COMPUTER INTEGRATED MANUFACTURING
LAB MANUAL
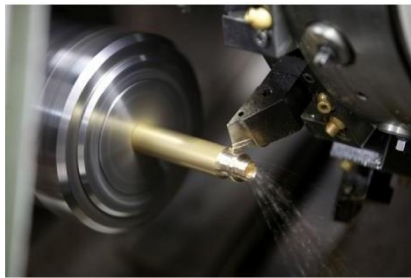
DEPARTMENT OF MECHANICAL ENGINEERING

NAME OF THE STUDENT : …………………………………….....................

REG.NO : …………………………………………………..

YEAR AND SECTION : …………………………………………………..

INDEX

DEFINITION OF CNC

"A system in which the actions are controlled by direct insertion of numerical data at some point .The system must automatically interpret at least some portion of this data"

WHY IT IS CALLED AS CNC?

Since the information required to actuate and control slides of the machine are coded numerically, this technology came to be known as Numerical Control.

WHAT IS CNC?

CNC is acronym for <u>Computer Numerical Control.</u>

A dedicated computer is used to perform all the basic NC functions. The complete part programme to produce a component is input and stored in the computer memory and the information for each operation is fed to the machine tools. The program can be stored and used in future

AXIS IN CNC MACHINES

THE BASIS OF AXIS IDENTIFICATION IS THE 3-DIMENSIONAL CARTESIAN CO-ORDINATE SYSTEM AND THREE AXIS OF MOVEMENT ARE IDENTIFIED AS X,YAND Z AXIS

Z AXIS.

Z- Axis The Z Axis of motion is always the axis of the main spindle of the machine.It doses not matters whether the spindle carries the work piece or the cutting tool . On vertical machining centers Z axis is vertical and on horizontal machining center and turning centers Z axis is horizontal

Positive Z Movement is away from the spindle

X-Axis The axis is always horizontal and is always parallel to the work holding surface. Positive X Axis movement is identified as being to the right, when looking from the spindle towards its supporting column.

Y- Axis The axis is always at right angle to both X-Axis and Z-Axis

Rotary axis .The rotary motion about the X,Y and Z-Axis are identified by A,B,C respectively .Clockwise is designated as +VE. .Positive rotation is identified looking in x ,y and z direction respectively

AXIS IN CNCLATHE

Max. turning diameter	Axis X
Max. turning length	Axis Z
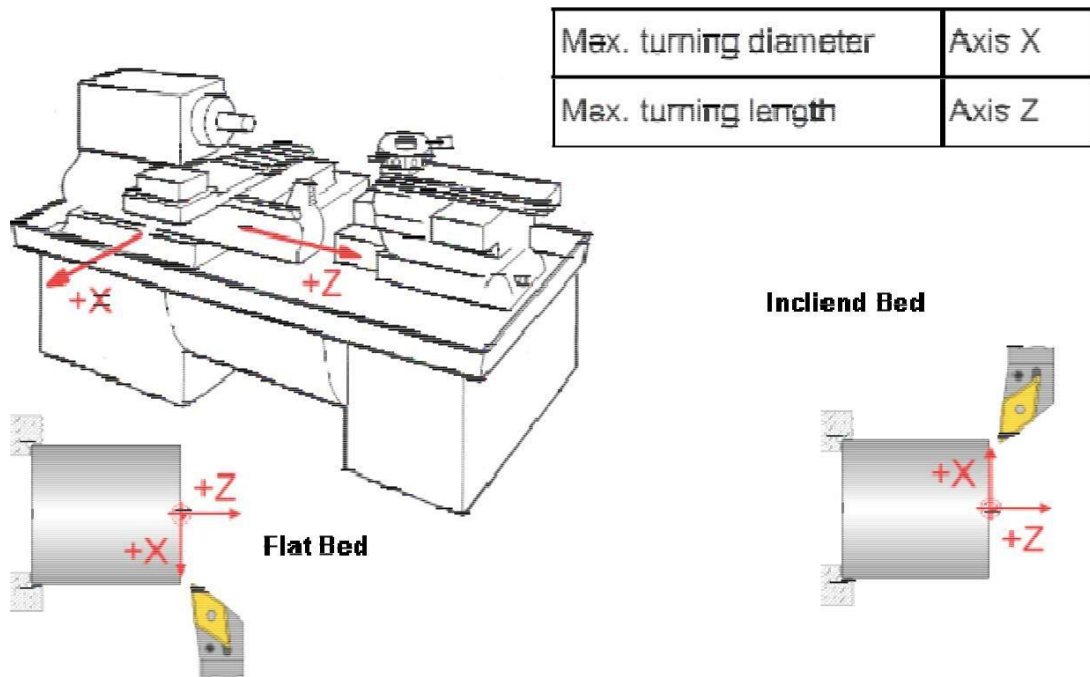
AXIS IN MILLING MACHINE

A milling machine has 3 axes of movement identified by X, Y & Z axes
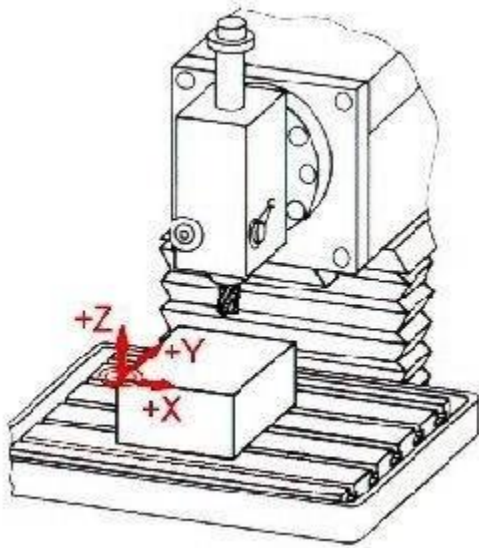
Max. workpiece length	Axis X
Max. workpiece width	Axis Y
Max. workpiece height	Axise Z

The maximum work piece dimensions correspond to the possible traversing path of the tool in the particular axis.

AXIS IN MILLING MACHINE

3

ZERO POINTS REFERENCE POINT
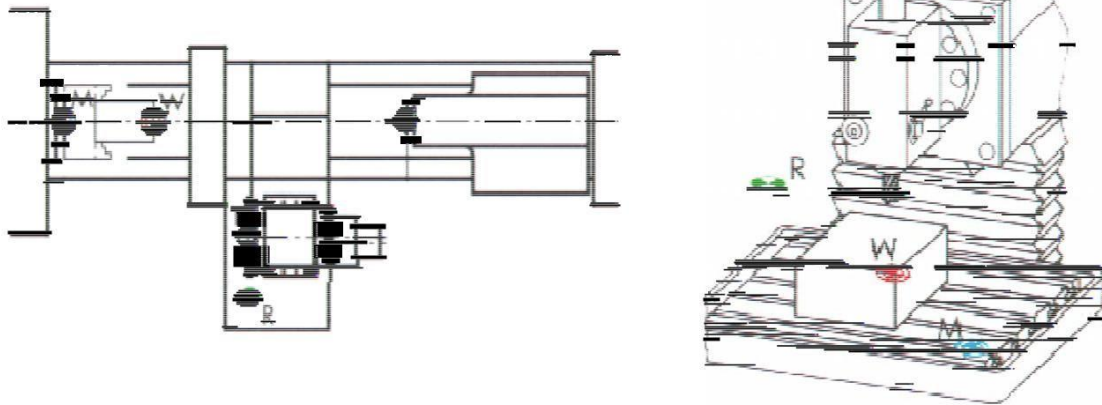
Machine zero M	The manufacturer defines the machine zero M and this cannot be changed. It is located at the origin of the machine coordinate system.

Workpiece zero W	The workpiece zero W (also known as program zero) is at the origin of the workpiece coordinate system. It can be freely selected, and should be located at the point where most of the dimensions originate in the drawing.

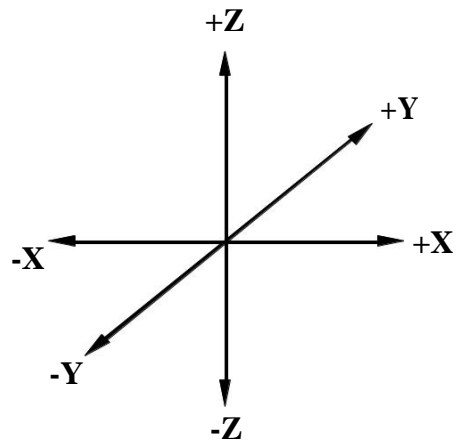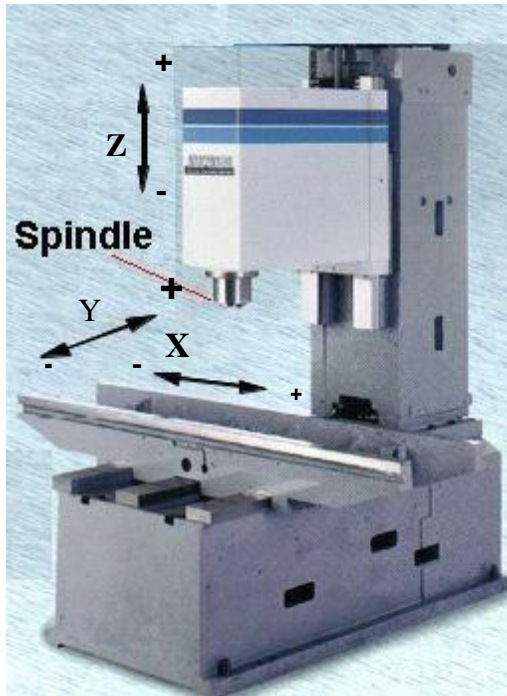| **Reference point R**	The reference point R is approached to set the measuring system to zero, as the machine zero point can generally not be approached. The control starts to count in its incremental position measuring system.

The reference point R serves for calibrating and for controlling of measuring systems of the slides and tool traverses. The position of the reference point is accurately predetermined in every traverse axis by the trip dogs and limit switches. Therefore, the reference point coordinates always have the same, precisely known numerical value in relation to the machine zero point.

After initiating the control system, the reference point must always be approached from all axes to calibrate the traverse measuring system.

DIMENSION SYSTEM

Dimensional information in a work piece drawing can be stated in two ways Absolute Dimension System and Incremental Dimension System.

Absolute Dimension System

Data in absolute dimension system always refer to a fixed reference point. This point has the function of a coordinate zero point. The dimension lines run parallel to the coordinate axes and always start at the reference point. Absolute dimensions are also called as 'Reference dimensions'

Incremental Dimension System

When using Incremental Dimension system, every measurement refers to a previously dimensioned position Incremental dimensions are distance between adjacent points. These distances are converted into incremental coordinates by accepting the last dimension point as the coordinate origin for the new point. This may be compared to a small coordinate system, i.e., shifted consequently form point to point (P1..P2..through P9). Incremental dimensions are also frequently called 'Relative dimensions' or 'Chain dimensions'.

DIMENSION SYSTEM

ABSOLUTE DIMENSIONING			INCREMENIAL DIMENSIONING		
POINTS	X	Z	POINTS	U	W
P1	10	0	P1	10	0
P2	10	-10	P2	0	-10
P3	20	-10	P3	10	0
P4	20	-25	P4	0	-15
P5	30	-25	P5	10	0
P6	30	-35	P6	0	-10
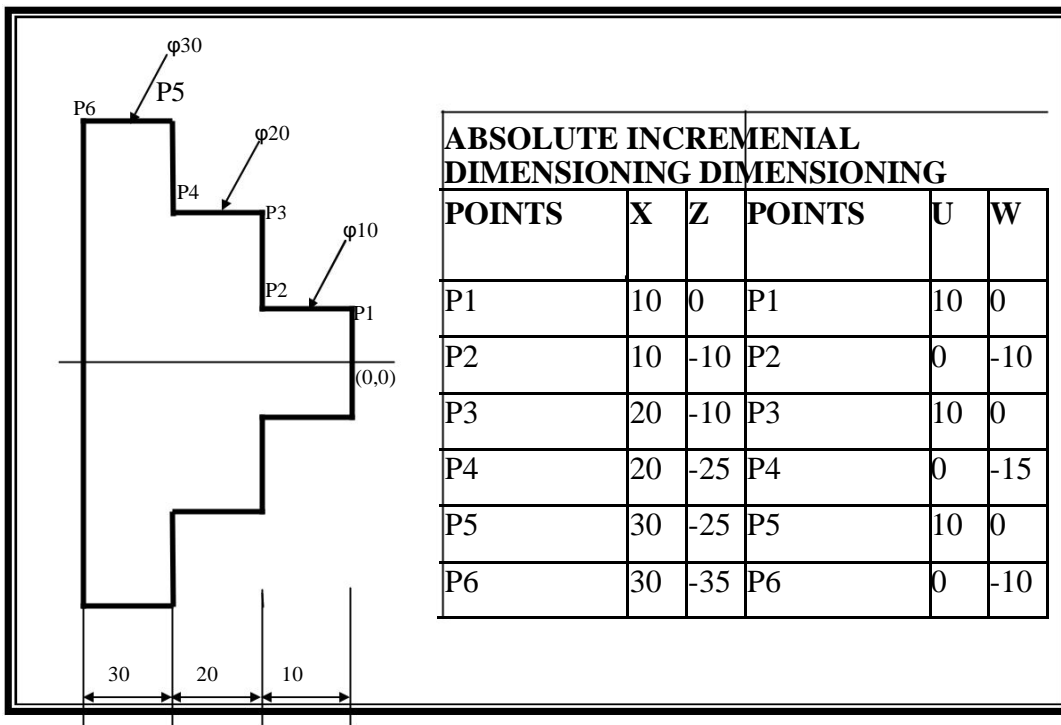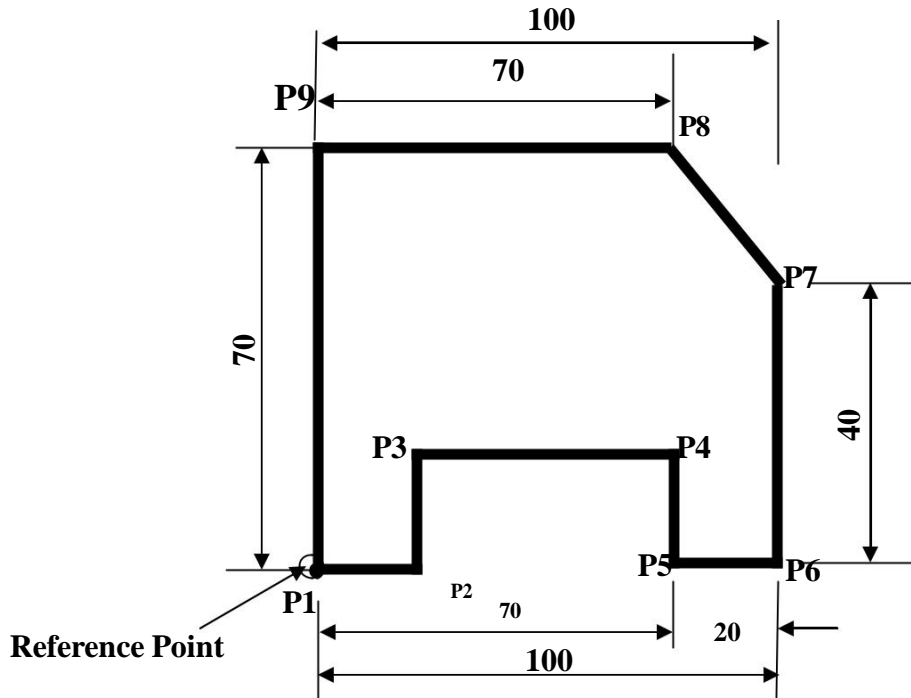
POINTS	X VALUE	Z VALUE
P1	0	0
P2	20	0
P3	20	20
P4	70	20
P5	70	0
P6	100	0
P7	100	40
P8	70	70
P9	0	70

ABSOLUTE SYSYEM (G90)

POINTS	X VALUE	Z VALUE
P1	0	0
P2	20	0
P3	0	20
P4	50	0
P5	0	-20
P6	30	0
P7	0	40
P8	-30	30
P9	-70	0

INCREMENTAL SYSTEM (G91)

STUDY OF ISO CODES

CNC MILLING

AIM: To study various CNC Milling codes and addresses

PREPARATORY FUNCTIONS (G CODES)

A number following address G determines the meaning of the command for the concerned block. G codes are divided into the following two types

Type	Meaning
One-shot G code	The G code is effective only in the block in which it is specified
Modal G code	The G code is effective until another G code of the same group is specified

LIST OF G CODES

G00: Point to point positioning (Rapid traverse)

G01: Linear interpolation

G02: Circular interpolation clockwise

G03: Circular interpolation counter clockwise

G04: Dwell, Exact stop

G17: X Y Plane selection

G18: Z X plane selection

G19: Y Z Plane selection

G20: Input in inch

G21: Input in metric (mm)

G28: Return to reference point

G40: Cutter compensation cancel

G41: Cutter Compensation left

G42: Cutter Compensation right

G43: Tool length compensation positive direction

G44: Tool length compensation negative direction

G49: Tool length compensation cancel

G50: Scaling OFF

G51: Scaling ON

G54: Datum shift

G68: Rotation ON

G69: Rotation cancel

G73: High speed Peck drilling cycle

G74: L.H Tapping cycle

G76: Fine Boring

G80: Canned cycle cancel

G81: Continuous drilling cycle, Stop boring

G82: Continuous drilling cycle, Stop boring with dwell

G83: Peck drilling cycle

G84: R.H. Tapping cycle

G85: Boring cycle with feed retraction

G86: Boring cycle with rapid retraction

G87: Back boring cycle

G88: Boring cycle

G89: Boring cycle with dwell & feed retraction

G90: Absolute coordinates

G91: Incremental coordinates

G92: Set Datum

G94: Feed per minute

G95: Feed per revolution

G98: Return to initial point in a canned cycle

G99: Return to R point in a canned cycle

G170, G171: Circular pocketing

G172, G173: Rectangular pocketing

MISCELLANEOUS FUNCTIONS (M CODES)

When a three digit figure is specified following address M, a 3-digit BCD code signal and a strobe signal are transmitted. These signals are used for ON/OFF control of a machine function such as tool change, spindle rotation change, coolant ON/OFF. One M code can be specified in one block. Selection of M codes for functions varies with the machine tool builder.

LIST OF M CODES

M00: Program stop

M01: Optional (planned) stop

M02: End of program

M03: Spindle ON clockwise

M04: Spindle ON counter clockwise

M05: Spindle stop

M06: Tool change

M07: Coolant 2 ON

M08: Coolant 1 ON

M09: Coolant OFF

M10: Vice open

M11: Vice close

M12: Synchronization code

M13: Spindle clockwise and coolant ON

M14: Spindle counterclockwise and coolant ON

M19: Orientates Spindle

M20: ATC arm IN (Towards spindle)

M21: ATC arm out (Retracts from spindle)

M22: ATC arm down

M23: ATC arm up

M24: Activates ATC draw bar

M25: Releases draw bar

M30: Program stop & rewind

M32: Rotates ATC clockwise

M33: Rotates ATC counter clockwise

M38: Opens the door

M39: Closes the door

M62: Output 1 ON

M63: Output 2 ON

M64: Output 1 OFF

M65: Output 2 OFF

M66: Wait input 1 ON

M67: Wait input 2 ON

M70: X mirror ON

M71: Y mirror ON

M76: Wait input 1 OFF

M77: Wait input 2 OFF

M80: X mirror OFF

M81: Y mirror OFF

M98: Sub program Call

M99: Sub program Exit

COMMON MILLING G-CODES

(* Indicates default start-up codes)

*G00 Rapid linear move. [X, Y, Z] Moves the machine at the fastest rate possible to the X, Y, Z location specified

G01 Linear feed move [X, Y, Z, F] moves the machine at the specified feed rate (F) to the X, Y, Z location specified

G02 Circular interpolation; CW. [X, Y, Z, F, R (or I, J, K)] Moves the machine, in a clockwise circular path to the X, Y, Z, location specified with radius R, or with a center point defined relative to the start point in the X, Y, & Z axis by I, J, & K respectively.

G03 Circular interpolation; CCW. [X, Y, Z, F, R (or I, J, K)] Same as G02, but opposite direction of movement.

G28 Machine home. Causes the machine to return to its X0, Y0, and Z0 position at a
rapid rate.

*G40 Cutter compensation, CANCEL.   [X, Y, F] Cancels the G41 or G42 Cutter
compensation listed below. Causes a feed move to X and/or Y at feed rate F (or at
modal feed F, if not specified). The distance of the move must be greater than the
radius of the tool.

G41 Cutter compensation, LEFT.   [X, Y, D, F] Looking from the spindle toward the
part, G41 offsets the position of the tool left of the programmed tool path by the
value stored in the offsets register position called by the D word. Causes a feed
move from the current position to the compensated position specified by X, Y, at
feed F (or at modal feed F if not specified). The distance of this move must be
greater than the radius of the tool.

G42 Cutter compensation, RIGHT. Same as G41 above except that the tool is
compensated to the right of the programmed tool path.
NOTE: Cutter compensation may be accomplished at the machine, through the
tool path generated by the CAM program, or both.

G43 Tool Length compensation.   [Z,H] Compensates for the length of the tool during
subsequent machine motion. Causes the spindle (Z axis) to move to designated Z
location offset by the value stored in the offsets register position called by the H
word. This compensation will remain in effect until cancelled during a tool
change by the G28 command.

*G53 Machine coordinate system (MCS). Cancels G-54 thru G59

G54-G59 Work coordinate offsets. Allows the user to establish one or more alternative
coordinate systems to correspond to individual work coordinate systems (WCS)

G80 Canned cycle cancel. This command cancels any active "canned cycle" commands
below.

CANNED CYCLE

A Canned cycle is a combination of machine movements that perform machining operations like drilling, milling, boring and tapping. For example, the drilling cycle consists of the following movements of the tool.

Fast approach to work
piece. Drill at feed rate
Rapid return to initial position.

These movements can be combined to form a cycle and give a code. When this code is invoked, the machine performs all these operations. The use of canned cycle reduces programming effort. This also saves the length of the program, thus saving the space required to store the program.

G81 Drill cycle. [X, Y, Z, R, F] Will drill a hole to Z depth at a location defined by X, Y, at feed rate F. R is the rapid plane, a Z-axis dimension "above the part" that denotes the point where the machine switches from rapid rate to feed rate. Will continue to drill the same hole profile at subsequent X, Y locations until cancelled by G80.

G82 Spot drill / Counter bore cycle. [X, Y, Z, R, F, P] Similar to above, but adds the P word to establish a dwell, or pause at the end of the Z-axis stroke.

G83 Deep Hole Peck drill cycle. [X, Y, Z, R, Q, (or I, J, K), R, F, P] Drills the hole in a series of steps, or pecks, at either a constant peck depth Q, or at a decreasing peck depth where I is the initial peck depth, J is the amount of decrease per peck and K is the minimum peck depth. Other parameters are as for other drill cycles.

G84 Rigid tap cycle (right hand threads). [X, Y, Z, J, R, F] Precisely coordinates the rotation of the spindle with the feed rate of the Z-axis for use with right hand taps. The J word is a multiplying factor for the retract speed. Feed rate is computed by multiplying rpm by thread pitch and is carried to 3 decimal places.

G85 Basic boring cycle. [X, Y, Z, R, F] Similar to the G81 drill cycle, except that the tool is advanced into and out of the work at the feed rate speed, instead of a feed in and rapid out of the G81.

*G90 Absolute Programming Mode. All X, Y, Z coordinates determined from a single origin.

G91 Incremental Programming Mode Next X, Y, Z, coordinates are determined from current location

G98 Canned cycle initial point return. Traverses from hole to hole at the initial point Z height, typically 1.0 inch.

G99 Canned cycle rapid (R) plane return. Traverses from hole to at the rapid plane Z height, typically 0.1 inch.

COMMON MILL M-CODES

M00 Program stops. Stops the machine, requiring the operator to restart the program to continue.

M01 Optional stop. Stops the machine as above only when the optional stop button has been pressed prior to this command in the program.

M03 Spindle start, CW. [S] Starts the spindle in a clockwise direction, at the RPM specified by the S word accompanying the code.

M04 Spindle start, CCW. [S] Similar to above, but rotation is reversed.

M05 Spindle stops.

M06 Tool change. [T] Executes a tool change to the tool specified with the T word. Automatically stops the spindle, returns the Z-axis to machine zero, and turns off the coolant. On machines with automatic tool changers, the tools will be swapped. On the Tool room machines, an alarm light signals the operator to switch tools manually.

M08 Coolant on.

M09 Coolant off.

M19 Spindle orientation.

M30 Program stop and rewind. to program start

M97 Local subroutine call. [N] Causes the program to skip to a subprogram contained inside the current program at line number N.

M98 Subprogram call [P, L] The program will call another program number, specified by the P word, and execute it L times.

Sub Program

When a program contains certain fixed sequences or frequently repeated patterns, these sequences or patterns may be entered into memory as a subprogram to simplify programming. A subprogram can tell another subprogram. When the main program call a subprogram, it is regarded as a one-loop subprogram call.

M99 Subprogram returns. Contained at the end of the subprogram (or subroutine) will return the control to the main program.

Note: only one M Code can be used per line and will be executed after all other operations specified in the line.

RESULT:

Thus various CNC Milling codes and addresses were studied.

MILLING EXERCISES

Write a manual part program for machining the component shown in **DWG.NO.1**.
Profile depth=2mm

POINT TO POINT MOTIONS
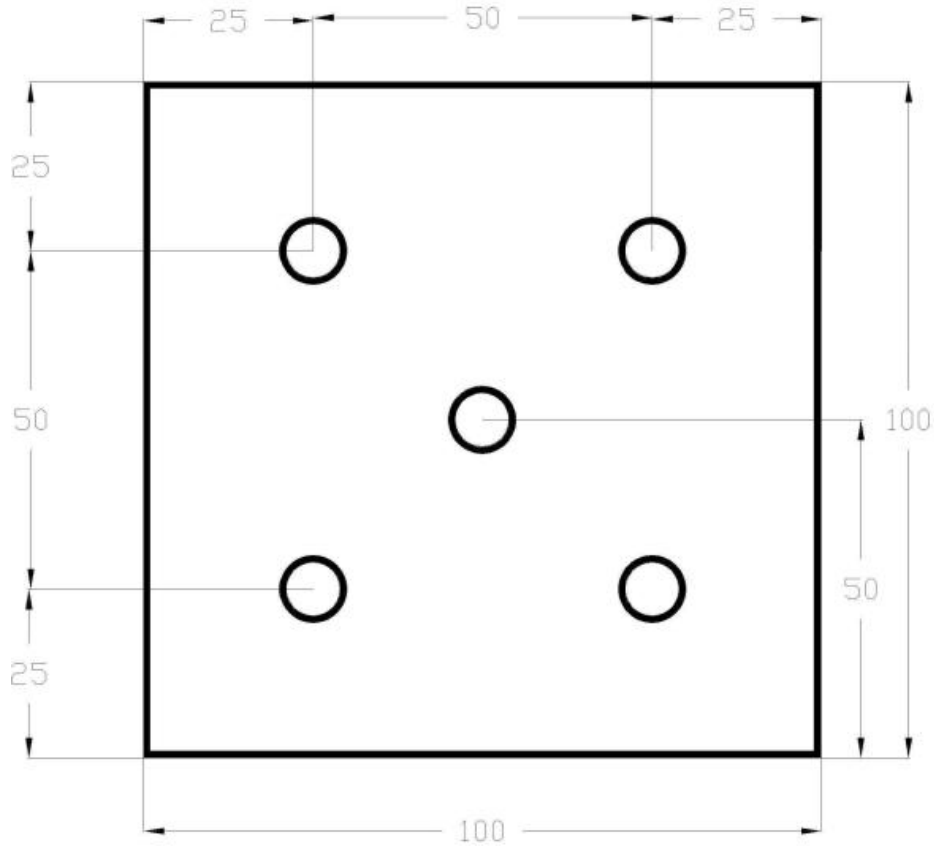
ALL DIMENSIONS ARE IN "mm"

DWG.NO.1

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 100x100x10				**MATERIAL:** Aluminium			
PROGRAM NO:1001				**DWG.NO:**1			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool length offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Contouring	Slot cutter	5	1	1	1500	30-50

16

EX.NO: M01

DATE:

AIM:

      To write a manual part program for machining the component shown in the DWG. NO.1

MATERIAL REQUIRED:

Material   : Aluminium
Size   : Length 100mm, Width 100mm, Thickness 10mm

PROGRAM:

```
O0001
G21 G94
G91 G28 X0 Y0 Z0
M06 T01
M03 S1500
G90 G00 X25 Y25 Z5
G01 Z-2 F50
G00 Z5
G00 X75 Y25
G01 Z-2 F50
G00 Z5
G00 X75 Y75
G01 Z-2 F50
G00 Z5
G00 X25 Y75
G01 Z-2 F50
G00 Z5
G00 X50 Y50
G01 Z-2
G00 Z5
G91 G28 X0 Y0 Z0
M05
M30
```

RESULT:

      Thus the manual part program was written to the given dimensions and executed in CNC milling.

Write a manual part program for machining the component shown in **DWG.NO.2**.
Profile depth=1mm

LINEAR MOTIONS
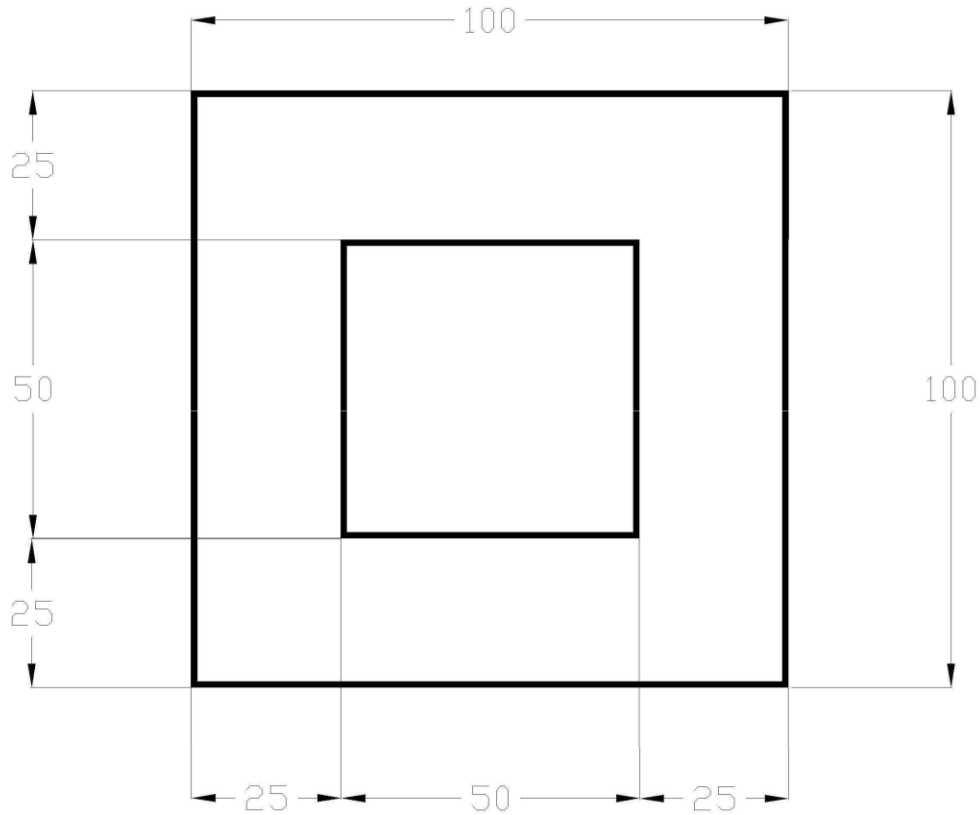
ALL DIMENSIONS ARE IN "mm"

DWG.NO.2

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 100x100x10				**MATERIAL:** Aluminium			
PROGRAM NO:1002				**DWG.NO:**2			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool length offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Contouring	Slot cutter	5	1	1	1500	30-50

LINEAR MOTIONS

EX.NO:M02

DATE:

AIM:
　　To write a manual part program for machining the component shown in the DWG. NO.2

MATERIAL REQUIRED:

Material　　　　: Aluminium
Size　　　　　　: Length 100mm, Width 100mm, Thickness 10mm

PROGRAM:

O0002
G21 G94
G91 G28 X0 Y0 Z0
M06 T01
M03 S1500
G90 G00 X25 Y25 Z5
G01 Z-1 F50
G01 X75
G01 Y75
G01 X25
G01 Y25
G00 Z5
G91 G28 X0 Y0 Z0
M05
M30

RESULT:

　　　　Thus the manual part program was written to the given dimensions and executed in CNC milling.

Write a manual part program for machining the component shown in **DWG.NO.3**.
Profile depth=1mm

CIRCULAR INTERPOLATION

ALL DIMENSIONS ARE IN "mm"

DWG.NO.3

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 100x100x10				**MATERIAL:** Aluminium			
PROGRAM NO:1003				**DWG.NO:3**			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool length offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Contouring	Slot cutter	5	1	1	1500	30-50

20

CIRCULAR INTERPOLATION

EX.NO: M03

DATE:

AIM:

      To write a manual part program for machining the component shown in the DWG. NO.3

MATERIAL REQUIRED:

Material   : Aluminium
Size   : Length 100mm, Width 100mm, Thickness 10mm

PROGRAM:

```
O0003
G21 G94
G91 G28 X0 Y0 Z0
M06 T01
M03 S1500
G90 G00 X35 Y0 Z5
G01 Z-1 F30
G02 X-35 Y0 R35
G02 X35 Y0 R35
G00 Z5
G91 G28 X0 Y0 Z0
M05
M30
```

RESULT:

      Thus the manual part program was written to the given dimensions and executed in CNC milling.

Write a manual part program for **Contouring operation** for the component shown in **DWG.NO.4**. Profile depth=1mm

CONTOURING
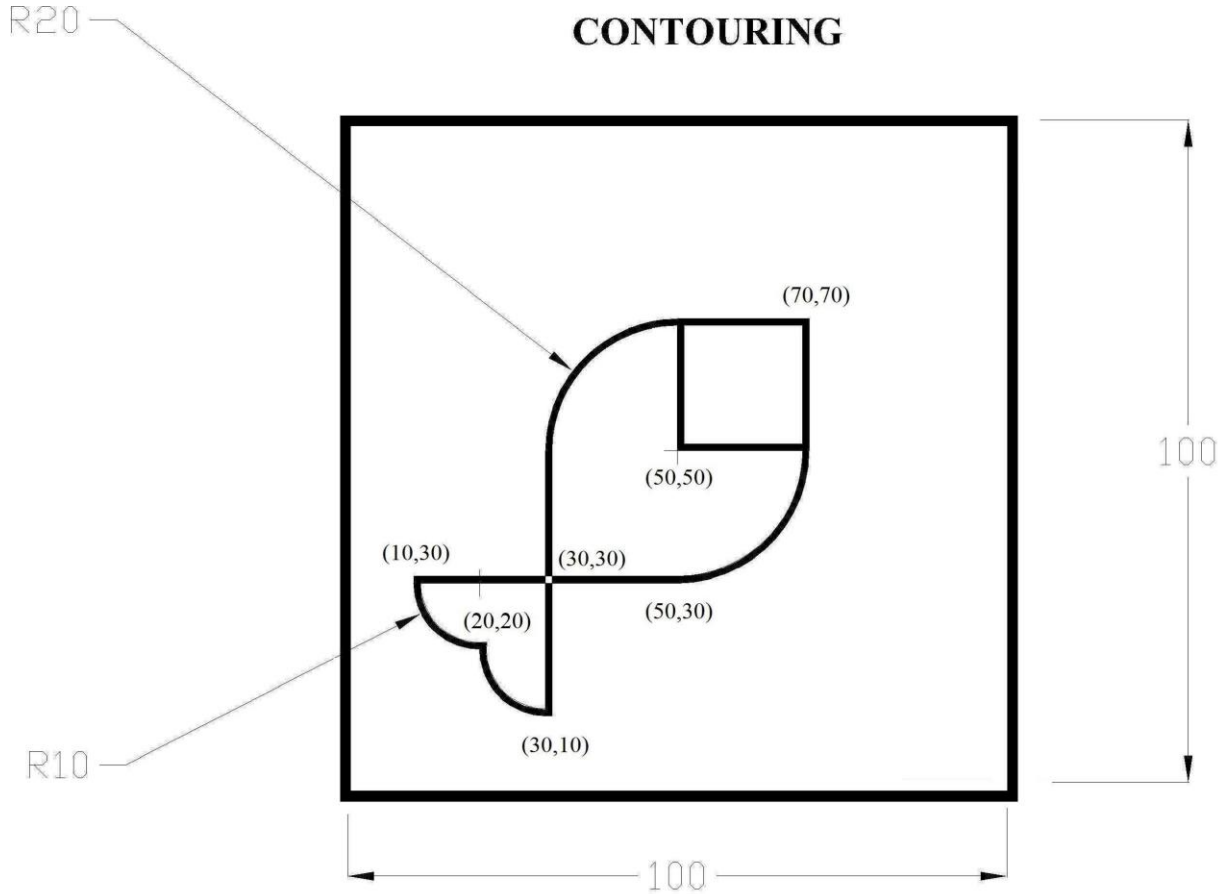
ALL DIMENSIONS ARE IN "mm"

DWG.NO.4

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 100x100x10				**MATERIAL:** Aluminium			
PROGRAM NO:1004				**DWG.NO:4**			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool length offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Contouring	Slot cutter	5	1	1	1500	30-50

CONTOUR MOTIONS

EX.NO: M04

DATE:

AIM:

      To write a manual part program for contouring operation for the component shown in the DWG. NO.4

MATERIAL REQUIRED:

Material   : Aluminium
Size   : Length 100mm, Width 100mm, Thickness 10mm
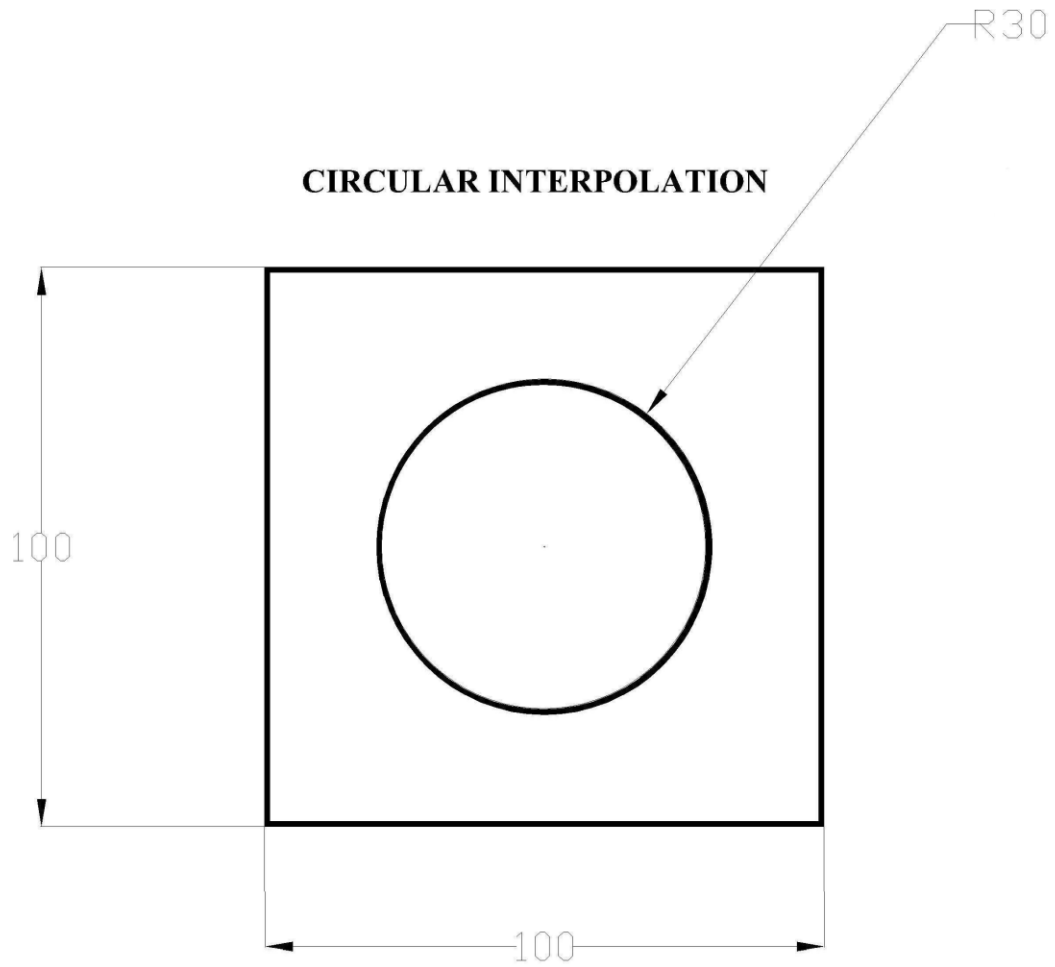
PROGRAM:

```
O0004
G21 G94
G91 G28 X0 Y0 Z0
M06 T01
M03 S1500
G90 G00 X50 Y70 Z5
G01 Z-1 F30
G01 X70
G01 Y50
G01 X50
G01 Y70
G03 X30 Y50 R20
G01 X30  Y10
G02 X20 Y20 R10
G02 X10 Y30 R10
G01 X50
G03 X70 Y50 R20
G00 Z5
G91 G28 X0 Y0 Z0
M05
M30
```

RESULT:

      Thus the manual part program was written to the given dimensions and executed in CNC milling.

Write a manual part program for **Rectangular Pocketing** operation for the component shown in **DWG.NO.5**. Profile depth=3mm

RECTANGULAR POCKETING

ALL DIMENSIONS ARE IN"mm"

DWG.NO.5

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 100x100x10				**MATERIAL:** Aluminium			
PROGRAM NO:1005				**DWG.NO:**5			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool length offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Slotting & Pocketing	Slot cutter	5	1	1	1500	35-45

RECTANGULAR POCKETING
G172 I(*i1) **J**(*j1) **K**(*k1) **P**(*p1) **Q**(*q1) **R**(*r1) **X**(*x1) **Y**(*y1) **Z**(*z1)
G173 I(*i2) **K**(*k2) **P**(*p2) **T**(*t2) **S**(*s2) **R**(*r2) **F**(*f2) **B**(*b2) **J**(*j2) **Z**(*z2)

INPUT DATA FOR G172

I – Length of the pocket in X direction
J – Length of the pocket in Y direction
K – Corner radius (always zero)
P – when P = 0 , roughing & when p = 1 , Finishing .
Q -- Depth of cut for each pass .
    i.e. Pocket Z increment.
R-- Absolute Z from surface
X -- Pocket corner X
Y – Pocket corner Y
Z – Absolute Z base of pocket

INPUT DATA FOR G173

I – Pocket side finish allowance
K--Pocket base finish allowance
P—Cut width Percentage T—
Pocket Tool
S—Spindle speed for roughing

R—Roughing feed for z
F--Roughing feed XY
B—Finishing spindle speed J—
Finishing feed & Z—Safety Z

24

RECTANGULAR POCKETING

EX.NO: M 05

DATE:

AIM:

      To write a manual part program for Rectangular Pocketing operation for the component shown in the DWG. NO.5

MATERIAL REQUIRED:

Material   : Aluminium
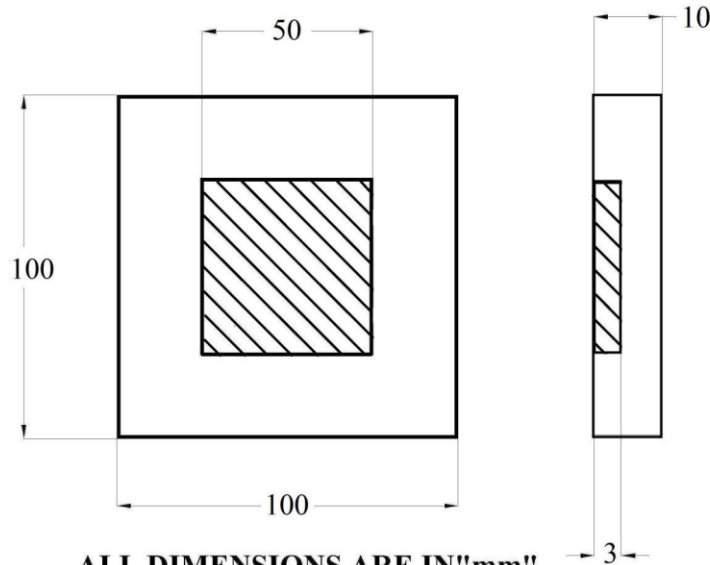Size   : Length 100mm, Width 100mm, Thickness 10mm

PROGRAM:

O0005
G21 G94
G91 G28 X0 Y0 Z0
M06 T01
M03 S1200
G90 G00 X0 Y0 Z5
G172 P0 Q1 R0.5 X-25 Y-25 Z-3 I50 J50 K0
G173 P80 T1 S1200 R30 I0.1 K0.1 F70 B2000 J40 Z5
G172 P1 Q1 R0.5 X-25 Y-25 Z-3 I50 J50 K0
G173 P80 T1 S1200 R30 I0.1 K0.1 F70 B2000 J40 Z5
G00 Z5
G91 G28 X0 Y0 Z0
M05
M30

.

RESULT:

      Thus the manual part program was written to the given dimensions and executed in CNC milling

Write the manual part program for **Contouring operation through subroutine** for the component shown in **DWG.NO.6.** Profile depth: 3mm. Depth of cut for each pass=1mm.

SUBPROGRAM
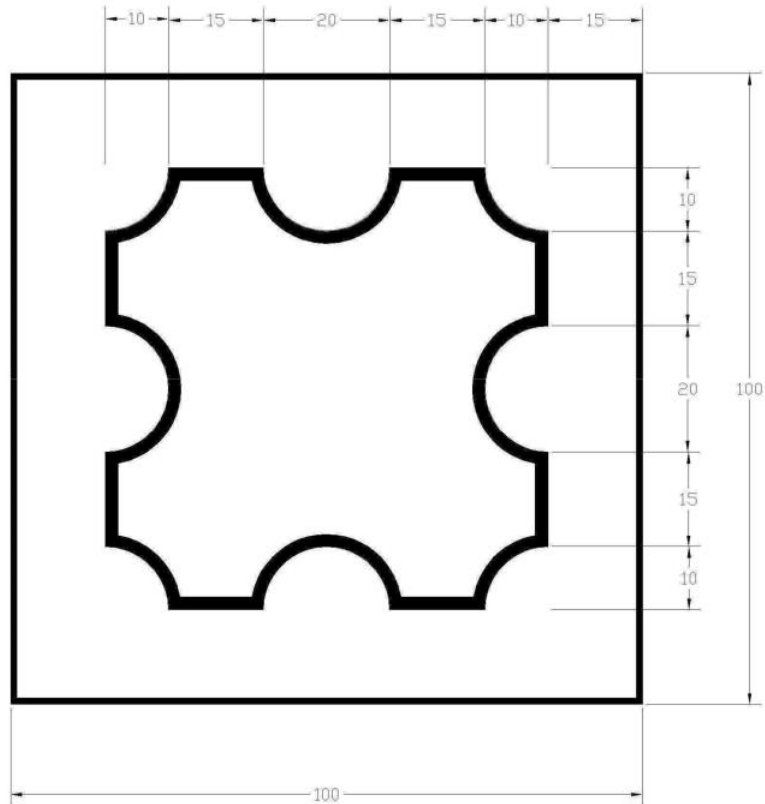
ALL DIMENSIONS ARE IN "mm"

DWG.NO.6

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 100x100x10				MATERIAL: Aluminium			
PROGRAM NO:1006				DWG.NO:6			
Sl.No	Operation	Tool type	Tool dia., mm	Tool Station No.	Tool length offset No.	Spindle speed, rpm	Feed, mm/min
1	Contouring	Slot cutter	5	1	1	1500	30-50

26

EX.NO: M06

DATE:

AIM:

To write the manual part program for contouring operation through subroutine for the component shown in DWG.NO.6

MATERIAL REQUIRED:

Material : Aluminium
Size : Length 100mm, Width 100mm, Thickness 10mm

PROGRAM:
```
O0006
G21 G94
G91 G28 X0 Y0 Z0
M06 T01
M03 S1500
G90 G00 X25 Y15 Z5
G01 Z0 F50
M98 P003 3242
G90 G00 Z5
G91 G28 X0 Y0 Z0
M05
M30
:3242
G91 G01 Z-1
G90 G01 X40 Y15
G02 X60 Y15 R10 F30
G01 X75 Y15
G02 X85 Y25 R10 F30
G01 X 85 Y40
G02 X85 Y60 R10
G01  X85 Y75
G02 X75 Y85 R10
G01  X60 Y85
G02 X40 Y85 R10
G01  X25 Y85
G02 X15 Y75 R10
G01  X15 Y60
G02 X15 Y40 R10
G01  X15 Y25
G02 X25 Y15 R10
M99
```

RESULT:

Thus the manual part program was written to the given dimensions and executed in CNC milling.

Write a manual part program for **Mirroring operation** for the component shown in **DWG.NO.7**. Profile depth=1mm

MIRRORING
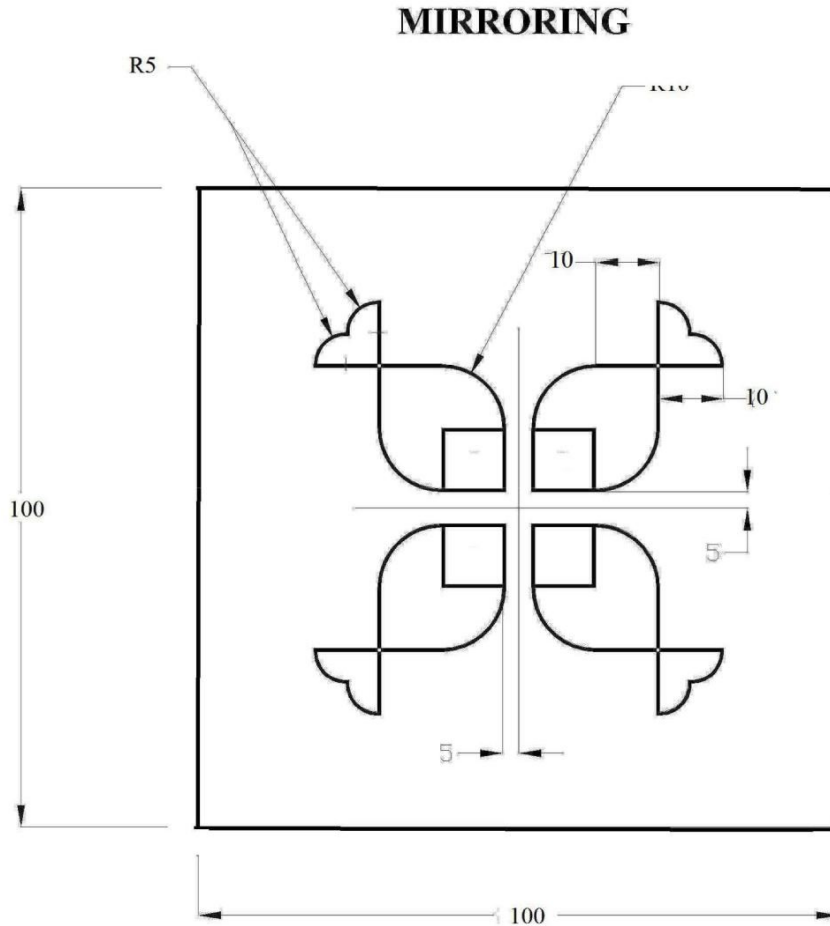
ALL DIMENSIONS ARE IN "mm"

DWG.NO.7

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 100x100x10				**MATERIAL:** Aluminium			
PROGRAM NO:1007				**DWG.NO:**7			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool length offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Contouring	Slot cutter	5	1	1	1500	30-50

EX.NO: M07

DATE:

AIM:
> To write the manual part program for Mirroring Operation for the component shown in DWG.NO.7

MATERIAL REQUIRED:

Material   : Aluminium
Size   : Length 100mm, Width 100mm, Thickness 10mm

PROGRAM:
```
O0007
G21 G94
G91 G28 X0 Y0 Z0
M06 T01
M03 S1200
G90 G00 X0 Y0 Z5
M98 P001 3535
M70
M98 P001 3535
M71
M98 P001 3535
M80
M98 P001 3535
M81
G91 G28 X0 Y0 Z0
M05
M30
:3535
G00 X5 Y15
G01 Z-1 F30
G01 X15
G01 Y5
G01 X5
G01 Y15
G02 X15 Y25 R10
G01 X35
G03 X30 Y30 R5
G03 X25 Y35 R5
G01 Y15
G02 X15 Y5 R10
G00 Z5
M99
```

RESULT:
> Thus the manual part program was written to the given dimensions and executed in CNC milling

Write the manual part program for **Circular Pocketing operation** for the component shown in **DWG.NO.8.** Profile depth: 3mm. Depth of cut for each pass=1mm.

CIRCULAR POCKETING
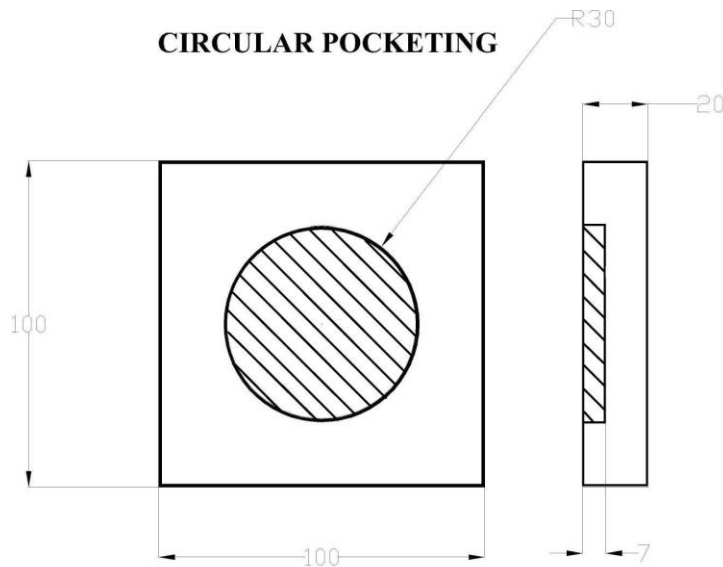
ALL DIMENSIONS ARE IN "mm"

DWG.NO.8

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 100x100x10				**MATERIAL:** Aluminium			
PROGRAM NO:1008				**DWG.NO:8**			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool length offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Slotting & Pocketing	Slot cutter	5	1	1	1500	30-50

CIRCULAR POCKETING:
G170 R(*r1) P(*p1) Q(*q1) X(*x1) Y(*y1) Z(*z1) I(*i1) J(*j1) K(*k1)
G171 P(*p2) S(*s2) R(*r2) F(*f2) B(*b2) J(*j2)

INPUT DATA FOR G170
R -- Position of tool to start cycle (i.e) 0(surface of job)
P -- When P = 0 the cycle is a roughing cycle,
        P = 1 the cycle is a finishing cycle.
Q -- Peck Increment for each cut
X – Pocket centre in X axis
Y – Pocket centre in Y axis
Z – Pocket base from job surface

I -- Side Finish Allowance
J – Base Finish Allowance
K – radius of pocket

INPUT DATA FOR G171
P -- Cuttter Movement percentage for next step.
S – Roughing Spindle speed

R – Roughing feed in Z direction for each cut
F – Roughing feed in X Y directions , mm/ min.
B – Finishing spindle speed, rpm.
J – Finishing feed, mm/min.

CIRCULAR POCKETING

EX.NO:M08

DATE:

AIM:
　　　　To write a manual part program for Circular Pocketing operation for the component shown in the DWG. NO.8
　　　.
MATERIAL REQUIRED:

Material　　　　: Aluminium
Size　　　　　　: Length 100mm, Width 100mm, Thickness 10mm

PROGRAM:

O0008
G21 G94
G91 G28 X0 Y0 Z0
M06 T01
M03 S1500
G90 G00 X0 Y0 Z5
G170 P0 Q1 R0 X0 Y0 Z-3 I0.1 J0.1 K35
G171 P80 S1500 R30 F70 B2000 J30
G170 P1 Q1 R0 X0 Y0 Z-3 I0.1 J0.1 K35
G171 P80 S1500 R30 F70 B2000 J30
G00 Z10
G91 G28 X0 Y0 Z0
M05
M30

RESULT:
　　　　　Thus the manual part program was written to the given dimensions and executed in CNC milling

Write a manual part program for machining the component **through Canned Cycle** for the component shown in the **DWG. NO.9**

DRILLING IN P.C.D (CANNED CYCLE)

(8) φ 6 C'BORE
φ10 3 mm DP
EQ.SPACED @ 70 PCD

R6

100

100

6

10

ALL DIMENSIONS ARE IN"mm"

DWG.NO.9

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 100x100x10				**MATERIAL:** Aluminium			
PROGRAM NO:1009				**DWG.NO:09**			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool length offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Drilling	Slot drill	6	1	1	1500	35
2	Drilling	Slot drill	10	2	2	1500	35

32

EX.NO:M09

DATE:

AIM:
>To write a manual part program for machining the component through canned cycle for the component shown in the DWG. NO.9

MATERIAL REQUIRED:

Material   : Aluminium
Size   : Length 100mm, Width 100mm, Thickness 10mm

PROGRAM:
O0009
G21 G94
G91 G28 X0 Y0 Z0
M06 T11
M03 S1500
G90 G00 X0 Y35 Z5
G98 G81 X0 Y35 Z-6 R2 F35
M98 P001 2525
G91 G28 X0 Y0 Z0
M05
M06 T01
M03 S1500
G90 G00 X0 Y35 Z5
G98 G81 X0 Y35 Z-3 R2 F35
M98 P001 2525
G91 G28 X0 Y0 Z0
M05
M30
:2525
X24.74 Y24.74
X35 Y0
X24.74 Y-24.74
X0 Y-35
X-24.74 Y-24.74
X-35 Y0
X-24.74 Y24.74
G00 Z10
G80
M99

RESULT:
>Thus the manual part program was written to the given dimensions and executed in CNC milling

Write the manual part program for **Mirroring operation** for the component shown in **DWG.NO.10.** Profile depth: 1mm.

MIRRORING-II
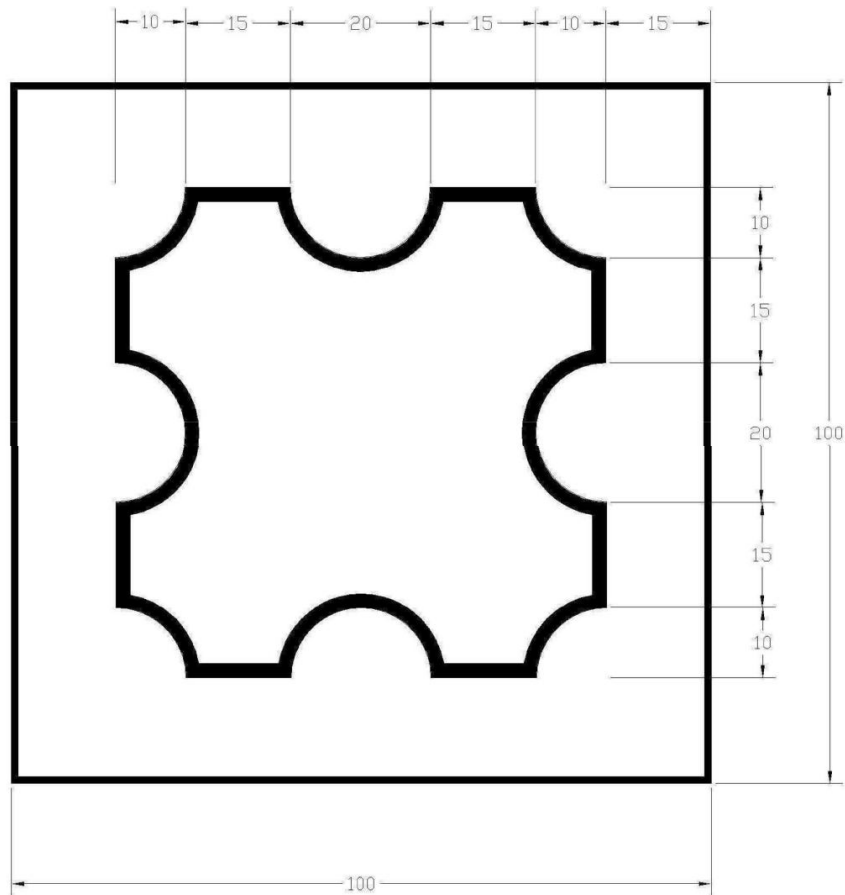
ALL DIMENSIONS ARE IN "mm"

DWG.NO.10

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 100x100x10				**MATERIAL:** Aluminium			
PROGRAM NO:1010				**DWG.NO:**10			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool length offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Contouring	Slot cutter	5	1	1	1500	30-50

34

MIRRORING-II

EX.NO: M10

DATE:

AIM:

      To write the manual part program for Mirroring Operation for the component shown in DWG.NO.10
    .

MATERIAL REQUIRED:

Material   : Aluminium
Size   : Length 100mm, Width 100mm, Thickness 10mm
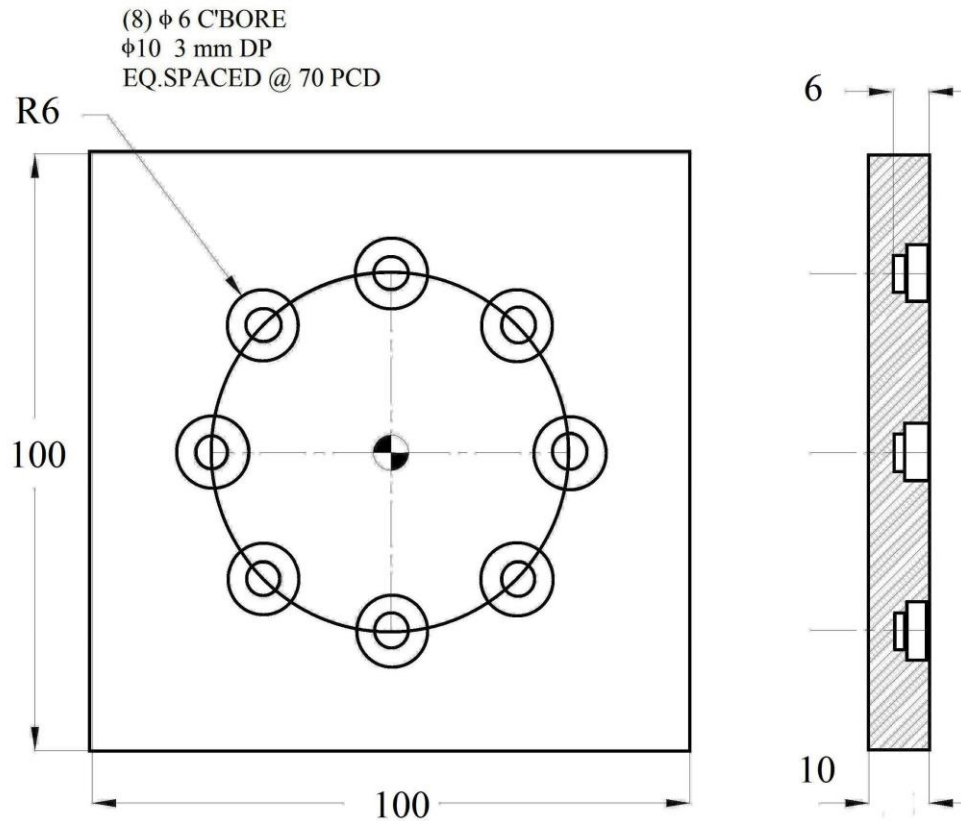
PROGRAM:

```
O0010
G21 G94
G91 G28 X0 Y0 Z0
M06 T11
M03 S1200
G90 G00 X0 Y0 Z5
M98 P001 3535
M70
M98 P001 3535
M71
M98 P001 3535
M80
M98 P001 3535
M81
G91 G28 X0 Y0 Z0
M05
M30
:3535
G00 X25 Y0 Z5
G01 Z-1 F30
G02 X35 Y10 R10
G01 X35  Y25
G02 X25 Y35 R10
G01 X10  Y35
G02 X0 Y25 R10
G00 Z5
M99
```

RESULT:

      Thus the manual part program was written to the given dimensions and executed in CNC milling.

Write the manual part program for **Circular Pocketing operation** for the component shown in **DWG.NO.11.** Profile depth: 3mm. Depth of cut for each pass=1mm.

CIRCULAR POCKETING -II
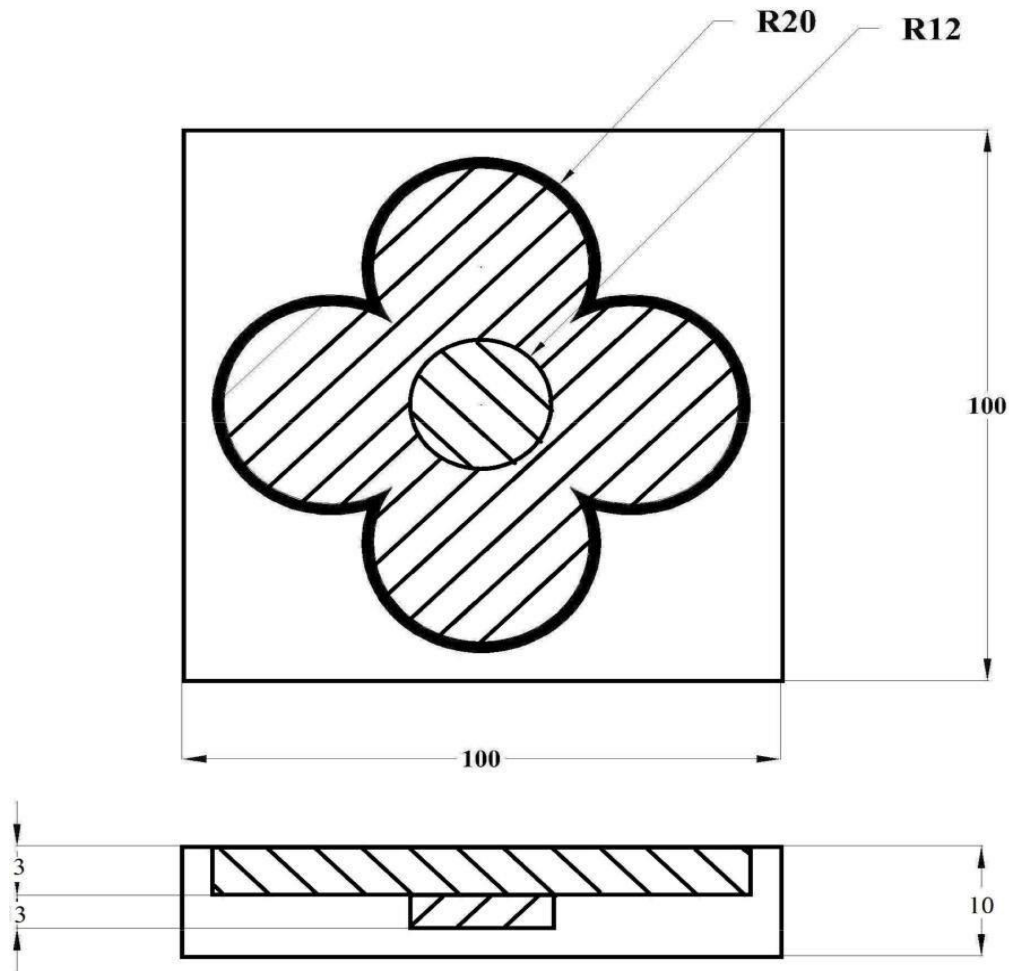
ALL DIMENSIONS ARE IN "mm"

DWG.NO.11

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 100x100x10				**MATERIAL:** Aluminium			
PROGRAM NO:1011				**DWG.NO:11**			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool length offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Contouring	Slot cutter	5	1	1	1500	30-50

36

CIRCULAR POCKETING-II

EX.NO:M11

DATE:

AIM:

   To write a manual part program for Circular Pocketing operation for the component shown in the DWG. NO.11

MATERIAL REQUIRED:

Material   : Aluminium
Size   : Length 100mm, Width 100mm, Thickness 10mm

PROGRAM:

O0011
G21 G94
G91 G28 X0 Y0 Z0
M06 T01
M03 S1200
G90 G00 X0 Y0 Z5
G170 P0 Q1 R0.5 X0 Y20 Z-3 I0.1 J0.1 K20
G171 P80 S1500 R30 F70 B2000 J30
G170 P1 Q1 R0.5 X0 Y20 Z-3 I0.1 J0.1 K20
G171 P80 S1500 R30 F70 B2000 J30
G170 P0 Q1 R0.5 X-20 Y0 Z-3 I0.1 J0.1 K20
G171 P80 S1500 R30 F70 B2000 J30
G170 P1 Q1 R0.5 X-20 Y0 Z-3 I0.1 J0.1 K20
G171 P80 S1500 R30 F70 B2000 J30
G170 P0 Q1 R0.5 X0 Y-20 Z-3 I0.1 J0.1 K20
G171 P80 S1500 R30 F70 B2000 J30
G170 P1 Q1 R0.5 X0 Y-20 Z-3 I0.1 J0.1 K20
G171 P80 S1500 R30 F70 B2000 J30
G170 P0 Q1 R0.5 X20 Y0 Z-3 I0.1 J0.1 K20
G171 P80 S1500 R30 F70 B2000 J30
G170 P1 Q1 R0.5 X20 Y0 Z-3 I0.1 J0.1 K20
G171 P80 S1500 R30 F70 B2000 J30
G170 P0 Q1 R-3 X0 Y0 Z-6 I0.1 J0.1 K12
G171 P80 S1500 R30 F70 B2000 J30
G170 P1 Q1 R-3 X0 Y0 Z-6 I0.1 J0.1 K12
G171 P80 S1500 R30 F70 B2000 J30
G00 Z5
G91 G28 X0 Y0 Z0
M05
M30

RESULT:

   Thus the manual part program was written to the given dimensions and executed in CNC milling.

STUDY OF ISO CODES

CNC TURNING

AIM: To study various CNC Turning codes and addresses

G Codes

G00: Point to point positioning (Rapid traverse)

G01: Linear interpolation

G02: Circular interpolation clockwise

G03: Circular interpolation counter clockwise

G04: Dwell, Exact stop

G17: X Y Plane selection

G18: Z X plane selection

G19: Y Z Plane selection

G20: Input in inch

G21: Input in metric (mm)

G28: Return to reference point

G32: Thread Cutting

G40: Cutter compensation cancel

G41: Cutter Compensation left

G42: Cutter Compensation right

G49: Tool length compensation cancel

G50: Work co-ordinate Change / Max Spindle speed setting

G70: Finishing cycle

G71: Stock removal in turning

G72: Stock removal in facing

G73: Pattern repeating

G74: Peck drilling in z axis

G75: Grooving in x axis

G76: Thread Cutting Cycle

G80: Canned cycle cancel

G90: Cutting cycle A

G92: Thread cutting cycle

G94: Cutting cycle B

G96: Constant surface speed control

G97: Constant surface speed control cancel

G98: Feed per minute

G99: Feed per revolution

M Codes

M00: Program stop

M01: Optional (planned) stop

M02: End of program

M03: Spindle forward clockwise

M04: Spindle forward counter clockwise

M05: Spindle stop

M06: Tool change

M08: Coolant ON

M09: Coolant OFF

M10: Chuck open

M11: Chuck close

M62: Output 1 ON

M63: Output 2 ON

M64: Output 1 OFF

M65: Output 2 OFF

M66: Wait input 1 ON

M67: Wait input 2 ON

M76: Wait input 1 OFF

M77: Wait input 2 OFF

M98: Sub program Call

M99: Sub program Exit

Common Lathe G-Codes

*G00 **Rapid linear move**. [X, Z, U, W] Moves the machine at the fastest rate possible to the X, Z location specified or incrementally U (X) W(Z) distance.

G01 **Linear feed move** [X, Z, U, W, F] Moves the machine at the specified feed rate (F) to the X, Z location specified or incrementally U (X) W(Z) distance.

G02 **Circular interpolation; CW.** [X, Z, U, W, F, R (or I, K)] Moves the machine, in a clockwise circular path to the X, Z, location (or incrementally by U,W) with radius R, or with a center point defined relative to the start point in the X,& Z axis by I, & K respectively.

G03 **Circular interpolation; CCW.** [X, Z, U, W, F, R (or I, K)] Same as G02, but opposite direction of movement.

G28 **Machine home**. Causes the machine to return to it's X0, Z0 position at a rapid rate.

*G40 **Tool nose compensation, *CANCEL.*** [X, Z, U, W, I, K, F] Cancels the G41 or G42 Cutter compensation listed below. Causes a feed move to X and/or Z at feed rate F (or at modal feed F, if not specified). The distance of the move must be greater than the radius of the tool.

G41 **Tool nose compensation, *LEFT***. [X, Z, U, W, D, F] Looking from the spindle toward the part, G41 offsets the position of the tool *left* of the programmed tool path by the value stored in the offsets register position called by the D word. Causes a feed move from the current position to the *compensated* position specified by X, Y, at feed F (or at modal feed F if not specified). The distance of this move must be greater than the radius of the tool.

G42 **Tool nose compensation, RIGHT**. Same as G41 above except that the tool is compensated to the *right* of the programmed tool path.

NOTE: Cutter compensation may be accomplished at the machine, through the tool path generated by the CAM program, or both.

G50 **Spindle speed clamp.** Specifies the maximum RPM the spindle can run during constant surface speed operation.

G70 **Finishing Cycle**

G71 **Multiple Turning Cycle.** It is used when the major direction of cut is along the Z axis. It causes the profile to be roughed out by turning.

G72 **Multiple Facing Cycle,** used when the major direction of cut is along the X axis. This cycle causes the profile to be roughed out by facing.

G73 **Pattern Repeating Cycle,** provides for roughing out of a form by repeating the desired tool path a set number of times, the tool path being incremented into the work piece until the full form is completed.

G75 **Grooving Cycle**

G92 **Single Thread cycle.** This is a Box type cycle producing a single pass of the threading tool.

G76 **Multiple Threading Cycle.** This is a box type cycle that is repeated a given number of times. After the first pass subsequent passes cut with one edge of the threading tool only to reduce the load at the tool tip.

G74 **End Face Peck Drilling.** This cycle is designed for deep hole drilling, the drill entering the work piece by a predetermined amount then by backing off by another set amount to provide breaking and allowing swarf to clear the drill flutes.

G90 **Single Turning Cycle**, used to produce either a parallel or tapered tool path. It performs four distinct moves with one line of information and it is equivalent to rapid to X position, feed to Z position, feed to start X position, rapid to start Z position

G94 **Facing Cycle** .This cycle is used for stock removal either parallel or at an angle to work piece face. It is equivalent of rapid to Z position, feed to X Position, and feed to start Z position and rapid to start X position. If R value is specified tapering will be performed.

Common Lathe M-Codes

M00 **Program stop**. Stops the machine, requiring the operator to restart the program to continue.

M01 **Optional stop**. Stops the machine as above only when the optional stop button has been pressed prior to this command in the program.

M03 **Spindle start**, **CW.** [S] Starts the spindle in a clockwise direction, at the RPM specified by the S word accompanying the code.

M04 **Spindle start, CCW**. [S] Similar to above, but rotation is reversed**.**

M05 **Spindle stop.**

M06 **Tool Change.** The M06 in conjunction with "T" word, is used to call up the required tool on an automatic indexing turret machine, and to activate its tool offsets.

M08 **Coolant on**.

M09 **Coolant off.**

M10 **Chuck Open**

M11 **Chuck Close**

M13 **Spindle Forward, Coolant ON**

M14 **Spindle Reverse, Coolant ON**

M19 **Spindle orientation.**

M25 **Quill** (Tailstock**) Extend**

M26 **Quill** (Tailstock**) Retract**

M30 **Program stops and rewind.** To program start

M38 **Door Open**

M97 **Local subroutine call.** [N] Causes the program to skip to a subprogram contained inside the current program at line number N.

M98 **Subprogram call** [P, L] the program will call another program number, specified by the P word, and execute it L times.

M99 **Subprogram return**. Contained at the end of the subprogram (or subroutine) will return the control to the main program.

Note: only one M Code can be used per line and will be executed after all other operations specified in the line.

RESULT:

Thus various CNC Turning codes and addresses were studied.

TURNING EXERCISES

Write a manual part program for simple **Turning and Facing** operation for component shown in **DWG.NO.T01**

TURNING AND FACING

DWG.NO.T01

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 22x60				**MATERIAL:** Aluminium			
PROGRAM NO:1001				**DWG.NO:**1			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Simple Facing	SDJCR121H11	DCMT11T304	1	1	1200	45

44

TURNING AND FACING

EX.NO: T01

DATE:

AIM:

      To write a manual part program for simple turning and facing operation for component shown in DWG.NO.T01

MATERIAL REQUIRED:

Material   : Aluminium
Size   : Length 70mm, Diameter 22mm

PROGRAM:

```
O0001
G21 G98
G28 U0 W0
M06 T0101
M03 S1200
G00 X23 Z3
G00 Z-0.5
G01 X-1 F45
G00 Z3
G00 X23
G00 Z-1
G01 X-1 F45
G00 Z3
G00 X23
G00 Z-1.5
G01 X-1 F45
G00 Z3
G00 X23
G00 Z-2
G01 X-1 F45
```

```
G00 Z3
G00 X21.5
G01 Z-30 F45
G00 X23
G00 Z3
G00 X21
G01 Z-30 F45
G00 X23
G00 Z3
G00 X20.5
G01 Z-30 F45
G00 X23
G00 Z3
G00 X20
G01 Z-30 F45
G00 X23
G28 U0 W0
M05
M30
```

RESULT:

      Thus the manual part program was written to the given dimensions and executed in CNC Lathe.

Write a manual part program for **Step Turning** operation with G90 cycle for component shown in **DWG.NO.T02**

STEP TURNING

ALL DIMENSIONS ARE IN "mm"
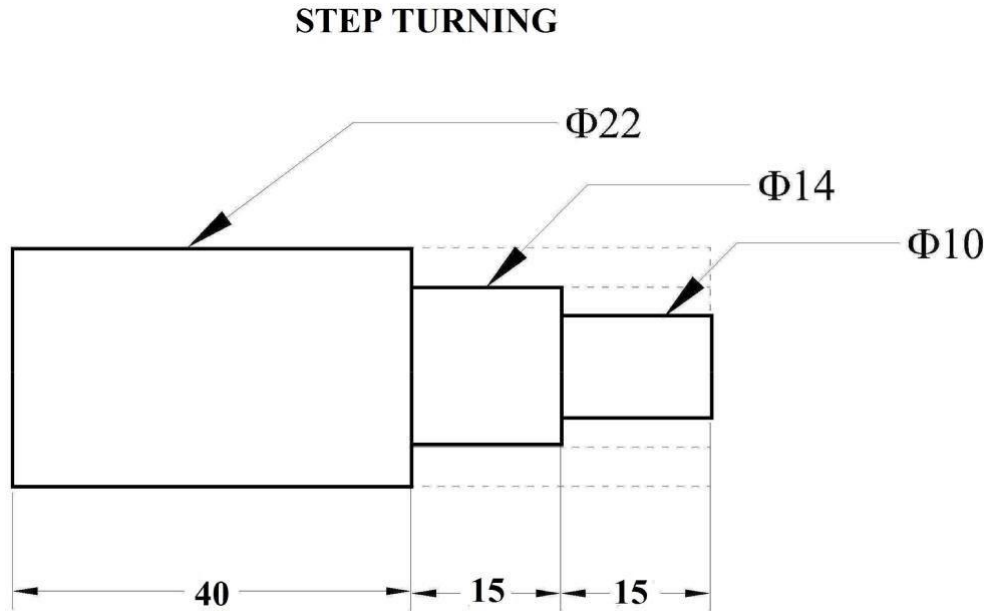
DWG.NO.T02

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 22x60				**MATERIAL:** Aluminium			
PROGRAM NO:1002				**DWG.NO:**2			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Step Turning	SDJCR121H11	DCMT11T304	1	1	1200	30

Cutting Cycle

G90 X(U) Z(W) F(*f)

X   Diameter to which the movement is being made.

U   The incremental distance from the current tool position to the required final Diameter Z The Z Axis Co-ordinate to which the movement is being made.

W   The incremental distance from the current tool position to the required Z axis Position fFeed rate

EX.NO:T02

DATE:

AIM:

      To write a manual part program for step turning operation with G90 cycle for component shown in DWG.NO.T02

MATERIAL REQUIRED:

Material   : Aluminium
Size   : Length 70mm, Diameter 22mm

PROGRAM:

O0001
G21 G98
G28 U0 W0
M06 T0101
M03 S1200
G00 X22 Z1
G90 X22 Z-30 F30
X21
X20
X19
X18
X17
X16
X15
X14
G00 X14 Z1
G90 X14 Z-15 F30
X13
X12
X11
X10
G28 U0 W0
M05
M30

RESULT:

      Thus the manual part program was written to the given dimensions and executed in CNC Lathe.

Write a manual part program for **Taper Turning** operation for component shown in **DWG.NO.T03**

TAPER TURNING
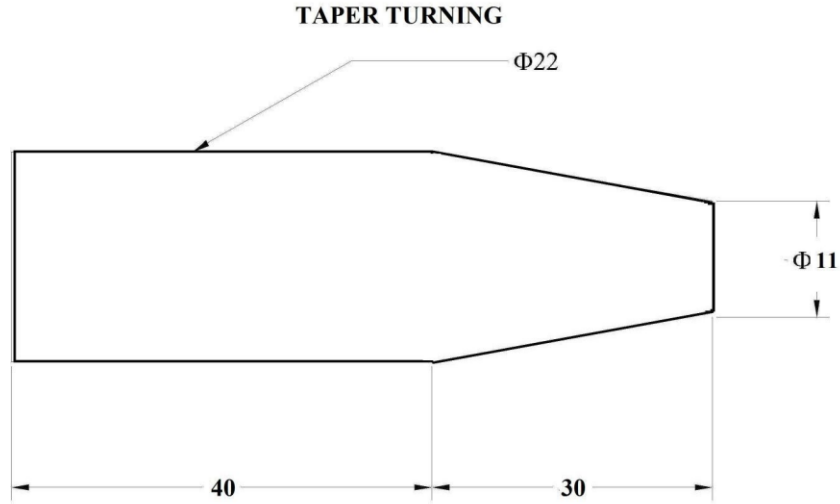
ALL DIMENSIONS ARE IN "mm"

DWG.NO.T03

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 22x60				**MATERIAL:** Aluminium			
PROGRAM NO:1003				**DWG.NO:3**			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Taper Turning	SDJCR121H11	DCMT11T302	3	3	1000	30

Cutting Cycle

G90 X(U) Z(W) R F

X   Diameter to which the movement is being made.

U   The incremental distance from the current tool position to the required final Diameter Z The Z Axis Co-ordinate to which the movement is being made.

W The incremental distance from the current tool position to the required Z axis Position R The difference in incremental of the cut start radius value and the cut finish radius value.

fFeed rate

TAPER TURNING

EX.NO:T03

DATE:

AIM:
     To write a manual part program for taper turning operation for component shown in DWG.NO.T03

MATERIAL REQUIRED:

Material   : Aluminium
Size   : Length 70mm, Diameter 22mm

PROGRAM:

O0002
G21 G98
G28 U0 W0
M06 T0101
M03   S1200
G00 X22 Z1
G90 X22 Z-30 R0 F30
X22 R-0.5
X22 R-1
X22 R-1.5
X22 R-2
X22 R-2.5
X22 R-3
X22 R-3.5
X22 R-4
X22 R-4.5
X22 R-5
X22 R-5.5
G28 U0 W0
M05
M30

RESULT:

     Thus the manual part program was written to the given dimensions and executed in CNC lathe.

Write a manual part program for **Multiple Turning** operation for component shown in **DWG.NO.T04**

MULTIPLE TURNING

ALL DIMENSIONS ARE IN "mm"
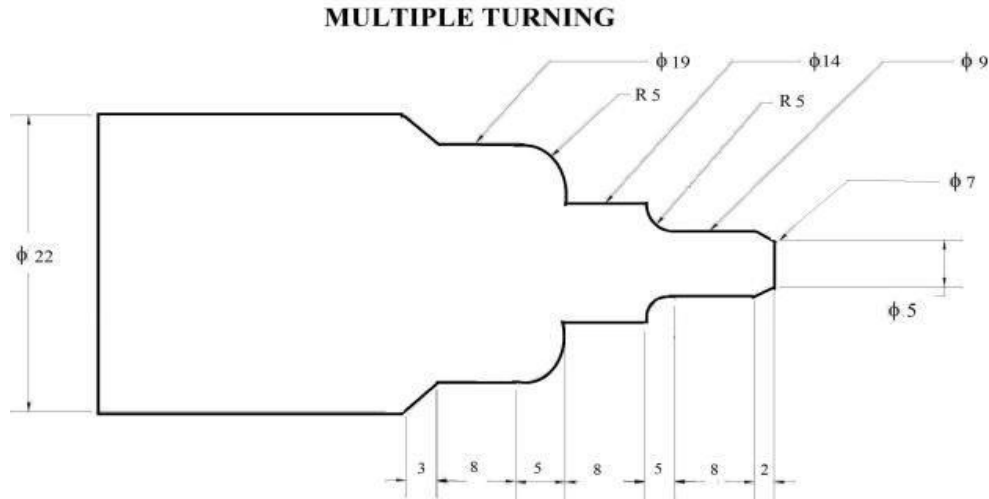
DWG.NO.T04

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 22x60				**MATERIAL:** Aluminium			
PROGRAM NO:1004				**DWG.NO:4**			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Multiple Rough turning	SDJCR121H11	DCMT11T304	1	1	1200	35
2	Finishing	SDJCR121H11	DCMT11T302	2	2	1450	25

Multiple Turning Cycle

G72 W (*w1) R (*r)   U(*u2) W(*w2) F(*f) S(*s) T(*t)

G72 P(*p) Q(*q)

where

*w1	depth of cut in z axis
*r	Relief Amount Starting
*P	Line Number Ending
	Line Number
*u2	the amount and direction of the finishing allowance left in the X Axis *q
*w2	the amount and direction of the finishing allowance left in the Z Axis
*f	Feed Rate
*s	spindle speed
*t	tool number

MULTIPLE TURNING (CANNED CYCLE)

EX.NO: T04

DATE:

AIM:
      To write a manual part program for multiple turning operation for component shown in DWG.NO.T04

MATERIAL REQUIRED:
Material   : Aluminium
Size   : Length 70mm, Diameter 22mm
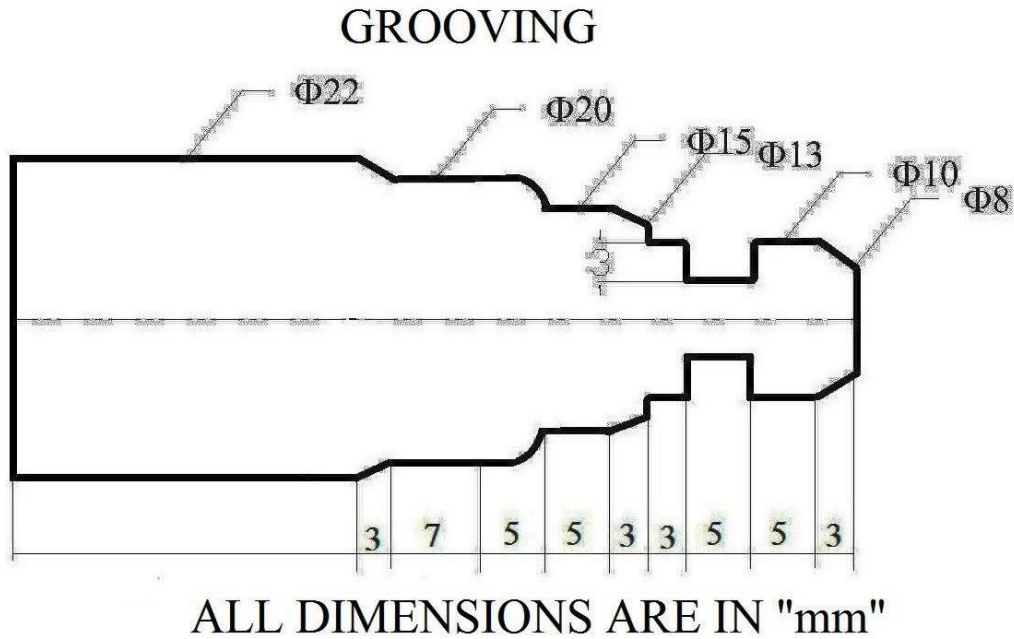
PROGRAM:

```
O000
G21 G98
G28  U0  W0
M06   T0303
M03   S1200
G00  X22  Z2
G71 U0.5 R1
G71 P10 Q20 U0.1 W0.1 F45
N10 G00 X7 Z0
G01 X9 Z-2
G01 Z-10
G02 X14 Z-15 R5
G01 Z-23
G03 X19 Z-28 R5
G01 Z-36
N20 G01 X22 Z-39
M03 S1600 F30
G70 P10 Q20
G28 U0 W0
M05
M30
```

RESULT
      Thus the manual part program was written to the given dimensions and executed in CNC lathe.

Write a manual part program for **External Grooving** operation for component shown in **DWG.NO.T05**

GROOVING

ALL DIMENSIONS ARE IN "mm"

DWG.NO.T05

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 22x60				**MATERIAL:** Aluminium			
PROGRAM NO:1005				**DWG.NO:**5			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Multiple Rough turning	SDJCR121H11	DCMT11T304	1	1	1200	35
2	Finishing	SDJCR121H11	DCMT11T302	2	2	1450	25
3	Grooving	HSS	3mm width	5	5	750	15

GROOVING CYCLE

G75 R (*r1)
G75 X(u) Z(w) P(*p) Q(*q) F (*f)

Where, r1-Return Amount
    X-Total Depth along X Axis (Absolute)
    u-Total Depth along X Axis (Incremental)
    z-Total Depth along Z Axis (Absolute)

w-Total Depth along Z Axis (Incremental)
p-Peck Increment in X Axis in microns
q -Stepping distance in Z Axis in microns
f -Feed Rate in mm

52

GROOVING

EX.NO:T05

DATE:

AIM:
To write a manual part program for external grooving operation for component shown in DWG.NO.T05

MATERIAL REQUIRED:
Material   : Aluminium
Size   : Length 70mm, Diameter 22mm

PROGRAM:
O0005
G21 G98
G28 U0 W0
M06 T03
M03 S1200
G00 X22 Z2
G71 U0.5 R1
G71 P10 Q20 U0.1 W0.1 F45
N10 G01 X8 Z0
G01 X10 Z-3
G01 Z-16
G01 X13
G01 X15 Z-19
G01 Z-24
G03 X20 Z-29 R5
G01 Z-36
N20 G01 X22 Z-39
M03 S1600
G70 P10 Q20 F30
G28 U0 W0
M06 T0707
M03 S600
G00 X10.5 Z-8
G75 R1
G75 X7 Z-10 P100 Q1000 F40
G28 U0 W0
M05
M30

RESULT
Thus the manual part program was written to the given dimensions and executed in CNC lathe.

Write a manual part program for **Multiple Threading** operations for component shown in **DWG.NO.T06**

MULTIPLE THREADING

ALL DIMENSIONS ARE IN "mm"

DWG.NO.T06

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 22x60				**MATERIAL:** Aluminium			
PROGRAM NO:1006				**DWG.NO:**6			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Multiple Rough turning	SDJCR121H11	DCMT11T304	1	1	1200	35
2	Finishing	SDJCR121H11	DCMT11T302	2	2	1450	25
3	Grooving	HSS	3mm width	5	5	750	15

MULTIPLE THREADING CYCLE

G76 P(m) (r)(a) Q(*q1) R(*r1)
G76 X(*x) Z(*z) P(*p2) Q(*q2) F(*f)

Where

m- Repetitive count in finishing (1 to 99)

r = Pull out angle

a= Angle of tool tip

*X = Minor diameter (core diameter)

*z = End position of thread

*p2 = Height of thread as a radius value x 1000, value in microns e.g. 1.02mm becomes p1020

*q2 = depth of the first cut as a radius value x 1000 , value is microns. E.g. 0.25 mm becomes Q250

*f = Lead or pitch of thread. E.g. lead 1.5 = F1.5

*q1 = Min. cutting depth

*r1 = Finishing allowance

MULTIPLE THREADING

EX.NO:T06

DATE:

AIM:

   To write a manual part program for multiple threading operation for component shown in DWG.NO.T06

MATERIAL REQUIRED:

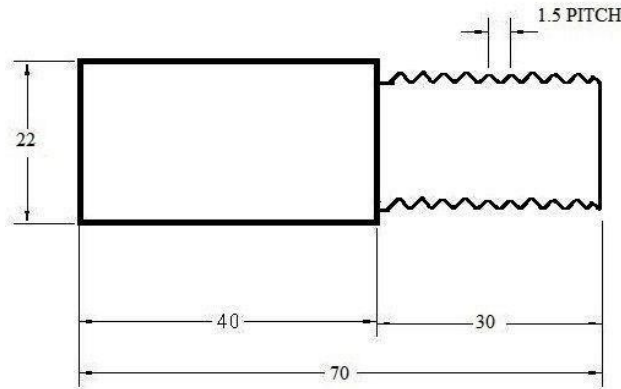Material : Aluminium
Size : Length 70mm, Diameter 22mm

PROGRAM:

O0006
G21 G98
G28 U0 W0
M06 T0404
M03 S600
G00 X22.5 Z2
G76 P031560 Q050 R0.02
G76 X20.162 Z-20 P920 Q100 F1.5
G28 U0 W0
M05
M30

RESULT

   Thus the manual part program was written to the given dimensions and executed in CNC lathe.

Write a manual part program for **Peck Drilling** operation for component shown in **DWG.NO.T07**

PECK DRILLING
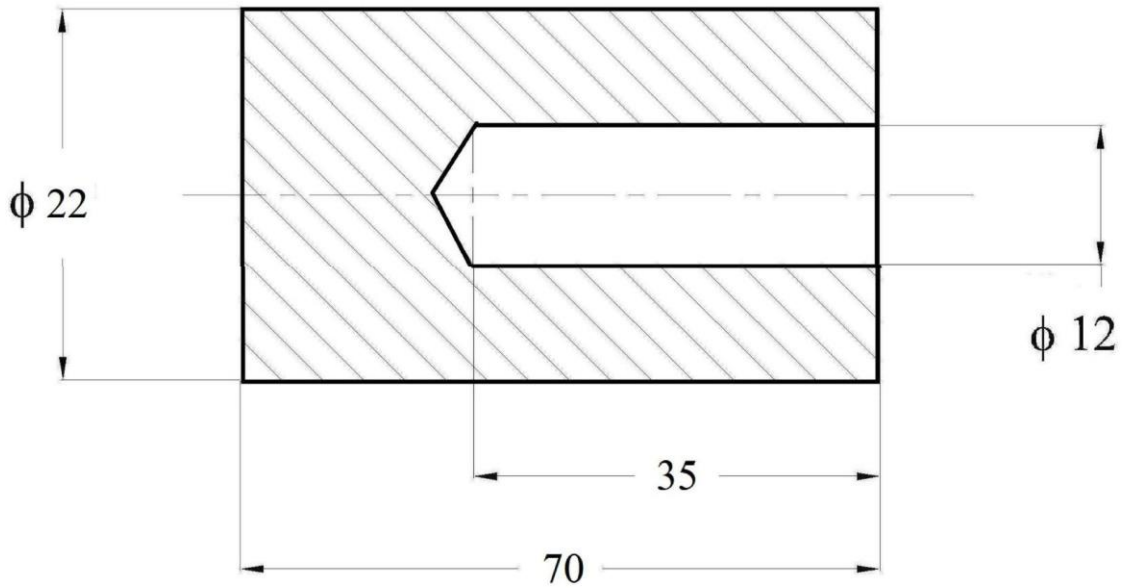
ALL DIMENSIONS ARE IN "mm"

DWG.NO.T07

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 22x60				**MATERIAL:** Aluminium			
PROGRAM NO:1007				**DWG.NO:**7			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Central Drill	6mm	-	6	6	1200	20
2	Drilling	12mm	-	8	8	800	20

PECK DRILLING

AIM:

To write a manual part program for peck drilling operation for component shown in DWG.NO.T07

MATERIAL REQUIRED:

Material   : Aluminium
Size   : Length 70mm, Diameter 22mm

PROGRAM:

```
O0007
G21 G98
G28 U0 W0
M06 T0606
M03 S1200
G00 X0
G00  Z2
G74 R1
G74 X0 Z-35 Q500 F20
G28 U0 W0
M06 T0808
M3 S800
G00 X0
G00 Z2
G74 R1
G74 X0 Z-35 Q500 F20
G28 U0 W0
M05
M30
```

RESULT

Thus the manual part program was written to the given dimensions and executed in CNC lathe.

VIVA QUESTIONS AND ANSWER

1. What Is CNC?

CNC router (Or Computer Numerical Control router) is a computer-controlled cutting machine related to the hand held router used for cutting various hard materials, such as wood, composites, aluminium, steel, plastics, and foams.

2. What Is The CNC Machine?

CNC Machining is a process used in the manufacturing sector that involves the use of computers to control machine tools. Tools that can be controlled in this manner include lathes, mills, routers and grinders. The CNC in CNC Machining stands for Computer Numerical Control.

3. How Does A CNC Work?

Computer Numerical Control – Taking digitized data, a computer and CAM program is used to control, automate, and monitor the movements of a machine. ... The CNC controller works together with a series of motors and drive components to move and control the machine axes, executing the programmed motions.

4. What Is A CNC Lathe Used For?

This style of lathe is commonly used under CNC control. Most CNC Swiss-style lathes today use one or two main spindles plus one or two back spindles (secondary spindles). The main spindleis used with the guide bushing for the main machining operations.

5. What Is The CNC Programming?

Most NC today is computer (or computerized) numerical control (CNC), in which computers play an integral part of the control. In modern CNC systems, end-to-end component design is highly automated using computer-aided design (CAD) and computer-aided manufacturing (CAM) programs.

6. What Is G Code?

G Code is the generic name for a control language for CNC (or Reprap) machines. It is a way for you to tell the machine to move to various points at a desired speed, control the spindle speed, turn on and off various coolants, and all sorts of other things.

7. What Is A CNC Controller?

The CNC controller is the brain of a CNC system. A controller completes the all important link between a computer system and the mechanical components of a CNC machine.

8. What Is The Machine Control Unit?

The machine control unit (MCU) is the heart of a CNC system. It is used to perform the following functions: To read the coded instructions. To decode the coded instructions. To implement interpolations (linear, circular, and helical) to generate axis motion commands.

9. What Is The Meaning Of Fanuc?

FANUC is one of the largest makers of industrial robots in the world. FANUC had its beginnings as part of Fujitsu developing early numerical control (NC) and servo systems. The company name is an acronym for Fuji Automatic NUmerical Control.

10. How Do You Program A CNC Machine?

- Learn all Fanuc CNC Programming Skills .
- Learn all parts of CNC G-Code programming:
- All details of Lathe-Mill Programming and Setup on a series of 2 hour long DVDs:
- Start learning for free:
- Go to www.youtube.com and put in Heinz Putz, CNC Part making.
- Each is 2 hours long and teaches CNC skills in great detail.

11. When Was The First CNC Machine?

The Invention of CNC Machining. In order to understand where CNC Machining first appeared, it is important to note that the said machine's advent can be traced back to the invention of the NC (numerical controlled) machine made by John T. Parsons during the 1940s-1950s.

12. What Is The Meaning Of G Code?

G-code (also RS-274), which has many variants, is the common name for the most widely used numerical control (NC) programming language. It is used mainly in computer-aided manufacturing for controlling automated machine tools. G-code is sometimes called G programming language.

13. What Is An NC Programmer?

An NC programmer is a person who is responsible for the preparation and development of numerical programs used to drive NC and CNC tools. The programmer will select the necessary tools that are used in cutting, and also in establishing the sequence of operations of the machine's operations.

14. What Is Turning Operation?

Turning is a machining process in which a cutting tool, typically a non-rotary tool bit, describes a helix tool path by moving more or less linearly while the work piece rotates.

15. What Is The Use Of Lathe Machine?

A lathe is a machine tool that rotates the work piece on its axis to perform various operations such as cutting, sanding, knurling, drilling, or deformation, facing, turning, with tools that are applied to the work piece to create an object with symmetry about an axis of rotation.

16. What Is Meant By Canned Cycle?

A Canned cycle is a way of conveniently performing repetitive CNC machine operations. Canned cycles automate certain machining functions such as drilling, boring, tapping, pocketing, etc... Canned cycles are so called because they allow a concise way to program a machine to produce a feature of a part.

17. What Is The Hardwired Control?

Hardwired control is a control mechanism that generates control signals by using an appropriate finite state machine (FSM). Micro programmed control is a control mechanism that generates control signals by reading a memory called a control storage (CS) that contains control signals.

18. What Is The CNC Milling?

CNC milling is a specific form of computer numerical controlled (CNC) machining. Milling itself is a machining process similar to both drilling and cutting, and able to achieve many of the operations performed by cutting and drilling machines. Like drilling, milling uses a rotating cylindrical cutting tool.

19. What Is A DNC Machine?

Direct numerical control (DNC), also known as distributed numerical control (also DNC), is a common manufacturing term for networking CNC machine tools.

20. What Are The Different Types Of CNC Machines?

There are basically five different types of CNC machines**:**

- CNC Plasma Cutting Machine.
- CNC Laser Cutting Machine.
- CNC Milling Machine.
- CNC Router Machine.
- CNC Lathe Machine.

21. What Is The D Girl?

A d-girl (a.k.a. development girl) is a non-influential, entry-level staffer in a film production company. Responsibilities include: finding and identifying story ideas worthy of adaptation into a script, and potentially a feature film; and writing script coverage for scripts submitted to the production company.

22. What Is Meant By VMC Machine?

A vertical machining center (VMC) is a machining center with its spindle in a vertical orientation. High-end VMCs are high-precision machines often used for tight-tolerance milling, such as fine die and mold work. Low-cost vertical machining centers are among the most basic CNC machine tools.

23. What Does A CNC Machine Operator Do?

CNC machine operators set up and work with computer numerically controlled (CNC) machines, a type of equipment that is usually found in the metalworking industry. Their jobs often involve repetitive tasks and the monitoring of multiple machines.

24. What Materials Can Be Cut On A CNC Machine?

Materials Used: Almost any material can be used in a CNC machine. It really depends on the application. Common materials include metals such as aluminum, brass, copper, steel, and titanium, as well as wood, foam, fiberglass, and plastics such as polypropylene.

25. What Is The Application Of CNC Machine?

- Electronics Parts Manufacturing.
- Engraving Machine Applications.
- Machining Composites.
- 5 Axis Machining.
- Dental Milling Applications.
- Micro Hole Drilling.
- Machining Aluminum.
- Machining Plastics.

26. What Is Fanuc In CNC?

The world standard CNC from FANUC powers. FANUC i Series CNC. A wide range of CNCs for simple machine tools to the most complex, as well as for other general industrial machines. FANUC has produced over 3.6 million CNCs and those CNCs are used to power machines around the globe.

27. Define numerical control machine

Numerical control machine cane be defined as a form of programmable machine in which the process are controlled by a program of numbers, letters, and symbols.

28. What is NC part programming?

NC part programming is the step by procedure of by which the sequence of processing steps to be performed on the NC machine is controlled by a program of numbers, letters, and symbols.

29. What is APT language?

APT [automated programming language]is a computer program, it automatically calculates the tool path, generates program and controls the machine by receiving general high level languages.

30. **Mention any 4 post processor statement in APT.**

COOLNT END FEDRAT/

MACHIN/

RAPID

31. **What is meant by machining centre?**

The machining centre is CNC system with automatic tool changing arrangement that is designed toperform a verity of machining operations, with large number of cutting tools.

32. **What is part program?**

Part program is a high level language containing the instructions for machining a part to variousstandard words, codes and symbols.

33. **What is post processing?**

Post processing is a computer program that takes a generalized part program output and adopts it to a particular machine control unit and machine tool combination. It is the basic intelligence required to change the program into computer language.

34. **Write the order of instructions in a part program.**
End of Preparatory function   Feed function   Tool function   block

35. **What is manual part programming?**

Manual part programming is a process of writing programs which consists of a set of instructions [contains codes, symbols and numbers] to carry out the machining of the work.

36. **What is preparatory function?**

It is word address format represented by the letter G, Followed by a numerical code for theoperation of the control unit to instruct the machine tool.

37. **What is canned cycle?**

Canned cycle simplifies and shortens the programming in such way whenever any one of theoperation is required.

38. **What are the major areas to be considered in the design of NC machine tools?**

i.Machine structure and frame
ii.Location of transducer
iii.Slideways
iv.Elements of transmission and positioning of sliders

v.Spindles

vi.Tool holding arrangements

39. How the heat effect on the machine bed, tool holder can be taken care?

i. Providing correctly designed mild steel structure with higher stiffness
ii. Use of ribs, braces, angle plates to increase stiffness
iii. Normal weight distribution over the entire frame
iv. The hollow cross section for beds, bases and columns with a number of ribs welded with the walls cater for the rigidity as well as opening for inspection, lubrication and collection of chip coolants.
v. Thermo symmetrical design of all parts.
Vi .Providing large heat removing surfaces vii. Use of excellent coolants
viii. Avoiding direct as well as local sources of heat such as sunlight and electrical motors.oil pumps respectively.
ix. Reduction of ambient temperature by using air conditioning units
x. Proper alignment of the machine elements relative to each other while in operation as well as in stationary conditions

40. Explain Slide and Slide ways

In general machine tools are provided with tables, slides, carriages etc., to carry the work pieces or cutting tools etc., These parts are sliding in nature and mounted on the ways that are fixed on the other parts (column, housing, bed or knee) of the machines known as sliding ways.

41. Explain the term "stick-up"

Conventional sliders operating under sliding friction do not have a constant coefficient of friction and the highest value of co-efficient tends to be at the lowest rates of slide velocity.

This phenomenon given to the familiar "sticking" of oil lubricated sliding surfaces when the fine adjustment is needed, a jerky action when movement takes place at low velocities. The term "stick- slip" is used to describe these situations

42. Define Milling

Milling may be defined as the machining process which is used to remove excess materialfrom the work-piece with a rotating multi-point cutter called milling cutter.

43. Name different types of Knee and Column type milling machines.

1. Horizontal Milling Machine
2. Vertical Milling Machine
3. Universal Milling Machine
4. Omniversal Milling Machine

44. Name different types of production milling machines.

1. Simplex Milling Machine
2. Duplex Milling Machine
3. Triplex Milling Machine

45. Name some special milling machines.

1. Rotary table milling machine
2. Drum type milling machine
3. Profile milling machine
4. Planetary milling machine
5. Pantograph milling machine
6. Offset milling machine

46. Name principal parts of knee and column type milling machine.

1. Base
2. Column
3. Knee
4. Saddle
5. Table
6. Spindle
7. Over arm
8. Arbor

47. What are the limitations of a milling machine?

The major limitations of a milling machine are as follows:

1. Milling machine is mostly used for machining jobs of smaller size
2. Its speed is slow for machining long jobs
3. Cutting tool is costlier

48. Name two milling methods:

1. Conventional milling (Up Milling)
2. Climb milling (Down Milling)

49. Define conventional milling

In conventional milling, the cutting tool rotates in the opposite direction to the movement of table. In the conventional or up milling, the chip starts at zero thickness gradually increases to the maximum size.

50. Define Climb milling

When the feed motion is in the same direction as that of motion of the cutter, the milling is known as climb milling or down milling. In climb milling, the chip starts at maximum thickness and goes to zero thickness gradually.

51. Define milling cutter.

A hardened steel disc or cylinder on which the cutting teeth are arranged either on itsperiphery or on its end or on both is called milling cutter. Usually the cutter is held in a fixed, but rotating position and the work-piece passes the cutter during the machining operation.

52. Name the materials used for milling cutters.

Milling cutters are generally made of high speed steel or sintered carbides.

53. What is plain milling cutter?

It is also called slab milling cutter and is basically cylindrical with the cutting teeth on its periphery. It is generally used for machining flat surfaces.

54. What is a side milling cutter?

This cutter is similar to plain milling cutter except that is has teeth on its sides also. These cutters may have straight, spiral or staggered teeth. Further these may be solid, inserted blade or tipped construction and may be profile sharpened or form relieved.

55. What is the use of metal slitting saw cutter?

It is used for cutting deep slots and parting off materials into pieces.

56. Define angle milling cutter.

Any milling cutter which has cutting teeth at an angle to the axis of rotation is known as angle milling cutter.

57. Define Numeraical Control Machine?

Numerical control machine cane be defined as a form of programmable machine in which the process are controlled by a program of numbers, letters, and symbols.

59. What Is Nc Part Programming?

NC part programming is the step by procedure of by which the sequence of processing steps to be performed on the NC machine is controlled by a program of numbers, letters, and symbols.

59. What Is Apt Language?

APT [automated programming language]is a computer program, it automatically calculates the tool path, generates program and controls the machine by receiving general high level languages.

60. What Is Meant By Machining Centre?

The machining centre is CNC system with automatic tool changing arrangement that is designed to perform a verity of machining operations, with large number of cutting tools.

61. What Is Part Program?

Part program is a high level language containing the instructions for machining a part to various standard words, codes and symbols.

62. What Is Post Processing?

Post processing is a computer program that takes a generalized part program output and adopts it to a particular machine control unit and machine tool combination. It is the basic intelligence required to change the program into computer language.

63. Write The Order Of Instructions In A Part Program?

End of Preparatory function Feed function Tool function block.

64. What Is Manual Part Programming?

Manual part programming is a process of writing programs which consists of a set of instructions [contains codes, symbols and numbers] to carry out the machining of the work.

65. What Is Preparatory Function?
It is word address format represented by the letter G, Followed by a numerical code for the operation of the control unit to instruct the machine tool.

66. What Is Canned Cycle?

Canned cycle simplifies and shortens the programming in such way whenever any one of theoperation is required.

67. What Are The Major Areas To Be Considered In The Design Of Nc Machine Tools?

- Machine structure and frame
- Location of transducer
- Slide ways
- Elements of transmission and positioning of sliders
- Spindles vi.Tool holding arrangements

68. How The Heat Effect On The Machine Bed, Tool Holder Can Be Taken Care?

- Providing correctly designed mild steel structure with higherstiffness
- Use of ribs, braces, angle plates to increase stiffness
- Normal weight distribution over the entire frame
- The hollow cross section for beds, bases and columns with a number of ribs welded with the walls cater for the rigidity as well as opening for inspection, lubrication and collection of chip coolants.
- Thermo symmetrical design of all parts.
- Providing large heat removing surfaces vii.Use of excellent coolants
- Avoiding direct as well as local sources of heat such as sunlight and electrical motors.oil pumps respectively.
- Reduction of ambient temperature by using air conditioning units
- Proper alignment of the machine elements relative to each other while in operation as well as in stationary conditions

69. Explain Slide And Slideways?

In general machine tools are provided with tables, slides, carriages etc., to carry the work pieces or cutting tools etc., These parts are sliding in nature and mounted on the ways that are fixed on the other parts (column, housing, bed or knee) of the machines known as sliding ways.

70. What Kind Of Pc Is The Minimum Hardware Requirement For Operate This CNC-program?

Use a PC 486 DX with 33 MHz or higher. It requires a DOS Operation System installed, e.g. MS-DOS 6.22. MS-Windows 3.1/95/98 also includes an use full DOS-Mode.

71. Which Stepper motor-driver boards are Compatible To This CNC-program?

Those with 5V CMOS/TTL logic and 2 Bit (direction and clock/enable) standard signals.

72. How Does It Work About The Reference- And End-switches?

- Pin11(reference-switch) must be on signal-ground when the program starts and ends.
- Pin10(end-switch) is normally closed to GND. If open, it can trigger a reference drive.
- When reference(Pin11) is open and end-switch(Pin10) opens also, then drive back toorgin!
- This saves time in case of interruption of processing. Pin10 is also good for overloaduse.
- All 9 axis will be driven back to orgin, one after the other. Order: W-V-U T-S-R Z-Y-X.

73. How Is The Order Of Signals Sending Out To The Printerports?

- First the direction signal send out, e.g. if X-forward and spindle enabled [binary 64].
- At second the direction signal inclusive the clock signal send out [binary 65].
- A step is executing with pos. edge of clock! Code in C: outp(0x378,64); outp(0x378,65);
- The databits 1,3,5 are the direction bits. If forward set on [0], if backwards set on [1].
- The signal of current boost should be connected to the clock signal in parallel.
- Make sure that the primary current (30% to 70% of boost) is enabled on the driver-unitto hold the motor in constant position while don't have motions.

74. How Do I Have To Connect The Step-encoders And How Does It Work?

- When the program starts Pin 10,11,12,13,15 must be on ground (Low)!
- Pin 12(X),13(Y),15(Z) will be checked every 4th full step whether they are on GND (Low).
- Otherwise it should exist a High-Level.
- If the Low-Level won't be reached at 4th full step (because of overload), then the program try to push more steps (micro steps) on the printer port until the Low-Level will be reached.
- The max. amount of failed micro steps until automatic program stop and beginning of auto-reference-drive can be adjusted in the file "paramet".
- An immediate reference-drive also can be triggered if any key is hit on the keyboard.
- You may design your step-encoders by yourself with a disk and photo gap.
- The position of the photo gap must be adjusted precisely that it does pass the current if the full step is reached. Then it pulls down the High-Level to Low-Level.

- If the design is an open design, please protect from daylight or other light on photo gap.

75. Some Producers Of Stepper motor-driver units Offer A "parallel-interface". Do I Need It?

Not required!

If you want to drive more than 3 axis, then you have to install a 2. or 3. port in your PC.The order

of parallel ports are: 378h 278h 3BCh (special requirements on request).

76. If Switch S1 (bit7 = Milling Spindle On) Is Enabled, Why Does Bit8 Pulsing While Motions?

Bit8 puts an impuls out, depending on length of motion (adjustable), while s1 is enabled.Good for

drive an electromechanic coolingpump, working with diaphragm or piston.

With s2 (Bit8) you can put an extraimpuls on coolingpump, e.g. short before drilling a hole.

77. Is It Possible Milling From Cad Data And Drawings?

Yes! With the Fullversion we deliver a tool for DXF- and HPGL-Import.
You may also use the integrated Teach-In function for draw figures and then plot with your machine.It is generated a programcode in the file "teachdat".
Call of Teach-In: "cnc teach 100 bs" The "100" means 100 percent screenscale!

With this function it is possible to scale the screen. So you can put a foil on the screen andthen use the mousepointer to move along the figures.

Call "cnc teachdat 0" in DOS Mode and your machine will mill the figure in 1:1 or other scale!The "0"

is the reduction of speed, means fastest gear (most slowly == "4.000.000.000").

78. How Do I Realise Effective Programming, Axis-transportation And Serial Works?

- Choose a cutter with suitable diameter for your most common works, e.g. Dia 4mm.
- Do coding only for this specific tool-diameter. Use the integrated cycles!
- The application of the cycles avoids often tool changes and cutter radius compensation.
- Axis-Transportation with self-coded assisting programs will transport the Start/Endposition.
- Serial works with batch procedures (*.bat) is possible!

79. Why The Z-axis Is Counting Positive When It Comes Down? (different To Din-iso Standard).

- A logical improvement! You save time not always type the "minus" in front of the figure.
- The basic scale system is absolute, means that the scale is measuring from a zero-position.
- All measurements of the triangle-, square- and sparkerosion-cycles are completely increment.
- From the circle-cycle just the Radius is increment, the other data is measured absolute.
- The startposition of a cycle is the actual position (automatic zero-point compensation).

- After finish a cycle the automatic zero-point compensation is reset to the old position.

80. Do The Axis Need All The Same Mechanic Gear And Spindle Slopes?

Each axis can have a different gear and spindle slope.

81. How Driving To More Precise Positions Than The Standard 0.1mm Raster?

Full values in the CNC-code are 1/10mm-Motions. This makes coding quick and easy. Decimalvalues, e.g. g 50.12 22.4 11.7 are in real X=5.012mm Y=2.24mm Z=1.17mm

82. Explain about G codes?

G-code is the common name for the most widely used numerical control (NC) programming language, which has many implementations. Used mainly in automation, it is part of computer-aided engineering. G-code is sometimes called G programming language

83. Mention few important G codes?

G00 - Positioning at rapid speed; Mill and Lathe G01 - Linear interpolation (machining a straight line); Mill and Lathe G02 - Circular interpolation clockwise (machining arcs); Mill and Lathe G03 - Circular interpolation, counter clockwise; Mill and Lathe G20 - Inch units; Mill and Lathe G21 - Metric units; Mill and Lathe

84. What is the use M codes?

A word used to signal an action from a miscellaneous group of commands. M codes change cutting tools, turn on or turn off the coolant, spindle, or work piece clamps, etc

85. Write about some important M codes?

M00 - Program stop; Mill and Lathe M01 - Optional program stop; Lathe and Mill M02 - Program end; Lathe and Mill M05 - Spindle off; Lathe and Mill

86. What is the use of box facing cycle?

Fanuc G94 facing cycle is used for simple facing (one-pass facing) however multiple passes are possible by specifying the Z-axis location of additional passes

87. What is the difference between G00 and G01codes?

G00 - Positioning at rapid speed; Mill and Lathe G01 - Linear interpolation (machining a straight line);Mill and Lathe

88. How to change the tool in CNC program?

M06 - Toolchange;

89. How to change the tool speed in cnc lathe?

M03 - Spindle on clockwise; Lathe and Mill M04 - Spindle on counterclockwise; Lathe and Mill

90. What is the difference between absolute and incremental system?

Absolute positioning means that the tool locations are always defined in relation to the zero point . incremental positioning means that the next tool location must be defined with reference to the previous tool location.

91. What are the axes to be considered while writing program for cnc lathe?

X and Z Axis

92. Mention the major components of the CNC machine?

CNC machine tools comprise the following main components:

- machine tools whose tables are controlled in X and Y axes and spindles controlled in Z axis.

- drive systems: drive motors, leading ball-screws and amplifier circuits.

- machine control units which read and decode part program instructions, generate axis motions commands, receive feedback signals regarding the position and the speed of each driven axis, etc.

- part programs that are a series of lines comprising the instructions needed to machine the part.

- program input devices which help to input the part program within CNC control.

- feedback systems that measure the position of cutting tool at each moment by mean of transducers of position and speed.

93. What is the code for Threading cycle?

G92

94. What is the code for Incremental and absolute co-ordinate system?

G90 and G91

95. What is the code for coolant control?

M7 - turn mist coolant on.• M8 - turn flood coolant on.• M9 - turn all coolant off.•

96. What is use of dry run option?

A dry run is a testing process where the effects of a possible failure are intentionally mitigated