



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



G-CODE & M-CODE



G Code	Function
G00	Positioning at rapid travel;
G01	Linear interpolation using a feed rate;
G02	Circular interpolation clockwise;
G03	Circular interpolation, counterclockwise;
G04	Dwell
G17	Select X-Y plane;
G18	Select Z-X plane;
G19	Select Z-Y plane;
G20	Imperial units;
G21	Metric units;
G27	Reference return check;
G28	Automatic return through reference point;
G29	Move to a location through reference point;
G31	Skip function;
G32	Thread cutting operation on a Lathe;
G33	Thread cutting operation on a Mill;
G40	Cancel cutter compensation;
G41	Cutter compensation left;
G42	Cutter compensation right;
G43	Tool length compensation;
G44	Tool length compensation;
G50	Set coordinate system (Mill);
G50	Maximum RPM (Lathe);
G52	Local coordinate system setting;
G53	Machine coordinate system setting;
G54	Set Datum;
G55	Set Datum;
G56	Set Datum;
G57	Set Datum;

G Code	Function
G58	Set Datum;
G59	Set Datum;
G70	Finish cycle (Lathe);
G71	Rough turning cycle (Lathe);
G72	Rough facing cycle (Lathe);
G73	Chip break drilling cycle;
G74	Left hand tapping (Mill);
G74	Face grooving cycle;
G75	OD groove pecking cycle (Lathe);
G76	Boring cycle (Mill);
G76	Screw cutting cycle (Lathe);
G80	Cancel cycles;
G81	Drill cycle;
G82	Drill cycle with dwell;
G83	Peck drilling cycle;
G84	Tapping cycle;
G85	Bore in, bore out;
G86	Bore in, rapid out;
G87	Back boring cycle;
G90	Absolute programming;
G91	Incremental programming;
G92	Reposition origin point (Mill);
G92	Screw thread cutting cycle (Lathe);
G94	Per minute feed;
G95	Per revolution feed;
G96	Constant surface speed (Lathe);
G97	Constant surface speed cancel;
G98	Feed per minute (Lathe);
G99	Feed per revolution (Lathe);

M code	Description
M00	Program stop
M01	Optional program stop
M02	End of program
M03	Spindle start forward CW
M04	Spindle start reverse CCW
M05	Spindle stop
M06	Tool change
M07	Coolant ON – Mist coolant/Coolant thru spindle
M08	Coolant ON – Flood coolant
M09	Coolant OFF
M19	Spindle orientation
M28	Return to origin
M29	Rigid tap
M30	End of program (Reset)
M41	Low gear select
M42	High gear select
M94	Cancel mirrorimage



MANUAL PART PROGRAMMING



%

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)

(Tool change - Replace T1 with the actual tool number)

T1 M06

(First, drill three holes in a triangular pattern)

G00 Z5 (Rapid move to safe Z height)

G00 X5 Y5 (Position for the first 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)

%



MANUAL PART PROGRAMMING



G00 X10 Y10 (Move to the second hole position)

G01 Z-1 F100

G81 R0.1 Z-5 F50

G80

G00 X15 Y5 (Move to the third hole position)

G01 Z-1 F100

G81 R0.1 Z-5 F50

G80

(Next, mill a rectangular pocket)

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)

%