K. S. INSTITUTE OF TECHNOLOGY

#14, Raghuvanahalli, Kanakapura Main Road,
Bengaluru-560109.

DEPARTMENT OF MECHANICAL ENGINEERING

VII SEMESTER

CIM LAB

(15MEL77)

NAME: ______________________________

SEM/SEC: ____________________________

BRANCH: ____________________________

USN: _________________________________

LAB MANUAL & OBSERVATION

SYLLABUS

COMPUTER INTEGRATED MANUFACTURING LAB B.E, VII Semester, Mechanical Engineering [As per Choice Based Credit System (CBCS) scheme]			
Course Code	15MEL77	**CIE Marks**	**40**
Number of Lecture Hours/Week	**03 (1 Hour Instruction+ 2 Hours Laboratory)**	**SEE Marks**	**60**
Total Hours	**40**	**Exam Hours**	**03**
Credits –02			

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 and cut part on virtual CNC machine simulator.
CLO3	To make the students understand the importance of automation in industries through exposure to FMS, Robotics, and Hydraulics and Pneumatics.

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,Millingoperations.3 typical simulations to be carried out using simulation packages like: **Cadem CAM Lab-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. Enter program, take tool offsets, cut part in single block and auto mode, measure the virtual part on screen in the virtual CNC machine simulator, for standard CNC control systems FANUC, FAGOR, HAAS and SINUMERIK.

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 or 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 topics to be conducted.

Course Outcomes:

After studying this course, students will be able to:

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; set up and cut part on.
CLO6	Understand & write programs for Robot control; understand the operating principles of hydraulics, pneumatics and electro pneumatic systems.

Scheme for Examination:
Two Questions from Part A - 60 Marks (30 +30)
Viva-Voce - 20 Marks

Total: 80 Marks

COURSE OBJECTIVES

The objectives of Computer Integrated Manufacturing and Automation laboratory is

- **To demonstrate the concepts discussed in Computer Integrated Manufacturing course.**
- **To introduce CNC part programming for simulation of various machining operations.**
- **To educate the students on Flexible Manufacturing System and Robot Programming.**
- **To educate the students on the hydraulics, pneumatics and electro– pneumatic systems.**

COURSE OUTCOMES

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

- **To practically relate to concepts discussed in Computer Integrated Manufacturing course.**
- **To write CNC part programs using CADEM simulation package for simulation of machining operations such as Turning, Drilling & Milling.**
- **To understand & write programs for Flexible Manufacturing Systems & Robotics.**
- **To understand the operating principles of hydraulics, pneumatics and electro–pneumatic systems.**
- **To apply these learnings to automate & improve efficiency of manufacturing process.**

VISION OF THE DEPARTMENT

"To groom the incumbent to be able to compete with the best in the Mechanical Engineering profession and to get recognized by peers as one of the best learning centers".

MISSION OF THE DEPARTMENT

- **To develop the Institution with research as the first preference.**
- **To impart sound technical knowledge with basic principles and concepts.**
- **To expose students to the recent development in the field.**
- **To provide an equal opportunity to every individuals to attain his/her potential.**
- **To achieve engineering excellence through R&D with team work.**

GENERAL INSTRUCTION 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.

CONTENTS

TURNING

MILLING

K. S. INSTITUTE OF TECHNOLOGY

#14, Raghuvanahalli, Kanakapura Main Road, Bengaluru – 109.

INDEX CARD

NAME:__________________________ USN: _______________________

LAB/CODE:_____________________ SEM: ______________________

BRANCH:________________________ BATCH: ___________________

NAME OF FACULTY: ______________________________

SL. NO	DATE OF CONDUCTION	PARTICULARS	MARKS
1			
2			
3			
4			
5			
6			
7			
8			
9			
10			
		TOTAL	
		AVG. RECORD MARKS OBTAINED	

AVG. RECORD MARKS(15)	TEST(10)	TOTAL INTERNAL MARKS (25)

Signature of Student Signature of Staff Signature of HOD

INTRODUCTION

NUMERICAL CONTROL : (NC)

It can be defined as 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 as 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 predefined 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.

PART PROGRAMMING FUNDAMENTALS

NC 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 carryout 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 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 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 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) Manual 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.

PART PROGRAMMING GEOMETRY

A. 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 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 :

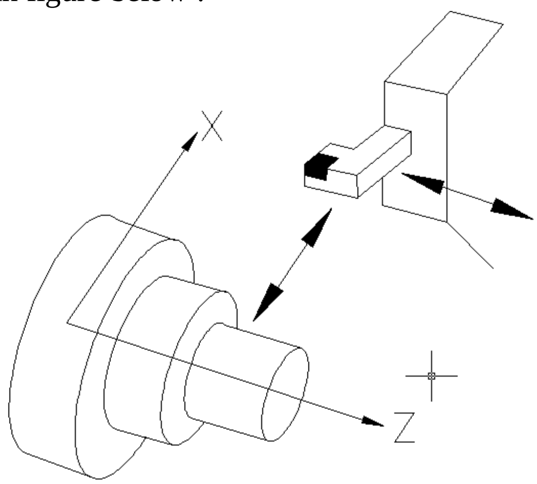
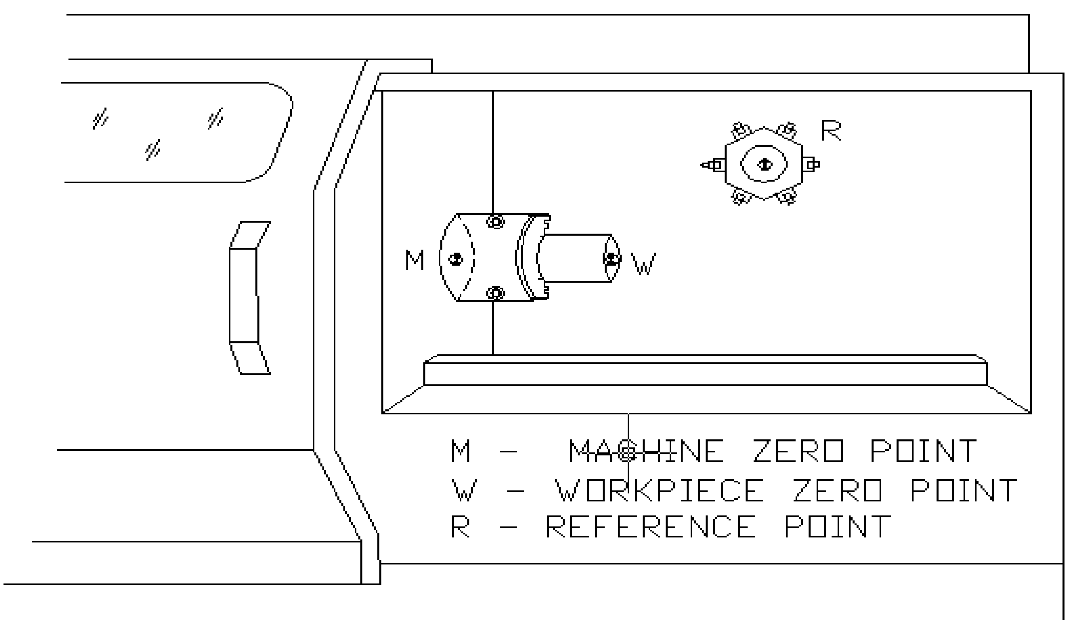


Fig 1. COORDINATE SYSTEM FOR TURNING OPERATIONS

B. ZERO POINTS AND REFERENCE POINTS

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

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



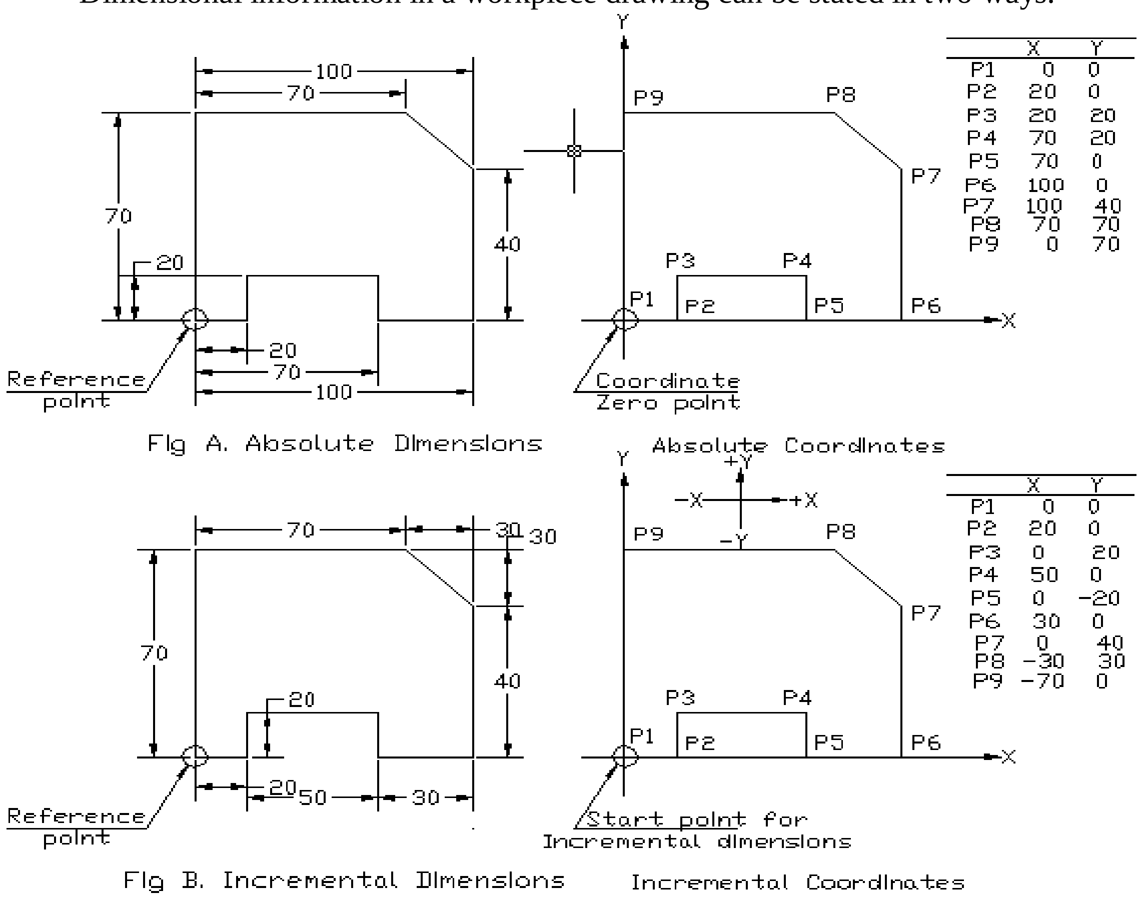
WORKPIECE ZERO POINT (W): This point determines the workpiece coordinate system in relation to the machine zero point. The workpiece zero pint is chosen by the programmer and input into the CNC system when setting up the machine. The position of the workpiece zero point can be freely chosen by the programmer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

For turned parts, the 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 figure. Occasionally the workpiece zero point is also called the "program zero point."

REFERNCE POINT (R): This point serves for calibrating and for controlling the measuring system of the slides and tool traverses. The position of the reference point as shown in figure below is accurately predetermined in every traverse axis by the trip dogs and limit switches. Therefore, the reference point coordinates always have the same, precisely known numerical value in relation to the machine zero point. After initiating the control system, the reference point must always be approached from all axes to calibrate the traverse measuring system. If current slide and tool position data should be lost in the control system as for example, through an electrical failure, the machine must again be positioned to the reference point to re-establish the proper positioning values.

C. NC- RELATED DIMENSIONING

Dimensional information in a workpiece drawing can be stated in two ways:



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

2. Incremental Dimension System: When using incremental dimension system, every measurement refers to a previously dimensioned position as shown in figure A below. Incremental dimensions are distance between adjacent points. These distances are converted into incremental coordinates by accepting the last dimension point as the coordinate origin for the new point. This may be compared to a small coordinate system, i.e. shifted consequently from point to point as shown in figurer B. Incremental dimensions are also frequently called “Relative dimensions” or “Chain dimensions”.

NC PROGRAM BUILD UP

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. An NC word is a collection of address letter and a sequence of numbers. Table shows the address letters:

Address Characters	
Character	**Meaning**
A	Rotation about, X-axis
B	Rotation about, Y-axis
C	Rotation about, Z-axis
D & E	Rotation about additional axis
F	Feed
G	Preparatory function, identifying the action to be executed
I	Interpolation parameter / Thread pitch parallel to X-axis.
J	Thread pitch parallel to Y-axis
K	Thread pitch parallel to Y-axis
M	Auxiliary function
N	Block Number
P,Q,R	Thread movement parallel to X,Y,Z axis respectively. P & Q are also used as parameters in cycles.
S	Spindle speed
T	Tool
U,V,W	Second movement parallel to X,Y,Z axis respectively
X	Movement in X-axis
Y	Movement in Y-axis
Z	Movement in Z-axis

All the NC words may not be used on every CNC machine. Using these words as an example, the composition of a block is assembled as follows:

	N	G	X	Z	F	S	T	M		;	

FANUC TURNING PROGRAMMING
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

M00 PROGRAM STOP: By inserting M00 in a program, the cutting cycle is stopped after the block containing M00 code. This facility is useful if an inspection check is necessary during an operation. The cycle is then continued by a cycle start.

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.

M02 PROGRAM END: This code is inserted at the end of the program, when encountered the cycle will end. To produce another, the system must be reset.

M03 SPINDLE FORWARD: Starts the spindle spinning forward, clockwise or negative direction at the last specified spindle rate.

M04 SPINDLE REVERSE: Starts the spindle spinning reverse, counter clockwise or positive direction at the last specified spindle rate.

M05 STOP SPINDLE: Stops the spindle without changing the spindle speed.

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

M08 COOLANT ON: Turns the coolant on.
M09 COOLANT OFF: Turns the coolant off.
M10 CHUCK OPEN: Opens pneumatic or similar automatic chuck to allow for bar feed.
M11 CHUCK CLOSE: Closes the chuck.

M13 SPINDLE FORWARD, COOLANT ON: Sets spindle rotation forward and sets the coolant on, both are performed by single code.

M14 SPINDLE REVERSE, COOLANT ON: Sets the spindle rotation in reverse direction and sets the coolant on.

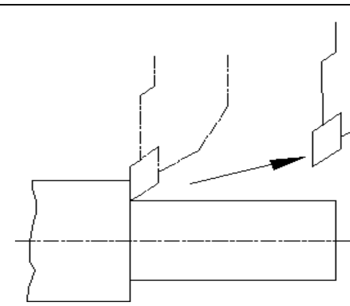
M30 PROGRAM END: Stops the spindle, turns the coolant off, terminates and resets the CNC program.

M38 DOOR OPEN: Opens the door, waiting until the door is open.
M39 DOOR CLOSE: Closes the door, waiting until the door is closed.

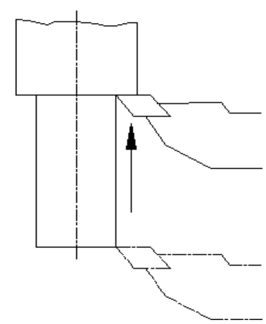
PREPARATORY FUNCTION (G-Codes).

G CODES	
G00 **G01** **G02** **G03**	Positioning (Rapid Transverse) Linear Interpolation (Feed) Circular Interpolation (CW) Circular Interpolation (CCW)
G04	Dwell
G20 **G21**	Inch Data Input Metric Data Input
G28	Reference point return
G40 **G41** **G42**	Tool nose radius compensation cancel Tool nose radius compensation left Tool nose radius compensation right
G50	Work coordinate change/ Max. Spindle speed setting
G70 **G71**	Finishing cycle Multiple Turning Cycle in turning
G72 **G73** **G74** **G75** **G76**	Stock removal in facing Pattern repeating Peck drilling in Z axis Grooving in X axis Thread cutting cycle
G90 **G94**	Cutting cycle A (Turning) Cutting cycle B (Facing)
G96 **G97**	Constant surface speed control Constant surface speed control cancel
G98 **G99**	Feed per minute Feed per revolution

G00 FAST TRAVERSE

Description	Illustration
A rapid traverse instruction traverses the tool to the target point at the maximum traverse rate. The tool normally takes the shortest path from the starting point to the destination point. The rapid traverse is used for movements where no tool is in engagement.	

G01 LINEAR MOTION

Description	Illustration
G01 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, the material being machined, the required surface finish and the drive rating and rigidity of the machine tool.	
Example : G01 X30 Z10 F100 S1000 Target point Feed Speed	

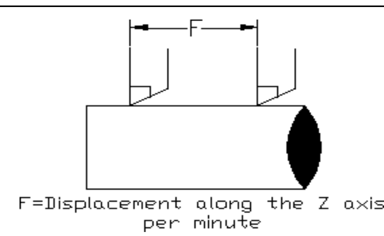
G20 INCH DATA INPUT: A G20 causes position to be as being in imperial units. All the input values are in inches. This can only be at the start of the main program.

G21 METRIC DATA INPUT: A G21 causes positions to be interpreted as being in metric units. All the input values are in mm. This can only be at the start of the main program.

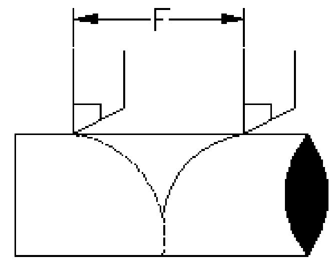
G28 REFERENCE POINT RETURN

Description	Illustration
A G28 causes a fast traverse to the specified position and then to the machine datum.	G28 X35 Z5 G28 U0 W0

G98 FEED PER MINUTE

Description	Illustration
This command coupled with the F word is used to specify feed rate per minute. This can be in either mm/min or inch/min. this is the default.	

G99 FEED PER REVOLUTION

Description	Illustration
This command coupled with the F word is used to specify a feed rate per revolution. This can be in mm/rev or inch/rev. the feed rates available in the machine simulation are 0.01-200 mm/min. Recommended feed rates are published by tool and cutter manufacturers, along with recommended cutting speeds. If the feed rate is expressed as mm/rev, a simple calculation can be used to convert to mm/min. FEED, mm/min=FEED(mm/rev) X spindle speed(RPM)	 F=Displacement along the Z axis per revolution of the workpiece

PROGRAM BUILD-UP FOR CNC LATHE (FANUC)

CNC program can be divided into 3 parts, Start-up, Body and End of the program.

START-UP OF CNC PROGRAM

O1000
[BILLET X20 Z60
G21/G20 G98/G99 G40
G28 U0 W0
M06 T0101
M03/M04 S1000
G00 X21 Z1

EXPLANATION

O1000	While writing a program on FANUC controller first line has to be started with letter 'O' followed by four digit number which specifies the program name.
[BILLET X20 Z60	This directive is used only for simulation purpose. It defines the work piece dimensions as 60 mm long and 20 mm in diameter.
G21/G20 G98/G99 G40	G21 – code specifies that program is done in metric units. G20 - code specifies that program is done in imperial units G98 – gives the unit of feed in mm/minute. G99 - gives the unit of feed in mm/revolution. G40 – Compensation cancel.
G28 U0 W0	Makes the tool to go to home position. U & W are Secondary movements about X and Z axis.
M06 T0101	Tool change. The first two digits specify the tool position in the turret and last two digits denotes the tool offset number.
M03/M04 S1000	M03- makes the spindle rotate in clockwise direction. M04 – makes the spindle rotate in counter-clockwise direction. S1000-Spindle rotates at 1000 rpm.

G00 X21 Z1	G00 gives rapid position of the tool to a point X21 Z1 which is just above the billet. This point is called as the tool entry point.

BODY OF THE PROGRAM: This is dealt operation wise in the succeeding pages.

END OF THE PROGRAM:
G28 U0 W0
M05
M02/30

EXPLANATION

G28 U0 W0	Makes the tool to go to home position. U & W are secondary movements about X and Z axis.
M05	Stops the spindle rotation
M02/30	M02 – Optional stop M30 – Program stop and rewind.

METAL CUTTING PARAMETERS FOR LATHE

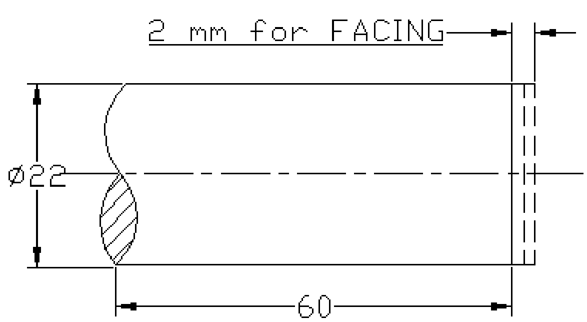
BILLET MATERIAL : Aluminum

OPERATIONS	SPEED, rpm	FEED mm/min	DEPTH OF CUT, mm
TURNING	1000-1500	45-55	0.5-1.0
GROOVING	600-800	15-25	0.25-0.5
THREADING	300-350	15-30	0.03-0.04

EXERCISE -1

SIMPLE FACING

Write a manual part program for Simple Facing Operation for the component shown in figure below.



DWG. NO. 1

PLANNING AND OPERATIONS SHEET	
BILLET SIZE : 22 x 60	**MATERIAL : Aluminum**

PROGRAM NO : 1001				**DWG NO : 1**			
SL.NO	**Operation**	**Tool Holder**	**Tool Tip**	**Tool Station No**	**Tool Offset No**	**Spindle Speed, rpm**	**Feed, mm/min**
1	Simple Facing	SDJCR 1212H11	DCMT 11T304	1	1	1200	45

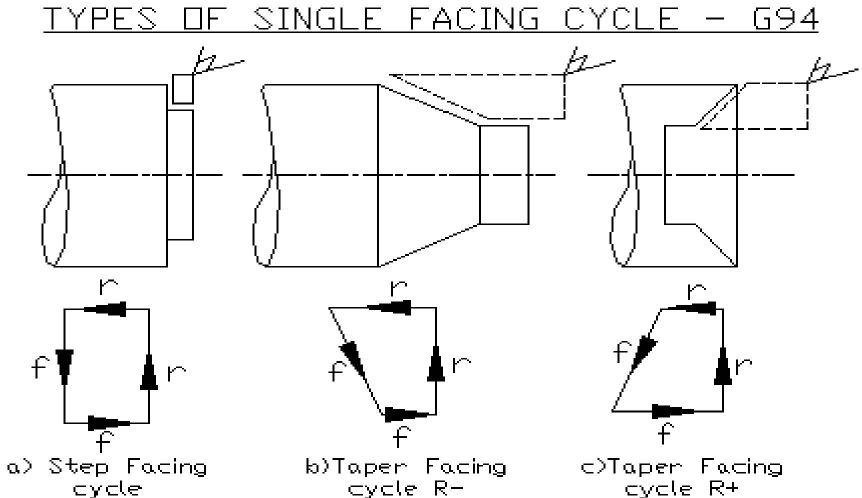
(Drawing No .1

(CNC program for Simple Facing) (Material to be removed by facing : 2mm)

```
O1001                 --------- Program Number 1001
[BILLET X22 Z60       --------- Defining Billet size dia : 22 length 60 mm
G21 G98               --------- Initial settings
G28 U0 W0             --------- Going to home position
M06 T0101             ---------- Selecting Tool No. 1 with offset No. 1
M03 S1200             ---------- Setting spindle speed at 1200 rpm
G00 X22 Z1            ---------- Tool moving to tool entry point X22 Z1 at
                                 rapid traverse.
G01 Z-0.5 F45         ---------- Giving depth of cut of 0.5 mm at a feed rate of
                                 45 mm/min.
G01 X0                ---------- Moving the tool to spindle center line
G01 Z1                --------- Retract back the tool
G00 X22               --------- Moving the tool to X22
G01 Z-1 F45           --------- Giving the second depth of cut.
G01 X0
G01 Z1
G00 X22
G01 Z-1.5
G01 X0
G01 Z1
G00 X22
G01 Z-2
G01 X0
G01 Z1
G00 X22               --------- Retract back the tool
G28 U0 W0             -------- Going to home position
M05                   -------- Stop the spindle
M30                   -------- Program stop and rewind.
```

Date of Commencement	SIGNATURE OF STUDENT	**Marks / Remarks:**
Date of finish	SIGNATURE OF STAFF	

G94 – FACING CYCLE

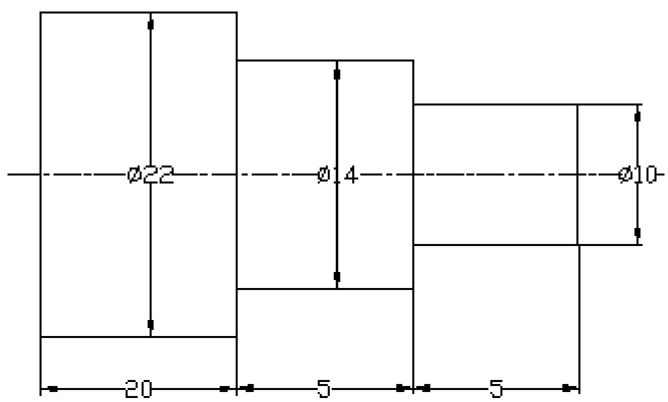


Description	Illustration
This cycle is used for stock removal either parallel or at an angle to workpiece face. It is the equivalent of rapid to Z position, feed to X position, feed to start Z position, and rapid to start X position. If the "R" value is specified tapering will be performed. The sign of "R" depends on direction of the taper.	**a)G94 X(U) Z(W) F** **b) G94 X(U) Z(W) R- F** **c) G94 X(U) Z(W) R+ F** Where X = diameter to which the movement is being made. Z= The Z axis coordinate to which the movement is being made. R= The difference in incremental of the cut start radius value and the cut finish radius value.

EXERCISE -2

SIMPLE FACING - CYCLE

Write a manual part program for Facing Operation for the component shown in figure below.



DWG. NO. 2

PLANNING AND OPERATIONS SHEET							
BILLET SIZE : 22 x 40				**MATERIAL : Aluminum**			
PROGRAM NO : 1004				**DWG NO : 2**			
SL.NO	**Operation**	**Tool Holder**	**Tool Tip**	**Tool Station**	**Tool Offset**	**Spindle Speed,**	**Feed, mm/min**

				No	No	rpm	
1	Facing	SDJCR 1212H11	DCMT 11T304	1	1	1200	45

(Drawing No .2

(CNC program for Facing cycle

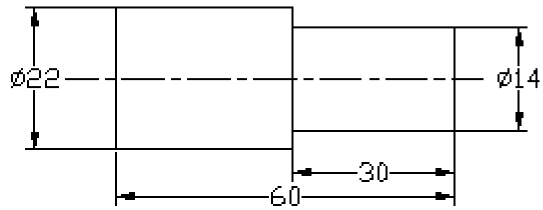
```
O1004                      --------- Program Number 1004
[BILLET X22 Z40            --------- Defining Billet size dia : 22 length 40 mm
G21 G98                    --------- Initial settings
G28 U0 W0                  --------- Going to home position
M06 T0101                  ---------- Selecting Tool No. 1 with offset No 1
M03 S1200                  ---------- Setting spindle speed at 1200 rpm
G00 X22 Z1                 ---------- Tool moving to tool entry point X22 Z1
G01 Z0
G94 X10 Z-0.5 F35          ----------- G94 Box Facing cycle
Z-1                                    G94 code Syntax : G94 X Z F
Z-1.5
Z-2
Z-2.5
Z-3
Z-3.5
Z-4
Z-4.5
Z-5
G00 X22 Z-5
G94 X14 Z-5.5 F35          ----------- G94 Facing cycle
Z-6
Z-6.5
Z-7
Z-7.5
Z-8
Z-8.5
Z-9
Z-9.5
Z-10
G28 U0 W0                  -------- Going to home position
M05                        -------- Stop the spindle
M30                        -------- Program stop and rewind.
```

Date of Commencement	SIGNATURE OF STUDENT	**Marks / Remarks:**
Date of finish	SIGNATURE OF STAFF	

EXERCISE -3

SIMPLE TURNING

Write a manual part program for Simple Turning Operation for the component shown in figure below.



DWG. NO. 3

PLANNING AND OPERATIONS SHEET							
BILLET SIZE : 22 x 60				**MATERIAL : Aluminum**			
PROGRAM NO : 1002				**DWG NO : 3**			
SL.NO	**Operation**	**Tool Holder**	**Tool Tip**	**Tool Station No**	**Tool Offset No**	**Spindle Speed, rpm**	**Feed, mm/min**
1	Simple Turning	SDJCR 1212H11	DCMT 11T304	1	1	1200	45

(Drawing No .3

```
(CNC program for Simple Turning)
 (Reducing the diameter from 22 mm to 14 mm
O1002                  --------- Program Number 1002
[BILLET X22 Z60        --------- Defining Billet size dia : 22 length 60 mm
G21 G98                --------- Initial settings
G28 U0 W0              --------- Going to home position
M06 T0101              ---------- Selecting Tool No. 1 with offset No 1
M03 S1200              ---------- Setting spindle speed at 1200 rpm
G00 X22 Z1             ---------- Tool moving to tool entry point X22 Z1 at
G01 X21                           rapid traverse.
G01 Z-30 F45
G00 X22                ---------- Simple Turning operation
G00 Z1
G01 X20
G01 Z-30 F45
G00 X22
G00 Z1
G01 X19
G01 Z-30 F45
G00 X22
G00 Z1
G01 X18
G01 Z-30 F45
G01 X15
G01 Z-30 F45
G00 X22
G00 Z1
```

```
G00 X22
G00 Z1
G01 X17
G01 Z-30 F45
G00 X22
G00 Z1
G01 X16
G01 Z-30 F45
G00 X22
G00 Z1
G01 X14
G01 Z-30 F45
G00 X22
G00 Z1
G28 U0 W0-------- Going to home position
M05      -------- Stop the spindle
M30      -------- Program stop and rewind.
```

Date of Commencement	SIGNATURE OF STUDENT	**Marks / Remarks:**
Date of finish	SIGNATURE OF STAFF	

G90 TURNING CYCLE

This cycle can be used to produce either a parallel or tapered tool path. This cycle performs four distinct moves with one line of information and it is equivalent of

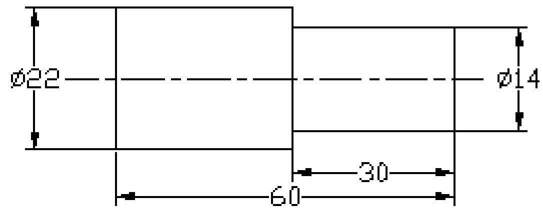
- Rapid to X position
- Feed to Z position
- Feed to start X position
- Rapid to start Z position.

Description	Illustration
With the above command the cycle will execute removing material to the required diameter and length. To repeat this cycle to reduce the diameter but maintain the same length, only the value to be changed need to be programmed.	**G90 X(U) Z(W) F** Where X – Diameter to which the movement is being made. Z- The Z axis coordinate to which the movement is being made. F- Feed

EXERCISE -4

SIMPLE TURNING CYCLE

Write a manual part program for Simple Turning Operation for the component shown in figure below.



DWG. NO. 4

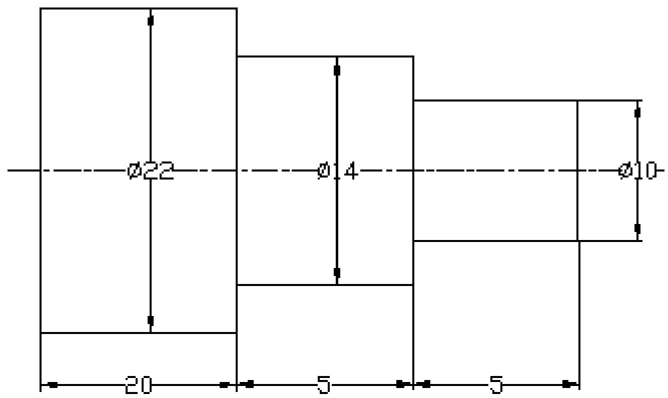
PLANNING AND OPERATIONS SHEET							
BILLET SIZE : 22 x 60				**MATERIAL : Aluminum**			
PROGRAM NO : 1002				**DWG NO : 4**			
SL.NO	**Operation**	**Tool Holder**	**Tool Tip**	**Tool Station No**	**Tool Offset No**	**Spindle Speed, rpm**	**Feed, mm/min**
1	Simple Turning	SDJCR 1212H11	DCMT 11T304	1	1	1200	45

```
(Drawing No .4
(CNC program for Simple Turning cycle
O1002                        --------- Program Number 1002
[BILLET X22 Z60              --------- Defining Billet size dia : 22 length 60 mm
G21 G98                      --------- Initial settings
G28 U0 W0                    --------- Going to home position
M06  T0101                   ---------- Selecting Tool No. 1 with offset No 1
M03  S1200                   ---------- Setting spindle speed at 1200 rpm
G00 X22 Z1                   ---------- Tool moving to tool entry point X22 Z1 at
G01 Z0                                  rapid traverse.
G90 X21 Z-30 F50             ----------  Turning cycle
X20
X19
X18
X17
X16
X15
X14
G28 U0 W0                    -------- Going to home position
M05                          -------- Stop the spindle
M30                          -------- Program stop and rewind.
```

EXERCISE -5

STEP TURNING

Write a manual part program for Step Turning Operation with G90 cycle for the component shown in figure below.



DWG. NO. 5

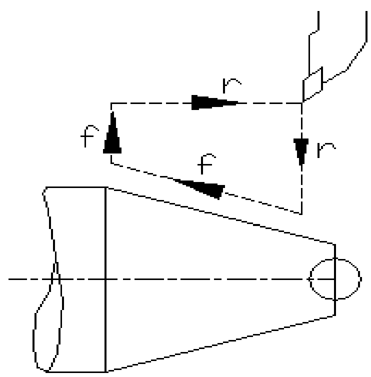
PLANNING AND OPERATIONS SHEET							
BILLET SIZE : 22 x 60				**MATERIAL : Aluminum**			
PROGRAM NO : 1007				**DWG NO : 5**			
SL.NO	**Operation**	**Tool Holder**	**Tool Tip**	**Tool Station No**	**Tool Offset No**	**Spindle Speed, rpm**	**Feed, mm/min**
1	Step Turning	SDJCR 1212H11	DCMT 11T304	1	1	1200	30

```
(Drawing No .5
(CNC program for Step Turning
O1007                        --------- Program Number 1007
[BILLET X22 Z60               --------- Defining Billet size dia : 22 length 60 mm
G21 G98                       --------- Initial settings
G28 U0 W0                     --------- Going to home position
M06  T0101                    ---------- Selecting Tool No. 1 with offset No 1
M03  S1200                    ---------- Setting spindle speed at 1200 rpm
G00 X22 Z1                    ---------- Tool moving to tool entry point X22 Z1
G01 Z0
G90 X21 Z-10 F30             ----------- G90 Step Turning cycle
X20                                       G90 code Syntax : G90 X Z  F
X19
X18
X17
X16
```

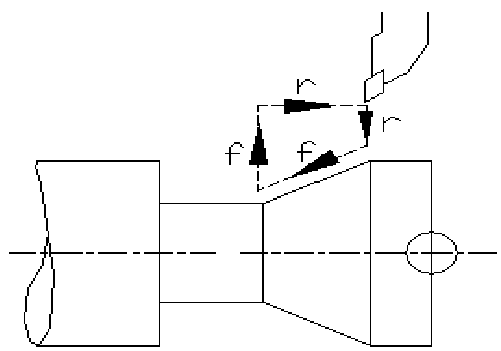
```
X15
X14
G00 X14 Z1
G90 X13 Z-5 F30
X12
X11
X10
G28 U0 W0          -------- Going to home position
M05                -------- Stop the spindle
M30                -------- Program stop and rewind.
```

Date of Commencement	SIGNATURE OF STUDENT	**Marks / Remarks:**
Date of finish	SIGNATURE OF STAFF	

G90 TAPER TURNING



b)Taper Turning cycle R-



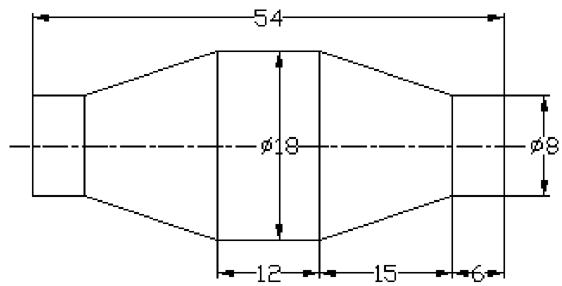
c)Taper Turning cycle R+

Description	Illustration
If an “R” value is specified in the command format of G90 cycle, tapering will be performed. The sign of “R” will depend on the direction of the taper. The initial rapid move will be to the X position plus the “R” value.	**G90 X(U) Z(W) R F** Where X – Diameter to which the movement is being made. Z- The Z axis coordinate to which the movement is being made. R- The difference in incremental of the cut start radius value and the cut finish radius value. F- Feed

EXERCISE -6

TAPER TURNING CYCLE

Write a manual part program for Taper Turning Operation for the component shown in figure below.

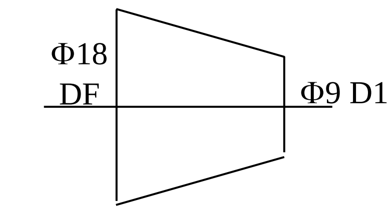


DWG. NO. 6

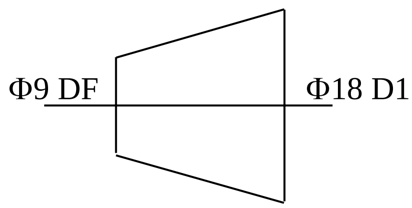
PLANNING AND OPERATIONS SHEET							
BILLET SIZE : 22 x 60				**MATERIAL : Aluminum**			
PROGRAM NO : 1008				**DWG NO : 6**			
SL.NO	**Operation**	**Tool Holder**	**Tool Tip**	**Tool Station No**	**Tool Offset No**	**Spindle Speed, rpm**	**Feed, mm/min**
1	Taper Turning	SDJCR 1212H11	DCMT 11T304	1	1	1200	35

```
(Drawing No .6
(CNC program for Taper Turning
O1008                   --------- Program Number 1008
[BILLET X22 Z60          --------- Defining Billet size dia : 22 length 60 mm
G21 G98                  --------- Initial settings
G28 U0 W0                --------- Going to home position
M06  T0101              ---------- Selecting Tool No. 1 with offset No 1
M03  S1200              ---------- Setting spindle speed at 1200 rpm
G00 X22 Z1               ---------- Tool moving to tool entry point X22 Z1
G01 Z0
G90 X21 Z-54 F35        ----------- G90 Step Turning cycle
X20                                 G90 code Syntax : G90 X Z  F
X19
X18
X17 Z-6
X16
X15
X14
X13
X12
```

```
X11
X10
X9
G00 X18 Z-6
G90 X18 Z-21 R0 F30        --------   Taper Turning – G90 R-
X18 R-0.5                                         R=(D1-DF)/2
X18 R-1                                             =(9-18)/2
X18 R-1.5                                           =-9/2 = -4.5
X18 R-2
X18 R-2.5
X18 R-3
X18 R-3.5
X18 R-4
X18 R-4.5
```




```
G01 X18 Z-33
G90 X18 Z-48 R0 F50        --------   Taper Turning – G90 R+
X17 R0.5                                          R=(D1-DF)/2
X16 R1                                              =(18-9)/2
X15 R1.5                                            =9/2 = 4.5
X14 R2
X13 R2.5
X12 R3
X11 R3.5
X10 R4
X9 R4.5
```




```
G00 X18 Z-48
G90 X18 Z-54 F30           --------   Taper Turning – G90
X17
X16
X15
X14
X13
X12
X11
X10
X9
G28 U0 W0                  -------- Going to home position
M05                        -------- Stop the spindle
M30                        -------- Program stop and rewind.
```

Date of Commencement	SIGNATURE OF STUDENT	**Marks / Remarks:**
Date of finish	SIGNATURE OF STAFF	

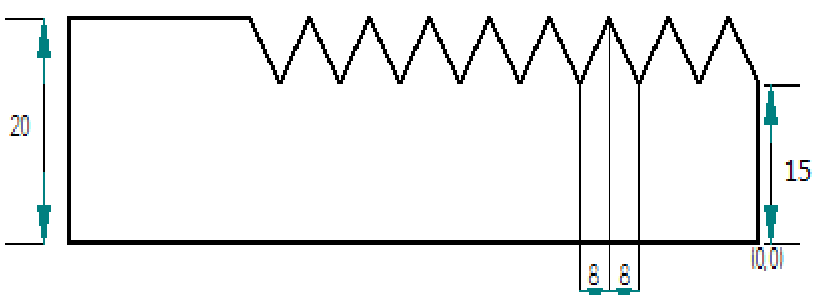
SUBPROGRAM CALL / EXIT – M98 / M99

Main Program	SubProgram	
A Program is divided into a main program and subprogram. Normally the CNC operates according to the main program but when a command calling a subprogram is encountered in the main program control is passed to the subprogram. When a command indicating to return to the main program is encountered in the subprogram, control is returned to the main program. The first block of program / subroutine must contain a program number "O".	When a program contains certain fixed sequences or frequently repeated patterns these sequences or patterns may be entered into memory as a subprogram to simplify programming. A subprogram can call another subprogram. When the main program call a subprogram, it is regarded as a one-loop subprogram call.	
Main program O0001; -------; -------; M98 P1000; --------; --------; M30;	Subprogram O1000; --------; --------; M98 P2000; ---------; ---------; M99; 1st Loop Nesting	Subprogram O2000; ---------; ---------; ---------; ---------; ---------; M99; 2nd Loop Nesting

EXERCISE -7

EXAMPLES ON SUBPROGRAMS

Write a manual part program by using Subprograms for the component shown in figure below.

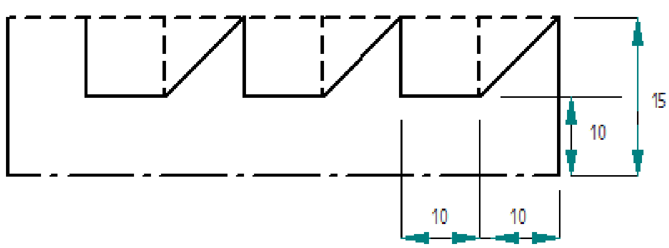


DWG. NO. 7

PLANNING AND OPERATIONS SHEET							
BILLET SIZE : 22 x 60				**MATERIAL : Aluminum**			
PROGRAM NO : 1008				**DWG NO : 7**			
SL.NO	**Operation**	**Tool Holder**	**Tool Tip**	**Tool Station No**	**Tool Offset No**	**Spindle Speed, rpm**	**Feed, mm/min**
1	Taper Turning	SDJCR 1212H11	DCMT 11T304	1	1	1200	35

```
(Drawing No .7
(CNC program on subprograms
O1008                    --------- Program Number 1008 –Main Program
[BILLET X20 Z70          --------- Defining Billet size dia : 20 length 70 mm
G21 G98                  --------- Initial settings
G28  U0  W0              --------- Going to home position
M06   T0101              ---------- Selecting Tool No. 1 with offset No 1
M03   S1200              ---------- Setting spindle speed at 1200 rpm
G00 X20 Z1               ---------- Tool moving to tool entry point X20 Z1
G01 Z0
M98 P0037777
G28 U0 W0                -------- Going to home position
M05                      -------- Stop the spindle
M30                      -------- Program stop and rewind.

O7777                    ----------- Subprogram 7777
G90 X20 W-8 R-0.5 F50
X20 R-1
X20 R-1.5
X20 R-2
X20 R-2.5
G00 X20 W-8
G90 X19 W-8 R0.5
X18 R1
X17 R1.5
X16 R2
X15 R2.5
G00 X20 W-8
M99
```

Write a manual part program by using Subprograms for the component shown in figure below.



```
 (CNC program on Subprograms
O1001                          --------- Program Number 1001 –Main Program
[BILLET X15 Z70                 --------- Defining Billet size dia : 15 length 70 mm
G21 G98                         --------- Initial settings
G28 U0 W0                      --------- Going to home position
M06  T0101                      ---------- Selecting Tool No. 1 with offset No 1
M03  S2000                      ---------- Setting spindle speed at 1200 rpm
G00 X15 Z1                      ---------- Tool moving to tool entry point X20 Z1
G01 Z0
M98 P0038888
G28 U0 W0                       -------- Going to home position
M05                             -------- Stop the spindle
M30                            -------- Program stop and rewind.

O8888                          ----------- Subprogram 7777
G90 X14 W-10 R0.5 F50
X13 R1
X12 R1.5
X11 R2
X10 R2.5
G00 X15 W-10
G90 X14 W-10
X13
X12
X11
X10
G00 X15 W-10
M99
```

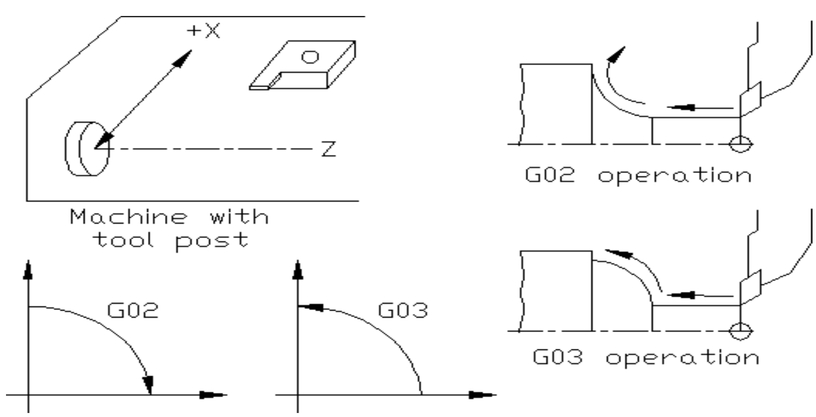
Date of Commencement	SIGNATURE OF STUDENT	**Marks / Remarks:**
Date of finish	SIGNATURE OF STAFF	

CIRCULAR INTERPOLATION – G02/G03

	Data to be given		Command	Meaning
1	Rotation Direction		G02	Clockwise direction (CW)
			G03	Counter clockwise direction (CCW)
2	End point position	Absolute command	X,Z	End point position in the work coordinate system
		Incremental command	U,W	Distance from start point to end point.
3	Radius of arc		R	Radius of arc.

The end point of an arc is specified by address X,Z or U, W and is expressed as an absolute or incremental value. For the incremental value, the coordinate of the endpoint which is viewed from the start point of the arc is specified. The arc center is specified by addresses I and K for the X and Z axis. The numerical value following I,J is always specified as an incremental value. I and K must be signed according to the direction. The radius is specified with address R, if the circular path is greater than 180^0, then R should be negative. For a lathe, because of the characteristics of the turning operation, the circular motion can only be less than 180^0.

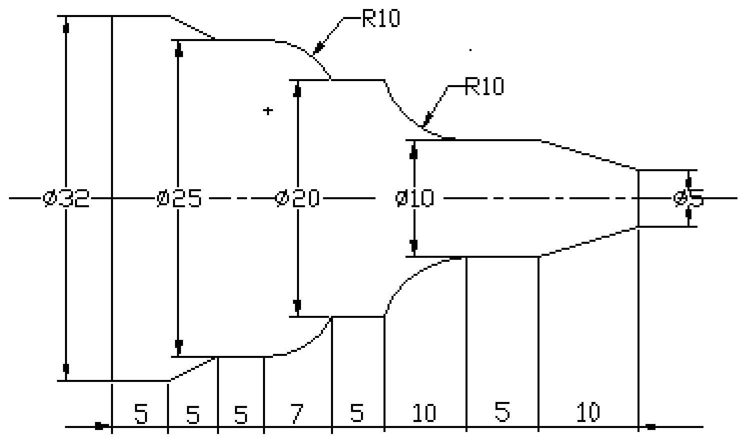
Clockwise and counter clockwise directions of rotations are distinguished on the basis of the rule that when one looks from the positive direction of the axis perpendicular to the plane on which the circular motion is performed, the motion is in clockwise and counter clockwise directions respectively. The clockwise or counter clockwise direction varies in right or left hand coordinate systems as shown in figures below :



EXERCISE -8

CONTOURING

Write a manual part program for Linear and Circular Contour Operation for the component shown in figure below.



DWG. NO. 8

PLANNING AND OPERATIONS SHEET							
BILLET SIZE : 32 x 60				**MATERIAL : Aluminum**			
PROGRAM NO : 1003				**DWG NO : 8**			
SL.NO	**Operation**	**Tool Holder**	**Tool Tip**	**Tool Station No**	**Tool Offset No**	**Spindle Speed, rpm**	**Feed, mm/min**
1	Turning	SDJCR 1212H11	DCMT 11T304	1	1	1200	45

```
(Drawing No .8      (CNC program for Linear and circular interpolation
O1003 ----------------------------------------Program Number 1003
[BILLET X32 Z60 -------------------------- Defining Billet size dia : 32 length 60 mm
G21 G98--------------------------------------Initial settings
G28 U0 W0 ---------------------------------- Going to home position
M06 T0101                         ---------- Selecting Tool No. 1 with offset No. 1
M03 S1200------------------------------------ Setting spindle speed at 1200 rpm
G00 X32 Z1
G00 X5     G01 Z0
G01 X10 Z-10 F45
G02 X20 Z-25 R10 F25              ---------- Clockwise Interpolation – G02
G01 Z-30 F45
G03 U5 Z-37 R10 F25               ---------- Counter Clockwise Interpolation – G03
G01 Z-42 F45
X30 Z-47
Z-52
G28 U0 W0                         -------- Going to home position
M05                               -------- Stop the spindle
M30                              -------- Program stop and rewind.
```

G71 MULTIPLE TURNING CYCLE

Description	Illustration
This multiple turning cycle is used when the major direction of cut is along the 'Z' axis. This cycle causes the profile to be roughed out by turning. Control passes on to after the last block of the profile. Two G71 blocks are needed to specify all the values.	**G71 U(*u1) R** **G71 P Q U(*u2) W F S T** Where u1 – Depth of cut (Radius Designation). R- Relief amount, F – Feed rate, S - Speed P- Line or block number of the start of the final profile. Q- Line or block number of the end point of the final profile, T – Tool number. U2 – Finishing allowance in the X axis. W- Finishing allowance in the Z axis.

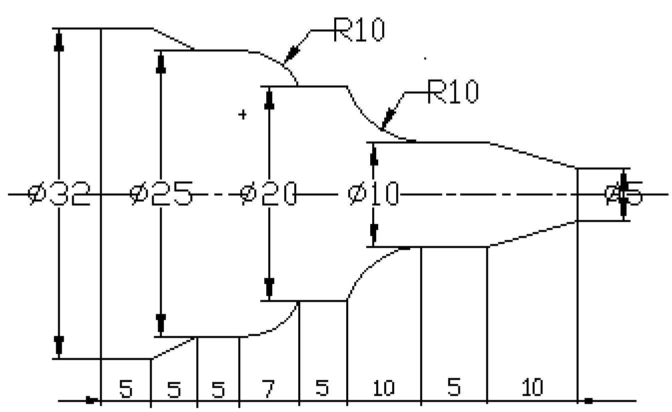
G70 FINISHING CYCLE

Description	Illustration
On completion of roughing out operation using cycles G71, G72 or G76, the material left as a finishing allowances is removed using the finishing cycle G70. the tool path program used as the finishing cycle are the same programming lines that the roughing cycle is based on. A G70 cycle causes a range of blocks to be executed, then control passes to the block after the G70.	**N40 G71 U(*u1) R** **N50 G71 P60 Q120 U(*u2) W F S** **…………….** **…………….** **…………….** **N130 G70 P60 Q120** The 'P' and 'Q' values specifies the 'N' block numbers at the start and end of the profile.

EXERCISE -9

MULTIPLE TURNING CYCLE

Write a manual part program for Multiple Turning Operation for the component shown in figure below.

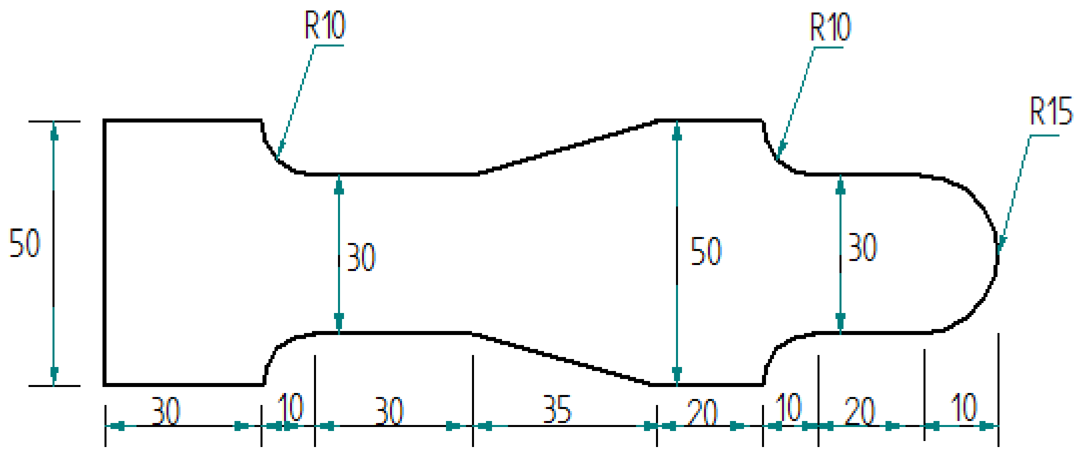


DWG. NO. 9

PLANNING AND OPERATIONS SHEET							
BILLET SIZE : 32 x 60				**MATERIAL : Aluminum**			
PROGRAM NO : 1009				**DWG NO : 9**			
SL.NO	**Operation**	**Tool Holder**	**Tool Tip**	**Tool Station No**	**Tool Offset No**	**Spindle Speed, rpm**	**Feed, mm/min**
1	Multiple Rough turning	SDJCR 1212H11	DCMT 11T304	1	1	1200	35
2	Finishing	SDJCR 1212H11	DCMT 11T302	2	2	1450	25

```
(Drawing No .9
(CNC program for Multiple Turning
O1009 --------------------------------------- Program Number 1009
[BILLET X32 Z60 -------------------------- Defining Billet size dia : 32 length 60 mm
G21 G98---------------------------------------- Initial settings
G28 U0 W0 ---------------------------------- Going to home position
M06 T0101                          ---------- Selecting Tool No. 1 with offset No 1
M03 S1200------------------------------------ Setting spindle speed at 1200 rpm
G00 X32 Z1----------------------------------- Tool moving to tool entry point X32 Z1
(G71 MULTIPLE TURNING
(Depth of cut for each pass U=0.5 mm
(Relief amount R= 1.0 mm
(P and Q: Beginning and end of cycle sequence Nos.
(Allowances on X(U) and Z(W) axis=0.1 mm respectively.
(Feed rate= 35 mm/min.
G71 U0.5 R1
G71 P10 Q20 U0.1 W0.1 F35
N10 G01 Z0
G01 Z0
G01 X10 Z-10
G01 Z-15
G02 X20 Z-25 R10
G01 Z-30
G03 X25 Z-37 R10
G01 Z-42
X30 Z-47
N20 Z-52
G28 U0 W0
M06 T0202---------------------------------- Using RH Finishing tool.
M03 S1450
G00 X32 Z1
G70 P10 Q20 U0 W0 S2500 F30
G28 U0 W0 ---------------------------------- Going to home position
M05 ------------------------------------------ Stop the spindle
M30 ------------------------------------------ Program stop and rewind.
```

Write a manual part program for Multiple Turning Operation for the component shown in figure below



Date of Commencement	SIGNATURE OF STUDENT	**Marks / Remarks:**
Date of finish	SIGNATURE OF STAFF	

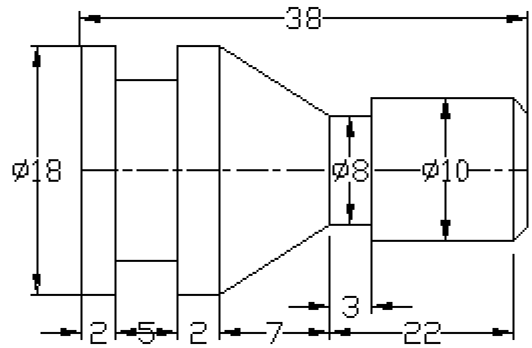
G75 EXTERNAL GROOVING CYCLE

Description	Illustration
This cycle is designed for Outer/ Inner diameter drilling. The drill entering the workpiece by a predetermined amount then backing off by another set amount to provide breaking and allowing swarf to clear the drill flutes. The cycle is commanded by two distinct lines of data.	**G75 R** **G75 X(u) Z(w) P Q F** Where R – Return amount X – Total depth along X axis(absolute) U – Total depth along X axis(Incremental) Z – Total width along Z axis(absolute) W – Total width along Z axis (Incremental) P – Peck increment in X axis in microns. Q – Stepping distance in Z axis in microns. F- Feed rate in mm.

EXERCISE -10

EXTERNAL GROOVING

Write a manual part program for External Grooving Operation for the component shown in figure below.

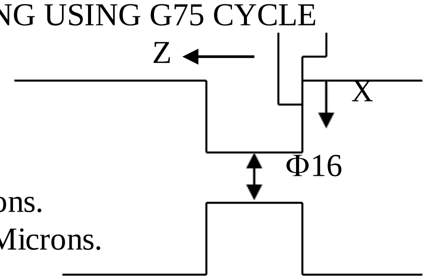


DWG. NO. 10

PLANNING AND OPERATIONS SHEET							
BILLET SIZE : 22 x 60				**MATERIAL : Aluminum**			
PROGRAM NO : 1010				**DWG NO : 10**			
SL.NO	**Operation**	**Tool Holder**	**Tool Tip**	**Tool Station No**	**Tool Offset No**	**Spindle Speed, rpm**	**Feed, mm/min**
1	Multiple Rough turning	SDJCR 1212H11	DCMT 11T304	1	1	1200	45
2	Finishing	SDJCR 1212H11	DCMT 11T302	2	2	1450	25
3	Grooving	HSS	3 mm width	5	5	750	15

(Drawing No .10
O1010 --------------------------------------- Program Number 1010
[BILLET X22 Z60 -------------------------- Defining Billet size dia : 22 length 60 mm
G21 G40 G98--------------------------------- Initial settings
G28 U0 W0 ---------------------------------- Going to home position
M06 T0101 ---------- Selecting Tool No. 1 with offset No 1
M03 S1200------------------------------------ Setting spindle speed at 1200 rpm
G00 X22 Z1----------------------------------- Tool moving to tool entry point X22 Z1
G71 U0.5 R1---------------------------------- MULTIPLE TURNING
G71 P10 Q20 U0.1 W0.1 F45
N10 G01 X8
G01 Z0

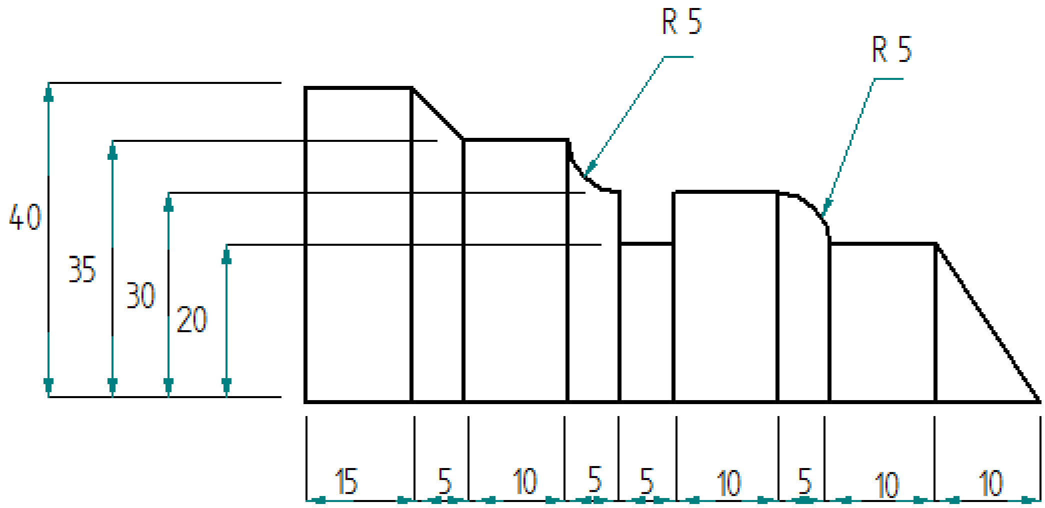
```
X10 Z-1
Z-22
X18 Z-29
N20 Z-38
G28 U0 W0
M06 T0202                    ----------- CALLING FINISHING TOOL
M03 S1450
G00 X22 Z1
G70 P10 Q20 F25              ------------ Calling Finishing cycle.
G28 U0 W0                    ------------ GROOVING OPERATION USING G81
M06 T0505                    ------------ Calling 3 mm GROOVING TOOL
M03 S750
G00 X12 Z-22
G81 X10 F20
X9.75
X9.5
X9.25
X9
X8.75
X8.5
X8.25
X8
G00 X19 Z-34
G75 R1                       ----------- GROOVING USING G75 CYCLE
G75 X16 W-2 P100 Q1500 F15
```

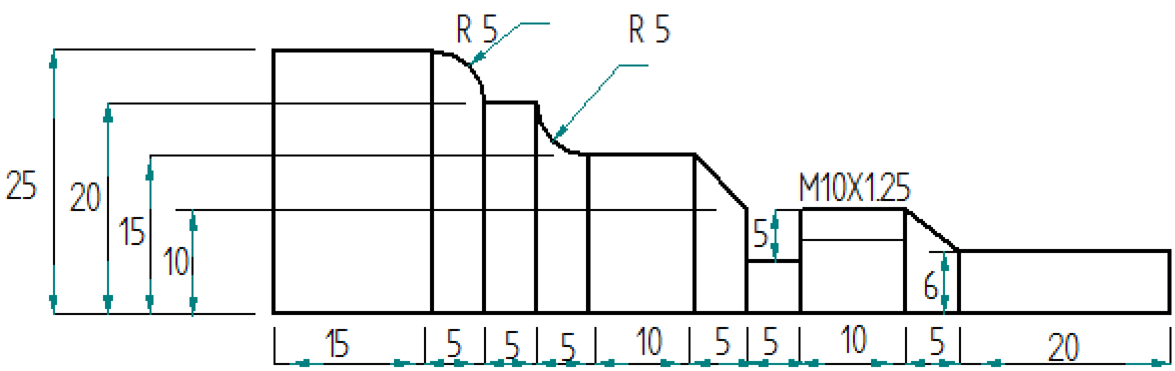



```
(Relief amount, R=1.0 mm.
(Depth of Groove, X= 2mm.
(P- Peck increment along X axis 0.1 mm = 100 Microns.
(Q – Stepping distance along Z axis 1.5 mm = 1500 Microns.

G28 U0 W0                    -------- Going to home position
M05                          -------- Stop the spindle
M30                          -------- Program stop and rewind.
```

Date of Commencement	SIGNATURE OF STUDENT	**Marks / Remarks:**
Date of finish	SIGNATURE OF STAFF	

Write a manual part program for the component shown in figures below.





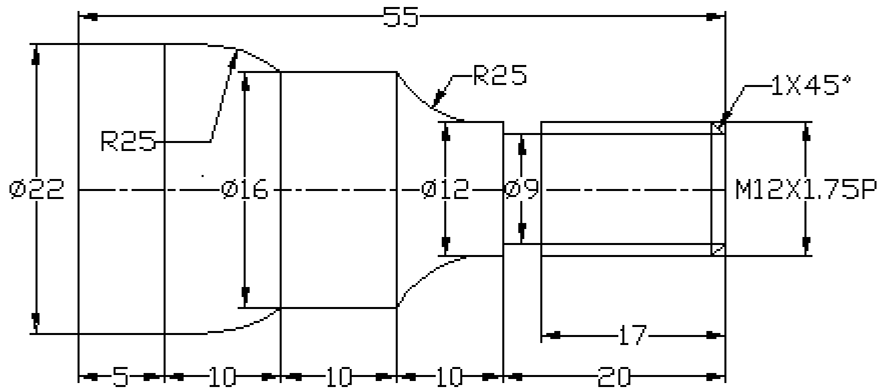
G76 THREADING CYCLE

Description	Illustration
This is a "Box type" cycle that is repeated a given number of times. After the first pass subsequent passes cut with one edge of the threading tool only to reduce the load at the tool tip. This cycle requires two distinct blocks of data. When the cutting depth of one cycle becomes smaller than the limit, the actual amount of cut is clamped at the minimum cut depth.	**G76 P(m)(r)(a) Q(q1) R(r1)** **G76 X(x) Z(z) P(p2) Q(q2) F** Where m – Repetitive count in finishing (1 to 99) r – Chamfering amount(0.01 to 9.91) a – Angle of tool tip(80^0,60^0,55^0,30^0,29^0 & 0^0) q1 – Minimum cutting depth. r1 – Finishing allowance. x – Finished depth of thread z – End position of thread p2 – Height of the thread as a radius value x 1000, as the controller accepts this value in microns. Eg. 1.02 mm becomes P1020 q2 – Depth of first cut as a radius value X 1000, value in microns, F- Lead or pitch of thread.

EXERCISE -11

EXTERNAL MULTIPLE THREADING

Write a manual part program for External Threading operation for the component shown in figure below.



DWG. NO. 11

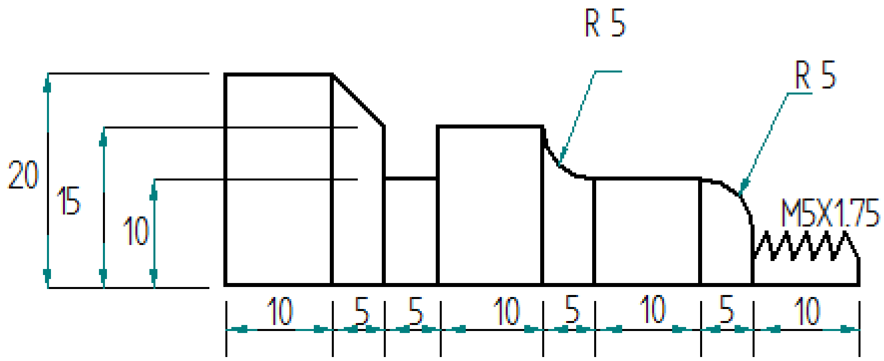
PLANNING AND OPERATIONS SHEET							
BILLET SIZE : 22 x 60				**MATERIAL : Aluminum**			
PROGRAM NO : 1011				**DWG NO : 11**			
SL.NO	**Operation**	**Tool Holder**	**Tool Tip**	**Tool Station No**	**Tool Offset No**	**Spindle Speed, rpm**	**Feed, mm/min**
1	Multiple Rough turning	SDJCR 1212H11	DCMT 11T304	1	1	1200	35
2	Finishing	SDJCR 1212H11	DCMT 11T302	2	2	1450	25
3	Grooving	HSS	3 mm width	5	5	750	25
4	Threading	HSS		7	7	500	25

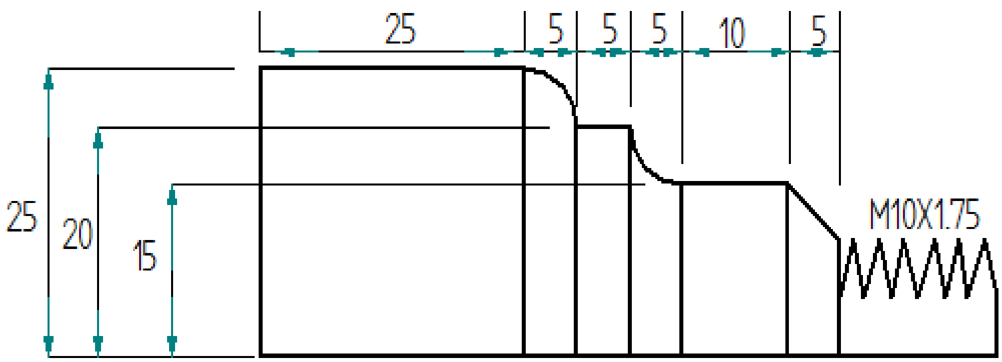
(Drawing No .11
(CNC program for Multiple Turning and Threading
O1011 --------------------------------------- Program Number 1011
[BILLET X22 Z60 -------------------------- Defining Billet size dia : 22 length 60 mm
G21 G98-- Initial settings
G28 U0 W0 ---------------------------------- Going to home position
M06 T0101 ---------- Selecting Tool No. 1 with offset No 1
M03 S1200------------------------------------ Setting spindle speed at 1200 rpm
G00 X22 Z1
G01 Z0

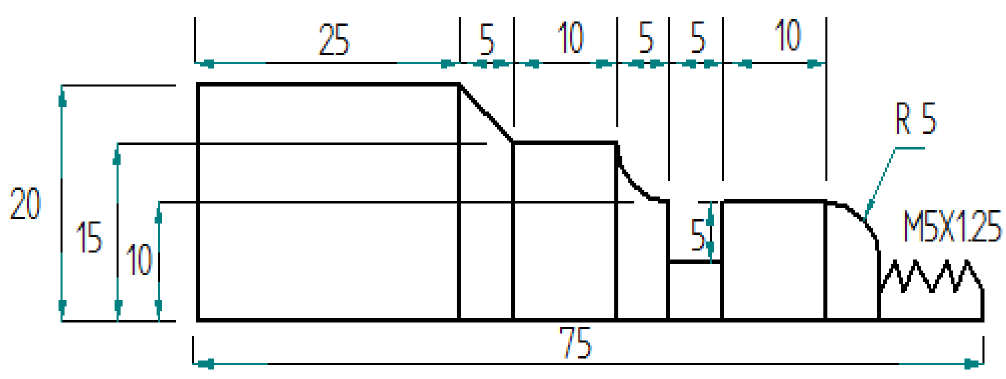
```
G71 U0.5 R1                        ---------- MULTIPLE TURNING
G71 P10 Q20 U0.1 W0.1 F35
N10 G01 X10
G01 Z0
X12 Z-1
Z-20
G02 X16 Z-30 R25
G01 Z-40
G03 X22 Z-50 R25
N20 G01 Z-55
G28 U0 W0
M06  T0202                         ----------- CALLING RH FINISHING TOOL
M03  S1450
G00 X22 Z1
G70 P10 Q20 F25                    ------------ FINISHING OPERATION
G28 U0 W0
M06 T0505                          ---------- CALLING 2mm Width Grooving tool
M03 S650
G00 X13 Z-19
G75 R1
G75 X9 Z-20 F25                    ---------- GROOVING OPERATION G75
G28 U0 W0
M06 T0707                          ----------- CALLING THREADING TOOL
G00 X17 Z23
G76 P031560 Q20 R0.15              ---------- MULTIPLE THREADING CYCLE.
G76 X9.853 Z-19 P1073 Q30 F1.75
(03 – Number of passes for finishing operation
(15 -  Chamfer amount or pull out angle        (60 – Angle of the thread, deg
(Q – Minimum cutting depth = 250 microns ( .25 mm)
(R -  Finishing allowances = 0.15 mm      (X – Core diameter = 9.853 mm for M12
(Z – Length of thread=19 mm        (P - Height of thread = 1073 microns (1.073 mm)
(Q – Depth of cut for first pass = 300 microns (0.3 mm)
(F – Pitch of the thread = 1.75 mm
G28 U0 W0 -------------------------------- Going to home position
M05 ------------------------------------------ Stop the spindle
M30 ------------------------------------------ Program stop and rewind.
```

Date of Commencement	SIGNATURE OF STUDENT	**Marks / Remarks:**
Date of finish	SIGNATURE OF STAFF	

Write a manual part program for External Threading operation for the component shown in figures below.







INTERNAL OPERATIONS

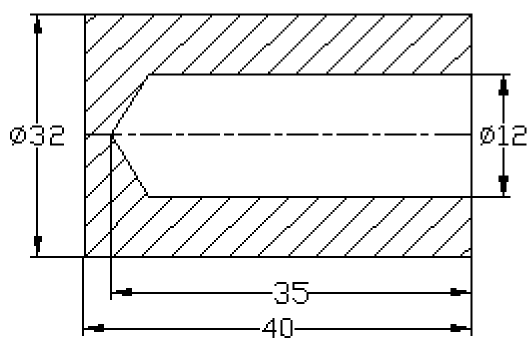
G74 END FACE PECK DRILLING

Description	Illustration
This cycle is designed for deep hole drilling, the drill entering the workpiece by a predetermined amount then backing off by another set amount to provide breaking and allowing swarf to clear the drill flutes. The cycle is commanded by two distinct lines of data	**G74 R(r1)** **G74 X0 Z(W) Q(q) R(r2) F** Where r1 – Return amount Z – Total depth(absolute) W – Total depth (Incremental) q – Depth of cut (incremental, unsigned) F- Feed rate

EXERCISE -12

PECK DRILLING

Write a manual part program for Peck drilling operation for the component shown in figure below.



DWG. NO. 12

PLANNING AND OPERATIONS SHEET							
BILLET SIZE : 32 x 60				**MATERIAL : Aluminum**			
PROGRAM NO : 1011				**DWG NO : 12**			
SL.NO	**Operation**	**Tool Holder**	**Tool Tip**	**Tool Station No**	**Tool Offset No**	**Spindle Speed, rpm**	**Feed, mm/min**
1	Center Drill	6 mm	-	6	6	1200	20
2	Drilling	12 mm	-	8	8	800	20

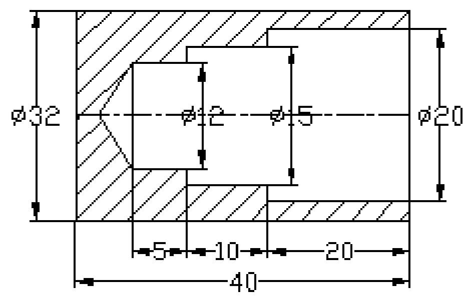
```
(Drawing No .12
(CNC program for Drilling operation – G74 CYCLE
O1011                          --------- Program Number 1011
[BILLET X32 Z60                --------- Defining Billet size dia : 32 length 60 mm
G21 G98                        --------- Initial settings
G28 U0 W0                      --------- Going to home position
M06 T0606                      ---------- Using 6 mm center drill with tool no 1.
M03 S1200                      ---------- Setting spindle speed at 1200 rpm
G00 X0 Z2                      ---------- Tool moving to tool entry point X0 Z2
G01 Z0
G74 R1                         ---------- PECK DRILLING CYCLE
G74 X0 Z-5 Q500 F20
(R = Relief amount = 1.0 mm
(X, Z = Position of the bottom of the hole 0, -5
(Q = Depth of cut for each pass – 500 microns (0.5 mm)
G28 U0 W0
M06 T0808                      ----------- Using 12 mm drill.
M03 S800
G00 X0 Z2
G74 R1
G74 X0 Z-35 Q500 F20
G28 U0 W0                      -------- Going to home position
M05                            -------- Stop the spindle
M30                            -------- Program stop and rewind.
```

Date of Commencement	SIGNATURE OF STUDENT	**Marks / Remarks:**
Date of finish	SIGNATURE OF STAFF	

EXERCISE -13

STEP BORING

Write a manual part program for Step Boring operation for the component shown in figure below.



DWG. NO. 13

PLANNING AND OPERATIONS SHEET							
BILLET SIZE : 32 x 60				**MATERIAL : Aluminum**			
PROGRAM NO : 1013				**DWG NO : 13**			
SL.NO	**Operation**	**Tool Holder**	**Tool Tip**	**Tool Station No**	**Tool Offset No**	**Spindle Speed, rpm**	**Feed, mm/min**
1	Center Drill	6 mm	-	1	1	1200	20
2	Drilling	12 mm Drill	-	8	8	800	15
3	Boring	10 mm Boring bar	-	2	2	1200	20

(Drawing No .13
(CNC program for Internal operation, Face Drilling, Step boring cycle
O1013 -------------------------------------- Program Number 1013
[BILLET X32 Z60 -------------------------- Defining Billet size dia : 32 length 60 mm
G21 G98--------------------------------------- Initial settings
G28 U0 W0 ---------------------------------- Going to home position
M06 T0101------------------------------------Using 6 mm center drill with tool no 1.
M03 S1200------------------------------------ Setting spindle speed at 1200 rpm
G00 X0 Z2 ------------------------------------Tool moving to tool entry point X0 Z2 at rapid traverse.
G01 Z0
G74 R1
G74 X0 Z-5 Q500 F20
(R = Relief amount = 1.0 mm

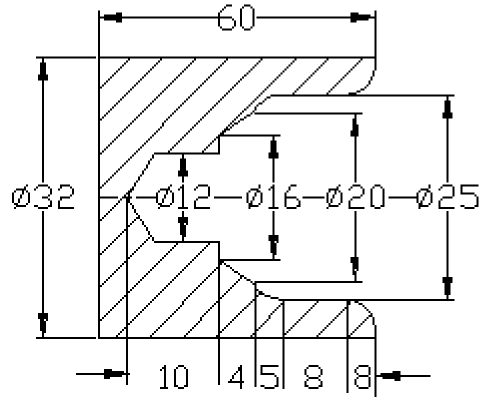
```
(X, Z = Position of the bottom of hole (0,-5)
(Q = Depth of cut for each pass – 500 microns (0.5 mm)
G28 U0 W0
M06 T0808                  ----------- Using 12 mm drill.
M03 S1000
G00 X0 Z1
G01 Z0
G74 R1
G74 X0 Z-35 Q500 F50
G00 X0 Z1
G28 U0 W0
M06 T0202                  --------- CALLING 10 MM DIA BORING TOOL
M03 S800
G00 X12 Z1
G01 Z0
G90 X13 Z-30 F20           ---------- INTERNAL BORING USING G90
X14
X15
X16 Z-20
X17
X18
X19
X20
G28 U0 W0                  -------- Going to home position
M05                        -------- Stop the spindle
M30                        -------- Program stop and rewind.
```

Date of Commencement	SIGNATURE OF STUDENT	**Marks / Remarks:**
Date of finish	SIGNATURE OF STAFF	

EXERCISE -14

INTERNAL MULTIPLE TURNING

Write a manual part program for Internal Multiple Turning operation for the component shown in figure below.



DWG. NO. 14

PLANNING AND OPERATIONS SHEET							
BILLET SIZE : 32 x 60				**MATERIAL : Aluminum**			
PROGRAM NO : 1014				**DWG NO : 14**			
SL.NO	**Operation**	**Tool Holder**	**Tool Tip**	**Tool Station No**	**Tool Offset No**	**Spindle Speed, rpm**	**Feed, mm/min**
1	Drilling	12 mm Drill	-	1	1	700	15
2	Boring	10 mm	-	2	2	800	20

```
(Drawing No .14
(CNC program for Internal operation, Boring cycle
O1014                    --------- Program Number 1014
[BILLET X32 Z60          --------- Defining Billet size dia : 32 length 60 mm
G21 G98                  --------- Initial settings
G28 U0 W0                --------- Going to home position
M06 T0101                ---------- Using 6 mm center drill with tool no 1.
M03 S1200                ---------- Setting spindle speed at 1200 rpm
G00 X0 Z2                ---------- Tool moving to tool entry point X0 Z2 at
G01 Z0                              rapid traverse.
G74 R1
G74 X0 Z-5 Q500 F50
G28 U0 W0
M06 T0202                ----------- Using 12 mm drill.
```

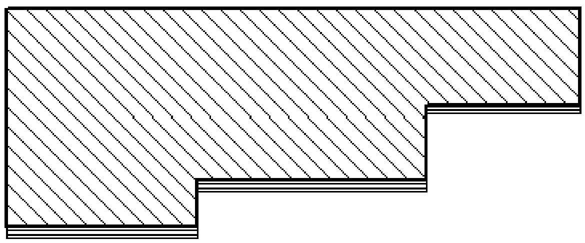
```
M03 S1000
G00 X12 Z1
G01 Z0
G74 R1
G74 X0 Z-35 Q500 F50
G28 U0 W0
M06 T0303                    --------- CALLING 10 MM DIA BORING TOOL
M03 S800
G00 X12 Z1
G01 Z0
G71 U0.2 R1                  --------- BORING OPERATION
G71 P10 Q20 U0.1 W0.1 F20
N10 G01 X30
G02 X25 Z-8 R8 F15
G01 Z-16 F20
G03 X20 Z-21 R8 F15
G01 X16 Z-31 F20
N20 G01 X12
G70 P10 Q20 U0 W0 S1000 F30          -------- CALLING FINISHING CYCLE.
G28 U0 W0                    -------- Going to home position
M05                          -------- Stop the spindle
M30 ------------------------------------------ Program stop and rewind.
```

Date of Commencement	SIGNATURE OF STUDENT	**Marks / Remarks:**
Date of finish	SIGNATURE OF STAFF	

EXERCISE -15

INTERNAL THREADING

Write a manual part program for Internal Threading operation for the component shown in figure below.



Dimension of workpiece 25 X 70
Internal Thread M 20x1

DWG. NO. 15

PLANNING AND OPERATIONS SHEET							
BILLET SIZE : 25 x 70				MATERIAL : Aluminum			
PROGRAM NO : 1011				DWG NO : 11			
SL.NO	Operation	Tool Holder	Tool Tip	Tool Station No	Tool Offset No	Spindle Speed, rpm	Feed, mm/min
1	Center Drill	6 mm	-	1	1	800	20
2	Drilling	12 mm Drill	-	2	2	1200	15
3	Boring	10 mm Boring bar	-	3	3	800	20
4	Threading	HSS		4	4	1000	25

```
(Drawing No .15
(CNC program for Internal Threading
O1011                        --------- Program Number 1011
[BILLET X25 Z70              --------- Defining Billet size dia : 25 length 70 mm
G21 G98                      --------- Initial settings
G28 U0 W0                    --------- Going to home position
M06 T0101                    ---------- Selecting Tool No. 1 with offset No 1
M03 S800                     ---------- Setting spindle speed at 1200 rpm
G00 X0 Z1                    ---------- Tool moving to tool entry point X22 Z1 at
G01 Z0                                  rapid traverse.
G74 R1                       ----------  6 mm Drill Bit
G74 X0 Z-5 Q500 F50
G28 U0 W0
```

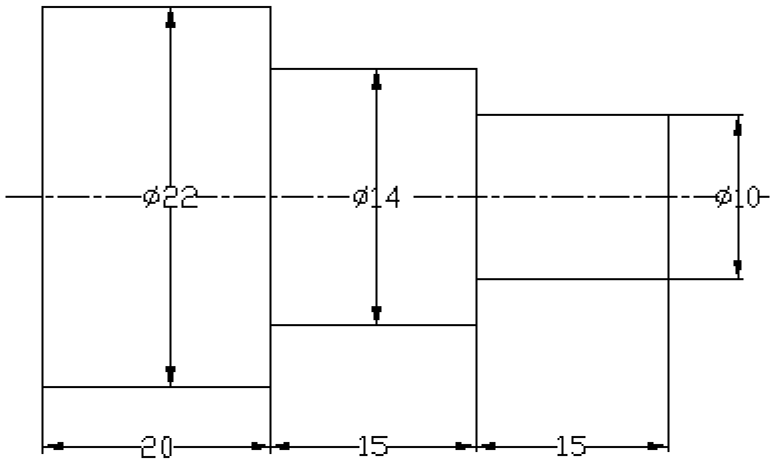
```
M06 T0202
M03 S1200
G00 X0 Z1
G01 Z0
G74 R1                          ---------- 12 mm Drill Bit
G74 X0 Z-30 Q500 F50
G00 X0 Z1
G28 U0 W0
M06 T0303                       ---------- Boring Tool
M03 S800
G00 X12 Z1
G01 Z0
G90 X13 Z-20
X14
X15
X16
X17
X18
X18.774
G28 U0 W0
M06  T0404                      ----------- Threading Tool
M03  S1000
G00 X16 Z1
G01 Z0
G76 P031560 Q50 R0.1
G76 X10 Z-20 P0613 Q100 F1
G28 U0 W0                       -------- Going to home position
M05                             -------- Stop the spindle
M30                             -------- Program stop and rewind.
```

Date of Commencement	SIGNATURE OF STUDENT	**Marks / Remarks:**
Date of finish	SIGNATURE OF STAFF	

EXERCISE -16

PARTING OFF

Write a manual part program for turning and parting off operation through subprograms for the component shown in figure below.



DWG. NO. 16

PLANNING AND OPERATIONS SHEET							
BILLET SIZE : 22 x 60				**MATERIAL : Aluminum**			
PROGRAM NO : 1015				**DWG NO : 16**			
SL.NO	**Operation**	**Tool Holder**	**Tool Tip**	**Tool Station No**	**Tool Offset No**	**Spindle Speed, rpm**	**Feed, mm/min**
1	Turning	SDJCR 1212H11	DCMT 11T304	1	1	1000	45
2	Grooving	HSS	2 mm	5	5	750	25

```
(Drawing No .16
(CNC program for parting off using subprograms
O1015                    --------- Program Number 1015
[BILLET X22 Z60          --------- Defining Billet size dia : 22 length 60 mm
G21 G98                  --------- Initial settings
G28 U0 W0                --------- Going to home position
M06 T0101                ---------- Using RH Roughing tool
M03 S1000                ---------- Setting spindle speed at 1200 rpm
G00 X22 Z0               ---------- Tool moving to tool entry point X22 Z0.
M98 P0101000             --------- Calling subprogram for turning[01000] 10 times
G00 X22 Z-15
M98 P0061000             ---------- Calling subprogram for turning[01000] 6 times
(PARTING OFF OPERATION
```

```
G28 U0 W0
M06 T0505                    ---------- Calling grooving tool with 2 mm width.
M03 S750
G00 X23 Z-32
M98 P0421021                 --------- Calling subprogram '1021' 42 times.
G00 X22
G28 U0 W0                    -------- Going to home position
M05                          -------- Stop the spindle
M30                          -------- Program stop and rewind.
O1000                        -------- SUBPROGRAM FOR TURNING
G90 U-1 W-15 F45
G01 U-1
M99
O1021                        -------- SUBPROGRAM FOR PARTING
G01 U-0.5 F50
M99
```

Date of Commencement	SIGNATURE OF STUDENT	**Marks / Remarks:**
Date of finish	SIGNATURE OF STAFF	

COMPUTERISED NUMERICAL CONTROL MILLING PART PROGRAMMING FUNDAMENTALS

PART PROGRAMMING GEOMETRY

A. COORDINATE SYSTEM FOR A CNC MILL

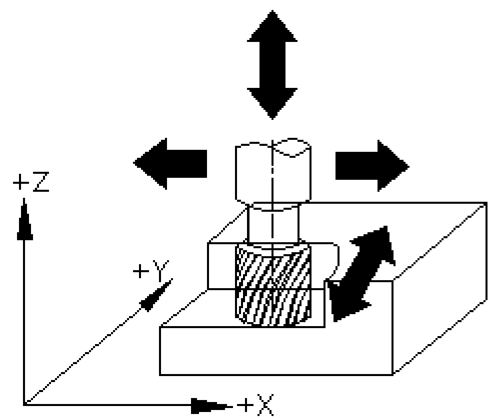
Machining of a workpiece by an NC program requires a coordinate system to be applied to the machine tool. There are three planes in which movement can take place.

- Longitudinal
- Vertical
- Transverse

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

- Axis X
- Axis Y
- Axis Z

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



COORDINATE SYSTEM FOR MILLING

B. ZERO POINTS AND REFERENCE POINTS

MACHINE ZERO POINT (M): This is specified by the manufacturer of the machine. This is the zero point for the coordinate systems and reference points in the machine. The machine zero point can be the center of the table or a point along the edge of the traverse range as shown in figure the position of the machine zero point generally varies from manufacture. The precise position of the machine zero point as well as the axis direction must therefore be taken from the operating instructions provided for each individual machine.

REFERENCE POINT (R): This point serves for calibrating and for controlling the measuring system of the slides as tool traverses. The position of the reference point is accurately predetermined in every traverse axis by the trip dogs and limit switches. Therefore, the reference point coordinates always have the same, precisely known numerical value in relation to the machine zero point. After initiating the control system, the reference point must always be approached from all axes to calibrate the traverse measuring system. If current slide and tool position data should be lost in the control systems, for example, through an electrical failure, the machine must again be positioned to the reference point to re-establish the proper positioning values.

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

CNC MILL PROGRAMMING

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

M70 X MIRROR ON : M70 sets x axis mirroring about the current x axis position.

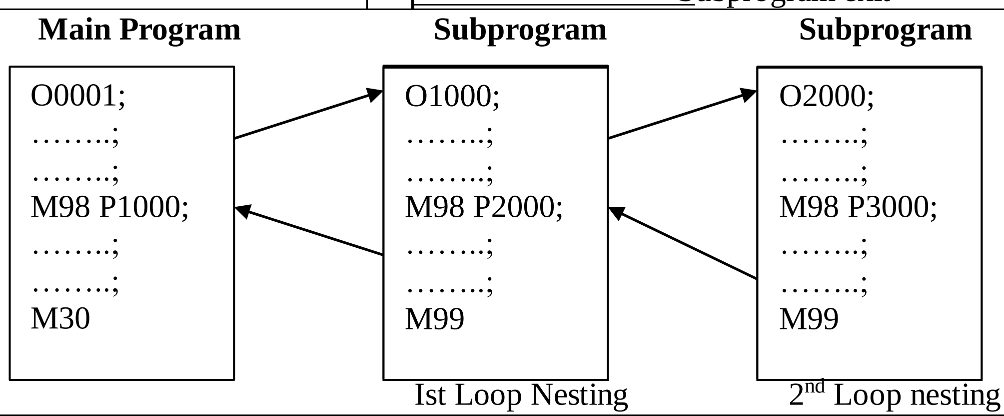
M71 Y MIRROR ON : M71 sets Y-axis mirroring about the current Y axis position.

M80 X MIRROR OFF : M80 disables X axis mirroring.

M80 Y MIRROR OFF : M80 disables X-axis mirroring.

SUBPROGRAM CALL / EXIT- M98 / M99

Main program	Subprogram
A Program is divided into main program and subprogram. Normally the CNC operates according to the main program but when a command calling a subprogram is encountered in the main program control is passed to the subprogram. When a command indicting to return to the main program is encountered in the subprogram is encountered in the subprogram, control is returned to the main program. The first block of program/subroutine must contain a program number "o"	When a program contains certain fixed sequences or frequently repeated patterns, these sequences or patterns may be entered into memory as a subprogram to simplify programming. A subprogram can call a subprogram it is regarded as a one-loop subprogram call. FORMAT: O0001; ………; ………; M98 POOOOOOO Subprogarm Number Number of repetitions Subprogram call ………. ………. M99 Subprogram exit



PREPARATORY FUNCTIONS (G CODES)

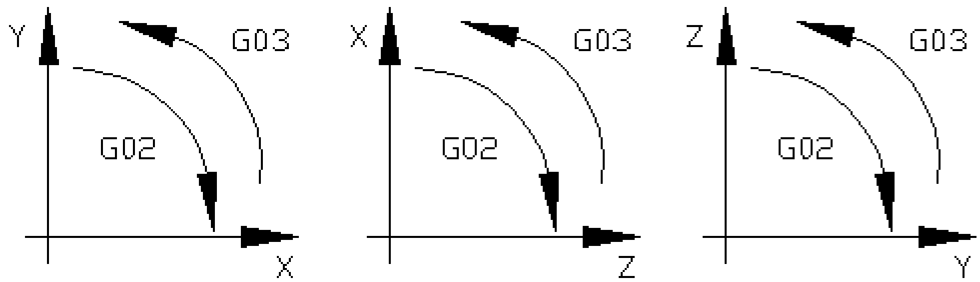
G CODES	
G00	Positioning (Rapid Transverse)
G01	Linear Interpolation (Feed)
G02	Circular Interpolation (CW)
G03	Circular Interpolation (CCW)
G04	Dwell
G20	Inch Data Input
G21	Metric Data Input
G28	Reference point return
G40	Tool nose radius compensation cancel
G41	Tool nose radius compensation left

G42	Tool nose radius compensation right
G43	Tool length compensation + direction
G44	Tool length compensation - direction
G73 **G74** **G76** **G80** **G81** **G82** **G83** **G84** **G85** **G86** **G87** **G88** **G89**	Peck drilling cycle Counter tapping cycle Fine Boring Canned cycle cancel Drilling cycle, spot boring Drilling cycle, counter boring Peck drilling cycle Tapping cycle Boring cycle Boring cycle Back boring cycle Boring cycle Boring cycle
G90 **G91**	Absolute command Incremental command
G92	Programming of Absolute zero point.
G94 **G95**	Feed per minute Feed per revolution
G98 **G99**	Return to initial point in canned cycle Return to R point in canned cycle.

CIRCULAR INTERPOLATION (G02/G03)

Sl no.	Data to be given		Command	Meaning
1	Plane selection		G17	Specification of arc on XY plane
			G18	Specification of arc on ZX plane
			G19	Specification of arc on YZ plane
2	Direction of rotation		G02	Clockwise direction
			G03	Counter clock wise direction
3	End point position	G90 Mode	Two of the X,Y and Z axis	End point position in the Work Coordinate System
		G91 Mode	Two of the X,Y and Z axes	
4	Dist. From start point to center		Two of the I,J and K axes	Distance from start point to end point
	Arc radius		R	Arc radius
5	Feed rate		F	Velocity along arc

The view is from the positive direction of the Z axis(Y axis or X axis) to the negative direction on XY plane (ZX plane or YZ plane) in the right hand Cartesian coordinate system. The following sketch shows the CW and CCW directions in different planes.

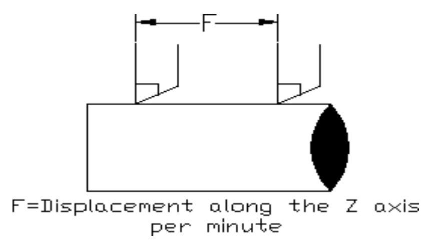


CW and CCW directions

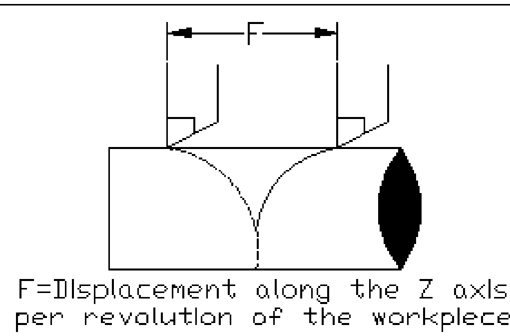
G40, G41, G42 TOOL NOSE RADIUS COMPENSATION

Description
On Modern CNC machines, special calculation functions or cutter radius compensation codes are provided to allow a user to utilize part profile coordinates obtainable from the part drawing to program a contouring motion. These are the G41 and G42 codes for tool radius compensation on the left and right hand sides of a profile. A left or right compensation is based on the fact that the tool is on left or right hand side when one goes along the part profile in the direction specified by the contouring motion statements in the program. A G40 code is provided to cancel the cutter radius compensation.

G94 PER MINUTE FEED

Description	Illustration
With the per minute feed mode, tool feed rate per minute is directly commanded by numerical value after F. the "F" value specifies the feed rate in millimeters, or inches per minute.	

G95 PER REVOLUTION FEED

Description	Illustration
This command coupled with the F word is used to specify a federate per revolution. The feed rate is changed whenever the spindle changes. This can be in mm/rev to inch/rev. The feed rates available in the DENFORD FANUC simulation are 0.01-200 mm/min. Recommended federates are published by tool and cutter manufacturers, along with recommended cutting speeds. If the feed rate is expressed as mm/rev. a simple calculation can be used to convert to mm/min. Feed, mm/min = feed(mm/rev) x spindle speed(rpm)	 F= the displacement along the z axis per revolution of the workpiece Example G(% S1200 G01 X10.0 f0.3 This sets the feed rate to 360(1200*0.3)

CANNED CYCLES (G73, G74, G76, G80-89)

A canned cycle simplifies the program using a single block with a G code to specify the machining operations usually specified in several blocks. The following table give list of canned cycles.

Canned cycles				
G code	**Drilling (-z direction)**	**Operation at the bottom of a hole**	**Retraction (+z direction)**	**Application**
G73	Intermittent feed	-	Rapid traverse	High speed peck Drilling cycle
G74	Feed	Dwell spindle CW	Feed	Left hand tapping cycle
G76	Feed	Oriented spindle stop	Rapid traverse	Fine boring cycle
G80	-	-	-	cancel
G81	Feed	-	Rapid traverse	Drilling cycle spot drilling cycle
G82	Feed	Dwell	Rapid traverse	Drilling cycle counter drilling cycle
G83	Intermittent feed	-	Rapid traverse	Peck drilling cycle
G84	Feed	Dwell spindle CCW	Feed	Tapping cycle
G85	Feed		Feed	Boring cycle
G86	Feed	Spindle stop	Rapid traverse	Boring cycle
G87	Feed	Spindle CW	Rapid traverse	Back Boring cycle
G88	Feed	Dwell-spindle stop	Manual	Boring cycle
G89	Feed	-	Feed	Boring cycle

G90 ABSOLUTE MOVEMENT

Description	Illustration
All future movement will be absolute until overridden by a G91 instruction. This is the default setting.	Example: G90 G01 X30 Y0 The position becomes X30 Y0

G91 INCREMENT MOVEMENT

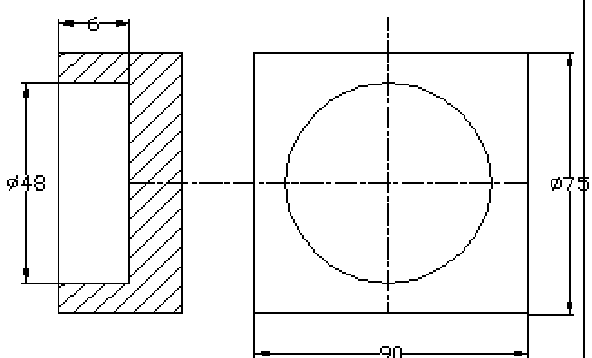
Description	Illustration
All future movement will be incremental until over-ridden by a G90 instruction	Example: G90 G01 X15 Y0 G91 G01 X2 Y0 The position becomes X2 Y0

G73 FAST PECK DRILLING CYCLE

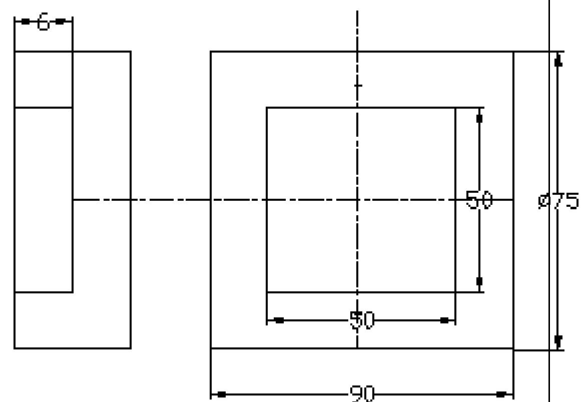
Description	Illustration
When drilling a deep hole, the drill should be retracted occasionally to avoid congestion of	G73 X(*x) Y(*z) P(*p) Q(*q) R(*r) F(*f) Where

chips between hole and the drill. Since the Z-axis direction intermittent feed simplifies chip disposal and permits a very small retraction value to be set in deep hole drilling, efficient machining is performed. Retraction is performed at the rapid traverse rate. Example: M06 T03 M03 S1500 G90 G00 X10 Y10 Z10 G99 G73 X10 Y10 z-20 P500 Q0.5 R2 F50 G80	*x and *y = the next hole position to drill at. *z= the depth of the hole, which is the absolute distance in G90 mode and the incremental coordinate from R point in the G91 mode. *p= dwell time in sec. *q= the depth of cut for each peck drill always a positive incremental value. *r= the Z coordinates of the R point (in G90 mode) or the incremental Z coordinates from the initial point to the R point(in G91 mode). *f = feed, mm/min.

G170-G171 CIRCULAR POCKETING

Description	Illustration
Using this cycle circular pocketing can be done. When the tool has finished cutting, the tool retracts 1mm in Z axis. Moves to the center of the circular pocket at rapid traverse, retracts again in the Z axis, then moves away to the datum point. It requires two blocks. **G170 R0 P0 Q3 X0 Y0 Z-6 I0 J0 K-24** **G171 P75 S2000 R50 F150 B2500 J150** 	G170 r(*r1) P(*p1) Q(*q1) X(*x1) Y(*y1) Z(*z1) I(*i1) J(*j1) K(*k1) G171 P(*p2) S(*s2) R(*r2) F(*f2) B(*b2) J(*j2) Where *r1= position of tool to start cycle ie., for flat surface *p1=0(roughing), = finishing *q1= peck increment for each cut(always a + value) *x1, *y1, *z1: coordinates of center of circular pocket *i1: finishing allowance for side J1: finishing allowance for pocket base *k1= radius of circular pocket(positive value for CW arc) *p2=cutter movement percentage for next step (ex:50,75) *s2= roughing spindle speed, rpm *r2=Roughing feed in z direction for each cut *f2= roughing feed in XY directions mm/min *b2= finish spindle speed, rpm *j2= finishing feed, mm/min

G172-G173 RECTANGULAR POCKETING

Illustration	Example
G172 I(*i1) J(*j1) K(*k1) P(*p1) Q(*q1) R(*r1) X(*x1) Y9*y1) Z(*z1) G173 I(*i2) K(*k2) P(*p2) T9*t2) S(*s2) R(*r2) F(*f2) B9*b2) J(*j2) Z(*z2) Where *I1= length of pocket in x direction *j1=length of pocket in Y direction *k1= corner radius(always zero) *p1-0 Roughing=1(finishing) *q1=depth of cut for each pass (1, 1.5,2) *r1=Absolute depth from the surface(Z R point) *x1= pocket cornerX *y1=Pocket corner Y *z1=absolute z base finish allowance *****p2= cutter width percentage(50,75 etc.,) *t2= tool number *s2= spindle speed rpm *r2= roughing feed in Z, mm/min *f2= roughing feed along XY mm/min *b2= finishing spindle speed rpm., *j2= finishing feed mm/min *z2=safety Z position.	**G172 I-50 J-50 K0 P0 Q3 R0 X-25 Y-25 Z-6** **G173 I0 K0 P75 T1 S2500 R75 F250 B2500 J200 Z5** 

STRUCTURE OF A CNC PROGRAM

CNC program consists of three parts: program start-up, body and end of the program.

1. **PROGRAM START-UP**

O1000
[BILLET X100 Y100 Z10
[TOOLDEF T1 D5 T2 D10
[EDGEMOVE X-50 Y-50
G21/G20 G40 G49 G80
G94/G95
G50 S3500
G91 G28 Z0
G28 X0 Y0
M06 T1
M03/M04 S2000

EXPLANATION

O1000	While writing a program on fanuc controller first line has to be stared with letter ‘O’ followed by four digit number which specifies the program name.
[BILLET X20 Z60	This directive is used only for simulation purpose. It defines the workpiece as 60mm long and 20mm in diameter.
[TOOLDEF T1 D5 T2 D10	This directive sets the length and diameter of the tool for simulation
[EDGEMOVE	This directive sets up the required offset from the program zero position to lower left hand corner of the billet. This is used for simulation.
G21/G20 G40 G49 G80	G21-this code specifies that program is done in metric units G20-this code specifies that program is done in imperial units G40- compensation cancel G49- length compensation cancel G80- canned cycle
G50 S3500	Clamps the spindle speed at 3500 rpm
G94/G95 G91 G28 z0	G94- gives the unit of feed n mm/min G95-gives the unit of feed in mm/rev G28- go to home position along Z-axis
G28 X0 Y0	Go to home position along X and Y axes
M06 T01	Tool change to tool No.1
M03/M04 S1000	M03- makes the spindle rotate in clockwise direction M04- makes the spindle rotate in counter-clockwise direction S1000-setting the spindle speed at 1000rpm
G00 X0 Y0 Z5	G00- gives rapid position of the tool to appoint X0 Y0 Z5 which is just above the billet. This point is called tool entry point.

2. **BODY OF THE PROGRAM:** This is dealt operation wise in the succeeding pages.

3. **PROGRAM END**

G91 G28 X0 Y0 Z0
M05
M30

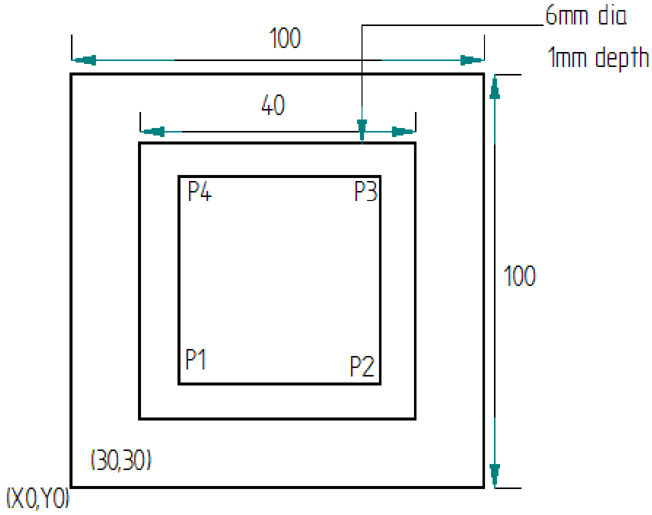
EXPLANATION

G91 G28 X0 Y0 Z0	Makes the tool to go to home position
M05	Stops the spindle rotation
M02/M30	M02 Optional stop M30 Program stop and rewind

EXERCISE – 17

LINEAR INTERPOLATION

Write a manual part program for Linear Interpolation for the component shown in figure below :



DWG. NO. 17

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 100X100X10				**MATERIAL: Aluminum**			
PROGRAM NO:1001				**DWGNO :17**			
SL.NO	**Operation**	**Tool type**	**Tool dia., mm**	**Tool station No.**	**Tool length offset No**	**Spindle speed rpm**	**Feed mm/min**
1	Linear	Slot cutter	6	1	1	2000	30-50

(Drawing No .17
O1001
(G01-Linear interpolation
[BILLET X100 Y100 Z10 ----------------- it defines the billet dimensions
[EDGEMOVE X0 Y0 ----------------This directive sets up the required offset from the program zero position to the middle of the billet.
[TOOLDEF T1 D6 -----------------------Defining tool

G21 G94	G21-this code specifies that program is done in metric units G94 gives the unit of feed in mm/min
G91 G28 Z0	G28 go to home position along Z-axis in incremental mode
G28 X0 Y0	Go to home position along X and Y axis
M06 T0101	Tool change to Tool No.1
M03 S1500	M03-makes the spindle rotate in clockwise direction
G90 G00 X30 Y30 Z5	G90-Absolute mode
G01 Z-1 F30	Giving the depth of cut along Z axis at a federate of 50mm/min
G01 X70 Y30 F60	
G01 X70 Y70	

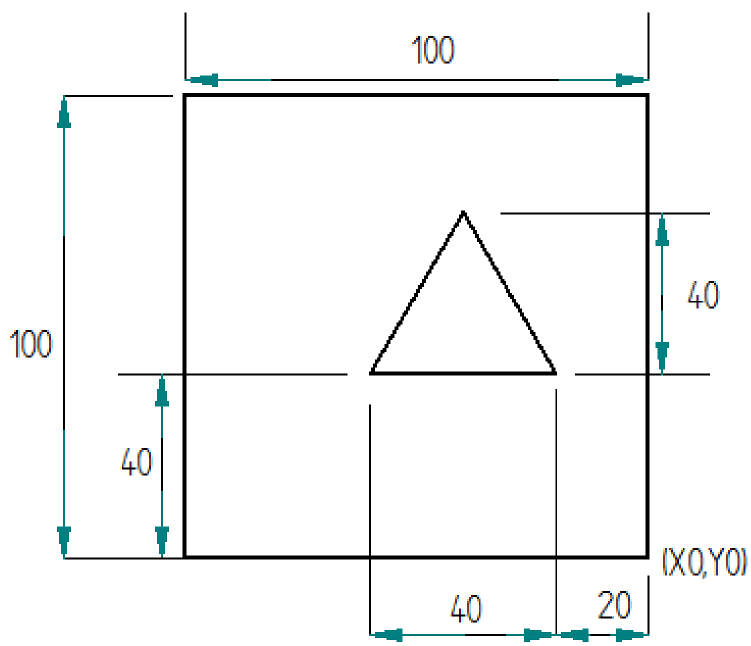
```
G01 X30 Y70
G01 X30 Y30
G00 Z5
G91 G28 Z0          ----- makes the toll to go to home position
G28 X0 Y0
M05                 ----- stops the spindle rotation
M30                 ----- Program stop and rewind
```

Date of Commencement	SIGNATURE OF STUDENT	**Marks / Remarks:**
Date of finish	SIGNATURE OF STAFF	

EXERCISE – 18

LINEAR INTERPOLATION

Write a manual part program for Linear Interpolation for the component shown in figure below :



DWG. NO. 18

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 100X100X10				**MATERIAL: Aluminum**			
PROGRAM NO:1002				**DWGNO :18**			
SL.NO	**Operation**	**Tool type**	**Tool dia., mm**	**Tool station No.**	**Tool length offset No**	**Spindle speed rpm**	**Feed mm/min**
1	Linear	Slot cutter	6	1	1	2000	30-50

(Drawing No .18
O1002
(G01-Linear interpolation
[BILLET X100 Y100 Z10 ----------------- it defines the billet dimensions
[EDGEMOVE X-100 Y0 --------------- This directive sets up the required offset from the program zero position to the middle of the billet.
[TOOLDEF T1 D6 ---------------------- Defining tool
G21 G94 G21-this code specifies that program is done in metric units
G94 gives the unit of feed in mm/min
G91 G28 Z0 G28 go to home position along Z-axis in incremental mode
G28 X0 Y0 Go to home position along X and Y axis
M06 T0101 Tool change to Tool No.1
M03 S1500 M03-makes the spindle rotate in clockwise direction
G90 G00 X-60 Y20 Z5 G90-Absolute mode
G01 Z-1 F30 Giving the depth of cut along Z axis at a federate of 50mm/min
G01 X-40 Y60 F60
G01 X-20 Y20
G01 X-60
G00 Z5
G91 G28 Z0 ----------------------- makes the toll to go to home position

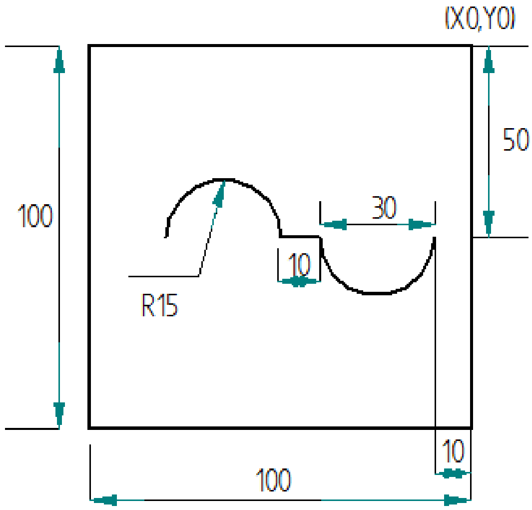
G28 X0 Y0
M05---------------------------------stops the spindle rotation
M30---------------------------------Program stop and rewind

Date of Commencement	SIGNATURE OF STUDENT	**Marks / Remarks:**
Date of finish	SIGNATURE OF STAFF	

EXERCISE – 19

CIRCULAR INTERPOLATION

Write a manual part program for Circular Interpolation for the component shown in figure below :



DWG. NO. 19

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 100X100X10				MATERIAL: Aluminum			
PROGRAM NO:1003				DWGNO :19			
SL.NO	**Operation**	**Tool type**	**Tool dia., mm**	**Tool station No.**	**Tool length offset No**	**Spindle speed rpm**	**Feed mm/min**
1	Linear	Slot cutter	6	1	1	2000	30-50

(Drawing No .19
O1003
(G01-Linear interpolation

[BILLET X100 Y100 Z10 ----------------- it defines the billet dimensions
[EDGEMOVE X-100 Y-100 --------------- This directive sets up the required offset from the program zero position to the middle of the billet.
[TOOLDEF T1 D6 -----------------------Defining tool

G21 G94	G21-this code specifies that program is done in metric units G94 gives the unit of feed in mm/min
G91 G28 Z0	G28 go to home position along Z-axis in incremental mode
G28 X0 Y0	Go to home position along X and Y axis
M06 T0101	Tool change to Tool No.1
M03 S1500	M03-makes the spindle rotate in clockwise direction

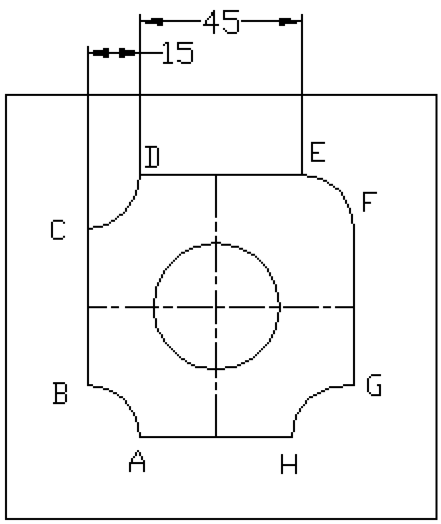
```
G90 G00 X-10 Y-50 Z5          G90-Absolute mode
G01 Z-1 F30                   Giving the depth of cut along Z axis at a federate of 50mm/min
G02 X-40 Y-50 R15
G01 X-40
G03 X-70 Y-50 R15
G00 Z5
G91 G28 Z0                    ----- makes the toll to go to home position


G28 X0 Y0
M05                           ----- stops the spindle rotation
M30                           ----- Program stop and rewind
```

Date of Commencement	SIGNATURE OF STUDENT	**Marks / Remarks:**
Date of finish	SIGNATURE OF STAFF	

EXERCISE – 20

LINEAR AND CIRCULAR INTERPOLATION

Write a manual part program for Contouring operation for the component shown in figure below :



A (X-22.5 Y-37.5)
B (X-37.5 Y-22.5)
C (X-37.5 Y22.5)
D (X-22.5 Y37.5)
E (X22.5 Y37.5)
F (X37.5 Y22.5)
G (X37.5 Y-22.5)
H (X22.5 Y-37.5)
I (X-15 Y0)
J (X15 Y0)

DWG. NO. 20

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 100X100X10				**MATERIAL: Aluminum**			
PROGRAM NO:1004				**DWGNO :20**			
SL.NO	**Operation**	**Tool type**	**Tool dia., mm**	**Tool station No.**	**Tool length offset No**	**Spindle speed rpm**	**Feed mm/min**
1	Contouring	Slot cutter	6	1	1	2000	30-50

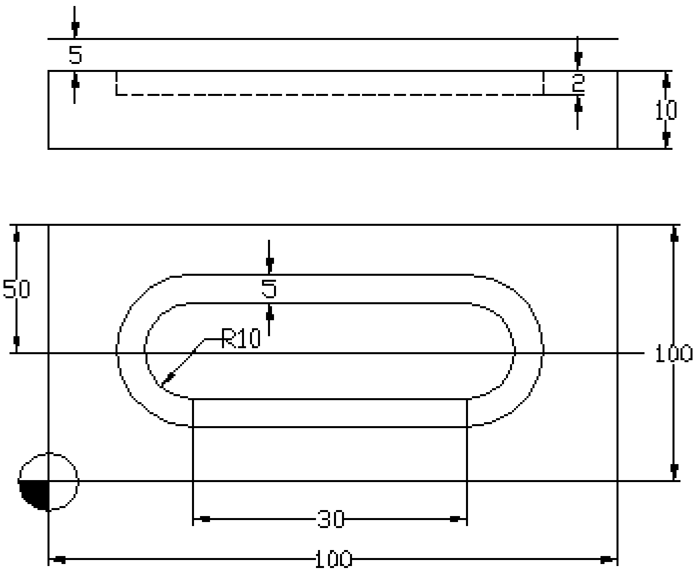
```
(Drawing No .20
O1004
[BILLET X100 Y100 Z10
[EDGEMOVE X-50 Y-50
[TOOLDEF T1 D6----------------------- Defining tool
G21 G94
G91 G28 Z0               G28 go to home position along Z-axis in incremental mode
G28 X0 Y0                Go to home position along X and Y axis
M06 T1                   Tool change to Tool No.1
M03 S2000                M03-makes the spindle rotate in clockwise direction
G90 G00 X0 Y-37.5 Z5     G90-Absolute mode
G01 Z-1 F50           Giving the depth of cut along Z axis at a federate of 50mm/min
G01 X-22.5 Y-37.5
G03 X37 Y-22.5 R15
G01 X-37.5
G01 X22.5 Y-37.5 R15
G03 X-22.5 Y37.5
G01 X22.5 Y37.5
G02 X37.5 Y22.5 R15
G01 X37.5 Y-22.5
G03 X22.5 Y-35.5 R15
G01 X0 Y-37.5
G00 Z5
G00 X-15 Y0
G01 Z-1 F50
G02 X15 Y0 R15
G02 X-15 Y0 R15      G00 Z5
G91 G28 Z0              ----- makes the toll to go to home position
G28 X0 Y0
M05    M30
```

Date of Commencement	SIGNATURE OF STUDENT	**Marks / Remarks:**
Date of finish	SIGNATURE OF STAFF	

EXERCISE – 21

CONTOURING WITH LEFT CUTTER DIAMETER COMPENSATION

Write a manual part program for contouring operation with left cutter diameter compensation for the component shown in fig



DWG. NO. 21

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 125X60X20				MATERIAL: Aluminum			
PROGRAM NO:1005				DWGNO :21			
SL.NO	Operation	Tool type	Tool dia., mm	Tool station No.	Tool length offset No	Spindle speed rpm	Feed mm/min
1	Contouring	Slot cutter	5	1	1	2000	35

(Drawing No .21
O1005
(G-40 cutter diameter compensation cancel
(G41-Cutter diameter compensation left
[BILLET X100 Y100 Z10 -----------------it defines the billet dimensions
[EDGEMOVE X0 Y0 --------------------- this directive sets up the required offset from the program zero position to the middle of the billet.
[TOOLDEF T1 D5----------------------- Defining tool

G21 G94	G21-this code specifies that program is done in metric units G94 gives the unit of feed in mm/min
G91 G28 Z0	G28 go to home position along Z-axis in incremental mode
G28 X0 Y0	Go to home position along X and Y axis
M06 T1	Tool change to Tool No.1
M03 S2000	M03-makes the spindle rotate in clockwise direction

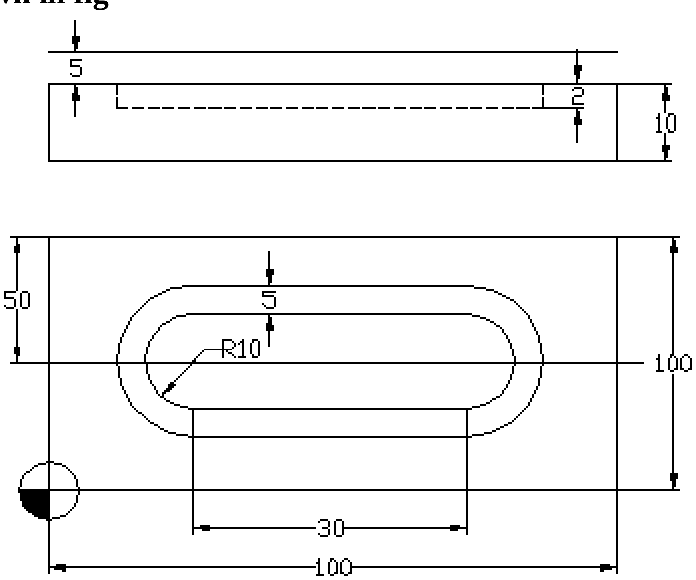
	S2000-setting the spindle speed at 2000 rpm
G90 G41 G00 X40 Y40 Z5	G90-Absolute mode G41 cutter diameter compensation left G00 gives rapid position of the tool to a point X40 Y40, Z5 which is just above the billet.
G01 Z-1 F50	Giving the depth of cut along Z axis at a federate of 50mm/min
G01 X70 Y40	
G03 X70 Y60 R10	
G01 X40 Y60	
G03 X40 Y40 R10	
G00 Z5	
G40	----- Cutter compensation cancel
G91 G28 Z0	----- makes the toll to go to home position
G28 X0 Y0	
M05	----- stops the spindle rotation
M30	----- Program stop and rewind

Date of Commencement	SIGNATURE OF STUDENT	**Marks / Remarks:**
Date of finish	SIGNATURE OF STAFF	

EXERCISE – 22

CONTOURING WITH RIGHT CUTTER DIAMETER COMPENSATION

Write a manual part program for contouring operation with left cutter diameter compensation for the component shown in fig



DWG. NO. 22

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 100X100X10				**MATERIAL: Aluminum**			
PROGRAM NO:1006				**DWGNO :22**			
SL.NO	**Operation**	**Tool type**	**Tool dia., mm**	**Tool station No.**	**Tool length offset No**	**Spindle speed rpm**	**Feed mm/min**
1	Contouring	Slot cutter	5	1	1	2000	35-50

(Drawing No .22

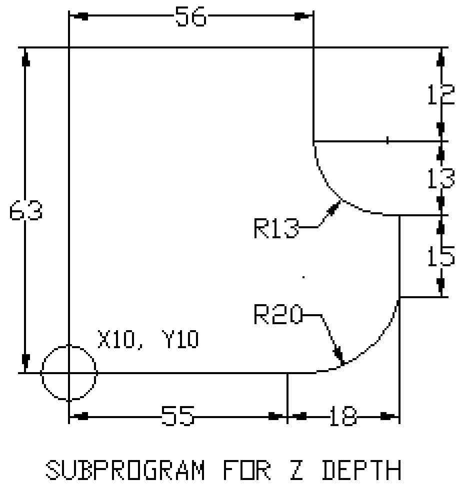
```
O1006
(G-40 cutter diameter compensation cancel
(G42-Cutter diameter compensation right
[BILLET X100 Y100 Z10 -----------------it defines the billet dimensions
[EDGEMOVE X0 Y0 --------------------- this directive sets up the required offset from the
                                      program zero position to the middle of the billet.
[TOOLDEF T1 D5----------------------- Defining tool
G21 G94                   G21-this code specifies that program is done in metric units
                          G94 gives the unit of feed in mm/min
G91 G28 Z0                G28 go to home position along Z-axis in incremental mode
G28 X0 Y0                 Go to home position along X and Y axis
M06 T1                    Tool change to Tool No.1
M03 S2000                 M03-makes the spindle rotate in clockwise direction
G90 G42 G00 X40 Y40 Z5    G90-Absolute mode
                          G42 cutter diameter compensation right
                          G00 gives rapid position of the tool to a point X40 Y40,
                          Z5 which is just above the billet.
G01 Z-1 F50           Giving the depth of cut along Z axis at a federate of 50mm/min
G01 X70 Y40
G03 X70 Y60 R10
G01 X40 Y60
G03 X40 Y40 R10
G00 Z5
G40 --------------------------------- Cutter compensation cancel
G91 G28 Z0----------------------- makes the toll to go to home position
G28 X0 Y0
M05 -------------------------------- stops the spindle rotation
M30 -------------------------------- Program stop and rewind
```

Date of Commencement	SIGNATURE OF STUDENT	**Marks / Remarks:**
Date of finish	SIGNATURE OF STAFF	

EXERCISE – 23

CONTOURING THROUGH SUBPROGRAM

Write a manual part program for contouring operation through subprogram for the component shown in fig.



DWG. NO. 23

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 100X100X10				**MATERIAL: Aluminum**			
PROGRAM NO:1007				**DWGNO :23**			
SL.NO	**Operation**	**Tool type**	**Tool dia., mm**	**Tool station No.**	**Tool length offset No**	**Spindle speed rpm**	**Feed mm/min**
1	Contouring	Slot cutter	5	1	1	2000	35-50

(Drawing No . 23
O1007
(PROGRAM FOR SUBPROGRAM
(MCODES USED M98, M99 (M98 SUB PROGRAM CALL
(M99 SUBPROGRAM EXIT
(TOTAL DEPTH OF CUT 5MM (DEPTH OF CUT FOR EACH PASS:0.5mm
[BILLET X100 Y100 Z10 ---- it defines the billet dimensions
[EDGEMOVE X0 Y0 ---- this directive sets up the required offset from the program zero position to the middle of the billet.
[TOOLDEF T1 D5---------------------- Defining tool
G21 G94 G21-this code specifies that program is done in metric units
G94 gives the unit of feed in mm/min
G91 G28 Z0 G28 go to home position along Z-axis in incremental mode
G28 X0 Y0 Go to home position along X and Y axis
M06 T1 Tool change to Tool No.1
M03 S2000 M03-makes the spindle rotate in clockwise direction
S2000-setting the spindle speed at 2000 rpm

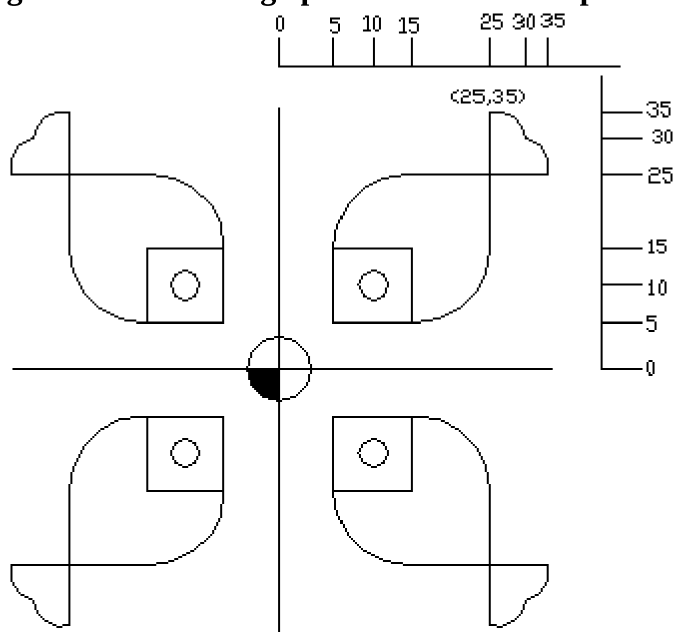
```
G90 G00 X10 Y10 Z5        G90-Absolute mode
                          G00 gives rapid position of the tool to a point X10 Y10,
                          Z5 which is just above the billet.
G00 Z0
M98 P0014000           ----- Calling the subprogram 4000 five times
M98 P0014000
M98 P0014000
M98 P0014000
M98 P0014000
G00 Z5
G91 G28 Z0              ----- makes the toll to go to home position
G28 X0 Y0
M05                     ----- stops the spindle rotation
M30                     ----- Program stop and rewind

O4000                   ----- Subprogram for increasing depth
M98 P0014001
G01 X10 Y73 F50
X66
Y61
G03 X83 Y44 R13
G01 X83 Y29
G02 X65 Y10 R20
G01 X10 Y10
M99                    ------ Exiting the subprogram O4000

O4001                  ------ Subprogram
G91 G01 Z-0.5 F50
G90
M99                    ------ Exiting the subprogram O4001
```

Date of Commencement	SIGNATURE OF STUDENT	**Marks / Remarks:**
Date of finish	SIGNATURE OF STAFF	

EXERCISE – 24

MIRRORING

Write a manual part program for Mirroring operation for the component shown in fig.



DWG. NO. 24

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 100X100X10				**MATERIAL: Aluminum**			
PROGRAM NO:1008				**DWGNO :24**			
SL.NO	**Operation**	**Tool type**	**Tool dia., mm**	**Tool station No.**	**Tool length offset No**	**Spindle speed rpm**	**Feed mm/min**
1	Contouring	Slot cutter	5	1	1	2000	35-50

```
(Drawing No . 24
O1008
(PROGRAM FOR MIRRORING
(MCODES USED M70,M71,M80,M81
(M70 X-AXIS MIRROR ON         (M71 Y- AXIS MIRROR ON
(M80 X-AXIS MIRROR OFF        (M81 Y AXIS MIRROR OFF
[BILLET X100 Y100 Z10           ---- it defines the billet dimensions
[EDGEMOVE X-50 Y-50             ---- this directive sets up the required offset from the
[TOOLDEF T1 D5                  ----Defining tool
G21 G94
G91 G28 Z0             G28 go to home position along Z-axis in incremental mode
G28 X0 Y0              Go to home position along X and Y axis
M06 T1                 Tool change to Tool No.1
M03 S2000              M03-makes the spindle rotate in clockwise direction
G90 G00 X0 Y0 Z5       G90-Absolute mode
G00 Z0
```

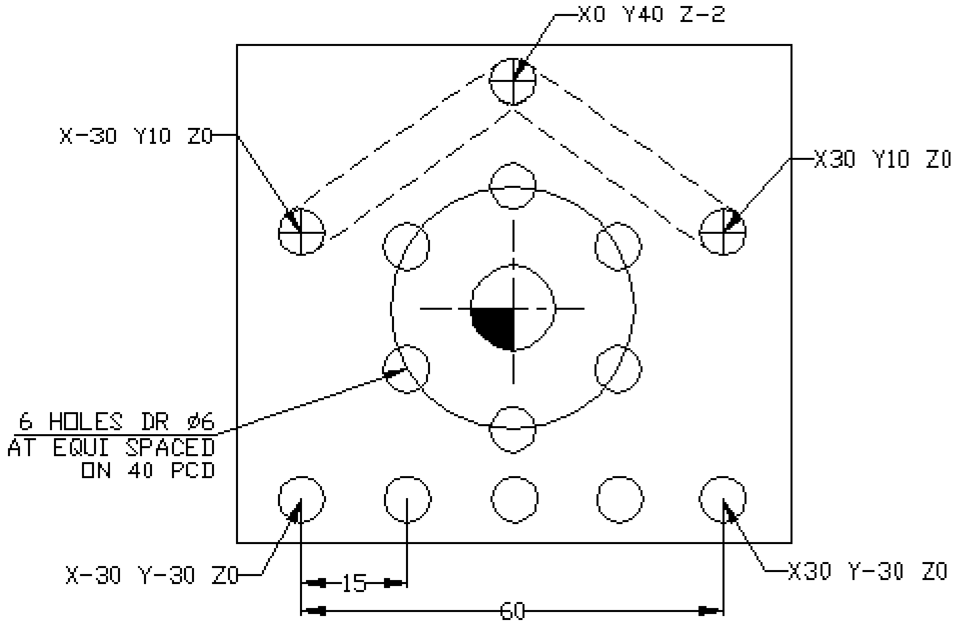
```
M98 P0015000            ----- Calling the subprogram 5000 one time
M70                     ----- Mirroring about X axis
M98 P0015000            ----- Calling the subprogram 5000 one time
M80                     ----- Canceling of mirroring about X axis
M70                     ----- Mirroring about X axis
M71                     ----- Mirroring about Y axis
M98 P0015000            ----- Calling the subprogram 5000 one time
M80                     ----- Canceling of mirroring about X axis
M81                     ----- Canceling of mirroring about Y axis
M71                     ----- Mirroring about Y axis
M98 P0015000            ----- Calling the subprogram 5000 one time
G00 Z5
G91 G28 Z0              ----- makes the toll to go to home position
G28 X0 Y0
M05                     ----- stops the spindle rotation
M30                     ----- Program stop and rewind

O5000                   ----- Subprogram
G00 X5 Y5 Z5            ----- Rapid traverse to 5,5,5
G01 Z-1 F50             ----- Feed rate at 35 mm/min.
X15 Y5
X15 Y15
X5 Y15
X5 Y5
G00 Z5
X10 Y10
G01 Z-1 F50
G00 Z5
X15 Y5
G01 Z-1 F50
G03 X25 Y15 R15         ----- CCW interpolation.
G01 X25 Y35
G02 X30 Y30 R5
G02 X35 Y25 R5
G01 X15 Y25
G03 X5 Y15 R15
G00 Z5
G00 X0 Y0
M99                     ------ Exiting the subprogram O5000
```

Date of Commencement	SIGNATURE OF STUDENT	**Marks / Remarks:**
Date of finish	SIGNATURE OF STAFF	

EXERCISE – 25

DRILLING

Write a manual part program for Drilling operation for the component shown in fig.



DWG. NO. 25

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 100X100X10				**MATERIAL: Aluminum**			
PROGRAM NO:1009				**DWGNO :25**			
SL.NO	**Operation**	**Tool type**	**Tool dia., mm**	**Tool station No.**	**Tool length offset No**	**Spindle speed rpm**	**Feed mm/min**
1	Slotting	Slot cutter	8	1	1	2000	35-50
2	Drilling	Slot drill	5	2	2	1500	35

(Drawing No . 25
O1009
(PROGRAM FOR DRILLING
(GCODES USED G73,G83,G98,G99
(G73 – FAST PECK DRILLING CYCLE (G83 – PECK DRILLING CYCLE
[BILLET X100 Y100 Z10----------------- it defines the billet dimensions
[EDGEMOVE X-50 Y-50---------------- this directive sets up the required offset from the program zero position to the middle of the billet.
[TOOLDEF T1 D8 T2 D5------------- Defining tool

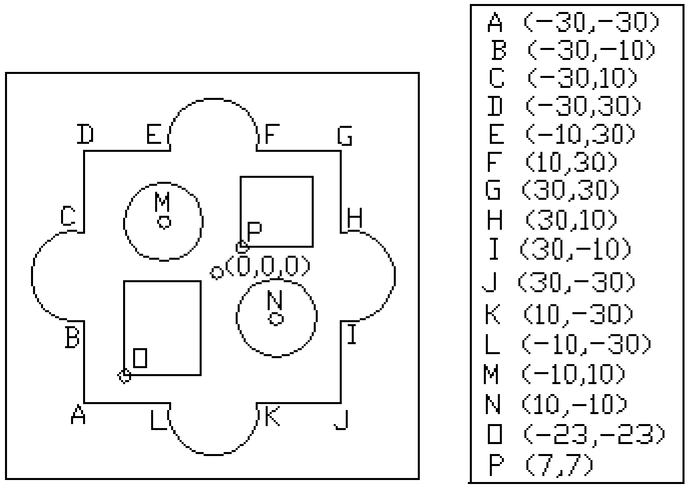
```
G21 G94                      G21-this code specifies that program is done in metric units
                             G94 gives the unit of feed in mm/min
G91 G28 Z0                   G28 go to home position along Z-axis in incremental mode
G28 X0 Y0                    Go to home position along X and Y axis
M06 T1                       Tool change to Tool No.1
M03 S2000                    M03-makes the spindle rotate in clockwise direction
G90 G00 X0 Y0 Z5             G90-Absolute mode
                             G00 gives rapid position of the tool to a point X0 Y0,
                             Z5 which is just above the billet.
G83 G99 X0 Y20 Z-5 Q0.5 R0.5 F50------------- Drilling cycle
X17.32 Y10
Y-10
X0 Y-20
X-17.32 Y-10
Y10
G00 X-30 Y-30
G91 G99 G73 X10 Y0 Z-5 P100 Q0.5 R0.5 K5 F50-----------Peck drilling cycle
G80
G91 G28 Z0                     ------ Going to home position
G28 X0 Y0
M06 T2
M03 S1500
G90 G00 X-30 Y10 Z0
G01 Z-1
G01 X0 Y40
G01 X30 Y10
G00 Z5
G91 G28 Z0
G28 X0 Y0
M05                          ----- stops the spindle rotation
M30                          ----- Program stop and rewind
```

Date of Commencement	SIGNATURE OF STUDENT	**Marks / Remarks:**
Date of finish	SIGNATURE OF STAFF	

EXERCISE – 26

POCKETING

Write a manual part program for Pocketing operation for the component shown in fig.



CIRCULAR AND RECTANGULAR POCKETING

DWG. NO. 26

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 100X100X10				**MATERIAL: Aluminum**			
PROGRAM NO:1010				**DWGNO :26**			
SL.NO	**Operation**	**Tool type**	**Tool dia., mm**	**Tool station No.**	**Tool length offset No**	**Spindle speed rpm**	**Feed mm/min**
1	Slotting & Pocketing	Slot cutter	5	1	1	2000	35-45

```
(Drawing No . 26
O1010
(PROGRAM FOR POCKETING
(GCODES USED G170 & G171 – CIRCULAR POCKETING
(G172 & G173 – RECTANGULAR POCKETING
[BILLET X100 Y100 Z10 -----------------it defines the billet dimensions
[EDGEMOVE X-50 Y-50 ----------------this directive sets up the required offset from the
                                     Program zero position to the middle of the billet.
[TOOLDEF T1 D5----------------------- Defining tool
G21 G94                  G21-this code specifies that program is done in metric units
G91 G28 Z0               G28 go to home position along Z-axis in incremental mode
G28 X0 Y0                Go to home position along X and Y axis
M06 T1                   Tool change to Tool No.1
M03 S2000                M03-makes the spindle rotate in clockwise direction
G90 G00 X-30 -30 Z5      G90-Absolute mode
G01 Z-1 F50
G01 X-30 Y-10
```

```
G02 X-30 Y10 R10
G01 X-30 Y30
G01 X-10 Y30
G02 X10 Y30 R10
G01 X30 Y30
G01 X30 Y10
G02 X30 Y-10 R10
G01 X30 Y-30
G01 X10 Y-30
G02 X-10 Y-30 R10
G01 X-30 Y-30
G00 Z5
G00 X-10 Y10
G170 R0 P1 Q0.5 X-10 Y10 Z-5 I0.2 J0.2 K9---------- Circular pocketing
G171 P50 S2000 R35 F45 B2500 J50
G170 R0 P1 Q0.5 X10 Y-10 Z-5 I0.2 J0.2 K9---------- Circular pocketing
G171 P50 S2000 R35 F45 B2500 J50
G172 I15 J15 K0 P1 Q0.5 R0 X7 Y7 Z-5-----------------Rectangular pocketing
G173 I0.2 K0.2 P50 T1 S2000 R35 F45 B2500 J50 Z5
G172 I15 J15 K0 P1 Q0.5 R0 X-23 Y-23 Z-5------------ Finishing pass
G173 I0.2 K0.2 P75 T1 S2000 R40 F60 B2500 J50 Z5
G00 Z0
G91 G28 Z0---------------------- makes the toll to go to home position
G28 X0 Y0
M05-------------------------------- stops the spindle rotation
M30-------------------------------- Program stop and rewind
G170 R0 P1 Q0.5 X10 Y-10 Z-5 I0.1 J0.1 K9----------Finishing pass Circular pocketing
G171 P80 S2000 R35 F45 B3000 J25
G172 I15 J15 K0 P0 Q0.5 R0 X7 Y7 Z-5-----------------Rectangular pocketing
G173 I0.1 K0.1 P85 T1 S2000 R35 F45 B3000 J25 Z5
G172 I15 J15 K0 P1 Q0.5 R0 X7 Y7 Z-5
G173 I0.1 K0.1 P85 T1 S2000 R35 F45 B3000 J25 Z5
G172 I15 J15 K0 P0 Q0.5 R0 X-23 Y-23 Z-5
G173 I0.1 K0.1 P85 T1 S2000 R35 F45 B3000 J25 Z5
G172 I15 J15 K0 P1 Q0.5 R0 X-23 Y-23 Z-5------------- Finishing pass
G173 I0.1 K0.1 P85 T1 S2000 R35 F45 B3000 J25 Z5
G91 G28 Z0---------------------- makes the toll to go to home position
G28 X0 Y0
M05    M30
```

Date of Commencement	SIGNATURE OF STUDENT	**Marks / Remarks:**
Date of finish	SIGNATURE OF STAFF	

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

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

CAD - CAM

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 are different product activities?
5. What is a product cycle?
6. **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.
7. **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
8. **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
9. Block diagram for general design process?
10. Block diagram of a design process with CAD?
11. **What is geometric modeling?**
    Geometric modeling is a branch of applied mathematics and geometry that studies methods and algorithms for the mathematical description of shapes.
12. What is the basic classification of modeling?
13. What is engineering analysis?
14. What is design review and its evaluation?
15. What is automated drafting?
16. What is the use of database?
17. Block diagram of a database with CAD/CAM?
18. **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.

19. What are the basic computer hardware units?

20. What is the hardware configuration for a CAD system?

21. Block diagram of main frame based CAD hardware?

22. Block diagram of HOST satellite CAD system?

23. Define NC?

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

24. 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

25. What is NC procedure?

- Process planning.
- Part programming

Manual part programming

Computer-assisted part programming

- Tape preparation.
- Tape verification.
- Production.

26. Discuss NC coordinate system?

27. 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.

28. 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.

29. 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.

30. What is absolute positioning and incremental positioning/

31. Discuss NC motion control systems?

32. Applications of NC systems?

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

33. 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

34. What is NC part programming?
35. What is manual part programming?
36. What is computer assisted part programming?
37. What are the various input mediums of an NC system?
38. What does N Word stands for?
N - Sequence number (Used for line identification)
39. What does G word stands for?
G - Preparatory function
40. What does M Word stands for?
M - Miscellaneous function
41. What does T word stands for?
T - Tool Designation
42. What is fixed sequential format?
43. What is tab sequential format?
44. What is word address format?
45. What are part programmers job?
46. 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

47. What is cutter offset compensations?
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.
48. What is a Robot?
A robot is a mechanical or virtual agent, usually an electro-mechanical machine that is guided by a computer program or electronic circuitry.
49. Physical configurations of robot.

- Cartesian configuration
- Cylindrical configuration
- Polar configuration
- Jointed-arm configuration

50. Basic robot motions.
1. Arm and body motions
   - Vertical traverse
   - Radial traverse
   - Rotational traverse

2. Wrist Motion
   - Wrist swivel
   - Wrist bend
   - Wrist yaw

51. Robot programming language.
   - The VALTM Language
   - The MCL Language

52. Basic commands for robot

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

53. 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

54. 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

55. 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.

56. 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.

57. 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)

****