

THE EXAMPLES CONTAINED WITHIN THIS DOCUMENT ARE BASED ON OUR EXPERIENCE WITH FANUC STYLE CONTROLS. THE HAAS LATHE MANUALS DO A FAIR JOB AT EXPLAINING BASIC PROGRAMMING WITHOUT BEING OVERLY COMPLEX AND HARD TO FOLLOW. G & M CODES MAY VARY FROM MACHINE TO MACHINE.

LINK TO THE HAAS MANUAL ARCHIVE

<https://www.haascnc.com/service/search-results.manual.html#contentType%3DInstruction%20Manual>

CANNED CYCLE PROGRAMS AND PROGRAMMING IN GENERAL CAN BE RELIANT ON SETTINGS AND PARAMETERS IN YOUR CONTROL. ALL PROGRAMS SHOULD BE RUN IN GRAPHICS FIRST, FOLLOWED BY CUTTING AIR WITH RAPIDS AT 5%. THE BEST WAY TO LEARN THIS TRADE IS BY DOING IT, THESE SAMPLES ARE PROVIDED TO HELP ACCELERATE THE LEARNING CURVE.

WE STRING PROGRAMS TOGETHER BY COPY AND PASTING MANY OF THESE SAMPLES MODIFYING AS NEEDED. AT THE END OF THE DAY, ALL PROGRAMMING IS SIMPLY STRINGING TOGETHER EITHER LINEAR MOVES OR ARCS IN COMBINATION. IT IS POSSIBLE TO HANDCODE NEARLY ANYTHING, BUT A GOOD CAD/CAM SYSTEM OR CONVERSATIONAL CONTROL IS HARD TO BEAT. KNOWING WHAT THE CODE IS GOING TO DO BEFORE PRESSING CYCLE START IS CRITICAL THOUGH, THERE WILL ALWAYS BE A NEED TO KNOW HOW TO DO BOTH.

IT IS A GOOD PRACTICE TO BEGIN WITH A SAFETY LINE OF CODE. SAFETY LINES ARE ADDED AT THE BEGINNING OF A PROGRAM OR ANY POINT IN A PROGRAM WHERE A TOOL COULD POTENTIALLY CRASH DUE TO ACTIVE COMMANDS. SAFETY LINES VARY BASED ON THE NEEDS OF THE PROGRAMMER, THE TYPE OF MACHINE, AND THE PROGRAM.

**SAFETY LINE EXAMPLE- G90 G54 G40 G18 G20 G99 G80**

**G90/G91 - Absolute/Incremental positioning**

**G54 - Workshift datum**

**G90.1/G91.1 - Absolute/incremental distance mode for I, J, and K offsets**

**G40 - Cancel cutter compensation**

**G17 - XY plane selection**

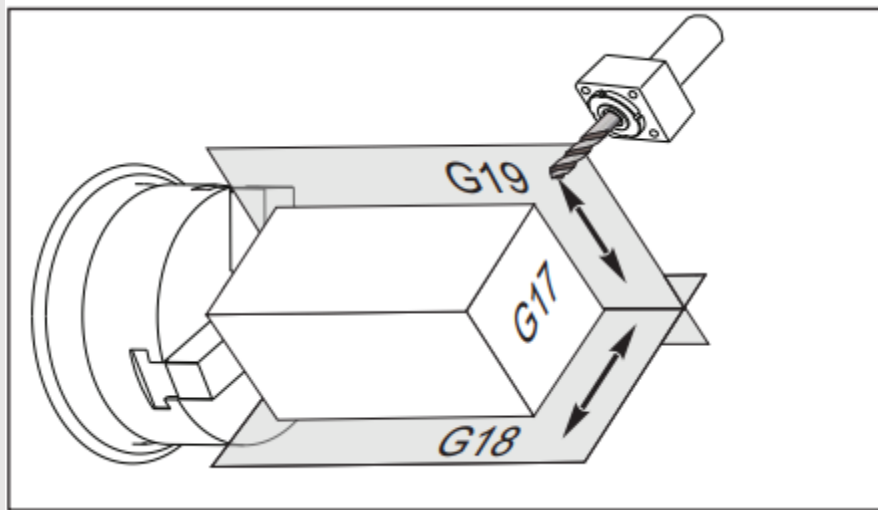
**G18 - ZX plane selection**

**G19 - YZ plane selection**

**G20/G21 - Imperial/metric units**

**G99 - puts the machine in Feed per Rev mode**

**G80 - Cancel any active cycle**




---

**(SIMPLE RADIUS AND FACING PASS EXAMPLE)  
 (1.00" OD STOCK-.02" RADIUS AND FACE TO CENTER WITH A .031" RADIUS INSERT)**

```

T101          (TOOL CALL)
G54           (G54 WORK OFFSET POSITIONING COORDINATE #1)
G50 S1500    (SPINDLE SPEED MAXIMUM RPM LIMIT)
G96 S500 M3   (CONSTANT SURFACE SPEED-CSS ON)
G00 X1.1 Z-.051 (RAPID APPROACH)
G01 X1. F.006 (FEED INTO MATERIAL IN X)
G02 X.898 Z0. R.051 (CLOCKWISE CIRCULAR INTERPOLATION WITH RADIUS VALUE)
G01 X-.0625   (FEED TO BELOW CENTER)
G00 Z.1       (RAPID RETRACT CLEAR OF PART FACE)
G00 X6. Z6.   (RETRACT FOR TOOLCHANGE)
M30          (PROGRAM END AND RESET)
  
```

THE APPROACH MOVE AT Z-.051" IS YOUR DESIRED RADIUS (.020" IN THIS CASE) + YOUR TOOL NOSE RADIUS (.031" IN THIS CASE).

THE (R) VALUE EQUALS YOUR DESIRED RADIUS (.020" IN THIS CASE) + YOUR TOOL NOSE RADIUS (.031" IN THIS CASE).

THE (X) VALUE IN THE G02 LINE EQUALS THE STOCK O.D. (1.00") MINUS .102" (WE SUBTRACT THE TOOL NOSE RADIUS AND DESIRED CORNER RADIUS VALUE TWICE ON THIS LINE)

(WE FINISH THE FACING PASS BY FEEDING TWICE THE TOOL NOSE RADIUS BELOW CENTER, OTHERWISE A NUB MAY REMAIN) (A .015R WOULD REQUIRE X-.03 TO CLEAN UP AS AN EXAMPLE)

---

---

**(SIMPLE RADIUS AND O.D. TURN EXAMPLE)**  
**(1.00" OD STOCK-.02" RADIUS WITH A .031" RADIUS INSERT)**  
**(THIS CYCLE WILL FEED INTO THE FACE, RADIUS, AND FEED TOWARD THE CHUCK)**

T101 (TOOL CALL)  
G54 (G54 WORK OFFSET POSITIONING COORDINATE #1)  
G50 S1500 (SPINDLE SPEED MAXIMUM RPM LIMIT)  
G96 S500 M3 (CONSTANT SURFACE SPEED-CSS ON)  
G00 X.898 Z.05 (RAPID APPROACH)  
G01 Z0. F.006 (FEED INTO MATERIAL IN X)  
G03 X1. Z-.051 R.051 (CCW CIRCULAR INTERPOLATION WITH RADIUS VALUE)  
G01 Z-1. (FEED TOWARD CHUCK)  
G01 X1.1 F.02 (HIGH FEED AWAY FROM STOCK)  
G00 X6. Z6. (RAPID RETRACT FOR TOOLCHANGE)  
M30 (PROGRAM END AND RESET)

---

**(USING A PROGRAM LOOP TO RUN X NUMBER OF PARTS WITH A BAR PULLER)**  
**(THE FOLLOWING CAN BE USED IN CONJUNCTION)**

**O00001**  
**(MASTER PROGRAM LOOP)**  
**(WHEN THIS PROGRAM IS RUN IT WILL PROCESS THE CODE FROM WHATEVER**  
**P/PROGRAM YOU SPECIFY)**

M98 P1 L75  
M30

(M98=SUBPROGRAM CALL)  
(P=PROGRAM NUMBER)  
(L=LOOPS/HOW MANY TIMES TO RUN)  
(M30=END OF PROGRAM AND RESET)

### **(BAR PULLER SAMPLE PROGRAM)**

(YOUR PROGRAM WOULD INCLUDE THE FOLLOWING TOOLPATH-THE PROGRAM WOULD BE RUN FROM THE MASTER LOOP PROGRAM TO REPEAT THE BAR PULL X NUMBER OF TIMES) (BAR PULLER Z OFFSET IS SET TO THE FURTHEST POSITION ON THE TOOL WHICH WILL CLEAR THE END OF THE STOCK) (THIS REDUCES THE LIKELIHOOD OF PROGRAMMING OR SETUP ERRORS)

T1616 (BAR PULLER PULLS STOCK BACK TO G54 Z0.)  
G54 G54 (WORK OFFSET POSITIONING COORDINATE #1)  
M05 (SPINDLE STOP)  
G98 (G98 FEED PER MINUTE)  
M14 (MAIN SPINDLE CLAMP)  
G00 X6. (RAPID CLEARANCE ON X WELL ABOVE PART)  
G01 Z-1.5 F250. (HIGH FEED TO PULL POSITION-WE USE THE PARTOFF Z POSITION)

(USING A FEED MOVE ALLOWS THE RAPIDS TO BE TURNED DOWN FOR PROOFING)

G01 X0. F100. (FEED MOVE IN X ONTO STOCK)  
M11 (CHUCK UNCLAMP)  
G04 P1. (DWELL)

G01 Z0.015 F150. (FEED BACK TO ZERO-.015 STOCK TO FACE FROM NEXT PART)  
M10 (CHUCK CLAMP)  
G01 X6. (FEED OFF PART)  
M15 (UNCLAMP SPINDLE)  
G99 (SWITCH BACK TO FEED PER REVOLUTION)  
G00 Z6. X6. (RETRACT MOVE)  
M99 (SUB PROGRAM/RETURN OR LOOP)  
M30 (PROGRAM END AND RESET)

---

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

**( C-AXIS LIVE TOOL-PECK DRILLING THE FACE OF A PART SAMPLE )**

T101 (TOOL CALL)  
G54 (WORK OFFSET POSITIONING COORDINATE #1)  
G98 (G98 FEED PER MINUTE)  
M08 (COOLANT ON)  
M154 (C AXIS ENGAGE)  
G97 M133 P500 (M133-ENGAGE LIVE TOOLING DRIVE FORWARD-G97-CONSTANT RPM - P = RPM)  
G00 C84. (C AXIS POSITION)  
M14 (HAAS SPINDLE CLAMP-PARAMETER 304 SETS DELAY 500 FROM FACTORY)  
G17 (CIRCULAR MOTION XYZ PLANE SELECTION LIVE TOOLING)  
G00 X7.5 Z0.25 (APPROACH MOVE)  
**G98 G83 Z-0.75 R0.1 Q0.1 P0.25 F10.** (G83 PECK DRILL CANNED CYCLE)  
C129. (REMAINING HOLES SIMPLY ADD MORE C POSITIONS-ADD X VALUE ON SAME LINE FOR DIFFERENT DIAMETERS)  
C174.  
C219.  
C264.  
C309.  
C354.  
C399.  
G80 (CANNED CYCLE CANCEL)  
M09 (COOLANT OFF)  
M135 (LIVE TOOL DRIVE STOP)  
M155 (C AXIS DISENGAGE)  
M89 (HIGH PRESSURE COOLANT OFF)  
G0 X6. Z6.  
M30

---

---

**(C-AXIS AXIAL/FACE- LIVE TOOLING HIGH SPEED PECK DRILLING SAMPLE- MICRO PECKS)**

(HAAS SETTING 22 CONTROLS PECK RETRACT)  
(THIS TOOLPATH IS EXTREMELY SIMPLE BUT VERY EFFECTIVE)

(G54 WORK OFFSET POSITIONING COORDINATE #1)  
(G98 FEED PER MINUTE)  
(WORK OFFSET POSITIONING COORDINATE #1)  
(M154 C AXIS ENGAGE)  
(M133-ENGAGE LIVE TOOLING DRIVE FORWARD-G97-CONSTANT RPM - P = RPM)  
(M14 HAAS SPINDLE CLAMP-PARAMETER 304 SETS DELAY- 500 FROM FACTORY)  
(K = PECK INTERVAL)  
(R = CLEARANCE POINT IN Z)  
(G74 FACE GROOVING OR HIGH SPEED PECK DRILL CYCLE)  
(G17 CIRCULAR MOTION XYZ PLANE SELECTION LIVE TOOLING)  
(F = FEEDRATE IN INCHES PER MINUTE)  
(G80 CANNED CYCLE CANCEL)  
(M09 COOLANT OFF)  
(M135 LIVE TOOL DRIVE STOP)  
(M155 C AXIS DISENGAGE)  
(M89 HIGH PRESSURE COOLANT OFF)

T101  
G54  
G98  
M08  
M154  
G97 M133 P500  
G00 C0.  
M14  
G17  
G00 X4. Z0.1  
G98 **G74** Z-2. R0.1 K0.05 F10.  
C120.  
G98 **G74** Z-2. R0.1 K0.05 F10.  
C240.  
G98 **G74** Z-2. R0.1 K0.05 F10.  
G80  
M09  
M135  
M155  
M89  
G00 Z7.  
G00 X8.  
M30

---

---

**(C-AXIS RADIAL/O.D. - LIVE TOOLING HIGH SPEED PECK DRILL SAMPLE- MICRO PECKS)**  
(HAAS SETTING 22 CONTROLS PECK RETRACT)

(G54 WORK OFFSET POSITIONING COORDINATE #1)  
(G19 CIRCULAR MOTION YZ PLANE SELECTION LIVE TOOLING)  
(M154 C AXIS ENGAGE)  
(M14 HAAS SPINDLE CLAMP-PARAMETER 304 SETS DELAY- 500 FROM FACTORY)  
(M15 HAAS SPINDLE UNCLAMP)  
(M133-ENGAGE LIVE TOOLING DRIVE FORWARD-G97-CONSTANT RPM - P = RPM)  
(G75 PECK GROOVING CYCLE O.D. OR I.D.-BEING USED TO DRILL IN THIS SAMPLE)  
(I = PECK INTERVAL)  
(G04 DWELL P = SECONDS)  
(G98 FEED PER MINUTE)  
(F = FEEDRATE IN INCHES PER MINUTE)  
(G80 CANNED CYCLE CANCEL)  
(M09 COOLANT OFF)  
(M135 LIVE TOOL DRIVE STOP)  
(M155 C AXIS DISENGAGE)

T101  
G19  
G98  
G54  
M154  
M133 P1000  
G00 X5. C36. (HOLE 1) Z-1. (APPROACH MOVE X = RETRACT POINT)  
G04 P0.5  
M14  
**G19 G75 X4. I0.05 F1.5**  
M15  
G00 P0.5  
(COPY PASTE BLOCK FROM HERE TO ADD MORE HOLES)  
G00 C126. (HOLE 2)  
G04 P0.5  
M14  
**G19 G75 X4. I0.05 F1.5**  
M15  
G04 P0.5  
G00 C216. (HOLE 3)  
G04 P0.5  
M14  
**G19 G75 X4. I0.05 F1.5**  
M15  
G04 P0.5  
G00 C306. (HOLE 4)  
G04 P0.5  
M14  
**G19 G75 X4. I0.05 F1.5**  
M15  
G80  
M135  
M09  
G00 X10. Z6.  
M30

---

**(C AXIS FACE TAP CYCLE 1/4-20 EXAMPLE 4 HOLES)**

T101  
G54  
G99  
M154  
G00 C0.  
G00 X6.15  
G00 Z0.25  
S250  
G95 Z-1. R0.25 F0.05  
G00 C90.  
G00 X6.15  
G00 Z0.25  
S250  
G95 Z-1. R0.25 F0.05  
G00 C180.  
G00 X6.15  
G00 Z0.25  
S250  
G95 Z-1. R0.25 F0.05  
G00 C270.  
G00 X6.15  
G00 Z0.25  
S250  
G95 Z-1. R0.25 F0.05  
G28 U0  
G28 W0  
G28  
M135  
M155  
M30

---

**(M6 X 1 LIVE TOOL TAP ON CENTER SAMPLE)**

(SPINDLE STATIONARY)  
T1919  
G54  
G99  
G00 X0.  
G00 Z0.25  
S250  
M19 P0.  
M14  
G95 Z-0.75 R0.25 F0.03937  
M09  
M15  
G00 X4. Z4.  
M00

---

---

**(PECK TAP 3/8-16 SAMPLE USING SPINDLE-TAP IS STATIONARY)**

(EACH -Z MOVEMENT IS A PECK)

T1313  
G54  
M08  
G97 S80  
G00 X0.  
Z0.2  
G84 X0. Z-0.25 R0.2 F0.0625  
X0. Z-0.5  
X0. Z-0.75  
X0. Z-1.  
G80  
M09  
G00 Z4.  
X6.  
M00

---

**(INSERT DRILL USING MAIN SPINDLE SAMPLE)**

T101 (TOOL CALL)  
G54 (G54 WORK OFFSET POSITIONING COORDINATE #1)  
G50 S2000 (SPINDLE SPEED MAXIMUM RPM LIMIT)  
G97 S1500 M03 (G97-CONSTANT RPM / S = RPM/ M3 = SPINDLE ON FORWARD)  
G00 X0. Z0.1 (APPROACH MOVE)  
M88 (HIGH PRESSURE COOLANT ON)  
G01 Z.01 F.0068 (FEED APPROACH)  
G01 Z-0.1 F0.001 (SLOW FEED TO ALLOW DRILL TO STABILIZE)  
G01 Z-6. F0.0068 (RESUME NORMAL FEED RATE)  
G00 Z0.1 (RAPID RETRACT/USE G01 FOR A FINER FINISH)  
M05 (SPINDLE STOP)  
M89 (HIGH PRESSURE COOLANT OFF)  
Z4.  
X6.  
M30 (PROGRAM END AND RESET)

---



---

**(CHIP BREAK TURNING CANNED CYCLE)**

(EXAMPLE TURNING STOCK FROM 4.000 TO 3.500 .100 DOC)  
(PASSES WILL BE MADE AT 3.800D/3.600D/3.500D)

```
T202
G54
G97 S750 M03
G00 X3.8 (APPROCH DIAMETER THIS IS THE FIRST PASS)
Z0.1
G74 X3.5 Z-0.5 D0.1 I0.1 K0.1 F0.006
G80
G00 Z0.1
M30
```

(G74 OD/ID PECK GROOVING CYCLE)

(X3.5 VALUE = TARGET TURNED DIAMETER)

(D VALUE DISTANCE TO RETRACT IN X+ BEFORE RETRACT IN Z) (I VALUE = DEPTH OF CUT)

(K VALUE = DISTANCE BETWEEN MICRO RETRACTS)

(F VALUE = FEEDRATE)

(HAAS SETTING 22 CONTROLS MICRO RETRACT DISTANCE)

(DO NOT MAKE THE 3.8 AND 3.5 NUMBERS IDENTICAL!!!!!! )

(THIS CAUSES THE D VALUE TO FEED X- INSTEAD OF X+!!!!!!!!!!!!!!)

---

**CANNED CYCLES CAN BE USED FOR THREADING CYCLES, BUT WE PREFER NOT TO USE THEM. G32 PROGRAMMING IS VERY EASY TO UNDERSTAND AND DOES NOT RELY ON SETTINGS IN THE CONTROL TO MANIPULATE. G32 CAN BEGIN AS A STRAIGHT THREAD AND TRANSITION TO A TAPER, AND BACK TO STRAIGHT.**

**(G32 THREAD CHAMFER OUT SAMPLE WITH SPRING PASSES)**

(THIS IS THE KIND OF CYCLE I WRITE FOR LOW VOLUME WORK, WHEN GETTING THE JOB DONE TROUBLE FREE IS THE PRIORITY OVER SPEED)

(SPRING PASSES MAKE CUTTING GENTLE)

N5

(1.146 OVER .072 WIRES)

( 1-1/8-8N THREAD )

T2020

G18

M31

M08

G97 S764 M03

G00 G54 X1.3229 Z0.2

X1.0959

G32 Z-2.625 E0.125

X1.25 Z-2.75

G00 X1.3229

Z0.2

X1.0774

G32 Z-2.625 E0.125

X1.25 Z-2.75

G00 X1.3229

Z0.2

X1.0623  
G32 Z-2.625 E0.125  
X1.25 Z-2.75  
G00 X1.3229  
Z0.2

X1.0494  
G32 Z-2.625 E0.125  
X1.25 Z-2.75  
G00 X1.3229  
Z0.2

X1.0378  
G32 Z-2.625 E0.125  
X1.25 Z-2.75  
G00 X1.3229  
Z0.2

(SPRING PASSES AFTER EVERY CUT DEPTH BEGIN-A SURPRISING AMOUNT OF MATERIAL IS REMOVED ON THE SPRING PASSES WHICH GREATLY REDUCES TOOL PRESSURE)

X1.0272  
G32 Z-2.625 E0.125  
X1.25 Z-2.75  
G00 X1.3229  
Z0.2

X1.0272  
G32 Z-2.625 E0.125  
X1.25 Z-2.75  
G00 X1.3229  
Z0.2

X1.0174  
G32 Z-2.625 E0.125  
X1.25 Z-2.75  
G00 X1.3229  
Z0.2

X1.0174  
G32 Z-2.625 E0.125  
X1.25 Z-2.75  
G00 X1.3229  
Z0.2

X1.0082  
G32 Z-2.625 E0.125  
X1.25 Z-2.75  
G00 X1.3229  
Z0.2

X1.0082  
G32 Z-2.625 E0.125  
X1.25 Z-2.75  
G00 X1.3229  
Z0.2

X0.9996  
G32 Z-2.625 E0.125  
X1.25 Z-2.75

G00 X1.3229  
Z0.2

X0.9996  
G32 Z-2.625 E0.125  
X1.25 Z-2.75  
G00 X1.3229  
Z0.2

X0.9914  
G32 Z-2.625 E0.125  
X1.25 Z-2.75  
G00 X1.3229  
Z0.2

X0.9914  
G32 Z-2.625 E0.125  
X1.25 Z-2.75  
G00 X1.3229  
Z0.2

X0.9836  
G32 Z-2.625 E0.125  
X1.25 Z-2.75  
G00 X1.3229  
Z0.2

X0.9836  
G32 Z-2.625 E0.125  
X1.25 Z-2.75  
G00 X1.3229  
Z0.2

X0.9761  
G32 Z-2.625 E0.125  
X1.25 Z-2.75  
G00 X1.3229  
Z0.2

X0.9761  
G32 Z-2.625 E0.125  
X1.25 Z-2.75  
G00 X1.3229  
Z0.2

X0.9741  
G32 Z-2.625 E0.125  
X1.25 Z-2.75  
G00 X1.3229  
Z0.2

X0.9741  
G32 Z-2.625 E0.125  
X1.25 Z-2.75  
G00 X1.3229  
Z0.2

M09  
G00 X6. Z6.  
M30

---

## CANNED CYCLE EXAMPLES

Using the following examples, simple programs can be created by copy/pasting as needed. These cycles are easy to modify and understand. The U-X and W-Z values can be modified for stock to leave when roughing/finishing. Negative U values = more off the O.D. and less off the I.D. Negative W values = more off the faces of a part.

---

### **(OD TURN)**

```
T101
G54
G50 S1500
G96 S450 M03
G00 X1.075
G00 Z0.05
G71 P101 Q102 U0 W0 D0.05 F0.006
N101 G00 X0.5
G01 X0.5 Z-1.
N102 G01 X1.075
G00 X1.075 Z0.05
M00
```

---

### **(OD TURN WITH .25 CORNER FILLET)**

```
T101
G54
G50 S1500
G96 S450 M03
G00 X1.075
G00 Z0.05
G71 P101 Q102 U0 W0 D0.05 F0.006
N101 G00 X0.5
G01 X0.5 Z-0.781
G02 X0.938 Z-1. R0.219
N102 G01 X1.075 Z-1.
G00 X1.075 Z0.05
M00
```

---

---

**(ID TURN)**

T101

G54

G50 S1500

G96 S450 M03

G00 X0.45

G00 Z0.05

G71 P101 Q102 U0 W0 D0.05 F0.006

N101 G00 X1.

G01 Z-1.

N102 G01 X0.45

G00 X0.45 Z0.05

M00

---

**(ID TURN WITH .250 CORNER FILLET)**

T101

G54

G50 S1500

G96 S450 M03

G00 X0.45

G00 Z0.05

G71 P101 Q102 U0 W0 D0.05 F0.006

N101 G00 X1.

G01 X1. Z-0.781

G03 X0.562 Z-1. R0.219

N102 G01 X0.45 Z-1.

G00 X0.45 Z0.05

M00

---

---

**(END FACE - PASSES TOWARD CENTERLINE)**

T101

G54

G50 S1500

G96 S450 M03

G00 X1.05

G00 Z0.05

G72 P101 Q102 U0 W0 D0.05 F0.006

N101 G00 Z-1.

G01 X-0.0625

N102 G01 X-0.0625 Z0.05

G00 X1. Z0.

M00

---

**(END FACE - PASSES TOWARD O.D.)**

**(BORING BAR I.D. PASSES IN X RATHER THAN Z)**

**(.500 BORE OPENED UP TO 1.00 BY 1.00 DEEP IN Z)**

T101

G54

G50 S1500

G96 S450 M03

G00 X.4

G00 Z0.05

G72 P101 Q102 U0 W0 D0.05 F0.006

N101 G00 Z-1.

G01 X1.

N102 G01 X1. Z0.05

G00 X1. Z0.

M00

---

---

**(OD RADIUS .250)**

(1.00 OD)

T101

G54

G50 S1500

G96 S450 M03

G00 X1.05

Z0.05

G71 P101 Q102 U0 W0 D0.05 F0.006

N101 G00 X0.438

G01 Z0.

N102 G03 X1. Z-0.281 R0.281

G00 X1.05 Z0.05

M00

---

**(ID RADIUS .250)**

(1.00 ID)

T101

G54

G50 S1500

G96 S450 M03

G00 X0.95

Z0.05

G71 P101 Q102 U0 W0 D0.05 F0.006

N101 G00 X1.562

G01 Z0.

N102 G02 X1. Z-0.281 R0.281

G00 X1. Z0.

M00

---

---

**(OD CHAMFER .250 X 45°)**

(1.00 OD)

T101

G54

G50 S1500

G96 S450 M03

G00 X1.05

G00 Z0.05

G71 P101 Q102 U0 W0 D0.05 F0.006

N101 G00 X0.438

G01 Z0.

N102 G01 X1. Z-0.281

G00 X1. Z0.

M00

---

**(OD CHAMFER .250 X 60° STEEP)**

(1.00 OD)

T101

G54

G50 S1500

G96 S450 M03

G00 X1.05

G00 Z0.05

G71 P101 Q102 U0 W0 D0.05 F0.006

N101 G00 X0.0266

G01 Z0.

N102 G01 X1. Z-0.281

G00 X1. Z0.

M00

---



**(OD CHAMFER .250 X 30° SHALLOW)**

(1.00 OD)

T101

G54

G50 S1500

G96 S450 M03

G00 X1.05

G00 Z0.05

G71 P101 Q102 U0 W0 D0.05 F0.006

N101 G00 X0.6755

G01 Z0.

N102 G01 X1. Z-0.281

G00 X1. Z0.

M00

---

**(ID CHAMFER .250 X 45°)**

(1.00 ID)

T101

G54

G50 S1500

G96 S450 M03

G00 X0.95

G00 Z0.05

G71 P101 Q102 U0 W0 D0.05 F0.006

N101 G00 X1.562

G01 Z0.

N102 G01 X1. Z-0.281

G00 X0.95 Z0.05

M00

---

---

**(OD THREAD G76 CYCLE)**

**(1" - 8)**

T101

G54

G97 S500 M03

G00 X1.1

Z0.375

G04 P1.

M24

G76 X0.8492 Z-1. K0.0954 I0. D0.02 F0.125

G00 X1.1 Z0.375

M09

M00

---

**(OD GROOVE .250 WIDTH)**

(.125 WIDE TOOL)

(Z-1.00-FURTHEST POINT OF GROOVE)

(Z-.750-CLOSEST POINT OF GROOVE)

T101

G54

G50 S1500

G96 S450 M03

G00 X1.1

G00 Z0.05

G00 X1.1 Z-0.877

G75 X0.505 Z-0.998 K0.115 F0.006

G00 X1.1

G00 Z-0.875

G01 X0.5 F0.006

G01 Z-1.

G01 X1.1

G00 Z0.05

M00

---

---

**(ID GROOVE .250 WIDTH .125 WIDE TOOL)**

(Z-1.00-FURTHEST POINT OF GROOVE)

(Z-.750-CLOSEST POINT OF GROOVE)

T101

G54

G50 S1500

G96 S450 M03

G00 X0.9

G00 Z0.05

G00 X0.9 Z-0.877

G75 X1.495 Z-0.998 K0.115 F0.006

G00 X0.9

G00 Z-0.875

G01 X1.5 F0.006

G01 Z-1.

G01 X0.9

G00 Z0.05

M00

---

**(PART OFF 1.00 OD 1.00 LONG)**

(.125 WIDE INSERT)

T101

G54

G50 S1500

G96 S450 M03

G00 X1.05

G00 Z0.05

G00 X1.05 Z-1.125

G01 X0. Z-1.125 F0.006

G00 X1.05

G00 X1.05 Z0.05

M00

---

(PART OFF WITH PECK 1.00 OD 1.00 LONG)

(.125 WIDE INSERT)

T101

G54

G50 S1500

G96 S450 M03

G00 X1.05

G00 Z0.05

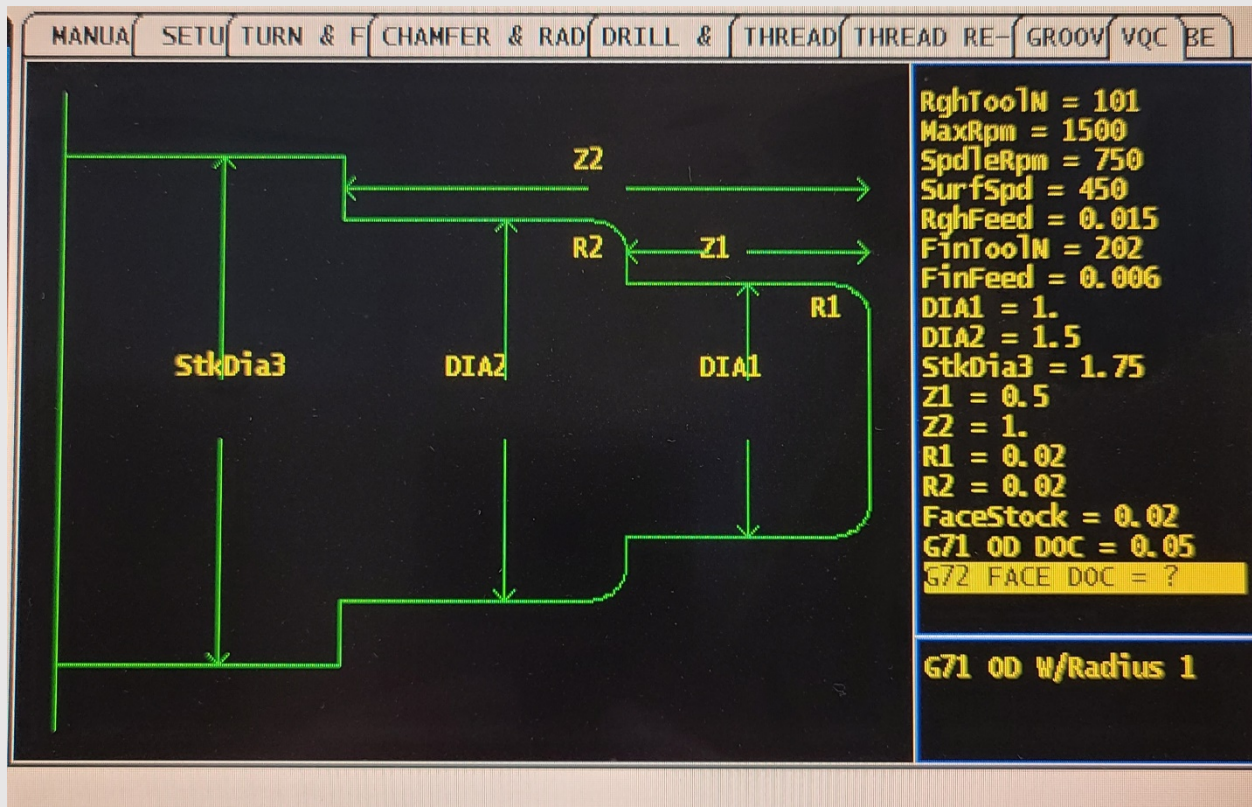
G00 X1.05 Z-1.125

G75 X0. Z-1.125 I0.025 F0.006

G00 X1.05

G00 X1.05 Z0.05

M00



(G71/72 CONTOURING EXAMPLE)

(G71 OD W/Radius 1)

(RghToolN = 101)  
(MaxRpm = 1500)  
(SpdleRpm = 750)  
(SurfSpd = 450)  
(RghFeed = 0.015)  
(FinToolN = 202)  
(FinFeed = 0.006)  
(DIA1 = 1.)  
(DIA2 = 1.5)  
(StkDia3 = 1.75)  
(Z1 = 0.5)  
(Z2 = 1.)  
(R1 = 0.02)  
(R2 = 0.02)  
(FaceStock = 0.02)  
(G71 OD DOC = 0.05)  
(G72 FACE DOC = 0.05)

**(ROUGHING CYCLE)**

T101

G50 S1500

G97 S750 M03

G96 S450 M08

G00 Z2.02

G00 X1.85 Z0.04

**G72** P101 Q102 D0.05 U0.02 W0.004 F0.015

N101 G00 Z0

G01 X-0.07

N102 G00 W0.1

**G71** P103 Q104 D0.05 U0.02 W0.004 F0.015

N103 G00 X0.94

G01 G42 Z0

G01 X0.96

G03 X1. Z-0.02 R0.02

G01 Z-0.5

G01 X1.46

G03 X1.5 Z-0.52 R0.02

G01 Z-1.

G01 X1.85

N104 G40 X1.95

G97 S750 M09

M00

**(FINISH PATH)**

T202

G50 S1500

G97 S750 M03

G96 S450 M08

G00 Z2.02

G00 X1.1 Z0.12

G01 Z0 F0.05

X-0.07 F0.005

G00 X1.85 W0.1

G70 P103 Q104 F0.006

G97 S750 M09

M30

---

# Haas Lathe G & M Codes

## G-Codes

G00	<u>Rapid Position Motion</u>
G01	<u>Linear Interpolation Motion OR Linear Motion, Chamfer and Corner Rounding – Modal</u>
G02	<u>CW Circulation Interpolation Motion – Modal</u>
G03	<u>CCW Circular Interpolation Motion – Modal</u>
G04	<u>Dwell (P) P=Seconds. Milliseconds</u>
G05	<u>Fine Spindle Control Motion (Live Tooling) – Optional</u>
G09	<u>Exact Stop</u>
G10	<u>Programmable Offset Setting</u>
G14	<u>Main-Spindal / Sub-Spindle Swap – Optional</u>
G15	<u>Main-Spindal / Sub-Spindle Swap Cancel – Optional</u>
G17	<u>Circular Motion XYZ Plane Selection Live Tooling (G02, G03) – Modal, Optional</u>
G18	<u>Circular Motion ZX Plane Selection (G02, G03)</u>
G19	<u>Circular Motion YZ Plane Selection Live Tooling (G02, G03) – Modal/Optional</u>
G20	<u>Verify Inch Coordinate Positioning</u>
G21	<u>Verify Metric Coordinate Positioning – Modal</u>
G28	<u>Rapid to machine zero return through Ref. Point</u>
G29	<u>Move to Location Through G29 Ref. Point</u>
G31	<u>Feed Until skip Function – Optional</u>
G32	<u>Thread cutting path – Modal</u>
G40	<u>Tool Nose Compensation Cancel G41/G42</u>
G41	<u>Tool Nose Compensation, Left – Modal</u>
G42	<u>Tool Nose Compensation, Right – Modal</u>
G50	<u>Spindle Speed Maximum RPM Limit (S) – Modal</u>
G51	<u>Rapid to Machine zero, Cancel offset</u>
G52	<u>Work offset Positioning Coordinate OR Global Work Offset Coordinate System Shift – Modal</u>
G53	<u>Machine Zero Positioning Coordinate</u>
G54	<u>Work Offset Positioning Coordinate #1</u>
G55	<u>Work Offset Positioning Coordinate #2 – Modal</u>
G56	<u>Work Offset Positioning Coordinate #3 – Modal</u>
G57	<u>Work Offset Positioning Coordinate #4 – Modal</u>
G58	<u>Work Offset Positioning Coordinate #5 – Modal</u>
G59	<u>Work Offset Positioning Coordinate #6 – Modal</u>
G61	<u>Exact Stop – Modal</u>
G64	<u>Exact Stop G61 Cancel</u>
G65	<u>Macro Sub-Routine Call – Optional</u>
G70	<u>Finishing Cycle</u>

G71	<u>O.D. / I.D. Stock Removal Cycle Example</u>
G72	<u>End Face Stock Removal Cycle</u>
G73	<u>Irregular Path Stock Removal Cycle</u>
G74	<u>Face Grooving OR High Speed Peck Drill Cycle</u>
G75	<u>Peck Grooving Cycle O.D. or I.D.</u>
G76	<u>Threading Cycle, Multiple Pass O.D. / I.D.</u>
G77	<u>Live Tooling Flattening Cycle – Optional</u>
G80	<u>Cancel Canned Cycle</u>
G81	<u>Drill Canned Cycle – Modal</u>
G82	<u>Spot Drill / Counterbore Canned Cycle – Modal</u>
G83	<u>Peck Drill Deep Hole Canned Cycle – Modal</u>
G84	<u>Tapping Canned Cycle – Modal</u>
G85	<u>Bore In-Bore Out Canned Cycle – Modal</u>
G86	<u>Bore In-Stop-Rapid Out Canned Cycle – Modal</u>
G87	<u>Bore In-Stop-Manual Retract Canned Cycle – Modal</u>
G88	<u>Bore In-Dwell-Manual Retract Canned Cycle – Modal</u>
G89	<u>Bore In-Dwell-Bore-out Canned Cycle – Modal</u>
G90	<u>O.D. / I.D. Turning Cycle – Modal</u>
G92	<u>Threading Cycle – Modal</u>
G94	<u>End Facing Cycle – Modal</u>
G95	<u>Live Tooling End Face Rigid Tap – Modal/Optional</u>
G96	<u>Constant Surface speed, CSS On – Modal</u>
G97	<u>Constant Non-Varying Spindle Speed, CSS Off (S)</u>
G98	<u>Feed Per Minute (F) – Modal</u>
G99	<u>Feed Per Revolution (F)</u>
G100	<u>Mirror Image Cancel G101</u>
G101	<u>Mirror Image</u>
G102	<u>Programmable Output to RS-232</u>
G103	<u>Limit Block Lookahead</u>
G105	<u>Servo Bar Command – Optional</u>
G110-G111	<u>Work Offset-positioning Coordinate #7-#8 – Modal</u>
G112	<u>Cartesian to Polar Transformation – Optional</u>
G113	<u>Cartesian to Polar Transformation Cancel – Optional</u>
G114-G129	<u>Work Offset Positioning Coordinate #9-#24 – Modal</u>
G154	<u>Select Work Offset Positioning Coordinate P1-99 (P) – Modal</u>
G159	<u>Background Pickup / Part Return – Optional</u>
G160	<u>APL Axis Command On – Optional</u>
G161	<u>APL Axis Command Off – Optional</u>
G184	<u>Reverse Tapping Canned Cycle – Modal</u>
G186	<u>Live Tooling Reverse Rigid Tap – Optional</u>
G187	<u>Accuracy Control for High Speed Machining (E)</u>
G194	<u>Sub-Spindle / Tapping Canned Cycle – Modal</u>
G195	<u>Live Tooling Radial Tapping – Optional</u>
G196	<u>Live Tooling Radial Tapping Reverse – Optional</u>
G200	<u>Index on the Fly</u>



## M-Codes

M00	Program Stop – Modal
M01	Optional Program Stop – Modal
M02	Program End – Modal
M03	Spindle on Forward (S) – Modal
M04	Spindle on Reverse (S) – Modal
M05	Spindle Stop – Modal
M08	Coolant On – Modal
M09	Coolant Off – Modal
M10	Chuck Clamp – Modal
M11	Chuck Unclamp – Modal
M12	Auto Air Jet On (P) – Modal/Optional
M13	Auto Air Jet Off – Modal/Optional
M14	Main Spindle Clamp – Modal/Optional
M15	Main Spindle Unclamp – Modal/Optional
M17	Rotate Turret Forward (T) – Modal
M18	Rotate turret Reverse (T) – Modal
M19	Orient Spindle – Modal/Optional Example
M21	Tailstock Advance – Modal/Optional
M22	Trailstock Retract – Modal/Optional
M23	Angle Out of Thread On – Modal
M24	Angle of Thread Off – Modal
M25-M28	Optional User M Code Interface with M-Fin Signal – Modal
M30	Program End and Reset – Modal
M31	Chip Auger Forward – Modal
M33	Chip Auger Stop – Modal
M36	Parts Catcher On – Modal/Optional
M37	Parts Catcher Off – Modal/Optional
M38	Specify Spindle Variation On – Modal
M39	Specify Spindle Variation Off – Modal
M41	Spindle Low Gear Override – Modal
M42	Spindle High Gear Override – Modal
M43	Turret Unlock – Modal
M44	Turret Lock – Modal
M51-M58	Optional User M Code Set – Modal
M59	Output Relay Set (N) – Modal
M61-M68	Optional User M Code Clear – Modal
M69	Output Relay Clear (N) – Modal
M76	Program Displays Inactive – Modal
M77	Program Displays Active – Modal
M78	Alarm in Skip Signal Found – Modal
M79	Alarm if Skip signal Not Found – Modal
M85	Automatic Door Open – Modal/Optional

M86	Automatic Door Close – Modal/Optional
M88	High Pressure Coolant ON – Modal/Optional
M89	High Pressure Coolant off – Modal/Optional
M93	Axis Position Capture Start – Modal/Optional
M94	Axis Position Capture Stop – Modal/Optional
M95	Sleep Mode – Modal
M96	Jump if no Signal – Modal
M97	Local Sub-Routine Call – Modal
M98	Sub-Program Call – Modal
M99	Sub-Program / Routine Return or Loop – Modal
M109	Inactive User Input (P) – Modal/Optional
M110	Tailstock Chuck Clamp – Modal/Optional
M111	Tailstock Chuck Unclamp – Modal/Optional
M119	Sub-Spindle Orient – Modal/Optional**
M121-M128	Optional User M Code Interface with M-Fin Signal – Modal
M133	Live tool Drive Forward (P) – Modal/Optional Example
M134	Live Tool Drive Reverse (P) – Modal/Optional
M135	Live Tool Drive Stop – Modal/Optional
M143	Sub-Spindle Forward (P) – Modal/Optional
M144	Sub-Spindle Reverse (P) – Modal/Optional
M145	Sub-Spindle Stop – Modal/Optional
M154	C Axis Engage – Modal/Optional Example
M155	C Axis Disengage – Modal/Optional
M164	Rotate APL Grippers To “n” Position – Modal/Optional
M165	Open APL Gripper 1 (Raw Material) – Modal/Optional
M166	Close APL Gripper 1 (Raw Material) – Modal/Optional
M167	Open APL Gripper 2 (Finished Material) – Modal/Optional
M168	Close APL Gripper 2 (Finished Material) – Modal/Optional