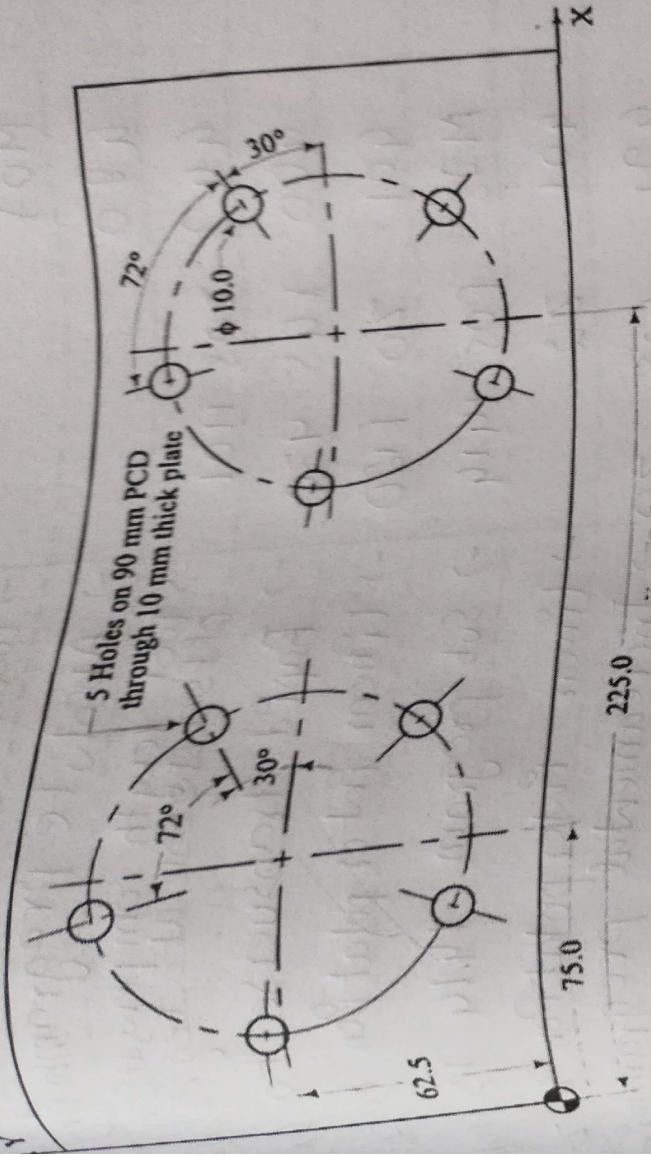


Exercise No.12 - Drilling along PCD

use a manual part programming to drill five holes on 90 mm PCD through 10 mm thick plate

use a manual part programming to drill five holes on 90 mm PCD through 10 mm thick



Operation	Tool	Machining Parameters		
		Speed (rpm)	Feed (mm/min)	Depth of cut (mm)
High speed peck drilling	Drill $\phi 10$	1200	120	2

G-1000E

Geometric Programming, XY plane Selection
Feed rate in mm/rev (Cutting compensation can be done)
Incremental programming, tool position moves to home position first along Z-axis then along X & Y axis
Tool selected from magazine 01
M06 T01

M03 S1200	→ Spindle on in CW direction at 1200
M07	→ Coolant 'ON'
G90	→ Absolute programming
G43 Z05 H01	→ Tool length compensation ON, offset stored at register 01
G00 X45 Y30	→ Rapid transverse to (X, Y, Z) : (45, 30)
G01 Z0 F120	→ Linear interpolation to Z=0.
M98 P05 1414	→ Sub program 1414 is called
G01 Z2	→ Linear interpolation to Z=2
G91 G28 Z0	→ Incremental programming tool moves to home position first along Z axis and then along X, Y axes.
M05	→ Spindle stop
M30	→ Program end with reset & rewind
01414	→ Sub program 1414
G91 G01 Z-1 F120	→ Incremental programming tool moves to depth of 1mm along Z axis, feed = 120 mm/min.
G90	→ Absolute programming
G52	→ Local coordinate system ON
G52 X72 Y62.5	
G16	→ Polar coordinate sys ON
G99 G73 X45 Y30 Z-12 R2 Q2 F10	→ location of peck drill with incremental depth of cut = 2mm, reference plane distance = 12mm

G102
 Y174
 Y246
 Y318
 G15 → Polar coordinates cancel
 G80 → Canned cycle cancel
 G52 X0 Y0 → Local coordinate system ON
 G52 X225 Y62.5
 G16 → Polar coordinate system ON

G99 G73 X45 Y30 Z-12 R2 G2
 → Location of peck drill with incremental depth of cut
 reference plane distance = 2mm, feed = 10mm/min
 depth of hole 12mm.

G102
 Y174
 Y246
 Y318
 G15 → Polar coordinates of peck drill

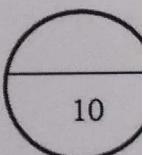


		Diagram: 1M	10 M	C01	L1
		Procedure: 2M		C02	L2
		Writing Program: 5M		C03	L3
		Simulation the program and print out: 2M		C04	L4

Faculty Signature with Date

G80
 G52 X0 Y0
 G91 G28 Z0
 G28 X0 Y0

Incremental programming, tool removes
 to home position first along Z axis,
 then along X, Y axis.

M05
 M30

→ Spindle stop
 → Program end with reset & rewind.

M07

→ Coolant switched 'ON'.

G00 X10 Y25 Z2

→ Rapid traverse to $(x, y, z) = (10, 25, 2)$

G01 Z0 F120

→ Linear interpolation along Z to Z=0

M98 P0S 3499

at Feed 120 mm/min.

→ Sub program '3499' called 5 times.

G00 Z2

→ Rapid traverse to Z=2.

G91 G28 Z0

→ Incremental programming to machine home position wrt Z axes.

G28 X0 Y0

→ Tool post to machine home position

M05

→ Spindle stop.

M30

→ Program ends with just {end}

O3499

→ Sub program '3499'

G91 G01 Z-1 F120

→ Incremental Programming, tool + depth processing by -1 mm, feed at

G90

→ Incremental program. at 120 mm/min

G01 X40

→ Tool at B

G03 X64 Y25 R12

→ Tool at C

G01 X82

→ Tool at D

G02 X118 Y25 R13

→ Tool at E

G01 X113 Y25

→ Tool at F

G01 X90 Y70

→ Tool at G

at H

10

G01	X50	→ Tool Diagram: 1M	10 M	C01	L1
G01	Y100	→ Tool at I Procedure: 2M		C02	L2
G03	X10 Y105 R5	→ Tool at J Writing Program: 5M Simulation the program and print out: 2M		C03	L3

G01 X30

→ Tool at K

Faculty Signature with Date

G03 X10 Y105 R5

G01 X30

FMS LAB, DR. NAGESH S & DR. KESHAV M

Page 29

G03 X10 Y105 R10

→ Tool at L

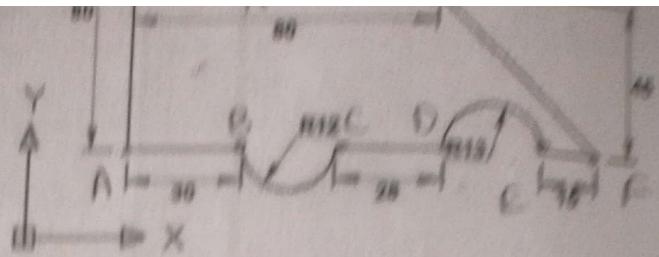
G01 X10 Y25

→ Tool at A.

M99

→ Return to main program to sub program.



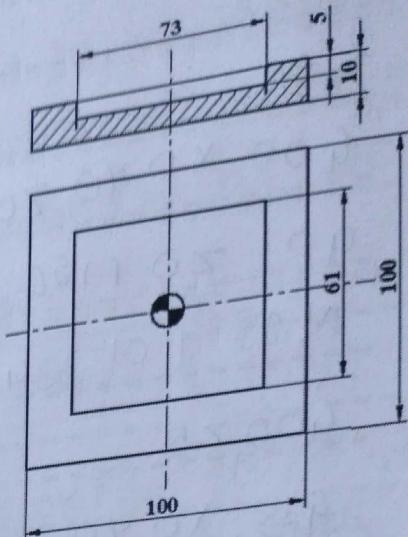
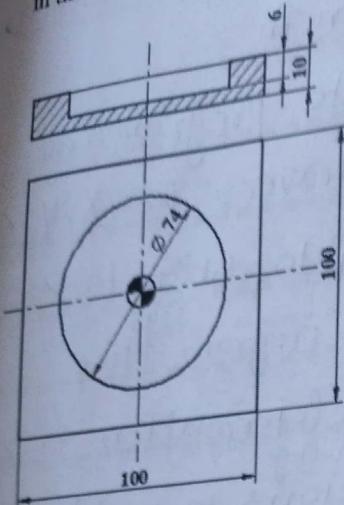


Operation	Tool	Machining Parameters			Remarks
		Speed r/min	Feed (mm/min)	Depth of cut (mm)	
Slot milling	End-mill 5 mm	1200	10	1	

CODE:

G17 G21 G94 G40	→ Program in metric system; XY plane; feed in feed/min; Tool radius compensation card.
G91 G28 Z0	→ Incremental programming & tool is moved home to Z=0.
G28 X0 Y0	→ Tool moved to machine home position with XY axes.
M06 T01	→ Tool selected from tool magazine post 01.
M03 S1200	→ Spindle switched 'ON' in CW direction at 1200 r/min.
G90	→ Absolute Programming
G43 Z05 H01	→ Tool length compensation

Exercise No.09 (Circular and Rectangular Pocket Milling)
 Write a part program to perform the linear slotting operation on the component as shown in the figure using subprogram,



Operation	Tool	Machining Parameters			Remarks
		Speed (rpm)	Feed (mm/min)	Depth of cut (mm)	
Circular Pocketing	End mill 10mm diameter	1200	120	1mm	
Rectangular Pocketing	End mill 10mm diameter	1200	120	1mm	

G CODE (Circular Pocketing)

G17 G21 G94 G40 → Program in metric system, XY plane,
 G91 G28 Z0 → Incremental programming & Tool is cancel
 G28 X0 Y0 → Tool moved to home position wrt
 FMS LAB, DR. NAGESH S & DR. KESHAV M X & Y axes. Page 37

M06 T01	→ Tool selected from tool mag
M03 S1200	→ Spindle on in (w direc)
M07	→ Coolant on
G90	→ Absolute programming
G00 X0 Y0 Z0	→ Rapid traverse to (X, Y)
G01 Z0 F120	→ Linear interpolation to Z <small>(3345)</small>
M98 P01 3345	→ Call sub program 1 time
G00 Z5	→ Linear interpolation to machining
G28 X0 Y0	→ Tool returns to machine
M05	→ Spindle stop.
M30	→ Program end with reset
03345	→ Sub program 3345
G91 G01 Z-1 F120	→ Incremental programming
G90	→ Absolute programming
G170 R0 R1 Q0.5 X0 Y0 Z-6 I0.2 J0.2	
G171 P80 S500 R0.1 F0.5 B1200 S0.1	→ Circular pocket: Z top = 0, Roughing cycle Incremental depth of cut 0.5 defines Pocket origin at (0,0) = (X) Total depth = 6mm, Spindle Σ bottom all = 0.2 each, radius = 37. Spindle Speed = 500 rpm.
M99	→ Return to main program from sub program.

G CODE (Rectangular Pocketing)

G17 G21 G94 F400	→ Program in metric system, XY plane, feed in Fe mm/sec
G91 G28 Z0	→ Incremental programming Tool moved to Z0.
G28 X0 Y0	→ Tool moved to machine home position
M06 T01	→ Tool '01' selected from tool magazine.
M03 S1200	→ Spindle on in CW at 1200rpm.
M07	→ Coolant 'ON'
G90	→ Absolute programming
G00 X0 Y0 Z0	→ Rapid traverse. → Tool moved to (X, Y, Z) = (0, 0, 0)
G01 Z0 F120	→ Linear Interpolation to Z0 at 120mm/min
M98 P1 3345	→ Calling of Sub program once ³³⁴⁵
G00 Z5	→ Rapid traverse to Z=5
G28 X0 Y0	→ Tool returns to machine home position
M05	→ Spindle Stop
M30	→ Program end with reset & rewind.

03345 → Sub program 3345

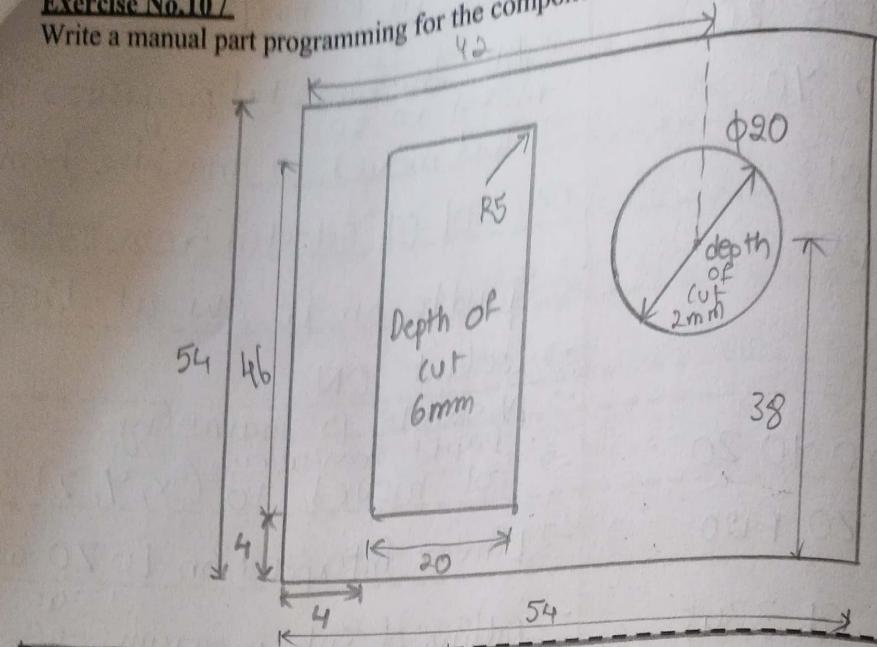
Rectangular pocket { G17 I73 J61 K0 P0 Q-0.5 R0 X13.5 Y19.5 Z-6
 G173 I0 K0 P50 T1 S3000 R75 F50 B3500 J500 Z5
 M99 → Return to main program from sub program

	Diagram: 1M	10 M	C01	L1
	Procedure: 2M		C02	L2
	Writing Program: 5M		C03	L3
	Simulation the program and print out: 2M		C04	L4

Faculty Signature with
Date

Rectangular pocket Parameters: length in X = 13 mm, length in Y = 61, Incremental depth of cut = 0.5 mm/s, Total depth = 6 mm, spindle speed: 3000 rpm, Retract after mill pass = 5 mm.

Exercise No. 10 /
Write a manual part programming for the component shown in figure below



Operation	Tool	Machining Parameters		
		Speed rpm	Feed mm/min	Depth of cut (mm)
Pocket Milling	End mill diameter 10mm	1200	120	1

G-CODE

G17 G21 G94 G40	→ Program in metric System, XY plane feed in mm/rev
G91 G28 Z0	→ Incremental programming, tool moves Z=0.
G28 X0 Y0	→ Tool post moves to (X, Y) (0,0) M06 T01
M06 T01	→ Tool '01' Selected from tool magazine.
M03 S1200	→ Spindle 'on' in CW direction

M07	→ coolant switched 'on'.
G90	→ Absolute programming.
G00 X 0 Y 0 Z 2	→ Tool moved to $(X, Y, Z) = (0, 0, 2)$
G01 Z 0 F 120	→ Linear Interpolation to $Z=0$ at $(X, Y) = (0, 0)$
M98 P01 3345	→ Subprogram 3345 called once. Feed 120mm/ min .
G00 Z 5	→ Rapid traverse to $Z=5$
G01 X 14 Y 27	→ Linear interpolation to $(X, Y) = (14, 27)$
M98 P01 3555	→ Subprogram '3555' called once. (rectangular)
G28 X 0 Y 0	→ Tool moves to machine home position
M05	→ Spindle stop.
M30	→ Program end with reset & rewind.
03345	→ Subprogram 3345
G91 G01 Z -1 F 120	→ Incremental programming for Z axis.
G90	→ Absolute programming.
G170 R0 P1 Q0.5 X 42 Y 38 Z -2 I 0.2 J 0.2 K 10	
G171 P80 S500 R0.1 F 0.5 B 1200 S0.1	→ Circular pocketing, Pocket from $Z=0$, Roughing cycle, Increasing depth of cut = 0.5mm, Pocket centred at $(42, 38) = (X, Y)$, Pocket depth = 2mm, radius of pocket = 10mm, Roughing speed = 500 rpm, feed = 0.5mm/rev
M99	→ Return to main program from sub program.

03555 → Sub program '3555'

G172 I20 J46 K0 P0 Q-0.5 R0 X4 Y4 Z

G173 I0 K0 P50 T1 S3000 R75 F50 B3500 J5
→ Pocket length 20mm $\frac{1}{2}$ x, Pocket length $\text{inf} = 46$,
corner radius = 0, Increasing depth of cut = 0.5 mm,
depth of base after cut = -6 mm

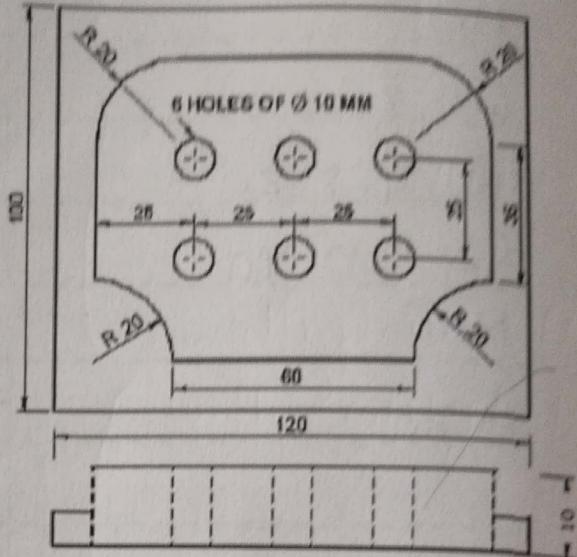
→ Side finishing allowance = 0, Base finishing all = 0, lateral
shift of tool = 50% , spindle speed = 3000, safe z position

M99 → Return to main program from Sub-program.

Diagram: 1M	10 M	C01	L1
Procedure: 2M		C02	L2
Writing Program: 5M		C03	L3
Simulation the program and print out: 2M		C04	L4
			10

Exercise No.08 slab milling and Drilling

Write a part program to perform the square slab milling operation on a workpiece of dimensions 120mm x 100mm x 10mm. the slab's dimension is 64mm x 64mm x 6 mm and is located at the center of workpiece. Use a cutter diameter of 10 mm. Take program zero at the left bottom end of the workpiece.



Operation	Tool	Machining Parameters			Remarks
		Speed (rpm)	Feed (mm/min)	Depth of cut (mm)	
Slab Milling	End mill diameter 10 mm	1200	120	1	
Drilling	Drill diameter 10mm	600	100	3	

G-CODE :

G17 G21 G94 G40
G91 G28 Z0.

→ Program in metric system, XY plane, Tool compensations cancel, feed in mm/min
→ Incremental programming and tool moved to Z=0.

G28 X0 Y0	→ Tool moved to machine home position.
M06 T01	→ Tool 01 selected from tool magazine. (Endmilling)
M03 S1200	→ Spindle 'on' at 1200 rpm
M07	→ Coolant switched 'ON'
G90	→ Absolute programming.
G43 Z05 H01	→ Tool length compensation on at 5mm
G01 G42 P18 X-40 Y-39.5 Z2 D65	Tool radius compensation 01 toward
G00 X-40 Y-39.5 Z2	→ Rapid traverse to $(X_1, Y_1) = (-40, -39.5)$ mm $(X_2, Y_2) = (-40, -39.5)$ mm
G01 Z0 F120	→ Linear interpolation to $Z=0$ at feed 120mm/min
M98 POS 1528	→ Subprogram '1528' called for 5 times
G01 Z2	→ Linear interpolation to $Z=2$
G01 Z2 F120	→ Linear interpolation to $Z=2$ at feed 120mm/min
G51 X0 Y0 P1.1	→ Scaling about $(X, Y) = (0, 0)$ with scaling factor 1.1
G00 X-40 Y-39.5 Z2	→ Rapid traverse to $(X_1, Y_1) = (-40, -39.5)$ mm
G01 Z0	→ Linear interpolation to $Z=0$
M98 POS 1528	→ Subprogram '1528' called.
G01 Z2	→ Linear interpolation to $Z=2$.
G50	→ Scaling 'off'
G01 Z2 F120	
G51 X0 Y0 P1.2	Scaling about $(X, Y) = (0, 0)$ with scaling factor 1.2
G00 X-40 Y-39.5 Z2	
G01 Z0	

M98 P05 1528

G01 Z2

G50

G01 Z2 F120

G51 X0 Y0 P1.3

G00 X-40 Y-39.5 Z2

G01 Z0

M98 P05 1528

G01 Z2

G50

G01 Z2 F120

G51 X0 Y0 P1.4

G00 X-40 Y-39.5 Z2

G01 Z0

M98 P05 1528

G01 Z2

G50

} Scaling about $(x,y) = (0,0)$
with scaling factor 1.2

} Scaling about $(x,y) = (0,0)$
with scaling factor 1.3

} Scaling about $(x,y) = (0,0)$
with scaling factor 1.4

10

	Diagram: 1M Procedure: 2M Writing Program: 5M Simulation the program and print out: 2M	10 M	CO1 CO2 CO3 CO4	L1 L2 L3 L4
--	---	------	--------------------------	----------------------

Faculty Signature with
Date

①

G01 Z2 F120

G51 X0 Y0 P1.5

G00 X-40 Y-39.5 Z2

G01 Z0

M98 POS 1528

G01 Z2

G50

G01 Z2 F120

G51 X0 Y0 P1.5

G00 X-40 Y-39.5 Z2

G01 Z0

M98 POS 1528

G01 Z2

G50

G90 G28 Z0

→ Absolute programming of tool to

Z0.

→ Tool moves to machine home position.

G28 X0 Y0

→ Tool 2 (drilling) selected from turret

M06 T02 M03 S600

→ Coolant 'ON'

Station '02

M07

→ Tool compensation off.

G40

→ Reference level return.

G99

G73 X-25 Y-12.5 Z-10 R2 G3 F100 → Peck drilling operation at $(X, Y) = (-25, -12.5)$

X0

→ Drill depth -10, retract 2mm, Feed = 100 mm/min.

X25

→ Peck drilling at $(X, Y) = (25, -12.5)$

Y12.5

→ Peck drilling → Y set to 12.5.

X25

→ Peck drilling at $(X, Y) = (25, 12.5)$

X0

→ Peck drilling at $(X, Y) = (0, 12.5)$



G00

M05

M30

→ Canned cycle cancel.

→ SPPndle stop.

→ Program end with reset & reini.

O 1528

→ Sub program '1528'

→ Incremental programming for

→ Absolute programming

G91 G01 Z-1 F120

G90

G01 X-30 Y-37.5

G01 X30 Y-37.5

G02 X50 Y-17.5 R20

G01 X50 Y17.5

G03 X30 Y37.5 R20

Body of the program

G01 X-30 Y 37.5

G03 X-50 Y17.5 R20

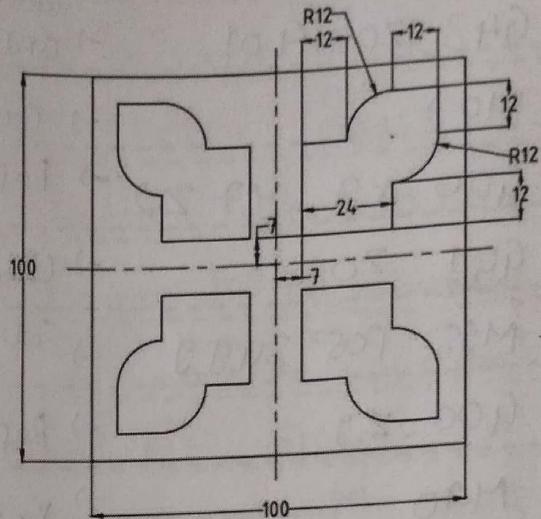
G01 X-50 Y17.5

G02 X-30 Y-37.5 R20

M99.

→ Return to main program from sub program

Exercise No. 07: Write a part program to perform the mirroring operation by assuming the profile using subprogram,



Operation	Tool	Machining Parameters			Remarks
		Speed (rpm)	Feed (mm/sec) min	Depth of cut (mm)	
Slot Milling	End mill diameter 5 mm	1200	10	1	

CODE:

G17 G21 G94 G40

→ Program in metric system, XY plane, feed in feed/min, tool radius compensation cancel.

G91 G28 Z0

→ Incremental Programming { tool is moved to home position wrt Z axis.

G28 X0 Y0

→ Tool moved to home position wrt XY axes

M06 T 01

→ Tool selected from tool magazine post 01.

M03 S1200

→ Spindle switched 'ON' in CW direction at 1200 rpm

G90	→ Program in absolute programming.
G43 Z05 H01	→ Tool length compensation wrt F2 axis at +5. → And tool offset stored at offset 01.
M07	→ Coolant switched 'ON'.
G00 X7 Y7 Z2	→ Rapid traverse to $(7, 7, 2) = (X, Y, Z)$
G01 Z0	→ Linear interpolation to $Z=0$.
M98 POS 34ggg	→ Sub program '34ggg' called 5 times.
G00 Z2	→ Rapid traverse to $Z=2$.
M70	→ X mirror ON
G00 X7 Y7 Z2	→ Rapid traverse to $(X, Y, Z) = (7, 7, 2)$
G001 Z0	→ Linear interpolation to $Z=0$.
M98 POS 34ggg	→ Sub program '34ggg' called 5 times.
G00 Z2	→ Rapid traverse to $Z=2$.
M80	→ X mirror OFF
M71	→ Y mirror 'On'
G00 X7 Y7 Z2	→ Rapid traverse to $(X, Y, Z) = (7, 7, 2)$
G01 Z0	→ Linear interpolation to $Z=0$.
M98 POS 34ggg	→ Sub program '34ggg' called 5 times.
G00 Z2	→ Rapid traverse to $Z=2$.
M180 M70	→ X mirror 'ON'
M71	→ Y mirror 'OFF' 'ON'
G00 X7 Y7 Z2	→ Rapid traverse to $(X, Y, Z) = (7, 7, 2)$
G01 Z0	→ Linear interpolation to $Z=0$.
M98 POS 34ggg	→ Sub program '34ggg' called for 5 times.

G00 Z2 → Rapid traverse to Z=2
 M80 → X mirror 'OFF'
 M81 → Y mirror 'OFF'
 G00 Z2 → Rapid interpolation to Z=t2
 G91 G28 Z0 → Incremental programming. Tool to
 machine home position wrt Z axis.
 G28 X0 Y0 → Tool moves to home position wrt
 X & Y axes
 M05 → Spindle Stop.
 M30 → Program end with reset & rewind.
 O3488 → Sub program '3488'
 G91 G01 Z-1 F120
 G90
 G00 X7 Y7
 G01 X31 Y7
 G01 Y19
 G03 X43 Y31 R12
 G01 X43 Y43
 G01 X81 Y43

Body of the program.

G03	X19	Y31	R12	Diagram: 1M	10 M	C01	L1
G01	X7	Y31		Procedure: 2M		C02	L2
Writing Program: 5M						C03	L3
Simulation the program and print out: 2M						C04	L4

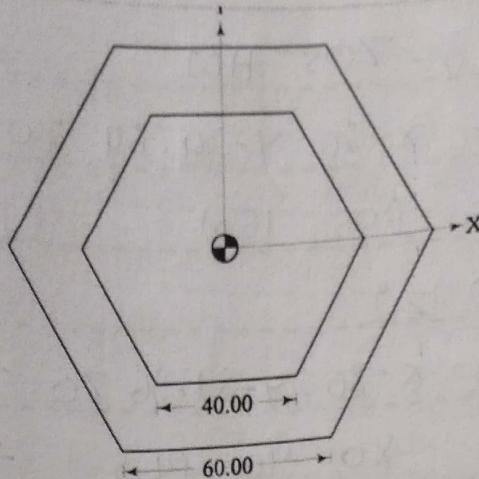
G01 X7 Y7	Faculty Signature with Date
-----------	-----------------------------

Mgg

→ Return to main program from sub program.

Exercise No.11 (Scaling)

Write a manual part programming to create two concentric hexagons of sides 40mm and 60mm respectively using scaling functions



Operation	Tool	Machining Parameters			Remarks
		Speed (crpm)	Feed (mm/min)	Depth of cut (mm)	
Slot milling with scaling	End mill 10 mm diameter	1200	120	1	

CODE:

G21 G17 G94 G40

→ Program in metric system, XY plane, feed in Feed 1/min, Tool radius compensation cancel

G91 G28 Z0

→ Incremental programming & tool is moved to machine home position wrt Z axis.

G0 G X0 Y0

→ Tool moved to machine home position wrt X & Y axes.

M06 T01

→ Tool selected from tool magazine posted

M03 S1200

→ Spindle switched on in CW direction at 1200 rpm.

M07	→ Coolant switched on.
G90	→ Absolute programming.
G43 Z05 H01	→ Tool length compensation wrt z axis at 5mm & stored in word offset table.
G00 X-20 Y-34.64 Z0	→ Rapid traverse to (-20, -34.64, 0)
M98 P05 1531	Sub program is called 5 times whose given '1531' name.
G00 Z2	→ Tool moves to Z=2 in rapid mode.
G00 X-30 Y-51.96 Z0	→ Rapid traverse to (-30, -51.96, 0)
G51 X0 Y0 P1.5	Scaling is switched on wrt X=0, Y=0 with a scaling factor 1.5
G00	
G01 Z1	→ Linear interpolation to Z=1
M98 P05 1531	Subprogram '1531' is called 5 times.
G50	→ Scaling cancel
G40	→ Tool radius compensation cancel
G28 Z0	→ Tool is brought to machine home position wrt z axes.
G28 X0 Y0	→ Tool is brought to machine home position wrt X=0, Y=0.
M05	→ Spindle Stop.
M30	→ Program end with reset & run
O15 31	→ Sub program '1531'
G91 G01 Z-1 F120	→ Incremental programming wrt z axis with increments of -1mm & feed 120
G90	→ Absolute program
G01 X-20 Y-34.64	} Body of the program.

601 X 20 Y -34.64 }
601 X 40 Y 0 }
601 X 20 Y 34.64 } Body of the program
601 X -20 Y 34.64 }
601 X -40 Y 0 }
601 X -20 Y -34.64 }
601 X 99 .

Return to main program
→ from sub program.

10

		Diagram: 1M		10 M		CO1		L1	
		Procedure: 2M		CO2		L2		L3	
		Writing Program: 5M		CO3		L4		CO4	