INTRODUCTION

Numerical control :(NC)

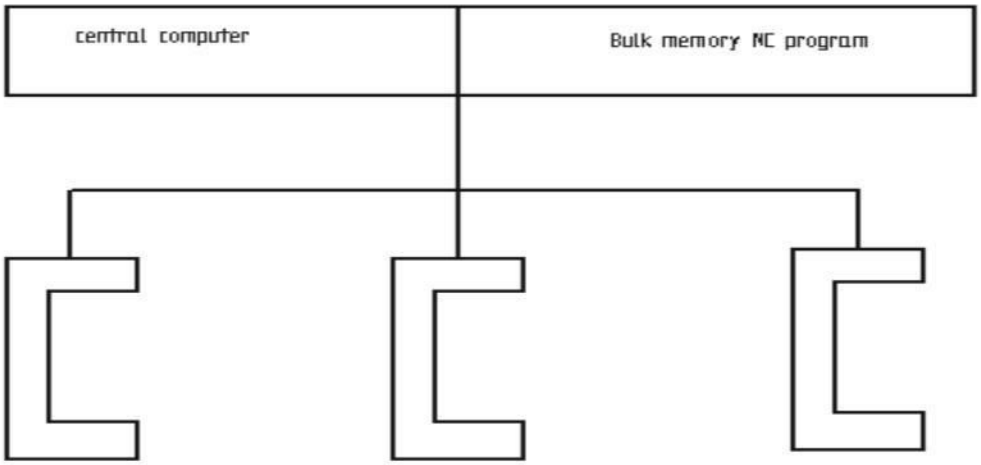
It can be defined has form of programmable automation in which the process is controlled by numbers, letters and symbols in NC the numbers forms a program of instructions designed for a particular work part or job.

When the job changes the program of instruction is changed. This capability will change program for each new job is what gives NC flexibility.

Ex: GOO XO YO ZO

Computer numerical control :(CNC)

Numerical control integrated computer control includes one or more microprocessor, mini computers. The logic function or program the control comprises a program that is stored in the memory.



Direct numerical control: (DNC)

It can be defined has a manufacturing system in which a number of machines are controlled by a computer through direct connection & in real time.

Dept. of ME,

NC motion control system:

In NC there are 3 basic types of machine control system

1. Point to Point

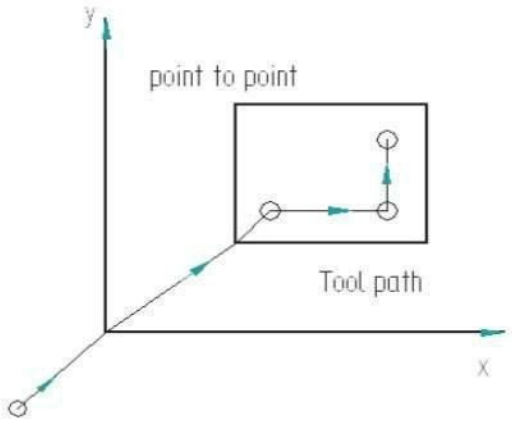
2. Straight cut

3. Contouring

1) Point to point

It is also sometimes called positioning system. In point to point the objective of the machine tool control system is to the cutting to pre defined location once the tool reaches the defined location the machining operation is performed at that position.

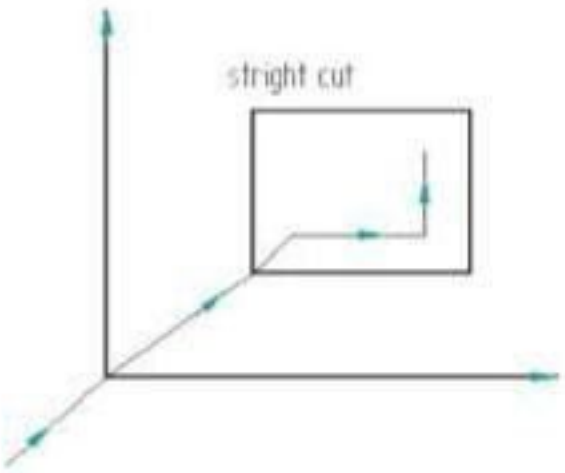
EX: NC drill presses.



2) Straight cut NC

Straight cut control system is capable of moving the cutting tool, parallel to one of the major axes at controlled rate suitable for machining. It is therefore appropriate for performing milling operation to fabricate work piece of rectangular configurations.

Dept. of ME,



FUNDAMENTALS OF PART PROGRAMMING

NUMERICAL CONTROL PROCEDURE

The following are the basic steps in NC procedure

- ∑ Process Planning
- ∑ Part Programming
- ∑ Part Program entry
- ∑ Proving the part program
- ∑ Production

A) PROCESS PLANNING

The part programmer will often carry out the task of process planning. Process planning is the procedure of deciding what operations are to be done on the component, in what order, and with what tooling and work holding facilities. Both the process planning and part programming aspects of manufacture occur after the detail drawings of a component have

been prepared. The following procedure may be used as a guide to assist the programmer, by describing each step required in preparing the method of production.

PROCESS PLANNING

∑ Receive the part drawing from part drawing information, check suitability of part to be machined against the machine capacity.

∑ Determine a method of driving the component (chuck type, chuck size, type of jaw) and the method of machining.

∑ Determine the tooling required to suit the method of machining and utilize as much as possible the tools which are permanently in the turret set upon the machine.

∑ Determine the order of machining and the tooling stations.

∑ Determine planned stops for checking dimensional sizes where required by operator

∑ Determine cutting speeds based on

- Component material, method of driving, rigidity of component
- Tooling selected for roughing and finishing

∑ Determine the depths of cut and feeds for roughing operations

∑ Determine surface finish requirements, the cutter nose radius most suited for finishing operations and determine feed rates.

∑ Allocates tool offsets as required

∑ Complete planning sheet

B) **PART PROGRAMMING**

∑ After completing the planning sheet, draw the component showing the cutter

∑ Indicate the ordering code for each tool and grade and type of inserts to be used.

∑ Write the part program according to the sequence of operations.

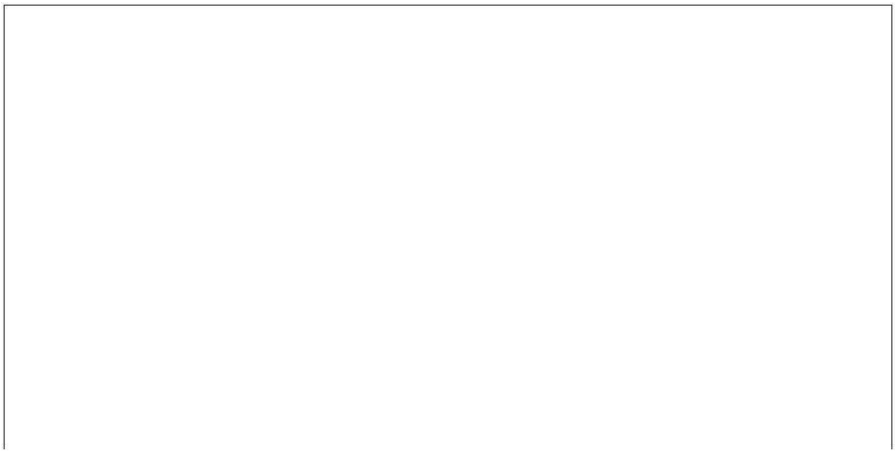
C) PART PROGRAM ENTRY (OR) TAPE PREPARATION

The part program is prepared / punched on a 25 mm wide paper tape with 8 tracks and is fed to MCU in order to produce a component of interest on machine tool. Other forms of input media include, punched cards, magnetic tape, 35 mm motion picture film. The input to the NC system can be in two ways:

1. Manual data input
2. Direct Numerical control.

1) **Direct Data Input (MDI):** Complete part programs are entered into CNC control unit via the console keyboard. It is suited only for relatively simple jobs. The most common application for MDI is the editing of part programs already resident in controllers memory.

One variation of MDI is a concept called "Conversational Programming". CNC machines are programmed via a question and answer technique whereby a resident software program asks the operator a series of questions. In response to the operators input, and by accessing a pre-programmed data file, the computer control can.



- Select numerical values for use within machining calculations
- Perform calculations to optimize machining conditions
- Identify standard tools and coordinates
- Calculate cutter paths and coordinates
- Generate the part program to machine the

component

A typical dialogue from the machine would be as follows for the operator to identify such things as:

- Machined shape

required

- Size of the raw material

blank

- Machining allowances,

cut directions

- Tools and tool detail

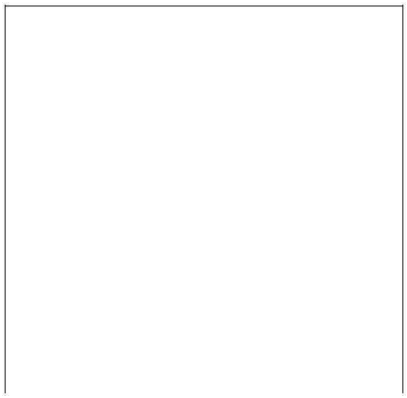
etc.

The operator may then examine and prove the program via computer graphics simulation on the console VDU. After this, the program is stored or punched on tape. Although there is some sacrifice in machine utilization, actual programming time is minimal and much tedious production engineering work is eliminated.

2) **Direct Numerical Control:** The process of transferring part programs into memory of a CNC machine tool from a host computer is called Direct Numerical Control or DNC

D) PROVING PART PROGRAMS

It is safe practice to check the programmed path for any interference between the tool and the work before using the part program for production. The proving part program is done by:



- Visual inspection
- Single step

execution

- Dry run
- Graphical simulation.

Visual Inspection: It represents the method of checking visually the program present in the memory of the CNC machine. In this, actual program is run and the programmed movements in all axes are to be checked along with ensuring the tool offset and cutter compensation feature. This method represents the least form of verification and should not be relied up on entirely.

Single Step Execution: Before auto-running the part program it should be executed in a step mode i.e. block by block. During this execution, spindle speed and feed rate override facilities are to be used so that axes movement can be easily monitored. This operation may be carried out with or without mounting the component on the machine.

Dry run: A dry run consists of running the part program in auto-mode. During this, the component is not installed on the machine table and the cutting is done in air. The purpose of this run is to verify the programmed path of the tool under continuous operation and to check whether adequate clearance exist between the clamping arrangement and other projections within the set up. Feed rate override facilities are used to slow down the speed of execution of the program.

Graphical simulation: A graphical simulation package emulates the machine tool and, using computer graphics, plots out the machine movements on a VDU screen. Machine movement often takes the form a cutting tool shape moving around the screen according to the programmed movements. When the tool shape passes over a shaded representation of the component, it erases that part of the component. The resulting shape, lest after the execution represents the shape of the finished component. Any gross deviations from the intended tool path can be observed and any potential interference can be highlighted.

Dept. of ME,

PART PROGRAMMING GEOMETRY FOR TURNING

A. COORDINATE SYSTEM FOR A CNC LATHE.

Machining of a work piece by an NC program requires a coordinate system to be applied to the machine tool. As all machine tools have more than one slide, it is important that each slide is identified individually. There are two planes in which movements can take place

- ∑ Longitudinal.
- ∑ Transverse.

Each plane is assigned a letter and is referred to as an axis,

- ∑ Axis X

Dept. of ME,

∑ Axis Z

The two axis are identified by upper case X, Z and the direction of movement along each axis (+) or (-). The Z axis is always parallel to the main spindle of the machine. The X axis is always parallel to the work holding surface, and always at right angles to the Z axis. The coordinate system for turning operations is shown in figure below

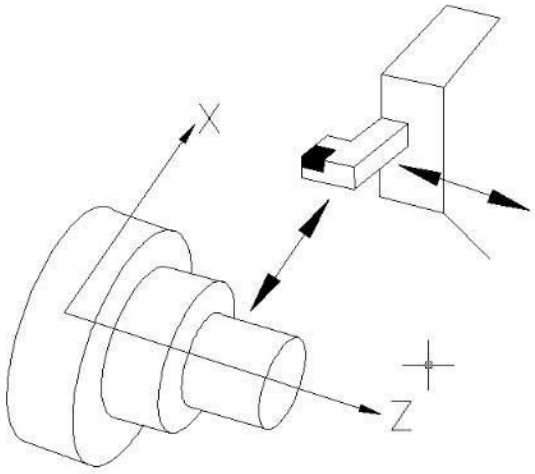


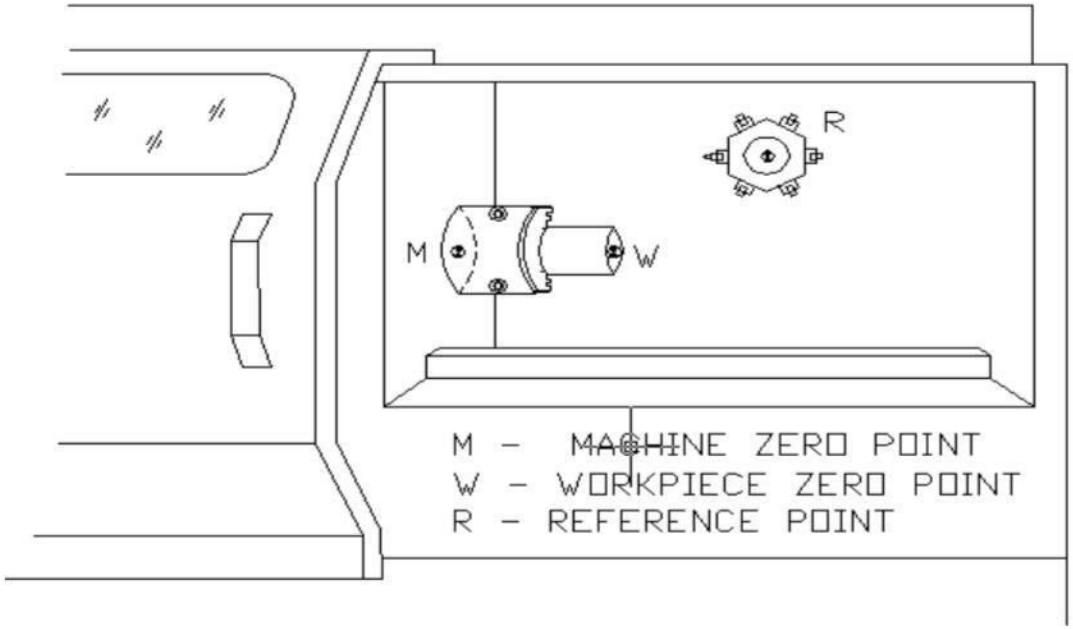
Fig 1. COORDINATE SYSTEM FOR TURNING OPERATIONS

B. ZERO POINTS AND REFERENCE POINTS

All CNC machine tool traverses are controlled by coordinating systems. Their accurate position within the machine tool is established by "ZERO POINTS".

MACHINE ZERO POINT (M): is specified by the manufacturer of the machine. This is the zero point for the coordinate systems and reference points in the machine. On turning lathes, the machine zero point is generally at the center of the spindle nose face. The main spindle axis (center line) represents the Z axis; the face determines the X axis. The directions of the positive X and Z axes point toward the working area as shown in figure below:

Dept. of ME,



WORKPIECE ZERO POINT (W): This point determines the workpiece coordinate system in relation to the machine zero point. The workpiece zero point is chosen by the programmer and input into the CNC system when setting up the machine. The position of the workpiece zero point can be freely chosen by the programmer within the workpiece envelope of the machine. It is however advisable to place the workpiece zero point in such a manner that the dimensions in the workpiece drawing can be conveniently converted into coordinate values and orientation when clamping / chucking, setting up and checking, the traverse measuring system can be effected easily.

For turned parts, the work piece zero point should be placed along the spindle axis (center line), in line with the right hand or left hand end face of the finished contour as shown in figure. Occasionally the work piece zero point is also called the "program zero point."

REFERNCE POINT (R): This point serves for calibrating and for controlling the measuring system of the slides and tool traverses. The position of the reference point as shown in figure below 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. If current slide and tool position data should be lost in the control system as for example, through an electrical failure, the machine must again be positioned to the reference point to re-establish the proper positioning values.

PREPARATORY FUNCTION (G-Codes).

G CODES

G00 Positioning (Rapid Transverse)

G01 Linear Interpolation (Feed)

G02	Circular Interpolation (CW)
G03	Circular Interpolation (CCW)
G04	Dwell
G20	Inch Data Input
G21	Metric Data Input
G28	Reference point return
G40	Tool nose radius compensation cancel
G41	Tool nose radius compensation left
G42	Tool nose radius compensation right
G50	Work coordinate change/ Max. Spindle speed setting
G70	Finishing cycle
G71	Multiple Turning Cycle 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

G90 Cutting cycle A (Turning)

G94 Cutting cycle B (Facing)

G96 Constant surface speed control

G97 Constant surface speed control cancel

G98 Feed per minute

G99 Feed per revolution

MISCELLANEOUS FUNCTION (M Codes)

M Codes are instructions describing machine functions such as calling the tool, spindle rotation, coolant on, door close/open etc.

M CODES

M00 Program Stop

M02 Optional Stop

M03 Spindle Forward (CW)

M04 Spindle Reverse (CCW)

M05 Spindle Stop

M06 Tool Change

M08 Coolant On

M09 Coolant Off

M10 Vice Open

M11 Vice Close

M13 Spindle Forward, Coolant On

M14 Spindle Reverse, Coolant On

M30 Program End

M38 Door Open

M39 Door Close

M98 Subprogram Call

M99 Subprogram Exit

COMPUTERISED NUMERICAL CONTROL MILLING

PART PROGRAMMING FUNDAMENTALS

1. PART PROGRAMMING GEOMETRY

COORDINATE SYSTEM FOR A CNC MILL

Machining of a work piece by an NC program requires a coordinate system to be applied to the machine tool. As all machine tools have more than one slide, it is important that each slide is identified individually. There are three planes in which movement can take place.

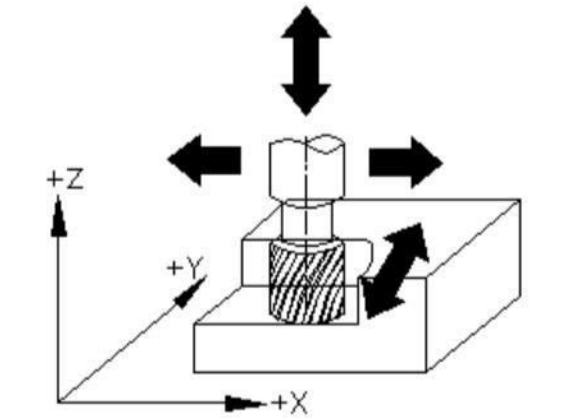
- Longitudinal

- Vertical
- Transverse

Each plane is assigned a letter and is referred to as an axis, i.e,

- Axis X
- Axis Y
- Axis Z

The three axes are identified by upper case X, Y and Z and the direction of movement along each axis is specified as either '+' or '-'. The Z axis is always parallel to the main spindle of the machine. The X axis is always parallel to the work holding surface, and always at right angles to the Z axis. The Y axis is at right angles to both Z and X axis. Figure shows the coordinate system for milling.



COORDINATE SYSTEM FOR MILLING

B. ZERO POINTS AND REFERENCE POINTS

MACHINE ZERO POINT (M): This is specified by the manufacturer of the machine. This is the x\zero point for the coordinate systems and reference points in the machine. The machine zero point can be the center of the table or a point along the edge of the traverse range as shown in figure the position of the machine zero point generally varies from

manufacture. The precise position of the machine zero point as well as the axis direction must therefore be taken from the operating instructions provided for each individual machine.

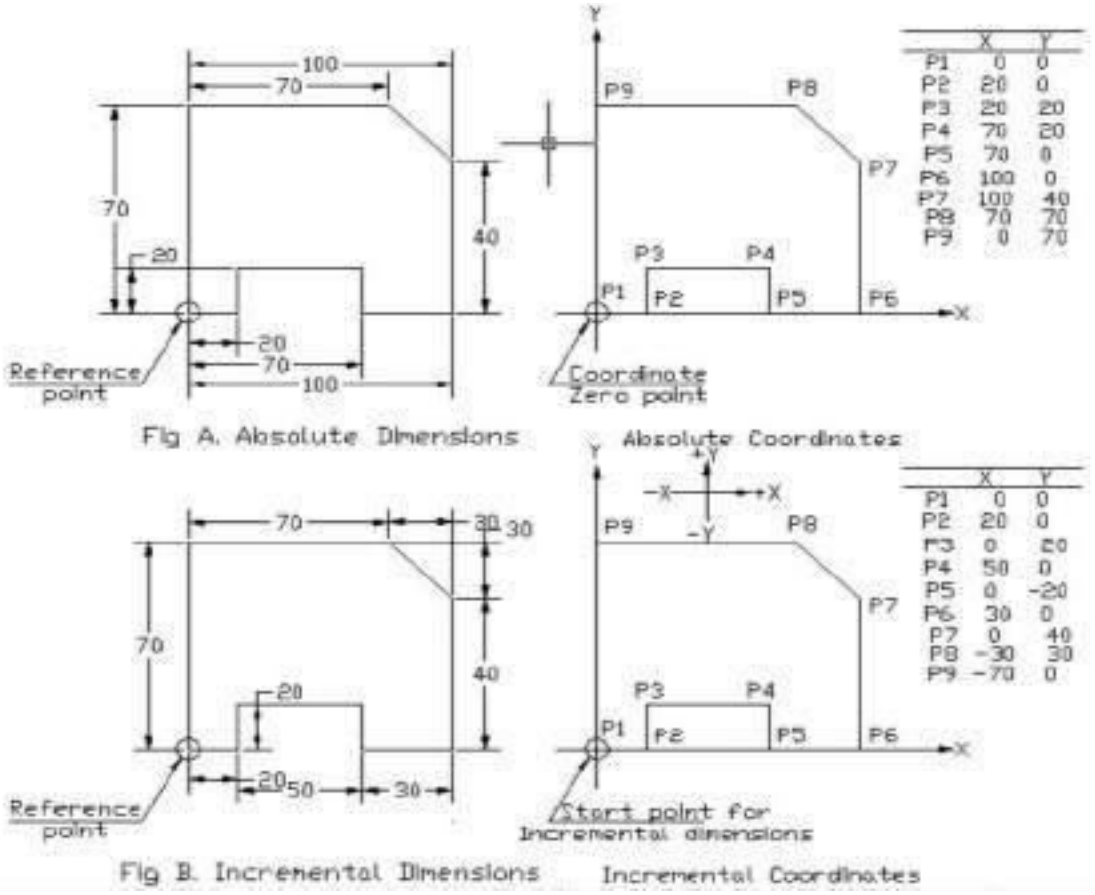
REFERENCE POINT (R): this point serves for calibrating and for controlling the measuring system of the slides as tool traverses. The position of the reference point is accurately predetermined in every traverse axis by the trip dogs and limit switches. Therefore, the reference point coordinates always 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. If current slide and tool position data should be lost in the control systems, for example, through an electrical failure, the machine must again be positioned to the reference point to re-establish the proper positioning values.

WORKPIECE ZERO POINT (W): This point determines the work piece coordinate system in relation to the machine zero point. The work piece zero point is chosen by the programmer and input into the CNC system when setting up the machine. The position of the work piece zero point can be freely chosen by the programmer within the work piece envelope of the machine. It is however, advisable to place the work piece zero point in such a manner that the dimensions in the work piece drawing can be conveniently converted into coordinate values and orientation when clamping/ chucking, setting up and checking the traverse measuring system can be affected easily. For milled parts, it is generally advisable to use an extreme corner point as the "work piece zero point". Occasionally, the work piece zero point is called the "program zero point"

Dept. of ME,

NC- RELATED DIMENSIONING

Dimensional information in a work piece drawing can be stated in two ways:



1. **Absolute Dimension System:** Data in absolute dimension system always refer to a fixed reference point in the drawing as shown in figure A above. This point has the function of a coordinate zero point as in figure B. The dimension lines run parallel to the coordinate axes and always start at the reference point. Absolute dimensions are also called as "Reference dimensions".

2. **Incremental Dimension System:** When using incremental dimension system, every measurement refers to a previously dimensioned position as shown in figure A below. 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 from point to point as shown in figurer B. Incremental dimensions are also frequently called "Relative dimensions" or "Chain dimensions".

PREPARATORY FUNCTIONS (G CODES)

G CODES	
G00	Positioning (Rapid Transverse)
G01	Linear Interpolation (Feed)
G02	Circular Interpolation (CW)
G03	Circular Interpolation (CCW)
G04	Dwell
G20	Inch Data Input
G21	Metric Data Input
G28	Reference point return
G40	Tool nose radius compensation cancel
G41	Tool nose radius compensation left
G42	Tool nose radius compensation right
G43	Tool length compensation + direction
G44	Tool length compensation - direction
G73	Peck drilling cycle
G74	Counter tapping cycle
G76	Fine Boring
G80	Canned cycle cancel
G81	Drilling cycle, spot boring
G82	Drilling cycle, counter boring
G83	Peck drilling cycle
G84	Tapping cycle
G85	Boring cycle
G86	Boring cycle
G87	Back boring cycle
G88	Boring cycle

Dept. of ME,

G89	Boring cycle
G90	Absolute command
G91	Incremental command
G92	Programming of Absolute zero point.
G94	Feed per minute
G95	Feed per revolution
G98	Return to initial point in canned cycle
G99	Return to R point in canned cycle.

MISCELLANEOUS AND PREPARATORY FUNCTIONS

M Codes are instructions describing machine functions such as calling the tool, spindle rotation, coolant on, door close/open etc.

M CODES	
M00	Program stop
M01	Optional stop

M02	Program end
M03	Spindle forward
M04	Spindle reverse
M05	Spindle stop
M06	Tool change
M08	Coolant on
M09	Coolant off
M10	Vice open
M11	Vice close
M13	Coolant, spindle fwd
M14	Coolant, spindle rev
M30	Program stop and rewind

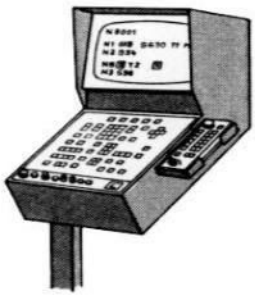
M70	X mirror On
M71	Y mirror On
M80	X mirror off
M81	Y mirror off
M98	Subprogram call
M99	Subprogram exit

From the drawing to the workpiece

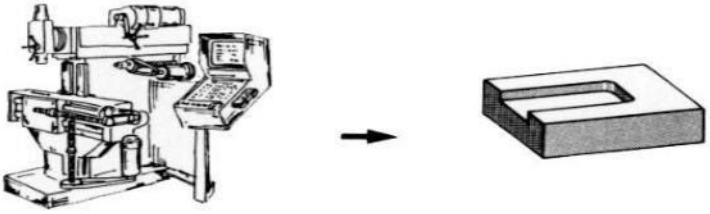
1. Reading drawing

2. Programming

3. Inputting program



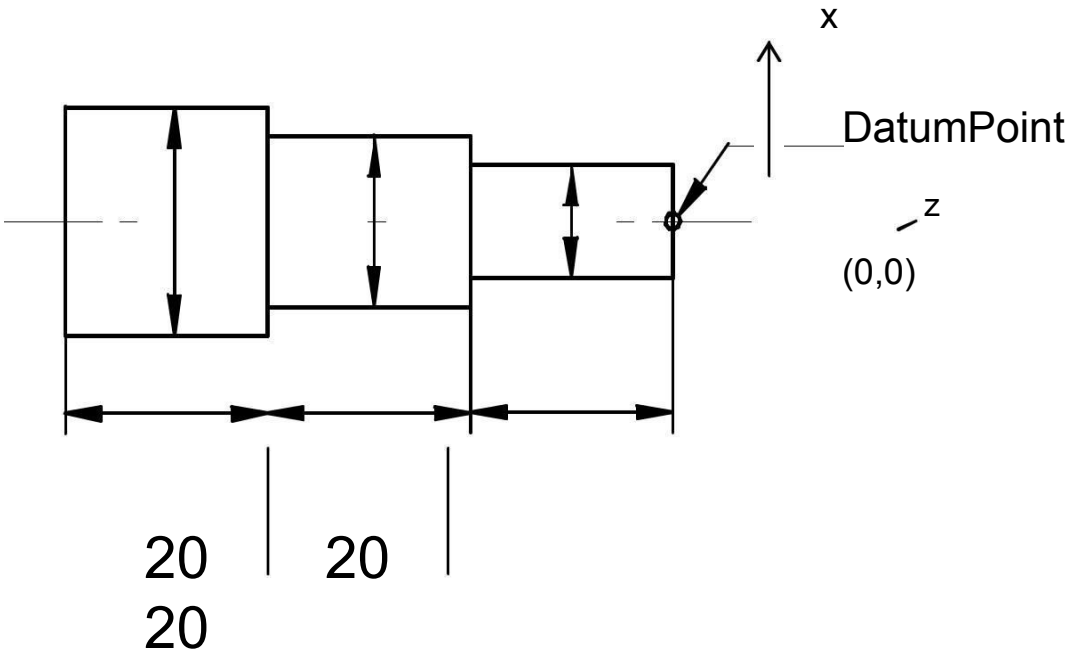
4. Manufacturing



Dept. of ME,

CNC TURNING

1. Write a manual part program for Linear Interpolation for the given part and execute.



Note: All dimensions are in mm only

CNC Part Program:

O1011

G21 G98

G28 X0 Z0

M06 T0101

M03 S100

G00 X10 Z1 M08

G00 X7.5

C.I.T	Department of ME	
Particulars	**Max. Marks**	**Marks Obtained**
Preparation	05	
Performance	10	
Viva -Voce	05	
Result/Output	05	
Total	**25**	
Student Signature with date	**Staff Signature with date**	

G01 Z-40 F0.2

G00 X10 Z1

G00 X5

G01 Z-20 F0.2

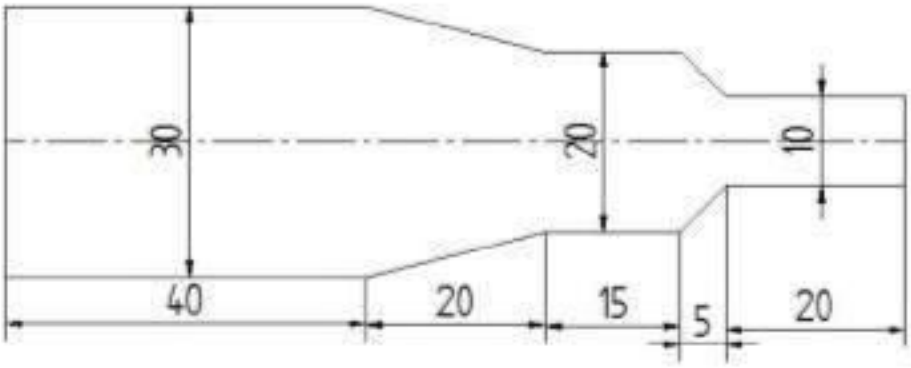
G00 X10 Z1

G28 X0 Z0

M05 M09

M30

2. Write a manual part program for Taper turning for the given part and execute.



Note: All dimensions are in mm only

CNC Part Program:

O1012

G21 G98

G28 X0 Z0

M06 T0101

M03 S100

G00 X15 Z1 M08

G00 X10

G01 Z-40 F0.2

G00 X15 Z1

G00 X5

G01 Z-20 F0.2

G00 X15 Z1

G00 X10 Z-40

C.I.T	Department of ME	
Particulars	**Max. Marks**	**Marks Obtained**
Preparation	05	
Performance	10	
Viva -Voce	05	
Result/Output	05	
Total	**25**	
Student Signature with date	**Staff Signature with date**	

G01 X15 Z-60 F0.2

G00 X15 Z1

G00 X5 Z-20

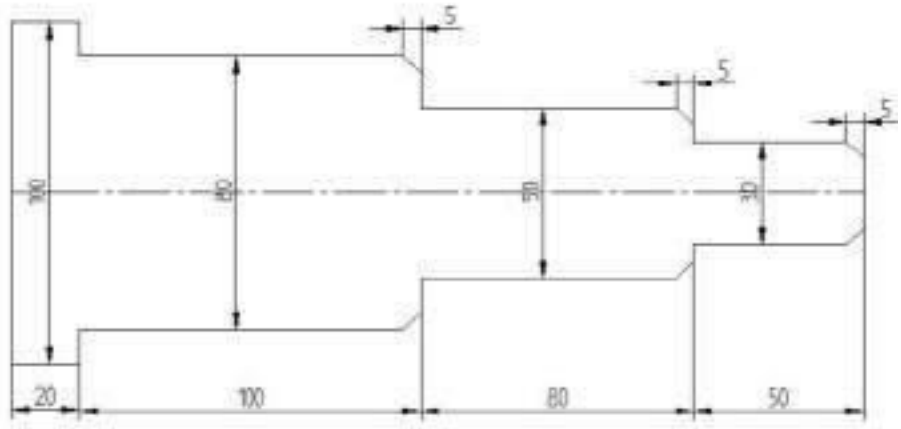
G01 X10 Z-25 F0.2

G00 X15 Z1G28 X0 Z0

M05 M09

M30

3. Write a manual part program on Chamfering& Step turning for the given part and execute.



Note: All dimensions are in mm only

CNC Part Program:

O1013
G21 G98
G28 X0 Z0
M06 T0101
M03 S100

G00 X50 Z1 M08

G00 X40
G01 Z-230 F0.2
G00 X50 Z1
G00 X25
G01 Z-130 F0.2
G00 X50 Z1
G00 X15
G01 Z-50 F0.2
G00 X50 Z1
G00 X40 Z-135
G01 X35 Z-130 F0.2
G00 X50 Z1
G00 X25 Z-55
G01 X20 Z-50 F0.2
G00 X50 Z1
G00 X15 Z-5
G01 X10 Z0 F0.2

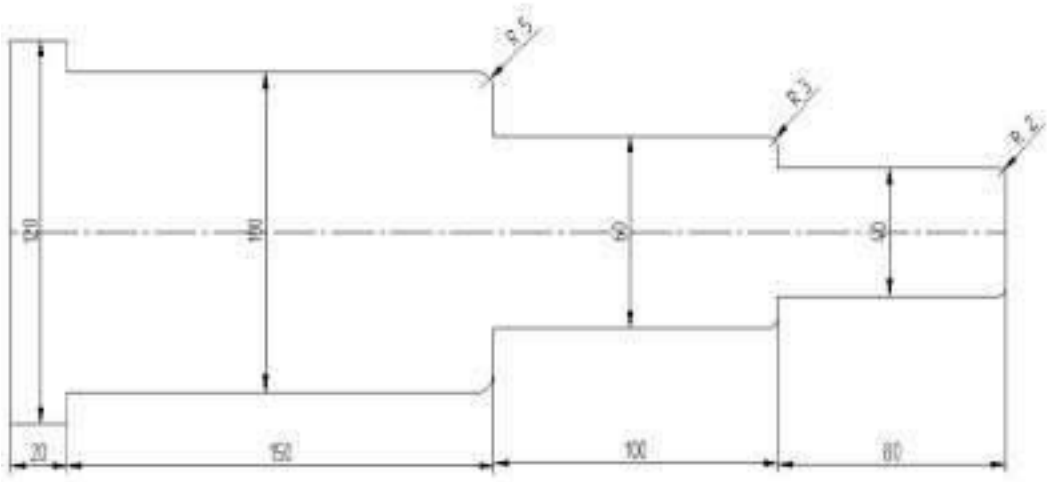
C.I.T	Department of ME	
Particulars	**Max. Marks**	**Marks Obtained**
Preparation	05	
Performance	10	
Viva -Voce	05	
Result/Output	05	
Total	**25**	
Student Signature with date	**Staff Signature with date**	

G00 X50 Z1
G28 X0 Z0
M05 M09

M30

---

4. Write a manual part program on fillet & Step turning for the given part.



Note: All dimensions are in mm only

CNC Part Program:

O1014
G21 G98
G28 X0 Z0
M06 T0101
M03 S100

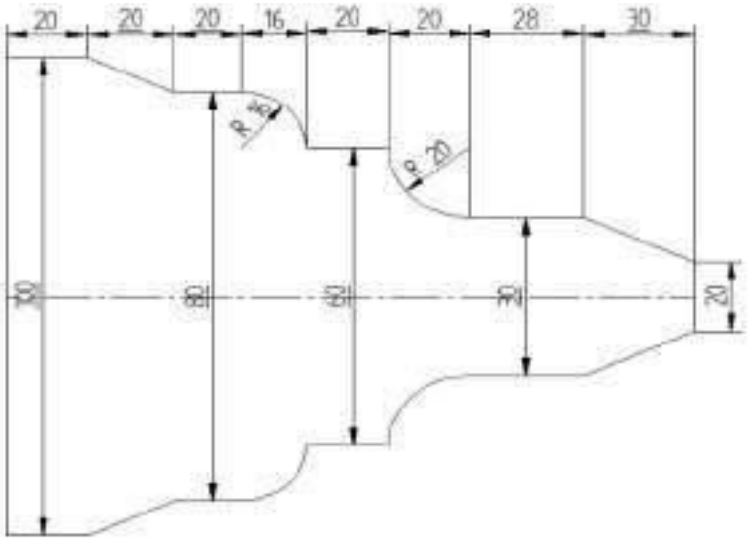
G00 X60 Z1 M08
G00 X50
G01 Z-330 F0.2
G00 X60 Z1

G00 X30
G01 Z-180 F0.2
G00 X60 Z1
G00 X20
G01 Z-80 F0.2
G00 X60 Z1
G00 X50 Z-185
G02 X45 Z-180 R5
G00 X60 Z1
G00 X30 Z-83
G02 X27 Z-80 R3
G00 X60 Z1
G00 X20 Z-2

C.I.T	Department of ME	
Particulars	**Max. Marks**	**Marks Obtained**
Preparation	05	
Performance	10	
Viva -Voce	05	
Result/Output	05	
Total	**25**	
Student Signature with date	**Staff Signature with date**	

G02 X18 Z0 R2
G00 X60 Z1
G28 X0 Z0
M05 M09
M30

5. Write a manual part program for the given profile and execute.



Note: All dimensions are in mm only

CNC Part Program:

O1015

G21 G98

G28 X0 Z0
M06 T0101
M03 S100
G00 X50 Z1 M08
G00 X40

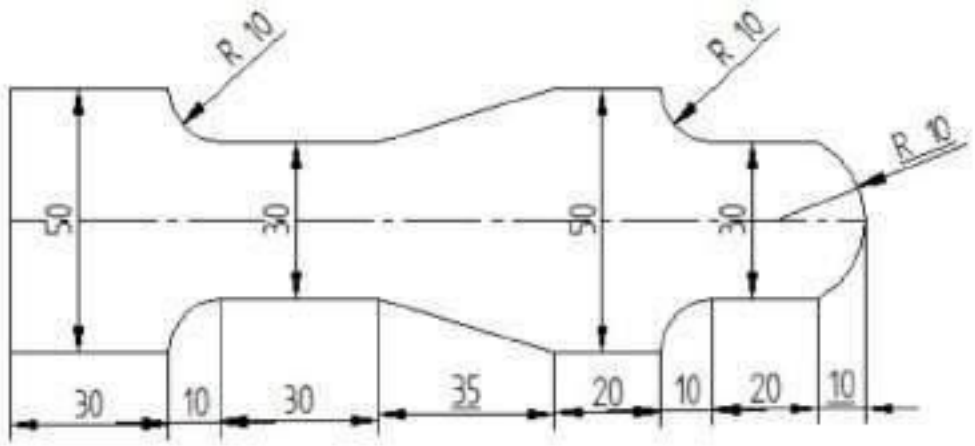
G01 Z-134 F0.2
G00 X50 Z1
G00 X30
G01 Z-98 F0.2
G00 X50 Z1
G00 X15
G01 Z-58 F0.2
G00 X50 Z1
G00 X40 Z-134
G01 X50 Z-154 F0.2
G00 X50 Z1
G00 X30 Z-98
G03 X40 Z-114 R16
G00 X10

G01 X15 Z-30
F0.2
G00 X50 Z1
G28 X0 Z0
M05 M09
M30

C.I.T	Department of ME	
Particulars	**Max. Marks**	**Marks Obtained**
Preparation	05	
Performance	10	
Viva -Voce	05	
Result/Output	05	
Total	**25**	
Student Signature with date	**Staff Signature with date**	

G00 X50 Z1
G00 X15 Z-58
G02 X30 Z-78 R20
G00 X50 Z1

6. Write a manual part program for the given profile and execute.



Note: All dimensions are in mm only

CNC Part Program:

O1016

G21 G98
G28 X0 Z0
M06 T0101
M03 S100

G00 X25 Z1 M08
G00 X15
G01 Z-30 F0.2
G00 X25 Z1

G00 X25 Z-60
G01 X15 Z-95 F0.2
G00 X25 Z-95
G00 X25 Z1
G00 X25 Z-95
G00 X15 Z-95
G01 X15 Z-125 F0.2
G00 X25 Z-125
G00 X25 Z1
G00 X25 Z-125
G00 X15 Z-125
G02 X25 Z-135 R10
G00 X25 Z-135
G00 X25 Z1

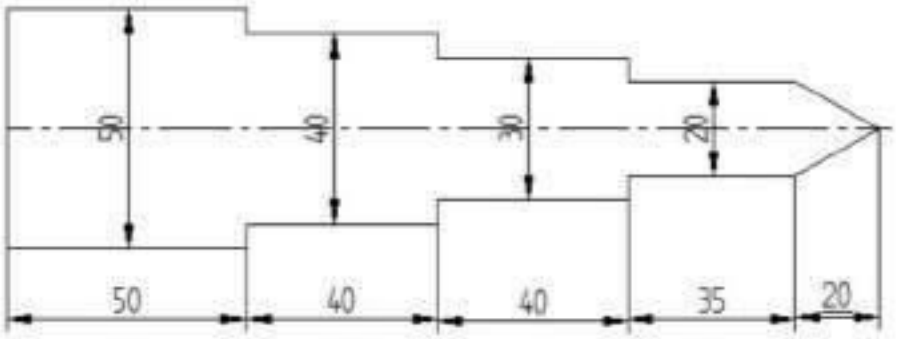
G02 X0 Z0 R10

G00 X25 Z1
G28 X0 Z0
M05 M09
M30

C.I.T	Department of ME	
Particulars	**Max. Marks**	**Marks Obtained**
Preparation	05	
Performance	10	
Viva -Voce	05	
Result/Output	05	
Total	**25**	
Student Signature with date	**Staff Signature with date**	

G00 X15 Z-30
G02 X25 Z-40 R10
G00 X25 Z1
G00 X15 Z-10

7. Write a manual part program for the given profile and execute.



Note: All dimensions are in mm only

CNC Part Program:

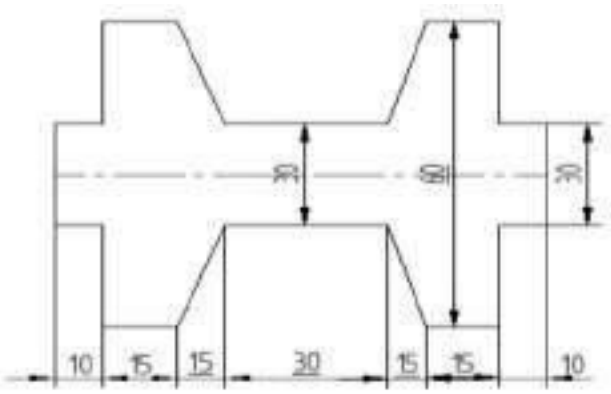
```
O1017
G28 X0 Z0
M06 T0101
M03 S100
G00 X25 Z1 M08

G00 X20

G01 Z-135 F0.2
G00 X25 Z1
G00 X15
G01 Z-95 F0.2
G00 X25 Z1
G00 X10
G01 Z-55 F0.2
G00 X25 Z1
G00 X10 Z-20
G01 X0 Z0 F0.2
G00 X25 Z1
G28 X0 Z0
M05 M09
M30
```

C.I.T	Department of ME	
Particulars	**Max. Marks**	**Marks Obtained**
Preparation	05	
Performance	10	
Viva -Voce	05	
Result/Output	05	
Total	**25**	
Student Signature with date	**Staff Signature with date**	

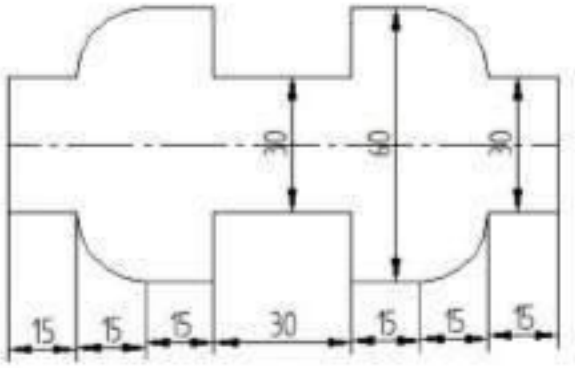
Dept. of ME,

PRACTICE PROGRAMS [CNC TURNING]

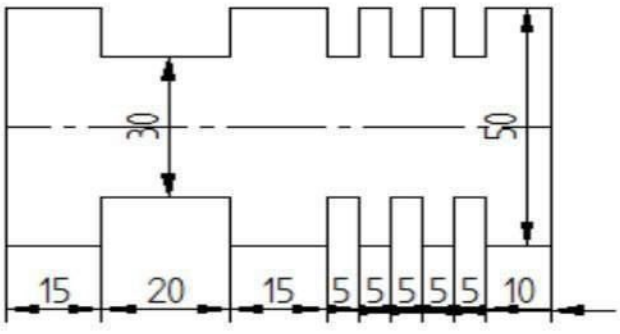
8.



9.



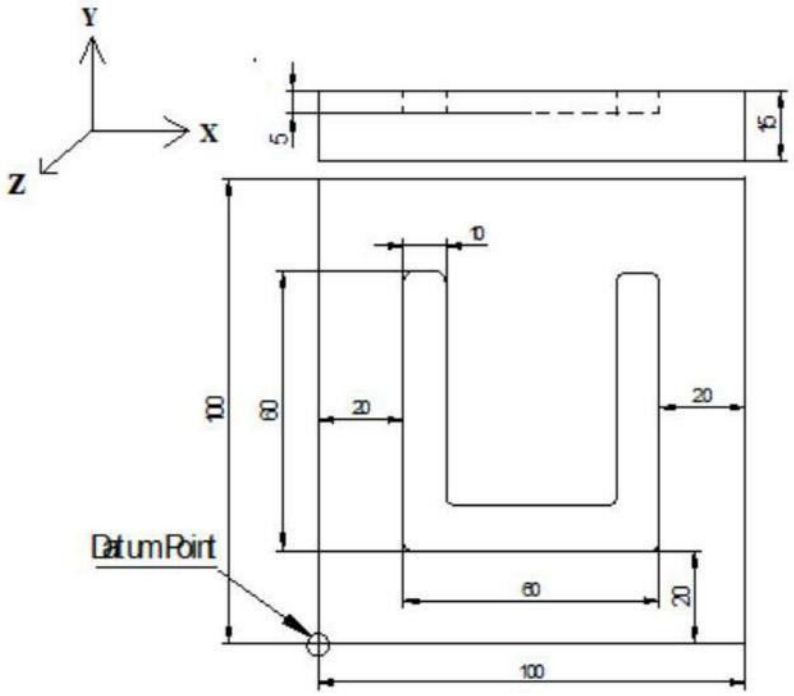
10.



Note: All dimensions are in mm only

CNC MILLING

1) Write a manual part program for Slotting operation for the component as shown in drawing and execute.



Material – Mild Steel,

Operation - Slotting

Billet size – 100 x 100 x 15mm

Tool – Radiused end mill Φ 10 mm

Note: All dimensions are in mm only

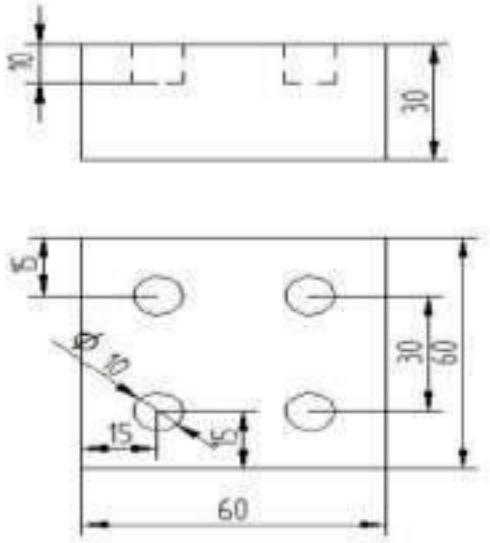
CNC Part Program:

O1021
G21 G98
G28 X0 Y0 Z0

C.I.T	Department of ME	
Particulars	Max. Marks	Marks Obtained
Preparation	05	
Performance	10	

```
M06 T0101
M03 S100
G00 X25 Y75 Z5 M08
G01 Z-5 F0.2
Y25
X75
Y75
G00 Z5
G28 X0 Y0 Z10
M05 M09
M30
```

2) Write a manual part program for Drillingoperationfor the component as shown in drawing.



Note: All dimensions are in mm only

Material – Mild Steel, Operation - Drilling

Billet size – 60 x 60 x 30mm Tool – Slot drill Φ 10 mm

CNC Part Program:

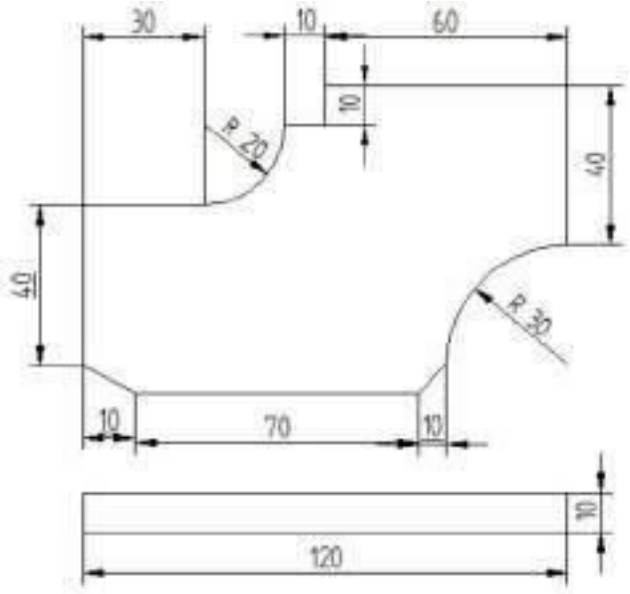
O1022
G21 G98
G28 X0 Y0 Z0
M06 T0101
M03 S100
G00 X15 Y15 Z5 M08
G01 Z-10 F0.2
G00 Z5
G00 X15 Y45
G01 Z-10 F0.2
G00 Z5
G00 X45 Y45
G01 Z-10 F0.2
G00 Z5
G00 X45 Y15
G01 Z-10 F0.2
G00 Z5

C.I.T	Department of ME	
Particulars	**Max. Marks**	**Marks Obtained**
Preparation	05	
Performance	10	
Viva -Voce	05	
Result/Output	05	
Total	**25**	
Student Signature with date	**Staff Signature with date**	

G28 X0 Y0 Z0
M05 M09
M30

Dept. of ME,

3) Write a manual part program for the profile as shown in the drawing and execute.



Note: All dimensions are in mm only

Material – Mild Steel, Operation – Milling

Billet size – 120 x 80 x 10mm Tool – Radiused end mill Φ 5mm

CNC Part Program:

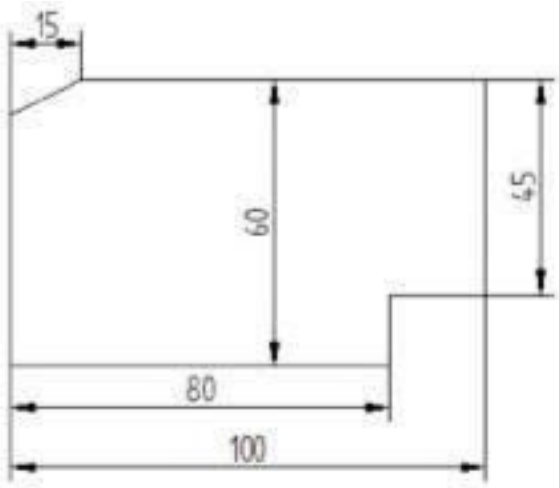
O1023
G21 G98
G28 X0 Y0 Z0

M06 T0101
M03 S100
G00 X0 Y10 Z5 M08
G01 Z-10 F60
X10 Y0
X80
X90 Y10
G02 X120 Y40 Z-10 R30
G01 X120 Y80 Z-10 F60
X60 Y80
X60 Y70
X50 Y70
G02 X30 Y60 Z-10 R20
G01 X0 Y60 Z-10 F60

C.I.T	Department of ME	
Particulars	**Max. Marks**	**Marks Obtained**
Preparation	05	
Performance	10	
Viva -Voce	05	
Result/Output	05	
Total	**25**	
Student Signature with date	**Staff Signature with date**	

X0 Y10
G28 X0 Y0 Z0
M05 M09
M30

4) Write a manual part program for the profile as shown in the drawing and execute.



Note: All dimensions are in mm only

Material – Mild Steel, Operation – Milling

Billet size – 100 x 60 x 15mm Tool – Radiused end mill Φ 10mm

CNC Part Program:

O1024

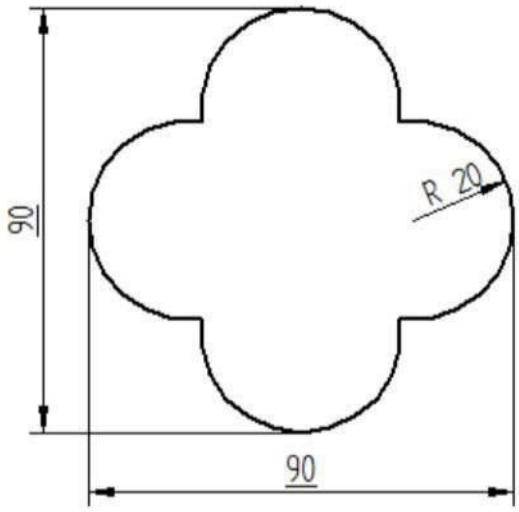
G21 G98
G28 X0 Y0 Z0
G41 M06 T0101
M03 S600
G00 X0Y0 Z5 M08
G01 Z-10 F0.2
G01 X0 Y45
G01 X15 Y60
G01 X100
G01 Y15
G01 X80
G01 Y0
G01 X0
G0 Z5
G28 X0 Y0 Z0
M05M09

C.I.T	Department of ME	
Particulars	**Max. Marks**	**Marks Obtained**
Preparation	05	
Performance	10	
Viva -Voce	05	
Result/Output	05	
Total	**25**	
Student Signature with date	**Staff Signature with date**	

M30

Dept. of ME,

5) Write a manual part program for the profile as shown in the drawing and execute.



Note: All dimensions are in mm only

Material – Mild Steel, Operation – Milling

Billet size – 100 x 60 x 15mm Tool – Radiused end mill Φ 10mm

CNC Part Program:

O1025

G21 G98

G28 X0 Y0 Z0

G42 M06 T0101

M03 S100

G00 X20 Y20 M08

G01 Z-10 F0.2

G03 X60 Y20 R20

G03 X60 Y60 R20

C.I.T	Department of ME	
Particulars	**Max. Marks**	**Marks Obtained**
Preparation	05	
Performance	10	
Viva -Voce	05	
Result/Output	05	
Total	**25**	
Student Signature with date	**Staff Signature with date**	

G03 X20 Y60 R20

G03 X20 Y20 R20

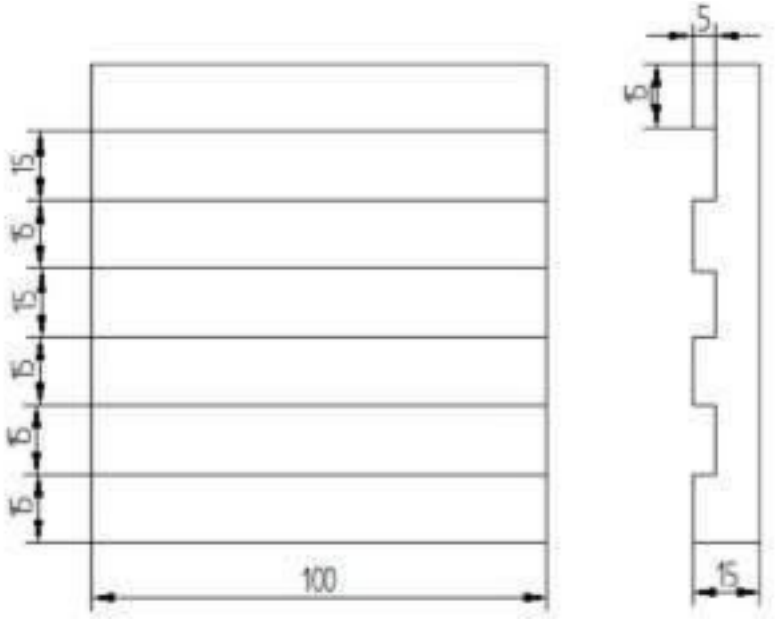
G00 Z5

G28 X0 Y0 Z0

M05 M09

M30

6) Write a manual part program for the profile as shown in the drawing and execute.



Note: All dimensions are in mm only

Material – Mild Steel, Operation – Slotting

Billet size – 100 x 105 x 15mm Tool – slot drill Φ 15mm

CNC Part Program:

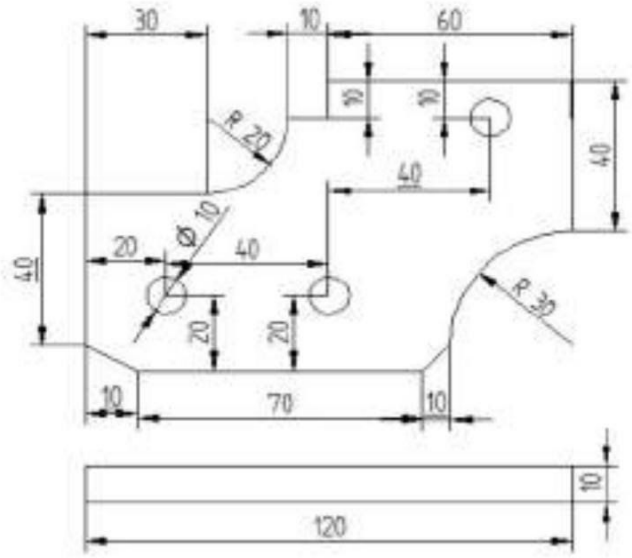
O1027
G21 G98
G28 X0 Y0 Z0
M06 T0101
M03 S100
G00 X-8 Y22.5 M08
G01 Z-5 F0.2
G01 X108 F0.2
G00 Y52.5
G01 X-8 F0.2
G00 Y82.5
G01 X108 F0.2
G00 Z10

C.I.T	Department of ME	
Particulars	**Max. Marks**	**Marks Obtained**
Preparation	05	
Performance	10	
Viva -Voce	05	
Result/Output	05	
Total	**25**	
Student Signature with date	**Staff Signature with date**	

G28 X0 Y0 Z10

M05 M09
M30

7) Write a manual part program for the profile as shown in the drawing and execute.



Material – Mild Steel,

Operation – milling & drilling

Billet size – 120 x 80 x 10mm

Tool – slot drill Φ 10mm

Note: All dimensions are in mm only

CNC Part Program:

O1027

G21 G98
G28 X0 Y0 Z0
M06 T0101
M03 S600
G00 X0 Y10 Z5 M08
G01 Z-10 F60

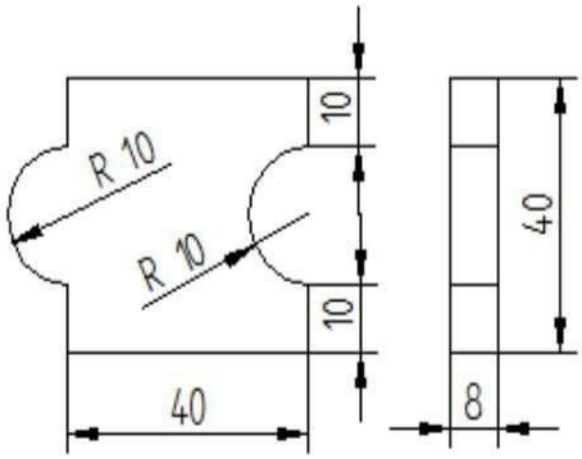
X10 Y0
X80
X90 Y10
G00 X105 Y70
Z5
G01 Z-10 F60
G00 Z5
G28 X0 Y0 Z0
M05 M09
M30

C.I.T	Department of ME	
Particulars	Max. Marks	Marks Obtained
Preparation	05	

```
G02 X120 Y40 Z-10 R30
G01 X120 Y80 Z-10 F60
X60 Y80
X60 Y70
X50 Y70
G02 X30 Y60 Z-10 R20
G01 X0 Y60 Z-10 F60
X0 Y10
G00 X0 Y0 Z5
G00 X25 Y20 Z5
G01 Z-10 F60
G00 Z5
G00 X65 Y20 Z5
G01 Z-10 F60
G00 Z5
```

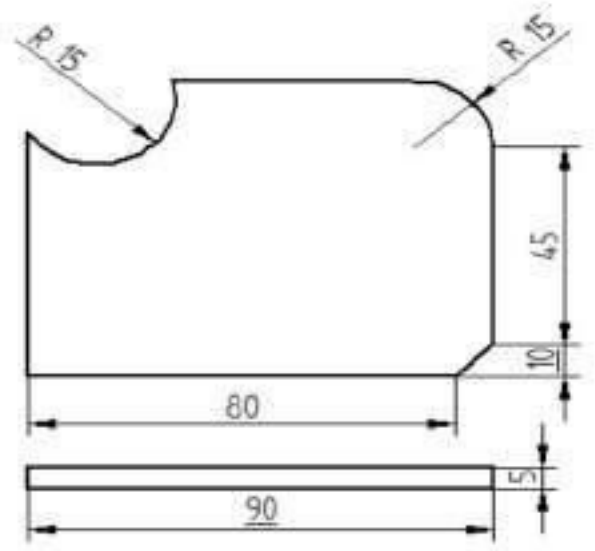
PRACTICE PROGRAMS [CNC MILLING]

8.



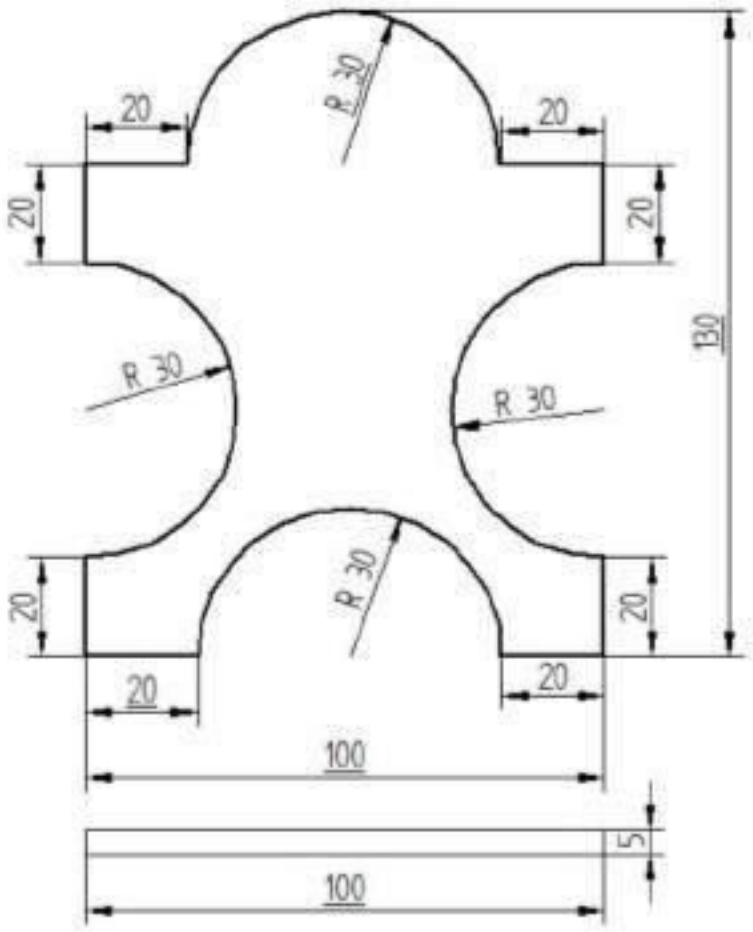
Note: All dimensions are in mm only

9.



Note: All dimensions are in mm only

10.



Note: All dimensions are in mm only

Dept. of ME,

TO GENERATE THE PROGRAM

8 steps in CAPSTURN/CAPSMILL NC programming

1. Start new program
2. Define work setup

} Geometry

3. Draw the part

4. Draw the blank

Machining

5. Perform machining →

6. Select machine

7. View tool path

} →

Tool path

8. Generate NC program

1. Start new program

Double click on the CAPSTURN icon

Or

Select start- program –CADEM –CAPSTURN

2. Define work setup

Setup data is required for machining, and documentation is related to the details of the program. The work setup data is divided into

Setup data 1,

Setup data 2 and

Documentation.

Entering the setup data I mandatory, while documentation is optional.

Dept. of ME,

3. Draw the part

Draw-use the drawing tools to construct the geometry of the part

Draw-define part – create part shape

4. Draw the blank

Draw –define blank

5. Perform machining

Switch to the machining menu clicking on the machining tab

Select appropriate machining operation and define tool details used for that operation

6. Select machine

Select suitable machine from the available list

7. View tool path.

Switch to tool path mode by clicking on tool path tab

Select tool path-start

8. Generate NC program

Click on NC PROGRAM ON THE menu bar