MACHINE DRAWING THROUGH CAD

LAB MANUAL

Year	:	2018 - 2019
Subject Code	:	AME105
Regulations	:	IARE-R16
Class	:	III Semester
Branch	:	ME

Prepared By

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Mr. M. Sunil Kumar, Assistant Professor

MECHANICAL ENGINEERING

INSTITUTE OF AERONAUTICAL ENGINEERING

(Autonomous)

Dundigal, Hyderabad - 500 043

INSTITUTE OF AERONAUTICAL ENGINEERING

(Autonomous)

Dundigal, Hyderabad – 500 043, Telangana

Certificate

This is to certify that Mr. / Ms. _______________ *bearing roll no*__________*of B. Tech*______*semester* _________________________________ *branch has satisfactorily* completed___________laboratory during the academic year______.

Signature of HOD

Signature of Faculty

Signature of Internal Examiner

Signature of External Examiner

Program Outcomes	
PO1	**Engineering Knowledge:** Capability to apply knowledge of Mathematics, Science Engineering in the field of Mechanical Engineering
PO2	**Problem Analysis:** An ability to analyze complex engineering problems to arrive at relevant conclusion using knowledge of Mathematics, Science and Engineering.
PO3	**Design/ Development of solution:** Competence to design a system, component or process to meet societal needs within realistic constants.
PO4	**Conduct investigation of complex problems:** To design and conduct research oriented experiments as well as to analyze and implement data using research methodologies.
PO5	**Modern Tool usage:** An ability to formulate, solve complex engineering problems using modern engineering and information technology tools.
PO6	**The Engineer society:** To utilize the engineering practices, techniques, skills to meet needs of health, safety legal, cultural and societal issues.
PO7	**Environment and Sustainability:** To understand the impact of engineering solution in the societal context and demonstrate the knowledge for sustainable development.
PO8	**Ethics:** An understanding and implementation of professional and Ethical responsibilities.
PO9	**Individual Team work:** To function as an effective individual and as a member or leader in multi-disciplinary environment and adopt in diverse teams.
PO10	**Communication:** An ability to assimilate, comprehends, communicate, give and receive instructions to present effectively with engineering community and society.
PO11	**Project Management and Finance:** An ability to provide leadership in managing complex engineering project at multi-disciplinary environment and to become a professional engineer.
PO12	**Life-Long learning:** Recognition of the need and an ability to engage in lifelong learning to keep abreast with technological changes.
Program Specific Outcomes	
PSO1	**Professional Skills:** To produce engineering professional capable of synthesizing and analyzing mechanical system including allied engineering streams.
PSO2	**Design/ Analysis:** An ability to adapt and integrate current technologies in the design and manufacturing domain to enhance the employability.
PSO3	**Successful Career and Entrepreneurship:** To build the nation by imparting technological inputs and managerial skills to become a technocrats.

ATTAINMENT OF PROGRAM OUTCOMES AND PROGRAM SPECIFIC OUTCOMES

Week No.	Experiment	Program Outcomes Attained	Program Specific Outcomes Attained
1	Conventional representation	PO1, PO2, PO3, PO5	PSO1, PSO2
2	Sectional views	PO1, PO2, PO3, PO5	PSO1, PSO2
3	Dimensioning	PO1, PO2, PO3, PO5	PSO1, PSO2
4	Working drawings	PO1, PO2, PO3, PO5	PSO1, PSO2
5	Machine elements	PO1, PO2, PO3, PO5	PSO1, PSO2
6	Keys and cotter joints	PO1, PO2, PO3, PO5	PSO1, PSO2
7	Riveted joints	PO1, PO2, PO3, PO5	PSO1, PSO2
8	Couplings	PO1, PO2, PO3, PO5	PSO1, PSO2
9	Bearings	PO1, PO2, PO3, PO5	PSO1, PSO2
10	Assembly drawings-Engine parts : Stuffing box	PO1, PO2, PO3, PO5	PSO1, PSO2
11	Assembly drawings : Connecting rod and eccentric	PO1, PO2, PO3, PO5	PSO1, PSO2
12	Assembly drawings: Screw jack	PO1, PO2, PO3, PO5	PSO1, PSO2
13	Assembly drawings: Machine vice and tailstock	PO1, PO2, PO3, PO5	PSO1, PSO2
14	Assembly drawings Rams-bottom Safety Valve, feed check valve	PO1, PO2, PO3, PO5	PSO1, PSO2

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Signature of Faculty – in-charge

MACHINE DRAWING THROUGH CAD LABORATORY

III Semester: ME									
Course Code	**Category**	**Hours / Week**			**Credits**	**Maximum Marks**		
AME105	**Core**	**L**	**T**	**P**	**C**	**CIA**	**SEE**	**Total**
		-	**-**	**3**	**2**	**30**	**70**	**100**
Contact Classes: Nil	**Tutorial Classes: Nil**	**Practical Classes: 42**				**Total Classes: 42**		

OBJECTIVES:
The course should enable students to

I. Understand Code of drawing practice as per BIS conventions for mechanical elements using AutoCAD.
II. Practice the drawing methods for sectioning of joints, couplings, bearings, keys.
III. Prepare assembly drawings, sectional views and bill of materials for selected assemblies.

LIST OF EXERCISES

Week-1	**CONVENTIONAL REPRESENTATION**
Conventional representation of materials, common machine elements and parts such as screws, nuts, bolts, keys, gears, webs and ribs; Introduction to AutoCAD.	
Week-2	**SECTIONAL VIEWS**
Types of sections, selection of section planes and drawing of sections and auxiliary sectional views, parts not usually sectioned.	
Week-3	**DIMENSIONING**
Methods of dimensioning, general rules for sizes, and placement of dimensions for holes, centers, and curved and tapered features.	
Week-4	**WORKING DRAWINGS**
Types of drawings–working drawings for machine parts.	
Week-5	**MACHINE ELEMENTS**
Drawing of machine elements and simple parts; Selection of orthogonal views and additional views for the following machine elements and parts with every drawing proportion, popular forms of screw threads, bolts, nuts, stud bolts.	
Week-6	**KEYS AND COTTER JOINTS**
Keys, cotter joints, and knuckle joint.	
Week-7	**RIVETED JOINTS**
Riveted joints for plates.	
Week-8	**COUPLINGS**
Shaft coupling and spigot joint.	
Week-9	**BEARINGS**
Journal, pivot, and collar bearing.	
Week-10	**ASSEMBLY DRAWINGS-ENGINE PARTS**

Assembly drawings for the following, using conventions and parametric drawing practice:
Engine parts, stuffing box.

Week-11	CONNECTING ROD AND ECCENTRIC

Eccentrics, I.C. engine connecting rod.

WeeK-12	SCREW JACK

Screw jack.

Week-13	TAIL STOCK AND MACHINE VICE

Machine vice and tailstock.

Week-14	SAFETY VALVES

Rams-bottom Safety Valve, feed check valve.

Text Books:

1. K.L. Narayana, P. Kannaiah, K. Venkata Reddy, "Machine Drawing", New Age Publishers, 3rd Edition, 2012.
2. K.C. John, "Text book of Machine Drawing'', PHI Eastern Economy, 1st Edition, 2010.
3. P.S Gill, "Machine Drawing'', S.K Kataria & Sons, 1st Edition, 2013.
4. Junnarkar N.D, "Machine Drawing'', Pearson Edu, 1st Edition, 2007.
5. Basudeb Bhattacharya, "Machine Drawing'', Oxford University Press, 1st Edition, 2011.
6. N. D. Bhatt, V. M Pancahal, "Machine Drawing'', Charotar, 2014.
7. R. K. Dhavan, "A Text book of Machine Drawing'', S.Chand Publication & Co, New Delhi, 2nd Edition, 2008.

Web References:

1. http://web.iitd.ac.in/~achawla/public_html/201/sheets/sheet5/sheet5.pdf
2. https://drive.google.com/file/d/0B_GCh7LMfHf6Z0VNWTNHU3pMSTg/view?pref=2&pli=1
3. http://www.uiet.co.in/downloads/20140911122818-Machine20Drawing.pdf
4. http://listpdf.com/ma/machine-drawing-book-pdf.html

Course Home Page:

SOFTWARE AND HARDWARE REQUIREMENTS FOR A BATCH OF 36 STUDENTS:

SOFTWARE: System Software: Microsoft Windows 7.
Application Software: AutoCAD.

HARDWARE: 36 numbers of Desktop Computer Systems

DESIGNATION AND RELATIVE POSITIONS OF VIEWS

An object positioned in space may be imagined as surrounded by six mutually perpendicular planes. So, for any object, six different views may be obtained by viewing at it along the six directions, normal to these planes. Figure 1.0 shows an object with six possible directions to obtain the different views which are designated as follows:

View in the direction **a** = view from the front

View in the direction **b** = view from above

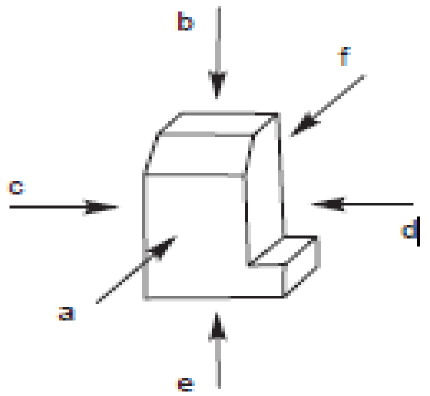


Fig. 1.0

View in the direction c = view from the left

View in the direction d = view from the right

View in the direction e = view from below

View in the direction f = view from the rear

Figure 1.1 *a* shows the relative positions of the above six views in the first angle projection and Fig.1.1 *b*, the distinguishing symbol of this method of projection. Figure 1.2 *a* shows the relative position of the views in the third angle projection and Fig. 1.2*b*, the distinguishing symbol of this method of projection

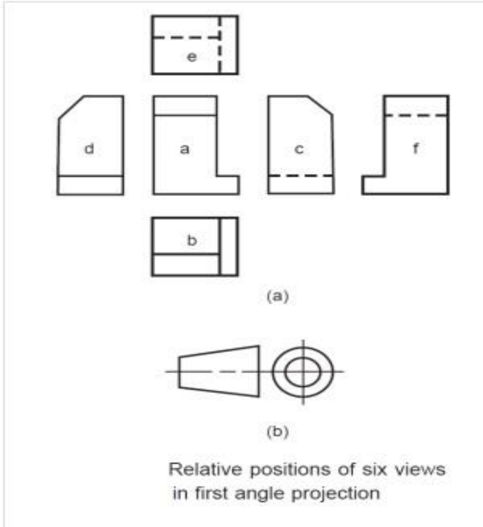


Fig. 1.1

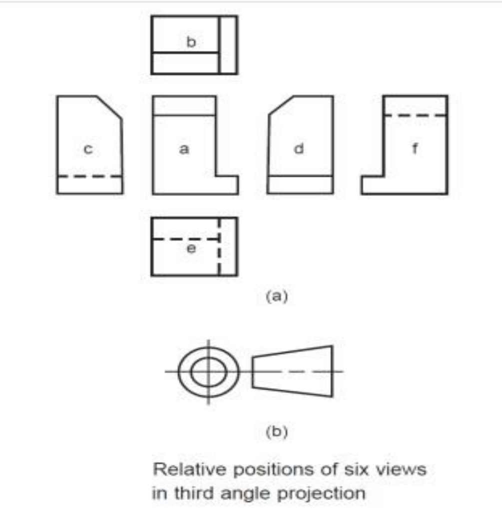


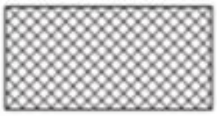
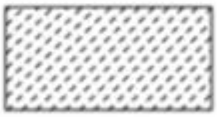
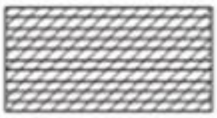
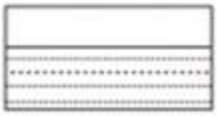
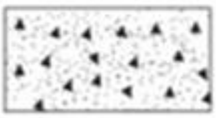
Fig. 1.2

TYPES OF LINES

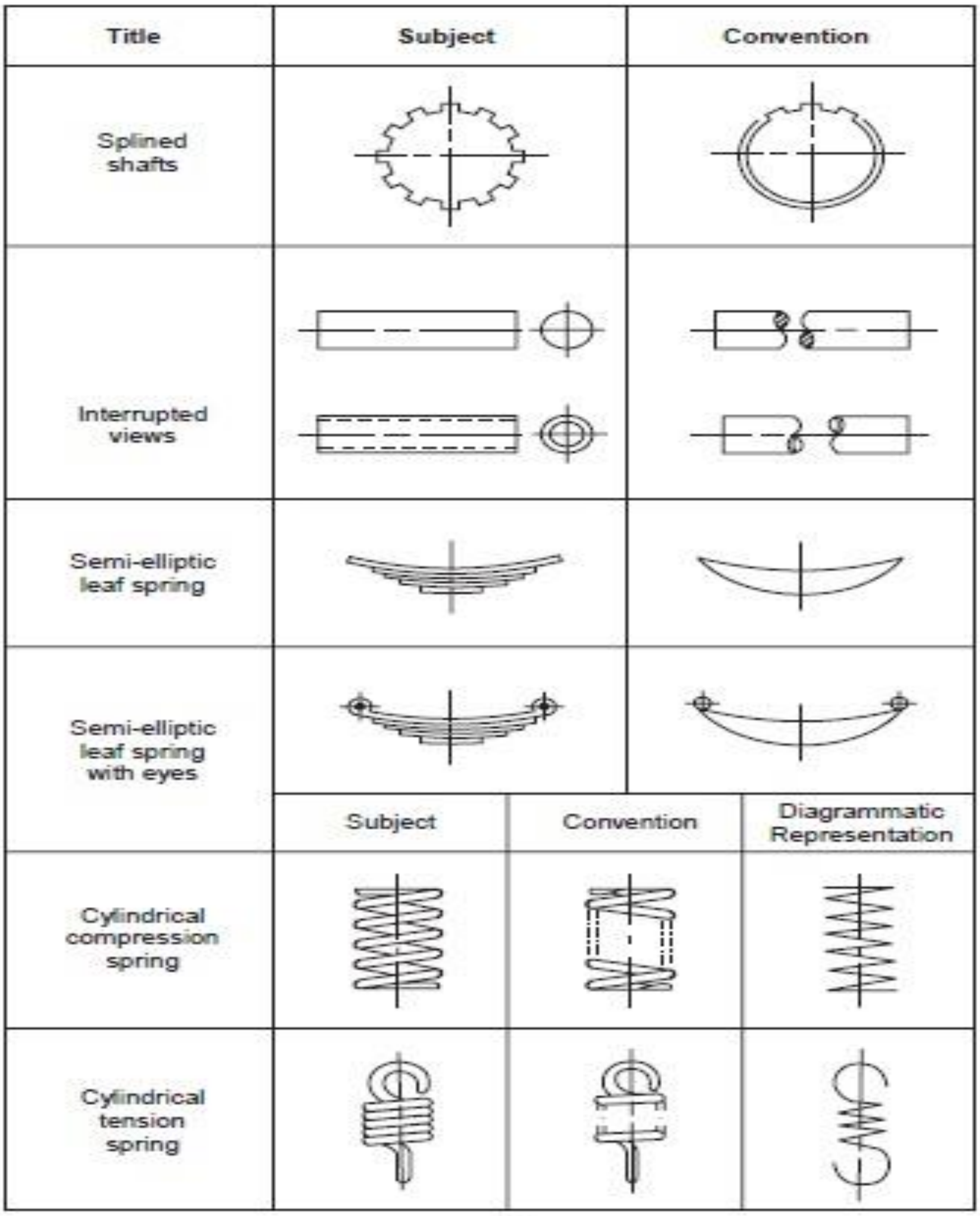
Illustration	Application
Thick	Outlines, visible edges, surface boundaries of objects, margin lines
Continuous thin	Dimension lines, extension lines, section lines leader or pointer lines, construction lines, boarder lines
Continuous thin wavy	Short break lines or irregular boundary lines – drawn freehand
Continuous thin with zig-zag	Long break lines
Short dashes, gap 1, length 3 mm	Invisible or interior surfaces
Short dashes	Center lines, locus lines Alternate long and short dashes in a proportion of 6:1,
Long chain thick at end and thin elsewhere	Cutting plane lines

WEEK-I - CONVENTIONAL REPRESENTATION

Exercise No. 1 Conventional Representation of Materials

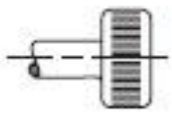
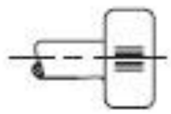
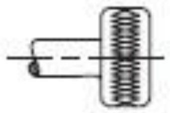
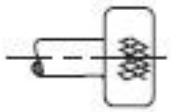
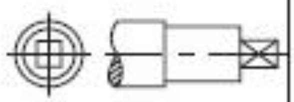
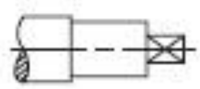
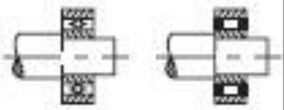
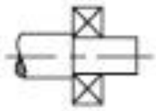
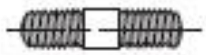
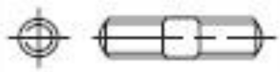
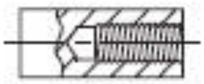
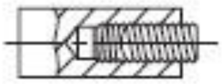
Type	Convention	Material
Metals		Steel, Cast Iron, Copper and its Alloys, Aluminium and its Alloys, etc.
		Lead, Zinc, Tin, White-metal, etc.
Glass		Glass
Packing and Insulating material		Porcelain, Stoneware, Marble, Slate, etc.
		Asbestos, Fibre, Felt, Synthetic resin products, Paper, Cork, Linoleum, Rubber, Leather, Wax, Insulating and Filling materials, etc.
Liquids		Water, Oil, Petrol, Kerosene, etc.
Wood		Wood, Plywood, etc.
Concrete		A mixture of Cement, Sand and Gravel

Exercise No. 2 Conventional Representation of Common Machine Elements



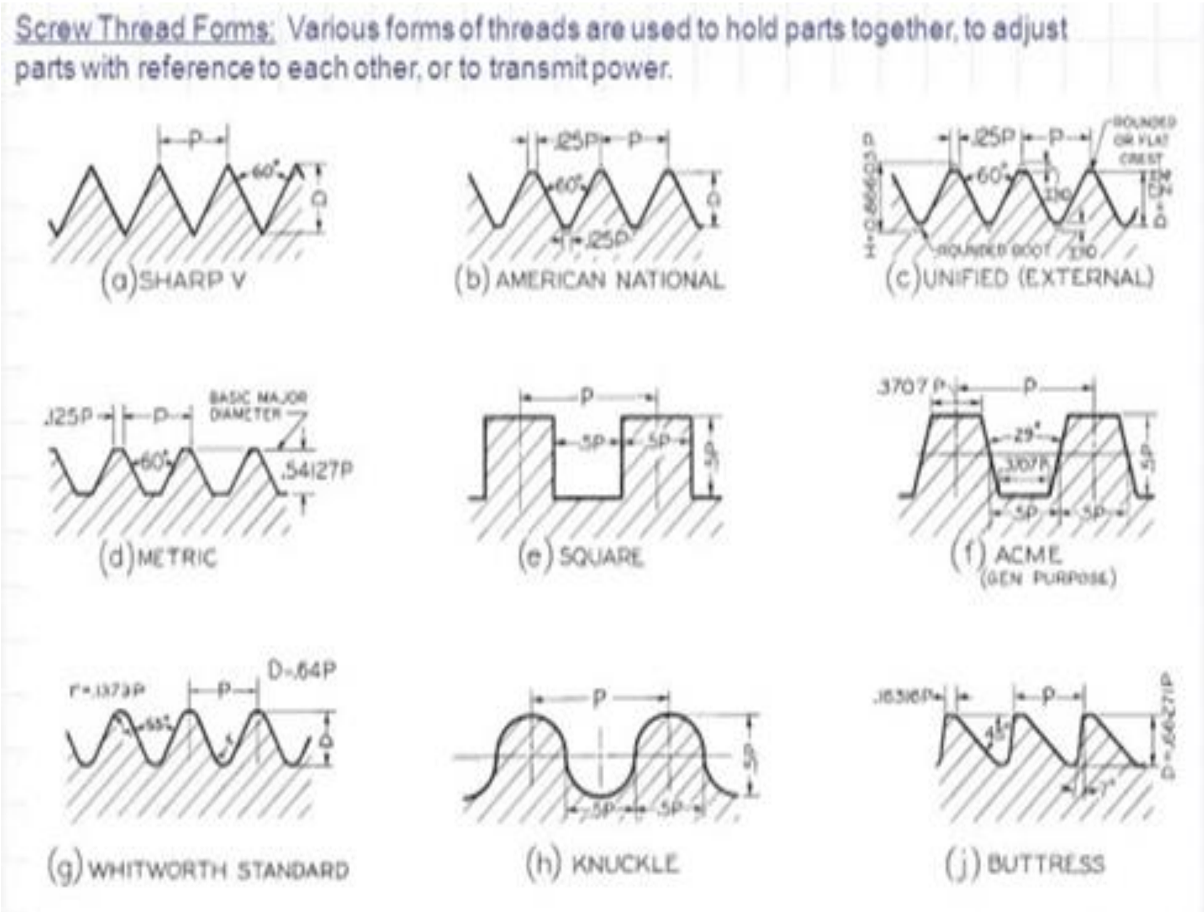
(b)

Exercise No. 3 Conventional Representation of Machine Parts

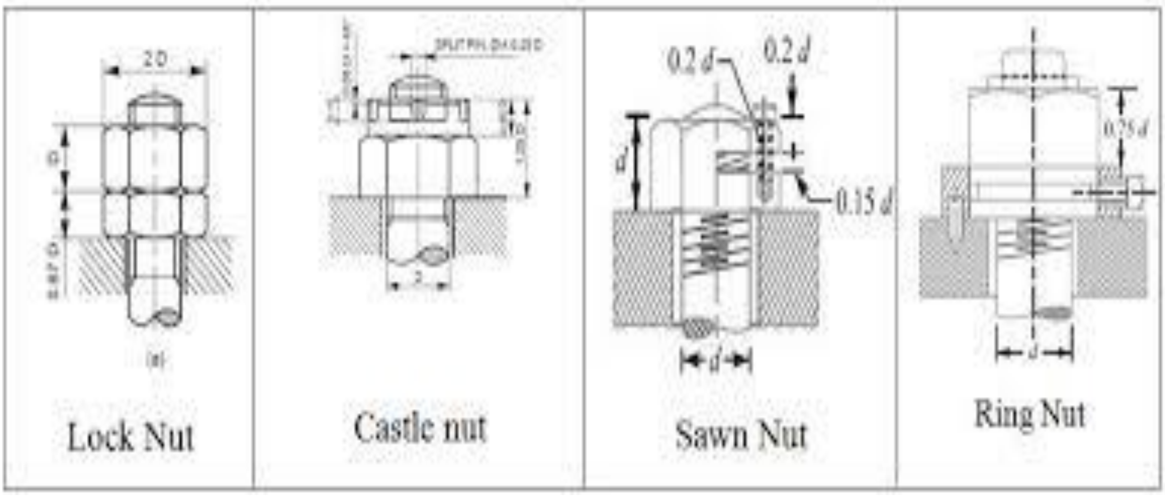
Title	Subject	Convention
Straight knurling		
Diamond knurling		
Square on shaft		
Holes on circular pitch		
Bearings		
External screw threads (Detail)		
Internal screw threads (Detail)		
Screw threads (Assembly)		

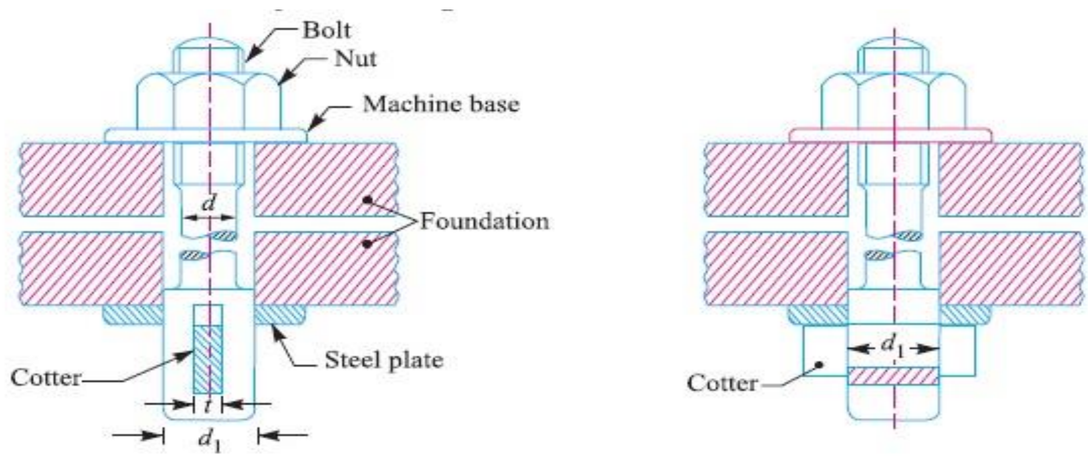
(c)

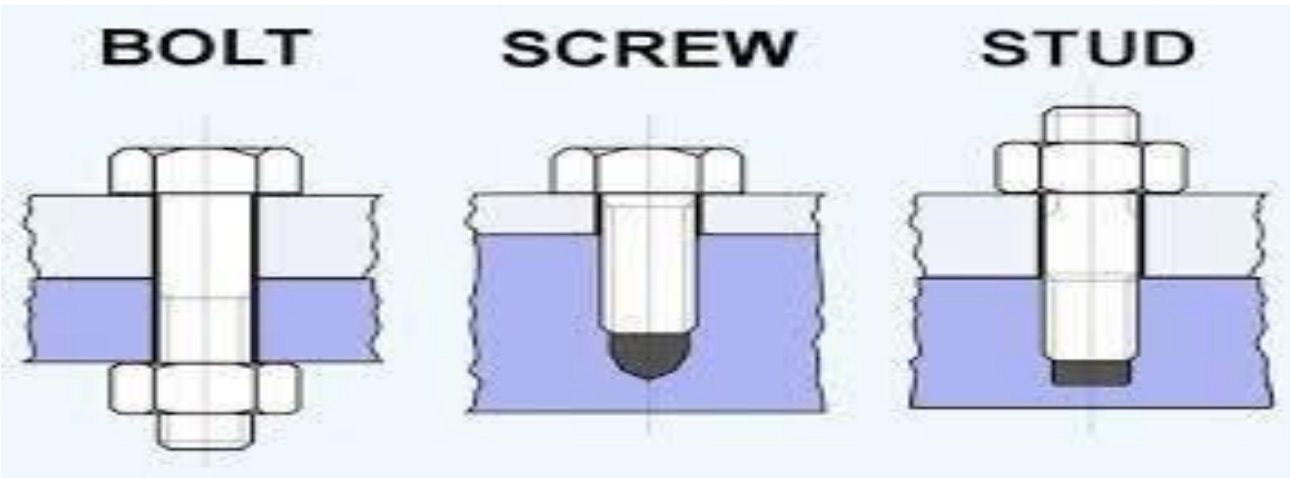
Exercise No. 4 Conventional Representation Of Screws.

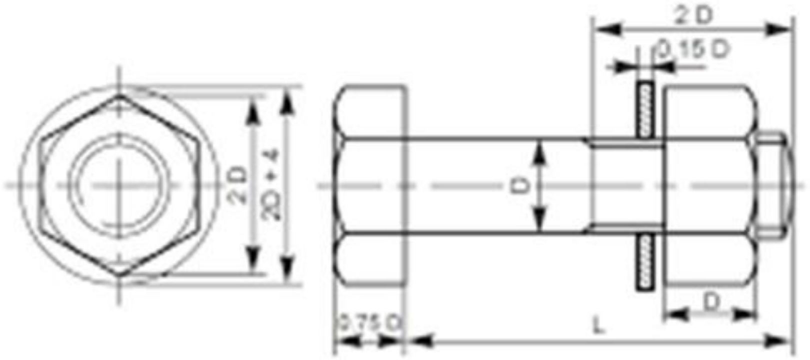


Exercise No. 5   Conventional Representation of Nuts And Bolts

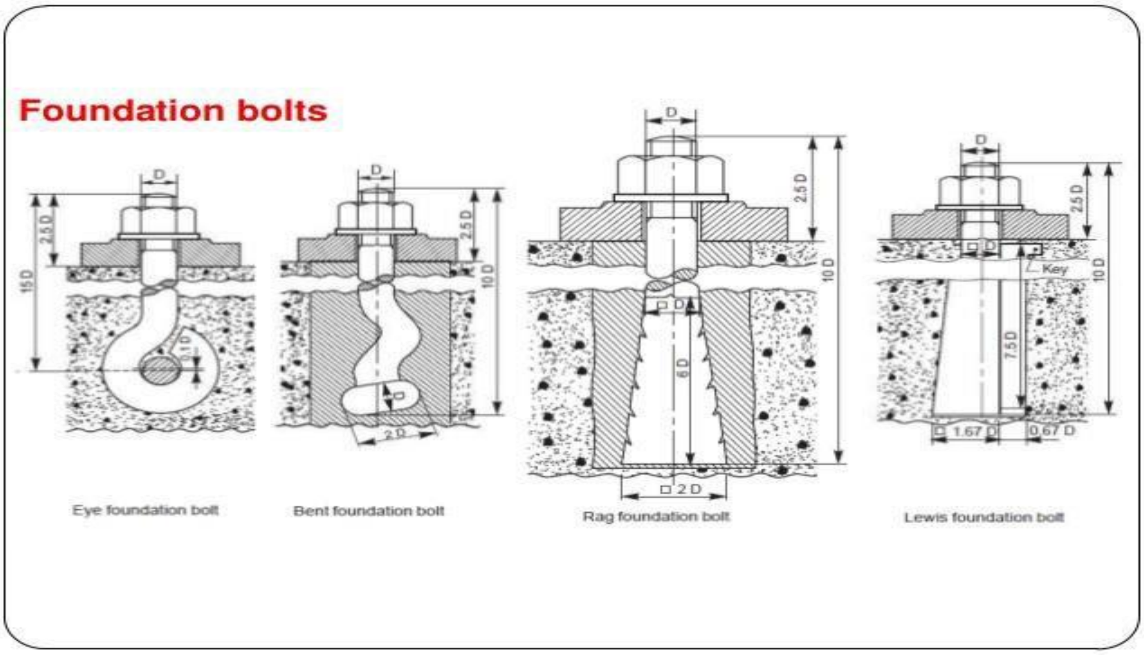




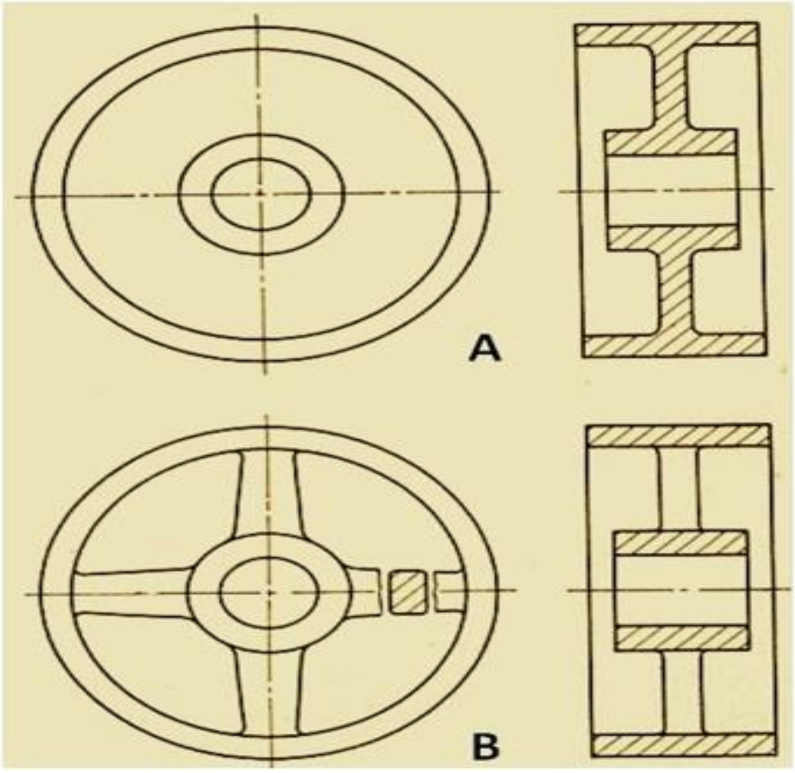




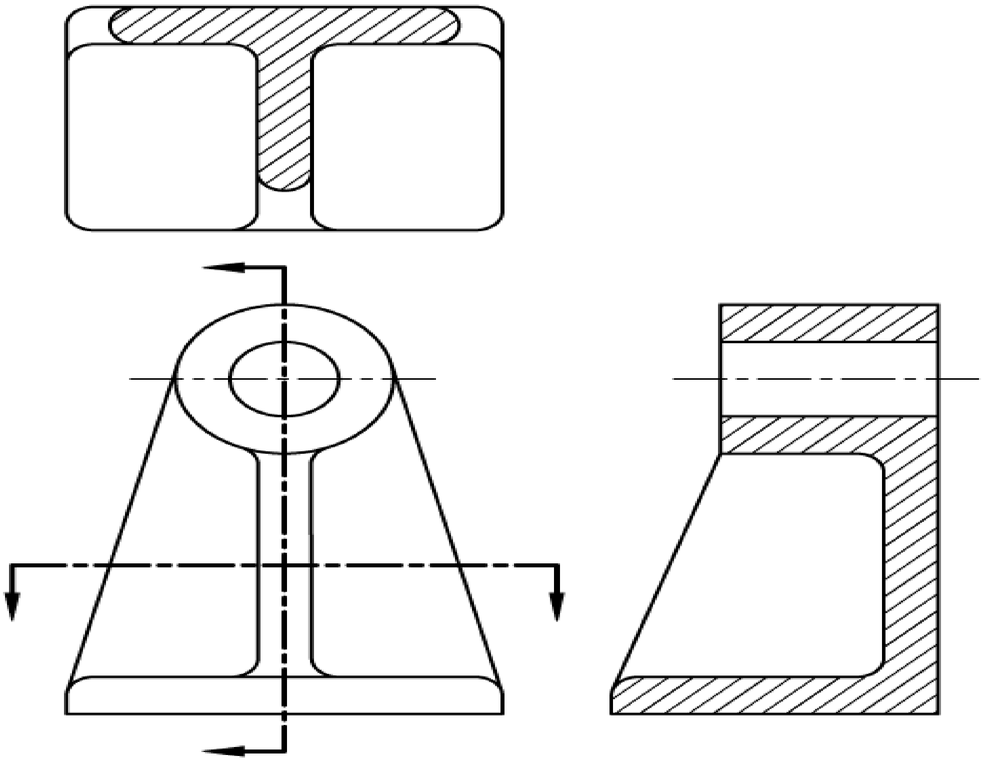
HEXAGONAL HEADED BOLT WITH NUT AND WASHER



Exercise No. 6   Conventional Representation of Webs and Ribs

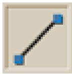
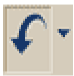
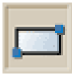
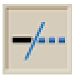
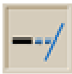


RIB

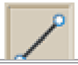
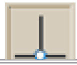
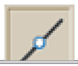
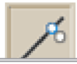
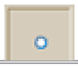
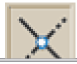
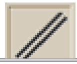
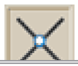
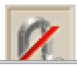
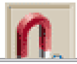
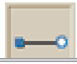
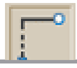


WEB

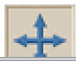
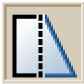
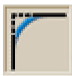
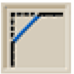
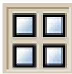
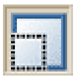
Exercise No. 7 Introduction to AutoCAD Commands

Command	Keystroke	Icon	Menu	Result
Line	**Line / L**		Draw > Line	Draw a straight line segment from one point to the next
Circle	**Circle / C**		Draw > Circle > Center, Radius	Draws a circle based on a center point and radius.
Erase	**Erase / E**		Modify > Erase	Erases an object.
Print	**Print / Plot Cntl+P**		File > Print	Enables the Print/Plot Configuration Dialog Box
Undo	**U** (Don't use 'Undo' for now)		Edit > Undo	Undoes the last command.
Rectangle	**RECTANGLE / REC**		Draw > Rectangle	Draws a rectangle after you enter one corner and then the second.
Multi Lines	**MLINE / ML**	No Icon	Draw > Multiline	Draw parallel lines based on the parameters you define.
Trim	**TRIM / TR**		Modify > Trim	Trims objects to a selected cutting edge.
Extend	**EXTEND / EX**		Modify > Extend	Extends objects to a selected boundary edge.
Offset	**OFFSET / O**		Modify > Offset	Offsets an object (parallel) by a set distance.
Object Snaps	**OSNAP / OS / F3**	CLICK OSNAP	Tools > Object Snap Settings	Brings up the OSNAP dialog box.

ICON	**SETTING**		**ICON**	**SETTING**

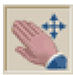
	Endpoint			Perpendicular
	Midpoint			Tangent
	Center			Nearest
	Node			Apparent Intersection
	Quadrant			Parallel
	Intersection			None
	Extension			Osnap Setting
	Insertion Point			Temporary Tracking Point
				Snap From

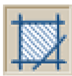
RIGHT TO LEFT	CROSSING SELECTION	GREEN (with a dotted outline	SECLECTS ANY OBJECT THAT EITHER CROSSES THE BOUNDARY OR IS INSIDE IT
LEFT TO RIGHT	WINDOW SELCTION	BLUE	SELECTS ON OBJECTS THAT ARE COMPLETELY WITHIN THE BOX

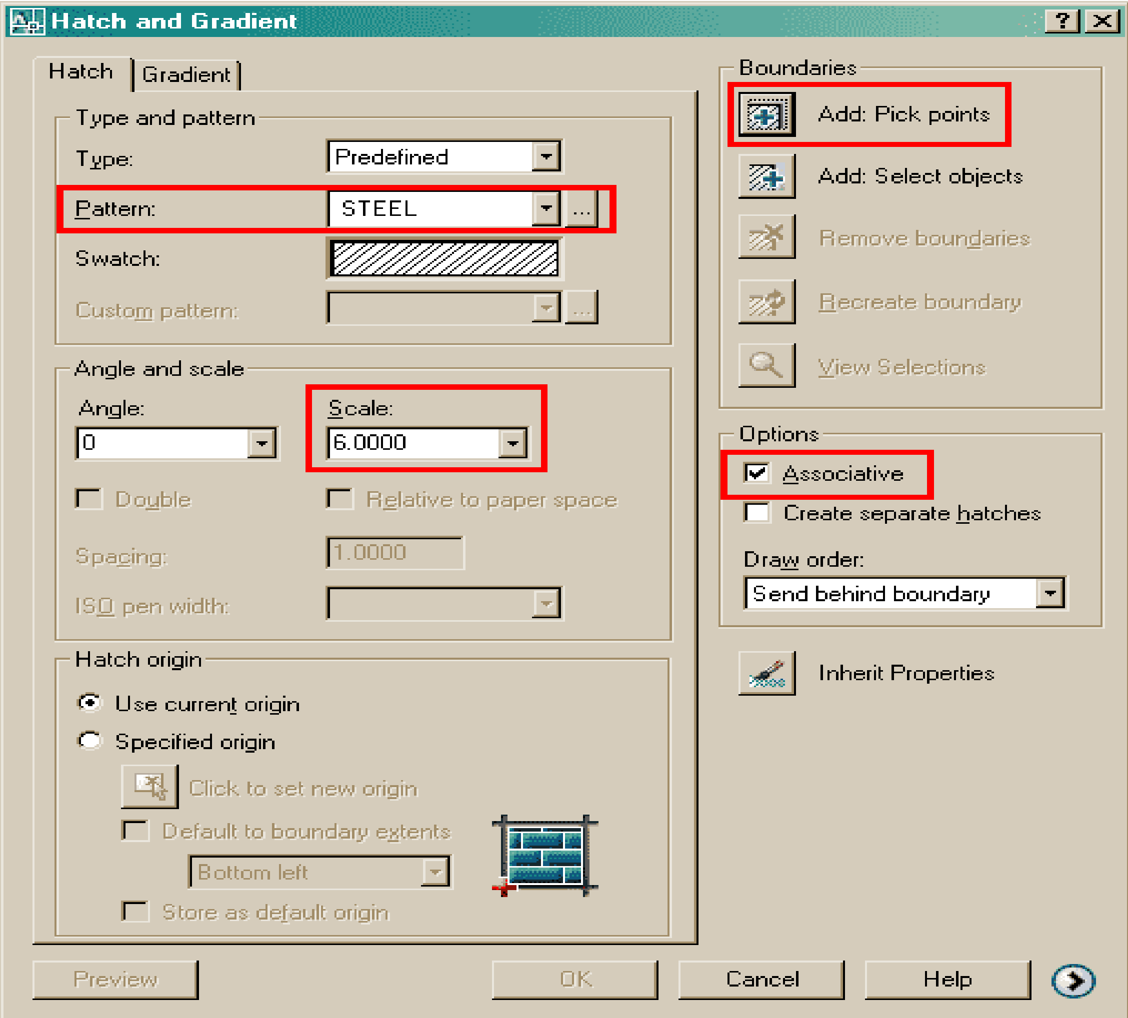
Command	Keystroke	Icon	Menu	Result
Move	**Move / M**		Modify > Move	Moves an object or objects
Copy	**Copy / CP**		Modify > Copy	Copies object(s) once or multiple times
Stretch	**Stretch / S**		Modify > Stretch	Stretches an object after you have selected a portion of it
Mirror	**Mirror / MI**		Modify > Mirror	Creates a mirror image of an object or selection set
Rotate	**Rotate / RO**		Modify > Rotate	Rotates objects to a certain angle
Fillet	**Fillet / F**		Modify > Fillet	Creates a round corner between two lines
Chamfer	**Chamfer / CHA**		Modify > Chamfer	Creates an angled corner between two lines
Array	**Array / AR**		Modify > Array	Creates a repeating pattern of the selected objects
Layer	**Layer / LA**		Format > Layer	Starts the Layer and Linetype property dialog box
Text	**Text**	No Icon	Draw > Single Line Text	Creates a single line of text
Dimension	**Dim**	Many	Dimension > (*pick one*)	Dimensions previously drawn objects
Scale	**Scale / SC**		Modify > Scale	Proportionately resizes (or scales) objects

COMMAND OPTION	ICON	DESCRIPTION
Zoom All		This option causes AutoCAD to display the whole drawing as far as its drawing limits (whichever is the greater of the two).
Zoom Center		This option requires two things: a point that is to be the center of the new display and a value to be its new height *in drawing units*. The existing height is the default for the new height to allow for panning across the drawing. If the new height value is followed by "X" (eg. 2x), then it is taken as a magnification factor relative to the current height. If followed by "XP", then it is taken as a scale factor relative to paper space and can be used for scaling the contents of paper space viewports.
Zoom Dynamic		This is a very useful ZOOM option once it is understood. It permits very quick movement around the drawing. Once selected, this option redraws the graphics area of the screen and displays two rectangles. The larger box shows the extents of the current drawing. The smaller box shows the current view with an "X" in the middle. This moves with the mouse. This view box should be positioned so that its lower left corner is at the lower left corner of the view required. By pressing the left button on the mouse, the "X" is replaced by an "> " pointing to the right side of the view box. This allows you to change the magnification. As the mouse is moved, the view box shrinks and expands so that the size of the required view can be set. The left mouse button toggles between PAN "X" and ZOOM "> " mode so that fine adjustments can be achieved. When the view required has been selected, press <ENTER> or right click to cause AutoCAD to display it.
Zoom Extents		This option will display all the graphics that are contained in the drawing (referred to as the *drawing extents*) with the largest image possible.
Zoom Previous		This option restores the displayed view prior to the current one. For the purpose of this option, up to 10 views are saved so that the last ten views can be recalled. This option includes every time you use the scroll bar, which is one reason to **avoid the scroll bars** for panning a lot in your drawing.
Zoom Scale		This is a 'hidden' default option. You do not have to type "S" to choose this option. It simply requires the entry of a number that represents a magnification factor. Note that the factor is applied to the entire drawing (as defined by the drawing's limits). Numbers less than 1 will reduce the displayed size of the drawing, while numbers greater than 1 will enlarge it. If "X" is inserted after the number (eg. 0.8x) then the factor is applied *to the current view*. If "XP" is inserted after the scale factor, then the view is scaled relative to paper space. This is useful for zooming a view within a paper space viewport to a specific scale, for example, "1/48XP" will produce a view of model space at a scale of ¼" = 1' relative to paper space.

COMMAND OPTION	ICON	DESCRIPTION
Zoom Window		This option (also a 'hidden' default) prompts the user to pick two corners of a box on the existing view in order to enlarge that area to fill the display.
Zoom Realtime		Zoom Real time provides interactive zooming capability. Pressing <ENTER> (after entering zoom) on the command line automatically places you in Real time mode. Hold the left mouse button down at the midpoint of the drawing and move the cursor vertically to the top (positive direction) of the window to zoom in up to 100% (2x magnification). Hold the left mouse button down at the midpoint of the drawing and move the cursor vertically to the bottom (negative direction) of the window to zoom out to100% (.5x magnification). *You cannot zoom out beyond the extents of the current view.* When you release the pick button, zooming stops. You can release the pick button, move the cursor to another location in the drawing, and then press the pick button again and continue zooming from that location. To exit Real time Zoom mode, press <ENTER> or (ESC).
Aerial View command: **DSVIEWER**	None	Aerial View is a zooming tool that displays a view of the drawing in a separate window so that you can quickly move to that area. If you keep the Aerial View window open as you work, you can zoom and pan without choosing a menu option or entering a command. You can change the view by creating a new view box in the Aerial View window. To zoom in to the drawing, make the view box smaller by left clicking a rectangle. To zoom out of the drawing, make the view box larger. As you zoom in or out of the drawing, a real-time view of the current zoom location is displayed in the graphics area. The screenshot shows how the view box looks. Right click in the box and you can move the box to where you want to zoom to.
Zoom Object		This option asks you to select an object or objects, then press <ENTER> and the screen will zoom to those objects only. This is great for when you want to work on object.
Zoom In		Clicking this icon will zoom in to the drawing by about 50%. This option is only available as an icon and cannot be invoked by the command line.
Zoom Out		Similar to 'Zoom In' - this icon will zoom out of your drawing and allow you to see about 50% more of your drawing space.
Mouse Scroll	-	If you have a scrolling wheel on your mouse, you can use it to zoom in and out of your drawing. Scroll towards you to zoom out and away from you to zoom in. You have the option to change the amount of zoom per wheel click with the Zoom factor system variable. Keep in mind that you will zoom in and out using your mouse location as a 'centre point'.

COMMAND OPTION	ICON	DESCRIPTION
PAN		Panning allows you to quickly move around the drawing area at the same magnification you currently have set. Type in **PAN**(or **P**) <ENTER> and a hand will appear on the screen. Left click and hold to move around your drawing.

Command	Keystroke	Icon	Menu	Result
Boundary Hatch	**Bhatch / H**		Draw > Hatch	Covers an area with a predefined pattern
Hatch Edit	**HatchEdit / HE**		Modify > Object > Hatch...	Edits an existing Hatch



WEEK-II **SECTIONAL VIEWS**

Orthographic views when carefully selected may reveal the external features of even the most complicated objects. However, there are objects with complicated interior details and when represented by hidden lines, may not effectively reveal the true interior details. This may be overcome by representing one or more of the views 'in section'.

A sectional view is obtained by imagining the object, as if cut by a cutting plane and the portion between the observer and the section plane being removed. Figure. 1 (*a)* shows an object, with the cutting plane passing through it and Fig. 1(*b)*, the two halves drawn apart, exposing the interior details.

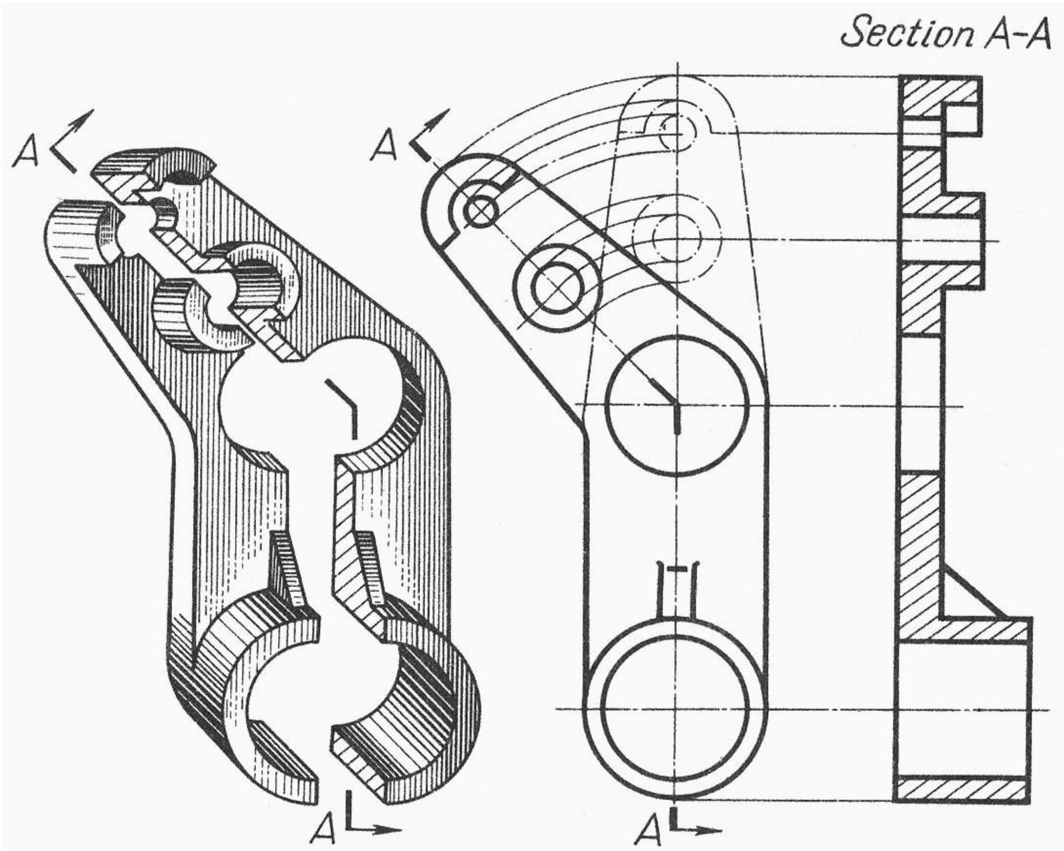


Fig 1 (a) Fig 1(b)

Fig.1.0 Principles of sectioning

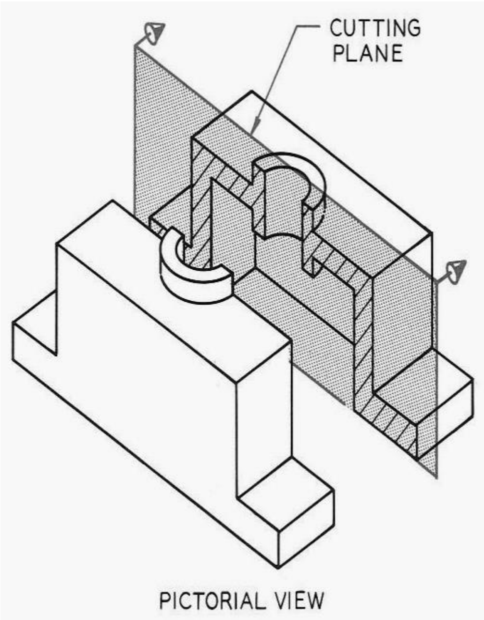
In order to show such features clearly, one or more views are drawn as if a portion had been cut away to reveal the interior This procedure is called sectioning and the view showing the cut away picture is called section view.

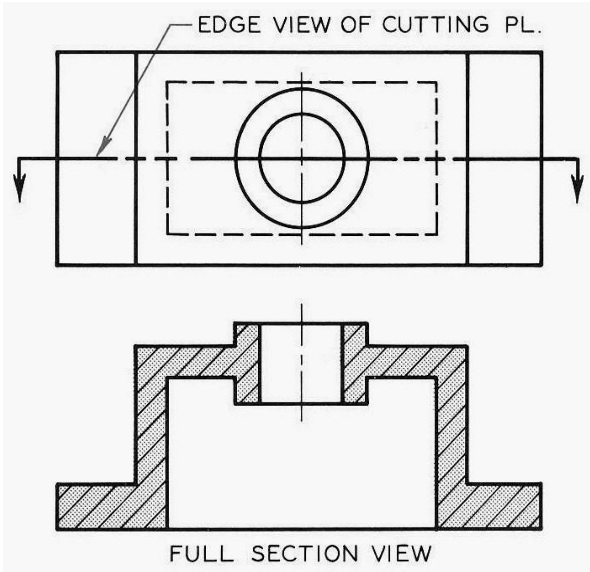
A section is an imaginary cut taken through an object to reveal the shape or interior construction.

Fig. 2a shows the imaginary cutting plane in perspective view.

The imaginary cutting plane is projected on a standard view so that the sectional view with orthographic representation is obtained as shown in Fig. 2 b .

A sectional view must show which portions of the object are solid material and which are spaces. This is done by section lining (cross-hatching) the solid parts with uniformly spaced thin lines generally at 45º.





(a) (b)

Fig. 2 sectional view

Types of Sections

Depending on the number of cutting planes, sectional views can be simple with one cutting plane (Fig. 3) or complex with two or more cutting planes (Fig.4).

If the cutting plane-line cuts entirely across the object, it is called a full section.
If the cutting plane cuts halfway through the object, it is a half section.
In addition to these, there are broken-out sections, rotated sections, removed sections, auxiliary sections, and assembly sections.

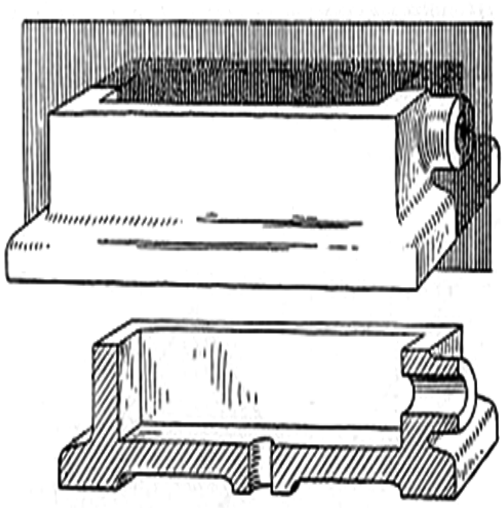


Fig.3 sectional views can be simple with one cutting plane

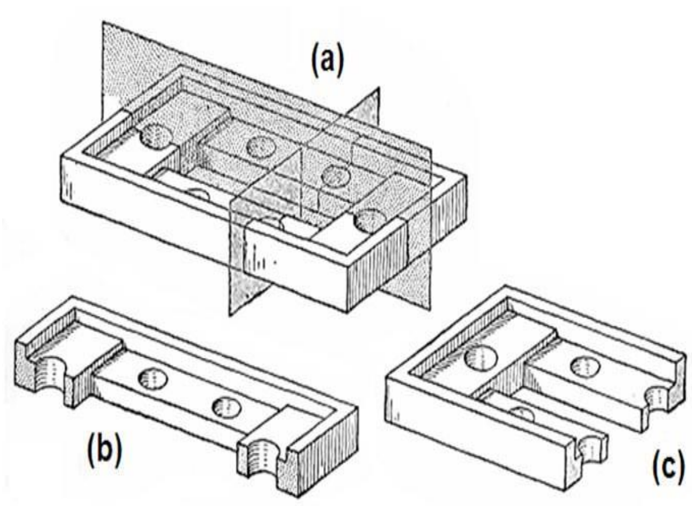


Fig.4 sectional views can be complex with two or more cutting planes

Exercise No. 8

Full Section

A sectional view obtained by assuming that the object is completely cut by a plane is called a full section or sectional view. shown in Fig. 5.1*a*, in full section. The sectioned view provides all the inner details, better than the unsectioned view with dotted lines for inner details (Fig. 5.1*b*). The cutting plane is represented by its trace (V.T) in the view from the front (Fig. 5.1c) and the direction of sight to obtain the sectional view is represented by the arrows.

When cutting plane passes fully through an object, it is called full section

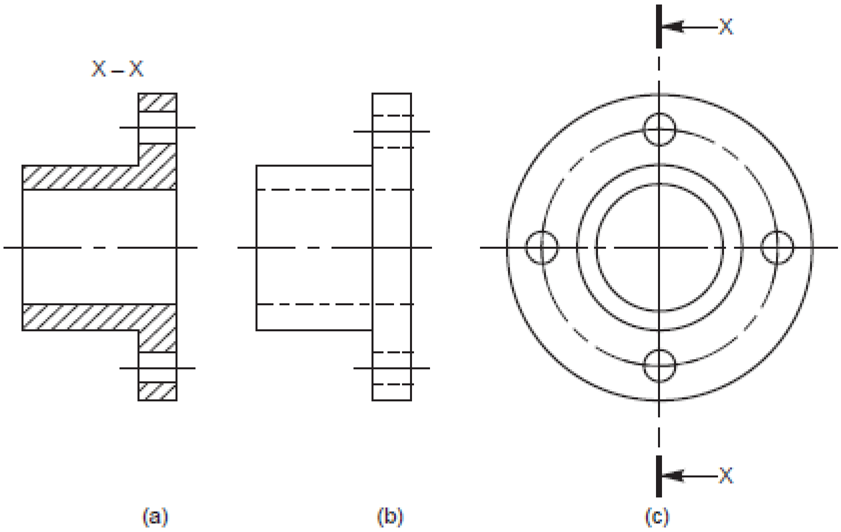


Fig. 5 Sectioned and un-sectioned views

It may be noted that, in order to obtain a sectional view, only one half of the object is imagined to be removed, but is not actually shown removed anywhere except in the sectional view. Further, in a sectional view, the portions of the object that have been cut by the plane are represented by section lining or hatching. The view should also contain the visible parts behind the cutting plane.

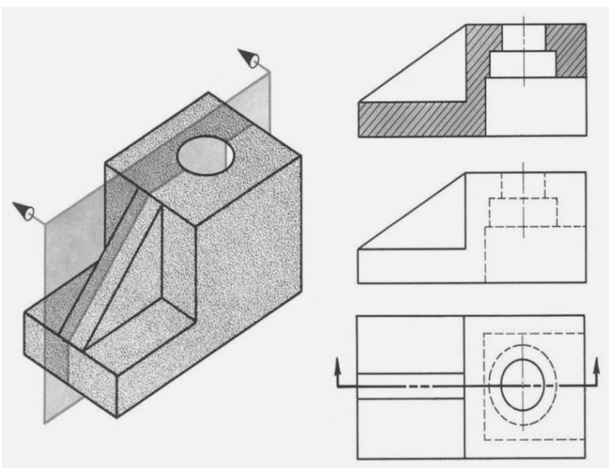


Figure.6 Full Sectional View

Exercise No. 9

Half Section

A half sectional view is preferred for symmetrical objects. For a half section, the cutting plane removes only one quarter of an object. For a symmetrical object, a half sectional view is used to indicate both interior and exterior details in the same view.

A half section is made by cutting halfway through an object (Fig. 7). Thus, one half is drawn in section and the other half is a side view, usually, hidden lines are not used (inside details are visible on the section view).

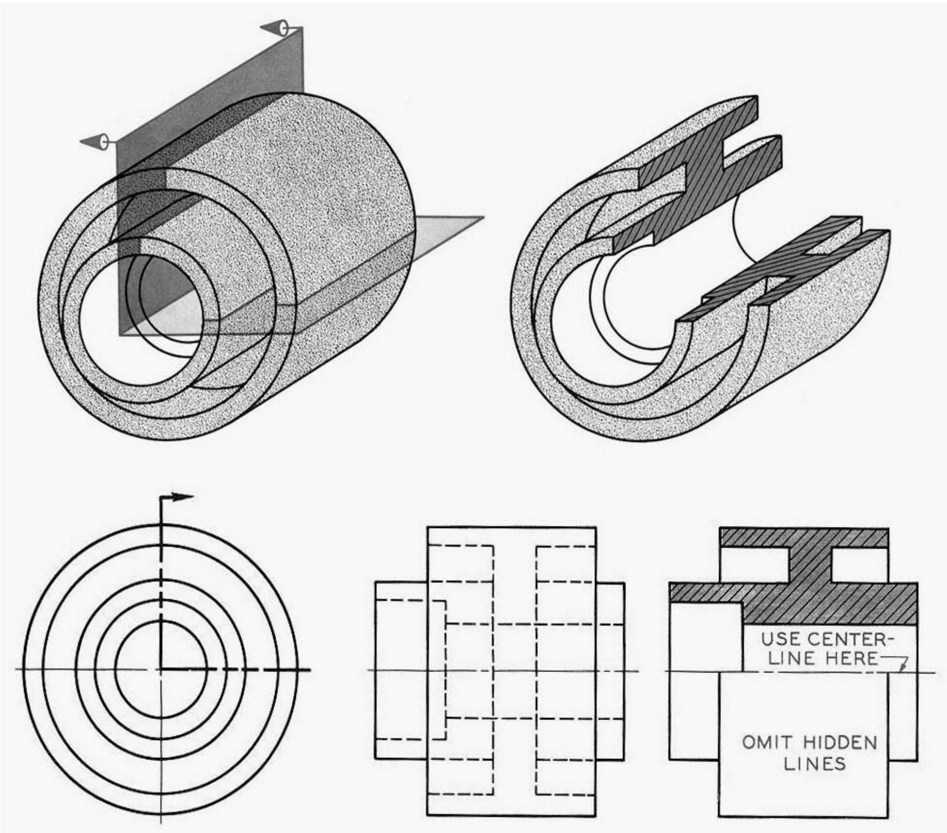


Fig. 7 Half Section

Auxiliary sections may be used to supplement the principal views used in orthographic projections. A sectional view projected on an auxiliary plane, inclined to the principal planes of projection, shows the cross-sectional shapes of features such as arms, ribs and so on. In Fig.8, auxiliary cutting plane X-X is used to obtain the auxiliary section X-X

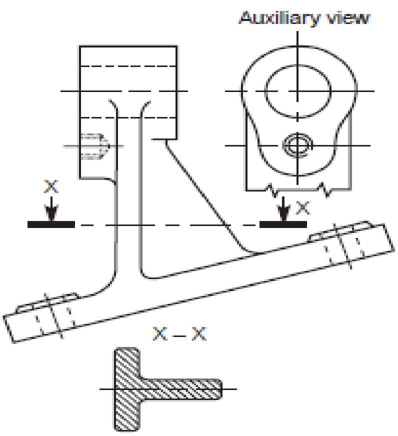


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Fig. 8 Auxiliary section

Exercise No. 10

Assembly Sections

Assembly sections consist of a combination of parts.

The purpose of an assembly section is to reveal the interior of a machine or structure so that the separate parts can be clearly shown and identified. However, the separate parts do not need to be described.

In assembly drawing, only such hidden details (as needed for part identification or dimensioning) are drawn.

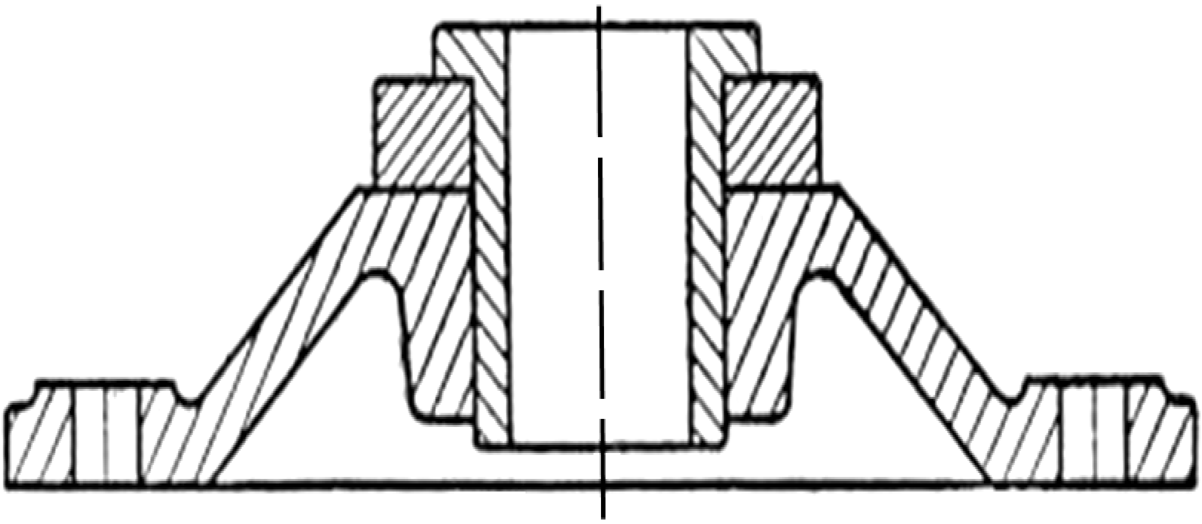


Fig. 9 Assembly section

Exercise No. 11

Hidden Edges and Surface in Section

Sections are primarily used to replace hidden lines with visible lines. As a rule, hidden lines and surfaces should be omitted in sectional views.

Thus, preferred sectional view should be as in Fig. 10.

In some cases, hidden edges and surfaces can be shown for describing the object

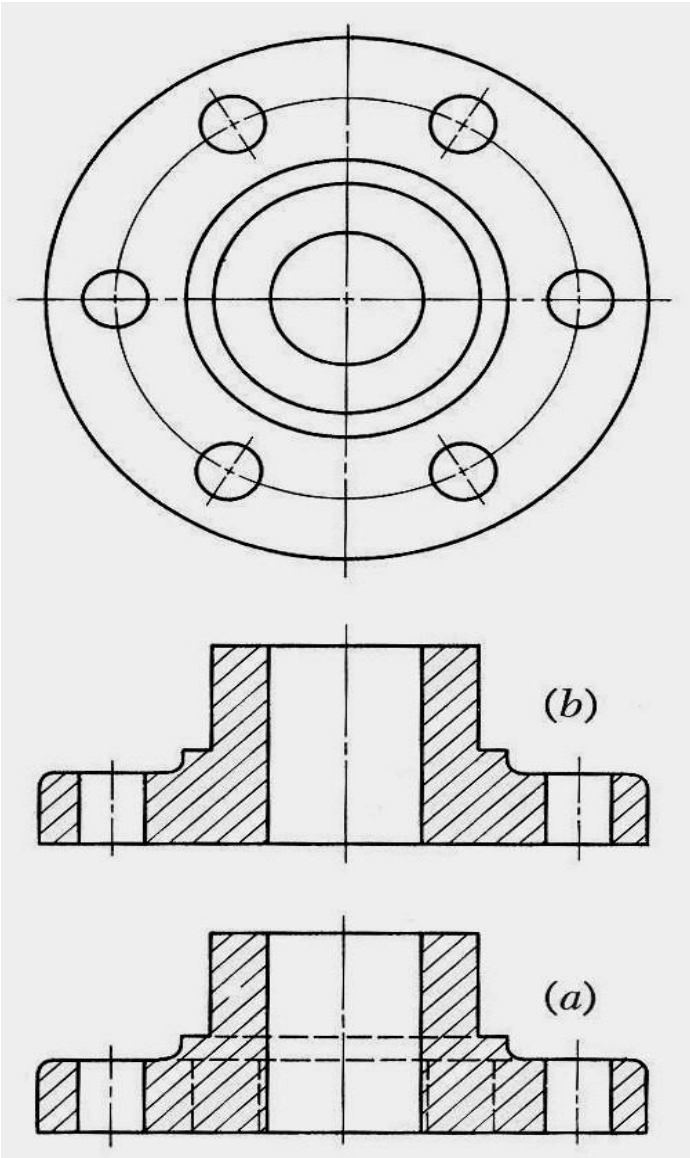


Fig .10 Hidden Edges and surfaces in section

Section Lining (Cross Hatching)

Section lining of a cut surface is indicated by fine lines, which are drawn as continuous lines usually at an angle of 45º with uniform distance (about 2 mm). For smaller or larger areas, distance between lines can be from 1 mm to 4 mm.

Section lining or cross-hatching lines should not be parallel or perpendicular to any main visible line bounding the sectioned area.

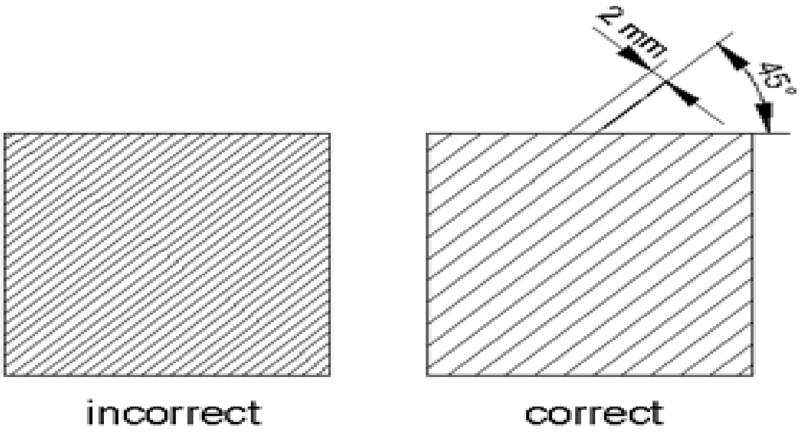


Fig. 11 Cross Hatching

Ribs and webs are used to strengthen the parts. When the cutting plane passes through the ribs lengthwise, cross-hatching would give the misleading impression that the section was conical (Fig.12).

Therefore, cross-hatching is eliminated from the ribs and webs (as if the cutting plane was just in front of them) when the cutting plane passes longitudinally through them (Fig. 12and 13).

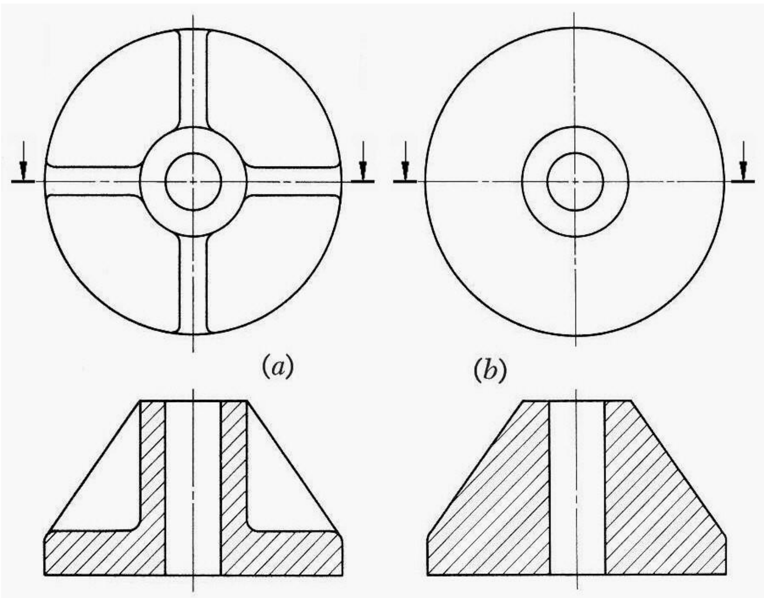


Fig. 12Web

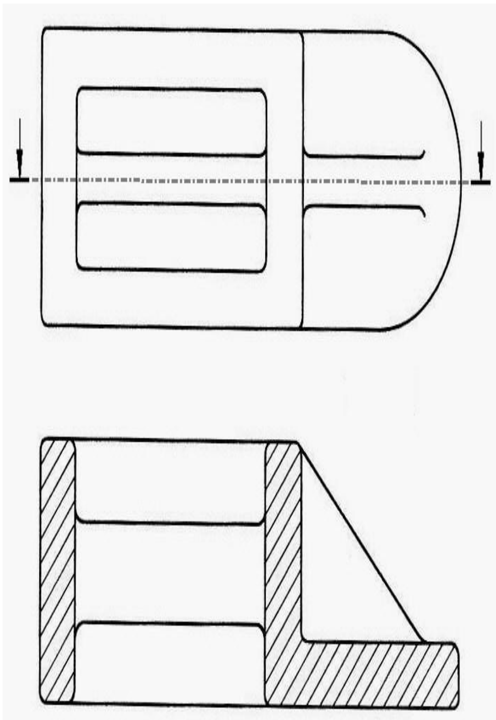


Fig. 13 Ribs

Exercise No. 12

SPOKES AND ARMS IN SECTION

When a cutting plane passes through pulley spokes or arms, cross-hatching is eliminated where the plane is thought of as being just in front of the spokes.

Even though the cutting plane passes through two of the spokes in Fig. 14, the sectional view in Fig. 14 a must be made without cross-hatching the spokes in order to avoid the appearance of a solid web as in Fig. 1 4 b.

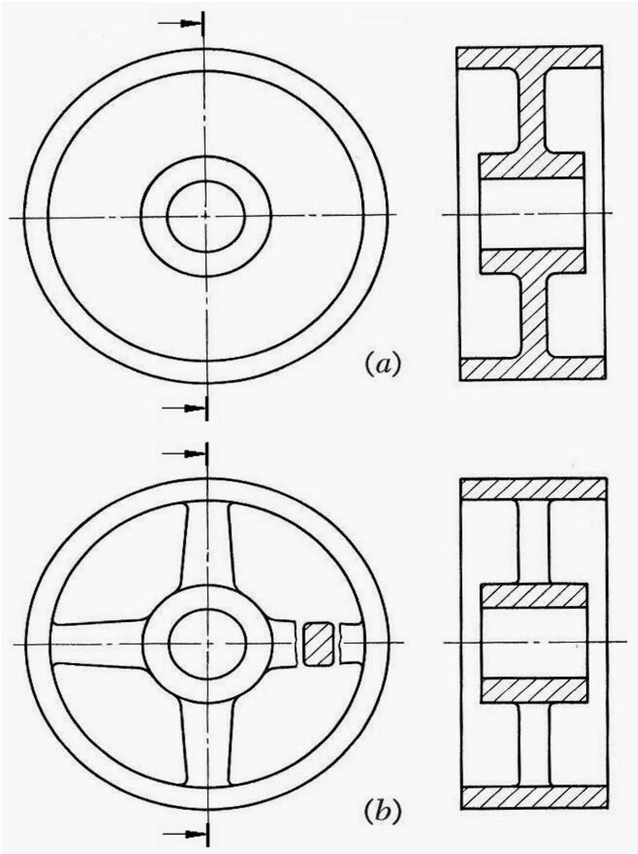


Fig 14 Spokes and Arms

Various cutting planes can be selected for obtaining clear sectional views

The plane may cut straight across (Fig.15) or be offset (changing direction forward and backward) to pass through features.

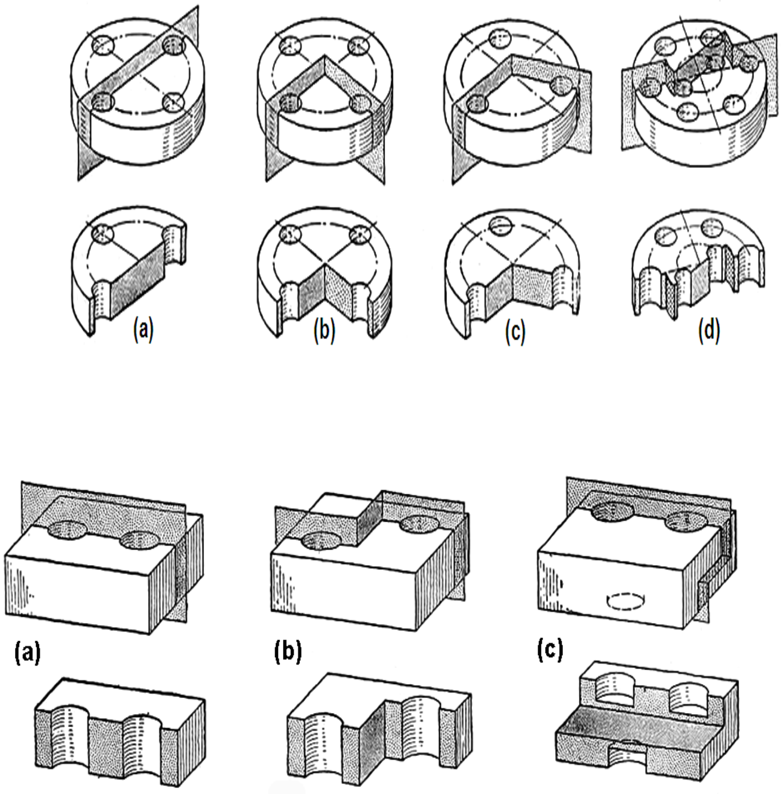
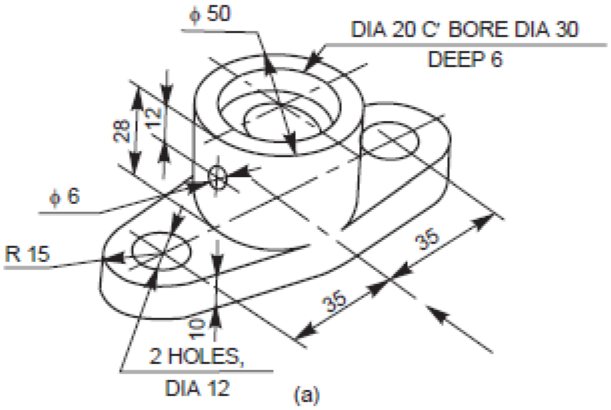


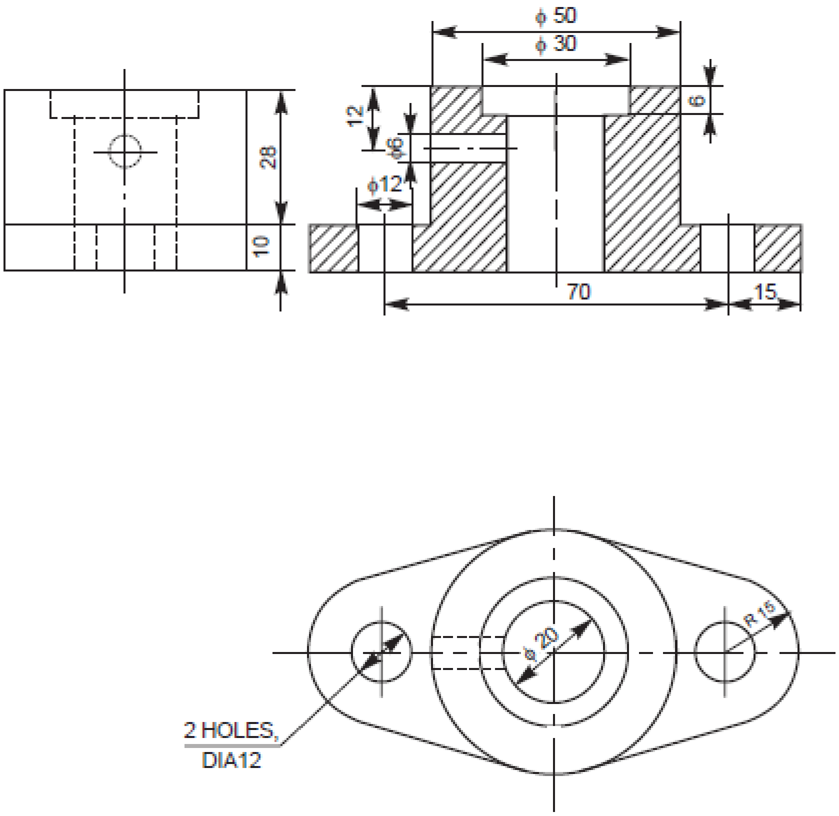
Fig. 15 : Various cutting planes obtaining clear sectional views.

Exercise No. 13

Draw (*i*) sectional view from the front, (*ii*) the view from above and (*iii*) the view from the right of the shaft support



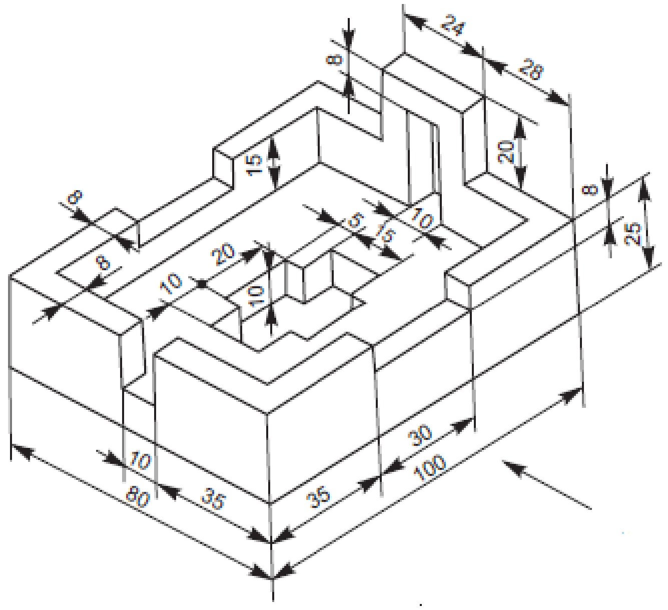
(a)



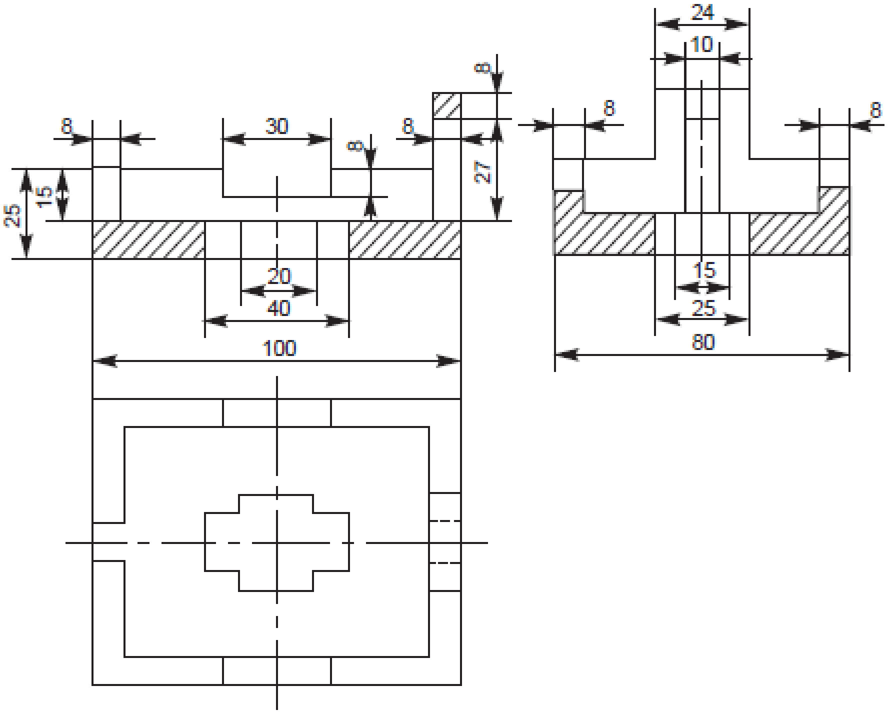
(b)

Exercise No. 14

Draw the (*i*) the sectional view from the front, (*ii*) the view from above and (*iii*) the sectional view from the left Of the given Machine block.



Isometric View of Machine Block



Sectional Views

Exercise No. 15

Week-III Dimensioning

A drawing of a component, in addition to providing complete shape description, must also furnish information regarding the size description. These are provided through the distances between the surfaces, location of holes, nature of surface finsih, type of material, etc. The expression of these features on a drawing, using lines, symbols, figures and notes is called dimensioning.

BASIC DIMENSIONING

In many applications, a drawing should contain annotations showing lengths or distances or angles between objects to convey the desired information. Dimensioning is the process of adding these annotations to a drawing. AutoCAD provides four basic types of dimensioning; linear, angular, diameter and radius.

DIM and DIMI Commands—DIMI command allows executing one dimensioning command and then returns to the normal command mode. If several dimensioning commands are to be executed, DIM command should be used. In this mode, the normal set of AutoCAD commands is replaced by a special set of dimensioning commands. To end the process of dimensioning, EXIT command has to be used.

The elements of dimensioning include the projection line, dimension line, leader line, dimension line termination, the origin indication and the dimension itself. The various elements of dimensioning are shown in Figs. 1 and 2. The following are some of the principles to be adopted during execution of dimensioning:

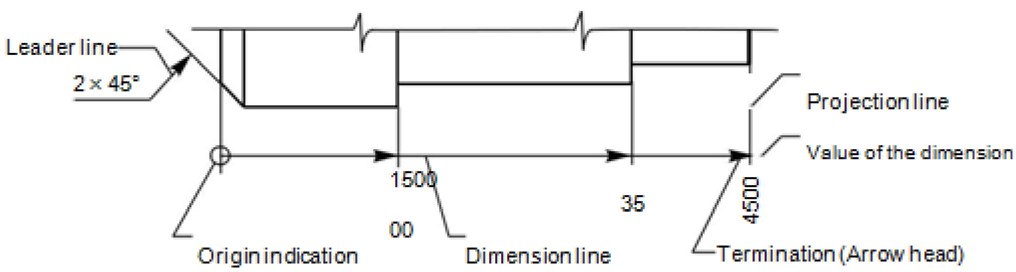


Fig. 1 Elements of dimensioning

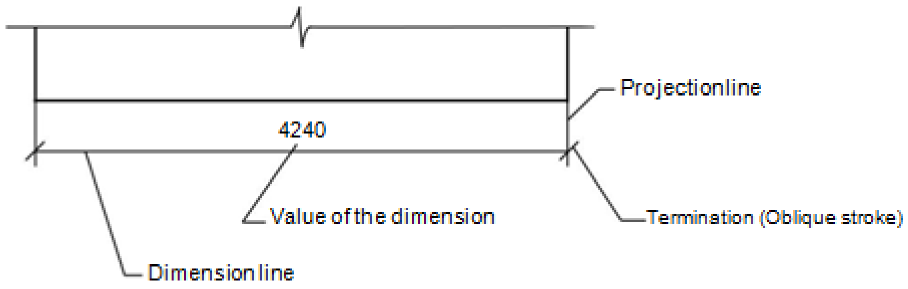


Fig. 2 Elements of dimensioning

Projection and dimension lines should be drawn as thin continuous lines.

Projection lines should extend slightly beyond the respective dimension lines.

Projection lines should be drawn perpendicular to the feature being dimensioned. Where necessary, they may be drawn obliquely, but parallel to each other (Fig. 2). However, they must be in contact with the feature.

Projection lines and dimension lines should not cross each other, unless it is unavoidable (Fig.4)

A dimension line should be shown unbroken, even where the feature to which it refers, is shown broken (Fig. 5).
A centre line or the outline of a part should not be used as a dimension line, but may be used in place of projection line (Fig. 6).

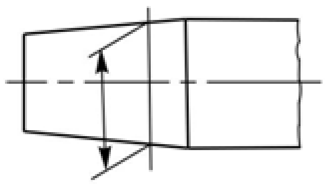


Fig. 4

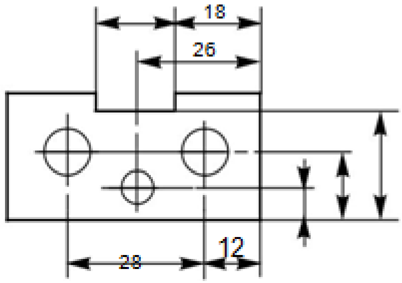


Fig. 5

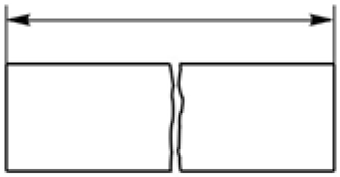


Fig. 6

Dimension lines should show distinct termination, in the form of arrow heads or oblique strokes or where applicable, an origin indication. Two dimension line terminations and an origin indication are shown in Fig. 7.

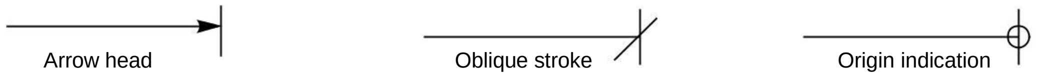


Fig. 8

METHOD–1 (Aligned System)

Dimensions should be placed parallel to their dimension lines and preferably near the middle, above and clear-off the dimension line (Fig. 9). An exception may be made where super-imposed running dimensions are used (Fig. 15)

Dimensions may be written so that they can be read from the bottom or from the right side of the drawing. Dimensions on oblique dimension lines should be oriented as shown in Fig. 10. Angular dimensions may be oriented as shown in Fig. 11.

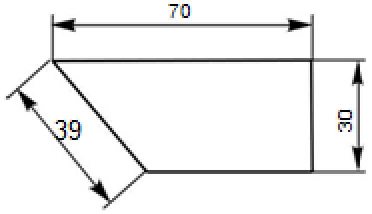


Fig. 9

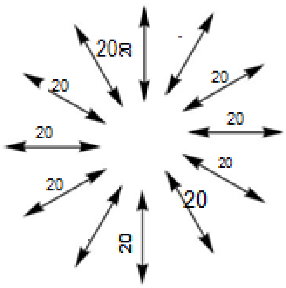


Fig. 10 Oblique dimensioning

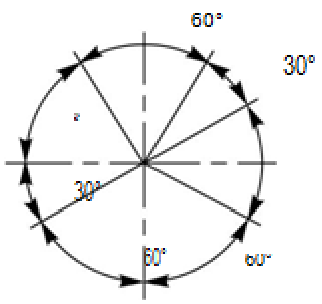


Fig. 11 Angular dimensioning

METHOD–2 (Uni-directional System)

Dimensions should be indicated so that they can be read from the bottom of the drawing only. Non-horizontal dimension lines are interrupted, preferably near the middle, for insertion of the dimension (Fig. 12). Angular dimensions may be oriented as in Fig. 13

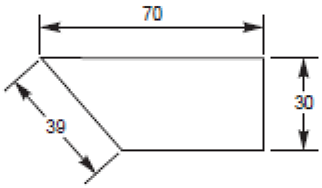


Fig. 12

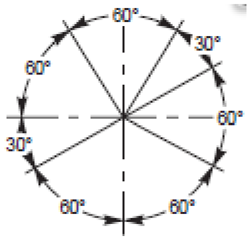


Fig. 13 Angular dimensioning

Arrangement of Dimensions

The arrangement of dimensions on a drawing must indicate clearly the design purpose. The following are the ways of arranging the dimensions.

Chain Dimensions

Chains of single dimensions should be used only where the possible accumulation of tolerances does not endanger the functional requirement of the part (Fig. 14).

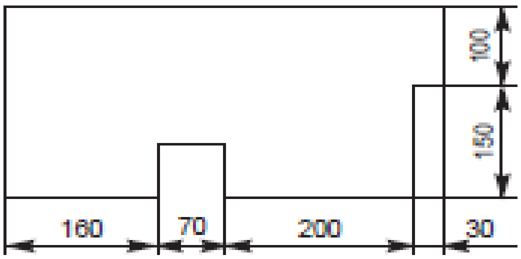


Fig. 14 Chain dimensioning

Parallel Dimensions

In parallel dimensioning, a number of dimension lines, parallel to one another and spaced -out are used. This method is used where a number of dimensions have a common datum feature (Fig. 15)

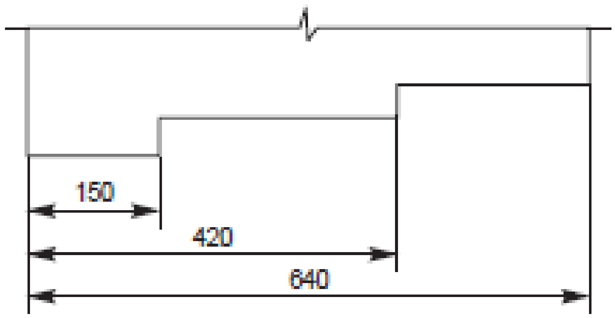


Fig. 15 Parallel dimensioning

Combined Dimensions

These are the result of simultaneous use of chain and parallel dimensions (Fig. 16).

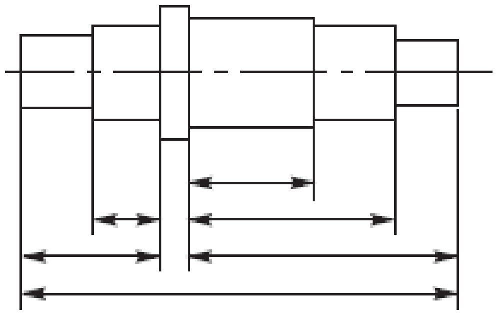
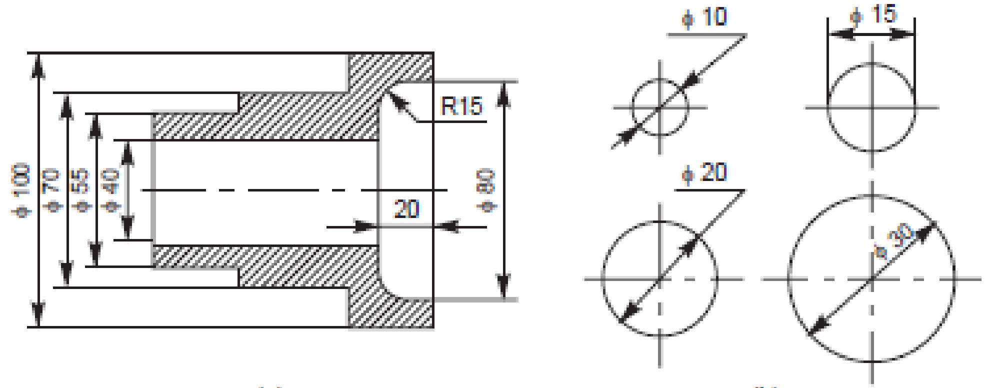


Fig. 16 Combined dimensioning

Exercise No. 16

Diameters

Diameters should be dimensioned on the most appropriate view to ensure clarity. The dimension value should be preceded by φ. Figure 17 shows the method of dimensioning diameters.



(a) (b)

Chords, Arcs, Angles and Radii

The dimensioning of chords, arcs and angles should be as shown in Fig. 18. Where the centre of an arc falls outside the limits of the space available, the dimension line of the radius should be broken or interrupted according to whether or not it is necessary to locate the centre (Fig. 19).

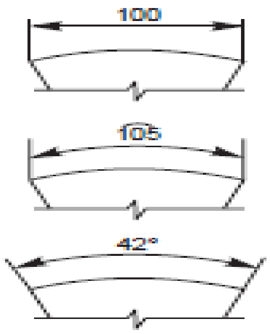


Fig. 18 Dimensioning of chords, arcs and angles

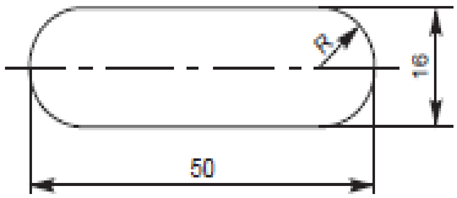


Fig. 19 Dimensioning of radius

Equi-distant Features

Linear spacings with equi-distant features may be dimensioned as shown in Fig. 20.

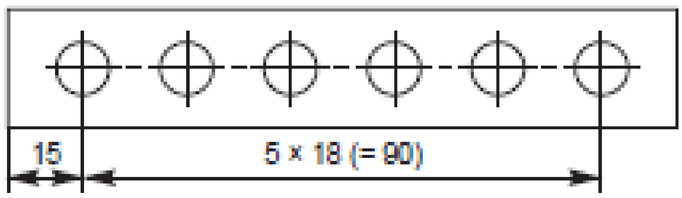


Fig. 20 Dimensioning equi-distant features

Tapered Features

Tapered features are dimensioned, either by specifying the diameters at either end and the length, or the length, one of the diameters and the taper or the taper angle (Fig. 21 *a*).A slope or flat taper is defined as the rise per unit length and is dimensioned by the ratio of the difference between the heights to its length (Fig 21 *b*).

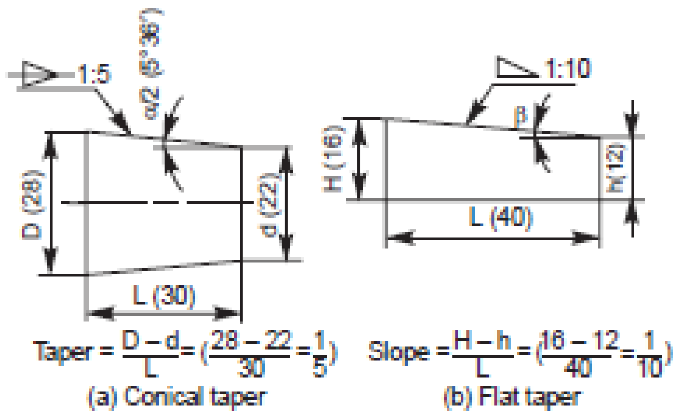


Fig. 21 Dimensioning tapered features

Exercise No. 17

EXAMPLES

Violations of some of the principles of drawing are indicated in Fig. 22 a. The corrected version of the same as per the BIS, SP–46: 1988 is given in Fig. 22 b and the reasons are given below:

1. Dimension should follow the shape symbol (Fig. 22).

2. and 3. As far as possible, features should not be used as extension lines for dimensioning.

4. Extension line should touch the feature.

5. Extension line should project beyond the dimension line.

6. Writing the dimension is not as per the aligned system.

7. Hidden lines should meet without a gap

8. Centre line representation is wrong. Dot should be replaced by a small dash.

9. Horizontal dimension line should not be broken to insert the value of the dimension

10. Dimension should be placed above the dimension line

11. Radius symbol should precede the dimension

12. Centre lines should cross at long dashes

13. Dimension should be written by symbol (not abbreviation) followed by its value

14. Note with dimensions should be written in capitals.

15. Elevation is not the correct usage.

16.Usage of the term ‘‘plan’’ is obsolete in graphic language.

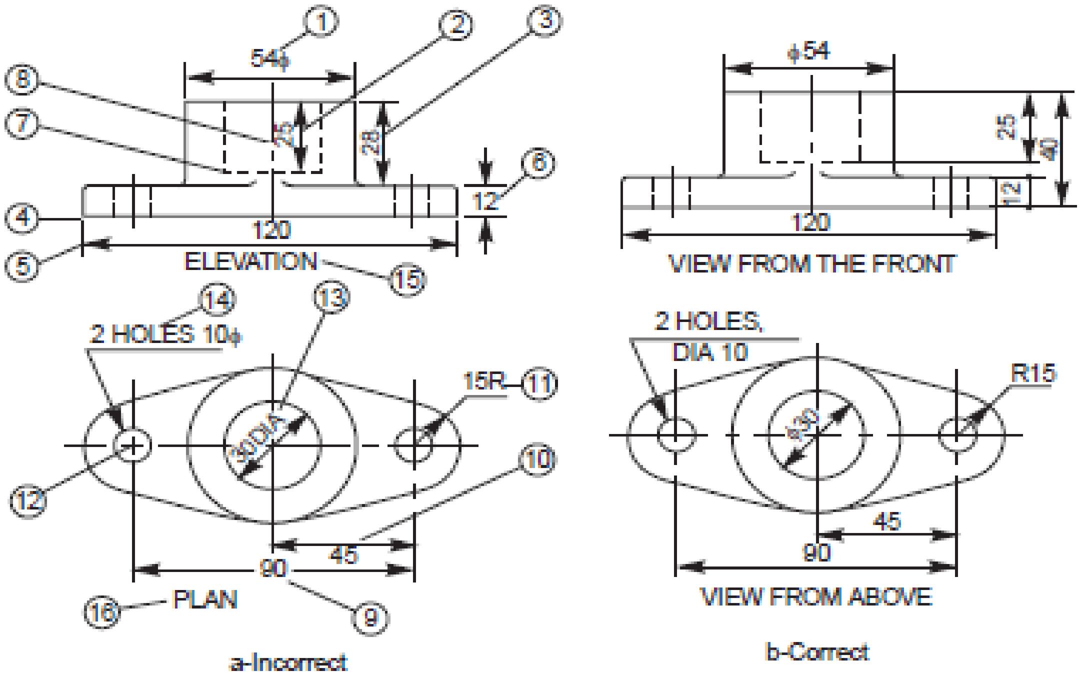

54ϕ
25
28
12
120
ELEVATION
2 HOLES 10ϕ
15R
30 DIA
45
90
PLAN
a-Incorrect
ϕ54
25
40
12
120
VIEW FROM THE FRONT
2 HOLES,
DIA 10
R15
ϕ30
45
90
VIEW FROM ABOVE
b-Correct

Fig. 22

Week-IV Working Drawings

A production drawing, also known as working drawing, supplies information and instructions for
the manufacture or construction of machines or structures. A production drawing should provide
all the dimensions, limits, special finishing processes, surface quality, etc. The particulars of material, the number of components required for the assembly, etc., are given in the title block. The production drawing of a component should also indicate the sub-assembly or main assembly
where it will be assembled. Since the working drawings may be sent to other companies to make or assemble the unit, the drawings should confirm with the standards followed in the country. For this reason, aproduction drawing becomes a legal document between the parties, in case of disputes in manufacturing.Working drawings may be classified into two groups : (i) detail or part drawings and (ii) assembly drawings

Types of working Drawings

A detail or part drawing is nothing but a production or component drawing, furnishing complete information for the construction or manufacture of the part. This information may be classified as:

1. Shape description This refers to the selection of number of views to describe the shape of the part. The part may be drawn in either pictorial or orthographic projection; the latter being used more frequently. Sectional views, auxiliary views and enlarged detailed views may be added to the drawing in order to provide a clear image of the part.
2. Size description Size and location of the shape features are shown by proper dimensioning. The manufacturing process will influence the selection of some dimensions, such as datum feature, tolerances, etc.
3. Specifications This includes special notes, material, heat treatment, finish, general tolerances and number required. All this information is mostly located near the title block.
4. Additional information Information such as drawing number, scale, method of projection, date, names of the parts, the draughter's name, etc., come under additional information which is included in the title block.

Since the craftsman will ordinarily make one component at a time, it is advisable to prepare the production drawing of each component, regardless of its size, on a separate sheet. Figures1 and 2 show the detailed drawings of a template jig and gear.

Exercise No. 18

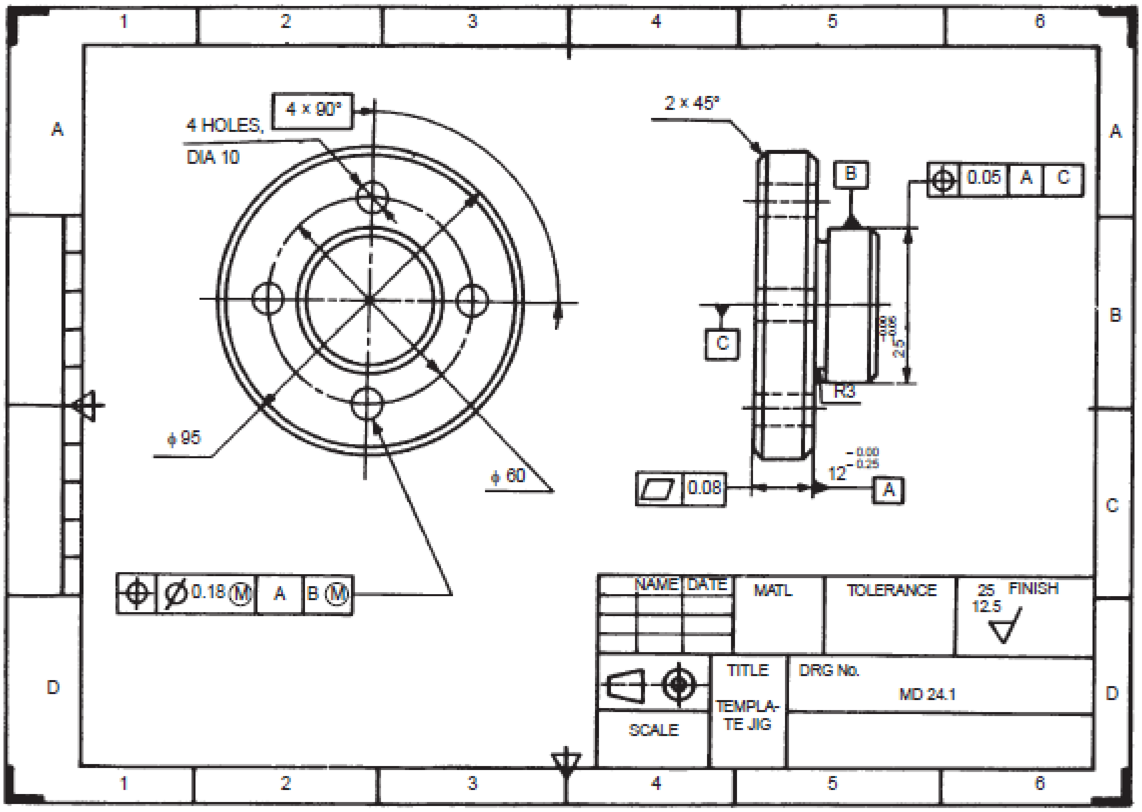


Fig. 1 Template Jig

Exercise No. 19

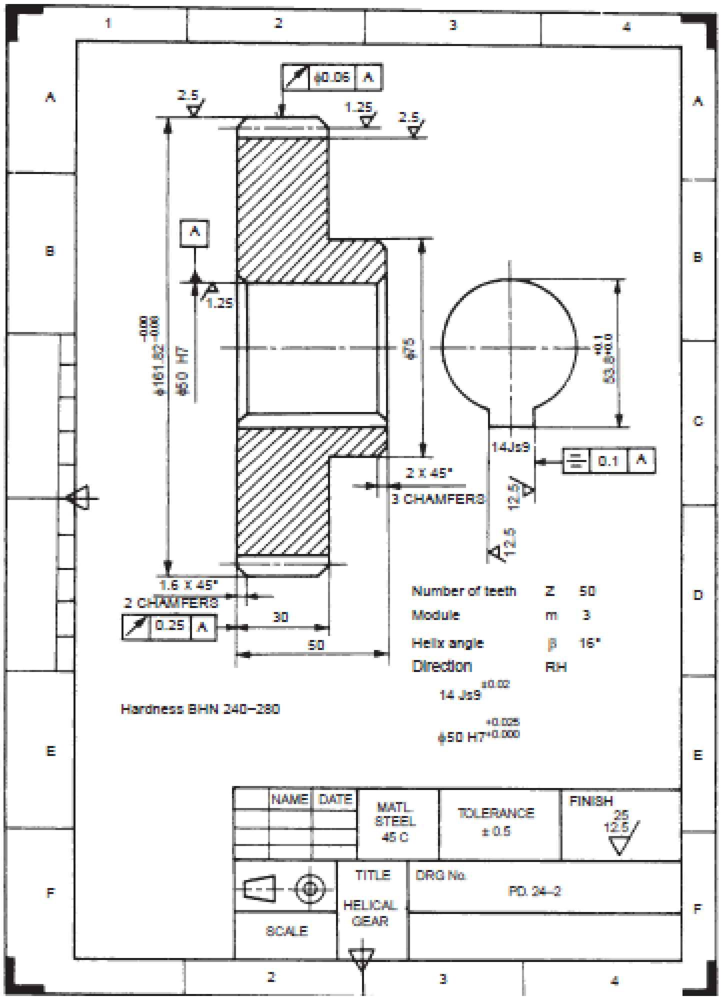


Fig.2 Gear

Exercise No. 20

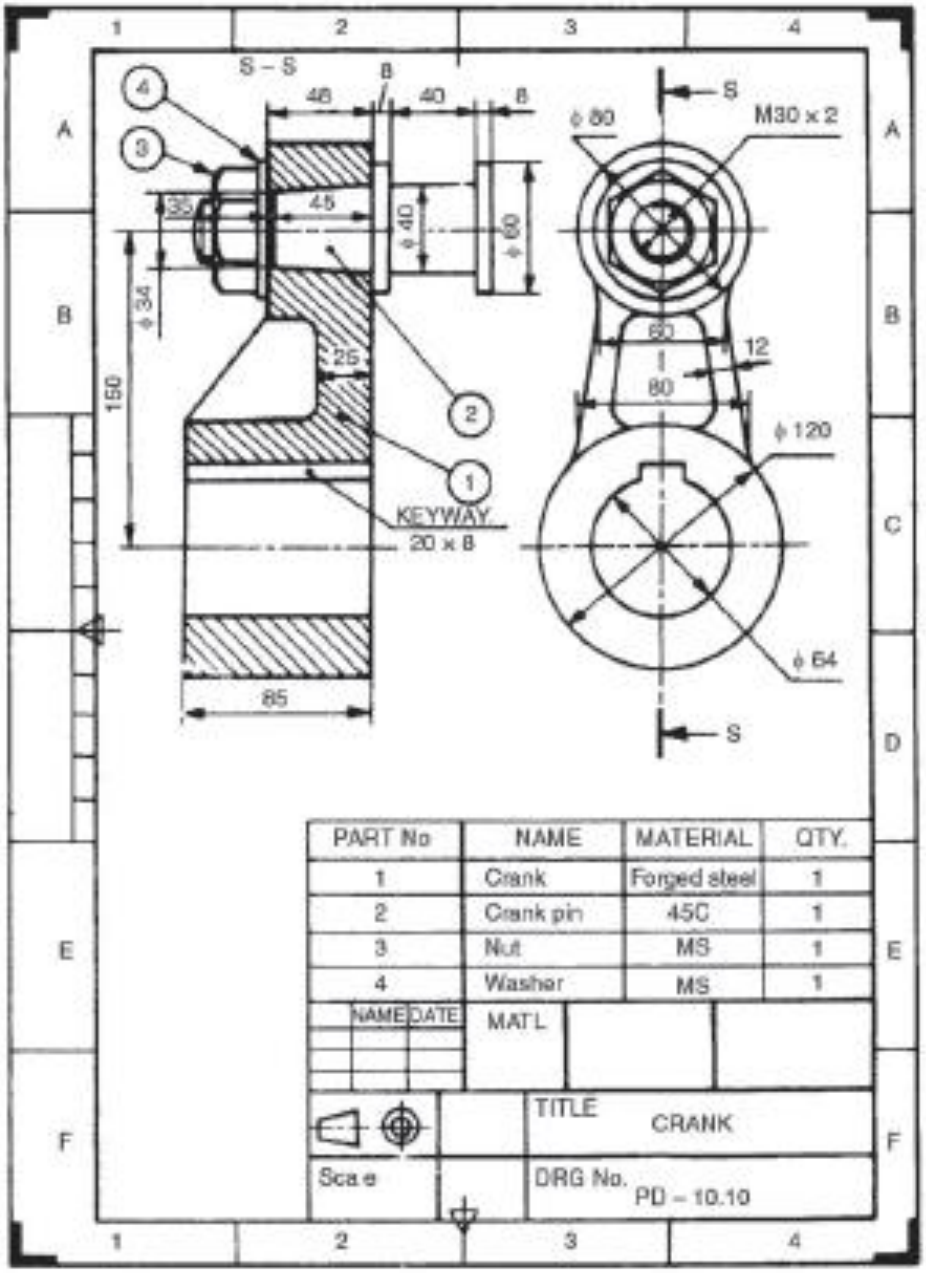


Fig. 3 Crank

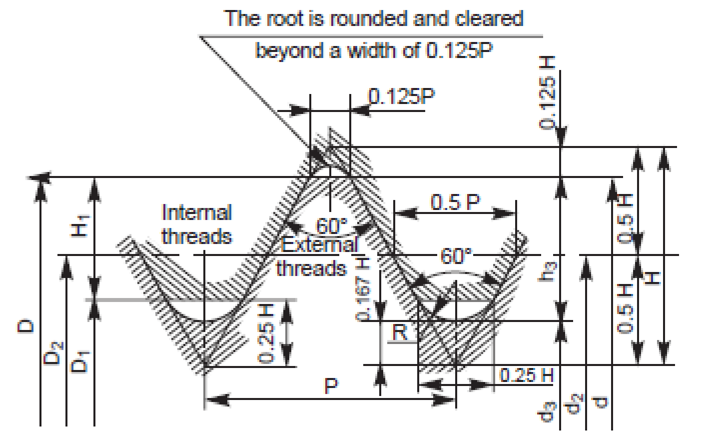
Exercise No. 21

Week-V MACHINE ELEMENTS

FORMS OF THREADS

Bureau of Indian Standards (BIS) adapts ISO (International Organisation for Standards) metric threads which are adapted by a number of countries apart from India.

The design profiles of external and internal threads are shown in Fig. 1. The following are the relations between the various parameters marked in the figure :



Internal thread diameters
D - Major diameter
D_2 - Pitch diameter
D_1 - Minor diameter

External thread diameters
d - Major diameter
d_2 - Pitch diameter
d_3 - Minor diameter

Fig. 5.2 Metric screw thread

P = Pitch

$H = 0.86\ P$

$D = d$ = Major diameter

$D_2 = d_2 = d - 0.75H$

$D_1 = d_2 - 2(H/2 - H/4) = d - 2H_1$

$d_3 = d_2 - 2\ (H/2 - H/6)$

$= d - 1.22P$

$H_1 = (D - D_1)/2 = 5H/8 = 0.54P$

$h_3 = (d - d_3)/2 = 17/24H = 0.61P$

$R = H/6 = 0.14P$

Other Thread Profiles

Apart from ISO metric screw thread profile, there are other profiles in use to meet various applications. These profiles are shown in Fig. 5.3, the characteristics and applications of which are discussed below :

V-Thread (sharp)

This thread profile has a larger contact area, providing more frictional resistance to motion.

Hence, it is used where effective positioning is required. It is also used in brass pipe work.

British Standard Whitworth (B.S.W) Thread

This thread form is adopted in Britain in inch units. The profile has rounded ends, making it less liable to damage than sharp V-thread.

Buttress Thread

This thread is a combination of V-and square threads. It exhibits the advantages of square thread, like the ability to transmit power and low frictional resistance, with the strength of the V-thread. It is used where power transmission takes place in one direction only such as screw press, quick acting carpenter's vice, etc.

Square Thread

Square thread is an ideal thread form for power transmission. In this, as the thread flank is at right angle to the axis, the normal force between the threads, acts parallel to the axis, with zero radial component. This enables the nut to transmit very high pressures, as in the case of a screw jack and other similar applications.

ACME Thread

It is a modified form of square thread. It is much stronger than square thread because of the wider base and it is easy to cut. The inclined sides of the thread facilitate quick and easy engagement and disengagement as for example, the split nut with the lead screw of a lathe.

Worm Thread

Worm thread is similar to the ACME thread, but is deeper. It is used on shafts to carry power to worm wheels.

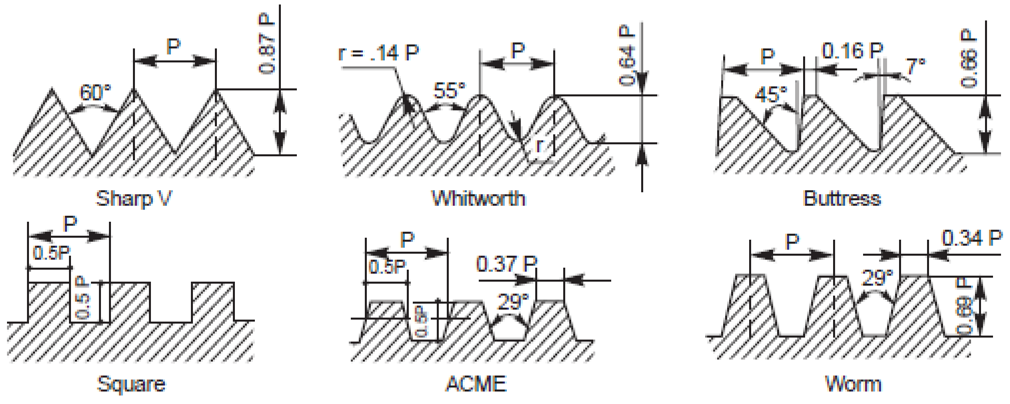


Fig. 5.3 Types of thread profiles

RIGHT HAND AND LEFT HAND THREADS

Screw threads may be right hand or left hand, depending on the direction of the helix. A right hand thread is one which advances into the nut, when turned in a clockwise direction and a left hand thread is one which advances into the nut when turned in a counter clockwise direction. An abbreviation LH is used to indicate a left hand thread. Unless otherwise stated, a thread should be considered as a right hand one. Figure 5.5 illustrates both right and left hand thread forms.

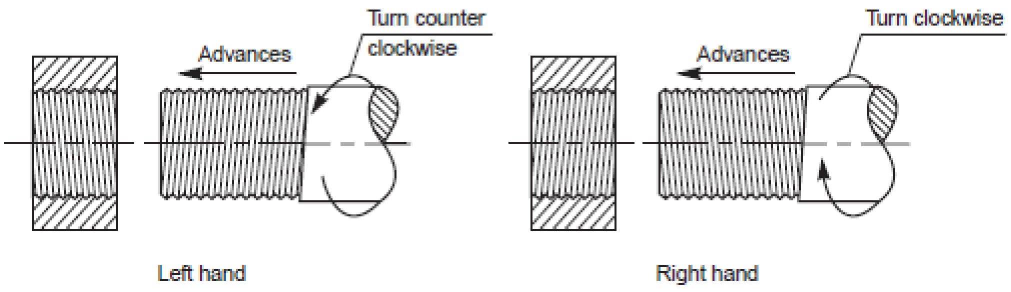


Fig. 5.5 Right hand and left hand threads

BOLTED JOINT

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 re-introduced in the same position. However, square nuts are used when frequent loosening and tightening is required, for example on job holding devices like vices, tool posts in machines, etc. The sharp corners on the head of bolts and nuts are removed by chamfering.

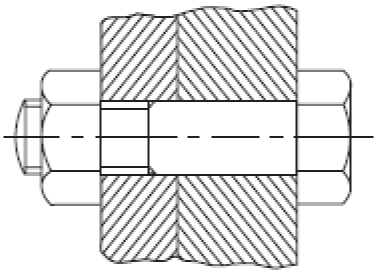


Fig. 5.11 Bolted joint

Exercise No. 22

Methods of Drawing Hexagonal (Bolt Head) Nut

Drawing hexagonal bolt head or nut, to the exact dimensions is labourious and time consuming. Moreover, as standard bolts and nuts are used, it is not necessary to draw them accurately. The following approximate methods are used to save the draughting time

Method 1 (Fig. 5.12)

Empirical relations :

Major or nominal diameter of bolt = D

Thickness of nut, T = D

Width of nut across flat surfaces, W = 1.5D + 3 mm

Radius of chamfer, R =1.5D

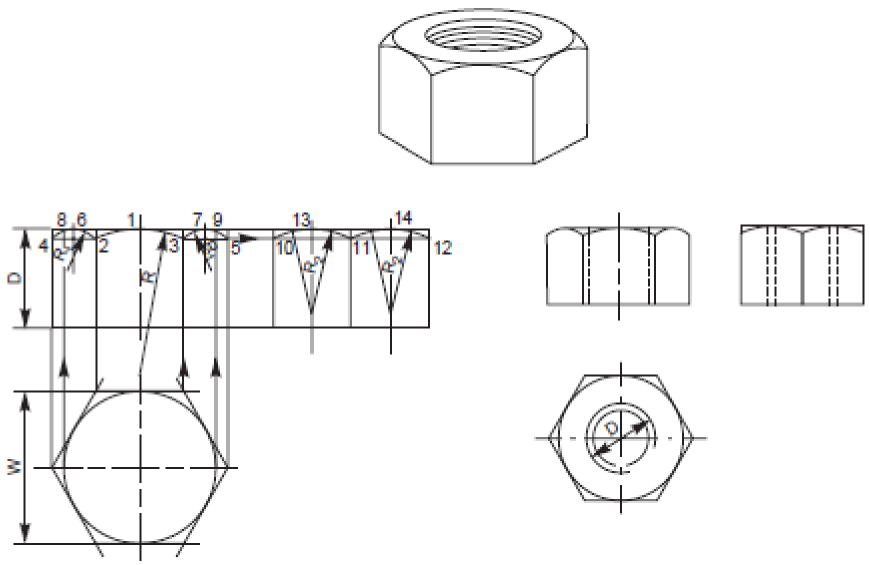


Fig. 5.12 Method of drawing views of a hexagonal nut (Method I)

PROCEDURE

Draw the view from above by drawing a circle of diameter, W and describe a regular hexagon on it, by keeping any two parallel sides of the hexagon, horizontal.

Project the view from the front, and the view from side, and mark the height equal to D.

With radius R, draw the chamfer arc 2-1-3 passing through the point 1 in the front face.

Mark points 4 and 5, lying in-line with 2 and 3.

Locate points 8,9 on the top surface, by projecting from the view from above.

Draw the chamfers 4–8 and 5–9.

Locate points 6 and 7, lying at the middle of the outer two faces.

Draw circular arcs passing through the points 4, 6, 2 and 3, 7, 5, after determining the radius R_1 geometrically.
Project the view from the side and locate points 10, 11 and 12.

Mark points 13 and 14, lying at the middle of the two faces (view from the side).

Draw circular arcs passing through the points 10, 13, 11 and 11, 14, 12, after determining the radius R_2 geometrically.

It may be noted that in the view from the front, the upper outer corners appear chamfered.

In the view from the side, where only two faces are seen, the corners appear square.

Method 2 (Fig. 5.13)

Empirical relations:

Major or nominal diameter of bolt = D

Thickness of nut, T	= D
Width of the nut across corners	= 2 D
Radius of chamfer arc, R	= 1.5 D

Figure 5.13 illustrates the stages of drawing different views of a hexagonal nut, following the above relations, which are self-explanatory.

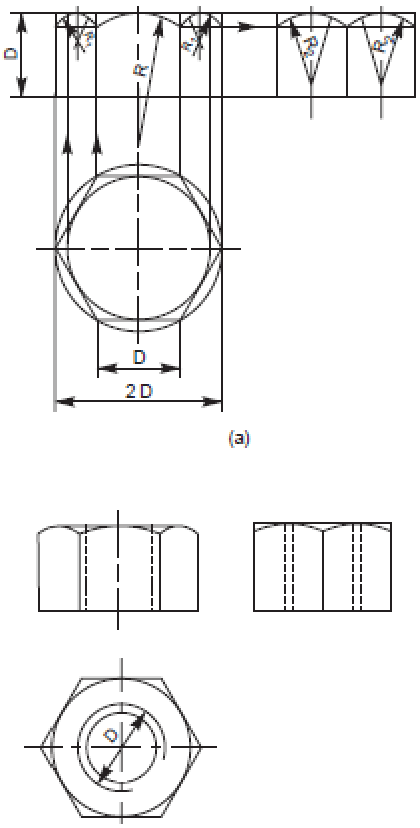


Fig. 5.13 Method of drawing views of a hexagonal nut (Method II)

Exercise No. 23

Method of Drawing Square (Bolt Head) Nut

A square bolt head and nut may be drawn, showing either across flats or corners. Following relations may be adopted for the purpose:

Major or nominal diameter of bolt	= D
Thickness of nut, T	= D
Width of the nut across flats, W	= 1.5 D + 3 mm
Radius of chamfer arc, R	= 2 D

Figure 5.14 illustrates the method of drawing views of a square nut, in two orientations.

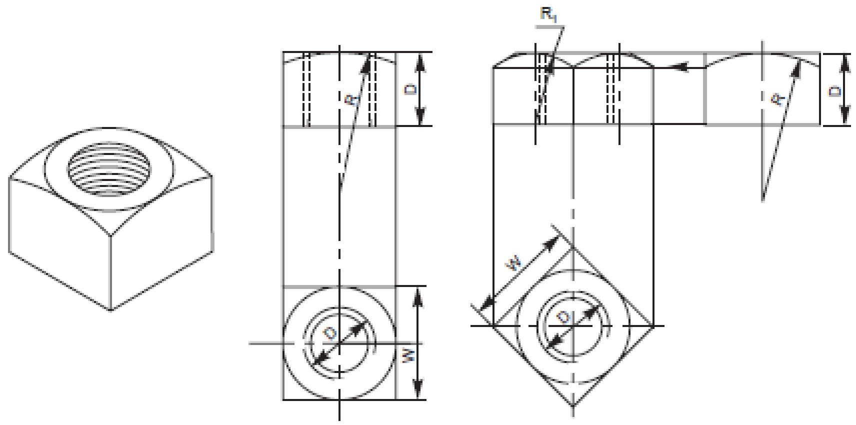
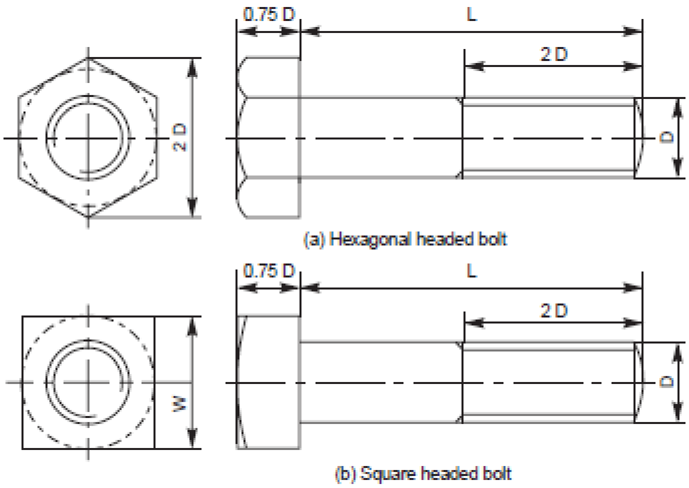


Fig.5.14 Method of drawing the views of a square nut

Hexagonal and Square Headed Bolts

Figure 5.15 shows the two views of a hexagonal headed bolt and square headed bolt, with the proportions marked.



Square Headed Bolt with Square Neck

It is provided with a square neck, which fits into a corresponding square hole in the adjacent part, preventing the rotation of the bolt (Fig. 5.18).

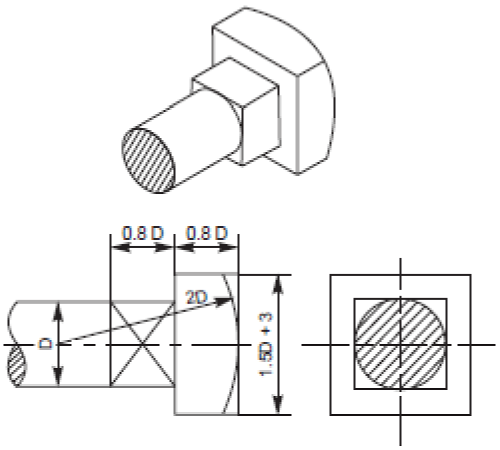


Fig. 5.18 Square headed bolt with square neck

Eye Bolt

In order to facilitate lifting of heavy machinery, like electric generators, motors, turbines, etc., eye bolts are screwed on to their top surfaces. For fitting an eye bolt, a tapped hole is provided, above the centre of gravity of the machine (Fig. 5.21).

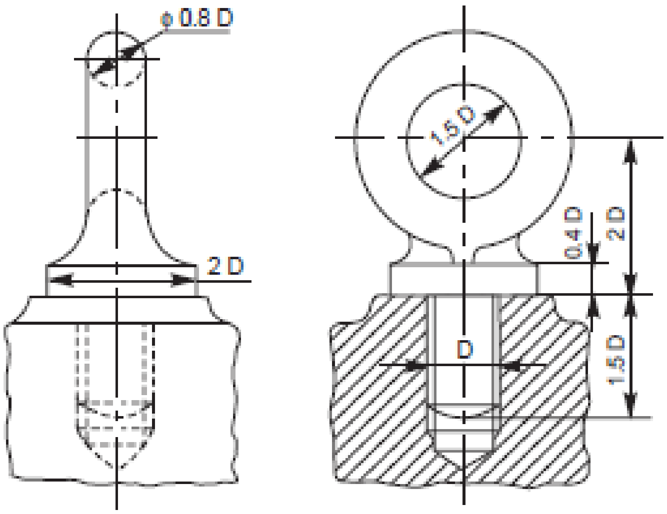


Fig. 5.21 Eye-bolt

Stud Bolt or Stud

It consists of cylindrical shank with threads cut on both the ends (Fig. 5.22*a*). It is used where there is no place for accommodating the bolt head or when one of the parts to be joined is too thick to use an ordinary bolt.

The stud is first screwed into one of the two parts to be joined, usually the thicker one. A stud driver, in the form of a thick hexagonal nut with a blind threaded hole is used for the purpose. After placing the second part over the stud, a nut is screwed-on over the nut end. It is usual to provide in the second part, a hole which is slightly larger than the stud nominal diameter. Figure 5.22*b* shows a stud joint.

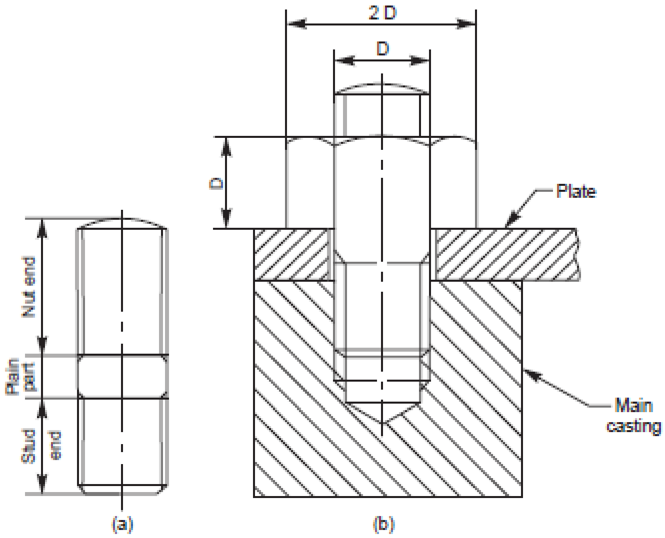


Fig. 5.22 (*a*)–Stud, (*b*)–Stud joint

Other Forms of Nuts

Flanged Nut

This is a hexagonal nut with a collar or flange, provided integral with it. This permits the use of a bolt in a comparitively large size hole (Fig. 5.23*a*).

Cap Nut

It is a hexagonal nut with a cylindrical cap at the top. This design protects the end of the bolt from corrosion and also prevents leakage through the threads. Cap nuts are used in smoke boxes or locomotive and steam pipe connections (Fig. 5.23*b*).

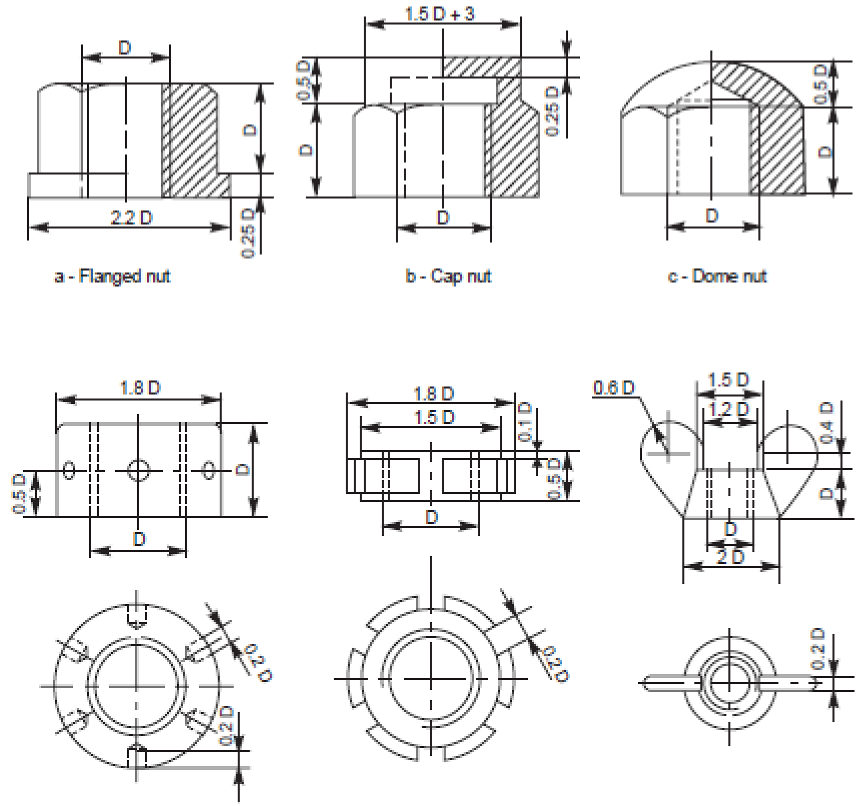
Dome Nut

It is another form of a cap nut, having a spherical dome at the top (Fig. 5.23*c*).

Capstan Nut

This nut is cylindrical in shape, with holes drilled laterally in the curved surface. A tommy bar may be used in the holes for turning the nut (Fig. 5.23*d*). Holes may also be drilled in the upper flat face of the nut.

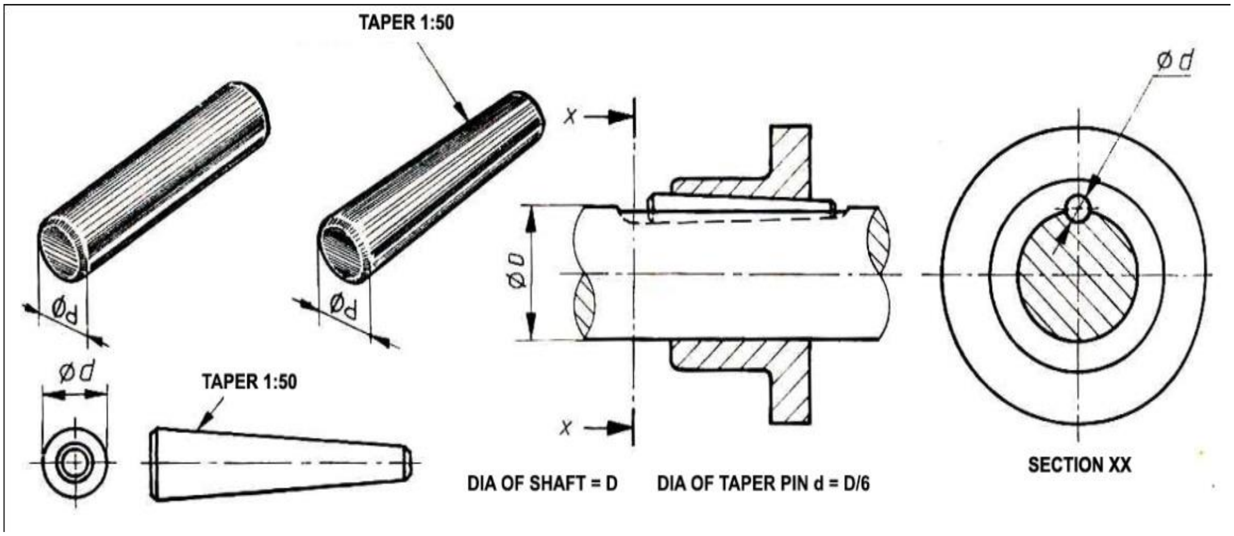
Exercise No. 24

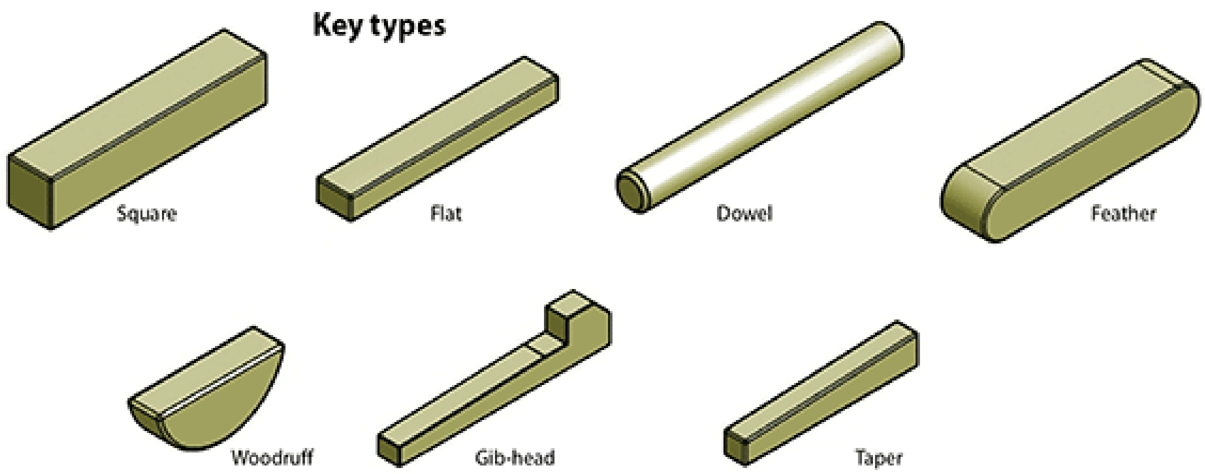


d - Capstan nut e - Ring nut f - Wing nut

Fig. 5.23 Other forms of nuts

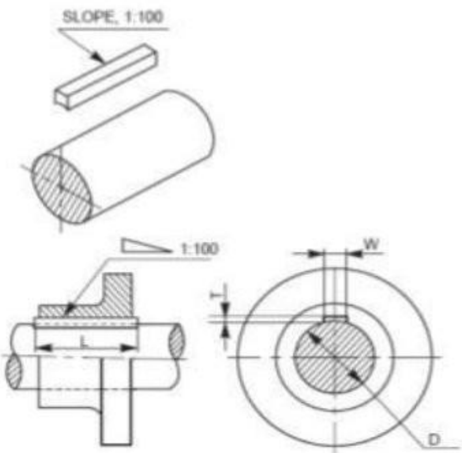
Week-VI KEYS AND COTTER JOINTS



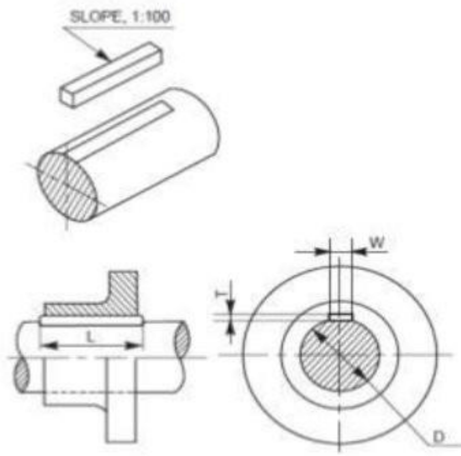


Saddle Keys

- These are taper keys, with uniform width but tapering in thickness on the upper side.
- suitable for light duty only



Hollow saddle key

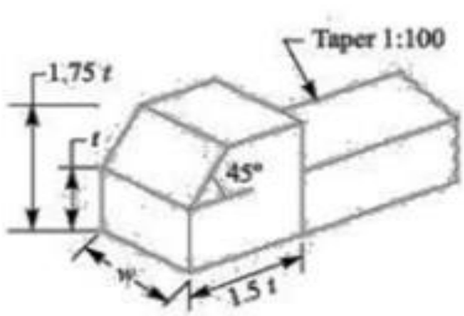


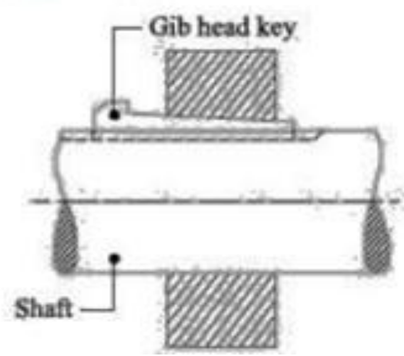
Flat saddle key

The rectangular sunk key with head at one end is known as **GIB HEAD KEY.** The head is provided to facilitate the removal of key.

Width $w = d/4$

Thickness at large end $t = 2w/3 = d/6$





A cotter is a flat wedge shaped piece, made of steel. It is uniform in thickness but tapering in width, generally on one side; the usual taper being 1:30. The lateral (bearing) edges of the cotter and the bearing slots are generally made semi-circular instead of straight (Fig. 1.0).

This increases the bearing area and permits drilling while making the slots. The cotter is locked in position by means of a screw as shown in Fig. 1.1.

Cotter joints are used to connect two rods, subjected to tensile or compressive forces along their axes. These joints are not suitable where the members are under rotation. The following are some of the commonly used cotter joints:

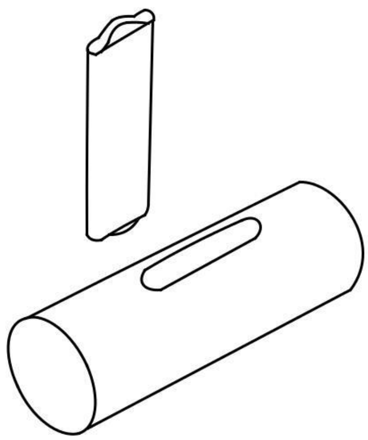


Fig. 1.0 Cotter and the bearing slot

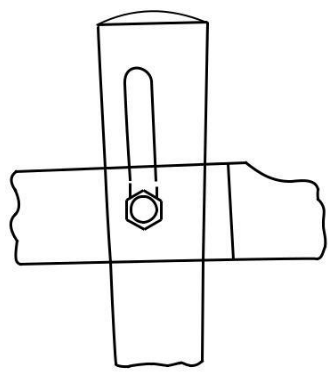


Fig. 1.1 Locking arrangement of cotter

Cotter Joint with Sleeve

This is the simplest of all cotter joints, used for fastening two circular rods. To make the joint, the rods are enlarged at their ends and slots are cut. After keeping the rods butt against each other, a sleeve with slots is placed over them. After aligning the slots properly, two cotters are driven-in through the slots, resulting in the joint (Fig. 1.1). The rod ends are enlarged to take care of the weakening effect caused by the slots.

The slots in the rods and sleeve are made slightly wider than the width of cotter. The relative positions of the slots are such, that when a cotter is driven into its position, it permits wedging action and pulls the rod into the sleeve.

Exercise No. 25

Draw the sectional view from the front, and view from the side of a cotter joint with sleeve used to connect two rods of 50 mm diameter each.

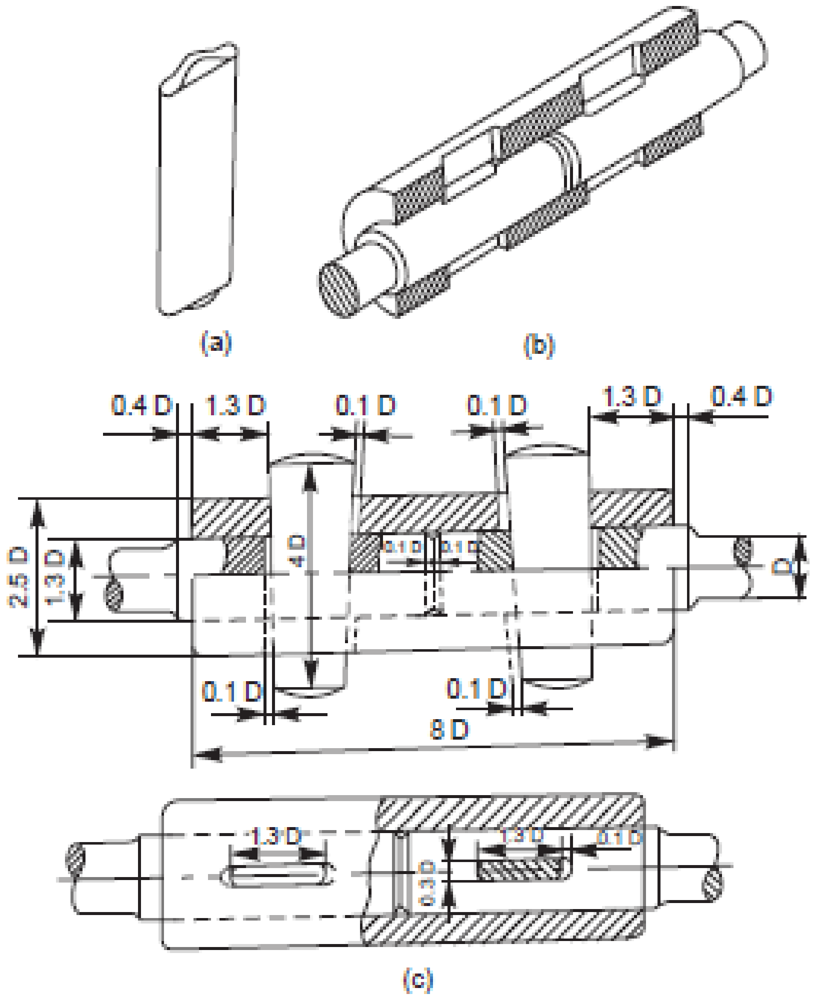


Fig. 1.2 Cotter joint with sleeve

Exercise No. 26

Cotter Joint with Socket and Spigot Ends

This joint is also used to fasten two circular rods. In this, the rod ends are modified instead of using a sleeve. One end of the rod is formed into a socket and the other into a spigot (Fig1.3) and slots are cut. After aligning the socket and spigot ends, a cotter is driven-in through the slots, forming the joint.

Draw the half sectional view from the front, with top half in section and the view from the side of a cotter joint with socket and spigot ends, to connect two rods of 50 mm diameter each.

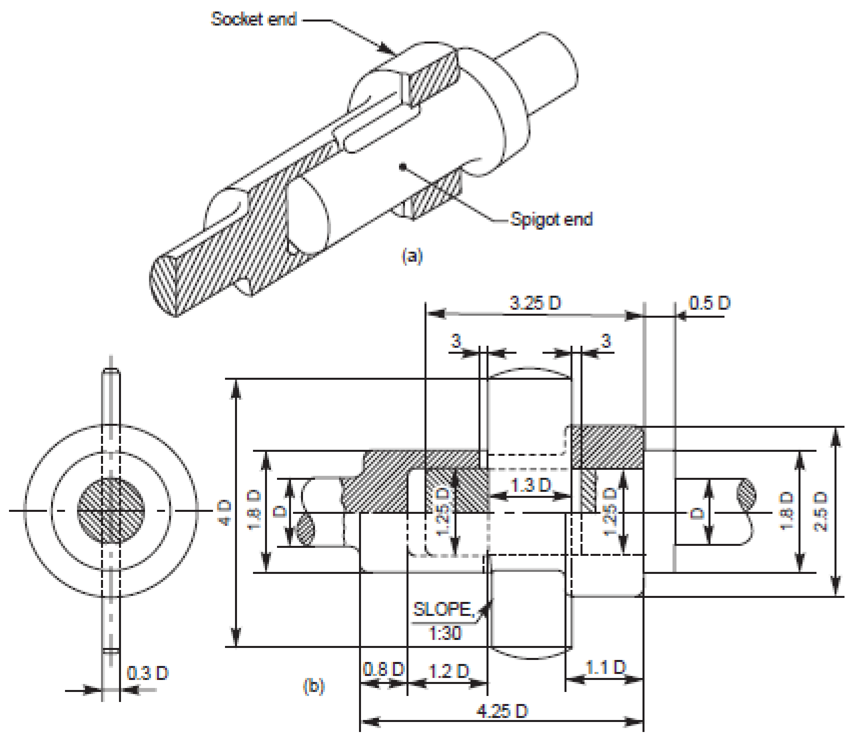


Fig. 1.3 Cotter joint with socket and spigot ends

Knuckle Joint

A knuckle joint is a pin joint used to fasten two circular rods. In this joint, one end of the rod is formed into an eye and the other into a fork (double eye). For making the joint, the eye end of the rod is aligned into the fork end of the other and then the pin is inserted through the holes and held in position by means of a collar and a taper pin (Fig. 1.4). Once the joint is made, the rods are free to swivel about the cylindrical pin.

Knuckle joints are used in suspension links, air brake arrangement of locomotives, etc.

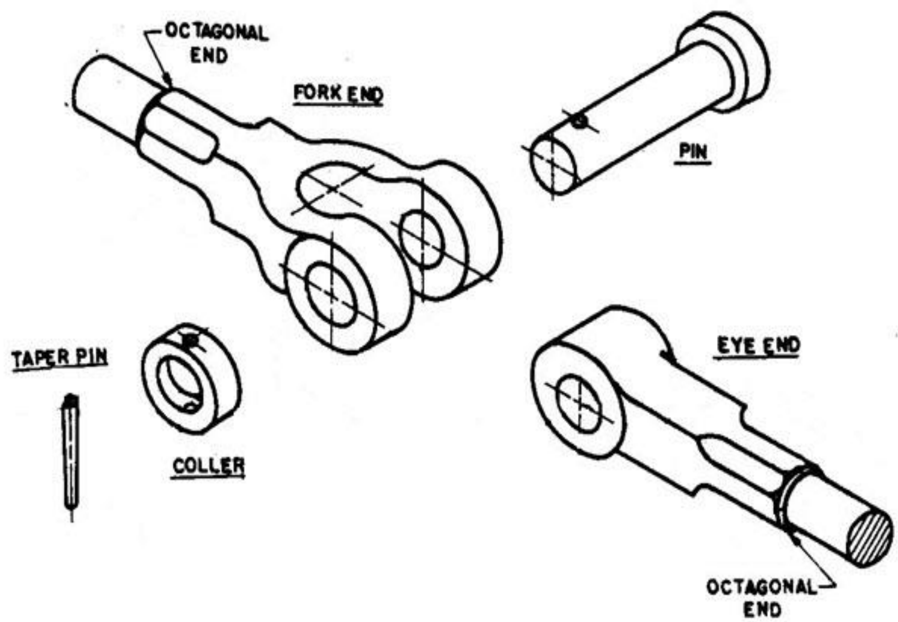


Fig.1.4

Exercise No. 27

Draw a Knuckle joint with 50 mm diameter (D)

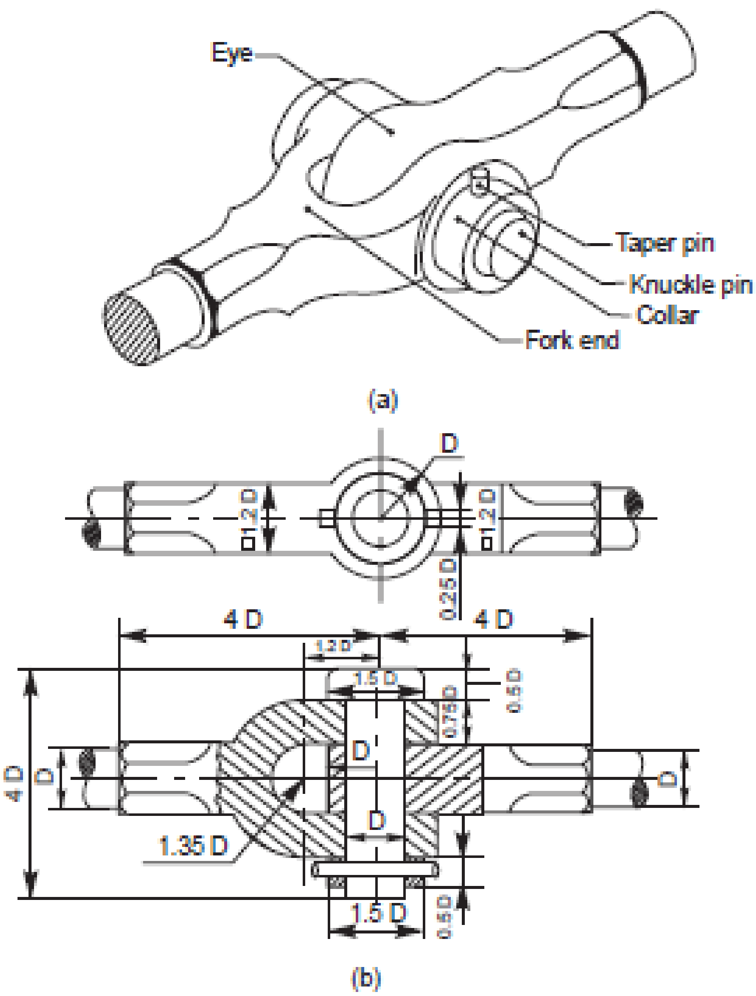
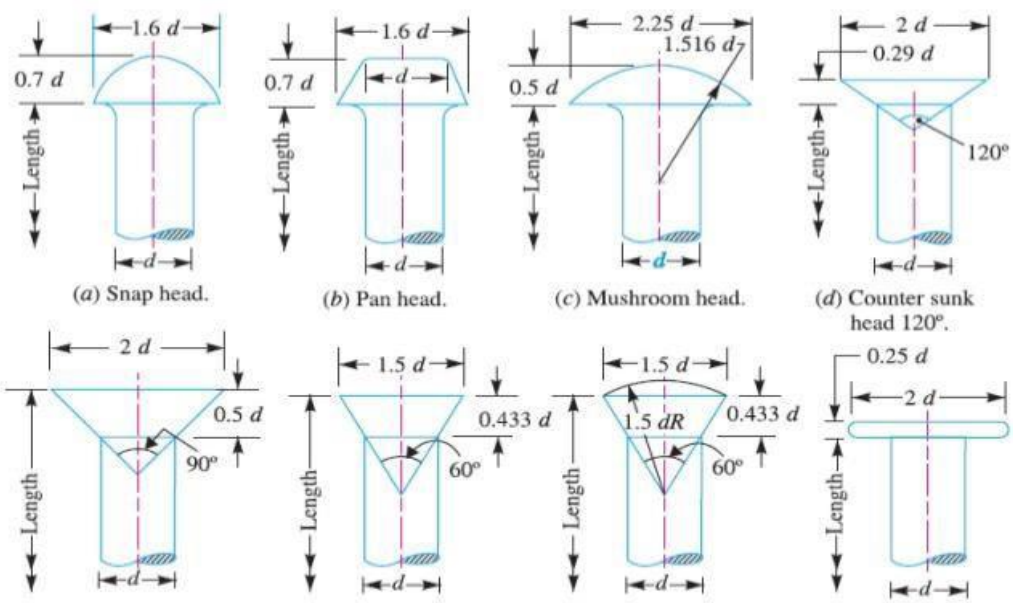


Fig. 1.5 Knuckle joint

Exercise No. 28

Week-VII Riveted Joints

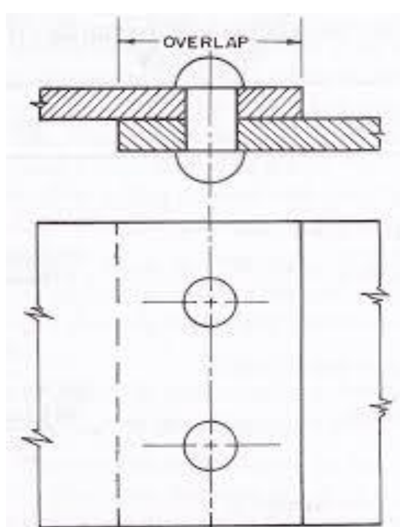
Types of Rivet Heads



(a) Snap head. (b) Pan head. (c) Mushroom head. (d) Counter sunk head 120°.

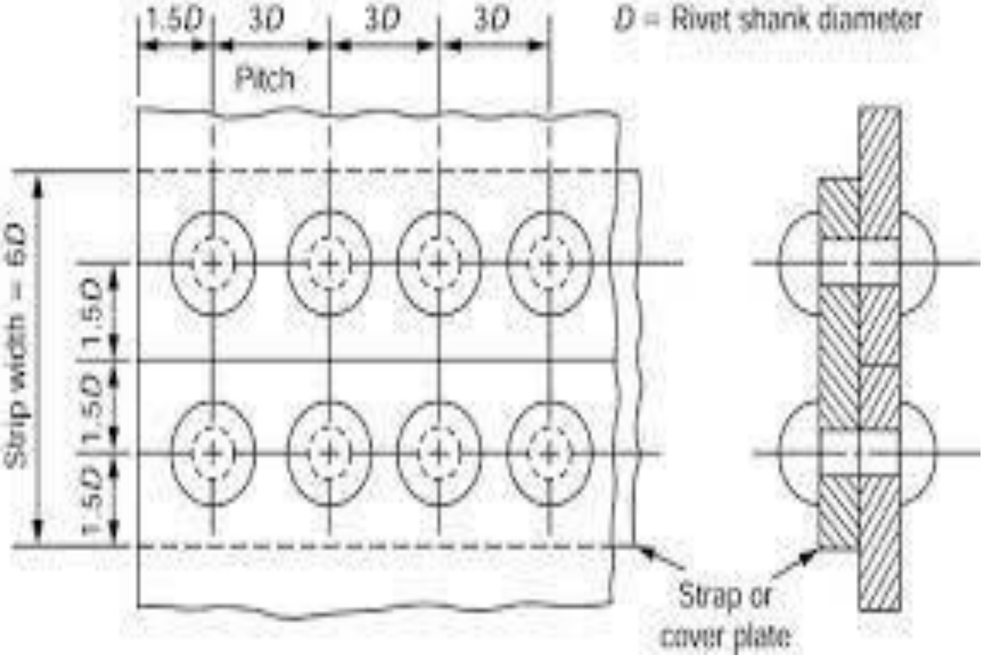
(e) Flat counter sunk head 90°. (f) Flat counter sunk head 60°. (g) Round counter sunk head 60°. (h) Flat head.

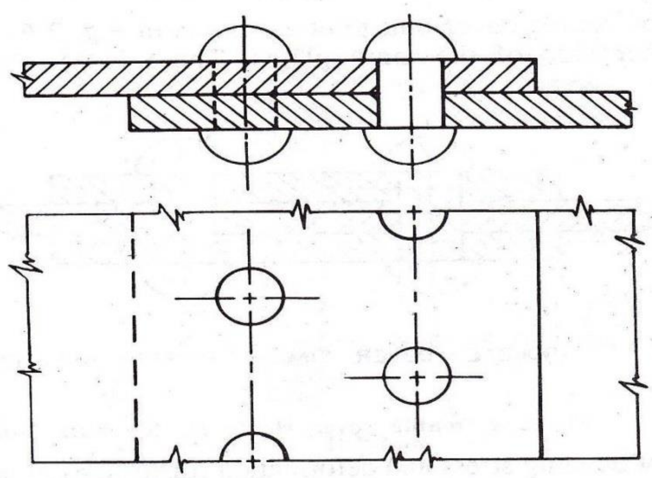
Riveted Joints



(A) LAP JOINT (SINGLE RIVETED)

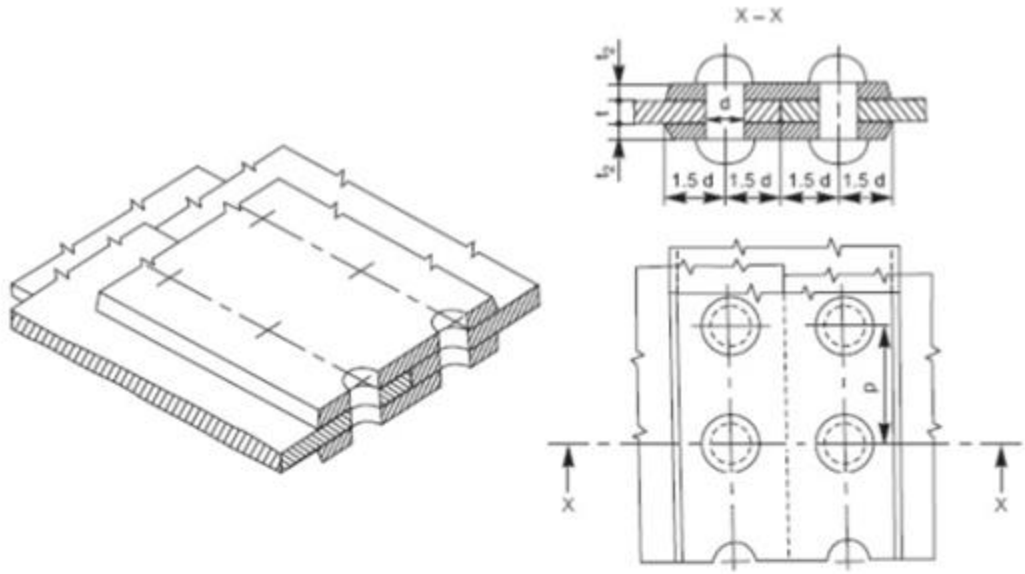
Single strap chain riveted butt joint (single row)



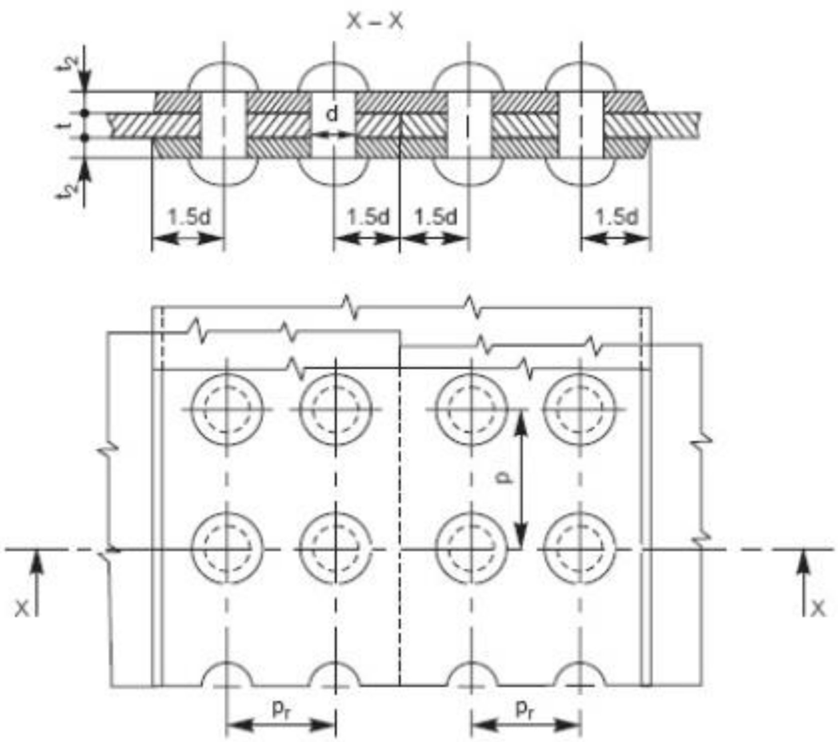


(B) ZIG-ZAG RIVETED LAP JOINT

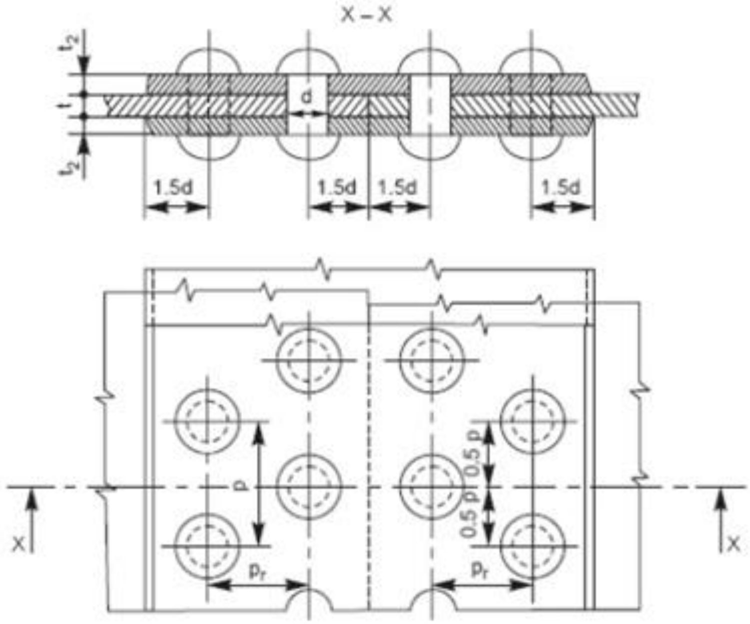
Single riveted, double strap butt joint



Double riveted, double strap chain butt joint

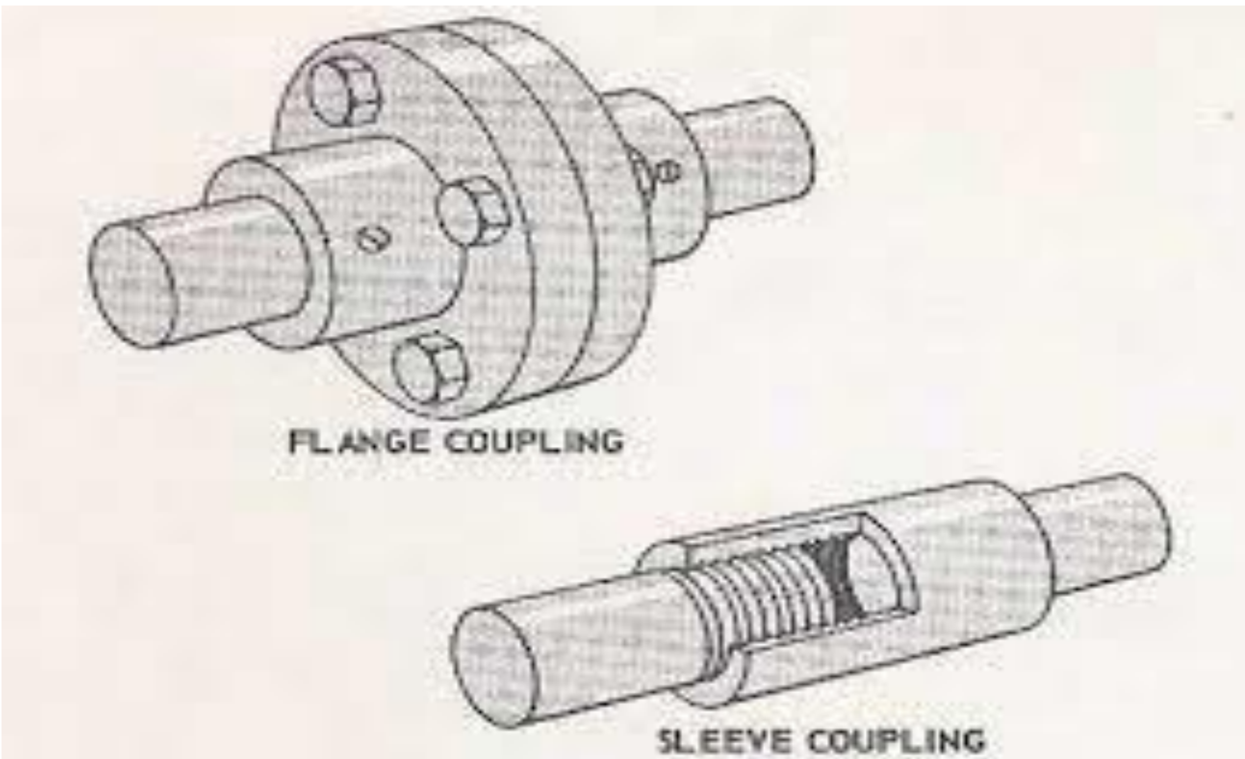


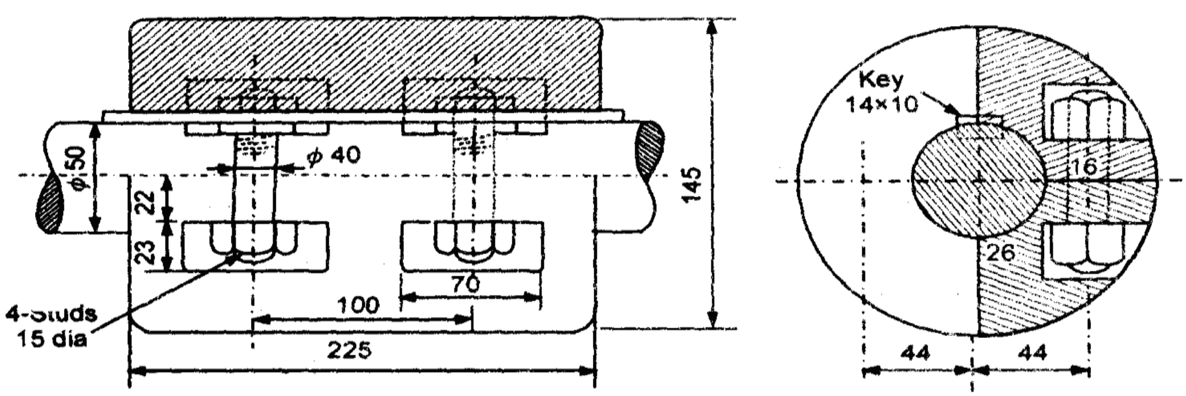
Double riveted, double strap zig zag butt joint



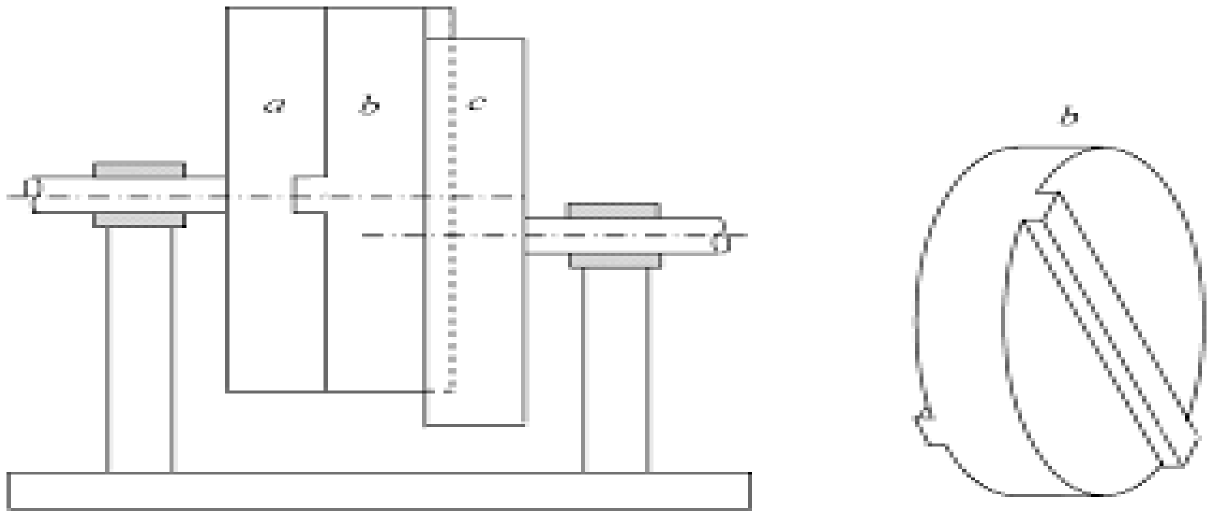
Week-VIII Couplings

Shaft couplings are used to join or connect two shafts in such a way that when both the shaftsrotate, they act as one unit and transmit power from one shaft to the other. Shafts to beconnected or coupled may have collinear axes, intersecting axes or parallel axes at a smalldistance. Based on the requirements, the shaft couplings are classified as: (*i*) rigid couplings,(*ii*) flexible couplings, (*iii*) loose or dis-engaging couplings and (*iv*) non-aligned couplings.



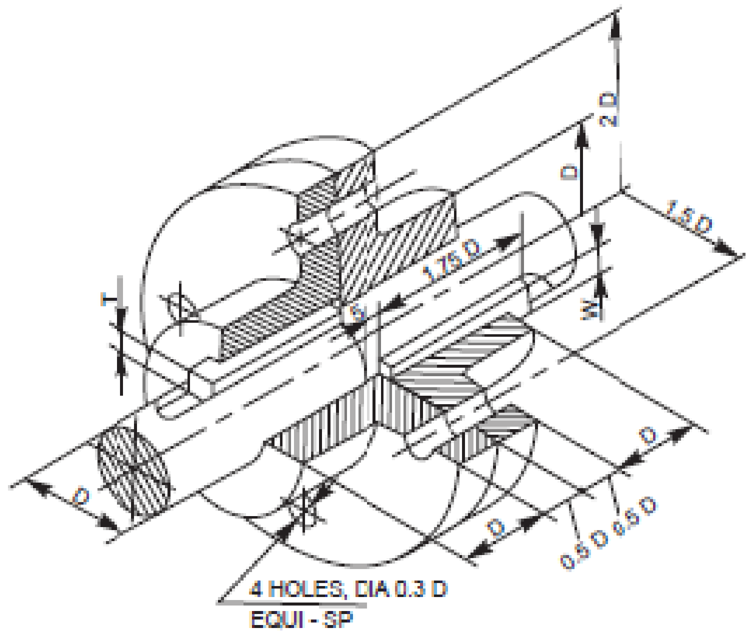


Split muff coupling

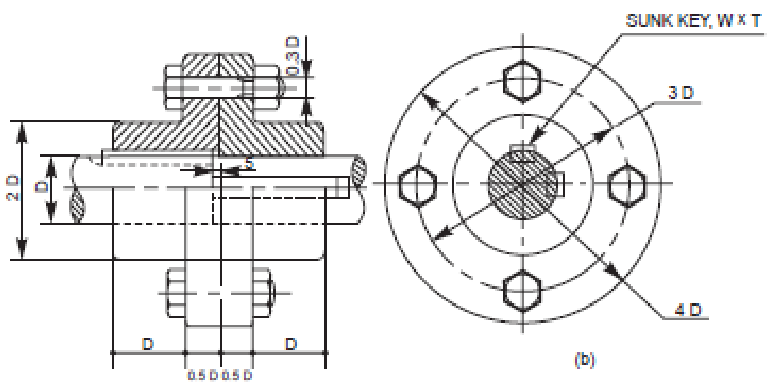


Oldham’s coupling

Exercise No. 29

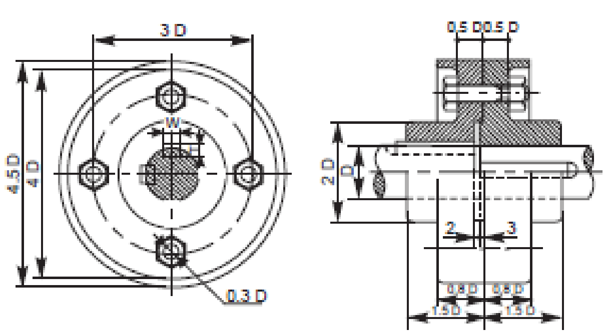


(a)

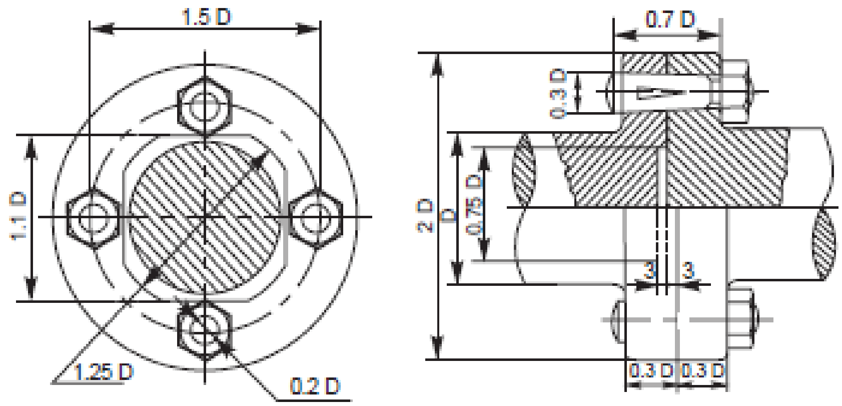


(b)

Fig. 7.4 Flanged coupling

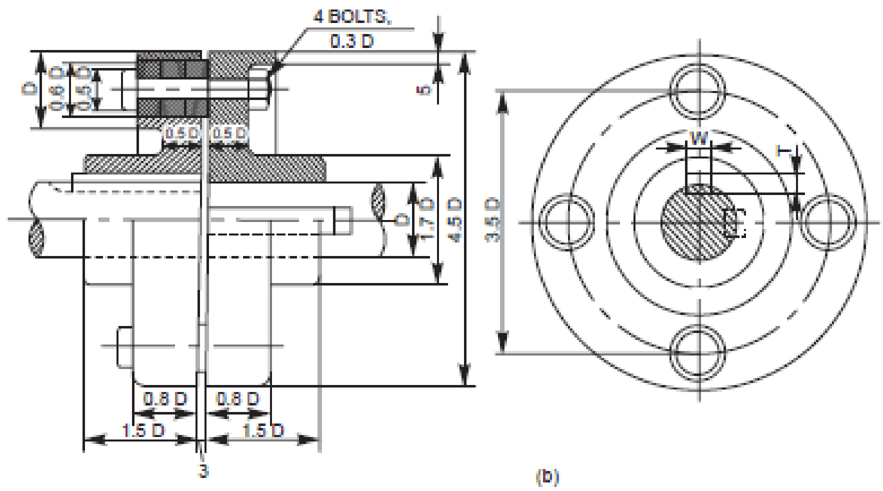


Exercise No. 30



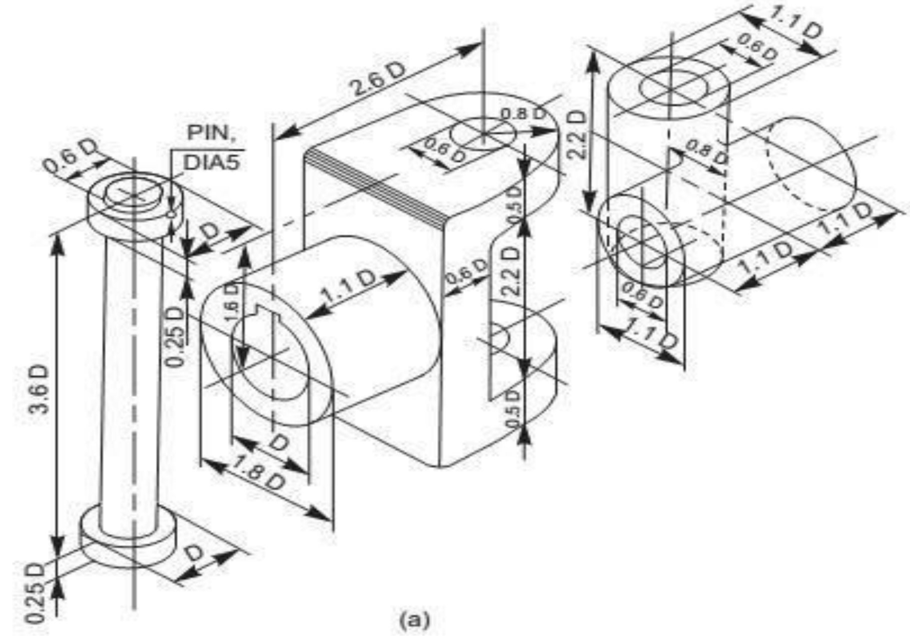
Solid flanged coupling

Draw (*a*) half sectional view from the front, top half in section and (*b*) view from the side of a bushed pin type flange coupling, indicating proportions to connect two shafts, each of diameter 30 mm.

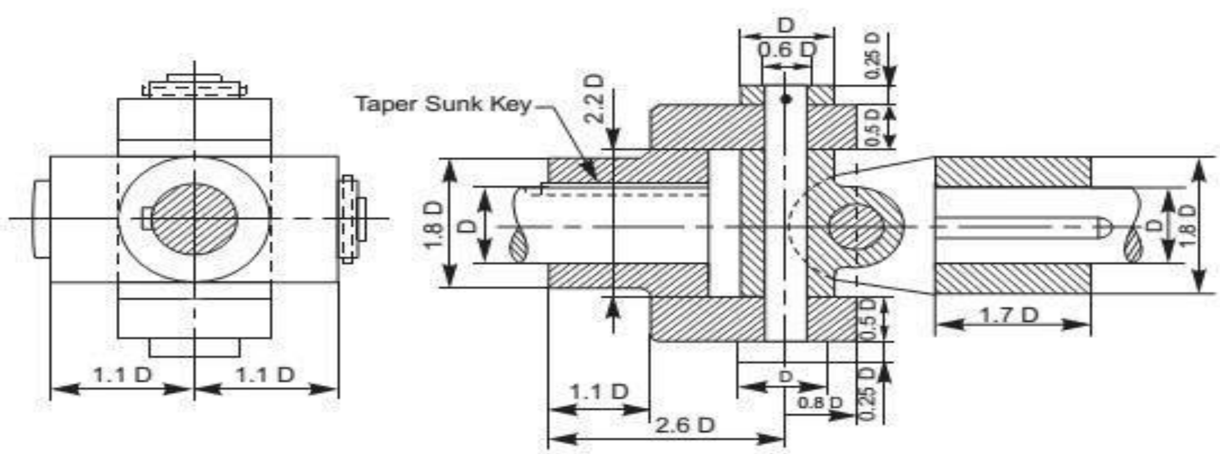


Bushed pin type flanged coupling

Exercise No. 31



(a)



(b)

Universal coupling

Exercise No. 32

Week-IX Bearings

Bearings are supports for shafts, providing stability, and free and smooth rotation. The importance of bearings may be understood from the supporting requirement of machine tool spindles, engine crankshafts, transmission or line shafts in workshops, etc. Bearings are broadly classified into two categories: sliding contact bearings and rolling contact bearings or antifriction bearings.

Journal Bearings

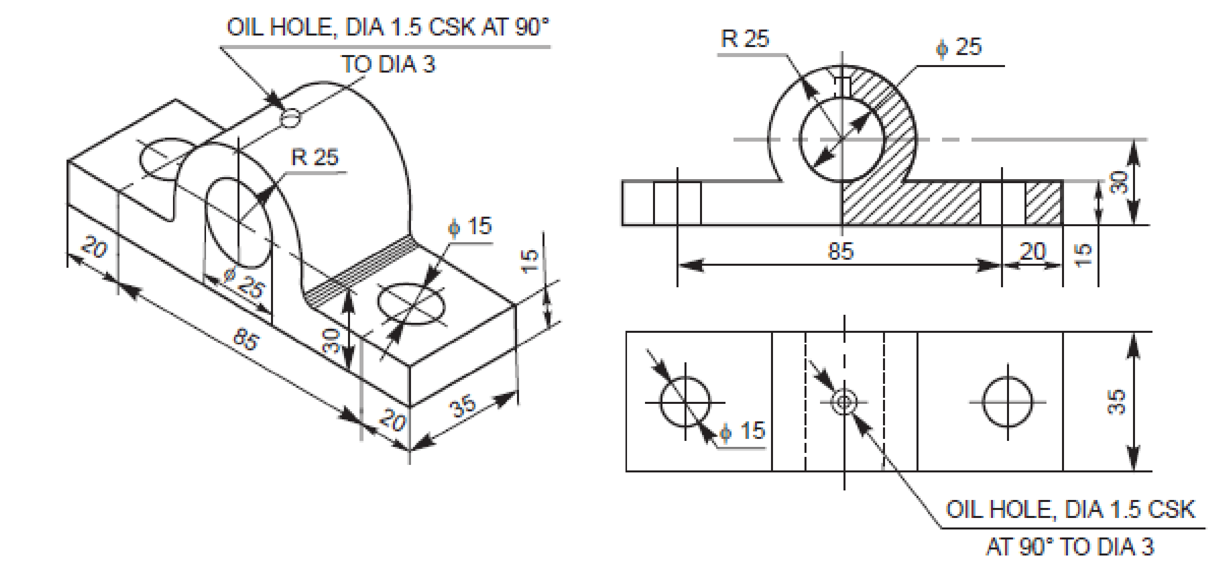


Fig. 1.0 Solid journal bearing

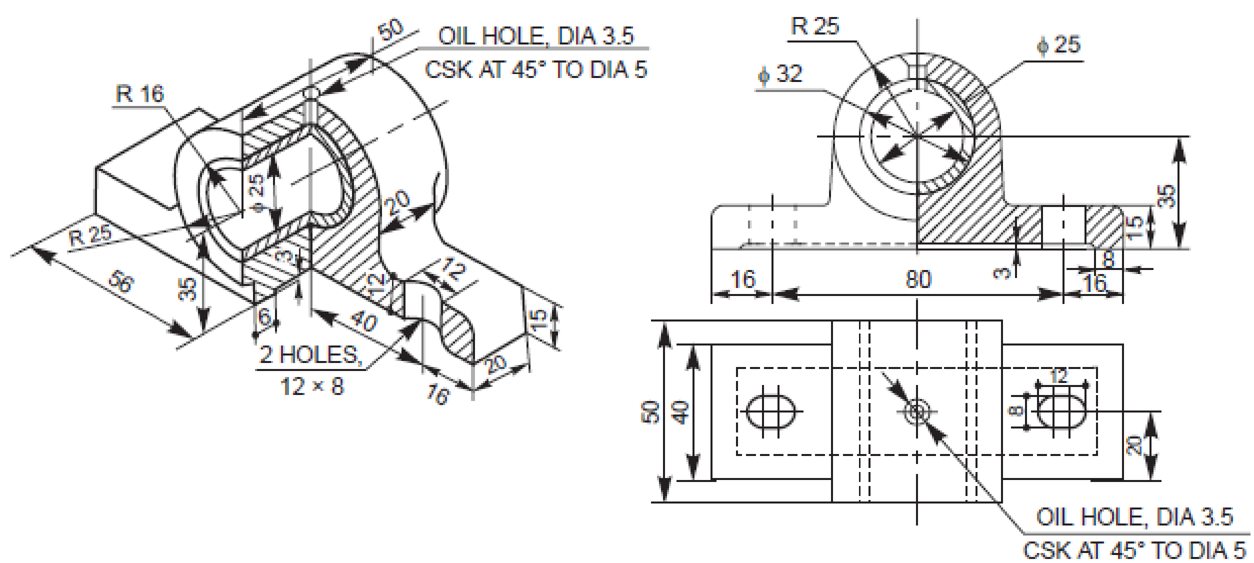
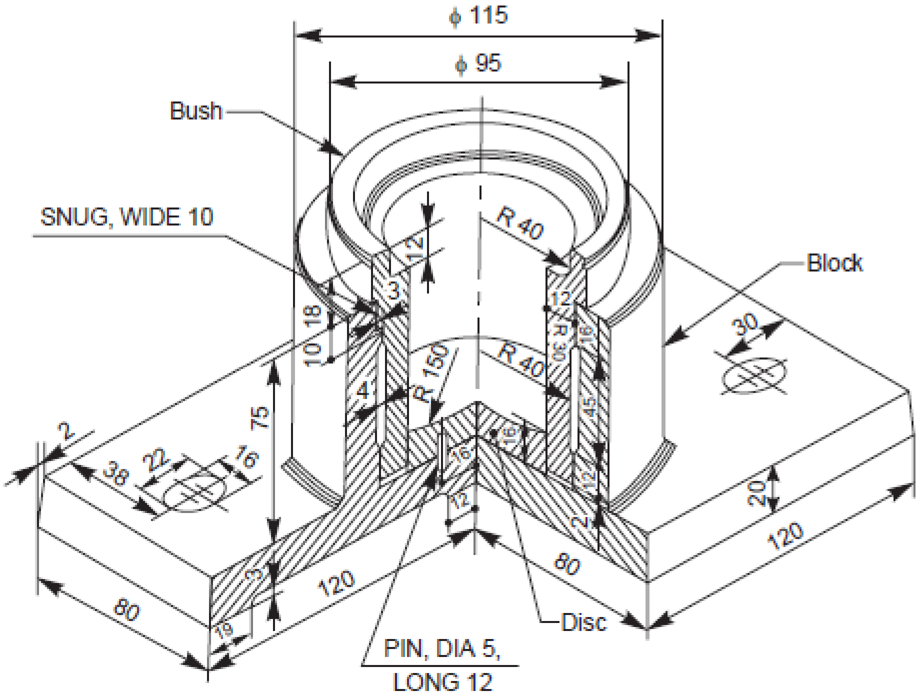


Fig. 1.1 Bushed journal bearing

Exercise No. 33

Pivot or foot-step bearing

This bearing is used to support a vertical shaft under axial load. Further, in this, the shaft is terminated at the bearing. The bottom surface of the shaft rests on the surface of the bearing which is in the form of a disc. The bush fitted in the main body supports the shaft in position and takes care of possible radial loads coming on the shaft.



Draw (*a*) sectional view from the front and (*b*) view from above of a foot-step bearing with radial and thrust ball bearings, suitable for supporting a shaft of diameter 60mm.

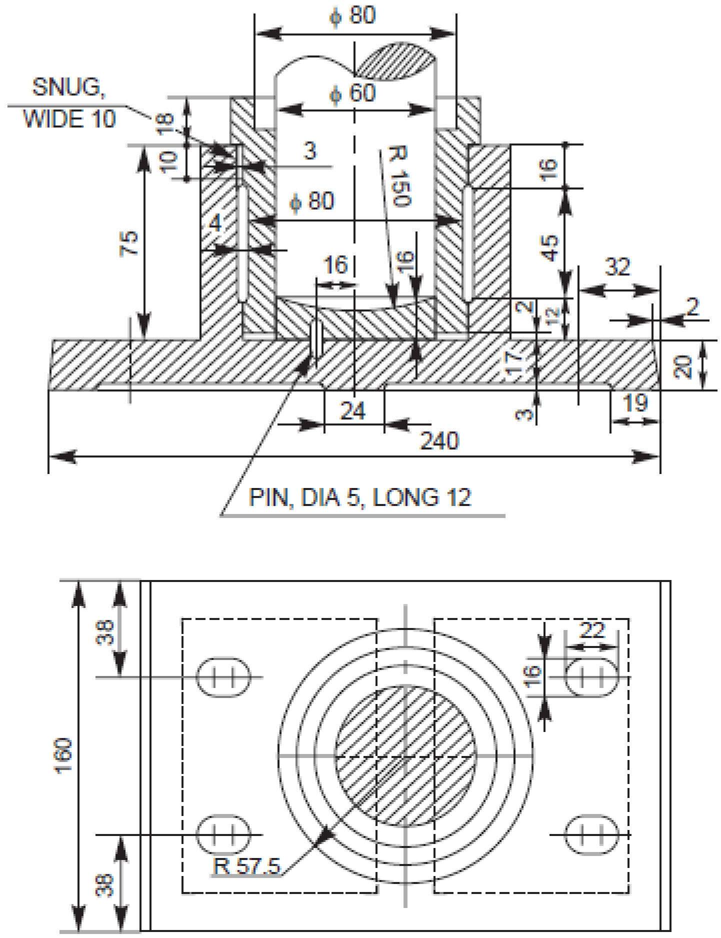


Fig. 1.2 Foot-step bearing

Collar thrust bearing

This is generally used for supporting a horizontal shaft under axial load. Further, in this, the shaft extends through and beyond the bearing. The shaft in a collar thrust bearing may consists of one or more collars which are either fitted to or integral with the shaft (Fig. 1.3). The collars rotate against the stationary split bearing surfaces.

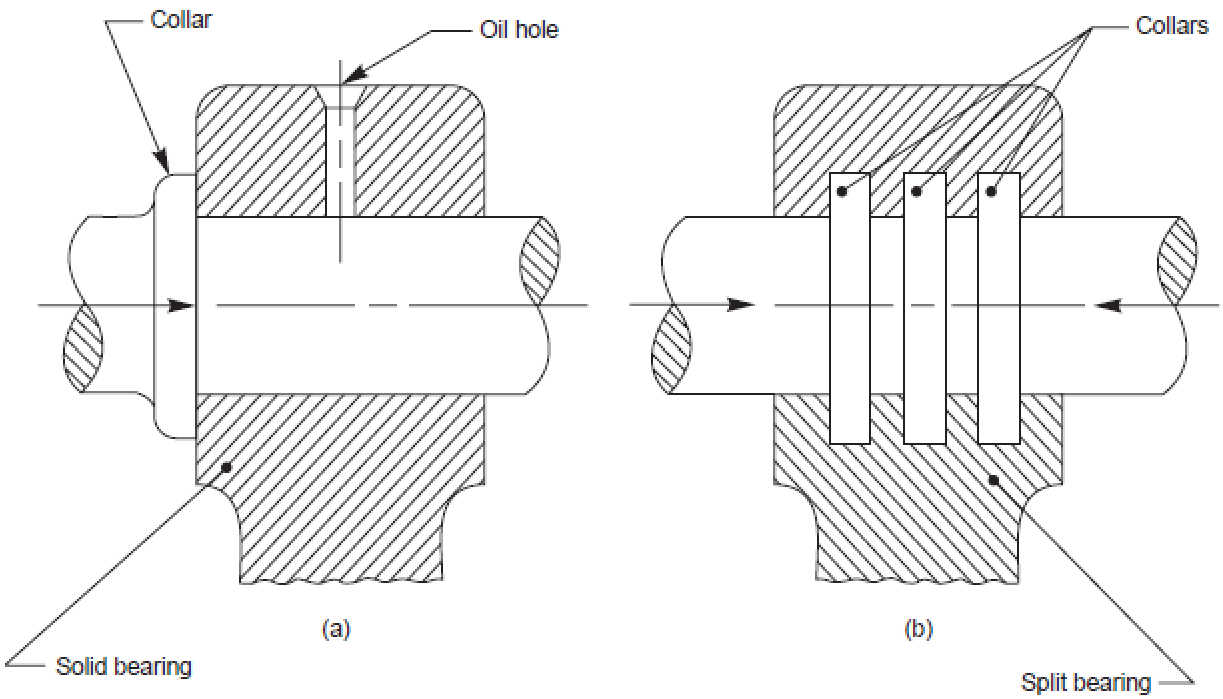


Fig. 1.3 (*a*) Single collar bearing, (*b*) Multi-collar bearing

Exercise No. 34

Week-X Assembly Drawings

ENGINE PARTS

Assembly of Stuffing Box

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 reciprocates through the cylinder cover; stuffing box provided in the cylinder cover, prevents leakage of steam from the cylinder.

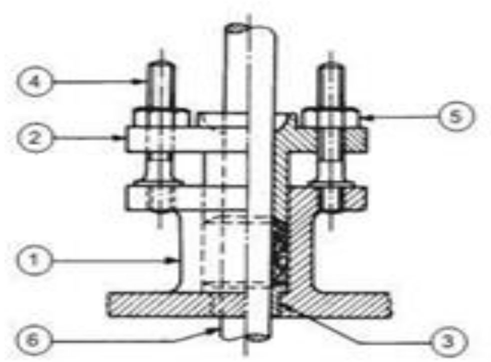
Figure 1.0 shows the various parts of a stuffing box. At the base of stuffing box body 1, a bush 3 is placed such that the bevelled edge of the bush is at the inner side of the body. Gland 2 is placed at the other end of the body and is connected to the main body by means of studs 4 and nuts

The space between the reciprocating rod and the bush and the gland is packed with a packing material such as mineral fibres, leather, rubber or cork.

Exercise

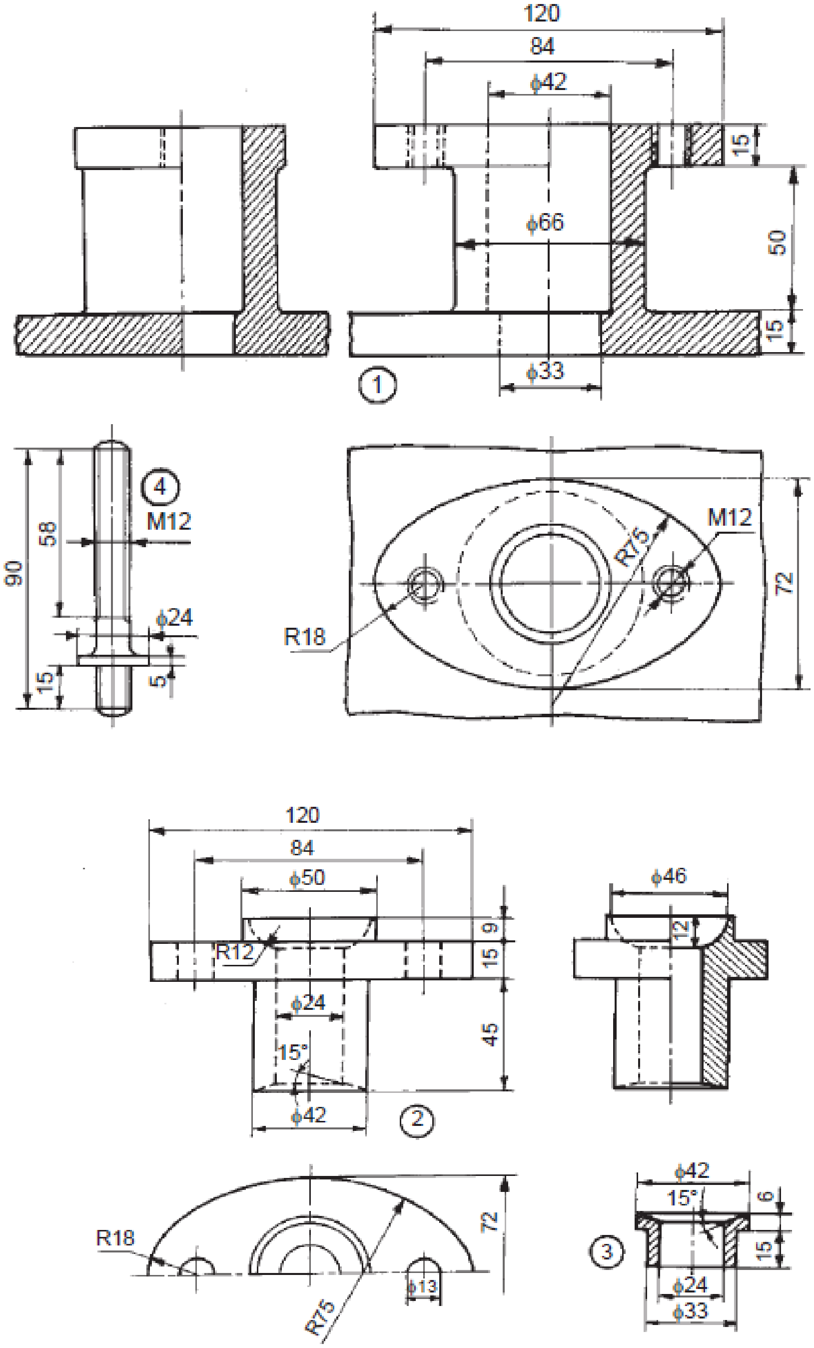
Assemble all parts of the stuffing box for a vertical steam engine, shown in Fig. 1.0 and draw, (*i*) half sectional view from the front, with left half in section, (*ii*) half sectional view from the right and (*iii*) view from above.

Stuffing Box Assembly



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

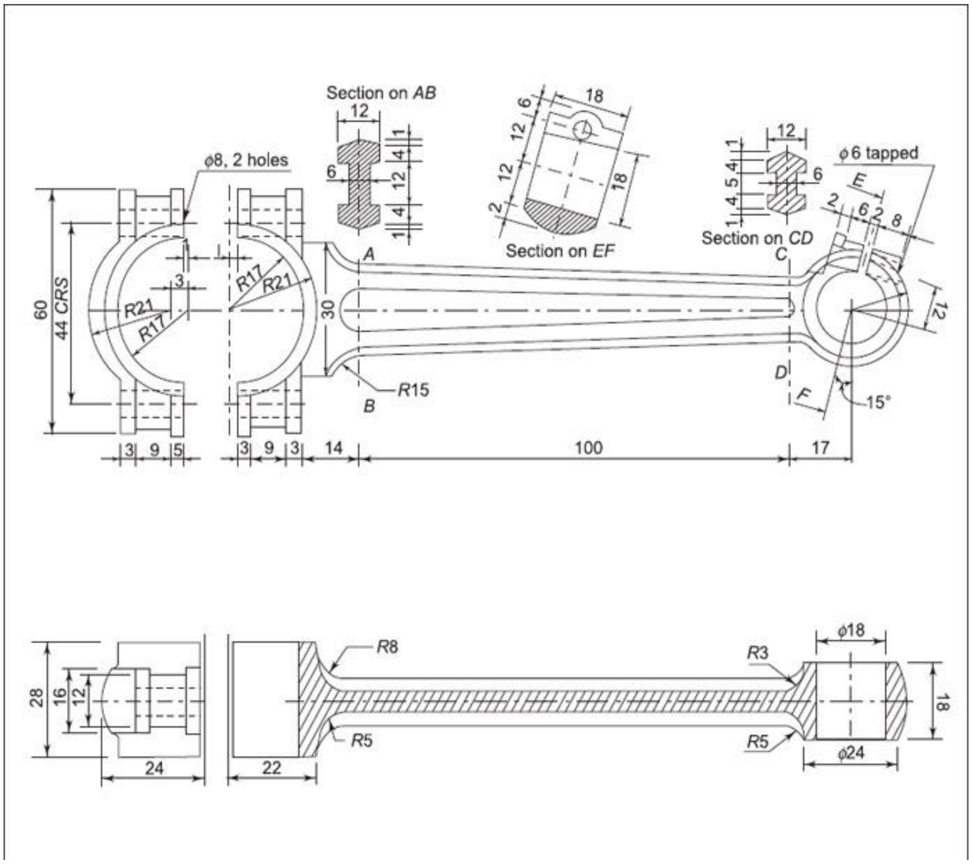


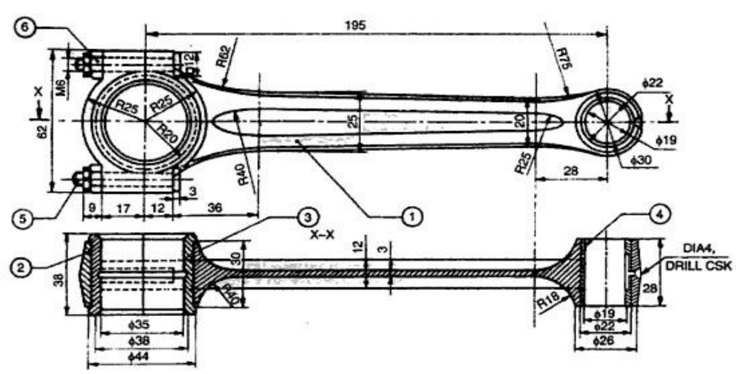
Parts list

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

Fig. 1.0 Stuffing box

Exercise No. 35 **Assembly of Connecting Rod**



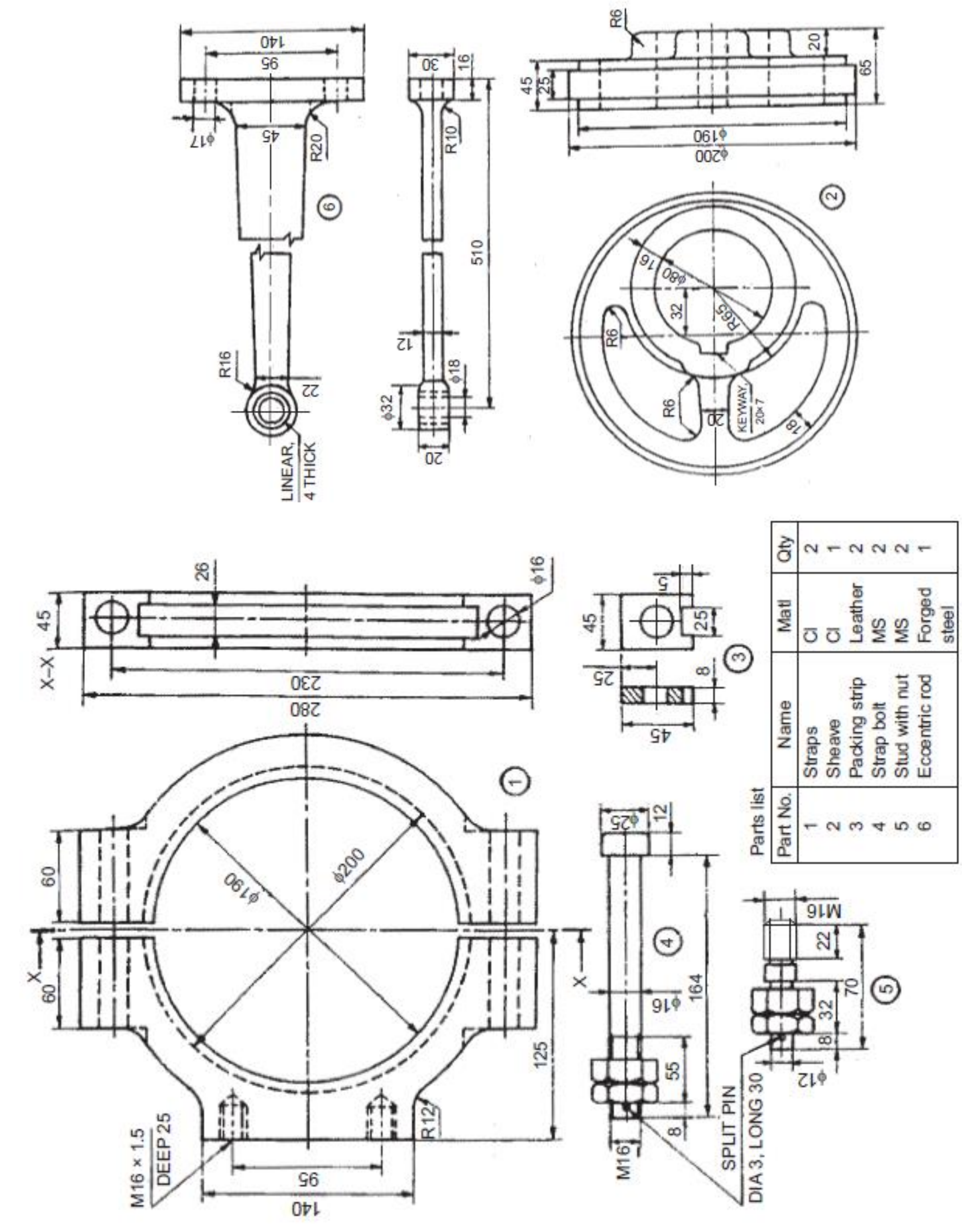


Parts list

Part No.	Name	Matl.	Qty.
1	Rod	FS	1
2	Cap	FS	1
3	Bearing brass	GM	2
4	Bearing bush	P Bronze	1
5	Bolt	MCS	2
6	Nut	MCS	2

Exercise No. 36 **Assembly of eccentric**

The details of an eccentric are shown in **Fig. 2.** Assemble the parts and draw, (*i*) half sectional view from the front, with top half in section, (*ii*) view from the right and (*iii*) view from above.

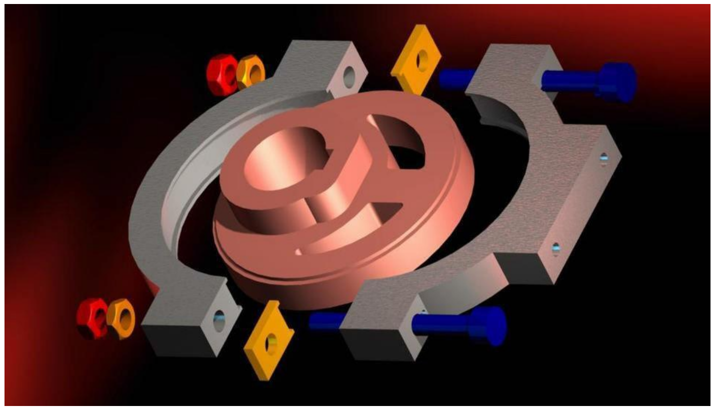


Part No.	Name	Matl	Qty
1	Straps	CI	2
2	Sheave	CI	1
3	Packing strip	Leather	2
4	Strap bolt	MS	2
5	Stud with nut	MS	2
6	Eccentric rod	Forged steel	1

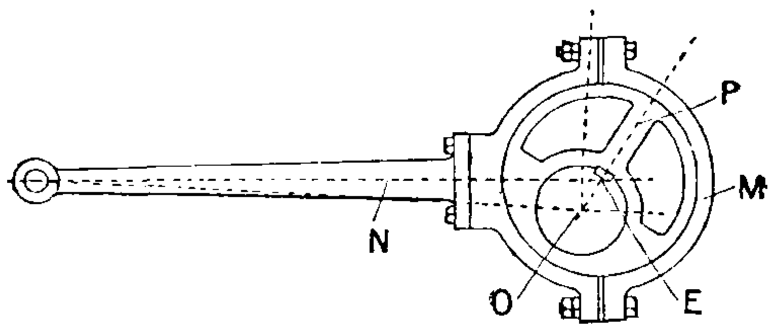
Parts list

Details of an eccentric

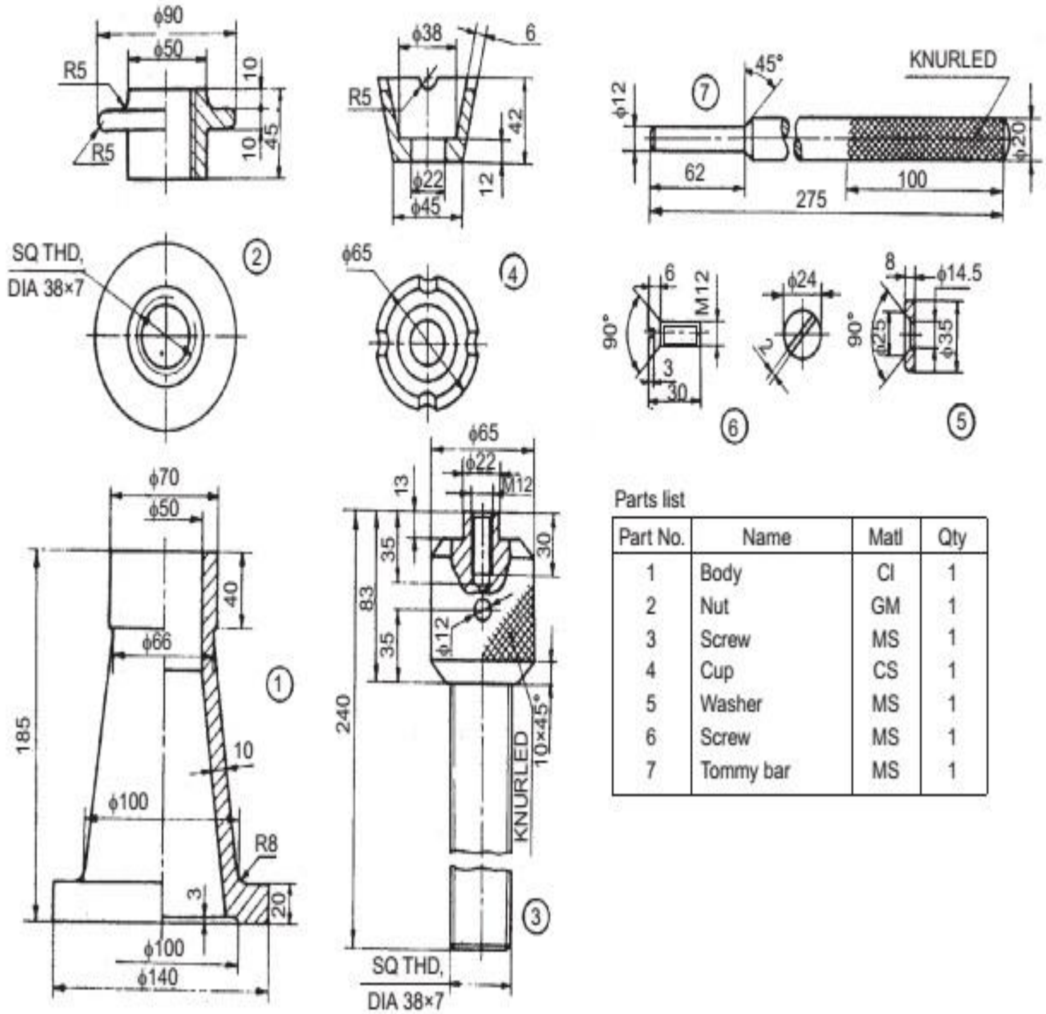
Assembly Model of an Eccentric



Assembly Drawing of an Eccentric

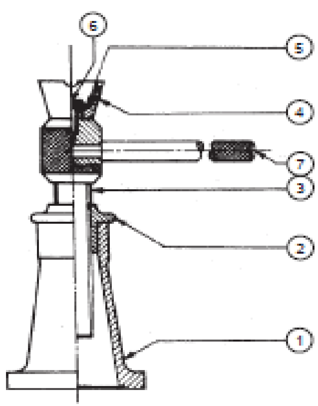


Exercise No. 37 Assembly of Screw Jack

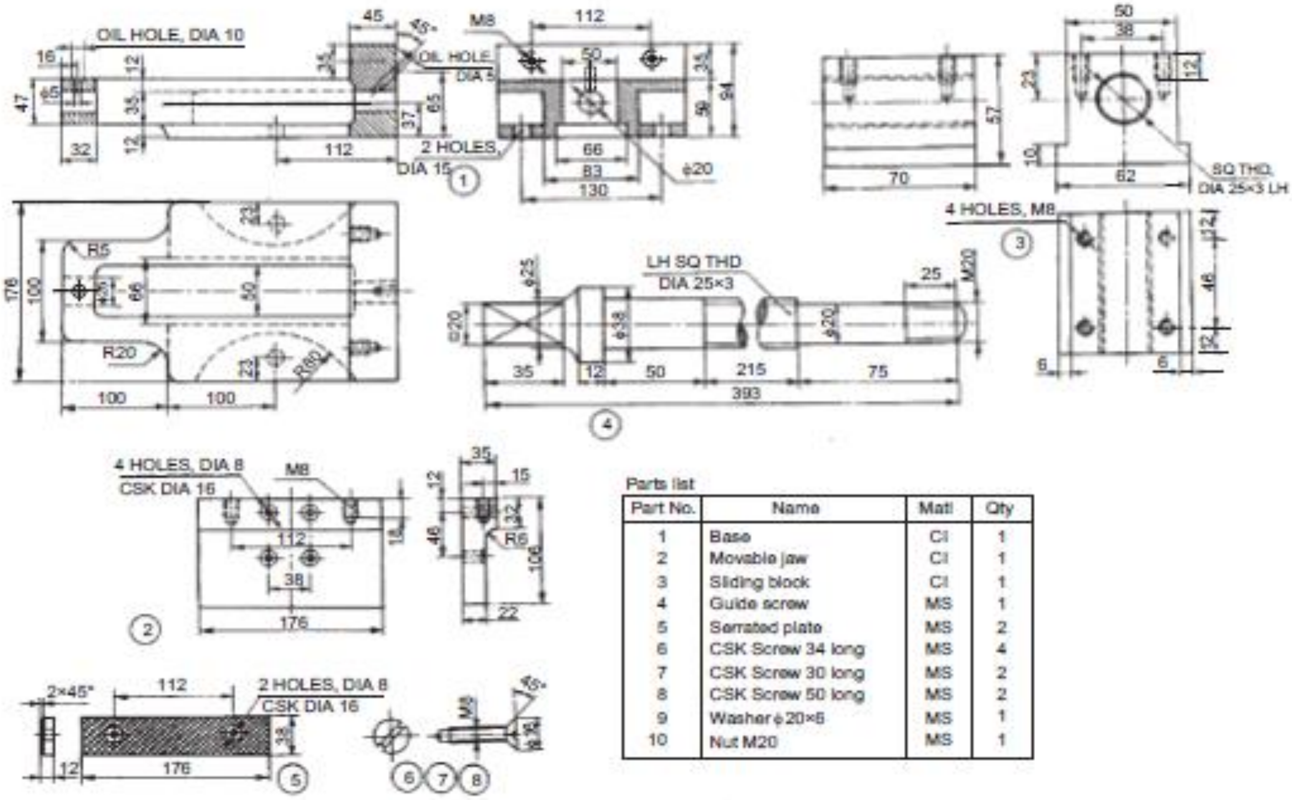


Part No.	Name	Matl	Qty
1	Body	CI	1
2	Nut	GM	1
3	Screw	MS	1
4	Cup	CS	1
5	Washer	MS	1
6	Screw	MS	1
7	Tommy bar	MS	1

Screw Jack Assembly



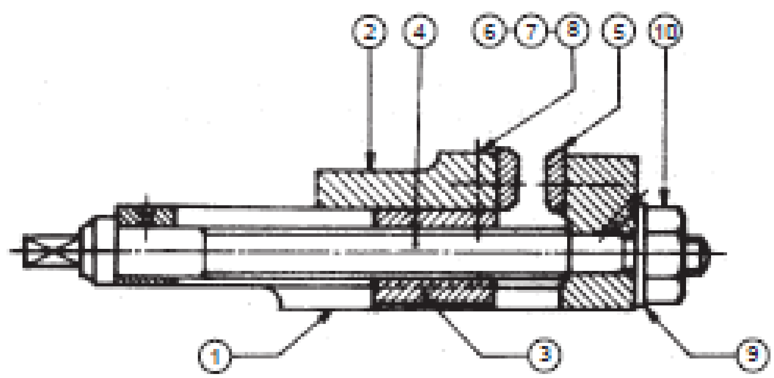
Exercise No. 38 Assembly of Machine vice



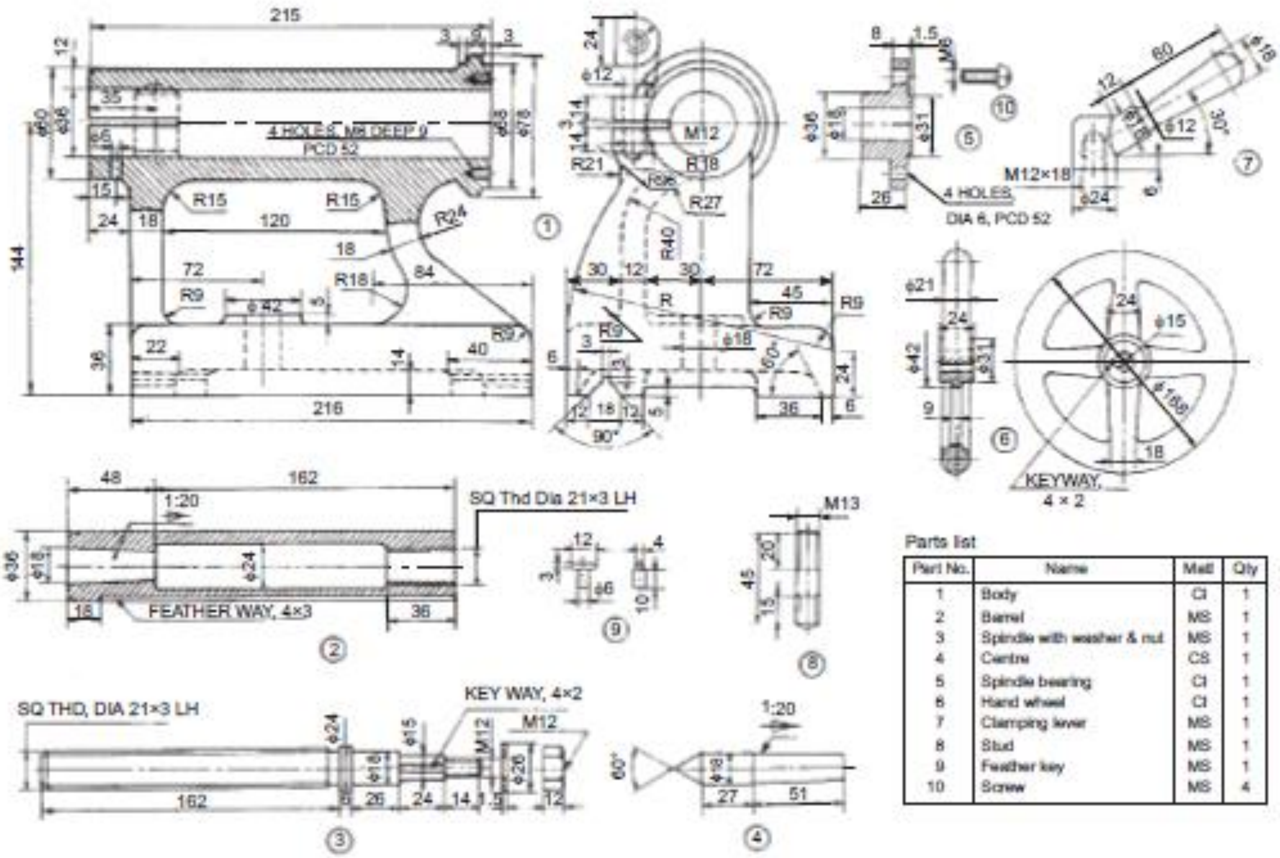
Parts list

Part No.	Name	Matl	Qty
1	Base	CI	1
2	Movable jaw	CI	1
3	Sliding block	CI	1
4	Guide screw	MS	1
5	Serrated plate	MS	2
6	CSK Screw 34 long	MS	4
7	CSK Screw 30 long	MS	2
8	CSK Screw 50 long	MS	2
9	Washer ϕ 20×6	MS	1
10	Nut M20	MS	1

Assembly Drawing :



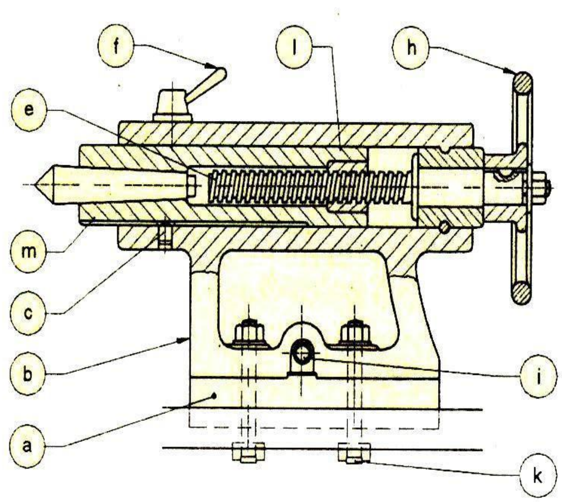
Exercise No. 39 Assembly of lathe tailstock



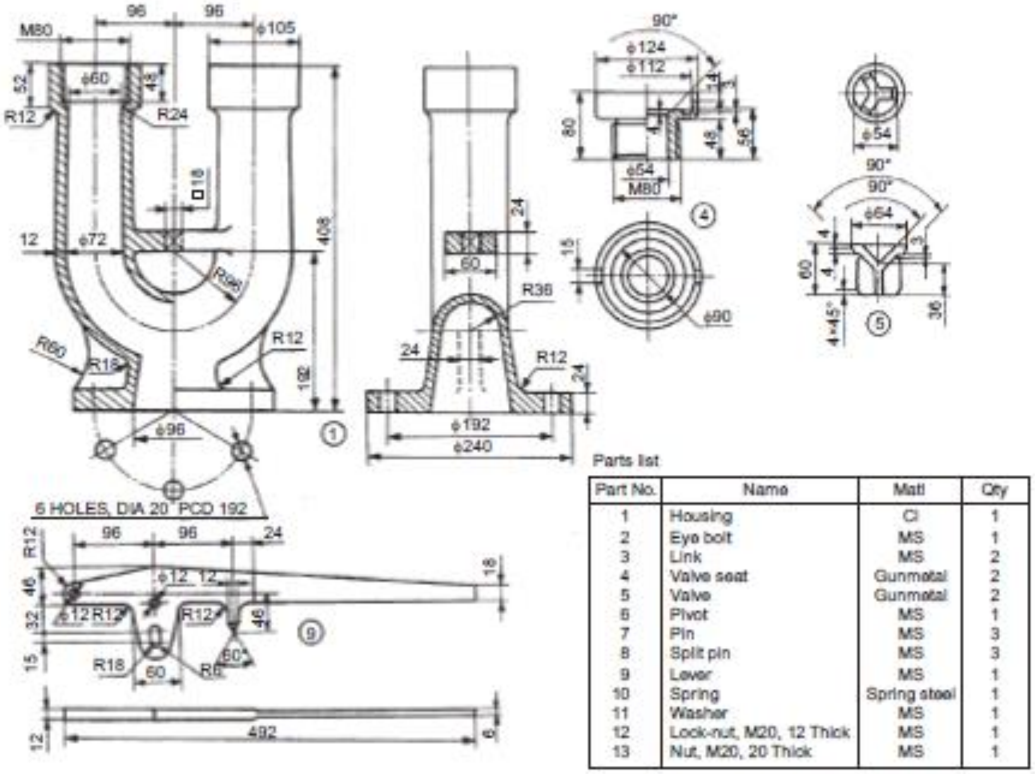
Parts list

Part No.	Name	Matl	Qty
1	Body	CI	1
2	Barrel	MS	1
3	Spindle with washer & nut	MS	1
4	Centre	CS	1
5	Spindle bearing	CI	1
6	Hand wheel	CI	1
7	Clamping lever	MS	1
8	Stud	MS	1
9	Feather key	MS	1
10	Screw	MS	4

Tail Stock Assembly



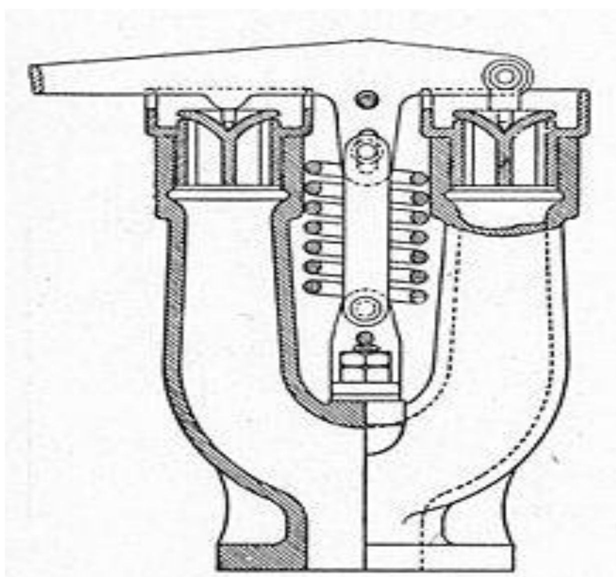
Exercise No.40 **Assembly of Ramsbottom Safety Valve**



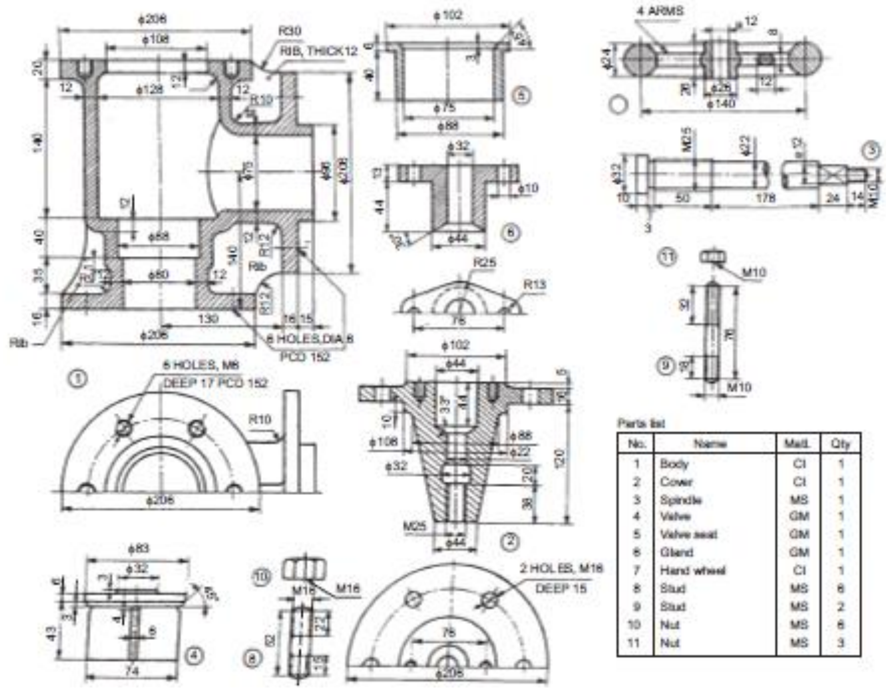
Parts list

Part No.	Name	Matl	Qty
1	Housing	CI	1
2	Eye bolt	MS	1
3	Link	MS	2
4	Valve seat	Gunmetal	2
5	Valve	Gunmetal	2
6	Pivot	MS	1
7	Pin	MS	3
8	Split pin	MS	3
9	Lever	MS	1
10	Spring	Spring steel	1
11	Washer	MS	1
12	Lock-nut, M20, 12 Thick	MS	1
13	Nut, M20, 20 Thick	MS	1

Assembly Of Ramsbottom Safety Valve



Exercise No.41 **Assembly of Feed Check Valve**



Parts list

No.	Name	Matl.	Qty
1	Body	CI	1
2	Cover	CI	1
3	Spindle	MS	1
4	Valve	GM	1
5	Valve seat	GM	1
6	Gland	GM	1
7	Hand wheel	CI	1
8	Stud	MS	6
9	Stud	MS	2
10	Nut	MS	6
11	Nut	MS	3

Feed Check Valve Assembly