1. Preparatory Function [G-code]

It determines how the tool is to move to the programmed target.

i.e. Linear, Circular, PTP (or) any Specific operation [Rough turning, Facing, finishing etc]

Ex:- G00 Rapid traverse of tool.
G01 Linear —11—11—
G02 Circular interpolation in C.W
G03 —11— in C.C.W.
G04 Dwell [tool resides for sometimes to get Flat surface at the bottom position of drilling].

G71 Rough turning
G70 - Finishing
G72 - Facing.
G83 - Peck drilling.

2. Miscellaneous Code [M-Code]

It is used to control the Machine operation other than for co-ordinate movement.

i.e. M06 Tool change
M00 - Program Stop.
M01 - optional Stop
M30 - End of program.
M98 - Sub program call.

NC Program :-

It is a manual program in which set of instruction are coded by number, letter, symbols which are executed by a sequential manner through a Simulation Software

Namely : See NC Mill 6.0v for Milling operation
See NC turn 6.0v for turning operation

3 Basic Motion [NC]

1. Linear Motion / Continuous Motion:
The cutting tool travels in a linear manner. Ex: G00
G01

2. Contour Motion:
the tool will travels in a Circular path. ex:- G02 C.w
G03 CCW

3. Point to Point [PTP]:- Cutting tool will move from one point to another Point to Perform the activity Specified
Ex:- G83 [Peck drilling

CNC Program :-

Here the CAD model is prepared and the programmer will define the tool path so that the computer itself generate the program through software

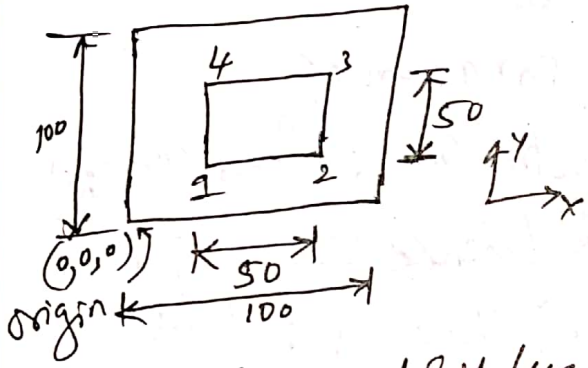
Ex:-

Software :- Caps mill

Caps turn.

2 Methods of Programming :-

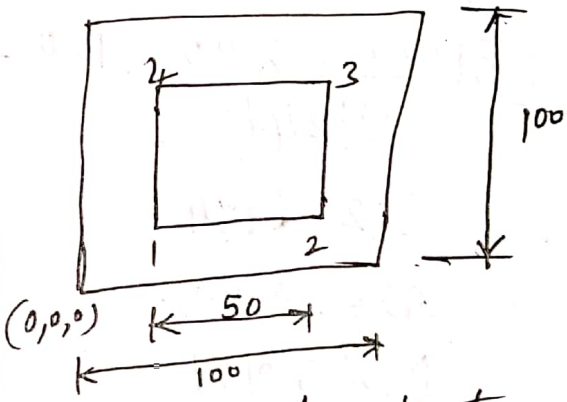
Absolute programming [G90]



Here Coordinates value are defined from the fixed origin

	X	Y
1	25	25
2	75	25
3	75	75
4	25	75

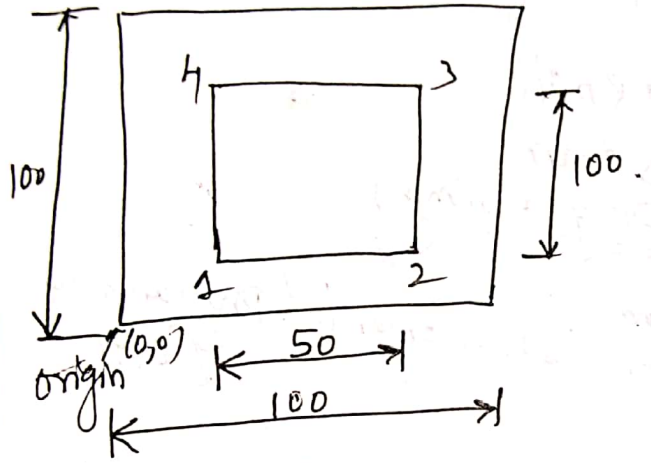
Incremental programming [G91]



Here the coordinates value are defined by taking previous value as origin.

	X	Y
1	25	25
2	50	0
3	0	50
4	-50	0
1	0	-50

1. Write a Manual Part programming to Perform Linear operation for the given profile. take cutter diameter 10mm & depth of Slot 2mm.

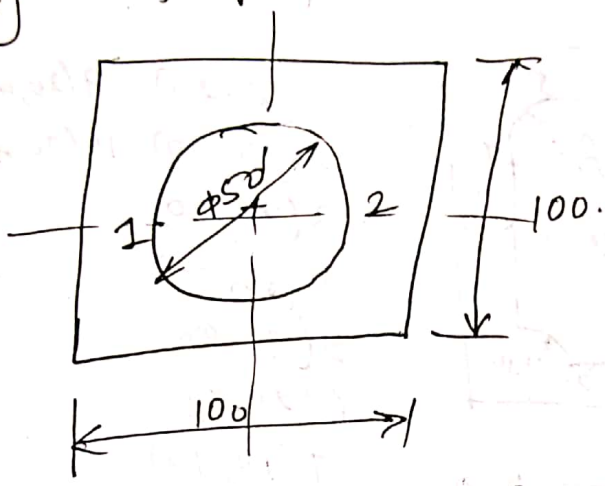


1 (25, 25)
2 (75, 25)
3 (75, 75)
4 (25, 75)

[BILLET X100 Y100 Z10.

```
O1244          Program No
G21 G94        metric(mm); feed(mm/min)
G28 X0 Y0 Z0   Reference position of tool.
M06 T0101      tool Change; tool no
M03 S1000      Spindle C.W; rpm.(speed)
G00 X25 Y25 Z5
G01 Z-2 F40    -(1)
G01 X75 Y25    -(2)
    X75 Y75    -(3)
    X25 Y75    (4)
    X25 Y25    (1)
G00 Z5
G28 X0 Y0 Z0
M05
M30]
```

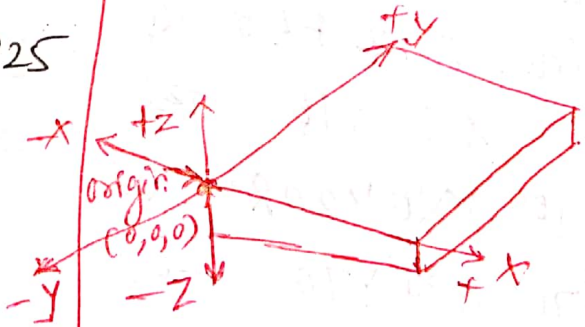
2. Write a Manual part program to perform Circular operation for the given profile. Take: BILLET 100X100X10



1 (25, 50)
2 (75, 50)
1 (25, 50)

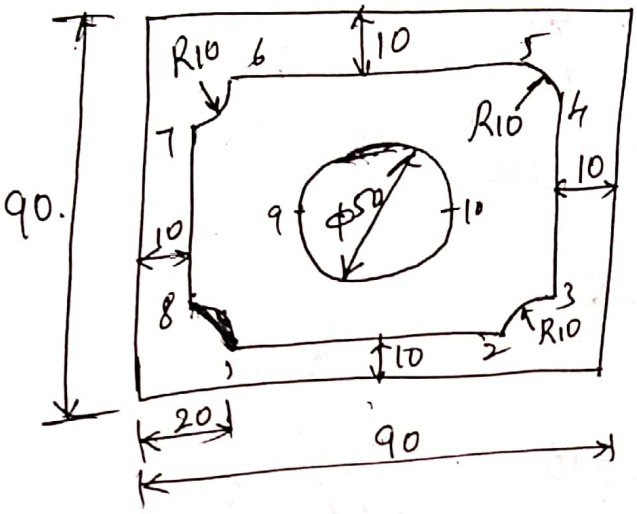
[BILLET X100 Y100 Z10.
O1445
G21 G94
G28 X0 Y0 Z0
M06 T0101
M03 S1000
G00 X25 Y50 Z5
G01 Z-2 F40
G02 X75 Y50 R25
G02 X25 Y50 R25
G00 Z5
G28 X0 Y0 Z0
M05
M30].

Coordination System for Milling operation



X, Y are dimension of w/p along x and y direction
Z represent depth / thickness of w/p.

3. write a manual part program to Perform Profile cutting operation for the given Profile take cutter dia 10mm

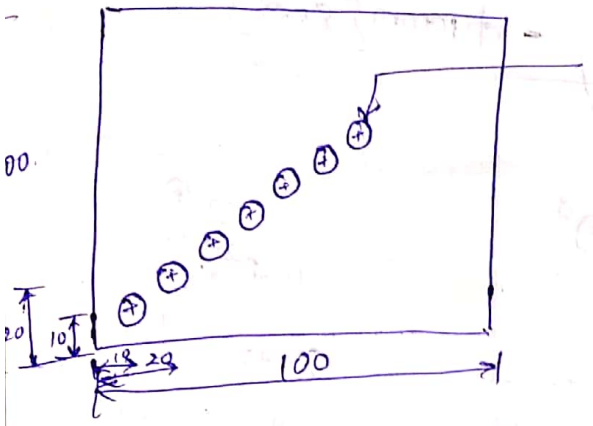


1 (10,20) 9 (20,45)
2 (70,10) 10 (70,45)
3 (80,20)
4 (80,70)
5 (70,80)
6 (20,80)
7 (10,70)
8 (10,20)

[BILLET X90 Y90 Z20.
O1475
G21 G94
G28 X0 Y0 Z0.
M06 T0101
M03 S1500
G00 X20 Y10 Z5
G01 Z-2 F40 (1)
X70 Y10 (2)
G02 X80 Y20 R10 (3)
G01 X80 Y70 (4)
G03 X70 Y80 R10 (5)
G01 X20 Y80 (6)
G02 X10 Y70 R10 (7)
G01 X10 Y20 (8)
X20 Y10. (9)

G00 Z5
G00 X20 Y45 Z2.
G01 Z-2 F40 (9)
G02 X70 Y45 R25 (10)
G02 X20 Y45 R25 (9)
G00 Z5
G28 X0 Y0 Z0
M05
M30]

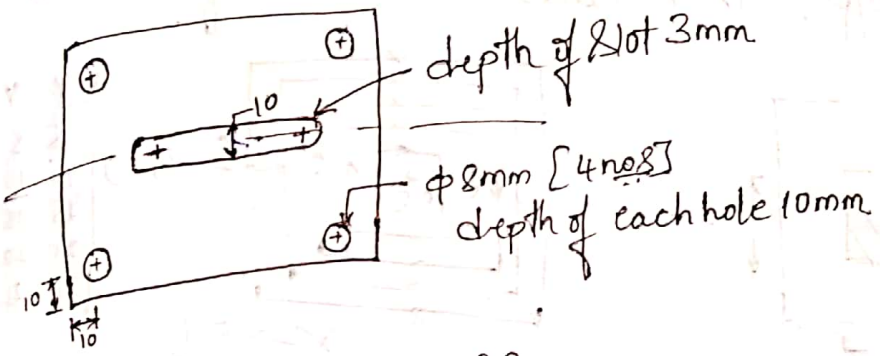
4. write a manual / NC part program to perform Peck drilling operation [G183].



Take Blank thickness = 10mm
Depth of each hole 8mm.

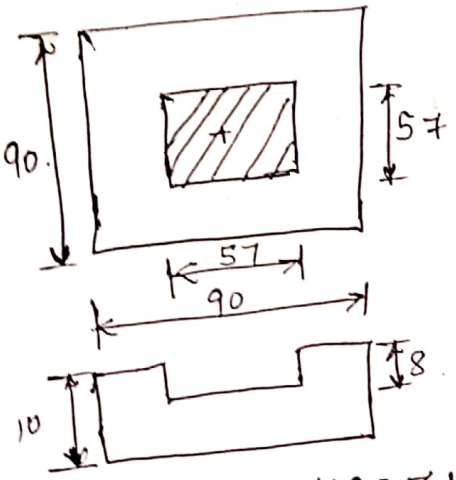
[BILLET X100 Y100 Z10
O1247
G21 G94
G28 X0 Y0 Z0.
M06 T0101 → Drill bit ϕ8mm
M03 S1000
G83 X10 Y10 Z-8 F40 Q4 R1 K1 -(1)
X20 Y20 -(2)
X30 Y30 -(3)
X40 Y40 -(4)
X50 Y50 -(5)
X60 Y60 -(6)
X70 Y70 -(7)
G80 Peck drilling Cancel
G28 X0 Y0 Z0
M05
M30]

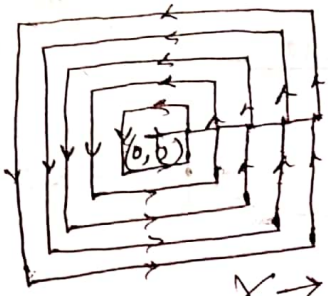
6) Write a manual part program to perform Profile Cutting and drilling operation for the given Profile Take BILLET SIZE 90X90X20 mm



[BILLET X90 Y90 Z20
O1245
G21 G94
G28 X0 Y0 Z0
M06 T01 → drill bit ϕ8mm
M03 S1500
G90 G00 X10 Y10 Z5
G83 X10 Y10 Z-10 F40 Q4 K1 R1
X10 Y80
X80 Y80
X80 Y10
G80
G28 X0 Y0 Z0.
M06T02 → End mill ϕ 10mm
M03 S1000
G90 G00 X30 Y45 Z2
G01 Z-3 F40
G01 X70 Y45
G00 Z5
G28 X0 Y0 Z0.
M05
M30].

7) write a manual part program to perform Square Pocket operation and workpiece Dimension 90X90X10mm, the pocket dimension 57X57X8mm and is located at the Centre of the w/p. use Cutter diameter of 10mm.





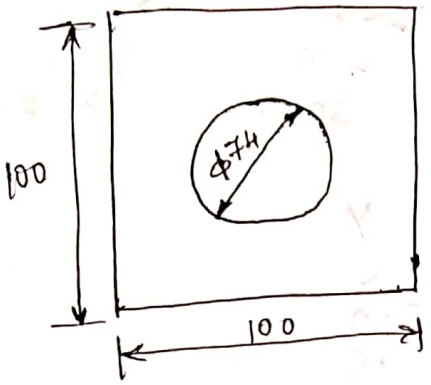
X	Y
5	5
10	10
15	15
20	20
23.5	23.5

X → 57 − 10 = 47 / 2 = 23.5

Y → 57 − 10 = 47/2 = 23.5

```
(BILLETX90 Y90 Z10
  01235
  G21 G94
  G28 X0 Y0 Z0
  M06 T0101 → Endmill ϕ10mm
  M03 S1000
G90 G00 Z5.
G01 X0 Y0 Z0 F30.
  M98 P0081277
  G00 Z5
G28 X0 Y0 Z0.
  M05
  M30.
  O1277
G91 G01 Z-1 F30
G90 G01 X5 Y0
  X5 Y5
  X-5 Y5
  X-5 Y-5
  X5 Y-5
  X5 Y0
  X10 Y0
  X10 Y10
  X-10 Y10
  X-10 Y-10
  X10 Y-10
  X10 Y0.
  X15 Y0
  X15 Y15
  X-15 Y15
  X-15 Y-15
  X15 Y-15
  X15 Y0.
  X20 Y0
  X20 Y20
  X-20 Y20
  X-20 Y-20
  X20 Y-20
  X20 Y0.
  X23.5 Y0
  X23.5 Y23.5
  X-23.5 Y23.5
  X-23.5 Y-23.5
  X23.5 Y-23.5
  X23.5 Y0.
G01 X0 Y0
  M99 ].
```

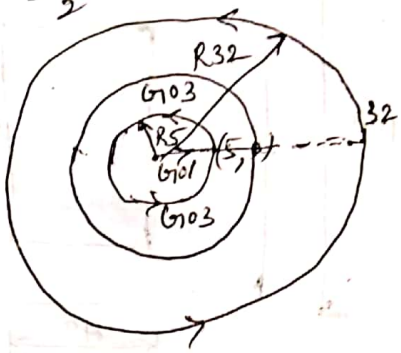
8) write a manual part program to perform Circular pocket having Dimension R32 X 6mm. Take Cutter diameter of 10mm.



$X \rightarrow 74 - 10 = 64$

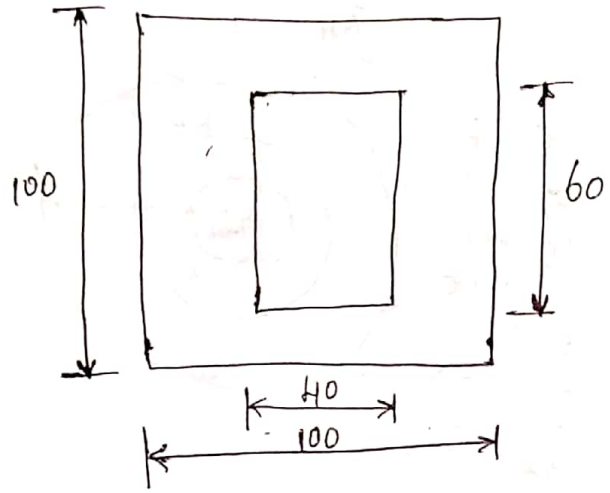
$\frac{64}{2} = 32$

X	R
5	5
10	10
15	15
20	20
25	25
30	30
32	32.



[BILLET X100 Y100 Z10
O1774
G21 G94
G28 X0 Y0 Z0.
M06 T0101 → Endmill ϕ10mm
M03 S1000.
G01 X0 Y0 Z0 F35
M98 P0061234
G00 Z5
G28 X0 Y0 Z0
M05
M30.
O1234
G91 G01 Z-1 F30
G90 G01 X5 Y0
G03 X-5 Y0 R5
G03 X5 Y0 R5
G01 X10 Y0.
G03 X-10 Y0 R10.
G03 X10 Y0 R10
G01 X15 Y0.
G03 X-15 Y0 R15
G03 X15 Y0 R15
G01 X20 Y0
G03 X-20 Y0 R20
G03 X20 Y0 R20
G01 X25 Y0
G03 X-25 Y0 R25
G03 X25 Y0 R25
G01 X30 Y0
G03 X-30 Y0 R30.
G03 X30 Y0 R30.
G01 X32 Y0
G03 X-32 Y0 R32
G03 X32 Y0 R32
G01 X0 Y0
M99]

9) Write a manual Part program for a Rectangular Pocketing as shown in figure. Take cutter diameter as 10mm and w/p centre as origin.



X → 40 − 10 = 30 / 2 = 15

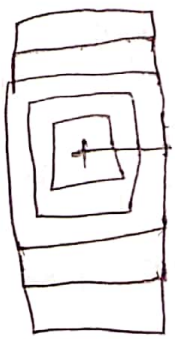
Y → 60 − 10 = 50 / 2 = 25

X	Y
5	5
10	10
15	15
15	20
15	25

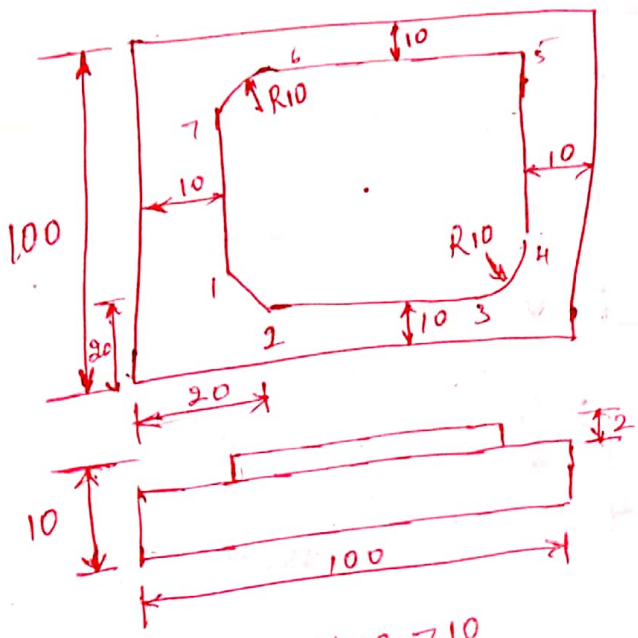
```
[BILLET X100 Y100 Z10
    O1774
  G21 G94
  G28 X0 Y0 Z0
  M06 T0101 → End mill ϕ10mm.
  M03 S1000
G00 X0 Y0 Z0 F40
  M98 P08 1245
    G00 Z5
* G28 X0 Y0 Z0
    M05
    M30.
  O1774
G91 G01 Z-1 F40
G90 G01 X5 Y0
    X5 Y5
    X-5 Y5
    X-5 Y-5
    X5 Y-5
    X5 Y0
```

```
X10 Y0
X10 Y10
X-10 Y10
X-10 Y-10
X10 Y-10
X10 Y0
.
.
.
X15 Y0
G01 X0 Y0
  M99]
```

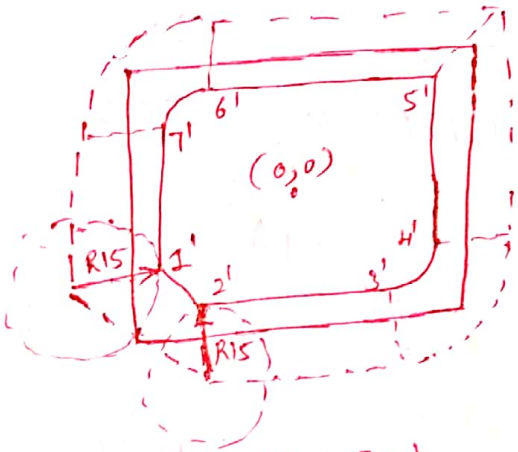
Tool Path Movement



10) Write an NC Program for the given profile to Perform Endmilling operation. Take cutter diameter 30mm.



1 (-40, -30)
2 (-30, -40)
3 (30, -40)
4 (40, -30)
5 (40, 40)
6 (-30, 40)
7 (-40, 30)

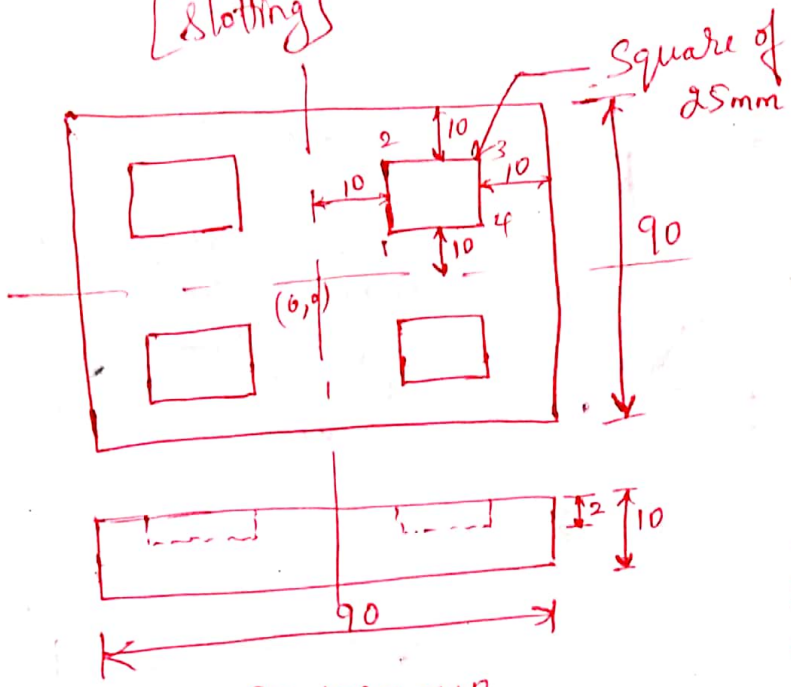


1' (-55, -30)
2' (-30, -55)
3' (30, -55)
4' (55, -30) R25
5' (55, 55)
6' (-30, 55)
7' (-55, 30) R25

[BILLET X100 Y100 Z10
O1145
G21 G94
G28 X0 Y0 Z0
M06 T0101
M03 S500
G90 G00 X-55 Y-30 Z2
G01 Z-2 F40 -(1)
X-30 Y-55 (2')
X30 Y-55 (3')
G03 X55 Y-30 R25 (4')
G01 X55 Y55 (5')
X-30 Y55 (6')
G03 X-55 Y30 R25 (7')
G01 X-55 Y-30 (1')
G00 Z5
G28 X0 Y0 Z0
M05
M30].

11) Write a manual part program to perform mirror operation for the given profile.

[Slotting]



1 (10, 10)
2 (10, 35)
3 (35, 35)
4 (35, 10)

[BILLET X 90 Y90 Z10

O1774

G21 G94

G28 X0 Y0 Z0

M06 T0101 → End mill ϕ10mm

M03 S1000

G90 G01 X10 Y10 Z0 F40

M98 P021234

G00 Z5

M70 → X mirror ON (All +Y → -Y)

G01 X10 Y10 Z0 F40

M98 P021234

G00 Z5

M71 → Y mirror ON (All +X → -X)

G01 X10 Y10 Z0 F40

M98 P021234

G00 Z5

M80 → X mirror OFF [All -Y → +ve Y]

G01 X10 Y10 Z0 F40

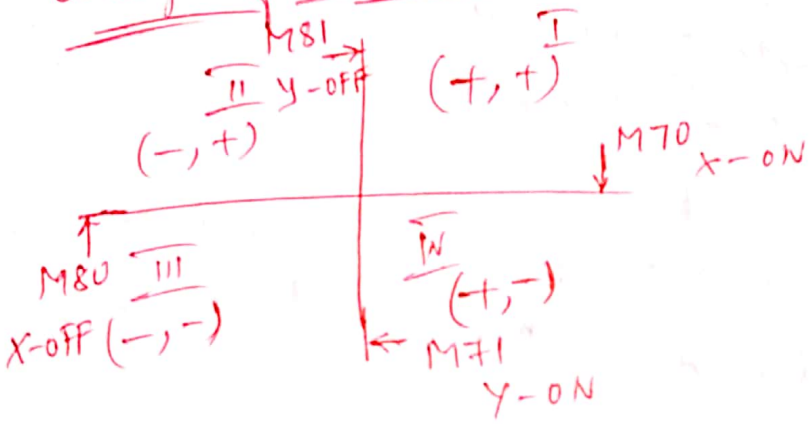
M98 P021234

G00 Z5

M81 → Y mirror OFF (All -X → +ve X)

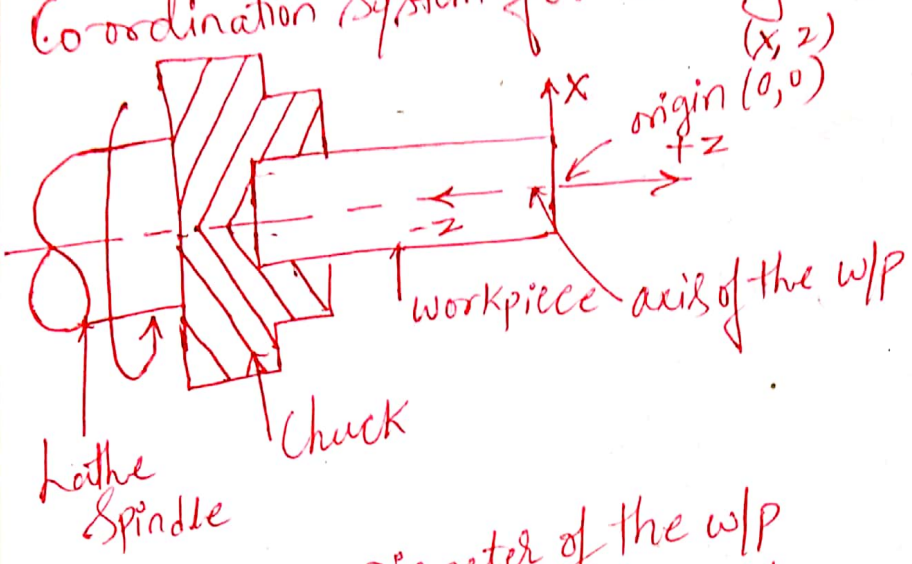
G28 X0 Y0 Z0
M05
M30
O1234
G91 G01 Z-1 F40
G90 G01 X10 Y35 -(2)
X35 Y35 -(3)
X35 Y10 (4)
X10 Y10 -(1)
M99].

Concept of Mirror operation



Turning :-

Coordination System for turning operation.



where X → Diameter of the w/p
Z → Length of the w/p.

In turning operations w/p will have rotational motion and tool is held stationary on the tool post.

Cutting parameters :-

1. **Depth of Cut [DOC]** :- It is the Perpendicular distance b/w machined and un machined surface. unit : mm.

2. **Feed (mm/min @ mm/rev)** :- F

Distance travelled by the tool per minute @ per revolution.

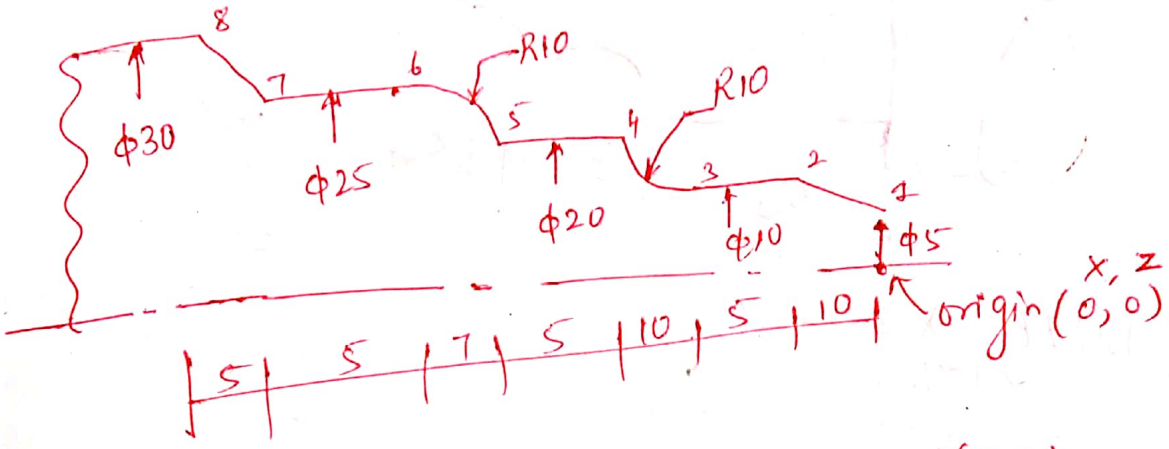
3. **Spindle Speed (rpm)** :- S

Number of rotation made by the tool / workpiece per minute.

1. Write a manual part program to perform turning operation without Canned cycle take cutting tool as External turn of nose radius 0.8mm.

(01)

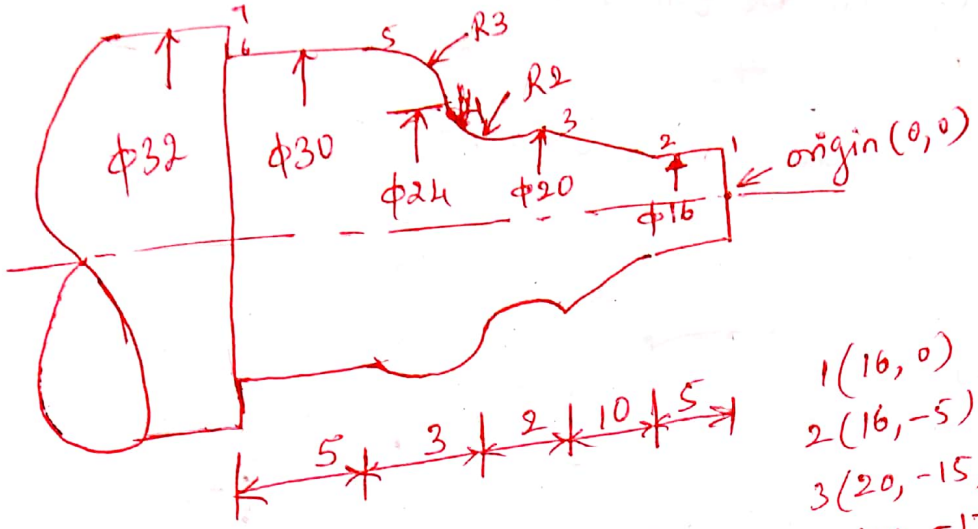
Linear and Circular interpolation.



1 (5, 0)
2 (10, -10)
3 (10, -15)
4 (20, -25)
5 (20, -30)
6 (25, -37)
7 (25, -42)
8 (30, -47)

[BILLET X30 Z50
01245
G21 G98 mm; mm/min
G28 U0 W0 Reference position of tool.
M06 T01 → Ext turn of nose radius 0.8mm
M03 S1500
G00 X30 Z2 initial safe position.
G00 X5 Z0 (1)
G01 X10 Z-10 F40 (2)
G01 X10 Z-15 (3)
G02 X20 Z-25 R10 (4)
G01 X20 Z-30 (5)
G03 X25 Z-37 R10 (6)
G01 X25 Z-42 (7)
X30 Z-47 (8)
G00 X30 Z2
G28 U0 W0
M05 M30]

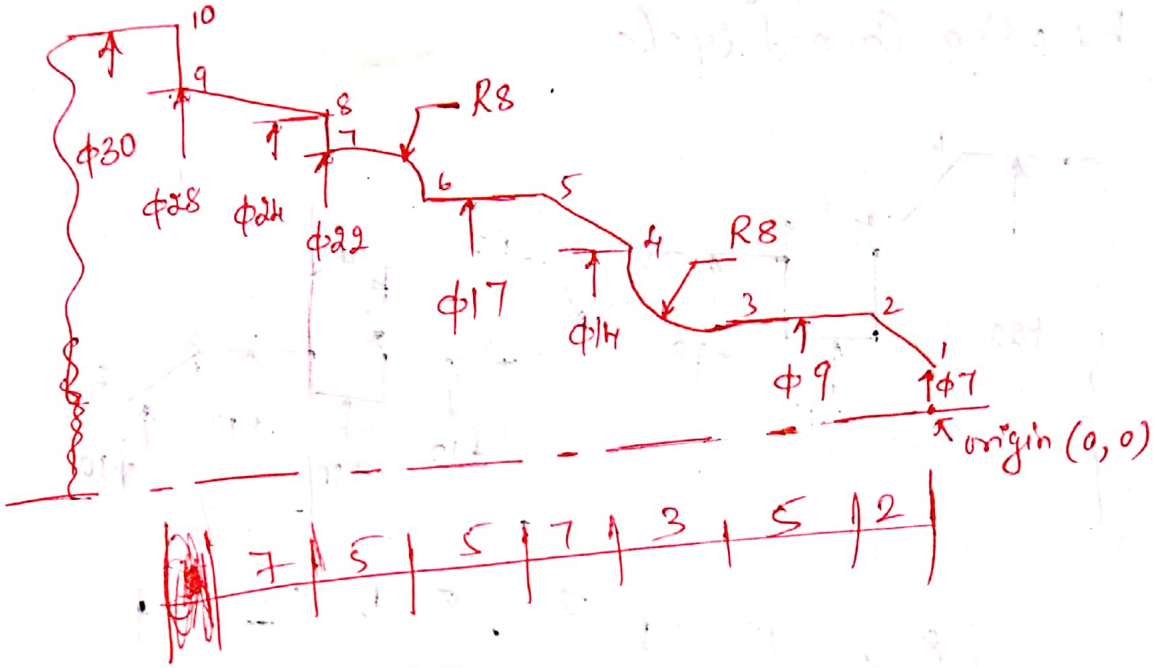
1. Write a manual part program to perform Multiple turning operation by using canned cycle.



1 (16, 0)
2 (16, -5)
3 (20, -15)
4 (24, -17)
5 (30, -20)
6 (30, -25)
7 (32, -25)

[BILLET X 32 Z 30
O1748
G21 G98 mm; mm/min
G28 U0 W0
M06 T01 → Ext tool 0.8mm
M03 S1200
G00 X32 Z2 → initial safe position of tool
G71 U0.5 R1 → U0.5 :- Amount of material removed in x-axis (0.5mm)
R :- Return (1mm)
G71 P10 Q20 U0.5 W0.5 F40 → P10 Q20 :- Profile starting & ending no.
U0.5 W0.5 :- Amount of stock maintained on the w/p for finishing cycle
N10 G00 X16 Z0 (1)
G01 X16 Z-5 (2)
X20 Z-15 (3)
G02 X24 Z-17 R2 (4)
G03 X30 Z-20 R3 (5)
G01 X30 Z-25 (6)
G01 X32 Z-25 (7)
N20 G00 X32 Z2
G70 P10 Q20
G28 U0 W0
M05
M30]

3. write a manual part program to perform Multiple turning operation by using Canned cycle [G71 & G70]

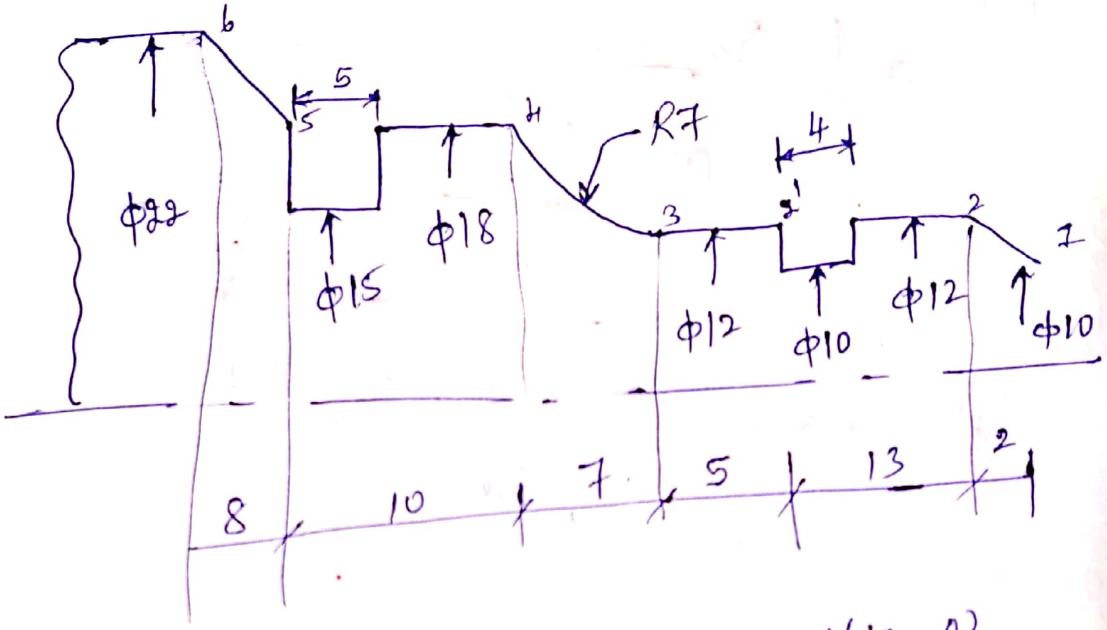


[BILLET X30 Z40
O8989
G21 G98 mm; mm/min
G28 U0 W0
M06 T01
M03 S1800
G00 X30 Z2
G71 U0.5 R1
G71 P10 Q20 U0.5 W0.5 F40
N10 G00 X7 Z0 (1)
G01 X9 Z-2 (2)
G01 X9 Z-7 (3)
G02 X14 Z-10 R8 (4)
G01 X17 Z-17 (5)
X17 Z-22 (6)
G03 X22 Z-27 R8 (7)
G01 X24 Z-27 (8)
X28 Z-34 (9)

1 (7, 0)
2 (9, -2)
3 (9, -7)
4 (14, -10) R8
5 (17, -17)
6 (17, -22)
7 (22, -27) R8
8 (24, -27)
9 (28, -34)
10 (30, -34)

X30 Z-34 (10)
N20 G00 X30 Z2
G70 P10 Q20
G28 U0 W0
M05
M30].

4) Write a manual part program to perform External grooving operation for the given profile by using Canned cycle

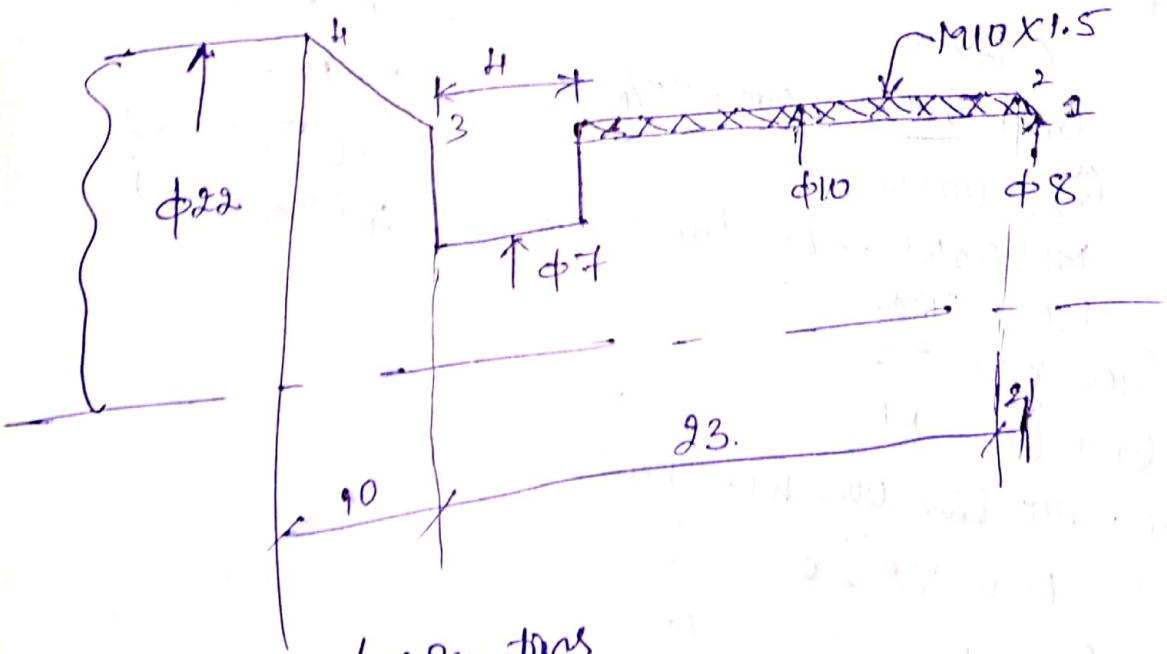


1(10, 0)
2(12, -2)
3(12, -20)
4(18, -27)
5(18, -37)
6(22, -45)
Grooving
2'(12, -15)
5(18, -37)

%BILLET X22 Z50
O1247
G21 G98
G28 U0 W0
M06 T0101 Ext tool 0.8mm
M03 S1000
G00 X22 Z2
G71 U0.5 R1
G71 P10 Q20 U0.5 W0.5 F40
N10 G00 X10 Z0 (1)
G01 X12 Z-2 (2)
X12 Z-20 (3)
G02 X18 Z-27 R7 (4)
G01 X18 Z-37 (5)
G01 X22 Z-45 (6)
N20 G00 X22 Z2

G70 P10 Q20
G28 U0 W0
M06 T0202 Ext groove [3mm width]
M03 S1200
G00 X12 Z-15 initial position of the grooving tool
G75 R1 → Relief for the tool.
G75 X10 Z-14 P250 Q500 F10 P250 and Q500 are the movement of tool along X & Z direction in 'μ'
G28 U0 W0
G00 X18 Z-37 initial position of the grooving tool.
G75 R1
G75 X15 Z-35 P250 Q500 F10.
G28 U0 W0
M05
M30]

5) Write a part program to perform threading operation on the component shown in figure



Soln:- Sequence of operations
G71 → Rough turning
G70 → finishing
G75 → External Grooving
G76 → External threading.

Threading Calculation:- for V-thread

Height of the thread

$h = P \times 0.61343$

$= 1.5 \times 0.61343$

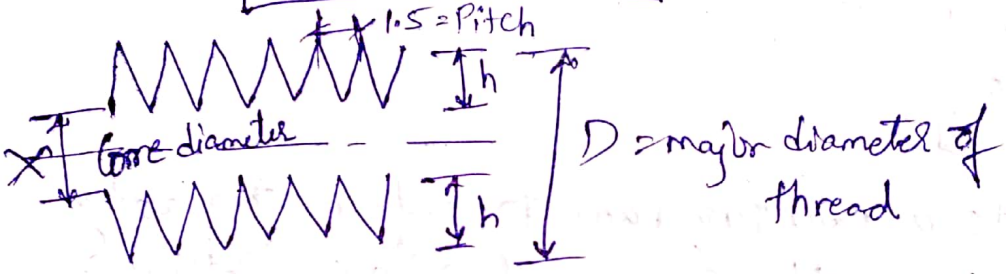
$= 0.920\ mm$

$\boxed{h = 920\mu = P}$

Core diameter = major diameter $- 2h$.

$= 10 - 2 \times 0.920$

$\boxed{X = 8.16mm}$




```
%
O1289
G21 G98          mm; mm/min
G28 U0 W0
M06 T0101 → Ext turn 0.8mm
M03 S1200
G00 X22 Z2
G71 U0.5 R1
G71 P10 Q20 U0.5 W0.5 F40
N10 G00 X8 Z0          (1)
    G01 X10 Z-2        (2)
    G01 X10 Z-25       (3)
    G01 X22 Z-35       (4)
N20 G00 X22 Z2
```

1(8, 0)
2(10, -2)
3(10, -25)
4(22, -35)

```
G97 P10 Q20
G28 U0 W0
M06 T0202      Ext Grooove (3mm)
M03 S800
G00 X10 Z-25   initial position of grooving tool
G115 R1
G75 X7 Z-24 P250 Q500 F40
G28 U0W0
M06 T0303  (Ext threading tool L.H thread angle 60°)
M03 S1000
G00 X10 Z0     initial position of threading tool.
G76 P041560 Q100 R0.2
G76 X8.16 Z-21 P920 Q200 F1.5
G28 U0 W0
M05
M30.
```

P041560
- 60 → Angle of tool tip (60°)
- 15 → Chamfering amount (15)
- 04 → No of idel passes (4)

Q100 → minm thread depth (0.1mm)

R0.2 → finishing allowance (0.2mm)

X8.16 → Coredia / minordia of thread

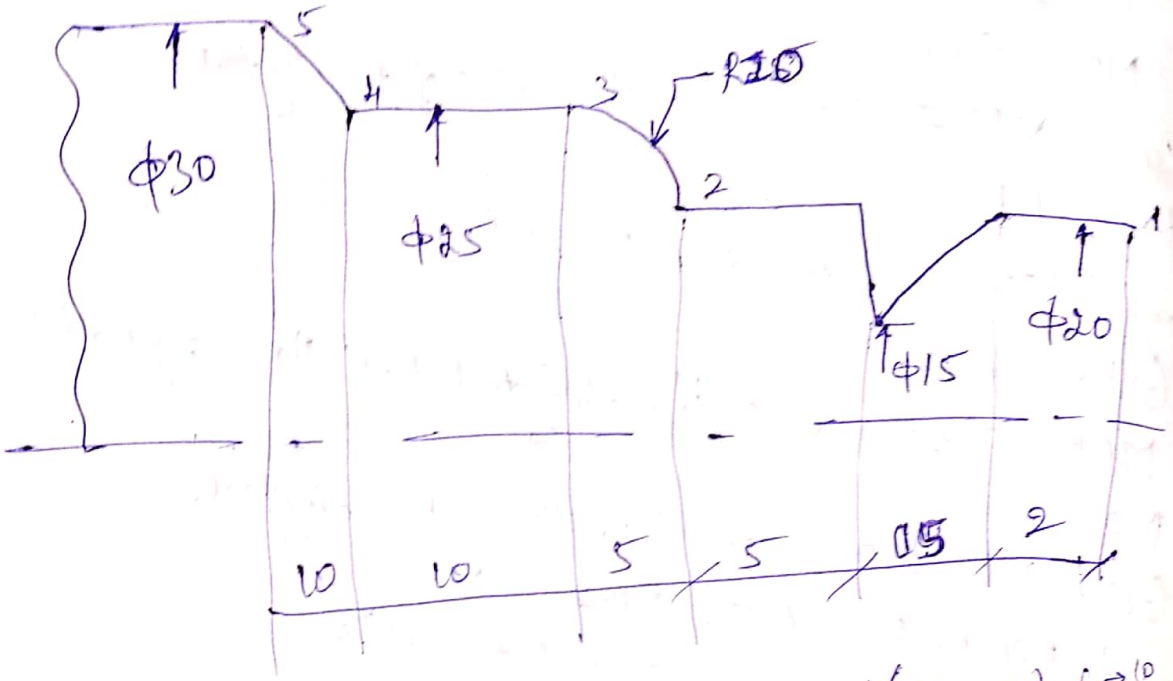
Z-21 → Length of thread

P920 → height of thread in 'µ'

Q200 → depth of cut in 1st Cut (0.2 mm)

F1.5 → Lead of thread on M/C tool (1.5mm).

6) Write a manual part program to Box turning / taper turning operation for the given profile



[BILLET X30 Z49
0 1 2 4 5
G21 G98
G28 U0 W0
M06 T0101
M03 S1200
G00 X30 Z2
G71 U0.5 R1
G71 P10 Q20 U0.5 W0.5 F40
N10 G00 X20 Z0 - (1)
G01 X20 Z-17 (2)
G03 X25 Z-22 R20 (3)
G01 X25 Z-32 (4)
X30 Z-42 (5)
N20 G00 X30 Z2
G70 P10 Q20.

1 (20, 0) L→10
2 (20, -17)
3 (25, -22)
4 (25, -32)
5 (30, -42)

L→5
1 (20, 0)
2 (20, -12)
3 (25, -17)
4 (25, -27)
5 (30, -37)

```
G28 U0 W0
G100 X20 Z-2        initial position for taper turning tool
G100 X20 Z-07 R0 F10
G90 X19 R0.5
    X18 R1
    X17 R1.5
    X16 R2
    X15 R2.5
G100 X30 Z2
G28 U0 W0
M05
M30
```

Z-07 → Length of taper from origin.

R → taper amount

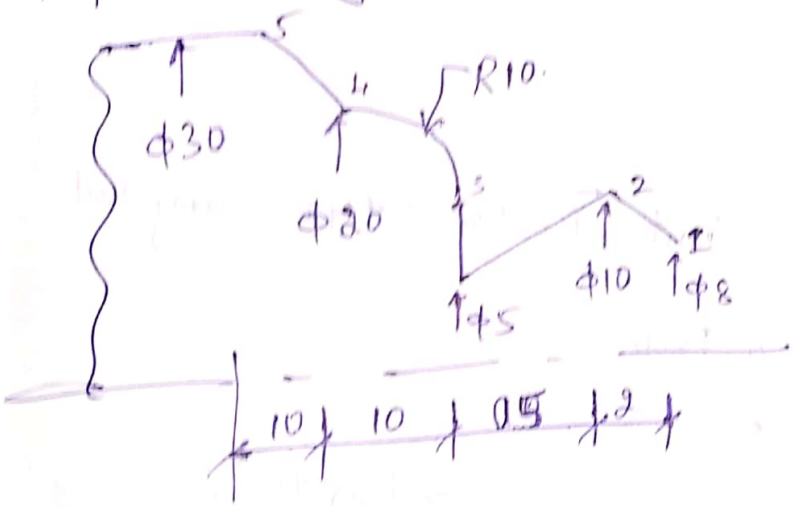
$$\tan\alpha = R = \frac{D-d}{2L}$$

D → 20
d → 15
L → 05

$$= \frac{20-15}{10}$$

$$\boxed{R = 0.5}$$

7) write a part program for Multiple turning operation for the component shown in figure by using taper turning cycle.



L → 05
5 (8, -0)
2 (10, -2)
3 (10, -7)
4 (20, -17)
5 (30, -27)

L → 10
1 (8, 0)
2 (10, -2)
3 (10, -12)
4 (20, -22)
5 (30, -32).

Soln:- taper amount (R)

D → 10
d → 5
L → 5

$$\tan\alpha = R = \frac{D-d}{2XL}$$

$$= \frac{10-5}{10}$$

$$\boxed{R = 0.5}$$

[BILLET X30 Z35

O1289

G21 G98

G28 U0 W0

M06 T0101

M03 S1500

G00 X30 Z2

G71 U0.5 R1

G71 P10 Q20 U0.5 W0.5 F40

N10 G00 X8 Z0 (1)

G01 X10 Z-2 (2)

X10 Z-07 (3)

G03 X20 Z-27 R10 (4)

G01 X30 Z-27 (5)

N20 G00 X30 Z2

G70 P10 Q20

G28 U0 W0

G00 X10 Z-2 initial position of taper turning tool

G90 X10 Z-07 R0 F40

X9 R0.5

X8 R1.0

X7 R1.5

X6 R2

X5 R2.5

G00 X30 Z2

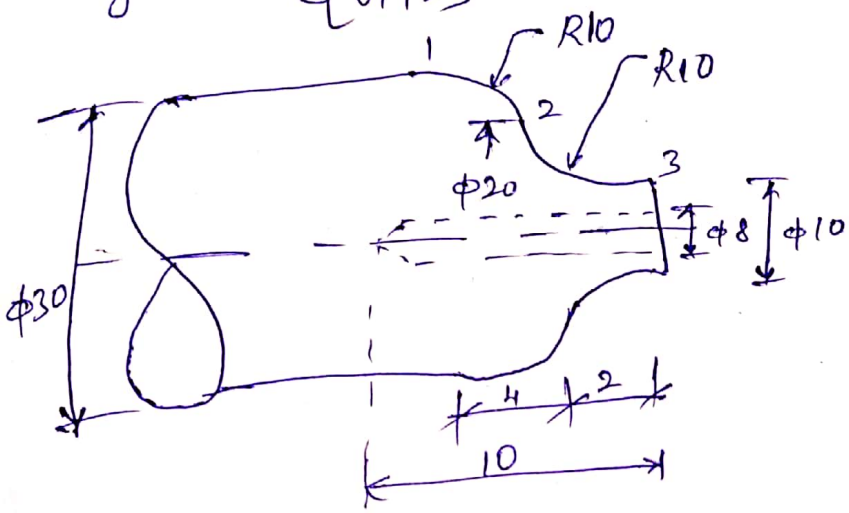
G28 U0 W0

M05

M30]

G90 → Box turning cycle/
Taper turning cy

8) write a part program for multiple turning operation for the component shown in figure using Facing and Drilling cycle. [G74] [G72]




```
[BILLET X30 Z20
  O1287
  G21 G98
  G28 U0 W0
  M06 T0101
  M03 S2000
  G00 X30 Z2
G72 W0.5 R1
G72 P10 Q20 U0.5 W0.5 F40
N10 G00 X30 Z-6      (1)
    G00 X20 Z-2 R10  (2)
    G03 X10 Z0 R10   (3)
N20 G00 X30 Z2
  G70 P10 Q20
  G28 U0 W0
  M06 T0202 → Tcentre drill φ8mm
  M03 S1000
  G00 X0 Z2
G74 R1
G74 X0 Z-10 Q500 R1 F15
    G28 U0W0 M05 M30]
```

1(30, -6)
2(20, -2)
3(10, 0).

G74 R1 → Return amount
Z-10 → depth of hole
Q500 → depth of cut in Z direction (0.5mm)
R1 → Relief for tool @ the cutting bottom
F15 → feedrate (mm/min)

CIM - Lab Viva Questions

1.) **CIM:-** Computer Integrated Manufacturing

Automation:- It is defined as technology concerned with mechanical, electrical, electronics and computer based System in order to control the Production by means of computer integrated System.

2.) **NC:-** Numerical Control.

It involves with G-Codes, M-codes, Numerical, Symbols which describe the motion of the tool to perform operation on Machine tool. It is a form of Programm-able automation.

3) **CNC:-** Computer Numerical Control.

In this CAD drawing is constructed and movement of tool is defined so that the Computer will control the motion of the tool and generate Programme.

Components of CNC:-

1.) Computer / Monitor

2.) Machine Control Unit [MCU]

It transfer the part programme into actual Machining on Machine tool.

3) **Machine tool:-**

where actual Machining will occur

For taper turning / Box turning Cycle

[G90] G90 X10 Z-30 R0 F30

R → taper amount

$$\tan\alpha = \frac{D-d}{2L} = R$$

R = taper amount

4) ~~Automation~~:-

Types of Automation:-

1. Fixed automation
2. Programmable Automation
3. Flexible Automation.

Software using:-
See NC Mill 6.0v
See NC turn 6.0v

5) CIM:- It is an Manufacturing approach of using Computer to control the entire production process.

It integrate electrical, electronic, computer and mechanical system to exchange information to control the production system.

NC Machines	CNC Machines
• Run by Servomotors, encoders, ball screw	• Run by computer [PC's]
• make use of punched tapes.	• Make use of floppy disk, Stored Programs.
• It Cannot import CAD files	• It Can import CAD files
• It Cannot use feedback Signal	• It make use of feedback Signal
• In order to modify the Programme, the tape have to be changed.	• programme Can be easily modify.
• Highly Skilled operator are required	• Less Skilled operator are required

Preparatory Codes [G-Codes]

It refers to Control of motion of the tool and which describe some specific operation.

Ex:- G00 - rapid traverse of tool
G01 - Linear traverse of tool
G71 - Rough turning cycle
G70 - finishing cycle

7) Miscellaneous Codes [M-Codes]

It refers to machine control rather than Control of motion of tool

Ex:- M03 - Spindle rotate in C.W.
M05 - Spindle off
M06 - Tool change

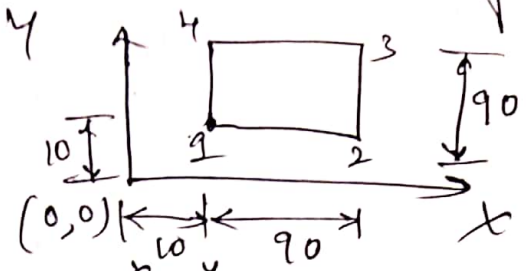
8) G-Codes

G00 - rapid traverse of tool
G01 - Linear —11—
G02 - Circular traverse (C.W)
G03 - —11— (C.C.W)
G04 - Dwell [Tool resides for a certain period to get flat surface during drilling operation]
G20 - Inches unit
G21 - Metri unit
G28 - Reference position of tool
G98 G99 - Feed (mm/min)
G83 - Peck drilling
G80 - Peck drilling Cancel
G70 - finishing cycle
G71 - Rough turning cycle
G72 - Facing cycle
G73 - High Peck drilling
G7H - Centre drill
G75 - External Grooving
G76 - External threading

X, Y, Z : Primary co-ordination System.
UW : Secondary Coordination System

Absolute Method of Programming :-

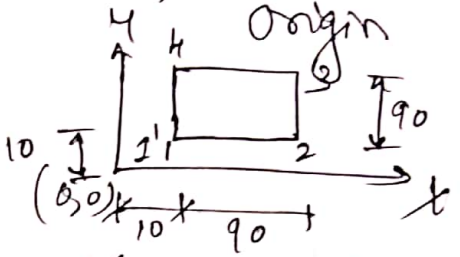
9) G90 :- Co-ordinates are obtained from the fixed origin.



1 (10, 10) 3 (100, 100)
2 (100, 10) 4 (10, 100)

Incremental method of Programming

G91 :- Co-ordinates are Obtained taking Previous location as Origin



1 (10, 10) 2 (90, 00)
3 (0, 90) 4 (-90, 0)
1' (0, -90)

10) Basic NC Motions :-

1.) Linear Interpolation
G00, G01

2) Circular Interpolation
G02 and G03

3) Point to Point [PTP] Motion
G83.

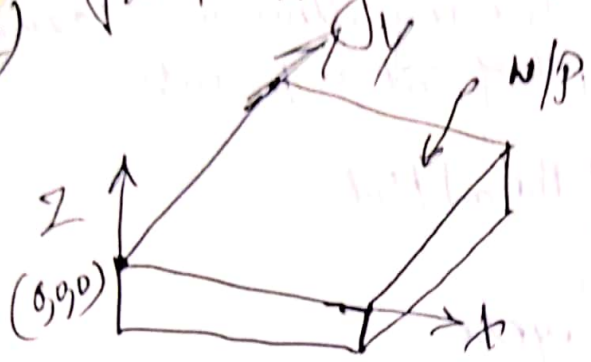
11) Miscellaneous Codes :-

M00 - optional program Stop
M30 - Programme end / End of programme
M05 - Spindle off
M03 - Spindle rotate in C.W
M04 - — 11 — C.C.W
M06 - tool change.
M08 - Coolant ON
M09 - Coolant OFF
M98 - Sub programm Call
M99 - End of Sub Program Call

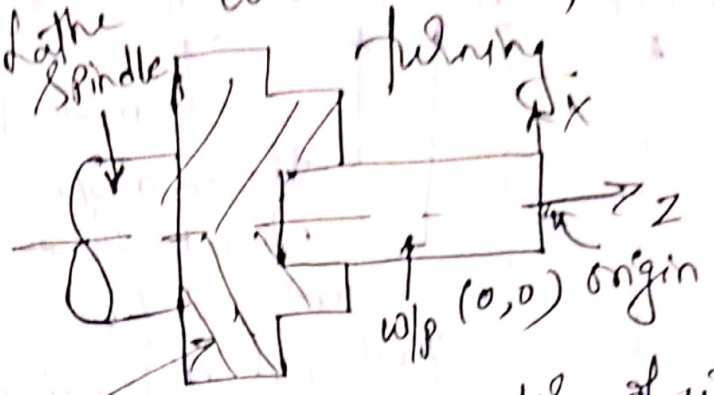
S - Spindle Speed (rpm)
F - Feed. (mm/min)
T - tool no
Z - thickness of w/p
depth
X, Y - Co-ordination System

In case of turning :-
X - diameter of w/p
Z - length of w/p

12) Coordination System for Milling



Coordination System for Turning



X - Diameter of W/P
Z - length of W/P

13) G71 - Rough turning.

G171 U0.5 R1
- R1 → Return of tool
- U0.5 → @ what rate material has to be removed

N10 - Profile Starting Block no
N20 - Profile ending Block no

G71 P10 Q20 U0.5 W0.5 F40
- P10 → Profile Starting no.
- Q20 → Profile ending no
- W0.5 → amount of stock maintained on w/p for finishing cycle
- F40 → feed rate (mm/min)

14) G75 - External grooving cycle

G175 R1
- R1 → Relief for tool (Sec)

G75 X5 Z-21 P250 Q500 F40
- X5 → Required diameter of groove
- Z-21 → Dimension from the origin (to achieve width of tool)
- P250 → movement of Grooving tool in x-direction (u)
- Q500 → movement of Grooving tool in z-direction (u)

15) G70 P10 Q20
- G70 → finishing cycle
- P10 Q20 → Profile Starting and Ending number.

16) G76 P04 15 60 Q100 R0.2

R0.2 → finishing allowance (0.2mm)
Q100 → min depth of cut in 1st pass
60 → thread angle
15 → chamfer length of thread / tool
04 → min no. of passes
G76 → External threading cycle

G76 X7.16 Z-21 P920 Q200 F1.5

F1.5 → lead of thread on m/c tool
Q200 → depth of cut (0.2mm)
P920 → height of the thread (0.920mm)
Z-21 → Length of the thread from origin
X7.16 → Core dia @ minor diameter of thread.

17) G83 X10 Y10 Z-12 Q4 R1 K1 F40

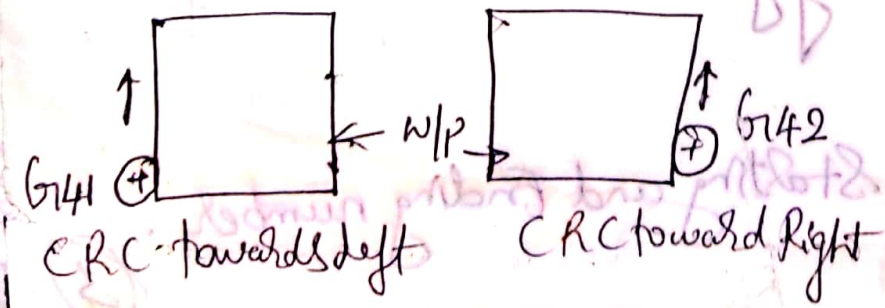
F40 → feed rate (mm/min)
K1 → no of repetation
R1 → Retract
Q4 → depth of peck increment (4mm)
Z-12 → depth of hole (mm)
X10 Y10 → Location / position of hole on w/p
G83 → Peck drilling

G80 - Peck drilling cancel

CRC :- Cutter Radius Compensation

used to obtain exact Profile Dimension on w/p.

G140 - CRC Cancel



Types of error :-

Syntax error :-
- It refer to improper defination of Code in the Programme
Ex :- If G03 line R is not defined.
G01 line F is not defined

Omission error :-
omitting decimal values

Setup error :-
Wrong entry of Dimension value in Blank.