

**DEPARTMENT OF
MECHANICAL ENGINEERING**

ME8381 – COMPUTER AIDED MACHINE DRAWING LABORATORY

III SEMESTER - R 2017

LABORATORY MANUAL

Name : _____

Register No. : _____

Section : _____

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VISION

College of Engineering is committed to provide highly disciplined, conscientious and enterprising professionals conforming to global standards through value based quality education and training.

MISSION

To provide competent technical manpower capable of meeting requirements of the industry

To contribute to the promotion of academic excellence in pursuit of technical education at different levels

To train the students to sell his brawn and brain to the highest bidder but to never put a price tag on heart and soul

DEPARTMENT OF MECHANICAL ENGINEERING**VISION**

Rendering the services to the global needs of engineering industries by educating students to become professionally sound mechanical engineers of excellent caliber

MISSION

To produce mechanical engineering technocrats with a perfect knowledge intellectual and hands on experience and to inculcate the spirit of moral values and ethics to serve the society

PROGRAMME EDUCATIONAL OBJECTIVES (PEOs)

1. Fundamentals

To impart students with fundamental knowledge in mathematics and basic sciences that will mould them to be successful professionals

2. Core competence

To provide students with sound knowledge in engineering and experimental skills to identify complex software problems in industry and to develop a practical solution for them

3. Breadth

To provide relevant training and experience to bridge the gap between theory and practice which enable them to find solutions for the real time problems in industry and organization and to design products requiring interdisciplinary skills

4. Professional skills

To bestow students with adequate training and provide opportunities to work as team that will build up their communication skills, individual, leadership and supportive qualities and to enable them to adapt and to work in ever changing technologies

5. Life-long learning

To develop the ability of students to establish themselves as professionals in mechanical engineering and to create awareness about the need for lifelong learning and pursuing advanced degrees

PROGRAMME OUTCOMES (POs)

On completion of the B.E. (Mechanical) degree, the graduate will be able

1. To apply the basic knowledge of mathematics, science and engineering
2. To design and conduct experiments as well as to analyze and interpret data and apply the same in the career or entrepreneurship
3. To design and develop innovative and creative software applications
4. To understand a complex real world problem and develop an efficient practical solution
5. To create, select and apply appropriate techniques, resources, modern engineering and IT tools
6. To understand the role as a professional and give the best to the society
7. To develop a system that will meet expected needs within realistic constraints such as economical environmental, social, political, ethical, safety and sustainability
8. To communicate effectively and make others understand exactly what they are trying to tell in both verbal and written forms
9. To work in a team as a team member or a leader and make unique contributions and work with coordination
10. To engage in lifelong learning and exhibit their technical skills
11. To develop and manage projects in multidisciplinary environments

ME8381 – COMPUTER AIDED MACHINE DRAWING LABORATORY**SYLLABUS****COURSE OBJECTIVES**

- To make the students understand and interpret drawings of machine components
- To prepare assembly drawings both manually and using standard CAD packages
- To familiarize the students with Indian standards on drawing practices and standard components
- To gain practical experience in handling 2D drafting and 3D modelling software systems

List of Experiments**A. Drawing standards & fits and tolerances**

Code of practice for Engineering drawing BIS specifications – welding symbols, riveted joints, keys, fasteners- reference to hand book for the selection of standard components like bolts, nuts, screws, keys etc, - limits, fits- tolerancing of individual dimensions-specification of fits-preparation of production drawings and reading of part and assembly drawings, basic principles of geometric dimensioning & tolerancing

B. Introduction to 2D drafting

Drawing, editing, dimensioning, layering, hatching, block, array, detailing, detailed drawing.
Bearings- Bush bearing, plummer block
Valves-safety and non-return valves

C. 3D Geometric modelling and assembly

Sketcher-datum planes-protrusion-holes-part modelling-extrusion-revolve-sweep-loft-blend-fillet-pattern-chamfer-round-mirror-section-assembly
Couplings – Flange, Universal, Oldham's, Muff, Gear couplings
Joints – Knuckle, Gib & cotter, strap, sleeve & cotter joints
Engine parts – piston, connecting rod, cross-head (vertical and horizontal), stuffing box, multi-plate clutch
Miscellaneous machine components – screw jack, machine vice, tail stock, chuck, vane and gear pump

COURSE OUTCOMES

Upon completion of this course, the students will be able to follow the drawing standards, fits and tolerances and re-create part drawings, sectional views and assembly drawings as per standards

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Expt. No.01
STUDY OF DRAWING STANDARDS AND FITS AND TOLERANCES

Aim:

To study about the drawing standards and fits and tolerance

1.1 Code of practice for Engineering Drawing
Abbreviations

<i>Term</i>	<i>Abbreviation</i>	<i>Term</i>	<i>Abbreviation</i>
Across corners	A/C	Maunfacture	MFG
Across flats	A/F	Material	MATL
Approved	APPD	Maximum	max.
Approximate	APPROX	Metre	m
Assembly	ASSY	Mechanical	MECH
Auxiliary	AUX	Millimetre	mm
Bearing	BRG	Minimum	min.
Centimetre	Cm	Nominal	NOM
Centres	CRS	Not to scale	NTS
Centre line	CL	Number	No.
Centre to centre	C/L	Opposite	OPP
Chamfered	CHMED	Outside diameter	OD
Checked	CHD	Pitch circle	PC
Cheese head	CH HD	Pitch circle diameter	PCD
Circular pitch	CP	Quantity	QTY
Circumference	OCE	Radius	R
Continued	CONTD	Radius in a note	RAD
Counterbore	C BORE	Reference	REF
Countersunk	CSK	Required	REQD
Cylinder	CYL	Right hand	RH
Diameter	DIA	Round	RD
Diametral pitch	DP	Screw	SCR
Dimension	DIM	Serial number	Sl. No.
Drawing	DRG	Specification	SPEC
Equi-spaced	EQUI-SP	Sphere/Spherical	SPHERE
External	EXT	Spot face	SF

Figure	FIG.	Square	SQ
General	GNL	Standard	STD
Ground level	GL	Symmetrical	SYM
Ground	GND	Thick	THK
Hexagonal	HEX	Thread	THD
Inspection	INSP	Through	THRU
Inside diameter	ID	Tolerance	TOL
Internal	INT	Typical	TYP
Left hand	LH	Undercut	U/C
Machine	M/C	Weight	WT

Standard Codes

Sl.No.	IS-CODE	DESCRIPTION
1	IS:9609-1983	Lettering on Technical Drawing
2	IS:10711-1983	Size of drawing sheets
3	IS:10713-1983	Scales for use on technical drawing
4	IS:10714-1983	General Principles of Presentation
5	IS:10715-1983	Presentation of threaded parts on technical drawing
6	IS:10716-1983	Rules for presentation of springs
7	IS:10717-1983	Conventional representation of gears on technical drawing
8	IS:11663-1986	Conventional representation of common features
9	IS:11664-1986	Folding of drawing prints
10	IS:11665-1986	Technical drawing – Title blocks
11	IS:11669-1986	General principles of dimension on technical drawing
12	IS:11670-1986	Abbreviations for use in Technical Drawing

1.2 Welding symbols

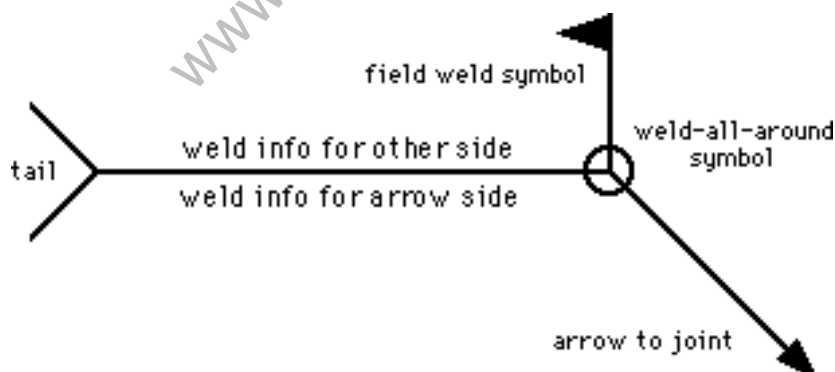
Welding is a process of fastening the metal parts together permanently by the application of heat (fusion welds) or pressure (pressure or forge welding) or both (resistance welding). Both ferrous (steel, cast iron) and Non-ferrous metals (like brass copper and alloy) can be joined by welding.

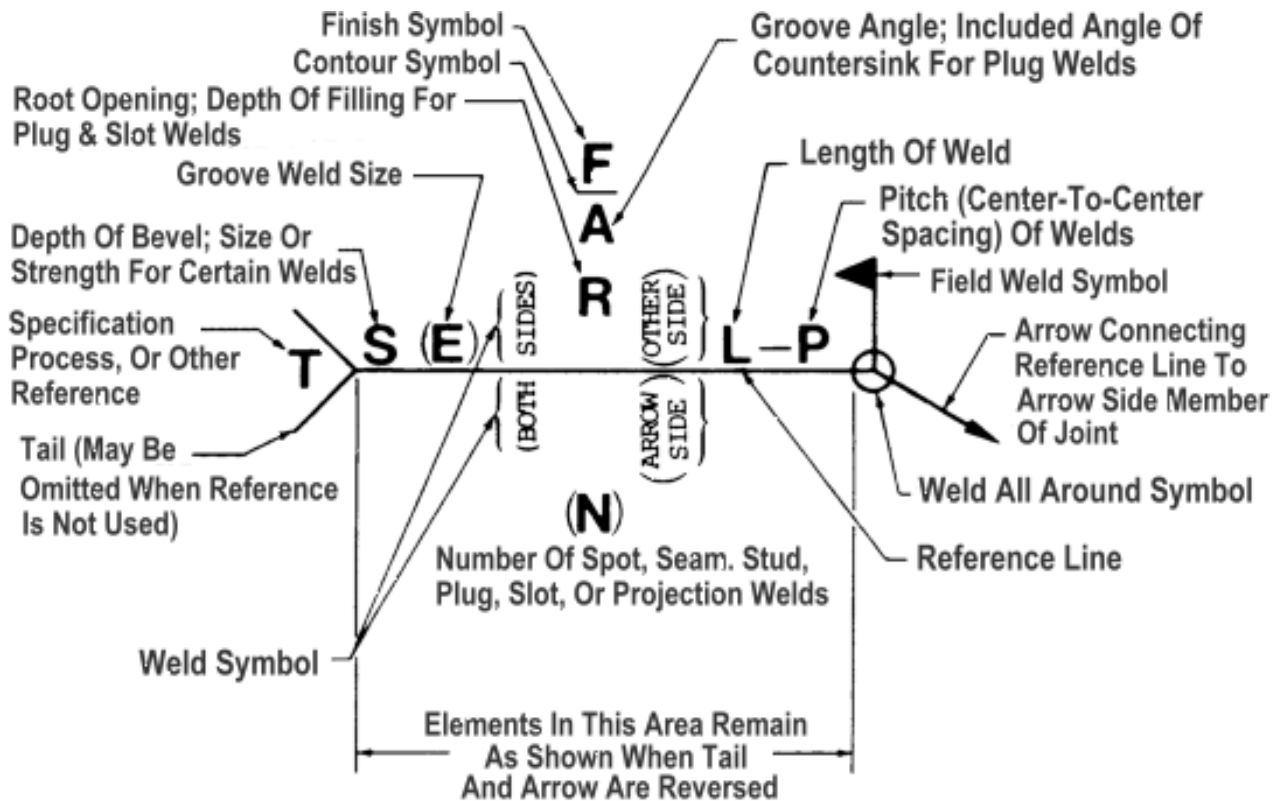
The various types of welding process are

- a. Gas welding
- b. Arc welding
 - Metal Arc Welding (MAW)
 - Gas metal Arc Welding (GMAW)
 - Submerged Arc Welding (SAW)
 - Tungsten Inert Gas Welding (TIG)
 - Metal Inert Gas Welding (MIG)
- c. Forge Welding
- d. Resistance Welding
- e. Thermit Welding
- f. High Energy Welding

A welding symbol may include the following elements:

- 1. Reference line
- 2. Arrow line
- 3. Basic weld symbol
- 4. Dimensions & other data
- 5. Supplementary symbols
- 6. Finish symbols
- 7. Tail
- 8. Specifications, process, or other references





















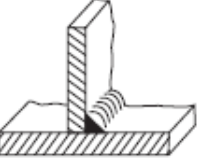



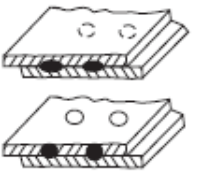

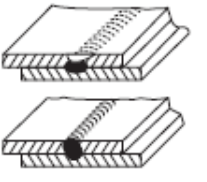



Location Of Welding Symbol Elements

The position of the arrow line with respect to the weld is of no special significance. The side of the joint on which the arrow line is drawn is called "arrow side". The side of the joint remote to the arrow line is called "other side". The reference line has significance on the weld side. If the weld symbol is placed BELOW the reference line, the welding should be done in the "ARROW SIDE". If the weld symbol is placed ABOVE the reference line, the welding should be done in the "OTHER SIDE". If the weld symbol is placed both ABOVE and BELOW the reference line, the welding should be done in both the "ARROW and OTHER SIDES".

Basic Weld Symbol

The basic symbols recommended by Bureau of Indian Standards (BIS) for specifies the type of weld are shown in the fig.

No.	Designation	Illustration	Symbol
1.	Butt weld between plates with raised edges (the raised edges being melted down completely)		
2.	Square butt weld		
3.	Single-V butt weld		
4.	Single-bevel butt weld		
5.	Single-V butt weld with broad root face		
6.	Single-bevel butt weld with broad root face		
7.	Single-U butt weld (parallel or sloping sides)		
8.	Single-U butt weld		
9.	Backing run; back or backing weld		
10.	Fillet weld		
11.	Plug weld; plug or slot weld		
12.	Spot weld		
13.	Seam weld		

1.3 Riveted Joints

A riveted joint is a permanent type of fastener used to join the metal plates or rolled steel sections together. Riveted joints are extensively used in structural works such as bridges and roof trusses and in the construction of pressure vessels such as storage tanks, boilers, etc. Although welded joints are best suited to several of these applications than the riveted joints, however, riveted joints are ideal in cases where the joints will be subjected to pronounced vibrating loads. Riveted joints are also used when a non-metallic plate and a metallic plate are to be connected together. They are also used when the joints are not expected to be heated while joining as in welding, which may cause warping and tempering of the finished surfaces of the joints.

The disadvantage of riveted joints are: (i) more metal is removed while making of the holes, which weakens the working cross sections along the line of the rivet holes, and (ii) weight of the rivets increases the weight of the riveted members.

A rivet is a round rod made either from mild steel or non-ferrous materials such as, copper, aluminium, etc., with a head is, and formed at one end during its manufacture and its tail end being slightly tapered. The length of the shank of the rivet must be sufficient enough to accommodate the connection plates and provide enough material for forming a head at its shank end. In general, the shank of the rivet will be equal to sum of the thickness of the connecting plates plus 1.5 to 1.7 times the diameter of the rivet. If,

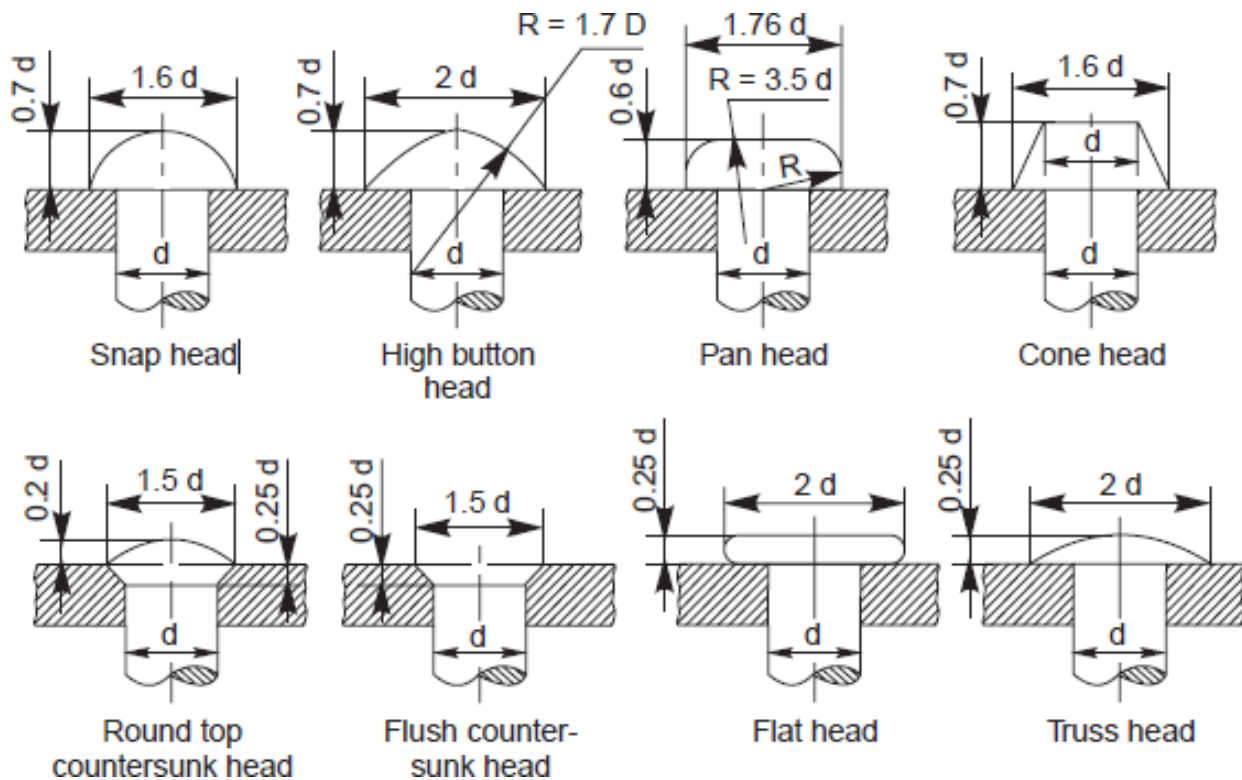
L = length of the shank of the rivet

d = diameter of rivet

t = thickness of each of the connecting plates

then, $L = \sum t + (1.5 \text{ to } 1.7)d$

Various types of rivet heads for the use in general engineering work and boiler work as recommended by the Bureau of Indian Standards. The different proportions of these rivet heads are given in terms of the nominal diameter d of the rivet. The rivet head to be used for general purposes for diameter below 12 mm are specified in the Indian Standard code IS:2155-1962 and for diameters between 12 and 48 mm are specified in the Indian Standard code IS:1929-1961. The rivet heads to be used for boiler work are specified in the Indian Standard code IS: 1928-1961. The rivet heads to be used for ship building are specified in the Indian Standard code IS: 4732-1968.



Types of rivet heads

1.4 Screw Threads, Keys and Fasteners

Screw Threads

A screw thread is a functional element used on bolt, stud, set screw, nut or any other threaded piece or component. Screw thread is a helical groove on a cylinder surface (outer side or inner side). Its function is to transform the input motion of rotation into output motion of translation.

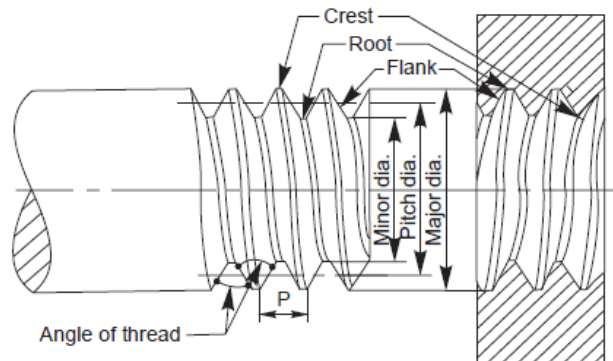
If a cylindrical rod is rotated at a constant speed simultaneously if a pointed tool touching the rod moving parallel to the axis of the rod at constant speed, the cut made by tool on the rod will be continuous and of helical form. The helical groove is called "thread" and the threaded rod is called a "screw".

Threads are cut using a lathe. Small size thread is often cut by means of a tool called die. A small size hole is threaded by means of a tool called a tap.

The principal uses of threads are,

1. for fastening
2. for adjusting
3. for transmitting power

Terms and Nomenclature



1. **Crest:** It is the peak edge of a screw thread that connects the adjacent flanks at the top.
2. **Root:** It is the bottom edge of the thread that connects the adjacent flanks at the bottom.
3. **Flank or side:** It is the straight portion of the surface, on either side of the screw thread.
4. **Angle of the thread:** It is the angle included between the sides of two adjacent threads measured on an axial plane.
5. **Depth of the thread:** It is the distance between the crest and the root measured at Right angle to the axis. It is equal to half the difference between the outer diameter and the core diameter.
6. **Major diameter or outside diameter:** It is the diameter of the imaginary coaxial cylinder, which would bind the crests of an external or internal thread.
7. **Minor or core or root diameter:** It is the diameter of the imaginary coaxial cylinder; this would bind the roots of an external thread or of an internal thread.
8. **Pitch diameter:** It is the diameter of the imaginary coaxial cylinder that can be passed so as to cut the thread, that the width of the cut thread will be equal to the Width of the groove.
9. **Pitch:** It is the axial distance between a point on one thread and the corresponding Point on the next thread. It may be indicated as the distance from crest or from root of two adjacent threads.
10. **Lead:** It is the distance measured parallel to the axis from a point on a thread to the corresponding point on the same thread for one complete revolution. In other words, it is axial distance a screw advances in one revolution.
11. **External thread:** It is the thread on the outside surface of a member such as bolt, studs or screw.
12. **Internal thread:** It is the thread on the inside surface of a member such as nut or threaded hole.
13. **Right hand thread:** Right hand thread if turned clockwise direction advances into a threaded hole. It abbreviated as RH.
14. **Left hand thread:** Left hand thread if turned anticlockwise direction advances into a threaded hole. It abbreviated as LH.

V- THREADS

British standard whit worth thread (BSW)

This thread was introduced by Sir Joseph whit worth, and was standardized as British standard thread. It has a thread angle of 55 degree and is rounded equally at crest and roots.

British Association threads (B.A Threads)

The angle between flanks is 47.5 degree. These threads are to supplement BSW and have fine pitches. They are used on screws for precision work.

American standard threads (or) sellers thread

The thread angle is 60 degree and both the crests and roots are kept flat. The sellers thread has been in use in USA and Canada.

Unified screw threads

The countries UK, U.S.A and Canada came to an arrangement for a common screw thread system with the included angle of 60 degree and designated as unified screw thread in the year 1949. The thread on the bolt is rounded off at the crest and root and the thread in the nut is rounded off at the root but the crest is left flat. This thread is very important in the motor and aeroplane industries and in chemical engineering. Unified thread can be either coarse (UNC) or fine (UNF) and unified national extra fine (UNEF).

ISO Metric Thread

This is Indian standard thread for ISO (International Standard Organization). The included angle is 60° and the crests are flat and roots are round. Metric threads are grouped into diameter pitch combination differentiated by the pitch applied to specific diameters. There are coarse, constant, fine pitch series available. ISO metric threads are defined by nominal size (Basic major diameter) and pitch both expressed in millimeters. for example, a 10mm diameter, 1.25 pitches is expressed as M10×1.25.

SQUARE THREADS

Basic square thread

The sides of these threads are normal to the axis and parallel to each other. The depth and the thickness of the thread are equal to half pitch. A square thread is designated by the letter SQ followed by nominal diameter pitch. For example a square thread of nominal diameter 30mm and pitch 6mm is designated as SQ 30×6.

Acme Thread

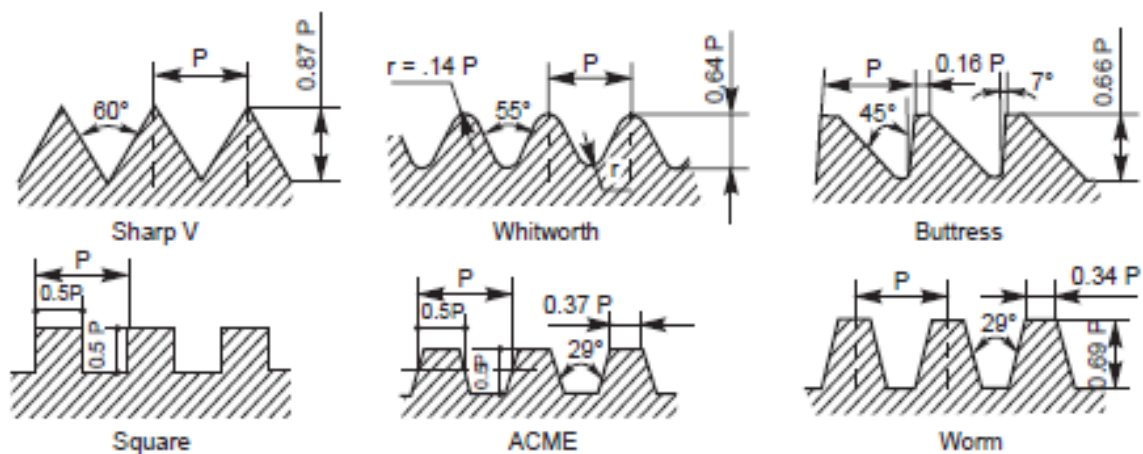
It is a modified form of square thread. It is easier to cut and is much stronger than square thread. It has a 29° thread angle. This inclined sides of the thread facilitate quick and early engagement and disengagement. It is used for power screws like lead screw of lathe, jackscrews, bench vices and valve operating screws.

Buttress Thread

The profile of this thread is a combination of square and V- threads. It combines the low frictional of square and ability to transmit power of square thread and the strength of V – thread. It is used to transmit load in uni-direction. These threads are used in screw press, vices.

Knuckle Thread

It is also a modification of square thread. The sharp corners of square thread are rounded off. This thread is used where heavy wear rough use is expected. The thread can be rolled or cast easily. It is used in railway carriage coupling screws, light bulbs and sockets, bottle caps etc and also objects made of brittle materials as glass, plastic, porcelains etc.



Types of thread profiles

Keys

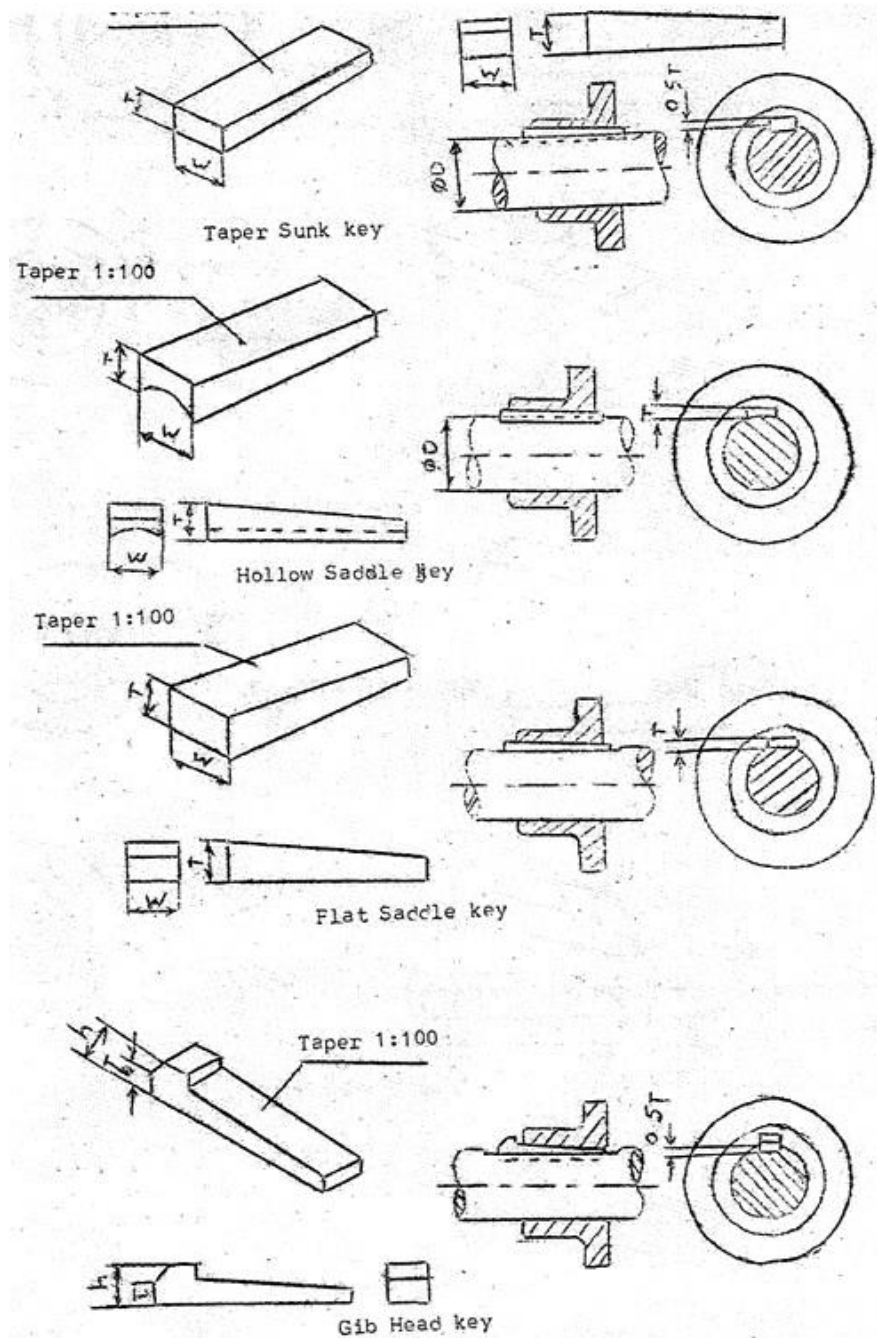
Keys are machine elements used to prevent relative rotational movement between a shaft and the parts mounted on it, such as pulleys, gears, wheels, couplings, etc. for making the joint, keyways are cut on the surface of the shaft and in the hub of the part to be mounted. After positioning the part on the shaft such that, both the keyways are properly aligned, the key is driven from the end, resulting in a firm joint.

For mounting a part at any intermediate location on the shaft, first the key is firmly placed in the keyways of the shaft and then the part to be mounted is slid from one end of the shaft, till it is fully engaged with the key.

Keys are classified into three types.

1. Saddle keys
 - I. flat saddle key
 - II. hollow saddle key

2. Sunk keys
 - I. taper sunk keys
 - II. parallel sunk keys
 - III. woodruff keys
3. Round keys



Types of keys

Bolt and nuts

A bolt and nut in combination is a fastening device used to hold two parts together. The body of the bolt, called shank is cylindrical in form, the head; square or hexagonal in shape, is formed by forging. Screw threads are cut on the other end of the shank. Nuts in general are square or hexagonal in shape. The nuts with internal threads engage with the corresponding size of the external threads of the bolt. However, there are other forms of nuts used to suit specific requirements.

For nuts, hexagonal shape is preferred to the square one, as it is easy to tighten even in a limited space. This is because, with only one-sixth of a turn, the spanner can be reintroduced in the same position. However, square nuts are used when frequent loosening and tightening is required. The sharp corners on the head of bolts and nuts are removed by chamfering.

Empirical portions of hexagonal and square head bolt & nut detail proportion

Nominal diameter d = size of bolt or nut, mm

Width across flats $s = 1.5d + 3$ mm

Width across corners $e = 2d$

Thickness of bolt head $k = 0.8d$

Thickness of nut, $n = 0.9d$

Root diameter, $d_1 = d - (2 \times \text{depth of thread})$

Length of the bolt L = as specified

Thread length $b = 2d + 6\text{mm}$ (for $l < 150\text{mm}$)
 $= 2d + 12\text{mm}$ (for $l > 150\text{mm}$)

Chamfer of bolt end, $Z = (\text{depth of thread} \times 45^\circ)$ or $= 0.1d$

Chamfer angle of bolt head & nut 30°

1.5 Limits, fits and tolerances

Limit System

Following are some of the terms used in the limit system:

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.

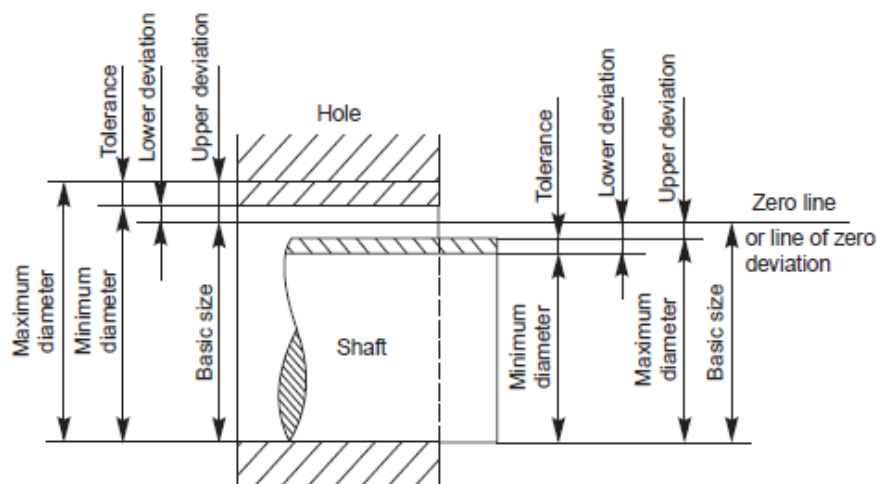
Actual deviation: It is the algebraic difference between the actual size and the corresponding basic size.

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. Here, the two limit dimensions of the shaft are deviating in the negative direction with respect to the basic size and those of the hole in the positive direction. The line corresponding to the basic size is called the zero line or line of zero deviation.



Fits

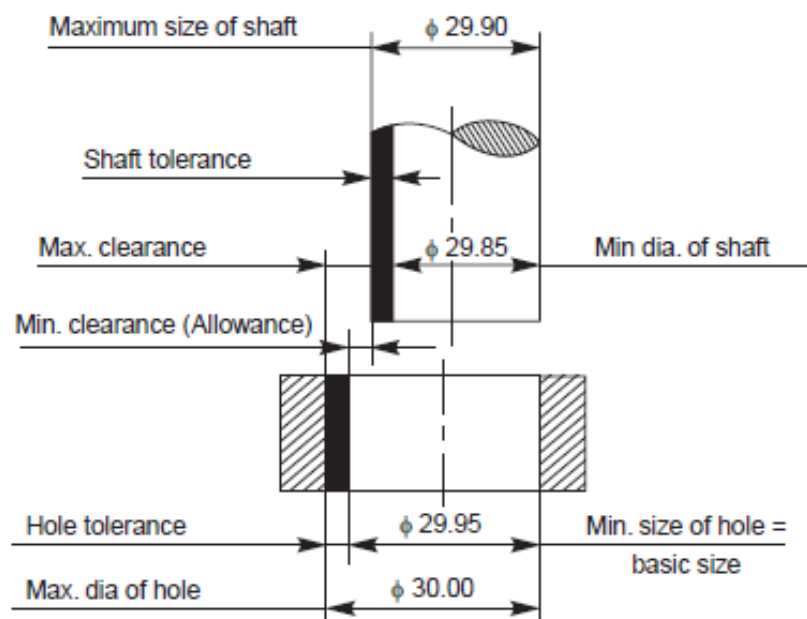
The relation between two mating parts is known as a fit. Depending upon the actual limits of the hole or shaft sizes, fits may be classified as clearance fit, transition fit and interference fit.

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

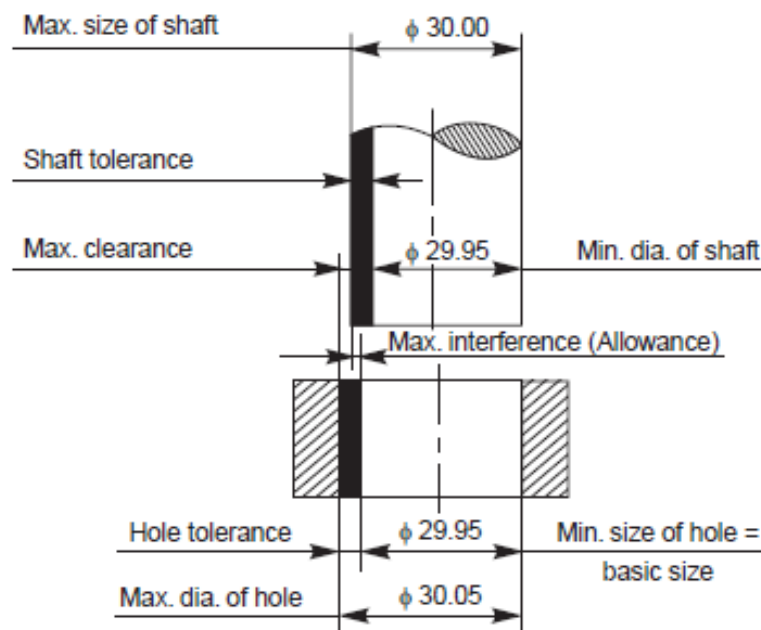
minimum clearance: It is the difference between the minimum size of the hole and the maximum size of the shaft in a clearance fit.

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.

The fit between the shaft and hole in Fig. is a clearance fit that permits a minimum clearance (allowance) value of $29.95 - 29.90 = + 0.05$ mm and a maximum clearance of $+ 0.15$ mm.



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).



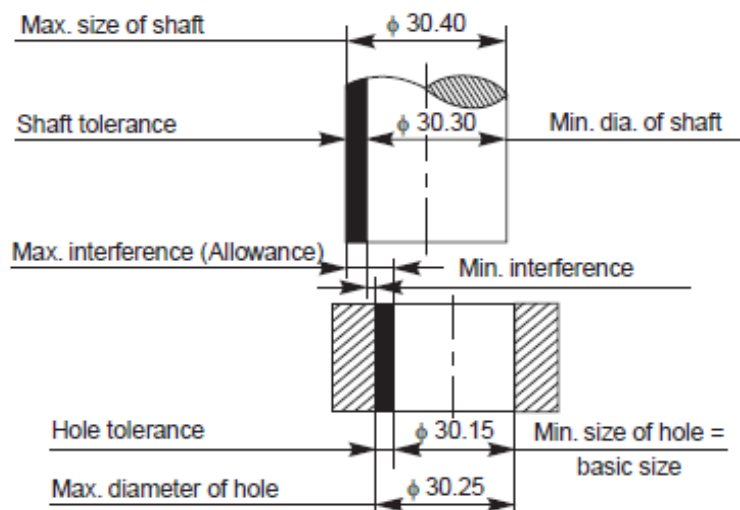
Interference fit: If the difference between the hole and shaft sizes is negative before assembly; an interference fit is obtained.

minimum interference: It is the magnitude of the difference (negative) between the maximum size of the hole and the minimum size of the shaft in an interference fit before assembly.

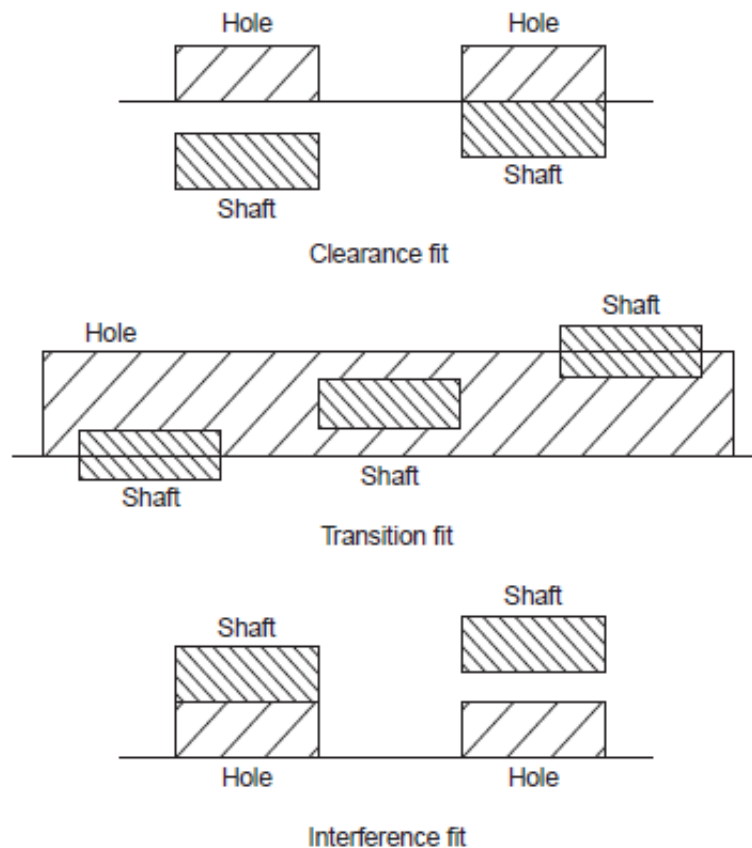
maximum interference: It is the magnitude of the difference between the minimum size of the hole and the maximum size of the shaft in an interference or a transition fit before assembly.

The shaft in fig. is larger than the hole, so it requires a press fit, which has an effect similar to welding of two parts.

The value of minimum interference is $30.25 - 30.30 = -0.05$ mm and maximum interference is $30.15 - 30.40 = -0.25$ mm.



Schematic representation of fits:



Tolerance

It is impossible to make anything to an exact size, therefore it is essential to allow a definite tolerance or permissible variation on every specified dimension.

Tolerances are specified because

- Variations in properties of the material being machined introduce errors.
- The production machines themselves may have some inherent inaccuracies.
- It is impossible for an operator to make perfect settings. While setting up the tools and workpiece on the machine, some errors are likely to creep in.

Form variation: It is a variation of the actual condition of a form feature (surface, line) from geometrically ideal form.

Position variation: It is a variation of the actual position of the form feature from the geometrically ideal position, with reference to another form (datum) feature.

Geometrical tolerance: Geometrical tolerance is defined as the maximum permissible overall variation of form or position of a feature.

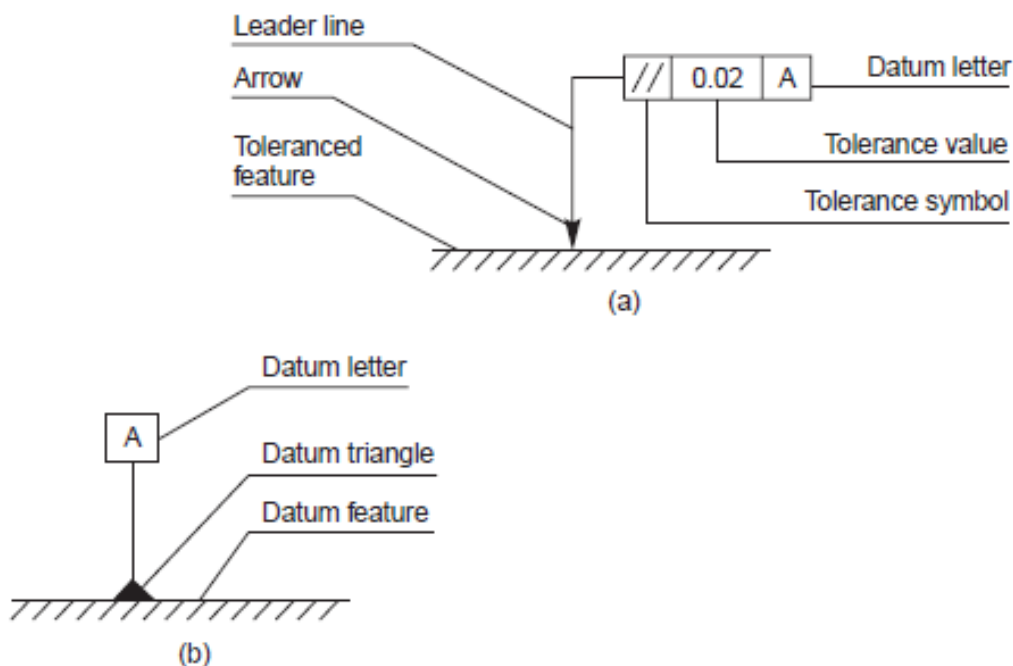
Geometrical tolerances are used,

- (i) to specify the required accuracy in controlling the form of a feature,
- (ii) to ensure correct functional positioning of the feature,
- (iii) to ensure the interchangeability of components, and
- (iv) to facilitate the assembly of mating components.

Tolerance zone: It is an imaginary area or volume within which the controlled feature of the manufactured component must be completely contained














Datum: It is a theoretically exact geometric reference (such as axes, planes, straight lines, etc.) to which the tolerance features are related.

Datum feature: A datum feature is a feature of a part, such as an edge, surface, or a hole, which forms the basis for a datum or is used to establish its location



Datum triangle: The datums are indicated by a leader line, terminating in a filled or an open triangle.

Datum letter: To identify a datum for reference purposes, a capital letter is enclosed in a frame, connected to the datum triangle.

<i>Characteristics to be toleranced</i>		<i>Symbols</i>
Form of single features	Straightness	
	Flatness	
	Circularity (roundness)	
	Cylindricity	
	Profile of any line	
	Profile of any surface	
Orientation of related features	Parallelism	
	Perpendicularity (squareness)	
	Angularity	
Position of related features	Position	
	Concentricity and coaxiality	
	Symmetry	
	Run-out	

Expt. No.02**INTRODUCTION TO CADD**

CADD is an electronic tool that enables us to make quick and accurate drawings. CADD has number of advantages over drawings created on a drawing board. Electronic drawings can be modified quite easily and can be represented in a variety of formats.

CADD extends its power to yet another branch of engineering called computer aided manufacturing (CAM). CADD and manufacturing program are often integrated into one system called CAD-CAM. This system import CADD drawings into CAM program to automate the manufacturing process. When the design is finalized, the drawings are brought into a CAD-CAM system that uses numerical data from the CADD drawing for actual manufacturing.

There is separate category of programs called Computer Aided Engineering (CAE) that can use CADD drawing for engineering analysis. The CAE programs have a number of applications in Structural Design, Civil Engineering, Mechanical Engineering and Electrical Engineering. The Mechanical engineer can test a machine assembly and also a prototype electronic model and test it without building a physical model.

We can do amazing things with CADD that we never thought possible while creating drawings with pen or pencil.

The following are some of the important capabilities that make CADD a powerful tool.

- Presentations
- Flexibility in editing
- Unit and accuracy levels
- Storage and access for drawings
- Sharing CADD drawings

Presentations

There are a number of ready-made presentations symbols available in CADD that can be used to enhance the look of drawings. In addition to prepare impressive presentations on paper, we can use CADD to make an on-screen presentations. Advanced CADD programs ever allow us to create an animated image.

Flexibility in editing

CADD allows us to work with great accuracy. If we need to create highly accuracy geometric shapes, CADD is the answer. It can help avoid time-consuming mathematical calculations.

Unit and accuracy level

We can work with as high precession as $1/1000^{\text{th}}$ of an inch.

Storage and access of drawing

A computer electronic filing system has the following advantages over the traditional filing system.

- It is quick and convenient to organize CADD drawing in a computer.
- It enables us to create a highly organized environment.
- An electronic drawing never gets old and faded.

Sharing CADD Drawing

The electronic drawing can be shared by a number of users, allowing them to Co-ordinate projects and work as a team. This is accomplished by connecting different computer via a network.

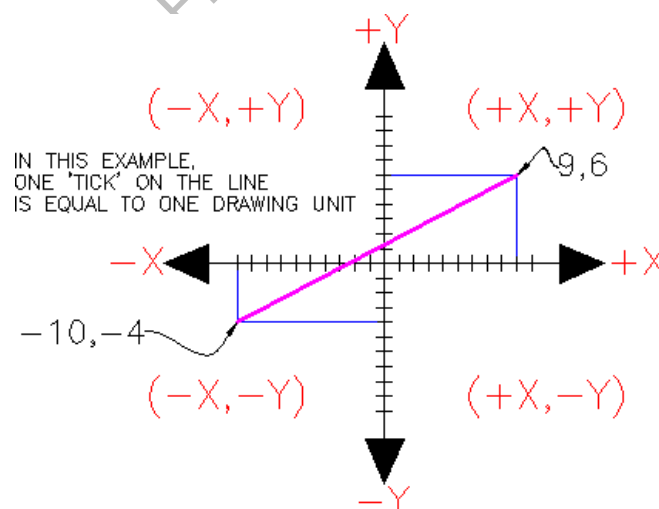
About AutoCAD

AutoCAD is a Computer Aided Design (CAD) program used by just about every Engineering and Design office in the world. Although there are alternative CAD packages, AutoCAD is by far the most widely used system. Autodesk's AutoCAD is the industry leader in CAD packages. Used by Civil Engineers, Architects, Mechanical and Electrical Engineers, Aeronautical Engineers plus many other disciplines.

There have been several versions of AutoCAD over the years, with each new version introducing new and more powerful features than its predecessor. The latest version of AutoCAD (at the time of writing) is AutoCAD 2011. Any courses, whether through community colleges or online universities, that are related to Engineering or Architecture should be considered incomplete if they do not introduce students to AutoCAD.

Accurate, scale drawings can be created and published using AutoCAD powerful features. 3D 'models' can also be created giving the designer absolute control over the design from start to finish. The computerized model can be viewed through a 360° angle, and even 'rendered' with a texture on screen to give an idea of the finished product.

Co-ordinate system



AutoCAD uses points to determine where an object is located. There is an origin where it begins counting from. This point is (0,0). Every object is located in relation to the origin. If we were to draw a line straight out to the right from

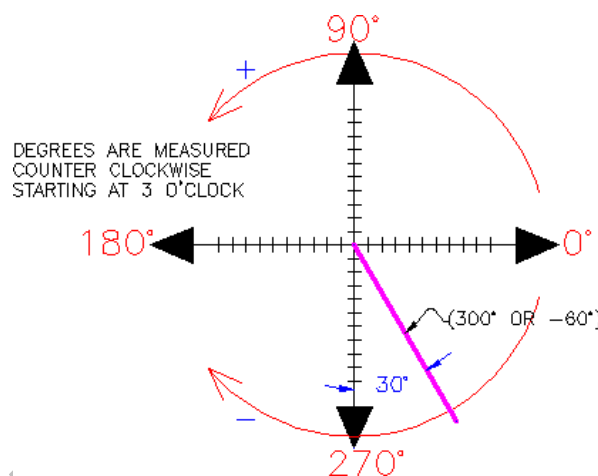
the origin, this would be considered the positive X-axis. If we were to draw a line straight up, this would be the positive Y-axis. The picture above shows a point located at (9, 6). This means that the point is 9 units over in the X-axis and 6 units up in the Y-axis. When we are working with points, X always comes first. The other point shown is (-10,-4). This means that the point is 10 units in the negative X-axis (left) and 4 units in the negative Y-axis (down).

A line has two points, a start point and an end point. AutoCAD works with the points to display the line on the screen. Most of the time we will not have an indication of where the origin is. We may need to draw a line from the endpoint of an existing line. To do this we use relative points. These work the same way, but we have to add the @ symbol (shift+2) to tell AutoCAD that this next point is relative from the last point entered. i.e.

1. ABSOLUTE POINTS are exact points on the drawing space.
2. RELATIVE POINTS are relative to an OBJECT on the drawing space

Angular Measurement

AutoCAD measures angles in a particular way also.



When drawing lines at an angle, we have to begin measuring the angle from 0 degrees, which is at the 3 o'clock position. If we drew a line at 90 degrees, it would go straight up. The example shows a line drawn at +300 degrees (270+30), or -60 degrees.

Entering Points in AutoCAD

We can enter points directly on the command line using three different systems. The one we use will depend on which is more applicable for the situation. The three systems are as follows:

ABSOLUTE CO-ORDINATES - Using this method, we enter the points as they relate to the origin of the WCS. To enter a point just enters in the exact point as X, Y.

RELATIVE CO-ORDINATES - This allows us to enter points in relation to the first point we have entered. After we've entered one point, the next would be entered as @X, Y. This means that AutoCAD will draw a line from the first point to another point X units over and Y units up relative to the previous point.

POLAR CO-ORDINATES - We would use this system if we know that we want to draw a line a certain distance at a particular angle. We would enter this as @D<A. In this case, D is the distance and A is the angle. Example: @10<90 will draw a line 10 units straight up from the first point.

The three ways of entering co-ordinates shown above are the ONLY way AutoCAD accepts input. First decide which style we need to use, and then enter as shown. Remember that X is always before Y (alphabetical). Don't forget the '@' symbol when we are entering relative points.

AutoCAD Basics

Application Button - This button displays commands for printing, saving, drawing utilities and other non-drawing tool.

Quick Access Toolbar - This is for quick access to common commands like New, Open, Save, Plot

Filename - The name of the current file we are working on.

Search Bar - Search for text in were drawing or search the help files.

Ribbon - The Ribbon has most of the commands/tools that we will use while we are working.

Tabs - A series of Tabs make up the Ribbon (Home, Insert, Manage, etc) and organize the Tools into common groups.

Panels - Contain a group of tools

Tools - These are the icon that starts the commands we use to draw, modify, etc.

Tool Tip - If we however mouse over a tool, a tool tip will appear to give us more information. Hold it longer for more info.

Drawing Space - These is where we draw were designs.

Command line - When we type a command, we will see it here. AutoCAD uses this space to 'prompt' us for information. It will give us a lot of information and tell us where we are in the command.

Status bar - This allows seeing and changing different modes of drawing such as Ortho, Osnaps, Grid, Otrack, etc. We can right click this area to toggle between icons and text for this area.

Basic Drawing Commands for Autocad

Measuring Commands

GRID (F7): Displays a grid of dots at a desired spacing on the screen.

Command: GRID (enter)

On/Off/Tick spacing(x)/Aspect: (enter value) (enter)

SNAP (F9): Specifies a "round off" interval so that points entered with the mouse can be locked into alignment with the grid spacing.

Command: SNAP (enter)

On/Off/Value/Aspect/Rotate/Style: (enter value) (enter)

Basic Draw Commands

CIRCLE (C): Draws circles of any size.

Command: Circle (enter)

3P/2P/TTR/<center point>: (pick a center point)

Diameter or <Radius>: (Pick a point on the circle)

LINE (L): Draws straight lines between two points

Command: LINE (enter)

From Point: (pick a point using the mouse) To Point: (Pick a point using the mouse)

To Point: (Press return to end the command)

ARC (A): Draws an arc (any part of a circle or curve) through three known points.

Command: ARC (enter)

Center/ < Start point > : (pick the first point on the arc)

Center/End/ < Second point >:C

Center: (pick the arc's center point)

Angle/Length of chord/ <End point >: (pick the arc endpoint)

Display Commands

LIMITS: Sets the size of the drawing paper. For size "A" drawing paper the limits should be set for 10.5 x 8.

Command: LIMITS (enter)

On/Off/Lower left corner <0.0000> (enter)

Upper right corner: 10.5,8 (enter)

ZOOM (Z): Enlarges or reduces the display of a drawing.

PAN: Allows we to move were view point around the drawing without changing the magnification factor.

Command: PAN (enter)

Editing Commands

CHANGE: Alters properties of selected objects

Command: CHANGE (enter)

Select objects or window or Last (select objects to be changed)

Properties/<Change point>: (type P)

Change what property (Color/Elev/ Layer/L Type/Thickness)? (Type Layer) New Layer:

(enter new layer name and press enter)

ERASE (E): Erases entities from the drawing.

Command: ERASE (enter)

Select objects or Window or Last: (Select objects to be erased and press enter when finished)

TRIM (TR): Trims a line to end precisely at a cutting edge.

Command: Trim (enter)

Select cutting edge(s)... Select Objects (pick the line which represents the cutting edge of line in which objects will be trimmed to)(press enter when finished selecting cutting edges)<Select object to trim>/Undo: (pick the line(s) that need to be trimmed).

Creating Layers

LAYER: Creates named drawing layers and assigns color and line type properties to those layers.

Command: LAYER (enter)

A Layer & Line type Properties dialog box will be displayed. To add a new layer, pick the new button. A new layer listing appears, using a default name of Layer1. The layer name can be changed by highlighting the layer name. Colors and Line types can be assigned to each new layer by picking the color box to assign a color and picking the line type box to assign a line type.

Standard AutoCAD colors

1 = Red	2 = Yellow	3 = Green	4 = Cyan
5 = Blue	6 = Magenta	7 = White	

Standard AutoCAD line types

Hidden2 = hidden lines

Center2 = center lines

Phantom2 = phantom or cutting-plane lines

Construction Commands

ARRAY (AR): Makes multiple copies of selected objects in a rectangular or circular pattern

Command: ARRAY (enter)

Select objects or Window or Last: (select object to array)

Rectangular or Polar array (R/P) <current>: (P)

Center point of array: (pick the point around which to form the array)

Angle to fill (+=CCW, -=CW) <360>: (enter)

COPY (CO): Draws a copy of selected objects.

Command: COPY (enter)

Select objects or Window or Last: (select objects to be copied)

Base point or displacement: (pick a point on the object to be use as a reference point)

Second point of displacement: (pick a point which represents the new location of the copied object)

MIRROR(MI): Makes mirror images of existing objects.

Command: MIRROR (enter)

Select objects or Window or Last: (select objects to be mirrored)

First point of mirror line: (pick a point on top of the mirror line)

Second point: (pick a point on the bottom of the mirror line) Delete old objects? <N> y or n (enter)

MOVE(M): Moves designated entities to another location.

Command: MOVE (enter)

Select objects or Window or Last: (select objects to move)

Base point or displacement: (pick a point on the object to be use as a reference point)

Second point of displacement: (pick a point which represents the new location of the object)

OFFSET (O): Constructs an entity parallel to another entity at a specified distance. Offset can be used with lines, circles, arcs, and polylines.

Command: OFFSET (enter)

Offset distance or Through<last>: (enter a distance value)

Select object to offset: (select object to offset)

Side to offset: (Pick any point on the side of the object we wish to offset)

FILLET (F): Changes any corner to a rounded corner.

Command: FILLET

Polyline/Radius/Angle/Trim/Method/ <Select first line >: (pick the first line) S

elect second line: (pick the second line)

CHAMFER (CHA): Changes any corner to an angled corner.

Command: CHAMFER

Polyline/Distance/Angle/Trim/Method/ < Select first line > : (pick the first line)

Select second line: (pick the second line)

OSNAP (F3): Instantly locates exact points relative to existing objects (points).

Object Snap Modes: Endpoint, Midpoint, Center, Quadrant, Intersection, Insertion, Perpendicular, Tangent, Nearest, Node, and None.

Placing lettering on a drawing

TEXT (M TEXT): Draws text characters of any size.

Command: TEXT (enter)

Justify/Style/<Start point>: (pick a starting point or enter a justification letter)

Height (0) (enter the height of the lettering)

Rotation Angle (0) (enter)

Text: (enter the desired lettering) (enter)

Summary of Options

<Start Point> Left-Justifies text along its baseline

Justify Justifies text according to the alignment options

Style Enters a new text style

Null reply Enters a new line of text below the previous text.

Alignment	Abbreviation	Orientation
Aligned	A	Aligns text between two points. Text height will adjust automatically
Fit	F	Fits text between two points. Text height will not change
Centered	C	Centers text at the baseline of a specified point
Middle	M	Centers text horizontally and vertically at the baseline of a specified point.
Right	R	Justify text at the baseline of a specified point
Top Left	TL	Left Justifies text at the top of text
Top Center	TC	Centers text at the top of text
Top Right	TR	Right justifies text at the top to text
Middle Left	ML	Left justifies text at the middle of text
Middle Center	MC	Centers text both horizontally and vertically at the middle of the text
Middle Right	MR	Right justifies text at the middle of text
Bottom Left	BL	Left justifies text at the bottom of text
Bottom Center	BC	Centers text a the bottom of text
Bottom Right	BR	Right justifies text at the bottom of text

Crosshatching a drawing

BHATCH: Allows the user to crosshatch areas of a section view.

Command: BHATCH (enter)

The Boundary Hatch dialogue Box will be displayed. Select the Hatch Options box. The

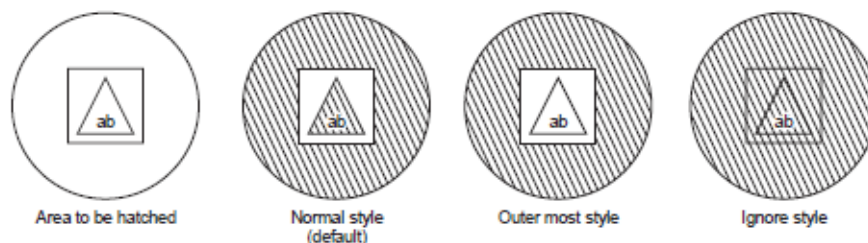
Hatch Options box will be displayed. Select the Patterns box.

The Choose Hatch Pattern box will be displayed. Select the desired hatch pattern.

The Hatch Options box will be displayed again. We can select a scale and rotation angle for the crosshatch pattern. Select the OK box when finished.

The Boundary Hatch dialogue box will be displayed again. Select the Pick Points box. When prompted select the internal point of the area to be crosshatched. Press the enter key when finished.

The Boundary Hatch dialog box will be displayed again. Select the Apply box to add the crosshatching to the drawing.



Expt. No.03**INTRODUCTION TO MODELLING SOFTWARE PRO-E****Introduction:**

Pro-E is a high end cad package. Pro-E is a parametric solid model. It was developed by parametric technology corporation. It is suite of programs that are used in design analysis and manufacturing of a virtually unlimited range of products.

Feature based:

Pro-E is a feature based solid modeling tool. It builds the model using individual building blocks one at a time. Designers can think at a high level and leave all the low level geometry details, for Pro-E to figure out.

Associative:

We can use Pro-E to document model by creating parts, assemblies and drawing all these functions are really associative.

Parametric:

Parametric means that the physical shape of the part is driven by the values assigned to the attributes of its features.

Solid modeling:

Solid modeling means that the computer model we create is able to contain all the information that a real solid object would have the material.

Parent - child Relationship:

An important aspect to feature based modeling in Pro-E is the concept of modeling in pro-E parent - child relationship.

Datum plane:

A datum plane representation of an infinite large power surface area of a user defined orientation and location that are continuously adjusted automatically to be slightly larger than the object.

Plane-Insert-Datum-Plane:

Default method creates three mutually perpendicular datum planes intersecting at the origin at the default coordinate system. These three planes represents XY, YZ & ZX planes are named as follows:

DTM1 → represents YZ plane

DTM2 → represents ZX plane

DTM3 → represents XY plane

Datum axis:

It can be used as a reference for features creation. It is particularly useful for making datum planes placing items concentrically and creating radial patterns.

Part modeling:

Pro-E is a feature based modeling software. Features in Pro-E are of two kinds. Place features extrude, revolve, sweep and blend are examples of sketched features.

Sketches features:

Protrusion features:

Protrusion is the method of adding a solid material of the method. Pro-E provides the following basic methods of adding material to a method.

Extrude: creates a solid feature by extruding a section named to the section.

Revolve: creates a solid surface by revolving a section about an axis.

Sweep: creates a solid feature by sweeping a section about a path.

Blend: creates a solid feature by blending various cross sections at various levels.

Extrusion:

Features → create → solid → protrusion → extrude

Attribute types:

One side: Adds the material in one side of the cross section only.

Both sides: Adds the material on both sides of the cross section.

Defining sketching plane: We have to specify the sketch plane to draw the cross section.

Direction of feature creation:

Once the direction of extrusion is defined Pro-E will promote for orienting the sketch plane. Orientation plane can lie horizontally or vertically in the section view sketch plane once the orientation plane is defined, tools needed for sketching the cross section will be provided by Pro-E.

Sketch reference:

Pro-E automatically selects two mutually perpendicular lines which are normal to the sketching plane as horizontal and vertical reference for dimensioning and constraining the sketch.

Extrusion depth:

Blind: Enter a dimension for the feature depth. We can then control the feature depth by changing the depth dimension

Through next: Terminate the features at the next part surface

Through all: The new surface intersects all surfaces

Rib: Feature → create → solid → rib

A rib is a special type of protrusion designed to create a thin wall or web to support two surface. All straight holes are created with constant diameter. A hole from feature always removes material from our model.

Round:

Features → create → solid → round

It creates smooth transition with circular or profile between adjacent surface. An edge round smoothness the hard edges between adjacent surface.

Sweep:

Feature → create → solid → protrusion → sweep

A sweep is creating by defining two sections. The section is the trajectory and the second cross section.

Sweep can be used on for protrusion cut and slots.

Blend:

Feature → create → solid → protrusion → blend

Blending is the method of defining a volume by connecting series of at least two planar sections together, at their edges with transitional surface.

Assembly creation:

Assembly mode in Pro-E enable us to place component parts and use assemblies together to form assemblies as well as to design parts based on how they should fit together.

Mate:

Selected surface point in opposite direction and become coplanar.

Mate offset:

Selected surface point in opposite direction and are offset by specific value.

Align:

Selected surface point in opposite direction and become coplanar axis may be co-axial with the align command.

Align offset:

Selected surface point in opposite direction and are offset by a specific value which is to be modified to provide design flexibility.

Orient:

Make the two surface of revolution co-axial.

Expt. No.04 2D DRAFTING OF PLUMMER BLOCK BEARING

Aim:

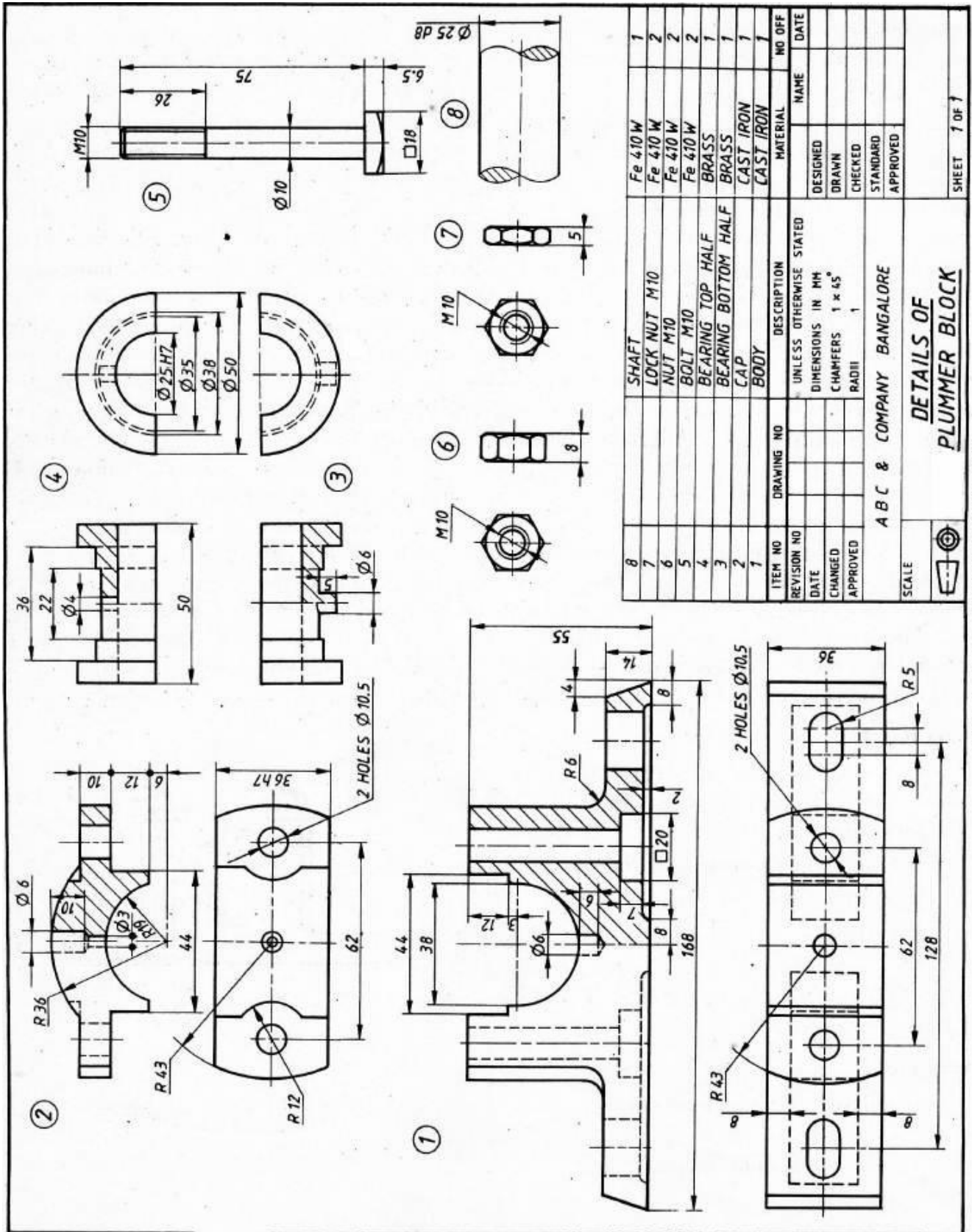
To create a 2D model of plummer block bearing by using standard CAD software

System configuration:

Ram	: 8 GB
Processor	: Core 2 Quad / Core 2 Duo
Operating system	: Windows 7
Software	: Auto CAD

Procedure:

1. Set the units to mm.
2. Using LIMITS Command set the limits to 420,297.
3. Using ZOOM Command fit the drawing to window.
4. Draw the part using LINE command and extrude it using REGION, EXTRUDE commands.
5. Draw the front view, top view and side view using drawing commands.
6. Detail the drawing as per dimensions.



DETAILS OF A PLUMMER BLOCK

Result:

Thus the 2D model of plummer block bearing is created by using standard CAD software

Outcome:

From this experiment, students will be able to create the detailed drawings which is used in automobile.

Application:

- 1.R & D
- 2.Automobile

Viva-voce

1. What is meant by Auto CAD?
2. What is the difference between CAD and CADD?
3. What are the applications of CAD?
4. Define – Absolute Coordinate System
5. Define – Polar Coordinate System
6. What is a hatch pattern?
7. What is the advantage of plummer block bearing over simple bushed bearing?

Expt. No.05 2D DRAFTING OF NON-RETURN VALVE

Aim:

To create a 2D model of non-return valve by using standard CAD software

System configuration:

Ram	: 8 GB
Processor	: Core 2 Quad / Core 2 Duo
Operating system	: Windows 7
Software	: Auto CAD

Procedure:

1. Set the units to mm.
2. Using LIMITS Command set the limits to 420,297.
3. Using ZOOM Command fit the drawing to window.
4. Draw the part using LINE command and extrude it using REGION, EXTRUDE commands.
5. Draw the front view, top view and side view using drawing commands.
6. Detail the drawing as per dimensions.



No.	Name	Matl	Qty
1	Body	Brass	1
2	Cover	Brass	1
3	Valve seat	Bronze	1
4	Valve	Brass	1
5	Stud with nut	MS	6
6	Set screw	MS	1

www.FirstRanker.com

Result:

Thus the 2D model of non-return valve is created by using standard CAD software

Outcome:

From this experiment, students will be able to create the detailed drawings which is used in pumps.

Application:

1. Pumps
2. Industrial processes

Viva-voce

1. What are the advantages of CAD?
2. What is a non return valve?
3. Name the components of non return valve.
4. Write the applications of non return valve.
5. Name the editing commands

Expt. No.06 2D DRAFTING OF SAFETY VALVE**Aim:**

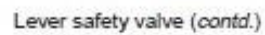
To create a 2D model of safety valve by using standard CAD software

System configuration:

Ram	: 8 GB
Processor	: Core 2 Quad / Core 2 Duo
Operating system	: Windows 7
Software	: Auto CAD

Procedure:

1. Set the units to mm.
2. Using LIMITS Command set the limits to 420,297.
3. Using ZOOM Command fit the drawing to window.
4. Draw the part using LINE command and extrude it using REGION, EXTRUDE commands.
5. Draw the front view, top view and side view using drawing commands.
6. Detail the drawing as per dimensions.



Result:

Thus the 2D model of safety valve is created by using standard CAD software

Outcome:

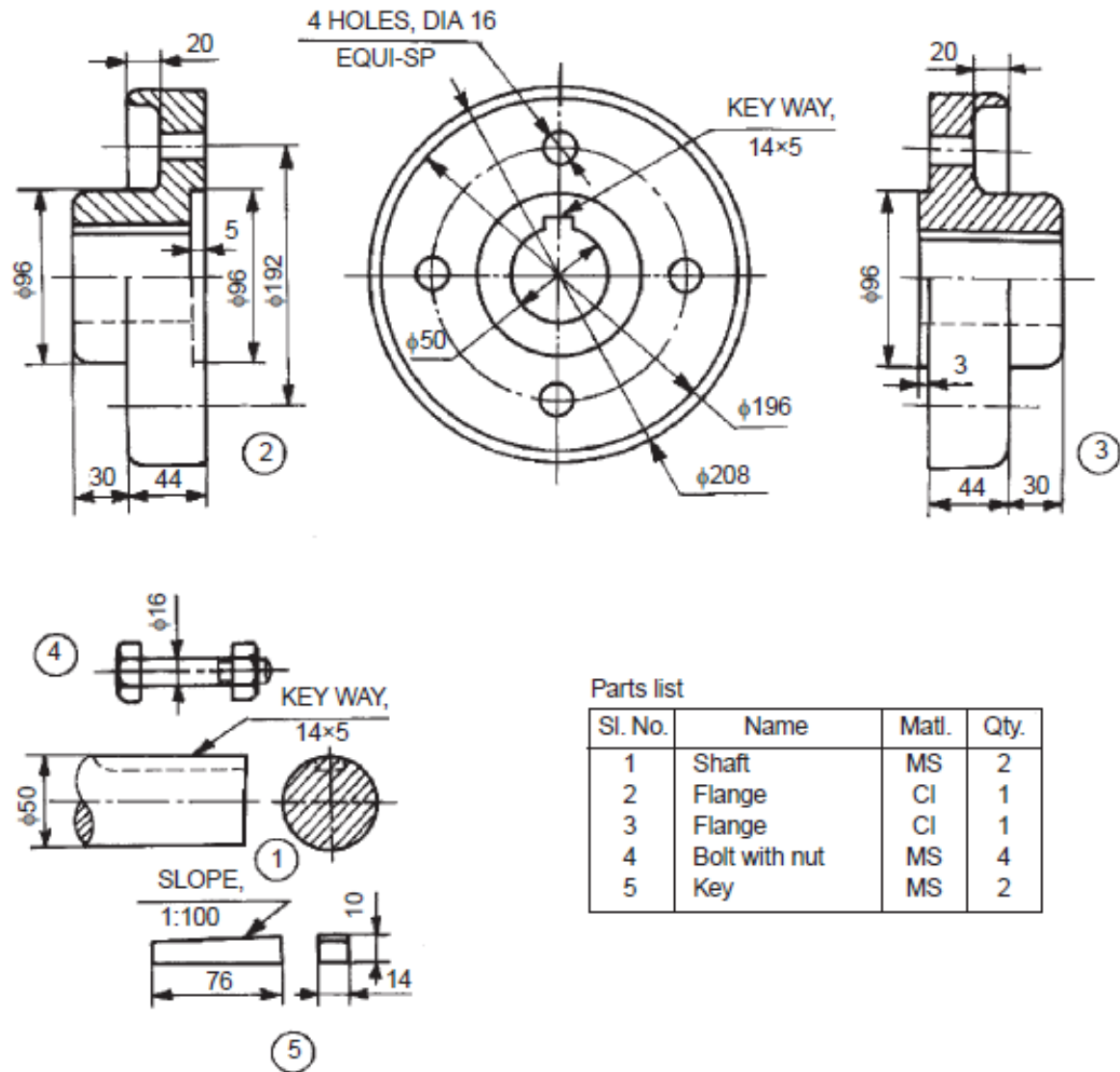
From this experiment, students will be able to create the detailed drawings which is used in boilers.

Application:

1. R & D
2. Boilers

Viva-voce

1. What is safety valve?
2. Name the types of safety valves.
3. Name the components of safety valve.
4. Write the applications of safety valve.
5. Name the utility commands
6. Name the types of modeling in CAD.



Details of Flange Coupling

Result:

Thus the given various part of flange coupling had been created and assembled by using standard CAD software Pro-E.

Outcome:

From this experiment. Students will be able to assemble the given flange coupling using standard CAD software.

Application:

1. Automobile
2. Transmission systems

Viva-voce

1. Name the components used in flange coupling.
2. What is the function of flange coupling?
3. What are the materials used for making flange coupling?
4. Define – Coupling
5. Name the different types of couplings.

Expt. No.08**3D ASSEMBLY OF UNIVERSAL COUPLING****Aim:**

To create various parts of universal coupling and assemble them by using standard CAD software Pro-E

System configuration:

Ram	: 8 GB
Processor	: Core 2 Quad / Core 2 Duo
Operating system	: Windows 7
Software	: Pro-E Wildfire 4.0

Description:

This is a rigid coupling and is used to connect two shafts, whose axes intersect if extended. The given figure shows the details of universal coupling. The forks 2 are mounted at the ends of two shafts 1, making use of sunk keys 6. The central block 3, having two arms at right angle to each other, is placed between the forks and connected to both of them by using pins 4 and collars 5. A taper pin is used to keep the pins 4 in position. During rotation of shafts, the angle between them can be varied.

Commands used:**Part diagram:**

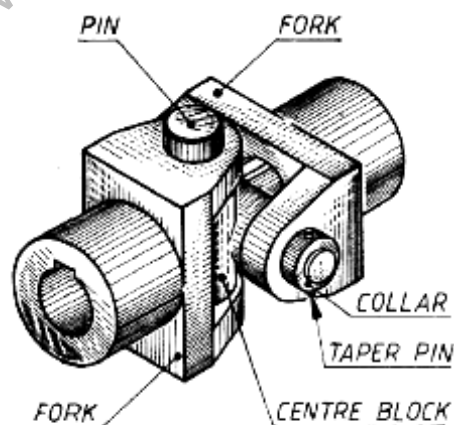
Extrude → Sketch → Sketch to dimension → Ok

Hole → Sketch → given dimension → Ok

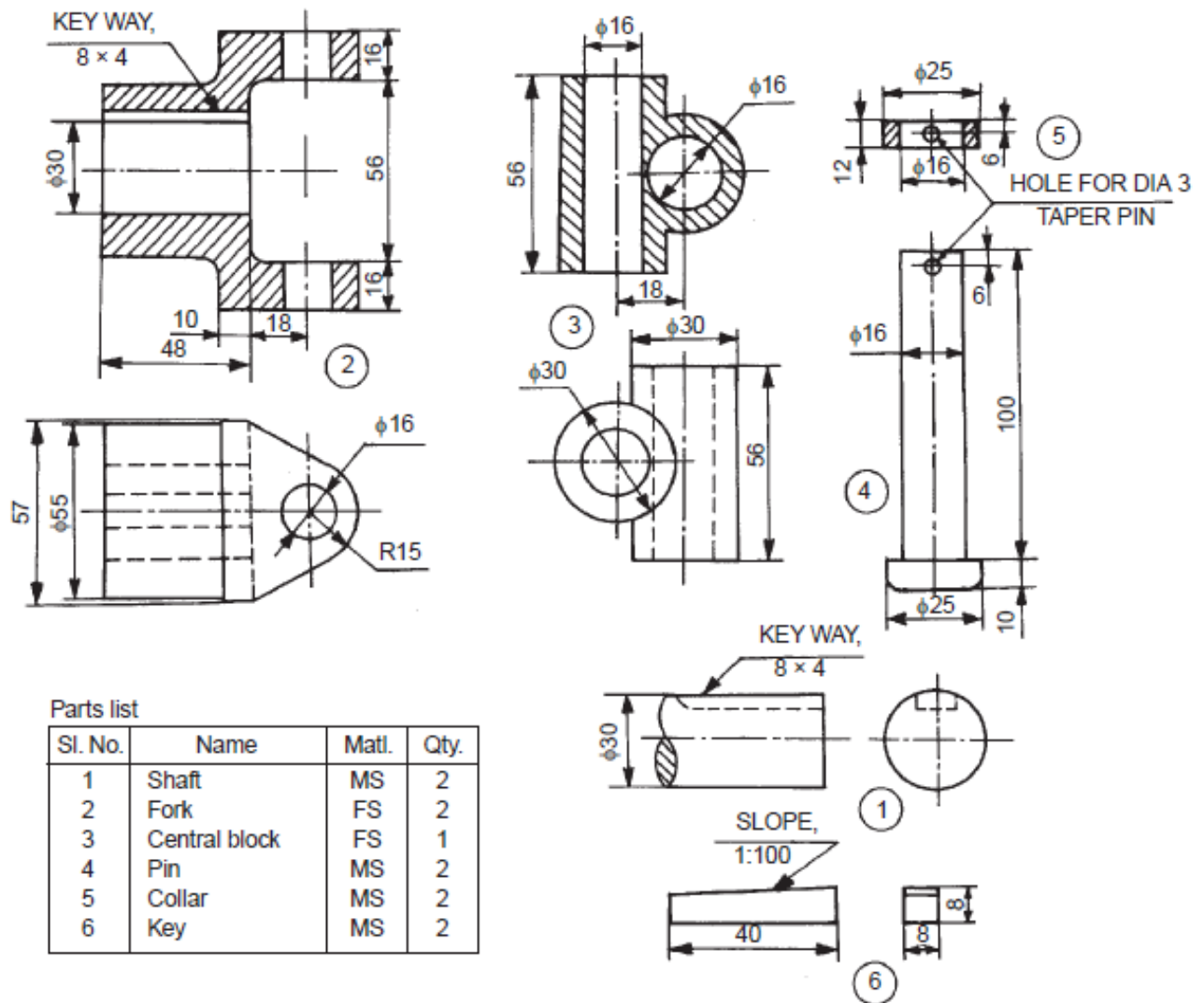
Revolve → Sketch plane → Sketches → Centerlines → Sketch to dimension → Ok

Assembly:

Assembly → Set default → Assembly according to alignment → fully constrained → Ok



Universal Coupling



Details of Universal Coupling

Result:

Thus the given various part of universal coupling had been created and assembled by using standard CAD software Pro-E.

Outcome:

From this experiment, Students will be able to assemble the given universal coupling using standard CAD software which is used in automobile.

Application:

1. Automobile

Viva-voce

1. Define – Coupling
2. What are the different types of couplings?
3. What is meant by universal coupling?
4. What are the components used in universal coupling?
5. What type of keyway is used in universal coupling?

Expt. No.09**3D ASSEMBLY OF OLDHAM'S COUPLING****Aim:**

To create various parts of Oldham's coupling and assemble them by using standard CAD software Pro-E

System configuration:

Ram	: 8 GB
Processor	: Core 2 Quad / Core 2 Duo
Operating system	: Windows 7
Software	: Pro-E Wildfire 4.0

Description:

This coupling is known as a non-aligned coupling and is used to connect two parallel shafts, whose axes are at a small distance apart. The given figure shows various parts of a Oldham's coupling. The two flanges 1 are mounted on the ends of shafts 3 by means of sunk keys 4. The flanges are having rectangular slots in them. These flanges are set such that, the slots in them are at right angle to each other. The circular disc 2 is now positioned in between them so that the projections in the circular disc, enter into the corresponding slots of the flanges. During rotation of the shafts, the central disc slides in the slots of the flanges.

Commands used:**Part diagram:**

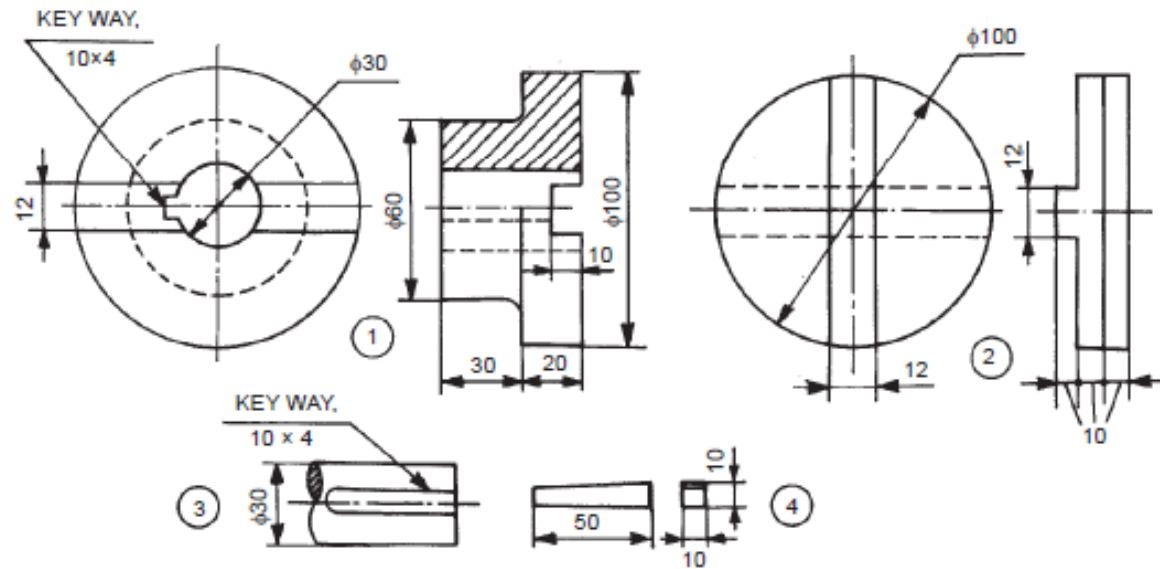
Extrude → Sketch → Sketch to dimension → Ok

Hole → Sketch → given dimension → Ok

Revolve → Sketch plane → Sketches → Centerlines → Sketch to dimension → Ok

Assembly:

Assembly → Set default → Assembly according to alignment → fully constrained → Ok



Parts list

Sl. No.	Name	Matl.	Qty.
1	Flange	MS	2
2	Disc	MS	1
3	Shaft	MS	2
4	Key	MS	2

Details of Oldham's Coupling

Result:

Thus the given various part of Oldham's coupling had been created and assembled by using standard CAD software Pro-E.

Outcome:

From this experiment, Students will be able to assemble the given Oldham's coupling using standard CAD software.

Application:

1. Automobile

Viva-voce

1. Define – Coupling
2. What is oldham's coupling?
3. What are the advantages of oldham's coupling?
4. What are the components used in oldham's coupling?
5. What are the materials used in oldham's coupling?

Expt. No.10**3D ASSEMBLY OF KNUCKLE JOINT****Aim:**

To create various parts of knuckle joint and assemble them by using standard CAD software Pro-E

System configuration:

Ram	: 8 GB
Processor	: Core 2 Quad / Core 2 Duo
Operating system	: Windows 7
Software	: Pro-E Wildfire 4.0

Description:

This is a pin joint and is used to connect two circular rods subjected to axial loads. Compared to a socket and spigot joint, wherein the axes of both the rods should be in the same plane; in the knuckle joint, one of the rods can be swiveled through some angle about the connecting pin, i.e., the axes of the two rods could be inclined to each other. The given figure shows the details of a knuckle joint. The eye end of the rod 2 is inserted into the fork end 1 of the other rod. Then, pin 3 is inserted through the holes in the ends of the rods and held in position by the collar 4 and taper pin 5.

Commands used:**Part diagram:**

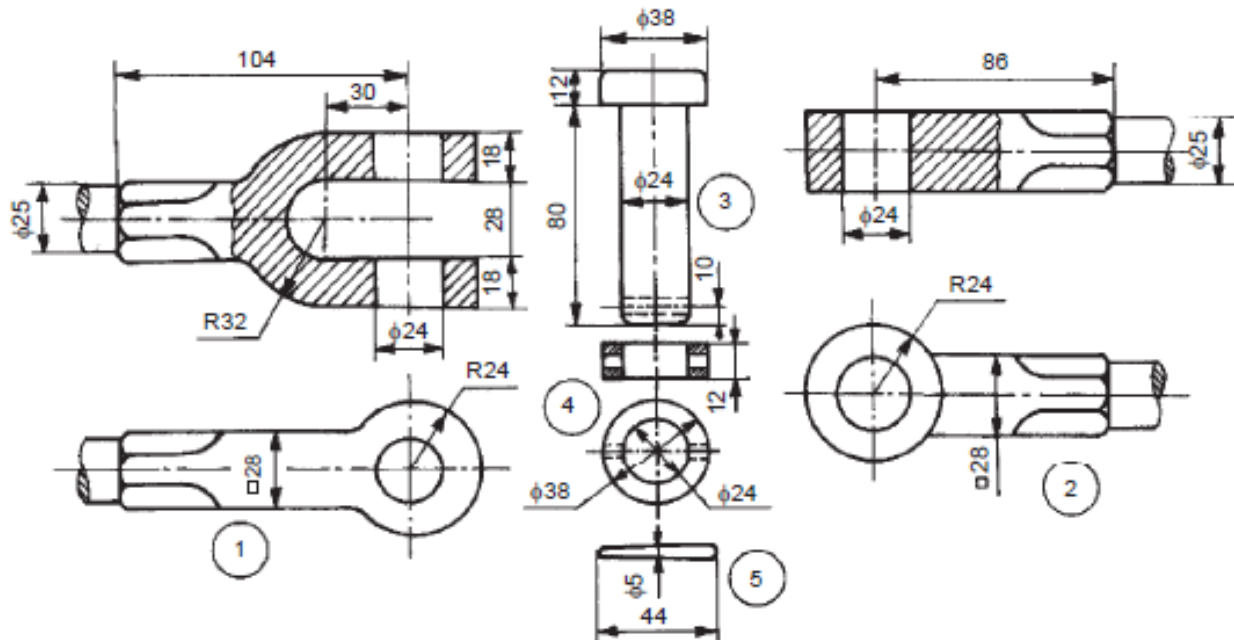
Extrude → Sketch → Sketch to dimension → Ok

Hole → Sketch → given dimension → Ok

Revolve → Sketch plane → Sketches → Centerlines → Sketch to dimension → Ok

Assembly:

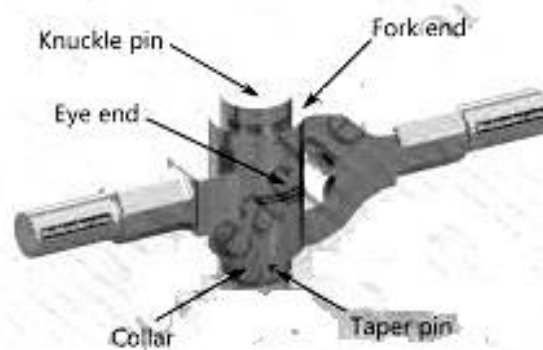
Assembly → Set default → Assembly according to alignment → fully constrained → Ok



Parts list

Sl. No.	Name	Matl.	Qty.
1	Fork end	Forged steel	1
2	Eye end	Forged steel	1
3	Pin	Mild steel	1
4	Collar	Mild steel	1
5	Taper pin	Mild steel	1

Details of Knuckle Joint



Result:

Thus the given various part of knuckle joint had been created and assembled by using standard CAD software Pro-E.

Outcome:

From this experiment, Students will be able to assemble the given knuckle joint using standard CAD software.

Application:

1. Automobile

Viva-voce

1. In what situation flexible coupling is used?
2. What are the applications of knuckle joint?
3. Why is taper given to key? Why is taper given only on one side?
4. Knuckle Joint is used to connect two rods _____
5. Knuckle joint can't be used for _____

Expt. No.11 3D ASSEMBLY OF SOCKET AND SPIGOT JOINT**Aim:**

To create various parts of socket and spigot joint and assemble them by using standard CAD software Pro-E

System configuration:

Ram	: 8 GB
Processor	: Core 2 Quad / Core 2 Duo
Operating system	: Windows 7
Software	: Pro-E Wildfire 4.0

Description:

This joint is used to connect two circular rods and subjected to axial loads. For this purpose, the ends of the rods are shaped suitably to form a socket at one end while the other rod end as a spigot and slots are made. The given figure shows the details of a socket and spigot joint. The spigot end of the rod 2 is inserted into the socket end of the rod 1. After aligning the socket and spigot ends, the cotter 3 is driven-in, forming the joint.

Commands used:

Part diagram:

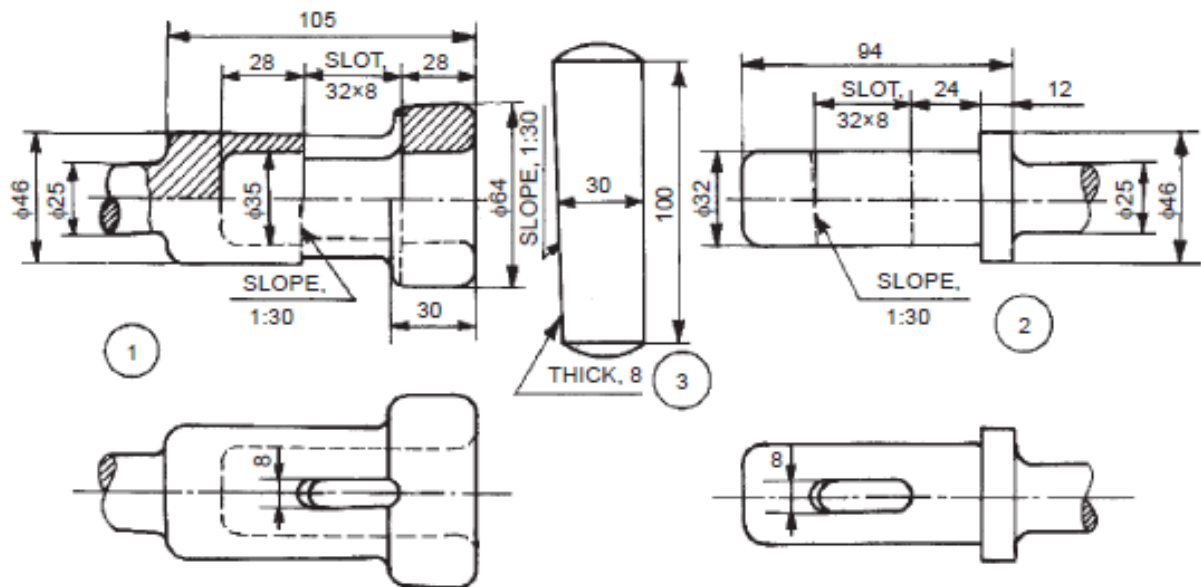
Extrude → Sketch → Sketch to dimension → Ok

Hole → Sketch → given dimension → Ok

Revolve → Sketch plane → Sketches → Centerlines → Sketch to dimension → Ok

Assembly:

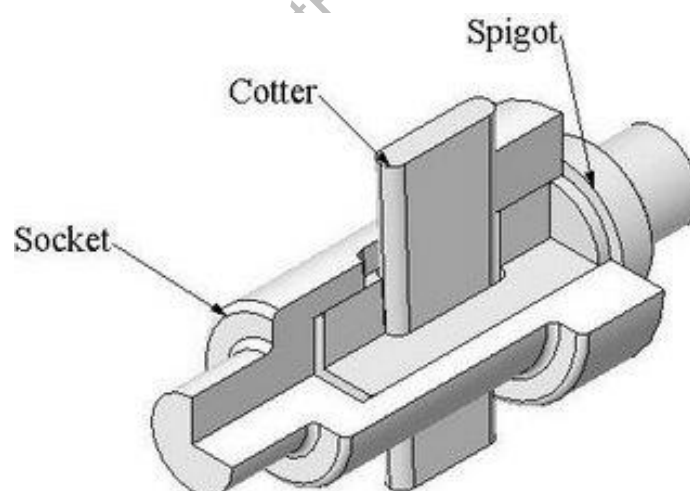
Assembly → Set default → Assembly according to alignment → fully constrained → Ok



Parts list

Sl. No.	Name	Matl.	Qty.
1	Socket end	MS	1
2	Spigot end	MS	1
3	Cotter	HCS	1

Details of Socket and Spigot Joint



Result:

Thus the given various part of socket and spigot joint had been created and assembled by using standard CAD software Pro-E.

Outcome:

From this experiment, Students will be able to assemble the given socket and spigot joint using standard CAD software.

Application:

1. Automobile

Viva-voce

1. What is socket?
2. What is spigot?
3. What is meant by cotter?
4. Why in a cotter joint the cotter is kept weakest?
5. State the advantages of cotter joint.
6. Why taper is provided on cotter?
7. How much taper is provided on cotter?

Expt. No.12 **3D ASSEMBLY GIB AND COTTER JOINT****Aim:**

To create various parts of gib and cotter joint and assemble them by using standard CAD software Pro-E

System configuration:

Ram	: 8 GB
Processor	: Core 2 Quad / Core 2 Duo
Operating system	: Windows 7
Software	: Pro-E Wildfire 4.0

Description:

Gib and cotter joints are used for rods of square or rectangular cross section, the ends of one rod fits the end of the other rod which is made in the form of a strap. A gib is used along with the cotter to make this joint. Gib is likely a cotter but with two gib heads at its ends. The thickness of the gib and cotter are same.

Commands used:**Part diagram:**

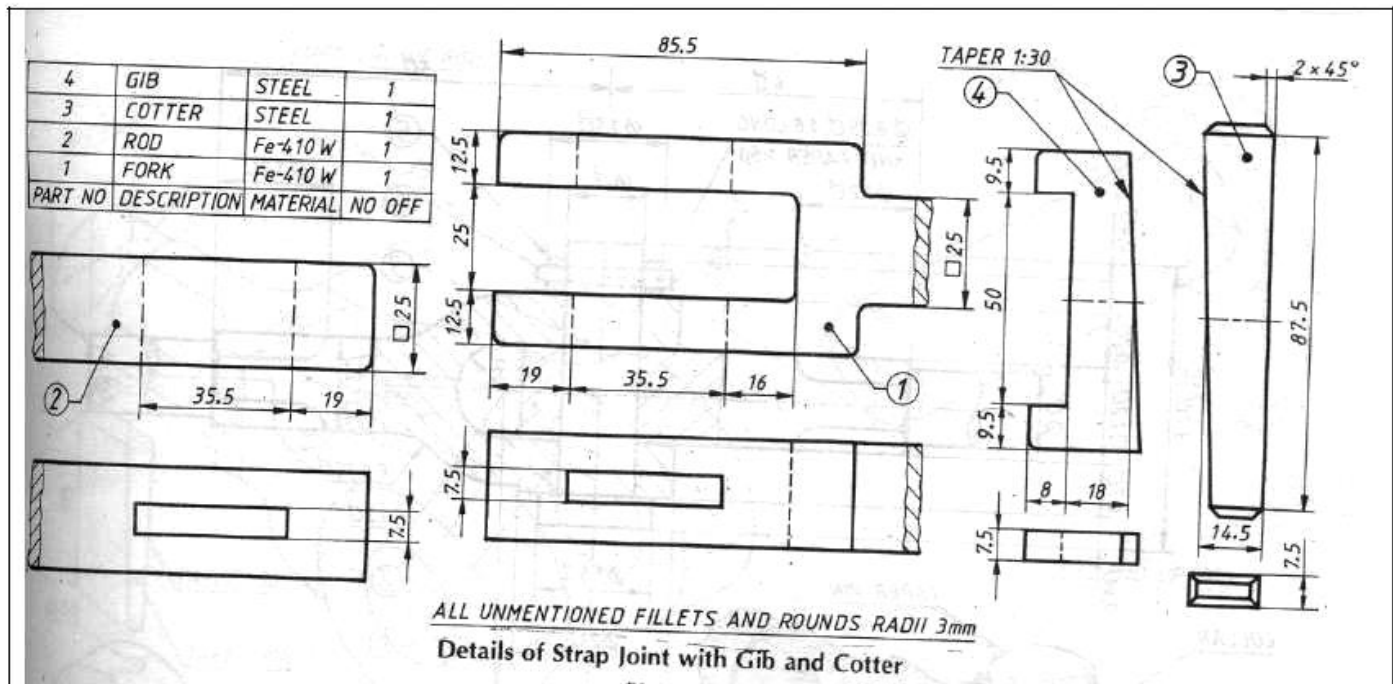
Extrude → Sketch → Sketch to dimension → Ok

Hole → Sketch → given dimension → Ok

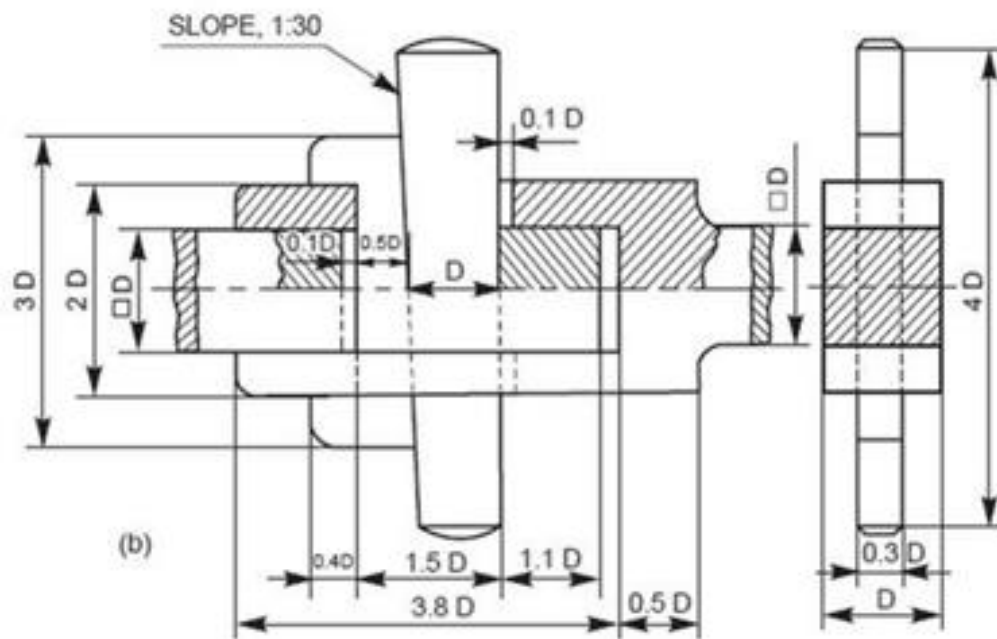
Revolve → Sketch plane → Sketches → Centerlines → Sketch to dimension → Ok

Assembly:

Assembly → Set default → Assembly according to alignment → fully constrained → Ok



Details of Gib and Cotter Joint



Result:

Thus the given various part of gib and cotter joint had been created and assembled by using standard CAD software Pro-E.

Outcome:

From this experiment, Students will be able to assemble the given gib and cotter joint using standard CAD software.

Application:

1. Connect piston rod in cross head
2. Bicycle to connect pedal to sprocket wheel

Viva-voce

1. What is the function of gib and cotter joint?
2. Name the components of gib and cotter joint.
3. State the applications of cotter joints.
4. Name the materials used for gib and cotter.
5. Name the material used for rod and fork.

Expt. No.13 **3D ASSEMBLY CONNECTING ROD****Aim:**

To create various parts of connecting rod and assemble them by using standard CAD software Pro-E

System configuration:

Ram	: 8 GB
Processor	: Core 2 Quad / Core 2 Duo
Operating system	: Windows 7
Software	: Pro-E Wildfire 4.0

Description:

A connecting rod is an engine component that transfers motion from the piston to the crankshaft and functions as a lever arm. Connecting rods are commonly made from cast aluminum alloy and are designed to withstand dynamic stresses from combustion and piston movement. The small end of the connecting rod connects to the piston with a piston pin. The piston pin, or wrist pin, provides a pivot point between the piston and connecting rod. Spring clips, or piston pin locks, are used to hold the piston pin in place. The big end of the connecting rod connects to the crankpin journal to provide a pivot point on the crankshaft. Connecting rods are produced as one piece or two-piece components. A rod cap is the removable section of a two-piece connecting rod that provides a bearing surface for the crankpin journal. The rod cap is attached to the connecting rod with two cap screws for installation and removal from the crankshaft.

Commands used:

Part diagram:

Extrude → Sketch → Sketch to dimension → Ok

Hole → Sketch → given dimension → Ok

Revolve → Sketch plane → Sketches → Centerlines → Sketch to dimension → Ok

Assembly:

Assembly → Set default → Assembly according to alignment → fully constrained → Ok



Result:

Thus the given various part of connecting rod had been created and assembled by using standard CAD software Pro-E.

Outcome:

From this experiment, Students will be able to assemble the given connecting rod using standard CAD software.

Application:

1. Internal Combustion Engines
2. Reciprocating Compressor

Viva-voce

1. What is the function of connecting rod?
2. Name the materials used for connecting rod.
3. Why I-section is mostly used in the connecting rod?
4. Small end bearing is connected to the _____.
5. Big end bearing is connected to the _____.

Expt. No.14 3D ASSEMBLY OF PISTON**Aim:**

To create various parts of piston and assemble them by using standard CAD software Pro-E

System configuration:

Ram	: 8 GB
Processor	: Core 2 Quad / Core 2 Duo
Operating system	: Windows 7
Software	: Pro-E Wildfire 4.0

Description:

In the cylinder of an engine, the energy bound up in the fuel is rapidly converted into heat and pressure during the combustion cycle. The heat and pressure values increase greatly within a very short period of time. The piston, as the moving part of the combustion chamber, has the task of converting this released energy into mechanical work.

The basic structure of the piston is a hollow cylinder, closed on one side, with the segments piston crown with ring belt, pin boss, and skirt. The piston crown transfers the gas forces resulting from the combustion of the fuel-air mixture via the pin boss, the piston pin, and the connecting rod, to the crankshaft.

Pistons are usually equipped with piston rings. These are circular metal rings that fit into grooves in the piston walls and assure a snug fit of the piston inside the cylinder. They help provide a seal to prevent leakage of compressed gases around the piston and to prevent lubricating oil from entering the combustion chamber.

Commands used:**Part diagram:**

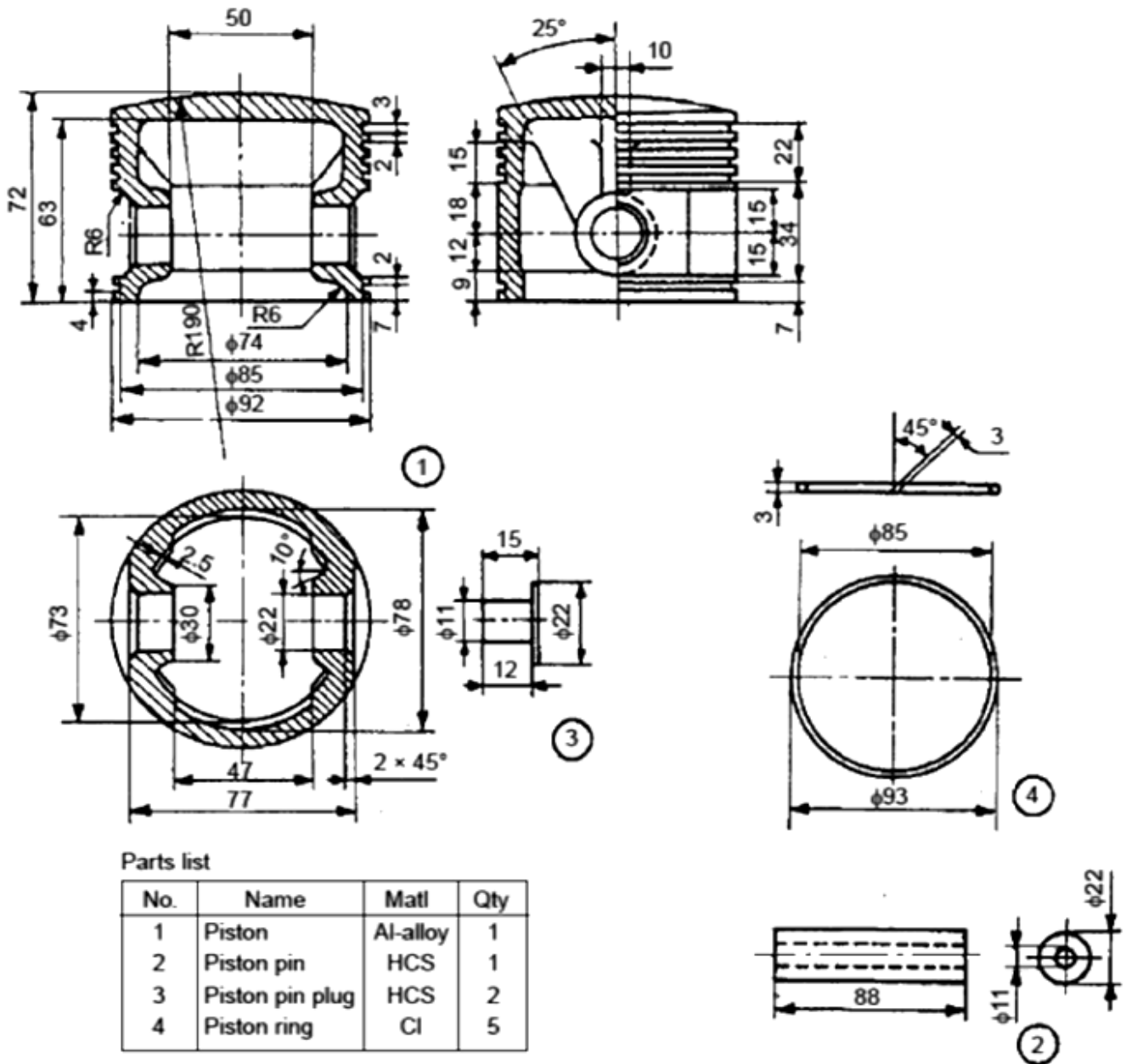
Extrude → Sketch → Sketch to dimension → Ok

Hole → Sketch → given dimension → Ok

Revolve → Sketch plane → Sketches → Centerlines → Sketch to dimension → Ok

Assembly:

Assembly → Set default → Assembly according to alignment → fully constrained → Ok



Details of Piston

Result:

Thus the given various part of piston had been created and assembled by using standard CAD software Pro-E.

Outcome:

From this experiment, Students will be able to assemble the given piston using standard CAD software.

Application:

1. Internal Combustion Engines
2. Reciprocating Compressor

Viva-voce

1. What is the function of piston?
2. What is meant by gudgeon pin?
3. What type of material is used for piston?
4. What is piston ring?
5. Name the types of piston rings.
6. What is the function of piston rings?

Expt. No.15 3D ASSEMBLY OF STUFFING BOX**Aim:**

To create various parts of stuffing box and assemble them by using standard CAD software Pro-E

System configuration:

Ram	: 8 GB
Processor	: Core 2 Quad / Core 2 Duo
Operating system	: Windows 7
Software	: Pro-E Wildfire 4.0

Description:

It is used to prevent loss of fluid such as steam, between sliding or turning parts of machine elements. In a steam engine, when the piston rod reciprocate through the cylinder cover; stuffing box provided in the cylinder cover, prevent leakage of steam from the cylinder.

The given figure shows the various parts of a stuffing box. At the base of stuffing box body 1, a bush 3 is placed such that the beveled edge of the bush is at the inner side of the body. Gland 2 is placed at the other end of the body and is connected to the main body by means of studs 4 and nuts 5. The space between the reciprocating rod and the bush and the gland is packed with a packing material such as fibres, leather, rubber or cork.

Commands used:

Part diagram:

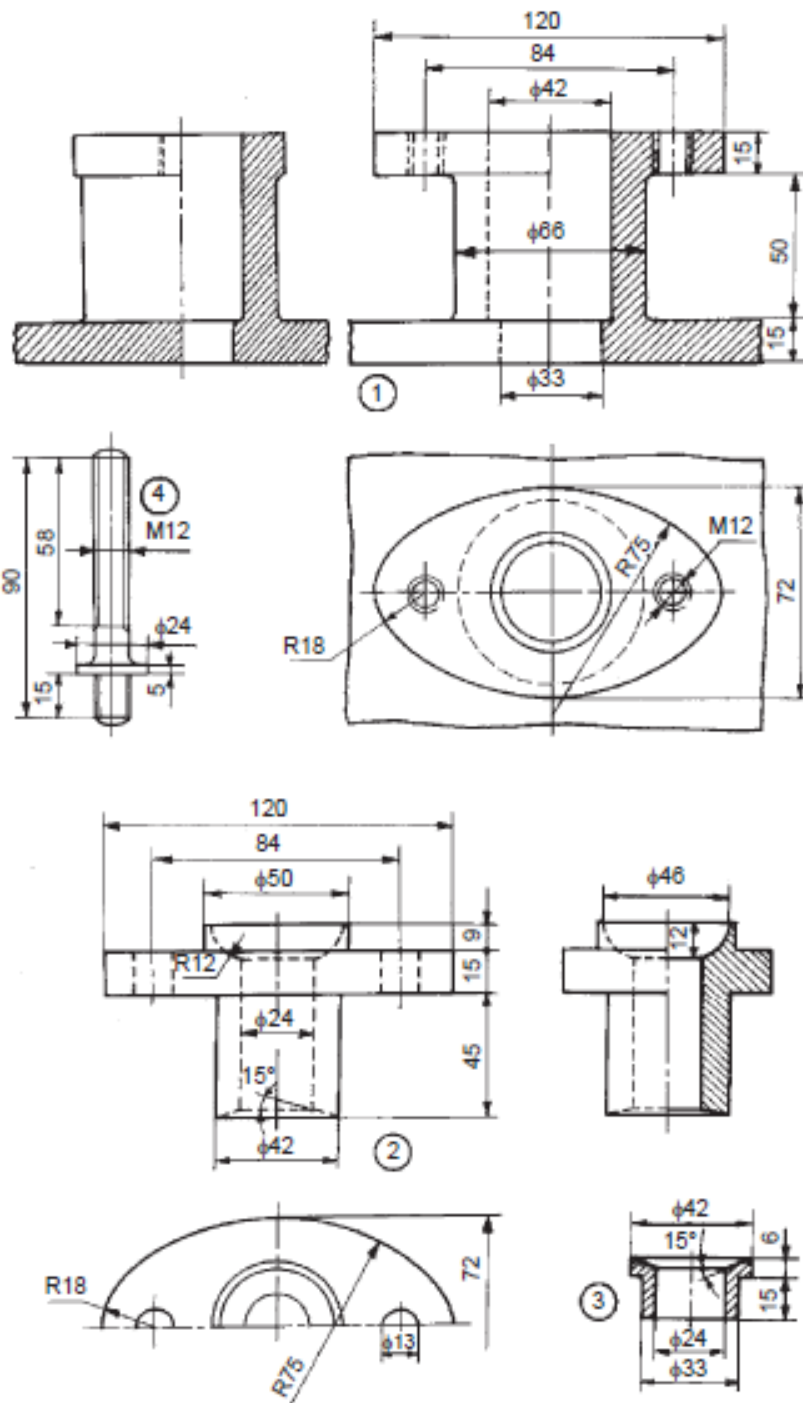
Extrude → Sketch → Sketch to dimension → Ok

Hole → Sketch → Given dimension → Ok

Revolve → Sketch plane → Sketches → Centerlines → Sketch to dimension → Ok

Assembly:

Assembly → Set default → Assembly according to alignment → fully constrained → Ok



Parts list

Part No.	Name	Matl	Qty
1	Body	CI	1
2	Gland	Brass	1
3	Bush	Brass	1
4	Stud	MS	2
5	Nut, M12	MS	2

Details of Stuffing box

Result:

Thus the given various part of stuffing box had been created and assembled by using standard CAD software Pro-E.

Outcome:

From this experiment, Students will be able to assemble the given stuffing box using standard CAD software.

Application:

1. Steam engine

Viva-voce

1. Name the components in stuffing box.
2. What is stuffing box?
3. What is meant by gland?
4. What is the application of stuffing box?
5. What are the materials required for stuffing box?

Expt. No.16 3D ASSEMBLY OF CROSSHEAD**Aim:**

To create various parts of crosshead and assemble them by using standard CAD software Pro-E

System configuration:

Ram	: 8 GB
Processor	: Core 2 Quad / Core 2 Duo
Operating system	: Windows 7
Software	: Pro-E Wildfire 4.0

Description:

Crosshead is used in horizontal steam engines for connecting the piston rod and connecting rod. The given figure shows the part drawings of a steam engine crosshead. The crosshead, with the help of slide block 4, reciprocates between two guides provided in the engine frame. The gudgeon pin 3 connects the slide blocks with the crosshead block 1. This acts as a pin joint for the connecting rod. The piston rod 2 is secured to the crosshead block by means of the cotter 5. The assembly ensures reciprocating motion along a straight line for the piston rod and reciprocating cum oscillatory motion for the connecting rod.

Commands used:**Part diagram:**

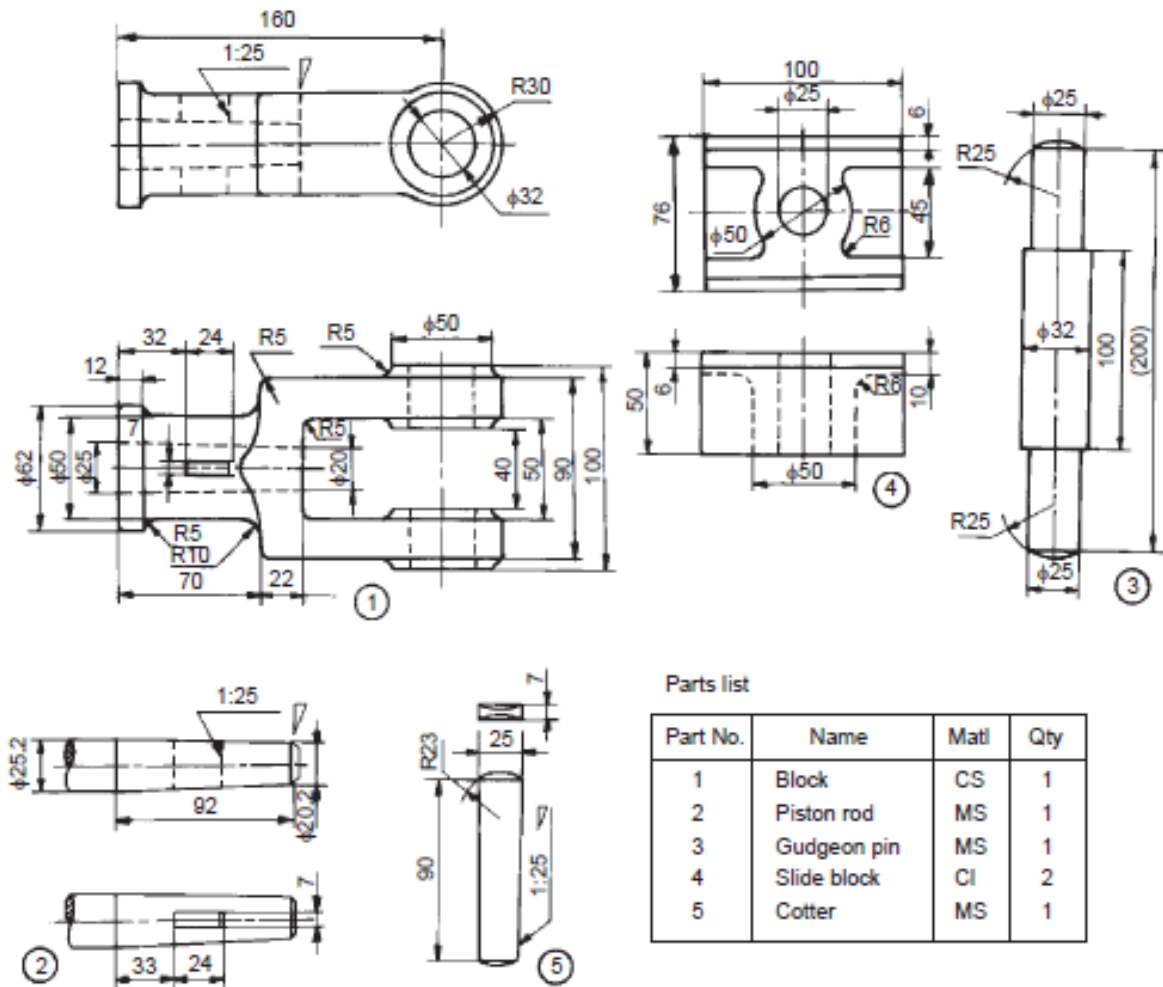
Extrude → Sketch → Sketch to dimension → Ok

Hole → Sketch → given dimension → Ok

Revolve → Sketch plane → Sketches → Centerlines → Sketch to dimension → Ok

Assembly:

Assembly → Set default → Assembly according to alignment → fully constrained → Ok



Details of Crosshead

Result:

Thus the given various part of crosshead had been created and assembled by using standard CAD software Pro-E.

Outcome:

From this experiment, Students will be able to assemble the given crosshead using standard CAD software which is used in steam engine.

Application:

1. Steam engine

Viva-voce

1. What is meant by cross head?
2. What is the function of cross head?
3. What are the components used in cross head?
4. What is meant by piston rod?
5. What is the function of gudgeon pin?

Expt. No.17 3D ASSEMBLY OF MULTIPLATE CLUTCH**Aim:**

To create various parts of multi plate clutch and assemble them by using standard CAD software Pro-E

System configuration:

Ram	: 8 GB
Processor	: Core 2 Quad / Core 2 Duo
Operating system	: Windows 7
Software	: Pro-E Wildfire 4.0

Description:

In a multi plate friction clutch, a number of discs or plates with frictional surfaces on either side are located alternately on the driving and driven shafts. By bringing these, close together by external pressure, power can be transmitted from the driving to the driven shaft. Multi plate clutches are normally wet type, whereas, single plate clutches are dry. The multi plate clutch is located on the driving shaft. The pulley/wheel mounted on the shell serves as a driven member. There is no driven shaft.

Figure A illustrates various parts of a multi plate clutch. The serrations on the internal diameter of the shell 1, receive the alternate friction discs called the outside plates 3. The shell along with a bush is mounted freely on the driving shaft, has provision to receive the driven pulley or wheel. A centre boss 5 is keyed onto the driving shaft. The serrations on the centre boss receive the alternate friction discs called the inside plates 4.

The front cover 2, is fixed to the shell by means of six small set screws 12. The front cover is able to either slide or rotate freely on the driving shaft with the bush provided. The sliding sleeve 9, rotates freely on the driving shaft and is connected to the front cover through three jaw levers 7, and three L-levers 8. The last outside plate called a stud carrier 6, is provided with the arrangement to carry three studs 10, which are operated by the sliding sleeve through the L-lever.

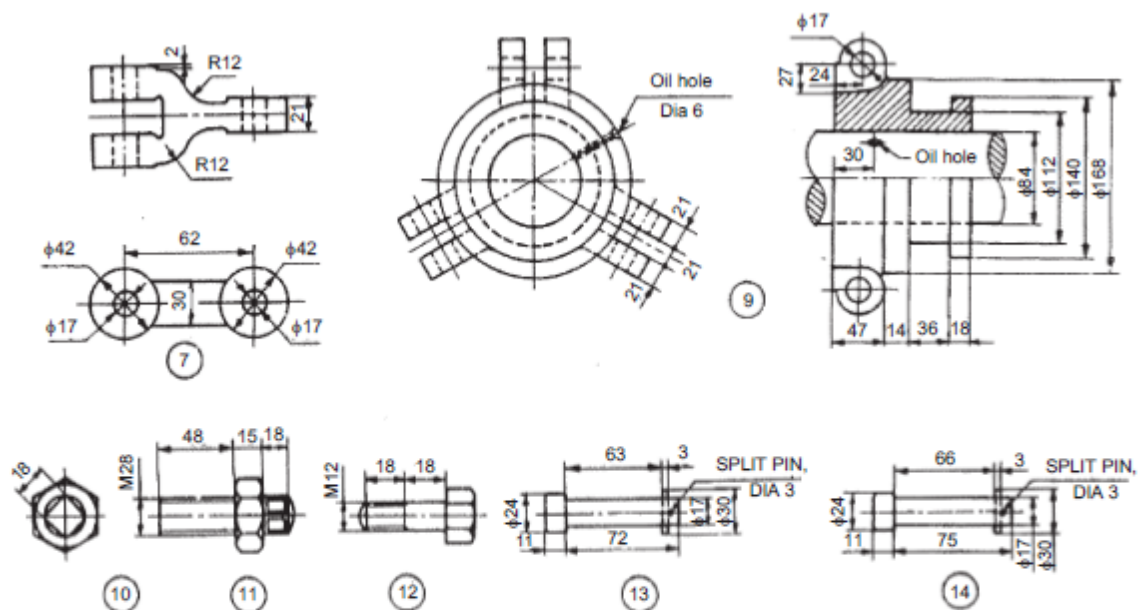
The sliding sleeve rotates with the clutch body, *i.e.*, the shell. The sleeve can be moved forward by an operating lever (not shown) in order to exert pressure on the studs. The plates will then get pressed together and the drive is completed. Figure B shows the assembly drawing of the multi plate clutch.

Commands used:

Part diagram:

Extrude → Sketch → Sketch to dimension → Ok

Hole → Sketch → given dimension → Ok



Parts list

Sl No.	Name	Matl	Qty	Sl No.	Name	Matl	Qty
1	Shell	CI	1	8	L-Lever	MS	3
2	Front cover	CI	1	9	Sliding sleeve	CI	1
3	Outside plate	CI	3	10	Stud	MS	3
4	Inside plate	CI	4	11	Nut	MS	3
5	Centre boss	CI	1	12	Set screw	MS	6
6	Stud carrier	CI	1	13	Sleeve pin	MS	3
7	Jaw lever	MS	3	14	Jaw lever and front cover pin	MS	6

Details of Multi-plate Clutch

Result:

Thus the given various part of multiplate clutch had been created and assembled by using standard CAD software Pro-E.

Outcome:

From this experiment, Students will be able to assemble the given multiplate clutch using standard CAD software.

Application:

1. Automobile

Viva-voce

1. What is the function of clutch?
2. Where do you use clutch?
3. Name the types of clutches.
4. Where do you use multi-plate clutch?
5. What are the advantages of single plate clutch over multi plate clutch?

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Expt. No.18 **3D ASSEMBLY OF SCREW JACK****Aim:**

To create various parts of screw jack and assemble them by using standard CAD software Pro-E

System configuration:

Ram	: 8 GB
Processor	: Core 2 Quad / Core 2 Duo
Operating system	: Windows 7
Software	: Pro-E Wildfire 4.0

Description:

Screw jacks are used for raising heavy loads through very small heights. Figure shows the details of one type of screw jack. In this, the screw 3 works in the nut 2 which is press fitted into the main body 1. The tommy bar 7 is inserted into a hole through the enlarged head of the screw and when this is turned, the screw will move up or down, thereby raising or lowering the load.

Commands used:

Part diagram:

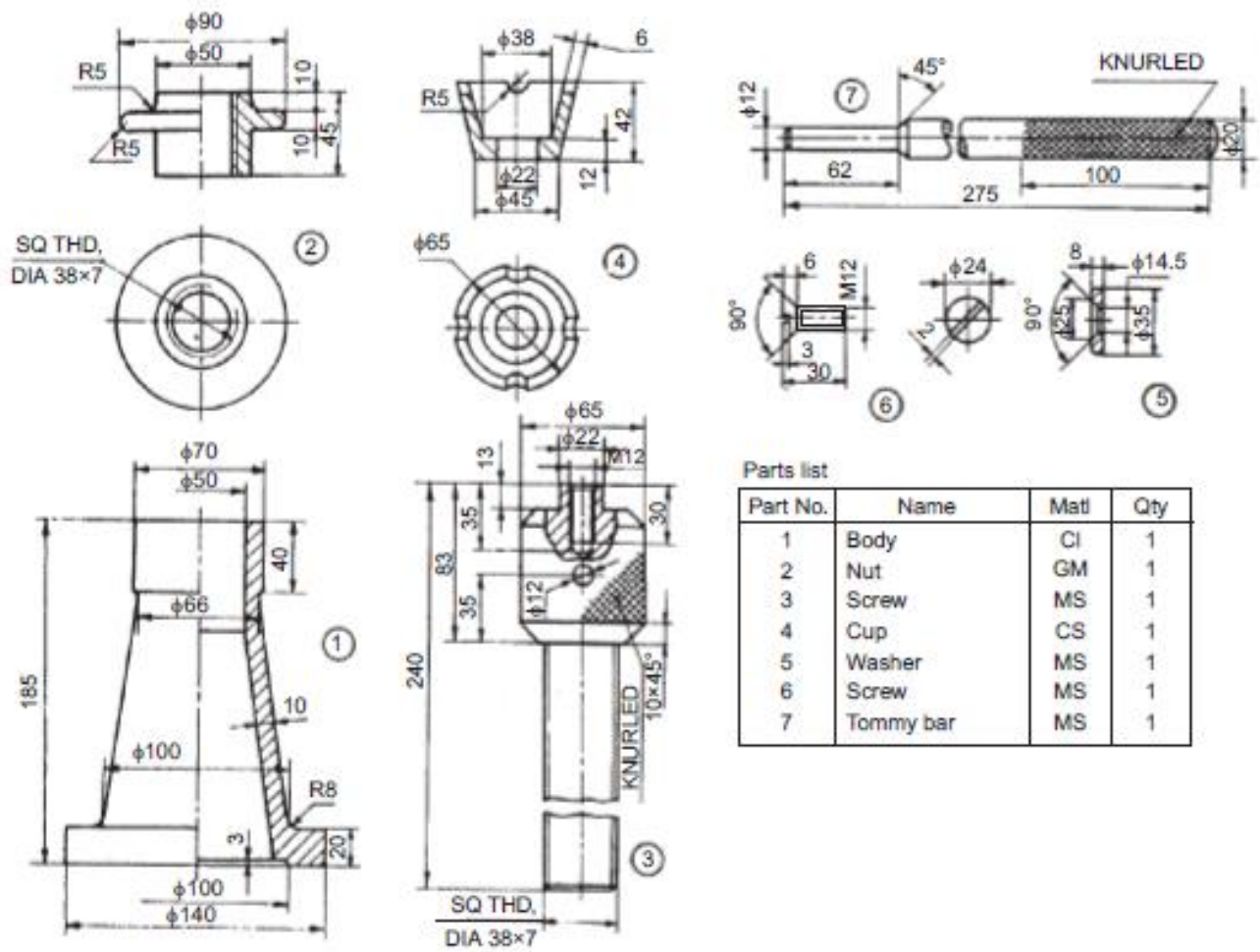
Extrude → Sketch → Sketch to dimension → Ok

Hole → Sketch → given dimension → Ok

Revolve → Sketch plane → Sketches → Centerlines → Sketch to dimension → Ok

Assembly:

Assembly → Set default → Assembly according to alignment → fully constrained → Ok



Details of Screw Jack

Result:

Thus the given various part of screw jack had been created and assembled by using standard CAD software Pro-E.

Outcome:

From this experiment, Students will be able to assemble the given screw jack using standard CAD software.

Application:

1. Automobile

Viva-voce

1. How will you designate square threads?
2. What is power screw?
3. What are the components used for screw jack?
4. What is the use of screw jack?
5. Name the materials used for making screw jack.

Expt. No.19**3D ASSEMBLY OF MACHINE VICE****Aim:**

To create various parts of screw jack and assemble them by using standard CAD software Pro-E

System configuration:

Ram	: 8 GB
Processor	: Core 2 Quad / Core 2 Duo
Operating system	: Windows 7
Software	: Pro-E Wildfire 4.0

Description:

A machine vice is a work holding device, used in machines such as drilling, milling, etc. A swiveling type machine vice permits swiveling about its vertical axis, so that the work may be clamped at any angular position required in the machining operation. T-bolts (not shown) are used through the base plate, to fix the vice to the machine table.

The details of a plain machine vice are shown in figure. It consists of the base 1 which is clamped to the machine table using two T-bolts. The sliding block 3 is fixed in the centre slot of the base by means of the guide screw 4. The movable jaw 2 is fixed to the sliding block with four screws 8 and 7. One of the serrated plates 5 is fixed to the jaw of the base by means of screws 6 and the other to the movable jaw by the screws 7. One end of the guide screw is fixed to the base by means of the washer 9 and nut 10 (not shown in figure). The movable jaw is operated by means of a handle (not shown) which fits onto the square end of the guide screw.

Commands used:**Part diagram:**

Extrude → Sketch → Sketch to dimension → Ok

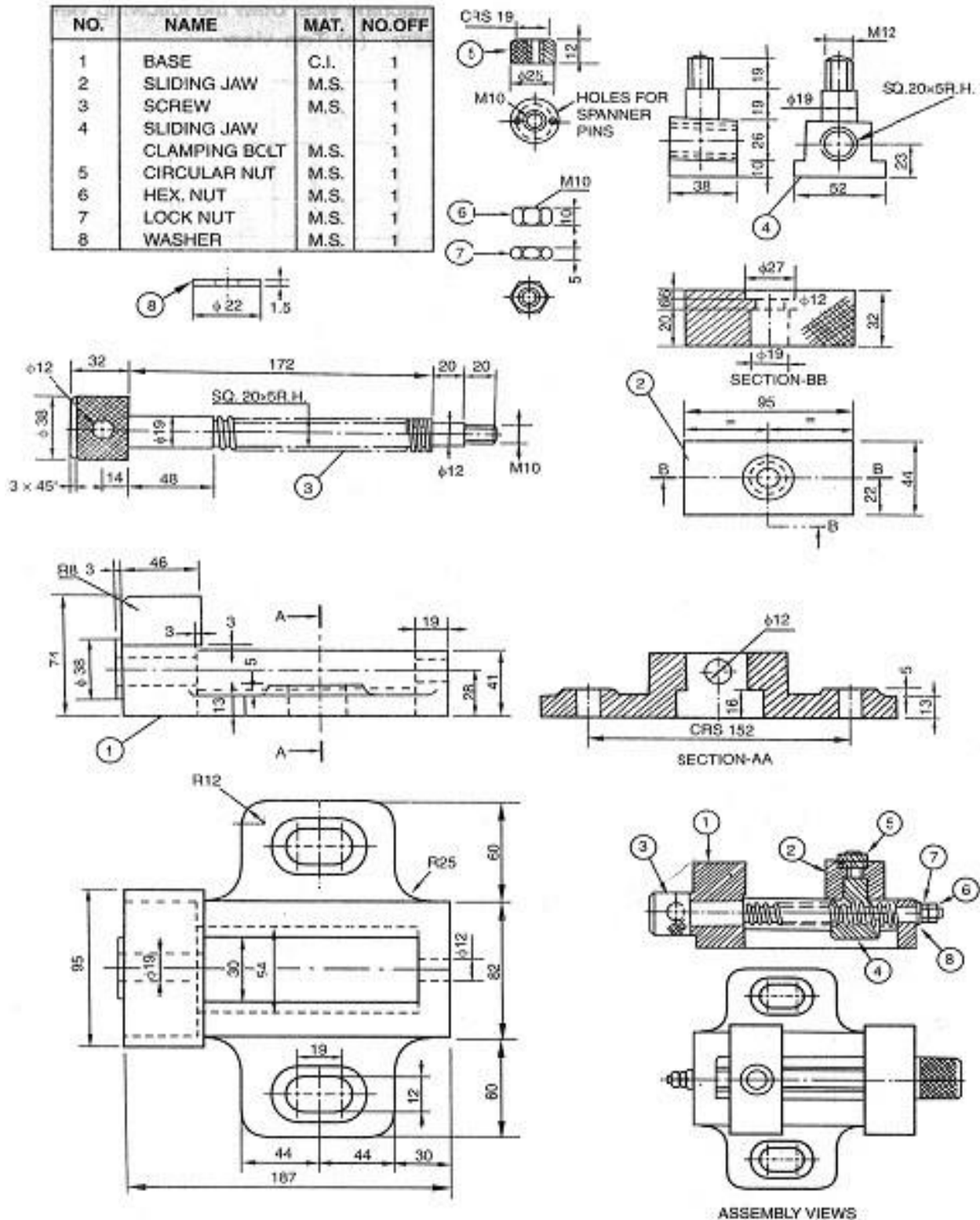
Hole → Sketch → given dimension → Ok

Revolve → Sketch plane → Sketches → Centerlines → Sketch to dimension → Ok

Assembly:

Assembly → Set default → Assembly according to alignment → fully constrained → Ok

NO.	NAME	MAT.	NO.OFF
1	BASE	C.I.	1
2	SLIDING JAW	M.S.	1
3	SCREW	M.S.	1
4	SLIDING JAW		1
	CLAMPING BCLT	M.S.	1
5	CIRCULAR NUT	M.S.	1
6	HEX. NUT	M.S.	1
7	LOCK NUT	M.S.	1
8	WASHER	M.S.	1



Details of a machine vice

Result:

Thus the given various part of machine vice had been created and assembled by using standard CAD software Pro-E.

Outcome:

From this experiment, Students will be able to assemble the given machine vice using standard CAD software.

Application:

1. Work holding devices

Viva-voce

1. Name the components of machine vice?
2. What is a Machine Vice?
3. Name the different types of vice.
4. What are the tools used in machine vice?
5. What is meant by sliding jaw?

Expt. No.20 3D ASSEMBLY OF TAIL STOCK**Aim:**

To create various parts of tail stock and assemble them by using standard CAD software Pro-E

System configuration:

Ram	: 8 GB
Processor	: Core 2 Quad / Core 2 Duo
Operating system	: Windows 7
Software	: Pro-E Wildfire 4.0

Description:

It is a part of a lathe machine and is used to support lengthy jobs. To accommodate works of different lengths between centres, the tail-stock may be moved on the lathe bed to the required position and clamped by means of a clamping bolt.

Figure shows various parts of a tail-stock. The barrel 2 is fitted into the bore of the tail-stock body 1 and is prevented from rotation by the feather key 9 placed underneath of it. The barrel has a threaded portion at its end and the spindle 3 is inserted into the barrel through this. The hand wheel 6 is mounted on the spindle by a key and is retained in position by a nut. The spindle bearing 5 is placed between the hand wheel and the tail-stock body. A tapered hole provided at the front end of the barrel, receives the dead centre 4 or a tapered shank of the drill or reamer.

When the hand wheel is operated, the barrel is made to move in or out of the tail-stock body. In the required position of the barrel, clamping may be made by means of the clamping lever 7 and stud 8 which is fitted to the tail-stock body. The spindle bearing is fixed to the body by means of the screws 10

Commands used:**Part diagram:**

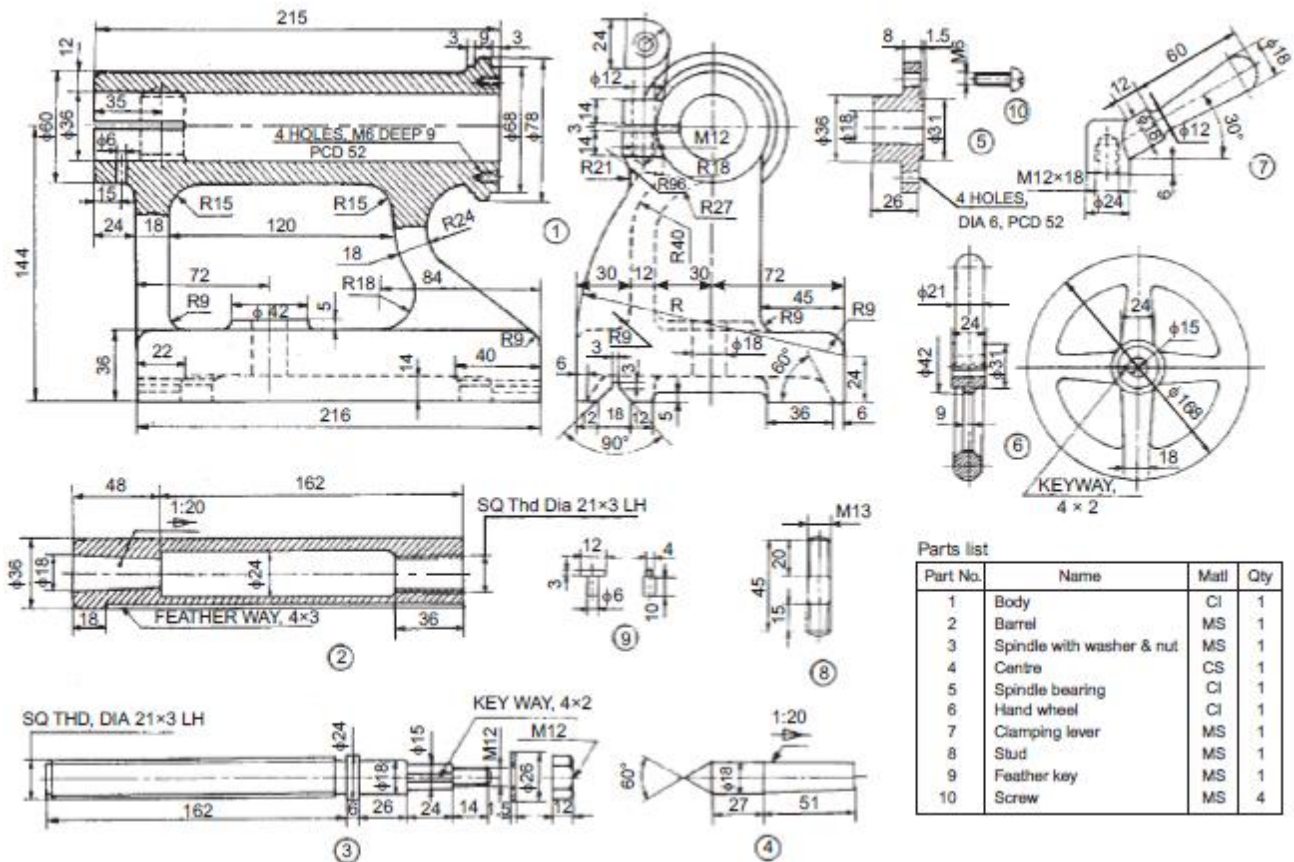
Extrude → Sketch → Sketch to dimension → Ok

Hole → Sketch → given dimension → Ok

Revolve → Sketch plane → Sketches → Centerlines → Sketch to dimension → Ok

Assembly:

Assembly → Set default → Assembly according to alignment → fully constrained → Ok



Details of Tail Stock

Result:

Thus the given various part of lathe tail stock had been created and assembled by using standard CAD software Pro-E.

Outcome:

From this experiment, students will be able to assemble the given lathe tail stock using standard CAD software Pro-E

Application:

1. Lathe machine

Viva-voce

1. What is the purpose of tail stock?
2. Name the components are used in tail stock.
3. What are the materials used for making tail stock?
4. What is meant by live centre?
5. What is meant by dead centre?

Expt. No.21 3D ASSEMBLY OF CHUCK**Aim:**

To create various parts of tail stock and assemble them by using standard CAD software Pro-E

System configuration:

Ram	: 8 GB
Processor	: Core 2 Quad / Core 2 Duo
Operating system	: Windows 7
Software	: Pro-E Wildfire 4.0

Description:

The self-centring chuck is a work holding device mounted on the headstock spindle of a lathe. It automatically centres the workpiece by the three jaws, moving simultaneously to and from the centre. Regular shaped objects such as rounds and hexagons are quickly held and centred in three jaw self-centring chucks.

The details of the assembly are given in figure It consists of a face plate 1, and the scroll plate 3 is fitted into the circular recess at the back of the face plate. Three pinions 4 are mounted in position in the face plate such that, their teeth engage with those on the back of the scroll plate. The back plate 2 is fastened to the face plate by six screws 7. This assembly is fastened to the flange 5 by three socket headed screws 8. The three jaws 6 are then engaged with the scroll plate. By rotating any one pinion by a chuck key (not shown), the three jaws move in the radial direction either to or from the centre. The threaded hole in the flange facilitates the mounting of the chuck in the threaded headstock spindle.

Commands used:**Part diagram:**

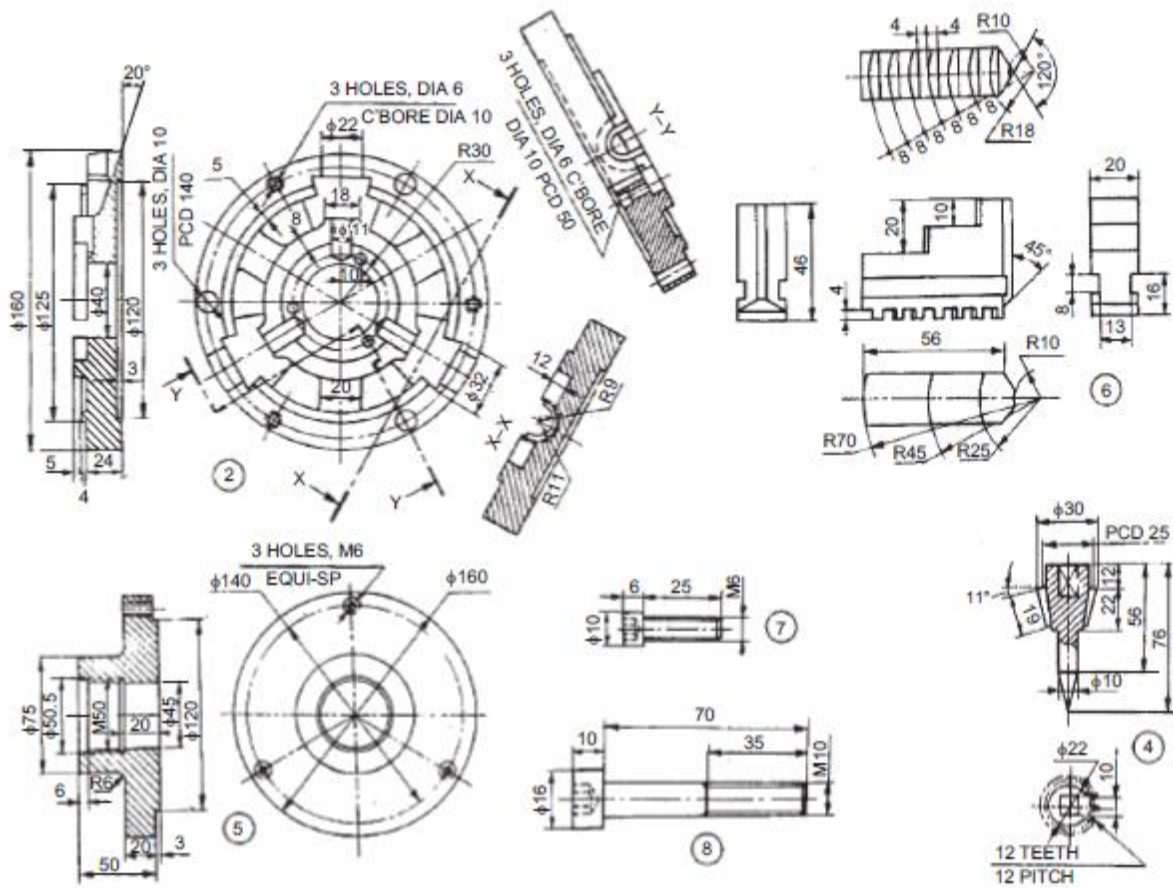
Extrude → Sketch → Sketch to dimension → Ok

Hole → Sketch → given dimension → Ok

Revolve → Sketch plane → Sketches → Centerlines → Sketch to dimension → Ok

Assembly:

Assembly → Set default → Assembly according to alignment → fully constrained → Ok



Details of Chuck

Result:

Thus the given various part of chuck had been created and assembled by using standard CAD software Pro-E.

Outcome:

From this experiment, students will be able to assemble the given chuck using standard CAD software Pro-E

Application:

1. Lathe machine
2. Milling machine

Viva-voce

1. What is the purpose of chuck?
2. Name the types of chuck.
3. What is a collet chuck?
4. What is meant by live centre?
5. What is meant by dead centre?

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Expt. No.22 3D ASSEMBLY OF GEAR PUMP**Aim:**

To create various parts of gear pump and assemble them by using standard CAD software Pro-E

System configuration:

Ram	: 8 GB
Processor	: Core 2 Quad / Core 2 Duo
Operating system	: Windows 7
Software	: Pro-E Wildfire 4.0

Description:

It is used to supply lubricating oil from the sump to the required places of an internal combustion engine, under pressure and is positive displacement type. Figure shows the details of a rotary gear pump.

The two gaskets 4 are placed; one on the end cover 3 and the other on the mounting bracket 1 in position. Two each of bushes 8 are placed on the end cover and mounting bracket to provide bearing surfaces for the gear shafts. End cover is connected to the pump body 2 after placing the dowel pins in the body. The shaft with gear 6 is placed in the bottom recess, whereas the shaft with gear 5 is located in the top recess of the end cover and pump body assembly. Finally, the mounting bracket is connected to the pump body after locating the dowels in the pump body. The gland 7 is placed on the mounting bracket, after locating the packing 9 between the mounting bracket and the shaft with gear 5. The end cover, pump body and the mounting bracket are connected or fastened, using bolts and nuts (not shown). Similarly, the gland is fastened to the mounting bracket by using two studs and nuts (not shown). At the end, oil inlet and outlet connections are made through the mounting bracket.

Commands used:**Part diagram:**

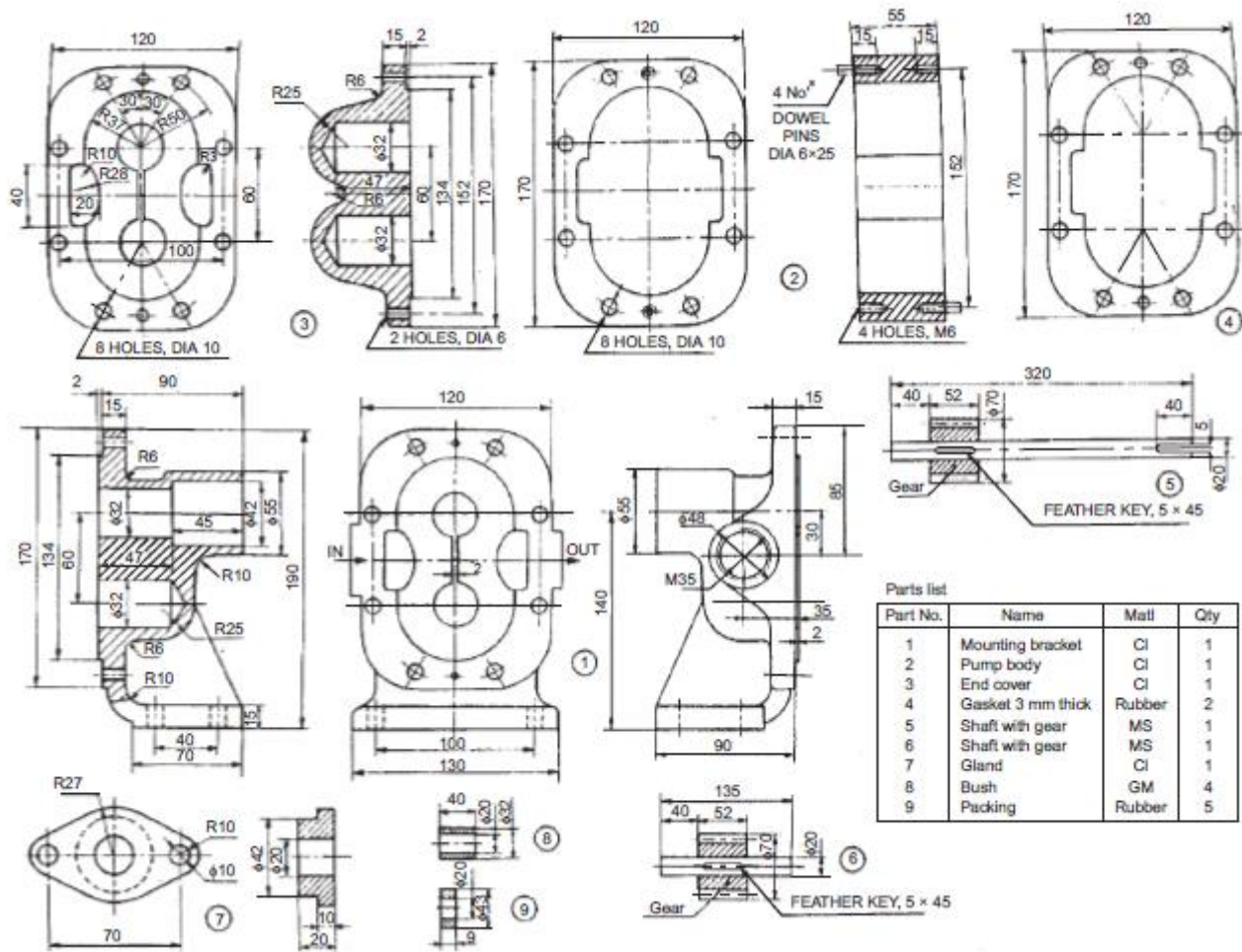
Extrude → Sketch → Sketch to dimension → Ok

Hole → Sketch → given dimension → Ok

Revolve → Sketch plane → Sketches → Centerlines → Sketch to dimension → Ok

Assembly:

Assembly → Set default → Assembly according to alignment → fully constrained → Ok



Details of Gear Pump

Result:

Thus the given various part of gear pump had been created and assembled by using standard CAD software Pro-E.

Outcome:

From this experiment, students will be able to assemble the given rotary gear pump using standard CAD software Pro-E which is used in IC engines.

Application:

1. Internal Combustion Engines

Viva-voce

1. What is the purpose of gear pump?
2. Name the types of gear pumps.
3. Name the components of gear pump.
4. What is meant by vane pump?
5. Give the applications of gear pump.

Expt. No.23**INTRODUCTION TO LS-DYNA**

LS-DYNA is a highly advanced general purpose nonlinear finite element program that is capable of simulating complex real world problems. The distributed and shared memory solver provides very short turnaround times on desktop computers and clusters operated using Linux, Windows, and UNIX. With LS-DYNA, Livermore Software Technology Corporation (LSTC) aims to provide methods to seamlessly solve problems requiring

Multi-Physics

Multi-Processing

Multiple Stages

Multi-Scale

LS-DYNA is suitable to investigate phenomena involving large deformations, sophisticated material models and complex contact conditions for structural dynamics problems. LS-DYNA allows switching between explicit and different implicit time stepping schemes. Disparate disciplines, such as coupled thermal analyses, Computational Fluid Dynamics (CFD), fluid-structure interaction, Smooth Particle Hydrodynamics (SPH), Element Free Galerkin (EFG), Corpuscular Method (CPM), and the Boundary Element Method (BEM) can be combined with structural dynamics.

By determining product characteristics before a prototype is built, for many products LS-DYNA is the key to reducing time to market. Carrying out investigations with the aid of LS-DYNA supports the design of robust products with superior performance. For pre- and post-processing, LS-DYNA comes with the LS-PrePost tool. LS-PrePost can be utilized to generate inputs and visualize numerical results. The software package LS-OPT for optimization and robust design is also supplied with LS-DYNA. With the option of multidisciplinary simulations, LS-DYNA significantly increases potentials for developing innovative products. These advantages contribute towards reducing development costs. All above-mentioned features and software packages are supplied as a single unit. LS-DYNA is not split for special applications, and the licensing scheme enables the different disciplines to be combined without limitations.

LS-DYNA has been developed in California for more than 20 years. It is the most frequently used code for many complex applications in structural nonlinear dynamics. Its usage is growing rapidly due to LS-DYNA's flexibility, enabling it to be applied to new disciplines. The new developments are driven in co-operation with leading universities from all over the world and new requirements requested by the vast customer base.

Analysis capabilities:

Different applications utilize one or a combination of the features listed below:

1. Nonlinear dynamics

2. Coupled rigid body dynamics
3. Quasi-static simulations
4. Normal modes
5. Linear and nonlinear statics
6. Eigen value analysis
7. Thermal analysis
8. Fluid analysis
9. Eulerian capabilities
10. Arbitrary LagrangianEulerian (ALE)
11. Fluid-structure interactions
12. Underwater Shock Analysis coupling (USA)
13. Failure analysis
14. Crack propagation
15. Real-time acoustics
16. Multi-physics coupling
17. Structural-thermal coupling
18. Adaptive re-meshing
19. Smooth Particle Hydrodynamics (SPH)
20. Element Free Methods (EFG)
21. X-FEM
22. CSE solver
23. 2-D and 3-D formulations
24. Arbitrary rigid to deformable

Applications:

1. LS-DYNA in metal forming:

The main application of LS-DYNA in metal forming is sheet metal stamping. The incremental approach of LS-DYNA allows the user to simulate multi-stage sheet metal stamping processes with a high degree of accuracy. The multiple core technology implemented enables even large components with very high accuracy requirements to be simulated within just one hour. In addition, the simulation of the forming process can be complemented by simulating the trimming and spring back of the part.

The simulation may have different targets. One of them could be to determine the feasibility of a part in a forming process and its final geometry on completion of the various manufacturing processes. The information gained

enables process parameters, forming sequences and optimal tool geometries to be determined. As a result, the part can be manufactured with a more accurate shape and better surface quality using fewer forming steps, thus lowering the cost per part.

Another target could be to design a hot forming process. The program allows the user to determine metal phase transformations encountered during cooling. Heating due to heat flow and radiation before, during, and after the forming process can all be analyzed. Thus, an analysis of the entire process can be carried out, from heating through forming right up to cooling, and only one model is required to predict the time needed for heating and cooling, the press requirements and part performance.

In addition to sheet metal forming, LS-DYNA is also capable of effectively analyzing other forming applications such as tube forming, cutting, extruding, impulse forming, forging, rolling, welding, hemming, flanging, electromagnetic forming and bending. Different disciplines are coupled for many of these applications. Features such as re-meshing, meshless methods, switching time-stepping schemes, ALE, thermal capacities, rigid body dynamics, among others, can be used simultaneously.

Many specific sheet metal stamping features come with LS-DYNA and the pre- and post-processing tool LS-Post, which is included in the software package. The LS-DYNA solver has also been integrated into various forming simulation tools. These tools are provided by third party companies which supply highly effective support for specific forming applications and localization. A one step solver is currently under preparation and will soon be available in the standard LS-DYNA version.

2. LS-DYNA – a multi-purpose program for automotive suppliers:

LS-DYNA is essential for the virtual testing of various components in vehicles. The explicit and implicit time stepping schemes are capable of simulating static and dynamic tests using the same model. Component manufacturing processes are investigated by LS-DYNA using the metal forming and thermal capabilities. Hence, only one model is required to consider different problems. Ultimately, this results in lower costs for training and model creation compared to other solutions.

One example of the successful application of LS-DYNA is seat design. Seat manufacturers are able to consider static and dynamic load cases for seat frames as well as analyze the stability of belt anchor age points. LS-DYNA enables them to determine maximal locking mechanism loads or failure loads of seat tracks. LS-DYNA allows to investigate the influence of the seat on an occupant in a crash as well as the stamping process of a gear wheel. To enhance design and find a robust solution, this user group also often utilizes LS-OPT, a state-of-the-art optimization tool.

Other examples of similar beneficial applications of LS-DYNA are the design and manufacture of crash boxes, bumpers, front ends, dashboards, trimmings, and tires.

3. LS-DYNA in aerospace and defense:

LS-DYNA is a state-of-the-art program which can simulate high speed impacts, blasts and explosions. ALE and SPH methods are well suited for investigating high speed impacts on textiles, metal sheets and composites. The large library of constitutive equations with multiple options for material failure and non-localization complete the features required for many defense and aerospace applications.

Additionally, 2-D capabilities and automatic re-meshing and rezoning enable users to investigate axi-symmetric problems. The multi-physics capabilities of LS-DYNA, in conjunction with features developed for the automotive industry, facilitate the investigation of splashdown loads on tanks and rockets and emergency airplane landings. These features can also be used to optimize the design of airplane turbines and their blades with regard to bird collision.

4. LS-DYNA for drop test analysis:

LS-DYNA is used to investigate the behavior of products under impact conditions due to dropping. The application range includes consumer products, tools and also container design. For example, in the field of packaging design LS-DYNA helps to develop food containers capable of sustaining dynamic loading conditions during transport and storage.

Besides the broad spectrum of material models equipped with complex failure mechanisms, the flexible coupling and switching capabilities of LS-DYNA are essential for many applications. For instance, a liquid in a container can be modeled with the ALE or SPH method coupled with the structure. This allows the behavior of the liquid to be modeled accurately during impact.

To investigate cracks, the Element Free Galerkin (EFG) method can be used to eliminate mesh influence during crack propagation. To determine steady state deformation effectively, LS-DYNA provides the flexibility to switch time-stepping scheme arbitrarily between explicit and implicit. Furthermore, LS-DYNA also enables parts to be altered from rigid to deformable and vice versa. This feature is often used to determine the position of one part in relation to another during a falling phase before the main impact occurs.

5. LS-DYNA for containment:

In certain industries, there is a risk of accidents causing severe damage to communities or the environment. LS-DYNA acts as a tool to reduce such risks by generating knowledge about how a system may fail. Thus, design changes can be made to reduce or even eliminate risks associated with the load cases considered.

One example is containers utilized to transport nuclear fuel elements. It is essential that containers remain closed and tightly sealed in the case of any predictable accident which might occur during transport. LS-DYNA is used to design the transport container, its interior and the energy-absorbing buffers around the enclosing hull of the container.

Another example is the high-speed impact on objects, like a turbine blade separating from a turbine that may not damage the embankment dam. LS-DYNA allows the user to estimate the damage caused by a turbine blade hitting the turbine housing. As such types of simulation require the exact prediction of the post-failing phase, extensive preliminary material testing is needed. However, due to the high potential of severe damage, this effort is well spent.

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Project Work

1. Design and Analysis of single composite lap joint
2. Design and Analysis of composite of leaf spring
3. Design and Analysis of unidirectional fibre orientation of under transverse static loading
4. Design and Analysis of unidirectional fibre orientation of bearing
5. Analysis of hybrid composite leaf spring using transverse static loading
6. Analysis of steel roof using composite material

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