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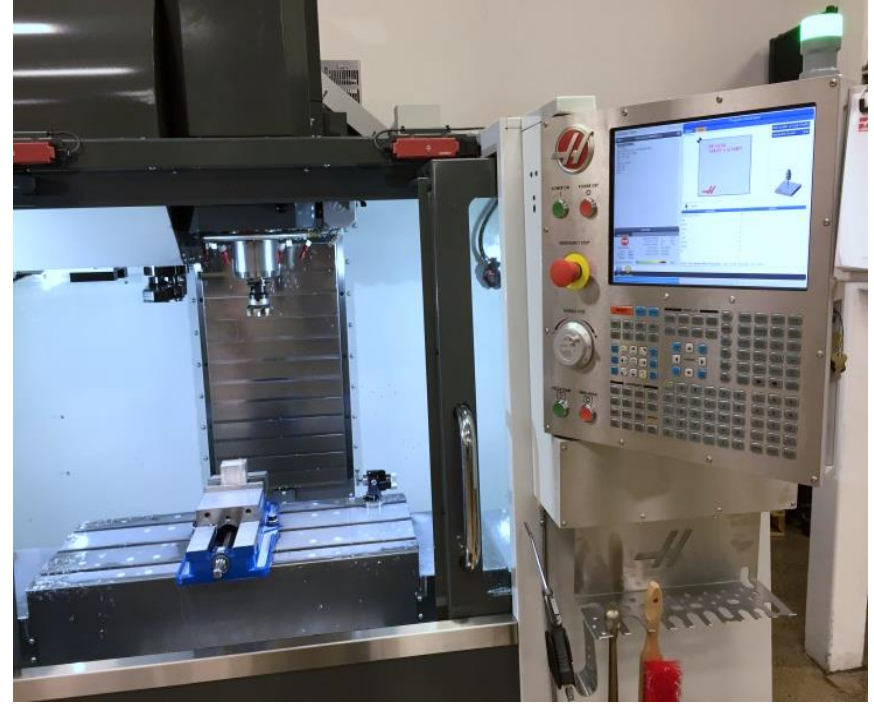
Prof. Steven S. Saliterman
Introductory Medical Device Prototyping
Department of Biomedical Engineering, University of Minnesota
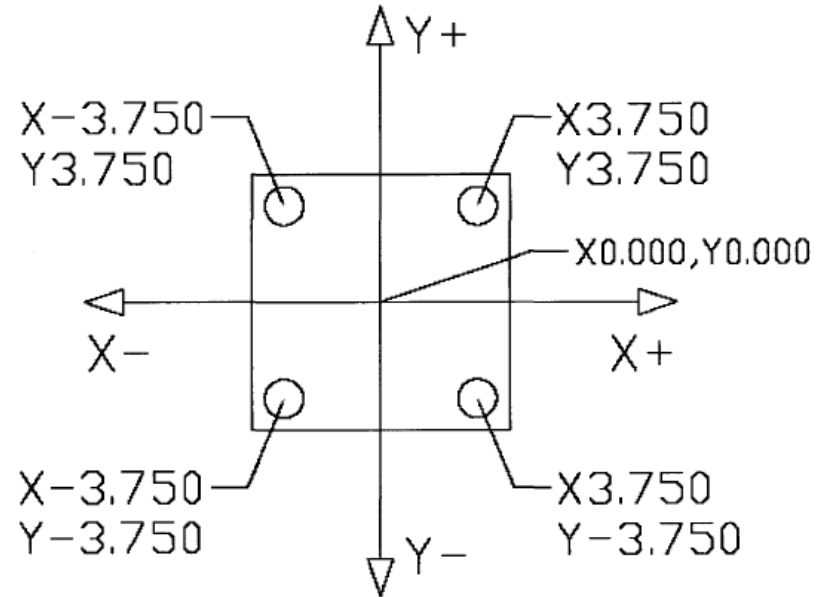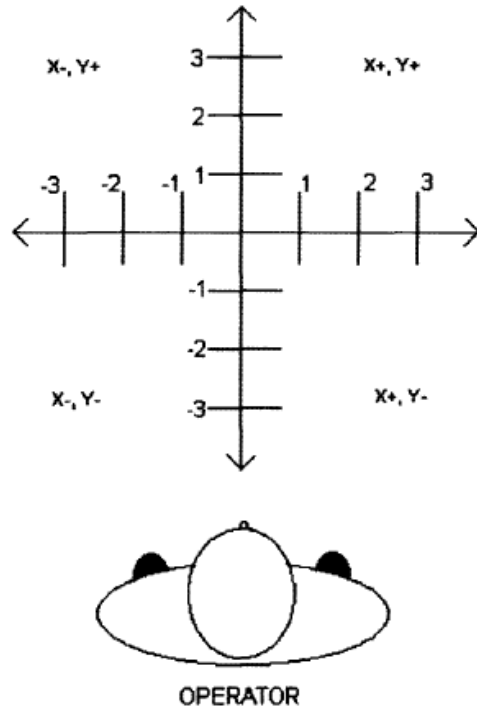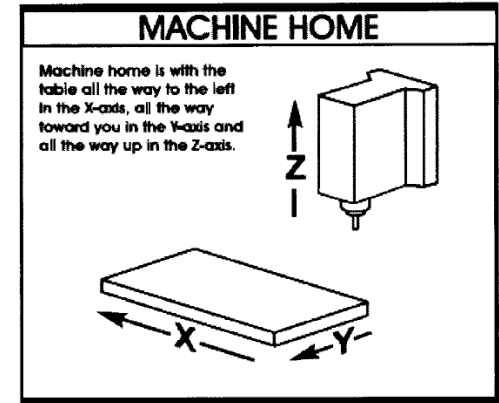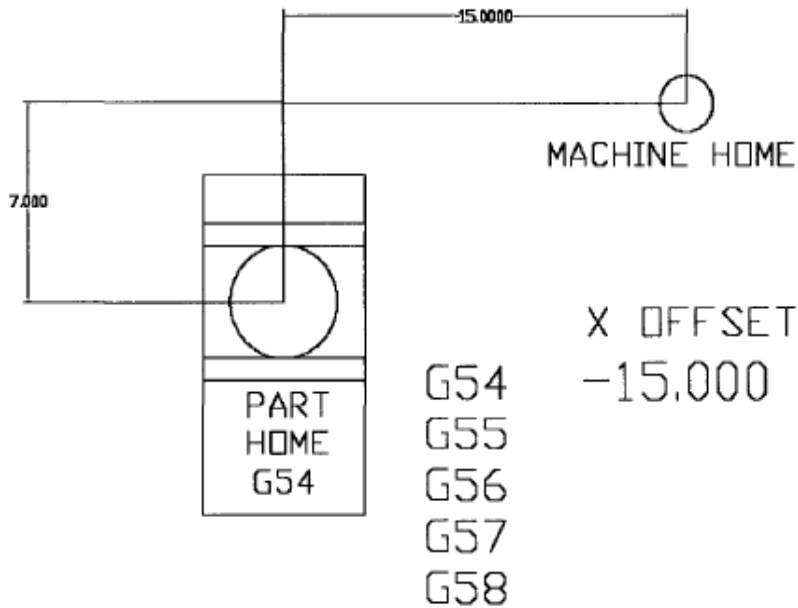http://saliterman.umn.edu/

Haas VF-2 CNC Mill

Safety Notice

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

Coordinate System

Image courtesy of Productivity and Haas

Work Offset

MACHINE HOME

Machine home is with the table all the way to the left in the X-axis, all the way toward you in the Y-axis and all the way up in the Z-axis.

—15.0000—

MACHINE HOME

7.000

PART
HOME
G54

WORK OFFSETS
(BASED FROM MACHINE HOME)

	X OFFSET	Y OFFSET	Z OFFSET
G54	−15.000	−7.000	0.000
G55			
G56			
G57			
G58			

Image courtesy of Productivity and Haas

Letter Codes

A Fourth Axis Rotation (Around the X axis)

B Fifth Axis Rotation (Around the Y axis)

C Auxiliary External Rotary Axis (Around Z)

D Tool diameter offset from the Tool Offset Page – D01 to D200.

F Speed of the movement of the spindle in the material in inches per minute (IPM). e.g. F10.0 = 10 linear IPM.

G Preparatory function – modes of operation

H Tool length offset from the Tool Offset Page – e.g. #01. H02 – Tool Length Value #2.

I Arc – distance from center to X axis

J Arc – distance from center Y axis

K Arc – distance from center to X axis

Letter Codes

L   Loop count for repeated cycles

M   Turning off and on machine functions. e.g. M3 and M4 turn the spindle on, M5 off. M8 coolant on, M9 coolant off.

N   Block line numbers. M97 functions may address an N line number.

O   Program name: Onnnnn

P   Delay time or program jump

R   Rapid plane – how far above the part to rapid the tool to. Also radius of an arc.

T   Tool selection position in the tool changer.

X, Y, Z Linear motions in these axis.

G Codes

- Come in groups – each establishing a mode of operation.
- No more than one G code in a group in a block.
- Model G codes remain effective until replaced with another code in the same group.
- Non-modal G codes are effective only in the calling block.

M Codes (One per Block)

M00	Program stop (spindle, axes, coolant)	M32	Chip conveyer backward
M01	Optional stop e.g. tool change	M33	Chip conveyer stop
M03	Spindle forward (clockwise)	M34	Increment spigot up
M04	Spindle reverse (ccw)	M35	Increment spigot down
M05	Spindle stop	M41	Low gear
M06	Load new tool e.g. T1 M06	M42	High gear
M08	Coolant on	M82	Tool unclamp
M09	Coolant off	M86	Tool clamp
M19	Orient spindle	M97	Local sub-Program call (P or L)
M30	Program end and rewind	M98	Sub-program call (P or L)
M31	Chip conveyer forward	M99	Return for subprogram or loop

Important student use setups: Go to settings and disable "tool drop" and ".1 tool rate." #76 lockout and #163 disable. Do not use M82 – tool will just fall out. Also, disable switch on controls – it can also cause this to happen.

Speed, Feed and Tapping Formulas

- M03 Spindle Forward, e.g. Snnnn
- M04 Spindle Reverse
- M05 Spindle Stop
- G94 Feed IPM (default – inches per minute)
- G95 Feed IPR (inches per revolution)

- $S = Spindle\ Speed\ in\ RPM$
- $RPM = 3.82\ x\ \dfrac{SFM}{Cutter\ Diameter}$ (revolutions per minutes)
- $SFM = 0.262\ x\ Cuttter\ Diameter\ x\ RPM$ (surface feet per minute)
- $Feed = IPM = IPR\ x\ RPM$ (inches per minute)

- $IPR = specified,\ or\ if\ chip\ load\ per\ flute\ x\ number\ of\ flutes$

- For tap, $F(inch\ per\ min) = \dfrac{RPM}{TPI}$
- For twist drill, $F(inch\ per\ min) = F(inches\ per\ revolution)x\ RPM$
- F mills, $F(inches\ per\ min) = \left(\dfrac{Feed}{tooth}x\ n\right) x\ RPM$

Safety Line

G00 G17 G40 G49 G80 G90 G98;

G00 Put machine in rapid motion.

G17 Selects X-Y plane for circular interpolation.

G40 Cancels cutter compensation.

G49 Cancels tool length compensation.

G80 Cancels any canned cycles.

G90 Absolute

G98 To initial start point in any canned operation.

Sample Program

Start up:
G00 G17 G40 G49 G80 G90 G98;   (Safety line)
G91 G28 Z0   (Return to machine zero)
T1 M06;   (Tool change)
G00 G90 G54 X0 Y0 S2500 M03;   (Rapid, Absolute, offset #1,
                                                   spindle speed, spindle on)

G43 H01 Z1.0 M08;   (Tool length comp on, tool #1,
                                                   go to Z1.0, coolant on)

Cutting Tool Path Lines:
Program Ending Lines:
GOO Z1. M09
GOO G91 G28 Z0 M5   (Retract tool tip 1.0 above part)
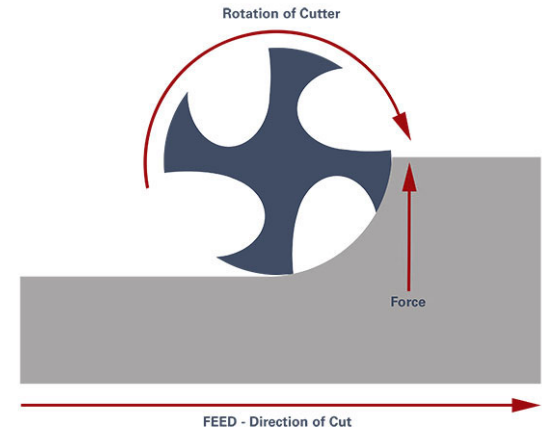G28 G91 YO   (G28 zero all axis, stop spindle)
M30   (Incremental to G28)
                                                 (End of program)

Machine Defaults at Power On

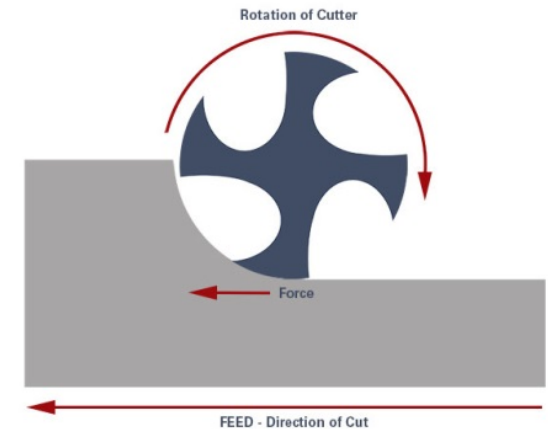|	
G00	Rapid traverse
G17	X, Y Circular plane selection
G40	Cutter Compensation cancel
G49	Tool length compensation cancel
G54	Work coordinate zero #1 (1 of 109 available)
G64	Exact stop cancel
G80	Canned cycle cancel
G90	Absolute programming
G98	Initial point return

Also, spindle speed set at zero.

G41 Climb Milling – CNC

- With the tool rotating clockwise climb milling goes WITH the rotation. Think of the flutes, or teeth of the cutter as pulling the material, or CLIMBING through the material.
- When climb milling the flute hits the material at the top of the cut, and the thickness of the chip decreases as the flute cuts.
- This results in the chips being deposited BEHIND the cut, which is important. The chips clear the cutter, which means you are not re-cutting chips. Since you are not re-cutting chips, the result is a better surface finish and longer tool life.
- Less power is required from the spindle to climb mill, and the result of the cut is down-force on the material, which can simplify work holding considerations.
- Also when finishing the floor of a feature or face milling thin material the down force can assist in stabilizing the part.
- Problematic with old manual lathes because of backlash in gears.
- Preferred method for CNC. For outside milling move around the work clockwise. When pocket milling move around the work counterclockwise.

Rotation of Cutter

Force

FEED - Direction of Cut

Images and text courtesy of Datron.
http://www.datron.com/blog/climb-milling-vs-conventional-milling/

Prof. Steven S. Saliterman

G42 Conventional Milling

- Used with manual Bridgeport lathe.
- With the tool rotating clockwise conventional milling goes AGAINST the rotation.
- The flutes of your cutter are hitting the material and pushing against the rotation, depositing chips IN FRONT of the cut. As expected, that will result in re-cutting of the chips which will both increase tool wear and decrease surface quality.
- Since the tool hits at the bottom of the part and the flute cuts upward with the chip getting heavier as it cuts, you are creating upward force on the part which can cause work holding issues.
- Preferred for hot rolled steel and cast iron.
- Tool deflection with a conventional mill tends to be parallel to the tool, it engages the rough surface at a more forgiving rate.

Rotation of Cutter

Force

FEED - Direction of Cut

Images and text courtesy of Datron.
http://www.datron.com/blog/climb-milling-vs-conventional-milling/

G00 Rapid Motion Positioning

- Use for rapid traverse of the axes to a specific location.
- Not used for cutting.
- All axis move at the same speed.
- May optionally select X, Y, Z, A axis motion.
- Motion will not necessarily be in a strait line.

G01 Linear Interpolation

- Used to cut strait lines.
- Requires a feed rate: Fnnn.nnnn
- Requires an address code: Xnn.nnnn, Ynn.nnnn, Znn.nnnn
- Subsequent axis motion will use the same feed code unless changed.
- Corners can be chamfered with Cnn.nnnn and rounded with Rnn.nnnn to define the radius of the arc.

G02 (cw) and G03 (ccw) Circular Interpolation

F Feed rate in inches (mm) per minute
I Optional distance along X-axis to center of circle
J Optional distance along Y-axis to center of circle
K Optional distance along Z-axis to center of circle
R Optional radius of circle
X Optional X-axis motion command
Y Optional Y-axis motion command
Z Optional Z-axis motion command
A Optional A-axis motion command
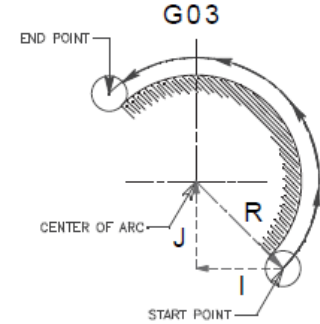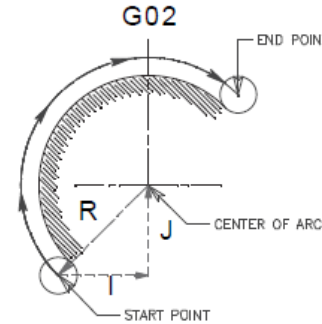
What's needed...
1) Plane selection
2) Arc start position coordinates
3) Rotation direction
4) Arc end position coordinates
5) Arc center coordinates or arc radius.

Image courtesy Haas
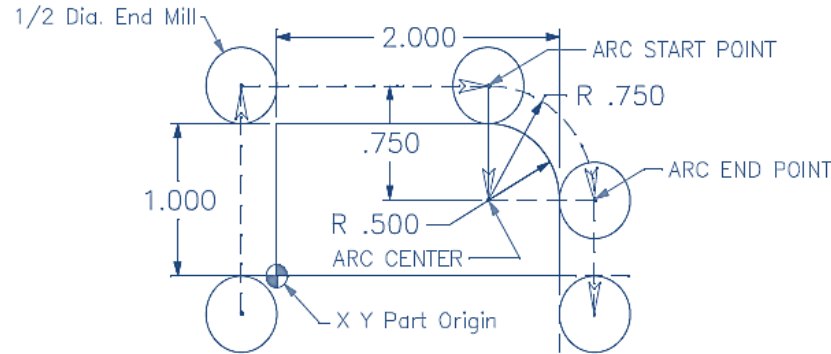
Example of G02

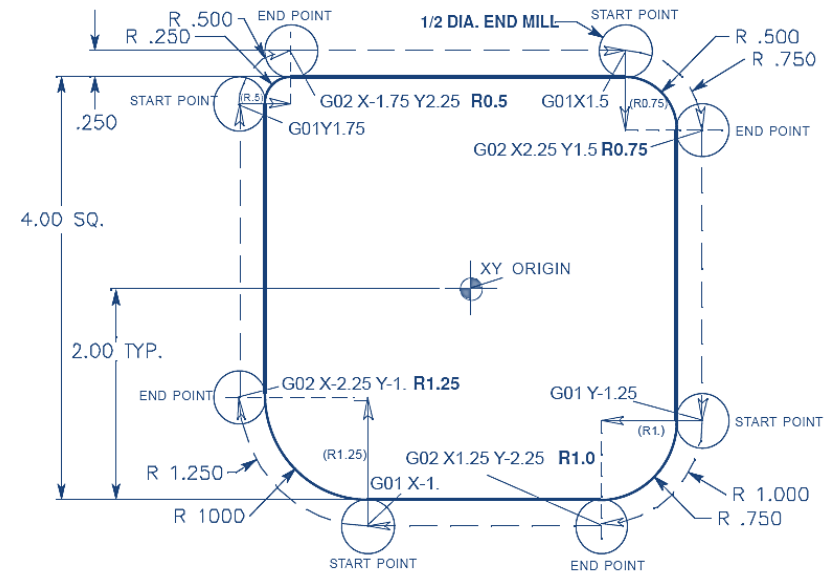(No cutter compensation – need to account for tool diameter)
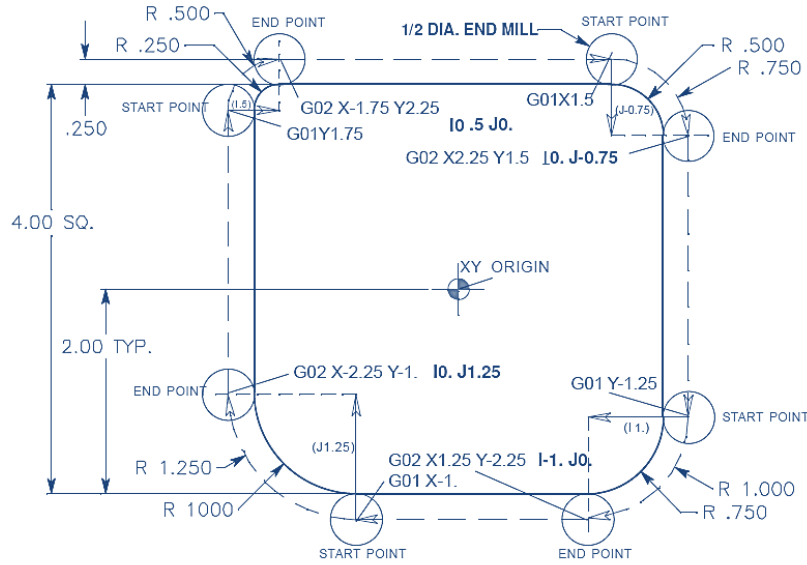N6 G01 Y1.25 F12.   (to start point in Y axis)
N7 X1.500   (to start point in X axis)
N8 G02 X2.25 Y0.5 (I0. J-0.75 or R0.75) (G02 using I J or Radius)
N9 G01 Y-0.25

Prof. Steven S. Saliterman   Image courtesy Haas

Examples of G02 with I & J vs R
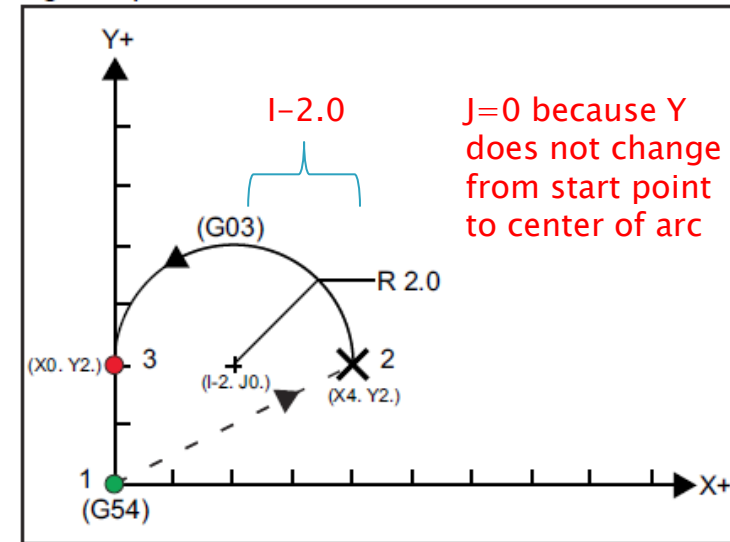
Images courtesy Haas

Example of G03

Method 1:
```
%
T01 M06 ;

...
G00 X4. Y2. ;
G01 F20.0 Z-0.1 ;
G03 F20.0 I-2.0 J0. X0. Y2. ; (see drawing)

...
M30 ;
%
```
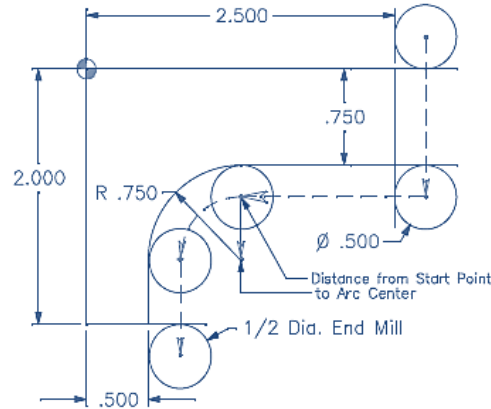
Method 2:
```
%
T01 M06 ;

...
G00 X4. Y2. ;
G01 F20.0 Z-0.1 ;
G03 F20.0 X0. Y2. R2. ;      (End point and radius)
...M30 ;
%
```


I-2.0

J=0 because Y does not change from start point to center of arc

Image courtesy Haas

Example of G03

(No tool compensation – need to allow for tool diameter.)
N6 G01 Y−1.0 F12.   (to start point in Y axis)
N7 X1.250   (to start point in X axis)
N8 G03 X0.750 Y−1.500 (I0. J−0.5 or R0.5)   (G03 using I J or Radius)
N9 G01 Y−2.250

Image courtesy Haas

Cutter Compensation

- Cutter compensation is used to offset the center of the cutter, and shift it the distance of the radius, to the specified side of the programmed path.
- Complex part geometries having angled lines, lines tangent to arcs, and lines intersecting arcs involve substantial trigonometric computations to determine the center of the cutter.
- Cutter compensation involves programming the part geometry directly instead of the tool center.

G41, G42 & G40 Cutter Compensation

1. **G40** Cutter Compensation Cancel
   1. G40 will cancel the G41 or G42 cutter compensation commands.
   2. Programming D00 will also cancel cutter compensation.
   3. Always cancel will moving away from the part.
2. **G41** Cutter Compensation Left – Climb Cutting Standard Right Hand Tool – Use this for CNC
   1. Tool is moving to the left of the programmed path to compensate for the radius of the tool.
   2. A Dnn must also be programmed to select the correct tool size from the Tool Offset Register (D01= Diameter Offset #1, D2 = #2, etc ..)
   3. The number to use with D is in the far-left column of the tool offsets table. The value that the control uses for cutter compensation is in the GEOMETRY column under D (if Setting 40 is DIAMETER) or R (if Setting 40 is RADIUS).

3. **G42** Cutter Compensation Right – Conventional Cutting, Standard Right Hand Tool
   1. Tool is moving to the right of the programmed path to compensate for the size of the tool.
   2. A Dnn must also be programmed to select the correct tool size from the Tool Offset Register (D01= Diameter Offset #1, D2 = #2, etc ..)

Cutter Compensation Example

```
(Program without cutter compensation.)
N104 G00 X-2.5 Y-2.0
N105 G01 Z-0.45 F50.
N106 X-2.25 F12.                    (feed is in IPM)
N107 Y1.75
N108 G02 X-1.75 Y2.25 R0.5
N109 G01 X1.5
N110 G02 X2.25 Y1.5 R0.75
N111 G01 Y????                     (Calculate point Y-1.25)
N112 X???? Y-2.25                  (Calculate point X-.75)
N113 G01 X-1.75
N114 G02 X-2.25 Y-1.75 R0.5
N115 G01 X-2.35 Y-2.0

(Program with cutter compensation.
Dia. value for D01 would be .500
in DIAMETER offset register #1.)
N104 G00 X-2.35 Y-2.0
N105 G01 Z-0.45 F50.
N106 G41 X-2. D01 F12.             (turn on C.C. with an X and/or Y move)
N107 Y1.75
N108 G02 X-1.75 Y2. R0.25
N109 G01 X1.5
N110 G02 X2. Y1.5 R0.5
N111 G01 Y-1.
N112 X-0.75 Y-2.
N113 X-1.75
N114 G02 X-2. Y-1.75 R0.25
N115 G40 G01 X-2.35               (turn off C.C. with an X and /or Y move)
```
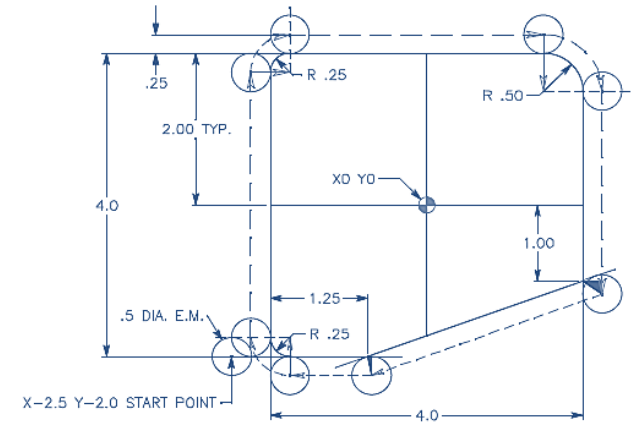

.25
R .25
R .50
2.00 TYP.
X0 Y0
4.0
1.00
.5 DIA. E.M.
R .25
1.25
X-2.5 Y-2.0 START POINT
4.0

Prof. Steven S. Saliterman

Image courtesy Haas

Another Compensation Example

```
O40006
                        (G54 X0 Y0 is at the lower left of part corner) ;
                        (Z0 is on top of the part) ;
                        (T1 is a .250 diameter of end mill) ;

(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) ;
```
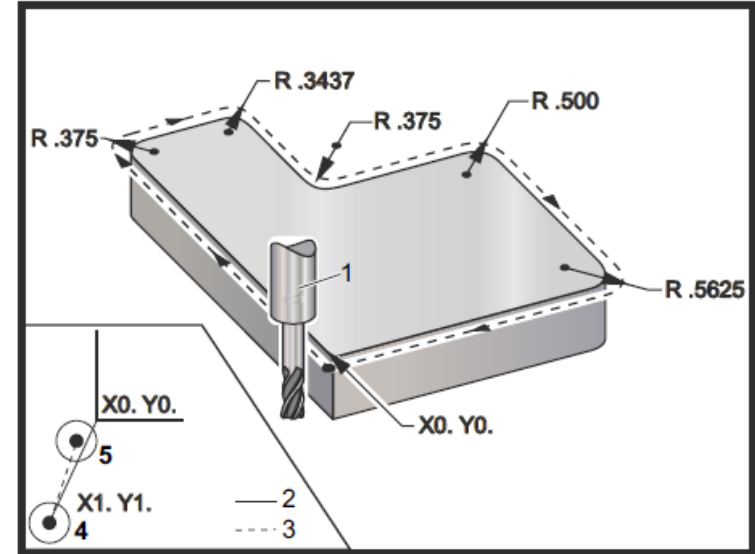

Circular Interpolation G02 and G03: [1] 0.250" diameter end mill, [2] Programmed path, [3] Center of Tool, [4] Start Position, [5] Offset Tool Path.

Image courtesy Haas

Canned Cycles

- G80   Cancels a canned cycle.
- G81   Drill
- G82   Spot drill
- G83   Peck drilling
- G84   Tap cycle

G81 Drill Canned Cycle

```
Nl T1 M06
N2 G90 G54 00 X.3 Y.3
N3 S1200 M03
N4 G43 H01 Z1. M08
NS G81 Z-.6 R.1 F10.
N6 X1.2 Y1.2
N7 G80 G00 Z1. M09
N8 G28 G91 Z0. M05
N9 M30
```
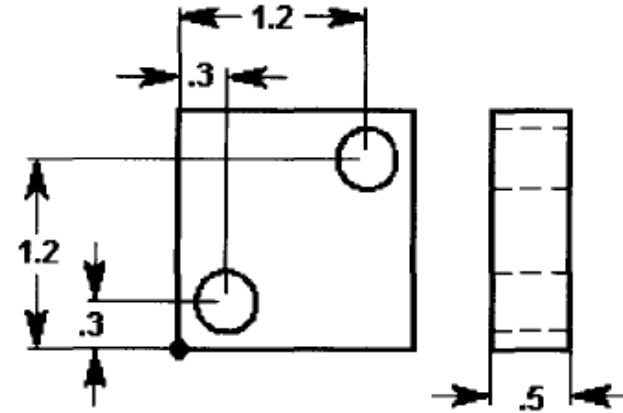

F Feed Rate in inches (mm) per minute
R Position of the R plane
X Optional X-axis motion command
Y Optional Y-axis motion command
Z Position of bottom of hole

Prof. Steven S. Saliterman

Image courtesy Haas
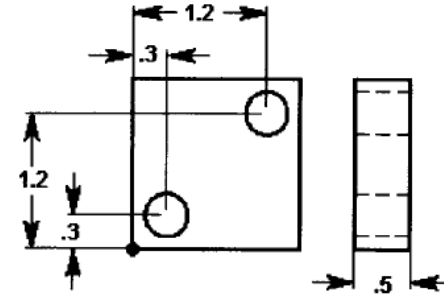
G82 Spot Drill Canned Cycle

```
Nl T1 M06
N2 G90 G54 G00 X.3 Y.3
N3 S1200 M03
N4 G43 H01 Z1. M08
NS G82 Z-.125 P1.5 R.1 F10.
N6 X1.2 Y1.2
N7 G80 G00 Z1. M09
N8 G91 G28 Z0. M05
N9 M30
```
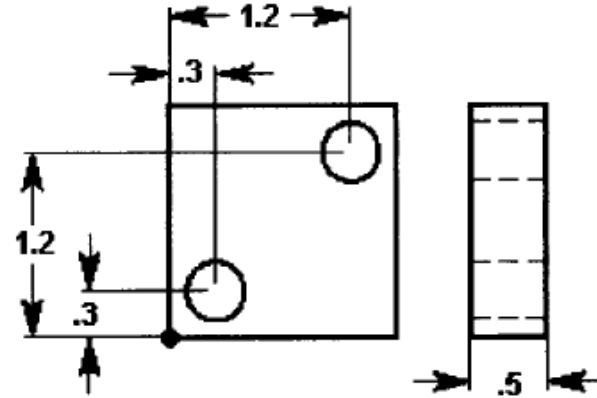
F Feed Rate in inches (mm) per minute
P The dwell time at the bottom of the hole in seconds
R Position of the R plane
X Optional X-axis motion command
Y Optional Y-axis motion command
Z Position of bottom of hole

Image courtesy Haas

G83 Peck Drilling Canned Cycle

F Feed Rate in inches (mm) per minute
Optional size of first cutting depth
J Optional amount to reduce cutting depth each pass
K Optional minimum depth of cut
Q The cut-in value, always incremental
R Position of the R plane
X Optional X-axis motion command
Y Optional Y-axis motion command
Z Position of bottom of hole


```
N1 T1 M06
N2 G90 G54 G00 X.3 Y.3
N3 S1200 M03
N4 G43 HO1 Z1. M08
NS G83 Z-.6 Q.10 R.1 F10.
N6 X1.2 Y1.2
N7 G80 G00 Z1. M09
N8 G91 G28 Z0. M05
N9 M30
```

Image courtesy Haas

G84 RH Threads

```
N1 T1 M06
N2 G90 G54 G00 X.3 Y.3
N3 S1200 (you do not need
to turn spindle on)
N4 G43 H01 Z1. M08
NS G84 Z-.85 R.1 F60.
N6 X1.2 Yl.2
N7 G80 G00 Z1. M09
N8 G91 G28 Z0. MO5
N9 M30
```
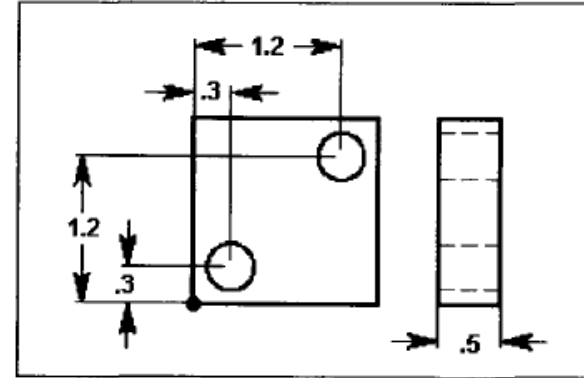

F Feed Rate in inches (mm) per minute
R Position of the R plane
X Optional X-axis motion command
Y Optional Y-axis motion command
Z Position of bottom of hole

Image courtesy Haas

Subroutines

- Commonly used for hole patterns. For example, the subroutine may contain only the X–Y coordinates, while the main program includes drilling, tapping and chamfering.
- M97 is local, with a P code defining the starting block line number. (M98 is a separate program.)
- M99 returns you to the main program – the next line after the subroutine was called.
- May contain an L (loop). If there is an Ln with the subroutine call, it is repeated n times before the main program begins with the next block.

Example of M97 Subroutine

```
O0300 (example local subroutine);
T1 M06;
G90 G54 G00 X.5 Y.5;
S2000 M03;
G43 Z1.0 H01;
G81 Z-0.1 R0.1 F20 (define canned cycle);
M97 P0500 (call local subroutine using
sequence  500);
T02 M06 (peck drill);
G90 G54 G00 X.5 Y.5;
S1000 M03;
G43 Z1. H02 M08;
G83 R0.l Z-1. F10. (define canned cycle);
M97 P0500 (call local subroutine using
sequence number 500);
G28 Y0.;
```

```
M30 (end of main program);
N0500 (local subroutine example
listing all hole positions);
Xl.5;
X2.5;
Y1. X2.;
X1.;
X0.5 Yl.5;
X1.5;
X2.5;
G80 G00 Z1. M09;
G91 G28 Z0. M05;
M99 (end of local subroutine);
```

Summary

- Safety
- Coordinate system and work offset.
- Letter, "G" and "M" codes.
- Formulas.
- Safety line and sample program.
- Climbing vs. conventional milling.
- G commands:
  - G00, G01, G02 and G03.
- Cutter compensation:
  - G41, G42 and G40.
- Canned cycles:
  - G81, G82, G83 and G84.
- Subroutines

References

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