HAAS FACTORY OUTLET
A DIVISION OF PRODUCTIVITY INC

HAAS MILL PROGRAMMING

Revised 05/25JP

PROGRAMMING MANUAL

Productivity Inc®

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 Mill Operator Manual

https://www.haascnc.com/service/online-operator-s-manuals/mill-operator-s-manual/mill---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

G54 WORK OFFSETS

Work Offsets define where a work piece is located on the table.

Work Offsets available are **G54-G59, G110-G129**, and **G154 P1-P99. G110-G129** and **G154 P1-P20** refer to the same Work Offsets.

A useful feature is to set up multiple work pieces on the table and machining multiple parts in one machine cycle. This is accomplished by assigning each work piece to a different Work Offset.

For more information, reference the G-code section of this manual. Below is an example of machining multiple parts in one cycle. The program uses M97 Local Sub-Program Call in the cutting operation.

```
%
O40005 (Work offsets ex-prog) ;
(G54 X0 Y0 is center left of part) ;
(Z0 is on top of the part) ;
(T1 is a drill) ;
(BEGIN PREPARATION BLOCKS) ;
T1 M06 (Select tool 1) ;
G00 G90 G40 G49 G54(Safe startup) ;
X0 Y0 ;
(Move to first work coordinate position-G54) ;
S1000 M03 (Spindle on CW) ;
G43 H01 Z0.1 (Tool offset 1 on) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
M97 P1000 (Call local Subprogram) ;
G00 Z3. (Rapid retract) ;
G90 G110 G17 G40 G80 X0. Y0. ;
(Move to second work coordinate position-G110)
;
M97 P1000 (Call local Subprogram) ;
G00 Z3. (Rapid Retract) ;
G90 G154 P22 G17 G40 G80 X0. Y0. ;
(Move to third work coordinate position-G154
P22) ;
M97 P1000 (Call local Subprogram) ;
(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) ;
N1000 (Local subprogram) ;
G81 F41.6 X1. Y2. Z-1.25 R0.1 (Begin G81) ;
(1st hole) ;
X2. Y2. (2nd hole) ;
G80 (Cancel G81) ;
M99 ;
%
```

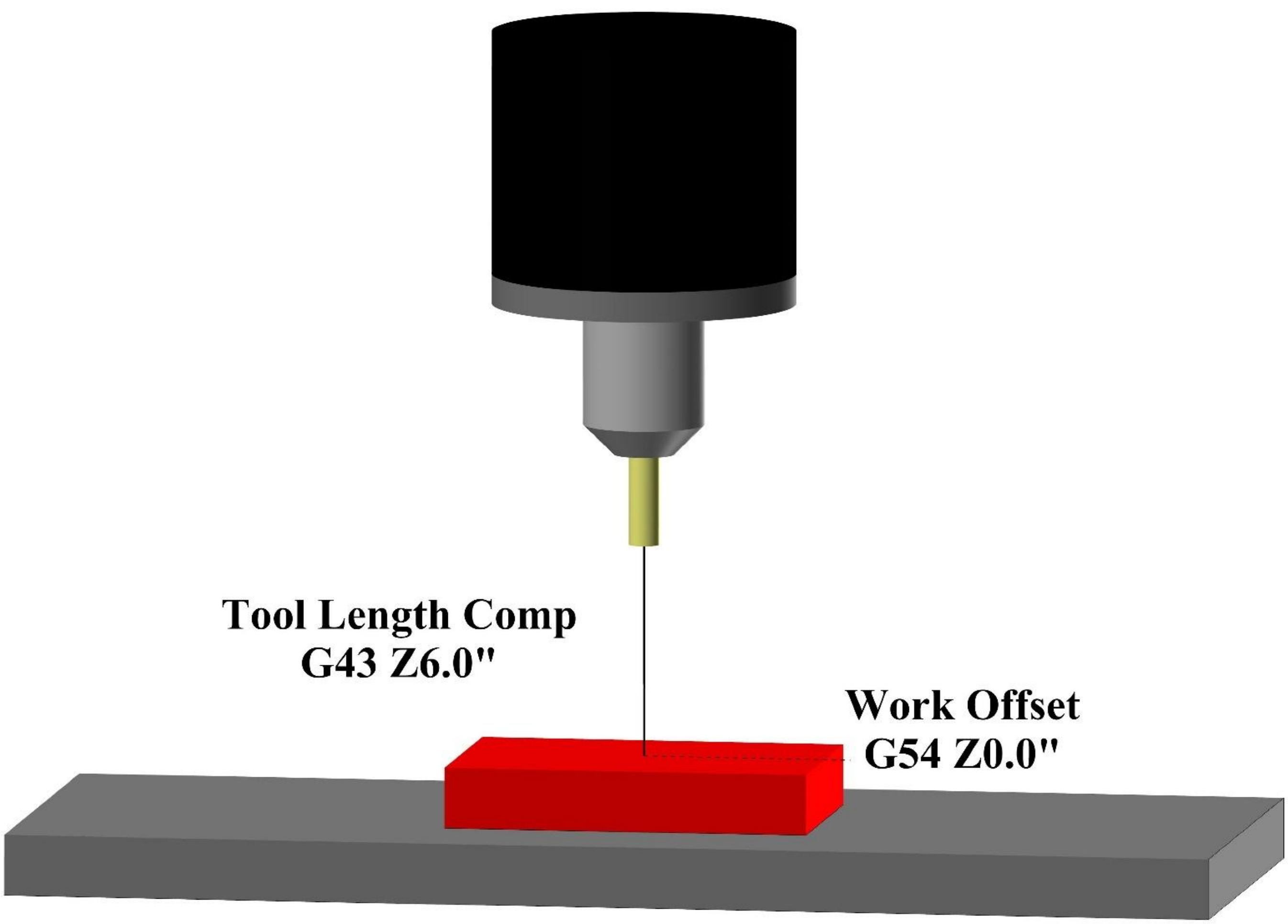
G43 TOOL LENGTH COMPENSATION + (ADD) / G44 TOOL LENGTH COMP - (SUBTRACT) (GROUP 08)

A G43 code selects tool length compensation in the positive direction; the tool length in the offsets page is added to the commanded axis position. A G44 code selects tool length compensation in the negative direction; the tool length in the offsets page is subtracted from the commanded axis position. A non-zero H address must be entered to select the correct entry from the offsets page.

NOTE Starting in NGC Software Version 100.21.000.1100, the tool length offset behavior has been modified on Haas machines in the following ways: By default, tool length offsets will now always be applied, unless a G49/H00 (Mill) or Txx00 offset (Lathe) is explicitly specified. On mills, when a tool change occurs, the tool length offset will automatically update to match the new tool. The current tool length offset and mill group 8 code will now persist through power cycles.

CAUTION: The tool length nn value should match the nn value from the M06 Tnn tool change command to avoid a possible collision.

Setting 15 - *H & T Code Agreement* controls whether the nn value needs to match in the Tnn and Hnn arguments. If Setting 15 is ON and the Tnn and Hnn do not match, Alarm 332 - H and T Not Matched is generated.



G52 SET WORK COORDINATE SYSTEM (GROUP 00 OR 12)

G52 works differently depending on the value of Setting 33. Setting 33 selects the Fanuc or Haas style of coordinates.

If FANUC is selected, *G52* is a group 00 G-code. This is a global work coordinate shift. The values entered in the *G52* line of the work offset page are added to all work offsets. All of the *G52* values in the work offset page will be set to zero (0) when powered on, reset is pressed, changing modes, at the end of the program, by an *M30*, *G92* or a *G52 X0 Y0 Z0 A0 B0*. When using a *G92* (Set Work Coordinate Systems Shift Value), in Fanuc format, the current position in the current work coordinate system is shifted by the values of *G92* (*X*, *Y*, *Z*, *A*, and *B*). The values of the *G92* work offset are the difference between the current work offset and the shifted amount commanded by *G92*.

If HAAS is selected, *G52* is a group 00 G-code. This is a global work coordinate shift. The values entered into the *G52* line of the work offset page are added to all work offsets. All of the *G52* values will be set to zero (0) by a *G92*. When using a *G92* (Set Work Coordinate Systems Shift Value), in Haas format, the current position in the current work coordinate system is shifted by the values of *G92* (*X*, *Y*, *Z*, *A*, and *B*). The values of the *G92* work offset are the difference between the current work offset and the shifted amount commanded by *G92* (Set Work Coordinate Systems Shift Value).

G53 NON-MODAL MACHINE COORDINATE SELECTION (GROUP 00)

This code temporarily cancels work coordinate offsets and uses the machine coordinate system. This code will also ignore tool offsets. In the machine coordinate system, the zero point for each axis is the position where the machine goes when a Zero Return is performed. G53 will revert to this system for the block in which it is commanded.

G92 SET WORK COORDINATE SYSTEMS SHIFT VALUE (GROUP 00)

This G-code does not move any of the axes; it only changes the values stored as user work offsets.

A G92 command cancels any G52 in effect for the commanded axes. Example: G92X1.4 cancels the G52 for the X-Axis. The other axes are not affected.

The G92 shift value is displayed at the bottom of the Work Offsets page and may be cleared there if necessary. It is also cleared automatically after power-up, and any time [ZERO RETURN] and [ALL] or [ZERO RETURN] and [SINGLE] are used.

G92 Clear Shift Value From Within a Program

G92 shifts may be canceled by programming another G92 shift to change the current work offset back to the original value.

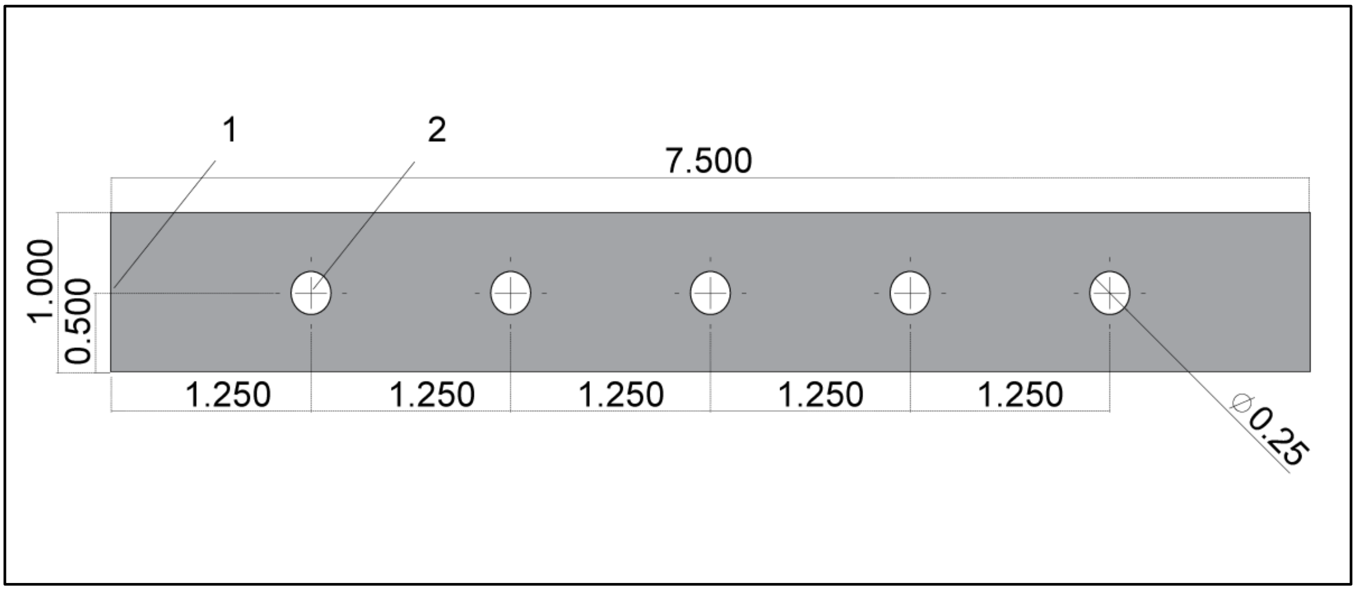
ABSOLUTE VS. INCREMENTAL POSITIONING (G90, G91)

Absolute (G90) and incremental positioning (G91) define how the control interprets axis motion commands.

When you command axis motion after a G90 code, the axes move to that position relative to the origin of the coordinate system currently in use.

When you command axis motion after a G91, 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.

Figure 1 shows a part with 5 equally spaced Ø0.25" (13 mm) diameter holes. The hole depth is 1.00" (25.4 mm) and the spacing is 1.250" (31.75 mm) apart.



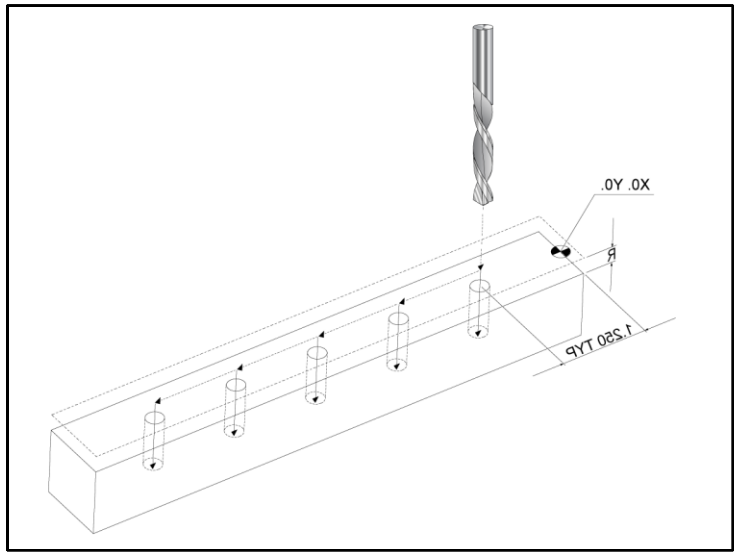
Absolute / Incremental Sample Program. G54 X0. Y0. for Incremental [1], G54 for Absolute [2]

Below are two example programs that drill the holes as shown in the drawing, with a comparison between absolute and incremental positioning.

We start the holes with a center drill, and finish drilling the holes with a 0.250" (6.35 mm) drill bit. We use a 0.200" (5.08 mm) depth of cut for the center drill and 1.00" (25.4 mm) depth of cut for the 0.250" drill. G81, Drill Canned Cycle, is used to drill holes.

MILL INCREMENTAL POSITIONING EXAMPLE

```
%
O40002 (Incremental ex-prog) ;
N1 (G54 X0 Y0 is center left of part) ;
N2 (Z0 is on top of the part) ;
N3 (T1 is a center drill) ;
N4 (T2 is a drill) ;
N5 (T1 PREPARATION BLOCKS) ;
N6 T1 M06 (Select tool 1) ;
N7 G00 G90 G40 G49 G54 (Safe startup) ;
N8 X0 Y0 (Rapid to 1st position) ;
N9 S1000 M03 (Spindle on CW) ;
N10 G43 H01 Z0.1(Tool offset 1 on) ;
N11 M08(Coolant on) ;
N12 (T1 CUTTING BLOCKS) ;
N13 G99 G91 G81 F8.15 X1.25 Z-0.3 L5 ;
N14 (Begin G81, 5 times) ;
N15 G80 (Cancel G81) ;
N16 (T1 COMPLETION BLOCKS) ;
N17 G00 G90 G53 Z0. M09 (rapid retract, clnt off);
N18 M01 (Optional stop) ;
N19 (T2 PREPARATION BLOCKS) ;
N20 T2 M06 (Select tool 2) ;
N21 G00 G90 G40 G49 (Safe startup) ;
N22 G54 X0 Y0 (Rapid to 1st position) ;
N23 S1000 M03 (Spindle on CW) ;
N24 G43 H02 Z0.1(Tool offset 2 on) ;
N25 M08(Coolant on) ;
N26 (T2 CUTTING BLOCKS) ;
N27 G99 G91 G81 F21.4 X1.25 Z-1.1 L5 ;
N28 G80 (Cancel G81) ;
N29 (T2 COMPLETION BLOCKS) ;
N30 G00 Z0.1 M09 (Rapid retract, clnt off) ;
N31 G53 G90 G49 Z0 M05 (Z home, spindle off) ;
N32 G53 Y0 (Y home) ;
N33 M30 (End program) ;
%
```



MILL ABSOLUTE POSITIONING EXAMPLE

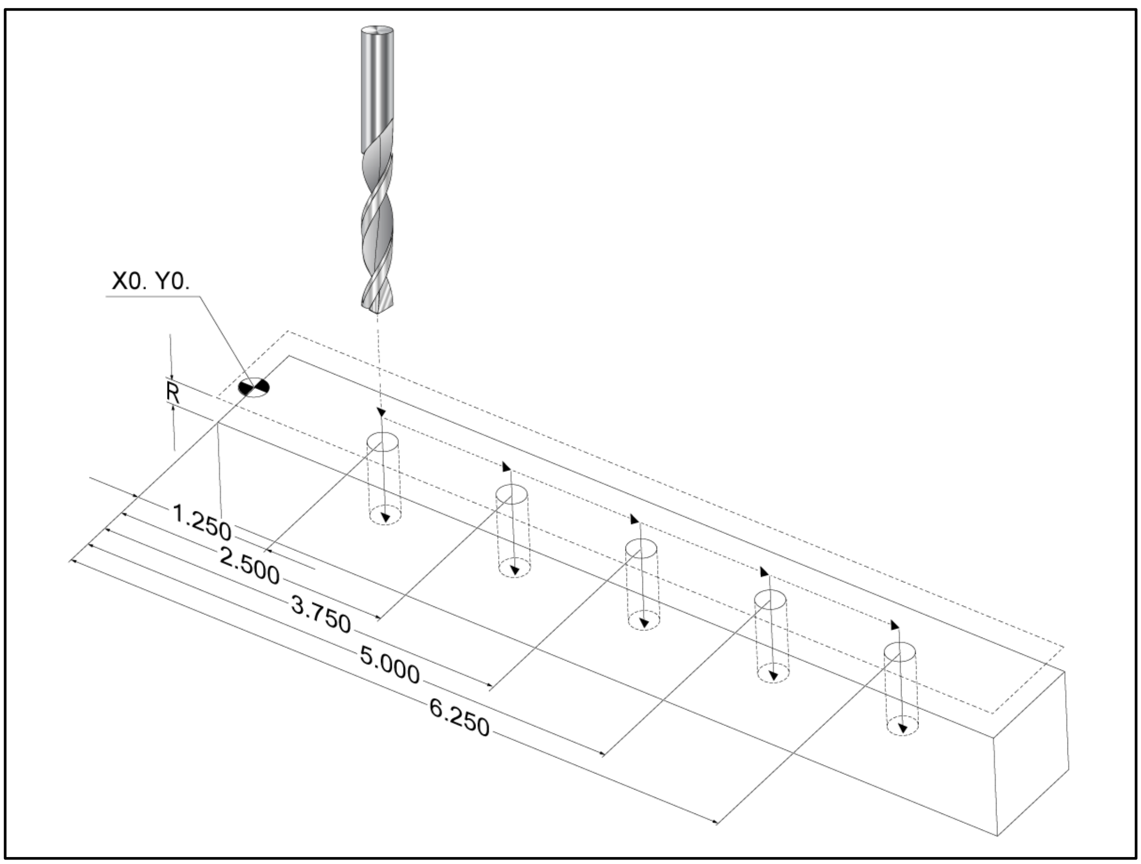
The absolute program method needs more lines of code than the incremental program. The programs have similar preparation and completion sections.

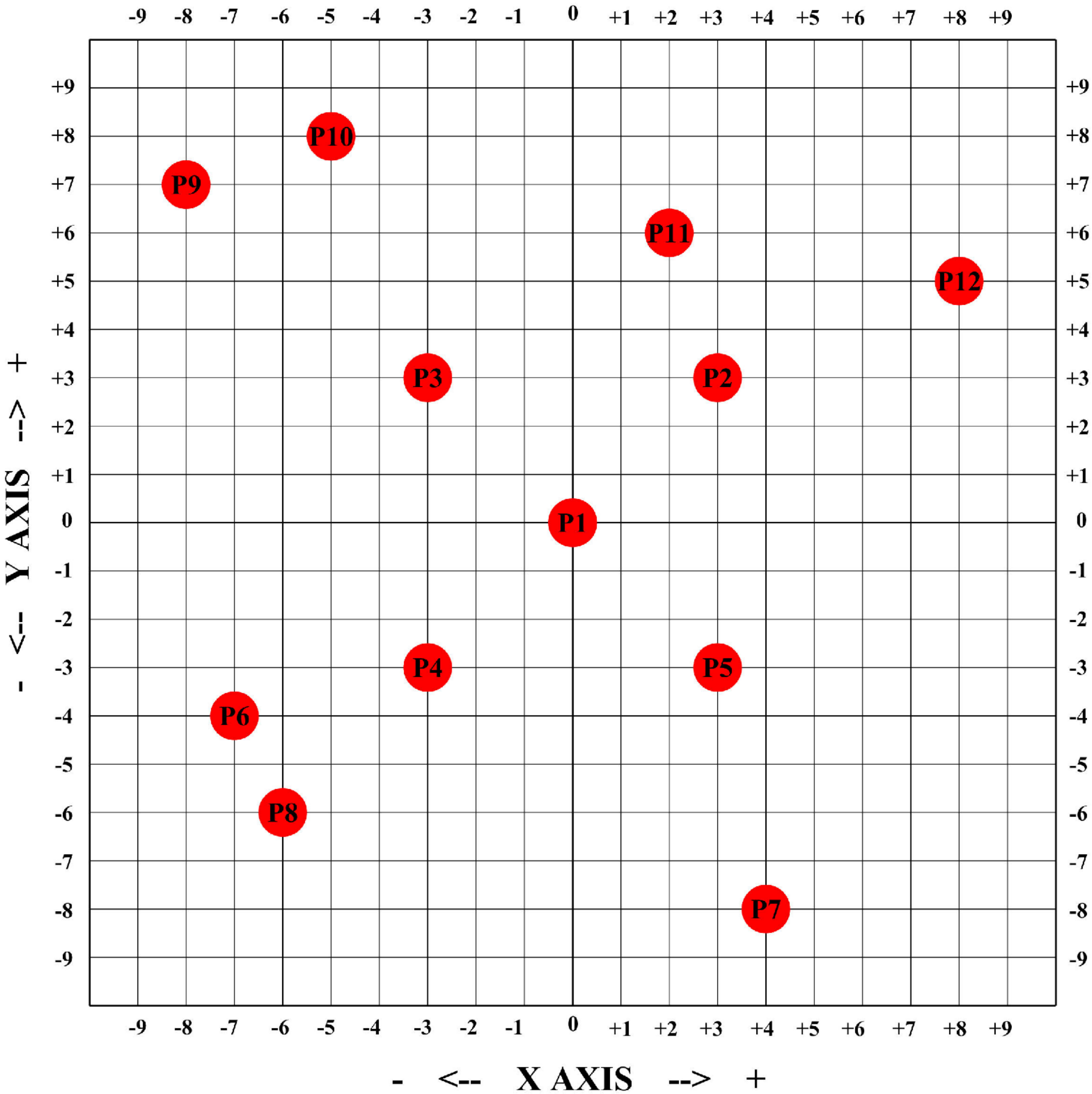
Look at line N13 in the incremental programming example, where the center drill operation begins. G81 uses the loop address code, Lnn, to specify the number of times to repeat the cycle. The address code L5 repeats this process (5) times. Each time the canned cycle repeats, it moves the distance that the optional X and Y values specify. In this program, the incremental program moves 1.25" in X from the current position with each loop, and then does the drill cycle.
For each drill operation, the program specifies a drill depth 0.1" deeper than the actual depth, because motion starts from 0.1" above the part.

In absolute positioning, G81 specifies the drill depth, but it does not use the loop address code. Instead, the program gives the position of each hole on a separate line. Until G80 cancels the canned cycle, the control does the drill cycle at each position.

The absolute positioning program specifies the exact hole depth, because the depth starts at the part surface (Z=0).

```
%
O40003 (Absolute ex-prog) ;
N1 (G54 X0 Y0 is center left of part) ;
N2 (Z0 is on top of the part) ;
N3 (T1 is a center drill) ;
N4 (T2 is a drill) ;
N5 (T1 PREPARATION BLOCKS) ;
N6 T1 M06 (Select tool 1) ;
N7 G00 G90 G40 G49 G54 (Safe startup) ;
N8 X1.25 Y0 (Rapid to 1st position) ;
N9 S1000 M03 (Spindle on CW) ;
N10 G43 H01 Z0.1 (Tool offset 1 on) ;
N11 M08 (Coolant on) ;
N12 (T1 CUTTING BLOCKS) ;
N13 G99 G81 F8.15 X1.25 Z-0.2 ;
N14 (Begin G81, 1st hole) ;
N15 X2.5 (2nd hole) ;
N16 X3.75 (3rd hole) ;
N17 X5. (4th hole) ;
N18 X6.25 (5th hole) ;
N19 G80 (Cancel G81) ;
N20 (T1 COMPLETION BLOCK) ;
N21 G00 G90 G53 Z0. M09 (Rapid retract, clnt off);
N22 M01 (Optional Stop) ;
N23 (T2 PREPARATION BLOCKS) ;
N24 T2 M06 (Select tool 2) ;
N25 G00 G90 G40 G49 (Safe startup) ;
N26 G54 X1.25 Y0 (Rapid to 1st position) ;
N27 S1000 M03 (Spindle on CW) ;
N28 G43 H02 Z0.1 (Tool offset 2 on) ;
N29 M08 (Coolant on) ;
N30 (T2 CUTTING BLOCKS) ;
N31 G99 G81 F21.4 X1.25 Z-1. (1st hole) ;
N32 X2.5 (2nd hole) ;
N33 X3.75 (3rd hole) ;
N34 X5. (4th hole) ;
N35 X6.25 (5th hole) ;
N36 G80 (Cancel G81) ;
N37 (T2 COMPLETION BLOCKS) ;
N38 G00 Z0.1 M09 (Rapid retract, Clnt off) ;
N39 G53 G49 Z0 M05 (Z home, Spindle off) ;
N40 G53 Y0 (Y home) ;
N41 M30 (End program) ;
%
```





ABSOLUTE AND INCREMENTAL EXERCISE

Absolute (G90) From X0 and Y0			Incremental (G91) From Starting P# to Ending P#		
	X	Y		X	Y
P1			P6-P7		
P2			P7-P8		
P3			P8-P9		
P4			P9-P10		
P5			P10-P11		
P6			P11-P12		

CUTTING G-CODES

The main cutting G-codes are categorized into interpolation motion and canned cycles. Interpolation motion cutting codes are broken down into:

G01 - Linear Interpolation Motion
G02 - Clockwise Circular Interpolation Motion
G03 - Counter-Clockwise Circular Interpolation Motion
G12 - Clockwise Circular Pocket Milling
G13 - Counter-Clockwise Circular Pocket Milling

Linear Interpolation Motion

G01 Linear Interpolation Motion is used to cut straight lines. It requires a feedrate, specified with the Fnnn.nnnn address code. Xnn.nnnn, Ynn.nnnn, Znn.nnnn, and Annn.nnn are optional address codes to specify cut. Subsequent axis motion commands will use the feed rate specified by G01 until another axis motion, G00, G02, G03, G12, or G13 is commanded.
Corners can be chamfered using the optional argument Cnn.nnnn to define the chamfer. Corners can be rounded using the optional address code Rnn.nnnn to define the radius of the arc. Refer to G01 Linear Interpolation Motion (Group 01) for more information

Circular Interpolation Motion

G02 and G03 are the G-codes for circular cutting motions. Circular Interpolation Motion has several optional address codes to define the arc or circle. The arc or circle begins cutting from the current cutter position [1] to the geometry specified within the G02/ G03 command.
Arcs can be defined using two different methods. The preferred method is to define the center of the arc or circle with I, J and/or K and to define the end point [3] of the arc with an X, Y and/or Z. The I J K values define the relative X Y Z distances from the starting point [2] to the center of the circle. The X Y Z values define the absolute X Y Z distances from the starting point to the end point of the arc within the current coordinate system. This is also the only method to cut a circle. Defining only the I J K values and not defining the end point X Y Z values will cut a circle.
The other method to cut an arc is to define the X Y Z values for the end point and to define the radius of the circle with an R value.

G01 LINEAR INTERPOLATION MOTION (GROUP 01)

F - Feedrate
* **X** - X-Axis motion command
* **Y** - Y-Axis motion command
* **Z** - Z-Axis motion command
* **A** - A-Axis motion command
* **B** - B-Axis motion command
* **C** - C-axis motion command
* **,R** - Radius of the arc
* **,C** - Chamfer distance

*indicates optional

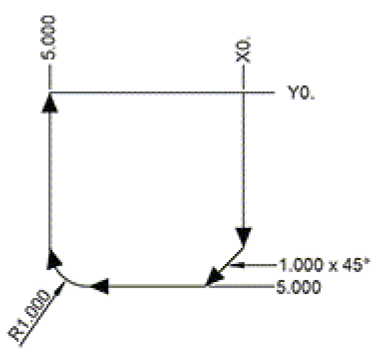
G01 moves the axes at a commanded feed rate. It is primarily used to cut the workpiece. A *G01* feed can be a single axis move or a combination of the axes. The rate of axes movement is controlled by feedrate (*F*) value. This *F* value can be in units (inch or metric) per minute (*G94*) or per spindle revolution (*G95*), or time to complete the motion (*G93*). The feedrate value (*F*) can be on the current program line, or a previous line. The control will always use the most recent *F* value until another *F* value is commanded. If in *G93*, an *F* value is used on each line. Refer also to *G93*.

G01 is a modal command, which means that it will stay in effect until canceled by a rapid command such as *G00* or a circular motion command like *G02* or *G03*.

Once a *G01* is started all programmed axes move and reach the destination at the same time. If an axis is not capable of the programmed feedrate the control will not proceed with the *G01* command and an alarm (max feedrate exceeded) will be generated.

Corner Rounding and Chamfering Example

Corner Rounding and Chamfering Example #1



%
O60011 (G01 CORNER ROUNDING & CHAMFER) ;
(G54 X0 Y0 is at the top-right of part) ;
(Z0 is on top of the part) ;
(T1 is an end mill) ;
(BEGIN PREPARATION BLOCKS) ;
T1 M06 (Select tool 1) ;
G00 G90 G40 G49 G54 (Safe startup) ;
G00 G54 X0 Y0 (Rapid to 1st position) ;
S1000 M03 (Spindle on CW) ;
G43 H01 Z0.1 (Activate tool offset 1) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;

```
G01 Z-0.5 F20. (Feed to cutting depth) ;
Y-5. ,C1. (Chamfer) ;
X-5. ,R1. (Corner-round) ;
Y0 (Feed to Y0.) ;
(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) ;
%
```

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). There must be a terminating linear interpolation block after the beginning block (a *G04* pause may intervene).

These two linear interpolation blocks specify a corner of intersection. If the beginning block specifies a *,C*, the value after the *,C* is the distance from the intersection to where the chamfer begins, and also the distance from the intersection to where the chamfer ends. If the beginning block specifies an *,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 and the endpoint of that arc. There can be consecutive blocks with chamfering or corner rounding specified. There must be movement on the two axes specified by the selected plane, whether the active plane is XY (*G17*), XZ (*G18*) or YZ (*G19*).

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

F - Feedrate
* **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 circle
* **X** - X-Axis motion command
* **Y** - Y-Axis motion command
* **Z** - Z-Axis motion command
* **A** - A-Axis motion command

*indicates optional

NOTE **I**,**J** and **K** is the preferred method to program a radius. **R** is suitable for general radii.

These G codes are used to specify circular motion. Two axes are necessary to complete circular motion and the correct plane, *G17-G19*, must be used. There are two methods of commanding a *G02* or *G03*, the first is using the **I**, **J**, **K** addresses and the second is using the **R** address.

Using I, J, K addresses

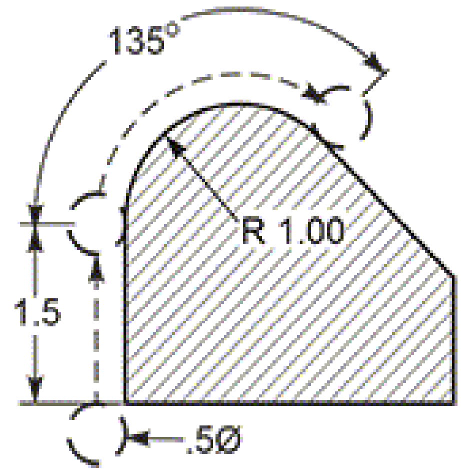
I, *J* and *K* address are used to locate the arc center in relation to the start point. In other words, the *I*, *J*, *K* addresses are the distances from the starting point to the center of the circle. Only the *I*, *J*, or *K* specific to the selected plane are allowed (*G17* uses *IJ*, *G18* uses *IK* and *G19* uses *JK*). The *X*, *Y*, and *Z* commands specify the end point of the arc. If the *X*, *Y*, and *Z* location for the selected plane is not specified, the endpoint of the arc is the same as the starting point for that axis.
To cut a full circle the *I*, *J*, *K* addresses must be used; using an R address will not work. To cut a full circle, do not specify an ending point (*X*, *Y*, and *Z*); program *I*, *J*, or *K* to define the center of the circle.
For example:
G02 I3.0 J4.0 (Assumes G17; XY plane) ;

Using the R address

The *R*-value defines the distance from the starting point to the center of the circle. Use a positive *R*-value for radii of 180 or less, and a negative *R*-value for radii more than 180.

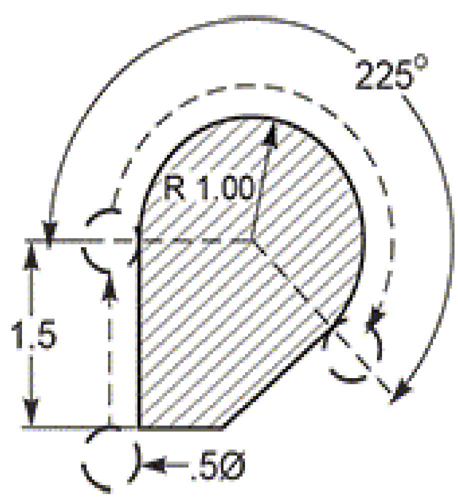
PROGRAMMING EXAMPLE

Positive *R* Address Programming Example




```
%
O60021 (G02 POSITIVE R ADDRESS) ;
(G54 X0 Y0 is at the bottom-left of part) ;
(Z0 is on top of the part) ;
(T1 is a .5 in dia endmill) ;
(BEGIN PREPARATION BLOCKS) ;
T1 M06 (Select tool 1) ;
G00 G90 G40 G49 G54 (Safe startup) ;
G00 G54 X-0.25 Y-0.25 (Rapid to 1st position) ;
S1000 M03 (Spindle on CW) ;
G43 H01 Z0.1 (Activate tool offset 1) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G01 Z-0.5 F20. (Feed to cutting depth) ;
G01 Y1.5 F12. (Feed to Y1.5) ;
G02 X1.884 Y2.384 R1.25 (CW circular 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) ;
%
```

Negative *R* Address Programming Example




```
%
O60022 (G02 NEGATIVE R ADDRESS) ;
(G54 X0 Y0 is at the bottom-left of part) ;
(Z0 is on top of the part) ;
(T1 is a .5 in dia endmill) ;
(BEGIN PREPARATION BLOCKS) ;
T1 M06 (Select tool 1) ;
G00 G90 G40 G49 G54 (Safe startup) ;
G00 G54 X-0.25 Y-0.25 (Rapid to 1st position) ;
S1000 M03 (Spindle on CW) ;
G43 H01 Z0.1 (Activate tool offset 1) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G01 Z-0.5 F20. (Feed to cutting depth) ;
G01 Y1.5 F12. (Feed to Y1.5) ;
G02 X1.884 Y0.616 R-1.25 (CW circular 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) ;
%
```

THREAD MILLING

Thread milling uses a standard *G02* or *G03* move to create the circular move in X-Y, then adds a Z move on the same block to create the thread pitch. This generates one turn of the thread; the multiple teeth of the cutter generate the rest. Typical block of code:
N100 G02 I-1.0 Z-.05 F5. (generates 1-inch radius for 20-pitch thread) ;
Thread milling notes:
Internal holes smaller than 3/8 inch may not be possible or practical. Always climb cut the cutter.
Use a *G03* to cut I.D. threads or a *G02* to cut O.D. threads. An I.D. right hand thread will move up in the Z-Axis by the amount of one thread pitch. An O.D. right hand thread will move down in the Z-Axis by the amount of one thread pitch. PITCH = 1/Threads per inch (Example - 1.0 divided by 8 TPI = .125)
This program I.D. thread mills a 1.5 diameter x 8 TPI hole with a 0.750" diameter x 1.0" thread hob.

1. To start, take the hole diameter (1.500). Subtract the cutter diameter .750 and then divide by 2. (1.500 - .75) / 2 = .375

The result (.375) is the distance the cutter starts from the I.D. of the part.

2. After the initial positioning, the next step of the program is to turn on cutter compensation and move to the I.D. of the circle.

3. The next step is to program a complete circle (*G02* or *G03*) with a Z-Axis command of the amount of one full pitch of the thread (this is called Helical Interpolation).

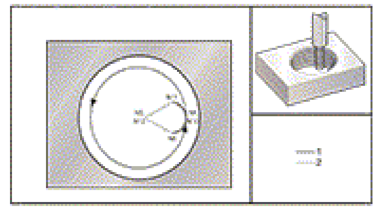
4. The last step is to move away from the I.D. of the circle and turn off cutter compensation.

You cannot turn cutter compensation off or on during an arc movement. You must program a linear move, either in the X or Y Axis, to move the tool to and from the diameter to cut. This move will be the maximum compensation amount that you can adjust.

THREAD MILLING EXAMPLE

Thread Milling Example, 1.5 Diameter X 8 TPI: [1]Tool Path, [2] Turn on and off cutter compensation.

NOTE Many thread mill manufacturers offer free online software to help you create your threading programs.



```
%
O60023 (G03 THREAD MILL 1.5-8 UNC) ;
(G54 X0 Y0 is at the center of the bore) ;
(Z0 is on top of the part) ;
(T1 is a .5 in dia thread mill) ;
(BEGIN PREPARATION BLOCKS) ;
T1 M06 (Select tool 1) ;
G00 G90 G40 G49 G54 (Safe startup) ;
G00 G54 X0 Y0 (Rapid to 1st position) ;
S1000 M03 (Spindle on CW) ;
G43 H01 Z0.1 (Activate tool offset 1) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G01 Z-0.5156 F50. (Feed to starting depth) ;
(Z-0.5 minus 1/8th of the pitch = Z-0.5156) ;
G41 X0.25 Y-0.25 F10. D01 (cutter comp on) ;
G03 X0.5 Y0 I0 J0.25 Z-0.5 (Arc into thread) ;
(Ramps up by 1/8th of the pitch) ;
I-0.5 J0 Z-0.375 F20. (Cuts full thread) ;
(Z moving up by the pitch value to Z-0.375) ;
X0.25 Y0.25 I-0.25 J0 Z-0.3594 (Arc out of thread) ;
(Ramp up by 1/8th of the pitch) ;
G40 G01 X0 Y1 (cutter comp off) ;
(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) ;
%
```

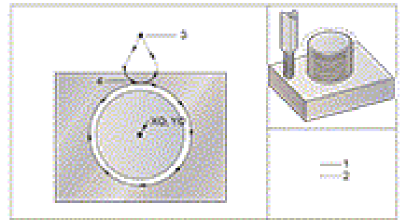
Specific line description:

N5 = XY at the center of the hole
N7 = Thread depth, minus 1/8 pitch
N8 = Enable Cutter Compensation
N9 = Arcs into thread, ramps up by 1/8 pitch
N10 = Cuts full thread, Z moving up by the pitch value
N11 = Arcs out of thread, ramps up 1/8 pitch
N12 = Cancel Cutter Compensation

NOTE Maximum cutter compensation adjustability is 0.175.

OUTSIDE DIAMETER (O.D.) THREAD MILLING

O.D. Thread Milling Example, 2.0 diameter post x 16 TPI: [1] Tool Path [2] Rapid Positioning, Turn on and off cutter compensation, [3] Start Position, [4] Arc with Z.



```
%
O60024 (G02 G03 THREAD MILL 2.0-16 UNC) ;
(G54 X0 Y0 is at the center of the post) ;
(Z0 is on top of the opost) ;
(T1 is a .5 in dia thread mill) ;
(BEGIN PREPARATION BLOCKS) ;
T1 M06 (Select tool 1) ;
G00 G90 G40 G49 G54 (Safe startup) ;
G00 G54 X0 Y2.4 (Rapid to 1st position) ;
S1000 M03 (Spindle on CW) ;
G43 H01 Z0.1 (Activate tool offset 1) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G00 Z-1. (Rapids to Z-1.) ;
G01 G41 D01 X-0.5 Y1.4 F20. (Linear move) ;
(Cutter comp on) ;
G03 X0 Y0.962 R0.5 F25. (Arc into thread) ;
G02 J-0.962 Z-1.0625 (Cut threads while lowering Z) ;
G03 X0.5 Y1.4 R0.5 (Arc out of thread) ;
G01 G40 X0 Y2.4 F20. (Linear move) ;
(Cutter comp off) ;
(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) ;
%
```

NOTE A cutter compensation move can consist of any X or Y move from any position as long as the move is greater than the amount being compensated.

SINGLE-POINT THREAD MILLING

This program is for a 1.0" diameter hole with a cutter diameter of 0.500" and a thread pitch of 0.125 (8TPI). This program positions itself in Absolute *G90* and then switches to G91 Incremental mode on line *N7*.

The use of an *Lxx* value on line *N10* allows us to repeat the thread milling arc multiple times, with a Single-Point Thread Mill.

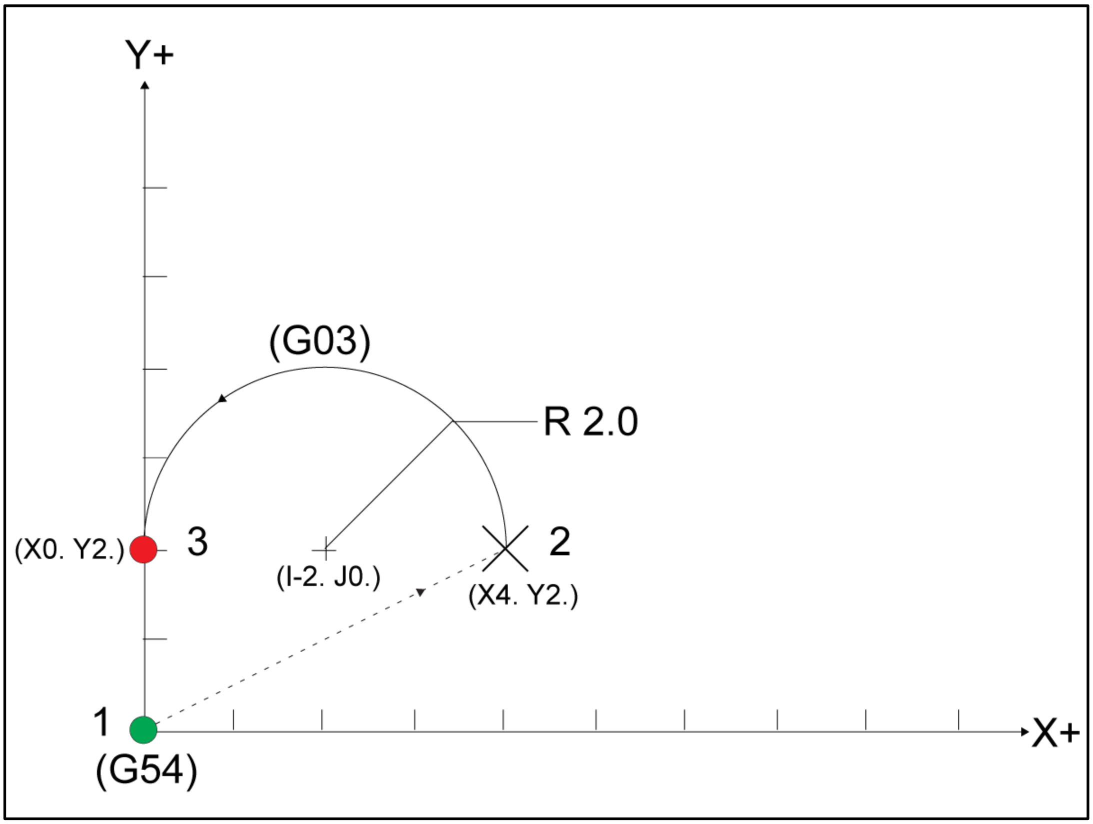
```
%
O60025 (G03 SNGL PNT THREAD MILL 1.5-8
UNC) ;
(G54 X0 Y0 is at the center of the bore) ;
(Z0 is on top of the part) ;
(T1 is a .5 in dia thread mill) ;
(BEGIN PREPARATION BLOCKS) ;
T1 M06 (Select tool 1) ;
G00 G90 G40 G49 G54 (Safe startup) ;
G00 G54 X0 Y0 (Rapid to 1st position) ;
S1000 M03 (Spindle on CW) ;
G43 H01 Z0.1 (Activate tool offset 1) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G91 G01 Z-0.5156 F50. (Feed to starting depth) ;
(Z-0.5 minus 1/8th of the pitch = Z-0.5156) ;
G41 X0.25 Y-0.25 F20. D01 (Cutter comp on) ;
G03 X0.25 Y0.25 I0 J0.25 Z0.0156 (Arc into
thread) ;
(Ramps up by 1/8th of the pitch) ;
I-0.5 J0 Z0.125 L5 (Thread cut, repeat 5 times) ;
X-0.25 Y0.25 I-0.25 J0 Z0.0156 (Arc out of thread)
;
(Ramps up by 1/8th of the pitch) ;
G40 G01 X-0.25 Y-0.25 (Cutter comp off) ;
(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) ;
%
```

Specific line description:
N5 = XY at the center of the hole
N7 = Thread depth, minus 1/8 pitch. Switches to G91
N8 = Enable Cutter Compensation
N9 = Arcs into thread, ramps up by 1/8 pitch
N10 = Cuts full thread, Z moving up by the pitch value
N11 = Arcs out of thread, ramps up 1/8 pitch
N12 = Cancel Cutter Compensation
N13 = Switches back to G90 Absolute positioning

Helical Motion
Helical (spiral) motion is possible with *G02* or *G03* by programming the linear axis that is not in the selected plane. This third axis will be moved along the specified axis in a linear manner, while the other two axes will be moved in the circular motion. The speed of each axis will be controlled so that the helical rate matches the programmed feedrate.

G02 / G03 EXAMPLES

Below are examples of using the two different methods to cut a 2" (or 2 mm) radius, 180 degree, counterclockwise arc. The tool starts at X0 Y0 [1], moves to the starting point of the arc [2], and cuts the arc to the end point [3]:



Method 1:

```
%
T01 M06 ;
G00 X4. Y2. ;
G01 F20.0 Z-0.1 ;
G03 F20.0 I-2.0 J0. X0. Y2. ;
M30 ;
%
```

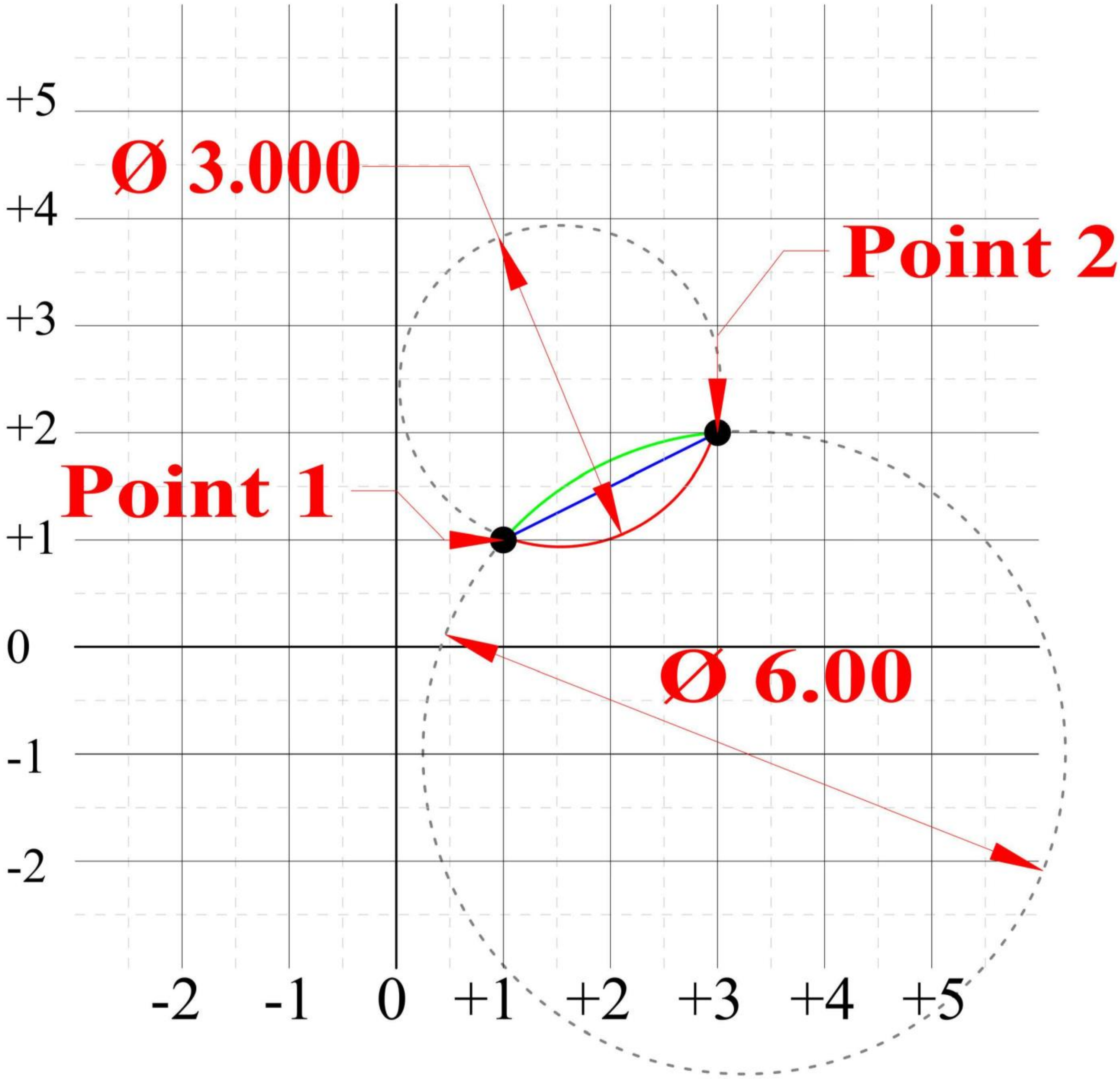
Method 2:

```
%
T01 M06 ;
...
G00 X4. Y2. ;
G01 F20.0 Z-0.1 ;
G03 F20.0 X0. Y2. R2. ;
M30 ;
%
```

Below is an example of how to cut a 2" (or 2 mm) **radius circle**:

```
%
T01 M06 ;
G00 X4. Y2. ;
G01 F20.0 Z-0.1 ;
G02 F20.0 I2.0 J0. ;
M30 ;
%
```

TOOL PATH G01 AND G02

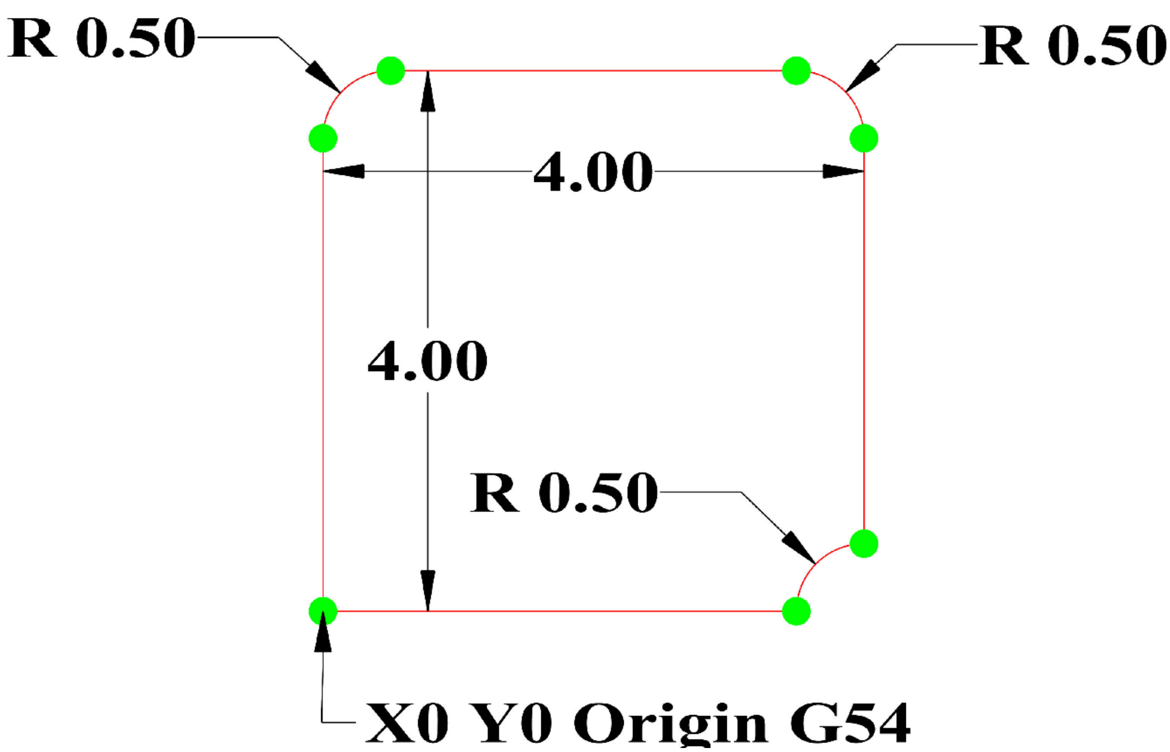


Point 1 to Point 2

Line 1	Line 2	Line 3
G00 X1.0 Y1.0 **G01 X3.0 Y2.0 F20.** M30	G00 X1.0 Y1.0 **G02 X3.0 Y2.0 R3.0 F20.** M30	G00 X1.0 Y1.0 **G03 X3.0 Y2.0 R1.5 F20.** M30

Notes

TOOL PATH EXERCISE 1



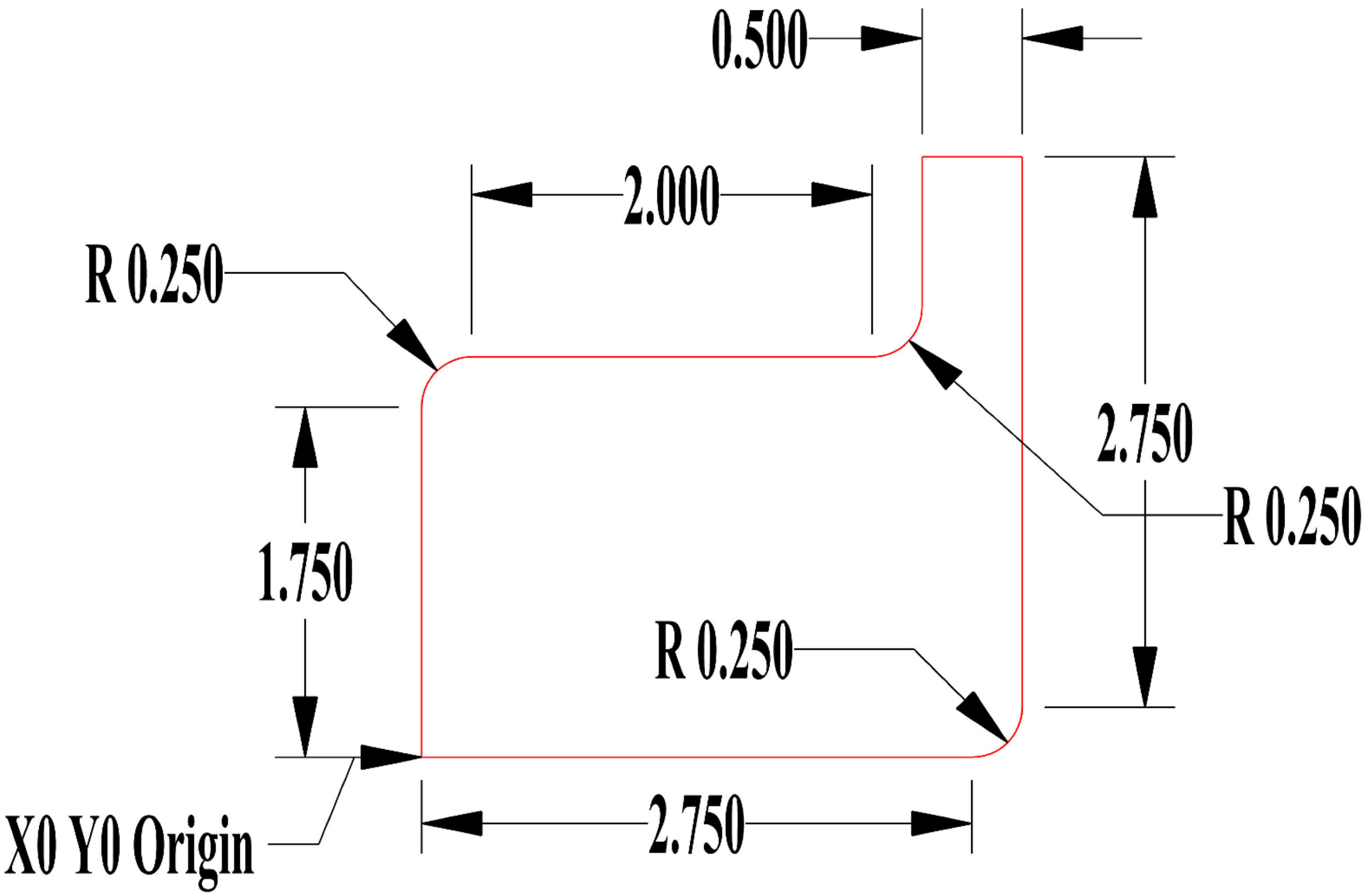
***** ALL EXERCISE THROUGH OUT THIS MANUAL WILL BE CLIMB MILLING. PROGRAM ALL YOUR TOOL PATH <u>CLOCKWISE</u> AROUND THE OUTSIDE OF PROFILE *****

HAAS Simulator: Go to MDI Mode:

T1 M06
G54 G90
G00 X0. Y0.
G01 X__ Y__ F20.0 →
…
…
…
…
…
…
M30

Run program in Graphic Mode.
Save program referencing page number.

TOOL PATH EXERCISE 2

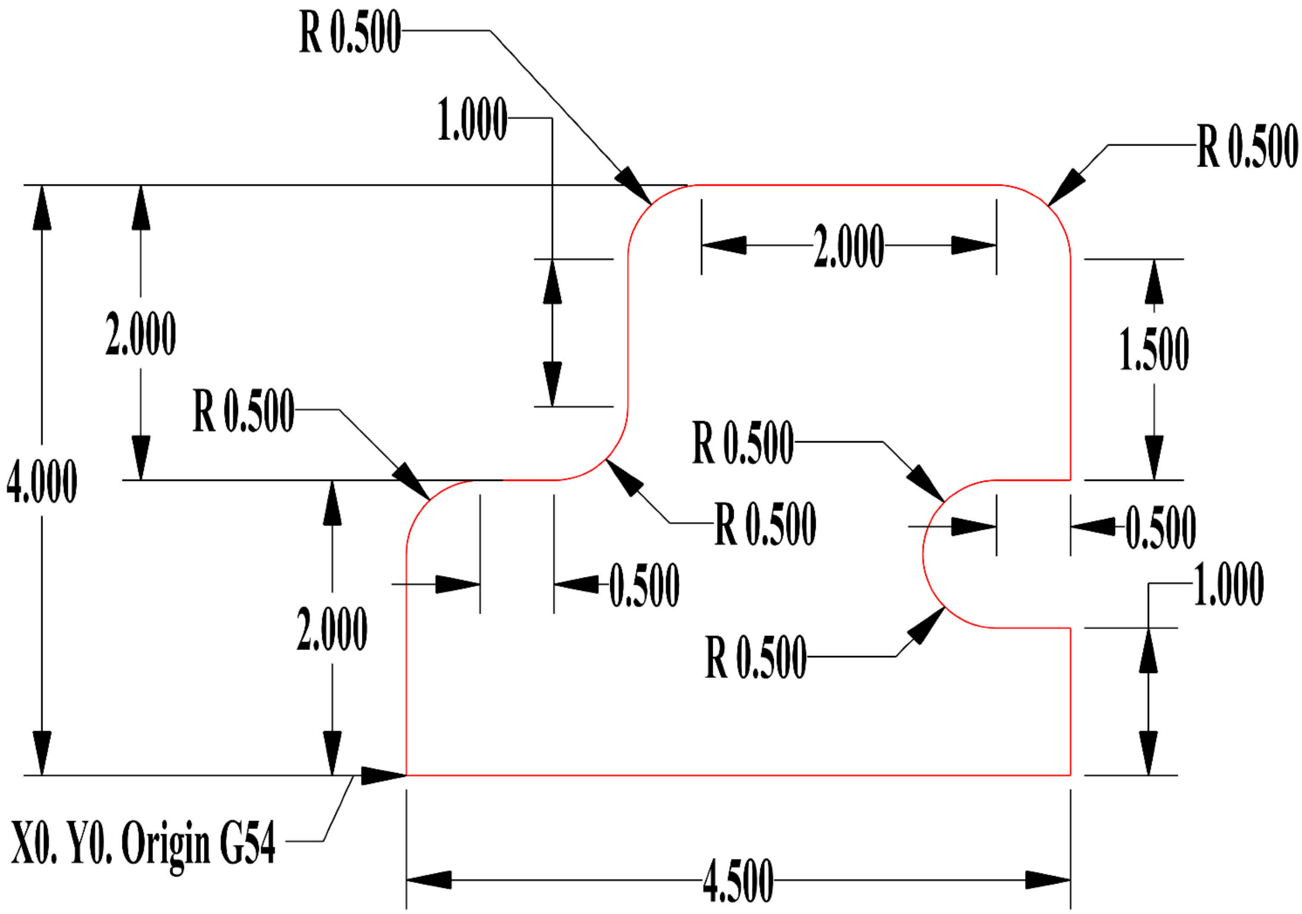


Use G90, Absolute with G02 and G03
HAAS Simulator: Go to MDI Mode:

T1 M06
G54 G90
G00 X0. Y0.
G01 X__ Y__ F20.0 →
…
…
…
…
…
…
M30

Run program in Graphic Mode.
Save program referencing page number.

TOOL PATH EXERCISE 3

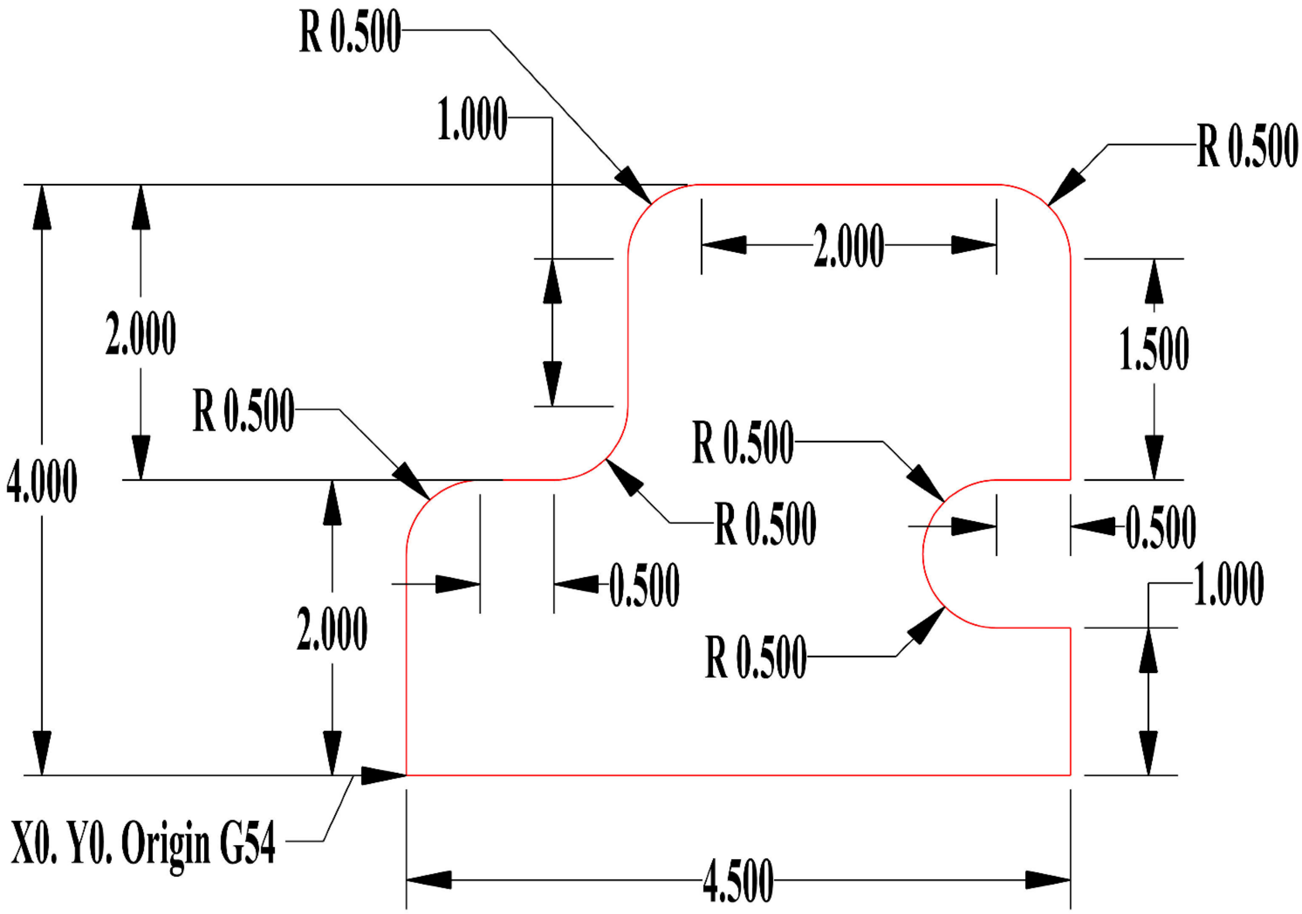


Use G02 and G03 with I and J ONLY

HAAS Simulator: Go to MDI Mode:

T1 M06
G54 G90
G00 X0. Y0.
G01 F20.0 .... →
...
...
...
...
...
...
M30

Run program in Graphic Mode.
Save program referencing page number.

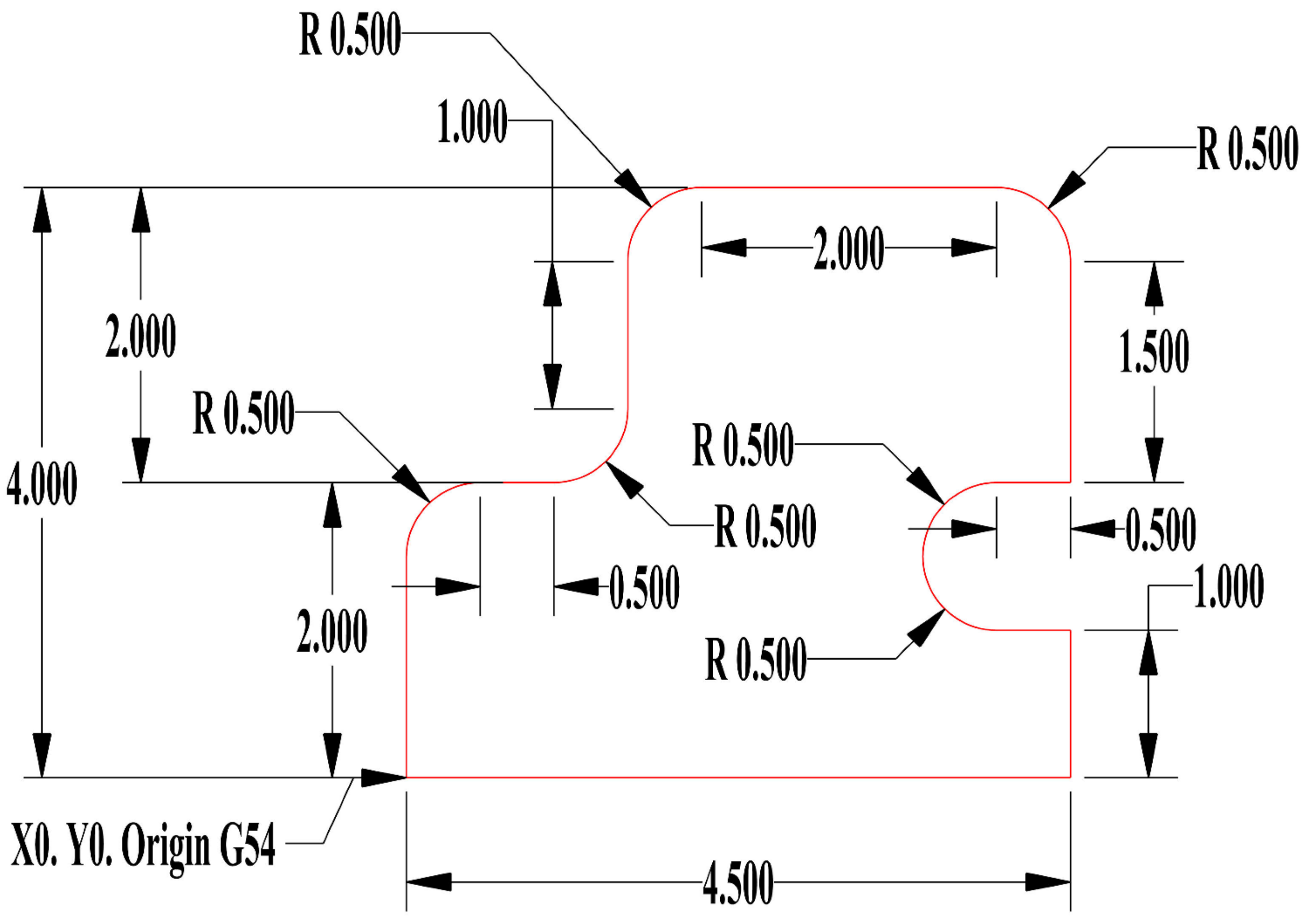


Use only G01 with ,R#.#
HAAS Simulator: Go to MDI Mode:

T1 M06
G54 G90
G00 X0. Y0.
G01 F20.0 .... →
...
...
...
...
...
...
M30

Run program in Graphic Mode.
Save program referencing page number.

TOOL PATH EXERCISE 5

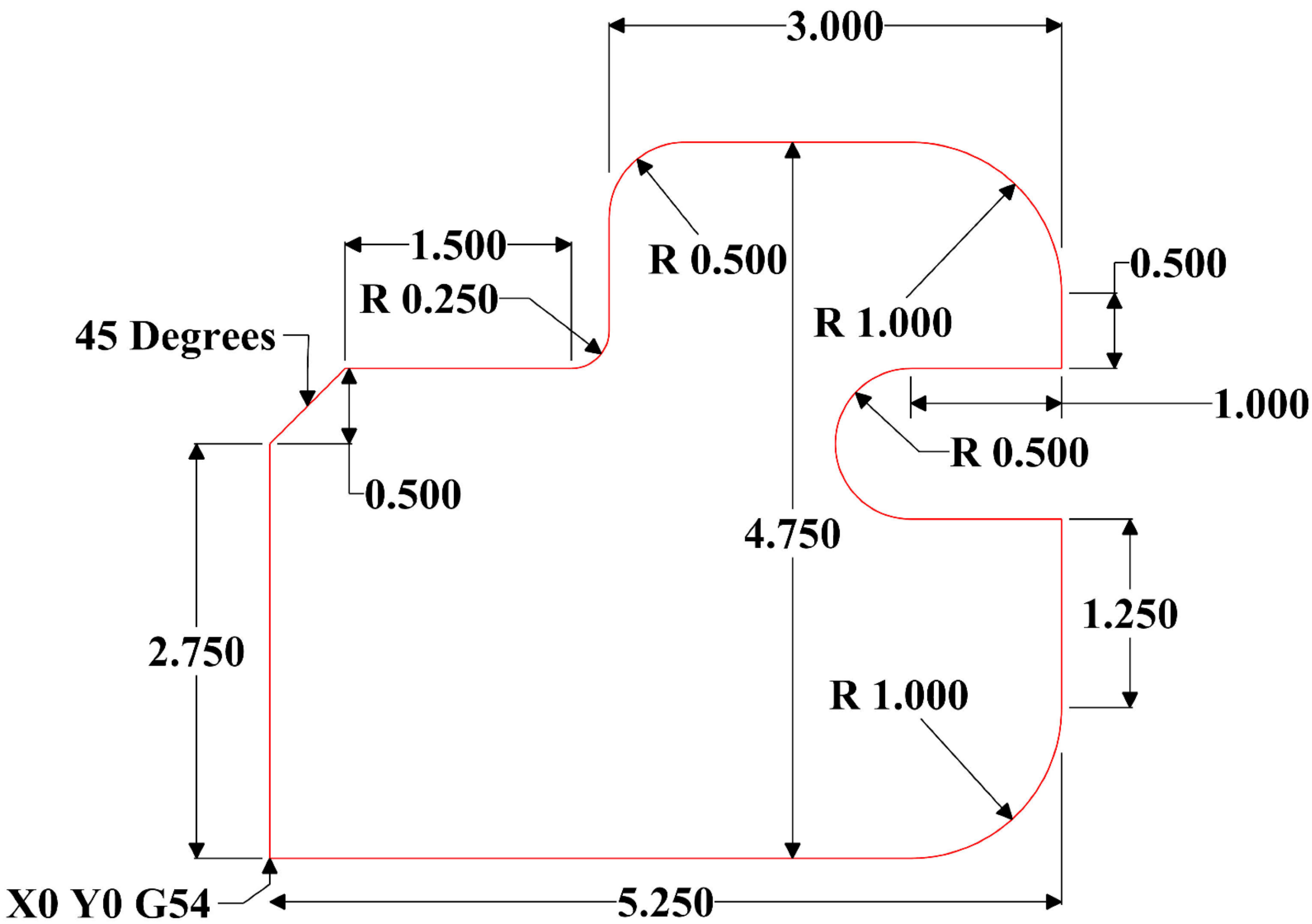


Use G91, Incremental with G01, R#.#, G02 or G03
HAAS Simulator: Go to MDI Mode:

T1 M06
G54 G90
G00 X0. Y0.
G01 F20.0 .... →
...
...
...
...
...
...
M30

Run program in Graphic Mode.
Save program referencing page number.

TOOL PATH EXERCISE 6



Program Tool Path Using any Method. G01 w/ ,R#.#, G02, or G03. Incremental or Absolute.
HAAS Simulator: Go to MDI Mode:

T1 M06
G54 G90
G00 X0. Y0.
G01 F20.0 .... →
...
...
...
...
...
M30

Run program in Graphic Mode.
Save program referencing page number.

MILL PROGRAMMING

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) ;
%
```

*****See Sample Program on USB Drive; O40001.NC**

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 Y0 is top right corner of part) ;	Comment
(Z0 is on top of the part) ;	Comment
(T1 is a 1/2" end mill) ;	Comment
(BEGIN PREPARATION BLOCKS) ;	Comment
T1 M06 (Select tool 1) ;	Selects tool T1 to be used. M06 commands the tool changer to load Tool 1 (T1) into the spindle.
G00 G90 G17 G40 G49 G54 (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 completed in Rapid Motion mode. G90 defines axis movements that will be completed in absolute mode (refer to Absolute vs. Incremental Positioning (G90, G91) for more information). G17 defines the cutting plane as the XY plane. G40 cancels Cutter Compensation. G49 cancels tool length compensation. G54 defines the coordinate system to be centered on the Work Offset stored in G54 on the Offset display.
X0 Y0 (Rapid to 1st position) ;	X0 Y0 commands the table to move to the position X = 0.0 and Y = 0.0 in the G54 coordinate system.
S1000 M03 (Spindle on CW) ;	M03 turns the spindle on in a clockwise direction. It takes the address code Snnnn, where nnnn is the desired spindle RPM. On machines with a gearbox, the control automatically selects high gear or low gear, based on the commanded spindle speed. You can use an M41 or M42 to override this. Refer to page M41 Low Gear Override / M42 High Gear Override for more information on these M-codes.
G43 H01 Z0.1 (Tool offset 1 on) ;	G43 H01 turns on Tool Length Compensation +. The H01 specifies to use the length stored for Tool 1 in the Tool Offset display. Z0.1 commands the Z Axis to Z=0.1.
M08 (Coolant on) ;	M08commands the coolant to turn on.

CUTTING

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

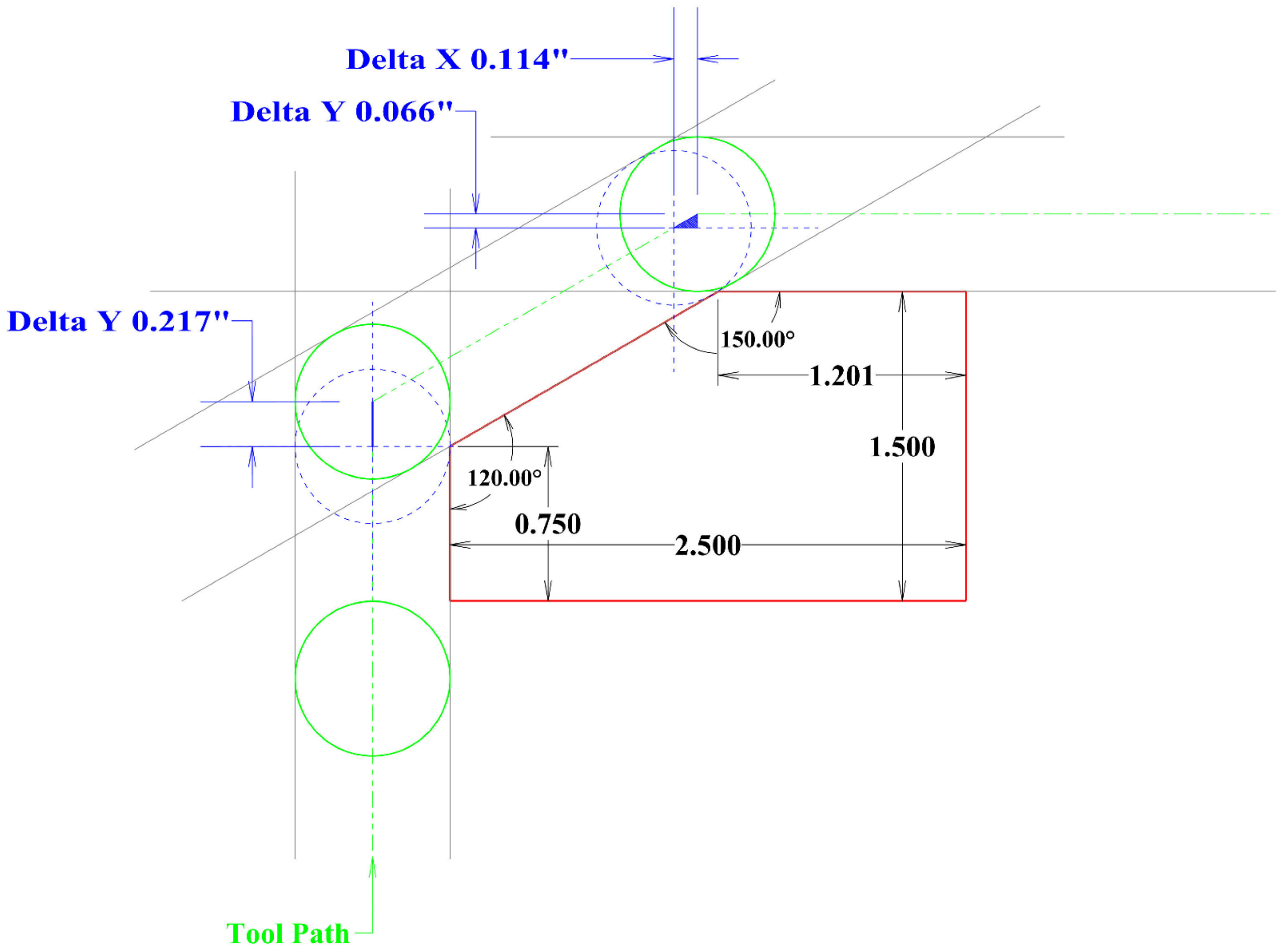
Cutting Code Block	Description
G01 F20. Z-0.1 (Feed to cutting depth) ;	G01 F20. defines axis movements after it to be completed in a straight line. G01 requires the address code Fnnn.nnnn. The address code F20. specifies that the feed rate for the motion is 20" (508 mm) / min. Z-0.1 commands the Z Axis to Z = - 0.1.
X-4. Y-4. (linear motion) ;	X-4. Y-4. commands the X Axis to move to X = - 4.0 and commands the Y Axis to move to Y = - 4.0.

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.
G53 G49 Z0 M05 (Z home, Spindle off) ;	G53 defines axis movements after it to be with respect to the machine coordinate system. G49 cancels tool length compensation. Z0 is a command to move to Z = 0.0. M05 turns the spindle off.
G53 Y0 (Y home) ;	G53 defines axis movements after it to be with respect to the machine coordinate system. Y0 is a command to move to Y = 0.0.
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.

CUTTER COMPENSATION G41 / G42

TOOL PATH WITHOUT CUTTER COMPENSATION – MANUAL COMPENSATION



The Tool Path above requires additional calculation for **0.75"** Diameter Tool.

Y Axis requires an additional **+0.217"** movement to clear the first **120** Degree angle.

Y Axis requires **+0.066"** and X Axis requires **+0.114"** to clear second **150** Degree angle.

CUTTER COMPENSATION

Cutter compensation is a method of shifting the tool path so that the actual centerline of the tool moves to either the left or right of the programmed path.

Normally, cutter compensation is programmed to shift the tool in order to control feature size. The offset display is used to enter the amount that the tool is to be shifted.

The offset can be entered as either a diameter or radius value, depending on Setting 40, for both the geometry and wear values. If diameter is specified, the shift amount is half of the value entered.

The effective offset values are the sum of the geometry and wear values. Cutter compensation is only available in the X Axis and the Y Axis for 2D machining (G17). For 3D machining, cutter compensation is available in the X Axis, Y Axis, and Z Axis (G141).

GENERAL DESCRIPTION OF CUTTER COMPENSATION

G41 selects cutter compensation left. This means that the control moves the tool to the left of the programmed path (with respect to the direction of travel) to compensate for the tool radius or diameter defined in the tool offsets table (Refer to Setting 40). G42 selects cutter compensation right, which moves the tool to the right of the programmed path, with respect to the direction of travel.

A **G41** or **G42** command must have a Dnnn value to select the correct offset number from the radius / diameter offset column. The number to use with D is in the far-left column of the tool offsets table.The value that the control uses for cutter compensation is in the GEOMETRY column under D (if Setting 40 is DIAMETER) or R (if Setting 40 is RADIUS).

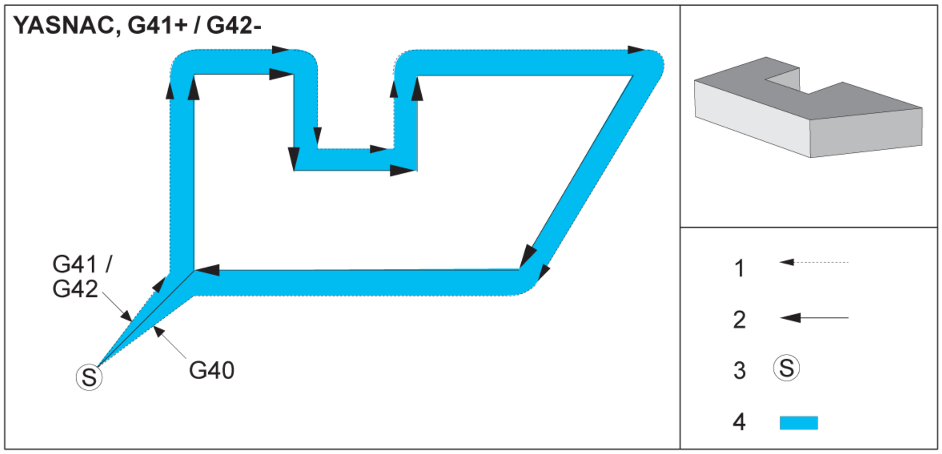
If the offset value is negative, cutter compensation operates as though the program specifies the opposite G code. For example, a negative value entered for a G41 will behave as if a positive value was entered for G42. Also, when cutter compensation is active (G41 or G42), you may use only the X-Y plane (G17) for circular motions. **Cutter Compensation is limited to only X-Y plane**.

G40 cancels cutter compensation and is the default condition when you power on your machine. When cutter compensation is not active, the programmed path is the same as the center of the cutter path. You may not end a program (M30, M00, M01, or M02) with cutter compensation active.
The control operates on one motion block at a time. However, it will look ahead at the next (2) blocks that have X or Y motions. The control checks these (3) blocks of information for interference. Setting 58 controls how this part of cutter compensation works. Available Setting 58 values are Fanuc or YASNAC.
If Setting 58 is set to YASNAC, the control must be able to position the side of the tool along all of the edges of the programmed contour without overcutting the next two motions. A circular motion joins all of the outside angles.

If Setting 58 is set to Fanuc, the control does not require that the tool cutting edge be placed along all edges of the programmed contour, preventing overcutting. However, the control will generate an alarm if the cutter's path is programmed so that it will be overcut. The control joins outside angles less than or equal to 270 degrees with a sharp corner. It joins outside angles of more than 270 degrees with an extra linear motion.
These diagrams show how cutter compensation works for the possible values of Setting 58. Note that a small cut of less than the tool radius and at a right angle to the previous motion will work only with the Fanuc setting.

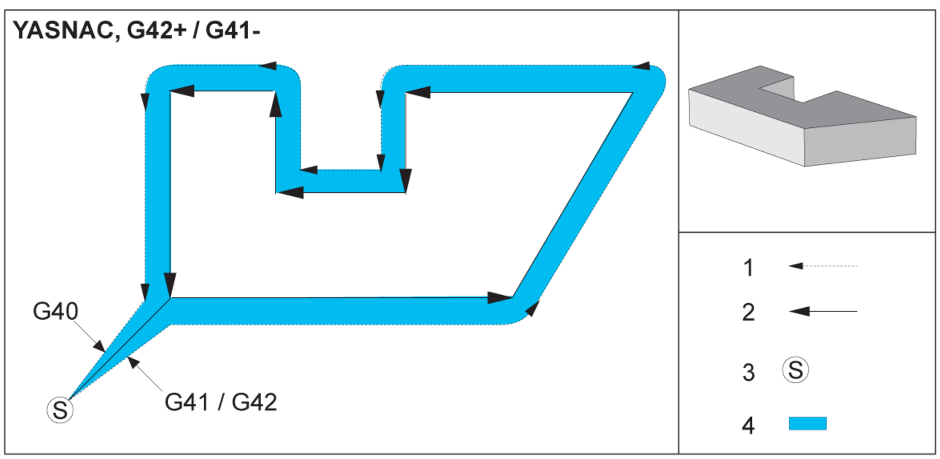
Cutter Compensation, YASNAC Style, G41 with a Positive Tool Diameter or G42 with a Negative Tool Diameter:

[1] Tool Path Actual Center,
[2] Programmed Tool Path,
[3] Start Point,
[4] Cutter Compensation. G41 / G42 and G40 are commanded at the start and end of the tool path.

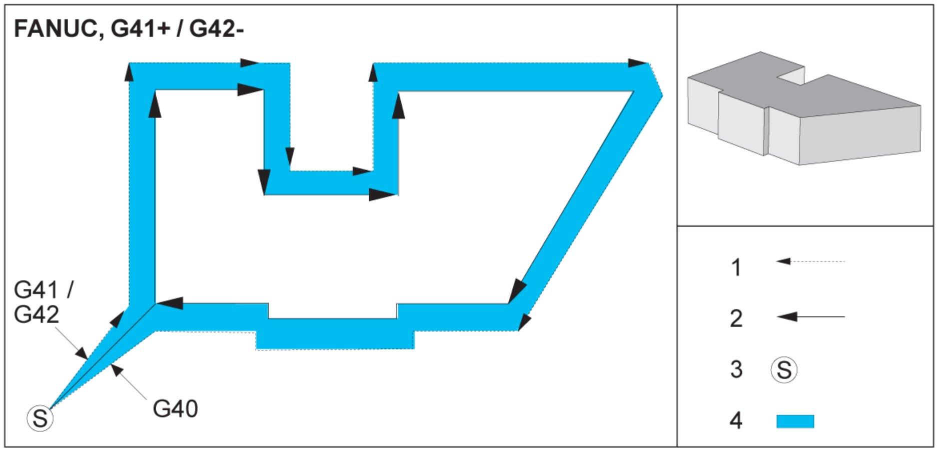


Cutter Compensation, YASNAC Style, G42 with a Positive Tool Diameter or G41 with a Negative Tool Diameter:

[1] Tool Path Actual Center,
[2] Programmed Tool Path,
[3] Start Point,
[4] Cutter Compensation. G41 / G42 and G40 are commanded at the start and end of the tool path.

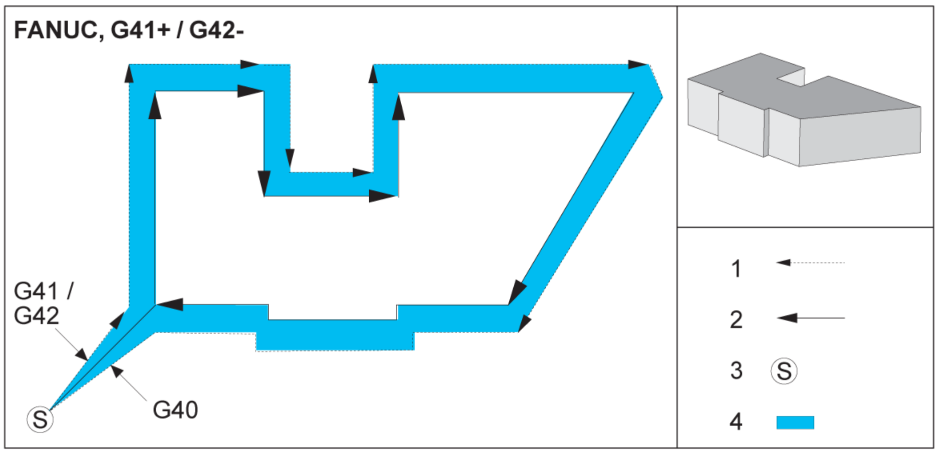


Cutter Compensation, FANUC Style, G41 with a Positive Tool Diameter or G42 with a Negative Tool Diameter:



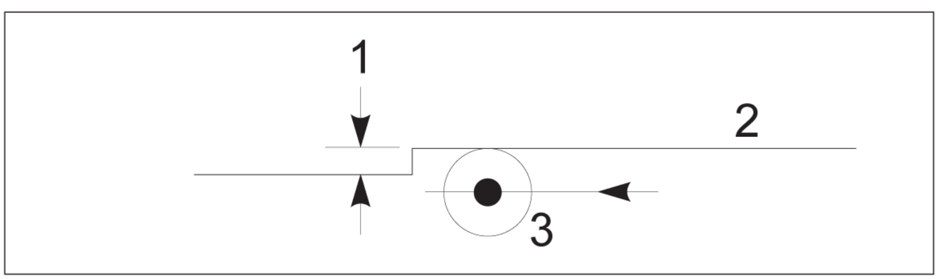
[1] Tool Path Actual Center,
[2] Programmed Tool Path,
[3] Start Point,
[4] Cutter Compensation. G41 / G42 and G40 are commanded at the start and end of the tool path.

Cutter Compensation, FANUC Style, G42 with a Positive Tool Diameter or G41 with a Negative Tool Diameter:



[1] Tool Path Actual Center,
[2] Programmed Tool Path,
[3] Start Point,
[4] Cutter Compensation. G41 / G42 and G40 are commanded at the start and end of the tool path.

Improper Cutter Compensation:



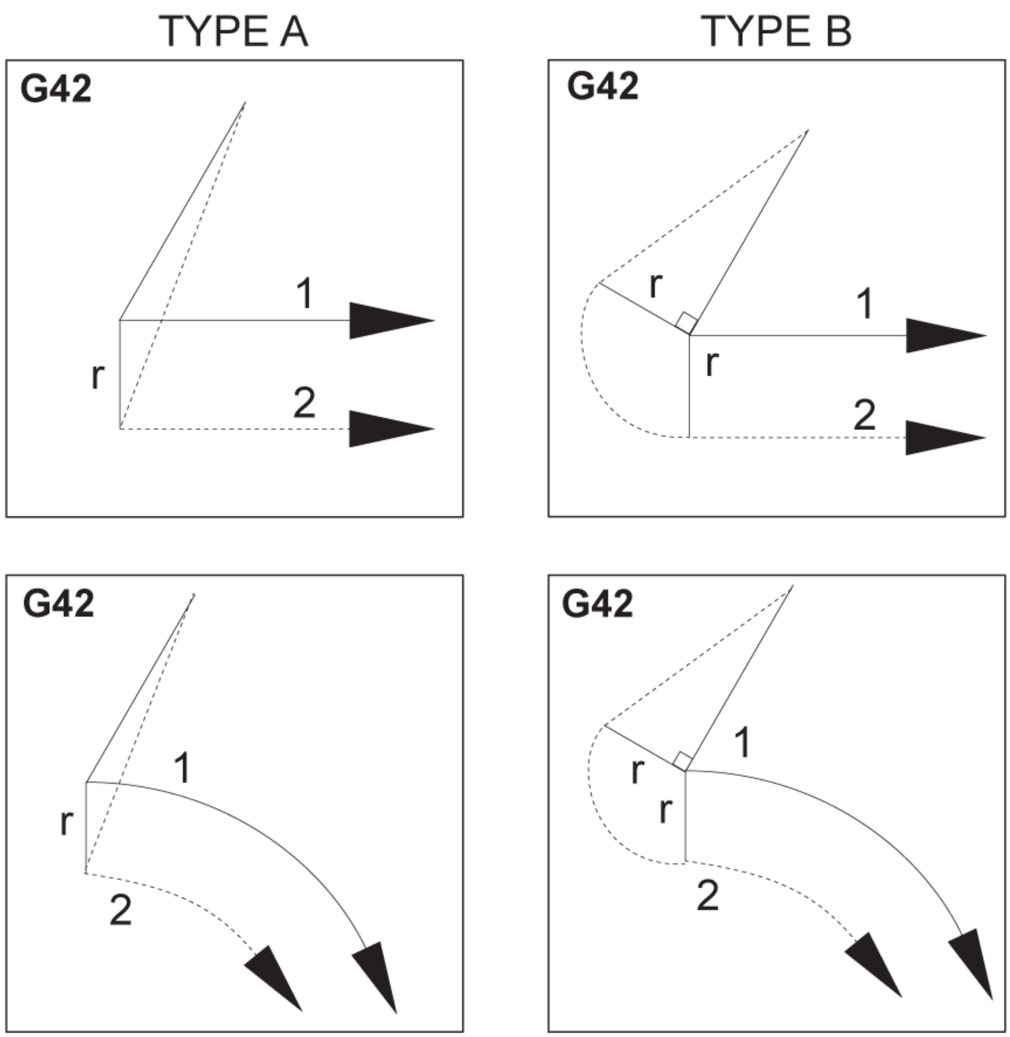
[1] Move is less than cutting comp radius,
[2] Workpiece,
[3] Tool.
NOTE: A small cut of less than tool radius and at a right angle to the previous motion will only work with the Fanuc setting. A cutter compensation alarm will be generated if the machine is set to the YASNAC setting.

FEED ADJUSTMENTS IN CUTTER COMPENSATION

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

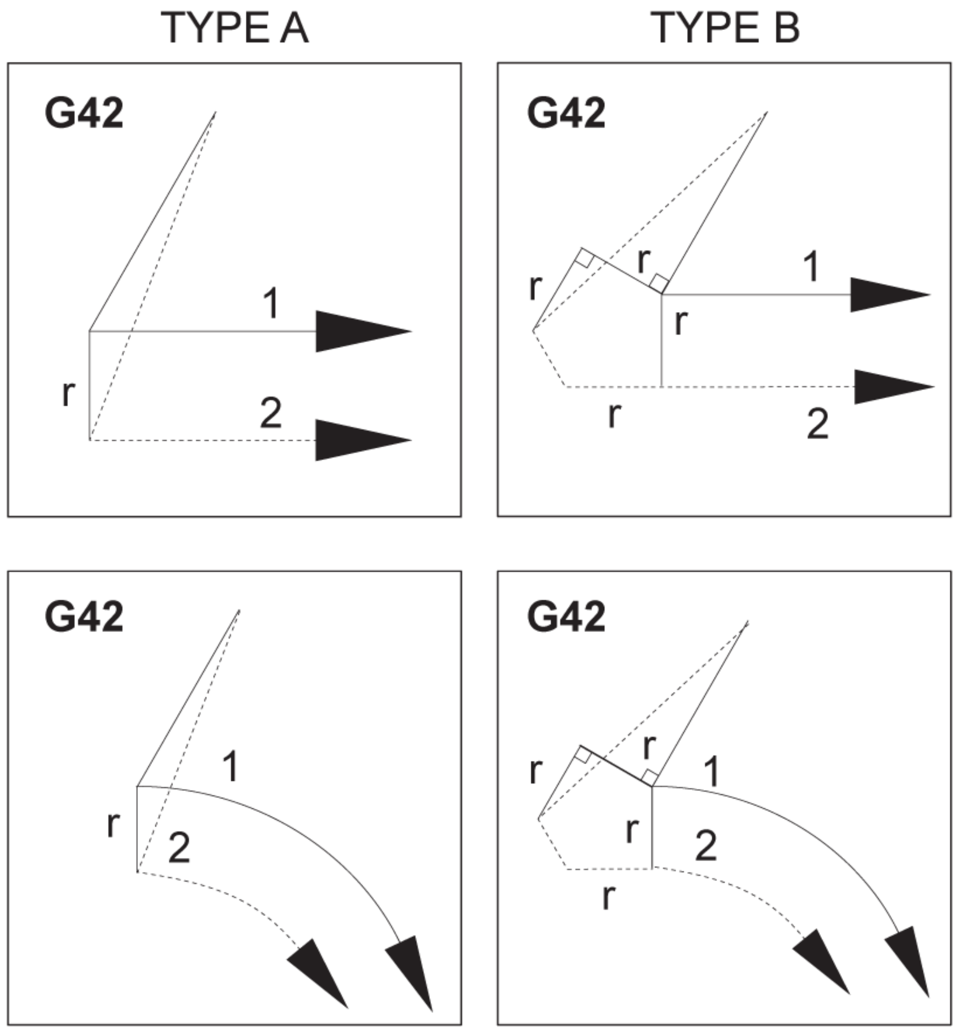
Cutter Compensation Entry (YASNAC) Type A and B:

[1] Programmed Path,
[2] Tool Center Path,
[r] Tool Radius



Cutter Compensation Entry (Fanuc style) Type A and B:

[1] Programmed Path,
[2] Tool Center Path,
[r] Tool Radius



Circular Interpolation and Cutter Compensation

In this section, the usage of G02 (Circular Interpolation Clockwise), G03 (Circular Interpolation Counterclockwise) and Cutter Compensation (G41: Cutter Compensation Left, G42: Cutter Compensation Right) is described.

Using G02 and G03, we can program the machine to cut circular moves and radii. Generally, when programming a profile or a contour, the easiest way to describe a radius between two points is with an R and a value. For complete circular moves (360 degrees), an I or a J with a value must be specified. The circle section illustration will describe the different sections of a circle.

By using cutter compensation in this section, the programmer will be able to shift the cutter by an exact amount and be able to machine a profile or a contour to the exact print dimensions. By using cutter compensation, programming time and the likelihood of a programming calculation error is reduced since real dimensions can be programmed, and part size and geometry can be easily controlled.

Here are a few rules about cutter compensation that you must follow closely for successful machining operations. Always refer to these rules when you write your programs.

1. Cutter compensation must be turned ON during a G01 X,Y move that is equal to or greater than the cutter radius, or the amount being compensated.

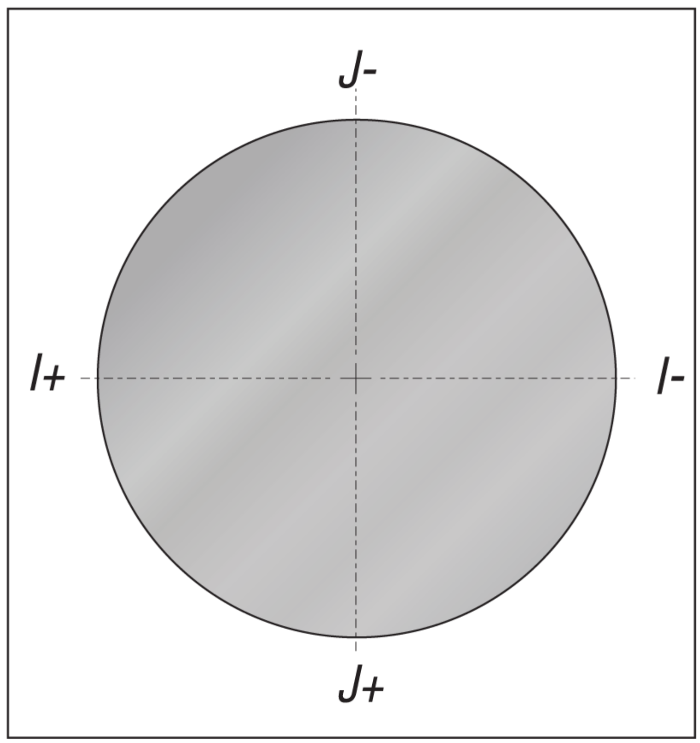
2. When an operation using cutter compensation is done, the cutter compensation will need to be turned OFF, using the same rules as the turn ON process, i.e., what is put in must be taken out.

3. In most machines, during cutter compensation, a linear X,Y move that is smaller than the cutter radius may not work. (Setting 58 - set to Fanuc - for positive results.)

4. Cutter compensation cannot be turned ON or OFF in a G02 or G03 arc movement.

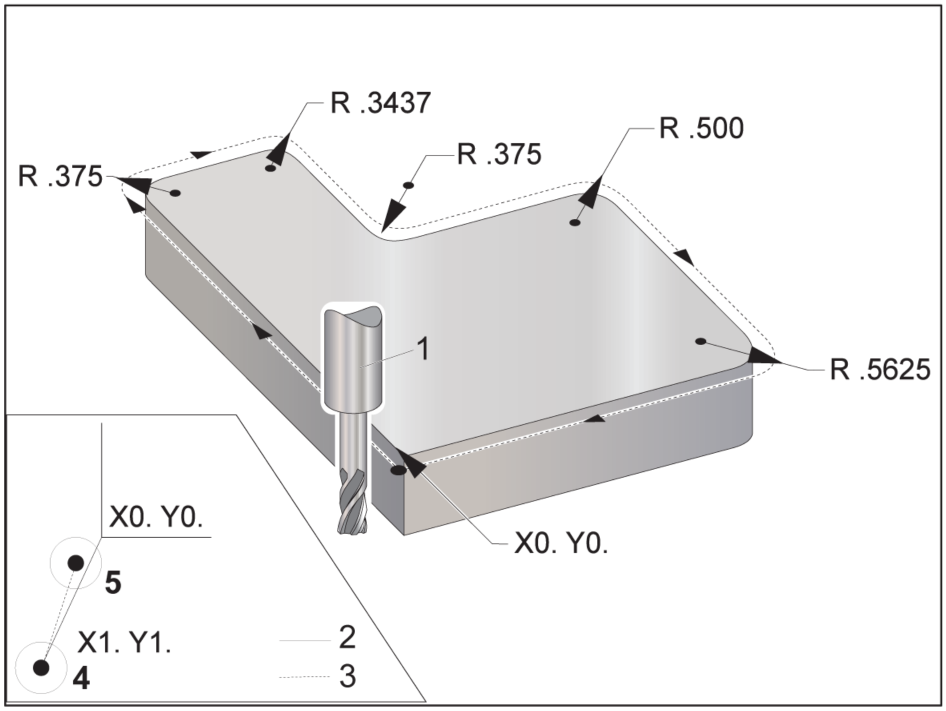
5. With cutter compensation active, machining an inside arc with a radius less than what is defined by the active D value causes the machine to alarm. Can not have too big of a tool diameter if the radius of arc is too small.

This illustration shows how the tool path is calculated for the cutter compensation.
The detail section shows the tool in the starting position and then in the offset position as the cutter reaches the workpiece.



Circular Interpolation G02 and G03:

[1] 0.250" diameter endmill,
[2] Programmed path,
[3] Center of Tool,
[4] Start Position,
[5] Offset Tool Path.



Programming exercise showing tool path.

This program uses cutter compensation. The toolpath is programmed to the centerline of the cutter. This is also the way the control calculates for cutter compensation.

```
%
O40006 (Cutter comp ex-prog) ;
(G54 X0 Y0 is at the lower left of part corner) ;
(Z0 is on top of the part) ;
(T1 is a .250 dia endmill) ;
(BEGIN PREPARATION BLOCKS) ;
T1 M06 (Select tool 1) ;
G00 G90 G40 G49 G54 (Safe startup) ;
X-1. Y-1. (Rapid to 1st position) ;
S1000 M03 (Spindle on CW) ;
G43 H01 Z0.1(Tool offset 1 on) ;
M08(Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G01 Z-1. F50. (Feed to cutting depth) ;
G41 G01 X0 Y0 D01 F50. (2D Cutter Comp left
on) ;
Y4.125 (Linear motion) ;
G02 X0.25 Y4.375 R0.375 (Corner rounding) ;
G01 X1.6562 (Linear motion) ;
G02 X2. Y4.0313 R0.3437 (Corner rounding) ;
G01 Y3.125 (Linear motion) ;
G03 X2.375 Y2.75 R0.375 (Corner rounding) ;
G01 X3.5 (Linear motion) ;
G02 X4. Y2.25 R0.5 (Corner rounding) ;
G01 Y0.4375 (Linear motion) ;
G02 X3.4375 Y-0.125 R0.5625 (Corner rounding)
;
G01 X-0.125 (Linear motion) ;
G40 X-1. Y-1. (Last position, cutter comp off) ;
(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) ;
%
```

*****See Sample Program on USB Drive; O40006.NC**

HARVI II TE APPLICATION DATA - Inch

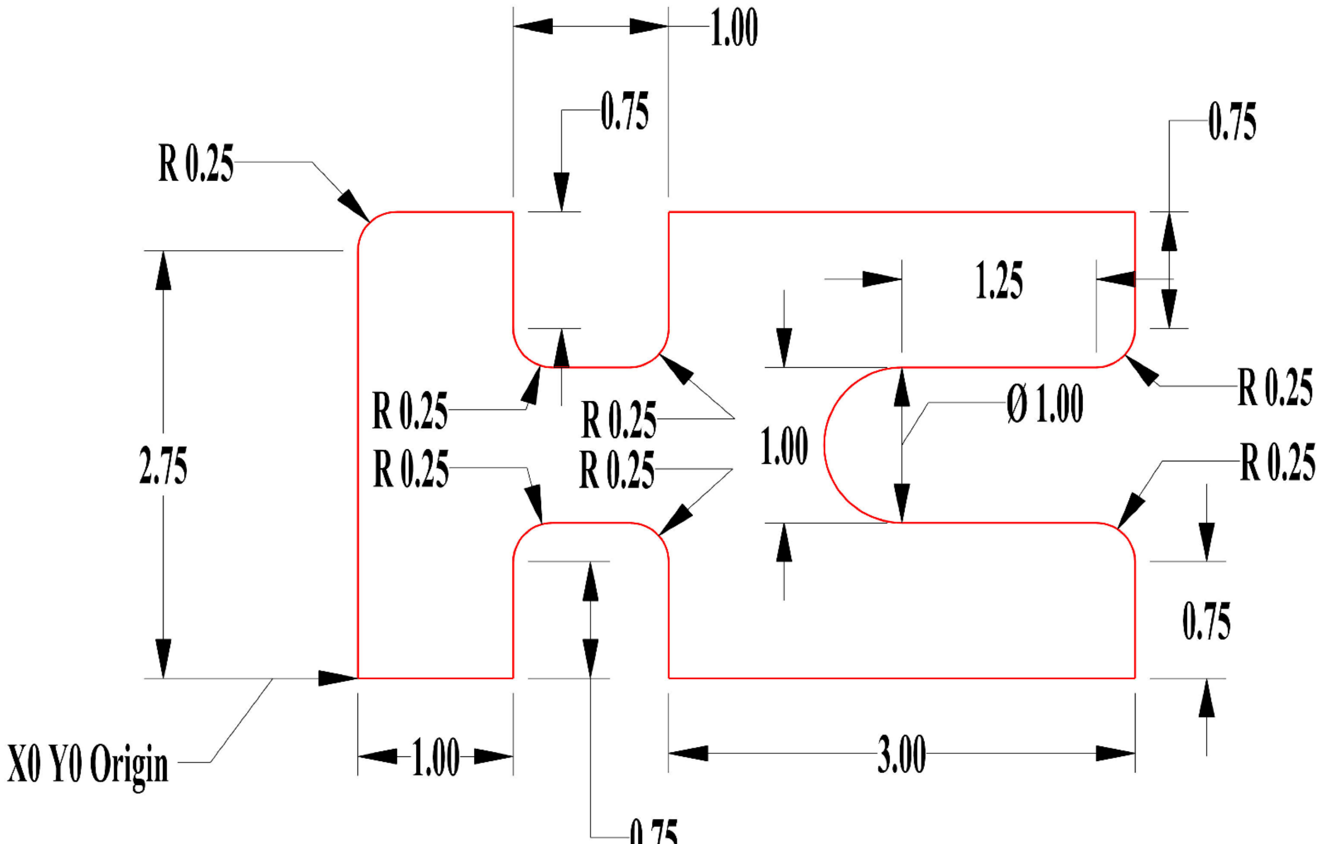
Material Group		Side Milling		Slotting	KCPM15A - KCSM15A Cutting Speed Vc SFM		Fraction	Recommended Feed per Tooth (IPT=Inch/th) is for Side Milling . For Slotting Reduce Fz by 20% — D1 - Diameter												
								1/8	5/32	3/16	7/32	1/4	9/32	5/16	3/8	1/2	5/8	3/4	1	1 1/4
		Ap	Ae	Ap	Min	Max	dec.	0.1250	0.1563	0.1875	0.2188	0.2500	0.2813	0.3125	0.3750	0.5000	0.6250	0.7500	1.0000	1.2500
P	P0	1.5xD	0.5XD	1.25xD	490	660	IPT	0.0010	0.0012	0.0015	0.0017	0.0020	0.0023	0.0026	0.0030	0.0037	0.0043	0.0048	0.0054	0.0054
	P1	1.5xD	0.5XD	1.25xD	490	660	IPT	0.0010	0.0012	0.0015	0.0017	0.0020	0.0023	0.0026	0.0030	0.0037	0.0043	0.0048	0.0054	0.0054
	P2	1.5xD	0.5XD	1.25xD	460	620	IPT	0.0010	0.0012	0.0015	0.0017	0.0020	0.0023	0.0026	0.0030	0.0037	0.0043	0.0048	0.0054	0.0054
	P3	1.5xD	0.5XD	1.25xD	390	520	IPT	0.0008	0.0010	0.0012	0.0014	0.0017	0.0019	0.0021	0.0025	0.0032	0.0038	0.0042	0.0050	0.0053
	P4	1.5xD	0.5XD	1.25xD	300	490	IPT	0.0007	0.0009	0.0011	0.0013	0.0015	0.0017	0.0019	0.0022	0.0028	0.0033	0.0037	0.0042	0.0044
	P5	1.5xD	0.5XD	1.25xD	200	330	IPT	0.0007	0.0008	0.0010	0.0012	0.0014	0.0015	0.0017	0.0020	0.0025	0.0030	0.0034	0.0040	0.0043
	P6	1.5xD	0.5XD	1.25xD	160	250	IPT	0.0006	0.0007	0.0008	0.0010	0.0011	0.0013	0.0014	0.0017	0.0021	0.0025	0.0027	0.0031	0.0032
M	M1	1.5xD	0.5XD	1.25xD	300	380	IPT	0.0008	0.0010	0.0012	0.0014	0.0017	0.0019	0.0021	0.0025	0.0032	0.0038	0.0042	0.0050	0.0053
	M2	1.5xD	0.5XD	1.25xD	200	260	IPT	0.0007	0.0008	0.0010	0.0012	0.0014	0.0015	0.0017	0.0020	0.0025	0.0030	0.0034	0.0040	0.0043
	M3	1.5xD	0.5XD	1.0xD	200	230	IPT	0.0006	0.0007	0.0008	0.0010	0.0011	0.0013	0.0014	0.0017	0.0021	0.0025	0.0027	0.0031	0.0032
K	K1	1.5xD	0.5XD	1.0xD	390	490	IPT	0.0010	0.0012	0.0015	0.0017	0.0020	0.0023	0.0026	0.0030	0.0037	0.0043	0.0048	0.0054	0.0054
	K2	1.5xD	0.5XD	1.0xD	360	460	IPT	0.0008	0.0010	0.0012	0.0014	0.0017	0.0019	0.0021	0.0025	0.0032	0.0038	0.0042	0.0050	0.0053
	K3	1.5xD	0.5XD	1.0xD	360	430	IPT	0.0007	0.0008	0.0010	0.0012	0.0014	0.0015	0.0017	0.0020	0.0025	0.0030	0.0034	0.0040	0.0043
S	S1	1.5xD	0.3XD	0.75xD	160	300	IPT	0.0008	0.0010	0.0012	0.0014	0.0017	0.0019	0.0021	0.0025	0.0032	0.0038	0.0042	0.0050	0.0053
	S2	1.5xD	0.3XD	0.75xD	80	160	IPT	0.0004	0.0005	0.0007	0.0008	0.0009	0.0010	0.0011	0.0013	0.0017	0.0020	0.0023	0.0027	0.0029
	S3	1.5xD	0.5XD	0.75xD	80	130	IPT	0.0004	0.0005	0.0007	0.0008	0.0009	0.0010	0.0011	0.0013	0.0017	0.0020	0.0023	0.0027	0.0029
	S4	1.5xD	0.5XD	1.25xD	160	200	IPT	0.0005	0.0007	0.0008	0.0010	0.0012	0.0014	0.0016	0.0019	0.0023	0.0028	0.0031	0.0036	0.0039
H	H1	1.5xD	0.5XD	1.0xD	260	460	IPT	0.0007	0.0009	0.0011	0.0013	0.0015	0.0017	0.0019	0.0022	0.0028	0.0033	0.0037	0.0042	0.0044
	H2	1.5xD	0.2XD	1.0xD	230	390	IPT	0.0006	0.0007	0.0008	0.0010	0.0011	0.0013	0.0014	0.0017	0.0021	0.0025	0.0027	0.0031	0.0032

HARVI II TE APPLICATION DATA - Inch

Material Group		Helical Interpolation / Ramping 15° - 30°	KCPM15A - KCSM15A Cutting Speed Vc SFM		Min - Max Diameter for Helical Interpola-tion	Recommended feed per tooth (IPT = Inch/th) for Helical Interpolation and Ramping - Zef=2 — D1 - Diameter												
						.144 - .238	.179 - .296	.216 - .356	.251 - .415	.288 - .475	.323 - .534	.359 - .594	.431 - .713	.575 - .950	.719 - 1.188	.863 - 1.425	1.150 - 1.900	1.437 - 2.375
		Max Depth	Min	Max	dec	0.1250	0.1563	0.1875	0.2188	0.2500	0.2813	0.3125	0.3750	0.5000	0.6250	0.7500	1.0000	1.2500
P	P0	1,25 x D1	490	660	IPT	0.0007	0.0009	0.0011	0.0013	0.0015	0.0017	0.0019	0.0022	0.0028	0.0033	0.0036	0.0040	0.0041
	P1	1,25 x D1	490	660	IPT	0.0007	0.0009	0.0011	0.0013	0.0015	0.0017	0.0019	0.0022	0.0028	0.0033	0.0036	0.0040	0.0041
	P2	1,25 x D1	460	620	IPT	0.0007	0.0009	0.0011	0.0013	0.0015	0.0017	0.0019	0.0022	0.0028	0.0033	0.0036	0.0040	0.0041
	P3	1,25 x D1	390	520	IPT	0.0006	0.0008	0.0009	0.0011	0.0013	0.0014	0.0016	0.0019	0.0024	0.0028	0.0032	0.0037	0.0040
	P4	1,25 x D1	300	490	IPT	0.0005	0.0007	0.0008	0.0010	0.0011	0.0013	0.0014	0.0017	0.0021	0.0025	0.0028	0.0032	0.0033
	P5	1,25 x D1	200	330	IPT	0.0005	0.0006	0.0007	0.0009	0.0010	0.0012	0.0013	0.0015	0.0019	0.0023	0.0025	0.0030	0.0032
	P6	1,25 x D1	160	250	IPT	0.0004	0.0005	0.0006	0.0007	0.0009	0.0010	0.0011	0.0013	0.0016	0.0018	0.0021	0.0023	0.0024
K	K1	1,0 x D1	390	490	IPT	0.0007	0.0009	0.0011	0.0013	0.0015	0.0017	0.0019	0.0022	0.0028	0.0033	0.0036	0.0040	0.0041
	K2	1,0 x D1	360	460	IPT	0.0006	0.0008	0.0009	0.0011	0.0013	0.0014	0.0016	0.0019	0.0024	0.0028	0.0032	0.0037	0.0040
	K3	1,0 x D1	360	430	IPT	0.0005	0.0006	0.0007	0.0009	0.0010	0.0012	0.0013	0.0015	0.0019	0.0023	0.0025	0.0030	0.0032
S	S1	0,75 x D1	160	300	IPT	0.0006	0.0008	0.0009	0.0011	0.0013	0.0014	0.0016	0.0019	0.0024	0.0028	0.0032	0.0037	0.0040
	S2	0,75 x D1	80	160	IPT	0.0003	0.0004	0.0005	0.0006	0.0007	0.0008	0.0009	0.0010	0.0013	0.0015	0.0017	0.0020	0.0022
	S3	0,5 x D1	80	130	IPT	0.0003	0.0004	0.0005	0.0006	0.0007	0.0008	0.0009	0.0010	0.0013	0.0015	0.0017	0.0020	0.0022
	S4	1,25 x D1	160	200	IPT	0.0004	0.0005	0.0006	0.0008	0.0009	0.0010	0.0012	0.0014	0.0018	0.0021	0.0023	0.0027	0.0029
H	H1	1,0 x D1	260	460	IPT	0.0005	0.0007	0.0008	0.0010	0.0011	0.0013	0.0014	0.0017	0.0021	0.0025	0.0028	0.0032	0.0033
	H2	1,0 x D1	230	390	IPT	0.0004	0.0005	0.0006	0.0007	0.0009	0.0010	0.0011	0.0013	0.0016	0.0018	0.0021	0.0023	0.0024

LETTER CODE	MATERIAL GROUP
P	STEEL
M	STAINLESS STEEL
K	CAST IRON
N	NONFERROUS (ALUMINUM, MAGNESIUM, ETC.)
S	SUPERALLOYS (TITANIUM, ETC.)
H	HARDENED STEEL

CUTTER COMPENSATION EXERCISE 1

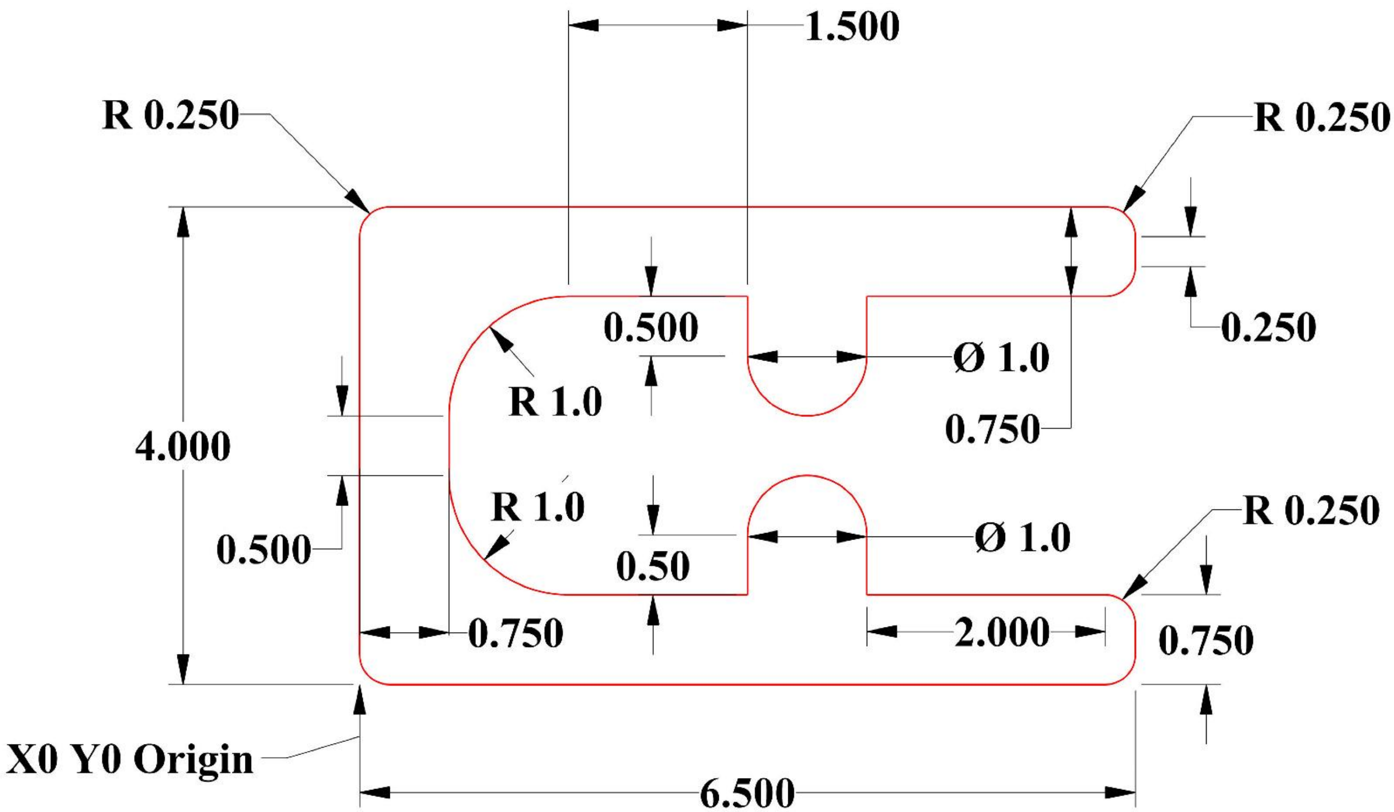


Tool	Length	Diameter	Flutes	SFM	FPT
	H2TE5SE0500S063HA	0.5 (1/2)	5	500	0.0037
Material	P0 - STEEL				

*** *Use the tool information above to calculate* ***speed*** *and* ***feed***
*** ***Side Milling*** *around Outside of Part Profile*

HAAS Simulator: Go to MDI Mode:	Save Program Referencing Page Number
G00 G90 G17 G40 G49 G54	________________________
T1 M06	________________________
X-0.5 Y-0.5	________________________
S______ M03	________________________
M08	________________________
G43 H____ Z1.0	________________________
G01 Z-0.5 F10. (Feed down to Z Depth)	________________________
G41 F____ D____ G01 X__ Y__ →	________________________
...	________________________
...	________________________
M30	________________________
Run program in Graphic Mode.	________________________

CUTTER COMPENSATION EXERCISE 2

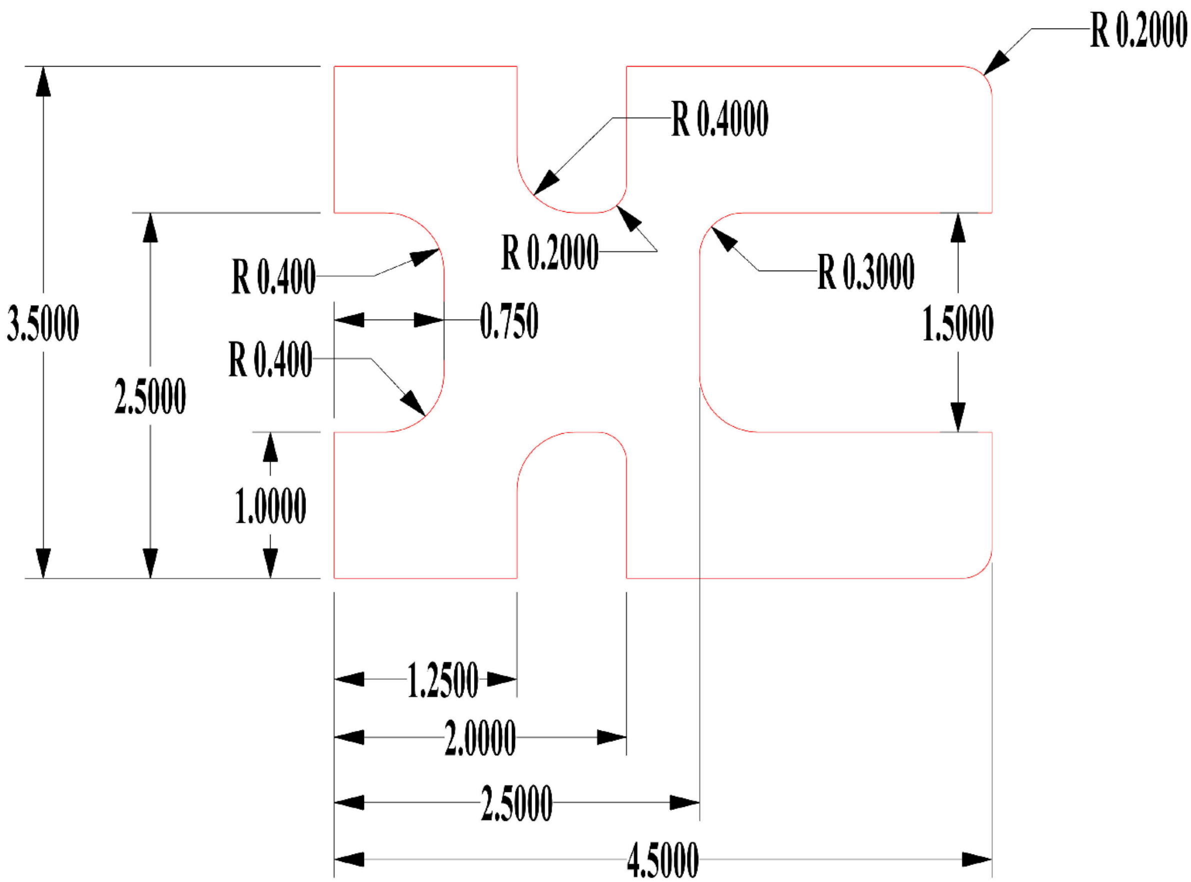


Tool	Length	Diameter	Flutes	SFM	FPT
	H2TE5SE0500S063HA	0.5 (1/2)	5	500	0.0037
Material	P0 - STEEL				

*** *Use the tool information above to calculate* ***speed*** *and* ***feed***

HAAS Simulator: Go to MDI Mode:	Save Program Referencing Page Number
G00 G90 G17 G40 G49 G54 **T1 M06** **X-0.5 Y-0.5** **S______ M03** **M08** **G43 H____ Z1.0** **G01 Z-0.5 F10. (Feed down to Z Depth)** **G41 F____ D____ G01 X__ Y__ →** **...** **...** **M30** **Run program in Graphic Mode.**	

CUTTER COMPENSATION EXERCISE 3



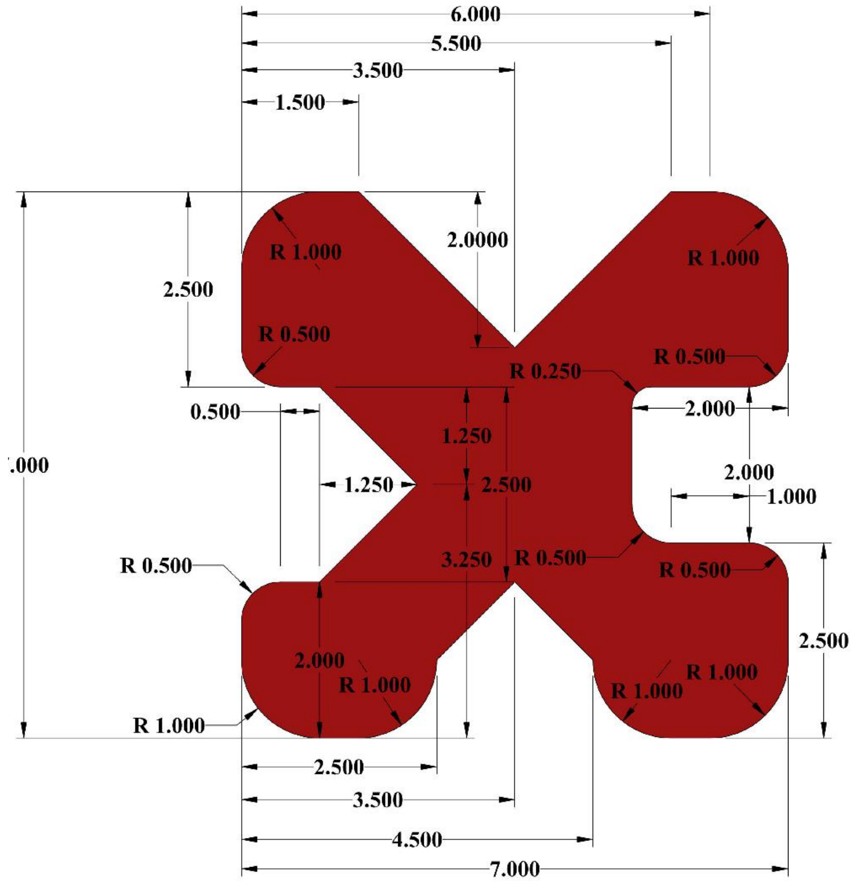
Tool	Length	Diameter	Flutes	SFM	FPT
	H2TE5SE0281R075HAV	0.3125 (5/16)	5	???	???
Material	P0 - STEEL				

**** Use the tool information above to calculate **speed** and **feed***

**** Top part is symmetrical to bottom half. Use dimensions from the top half.*

HAAS Simulator: Go to MDI Mode:	Save Program Referencing Page Number
G00 G90 G17 G40 G49 G54 T1 M06 X-0.5 Y-0.5 S______ M03 M08 G43 H____ Z1.0 G01 Z-0.5 F10. (Feed down to Z Depth) G41 F____ D____ G01 X__ Y__ → ... ... M30 Run program in Graphic Mode.	

CUTTER COMPENSATION EXERCISE 4



Tool	Length	Diameter	Flutes	SFM	FPT
	H2TE5SE0375S075HA	0.375 (3/8)	5	???	???
Material	P0 - STEEL				

*** *Use the tool information above to calculate* ***speed*** *and* ***feed***

*** *This is an* ***extra Exercise****. Sharp corners will be* ***rounded off*** *based on the tool diameter.*

HAAS Simulator: Go to MDI Mode:	Save Program Referencing Page Number
G00 G90 G17 G40 G49 G54 T1 M06 X-0.5 Y-0.5 S______ M03 M08 G43 H____ Z1.0 G01 Z-0.5 F10. (Feed down to Z Depth) G41 F____ D____ G01 X__ Y__ → ... ... M30 Run program in Graphic Mode.	____________________ ____________________ ____________________ ____________________ ____________________ ____________________ ____________________ ____________________ ____________________ ____________________ ____________________ ____________________

G12 CIRCULAR POCKET MILLING CW / G13 CIRCULAR POCKET MILLING CCW (GROUP 00)

These G-codes mill circular shapes. They are different only in that *G12* uses a clockwise direction and *G13* uses a counterclockwise direction. Both G-codes use the default *XY* circular plane (*G17*) and imply the use of *G42* (cutter compensation) for *G12* and *G41* for *G13*. *G12* and *G13* are non-modal.

* **D** - Tool radius or diameter selection**
F - Feedrate
I - Radius of first circle (or finish if no *K*). *I* value must be greater than Tool Radius, but less than *K* value.
* **K** - Radius of finished circle (if specified)
* **L** - Loop count for repeating deeper cuts
* **Q** - Radius increment, or stepover (must be used with K)
Z - Depth of cut or increment

*indicates optional

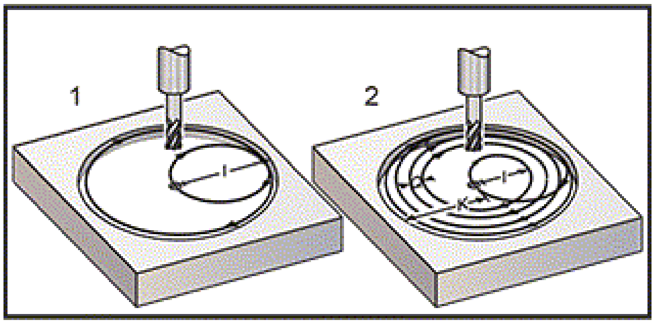
**To get the programmed circle diameter, the control uses the selected D code tool size. To program tool centerline select *D0*.

NOTE

Specify D00 if you do not want to use cutter compensation. If you do not specify a D value in the G12 / G13 block, the control uses the last commanded D value, even if it was previously canceled with a G40 .

Rapid-position the tool to the center of the circle. To remove all the material inside the circle, use *I* and *Q* values less than the tool diameter and a *K* value equal to the circle radius. To cut a circle radius only, use an *I* value set to the radius and no *K* or *Q* value.

CIRCULAR POCKET MILLING, G12 CLOCKWISE SHOWN: [1] I ONLY, [2] I, K AND Q ONLY.



***** Program O60121 in G12 & G13 Sample Program Folder on USB Drive *****

These G codes assume cutter compensation, so you do not need to program *G41* or *G42* in the program block. However, you must include a *D* offset number, for cutter radius or diameter, to adjust the circle diameter.

These program examples show the *G12* and *G13* format, and the different ways that you can write these programs.

Single Pass: Use *I* only.

Applications: One-pass counter boring; rough and finish pocketing of smaller holes, ID cutting of O-ring grooves.

Multiple Pass: Use *I*, *K*, and *Q*.

Applications: Multiple-pass counter boring; rough and finish pocketing of large holes with cutter overlap.

Multiple Z-Depth Pass: Using *I* only, or *I*, *K*, and *Q* (*G91* and *L* may also be used).

Applications: Deep rough and finish pocketing.

The previous figures show the tool path during the pocket milling G-codes.

EXAMPLE *G13* MULTIPLE-PASS USING *I*, *K*, *Q*, *L*, AND *G91*:

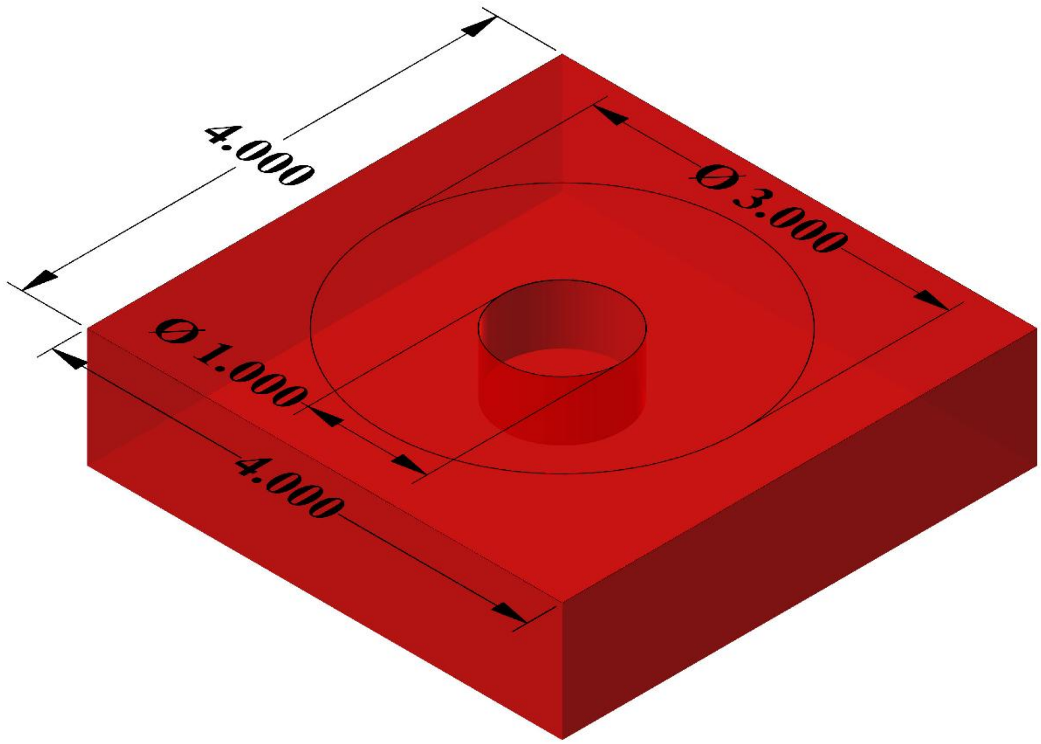
***** Program O60131 in G12 & G13 Sample Program Folder on USB Drive *****

This program uses *G91* and an *L* count of *4*, so this cycle will execute a total of four times. The Z depth increment is 0.500. This is multiplied by the *L* count, making the total depth of this hole 2.000.

The *G91* and *L* count can also be used in a *G13I* only line.

G13 AND G12 (GROUP 00) EXERCISE

Part Drawing



Tool Information for G13 and G12 (Group 00) Exercise			Surface Feed	FPT / IPT
Tool 1		1/2 (.5000) Dia, 4 Flt, SE Square Carbide Endmill	250	.0013

*** *Use the tool information above to calculate* ***speed (RPM)*** *and* ***feed (IPM)***

*** *Locate your own Work Offset* ***(G54 – X0 Y0)*** *on the part*

*** *Open* ***1.0"*** *Bore to* ***3.0"***

- ***Circular Pocket Milling G13 for ID to maintain climb milling***
- ***Cutter comp is not required. Use Tool diameter for compensation.***
- ***Z Depths is -0.50"***

HAAS Simulator: Go to MDI Mode:	Save Program Referencing Page Number
G00 G90 G17 G40 G49 G54 (Safety Line)	________________
T___ M06	________________
S______ M03	________________
X__ Y__	________________
M08	________________
G43 H____ Z1.0 (Feed to part surface)	________________
G01 _______ → (Feed down to Z depth)	________________
G13 D____ →	________________
...	________________
M30	________________
...	________________
Run program in Graphic Mode.	________________

CANNED CYCLES

Canned cycles are G-codes that do repetitive operations such as drilling, tapping, and boring. You define a canned cycle with alphabetic address codes. While the canned cycle is active, the machine does the defined operation every time you command a new position, unless you specify not to.

Canned cycles simplify part programming. Most common Z-axis repetitive operations, such as drilling, tapping, and boring, have canned cycles. When active, a canned cycle executes at every new axis position. Canned cycles execute axis motions as rapid commands (G00) and the canned cycle operation is performed after the axis motion. This applies to G17, G19 cycles, and Y-Axis movements on Y-Axis lathes.

DRILLING CANNED CYCLES

All four drill canned cycles can be looped in G91, Incremental Programming mode.

- The G81 Drill Canned Cycle is the basic drilling cycle. It is used for drilling shallow holes or for drilling with Through Spindle Coolant (TSC).

- The G82 Spot Drill Canned Cycle is the same as the G81 Drill Canned Cycle except that it can dwell at the bottom of the hole. The optional argument Pn.nnn specifies the duration of the dwell.

- The G83 Normal Peck Drilling Canned Cycle is typically used for drilling deep holes. Peck depth can be variable or constant and always incremental. Qnn.nnn. Do not use a Q value when programming with I, J, and K.

- The G73 High-Speed Peck Drilling Canned Cycle is the same as the G83 Normal Peck Drilling Canned Cycle except that tool peck retraction is specified with Setting 22 - Can Cycle Delta Z. Peck drilling cycles are advised for hole depths greater than 3 times the diameter of the drill bit. The initial peck depth, defined by I, should generally be a depth of 1 tool diameter.

TAPPING CANNED CYCLES

There are two tapping canned cycles. All tapping canned cycles can be looped in G91, Incremental Programming mode.

The G84 Tapping Canned Cycle is the normal tapping cycle. It is used for tapping right-hand threads. G74 Reverse Tap Canned Cycle is the reverse thread tapping cycle. It is used for tapping left-hand threads.

BORING AND REAMING CYCLES

There are (5) boring canned cycles. All boring canned cycles can be looped in G91, Incremental Programming mode.

- The G85 Boring Canned Cycle is the basic boring cycle. It will bore down to the desired height and return to the specified height.
- The G86 Bore and Stop Canned Cycle is the same as the G85 Boring Canned Cycle except that the spindle will stop at the bottom of the hole before returning to the specified height.
- The G89 Bore In, Dwell, Bore Out Canned Cycle is the same as G85 except that there is a dwell at the bottom of the hole, and the hole continues to be bored at the specified feed rate as the tool returns to the specified position. This differs from other boring canned cycles where the tool either moves in Rapid Motion or hand jog to return to the return position.
- The G76 Fine Boring Canned Cycle bores the hole to the specified depth and after boring the hole, moves to clear the tool from hole before retracting.
- The G77 Back Bore Canned Cycle works similar to G76 except that before beginning to bore the hole, it moves the tool to clear the hole, moves down into the hole, and bores to the specified depth.

R PLANES

R Planes, or return planes, are G-code commands that specify the Z-Axis return height during canned cycles.

The R Plane G-codes remain active for the duration of the canned cycle it is used with. G98 Canned Cycle Initial Point Return moves the Z axis to the height of the Z axis prior to the canned cycle.
G99 Canned Cycle R Plane Return moves the Z axis to the height specified by the Rnn.nnnn argument specified with the canned cycle.

CANNED CYCLES G81 AND G82

Canned cycles are used to automate a series of movements, reducing excessive lines of code. The process limits to Z Axis operation only. **X** and **Y** Axes (**G17**) specify the starting location. The next position moves in Absolute **(G90)** or Incremental **(G91)** movement. **G98 (Canned Cycle Initial Point Return)** and **G99 (Canned Cycle R Plane Return)** returns Z Axis to the specified height. Note that G99 and G98 are modal commands.

G80 (Canned Cycle Cancel) is required after completion of cycle.
See sample program below for **G90** and **G91** movements

***** Program O34000 in G91 Looping Canned Cycle Sample Program Folder on USB Drive *****

G81 DRILL CANNED CYCLE (GROUP 09)

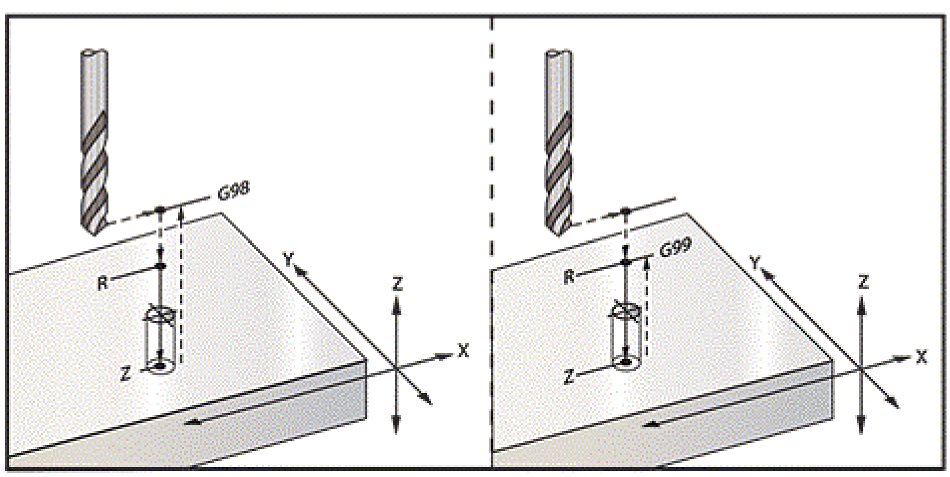
* **E** - Chip-clean RPM (Spindle reverses to remove chips after each cycle)
F - Feedrate
* **L** - Number of holes to drill if *G91* (Incremental Mode) is used
* **R** - Position of the R plane (position above the part)
* **X** - X-Axis motion command
* **Y** - Y-Axis motion command
Z - Position of the Z Axis at the bottom of hole

* indicates optional

CAUTION

Unless you specify otherwise, this canned cycle uses the most recently commanded spindle direction (M03 , M04 , or M05). If the program did not specify a spindle direction before it commands this canned cycle, the default is M03 (clockwise). If you command M05 , the canned cycle will run as a no-spincycle. This lets you run applications with self-driven tools, but it can also cause a crash. Be sure of the spindle direction command when you use this canned cycle.

G81 **Drill Canned Cycle**



This is a program to drill through an aluminum plate:

***** Program is in G81 Sample Program Folder on USB Drive *****

G82 SPOT DRILL CANNED CYCLE (GROUP 09)

* **E** - Chip-clean RPM (Spindle reverses to remove chips after each cycle)
  F - Feedrate
* **L** - Number of holes if *G91* (Incremental Mode) is used.
* **P** - The dwell time at the bottom of the hole
* **R** - Position of the R plane (position above the part)
* **X** - X-Axis location of hole
* **Y** - Y-Axis location of hole
  Z - Position of bottom of hole

* indicates optional

NOTE

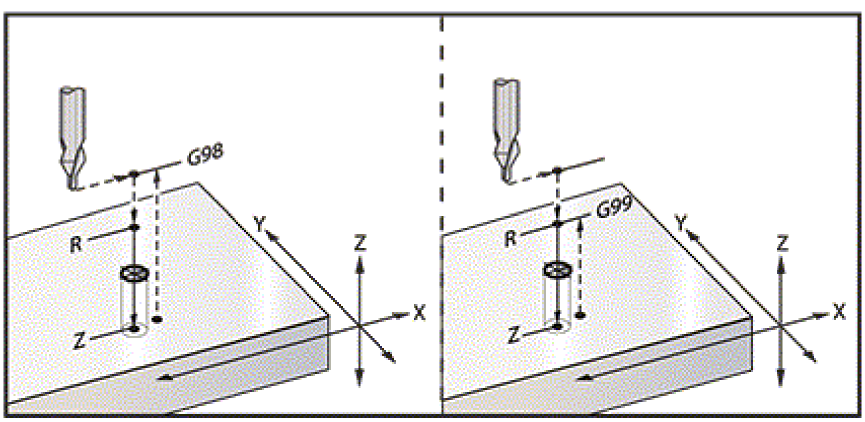
The P values are modal. This means if you are in the middle of a canned cycle and a G04 Pnn or an M97 Pnn is used the P value will be used for the dwell / subprogram as well as the canned cycle.

CAUTION

Unless you specify otherwise, this canned cycle uses the most recently commanded spindle direction (M03 , M04 , or M05). If the program did not specify a spindle direction before it commands this canned cycle, the default is M03 (clockwise). If you command M05 , the canned cycle will run as a no-spincycle. This lets you run applications with self-driven tools, but it can also cause a crash. Be sure of the spindle direction command when you use this canned cycle.

NOTE

G82 is similar to G81 except that there is the option to program a dwell (P).

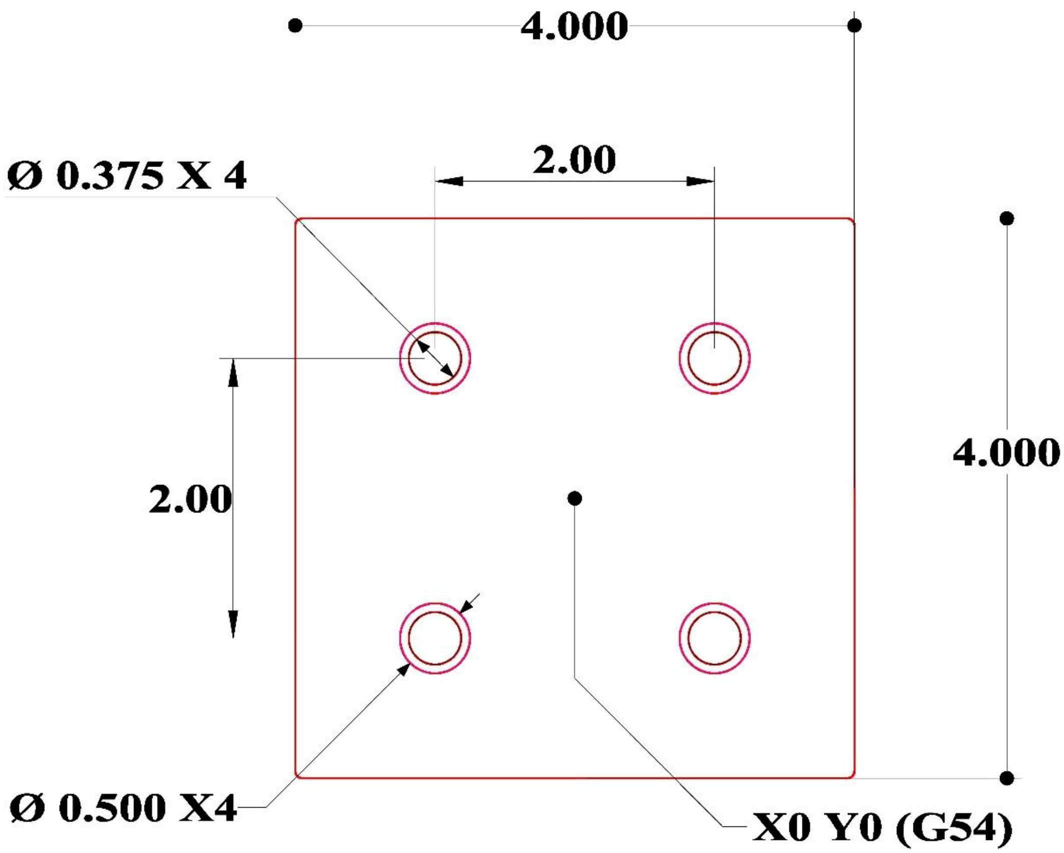


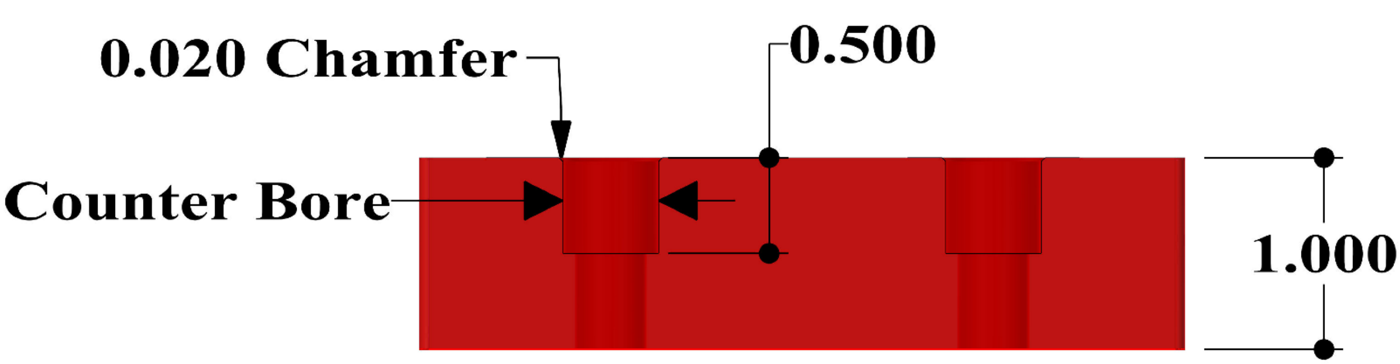
***G82* Spot Drilling Example**

***** Program is in G82 Sample Program Folder on USB Drive *****

Notes

Part Drawing





Tool Information for G81 and G82 Canned Cycle Exercise			Surface Feed	FPT / IPT
Tool 1		3/8 (.3750) Dia, 2 Flt, Screw Machine	100	.004
Tool 2		HSS Countersink, 3/4 (.7500) Dia, 82 Degree	60	.002
Tool 3		1/2 (.5000) Dia, 4 Flt, SE Square Carbide Endmill	250	.0013

**** Use the tool information above to calculate **speed (RPM)** and **feed (IPM)***
**** Put a **0.020"** Chamfer on Counter Bore using **Drill Point Depth and Countersink Diameter Formula***

Operation Order:

1. ***Spot Drill (G82 – Spot and Chamfer)***
2. ***Drill (G81)***
3. ***Counter Sink (G81)***

HAAS Simulator: Go to MDI Mode:	Save Program Referencing Page Number
G00 G90 G17 G40 G49 G54 (Safety Line) T___ M06 S______ M03 X_____ Y____ M08 G43 H____ Z1.0 G01 _______ → ... G82 ... ... G81 ... ... G81 ... ... M30 Run program in Graphic Mode.	

G83 NORMAL PECK DRILLING CANNED CYCLE (GROUP 09)

* * **E** - Chip-clean RPM (Spindle reverses to remove chips after each cycle)
* **F** - Feedrate
* * **I** - Size of first peck depth
* * **J** - Amount to reduce peck depth each pass
* * **K** - Minimum depth of peck
* * **L** - Number of holes if *G91* (Incremental Mode) is used, also G81 through G89.
* * **P** - Pause at end of last peck, in seconds (Dwell)
* * **Q** - Peck depth, always incremental
* * **R** - Position of the R plane (position above the part)
* * **X** - X-Axis location of hole
* * **Y** - Y-Axis location of hole
* **Z** - Position of the Z-Axis at the bottom of hole

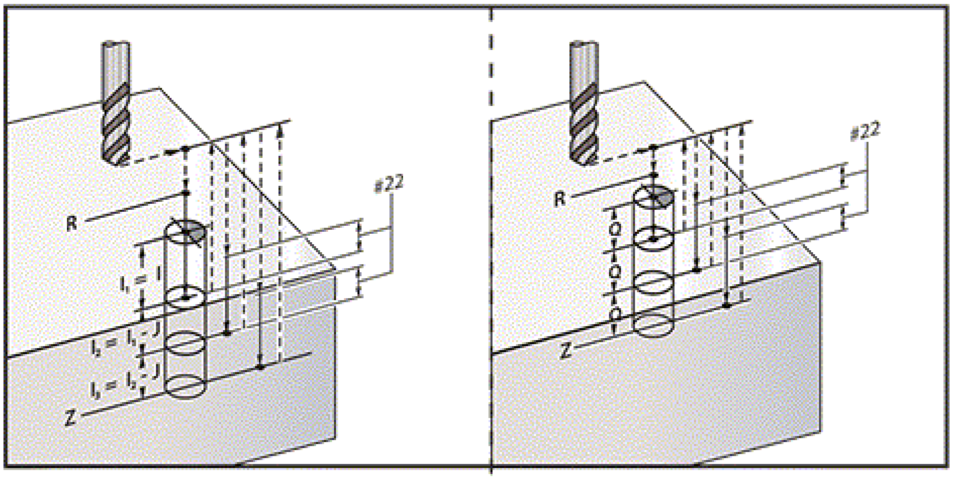
* indicates optional

If *I*, *J*, and *K* are specified, the first pass will cut in by the amount of I, each succeeding cut will be reduced by amount J, and the minimum cutting depth is *K*. Do not use a *Q* value when programming with *I*, *J*, and *K*.

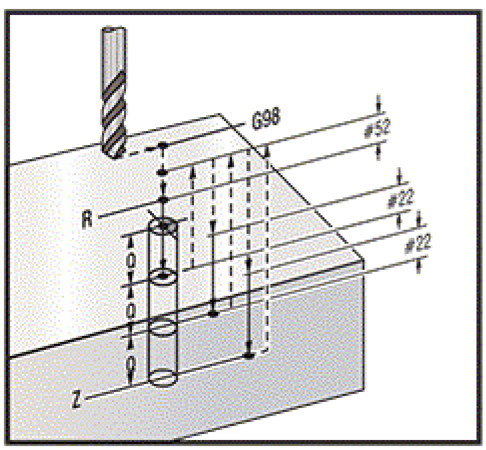
If *P* is specified, the tool will pause at the bottom of the hole for that amount of time. The following example will peck several times and dwell for 1.5 seconds:
G83 Z-0.62 F15. R0.1 Q0.175 P1.5 ;

The same dwell time will apply to all subsequent blocks that do not specify a dwell time.

G83 PECK DRILLING WITH *I*, *J*, *K* AND NORMAL PECK DRILLING: [#22] SETTING 22.



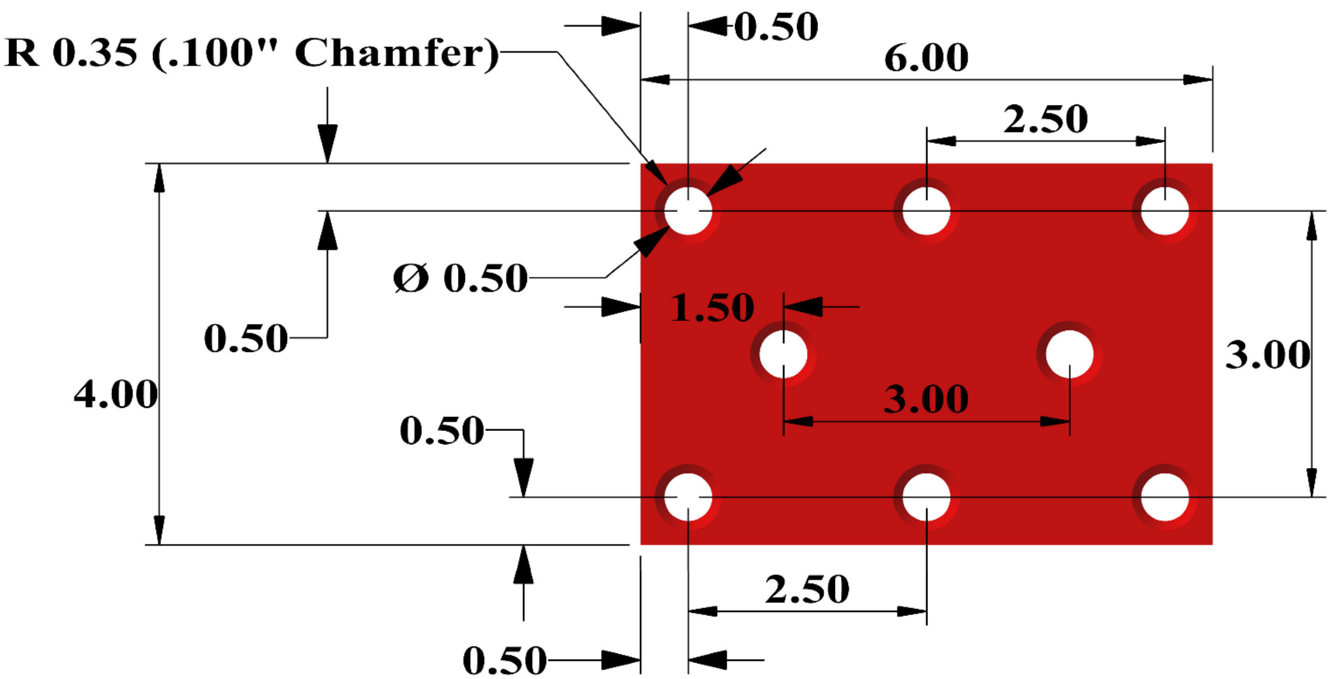
Setting 52 changes the way *G83* works when it returns to the R plane. Usually the R plane is set well above the cut to ensure that the peck motion allows the chips to get out of the hole. This wastes time as the drill starts by drilling empty space. If Setting 52 is set to the distance required to clear chips, you can set the R plane much closer to the part. When the chip-clearing move to R occurs, Setting 52 determines the Z-Axis distance above R.

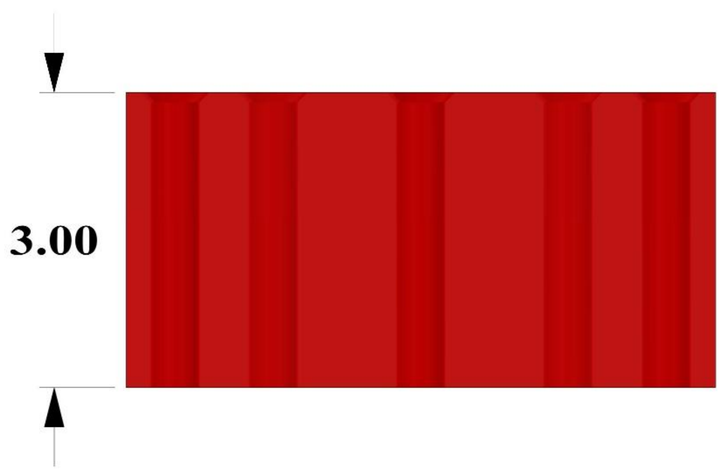


G83 PECK DRILLING CANNED CYCLE WITH SETTING 52 [#52]

***** Program is in G83 Sample Program Folder on USB Drive *****

Part Drawing





Tool Information for G83 and G82 Canned Cycle Exercise			Surface Feed	FPT / IPT
Tool 1		1/2 (.5000) Dia, 2 Flt, Jobber Length Drill, HSS Oxide Finish, 4.5000 LOC	95	.006
Tool 2		HSS Countersink, 3/4 (.7500) Dia, 82 Degree	60	.002

**** Use the tool information above to calculate* ***speed (RPM)*** *and* ***feed (IPM)***
**** Put a* ***0.100"*** *Chamfer on Counter Bore using* ***Drill Point Depth and Countersink Diameter Formula***
**** Set your Work Offset* ***(G54)***

Operation Order:

1. ***Spot Drill (G82 – Spot and Chamfer)***
2. ***Drill (G83)***

HAAS Simulator: Go to MDI Mode:	Save Program Referencing Page Number
G00 G90 G17 G40 G49 G54 (Safety Line) T___ M06 S______ M03 X_____ Y____ M08 G43 H____ Z1.0 G01 _______ → ... G82 ... ... G83 ... ... M30 Run program in Graphic Mode.	

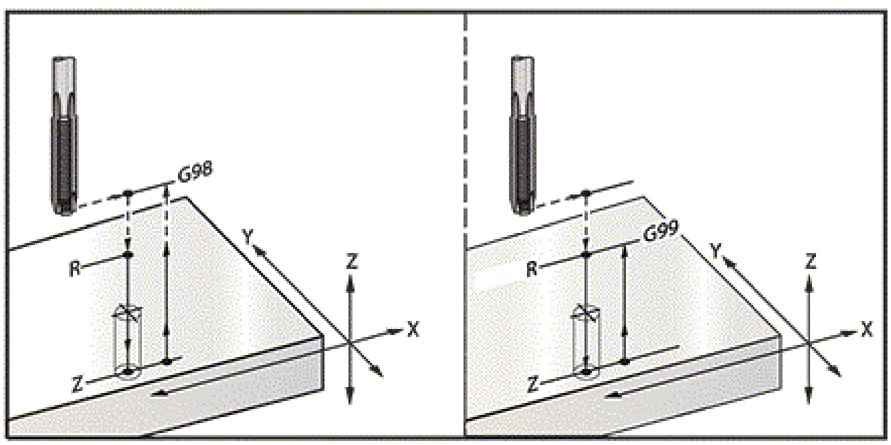
G84 TAPPING CANNED CYCLE (GROUP 09)

* **E** - Chip-clean RPM (Spindle reverses to remove chips after each cycle)
F - Feedrate
* **J** - Retract Multiple (Example: *J2* retracts twice as fast as the cutting speed, also refer to Setting 130)
* **L** - Number of holes if *G91* (Incremental Mode) is used
* **R** - Position of the R plane (Position above the part)
* **X** - X-Axis location of hole
* **Y** - Y-Axis location of hole
Z - Position of the Z Axis at the bottom of hole
* **Q** - Peck Depth (always incremental)
* **S** - Spindle speed

* indicates optional

NOTE You do not need to command a spindle start (M03 / M04) before G84. The canned cycle starts and stops the spindle as needed.

***G84* Tapping Canned Cycle**



*** Program is in G84 Sample Program Folder on USB Drive ***

G70 BOLT HOLE CIRCLE (GROUP 00)

I - Radius
* **J** - Starting angle (0 to 360.0 degrees CCW from horizontal;
or 3 oclock position)
L - Number of holes evenly spaced around the circle

*indicates optional

This non-modal G code must be used with one of the canned cycles *G73*, *G74*, *G76*, *G77*, or *G81*-*G89*. A canned cycle must be active so that at each position, a drill or tap function is performed. See also G-code Canned Cycles section.

G71 BOLT HOLE ARC (GROUP 00)

I - Radius
* **J** - Starting angle (degrees CCW from horizontal)
K - Angular spacing of holes (+ or --)
L - Number of holes

*indicates optional

This non-modal G code is similar to *G70* except that it is not limited to a complete circle. *G71* belongs to Group 00 and thus is non-modal. A canned cycle must be active so that at each position, a drill or tap function is performed.

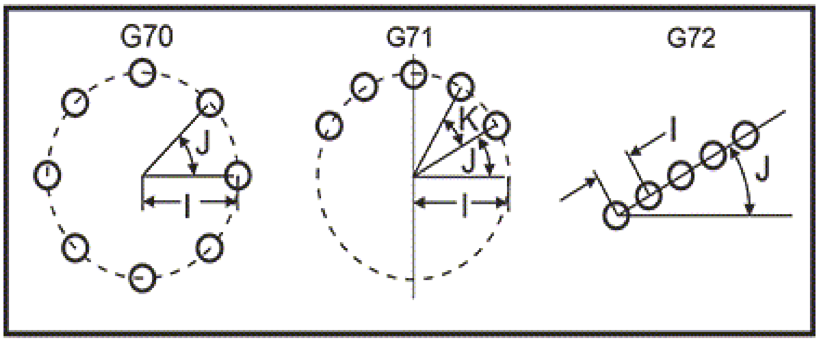
G72 BOLT HOLES ALONG AN ANGLE (GROUP 00)

I - Distance between holes
* **J** - Angle of line (degrees CCW from horizontal)
L - Number of holes

*indicates optional

This non-modal G code drills *L* number of holes in a straight line at the specified angle. It operates similarly to *G70*. For a *G72* to work correctly, a canned cycle must be active so that at each position, a drill or tap function is performed.

G70, *G71*, and *G72* Bolt Holes: [I] Radius of bolt circle (G70, G71), or distance between holes (G72), [J] Starting angle from the 3 oclock position, [K] Angular spacing between holes, [L] Number of holes.



Rules for Bolt Pattern Canned Cycles

1. Place the tool at the center of the bolt pattern (for G70 or G71), or at the starting hole location (for G72), before the canned cycle execution.

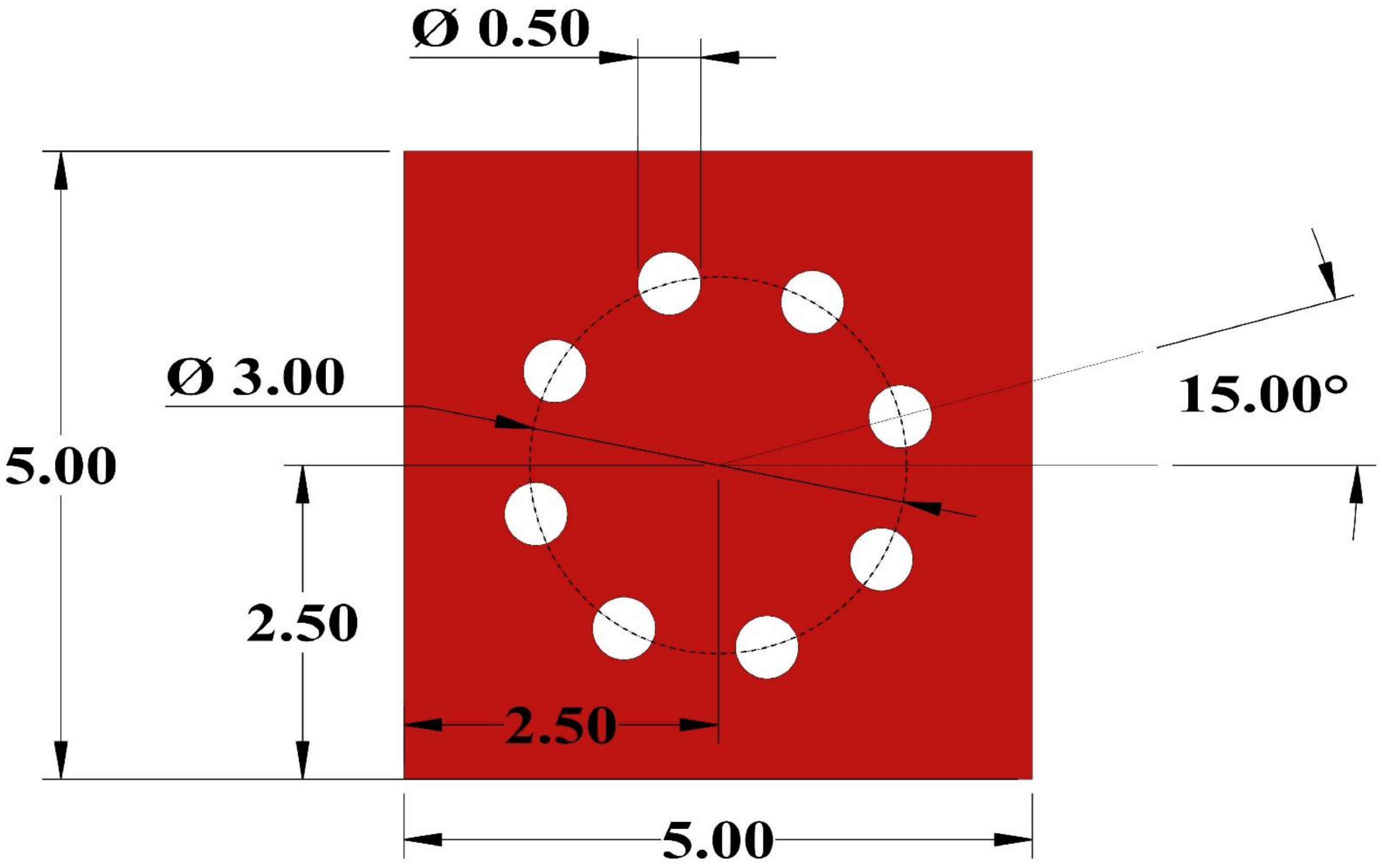
2. The *J* code is the angular starting position and is always 0 to 360 degrees counterclockwise from the three oclock position.

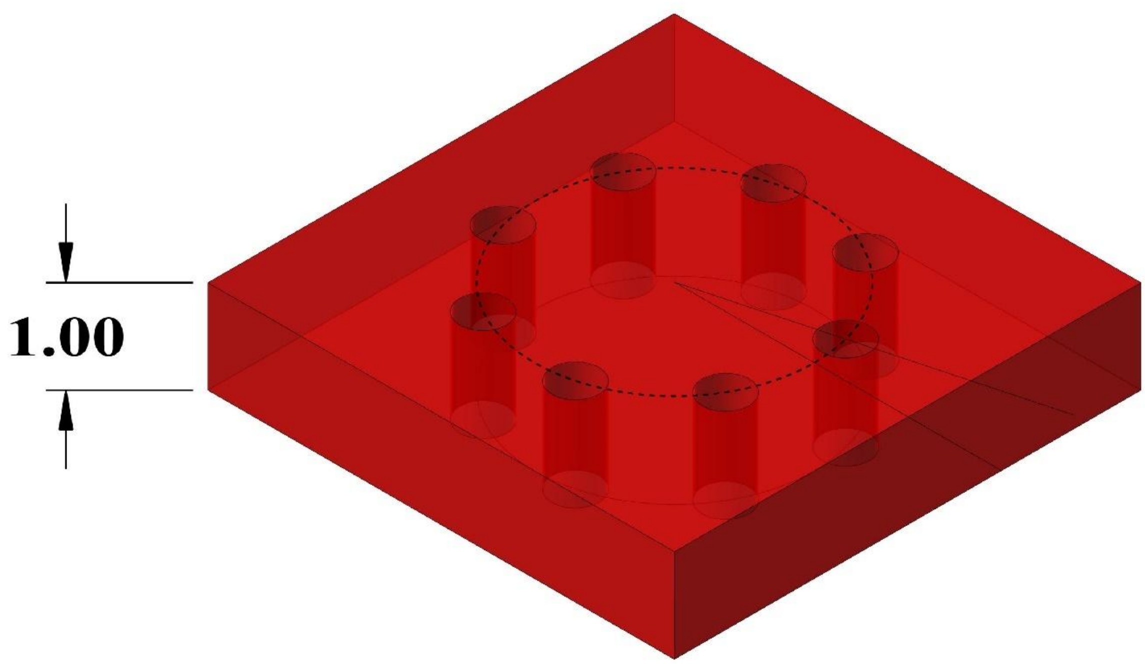
3. For G70 and G71 cycles, put an *L0* on the initial canned cycle line to skip drilling at the center of the hole pattern. You can also turn off Setting 28 to prevent a hole from being drilled at the initial X/Y position. Refer to 28 - Can Cycle No Linear Axes for more information on Setting 28.

Notes

G81 CANNED CYCLE WITH G70 (BOLT HOLE CIRCLE) EXERCISE

Part Drawing





Tool Information for G81 with G70			Surface Feed	FPT / IPT
Tool 1		1/2 (.5000) Dia, 2 Flt, 2.0000 LOC	100	.003

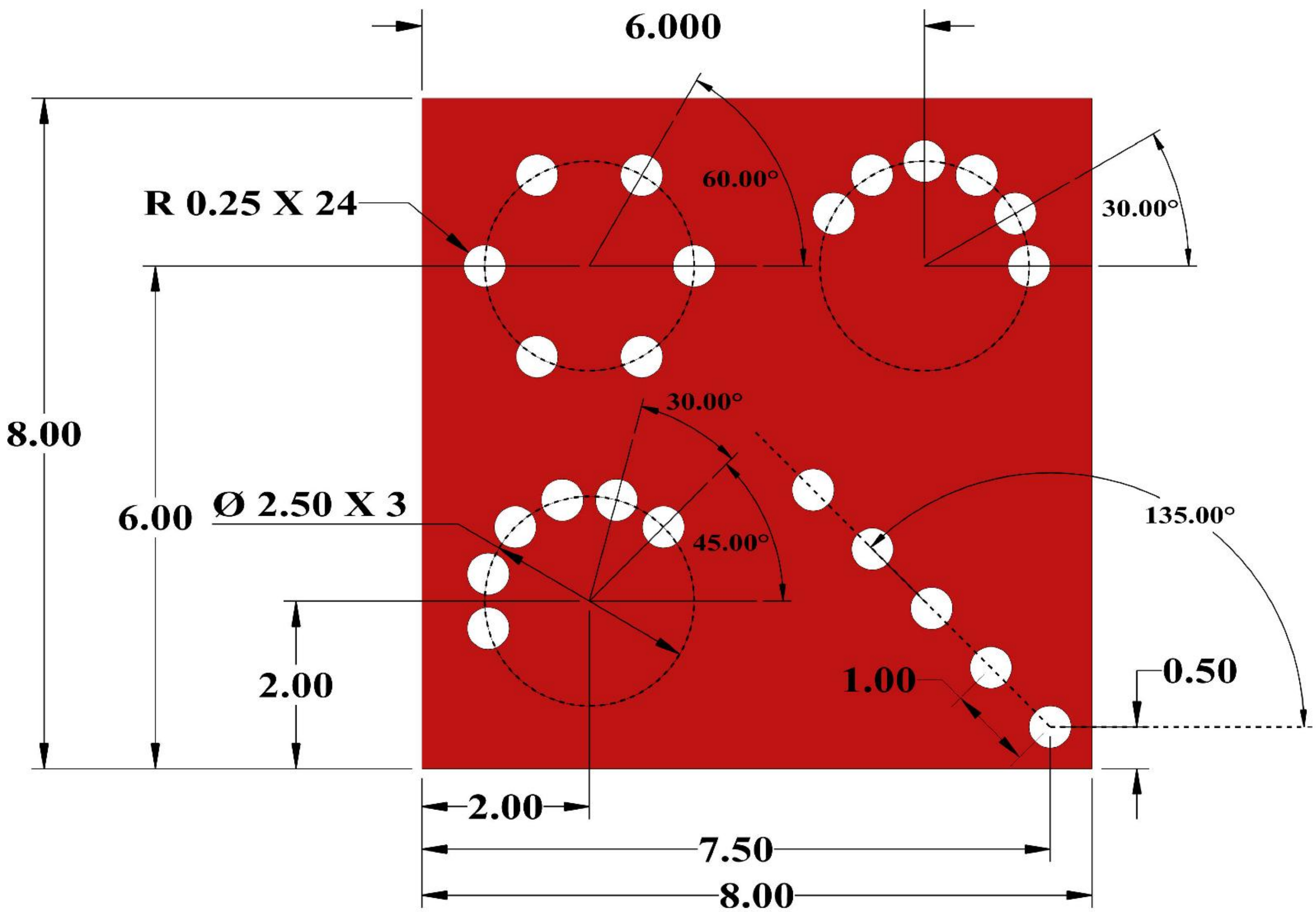
**** Use the tool information above to calculate* ***speed (RPM)*** *and* ***feed (IPM)***
**** Set your Work Offset* ***(G54)***

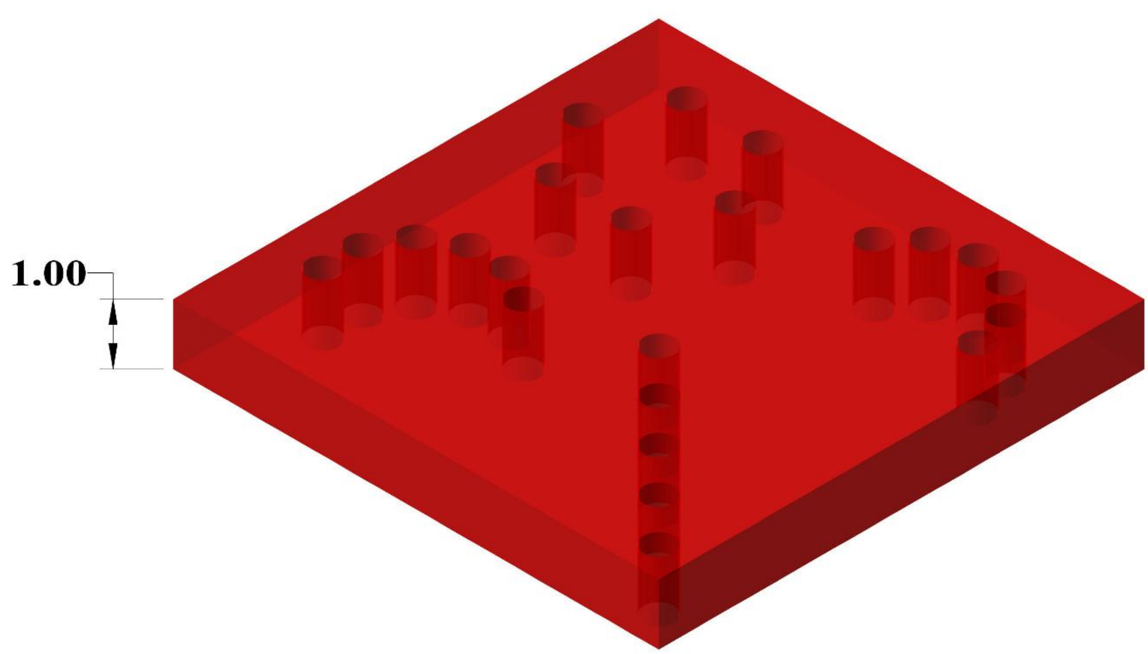
1. ***Drill (G81) with G70 Bolt Hole Circle***

HAAS Simulator: Go to MDI Mode:	Save Program Referencing Page Number
G00 G90 G17 G40 G49 G54 (Safety Line) **T___ M06** **S______ M03** **X_____ Y____** **M08** **G43 H____ Z1.0** **G01 _______ →** **...** **...** **G81 ... L0 _______________ (L0 does not drill a hole in the center of bolt hole pattern.)** **...** **...** **M30** **Run program in Graphic Mode.**	

G81 CANNED CYCLE WITH G70, G71 AND G72 EXERCISE

Part Drawing





Tool Information for G81 With G70, G71, and G72			Surface Feed	FPT / IPT
Tool 1		1/2 (.5000) Dia, 2 Flt, 2.0000 LOC	100	.003

**** Use the tool information above to calculate **speed (RPM)** and **feed (IPM)***
**** Set your Work Offset **(G54)***

1. Drill (G81) with G70, G71 and G72

HAAS Simulator: Go to MDI Mode:	Save Program Referencing Page Number
G00 G90 G17 G40 G49 G54 (Safety Line) T___ M06 S______ M03 X_____ Y____ M08 G43 H____ Z1.0 G01 _______ → ... ... G81... L0 _______________ (L0 does not drill a hole in the center of bolt hole pattern.) ... ... M30 Run program in Graphic Mode.	

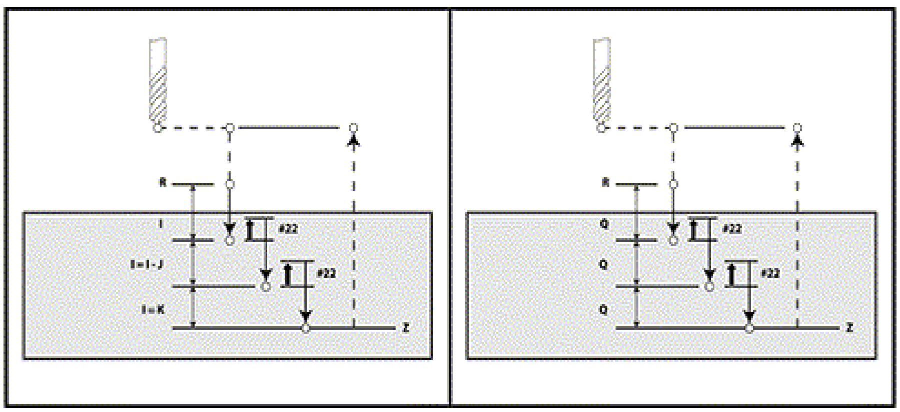
G73 HIGH-SPEED PECK DRILLING CANNED CYCLE (GROUP 09)

F - Feedrate
* **I** - First peck depth
* **J** - Amount to reduce pecking depth for pass
* **K** - Minimum peck depth (The control calculates the number of pecks)
* **L** - Number of loops (Number of holes to drill) if *G91* (Incremental Mode) is used
* **P** - Pause at the bottom of the hole (in seconds)
* **Q** - Peck Depth (always incremental)
* **R** - Position of the R plane (Distance above part surface)
* **X** - X-Axis location of hole
* **Y** - Y-Axis location of hole
Z - Position of the Z-Axis at the bottom of hole

* 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 M97 Pnn is used the P value will be used for the dwell / subprogram as well as the canned cycle.

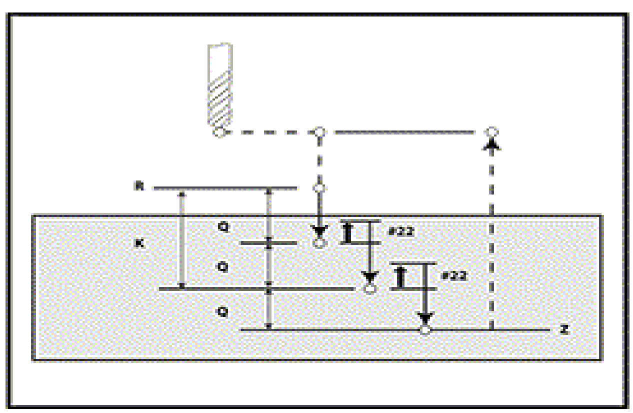
***G73* Peck Drilling. Left: Using *I*, *J*, and *K* Addresses. Right: Using Only the *Q* Address. [#22] Setting 22.**



I, *J*, *K*, and *Q* are always positive numbers.

There are three methods to program a *G73*: using the *I*, *J*, *K* addresses, using the *K* and *Q* addresses, and using only a *Q* address.

If *I*, *J*, and *K* are specified, The first pass will cut in by the value *I*, each succeeding cut will be reduced by the value of *J*, and the minimum cutting depth is *K*. If *P* is specified, the tool will pause at the bottom of the hole for that amount of time.



If *K* and *Q* are both specified, a different operating mode is selected for this canned cycle. In this mode, the tool is returned to the R plane after the number of passes totals up to the *K* amount.

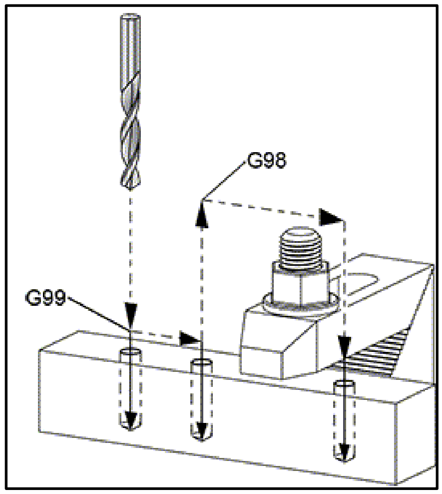
If only *Q* is specified, a different operating mode is selected for this canned cycle. In this mode, the tool is returned to the *R* plane after all pecks are completed, and all pecks will be equal to the *Q* value.

***G73* Peck Drilling Canned Cycles using the *K* and *Q* Addresses: [#22] Setting 22.**

G98 CANNED CYCLE INITIAL POINT RETURN (GROUP 10)

Using *G98*, the Z-Axis returns to its initial starting point (the Z position in the block before the canned cycle) between each X/Y position. This lets you program up and around areas of the part, clamps, and fixtures.

G98 Initial Point Return. After the second hole, the Z Axis returns to the starting position [G98] to move over the toe clamp to the next hole position.

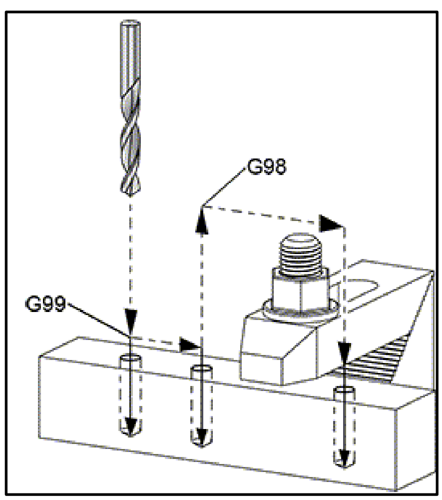


***** Program is in G98 Sample Program Folder on USB Drive *****

G99 CANNED CYCLE R PLANE RETURN (GROUP 10)

Using *G99*, the Z-Axis will stay at the R plane between each X and/or Y location. When obstructions are not in the path of the tool *G99* saves machining time.

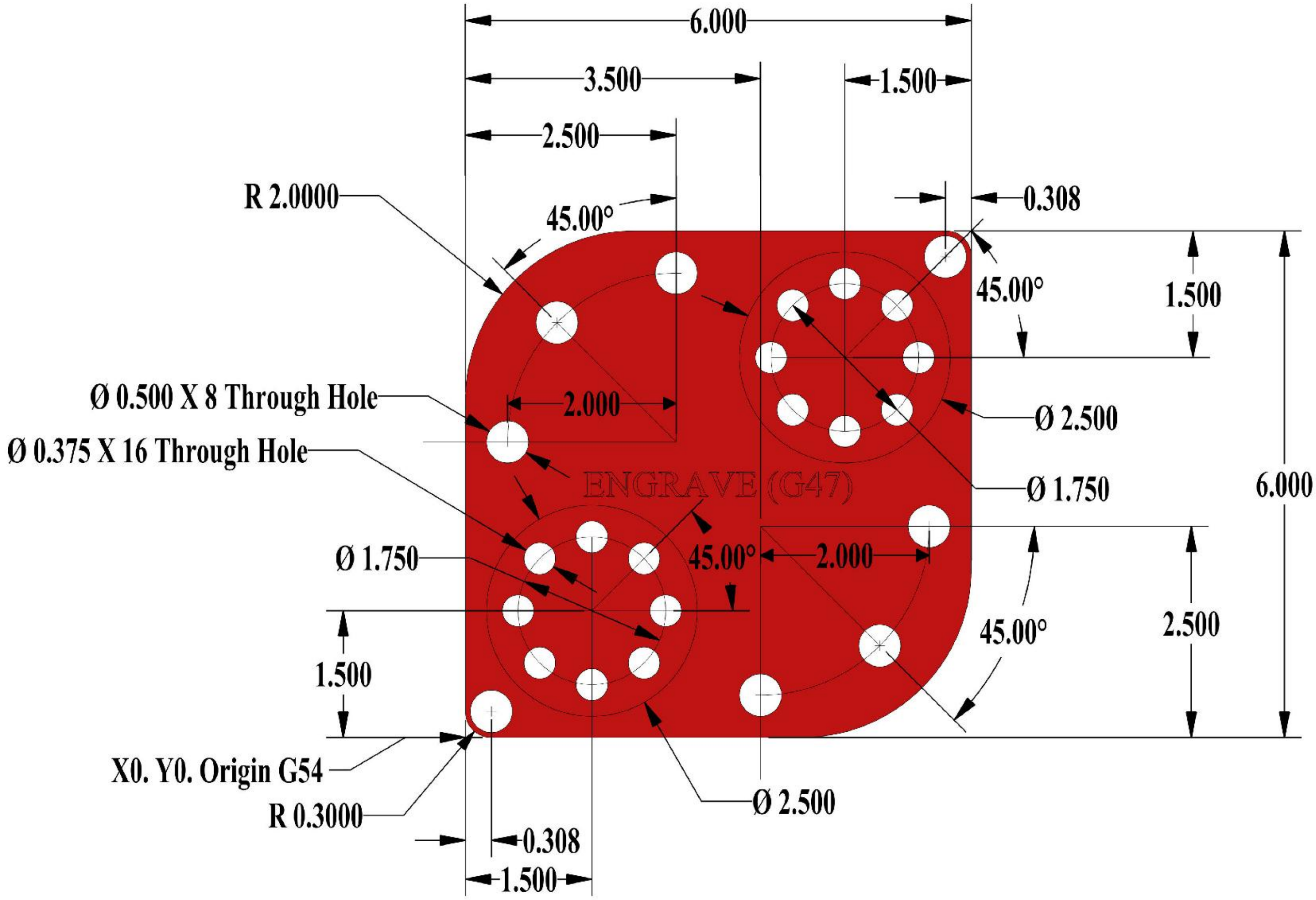
*G99*R Plane Return. After the first hole, the Z Axis returns to the R plane position [G99] and moves to the second hole position. This is a safe move in this case because there are no obstacles.

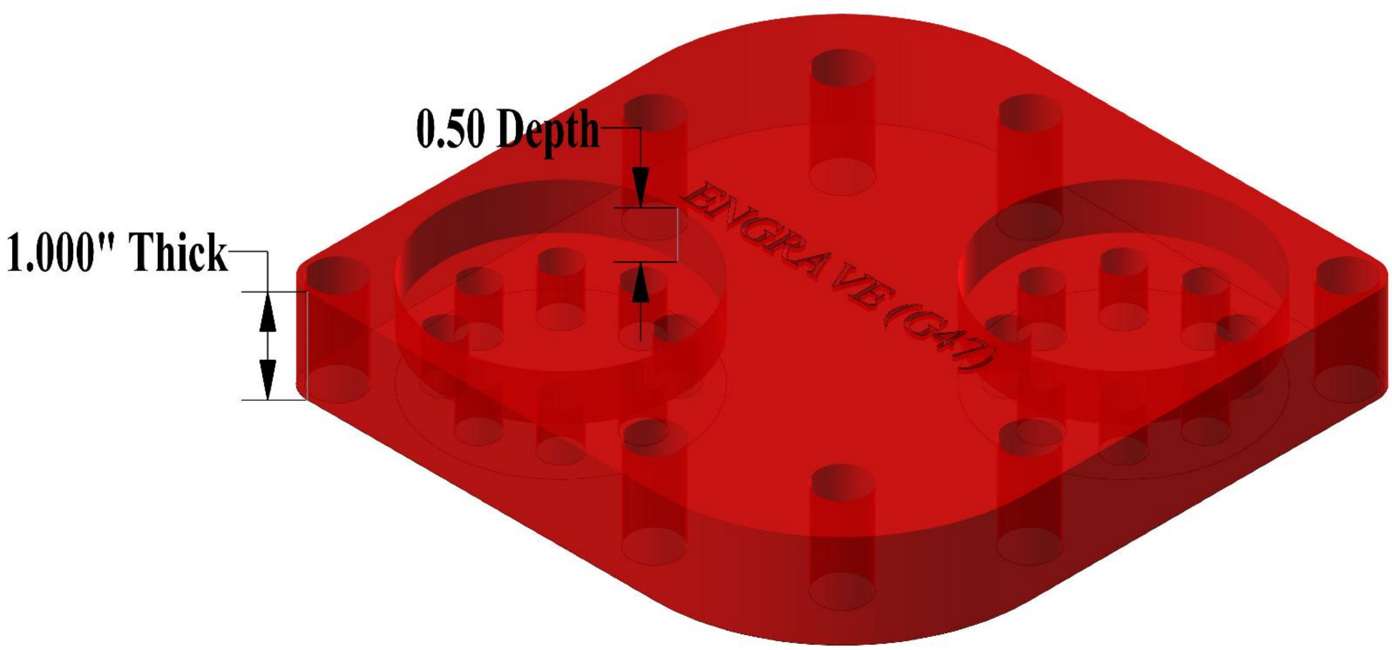


***** Program is in G99 Sample Program Folder on USB Drive *****

FINAL EXERCISE

Part Drawing





Tool Information for Final Exercise			Surface Feed	FPT / IPT
Tool 1		**HSS Countersink, 3/4 (.7500) Dia, 82 Degree**	**60**	**.002**
Tool 2		**1/2 (.5000) Dia, 2 Flt, 2.0000 LOC**	**100**	**.003**
Tool 3		**3/8 (.3750) Dia, 2 Flt, Screw Machine**	**100**	**.004**
Tool 4		**1/2 (.5000) Dia, 4 Flt, SE Square Carbide Endmill**	**250**	**.0013**
Tool 5		**90° Angle, .0050 Tip Dia, .1250 Shank**	**60**	**.001**

**** Create part using Canned Cycles.*
**** Outside Profile is an option.*
**** Engraving using **G47**. Adjust TEXT to fit center of part.*
**** **0.020”** Chamfer on all holes*
**** Mill out circular pockets using **VPS**.*
**** Use the tool information above to calculate **speed (RPM)** and **feed (IPM)***
**** Set your Work Offset **(G54)***

HAAS Simulator: Go to MDI Mode:	Save Program Referencing Page Number
G00 G90 G17 G40 G49 G54 (Safety Line) **T___ M06** **S______ M03** **X____ Y____** **M08** **G43 H____ Z1.0** **G01 _______ →** **...** **...** **...** **...** **M30** **Run program in Graphic Mode.**	

G150 GENERAL PURPOSE POCKET MILLING (GROUP 00)

D - Tool radius/diameter offset selection
F - Feedrate
I - X-Axis cut increment (positive value)
J - Y-Axis cut increment (positive value)
K - Finishing pass amount (positive value)
P - Subprogram number that defines pocket geometry
Q - Incremental Z-Axis cut depth per pass (positive value)
* **R** - Position of the rapid R-plane location
* **S** - Spindle speed
X - X start position
Y - Y start position
Z - Final depth of pocket
* indicates optional

The *G150* starts by positioning the cutter to a start point inside the pocket, followed by the outline, and completes with a finish cut. The end mill will plunge in the Z-Axis. A subprogram *P###* is called, which defines the pocket geometry of a closed area using *G01*, *G02*, and *G03* motions in the X and Y axes on the pocket. The *G150* command will search for an internal subprogram with a N-number specified by the P-code. If that is not found the control will search for an external subprogram. If neither are found, alarm 314 Subprogram Not In Memory will be generated.

NOTE When defining the G150 pocket geometry in the subprogram, do not move back to the starting hole after the pocket shape is closed.

NOTE The pocket geometry subprogram cannot use macro variables.

An *I* or *J* value defines the roughing pass amount the cutter moves over for each cut increment. If *I* is used, the pocket is roughed out from a series of increment cuts in the X-Axis. If *J* is used, the increment cuts are in the Y-Axis.

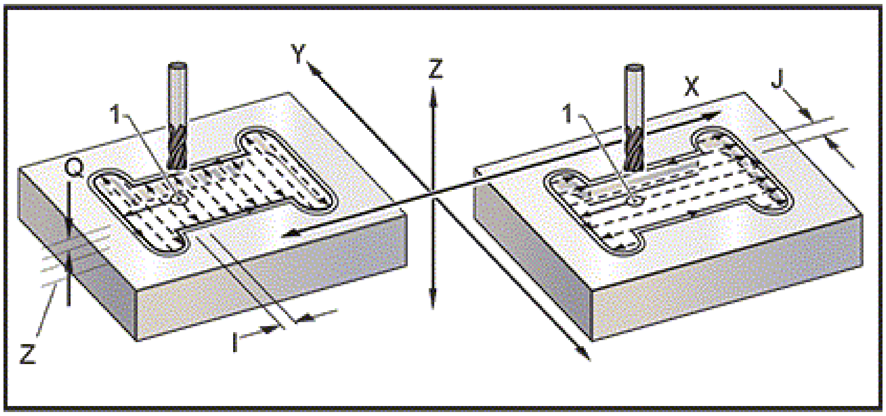
The *K* command defines a finish pass amount on the pocket. If a *K* value is specified, a finish pass is performed by *K* amount, around the inside of pocket geometry for the last pass and is done at the final Z depth. There is no finishing pass command for the Z depth.

The *R* value needs to be specified, even if it is zero (*R0*), or the last *R* value that was specified will be used.
Multiple passes in the pocket area are done, starting from the R plane, with each Q (Z-Axis depth) pass to the final depth. The *G150* command will first make a pass around pocket geometry, leaving stock with *K*, then doing passes of *I* or *J* roughing out inside of pocket after feeding down by the value in Q until the Z depth is reached.

The *Q* command must be in the *G150* line, even if only one pass to the Z depth is desired.
The *Q* command starts from the R plane.

Notes: The subprogram (*P*) must not consist of more than 40 pocket geometry moves.
It may be necessary to drill a starting point, for the *G150* cutter, to the final depth (Z). Then position the end mill to the start location in the XY axes within the pocket for the *G150* command.

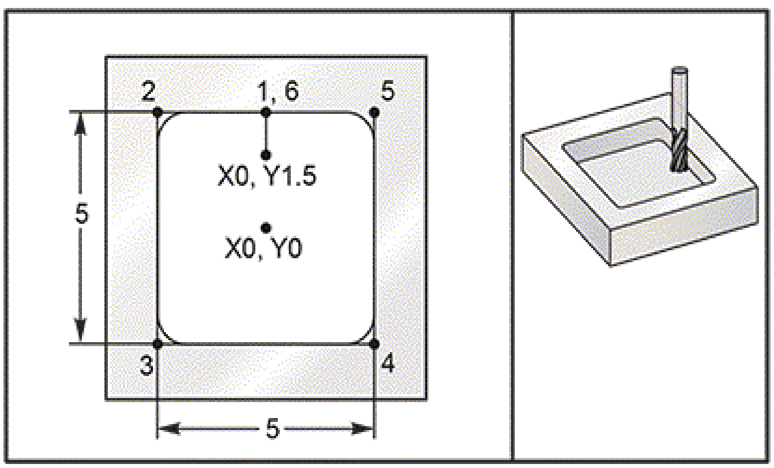
G150 GENERAL POCKET MILLING: [1] START POINT, [Z] FINAL DEPTH.



Main Program O61501 (G150 GENERAL POCKET MILLING)
Sub Program O61502 (G150 GENERAL POCKET MILL SUBPROGRAM)
***** Programs are in G150 Sample Programs Folder on USB Drive *****

Square Pocket

G150 GENERAL PURPOSE POCKET MILLING: 0.500 DIAMETER ENDMILL.



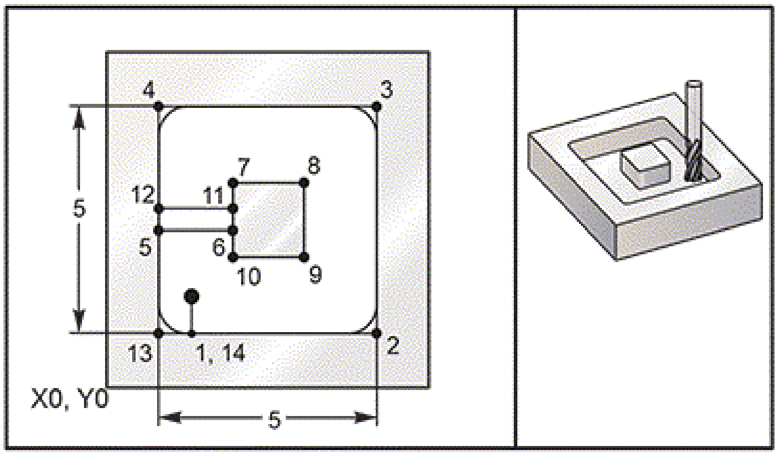
5.0 x 5.0 x 0.500 DP. Square Pocket
Main Program O61503 (G150 SQUARE POCKET MILLING)
Subprogram O61505 (G150 INCREMENTAL SQUARE POCKET MILLING SUBPROGRAM)
***** Programs are in G150 Sample Programs Folder on USB Drive *****

ABSOLUTE AND INCREMENTAL EXAMPLES OF A SUBPROGRAM CALLED UP BY THE *P####* COMMAND IN THE *G150* LINE:

Absolute Subprogram O61504 (G150 ABSOLUTE SQUARE POCKET MILLING SUBPROGRAM)
Incremental Subprogram O61511 (G150 INCREMENTAL SQUARE POCKET MILLING SUBPROGRAM) ;
***** Programs are in G150 Sample Programs Folder on USB Drive *****

SQUARE ISLAND

G150 **Pocket Milling Square Island: 0.500 diameter endmill.**



5.0 x 5.0 x 0.500 DP. Square Pocket with Square Island

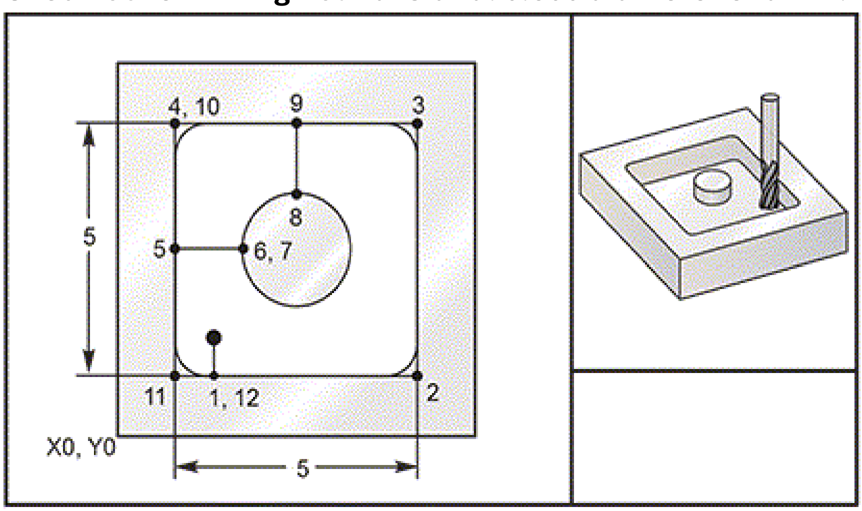
Main Program O61506 (G150 SQUARE ISLAND POCKET MILLING)

Subprogram O61507 (G150 SQUARE ISLAND POCKET MILLING SUBPROGRAM)

***** Programs are in G150 Sample Programs Folder on USB Drive *****

ROUND ISLAND

G150 **Pocket Milling Round Island: 0.500 diameter endmill.**



5.0 x 5.0 x 0.500 DP. Square Pocket with Round Island

Main Program O61508 (G150 SQ POCKET W/ ROUND ISLAND MILLING)

Subprogram O61509 (G150 SQ POCKET W/ ROUND ISLAND MILLING SUBPROGRAM)

***** Programs are in G150 Sample Programs Folder on USB Drive *****

SPECIAL G-CODES

Special G-codes are used for complex milling. These include:

- Engraving (G47)
- Pocket Milling (G12, G13, and G150)
- Rotation and Scaling (G68, G69, G50, G51)
- Mirror Image (G101 and G100)

ENGRAVING

The G47 Text Engraving G-code lets you engrave text (including some ASCII characters) or sequential serial numbers with a single block of code.

Refer to G47 Text Engraving (Group 00) for more information on engraving.

POCKET MILLING

There are two types of pocket milling G-codes on the Haas control:

Circular Pocket Milling is performed with the G12 Clockwise Circular Pocket Milling Command and the G13 Counter-Clockwise Circular Pocket Milling Command G-codes.

The G150 General Purpose Pocket Milling uses a subprogram to machine user-defined pocket geometries.

Make sure that the subprogram geometry is a fully closed shape. Make sure that the X-Y starting point in the G150 command is within the boundary of the fully closed shape. Failure to do so may cause Alarm 370 - Pocket Definition Error.

Refer to G12 Circular Pocket Milling CW / G13 Circular Pocket Milling CCW (Group 00) for more information on the pocket milling G-codes.

ROTATION AND SCALING

NOTE: You must purchase the rotation and scaling option to use these features. A 200-hour option tryout is also available.

G68 Rotation is used to rotate the coordinate system in the desired plane. You can use this feature together with G91 Incremental Programming mode to machine symmetrical patterns. G69 cancels rotation.

G51 applies a scaling factor to the positioning values in blocks after the G51 command. G50 cancels scaling. You can use scaling together with rotation, but be sure to command scaling first.
Refer to G68 Rotation (Group 16) for more information on the rotation and scaling G-codes.

MIRROR IMAGE

G101 Enable Mirror Image will mirror axis motion about the specified axis. Settings 45-48, 80 and 250 enable mirror imaging about the X, Y, Z, A, B, and C axes.

The mirror pivot point along an axis is defined by the Xnn.nn argument. This can be specified for a Y Axis that is enabled on the machine and in the settings by using the axis to mirror as the argument. G100 cancels G101.

Refer to G100/G101 Disable/Enable Mirror Image (Group 00) for more information on the mirror image G-codes.

ROTATION AND SCALING

Rotation lets you rotate a pattern to another location or around a circumference. Scaling reduces or enlarges a toolpath or pattern.

Refer to G68 Rotation to learn how to use this control feature.

RIGID TAPPING

This option synchronizes the spindle's RPM with the feedrate during a tapping operation. Refer to G84 Tapping Canned Cycle to learn how to use this control feature.

Refer to Tap Breakage - Troubleshooting guide to view solutions to some of the most common causes for tap breakage.

M19 SPINDLE ORIENTATION

The Spindle Orientation lets you position the spindle to a programmed angle. This option provides inexpensive, accurate positioning.

Refer to the M19 Orient Spindle for more information on how to use this software control feature.

HIGH-SPEED MACHINING

The Haas high-speed machining option allows faster feedrates and more complex toolpaths.

HSM uses a motion algorithm called Acceleration Before Interpolation combined with full look-ahead to provide contouring feeds up to 1200 ipm (30.5 m/min) without risk of distortion to the programmed path.

This reduces cycle times, improves accuracy, and smooths motion.

Refer to G187 Accuracy Control to control the smoothness and max corner rounding value when cutting a part.

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 a 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:

In the Device Manager (LIST PROGRAM), select the directory that you want to add to the list.
Press F3.

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)

A local subprogram is a block of code in the main program that is referenced several times by the main program. Local subprograms are commanded (called) using an M97 and Pnnnnn, which refers to the N line number of the local subprogram.

The local subprogram format is to end the main program with an M30 then enter the local subprograms after the M30. Each subprogram must have an N line number at the start and a M99 at the end that will send the program back to the next line in the main program.

***** See sample Program in M97 Sample Program Folder on USB Drive; O40009.NC *****

EXTERNAL SUBPROGRAM (M98)

An external subprogram is a separate program that the main program references. Use M98 to command (call) an external subprogram, with Pnnnnn to refer to the program number you want to call.

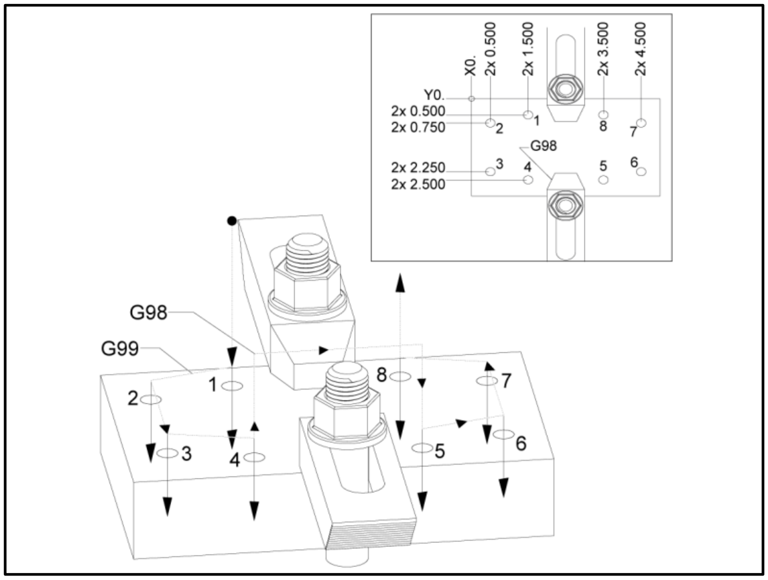
When your program calls an M98 subprogram, the control looks for the subprogram in the main program's directory. If the control cannot find the subprogram in the main program's directory, it then looks in the location specified in Setting 251. Refer to page5 for more information. An alarm occurs if the control cannot find the subprogram.

In this example, the subprogram (program O40008) specifies (8) positions. It also includes a G98 command at the move between positions 4 and 5. This causes the Z Axis to return to the initial starting point instead of the R plane, so the tool passes over the work holding.

THE MAIN PROGRAM (PROGRAM O40007) SPECIFIES (3) DIFFERENT CANNED CYCLES:

- G81 Spot drill at each position
- G83 Peck drill at each position
- G84 Tap at each position

Each canned cycle calls the subprogram and does the operation at each position.



***** Main Program O40007.NC is in M98 Sample Program Folder on USB Drive *****

***** Sub Program O40008.NC is in M98 Sample Program Folder on USB Drive *****

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.

Here are a few examples of the applications for Macros. The examples are outlines and not complete macro programs.

Tools For Immediate, On-Table Fixturing - You can semi-automate many setup procedures to assist the machinist. You can reserve tools for immediate situations that you did not anticipate in your application design. For instance, suppose a company uses a standard clamp with a standard bolt hole pattern. If you discovered, after setup, that a fixture needs an additional clamp, and suppose that you programmed macro subprogram 2000 to drill the bolt pattern of the clamp, then you only need this two-step procedure to add the clamp to the fixture:

a) Jog the machine to the X, Y, and Z coordinates and angle where you want to place the clamp. Read the position coordinates from the machine display.
b) Execute this command in MDI mode:

G65 P2000 Xnnn Ynnn Znnn Annn ;

Where nnn are the coordinates determined in Step a). Here, macro 2000 (P2000) does the work since it was designed to drill the clamp bolt hole pattern at the specified angle of A. Essentially, this is a custom canned cycle.

Simple Patterns That Are Repeated- You can define and store repeated patterns with macros. For example:

a) Bolt hole patterns
b) Slotting
c) Angular patterns, any number of holes, at any angle, with any spacing
d) Specialty milling such as soft jaws
e) Matrix Patterns, (e.g. 12 across and 15 down)
f) Fly-cutting a surface, (e.g. 12 inches by 5 inches using a 3 inch fly cutter)

Automatic Offset Setting Based On The Program - With macros, coordinate offsets can be set in each program so that setup procedures become easier and less error-prone (macro variables #2001-2800).

MILL G-CODES

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

*** Click on **Description** in **BLUE** for more information ***

Code	Description	Group
G00	Rapid Motion Positioning	01
G01	Linear Interpolation Motion	01
G02	Circular Interpolation Motion CW	01
G03	Circular Interpolation Motion CCW	01
G04	Dwell	00
G09	Exact Stop	00
G10	Set Offsets	00
G12	Circular Pocket Milling CW	00
G13	Circular Pocket Milling CCW	00
G17	XY Plane Selection	02
G18	XZ Plane Selection	02
G19	YZ Plane Selection	02
G20	Select Inches	06
G21	Select Metric	06
G28	Return To Machine Zero Point	00
G29	Return From Reference Point	00
G31	Feed Until Skip	00
G35	Automatic Tool Diameter Measurement	00
G36	Automatic Work Offset Measurement	00
G37	Automatic Tool Offset Measurement	00
G40	Cutter Compensation Cancel	07
G41	2D Cutter Compensation Left	07
G42	2D Cutter Compensation Right	07
G43	Tool Length Compensation + (Add)	08
G44	Tool Length Compensation - (Subtract)	08
G47	Text Engraving	00
G49	G43/G44/G143 Cancel	08
G50	Cancel Scaling	11
G51	Scaling	11
G52	Set Work Coordinate System	00 or 12

G53	Non-Modal Machine Coordinate Selection	00
G54	Select Work Coordinate System #1	12
G55	Select Work Coordinate System #2	12
G56	Select Work Coordinate System #3	12
G57	Select Work Coordinate System #4	12
G58	Select Work Coordinate System #5	12
G59	Select Work Coordinate System #6	12
G60	Uni-Directional Positioning	00
G61	Exact Stop Mode	15
G64	G61 Cancel	15
G65	Macro Subprogram Call Option	00
G68	Rotation	16
G69	Cancel G68 Rotation	16
G70	Bolt Hole Circle	00
G71	Bolt Hole Arc	00
G72	Bolt Holes Along an Angle	00
G73	High-Speed Peck Drilling Canned Cycle	09
G74	Reverse Tap Canned Cycle	09
G76	Fine Boring Canned Cycle	09
G77	Back Bore Canned Cycle	09
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 In, Dwell, Bore Out Canned Cycle	09
G90	Absolute Position Command	03
G91	Incremental Position Command	03
G92	Set Work Coordinate Systems Shift Value	00
G93	Inverse Time Feed Mode	05
G94	Feed Per Minute Mode	05
G95	Feed per Revolution	05
G98	Canned Cycle Initial Point Return	10
G99	Canned Cycle R Plane Return	10
G100	Cancel Mirror Image	00
G101	Enable Mirror Image	00
G103	Limit Block Buffering	00
G107	Cylindrical Mapping	00
G110	#7 Coordinate System	12
G111	#8 Coordinate System	12
G112	#9 Coordinate System	12
G113	#10 Coordinate System	12
G114	#11 Coordinate System	12
G115	#12 Coordinate System	12
G116	#13 Coordinate System	12

G117	#14 Coordinate System	12
G118	#15 Coordinate System	12
G119	#16 Coordinate System	12
G120	#17 Coordinate System	12
G121	#18 Coordinate System	12
G122	#19 Coordinate System	12
G123	#20 Coordinate System	12
G124	#21 Coordinate System	12
G125	#22 Coordinate System	12
G126	#23 Coordinate System	12
G127	#24 Coordinate System	12
G128	#25 Coordinate System	12
G129	#26 Coordinate System	12
G136	Automatic Work Offset Center Measurement	00
G141	3D+ Cutter Compensation	07
G143	5-Axis Tool Length Compensation +	08
G150	General Purpose Pocket Milling	00
G154	Select Work Coordinates P1-P99	12
G156	Broaching Canned Cycle	09
G167	Modify Setting	00
G174	CCW Non-Vertical Rigid Tap	00
G184	CW Non-Vertical Rigid Tap	00
G187	Setting the Smoothness Level	00
G234	Tool Center Point Control (TCPC)	08
G253	G253 Orient Spindle Normal To Feature Coordinate System	00
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	14
G269	Disable Feature Coordinate System	14

MILL M-CODES

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.

*** Click on **Description** in **BLUE** for more information ***

M - Code	Description
M00	Stop Program
M01	Optional Program Stop
M02	Program End
M03	Spindle Foward Command
M04	Spindle Reverse Command
M05	Spindle Stop Command
M06	Tool Change
M07	Shower Coolant On
M08 / M09	Coolant On / Off
M10 / M11	Engage / Release 4th Axis Brake
M12 / M13	Engage / Release 5th Axis Brake
M16	Tool Change
M19	Orient Spindle
M21-M25	Optional User M Function with M-Fin
M29	Set Output Relay with M-Fin
M30	Program End and Reset
M31	Chip Conveyor Forward
M33	Chip Conveyor Stop
M34	Coolant Increment
M35	Coolant Decrement
M36	Pallet Part Ready
M39	Rotate Tool Turret
M41 / M42	Low / High Gear Override
M46	Qn Pmm Jump to Line
M48	Validate That The Current Program is Appropriate for Loaded Pallet
M50	Pallet Change Sequence

M51-M55	Set Optional User M-codes
M59	Set Output Relay
M61-M65	Clear Optional User M-codes
M69	Clear Output Relay
M70/M71	Workholding Clamp / Unclamp
M73 / M74	Tool Air Blast (TAB) On / Off
M75	Set G35 or G136 Reference Point
M78	Alarm if Skip Signal Found
M79	Alarm if Skip Signal Not Found
M80 / M81	Auto Door Open / Close
M82	Tool Unclamp
M83 / M84	Auto Air Gun On / Off
M86	Tool Clamp
M88 / M89	Through-Spindle Coolant On / Off
M90 / M91	Fixture Clamp Input On / Off
M95	Sleep Mode
M96	Jump If No Input
M97	Local Sub-Program Call
M98	Sub-Program Call
M99	Sub-Program Return or Loop
M104 / M105	Probe Arm Extend/ Retract
M109	Interactive User Input
M116 / M117	Vise Air Chips Blast On/Off
M130 / M131	Display Media / Cancel Display Media
M138 / M139	Spindle Speed Variation On/Off
M158 / M159	Mist Condenser On/Off
M160	Cancel Active PulseJet
M161	PulseJet Continuous Mode
M162	PulseJet Single Event Mode
M163	PulseJet Modal Mode
M180 / M181	Auto Window Open / Close
M199	Pallet / Part Load or Program End
M300	M300 - APL/Robot Custom Sequence

MILL SETTINGS INTRODUCTION

This page gives detailed descriptions of the settings that control the way that your machine works.

List of Settings

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.

User Positions

This tab collects settings that control user-defined positions such as second home, tool change mid-positions, spindle center line, tailstock and travel limits.

Refer to the Settings section of this manual for more information about these position settings.

CAUTION: Incorrectly set user positions can cause machine crashes. Set user positions with caution, especially after you have changed your application in some way (new program, different tools, etc.). Verify and change each axis position separately.

To set a user position, jog the axis into the position you want to use, and then press F2 to set the position. If the axis position is valid, a crash warning appears (except for user travel limits). After you verify that you want to make the change to the position, the control sets the position and makes the setting active.
If the position is not valid, the message bar at the bottom of the screen gives a message to explain why the position is not valid.

To inactivate and reset user position settings, press ORIGIN while the user positions tab is active, then choose from the menu that appears.

- Press 1 to remove the value of the currently selected position setting and make it inactive.
- Press 2 to remove the values of all second home position settings and make them inactive.
- Press 3 to remove the values of all Tool Change Mid-Position settings and make them inactive.
- Press 4 to remove the values of all Max User Travel Limit settings and make them inactive.
- Press CANCEL to exit the menu without making changes.

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%
15	H and T Code Agreement
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
27	G76 / G77 Shift Dir.
28	Can Cycle Act w/o X/Y
29	G91 Non-modal
31	Reset Program Pointer
32	Coolant Override
33	Coordinate System
34	4th Axis Diameter
35	G60 Offset
36	Program Restart
39	Beep @ M00, M01, M02, M30
40	Tool Offset Measure
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
48	Mirror Image A 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-
61	Probe Offset Y+
62	Probe Offset Y-
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
76	Tool Release Lockout
77	Scale Integer F
79	5th-Axis Diameter
80	Mirror Image B Axis
81	Tool At Power Up
82	Language
83	M30/Resets Overrides
84	Tool Overload Action
85	Maximum Corner Rounding
86	M39 Lockout
87	Tool Change Resets Override
88	Reset Rests Override
90	Max Tools To Display
101	Feed Override -> Rapid
103	Cyc Start/Fh Same Key
104	Jog Handle to SNGL BLK
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
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
164	Rotary Increment

165	Ssv Variation (RPM)
166	Ssv Cycle
188	G51 X Scale
189	G51 Y Scale
190	G51 Z Scale
191	Default Smoothness
196	Conveyor Shutoff
197	Coolant Shutoff
199	Backlight Timer
216	Servo and Hydraulic Shutoff
238	High Intensity Light Timer (minutes)
239	Worklight Off Timer (minutes)
240	Tool Life Warning
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 Starup Screen
250	Mirror Image C Axis
251	Subprogram Search Location
252	Custom Subprogram Search Location
253	Default Graphics Tool Width
254	5-Axis Rotary Center Distance
255	MRZP X Offset
256	MRZP Y Offset
257	MRZP Z Offset
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 Postion X
269	Second Home Position Y
270	Second Home Position Z
271	Second Home Position A
272	Second Home Position B
273	Second Home Position C
276	Workholding Input Monitor
277	Lubrication Cycle Interval
291	Main Spindle Speed Limit
292	Door Open Spindle Speed Limit
293	Tool Change Mid Position X
294	Tool Change Mid Position Y
295	Tool Change Mid Position Z
296	Tool Change Mid Position A

297	Tool Change Mid Position B
298	Tool Change Mid Position C
300	MRZP X Offset Master
301	MRZP Y Offset Master
302	MRZP Z Offset Master
303	MRZP X Offset Slave
304	MRZP Y Offset Slave
305	MRZP Z Offset Slave
306	Minimum Chip Clear Time
310	Min User Travel Limit A
311	Min User Travel Limit B
312	Min User Travel Limit C
313	Max User Travel Limit X
314	Max User Travel Limit Y
315	Max User Travel Limit Z
316	Max User Travel Limit A
317	Max User Travel Limit B
318	Max User Travel Limit C
323	Disable Notch Filter
325	Manual Mode Enabled
330	MultiBoot Selection Time out
335	Linear Rapid Mode
356	Beeper Volume
357	Warm Up Cycle Start Idle Time
369	PulseJet Injection Cycle Time
370	PulseJet Single Squirt Count
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 Z
381	Enable Touchscreen
382	Disable Pallet Changer
383	Table Row Size
389	Vise Unclamped Safety Check
396	Enable / Disable Virtual Keyboard
397	Press and Hold Delay
398	Header Height
399	Header Tab
400	Pallet Ready Beep Type
403	Change Popup Button Size
408	Exclude Tool From Safe Zone
409	Default Coolant Pressure
416	Media Destination
420	ATC Button Behavior

421	General Orient Angle
422	Lock Graphics Plane
423	Help Text Icon Size
424	Mist Extractor Condenser Time Out
430	Enable - Broken Tool Detection
431	Region of Interest - Broken Tool Detection
433	Behavior - Broken Tool Detection