRAPHICS.

1. Press either **MEM** or **MDI** and select the program that you want to run in Graphics. Graphics will also run in the Edit Mode.
2. Press **SETNG/GRAPH** twice.
   - The top left line of the screen will list the GRAPHICS title. Above that line will list the mode you are in (MEM or MDI). The bottom lists explanations for use of function keys **F1** through **F4**.
   - The small window on the lower right side of the screen displays the whole table area during the simulation run, indicating the location of the tool and any zoom window. The center window of the display is a large window that represents a top-down perspective of the X and Y axes. This is where the tool path is displayed during graphic simulation of a CNC program.
3. Press **CYCLE START** to see all the X and Y-axis moves demonstrated.
   Note machine axis and spindle will not when graphic window is up.
4. To step through a program one block at a time in Graphics, press **SINGLE BLOCK**.
5. **F1** is a help key.
6. Press **F2** to zoom in on the Graphics view screen.
   - Use **PAGE DOWN** to zoom in further and **PAGE UP** to expand the view.
   - Use the **Cursor Keys** to position the new zoom window over the area you wish to zoom in on using the small window in the bottom right hand corner. Pressing **HOME** will display the whole table.
   - After positioning the desired zoom window, press **WRITE/ENTER** to accept the view and CYCLE START to see the new view.
   - F3 slows the execution speed of the graphic simulation
   - F4 speeds up the execution speed of simulation.

Use **SINGLE BLOCK** to step through a program in graphics to find any mistakes. During single block you can re-zoom your window to look at tool paths in tight corners etc. Also use position display to see find any discrepant values.

HELP/CALC – Will bring up a help POP UP relevant to the screen you are in. This provides information only pertaining to that screen. Pressing the **HELP/CALC** button again brings up a tabbed menu. With tabulated screens highlighting tab and pressing **WRITE/ENTER** key will open up respective tab. Pressing the **CANCEL** key will close the tab.

Help Opening up the Help tab brings you to the table of contents of the entire Mill Operators Manual. High light the topic of interest and press **WRITE/ENTER** will bring up subtopics on the area of interest. Select subtopic in similar fashion will bring up the relevant page in the manual.

Search The search tab will do a search of the manuals content for relevant information on a keyword. Type in the search term and press **F1.** Topics relevant to the keyword will appear. Highlight the topic and press **WRITE/ENTER** key to open.

Drill Table Displays a common drill sizes, decimal information and tap drill sizes.

Calculator Different calculator functions are available under this tab. The calculator gives ordinary calculations like addition, subtraction, multiplication and division in all tabs. It also will solve trig problems with information about triangles, circles, circle line tangent and circle- circle tangent. A milling and tapping tab will give you suggested cutting speeds and feeds per different materials and sized tools.

Simple Calculator It will calculate simple addition, subtraction, multiplication and division operations. Operations
are listed as: LOAD + - * /. These are selected using the left or right cursor arrow.

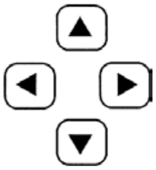
- To enter a number cursor on to LOAD; type the number you want to load and press **WRITE/ENTER**.
- To perform one of the arithmetic functions, enter the first number into the calculator window. Select the operation you want (+ - * /). Finally, enter the second number into the input buffer, press **WRITE/ENTER** to perform the calculation.

Milling and Tapping Help you solve values for feed rates SFM, RPM, and chip load under different conditions. It uses the three equations related to milling and tapping. The first one includes cutter diameter with SFM and RPM. The second one includes RPM, number of flutes, feed rate and chip load. The third one includes thread pitch, RPM and feed rate.

The Milling & Tapping Tab

MILLING:	Cutter Diameter	1.2500 IN	(entered)
	Surface Speed	210.0000 FT/MIN	(entered)
	RPM	**642**	**(calculated)**
	Flutes	4	(entered)
	Feed	**12.8343 FT/MIN**	**(calculated)**
	Chip Load	0.0005 IN	(entered)
TAPPING:	Threads	16.0/IN	(entered)
	RPM	500	(entered)
	FEED	**31.2500 IN/MIN**	**(calculated)**

5 – Cursor Keys



Cursor Keys The cursor keys are in the center of the control panel. They give the user the ability to move to and through various screens and fields in the control. They are used extensively for editing and searching CNC programs. They may be arrows or commands.

Up/Down – Moves up/down one item, block or field.
Page Up/Down – Used to change displays or move up/down one page when viewing a program.
HOME – Will move the cursor to the top-most item on the screen; in editing, this is the top left block of the program.
END – Will take you to the bottom-most item of the screen. In editing, this is the last block of the program.

6 and 7 – Alpha Keys and Numeric Keys

The **Alpha Keys** allow the user to enter the 26 letters of the alphabet along with some special characters. Depressing any Alphabet Key automatically puts that character in the Input Section of the control (lower left-hand corner).

SHIFT key provides access to the yellow characters shown in the upper left corner of some of the alphanumeric buttons on the keyboard. Pressing **SHIFT** and then the desired white character key will enter that character into the input buffer.

EOB key enters the end-of-block character, which is displayed as a semicolon on the screen and signifies the end of a programming block. It also moves the cursor to the next line.

Parentheses are used to separate CNC program commands from user comments. They must always be entered as a pair. Example: (T1 ½” End Mill)

Also any time an invalid line of code is received through the RS-232 port, it is added to the program between parentheses.

(–) and **(.)** These keys are used to define negative numbers and give decimal position.

+ = # * [ ] These symbols are accessed by first pressing the **SHIFT** key and then the key with the desired symbol. They are used in macro expressions (Haas option) and in parenthetical comments within the program.

, ? % $! & @ : These are additional symbols, accessed by pressing the **SHIFT** key, that can be used in parenthetical comments.

6 and 7 – Alpha Keys and Numeric Keys (continued)

The **Numeric Keys** allow the user to enter numbers and a few special characters into the control. Depressing any number key automatically puts it into the Input Section of the Control.

Cancel The **Cancel** key will delete the last character put into the Input Section of the control display.

Space Is used to format comments placed into the Input Section of the control display.

Write/ Enter General purpose "Enter" key. It inserts code from the input section into a program when the program display is in EDIT mode. With offsets pages active, pressing the **WRITE/ENTER** key adds a number in the Input Section to the highlighted cell. Pressing the **F1** key will input the number into the cell.

- The (Minus Sign) is used to enter negative numbers.

. The (Decimal Point) is used to note decimal places.

8 – Mode Keys

Mode keys set the operational state of the machine tool. Once a mode is set the keys to the right may be used. The current operation mode of the machine is displayed at the top thin pane of the CRT.

EDIT The edit mode is used to make changes in a program stored in memory. When you press **EDIT** two panes appear at the top of the screen. In the left pane the active program appears. In the right an inactive program appears or the select program screen appears. On the bottom left a editor help pane appears and on the right a clipboard pane. Editing may be performed in either the active or inactive panes. Pressing **EDIT** toggles between the two panes, (changes background to white). To call up a program from memory and put it in one of the edit panes press **SELCT/PROG.** Highlight the program desired by using the up or down cursor buttons and press **WRITE/ENTER.**

In the edit mode you are able to use the edit keys in the **same row** as the **EDIT** key.

INSERT Enters commands keyed into the input panel in lower left pane of CRT after the cursor highlighted word in a program.

ALTER Highlighted words are replaced by text input into the input panel.

DELETE Highlighted words are deleted from a program.

UNDO Will undo up to the last 9 edit changes**.**

F1 KEY While in the edit mode pressing **F1** will bring up an edit pop up window. Using the sideways cursor buttons will toggle thru HELP, MODIFY, SEARCH, EDIT AND PROGRAM MENUS. The up and down buttons will cursor thru the different options in each of the above.

MODIFY Gives options on changing line numbers.

SEARCH Will perform a search and gives the option of replacing text.

EDIT Gives option of cutting or copying and pasting to a clipboard and to another program.

PROGRAM Gives options of creating new program, selecting a program from list to edit, duplication of programs, switching from left to right side of window panes.

Background Edit When a program is being run pushing the edit will bring up the Background Edit pane in the Main Display Pane. Simple edits may be performed on the program that is being run or another program. The edits on the running program will not take place until after the current cycle has completed.

<u>MEM</u> The memory mode is the mode used when running the machine and making a part. The active program is shown in the Program Display Pane. Keys in the memory mode line reflect different ways of running a part in memory. When the keys to the right are depressed they will show up highlighted in black on the bottom right of the CRT.

SINGLE BLOCK When depressed **SINGLE BLOCK is** highlighted in black and will appear on the bottom of the CRT. When the machine is in SINGLE BLOCK mode only one block of the program is executed every time the cycle start button is depressed. Used when first test running a program or temporarily stopping a program when it is running.

DRY RUN Used to check machine movement without cutting a part. In dry run the machine runs at one feed rate. With the availability of graphics which show visually what the machine tool path is this mode is rarely used.

OPTION STOP When **OPTION STOP** is depressed program will stop at any M01 which is in the program. Normally M01s are placed after a tool is run in a program. When a job is being set up the operator may put machine in op stop mode to check dimensions after every tool has completed cutting.

BLOCK DELETE When this button is depressed any block with a slash (/) in it is ignored of skipped.

<u>MDI</u>
DNC **(**MANUAL DATA INPUT mode) – Usually short programs are written in MDI but are not put into memory. DNC mode allows large programs to be drip fed from a computer into the control.

COOLNT Turns coolant on and off manually

ORIENT SPINDLE Rotates and locks spindle to specific angle. Used when lining up tools where spindle orientation may be a issue such as boring heads.

ATC FWD Rotates turret to next tool and performs tool change - also used to call up specific tools or pots. Enter tool number (T1) and press **ATC FWD**.

ATC REV Rotates turret to previous tool and performs tool change - also used to call up specific tools or pots. Enter tool number (T1) and press **ATC REV**.

HAND JOG Puts machine in jog mode for set ups. Top values (.0001, .001, .01, .1) represent distance traveled per click of jog handle. Bottom values (.1, 1., 10., 100) represent feed in inches/minute when jogging axis using jog buttons.

ZERO RET On pressing position display becomes highlighted in Zero Return mode.

ALL Returns all axes to machine home similar in similar fashion as a Power Up/Restart.

ORIGIN Sets selected displays to zero or other functions.

SINGL Returns a single axis to machine home. Select desired axis (X, Y, or Z) then press **Singl** axis button.

Home/G28 Rapid motion to machine home; will make a rapid move in all axes at once - may also be used for a rapid home in one-axis. Press axis to home then **G28.** ***Caution must be used that extended tools, tailstock or parts are out of the way before initiating this rapid move to home.***

LIST PROG Will bring up list of programs in a tab format. Pressing **Cancel** will return you to tab at top usually MEM or USB. Cursor to left or right for which list one wants. Pressing **Enter** will open a list of programs. Cursor UP (∧) or DOWN (∨) to program desired. Select the desired programs to be moved by pressing **WRITE/ENTER.** This will put a check mark beside it. **F2** will copy selected program or programs to be moved. A pop up menu will ask where you want the selected programs to be copied.

SELECT PROG After highlighting a program from List Program with up or down cursor pressing this button will place the program in the Active Program Pane. This is the program that will run the CNC machine in the Memory mode. Use in the Edit mode in the Main Display will enter selected program in the Main Display pane for editing.

SEND Will send a selected program or programs out thru RS-232 serial port

RECV Will get machine ready to receive program from RS-232 serial port.

ERASE PROG Will erase highlighted program or programs. A prompt will appear asking if you want to delete selected program asking for Y/N.

Settings

Scrolling through Settings with Jog Handle - The jog handle can now be used to scroll through the settings. In previous versions, the jog handle could only be used to scroll through (cursor-highlight) the parameters, but not the settings. This has been corrected. (Any Mill Control Ver. 10.15 and above; any Lathe Control Ver. 3.05 and above.)

There are many settings which give the user various options over the control of their machine tool. Read the Settings section of the operator's manual for all the possible options. Here are some of the more useful settings.

Setting 1 ***AUTO POWER*** *OFF* – This turns the machine off after it is idle for the number of minutes defined in this setting.

Setting 2 ***POWER OFF AT*** *M30* – This option will power off the machine tool when an M30 command 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.

Setting 7 ***PARAMETER LOCK –*** When On parameter changes are locked out. When off parameter changes may be made. When control powered up switch turns to ON.

Setting 8 ***PROG MEMORY LOCK –*** When this is **Off**, control program memory can be modified. When this setting is turned **On**, memory edits cannot be done and programs cannot be erased.

Setting 9 ***DIMENSIONING*** – This changes the machine control from inch to metric, which will change all offset values and position displays accordingly. This setting *will not* change your program to either inch or metric.

Setting 23 ***9XXX PROGS EDIT LOCK*** – This is an On/Off setting. When it is On, the 9000 series programs (usually the Quick Code source file or macro programs) are invisible to the operator and cannot be uploaded or downloaded. They also cannot be listed, edited, or deleted.

Setting 31 ***RESET PROGRAM POINTER –*** When this is On, the **RESET** key will send the cursor (program pointer) back to the beginning of the program. Normally set to on.

Setting 32 ***COOLANT OVERRIDE –*** This setting controls how the coolant pump operates. The settings are: Normal, Ignore and Off. When it is set on Normal, coolant commands respond as programmed. If set on Ignore, an M08 or M88 command in the program will not turn the coolant on (i.e., the command will be ignored), but it can be turned on manually using the **COOLNT** key. If this setting is Off, the coolant cannot be turned on at all, and the control will give an alarm when it reads an M08 or M88 command in a program.

Setting 33 **_COORDINATE SYSTEM_** – This setting changes the way the G92/G52 offset system works. It can be set to Fanuc, or Yasnac. Normally it is set to Fanuc.

Setting 36 **_PROGRAM RESTART –_** When it is OFF, starting a program from anywhere other than the beginning of a program or a tool sequence may produce inconsistent results or crashes. When it is ON, you are able to start a program from the middle of a tool sequence. You cursor onto the line you want to start on and press CYCLE START. It will cause the entire program to be scanned to ensure that the correct tools, offsets, G codes, and axes positions are set correctly before starting and continuing at the block where the cursor is positioned. Some alarm conditions are not detected prior to motion starting. You could leave this setting ON all the time if you want, but it might do some things unnecessarily, so you would probably prefer to turn it OFF when you're done using it.

Setting 42 **_M00 AFTER TOOL CHANGE_** – When off tool changes are normal. When **ON** a program stop M)) will occur after a tool index**.** **M00 FOUND** will be displayed at the bottom left.

Setting 51 **_DOOR HOLD OVERRIDE –_** This setting is **no longer available** to use in new machines. On older machine when it is off, a program cannot be started if the doors are open, and opening the doors will cause a running program to stop – just like a feed hold. When this setting is On, the door condition is ignored. This setting will always be Off when the control is powered up.

Setting 76 **_FOOT PEDAL LOCK OUT –_** When set to Off the foot pedal operates normally. When ON the foot pedal is ignored by the control.

Setting 84 **_TOOL OVERLOAD ACTION –_** This is used to determine tool overload conditions as defined by the Tool Load monitor page in the CURNT COMDS display (use **PAGE DOWN** to get there). A tool overload condition can result in one of four actions by the control, depending on Setting 84. ALARM will generate an alarm when overload occurs; FEED HOLD will stop with a Feed Hold when overload occurs; BEEP will sound an audible alarm when overload occurs; or AUTOFEED will automatically decrease the feed rate.

Setting 85 **_MAX CORNER ROUNDING –_** This setting is used to set the corner rounding accuracy required by the user. The accuracy defined in Setting 85 will be maintained even at maximum feed rate. The control will only slow at corners *when it is needed*. If it is set at 0 the machine will operate in the exact stop mode, slowing speed of machine.

Setting 88 **_RESET RESETS OVERRIDE –_** When this is On, the RESET key sets all overrides back to 100%.

Setting 92 **_CHUCK CLAMPING –_** Specifies **OD** (outside diameter) **or ID** (inside diameter clamping.

Setting 93 ***TAIL STOCK X CLEARANCE–*** Works in conjunction with setting 9o. Defines X axis travel limit for turret so it won't crash into tail stock. When jogging no alarm is generated.

Setting 94 ***TAIL STOCK Z CLEARANCE–*** Works in conjunction with setting 94. Defines Z axis travel limit for turret so it won't crash into tail stock. When jogging no alarm is generated.

Setting 101 ***FEED OVERRIDE > RAPID –*** When this setting is OFF, the machine will behave normally. When it is ON and HANDLE CONTROL FEED RATE is active, 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. (Any Mill Control Ver. 10.22 and above; any Lathe Control Ver. 4.11 and above.)

Setting 103 ***CYC START/FH HUCK CLAMPING –*** When **ON** cycle start must be presses and held in to run a program. When **CYCLE START** key is released machine goes into a feed hold. When OFF the machine operates normal.

Setting 104 ***JOG HANDLE TO SNGLBLK*** When this is **ON** and **SHGLBLK** is selected , the jog handle can be used to single step through a program.

Setting 103 ***CYC START / FH SAME KEY –*** When this setting is ON, the CYCLE START button functions as the Feed Hold key as well. When CYCLE START is pressed and held in, the machine will run through the program; when it's released, the machine will stop in a Feed Hold. This gives you much better control when testing a new program. When you are done using this feature, turn it Off. This setting can be changed while running a program. It cannot be ON when Setting 104 is ON. When one of them is turned ON, the other will automatically turn OFF. *(Any Mill Control Ver. 9.06 and above; any Lathe Control Ver. 4.11 and above.)*

Setting 104 ***JOG HANDL TO SNGL BLK –*** When running a program in MEM mode in the Program or Graphics display, you can use the SINGLE BLOCK key to cycle through your program one line at a time with each press of the CYCLE START button, when the machine is running or you are in Graphics. If you turn Setting 104 ON, and SINGLE BLOCK has been selected. You first press the CYCLE START button, then each counterclockwise click of the jog handle will step you through a program line by line. Turning the handle clockwise will cause a FEED HOLD. This setting can be changed while running a program. It cannot be ON when Setting 103 is ON. When one of them is turned ON, the other will automatically turn OFF. *(Any Mill Control Ver. 9.06 and above; any Lathe Control Ver. 4.11 and above.)*

Setting 114 ***CONVEYOR CYCLE (MIN) –*** If this is set to zero, the conveyor will operate normally. If another number is entered, it defines how long (in minutes) each cycle will be when the chip conveyor is turned on. The chip conveyor cycle is started with either an M code (M31 or M32) or with the control **CHIP FWD/REV** keys. It will stay on for the time defined in Setting 115, then turn off and not restart until the cycle time in Setting 114 has elapsed.

Setting 115 ***CONVEYOR ON TIME (MIN) –*** This setting works with Setting 114, which defines the conveyor cycle time. Setting 115 defines how long the chip conveyor will stay on during each cycle.

Setting 163 ***Disable 0.1 Jog Rate –*** This disables the .10 jog rate key. With this disabled, the fastest job rate will be .01 in/min. With a jog rate of .10 in/min it is easy to go past the intended location and possibly result in a crash.

Setting 201 ***SHOW ONLY WORK and Tool Offsets in Use –*** With this feature turned on only the Work and Tool Offsets used within a program will be shown on the respective pages. To activate the program first must be run in graphics or memory regular mode.

Hydraulic Tailstock Operation

The optional Haas Hydraulic Tailstock is a hydraulically actuated cast iron member which runs along two linear guides. The 20 inches of travel allows a long part to be machined. Tailstock motion is controlled in one of three ways:

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

The tailstock is designed to travel to position at two rates:

1. High pressure is called "rapid" and can be programmed with G0.
2. Low pressure is called "feed" and can be programmed with G1. 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. **Recommended minimum tailstock operating pressure is 120 psi. Pressure below 120 psi may result in unreliable operation.**
2. **If Settings 93 and 94 are not used. There is NO RESTRICTED ZONE for the tailstock. The turret and tailstock can be crashed together if improperly operated.**
3. **It is important to verify tailstock and turret clearance before operating machine or serious damage could occur. Settings 93 and 94 are set at the factory but with tools sticking out these must be reset to prevent collisions.**
4. **FEED HOLD will *NOT* stop the hydraulic tailstock.**

Jogging

- In JOG mode, the keys "TS ←" AND "TS →" are used to jog the tailstock at low pressure (feed).
- The jog handle cannot be used to jog the tailstock.
- By pressing TS ← and 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.

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 FWS = foot pedal input

Hydraulic Pressure

The following information can also be found on the side panel of the machine:

RECOMMENDED HYDRAULIC TAILSTOCK OPERATING PRESSURE IS 120 PSI.
OPERATING PRESSURES BELOW 120 PSI MAY CAUSE THE TAILSTOCK TO FUNCTION UNRELIABLY.

Settings

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 it's 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.

M Codes

M21 Tailstock Advance (Standard M code)
Settings 105, 106, and 107 used to advance to the HOLD POINT.

M22 Tailstock Retract (Standard M code)
Setting 107 used to withdraw to the RETRACT POINT.

Foot Pedal Operation

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 the 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 stopped and the sequence must begin anew.

Machine Defaults

When the machine is first powered on, the control doesn't remember the program we were running prior to shutting the machine off. The machine has to assume some sort of "default" mode. These codes are the ones the machine "defaults to" when we power it on.

G00	Rapid traverse
G18	X-Z Circular plane selection
G40	Cutter Compensation cancel
G54	Work offset 54
G64	Exact stop cancel
G80	Canned cycle cancel
G97	Constant surface speed cancel
G99	Feed per revolution

Transfer of Programs: USB Device

Transfer program from USB card to Memory Directory

1. Plug USB card into CNC Control Panel
2. Press LIST PROG button
3. Curser to USB DEVICE (which should become highlighted in red)
4. Press **WRITE** key (Directory for USB device should appear.
5. Cursor to program to be used (should become highlighted)
6. Press **WRITE** key to select (check mark will appear)
7. Press **F2**
8. A drop down window will appear asking you where to transfer program to. Use up and down cursor to highlight MEMORY
9. Press **WRITE**
10. Program is now in memory directory

Note: On using Tabbed menus. **WRITE** will open up a tab
CANCEL will close directory in a tab and get you to the tab selection mode.
Note**: ORIGON** button should be pressed before removing USB device

Send program from Memory Directory to USB card

1. Press LIST PROG button
2. Curser to MEMORY TAB (which should become highlighted in red)
3. Press **WRITE** (Directory for MEMORY should appear).
4. Cursor to program to be transferred (should become highlighted)
5. Press **WRITE** key
6. Press **F2**
7. A drop down window will appear asking you where to transfer program to. Use up and down cursor to highlight USB device
8. Press **WRITE**
9. Program is now in the USB directory.

HAAS LATHE CONTROL TIPS

GENERAL TIPS

- **Cursor Searching for a Program** - When in EDIT or MEM mode, you can select and display another program quickly by entering the program number (Onnnnn) you want and pressing either the up or down cursor arrow or **F4**.

- **Searching for a Program Command** - Searching for a specific command in a program can be done in either MEM or EDIT mode. Enter the address letter code (A, B, C, etc.) or address letter code with the value (A1.23), and press the up or down cursor arrow. If you enter just the address code and no value, the search will stop at the next use of that letter, regardless of the value.

- **Spindle Command** - You can stop or start the spindle with **CW** or **CCW** any time you are at a Single Block stop or a Feed Hold. When you restart the program with **CYCLE START**, the spindle will be turned back on to the previously defined speed.

- **Coolant Pump** - The coolant pump can be turned on or off manually while a program is running, by pressing the **COOLNT** button. This will override what the program is doing until another M08 or M09 coolant command is executed. This also applies to the chip conveyor.

CONTROL TIPS

- **Optional Stop** - Takes effect on the line after the highlighted line when pressed.

- **A Block Delete** - Takes effect four lines after that key is pressed when cutter compensation is in use, or two lines later when cutter compensation is not in use.

- **Block Look-Ahead** - This control actually does look ahead for block interpretation, up to 20 blocks. This is not needed for high-speed operation. It is instead used to ensure 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** - This is a customer machine option that prevents the operator from editing or deleting programs, and from altering settings when in the locked position. Since the Key switch locks out the Settings, it also allows you to lock out other areas within the settings: Setting 7 locks parameters: Parameter 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.

- **Chip Conveyor** - The chip conveyor can be turned on or off when a program is running, either manually using the control keys or in the program using M codes. The M code equivalent to **CHIP FWD** is M31, **CHIP REV** is M32, and **CHIP STOP** is M33. You can set the Conveyor Cycle time (in minutes) with Setting 114, and the Conveyor On-Time (in minutes) with Setting 115.

- **Transferring an MDI Program** - You can transfer and save a program in MDI to your list of programs. When in the MDI display, *make sure that the cursor is at the beginning* of the MDI program. Enter a program number (Onnnnn) that's not being used. Then press **ALTER** and this will transfer the MDI data into your list of programs under that program number.

- **To Rapid an Axis Home** - You can rapid *all* axes to machine zero by pressing the **HOME G28** key. You can also send just one axis (X, or Z,) to machine zero in rapid motion. Enter the letter X or Z, then press **HOME G28** and that axis alone will rapid home. ***CAUTION***! There is no warning to alert you of any possible collision! For example, if the Z axis is down near the tail stock and then sent home using HOME G28, a crash may result. Care must be exercised. **Tailstock may be inadvertently removed from machine!**
(Any Mill Control ver. 9.49 and above; any Lathe Control ver. 2.24 and above.)

POSIT

- **Quick Zero on DIST-TO-GO Display** - To clear out and get a quick zero position display, for a distance reference move, use the DIST-TO-GO position display. When you are in the POSIT display and in HANDLE JOG mode, press any other operation mode (EDIT, MEM, MDI, etc.) and then go back to HANDLE JOG. This will zero out all axes on the DIST-TO-GO display and begin showing the distance moved.

- **To Origin the POS-OPER Display** - This display is used for reference only. Each axis can be zeroed out independently, to then show its position relative to where you selected to zero that axis. To zero out a specific axis, **PAGE UP** or P**AGE DOWN** in the POSIT display to the POS-OPER large-digit display page. When you Handle Jog the X, Y or Z axis and then press **ORIGIN**, the axis that is selected will be zeroed. Or, you can press an **X**, **Y** or **Z** letter key and then **ORIGIN** to zero that axis display. You can also press the **X**, **Y** or **Z** key and enter a number (X2.125), then press **ORIGIN** to enter the number in that axis display.

- **Jog Keys**. The **JOG** keys (+X, -X, +Y, -Y, +Z, -Z, +A, -A, +B, -B) use the jog speeds of 100., 10., 1. and .1 inches per minute listed next to the **HANDLE JOG** key (jogging with the handwheel uses the .1, .01, .001 and .0001 inch increments). You can also adjust feed rate using the **FEED RATE OVERRIDE** buttons, which allow you to increase or decrease feed rate in 10% increments, up to 200% or using the **HANDLE CONTROL FEED RATE** or **HANDLE CONTROL SPINDLE** keys to adjust the programmed feed or speed 1% up or down with every increment of the Handle.

- **Jog Keys** - You can also select an axis for jogging by entering the axis letter on the input line and then pressing the **HANDLE JOG** button. This works for the X, Y, Z, and A axes as well as the B, C, U, and V auxiliary axes.

ALARM

- **Alarm History Display** - There is an alarm history that displays the previous 100 alarms. Pressing the right or left cursor arrow (†t†††u†) while in the Alarm display will list the last 100 alarms, with their date and time. You will need to use the cursor up arrow (†p†) to see the alarms previous to the last one. Pressing either the left or right arrow again will bring you back to the normal Alarm display.

- **Alarm History saved to RS-232 and Disk** - This is a new feature. From the alarms history screen, the user can now save the alarm history (the last 100 alarms) to a floppy disk file by entering a file name and pressing F2. Alternately, the alarm history can be sent to a PC using RS-232 by pressing SEND RS232. The output from either method will contain a percent sign (%) on the first and last lines. (Any Mill control ver. 10.22 and above; any Lathe control ver. 4.02 and above.)

MESGS

- **Leaving Messages** - You can enter a message in the MESGS display for the next person, or for yourself. It will be the first display shown when you power up the machine, *if there are no alarms* other than the usual 102 SERVOS OFF alarm. If the machine was powered down using **EMERGENCY STOP**, the MESGS display will not show up when you turn the machine on again. Instead, the control will display the active alarm generated by the emergency stop. In this case, you would have to press the **ALARM/MESGS** key to view a message.

PROGRAMMING

- **Program Format at the Beginning and End** - Programs written on a PC and sent to the control from a floppy disk or through the RS-232 port must start and end with a % sign, on a line by itself. The second line in a program received via floppy or RS-232 (which will be the first line the operator sees) must be **O**nnnnn, a six-character program number that starts with the letter **O** followed by five digits. When you create a program on the Haas control the percent (%) signs will be entered automatically, though you won't see them displayed.

- **M19 (Orient Spindle) with a P Value** - This feature works on any vector drive mill. Previously, the M19 command would simply orient the spindle to only one position ó that suitable for a tool change. Now, a P value can be added that will cause the spindle to be oriented to a particular position (in degrees). If a whole number is used for the P value, no decimal point is needed. For example, M19 P270 will orient the spindle to 270 degrees. Note that P270.001 (or any other fraction) will be truncated to P270, and P365 will be treated as P5. (Any Mill Control ver. 9.49 and above. Any Lathe Control ver. 2.21 and above.)

- **M19 (Orient Spindle) with a Fractional R Value** - This feature works on any vector drive mill. An M19 R123.4567 command will position the spindle to the angle specified by the R fractional value; up to 4 decimal places will be recognized. This R command now *needs a decimal point*: if you program M19 R60, the spindle will orient to 0.060 of a degree. Previously, R commands were not used for this purpose and only integer P values could be used. (Any Mill Control ver. 9.49 and above; any Lathe Control ver. 2.29 and above.)

- **Duplicating a Program in LIST PROG** - In the LIST PROG mode, you can duplicate an existing program by cursor-selecting the program number you wish to duplicate,typing in a new program number (**O**nnnnn), and then pressing **F2**. You can also go to the Advanced Editor menu (**F1**) to duplicate a program, using the PROGRAM menu and the DUPLICATE ACTIVE PROGRAM item.

COMMUNICATIONS

- **Program Format to Receive** - You can receive program files from a floppy disk or the RS-232 port into the Haas control. Each program must begin and end with a % sign on a line with nothing else on that line. There also must be an **O**nnnnn program number on the line after the % sign in each program. If there is a (Program Name), it should be entered between parentheses, either after the program number on the same line or on the next line. The program name will appear in the program list.

- **Receiving Program Files from a Floppy Disk** - You can load program files from a floppy disc using the I/O menu and the FLOPPY DIRECTORY item of the Advanced Editor. Pressing **WRITE/ENTER** when this menu item is selected will display a list of the programs on the floppy disk. Use the cursor arrow keys or the handwheel to select the file you need to load, and press **WRITE/ENTER**. After loading that file, the floppy directory will remain on display to allow more files to be selected and loaded into the control. **RESET** or **UNDO** will exit this display.

- **Sending Multiple Programs from LIST PROG Using SEND RS232** - Several programs can be sent to the serial port by typing all the program names together on the input line without spaces (e.g., O12345O98765O45678) and pressing SEND RS232.

- D **I/O Menu SEND RS232 or SEND FLOPPY Commands** - You can send programs to the RS232 port or a floppy disk from the Advanced Editor. After selecting the menu item you want (SEND RS232 or SEND FLOPPY), a program list will appear. Select the program you want to save, or “ALL” (at the end of the list) if you wish to send all programs under one file name. You can also select any number of programs using the up and down cursor arrow keys or the handwheel and the **INSERT** key to mark the specific programs to send. If no programs are selected from the list using the **INSERT** key, the currently highlighted program will be sent.

- **Sending Multiple Programs Under One File Name** - In the Advanced Editor, you can send multiple program files via the RS232 port or a floppy disk, using the SEND RS232 or SEND FLOPPY commands under the I/O menu (see the previous paragraph for how to do this). The Advanced Editor allows you to choose several programs (select them using the cursor and the **INSERT** key) and save them under one file name that you type in; then press **WRITE/ENTER** to save and send it. (Any Mill Control ver. 9.49 and above; any Lathe Control ver. 3.00 and above.)

- **Sending Multiple Programs Using Program Numbers** - The SEND FLOPPY item from the I/O menu of the Advanced Editor allows the operator to select one or more programs to be saved to floppy disk. It will prompt you to "ENTER FLOPPY FILENAME". In previous versions, the control would insist on a file name. Now, however, if you do not enter a file name, but simply press **WRITE/ENTER**, the control will save each program (the ones you selected using the cursor and the **INSERT** key) to a separate file on the floppy and use the five-digit program number as the file name. For example, if programs O00123 and O45678 are selected, the new file names created will be O00123 and O45678. (Any Lathe Control ver. 3.00 and above.)

- **Sending a Program File from LIST PROG Display** - You can send a file or files to a floppy disk or through the RS-232 port from the LIST PROG display. Use the cursor arrows and the **INSERT** key to select the program(s) you want, or “ALL” if you want to send all of them under one file name. When you press **F2** to send the selected program(s), the control will ask for a floppy file name, which can be up to eight characters long with a three-letter extension (8CHRCTRS.3XT). Then press **F2** again to send it. You can also use the I/O menu in the Advanced Editor to send and receive program files.

- **RS-232 Communications Using X-Modem** - If you are seeing occasional errors when using RS-232 communications, X-Modem (Setting 14) is a standard communications mode which is very reliable when only a few errors occur. Our control supports this, as do almost all software communication packages for PCs.

- **Send and Receive Offsets, Settings, Parameters and Macro Variables to/from Disk** - You can save offsets, settings, and parameters to a floppy disk. *Press* **LIST PROG** *first*, then select an OFSET, SETNG or PARAM display page. Type in a file name and then press **F2** to write that display information to disk (or F3 to read that file from a disk). You can also do this with the macro variables by pressing **LIST PROG** *first*, then selecting the macro variable display page (**PAGE DOWN** from CURNT COMDS).

- **Send and Receive Offsets, Settings, Parameters and Macro Variables to/from RS232** – You can also save offsets, settings, and parameters via the RS-232 port. *Press* **LIST PROG** *first*, and then select an OFSET, SETNG, or PARAM display page. Press **SEND RS232** to send that display page to the RS-232 port under the file name that you enter. Press **RECV RS232** to read the file via RS-232. You can also do this with the macro variables by pressing **LIST PROG** *first*, then selecting the macro variable display page (**PAGE DOWN** from CURNT COMDS).

- **Deleting a Program File from a Floppy Disk** - A file can be erased from the floppy drive. On the LIST PROG display, type "DEL file name" where "file name" is the name of the file on the floppy disk. Do not use the program number, unless its also the file name. Press **WRITE/ENTER**. The message "FLOPPY DELETE" will appear, and the file will be deleted from the floppy disk. Note that this feature requires the latest floppy driver EPROM chip version 2.11. (Any Mill Control ver. 10.02 and above; any Lathe Control ver. 3.00 and above.)

G Code

At Productivity Inc, we have two classes that pertain to Haas Turning Centers. The one we are in right now is the Operator Class, and we also have a Haas Lathe Programming class.

We have two classes to fill the different needs of our customers as not all people that require training require programming training.

We do include the entire Programming manual in order to give everyone the opportunity to study G&M code lathe programming, but we also go over SOME programming basics so that the operator can understand the programmer's intensions.

The definition of a part program for any CNC consists of movements of the tool to coordinates and speed changes to the spindle RPM. It also contains auxiliary command functions such as tool changes, coolant on or off commands, or external M codes commands.

Tool movements consist of rapid positioning commands, straight line movement of the tool at a controlled speed, and movement along an arc.

This machine has two (2) linear axes named X and Z. The X-axis moves the tool turret toward and away from the spindle center line, while the Z axis moves the tool turret along the spindle axis. The machine zero position is where the tool is at the right corner of the work cell farthest away from the spindle axis. Motion in the X-axis will move the table toward the spindle centerline for negative numbers and away from spindle center for positive numbers. Motion in the z-axis will move the tool toward the spindle chuck for negative numbers and away from the chuck for positive numbers.

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 fractional decimal positions. If no decimal point is entered, the last digit is assumed to be 1/1000 degrees. The smallest magnitude is 0.001 degrees, 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 fractional 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 inches or 1/1000 millimeters.

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 fractional decimal positions. If no decimal point is entered, the last digit is assumed to be 1/1000 degrees. The smallest magnitude is 0.001 degrees, the most negative value is -8380.00, and the largest value is 8380.00 degrees.

D NOT USED, OPTIONAL MACRO PARAMETER

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. 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 the block containing the G code. The G is followed by a two or three digit number between 0 and 255. 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. More than one G code can be placed in a block in order to specify all of the setup conditions for an operation. See the following section (Preparatory Functions (G Codes)) for a detailed list of G codes. The G codes are specified in an A/B/C format to show the three different G coding systems. Each system uses a different G code for the same function. In this manual, programming is described based on the A system, however, Setting 34 (G CODE SYSTEM) can be used to select another system.

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. It is either in inches with four fractional positions or mm with three fractional positions. It is followed by a signed number in inches between -8380.00 and 8380.00 for inches or between -8380.00 and 8380.00 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.

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. Not 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, M98 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 a Onnnn identification in the first block; it cannot be deleted. Altering the O in the first block causes the program to be renamed. An Onnnn can be placed in other blocks of a program but will have no effect and can be confusing to the reader. A color (:) 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 either a time in seconds or a program number for a subroutine call. If it is used as a time (for a G04 dwell) or a program name (for a M97), the value may be either a positive number without decimal point up to 9999. If it is used as a time, it may be a positive decimal with fraction between 0.001 and 1000.00.

Q CANNED CYCLE OPTIONAL DATA

The Q address character is used in canned cycles and is always a positive number in inches/mm between 0 and 100.0.

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. It is followed by a signed number in inches between -8380.00 and 8380.00 for inches or between -8380.00 and 8380.00 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. The S is followed by an unsigned number between 1 - 99999. The S command does not turn the spindle on or off; it only sets the desired speed. If a gear change is required in order to set the commanded speed, this command will cause a gear change to occur even if the spindle is stopped. If the spindle is running, a gear change operation will occur and the spindle will continue running at the new speed.

T TOOL SELECTION CODE

When the T address is in a G50 block the number format following T is Txx00. Where xx is a positive number between 51 and 100. Otherwise the T address selects a tool and offset while initiating the tool change process. When not in a G50 block the number following T must be in the following format Txxyy. Where xx selects the tool and is a positive number between 1 and the total turret tools (i.e. Parameter 65); and yy selects the tool offset and is a positive number between 1 and 50.

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. It is followed by a signed number in inches between -838.0000 and 838.0000 for inches or between -8380.000 and 8380.000 for metric. If no decimal point is entered, the last digit is assumed to be 1/100000 inches or 1/1000 mm.

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.

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. It is followed by a signed number in inches between -8380.00 and 8380.00 for inches or -8380.00 and 8380.00 for metric. If no decimal point is entered, the last digit is assumed to be 1/10000 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 or mm with three fractional positions. It is followed by a signed number in inches between -8380.00 and 8380.00 for inches or between -8380.00 and 8380.00 for metric. If no decimal point is entered, the last digit is assumed to be 1/10000 inches or 1/100

Rules of Grouping Codes

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

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

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

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

1) Codes come in groups. Each group of codes will have a specific group number.

2) A “G” code from the same group can be replaced by another code in the same group. By doing this the programmer establishes modes of operation. The universal rule here is that codes from the same group cannot be used more than once on the same line.

3) There are modal G codes which, once established, remain effective until replaced with another code from the same group.

4) There are non-modal G codes which, once called, are effective only in the calling block, and are immediately forgotten by the control.

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

G CODES

Code:	Group:	Function:
G00	01	Rapid Motion
G01	01	Linear Interpolation Motion
G02	01	CW Interpolation Motion
G03	01	CCW Interpolation Motion
G04	00	Dwell
G09	00	Exact Stop
G10	00	Programmable Offset Setting
G17	02	XY Plane Selection (not available)
G18	02	ZX Plane Selection
G19	02	YZ Plane Selection (not available)
G20	06	Inch Programming Selection
G21	06	Metric Programming Selection
G28	00	Return To Reference Point
G29	00	Set Return Reference Point
G31	00	Skip Function
G40	07	Tool Nose Compensation Cancel
G41	07	Tool Nose Compensation Left
G42	07	Tool Nose Compensation Right
G50	11	Spindle Speed Clamp
G51	11	Cancel Offset (Yasnac)
G52	00	Set Local Coordinate System (Fanuc)
G53	00	Non-Modal Machine Coord. Selection
G54	12	Select Work Coordinate System 1
G55	12	Select Work Coordinate System 2
G56	12	Select Work Coordinate System 3
G57	12	Select Work Coordinate System 4
G58	12	Select Work Coordinate System 5
G59	12	Select Work Coordinate System 6
G61	13	Exact Stop Modal
G64	13	G61 Cancel
G65	00	Macro Subroutine Call
G70	00	Finishing Cycle
G71	00	O.D./I.D. Stock Removal Cycle
G72	00	End Face Stock Removal Cycle
G73	00	Irregular Path Stock Removal Cycle
G74	00	end Face Grooving Cycle, Peck Drilling
G75	00	O.D./I.D. Grooving cycle, Peck Drilling
G76	00	Thread Cutting Cycle, Multiple Pass
G80	09	Canned Cycle Cancel
G81	09	Drill Canned Cycle
G82	09	Spot Drill Canned Cycle

G-Codes (continued)

Code:	Group:	Function:
G83	09	Peck Drill Canned Cycle
G84	09	Tapping Canned Cycle
G85	09	Boring Canned Cycle
G86	09	Bore/Stop Canned Cycle
G87	09	Bore/Manual Retract Canned Cycle
G88	09	Bore/Dwell Canned Cycle
G89	09	Bore Canned Cycle
G90	01	O.D.I.D. Turning cycle, Modal
G92	01	Thread Cutting Cycle, Modal
G94	01	End Face Cutting cycle, Modal
G96	12	Constant Surface Speed On
G97	12	Constant Surface Speed Cancel
G98	05	Feed per Minute
G99	05	Feed per Revolution
G102	00	Programmable Output to RS-232
G103	00	Block Lookahead Limit
G110-G129	12	Extra Work Coordinate System 7 through 26
G154P1 to G154P99	12	Extra Work Coordinates on new machines
G184	09	Reverse Tap Canned Cycle
G187	00	Accuracy Control for High Speed Machining

M CODES

M00 Program stop. Press CYCLE START button to continue.

M01 Optional program stop. When optional stop key depressed program will stop on M01 code.

M02 End of program.

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

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

M05 Spindle stop.

M08 Coolant **ON** command.

M09 Coolant **OFF** command.

M10 Clamp spindle chuck.

M11 Unclamp spindle chuck.

M30 Program end and rewind to beginning of program.

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

Advanced Lathe Programming

Program format, or program style, is an important part of CNC machining. Each individual will format their programs differently. The point is that a programmer needs to be consistent and efficient. For example:

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

The first line or block in a program should be a return to machine zero (using G28 or G51 codes). Any tool change should be after 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 selections, tool geometry or tool shifts.

The third line may contain a spindle speed maximum for the tool being used.

The fourth line or block should cancel any constant surface speed mode (G97) and specify a spindle speed command (S____) along with a spindle ON clockwise command (M03).

The fifth line should contain a preparatory code (G00) for X and Z positioning command, specify the work coordinate being used, and to turn on the coolant (M08).

The sixth line may optionally specify a Constant Surface Speed (G96) and a surface feet per minute (SFM) defined with an (S____) command.

An example of the first five lines of a program might look like this:

```
N1 G28;
N2 T101;
N3 G50 S2000;
N4 G97 S1146 M03;
N5 G54 G00 X1.5 Z.02 M08;
N6 G96 S450;
```

Listed in the following pages are all the necessary codes for each operation. This format is a good example and defines a commonly used program style.

The format defines the "language of the machine tool." A description of the particular words a machine may accept, the order in which they must appear, the number of numeric digits associated with each word, the location of a decimal point, and the presence or absence of signs is explained in the following pages.

DEFINITIONS WITHIN THE FORMAT:

Character A single alphanumeric character value or the "+" and "-" sign.

Word A series of characters defining a single command such as "X" displacement or "F" feedrate. Unique letters are assigned as the first character of a word, and each letter has either a plus (+) or minus (-) sign value of numbers.

Block Series of words defining a single instruction. An instruction may consist of a single linear or circular motion, plus additional information such as a feedrate or stop command.

Positive Signs If a number value following the address command letters, such as I, K, R, U, W, X, Z, is positive, the plus sign need not be programmed. If the number value is negative, it must be programmed with a minus (-) sign.

Leading Zero's There is no need to program zeros proceeding a number.
EXAMPLE: G00 (G0) and M01 (M1). Trailing zeros must be programmed in M30 not M3, G70 or G7.

Modal Commands Once a particular word, such as G, X, Z, F, S, T, and M is programmed, it is not necessary to repeat the word in the following blocks until a different word or change of value is required in subsequent blocks of information.

Non-Modal Commands A non-modal command is one that is active only in the program block in which it is issued. M00 program stop is an example of a non-modal command.

Preparatory Functions "G" codes use the information contained on the line to make the machine tool do specific operations, such as:

1) Move the tool at rapid traverse.
2) Move the tool at feedrate along a straight line.
3) Move the tool along and arc at a feedrate in a clockwise direction.
4) Move the tool along an arc at a feedrate in a counterclockwise direction.
5) Move the tool through a series of repetitive operations controlled by "fixed cycles" such as, spot drilling, boring, and tapping.

Miscellaneous Functions "M" codes cause an action to occur at the end of the block. Only one M code is allowed in each block.

Sequence Numbers Sequence numbers are codes N1 through N9999 and are only used to locate a certain block or line within a CNC program. A program may be input without sequence numbers.

MACHINE DEFAULTS

A default is an automatic function of the machine tool control. After powering up the machine, the control will recognize the default G code values. The machine will go to the part zero entered for G54 if no other work coordinate code was specified in the actual program. The machine automatically recognizes the G54 column upon start-up. This is known as a default. The defaults for the Haas mill are indicated by an asterisk (*) in the "Preparatory Function G Codes" list of this workbook.

The control automatically recognizes these G codes when your HAAS lathe is powered up:

G00	Rapid Traverse
G18	X, Z Circular Plane Selection
G40	Cutter Compensation Cancel
G54	Work Coordinate Zero #1 (1 of 26 available)
G64	Exact Stop Cancel
G80	Canned Cycle Cancel
G97	Constant Surface Speed Cancel
G99	Feed Per Revolution

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

MACHINING CYCLES FOR THE LATHE

A machining cycle is used to simplify the programming of a part. Machine cycles are defined for the most common machining operations. There are two types: machining cycles for turning and grooving and canned cycles for drilling, tapping, and boring.

DRILLING, TAPPING, AND BORING CANNED CYCLES

These cycles are used to define and simplify programming for the most common Z-axis, repetitive operations such as drilling, tapping, and boring. Once selected, a canned cycle is active until canceled with a G80 code. There are six operations involved in every canned cycle:

1) Positioning of 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.
6) Rapid traverse to the initial starting point.

These cycles are modal, which remain in effect after they are defined and executed in the Z-axis for each positioning of X axes in a program. Some of the cycle command values can be changed after these cycles have been defined. The command values most often changed during a cycle are the R plane value and the Z depth value. These modal cycles will be canceled with the G80, G01 or G00 commands. The X-axis moves in these modal machine cycles are performed as rapid moves.

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 (Lnn code number) is defined within the block, the canned cycle will repeat that many times with an incremental U (X-axis move) move between each cycle.

The following is a list of the canned cycles that can be used on the HAAS lathe.

G80	Canned Cycle Cancel
G81	Drill Canned Cycle
G82	Spot Drill Canned Cycle
G83	Peck Drill Canned Cycle
G84	Tapping Canned Cycle
G85	Bore in Bore out Canned Cycle
G86	Bore in Rapid out Canned Cycle
G87	Bore with Manual Retract Canned Cycle
G88	Bore in Dwell with Manual Retract Canned Cycle
G89	Bore in Dwell Bore out Canned Cycle

MACHINE CYCLES FOR TURNING AND GROOVING

The following is a list of the canned cycles that can be used for turning and grooving for the HAAS lathe controls.

MACHINING CYCLES

G70	Finishing Cycle
G71	O.D./I.D. Stock Removal cycle
G72	End Face Stock Removal Cycle
G73	Irregular Path Stock Removal Cycle
G74	End Face Grooving or Turning with Chip Break Cycle
G75	O.D./I.D. Grooving or Turning with Chip Break cycle
G76	Thread Cutting Cycle, Multiple Pass
G90	O.D./I.D. Turning Cycle Modal
G92	Thread Cutting Cycle Modal
G94	End Face Cutting Cycle Modal

A machine cycle is used to simplify programming of a part. Machine cycles are defined for the most common, repetitive operations such as turning, facing, threading and grooving. There are both modal and non-modal machine cycles. Modal cycles, such as turning cycle G90, remain in effect after they are defined. After any subsequent X or Z-axis positioning, the canned cycle is executed again. Modal canned cycles remain in effect until canceled by a G80, G00, end of program, or RESET. Non-modal machine cycles are effective only for the block that contains them, but will perform a series of machining moves to perform that command.

Linear Movement – Creating Tool Paths

CNC programs consist of lines and arcs. This unit will discuss the proper methods to create tool paths applied to CNC programs to create the desired results in addition to creating tool paths with and without cutter compensation.

Objectives:

Upon completion of this unit, the student will:

1) Understand the major differences in G01 and G00.

2) Understand and apply the five criteria needed to produce an arc.

3) Understand the principles of programming and applications with and without cutter compensation in addition to the advantages and disadvantages of each.

4) Be capable of producing a tool path program, containing lines and arcs, with and without cutter compensation.

5) Be capable of determining feeds and speeds given an SFM and cutting tool, and be capable of determining a feed rate given an RPM and chip load.

6) Be able to integrate a tool path into a part program.

7) Understand the rules governing the use of cutter compensation.

8) Have a basic understanding of the concept of arc in/arc out and some of its applications.

Rapid Position Commands

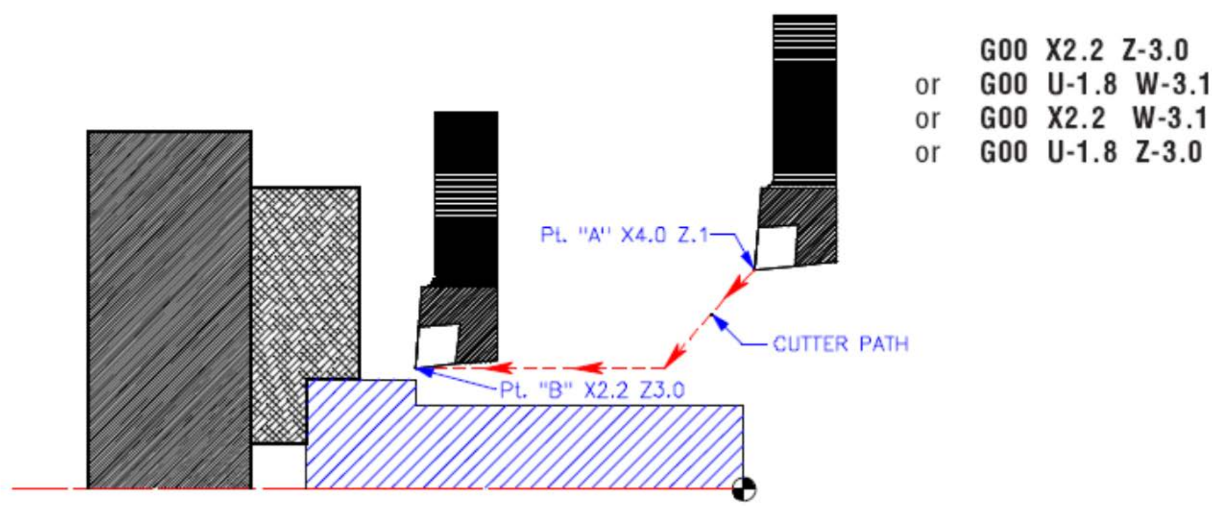
G00 RAPID MOTION POSITIONING

X	Absolute X-axis rapid motion command
Z	Absolute Z-axis rapid motion command
U	Incremental X-axis rapid motion command
W	Incremental Z-axis rapid motion command
B	Absolute tailstock rapid motion command

The G code is used to cause a rapid traverse of the two axes of the machine from one programmed point to the next programmed point. The auxiliary (tailstock) B-axis can also be moved with a G00. This G code is modal so that all following blocks will be in rapid motion until another group 01 is specified.

Generally, rapid motions done together in both X and Z-axes will not be in a straight line from one program location to the next program location. All of the axes specified are moved at the same speed (710 IPM), but will not necessarily complete their motions at the same time. Each axis slide is activated independently and the axis with the shortest distance to move will reach its endpoint first; thus, giving the impression of a 45-degree angle move. The control will wait until all motions are complete. These rapid moves may be made in ABSOLUTE or INCREMENTAL coordinate command values, which will change how those values are interpreted. The “U” letter address relates to X-axis incremental moves and the “W” letter address relates to Z-axis incremental moves.

To move from point “A” to point “B”, the programmed line can be either:



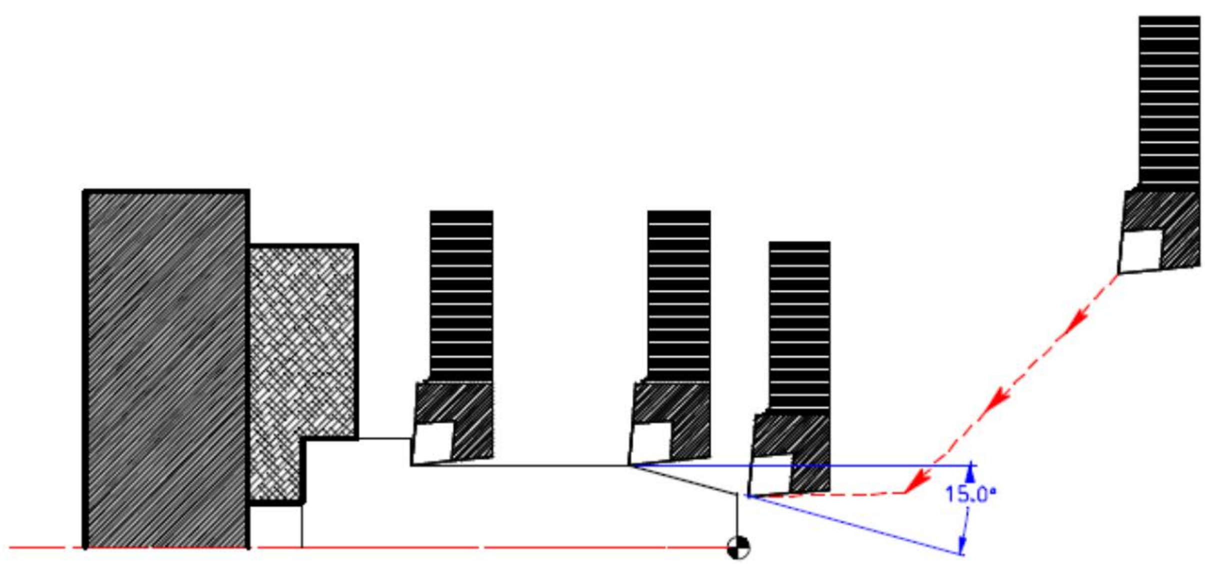
Interpolation Commands

G01 LINEAR INTERPOLATION MOTION

X	Absolute X-axis feed motion command
Z	Absolute Z-axis feed motion command
U	Incremental X-axis feed motion command
W	Incremental Z-axis feed motion command
B	Absolute tailstock feed motion command
F	Feed rate

This G code provides for straight-line (linear) motion from point-to-point. Motion can occur in one or two axes. Both axes will start and finish motion at the same time to move the tool along a straight-line path parallel to an axis or at a slope (angled) line. The speeds of all axes are controlled so that the feed rate specified is achieved along the actual path. The F (Feedrate) command is modal and may be specified in a previous block. These moves may be made in ABSOLUTE or INCREMENTAL coordinate command values, which will change how those values are interpreted. The “U” letter address relates to X-axis incremental moves and the “W” letter address relates to Z-axis incremental moves. Only those axes specified are moved in either absolute XZ or incremental UW commands.

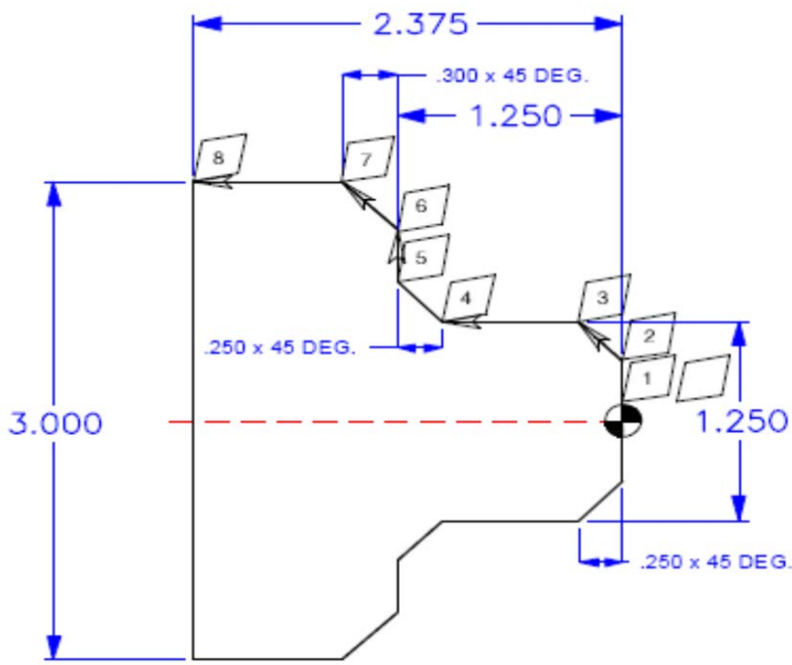
G00 X.9105 Z.10 (ABSOLUTE) G01 X1.5 Z-1.1 F.006 Z-3.0 X2.0	G00 X.9105 Z.10 (INCREMENTAL) G01 U.5895 W-2.1 F.006 W-1.9 U.5	G00 X.9105 Z310 (ABS. AND INC.) G01 X1.5 W-2.1 F.006 Z-3.0 U.5



Linear Interpolation Exercise

FORMAT: G01 (Linear interpolation [feed] Corner Chamfering for Absolute or Incremental).

With what we have learned about G01, we can move a cutter along the part profile by a series of Absolute X and Z-axis movements or Incremental U (X-axis) and W (Z-axis) movements, which can be a combination of both X and Z absolute movements and U and W incremental movements. Define the actual points around this part using both incremental and absolute.



ABSOLUTE PROGRAMMING	**INCREMENTAL AND ABSOLUTE PROGRAMMING**
G00 X.25 Z.1	G00 X.25 Z.1
G___ Z0. F.006	G___ Z0. F.006
X______	X______
X______ Z-______	U______ W-______
Z-_______	Z-_______
X______ Z-______	U_______ Z-______
X______	X_______
X______ Z-______	X_______ W-______
Z-_______	Z-_______

Circular Interpolation Commands

G02 CW Circular Interpolation Motion

X	Absolute X-axis Arc end point motion command
Z	Absolute Z-axis Arc end point motion command
U	Incremental X-axis Arc end point motion command
W	Incremental Z-axis Arc end point motion command
I	Distance from start point to arc center in the X-axis
K	Distance from start point to arc center in the Z-axis
R	Radius of circle (If I and K are not used)
F	Feed rate

G03 CCW Circular Interpolation Motion

X	Absolute X-axis Arc end point motion command
Z	Absolute Z-axis Arc end point motion command
U	Incremental X-axis Arc end point motion command
W	Incremental Z-axis Arc end point motion command
I	Distance from start point to arc center in the X-axis
K	Distance from start point to arc center in the Z-axis
R	Radius of circle (If I and K are not used)
F	Feed rate

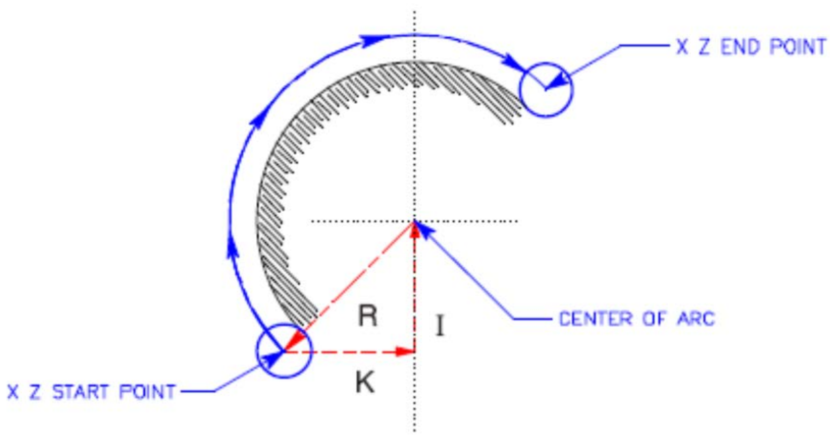
G03 will generate a counterclockwise circular motion, but is otherwise defined the same as a G02 clockwise circular motion.

Circular interpolation commands are used to move a tool along a circular arc to the commanded end position. Five pieces of information are required for executing a circular interpolation command: plane selection, arc start position coordinates, rotation direction, arc end position coordinates, and arc center coordinates or arc radius.

There are two basic command formats for defining circular interpolation depending on whether the I and K method or the R method is used to define the arc center.

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

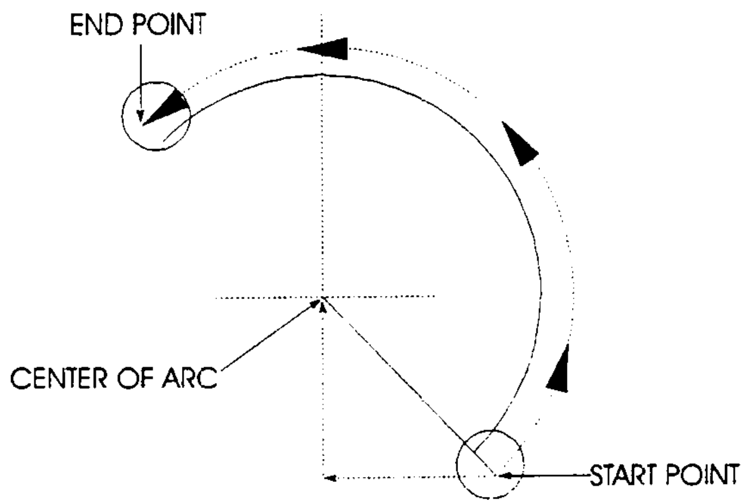
G02 CW Circular Interpolation Motion



Five pieces of information are required for executing a circular interpolation command.

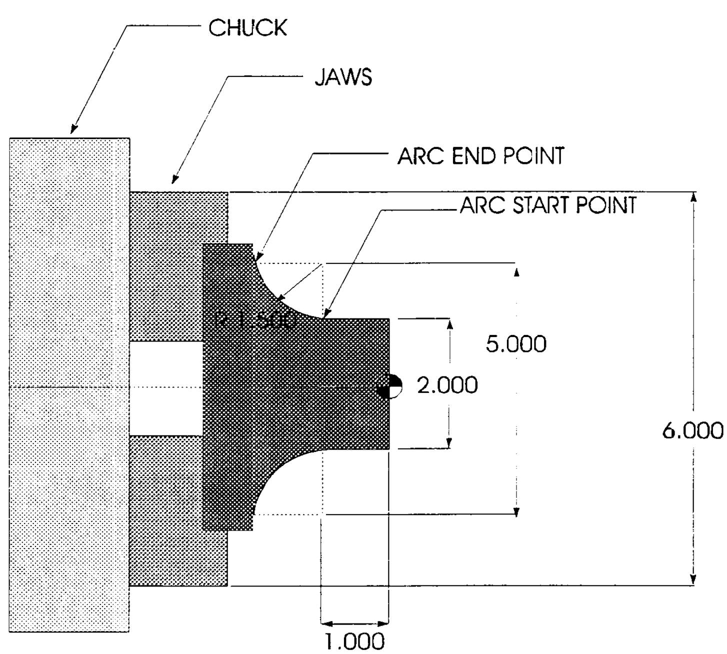
Plane selection command Plane selection command Plane selection command	G17 G18 G19	Arc parallel to XY-plane (not available) Arc parallel to ZX-plane (default) Arc parallel to YZ-plane (not available)
Arc start position coordinates	X,Z	Coordinates to the start position of arc
Rotation direction	G02 G03	Clockwise interpolation direction Counterclockwise interpolation direction
Arc end position Absolute or Arc end position Incremental	X,Z U,W	Coordinates of the end position of arc defined from the part zero or part origin Incremental distance and direction from start Point of arc to end point of arc in X and Z-axes
I and K method (arc center coordinates) R method (arc radius)	I,K R	Incremental distance and direction from start position of arc to the arc center for X and Z-axes. "I" is for the X-axis; "K" is for the Z-axis. Arc radius value of arc.

G03 CWW Circular Interpolation Motion



To move the cutter in a clockwise direction from the START POINT to the END POINT, the programming coding would be like this:

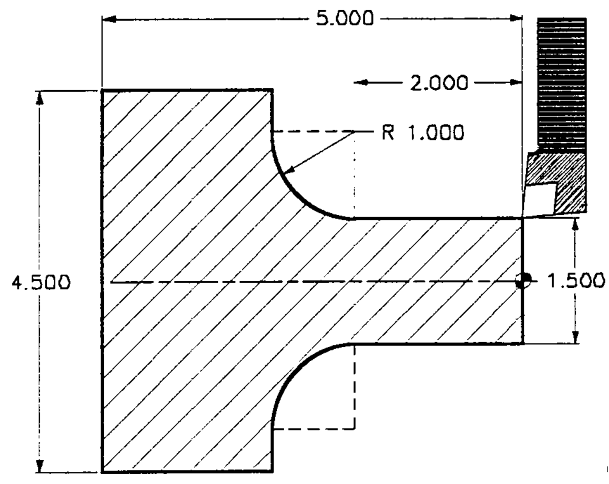
N11………………….	
N12 G00 X0. Z.1	(tool rapids to X0 Z.1 in front of part)
N13 G01 Z0. F.012	(tool feeds to Z0. face of part)
N14 X2.	(tool feeds up in X-axis to a 2. dia.)
N15 Z-1.	(tool at 2 dia. machines to the arc start point at Z-1.)
N16 G02 X5. Z-2.5 R1.5	(cuts the 1.5 radius to the arc end point at X5. Z-2.5)
N17 G01 X6.0	(tool feeds up in X-axis to a 6 dia.)
N18…………….	



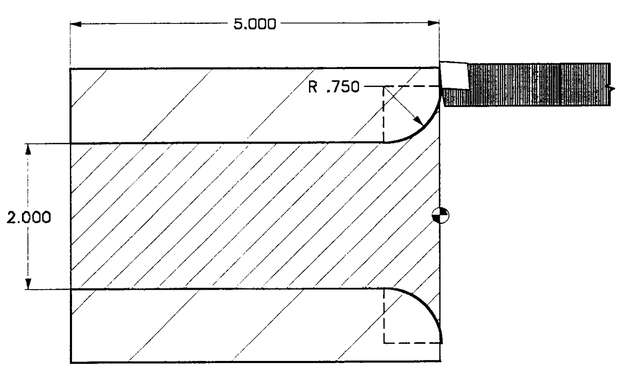
Circular Interpolation Motion Exercise

Exercise for G01 and G02

From the tools current position, program the cutter path to turn the 1.500 diameter up to the START POINT of the 1.000 radius and then interpolate the radius. Finish the cut by feeding up to the 4.500 diameter.



N9 G01 X1.5 Z0
N10 G01 Z________ F.01
N11 G____ X________ Z________ I________ K________
N12 G____ X________



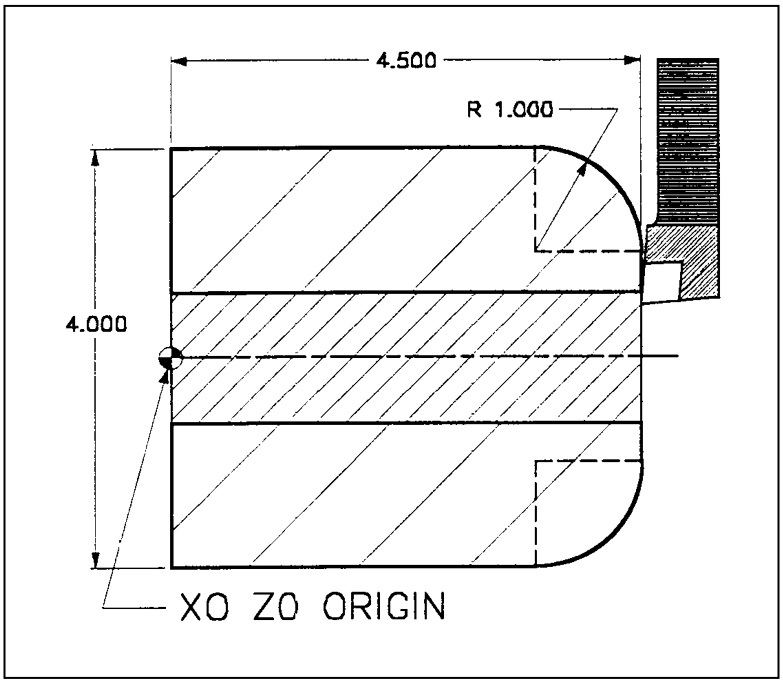
From the tool's current position, feed into face, face down to the START POINT of radius, machine the radius down into the bore, and then bore through to the end of the part.

N19 G00 X4.0 Z.10
N20 G01 Z0. F.01
N21 X________
N22 G____ X________ Z________ I________ K________
N23 G____ Z________

Circular Interpolation Exercises

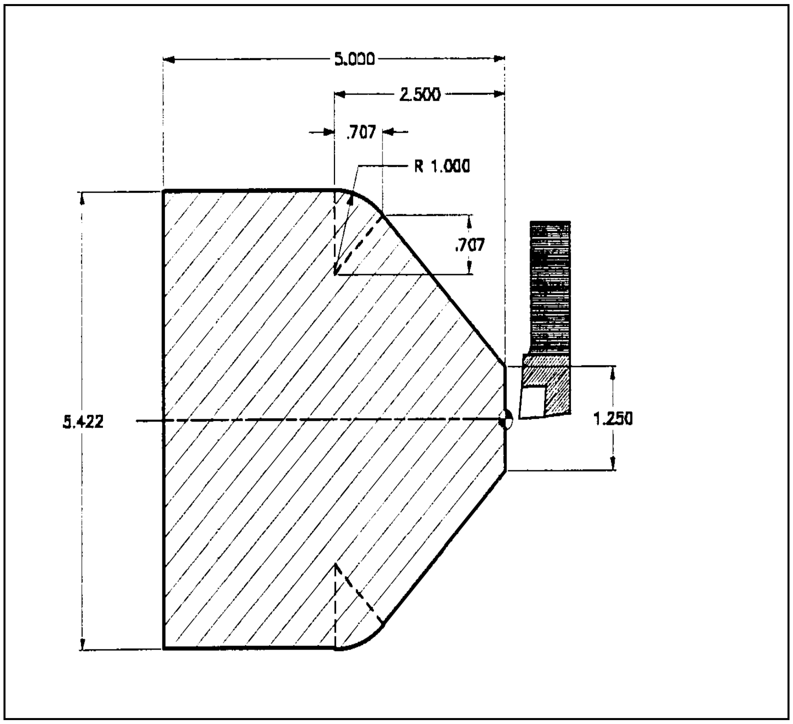
Exercise for G01 and G03:

From the tool's current position, program the cutter path to feed up the face to the START POINT of the 1.0 radius, machine the radius and then turn the 4.000 diameter to the end on the part.



N30 G___ X______
N01 X_____ Z_____
N29 G00 X1.0 Z.1
N30 G01 Z4.0 F.01
N32 X2.0
N33 G___ X_____ Z______ I_____ K______
N34 G___ Z-_____

Feed to face of part at X0, feed up to the 45 degree angle to START POINT of the radius, machine the 1.00 radius and turn the 5.422 diameter back to the end of the part back to 5.0



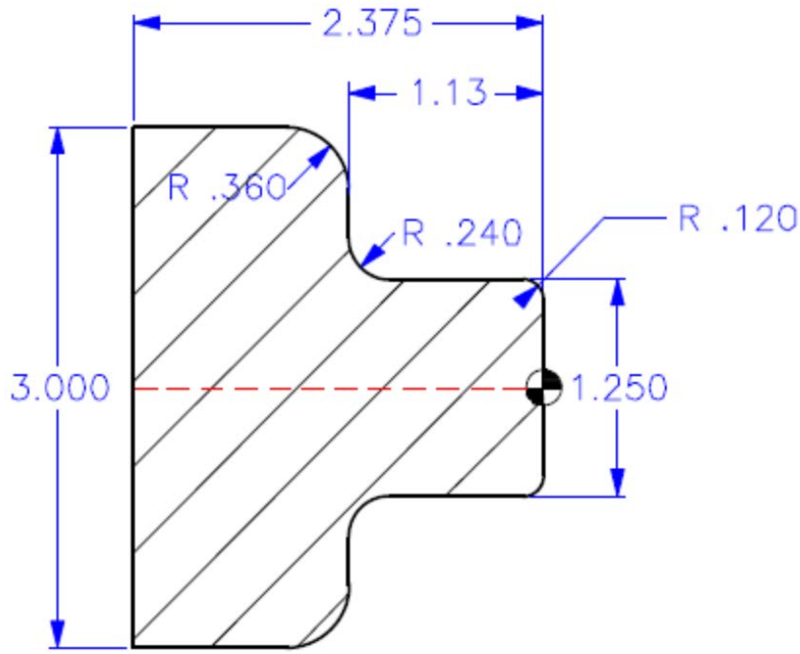
N39 G00 X0 Z.100
N40 G01 Z_____ F.01
N41 G0__ X_____Z _____I_____K____
N42 G01 X_______ Z________
N43 G___ X_____ Z_____ I_____ K_____
N44 G 01 Z______

Circular Interpolation Exercises

G02 and G03 (Circular Interpolation Clockwise/Counterclockwise)
Using “R” (radius) in place of “I” and “K”.

It is sometimes easier to program the “R” (radius) word in place of the “I” and “K” words. The “I” and “K” words are used to define the incremental distance and direction from START POINT to the ARC CENTER, which can also be done by using the “R” word in place of “I” and “K”.

NOTE: Any radius being cut using the “R” word needs a positive (+) value for a radius of 180 degrees or less. For a radius larger than 180 degrees, it needs to have a minus (-) sign with a radius value.



Program Example **(G02 & G03 using I and K)**	**Program Example** **(G02 & G03 using R)**
G01 Z0	G01 Z0
X1 .01	X1 .01
G03 X1.25 Z-.12 I_____ K_____	G03 X1.25 Z-.12 R_____
G01 Z-.89	G01 Z-.89
G02 X1.73 Z-1.13 I_____ K_____	G02 X1.73 Z-1.13 R_____
G01 X2.28	G01 X2.28
G03 X3. Z-1.49 I_____ K_____	G03 X3. Z-1.49 R_____
G01 Z-2.375	G01 Z-2.375

Manually Programming Tool Nose Compensation

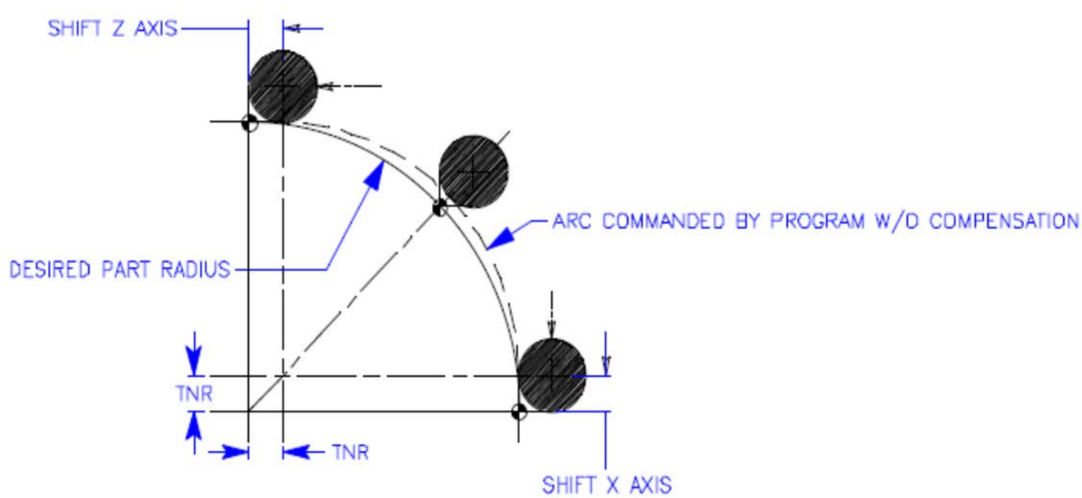
This section is to help those who have not had the time or patience to learn to use TNC. Our suggestion is, if you want to save time and money to learn how to use TNC.

Calculating Compensation for a Radius on Your Part

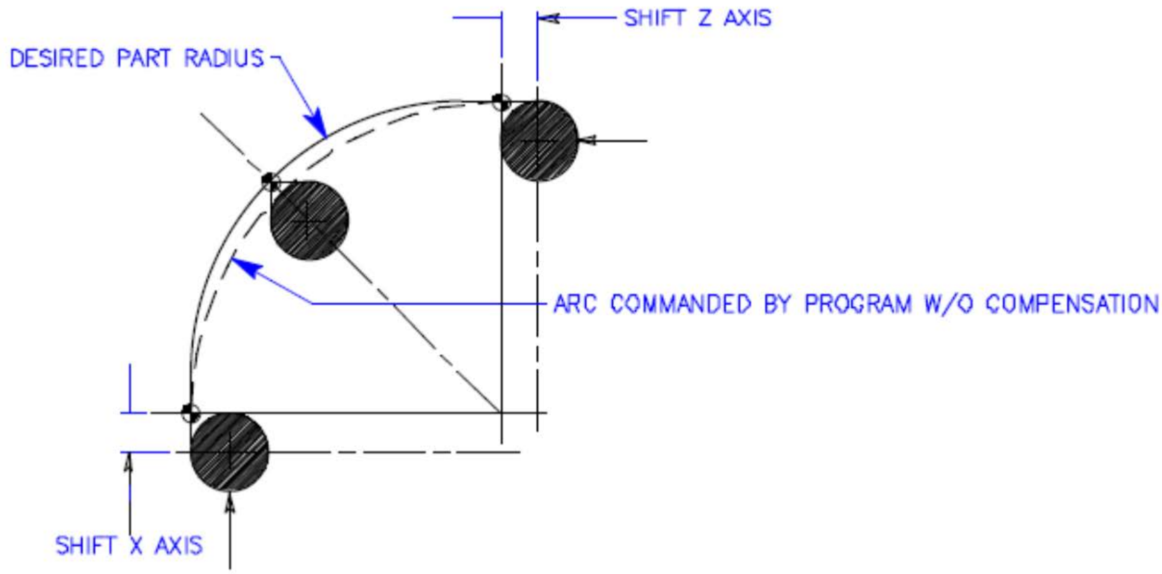
When you program a straight line in either X or Z, the tool tip touches the part at the same point where you touched your original tool offsets in X and Z. However, when you program a radius, the theoretical tool tip does not touch the part radius. Where the tip actually touches the part is dependent upon the radius of the tool and the point around the radius being cut. If you were to try to program a part without using any compensation and you programmed to the finish radius size of your part, you would see a smaller radius on the outside corners of your part and larger radiuses on the fillet radiuses of your part. The amount that will be either a smaller corner radius or a larger fillet radius will be the amount that is the radius of the tool you are machining.

Refer to the illustrations on the pages while reading the text below.

For a 90-degree corner radius, calculate the correct tool path position for a tool with a .031 TNR in your program. Manually calculate the compensated path by adding .031 to the radius to be machined. You will also need to re-calculate the start point and end point of this larger programmed radius.



For a 90-degree fillet radius, calculate the correct tool path position for a tool with a .031 TNR in your program. Manually calculate the compensated path by subtracting .031 from the radius to be machined. You will also need to re-calculate the start point and end point of this smaller programmed radius.

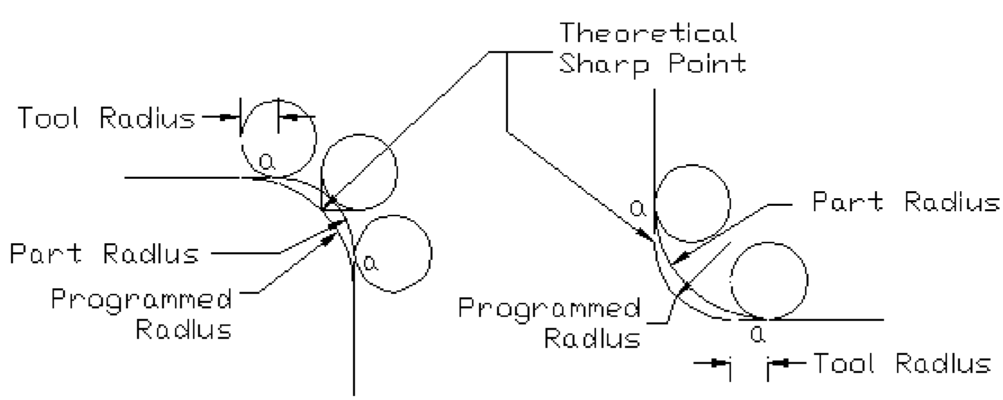


Radius Calculation

When a radius is required TNRC has to be applied.
The basic rule is as follows:

An external radius has the Tool Nose Radius added to it.
An internal radius has the Tool Nose Radius subtracted from it.

The example below shows this:



External Radius Calculation

If a radius of .25 is required on the external corner of a part and the tool nose radius is .031 the programmed radius will be:

.25 + .031 = .281

If the radius is on a 1.0 diameter and at the Z0 face of the part the program will follow the example below.

G1 X0 Z0	**Start of program**
G1 X.438	1.0 –(281 X 2)=.438
G3 X1.0 Z-.281 R.281	Move to the X and Z axis end point
G1 Z-?	Next axis parallel move

Internal Radius Calculation

If a radius of .25 is required on the internal corner of a part and the tool nose radius is .031 the programmed radius will be:

.25-.031 =.219

If the radius is on a 1.0 diameter and at the Z0 face of the part the program will follow the example below.

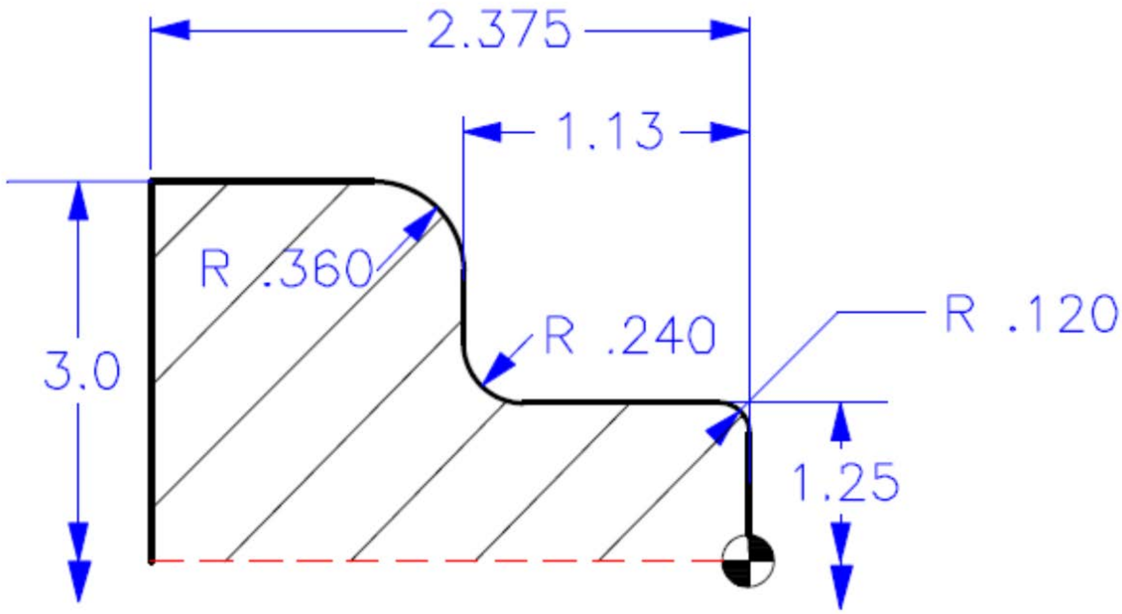
G1 X1. Z0
G1 Z-.781
G2 X 1.438 Z-1. R.219
G1 X-?

Note:

When calculating the end points in a part program you must **double the X axis** calculation to allow for both sides of the part. Any trigonometry **calculation is only generating a radial value** for X.

Circular Interpolation Calculation

Manually calculate compensation for a lathe tool with a .031 TNR to machine a radius on your part.



Manual Programming A Radius
(G02 & G03 using I and K)

```
O0102
 G28
 T101
 G97  S1200  M03
 G54  G00  X.92  Z.05
 G01  Z0.  F.01
 X.948  F.006
 G03  X1.25 Z-.151  I0.  K-.151
 G01  Z-.921
 G02  X1.668  Z-1.13  I.209  K0.
 G01  X2.218
 G03  X3.  Z-1.521  I0.  K-.391
 G01  Z-2.375
 G00  U.01  Z1.0
 G28
```

Manual Programming A Radius
(G02 & G03 using R)

```
O0103
G28
T101
G97  S1200  M03
G54  G00  X.92  Z.05
G01  Z0.  F.01
X.948  F.006
G03  X1.25  Z-.151  R.151
G01  Z-.921
G02  X1.668  Z-1.13  R.209
G01  X2.218
G03  X3.  Z-1.521  R.391
G01  Z-2.375
G00  U.01  Z1.0
G28
```

Types of Calculations

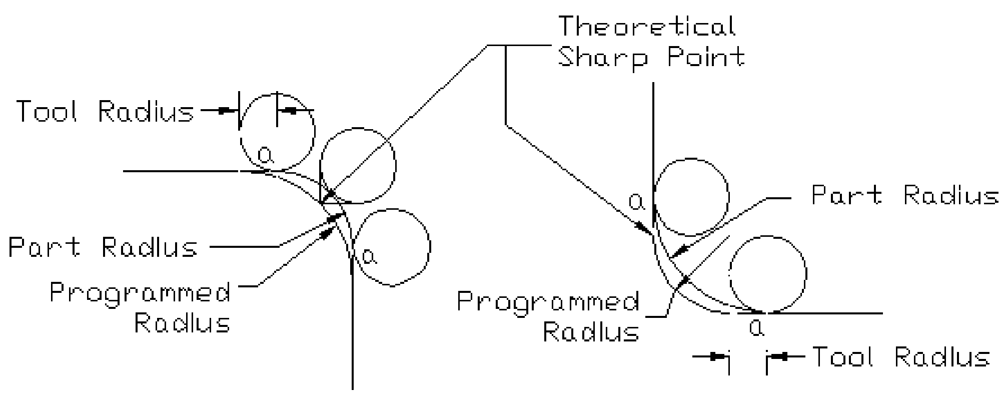
There are calculations for Radii, Tapers, reverse tapers (rear of tool tip) and combinations of both these will be explained in the following section.

Radius Calculation

When a radius is required TNRC has to be applied.
The basic rule is as follows:

An external radius has the Tool Nose Radius added to it.
An internal radius has the Tool Nose Radius subtracted from it.

The example below shows this:



In the above example above the Theoretical start and end points are calculated based on the fact that the tangent point for 2 radii is always through their center.

In the external radius we add the tool nose radius to the part radius and then calculate the actual start and end points.

External Radius Calculation

If a radius of .25 is required on the external corner of a part and the tool nose radius is .031 the programmed radius will be:

.25 + .031 = .281

If the radius is on a 1.0 diameter and at the Z0 face of the part the program will follow the example below.

G1 X0 Z0	*Start of program*
G1 X.438	1.0 –(281 X 2)=.438
G3 X1.0 Z-.281 R.281	Move to the X and Z axis end point
G1 Z-?	Next axis parallel move

Internal Radius Calculation

If a radius of .25 is required on the internal corner of a part and the tool nose radius is .031 the programmed radius will be:

.25-.031 =.219

If the radius is on a 1.0 diameter and at the Z0 face of the part the program will follow the example below.

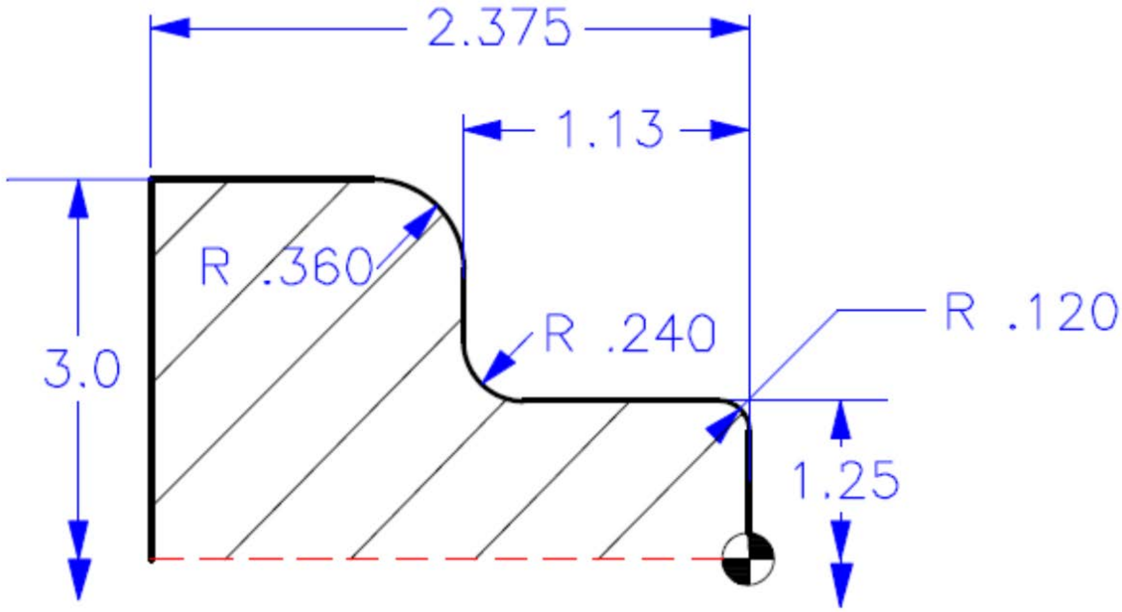
G1 X1. Z0
G1 Z-.781
G2 X 1.438 Z-1. R.219
G1 X-?

Note:

When calculating the end points in a part program you must double the X axis calculation to allow for both sides of the part. Any trigonometry calculation is only generating a radial value for X.

Circular Interpolation Calculation

Manually calculate compensation for a lathe tool with a .031 TNR to machine a radius on your part.



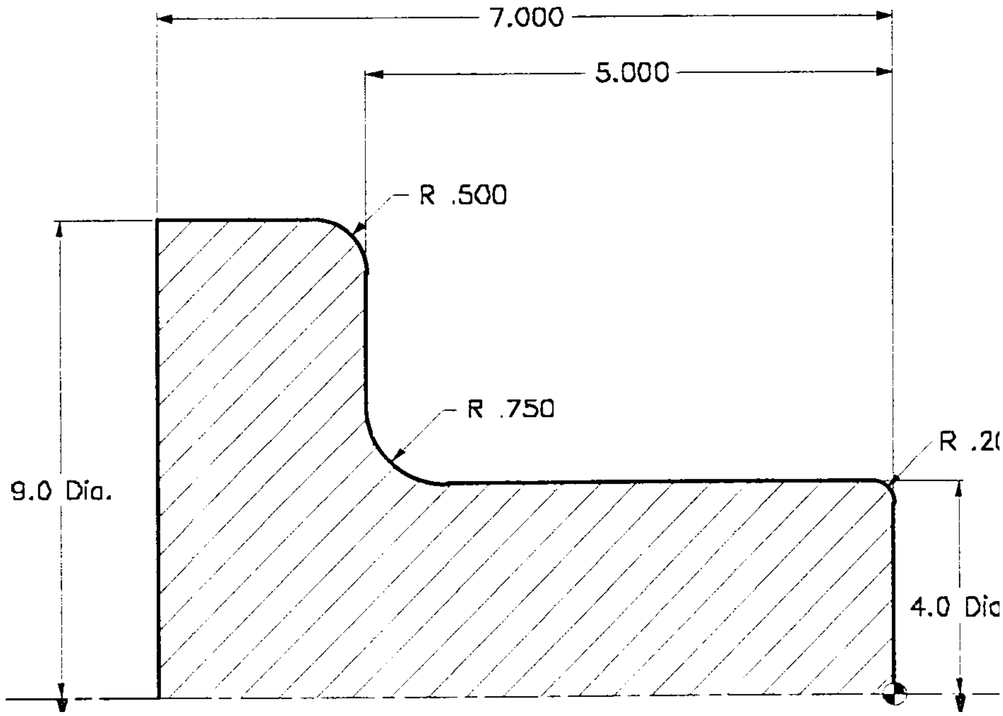
Manual Programming A Radius
(G02 & G03 using I and K)

```
O0102
G28
T101
G97  S1200  M03
G54  G00  X.92  Z.05
G01  Z0.  F.01
X.948  F.006
G03  X1.25 Z-.151  I0.  K-.151
G01  Z-.921
G02  X1.668  Z-1.13  I.209  K0.
G01  X2.218
G03  X3.  Z-1.521  I0.  K-.391
G01  Z-2.375
G00  U.01  Z1.0
G28
```

Manual Programming A Radius
(G02 & G03 using R)

```
O0103
G28
T101
G97  S1200  M03
G54  G00  X.92  Z.05
G01  Z0.  F.01
X.948  F.006
G03  X1.25  Z-.151  R.151
G01  Z-.921
G02  X1.668  Z-1.13  R.209
G01  X2.218
G03  X3.  Z-1.521  R.391
G01  Z-2.375
G00  U.01  Z1.0
G28
```

Circular Interpolation Exercise



Fill in the blanks to complete one finish pass of the cutter path for the above part defining the actual line of part “without” using cutter compensation (G41 or G42) and adding in the actual radius compensation for a .031 radius O.D. turning tool.

O0010
(Circular Interpolation “Without” Cutter Comp)
N1 (Finish O.D.)
G28
T101 (O.D. Turning Tool .031 TNR)
G50 S2200
G97 S291 M03
G54 G00 X4.2 Z0.1 M08
G96 S750
G00 Z0.
G01 X-0.062 F0.004
G0 X3.5 Z0.02

G01 Z0. F0.005
X__________
G03 X________ Z________ R________
G01 Z________
G02 X________ Z________ R________
G01 X________
G03 X________ Z________ R________
G01 Z________
G00 U0.02 Z1. M09
G97 S320
G28
M30

Calculating Compensation For "An Angle" On Your Part

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 the size insert you are using. If you were to try to program your part without using any compensation you would see overcutting and undercutting on your part.

The following pages contain tables and illustrations demonstrating how to calculate the compensation in order to program your part accurately. There are two tables with 1/32 and 1/64 insert radius values in both the X and Z-axes.

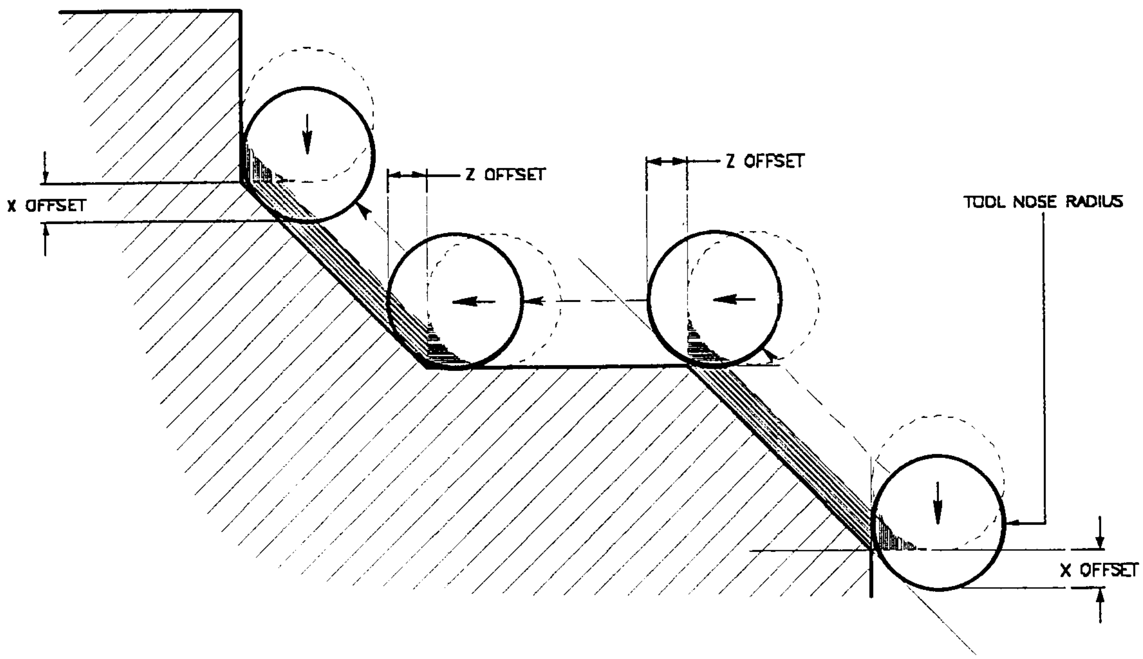
Along with the charts is an illustration example of compensation using offset values added to your program moves. Next is a program exercise using these values to define your part for machining an angle.

Taper Calculation

To simplify tool radius compensation on tapers a chart is provided that gives a range of radii and angle between 0°-45° if a radius greater than 45° is encountered simply use the opposite angle in the triangle and reverse the Xc and Zc values.

Tool Nose Radius Calculation Diagram

Machining a chamfer when cutter compensation is NOT used on the control requires that calculations must be made for the tool tip geometry for the programmed moves on your part angles.



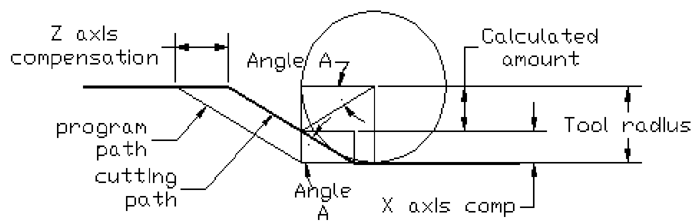
ANGLE	X-OFFSET .015 N/R	Z-OFFSET .015 N/R	X-OFFSET .031 N/R	Z-OFFSET .031 N/R
15	.0072	.0136	.0146	.0271
30	.0132	.0114	.0264	.0229
45	.0184	.0092	.0366	.0183
60	.0228	.0066	.0458	.0132

X-OFFSET numbers above are diametric.

Tool Nose Compensation Taper Calculation

If an angle has to be calculated the formula is described below:

TNRC for X= R-(Tan θ x R)



If the angle is 30° and the radius is .03125, the X-axis calculation is:

X = R-(Tan θ x R)
X = .03125-(Tan30 x .03125)
X = .03125-(.57735 x.03125)
X = .03125-(.01804)
X = .013207 (Note this is a radial number)

TNRC for Z = $\frac{\text{Compensation in X}}{\textbf{Tan } \theta}$

The Z-axis calculation is:

Z = $\frac{.013207}{.57735}$

Z = .02287

If these calculations are compared to your chart, the numbers match.

In practice, it is far easier to use a chart and apply one axis of compensation, and then calculate the end point of the other axis. This method ensures that the correct angle is machined, and any rounding of numbers is taken into consideration.

The following tables are taken from SL-Series Programming and Operation Manual.
Note that Xc values do not have to be multiplied by 2 as they are already calculated as diametric numbers.

TOOL RADIUS AND ANGLE CHART
1/32 RADIUS

NOTE: THE MEASUREMENT CALCULATED IS BASED ON PART DIAMETER

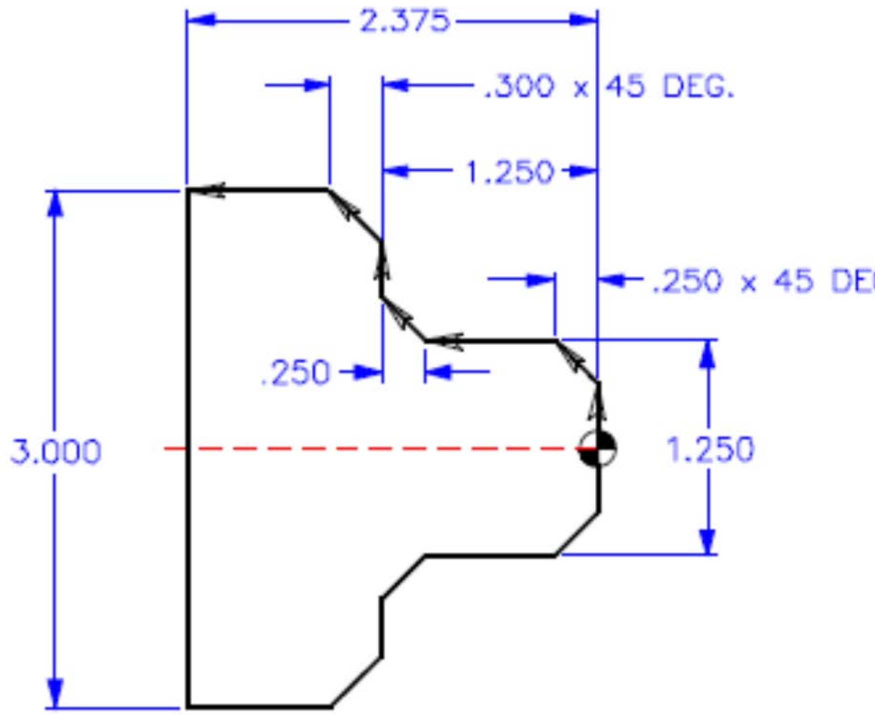
ANGLE	Xc CROSS	Zc LONGITUDINAL	ANGLE	Xc CROSS	Zc LONGITUDINAL
1.	.0010	.0310	46.	.0372	.0180
2.	.0022	.0307	47.	.0378	.0177
3.	.0032	.0304	48.	.0386	.0173
4.	.0042	.0302	49.	.0392	.0170
5.	.0052	.0299	50.	.0398	.0167
6.	.0062	.0296	51.	.0404	.0163
7.	.0072	.0293	52.	.0410	.0160
8.	.0082	.0291	53.	.0416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

TOOL 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			

Angular Straight Line Interpolation Exercise
(Manually Adding Radius Compensation)



Manual Programming a Finish Pass
With an Angle using a .031 TNR:

O0020
T101
G97 S1200 M03
G54 G00 X.85 Z.1 M08
G01 Z0. F.03
X-_____ F.01 (Face)
G00 X_____ Z.02
G___ Z0. F.006
X______
X______ Z-______
Z-______
X______ Z-______
X______
X______ Z-______
Z-______
G00 U.01 Z1.0 M09
G28

MISCELLANEOUS G CODES

G04 DWELL

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; coolant and spindle will remain on. If the P has a fraction part, the delay is in milliseconds (0.01 seconds); otherwise, the delay is in seconds.

EXAMPLE: G04 P1.0 (for a delay of 1.0 second)
or
G04 P25 (for a delay of 25 milliseconds)

INCH / METRIC SELECTION (G20, G21)

Selecting between inch and metric programming can only be done from the Setting page, Setting 9. When changing the setting from inches to metric all work offsets and tool offsets will be converted to mm. The machine will not convert the programs however to metric. This must be done by the programmer.

The G codes G20 and G21 are sometimes used to select between inch and metric. In the HAAS control, the G20 (inch) and G21 (mm) codes can only be used to make sure that the inch/metric setting is set correctly for that program.

Reference Point Definition and Return

G28 RETURN TO MACHINE ZERO AND CANCEL OFFSET (FANUC)

The G28 code is used to machine zero position on all axes. If an X or Z-axis is specified on the same block, only those axes will move and return to the machines' zero point. If X or Z specifies a different location than the current position, then the movement to machine zero will be through the specified point. If no X or Z is specified, all axes will be moved directly to machine zero along with the tailstock. If you do not want to position through an intermediate point while specifying an X or Z-axis to position to machine home, than use an incremental U0 and/or W0 command for the specific axis you want to send to machine zero. This will command those axes specified to position incrementally to a zero distance as an intermediate point, and then those axes will go directly to machine zero. G28 also cancels tool offsets.

G51 RETURN TO MACHINE ZERO AND CANCEL OFFSET (YASNAC)

G51, used on a YASNAC control, cancels out any existing tool wear and work coordinate shift offsets and then returns to the machine zero position.

Spindle Speed Commands

G50 SPINDLE SPEED CLAMP

G50 can be used to clamp 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 in constant surface feed mode G96.

```
N1    G50  S2500;              (Spindle RPM will not exceed 2500 RPM)
N2    G96  S650  M03;          (Enter constant surface speed mode, spindle on)
```

The effect of centrifugal force on its gripping force, unbalanced condition of the work piece, etc., can be constrained by programming a G50 code.

The MAXIMUM spindle speed is designated by the “S” word along with the G50 preparatory command and must be in the same block.

Once the MAXIMUM spindle speed is established, any direct RPM programmed in G97 (direct revolution per minute) or control-calculated RPM from the G96 (constant surface footage mode) exceeding the RPM established by the G50 block is ignored and the G50 “S” word spindle speed is used.

The second use of G50 is for older equipment not having work zero and geometry offset capabilities. This will not be covered because it is not desirable or as easy to use.

G96 CONSTANT SURFACE SPEED ON

G96 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 (radius of the cut). Constant surface speed is maintained by adjusting the spindle speed based on the radius of the cut. The current S code is used to determine the surface speed. G96 is modal.

G97 CONSTANT SURFACE SPEED OFF

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

Work Coordinate System Selection

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.

G52 CHILD OR LOCAL COORDINATE SYSTEM

G52 selects the child coordinate system. It is non-modal and FANUC compatible. M30, Reset or power down will remove any values in G52

G53 NON-MODAL MACHINE COORDINATE SELECTION

G53 temporarily cancels work coordinates offset and uses the machine coordinate system. It is non-modal, therefore, the next block will revert to the work coordinate that was active.

G54-59 SELECT COORDINATE SYSTEM #1 THRU #6

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

G110-G129 COORDINATE SYSTEM #7 THRU #26 OR G154P1 to G154 P99

Feed Command Functions

G98 FEED PER MINUTE

G98 affects how the F address code is interpreted. The value of F indicates inches per minute when Setting 9 is set to INCH and millimeters-per-minute when Setting 9 is set to METRIC. This code is modal.

IPM = Current RPM X IPR

G99 FEED PER REVOLUTION

G99 affects how the F address is interpreted. The value of F indicates inches-per-revolution of the spindle when Setting 9 is set to INCH and millimeters-per-revolution of the spindle when Setting 9 is set to METRIC. This code is modal and is the default feed mode.

IPR = IPM / Current RPM

TOOL NOSE COMPENSATION G CODES

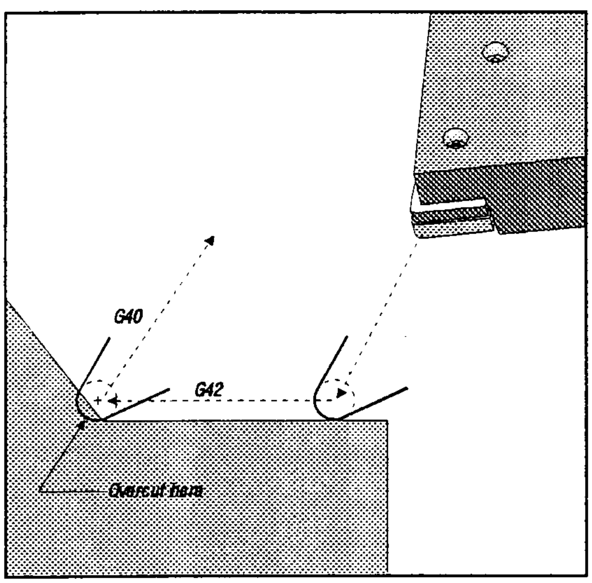
G40 TOOL NOSE COMPENSATION CANCEL

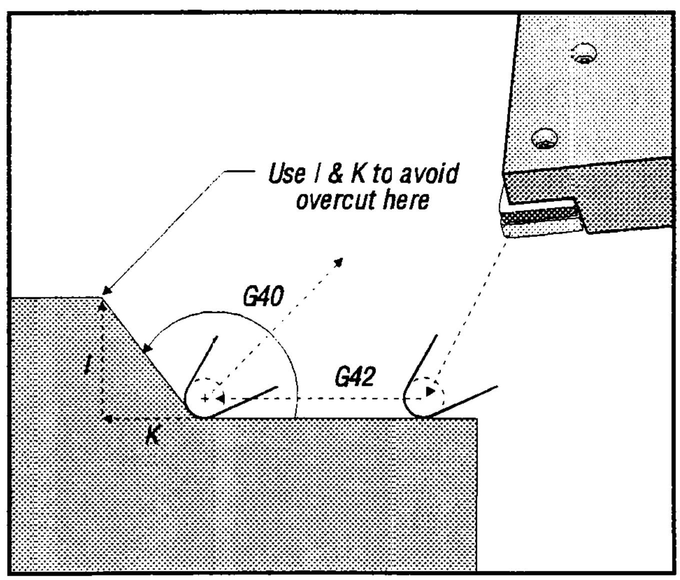
X	X-axis absolute departure location
Z	Z-axis absolute departure location
U	X-axis incremental departure distance
W	Z-axis incremental departure distance
I	X-axis intersection vector direction, (radius)
K	Z-axis intersection vector direction

G40 cancels G41 or G42. Programming Txx00 also cancels tool nose compensation. You must always cancel tool nose compensation before the end of a program.

The departure target location usually does not correspond with a point on the part. In many cases, overcutting or undercutting can occur. Figure 3 illustrates this.

When address codes I and K are used in a G40 departure block, the control will use these values as an intersection vector for the end point of the last, completely compensated, motion stroke. Figure 4 illustrates where I and K lie in relation to the departure stroke. Usually I and K lie along a face of the machined part.



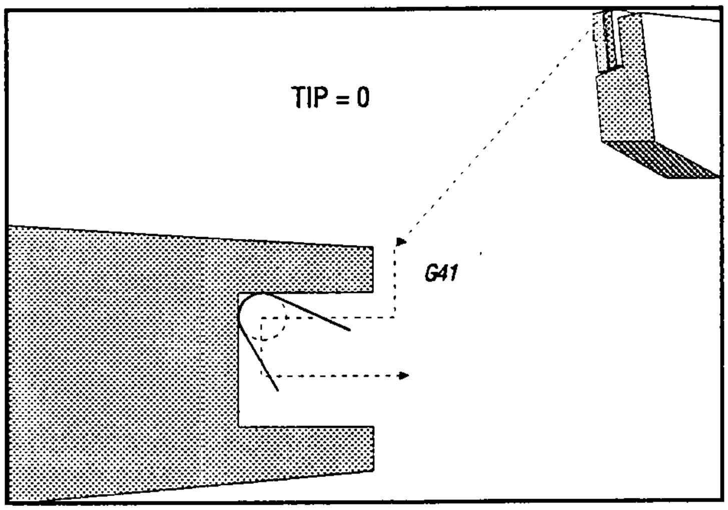


G40 without I and K

G40 with I and K

G41 TOOL NOSE COMPENSATION – LEFT

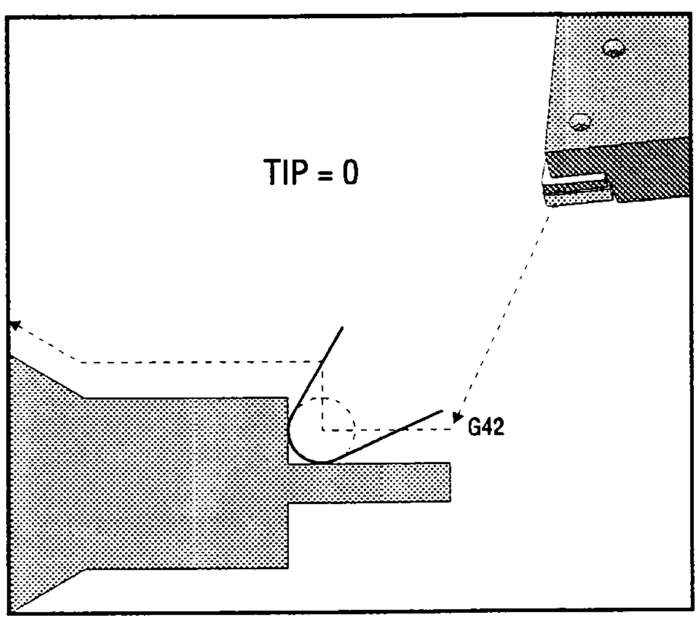
G41 will select tool nose compensation left; that is, the tool is moved to the left of the programmed path to compensate for the size of a tool. A tool offset must be selected with a Tnnxx code, where xx corresponds to the offsets to be used with the tool. If a negative radius is specified, tool nose compensation will operate as though a G42 was programmed.



G41 TNC Left

G42 TOOL NOSE COMPENSATION – RIGHT

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 to be used with the tool.



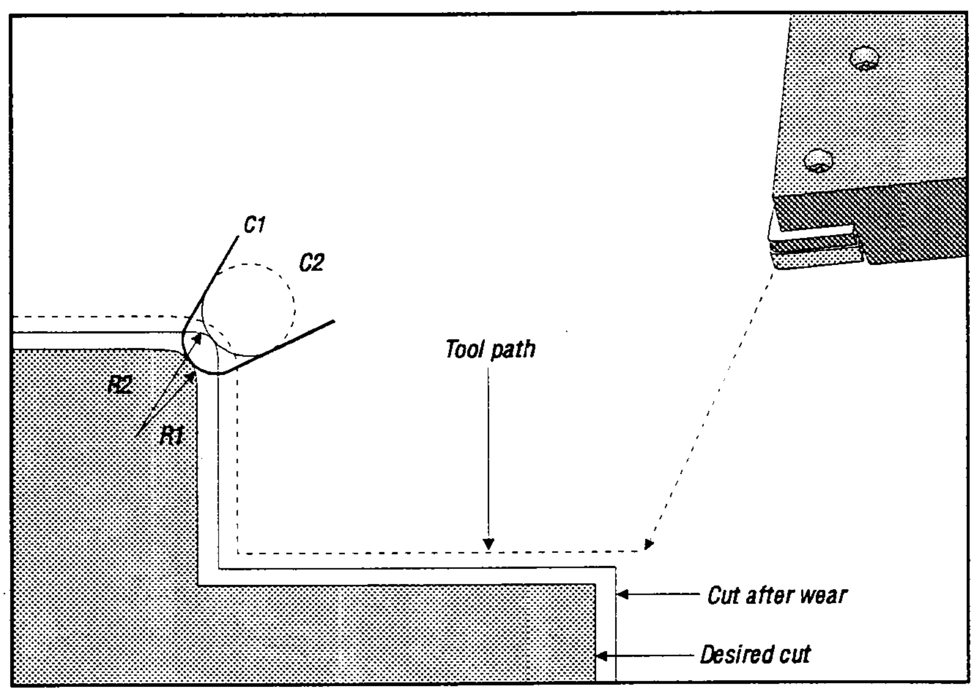
Tool Nose Compensation Programming

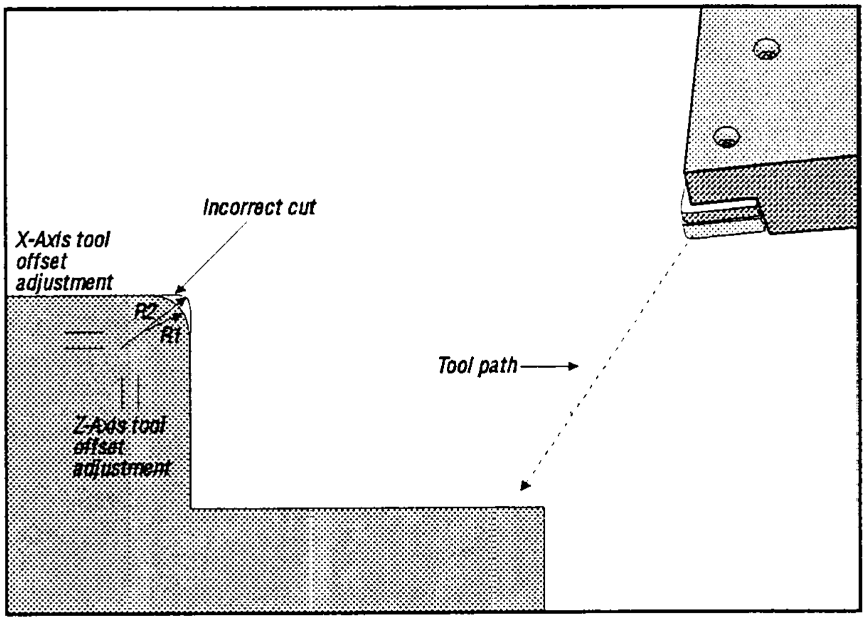
Overview

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

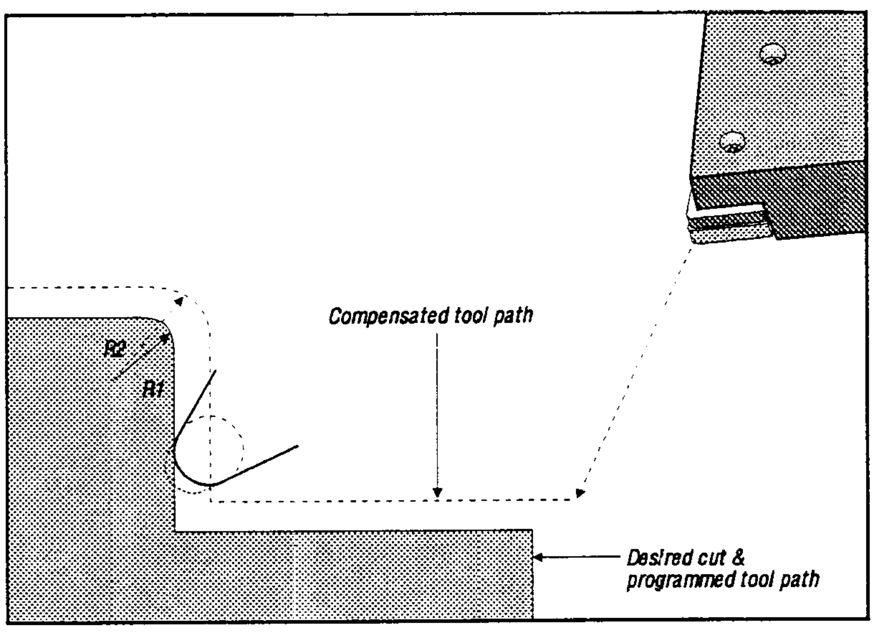
When To Use Tool Nose Compensation

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-axis (diameters) or Z-axis (faces). For angled and circular cuts, as the tool nose radius changes, undercutting or overcutting can occur. In the figure below, 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 8, a smaller radius would occur. If tool nose compensation is used, an incorrect cut does not occur. 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.





Two cuts overlaid to show cutting error

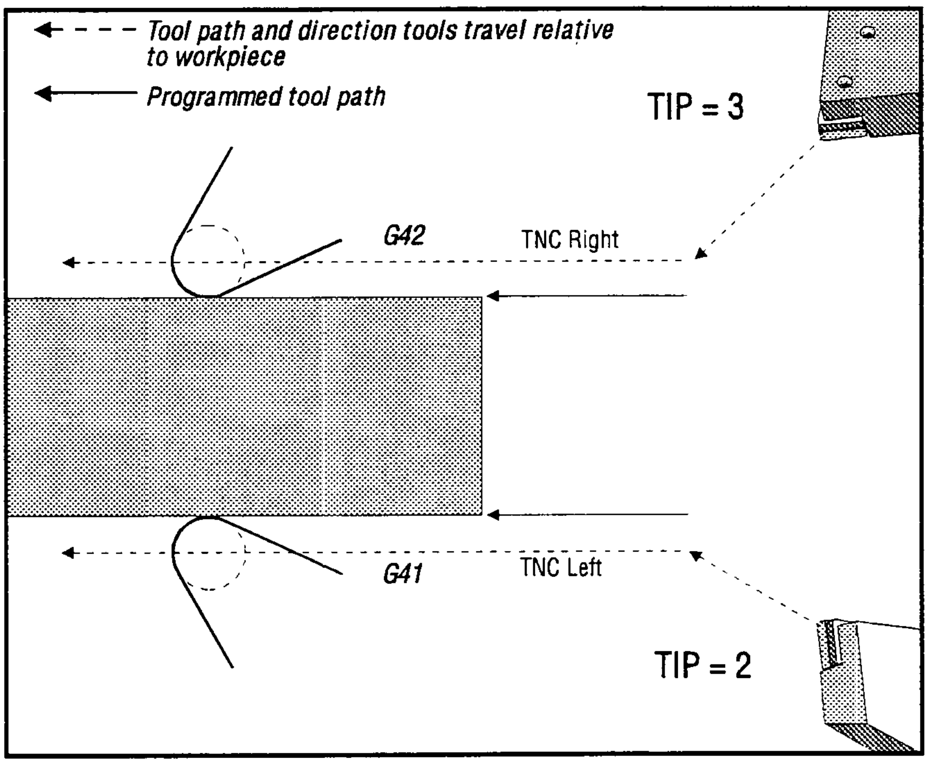


Path generated when tool nose compensation is used

Note that in the above figure, 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 the diameter of a tool based on special instructions written into the program. Two G-code commands are used to do this for compensation within a two-dimensional plane. G41 commands the control to shift to the left of the programmed tool path, and G42 commands the control to shift to the right of the programmed tool path. Another command, G40, is provided to cancel any shift made by tool nose compensation. G40, G41, and G42 are described in detail later in the “Using Tool Nose Compensation G Codes” section.



Shift direction

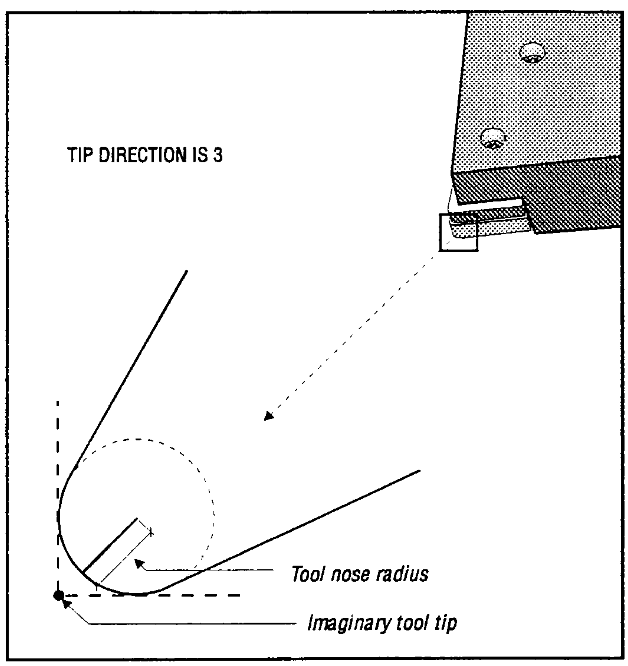
The shift direction is based on the direction of the tool movement relative to the tool and the side of the part it is located. When thinking about which direction the compensated shift will occur in tool nose compensation imagine yourself standing on the line of the part to be cut, looking in the direction the cutter is traveling. Which side of the line is the cutter going to pass you on, left or right? G41 will shift the tool tip on the left and G42 will shift the tool tip on the right. For a lathe, this means that normal O.D. turning, which is machining from the face of the part toward the chuck will require a G42 for the correct tool compensation, while normal I.D. turning will require a G41.

Imaginary Tool Tip and Direction

Tool nose compensation assumes that a compensated tool has a radius at the tool tip that it must compensate for. This is known as the TOOL NOSE RADIUS (TNR). For a lathe, it is not easy to determine the center of a tool nose 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 tool geometry offset information, the radius of the tool, the shift direction the cutter is expected to cut, and the direction the cutter is pointing. The X and Z-axis' geometry offsets intersect at a point, called the IMAGINARY TOOL TIP, which determines 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.

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 14 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.



Imaginary Tool Tip

Using Tool Nose Compensation

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

When the control is first powered on or in the reset condition, tool nose compensation is not active. Tool nose compensation is turned on in a program by the 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.

Follow the steps below when using tool nose compensation.

PROGRAM the part to finished dimensions.

1) Approach and Departure Moves:
   Ensure that there is an approach move for each compensated path that is executed and determine if G41 or G42 is to be used. Ensure that there is also a departure move for each compensated path. Approach and departure moves should be equal to or greater than the tool nose radius of the tool.

2) Tool Geometry Offsets:
   Set the tool length geometry offsets and clear the length wear offsets of each tool.

3) Tool Nose Radius and Wear Offsets:
   Select a standard insert (with a defined radius) that will be used for each tool. Set the tool nose radius offset for each compensated tool. Clear the corresponding tool nose wear offset to start at zero for each tool.

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

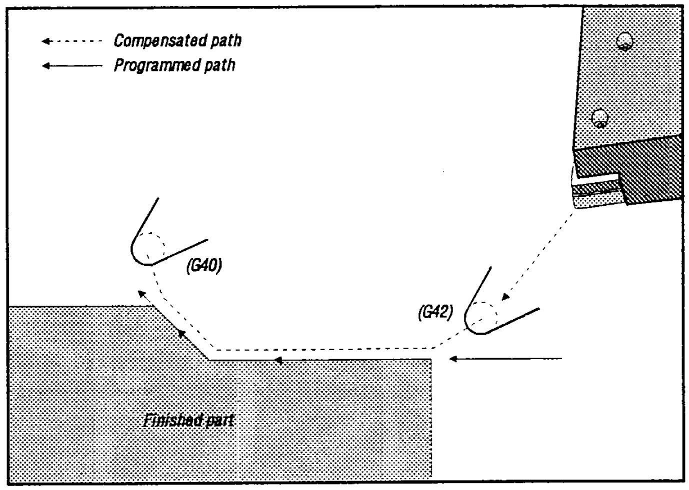
2) Check Compensation Geometry:
   Debug the program in graphics mode and correct any tool nose compensation geometry problems that may occur. A problem can be detected in two ways: an alarm will be generated indicating compensation interference or the incorrect geometry will be seen generated in graphics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

3) Run and Inspect First Article:
   Adjust wear offsets for the setup part.

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

1. *Approach And Departure Moves*

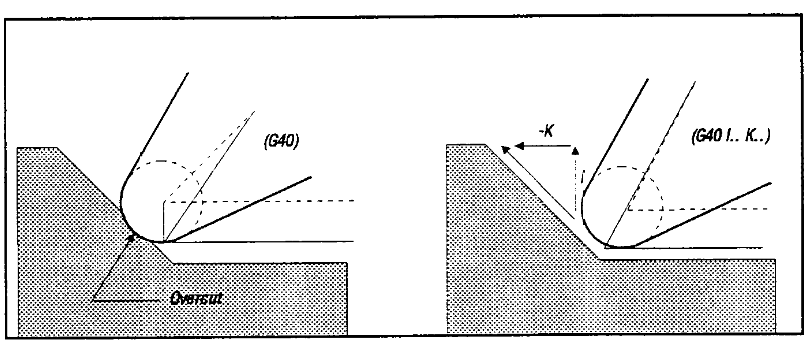
The first X or Z motion in the same line that contains a G41 or G42 is called the APPROACH move. The first move must be a **linear move**, a **G01** or **G00**. At the start of an approach move, the current position is not compensated. At the end of the approach move, the machine position will be fully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



Approach and Departure moves

Any line containing a G40 will cancel tool nose compensation. This is called the DEPARTURE move. The last move must be a linear move, a G01 or G00. At the start of a departure move, the current position is fully compensated. At the end of the departure move, the machine position is not compensated.

NOTE: Figure 13 shows the normal condition just prior to cancellation of tool nose compensation. Some geometry's will result in overcutting 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 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 13 shows how I and J can correct undesired cutting in a departure move. Refer to the G40 command description for instructions on calculating values of I and K.



Use of I and K in a G40 block.

2. Tool Geometry Offsets

The length geometries offsets 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 your operation 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 OFFSETS rather than the RADIUS WEAR OFFSETS. 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 part design "may not allow" the operator to compensate for wear by using the 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.

3. Tool Nose Radius And Wear Offsets

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.

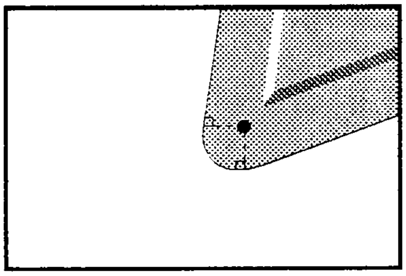
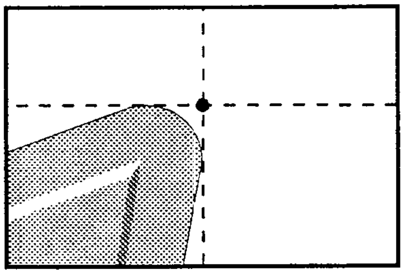
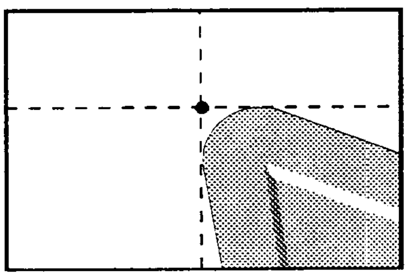
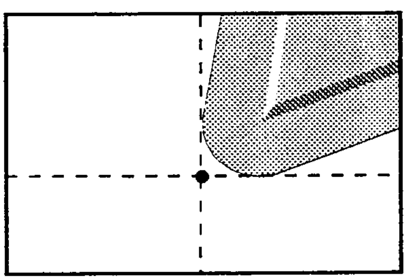
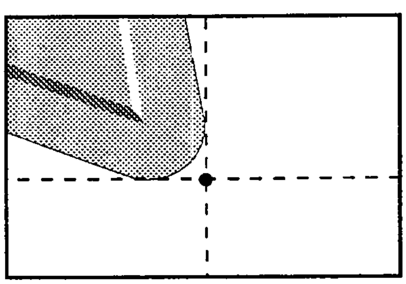
Associated with each radius offset is a RADIUS WEAR OFFSET. It is located on the wear offset page. The control adds the wear offset to the radius offset to obtain an effective radius that will be used for generating compensated values.

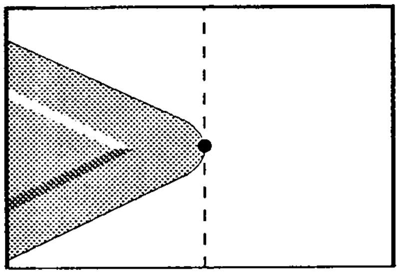
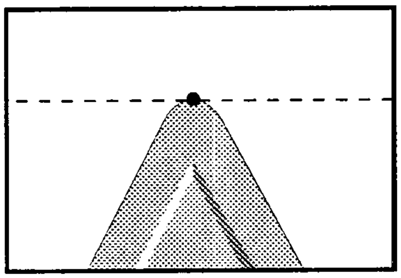
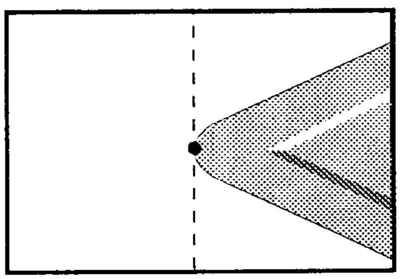
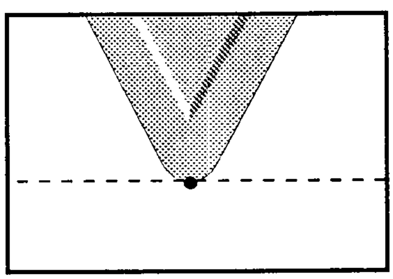
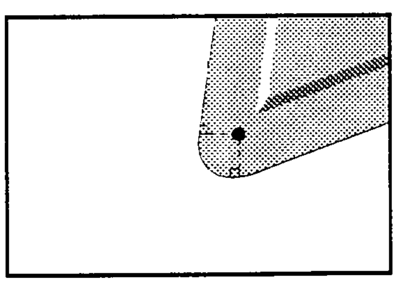
Small adjustments to the radius offset during production runs should be placed in the wear offset page. This allows the operator to easily track the wear for a given tool. As a tool is used, the insert will generally wear so that there is a larger radius at the end of the tool. This should place positive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

4. Tool Tip Direction

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

Note that the tip indicates to the setup person how the programmer intends 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 TIP CODE #	TOOL TIP DIRECTION ORIENTATION	COMMENT TOOL CENTER LOCATION
0		**Zero (0) indicates no Specified direction. It is usually not used when tool nose compensation is desired.**
1		**Tool tip off in the X+, Z+ direction.**
2		**Tool tip off in the X+, Z- direction**.
3		**Tool tip off in the X-, Z- direction**.
4		**Tool tip off in the X-, Z+ direction.**

TOOL TIP CODE #	TOOL TIP DIRECTION ORIENTATION	COMMENT TOOL CENTER LOCATION
5		**Tool Edge** **Z+ direction**
6		**Tool Edge** **X+ direction**
7		**Tool Edge** **Z- direction**
8		**Tool Edge** **X- direction**
9		**Same as tip 0**

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 or turned on. **If the incremental distance of a departure or approach move in a compensated path is not twice the radius of the cutting tool's radius, overcutting may occur.** Always remember that programmed paths are in terms of diameter and allow for twice the tool radius on approach and departure moves. The block of canned cycles requiring a PQ sequence can often be a departure move. The following example illustrates how incorrect programming will result in overcutting.

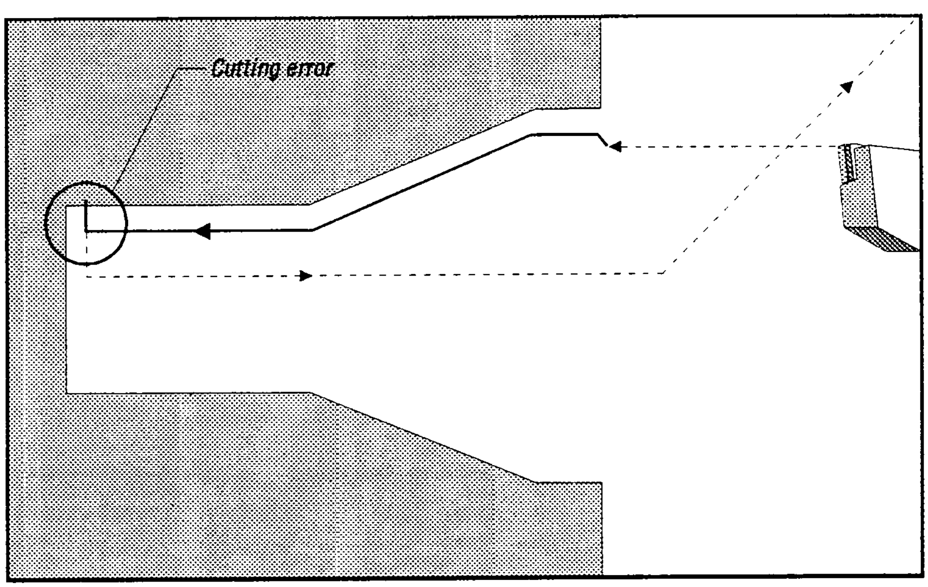
Example

Setting 33 is FANUC:	**X**	**Z**	**Radius**	**Tip**
Tool Geometry Offset 8:	-6.0000	-8.000	0.0156	2

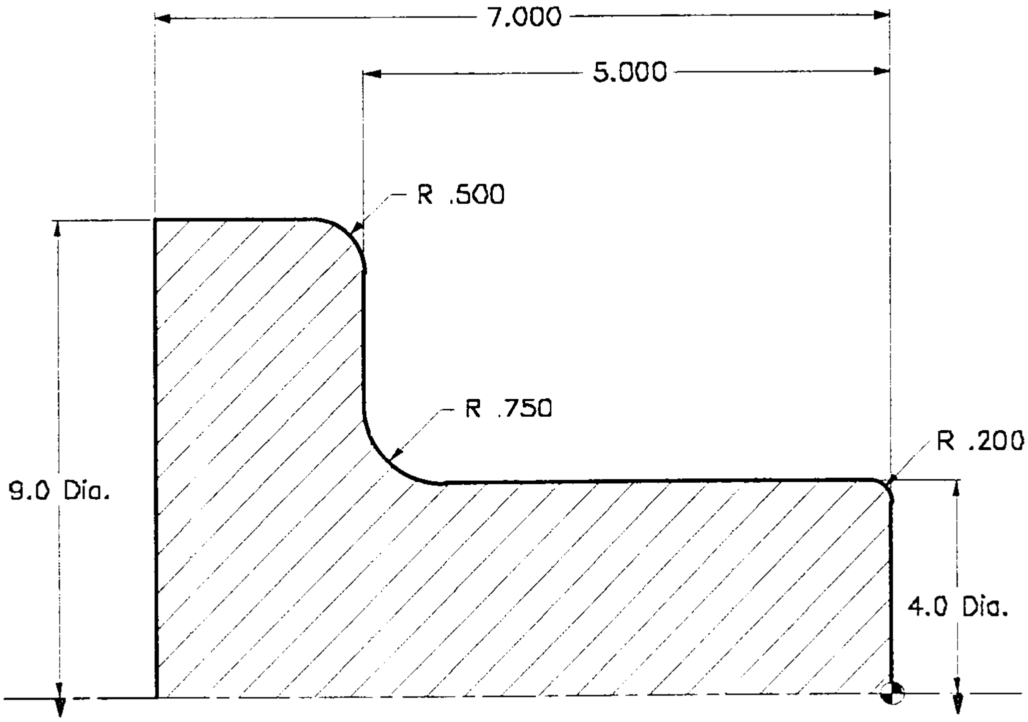
O0104;
G28;
T808; (Boring Bar);
G97 2400 M03;
G54 G00 X.49 Z.05;
G51 G01 X.5156 F.004;
Z-.05;
X.3438 Z-.25;
Z-.5;
X.33;

Note: 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;



Circular Interpolation Exercise with Tool Compensation



Fill in the blanks to complete one finish pass of the cutter path for the above part. Assume the part has already been roughed out. Define the tool path of the part using cutter compensation (G41 or G42) where the tool nose radius is .031 radius for an O.D. turning tool.

```
O0057
(Circular Interpolation With Cutter Compensation)
N1      (Finish O.D.)
G28
T101 (O.D. Turning Tool .031 TNR)
G50  S2200
G97  S____  M03
G54  G00  X4.2  Z0.1  M08
G96  S750
G00  Z0.
G01  X-0.062  F0.004
G0  X3.5  Z0.10
```

```
G01 G____ Z0.  F0.005 (TURN ON COMP)
 X_______
G03  X_______ Z-_______ R_______
G01  Z_______
G02  X_______ Z-_______ R_______
G01  X_______
G03  X_______ Z-_______ R_______
G01  Z-_______
G00  G40  X_____  Z1.  M09 (COMP OFF)
G97  S320
G28
M30
```

Canned Cycles and Additional G Codes

One of the most common types of codes in CNC programming are "canned cycles." This unit will define and discuss canned cycles and state some uses and applications.

Objectives:

Upon completion of this unit, the student will:

1) Be able to define "canned cycle."

2) Be familiar with common canned cycles, their variables, the proper applications, and the correct use of them.

3) Complete final exercise.

Machining Cycles for the Lathe

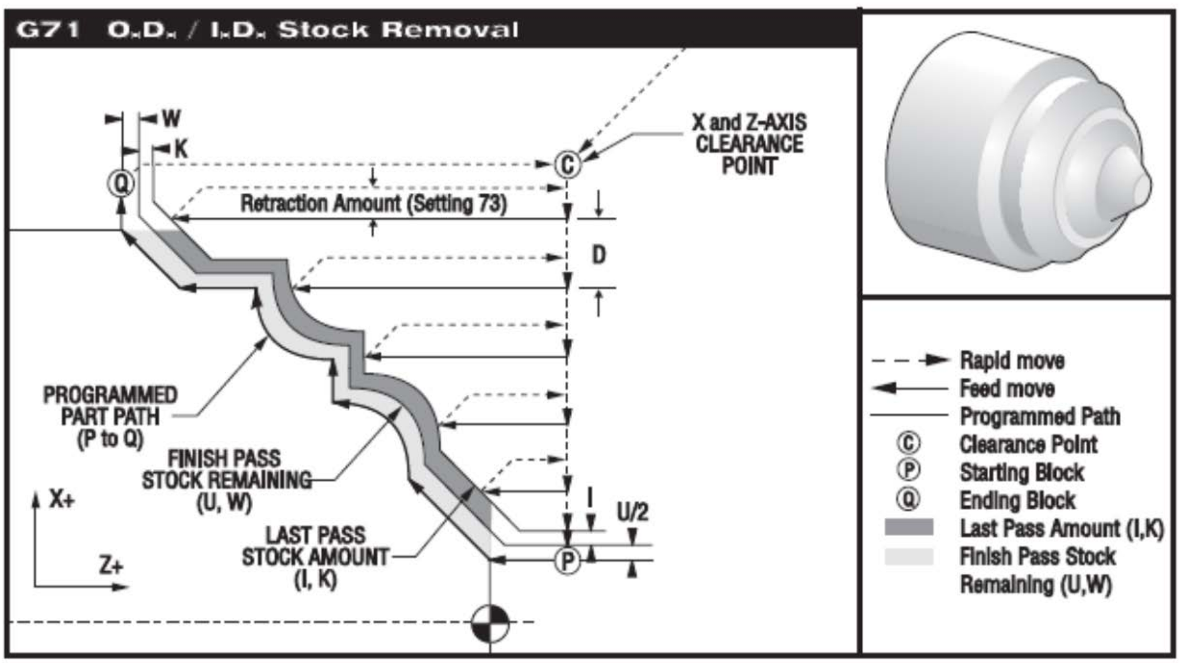
A machining cycle is used to simplify the programming of a part. Machine cycles are defined for the most common machining operations. They can be divided into two types. There are machine cycles for turning and grooving. There are cycles for drilling and tapping and can be either single block canned cycles or modal canned cycles.

The following is a list of the canned cycles that can be used for turning and grooving for the HAAS lathe controls:

G70	Finishing Cycles
G71	O.D./I.D. Stock Removal Cycle
G72	End Face Stock Removal Cycle
G73	Irregular Path Stock Removal Cycle
G74	End Face Grooving or Turning with Chip Break Cycle
G75	O.D./I.D. Grooving or Turning with Chip Break Cycle
G76	Thread Cutting Cycle, Multiple Pass
G90	O.D./I.D. Turning Cycle Modal
G92	Thread Cutting Cycle Modal
G94	End Face Cutting Cycle Modal

G71 O.D./I.D. Stock Removal Cycle

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



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, are 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 defined within the PQ block definition.

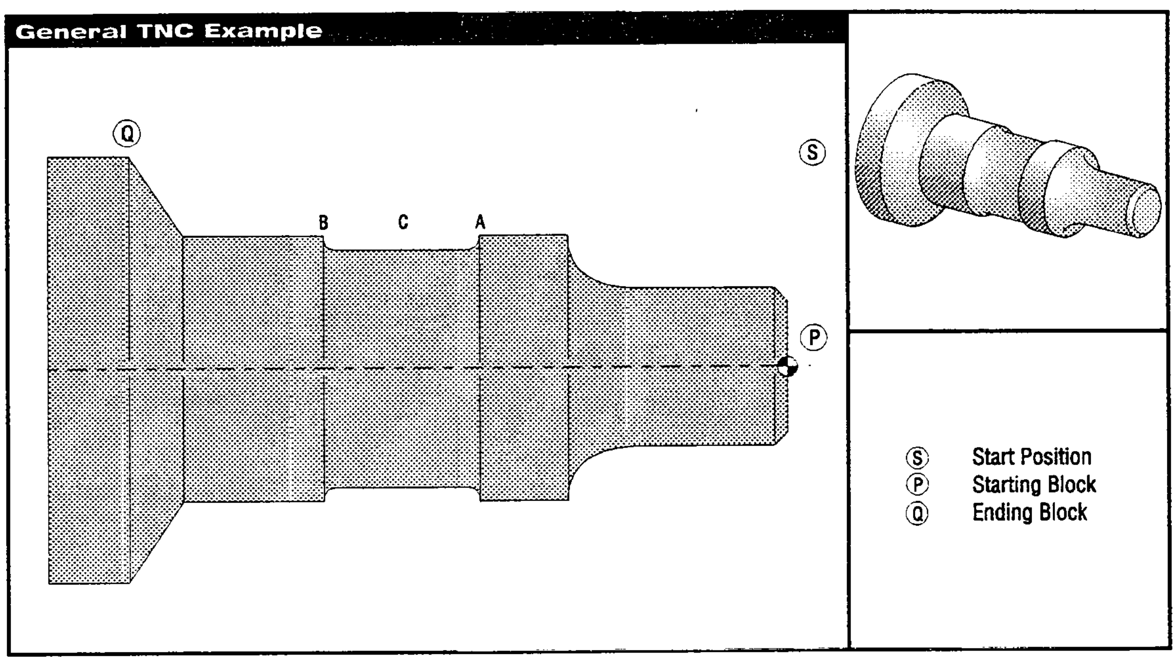
Two types of machining paths are addressed with a G71 command. The first type of path, a **Type I**, is when the X-axis of the programmed path does not change direction. This type of path is called a monotonic path. The second type of path, a **Type II**, allows the X-axis to change direction. For both the first and second type of programmed path, the Z-axis must be monotonous; it cannot change direction.

For FANUC, **Type I** is selected by having only an **X-axis motion** in the block specified by the **P block** in the G71 call. For FANUC, **Type II** is selected by having both an **X-axis** and **Z-axis** motion in the **P block**. When in YASNAC mode, Type II roughing is selected by including R1 on the G71 command block.

G71 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 Z-axis at the specified G71 command line feed rate. The finishing phase consists of a G70 block using the programmed feeds, speeds, tools and offsets defined within the PQ block definition. **The final motion** in either types is a **return to the starting position S**.

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.

Program Example using G70 andG71



Tool Description:

Tool 1	80 Degree CNMG 432 Roughing insert .0312 Radius
Tool 2	55 Degree TPG 432 Finishing insert .0312 Radius
Tool 3	.250 Wide Grooving Tool with .0156R

Tools	Offsets	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	-8.8400	-12.5880	.016	4

Program Example

```
%
O0105 (General TNC Example)
N1
G28                                      (Return to machine zero for tool change)
T0101                                     (Select Tool 1 Offset 1)
G50  S1000
G97  S500  M03
G54 G00  X2.1  Z0.1                      (Rapid to Point S, Defines Stock)
G96  S300
G71  P10  Q20  U0.02  W0.01  D.1  F0.01  (Rough P to Q using G71 and TNC)
                                         (Define part path PQ sequence)
N10  G42 G00  X0  Z0.1  F.01             (P  Block) (G71 type II, TNC approach)
G01  Z0  F.005
X0.65
X0.75  Z-0.05
Z-0.75
G02  X1.25  Z-1.  R0.25
G01  Z-1.5
G02  X1.  Z-1.625  R0.125
G01  Z-2.5
G02  X1.25  Z-2.625  R0.125
G01  Z-3.5
X2.  Z-3.75
N20  G00  G40  X2.1                      (TNC departure) (Q block)
G97  S500
G28                                      (Zero for tool change clearance)
M01

N2
G28                                      (Return to machine zero for tool change)
T202                                     (Select Tool 2 Offset 2)
G50  S1000
G97  S750  M03
G00 X2.1  Z0.1                           (Rapid to point S)
G96  S400
G70  P10  Q20                            (Finish P to Q using G70 and TNC)
G97  S750
G28                                      (Return to machine zero for tool change)
M01
```

```
N3
G28                          (Return to machine zero for tool change)
T303                         (Select Tool 3 Offset 3)
G50  S1000                   (Groove to Point B using Offset 3)
G97  S500 M03
G00  G42 X1.5  Z-2.0         (Rapid to Point C)(TNC approach)
G96  S200
G01  X1.  F0.003
G01  Z-2.5
G02  X1.25 Z-2.625 R0.125
G40  G01  X1.5               (TNC Depart)

                             (Move to right side of groove, to
                             Point A, using Offset 13)
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 Depart)
G97  S500
G28                          (Zero for tool change clearance)
M30
%
```

Note: Two offsets must be used when finishing the groove using tool nose compensation. Here, the same tool is used, but the tool cuts on different sides along the Z-axis. Offset 3 Z length geometry is set on the left side of the tool whereas offset 13 Z length geometry is set on the right side of the tool.

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 the canned cycle is to work with.

G70 Finishing Cycle

Usually, G70 is used following the use of a G71, G72 or G73, but it can be used alone. Below is the template for using tool nose compensation with G70 alone. Note that TNC approach is part of the PQ path definition sequence, whereas TNC departure is after the execution of G70.

G71 and G72 are similar canned cycles with regard to tool nose compensation. The finishing and rough finishing passes of G71 and G72 recognize tool nose compensation; however, the roughing pass of these two G codes does not. The template below can be applied to either G71 or G72.

G73 is similar to G71 and G72. G73 recognizes TNC on all passes.

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

The G70 Finishing cycle can be used to finish cut paths that are defined and roughed out with stock removal cycles G71, G72 and G73.

The G70 requires a beginning block number (P code) and an ending block number (Q code) be specified for the machine code defining the part geometry to be machined.

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 between the P and Q block are effective. The PQ sequence is searched for in the current program starting from the beginning of the program. 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 containing an N code matching the Q code in the G70 calling block is found and executed.

After execution of the Q block, a rapid (G00) is executed returning the machine to the start position saved earlier during G70 initialization. The program then returns to the block following the G70 call.

EXAMPLE G70 WITH TNC

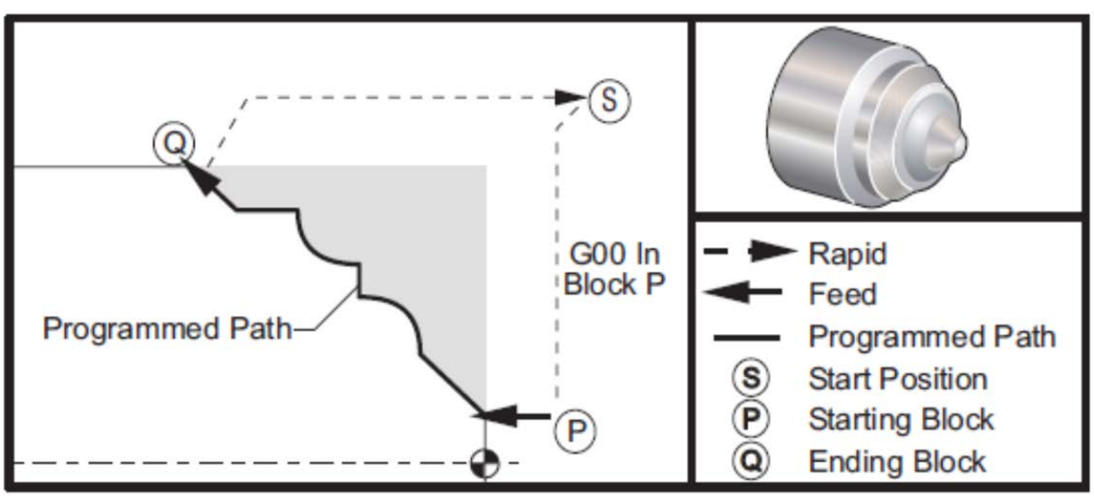
A traditional calling sequence using G70 appears below.

Rough cycle feedrate to rough out part between N10 and N50 for the G71 roughing cycle will be F.012, which is defined in the G71 block.

Finish Feedrate for the G70 finishing cycle will be whatever is defined between N10 and N50.

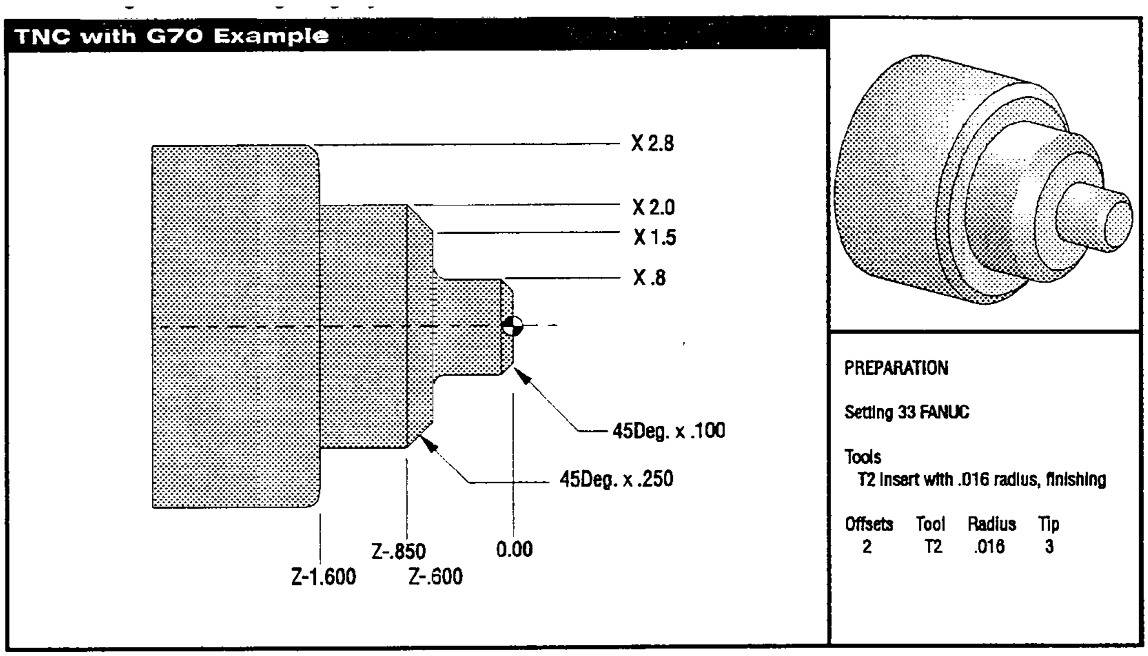
G71 P10 Q70 U.01 W.005 D.08 F.012	(Roughing cycle feedrate will be F.012 which is defined this block.)
N10 G00	(Machine code that defines part path geometry)
….	" " " " F.005
….	" " " "
….	" " " "
….	" " " " F.003
….	" " " " F.008
N70	" " " "
G70 P10 Q70	(Finish cycle feed rate is defined between N10 to N50).

This example demonstrates the use of a G70 finishing canned cycle. The basic shape should be roughed out using a G71, G72, or G73 roughing cycle.



Part program using a G70 with TNC

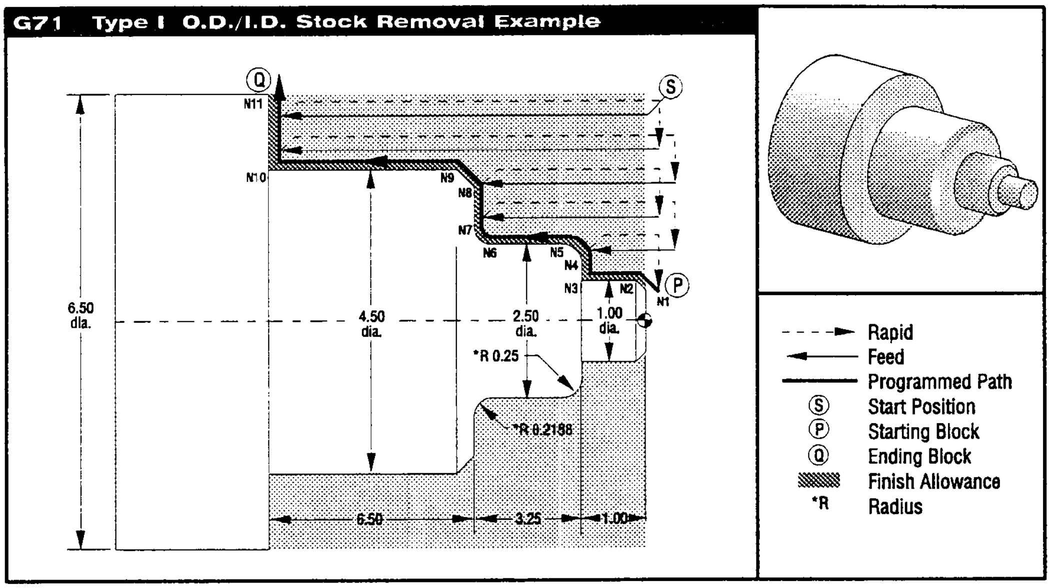
This example demonstrates tool nose compensation with the G70 finishing cycle. The basic shape has been roughed out using a G71 roughing cycle.



Part drawing for TNC with G70 example Type 1

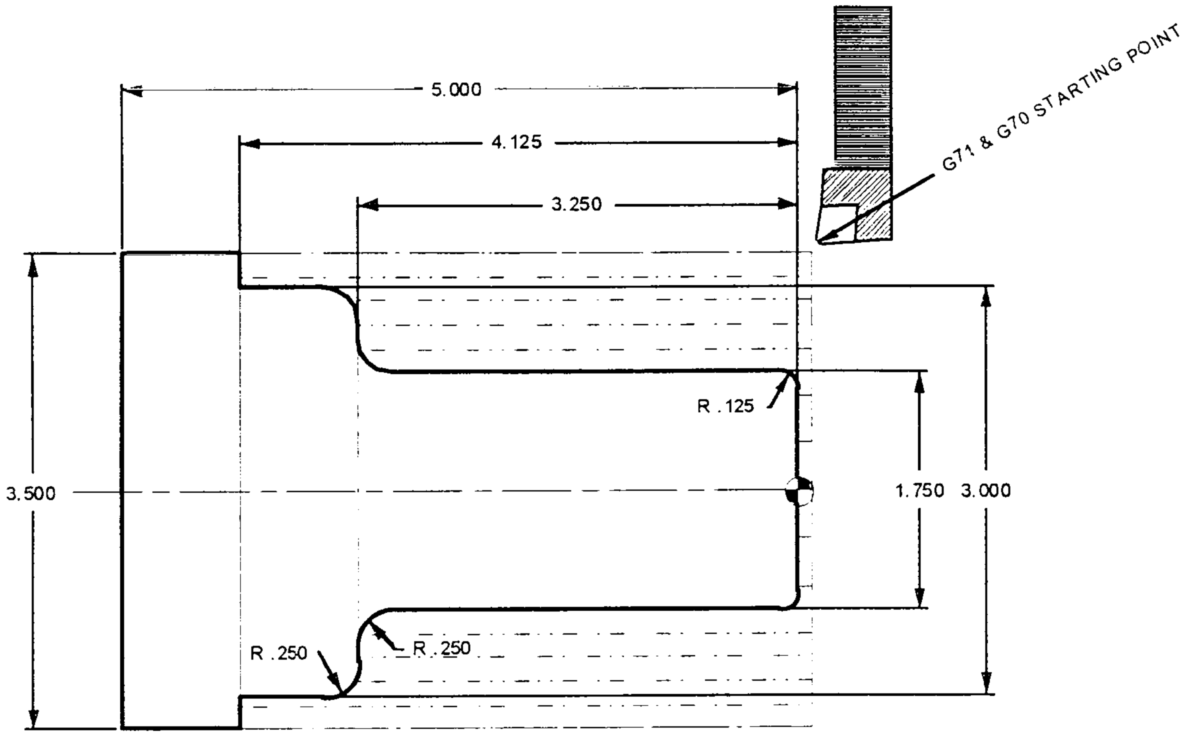
Program Example	Description
%	
O0106	
G28	
T0202	(Select Tool 2)
G50 S1750	
G97 S320 M03	
G54 G00 X3. Z0.1 M08	
G96 S300	
G71 P10 Q20 U.02 W.01 D.1 F.012	(G71 Define part path lines P thru Q)
N10 G42 G00 X0.5	(P Block)(Approach move, turn on tool nose compensation)
G01 Z0. F.012	
X.6	
X0.8 Z-0.1 F.008	
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	
N20 G00 G40 X3.0	(Q)(Departure move turning off tool nose compensation)
G70 P10 Q20	(G70 Finishing Cycle)
G28	(Return to machine zero for tool change clearance)
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.



O0107 (O.D. Roughing and Finishing)	(FANUC G71 – TYPE I example)
G28	(Rapid to Home Position)
T101 (CNMG 432)	(Tool change & apply offsets)
G50 S2000	(Set Max RPM 2000)
G97 S400 M03	(Spindle On)
G54 G00 X6.6 Z0.1 M08	(Rapid to Start Position, Coolant On)
G96 S630	(Constant surface speed on)
G71 P10 Q20 U0.01 W0.005 D0.15 F0.012	(G71 Roughing Cycle)
N10 G00 X0.6634	(P Block)(Begin definition)
G01 X1. Z-0.1683 F0.004	
Z-1.	
X1.9376	
G03 X2.5 Z-1.2812 R0.2812	
G01 Z-40312	
G02 X2.9376 Z-4.25 R0.2188	
G01 X3.9634	
X4.5 Z-4.5183	
Z-10.75	
N20 X6.5	(Q Block)(End definition)
G97 S400 M09	
G28	(Rapid to tool change position)
T202	(Finish tool)
G50 S2500	
G97 S400 M03	
G54 G00 X6. Z0.1 M08	
G96 S730	
G70 P10 Q20	(G70 Finishing Cycle)
G97 S400 M09	
G28	
M30	
%	

G71 and G70 Exercise with Tool Nose Compensation



Fill in the blank line for tool #1 using a G71 roughing cycle command to define roughing passes for the part geometry defined between N10 and N20. Then, define a finish pass using a G70 finishing cycle command, with tool #2, to define a finish pass for the part geometry defined between N10 and N20. On the G71 command line, leave .010 stock on diameters and .005 on the faces. Take .120 depth of cut at .012 feed.

O0030	(Program number)
N1	(First Operation)
T101 (O.D. Tool X.031 TNR)	(Tool #1 and Offset #1)
G50 S2500	(Spindle speed clamp at 2500 RPM)
G97 S400 M03	(Cancel CCS, define a 400 RPM, spindle ON forward)
G54 G____ X____ Z____ M08	(Rapid X and Z to starting location, coolant ON)
G____ S_____	(Turn on CSS to 375)
G_____ X______ Z______	(Rapid to start position above part)
G71 P____ Q____ U____ W____ D____ F____	(Roughing O.D. G71 cycle command)
N____ G____ G____ X______ Z_______	(Pnn starting number, Rapid X and Z axis, cutter comp ON)
G____ Z____ F.006	(G71 Part Geometry)
X______	(G71 Part Geometry)
G____ X______ Z______ R_____	(G71 Part Geometry)
G_____ Z______	(G71 Part Geometry)
G____ X______ Z______ R_____	(G71 Part Geometry)
G_____ X______	(G71 Part Geometry)
G____ X______ Z______ R_____	(G71 Part Geometry)
G____ Z______ F.003	(G71 Part Geometry)
X______ F.015	(G71 Part Geometry)
N____ G____ X____	(Qnnnn ending number, Cancel Cutter Comp.)
G____ S_____ M____	(Cancel CSS, define 400 RPM, spindle ON forward, Coolant Off)
G____	(Return to reference point)
M_____	(Optional stop command)
N2	(Second operation)
T_____	(Tool #2 and Offset #2)
G____ S_____	(Spindle speed clamp at 2500 RPM)
G____ S_____ M____	(Cancel CSS, turn on spindle 1200 RPM)
G____ X______ Z______	(Rapid to start position above part)
G____ S_____	(Turn on CSS to 650)
G____ P_____ Q_____	(Define finish pass using part geometry)
G____ S____ M____	(Cancel CSS, define 400 RPM, spindle on, Coolant Off)
G____	(Return to reference point)
M____	(End of program rewind)

Type 1 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

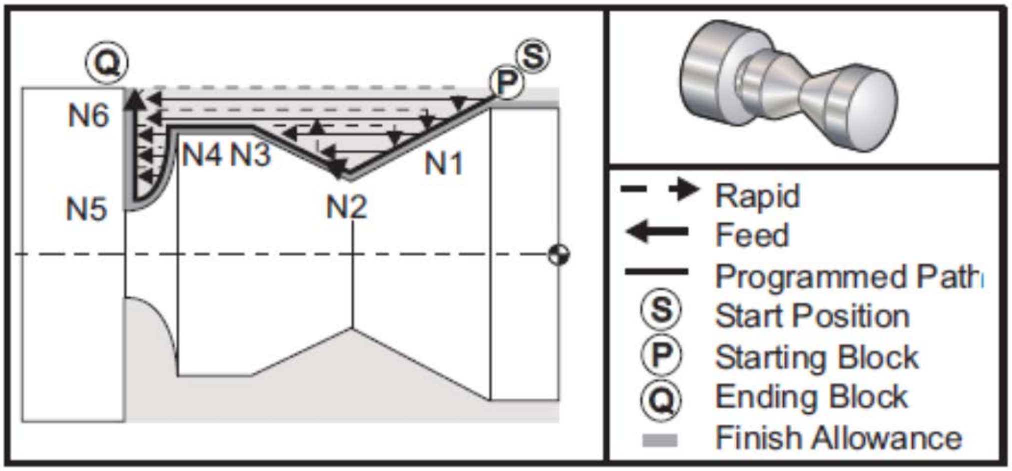
When Type II is specified by the programmer the X-axis PQ path is allowed to vary non-monotonically. In other words, the X-axis can change direction throughout the PQ path. Z must continue along in the same direction as the initial Z direction. The PQ path is checked prior to the start of any cutting and an alarm is generated if a problem exists.

When Setting 33 is set to FANUC, placing a reference to both the X and Z-axis in the block specified by P specifies Type II.

Roughing is similar to Type I except that 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.

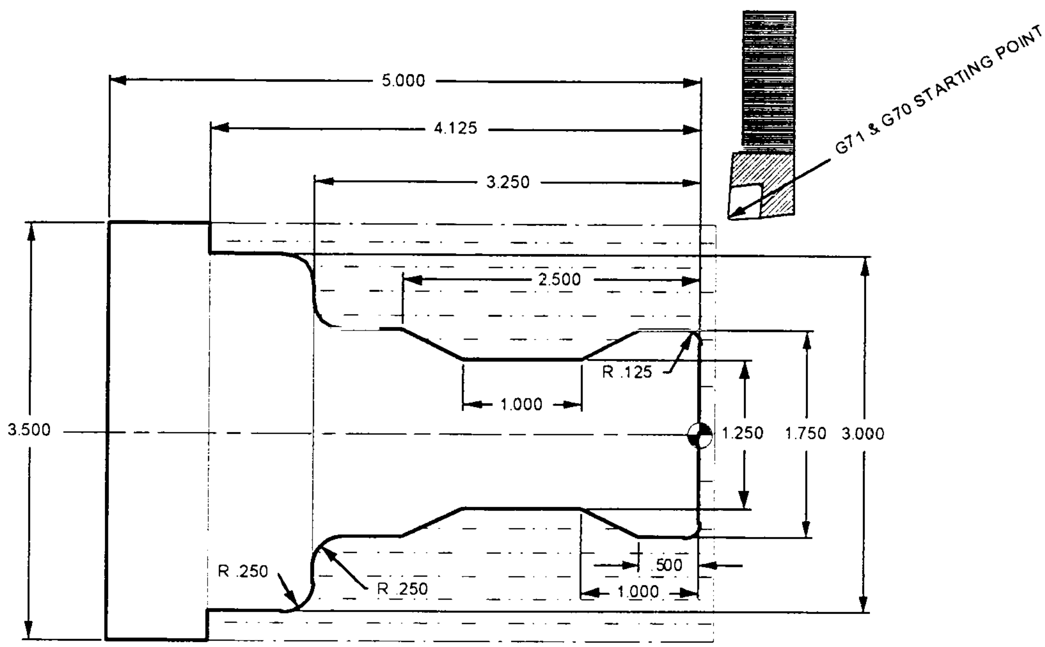
There is virtually no limit to the number of blocks used to define a Type II PQ path. There is a limit to the number of troughs included in a PQ path definition. A trough can be defined as a change in direction creating a concave surface in the material being cut. If successive troughs are on the same 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.

Example G71 Type II Roughing




```
%
O0135                                   (Fanuc G71 Type II roughing)
G28
T101                                    (Roughing Tool)
G97 S1200  M03
G00  G54 X2.  Z0.05                     (Start position)
G71  P1  Q6  D.035  U.03  W.01  F.01
N1  G1  X1.5  Z-.5 F.004                (PQ path defnition)
N2  X1.0  Z-1.
N3 X1.5  Z-1.5
N4 Z-2.
N5 G02  X.5  Z-2.5  R.5
N6 G1 X2.
G28
T202                                    (Finishing tool)
G97 S1500  M03
G70  P1  Q6                             (Finish pass)
G28
M30
%
```

G71 & G72 Type II Roughing Exercise



Fill in the blank line for tool #1 using a G71 Roughing Type II cycle command, to define Roughing passes for the part geometry defined between N10 and N20. Then define a finish pass using a G70 Finishing cycle command, with tool #2. Define a finish pass for the part geometry defined between N10 and N20. On the G71 command line leave .010 stock on diameters and .005 on the faces. Take a .120 depth of cut at .012 feed.

```
O0040                                   (Program number)
N1                                      (First operation)
T101 (O.D. Tool X .031 TNR)             (Tool #1 and Offset)
G50  S2500                              (Spindle speed clamp at 2500 RPM)
G97  S400  M03                          (Cancel CSS, define a 400 RPM, spindle
                                          ON forward)
G54  G____ X_____ Z_____ M08          (Rapid X & Z to starting location, coolant ON)
G____ S______                           (Turn on CSS to 375)
Z______                                 (Position .005 from end of part)
G_____ X______ F.005                    (Feed down to rough face end of part)
G_____ X______ Z______                  (Rapid to start position above part)
G71 P____ Q____ U____ W____ D____ F____ (Roughing O.D. G71 cycle command)
N____ G____ X______       Z_______      (Pnn starting number, Rapid X axis,
                                         Cutter comp ON)
G____ Z_____ F.006                      (G71 part geometry)
X______                                 (G71 part geometry)
G____ X_____ Z_____ R_____              (G71 part geometry)
G_____ Z_____                           (G71 part geometry)
X_____ Z_____ F.012                     (G71 Type II part geometry)
Z_____                                  (G71 Type II part geometry)
X_____ Z_____                           (G71 Type II part geometry)
Z_____                                  (G71 part geometry)
G_____ X______ Z______ R______          (G71 part geometry)
G_____ X______                          (G71 part geometry)
G_____ X______ Z______ R______          (G71 part geometry)
G_____ Z______ F.003                    (G71 part geometry)
X_____ F.015                            (G71 part geometry)
N_____ G_____ X_____                    (Qnnnn ending number, Cancel Cutter
                                        Comp)
G_____ S_____ M_____                    (Cancel CSS, define 400 RPM, spindle on
                                        (Forward, Coolant Off)
G_____                                  (Return to reference point)
M_____                                  (Optional stop command)

N2                                      (Second operation)
T_____                                  (Tool #2 and Offset #2)
G_____ S_____                           (Spindle speed clamp at 2500 RPM)
G_____ S_____ M____                     (Cancel CSS, turn on spindle 1200
G_____ X______ Z______
G_____ S______                          (Turn on CSS to 650)
                                        (Rapid to start position above part)
G_____ P______ Q______                  (Define finish pass using part geometry)
G_____ S______ M______                  (Cancel CSS, define 400 RPM, spindle
                                          On, coolant off)
G_____                                  (Return to reference point)
M_____                                  (End of program rewind)
```

G72 End Face Stock Removal Cycle

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

*	Indicates optional

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 define the shape of a part by programming the finished tool path and 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 are 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 3-12. 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 and second type of programmed path, the X-axis must be monotonic; 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, TYPE II roughing is assumed.

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.

Type I Details

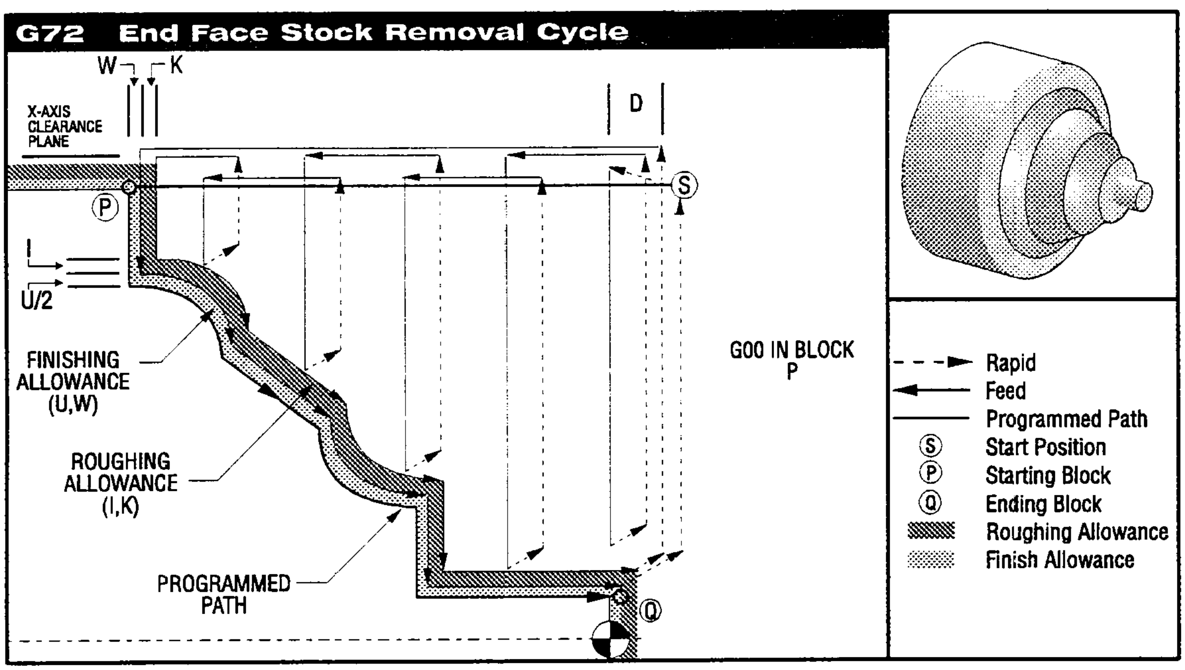
When Type I is specified by the programmer, it is assumed that the tool path is monotonic in the X-axis. Prior to any roughing motion, the tool path is checked for monotonicity and G code compliance. An alarm is generated if a problem is found.

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 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.

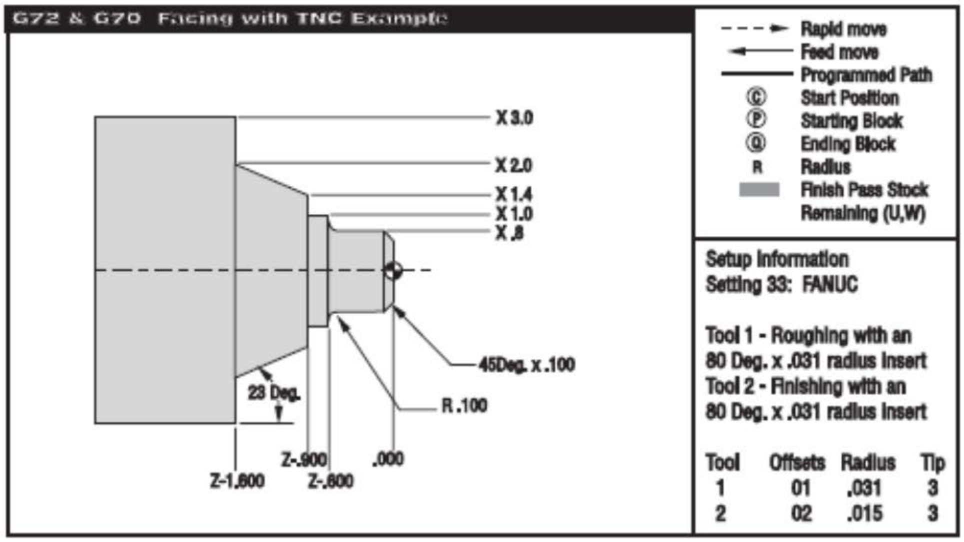
Each roughing pass is stopped before it intersects the programmed tool path allowing for both roughing and finishing allowances. The tool is then retracted from the material at a 45-degree angle by the distance specified in Setting 73. The tool then moves in rapid mode to the X-axis clearance plane.

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.



Example: TNC with G72

This example uses tool nose compensation with a G72 roughing canned cycle.



Part Drawing for TNC with G72 example

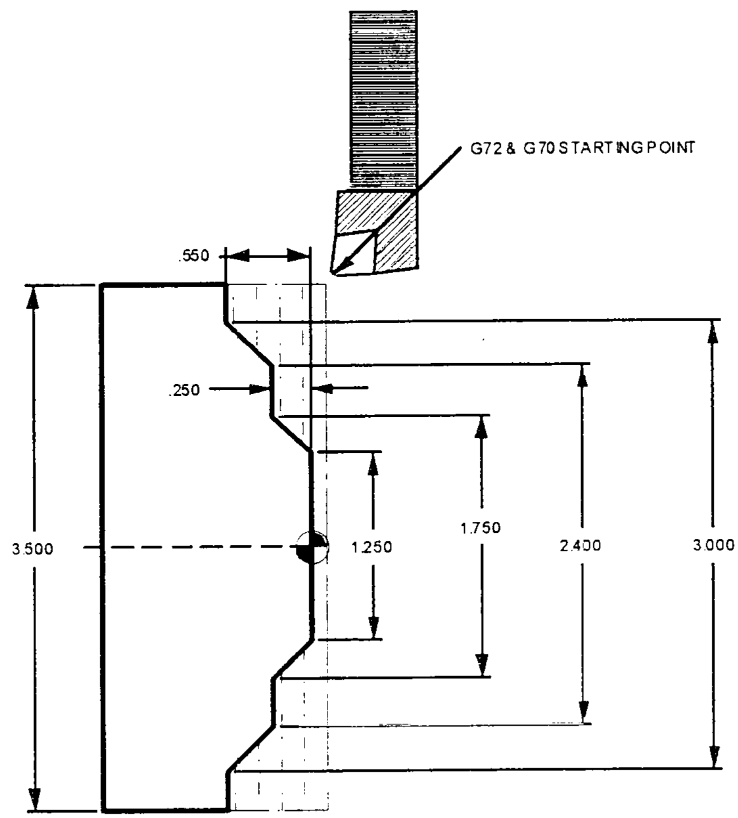
Program Example	Description
%	
O0814	
G50 S1000	
T0101	(Select Tool 1)
G0 X3.5 Z.1	(Move to point S)
G96 S100 M03	
G72 P80 Q180 U.005 W.01 D.05 F.010	(Rough P to Q with T1 using G72 and TNC)
(Define part path PQ sequence)	
N80 G41 G0 Z-1.6	(P)(G72 Type I, TNC Approach)
G1 X2.0 F0.005	
X1.4 Z-0.9	
X1.0	
Z-.60	
G3 X0.8 Z-0.5 K0.1	

Machining a chamfer, when cutter compensation is NOT used on the control, requires that calculations must be made for the tool tip dimensions.

G1 Z-0.1	
X0.6 Z0.0	
N180 X0.0	
G40 G0 X3.1 Z0.1 M5	(TNC departure)
(*****OPTIONAL FINISHING SEQUENCE*****)	
G28	(Zero for tool change clearance)
M01	
N2 G50 S1000	
T0202	(Select tool 2)
G0 X3.5 Z.1	(Move to point S)
G96 S100 M03	
(Finish P to Q with T2 using G70 and TNC)	
G70 P80 Q180	
G0 G40 Z.5 M5	(TNC departure)
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. Caution should be used however because of the direction of the force used in machining with face cutting. This is especially true with large parts held on with small jaw engagement and low jaw pressures because of part deformation. These parts may be thrown out of the chuck using face cutting canned cycle.

TYPE I G72 and G70 Exercise with Tool Nose Compensation



Fill in the blank lines for Tool #1 using a G72 roughing cycle command to define multiple rough facing passes for the part geometry defined between N10 and N20. Define tool #2 to do a finish pass using a G70 finishing cycle command. On the G72 command line, leave .010 stock on diameters and .005 on the faces. Take .12 depth of cut at .012 feed.

```
O0050                                       (Program number)
G28
T101 (O.D. Turning Tool .031R)              (Tool #1 and Offset #1)
G50  S2500                                  (Spindle speed clamp at 2500 RPM)
G97  S400  M03                              (Cancel CSS, turn on spindle 700 RPM)
G54  G00  X_____ Z_____ M08                 (Rapid, X, Z location, Coolant ON)
G____ S_____                                (Turn on CSS to 375)
Z_____                                      (Start position .050 from face of part)
G72  P___ Q___ U____ W____ D___ F___        (G72 Facing cycle command)
N___ G____ G____ Z______                    (Pnn starting number, Rapid to start, cutter
                                              comp ON)
G____ X______ F.006                         (G72 part geometry)
X______ Z______                             (G72 part geometry)
X______                                     (G72 part geometry)
X______ Z______                             (G72 part geometry)
X______                                     (G72 part geometry)
N____ G____ Z_____                          (G72 Qnnnn ending number, Cancel cutter
                                              comp.)
G____ S_____                                (Cancel constant surface speed)
G____                                       (Return to reference point)
M____                                       (Optional stop command)

T_____                                      (Tool #2 and Offset #2)
G____ S_____                                (Spindle speed clamp at 2500 RPM)
G____ S_____ M____                          (Cancel CSS, turn on spindle 1200 RPM)
G____ G____ X_____ Z_____ M____             (Rapid, X, Z location, Coolant ON)
G____ S_____                                (Turn on CSS to 650)
Z______                                     (Position to .050 from end of part)
G____ P_____ Q_____                         (Define finish pass using part geometry)
G____ S_____                                (Cancel constant surface speed)
G____ Z______ M____                         (Rapid 1.0 away from part in the Z-axis)
G____                                       (Return to reference point)
M____                                       (End of program rewind)
```

G73 Irregular Tool Path Stock Removal Cycle

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

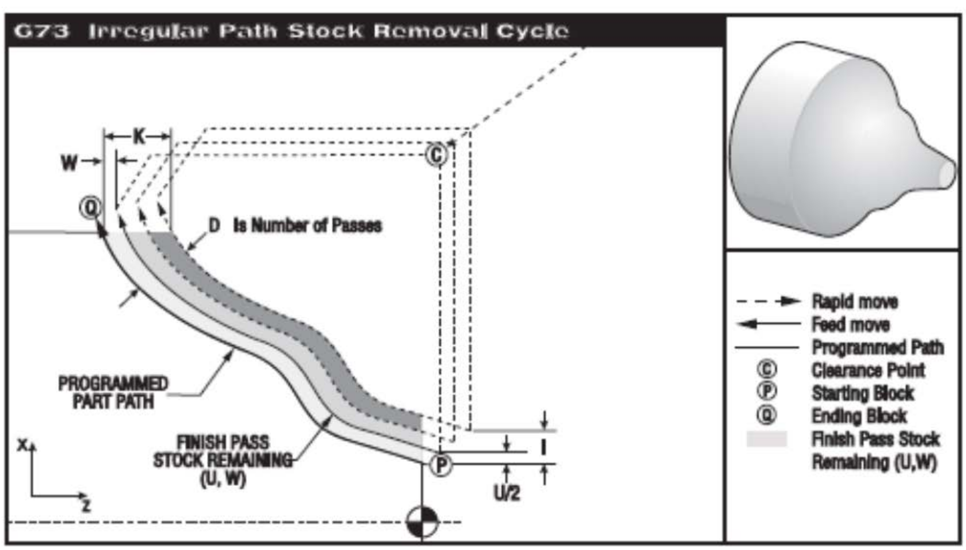
The G73 canned cycle can be used for rough cutting of castings or forgings. The canned cycle assumes that the extra material will follow the profile that is programmed between the tool path P through Q.

Machining either starts at the start position S and 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 roughing pass 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 programmed tool path P through Q does not have to be monotonic in X or Z, but care has to be taken to ensure that existing material does not interfere with tool movement during approach and departure moves.

This canned cycle is intended for use with the G70 finishing canned cycle.



EXAMPLE TNC WITH G73

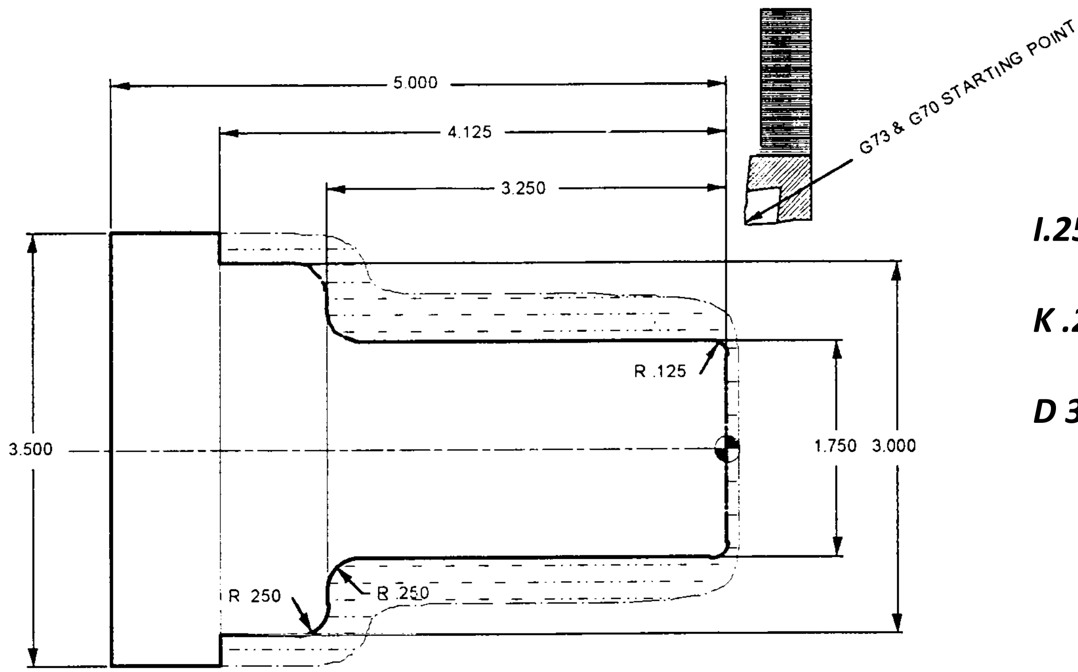
This example uses tool nose compensation with a G73 roughing canned cycle.

Preparation: Setting 33 FANUC

Tools:	T1	Insert with .064 radius, roughing		
	T2	Insert with .016 radius, finishing		
Offsets		Tool	Radius	Tip
1		T1	.064	3
2		T2	.016	3

Program Example	Description
%	
O00815	
G50 S1000	(Example section 8.10 example 5)
T101	(Select Tool 1)
G00 X3.5 Z.1	(Move to point S)
G96 S250 M03	
(Rough P to Q with T1 using G73 and TNC)	
G73 P80 Q180 U.01 W.005 I0.3 K0.15 D4 F.012	
(Define part path PQ sequence)	
N80 G42 G00 X0.6	(P)(G72 Type I, TNC Approach)
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	
N180 G40 G01 Z-2.1	(Q)
G00 X3.0 Z0.1 M05	(TNC departure)
(*****OPTIONAL FINISHING SEQUENCE*****)	
G28	(Zero for tool change clearance)
M01	
N2 G50 S1000	
T0202	(Select tool 2)
G00 X3.0 Z.1	(Move to point S)
G96 S250 M03	
(Finish P to Q with T2 using G70 and TNC)	
G70 P80 Q180	
G00 G40 Z.5 M05	(TNC departure)
G28	(Zero for tool change clearance)
M30	
%	

G73 and G70 Exercise with Tool Nose Compensation



I.25

K .25

D 3

Fill in the blank line for tool #1 using a G73 roughing cycle command defining roughing passes for the part geometry N10 and N20. Then define a finish pass using a G70 finishing cycle command, with tool #2, to define a finish pass for the part geometry defined between N10 and N20. On the G73 command line leave .010 stock on diameters and .005 on the faces. Take three passes at .012 feed.

```
O0060                                   (Program number)
N1                                      (First operation)
T101  (O.D. Tool X .031 TNR)            (Tool #1 and Offset #1)
G50  S2500                              (Spindle speed clamp at 2500 RPM)
G97  S400  M03                          (Cancel CSS, define a 400 RPM, spindle ON
                                        forward)
G54  G___ X_____ Z____ M08            (Rapid X & Z to starting location, coolant ON)
G____ S______                           (Turn on CSS to 375)
Z______                                 (Position .005 from end of part)
G____ X_____ F.005                      (Feed down to rough face End of part)
G____ X______ Z______                   (Rapid to start position above Part)
G73  P___ Q___ U____ W____ D____ F____ I_____ K_____      (Roughing O.D.  G73 cycle
                                                           command)
N___ G____ G____ X______                (Pnn starting number, Rapid X-axis,
                                                 cutter comp ON)
G____ Z_____ F.006                      (G73 part geometry)
X______                                 (G71 part geometry)
G____ X______ Z______ R_____            (G71 part geometry)
G_____ Z_______                         (G71 part geometry)
 G____ X______ Z______ R_____           (G71 part geometry)
G____ X______                           (G71 part geometry)
G____ X______ Z______ R______           (G71 part geometry)
G____ Z_____ F.003                      (G71 part geometry)
X______ F.015                           (G71 part geometry)
N____ G____ X_____                     (Qnnnn ending number, cancel Cutter comp)
G____ S_____ M____                      (Cancel CSS, define 400 RPM, spindle ON
                                        forward, Coolant Off)
G____                                   (Return to reference point)
M____                                   (Optional stop command)

N2                                      (Second operation)
T_____                                  (Tool #2 and offset #2)
G____ S_____                            (Spindle speed clamp at 2500 rpm)
G____ S_____ M____                      (Cancel CSS, turn on spindle 1200 RPM)
G54 G____ X_____ Z_____ M____           (Rapid, X, Z location, Turn coolant on)
G____ S_____                            (Turn on CSS to 650)
G____ P_____ Q_____                     (Define finish pass using part geometry)
G____ S_____ M____                      (Cancel CSS, define 400 RPM, spindle ON
                                        forward, Coolant Off)
G____                                   (Return to reference point)
M____                                   (End of program rewind)
```

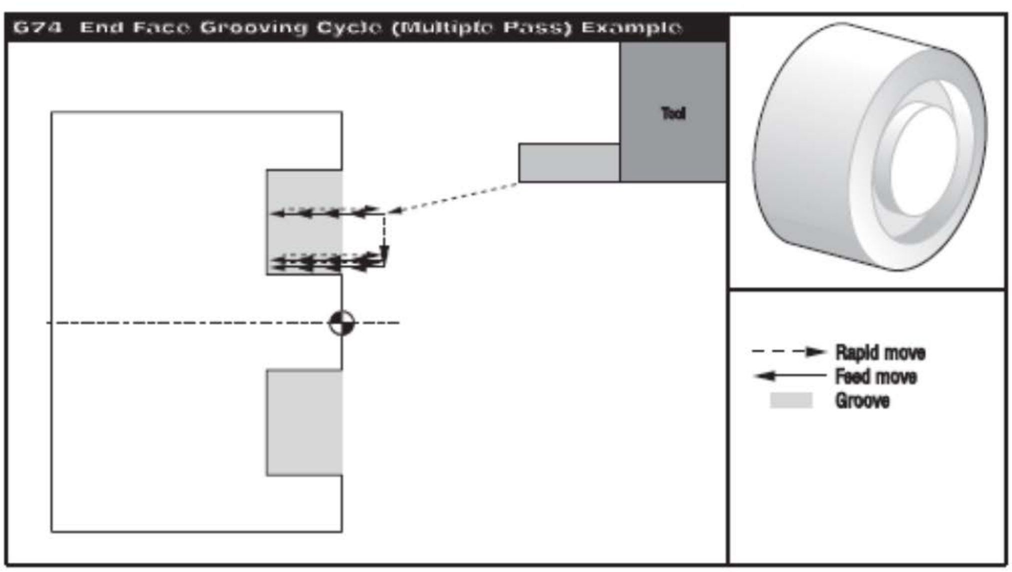
G74 End Face Grooving Cycle, Peck Drilling

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

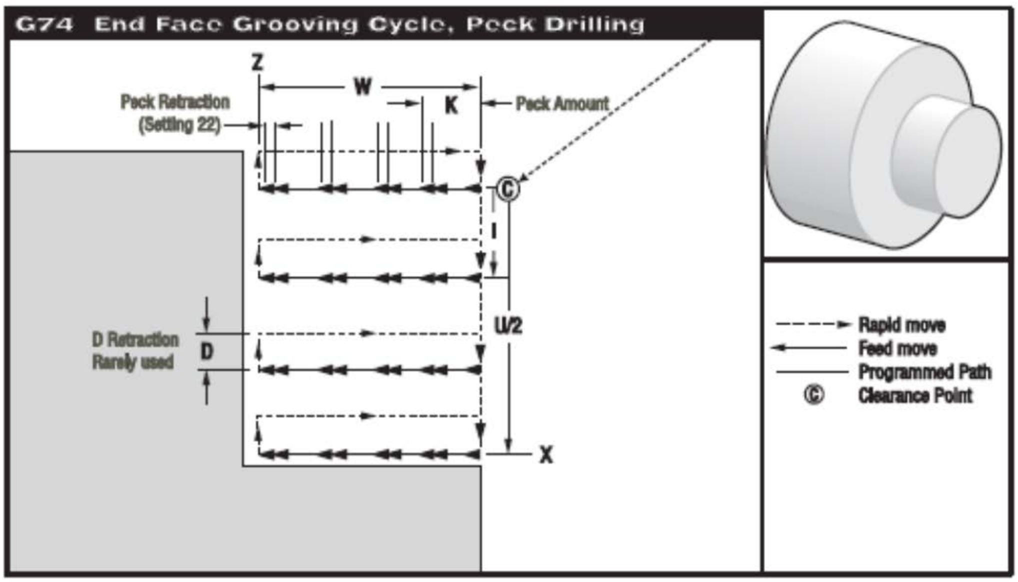
The G74 canned cycle can be used for grooving on the face of a part for peck drilling or for turning with a chip break. With this canned cycle, either a single pecking cycle can be executed, as for drilling on the spindle centerline, or a series of pecking cycles can be performed.

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.

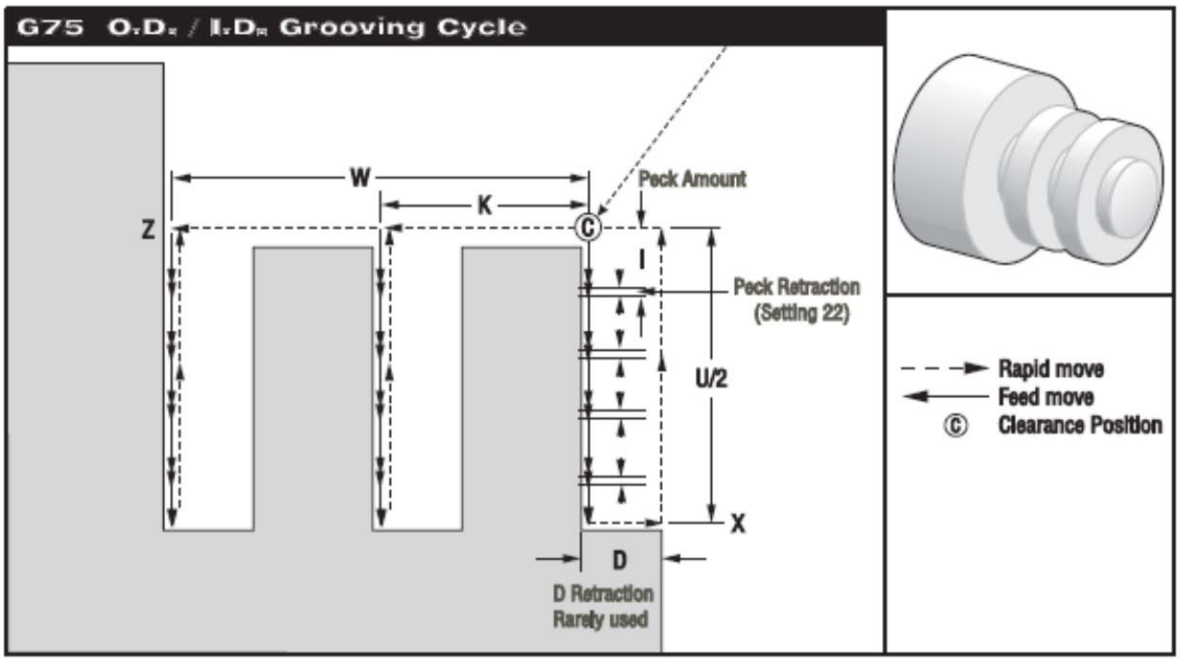


Program Example	Description
%	
O0074	
T101 M03 S750	
G00 X3. Z0.05	(Rapid to start position)
G74 I0.2 K0.1 X1.75 Z-0.5 F0.01	(Face grooving cycle multiple pass)
G28	
M30	
%	



G75 O.D./I.D. Grooving Cycle, Peck Drilling

Z	Z-axis absolute location to furthest peck cycle, signed
W	Z-axis incremental distance to furthest peck cycle, signed
*X	X-axis absolute location total pecking depth, signed diameter
*U	X-axis incremental distance to total pecking depth
*D	Tool clearance when returning to starting plane, positive
*I	X-axis size of increment b
*K	
*F	Feed Rate
*	Indicates optional



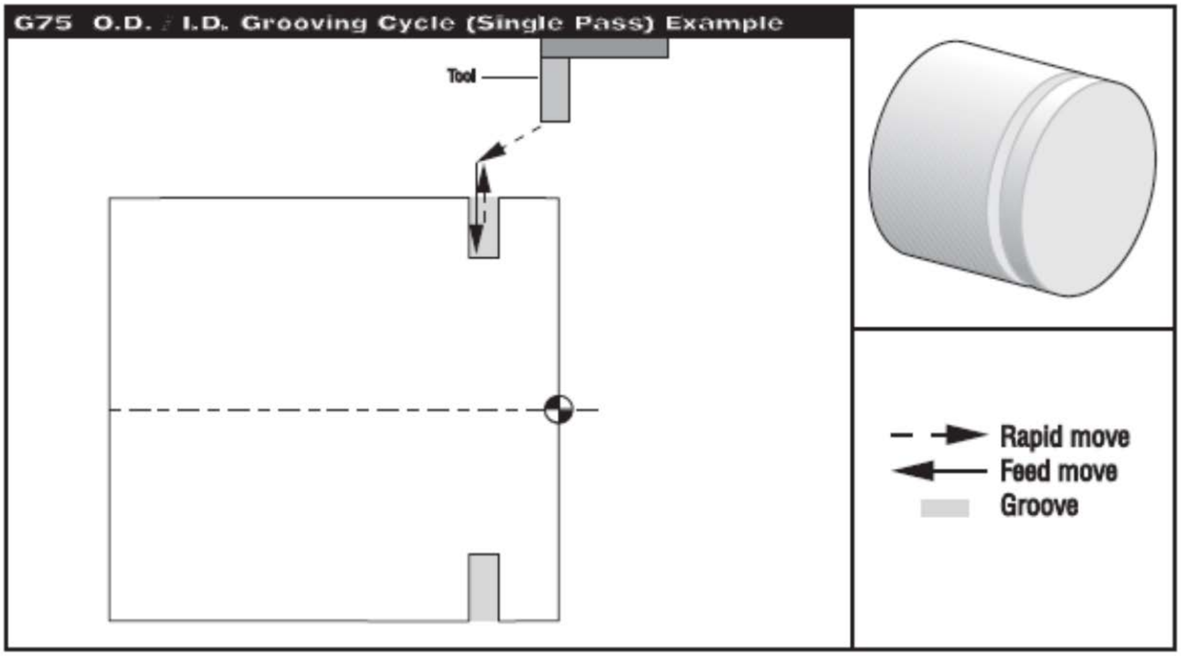
The G75 canned cycle can be used for grooving an outside diameter with a chip break. With this canned cycle, either a single pecking cycle can be executed (as for a single groove), or a series of pecking cycles can be performed (as for multiple grooves).

When adding a Z or W code 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**.

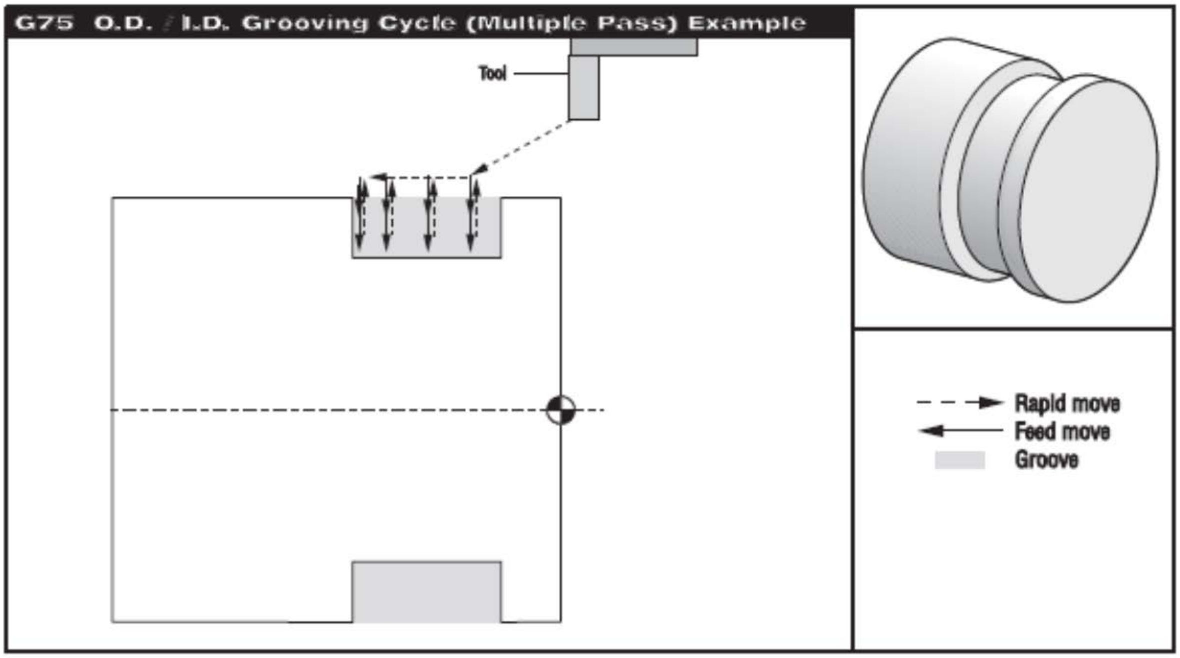
The D code can be used for grooving to provide material clearance when returning to starting plane S.

G75 Single Pass



O00109 (G75 O.D./I.D. SINGLE PASS GROOVE CYCLE)
(Machine a .25 wide O.D. Groove with .25 Groove Tool)
N1 G28
N2 T505 (.25 WIDE O.D. GROOVE TOOL)
N3 G97 S960 M03
N4 G54 G00 X2.1 Z0.1 M08 (Rapid to clearance point)
N5 Z-0.75 (Rapid to a start point of groove)
N6 G75 X1.75 I.05 F0.005 (G75 Single pass O.D. grooving cycle)
N7 M09
N8 G28
N9 M30

G75 Multiple Pass



(Machine a 1. wide O.D. Groove with .25 Groove Tool)

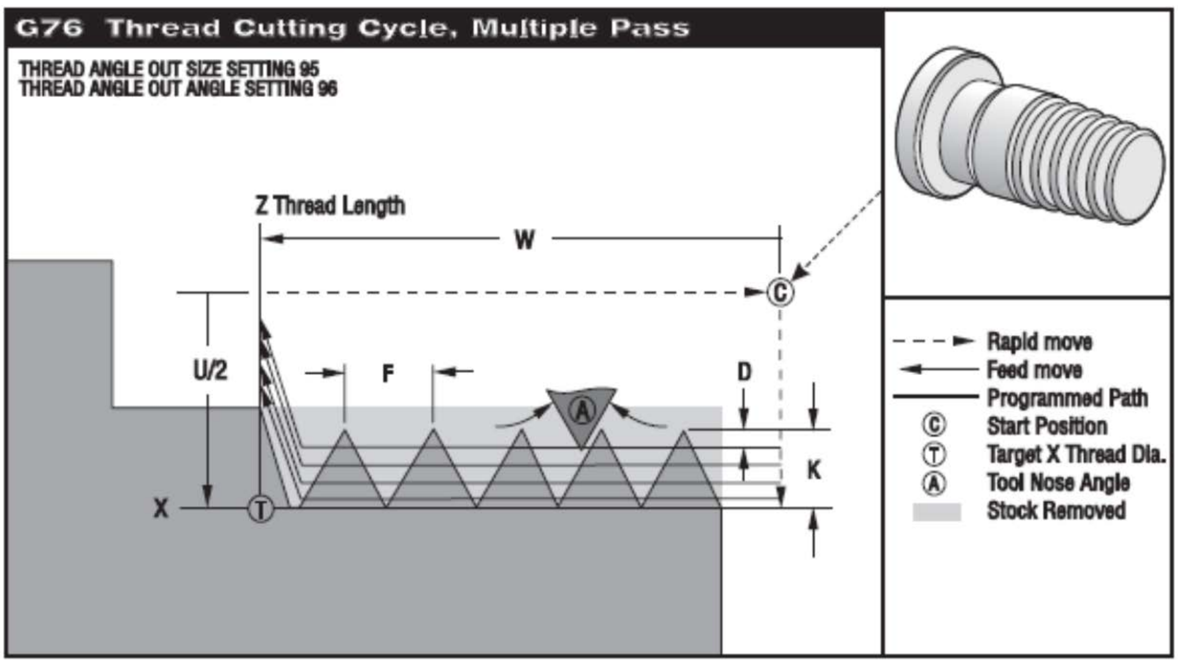
```
O00110 (G75  O.D./I.D.  MULTIPLE PASS GROOVING CYCLE)
N1 G28
N2 T505 (.25 WIDE O.D. GROOVE TOOL)
N3 G97 S960 M03
N4 G54 G00 X2.1 Z0.1 M08                (Rapid to front of part)
N5 Z-0.75                               (Rapid to start point of groove)
N6 G75 X1.75 Z-1.5 I0.05 K0.2 F0.005    (G75 Multiple pass O.D. grooving cycle)
N7 M09
N8 G28
N9 M30
```

SETTING 22 (CAN CYCLE DELTA Z) - *As the groove tool pecks deeper into the part, with each peck value of I, it pulls back a constant specified distance above the bottom of the groove created by the previous peck to break the chip. That specified distance it pulls back is defined in Setting 22.*

G76 Thread Cutting Cycle, Multiple Pass

G00 X1.1 Z0.3
G76 X0.913 Z-.85 K.0451 D.0136 A58 F0.0714 (1-14 THREAD Minor .9132)

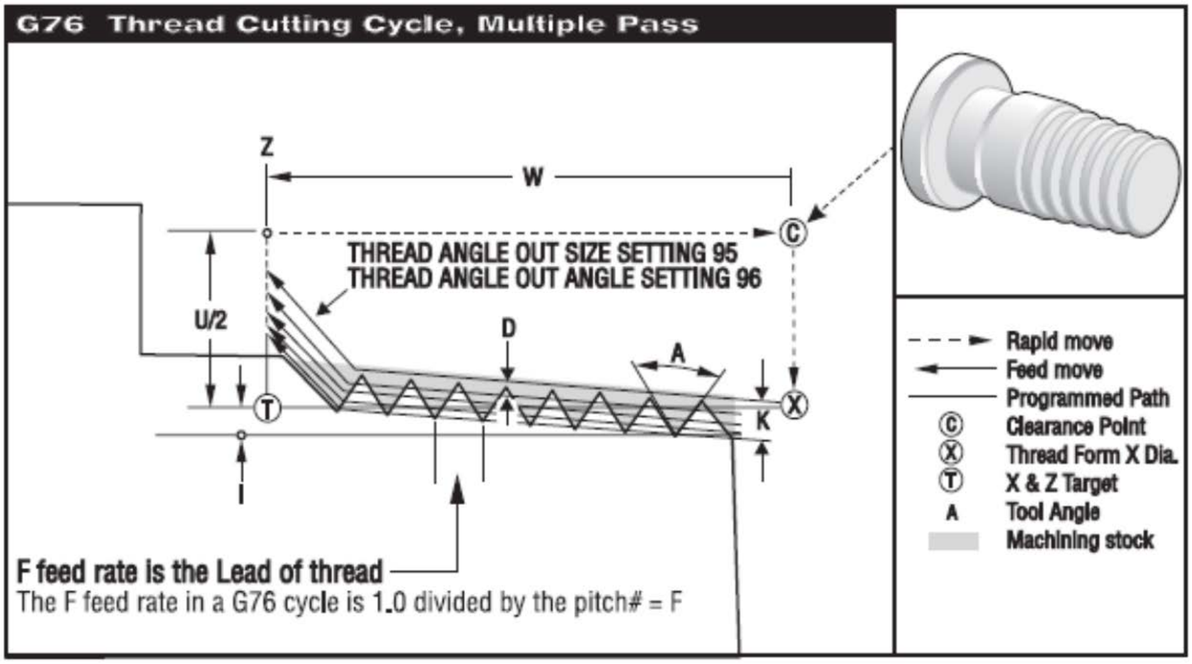
D	First pass cutting depth
K	Thread height, defines limit of multiple passes, radius measure
*X	X-axis absolute location, maximum thread I.D.
*Z	Z-axis absolute location, maximum thread length
*U	X-axis incremental distance, start to maximum thread I.D.
*W	Z-axis incremental distance, start to maximum thread length
*I	Thread taper amount, radius measure
*P	Subsequent pass positioning algorithm, cutting method
*F(E)	Feedrate (specifies feedrate for threading – also indicates the pitch or lead)
*A	Tool nose angle (from 0 to 120 degrees – if not used, then defaults to 0)
*	Indicates Optional



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**. The height of the thread is defined as the distance from the crest to the root of the thread. K must agree with the direction that the X-axis is being cut. The actual depth of cut 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 the beginning **position C**. A **conventional O.D. taper thread** will have a **negative I value**.

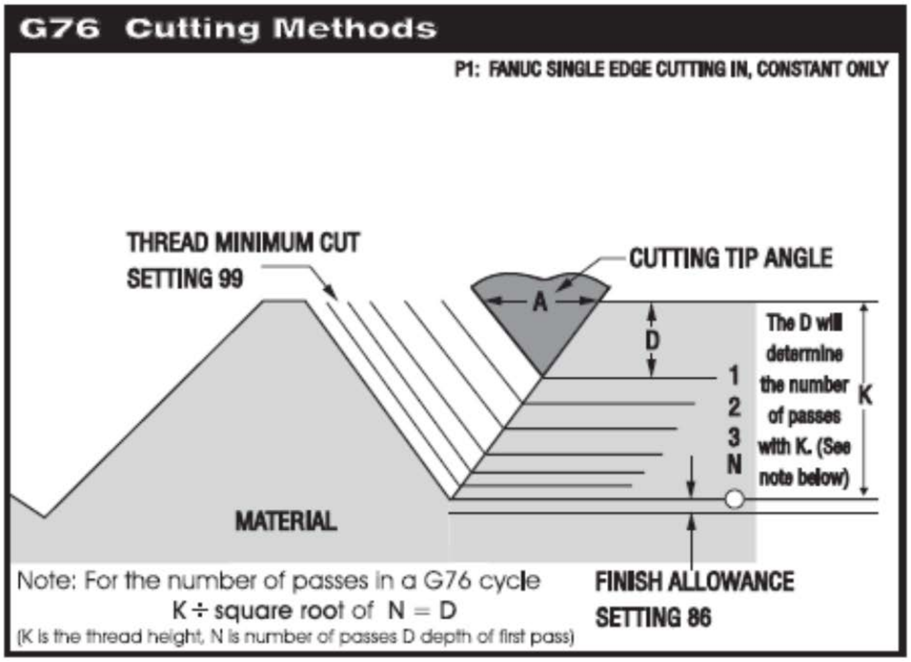


The **depth of the first cut** through the thread is specified in **D**, which also determines the number of passes over the thread based on the value of K and the cutting method used.

The depth of the last cut through the thread can be controlled with **Setting 99** (THREAD MINIMUM CUT). 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. On most carbon and alloy steels the value of minimum pass of .005 will produce acceptable finishes. For some materials .001 minimum pass will improve the surface finish of the thread.

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 feedrate 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. M23 commands chamfer on. M24 commands chamfer off. M23 is the default value.

The **tool nose angle** for the thread is **A**. The value can run from 0 to 120. If no A is given 0 degrees is assumed. If no A value is given then the thread is created with a radial cut. With a radial cut both sides of the threading tool are cutting. This creates a v shaped chip which may be hard to evacuate. This may cause the insert to chip. Also chattering may occur as the entire tool is engaged at the end of the thread cycle.



If **A** has a **60** degree value, the same as the thread cutting tool, the Haas machine will make a flank cut. In a flank cut just the leading edge of the tool will do the cutting. Each successive pass will in feed the tool at 30 degrees as shown above. The advantage of using a flank cut is that the chip will flow out of thread form area easier than a radial cut. This also reduces a burr from forming on the trailing edge of the tool. To avoid rubbing on the trailing edge of the tool a modified flank cut is recommended. With a modified flank cut the value of **A** is **58** (In feed angle 29 degrees). A channel chip is formed but because of uneven thickness on the chip flows out similar to a flank cut. The disadvantage to flank or modified flank cutting is torn or poor finish may result when soft or gummy materials are cut such as low carbon steels, aluminum and stainless steels.

P Options:

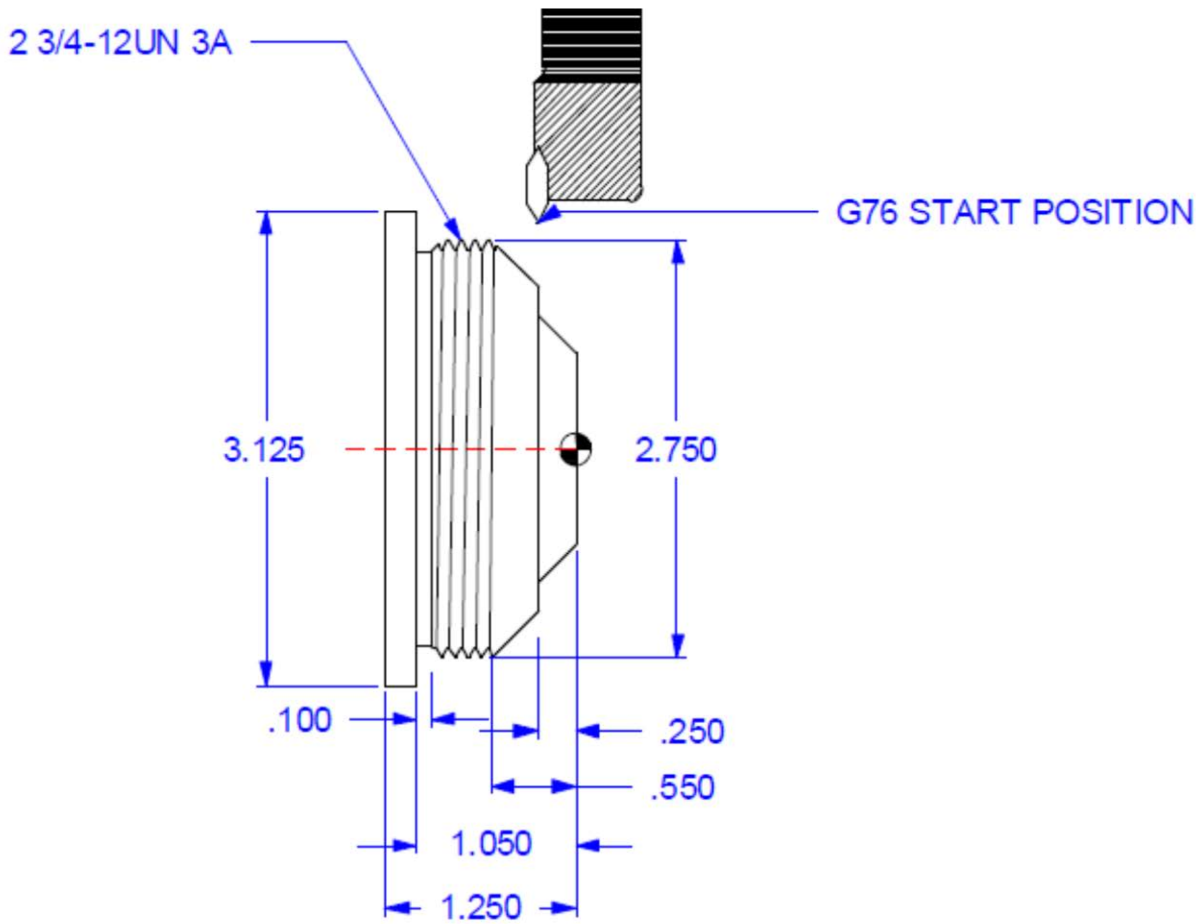
- **P1:** Single cutting, cutting amount constant (default)
- **P2:** Double edge cutting, cutting amount constant
- **P3:** Single edge cutting, cutting depth constant
- **P4**: Double edge cutting, cutting depth constant

Alternating flank cutting is possible with P2 and P4. Alternating flank cutting will increase tool life as both sides of the inserts are being used equally. Option **P2** may give superior results according to Haas.

Example of G76 code for a 1-8 Thread:

```
T404
S50 S2000
G97 S1200 M03
G00 X1.2 Z.3 M08
G76 X.8492 Z-1.50 K.0676 D.0169 A58 P2 F.125
G00 G28 U0
G28 W0.
M30
```

Thread Programming Exercise



2 ¾-12UN 3A Thread

Major Diameter 2.7500/2.7386 Minor Diameter 2.6478

O0010
N1 G28
N2 T101 (Thread Tool)
N3 G97 S590 M03
N4 G54 G00 X________Z________M08 (Rapid X and Z above part)
N5 Z_________ M____ (Z-.25 is start of thread, M code for no chamfer)
N6 G76 X_______ Z_______ K_____D_____ A____ P__ F____
(12 passes, alternating flank, in feed cut 29 degrees)
N7M_____ (Turn Coolant off)
N8 G_____ U______ (Take Home in X)
N9 G_____ W______ (Take Home in Z)
N10 M____ (End of Program)

Drilling, Boring and Tapping Canned Cycles

These canned cycles are used to define and simplify programming for the most common Z-axis repetitive operations. Once selected, a canned cycle is active until canceled with a G80 code. There are six operations involved in every canned cycle:

1) Positioning of 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.
6) Rapid traverse to the initial starting point.

Modal canned cycles remain in effect after they are defined and executed for each positioning of X axes in the program. Some of the canned cycle command values can also be changed after the canned cycle is defined. The commands most often changed during a canned cycle are the R plane value and the Z depth value. Modal canned cycles will be canceled with the G80, G01 or G00 command. Positioning moves during a canned cycle is performed as a rapid motion.

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

A canned cycle will only be executed in the Z-axis when positioning to a new X-axis position during a canned cycle command.

Canned Cycles for Drilling and Tapping

The following is a list of the canned cycles that can be used for drilling and tapping for the HAAS lathe controls.

Canned Cycles

G80 Canned Cycle Cancel
G81 Drill Canned Cycle
G82 Spot Drill Canned Cycle
G83 Peck Drill Canned Cycle
G84 Tapping Canned Cycle
G85 Bore in Bore out Canned Cycle
G86 Bore in Rapid out Canned Cycle
G87 Bore with Manual Retract Canned Cycle
G88 Bore in Dwell with Manual Retract Canned Cycle
G89 Bore in Dwell Bore out Canned Cycle

G Code	Z Drilling Direction	Operation at Bottom of Hole	Retraction Z Direction	Application
G81	Feed	None	Rapid	Spot drilling
G82	Feed	Dwell	Rapid	Counter boring
G83	Intermittent feed	None	Rapid	Peck drilling
G84	Feed	Spindle CCW	Feed	Tapping cycle
G85	Feed	None	Feed	Boring cycle
G86	Feed	Spindle stop	Rapid	Boring cycle
G87	Feed	Spindle stop	Manual/rapid	Back cycle
G88	Feed	Dwell, then spindle stop	Manual/rapid	Boring cycle
G89	Feed	dwell	feed	Boring cycle

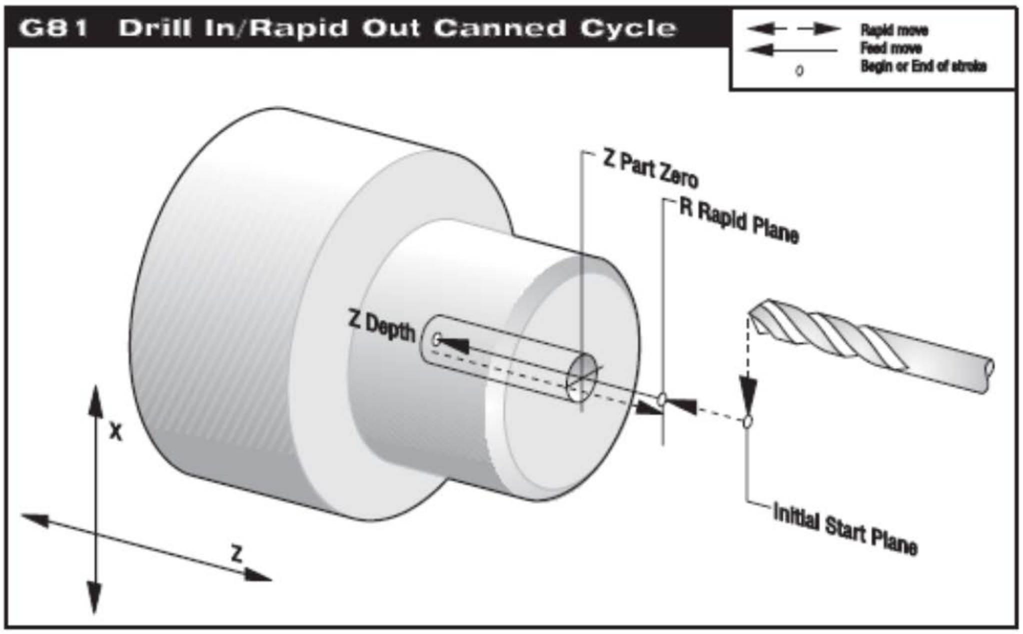
A canned cycle is presently limited to operations in the Z-axis. That is, only the G18 plane is allowed. This means that the canned cycle will be executed in the Z-axis whenever a new position is selected in the X-axis. The following is a summary of the canned cycles used on HAAS lathe controls.

G80 Canned Cycle Cancel

The G code is modal; deactivating 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

X	Optional X-axis motion command
Z	Position of bottom of hole
R	Position of the R plane
F	Feed Rate

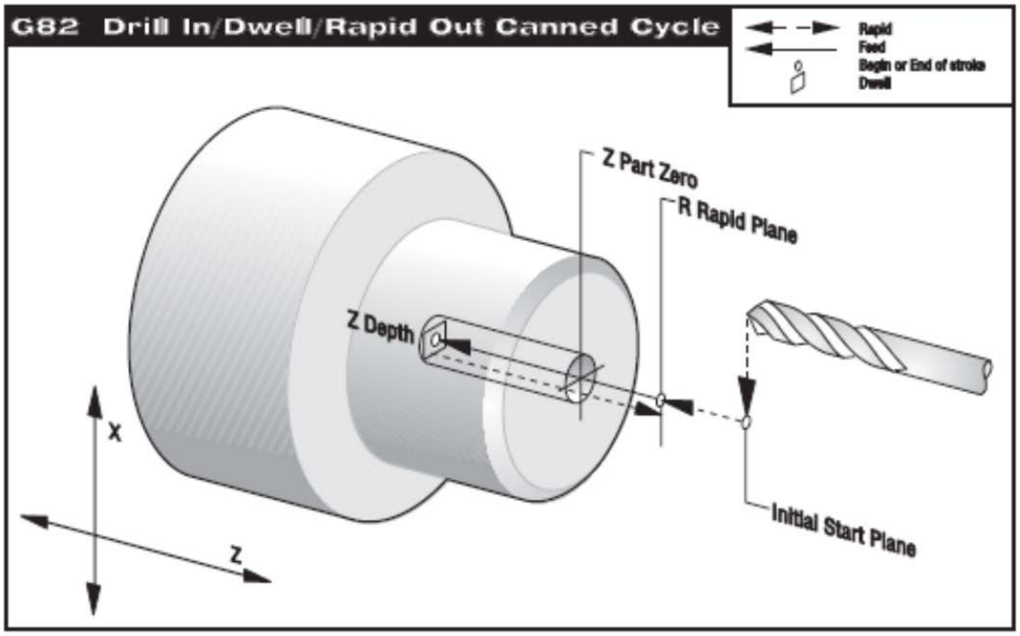


TOOL DESCRIPTION	TOOL	OFFSET	RADIUS	TIP
1/2 DIA. DRILL	1	01	0	0

```
O00119  (G81 Drilling)
N1 G28
N2 T101 (1/2 DIA. DRILL)         (Tool 1 Offset 1)
N3 G97 S1450 M03
N4 G54 G00 X0. Z1. M08           (Rapid to Initial Start Point)
N5 G81 Z-0.625 R0.1 F0.005       (G81 Drilling Cycle)
N6 G80 G00 Z1.0 M09
N7 G28
N8 M30
```

G82 Spot Drill Counter Bore Canned Cycle

X	Optional X-axis motion command
Z	Position of bottom of hole
P	The dwell time at the bottom of the hole
R	Position of the R plane
F	Feed Rate



TOOL DESCRIPTION	TOOL	OFFSET	RADIUS	TIP
1/2 DIA. FLAT BOTTOM DRILL	2	02	0	0

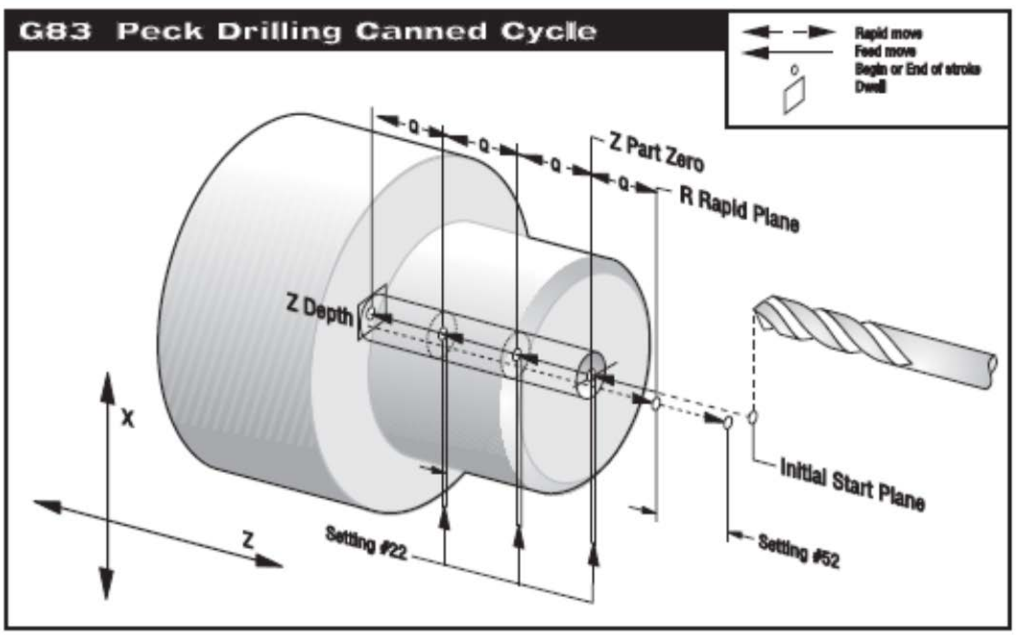
```
O00120 (G82 Drilling with a Dwell)
N1 G28
N2 T202 (1/2 DIA. FLAT BOTTOM DRILL)   (Tool 2 Offset 2)
N3 G97 S1450 M03
N4 G54 G00 X0. Z1. M08              (Rapid to Initial Start Point)
N5 G82 Z-0.625 P0.5 R0.1 F0.005   (G82 Drill with a Dwell at Z Depth Cycle)
N6 G80 G00 Z1.0 M09
N7 G28
N8 M30
```

G83 Deep Hole Peck Drilling Canned Cycle

X	Optional X-axis motion command
Z	Position of bottom of hole
I	Optional size of first cutting depth
J	Optional amount to reduce cutting depth each pass
K	Optional minimum depth of cut
Q	The cut-in value, always incremental
R	Position of the R plane
F	Feedrate

If I, J, and K are specified, a different operating mode is selected. The first pass will cut in by I. J will reduce each succeeding cut and K is the minimum cutting depth.

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 ensure that the chip clear motion actually allows the chips to get out of the hole. 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.




```
N4 G54 G00 X0. Z1. M08          (Rapid to Initial Start Point)
N5 G83 Z-1.5 Q0.2 R0.1 F0.005   (G83 Peck Drilling Cycle with Q)
```

The operation of this cycle is different if the rigid tapping option is installed. When rigid tapping is used, the ration between the feed rate and spindle speed must be precisely the thread pitch being cut.

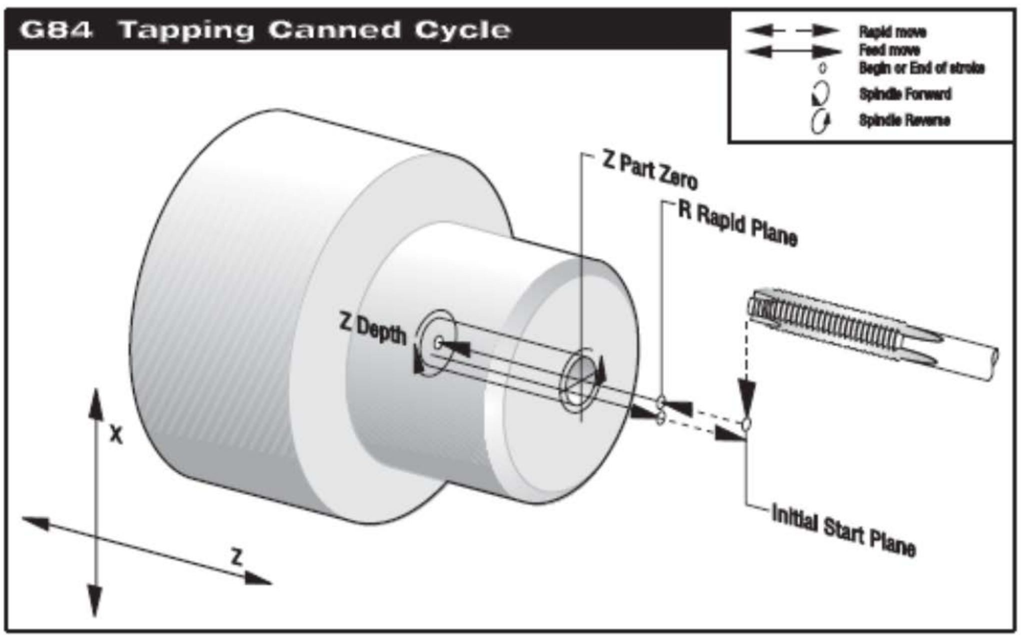
You do not need to start the spindle CW before this canned cycle. The control does this automatically.

G84 Tapping Canned Cycle

X	Optional X-axis motion command
Z	Position of bottom of hole
R	Position of the R plane
F	Feed rate

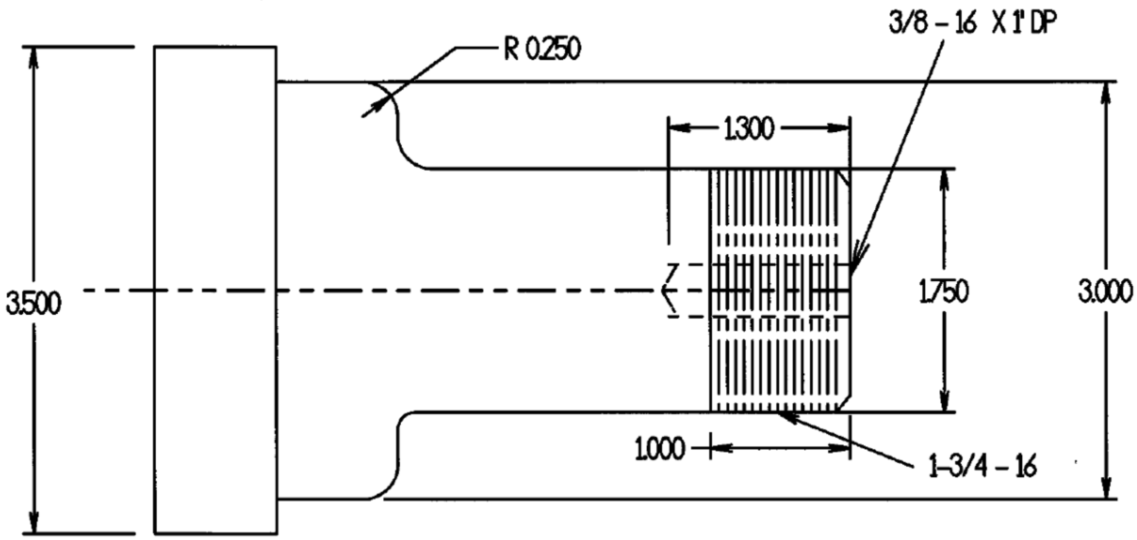
Feed rate is the **Lead** of the thread. Feed Rate = 1/ Threads per inch

G184 Reverse Tapping Canned Cycle
(the same as a G84, but is used for left handed taps)

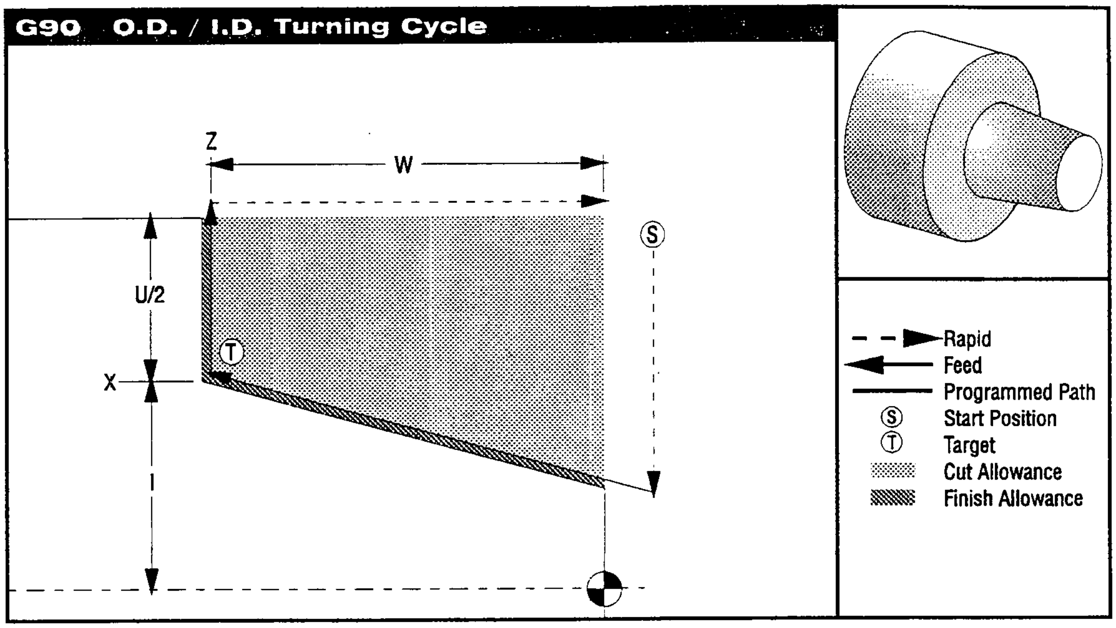



```
N2 T404 (3/8-16 TAP)          (Tool 4 Offset 4)
N3 G97 S650 M05               (G84 will turn on the spindle for you)
N4 G54 G00 X0. Z1. M08     (Rapid to Initial Start Point)
N5 G84 Z-0.5 R0.2 F0.0625    (G84 Tapping Cycle)
```

Turn Threads, Drill and Tap Exercise



Turn Thread 1-3/4 -16 UN 2A .6” back from face at a sfm=300
Major diameter 1.748, Minor diameter 1.6739
Drill and Tap 3/8-16 x 1.0” deep
Drill high speed 5/16 drill 1.3 deep F.006”/rev at sfm=80
Peck drill at depth of the diameter of the drill
Tap at 200 rpm
O0096
(OD THREAD)
G___ S_____ Clamp speed at 2000
G____ Take Home
T404
G___ S_____ M___ Start Spindle
G___ G___ X_____ Z___ M___ Start Position, Turn on coolant
G___ X______ Z-_____ K______ D_____ A___ F_____ Thread cycle
M09
G28
M01
T1111 (5/16 DRILL)
G___ S____ M___ Start spindle
G___ G___ X___ Z___ M___ Rapid to start position, Turn on coolant
G___ X___ Z____R____ Q_____ F____ Drill Peck cycle
G___ Cancel Canned cycle
G00 Z0.2
G28
M09
M01
T1010 (3/8-16 TAP)
G97 S____ M___ Start spindle
G54 G____ X__ Z____ M08 Rapid to start position, turn on coolant
G____ X__ Z-____ R0.__ F0.0____ Tap cycle
G80
G28
M09
M30



G90 O.D./I.D. Turning Cycle *Group 01*

X	X-axis absolute location of target
Z	Z-axis absolute location of target
*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
F (E)	Feedrate

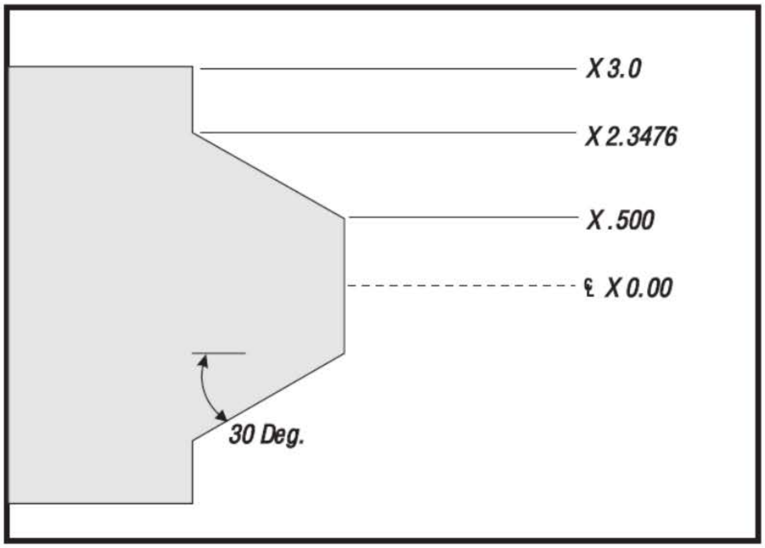
* indicates optional

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 specifying the X locations of successive passes.

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

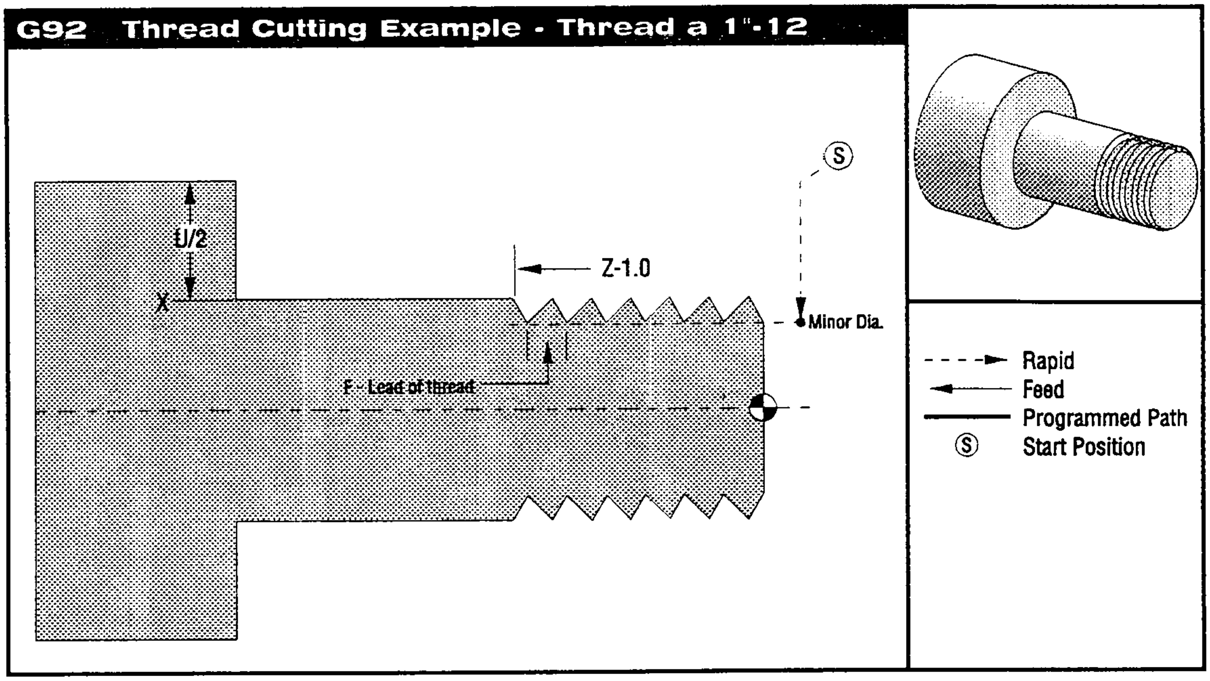
Any of the four ZX quadrants can be programmed by varying U, W, X, and Z. The taper can be positive or negative. Selecting the sign direction is not intuitive.

EXAMPLE: G90 MODAL TURNING CYCLE WITH TNC



This example uses tool nose compensation with a G90 modal rough turning cycle.

```
O00131 (G90 Modal Turning with TNC)
N11 G28
N12 T101  (O.D. TURNING TOOL)
N13 G50 S3000
N14 G97 S480 M03
N15 G54 G00 X3.1 Z1. MO8   (Rapid to Start Point)
N16 G96 S390
N17 Z0.1
N18 G90 G42 X2.8 Z-1.6001 I-0.9238 F0.01 (Rough 30 Deg. angle to X2.3476
N19 X2.65            (Additional Pass)          Dia. using G90 and TNC)
N20 X2.55            (Additional Pass)
N21 X2.45            (Additional Pass)
N22 X2.355           (Additional Pass)
N23 X2.3476          (Additional Pass)
N24 G00 G40 X3.1 Z1. M09       (TNC Departure)
N25 M05
N26 G28
N27 M30
```



G92 Thread Cutting Cycle *Group 01*

X	X-axis absolute location of target
Z	Z-axis absolute location of target
*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
F (E)	Feedrate, the lead of the thread
*	indicates optional

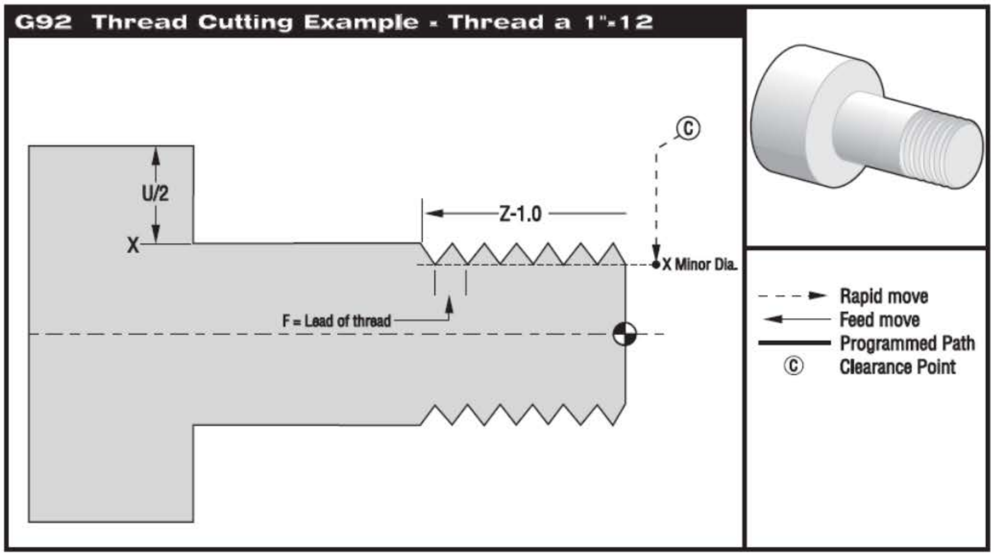
G92 is a modal canned cycle. It can be used for simple threading. Since it is modal, you can do multiple passes for threading by specifying the X locations of successive passes.

Straight threads can be made by 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; I is added to the value of X at the target.

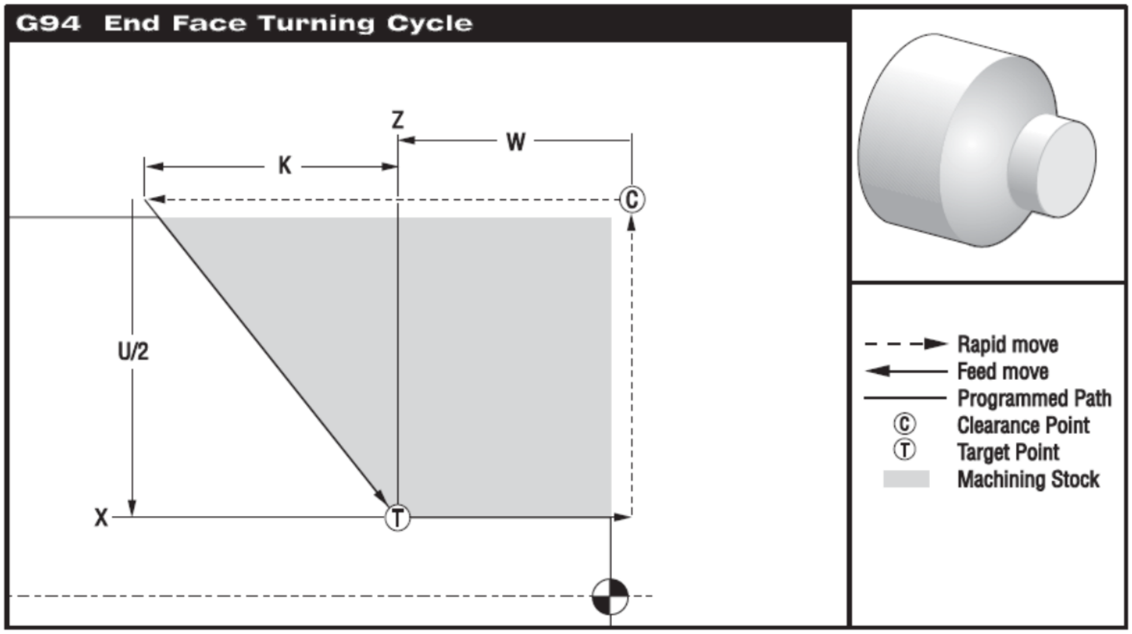
At the end of the thread, an automatic chamfer is executed before reaching the target default. This chamfer is one thread at 45 degrees. These values can be changed with Setting 95 and Setting 96.

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. The figure shows a few examples of the values required for machining in each of the four quadrants.

EXAMPLE: G92 MODAL THREADING CYCLE




```
O00133 (G92 Modal Threading)
N10 (1.0-12UN Thread)
N11 G28
N12 T404  (O.D. THREADING TOOL)
N13 G97 S825 M03
N14 G54 G00 X1.1 Z1. M08 (Rapid to Start Point)
N15 Z0.25
N16 G92 X.98 Z-1.05 F0.08333 M23   (First Pass of a G92 O.D. Thread Cycle)
N17 X.96                (Additional Pass)
N18 X.94                (Additional Pass)
N19 X.935               (Additional Pass)
N20 X.93                (Additional Pass)
N21 X.925               (Additional Pass)
N22 X.9225              (Additional Pass)
N23 X.92                (Additional Pass)
N24 X.9175              (Additional Pass)
N25 X.9155              (Additional Pass)
N26 X.915               (Additional Pass)
N27 X.9148              (Additional Pass)
N28 G00 X1.1 Z1. M09
N29 M05
N30 G28
N31 M30
```



G94 End Face Cutting Cycle *Group 01*

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

* indicates optional

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 specifying the Z locations of successive passes.

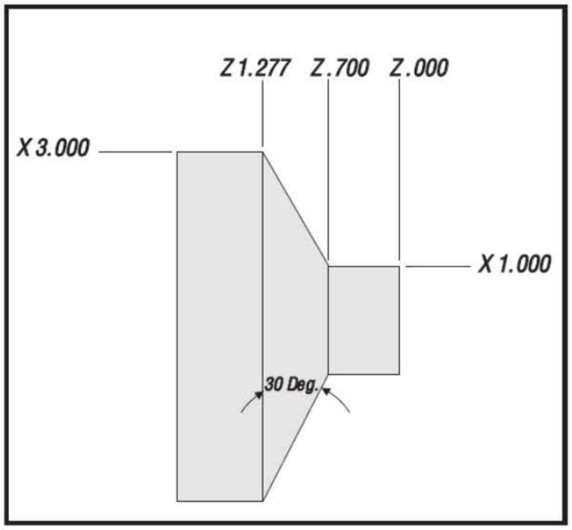
Straight end facing cuts can be made by specifying X, Z, and F. By adding K, a conical face can be cut. The amount of coning is referenced from the target. K is added to the value of X at the target.

Any of the four ZX quadrants can be programmed by varying U, W, X, and Z. The coning can be positive or negative. Selecting the sign direction is not intuitive.

EXAMPLE 7 TNC WITH G94

This example uses tool nose compensation with a G94 modal, rough turning cycle.

EXAMPLE: G94 MODAL END FACING WITH TNC



This example uses tool nose compensation with a G94 modal rough facing cycle.

```
O00135  (G94 Modal End Facing with TNC example)
N11 G28
N12 T101  (O.D. FACING TOOL)
N13 G50 S3000
N14 G97 S480 M03
N15 G54 G00 X3.1 Z1. M08        (Rapid above part)
N16 G96 S390
N17 Z.1  (rapid to start point)
N18 G94 G41 X1.0 Z-0.3 K-0.5774 F0.01      (Rough 30 Deg. angle to  X1. and
N19 Z-0.4                 (Additional Pass)          Z-0.7 using G94 and TNC)
N20 Z-0.5                 (Additional Pass)
N21 Z-0.6                 (Additional Pass)
N22 Z-0.69                (Additional Pass)
N23 Z-0.7                 (Additional Pass)
N24 G40 G00 X3.1 Z1. M09        (Cancel TNC)
N25 M05
N26 G28
N27 M30
```

M Code Detailed Description

M00 STOP PROGRAM

The M00 code is used to stop a program. It also stops the spindle, 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.

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.

M02 PROGRAM END

The M02 code will stop program operation the same as the 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 31 of parameter 209 (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 31 of a parameter 209 9CNCR 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 stops the spindle. The block is delayed until the spindle slows below 10 RPM.

M08 COOLANT ON

The M08 code will turn on the coolant. 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.

M09 COOLANT OFF

The M09 code will turn off the coolant.

M10 CLAMP CHUCK

The M10 code is used to clamp the chuck. It is only used when M11 is used to unclamp the chuck.

M11 UNCLAMP CHUCK

The M11 code will unclamp the chuck.

M17 TURRET ROTATION ALWAYS FORWARD

The M17 code is modal and forces the turret to rotate in the forward direction when a tool change is made. Setting 97 TOOL CHANGE DIRECTION needs to be switched from SHORTEST to M17/M18.

T1010 M17:
This command will advance the tool turret in a FORWARD direction to position #10. Because M17 is modal, any subsequent T command will cause the turret to rotate in the forward direction to the commanded tool.

M18 TURRET ROTATION ALWAYS REVERSE

The M18 code is modal and forces the turret to rotate in the forward direction when a tool change is made. Setting 97 TOOL CHANGE DIRECTION needs to be switched from SHORTEST to M17/M18.

T1010 M18:
This command will advance the tool turret in a REVERSE direction to position #10. Because M18 is modal, any subsequent T command will cause the turret to rotate in the reverse direction to the commanded tool.

M21-M22 OPTIONAL USER M

The M21 through M22 codes are optional for user interfaces.

M25-M28 OPTIONAL USER M

The M25 through M28 codes are optional for user interfaces.

M30 PROG END AND REWIND

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 is also incremented. M30 will also cancel tool length offsets.

M31 CHIP CONVEYOR FORWARD

M31 starts the chip conveyor motor in the forward direction.

M32 CHIP CONVEYOR REVERSE

M32 starts the chip conveyor motor in the reverse direction.

M33 CHIP CONVEYOR STOP

M33 stops chip conveyor motion.

M51- M58 OPTIONAL USER M ON

The M51 through M58 codes are optional for user interfaces.

M61- M68 OPTIONAL USER M OFF

The M61 through M68 codes are optional for user interfaces.

M75 SET G35 OR G136 REFERENCE POINT

M76 DISABLE DISPLAYS

M77 ENABLE DISPLAYS

M78 ALARM IF SKIP SIGNAL FOUND

M79 ALARM IF SKIP SIGNAL NOT FOUND

M85 OPEN AUTOMATIC DOOR

M86 CLOSE AUTOMATIC DOOR

M97 LOCAL SUB-PROGRAM CALL

This code is used to call a subroutine referenced by a line N number within the same program. 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 end 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.

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.