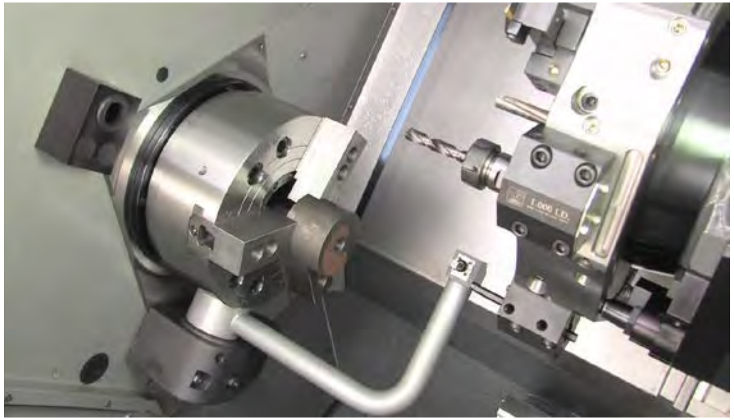
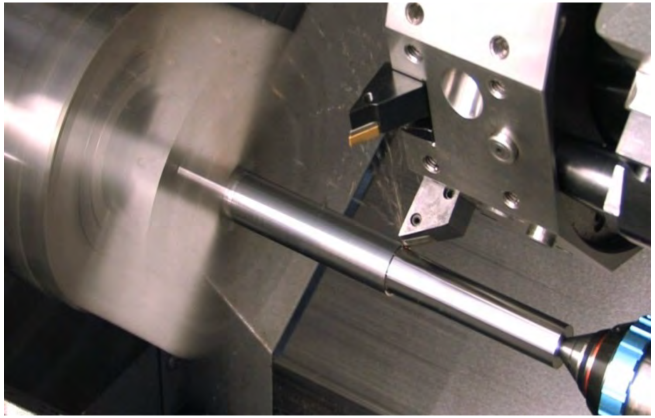
CNC Lathe Programming

Prof. Steven S. Saliterman
Introductory Medical Device Prototyping
Department of Biomedical Engineering, University of Minnesota
http://saliterman.umn.edu/

Haas ST-10 CNC Lathe





Images courtesy of Haas

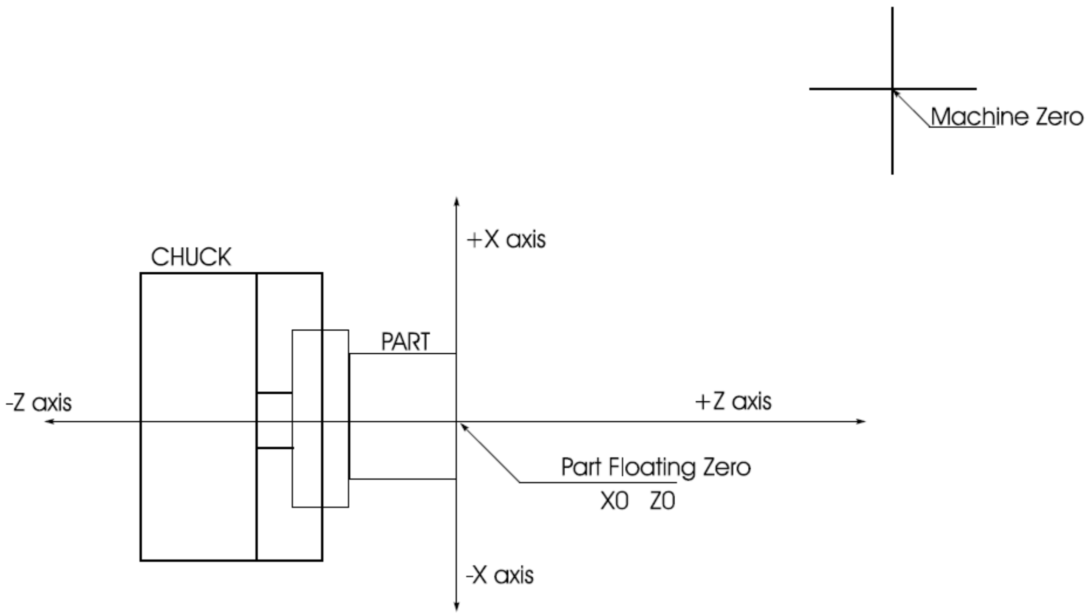
Safety Notice

- You must complete safety instruction before using tools and equipment in the Medical Device Center, ME Student Shop and CSE Workshops.
- All machinery can be dangerous. You must have a trained individual instruct you first when using unfamiliar equipment.
- Only authorized and trained individuals may operate CNC equipment.
- Code examples shown are for illustration purposes only, and are not meant for operation or programming actual equipment. They may be incomplete or contain errors.
- Always abide by shop safety instructions and never engage in horseplay.
- Remember to wear OSHA approved eye protection in the shop, short sleeves, leather or steel toed shoes, and secure long hair, avoid loose clothing, and take off rings, watches and bracelets when using power equipment.
- These slides are part of the "Introductory Medical Device Prototyping" course at the University of Minnesota, and are not meant for any other purpose.
- Formal training in Haas is available from Productivity, Inc.

CNC Lathe Axis

- X is the back to front motion, with the *part* X0 being coincident with the Z axis. *Determines diameter.*
- Z is the spindle axis, and the *part* Z0 is normally the front finished face. *Determines location of faces, shoulders and grooves.*
- *Home reference position* – movement at startup to extreme limits. A zero-return at POWER/STARTUP. This is the *machine* zero.
- We use a *floating zero* referred to as the *part zero* or *part origin*.
- We touch off the face and diameter, and store the offset from the machine zero in X and Z register of the *Tool Offsets Page*. (Not the same as mill which uses G54!)

Axis



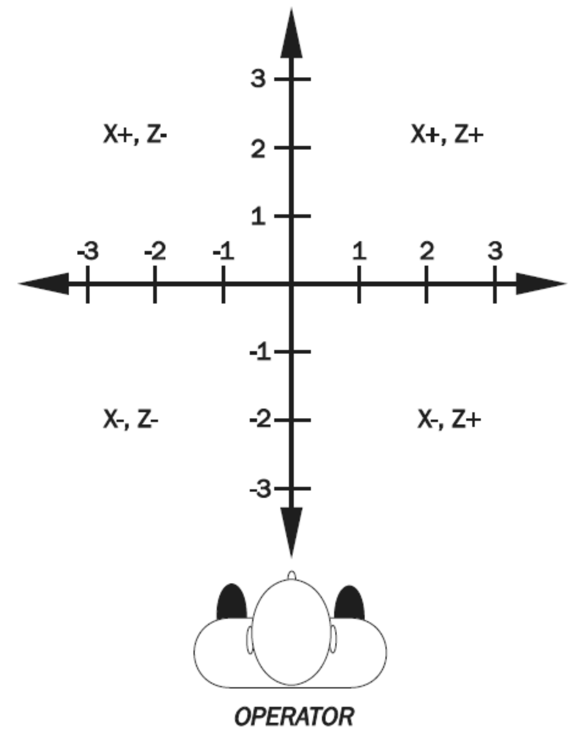


Image courtesy of Productivity and Haas

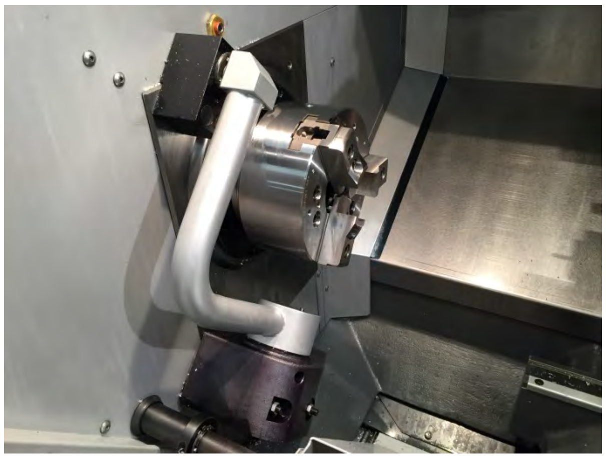
Axis

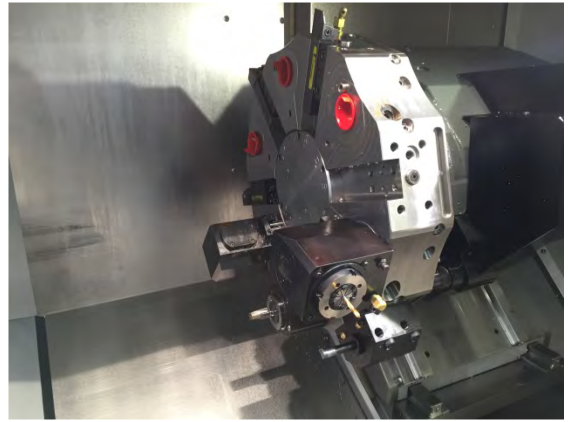
- Normally most of the X values are going to be positive in a part program. A negative X will occur only if you face a part past the centerline.
  - X values are in <u>diameters (not radius)</u>, hence a X1.0 will be moving the machine along the x-axis 0.5” in the positive direction.
- Z values will tend to be negative since zero is at the face. (Max. –1” travel beyond the *spindle* zero.)

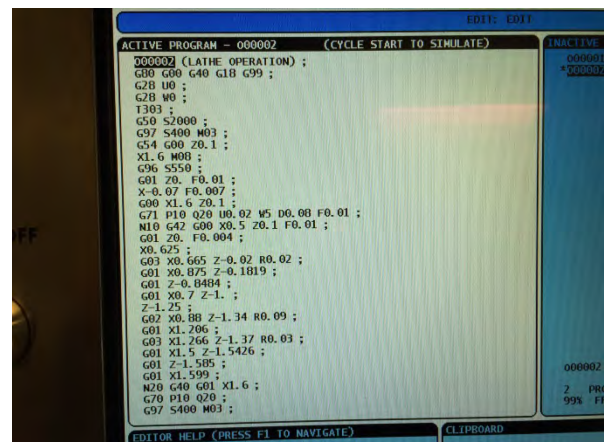
Absolute & Incremental Positioning

- Absolute positioning: X and Z codes are based on the Zero point of the part.
  - If a diameter of 1.0000 inches is needed, you input X1.0000.
  - If you are facing a shoulder 0.3” back from the face of the part, Z–3.0000 is input.
- Incremental positioning: U and W based on current position of the machine.
  - So a change of U–.5000 would be a smaller diameter of 0.5” from where the machine is presently at.
  - A grooving tool moving back ¾” behind the previous groove would be input as W–.7500.
  - Also called point to point.
- *Simultaneous* moves are possible e.g. G01 X2.000 W–.25 Moves X in absolute and Z in incremental at the same time.

Lathe Chuck and Tool Turret







Programs

- A block is a series of words on a single line ended with a ";" also known as the end-of-block (EOB) symbol.
  - Leading zeros and + signs are not needed.
  - Modal commands with G, X, Z,F, S, T and M need not be repeated in the following blocks unless a different word or change of value is needed.
  - Only one M code at the end of the block is permitted.
- Preparatory "G" codes make the tool do specific operations.
- "M" codes cause action to occur at the end of the block. Always executed last.
- N1-N9999 are optional sequence (line) numbers especially useful in macro subroutine.
- Programs begin and end with "%".
- The second line is the title of the program. Format is "O" followed by 5 digits.
- Enter comments by enclosing in parenthesis.
- A forward slash "/" denotes an optional block. BLOCK DELETE will skip these lines when running.

Address Letter Codes

A	Forth Axis
B	Linear B-axis motion
C	Fifth axis rotary motion
D	Canned cycle data
E	Feed rate
F	Feed rate
G	Type of operation, 0 to 255. G0x are non-modal, referring only to that block.
I, J, K	Canned cycle and circular optional data
L	Loop count for repeated cycles.
M	Control miscellaneous functions
N	Line number
O	Program number (name) Oxxxxx
P	Delay time
Q	Canned cycle optional data.
R	Canned cycle and circular optional data
S	Spindle speed (1-99999) No decimals allowed.
T	Tool selection Txxyy, where xx is tool location with respect to the turret position and yy selects the tool offset (1-50).
U	Incremental X axis motion
V	Optional macro parameter
W	Incremental Z axis motion
X	Linear X axis motion
Y	Linear Y-axis motion
Z	Linear Z-axis motion

M Codes (One per Block)

M00	Program stop (spindle, axes, coolant)
M03	Start spindle clockwise
M04	Spindle counterclockwise
M05	Spindle stop
M08	Coolant on
M09	Coolant off
M10	Clamp spindle chuck
M11	Unclamp spindle chuck
M21	Tailstock forward
M22	Reverse tailstock
M23	Thread chamfer (G76 or G92)
M24	Thread chamfer off
M30	Program end and rewind
M41	Low gear
M42	High gear
M85	Automatic door open
M86	Automatic door close
M88	High pressure coolant off
M97	Local sub-Program call (P or L)
M98	Sub-program call (P or L)
M99	Return for subprogram or loop

Machine Defaults

- On power up the machine will go to part zero entered into G54 from prior probing.
- Automatic G Codes
  - G00 Rapid traverse
  - G18 X, Y circular plane section
  - G40 Cutter compensation cancel
  - G54 Work coordinate Zero #1 (1 of 26 available)
  - G64 Exact stop cancel
  - G97 Constant surface speed cancel
  - G99 Feed per revolution
- Safety block used in mill generally not needed.
  - G18 G20 G40 G54 G80 G97 G99

Sample Program

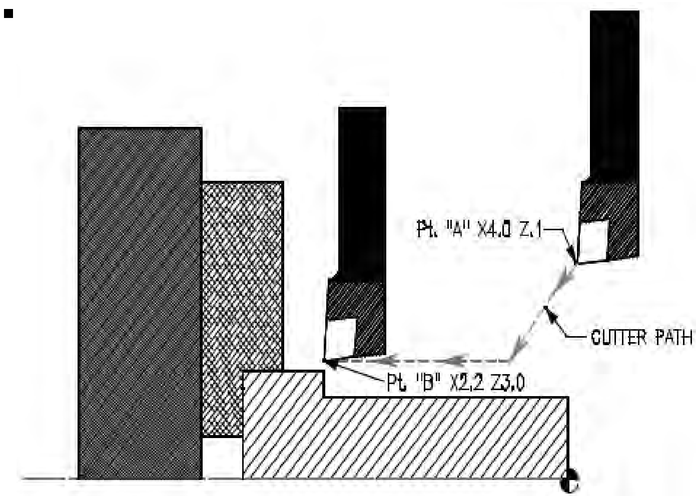
```
G28;                          (Rapid all axis machine zero)
G53 G00 X-3. Z-4.;            (Safe locate tool change turret)
T101;                         (Indexes turret to tool)
G50 S2000;                    (Spindle speed max. rpm)
G97 S1146 M03;                (Cancel surface speed mode;
                                spindle speed; clockwise; rpm)
G54 G00 X1.5 Z.02 M08;        (Rapid movement to start X, Z;
                                coolant on)
G96 S450;                     (Constant surface speed;
                                surface feet per minute (SFM))
G00 Z1. M09;                  (Move from part. coolant off)
G28;                          (Rapids to machine home)
G53 G00 X-3. Z-4. T0;         (Back to tool change location)
M30;                          (End of program. Stop spindle, turn
                              off coolant, cancel tool length offsets)
```

Spindle & Feed Codes

- M03 Spindle Forward
- M04 Spindle Reverse
- M05 Spindle Stop
- G50 Maximum spindle speed (RPM – revolutions per min.)
- G96 Constant surface speed ON (SFM),
  - Tool speed increases automatically as diameter decreases.
  - SFM $= 0.2618\ x\ Diameter\ x\ RPM$
- G97 Constant surface speed OFF (DEFAULT) & RPM ON,
  - RPM $= \frac{3.82\ x\ SFM}{Diameter}$
- G98 Feed (inches) per minute, IPM = Current RPM x IPR
- G99 Feed (inches) per revolution(DEFAULT), IRP = IPM/Current RPM

G00 Rapid Motion Positioning

- A move from point A to B can be done by positioning with absolute, incremental or both.



G00 X2.2 Z-3.0
or **G00** U-1.8 W-3.1
or **G00** X2.2 W-3.1
or **G00** U-1.8 Z-3.0

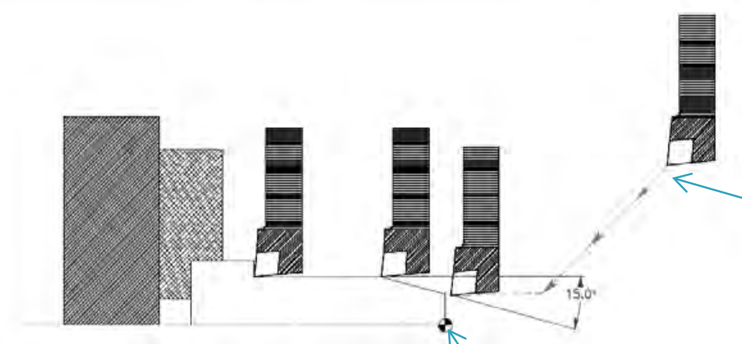
Image courtesy of Haas

G01 Linear Interpolation

- Straight line (linear) point to point motion.
  - Both X and Z axis start and stop at the same time.
  - Specified feed rate (IPR) is maintained along the line.
  - May be made in ABSOLUTE or INCREMENTAL commands.
  - Optional ways of programming:

G00 X0. 9106 Z0.1 (ABSOLUTE) G01 X1.5 Z-1. F0.006 Z-3 X2.	G00 X0. 9106 Z0.1 (INCREMENTAL) G01 U.5894 W-1.1. F0.006 W-1.0 U0.5	G00 X0. 9105 Z0.1 (ABS. AND INC.) G01 X1.5 W-1.1 F0.006 Z-3 U0.5

Back-off diameter

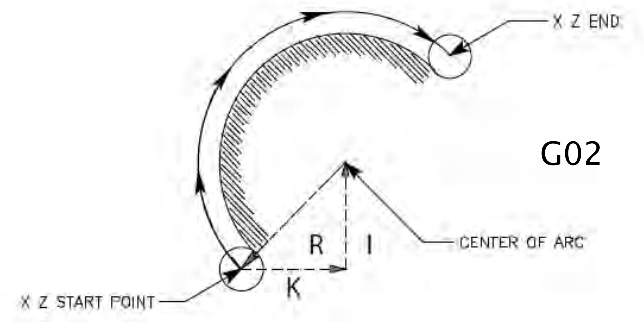


Machine home

Part zero

Image courtesy of Haas

G02(CW) & G03 (CCW) Circular Commands



- X Absolute arc *end point.*
- Y Absolute arc *end point.*
- U Incremental arc *end point.*
- W Incremental arc *end point.*
- I Distance from *start point* to *arc center* X axis.
- K Distance from *start point* to *arc center* Z axis.
- R Radius of circle (if I and K not used).
- F Feed rate (IPR).

Image courtesy of Haas

What's needed..

Plane selection command Plane selection command Plane selection command	G17 G18 G19	Arc parallel to XY-plane (not available) Arc parallel to ZX-plane (default) Arc parallel to YZ-plane (not available)
Arc start position coordinates	X,Z	Coordinates to the start position of arc
Rotation direction	G02 G03	Clockwise interpolation direction Counterclockwise interpolation direction
Arc end position Absolute or Arc end position Incremental	X,Z U,W	Coordinates of the end position of arc defined from the part zero or part origin Incremental distance and direction from start Point of arc to end point of arc in X and Z-axes
I and K method (arc center coordinates) or R method (arc radius)	I,K R	Incremental distance and direction from start position of arc to the arc center for X and Z-axes. "I" is for the X-axis; "K" is for the Z-axis. "I" is a radial value. Arc radius value of arc.
	−R	For circular path over 180 degrees.

Example G02…

```
…
G00 X0. Z.1          (rapids to X0 Z.1)
G01 Z0. F.012        (feeds to face)
X2.                  (feeds to x axis 2” dia.)
Z-1.                 (feeds to start of arc)
G02 X5. Z-2.5 R1.5   (cuts arc)
G01 X6.0             (moves out of way)
…
```

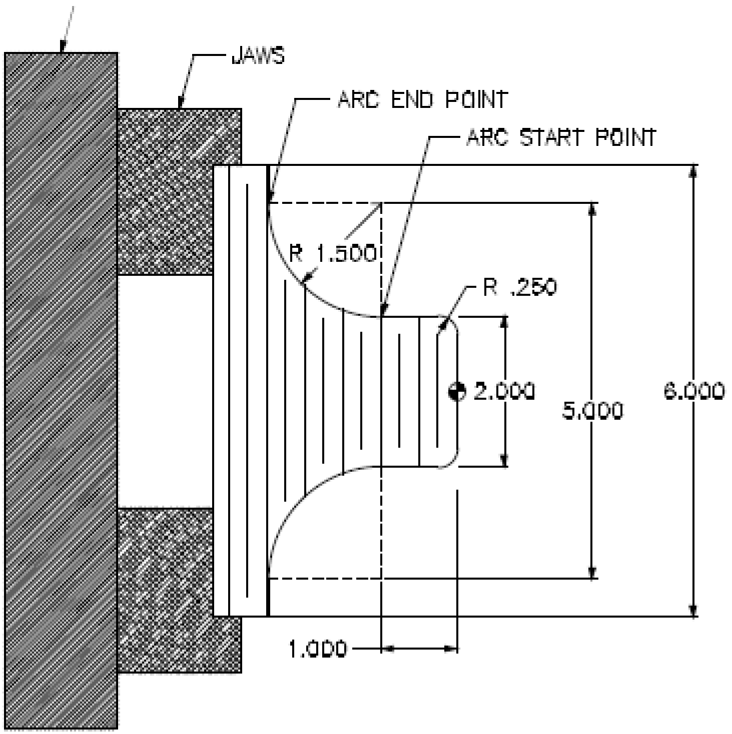
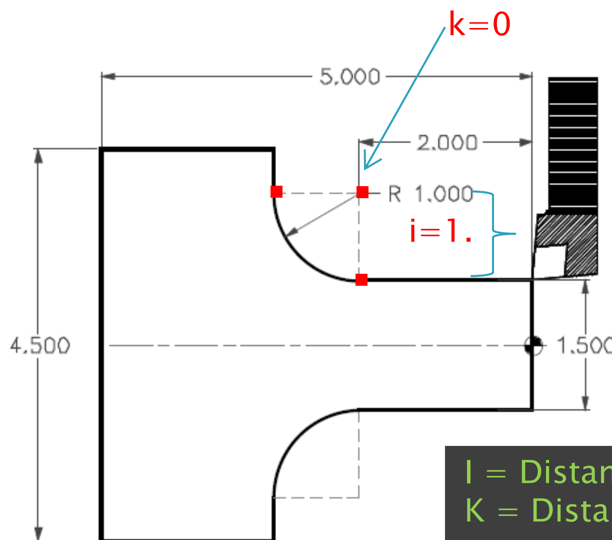


Image courtesy of Haas

Example Exercise G02…

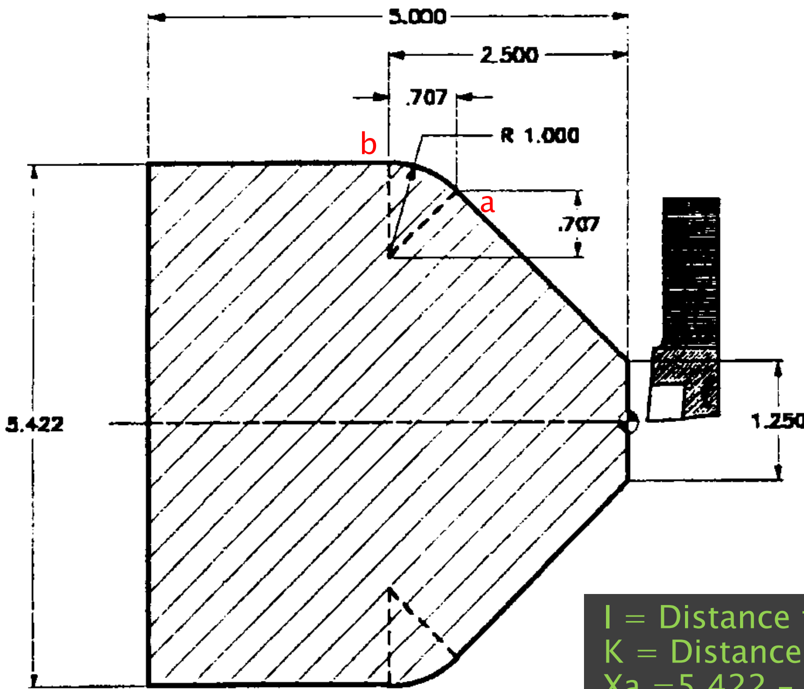


I = Distance from start point to arc center X axis = 1
K = Distance from start point to arc center Z axis = 0

N9 G00 X1.5 Z.100
N10 G01 Z-_2.____ F.01
N11 G_02_ X_3.5__ Z_-3.___ I_1.___ K_0___
N12 G_01_ X_5.___

Image courtesy of Haas

Example Exercise G03…



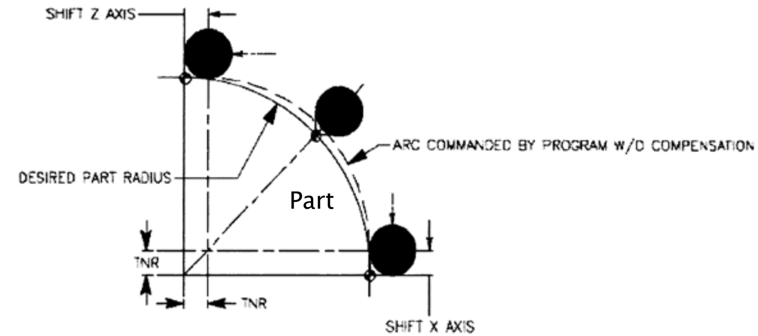
N39 G00 X0 Z.100
N40 G01 Z___0.___ F.01
N41 G01___ X_1.25___
N42 G01 X__4.836__ Z_1.793_____
N43 G_03___ X_5.422_ Z_2.5____ I_.707___
K_.707_____
N44 G 01 Z___5.____

I = Distance from start point to arc center X axis =.707
K = Distance from start point to arc center Z axis = 0
Xa =5.422 – 2 x (1 – .707) = 4.836
Za =2.5 – .707 = 1.793
Xb = 5.422
Zb = 2.5

Image courtesy of Haas

Tool Nose Compensation

- An external radius (chamfer) has the *Tool Nose Radius (TNR)* added to it.



- An internal radius (fillet) has the *Tool Nose Radius* subtracted from it.

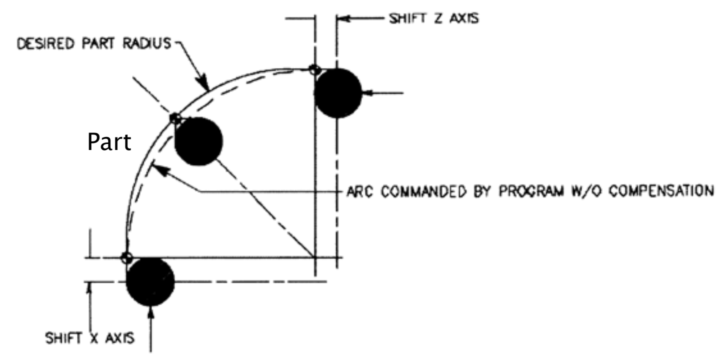


Image courtesy of Haas

Adjusting Radius for Tool Nose

- External Radius Calculation (Add the Tool Tip Nose Radius)
  - If a radius of .25 is required on the external corner of a part and the tool nose radius is .031 the programmed radius will be: .25 + .031 = .281
  - If the radius is on a 1.0 diameter and at the ZO face of the part the program will follow the example below:

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

- Internal Radius Calculation (Subtract the Tool Nose Radius)
  - If a radius of .25 is required on the internal corner of a part and the tool nose radius is .031 the programmed radius will be: .25 – .031 =.219
  - If the radius is on a 1.0 diameter and at the ZO face of the part the program will follow the example below:

```
G1 X1. Z0
G1 Z-.781
G2 X 1.438 Z-1. R.219
G1X-?
```

- Note: When calculating the end points in a part program you must double the X axis calculation to allow for both sides of the part.

Prof. Steven S. Saliterman

Example G02 and G03…

G01 Z0. F0.01	
X.948 F.006	$(1.24-2R_1')$
G03 X1.25 Z-.151 R .151	$(0 - R_1')$
G01 Z-.921	$(1.13-R_2')$
G02 X1.668 Z-1.13 R.20	$(1.25+2R_2')$
G01 X2.218	$(3 - 2R_3')$
G03 X3. Z-1.521 R.391	$(1.13 + R_3')$
G01 Z-2.375	
G00 U.01 Z1.0	
G28	

Lathe tool has a .031 TNR

$R_1' = .120+.031=.151$

$R_2' = .240-.031=.209$

$R_3' = .360+.031=.391$

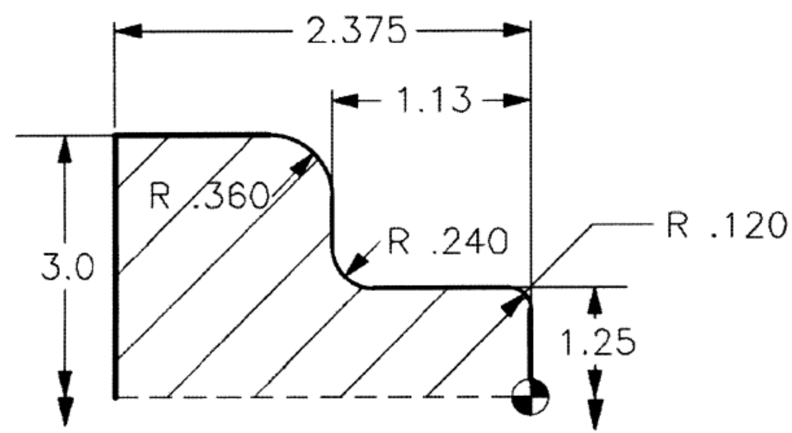
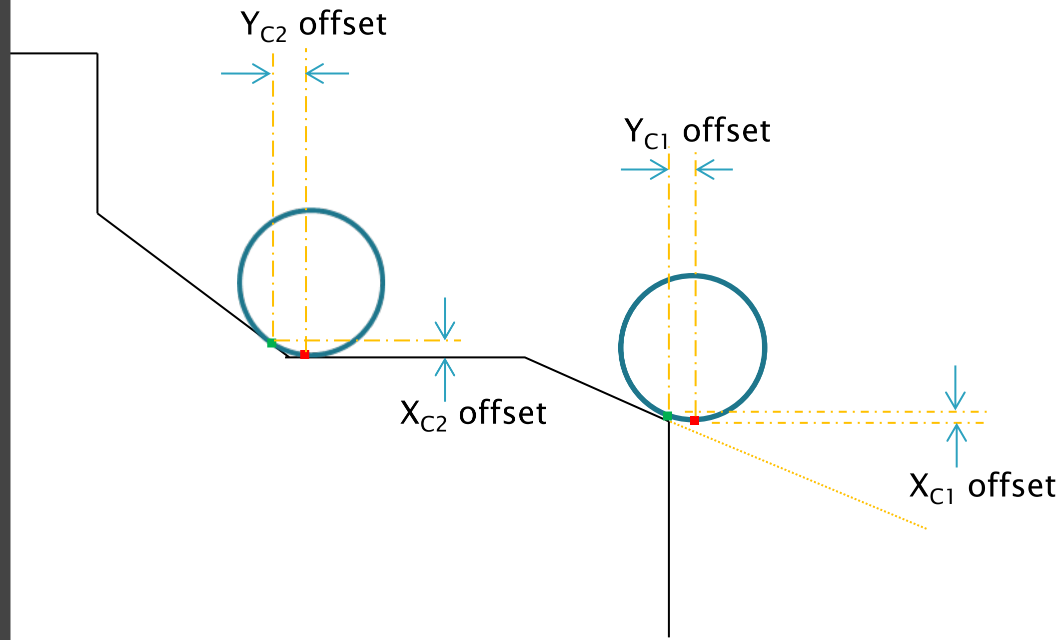


Image courtesy of Haas

Taper Offsets for Angle and TNR

1. When you *chamfer* (cut an angle) the cutting tip is not the same as where you touched off the tip. The tool was touched off at the red dots, but begins cutting the diagonal at the green dots.
2. The offsets depend on the cut angle and tool nose radius (either 0.31 or .0156).
3. Published Xc values are diametric and do not need to be multiplied by 2.
4. You will need to subtract Xc and add Zc.



Compensation for a Part Angle

- Example offsets of various angles and tool nose radii:

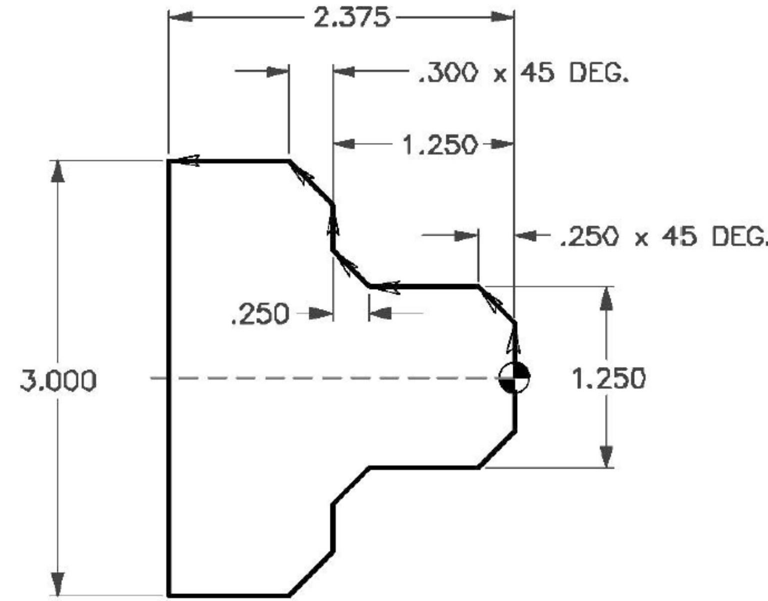
ANGLE	X-OFFSET .015 N/R	Z-OFFSET .015 N/R	X-OFFSET .031 N/R	Z-OFFSET .031 N/R
15	.0072	.0136	.0146	.0271
30	.0132	.0114	.0264	.0229
45	.0184	.0092	.0366	.0183
60	.0228	.0066	.0458	.0132

X-OFFSET numbers above are diametric.

Image courtesy of Haas

Manual Radius Compensation

```
O00055 (LINEAR INTERPOLATION "WITHOUT" CUTTER COMP.)
N1 G53 G00 X0. Z0. T0
N2 T101                    (O.D. TURNING TOOL .031 TNR)
N3 G97 S1450 M03
N4 G54 G00 X0.85 Z0.1 M08
N5 Z0.
N6 G01 X-.062 F0.01        (Face down end of part)
N7 G00 X.7134 Z0.02        (Rapid to start point of angle
subtracting X compensation value: .75-.0366)
N8 G01 Z0. F0.006          (Feed into face)
N9 X1.250 Z-.2683          (Feed up angle adding Z
compensation amount: .25+.0183)
N10 Z-1.0183 (Feed to angle adding Z compensation amount)
N11 X1.7134 Z-1.250        (Feed up to angle subtracting X
compensation amount)
N12 X2.3634                (Feed up to angle subtracting X
compensation amount)
N13 X3.0 Z-1.5683          (Feed up angle adding Z
compensation amount)
N14 Z-2.375                (Feed to finish to end in Z axis)
N15 G00 U-0.01 Z1.0 M09
N16 G53 G00 X0. Z0. T0
N17 M30
```



Offset .031TNR/45 degree is:
Xc = .0366,
Zc = .0183

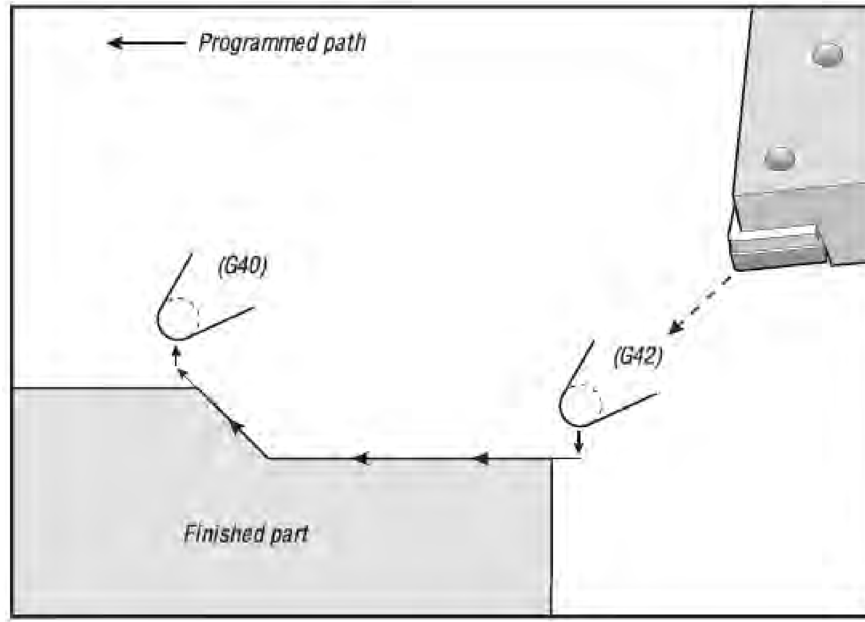
Image courtesy of Haas

Tool Nose Compensation

1) Approach Moves and Departure Moves
2) Tool Geometry and Wear Offsets
   - When setting up the tools for a part program, zero any Tool Geometry, Tool Wear, and Work Zero offsets that remain from an earlier job. Then you must touch off and enter the tool geometry (distance from machine home to part zero) length offsets for each tool being used.
3) Tool Nose Radius Geometry
   - Select a standard insert (with a defined radius) that will be used for each tool that is using tool nose compensation. Enter the tool nose radius of each compensated tool in the TOOL GEOMETRY offset display under RADIUS.
4) Tool Tip Direction
   - Input the tool tip direction, in the TOOL GEOMETRY offset display under TIP, for each tool that is using tool nose compensation, G41 or G42.
5) Test Run Compensation Geometry
   - Run the program in graphics mode and correct any tool nose compensation geometry problems that may occur. A problem can be detected in two ways: either an alarm will be generated indicating compensation interference, or you will see the incorrect geometry generated and seen in graphics mode.
6) Run and Inspect Part First

Approach & Departure

- Program an approach move for each tool path that needs tool nose compensation and determine if G41 or G42 is to be used.
- Be sure there is a departure move for each compensated tool path by using a G40 command to cancel tool nose compensation.
- At the end of the departure move the machine position is not compensated.

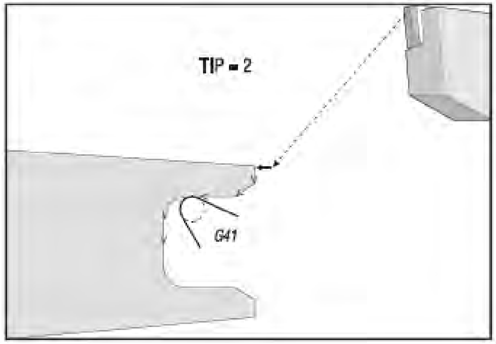


Approach and Departure moves.

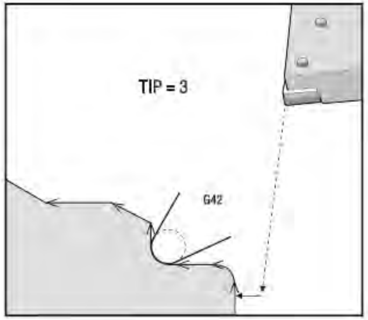
Image courtesy of Haas

G41, G42, G40 Cutter Compensation

- G41 will select tool nose compensation left; that is, the tool is programmed to the left of a tool path part line, to compensate for the tool tip radius. A tool offset must be selected with a Tnnxx code, where xx corresponds to the offsets that are to be used for the nn tool turret location.
- G42 will select tool nose compensation right; that is, the tool is programmed to the right of a tool path part line, to compensate for the tool tip radius. A tool offset must be selected with a Tnnxx code, where xx corresponds to the offsets that are to be used for the nn tool turret location.
- G40 Turn cutter compensation off.



G41 TNC Left of the programmed part line.



G42 TNC Right of the programmed part line.

Image courtesy of Haas

Automatic G01 Corner Rounding

(Using G42 and R plus or minus)

```
O00042 (Linear G01 with Radius using R)
N1 G53 G00 X0. Z0. T0
N2 T101 (O.D. TOOL x .031 TNR)
N3 G50 S3000
N4 G97 S3000 M03
N5 G54 G00 X0.3 Z0.1 M08
N6 G96 S390
N7 G42 G01 Z0. F0.01 (Tool nose compensation right)
N8 G01 X0.5 R-0.05
N9 G01 Z-0.5
N10 G01 X0.75 R-0.05
N11 G01 Z-1.0 R0.05     (R is to a positive X axis)
N12 G01 X1.25 R-0.05   (R- is to a negative Z axis)
N13 G01 Z-1.5
N14 G40 G00 U0.01 Z0.1 M09 (cancel tool nose comp)
N15 G53 G00 X0. Z0. T0
N16 M30
```

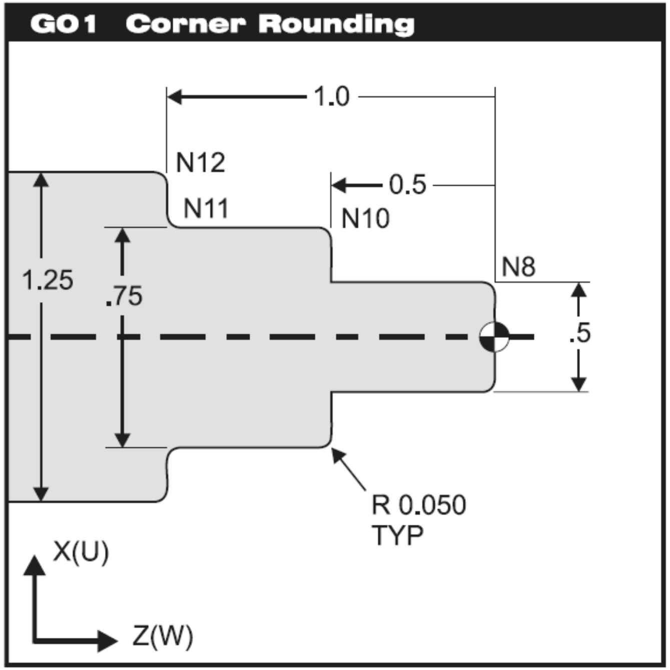


Image courtesy of Haas

Rules for Automatic Corner Rounding

1. R+ is to machine a radius into the plus direction on an axis. R– is to machine a radius into the minus direction on an axis.
2. The linear G01 block must be a single X(U) or Z(W) move with an R that is perpendicular to the previous move for corner rounding.
3. When using R for corner rounding, do not use I, K, or A.
4. This Radius (R) command is not supported in any of the roughing passes of a G71 or G72 canned cycles. The last pass of the G71 and G72 will be executed with the radius defined with R in the G01 command .
5. A G70 or G73 will support this type of radius command.
6. The + or – sign define the direction of the axis your moving into at the end of the arc move.

Automatic G01 Chamfering of 45° Angles

```
O00043 (Linear G01 with 45 Degree Chamfer
using I or K)
N1 G53 G00 X0. Z0. T0
N2 T101 (O.D. TOOL x .031 TNR)
N3 G50 S3000
N4 G97 S3000 M03
N5 G54 G00 X0.3 Z0.1 M08
N6 G96 S390
N7 G42 G01 Z0. F0.01
N8 G01 X0.5 K-0.05  (face - X into Z, right hand)
N9 G01 Z-0.5
N10 G01 X0.75 K-0.05 (X into Z, right of tool)
N11 G01 Z-1.0 I0.05    (Z into X,  up)
N12 G01 X1.25 K-0.05 (X into Z, right to tool)
N13 G01 Z-1.5
N14 G40 G00 U0.01 Z0.1 M09
N15 G53 G00 X0. Z0. T0
N16 M30
```

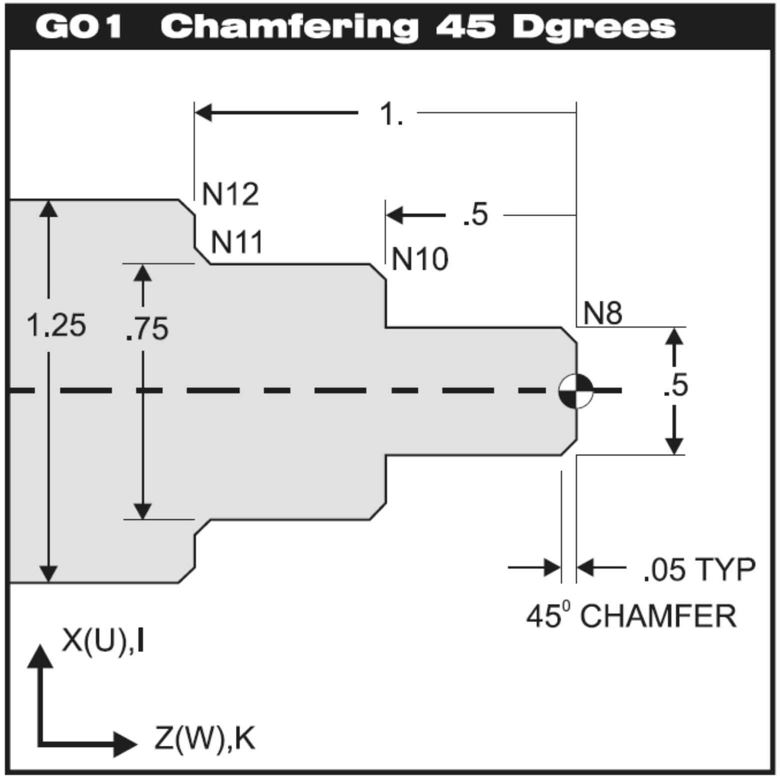


Image courtesy of Haas

Rules for Automatic Chamfering

1. I is for the 45-degree chamfer size from Z into X. I+ is up, I- is down into the X axis.
2. K is for the 45-degree chamfer size from X into Z. K- is left, K+ is right into the Z axis.
3. The linear G01 block must be a single X(U) or Z(W) move with an I or K that is perpendicular to the previous move for chamfering.
4. When using I or K for chamfering, do not use A or R. This chamfer (I,K) command is not supported in any of the roughing passes of a G71 or G72 canned cycles. The last pass of the G71 and G72 will be executed with the chamfer defined with I and K in the G01 command.
5. A G70 or G73 will support this type of chamfer command.
6. The + or - sign defines the direction of the axis your moving into at the end of chamfer move.

Chamfering with Angle

- O00044 (Linear G01 Chamfer 10 Degree Angle using A)
- N1 G53 G00 X0. Z0. T0
- N2 T101 (O.D. TOOL x .031 TNR)
- N3 G50 S2800
- N4 G97 S1490 M03
- N5 G54 G00 X1.25 Z0.1 M08
- N6 G96 S390
- N7 G42 G01 Z0. F0.01
- N8 G01 X1.375 (Start point)
- N9 G01 X1.5 A170. (10 Degree Angle using A)
- N10 Z-0.5
- N11 G40 G00 U0.01 Z0.1 M09
- N12 G53 G00 X0. Z0. T0
- N13 M30

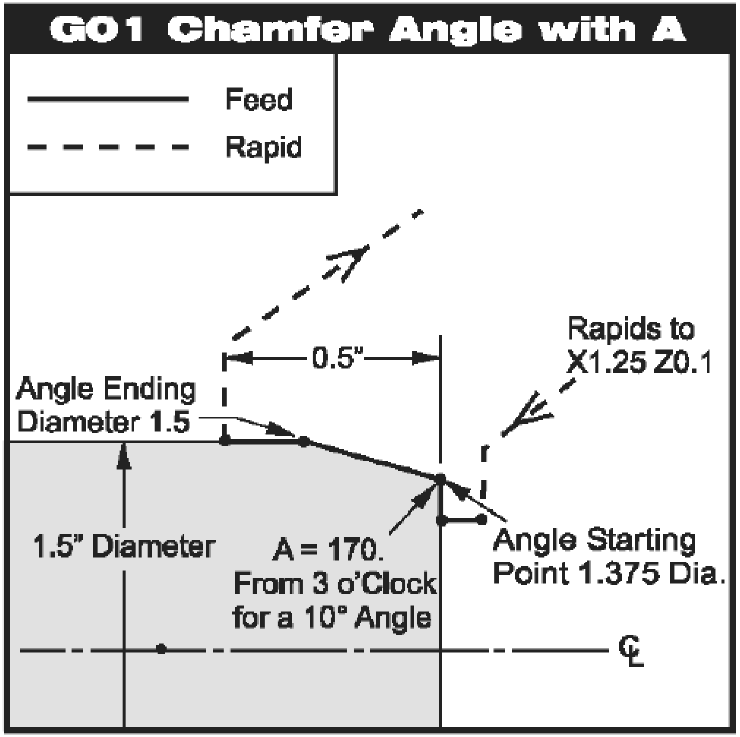


Image courtesy of Haas

Rules for Chamfering with Angles

1. The linear G01 block must be a single X(U) or Z(W) move that is perpendicular to the previous move with an A to do a specific angle.
2. When using A for an angle, do not use I, K or R.
3. This angle (A) command is "not supported in any of the roughing passes of a G71 or G72 canned cycles, though the last pass in the G71 and G72 will be executed with the angle defined with A in the G01 command.
4. A G70 or G73 will support this type of chamfer command.
5. You can use a minus value to define an angle clockwise from three o'clock: A-30. = A150. and A-45. = A135.
6. Be sure to enter in a decimal point for angles.

Canned Cycles Turning & Grooving

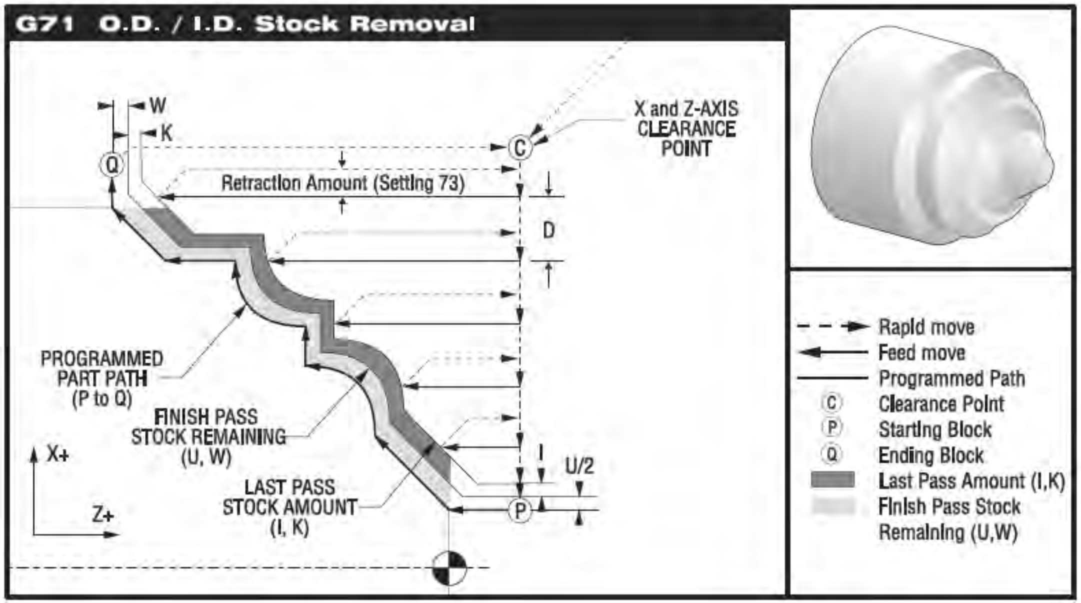
- **G70** Finishing
- **G71** O.D./I.D. Stock Removal (roughing)
- **G72** End Face Stock Removal (not recommended)
- **G73** Irregular Path Stock
- **G74** End Face Grooving
- **G75** O.D./I.D. Grooving
- **G76** O.D./I.D. Thread Cutting Cycle, Multiple Pass
- **G90** O.D./I.D. Turning Cycle
- **G92** Thread Cutting Cycle
- **G94** End Face Cutting Cycle

Modal

G70 Finishing

- The G70 Finishing cycle can be used to finish cut paths that are defined and roughed out with stock removal cycles G71, G72 and G73.
- The G70 requires that a beginning block number (P code) and an ending block number (Q code) be specified for the machine code that defines the part geometry to be machined.
- The G70 cycle is usually used after a G71, G72 or G73 has been performed using the blocks specified by P and Q.
- All codes in the block defined by P and Q are executed.
- Any F, S or T codes between the P and Q block are effective.
- The PQ sequence is searched for in the current program starting from the beginning of the program.

G71 Stock Removal



P Starting block number of part path to machine.

Q Ending block number of part path to machine.

U* Finish stock remaining with direction (+or –), X-axis diameter value.

W* Finish stock remaining with direction (+or –), Z-axis value.

I * Last pass amount with direction (+or –), X-axis radius value.

K* Last pass amount with direction (+or –), Z-axis value.

D* Depth of cut stock removal each pass, positive radius value (Setting 72).

F Roughing passes feed rate throughout this cycle.

R1* YASNAC type II roughing (only if setting 33 is on Yasnac).

S** Spindle speed in this cycle.

T** Tool and offset in this cycle.

* Indicates optional

** Rarely defined in a G71 line

Image courtesy of Haas

G70 Roughing and G71 Finishing

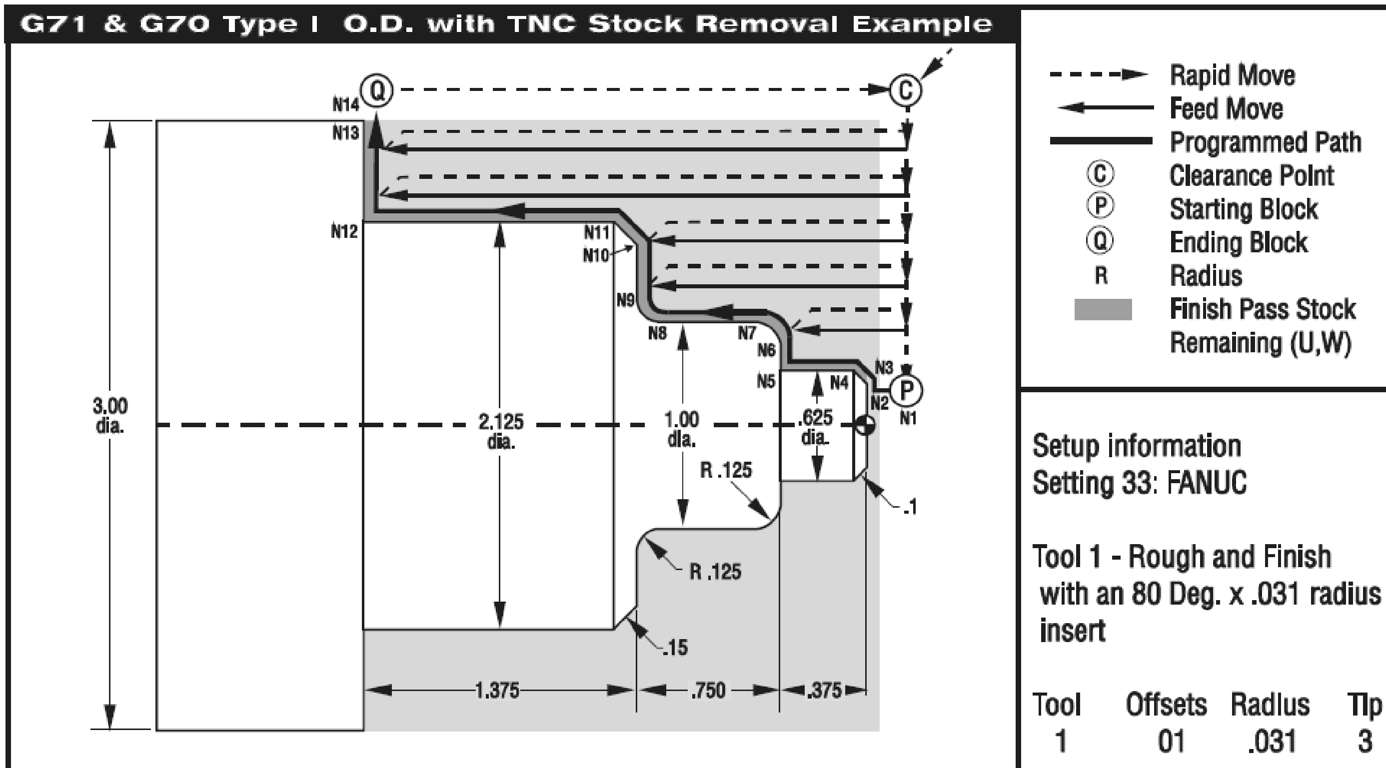


Image courtesy of Haas

Example G70 and G71…

```
O0106
G28
T0202
G50 S1750
G97 S320 M03
G54 G00 X3. Z0.1 M08            (Rapid to start point)
G96 S300
G71 P10 Q20 U.02 W.01 D.1 F.012   (G71 Define part path lines P thru Q, blocks 10 to 20, .01 depth each pass)
N10 G42 G00 X0.5                (Turn on TNC, P Block, TNC on, type 1 - X-only specified for roughing)
G01 ZO. F .012
X.6
X0.8 Z-0.1 F.008
Z-0.5
G02 X1.0 Z-0.6 10.1
G01 X1.5
X2.0 Z-0.85
Z-1.6
X2.3
G03 X2.8 Z-1.85 K-0.25
G01 Z-2.1
N20 G00 G40 X3.0                (Cancel TNC, go back to start point)
G70 P10 Q20                     (Go back and finish cut using same P block)
G28
M30
```

(Stock to leave for finish cut in X axis (U) and Z axis (W).)

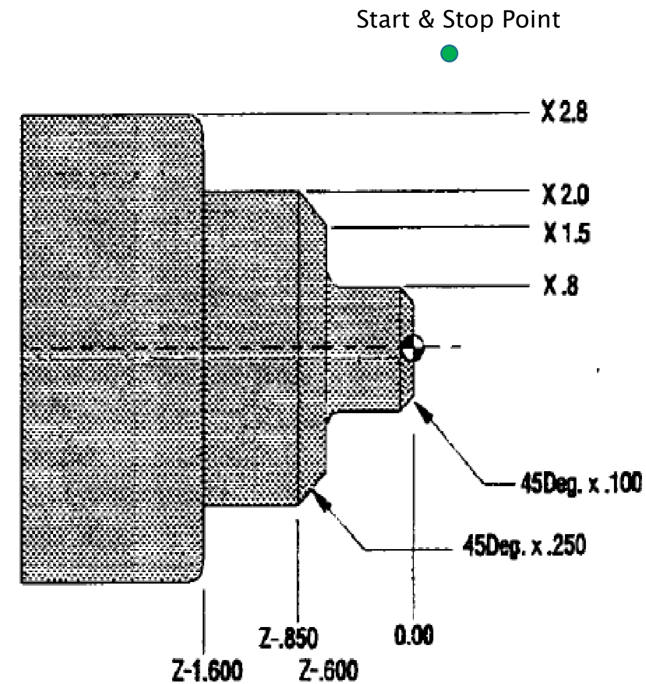
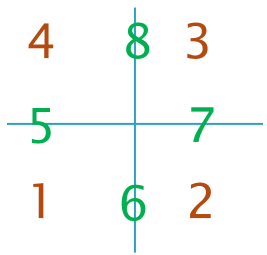


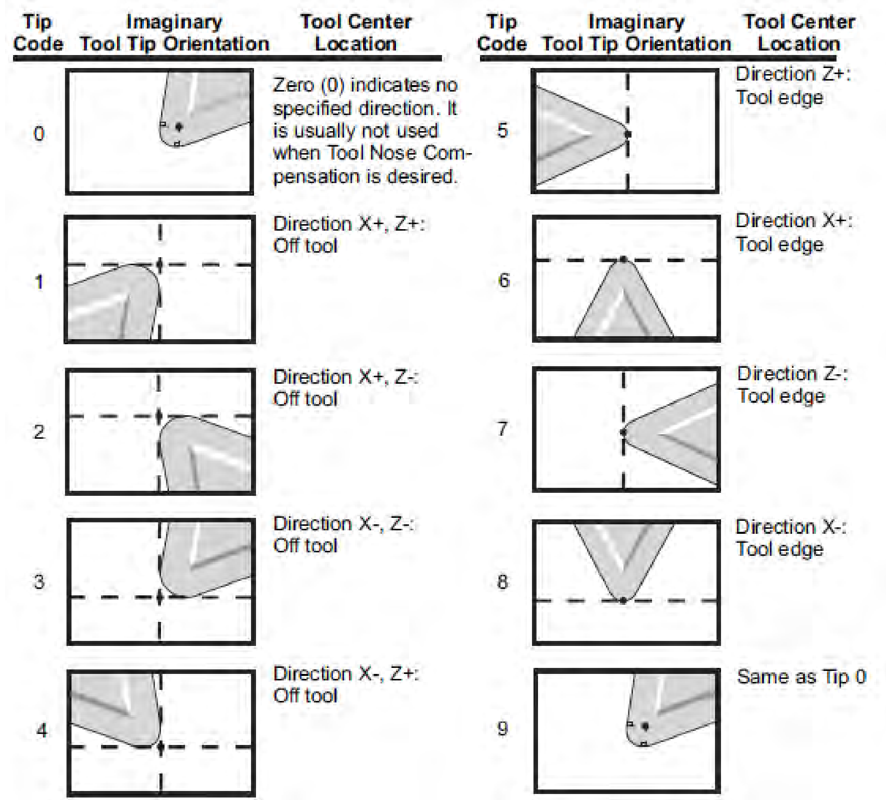
Image courtesy of Haas

G71 Type

- Type 1
  - All roughing passes start and end at the Z clearance plane.
  - Each roughing pass X-axis location is determined by applying the value specified in D to the current X location. The direction that D is applied is determined by the signs of U and W.
  - The nature of the movement along the Z clearance plane for each roughing pass is determined by the G code in block P. If block P contains a G00 code, then movement
  - along the Z clearance plane is a rapid mode. If block P contains a G01, then movement will be at the G71 feed rate.
  - Roughing continues until the X-axis position in block P is exceeded.
  - Each roughing pass is stopped before it intersects the programmed tool path allowing for both roughing and finishing allowances. The tool is then retracted from the material at a 45-degree angle by the distance specified in setting 73. The tool then moves in rapid mode to the Z-axis clearance plane.
  - When roughing is completed, the tool is moved along the tool path to clean up the rough cut.
  - If I and K are specified, an additional rough finish cut parallel to the tool path is performed.

Tool Tip Direction



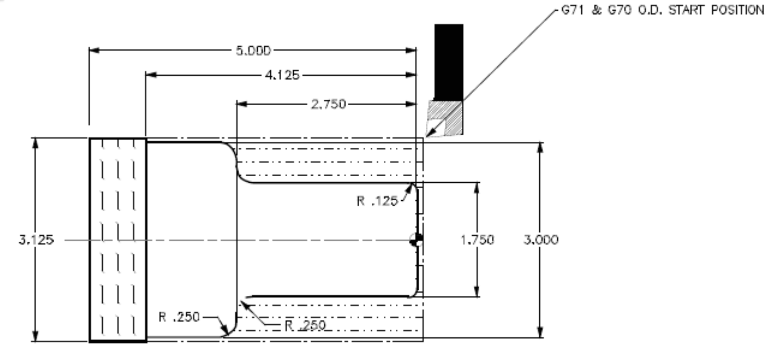


Tip Code	Imaginary Tool Tip Orientation	Tool Center Location	Tip Code	Imaginary Tool Tip Orientation	Tool Center Location
0		Zero (0) indicates no specified direction. It is usually not used when Tool Nose Compensation is desired.	5		Direction Z+: Tool edge
1		Direction X+, Z+: Off tool	6		Direction X+: Tool edge
2		Direction X+, Z-: Off tool	7		Direction Z-: Tool edge
3		Direction X-, Z-: Off tool	8		Direction X-: Tool edge
4		Direction X-, Z+: Off tool	9		Same as Tip 0

Image courtesy of Haas

Example G71/G72 Type 1 with TNC

```
O00060                              (Program number)
N1 (ROUGH O.D.)                     (Rough O.D.)
G53 G00 X0. Z0. T0                  (Sending home for a tool change)
T101 (O.D. TOOL x .031 TNR) (Tool #1 and Offset #1)
G50 S3200                           (Spindle speed clamp at 3200 RPM)
G97 S500 M03                        (Cancel CSS, 500 RPM, spindle ON forward)
G54 G00 X3.2 Z0.1 M08               (Rapid X3.2, Z0.1 to start point, coolant ON)
G96 S420                            (CSS ON, at 420 surface speed)
Z0.005                              (Position .005 from end of part)
G01 X-0.063 F.008                   (Feed down X-.063 to face end of part)
G00 X3.2 Z0.1                       (Rapid to X3.2, Z0.1 start point above part)
G71 P10 Q29  U0.01 W0.005 D0.12 F0.01 (G71 Rough cycle)
N10 G00 G42 X1.4                    (Pnn start #, rapid X1.4, Cutter Comp. ON)
G01 Z0. F0.006                      (G71 Part Geometry)
X1.5                                ( " " " )
G03 X1.75 Z-0.125 R0.125            ( " " " )
G01 Z-2.5                           ( " " " )
G02 X2.25 Z-2.75 R0.25              ( " " " )
G01 X2.5                            ( " " " )
G03 X3.0 Z-3.0 R0.25                ( " " " )
G01 Z-4.125 F.004                   ( " " " )
N20 G40 X3.2 F.02                   (Qnn end #, cancel Cutter Comp. feed X to
3.2)
G97 S500 M09                        (Cancel CSS, 500 RPM, Coolant Off)
G53 G00 X0. Z0. T0                  (Sending home for a tool change)
M01                                 (Optional stop command)
```

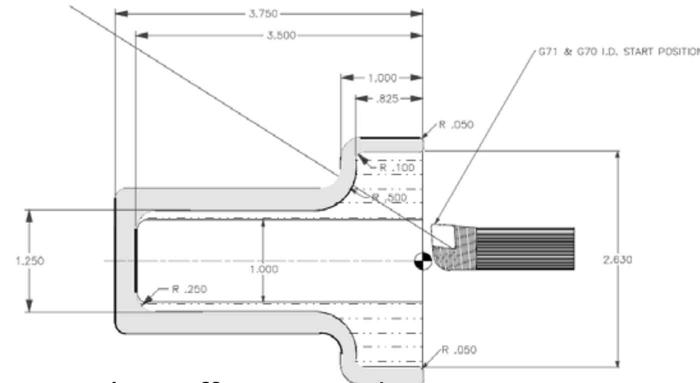



```
N2 (FINISH O.D.)
G53 G00 X0. Z0. T0          (Sending home for a tool change)
T202 (O.D. TOOL x .031 TNR) (Tool #2 and Offset #2)
G50 S2800                   (Spindle speed clamp at 3200 RPM)
G97 S890 M03                (Cancel CSS, 890 RPM, spindle on)
G54 G00 X1.8 Z0.1 M08       (Rapid, X1.8, Z.1 location, coolant
ON)
G96 S240                    (Turn on CSS to 420)
Z0.                         (Position to Z0 end of part)
G01 X-0.062 F0.005          (Feed down face of part)
G00 X3.2 Z0.1               (Rapid to X3.2, Z0.1 start point
above part)
G70  P10 Q20                (Define a G70 finish pass of part
geometry)
G97 S500 M09                (Cancel CSS, define 500 RPM,
Coolant Off)
G53 G00 X0. Z0. T0          (Sending home for a tool change)
M30                         (End of program and reset)
```

Image courtesy of Haas

Example G71 Type 1with I.D.

```
O00088 (Example of using a G71 on an I.D. with TNC)
N1 G53 G00 X0. Z0. T0 (Sending home for a tool change)
N2 T404                    (Select Tool 4 Offset 4)
N3 G50 S3000
N4 G97 S1780 M03
N5 G54 G00 X0.9 Z0.1 M08 (Rapid to start point below the I.D. stock diameter)
N6 G96 S420
N7 G71 P8 Q18 U-0.01 W0.005 D0.12 F0.012
                     (U is minus for G71 I.D. Roughing)
N8 G41 G00 X2.83        (N8, Start of part path geometry defined by P8 in G71 line)
N9 G01 Z0. F0.02
N10 X2.73 F0.005
N11 G02 X2.63 Z-.05 R0.05
N12 G01 Z-.725
N13 G03 X2.43 Z-.825 R.1
```



Tool=4, Offset=04, Radius=0.032, Tip=2
Remember to enter tool 4 offset & tip data.

```
N14 G01 X2.25
N15 G02 X1.25 Z-1.325 R0.5
N16 G01 Z-3.25
N17 G03 X.75 Z-3.5 R0.25
N18 G01 G40 X0.7 (N18 End of part path geometry defined by Q18 in G71 line)
N19 G70 P8 Q18
N20 G97 S1780 M09
N21 G53 G00 X0. Z0. T0 (Sending home for a tool change)
N22 M30 (End of Program)
```

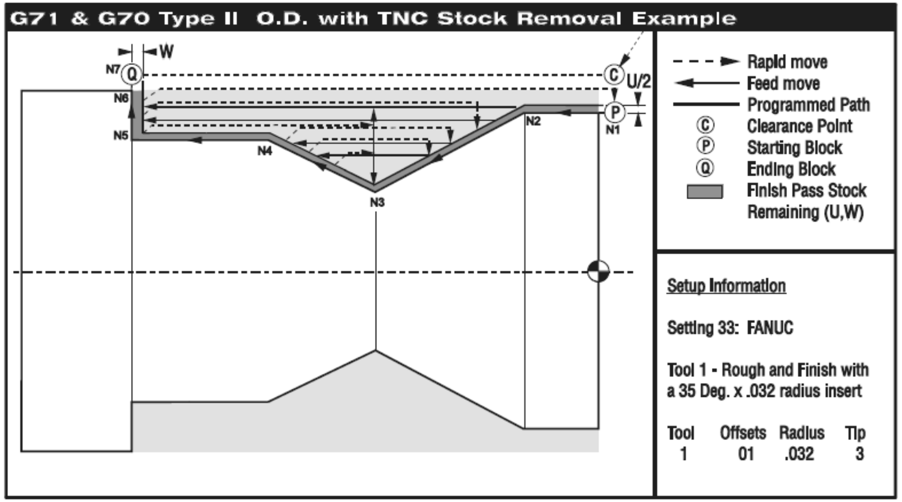
Image courtesy of Haas

G71 Type 2

- Type 2
  - The X-axis can change direction throughout the PQ path. Z must continue along in the same direction as the initial Z direction.
  - When Setting 33 is set to FANUC, placing a reference to both the X and Z-axis in the block specified by P specifies Type II.
  - A trough can be defined as a change in direction creating a concave surface in the material being cut. If successive troughs are on the same level, there can be an unlimited number of troughs. When troughs are within troughs (nested), there can be no more than 10 levels of trough nesting.

Example G71 Type 2 Roughing...

```
O00090
N101 G28                      (FANUC TYPE II G71 ROUGHING CYCLE)
N102 T101                     (Roughing Tool)
N103 G50 S3000
N104 G97 S746 M03
N105 G54 G00 X2.1 Z0.1        (Start Position)
N106 G96 S380
N107 G71 P108 Q113 U0.02 W0.005 D0.05 F0.01
N108 G00 X1.75 Z0.1           (A G71 Type II has moves in both X and Z in the P block)
N109 G01 Z0-.25 F0.006
N110 X1. Z-1.
N111 X1.5 Z-1.5
N112 Z-2.25
N113 G01 X2.1                 (End of PQ part Definition)
N114 G97 S746 M09
N115 G28
N116 G28                      (G70 FINISHING O.D.)
N117 T202                     (Finishing Tool)
```

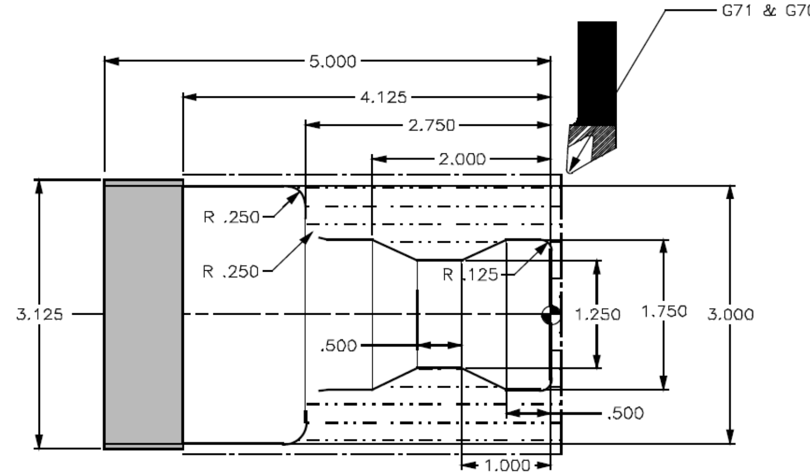



```
N118 G50 S3000
N119 G97 S690 M03
N120 G54 G00 X2.1 Z0.1        (Start Position)
N121 G96 S410
N122 G70 P108 Q113            (Finishing Cycle)
N123 G97 S746 M09
N124 G28
N125 M30
```

Image courtesy of Haas

Example G71 Type 2...

```
O00070                          (Program number)
N1 (ROUGH O.D.)                 (G71 type II roughing operation)
G53 G00 X0. Z0. T0              (Sending home for a tool change)
T101 (O.D. TOOL .031 TNR) (Tool #1 and offset #1)
G50 S3200                       (Spindle speed max. clamp at 3200 RPM)
G97 S500 M03                    (Cancel CSS, 500 RPM, spindle ON forward)
G54 G00 X3.2 Z1. M08            (Rapid X and Z to start location, coolant ON)
G96 S425                        (CSS on at 425)
Z0.005                          (Position .005 from end of part)
G01 X-0.063 F0.008              (Feed down to rough face end of part)
G00 X3.2 Z0.1                   (Rapid to start position above part)
G71 P10 Q20 U0.01 W0. D0.12 F0.01 (Rough O.D. G71 cycle command)
*N10 G42 G00 X1.4 Z.1           (Pnn starting number, Rapid X and Z axis for
                                 type II G71, G42 cutter comp ON)
G01 Z0. F0.006                  (G71 Part Geometry)
X1.5                            (1.750 - 2 face radii)
G03 X1.75 Z-0.125 R0.125        (G71 Part Geometry)
G01 Z-.5               (Type II G71 Geometry changes direction in X axis)
X1.25 Z-1. F.01                 ( " " " " " )
Z-1.5                           ( " " " " " )
X1.75 Z-2.                      ( " " " " " )
Z-2.5 F.006                     (G71 Part Geometry; 2.75-.25)
G02 X2.25 Z-2.75 R0.25          ( " " " )
G01 X2.5                        ( " " " )
G03 X3. Z-3. R0.25              ( " " " )
G01 Z-4.125 F0.004              ( " " " )
N20 G40 X3.2 F0.02  (Qnn ending number, cancel Cutter Comp. to X3.2)
G97 S500 M09                    (Cancel CSS, 500 RPM, coolant Off)
G53 G00 X0. Z0. T0              (Sending home for a tool change)
M01                             (Optional stop command)
```

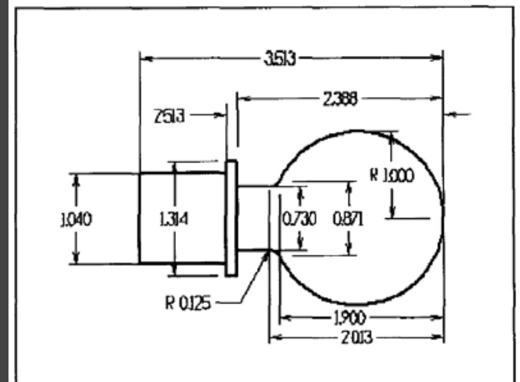



```
N2 (FINISH O.D.)         (G71 type II finishing operation)
G53 G00 X0. Z0. T0       (Sending home for a tool change)
T202            (O.D. TOOL .031 TNR) (Tool #2 and Offset #2)
G50 S3200                (Spindle speed clamp at 3200 RPM)
G97 S890 M03             (Cancel CSS, 890 RPM, spindle ON forward)
G54 G00 X1.8 Z1. M08 (Rapid, X,Z location, coolant on)
G96 S420                 (Turn on CSS to 420)
Z0.                      (Position to end of part)
G01 X-.062 F0.006        (Feed down face of part)
G00 X3.2 Z.1             (Rapid to start position above part)
G70 P10 Q20              (G70 finish pass using part geometry)
G97 S500 M09             (Cancel CSS, 500 RPM, coolant Off)
G53 G00 X0. Z0. T0       (Sending home for a tool change)
M30                      (End program and rewind)
```

Image courtesy of Haas

G71 Type 2 Ball Joint Side 1

```
O00004 (BALLJOINT SIDE 1)
N1 (STOCK STICKOUT 4.0 FROM JAWS)
T101 (80 DEG OD TOOL .0312R)
G50 S1500 (Clamp spindle speed at 1500 rev/min)
G97 S682 M03 (Turn on spindle speed so at 2.1d. S375ft/min)
G54 G00 X2.1 Z0.1 M08 (Rapid to position in front of part)
G96 S375 (Turn on constant surface speed to 375ft/min)
G01 Z0 F0.012 (Feed to face off position)
G01 X-0.1 F0.007 (Face off front of part)
G54 G00 X2.1 Z0.1 M08 (Rapid to G71 Start Position)
G71 P11 Q21 U0.01 W0.005 00.06 F0.012 (Leave .010 on OD and
.005 on face for finish)
N11 G00 G42 XO ZO.1
G01 ZO F0.006
G01 X0.98
G01 X1.04 Z-0.03 (Turn .03 chamfer)
G01 Z-1.
G01 X1.254
G01 X1.314 Z-1.03 (Turn .03 chamfer on 1.314 diam.)
Z-1.4
N21 G00 G40 X2.1 (Turn off tool nose compensation to 2.1 diam.)
G70 P11 Q21 (Cut Finish Pass)
G97 S400 M09
G53 XO
G53 Z-30.
M05
MO1
```



Stock is 2.06 in diameter.

Use CNMG 432 in 80 degree holder to face. rough and finish 1.04 and 1.314 diameters.

Run at surface feed 375ft/min.

F .01 in/rev rough F.007 for finish pass.

Turn back to 1.4 inch from face.

Cut off with .125 tool leaving OAL 3.708.

Cut off at S300 ft/min and feed of .003 In/rev.

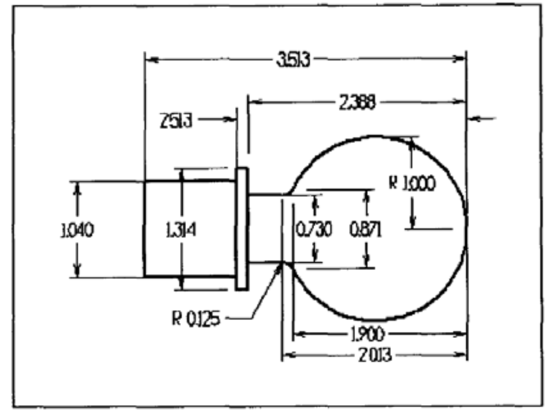
1/32 radius tool

```
(PART OFF WITH PECK)
T202 (.125 CUT OFF)
G54
G50 S400
G96 S200 M03
G54 G00 X2.1 Z0.1 (Rapid to start position)
G00 Z-3.833 (Rapid to cut off point in Z)
G75 X0.1 Z-3.833 I0.05 F0.003 (Cut off Part)
G00 X2.1 Z0.05
G53 X0     (Take home in Z)
G53 Z-30. (Rapid to safe in Z-30.)
M30
```

Image courtesy of Haas

Ball Joint Side 2

```
O00175 (BALLJOINT Side 1)
N1 (35 DEG TURN)
T101 (OD TOOL .0312R)
N2 G50 S2500      (CLAMP SPINDLE SPEED AT 2500)
N3 G97 S636 M03
N4 G54 G00 X2.25 Z0.1 M08 (START POSITION FOR
G71)
N5 G96 S375
N6 G71 / P1O Q20 UO.O1 W0.005 00.05 F0.012
N10 G00 G42 X-0.070 Z0.l
N11 G0l ZO F0.006
N12 X0
N13 G03 X0.871 Z-1.9 R1.
N14 G02 X0.73 Z-2.013 R0.125
N15 G01 Z-2.388
N16 G01 X1.285
N18 G01 X1.385 Z-2.438
N20 G00 G40 X2.25
N21 G97 S400 M09
N22 G28
M01
N2
```



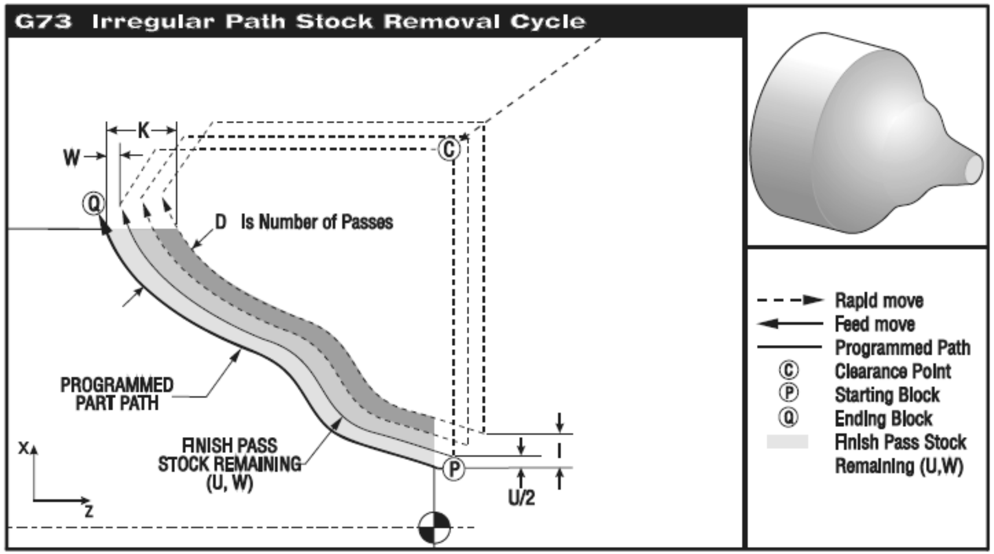
The first side 1.040 and 1.314 have already been turned.

Rough turn .05 depth of cut, leave .005 one face .01 on diameter for G70.

```
T202 (60 DEG FIN)
N23 G50 S2500
N24 G97 S1200 M03
N25 G00 X2.25 Z0.1
N26 G96 S650
N27 G70 P1O Q20
N28 G97 S400 M09
N29 G28
M30
```

Image courtesy of Haas

G73 Irregular Path Stock Removal



P Starting block number of part path to machine
Q Ending block number of part path to machine
U* Finish stock remaining with direction (+or –), X-axis diameter value
W* Finish stock remaining with direction (+or –), Z-axis value
I Distance and direction from first cut to last cut amount, X-axis radius value
K Distance and direction from first cut to last cut amount, Z-axis value
D Number of roughing passes, positive number
F Roughing passes feed rate throughout this cycle
S** Spindle speed to use in this cycle
T** Tool and offset to use in this cycle

* Indicates optional
** Rarely defined in a G73 line

Image courtesy of Haas

Example G73 & G70…

```
O00101 (G73 O.D. ROUGHING)
N101 G53 G00 X-3. Z-4. T0 (Rapid to tool change location)
N102 T101 (Tool 1 Offset 1)
N103 G50 S3000
N104 G97 S450 M03
N105 G54 G00 X3.1 Z.1 M08 (Rapid to Start Point)
N106 G96 S370
N107 G73 P108 Q121 U.01 W.005 I0.3 K0.15 D4 F.012 (G73
Stock Removal)
N108 G42 G00 X0.325 (P) (Start of geometry P number in G73
line. G42 C.C. right)
N109 G01 Z0. F0.01
N110 X0.425
N111 X.625 Z-0.1 F0.005
N112 Z-0.375
N113 X0.75
N114 G03 X1. Z-0.5 R.125
N115 G01 Z-1. F0.003
N116 G02 X1.25 Z-1.125 R.125
N117 G01 X1.825
N118 X2.125 Z-1.275
N119 Z-2.5 F0.008
N120 X3.
N121 G40 G00 X3.1 (Q) (End of geometry Q number in G71
line. G40 cancels C.C.)
N122 G70 P108 Q121 (G70 Finishing cycle N108 thru N121)
N123 G97 S450 M09
N124 G53 G00 X-3. Z-4. T0 (Rapid to tool change location)
N125 M30 (End of Program)
```

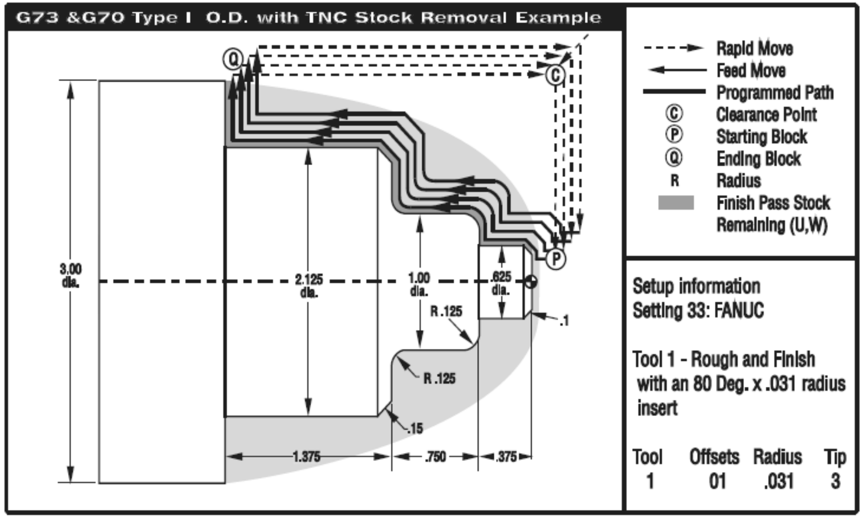
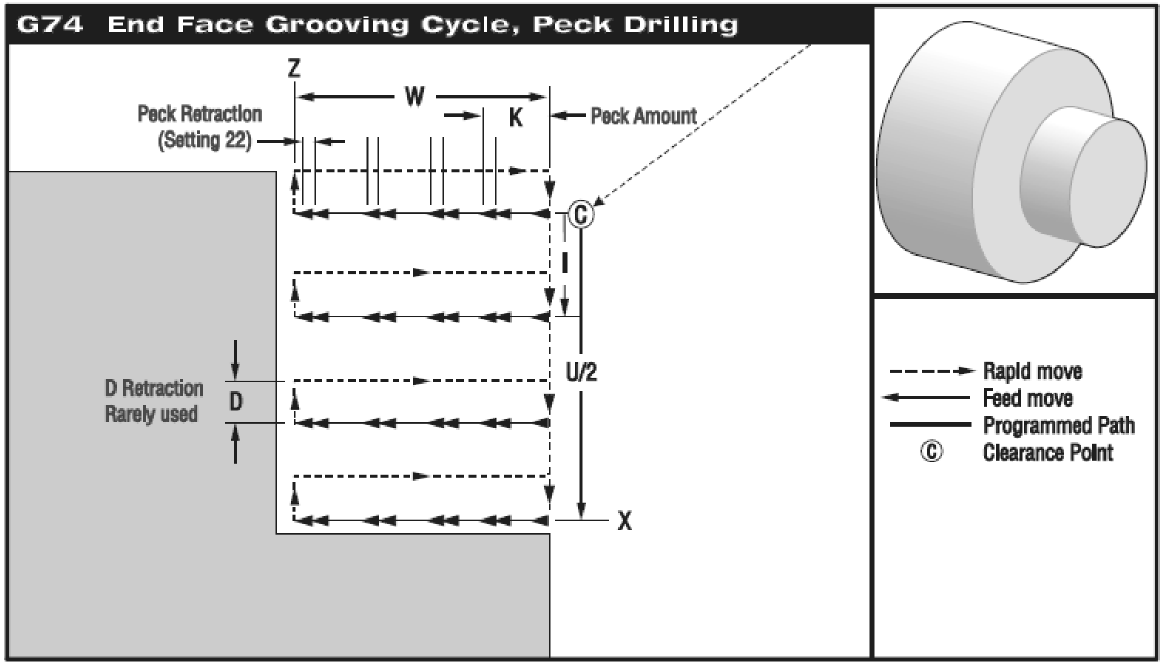


Image courtesy of Haas

G74 End Face Grooving or High Speed Peck Drill



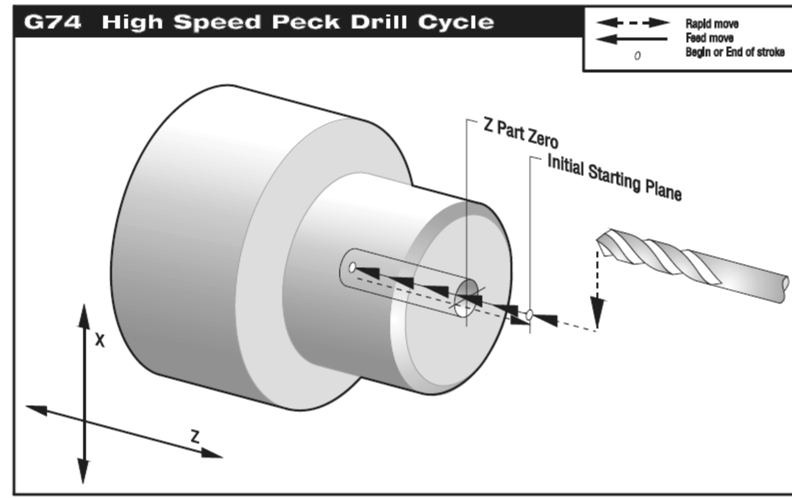
- X   X axis absolute pecking depth, diameter value.
- Z* Z axis absolute location to the furthest peck.
- U* X axis incremental pecking depth, diameter value.
- W* Z axis incremental distance and direction (+or –) to the furthest peck.
- I * X axis pecking depth increment, radius value.
- K* Z axis shift increment between pecking cycles.
- D* Tool shift amount when returning to clearance plane
- F   Feed rate

* Optional

Image courtesy of Haas

Example G74…

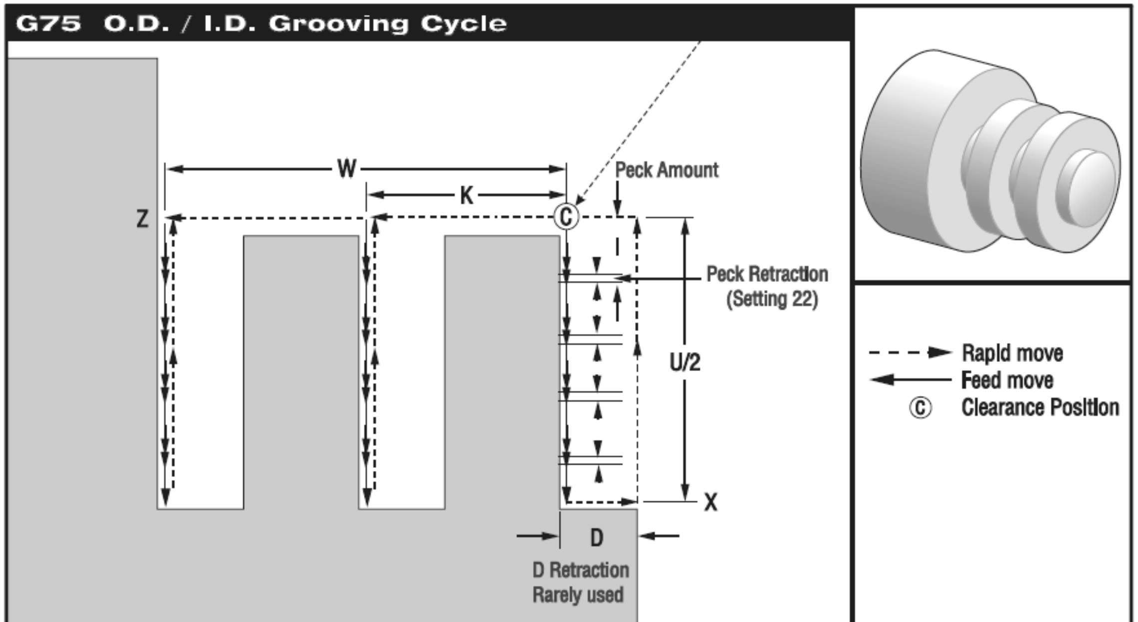
```
O00107 (G74 High Speed Peck Drilling example)
(Drill a .500 Diameter to a .525 Depth)
N1 G28
N2 T404 (1/2 DIA. DRILL)
N3 G97 S2445 M03
N4 G54 G00 X0. Z0.1 M08 (Rapid to X0 and Z start point)
N5 G74 Z-.525 K0.1 F0.006 (Drills to Z-.525 depth, pecking every .1 to pull)
N6 G00 Z1.0 M09 (back after each peck the amount in Setting 22.)
N7 G28
N8 M30
```



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

Image courtesy of Haas

G75 Grooving



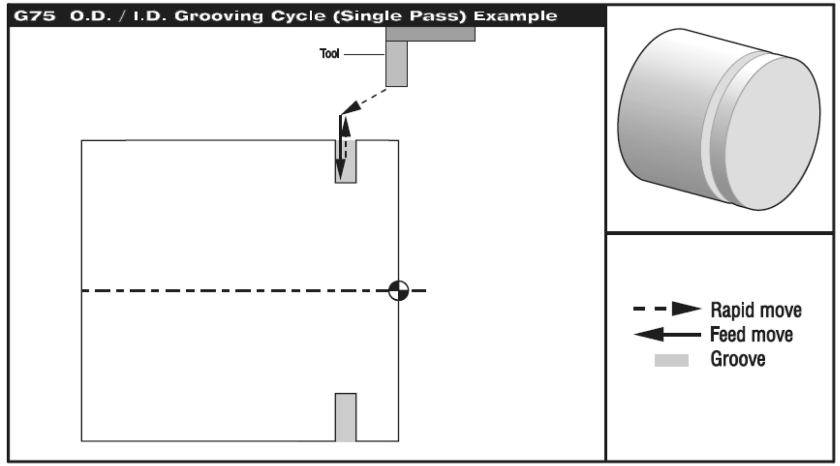
X X-axis absolute pecking depth, diameter value.
Z* Z-axis absolute location to the furthest peck.
U* X-axis incremental pecking depth, diameter value.
W* Z-axis incremental distance and direction (+or –) to the furthest peck.
I * X-axis pecking depth increment, radius value.
K* Z-axis shift increment between pecking cycles.
D* Tool shift amount when returning to clearance plane .(Caution see NOTE)
F Feed rate

* Indicates optional

Image courtesy of Haas

Example G75 Single Pass…

```
O00109 (G75 O.D./I.D. SINGLE PASS GROOVE CYCLE)
(Machine a .25 wide O.D. Groove with .25 Groove Tool)
N1 G28
N2 T505 (.25 WIDE O.D. GROOVE TOOL)
N3 G97 S960 M03
N4 G54 G00 X2.1 Z0.1 M08 (Rapid to clearance point)
N5 Z-0.75 (Rapid to a start point of groove)
N6 G75 X1.75 I0.05 F0.005 (G75 Single pass O.D. grooving cycle)
N7 M09
N8 G28
N9 M30
```



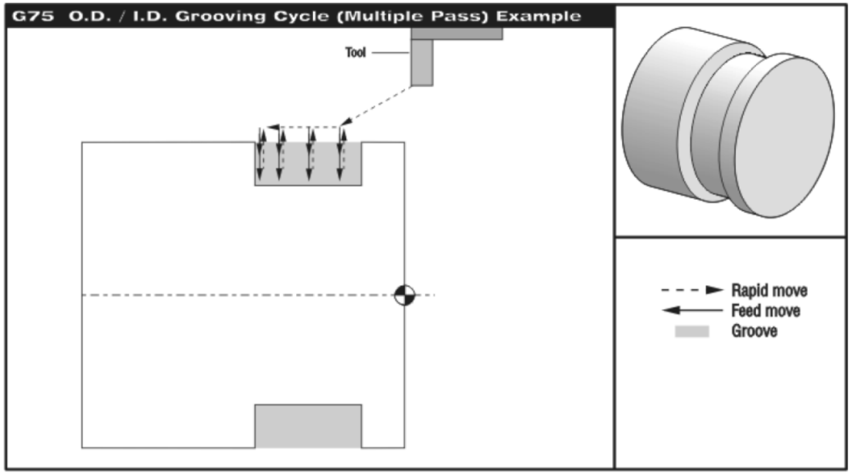
SETTING 22 (CAN CYCLE DELTA Z) – As the groove tool pecks deeper into the part, with each peck value of I, it pulls back a constant specified distance above the bottom of the groove created by the previous peck to break the chip. That specified distance it pulls back is defined in Setting 22.

Image courtesy of Haas

Example G75 Multiple Pass…

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

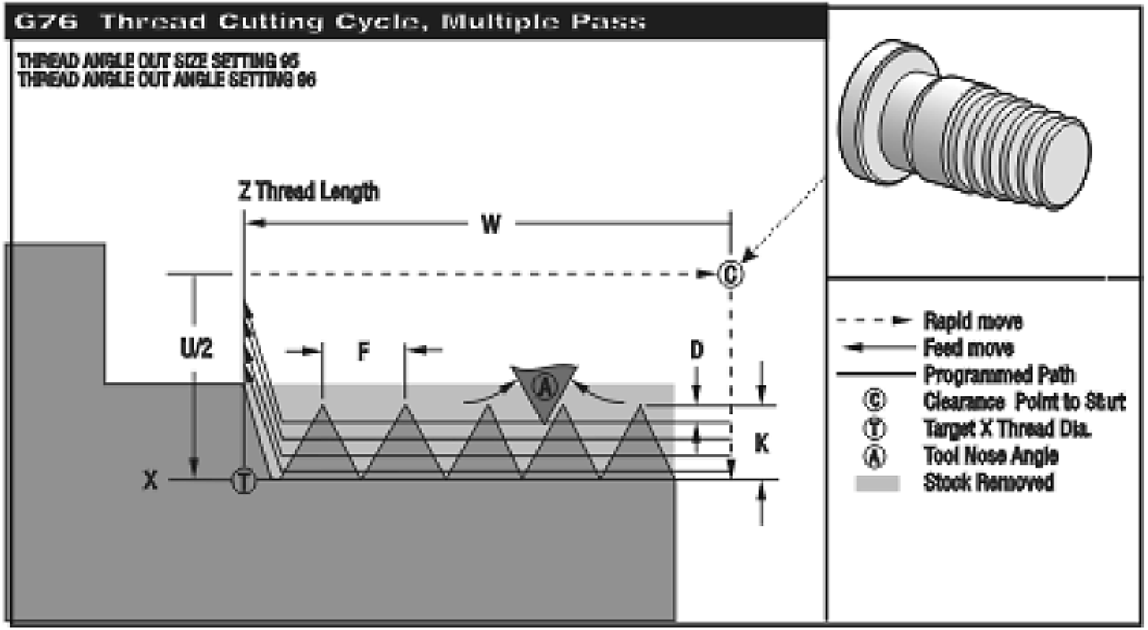
```
O00110 (G75 O.D./I.D. MULTIPLE PASS GROOVING CYCLE)
N1 G28
N2 T505 (.25 WIDE O.D. GROOVE TOOL)
N3 G97 S960 M03
N4 G54 G00 X2.1 Z0.1 M08 (Rapid to front of part)
N5 Z-0.75 (Rapid to start point of groove)
N6 G75 X1.75 Z-1.5 I0.05 K0.2 F0.005 (G75 Multiple pass O.D. grooving cycle)
N7 M09
N8 G28
N9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



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

Image courtesy of Haas

G76 Multiple Pass Thread Cutting



X* X-axis absolute thread finish point, diameter value.
Z* Z-axis absolute distance, thread end point location.
U* X-axis incremental total distance to finish point, diameter.
W* Z-axis incremental thread length finish point.
K Thread height, radius value
I* Thread taper amount, radius value.
D First pass cutting depth.
P Thread Cutting Method P1-P4 (added in software ver. 6.05)
A* Tool nose angle, no decimal with A command. (0 to 120 degrees, If not used then 0 degrees is assumed)
F Feed rate (Threading feed rate, is the thread distance per revolution)

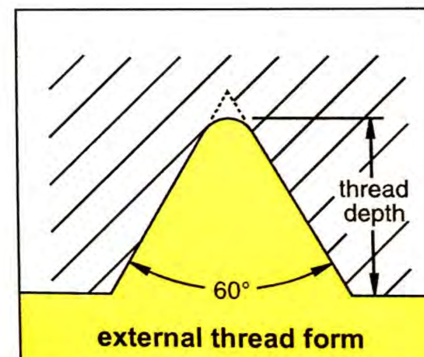
* Indicates optional

Image courtesy of Haas

Infeed for External Threading

External UN Threads – Recommendations for Steel Workpieces (<300 BHN)

tpi	4	5	6	7	8*	9	10	11	12	13	14	16	18	20	24	28	32	36	40	44	48
thread depth	.1578	.1262	.1052	.0902	.0789	.0701	.0631	.0574	.0526	.0485	.0451	.0394	.0350	.0315	.0263	.0225	.0197	.0175	.0157	.0143	.0131
# passes 1	.0353	.0298	.0248	.0213	.0197	.0175	.0169	.0157	.0152	.0142	.0136	.0125	.0124	.0119	.0118	.0112	.0098	.0087	.0078	.0073	.0065
2	.0146	.0122	.0105	.0088	.0082	.0073	.0070	.0066	.0064	.0057	.0059	.0054	.0053	.0049	.0048	.0046	.0042	.0036	.0032	.0028	.0027
3	.0113	.0094	.0078	.0077	.0063	.0056	.0053	.0048	.0048	.0044	.0043	.0039	.0039	.0039	.0039	.0036	.0031	.0028	.0024	.0022	.0020
4	.0095	.0079	.0067	.0059	.0053	.0047	.0045	.0041	.0042	.0037	.0036	.0034	.0033	.0032	.0031	.0031	.0026	.0024	.0020	.0020	.0019
5	.0084	.0070	.0058	.0050	.0047	.0042	.0039	.0036	.0036	.0033	.0032	.0029	.0029	.0028	.0027						
6	.0076	.0063	.0052	.0045	.0043	.0037	.0036	.0031	.0032	.0030	.0029	.0026	.0026	.0025							
7	.0070	.0058	.0048	.0041	.0039	.0034	.0031	.0028	.0029	.0027	.0026	.0024	.0024	.0023							
8	.0065	.0054	.0045	.0038	.0036	.0032	.0030	.0026	.0027	.0025	.0024	.0022	.0022								
9	.0061	.0051	.0042	.0036	.0034	.0030	.0029	.0025	.0026	.0024	.0023	.0021									
10	.0057	.0048	.0040	.0034	.0032	.0028	.0028	.0024	.0025	.0023	.0022	.0020									
11	.0054	.0045	.0038	.0032	.0031	.0027	.0027	.0023	.0023	.0022	.0021										
12	.0052	.0043	.0036	.0031	.0029	.0026	.0026	.0022	.0022	.0021											
13	.0049	.0042	.0035	.0030	.0027	.0025	.0025	.0021													
14	.0048	.0041	.0034	.0029	.0026	.0024	.0024	.0020													
15	.0046	.0040	.0033	.0028	.0025	.0023															
16	.0044	.0039	.0032	.0027	.0025	.0022															
17	.0043	.0038	.0031	.0026																	
18	.0042	.0037	.0030	.0025																	
19	.0041																				
20	.0039																				



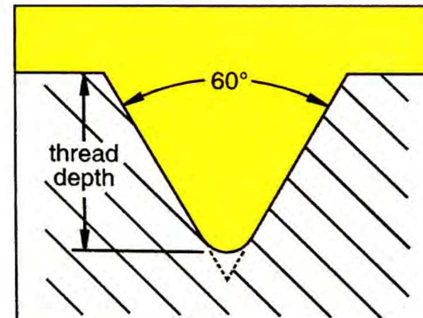
NOTE: These are nominal thread depths for full profile inserts. When using partial profile inserts, reduce the initial doc and increase the number of passes. When threading work-hardening materials, e.g. stainless austenitic steel, the infeed should not be less than .003 of an inch.

Image courtesy of Kennametal

Infeed for Internal Threading

Internal UN Threads – Recommendations for Steel Workpieces (<300 BHN)

tpi	4	5	6	7	8	9	10	11	12	13	14	16	18	20	24	28	32	36	40	44	48
thread depth	.1353	.1082	.0902	.0773	.0676	.0601	.0541	.0492	.0451	.0416	.0386	.0338	.0300	.0270	.0225	.0193	.0169	.0150	.0135	.0123	.0112
# passes 1	.0303	.0255	.0213	.0183	.0169	.0150	.0145	.0132	.0131	.0120	.0117	.0107	.0106	.0102	.0101	.0096	.0084	.0075	.0067	.0061	.0056
2	.0125	.0105	.0090	.0076	.0073	.0062	.0064	.0055	.0054	.0050	.0048	.0043	.0044	.0042	.0042	.0039	.0035	.0031	.0029	.0025	.0023
3	.0096	.0083	.0069	.0058	.0053	.0047	.0046	.0044	.0041	.0038	.0037	.0034	.0033	.0032	.0032	.0033	.0027	.0023	.0021	.0019	.0017
4	.0081	.0068	.0057	.0049	.0047	.0040	.0038	.0035	.0035	.0032	.0031	.0028	.0028	.0027	.0027	.0025	.0023	.0021	.0018	.0018	.0011
5	.0071	.0060	.0050	.0043	.0041	.0035	.0034	.0031	.0031	.0028	.0027	.0025	.0025	.0024	.0023						
6	.0064	.0054	.0045	.0039	.0036	.0032	.0031	.0028	.0028	.0025	.0025	.0029	.0023	.0022							
7	.0059	.0050	.0041	.0036	.0033	.0029	.0028	.0026	.0026	.0023	.0023	.0021	.0021	.0021							
8	.0055	.0046	.0038	.0033	.0030	.0027	.0026	.0024	.0024	.0022	.0021	.0020	.0029								
9	.0052	.0043	.0036	.0031	.0028	.0025	.0024	.0022	.0022	.0021	.0020	.0019									
10	.0049	.0041	.0034	.0029	.0027	.0024	.0023	.0021	.0021	.0020	.0019	.0018									
11	.0046	.0039	.0032	.0028	.0026	.0023	.0022	.0020	.0020	.0019	.0018										
12	.0044	.0037	.0031	.0027	.0025	.0022	.0021	.0019	.0019	.0018											
13	.0042	.0036	.0030	.0026	.0024	.0021	.0020	.0018													
14	.0041	.0035	.0029	.0025	.0023	.0020	.0019	.0017													
15	.0040	.0034	.0028	.0024	.0022	.0019															
16	.0039	.0033	.0027	.0023	.0021	.0019															
17	.0038	.0032	.0026	.0022																	
18	.0037	.0031	.0025	.0021																	
19	.0036																				
20	.0035																				



internal thread form

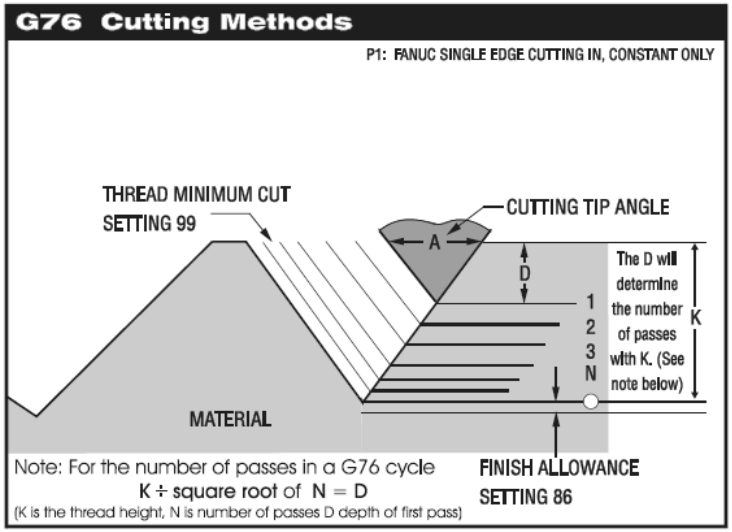
NOTE: These are nominal thread depths for full profile inserts. When using partial profile inserts, reduce the initial doc and increase the number of passes. When threading work-hardening materials, e.g. stainless austenitic steel, the infeed should not be less than .003 of an inch.

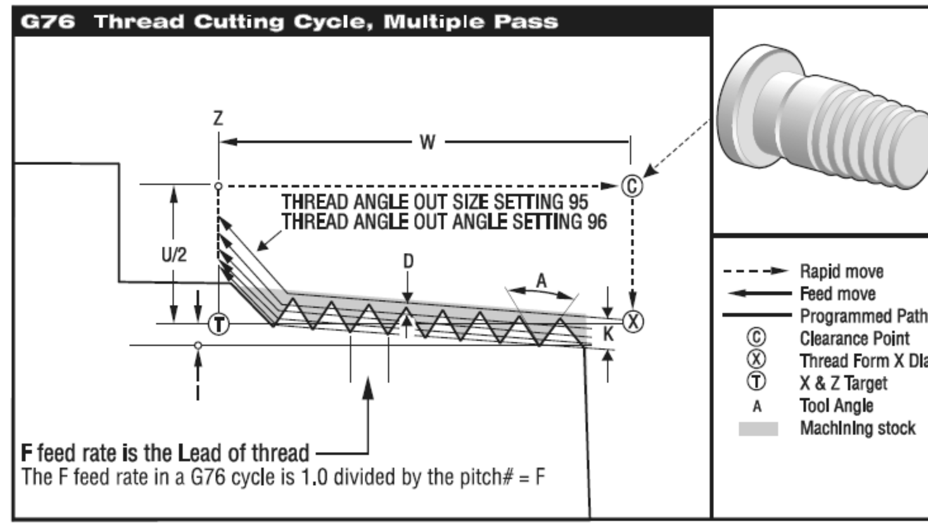
Image courtesy of Kennametal

Notes G76

1. The G76 canned cycle can be used for threading both straight or tapered (pipe) threads. With G76 a programmer can easily command multiple cutting passes along the length of a thread.
2. The height of the thread is specified in K. The height of the thread is defined as the distance from the crest of the thread to the root. The calculated depth of the thread will be K less the finish allowance. Setting 86 (THREAD FINISH ALLOWANCE) is this stock allowance for a finish pass allowance, if needed.
3. The depth of the first cut of the thread is specified in D. This also determines the number of passes over the thread based on the value of K and the cutting method used. $D = k/\sqrt{N}$ where N=passes.
    1. The depth of the last cut on the thread can be controlled with Setting 99 (THREAD MINIMUM CUT).
    2. The last cut will never be less than this value. The default value is .001 inches/.01 mm.
4. The feed rate F : $F = \frac{1}{TPI}$; e.g. ½-13, F=1/13=.0769

G76 Cutting Methods



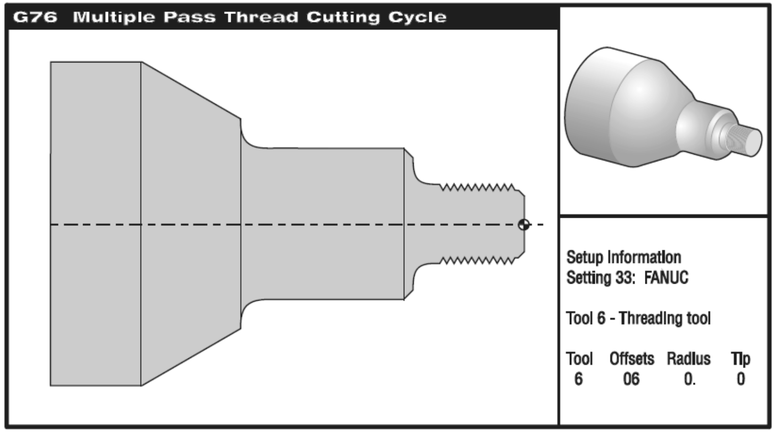


The thread taper distance amount is specified with the I command. It is measured from the target end position in X and Z axis down to the point in X axis where this cycle begins and is a radius amount. A conventional O.D. taper thread will have a negative I value and a conventional I.D. taper thread will have a positive I value.

Image courtesy of Haas

Example G76…

```
O00113 (G76 Multiple threading cycle to
   machine a 3/4-16 O.D. thread)
N10 G28
N20 T606 (O.D. THREADING TOOL)
N30 G97 S720 M03
N40 G54 G00 X0.85 Z1. M08 (Rapid to
   start point above diameter of the part)
N50 Z0.2 M23 (Z start point, chamfer at
   end of thread ON)
N60 G76 X0.674 Z-1.25 K0.0383
D0.0122 F0.0625 (G76 Multiple pass
O.D. thread)
N70 M09
N80 G28
N90 M30
```



Angle out of threads at end:
M23 Chamfer On (DEFAULT)
M24 Chamber Off

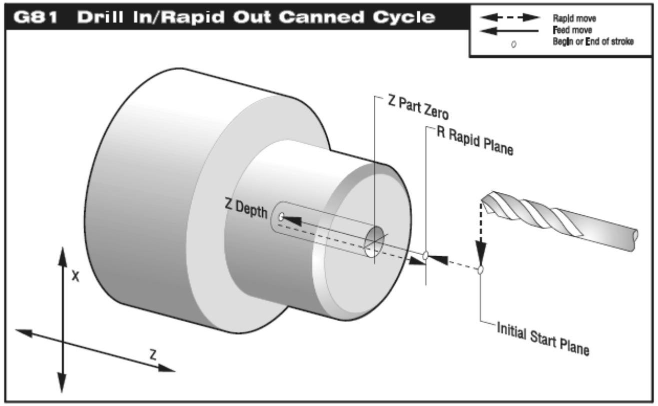
Image courtesy of Haas

G76 M23 and M24 Explained

- **M23 Chamfer (angle out of thread) at End of Thread is ON (DEFUALT)**
  - An angle out of thread move can improve the appearance and functionality of a thread. This M23 commands the control to exit the thread with angle out move on a thread executed by a G76 or G92. This M code is modal and is also the default. It remains in effect until changed by M24. Refer to Settings 95 and 96 to control the move distance and angle. M23 will again be active, with an M30, RESET, or a POWER ON condition.
  - SETTING 95 (THREAD CHAMFER SIZE) – The distance of angling out of the thread.
    - The distance is designated thread pitch, so that if 1.0 is in Setting 95 and the threading feed rate is .05, then the angle out distance will be .0500. The default in Setting 95 is 1.000.
  - SETTING 96 (THREAD CHAMFER ANGLE) – Angle out of thread chamfer.
    - The default angle of 45 degrees is in Setting 96.

- **M24 Chamfer (angle out of thread) at End of Thread is OFF**
  - An M24 commands the control to perform no angle out departure move at the end of a G76 or G92 threading cycle. This M code is modal. M24 is cancelled with an M23 (Chamfer at End of Thread ON), RESET, M30 or a POWER ON condition.

G81 Drilling

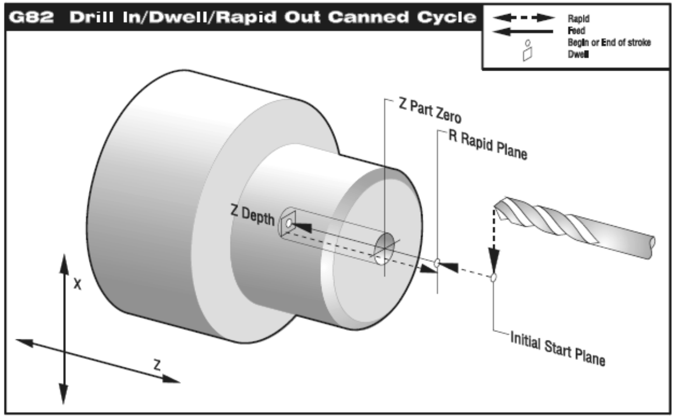



```
O00119 (G81 Drilling; ½" drill)
N1 G28
N2 T101 (1/2 DIA. DRILL) (Tool 1 Offset 1)
N3 G97 S1450 M03
N4 G54 G00 X0. Z1. M08 (Rapid to Initial Start Point)
N5 G81 Z-0.625 R0.1 F0.005 (G81 Drilling Cycle)
N6 G80 G00 Z1. M09 (G80 cancels G81)
N7 G28
N8 M30
```

X* Absolute X-axis rapid location.
Z* Absolute Z-depth (feeding to Z-depth starting from R-plane).
W* Incremental Z-depth (feeding to Z-depth starting from R-plane).
R Rapid to R-plane (where you rapid, to start feeding).
F Feed rate.
* Indicates optional

Image courtesy of Haas

G82 Spot Drill/Counterbore



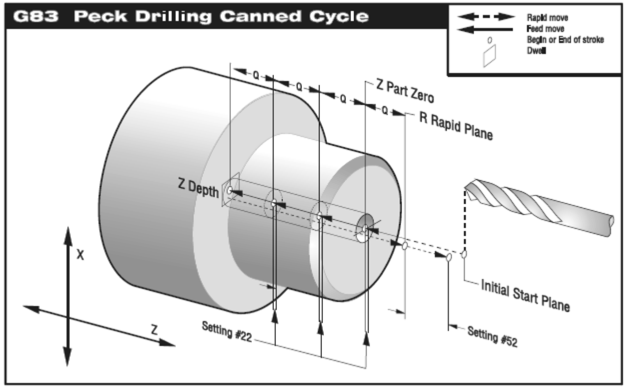

```
O00120 (G82 Drilling with a Dwell; ½" FB drill)
N1 G28
N2 T202 (1/2 DIA. FLAT BOTTOM DRILL) (Tool 2 Offset 2)
N3 G97 S1450 M03
N4 G54 G00 X0. Z1. M08 (Rapid to Initial Start Point)
N5 G82 Z-0.625 P0.5 R0.1 F0.005 (G82 Drill with a Dwell at Z Depth Cycle)
N6 G80 G00 Z1. M09 (G80 cancels G82)
N7 G28
N8 M30
```

X* Absolute X-axis rapid location.
Z* Absolute Z-depth (feeding to Z-depth starting from R-plane).
W* Incremental Z-depth (feeding to Z-depth starting from R-plane).
P Dwell time at Z-depth
R Rapid to R-plane (where you rapid, to start feeding).
F Feed rate.
* Indicates optional

Prof. Steven S. Saliterman

Image courtesy of Haas

G83 Deep Hole Peck Drill

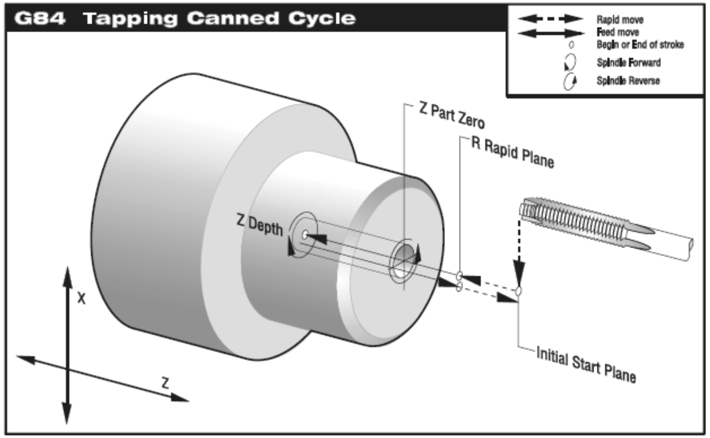



```
O00121 (G83 Peck Drilling; ½" drill)
N1 G28
N2 T303 (1/2 DIA. DRILL) (Tool 3 Offset 3)
N3 G97 S1820 M03
N4 G54 G00 X0. Z1. M08 (Rapid to Initial
Start Point)
N5 G83 Z-1.5 Q0.2 R0.1 F0.005 (G83 Peck
Drilling Cycle with Q)
N6 G80 G00 Z1. M09
N7 G28
N8 M30
```

X* Absolute X-axis rapid location.
Z* Absolute Z-depth (feeding to Z-depth starting from R-plane).
W* Incremental Z-depth (feeding to Z-depth starting from R-plane).
Q* Pecking depth amount, always incremental (if I, J and K are not used).
I* Size of first peck depth (if Q is not used).
J* Amount reducing each peck after first peck depth (if Q is not used).
K* Minimum peck depth (if Q is not used).
P Dwell time at Z-depth.
R Rapid to R-plane (where you rapid, to start feeding).
F Feed rate.
*** Indicates optional**

Image courtesy of Haas

G84 Tapping




```
O00123 (G84 Tapping)
N1 G28
N2 T404 (3/8-16 TAP) (Tool 4 Offset 4)
N3 G97 S650 M05 (G84 will turn on the spindle for you)
N4 G54 G00 X0. Z1. M08 (Rapid to Initial Start Point)
N5 G84 Z-0.75 R0.2 F0.0625 (G84 Tapping Cycle)
N6 G80 G00 Z1. M09
N7 G28
N8 M30
```

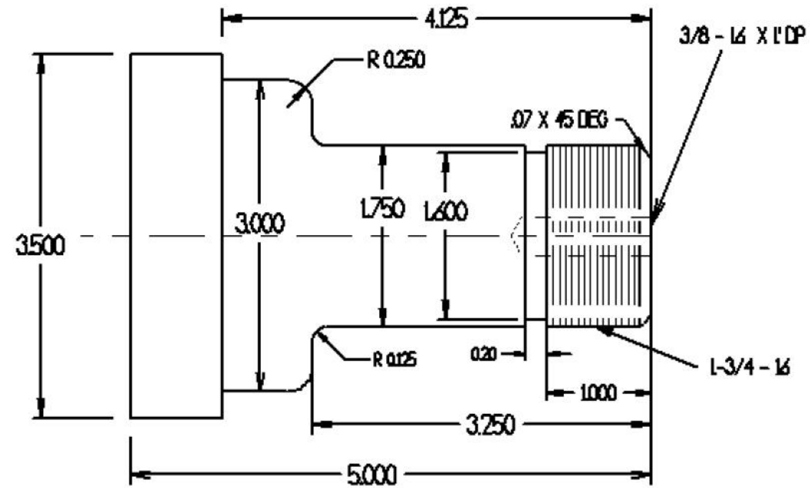
X* Absolute X-axis rapid location.
Z* Absolute Z-depth (feeding to Z-depth. starting from R-plane).
W* Incremental Z-depth (feeding to Z-depth starting from R-plane).
R Rapid to R-plane (where you rapid, to start feeding).
F Feed rate.

* Optional
Use G184 for left handed taps.

Image courtesy of Haas

Thread G76, Drill G83 and Tap G84

```
G50 S200                          (Clamp spindle speed 200 rpm)
G28                               (Rapid to machine zero)
M01                               (Optional program stop)
T404                              (OD THREAD tool)
G97 S655  M3                      (Start spindle, D=1.748, SFM=300, RPM=655)
G54 G00 X1.848 Z0.2 M08           (Rapid to start position, coolant on)
G76 X1.673 Z-1.1 K.039 D.0125 A58  F.0625   (Thread cycle, X=minor diam., Z=into
groove,  K=thread depth from table, D=first pass from table, A=60-2 degrees to cut on
both sides, F=feed=1/TPI=1/16, using Kennametal table )
M09                               (Coolant off)
G28
M01
T1111                              (5/16 DRILL)
G97 S976  M3                      (Start spindle, D=5/16=.313, SFM=80, RPM=976)
G54 G00 X0 Z0.2 M08               (Rapid to start position, coolant on)
G83 X0 Z-1.3 R.1 Q0.3125 F.006    (Drill peck cycle)
G80                               (Cancel canned cycle)
G00 Z0.2                          (Return to start)
G28
M09
M01
T1010                             (3/8-16 TAP)
G97 S200 M05                      (RPM given and stop spindle)
G54 G00 X0 Z0.5  M08              (Rapid to start position, coolant on)
G84 X0 Z-1 R0.5 F0.0625           (Tap cycle, R plane 500 in front, feed=1/TPI=1/16)
G80
G28
M30                               (End)
```



T0404 Thread G761-3/4 -16 UN 2A 1.0" back from face SFM=300 ft/min. Major diameter 1.748, Minor diameter 1.673

T1111 5/16 Drill G83 1.3" deep SFM=80 ft/min F.006"/rev peck diameter of drill

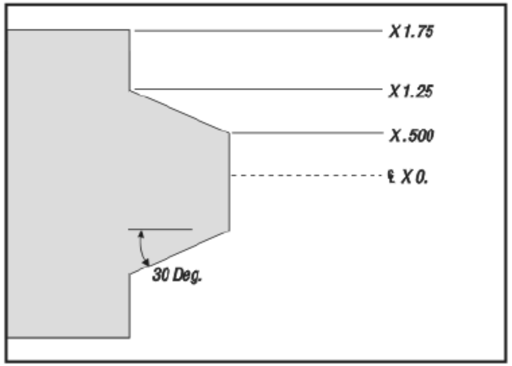
T1010 Tap 3/8-16 x 1.0" deep at 200 rpm

Image courtesy of Haas

G90, G92 & G94– Modal Cycles

- G90 O.D./I .D. Turning
  - It can be used for simple turning. Since it is modal, you can do multiple passes for turning by just specifying a new X location for successive passes.
- G92 Thread Cutting
  - It can be used for simple threading. Since it is modal, you can do multiple passes for threading by just specifying a new X location for successive passes.
  - Straight threads can be made by just specifying X, Z and F.
  - By adding I a pipe or taper thread can be cut. The amount of taper is defined with the I value added to the X value target point.
  - At the end of the thread, an automatic chamfer is executed before reaching the target default for this chamfer is one thread at 45 degrees.
- G94 End Face Cutting
  - You can do multiple passes for facing by just specifying a new Z location for successive passes.
  - Straight end facing cuts can be made by just specifying X, Z and F. By adding K a conical face can be cut. The coning amount is defined with the K value that is added to the Z value target point.

G90 Modal Turning with TNC

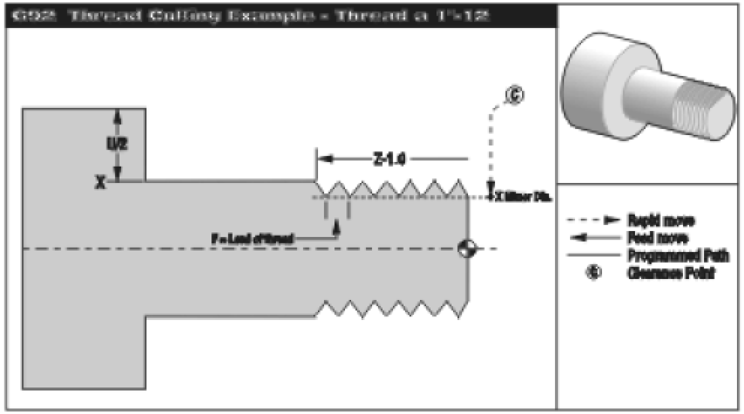


X* Absolute X-axis target location
Z* Absolute Z-axis target location
U* Incremental X-axis target distance, diameter
W* Incremental Z-axis target distance
I * Distance and direction of X axis ta per, radius value
F Feed rate
* Indicates optional

```
O00131 (G90 Modal Turning with TNC)
N11 G28
N12 T101 (O.D. TURNING TOOL)
N13 G50 S3000
N14 G97 S480 M03
N15 G54 G00 X1.85 Z1. M08 (Rapid to Start Point)
N16 G96 S390
N17 Z0.1
N18 G90 G42 X1.65 Z-0.6495 I-0.375 F0.006 (Rough 30 Deg. angle to X2.3476)
N19 X1.55 (Additional Pass) (Dia. using G90 and TNC)
N20 X1.45 (Additional Pass)
N21 X1.35 (Additional Pass)
N22 X1.25 (Additional Pass)
N23 G00 G40 X3.1 Z1. M09 (TNC Departure)
N24 M05
N25 G28
N26 M30
```

Image courtesy of Haas

G92 Modal Threading

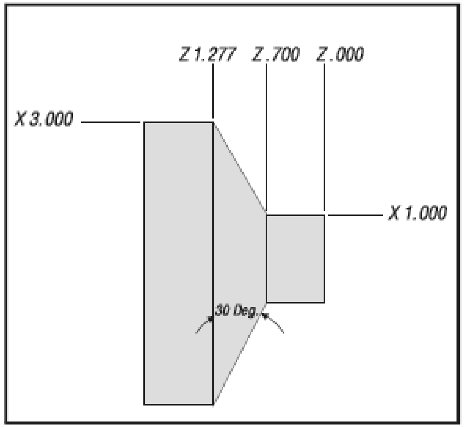


X* Absolute X-axis target location
Z* Absolute Z-axis target location
U* Incremental X-axis target distance, diameter
W* Incremental Z-axis target distance
I* Distance and direction of X axis taper, radius value
F Feed rate
* Indicates optional

```
O00133 (G92 Modal Threading)
N10 (1.0-12UN Thread)
N11 G28
N12 T404 (O.D. THREADING TOOL)
N13 G97 S825 M03
N14 G54 G00 X1.1 Z1. M08 (Rapid to Start Point)
N15 Z0.25
N16 G92 X.98 Z-1.05 F0.08333 M23 (First Pass of a G92 O.D. Thread Cycle)
N17 X.96   (Additional Pass)
N18 X.94   (Additional Pass)
N19 X.935 (Additional Pass)
N20 X.93   (Additional Pass)
N21 X.925 (Additional Pass)
N22 X.9225 (Additional Pass)
N23 X.92   (Additional Pass)
N24 X.9175 (Additional Pass)
N25 X.9155 (Additional Pass)
N26 X.915 (Additional Pass)
N27 X.9148 (Additional Pass)
N28 G00 X1.1 Z1. M09
N29 M05
N30 G28
N31 M30N31 M30
```

Image courtesy of Haas

G94 Modal End Facing with TNC



X* Absolute X-axis target location
Z* Absolute Z-axis target location
U* Incremental X-axis target distance, diameter
W* Incremental Z-axis target distance
K* Distance and direction of Z axis coning
F Feed rate
* Indicates optional

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

Image courtesy of Haas

Summary

- Axis X & Z, Absolute & Incremental Positioning
- Programming
  - G Commands
    - G00, G01, G02, G03, G04, G28, G50, G51, G96, G97, G98, G99
  - M Machine Controls
  - Letter Address Codes
- Tool Nose Compensation
  - G40, G41, G42
- Canned Cycles Turning and Grooving
  - G70, G71, G73, G74, G75, G76
- Canned Cycles for Drilling and Tapping
  - G80, G81, G82, G84

References

- Haas CNC Lathe Operator 2014
- Haas CNC Lathe Programming 2015
- Haas Lathe Operator's Manual 2016
- Haas Programming Workbook 2015
- Haas EBay Tutorials