

# LABORATORY MANUAL

18MEL76 COMPUTER INTEGRATED MANUFACTURING LAB

2019-2020



DEPARTMENT OF MECHANICAL ENGINEERING  
ATRIA INSTITUTE OF TECHNOLOGY  
Adjacent to Bangalore Baptist Hospital  
Hebbal, Bengaluru-560024

**Department of Mechanical Engineering**

**Vision**

To be a center of excellence in Mechanical Engineering education and interdisciplinary research to confront real world societal problems with professional ethics.

**Mission**

1. To push the frontiers of pedagogy amongst the students and develop new paradigms in research.
2. To develop products, processes, and technologies for the benefit of society in collaboration with industry and commerce.
3. To mould the young minds and build a comprehensive personality by nurturing strong professionals with human ethics through interaction with faculty, alumni, and experts from academia/industry.



**Computer-integrated manufacturing (CIM)** is the manufacturing approach of using computers to control entire production process. This integration allows individual processes to exchange information with each part. The idea of "digital manufacturing" became prominent in the early 1970s, with the release of Dr. Joseph Harrington's book, *Computer Integrated Manufacturing*. However, it was not until 1984 when computer-integrated manufacturing began to be developed and promoted by machine tool manufacturers and the Computer and Automated Systems Association and Society of Manufacturing Engineers (CASA/SME)

**ATRIA INSTITUTE OF TECHNOLOGY**  
**Department of Mechanical Engineering**

**VII Semester**

**COMPTER INTEGRATED MANUFACTURING LAB**

**ACADEMIC YEAR**  
**2020 – 2021**

**NAME OF THE STUDENT** : .....

**UNIVERSITY SEAT NO.** : .....

**BATCH** : .....

**PREPARED BY**

**PROF. PRAVEEN KUMAR. BC**

**ATRIA INSTITUTE OF TECHNOLOGY**  
**Department of Mechanical Engineering**

**LABORATORY CERTIFICATE**

This is to certify that Mr.  
/Ms. \_\_\_\_\_ bearing  
USN \_\_\_\_\_ of \_\_\_\_\_ semester and \_\_\_\_\_ section has  
satisfactorily completed the course of experiments in **COMPTER**  
**INTEGRATED MANUFACTURING LAB**, code **18MEL76**  
prescribed by the Visvesvaraya Technological University, Belagavi of this  
Institute for the academic year 2020 – 2021.

<b>MARKS</b>	
<b>Maximum Marks</b>	<b>Marks Obtained</b>

Signature of Faculty-In-Charge

Head of the Department

Date:

## INDEX PAGE

SI.N o	Name of the Experiment	Date			Manual Marks Max .	Record Marks Max.	Signature Student	Signature Faculty
		Conduction	Repetition	Submission of Record				
<b>Average</b>								

### **Instructions to students**

Users of Computer Lab must follow the following instructions for smooth conduction of laboratory.

1. Wear always safety shoes when you operate any System. Sandals are not allowed at all.
2. There should be no over-crowding.
3. Listen to the instructions first and act Accordingly.
4. Before operating any System, you must be aware of the Power switches.
5. Maintain cleanliness around the System.
6. In case of emergency, Consult the instructor for safety precautions to be followed.
7. Always Use the designated system provided to you, don't change your system.
8. Handle the equipment's with extreme care.
9. Switch off the system and Keep the chairs proper position, while leaving lab.
10. Do not remove mouse and keyboard from any system.

## PREFACE

Computer Integrated Manufacturing (CIM) embodies three components essential to the implementation of flexible design & manufacturing the means for information storage, retrieval, manipulation and presentation, the mechanisms by which to sense state, and modify substance and the methodologies by which to unite them. With the inception of Computer Integrated Manufacturing (CIM) machine in manufacturing setup, precise and reliable machine with out human during the machining process had enabled reduction in operational hazards and in job time through automated machining and tool changing.

This manual gives the perception to build technical knowledge by acting as a guide for imparting fundamental awareness. Numerous neatly drawn illustrations provided in the manual will help the students in understanding the subject, and the concepts related it, better. Sincere attempts have been made to present the contents in a simple language.

**PRAVEEN KUMAR B.C.**



# Department of Mechanical Engineering

## Vision

To be a center of excellence in Mechanical Engineering education and interdisciplinary research to confront real world societal problems with professional ethics.

## Mission

1. To push the frontiers of pedagogy amongst the students and develop new paradigms in research.
2. To develop products, processes and technologies for the benefit of the society in collaboration with industry and commerce.
3. To mould the young minds and build a comprehensive personality by nurturing strong professionals with human ethics through interaction with faculty, alumni and experts from academia/industry.

---

### **OBJECTIVES**

---

The objectives of Computer Integrated Manufacturing and Automation laboratory is

- 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.
- 

### **OUTCOMES**

---

The expected outcome of Computer Integrated Manufacturing and Automation lab is that the students will be able

- CLO1:** Generate CNC Lathe part program for Turning, Facing, Chamfering, Grooving, Step turning, Taper turning, Circular interpolation etc.
- CLO2:** Generate CNC Mill Part programming for Point to point motions, Line motions, Circular interpolation, Contour motion, Pocket milling- circular, rectangular, Mirror commands etc.
- CLO3:** Use Canned Cycles for Drilling, Peck drilling, Boring, Tapping, Turning, Facing, Taper turning thread cutting etc.
- CLO4:** Simulate Tool Path for different Machining operations of small components using CNC Lathe & CNC Milling Machine.
- CLO5:** Use high end CAM packages for machining complex parts; use state of art cutting tools and related cutting parameters; optimize cycle time.
- CLO6:** Understand & write programs for Robot control; understand the operating principles of hydraulics, pneumatics and electro pneumatic systems. Apply this knowledge to automate & improve efficiency of manufacturing.

**SYLLABUS****COMPTER INTEGRATED MANUFACTURING LAB****Subject. Code : 15 MEL 77/17ME77****No of Practical Hrs/ Week: 03****Time : 3 hours****Total No of Practical Hrs. 42****PART-A**

**Manual CNC part programming** for 2 turning and 2 milling parts. Selection and assignment of tools, correction of syntax and logical errors, and verification of tool path.

**CNC part programming using CAM packages.** Simulation of Turning, Drilling, Milling operations

3 typical simulations to be carried out using simulation packages like: **Cadem CAMLab-Pro, Master- CAM.**

Program generation using software. Optimize spindle power, torque utilization, and cycle time. Generation and printing of shop documents like process and cycle time sheets, tool list, and tool layouts. Cut the part in single block and auto mode and measure the virtual part on screen. Post processing of CNC programs for standard CNC control systems like FANUC, SINUMERIC and MISTUBISHI.

**PART-B**

**(Only for Demo/Viva voce)**

**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.

**(Only for Demo/Viva voce)**

**Robot programming:** Using Teach Pendent & Offline programming to perform pick and place, stacking of objects (2 programs).

**Pneumatics and Hydraulics, Electro-Pneumatics:** 3 typical experiments on Basics of these

**CONTENTS**

Chapter-1 01- 12

Chapter-2 13- 23

**Chapter-3 CNC PROGRAMS**

<b>SL. No.</b>	<b>Name of The Experiment</b>	<b>Page No.</b>
<b><u>TURNING EXERCISE</u></b>		
1	<b>Turning Exercise 1</b>	24
2	<b>Turning Exercise 2</b>	26
3	<b>Turning Exercise 3</b>	28
4	<b>Turning Exercise 4</b>	30
5	<b>Turning Exercise 5</b>	32
6	<b>Turning Exercise 6</b>	34
7	<b>Turning Exercise 7</b>	36
<b><u>MILLING EXERCISE</u></b>		
8	<b>Milling Exercise 1</b>	38
9	<b>Milling Exercise 2</b>	40
10	<b>Milling Exercise 3</b>	42
11	<b>Milling Exercise 4</b>	44
12	<b>Milling Exercise 5</b>	46
13	<b>Milling Exercise 6</b>	48
14	<b>Milling Exercise 7</b>	50

Chapter-5 FLEXIBLE MANUFACTURING SYSTEMS (FMS) 52-57

Chapter-6 A HYDRAULIC SYSTEM 58-61

Chapter-7 VIVA VOCE QUESTIONS 62-64

## Chapter-1

### INTRODUCTION

#### **Numerical control :( NC)**

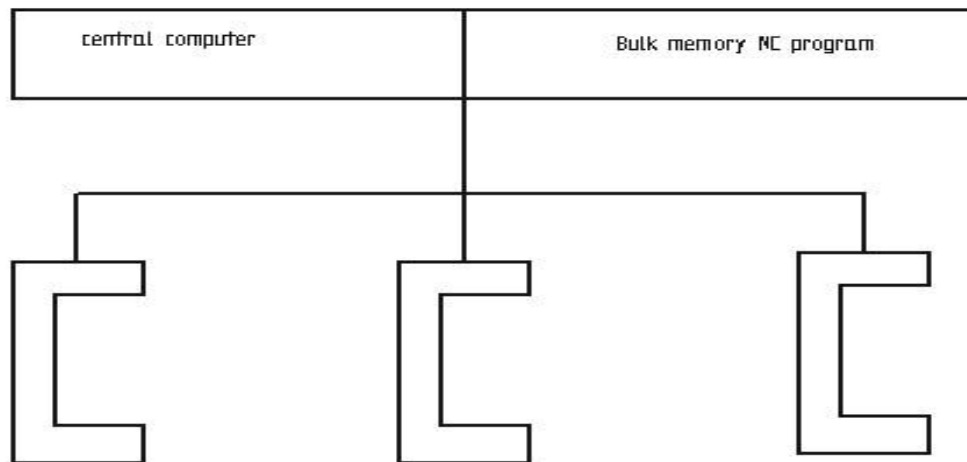
It can be defined has form of programmable automation in which the process is controlled by numbers, letters and symbols in NC the numbers forms a program of instructions designed for a particular work part or job.

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.

#### **NC motion control system:**

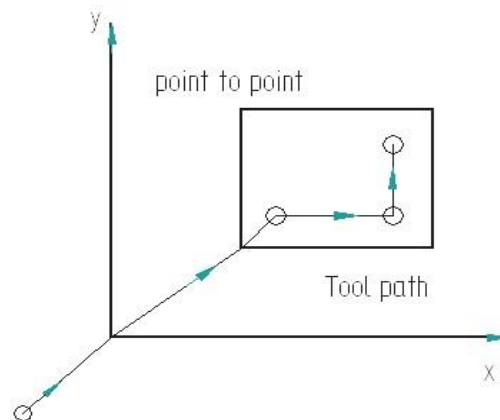
In NC there are 3 basic types of machine control system

1. Point to Point
2. Straight cut
3. Contouring

### 1) Point to point

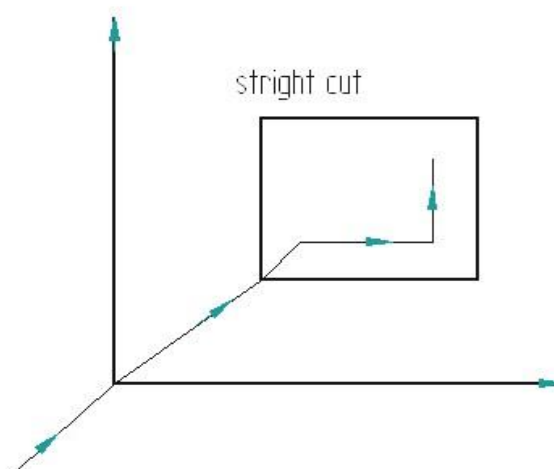
It is also sometimes called positioning system. In point to point the objective of the machine tool control system is to the cutting to pre defined location once the tool reaches the defined location the machining operation is performed at that position.

**EX:** NC drill presses.



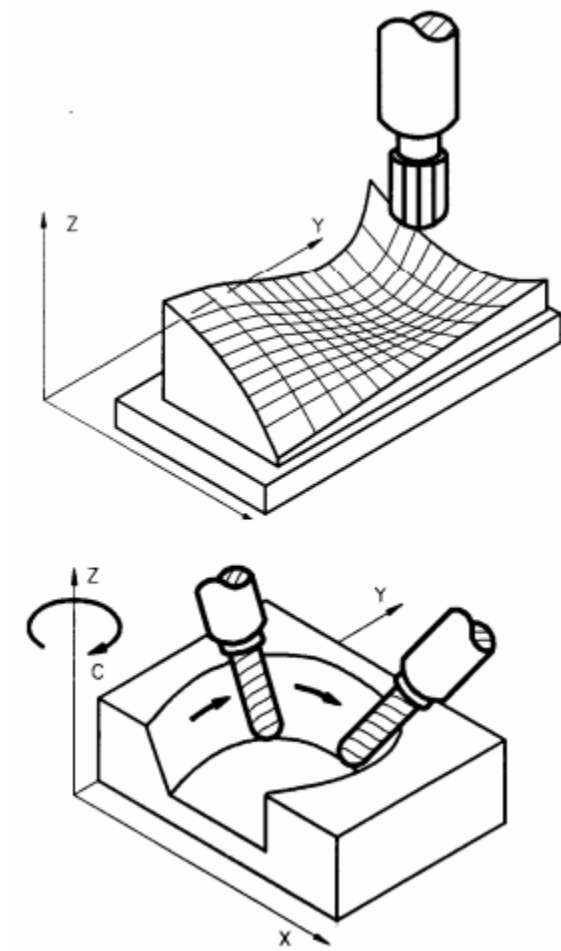
### 2) Straight cut NC

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.



FUNDAMENTALS OF PART PROGRAMMING

**2) Contouring:** The continuous-path control system is also known as the contouring system. It involves simultaneous motion control of two or more axes. The contouring system is more complex because each axis of motion requires separate position and velocity loops. The contouring along a predefined tool path is implemented by means of interpolation, in which the system generates a set of intermediate data points between given coordinate positions.



### **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.

<b>PROCESS PLANNING</b>	
<input type="checkbox"/>	Receive the part drawing from part drawing information, check suitability of part to be machined against the machine capacity.
<input type="checkbox"/>	Determine a method of driving the component (chuck type, chuck size, type of jaw) and the method of machining.
<input type="checkbox"/>	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.
<input type="checkbox"/>	Determine the order of machining and the tooling stations.
<input type="checkbox"/>	Determine planned stops for checking dimensional sizes where required by operator
<input type="checkbox"/>	Determine cutting speeds based on <ul style="list-style-type: none"> <li>- Component material, method of driving, rigidity of component</li> <li>- Tooling selected for roughing and finishing</li> </ul>
<input type="checkbox"/>	Determine the depths of cut and feeds for roughing operations
<input type="checkbox"/>	Determine surface finish requirements, the cutter nose radius most suited for finishing operations and determine feed rates.
<input type="checkbox"/>	Allocates tool offsets as required
<input type="checkbox"/>	Complete planning sheet

**B) PART PROGRAMMING**

- |                          |  |
|--------------------------|--|
| <input type="checkbox"/> | After completing the planning sheet, draw the component showing the cutter paths (a simple sketch is sufficient for simple components) |
|--------------------------|--|



- |  |
|--|
| <input type="checkbox"/> Select a component datum and carryout the necessary calculations at slopes and arcs.                                |
| <input type="checkbox"/> Prepare tooling layout sheet showing tools to be used in the program and indicate the station number for each tool. |
| <input type="checkbox"/> Indicate the ordering code for each tool and grade and type of inserts to be used.                                  |
| <input type="checkbox"/> 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 preprogrammed data file, the computer control can.

- |   |
|---|
| <ul style="list-style-type: none"><li>- Select numerical values for use within machining calculations</li><li>- Perform calculations to optimize machining conditions</li><li>- Identify standard tools and coordinates</li><li>- Calculate cutter paths and coordinates</li><li>- Generate the part program to machine the component</li></ul> |
|---|

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, left 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:

1. **Coordinate System for a CNC Lathe:** Machining of a workpiece 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 0 movement along each axis is specified as either PLUS (+) or MINUS (-). The Z-axis is: always parallel to the main spindle of the machine. The X-axis is always parallel to tilt work-holding surface, and always at right angles to the Z-axis. The Y-axis is at right angle to both Z and X-axis. The Coordinate System for turning operations is shown in Fig. 1.

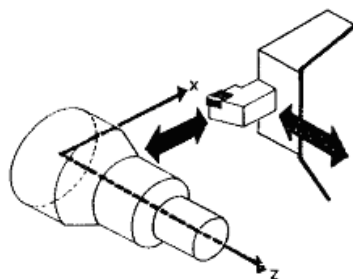


Fig. 1: Coordinate System for Turning Operations  
(Tool Positioned Behind Center Line)

The coordinate system for designating the axes is the conventional “**Right Hand Coordinate System**” as shown in Fig.2. A labeling of the axes is a right hand coordinate system whenever the fingers of the right hand are aligned with the positive X axis and are then rotated (through the smaller angle) toward the positive Y axis, then the thumb of the right hand points in the direction of the positive Z axis. Otherwise, the orientation is a “**Left Hand Coordinate System**”.

The right hand coordinate system is also known as “**Clockwise Rotating Coordinate System**”. The reason for this is the sequence of the axis definitions if the X-axis is rotating in the direction of the Y-axis, the movement is the same as if a screw is turned in the Z direction as shown in **Fig.3**.

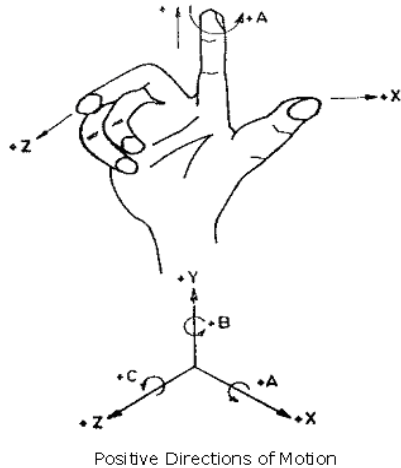


Fig.2: Right Hand Coordinate Systems

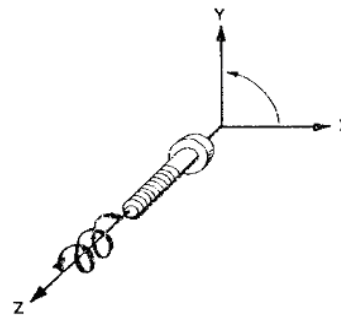


Fig.3: Clockwise Rotating Coordinate System

One could use his right hand to get these alternative relative positions of the same right hand coordinate system as shown in **Fig.4**

## 2. 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

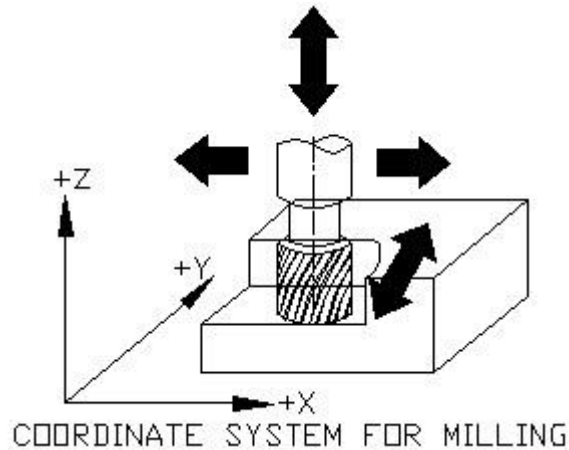
Each plane is assigned a letter and is referred to as an axis, i.e.,

- 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.

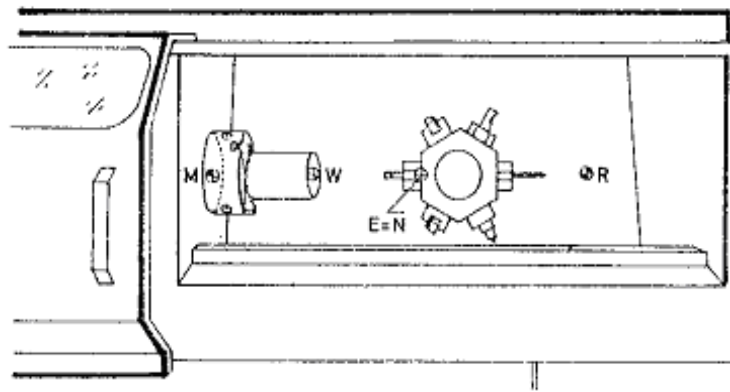
The X axis is always parallel to the work holding surface, and always at right angles to the Z axis.

The Y axis is at right angles to both Z and X axis. Figure shows the coordinate system for milling.



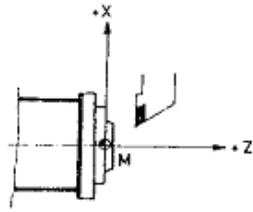
### Zero Points & Reference Points:

On CNC coordinating systems controls machines tool traverses. Their accurate position within the machine tool is established by “Zero Points” as shown in **fig. 5**.

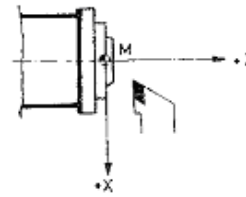


**Fig. 5: Zero & Reference Points on a CNC Machine**

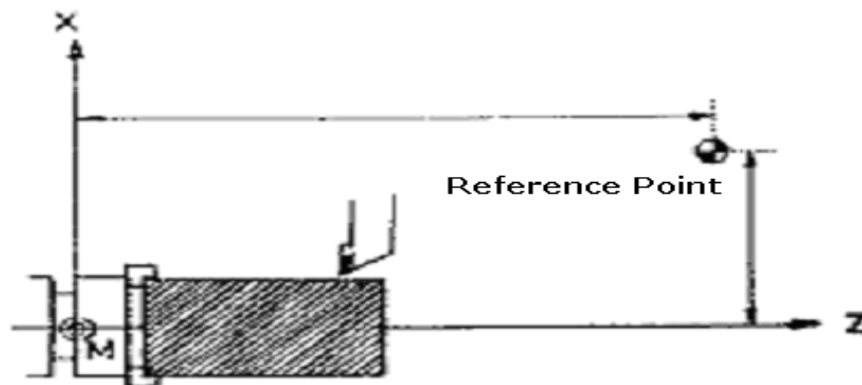
Machine zero point (M): is specified by the manufacturer of the machine. This is the zero point for the coordinate systems & reference points in the machine. On turning lathes, the machine zero point as shown in F/G.6 & FIG.7 is generally at the center of the spindle nose face. The main spindle axis (center line) represents the Z-axis; the face determines the X-axis. The directions of the positive X and Z-axis point toward the working area. When the tool traverses in the positive direction, it moves away from the workpiece



**Fig. 6: Position of Machine Zero Point on Turning Lathes**  
(When Machining with tools behind Center Line)



**Fig. 7: Position of Machine Zero Point on Turning Lathes**  
(When Machining with tools on Center Line)



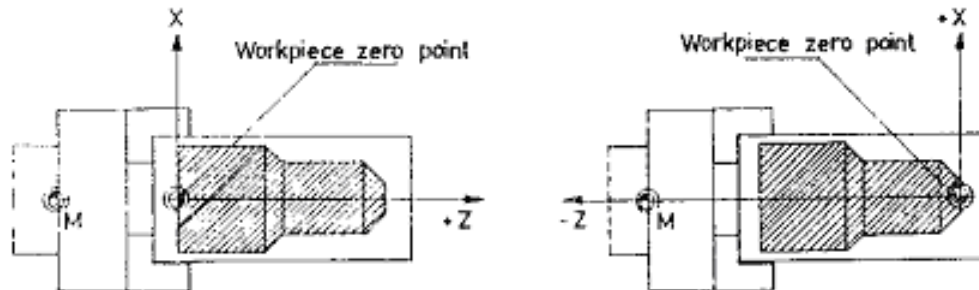
**Fig. 8: Position of Reference Point on a Lathe**

**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 FIG.8 is accurately predetermined 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.

**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

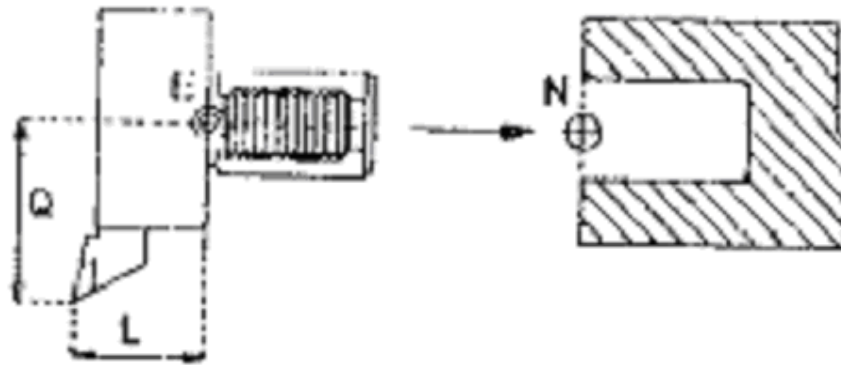
to place the workpiece zero point in such a manner that the dimensions in the workpiece drawing can be conveniently converted into coordinate values and orientation when damping/chucking, setting up and checking, the traverse measuring system can be effected easily. For turned parts, the workpiece zero 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 FIG.9. Occasionally, the workpiece zero point is also called the “**Program Zero Point**”.



**Fig. 9: Examples of Workpiece Zero Point Position for Turned Parts**

**Tool Point:** Which machining a workpiece, it is essential to able to control the tool point or the tool cutting edges in precise relationship to the workpiece along the machining path Since tools have different shapes and dimensions, precise tool dimensions have to be established beforehand and input into the control system. The tool dimensions arc related to a fixed tool setting point during pre-setting.

The tool selling point .E is located at a certain point on the tool holder as shown in **Fig. 10**. This setting point permits measuring of tools away from the CNC machine. The data thus measured such as tool length, tool point offset or tool radius is input into the tool data storage (memory) or the control system. The mate of the tool setting point is the Socket point N on the tool carrier. When the tool or tool holder is inserted into the tool carrier (e.g., turret), the setting point and the tool socket point coincide.

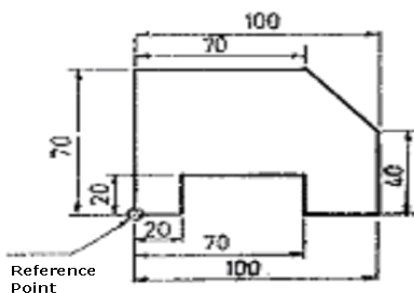


**Fig. 10: Tool Reference Points**

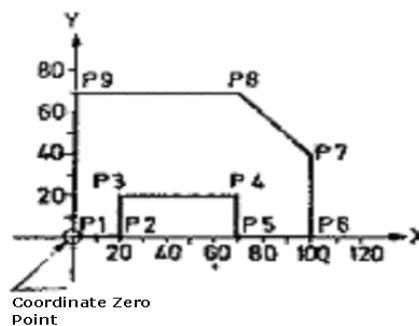
### NC Related Dimensioning:

Dimensional information in a workpiece 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 in the drawing as shown in Fig.11. This point has the function of a coordinate zero point as shown in Fig. 12. The dimension lines run parallel to the coordinate axes and always start at the reference point. Absolute dimensions are also called as “Reference Dimensions”



**Fig 11: Absolute Dimensions**



**Fig 12: Absolute Coordinates**

	X	Y
P1	0	0
P2	20	0
P3	20	20
P4	70	20
P5	70	0
P6	100	0
P7	100	40
P8	70	40
P9	0	40

### Advantages of Absolute Dimension System:

1. In case of interruptions that force the operator to stop the machine, the cutting tool automatically returns to previous position, since it always moves to the absolute coordinate called for, and the machining proceeds from the same block where it was interrupted.
2. Possibility of easily changing the dimensional data in the part program whenever require
3. When describing contours and positions, it is always preferable to employ absolute dimensions, because the first incorrect dimensioning of an individual point has no effect on the remaining dimensions & the absolute system is easier to check for error



## CHAPTER 2

### CNC PROGRAMMING

#### NC Program Buildup:

**Table. 1: Address Characters**

CHARACTER	MEANING
A,B,C	Co-ordinate axis move command
F	Feed
G	Preparatory function, identifying the action to be executed
I,J,K	Co-ordination for arc center
M	Auxiliary function
N	Block number/Sequence Number
S	Spindle speed
T	Tool
U, V, W	Second movement parallel to X, Y, Z axes respectively.
X	Movement in X-axis
Y	Movement in Y-axis
Z	Movement in Z-axis
O	Program
P	Subprogram
L	Number of Subprogram repetitions

In an NC program, the machining steps (operations) for producing a part on the machine tool are laid down in a form that the control system can understand. A program is composed of several blocks. A block is a collection of NC words

**MISCELLANEOUS FUNCTION (M Codes) For Turning**

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

<b>M CODES</b>	
<b>M00</b>	Program Stop
<b>M02</b>	Optional Stop
<b>M03</b>	Spindle Forward (CW)
<b>M04</b>	Spindle Reverse (CCW)
<b>M05</b>	Spindle Stop
<b>M06</b>	Tool Change
<b>M08</b>	Coolant On
<b>M09</b>	Coolant Off
<b>M10</b>	Vice Open
<b>M11</b>	Vice Close
<b>M13</b>	Spindle Forward, Coolant On
<b>M14</b>	Spindle Reverse, Coolant On
<b>M30</b>	Program End
<b>M38</b>	Door Open
<b>M39</b>	Door Close
<b>M98</b>	Subprogram Call

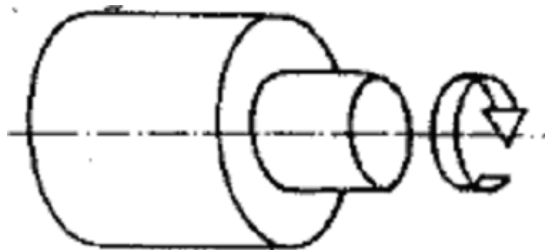
<b>M99</b>	Subprogram Exit
------------	-----------------

**M01 Optional Stop:** Cycle operation is stopped after a block containing M01 is executed. This code is only effective when the optional stop switch on the machine control panel has been pressed. **Example:** M01

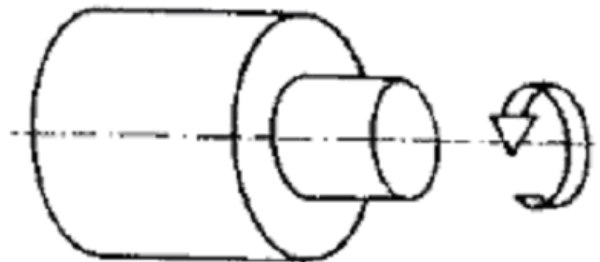
**M02 Program End:** This code is inserted at the end of the program. When encountered the cycle will encl. To produce another the system must be reset.

**Example:** M02

**M03 Spindle Forward:** Starts the spindle spinning forward, clockwise or negative direction at the last specified spindle rate. **Example:** M03 5120



**M04 Spindle Reverse:** Starts the spindle spinning reverse, counter clockwise of positive direction at the last specified spindle rate. **Example:** M04 51000

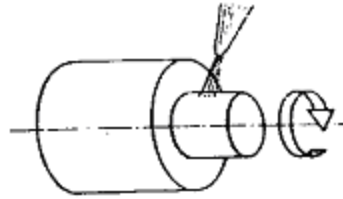


**M05 Stop Spindle:** Stops the spindle without changing the spindle speed. **Example:** MOS

**M06 Tool Change:** The M06 in conjunction with “T” WORD, is used to can up the required tool on an automatic indexing turret machine, and to activate its tool offsets. The left most digit of the “T” ignoring zeros selects the new tool. Tool changes are normally performed with the tool post at a safe position away from the workpiece, so the code G28 REFERENCE POINT RETURN would be used in the block prior to M06.

**Example:**      M06 and      T0200  
                         &      T20  
                         &      T2 all select tool 2

**M08 Coolant On:** M08 turns the coolant on.



M08

**M09 Coolant Off:** M09 turns the coolant off

**M10 Chuck Open:** M10 opens pneumatic or similar automatic chuck to allow for bar feed.

**M11 Chuck Close:** M11 closes the chuck

**M13 Spindle Forward, Coolant On:** Sets spindle rotation forward and coolant all. **Example: M13 51000**

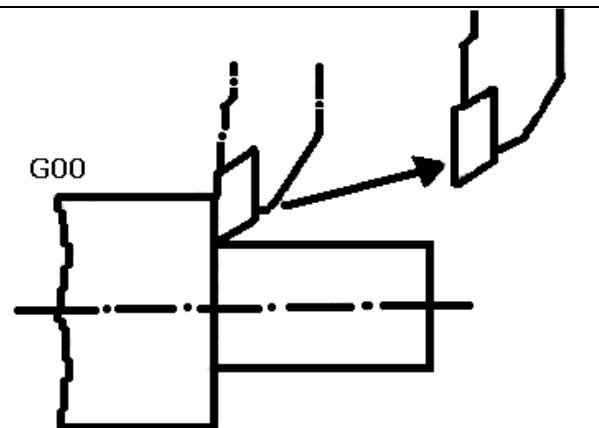
**PREPARATORY FUNCTION (G-Codes) For Turning.**

<b>G CODES</b>	
<b>G00</b>	Positioning (Rapid Transverse)
<b>G01</b>	Linear Interpolation (Feed)
<b>G02</b>	Circular Interpolation (CW)
<b>G03</b>	Circular Interpolation (CCW)
<b>G04</b>	Dwell
<b>G20</b>	Inch Data Input
<b>G21</b>	Metric Data Input
<b>G28</b>	Reference point return
<b>G40</b>	Tool nose radius compensation cancel
<b>G41</b>	Tool nose radius compensation left
<b>G42</b>	Tool nose radius compensation right
<b>G50</b>	Work coordinate change/ Max. Spindle speed setting
<b>G70</b>	Finishing cycle
<b>G71</b>	Multiple Turning Cycle in turning

<b>G72</b>	Stock removal in facing
<b>G73</b>	Pattern repeating
<b>G74</b>	Peck drilling in Z axis
<b>G75</b>	Grooving in X axis
<b>G76</b>	Thread cutting cycle
<b>G90</b>	Cutting cycle A (Turning)
<b>G94</b>	Cutting cycle B (Facing)
<b>G96</b>	Constant surface speed control
<b>G97</b>	Constant surface speed control cancel
<b>G98</b>	Feed per minute
<b>G99</b>	Feed per revolution

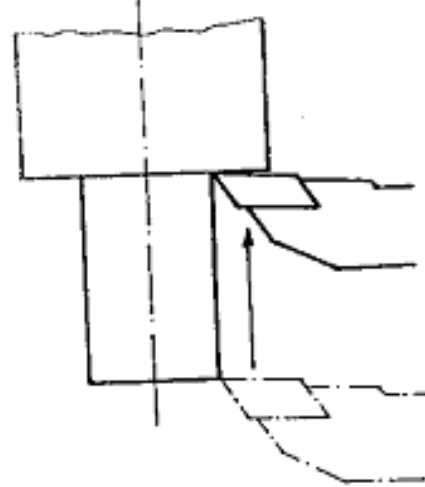
### FAST TRAVERSE – G00

**Description:** The rapid traverse instruction (function) is identified by the program word G00. A rapid traverse instruction traverses the tool to the target point at maximum traverse rate. As supplementary functions it will be necessary to input the coordinates of the target point. The tool normally takes the shortest path from the starting point to the destination point. The tool path is determined by the non-linear interpolation type positioning. Positioning is done separately with each axis. The rapid traverse is used for movements where no tools in engagement.



**Note:** The rapid traverse rate in the G00 command is set for each axis independently by the machine tool builder. Accordingly, the rapid traverse rate cannot be specified in the address F when programming.

**G01 Linear Motion**

<p><b>Description:</b> In accordance with the established standards, the instruction "straight-line at feed rate" requires the program word G01. The following <b>Supplementary Functions</b> are also needed: Target point coordinates feed rate, spindle speed or cutting speed. GO I traverses the tool along a linear path to the given target point with the feed rate input as a supplementary function. The feed rate determines the speed with which the workpiece is machined. The choice of feed rate depends on the tool (tool point geometry and tool material), the material being machined, and the required surface finish and the drive rating and rigidity of the machine tool.</p>	 <p>When giving the instructions G01, the coordinates of the destination point can be expressed using either absolute or incremental dimensions.</p>
--	--

**Note on G02/G03:**

- I0 and K0 can be omitted.
- If X (U) all Z (W) are both omitted or if the end point is located at the same position as the start point, and when the center is commanded by I & K, an arc of 360° (q complete circle) is assumed.
- If I, K & R addresses are specified simultaneously the arc specified by address R takes precedence and the others are ignored.

**G04 DWELL**

Description	Illustration
<p>A G04 causes the program to wait for a specified amount of time. The time can be specified in seconds with the "X" or "U" prefixes or in milliseconds with the "p" prefix. During cutter motion the NC controller usually applies a deceleration at the end of the motion specified by one statement and acceleration at the start of the motion specified by the next statement. A G04 code can be inserted between the two statements to make a sharp corner.</p>	<p>G04 X 1.5 G04 U 1.5 G04 P 1500</p>

**G20 Imperial:** A G20 causes position to be as being in imperial units. All the input values are in inches. This can only be at the start of the main program.

**G21 Metric:** A G21 causes positions to be interpreted as being in metric units. All the input values are in nun.

This can only be at the start of the main program.

**G28 Go to Reference Point:**

Description	Illustration
A G28 causes a fast traverse 10 the specified, position and then to the machine datum	G28 X35 Z5 G28 DO WO

**Compensate Function:**

**Description:** Tool offset is used to compensate for the difference when the tool actually used differs from the imagined tool used in programming (usually, standard tool). During programming, a four digit number is programmed following the letter T, the first two digits represent toe Tool number, and the second two digits represents the Tool offset number. Fig.18 illustrate the concept of Tool offsets

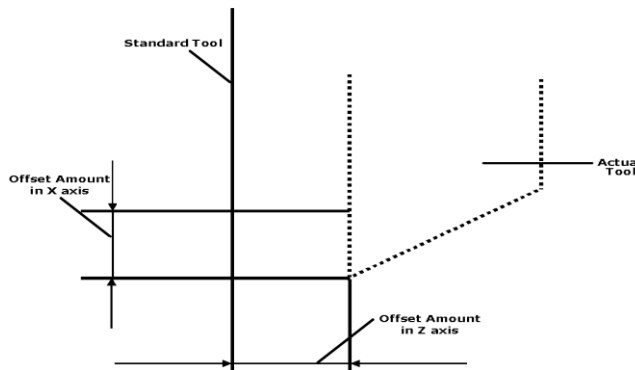


Fig. 18: Tool Offsets

**Tool Geometry Offset & Tool Wear Offset:** With the option of tool geometry and wear compensation, it is possible to divide the tool offset for compensating the tool shape or mounting position to the geometry offset, & tool wear to the wear offset. The total value of the tool geometry offset and tool wear offset are set as the tool offset value if the tool geometry and wear compensation option is not equipped Fig's. 19 & 20 illustrate the method of differentiation of tool geometry offset from tool wear offset

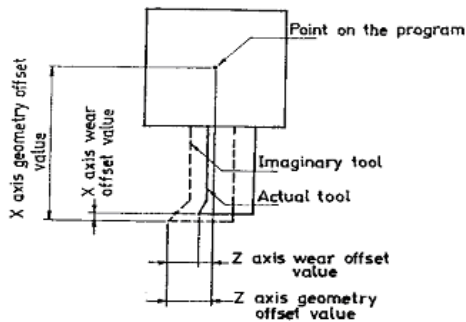


Fig. 19: Differentiate the Tool Geometry Offset from Tool Wear Offset

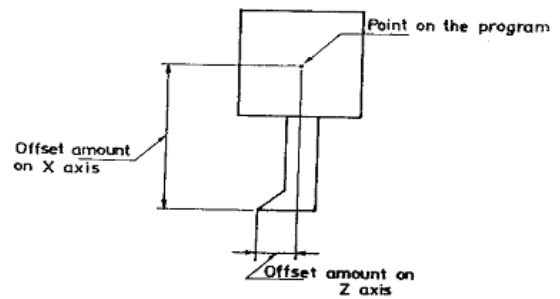


Fig. 20: Don't Differentiate the Tool Geometry Offset from Tool Wear Offset

**PREPARATORY FUNCTIONS (G CODES) For Milling**

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



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

### **MISCELLANEOUS AND PREPARATORY FUNCTIONS FOR MILLING**

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

<b>M CODES</b>	
<b>M00</b>	Program stop
<b>M01</b>	Optional stop
<b>M02</b>	Program end
<b>M03</b>	Spindle forward
<b>M04</b>	Spindle reverse
<b>M05</b>	Spindle stop
<b>M06</b>	Tool change
<b>M08</b>	Coolant on
<b>M09</b>	Coolant off
<b>M10</b>	Vice open
<b>M11</b>	Vice close
<b>M13</b>	Coolant, spindle fwd
<b>M14</b>	Coolant, spindle rev

<b>M30</b>	Program stop and rewind
<b>M70</b>	X mirror On
<b>M71</b>	Y mirror On
<b>M80</b>	X mirror off
<b>M81</b>	Y mirror off
<b>M98</b>	Subprogram call
<b>M99</b>	Subprogram exit

**Sub program:** If a program consists of sequence or frequently repeated pattern, such a sequence or Pattern can be stored as a sub program is ----- to simplify the program. A sub program can be called from the main program.

### Sub program cycle.

A Subprogram is called by the calling statement of the form  
M98 P\_\_\_ L\_\_\_\_\_ Where P = Subprogram Number L = Number of times the subprogram is called repeatedly.

## 3. PROGRAMMING CNC TURNING CENTRES

### Canned cycle for turning centers

Canned cycles are set of programmed instruction that eliminate the need for many lines of programming. Canned cycles or fixed cycles are used to simplify the programming of repetitive turning operations such as rough turning, grooving and threading.

### Rough Turning cycle G71 (for horizontal)

G71 P<sub>ns</sub> Q<sub>nf</sub> U\_\_\_ W\_\_\_ D\_\_\_ F\_\_\_ R\_\_\_\_\_

n<sub>s</sub> = Block or line number the designated the start of the part profile

n<sub>f</sub> = Block or line number the designated the end of the part profile

U = Amount of stock to be left on the X- Axis for finishing

W = Amount of stock to be left on the Z- Axis for finishing

D = Depth of cut per pass

F = Feed rate

R = Distance by which the tool will be withdrawn from the part for the return pass

## Finishing cycle G70

The general form of the finishing cycle is

G70 P<sub>ns</sub> Q<sub>nf</sub>

n<sub>s</sub> = Block or line number the designated the start of the part profile

n<sub>f</sub> = Block or line number the designated the end of the part profile

A finishing feed rate can also be included in this block. Like the roughing cycle, the tool needs to be positioned to stock boundary with Z clearance prior to the calling of the G70 finishing cycle. When the cycle G70 is terminated the tool is returned to the start point and the next block is real

## Turning cycle G72 (for vertical)

This cycle is used to cut the groove of varying widths. To use the turning cycle (vertical), the tool must be positioned to the start of the groove prior to calling. During cutting operation, the tool moves perpendicular to the axis or rotation. The general for of the turning cycle vertical is

**G72 P<sub>ns</sub> Q<sub>ns</sub> U\_\_\_\_\_ W\_\_\_\_\_ D\_\_\_\_\_ F\_\_\_\_\_ R\_\_\_\_\_**

ns = Block or line number the designates the start of the part profile

nf = Block or line number the designates the end of the part profile

U = Amount of stock to be left on the X- Axis for finishing

W = Amount of stock to be left on the Z- Axis for finishing

D = Depth of cut per pass

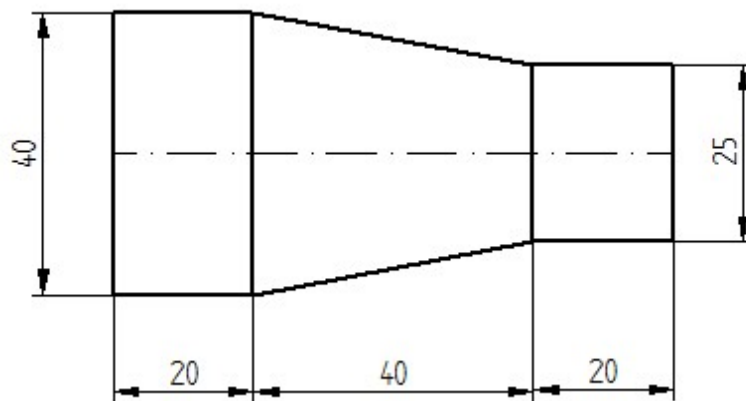
F = Feed rate

R = Distance by which the tool will be with drawn from the part for the return pass

## Chapter-3

**CAM - EXERCISES****TURNING EXERCISE****EXERCISE - 1**

Write the NC part program to turn the work piece as shown in figure with the use FANUC controller. The diameter of the stock is 50mm.



% TURNING

O001

N10 G90 G21

N20 T01

N30 G27 X100 Z100

N40 G96 G94 S100 M03 M08 F10

N50 G00 X50 Z5

N60 G71 P70 Q100 U1.0 W1.0 D1.0 R2.0

N70 G00 X25

N80 G01 Z-20

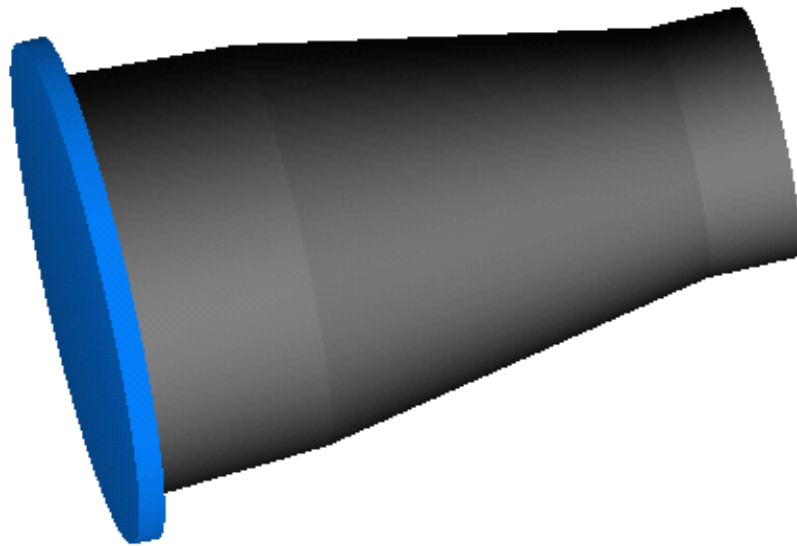
N90 X40 Z-60

N100 Z-80

N110 G70 P70 Q100 S200 F5.0

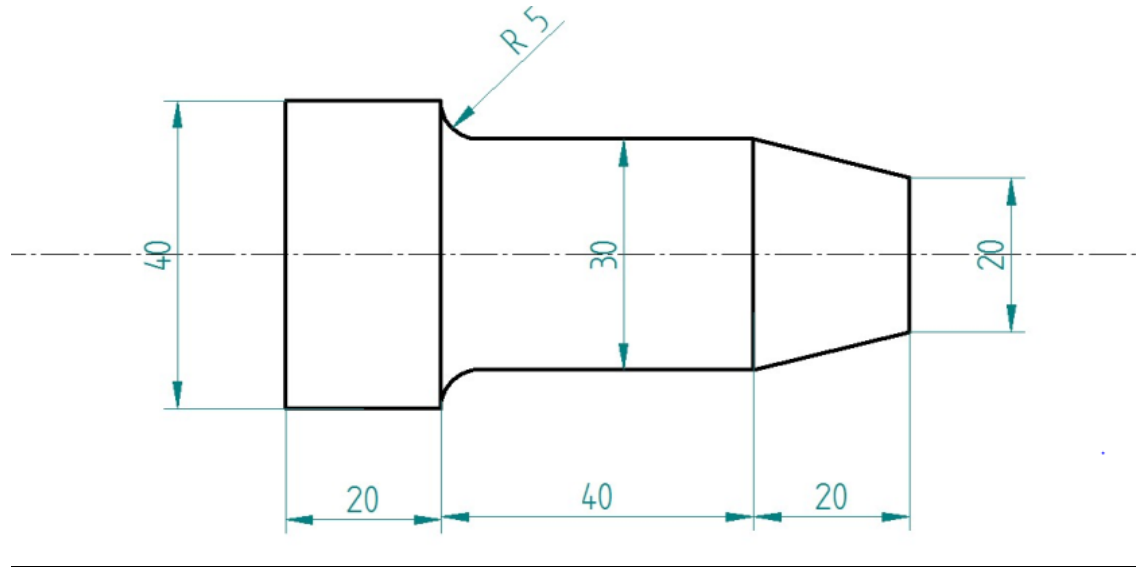
N120 G27 X100 Z100

N130 M30



**EXERCISE -2**

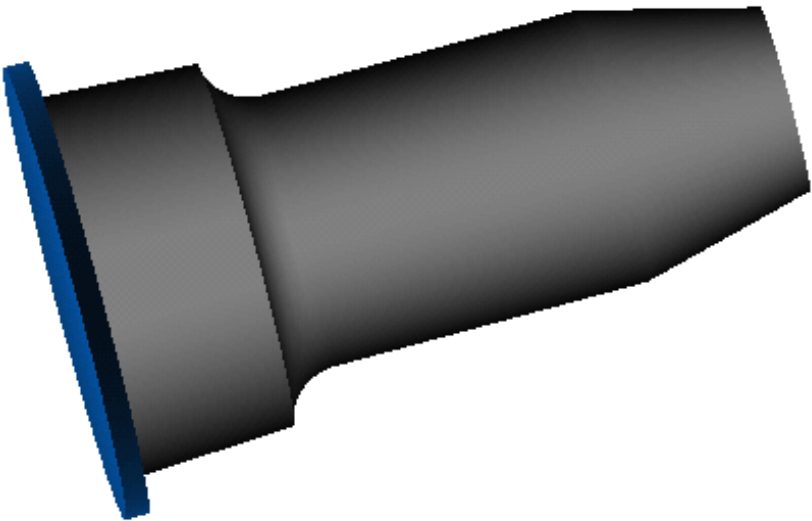
Write the NC part program to turn the work piece as shown in figure with the use FANUC controller.  
The diameter of the stock is 50mm



```

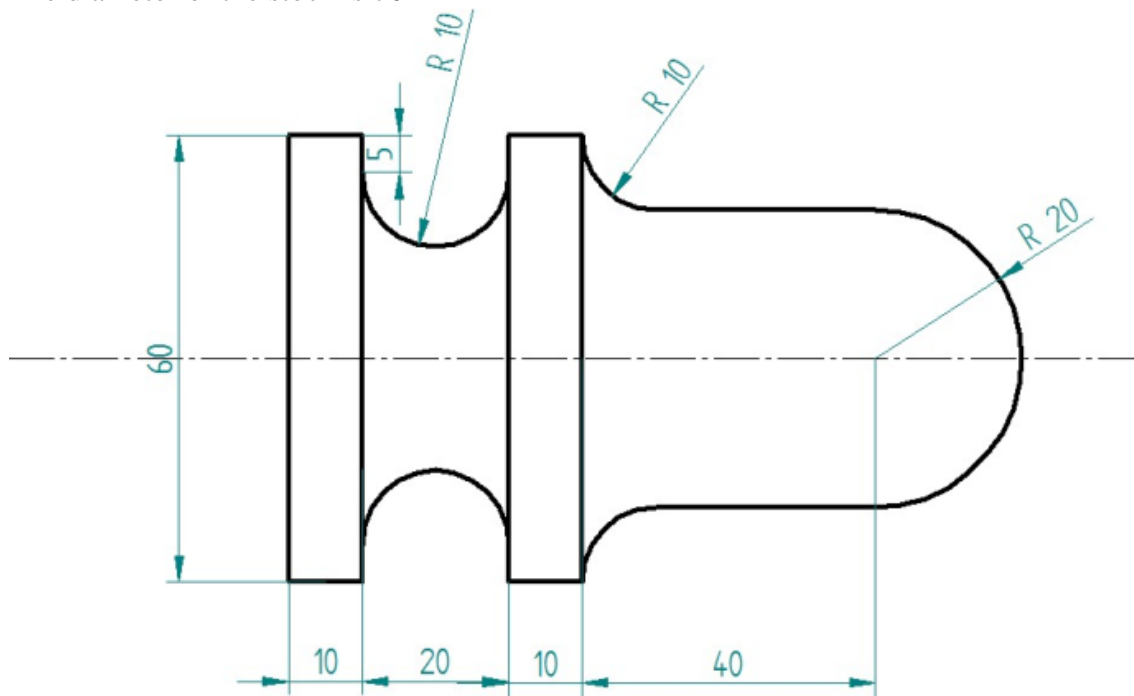
% TURNING
0002
N10 G90 G21
N20 T01
N30 G27 X100 Z100
N40 G96 G94 S100 M03 M08 F10
N50 G00 X50 Z5
N60 G71 P70 Q120 U1.0 W1.0 D1.0 R1.0
N70 G00 X20
N80 G01 Z0
N90 X30 Z-20
N100 Z-55
N110 G02 X40 Z-60 I5 K0
N120 G01 Z-80
N130 G70 P70 Q120 S200 F5.0
N140 G27 X100 Z100
N150 M30

```



**EXERCISE -3**

Write the NC part program to turn the work piece as shown in figure with the use FANUC controller.  
The diameter of the stock is 70mm



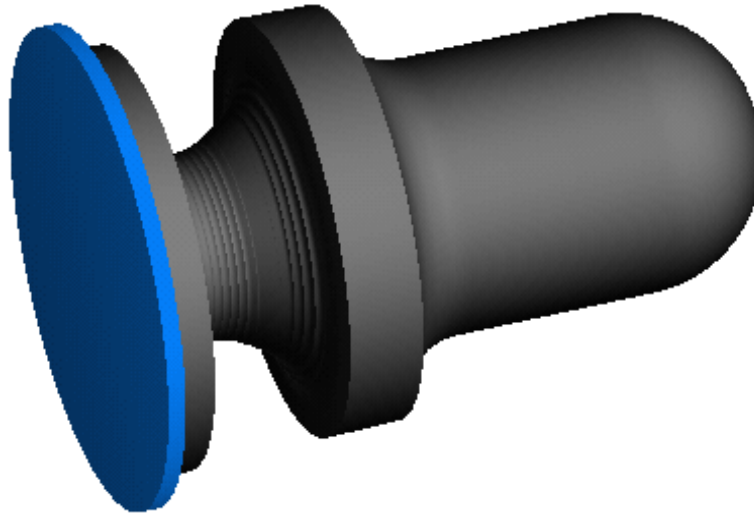
```

O003
N10 G90 G21
N20 T01
N30 G27 X100 Z100
N40 G96 G94 S100 M03 M08 F10
N50 G00 X70 Z5
N60 G71 P70 Q120 U1.0 W1.0 D1.0 R2.0
N70 G00 X0
N80 G01 Z0
N90 G03 X40 Z-20 I0 K-20
N100 G01 Z-50
N110 G02 X60 Z-60 I10 K0
N120 G01 Z-100
N130 G70 P70 Q120 S200 F5.0
N140 G27 X100 Z100
N150 T02

```



```
N160 G00 X70 Z-74
N170 G72 P180 Q200 U0.0 W0.0 D1.0 R3.0
N180 G01 X50
N190 G02 X50 Z-90 I0 K-10
N200 G01 X60
N210 G97 X100 Z100
N220 M30
```



### Grooving Cycle (G87)

The Grooving cycle is used to cut grooving of varying width. To use the grooving cycle, the tool must be positioned to the start the groove prior to calling the grooving cycle. The general form of grooving cycle is

**G87 R\_\_ X\_\_ Z\_\_ U\_\_ W\_\_ P\_\_ Q\_\_ F\_\_**

R = Return amount in the first line

X = Diameter at the bottom of the groove

Z = End position of the groove in Z- Direction

U = Amount of the stock to be left on the X- Axis for finishing

W = Amount of stock to be left on the Z- Axis for finishing

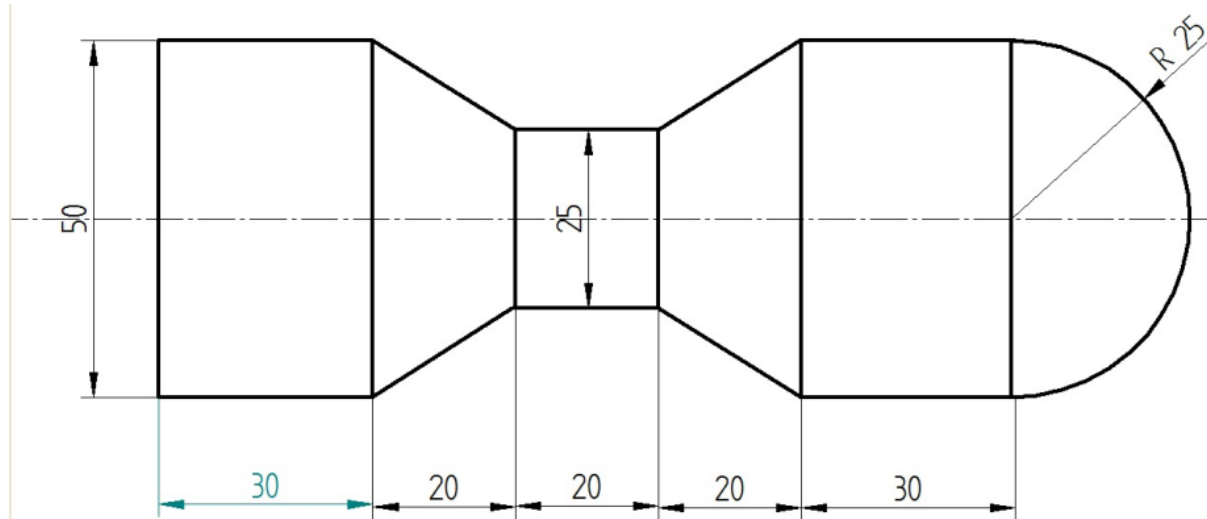
P = Depth of cut on the X-Axis

Q = Depth of cut on the z-Axis

F = Feed rate

**EXERCISE -4**

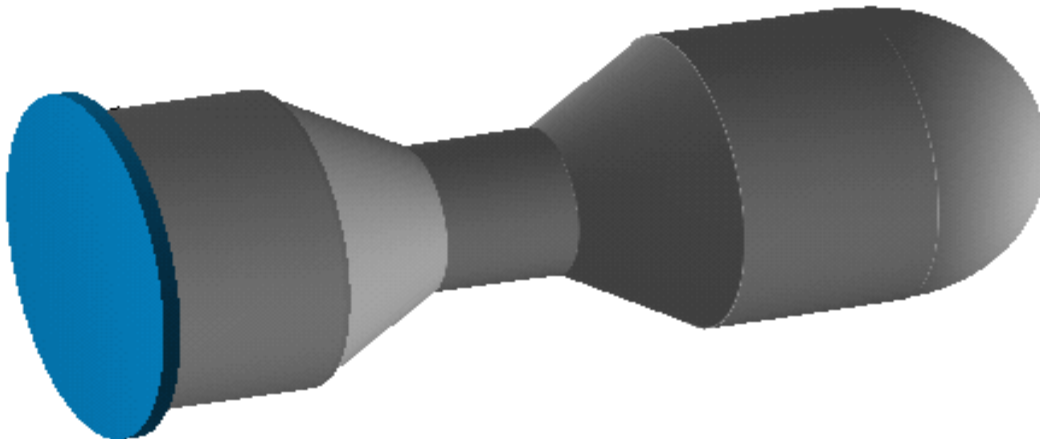
Write the NC part program to turn the work piece as shown in figure with the use FANUC controller. The diameter of the stock is 60mm



```

O005
N10 G90 G21
N20 T01
N30 G27 X100 Z100
N40 G96 G94 S100 M03 M08 F10.0
N50 G00 X60 Z5
N60 G71 P70 Q100 U1.0 W1.0 D1.0 R1.0
N70 G00 X0
N80 G01 Z0
N90 G03 X50 Z-25 T0 K-25
N100 G01 Z-145
N110 G70 P70 Q100 S200 F50
N120 G27 X100 Z100
N130 T02
N140 G00 X60 Z-55
N150 G87 R2 X25 Z-115 U0 W0 P1 Q1 F5
N160 G01 X50
N170 X25 Z-75
N180 Z-95
N190 X50 Z-115
N200 G27 X100 S200
N210 M30

```



### Thread Cutting Cycle (G76)

The G76 thread cutting cycle can cut multi pass threads with one block of information. The control will automatically calculate the correct number of passes. The general form of the thread cutting cycle is,

**G76 X\_\_ z\_\_ F\_\_ P\_\_ Q\_\_ R\_\_**

X = Minor diameter of the thread  $d_1$

Z = Absolute Z- Position of the end of the thread

F = Thread lead = pitch  $p$  for single start thread =  $2p$  for double start thread

P = Total thread height or total depth of cut expressed in radius value  
 $= (d-d_1)/2$

Q = Depth of cut in first pass in the radius value

R = Thread finishing allowance in radius value

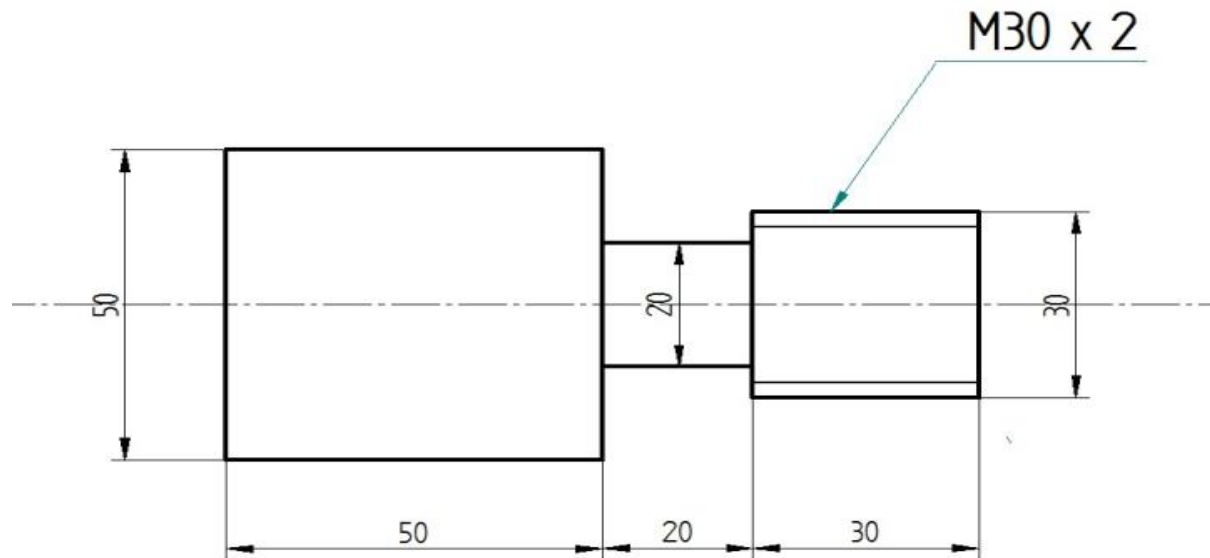
Thread angle  $\alpha = 60^\circ$

Thread Height  $h = 0.86603p$

Minor diameter  $d_1 = \text{Major diameter } d - 2 \times \text{thread height } h = d - 2h$

**EXERCISE -5**

Write the NC part program to turn the work piece as shown in figure with the use FANUC controller. The diameter of the stock is 60mm



```

O0012
N10 G90 G21
N20 T01
N30 G27 X100 Z100
N40 G96 G94 S100 M03 M08 F10.0
N50 G00 X60 Z5
N60 G71 P70 Q100 U1.0 W1.0 D1.0 R2.0
N70 G00 X30
N80 G01 Z-50
N90 X50
N100 Z-100
N110 G70 P70 Q100 S200 F5.0
N120 G27 X100 Z100
N130 T02
N140 G00 X60 Z-34
N150 G72 P160 Q180 U0.0 W0.0 D1.0 R2.0
N160 G01 X20
N170 Z-50.0
N180 X50
N190 G27 X100 Z100
N200 G97
N210 T03

```

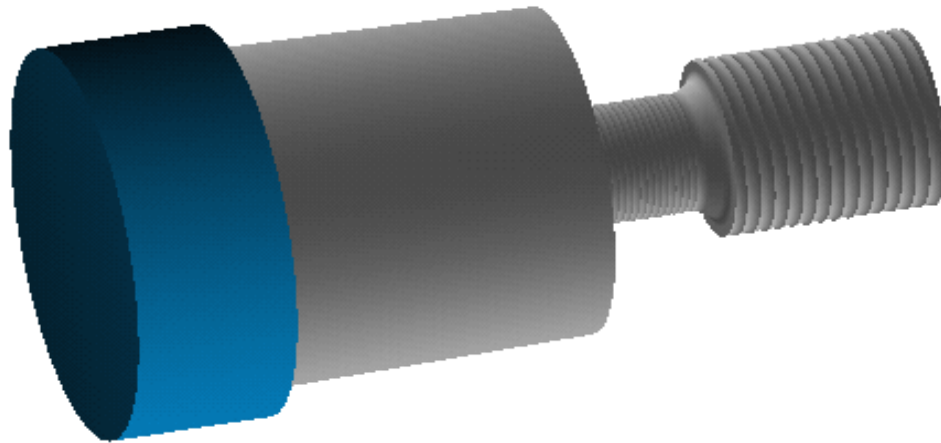
N220 M03 S100

N230 G00 X30 Z5

N240 G76 X26.536 Z-35 F2.0 P1.732 Q1.0 R0.0

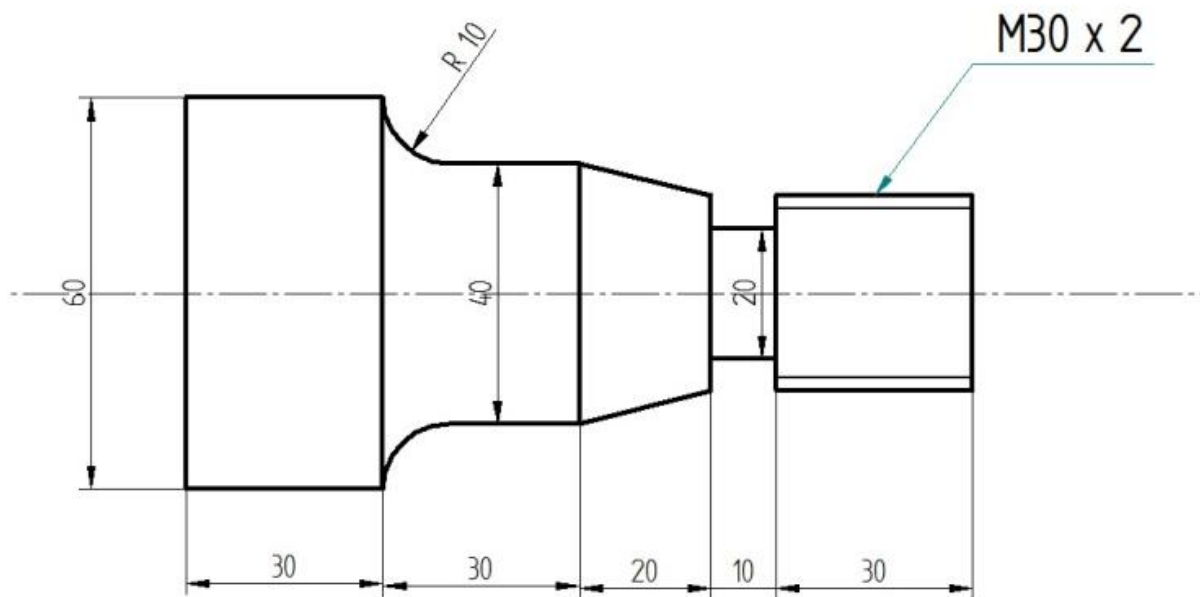
N250 G27 X100 Z100

N260 M30



**EXERCISE -6**

Write the NC part program to turn the work piece as shown in figure with the use FANUC controller.  
The diameter of the stock is 70mm



```

N10 G90 G21
N20 T01
N30 G27 X100 Z100
N40 G96 G94 M03 S100 M08 F10.0
N50 G00 X70 Z5
N60 G71 P70 Q120 U1.0 W1.0 D1.0 R2.0
N70 G00 X40
N80 G01 Z-35
N90 G02 X45 Z-40 I5 K0
N100 G01 X65
N110 G03 X70 Z-45 I0 K-5
N120 G01 Z-80
N130 G70 P70 Q120 S200 F5.0
N140 G27 X100 Z100
N150 G97
N160 T02
N170 M03 S200
N180 G00 X0.0 Z5
N190 G83 X0.0 Z-20.0 Q5.0 R5.0 L4 P10 F10

```

N200 G27 X100 Z100

N210 M30



### Peck Drilling Cycle G83

The peck drilling cycle will peck drill holes with automatic retract and incremental depth of cut. The general form of the peck drilling cycle is

**G83 X\_\_ Z\_\_ Q\_\_ R\_\_ L\_\_ P\_\_ F\_\_**

**X** = 0.0, Which is the centre of the work piece

**Z** = Full depth of the drilled hole (real)

**Q** = Peck increment (Depth of each peck)

**R** = Peck retract

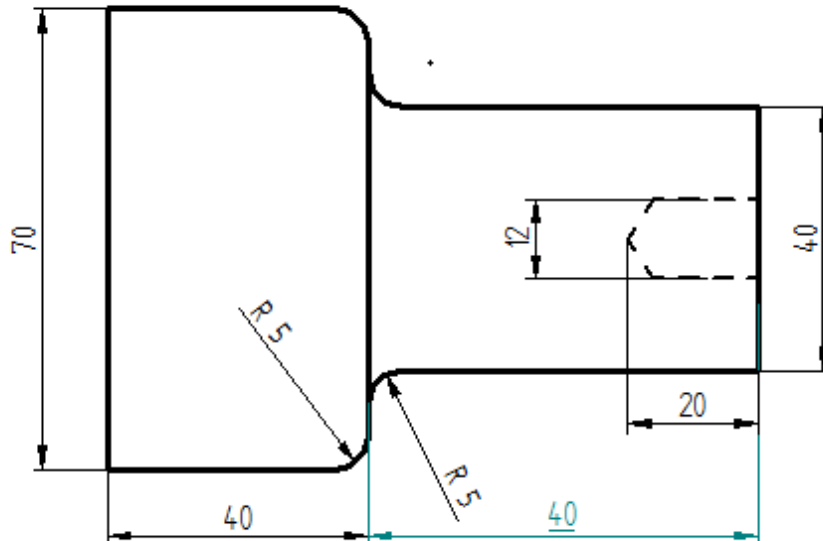
**L** = Peck Repeats in integer

**P** = me Dwell time at the bottom of the hole

**F** = Feed rate

**EXERCISE -7**

Write NC manual part program to turn the work piece as shown in fig. Then drill a 12 mm diameter hole at the center to a depth of 20mm. Take the diameter as 80mm.

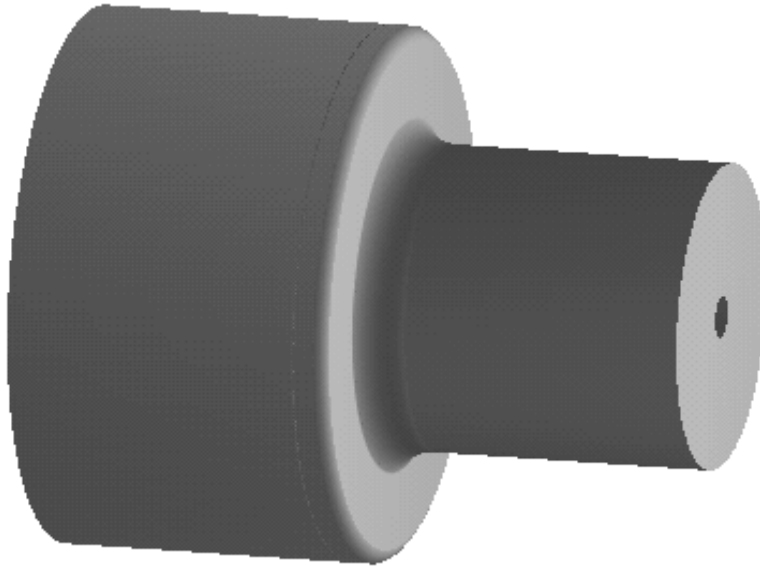


```

N10 G90 G21
N20 T01
N30 G27 X100 Z100
N40 G96 G94 M03 S100 M08 F10.0
N50 G00 X70 Z5
N60 G71 P70 Q120 U1.0 W1.0 D1.0 R2.0
N70 G00 X40
N80 G01 Z-35
N90 G02 X45 Z-40 I5 K0
N100 G01 X65
N110 G03 X70 Z-45 I0 K-5
N120 G01 Z-80
N130 G70 P70 Q120 S200 F5.0
N140 G27 X100 Z100
N150 G97
N160 T02
N170 M03 S200
N180 G00 X0.0 Z5
N190 G83 X0.0 Z-20.0 Q5.0 R5.0 L4 P10 F10
N200 G27 X100 Z100
N210 M30

```





### Drill Cycle

The most commonly used cycle to drill a hole is G81 canned drilling cycle. The Z position is very important when we call the canned cycle. The machine will normally rapid back to the Z initial position before a rapid to the next hole. The general form of the drill cycle is

**G81 X\_\_ Y\_\_ Z\_\_ R\_\_ F\_\_**

X = X- Coordinate of the hole

Y = Y- Coordinate of the hole

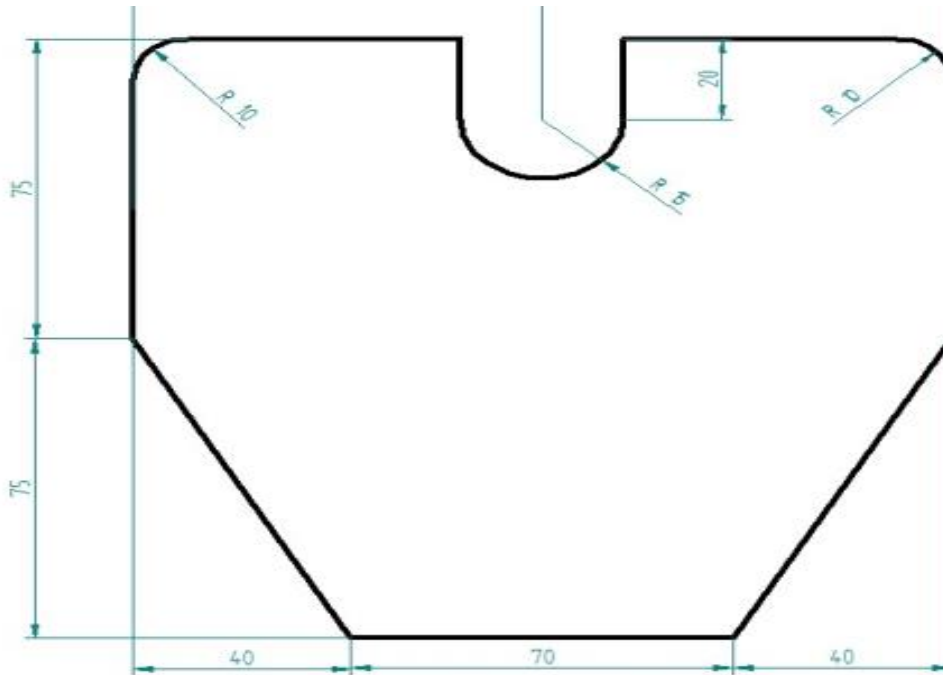
Z = Depth of the hole

R = Reference plane position above the Z<sub>0</sub>

F = Feed rate of the tool

**MILLING EXERCISE****EXERCISE -1**

Write the NC program to mill the profile of the part shown in figure. The stock is 160X160X10mm.



```

N10 G90      G21
N20 T01 M06
N30 G97      G94  S1000 M03  M08  F10
N40 G00      X-20  Y-20  Z20
N50 M98      P333  L6
N60 G00      X-20  Y-20  Z20
N70 M30
O333
N80 G91 G01 Z-4 R5 F10
N90 G90      G01  X0    Y75
N100         X0    Y140
N110 G02  X10  Y150 I10  J0
N120 G01  X60  Y150
N130         X60  Y130
N140 G03  X90  Y130 I15  J0
N150 G01  X90  Y150
N160         X140 Y150

```

```
N170 G02 X150 Y140 I0 J-10  
N180 G01 X150 Y75  
N190 X110 Y0  
N200 X40 Y0  
N210 X0 Y75  
N220 M99
```



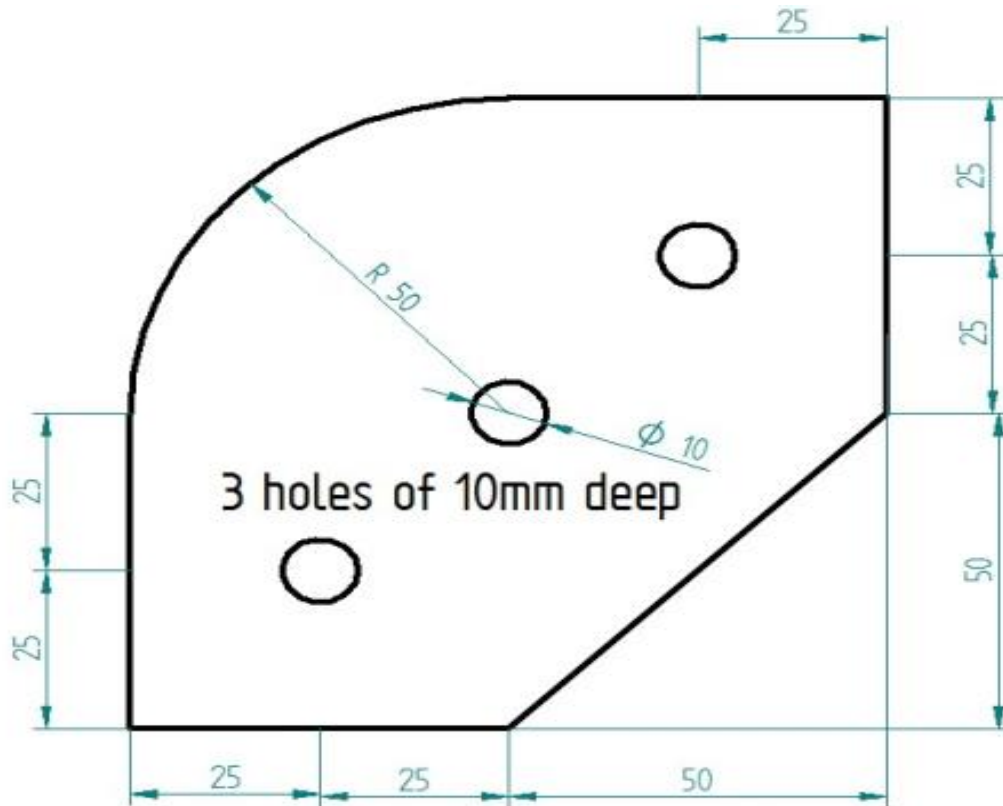
**EXERCISE -2**

Write the NC program to perform the following operations.

(i) Mill the profile 10mm deep.

(ii) Drill 10mm diameter holes at 3 places as shown in figure.

The stock is 160X160X10mm.



```

N10 G90      G21
N20 T01      M06
N30 G97      G94  S100  F10   M03  M08
N40 G00      X-20  Y-20  Z5
N50 M98      P222  L6
N60 G00      X-20  Y-20  Z20
N70 T02      M06
N80 G81      X25   Y25   Z-15  R5   F10
N90  X50     Y50
N100 X75     Y75
N110 G80
N120 G00     X-20  Y-20  Z20
N130 M30
  
```

**O222**

```
N140 G91 G01 Z-4 R5 F10
```

```
N150 G90 G01 X0 Y0
```

```
N160 X0 Y50
```

```
N170 G02 X50 Y100 I50 J0
```

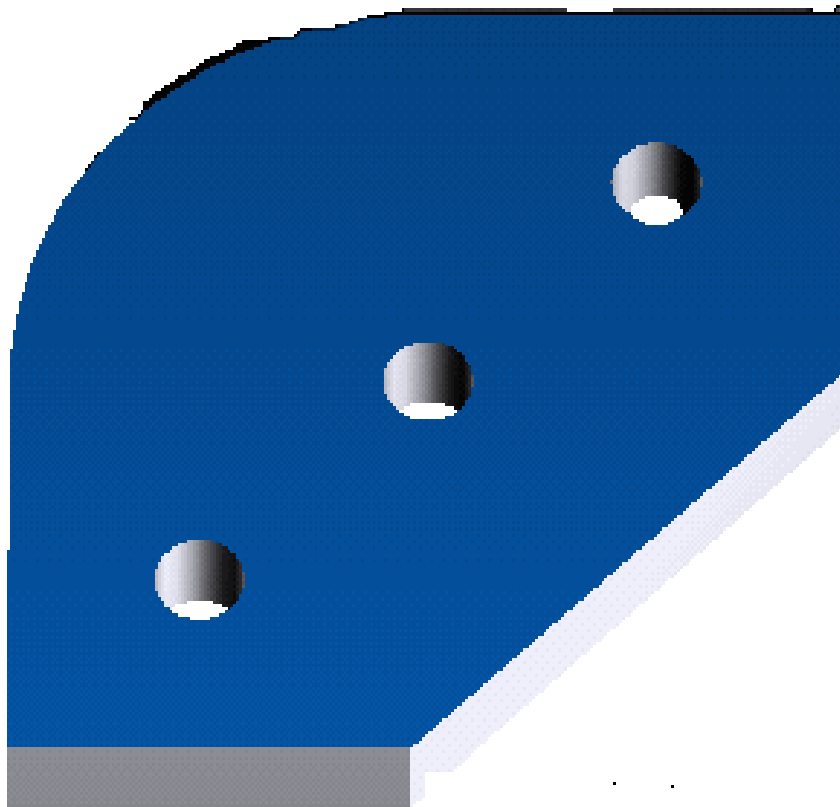
```
N180 G01 X100 Y100
```

```
N190 G01 X100 Y50
```

```
N200 G01 X50 Y0
```

```
N210 G01 X0 Y0
```

```
N220 M99
```

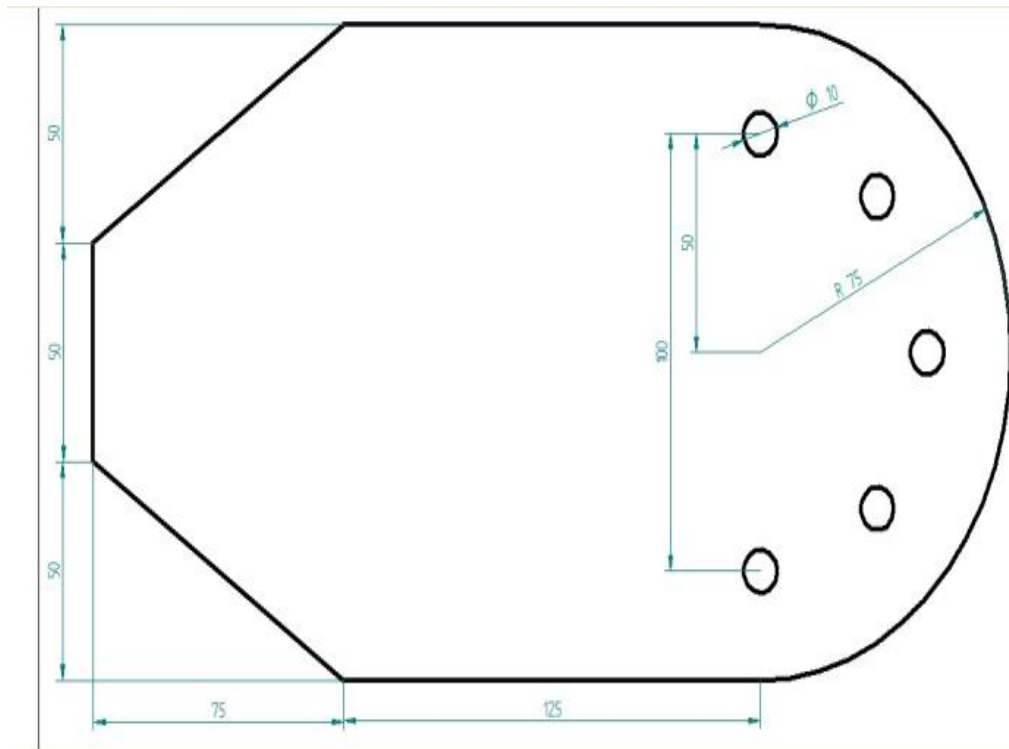


**EXERCISE -3**

(3) Write the NC program to perform the following operations.

(i) Mill the profile 10mm deep.

(ii) Drill 10mm diameter holes at 5 places as shown in figure. Take stock size as 285X160X10mm. 5 Holes 10mm deep.



```

N10 G90      G21
N20 T01      M06
N30 G97      G94  S100  M03  M08
N40 G00      X-20  Y-20  Z5
N50 M98      P123  L6
N60 G00      X-20  Y-20  Z20
N70 T02      M06
N80 G81      X200  Y25   Z-15  R5   F10
N90  X200  Y125
N100 X235.35  Y42.677
N110 X235.35  Y 107.323
N120 X250  Y75
N130 G80
N140 G00      X-20  Y-20  Z20
  
```

N150M30

O123

N160 G91 G01 Z-4 R5 F10

N170 G90 G01 X75 Y0

N180 X0 Y50

N190 Y100

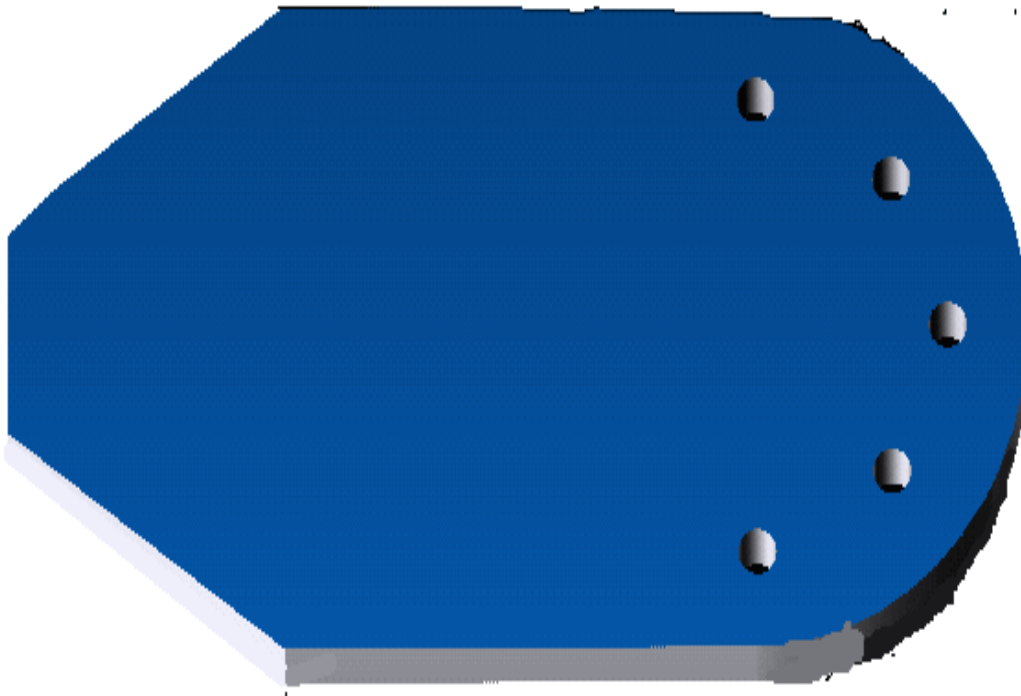
N200 X75 Y150

N210 X200

N220 G02 Y0 J-75

N230 G01 X75

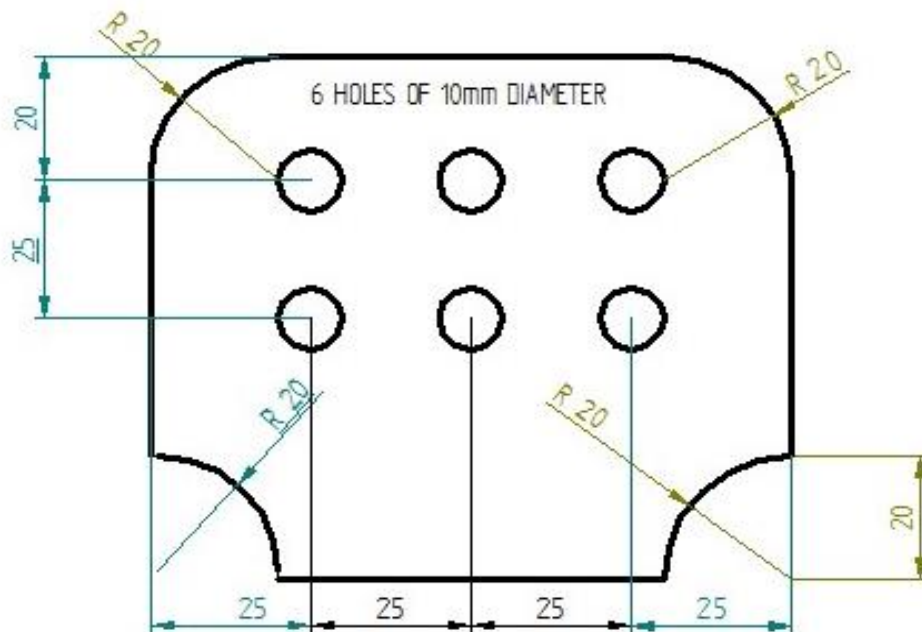
N240 M99



**EXERCISE -4**

Write the NC program to perform the following operations.

- (i) Mill the profile 10mm deep.
- (ii) Drill 10mm diameter holes at 6 places as shown in figure.



```

N10 G90 G21
N20 T01 M06
N30 G97 G94 S100 M03 M08 F10
N40 G00 X-20 Y-20 Z5
N50 M98 P333 L6
N60 G00 X-20 Y-20 Z20
N70 T02 M06
N80 G81 X25 Y30 Z-12 R2 F10
N90 X50 Y30
N100 X75 Y30
N110 X75 Y55
N120 X50 Y55
N130 X25 Y55
N140 G80
N150 G00 X-20 Y-20 Z20
N160 M30

```



O333

N170 G91 G01 X0 Y0 Z-2 F40

N180 G90 G01 X20 Y0

N190 G03 X0 Y20 I-20 J0

N200 G01 X0 Y55

N210 G02 X20 Y75 I20 J0

N220 G01 X80 Y75

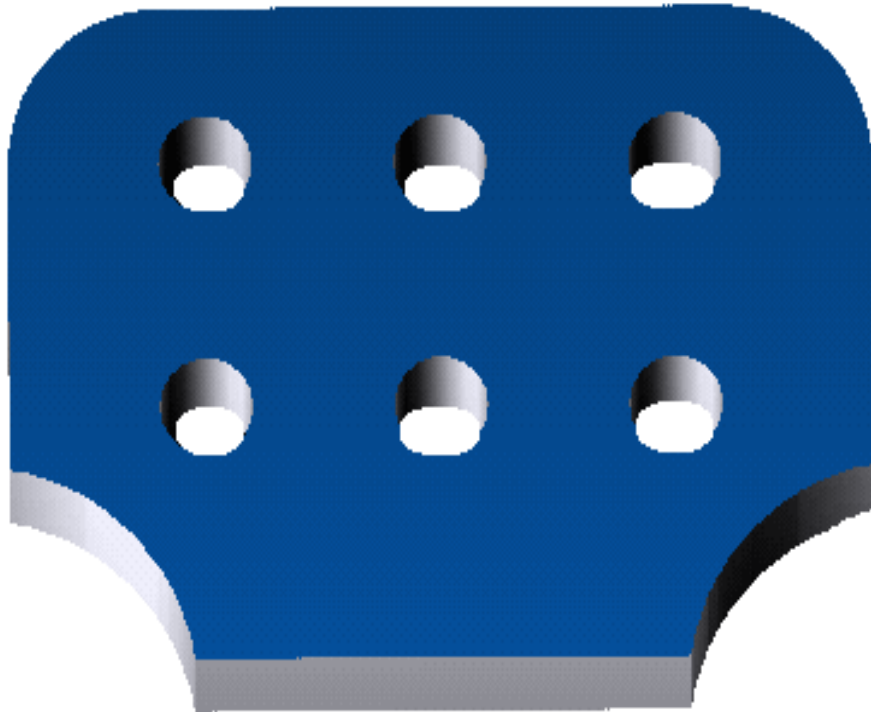
N 230 G02 X100 Y55 I0 J-20

N240 G01 X100 Y20

N250 G03 X80 Y0 J-20

N260 G01 X20 Y0

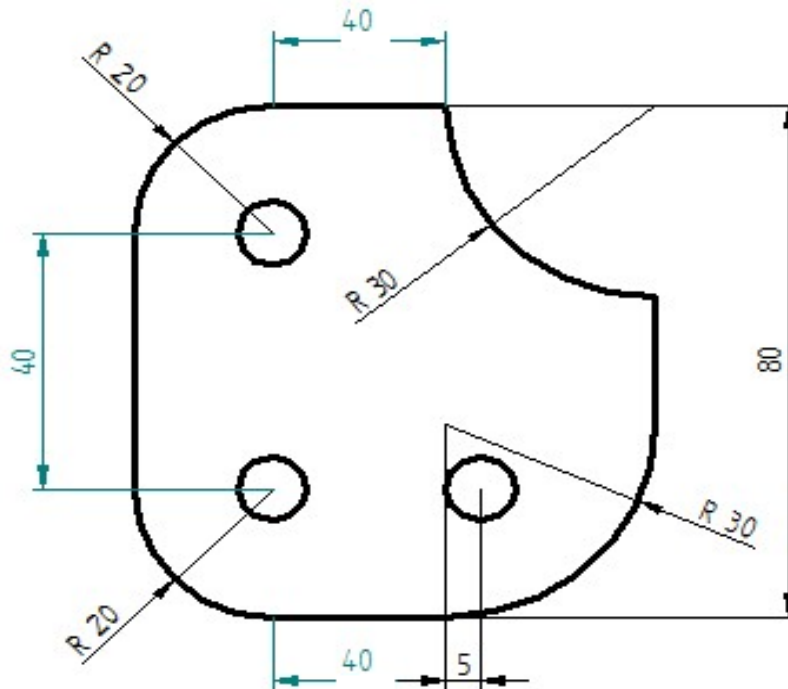
N270 M99



**EXERCISE -5**

Write the NC program to perform the following operations.

- (i) Mill the profile 10mm deep.
- (ii) Drill 10mm diameter holes at 3 places as shown in figure.



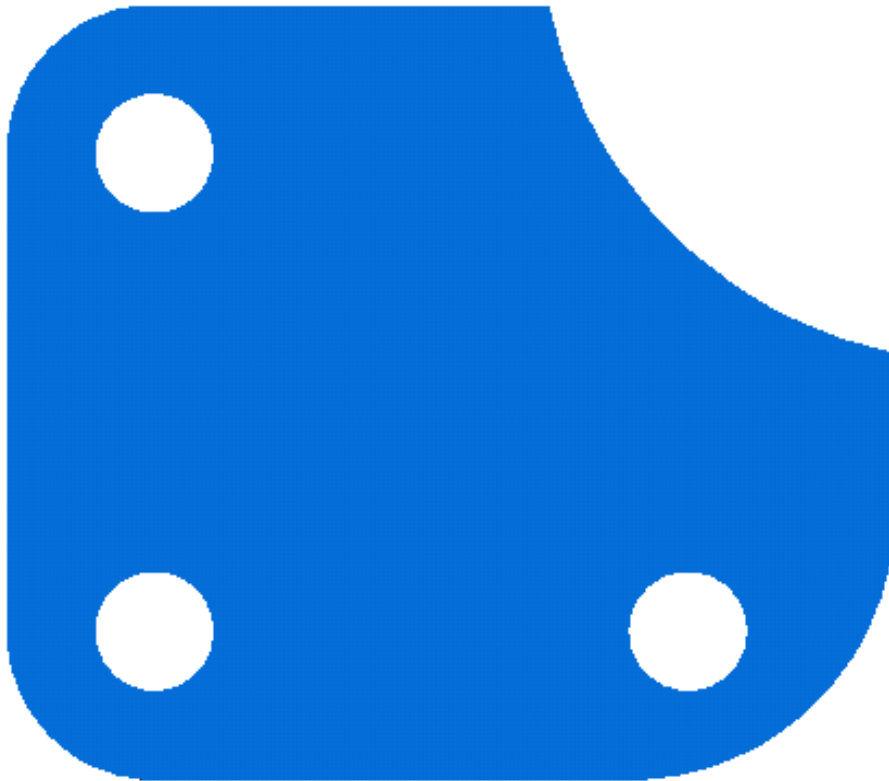
```

N10 G90 G21
N20 T01 M06
N30 G97 G94 S1000 M03 M08 F10
N40 G00 X-20 Y-20 Z5
N50 M98 P333 L6
N60 G00 X-20 Y-20 Z20
N70 T02 M06
N80 G81 X20 Y20 Z-12 R2 F10
N90 X65 Y20

N100 X20 Y60
N110 G80
N120 G00 X-20 Y-20 Z20
N130 M30
O333
N140 G91 G01 Z-2 F10
N150 G90 G01 X20
N160 G02 X0 Y20 I0 J20
N170 G01 X0 Y60
N180 G02 X20 Y80 I20 J0

```

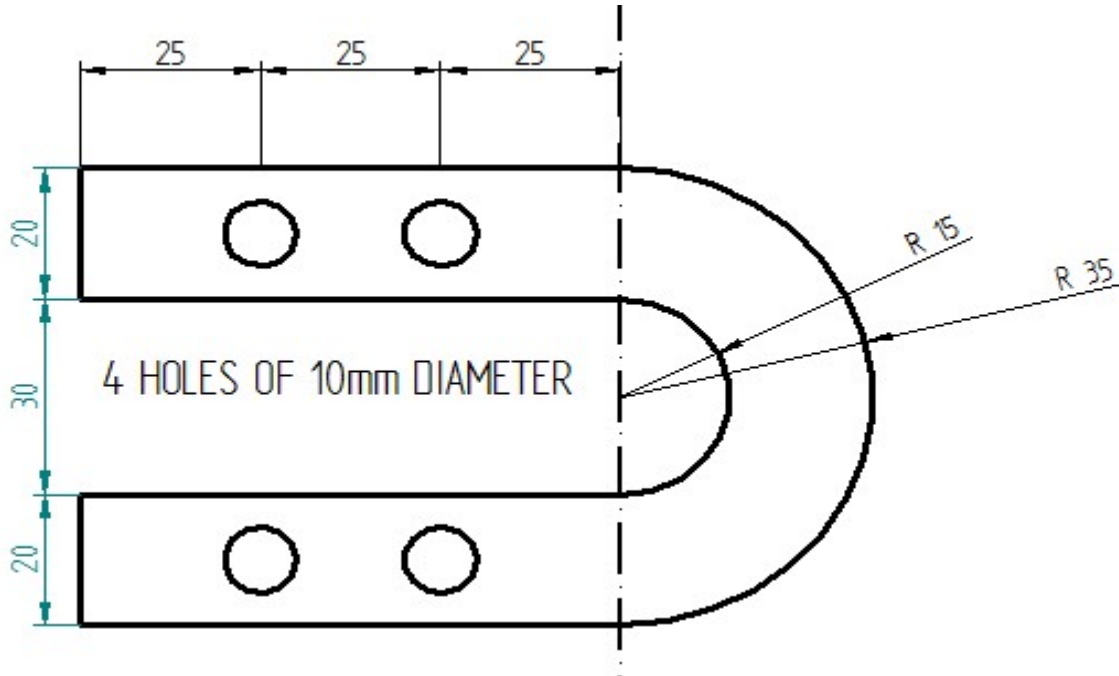
```
N190 G01 X60 Y80
N200 G03 X90 Y50 I30 J0
N210 G01 X90 Y30
N220 G02 X60 Y0 I-30
N230 G01 X20 Y0
N240 M99
```



**EXERCISE -6**

Write the NC program to perform the following operations.

- (i) Mill the profile 10mm deep.
- (ii) Drill 10mm diameter holes at 4 places as shown in figure.

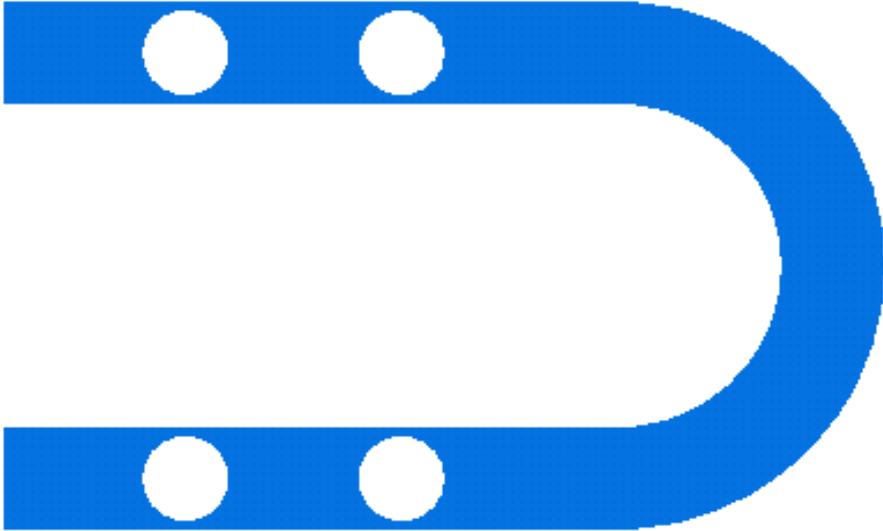


```

N10 G90 G21
N20 T01 M06
N30 G97 G94 S1000 M03 M08 F10
N40 G00 X-20 Y-20 Z5
N50 M98 P333 L6
N60 G00 X-20 Y-20 Z20
N70 T02 M06
N80 G81 X25 Y10 Z-15 R2 F10
N90 X50 Y10
N100 X50 Y60
N110 X25 Y60
N120 G80
N130 G00 X-20 Y-20 Z20
N140 M30
O333
N150 G91 G01 Z-5 F10
N160 G90 G01 X75 Y0
N170 G03 X75 Y70 J35
N180 G01 X0 Y70
N190 G01 X0 Y50
N200 G01 X75 Y50
N210 G02 X75 Y20 J-25
N220 G01 X0 Y20
N230 G01 X0 Y0

```

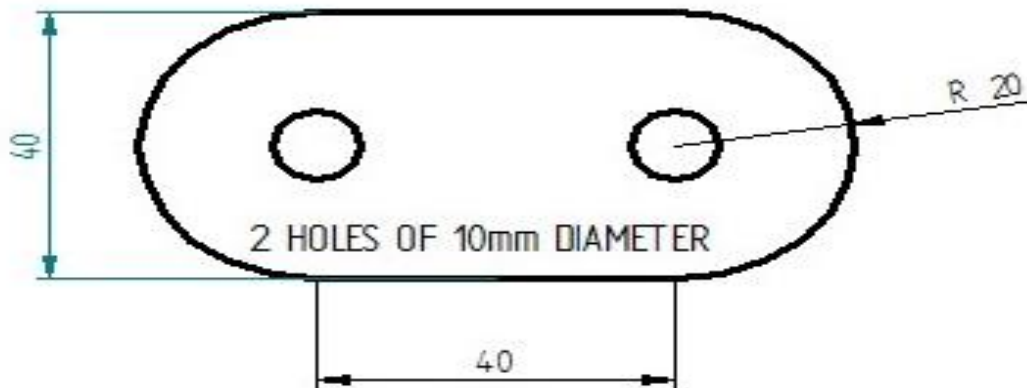
N240 M99



**EXERCISE -7**

Write the NC program to perform the following operations.

- (i) Mill the profile 10mm deep.
- (ii) Drill 10mm diameter holes at 2 places as shown in figure.



```

N10 G90 G21
N20 T01 M06
N30 G97 G94 S1000 M03 M08 F10
N40 G00 X-20 Y-20 Z10
N50 M98 P333 L6
N60 T02 M06
N70 G81 X20 Y20 Z-12 R2 F10
N80 X60 Y20
N90 M30
N100 G90
N110 G00 X-20 Y-20 Z20
N120
O333
N130 G91 G01 Z-10F10
N140 G90 G01 X20
N150 G02 X20 Y40 J20 I0
N160 G01 X60
N170 G02 X60 Y0 J-20 I0
N180 G01 X20
N190 M99

```



## Chapter-5

### FLEXIBLE MANUFACTURING SYSTEMS (FMS)

#### **Introduction**

In the middle of the 1960s, market competition became more intense. During 1960 to 1970 cost was the primary concern. Later quality became a priority. As the market became more and more complex, speed of delivery became something customer also needed. A new strategy was formulated: Customizability. The companies have to adapt to the environment in which they operate, to be more flexible in their operations and to satisfy different market segments (customizability). Thus the innovation of FMS became related to the effort of gaining competitive advantage. First of all, FMS is a manufacturing technology.

Secondly, FMS is a philosophy. "System" is the key word. Philosophically, FMS incorporates a system view of manufacturing. The buzz word for today's manufacturer is "agility". An agile manufacturer is one who is the fastest to the market, operates with the lowest total cost and has the greatest ability to "delight" its customers. FMS is simply one way that manufacturers are able to achieve this agility.

An MIT study on competitiveness pointed out those American companies spent twice as much on product innovation as they did on process innovation. Germans and Japanese did just the opposite. In studying FMS, we need to keep in mind what Peter Drucker said: "We must become managers of technology not merely users of technology". Since FMS is a technology, well adjusted to the environmental needs, we have to manage it successfully.

#### 1. Flexibility concept. Different approaches

Today flexibility means to produce reasonably priced customized products of high quality that can be quickly delivered to customers. Different approaches to flexibility and their meanings are shown Table 1. So, what is flexibility in manufacturing? While variations abound in what specifically constitutes flexibility, there is a general consensus about the core elements. There are three levels of manufacturing flexibility



Approach	Flexibility meaning
Manufacturing	<ul style="list-style-type: none"> <li>• The capability of producing different parts without major retooling</li> <li>• A measure of how fast the company converts its process (es) from making an old line of products to produce a new product</li> <li>• The ability to change a production schedule, to modify a part, or to handle multiple parts</li> </ul>
Operational	<ul style="list-style-type: none"> <li>• The ability to efficiently produce highly customized and unique products</li> </ul>
Customer	<ul style="list-style-type: none"> <li>• The ability to exploit various dimension of speed of delivery</li> </ul>
Strategic	<ul style="list-style-type: none"> <li>• The ability of a company to offer a wide variety of products to its customers</li> </ul>
Capacity	<ul style="list-style-type: none"> <li>• The ability to rapidly increase or decrease production levels or to shift capacity quickly from one product or service to another</li> </ul>

#### (a) Basic flexibilities

- Machine flexibility - the ease with which a machine can process various operations
- Material handling flexibility - a measure of the ease with which different part types can be transported and properly positioned at the various machine tools in a system
- Operation flexibility - a measure of the ease with which alternative operation sequences can be used for processing a part type

#### (b) System flexibilities

- Volume flexibility - a measure of a system's capability to be operated profitably at different volumes of the existing part types
- Expansion flexibility - the ability to build a system and expand it incrementally
- Routing flexibility - a measure of the alternative paths that a part can effectively follow through a system for a given process plan
- Process flexibility - a measure of the volume of the set of part types that a system can produce without incurring any setup

- Product flexibility - the volume of the set of part types that can be manufactured in a system with minor setup

(c) Aggregate flexibilities

- Program flexibility - the ability of a system to run for reasonably long periods without external intervention
- Production flexibility - the volume of the set of part types that a system can produce without major investment in capital equipment
- Market flexibility - the ability of a system to efficiently adapt to changing market conditions

## 2. Seeking benefits on flexibility

Today's manufacturing strategy is to seek benefits from flexibility. This is only feasible when a production system is under complete control of FMS technology. Having in mind the Process-Product Matrix you may realize that for an industry it is possible to reach for high flexibility by making innovative technical and organizational efforts. See the Volvo's process structure that makes cars on movable pallets, rather than an assembly line. The process gains in flexibility. Also, the Volvo system has more flexibility because it uses multi-skill operators who are not paced by a mechanical line.

So we may search for benefits from flexibility on moving to the job shop structures. Actually, the need is for flexible processes to permit rapid low cost switching from one product line to another. This is possible with flexible workers whose multiple skills would develop the ability to switch easily from one kind of task to another. As main resources, flexible processes and flexible workers would create flexible plants as plants which can adapt to changes in real time, using movable equipment, knockdown walls and easily accessible and re-routable utilities.

## 3. FMS- an example of technology and an alternative layout

The idea of an FMS was proposed in England (1960s) under the name "System 24", a flexible machining system that could operate without human operators 24 hours a day under computer control. From the beginning the emphasis was on automation rather than the "reorganization of workflow". Early FMSs were large and very complex, consisting of dozens of Computer Numerical Controlled machines (CNC) and sophisticated material handling systems. They were very automated, very expensive and controlled by incredibly complex software. There were only a limited number of industries that could afford investing in a traditional FMS as described above. Currently, the trend in FMS is toward small versions of the traditional FMS, called flexible manufacturing cells (FMC). Today two or more CNC machines are considered a flexible cell and two or more cells are considered a flexible manufacturing system.

Thus, a Flexible Manufacturing System (FMS) consists of several machine tools along with part and tool handling devices such as robots, arranged so that it can handle any family of parts for which it has been designed and developed.

Different FMSs levels are: Flexible Manufacturing Module (FMM).

Example : a NC machine, a pallet changer and a part buffer;

Flexible Manufacturing (Assembly) Cell (F(M/A)C).

Example : Four FMMs and an AGV(automated guided vehicle);

Flexible Manufacturing Group (FMG).

Example : Two FMCs, a FMM and two AGVs which will transport parts from a Part Loading area, through machines, to a Part Unloading Area;

Flexible Production Systems (FPS).

Example : A FMG and a FAC, two AGVs, an Automated Tool Storage, and an Automated Part/assembly Storage;

Flexible Manufacturing Line (FML).

Example : multiple stations in a line layout and AGVs.

#### 4. Advantages and disadvantages of FMSs implementation

##### Advantages

- Faster, lower- cost changes from one part to another which will improve capital utilization
- Lower direct labor cost, due to the reduction in number of workers
- Reduced inventory, due to the planning and programming precision
- Consistent and better quality, due to the automated control
- Lower cost/unit of output, due to the greater productivity using the same number of workers
- Savings from the indirect labor, from reduced errors, rework, repairs and rejects

##### Disadvantages

- Limited ability to adapt to changes in product or product mix (ex. machines are of limited capacity and the tooling necessary for products, even of the same family, is not always feasible in a given FMS)
- Substantial pre-planning activity
- Expensive, costing millions of dollars
- Technological problems of exact component positioning and precise timing necessary to process a component
- Sophisticated manufacturing systems

FMSs complexity and cost are reasons for their slow acceptance by industry. In most of the cases FMCs are favored.

Robot Programming: The methods used for development of robot programs or more generally called as teaching a robot is as follows

1. Lead by nose
2. Tech pendant
3. Offline programming

## 7. Robot programming

**INTRODUCTION** Robots are devices that are programmed to move parts, or to do work with a tool. Robotics is a multidisciplinary engineering field dedicated to the development of autonomous devices, including manipulators and mobile vehicles.

There are three basic methods for programming industrial robots but currently over 90% are programmed using the teach method.

### Teach Method

The logic for the program can be generated either using a menu based system or simply using a text editor but the main characteristic of this method is the means by which the robot is taught the positional data. A teach pendant with controls to drive the robot in a number of different co-ordinate systems is used to manually drive the robot to the desired locations.

These locations are then stored with names that can be used within the robot program. The co-ordinate systems available on a standard jointed arm robot are :-

### Joint Co-ordinates

The robot joints are driven independently in either direction.

### Global Co-ordinates

The tool centre point of the robot can be driven along the X, Y or Z axes of the robots global axis system. Rotations of the tool around these axes can also be performed

### Tool Co-ordinates

Similar to the global co-ordinate system but the axes of this one are attached to the tool centre point of the robot and therefore move with it. This system is especially useful when the tool is near to the work piece.

### Work piece Co-ordinates

With many robots it is possible to set up a co-ordinate system at any point within the working area. These can be especially useful where small adjustments to the program are required as it is easier to make them along a major axis of the co-ordinate system than along a general line. The effect of this is similar to moving the position and orientation of the global co-ordinate system.

This method of programming is very simple to use where simple movements are required. It does have the disadvantage that the robot can be out of production for a long time during reprogramming. While this is not a problem where robots do the same task for their entire life, this is becoming less common and some robotic welding systems are performing tasks only a few times before being reprogrammed.

### **Lead Through**

This system of programming was initially popular but has now almost disappeared. It is still however used by many paint spraying robots. The robot is programmed by being physically moved through the task by an operator. This is exceedingly difficult where large robots are being used and sometimes a smaller version of the robot is used for this purpose. Any hesitations or inaccuracies that are introduced into the program cannot be edited out easily without reprogramming the whole task. The robot controller simply records the joint positions at a fixed time interval and then plays this back.

### **Off-line Programming**

Similar to the way in which CAD systems are being used to generate NC programs for milling machines it is also possible to program robots from CAD data. The CAD models of the components are used along with models of the robots being used and the fixturing required. The program structure is built up in much the same way as for teach programming but intelligent tools are available which allow the CAD data to be used to generate sequences of location and process information. At present there are only a few companies using this technology as it is still in its infancy but its use is increasing each year. The benefits of this form of programming are:-Reduced down time for programming. Programming tools make programming easier. Enables concurrent engineering and reduces product lead time. Assists cell design and allows process optimization.

## Chapter-6

## PART C

**A hydraulic system**

A suitable hydraulic system is shown in Figure. The system requires a liquid fluid to operate: expensive and messy and, consequently, the piping must act as a closed loop, with fluid transferred from a storage tank to one side of the piston, and returned from the other side of the piston to the tank. Fluid is drawn from the tank by a pump which produces fluid flow at the required 150 bar. Such high pressure pumps, however, cannot operate into a dead end load as they deliver constant volumes of fluid from input to output ports for each revolution of the pump shaft. With a dead-end load, fluid pressure rises indefinitely, until a pipe or the pump itself fails. Some form of pressure regulation, as shown, is therefore required to spill excess fluid back to the tank.

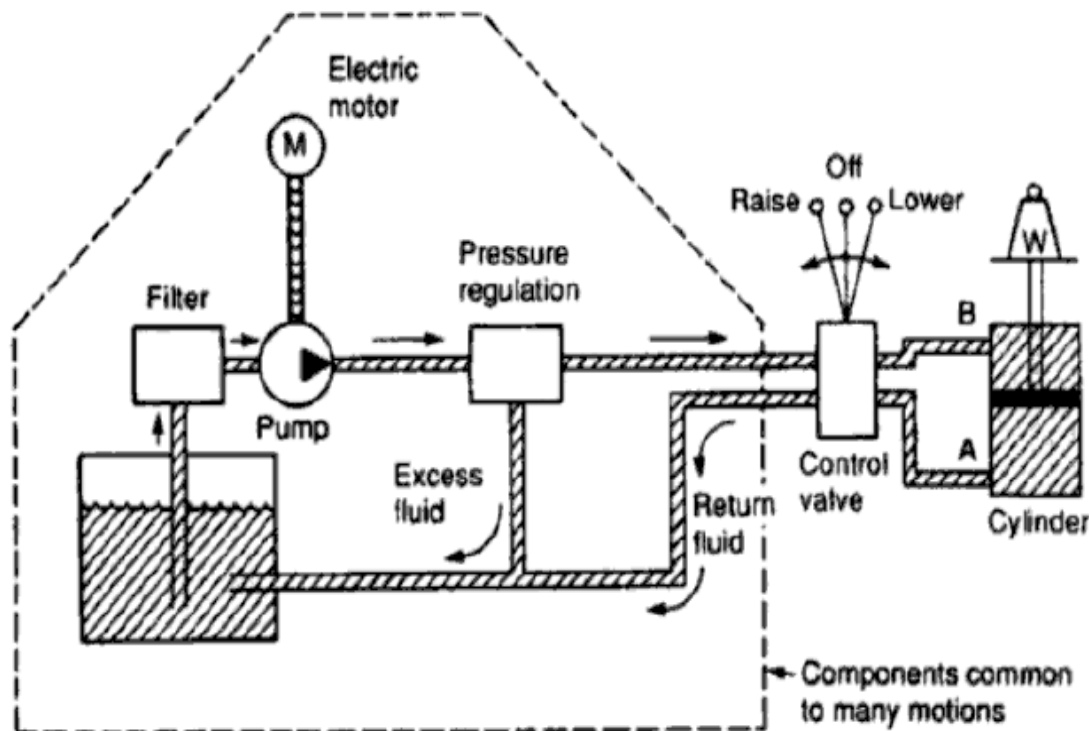
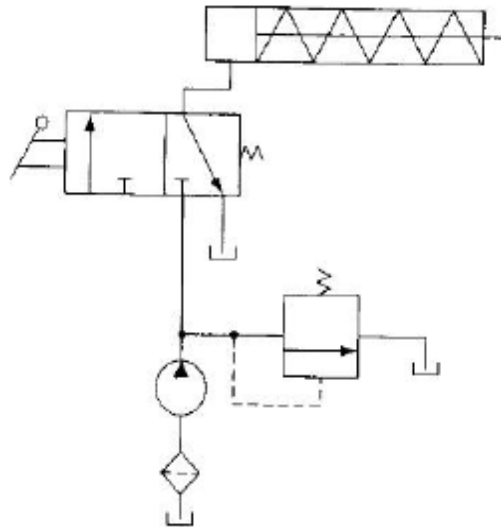
**HYDRAULIC CIRCUIT DESIGN****Control of a single acting hydraulic cylinder**

Figure 5.1 shows how a two-position, three-way, manually actuated, spring-offset directional control valve (DCV) can be used to control the operation of a single acting cylinder. In the spring-offset

mode, full pump flow goes to the tank via the pressure relief valve. The spring in the rod end of the cylinder retracts the piston as oil from the blank end drains back to the tank. When the valve is manually actuated into its left envelope flow path configuration, pump flow extends the cylinder. At full extension, pump flow goes through the relief valve. Deactivation of the DCV allows the cylinder to retract as the DCV shifts into -its spring-offset mode.



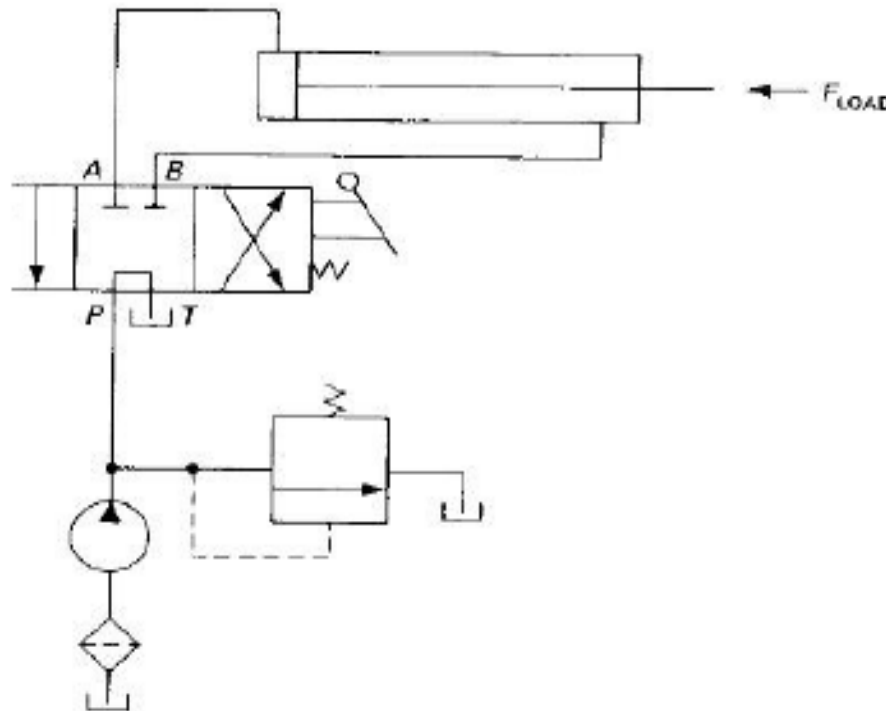
**Figure 5.1: control of a single acting hydraulic cylinder**

### Control of a double acting hydraulic cylinder

Figure 5.2 gives a circuit used to control a double acting hydraulic cylinder. The operation is described as follows:

- 1) When the four-way valve is in its spring-centered position, the cylinder is hydraulically locked. Also the pump is unloaded back to the tank at essentially atmospheric pressure.
- 2) When the four-way valve is actuated into the flow path configuration of the left envelope, the cylinder is extended against its load force  $F_{load}$  as oil flows from port P through port A. Also, oil in the rod end of the cylinder is-free to flow back to the tank via the four-way valve from port B through port T. Note that the cylinder could not extend if this oil were not allowed to leave the rod end of the cylinder.
- 3) When the four-way valve is deactivated, the spring-centered envelope prevails, and the cylinder is once again hydraulically locked.
- 4) When the four-way valve is actuated into the right envelope configuration, the cylinder retracts as oil flows from port P through port B. Oil in the blank end is returned to the tank via the flow path from port A to port T.

5) At the end of the stroke, there is no system demand for oil. Thus, the pump flow goes through the relief valve at its pressure-level setting unless the four way valve is deactivated. In any event, the system is protected from any cylinder overloads.

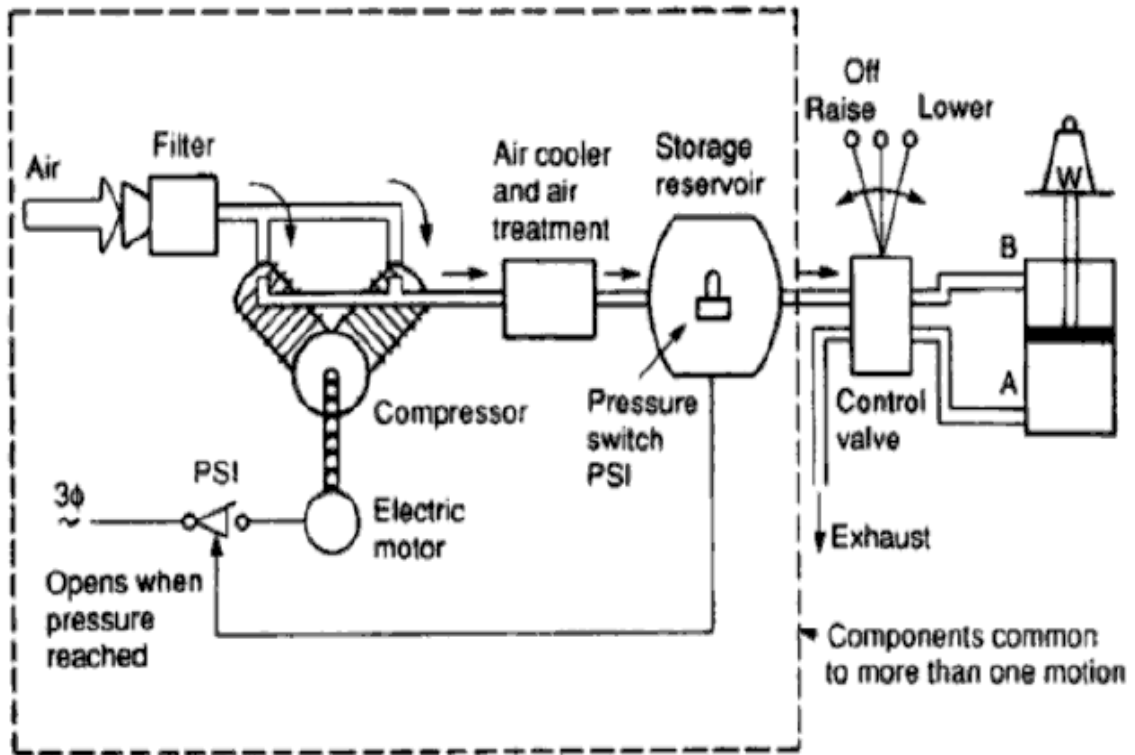


**Figure 5.2: control of a single acting hydraulic cylinder**

### A pneumatic system

Figure shows the components of a pneumatic system. The basic actuator is again a cylinder, with maximum force on the shaft being determined by air pressure and piston cross sectional area. Operating pressures in pneumatic systems are generally much lower than those in a hydraulic systems; 10 bar being typical which will lift 10 kg cm<sup>-2</sup> of piston area, so a 16 cm diameter piston is required to lift the 2000 kg load specified in the previous section. Pneumatic systems therefore require larger actuators than hydraulic systems for the same load. The valve delivering air to the cylinder operates in a similar way to its hydraulic equivalent. One notable difference arises out of the simple fact that air is free; return air is simply vented to atmosphere.





## Chapter-7

### VIVA QUESTIONS

1. Define the term CAD & CAM.
2. Give the abbreviations of CAD,CAM ,CIM ,CAPP,CAQC,CNC, EIA,ACC & ACO
3. List some of the commonly used Modeling packages.
4. Difference between Solid & Wire frame modeling.
5. What are the advantages of solid modeling over wire frame modeling.
6. Differentiate between C-rep and B-rep.
7. List the benefits of CAD
8. List the achievements of CAD
9. Differentiate between topology and geometry.
10. Define the term Numerical control
11. Differentiate between NC and CNC.
12. List the operation devices that are commonly used in a CAD system.
13. Distinguish between ACC and ACO
14. Enlist the commonly used NC words.
15. What are the different NC motion controllers.
16. Configuration of hardware components in a CAD systems.
17. Give basic configuration of the system for usages of CAD/CAM.
18. List some of the commonly used analysis packages.
19. Where NC is most Appropriate
20. Advantages and Disadvantages of NC machines.
21. What is Fixed zero and Floating zero.
22. What is Absolute & Incremental; NC machine tool system.
23. What are G Codes & M Codes.
24. Give the General procedure for NC programming
25. Give the general procedure for Manual part programming for turning
26. What is FMS
27. What are the benefits of FMS
28. What are the major elements of FMS

29. Discuss the importance of material handling system in FMS
30. Give the name of materials handling devices in a FMS
31. How is FMS optimized
32. What are different types of data associated with FMS
33. How are jobs scheduled in FMS
34. How NC program control is effective in a FMS
35. What are the significant advantages of using a robot in a computer integrated manufacturing system
36. Describe the elements of a robot system
37. How do you justify the use of a robot
38. How do you classify robot
39. What are the different types of drives used in robot
40. What are the different types of control systems used in robots
41. How do you specify a robot
42. Discuss the basic types of robot programming languages
43. Briefly describe significant application areas of robot.
44. When do we use G01
45. Differentiate G2 and G3.
46. Which code is used for rough turning cycle
47. What is G70
48. Differentiate G71 and G72
49. When do we use Grooving Cycle
50. Which code is used for Grooving Cycle
51. When do we use G83
52. What is the function for Preparatory codes
53. Describe absolute programming
54. Describe incremental programming
55. Briefly Explain the circular interpolation
56. What is canned cycle? Give Example
57. What is the use of G20
58. Which code is used for Absolute programming
59. What is the use M00

60. When do we use M30
61. Differentiate M04 and M05
62. When do we G21
63. What is cutter compensation
64. What is the use of G27
65. What is the use of N codes
66. Describe Drilling cycle
67. Explain Rapid positioning
68. What machining operations can be performed on machining center
69. Write note on high speed machining and high speed machine tool
70. Explain word address programming
71. Explain the configuration of main program and subprogram