HAAS FACTORY OUTLET
A DIVISION OF PRODUCTIVITY INC

HAAS LATHE PROGRAMMING

Revised 12/2024 JP

PROGRAMMING MANUAL

Productivity Inc®

This Manual is the Property of Productivity Inc

The document may not be reproduced without the express written permission of Productivity Inc, nor may it be sold. The content must not be altered, nor may the Productivity Inc name be removed from the materials. This material is to be used as a guide to the operation of the machine tool. The Operator is responsible for following Safety Procedures as outlined by their instructor or manufacturer's specifications.

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

To obtain permission, please contact trainingmn@productivity.com.

For more information on
Additional Training Opportunities or our Classroom Schedules
Contact the Productivity Inc Applications Department
Minneapolis: ☏ 763.476.8600
Cedar Rapids: ☏ 319.734.3403
Omaha: ☏ 402.330.2323
Denver: ☏ 800-947-8665
Salt Lake City: ☏ 801.886.2221
Visit us on the Web: www.productivity.com

HAAS Interactive Lathe Operator Manual

https://www.haascnc.com/service/online-operator-s-manuals/lathe-operator-s-manual/lathe---introduction.html

Note: Some of the content, images and screen shots included in this manual are taken from Haas manuals, controllers and web information with permission from Haas Automation Inc.
2800 Sturgis Road Oxnard CA 93030-8933

CONTENTS

TOOL AND WORK OFFSETS

WORK OFFSETS

G54 - G59 codes are user-settable coordinate systems, #1 - #6, for work offsets. All subsequent references to axes positions are interpreted in the new coordinate system. Work coordinate system offsets are entered from the Active Work Offset display page.

G154 Select Work Coordinates **P1-P99 (Group 12)**
This feature provides 99 additional work offsets. *G154* with a *P* value from 1 to 99 activates additional work offsets. For example, *G154 P10* selects work offset 10 from the list of additional work offsets.

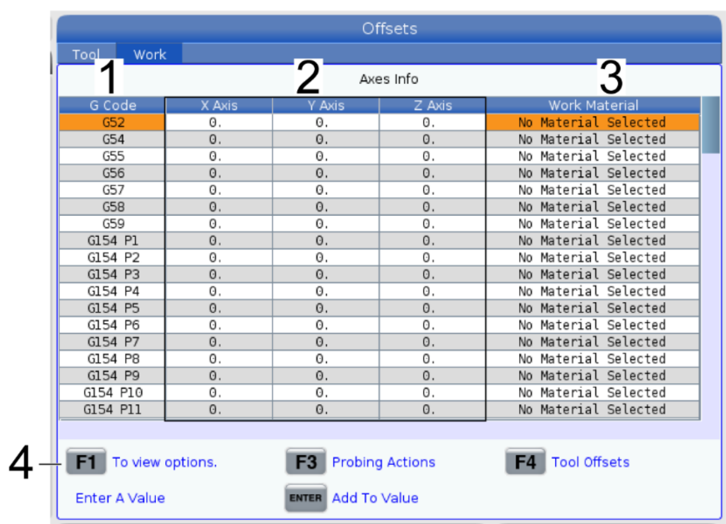
NOTE G110 to G129 refer to the same work offsets as G154 P1 through P20 ; they can be selected by using either method.

When a *G154* work offset is active, the heading in the upper right work offset will show the *G154P* value.

NOTE The P values are modal. This means if you are in the middle of a canned cycle and a G04 Pnn or an M97 Pnn is used the P value will be used for the dwell / subprogram as well as the canned cycle.

Press the **OFFSET**, then the **F4** to view the work offset values. The work offsets can be entered manually or automatically with a probe. The list below will shows how each work offset setting works.

1) G Code - This column displays all the available work offset G-codes. For more information on these work offsets, refer to the G52 Set Work Coordinate System (Group 00 or 12), G54 Work Offsets, G92 Set Work Coordinate Systems Shift Value (Group 00).



2) X, Y, Z, Axis - This column displays the work offset value for each axis. If rotary axis are enabled the offsets for these will be displayed on this page.

3) Work Material - This column is used by the VPS feeds and speeds library.

4) These functions buttons allow you to set the offset values. Type in the desired work offset value and press [F1] to set the value. Press [F3] to set a probing action. Press [F4] to toggle from work to tool offset tab. Type in a value and press Enter to add to the current value.

TOOL OFFSET CALLS

Tool Functions:
The Tnnoo code selects the next tool (nn) and offset (oo). **Example: T0101 → Tool 1, Tool Offset 1**

FANUC Coordinate System:
T-codes have the format Txxyy where xx specifies the tool number from 1 to the maximum number of stations on the turret; and yy specifies the tool geometry and tool wear indices from 1 to 50. The tool geometry X and Z values are added to the work offsets. If tool nose compensation is used, yy specifies the tool geometry index for radius, taper, and tip. If yy = 00 no tool geometry or wear is applied.

Tool Offsets Applied by FANUC:
Setting a negative tool wear in the tool wear offsets moves the tool further in the negative direction of the axis. Thus, for O.D. turning and facing, setting a negative offset in the X-axis results in a smaller diameter part and setting a negative value in the Z-axis results in more material being taken off the face.
Note: There is no X or Z motion required prior to performing a tool change and it wastes time in most cases to return X or Z to the home position. However, you must position X or Z to a safe location prior to a tool change to prevent a crash between the tools and the fixture or part.
Low air pressure or insufficient volume reduces the pressure applied to the turret clamp/unclamp piston and slows down the turret index time or does not unclamp the turret.

To load or change tools:

1. Press **[POWER UP/RESTART]** or **[ZERO RETURN]** and then **[ALL]**. The control moves the tool turret to a normal position.

2. Press **[MDI/DNC]** to toggle to MDI mode.

3. Press **[TURRET FWD]** or **[TURRET REV]**. The machine indexes the turret to the next tool position. Shows the current tool in the Active Tool window in the lower right of the display.

4. Press **[CURRENT COMMANDS]**. Shows the current tool in the Active Tool display in the upper right of the screen.

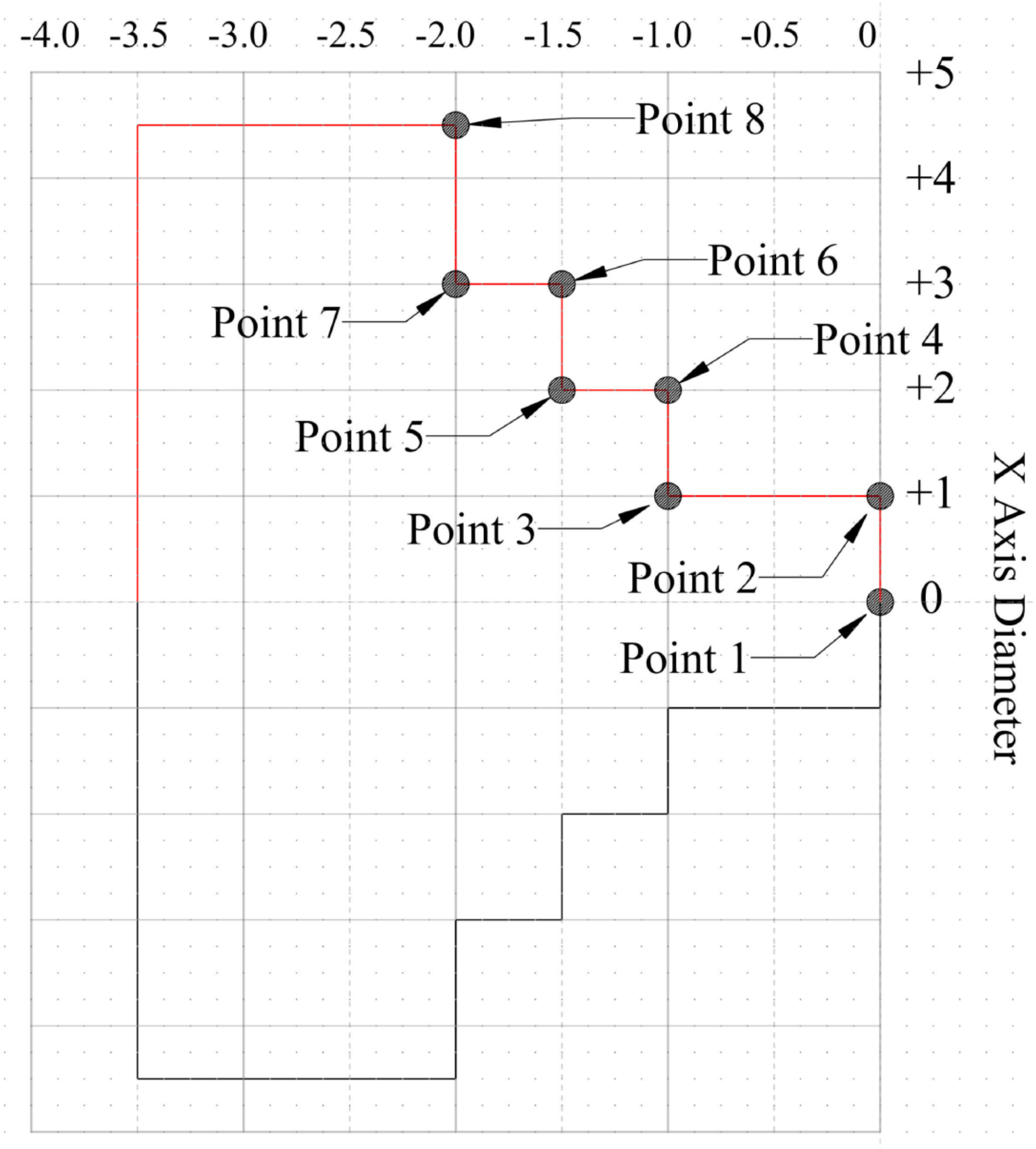
G53 NON-MODAL MACHINE COORDINATE SELECTION (GROUP 00)

This code temporarily cancels work coordinates offsets and uses the machine coordinate system. This code will also ignore tool offsets.

ABSOLUTE VS. INCREMENTAL POSITIONING (XYZ VS. UVW)

Absolute (**XYZ**) and incremental positioning (**UVW**) define how the control interprets axis motion commands. When you command axis motion using **X, Y, or Z**, the axes move to that position relative to the origin of the coordinate system currently in use. When you command axis motion using **U(X), V(Y), or W(Z),** the axes move to that position relative to the current position. Absolute programming is useful in most situations. Incremental programming is more efficient for repetitive, equally spaced cuts.

ABSOLUTE AND INCREMENTAL EXERCISE



Starting from X0, Z0 to point 5. Fill in the tool path positions below for both absolute and incrementle movements.

Absolute Position (X, Z)			Incremental Position (U, W)		
Point 1	X	Z	**Point 1 TO 2**	U	W
Point 2	X	Z	**Point 2 TO 3**	U	W
Point 3	X	Z	**Point 3 TO4**	U	W
Point 4	X	Z	**Point 4 TO5**	U	W
Point 5	X	Z	**Point 5 TO6**	U	W
Point 6	X	Z	**Point 6 TO7**	U	W
Point 7	X	Z	**Point 7 TO8**	U	W
Point 8	X	Z			

G00 RAPID MOTION POSITIONING (GROUP 01)

G00 Rapid Motion Positioning (Group 01)

* **B** - B-axis motion command
* **C** - C-Axis motion command
* **U** - X-axis incremental motion command
* **W** - Z-axis incremental motion command
* **X** - X-axis absolute motion command
* **Y** - Y-axis absolute motion command
* **Z** - Z-axis absolute motion command
* **E** - Optional code to specify the rapid rate of the block as a percent.

* indicates optional

This G code is used to move the machines axes at the maximum speed. It is primarily used to quickly position the machine to a given point before each feed (cutting) command. This G code is modal, so a block with *G00* causes all following blocks to be rapid motions until another cutting move is specified.

NOTE Generally, rapid motion will not be in a straight line. Each axis specified is moved at the same speed, but all axes will not necessarily complete their motions at the same time. The machine will wait until all motions are complete before starting the next command.

G01 LINEAR INTERPOLATION MOTION (GROUP 01)

G01 Linear Interpolation Motion (Group 01)

F - Feed rate
* **B** - B-axis motion command
* **C** - C-Axis motion command
* **U** - X-axis incremental motion command
* **W** - Z-axis incremental motion command
* **X** - X-axis absolute motion command
* **Y** - Y-axis absolute motion command
* **Z** - Z-axis absolute motion command
* **A** - Optional angle of movement (used with only one of *X*, *Z*, *U*, *W*)
* **I** - X-axis chamfering from Z to X (the sign does not matter, only for 90 degree turns)
* **K** - Z-axis chamfering from X to Z (the sign does not matter, only for 90 degree turns)

* **,C** - Distance from center of intersection where the chamfer begins (the sign does not matter, can chamfer non-90 degree lines)
* **,R / R** - Radius of the fillet or arc (the sign does not matter)

This G code provides for straight line (linear) motion from point to point. Motion can occur in 1 or more axes. You can command a *G01* with 3 or more axes All axes will start and finish motion at the same time. The speed of all axes is controlled so that the feed rate specified is achieved along the actual path. The C-Axis may also be commanded and this will provide a helical (spiral) motion. A C-Axis feed rate is dependent on the C-Axis diameter setting (Setting 102) to create a helical motion.
The *F* address (feedrate) command is modal and may be specified in a previous block. Only the axes specified are moved.

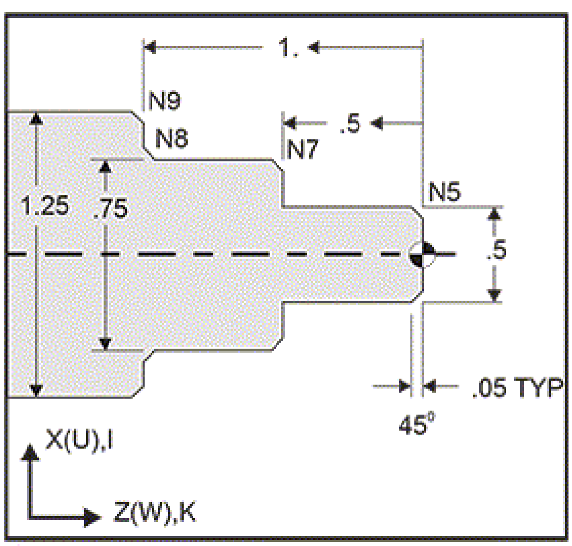
CORNER ROUNDING AND CHAMFERING EXAMPLE

A chamfer block or a corner rounding block can be automatically inserted between two linear interpolation blocks by specifying *,C* (chamfering) or *,R* (corner rounding).

NOTE Both of these variables use a comma symbol (,) before the variable.

There must be a terminating linear interpolation block after the beginning block (a *G04* pause may intervene). These two linear interpolation blocks specify a theoretical corner of intersection. If the beginning block specifies a *,C* (comma C) the value after the *C* is the distance from the corner of intersection where the chamfer begins and also the distance from that same corner where the chamfer ends. If the beginning block specifies a *,R* (comma R) the value after the R is the radius of a circle tangent to the corner at two points: the beginning of the corner rounding arc block that is inserted and the endpoint of that arc. There can be consecutive blocks with chamfer or corner rounding specified. There must be movement on the two axes specified by the selected plane (the active plane X-Y (*G17*), X-Z (*G18*) or Y-Z (*G19*). For chamfering a 90 angle only, an *I* or *K* value can be substituted where *,C* is used.

Chamfering



***** Program on USB Drive; G01 LINEAR INTERPOLATION MOTION (GROUP 01)\O60011.NC *****

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

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

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

Addresses:
I = chamfering, Z to X
K = chamfering, X to Z
R = corner rounding (X or Z axis direction)

Notes:

1. Incremental programming is possible if *U* or *W* is specified in place of *X* or *Z*, respectively. So its actions are as follows:

   X(current position + i) = Ui
   Z(current position + k) = Wk
   X(current position + r) = Ur
   Z(current position + r) = Wr

2. Current position of X or Z Axis is added to the increment.

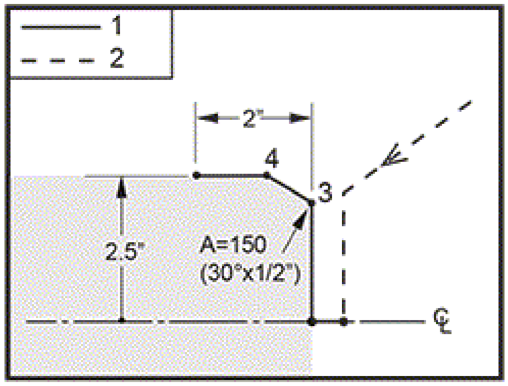
3. *I*, *K* and *R* always specify a radius value (radius programming value).

Rules:

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

G01 Chamfering with A

When specifying an angle (*A*), command motion in only one of the other axes (X or Z), the other axis is calculated based on the angle.
G01 Chamfering with A: [1] Feed, [2] Rapid, [3] Start Point, [4] Finish Point.

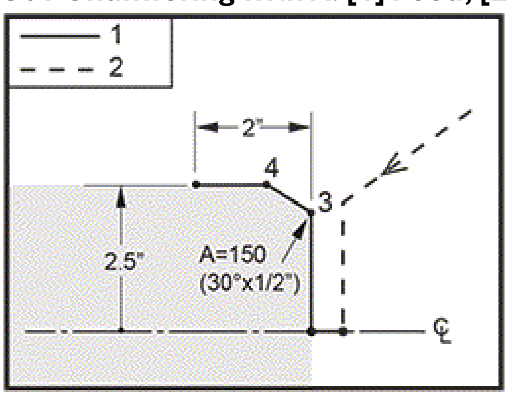


***** Program on USB Drive; G01 LINEAR INTERPOLATION MOTION (GROUP 01)\O60012 (Chamfering).NC *****

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

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

***G01* Chamfering with A: [1] Feed, [2] Rapid, [3] Start Point, [4] Finish Point.**



***** Program on USB Drive; G01 LINEAR INTERPOLATION MOTION (GROUP 01)\O60012 (Chamfering with 'A').NC *****

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

G02 CW/G03 CCW CIRCULAR INTERPOLATION MOTION (GROUP 01)

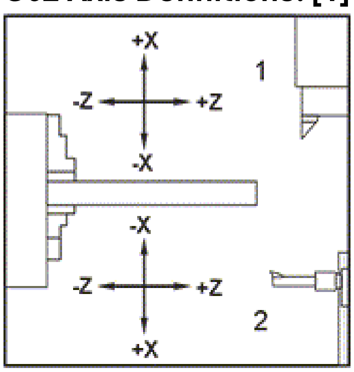
G02 CW/G03 CCW Circular Interpolation Motion (Group 01)

F - Feed rate
* **I** - Distance along X-axis to center of circle
* **J** - Distance along Y-axis to center of circle
* **K** - Distance along Z-axis to center of circle
* **R** - Radius of arc
* **U** - X-axis incremental motion command
* **W** - Z-axis incremental motion command
* **X** - X-axis absolute motion command
* **Y** - Y-axis absolute motion command
* **Z** - Z-axis absolute motion command

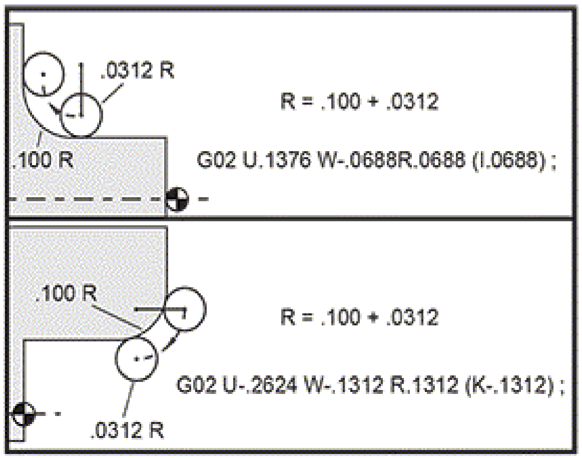
* indicates optional

These G codes are used to specify a circular motion (CW or CCW) of the linear axes (Circular motion is possible in the X and Z axes as selected by *G18*). The *X* and *Z* values are used to specify the end point of the motion and can use either absolute (*X* and *Z*) or incremental motion (*U* and *W*). If either the *X* or *Z* is not specified, the endpoint of the arc is the same as the starting point for that axis. There are two ways to specify the center of the circular motion; the first uses *I* or *K* to specify the distance from the starting point to the center of the arc; the second uses *R* to specify the radius of the arc. For information on *G17* and *G19* Plane Milling, see the Live Tooling section.

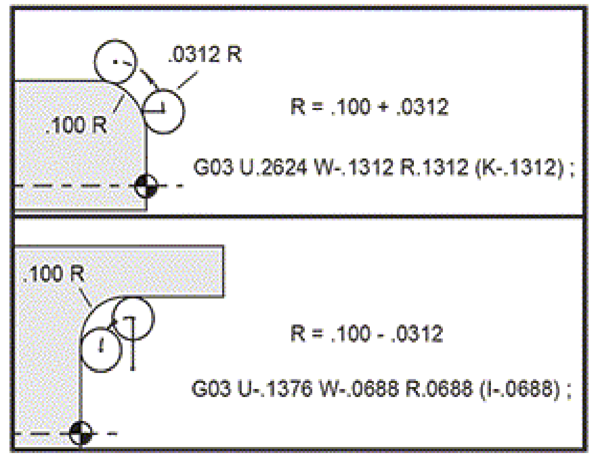
***G02* Axis Definitions: [1] Turret Lathes, [2] Table Lathes.**



G02 and G03 Programs



G02

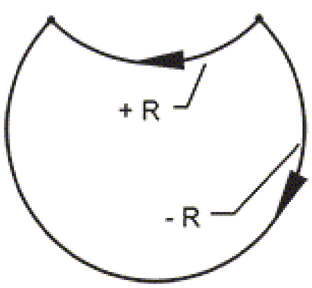


G03

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

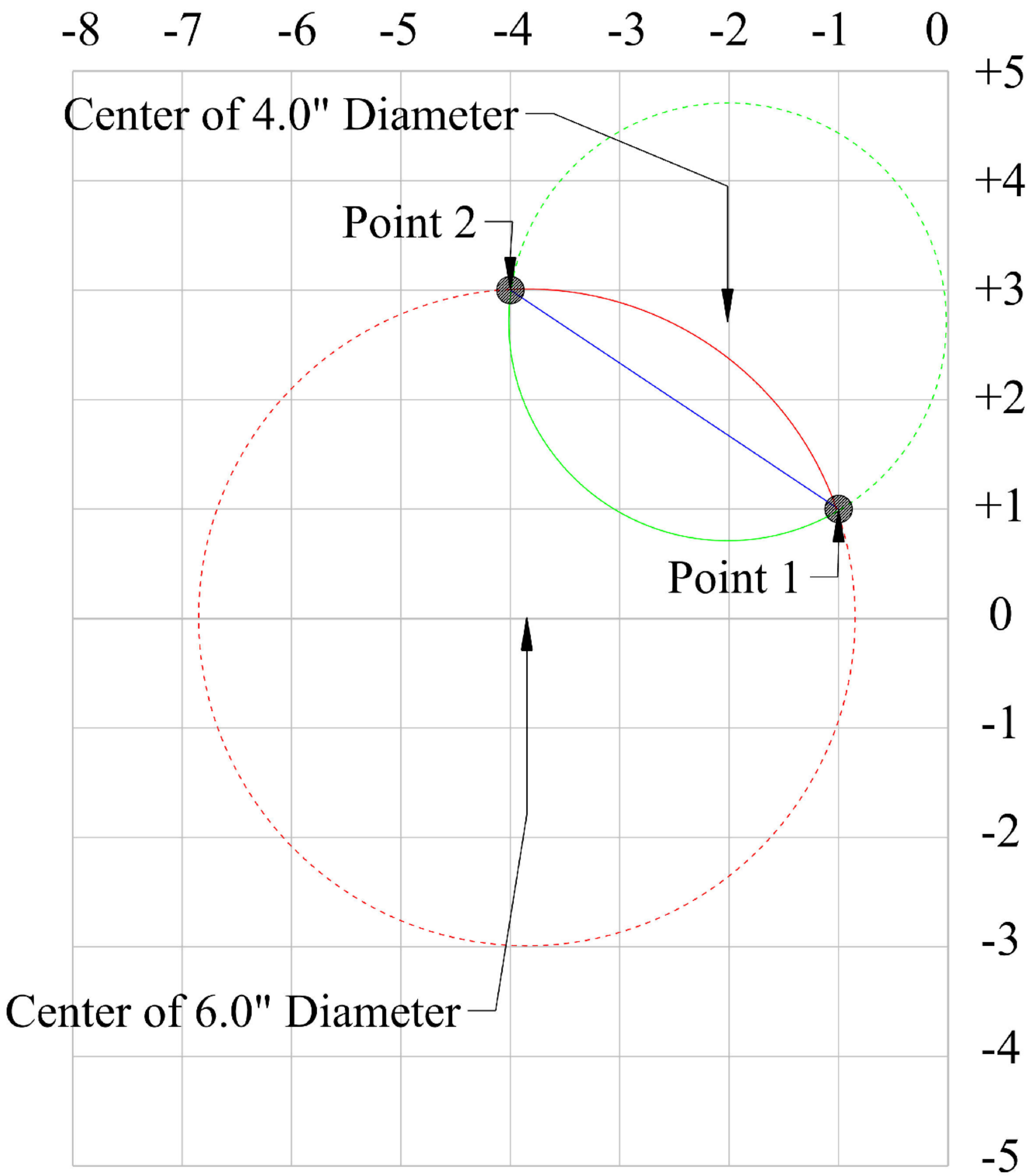
The following lines cut an arc of less than 180 degrees:

G01 X3.0 Z4.0 ;
G02 Z-3.0 R5.0 ;
G02 Arc Using Radius



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

G02 Defined X and Z: [1] Start.

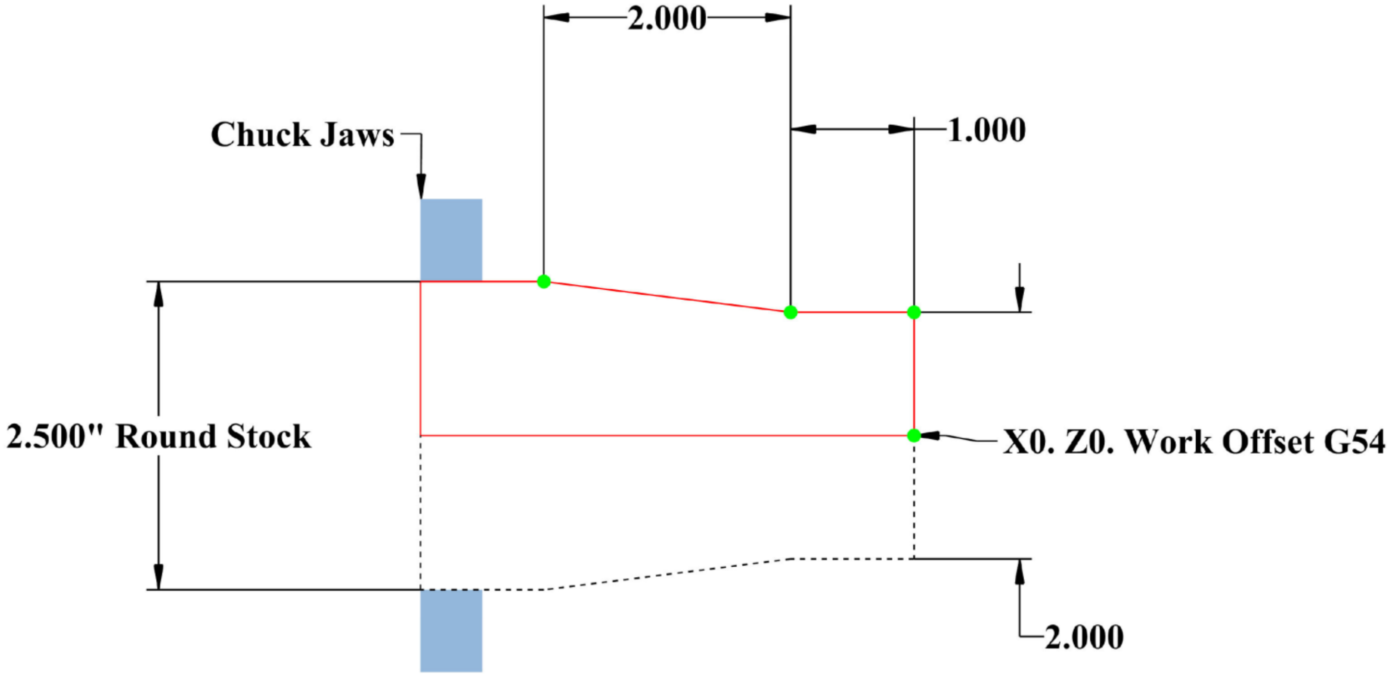


Point 1 to Point 2

Line 1 (G01) LINEAR	Line 2 (G02) CW	Line 3 (G03) CCW
…… G01 X1.0 Z-1.0 F0.005 **G01 X3.0 Z-4.0** M30	…… G01 X1.0 Z-1.0 F0.005 **G02 X3.0 Z-4.0 R2.0** M30	…… G01 X1.0 Z-1.0 F0.005 **G03 X3.0 Z-4.0 R3.0** M30

Notes

TOOL PATH EXERCISE 1



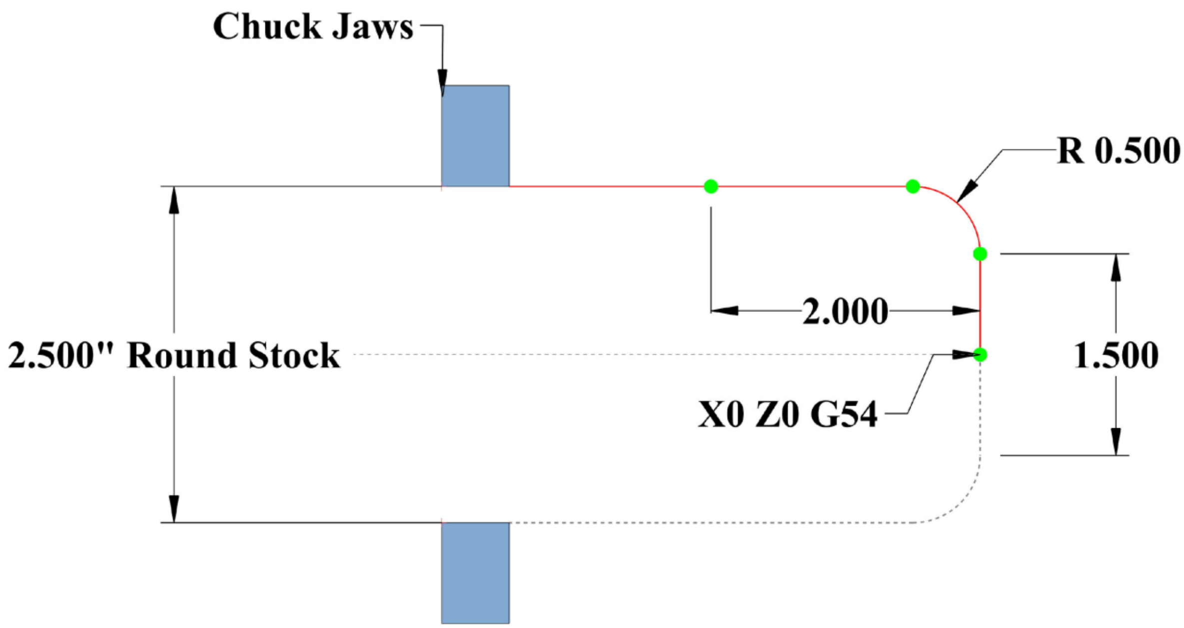
***** Start your program at X0 Z0 and follow the green dots along the path.**

HAAS Simulator: Go to MDI Mode:

T101
G54
S300 M03
G00 X0. Z0.1
G01 Z0 F0.005
…
…
…
…
…
…
M30

Run program in Graphic Mode.

TOOL PATH EXERCISE 2



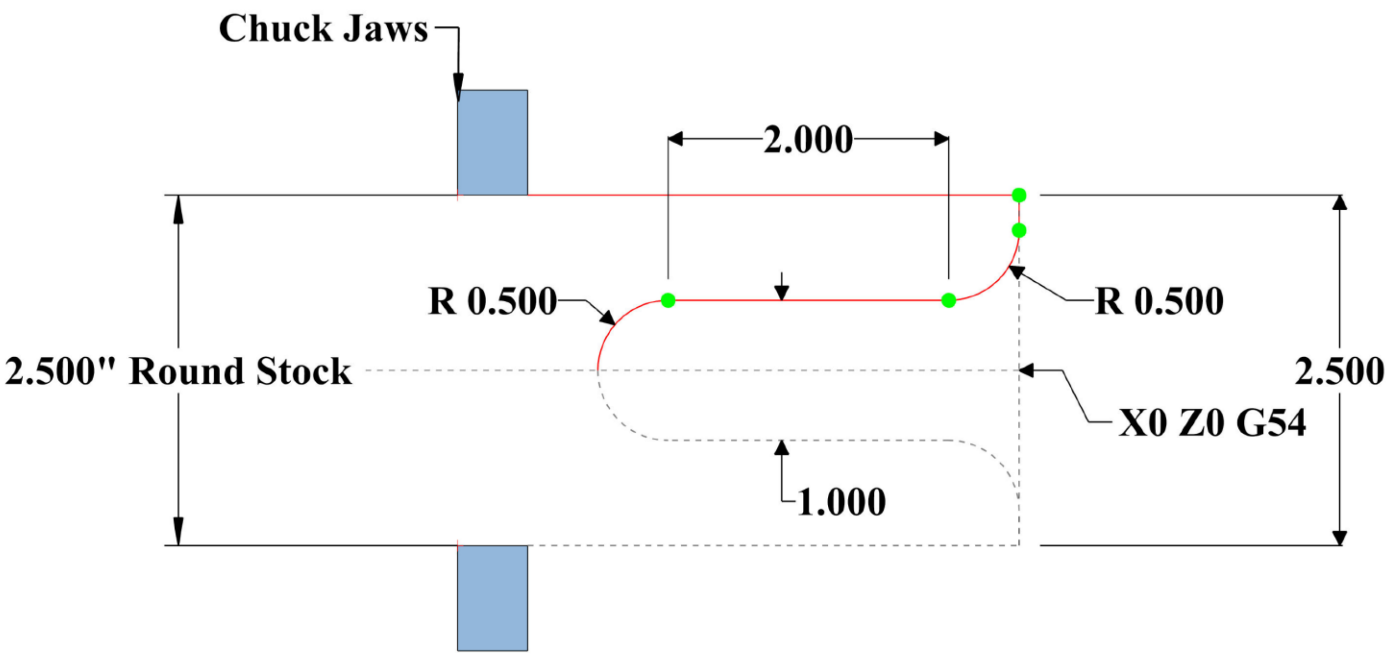
***** Start your program at X0 Z0 and follow the green dots along the path.**
***** Program G02 and G03 with R**

HAAS Simulator: Go to MDI Mode:

T101
G54
S300 M03
G00 X0. Z0.1
G01 Z0. F0.005
...
...
...
...
...
...
M30

Run program in Graphic Mode.

TOOL PATH EXERCISE 3



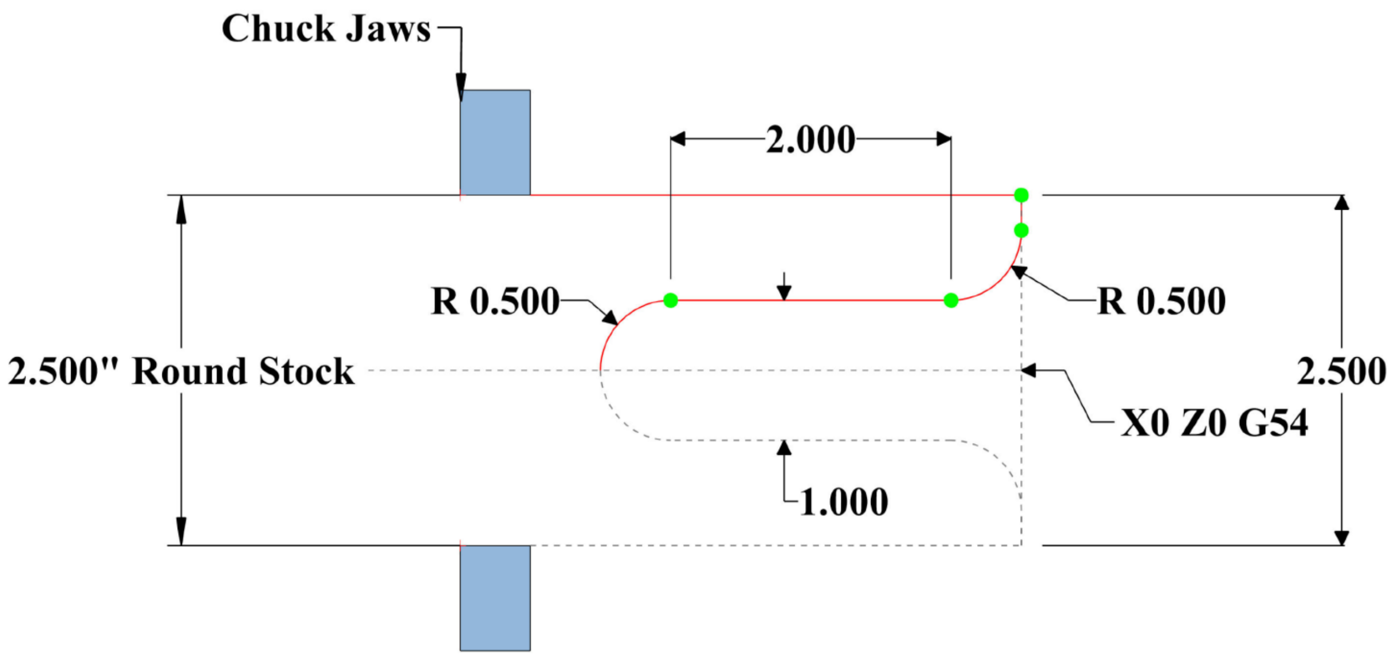
***** Start your program at X2.5 and Z0. Follow the green dots along the tool path.**
***** Program G02 and G03 with R**

HAAS Simulator: Go to MDI Mode:

T101
G54
S300 M03
G00 X2.5 Z0.1
G01 Z0. F0.005
…
…
…
…
…
…
M30

Run program in Graphic Mode.

TOOL PATH EXERCISE 4



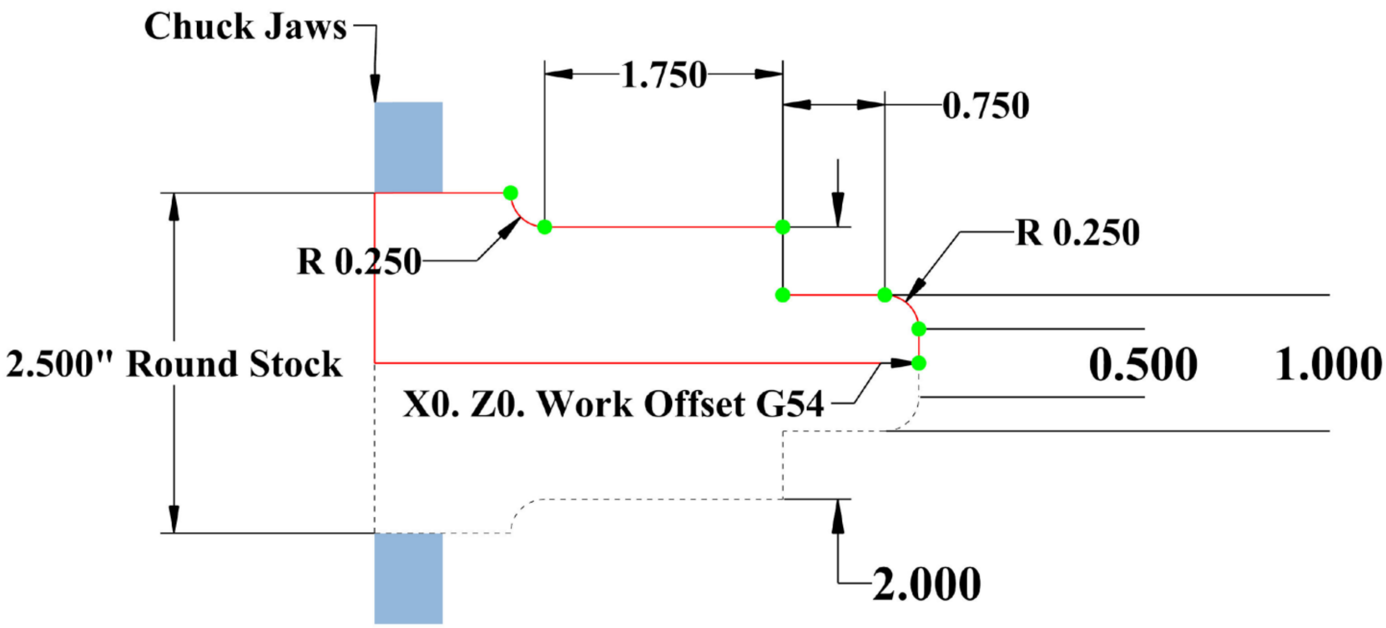
***** Start your program at X2.5 and Z0. Follow the green dots along the tool path.**
***** Program G02 and G03 with I and K ONLY**

HAAS Simulator: Go to MDI Mode:

```
T101
G54
S300 M03
G00 X2.5 Z0.1
G01 Z0. F0.005
…
…
…
…
…
…
M30
```

Run program in Graphic Mode.

TOOL PATH EXERCISE 5

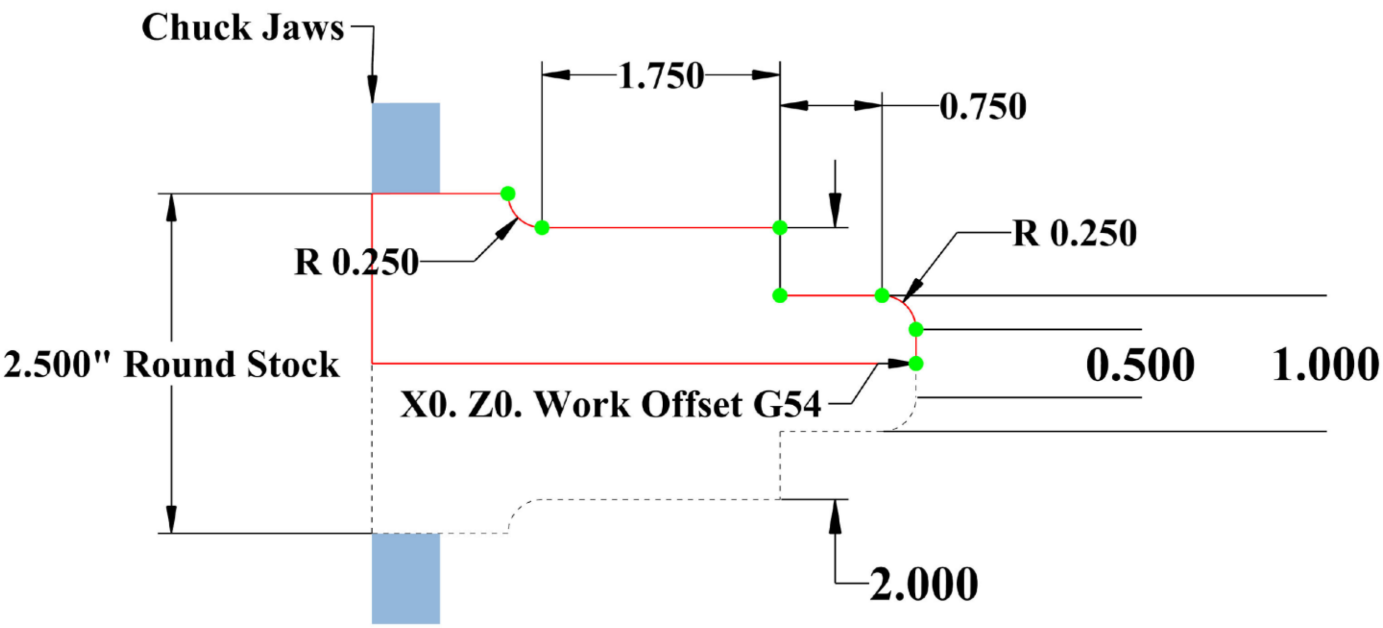


***** Start your program at X0 Z0 and follow the green dots along the path.**
***** Use ONLY G01 with R#.#**

HAAS Simulator: Go to MDI Mode:

T101
G54
S300 M03
G00 X0. Z0.1
G01 Z0. F0.005
…
…
…
…
…
…
M30

Run program in Graphic Mode.



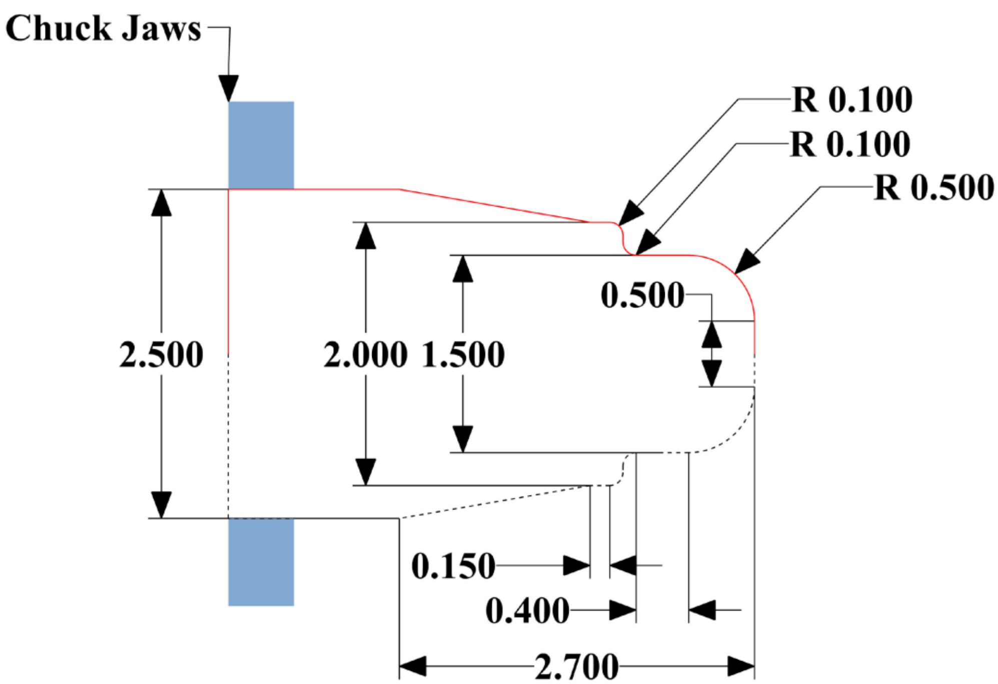
***** Start your program at X0 Z0 and follow the green dots along the path.**
***** Use ANY method for the radius**
***** Incremental moves only; U and W. Must move to absolute X0 Z0 first.**

HAAS Simulator: Go to MDI Mode:

T101
G54
S300 M03
G00 X0. Z0.1
G01 Z0. F0.005
U__ W__ ...
...
...
...
...
...
...
M30

Run program in Graphic Mode.

TOOL PATH EXERCISE 7



***** Start your program at X0 Z0, Spindle Center Line**
***** Use ANY method for the radius**
***** Use Incremental and/or Absolute**

HAAS Simulator: Go to MDI Mode:

T101
G54
S300 M03
G00 X0. Z0.1
G01 Z0. F0.005
…
…
…
…
…
…
M30

Run program in Graphic Mode.

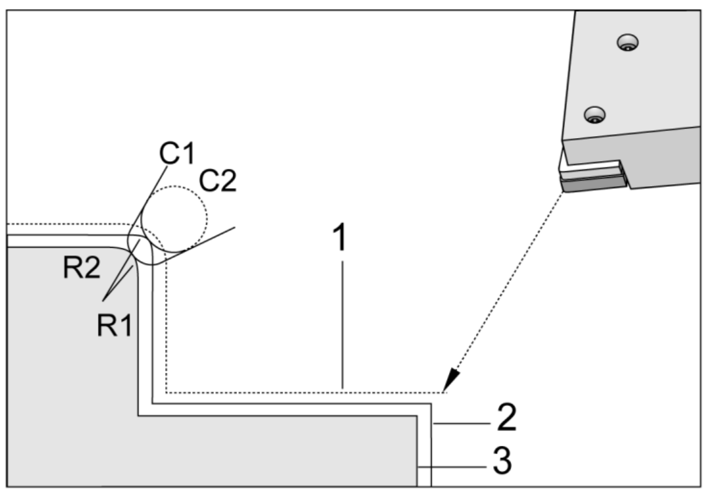
TOOL NOSE COMPENSATION - PROGRAMMING

Tool Nose Compensation (TNC) is a feature that lets you adjust in a programmed tool path for different cutter sizes, or for normal cutter wear. With TNC you only need to enter minimal offset data when you run a program. You do not need to do additional programming.

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

Cutting path without tool nose compensation:

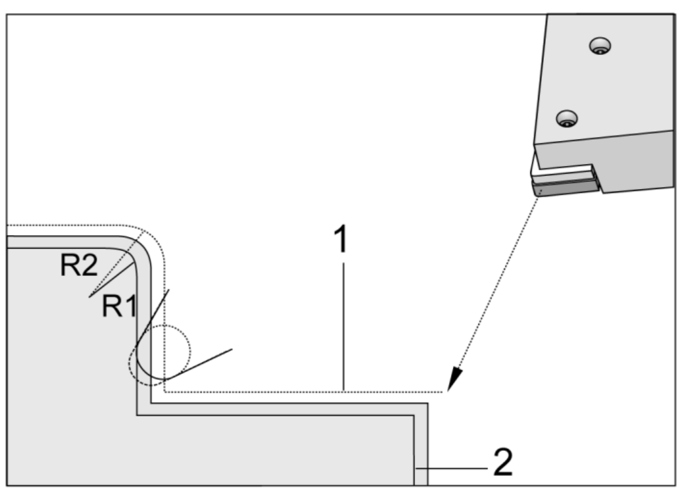
[1] Tool Path
[2] Cut after wear
[3] Desired cut.



Cutting path with tool nose compensation:

[1] Compensated tool path
[2] Desired cut and programmed tool path.

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



Approach and Departure Moves For TNC

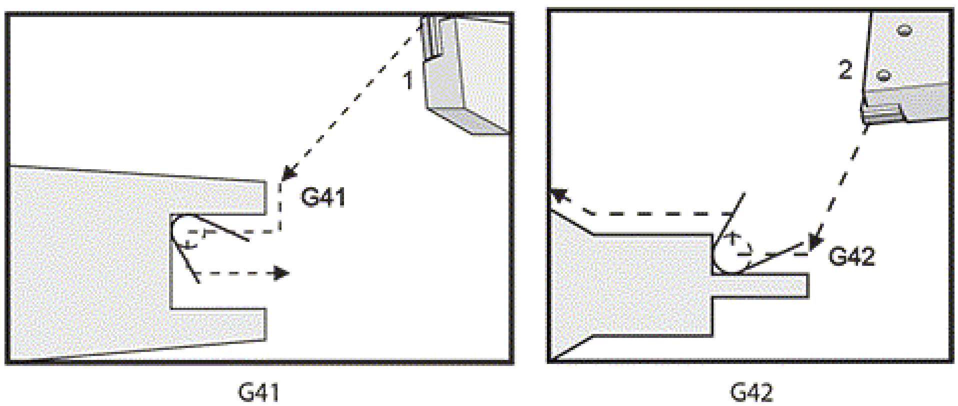
The first X or Z motion in the same line that contains a G41 or G42 is called the Approach move. The approach must be a linear move, that is a G01 or G00. The first move is not compensated, yet at the end of the approach move the machine position is fully compensated. See the following figure.

F5.11: TNC Approach and Depart Moves: [1] Compensated Path, [2] Programmed path.

G41 TOOL NOSE COMPENSATION (TNC) LEFT / G42 TNC RIGHT (GROUP 07)

G41 or *G42* will select tool nose compensation. *G41* moves the tool to the left of the programmed path to compensate for the size of a tool and vice versa for *G42*. A tool offset must be selected with a Tnnxx code. where xx corresponds to the offsets that are to be used with the tool. For more information, see Tool Nose Compensation in the Operation section of this manual.

G41 **TNC Right and** ***G42*** **TNC Left: [1] Tip = 2, [2] Tip = 3.**



TOOL NOSE COMPENSATION IN CANNED CYCLES

Some canned cycles ignore tool nose compensation, expect a specific coding structure, or perform their own specific canned cycle activity (also refer to page 304 for more information on using canned cycles).

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

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

BASIC PROGRAMMING

A typical CNC program has (3) parts:

1) Preparation: This portion of the program selects the work and tool offsets, selects the cutting tool, turns on the coolant, sets spindle speed, and selects absolute or incremental positioning for axis motion.
2) Cutting: This portion of the program defines the tool path and feed rate for the cutting operation.
3) Completion: This portion of the program moves the spindle out of the way, turns off the spindle, turns off the coolant, and moves the table to a position from where the part can be unloaded and inspected.
This is a basic program that makes a 0.100" (2.54 mm) deep cut with Tool 1 in a piece of material along a straight-line path from X = 0.0, Y = 0.0 to X = - 4.0, Y = - 4.0.
NOTE: A program block can contain more than one G-code, as long as those G-codes are from different groups. You cannot place two G-codes from the same group in a program block. Also note that only one M-code per block is allowed.

```
%
O40001 (Basic program) ;
(G54 X0 Y0 is top right corner of part) ;
(Z0 is on top of the part) ;
(T1 is a 1/2" end mill) ;
(BEGIN PREPARATION BLOCKS) ;
T1 M06 (Select tool 1) ;
G00 G90 G17 G40 G49 G54 (Safe startup) ;
X0 Y0 (Rapid to 1st position) ;
S1000 M03 (Spindle on CW) ;
G43 H01 Z0.1 (Tool offset 1 on) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G01 F20. Z-0.1 (Feed to cutting depth) ;
X-4. Y-4. (linear motion) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.1 M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%
```

***** Program on USB Drive *****

PREPARATION

These are the preparation code blocks in the sample program **O40001**:

Preparation Code Block	Description
%	Denotes the beginning of a program written in a text editor.
O40001 (Basic program) ;	O40001 is the name of the program. Program naming convention follows the Onnnnn format: The letter "O", or "o" is followed by a 5-digit number.
(G54 X0 is at the center of rotation) ;	Comment
(Z0 is on face of the part) ;	Comment
(T1 is an end face cutting tool) ;	Comment
T101 (Select tool and offset 1) ;	T101 selects the tool, the offset 1, and commands the tool change to Tool 1.
G00 G18 G20 G40 G80 G99 (Safe startup) ;	This is referred to as a safe startup line. It is good machining practice to place this block of code after every tool change. G00 defines axis movement following it to be in Rapid Motion mode. G18 defines the cutting plane as the XZ plane. G20 defines the coordinate positioning to be in Inches. G40 cancels Cutter Compensation. G80 cancels any canned cycles. G99 puts the machine in Feed per Rev mode.
G50 S1000 (Limit spindle to 1000 RPM) ;	G50 limits the spindle to a max of 1000 RPM. S1000 is the spindle speed address. Using Snnnn address code, where nnnn is the desired spindle RPM value.
G97 S500 M03 (CSS off, Spindle on CW) ;	G97 cancels constant surface speed (CSS) making the S value a direct RPM of 500. S500 is the spindle speed address. Using Snnnn address code, where nnnn is the desired spindle RPM value. M03 turns on the spindle. **Note:** Lathes equipped with a gearbox, the control will not select high gear or low gear for you. You must use a M41 Low Gear or M42 High Gear on the line before the Snnnn code. Refer to M41 / M42 Low / High Gear Override for more information on these M-codes.
G00 G54 X2.1 Z0.1 (Rapid to 1st position) ;	G00 defines axis movement following it to be in Rapid Motion mode. G54 defines the coordinate system to be centered on the Work Offset stored in G54 on the Offset display. X2.0 commands the X Axis to X = 2.0. Z0.1 commands the Z Axis to Z = 0.1.
M08 (Coolant on) ;	M08 turns on the coolant.
G96 S200 (CSS on) ;	G96 turns on CSS. S200 specifies a cutting speed of 200 ipm to be used along with the current diameter to calculate the correct RPM.

CUTTING

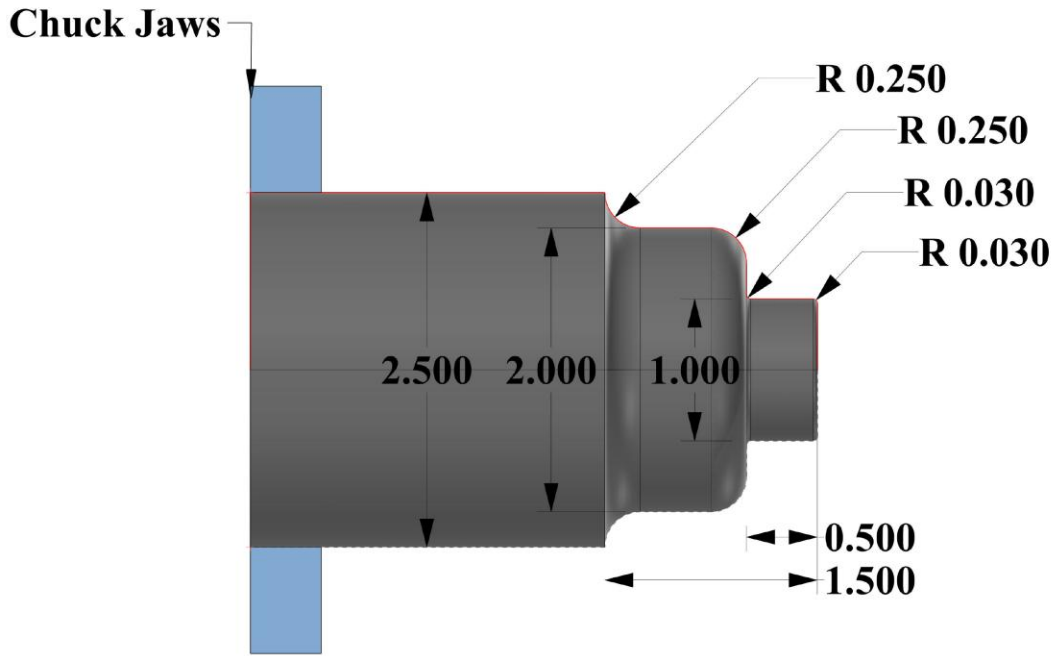
These are the cutting code blocks in the sample program **O40001**:

Cutting Code Block	Description
G01 Z-0.1 F.01 (Linear feed) ;	G01 defines axis movements after it to be in a straight line. Z-0.1 commands the Z Axis to Z = -0.1. G01 requires address code Fnnn.nnnn. F.01 specifies the feedrate for the motion is .0100" (.254 mm)/Rev.
X-0.02 (Linear feed) ;	X-0.02 commands the X Axis to X = -0.02.

COMPLETION

Completion Code Block	Description
G00 Z0.1 M09 (Rapid retract, Coolant off) ;	G00 commands the axis motion to be completed in rapid motion mode. Z0.1 Commands the Z Axis to Z = 0.1. M09 commands the coolant to turn off.
G97 S500 (CSS off) ;	G97 cancels constant surface speed (CSS) making the S value a direct RPM of 500. On machines with a gearbox, the control automatically selects high gear or low gear, based on the commanded spindle speed. S500 is the spindle speed address. Using Snnnn address code, where nnnn is the desired spindle RPM value.
G53 X0 (X home) ;	G53 defines axis movements after it to be with respect to the machine coordinate system. X0 commands the X Axis to move to X = 0.0 (X home).
G53 Z0 M05 (Z home, spindle off) ;	G53 defines axis movements after it to be with respect to the machine coordinate system. Z0 commands the Z Axis to move to Z = 0.0 (Z home). M05 turns off the spindle.
M30 (End program) ;	M30 ends the program and moves the cursor on the control to the top of the program.
%	Denotes the end of a program written in a text editor.

TOOL PATH WITH TOOL NOSE COMP (TNC) EXERCISE



Tool Information			FEED: FPR
Tool 1		Tungaloy CNGM, HM Chipbreaker, BXM20 Grade, Carbide Turning Insert, .0310 CR	.008

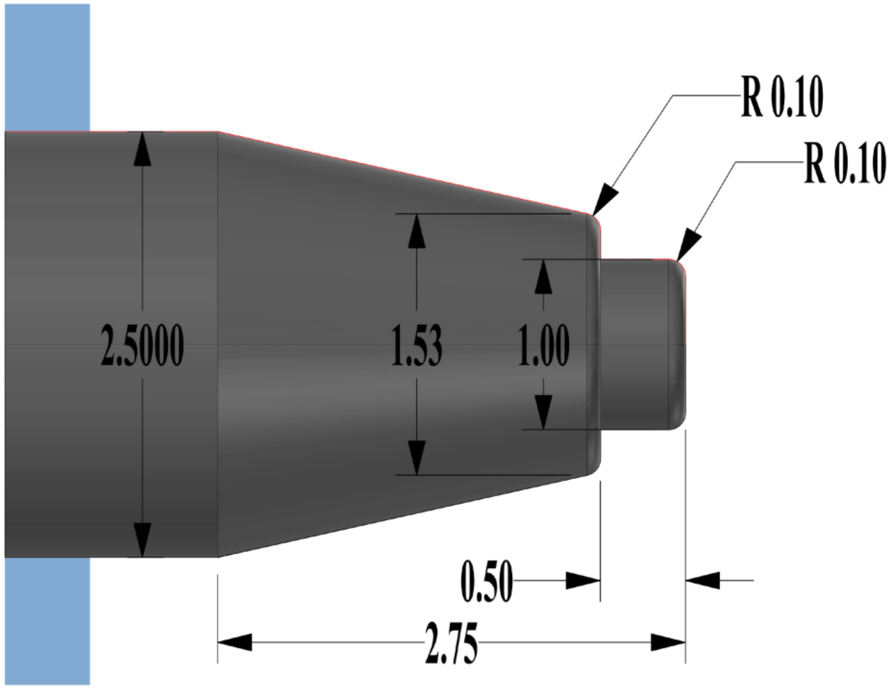
**** Use the information above for feed.*
**** Start at **X0 Z0, G54**. End at last green dot along the path.*

HAAS Simulator: Go to MDI Mode: **Save Program Referencing Page Number**

G00 G18 G20 G40 G80 G99 (Safety Line)
T101
G54
G50 S2000
G97 S500 M03
M08
G00 Z___ X___ → (Rapid Close to Stock)
G96 S300 M03
G42 G01 Z___ X___ F___ (Feed to starting point)
...
M30

Run program in Graphic Mode.

TOOL PATH WITH TOOL NOSE COMP (TNC) EXERCISE 2



Tool Information			FEED: FPR
Tool 1		Tungaloy CNGM, HM Chipbreaker, BXM20 Grade, Carbide Turning Insert, .0310 CR	.008

**** Use the information above for feed.*
**** Start at* ***X0 Z0, G54****. End at last green dot along the path.*
**** Stock Size is* ***2.5”***

HAAS Simulator: Go to MDI Mode:	Save Program Referencing Page Number
G00 G18 G20 G40 G80 G99 (Safety Line) **T101** **G54** **G50 S2000** **G97 S500 M03** **M08** **G00 Z___ X___ → (Rapid Close to Stock)** **G96 S300 M03** **G42 G01 Z___ X___ F___ (Feed to starting point)** **M30** **Run program in Graphic Mode.**	

CANNED CYCLES G70 / G71 / G72 / G73 / G74 / G75 / G76 / G90 / G92 / G94

G70 FINISHING CYCLE (GROUP 00)

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

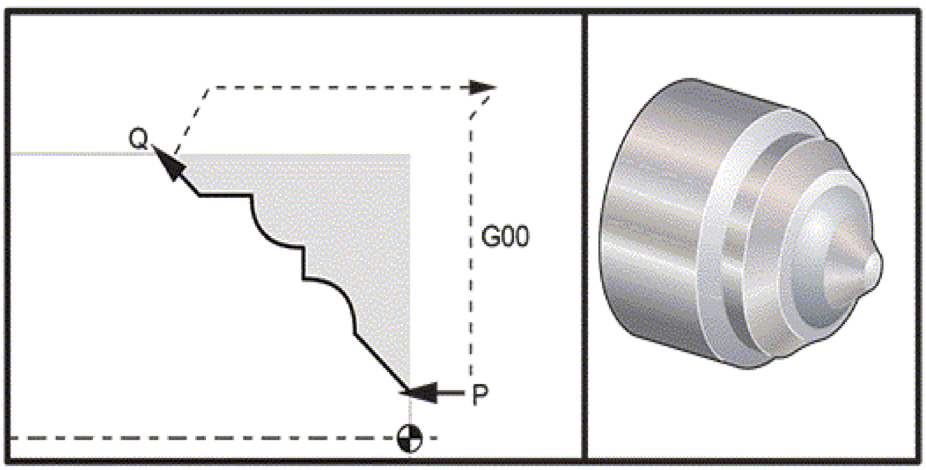
P - Starting Block number of routine to execute
Q - Ending Block number of routine to execute

G18 Z-X plane must be active

NOTE The P values are modal. This means if you are in the middle of a canned cycle and a G04 Pnn or an M97 Pnn is used the P value will be used for the dwell / subprogram as well as the canned cycle.

G70 **Finishing Cycle: [P] Starting block, [Q] Ending Block.**

```
%
G71 P10 Q50 F.012 (rough out
N10 to N50 the path) ;
N10 ;
F0.014 ;
... ;
N50 ;
... ;
G70 P10 Q50 (finish path defined
by N10 to N50) ;
%
```



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

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

After a *G70*, the block following the *G70* will be executed, not the block with an *N* code matching the *Q* code specified by the *G70* call.

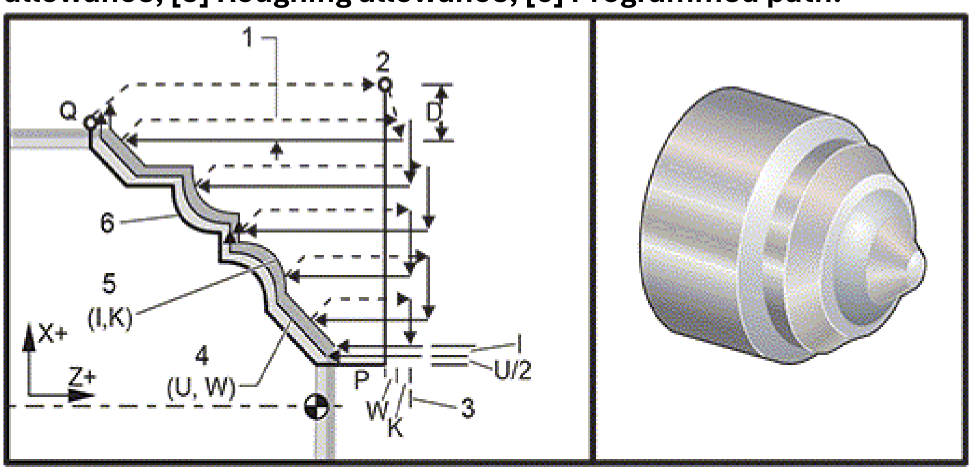
G71 O.D./I.D. STOCK REMOVAL CYCLE (GROUP 00)

* **D** - Depth of cut for each pass of stock removal, positive radius (Only use when using one block G71 notation)
* **F** - Feedrate in inches (mm) per minute (*G98*) or per revolution (*G99*) to use throughout *G71 PQ* block
* **I** - X-axis size and direction of *G71* rough pass allowance, radius
* **K** - Z-axis size and direction of *G71* rough pass allowance
P - Starting Block number of path to rough
Q - Ending Block number of path to rough
* **S** - Spindle speed to use throughout *G71 PQ* block
* **T** - Tool and offset to use throughout *G71 PQ* block
* **U** - X-axis size and direction of *G71* finish allowance, diameter
* **W** - Z-axis size and direction of *G71* finish allowance

* indicates optional
G18 Z-X plane must be active.

NOTE The P values are modal. This means if you are in the middle of a canned cycle and a G04 Pnn or an M97 Pnn is used the P value will be used for the dwell / subprogram as well as the canned cycle.

***G71* Stock Removal: [1] Setting 287, [2] Start position, [3] Z-Axis clearance plane, [4] Finishing allowance, [5] Roughing allowance, [6] Programmed path.**



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

Two types of machining paths are addressed with a *G71* command. The first type of path (Type 1) is when the X-Axis of the programmed path does not change direction. The second type of path (Type 2) allows the X-Axis to change direction. For both Type 1 and Type 2, the programmed path of the Z-axis cannot change direction. If the *P* block contains only an X-Axis position, then Type 1 roughing is assumed. If the *P* block contains both an X-Axis and Z-Axis position, then Type 2 roughing is assumed.

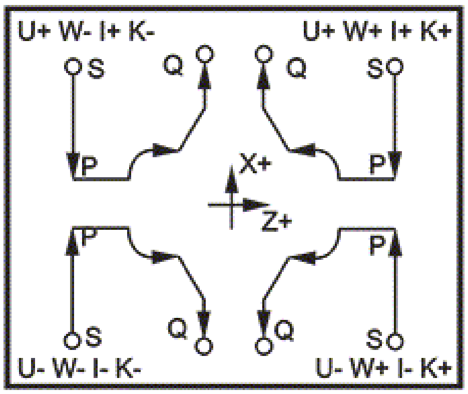
NOTE

The Z-Axis position given in the P block to specify Type 2 roughing does not have to cause axis motion. You can use the current Z-Axis position. In Figure G71 Type 2 O.D./I.D. Stock Removal Example, the P1 block (indicated by the comment in parentheses) contains the same Z-Axis position as the start position G00 block above.

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

In the figures, the start position *S* is the position of the tool at the time of the *G71* call. The *Z* clearance plane [3] is derived from the Z-axis start position and the sum of *W* and optional *K* finish allowance.

***G71* Address Relationships**



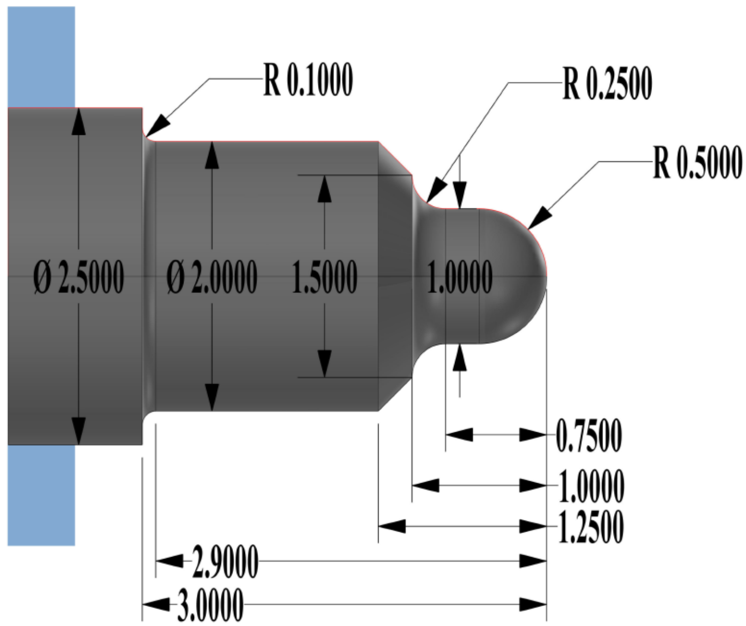
Type I Details
When Type I is specified by the programmer it is assumed that the X-axis tool path does not reverse during a cut. Each roughing pass X-axis location is determined by applying the value specified in *D* to the current X location. The nature of the movement along the Z clearance plane for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

Each roughing pass is stopped before it intersects the programmed tool path allowing for both roughing and finishing allowances. The tool is then retracted from the material, at a 45 degree angle. The tool then moves in rapid mode to the Z-axis clearance plane.
When roughing is completed the tool is moved along the tool path to clean up the rough cut. If I and *K* are specified an additional rough finish cut parallel to the tool path is performed.
Type II Details
When Type II is specified by the programmer the X axis *PQ* path is allowed to vary (for example, the X-axis tool path can reverse direction).
The X axis *PQ* path must not exceed the original starting location. The only exception is the ending *Q* block.

Type II, must have a reference move, in both the X and Z axis, in the block specified by P.
Roughing is similar to Type I except after each pass along the Z axis, the tool will follow the path defined by *PQ*. The tool will then retract parallel to the X axis. The Type II roughing method does not leave steps in the part prior to finish cutting and typically results in a better finish.

Notes

G71 AND G70 TOOL PATH EXERCISE



Tool Information for G71 with G70 Finish			FEED: FPR
Tool 1		Tungaloy CNGM, HM Chipbreaker, BXM20 Grade, Carbide Turning Insert, **0.015" Corner Radius**	Rough Feed 0.01" Finish Feed 0.005"

**** Use the information above for feed.*
**** Start at **X0 Z0, G54**. End at last green dot along the path.*
**** Leave **0.030"** in **X** and **0.015"** in **Z Axis** for finish pass*

HAAS Simulator: Go to MDI Mode:	Save Program Referencing Page Number
G00 G18 G20 G40 G80 G99 (Safety Line)	
T101	
G50 S2000	
G97 S500 M03	
M08	
G54 G00 Z___ X___ (Rapid Close to Stock)	
G96 S300	
G71 P___ Q___ →	
...	
N___	
...	
N___	
...	
G70 P___ Q___	
...	
M30	
Run program in Graphic Mode.	

G72 END FACE STOCK REMOVAL CYCLE (GROUP 00)

First Block (Only use when using two block G72 notation)
* **W** - Depth of cut for each pass of stock removal, positive radius
* **R** - Retract height for each pass of stock removal

Second Block
* **D** - Depth of cut for each pass of stock removal, positive radius (Only use when using one block G72 notation)
* **F** - Feedrate in inches (mm) per minute (*G98*) or per revolution (*G99*) to use throughout *G71 PQ* block
* **I** - X-axis size and direction of *G72* rough pass allowance, radius
* **K** - Z-axis size and direction of *G72* rough pass allowance

P - Starting Block number of path to rough
Q - Ending Block number of path to rough
* **S** - Spindle speed to use throughout *G72 PQ* block
* **T** - Tool and offset to use throughout *G72 PQ* block
* **U** - X-axis size and direction of *G72* finish allowance, diameter
* **W** - Z-axis size and direction of *G72* finish allowance

*indicates optional

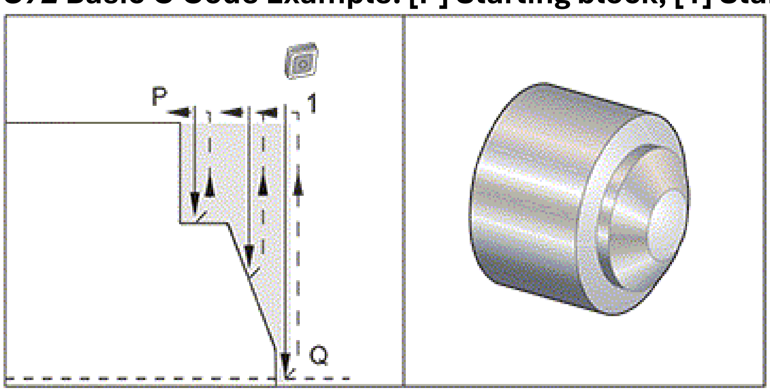
G18 Z-X plane must be active.

2 Block G72 Programming Example:

G72 W... R... G72 F... I... K... P... Q... S... T... U... W...

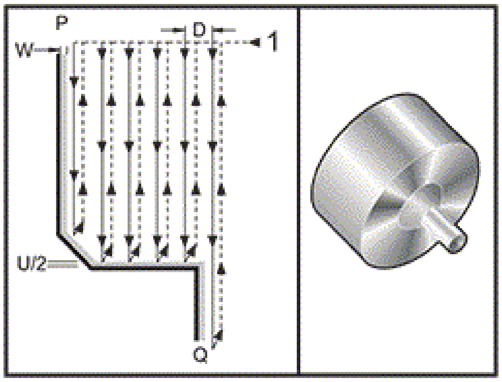
NOTE The P values are modal. This means if you are in the middle of a canned cycle and a G04 Pnn or an M97 Pnn is used the P value will be used for the dwell / subprogram as well as the canned cycle.

***G72* Basic G Code Example: [P] Starting block, [1] Start position, [Q] Ending block.**



*** See Sample Program on USB Drive; O60721.NC ***

G72 Tool Path: [P] Starting block, [1] Start position, [Q] Ending block.



***** See Sample Program on USB Drive; O60722.NC *****

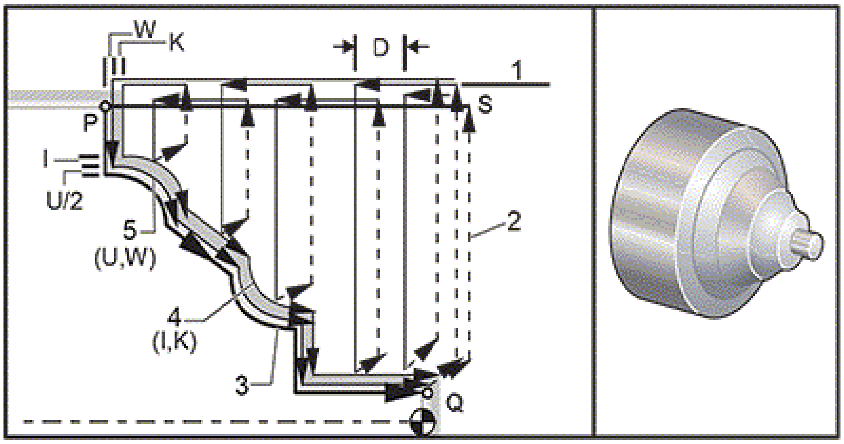
This canned cycle removes material on a part given the finished part shape. It is similar to *G71* but removes material along the face of a part. Define the shape of a part by programming the finished tool path and then use the *G72 PQ* block. Any *F*,*S* or *T* commands on the *G72* line or in effect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

Two types of machining paths are addressed with a *G72* command.

- The first type of path (Type 1) is when the Z Axis of the programmed path does not change direction. The second type of path (Type 2) allows the Z Axis to change direction. For both the first type and the second type of programmed path the X Axis cannot change direction. If Setting *33* is set to FANUC, Type 1 is selected by having only an X-axis motion in the block specified by *P* in the *G72* call.

- When both an X-axis and Z-axis motion are in the *P* block then Type 2 roughing is assumed.

_G72_ End Face Stock Removal Cycle: [P] Starting block, [1] X-Axis clearance plane, [2] G00 block in P, [3] Programmed path, [4] Roughing allowance, [5] Finishing allowance.

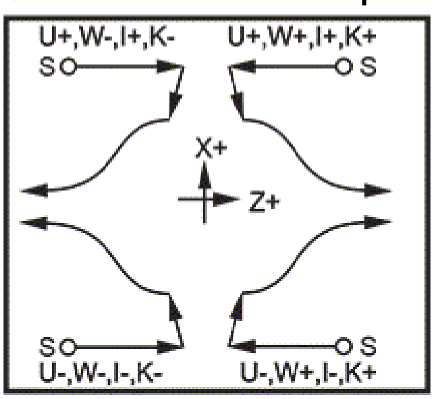


The *G72* consists of a roughing phase and a finishing phase. The roughing and finishing phase are handled differently for Type 1 and Type 2. Generally the roughing phase consists of repeated passes along the X-axis at the specified feed rate. The finishing phase consists of a pass along the programmed tool path to remove excess material left by the roughing phase while leaving material for a *G70* finishing cycle. The final motion in either type is a return to the starting position *S*.

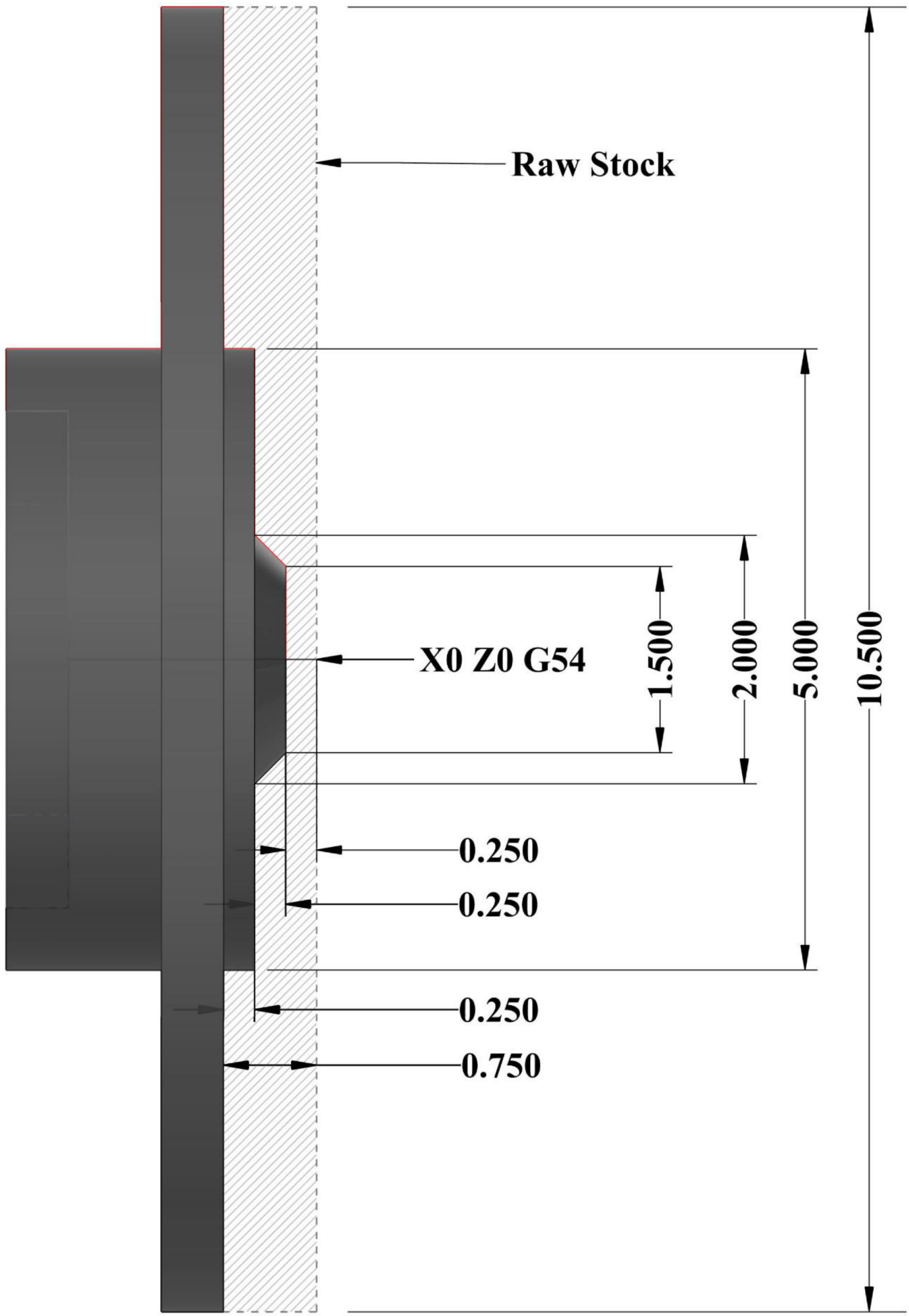
In the previous figure the start position *S* is the position of the tool at the time of the *G72* call. The *X* clearance plane is derived from the X-axis start position and the sum of *U* and optional *I* finish allowances.

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

_G72_ Address Relationships



G72 TOOL PATH EXERCISE



Tool Information for G72 wih G70 Finish			FEED: FPR
Tool 1		Tungaloy CNGM, HM Chipbreaker, BXM20 Grade, Carbide Turning Insert, .0310 CR	Rough Feed 0.01” Finish Feed 0.005”

**** Use the information above for feed.*
**** Start at **G54 X10.6 Z0.***
**** Leave **0.030”** in **X** and **0.015”** in **Z Axis** for finish pass*
**** See Sample program for assistance **(G72 Canned Cycle (O00001).nc)***

HAAS Simulator: Go to MDI Mode:	Save Program Referencing Page Number
G00 G18 G20 G40 G80 G99 (Safety Line)	___
T101	___
G50 S1500	___
G97 S300 M03	___
M08	___
G54 G00 Z___ X___	___
G96 S200	___
G72 P___ Q___ →	___
...	___
N___	___
...	___
N___	___
...	___
G70 P___ N___	___
...	
M30	
Run program in Graphic Mode.	

G73 IRREGULAR PATH STOCK REMOVAL CYCLE (GROUP 00)

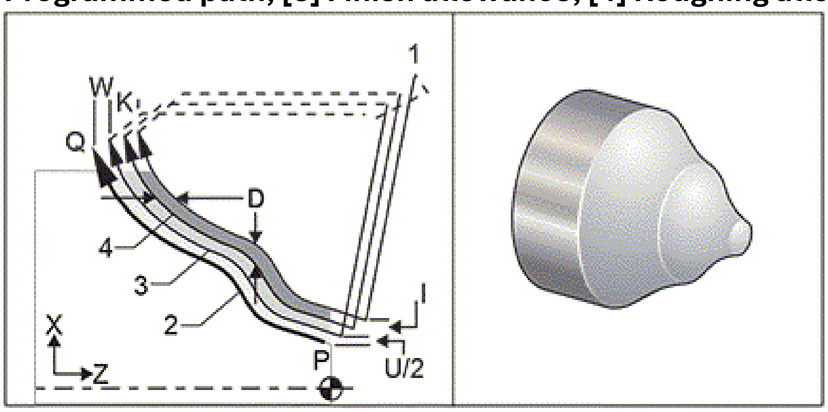
D - Number of cutting passes, positive integer
F - Feedrate in inches (mm) per minute (*G98*) or per revolution (*G99*) to use throughout *G73 PQ* block
I - X-axis distance and direction from first cut to last, radius
K - Z-axis distance and direction from first cut to last
P - Starting Block number of path to rough
Q - Ending Block number of path to rough
* **S** - Spindle speed to use throughout G73 PQ block
* **T** - Tool and offset to use throughout G73 PQ block
* **U** - X-axis size and direction of G73 finish allowance, diameter
* **W** - Z-axis size and direction of G73 finish allowance

* indicates optional

G18 Z-X plane must be active

NOTE The P values are modal. This means if you are in the middle of a canned cycle and a G04 Pnn or an M97 Pnn is used the P value will be used for the dwell / subprogram as well as the canned cycle.

G73 **Irregular Path Stock Removal: [P] Starting block, [Q] Ending block [1] Start position, [2] Programmed path, [3] Finish allowance, [4] Roughing allowance.**



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

Machining starts from the current position (*S*), and either rapids or feeds to the first rough cut. The nature of the approach move is based on whether a *G00* or *G01* is programmed in block *P*. Machining continues parallel to the programmed tool path. When block *Q* is reached a rapid departure move is executed to the Start position plus the offset for the second roughing pass. Roughing passes continue in this manner for the number of rough passes specified in *D*. After the last rough is completed, the tool returns to the starting position S.
Only *F*, *S* and *T* prior to or in the *G73* block are in effect. Any feed (*F*), spindle speed (*S*) or tool change (*T*) codes on the lines from *P* to *Q* are ignored.

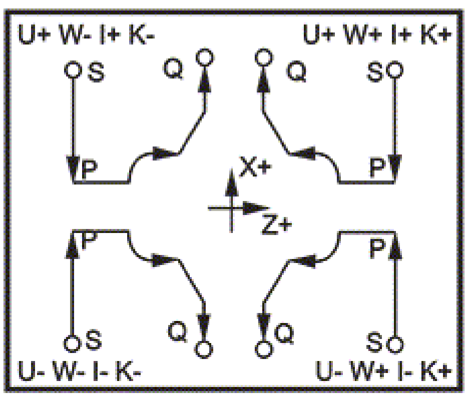
The offset of the first rough cut is determined by (*U*/2 + *I*) for the X Axis, and by (*W* + *K*) for the Z Axis. Each successive roughing pass moves incrementally closer to the final roughing finish pass by an amount of (*I*/(*D*- 1)) in the X Axis, and by an amount of (*K*/(*D*-1)) in the Z Axis.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

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

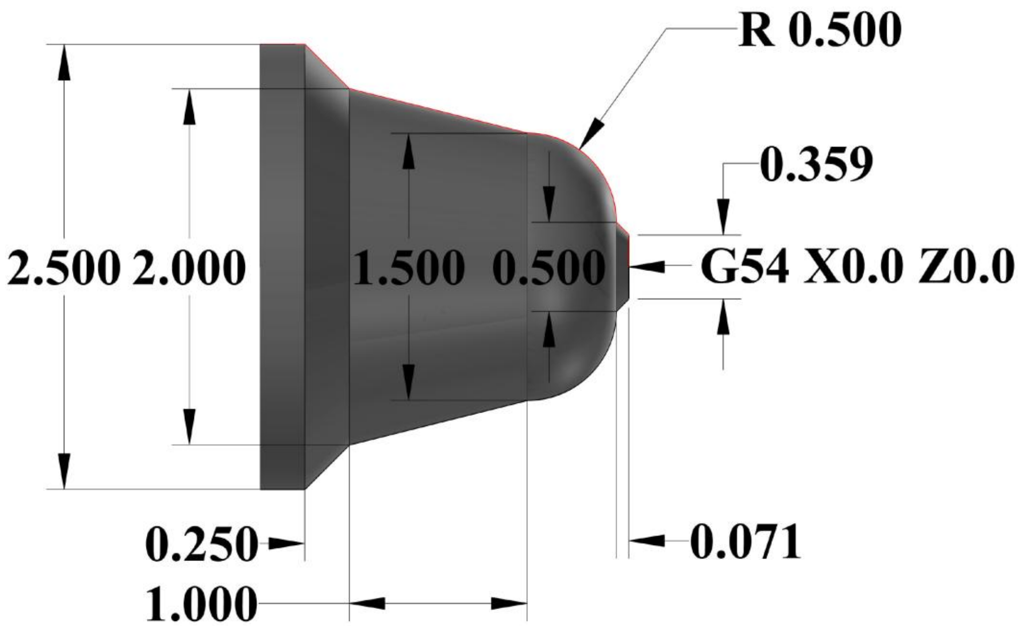
NOTE Monotonic curves are curves that tend to move in only one direction as x increases. A monotonic increasing curve always increases as x increases, i.e. f(a)>f(b) for all a>b. A monotonic decreasing curve always decreases as x increases, i.e. f(a)<f(b) for all a>b. The same sort of restrictions are also made for the monotonic non-decreasing and monotonic non-increasing curves.

The value of *D* must be a positive integral number. If the *D* value includes a decimal, an alarm is generated. The four quadrants of the *ZX* plane can be machined if the following signs for *U*, *I*, *W*, and *K* are used.

***G71* Address Relationships**



G73 TOOL PATH EXERCISE



Tool Information for G73 wih G70 Finish			FEED: FPR
Tool 1		Tungaloy CNGM, HM Chipbreaker, BXM20 Grade, Carbide Turning Insert, .0310 CR	Rough Feed 0.01” Finish Feed 0.005”

**** Use the information above for feed.*
**** Start at **G54 X0. Z0.***
**** Leave **0.030”** in **X** and **0.015”** in **Z Axis** for finish pass; U and W*
**** Remove 0.100” Materials in both I and K*
**** See Sample program for assistance **(G73 Canned Cycle (O00001).nc)***

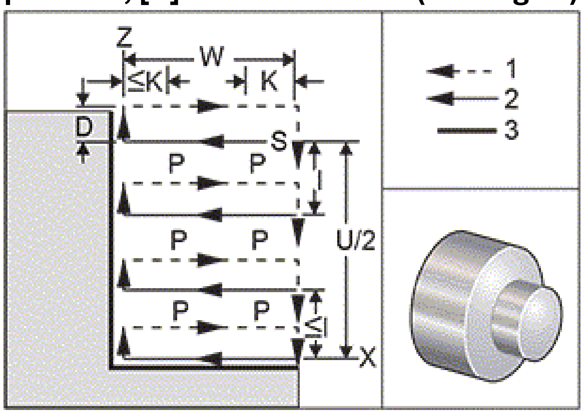
HAAS Simulator: Go to MDI Mode:	Save Program Referencing Page Number
G00 G18 G20 G40 G80 G99 (Safety Line)	
T101	
G50 S1500	
G97 S300 M03	
M08	
G54 G00 Z___ X___	
G96 S200	
G73 P___ Q___ →	
N___	
...	
N___	
G70 P___ N___	
...	
M30	
Run program in Graphic Mode.	

G74 END FACE GROOVING CYCLE (GROUP 00)

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

*indicates optional

***G74* End Face Grooving Cycle Peck Drilling: [1] Rapid, [2] Feed, [3] Programmed Path, [S] Start position, [P] Peck retraction (Setting 22).**



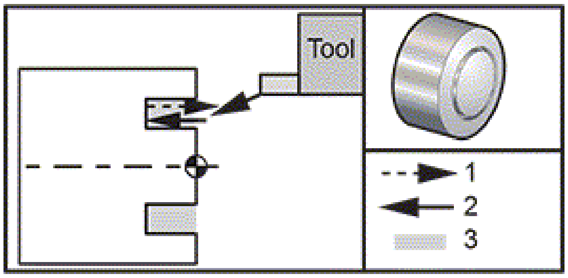
The *G74* canned cycle is used for grooving on the face of a part, peck drilling, or turning.

*****Warning**: The D code command is rarely used and should only be used if the wall on the outside of the groove does not exist like the figure above. The D code can be used in grooving and turning to provide a tool clearance shift, in the X axis, before returning in the Z axis to the C clearance point. But, if both sides to the groove exist during the shift, then the groove tool would break. So you wouldnt want to use the D command.

A minimum of two pecking cycles occur, if an *X*, or *U*, code is added to a *G74* block and *X* is not the current position. One at the current location and then at the *X* location. The *I* code is the incremental distance between X-Axis pecking cycles. Adding an *I* performs multiple 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 is less than *I*.

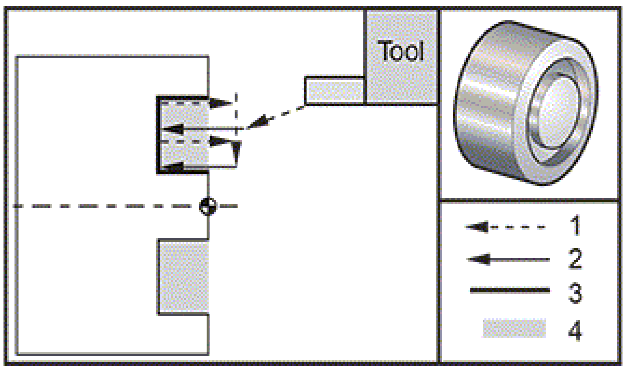
When *K* is added to a *G74* block, pecking is performed at each interval specified by *K*, the peck is a rapid move opposite the direction of feed with a distance defined by Setting 22. The *D* code can be used for grooving and turning to provide material clearance when returning to starting plane *S*.

_G74_ End Face Grooving Cycle: [1] Rapid, [2] Feed, [3] Groove.



***** See Sample Program on USB Drive; O60741.NC *****

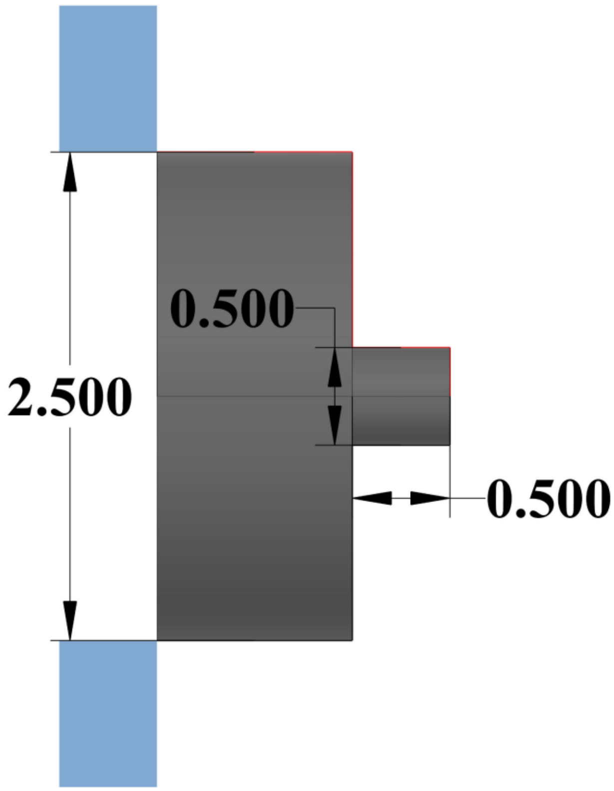
_G74_ End Face Grooving Cycle (Multiple Pass): [1] Rapid, [2] Feed, [3] Programmed path, [4] Groove.



***** See Sample Program on USB Drive; O60742.NC *****

Notes

G74 TOOL PATH EXERCISE



Tool Information for G74			FEED: FPR
Tool 1		Kennametal Carbide Grooving & Cut-Off Insert, .0080 CR, GUP Chipbreaker, KCU10 Grade, .1250 Groove Width, AlTiN Coated	0.01”

******* *Use the information above for feed.*
**** Start at* ***G54 X2.5 Z0.***
**** See Sample program for assistance* ***(G74 Canned Cycle (O00001).nc)***

HAAS Simulator: Go to MDI Mode:	Save Program Referencing Page Number
G00 G18 G20 G40 G80 G99 (Safety Line)	
T101	
G50 S1000	
G97 S500 M03	
M08	
G54 G00 Z___ X2.5 (Starting point)	
G96 S200	
G74 X___ Z___ →	
...	
...	
M30	
Run program in Graphic Mode.	

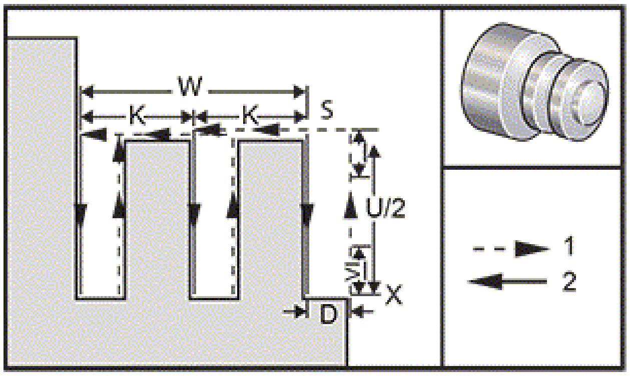
G75 O.D./I.D. GROOVING CYCLE (GROUP 00)

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

W - Z-axis incremental distance to furthest peck cycle
X - X-axis absolute location total pecking depth (diameter)
Z - Z-axis absolute location to furthest peck cycle

* indicates optional

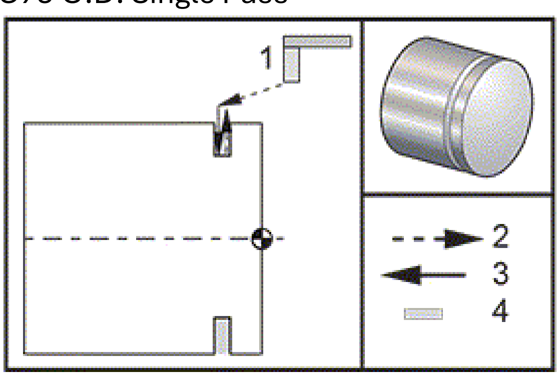
G75 O.D./I.D. Grooving Cycle: [1] Rapid, [2] Feed, [S] Start position.



The *G75* canned cycle can be used for grooving an outside diameter. When a *Z*, or *W*, code is added to a *G75* block and *Z* is not the current position, then a minimum of two pecking cycles 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* performs multiple, evenly spaced, grooves. If the distance between the starting position and the total depth (*Z*) is not evenly divisible by *K* then the last interval along *Z* is less than *K*.

NOTE	Chip clearance is defined by Setting 22.

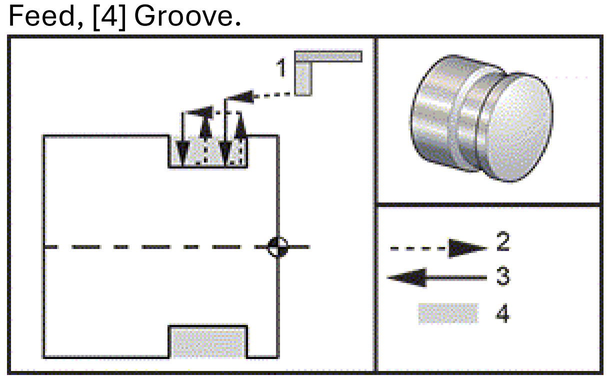
G75 O.D. Single Pass



***** See Sample Program on USB Drive; O60751.NC *****

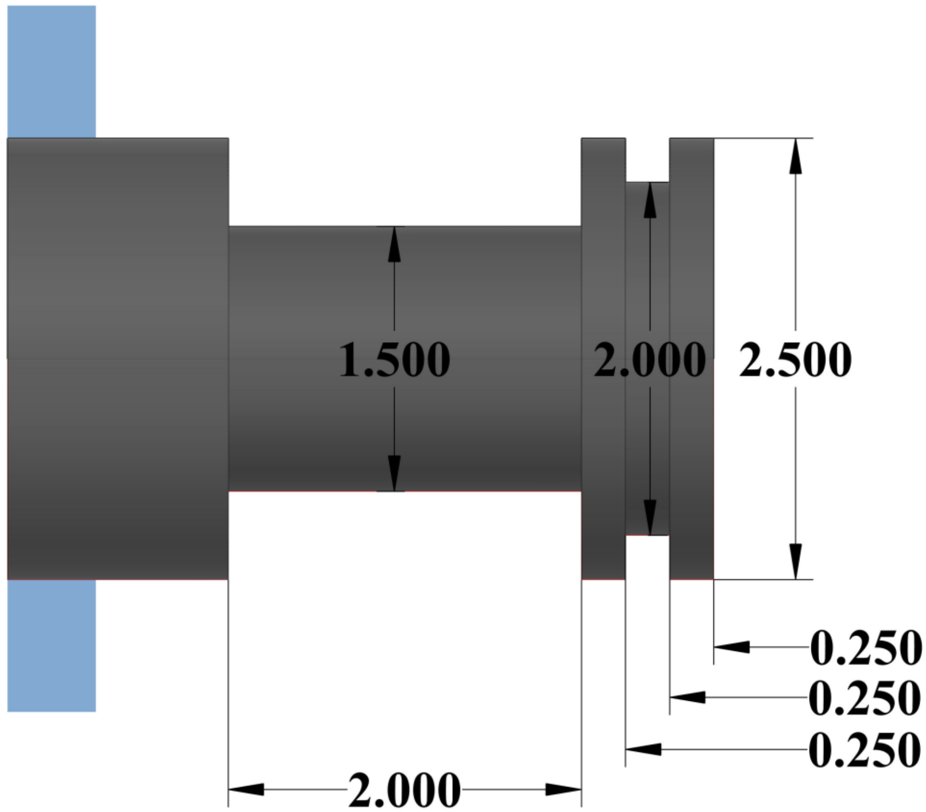
The following program is an example of a *G75* program (Multiple Pass):

G75 O.D. Multiple Pass: [1] Tool, [2] Rapid, [3] Feed, [4] Groove.



***** See Sample Program on USB Drive; O60752.NC *****

G75 TOOL PATH EXERCISE



Tool Information for G74			FEED: FPR
Tool 1		Kennametal Carbide Grooving & Cut-Off Insert, .0080 CR, GUP Chipbreaker, KCU10 Grade, .1250 Groove Width, AlTiN Coated	0.01"

*** *Use the information above for feed.*

*** *Start at **G54 X2.6 Z0.1***

*** ***2 G75s Operations***

*** *See Sample program for assistance **(G75 Canned Cycle (O00001).nc)***

HAAS Simulator: Go to MDI Mode:	Save Program Referencing Page Number
G00 G18 G20 G40 G80 G99 (Safety Line)	______
T101	______
G50 S1000	______
G97 S500 M03	______
M08	______
G54 G00 Z___ X___ (Starting point)	______
G96 S200	______
G75 X___ Z___ → (Front Grove)	______
...	______
G75 X___ Z___ →(Second Grove)	______
...	______
M30	______
	______
Run program in Graphic Mode.	______
	______

G76 THREADING CYCLE, MULTIPLE PASS (GROUP 00)

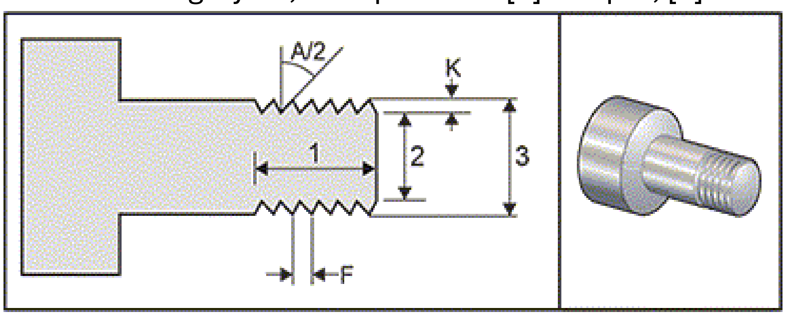
* **A** - Tool nose angle (value: 0 to 120 degrees) Do not use a decimal point
D - First pass cutting depth
F(E) - Feed rate, the lead of the thread
* **I** - Thread taper amount, radius measure
K - Thread height, defines thread depth, radius measure
* **P** - Single Edge Cutting (load constant)
* **Q**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

* indicates optional

NOTE

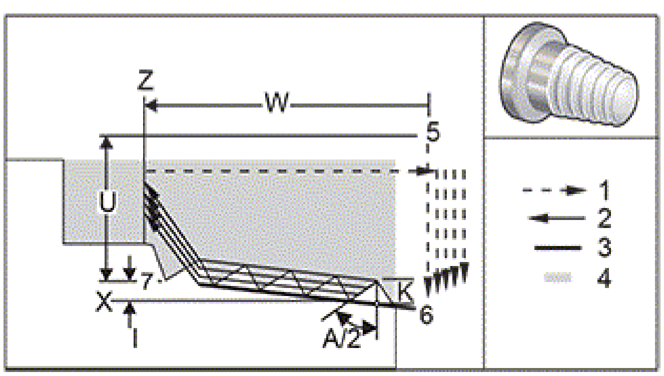
The P values are modal. This means if you are in the middle of a canned cycle and a G04 Pnn or an the canned cycle.

G76 Threading Cycle, Multiple Pass: [1] Z depth, [2] Minor diameter, [3] Major diameter.



Setting 95/Setting 96 determine chamfer size/angle; *M23/M24* turn chamfering ON/ OFF.

G76 Threading Cycle, Multiple Pass Tapered: [1] Rapid, [2] Feed, [3] Programmed path, [4] Cut allowance, [5] Start position, [6] Finished diameter, [7] Target, [A] Angle.



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

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

The thread taper amount is specified in *I*. Thread taper is measured from the target position *X*, *Z* at point [7] to position [6]. The I value is the difference in radial distance from the start to the end of the thread, not an angle.

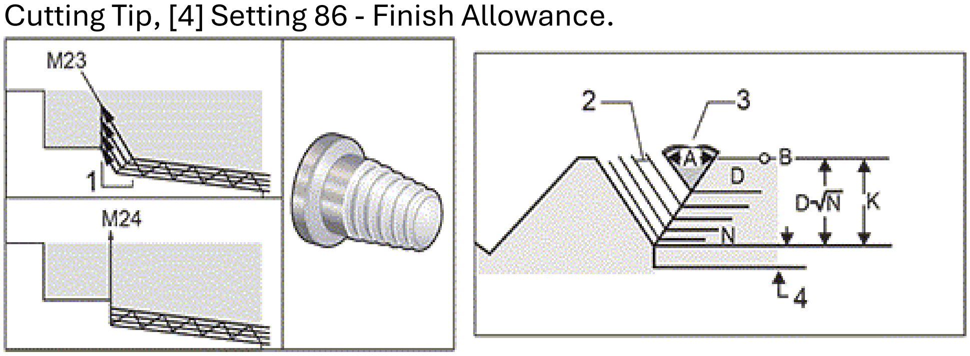
NOTE A conventional O.D. taper thread will have a negative I value.

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

The tool nose angle for the thread is specified in *A*. The value can range from 0 to 120 degrees. If *A* is not used, 0 degrees is assumed. To reduce chatter while threading use *A59* when cutting a 60 degree included thread.

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

At the end of the thread an optional chamfer is performed. The size and angle of the chamfer is controlled with Setting 95 (Thread Chamfer Size) and Setting 96 (Thread Chamfer Angle). The chamfer size is designated in number of threads, so that if 1.000 is recorded in Setting 95 and the feed rate is .05, then the chamfer will be .05. A chamfer can improve the appearance and functionality of threads that must be machined up to a shoulder. If relief is provided for at the end of the thread then the chamfer can be eliminated by specifying 0.000 for the chamfer size in Setting 95, or using *M24*. The default value for Setting 95 is 1.000 and the default angle for the thread (Setting 96) is 45 degrees.
G76 Using an *A* Value: [1] Setting 95 and 96 (see Note), [2] Setting 99 (Thread Minimum Cut), [3] Cutting Tip, [4] Setting 86 - Finish Allowance.



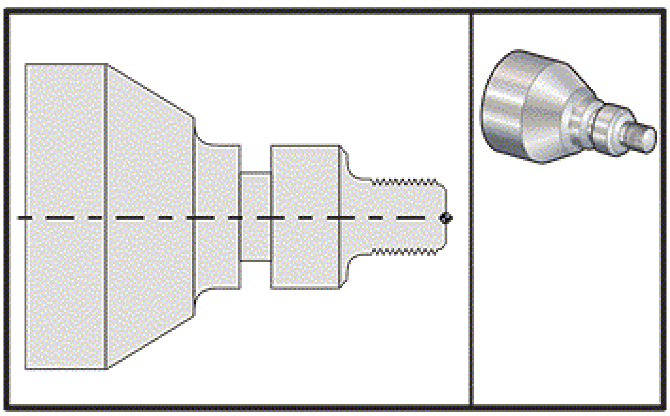
NOTE Setting 95 and 96 will affect the final chamfer size and angle.

Four options for *G76* Multiple Thread Cutting are available:

1. *P1*:Single edge cutting, cutting amount constant
2. *P2*:Double edge cutting, cutting amount constant
3. *P3*: Single edge cutting, cutting depth constant
4. *P4*: Double edge cutting, cutting depth constant

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

D specifies the depth of the first cut. Each successive cut is determined by the equation *D**sqrt(*N*) where *N* is the Nth pass along the thread. The leading edge of the cutter does all of the cutting. To calculate the *X* position of each pass you have to take the sum of all the previous passes, measured from the start point the X value of each pass
G76 Thread Cutting Cycle, Multiple Pass



***** See Sample Program on USB Drive; O60761.NC *****

UN Thread, External Thread Cutting

TPI	4	4,5	5	6	7	8	9	10	11	12	13	14	16	18	20	24	28	32
Total depth (inch)	.162	.143	.130	.107	.092	.082	.072	.065	.060	.055	.051	.047	.041	.037	.033	.028	.024	.021
Pass 1 (inch)	.019	.018	.017	.014	.014	.012	.011	.011	.011	.011	.010	.009	.009	.009	.008	.007	.007	.007
2	.017	.016	.016	.013	.013	.011	.010	.010	.010	.010	.009	.009	.008	.008	.007	.007	.006	.006
3	.016	.015	.014	.011	.010	.010	.008	.008	.008	.008	.007	.007	.006	.006	.006	.006	.004	.005
4	.014	.012	.012	.009	.009	.008	.007	.007	.007	.007	.007	.006	.006	.006	.005	.005	.004	.003
5	.013	.010	.010	.009	.008	.007	.007	.006	.006	.006	.006	.005	.005	.005	.004	.003	.003	
6	.011	.009	.009	.008	.007	.006	.006	.006	.006	.005	.005	.004	.004	.003	.003			
7	.010	.008	.008	.007	.007	.006	.006	.005	.005	.005	.004	.004	.003					
8	.010	.008	.008	.006	.006	.005	.005	.005	.004	.003	.003	.003						
9	.009	.007	.007	.006	.006	.005	.005	.004	.003									
10	.008	.007	.007	.006	.005	.005	.004	.003										
11	.007	.006	.007	.005	.004	.004	.003											
12	.007	.006	.006	.005	.003	.003												
13	.006	.006	.005	.004														
14	.006	.006	.004	.004														
15	.005	.005																
16	.004	.004																

UN Thread, Internal Thread Cutting

TPI	4	4,5	5	6	7	8	9	10	11	12	13	14	16	18	20	24	28	32
Total depth (inch)	.147	.131	.118	.097	.084	.074	.065	.059	.054	.049	.045	.042	.037	.033	.030	.025	.022	.019
Pass 1 (inch)	.017	.016	.017	.014	.013	.012	.011	.011	.011	.011	.010	.009	.009	.009	.008	.007	.007	.007
2	.016	.015	.015	.013	.013	.011	.010	.010	.009	.009	.008	.007	.007	.007	.006	.006	.005	.005
3	.015	.013	.013	.010	.009	.009	.007	.007	.007	.007	.006	.006	.006	.006	.005	.005	.004	.004
4	.013	.011	.011	.008	.008	.007	.006	.006	.006	.006	.005	.005	.005	.005	.004	.004	.003	.003
5	.011	.009	.009	.007	.007	.006	.006	.005	.005	.005	.005	.004	.004	.003	.004	.003	.003	
6	.009	.008	.008	.006	.006	.005	.005	.005	.005	.004	.004	.004	.003	.003	.003			
7	.009	.007	.007	.006	.006	.005	.005	.004	.004	.004	.004	.004	.003					
8	.008	.007	.007	.006	.005	.004	.004	.004	.004	.003	.003	.003						
9	.008	.007	.006	.005	.005	.004	.004	.004	.003									
10	.007	.006	.005	.005	.005	.004	.004	.003										
11	.007	.006	.005	.005	.004	.004	.003											
12	.006	.006	.006	.004	.003	.003												
13	.006	.006	.005	.004														
14	.006	.005	.004	.004														
15	.005	.005																
16	.004	.004																

Recommendations are for Steel Below 300 HB

Infeed Tables

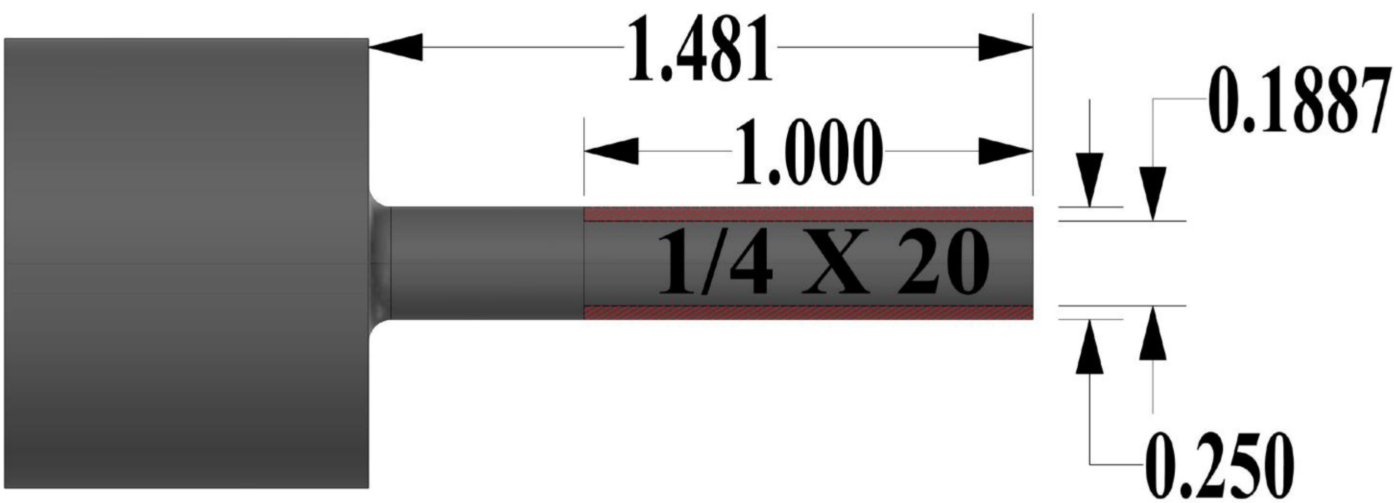
Metric ISO, External Thread Cutting

Pitch (mm)	6.0	5.5	5.0	4.5	4.0	3.5	3.0	2.5	2.0	1.75	1.5	1.25	1.0	0.75	0.50
Total depth (inch)	.151	.139	.126	.113	.100	.088	.076	.063	.049	.044	.037	.032	.026	.019	.014
Pass 1 (inch)	.018	.017	.016	.015	.003	.013	.011	.011	.009	.008	.008	.008	.008	.006	.005
2	.017	.016	.015	.013	.013	.012	.010	.009	.009	.008	.008	.007	.006	.006	.004
3	.014	.013	.013	.011	.010	.010	.008	.008	.007	.007	.007	.006	.005	.004	.003
4	.012	.011	.011	.009	.009	.008	.007	.007	.006	.006	.006	.004	.004	.003	.002
5	.011	.010	.010	.009	.008	.008	.007	.006	.006	.005	.005	.004	.003		
6	.010	.009	.009	.008	.007	.007	.006	.006	.005	.004	.003	.003			
7	.009	.008	.009	.008	.007	.006	.006	.005	.004	.003					
8	.009	.008	.008	.007	.006	.006	.005	.004	.003	.003					
9	.009	.007	.007	.007	.006	.006	.005	.004							
10	.008	.007	.007	.006	.005	.005	.004	.003							
11	.007	.007	.006	.006	.005	.004	.004								
12	.006	.006	.006	.005	.004	.003	.003								
13	.006	.006	.005	.005	.004										
14	.006	.005	.004	.004	.003										
15	.005	.005													
16	.004	.004													

Metric ISO, Internal Thread Cutting

Pitch (mm)	6.0	5.5	5.0	4.5	4.0	3.5	3.0	2.5	2.0	1.75	1.5	1.25	1.0	0.75	.50
Total depth (inch)	.139	.128	.117	.104	.092	.081	.070	.058	.045	.041	.033	.030	.024	.018	.012
Pass 1 (inch)	.018	.017	.017	.015	.013	.013	.011	.010	.009	.009	.008	.007	.007	.006	.004
2	.017	.016	.016	.013	.012	.012	.010	.010	.008	.007	.007	.007	.006	.005	.003
3	.014	.013	.013	.011	.009	.009	.008	.007	.007	.006	.006	.006	.004	.004	.003
4	.012	.010	.010	.009	.008	.007	.006	.006	.006	.005	.005	.004	.004	.003	.002
5	.010	.009	.009	.008	.007	.007	.006	.005	.005	.004	.004	.003	.003		
6	.009	.008	.008	.007	.007	.006	.005	.005	.004	.004	.003	.003			
7	.008	.007	.007	.006	.006	.006	.005	.004	.003	.003					
8	.007	.006	.006	.006	.005	.005	.004	.004	.003	.003					
9	.007	.006	.006	.006	.005	.005	.004	.004							
10	.006	.006	.006	.005	.005	.004	.004	.003							
11	.006	.006	.005	.005	.004	.004	.004								
12	.006	.006	.005	.005	.004	.003	.003								
13	.005	.005	.005	.004	.004										
14	.005	.005	.004	.004	.003										
15	.005	.004													
16	.004	.004													

Recommendations are for Steel Below 300 HB



Tool Information for G76		
Tool 1		Tungaloy 16, B Chipbreaker, AH725 Grade, Carbide Threading Insert, 48-8 TPI, 0.5-3mm Pitch, .0031 CR, .3750 IC, TiAlN Coated

**** See Sample program for assistance* ***(G76 Canned Cycle O60761.txt)***

G76 X(Minor) K (Thread Height) Z (Length of thread) D (Depth of First Cut) F (Feedrate)

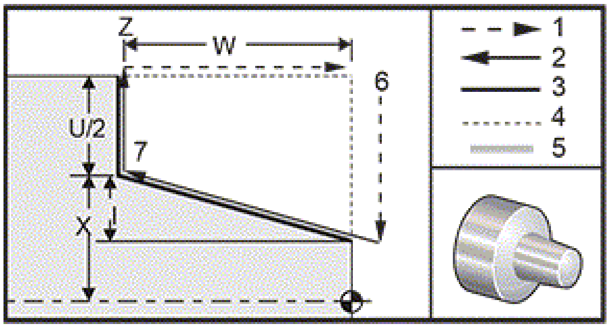
HAAS Simulator: Go to MDI Mode:	Save Program Referencing Page Number
G00 G18 G20 G40 G80 G99 (Safety Line)	
T101	
G50 S2000	
G97 S300 M03	
M08	
G54 G00 Z___ X2.6 (Starting point 0.200" in front of thread)	
G00 X__ → (Position X Axis 0.200 above Major Thread OD)	
G76 X___ F___ →	
...	
...	
M30	
Run program in Graphic Mode.	

G90 O.D./I.D. TURNING CYCLE (GROUP 01)

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

*indicates optional

G90 O.D./I.D. Turning Cycle: [1] Rapid, [2] Feed, [3] Programmed path, [4] Cut allowance, [5] Finish allowance, [6] Start position, [7] Target.

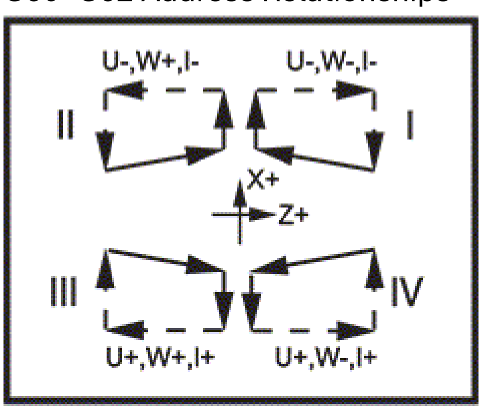


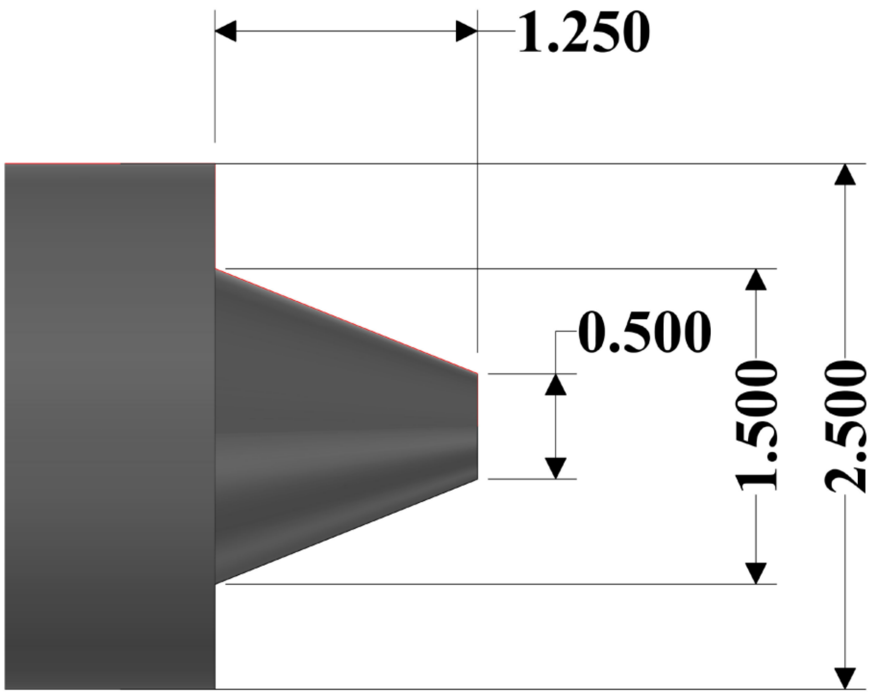
G90 is used for simple turning, however, multiple passes are possible by specifying the *X* locations of additional passes.

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

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

G90- G92 Address Relationships





Tool Information for G90			FEED: FPR
Tool 1		Tungaloy CNMG, HRM-R08 Chipbreaker, AH8015 Grade, Carbide Turning Insert, .0314 CR, .5000 IC, .1870 Thick, TiAlN Coated, Rhomboid Shape	0.01”

**** Use the information above for feed.*
**** See Sample program for assistance* ***(G90 Canned Cycle (O00001).nc)***

HAAS Simulator: Go to MDI Mode:	Save Program Referencing Page Number
G00 G18 G20 G40 G80 G99 (Safety Line)	______________________
T101	______________________
G50 S2000	______________________
G97 S500 M03	______________________
M08	______________________
G54 G00 Z___ X2.6 (Starting point)	______________________
G96 S200	______________________
G01 X2.0 Z0.0 F0.01→	______________________
G90 F0.01 I-0.5 Z___	______________________
…	______________________
…	______________________
M30	______________________
Run program in Graphic Mode.	______________________
	______________________

G92 THREADING CYCLE (GROUP 01)

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

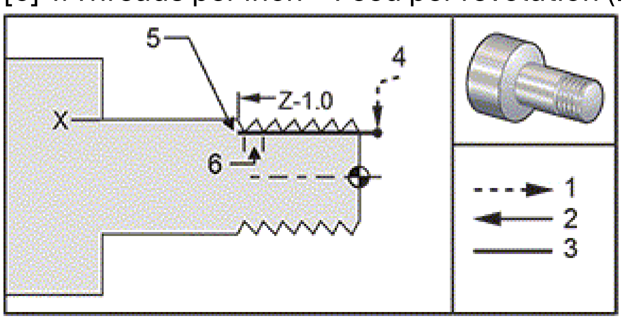
* indicates optional
Programming Notes:

- Setting 95/Setting 96 determine chamfer size/angle. *M23*/*M24* turn chamfering on/off.

- *G92* is used for simple threading, however, multiple passes for threading are possible by specifying the *X* locations of additional passes. Straight threads are made by specifying *X*, *Z*, and *F*. By adding an *I* value, a pipe or taper thread is cut. The amount of taper is referenced from the target. That is, *I* is added to the value of *X* at the target. At the end of the thread, an automatic chamfer is cut before reaching the target; default for this chamfer is one thread at 45 degrees. These values can be changed with Setting 95 and Setting 96.

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

G92 Threading Cycle: [1] Rapid, [2] Feed, [3] Programmed path, [4] Start position, [5] Minor diameter, [6] 1/Threads per inch = Feed per revolution (Inch formula; F = lead of thread) .

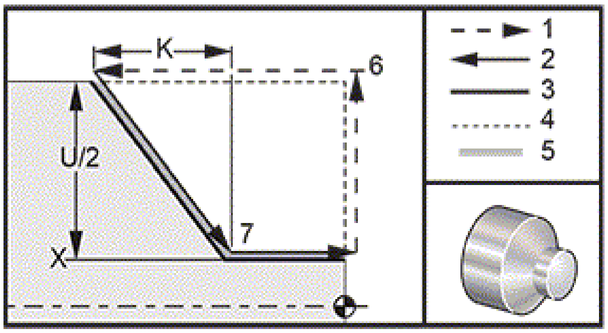


***** See Sample Program on USB Drive; O60921.NC *****

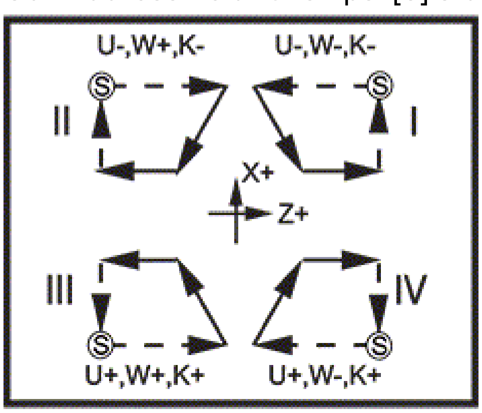
G94 END FACING CYCLE (GROUP 01)

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

*indicates optional
G94 End Facing Cycle: [1] Rapid, [2] Feed, [3] Programmed path, [4] Cut allowance, [5] Finish allowance, [6] Start position, [7] Target.



Straight end facing cuts can be made by specifying *X*, *Z* and *F*. By adding *K* a cone-shaped face is cut. The amount of coning is referenced from the target. That is *K* is added to the value of *X* at the target. Any of the four ZX quadrants is programmed by varying *U*, *W*, *X*, and *Z*. The coning is positive or negative. The following figure gives a few examples of the values required for machining in each of the four quadrants.
During incremental programming, the sign of the number following the *U* and *W* variables depends on the direction of the tool path. If the direction of a path along the X-axis is negative, the value of *U* is negative.
G94 Address Relationships: [S] Start position.

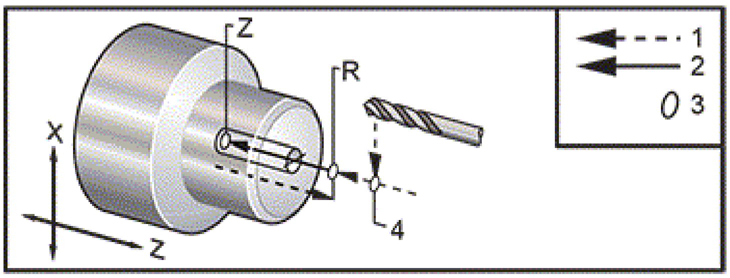


CANNED CYCLES G81 / G82 / G83 / G84 / G85 / G86

G81 DRILL CANNED CYCLE (GROUP 09)

* **C** - C-Axis absolute motion command (optional)
F - Feed Rate
* **L** - Number of repeats
R - Position of the R plane
* **X** - X-axis motion command
* **Y** - Y-axis absolute motion command
Z - Position of bottom of hole

* indicated optional
Also see *G241* for radial drilling and *G195/G196* for radial tapping with live tooling.
G81 Drill Canned Cycle: [1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Starting plane, [R] R plane, [Z] Position at the bottom of the hole.



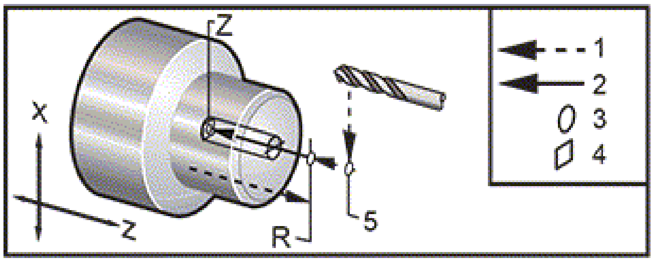
G82 SPOT DRILL CANNED CYCLE (GROUP 09)

* **C** - C-Axis absolute motion command (optional)
F - Feed Rate in inches (mm) per minute
* **L** - Number of repeats
P - The dwell time at the bottom of the hole
R - Position of the R plane
* **X** - X-axis motion command
* **Y** - Y-axis motion command
Z - Position of bottom of hole

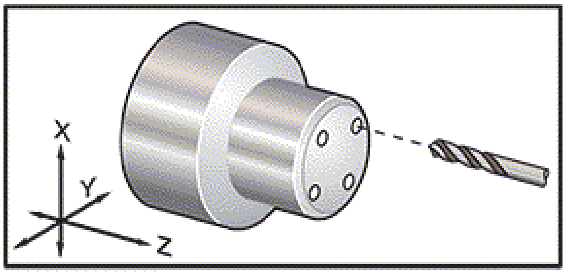
* indicates optional
This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X will cause this canned cycle to be executed.
Also, see *G242* for radial live tool spot drilling.

NOTE The P values are modal. This means if you are in the middle of a canned cycle and a G04 Pnn or an M97 Pnn is used the P value will be used for the dwell / subprogram as well as the canned cycle.

G82 Spot Drill Canned Cycle:[1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Dwell, [5] Starting plane, [R] R plane, [Z] Position of the bottom of the hole.



G82 Y-Axis Drill



***** See Sample Program on USB Drive; O60821.NC *****

To calculate how long you should dwell at the bottom of your spot drill cycle, use the following formula:

P = Dwell Revolutions x 60000/RPM

If you want the tool to dwell for two full revolutions at its full Z depth in the program above (running at 1500 RPM), you would calculate:

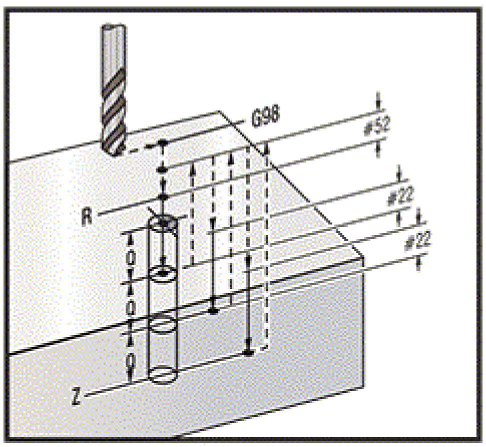
2 x 60000 / 1500 = 80

Enter *P80* (80 milliseconds or P.08 (.08 seconds) on the *G82* line, to dwell for 2 revolutions at 1500 RPM.

G83 NORMAL PECK DRILLING CANNED CYCLE (GROUP 09)

* **C** - C-Axis absolute motion command (optional)
F - Feed Rate in inches (mm) per minute
* **I** - Size of first cutting depth
* **J** - Amount to reduce cutting depth each pass
* **K** - Minimum depth of cut
* **L** - Number of repeats
* **P** - The dwell time at the bottom of the hole
* **Q** - The cut-in value, always incremental
* **R** - Position of the R plane
* **X** - X-axis motion command
* **Y** - Y-axis motion command
Z - Position of bottom of hole

* indicates optional
G83 Peck Drilling Canned Cycle: [1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Dwell, [#22] Setting 22, [#52] Setting 52.



NOTE

If I , J , and K are specified, a different operating mode is selected. The first pass will cut in the valu
cutting depth is K . Do not use a Q value when programming with I , J , and K .

Setting 52 changes the way *G83* works when it returns to the *R* plane. Usually the *R* plane is set well outside the cut to insure that the chip clearing motion allows the chips to clear the hole. However, this is wasted motion when first drilling through this empty space. If Setting 52 is set to the distance required to clear chips, the *R* plane can be put much closer to the part being drilled. When the clear move to *R* occurs, the *Z* will be moved past *R* by this value in Setting 52. Setting 22 is the amount to feed in *Z* to get back to the same point at which the retraction occurred.

***** See Sample Program on USB Drive; O60831.NC *****
***** See Sample Program on USB Drive; LIVE PECK DRILL - AXIAL.NC *****

G84 TAPPING CANNED CYCLE (GROUP 09)

F - Feed Rate
* **R** - Position of the R plane
S - RPM, called prior to G84
* **X** - X-axis motion command
* **Q** - Peck Depth (always incremental)
Z - Position of bottom of hole

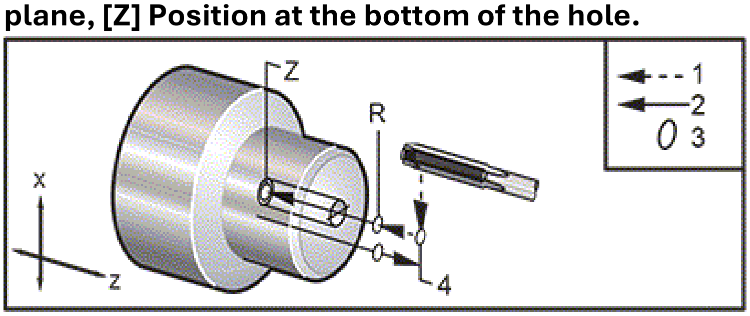
* indicates optional
Programming Notes:

- It is not necessary to start the spindle CW before this canned cycle. The control does this automatically.
- When *G84* tapping on a lathe, it is simplest to use *G99* Feed Per Revolution.
- The Lead is the distance traveled along a screw's axis, with each full revolution.
- The feedrate, when using *G99*, is equal to the Lead of the tap.
- An *S* value must be called prior to the *G84*. The *S* value determines the RPM of the tapping cycle.
- In Metric Mode (*G99*, with Setting 9 = MM), the feedrate is the metric equivalent of the lead, in MM.
- In Inch Mode (*G99*, with Setting 9 = INCH), the feedrate is the Inch equivalent of the lead, in inches.
- The lead (and *G99* feedrate) of an M10 x 1.0mm tap is 1.0mm, or .03937" (1.0/25.4=.03937).

Examples:

1. The lead of a 5/16-18 tap is 1.411 mm (1/18*25.4 = 1.411), or .0556" (1/18 = .0556)
2. This canned cycle can be used on the secondary spindle of a Dual Spindle DS lathe, when prefaced by a *G14*.
   Refer to the *G14* Secondary Spindle Swap for more information.
3. For Axial Live-Tool tapping, use a *G95* or *G186* command.
4. For Radial Live-Tool tapping, use a *G195* or *G196* command.
5. For Reverse Tapping (left-hand thread) on the Main or Secondary Spindle,
   refer to G184 Reverse Tapping Canned Cycle For Left Hand Threads (Gr... .

More programming examples, in both Inch and Metric, are shown below:
G84 **Tapping Canned Cycle: [1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Starting plane, [R] R plane, [Z] Position at the bottom of the hole.**



***** See Sample Program on USB Drive; O60841.NC *****
***** See Sample Program on USB Drive; O60842.NC *****
***** See Sample Program on USB Drive; O60843.NC *****
***** See Sample Program on USB Drive; O60844.NC *****

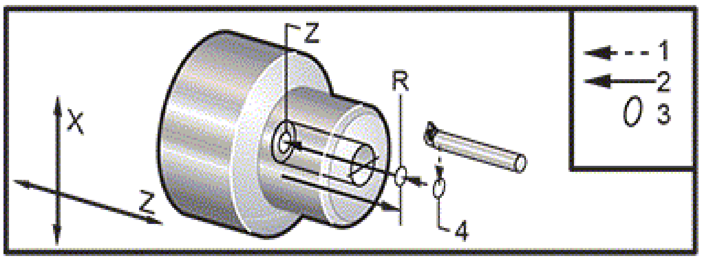
G85 BORING CANNED CYCLE (GROUP 09)

NOTE This cycle feeds in and feeds out.

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

* indicates optional

G85 Boring Canned Cycle: [1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Starting plane, [R] R plane, [Z] Position of the bottom of the hole.

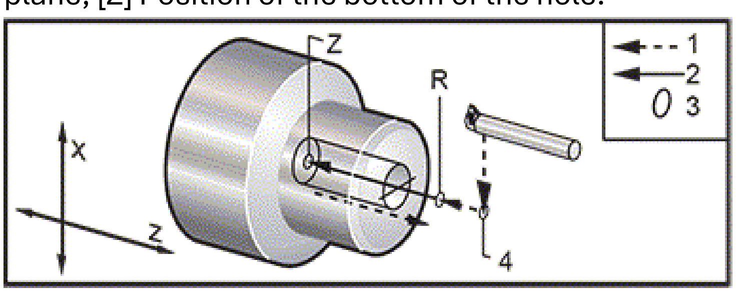


G86 BORE AND STOP CANNED CYCLE (GROUP 09)

NOTE The spindle stops and it rapids out of the hole.

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

* indicates optional
This G code stops the spindle once the tool reaches the bottom of the hole. The tool retracts once the spindle has stopped.
G86 Bore and Stop Canned Cycle: [1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Starting plane, [R] R plane, [Z] Position of the bottom of the hole.



SUBPROGRAMS

Subprograms:

- Are usually a series of commands that are repeated several times in a program.
- Are written in a separate program, instead of repeating the commands many times in the main program.
- Are called in the main program with an M97 or M98 and a P code.
- Can include an L for repeat count. The subprogram call repeats L times before the main program continues with the next block.

When you use M97:

- The P code (nnnnn) is the same as the block number (Nnnnnn) of the local subprogram.
- The subprogram must be within the main program

When you use M98:

- The P code (nnnnn) is the same as the program number (Onnnnn) of the subprogram.
- If the subprogram is not in memory, the file name must be Onnnnn.nc. The file name must contain the O, leading zeros and .nc for the machine to find the subprogram.
- The subprogram must reside in the active directory, or in a location specified in Settings 251/252. Refer to page 5 for more information on subprogram search locations.

Canned Cycles are the most common use of subprograms. For example, you might put the X and Y locations of a series of holes in a separate program. Then you can call that program as a subprogram with a canned cycle. Instead of writing the locations once for each tool, you write the locations only once for any number of tools.

SETTING UP SEARCH LOCATIONS

When program calls a subprogram, the control first looks for the subprogram in the active directory. If the control cannot find the subprogram, the control uses Settings 251 and 252 to determine where to look next. Refer to those settings for more information.

To build a list of search locations in Setting 252:

1. In the Device Manager (LIST PROGRAM), select the directory that you want to add to the list.
2. Press F3.
3. Highlight the SETTING 252 option in the menu, and then press ENTER.

   The control adds the current directory to the list of search locations in Setting 252.

Result:

To see the list of search locations, look at the values of Setting 252 on the Settings page.

LOCAL SUBPROGRAM (M97)

This code calls a subprogram (subprogram) referenced by a line number (N) within the same program. A Pnn code is required and must match a line number within the same program. This is useful for subprograms within a program as it does not require a separate program. The subprogram must end with an M99. An Lnn code in the M97 block will repeat the subprogram call nn times.

```
%
O69701 (M97 LOCAL SUBPROGRAM CALL) ;
M97 P1000 L2 (L2 will run the N1000 line twice) ;
M30 ;
N1000 G00 G55 X0 Z0 (N line that will run after M97 P1000 is run) ;
S500 M03 ;
G00 Z-.5 ;
G01 X.5 F100. ;
G03 ZI-.5 ;
G01 X0 ;
Z1. F50. ;
G28 U0 ;
G28 W0 ;
M99 ;
%
```

***** Program on USB Drive *****

EXTERNAL SUBPROGRAM (M98)

P - The subprogram number to run
L - Repeats the subprogram call (1-99) times.
(<PATH>) - The Subprogram's directory path

M98 calls a subprogram in the format M98 Pnnnn, where Pnnnn is the number of the program to call, or M98 (/Onnnnn), where is the device path that leads to the subprogram.

The subprogram must contain an M99 to return to the main program. You can add an Lnn count to the M98 block M98 to call the subprogram nn times before continuing to the next block.

When your program calls an M98 subprogram, the control looks for the subprogram in the main program's directory. If the control cannot find the subprogram, it then looks in the location specified in Setting 251. An alarm occurs if the control cannot find the subprogram.

M98 EXAMPLE:

The subprogram is a separate program (O00100) from the main program (O00002).

```
%
O00002 (PROGRAM NUMBER CALL);
M98 P100 L4 (CALLS O00100 SUB 4 TIMES) ;
M30 ;
%

%
O00100 (SUBPROGRAM);
M00 ;
M99 (RETURN TO MAIN PROGRAM) ;
%

%
O00002 (PATH CALL);
M98 (USB0/O00001.nc) L4 (CALLS O00100 SUB 4 TIMES) ;
M30 ;
%

%
O00100 (SUBPROGRAM);
M00 ;
M99 (RETURN TO MAIN PROGRAM) ;
%
```

MACROS INTRODUCTION

NOTE: This control feature is optional; call your HFO for information on how to purchase it.

Macros add capabilities and flexibility to the control that are not possible with standard G-code. Some possible uses are: families of parts, custom canned cycles, complex motions, and driving optional devices. The possibilities are almost endless.

A Macro is any routine/subprogram that you can run multiple times. A macro statement can assign a value to a variable, read a value from a variable, evaluate an expression, conditionally or unconditionally branch to another point within a program, or conditionally repeat some section of a program.

USEFUL G AND M CODES

M00, M01, M30 - Stop Program
G04 - Dwell
G65 Pxx - Macro subprogram call. Allows passing of variables.
M29 - Set output relay with M-FIN.
M129 - Set output relay with M-FIN.
M59 - Set output relay.
M69 - Clear output relay.
M96 Pxx Qxx - Conditional Local Branch when Discrete Input Signal is 0
M97 Pxx - Local Sub Routine Call
M98 Pxx - Sub Program Call
M99 - Sub Program Return or Loop
G103 - Block Lookahead Limit. No cutter comp allowed.
M109 - Interactive User Input

ROUND OFF

The control stores decimal numbers as binary values. As a result, numbers stored in variables can be off by 1 least significant digit. For example, the number 7 stored in macro variable #10000, may later be read as 7.000001, 7.000000, or 6.999999. If your statement was

IF [#10000 EQ 7]... ;

it may give a false reading. A safer way of programming this would be

IF [ROUND [#10000] EQ 7]... ;

This issue is usually a problem only when you store integers in macro variables where you do not expect to see a fractional part later.

LOOK-AHEAD

Look-ahead is a very important concept in macro programming. The control attempts to process as many lines as possible ahead of time in order to speed up processing. This includes the interpretation of macro variables. For example,

```
#12012 = 1 ;
G04 P1. ;
#12012 = 0 ;
```

This is intended to turn an output on, wait 1 second, and then turn it off. However, lookahead causes the output to turn on then immediately back off while the control processes the dwell. G103 P1 is used to limit lookahead to 1 block. To make this example work properly, modify it as follows:

```
G103 P1 (See the G-code section of the manual for a further explanation of G103) ;
;
#12012=1 ;
G04 P1. ;
;
;
;
#12012=0 ;
```

BLOCK LOOK-AHEAD AND BLOCK DELETE

The Haas control uses block look-ahead to read and prepare for blocks of code that come after the current block of code. This lets the control transition smoothly from one motion to the next. G103 limits how far ahead the control looks at blocks of code. The Pnnaddress code in G103 specifies how far ahead the control is allowed to look. For additional information, refer to G103 Limit Block Look-Ahead (Group 00)

Block Delete mode lets you selectively skip blocks of code. Use a / character at the beginning of the program blocks that you want to skip. Press BLOCK DELETE to enter the Block Delete mode. While Block Delete mode is active, the control does not execute the blocks marked with a / character. For example:

Using; **/M99 (Sub-Program Return)**
Before a block with; **M30 (Program End and Rewind)**

Makes the sub-program a main program when BLOCK DELETE is on. The program is used as a sub-program when Block Delete is off.

When a block delete token "/" is used, even if Block Delete mode is not active, the line will block look ahead. This is useful for debugging macro processing within NC programs.

LATHE G-CODES INTRODUCTION

This section gives detailed descriptions of the G-codes that you use to program your machine.

CAUTION: *The sample programs in this manual have been tested for accuracy, but they are for illustrative purposes only.* The programs do not define tools, offsets, or materials. They do not describe workholding or other fixturing. If you choose to run a sample program on your machine, do so in Graphics mode. Always follow safe machining practices when you run an unfamiliar program.

NOTE: The sample programs in this manual represent a very conservative programming style. The samples are intended to demonstrate safe and reliable programs, and they are not necessarily the fastest or most efficient way to operate a machine. The sample programs use G-codes that you might choose not to use in more efficient programs.

- **Lathe - G-Codes**

G-Code	Description	Group
G00	Rapid Motion Positioning	01
G01	Linear Interpolation Motion	01
G02	CW Circular Interpolation Motion	01
G03	CCW Circular Interpolation Motion	01
G04	Dwell	00
G09	Exact Stop	00
G10	Set Offsets	00
G12	Circular Pocket Milling CW	
G13	Circular Pocket Milling CCW	
G14	Secondary Spindle Swap	17
G15	Secondary Spindle Swap Cancel	17
G17	XY Plane	02
G18	XZ Plane	02
G19	YZ Plane	02
G20	Select Inches	06
G21	Select Metric	06
G28	Return To Machine Zero Point	00
G29	Return From Reference Point	00
G31	Skip Function	00
G32	Thread Cutting	01
G40	Tool Nose Compensation Cancel	07
G41	Tool Nose Compensation (TNC) Left	07
G42	Tool Nose Compensation (TNC) Right	07
G43	Tool Length Compensation + (Add)	08
G50	Spindle Speed Limit	00
G50	Set Global coordinate Offset FANUC	00
G52	Set Local Coordinate System FANUC	00
G53	Machine Coordinate Selection	00
G54	Coordinate System #1 FANUC	12
G55	Coordinate System #2 FANUC	12

G56	Coordinate System #3 FANUC	12
G57	Coordinate System #4 FANUC	12
G58	Coordinate System #5 FANUC	12
G59	Coordinate System #6 FANUC	12
G61	Exact Stop Modal	15
G64	Exact Stop Cancel G61	15
G65	Macro Subprogram Call Option	00
G68	Rotation	16
G69	Cancel G68 Rotation	16
G70	Finishing Cycle	00
G71	O.D./I.D. Stock Removal Cycle	00
G72	End Face Stock Removal Cycle	00
G73	Irregular Path Stock Removal Cycle	00
G74	End Face Grooving Cycle	00
G75	O.D./I.D. Grooving Cycle	00
G76	Threading Cycle, Multiple Pass	00
G80	Canned Cycle Cancel	09
G81	Drill Canned Cycle	09
G82	Spot Drill Canned Cycle	09
G83	Normal Peck Drilling Canned Cycle	09
G84	Tapping Canned Cycle	09
G85	Boring Canned Cycle	09
G86	Bore and Stop Canned Cycle	09
G89	Bore and Dwell Canned Cycle	09
G90	O.D./I.D. Turning Cycle	01
G92	Threading Cycle	01
G94	End Facing Cycle	01
G95	Live Tooling Rigid Tap (Face)	09
G96	Constant Surface Speed On	13
G97	Constant Surface Speed Off	13
G98	Feed Per Minute	10
G99	Feed Per Revolution	10
G100	Disable Mirror Image	00
G101	Enable Mirror Image	00
G103	Limit Block Lookahead	00
G105	Servo Bar Command	09
G107	G107 Cylindrical Mapping	00
G110	Coordinate System #7	12
G111	Coordinate System #8	12
G112	XY to XC Interpolation	04
G113	Cancel G112	04
G114	Coordinate System #9	12
G115	Coordinate System #10	12
G116	Coordinate System #11	12
G117	Coordinate System #12	12
G118	Coordinate System #13	12
G119	Coordinate System #14	12

G120	Coordinate System #15	12
G121	Coordinate System #16	12
G122	Coordinate System #17	12
G123	Coordinate System #18	12
G124	Coordinate System #19	12
G125	Coordinate System #20	12
G126	Coordinate System #21	12
G127	Coordinate System #22	12
G128	Coordinate System #23	12
G129	Coordinate System #24	12
G154	Select Work Coordinates P1-99	12
G156	Broaching Canned Cycle	09
G167	Modify Setting	00
G170	G170 Cancel G171/G172	20
G171	G171 Radius Programming Override	20
G172	G172 Diameter Programming Override	20
G184	Reverse Tapping Canned Cycle For Left Hand Threads	09
G186	Reverse Live Tool Rigid Tap (For Left Hand Threads)	09
G187	Accuracy Control	00
G195	Forward Live Tool Radial Tapping (Diameter)	09
G196	Reverse Live Tool Radial Tapping (Diameter)	09
G198	Disengage Synchronous Spindle Control	00
G199	Engage Synchronous Spindle Control	00
G200	Index on the Fly	00
G211	Manual Tool Setting	-
G212	Auto Tool Setting	-
G234	Tool Center Point Control (TCPC)	08
G241	Radial Drill Canned Cycle	09
G242	Radial Spot Drill Canned Cycle	09
G243	Radial Normal Peck Drilling Canned Cycle	09
G245	Radial Boring Canned Cycle	09
G246	Radial Bore and Stop Canned Cycle	09
G249	Radial Bore and Dwell Canned Cycle	09
G250	Cancel Scaling	11
G251	Scaling	11
G254	Dynamic Work Offset (DWO)	23
G255	Cancel Dynamic Work Offset (DWO)	23
G266	Visible Axes Linear Rapid %Motion	00
G268	Enable Feature Coordinate System	02
G269	Disable Feature Coordinate System	02
G390	Absolute Position Command	03
G391	Incremental Position Command	03

LATHE M-CODES INTRODUCTION

This page gives detailed descriptions of the M-codes that you use to program your machine.

CAUTION: The sample programs in this manual have been tested for accuracy, but they are for illustrative purposes only. The programs do not define tools, offsets, or materials. They do not describe workholding or other fixturing. If you choose to run a sample program on your machine, do so in Graphics mode. Always follow safe machining practices when you run an unfamiliar program.

NOTE: The sample programs in this manual represent a very conservative programming style. The samples are intended to demonstrate safe and reliable programs, and they are not necessarily the fastest or most efficient way to operate a machine. The sample programs use G-codes that you might choose not to use in more efficient programs.

M-codes are miscellaneous machine commands that do not command axis motion. The format for an M-code is the letter M followed by two to three digits; for example M03. Only one M-code is allowed per line of code. All M-codes take effect at the end of the block.

- **Lathe - M-Codes**

M- Code	Description
M00	Stop Program
M01	Optional Stop Program
M02	Program End
M03	Spindle On Fwd
M04	Spindle On Rev
M05	Spindle Stop
M08 / M09	Coolant On / Off
M10 / M11	Chuck Clamp / Unclamp
M12 / M13	Auto Jet Air Blast On / Off (Optional)
M14 / M15	Main Spindle Brake On /Off (Optional C-Axis)
M17	Turret Rotation Fwd
M18	Turret Rotation Rev
M19	Orient Spindle (Optional)
M21	Tailstock Advance (Optional)
M22	Tailstock Retract (Optional)
M23	Chamfer Out of Thread On
M24	Chamfer Out of Thread Off
M30	End of Program and Reset
M31	Chip Auger Forward (Optional)
M33	Chip Auger Stop (Optional)
M35	Parts Catcher Part-Off Position
M36	Parts Catcher On (Optional)
M37	Parts Catcher Off (Optional)
M38 / M39	Spindle Speed Variation On / Off
M41 / M42	Low / High Gear (Optional)
M43	Turret Unlock (Service Use Only)

M44	Turret Lock (Service Use Only)
M51 - M56	Turn On Built-In M-Code Relay
M59	Turn On Output Relay
M61 - M66	M61 - M66 Turn Off Built-In M-Code Relay
M69	Turn Off Output Relay
M78	Alarm if Skip Signal Found
M79	Alarm if Skip Signal Not Found
M85 / M86	Automatic Door Open / Close (Optional)
M88 / M89	High Pressure Coolant On / Off (Optional)
M90 / M91	Fixture Clamp Input On / Off
M95	Sleep Mode
M96	Jump If No Signal
M97	Local Subprogram Call
M98	Subprogram Call
M99	Subprogram Return Or Loop
M104 / M105	Probe Arm Extend / Retract (Optional)
M109	Interactive User Input
M110	Secondary Spindle Chuck Clamp (Optional)
M111	Secondary Spindle Chuck Unclamp (Optional)
M112 / M113	Secondary Spindle Air Blast On / Off (Optional)
M114 / M115	Secondary Spindle Brake On / Off (Optional)
M119	Secondary Spindle Orient (Optional)
M121- M126	M121 - M126 Built-In M-Codes Relays with M-Fin
M129	Turn On M-Code Relay with M-Fin
M130 / M131	Display Media / Cancel Display Media
M133	Live Tool Fwd (Optional)
M134	Live Tool Rev (Optional)
M135	Live Tool Stop (Optional)
M138	Spindle Speed Variation On
M139	Spindle Speed Variation Off
M143	Secondary Spindle Forward (Optional)
M144	Secondary Spindle Reverse (Optional)
M145	Secondary Spindle Stop (Optional)
M146 / M147	Steady Rest Clamp / Unclamp (Optional)
M158 / M159	Mist Condenser On/Off
M170 / M171	Engage 4th Axis Brake / Release 4th Axis Brake
M214 / M215	Live Tool Brake On/Off
M219	Live Tool Orient (Optional)
M299	APL / Part Load / or Program End
M300	M300 - APL/Robot Custom Sequence
M334 / M335	P-Cool Increment / P-Cool Decrement
M373 / M374	Tool Air Blash (TAB) On/OFF
M388 / M389	Through-Spindle Coolant On / Off

LATHE SETTINGS INTRODUCTION

This page gives detailed descriptions of the settings that control the way that your machine works. Inside the **SETTINGS** tab, the settings are organized into groups. Use the **[UP]** and **[DOWN]** cursor arrow keys to highlight a setting group. Press the **[RIGHT]** cursor arrow key to see the settings in a group, . Press the **[LEFT]** cursor arrow key to return to the setting group list.

To quickly access a single setting, make sure the **SETTINGS** tab is active, type the setting number, and then press **[F1]** or, if a setting is highlighted, press the **[DOWN]** cursor.

Some settings have numerical values that fit in a given range. To change the value of these settings, type the new value and press **[ENTER].** Other settings have specific available values that you select from a list. For these settings, use the **[RIGHT]** cursor to display the choices.

Press **[UP]** and **[DOWN]** to scroll through the options. Press **[ENTER]** to select the option.

- **Lathe - Settings**
- **Network Setup**
- **User Positions**

Setting Number	Description
1	Auto Power Off Timer
2	Power Off at M30
4	Graphics Rapid Path
5	Graphics Drill Point
6	Front Panel Lock
8	Prog Memory Lock
9	Dimensioning
10	Limit Rapid at 50%
17	Opt Stop Lock Out
18	Block Delete Lock Out
19	Feedrate Override Lock
20	Spindle Override Lock
21	Rapid Override Lock
22	Can Cycle Delta Z
23	9xxx Progs Edit Lock
28	Can Cycle Act w/o X/Y
29	G91 Non-modal
31	Reset Program Pointer
32	Coolant Override
39	Beep @ M00, M01, M02, M30
42	M00 After Tool Change
43	Cutter Comp Type
44	Min F Radius CC%
45	Mirror Image X Axis
46	Mirror Image Y Axis
47	Mirror Image Z Axis
52	G83 Retract Above R
53	Jog w/o Zero Return
56	M30 Restore Default G
57	Exact Stop Canned X-Y
58	Cutter Compensation
59	Probe Offset X+

60	Probe Offset X-
63	Tool Probe Width
64	Tool Offset Measure Uses Work
71	Default G51 Scaling
72	Default G68 Rotation
73	G68 Incremental Angle
74	9xxx Progs Trace
75	9xxx Progs Single BLK
77	Scale Integer F
80	Mirror Image B Axis
82	Language
83	M30/Resets Overrides
84	Tool Overload Action
85	Maximum Corner Rounding
87	Tool Change Resets Override
88	Reset Resets Override
90	Max Tools To Display
93	Tailstock X Clearance
94	Tailstock Z Clearance
95	Thread Chamfer Size
96	Thread Chamfer Angle
97	Tool Change Direction
99	Thread Minimum Cut
101	Feed Override -> Rapid
102	C Axis Diameter
103	Cyc Start/Fh Same Key
104	Jog Handle to SNGL BLK
105	TailStock Retract Distance
108	Quick Rotary G28
109	Warm-Up Time in Min.
110	Warmup X Distance
111	Warmup Y Distance
112	Warmup Z Distance
113	Tool Change Method
114	Conveyor Cycle Time (minutes)
115	Conveyor On-Time (minutes)
117	G143 Global Offset
118	M99 Bumps M30 Cntrs
119	Offset Lock
120	Macro Var Lock
130	Tap Retract Speed
131	Auto Door
133	Repeat Rigid Tap
142	Offset Chng Tolerance
143	Machine Data Collection Port
144	Feed Override -> Spindle
145	Tailstock At Part For Cycle Start

155	Load Pocket Tables
156	Save Offsets with Program
158	X Screw Thermal Comp%
159	Y Screw Thermal Comp%
160	Z Screw Thermal Comp%
162	Default To Float
163	Disable .1 Jog Rate
165	Ssv Variation (RPM)
166	Ssv Cycle
191	Default Smoothness
196	Conveyor Shutoff
197	Coolant Shutoff
199	Backlight Timer
216	Servo and Hydraulic Shutoff
232	G76 Default P Code
238	High Intensity Light Timer (minutes)
239	Worklight Off Timer (minutes)
240	Tool Life Warning
241	Tailstock Hold Force
242	Air Water Purge Interval
243	Air Water Purge On-Time
245	Hazardous Vibration Sensitivity
247	Simultaneous XYZ Motion in Tool Change
249	Enable Haas Startup Screen
250	Mirror Image C Axis
251	Subprogram Search Location
252	Custom Subprogram Search Location
253	Default Graphics Tool Width
261	DPRNT Store Location
262	DPRNT Destination File Path
263	DPRNT Port
264	Autofeed Step Up
265	Autofeed Step Down
266	Autofeed Minimum Override
267	Exit Jog Mode After Idle TIme
268	Second Home Position X
269	Second Home Position Y
270	Second Home Position Z
276	Workholding Input Monitor
277	Lubrication Cycle Interval
281	Chuck Foot Pedal Lock Out
282	Main Spindle Chuck Clamping
283	Chuck Unclamp RPM
284	Cycle Start Allowed With Chuck Unclamped
285	X Diameter Programming
286	Canned Cycle Cut Depth
287	Canned Cycle Retraction

289	Thread Finish Allowance
291	Main Spindle Speed Limit
292	Door Open Spindle Speed Limit
306	Minimum Chip Clear Time
313	Max User Tavel Limit X
314	Max User Travel Limit Y
315	Max User Travel Limit Z
319	VDI Spindle Center Line X
320	BOT Spindle Center Line X
321	Spindle Center Line Y
322	Foot Pedal Tailstock Alarm
323	Disable Notch Filter
325	Manual Mode Enabled
326	Graphics X Zero Location
327	Graphics Z Zero Location
328	eHandwheel Rapid Limit
329	Main Spindle Jog Speed
330	MultiBoot Selection Time out
331	Sub Spindle Jog Speed
332	Foot Pedal Lockout
333	Probe Offset Z+
334	Probe Offset Z-
335	Linear Rapid Mode
336	Bar Feeder Enable
337	Safe Tool Change Location X
338	Safe Tool Change Location Y
339	Safe Tool Change Location Z
340	Chuck Clamp Delay Time
341	Tailstock Rapid Position
342	Tailstock Advance Distance
343	Sub Spindle SSV Variation
344	Sub Spindle SSV Cycle
345	Sub Spindle Chuck Clamping
346	Sub Spindle Chuck Unclamp RPM
347	Live Tooling SSV Variation
348	Live Tooling SSV Cycle
349	Live Tooling Chuck Clamping
350	Live Tooling Chuck Unclamp RPM
352	Live Tooling Speed Limit
355	Sub Spindle Speed Limit
356	Beeper Volume
357	WarmUp Compensation Cycle Start Idle Time
358	Steady Rest Clamp/Unclamp Delay Time
359	SS Chuck Clamp Delay Time
360	Steady Rest Foot Pedal Lockout
361	Bar Pusher Vent Time
368	Live Tooling Type

372	Parts Loader Type
375	APL Gripper Type
376	Light Curtain Enable
377	Negative Work Offsets
378	Safe Zone Calibrated Geometry Reference Point X
379	Safe Zone Calibrated Geometry Reference Point Y
380	Safe Zone Calibrated Geometry Reference Point X
381	Enable Touchscreen
383	Table Row Size
396	Enable / Disable Virtual Keyboard
397	Press and Hold Delay
398	Header Height
399	Tab Height
403	Change Popup Button Size
409	Default Coolant Pressure
410	Safe Tool Change Location B
413	Main Spindle Load Type
414	Sub Spindle Load Type
416	Media Destination
417	Chuck Unclamp Delay Time
418	SS Chuck Unclamp Delay Time
421	General Orient Angle
422	Lock Graphics Plane
423	Help Text Icon Size
424	Mist Extractor Condenser Time Out

OTHER MANUALS

Interactive Manuals

Product	Lathe Operator's Manual Supplements	Service Manual
VMT-750	VMT- Interactive Operator's Manual Supplement	N/A
Haas Bar Feeder	Haas Bar Feeder - Interactive Operator's Manual Supplement	Haas Barfeeder - Interactive Service Manual
Lathe APL	Lathe - APL - Interactive Operator's Manual Supplement	Haas Automatic Parts Loader - Interactive Service Manual
Toolroom Lathe	Toolroom Lathe - Interactive Operator's Manual Supplement	N/A
Chucker Lathe	Chucker Lathe - Interactive Operator's Manual Supplement	N/A
Other Equipment	**Operator's Manual**	**Service Manual**
Autodoor	N/A	Autodoor - Interactive Service Manual
Haas Air Compressor	Haas Air Compressor - Operators/Service Manual	Haas Air Compressor - Operators/Service Manual
Haas Cobot Package	Haas Cobot Package - Operator's/Service Manual	Haas Cobot Package - Operator's/Service Manual
Haas Automatic Bandsaw	HAB - Operator's/Service Manual	HAB - Operator's/Service Manual
Haas Laser Engraver	Haas Laser Engraver - Operator's/Service Manual	Haas Laser Engraver - Operator's/Service Manual
Haas Spindle Chiller	Haas Spindle Chiller - Operator's/Service Manual	Haas Spindle Chiller - Operator's/Service Manual
Haas Robot Package	Haas Robot Package - Interactive Operator's Manual	Haas Robot Package - Interactive Service Manual
Haas Robot Pallet Loader	Haas Robot Pallet Loader - Operator's/Service Manual	Haas Robot Pallet Loader - Operator's/Service Manual
HSF-325	HSF-325 - Interactive Operator's/Service Manual	HSF-325 - Interactive Operator's/Service Manual
HSF-450	HSF-450 - Interactive Operator's/Service Manual	HSF-450 - Interactive Operator's/Service Manual
HTS400	HTS400 - Interactive Operator's/Service Manual	HTS400 - Interactive Operator's/Service Manual
Haas Tooling and Workholding	N/A	Haas Tooling and Workholding - Interactive Service Manual
Lubrication Systems	N/A	Lubrication Systems - Interactive Service Manual
Chip Removal and Coolant	N/A	Chip Removal and Coolant - Interactive Service Manual
WIPS and WIPS-L	WIPS - Interactive Operator's Manual Supplement	N/A
CAN Bus Systems	N/A	CAN Bus Systems - Interactive Service Manual