SRI SHANMUGHA COLLEGE OF ENGINEERING AND TECHNOLOGY

(Approved by AICTE, Affiliated to Anna University and Accredited by NAAC & NBA (ECE)

Pullipalayam, Morur (P.O), Sankari (T.k), Salem (D.T) – 637 304

DEPARTMENT OF MECHANICAL ENGINEERING

ME8681 - CAD / CAM Laboratory

Vision and Mission of the institute

VISION

To be an Institute of repute in the field of Engineering and Technology by implementing the best educational practices akin to global standards for fostering domain knowledge and developing research attitude among students to make them globally competent

MISSION

M1: Achieving excellence in Teaching Learning process using state of the art resources.

M2: Extending opportunity to upgrade faculty knowledge and skills.

M3: Implementing best student training practices for requirements of Industrial scenario of the State.

M4: Motivating faculty and students in research activity for real-time application.

Vision and Mission of the Department

VISION

To prepare competent mechanical engineers capable of working in an interdisciplinary environment contributing to society through innovation, leadership and entrepreneurship

MISSION

M1: To offer quality education which enables them in professional practice and career

M2: To provide learning opportunities in the state-of-the-art research facilities to create, interpret, apply and disseminate knowledge in their profession

M3: To prepare the students as professional engineers in the society with an awareness of environmental and ethical values

PROGRAMOUTCOMES (POs):

PO1 Engineering knowledge: Apply the knowledge of mathematics, science, engineering fundamentals and an engineering specialization to the solution of complex engineering problems.

PO2 Problem analysis: Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences and engineering sciences.

PO3 Design/development of solutions: Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health, safety, cultural, societal and environmental considerations.

PO4 Conduct investigations of complex problems: Use research-based knowledge and research methods including design of experiments, analysis, and interpretation of data and synthesis of the information to provide valid conclusions.

PO5 Modern tool usage: Create, select, apply appropriate techniques, resources, modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.

PO6 The engineer and society: Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal, cultural issues and the consequent responsibilities relevant to the professional engineering practice.

PO7 Environment and sustainability: Understand the impact of the professional engineering solutions in societal, environmental contexts, demonstrate the knowledge and need for sustainable development.

PO8 Ethics: Apply ethical principles, commit to professional ethics, responsibilities and norms of the engineering practice.

PO9 Individual and team work: Function effectively as an individual, as a member or leader in diverse teams and in multidisciplinary settings.

PO10 Communication: Communicate effectively on complex engineering activities with the engineering community with society at large being able to comprehend, write effective reports, design documentation, make effective presentations and receive clear instructions.

PO11 Project management and finance: Demonstrate knowledge, understanding of the engineering and management principles and apply these to one’s own work, as a member and leader in a team, to manage projects and in multidisciplinary environments.

PO12 Life-long learning: Recognize the need, ability to engage in independent and life-long learning in the broadest context of technological change.

PROGRAM SPECIFIC OUTCOMES (PSOs)

PSO1 Manufacturing: Modelling, Simulation and Analysis in the field of Manufacturing.

PSO2 Design: Develop and implement new ideas on product design with help of modern CAD tools.

PROGRAM EDUCATIONAL OBJECTIVES (PEOs)

PEO1: To prepare students to take up career in Industry, Academia as well as in Public services.

PEO2: To provide core domain and interpersonal skills to design & develop mechanical systems for Interdisciplinary applications following ethical code.

PEO3: To develop qualities to progress in entrepreneurship and research activities.

COURSE OUTCOMES:

Upon the completion of this course the students will be able to

C320.1	Understand and interpret machine manufacturing drawings
C320.2	Develop 2D and 3D models using high end modeling software's
C320.3	Apply engineering drawing standards as per BIS conventions
C320.4	Understand the CNC control in modern manufacturing system
C320.5	Prepare CNC part programming and perform manufacturing

CO-PO MAPPING MATRIX:

Course Outcomes	Program Outcomes													
	PO 1	PO 2	PO 3	PO 4	PO 5	PO 6	PO 7	PO 8	PO 9	PO 10	PO 11	PO 12	PSO 1	PSO 2
C320.1	3	2	3		1	1		1	1	1		3	2	3
C320.2	3	3	3		3	1			2	1		3	2	3
C320.3	3	2	3		2	1		2	2	3		3	2	3
C320.4	3	2	3	3	2	1		2	2			3	2	3
C320.5	3	2	3	3	2	1		2	2			3	2	3
C320	**3.0**	**2.2**	**3.0**	**3.0**	**2.0**	**1.0**		**1.8**	**1.8**	**1.7**		**3.0**	**2.0**	**3.0**

LIST OF EXPERIMENTS

Sl. No	K Level	Name of the Experiment	Relevance to COs	Page No
1.	K2	Introduction- Role of CAD in product design process- GD&T,Limits,Fits- Basics	CO1	
2.	K4	Detailing and assembly of flange coupling	CO1,CO2,CO3	
3.	K4	Detailing and assembly of universal coupling	CO1,CO2,CO3	
4.	K4	Detailing and assembly of screw jack	CO1,CO2,CO3	
5.	K4	Detailing and assembly of stuffing box	CO1,CO2,CO3	
6.	K4	Detailing and assembly of Plummer block	CO1,CO2,CO3	
7.	K2	Introduction-CAM-Manual part programming-Computer aided part programming basics	CO4	
8.	K4	Manual part programming for step turning operation in CNC turning center	CO4,CO5	
9.	K4	Manual part programming for taper turning operation in CNC turning center	CO4,CO5	
10.	K4	NC code generation for step turning and facing operation using cadem software	CO4,CO5	
11.	K4	NC code generation for grooving and thread cutting operation using cadem software	CO4,CO5	
12.	K4	Manual part programming for drilling operation	CO4,CO5	
13.	K4	NC code generation for drilling operation using cadem software	CO4,CO5	
14.	K4	NC code generation for side milling operation using cadem software	CO4,CO5	
15.	K4	NC code generation for pocket milling, drilling and tapping operation using cadem software	CO4,CO5	
16.	K3	NC code generation for mirroring and pocket milling operation using cadem software	CO4,CO5	
Content Beyond the Syllabus				
1.	K4	NC code generation for industry components	CO4,CO5	

COMPUTER AIDED DESIGN (CAD)

EX NO: 1 **INTRODUCTION-DESIGN PROCESS AND ROLE OF CAD**

According to Shingly, the design process is an iterative procedure involving the following six phases:

1. Recognition of need
2. Definition of problem
3. Synthesis
4. Analysis and optimization
5. Evaluation
6. Presentation

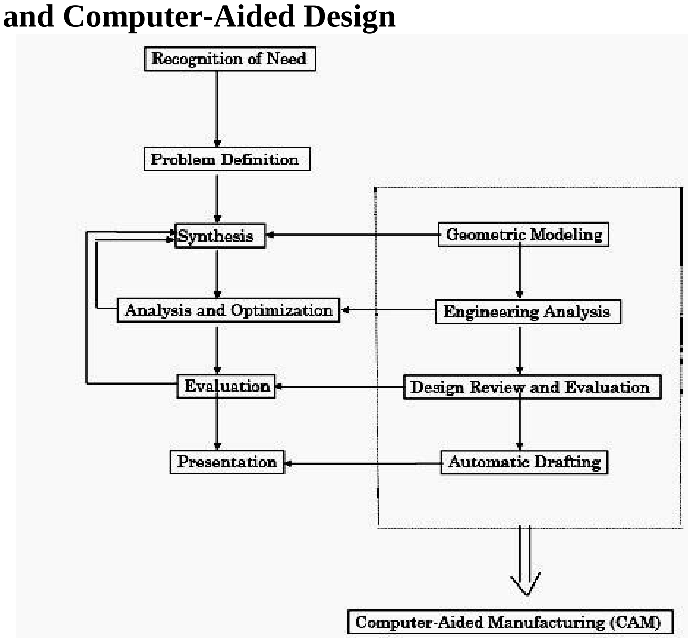
Phase 3 (synthesis) includes defining the design problem, design conceptualization, searching for design information, modeling and simulation.

Phase 4 (analysis and optimization) may include parameter study, finite element analysis, etc. Although computers are being utilized more and more in the design process, their use is still limited to the last four steps in the design and they are mainly used as a tool that helps the designer, rather than as a replacement for the designer.

BENEFITS OF USING CAD:

(1) Increasing productivity
(2) Improving quality of design
(3) Improving communications
(4) Creating data-base for manufacturing

The Design Process and Computer-Aided Design



GEOMETRIC MODELING

The term geometric modeling (or representation) means a method of describing commonly used curves and surfaces in terms of values of a few parameters.

THREE TYPES OF GEOMETRIC MODELS

Wireframe Model: connect 3D vertex points, sometimes ambiguous.
Surface Model : define surface to form an object.
Solid Model: various representation schemes are used to describe a solid object

SOLID MODELING

A solid modeling system is usually an interactive computer graphics system that is intended to create true three-dimensional components and assemblies. Recent advances in CAD software, computers, and graphical displays have made it possible to use solid representations of components being considered in the design process. These solid models can be employed innumerous ways.

ADVANTAGES OF SOLID MODELING

A realistic visual display: By producing a shaded visible surface image of the solid, solid modeling allows a designer to see exactly what has been created. Easy to deal with different views: Once a part has been created, we have the ability to rotate, shade, section, or produce almost any view required by a designer. Single associated model database: The solid modeler provides the only database suitable for all CAD operations. Almost all information needed for part generation is contained in the solid model. The algorithm should be able to ensure that it represents physically possible shape that is complete and unambiguous Applications. e.g., automatic generation of a mesh for a finite element analysis.

REQUIREMENTS FOR MODELING ASSEMBLING

1.Part modeling and analysis

The part analysis includes the material type, mass and inertial properties, functional properties of the faces, etc.

2.Hierarchical relationships

An assemble tree and assemble sequence must be given.

3.Mating conditions.

There are two methods for specifying mating conditions: Specify the location and orientation of each part in the assembly, together with the representation of the part itself, by providing a 4 x 4 homogeneous transformation matrix. (i.e., transformation from MCS to WCS)Specify the spatial relationships between its individual parts as mating conditions. For example, a mating condition can consist of planar faces butting up against one another or requiring centerlines of individual parts to be collinear (—fits|| condition).

CAD/CAE/CAM Data Exchange

Computer databases are now replacing paper blueprints in defining product geometry and non-geometry for all phases of product design, analysis, and manufacturing. It becomes increasingly important to find effective procedures for transferring data among CAD/CAE/CAM systems. The need to exchange modeling data is directly motivated by the need to integrate and automate the design and manufacturing process to obtain the maximum benefits from CAD/CAE/CAM systems.

FOUR TYPES OF MODELING DATA TO BE TRANSFERRED

(1) Shape
(2) Non shape
(3) Design
(4) Manufacturing

(1) Shape data consists of both geometrical and topological information as well as part features. Entity attributes such as font, color, and layers as well as annotation are considered part of the entity geometrical information. Topological information applies only to products described via solid modeling. Features allow high-level concept communication about parts. Examples are hole, flange, web, pocket, chamfer, etc.

(2) Non shape data includes graphics data such as shaded images, and model global data as measuring units of the database and the resolution of storing the database numerical values.

(3) Design data has to do with the information that designers generate from geometric models for analysis purposes. e.g., mass property and finite element mesh data.

(4) Manufacturing data consists of information such as tooling, NC tool paths, tolerance, process planning, tool design, and bill of materials.

Commonly Used CAD Data Exchange Format IGES (Initial Graphics Exchange Specification) PDES (Product Data Exchange Using STEP) IGES is focused on CAD-to-CAD exchange where primarily shape and non-shape data were to be transferred from one system to another. PDES is previous called Product Data Exchange Standard. It is for the exchange of complete product descriptions which covers the four types of modeling data (i.e., shape, non-shape, design and manufacturing).Other data exchange interfaces include: STL, Neutral, SET, ECAD, VDA, STEP, PDGS, CATIA, Render, CGM, VRML, PATRAN, TIFF, etc.

LIMITS, TOLERANCES AND FITS

The manufacture of interchangeable parts requires precision. Precision is the degree of accuracy to ensure the functioning of a part as intended. However, experience shows that it is impossible to make parts economically to the exact dimensions. This may be due to, (i) inaccuracies of machines and tools, (ii) inaccuracies in setting the work to the tool, and (iii) error in measurement, etc. The workman, therefore, has to be given some allowable margin so that he can produce a part, the dimensions of which will lie between two acceptable limits, a maximum and a minimum. The system in which a variation is accepted is called the limit system and the allowable deviations are called tolerances. The relationships between the mating parts are called fits.

Tolerance -The permissible variation of a size is called tolerance. It is the difference between the maximum and minimum permissible limits of the given size. If the variation is provided on one side of the basic size, it is termed as unilateral tolerance. Similarly, if the variation is provided on both sides of the basic size, it is known as bilateral tolerance.

Limits-The two extreme permissible sizes between which the actual size is contained are called limits. The maximum size is called the upper limit and the minimum size is called the lower limit.

Deviation-It is the algebraic difference between a size (actual, maximum, etc.) and the corresponding basic size.

Actual deviation-It is the algebraic difference between the actual size and the corresponding basic size.

Upper deviation-It is the algebraic difference between the maximum limit of the size and the corresponding basic size.

Lower deviation-It is the algebraic difference between the minimum limit of the size and the corresponding basic size.

Allowance -It is the dimensional difference between the maximum material limits of the mating parts, intentionally provided to obtain the desired class of fit. If the allowance is positive, it will result in minimum clearance between the mating parts and if the allowance is negative, it will result in maximum interference.

Basic size -It is determined solely from design calculations. If the strength and stiffness requirements need a 50mm diameter shaft, then 50mm is the basic shaft size. If it has to fit into a hole, then 50 mm is the basic size of the hole. Figure illustrates the basic size, deviations and tolerances.

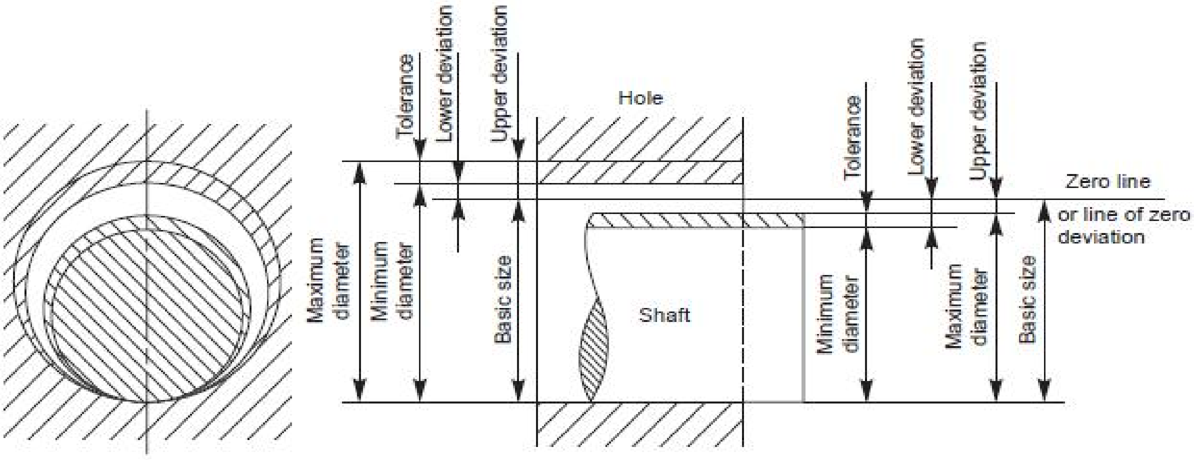


Diagram illustrating basic size deviations and tolerances

<u>Fits</u>

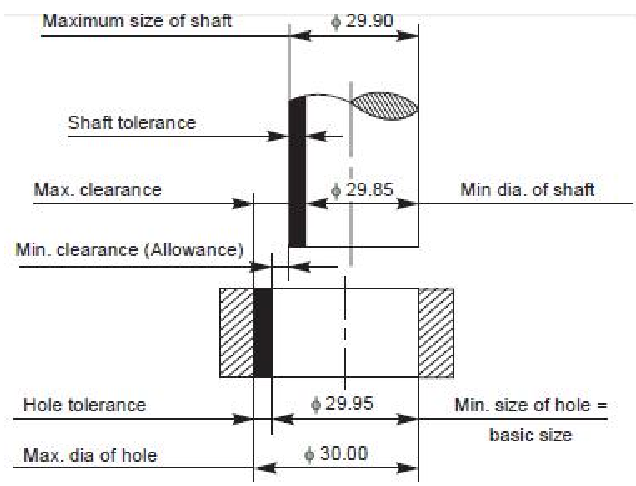
The relation between two mating parts is known as a fit. Depending upon the actual limits of the hole or shaft sizes, fits may be classified as clearance fit, transition fit and interference fit.

Clearance fit-It is a fit that gives a clearance between the two mating parts.

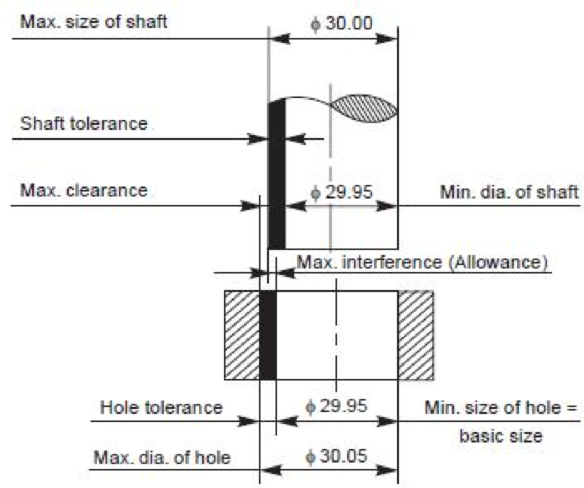
Minimum clearance-It is the difference between the minimum size of the hole and the maximum size of the shaft ina clearance fit.

Maximum clearance-It is the difference between the maximum size of the hole and the minimum size of the shaft in a clearance or transition fit.

Transition fit-This fit may result in either interference or a clearance, depending upon the actual values of the tolerance of individual parts. The shaft in Fig. may be either smaller or larger than the hole and still be within the prescribed tolerances. It results in a clearance fit, when shaft diameter is 29.95 and hole diameter is 30.05 (+ 0.10 mm) and interference fit, when shaft diameter is 30.00 and hole diameter 29.95 (– 0.05 mm).

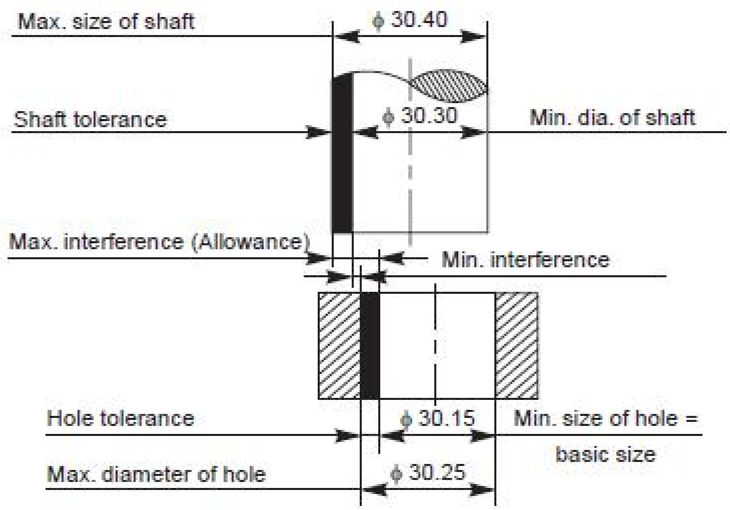


Clearance fit



Transition fit

Interference fit-If the difference between the hole and shaft sizes is negative before assembly; an interferencefit is obtained.

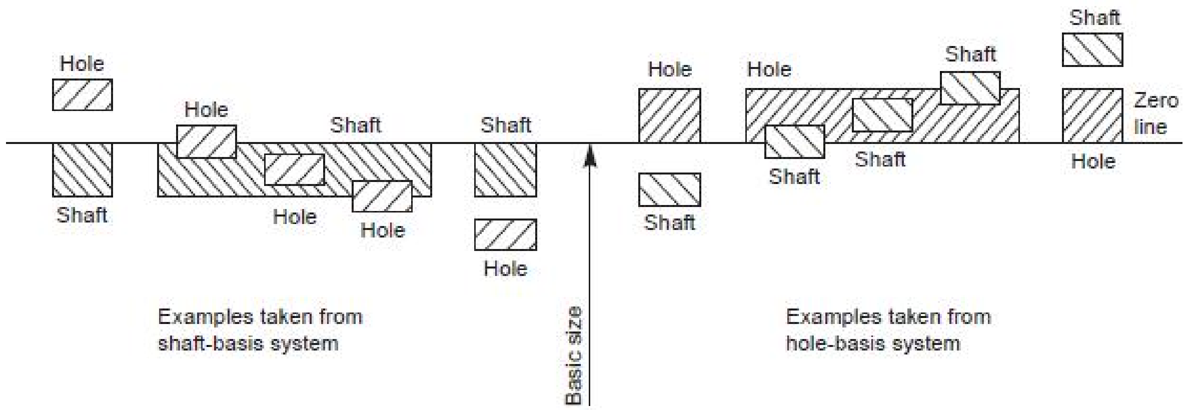


Interference fit

Hole and shaft basis system

Hole basis system-In this system, the size of the shaft is obtained by subtracting the allowance from the basic size of the hole. This gives the design size of the shaft. Tolerances are then applied to each part separately. In this system, the lower deviation of the hole is zero. The letter symbol for this situation is ‘H’. The hole basis system is preferred in most cases, since standard tools like drills, reamers, broaches, etc., are used for making a hole.

Shaft basis system-In this system, the size of the hole is obtained by adding the allowance to the basic size of the shaft. This gives the design size for the hole. Tolerances are then applied to each part. In this system, the upper deviation of the shaft is zero. The letter symbol for this situation is ‘h’. The shaft basis system is preferred by (i) industries using semi-finished shafting as raw materials, e.g., textile industries, where spindles of same size are used as cold-finished shafting and (ii) when several parts having different fits but one nominal size is required on a single shaft.



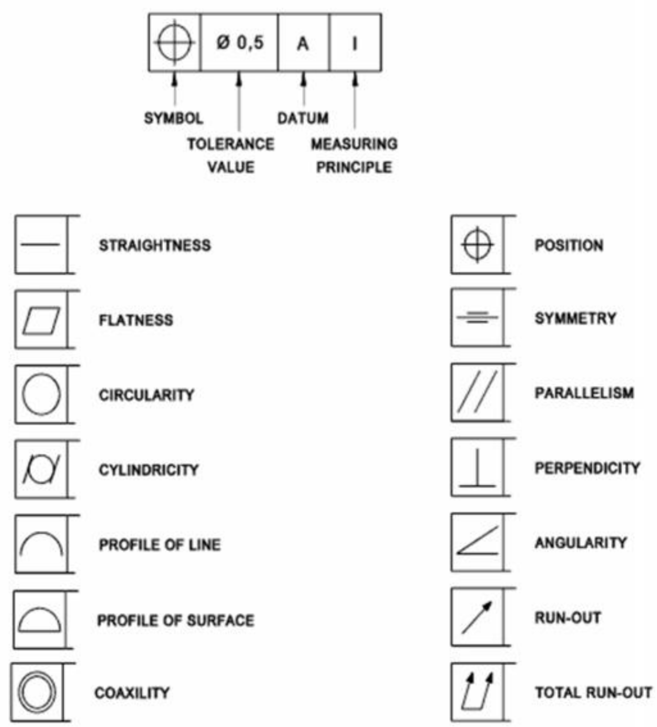
Examples illustrating shaft basis and hole basis systems

Geometric Dimensioning & Tolerance (GD&T)

Geometric dimensioning and tolerance (GD&T) is a language used on mechanical engineering drawings composed of symbols that are used to efficiently and accurately communicate geometry requirements for associated features on components and assemblies. GD&T is, and has been, successfully used for many years in the automotive, aerospace, electronics and the commercial design and manufacturing industries. In today's modern and technically advanced design, engineering and manufacturing world, effective and accurate communication is required to ensure successful end products. Success oriented industries and organizations which require accurate and common lines of communications between engineering, design, manufacturing and quality should consider geometric dimensioning and tolerance (GD&T) as their mechanical drawing standard. Some advantages of GD&T (geometric dimensioning and tolerance) are;

1. Provides a clear and concise technique for defining a reference coordinate system (datum's) on a component or assembly to be used throughout the manufacturing and inspection processes.

2. Proper application of geometric dimensioning closely dovetails accepted and logical mechanical design process and design for manufacturing considerations.
3. Geometric dimensioning dramatically reduces the need for drawing notes to describe complex geometry requirements on a component or assembly by the use of standard symbol that accurately and quickly defines design, manufacturing and inspection requirements.
4. GD&T concepts such as MMC (maximum material condition) when applied properly will facilitate and simplify the design of cost saving functional check gages, manufacturing fixtures and jigs.



EX.NO:2 **FLANGE COUPLING ASSEMBLY**

DATE:

AIM:

To draw the detail view of the flange coupling and assemble the parts by using the Pro-E software and obtain its respective views.

PROCEDURE:

- The drawings of Flanges, Shaft, Taper key, Hexagonal Bolt and Nut are studied
- 3D models of Flanges, Shaft, Taper key, Hexagonal Bolt and Nut are created using Pro-E software.
- The Assembly of Flanged Coupling was created as per the drawing specification
- Detail all the components of the assembly as per the drawing standards.

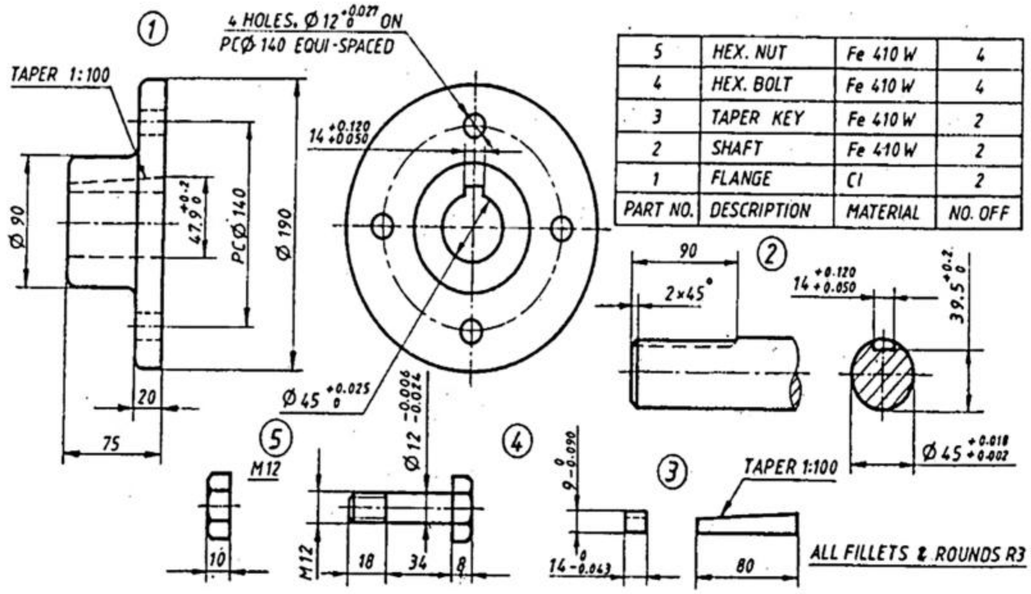
COMMANDS USED:

Sketcher Commands: Line, Circle, Arc, Fillet, Trim, Smart Dimension, Relations, Show, and View

Features Commands: Extrude and Cut, Revolve, Fillet/Round, Chamfer, Hole - Simple, Pattern Fastening Features

Assembly Commands: Insert, Component, Existing Part/Assembly

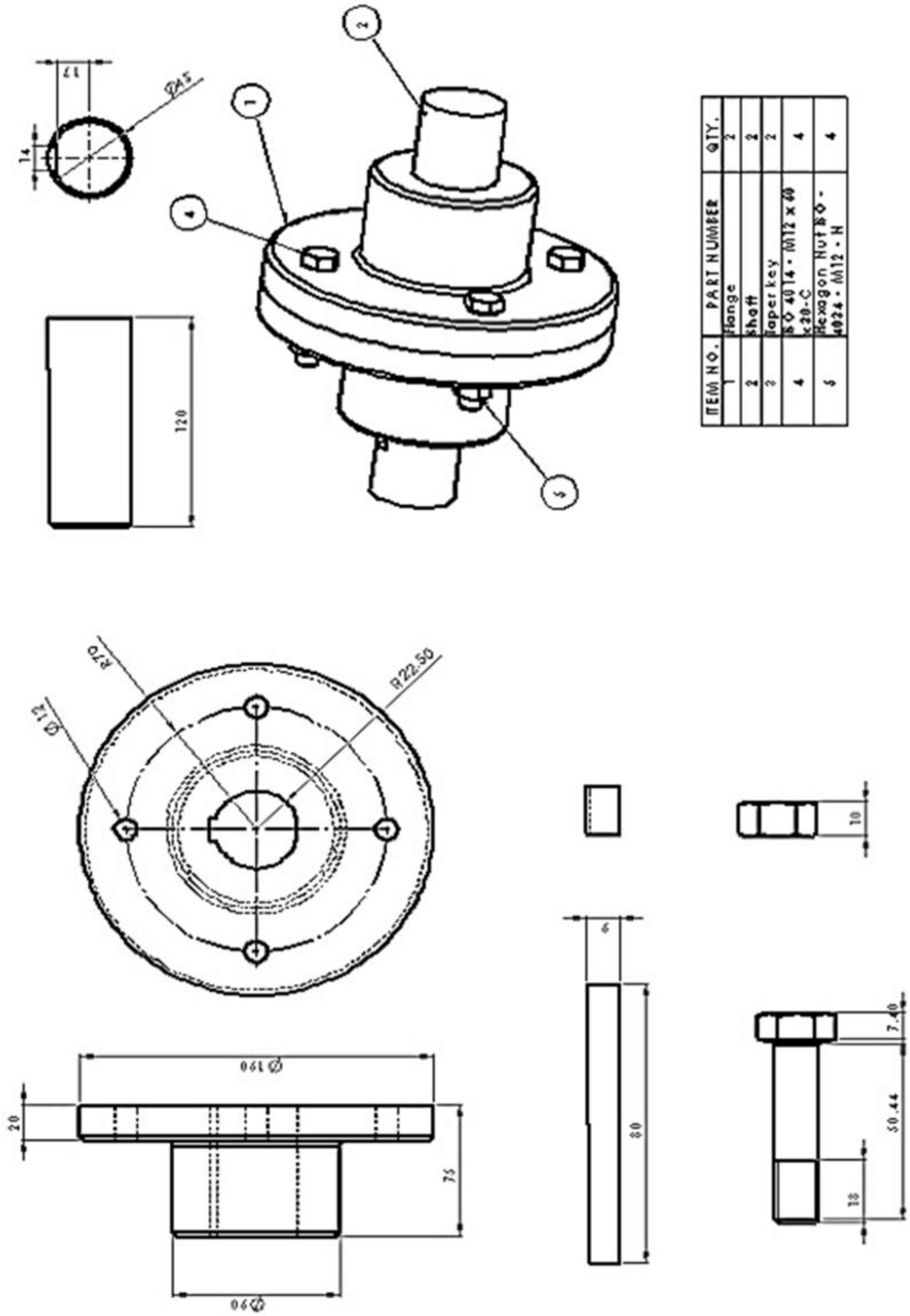
Mating Commands: Coincident, Concentric, Distance



PART NO.	DESCRIPTION	MATERIAL	NO. OFF
5	HEX. NUT	Fe 410 W	4
4	HEX. BOLT	Fe 410 W	4
3	TAPER KEY	Fe 410 W	2
2	SHAFT	Fe 410 W	2
1	FLANGE	CI	2

All Dimensions in mm

Details of Flanged Coupling — Unprotected Type



RESULT: Thus the Detail View of the flange coupling assembly and its respective views has been drawn.

EX.NO:3 **UNIVERSAL COUPLING ASSEMBLY**

DATE:

AIM:

To draw the detail view of the universal coupling and assemble the parts by using the Pro-E software and obtain its respective views.

PROCEDURE:

- The drawings of Fork, Shaft, Centre, Parallel key, Pin, Collar and Taper pin are studied.
- 3D models of all the parts are created using Pro-E software.
- The Assembly of Universal Joint was created as per the drawing specification.
- Detail all the components of the assembly as per the drawing standards.

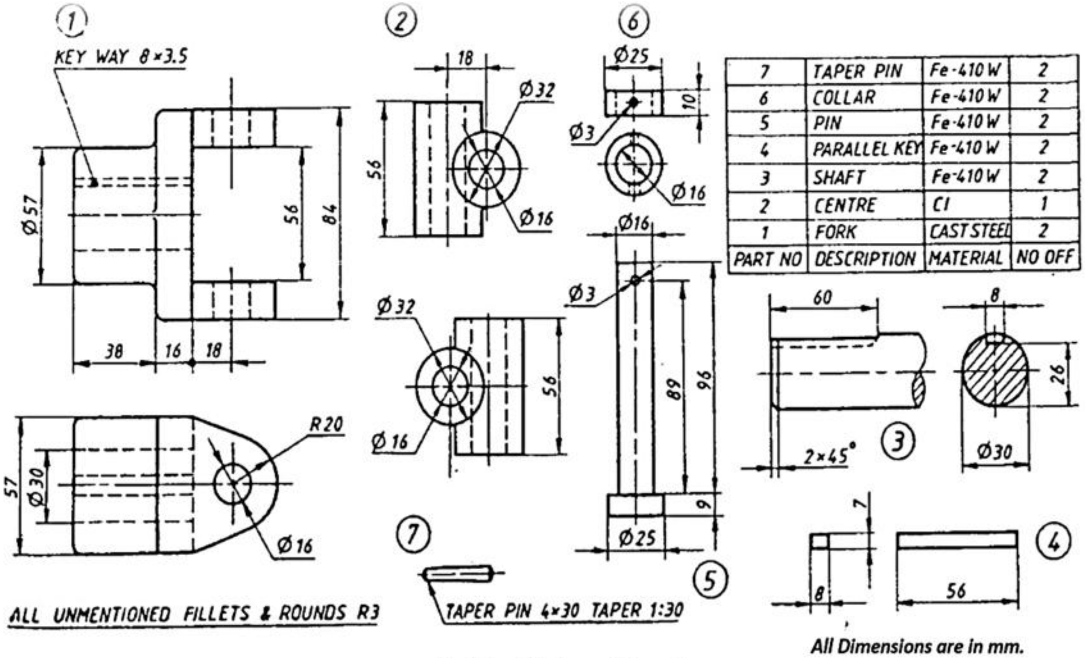
COMMANDS USED:

Sketcher Commands: Line, Circle, Arc, Fillet, Trim, Smart Dimension, Relations, Show, and View

Features Commands: Extrude and Cut, Revolve, Fillet/Round, Chamfer, Hole - Simple, Pattern Fastening Features

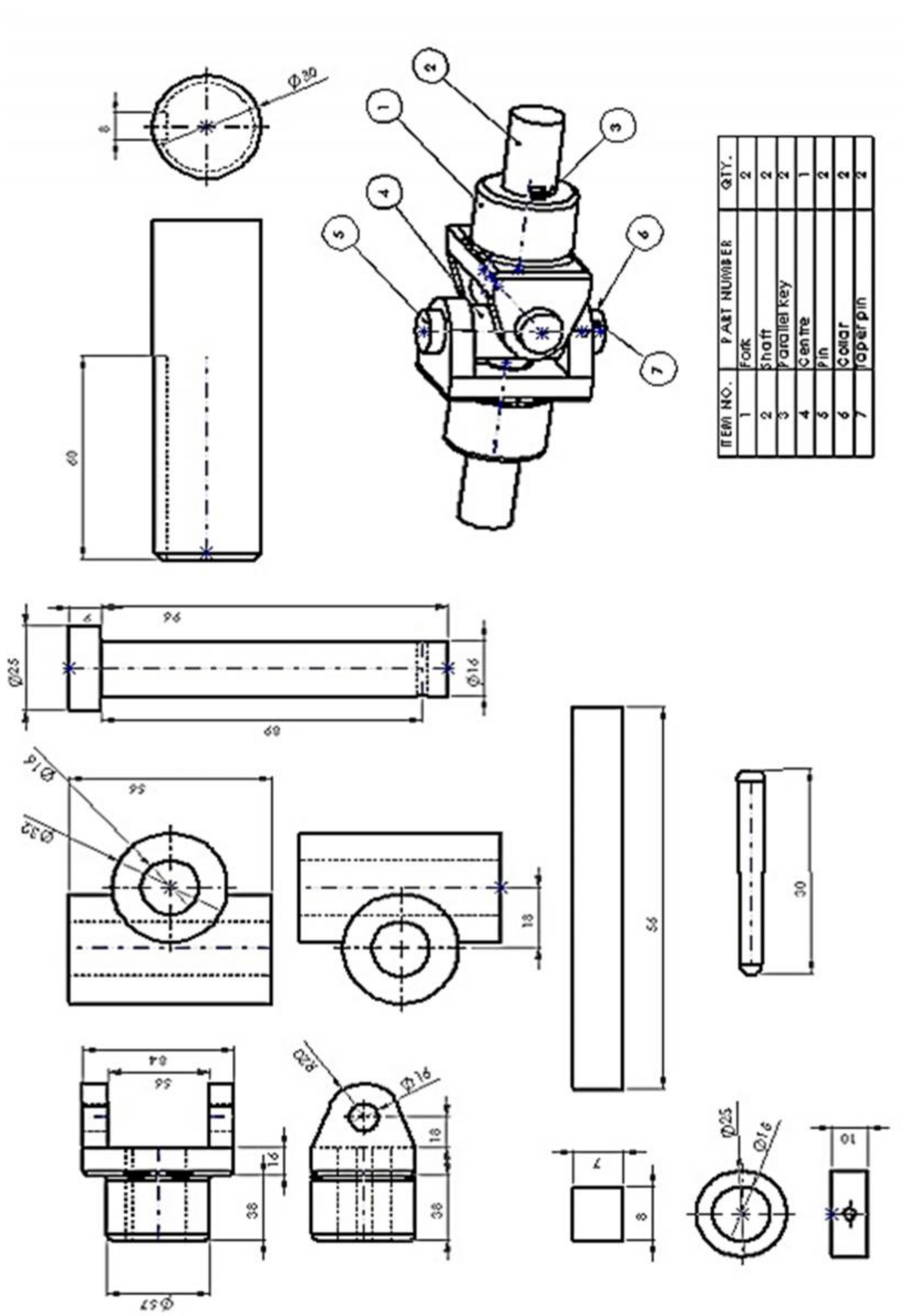
Assembly Commands: Insert, Component, Existing Part/Assembly

Mating Commands: Coincident, Concentric, Distance



PART NO	DESCRIPTION	MATERIAL	NO OFF
7	TAPER PIN	Fe-410 W	2
6	COLLAR	Fe-410 W	2
5	PIN	Fe-410 W	2
4	PARALLEL KEY	Fe-410 W	2
3	SHAFT	Fe-410 W	2
2	CENTRE	CI	1
1	FORK	CAST STEEL	2

Details of Universal Coupling



RESULT: Thus the Detail View of the universal coupling assembly and its respective views has been drawn.

EX.NO:4 **SCREW JACK ASSEMBLY**

DATE:

AIM:

To draw the detail view of the screw jack and assemble the parts by using the Pro-E software and obtain its respective views.

PROCEDURE:

- The drawings of Body, Nut, Screw Spindle, Cup, Washer Special, CSK Screw, and Tommy Bar are studied
- 3D models of Body, Nut, Screw Spindle, Cup, Washer Special, CSK Screw, and Tommy Bar are created using Pro-E software.
- The Assembly of Screw Jack was created as per the drawing specification
- Detail all the components of the assembly as per the drawing standards.

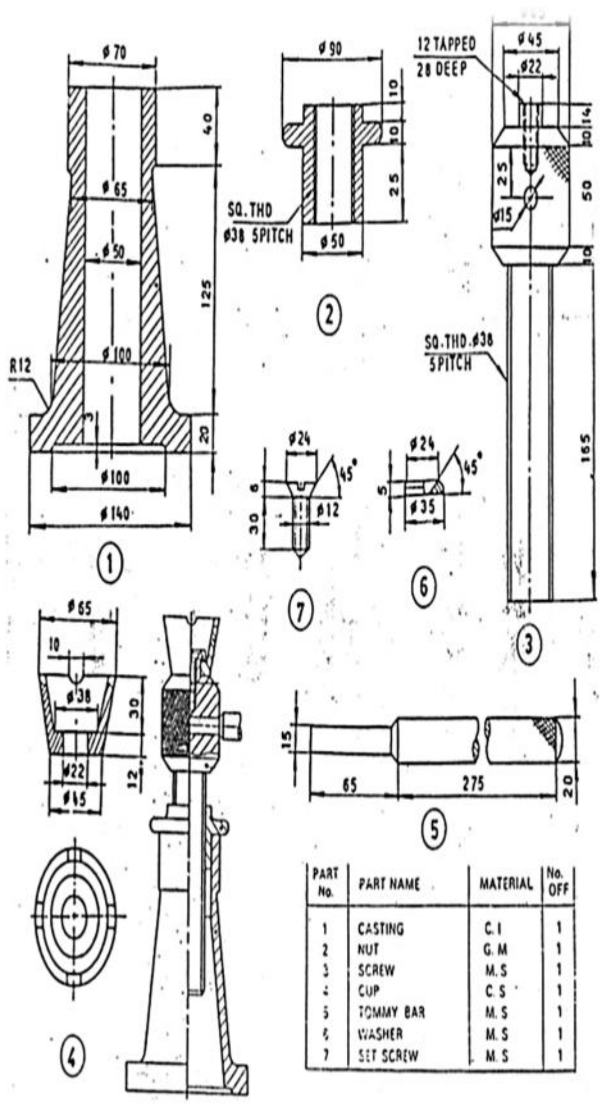
COMMANDS USED:

Sketcher Commands: Line, Circle, Arc, Fillet, Trim, Smart Dimension, Relations, Show, and View

Features Commands: Extrude and Cut, Revolve, Fillet/Round, Chamfer, Hole - Simple, Pattern Fastening Features

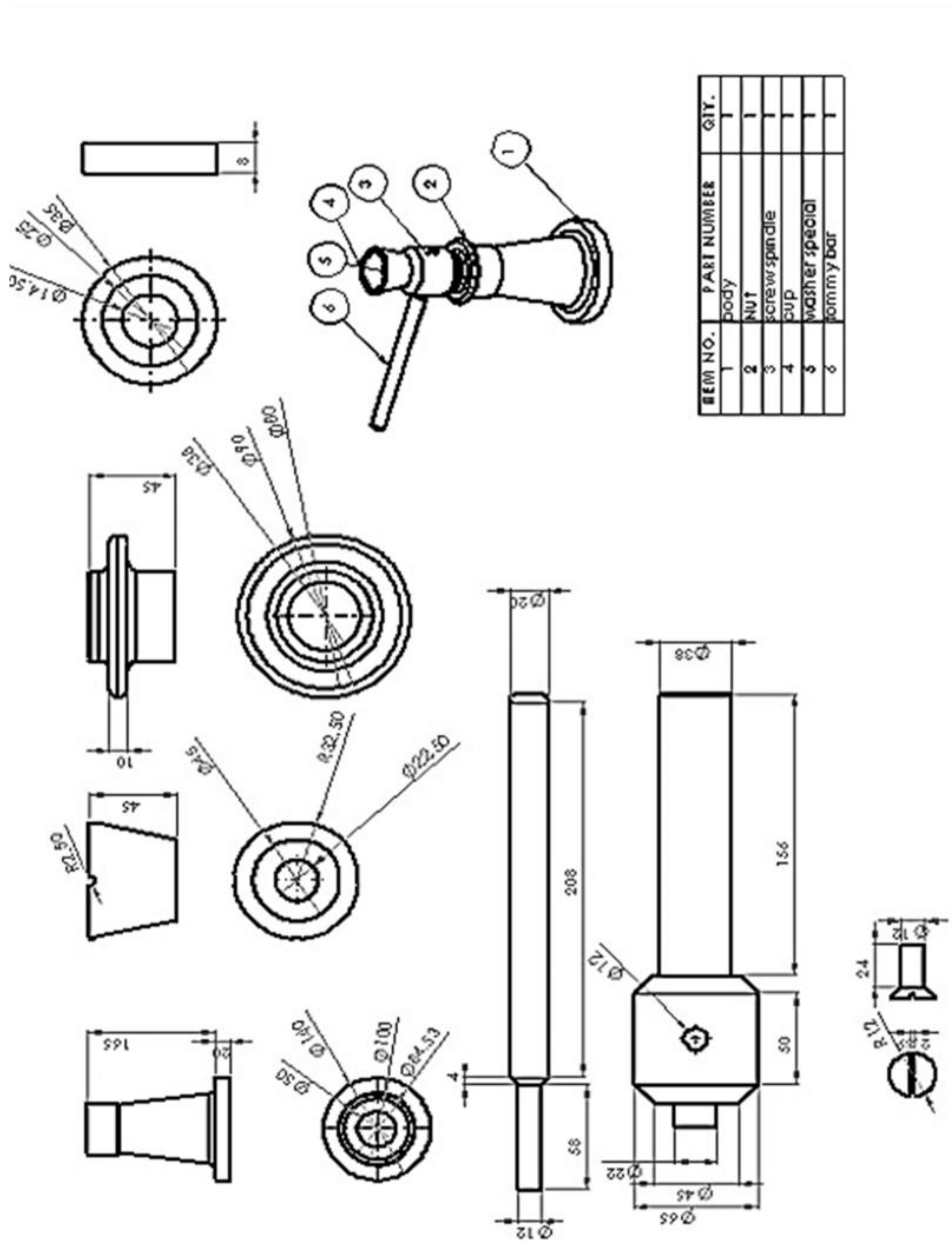
Assembly Commands: Insert, Component, Existing Part/Assembly

Mating Commands: Coincident, Concentric, Distance



PART No.	PART NAME	MATERIAL	No. OFF
1	CASTING	C. I	1
2	NUT	G. M	1
3	SCREW	M. S	1
4	CUP	C. S	1
5	TOMMY BAR	M. S	1
6	WASHER	M. S	1
7	SET SCREW	M. S	1

All Dimensions are in mm.



RESULT: Thus the Detail View of the screw jack assembly and its respective views has been drawn.

EX.NO:5 **STUFFING BOX ASSEMBLY**

DATE:

AIM:

To draw the detail view of the stuffing box and assemble the parts by using the Pro-E software and obtain its respective views.

PROCEDURE:

- The drawings of stuffing box, Nut, Gland, Piston rod, and Packing are studied.
- 3D models of all the parts are created using Pro-E software.
- The Assembly of Stuffing Box was created as per the drawing specification.
- Detail all the components of the assembly as per the drawing standards.

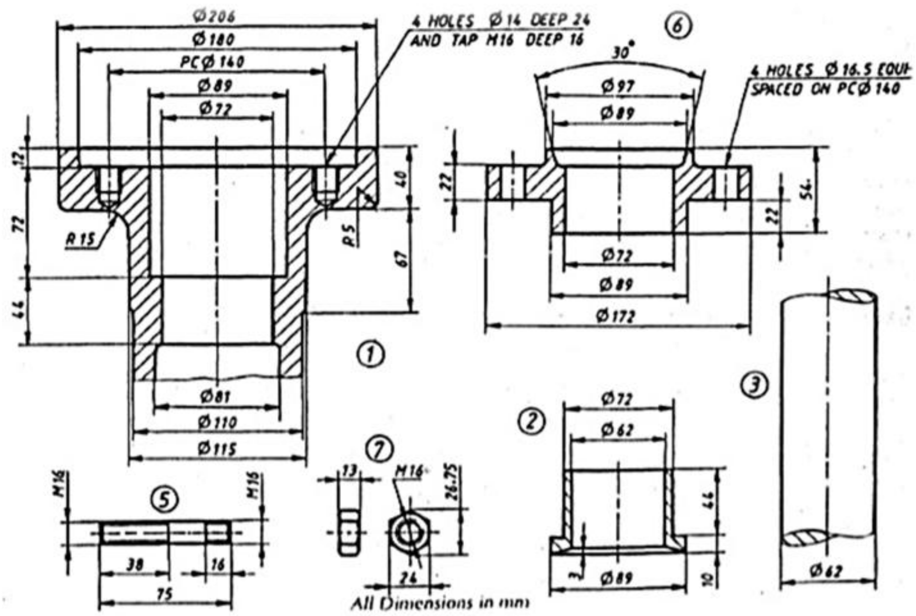
COMMANDS USED:

Sketcher Commands: Line, Circle, Arc, Fillet, Trim, Smart Dimension, Relations, Show, and View

Features Commands: Extrude and Cut, Revolve, Fillet/Round, Chamfer, Hole - Simple, Pattern Fastening Features

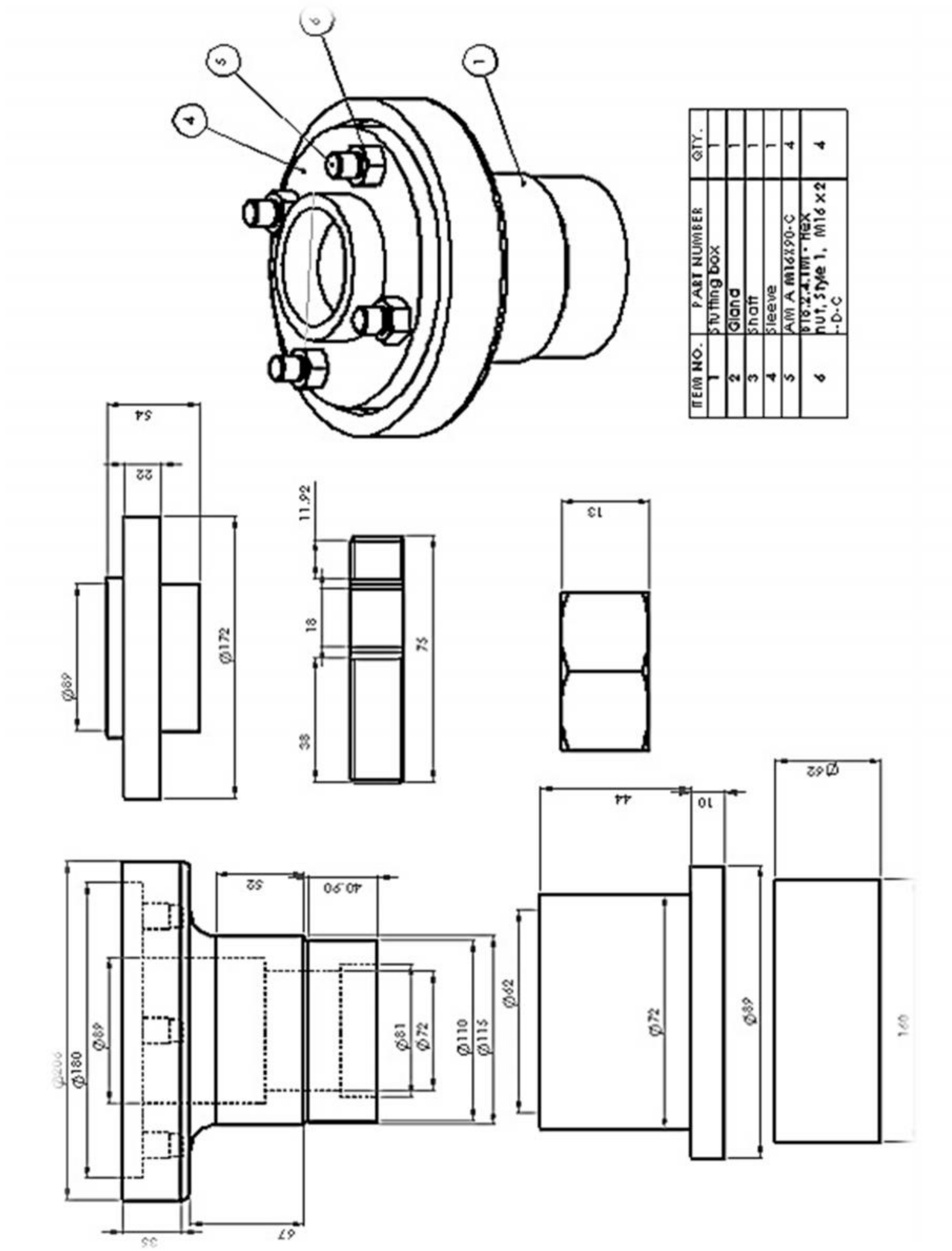
Assembly Commands: Insert, Component, Existing Part/Assembly

Mating Commands: Coincident, Concentric, Distance



7	Nut	Fe-410W	4
6	Sleeve	Cast Iron	1
5	Stud	Fe-410W	4
4	Packing - Sq.9	Asbestor Rope	-
3	Shaft	Fe-410W	1
2	Gland	Brass	1
1	Stuffing Box	Cast Iron	1
Sl. No.	Part Name	Material	No. Off

Continued...



RESULT: Thus the Detail View of the stuffing box assembly and its respective views has been drawn.

EX.NO:6 **PLUMMER BLOCK ASSEMBLY**

DATE:

AIM:

To draw the detail view of the Plummer block and assemble the parts by using the Pro-E software and obtain its respective views.

PROCEDURE:

- The drawings of Body, Cap, Bearing top & Bottom half, Nuts and shaft are studied
- 3D models of Body, Cap, Bearing top & Bottom half Nuts and shaft are created using Pro-E software.
- The Assembly of Plummer block was created as per the drawing specification.
- Detail all the components of the assembly as per the drawing standards.

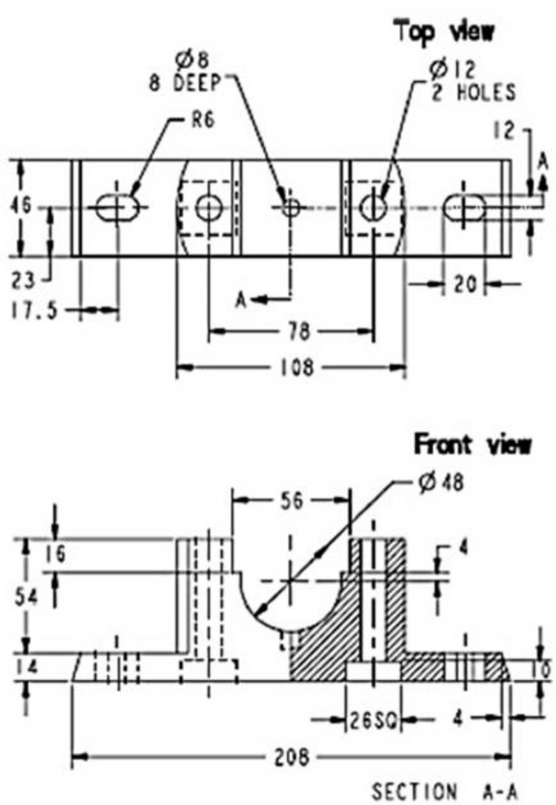
COMMANDS USED:

Sketcher Commands: Line, Circle, Arc, Fillet, Trim, Smart Dimension, Relations, Show, and View

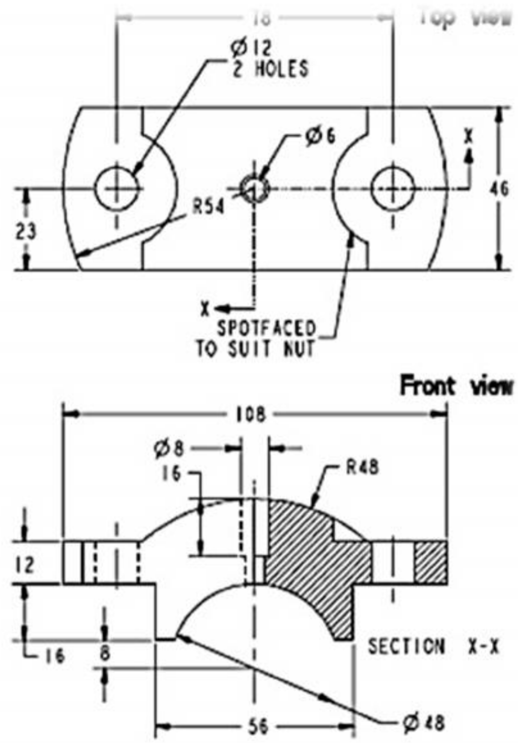
Features Commands: Extrude and Cut, Revolve, Fillet/Round, Chamfer, Hole - Simple, Pattern Fastening Features

Assembly Commands: Insert, Component, Existing Part/Assembly

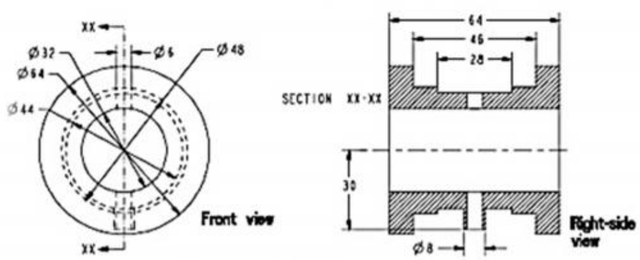
Mating Commands: Coincident, Concentric, Distance



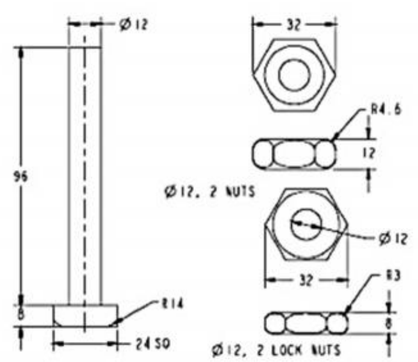
Dimensions for Casting



Dimensions for Cap



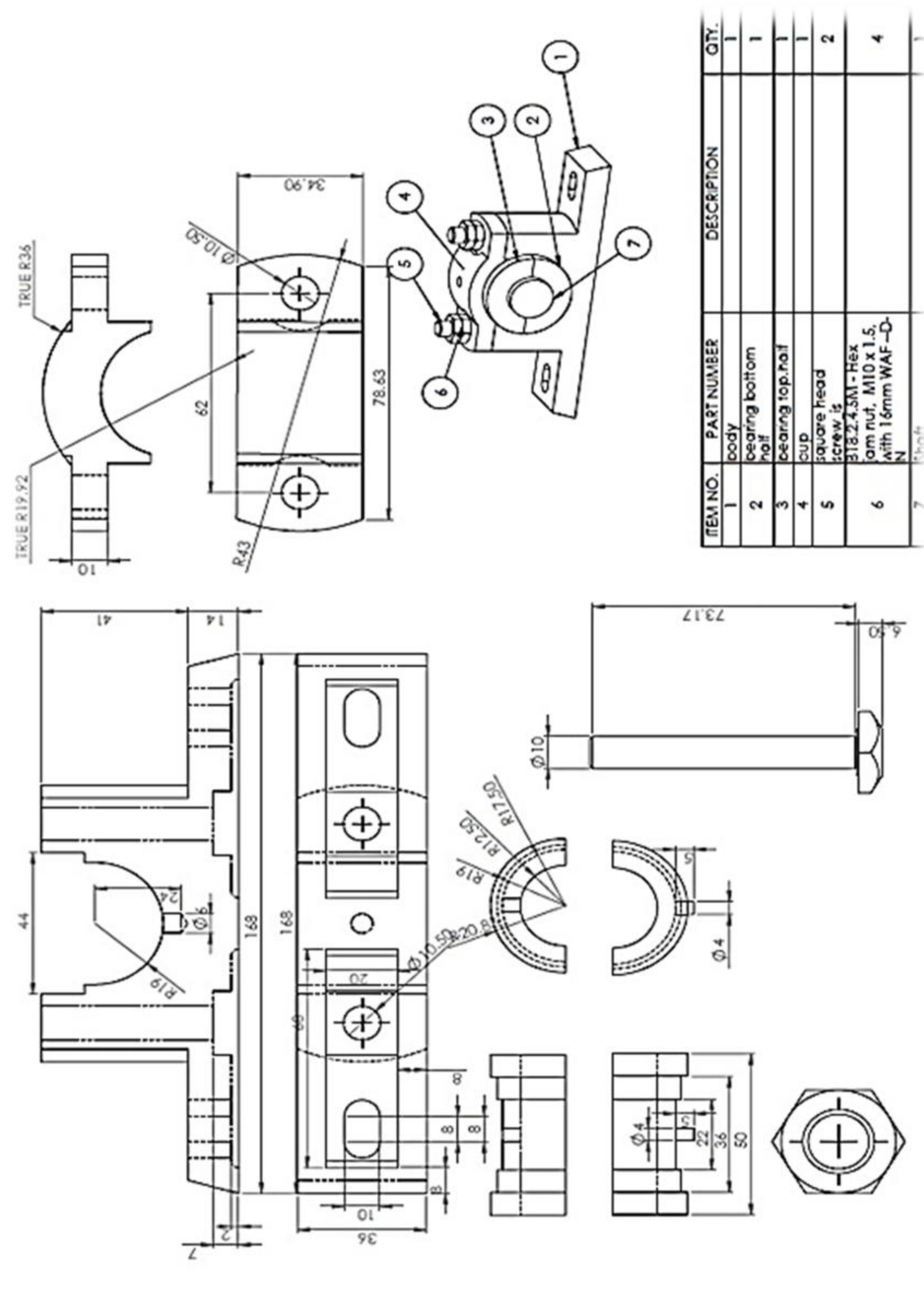
Dimensions for the Brasses



Dimensions for the components of the assembly

S. NO.	NO. OFF	PART'S NAME	MATERIAL
1	1	BODY	CAST IRON
2	1	CAP	CAST IRON
3	1	BRASSES	GUN METAL
4	2	SQUARE HEADED BOLT	MILD STEEL
5	2	NUT	MILD STEEL
6	2	LOCK NUT	MILD STEEL

The Bill Of Material for Pedestal Bearing



RESULT: Thus the Detail View of the Plummer block assembly and its respective views has been drawn.

COMPUTER AIDED MANUFACTURING (CAM)

EX.NO:7 **INTRODUCTION**

The part program is a sequence of instructions, which describe the work, which has to be done on a part, in the form required by a computer under the control of a numerical control computer program. It is the task of preparing a program sheet from a drawing sheet. All data is fed into the numerical control system using a standardized format. Programming is where all the machining data are compiled and where the data are translated into a language which can be understood by the control system of the machine tool. The machining data is as follows:

(a) Machining sequence classification of process, tool start up point, cutting depth, tool path, etc.

(b) Cutting conditions, spindle speed, feed rate, coolant, etc.

(c) Selection of cutting tools.

While preparing a part program, need to perform the following steps :

(a) Determine the startup procedure, which includes the extraction of dimensional data from part drawings and data regarding surface quality requirements on the machined component.

(b) Select the tool and determine the tool offset.

(c) Set up the zero position for the work piece.

(d) Select the speed and rotation of the spindle.

(e) Set up the tool motions according to the profile required.

(f) Return the cutting tool to the reference point after completion of work.

(g) End the program by stopping the spindle and coolant.

The part programming contains the list of coordinate values along the X, Y and Z directions of the entire tool path to finish the component. The program should also contain information, such as feed and speed. Each of the necessary instructions for a particular operation given in the part program is known as an NC word. A group of such NC words constitutes a complete NC instruction, known as block. The commonly used words are N, G, F, S, T, and M. The same is explained later on through examples.

Hence the methods of part programming can be of two types depending upon the two techniques as below:

(a) Manual part programming, and

(b) Computer aided part programming.

MANUAL PART PROGRAMMING

The programmer first prepares the program manuscript in a standard format. Manuscripts are typed with a device known as flexo writer, which is also used to type the program instructions. After the program is typed, the punched tape is prepared on the flexo writer. Complex shaped components require tedious calculations. This type of programming is carried out for simple machining parts produced on point-to-point machine tool.

To be able to create a part program manually, need the following information:

(a) Knowledge about various manufacturing processes and machines.

(b) Sequence of operations to be performed for a given component.

(c) Knowledge of the selection of cutting parameters.

(d) Editing the part program according to the design changes.

(e) Knowledge about the codes and functions used in part programs.

FUNDAMENTAL ELEMENTS FOR DEVELOPING MANUAL PART PROGRAMME

- Type of Dimensioning System
- Axis Designation
- NC Words
- Standard G and M Codes
- Tape Programming Format
- Machine Tool Zero Point Setting
- Coordinate Word
- Parameter for Circular Interpolation
- Spindle Function
- Feed Function
- Tool Function
- Work Settings and Offsets
- Rapid Positioning
- Linear Interpolation
- Circular Interpolation

MANUAL PART PROGRAMMING IN CNC LATHE

G-CODES (PREPARATORY FUNCTIONS)

CODE	FUNCTION
G00	Rapid positioning
G01	Linear interpolation
G02	Circular interpolation clockwise (CW)
G03	Circular interpolation counterclockwise (CCW)
G20	Inch input (in.)
G21	Metric input (mm)
G24	Radius programming
G28	Return to reference point
G29	Return from reference point
G32	Thread cutting
G40	Cutter compensation cancel
G41	Cutter compensation left
G42	Cutter compensation right
G43	Tool length compensation positive (+) direction
G44	Tool length compensation minus (-) direction
G49	Tool length compensation cancels
G 53	Zero offset or M/c reference
G54	Settable zero offset
G84	canned turn cycle
G90	Absolute programming
G91	Incremental programming

M-CODES (MISCELLANEOUS FUNCTIONS)

CODE	FUNCTION
M00	Program stop
M02	End of program
M03	Spindle start (forward CW)
M04	Spindle start (reverse CCW)
M05	Spindle stop
M06	Tool change
M08	Coolant on
M09	Coolant off
M10	Chuck - clamping
M11	Chuck - unclamping
M12	Tailstock spindle out
M13	Tailstock spindle in
M17	Tool post rotation normal
M18	Tool post rotation reverse
M30	End of tape and rewind or main program end
M98	Transfer to subprogram
M99	End of subprogram

MANUAL PART PROGRAMMING IN CNC MILLING

G-CODES (PREPARATORY FUNCTIONS)

CODE	FUNCTION
G00	Positioning (Rapid traverse)
G01	Linear interpolation (Cutting feed)
G02	Circular interpolation / Helical CW
G03	Circular interpolation / Helical CCW
G04	Dwell Exact stop
G20	Imperial units (inches)
G21	Metric units (mm)
G28	Return to reference point
G40	Tool radius compensation cancel
G41	Left hand radius compensation
G42	Right hand radius compensation
G49	Tool length compensation cancel
G90	Absolute command
G91	Incremental command
G92	Set datum
G94	Feed per minute
G94	Feed per rotation
$G_1$70-$G_1$71	Circular Pocketing
$G_1$72-$G_1$73	Rectangular Pocketing

M-CODES (MISCELLANEOUS FUNCTIONS)

CODE	FUNCTION
M00	Program Stop
M02	Program End
M03	Spindle Forward
M04	Spindle Reverse
M05	Spindle Stop
M06	Tool Change
M70	X Mirror on
M71	Y Mirror on
M80	X Mirror Off
M81	Y Mirror Off
M98	Subprogram Call
M99	Subprogram Exit

COMPUTER AIDED PART PROGRAMMING

If the complex-shaped component requires calculations to produce the component are done by the programming software contained in the computer. The programmer communicates with this system through the system language, which is based on words. There are various programming languages developed in the recent past, such as APT (Automatically Programmed Tools), ADAPT, AUTOSPOT, COMPAT-II, 2CL, ROMANCE, SPLIT is used for writing a computer programme, which has English like statements. A translator known as compiler program is used to translate it in a form acceptable to MCU.

The programmer has to do only following things :

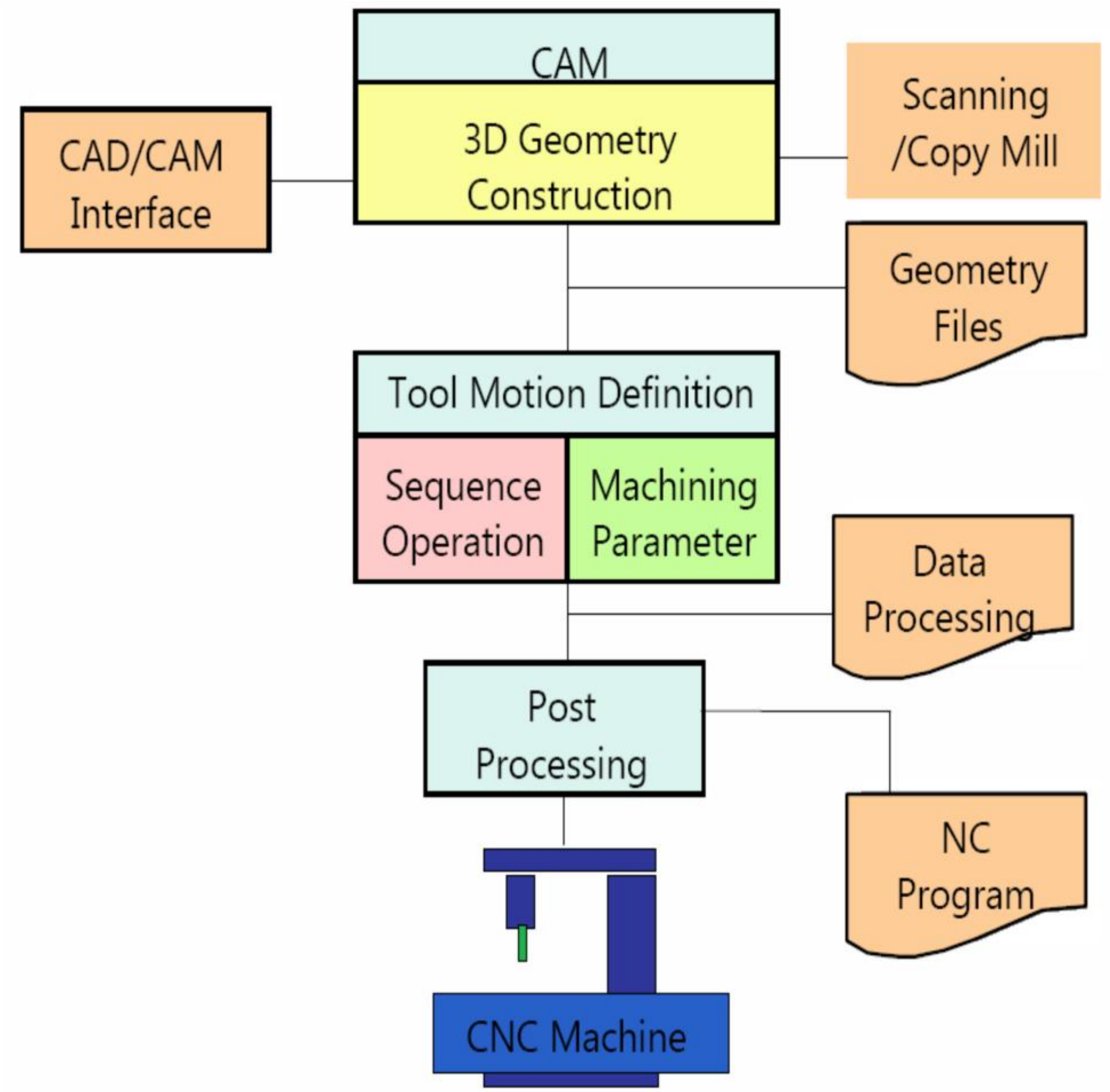
(a) Define the work part geometry.

(b) Defining the repetition work.

(c) Specifying the operation sequence.

Over the past years, lot of effort is devoted to automate the part programme generation. With the development of the CAD (Computer Aided Design)/CAM (Computer Aided Manufacturing) system, interactive graphic system is integrated with the NC part programming. Graphic based software using menu driven technique improves the user friendliness. The part programmer can create the geometrical model in the CAM package or directly extract the geometrical model from the CAD/CAM database. Built in tool motion commands can assist the part programmer to calculate the tool paths automatically. The programmer can verify the tool paths through the graphic display using the animation function of the CAM system. It greatly enhances the speed and accuracy in tool path generation.

INTERACTIVE GRAPHIC SYSTEM IN COMPUTER AIDED PART PROGRAMMING



EX.NO:8 MANUAL PART PROGRAMMING FOR STEP TURNING OPERATION IN CNC TURNING CENTER

DATE:

AIM:

To write the manual part programming for the given diagram and execute the same by using CNC lathe.

MATERIALS REQUIRED:

Mild steel	Size
Diameter	35 mm
Length	100 mm

MANUAL PART PROGRAM:

N01 G54 G90 G71 G94 M03 S800;

N05 G01 X-12.5 Z0 F2;

N10 G00 Z1;

N15 G00 X00;

N20 G01 Z-100;

N25 G00 X1 Z1;

N30 G00 X-2;

N35 G01 Z-60;

N40 G00 X-1 Z1;

N45 G00 X-3;

N50 G01 Z-60;

N55 G00 X-2 Z1;

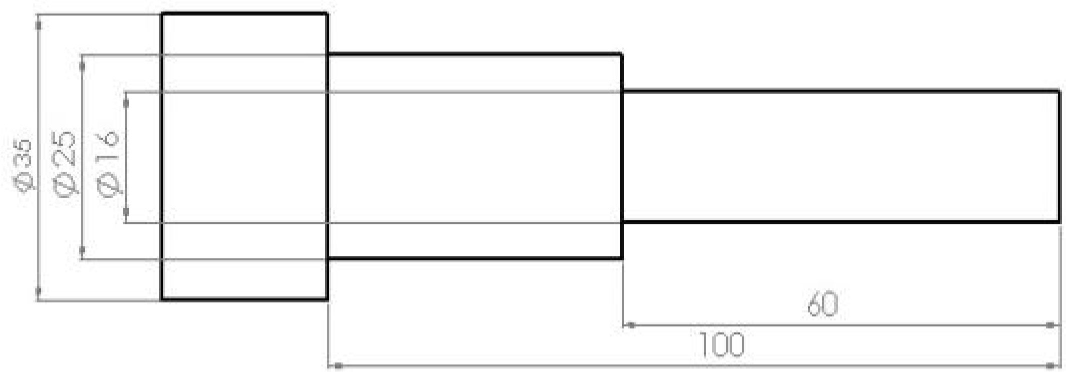
N60 G00 X-4;

```
N65 G01 Z-60;
N70 G00 X-3 Z1;
N75 G00 X-4.5;
N80 G01 Z-60;
N85 G00 X5 Z5;
N90 M02;
```

RESULT:

Thus the part program has been written and executed by using the CNC Lathe.

STEP TURNING OPERATION IN CENTRE LATHE



All the dimensions are in mm

EX.NO:9 MANUALPART PROGRAMMING FOR TAPER TURNING OPERATION IN IN CNC TURNING CENTER

DATE:

AIM:

To write the manual part programming for the given diagram and execute the same by using CNC lathe.

MATERIALS REQUIRED:

Mild steel	Size
Diameter	30 mm
Length	95 mm

MANUAL PART PROGRAM:

N01 G54 G91 G71 G94 M03 S800;

N05 G01 X-15 Z0 F2;

N10 G00 Z1;

N15 G00 X10;

N20 G01 Z-36;

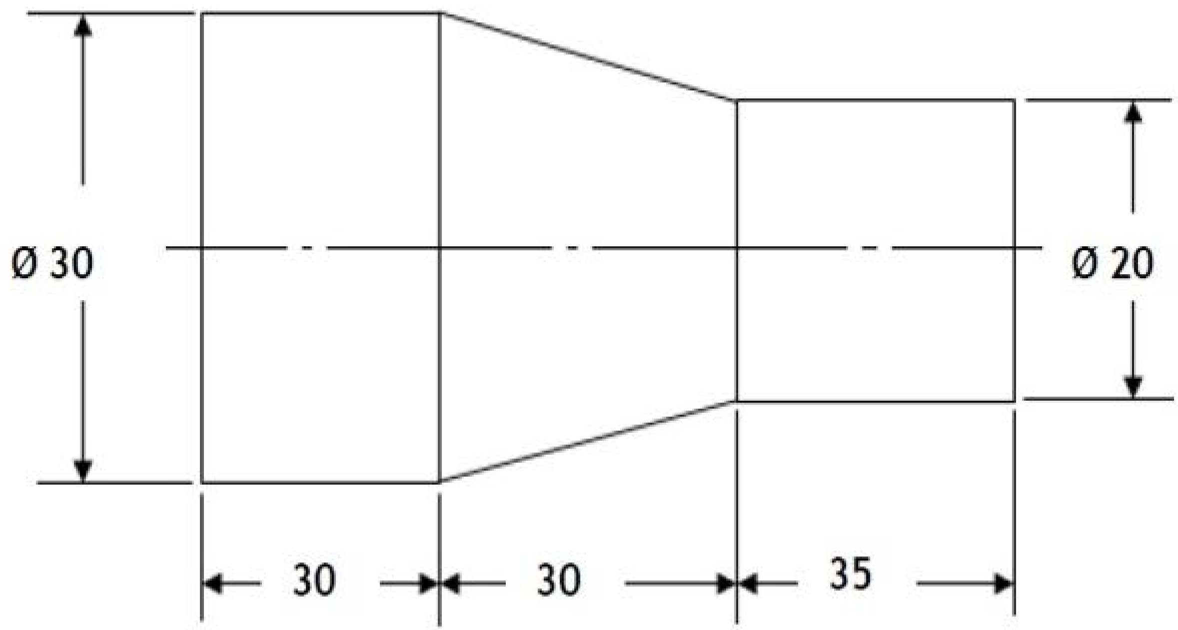
N25 G01 X5 –Z30;

N30 G00 X1 Z66;

N35 M02;

RESULT:

Thus the part program has been written and executed by using the CNC Lathe.


Ø 30
Ø 20
30
30
35

EX.NO:10 NC CODE GENERATION FOR STEP TURNING AND FACING OPERATION BY USING CADEM SOFTWARE

DATE:

AIM:

To generate the NC code for step turning and facing operation by using cadem-capsturn software.

PROCEDURE:

Step 1: open the capsturn software

Step 2: Create the required part as per the given diagram in geometry mode

Step 3: Define the blank as per the requirement

Step 4: Select the suitable tool for turning and facing operation in machining mode

Step 5: Select the required machining operation and define the data

Step 6: Simulate the tool path for the given part

Step 7: Generate the NC code and save the program

NC CODE GENERATION:

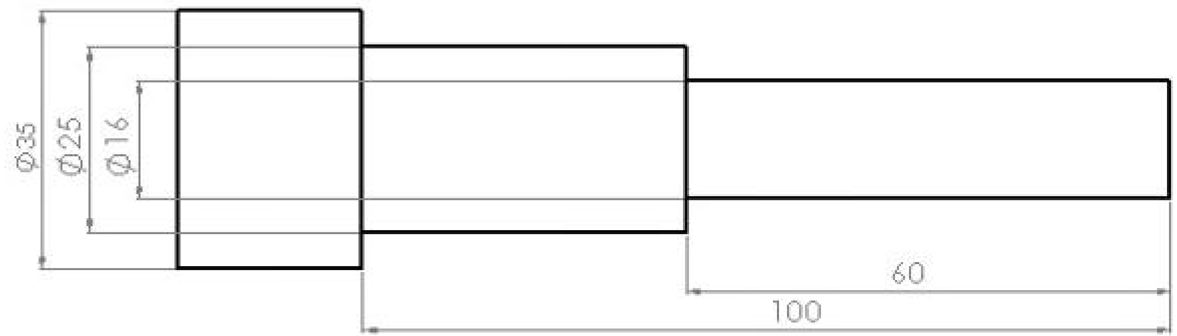
```
%
O1234
G21 G95
G0 X200 Z100
N1 T11 (PCLNL 1616H12 R0.8)
G50 S3000
G96 S139 M04
(PLAIN FACE)
X38 Z4 M08
X39
```

```
G72 W3 R0.5
G72 P25 Q40 U0 W0 F0
N25 G0 Z0
N30 G01 X35 Z0
N35 X0
N40 Z2
G0 X38
(CONTOUR TURN)
Z2
X39
G71 U3 R0.5
G71 P45 Q80 U0 W0 F0
N45 G00 X16
N50 G01 X16 Z0
```

RESULT:

Thus the NC code has been generated by using the cadem-capsturn software.

STEP TURNING AND FACING OPERATION



S.No	Operation	Tool Name
1	OD Facing	PCLNL 1212 H09 R0.8
2	OD Turning	PCLNL 1212 H09 R0.8

All the dimensions are in mm

EX.NO:11 NC CODE GENERATION FOR GROOVING AND THREAD CUTTING OPERATION BY USING CADEM SOFTWARE

DATE:

AIM:

To generate the NC code for grooving and thread cutting operation by using cadem-capsturn software.

PROCEDURE:

Step 1: open the capsturn software

Step 2: Create the required part as per the given diagram in geometry mode

Step 3: Define the blank as per the requirement

Step 4: Select the suitable tool for grooving and thread cutting operation in machining mode

Step 5: Select the required machining operation and define the data

Step 6: Simulate the tool path for the given part

Step 7: Generate the NC code and save the program

NC CODE GENERATION:

```
%O1234
G21 G95
G0 X200 Z100
N1 T11 (PCLNL 2525M12 R0.8)
G50 S3000
G96 S139 M04
(PLAIN FACE)
X38 Z4 M08
X56
G72 W3 R0.5
G72 P25 Q40 U0 W0 F0
N25 G0 Z0
N30 G01 X52 Z0
N35 X0
N40 Z2
G0 X38
```

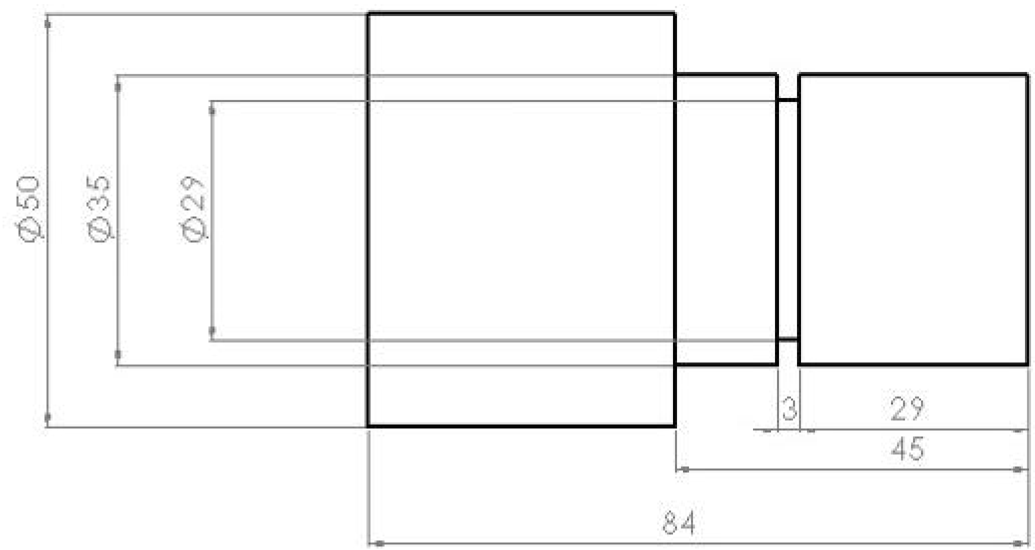
```
(CONTOUR TURN)
Z2
X56
G71 U3 R0.5
G71 P45 Q90 U0 W0 F0
N45 G00 X35
N50 G01 X35 Z0
N55 Z0
N60 Z-29
N65 Z-32
N70 Z-45
N75 X50
N80 Z-84
N85 X52
N90 X55 Z-84
G0 X38
M09
M05
M01
G0 X200 Z100
N2 T22 (12X12, 2.00W, 0.20R, 08DEPTH, LH)
G50 S3000
G96 S190 M04
(EXTERNAL GROOVE)
X39 Z-31 M08
Z-31
G01 X29 F0
G04 X0
G0 X39
Z-31
G01 X29 F0
G04 X0
G0 X39
Z-31
G01 X29
G04 X0
G0 X39
M09
M05
M01
```

```
G0 X200 Z100
N3 T33 (THREAD 16 X 16, 60 DEG., DEPTH 3.0, LH)
G97 S1182 M04
(THREADING)
X38 Z-31 M08
X39
G76 P20060 Q0 R0
G76 X29 Z0 R0 P3000 Q1732 F2
X38
M09
M05
X200 Z100
M30
%
```

RESULT:

Thus the NC code has been generated by using the cadem-capsturn software.

GROOVING AND THREAD CUTTING OPERATION



S.No	Operation	Tool Name
1	Grooving	2 mm width
2	Thread cutting	60 deg,3 mm

All the dimensions are in mm

EX.NO:12 MANUALPART PROGRAMMING FOR DRILLING OPERATION

DATE:

AIM:

To write the manual part programming for the given diagram and execute the same by using CNC milling center.

MATERIALS REQUIRED:

Mild steel	Size
Breadth	124 mm
Length	124 mm
Thick	24 mm

MANUAL PART PROGRAM:

N5 G17 G71 G90 G94 G55;

N10 T1 L90;

N15 G00 D5 Z5 M3 S600 X27 Y27;

N20 G81 R02=5, R03=-33, R11=3, F50 M7;

N25 X97;

N30Y97;

N35 X27;

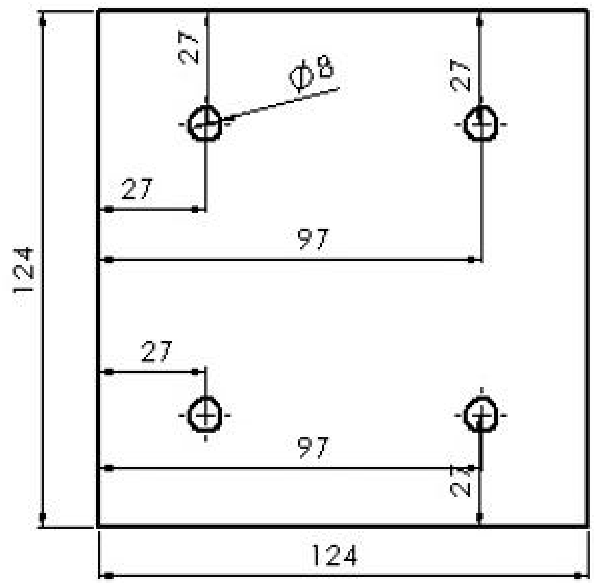
N40 G00 G80 Z100 M9;

N45 M02;

RESULT:

Thus the part program has been written and executed by using the CNC milling center.

DRILLING CYCLE



All the dimensions are in mm

EX.NO:13 NC CODE GENERATION FOR DRILLING OPERATION BY USING CADEM SOFTWARE

DATE:

AIM:

To generate the NC code for drilling operation by using cadem-capsmill software.

PROCEDURE:

Step 1: open the capsmill software

Step 2: Create the required part as per the given diagram in geometry mode

Step 3: Define the blank as per the requirement

Step 4: Select the suitable tool for drilling operation in machining mode

Step 5: Select the required machining operation and define the data

Step 6: Simulate the tool path for the given part

Step 7: Generate the NC code and save the program

NC CODE GENERATION:

```
%
O1234
N1 T1 (8.00 MM. DIA. TWIST DRILL)
M98 P9999
T1
(PECK DRILL)
G90 G00 G54 X27 Y27 M8
G43 H1 Z100
Z3
S3342 M3
G99 G73 Z-12 R3 Q5 F434
Y97
```

X97

Y27

G80

M5

M9

Z100

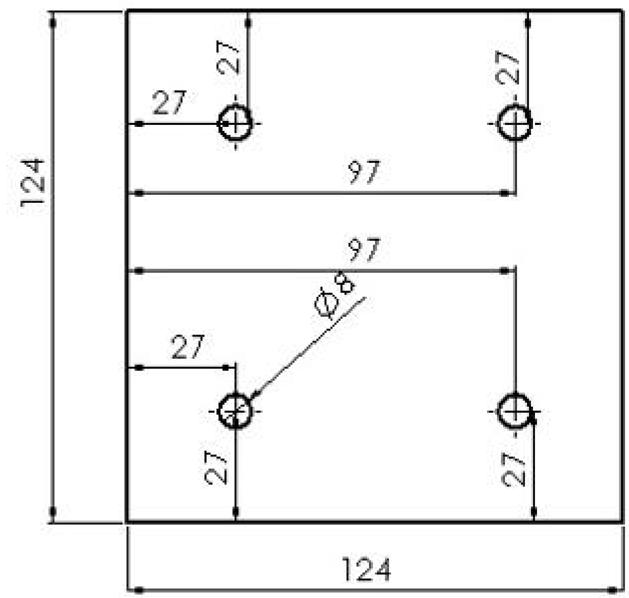
M30

%

RESULT:

Thus the NC code has been generated by using the cadem-capsmill software.

DRILLING OPERATION



S.No	Operation	Tool Name
1	Drilling	Dia 8.0 mm HSS twist drill

All the dimensions are in mm

EX.NO:14 NC CODE GENERATION FOR SIDE MILLING OPERATION BY USING CADEM SOFTWARE

DATE:

AIM:

To generate the NC code for side milling operation by using cadem-capsmill software.

PROCEDURE:

Step 1: open the capsmill software

Step 2: Create the required part as per the given diagram in geometry mode

Step 3: Define the blank as per the requirement

Step 4: Select the suitable tool for side milling operation in machining mode

Step 5: Select the required machining operation and define the data

Step 6: Simulate the tool path for the given part

Step 7: Generate the NC code and save the program

NC CODE GENERATION:

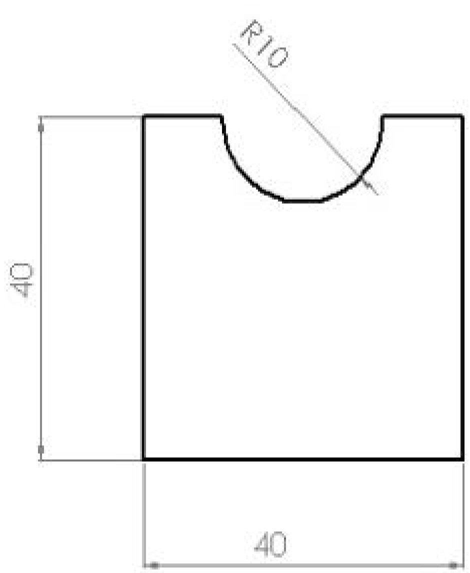
```
%
O1234
N1 T1 (10.00 MM. DIA. END MILL-ROUGH-3 FLUTE)
M98 P9999
T1
(SIDE MILLING)
S891 M3
G90 G00 G54 X17 Y40 M8
G43 H1 Z100
Z3
G01 Z-5 F131
```

```
G03 X23  I3 J0 F187
G00
Z3
X15
G01 Z-5 F131
G03 X25  I5 J0 F187
G00
Z3
M5
M9
Z100
M30
%
```

RESULT:

Thus the NC code has been generated by using the cadem-capsmill software.

SIDE MILLING OPERATION



S.No	Operation	Tool Name
1	Side milling	Dia 10.0 mm end mill
Billet thickness=10 mm,Depth of cut=5 mm		

All the dimensions are in mm

EX.NO:15 NC CODE GENERATION FOR POCKET MILLING, DRILLING AND TAPPING OPERATION BY USING CADEM SOFTWARE

DATE:

AIM:

To generate the NC code for pocket milling, drilling and tapping operation by using cadem-capsmill software.

PROCEDURE:

Step 1: open the capsmill software

Step 2: Create the required part as per the given diagram in geometry mode

Step 3: Define the blank as per the requirement

Step 4: Select the suitable tool for milling, drilling and tapping operation in machining mode

Step 5: Select the required machining operation and define the data

Step 6: Simulate the tool path for the given part

Step 7: Generate the NC code and save the program

NC CODE GENERATION:

```
%
O1234
N1 T1 (10.00 MM. DIA. END MILL-ROUGH-3 FLUTE)
M98 P9999
T2
(POCKET MILLING)
S891 M3
G90 G00 G54 X26 Y21 M8
G43 H1 Z100
Z3
```

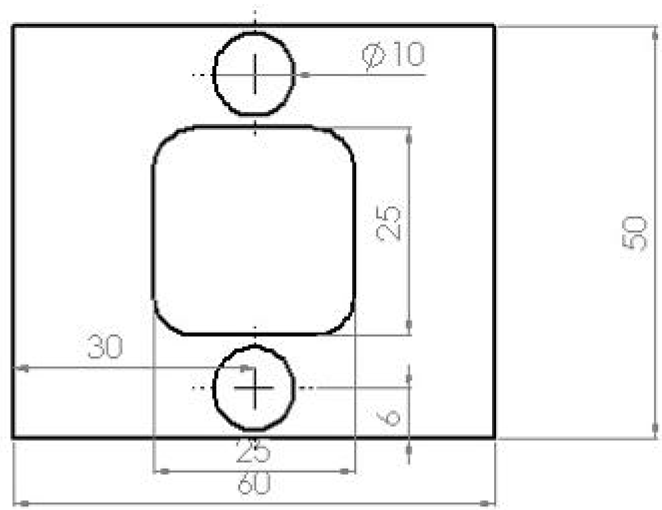
```
G01 Z-4 F131
M98 P155
G90 G00 X22 Y29
Z3
X26 Y21
Z-3
G01 Z-8 F131
M98 P155
G90 G00 X22 Y29
Z3
X26 Y21
Z-7
G01 Z-10 F131
M98 P155
G90 G00 X22 Y29
Z35
M5
M9
Z100
M01
N2  (9.00 MM. DIA. TWIST DRILL)
M98 P9999
T3
(PECK DRILL)
G54 X30 Y6 M8
G43 H2 Z100
Z3
```

```
S2970 M3
G99 G73 Z-17 R3 Q5 F445
Y44
G80
M5
M9
Z100
M01
N3  (M10.00 X 1.50 TAP)
M98 P9999
T1
(TAP)
G54 X30 Y6 M8
G43 H3 Z100
Z3
S477 M3
G99  G84 Z-8 R3 F715
Y44
G80
M5
M9
Z100
M30
%
```

RESULT:

Thus the NC code has been generated by using the cadem-capsmill software.

POCKET MILLING,DRILLINGAND TAPPING OPERATION



S.No	Operation	Tool Name
1	Pocket milling	Dia 10.00 mm end mill
2	Drilling	Dia 9.0 mm HSS twist drill
3	Tapping	M10 x 1.5 mm HSS tap

All the dimensions are in mm

EX.NO:16 NC CODE GENERATION FOR MIRRORING AND POCKET MILLING OPERATION BY USING CADEM SOFTWARE

DATE:

AIM:

To generate the NC code for mirroring and pocket milling operation by using cadem-capsmill software.

PROCEDURE:

Step 1: open the capsmill software

Step 2: Create the required part as per the given diagram in geometry mode

Step 3: Define the blank as per the requirement

Step 4: Select the suitable tool for pocket milling operation in machining mode

Step 5: Select the required machining operation and define the data

Step 6: Simulate the tool path for the given part

Step 7: Generate the NC code and save the program

NC CODE GENERATION:

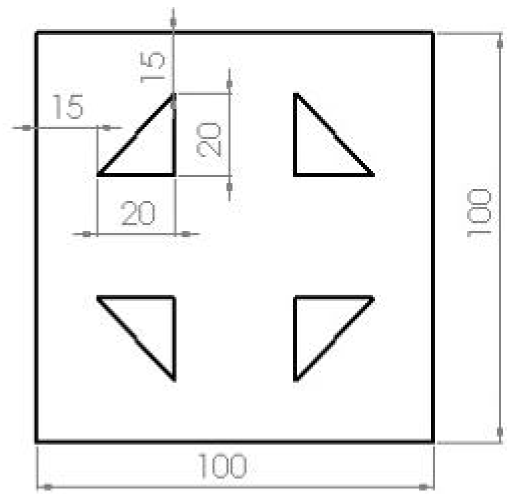
```
%
O1234
N1 T1 (10.00 MM. DIA. END MILL-ROUGH-3 FLUTE)
M98 P9999
T1
(POCKET MILLING)
S891 M3
G90 G00 G54 X30 Y27 M8
G43 H1 Z100
Y27
Z3
G01 Y27 Z-2 F131
M98 P155
```

```
G90 G00 X30 Y27
Z3
(POCKET MILLING)
X27 Y70
G01 X27 Z-2 F131
M98 P156
G90 G00 X27 Y70
Z3
(POCKET MILLING)
X72
G01 X72 Z-2 F131
M98 P157
G90 G00 X72 Y70
Z3
(POCKET MILLING)
X70 Y27
G01 Y27 Z-2 F131
M98 P158
G90 G00 X70 Y27
Z3
M5
M9
Y27
Z100
M30
%
```

RESULT:

Thus the NC code has been generated by using the cadem-capsmill software.

POCKET MILLING,DRILLINGAND TAPPING OPERATION



S.No	Operation	Tool Name
1	Pocket milling	Dia 10.00 mm end mill

All the dimensions are in mm