



SNS COLLEGE OF TECHNOLOGY

Coimbatore-35

An Autonomous Institution

Accredited by NBA – AICTE and Accredited by NAAC – UGC with 'A++' Grade

Approved by AICTE, New Delhi & Affiliated to Anna University, Chennai



DEPARTMENT OF MECHATRONICS ENGINEERING

UNIT IV – CNC PROGRAMMING

G & M CODE & MANUAL PART PROGRAMMING



MILLING



%

O0001 (Program Number)

G90 G54 (Absolute coordinates, work coordinate system G54)

G00 X0 Y0 Z0 (Rapid positioning to the starting point)

M03 S1000 (Spindle on, clockwise, at 1000 RPM)

G01 X10 Y5 Z-2 F200 (Linear cut to point (10, 5, -2) at a feed rate of 200 units/minute)

G02 X15 Y10 I2 J3 F150 (Clockwise circular cut with a radius of 2 units, center at (15, 10),
feed rate 150)

G01 X20 Y5 Z-2 (Linear cut to point (20, 5, -2))

M05 (Spindle off)

G00 Z5 (Rapid retract to Z5)

M30 (End of program)

%



MILLING



- The machine is in absolute positioning mode (**G90**).
- The work coordinate system is set to G54.
- Rapid positioning (**G00**) is used for non-cutting moves.
- Linear interpolation (**G01**) is used for straight-line cutting moves.
- Circular interpolation (**G02**) is used for clockwise circular cuts.
- Spindle is turned on in clockwise direction at 1000 RPM (**M03**).
- Feed rates are specified with **F**.
- The spindle is turned off (**M05**) after the cutting operations.
- Rapid retract to Z5 after the cutting is done.
- The program ends with **M30**.



POCKET CUTTING & DRILLING



%

O0001 (Program Number)

G90 G54 (Absolute coordinates, work coordinate system G54)

G00 X0 Y0 Z0 (Rapid positioning to the starting point)

M03 S1000 (Spindle on, clockwise, at 1000 RPM)

(First, let's drill a hole at the center of the pocket)

G00 Z5 (Rapid move to safe Z height)

G00 X10 Y10 (Position for the hole)

G01 Z-1 F100 (Move down to drill depth at a feed rate of 100 units/minute)

G81 R0.1 Z-5 F50 (Drill a hole with a 0.1-inch retract, to a total depth of 5 units, at a %
feed rate of 50 units/minute)

G80 (Cancel drilling cycle)

(Now, let's pocket cut a rectangular shape)

G00 Z2 (Rapid move to a safe Z height)

G00 X5 Y5 (Position for the pocket)

G01 Z-2 F200 (Move down to cut depth at a feed rate of 200 units/minute)

G01 X15 (Cut along the X-axis to create the pocket)

G01 Y15 (Cut along the Y-axis to create the pocket)

G01 X5 (Complete the pocket shape)

G00 Z2 (Rapid retract to a safe Z height)

M05 (Spindle off)

G00 X0 Y0 Z5 (Rapid move to home position)

M30 (End of program)



POCKET CUTTING & DRILLING



- A hole is drilled at the center of the pocket first.
- Then, a rectangular pocket is cut with a depth of 2 units.
- The G81 command is used for drilling. The R parameter specifies the retract distance, and the Z parameter specifies the total depth.
- G80 cancels the drilling cycle.
- Feed rates (F values) are set according to the desired cutting and drilling speeds.



APT LANGUAGE



APT (Automatically Programmed Tool) is a high-level programming language that was developed in the 1950s and 1960s for generating toolpaths in the field of computer-aided design and manufacturing (CAD/CAM). It was one of the earliest attempts to create a programming language specifically for controlling machine tools through numerical control.

APT is primarily used for defining tool motion and machining operations in a more abstract and human-readable way compared to low-level G-code. Here is a brief overview of APT with some examples:

APT Structure:

APT programs consist of a series of statements that describe machining operations. These statements are typically organized into blocks.



APT LANGUAGE



N1 GOTO 10
N2 GOTO 20
N10 CUTTER/ENDMILL DIA 0.5 LOC 0 0 0
N20 POCKET X 2 Y 3 Z -1 DEPTH 0.5
N30 GOTO 50
N40 DRILL LOC 1 2 0 DEPTH -1
N50 END



APT LANGUAGE



Explanation of Example Program:

- 1.The program starts by jumping to label 10 (**N1 GOTO 10**).
- 2.It defines a cutter/endmill with a diameter of 0.5 inches at the origin (**N10 CUTTER/ENDMILL DIA 0.5 LOC 0 0 0**).
- 3.It then specifies a pocket milling operation at coordinates (2, 3, -1) with a depth of 0.5 inches (**N20 POCKET X 2 Y 3 Z -1 DEPTH 0.5**).
- 4.The program jumps to label 50 (**N30 GOTO 50**).
- 5.It defines a drilling operation at coordinates (1, 2, 0) with a depth of -1 inch (**N40 DRILL LOC 1 2 0 DEPTH -1**).
- 6.The program ends (**N50 END**).