QMP 7.1 D/F

Channabasaveshwara Institute of Technology

(Affiliated to VTU, Belagavi & Approved by AICTE, New Delhi)

(**NAAC Accredited & ISO 9001:2015 Certified Institution**)

NH - 206, (B.H ROAD), GUBBI, TUMAKURU – 572 216, Karnataka

Department of Mechanical Engineering

CNC Programming & 3D Printing Lab

[21MEL66]

VI Semester

LAB MANUAL

2023-24 (Even)

Name:____

USN:____

Batch:____________________Section:_____________

QMP 7.1 D/F

Channabasaveshwara Institute of Technology

(Affiliated to VTU, Belagavi & Approved by AICTE, New Delhi)

(**NAAC Accredited & ISO 9001:2015 Certified Institution)**

NH - 206, (B.H ROAD), GUBBI, TUMAKURU – 572 216, Karnataka

Department of Mechanical Engineering

CNC Programming & 3D Printing Lab

2023-24

Prepared by:

Mr. Natesh C P

Asst. Professor

Dept. of ME,

CIT, Gubbi, Tumakuru

Approved by:

Dr. Giridhar S Kulkarni

Professor & Head,

Dept. of ME,

CIT, Gubbi, Tumakuru

CONTENTS

College Vision

To create centers of excellence in education and to serve the society by enhancing the quality of life through value based professional leadership.

College Mission

- ❖ To provide high quality technical and professionally relevant education in a diverse learning environment.
- ❖ To provide the values that prepare students to lead their lives with personal integrity, professional ethics and civic responsibility in a global society.
- ❖ To prepare the next generation of skilled professionals to successfully compete in the diverse global market.
- ❖ To promote a campus environment that welcomes and honors women and men of all races, creeds and cultures, values and intellectual curiosity, pursuit of knowledge and academic integrity and freedom.
- ❖ To offer a wide variety of off-campus education and training programmes to individuals and groups.
- ❖ To stimulate collaborative efforts with industry, universities, government and professional societies.
- ❖ To facilitate public understanding of technical issues and achieve excellence in the operations of the institute.

Department Vision

To create state of the art learning environment to nurture the learning, blending human values, academic professionalism and research process in the field of mechanical engineering for the betterment of society.

Department Mission

The mission of the department is to,

- ❖ Provide requisite foundation to our students in Mechanical Engineering.
- ❖ Provide cutting edge laboratory resources to bridge the gap between theoretical and practical concepts.
- ❖ Provide exposure to various mechanical industries through periodic industrial visits.
- ❖ Enhance our student's skill set and to make them industry ready by systematic skill development program.

SYLLABUS

CNC PROGRAMMING AND 3-D PRINTING LAB			
Course Code	21MEL66	CIE Marks	50
Teaching Hours/Week (L:T:P: S)	0:0:2*:0	SEE Marks	50
Credits	01	Exam Hours	03

*** Additional one hour may be considered for Instructions if required**

Course objectives:

- To expose the students to the techniques of CNC programming and cutting tool path generation through CNC simulation software by using G-Codes and M-codes.
- To educate the students on the usage of CAM packages.
- To expose the students on the usage of 3D Printing Technology
- To make the students understand the importance of automation in industries through exposure to FMS, Robotics, and Hydraulics and Pneumatics.

Sl.NO	Experiments
1	Manual CNC part programming using ISO Format G/M codes for 2 turning and 2 milling parts. Selection and assignment of tools, correction of syntax and logical errors, and verification of tool path using CNC program verification software.
2	CNC part programming using CAM packages : Simulation of Turning simulations to be carried out using simulation packages like: CademCAMLab-Pro, Master-CAM.
3	CNC part programming using CAM packages : Simulation of Drilling simulations to be carried out using simulation packages like: CademCAMLab-Pro, Master-CAM.
4	CNC part programming using CAM packages : Simulation of Milling simulations to be carried out using simulation packages like: CademCAMLab-Pro, Master-CAM.
5	Internal and external threading : Write a CNC program to create internal and external threading on a cylindrical block.s
6	Simple 3D Printing Model : Creating Simple 3D model (example cube, gear, prism etc) in CAD software and printing the model using any 3D Printer (FDM/SLA/SLS printer)
7	Assembly Model-1: Creating an 3D CAD model of NUT and Bolt (example size M12x50), print the model using any 3D Printer and Check the assembly
8	Assembly Model-2: Creating an 3D CAD assembly model containing four or more parts (example Screw jack, plumber block etc) print the model using any 3D Printer and Check the assembly
	Demonstration Experiments (For CIE)
9	Robot programming: Using Teach Pendent & Offline programming to perform pick and place, stacking of objects (2 programs).
10	Pneumatics and Hydraulics, Electro-Pneumatics: 3 typical experiments on Basics of these topics to be conducted.
11	FMS (Flexible Manufacturing System): Programming of Automatic storage and Retrieval system (ASRS) and linear shuttle conveyor Interfacing CNC lathe, milling with loading unloading arm and ASRS to be carried out on simple components.
12	Simple strength testing of 3D Printed Parts

CIE: 50 Marks
SEE: 50 Marks

Total: 100 Marks

COURSE OBJECTIVES:

- **CLO1:** To expose the students to the techniques of CNC programming and cutting tool path generation through CNC simulation software by using G-Codes and M-codes.
- **CLO2:** To educate the students on the usage of CAM packages.
- **CLO3:** To make the students understand the importance of automation in industries through exposure to FMS, Robotics, and Hydraulics and Pneumatics.

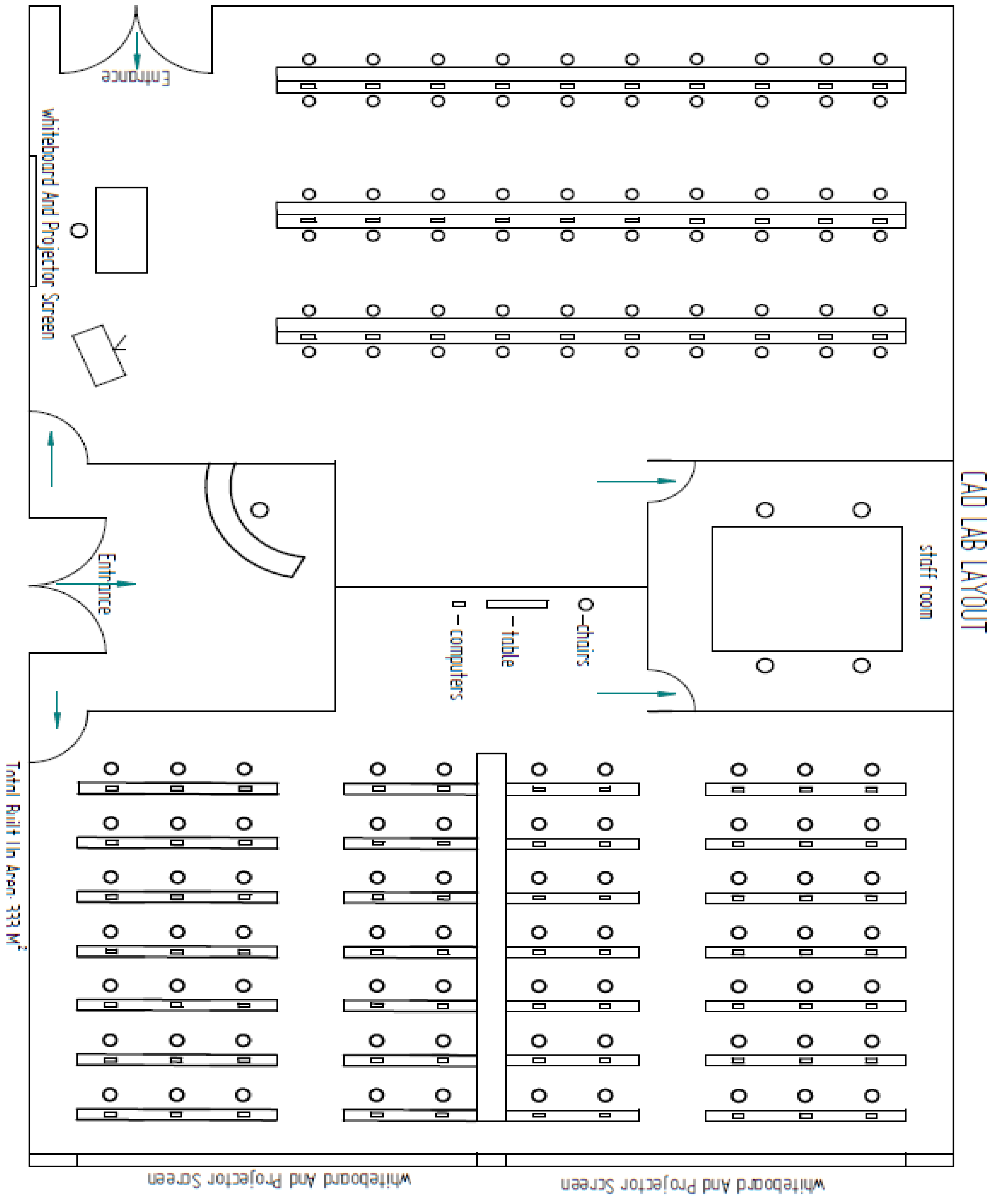
COURSE OUTCOMES:

- **CO1:** Generate CNC Lathe and Mill part programs involving different operations using different motion control systems with the help of CAM Packages.
- **CO2:** Simulate Tool Path for different Machining operations on small components using CAM Packages for CNC Lathe & CNC Milling Machines.
- **CO3:** Understand programming of Robots; understand the operating principles of hydraulics, pneumatics and electro pneumatic systems. Apply this knowledge to automate & improve efficiency of manufacturing.

General Instructions to Students

- ✓ Students are informed to present 5 min before the commencement of lab.
- ✓ Students must enter their name in daily book before entering into lab.
- ✓ Students must leave Foot wares before entering lab.
- ✓ Students must not carry any valuable things inside the lab.
- ✓ Students must inform lab assistant before He/She uses any computer.
- ✓ Do not touch anything with which you are not completely familiar. Carelessness may not only break the valuable equipment in the lab but may also cause serious injury to you and others in the lab.
- ✓ For any software/hardware/ Electrical failure of computer during working, report it immediately to your supervisor. Never try to fix the problem yourself because you could further damage the equipment and harm yourself and others in the lab.
- ✓ Students must submit Record book for evaluation before the commencement of lab.
- ✓ Students must keep observation book (if necessary).
- ✓ Students must keep silent near lab premises.
- ✓ Students are informed to follow safety rules.
- ✓ Students must obey lab rules and regulations.
- ✓ Students must maintain discipline in lab.
- ✓ Do not crowd around the computers and run inside the laboratory.
- ✓ Please follow instructions precisely as instructed by your supervisor.
- ✓ Do not start the experiment unless your setup is verified & approved by your supervisor.

DEPARTMENT OF MECHANICAL ENGINEERING



Channabasaveshwara Institute of Technology
(Affiliated to VTU, Belagavi & Approved by AICTE, New Delhi)
(**NAAC Accredited & ISO 9001:2015 Certified Institution**)
NH - 206, (B.H ROAD), GUBBI, TUMAKURU – 572 216, Karnataka

LECTURE PLAN

Sem.: **VI** Section: **A1** Sub. & Sub. Code: **CNC Lab (21MEL66)**

Sl. No.	Date	Lesson Plan No.	Topic	Remarks
01		LP.1	Introduction to CNC Programming	
02		LP.2	Manual Part Programming and Simulation of Turning Part Programs: Ex.01 & Ex.02	
03		LP.3	Manual Part Programming and Simulation of Turning Part Programs: Ex.03 & Ex.04	
04		LP.4	Manual Part Programming and Simulation of Turning Part Programs: Ex.05 & Ex.06	
05		LP.5	Manual Part Programming and Simulation of Turning Part Programs: Practice Ex.07& Ex.08	
06		LP.6	Manual Part Programming and Simulation of Turning Part Programs: Practice Ex.09 & Ex.10 **Internal Assessment (IA 1)**	
07		LP.7	Manual Part Programming and Simulation of Milling Part Programs: Ex.01 & Ex.02	
08		LP.8	Manual Part Programming and Simulation of Milling Part Programs: Ex.03 & Ex.04	
09		LP.9	Manual Part Programming and Simulation of Milling Part Programs: Ex.05 & Ex.06	

10		LP.10	Manual Part Programming and Simulation of Milling Part Programs: Practice Ex.07& Ex.08	
11		LP.11	Manual Part Programming and Simulation of Milling Part Programs: Practice Ex.09 & Ex.10 **Internal Assessment (IA 2)**	
12		LP.12	FMS -Flexible Manufacturing System	
13		LP.13	Robot programming	
14		LP.14	Pneumatics and Hydraulics, Electro-Pneumatics.	
15		LP.15	3D Printing	

EXTRA LABS

Month	Date
April	
May	
June	
July	

Staff In-charge **HOD**

Channabasaveshwara Institute of Technology
(Affiliated to VTU, Belagavi & Approved by AICTE, New Delhi)
(**NAAC Accredited & ISO 9001:2015 Certified Institution**)
NH - 206, (B.H ROAD), GUBBI, TUMAKURU – 572 216, Karnataka

LECTURE PLAN

Sem.: **VI** Section: **A2** Sub. & Sub. Code: **CNC Lab (21MEL66)**

Sl. No.	Date	Lesson Plan No.	Topic	Remarks
01		LP.1	Introduction to CNC Programming	
02		LP.2	Manual Part Programming and Simulation of Turning Part Programs: Ex.01 & Ex.02	
03		LP.3	Manual Part Programming and Simulation of Turning Part Programs: Ex.03 & Ex.04	
04		LP.4	Manual Part Programming and Simulation of Turning Part Programs: Ex.05 & Ex.06	
05		LP.5	Manual Part Programming and Simulation of Turning Part Programs: Practice Ex.07& Ex.08	
06		LP.6	Manual Part Programming and Simulation of Turning Part Programs: Practice Ex.09 & Ex.10 **Internal Assessment (IA 1)**	
07		LP.7	Manual Part Programming and Simulation of Milling Part Programs: Ex.01 & Ex.02	
08		LP.8	Manual Part Programming and Simulation of Milling Part Programs: Ex.03 & Ex.04	
09		LP.9	Manual Part Programming and Simulation of Milling Part Programs: Ex.05 & Ex.06	

10		LP.10	Manual Part Programming and Simulation of Milling Part Programs: Practice Ex.07& Ex.08	
11		LP.11	Manual Part Programming and Simulation of Milling Part Programs: Practice Ex.09 & Ex.10 **Internal Assessment (IA 2)**	
12		LP.12	FMS -Flexible Manufacturing System	
13		LP.13	Robot programming	
14		LP.14	Pneumatics and Hydraulics, Electro-Pneumatics.	
15		LP.15	3D Printing	

EXTRA LABS

Month	Date
April	
May	
June	
July	

Staff In-charge **HOD**

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 figure shown below,

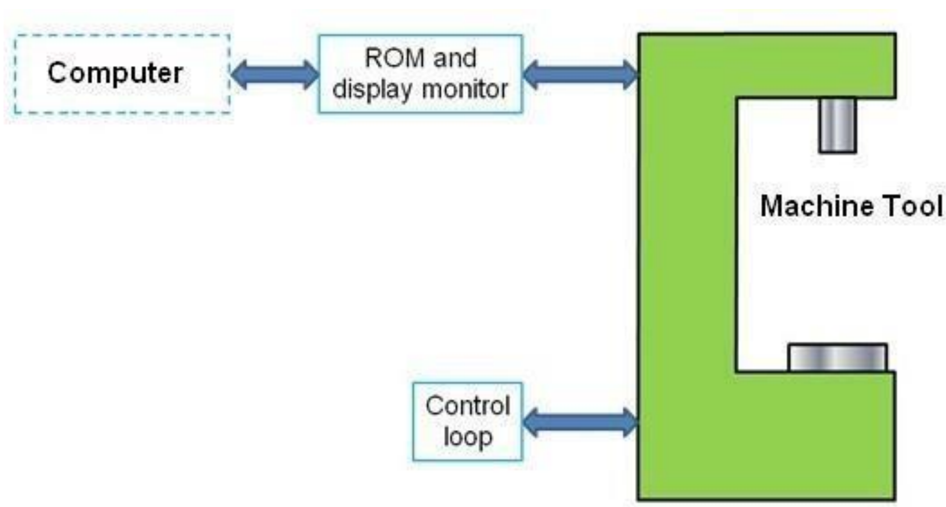


Fig. 01 CNC Machine Tool

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.

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.

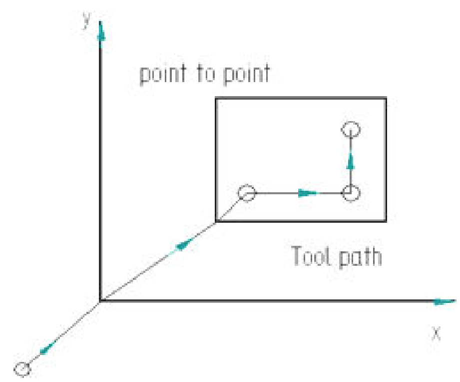


Fig. 02 Point to point

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.

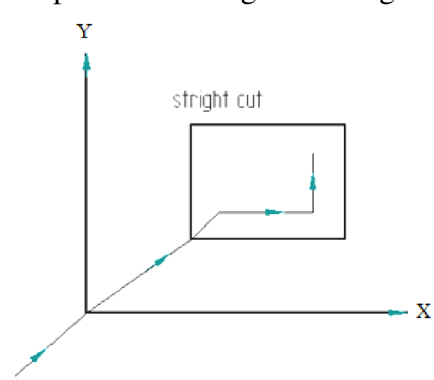


Fig. 03 Straight cut NC

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 paths (a simple sketch is sufficient for simple components)
• Select a component datum and carryout the necessary calculations at slopes and arcs.
• Prepare tooling layout sheet showing tools to be used in the program and indicate the station number for each tool.
• 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:

- Material to be cut
- Surface roughness tolerance
- 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.

Part Programming Geometry for Turning

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
- 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,

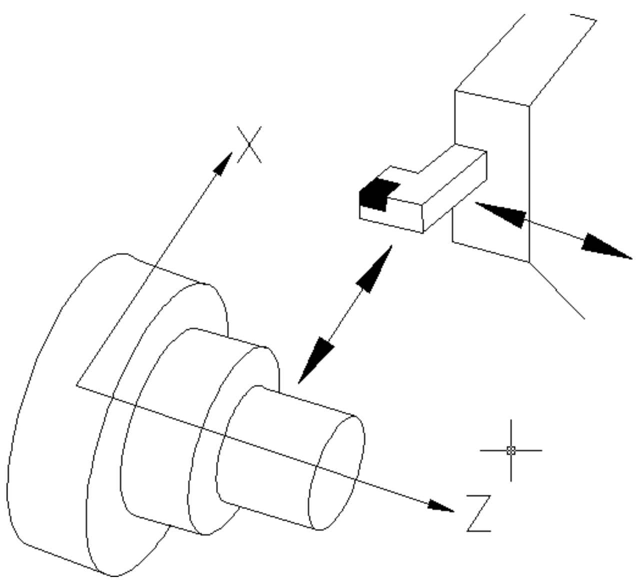


Fig. 04 Coordinate system for turning operations

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 centre of the spindle nose face. The main spindle axis (centre 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:

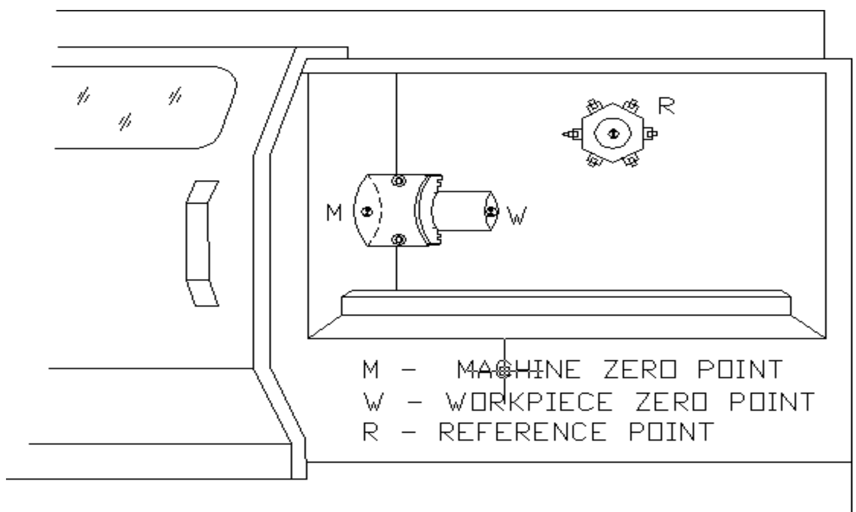


Fig. 05 CNC Machine Zero, Work piece Zero and Reference Point

Work piece 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 effected easily. For turned parts, the work piece zero point should be placed along the spindle axis (centre 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."

Reference 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 **G01** **G02** **G03**	Positioning (Rapid Transverse) Linear Interpolation (Feed) Circular Interpolation (CW) Circular Interpolation (CCW)	**G72** **G73** **G74** **G75** **G76**	Stock removal in facing Pattern repeating Peck drilling in Z axis Grooving in X axis Thread cutting cycle
G04	Dwell	**G90** **G94**	Cutting cycle A (Turning) Cutting cycle B (Facing)
G20 **G21**	Inch Data Input Metric Data Input	**G96** **G97**	Constant surface speed control Constant surface speed control cancel
G28	Reference point return	**G98**	Feed per minute
G40 **G41** **G42**	Tool nose radius compensation cancel Tool nose radius compensation left Tool nose radius compensation right	**G99**	Feed per revolution
G50	Works coordinate change/ Max. Spindle speed setting		
G70 **G71**	Finishing cycle Multiple Turning Cycle in turning		

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

Structure of CNC Program Block

A CNC program block is normally written as,

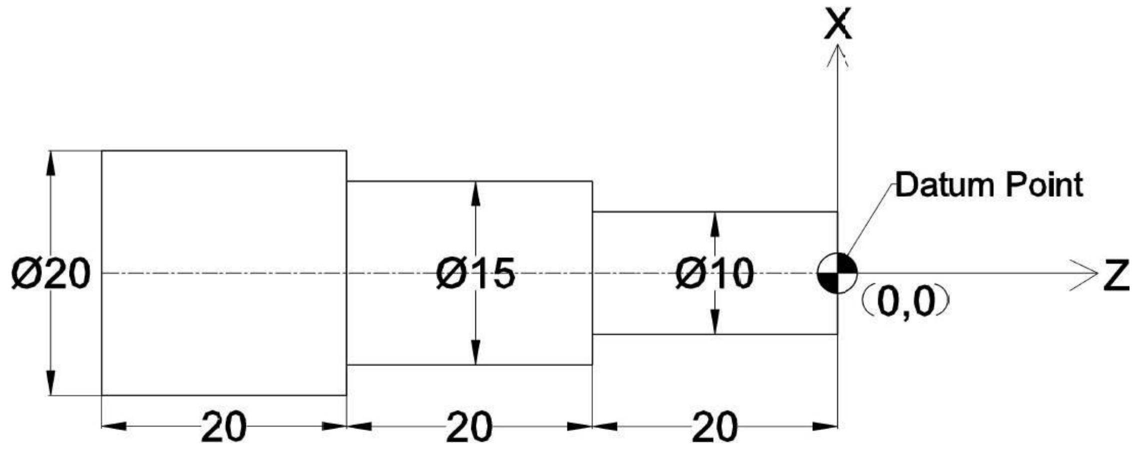
```
N50 G01 X100 Y-50.02 F100 M08
```

As you can see in above CNC program block these are Six Words separated by Spaces. Every word is a group of alphanumeric characters, every word is lead by a character as above N, G, X, Y, F, M and remaining part consist of some signed/unsigned numeric value as above give 50, 01, 100, –50.02, 100, 08

CNC TURNING

Exercise No. 1 **Date:__/__/ _____**

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



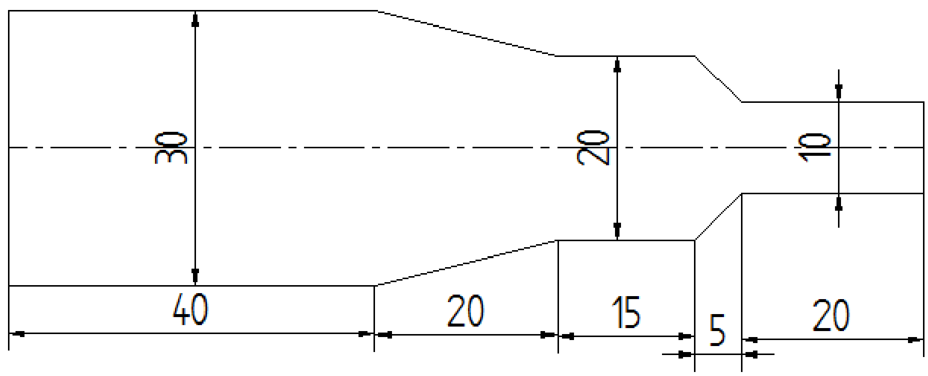
Note: All dimensions are in mm only

CNC Part Program:

O1011	[Program Number]
G21 G98	[Metric Data Input (MDI), Feed/Minute]
G28 X0 Z0	[Reference Point Return (Tool), X = 0, Z = 0]
M06 T0101	[Tool Change, Tool Number]
M03 S100	[Spindle Start, Spindle Speed]
M08	[Coolant On]
G00 X10 Z1	[Rapid Traverse/Tool Parking, X = 10, Z = 1]
G00 X7.5	[Rapid Traverse/Tool Parking, X = 7.5]
G01 Z-40 F0.2	[Linear Interpolation, Z = - 40, Feed Rate = 0.2]
G00 X10 Z1	[Rapid Traverse/Tool Parking, X = 10, Z = 1]
G00 X5	[Rapid Traverse/Tool Parking, X = 5]
G01 Z-20 F0.2	[Linear Interpolation, Z = -20, Feed Rate = 0.2]
G00 X10 Z1	[Rapid Traverse/Tool Parking, X = 10, Z = 1]
G28 X10 Z0	[Reference Point Return (Tool), X = 10, Z = 0]
M05 M09	[Spindle Stop, Coolant off]
M30	[Program End]

Exercise No. 2 **Date:___/___/ ______**

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

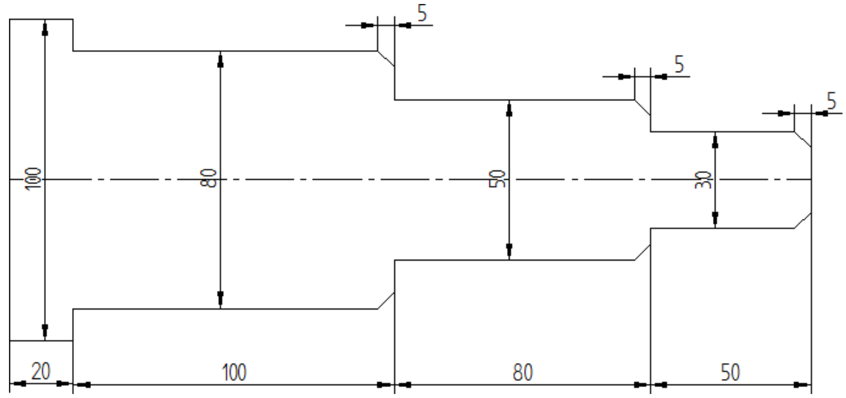


Note: All dimensions are in mm only

CNC Part Program:

Exercise No. 3 **Date:___/___/ ______**

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

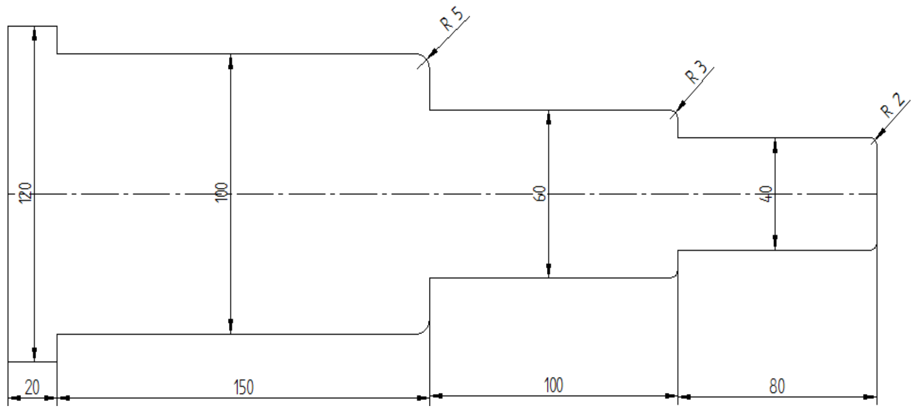


Note: All dimensions are in mm only

<u>CNC Part Program:</u>

Exercise No. 4 **Date:___/___/ ______**

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

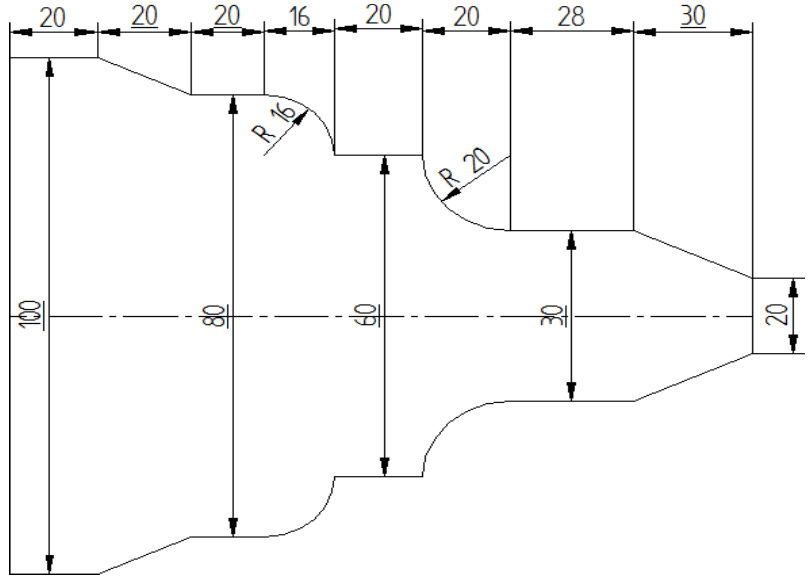


Note: All dimensions are in mm only

<u>CNC Part Program:</u>

Exercise No. 5 **Date:___/___/ ______**

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

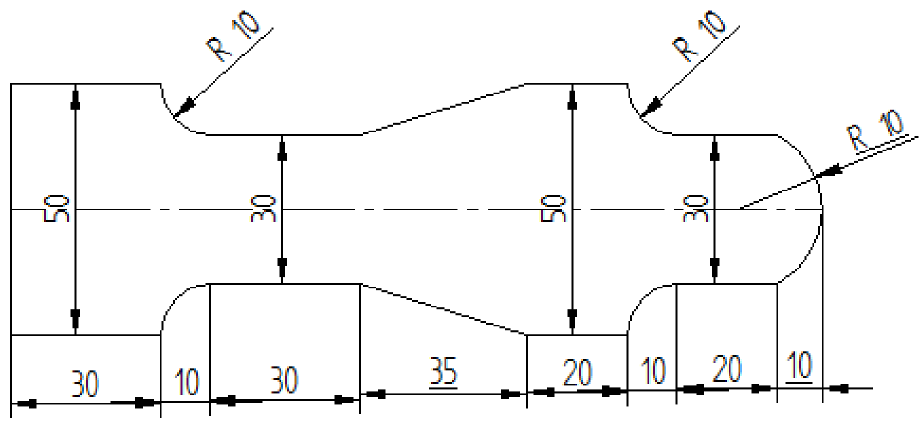


Note: All dimensions are in mm only

CNC Part Program:

Exercise No. 6 **Date:__/__/ ______**

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

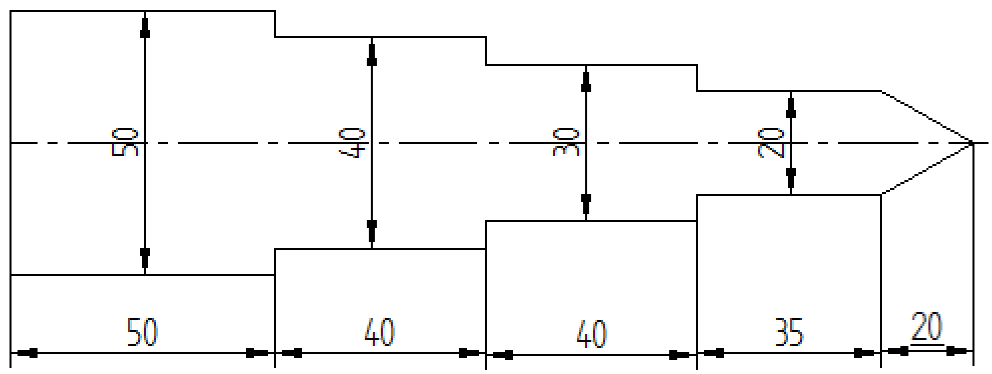


Note: All dimensions are in mm only

CNC Part Program:

Exercise No. 7 **Date:___/___/ ______**

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

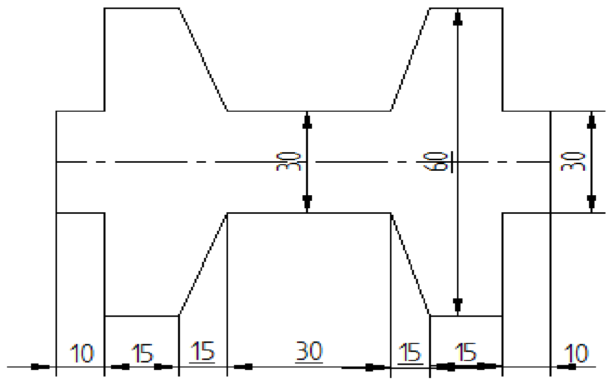


Note: All dimensions are in mm only

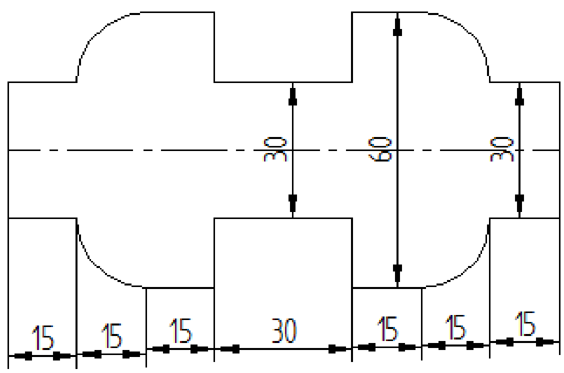
CNC Part Program:

PRACTICE PROGRAMS [CNC TURNING]

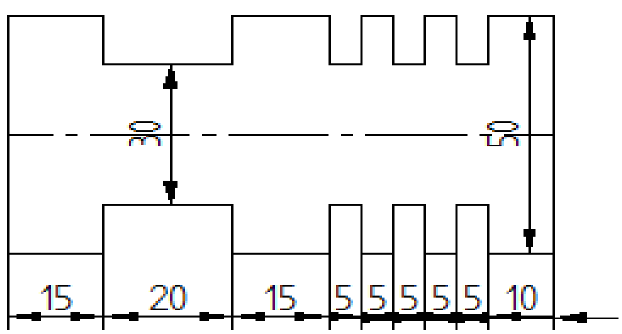
Exercise No. 8 **Date:___/___/ ______**



Exercise No. 9 **Date:___/___/ ______**



Exercise No. 10 **Date:___/___/ ______**



Note: All dimensions are in mm only

Computerized 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.

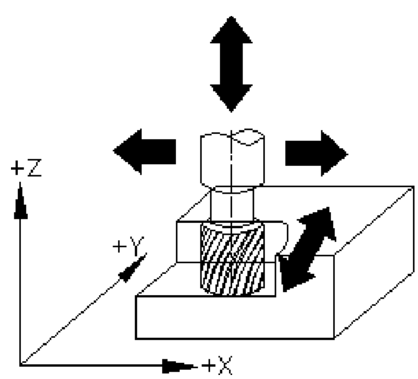


Fig. 06 Coordinate system for milling operations

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 centre 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.

Work piece 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”

NC- Related Dimensioning

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

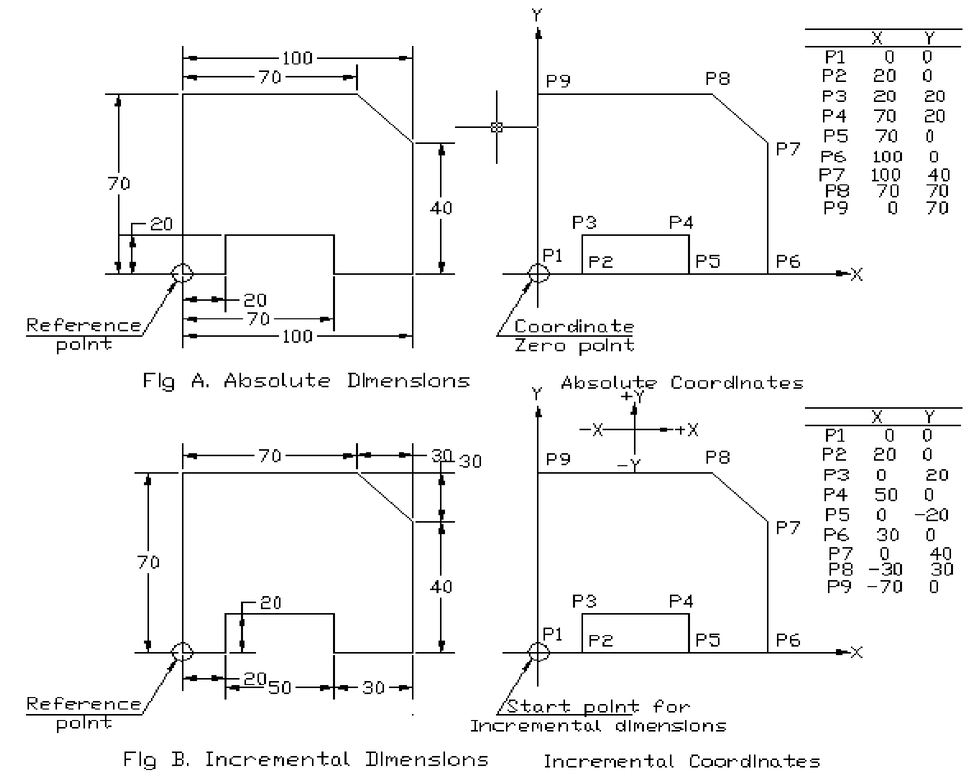


Fig. 07 Absolute and Incremental Dimension System

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
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

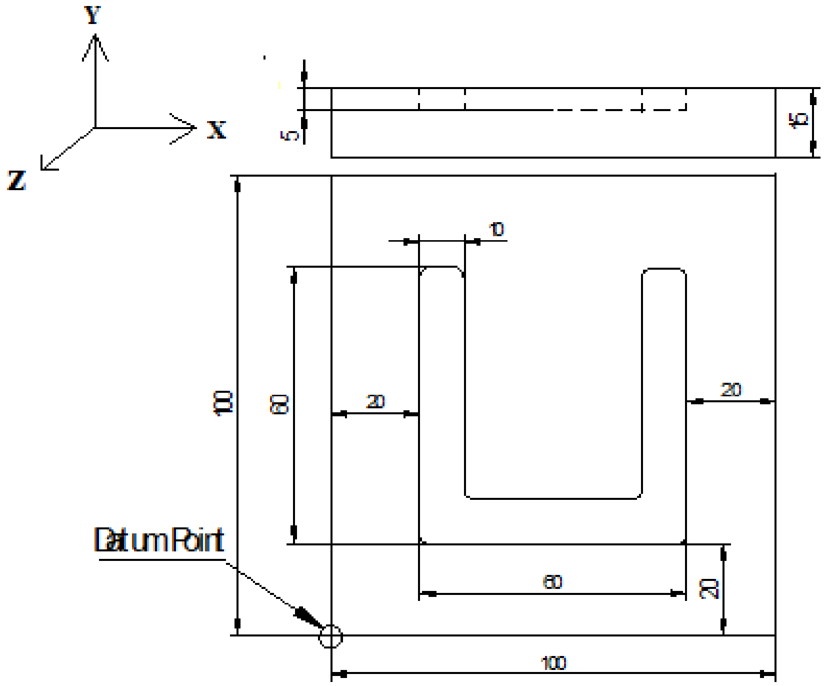
M Codes are instructions describing machine functions such as calling the tool, spindle rotation, coolant on, door close/open etc.

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

CNC MILLING

Exercise No. 1 **Date:___/___/ ______**

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



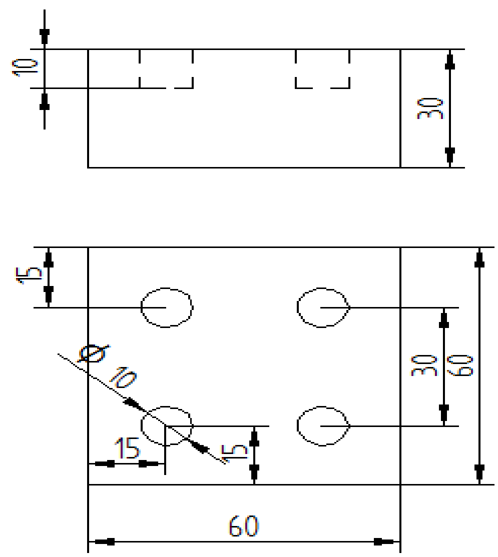
Note: All dimensions are in mm only

CNC Part Program:

O1021 [Program Number]
G21 G98 [Metric Data Input (MDI), Feed/Minute]
G28 X0 Y0 Z0 [Reference Point Return (Tool), X = 0, Y = 0, Z = 0]
M06 T0101 [Tool Change, Tool Number]
M03 S100 [Spindle Start, Spindle Speed]
G00 X25 Y75 Z5 [Rapid Traverse/Tool Parking, X = 25, Y = 75, Z= 5]
G01 Z-5 F0.2 [Linear Interpolation, Z = -5, Feed Rate = 0.2]
Y25 [Y = 25]
X75 [X = 75]
Y75 [Y = 75]
G00 Z5 [Rapid Traverse, Z= 5]
G28 X0 Y0 Z10 [Reference Point Return (Tool), X = 0, Y = 0, Z = 10]
M05 M09 [Spindle Stop, Coolant off]
M30 [Program End]

Exercise No. 2 **Date:___/___/ ______**

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

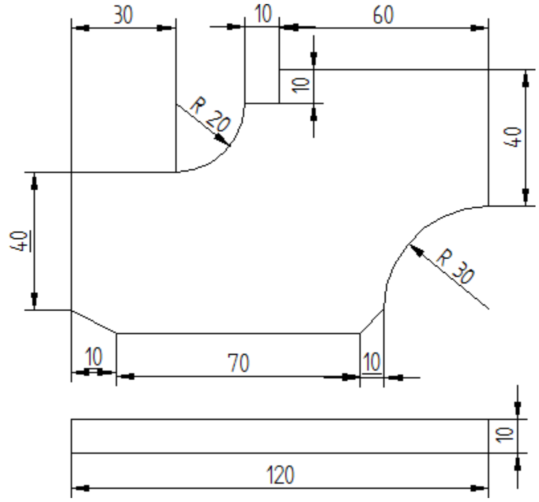


Note: All dimensions are in mm only

CNC Part Program:

Exercise No. 3 **Date:___/___/ ______**

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

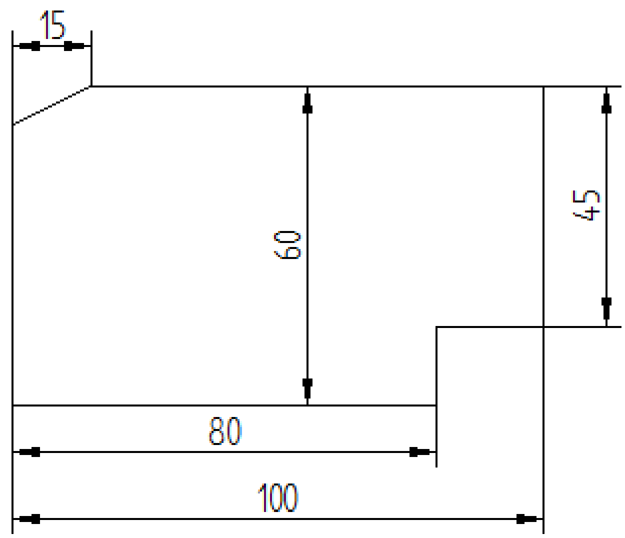


Note: All dimensions are in mm only

CNC Part Program:

Exercise No. 4 **Date:___/___/ ______**

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

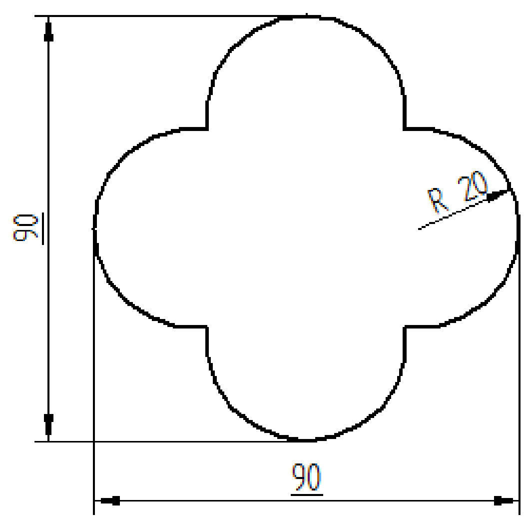


Note: All dimensions are in mm only

<u>CNC Part Program:</u>

Exercise No. 5 **Date:___/___/ ______**

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

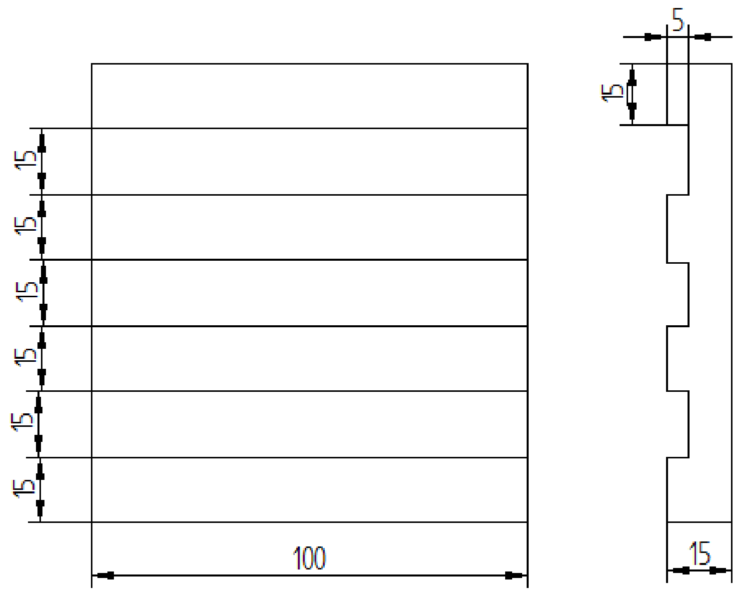


Note: All dimensions are in mm only

CNC Part Program:

Exercise No. 6 **Date:___/___/ ______**

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

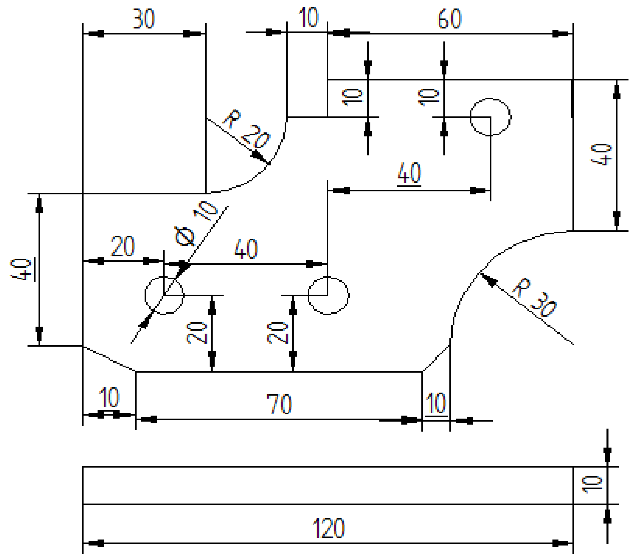


Note: All dimensions are in mm only

CNC Part Program:

Exercise No. 7 **Date:___/___/ ______**

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

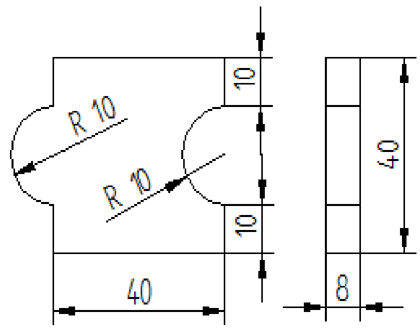


Note: All dimensions are in mm only

CNC Part Program:

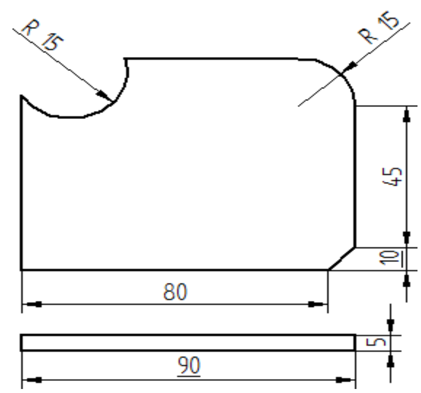
Practice Programs [CNC Milling]

Exercise No. 8 **Date:___/___/ ______**



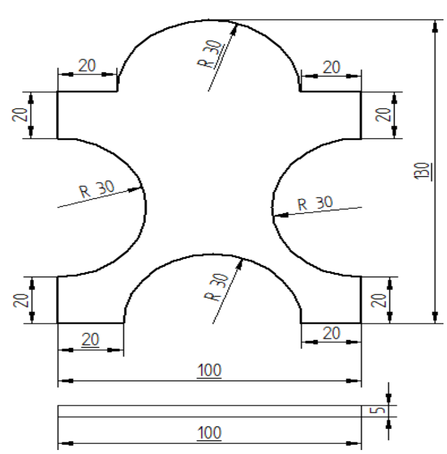
Note: All dimensions are in mm only

Exercise No. 9 **Date:___/___/ ______**



Note: All dimensions are in mm only

Exercise No. 10 **Date:___/___/ ______**



Note: All dimensions are in mm only

CAPS turn and CAPS mill

TO GENERATE THE PROGRAM

8 steps in CAPSTURN/CAPSMILL NC programming

1. Start new program
2. Define work setup
3. Draw the part
4. Draw the blank
5. Perform machining
6. Select machine
7. View tool path
8. Generate NC program

Geometry

→ Machining

→ Tool path

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.

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

VIVA QUESTIONS:

1. What is CAD?

Computer-aided design (CAD) is the use of computer systems to assist in the creation, modification, analysis, or optimization of a design.

2. What is CAM?

Computer-aided manufacturing (CAM) is the use of computer software to control machine tools and related machinery in the manufacturing of work pieces.

3. What is CAE?

Computer-aided engineering (CAE) is the broad usage of computer software to aid in engineering tasks.

4. What is Automation?

Automation is the use of machines, control systems and information technologies to optimize productivity in the production of goods and delivery of services.

5. What are the benefits of CAD?

- Improved engineering productivity
- Reduced engineering personnel requirements
- Customer modifications are easier to make
- Faster response to requests for quotations
- Minimized transcription errors
- Improved accuracy of design
- Improved productivity in tool design

6. What is design process?

- Define the Problem
- Do Background Research
- Specify Requirements
- Create Alternative Solutions
- Choose the Best Solution
- Do Development Work
- Build a Prototype
- Test and Redesign

7. What is geometric modelling?

Geometric modelling is a branch of applied mathematics and geometry that studies methods and algorithms for the mathematical description of shapes.

8. Advantages of CAD/CAM?

- Savings in geometry definition.
- Immediate visual verification.
- Use of automatic programming routines.
- One-of-a-kind jobs.
- Integration with other related functions.

09. Define NC?

Numerical Control (NC) is the automation of machine tools that are operated by abstractly programmed commands encoded on a storage medium.

10. What are the basic components of NC system?

An operational numerical control system consists of the following three basic components:

1. Program of instructions
2. Controller unit, also called a machine control unit (MCU)
3. Machine tool or other controlled process

11. What is NC procedure?

- Process planning.
- Part programming

Manual part programming

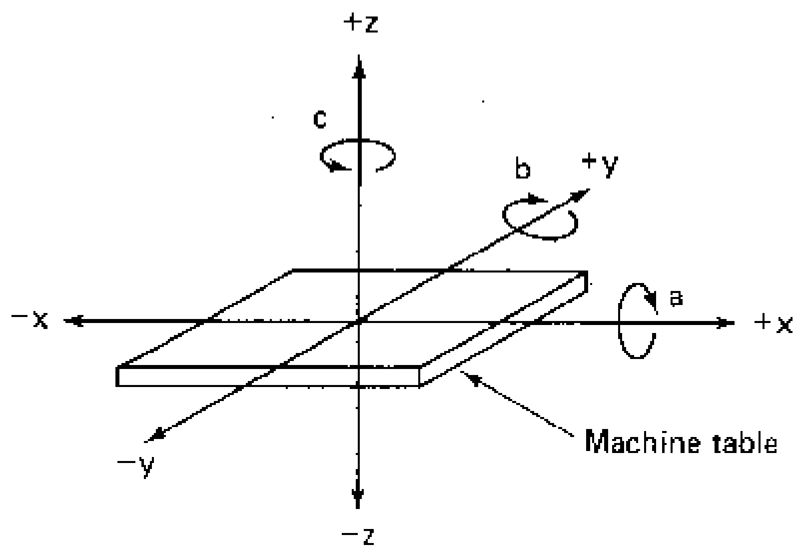
Computer-assisted part programming

- Tape preparation.
- Tape verification.
- Production.

12. What is cutter offset compensation?

An offset used on the mill that accounts for variations in tool diameter. Cutter compensation is necessary only for tools that travel in the X- or Y-axes.

13. Discuss NC coordinate system?



14. What is work piece Zero point?

The origin of both the work piece coordinates system and the part program for a particular work piece. Work piece zero, commonly called program zero, is unique to each work piece design and is selected by a part programmer.

15. What is Machine zero point?

The origin of the machine coordinates system located above the far upper right-hand corner of the mill table. The unchangeable machine zero point is also known as the home position.

16. What Home zero point?

The origin of the machine coordinate system located above the lathe spindle and to the far upper right-hand corner of the lathe work area. The unchangeable machine zero point is also known as the home position.

17. Applications of NC systems?

- Batch and high volume production
- Repeat and repetitive order
- Complex part geometries
- Many separate operations on one part

18. Advantages and disadvantages of NC machine?

Advantages

- Part program tape and tape reader

- Editing the program
- Metric conversion
- Highly flexible
- Easier programming

Disadvantages

- Higher investment cost.
- Higher maintenance cost
- Finding and/or training NC personnel

19. What does N Word stands for?

N - Sequence number (Used for line identification)

20. What does G word stands for?

G - Preparatory function

21. What does M Word stands for?

M - Miscellaneous function

22. What does T word stands for?

T - Tool Designation

23. Steps in computer assisted part programming?

- Typically starts with the receipt (by the manufacturing department) of a design in the form of a CAD/NC drawing or model
- Review of the model by a production planner and then design/selection of the tools
- Selection of cutting process parameters (cutting conditions, direction of cut, roughing and finishing, etc)
- Generation of cutter path
- Verification of the cutter path by replaying the path – computer assists the programmer by animating the entire path, showing the location of the cutter visually and displaying the XYZ coordinates

24. What is robot?

A robot is a mechanical or virtual agent, usually an electro-mechanical machine that is guided by a computer program or electronic circuitry.

25. Physical configurations of robot.
    - Cartesian configuration
    - Cylindrical configuration
    - Polar configuration
    - Jointed-arm configuration

26. Basic robot motions.
    1. Arm and body motions
        - Vertical traverse
        - Radial traverse
        - Rotational traverse
    2. Wrist Motion
        - Wrist swivel
        - Wrist bend
        - Wrist yaw

27. Robot programming language.
    - The VAL™ Language
    - The MCL Language

28. Basic commands for robot

MOVE, HERE, APPROACH, DEPART, MOVE PATH, SPEED, EXECUTE PROGRAM

29. Applications of robot
    - Hazardous work environment for humans
    - Repetitive work cycle
    - Difficult handling task for humans
    - Multi shift operations
    - Infrequent changeovers
    - Part position and orientation are established in the work cell

30. Advantages and disadvantages of robot

Advantages

- Robotics and automation can, in many situation, increase productivity, safety,
- efficiency, quality, and consistency of Products

- Robots can work in hazardous environments
- Robots need no environmental comfort
- Robots work continuously without any humanity needs and illnesses
- Robots have repeatable precision at all time

Disadvantages

- Robots lack capability to respond in emergencies, this can cause:
  - Inappropriate and wrong responses
  - A lack of decision-making power
  - A loss of power
- Robots may have limited capabilities in
  - Degrees of Freedom
  - Sensors
- Robots are costly, due to
  - Initial cost of equipment
  - Installation Costs

31. What is FMS?

A flexible manufacturing system (FMS) is a manufacturing system in which there is some amount of flexibility that allows the system to react in the case of changes, whether predicted or unpredicted.

32. What is automatic storage and retrieval system?

An automated storage and retrieval system (ASRS or AS/RS) consists of a variety of computer-controlled systems for automatically placing and retrieving loads from defined storage locations.

33. What is meant by canned cycle (or) fixed cycle? Give an example.

A canned cycle simplifies a program by using a few blocks containing G code functions to specify the machining operations usually specified in several blocks.

Ex. Drilling (G81), Peck drilling (G83), Tapping (G84), Boring (G86)

References

1. Automation, Production system & Computer Integrated manufacturing, M. P. Groover Person India, 2007 2nd edition.
2. Principles of Computer Integrated Manufacturing, S. Kant Vajpayee, Prentice Hall India.
3. Cadem software manuals.