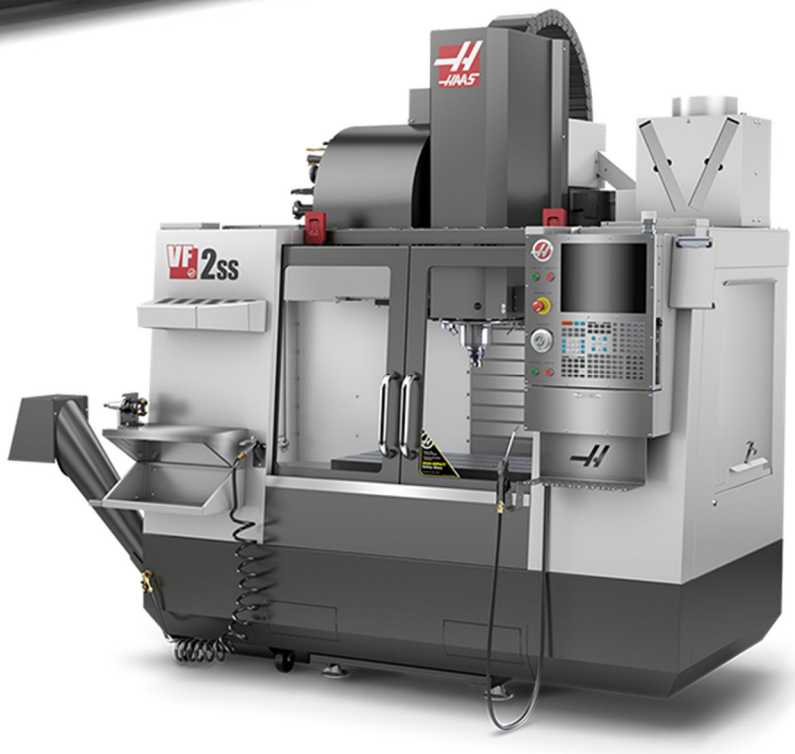
HAAS FACTORY OUTLET
A DIVISION OF PRODUCTIVITY INC

HAAS MILL PROGRAMMING



Revised 06/26 LL

PROGRAMMING MANUAL

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.

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

TABLE OF 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 practical method for increasing efficiency is to fixture multiple workpieces on the table and machine them within a single cycle. This is achieved by assigning each workpiece to its own Work Offset.

Refer to the G-code section of this manual for additional details. The following example demonstrates multi-part machining using an M97 local subprogram call during the cutting sequence.

```
%
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 ;
%
```

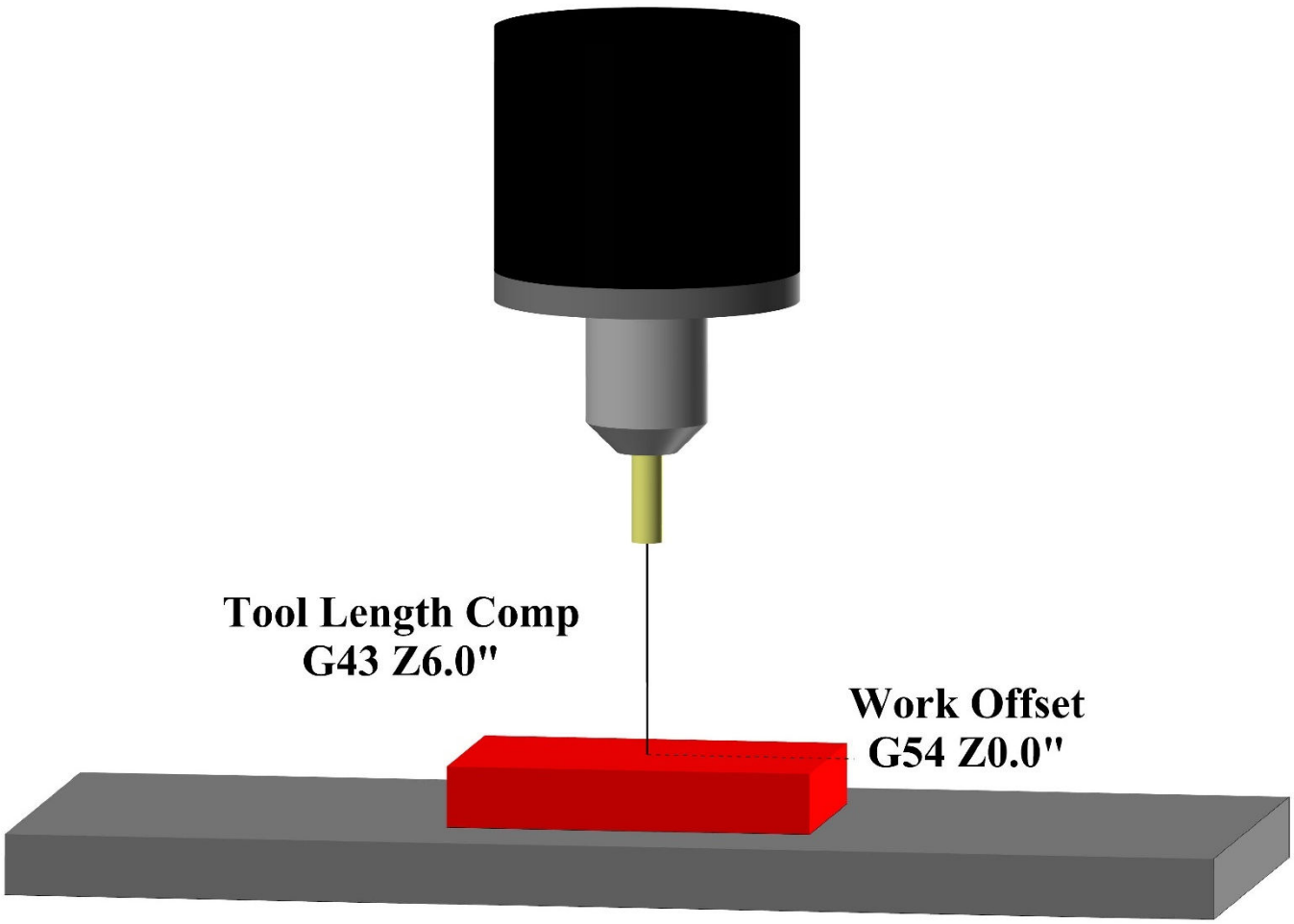
G43 TOOL LENGTH COMPENSATION + (ADD) / G44 TOOL LENGTH COMP - (SUBTRACT) (GROUP 08)

A G43 command applies positive tool length compensation by adding the tool length value from the offsets page to the commanded axis position. A G44 command applies negative tool length compensation by subtracting the tool length value. A non-zero H value must be specified to reference the correct tool offset entry.

NOTE Starting with NGC Software Version 100.21.000.1100, HAAS machines apply tool length offsets by default. The offset remains active unless explicitly canceled with G49/H00 on mills or a Txx00 offset on lathes. On mill machines, the tool length offset automatically updates after each tool change, and both the active tool length offset, and the associated Group 8 code persist through power cycles.

CAUTION: The tool length **nn** value must match the **nn** value specified in the **M06 Tnn** tool change command to prevent potential collisions.

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 behaves differently depending on the value of Setting 33, which selects either Fanuc-style or Haas-style coordinates.

Fanuc Mode (Setting 33 = FANUC)
G52 is a Group 00 G-code used to apply a global work coordinate shift. The values entered on the G52 line are added to all work offsets. These G52 values are automatically cleared at power-up, when reset is pressed, when changing modes, at the end of a program, or by issuing M30, G92, or G52 X0 Y0 Z0 A0 B0.

Haas Mode (Setting 33 = HAAS)
G52 also functions as a global work coordinate shift, and the entered values are added to all work offsets. Unlike Fanuc mode, G52 values are cleared only by a G92 command.
When G92 is used in Haas format, the current position in the active work coordinate system is shifted by the programmed X, Y, Z, A, and B values, and the G92 work offset reflects the difference between the current work offset and the commanded shift.

G53 NON-MODAL MACHINE COORDINATE SELECTION (GROUP 00)

G53 temporarily cancels all work coordinate offsets and moves the machine using the machine coordinate system. It also **ignores any active tool length offsets**. In machine coordinates, the zero point for each axis corresponds to the position reached during a Zero Return. G53 applies only to the block in which it is commanded.

G92 SET WORK COORDINATE SYSTEMS SHIFT VALUE (GROUP 00)

A G92 command cancels any active G52 shift for the specified axes. For example, **G92 X1.4** clears the G52 shift on the X-axis only.

The current G92 shift value appears at the bottom of the Work Offsets page and can be cleared from that display if needed. It is also cleared automatically at power-up and whenever **Zero Return → All** or **Zero Return → Single** is performed.

To clear a G92 shift within a program, issue another G92 command that returns the work offset to its original value.

ABSOLUTE VS. INCREMENTAL POSITIONING (G90, G91)

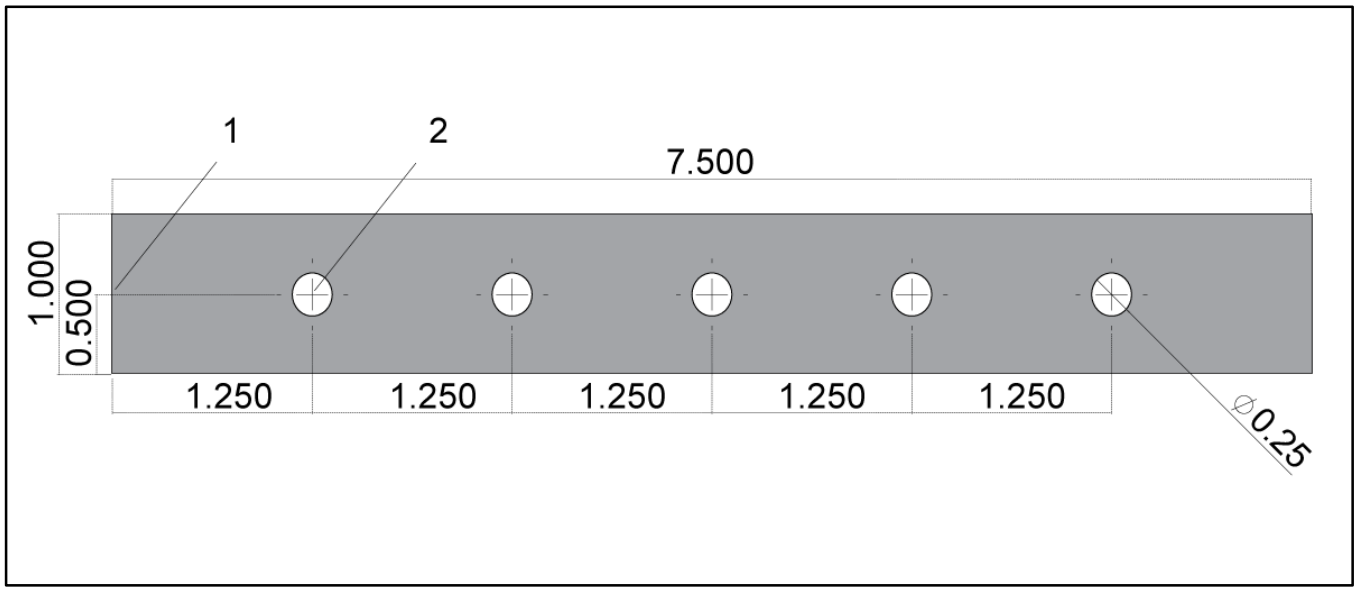
Absolute (**G90**) and incremental (**G91**) positioning determine how the control interprets axis motion commands.

With **G90**, commanded positions are interpreted relative to the origin of the active coordinate system.

With **G91**, commanded positions are interpreted relative to the tool's current position.

Absolute programming is appropriate for most operations, while incremental programming is effective for repetitive, equally spaced movements.

Figure 1 illustrates a part with five equally spaced Ø0.25" (13 mm) holes, each 1.00" (25.4 mm) deep and spaced 1.250" (31.75 mm) apart.



Absolute and incremental sample programs are shown below. **G54 X0. Y0.** is used for the incremental example [1], and **G54** is used for the absolute example [2].

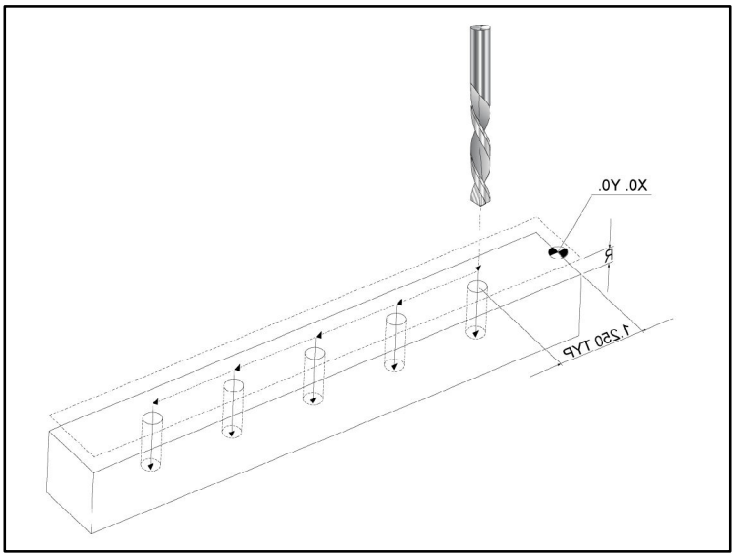
The two programs demonstrate how to drill the holes shown in the drawing, allowing comparison between absolute and incremental positioning.

The holes are first started with a center drill, followed by drilling with a 0.250" (6.35 mm) drill bit. A depth cut of 0.200" (5.08 mm) is used for the center drill, and 1.00" (25.4 mm) for the 0.250" drill.

The **G81** drill canned cycle is used for all 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) ;
%
```



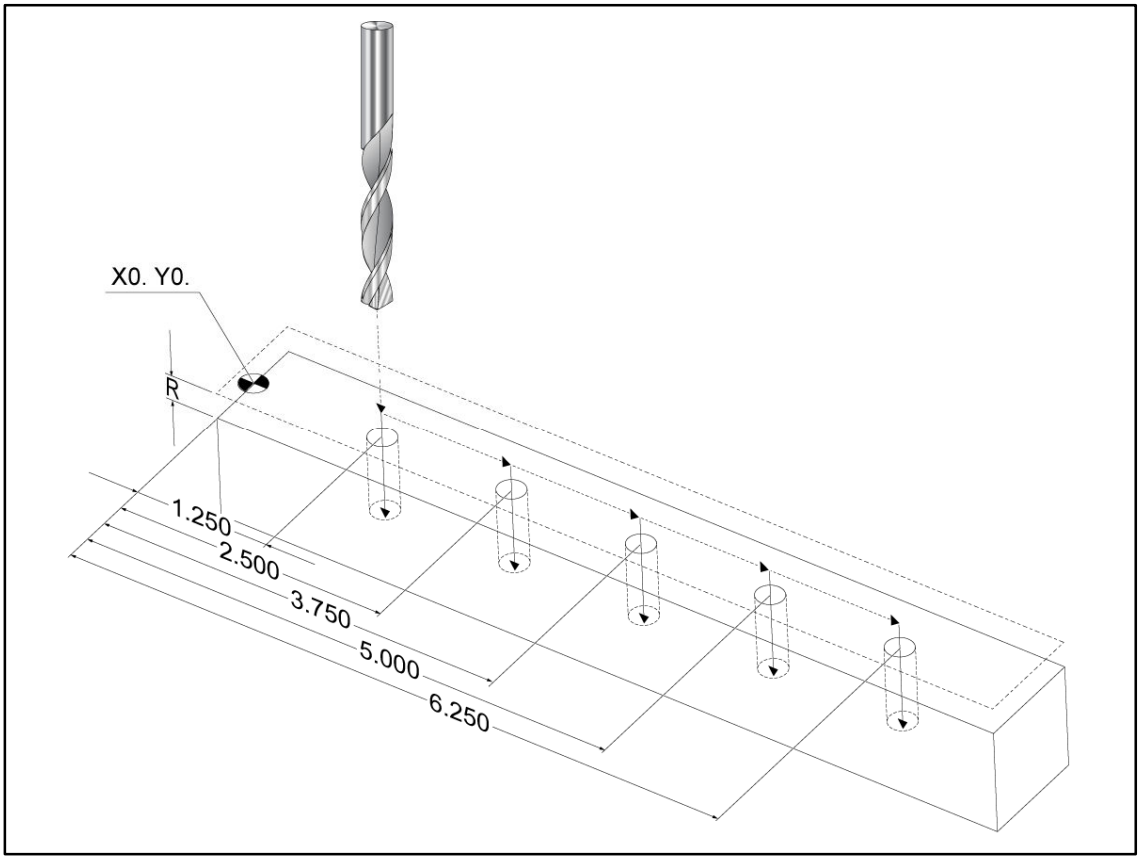
MILL ABSOLUTE POSITIONING EXAMPLE

The absolute program method requires more lines of code than the incremental method, although both use similar preparation and completion sections.

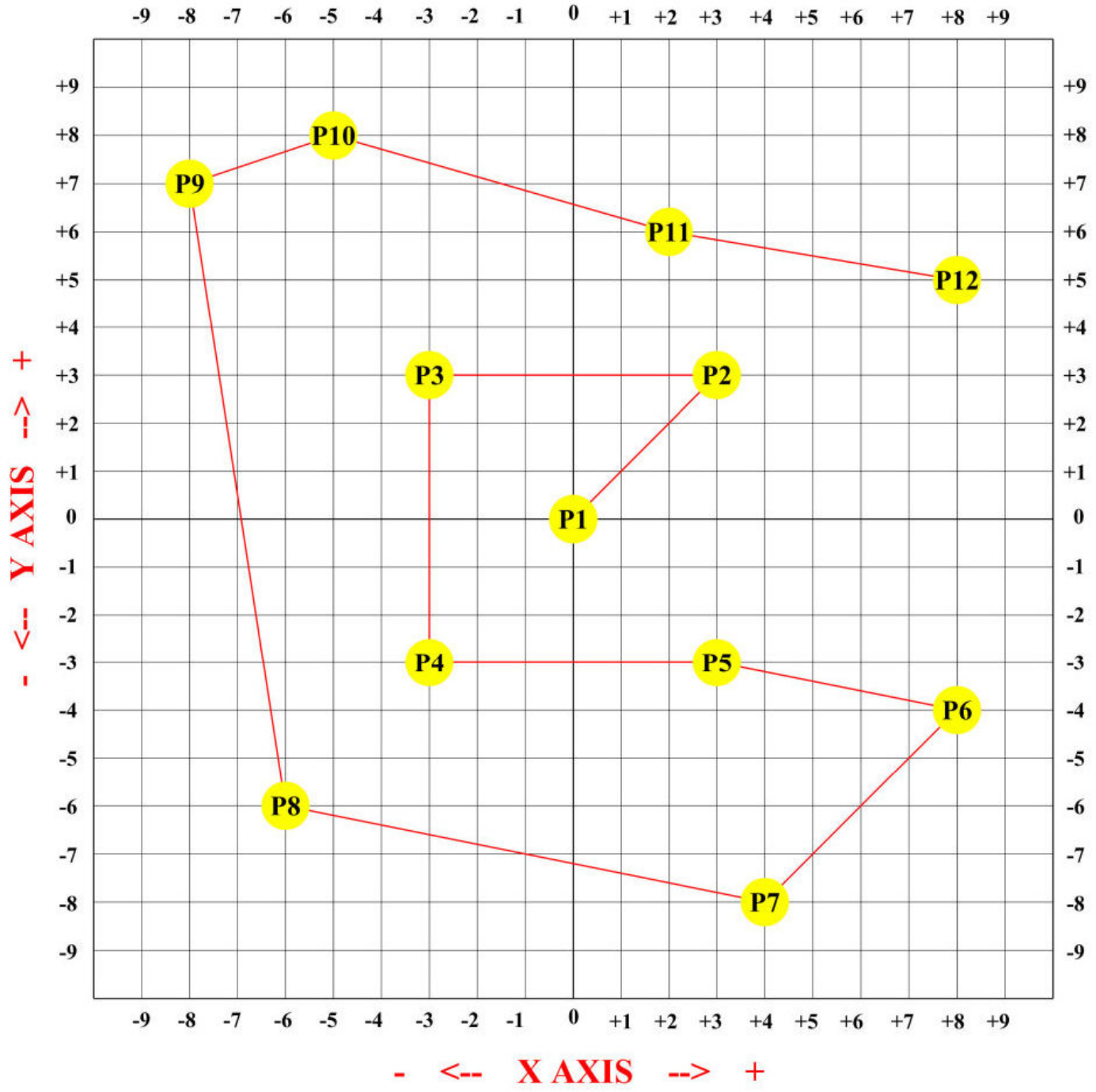
In the incremental example, the center-drill operation begins at line N13. The **G81** canned cycle uses the loop address **Lnn** to repeat the cycle a specified number of times. In this case, **L5** repeats the cycle five times. With each repetition, the machine moves the programmed X and Y increment—in this example, **1.25" in X**—from the current position before performing the drill cycle. The programmed drill depth is **0.1" deeper** than the actual depth to account for starting **0.1" above the part surface**.

In the absolute program, **G81** specifies the drill depth but does not use an L-code loop. Instead, each hole location is listed on a separate line, and the control performs the drill cycle at each position until **G80** cancels the canned cycle. The absolute method specifies the exact drill depth because motion begins 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



Tool path example above starts from Point 1 and ends on Point 12. P1 through P6 are in ABSOLUTE (G90) and P6 through P12 are INCREMENTAL (G91).

Absolute (G90) From X0 and Y0			Incremental (G91) Starting from P6		
	X	Y		X	Y
P1			P7		
P2			P8		
P3			P9		
P4			P10		
P5			P11		
P6			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)

G01 is used to cut straight-line toolpaths. It requires a programmed feedrate (**F**). Optional axis words—**X**, **Y**, **Z**, and **A**—specify the target position. Once commanded, G01 remains active and continues to use the last programmed feedrate until another motion mode is selected, such as **G00**, **G02**, **G03**, **G12**, or **G13**.

Corners may be modified by using **C** for chamfers or **R** for corner rounding. Refer to *G01 Linear Interpolation Motion (Group 01)* for additional details.

Circular Interpolation Motion (G02/G03)

G02 (clockwise) and **G03** (counterclockwise) command circular cutting moves. Several optional address codes can be used to define the arc geometry. The motion begins at the tool's current position and moves along the programmed arc to the specified endpoint.

Arcs may be defined using either of two methods:

1. **I, J, K Method (Preferred)**
    - **I**, **J**, and **K** define the relative distances from the start point to the arc center.
    - **X**, **Y**, and **Z** specify the absolute endpoint of the arc.
    - This is the only method that allows programming of a full 360° circle; omitting the endpoint and specifying only I, J, or K produces a complete circle.
2. **R Method (Radius)**
    - The **R** value defines the radius from the start point to the center of the arc.
    - The endpoint must still be defined with X, Y, or Z.
    - Positive **R** values produce arcs ≤ 180°, while negative **R** values are used for arcs > 180°.

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

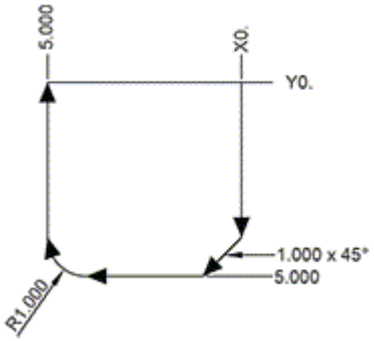
G01 *moves the axes at a commanded feedrate and is primarily used for cutting. A G01 move may involve a single axis or multiple axes simultaneously. The feedrate (**F**) can be specified in units per minute (**G94**), per spindle revolution (**G95**), or as inverse-time (**G93**). The control uses the most recently commanded **F** value until a new one is programmed; in **G93**, an **F** value must be specified on every line.*

*G01 is a modal command and remains active until canceled by another motion mode such as **G00**, **G02**, or **G03**.*

When G01 is active, all programmed axes move and arrive at the endpoint simultaneously. If any axis cannot achieve the commanded feedrate, the machine will stop the move and generate a "maximum feedrate exceeded" alarm.

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 or corner-rounding move can be automatically inserted between two linear interpolation blocks by adding **,C** (chamfer) or **,R** (corner rounding) to the beginning block. A terminating linear interpolation block must follow, although a **G04** dwell may appear between these blocks.

The two linear blocks define the intersection corner. When **,C** is used, the value specifies the distance from the intersection to both the start and end of the chamfer. When **,R** is used, the value specifies the radius of a circle tangent to the corner at the start and end of the rounding arc. Consecutive chamfer or corner-rounding blocks are permitted. These moves require motion in the two axes of the currently selected plane—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 specify circular motion. Circular interpolation requires movement in two axes and must use the correct plane (**G17–G19**). There are two methods for programming a **G02** or **G03** move: using the **I, J, K** addresses or using the **R** address.

Using I, J, K Addresses
I, **J**, and **K** define the distances from the starting point to the arc center. Only the addresses relevant to the selected plane are allowed—**I/J** for **G17**, **I/K** for **G18**, and **J/K** for **G19**. The **X**, **Y**, and **Z** commands specify the arc endpoint; if an axis endpoint is omitted, that axis remains at its starting value.

To cut a full circle, the **I, J, K** method must be used; an **R** value cannot produce a complete circle. A full circle is programmed by specifying only the I/J/K center values without an endpoint.

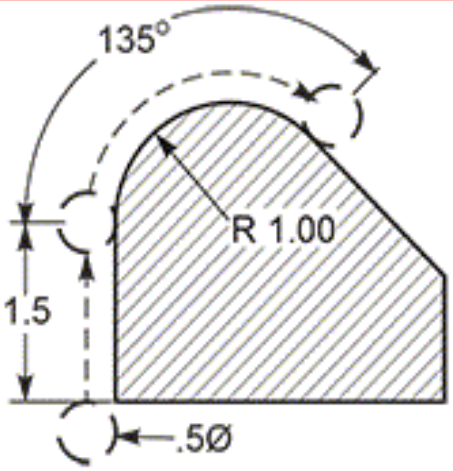
For example:
G02 I3.0 J4.0 (Assumes G17; XY plane)

Using the R Address

The **R** value specifies the radius of the arc by defining the distance from the starting point to the center of the circle. A **positive R** value is used for arcs of **180° or less**, while a **negative R** value is used for arcs **greater than 180°**.

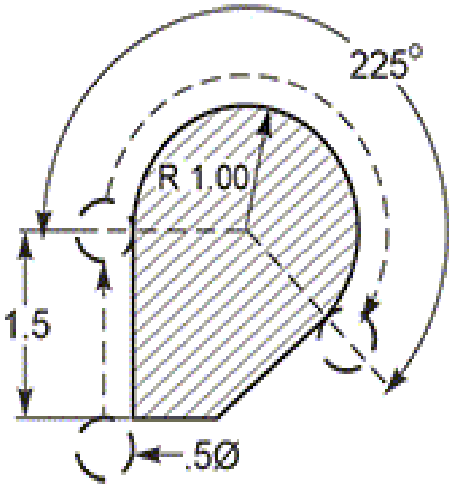
PROGRAMMING EXAMPLE

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 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 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%
```

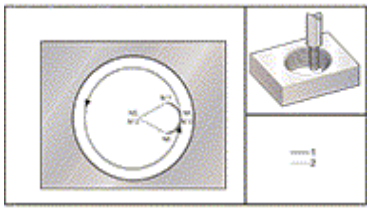
THREAD MILLING

Thread milling uses a standard **G02** or **G03** move to generate the circular motion in the X-Y plane, while simultaneously adding a Z-axis move on the same block to produce the thread pitch. This helical move generates one turn of the thread; the remaining threads are formed by the multiple teeth of the cutter. A typical example is:

N100 G02 I-1.0 Z-.05 F5. (Generates 1-inch radius for a 20-pitch thread)
```[1](L83)

**Thread Milling Notes**

- Internal holes smaller than **3/8 inch** may not be practical.
- Always climb cut.
- Use **G03** for internal (I.D.) threads and **G02** for external (O.D.) threads.
- A right-hand I.D. thread moves **up** in Z by one pitch; a right-hand O.D. thread moves **down** in Z by one pitch.
- Pitch is calculated as:
    - PITCH = 1 / TPI** (e.g., 1 ÷ 8 TPI = 0.125").[1](L85)

This example thread-mills a **1.5" diameter × 8 TPI** internal thread using a **0.750" diameter × 1.0"** thread hob. [1](L88)

**Thread Milling Steps**

1. **Calculate the start position.**
   Subtract the cutter diameter from the hole diameter and divide by two:
   (1.500 − 0.750) / 2 = **0.375**.
   This value is the offset distance from the hole center to begin cutting.[1](L89)

2. **Turn on cutter compensation and move to the I.D.**
   After initial positioning, enable cutter compensation and position the cutter on the thread diameter.[1](L91)

3. **Program the helical move.**
   Execute a full circular move (**G02** or **G03**) while commanding a Z-axis move equal to one thread pitch. This creates the helical interpolation required for the thread form.
   [1](L92)

4. **Exit the thread and cancel cutter compensation.**
   Move off the diameter using a linear move and then cancel compensation.[1](L93)

**Important Rule**

Cutter compensation cannot be turned on or off during an arc move. A linear X or Y move must be programmed to transition onto or off the compensated diameter. This linear move defines the maximum allowable adjustment.[1](L94)
```

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 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%
```

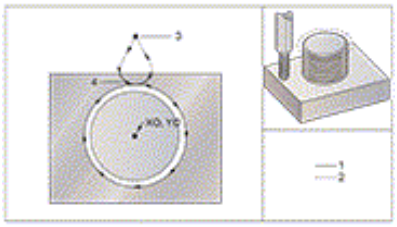
Specific line description:

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 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 thread-mills a 1.0" diameter hole using a 0.500" cutter and a 0.125" (8 TPI) thread pitch. It begins in absolute positioning (**G90**) and switches to incremental positioning (**G91**) at 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.

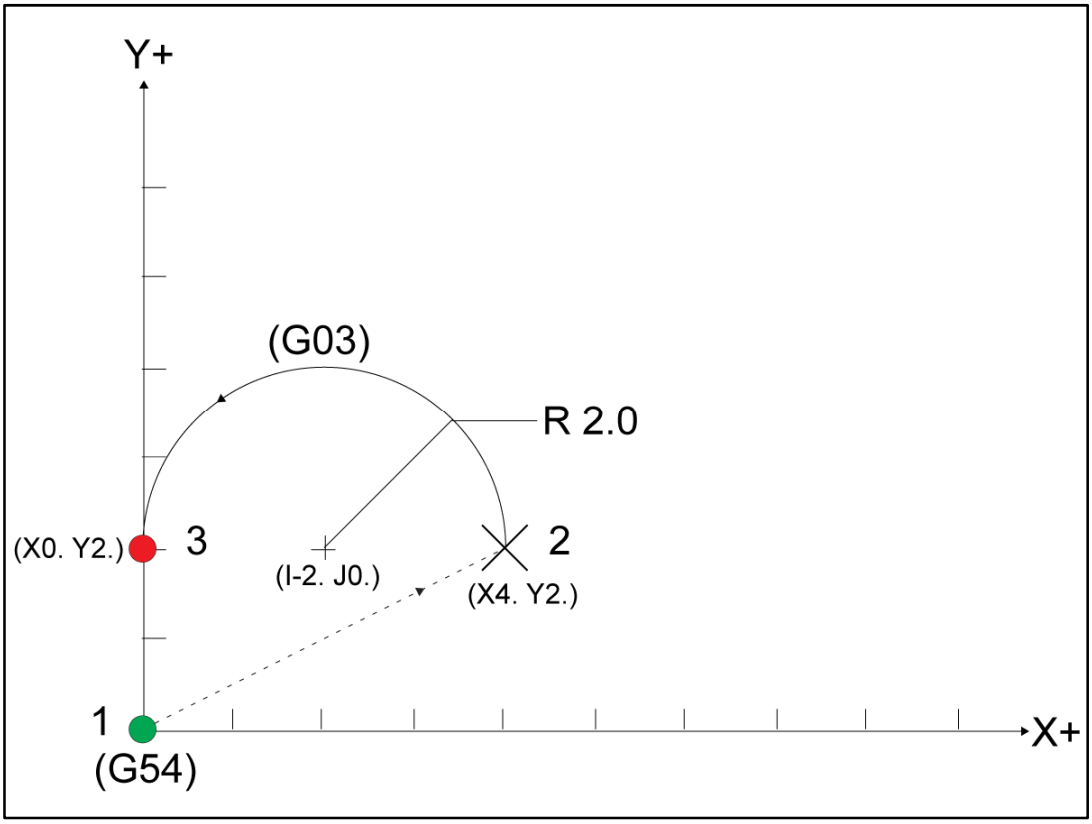
```
%
O60025 (G03 SNGL PNT THREAD MILL 1.5-8
UNC) ;
(G54 X0 Y0 is at the center of the bore) ;
(Z0 is on top of the part) ;
(T1 is a .5 in dia thread mill) ;
(BEGIN PREPARATION BLOCKS) ;
T1 M06 (Select tool 1) ;
G00 G90 G40 G49 G54 (Safe startup) ;
G00 G54 X0 Y0 (Rapid to 1st position) ;
S1000 M03 (Spindle 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 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 demonstrating the two methods for programming a 2" (or 2 mm) radius, 180-degree counterclockwise arc. The tool begins at **X0 Y0** [1], moves to the arc's starting point [2], and then cuts the arc to the endpoint [3].



Method 1:

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

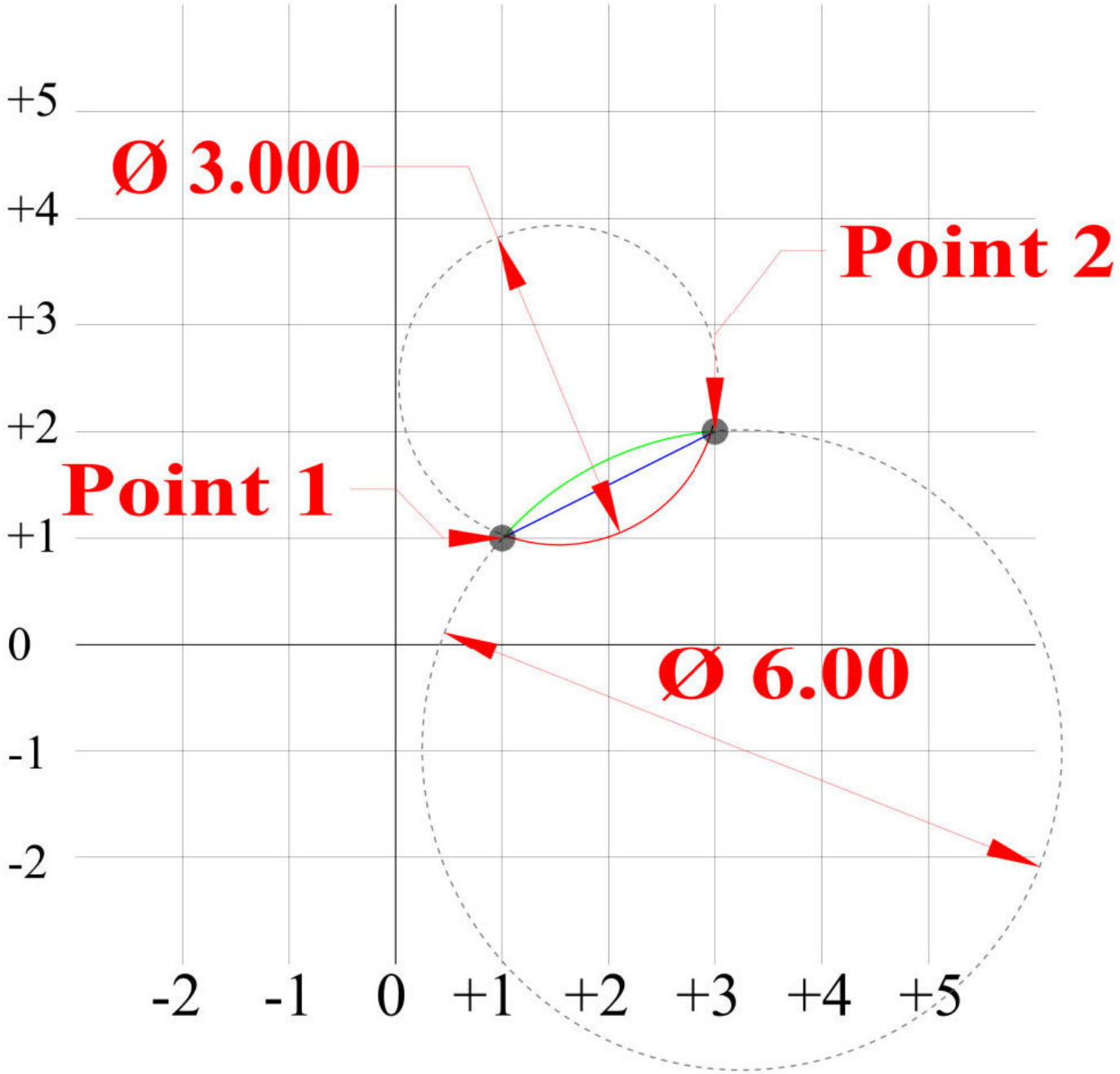
Method 2:

```
%
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 ;
%
```

G01, G02 AND G03 MOTION

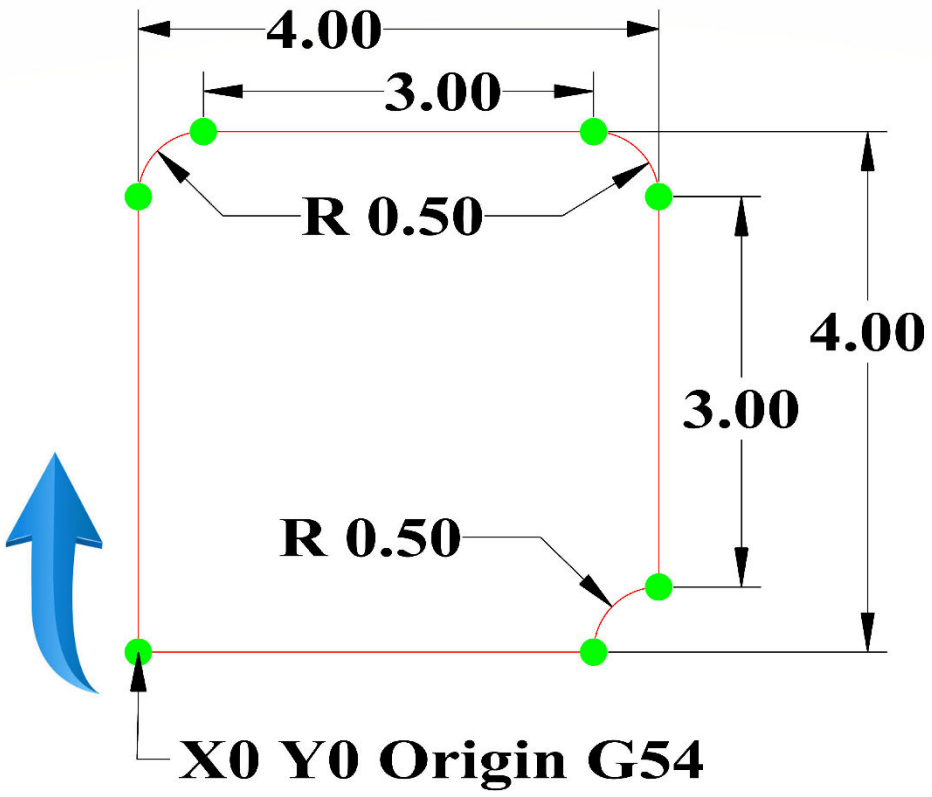


All Feed moves below have the same start position, X1.0 Y1.0, and the same end position, X3.0 Y2.0.

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



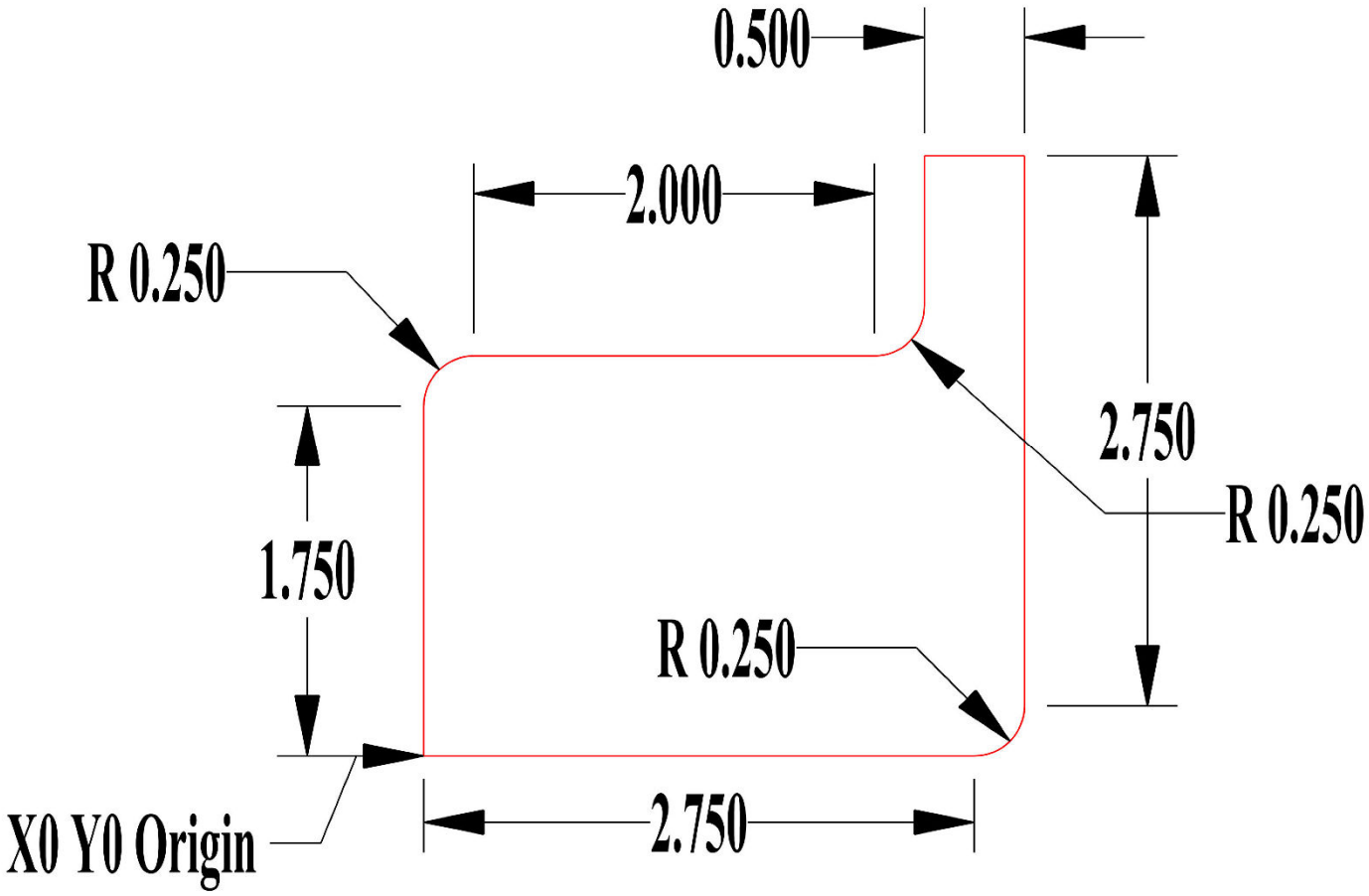
All exercises in this manual use climb milling. Program all toolpaths clockwise around the outside of the part profile.

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

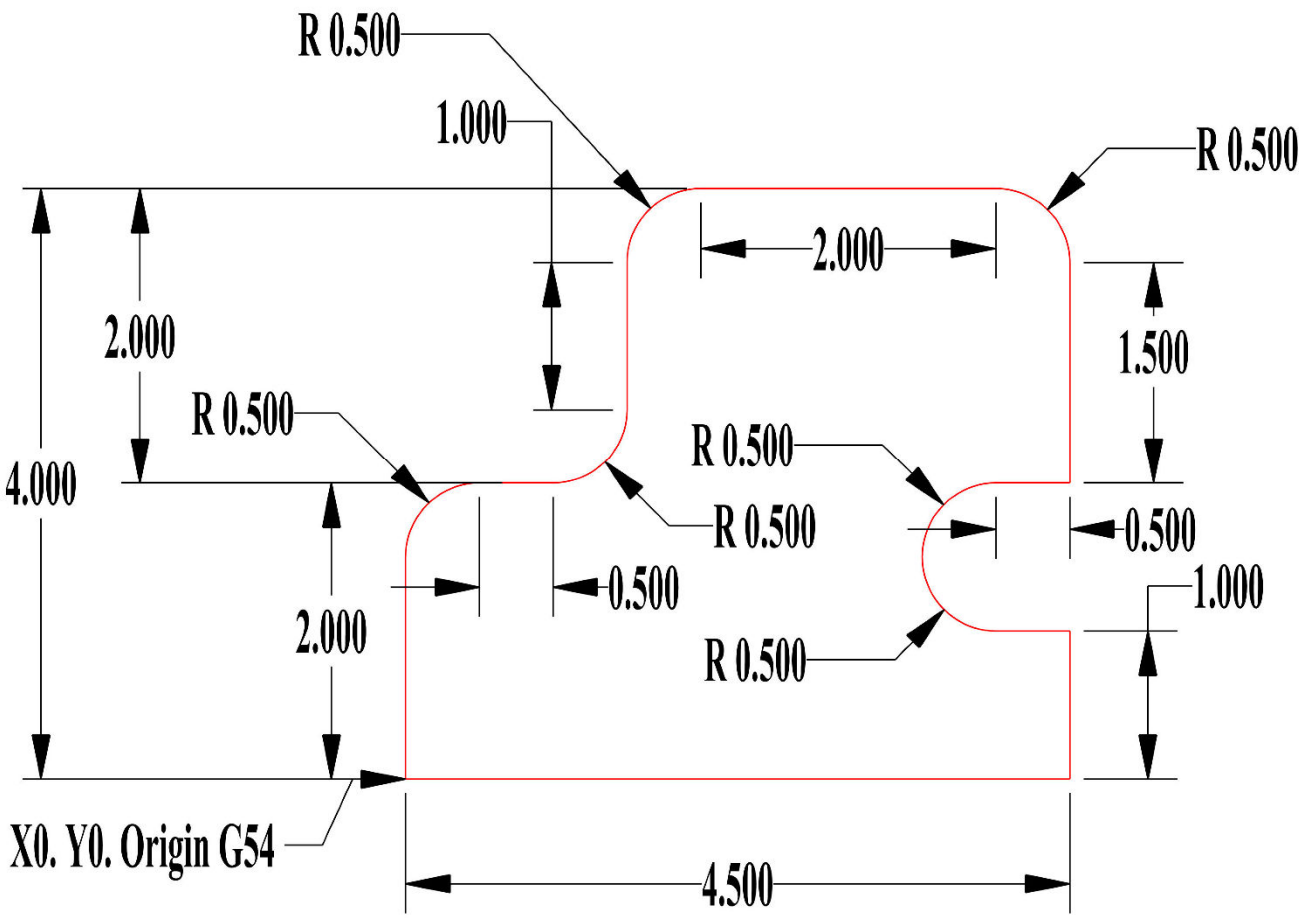


Use G90, Absolute with G02 and G03
HAAS Simulator: Go to MDI Mode:

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



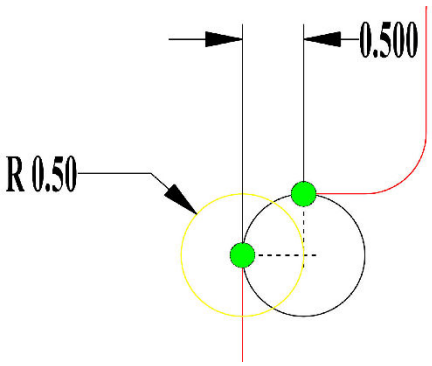
Use G02 and G03 with I and J ONLY

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.

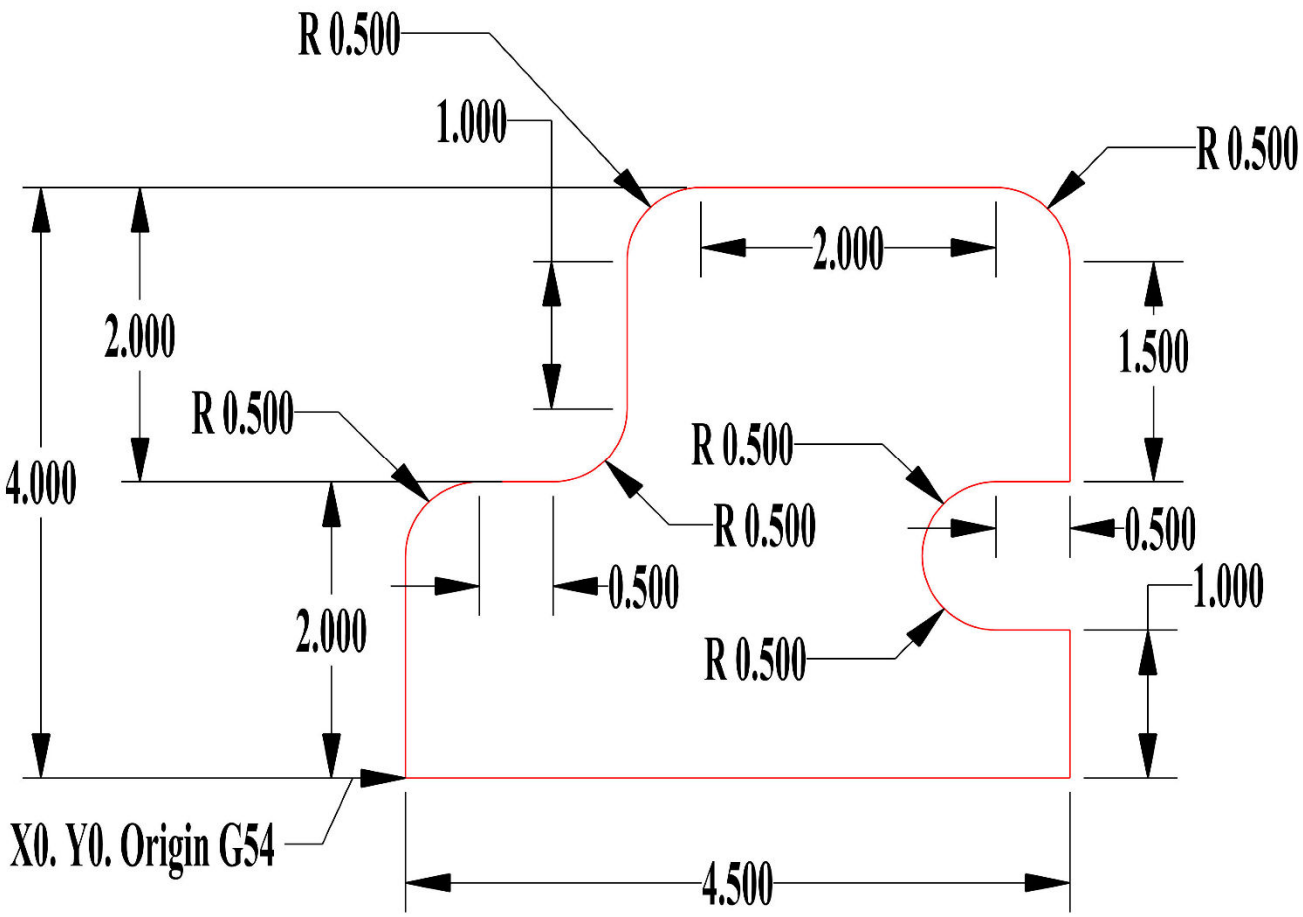
Example for 1st Radius tool path:



Distance shifted in I or J Determines the size of the Arc / Circle

G02 X0.5 Y2.0 I0.5 ; (Arc Center Shifted 0.5" along X +)

TOOL PATH EXERCISE 4



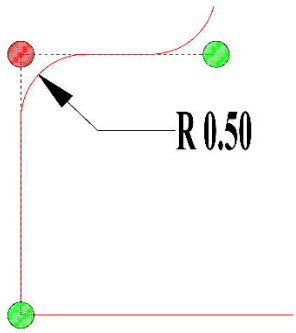
Use only G01 with ,R#.#

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.

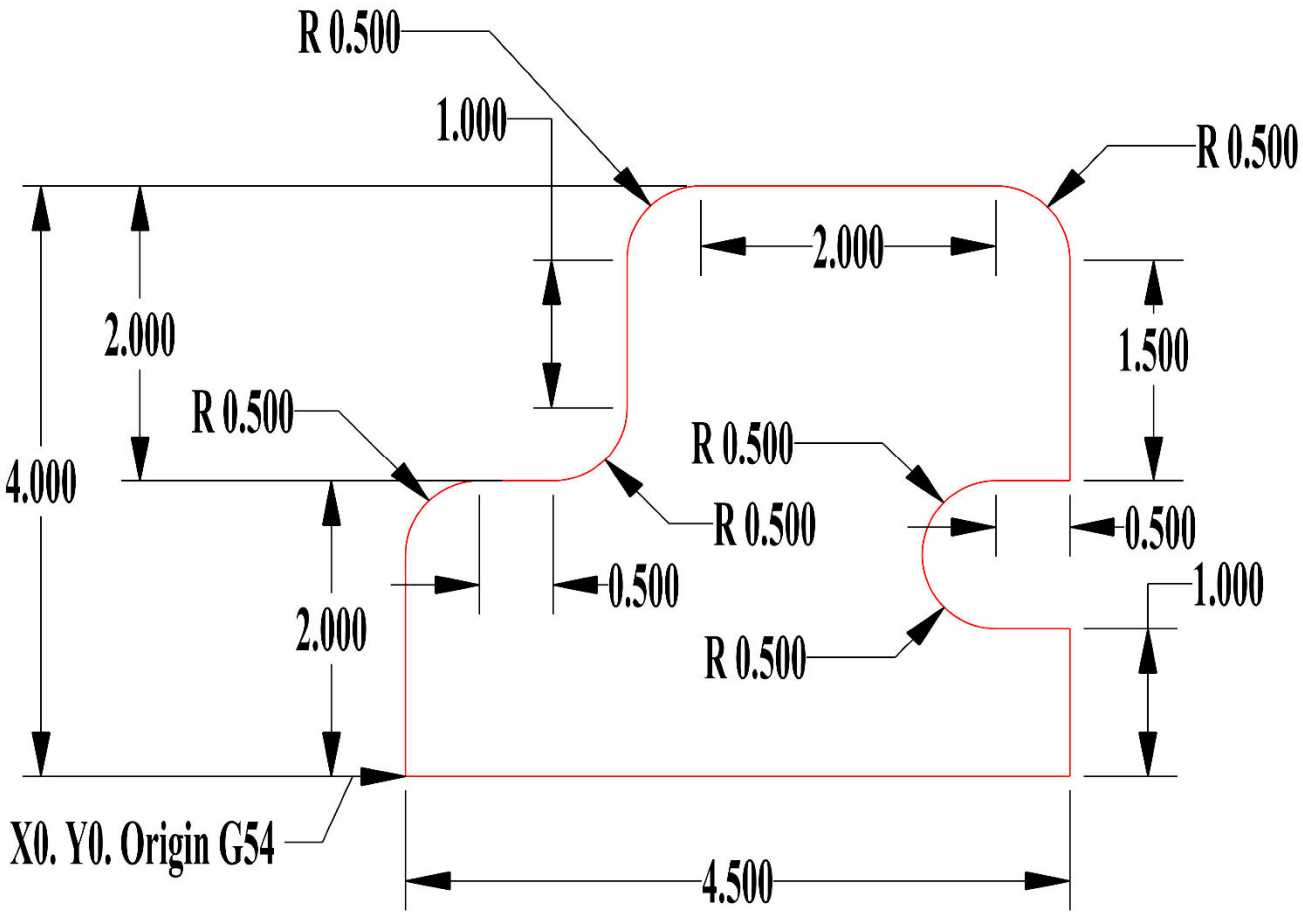
Example for 1st Radius tool path:



0.5" radius at the end of linear move

G01 Y2.0 ,R0.5 ;
X2.0 ;

TOOL PATH EXERCISE 5

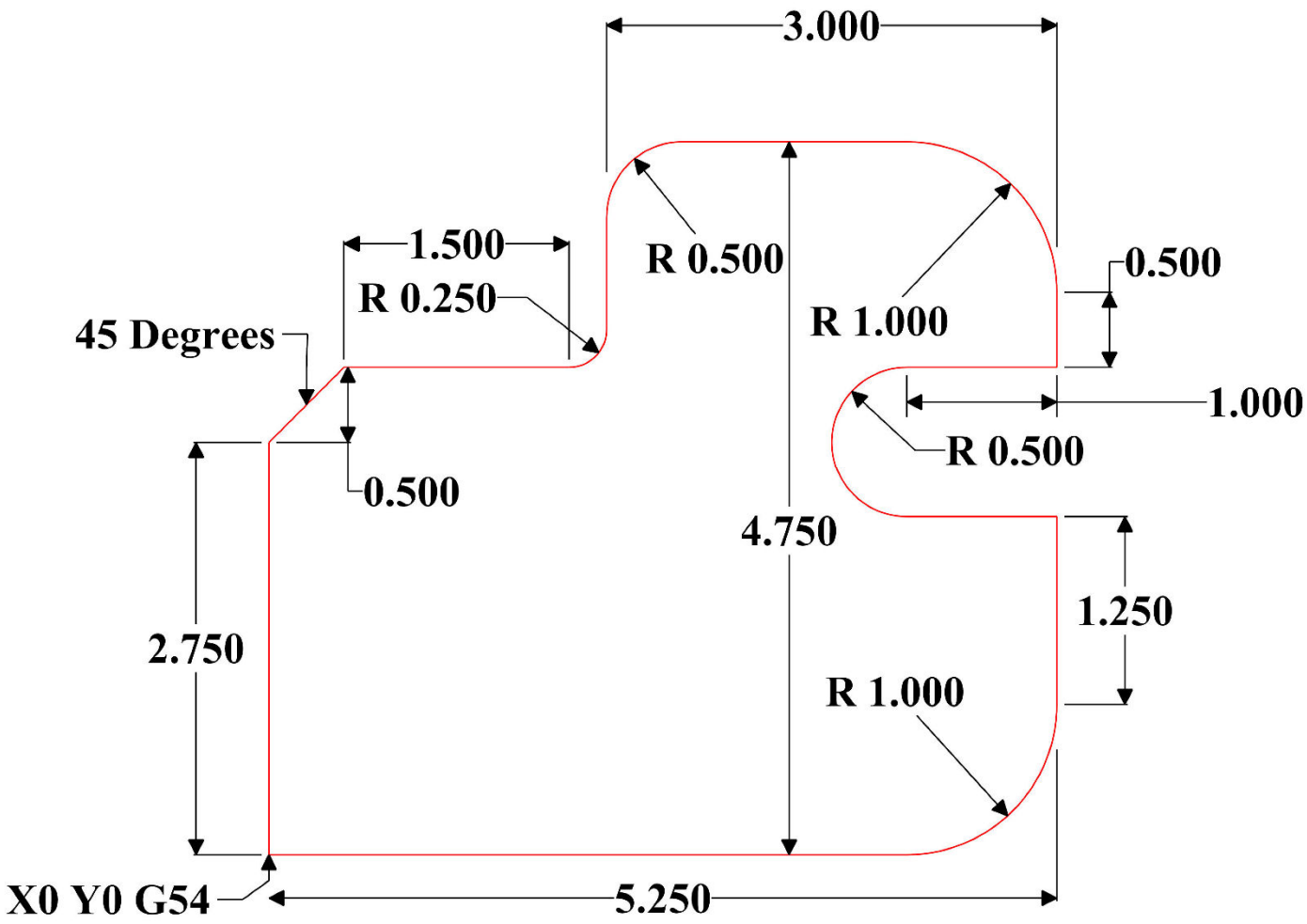


Use G91, Incremental with G01, R#.#, G02 or G03
HAAS Simulator: Go to MDI Mode:

T1 M06
G54 G90
G00 X0. Y0. **(Must rapid to Absolute starting origin)**
G91
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:

Preparation: Defines the work and tool offsets, loads the required cutting tool, sets spindle speed, enables coolant, and selects absolute or incremental positioning for axis motion.

Cutting: Specifies the toolpath and feedrate for the machining operation.

Completion: Retracts the spindle, turns off the spindle and coolant, and moves the table to a safe location for part unloading and inspection.

This basic program makes a 0.100" (2.54 mm) deep cut with Tool 1 along a straight-line path from X = 0.0, Y = 0.0 to X = –4.0, Y = –4.0.

NOTE: A program block may contain multiple G-codes, provided they belong to different groups. Two G-codes from the same group cannot appear in the same block. Only one M-code is permitted per block.

```
%
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) ;
G00 G90 G17 G40 G49 G54 (Safe startup) ;
T1 M06 (Select tool 1) ;
G43 H01 (Tool offset 1 on) ;
X0 Y0 (Rapid to 1st position) ;
Z2.0 (Rapid to 2.0 Inch Above the part)
S1000 M03 (Spindle on CW) ;
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 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.
O40001 (Basic program) ;	O40001 is the program name. Program names follow the Onnnnn format, where the letter "O" is followed by a five-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
G00 G90 G17 G40 G49 G54 (Safe startup) ;	This line is referred to as a **safe startup block**, and it is good machining practice to include it after every tool change. **G00** sets subsequent axis motion to rapid mode. **G90** selects absolute positioning. **G17** selects the XY cutting plane, **G40** cancels cutter compensation, **G49** cancels tool length compensation, and **G54** activates the work coordinate system defined in the G54 offset.
T1 M06 (Select tool 1)	Selects tool T1 to be used. M06 commands the tool changer to load Tool 1 (T1) into the spindle.
G43 H01 (Tool offset 1 on)	**G43 H01** activates positive tool length compensation. The **H01** value instructs the control to apply the tool length stored for Tool 1 in the Tool Offset table. **NOTE:** Z Axis will move to position if there's any Z Axis command.
X0 Y0 (Rapid to 1st position)	X0 Y0 commands the table to move to X = 0.0 and Y = 0.0 using Work Offset **G54**.
Z2.0	Rapid Z Axis to 2.0" above the part using both **Tool and Work Offset combined**.
S1000 M03 (Spindle on CW)	**M03** turns the spindle on in a clockwise direction. The **Snnnn** address specifies the desired spindle RPM. On machines equipped with a gearbox, the control automatically selects high or low gear based on the commanded spindle speed. **M41** and **M42** may be used to override this behavior when needed.
M08 (Coolant on)	M08 turns the coolant on.

CUTTING

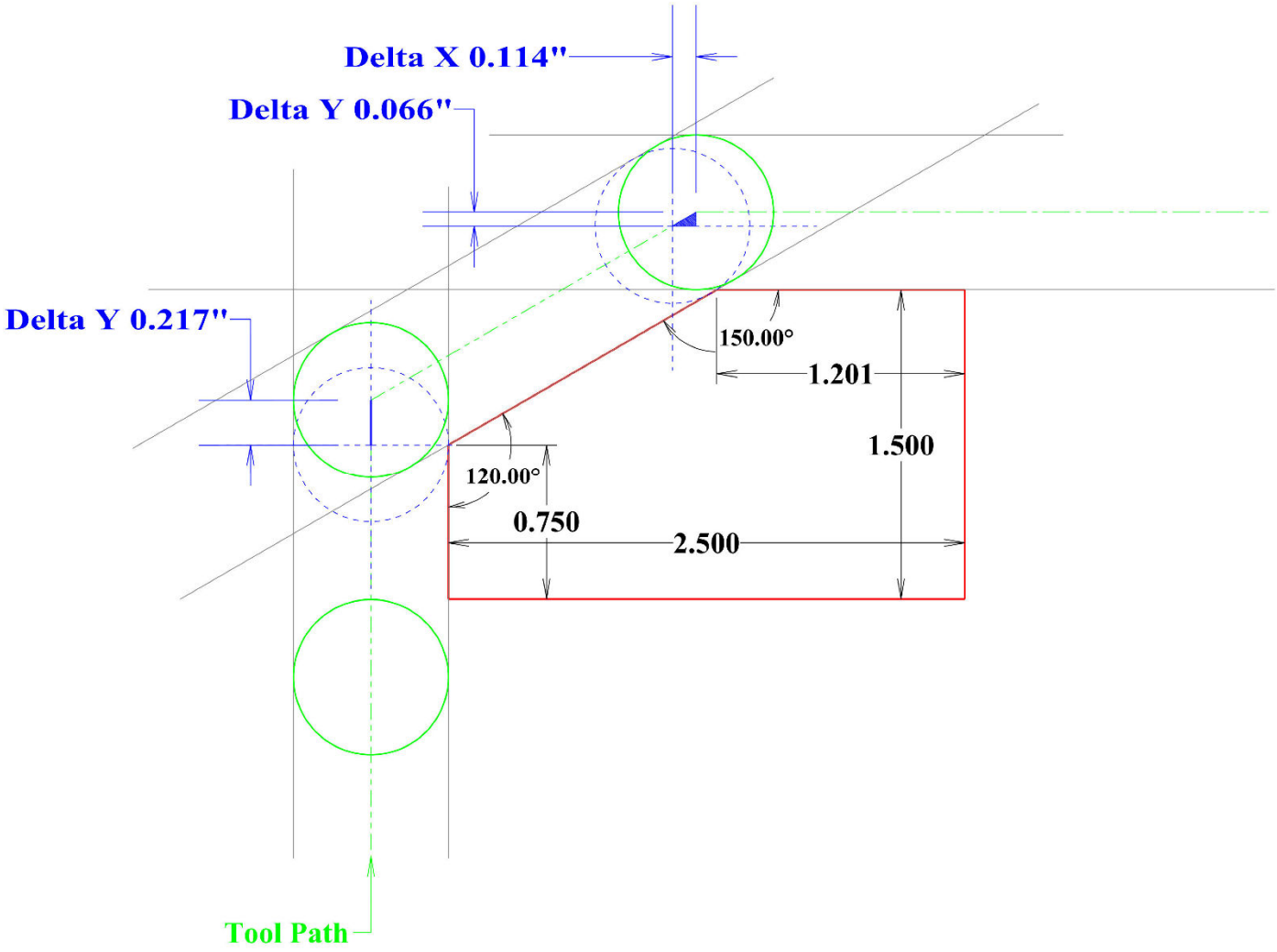
These are the cutting code blocks in the sample program O40001:

Cutting Code Block	Description
G01 F20. Z-0.1 (Feed to cutting depth) ;	**G01 F20.** sets linear interpolation at a feedrate of 20" (508 mm) per minute. **Z-0.1** commands the Z-axis to move 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 the Y-axis to move to Y = –4.0.

COMPLETION

Completion Code Block	Description
G00 Z0.1 M09 (Rapid retract, Coolant off) ;	**G00** commands rapid-motion positioning, **Z0.1** moves the Z-axis to **Z = 0.1**, and **M09** turns the coolant off.
G53 Z0 M05 (Z home, Spindle off) ;	**G53** commands the subsequent axis motion to be executed in the machine coordinate system. The **Z0** command moves the **Z-axis** to the machine home position at **Z = 0.0**, and **M05** turns the spindle off.
G53 Y0 (Y home) ;	**G53** commands the subsequent axis motion to be executed in the machine coordinate system. The **Y0** command moves the **Y-axis** to the machine home position at **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.

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 offsets the toolpath so the tool centerline runs to the left or right of the programmed path, allowing the control to adjust feature size using the wear values entered on the offsets page.

The offset may be specified as either a D**iameter** or a R**adius**, depending on **Setting 40**. In diameter mode, the control applies half of the entered value as the actual offset. Total compensation is the sum of the geometry and wear values.

Cutter compensation is available in X and Y for 2D machining (G17), and in X, Y, and Z for 3D machining with G141.

GENERAL DESCRIPTION OF CUTTER COMPENSATION

G41 (Climb Milling) selects cutter compensation left, shifting the tool to the left of the programmed path.

G42 (Conventional Milling) selects cutter compensation right, shifting the tool to the right of the programmed path.

A G41 or G42 command must include a **Dnnn** value to select the correct offset from the tool offset table. The D-number corresponds to the entry in the far-left column, and the control applies the value found in the GEOMETRY column—either **D** (diameter mode) or **R** (radius mode), depending on **Setting 40**.

If a negative value is entered for the offset, cutter compensation behaves as though the opposite G-code were commanded. For example, a negative value used with G41 will act like a positive value used with G42. When cutter compensation is active, circular motions are allowed only in the **X-Y plane (G17)**; cutter compensation is limited to 2D motion in that plane.

G40 cancels cutter compensation and is the default state at power-on. When cutter compensation is not active, the tool centerline follows the programmed path. A program cannot end (M30, M00, M01, or M02) with cutter compensation active.

The control evaluates cutter compensation one block at a time but looks ahead to the next two blocks containing X or Y motion to check for interference. **Setting 58** determines how this behavior is applied, with available options **Fanuc** or **YASNAC**.

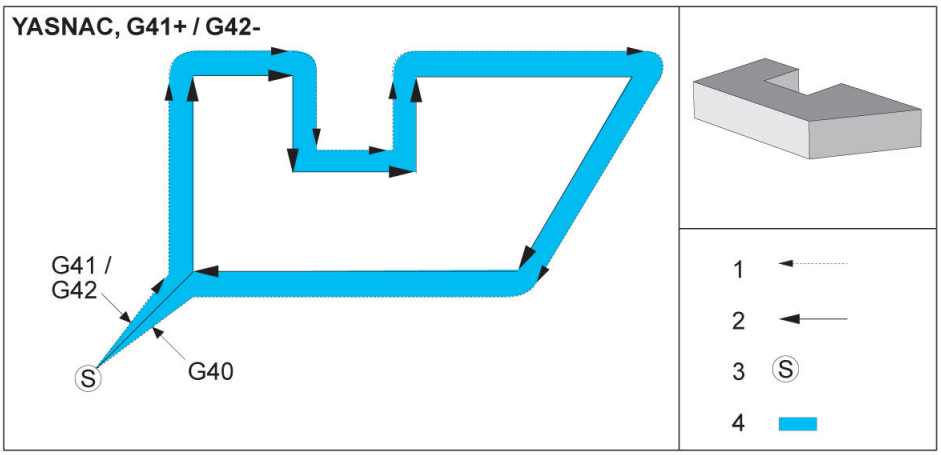
If **Setting 58** is set to **YASNAC**, the control must be able to place the side of the tool along the programmed contour without overcutting the following two motions. In this mode, a circular motion is inserted to connect all outside angles.

When **Setting 58** is configured to *Fanuc*, the control does not require the tool's cutting edge to remain in contact with all segments of the programmed contour, reducing the likelihood of overcutting. However, if the programmed path would cause an overcut, the control will stop and issue an alarm. For outside angles of 270° or less, the control produces a sharp corner; for angles greater than 270°, it automatically inserts an additional linear move.

The diagrams illustrate how cutter compensation behaves under the different Setting 58 modes. Notably, a short linear move—smaller than the tool radius and perpendicular to the previous motion—is permitted only when using the Fanuc option.

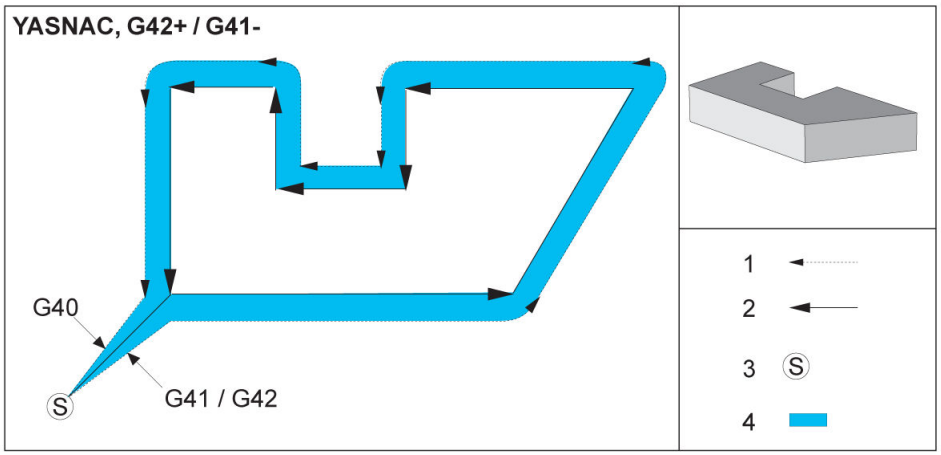
Cutter Compensation, YASNAC Style, G41 with a Positive Tool Diameter or G42 with a Negative Tool Diameter:

[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.



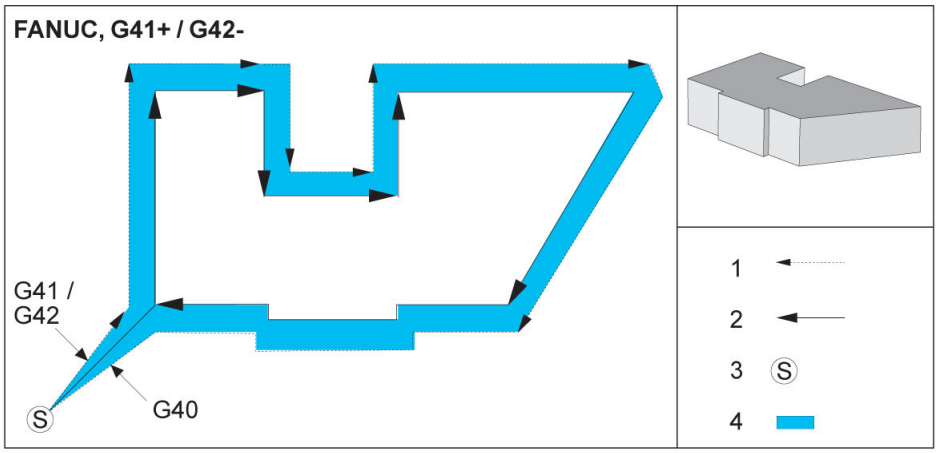
Cutter Compensation, YASNAC Style, G42 with a Positive Tool Diameter or G41 with a Negative Tool Diameter:

[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.



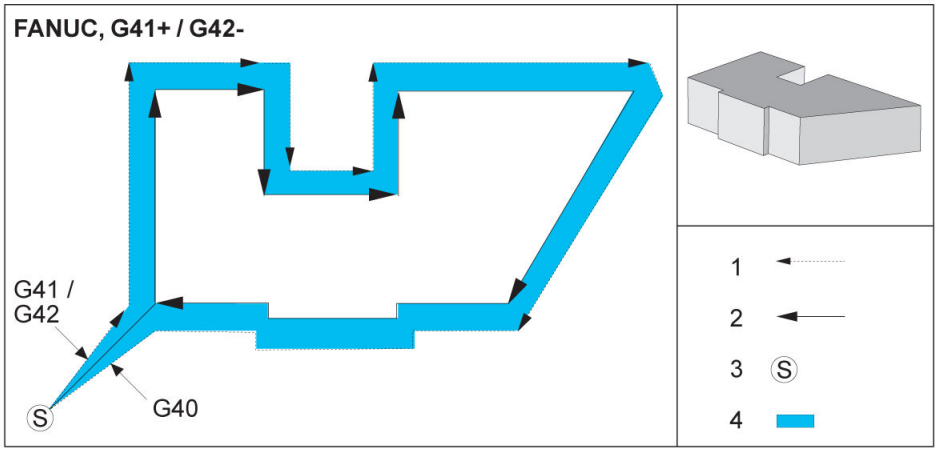
Cutter Compensation, FANUC Style, G41 with a Positive Tool Diameter or G42 with a Negative Tool Diameter:

[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.



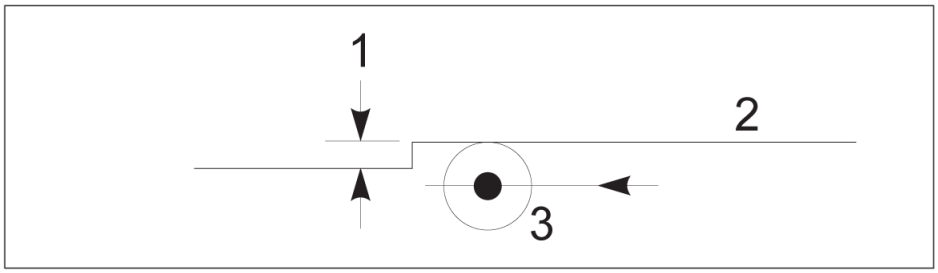
Cutter Compensation, FANUC Style, G42 with a Positive Tool Diameter or G41 with a Negative Tool Diameter:

[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.

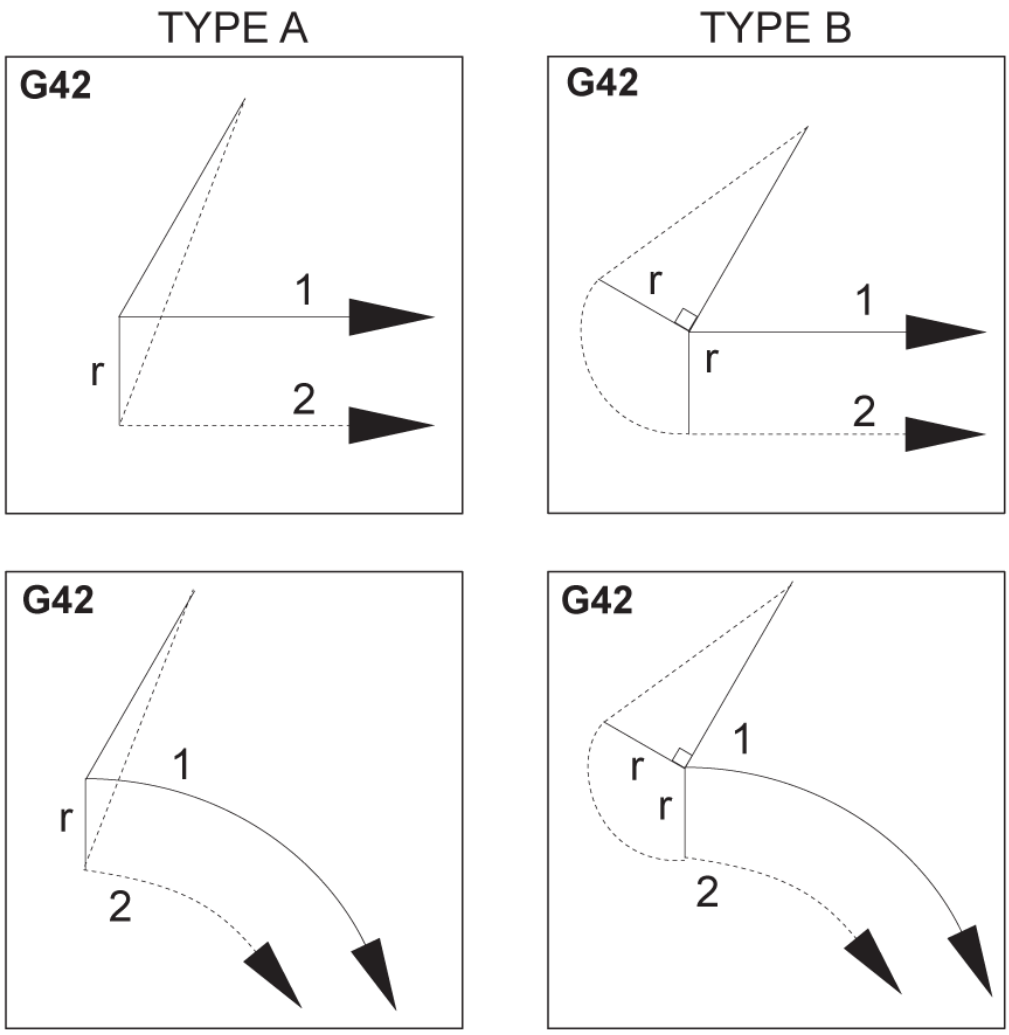
FEED ADJUSTMENTS IN CUTTER COMPENSATION

When cutter compensation is active during circular interpolation, the control may adjust the programmed feedrate. For inside arcs, the feedrate is reduced to prevent surface speed from exceeding the intended cutting conditions. Excessive reduction can impact performance, so Setting 44 limits the allowable slowdown from 1% to 100%. At 100%, no feed reduction is applied; at 1%, the feedrate may be reduced to as little as 1% of the programmed value.

For outside arcs, the control does not increase the feedrate—no speed-up adjustment is applied.

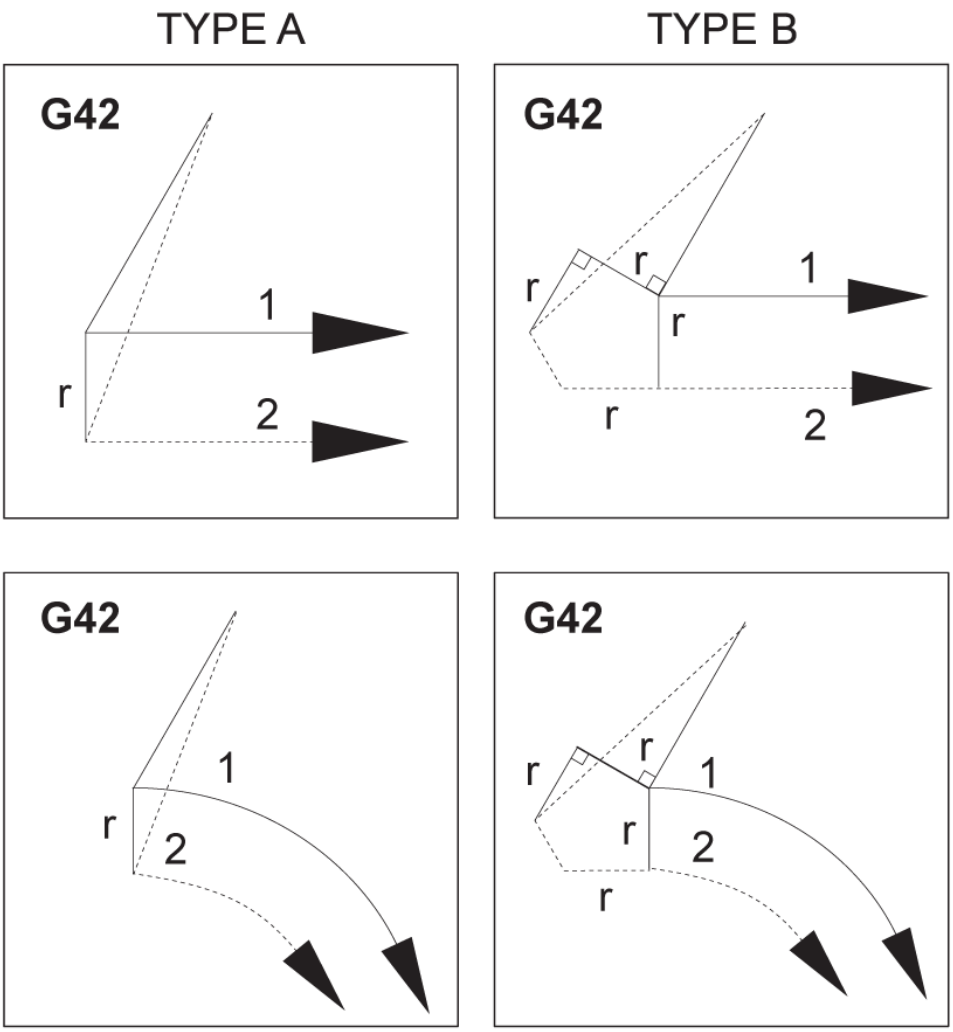
Cutter Compensation Entry (YASNAC) Type A and B:

[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

This section describes the use of **G02** (clockwise circular interpolation), **G03** (counterclockwise circular interpolation), and cutter compensation (**G41** left, **G42** right).

Using **G02** and **G03**, the machine can cut circular moves and radii. For most profiles or contours, the simplest way to program a radius between two points is by using an **R** value. For full 360° circular moves, an **I** or **J** value must be specified. The accompanying circle-section illustration identifies the various arc segments.

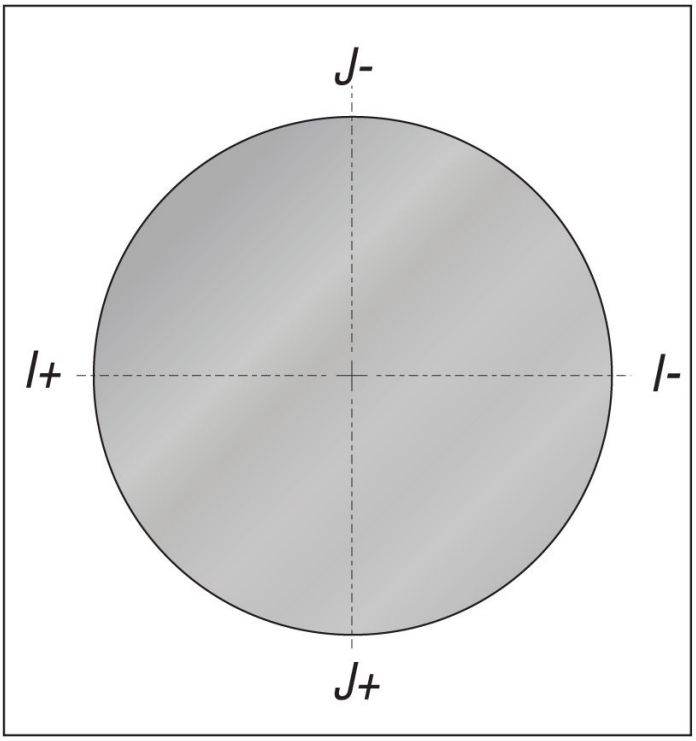
Cutter compensation allows the programmer to shift the toolpath by an exact amount so the machine can cut profiles or contours to the print's true dimensions. By using cutter compensation, real dimensions can be programmed directly, reducing calculation time and the risk of programming errors while providing precise control of part size and geometry.

Here are several important rules you must follow for successful cutter compensation. Always refer to these guidelines when writing programs.

1. **Turn cutter compensation ON with a G01 X,Y move that is equal to or greater than the cutter radius (or the compensation amount).**
2. **When the operation is complete, turn cutter compensation OFF using the same method required to turn it on.** The entry and exit moves must follow the same rules—what is put in must be taken out.
3. **On most machines, a linear X,Y move smaller than the cutter radius may not work while compensation is active.** Set **Setting 58 to Fanuc** for best results in these cases.
4. **Cutter compensation cannot be turned ON or OFF during a G02 or G03 arc move.**
5. **With cutter compensation active, machining an inside arc with a radius smaller than the active D-value will generate an alarm.** A tool that is too large cannot cut an inside radius that is too small.

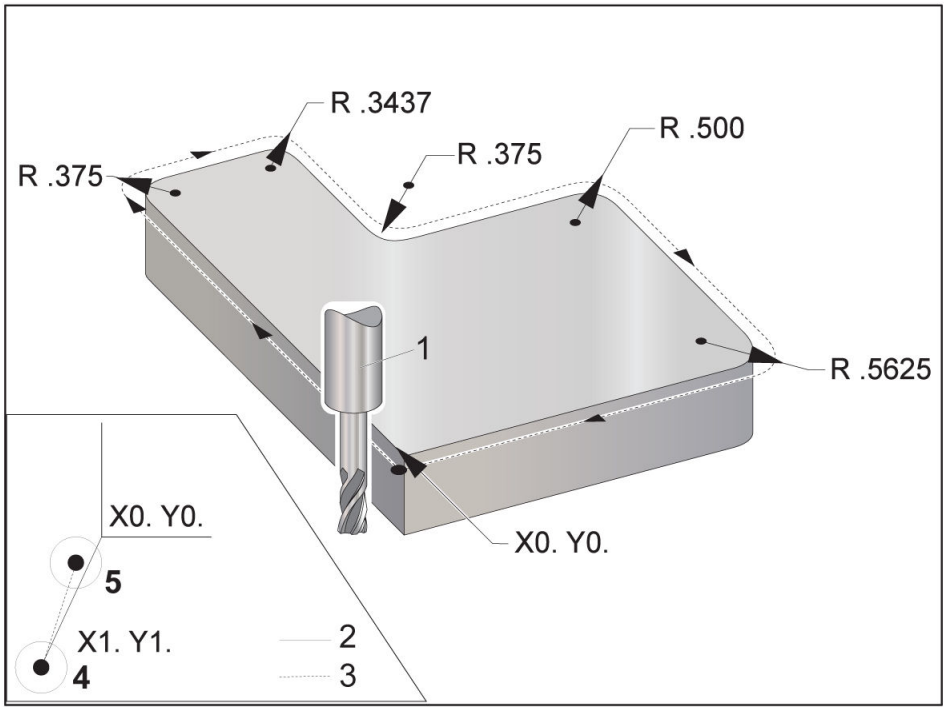
This illustration shows how the toolpath is calculated when cutter compensation is applied.

The detail view displays the tool in its starting position and then in the offset position as it engages 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

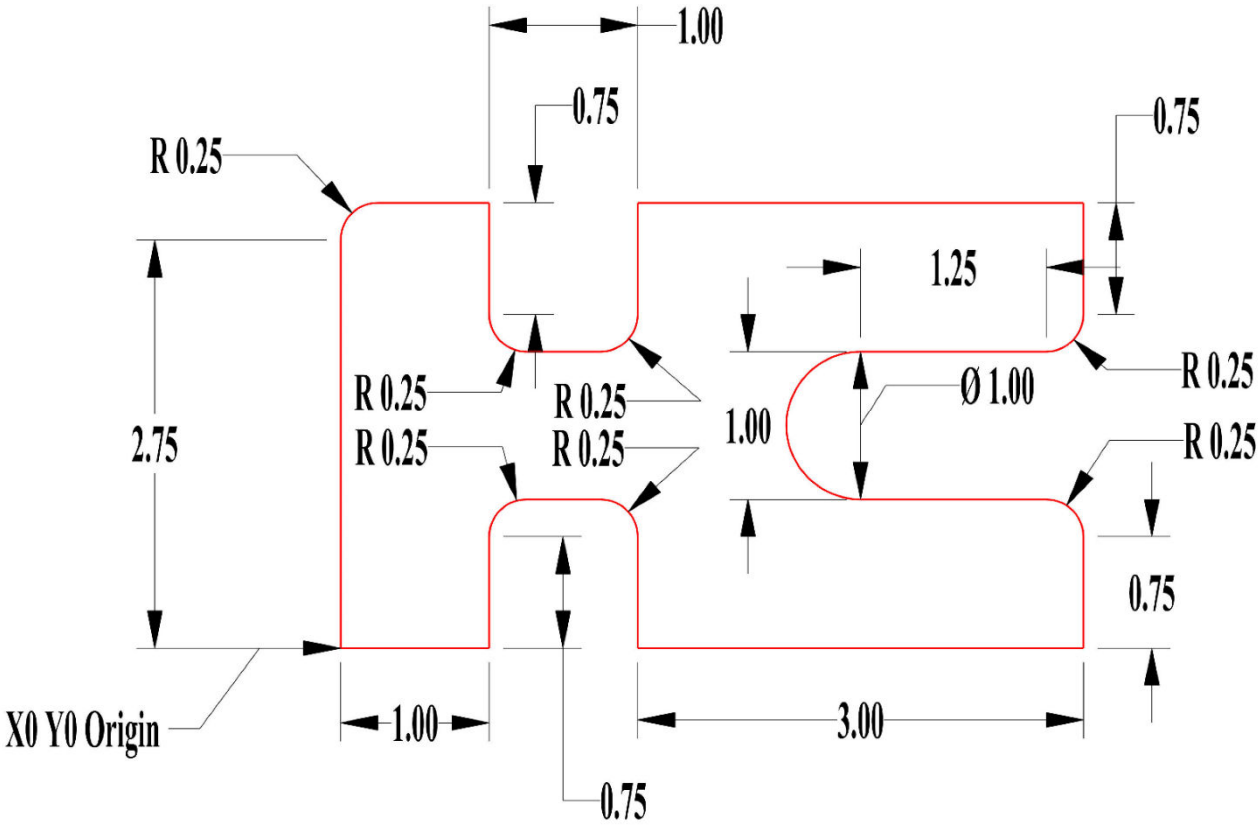
Material Group		Side Milling Ap	Side Milling Ae	Slotting Ap	KCPM15A - KCSM15A Cutting Speed Vc SFM Min	SFM Max	Fraction dec.	Recommended Feed per Tooth (IPT=Inch/th) is for Side Milling . For Slotting Reduce Fz by 20% D1 - Diameter 1/8	5/32	3/16	7/32	1/4	9/32	5/16	3/8	1/2	5/8	3/4	1	1 1/4
								0.1250	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° Max Depth	KCPM15A - KCSM15A Cutting Speed Vc SFM Min	SFM Max	Min - Max Diameter for Helical Interpolation dec	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
						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

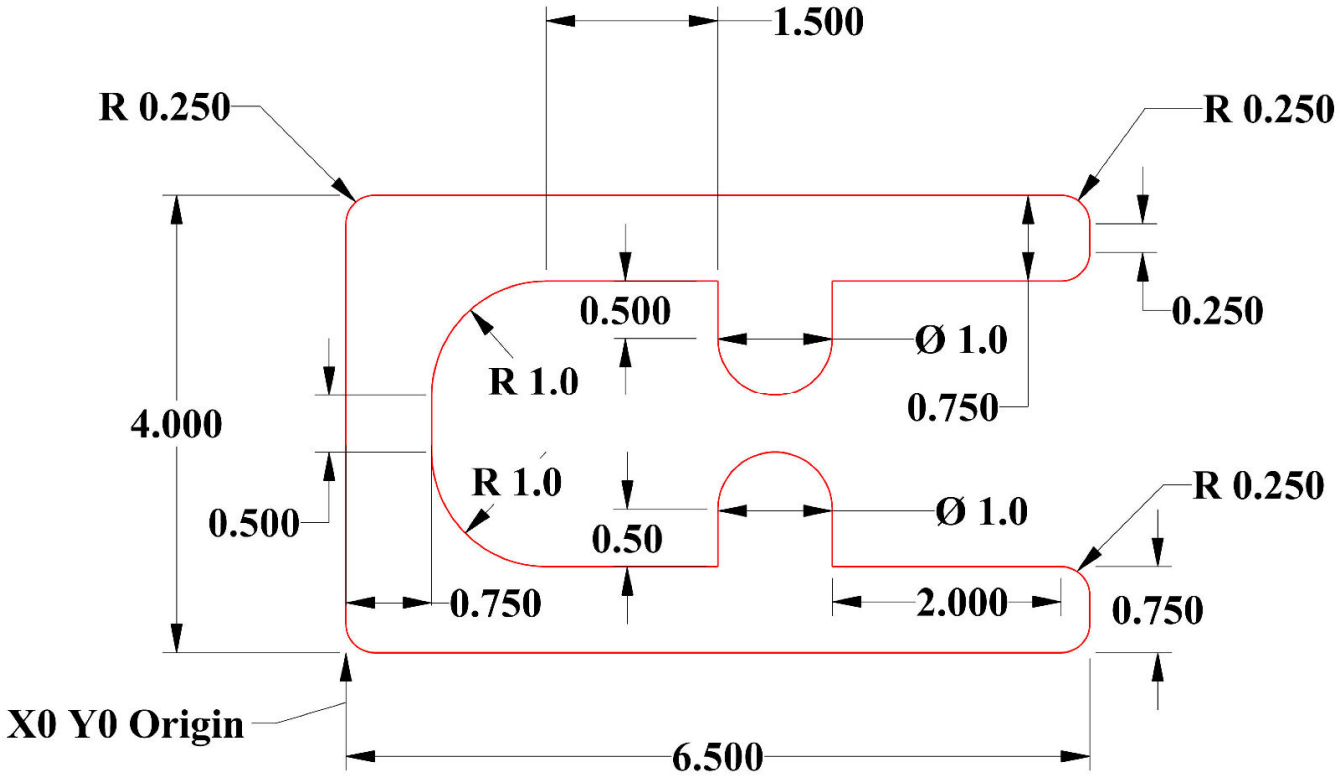


Tool	Description	Diameter	Flutes	SFM	Chip Load
	H2TE5SE0500S063HA	0.5 (1/2)	5	500	0.0037 IPT
Material	**P0 - STEEL**				

*** *Calculate* ***Speed*** ***(RPM = 3.82 x SFM / DIA)*** *and* ***Feed*** ***(IPM = FTP x T x RPM)***
*** *Outside Part Profile Tool Path*

HAAS Simulator: Go to MDI Mode:	Save Program Referencing Page Number
G00 G90 G17 G40 G49 G54	
T1 M06	
S_____ M03	
G43 H____	
X-0.5 Y-0.5	
Z1.0	
M08	
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

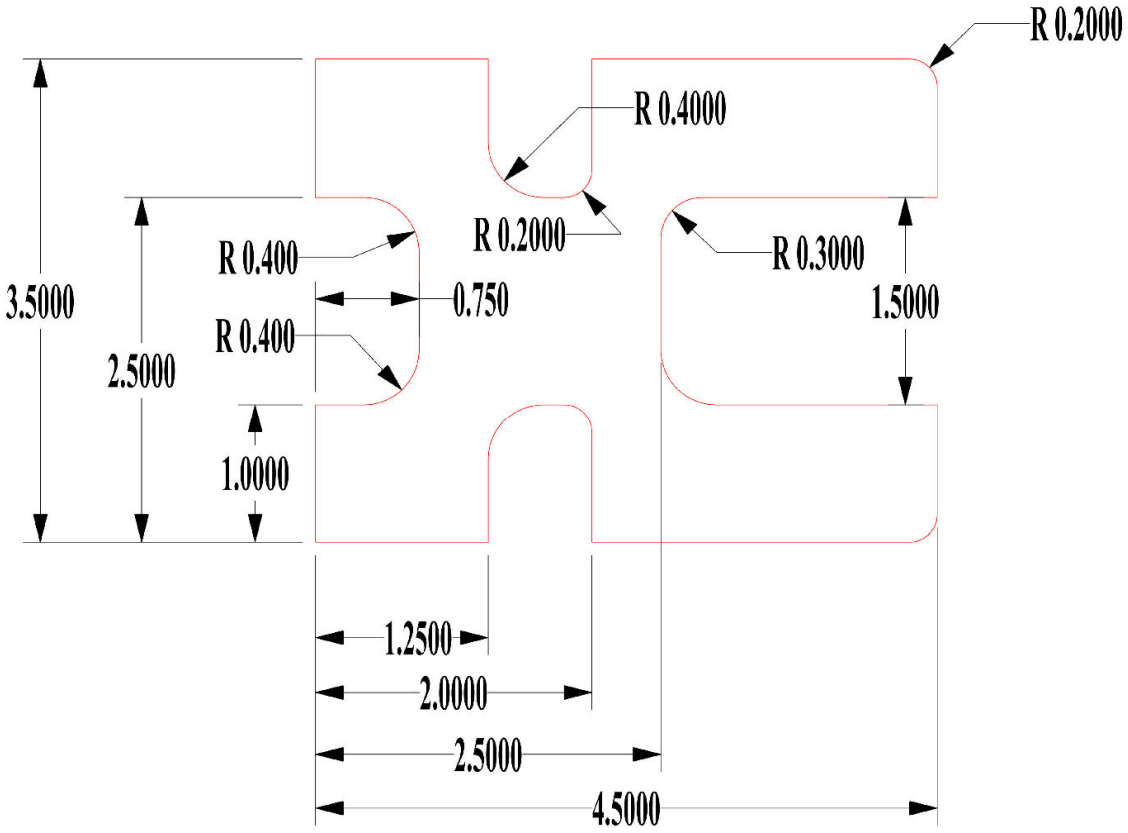


Tool	Description	Diameter	Flutes	SFM	Chip Load
	H2TE5SE0500S063HA	0.5 (1/2)	5	500	0.0037 IPT
Material	**P0 - STEEL**				

*** *Calculate* ***Speed*** ***(RPM = 3.82 x SFM / DIA)*** *and* ***Feed*** ***(IPM = FTP x T x RPM)***

HAAS Simulator: Go to MDI Mode:	Save Program Referencing Page Number
G00 G90 G17 G40 G49 G54	
T1 M06	
S_____ M03	
G43 H____	
X-0.5 Y-0.5	
Z1.0	
M08	
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	Description	Diameter	Flutes	SFM	Chip Load
	H2TE5SE0281R075HAV	0.3125 (5/16)	5	???	???
Material	**P0 - STEEL**				

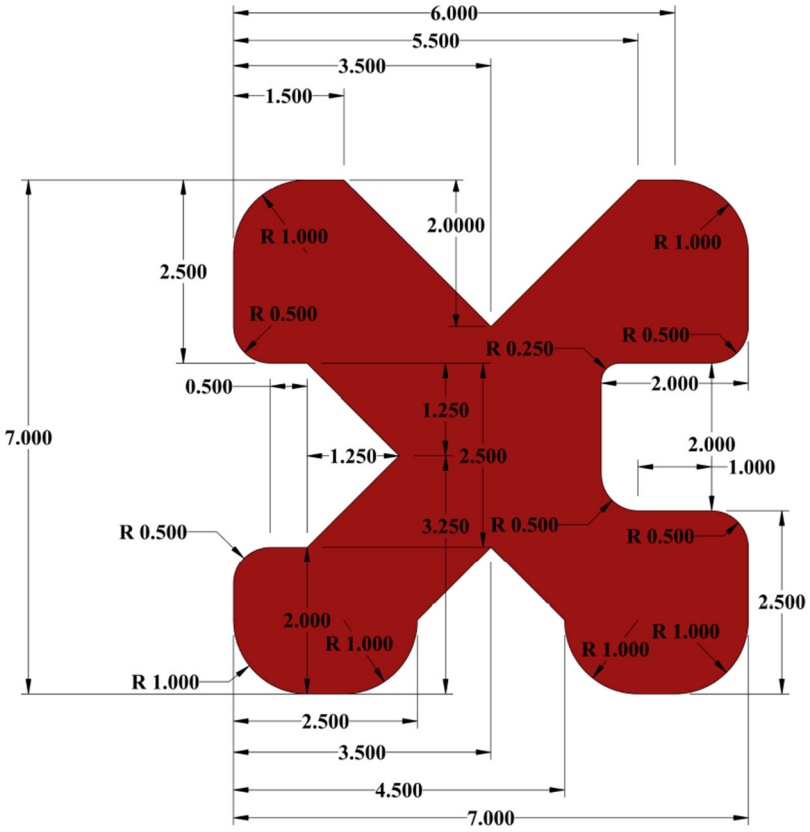
*** *Choose **SFM** and **FPT** from Kennametal Tech Info chart.*

*** *Calculate **Speed** **(RPM = 3.82 x SFM / DIA)** and **Feed** **(IPM = FTP x T x RPM)***

*** *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	
S_____ M03	
G43 H____	
X-0.5 Y-0.5	
Z1.0	
M08	
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	Description	Diameter	Flutes	SFM	Chip Load
	H2TE5SE0375S075HA	0.375 (3/8)	5	???	???
Material	**P0 - STEEL**				

*** *Choose **SFM** and **FPT** from Kennametal Tech Info chart.*

*** *Calculate **Speed** **(RPM = 3.82 x SFM / DIA)** and **Feed** **(IPM = FTP x T x RPM)***

*** *This is an **extra Exercise**. Sharp corners will be **round 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	
S_____ M03	
G43 H____	
X-0.5 Y-0.5	
Z1.0	
M08	
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. **G12** cuts in a clockwise direction, while **G13** cuts counterclockwise. Both commands use the default **XY plane (G17)** and automatically apply cutter compensation—**G42** for G12 and **G41** for G13. Both G-codes 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

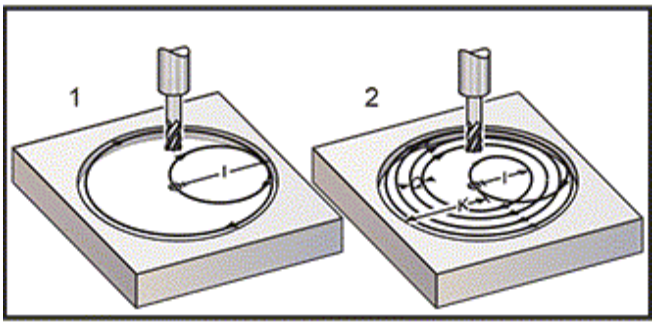
** To calculate the programmed circle diameter, the control uses the tool size associated with the selected D-code. To program the tool centerline, specify D0.

NOTE

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

Rapid-position the tool to the center of the circle. To remove all material inside the circle, use **I** and **Q** values that are less than the tool diameter and a **K** value equal to the circle radius. To cut only the circle radius, use an **I** value set to the radius and do **not** specify **K** or **Q**.

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 G41 or G42 do not need to be programmed in the block. However, a **D** offset number must be included to apply the cutter radius or diameter and adjust the circle diameter.

The following examples illustrate the **G12** and **G13** formats and the various ways they can be written.

Single Pass: Use **I** only.
Applications: One-pass counterboring; rough and finish pocketing of smaller holes; I.D. cutting of O-ring grooves.

Multiple Pass: Use **I, K, and Q**.
Applications: Multiple-pass counterboring; rough and finish pocketing of large holes with cutter overlap.

Multiple Z-Depth Pass: Use **I** only, or **I, K, and Q** (G91 and **L** may also be used).
Applications: Deep rough and finish pocketing.

The previous figures illustrate the toolpath produced by the pocket-milling G-codes.

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

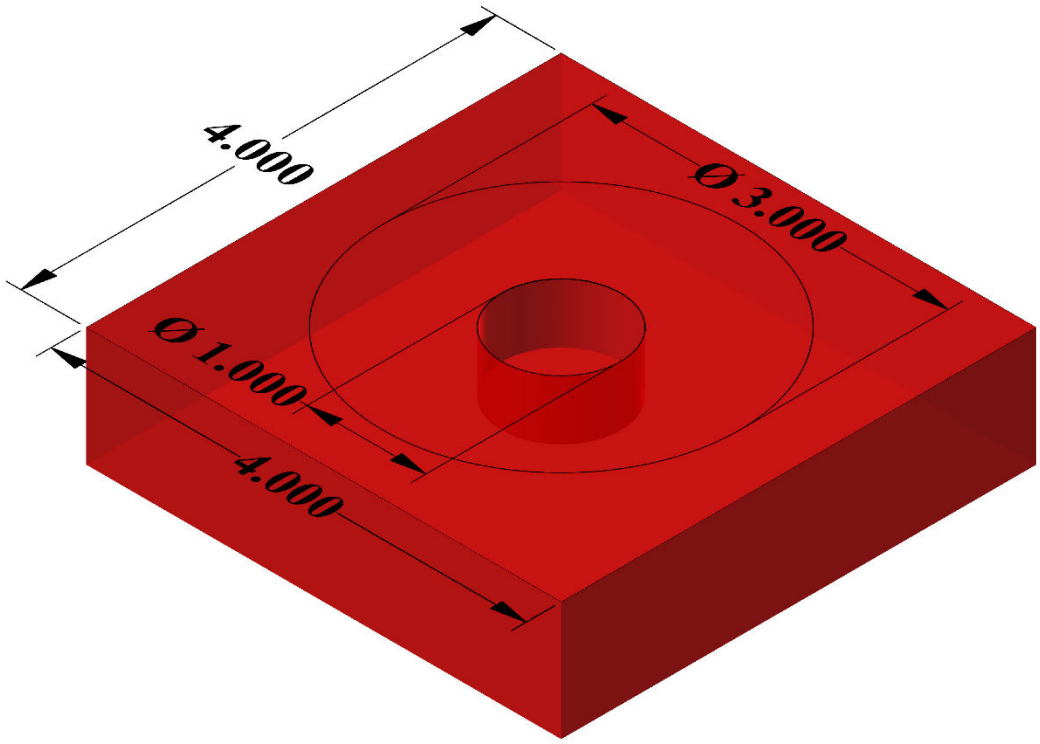
***** Program O60131 in G12 & G13 Sample Program Folder on USB Drive *****

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	Chip Load
Tool 1		1/2 (.5000) Dia, 4 Flt, SE Square Carbide Endmill	250	.0013 IPT

*** *Calculate* ***Speed*** ***(RPM = 3.82 x SFM / DIA)*** *and* ***Feed*** ***(IPM = FTP x T x RPM)***

*** *Choose your origin* ***(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	
G43 H____	
X__ Y__	
Z__	
M08	
G01 _____ → (Feed down to Z depth)	
G13 D____ →	
…	
M30	
…	
Run program in Graphic Mode.	

CANNED CYCLES

Canned cycles are G-codes used for repetitive operations such as drilling, tapping, and boring. They are defined using alphabetic address codes, and while the cycle is active, the machine performs the specified operation each time a new position is commanded—unless otherwise directed.

Canned cycles simplify programming because most common Z-axis repetitive operations use them. Once active, the cycle executes at every new axis location. Axis motion occurs in rapid mode (**G00**), and the canned cycle action is performed after the move. This applies to **G17** and **G19** plane cycles, as well as Y-axis movements on Y-axis lathes.

DRILLING CANNED CYCLES

All four drill canned cycles can be looped when using **G91**, Incremental Programming mode.

- **G81 Drill Canned Cycle** is the basic drilling cycle. It is used for drilling shallow holes or for drilling with Through-Spindle Coolant (TSC).
- **G82 Spot Drill Canned Cycle** operates the same as G81 but includes an optional dwell at the bottom of the hole. The **P** address specifies the dwell duration.
- **G83 Normal Peck Drilling Canned Cycle** is typically used for deep holes. Peck depth may be variable or constant, and it is always incremental. Use **Q** for peck depth, but do **not** use Q with **I, J, or K** values.
- **G73 High-Speed Peck Drilling Canned Cycle** is similar to G83, except that peck retraction is controlled by **Setting 22 – Can Cycle Delta Z**. Peck drilling is recommended for holes deeper than three times the drill diameter. The initial peck depth (**I**) should generally equal one tool diameter.

TAPPING CANNED CYCLES

There are two tapping canned cycles, and both can be looped when using **G91**, Incremental Programming mode.

- **G84 Tapping Canned Cycle** is the standard tapping cycle used for cutting **right-hand threads**.
- **G74 Reverse Tap Canned Cycle** is used for tapping **left-hand threads**.

BORING AND REAMING CYCLES

There are five boring canned cycles, and all of them can be looped when using **G91**, Incremental Programming mode.

- **G85 Boring Canned Cycle** is the basic boring cycle. It bores to the programmed depth and then returns to the specified height.
- **G86 Bore and Stop Canned Cycle** functions like G85, except the spindle stops at the bottom of the hole before the tool retracts.
- **G89 Bore In, Dwell, Bore Out Canned Cycle** is similar to G85 but includes a dwell at the bottom of the hole. During retraction, the tool continues boring at the programmed feedrate rather than retracting in rapid or jog mode, as other boring cycles do.
- **G76 Fine Boring Canned Cycle** bores to the programmed depth, then moves to clear the tool from the hole before retracting.
- **G77 Back Bore Canned Cycle** operates similarly to G76, but it first moves the tool to clear the hole, then positions it inside the hole before boring to depth.

R PLANES

R planes (return planes) specify the Z-axis return height used during canned cycles.

R-plane commands remain active for the duration of the canned cycle in which they are used. **G98** (Canned Cycle Initial Point Return) returns the Z-axis to the height it was at before the canned cycle began.

G99 (Canned Cycle R-Plane Return) returns the Z-axis to the height specified by the **R** value programmed in the canned cycle block.

CANNED CYCLES G81 AND G82

Canned cycles automate a series of movements and reduce the amount of code required. These cycles apply only to **Z-axis** operations, while **X** and **Y** (in **G17**) specify the hole location. The next commanded position may use either **G90** (absolute) or **G91** (incremental) positioning. **G98** (Canned Cycle Initial Point Return) and **G99** (Canned Cycle R-Plane Return) control the Z-axis return height, and both are modal commands.

G80 (Canned Cycle Cancel) must be used to cancel the cycle after completion.

See the sample program below for examples using **G90** and **G91** movements.

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

G81 DRILL CANNED CYCLE (GROUP 09)

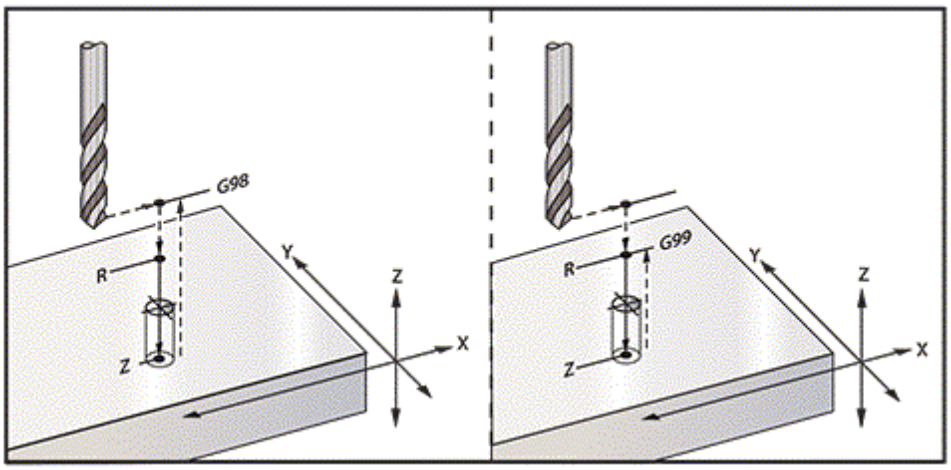
* **E** - Chip-clean RPM (Spindle reverses to remove chips after each cycle)
F - Feedrate
* **L** - Number of holes to drill if *G91* (Incremental Mode) is used
* **R** - Position of the R plane (position above the part)
* **X** - X-Axis motion command
* **Y** - Y-Axis motion command
Z - Position of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

* indicates optional

CAUTION

Unless otherwise specified, this canned cycle uses the most recently commanded spindle direction (M03, M04, or M05). If no spindle direction is commanded beforehand, the machine defaults to M03 (clockwise). If M05 is active, the cycle runs with no spindle rotation, which is appropriate for self-driven tools but may cause a crash in other situations. Always verify spindle direction before using 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

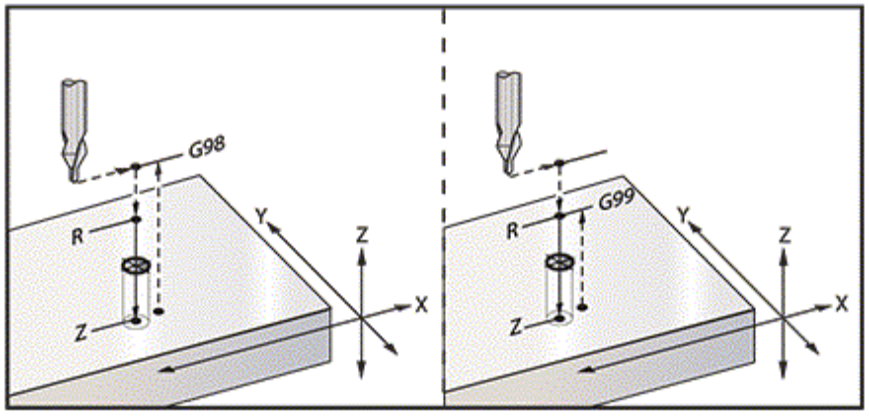
The P values are modal. This means that if a G04 Pnn or an M97 Pnn is used during a canned cycle, the P value will apply to the dwell or subprogram as well as the canned cycle.

CAUTION

Unless otherwise specified, this canned cycle uses the most recently commanded spindle direction (M03, M04, or M05). If no spindle direction is commanded beforehand, the machine defaults to M03 (clockwise). If M05 is active, the cycle runs with no spindle rotation, which is appropriate for self-driven tools but may cause a crash in other situations. Always verify spindle direction before using this canned cycle.

NOTE

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

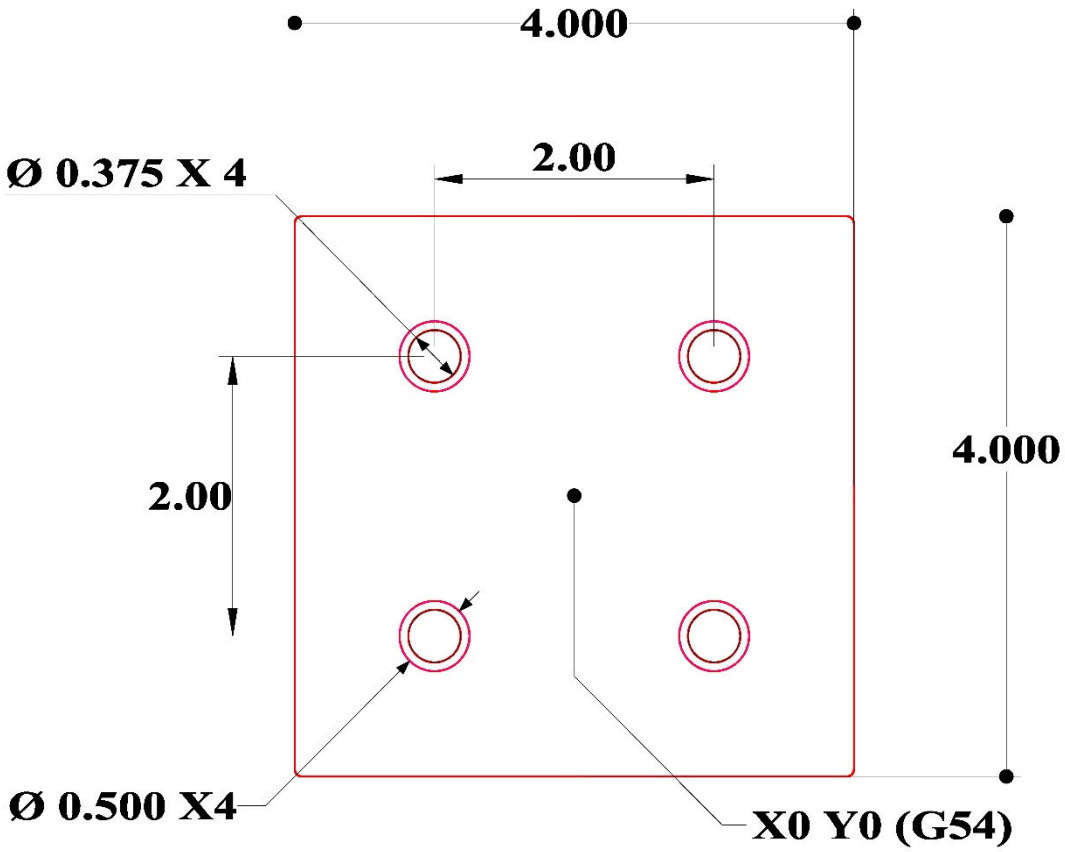


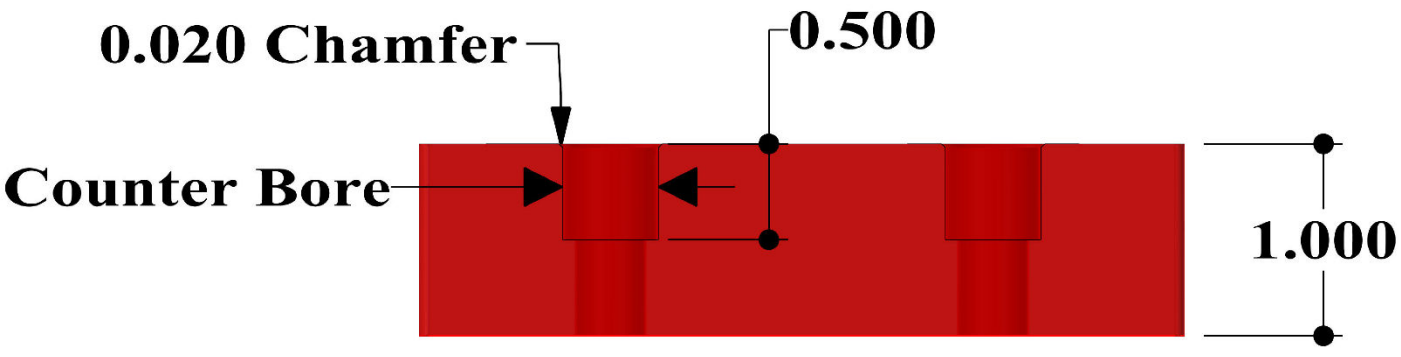
***G82* Spot Drilling Example**

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

Notes

Part Drawing





Tool Information for G81 and G82 Canned Cycle Exercise			Surface Feed	Chip Load
Tool 1		**3/8 (.3750) Dia, 2 Flt, Screw Machine**	**100**	**.007 IPR**
Tool 2		**HSS Countersink, 3/4 (.7500) Dia, 82 Degree**	**60**	**.005 IPR**
Tool 3		**1/2 (.5000) Dia, 4 Flt, SE Square Carbide Endmill**	**250**	**.0013 IPT**

**** Calculate* ***Speed*** ***(RPM = 3.82 x SFM / DIA)*** *and* ***Feed*** ***(IPM = FTP x T x RPM) / (IPM = IPR x RPM)***
**** Put a* ***0.020”*** *Chamfer on Counter Bore using* ***Drill Point Depth and Countersink Diameter Formula from HAAS Machinist Handbook***

Operation Order:

1. ***Drill with Tool 1 (G81)***
2. ***Counter Bore with Tool 3 (G81 or G82) *** G82 allows you to dwell at Z depth.***
3. ***Chamfer holes with Tool 2 using formula from HAAS Machinist Handbook***

HAAS Simulator: Go to MDI Mode:	Save Program Referencing Page Number
G00 G90 G17 G40 G49 G54 (Safety Line) **T___ M06** **G43 H____** **S_____ M03** **X___ Y____** **Z5.0** **M08** **G01 _____ →** **…** **G81 …** **…** **G82 …** **…** **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

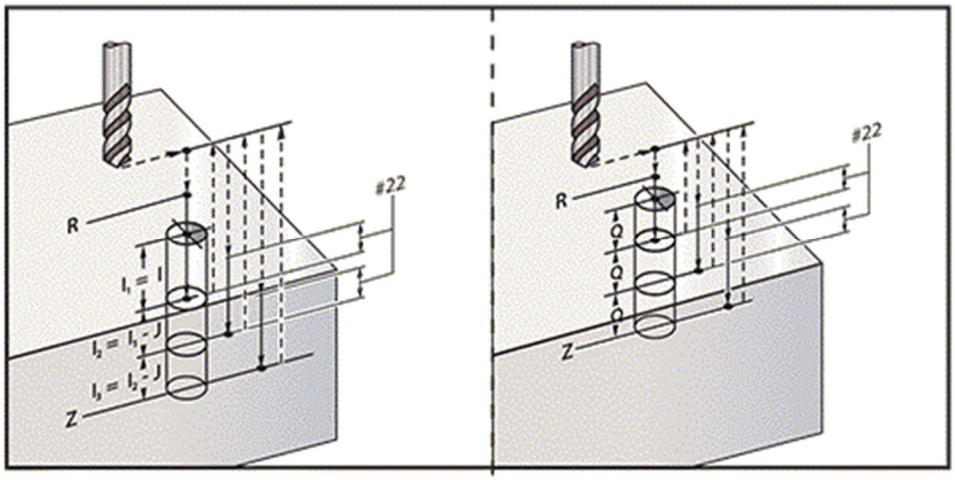
If **I**, **J**, and **K** are specified, the first pass will cut in by the amount of **I**, each succeeding pass will be reduced by **J**, and the minimum cutting depth is **K**. Do **not** use a **Q** value when programming with **I**, **J**, and **K**.

If **P** is specified, the tool will pause at the bottom of the hole for the programmed duration. For example, the following block will peck several times and include a 1.5-second dwell:

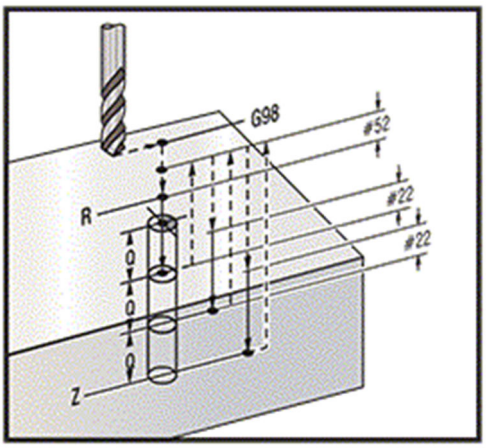
G83 Z-0.62 F15. R0.1 Q0.175 P1.5 ;

The same dwell time applies to all subsequent blocks that do not specify a new **P** value.

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



Setting 52 modifies how **G83** handles retracts to the **R-plane**. Normally, the R-plane is positioned well above the part so each peck retract clears chips, but this also adds unnecessary travel as the tool repeatedly re-enters empty space. When Setting 52 is set to the minimum distance required for chip clearance, the R-plane can be set closer to the part. During chip-clearing retracts, Setting 52 determines how far above the programmed R-plane the Z-axis retracts.

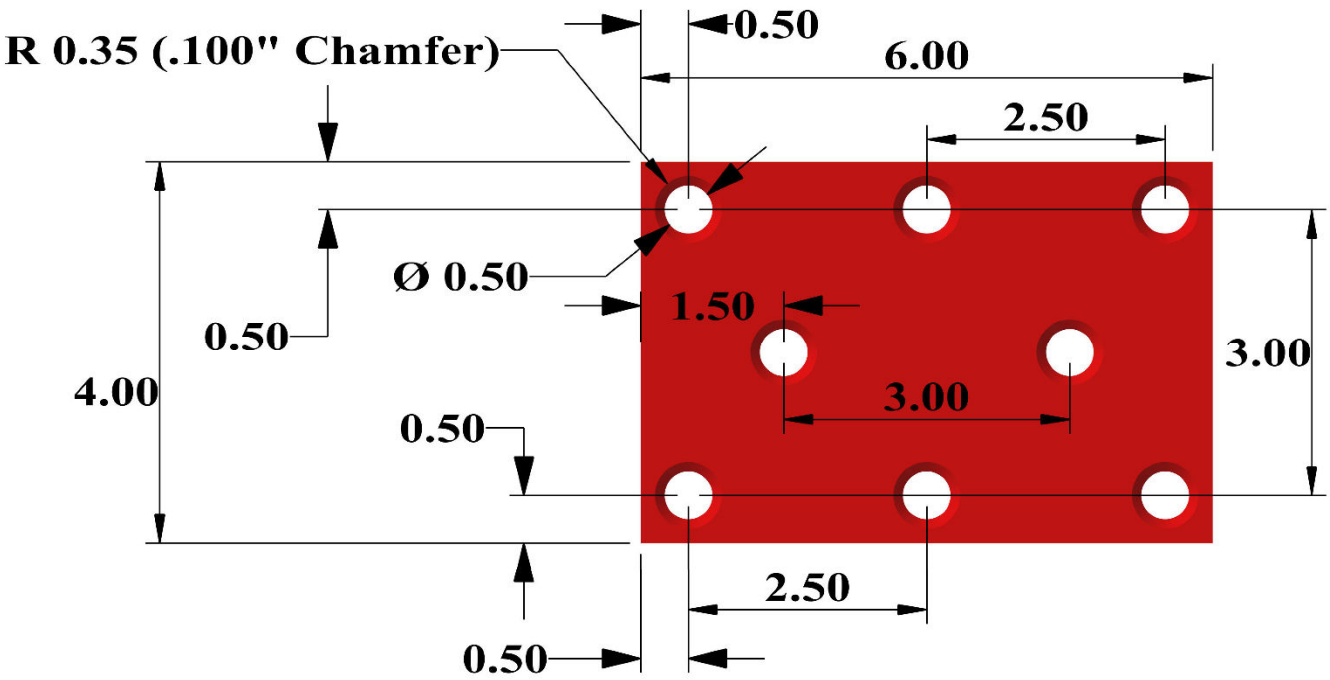


***G83* peck Drilling Canned Cycle with Setting 52 [#52]**

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

G83 AND G82 CANNED CYCLE EXERCISE

Part Drawing



Tool Information for G83 and G82 Canned Cycle Exercise			Surface Feed	Chip Load
Tool 1		1/2 (.5000) Dia, 2 Flt, Jobber Length Drill, HSS Oxide Finish, 4.5000 LOC	95	.006 IPR
Tool 2		HSS Countersink, 3/4 (.7500) Dia, 82 Degree	60	.005 IPR

*** *Calculate* ***Speed*** ***(RPM = 3.82 x SFM / DIA)*** *and* ***Feed*** ***(IPM = IPR x RPM)***
*** *Put a* ***0.100”*** *Chamfer on Counter Bore using* ***Drill Point Depth and Countersink Diameter Formula from HAAS Machinist Handbook***

Operation Order:

1. ***Drill with Tool 1 (G81)***
2. ***Chamfer holes with Tool 2 using formula from HAAS Machinist Handbook***

HAAS Simulator: Go to MDI Mode:	Save Program Referencing Page Number
G00 G90 G17 G40 G49 G54 (Safety Line) **T___ M06** **G43 H____** **S_____ M03** **X___ Y____** **Z5.0** **M08** **G01 _____ →** **…** **G81 …** **…** **M30** **Run program in Graphic Mode.**	

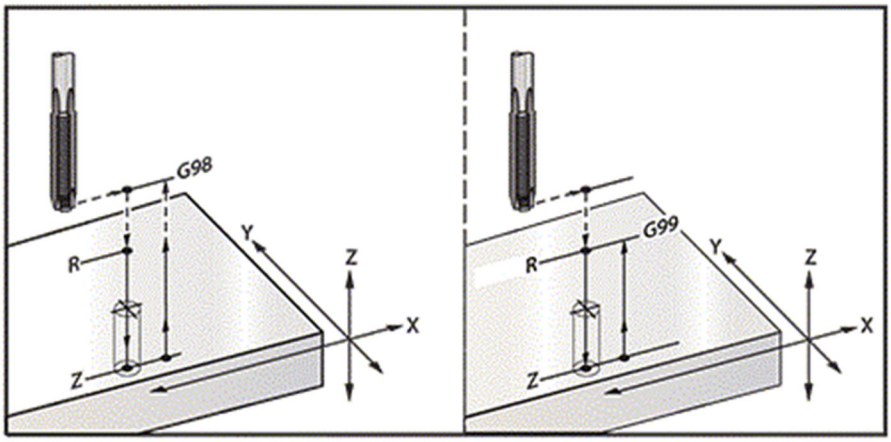
G84 TAPPING CANNED CYCLE (GROUP 09)

* **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 automatically starts and stops the spindle as required.

_G84_ Tapping Canned Cycle



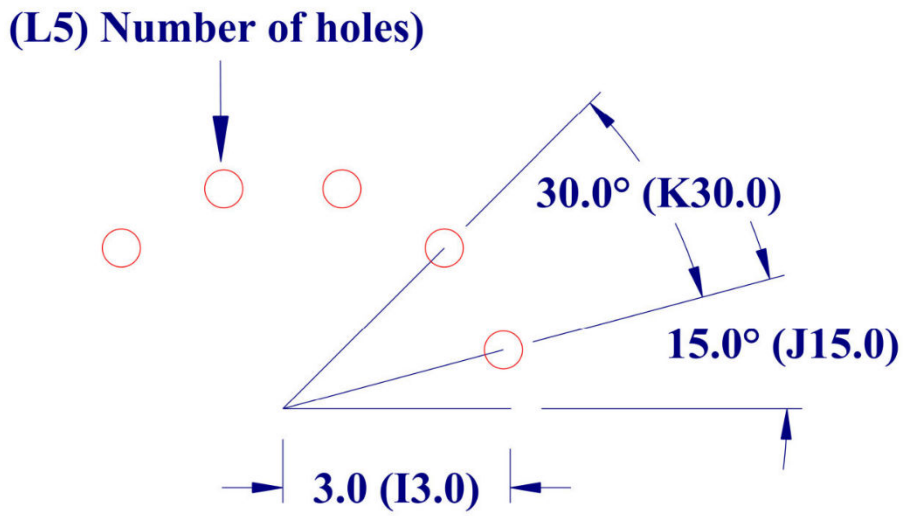
***** Program is in G84 Sample Program Folder on USB Drive *****

G70 / G71 / G72 BOLT HOLE PATTERN

G70, **G71**, and **G72** are use with Canned Cycles. Must execute Canned cycle first before providing Bolt hole pattern.

Example below Uses **G81** with **G71** Bolt Hole Pattern.

G00 X0 Y0
Z5.0
G81 Z-0.5 R0.1 F10.0
G71 I3.0 J15.0 K30.0 L5.0



Rules for Bolt Pattern Canned Cycles

1. Position the tool at the center of the bolt pattern (G70 or G71) or at the starting hole location (G72) before executing the canned cycle.

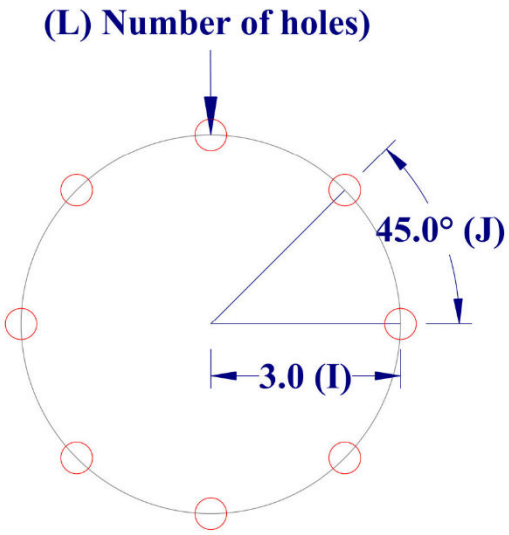
2. The J code defines the starting angle and is always specified from 0° to 360°, measured counterclockwise from the three-o'clock position.

3. For G70 and G71, program **L0** on the initial canned-cycle line to skip drilling at the center of the pattern. Alternatively, disabling Setting 28 prevents drilling at the initial X/Y location. Refer to *Setting 28 – Can Cycle No Linear Axes* for details.

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

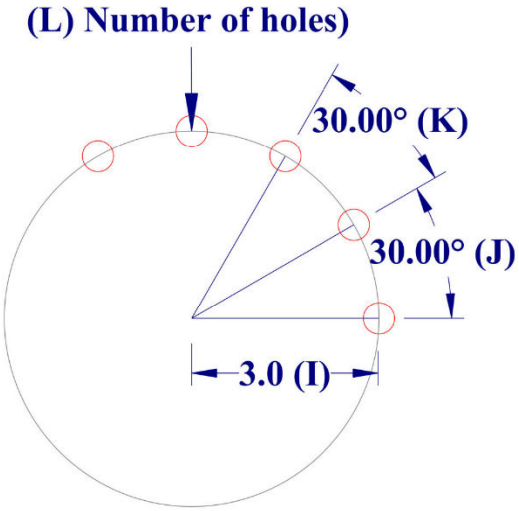
*Indicates optional



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

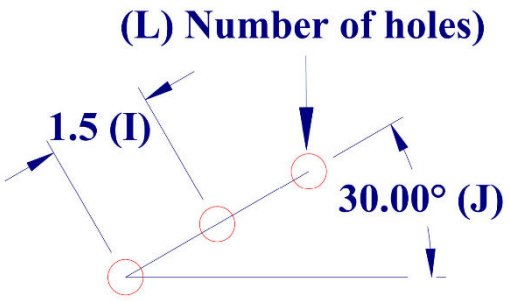
*Indicates optional



G72 BOLT HOLES ALONG AN ANGLE (GROUP 00)

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

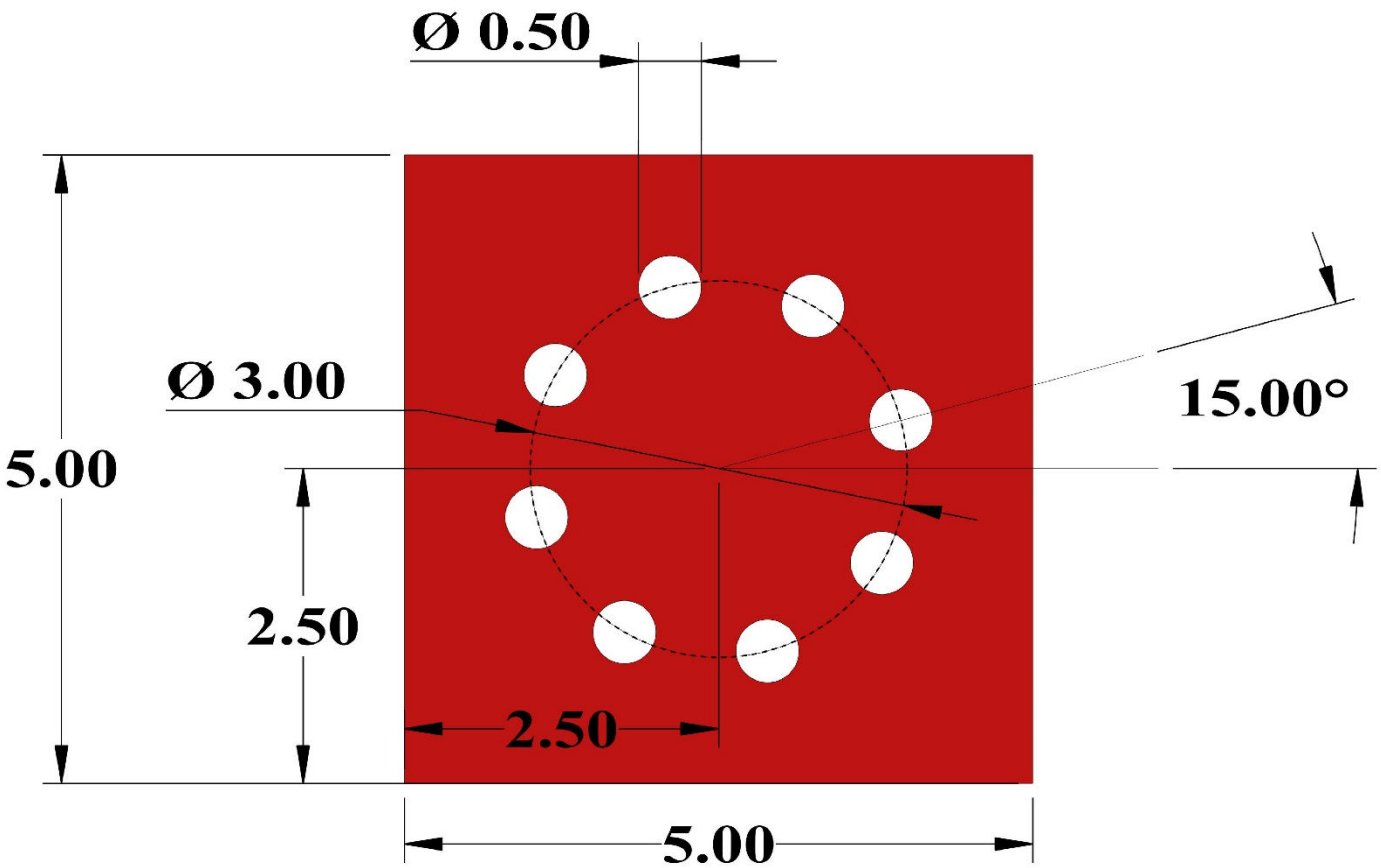
*Indicates optional

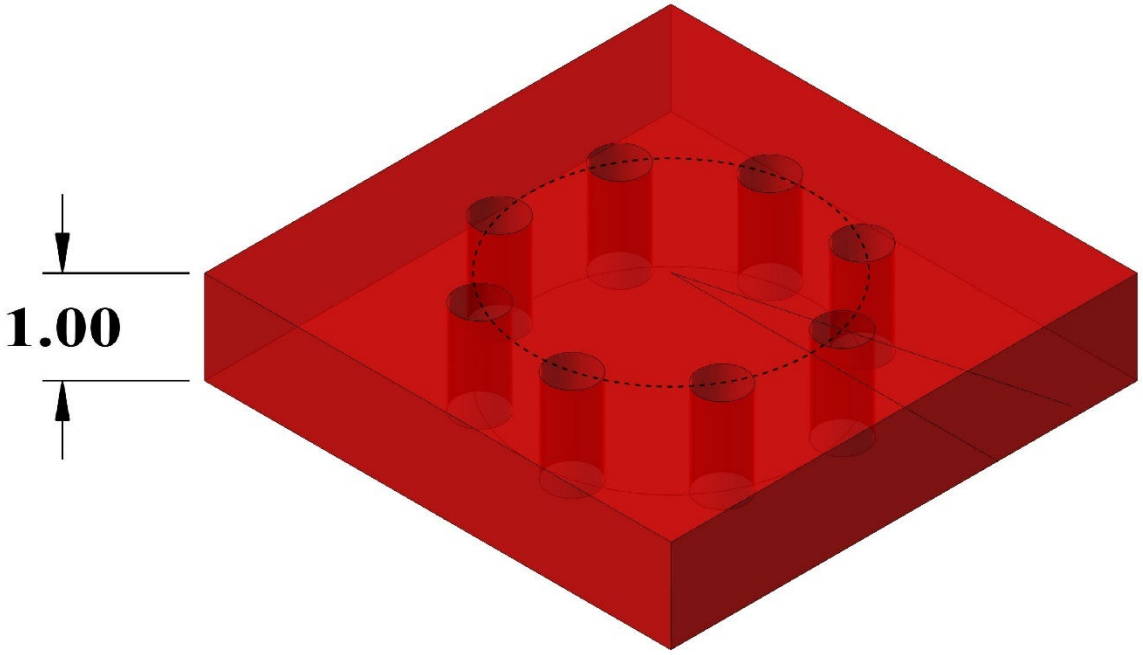


Notes

G81 CANNED CYCLE WITH G70 (BOLT HOLE CIRCLE) EXERCISE

Part Drawing





Tool Information for G81 with G70			Surface Feed	Chip Load
Tool 1		1/2 (.5000) Dia, 2 Flt, 2.0000 LOC	100	.007 IPR

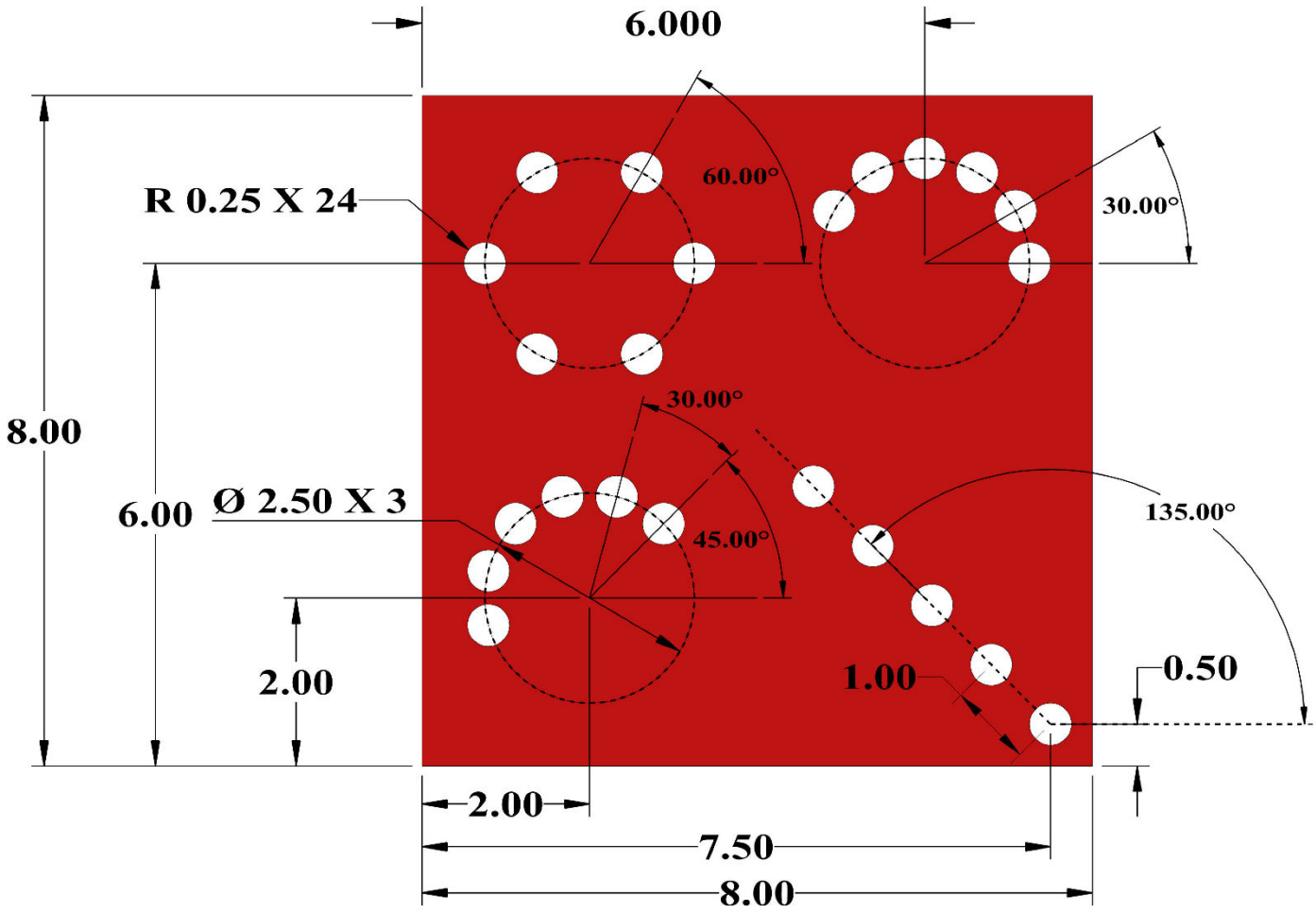
*** *Calculate* ***Speed*** ***(RPM = 3.82 x SFM / DIA)*** *and* ***Feed*** ***(IPM = IPR x RPM)***
*** *Choose your Origin on part*

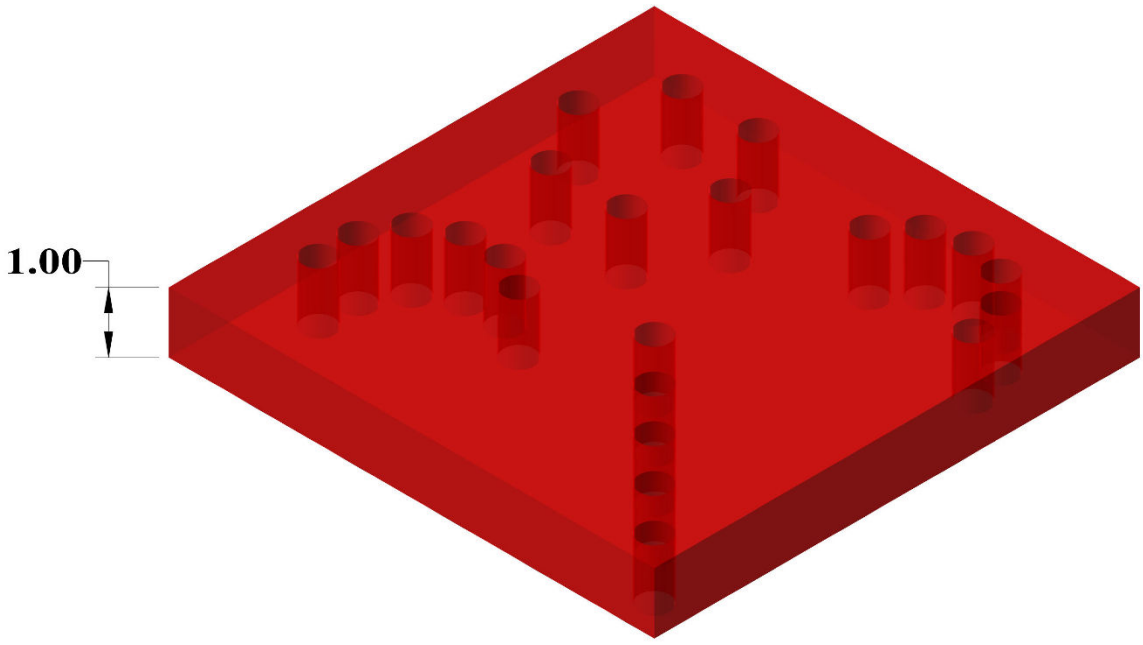
Drill (G81) with G70 Bolt Hole Circle

HAAS Simulator: Go to MDI Mode:	Save Program Referencing Page Number
G00 G90 G17 G40 G49 G54 (Safety Line) **T___ M06** **G43 H____** **S_____ M03** **X_____ Y____** **Z5.0** **M08** **…** **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	Chip Load
Tool 1		1/2 (.5000) Dia, 2 Flt, 2.0000 LOC	100	.010 IPR

*** *Calculate* ***Speed*** ***(RPM = 3.82 x SFM / DIA)*** *and* ***Feed*** ***(IPM = IPR x RPM)***
*** *Choose your Origin on part*

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** **G43 H____** **S_____ M03** **X_____ Y____** **Z5.0** **M08** **…** **…** **G81… L0 _____ (L0 does not drill a hole in the center of bolt hole pattern.)** **…** **X___ Y___ L0 (Does not execute Canned Cycle with L0)** **…** **M30** **Run program in Graphic Mode.**	

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

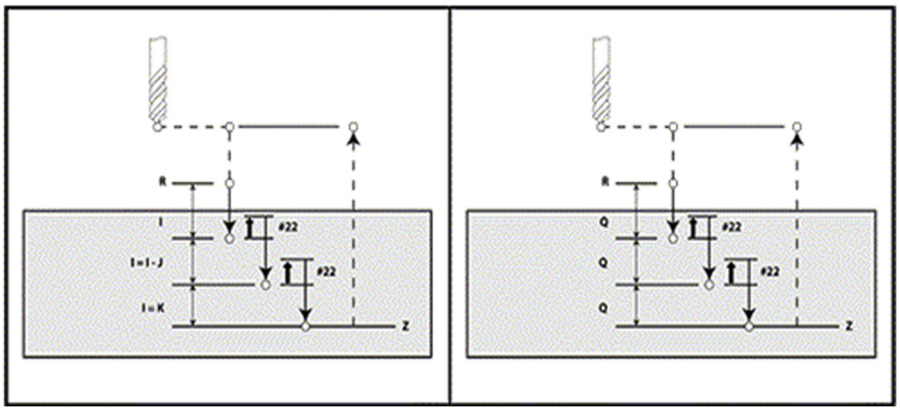
F - Feedrate
* **I** - First peck depth
* **J** - Amount to reduce pecking depth for pass
* **K** - Minimum peck depth (The control calculates the number of pecks)
* **L** - Number of loops (Number of holes to drill) if *G91* (Incremental Mode) is used
* **P** - Pause at the bottom of the hole (in seconds)
* **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.

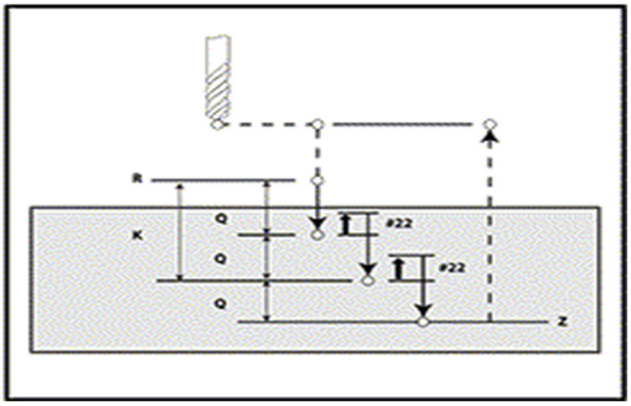
***G73* Peck Drilling. Left: Using *I*, *J*, and *K* Addresses. Right: Using Only the *Q* Address. [#22] Setting 22.**



I, J, K, and Q are always positive values.

There are three methods for programming a G73 cycle: using the I, J, and K addresses; using the K and Q addresses; or using only a Q address.

When I, J, and K are specified, the first peck advances by I, each subsequent peck is reduced by J, and K defines the minimum peck depth. If a P value is included, the tool dwells at the bottom of the hole for the programmed duration.



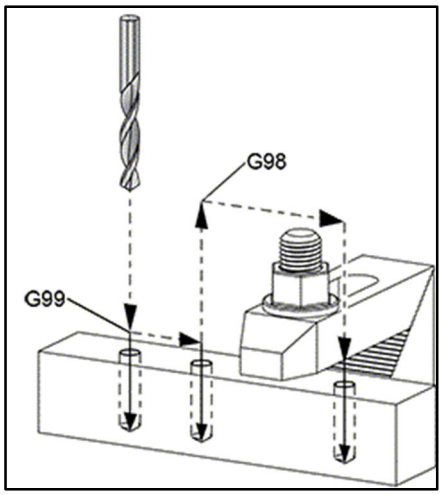
When K and Q are both specified, the cycle operates in an alternate mode in which the tool returns to the R-plane after the cumulative peck depth reaches the K value.

When only Q is specified, the cycle uses a uniform peck depth. The tool returns to the R-plane only after all pecks are completed.

G98 CANNED CYCLE INITIAL POINT RETURN (GROUP 10)

Using **G98**, the Z-axis returns to its initial starting point (the Z-position from the block before the canned cycle) between each X/Y move. This allows the tool to clear part features, clamps, and other obstructions.
In the example, after drilling the second hole, the Z-axis returns to the initial Z position under **G98** so the tool can safely move over the toe clamp to the next hole location.

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



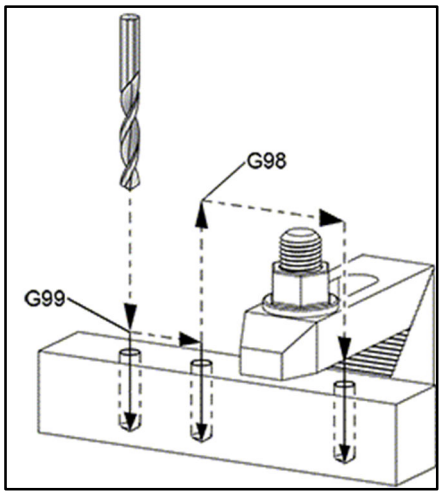
G99 CANNED CYCLE R PLANE RETURN (GROUP 10)

Using **G99**, the Z-axis remains at the programmed R-plane between each X/Y move. When there are no obstructions in the toolpath, **G99** reduces unnecessary motion and shortens cycle time.

In the example, after the first hole, the Z-axis returns to the R-plane under **G99** and moves to the second hole location. This is safe in this case because no obstacles are present.

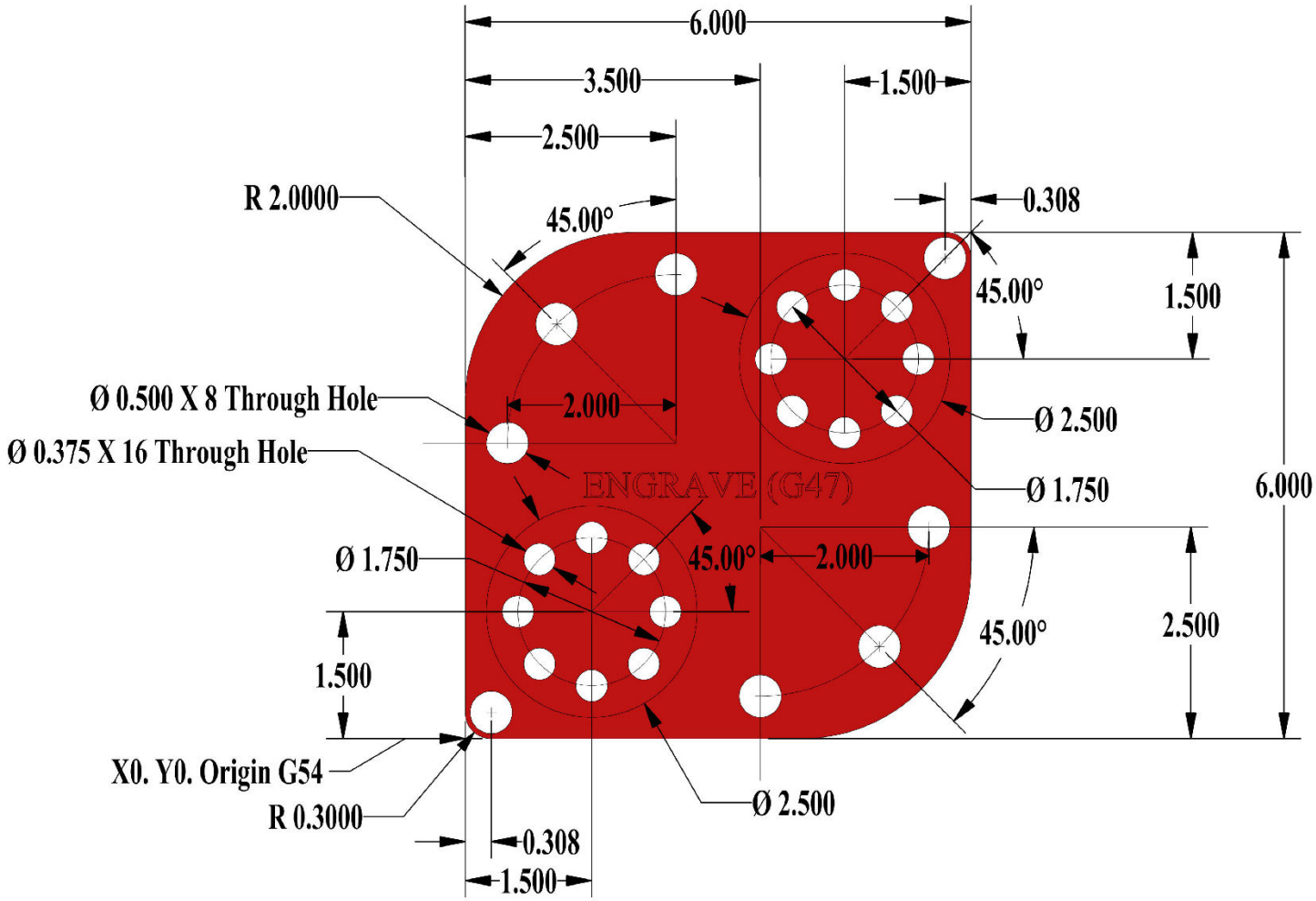
***** Program is in G99 Sample**

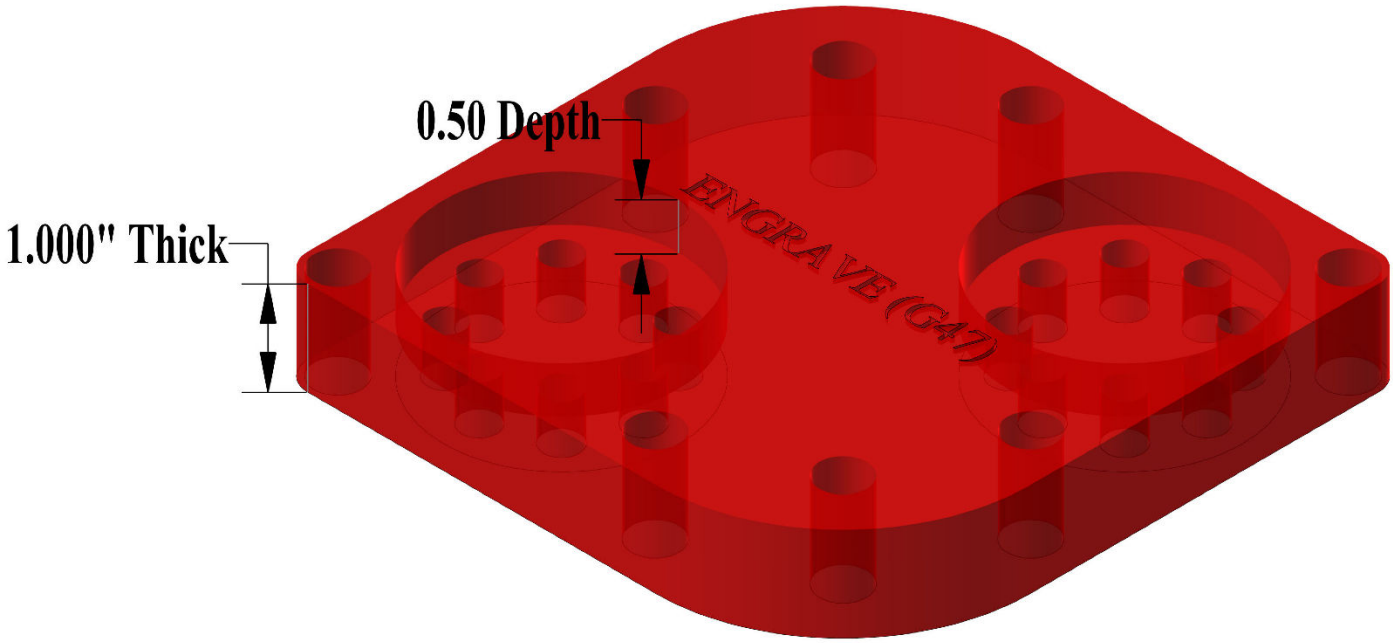
Program Folder on USB Drive ***



FINAL EXERCISE (OPTIONAL)

Part Drawing





Tool Information for Final Exercise			Surface Feed	Chip Load
Tool 1		HSS Countersink, 3/4 (.7500) Dia, 82 Degree	60	.005 IPR
Tool 2		1/2 (.5000) Dia, 2 Flt, 2.0000 LOC	100	.010 IPR
Tool 3		3/8 (.3750) Dia, 2 Flt, Screw Machine	100	.007 IPR
Tool 4		1/2 (.5000) Dia, 4 Flt, SE Square Carbide Endmill	250	.0013 IPT
Tool 5		90° Angle, .0050 Tip Dia, .1250 Shank	60	.001 IPT

*** *Create part using Canned Cycles.*
*** *Outside Profile is optional.*
*** *Engraving using **G47**. Adjust TEXT to fit center of part.*
*** ***0.020”** Chamfer on all holes*
*** *Mill out circular pockets using **VPS**.*
*** *Calculate **Speed** **(RPM = 3.82 x SFM / DIA)** and **Feed** **(IPM = FTP x T x RPM) / (IPM = IPR x RPM)***
*** *Choose your Origin on part*

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** cycle begins by positioning the cutter at a start point inside the pocket, then roughs the pocket outline and finishes with a final cleanup pass. The tool plunges in the Z-axis before pocketing. A subprogram (**P###**) defines the closed-geometry pocket shape using **G01**, **G02**, and **G03** motions in X and Y.

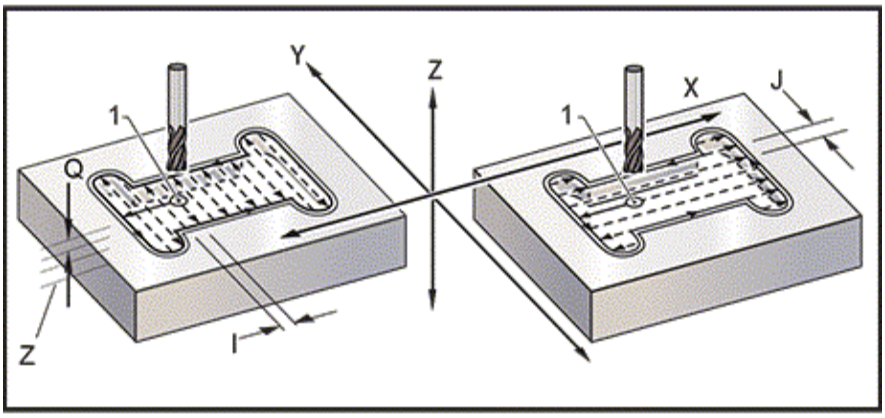
When G150 is executed, the control first searches for an internal subprogram whose **N-number** matches the P-code. If none is found, it searches for an external subprogram. If the subprogram cannot be located, the control generates **Alarm 314 – Subprogram Not In Memory**.

NOTE Do not return to the starting point after closing the pocket geometry in the G150 subprogram.

NOTE The pocket-geometry subprogram used with G150 cannot contain macro variables.

An **I** or **J** value defines the roughing-pass stepover for each cut increment. **I** produces incremental cuts in the X-axis, while **J** produces incremental cuts in the Y-axis.
The **K** command defines the finish-pass allowance. When **K** is specified, the control makes a final finishing pass by the K amount around the inside of the pocket at the final Z-depth. There is no separate finishing-pass command for Z.
The **R** value must be specified—even **R0**—otherwise the control uses the last commanded R value.
Pocketing is performed in multiple passes, starting at the R-plane. With each **Q** depth increment, the tool feeds down and removes material until the final Z-depth is reached. G150 first makes a contour pass leaving the K amount of stock, then performs roughing passes based on the I or J increment.
The **Q** command is required on the G150 line, even if only a single pass to the final Z-depth is desired. The Q value is applied from the R-plane.
Notes:
• The pocket-geometry subprogram (P) must not contain more than 40 geometry moves.
• A starting hole to full depth may be required before running G150. After drilling, position the end mill at the pocket's XY start point and then execute the G150 command.

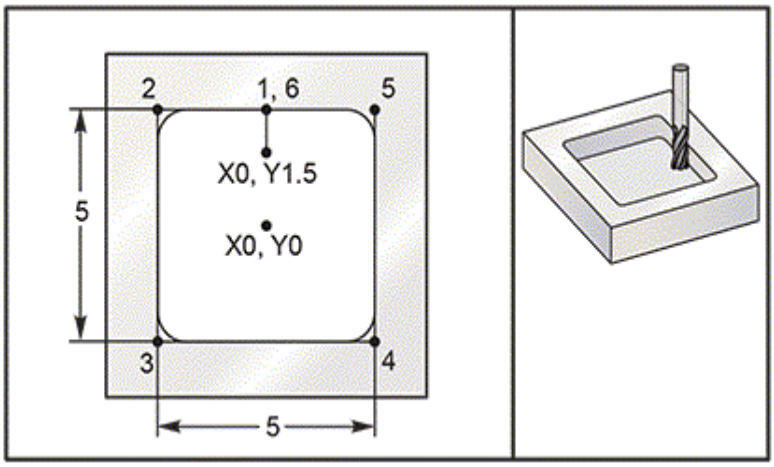
G150 GENERAL POCKET MILLING: [1] START POINT, [Z] FINAL DEPTH.



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.



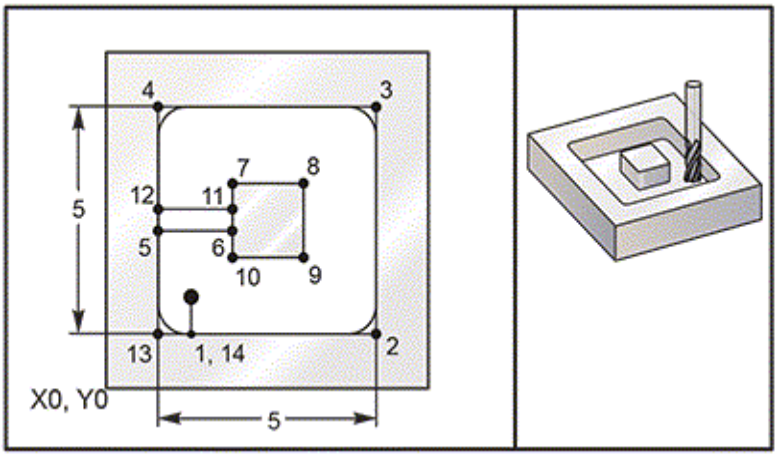
5.0 x 5.0 x 0.500 DP. Square Pocket
Main Program O61503 (G150 SQUARE POCKET MILLING)
Subprogram O61505 (G150 INCREMENTAL SQUARE POCKET MILLING SUBPROGRAM)
***** Programs are in G150 Sample Programs Folder on USB Drive *****

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

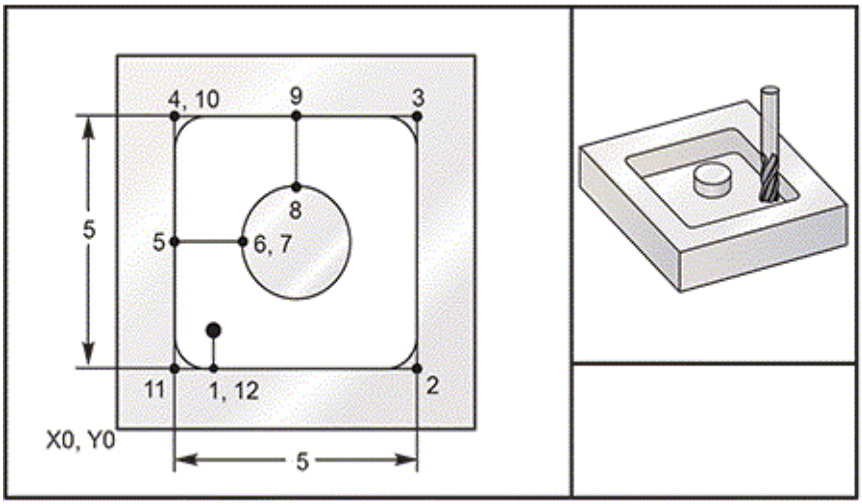
Main Program O61506 (G150 SQUARE ISLAND POCKET MILLING)

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.

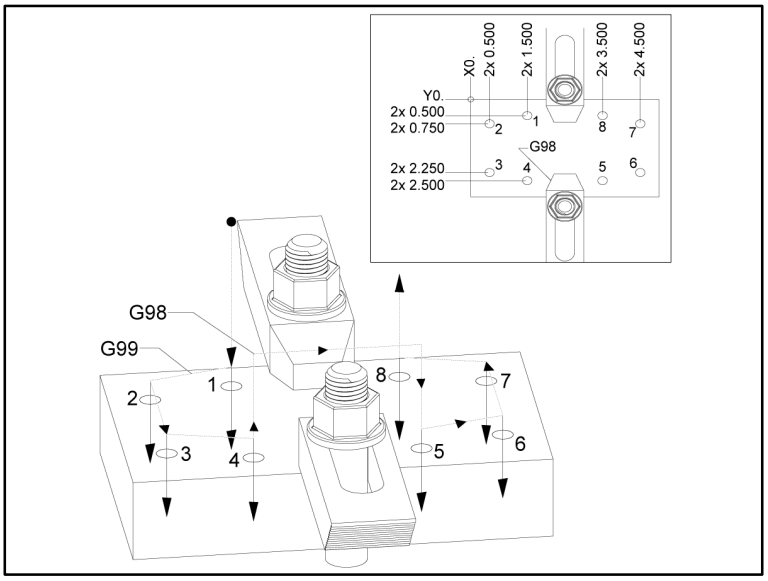
When your program calls an M98 subprogram, the control looks for the subprogram in the main program's directory. If the control cannot find the subprogram in the main program's directory, it then looks in the location specified in Setting 251. Refer to page5 for more information. An alarm occurs if the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