COMPUTER NUMERICAL CONTROL OF MACHINE TOOLS

Laboratory for Manufacturing Systems and Automation
Department of Mechanical Engineering and Aeronautics
University of Patras, Greece

Dr. Dimitris Mourtzis
Associate professor

Patras, 2017

Chapter 9:
Three – Axis Programming

Table of Contents

Objectives

- Write simple programs to perform hole operations using **three machine axes**
- Explain what a **canned cycle** is
- Explain the difference between **initial level** and **reference level** on CNC machinery
- Explain the difference between a **modal** and **non-modal** command
- Write simple programs to perform **milling** operations using **three machine axes**
- Write simple programs involving a **machine indexer**

Parts of a CNC Program

Three-axis Programming

- Three-axis programming is used for a program sequence in which **all three machine axes are used at the same time**

Two-and-half axis programming

- Use all three axes **BUT** Primarily position a location using X and Y axis
- Use **Z axis to perform a drilling or milling operation**
- Is the most common CNC milling programming
- **90% of the CNC** machining center programming
- It is the **practical limit** for manual programming
- Mathematical calculations for 3-axis are **very time consuming**

Parts of a CNC Program

- 3-axis, 4-axis and 5-axis programming are performed **using CAD / CAM systems**
- **Tool length offset** is used
- Operator enters the **tool lengths into the appropriate tool length offset registers** in the CNC controller
- Tool length compensation **adjust Z-axis zero point** to account for the differences in the lengths of the various cutting tools used in the program

A Programming Task Using Three Axes

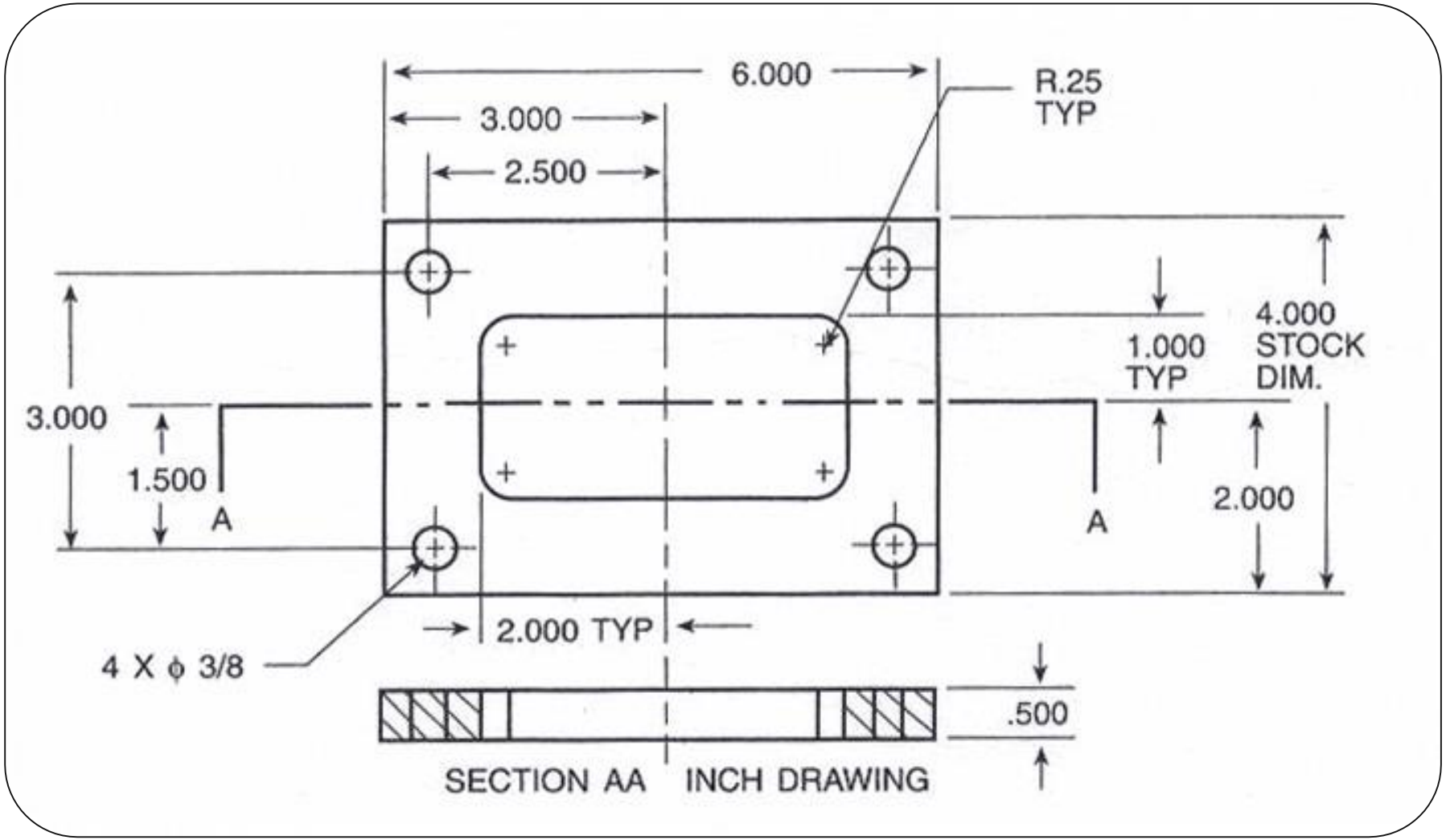
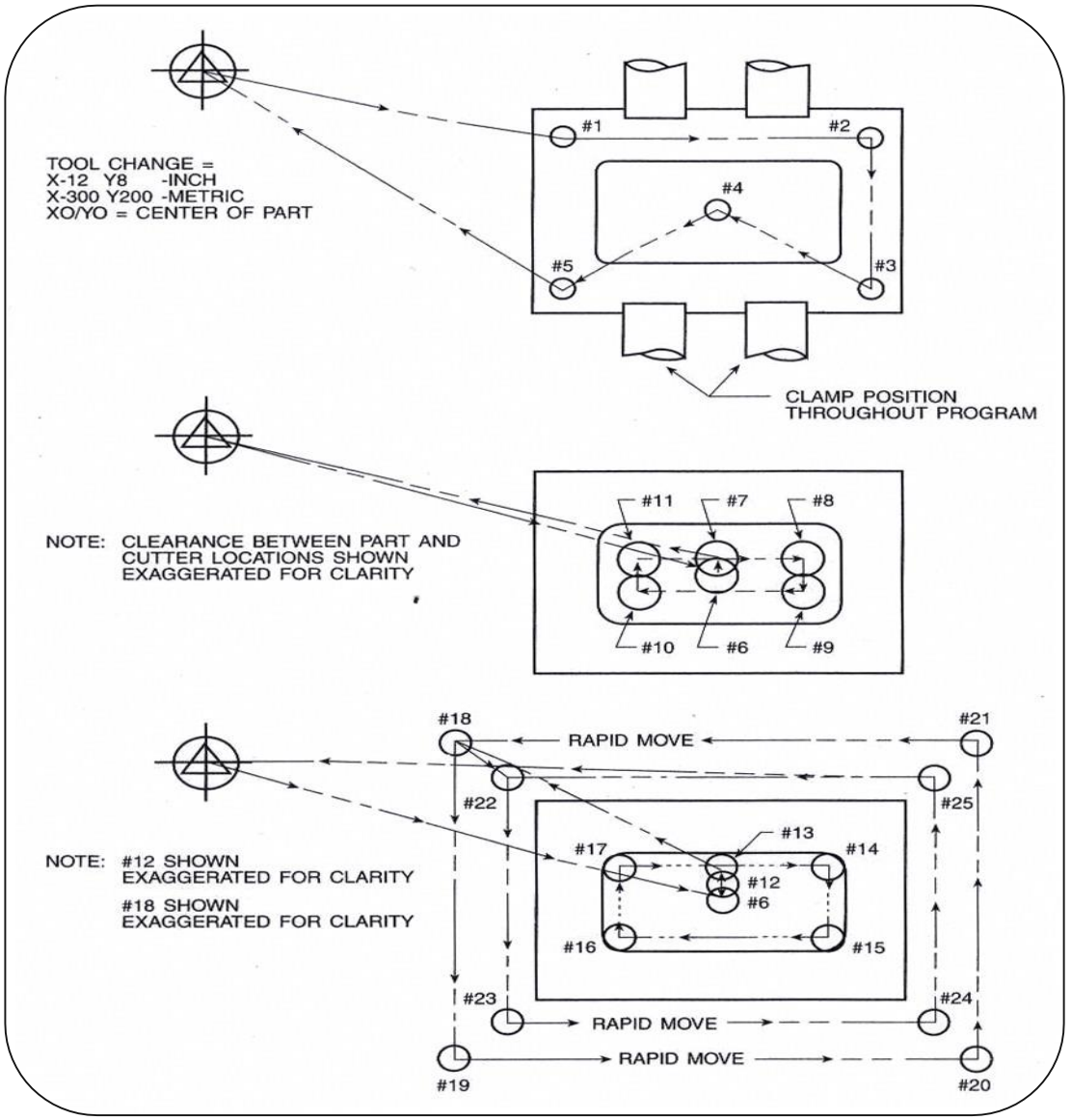


Figure 1: Part drawing for three-axis programming task

Seams W., "Computer Numerical Control, Concepts & Programming"

A Programming Task Using Three Axes



Process Data

- The part is to be **milled**
- **Absolute** positioning
- **Word Address** Format
- **Vertical** Machining Center
- **CMU**: FANUC 11M

Figure 2: Cutter paths for part drawing in figure 1

Sequence of Events

1. Move to the machine ***home zero*** and preset the ***part coordinate*** system
2. Change tools, placing a ***drill into the spindle***, and turn on the ***tool length offset compensation***
3. Drill hole **#1**
4. Drill hole **#2**
5. Drill hole **#3**
6. Drill hole **#4**
7. Drill hole **#5**
8. Cancel the tool length offset compensation
9. Change tools, placing ***a 1.000-inch diameter end mill*** in the spindle and turning on the tool length offset compensation
10. Move to location 6 at rapid traverse and ***plunge cut a hole*** through the part
11. Feed from **#6** to **#7**
12. Feed from **#7** to **#8**
13. Feed from **#8** to **#9**
14. Feed from **#9** to **#10**
15. Feed from **#10** to **#11**
16. Feed from **#11** to **#7**
17. ***Retract the spindle*** and cancel the tool length offset compensation
18. Change tools, placing ***a .500-inch diameter end mill*** in the spindle; turn on the tool length offset compensation
19. ***Rapid traverse*** to location #6
20. Lower the spindle to depth
21. Feed from **#6** to **#13**
22. Feed from **#13** to **#14**
23. Feed from **#14** to **#15**
24. Feed from **#15** to **#16**
25. Feed from **#16** to **#17**
26. Feed from **#17** to **#13**
27. Feed from **#13** to **#12**
28. Retract the spindle
29. ***Rapid traverse*** from **#12** to **#18**
30. Lower the spindle to depth
31. Feed from **#18** to **#19**
32. Retract the spindle
33. Rapid traverse from **#19** to **#20**, ***jumping over the clamps***

Sequence of Events

34. Lower spindle to depth
35. Feed from #20 to #21
36. Retract the spindle
37. Rapid traverse from #21 to #18, jumping the clamps
38. Lower spindle to depth
39. Feed from #18 to #22
40. Feed from #22 to #23
41. Retract the spindle
42. Rapid move from #23 to #24
43. Lower spindle to depth
44. Feed from #24 to #25
45. Retract the spindle
46. Turn off the tool length offset compensation
47. Return the spindle to the home zero location

Seams W., "Computer Numerical Control, Concepts & Programming"

A Programming Task Using Three Axes 1/2

```
%
07003
(* ********************************)
(* COORDINATE SYSTEM ORIGIN)
(* X/Y 0 - CENTERLINE OF PART)
(* Z0 - .100 ABOVE TOP OF PART)
(* ********************************)
(* ABSOLUTE ZERO SHIFT TO PART SYSTEM)
(* ********************************)
N10 G80 G90 G70
N20 G28 G91 X0. Y0. Z0.
N30 G92 X10.625 Y7.5Z6.
(* ************)
(* TOOL 1 - 3/8 STUB DRILL)
(* DRILL HOLES)
(* ************)
N40 G00 G90 G98 G70
N50 T01 M06
N60 G00 X-2.5 Y1.5 S1066 M03 T02
N70 G44 Z0. H01 M08
N80 G81 G99 X-2.5 Y1.5 Z-.62 R0. F12.8
N90 X2.5
N100 Y-1.5
N110 X0. Y0.
N120 X-2.5 Y-1.5
N130 G80 G00 Z0.
N140 G49 M01
```

```
(**************)
(* TOOL 2 - 1.0 DIA. 4-FLT. END MILL)
(* ROUGH MILL INSIDE OF SLOT)
(* ************)
N150 G00 G90 G98 G70
N160 T02 M06
N170 G00 X0. Y0.3 S425 M03 T03
N180 G44 Z0. H02 M08
(FEED TO DEPTH)
N190 G01 Z-.62 F6.8

(ROUGH MILL INSIED)
N200 Y.48
N210 X1.48
N220 Y-.48
N230 X-1.48
N240 Y.48
N250 X0.
N260 G80 G00 Z0. M09
N270 G49 M01
(* ************)
(* TOOL 3 - 1/2 DIA. 4-FLT. END MILL)
(* FINISH INSIDE SLOT)
(* ROUGH/FINISH OUTSIDE OF PART)
(* ************)
N280 G00 G90 G98 G70
N280 T03 M06
N300 G00 X0. Y0. S800 M03 T03
N310 G44 Z0. H03 M08
```

```
(FEED TO DEPTH)
N320 G01 Z-.62 F12.8
(FINISH MILL INSIDE SLOT)
N330 Y.75
N340 X1.75
N350 Y-.75
N360 X-1.75
N370 Y.75
N380 X0.
(PULL AWAY FROM PART AND RETRACT SPINDLE)
N390 Y.74
N400 G00 Z3.
(POSITION TO START OF OUTSIDE MILL CUT)
N410 X-3.26 Y2.26
N420 Z0.
(FEED TO DEPTH AND ROUGH MILL 1ST SIDE)
N430 G01 Z-.62 F12.8
N440 Y-2.26
```

Figure 3(a):Three-axis program, inch, for the part in figure 1 (1/2)

Seams W., "Computer Numerical Control, Concepts & Programming"

A Programming Task Using Three Axes 2/2

```
(RETRACT SPINDLE AND JUMP OVER CLAMP)
(POSITION FOR ROUGH CUT ON 2ND SIDE)
N450 G00 Z3.
N460 X3.26
N470 Z0.
(FEED TO DEPTH AND ROUGH MILL 2ND SIDE)
N480 G01 Z-.62 F12.8
N490 Y2.26
(RETRACT SPINDLE AND JUMP OVER CLAMP)
N500G00Z3.
N510 X-3.26 Y2.26
N520 Z0.
(FEED TO DEPTH - MOVE TO PART SURFACE)
(AND FINISH MILL FIRST SIDE)
N530 G01 Z-6.2 F12.8
N540 X-3.25 Y2.25
N550 Y-2.25
(RETRACT SPINDLE AND JUMP OVER CLAMP)
(POSITION FOR FIISH CUT ON 2ND SIDE)
N560 G00 Z3.
N570 X3.25
N580 Z0.
```

```
(FEED TO DEPTH AND FINISH MILL 2ND SIDE)
N590 G01 Z-.62 F12.8
N600 Y2.25
N610 G80 G00 Z0 M09
N620 G49
N630 G28 G91 Z0. M05
N640 G28 X0. Y0.
N650 M30
%
```

Figure 3(b): Three-axis program, inch, for the part in figure 1 (2/2)

A Programming Task Using Three Axes

- **Several new word address commands used in this program**
- **G28 - Return to reference point command**
- **G28** is used in conjunction with other commands to **cause the spindle to position at the machine's coordinate system origin**
- This point is referred to as **home zero** in most CNC shops
- **If coordinates are specified on the G28 line, the spindle will first move to the coordinates, then to home zero**
- In this manner the spindle may be moved to a **known safe position** before moving to **home zero**

Seams W., "Computer Numerical Control, Concepts & Programming"

A Programming Task Using Three Axes

- **G44 - Calls up a tool length offset register**
- A **G44** accomplishes a **Z-zero** shift toward the workpiece
- **H - Used to assign a tool register**
- **H01** would assign the information stored in **tool length register #1**
- **H02** would assign the information stored in **tool length register #2**
- **G49** - This is the **tool length offset cancel code**

A Programming Task Using Three Axes

- **Several new word address commands used in this program**
- **G81 - This is the canned drill cycle**
- When a **G81** is issued:
  - The spindle rapids to the (X,Y) coordinates specified on the drill cycle line
  - The Z axis then rapids to the specified feed engagement point
  - Feeds to the final drill depth
  - Then rapids out of the hole to either the rapid or initial level
- **G80 - This is the canned cycle cancel code**
- When a **G80** is issued, the active canned cycle code is turned off

A Programming Task Using Three Axes

- **R** - This address stands for the **canned cycle reference level**
- The **reference level** is the spot where the programmer desires the **canned cycle to start feeding into the workpiece**
- The **reference level** is also called the **rapid or gage level**
- **G92 - Absolute zero set command**
- This command tells the control to **reset the part coordinate system origin - Coordinates must be specified on the G92 block -** The coordinates tell the machine where **to set the origin, relative to the current spindle position**

A Programming Task Using Three Axes

- **G99/G98**
- **G98** is the **return to initial level** command
- **G99** is the **return to rapid (reference) level** command
- When a **canned cycle is active**, the spindle may be directed to **return to the rapid level** when it exits a hole with a **G99**
- If the programmer desires the **spindle to return to the original starting point Z height**, the **G98** command is issued
- **G99** results in the **faster cycle**
- **G98** is particularly useful for **jumping over clamps** and other obstructions while in a cycle

A Programming Task Using Three Axes

- **M01** - **Program optional stop** code
- **M01** functions as an **M00** with one exception: **it is only effective if the optional stop switch on the machine control is turned on**
- When this switch, -called an **opstop switch-**, is off, the **M01** is ignored by the control
- **M03** - is the code for **turning the spindle on in the clockwise direction**
- **M05** - Turns the **spindle off**

A Programming Task Using Three Axes

- **M06** - **Tool change** code
- When **M06** is issued, the machine's **automatic tool changer sequence will be initiated**
- **M08** - Turns the **flood coolant on**
- **M09** - Turns the **coolant off**
- **T** - **Selects the tool** to be put in the spindle by the tool changer
- **F** - **Assigns feedrates**, as in two-axis programming
- **S** - **Designates the spindle speed**

Modal / Non-Modal Commands

Modal Commands

- Codes that are **active for more than one line** in which they are issued
- **Rapid transverse, Feedrate moves and canned cycle** codes are examples of **modal commands**

Non-Modal command

- Is the one that is **active only in the program block in which it is issued**
- **M00**: **Program Stop** is an example of a **Non-Modal command**

Canned Cycles

- Are **routines** (e.g. **G81**) built into the control to **perform standard operations**
- Drilling, boring and taping are common operations
- The programmer can **call a canned cycle instead of repetitive programming**

Modal Commands

- Most G codes put the machine in a "permanent" status, which remains in effect until it is changed or canceled by another G command
- Those are the **modal commands**

G00	Rapid Transverse	G43	Tool length compensation (plus)
G01	Linear Interpolation	G44	Tool length compensation (minus)
G02	Circular Interpolation, CW	G49	Tool length compensation cancel
G03	Circular Interpolation, CCW	G80	Cancel canned cycles
G17	XY Plane	G81	Drilling cycle
G18	XZ Plane	G82	Counter boring cycle
G19	YZ Plane	G83	Deep hole drilling cycle
G20/G70	Inch units	G90	Absolute positioning
G21/G71	Metric Units	G91	Incremental positioning
G40	Cutter compensation cancel		
G41	Cutter compensation left		
G42	Cutter compensation right		
G43	Tool length compensation (plus)		

Figure 4: Example showing G00 and G01 modal commands

Canned Cycles

- **A canned Cycle for Tapping:**

G84G99X1.Y.375Z-.753R.1F10

- **G84 - G-code to turn on the tapping cycle**

The spindle will feed into the work-piece with the **spindle rotating clockwise**, stop at the programmed Z axis coordinate, reverse the spindle, then feed back out of the workpiece until it reaches the programmed feed engagement point

- **G99 -** Specifies that the **spindle should return to the reference level** (the feed engagement point) when retracting out of the hole

- **X/Y coordinates** - **Indicate the location where the cycle is to begin**. The spindle will first position here at rapid traverse before moving the Z - axis.

- **Z coordinates** - Tells the **control how deep to feed the Z - axis**. It is the actual Z coordinate to which the spindle is to move

Canned Cycles

- **R -** Specifies the **Z coordinate where the spindle is to begin feeding** Until the spindle reaches this coordinate, it will move in rapid traverse.

- **F -** Sets the **feedrate for the Z - axis feedrate moves**

- A **complete cycle** would look like following:

```
G84 G99 X1.Y.375Z-.753R.1F10
X1.5
Y1.375
X1.
G80
```

Seams W., "Computer Numerical Control, Concepts & Programming"

Program Explanation

%

Program start code.

Program Explanation

07003

Program number.

N10 - N30 *are the tape startup blocks*

N10

N10 - The block sequence number.
G80 - Canned cycle cancel command, turns off any active canned cycles.
G90 - Selects absolute positioning mode.
G70 - Selects inch input.

N20

N20 - The block sequence number
G28 - Return to reference point. On FANUC-style controls, the reference point is the machine home zero. The machine is returned to home zero prior to issuing a G92 coordinate system preset in the next block. Even if the operator moved the spindle between program ccycles, the spindle would be positioned to home zero because this block was included in the program
G91 - Selects absolute positioning.
X0.Y0. coordinates - Because the machine is in incremental mode, the spindle will not move anywhere
G28 command simply will return the spindle to home zero

N30

N30 - The block sequence number
G92 - Absolute zero preset code. The G92 cause the control to reset the part coordinate system. The
G28 - Command in the previous block insured the spindle was at home zero prior to issuing the G92
X/Y coordinates - Specify where the part origin should be set, incrementally from the current spindle position.

Program Explanation

N40 - *Tool sequence safety block*

N40 - The block sequence number
G00 -Selects rapid traverse mode
G90 - Selects absolute positioning
G98 - Selects return to initial level mode on canned cycle Z-axis retract moves
G70 - Selects inch input

N50 through N70 are the tool change blocks

N50

N50 - The block sequence number
T01 - Places tool 1 in standby mode for the next tool change command
M06 - Initiates an automatic tool change cycle
The tool in standby (in this case tool 1) is placed in the spindle, and the previous tool put away in the tool storage magazine

N60

N60 - The block sequence number
G00 - Selects rapid 1 reverse mode
X/Y coordinates - Move the spindle from home zero to the position #1
S1066 - Sets the spindle speed to 1066 rpm
M03 - Turns the spindle on clockwise
T02 - Places tool 2 in tool change standby

N70

N70 - The block sequence number
G44 - Turns on tool length compensation
Z coordinate - Moves the Z-axis to .100 above the top of the part (the part Z-zero point)
H01 - instructs the control to use the values in tool length offset register #1 for tool length compensation - M08--Turns on the flood coolant.

Program Explanation

N80

N80 - The block sequence number
G81 - Turns on the canned drilling cycle
G99 - Instructs the control to return the Z-axis to the feed engagement point (the reference level) when retracting the spindle out of a hole
The G99 is effective only when the canned cycle is turned on
X/Y coordinates - Coordinates of hole #1
Z coordinate - The depth of the drilled hole. Note that this is the programmed depth (i.e.where the tip of the tool is to be sent)
R - Specifies the Z-axis coordinate for the feed engagement point
This point is also referred to as the r-plane
In this example, the spindle is commanded to begin feeding into the workpiece at the Z-zero point (.100 above the top of the part). F12.8 - Sets feedrate at 12.8 inches per minute

N90

N90 - The block sequence number
X coordinate - Moves the spindle from position #1 to #2

N100

N100 - The block sequence number.
Y coordinate - Moves the spindle from position #2 to #3

N110

N110 - The block sequence number
X/Y coordinate - Moves the spindle from position #3 to #4

N120

N120 - The block sequence number
X/Y coordinate - Moves the spindle from position #4 to #5

Program Explanation

N130 and N140 are the tool cancel sequence

N130

N130 - The block sequence number
G80 - Turns off the canned drill cycle
G00 - Selects rapid traverse mode.
Z0. - Returns the spindle to the part 2-zero point

N140

N140 - The block sequence number
G49 - Cancels the tool length compensation
M01 - Optional program stop code. The M01 is included for operator convenience. If the operator desired to check the part or setup for any reason, he or she need only turn on the opstop switch on the control The program will halt execution at the end of the tool cycle

N150 - *Tool sequence safety block*

N150

N150 - The block sequence number
G00 - Selects rapid traverse mode
G90 - Selects absolute positioning
G98 - Selects return to initial level mode on canned cycle Z-axis retract moves
G70 - Selects inch input

N160 through N180 are the tool change blocks

N160

N160 - The block sequence number.
T02 - Places tool 2 in standby mode for the next tool change command
M06 - Initiates an automatic tool change cycle

Program Explanation

N170

N170 - The block sequence number
G00 - Selects rapid traverse mode
X/Y coordinates - Move the spindle rpm home zero to the position #6
S - Sets the spindle speed rpm
M03 - Turns the spindle on clockwise. T03 - Places tool 3 in tool change standby

N180

N180 - The block sequence number
G44 - Turns on tool length compensation
Z coordinate - Moves the Z-axis to .100 above the top of the part (the part Z-zero point)
H02 - Instructs the control to use the values in tool length offset register #2 for tool length compensation
M08 - Turns on the flood coolant

N190 through N250 are the tool motion sequence

N190

N190 - Block sequence number
G01 - Selects feedrate mode
Z coordinate - Positions the spindle, at feedrate. to the milling depth
F - Sets the feedrate

N200

N200 - Block sequence number
Y Coordinate - Moves the spindle from position #6 to #7

N210

N210 - Block sequence number
X coordinate - Moves the spindle from position #7 to #8

Program Explanation

N220

N220 - Block sequence number
Y coordinate- Moves the spindle from position #8 to #9

N230

N230 - Block sequence number
X coordinate - Moves the spindle from position #9 to #10

N240

N240 - Block sequence number
Y coordinate - Moves the spindle from position #10 to #11

N250

N250 - Block sequence number
X coordinate - Moves the spindle from position #11 to #7

N260 & N270 are the tool cancel sequence

N260

N260 - The block sequence number
G80 - Turns off the canned drill cycle
G00 - Selects rapid traverse mode
Z0 - Returns the spindle to the part Z-zero point

N270

N140 - The block sequence number
G49 - Cancels the tool length compensation
M01 - Optional program stop code

Program Explanation

N280 - *Tool sequence safety block*

N280 - The block sequence number.
G00 - Selects rapid traverse mode.
G90 - Selects absolute positioning.
G98 - Selects return to initial level mode on canned cycle Z-axis retract moves.
G70 - Selects inch input.

N290 through N310 are the tool change blocks

N290

N290 - The block sequence number.
T03 - Places tool 3 in standby mode for the next tool change command.
M06 - Initiates an automatic tool change cycle.

N300

N300 - The block sequence number.
G00 - Selects rapid traverse mode.
X/Y coordinates - Move the spindle from home zero to the position #6.
S - Sets the spindle speed rpm.
M03 - Turns the spindle on clockwise.
T01 - Places tool 1 in tool change standby.

N310

N310 - The block sequence number.
G44 - Turns on tool length compensation.
Z coordinate - Moves the Z-axis to 100 above the top of the part (the part Z-zero point).
H03 - Instructs the control to use the values in tool length offset register #3 for tool length compensation.
M08 - Turns on the flood coolant.

Program Explanation

N320 through N600 are the tool motion sequence

N320

N320 - Block sequence number
G01 - Selects feedrate mode.
Z coordinate - Positions the spindle, at feedrate. to the milling depth.
F - Sets the feedrate,

N330

N330 - Block sequence number.
Y coordinate - Moves the spindle from position #6 to #13.

N340

N340 - Block sequence number.
X coordinate - Moves the spindle from position #13 to #14.

N350

N350 - Block sequence number.
Y coordinate - Moves the spindle from position #14 to #15.

N360

N360 - Block sequence number.
X coordinate - Moves the spindle from position #15 to #16.

N370

N370 - Block sequence number.
Y coordinate - Moves the spindle from position #16 to #7.

N380

N380 - Block sequence number.
X coordinate - Moves the spindle from position #17 to #13.

Program Explanation

N390

N390 - Block sequence number.
X coordinate - Moves the spindle from position #13 to #12.

N400

N400 - Block sequence number.
G00 - Selects rapid traverse mode.
2 coordinate - Retracts spindle to 23.000.

N410

N410 - Block sequence number
X/Y coordinates - Move the spindle from position #12 to #18.

N420

N420 - Block sequence number.
Z coordinates - Rapids spindle to .100 above the part.

N430

N430 - Block sequence number.
G01 - Selects feedrate mode.
2 coordinate - Moves the spindle at feedrate to the milling depth.
F - Sets the feedrate.

N440

N440 - Block sequence number.
Y coordinate - Moves the spindle from position #18 to #19.

N450

N450 - Block sequence number.
G00 - Selects rapid traverse mode.
2 coordinate - Retracts spindle to Z3.000.

Program Explanation

N460

N460 - Block sequence number.
X/Y coordinates - Move the spindle from position #19 to #20.

N470

N470 - Block sequence number.
Z coordinate - Rapids spindle to .100 above the part.

N480

N480 - Block sequence number.
G01 - Selects feedrate mode.
Z coordinate - Moves the spindle at feedrate to the milling depth.
F - Sets the feedrate.

N490

N490 - Block sequence number.
Y coordinate - Moves spindle from position #20 to #21.

N500

N500 - Block sequence number.
G00 - Selects rapid traverse mode.
Z coordinate - Retracts spindle to Z3.000.

N510

N510 - Block sequence number.
X/Y coordinates - Move the spindle from position #21 to #18.

N520

N520 - Block sequence number.
Z coordinate - Rapids spindle to .100 above the part.

Program Explanation

N530

N530 - Block sequence number.
G01 - Selects feedrate mode.
Z coordinate - Moves the spindle at feedrate to the milling depth.
F - Sets the feedrate.

N540

N540 - Block sequence number.
X/Y coordinates - Moves spindle from position #18 to #2.

N550

N550 - Block sequence number.
Y coordinate - Moves spindle from position #22 to #23

N560

N560 - Block sequence number.
G00 - Selects rapid traverse mode.
Z coordinate - Retracts spindle to Z3.000

N570

N570 - Block sequence number.
X/Y coordinates - Move the spindle from position #23 to #24.

N580

N580 - Block sequence number.
2 coordinate - Rapids spindle to .100 above the part.

N590

N590 - Block sequence number
G01 - Selects feedrate mode.
z coordinate - Moves the spindle at feedrate to the milling depth. F - Sets the feedrate.

Program Explanation

N600

N600 - Block sequence number
Y coordinate - Moves spindle from position #24 to #25

N610 and N620 are the tool cancel sequence

N610

N610 - The block sequence number.
G80 - Turns off the canned drill cycle.
G00 - Selects rapid traverse mode.
Z0. - Returns the spindle to the part *Z-zero* point.
M09 - Turns off the tool coolant.

N620

N620 - The block sequence number.
G49 - Cancels the tool length compensation.

N630 through N650 are the end of tape sequence

N630

N630 - Block sequence number.
G28 - Return to reference code.
G91 - Selects Incremental positioning.
Z0. - In conjunction with G28G91 commands the Z-axis to the home zero position.
M05 - Turns off the spindle.

N640

N640 - Block sequence number.
G28 - Return to reference code.
X0.Y0. -In conjunction with G28 commands the X and Y axis to the home zero position.

Program Explanation

N650

N650 - Block sequence number.

M30 - Signals end of program.

%

Program stop code.

Seams W., "Computer Numerical Control, Concepts & Programming"

Milling Example 1

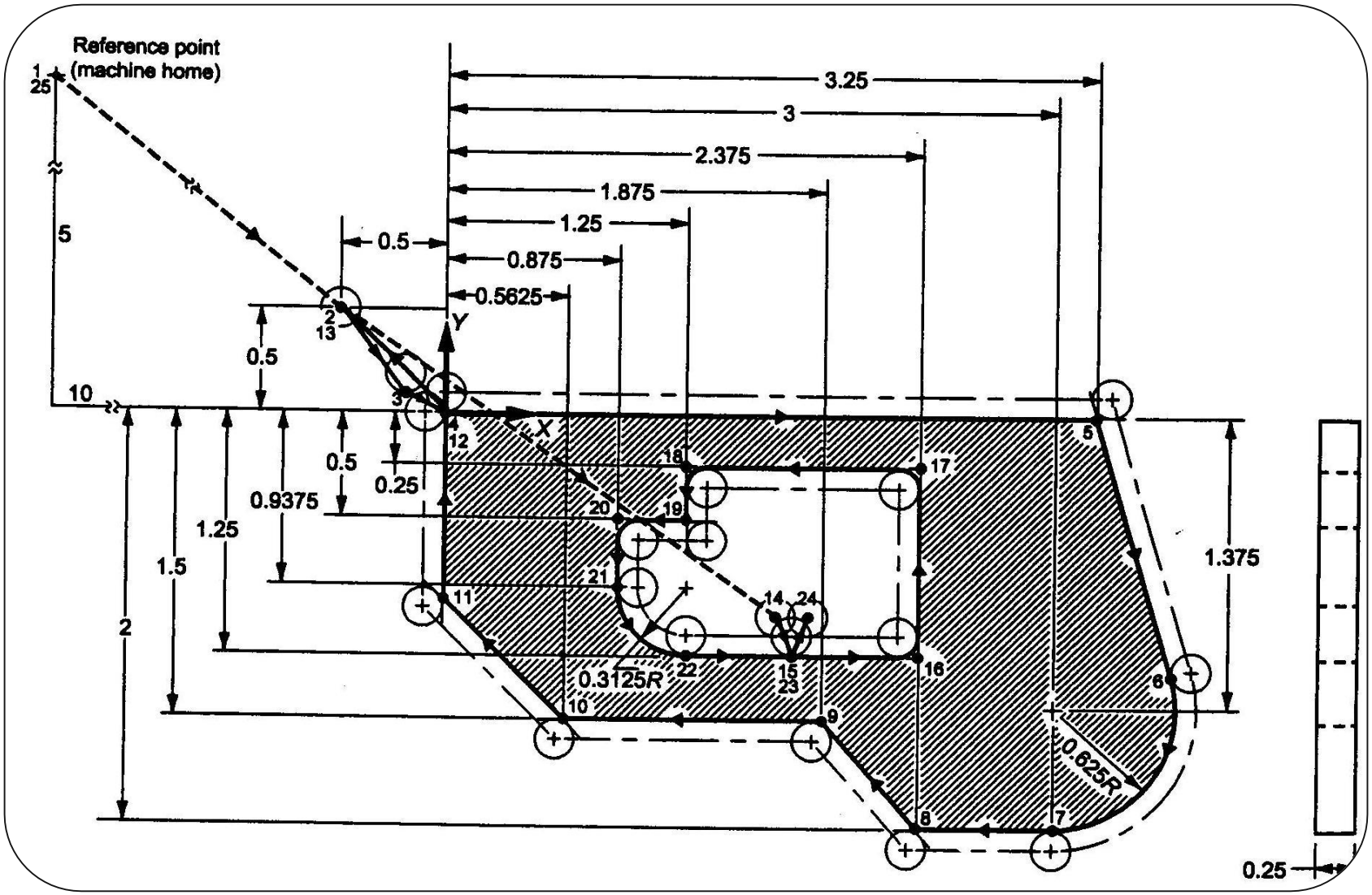


Figure 5:The part to be milled *(Seams W., "Computer Numerical Control, Concepts & Programming")*

Milling Example 1

Word Address command	Meaning
%	End of tape
01313	Program number
N0010 (X0Y0 IS THE UPPER LEFT HAND CORNER)	Comments
N0020 (Z0 IS THE TOP OF THE PART)	
N0030 (TOOL 1: USE REGISTER 21 TO SET CUTTER DIA)	
N0040 G90 G20 G40 G80	Absolute, Inch mode, cancel cutter diameter, compensation and fixed cycles
N0050 G91 G28 X0 Y0 Z0	Return to reference point 1
N0060 G92 X-10.0 Y5.0 Z0	Preset absolute zero point
N0070 T1 M06	Change to tool 1
N0080 (TOOL 1: FINISH INSIDE AND OUTSIDE PROFILES)	
N0090 G00 G90 X-0.5 Y0.5 Z0 S1200 M03	Rapid to 2. Spindle on CW at 1200 rpm

Milling Example 1

Word Address command	Meaning
N0100 G43 Z0.1 H01	**Rapid tool 1 to 0.1 above part**
N0110 M08	**Coolant on**
N0120 G01 Z0.1 H01	**Plunge to -0.25 at feed rate**
N0130 G41 X-0.25 D21	**Ramp on at 3. Offset tool by the value in D21**
N0140 X0 Y0	**Cut to 4**
N0150 X3.25	**Cut to 5**
N0160 X3.6 Y-1.2	**Cut to 6**
N0170 G02 X3.0 Y-2.0 I-0.625 J-0.175	**Cut arc to 7**
N0180 G01 X2.376	**Cut to 8**
N0190 X1.875 Y-1.5	**Cut to 9**
N0200 X0.5625	**Cut to 10**
N0210 X0 Y-0.9375	**Cut to 11**

Milling Example 1

Word Address command	Meaning
N0220 Y0	Cut to 12
N0230 G40 X-0.5 Y0.5	Ramp off at 13
N0240 G00 Z0.1	Rapid to 0.1 above part
N0250 X1.625 Y-1.05	Rapid to 14
N0260 G01 Z-0.25	Plunge to -0.25 at feed rate
N0270 G41 X1.825 Y-1.25 D21	Ramp on at 15. Offset cutter by value in D21
N0280 X2.375	Cut to 16
N0290 Y-0.25	Cut to 17
N0300 X1.25	Cut to 18
N0310 Y-0.5	Cut to 19
N0320 X0.875	Cut to 20
N0330 Y-0.9375	Cut to 21

Milling Example 1

Word Address command	Meaning
N0340 G03 X1.1875 Y-1.25 I0.3125 J0	**Cut arc to 22**
N0350 G01 X1.875	**Cut to 23**
N0360 G40 X2.075 Y-0.105	**Ramp off at 24**
N0370 G00 G90 Z1.0 M05	**Rapid to 1.0 above part**
N0380 M09	**Coolant off**
N0390 G91 G28 X0 Y0 Z0	**Return to XYZ reference point 25**
N0400 M30	**Program end, reset memory**
%	**End of tape**

Milling Example 2

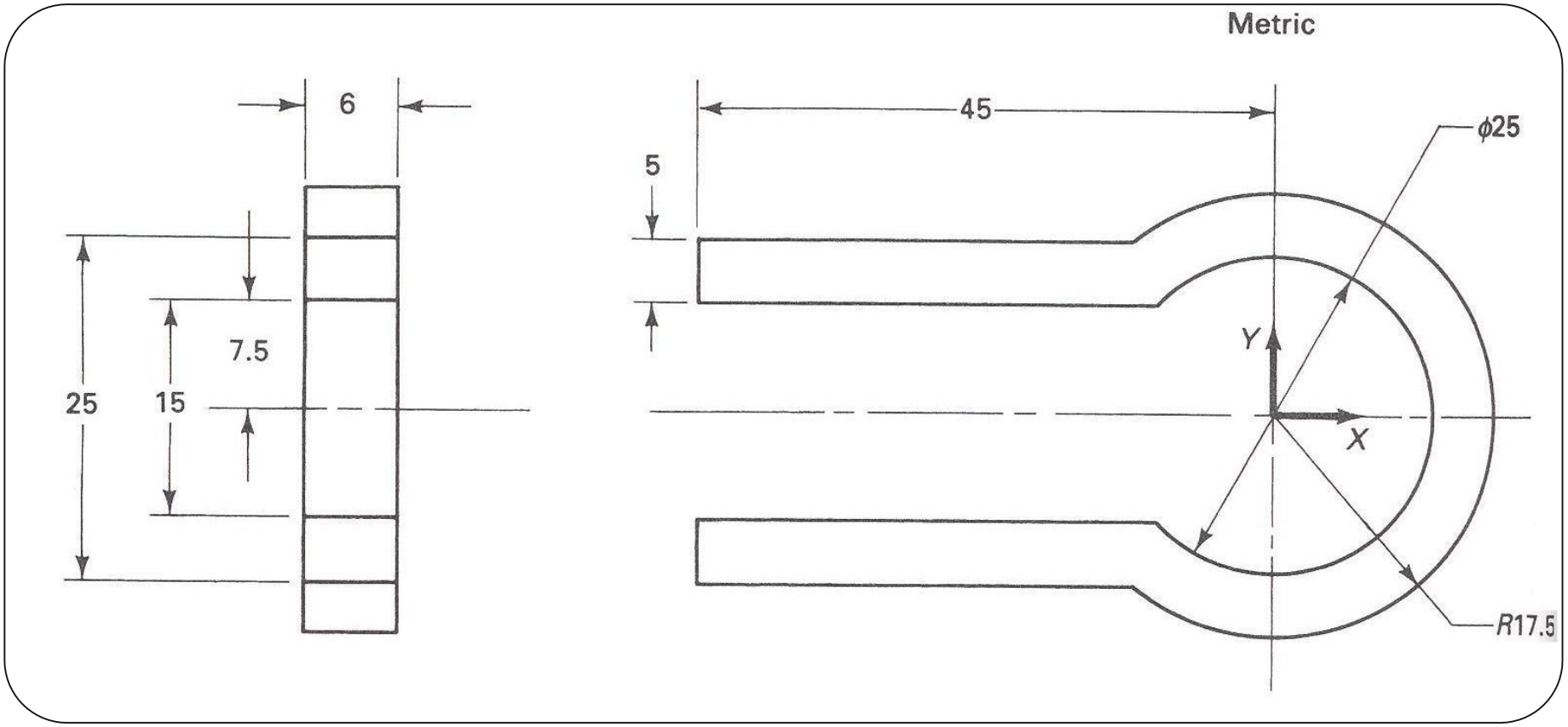


Figure 6 :The part to be milled

Seams W., "Computer Numerical Control, Concepts & Programming"

Milling Example 2

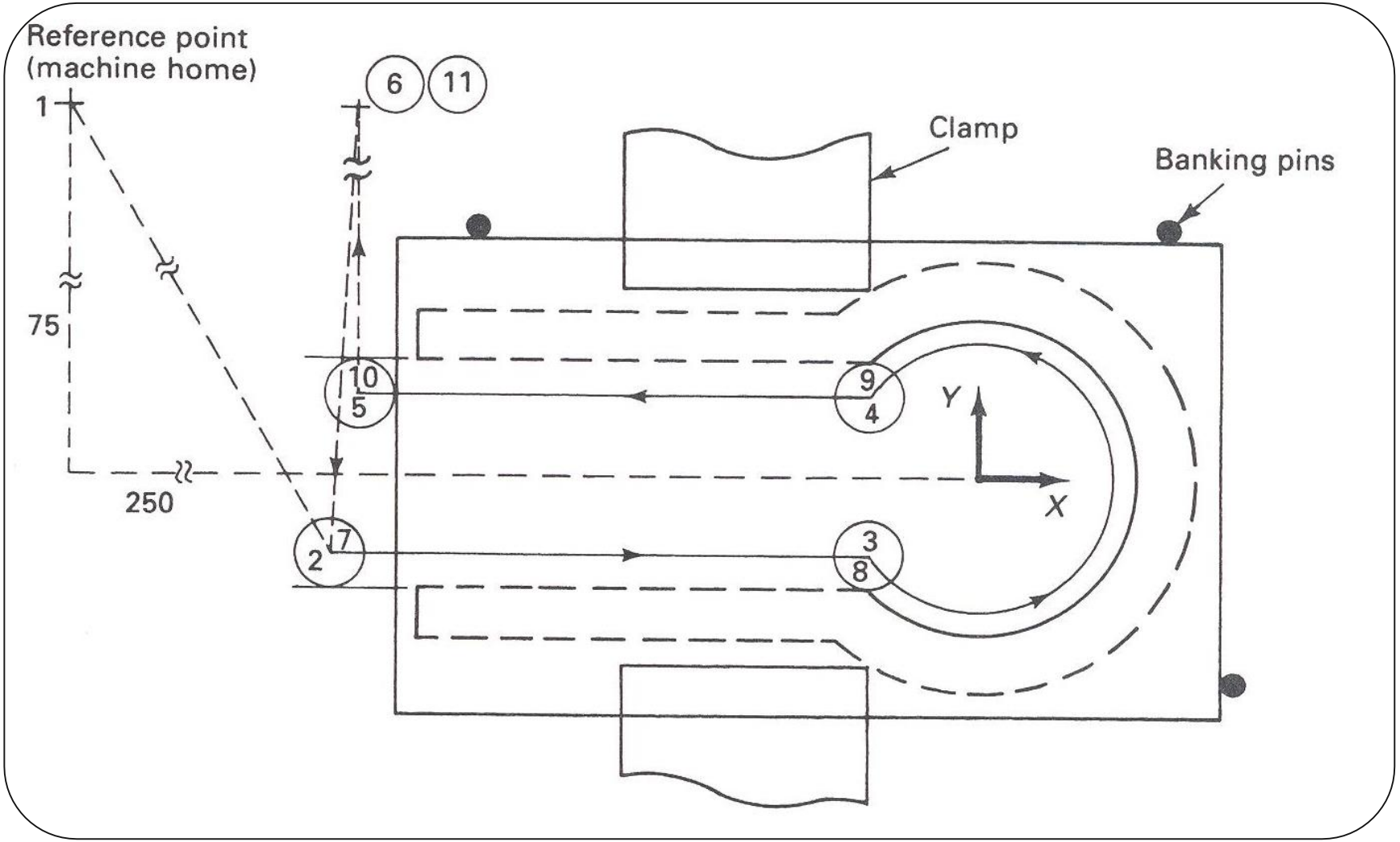


Figure 7(a):Cutter path for milling the inside of the part the part of Fig. 6

Seams W., "Computer Numerical Control, Concepts & Programming"

Milling Example 2

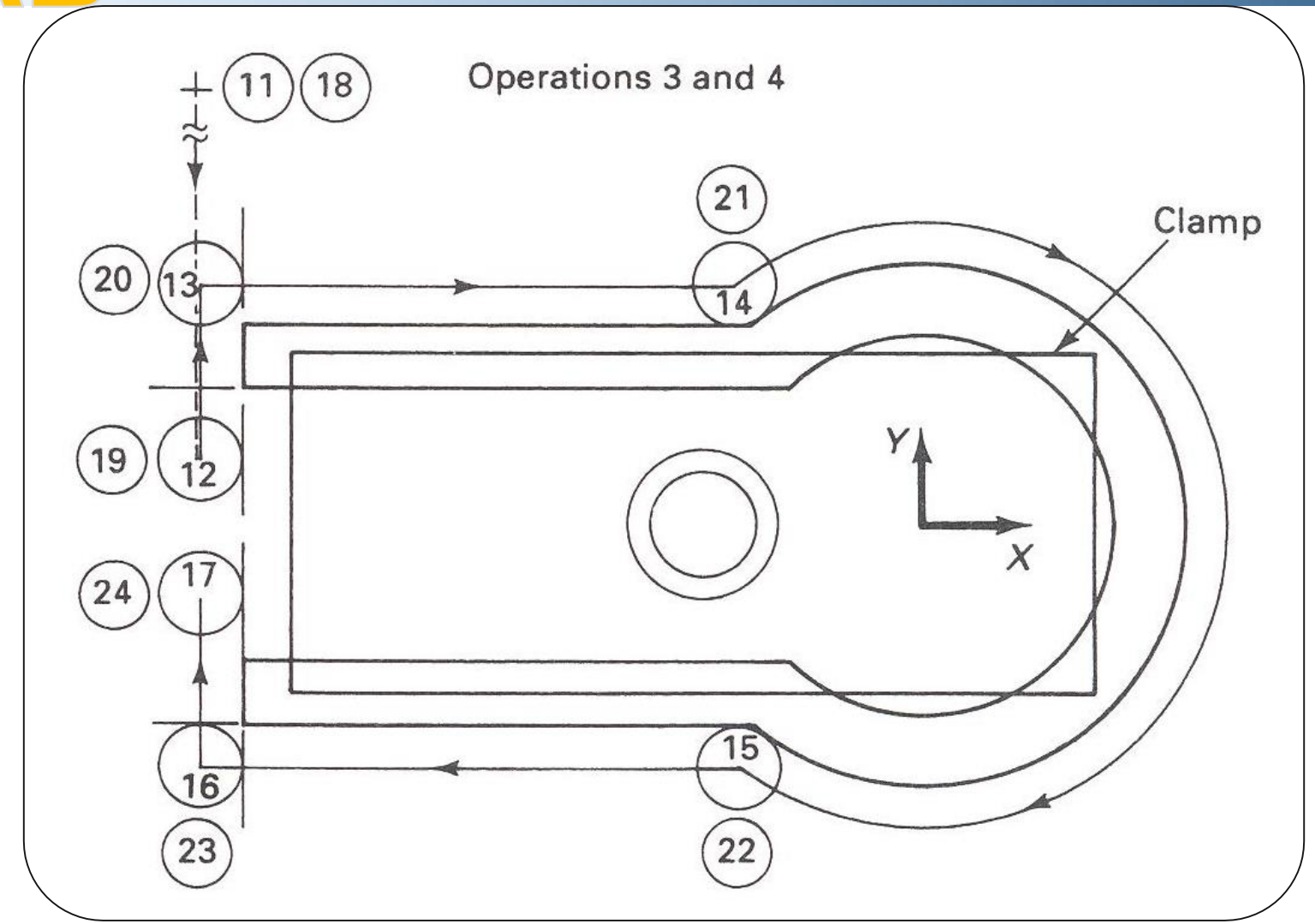


Figure 7(b):Cutter path for milling the outside of the part the part of Fig. 6

Seams W., "Computer Numerical Control, Concepts & Programming"

Milling Example 2

Word Address command	Meaning
%	**End of tape**
O1204	**Program number**
N0010 (X0Y0 IS 18.5MM FROM UPPER RIGHT CORNER)	**Comments**
N0020 (Z0 IS THE TOP OF THE PART)	
N0030 (TOOL 1:6MM ROUGHING END MILL)	
N0040 (TOOL 2:6MM FINISHING END MILL)	
N0050 G90 G21 G40 G80	**Absolute, metric mode, cancel cutter diameter compensation and fixed cycles**
N0060 G91 G28 X0 Y0 Z0	**Return to reference point 1**
N0070 G92 X-250 Y75 Z0	**Present absolute zero point**
N0080 T1 M06	**Change to Tool 1**
N0090 (TOOL 1 : ROUGH INSIDE CONTOUR 6.5MM DEEP)	

Milling Example 2

Word Address command	Meaning
N0100 G00 G90 X-50.2 Y-4.3 Z0 S2100 M03 T2	**Rapid to 2. Spindle on (CW) at 2100 rpm. Prepare tool 2 in ready position**
N0110 G43 Z2.0 H01	**Rapid tool 1 to 2mm above part**
N0120 M08	**Coolant on**
N0130 G01 Z-6.5 F120	**Linear profile mode. Plunge to Z-6.5 mm at feed rate 120mm/min**
N0140 X-8.2462	**Cut to position 3**
N0150 G03 X-8.2462 Y4.3 I8.2462 J4.3	**Rough 12.5R arc to 4**
N0160 G01 X-48.2	**Cut to position 5**
N0170 G00 G90 Z1 M05	**Rapid to 1mm above part. Spindle off**
N0180 M09	**Coolant off**
N0190 G91 G28 Z0 Y0	**Rapid to tool change position 6**

Milling Example 2

Word Address command	Meaning
N0200 T2 M06	Change to tool 2
N0210 (TOOL 2: FINISH INSIDE CONTOUR 6MM DEEP)	
N0220 G00 G90 X-50 Y-4.5 Z0 S2600 M03	Rapid to 7. Spindle on (CW) at 2600 rpm
N0230 G43 Z2 H02	Rapid tool 2 to 2mm above part
N0240 M08	Coolant on
N0250 G01 Z-6.5 F100	Linear profile mode.Plunge to Z-6.5 mm at feed rate 100mm/min
N0260 X-8.366	Cut to position 8
N0270 G03 X-8.366 Y 4.5 I8.366 J4.5	Finish 17.5R arc to 9
N0280 G01 X-48	Cut to position 10
N0290 G00 Z1 M05	Rapid to 1mm above part. Stop spindle
N0300 M09	Coolant off

Milling Example 2

Word Address command	Meaning
N0310 G91 G28 Z0 Y0	Rapid to tool change position 11
N0320 T1 M06	Change to tool 1
N0330 M00 (RECLAMP PART FROM INSIDE)	Program stop
N0340 (ROUGH OUTSIDE CONTOUR 6.5MM DEEP)	
N0350 G00 G90 X-48.2 Y2.3 Z0 S2100 M03	Rapid to 12. Spindle on (CW) at 2100 rpm
N0360 G43 Z2 H01	Rapid tool 1 to 2mm above part
N0370 M08	Coolant on
N0380 G01 Z-6.5 F120	Linear profile mode. Plunge to Z-6.5 mm at feed rate 120mm/min
N0390 Y15.7	Cut to position 13
N0400 X-13.4907	Cut to position 14
N0410 G02 X-13.4907 Y-15.7 I13.4907 J-15.7	Rough 17.5R arc to 15

Milling Example 2

Word Address command	Meaning
N0420 G01 X-48.2	Cut to position 16
N0430 Y-4.3	Cut to position 17
N0440 G00 G90 Z1 M05	Rapid to 1mm above part. Spindle off
N0450 M09	Coolant off
N0460 G91 G28 Z0 Y0	Rapid to tool change 18
N0470 T2 M06	Change to tool 2
N0480 (TOOL 2: FINISH OUTSIDE CONTOUR 6.5MM DEEP)	
N0490 G00 G90 X-48 Y2.5 Z0 S2600 M03	Rapid to 19. Spindle on (CW) at 2600rpm
N0500 G43 Z0.1 H03	Rapid tool 2 to 0.1mm above part
N0510 M08	Coolant on
N0520 G01 Z-6.5 F100	Linear profile mode. Plunge to Z-6.5 mm at feed rate 100 mm/min

Milling Example 2

Word Address command	Meaning
N0530 Y15.5	Cut to position 20
N0540 X-13.4164	Cut to position 21
N0550 G02 X-13.4164 Y-15.5 I 13.4164 J-15.5	Finish 17.5R arc to 22
N0560 G01 X-48	Cut to position 23
N0570 Y-4.5	Cut to position 24
N0580 G00 G90 Z1 M05	Rapid to 1mm above part Stop spindle
N0590 M09	Coolant off
N0600 G91 G28 X0 Y0 Z0	Return to XYZ reference point
N0610 M30	Program end, memory reset
%	End of tape

A Programming Task Using Three Axes

Figure 8 :A Light-duty vertical machining centre (Photo courtesy of Hardinge Bridgeport)

Other G Codes used in CNC Programming

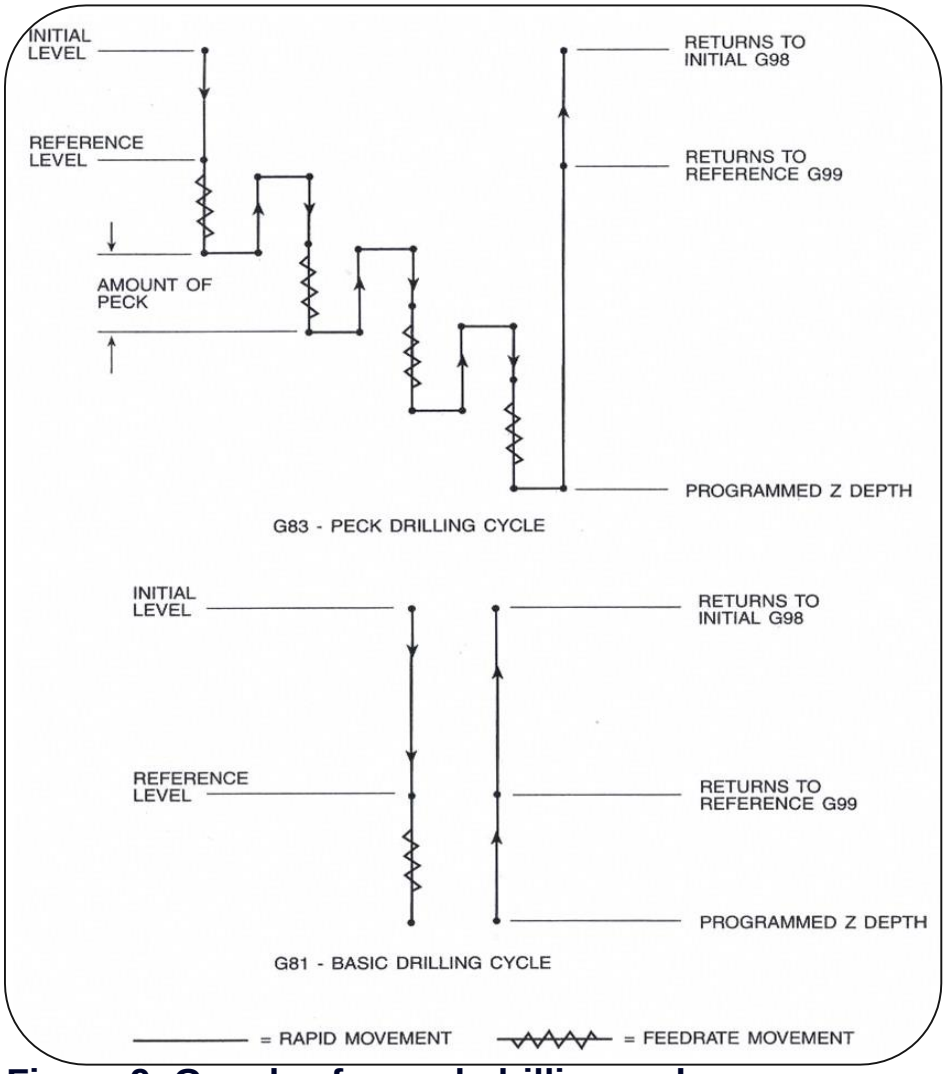


Figure 9: G codes for peck drilling and basic Drilling cycles

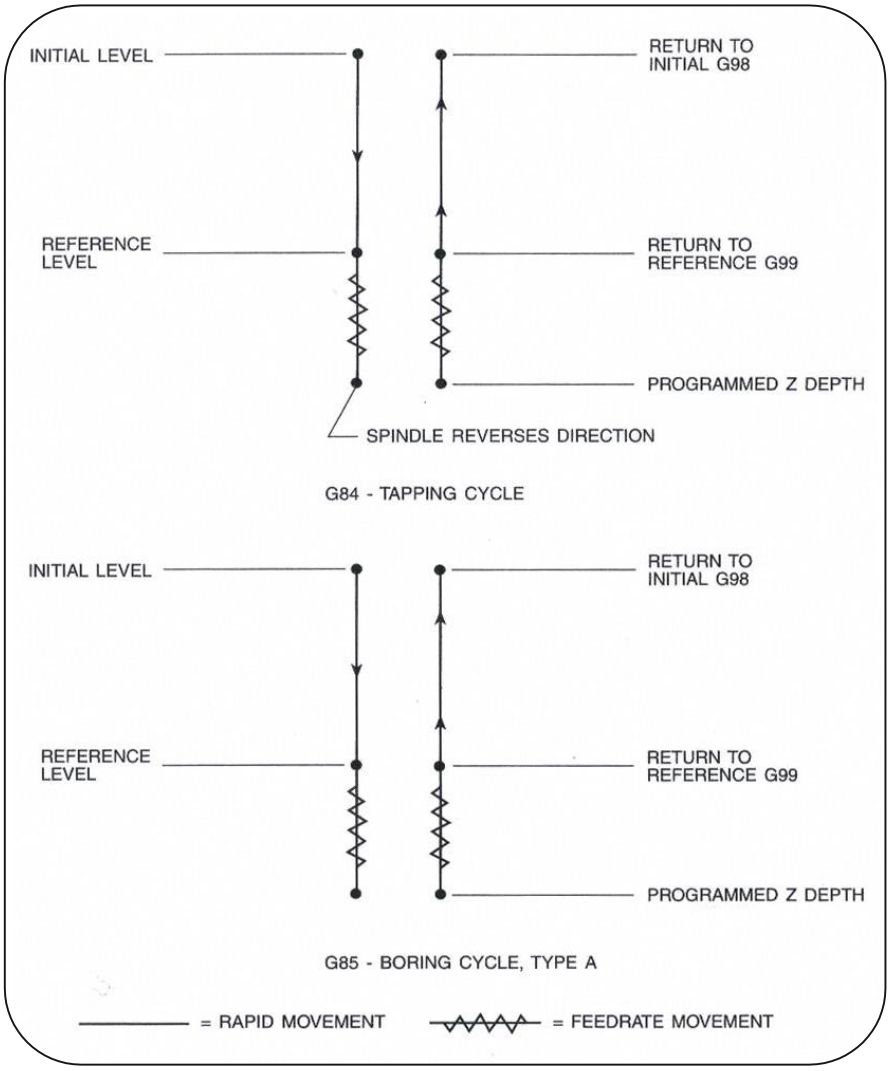


Figure 10:G codes for tapping and boring cycles

Other G Codes used in CNC Programming

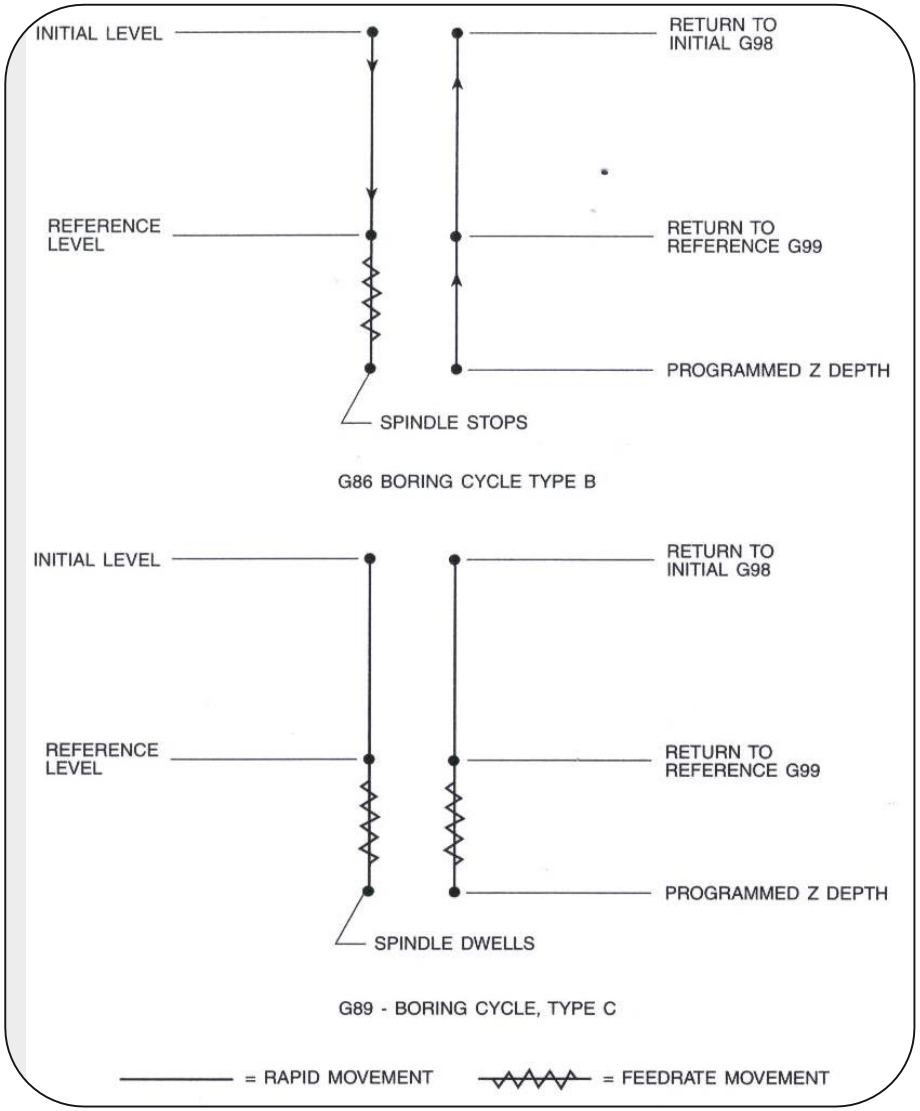
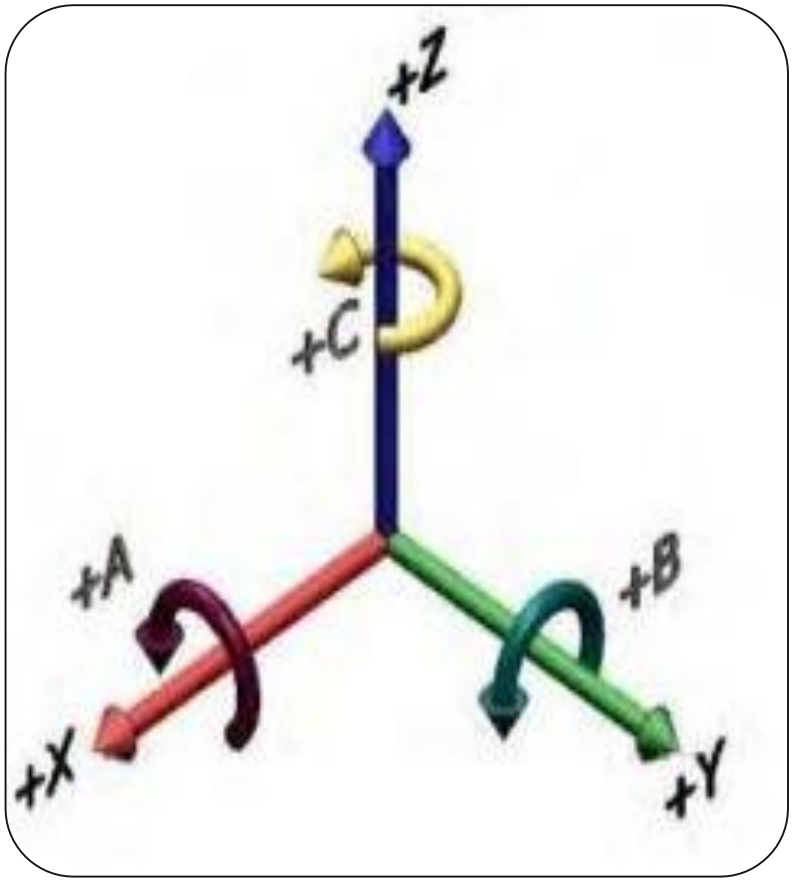


Figure 11: G codes for boring cycles

Using an Indexer



The common rotational axis are:

- **A-axis** = rotation around the X-axis
- **B-axis** = rotation around the Y-axis
- **C-axis** = rotation around the Z-axis

Figure 12:Common rotational axis

(photo courtesy of www.sae.org/)

Programming Example

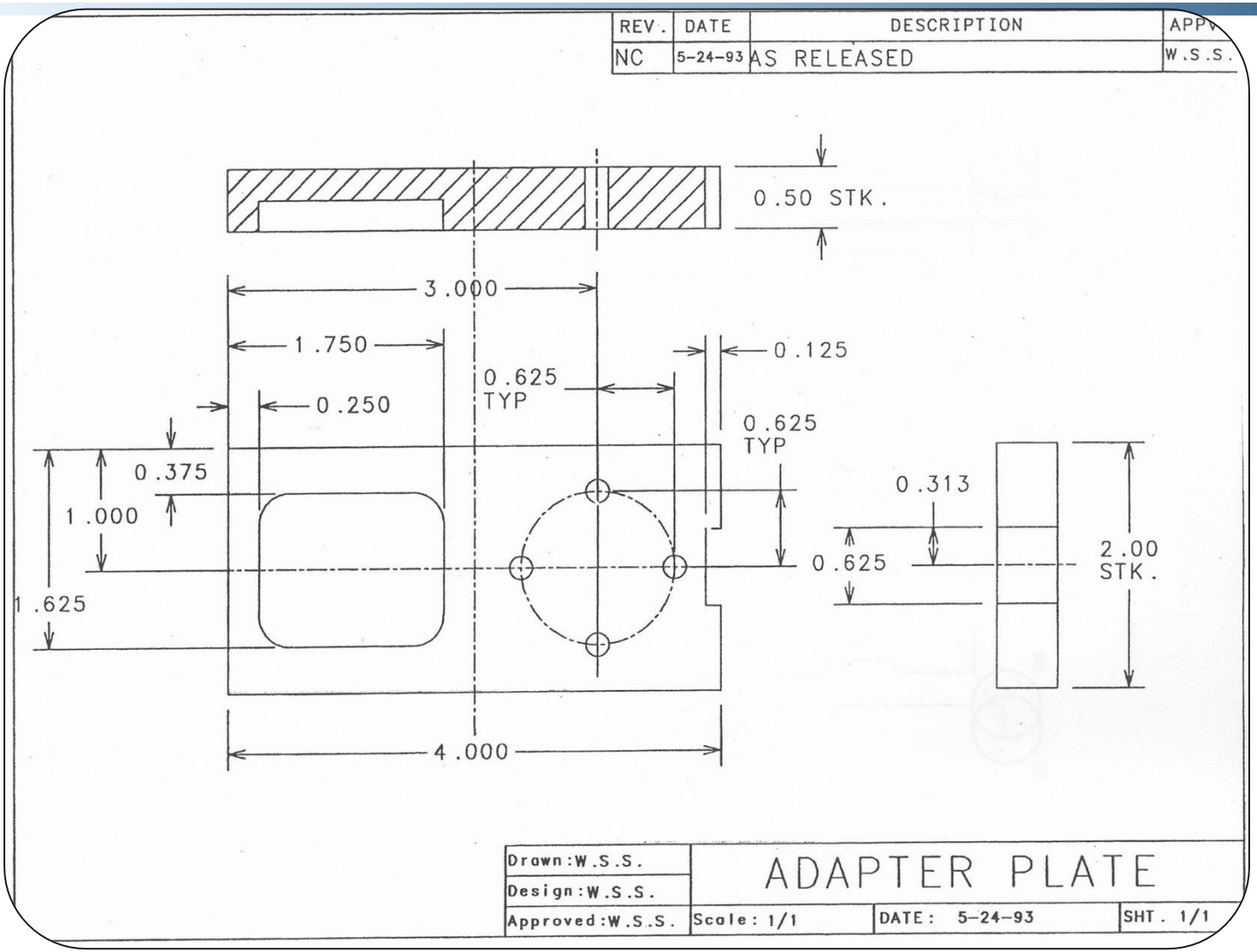


Figure 13: Part drawing for three-axis program using an indexer

Seams W., "Computer Numerical Control, Concepts & Programming"

Programming Example

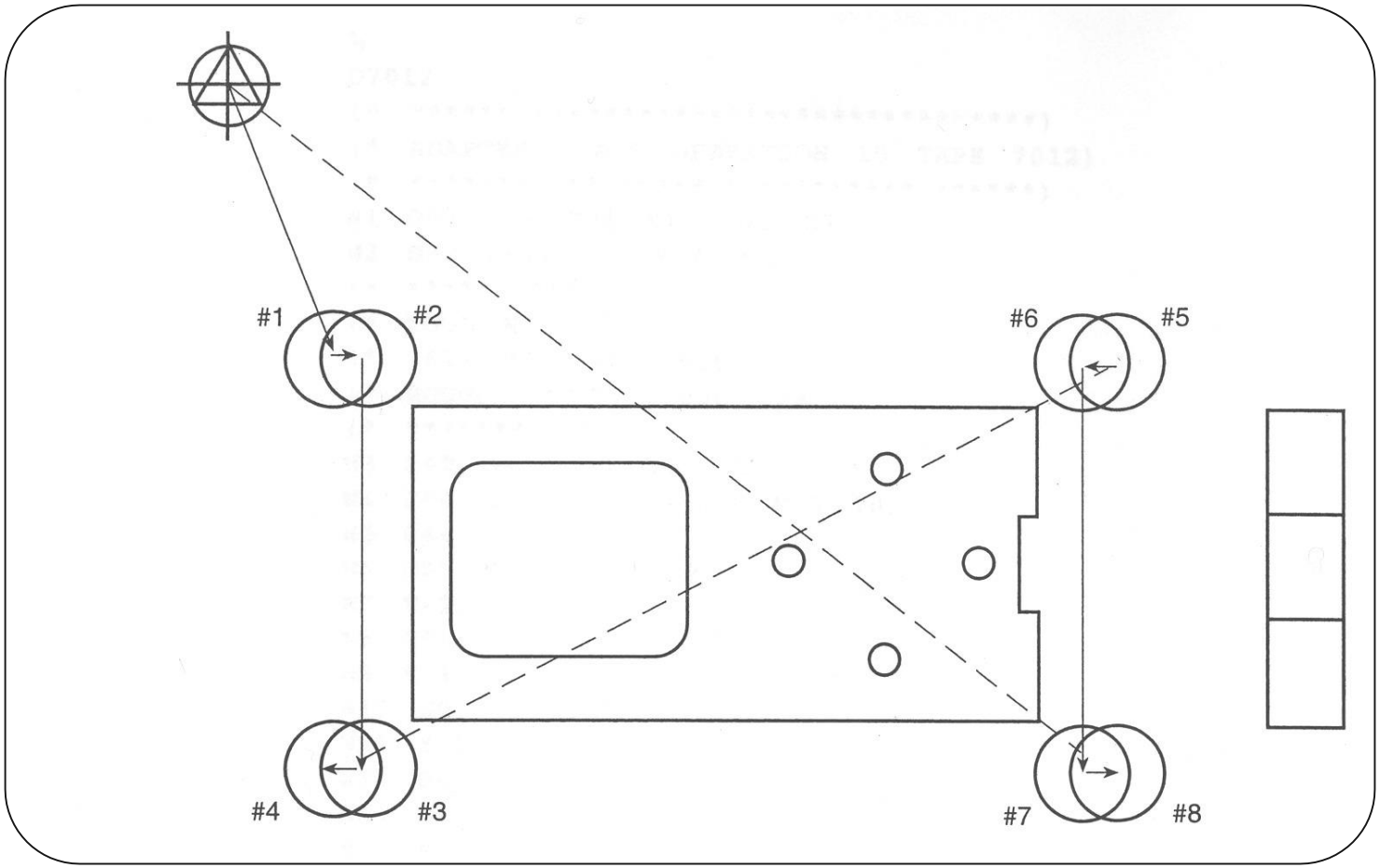


Figure 14: Cutter path for milling the outside of the part in Figure 13

Seams W., "Computer Numerical Control, Concepts & Programming"

Programming Example

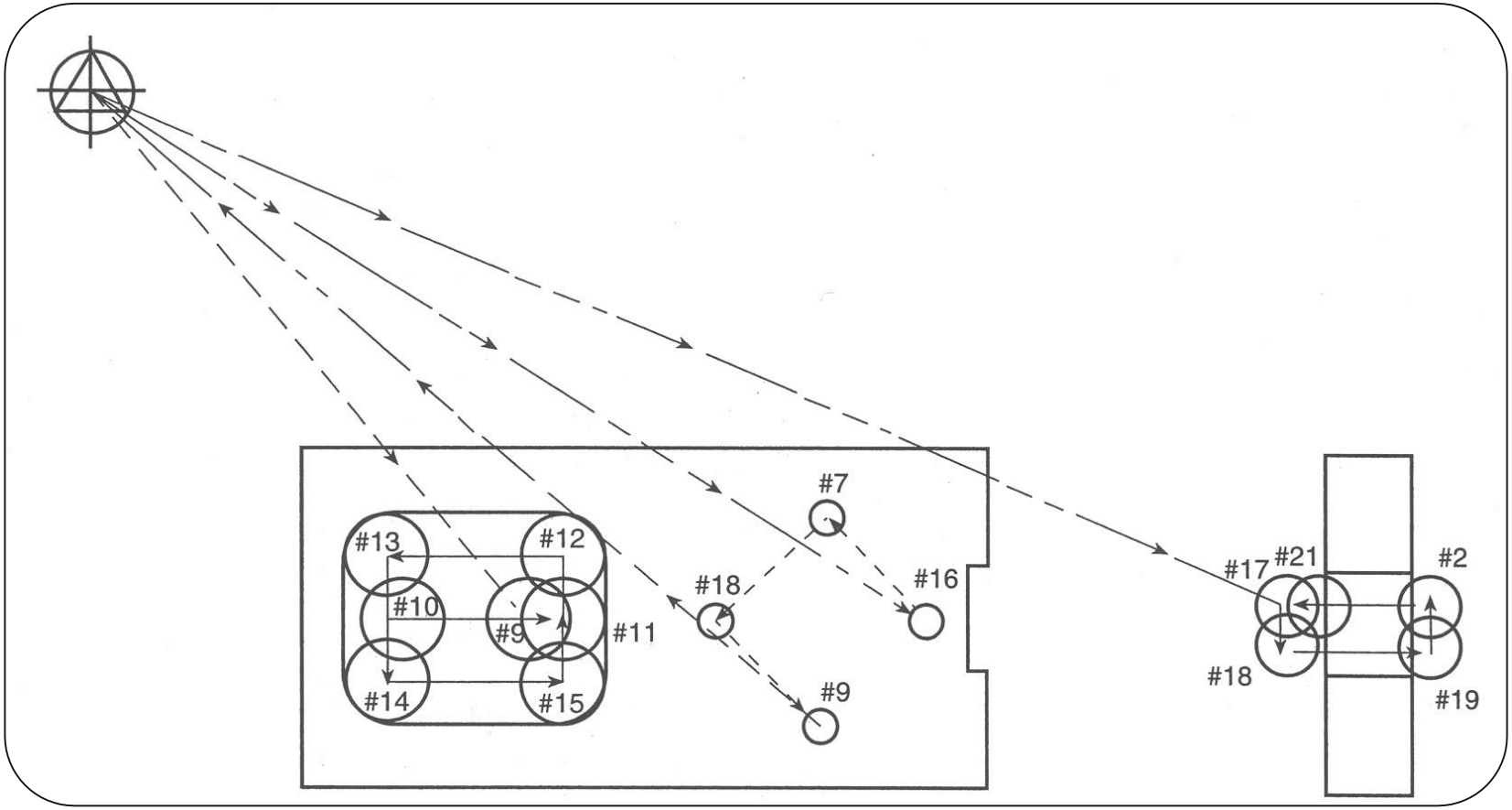


Figure 15: Cutter path for milling the slots and drilling the holes in the part in Figure 13

Seams W., "Computer Numerical Control, Concepts & Programming"

LAB

Programming Example

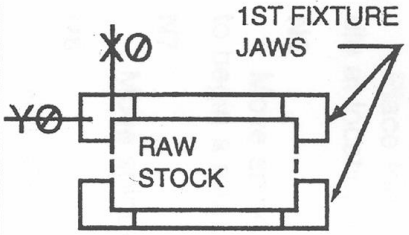
STA. NO.	TOOL DESCRIPTION
1.	5/8 HSS 4-FLT END MILL (Ø.625)
2.	1/2 HSS 4-FLT CENTER CUTTING END MILL (Ø.500)
3.	3/16 HSS STUB DRILL (Ø.1875)

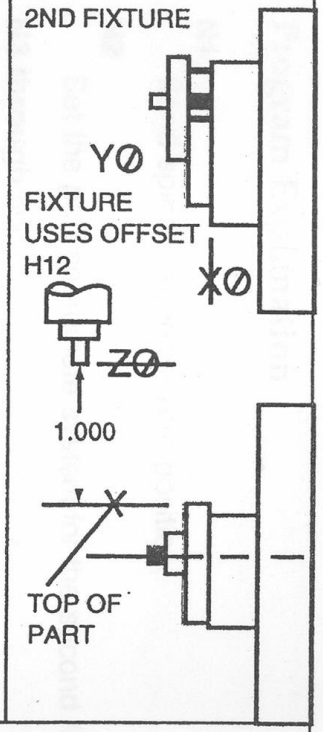
TAPE NUMBER: 7012

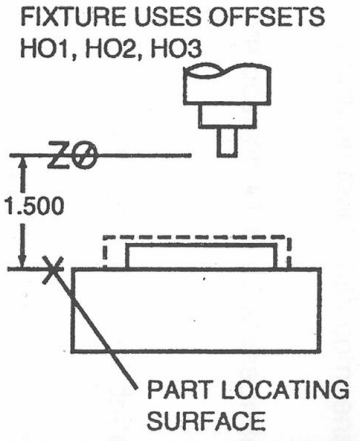
FIXTURE: FIX 1=J1000 FIX 2 = F1627

WORK COORDINATES/FIXTURE OFFSETS

92 FIX#1		92 FIX#2	
BANK SURF		BANK SURF	
BANK SURF		C/L INDEXER	
1.5 ABOVE BANK		1.0 ABOVE TOP	







REMARKS USE REMOVABLE VISE STOP ON FIX #1 TO LOCATE PART IN JAWS.

DRWN: WSS	PART NUMBER: ADAPTER PLATE	OP: 030
PRGM: WSS		
APVD: 7/20/93	MACHINE: UNIVERSAL VNC	

Figure 16:Setup sheet for part in Figure 13

Seams W., "Computer Numerical Control, Concepts & Programming"

LAB Programming Example

```
%
O7012
(* ************************************)
(* ADAPTER PLATE OPERATION 10 TAPE 7012)
(* ************************************)
N1 G07 G08 G04 X1. Y2. Z3.
N2 G92 X-12.752 Y-7.453 Z.0
(* **********)
(* TOOL NO. 1)
(* .615 DIA. END MILL)
(* ROUGH/FINISH 4.000 DIM.)
(* **********)
N3 G40 G90 G00 G80 T01
N4 G00 X-.5 Y.1 S1200 M03 T02
N5 G44 Z-.75 H01
N6 G01 X-.3225 F7.2
N7 Y-2.1
N8 G00 X-.5
N9 Y.1
N10 G01 X-.3125
N11 Y-2.1
N12 G00 Y.1
N13 Z.0
N14 X4.5 Y-2.1
N15 Z-1.05
N16 G01 X4.3225
N17 Y.1
N18 G00 X4.5
N19 Y-2.1
N20 G01 X4.3125
N21 Y.1
N22 G00 X4.5
N23 Z.0
N24 G00 G80 Z.0
N25 G49
N26 M01
(* **********)
(* TOOL NO. 2)
(* .500 DIA. END MILL)
(* ROUGH/FINISH MILL POCKET)
(* **********)
N27 G40 G90 G00 G80 T02
N28 G00 X1.33Y-1. S1500 M03 T03
N29 G44 Z.0 H02
N30 G01 X.67 Z-.74 F9.
N31 X1.49
N32 Y-.635
N33 X.51
N34 Y-1.365
N35 X1.49
N36 Y-1.
N37 X1.33
N38 Z-.75
N39 X1.5
N40 Y-.625
N41 X.5
N42 Y-1.375
N43 X1.5
N44 Y-1.
N45 Z-.753 F.05
N46 X.67
N47 G00 G80 Z.0
N48 G49
N49 M01
(* **********)
(* TOOL NO. 3)
(* .188 DIA. STUB DRILL)
(* DRILL HOLE PATTERN)
(* **********)
N50 G40 G90 G00 G80 T03
N51 G00X3. Y.0 S1800 M03
N52 G44 Z.0 H03
N53 G83 G99 X3.625 Y.0 Z-1.1064 R-.4 F7.2
N54 X3. Y.625
N55 X2.375 Y.0
N56 X3. Y-.625
N57 G80
N58 G00 G80 Z.0
N59 G49
N60 M01
N61 G91 G28 X.0 Y.0Z.0
N62M30
%
```

Figure 17: Program for part in figure 13

LAB Programming Example

Program Explanation

N1

Send spindle to home zero position.

N2

Set the part coordinate system for the second fixture.

N3 through N5

Place tool 1 in spindle, put tool 2 in tool change standby, position spindle at location #1, and turn on tool length compensation.

N6

Move spindle from position #1 to #2 at feedrate, positioning the spindle to begin a rough pass.

N7

Move spindle from position #2 to #3 at feedrate.

N8

Move spindle away from the part surface, from #3 to #4 at rapid.

N9

Rapid spindle from position #4 back to #1.

N10

Feed spindle from position #1 to #2, positioning the spindle for the start of a finish pass.

N11

Feed the spindle from #2 to #3, completing the finish pass.

N12

Rapid the spindle away from the part surface, from #3 to #4.

Seams W., "Computer Numerical Control, Concepts & Programming"

Programming Example

N13

Retract the spindle at rapid above the part.

N14

Rapid from position #4 to #5.

N15

Rapid the Z-axis to the final milling depth.

N16

Move spindle from position #5 to #6 at feedrate, positioning the spindle to begin a rough pass.

N17

Move spindle from position #6 to #7 at feedrate

N18

Move spindle away from the part surface, from #7 to #8 at rapid.

N19

Rapid spindle from position #8 back to #5.

N20

Feed spindle from position #5 to #6, positioning the spindle for the start of a finish pass.

N21

Feed the spindle from #6 to #7, completing the finish pass.

N22

Rapid the spindle away from the part surface, from #7 to #8.

N23

Retract the spindle at rapid above the part.

N24 through N26

Turn off the tool length compensation, and issue an M01 program optional stop code for operator convenience.

LAB

Programming Example

N27 through N29

Place tool 2 in spindle, tool 3 in standby. Position to location #9, and turn on too! length compensation.

N30

Move spindle from position #9 to #10 at feedrate.
Notice that this move was made in both X and Z. In this manner the cutter is ramped into the part rather than plunged. This requires much less cutting too! pressure. Although on a shallow pocket such as this one a straight plunge would work effectively, on deep pockets this ramping move prevents tool breakage.

N31

Move spindle from position #10 to #11 at feedrate. Not only does this rough the center area of the pocket, but t also removes the ramp left from the previous block.

N32

Move spindle from #11 to #12 at feedrate.

N33

Move spindle from #12 to #13.

N34

Move spindle from #13 to #14.

N35

Move spindle from #14 to #15.

N36

Move spindle from #15 to #11.

N37

Move spindle form #11 to #10.

N38

Position Z-axis to final milling depth.

N39

Move spindle from position #10 to #11.

Programming Example

N40

Move spindle from #11 to #12.

N41

Move spindle from #12 to #13.

N42

Move spindle from #13 to #14.

N43

Move spindle from #14 to #15.

N44

Move spindle from #15 to #11.

N45

Move spindle away from the part, .003 in X, .100 in Y. This small X-axis departure prevents the cutter from leaving a mark on the side of the pocket when the cutter stops its motion, prior to retracting the Z-axis.

N46 through N48

Tool cancel blocks.

N49 through N51

Place tool 3 in the spindle, tool 2 in standby. Turn on the tool length compensation, and position spindle over hole location #16.

N52

Turn on canned drill cycle and drill hole #16.

N53 through N55

Drill holes #17, 18, 19, and 20.

N56

Cancel the canned drill cycle.

N57 through N59

Tool cancel blocks.

Programming Example

N60 through 61

Return spindle to home zero. Establish a new part coordinate system for the second fixture.

N62

Put tool 2 in the spindle.

N63

Rotate the indexer (the A-axis) to zero degrees. This orients the part with the end to be machined up.

N64 through N65

Move to position #17, and turn on the tool length compensation. Notice that a different offset register is used for tool 2 this time. The original tool length offset was set relative to the first fixture's part coordinate system. It was necessary, therefore, to use a different register for fixture 2.

N66

Move from position #17 to #18. This positions the spindle for a roughing pass. There is .010 stock left on the sides of the slot at this point.

N67

Move from position #18 to #19.

N68

Move from #19 to #20.

N69

Move from #20 to #17.

N70

Move from #17 to #18. This positions the spindle for the finish pass.

N71

Move from #18 to #19.

N72

Move from #19 to #20.

Programming Example

N73 through N75

Tool cancel blocks.

N76 through N78

End of program blocks.

Summary 1/2

The important concepts presented in this chapter are:

- **Tool lengths** in three-axis machines must be **pre-set** by the operator. On some controls they can be pre-set in the program
- ***The initial level is the Z-axis spindle position*** when an 80 series canned cycle commences
- A **reference (or rapid) level is the Z-axis feedrate engagement point**, selected by the programmer
- **G98** selects a **return to initial level**, and **G99** selects a **return to reference level** when using 80 series G codes (canned cycle codes)
- **Canned cycles** are **routines built into the controller to simplify programming**.

Summary 2/2

- Values, called **parameters**, are passed to the control indicating **how the cycle is to perform, where the cycle is to begin, and how it should repeat**

- Positioning the spindle in two axis, then feeding with the third is called **$2^1/_2$-axis programming**

- Feeding with all three axes simultaneously is called **3-axis programming**

- **Indexers** often are used on CNC machinery. Positioning an index usually is just a matter of calling out the axis designator and a coordinate (i.e., AC, B270., A135.)

Vocabulary Introduced in this Chapter

- $2^1/_2$-axis programming
- 3-axis programming
- 4-axis programming
- Canned cycle
- Indexer
- Initial level
- Rapid level
- Reference level
- Rotary table

References

1. Chryssolouris G., «Manufacturing Systems: Theory and Practice», 2nd Edition, 2006, Springer-Verlag
2. http://www.dptechnology.com/
3. http://www.hardinge.com/
4. Kalpakjian S., «Manufacturing Engineering and Technology», 2nd Edition, 1992, Addison-Wesley Publishing company
5. Mattson M., “CNC Programming, Principles and Applications”, Delmar, 2002
6. Seams W., “Computer Numerical Control, Concepts & Programming”, 4th Edition, Delmar, 2002
7. Γ. Χρυσολούρης, «Συστήματα Παραγωγής Θεωρία και Πράξη» Μέρος Ι και ΙΙ, Εκπαιδευτικές Σημειώσεις, Πανεπιστήμιο Πατρών, 2001,
8. Σύγχρονες μέθοδοι κατεργασίας υλικών και προγραμματισμός με Ηλεκτρονικό Υπολογιστή (Η/Υ) ,Δ. Μούρτζης ,Κ. Σαλωνίτης