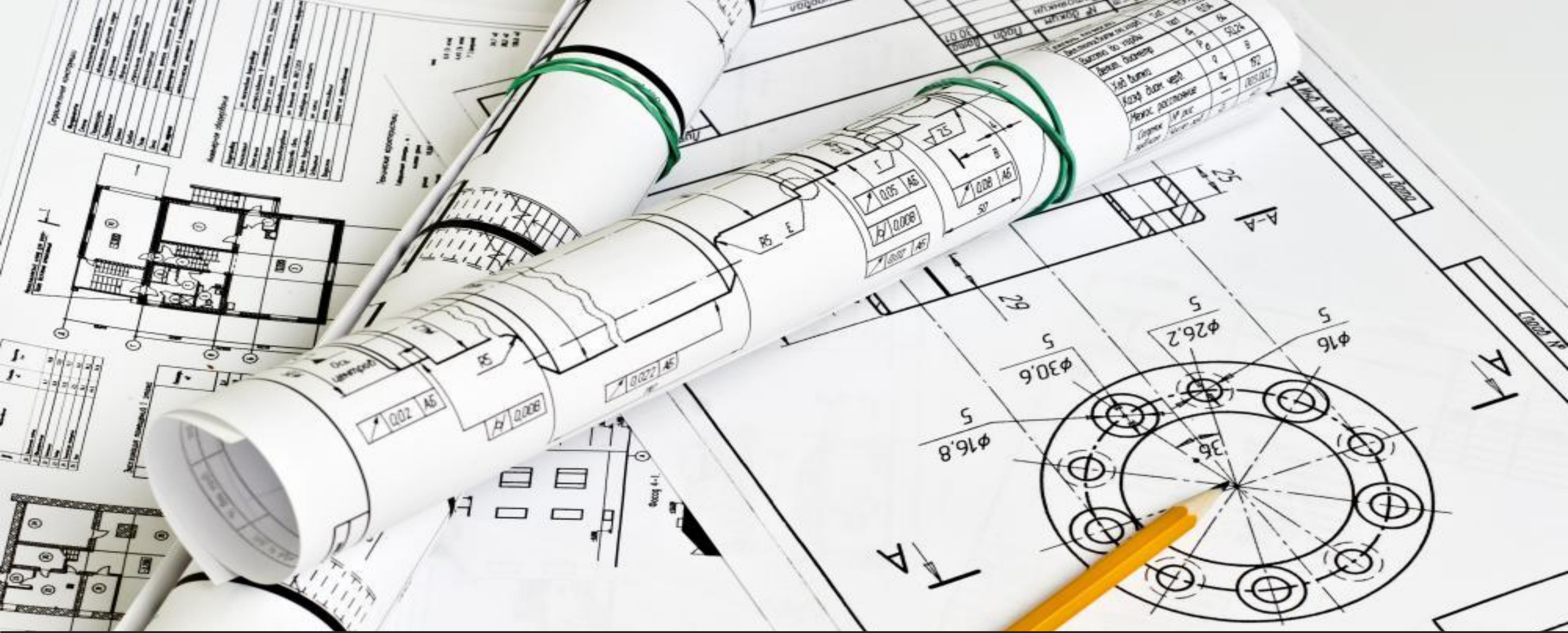

ENGINEERING DRAWING

MR/ES/CE/RN 156

STUDY SLIDES COMPILED BY

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Course Outline

- Lesson 1. INTRODUCTION TO AUTOCAD
- Lesson 2. CONVENTIONAL REPRESENTATION
- Revision. ORTHOGRAPHIC PROJECTION
- Lesson 3. SECTIONS
- Lesson 4. ASSEMBLY DRAWINGS
- Lesson 5. PROJECTION OF SOLIDS
- Lesson 6. DEVELOPMENT OF SOLIDS/SURFACES
- Lesson 7. CURVES OF INTERSECTION

Course Objectives

Objective 1. To understand the conventional symbols and how to use them in detail drawings.

Objective 2. To understand what sectioning a part or assembly is and when to apply them.

Objective 3. To understand the term assembly drawings, types of assembly drawings and how to assemble parts together, creating part lists.

Objective 4. To be enlightened about the need to develop solids and how to develop various shades of solids.

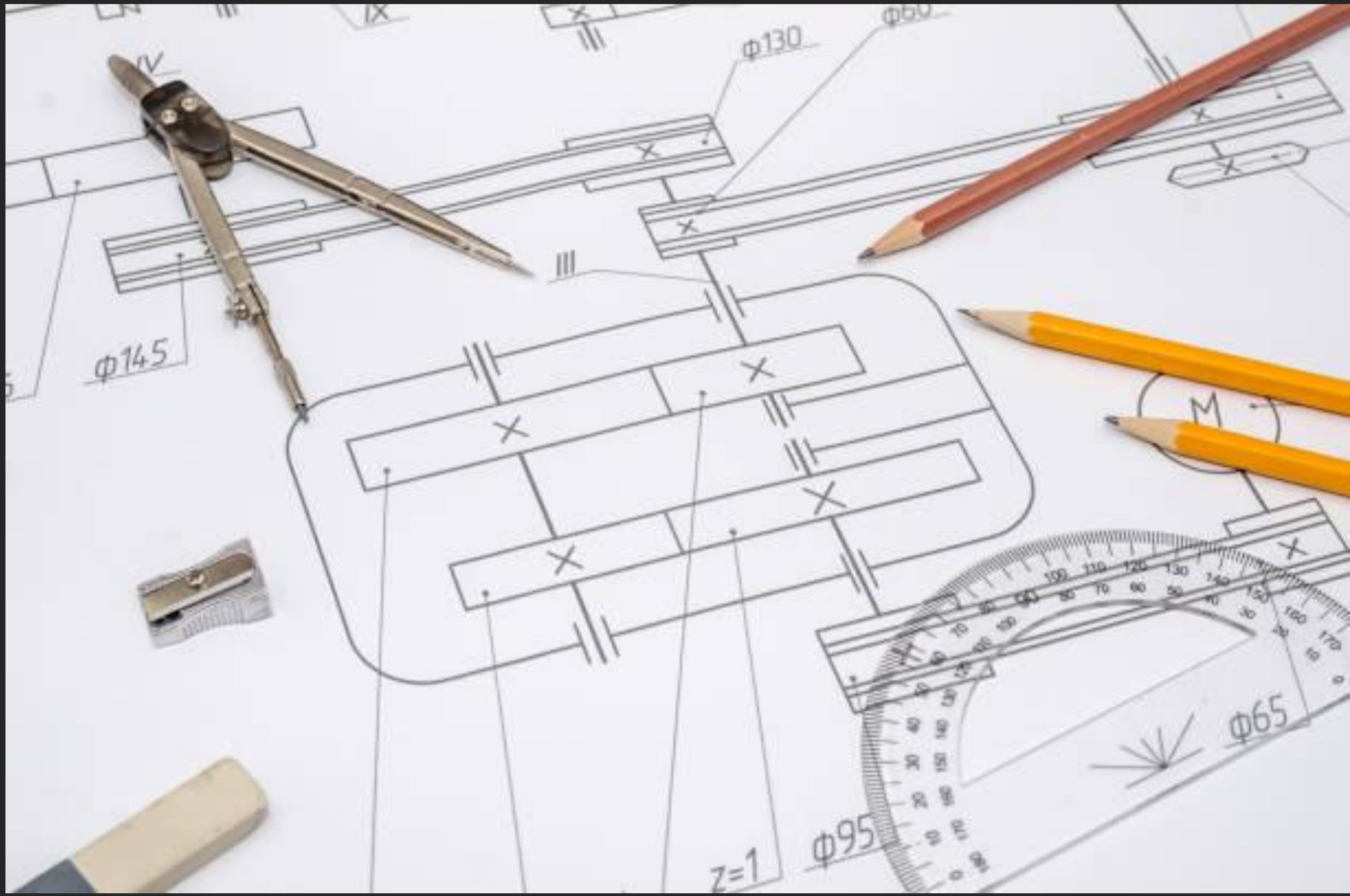
Objective 5. To understand what CAD is, the various kinds of CAD software and their applications.

Objective 5. To understand what CAD is, the various kinds of CAD software and their applications.

INTRODUCTION TO AUTOCAD

We will cover these skills:

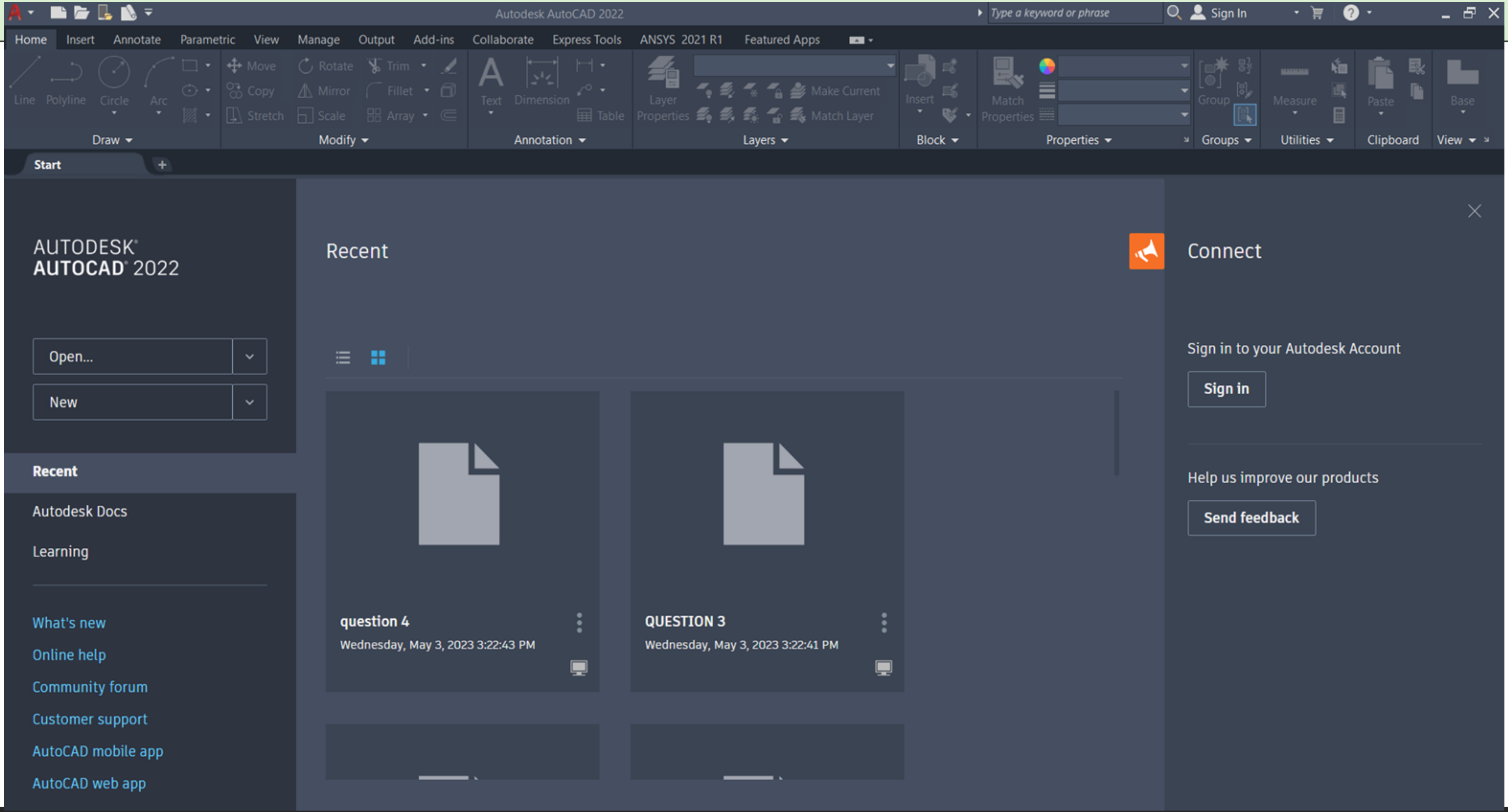
- USER INTERFACE AND TEMPLATES.
- DRAWING AND MODIFYING TOOLS.
- TEXT, DIMENSION AND TABLES.
- LAYERS, TITLE BLOCK AND VIEWPORT CREATION.
- DRAWING WITH AUTOCAD



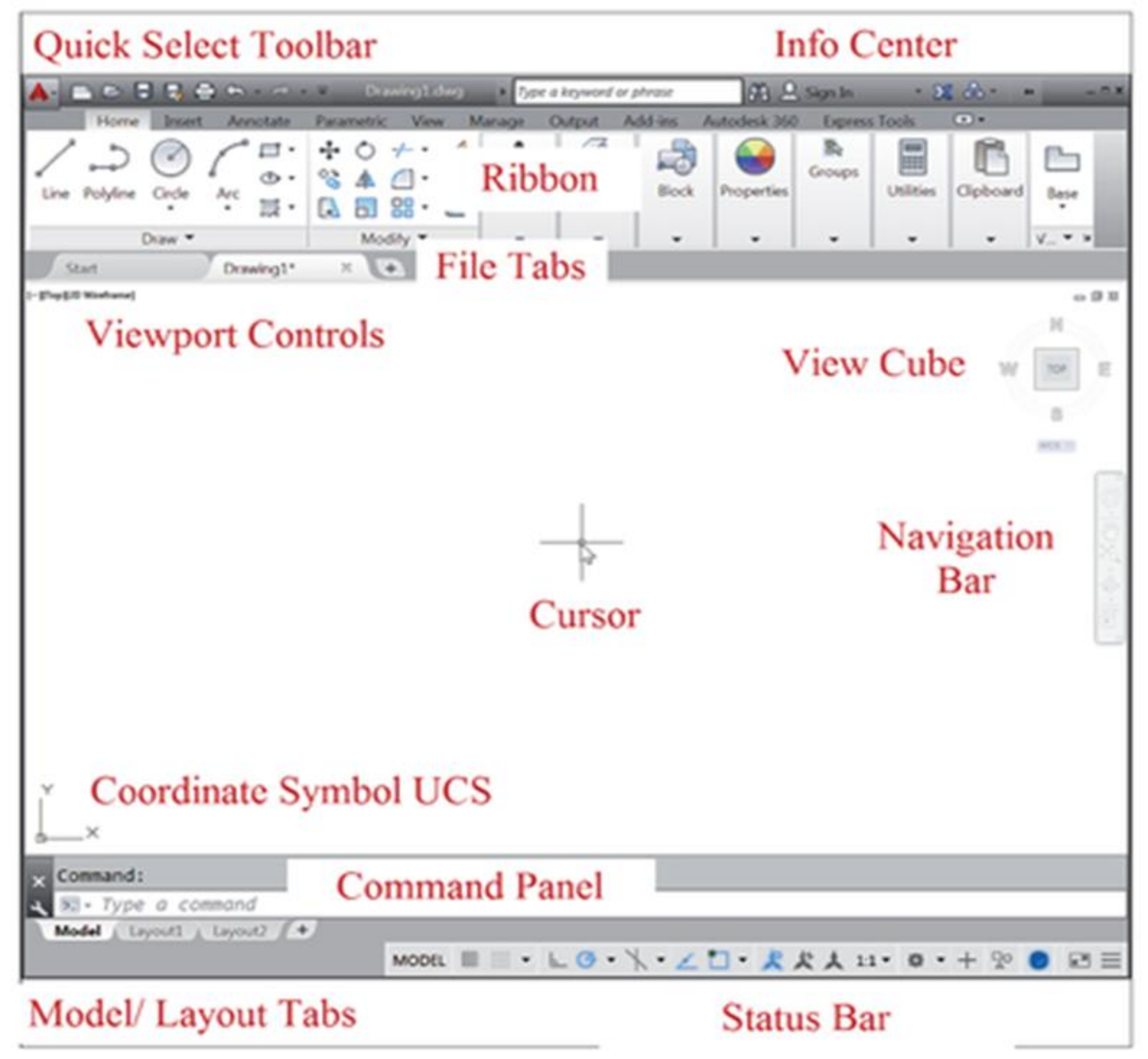
WHAT IS AUTOCAD?

Automatic Computer Aided Drawing/Drafting software as its full name reads, is a computer software used for drawing and designing. Its is the most widely used drafting software in the world of Engineering. It can also be used by other Drawing Professionals like Graphic Designers, Fashionista, Interior Designer, Landscaper and Educators.

USER INTERFACE – START PAGE



START PAGE



Quick Select Toolbar
Info Center
Ribbon
File Tabs
Viewport Controls
View Cube
Navigation Bar
Cursor
Coordinate Symbol UCS
Command Panel
Model/ Layout Tabs
Status Bar

ACCURACY TOOLS/DRAWING AIDS

Grid: also toggled using the F7 key. When set on, a grid pattern appears in the drawing area.

Snap Mode: also toggled using the F9 key. When set on, the cursor under mouse control can only be moved in jumps from one snap point to another. Ortho Mode: also toggled using the F8 key. When set on, features can only be drawn vertically or horizontally.

Polar Tracking: also toggled using the F10 key. When set on, a small tip appears showing the direction and length of lines etc. in degrees and units.

Object Snap Tracking: also toggled by the F11 key. When set on, lines etc. can be drawn at exact coordinate points and precise angles.

2D Object Snap: also toggled using the F3 key. When set on, an osnap icon appears at the cursor pick box.

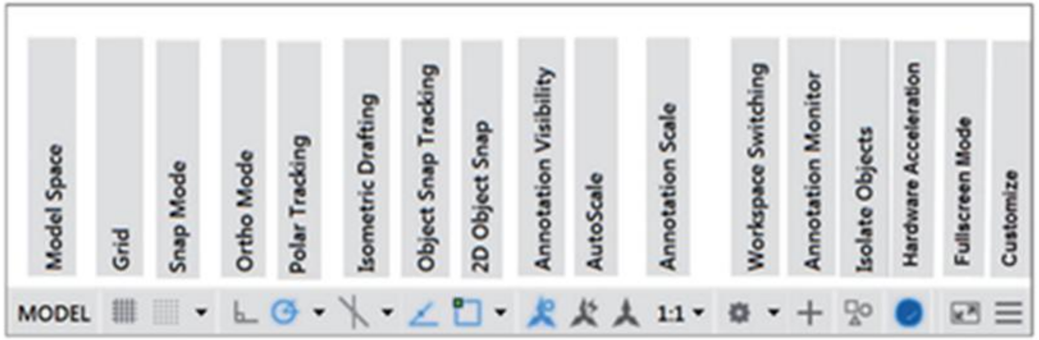
Isoplane: Cycles through 2-1/2D isoplane settings.

Dynamic UCS: turns on UCS alignment with planar surfaces.

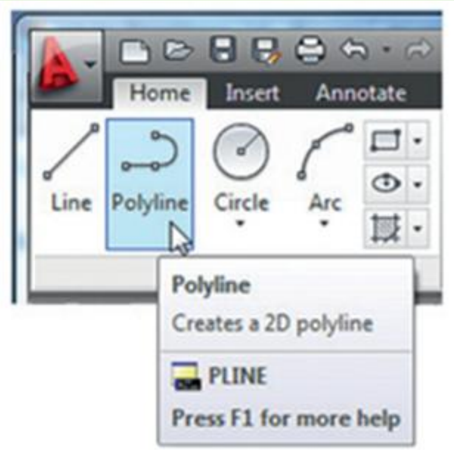
Dynamic input: displays distances and angles near the cursor and accepts input as you use Tab between the fields.

Object Snap Tracking: also toggled by the F11 key. When set on, lines etc. can be drawn at exact coordinate points and precise angles.

2D Object Snap: also toggled using the F3 key. When set on, an osnap icon appears at the cursor pick box.



DRAWING TOOLS



1. Line

You can invoke the LINE command by choosing the LINE tool from the Draw panel, or you can also invoke the LINE tool by entering LINE or L at the Command Prompt. You will have to specify the starting point of the line by clicking the mouse then you will be prompted to specify the second point. You can terminate the LINE command by pressing ENTER, ESC or SPACEBAR.

2. Circle

A circle is drawn by using the CIRCLE command. You can draw a circle by using six different tools, i.e., by specifying center and radius, by specifying center and diameter, by specifying two diametrical ends, by specifying three points on a circle, tangent to two objects, tang

3. Rectangle

You can draw rectangles by specifying two opposite corners of the rectangle, specifying the area and the size of one of the sides, or specifying the rectangle’s dimensions.

4. Polyline

Polylines means many lines. To draw a polyline, you need to invoke the PLINE command. A rectangle drawn with the **Polyline** tool is a single object. Lines of different thickness, arcs, arrows and circles can all be drawn using this tool.

DRAWING TOOLS

5. Arc

The Arc tool is used to create an arc. It provides various ways in which to draw the arc.

6. Rectangle

The rectangle tool is used to draw a closed rectangle polyline.

7. Polygon

The polygon tool is used to draw objects from 3 sides.

8. Spline

Used for drawing a single wavy line with fit points.

9. Donut

It is used to create a filled circle or wide ring.

10. Ellipse

Creates an ellipse or an elliptical arc.

MODIFYING TOOLS

1. Move

The Move Tool is used to move one or more objects from their current location to a new location without changing their size or orientation.

2. Copy

A circle is drawn by using the CIRCLE command. You can draw a circle by using six different tools, i.e., by specifying center and radius, by specifying center and diameter, by specifying two diametrical ends, by specifying three points on a circle, tangent to two objects, tangent to three objects.

3. Rotate

On invoking this tool, you will be prompted to select the objects and the base point about which the selected objects will be rotated. By default, a positive angle results in counterclockwise rotation, whereas a negative angle results in a clockwise rotation.

4. Mirror

This tool is used to create a mirror copy of the selected objects. The objects can be mirrored at any angle. This tool is helpful in drawing symmetrical figures. On invoking this tool, you will be prompted to select objects. On selecting objects to be mirrored, you will be prompted to enter the first point of the mirror line and the second point of the mirror line. A mirror line is an imaginary line about which the objects are mirrored.

MODIFYING TOOLS

5. Scale

Sometimes you need to change the size of objects in a drawing. For this purpose, the Scale tool comes in handy.

6. Trim

When creating a design, you may need to remove the unwanted and extending edge. In such cases, you can use the Trim tool. On invoking the Trim tool, you will be prompted to select the cutting edges. These edges can be lines, polylines, circles, arcs, ellipses, rays, splines, text, blocks, xlines or even viewports. After the cutting edge/edges are selected, you must select each object to be trimmed.

7. Extend

The Extend tool may be considered as the opposite of the Trim tool. You can extend lines, polylines, rays, and arcs to meet the other objects using the Extend tool. You can use this option whenever you want to extend the objects that do not actually intersect the boundary edge but would intersect its edge if the boundary edges were extended.

8. Fillet

The edges in a model are generally filleted to reduce the area of stress concentration. The fillet tool helps form round corners between any two entities that form a sharp vertex.

9. Chamfer

The chamfer is used to bevel the edges of objects.

10. Rectangular Array

It is used to create copies of a selected object in rows, columns and levels.

11. Polar Array

It creates an array by copying the selected objects around a specified center point or axis of rotation. Evenly distributing copied objects in circular form.

12. Erase

It is used to erase the unwanted objects from the objects drawn. To erase an object, choose Erase tool from the Modify panel. To invoke the Modify toolbar, choose View>Windows>Toolbars>AutoCAD>Modify from the ribbon. A small box, known as a pick box, replaces the screen cursor on invoking the Erase tool. To erase the object, select it by using the pick box; the selected object will be displayed in dashed lines, and the Select objects prompt will be displayed again. You can either continue selecting the objects or press ENTER to terminate the object selection process and erase the selected objects.

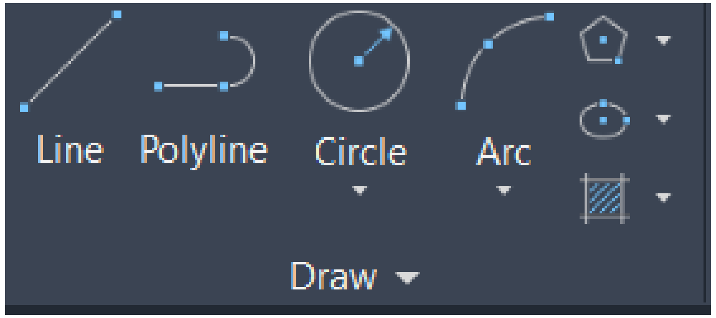
13. Explode

This tool is useful when you have inserted an entire drawing, and you need to alter a small detail. After you invoke the Explode tool, you are prompted to select the objects you want to explode. After selecting the objects, press ENTER or right-click to explode the selected objects and then end the command.

MODIFYING TOOLS

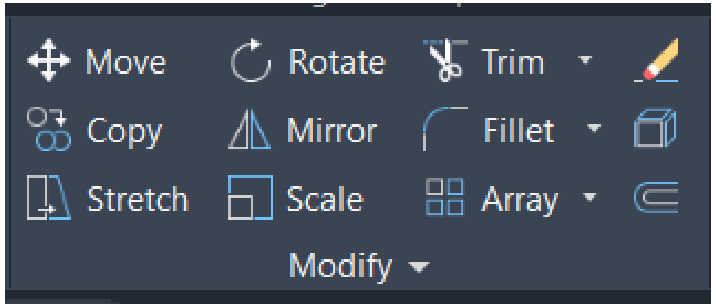
14. Offset

You can use the Offset tool to draw parallel lines, polylines, concentric circles, arcs, curves, etc., While offsetting an object, you need to specify the offset distance and the side to offset.



15. Break

It is used to break objects between two points.

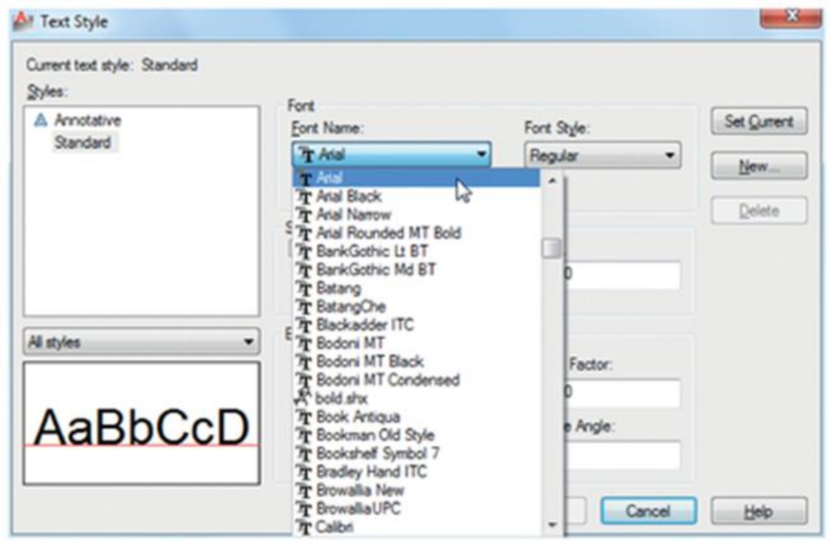


16. Join

It is used to combine series of finite linear and open curved objects at their common endpoints to create a single 2D or 3D object. It joins similar objects to form a single and unbroken object.

SETTING TEXT (TEXT STYLE)

1. At the keyboard: **Enter st (Style) right-click**
2. The **Text style dialog** appears (Fig. 32). In the dialog, enter 6 in the Height field. Then left-click on **Arial in the Font name popup list**. Arial font letters appear in the Preview area of the dialog.
3. Left-click the **New button** and enter **Arial** in the **New text style sub-dialog** that appears (Fig. 33) and click the **OK** button.
4. Left-click the **Set Current button** of the Text Style dialog.
5. Left-click the **Close button** of the dialog.



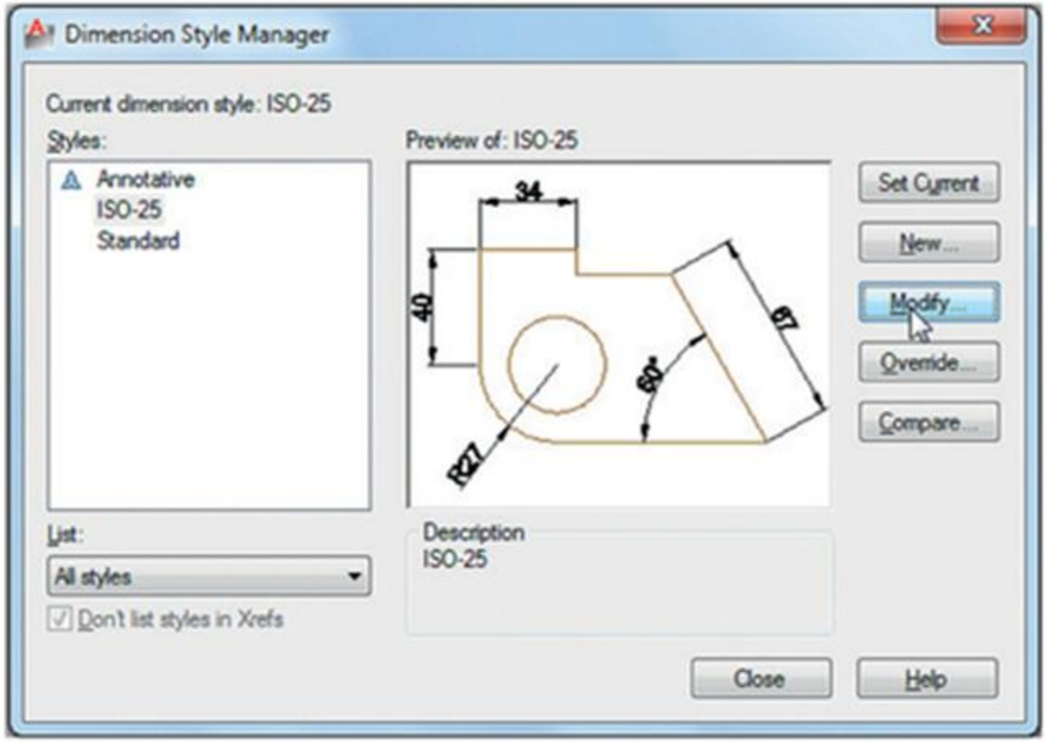
SETTING DIMENSION STYLE

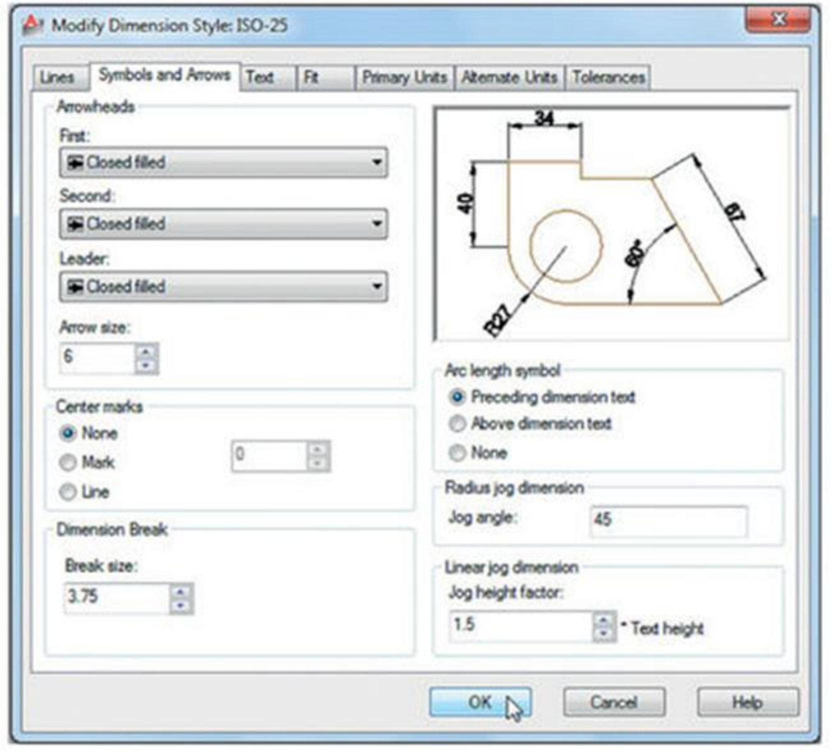
Settings for dimensions require making entries in a number of sub-dialogs in the **Dimension Style Manager**. To set the **dimension style**:

1. At the keyboard: Enter **d right-click** and the **Dimensions Style Manager** dialog appears (Fig. 35).
2. In the dialog, click the **Modify** . . . button.
3. The **Modify Dimension Style** dialog appears (Fig. 36). This dialog shows a number of tabs at the top of the dialog. Click the **Symbols and Arrows tab** and make settings as shown. Then click the **OK button** of that dialog.
4. The **original Dimension Style Manager** reappears. Click its **Modify button** again.
5. **The Modify Dimension Style dialog** reappears, click the **Line tab**. Set **Line to colour Magenta**. **Set Text style to Arial**, set **Color to Magenta**, set **Text Height to 6** and **click the ISO check box in the bottom right-hand corner** of the dialog.
6. Then click the **Primary Units tab** and **set the units Precision to 0**, that is no units after decimal point and **Decimal separator to Period**. Click the sub -dialogs **OK** button.
7. The **Dimension Styles Manager** dialog reappears showing dimensions, as they will appear in a drawing, in the Preview of my-style box. Click the New . . . button. The Create New Dimension Style dialog appears.

SETTING DIMENSION STYLE

8. Enter a suitable name in the **New style name field** – in this example, this is **My-style**. Click the **Continue button** and the **Dimension Style Manager** appears. This dialog now shows a preview of the **My-style dimensions**. Click the dialog's **Set Current button**, followed by another click on the **Close button**.





SETTING LAYERS (LAYER PROPERTIES MANAGER)

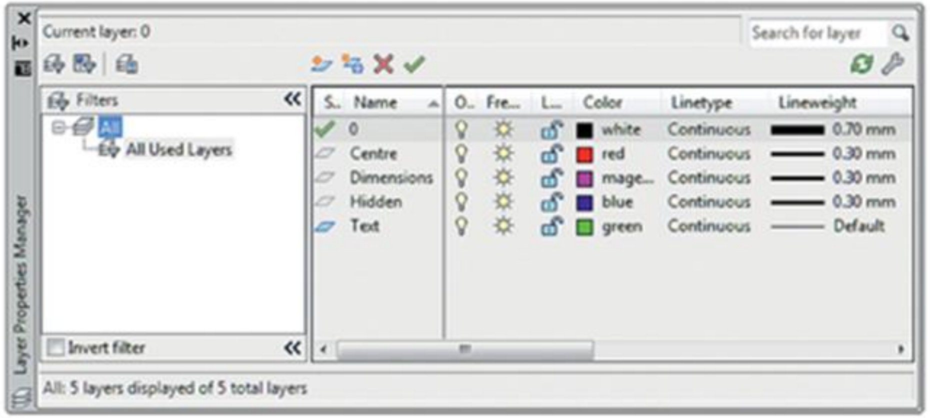
1. At the keyboard, **enter layer or la** and right-click. **The Layer Properties Manager** palette appears (Fig. 37).
2. Click the **New Layer** icon. **Layer1** appears in the layer list. Overwrite the name **Layer1** entering **Centre.**
3. Repeat step **2 four times** and make **four more layers** entitled **Construction, Dimensions, Hidden and Text.**
4. Click one of the squares under the **Color** column of the dialog. The **Select Color dialog** appears. Double-click on one of the **colours** in the **Index Color squares**. The selected **colour** appears against the layer name in which the square was selected. Repeat until all five new layers have a colour.

5. Click on the **linetype Continuous** against the layer name **Centre**. The Select **Linetype** dialog appears. Click its **Load . . . button** and from the **Load or Reload Linetypes** dialog double-click **CENTER2**. The dialog disappears and the name appears in the **Select Linetype** dialog. Click the **OK** button and the **linetype CENTER2** appears against the **layer Centre**.
6. Repeat with **layer Hidden**, load the **linetype HIDDEN2** and make **the linetype against this layer HIDDEN2**.

7. Click on the any of the **lineweights** in the **Layer Properties Manager**. This brings up the **Lineweight dialog**. Select the **lineweight 0.7** for **Layer 0**. Set at 0.3 for all other the layers, except Text. Then click the Close button of the Layer Properties Manager.

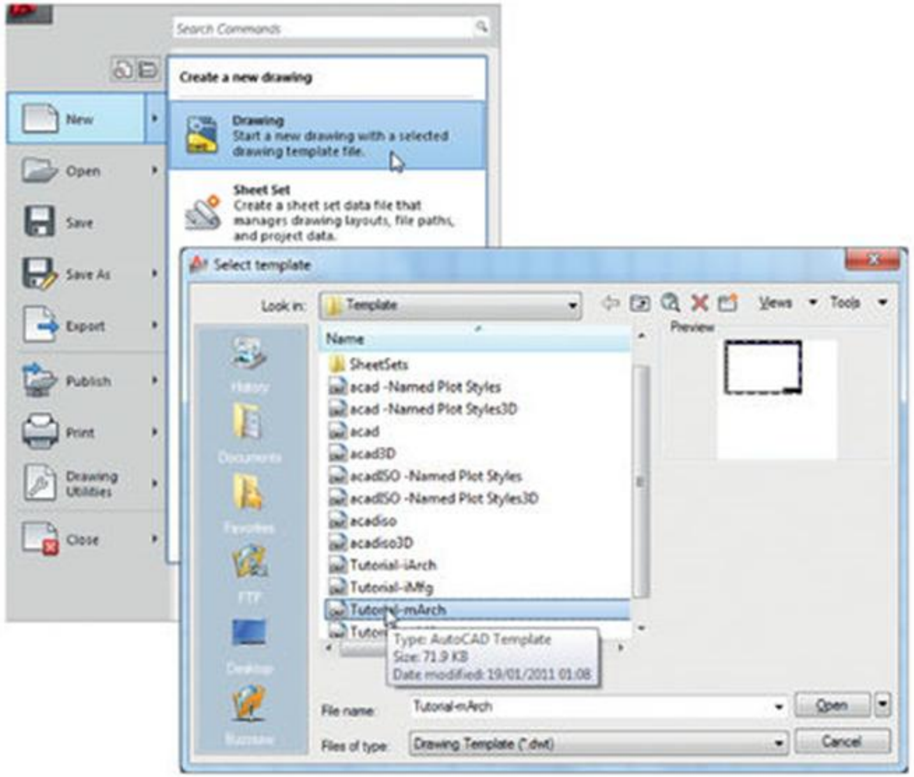
SETTING LAYERS (LAYER PROPERTIES MANAGER)

The **Layer Properties Manager dialog box** can also be accessed at the **HOME/LAYERS** on the **Ribbon Menu.**



DRAWING TEMPLATE

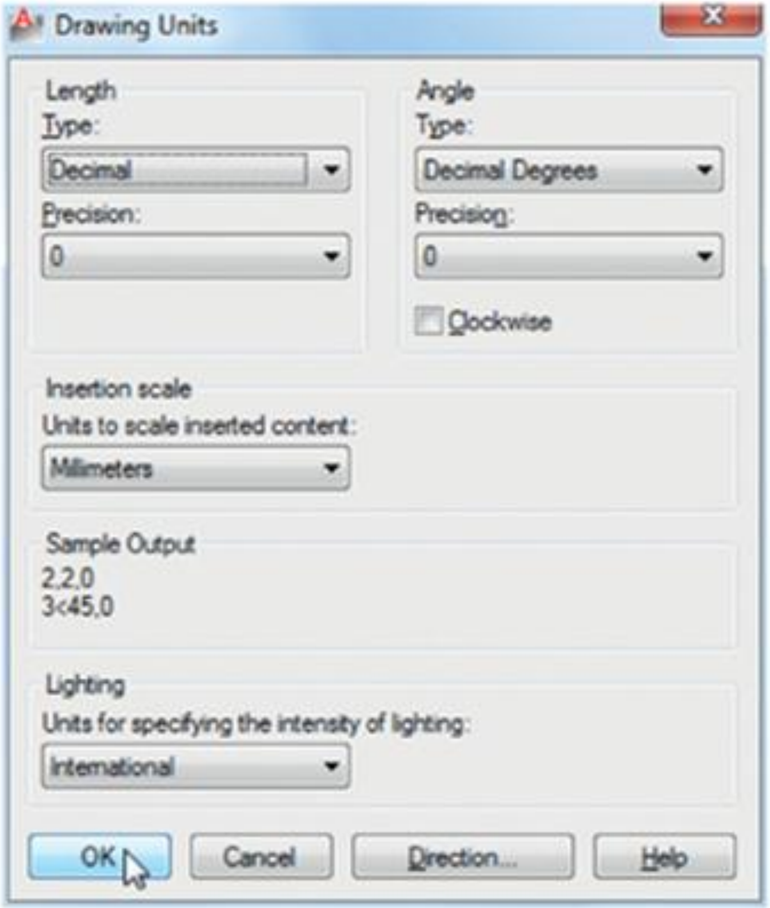
Drawing templates are files with an extension .dwt. Templates are files that have been saved with predetermined settings – such as Grid spacing, Snap spacing, Text style, Dimension style etc. Templates can be opened from the **Select template dialog**, called by clicking the **New . . . icon in the Quick Access toolbar**.



HOW TO CREATE CUSTOMIZED DRAWING TEMPLATES

There are **two** default drawing template formats in AutoCAD, Namely: **acad and acadiso. Acad** unit formats include **inches and feet** while **acadiso** units formats include **millimeters and meters.**

To check the current units in use, type UNITS in command line and press ENTER, the Drawing unit dialog box will appear showing the current units been used.



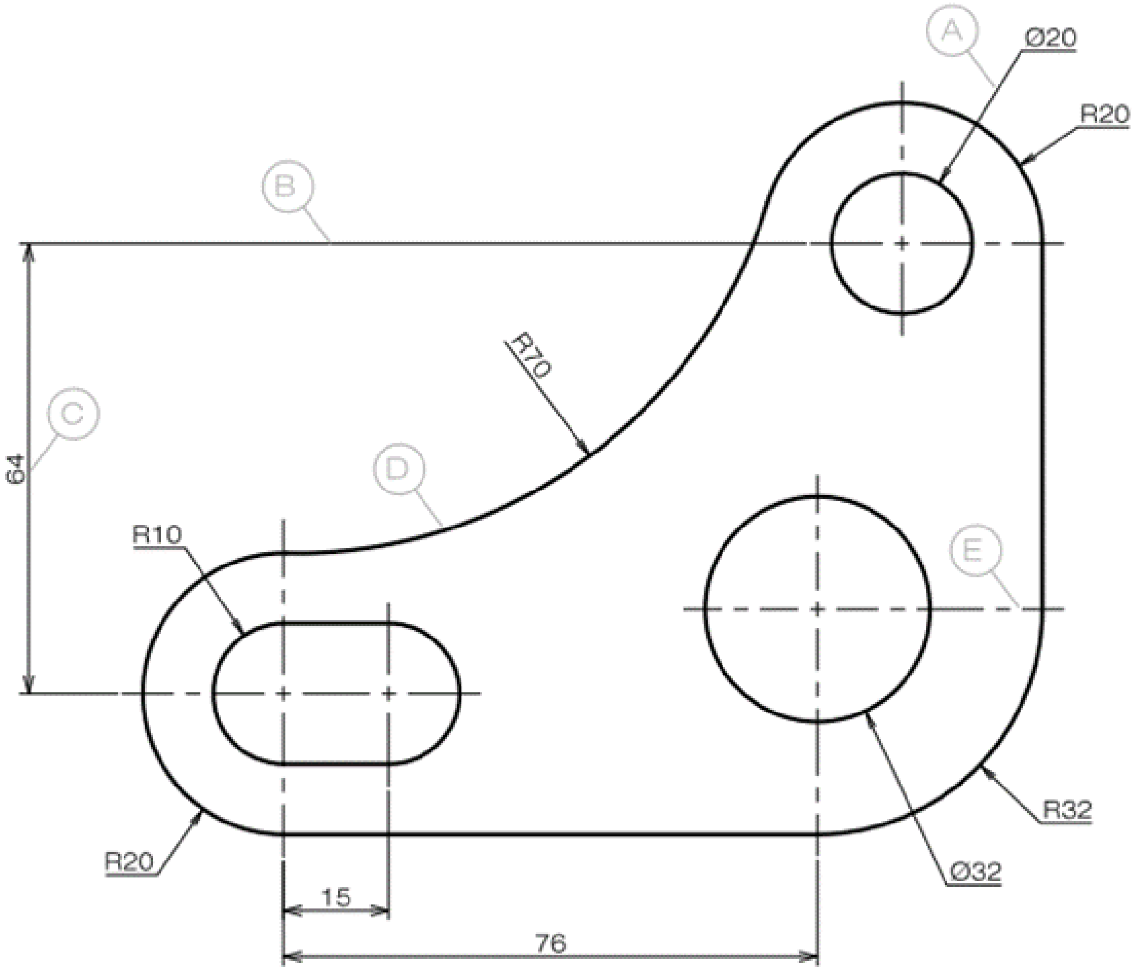
TO CREATE A CUSTOMIZED DRAWING TEMPLATE

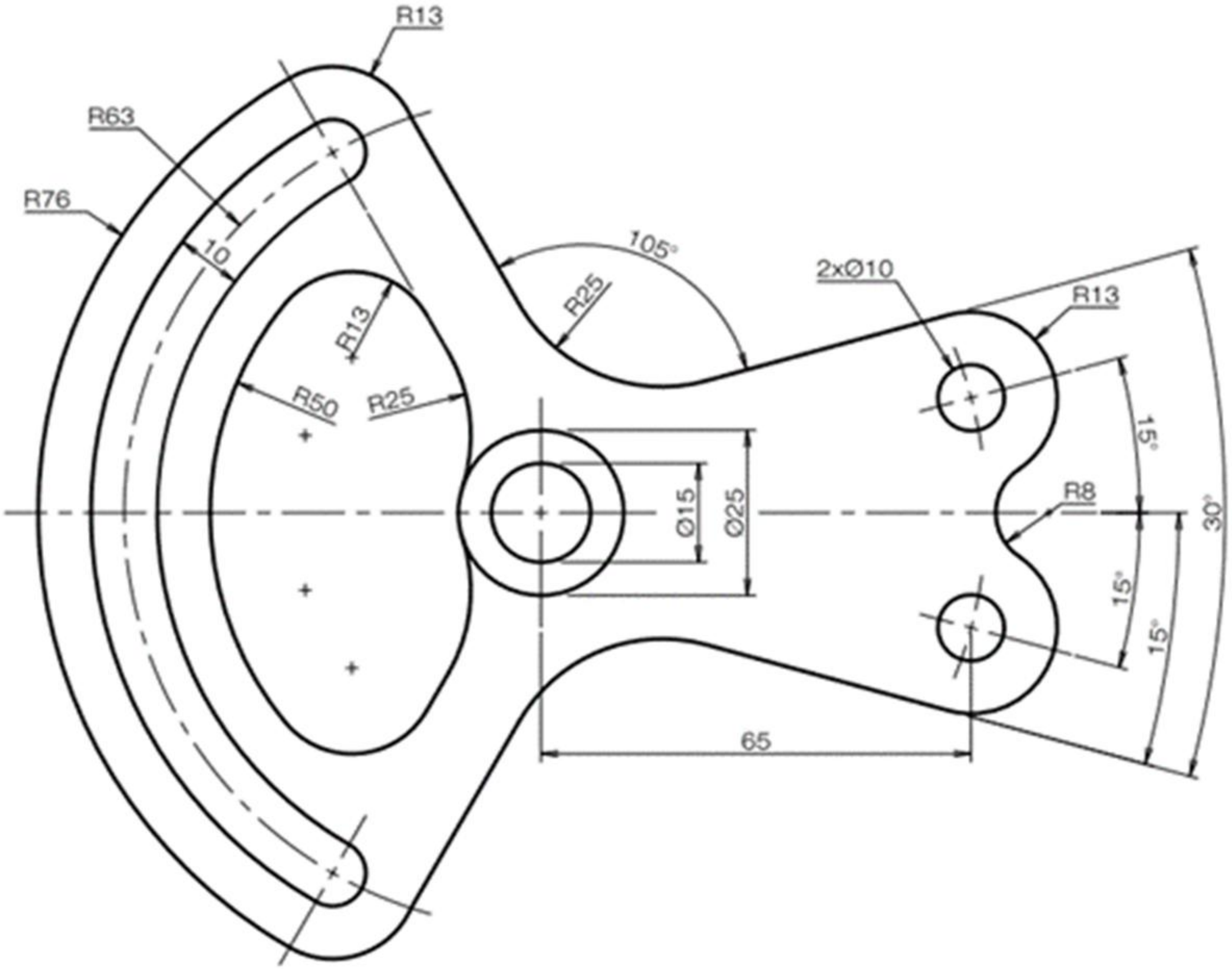
1. Open AutoCAD, Right Click on the **Start Tab** on the **Start Page** and click on **NEW**. The **Select Template dialog box** will pop up. (fig. 13).
2. Since we will be working in metric units, Select **acadiso.dwt** from the list of Templates.
3. A new drawing workspace will open. Type **Z for zoom** and press **ENTER**. Select **All** by typing **A** and press **ENTER**.
4. Set your layers (Line types).
5. Set your Text Styles.
6. Set your Dimension, leader and Table styles if necessary.
7. Click on the **Layout 1 tab** and then Right Click on it. Click on **Page Setup Manager**. (Fig. 14)
8. Click on **Modify** from the **Page Setup Manager**, Select **Plotter or Printer type, Paper size and Paper orientation**.
9. Click **OK** when done and **Close** on the next dialog box.
10. Back to **layout 1**, Create your boundary using the **Rectangle Tool** and your **Title box** using the suitable drawing tool.
11. Set **Viewport** by typing **MV or MVIEW** in the command palette and press **ENTER**.
12. To Activate a viewport, **double click inside the viewport**. To deactivate it, double click **outside the viewport**.
13. Go to **File** and Save as template by Clicking on **SAVE AS**. At the SAVE AS dialog box, Select **dwt template** as your file type.
14. Add the name of the Template and save the file to a suitable location.

HOW TO DRAW ON AUTOCAD

1. Launch AutoCAD software
2. Click on the Action Menu, select New to open the Template dialog box.
3. Select the right template needed for the drawing especially based on the units of the dimensions.
4. In the model view, type Z in the command line and select all.
5. Create your layers using the layer manager.
6. Define your default layer.
7. Combine both draw and modify tools to create your drawing.
8. Create a dimension style and use that to place dimensions on your drawing.

EXERCISE

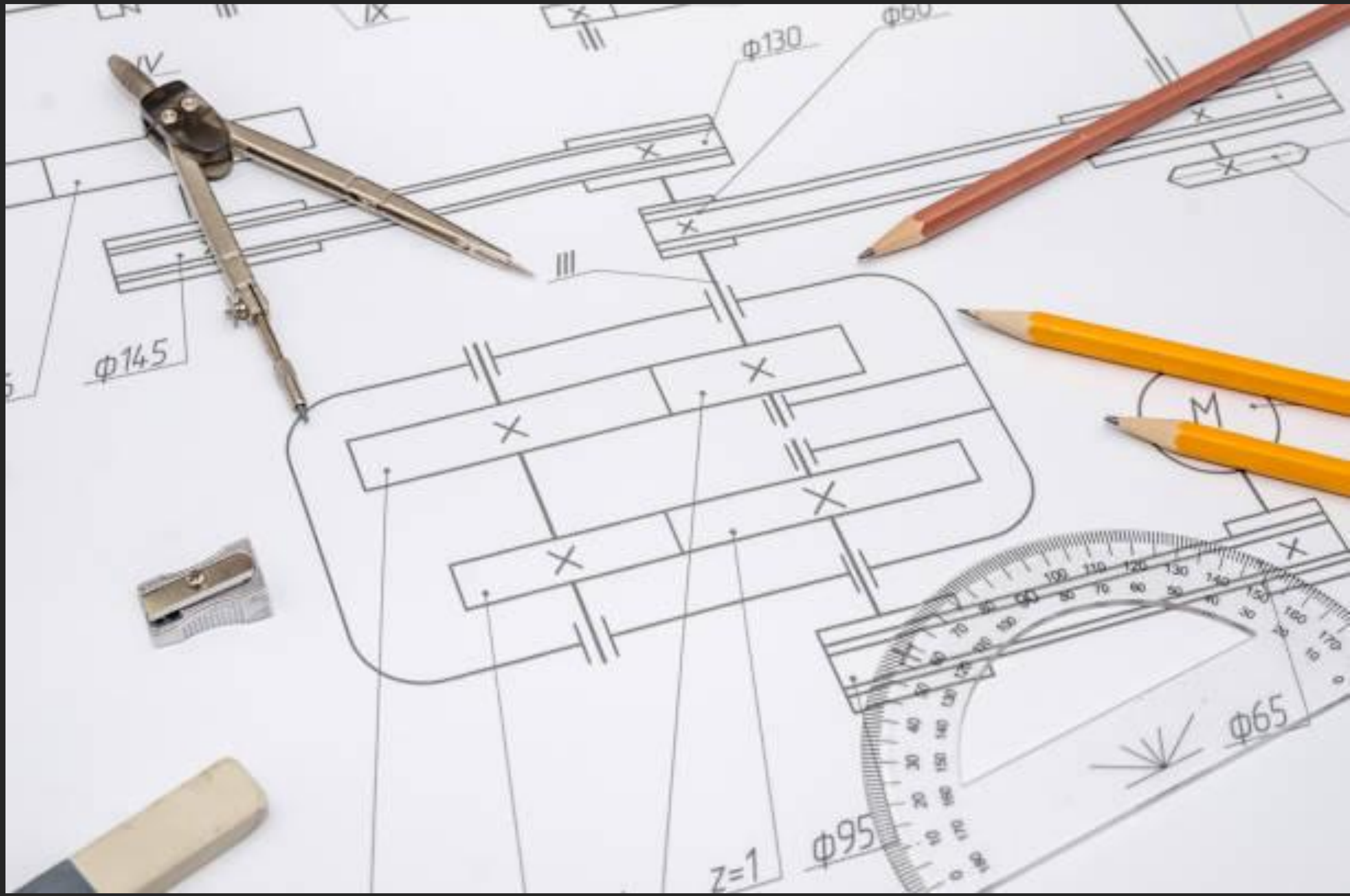

A
Ø20
R20
B
R70
C
64
D
R10
E
R32
Ø32
R20
15
76


R13
R63
R76
10
105°
R25
2xØ10
R13
R13
R50
R25
Ø15
Ø25
15°
R8
30°
15°
15°
65

CONVENTIONAL DRAWINGS

We will cover these skills:

- Features and conventional representation.
- Pictures of machine parts with features.
- Exercises.



CONVENTIONAL REPRESENTATION OF COMMON FEATURES

There are many common engineering features which it is difficult and tedious to draw in full. In order to save draughting time and space on drawings, these features are represented in a simple conventional forms as show.

- **External screw threads**

The crests of the male thread of a stud are defined by a continuous thick line, and the roots of threads by a parallel continuous thin line. The distance between these parallel lines should be approximately equal to the depth of the thread. i.e. approximately one eighth of the major diameter of the thread.

The limit of the useful length of the thread — 'full thread' — is shown by a continuous thick line.

- **Internal screw threads**

The tapped hole initially is drilled, which is indicated by the thick outlines. When the hole is tapped, the roots of the threads are defined by a parallel continuous thin line.

CONVENTIONAL REPRESENTATION OF COMMON FEATURES

In an end view, the thread roots inside the material are represented by an outer thin broken circle.

In a sectional view, the hatching lines are drawn across the thin lines (see pages 38 and 77).

- **A screw-thread assembly**

The male thread of an inserted stud takes precedence over the female thread of the hole.

The hatching lines are not drawn across the thick lines.

In an end view, the male part which is nearest to the observer is represented.

- **Interrupted views**

To save space, it is permissible to show only those parts of a long component which are sufficient for its definition. All break lines are thin and continuous.

CONVENTIONAL REPRESENTATION OF COMMON FEATURES

- **Cylindrical compression springs**

Coils are drawn only at each end of the spring and are connected by parallel thin chain lines indicating the repetition of coils.

- **Splined and serrated shafts**

Only a few splines or serrations need be shown, with the root circle represented by an inner thin circle.

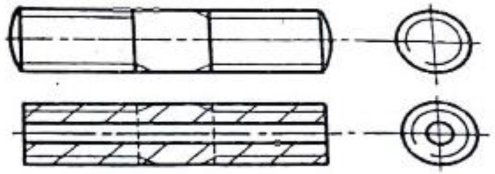
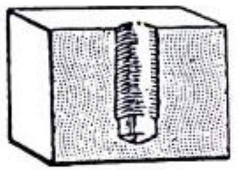
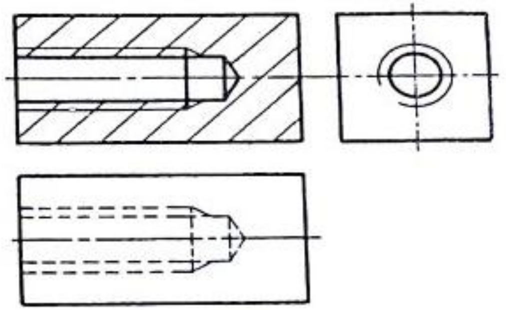
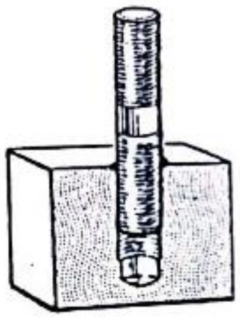
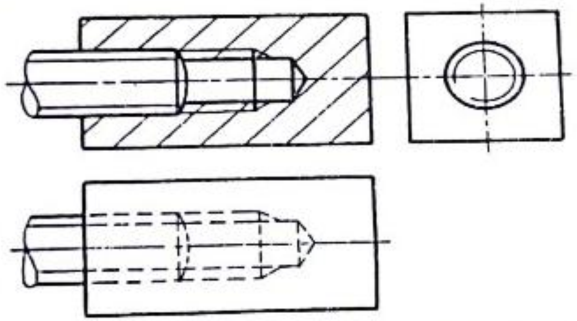
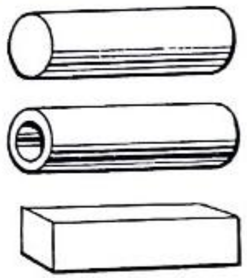
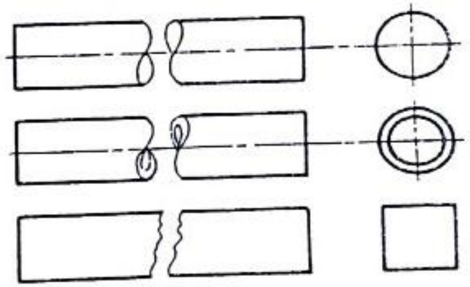
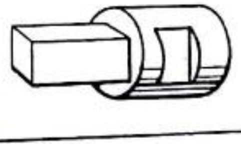
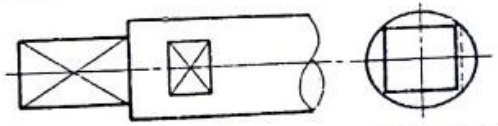
- **Diamond and straight knurling**

Knurling provides a rough surface to facilitate the operation of the com ponent by hand. (A spring-bow compass usually incorporates both types of knurling.)

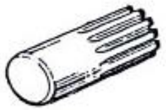
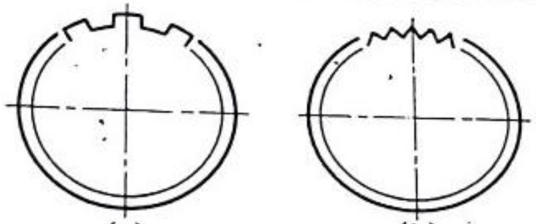
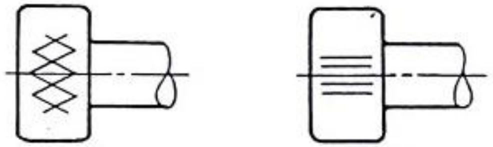
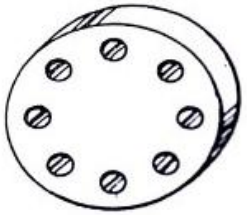
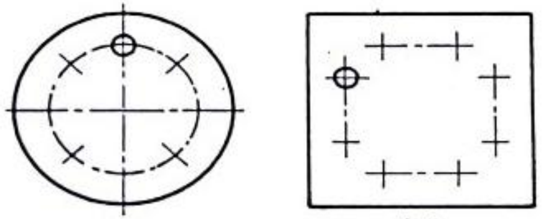
- **Holes on circular and linear pitch**

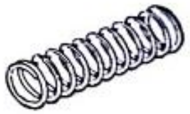
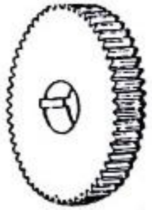
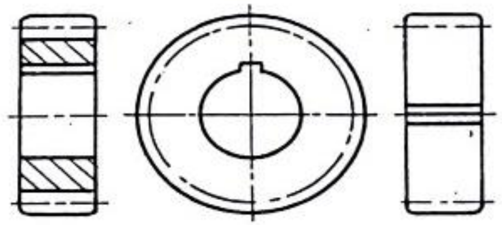
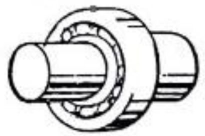
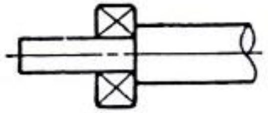
When a large number of repeated holes are required, it is sufficient to draw only one hole and to indicate the position of the remainder.

CONVENTIONAL REPRESENTATION OF COMMON FEATURES

Title	Subject	Convention
External screw threads (detail)		
Internal screw threads (detail)		
Screw threads (assembly)		
Interrupted views of: round shaft, hollow shaft, rectangular block		
Squared end or a local flat on a shaft		

CONVENTIONAL REPRESENTATION OF COMMON FEATURES

Title	Subject	Convention
Shafts: (a) splined, (b) serrated		 (a) (b)
Knurling: (a) diamond, (b) straight		 (a) (b)
Holes on (a) circular pitch (b) linear pitch		 (a) (b)

Cylindrical compression spring		 Convention / Diagrammatic representation
Spur gear		
Bearing		

CONVENTIONAL REPRESENTATION OF COMMON FEATURES

- **Cylindrical compression springs**

Coils are drawn only at each end of the spring and are connected by parallel thin chain lines indicating the repetition of coils.

- **Spur gears**

The front view of a spur gear consists of the outside thick circle and the pitch ‘center-line’ circle.

In the sectional end view the gear teeth are not sectioned, but the center line and the root of the teeth are shown.

In the remaining end view the center lines are shown and the direction of teeth is indicated by thick parallel lines.

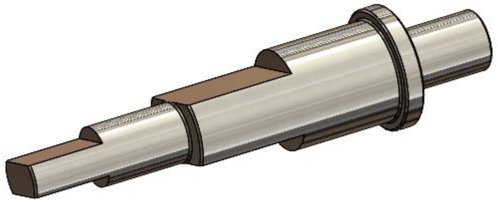
- **Bearings**

Sometimes it is necessary to draw sectional views of a number of ball and roller bearings on one drawing. The complicated sectional views can be replaced by a conventional representation consisting of the bearing outline and thin-line diagonals. This conventional representation does not imply any particular type of bearing.

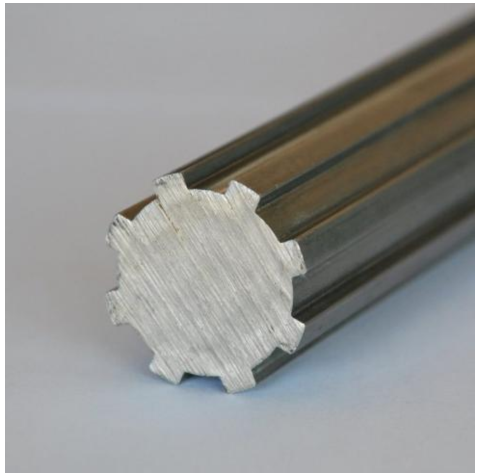
CONVENTIONAL REPRESENTATION OF COMMON FEATURES

Thread (Internal and external)

Shaft



Splined shaft



Serrated shaft

Knurl

Holes on circular pitch

Bearing

EXERCISES

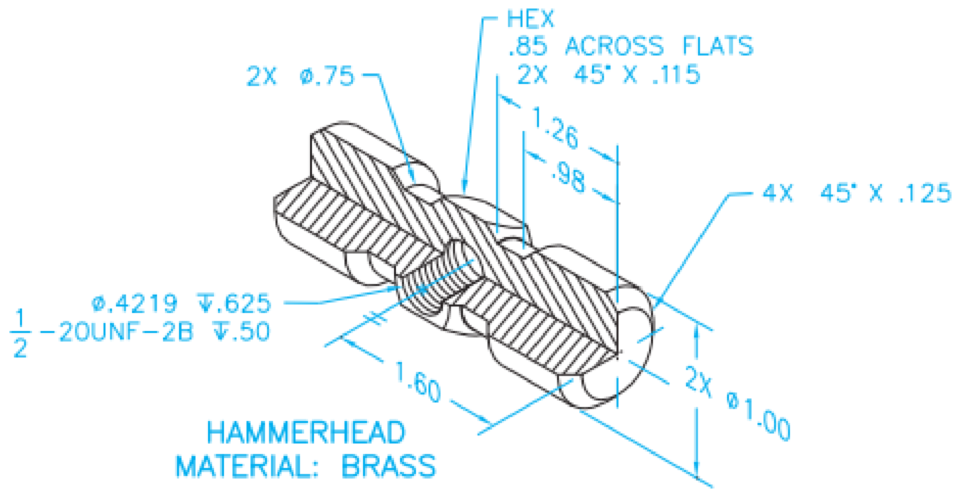
Produce the Front, left and Right End View of these images.

EXERCISES

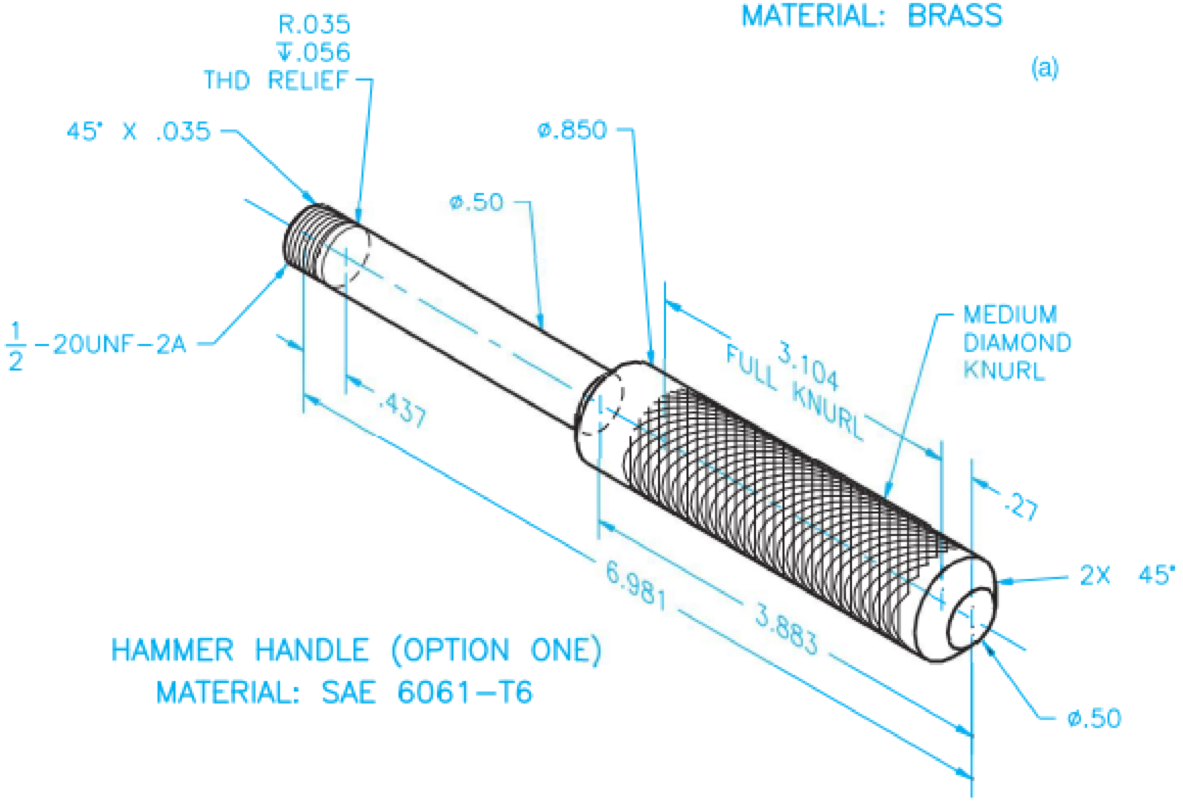
PROBLEM 15.2 **Working drawing (in.)**

Assembly Name: Hammer

SPECIFIC INSTRUCTIONS: Prepare a detail drawing for the hammer head and two optional hammer handles on one sheet. Make the assembly drawing and parts list on another sheet.



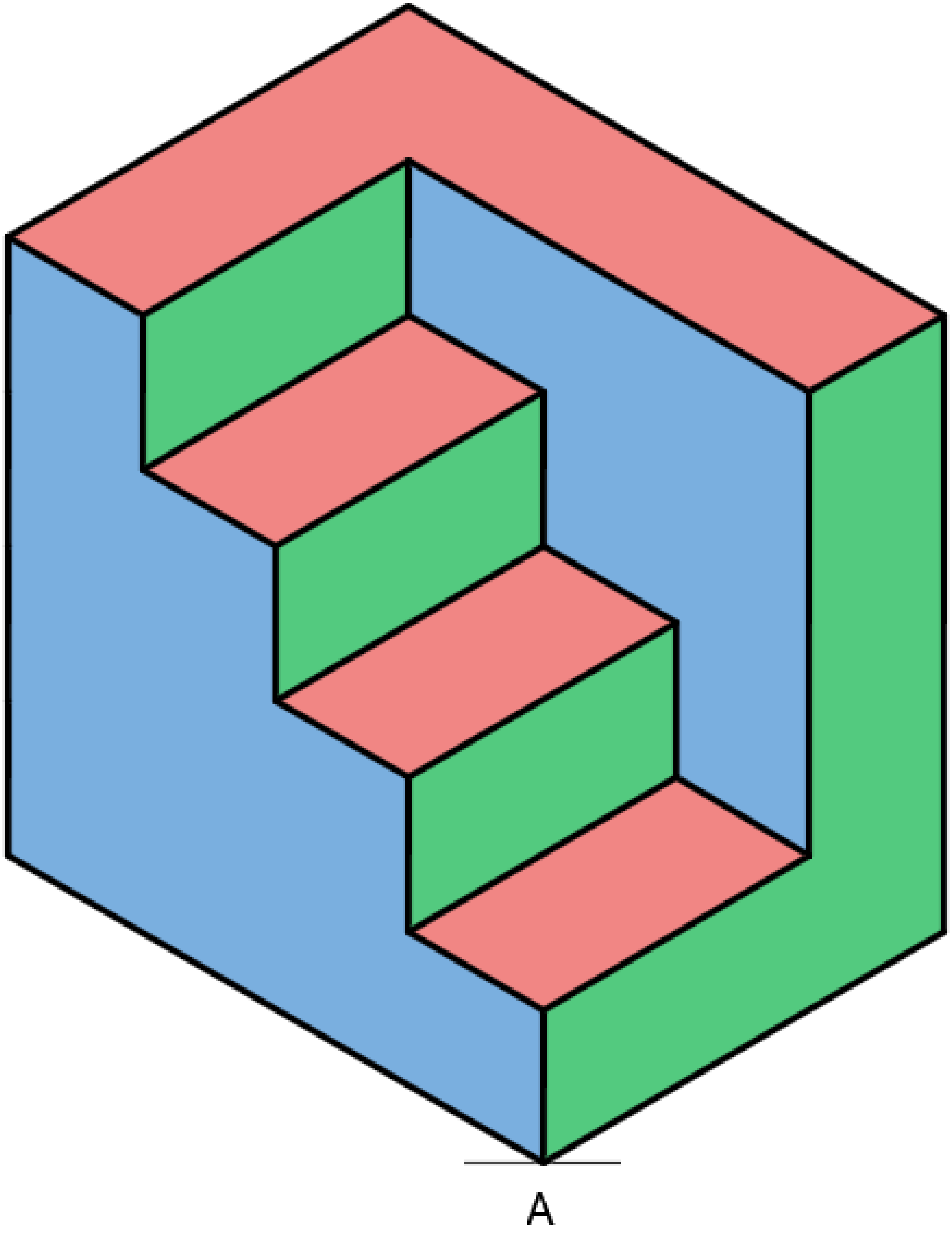
(a)



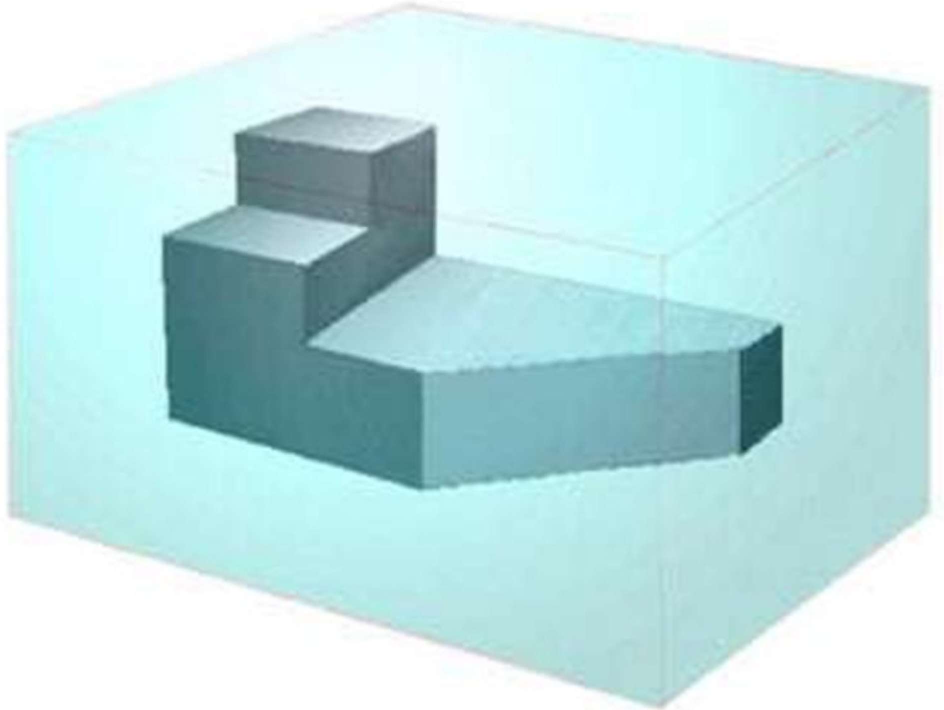
ORTHOGRAPHIC PROJECTION (REV)

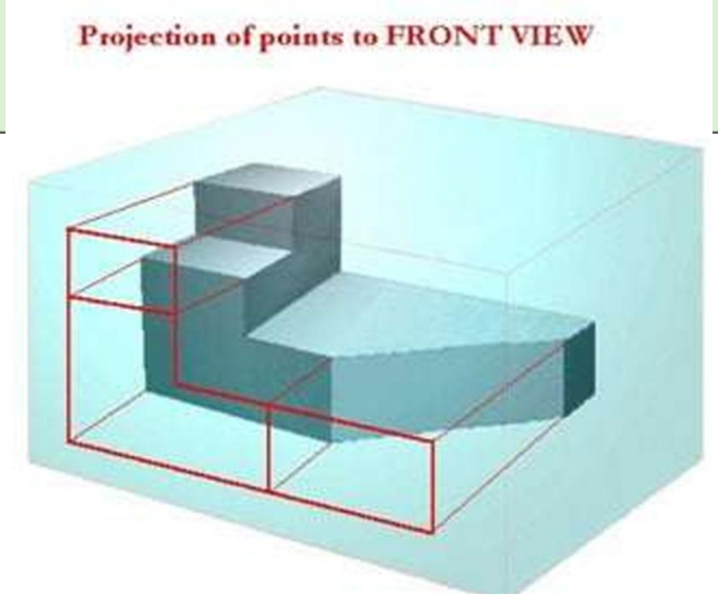
We will cover these skills:

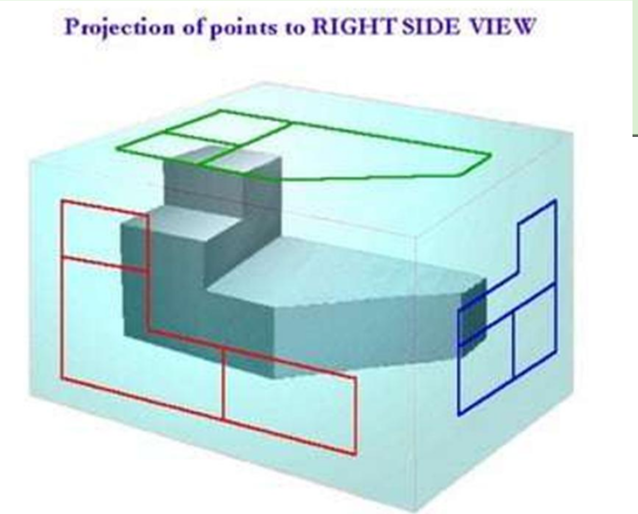
- Producing Orthographic views.
- First Angle Projection
- Third Angle Projection

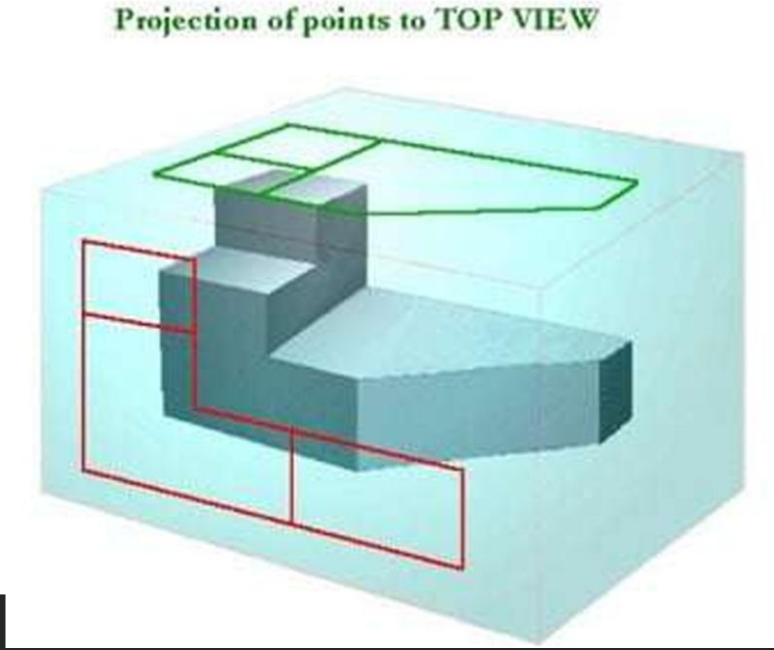


ORTHOGRAPHIC PROJECTION

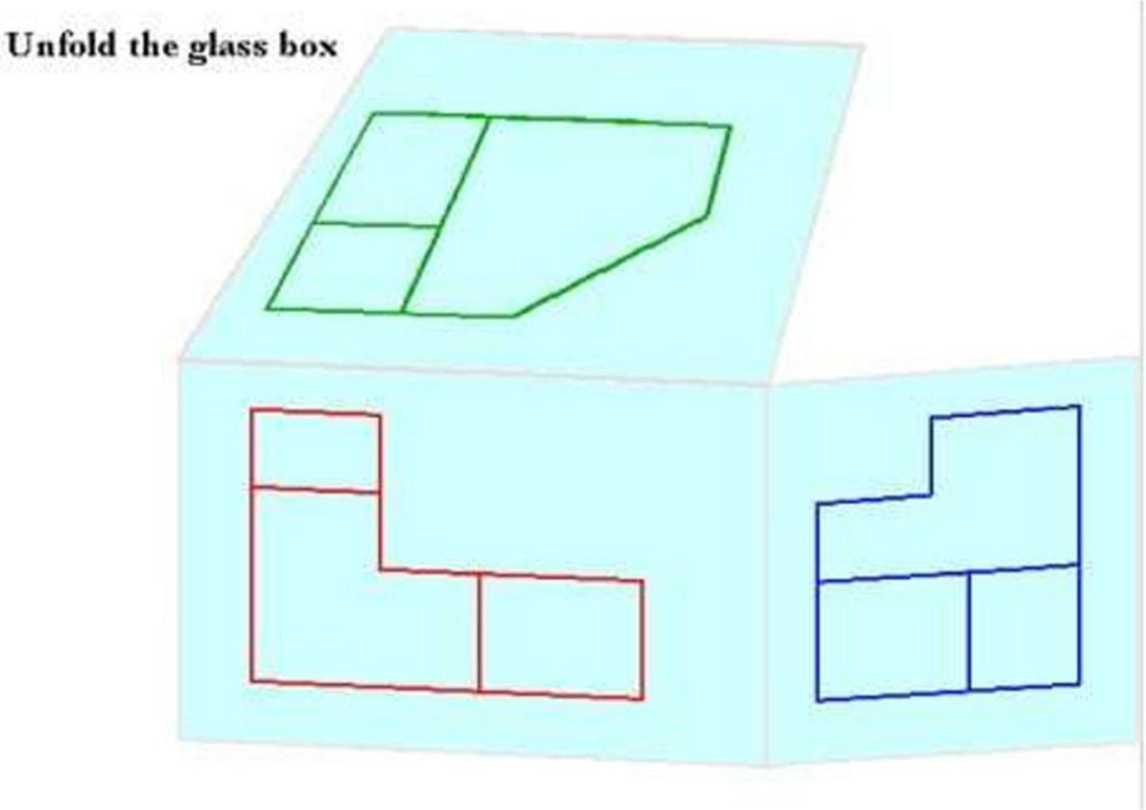


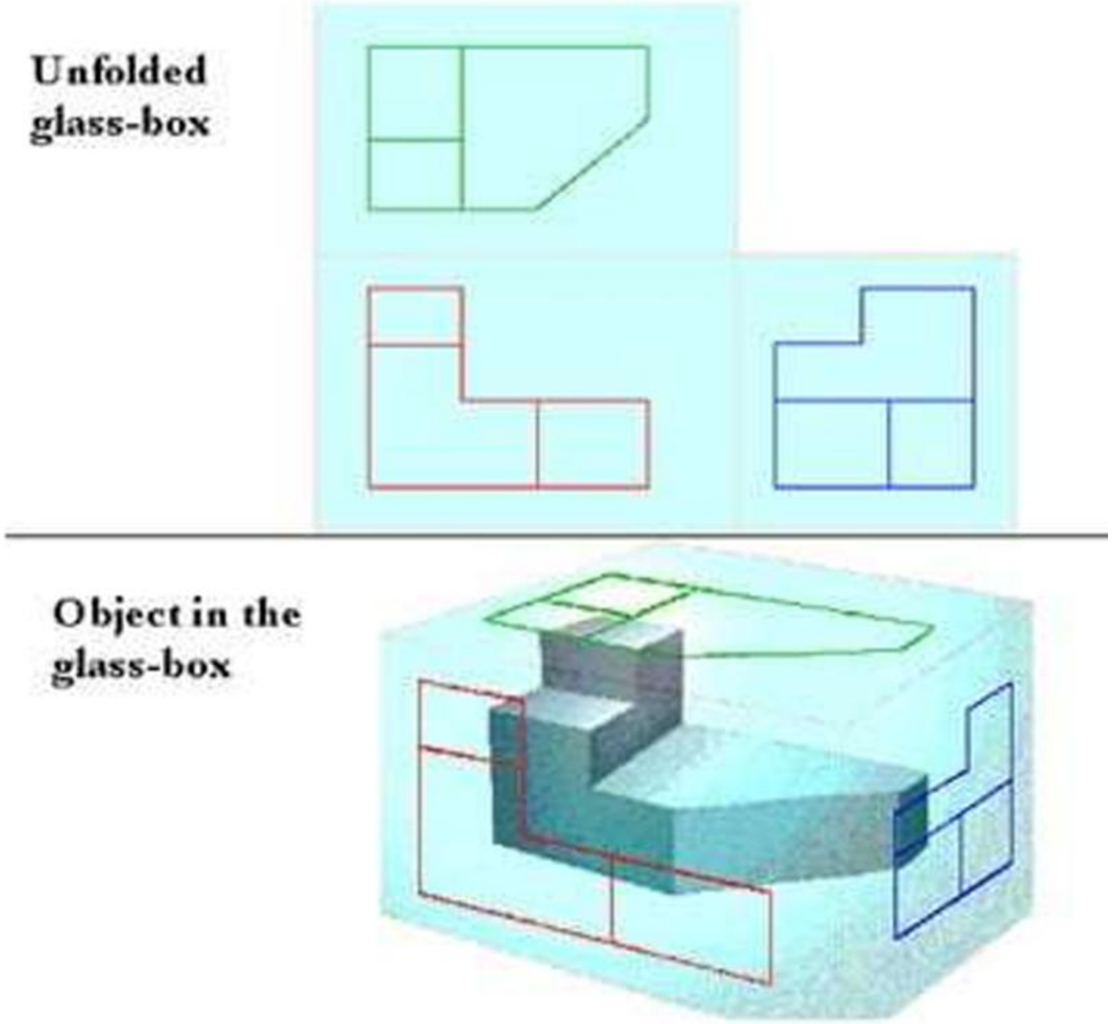




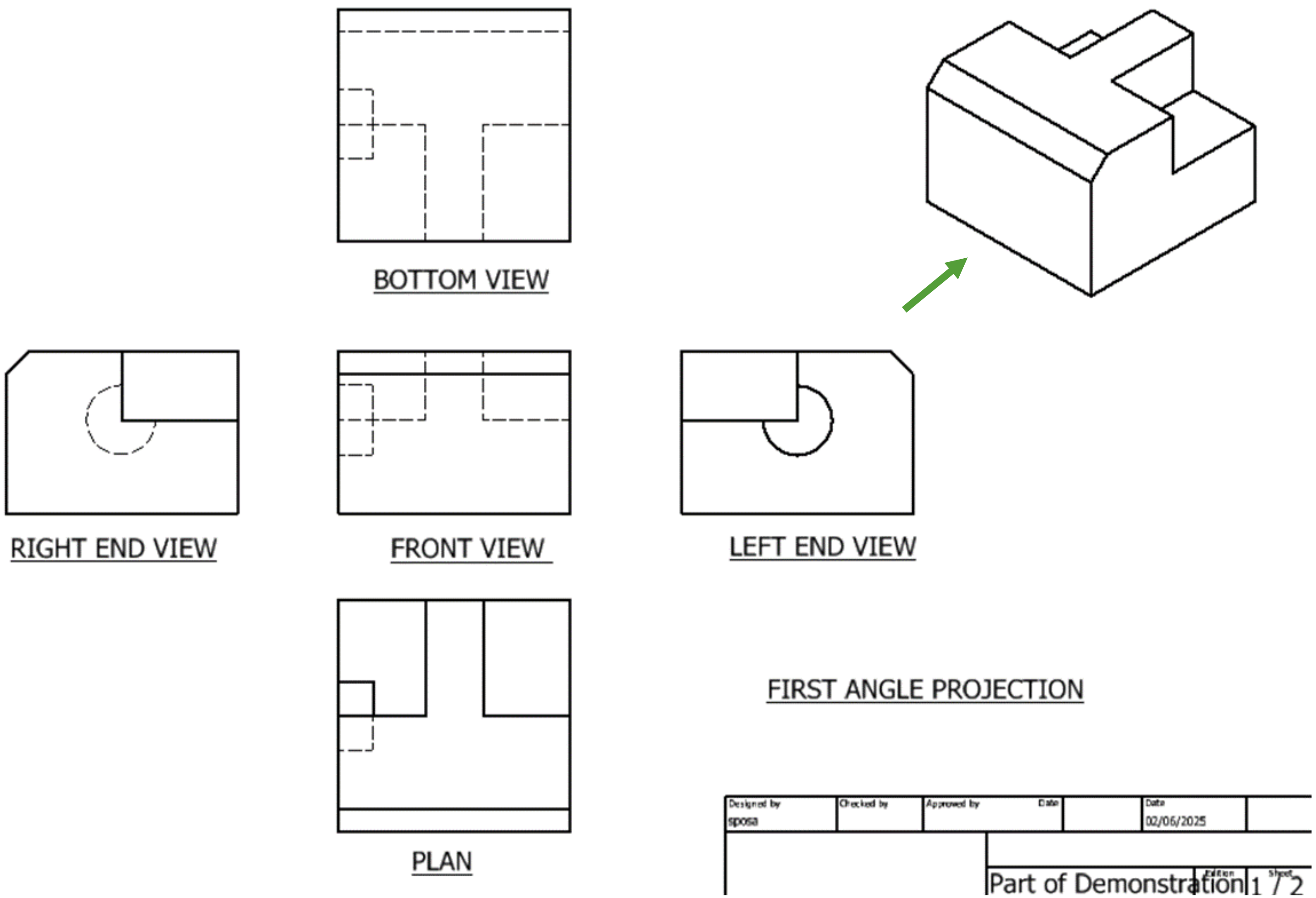


ORTHOGRAPHIC PROJECTION

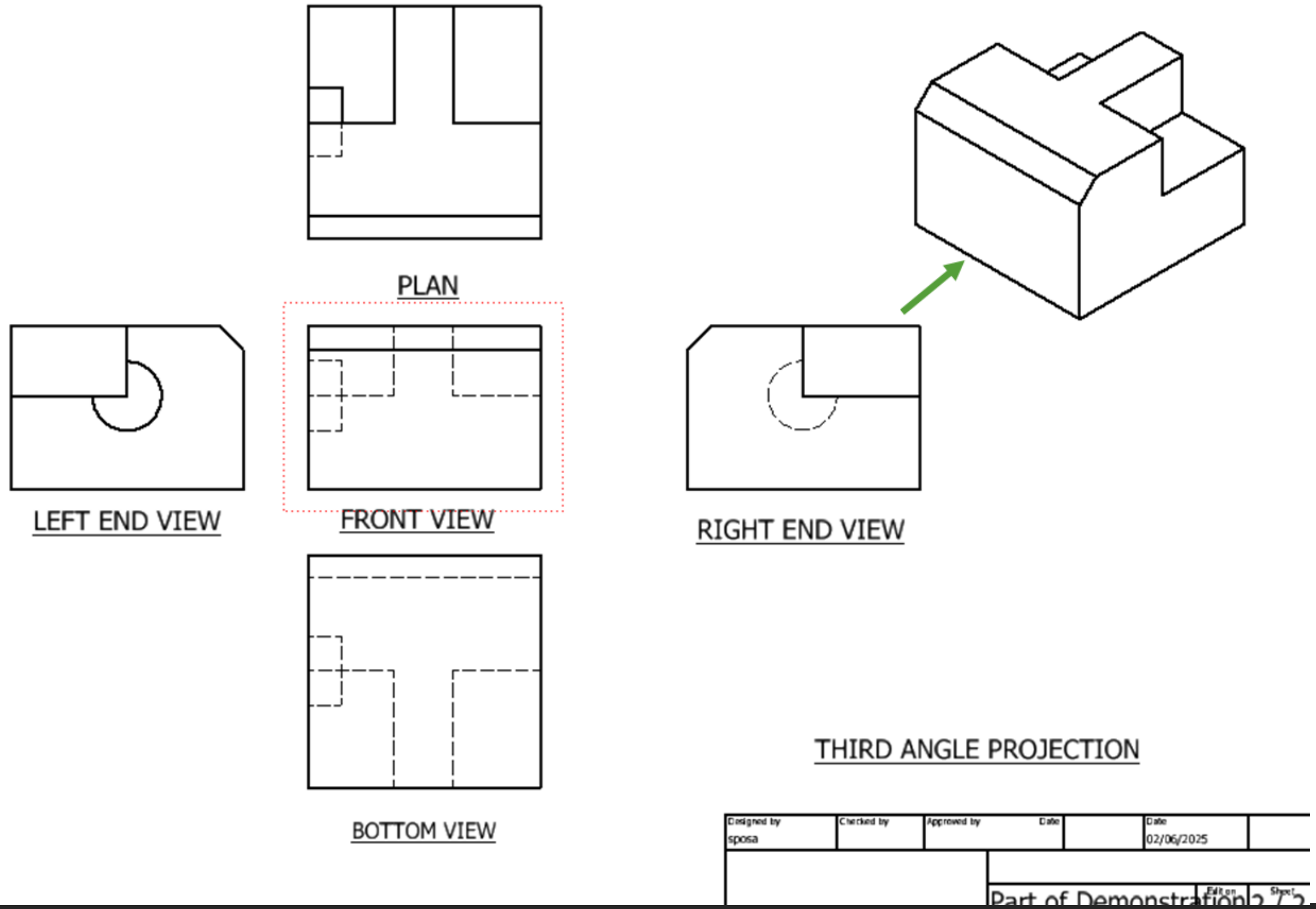




ORTHOGRAPHIC PROJECTION



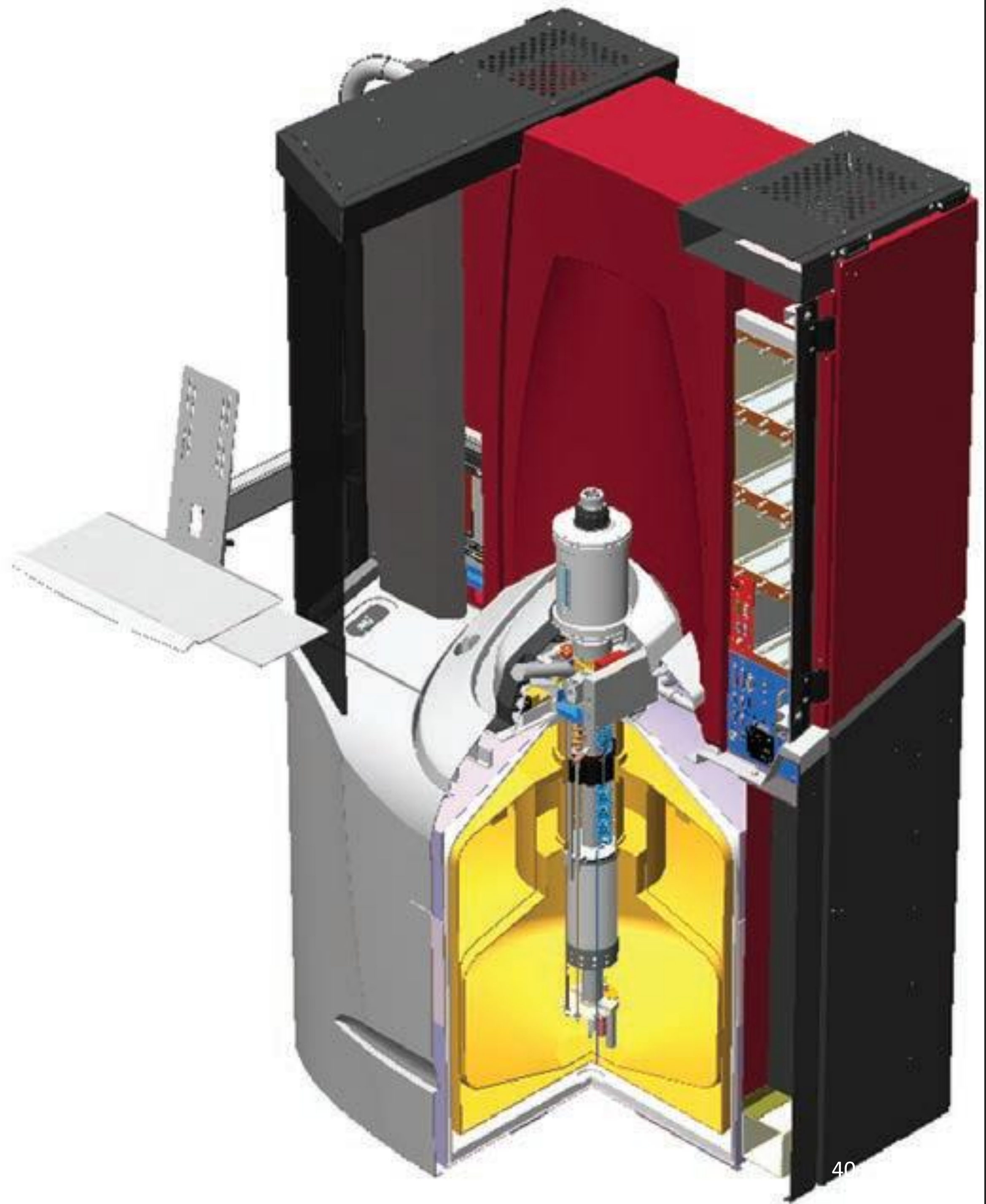
ORTHOGRAPHIC PROJECTION



SECTIONAL VIEWS

We will cover these skills:

- Introduction.
- Full Sections.
- Placement of Section view.
- Labeling Cutting planes.
- Half Sections.
- Broken out sections.
- Revolved sections.
- Removed sections.
- Offset sections.
- Ribs in section.
- Aligned sections.
- Partial sections.
- Parts not sectioned.
- Exercises.



SECTIONAL VIEWS

Technical drawings often represent a single part with a complex interior structure, or many different parts in a mechanical assembly, building, bridge, toy, or other product. When you are creating a drawing, if the interior structure cannot be shown clearly by using hidden lines, you should use a section view to reveal the internal features of the part.

To visualize a section view, think of slicing through the object as if you were cutting through an apple or melon.

This familiar cutaway view—looking onto the cut portion of the object—is called **a section view, or sometimes a cross section.**

In order to show the internal features without excessive use of hidden- detail lines, the object is imagined to be cut along a plane called **a cutting plane**. The cut portion nearer to the observer is removed and the remaining part is shown as a sectional view.

SECTIONAL VIEWS

PURPOSE OF SECTIONS

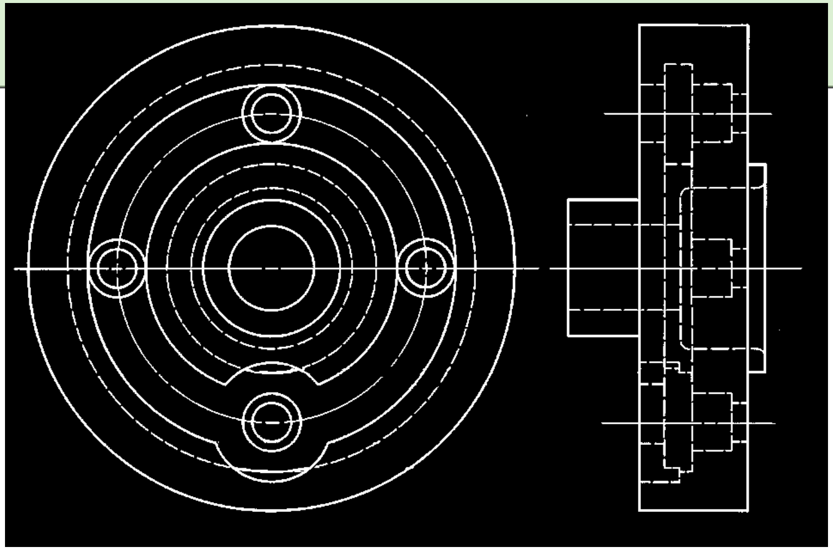
- to clarify details of the object,
- to illustrate internal features clearly,
- to reduce the number of hidden-detail lines,
- to facilitate the dimensioning of internal features,
- to show the shape of the cross-section,
- to show clearly the relative positions of parts forming an assembly.

Full Section of a Melon

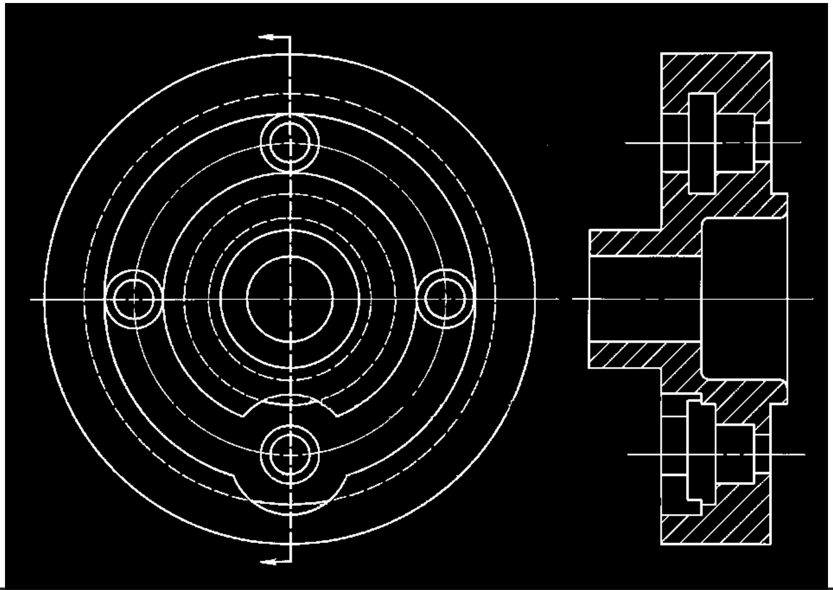
SECTIONAL VIEWS

Full Sections

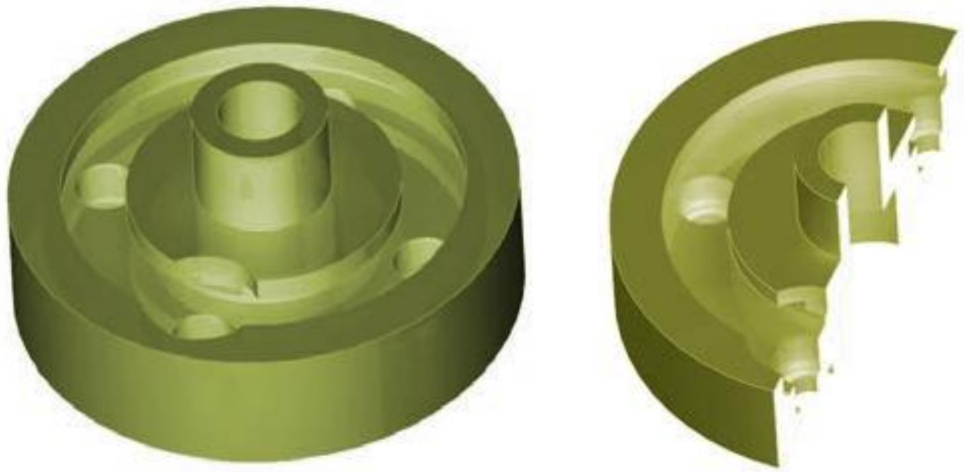
When the part is cut fully in half, the resulting view is called a full section, as shown in Figure 1. Figure 2 shows a technical drawing of the part from Figure 1 that does not use a section view.



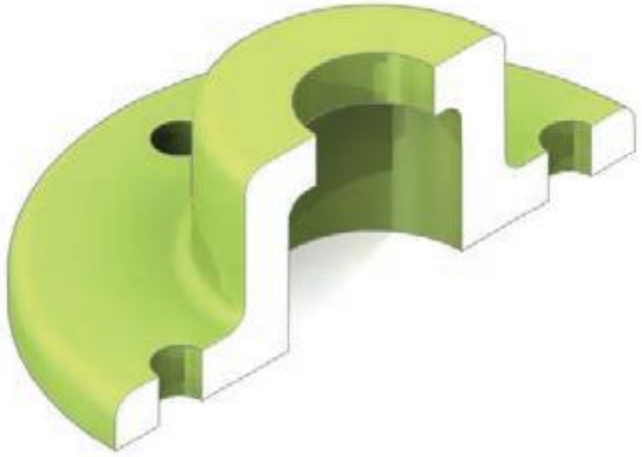
Front and Right-side view with hidden lines.

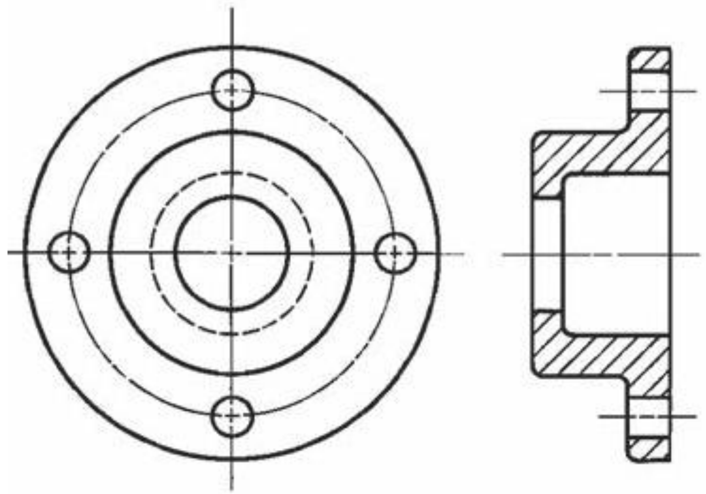


Front and Right-side view sectioned.



SECTIONAL VIEWS





PLACEMENT OF SECTION VIEWS

Section views can replace the normal top, front, side, or other standard orthographic views in the standard view arrangement. Figure 1 shows an example. In this drawing, the front view of the object is shown in section. Only two views are necessary. The front view is shown as a section view, and the cutting plane line is shown in the right-side view.

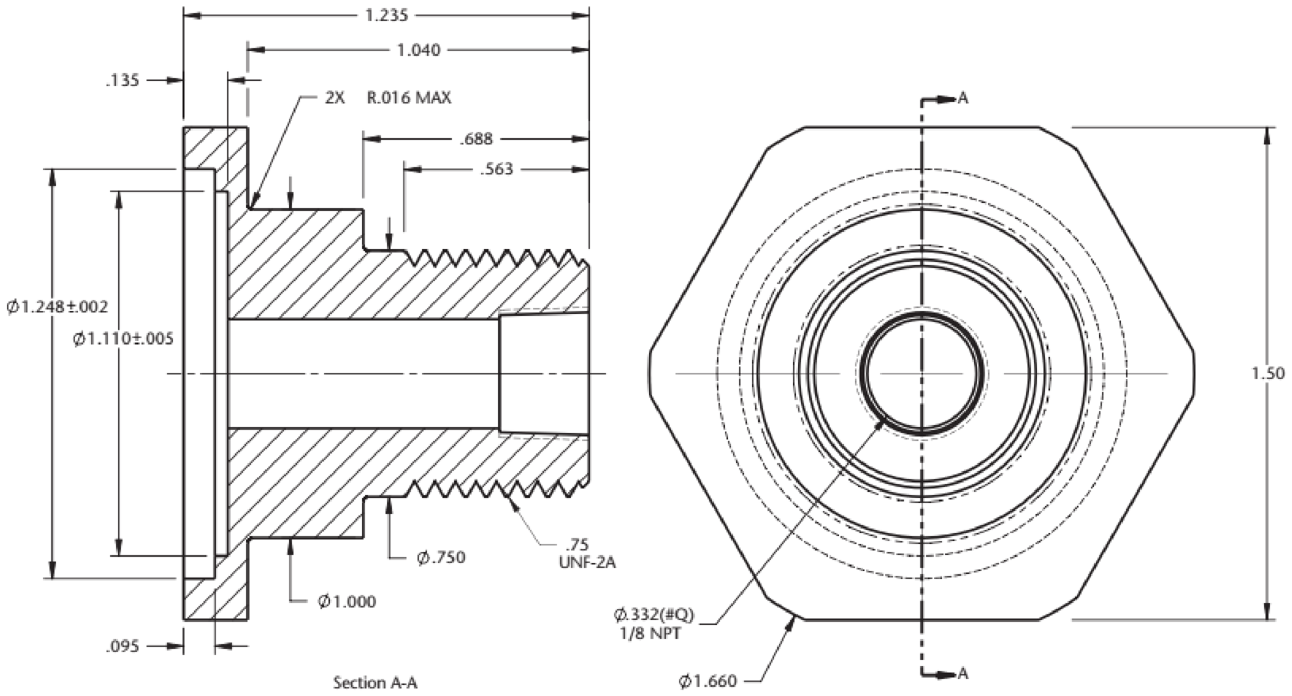
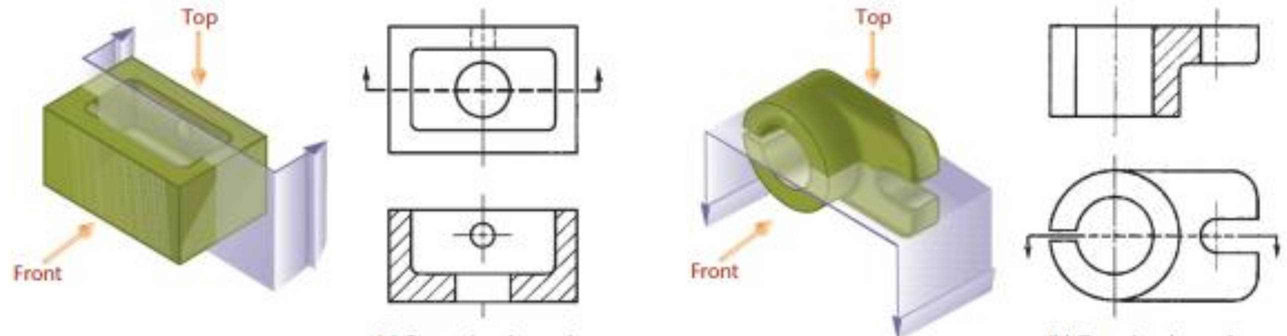


Figure. 1

SECTIONAL VIEWS

In Figure 2a, the object is cut through with a plane parallel to the front view. The front half of the object is imagined removed. The resulting full section may be referred to as the "front view in section" because it occupies the front view position. In Figure 2b, the cutting plane is a horizontal plane (which would appear as a line in the front view). The upper half of the object is imagined removed. The resulting full section is shown in place of the top view.



(a) Front view in section

(b) Top view in section

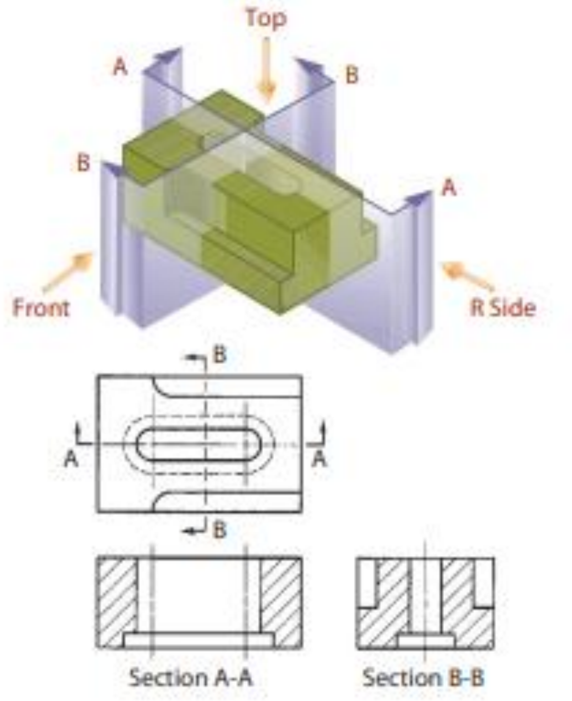
When adding a section view to your drawing, keep in mind that your purpose is to document and convey information about your design and show the information in the way that best achieves this objective.

SECTIONAL VIEWS

LABELING CUTTING PLANES

In Figure 3, two cutting planes are shown, one a plane parallel to the front view and the other a plane parallel to the side view, both of which appear edgewise in the top view. Each section is completely independent of the other and drawn as if the other were not present.

For section A–A, the front half of the object is imagined removed. The back half is then viewed in the direction of the arrows for a front view, and the resulting section is a front view in section.



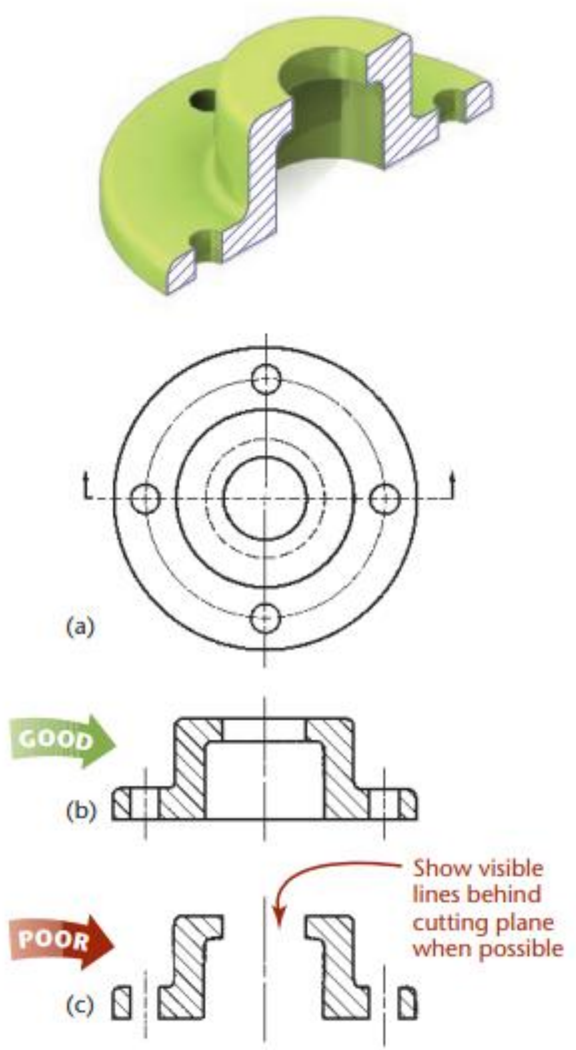
For section B–B, the right half of the object is imagined removed. The left half is then viewed in the direction of the arrows for a right-side view, and the resulting section is a right-side view in section. The cutting-plane lines are preferably drawn through an exterior view (in this case the top view, as shown) instead of a section view.

SECTIONAL VIEWS

RULES FOR LINES IN SECTION VIEWS

When creating section views follow these general rules:

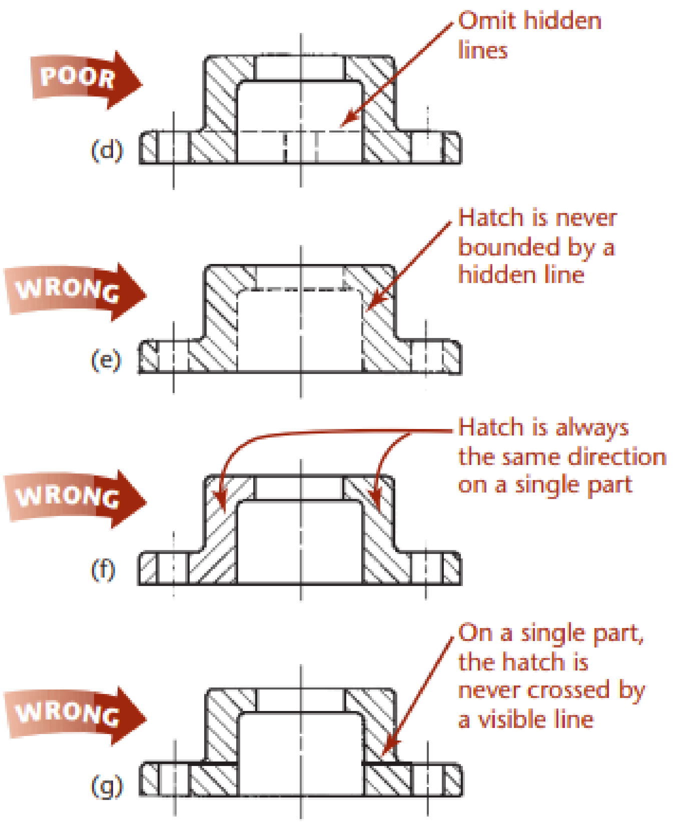
- Show edges and contours that are now visible behind the cutting plane; otherwise a section will appear to be made up of disconnected and unrelated parts. (Occasionally, visible lines behind the cutting plane may be omitted, particularly from those generated from 3D models).
- Omit hidden lines in section views. Section views are used to show interior confusion of hidden lines, so add them only if necessary to understand the part.
- Sometimes hidden lines are necessary for clarity and should be used in such cases, especially if their use will make it possible to omit a view (Figure 4d).
- A sectioned area is always completely bounded by a visible outline—never by a hidden line, because in every case the cut surfaces will be the closest surface in



SECTIONAL VIEWS

the section view and therefore their boundary lines will be visible (Figure 4e).

- In a section view of an object, the section lines in all hatched areas for that object must be parallel, not as shown in Figure 4f. The use of section lining in opposite directions is an indication of different parts, as when two or more parts are adjacent in an assembly drawing.
- A visible line can never cross a sectioned area in a view of a single part. This would be impossible on the full section of a single part because the section lines are all in the same plane. A line across it would indicate a change of plane (Figure 5g). In an assembly section, this would be possible. You will learn about assemblies later in the chapter.



SECTIONAL VIEWS

SECTION LINING OR HATCHING

The correct method of drawing section lines is shown in Figure 5a. When drawing by hand, use a sharp, medium grade pencil (H or 2H) to draw uniformly thin section lines, or hatching (a term meaning closely spaced parallel lines). There should be a marked contrast between the thin section lines and the thick visible outlines of the part.

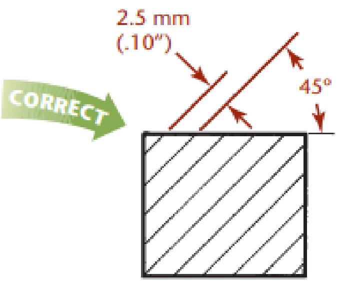
Draw section lines at 45° from horizontal unless they would be parallel or perpendicular to major edges of the part, in which case use a different angle.

Figure 5b shows an example of section lines drawn at a different angle to prevent them from being parallel or perpendicular to visible outlines.

Space the lines as evenly as possible by eye (for most drawings, about 2.5 mm (1/10”) apart). The spacing interval depends on the size of the drawing or of the sectioned area, with larger drawings having wider spacing. In a smaller draw ng the spacing interval may be as small as 1.5 mm (1/16”); in a large drawing, it may be 3 mm (1/8”) or more.

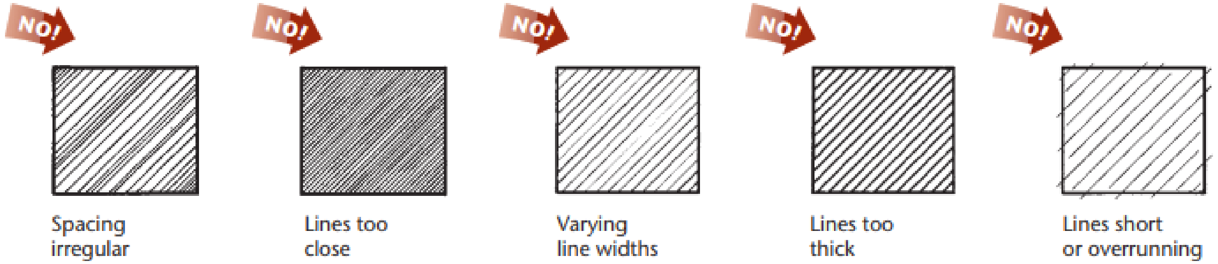
SECTIONAL VIEWS

As a rule, space the lines as generously as possible, yet close enough to clearly distinguish the sectioned areas. Keep extension lines and dimension values off sectioned areas. If there is no alternative, omit the section lines behind the dimensions (Figure 5c).



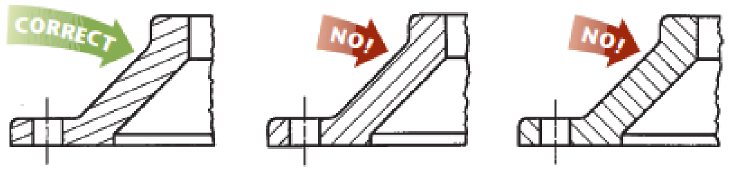
- Uniformly spaced by an interval of about 2.5 mm
- Not too close together
- Uniformly thin, not varying in thickness
- Distinctly thinner than visible lines
- Neither running beyond nor stopping short of visible outlines

(a) Correctly drawn section lines



(b) Direction of Section Lines

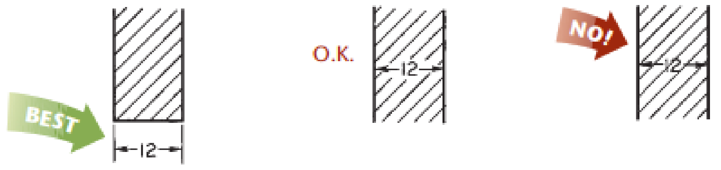
If section lines drawn at 45° from horizontal would be parallel or perpendicular (or nearly so) to a prominent visible outline, the angle should be changed to 30°, 60°, or some other angle.



Angle of section lines is adjusted | Lines should not be parallel to outline | Lines should not be perpendicular to outline

(c) Dimensions and Section Lines

Keep extension lines and values for dimensions off crosshatched areas, but when this is unavoidable, the cross-hatching should be omitted where the dimension value is placed.



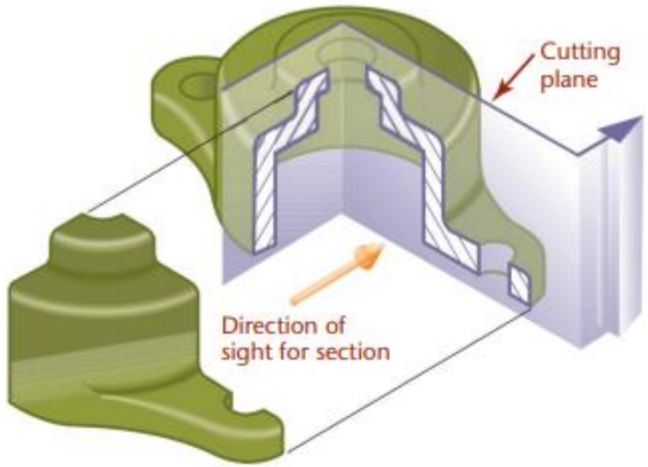
Extension lines and dimension values are not on hatched area | Section lines are omitted behind dimensioning | Dimensioning should not be on hatched area

HALF SECTIONS

Symmetrical objects can be shown effectively using a special type of section view called a ***half section*** (Figure 6). A half section exposes the interior of half of the object and the exterior of the other half. This is done by removing one quarter of the object. Half sections are not widely used to create detail drawings showing how to make a single part because it can be difficult to show all the dimensions clearly when some internal features are only partly shown in the sectioned half (Figure 6b).

In general,

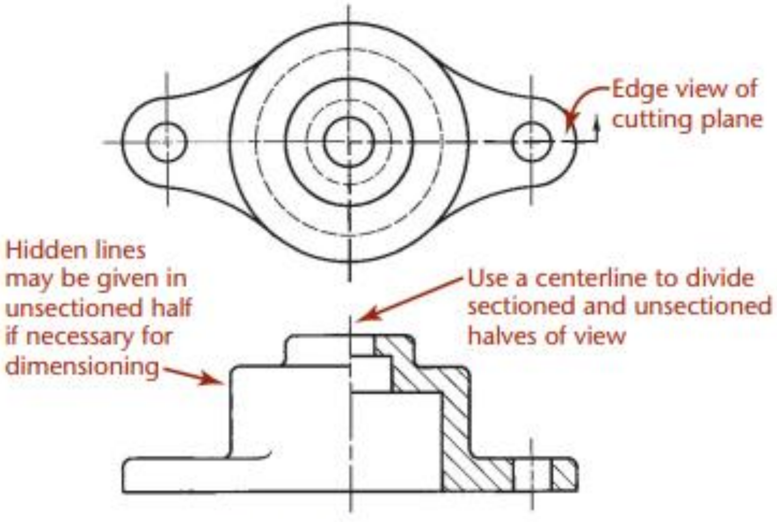
- Omit hidden lines from both halves of a half section, whenever possible.
- Use a centerline to divide the sectioned half and the unsectioned half, as shown in Figure 6b.



(a) Cutting plane

SECTIONAL VIEWS

Half-section drawings are most useful in showing an assembly where it is often necessary to show both internal and external construction in one drawing view and usually without dimensioning. A **broken-out section** may be preferred in some cases.



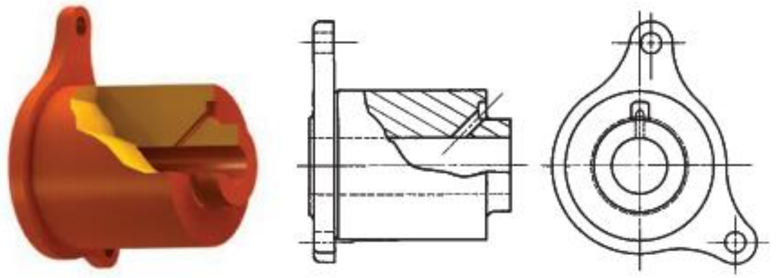
(b) Half section

BROKEN OUT SECTIONS

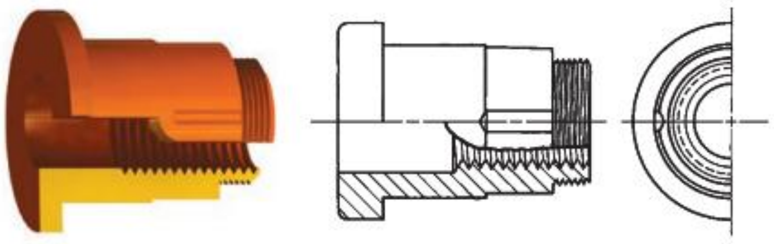
It often happens that only a partial section of a view is all that is needed to expose interior shapes. Such a section, limited by a break line, is called a **broken out** section.

In Figure 7, a full or half section is not necessary, and a small broken out section is sufficient to explain the construction.

In Figure 8, a half section would have caused the removal of half the keyway. The keyway is preserved by breaking out around it. In this case, the section is limited partly by a break line and partly by a centerline in the drawing.



Broken Out Section



Break around keyway

SECTIONAL VIEWS

REVOLVED SECTIONS

The shape of the cross section of a bar, arm, spoke, or other elongated object can be shown in the longitudinal view by using a ***revolved section***.

To create a revolved section, first imagine a cutting plane perpendicular to the centerline or axis of the object, as shown in Figure 9a. Next, revolve the plane 90° about a centerline at right angles to the axis, as shown in Figures 9b and c.

The visible lines adjacent to a revolved section may be broken out if desired, as shown in Figure 10.

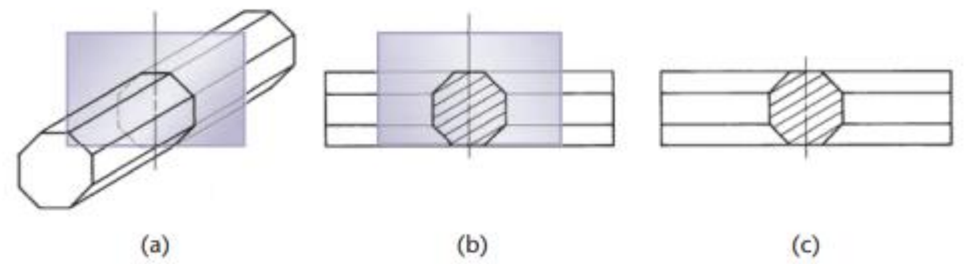


Figure 9

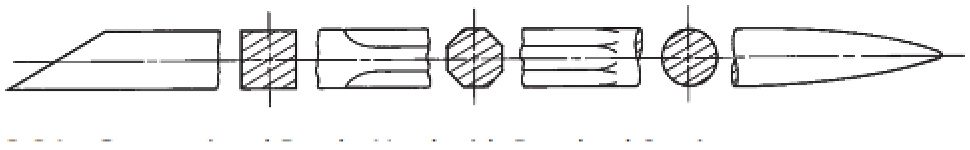


Figure 10

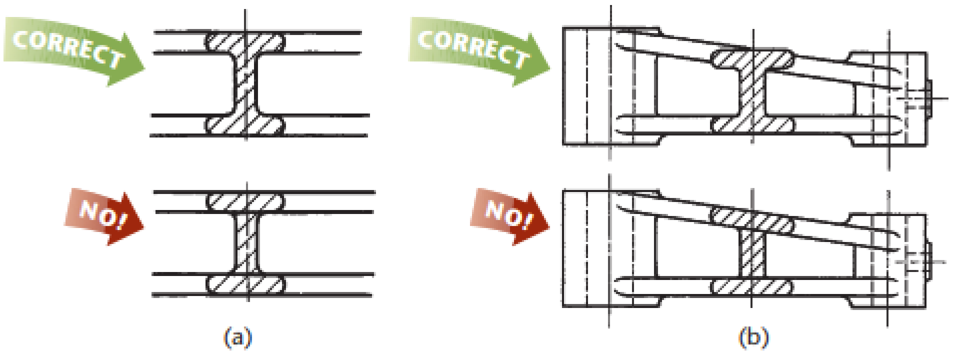


Figure 11

SECTIONAL VIEWS

When you superimpose the revolved section over the top of the view, be sure that any original lines of the view covered by the revolved view are removed (Figure 11a). Show the true shape of the revolved section, regardless of the direction of the lines in the view (Figure 11b).

Figure 12 shows examples of how revolved sections look in a drawing.

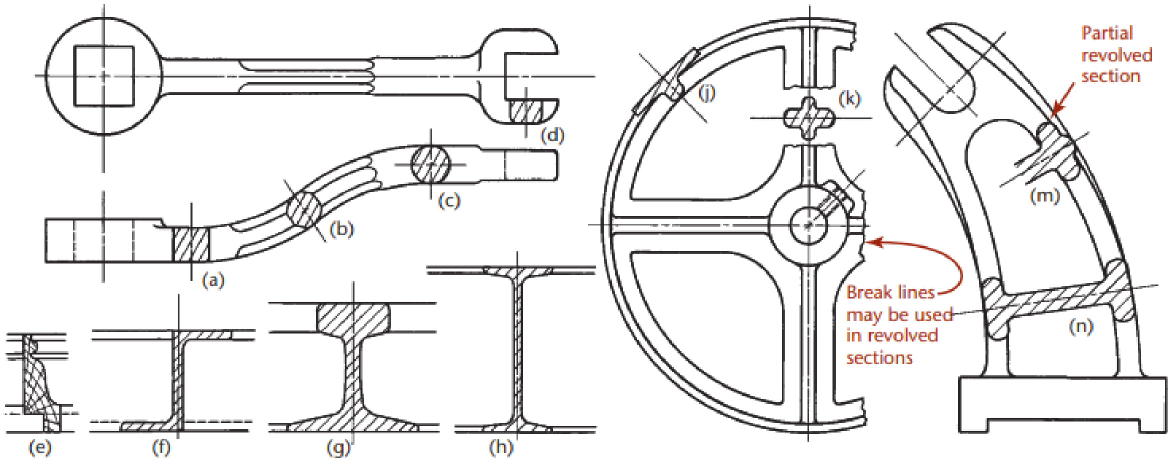


Figure 12

SECTIONAL VIEWS

REMOVED SECTIONS

A removed section (Figure 13) is one that is not in direct projection from the view containing the cutting plane—that is, it is not positioned in agreement with the standard arrangement of views. Be sure to keep the section in its normal orientation and do not turn it a different direction on the sheet. If you must rotate the view, use a rotation arrow as shown in Figure 14 and note the angle the view was rotated.

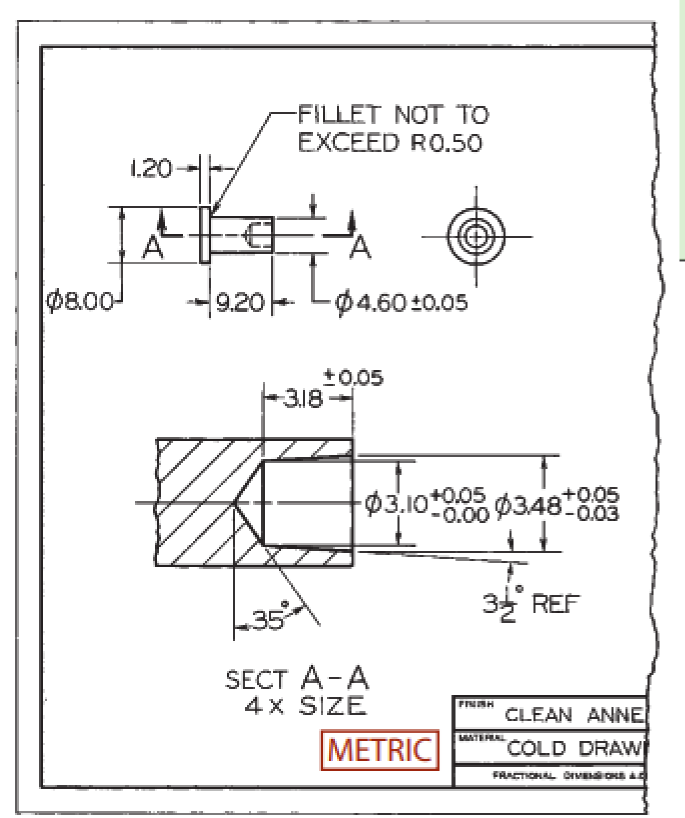


Figure 13

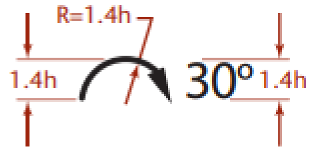


Figure 14

SECTIONAL VIEWS

Removed sections should be labeled, such as section A–A and section B–B, corresponding to the letters at the ends of the cutting-plane line (Figure 13). They should be arranged in alphabetical order from left to right on the sheet. Section letters should be used in alphabetical order, but letters I, O, and Q should not be used because they are easily confused with the numeral 1 or zero. Figure 14 shows several removed sections.

A removed section is often a **partial section**, in which only a portion of the section view is drawn.

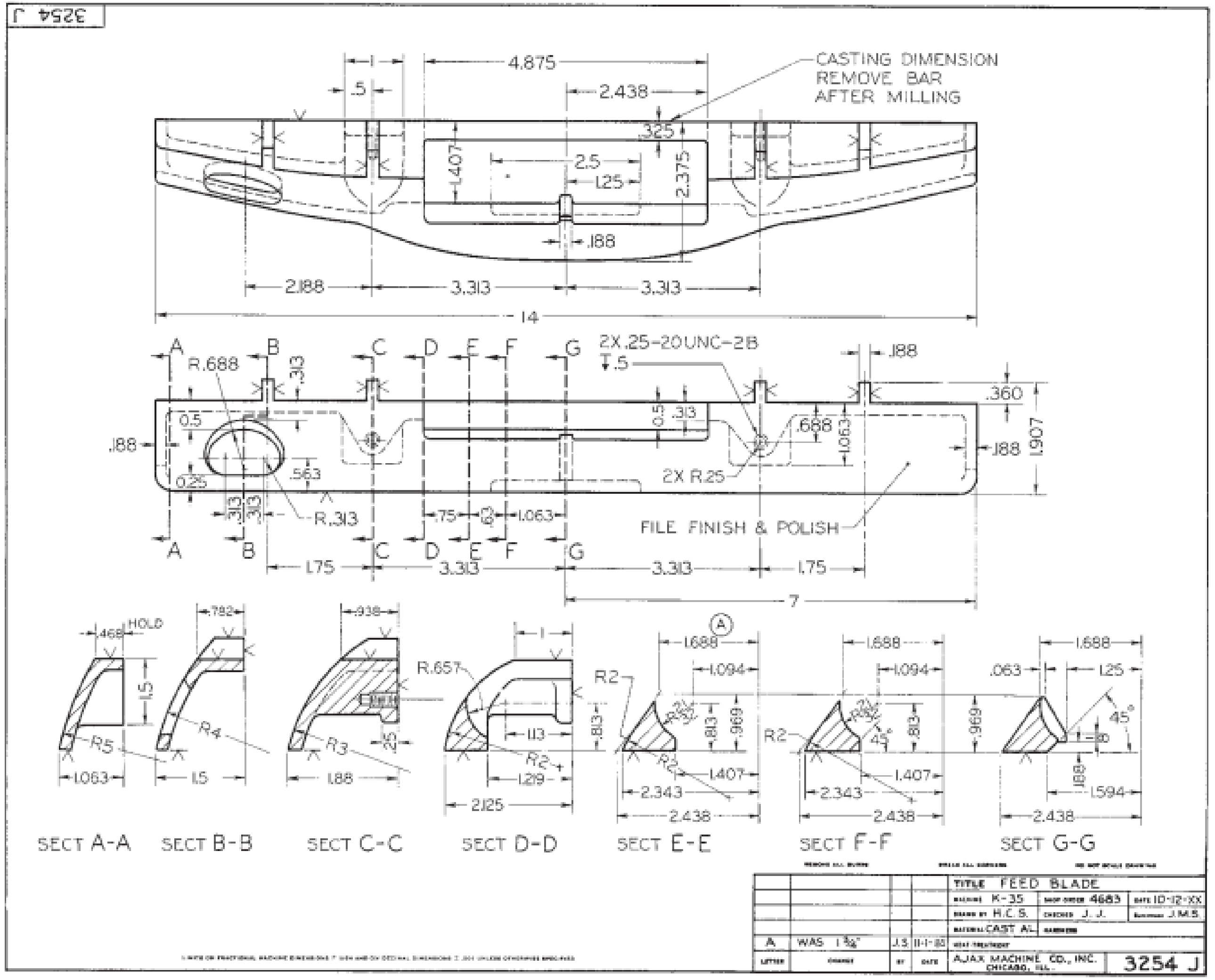
Removed sections are frequently drawn to an enlarged scale (Figure 15) to show detail and provide space for dimensions. When using an enlarged scale be sure to indicate the scale below the section view's title, as in Figure 13. A removed section should be placed so that it no longer lines up in projection with any other view. It should be separated clearly from the standard arrangement of views (see Figure 16). Whenever possible, removed sections should be on the same sheet as the regular views. If a section must be placed on a different sheet, cross-references

SECTIONAL VIEWS

should be given on the related sheets. A note should be given below the section title, such as

SECTION B-B ON SHEET 4, ZONE A3

Figure 15



SECTIONAL VIEWS

OFFSET SECTIONS

In sectioning complex objects, it is often desirable to show features that do not lie in a straight line by "offsetting" or bending the cutting plane. These are called offset sections. In Figure the cutting plane is offset in several places to include the hole at the left end, one of the parallel slots, the rectangular recess, and one of the holes at the right end. The front portion of the object is then imagined to be removed (Figure 16b). The path of the cutting plane is shown by the cutting-plane line in the top view (Figure 16c), and the resulting offset section is

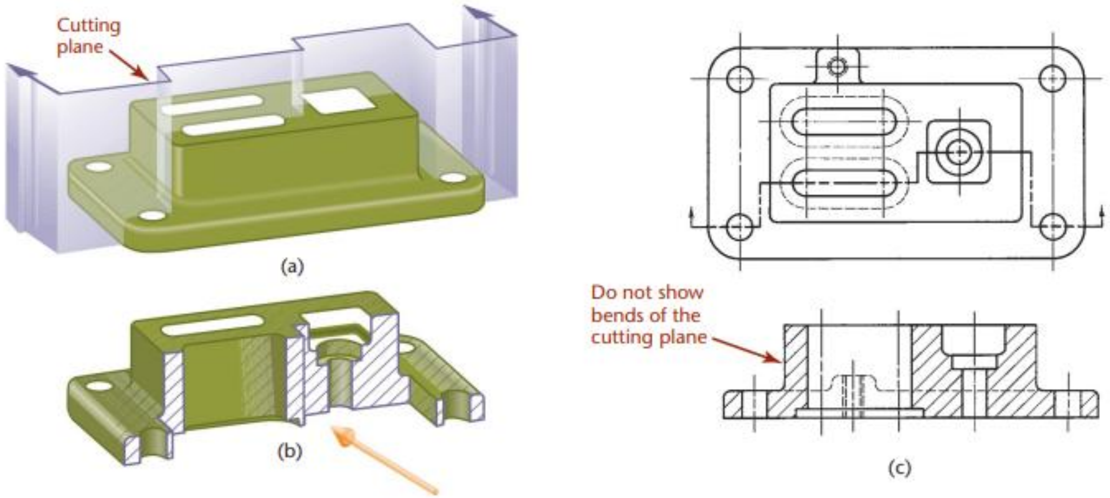


Figure 16

shown in the front view.

• The offsets or bends in the cutting plane are all 90°.

• The bends in the cutting plane are never shown in the section view.

Figure 16 also illustrates how hidden lines in a section eliminate the need for an additional view. In this case, an extra view would be needed to show the small boss on the back if hidden lines were not shown.

RIBS IN SECTION

To avoid giving a false impression of thickness and solidity, ribs, webs, gear teeth, and other similar flat features are not hatched with section lining even though the cutting plane slices them. For example, in Figure 17, the cutting plane A–A slices through the center of the vertical web, or rib, and the web is not sectioned (Figure 17b). Thin features are not hatched even though the cutting plane passes lengthwise through them. The incorrect section is shown in Figure 17c. Note the false impression of thickness or solidity resulting from section-lining the rib

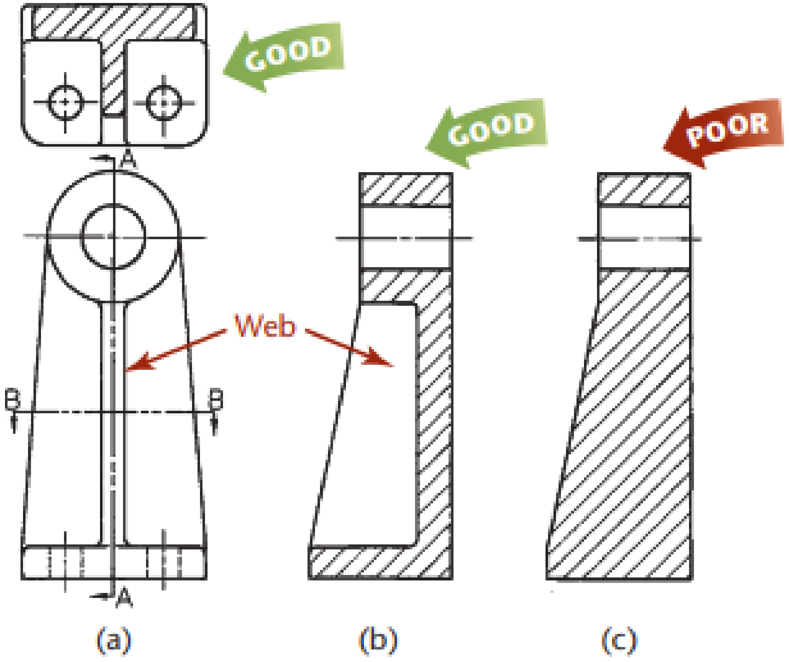


Figure 17

SECTIONAL VIEWS

If the cutting plane passes crosswise through a rib or any thin member, as in section B–B, section-line the feature in the usual manner, as in the top view of Figure 17a. If a rib is not sectioned when the cutting plane passes through it flatwise, it is sometimes difficult to tell whether the rib is actually present, as, for example, ribs A in Figure 18a and b. It is difficult to distinguish spaces B as open spaces and spaces A as ribs. In such cases, double-spaced section lining of the ribs should be used (Figure 18c). This consists simply of continuing alternate section lines through the ribbed areas, as shown.

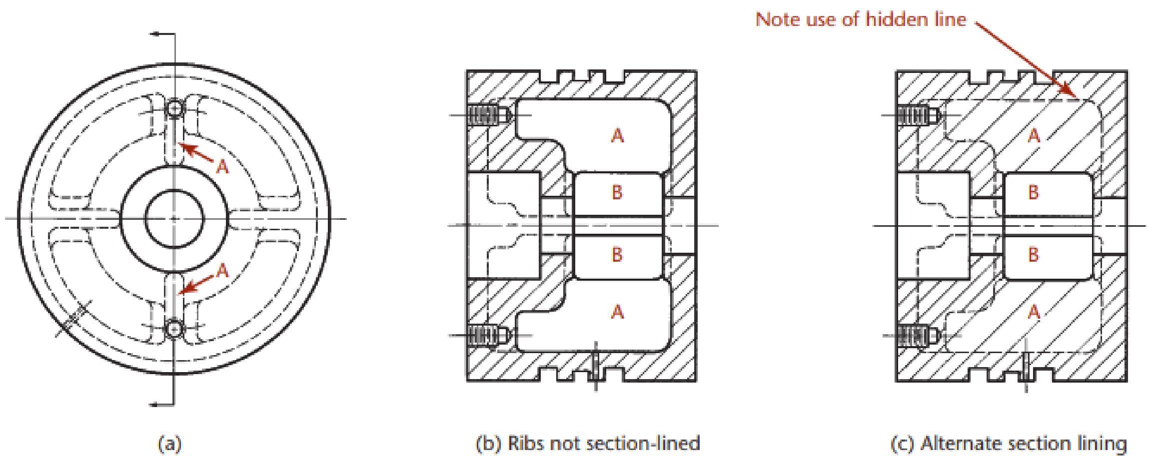


Figure 18

ALIGNED SECTIONS

When parts with angled elements are sectioned, the cutting plane may be bent to pass through those features. The plane and features are then imagined to be revolved into the original plane. For example, Figure 19 shows an **aligned section**. The cutting plane was bent to pass through the angled arm and then revolved to a vertical position (aligned), from where it was projected across to the section view.

The angle of revolution should always be less than 90° for an aligned section.

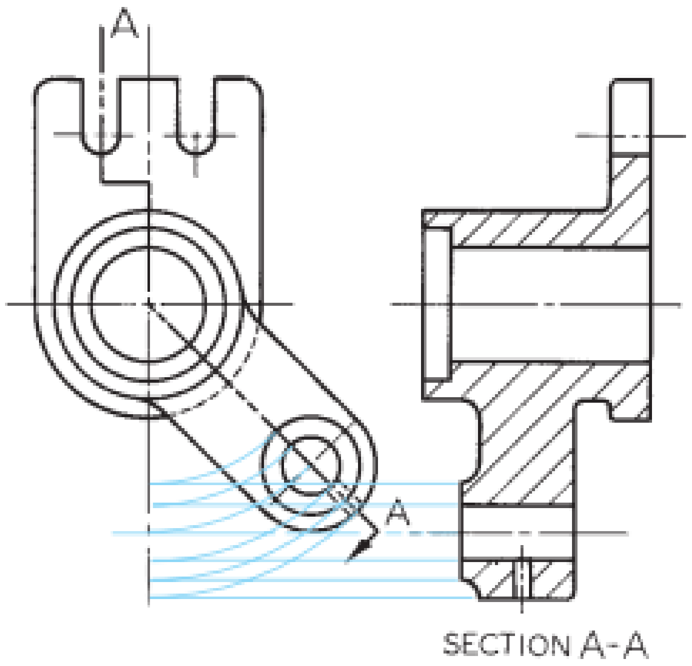


Figure 19

SECTIONAL VIEWS

Do not revolve features when the clarity of your drawing is not improved. In the exercises later in the chapter, you will see examples of when revolution should not be used.

In Figure 20 the cutting plane is bent to include one of the drilled and counterbored holes in the section view. The correct section view in Figure 20b gives a clearer and more complete description than does the section in Figure 20c, which is shown without any bend in the cutting plane.

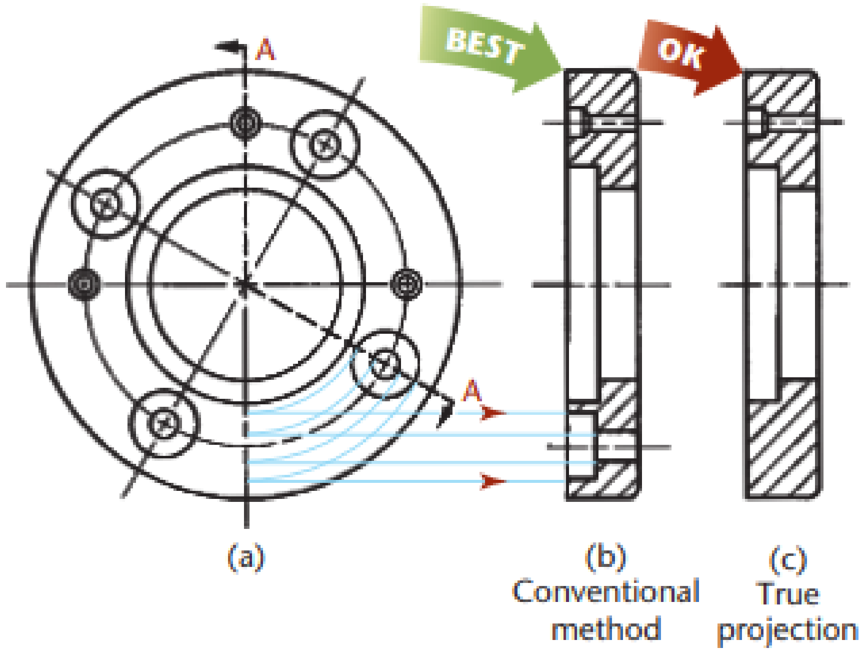


Figure 20

PARTIAL VIEWS

If space is limited on the paper or to save time, partial views may be used with sectioning (Figure 21). Half views are shown in Figures 21a and b in connection with a full section and a half section, respectively. In each case the back half of the object in the circular view is shown, to remove the front portion of the object and expose the back portion in the section. Another method of drawing a partial view is to break out much of the circular view, retaining only those features that are needed for minimum representation (Figure 21c).

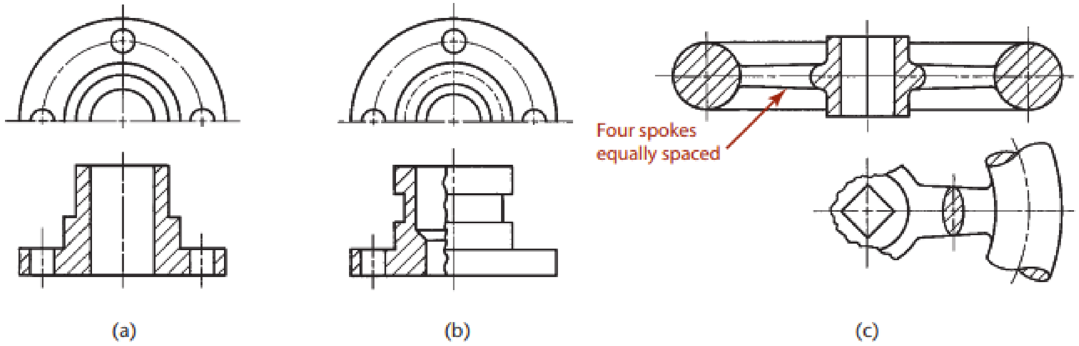


Figure 21

PARTS NOT SECTIONED

Section lines are not drawn on the following objects when the cutting plane passes lengthwise through them.

- SPOKES
- GEAR TEETH
- SHAFTS
- FASTENERS (BOLTS, NUTS, PINS, RIVETS, STUDS, WASHERS, SET SCREWS)
- SPINDLES

The objects above are more easily recognized by their external features, so they are not shown in section if

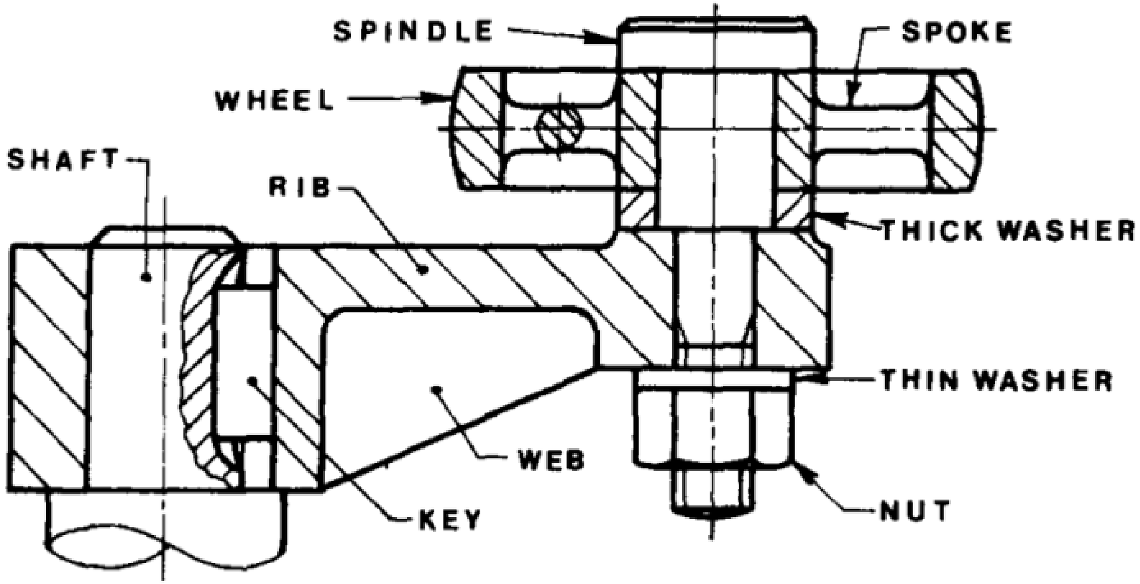


Figure 22

SECTIONAL VIEWS

longitudinally. However, section lines are drawn when the cutting plane cuts across the axis of the objects Also, when webs or ribs are cut along their length by a cutting plane, they are not sectioned, in order to avoid a false appearance of solidity. The webs and ribs are usually thin in comparison to the overall thickness of the main body. If a cutting plane cuts across the webs or ribs, then they are shown in section in the usual way.

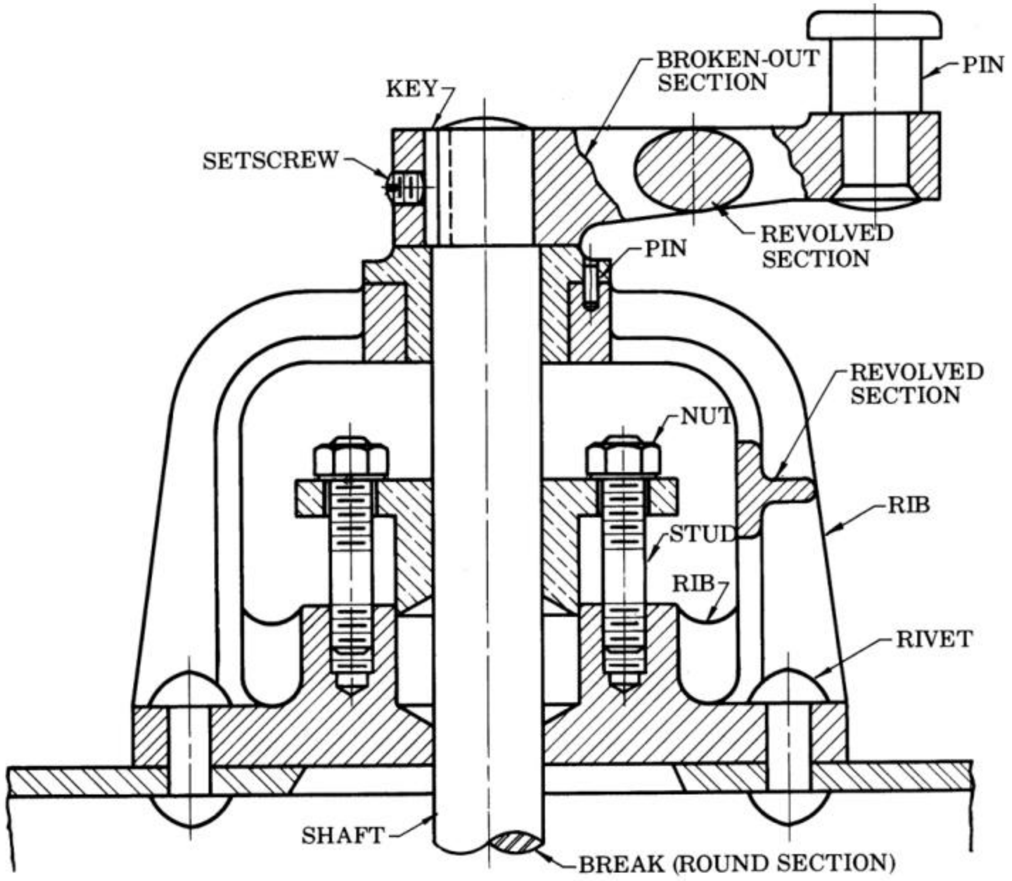
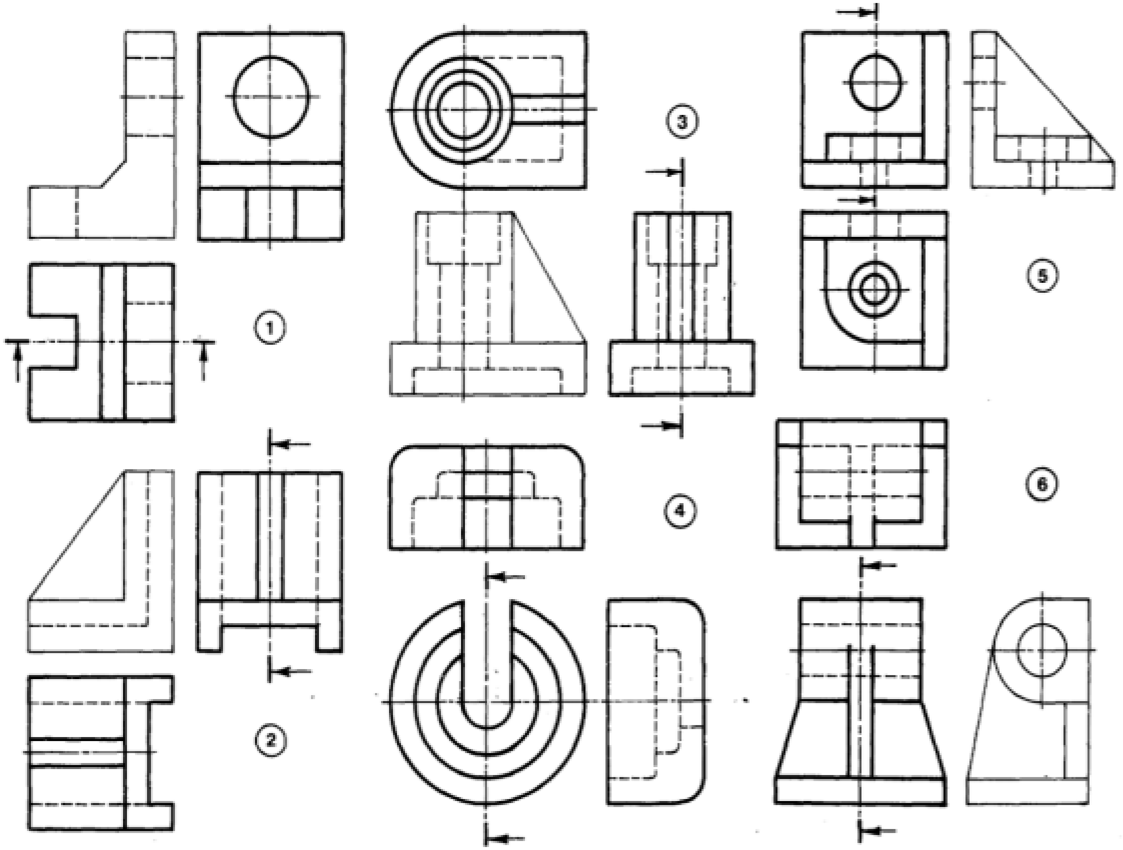


Figure 23

SECTIONAL VIEWS EXERCISES

1. The components shown in Figure 24 are drawn in first-angle or third-angle projection. Sketch the sectional view of each component.

Figure 24



SECTIONAL VIEWS EXERCISES

2. With a scale of 1:1, draw in first-angle projection the following views of the bracket shown in Figure 25:

(a) the front view in the direction of arrow F,

(b) (b) the plan,

(c) (c) a sectional end view along cutting plane X-X.

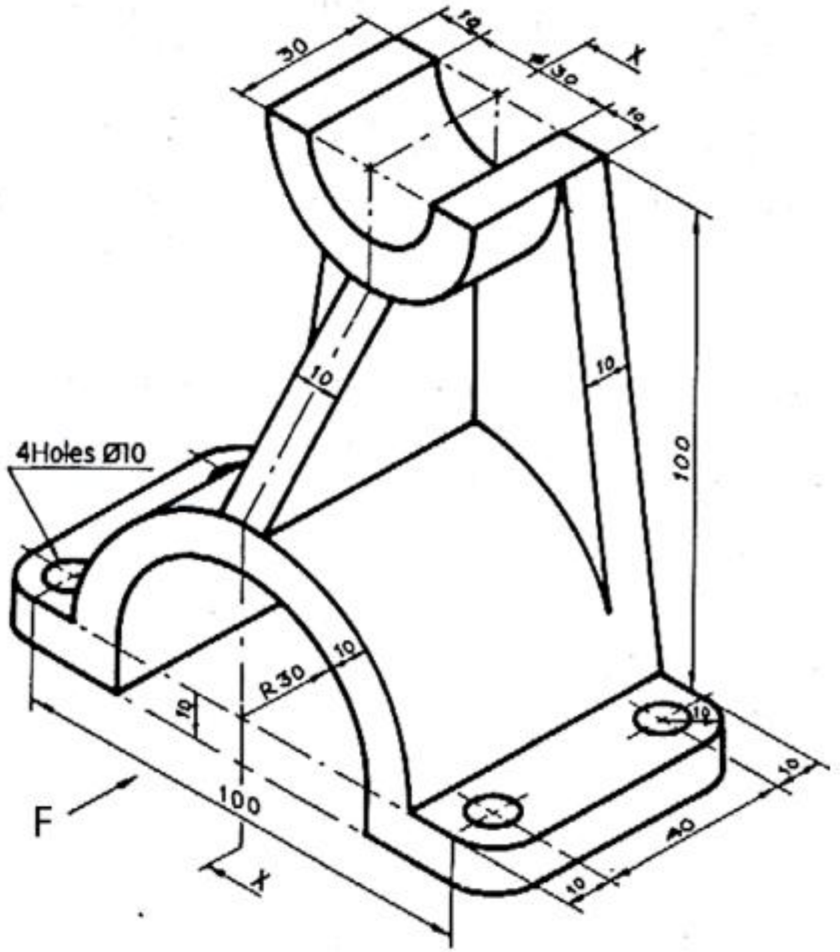


Figure 25

3. With a scale of 1:1, draw in third-angle projection the following views of the bracket shown in Figure 26:

(a) the front view in the direction of arrow,

(b) (b) the plan,

(c) (c) a sectional end view along cutting plane A-A.

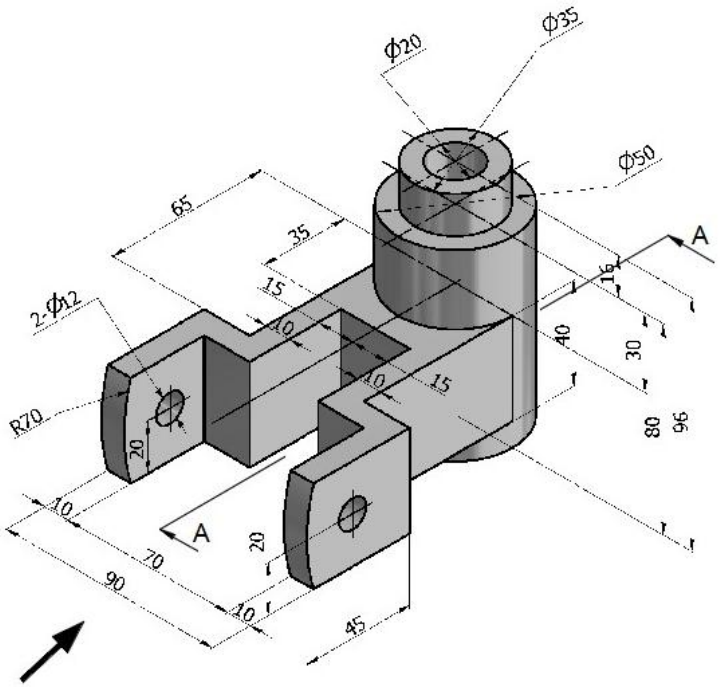


Figure 26

4. With a scale of 1:1, draw in third-angle projection the following views of the bracket shown in Figure 27:

(a) the sectional front view in the direction of arrow X, sectional view A-A.

(b) the plan,

(c) a sectional end view along cutting plane B-B.

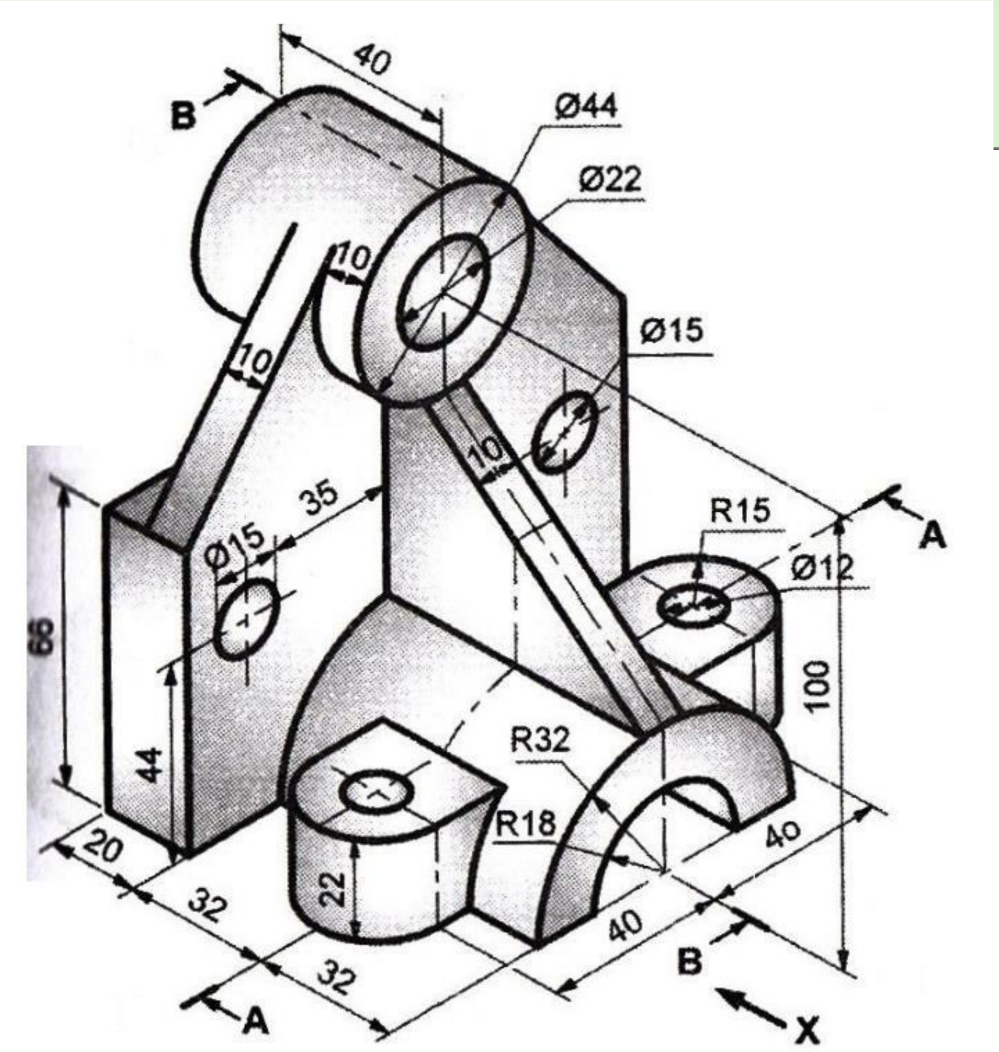
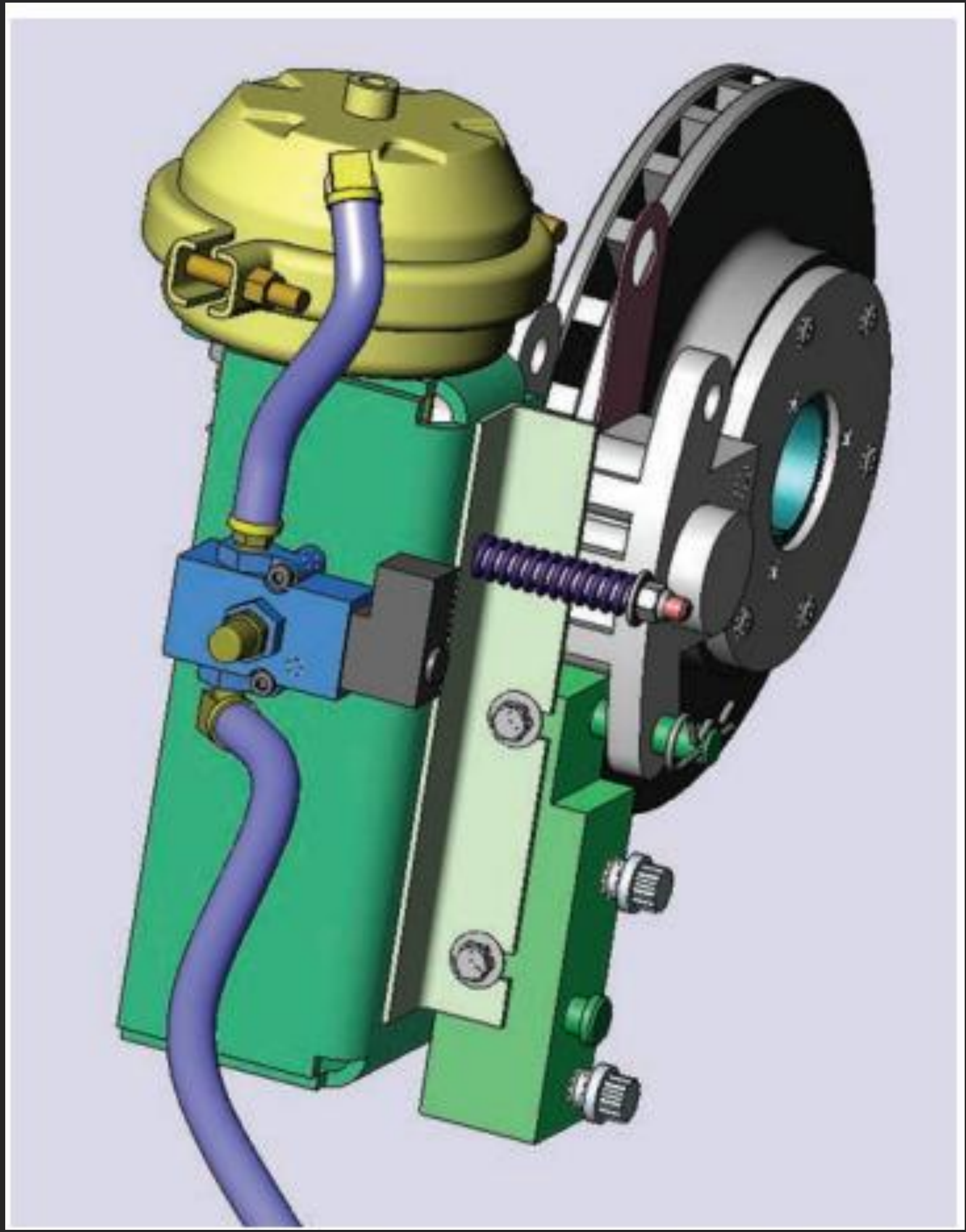


Figure 27

ASSEMBLY DRAWINGS

We will cover these skills:

- Introduction to detailed drawings.
- Introduction to Assembly drawing.
- Types of Assembly Drawings
- Identification numbers.
- Parts List.
- Exercises.



ASSEMBLY DRAWINGS

INTRODUCTION

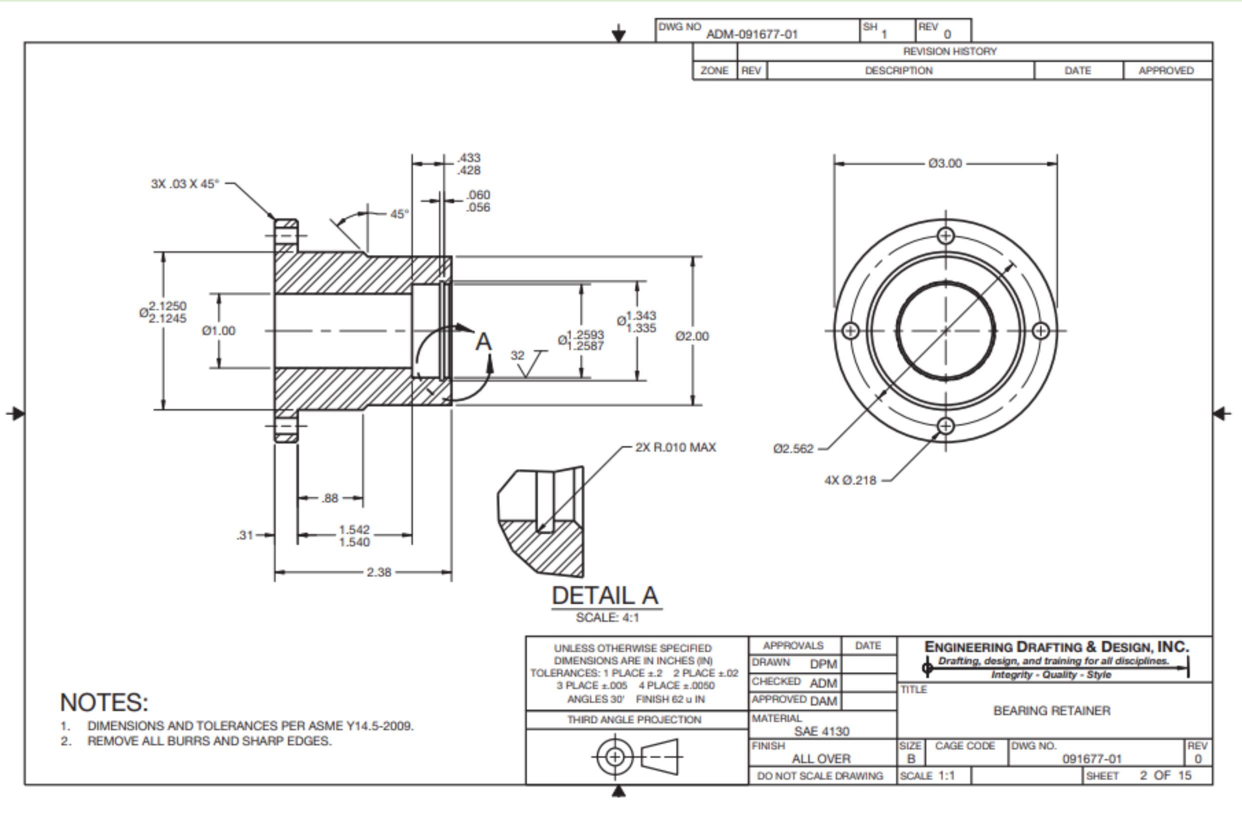
Most of the drawings that have been shown as examples or assigned as problems in this text are called **DETAIL DRAWINGS**. Detail drawings are the kind of drawings that most entry-level mechanical drafters prepare. When a product is designed and drawings are made for manufacturing, each part of the product must have a drawing. These drawings of individual parts are referred to as **detail drawings**. Component parts are assembled to create a final product, and the drawing that shows how the parts go together is called **the assembly drawing**. Associated with the assembly drawing and coordinated to the detail drawings is the parts list that identifies all of the parts in the assembly. When the detail drawings, assembly drawing, and parts list are combined, they are referred **to as a complete set of working drawings**. **Working drawings are a set of drawings that supply all the information necessary to manufacture any given product**. A set of working drawings includes all the information and instructions needed for the purchase or

ASSEMBLY DRAWINGS

manufacture of parts and the assembly of those parts into a product.

DETAIL DRAWINGS

Detail drawings, which are used by workers in manufacturing, are drawings of each part contained in the assembly of a product. **The only parts that do not have to be drawn are standard parts. Standard parts, also known as purchase parts, are items that can be purchased from an outside supplier more economically than they can be manufactured**. Examples of standard or purchased parts are common



bolts, screws, pins, keys, and any other product that can be purchased from a vendor. Standard parts do not have to be drawn because a written description clearly identifies the part.

ASSEMBLY DRAWINGS

INTRODUCTION

Most products are composed of several parts. A drawing showing how all of the parts fit together is called an **ASSEMBLY DRAWING**. Assembly drawings differ in the amount of information provided, and this decision often depends on the nature or complexity of the product. Assembly drawings are generally multiview drawings. Your goal in the preparation of assembly drawings is to use as few views as possible to completely describe how each part goes together. In many cases, a single front view is the only view

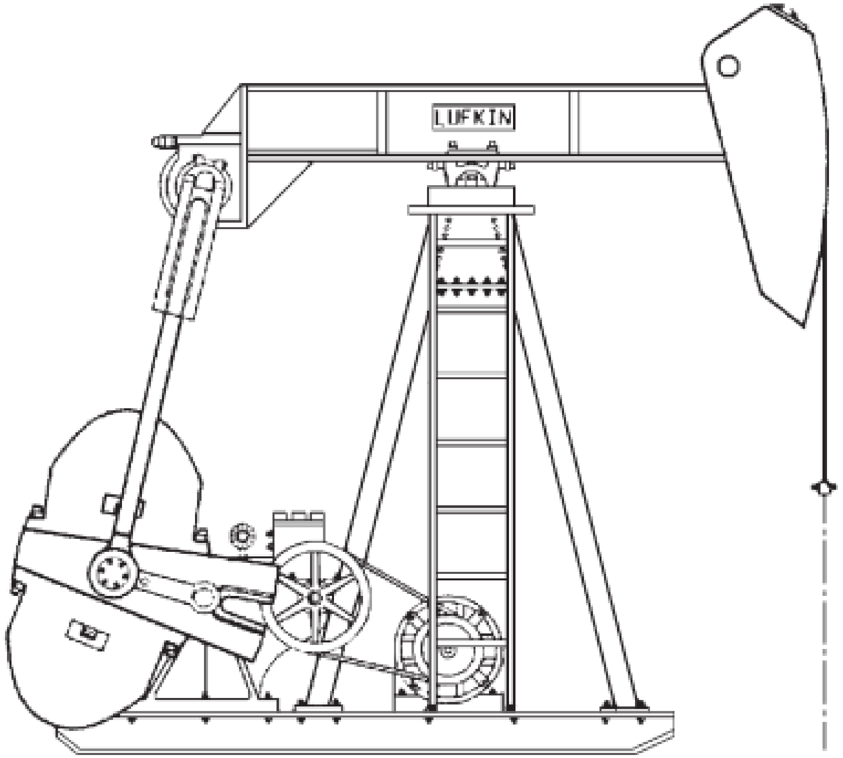


Figure 28: Assembly drawing with single front view

ASSEMBLY DRAWINGS

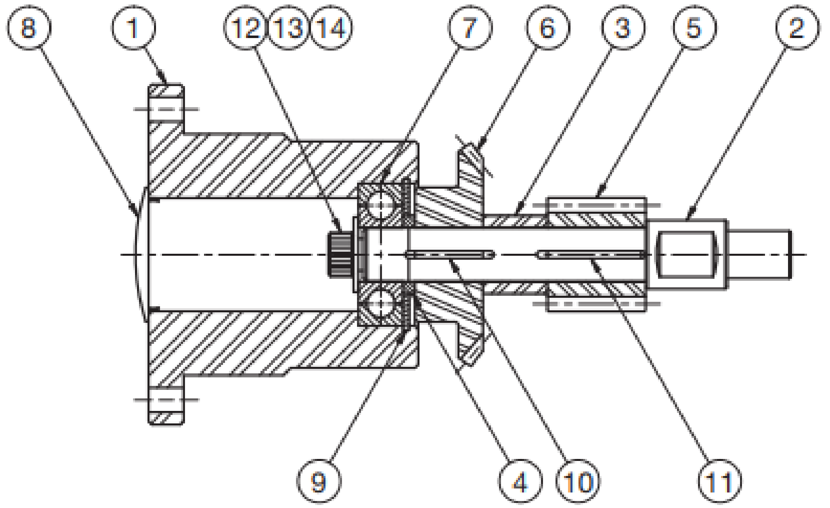
necessary to describe the assembly (see Figure 28). Full sections are commonly associated with assembly drawings, because a full section exposes the assembly of most or all of the internal features as shown in Figure 29. If one section or view is not enough to show how the parts fit together, then a number of views or sections can be necessary. In some situations, a front view or group of views with broken-out sections is the best method of showing the external features while exposing some of the internal features (see Figure 30).

You must make the assembly drawing clear enough for the assembly department to put the product together. Other elements of assembly drawings that make them different from detail drawings are that they usually contain few or no hidden lines or dimensions. <u>Hidden lines should be avoided on assembly drawings unless necessary for clarity</u>. The common practice is to draw an exterior view to clarify outside features and a sectional view to expose interior features. <u>Dimensions serve no purpose on an assembly drawing unless the</u>

ASSEMBLY DRAWINGS

DWG NO 091677 | SH 1 | REV 0

REVISION HISTORY				
ZONE	REV	DESCRIPTION	DATE	APPROVED



FIND NO	QTY REQD	PART OR IDENT NO	NOMENCLATURE OR DESCRIPTION	NOTES OR REMARKS
14	1	091677-14	FLAT WASHER	1/4 NOMINAL
13	1	091677-13	LOCK WASHER	1/4 NOMINAL
12	1	091677-12	SHCS	1/4-20 UNC-2A X .750
11	1	091677-11	KEY STOCK	□.125 X 1.000
10	1	091677-10	KEY STOCK	□.125 X .750
9	1	N5000-125	SNAP RING	WALDES-TRUARC
8	1	091677-08	END PLUG	SAE 1020
7	1	MRC201-S22	BALL BEARING	Ø.4724 BORE TRW
6	1	HL149Y-G	BEVEL GEAR	BOSTON 20° PA 2IN PD
5	1	H1056 RH	WORM GEAR	BOSTON 12 DP SGL THD
4	1	091677-04	SPACER	TS .750 OD X 11 GA
3	1	091677-03	SPACER	TS .750 OD X 11 GA
2	1	091677-02	CROSS SHAFT	BAR Ø.750 SAE 4130
1	1	091677-01	BEARING RETAINER	BAR Ø3.00 SAE 4130

PARTS LIST

APPROVALS		DATE
DRAWN	DPM	
CHECKED	ADM	
APPROVED	DAM	

MATERIAL

FINISH

DO NOT SCALE DRAWING

THIRD ANGLE PROJECTION

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TITLE
CROSS SHAFT ASSEMBLY

SIZE B | CAGE CODE | DWG NO. 091677 | REV 0

SCALE 1:1 | SHEET 1 OF 15

Figure 29: Assembly in full section.

ASSEMBLY DRAWINGS

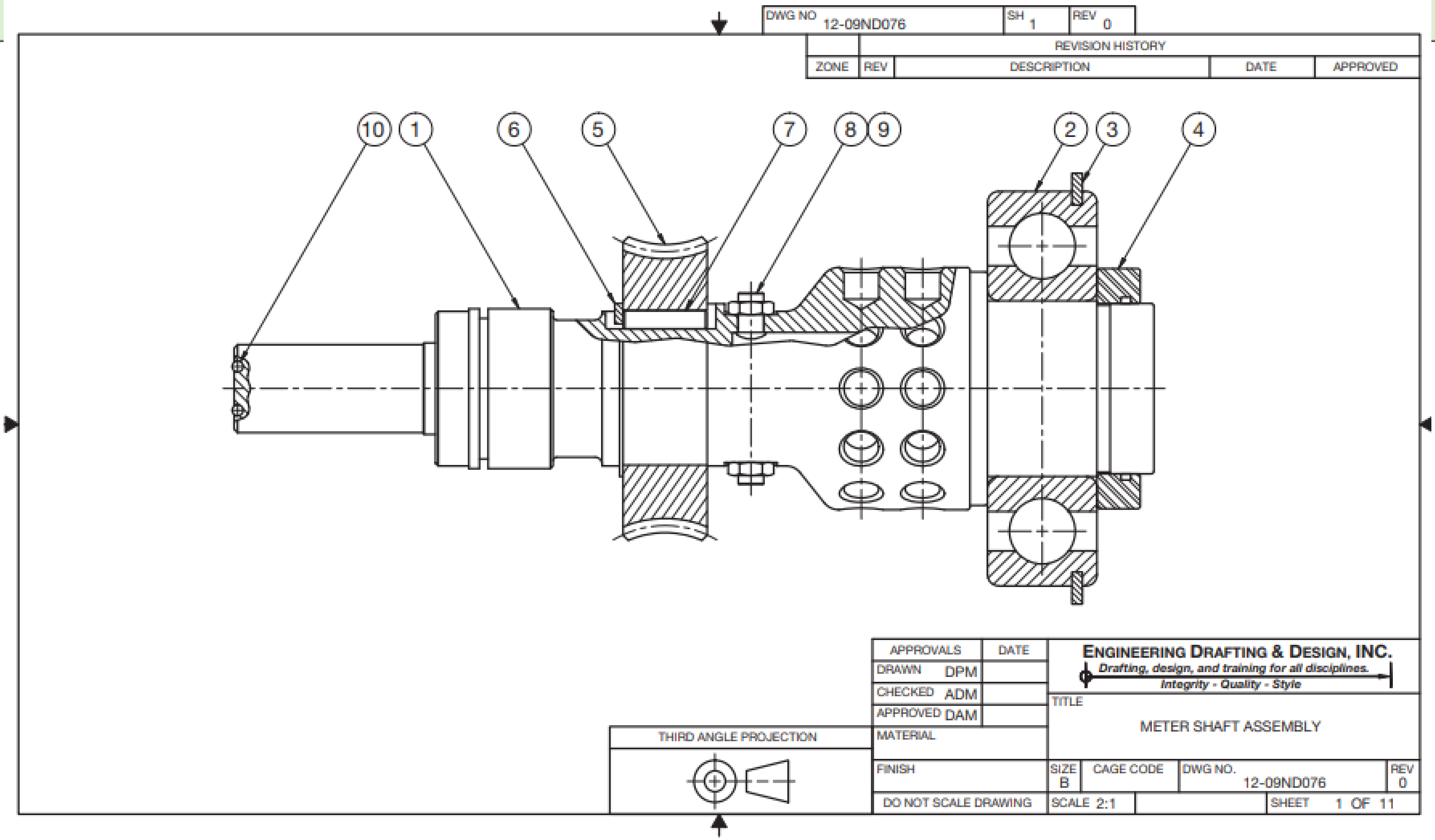


Figure 30: Assembly with broken-out sections.

ASSEMBLY DRAWINGS

dimensions are used to show the assembly relationship of one part to another. Assembly dimensions are only necessary when a certain distance between parts must exist before proper assembly can take place.

Assembly drawings can contain some or all of the following information:

• One or more views. Auxiliary views are used as needed.

• Sections necessary to show internal features, function, and assembly.

• Enlarged views necessary to show adequate detail.

• Arrangement of parts.

• Overall size and specific dimensions necessary for assembly.

• Manufacturing processes necessary for, during, or after assembly.

• Reference or item numbers that key the assembly to a parts list and to the details.

• Parts list or bill of materials.

ASSEMBLY DRAWINGS

TYPES OF ASSEMBLY DRAWINGS

There are several different types of assembly drawings used in industry, including:

- Layout, or design, assembly.
- General assembly.
- Working-drawing or detail assembly.
- Erection assembly.
- Subassembly.
- Pictorial assembly.

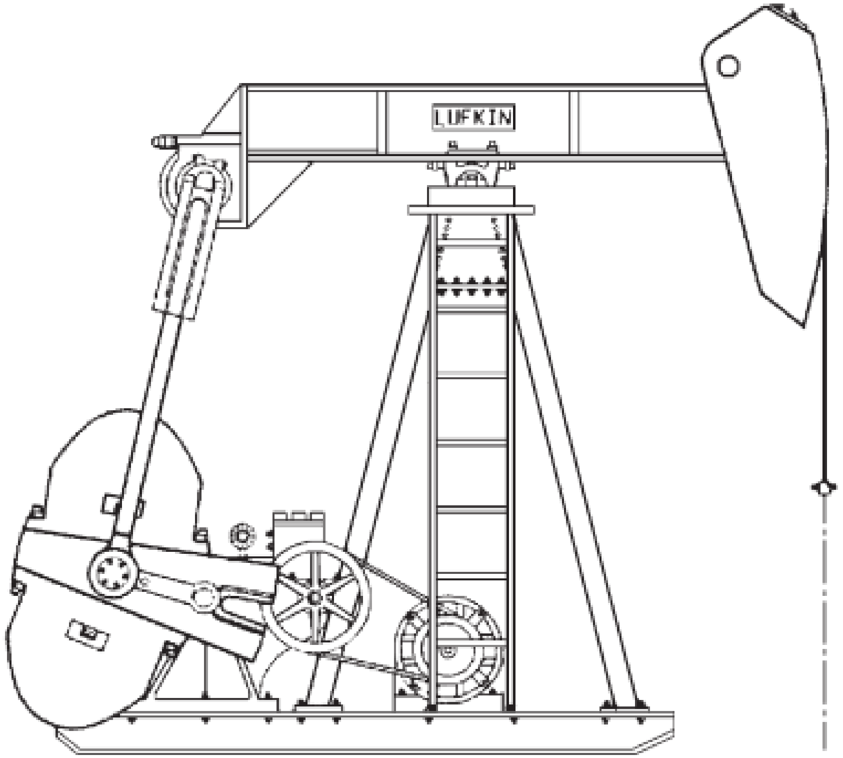
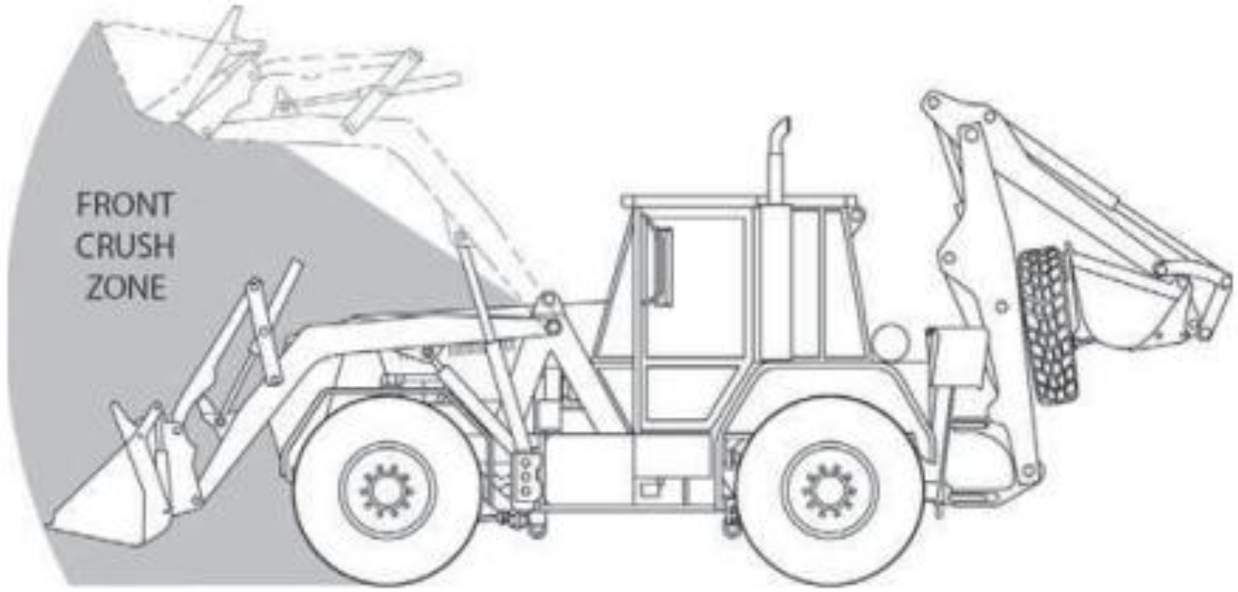


Figure 34: Assembly drawing with single front view

ASSEMBLY DRAWINGS

LAYOUT ASSEMBLY

Engineers and designers can prepare a design layout in the form of a sketch or as an informal drawing. These engineering design drawings are used to establish the relationship of parts in a product assembly. From the layout, the engineer prepares sketches or informal detail drawings for prototype construction. Layout or design assemblies can take any form, depending on the drafting ability of the engineer, the period for product implementation, the complexity of the product, or company procedures.



Layout assembly. The limits of operation are shown with phantom lines and with a shaded area.

ASSEMBLY DRAWINGS

GENERAL ASSEMBLY

They are the most common types of assemblies that are used in a complete set of working drawings. A set of working drawings contains three parts: detail drawings, an assembly drawing, and a parts list. Each part is identified with a balloon containing a number keyed to the parts list. A balloon is a circle placed on the drawing with a part identification number inside the circle. Each balloon is connected to its related part with a leader line. The balloon part identification number correlates to the same number identifying

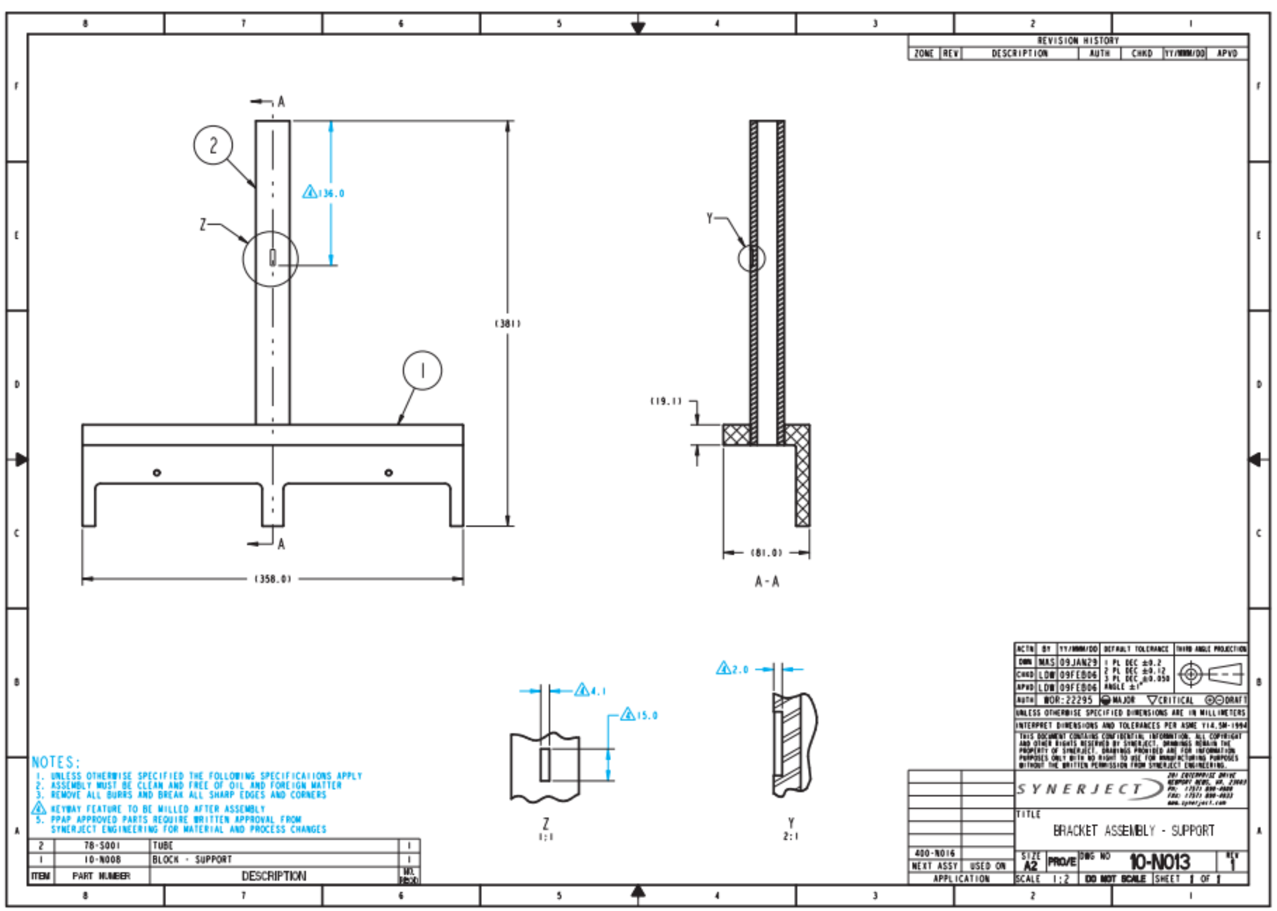


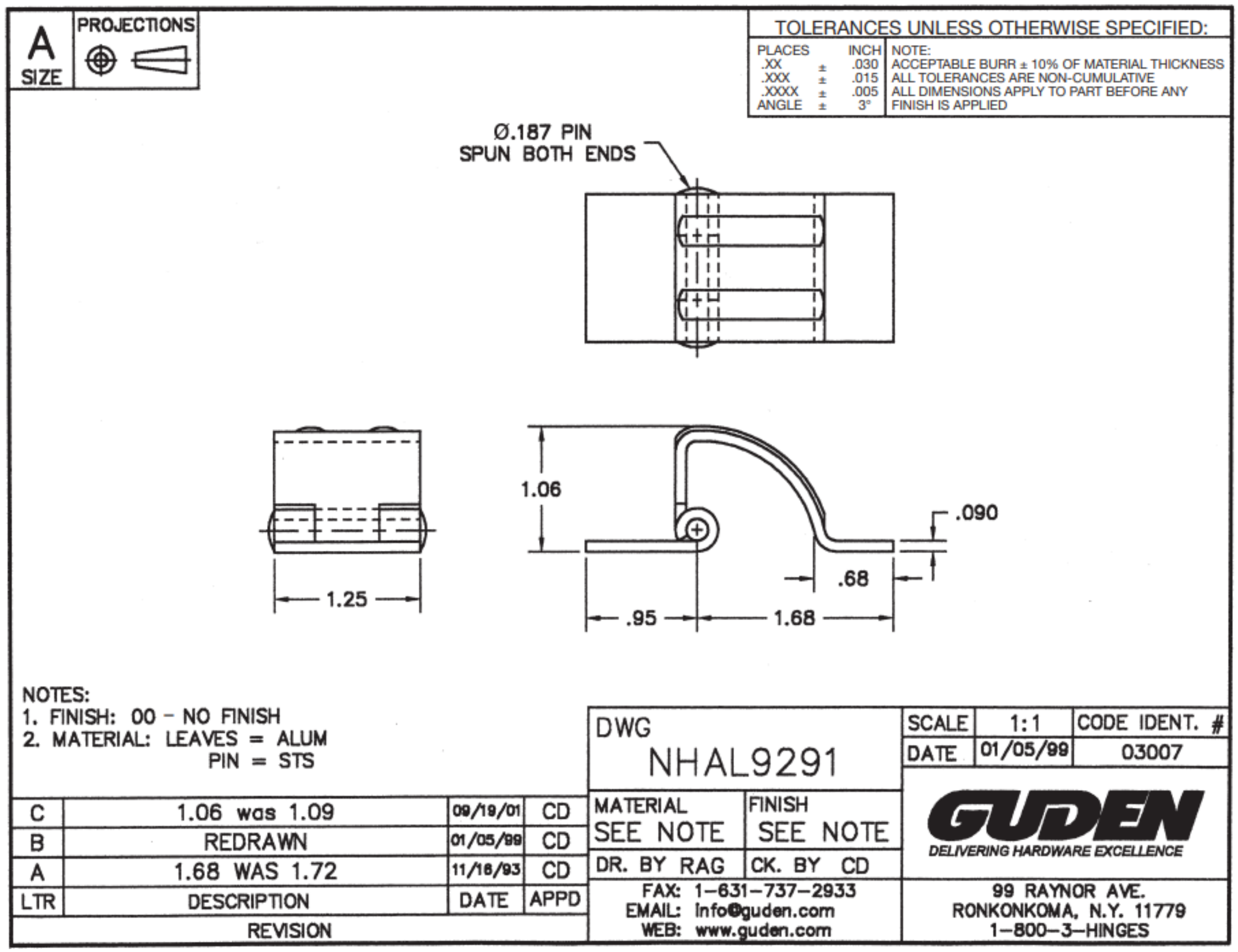
Figure 31: Assembly drawing with single front view

the part in the parts list.

ASSEMBLY DRAWINGS

DETAIL ASSEMBLY

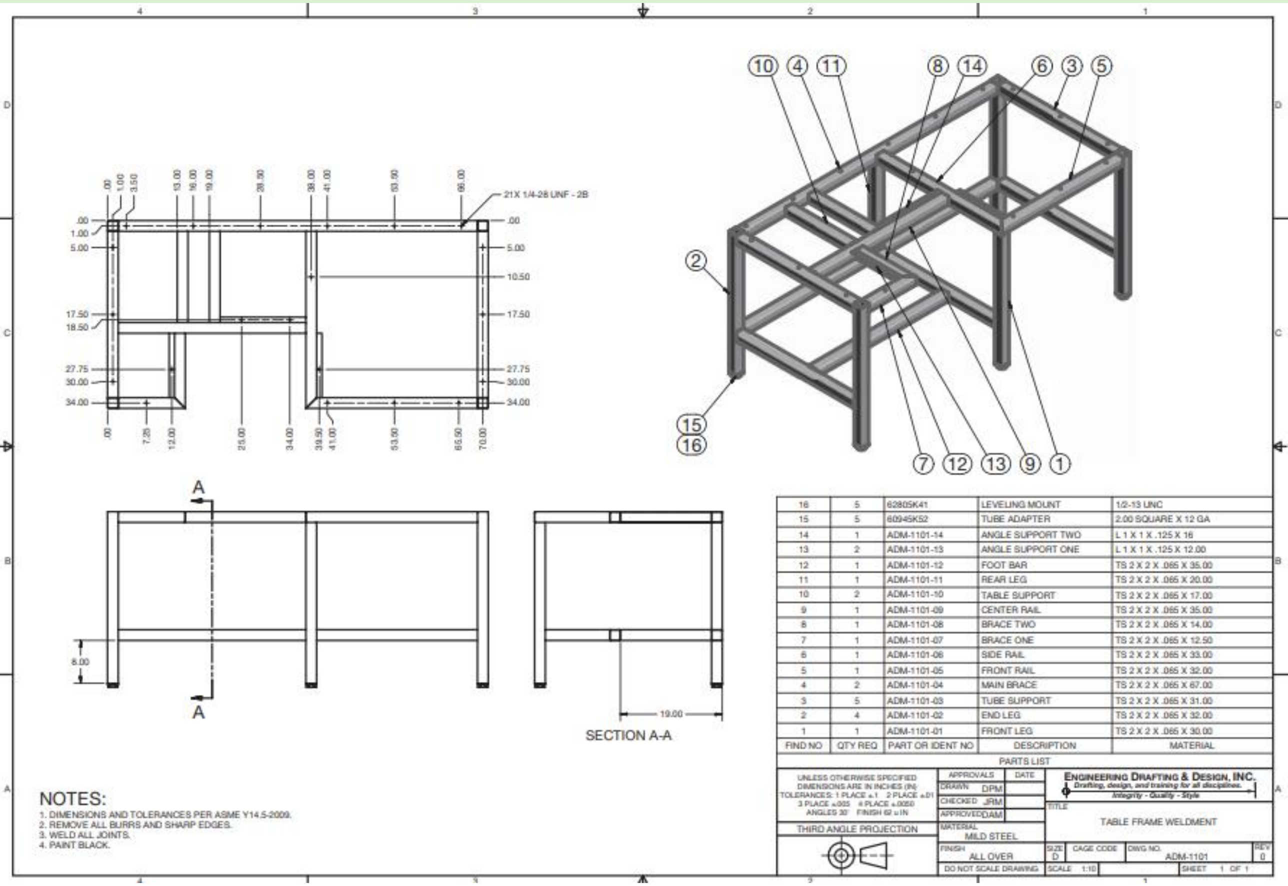
It shows the details of parts combined on the same sheet with an assembly of the parts. This practice is also referred to as a **working-drawing assembly**. Although this application is not as common as a general assembly, it is a practice used at some companies. The use of working-drawing assemblies can be a company standard, or this technique can be used in a specific situation even when it is not considered a normal procedure at a particular company.



This product is an example of a detail assembly with its component parts assembled in a manufacturer's catalog or on a Web site.

ERECTION ASSEMBLY

Erection assemblies usually differ from general assemblies in that dimensions and fabrication specifications are commonly included. Erection assemblies are used for fabrication and assembly, and they are typically associated with products that are made of structural steel or with cabinetry. The figure shows an erection assembly with multiviews, fabrication dimensions, and an isometric drawing that helps display how the parts fit together.

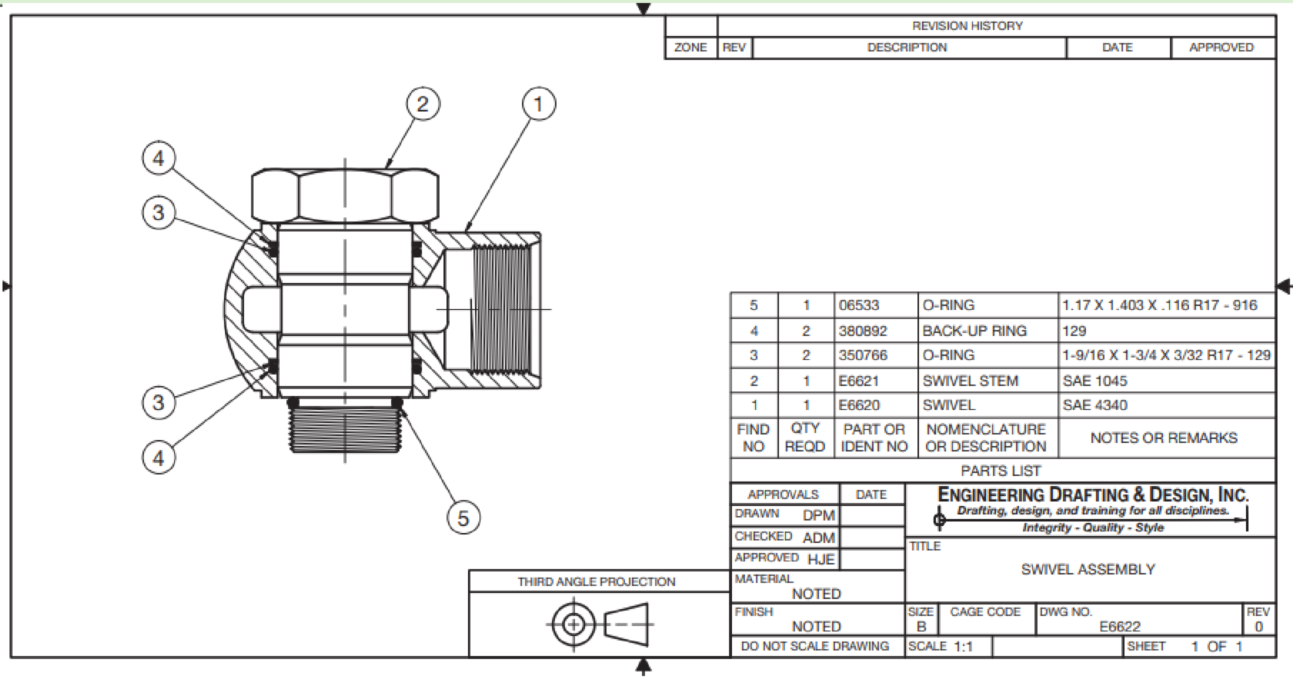


Erection assembly. Dimension values in this figure are in inches.

ASSEMBLY DRAWINGS

SUB ASSEMBLY

The complete assembly of a product can be made up of several separate component assemblies. These individual unit assemblies are called subassemblies. A complete set of working drawings can have several subassemblies, each with its own detail drawings. The subassemblies are put together to form the general assembly. The general assembly of an automobile, for example, includes the subassemblies of the drive components, the engine components, and the steering column, just to name a few.

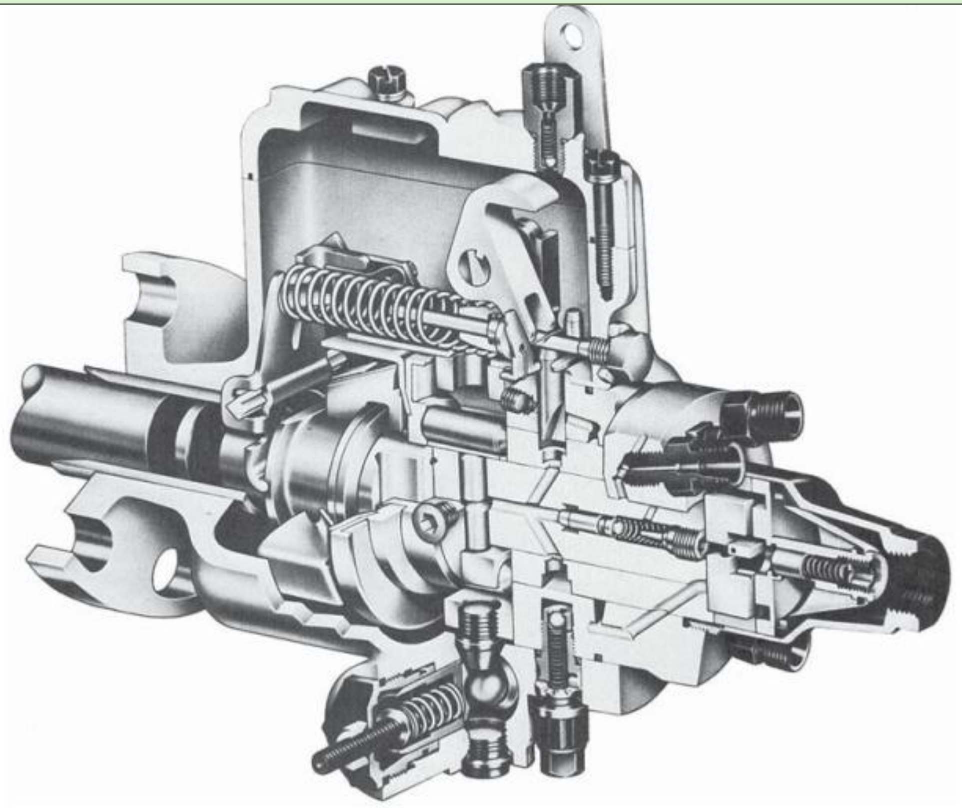


Subassembly with parts list.

ASSEMBLY DRAWINGS

PICTORIAL ASSEMBLY

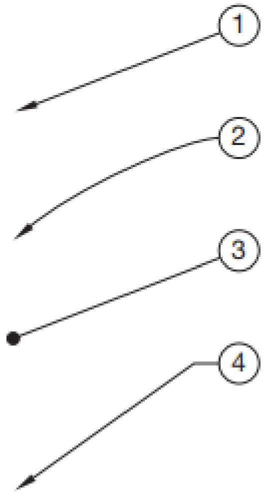
Pictorial assemblies are used to display a pictorial rather than multi-view representation of the product, which can be used in other types of assembly drawings. Pictorial assemblies can be made from photographs, artistic renderings, or CADD models. The pictorial assembly can be as simple as the isometric drawing, which is used to assist workers in the assembly of the product. Pictorial assemblies are commonly used in product catalogs or brochures.



Pictorial assembly

IDENTIFICATION NUMBERS

Identification or item numbers are used to key the parts from the assembly drawing to the parts list. Identification numbers are generally placed in balloons. Balloons are circles that are connected to the related part with a leader line. Numbers in balloons are common, although some companies prefer to use identification letters. All balloons on the drawing are the same size. The leaders connecting the balloons to the parts are thin lines that can be presented in any one of several formats, depending on company standards. Notice that the leaders can terminate with arrowheads or dots. Whichever method of connecting balloons is used, the same technique should be used throughout the entire drawing. An exception is the common practice of using leader arrowheads when pointing to the profile of a part and using a leader with a dot when pointing to the surface of a part.



BALLOON OPTIONS

PARTS LISTS

The parts list, also known as a bill of materials or list of materials, is usually combined with the assembly drawing, but it remains one of the individual components of a complete set of working drawings. The information associated with the parts list generally includes:

• Item number—from balloons. Item number is also called find number.

• Quantity—the number of that particular part used on this assembly

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ASSEMBLY: **CRYOGENIC VALVE** ASSEMBLY NUMBER: **MSIV12-09-0087**
NUMBER OF UNITS: **250** MANAGEMENT: ____ DATE: _______

PARTS LIST

HAVE	NEED	FIND NO	QTY REQD	DIA	PART OR IDENT NO	NOMENCLATURE OR DESCRIPTION	NOTES OR REMARKS
✓		1	1		087930-01	HOUSING	
✓		2	1		087930-02	HOUSING COVER	
	✓	3	4		224707-10	3/4 X 1/4-20UNC HEX BOLT	
	✓	4	4		224707-20	3/4 FLAT WASHER	
	✓	5	4		224707-33	3/4 X 1/4-20UNC HEX NUT	
✓		6	2		224707-34	SHAFT ADAPTER	

ASSEMBLY DRAWINGS

• Part or drawing number, which is a reference back to the detail drawing.

• Description, which is usually a part name or complete description of a purchase part or stock specification, including sizes or dimensions.

• Material identification—the material used to make the part.

• Information about vendors for purchase parts.

• Sheet number.

The first four elements listed are the most common items. When all six elements are provided, the parts list can also be called **a list of materials or a bill of materials**.

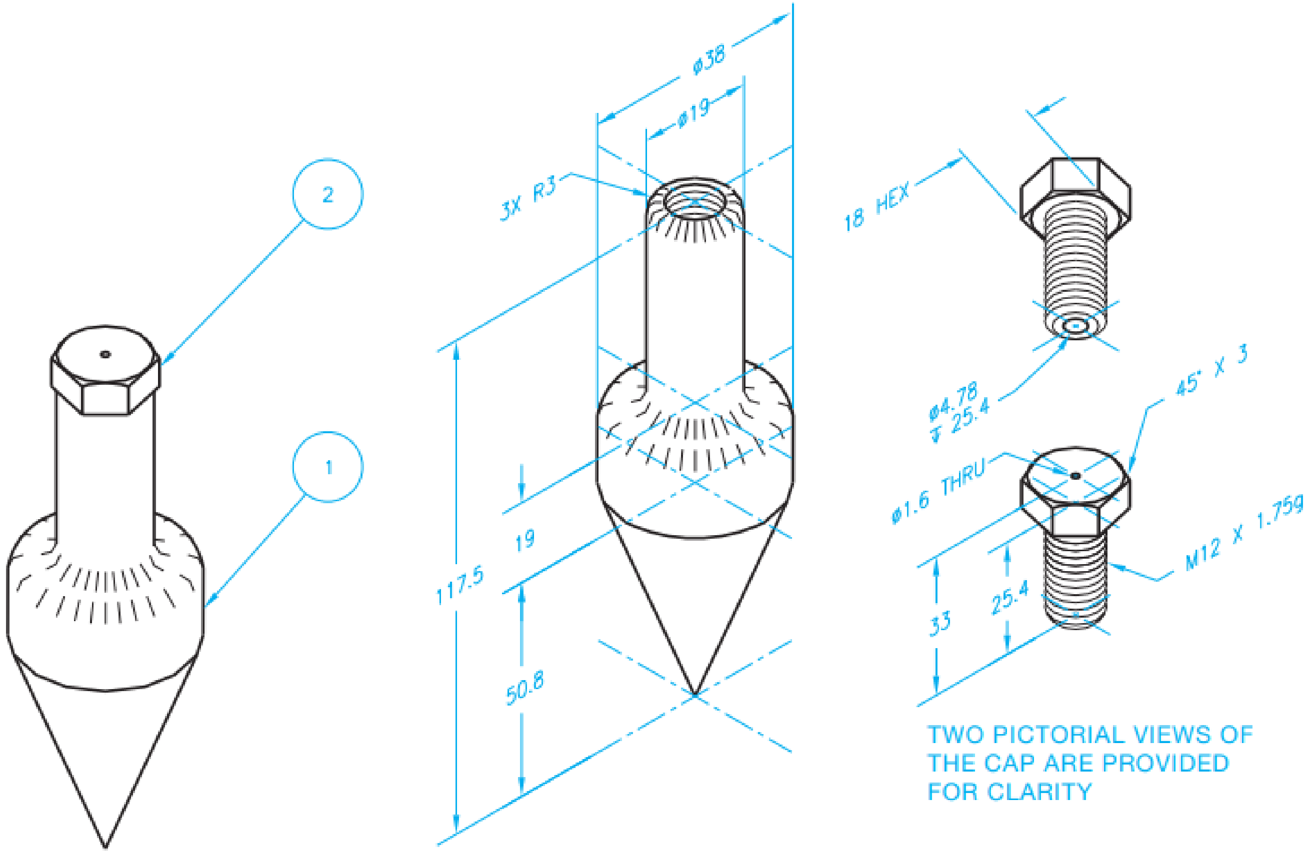
FIND NO	QTY REQD	PART OR IDENT NO	NOMENCLATURE OR DESCRIPTION	NOTES OR REMARKS
5	1	06533	O-RING	1.17 X 1.403 X .116 R17 - 916
4	2	380892	BACK-UP RING	129
3	2	350766	O-RING	1-9/16 X 1-3/4 X 3/32 R17 - 129
2	1	E6621	SWIVEL STEM	SAE 1045
1	1	E6620	SWIVEL	SAE 4340

PARTS LIST

APPROVALS		DATE	ENGINEERING DRAFTING & DESIGN, INC. Drafting, design, and training for all disciplines. Integrity - Quality - Style			
DRAWN	DPM		TITLE SWIVEL ASSEMBLY			
CHECKED	ADM					
APPROVED	HJE					
MATERIAL NOTED						
FINISH NOTED			SIZE B	CAGE CODE	DWG NO. E6622	REV 0
DO NOT SCALE DRAWING			SCALE 1:1		SHEET 1 OF 1	

EXERCISES

1. Working-drawing assembly (metric) Assembly Name: Plumb Bob SPECIFIC INSTRUCTIONS: Prepare a working-drawing assembly that has a detail drawing of each part, an assembly drawing, and a parts list on one sheet

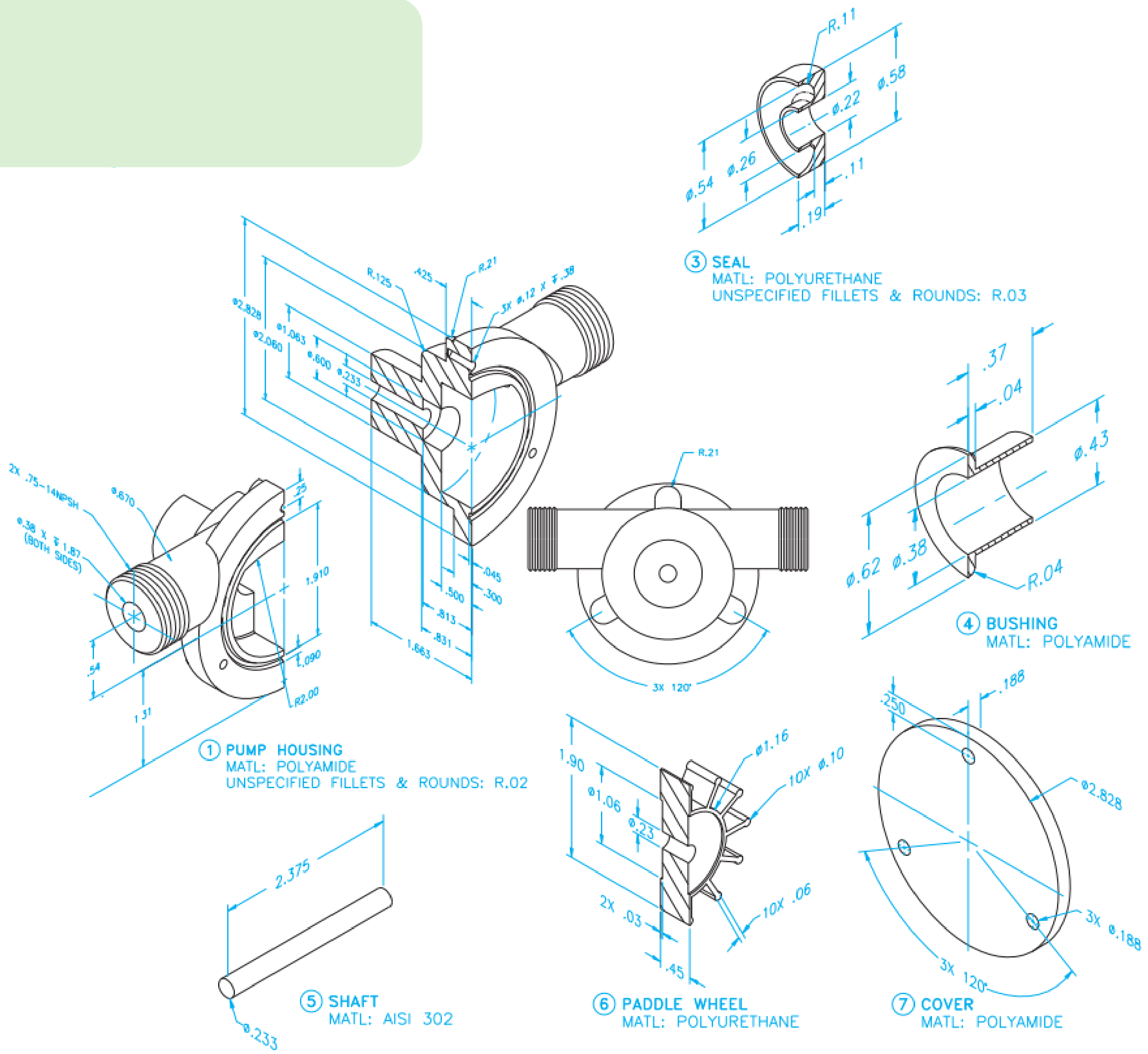


PARTS LIST			
ITEM	QTY	NAME	MATERIAL
1	1	PLUMB BOB	BRONZE
2	1	CAP	BRONZE

EXERCISES

2. Assembly Name: Drill Pump

SPECIFIC INSTRUCTIONS: Prepare the assembly drawing and bill of materials on the same sheet. Use multi-view projection for view layout.

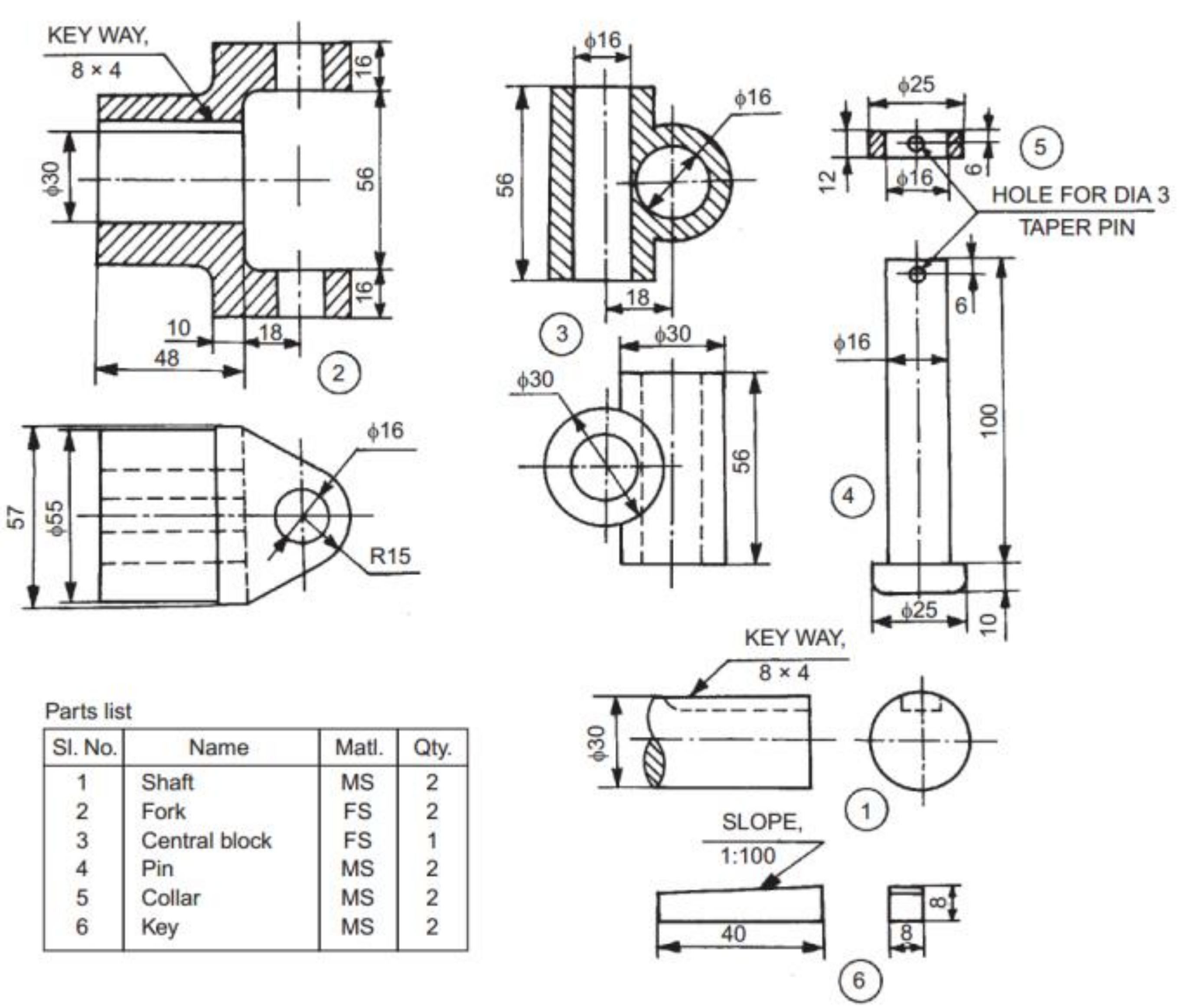


DRILL PUMP BILL OF MATERIALS		
ITEM	QTY	DESCRIPTION
1	1	PUMP HOUSING
2	1	.070 O-RING, Ø2.00
3	1	SEAL
4	1	PLASTIC BUSHING
5	1	SHAFT
6	1	PADDLE WHEEL
7	1	COVER
8	3	1/8" X 5/8" LONG PAN HEAD SCREW, THREAD CUTTING

EXERCISES

Universal Joint

This is a rigid coupling and is used to connect two shafts, whose axes intersect if extended. Figure 1 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 (not shown) is used to keep the pins (4) in position. During rotation of shafts, the angle between them can be varied.



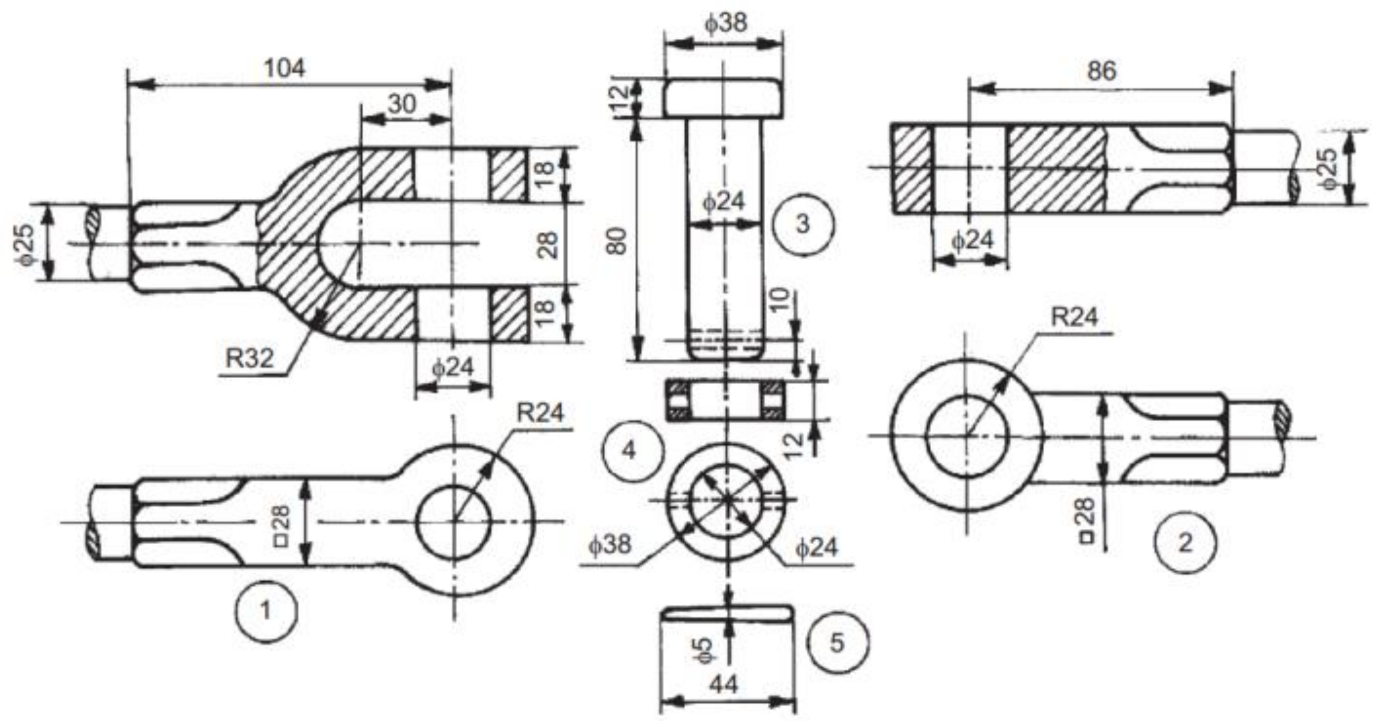
Parts list

Sl. No.	Name	Matl.	Qty.
1	Shaft	MS	2
2	Fork	FS	2
3	Central block	FS	1
4	Pin	MS	2
5	Collar	MS	2
6	Key	MS	2

EXERCISES

KNUCKLE JOINT

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 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.



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

EXERCISES

BOLTS AND NUTS

The bolt has an external thread which extends along only part of the shank. Bolts generally pass completely through the work to be fastened and on the other side are secured by a nut, which has an internal mating thread. Nuts and bolts are usually hexagonal-headed and are adjusted with a standard spanner of the open-ended, ring, or socket type. Nuts and bolts often have to be drawn by draftsmen, so it is very useful to learn a quick method to obtain an approximate shape.

Approximate Method of Drawing a Hexagonal Nut

1. Start with the plan view. Draw a circle of diameter 2d, where d is the major diameter of the thread (nominal size).

2. Using a 60^{O} set square, construct a hexagon inside the circle and then draw a chamfer circle inside the hexagon.

3. Complete the plan by drawing concentric circles representing the threaded hole.

4. Project the front and end views, making the height of the nut equal to 0.8d.

. From the points marked A in the front view, draw construction lines at 30^{O} to the main centre line to intersect this centre line at point B.

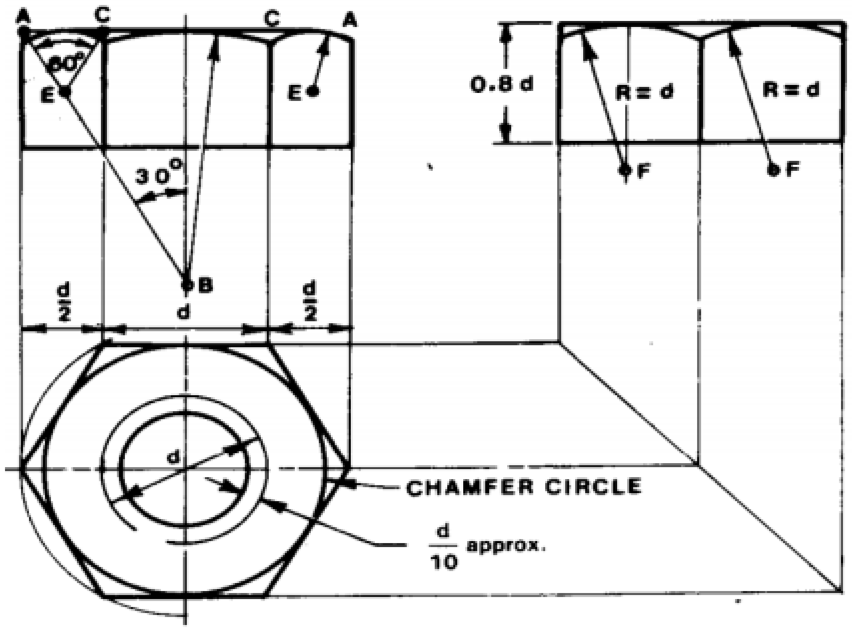
6. With the centre at point B, draw the chamfer curve tangential to the top surface of the nut.

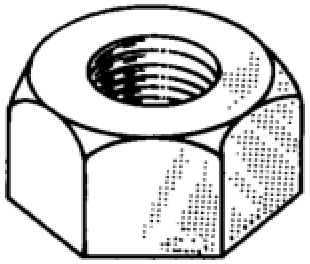
7. From the points marked C, draw construction lines at 30^{O} to the main centre line, intersecting the initial construction line at the points E.

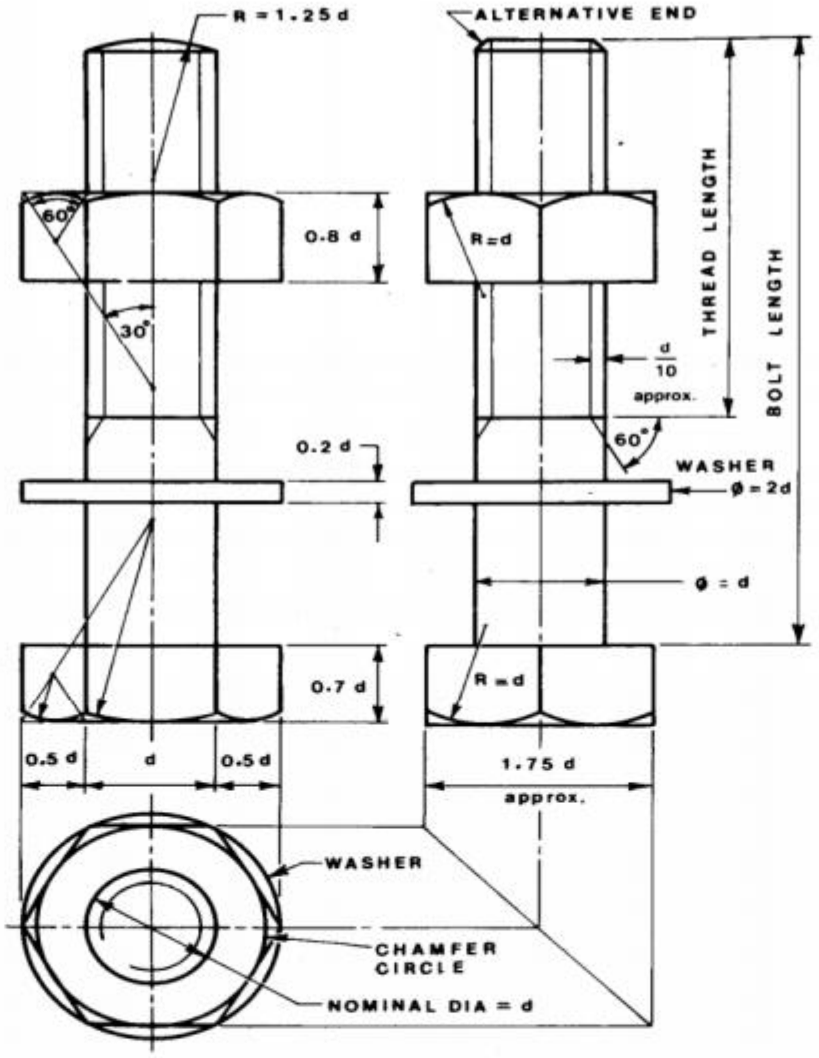
EXERCISES

8. With centres at points E, draw two small chamfer curves tangential to the top surface of the nut.

9. In the end view, bisect the distances between the main centre line and the two extreme sides. Starting at the top surface of the nut, mark along the bisectors a distance equal to d, giving the intersection points F.







Method for Drawing Hexagonal Bolt and Nut

PROJECTION OF SOLIDS

We will cover these skills:

- Classification of Solids.
- Orientation of Planes in Space.
- Position of Projection in Solids.
- Positions solids with respect to reference plane.
- Exercises.

PROJECTION OF SOLIDS - CLASSIFICATION

CLASSIFICATION OF SOLIDS

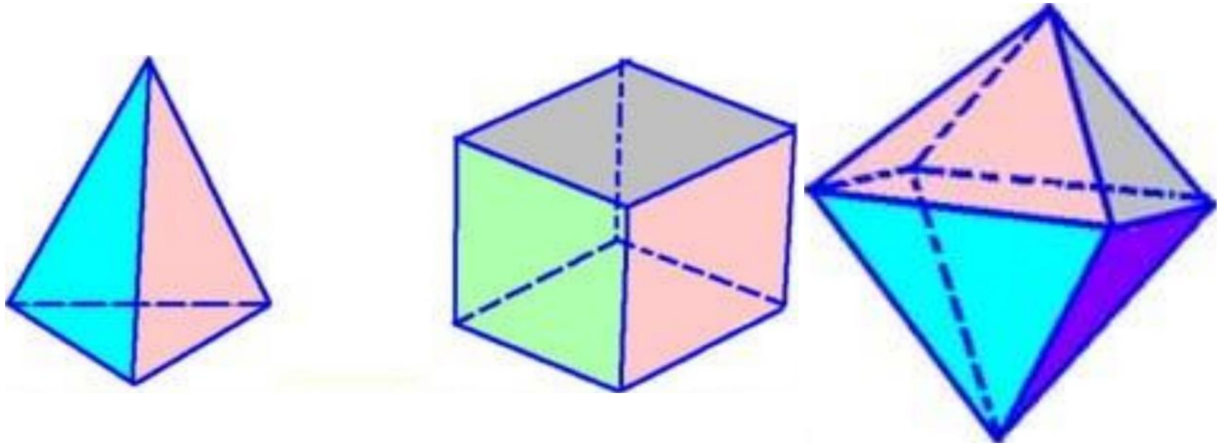
Solids are classified into two groups. They are

1. Polyhedra
2. Solids of Revolution

❖ **Polyhedra**- A solid, which is bounded by plane surfaces or faces, is called a polyhedron, which meet in straight lines called **EDGES**

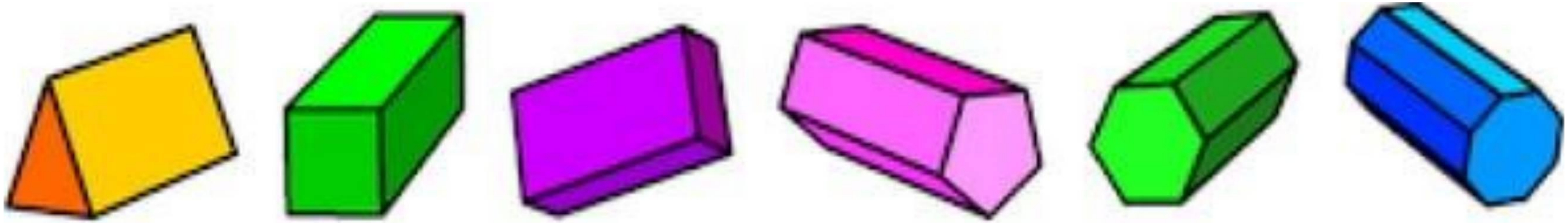
Polyhedra are classified into three sub groups, these are 1. Regular Polyhedra - The regular plane surfaces are called "Faces" and the lines connecting adjacent faces are called "edges".

Example . Tetrahedran, Hexahedran , Octahedran.



PROJECTION OF SOLIDS - CLASSIFICATION

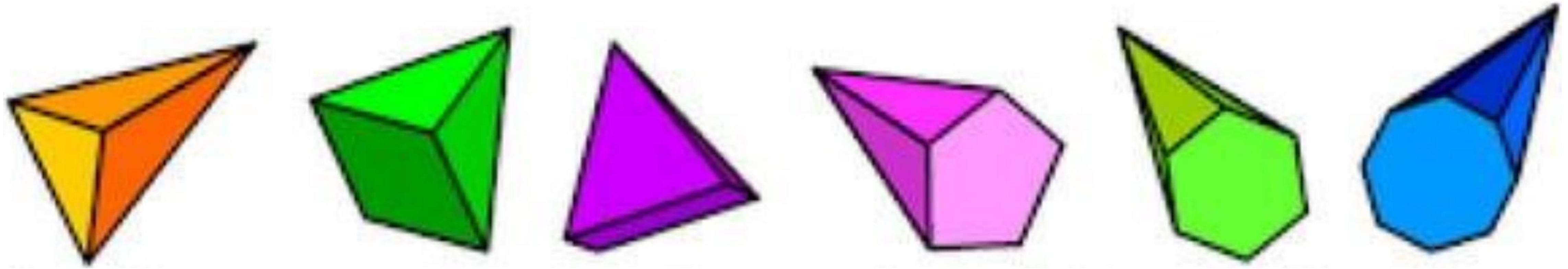
- **Prisms** - A prism has two equal and similar end faces called the top face and the bottom face or (base) joined by the other faces, which may be rectangles or parallelograms.



Triangular Prism | Square Prism | Rectangular Prism | Pentagonal Prism | Hexagonal Prism | Octagonal Prism

PROJECTION OF SOLIDS - CLASSIFICATION

- **Pyramids** - A pyramid has a plane figure as at its base and an equal number of isosceles triangular faces that meet at a common point called the "vertex" or "apex". The line joining the apex and a corner of its base is called **the slant edge**.



Triangular Pyramid

Square Pyramid

Rectangular Pyramid

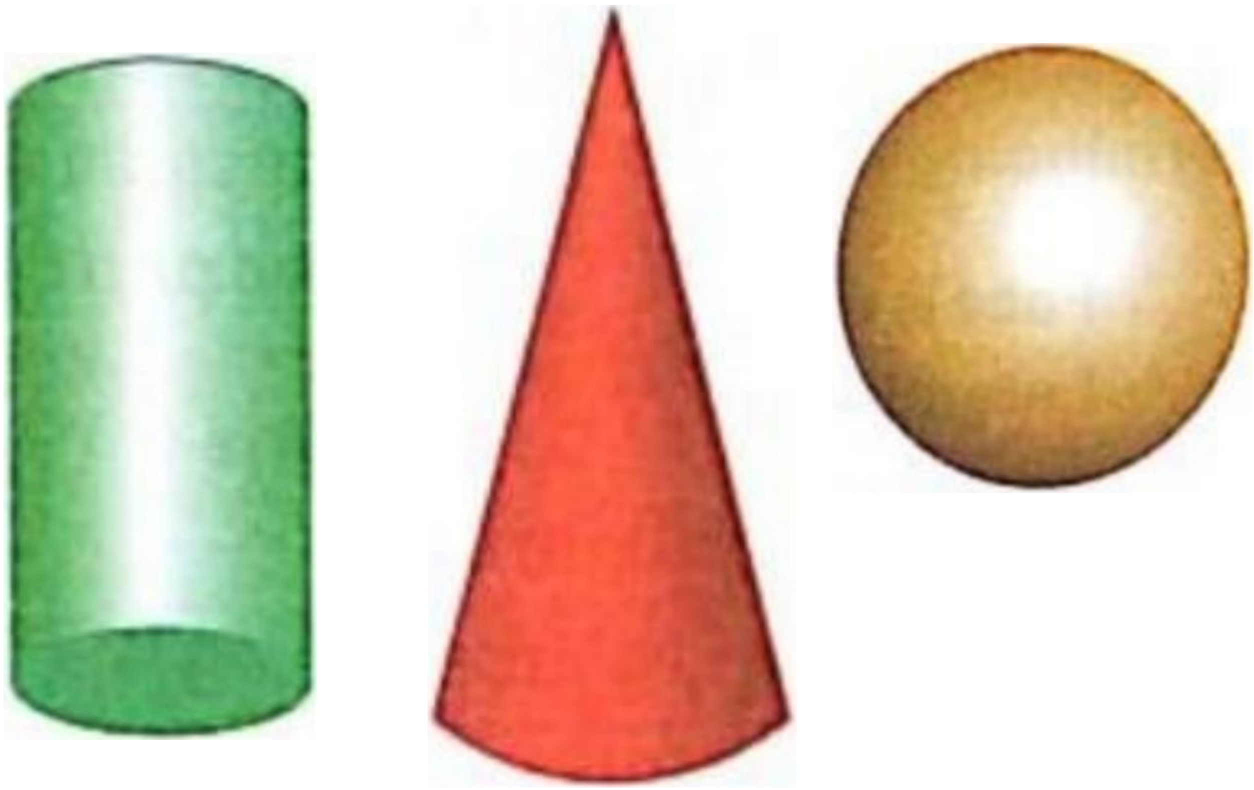
Pentagonal Pyramid

Hexagonal Pyramid

Octagonal Pyramid

PROJECTION OF SOLIDS - CLASSIFICATION

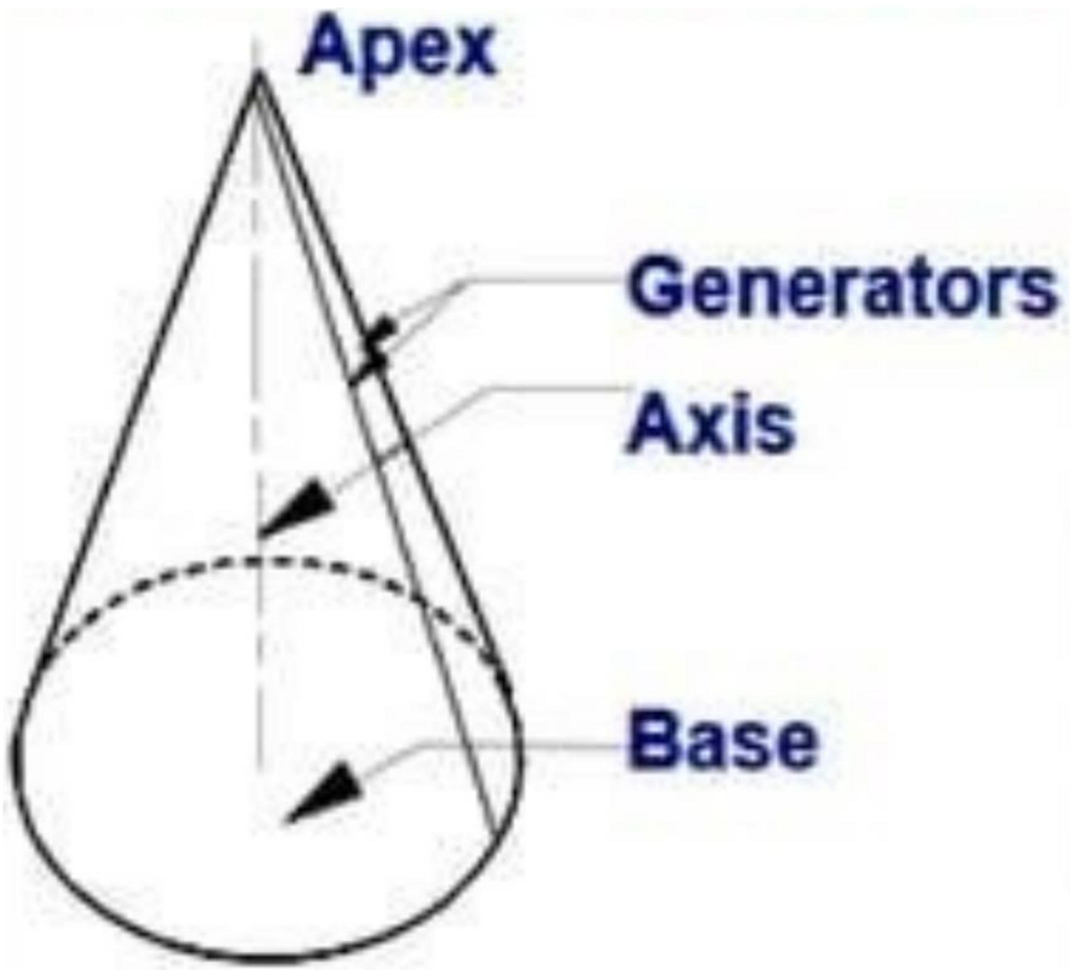
- **Solid of Revolution -** If a plane surface is revolved about one of its edges, the solid generated is called a Solid of Revolution. Eg. Cylinder, Cone and Sphere.



PROJECTION OF SOLIDS - CLASSIFICATION

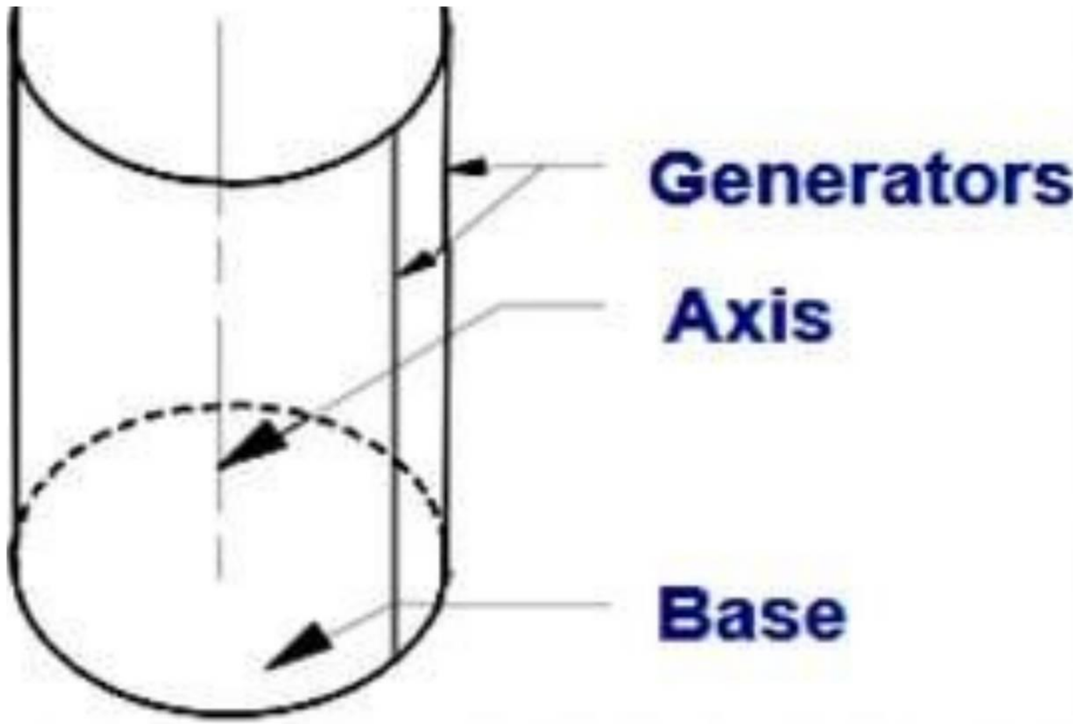
- **Cone-** A cone can be generated by the revolution of a right-angled triangle about one of its perpendicular sides, which remains fixed.

A cone has a circular base and an apex. The line joining apex and the center of the base is called the "Axis" of the cone.



PROJECTION OF SOLIDS - CLASSIFICATION

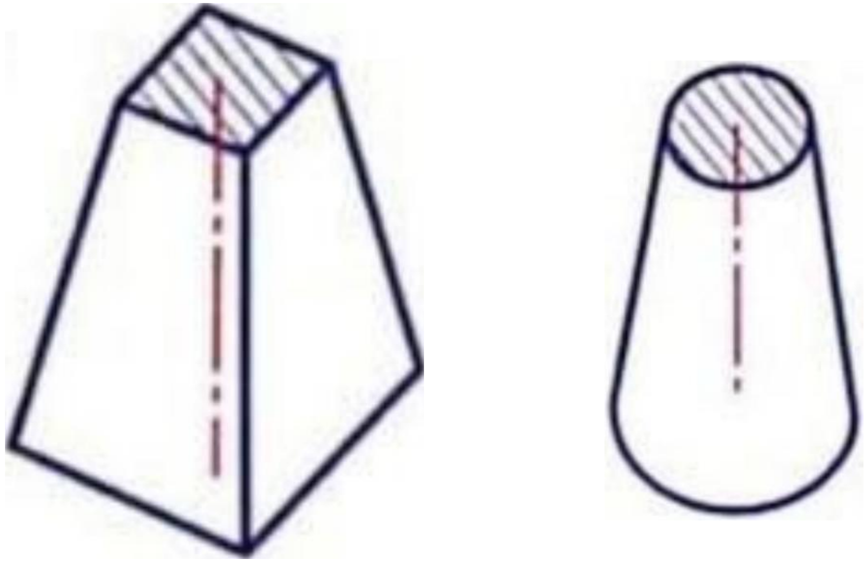
- **Sphere** – A sphere can be generated by the revolution of a semi-circle about its diameter that remains fixed.
- **Cylinder** – A right circular cylinder is a solid generated by the revolution of a rectangular surface about one of its sides, which remains fixed. It has two circular faces. The line joining the centers of the top and the bottom faces is called "Axis".



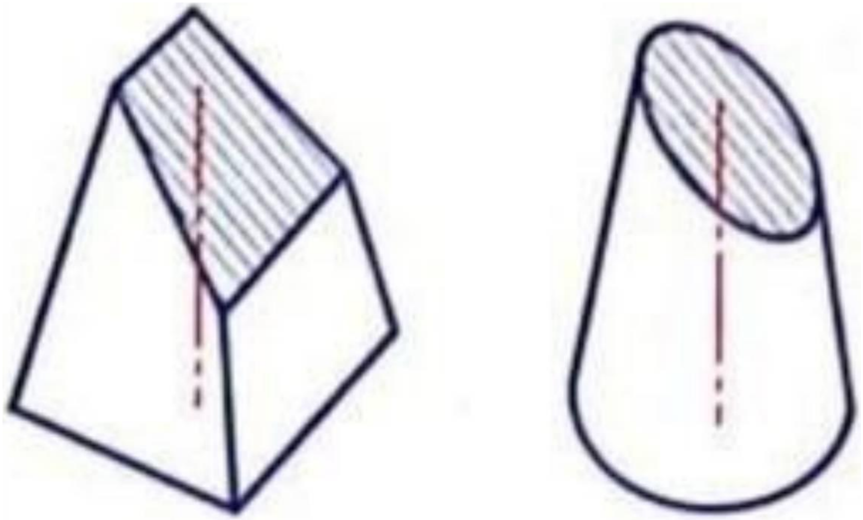
PROJECTION OF SOLIDS - CLASSIFICATION

❖ FRUSTUMS AND TRUNCATED SOLIDS

When a solid is cut by a plane parallel to its base, thus removing the top portion, the remaining lower portion is called **frustum**. When a solid is cut by a plane inclined to its base, thus removing the top cut portion, the remaining lower portion of the solid is called **truncated**.



Frustum of solids



Truncated Solids

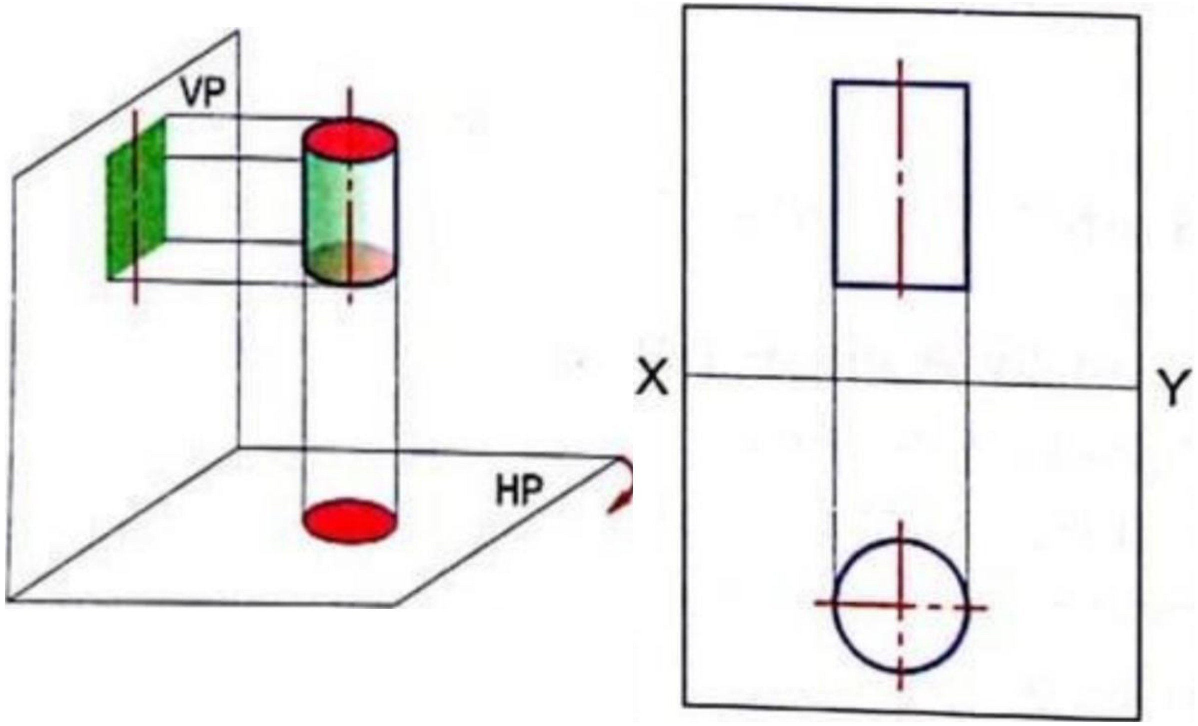
ORIENTATION OF PLANES IN SPACE

The following position of Planes in space

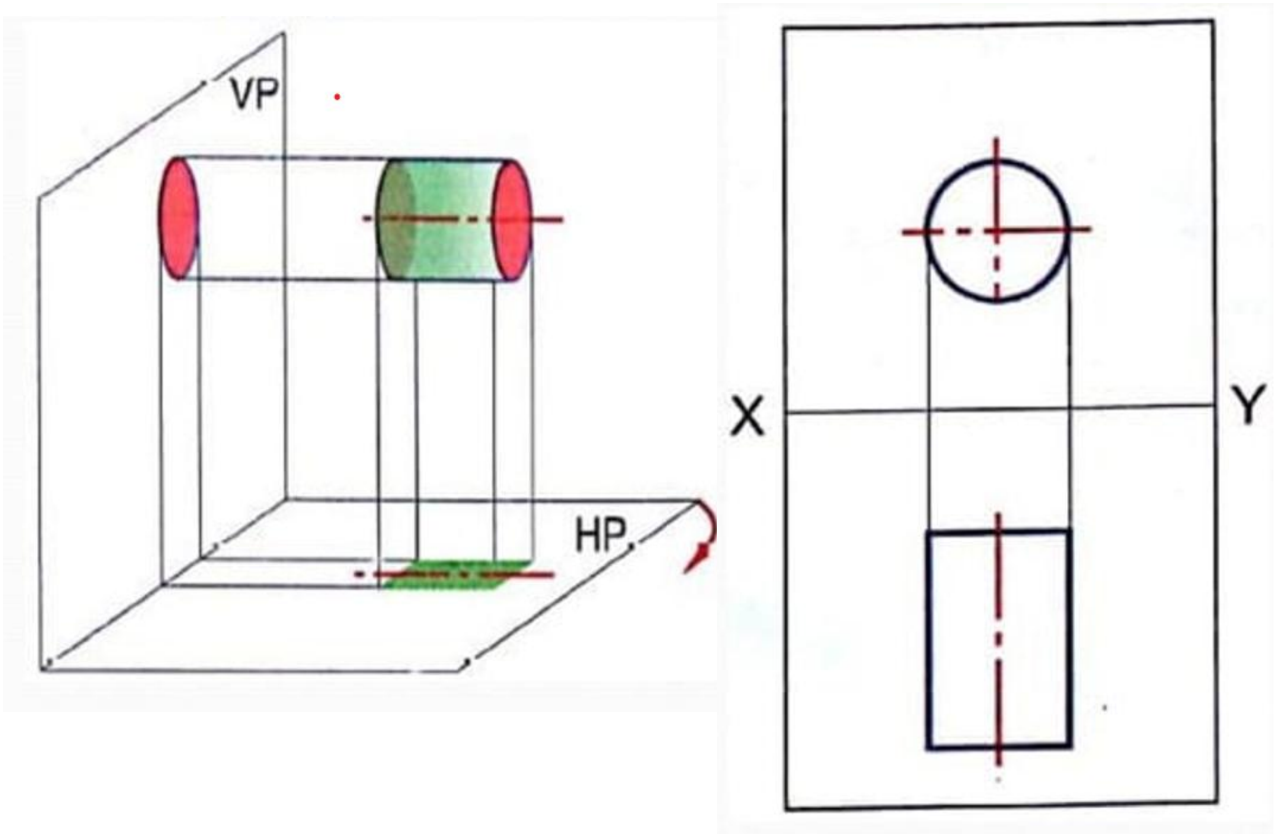
- ❖ Axis Parallel to VP and Perpendicular to HP
- ❖ Axis Perpendicular to VP and Parallel to HP
- ❖ Axis Parallel to both VP & HP or both Perpendicular VP&HP
- ❖ Axis Perpendicular to VP and Inclined to HP
- ❖ Axis Inclined to VP and Perpendicular to HP
- ❖ Axis Inclined to both VP and HP

POSITION IN PROJECTION OF SOLIDS

1. Axis perpendicular to HP &parallel to VP

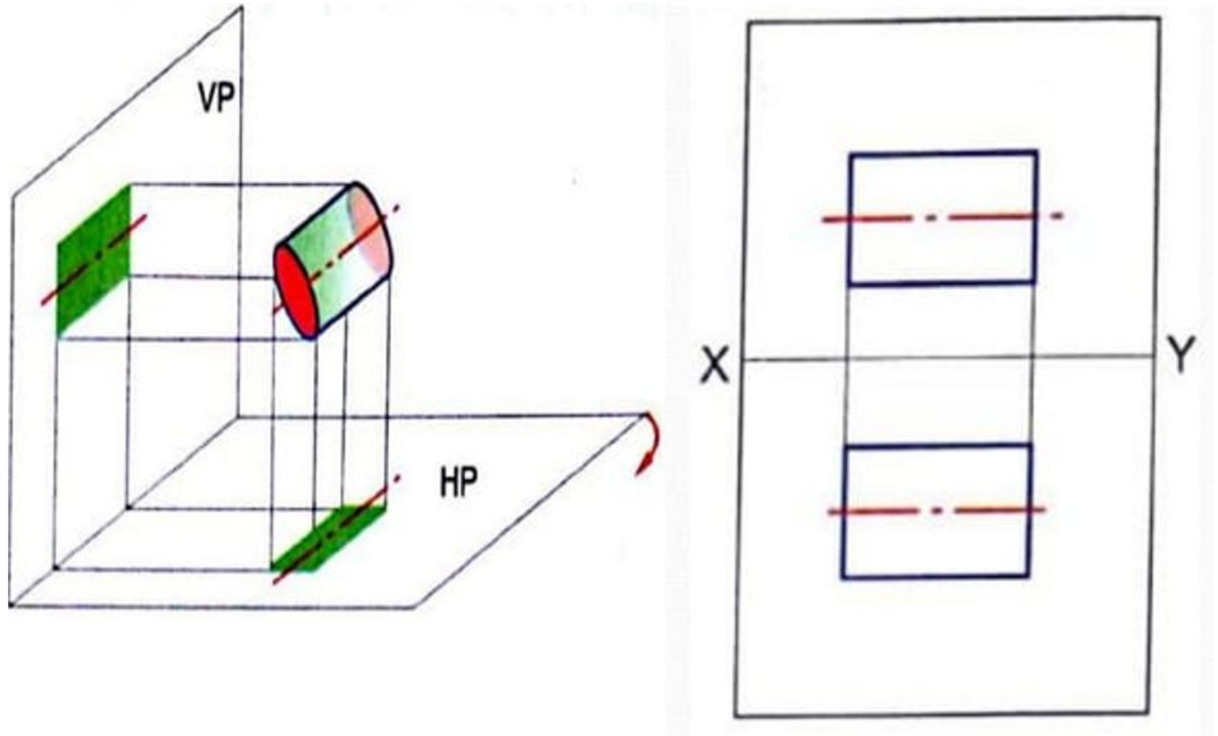


2. Axis perpendicular to VP & parallel to HP

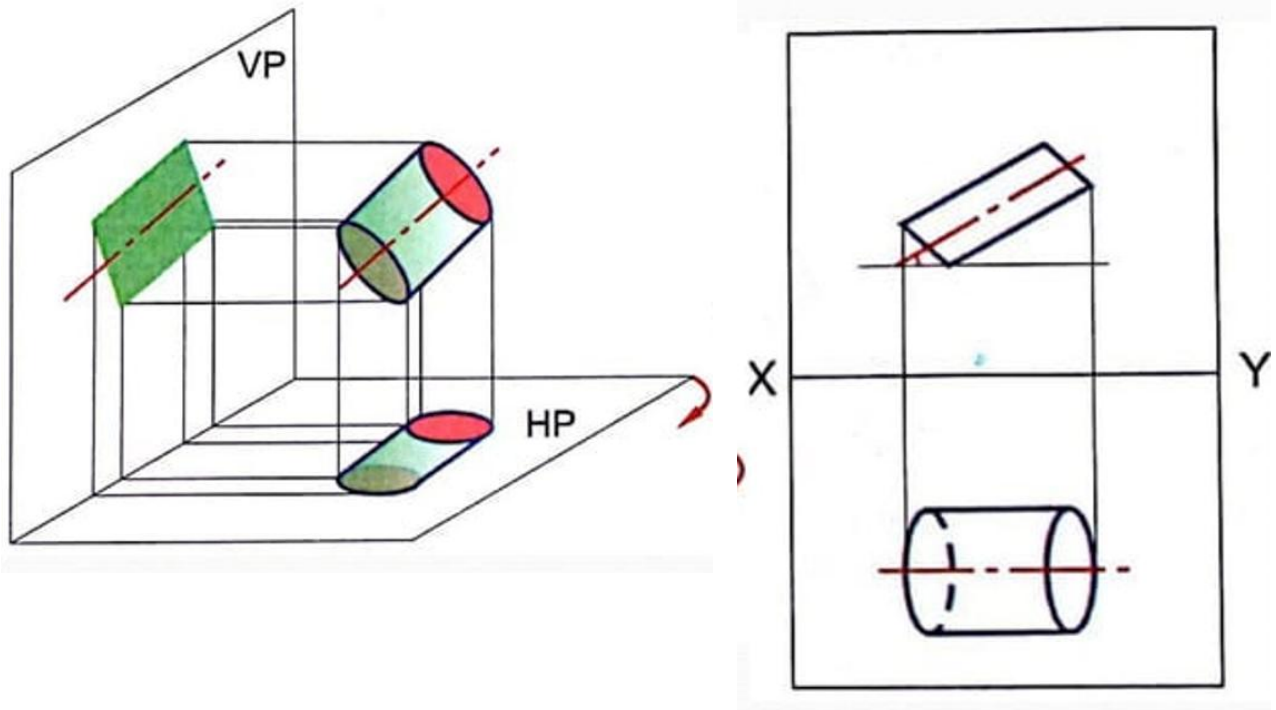


POSITION IN PROJECTION OF SOLIDS

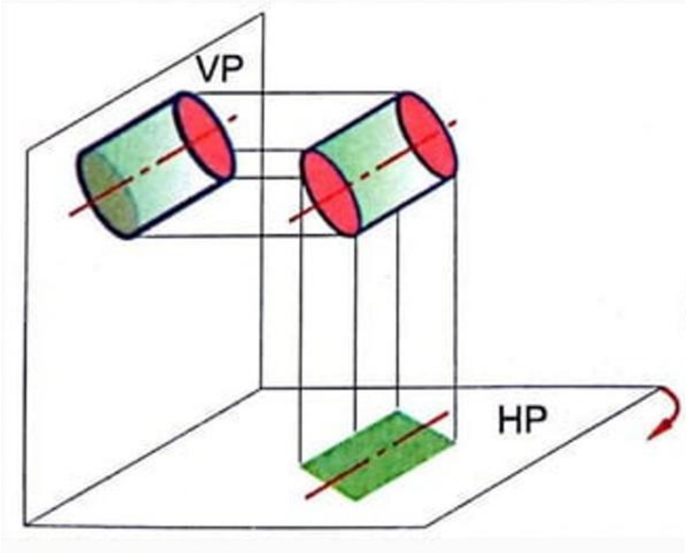
3. Axis parallel to both VP and HP

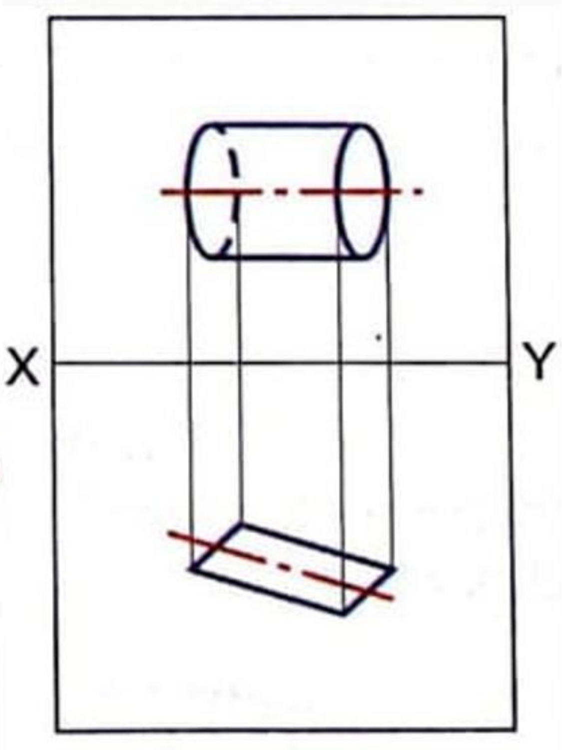


4. Axis inclined to both HP & perpendicular to VP

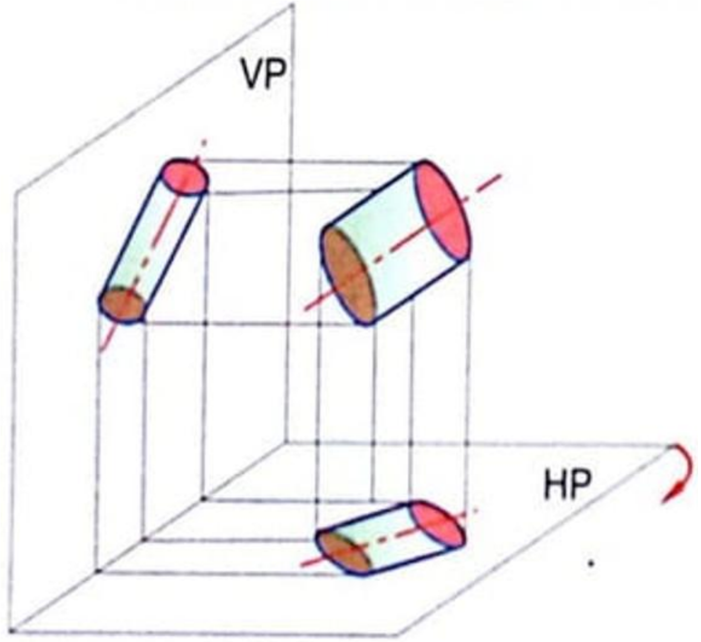


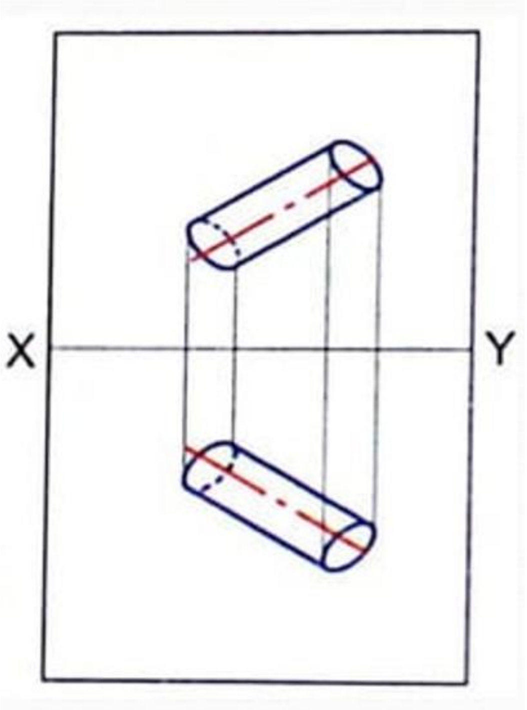
5. Axis inclined to VP and perpendicular to HP





6. Axis inclined to both VP and HP





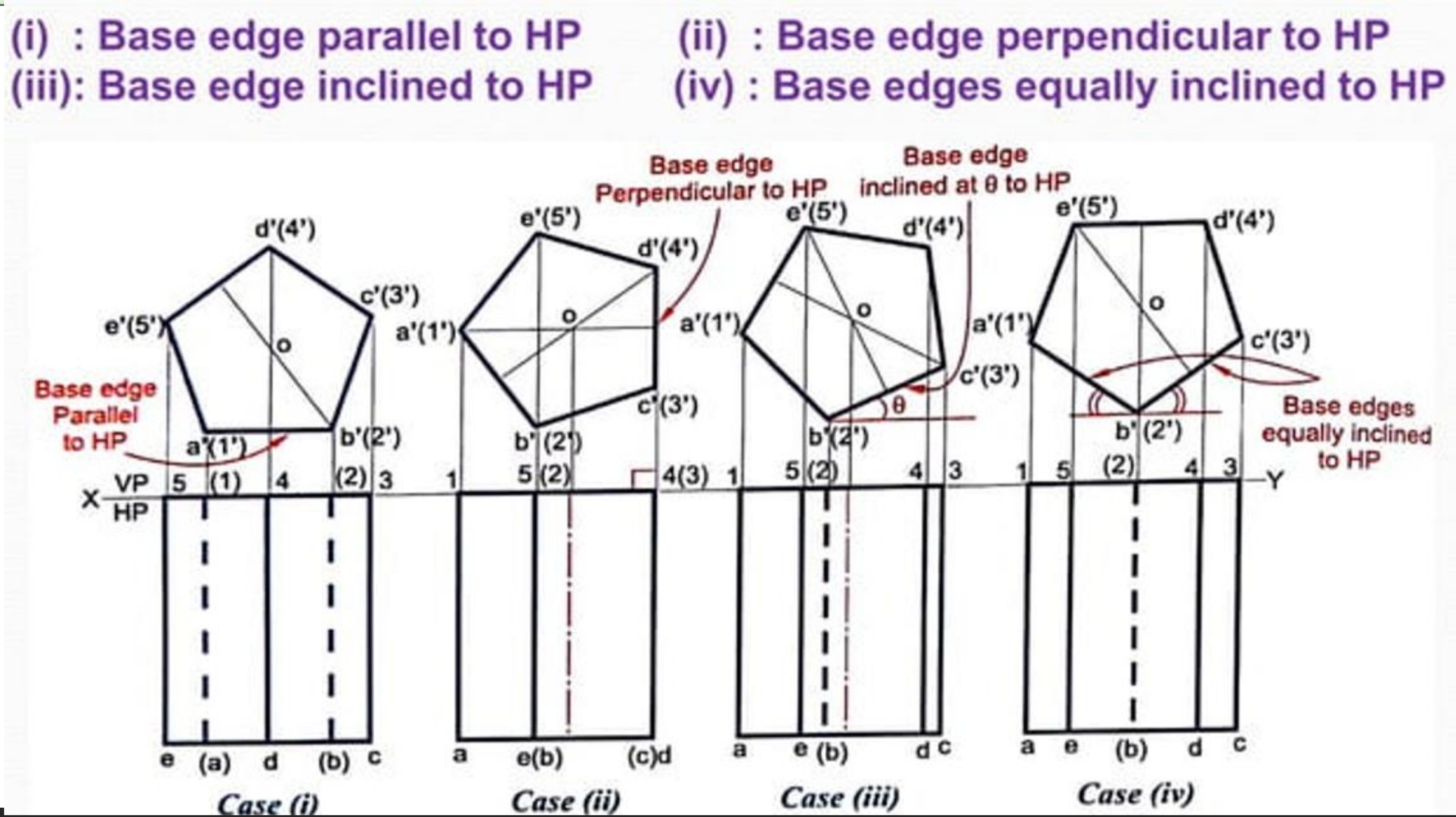
POSITION IN PROJECTION OF SOLIDS

Positions of Solids with respect to reference plane

S.No	Positions of solids	Step -1	Step -2	Step -3
1	Axis of the solid perpendicular to HP and parallel to VP	Draw plan first	Draw elevation next	--
2	Axis of the solid perpendicular to VP and parallel to HP	Draw elevation First	Draw plan next	---
3	Axis parallel to both VP and HP	Side view	Elevation	Plan
4	Axis of the solid inclined to VP and parallel to HP	Draw elevation axis perpendicular to VP	Tilt the plan	Get final elevation
5	Axis of the solid inclined to HP and parallel to VP	Draw plan axis perpendicular to HP	Tilt the elevation	Get final elevation
6	Axis of the solid inclined to both HP and VP	Draw plan , edge perpendicular to VP	Tilt the elevation get plan	Tilt the plan get elevation

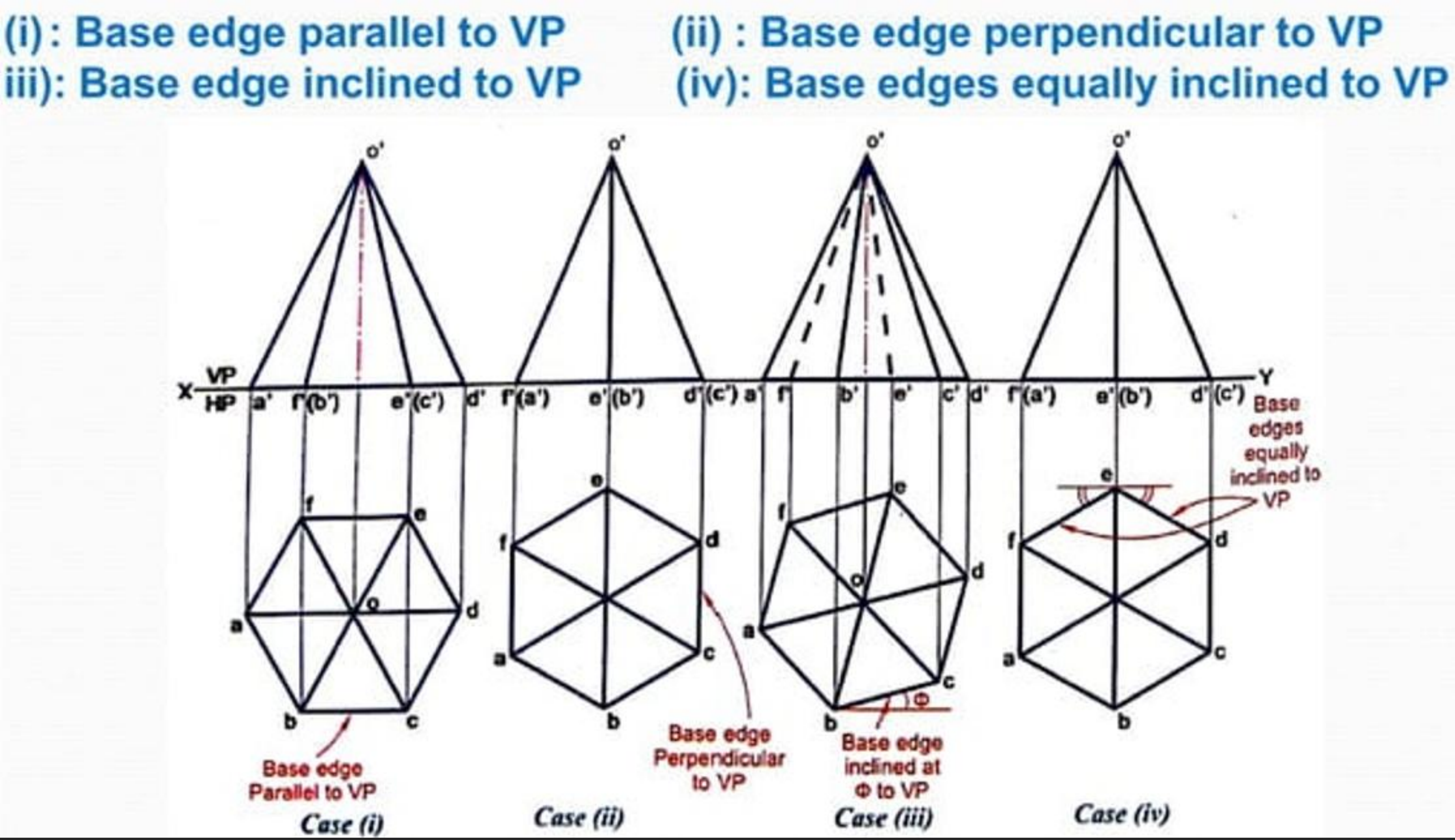
POSITION IN PROJECTION OF SOLIDS

Axis of solid perpendicular to VP & parallel to HP



POSITION IN PROJECTION OF SOLIDS

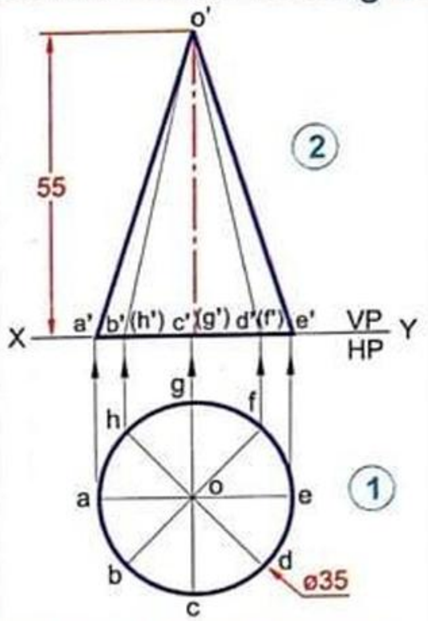
Axis of solid perpendicular to HP & parallel to VP



EXERCISES

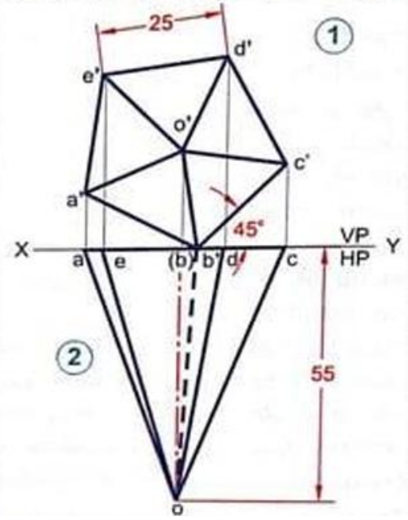
Axis is perpendicular to HP and parallel to VP

2.Draw the projections of a right circular cone of the base 35 mm diameter and height 55 mm when resting with its base on the HP.



Axis perpendicular to VP and parallel to HP

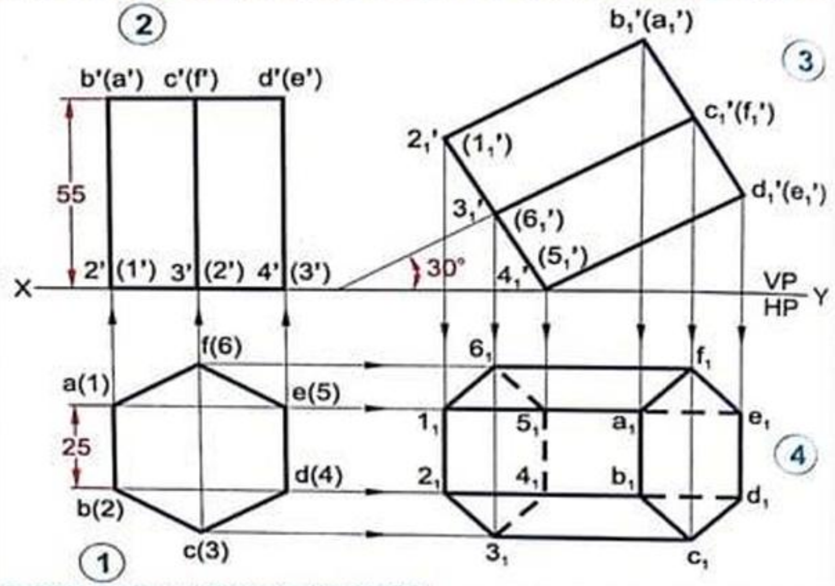
6.A pentagonal pyramid of base side 25 mm and axis 55 mm rests on the HP with a corner of the base, such that one of the base edge containing the corner makes **45⁰** with the HP. draw the projections when the axis perpendicular to the VP and the base is touching the VP.



EXERCISES

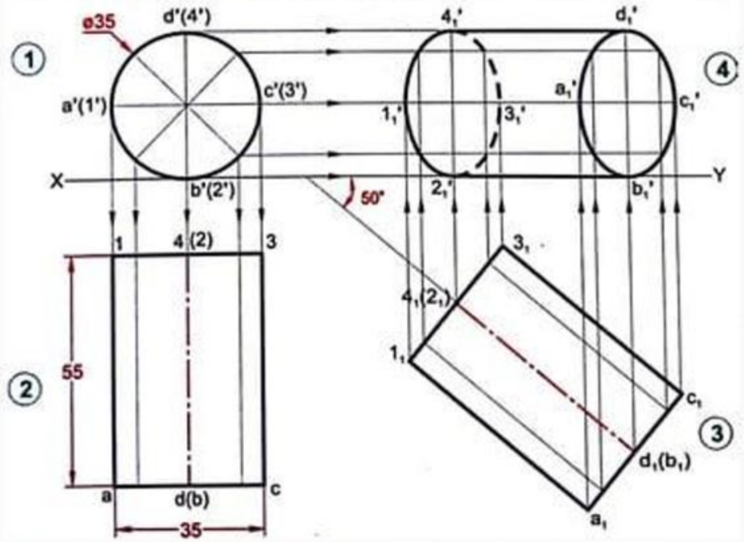
Axis parallel to VP and inclined to HP

9. A hexagonal prism of base side 25 mm and axis height 55 mm resting on HP with one of its base edges, such that the axis is inclined at **30⁰** to the HP and parallel to VP. Draw the projections of the prism.



Axis parallel to HP and inclined to VP

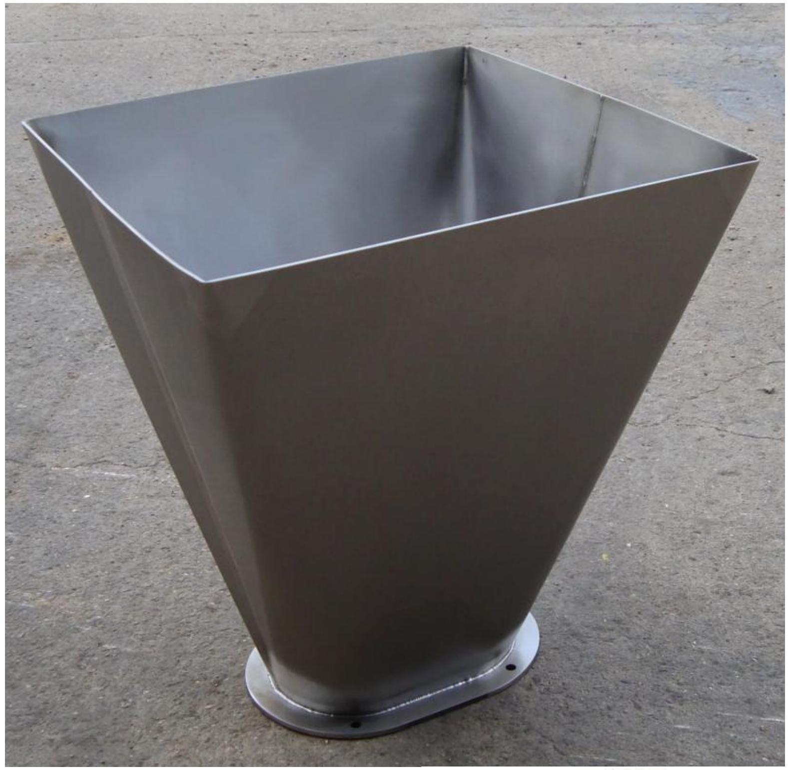
13. Draw the projections of a cylinder of diameter 35 mm and axis 55 mm long is resting on HP on one of its generators with its axis inclined at 5**0⁰** to VP. Draw its projections



DEVELOPMENT OF SOLIDS/SURFACES

We will cover these skills:

- Introduction to Development
- Developable surfaces.
- Methods of Development.
- Development of a Prism.
- Development of a Cylinder.
- Development of a Square Pyramid.
- Development of a Pentagonal Pyramid.
- Development of a Cone.
- Development of a truncated Pentagonal Prism.
- Development of a truncated Hexagonal Prism.
- Development of a frustum of a square Pyramid.
- Exercises.



DEVELOPMENT OF SOLIDS/SURFACES

DEVELOPMENT

A **development** is a flat representation or pattern that when folded together creates a 3D object (Figure 50). An intersection is the result of two objects that intersect each other (Figure 51). Sheet metal construction is the most common application for developments and intersections. A development of surfaces, such as those found in sheet metal fabrication, is a flat pattern that represents the unfolded or unrolled surface of the form. The resulting flat pattern gives the true size of each connected area of the form so that the part or structure can be fabricated. Auxiliary views are a primary tool for creating developments.

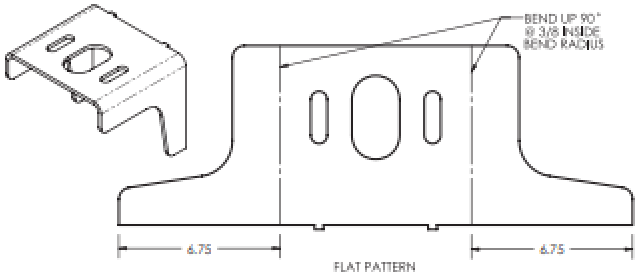


Figure 50: A Flat Pattern

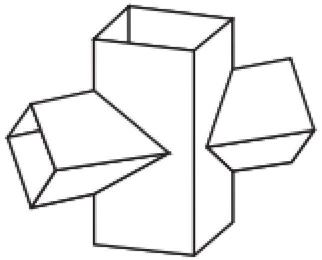


Figure 51: Intersecting Prisms

DEVELOPMENT OF SOLIDS /SURFACES

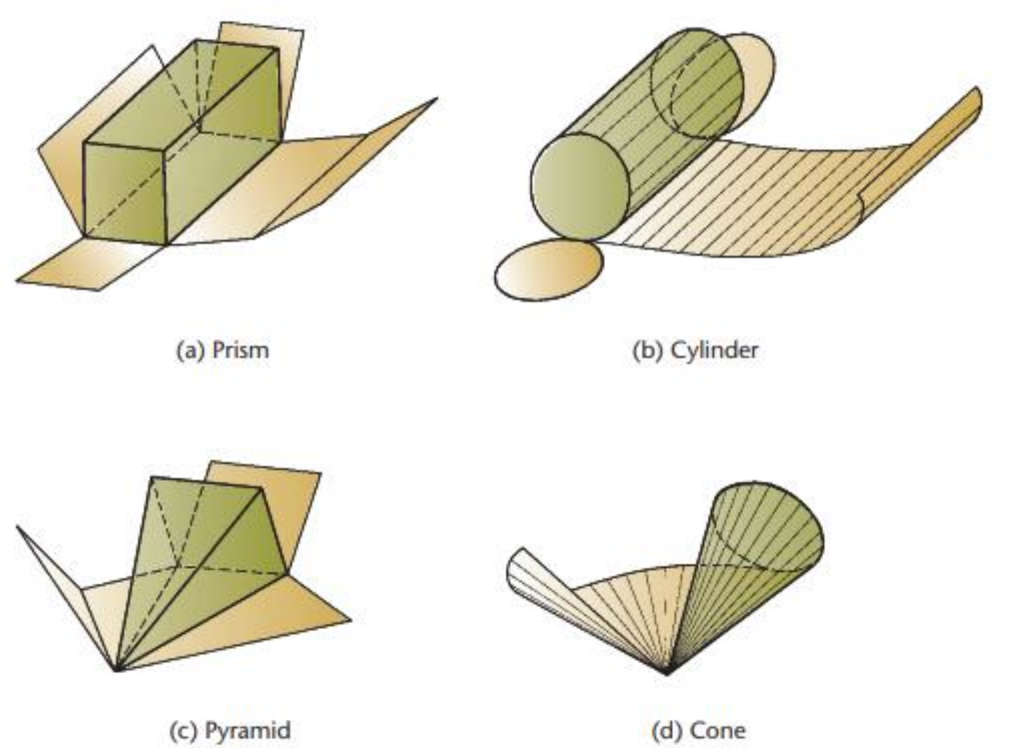


Figure 52: Development of Surfaces.

DEVELOPABLE SURFACES

A **developable surface** may be unfolded or unrolled to lie flat. Surfaces composed of single-curved surfaces, of planes, or of combinations of these types are developable. Warped surfaces and double-curved surfaces are not directly developable. They may be developed by approximating their shape using developable surfaces. If the material used in the actual manufacturing is sufficiently pliable, the flat sheets may be stretched, pressed, stamped, spun, or otherwise forced to assume the desired shape.

Nondevelopable surfaces are often produced by a combination of developable surfaces that are then formed slightly to produce the required shape. Figure 52 shows examples of developable surfaces.

DEVELOPMENT OF SOLIDS/SURFACES

METHODS OF DEVELOPMENT

- Parallel-line development.

It is used for developing prisms and single curved surfaces like cylinders in which all the edges/generators of lateral surfaces are parallel to each other.

- Radial-line development .

It is employed for pyramids and single curved surfaces like cones in which the apex is taken as centre and the slant edge or generator (which are the true lengths)as radius for its development.

- Triangulation development.

For developing transition pieces.

- Approximate development.

For developing spheres

SURFACE TERMINOLOGY

The following terminology describes objects and concepts used in developments and intersections:

A **ruled surface** is one that may be generated by sweeping a straight line, called the **generatrix**, along a path, which may be straight or curved (Figure 53). Any position of the generatrix is an element of the surface. A ruled surface may be a plane, a single-curved surface, or a warped surface.

- A **plane** is a ruled surface that is generated by a line, one point of which moves along a straight path while the generatrix remains parallel to its original position. Many geometric solids are bounded by plane surfaces (Figure 54).
- A single-curved surface is a developable ruled surface; that is, it can be unrolled to coincide with a plane. Any two adjacent positions of the generatrix lie in the same plane. Examples are the cylinder (Figure 55) and the cone.

DEVELOPMENT OF SOLIDS/SURFACES

A **double-curved surface** is generated by a curved line and has no straight-line elements (Figure 56). A surface generated by revolving a curved line about a straight line in the plane of the curve is **called a double-curved surface of revolution**. Common examples are the **sphere, torus, ellipsoid, and hyperboloid.**

- A warped surface is a ruled surface that is not developable. Some examples are shown in Figure 57. No two adjacent positions of the generatrix lie in a flat plane. Warped surfaces cannot be unrolled or unfolded to lie flat. Many exterior surfaces on an airplane or automobile are warped surfaces.

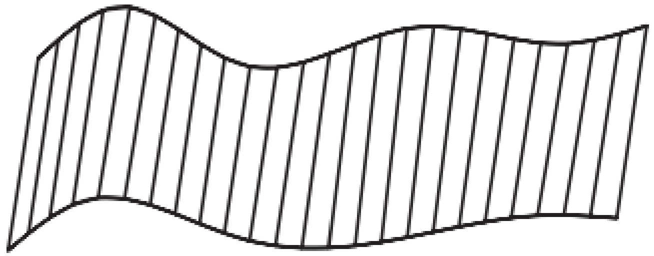


Figure 53: Ruled Surface

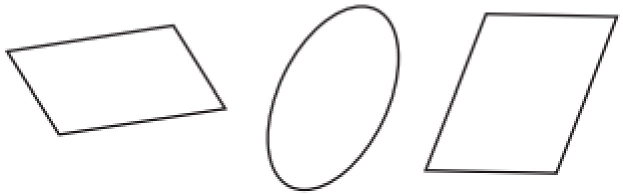


Figure 54: Plane surfaces

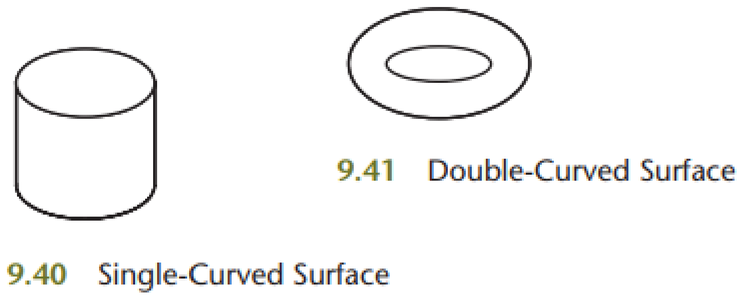
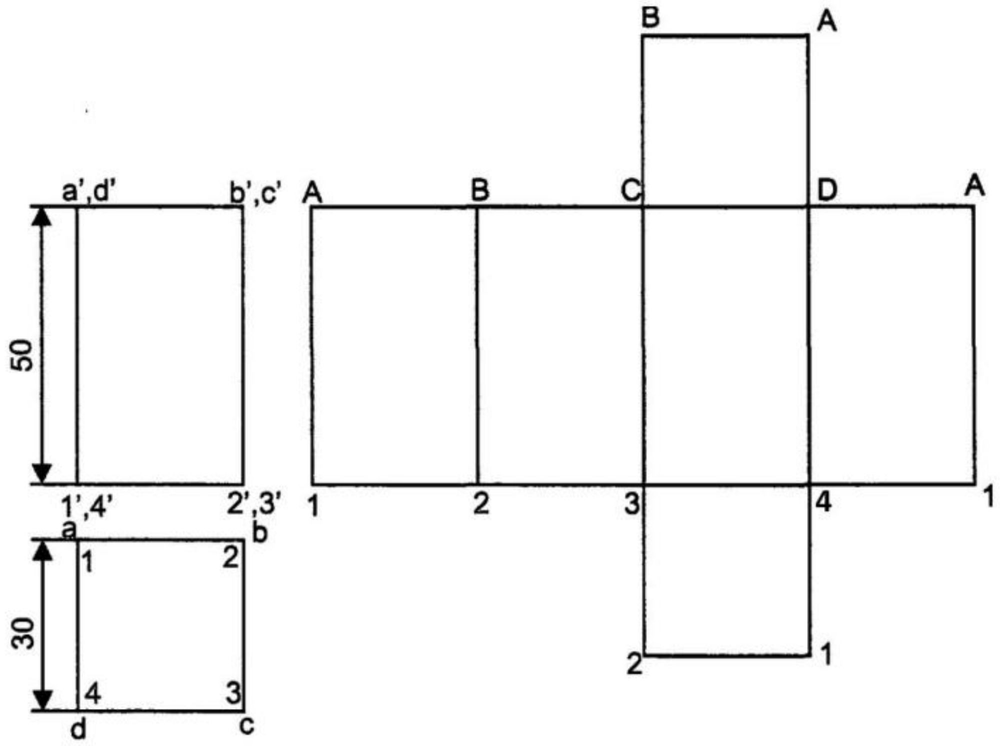


Figure 55: Curved surfaces

DEVELOPMENT OF SOLIDS /SURFACES

DEVELOPMENT OF A PRISM

To draw the development of a square prism of side of base 30 mm and height 50 mm.

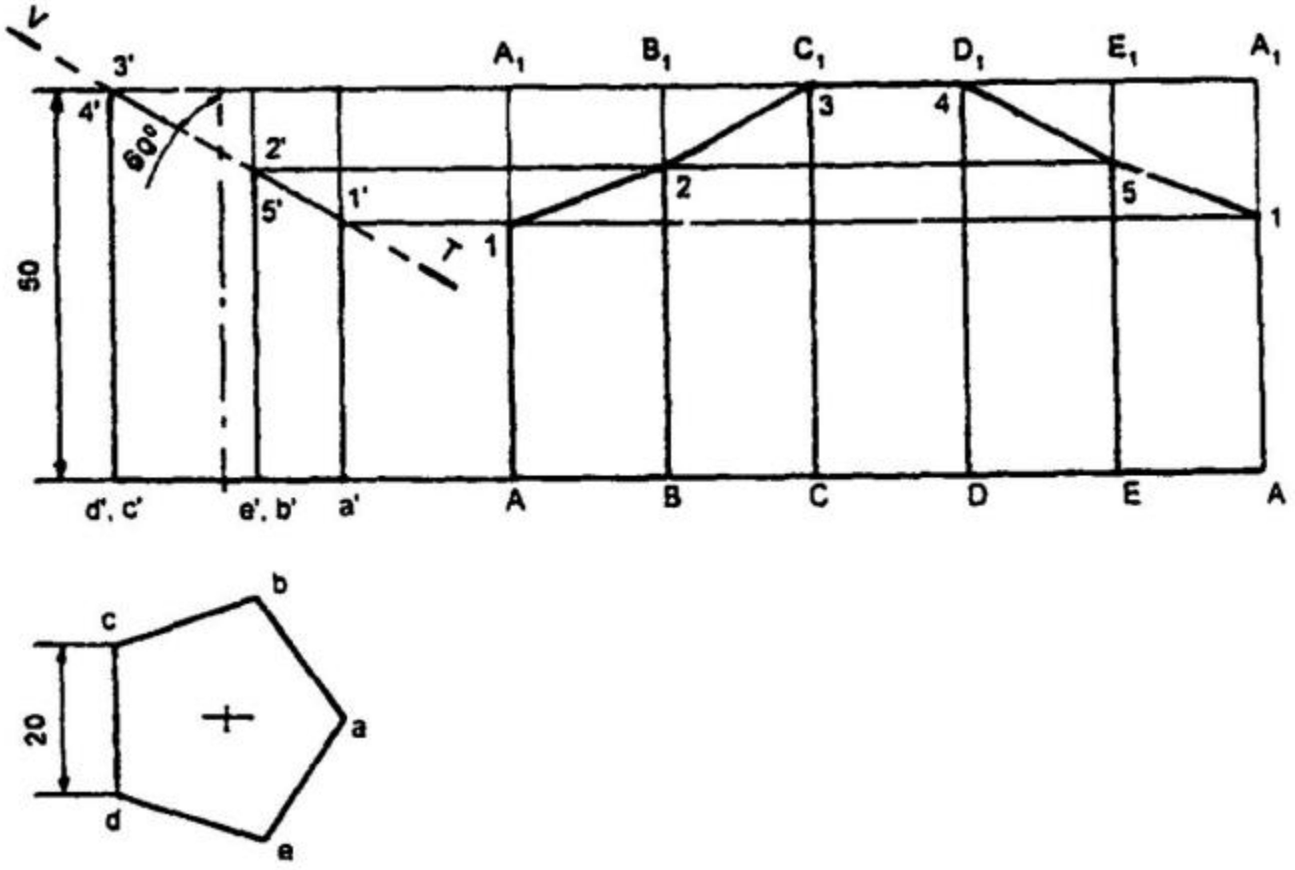


INSTRUCTIONS

1. Draw the stretch-out line 1-1 (equal in length to the circumference of the square prism) and mark off the sides of the base along this line in succession, that is 1-2, 2-3, 3-4 and 4-1. 3.
2. Erect perpendiculars through 1,2,3 etc., and mark the edges (folding lines) 1-A, 2-B, etc., equal to the height of the prism (50 mm).
3. Add the bottom and top bases 1234 and ABCD by the side of any of the base edges.

DEVELOPMENT OF A TRUNCATED PENTAGONAL PRISM

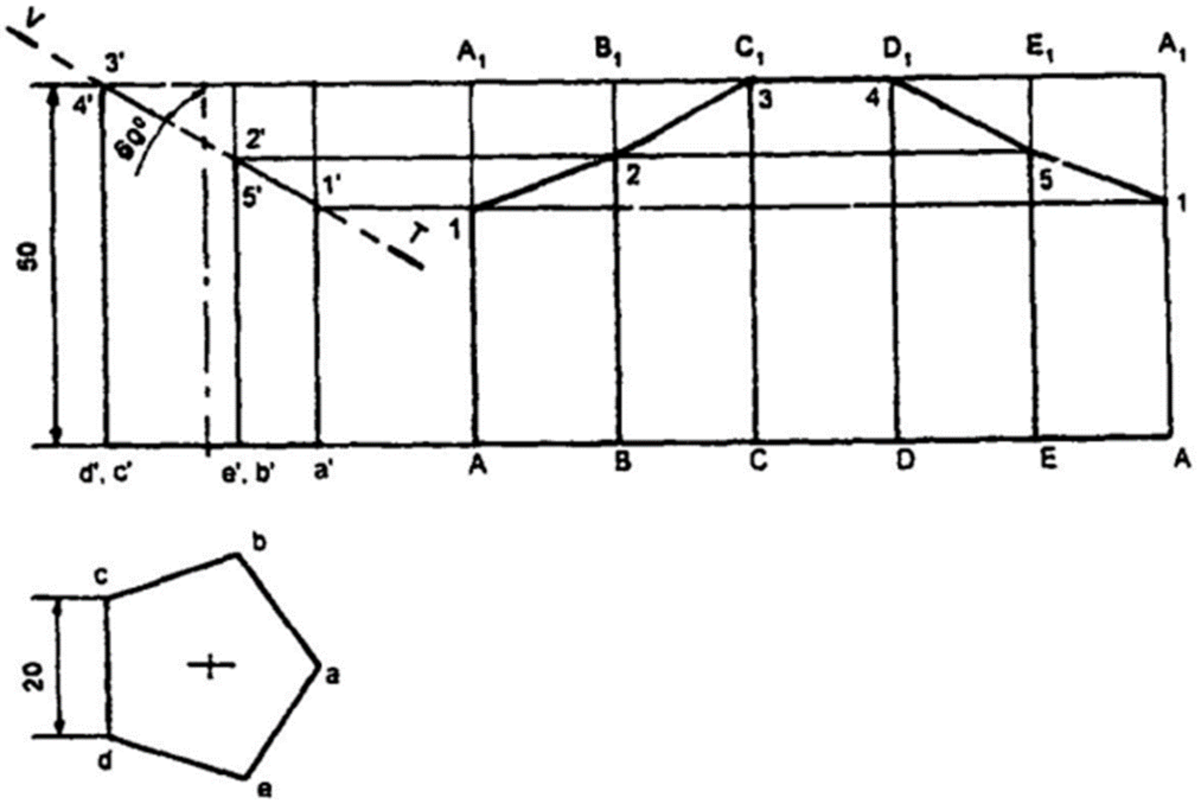
A Pentagonal prism of the side of base 20 mm and height 50 mm stands vertically on its base with a rectangular face perpendicular to V.P. A cutting plane perpendicular to V.P and inclined at 60^o from the axis and passes through the edges of the top base of the prism. Develop the lower portion of the lateral surface of the prism.



DEVELOPMENT OF SOLIDS/SURFACES

Instructions

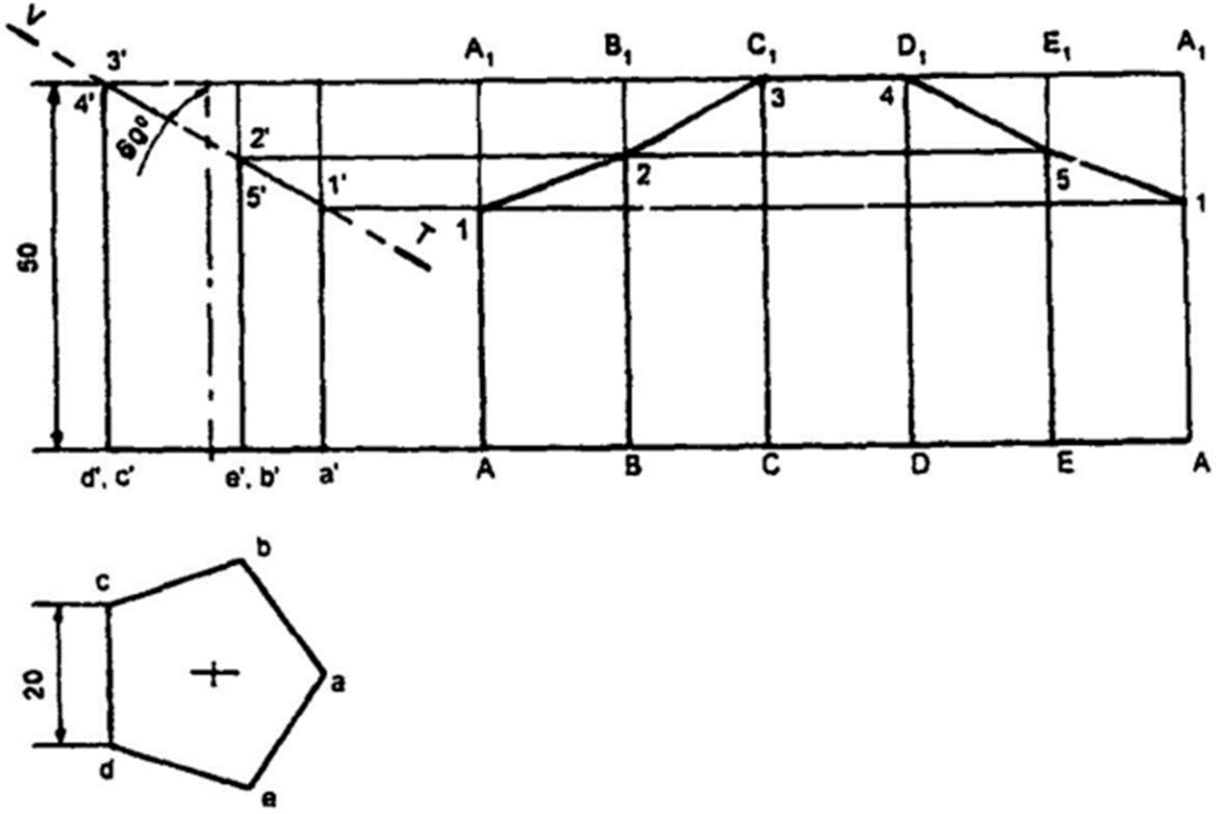
1. Draw the projections of the prism.
2. Draw the trace (VT) of the cutting plane intersecting the edges at points 1,2,3, etc.
3. Draw the stretch-out AA and mark-off the sides of the base along this in succession i.e., AB, BC, CD, DE and EA.
4. Erect perpendiculars through A,B,C etc., and mark the edges AA1, BB1 etc., equal to the height of the prism.



DEVELOPMENT OF SOLIDS/SURFACES

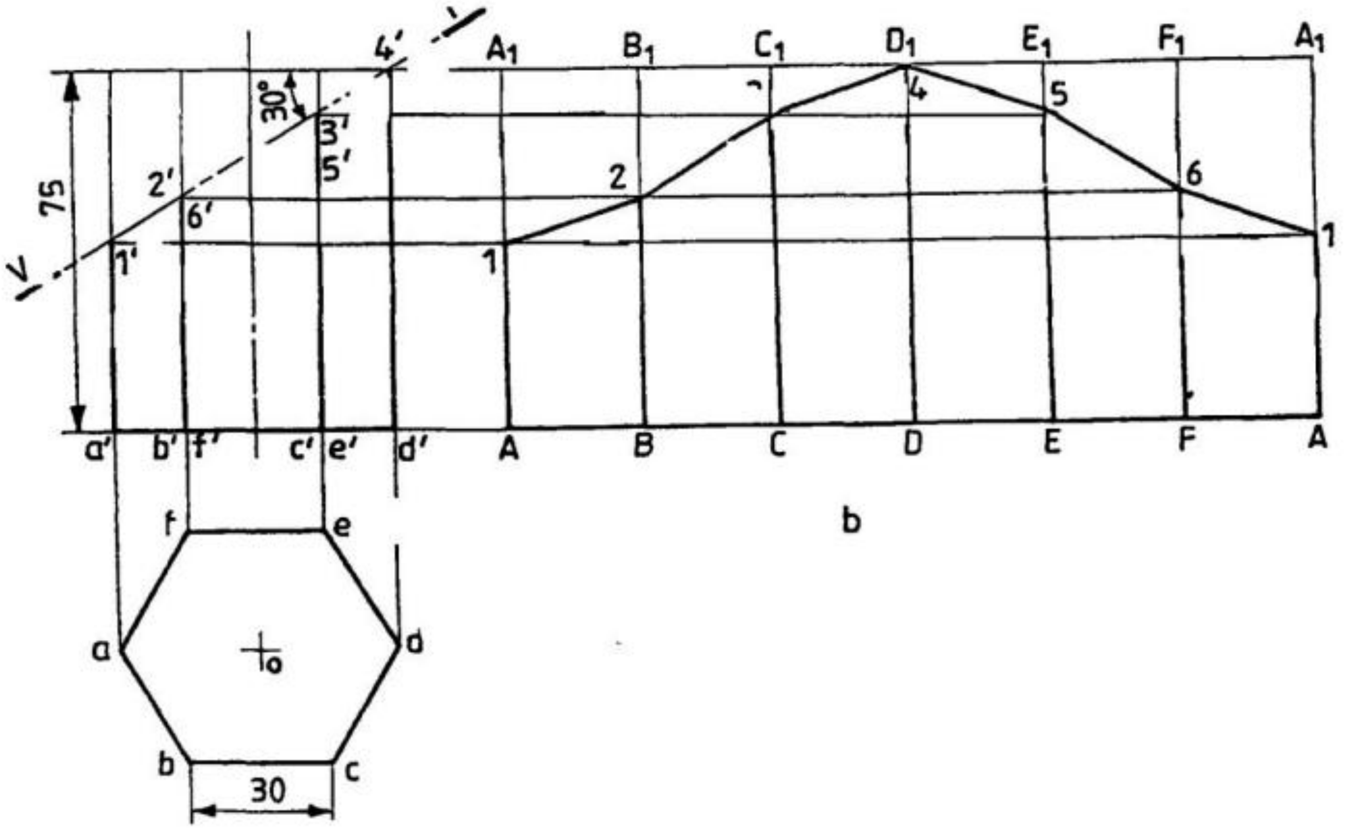
5. Project the points 1^1 , 2^1 , 3^1 etc., and obtain 1,2,3 etc., respectively on the corresponding edges in the development.

6. Join the points 1,2,3 etc., by straight lines and darken the sides corresponding to the truncated portion of the solid.



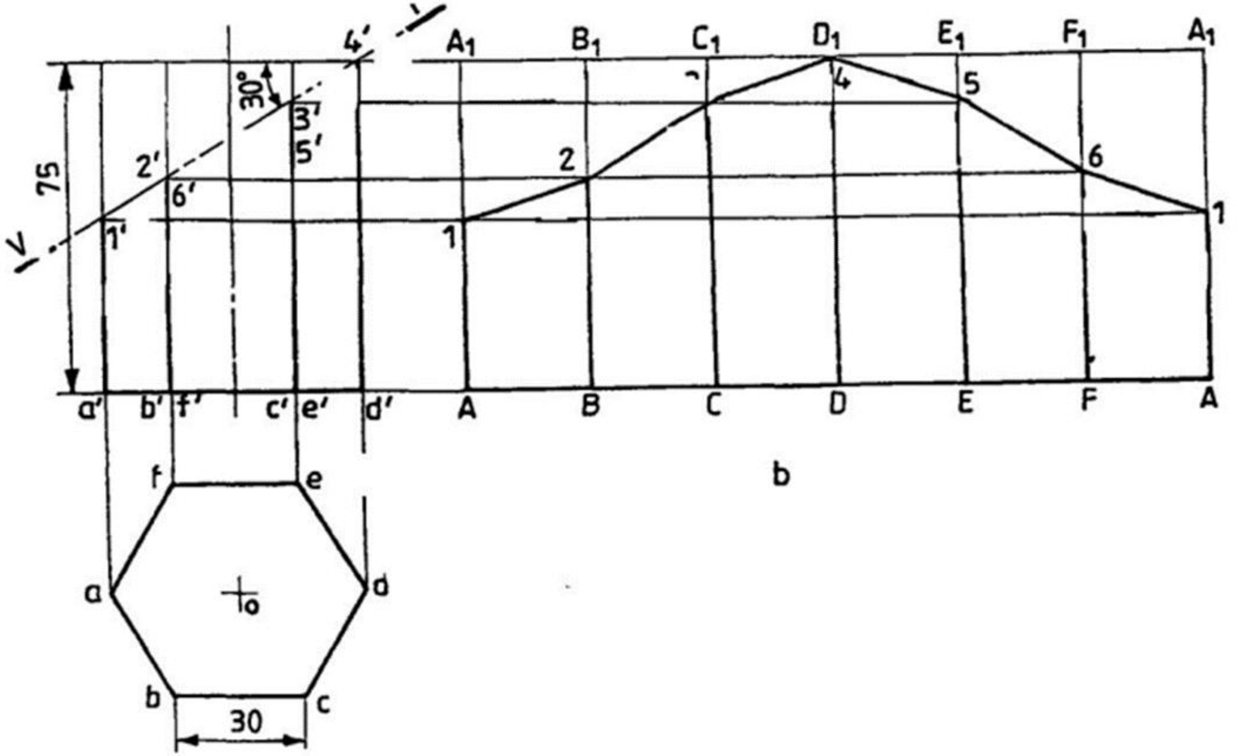
DEVELOPMENT OF A TRUNCATED HEXAGONAL PRISM

A hexagonal prism of side of base 30 mm and axis 75 mm long is resting on its base on HP. such that a rectangular face is parallel to HP. It is cut by a section plane perpendicular to VP and inclined at 30^{O} to HP. The section plane is passing through the top end of an extreme lateral edge of the prism. Draw the development of the lateral surface of the cut prism.



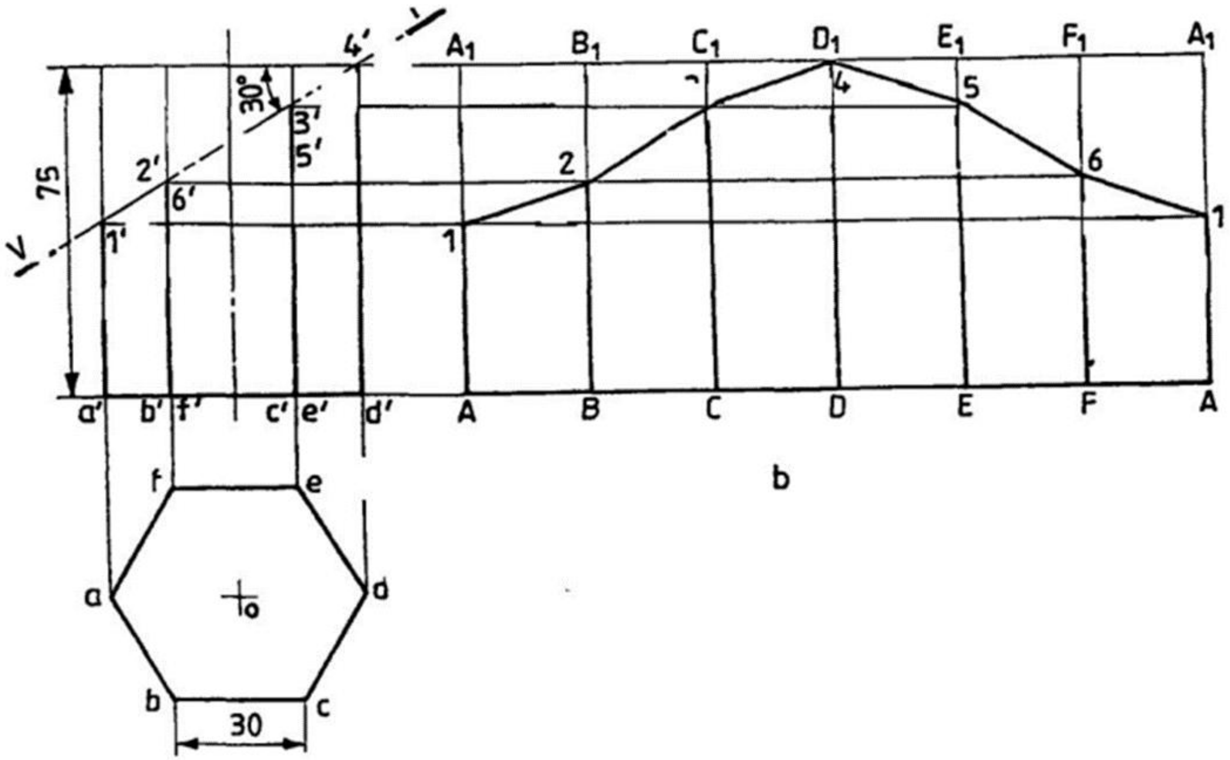
Instructions

1. Draw the projections of the prism.
2. Draw the section plane VT.
3. Draw the development AA_1-A_1A of the complete prism following the stretch out line principle.
4. Locate the point of intersection 1^1, 2^1 etc., between VT and the edges of the prism.
5. Draw horizontal lines through 1^1, 2^1 etc., and obtain 1,2, etc., on the corresponding edges in the development.



Instructions

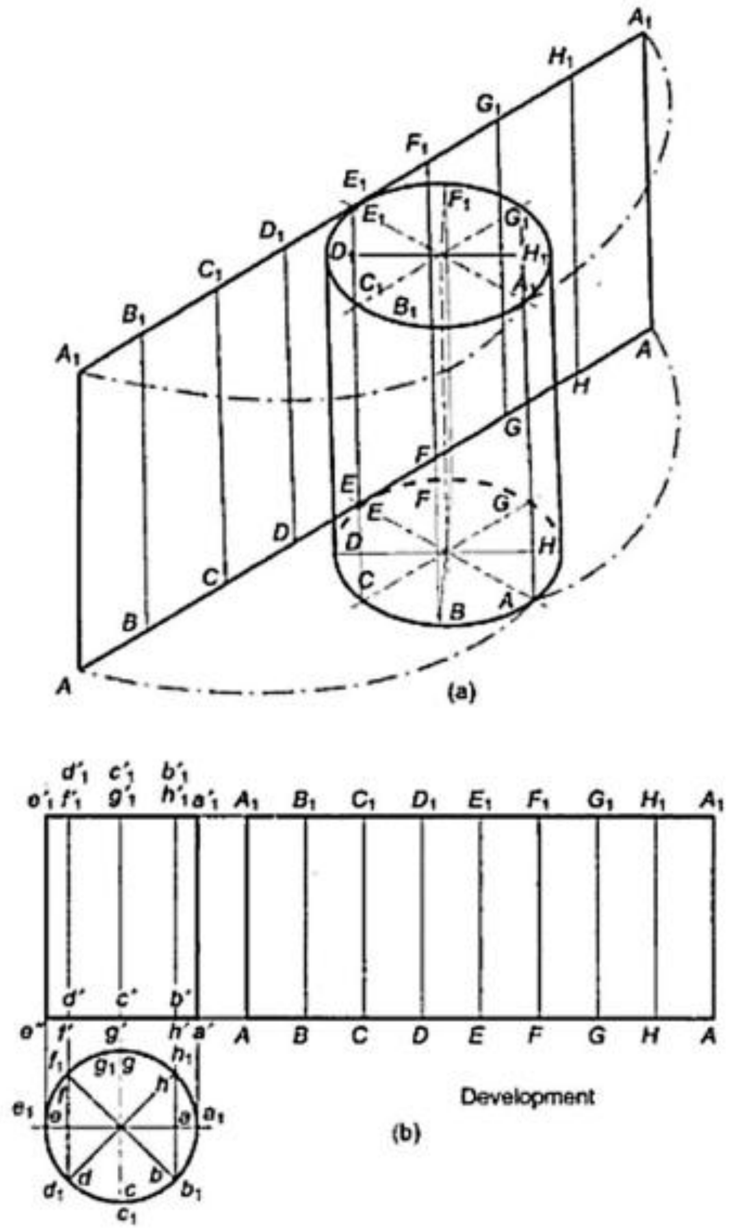
6. Join the points 1,2, etc., by straight lines and darken the sides corresponding to the retained portion of the solid.



DEVELOPMENT OF A CYLINDER

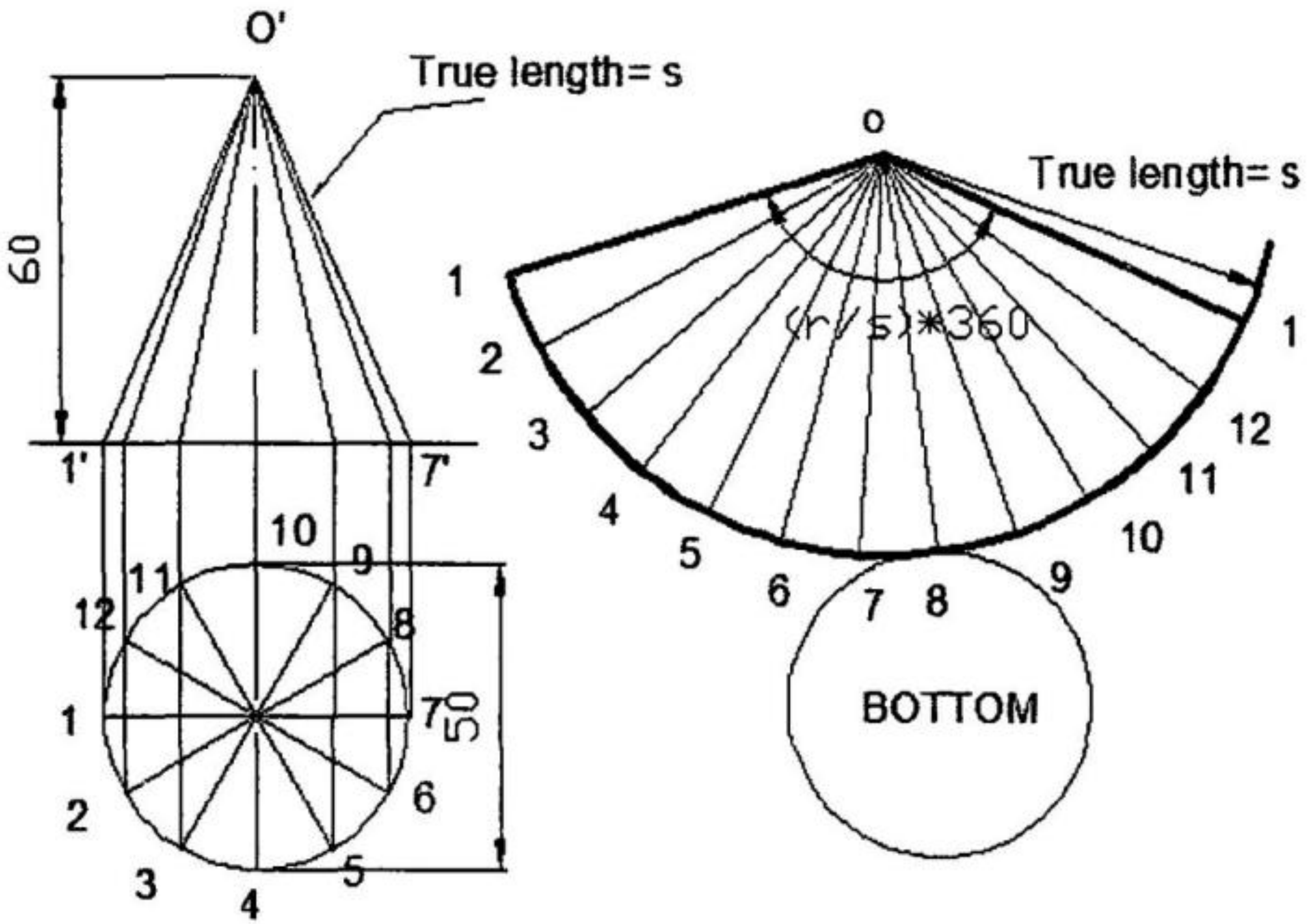
A cylinder of 25 mm in diameter, with an axis 40 mm long is resting on its base, with its axis perpendicular to the horizontal plane (HP). Draw the projections of the cylinder and develop the lateral surfaces of the cylinder.

The Figure shows the development of a cylinder. In this the length of the rectangle representing the development of the lateral surface of the cylinder is equal to the 4 circumference (here πd is the diameter of the cylinder) of the circular base.



DEVELOPMENT OF A CONE

The development of the lateral surface of a cone is a sector of a circle. The radius and length of the arc are equal to the slant height and circumference of the base of the cone respectively. The included angle of the sector is given by (r/s) × 360^{o} , where r is the radius of the base of the cone and s is the true length.



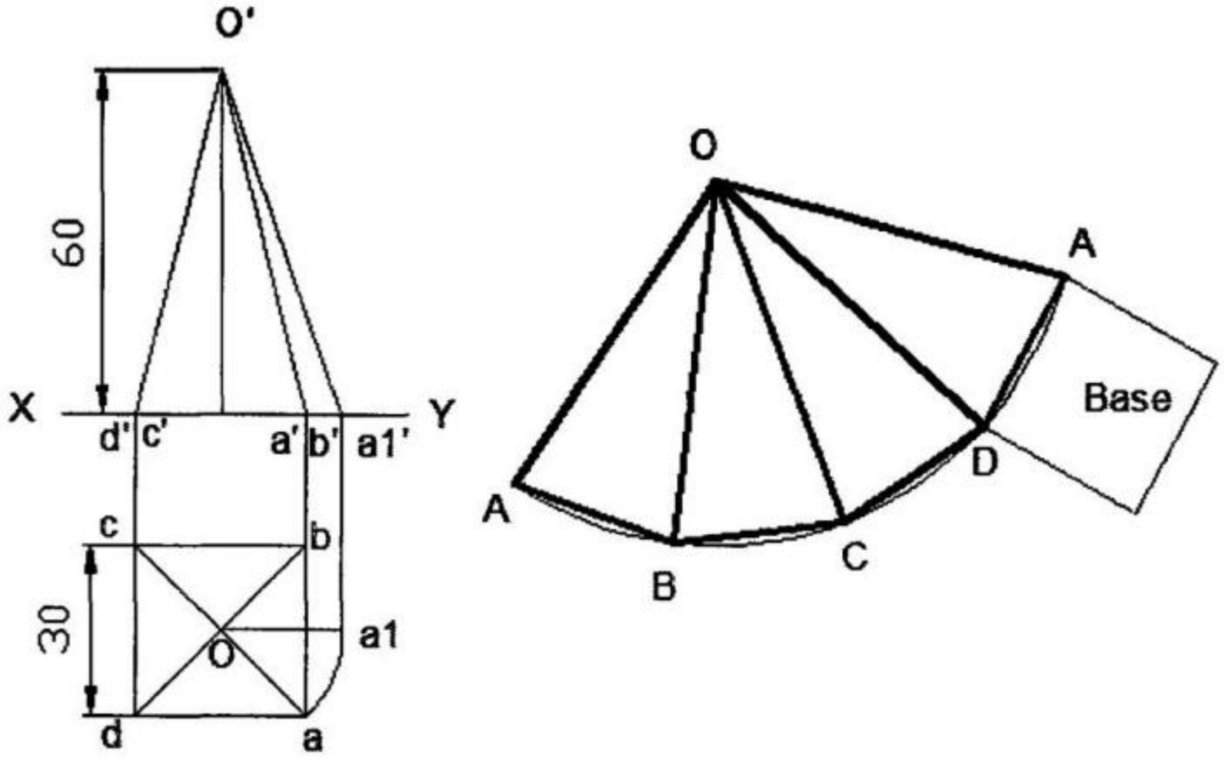
DEVELOPMENT OF SOLIDS/SURFACES

DEVELOPMENT OF A SQUARE PYRAMID

The development of a square pyramid with side of base 30 mm and height 60 mm.

Instructions

1. Draw the views of the pyramid assuming that it is resting on H.P and with an edge of the base parallel to V.P.
2. Determine the true length o-a of the slant edge. Note: In the orientation given for the solid, all the slant edges are inclined to both H.P and V.P.



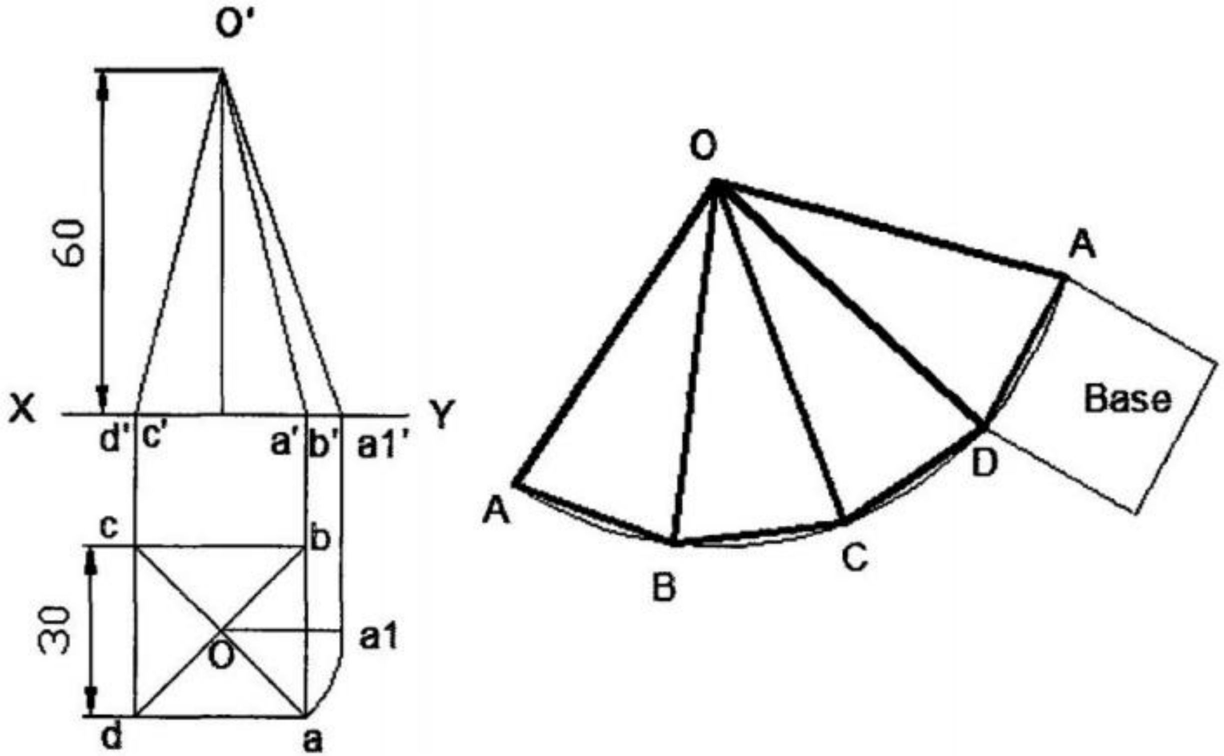
Hence, neither the front view nor the top view provides the

DEVELOPMENT OF SOLIDS/SURFACES

true length of the slant edge. To determine the true length of the slant edge, say OA, rotate oa till it is parallel to xy to the position oa_1. Through a_1, draw a projector to meet the line xy at a'_1 . Then o'_1 and a'_1 , all represents the true length of the slant edge OA. This method of determining the true length is also known as **rotation method**.

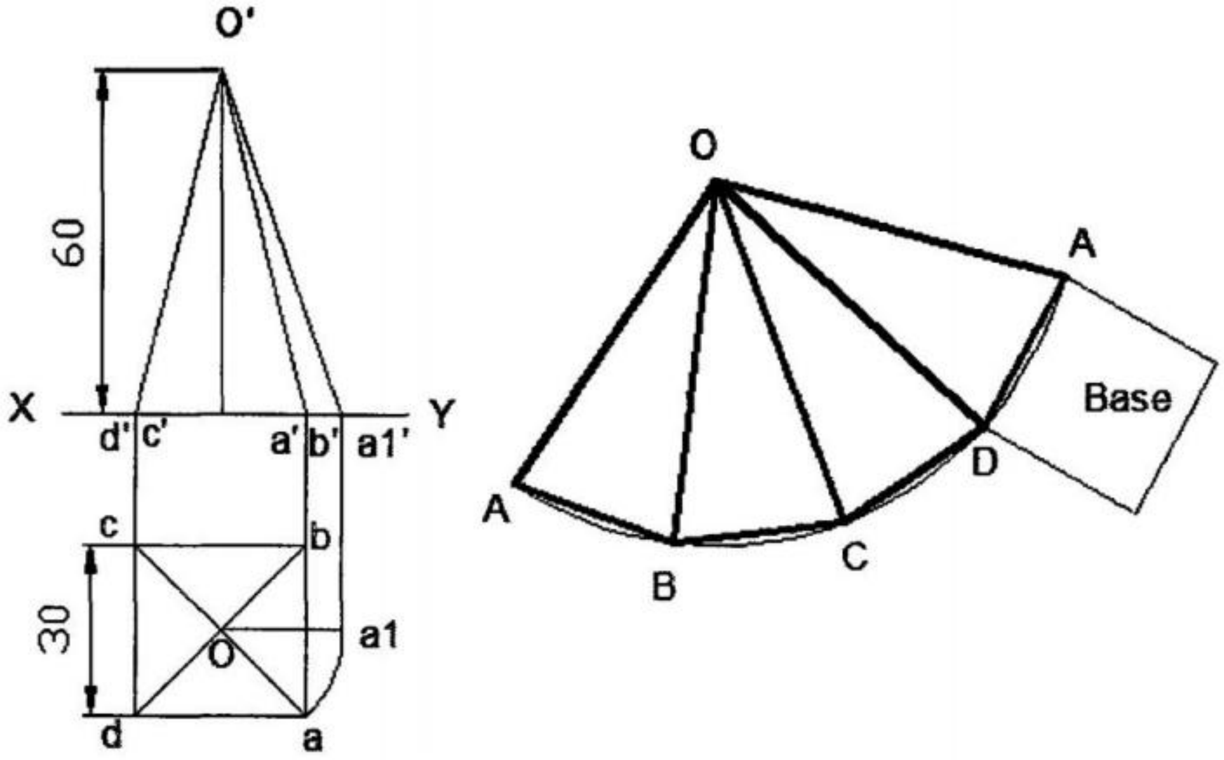
3. With centre O and radius o_1a_1 draw an arc.

4. Starting from A along the arc, mark the edges of the base i.e, AB, BC, CD and DA.



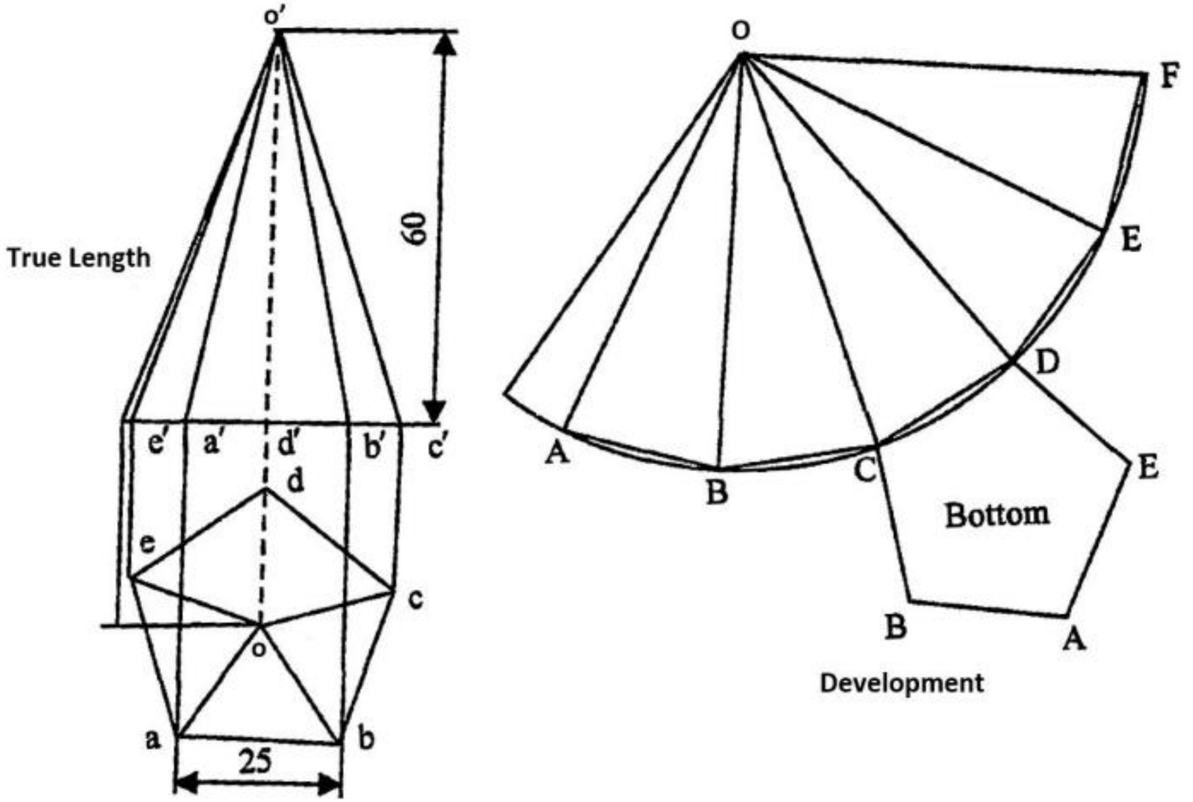
DEVELOPMENT OF SOLIDS/SURFACES

5. Join O to A,B,C, etc., representing the lines of folding and thus completing the development.



DEVELOPMENT OF PENTAGONAL PYRAMID

The development of a pentagonal pyramid with side of base 25 mm and height 60 mm.



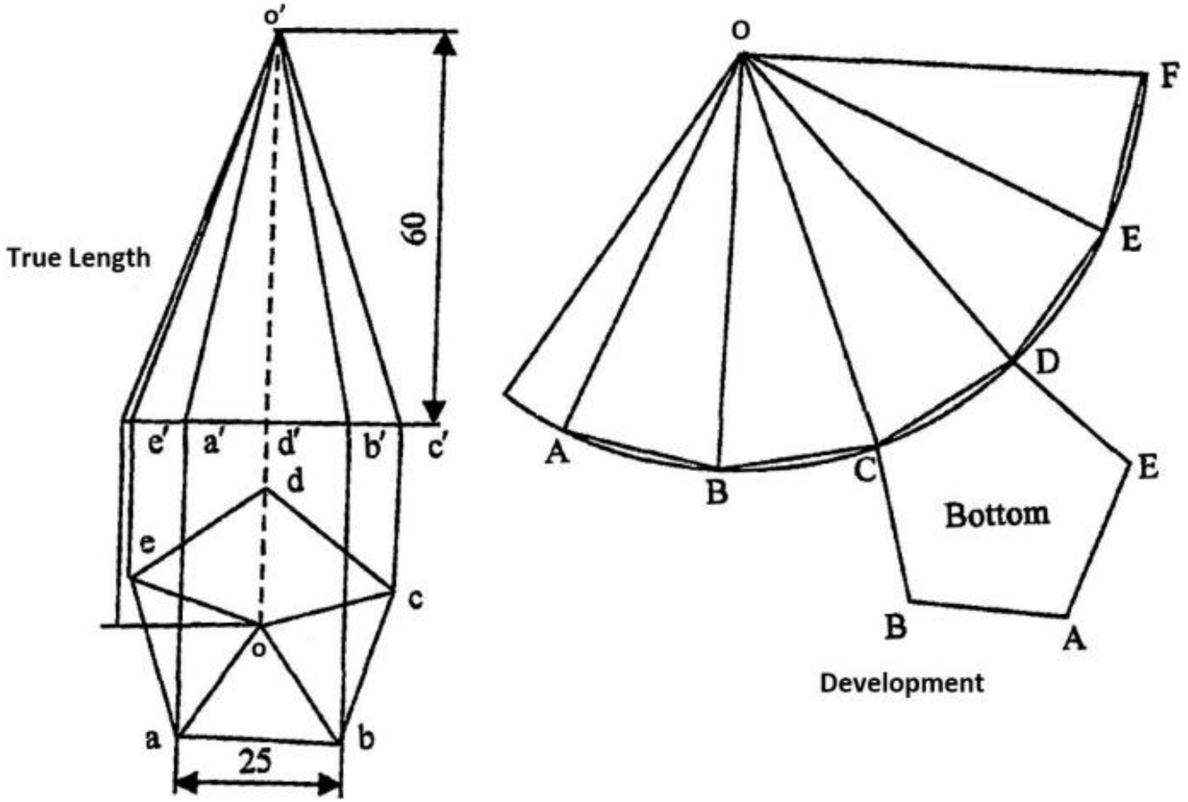
Instructions

1. Draw the orthographic views of the pyramid ABCDE with its base on H.P and axis parallel to V.P.
2. With centre D of the pyramid and radius equal to the true length of the slant edge draw an arc.
3. Mark off the edges starting from A along the arc

DEVELOPMENT OF SOLIDS/SURFACES

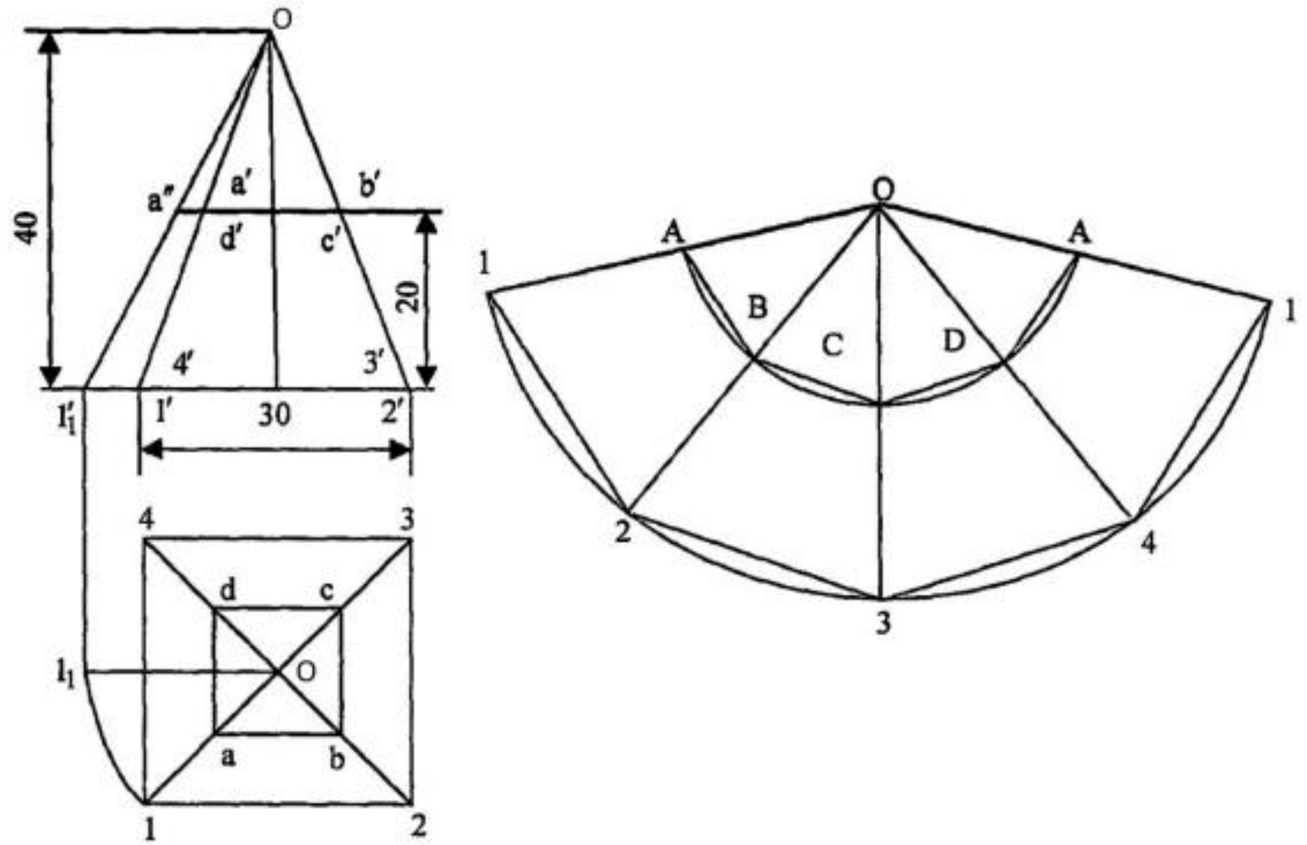
and join them to D representing the lines of folding.

4. Add the base at a suitable location.



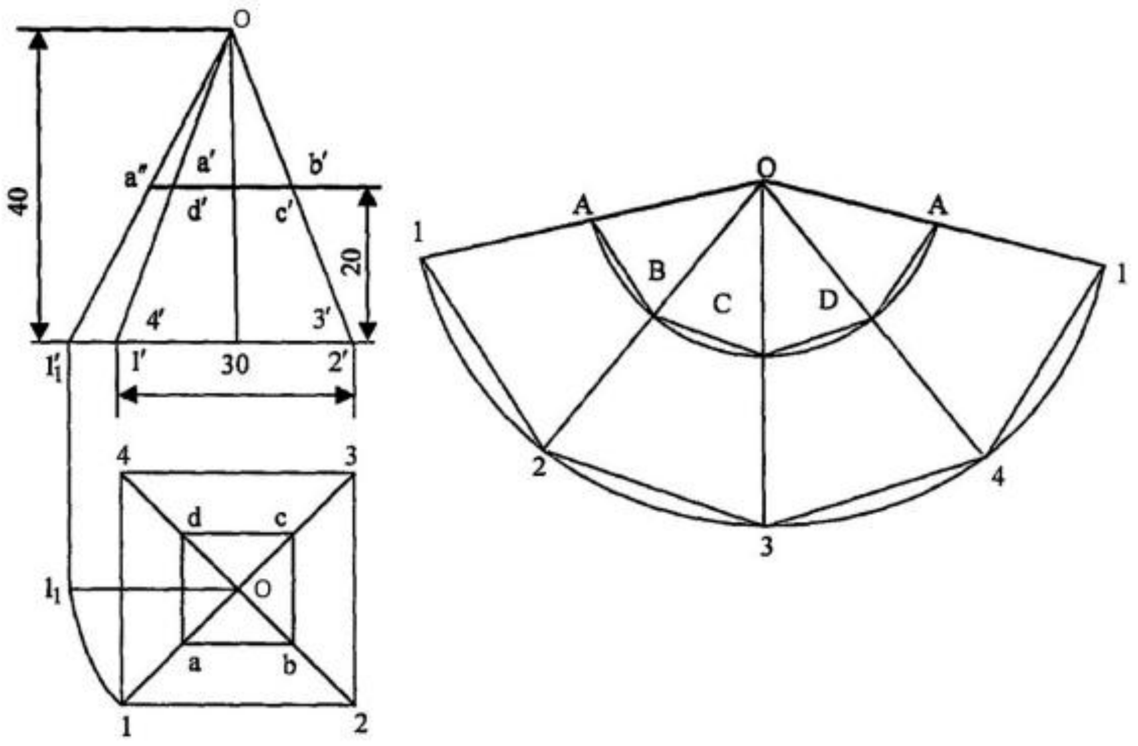
DEVELOPMENT OF FRUSTUM OF SQUARE PYRAMID

Draw the development of the lateral surface of the frustum of the square pyramid of side of base 30 mm and axis 40 mm, resting on HP with one of the base edges parallel to VP. It is cut by a horizontal cutting plane at a height of 20 mm.



Instructions

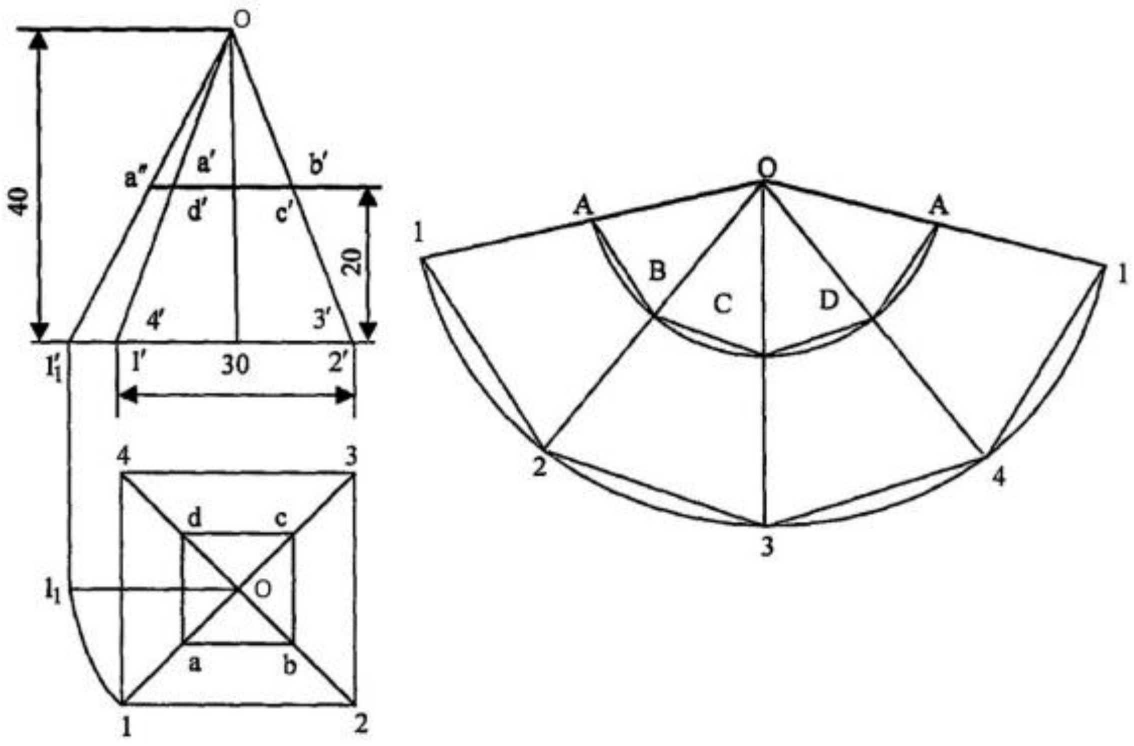
1. Draw the projections of the square pyramid.
2. Determine the true length of the slant edge.
3. Draw the trace of the cutting plane.
4. Locate the points of intersection of the cutting plane on the slant edges $a^1b^1c^1d^1$ of the pyramid.
5. With any point O as center and radius equal to the true length of the slant edge draw an arc of the circle.
6. With radius equal to the side of the base 30 mm, step-off divisions on the above arc.



Instructions

7. Join the above division points 1,2,3 etc., in the order with the center of the arc o. The full development of the pyramid is given by 012341.

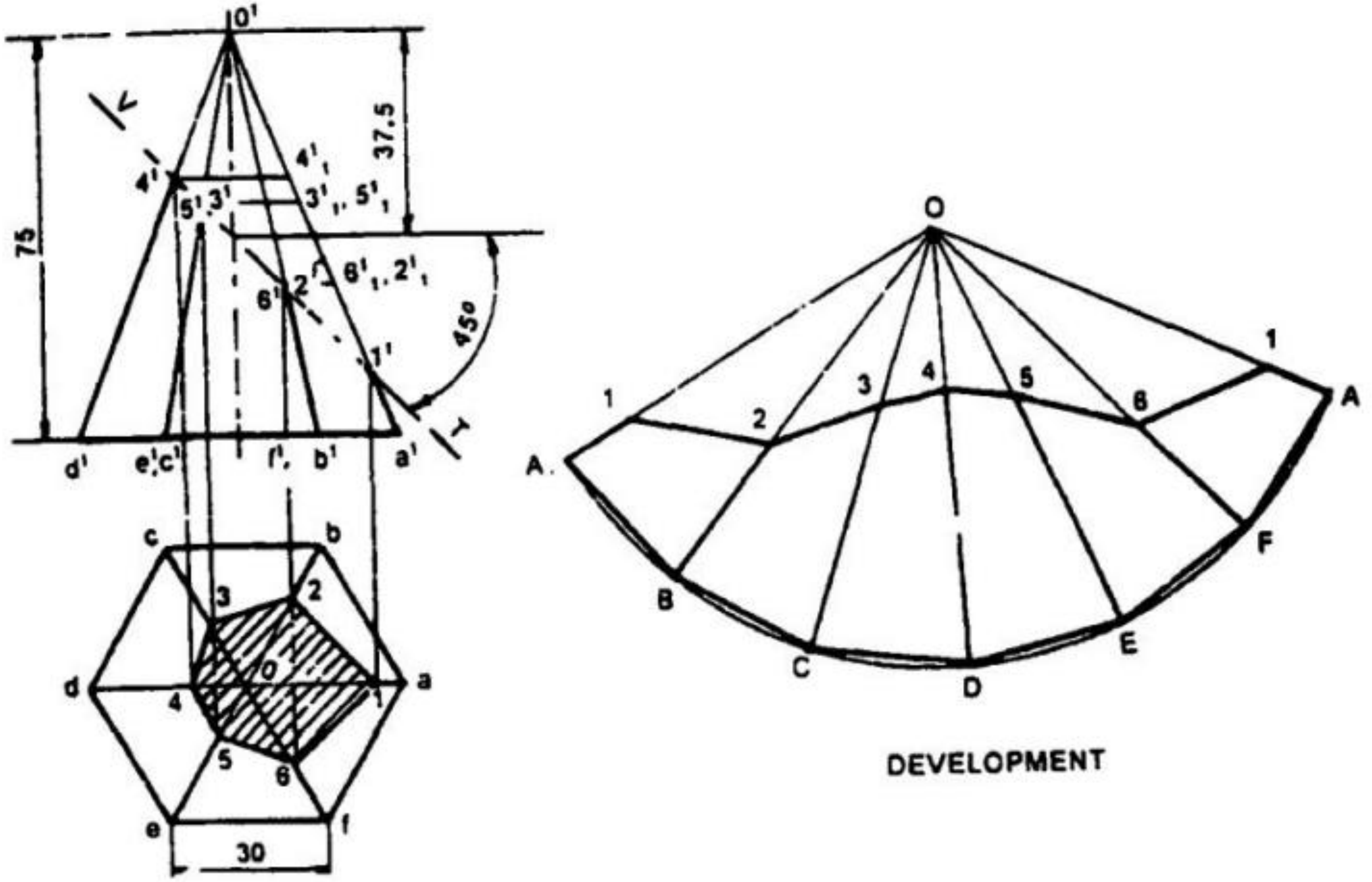
8. With centre O and radius equal to o-a mark-off these projections at A, B, C, D, A. Join A-B, B-C etc. ABCDA-12341 is the development of the frustum of the square pyramid.



EXERCISE 1

TRUNCATED HEXAGONAL PYRAMID

A hexagonal pyramid with side of base 30 mm and height 75 mm stands with its base on HP and an edge of the base parrallel to V.P. It is cut by a plane perpendicular to VP, inclined at 45^{O} to H.P and passing through the mid-point of the axis. Draw the (sectioned) top view and develop the lateral surface of the truncated pyramid.

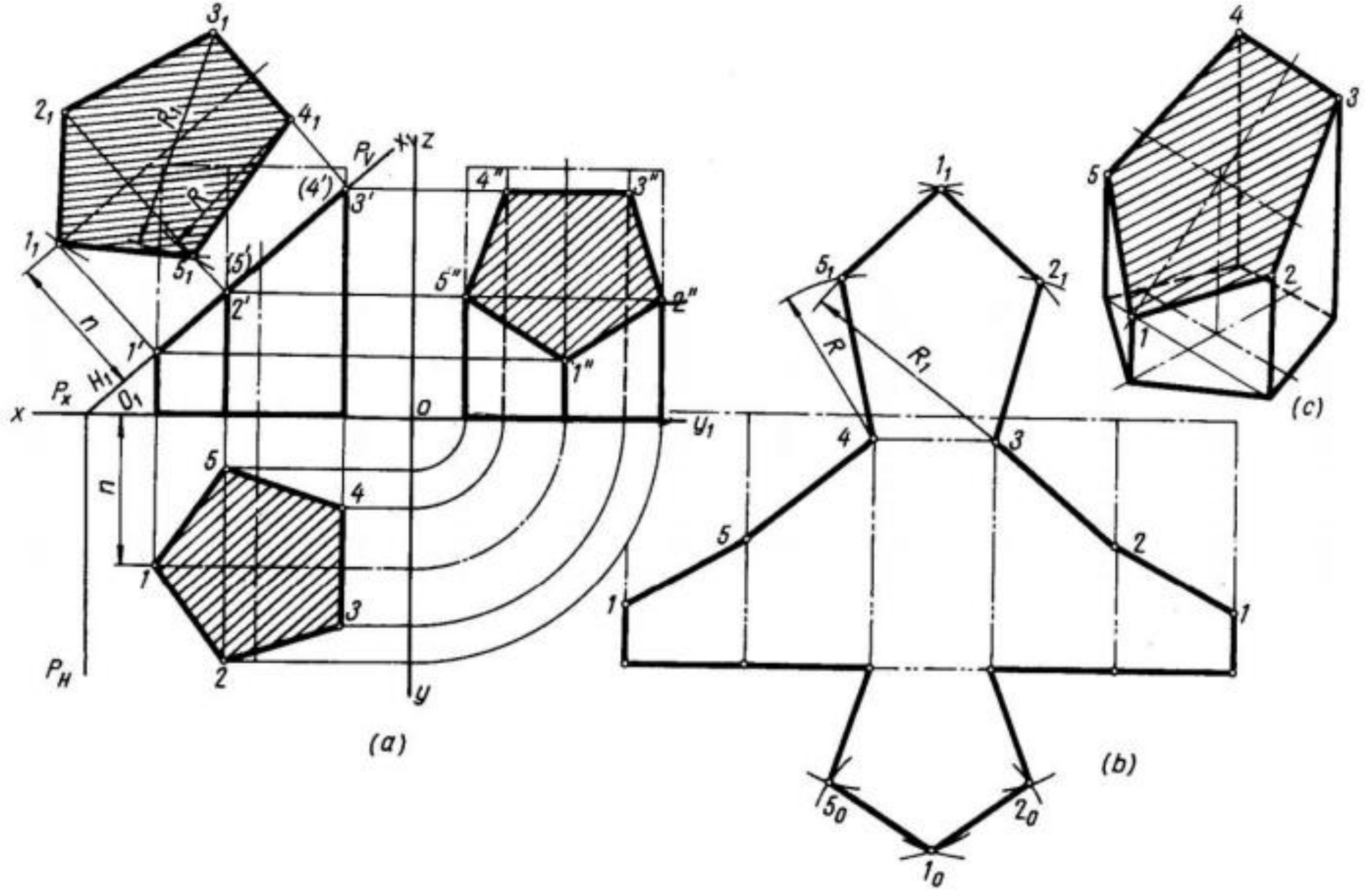


EXERCISE 2

QUESTION

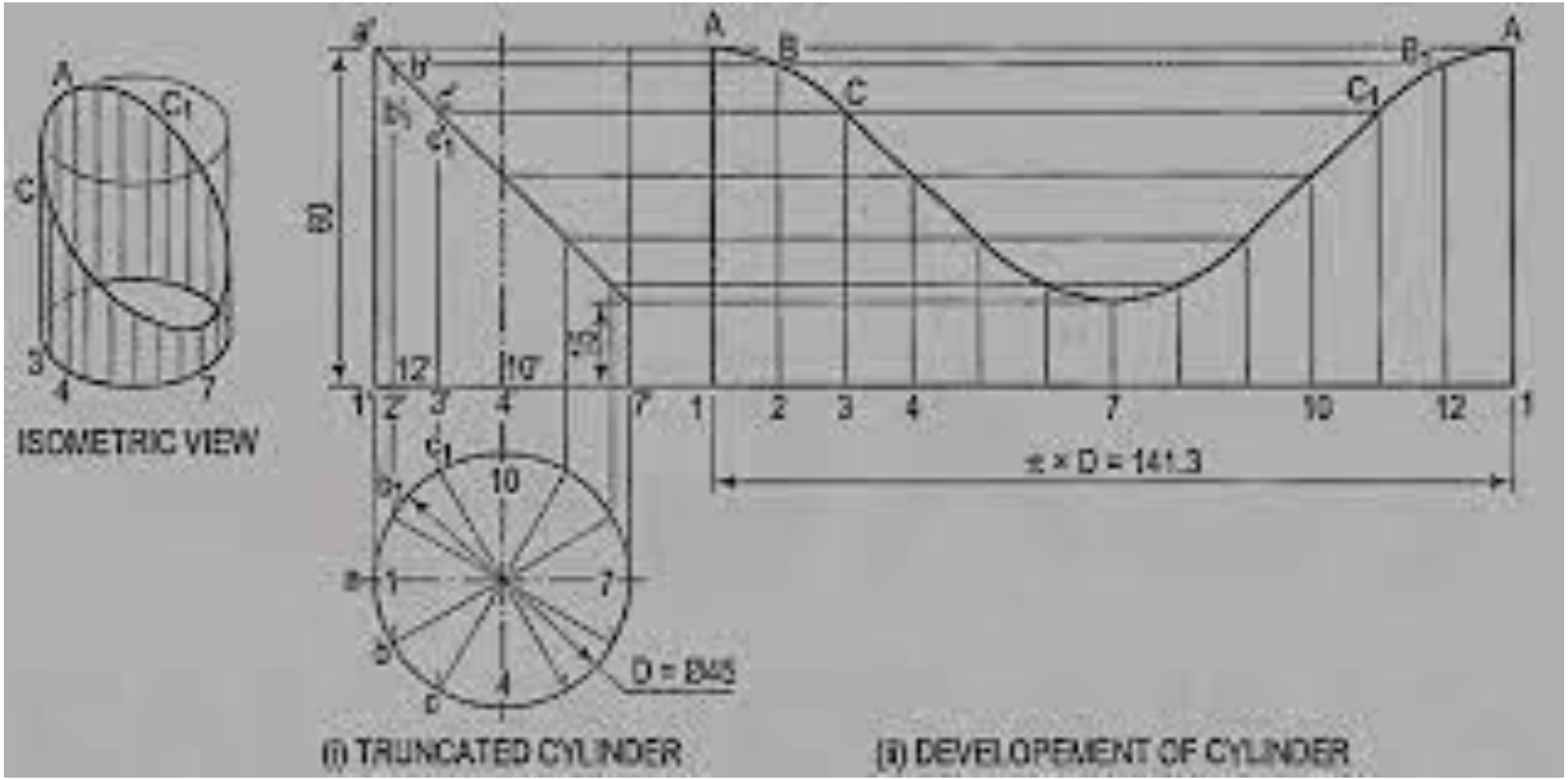
A Pentagonal prism of base 45 mm and height of 80 mm lies with its base on the HP such that one of its base edges is perpendicular to the vertical plane. The pentagonal prism is truncated by a vertical projecting plane which makes an angle of 45° with the horizontal plane at a point 50 mm from the base through the axis. Construct the three (3) projections, the true shape of the section and develop the surface of the truncated pentagonal prism (Pentagonal Pyramid) completely.

EXERCISE 2



3_1
2_1
4_1
1_1
5_1
R_1
R
P_V
$x \vee z$
(4')
3'
(5')
2'
1'
4''
3''
5''
2''
1''
n
H_1
P_x
O_1
x
o
y_1
5
4
1
3
2
P_H
y
(a)
1_1
5_1
2_1
R
R_1
4
3
5
2
1
1
5_0
2_0
1_0
(b)
4
3
5
2
1
(c)

EXERCISE 3



ISOMETRIC VIEW
(i) TRUNCATED CYLINDER
(ii) DEVELOPEMENT OF CYLINDER
D = Ø45
π × D = 141.3

CURVES OF INTERSECTION

We will cover these skills:

- Introduction.
- Methods of determining the curves of intersection.
- Line Method.
- Cutting Plane Method.
- Drawing line of intersection of surfaces of two solids by the Line method
- Drawing the curve of intersection of surfaces in orthographic projections using the Cutting plane method.

CURVE OF INTERSECTION

INTRODUCTION

When the surface of one solid meets that of another solid, the line along which the two surfaces meet each other, is known as the **curve of intersection** of the surfaces of the two solids. Similarly, if a hole is cut in a solid, the line along which the surface of the hole meets that of the solid is also known as the curve of intersection of the surfaces. If a solid completely penetrates another solid, the line along which the two surfaces meet is known as **line of interpenetration or the curve of interpenetration**

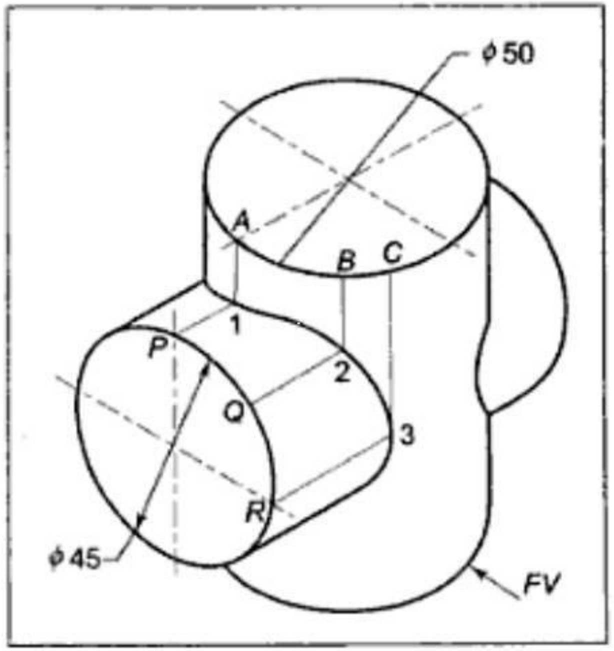


Figure 8.1 Curve of Intersection of Surfaces

CURVE OF INTERSECTION

METHODS OF DETERMINING THE CURVE OF INTERSECTION OR THE CURVE OF INTERPENETRATION

There are two methods available for determining the curve of intersection of two solids:

1. Line method or generator method
2. Cutting plane method

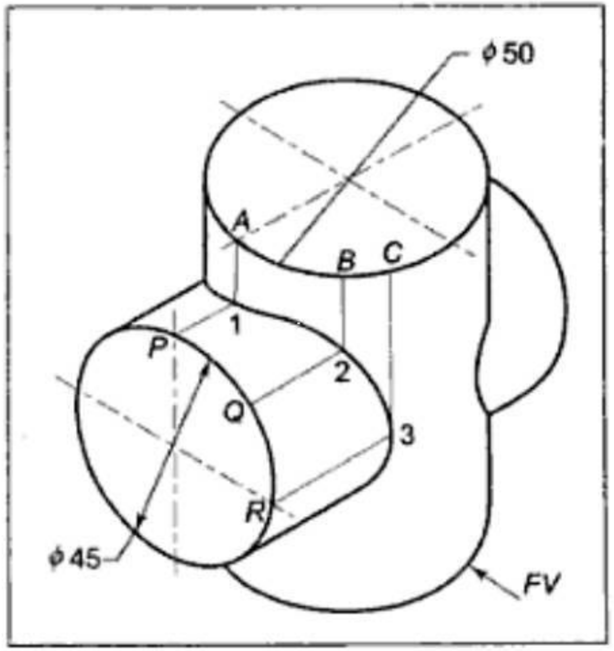


Figure 8.1 Curve of Intersection of Surfaces

CURVE OF INTERSECTION

LINE METHOD

The curve of intersection of surfaces being the line or lines along which the surfaces of the two solids meet, it is made up of points common between the two surfaces. In other words, each point on the line of intersection is located on the surfaces of both the solids. As shown in Figure 8.1, the points 1,2, 3 and so on the curve of intersection are all located on each of the two solids. The surface of each solid can be divided into a number of convenient lines, which may be generators in case of cylinders and cones. If these lines of the two intersect, they must intersect at points that are on the curve of intersection.

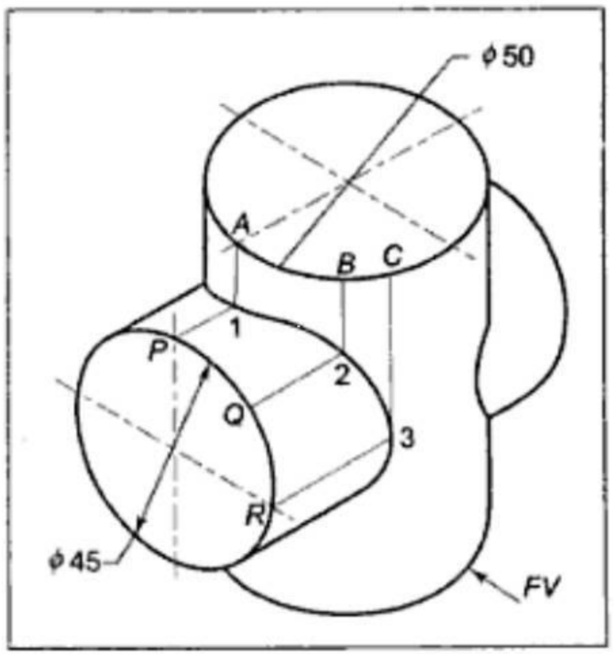


Figure 8.1 Curve of Intersection of Surfaces

CURVE OF INTERSECTION

In Figure 8.1, lines P-1, Q-2, P-3 and so on drawn on the surface of horizontal cylinder, respectively, intersect surface lines A-l, B-2, C-3, etc. on the vertical cylinder at points 1, 2, 3 and so on which are the points on the curve of intersection. It is therefore possible to locate points on the curve of intersection by drawing convenient surface lines on the two solids and finding their points of intersection. Usually, the following lines are drawn as surface lines: lines starting from points on the base edges and drawn parallel to the side edges in case of prisms, lines starting from points on base edges and joining the apex in the case of pyramids, and generators in case of cylinders and cones. All edges of the solids are also utilised as surface lines.

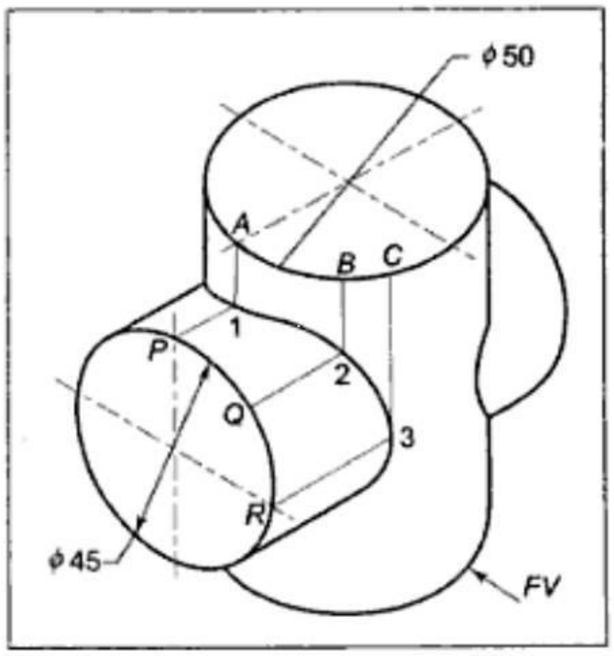


Figure 8.1 Curve of Intersection of Surfaces

CURVE OF INTERSECTION

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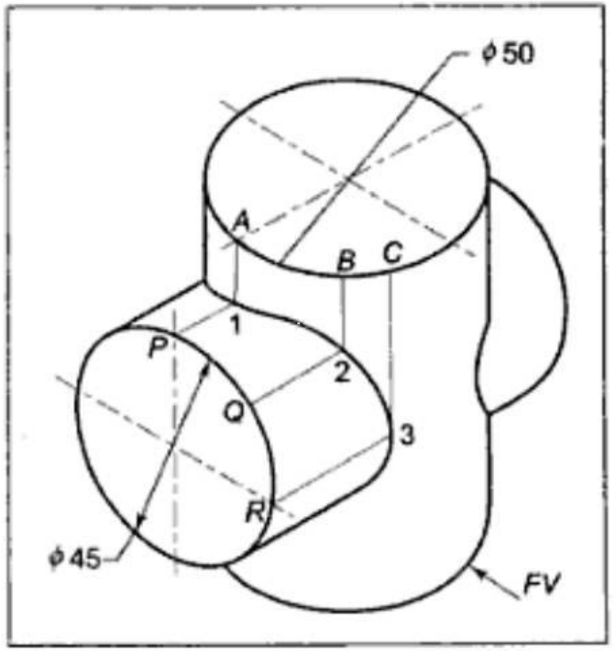

Figure 8.1 Curve of Intersection of Surfaces

CURVE OF INTERSECTION

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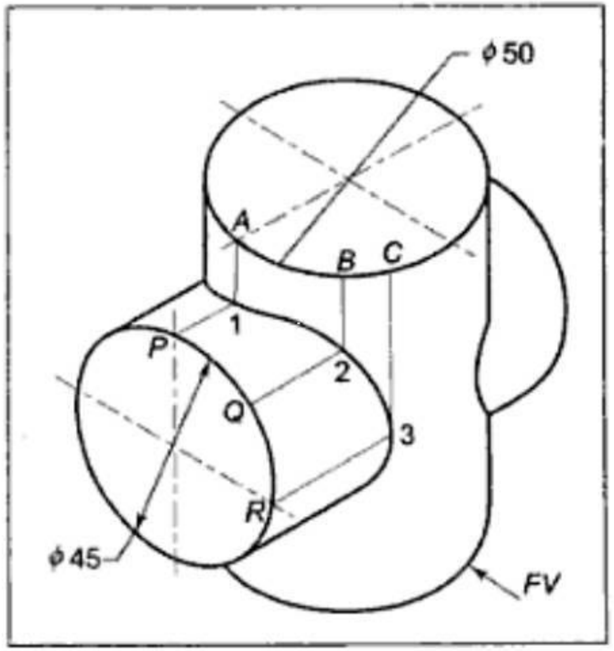


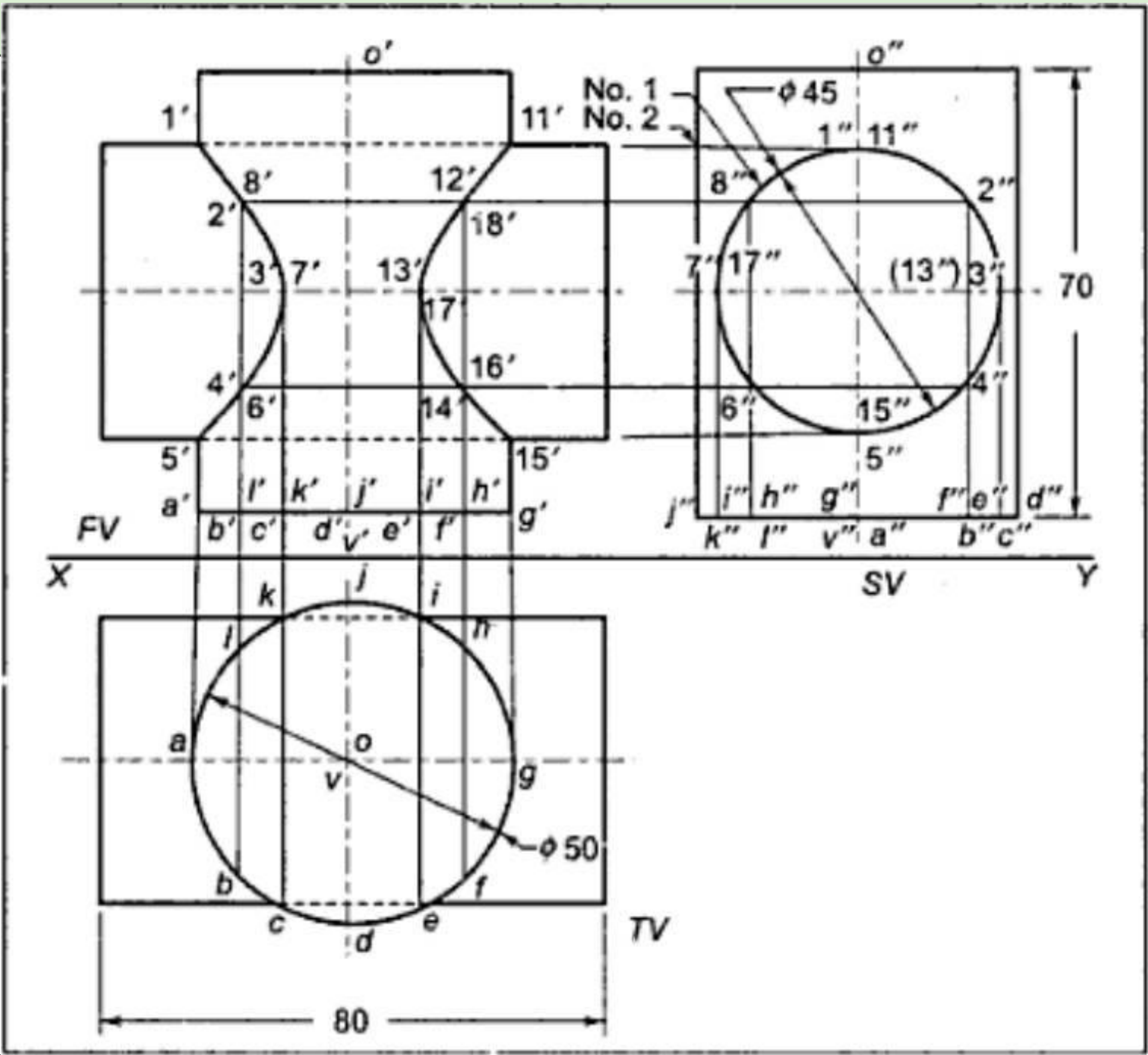
Figure 8.1 Curve of Intersection of Surfaces

apex in the case of pyramids, and generators in case of cylinders and cones. All edges of the solids are also utilised as surface lines.

CURVE OF INTERSECTION

<u>PROCEDURE FOR DRAWING LINE OF INTERSECTION OF SURFACES OF TWO SOLIDS BY THE LINE METHOD</u>

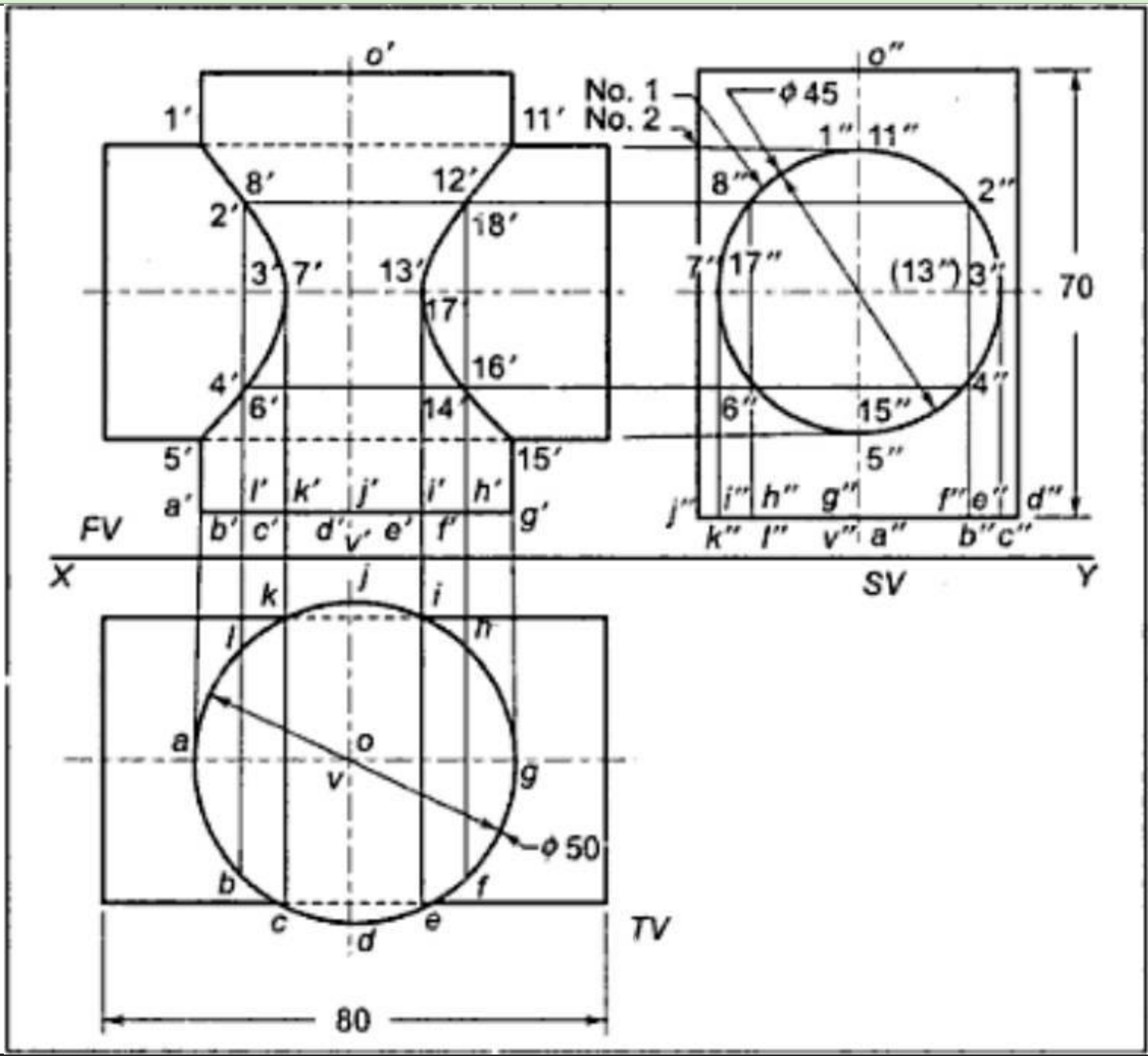
A vertical cylinder, 50 mm in diameter and 70 mm in length, is resting on its base, with its axis perpendicular to the HP. It is completely penetrated by another horizontal cylinder 45 mm in diameter and 80 mm in length. The axis of the horizontal cylinder is parallel to the VP and the two axes bisect each other. Draw the projections showing the curves of intersection cylinders and cones. If these lines of the two



CURVE OF INTERSECTION

Step 1: Draw, by thin lines the projections of two uncut cylinders in proper relative positions. The vertical cylinder is projected as a circle in top view and the horizontal one as a circle in side view. The other two views are rectangles for both the solids. (See Figure 8.2).

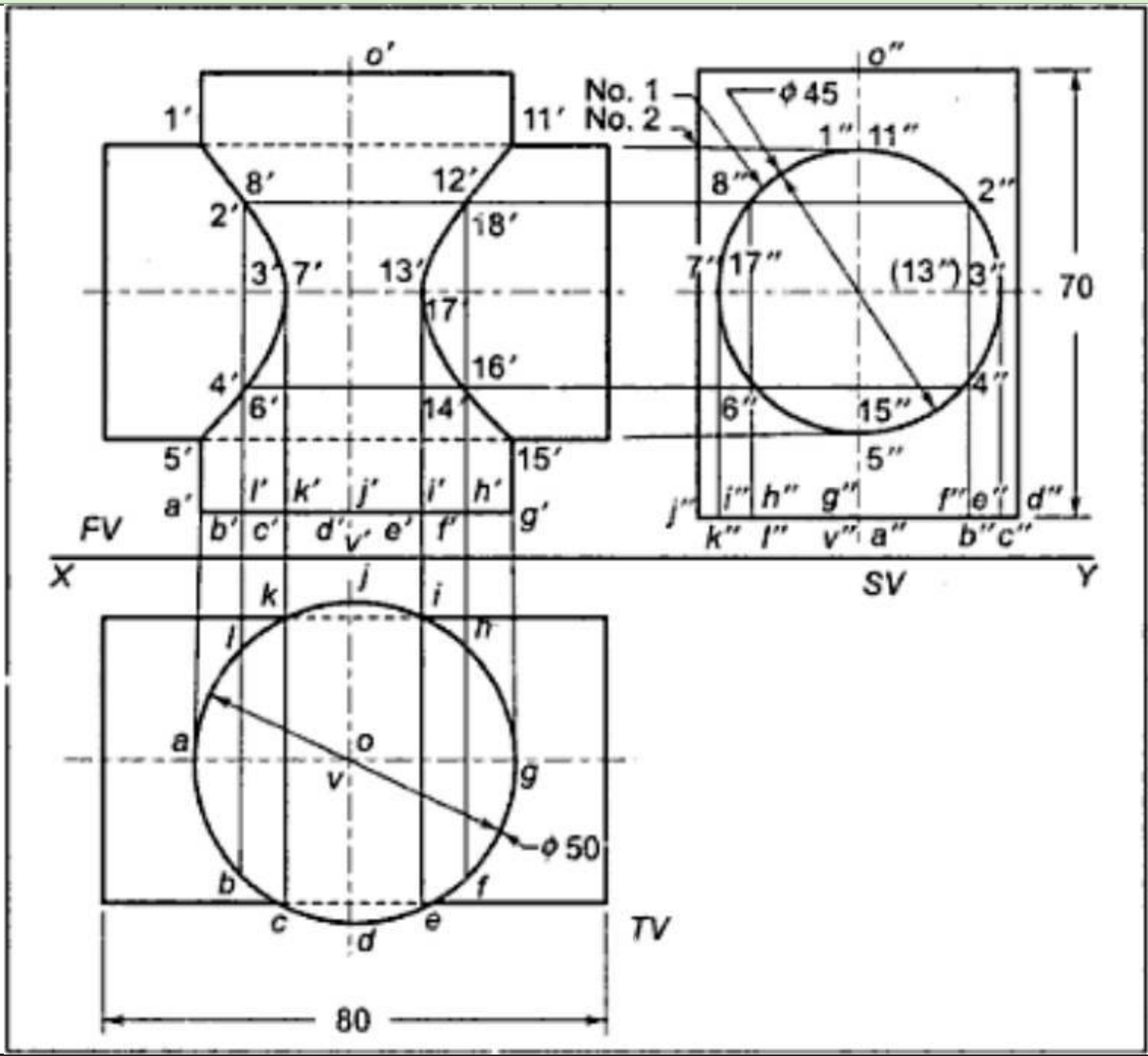
Step 2: The axial view is a circle for the horizontal cylinder in the side view and for the vertical one it is in the top view. The solution can, therefore, be started either from the side view or from the top view. Let the horizontal cylinder be numbered as solid 1 and the



CURVE OF INTERSECTION

vertical one as 2. In the side view, the circle of solid 1 is completely within the boundary of solid 2. Hence, this circle represents the curve of intersection in the side view.

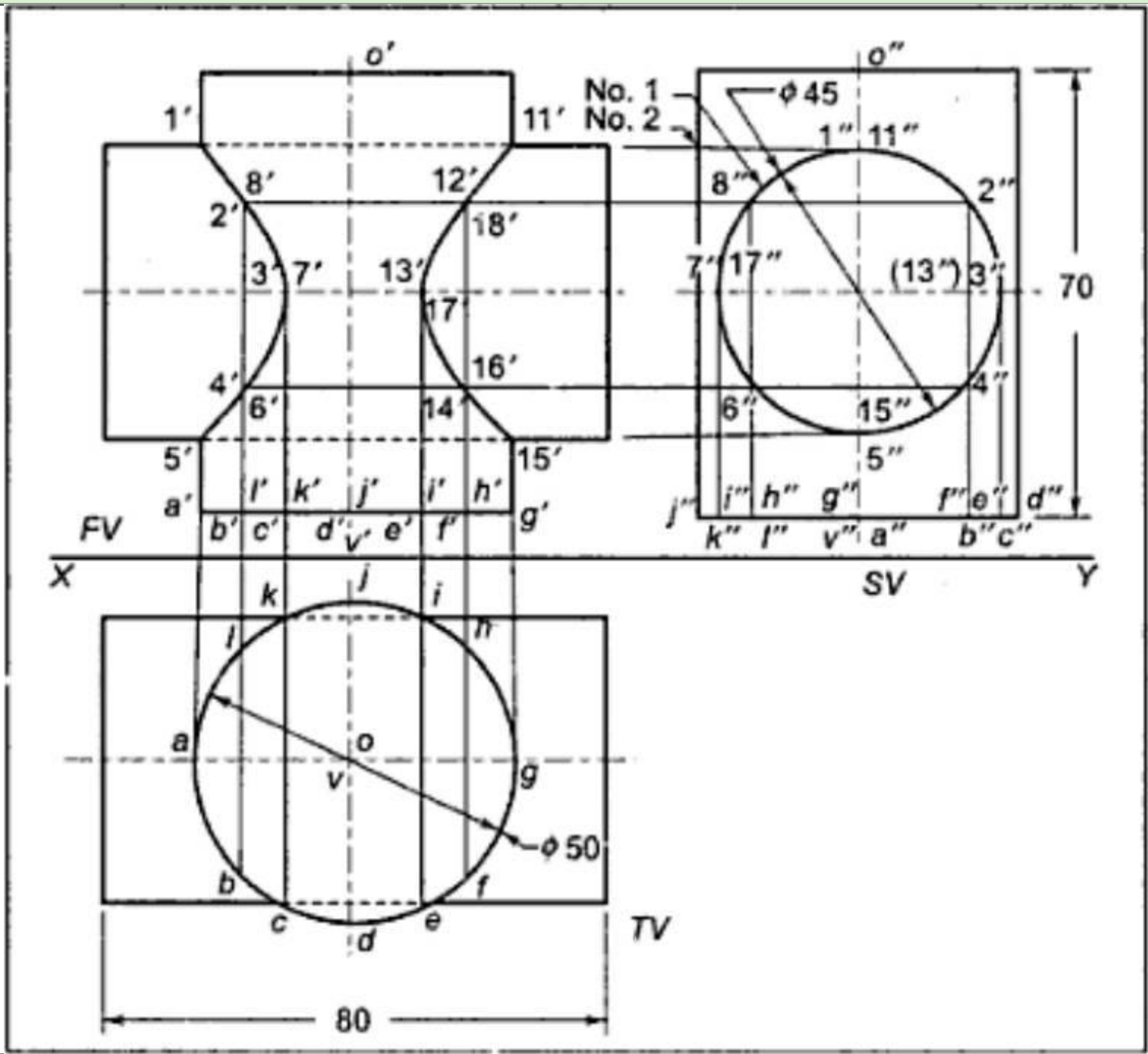
Step 3: Draw a number of generators on solid 2. Highest, lowest, extreme left and right points on the curve in the side view are critical points. Points on central generators in that view are also critical points but those points are the highest and lowest as well. Ascertain that a surface line passes through each and every critical point. The solids being curved, additional



CURVE OF INTERSECTION

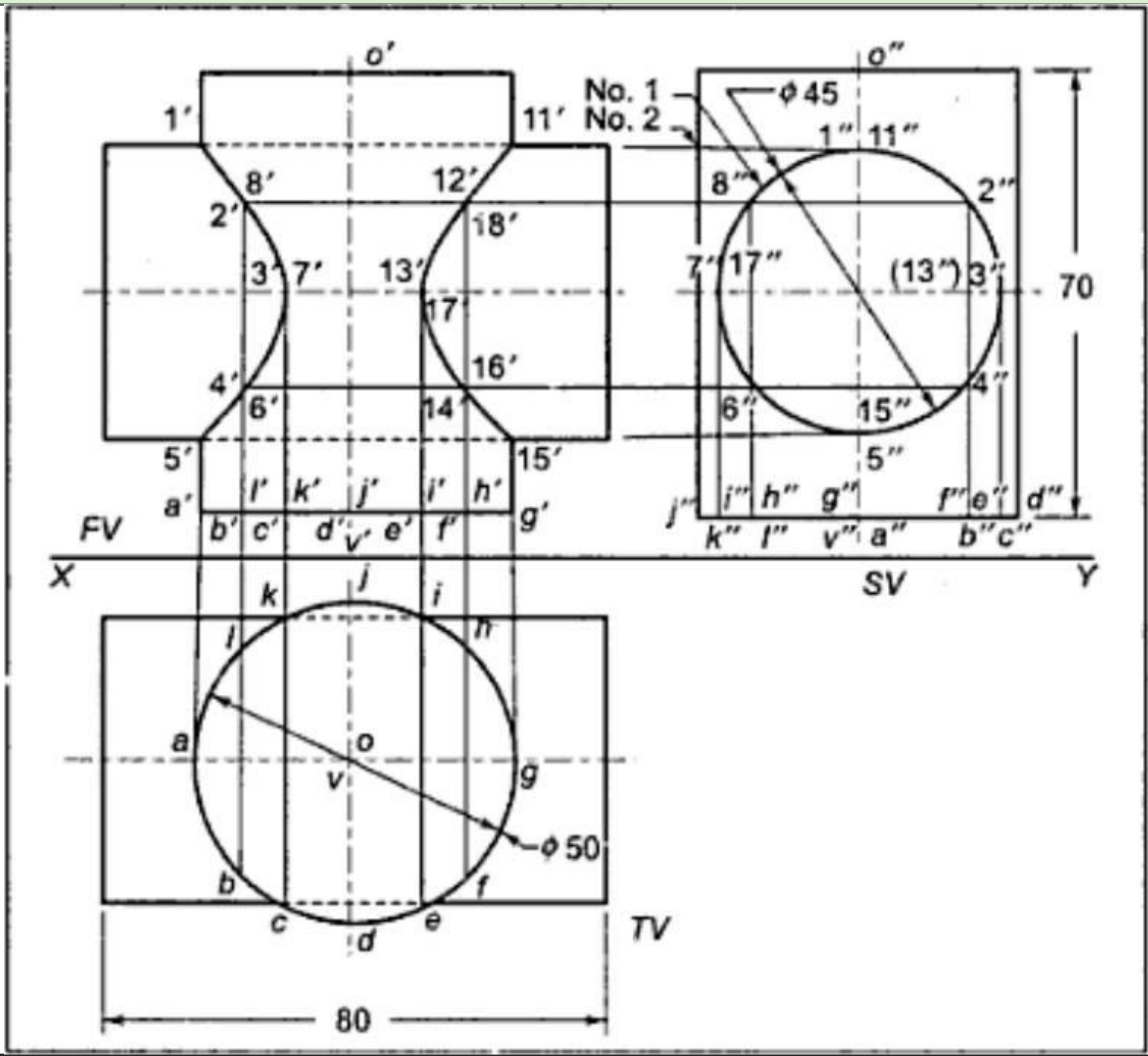
generators should also be drawn. Obtain the projections of all the surface lines in other views.

Step 4: Locate the points common between the surface lines and the curve of intersection and name them. The curve is a closed ended type. Hence, two sets of points are used — one set for the points on the visible surface lines and the other for those on the hidden surface lines.



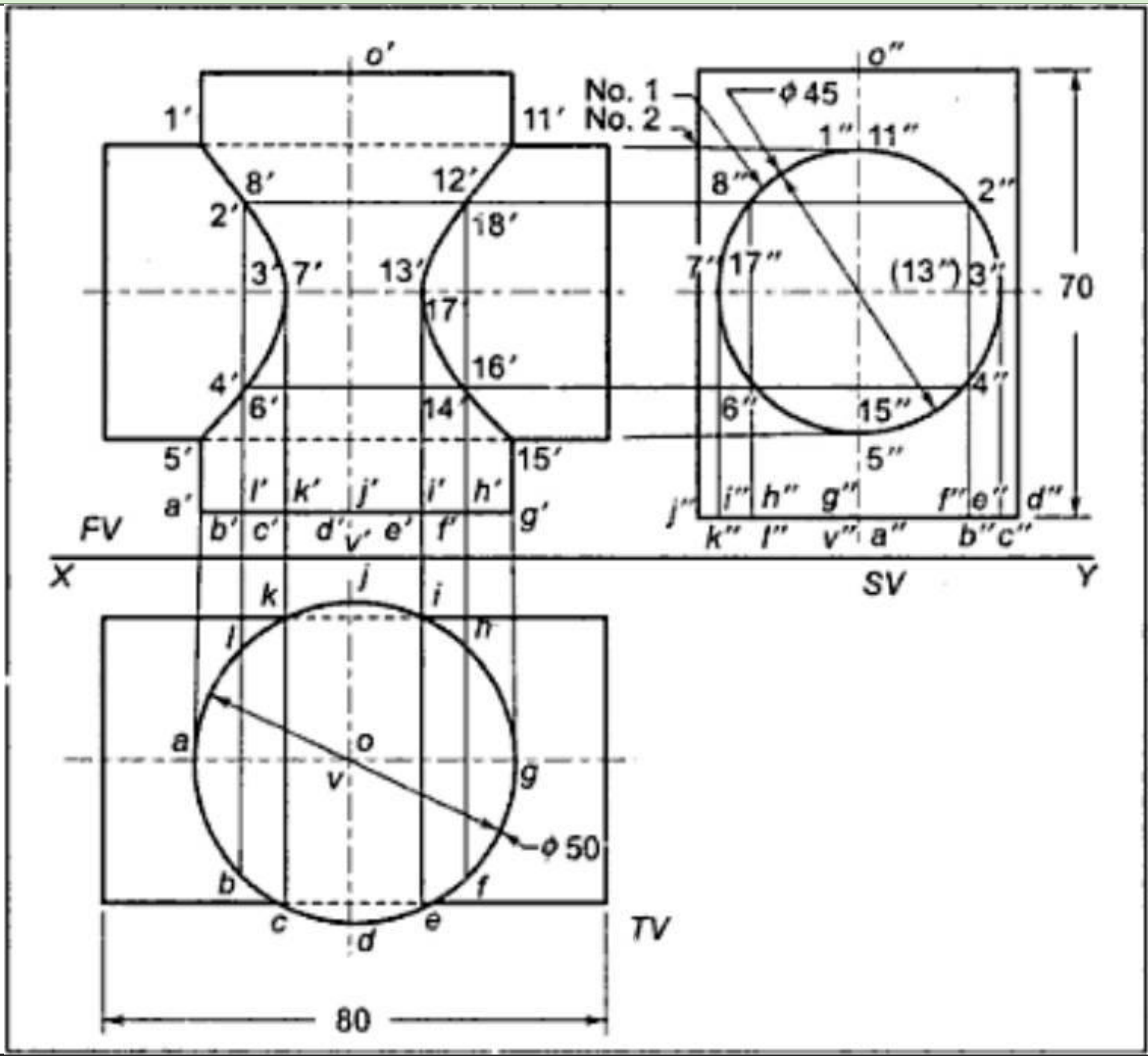
CURVE OF INTERSECTION

Step 5: Obtain the projections of all the points by drawing horizontal lines (as interconnecting projectors between the side view and the front view) and intersecting the concerned surface lines. Join the points so obtained by a thin curved line. There being neither edges on solid 2 nor any comers in the curve of intersection located in the side view, there will be a single continuous curved line obtained in the front view, for each set of points being joined in serial cyclic order.



CURVE OF INTERSECTION

Step 6: Complete the projections by drawing proper conventional lines for all the existing edges and surface boundaries taking due care of visibility.

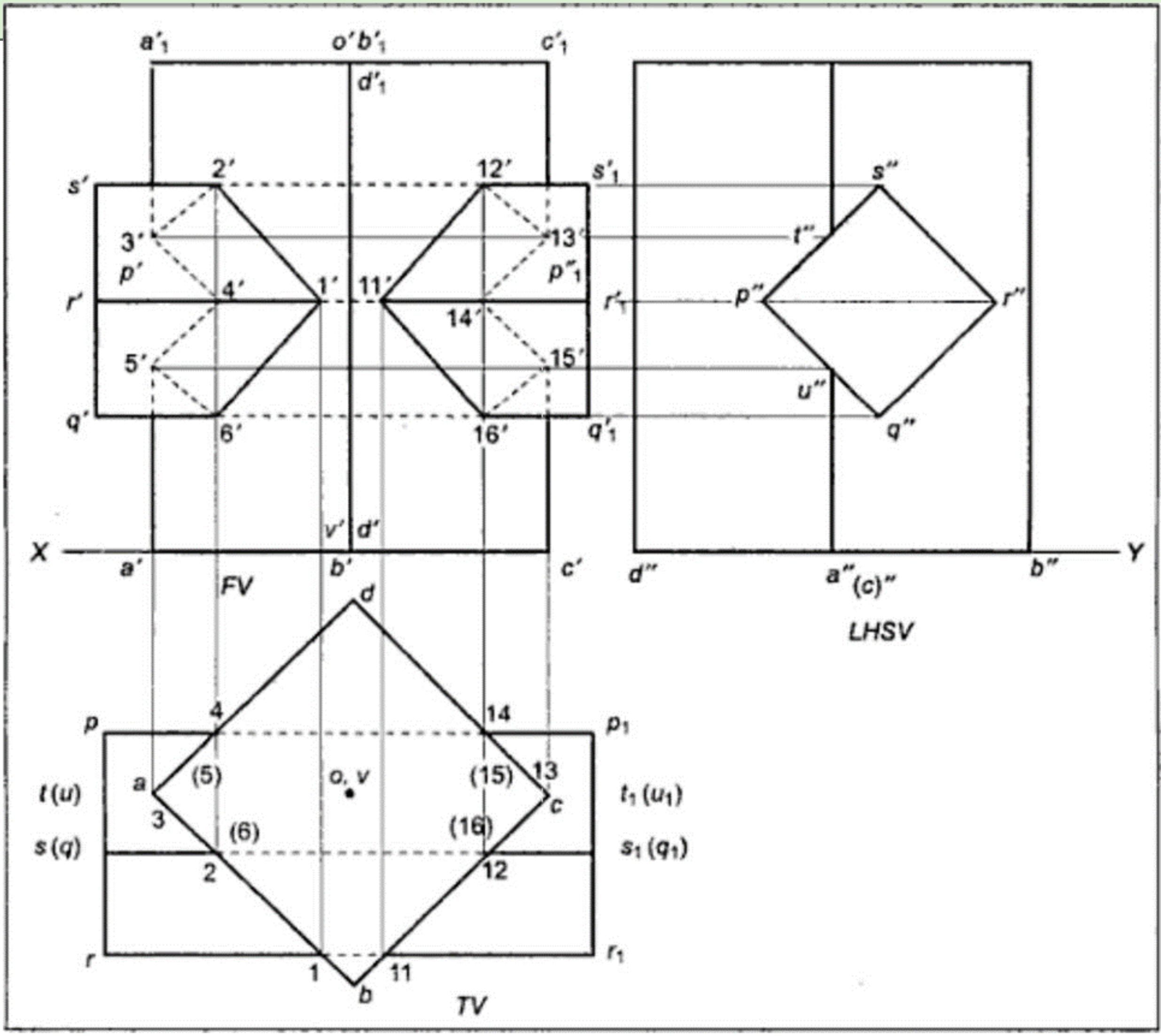


CURVE OF INTERSECTION

PROCEDURE FOR DRAWING LINE OF INTERSECTION OF SURFACES OF TWO SOLIDS BY THE LINE METHOD

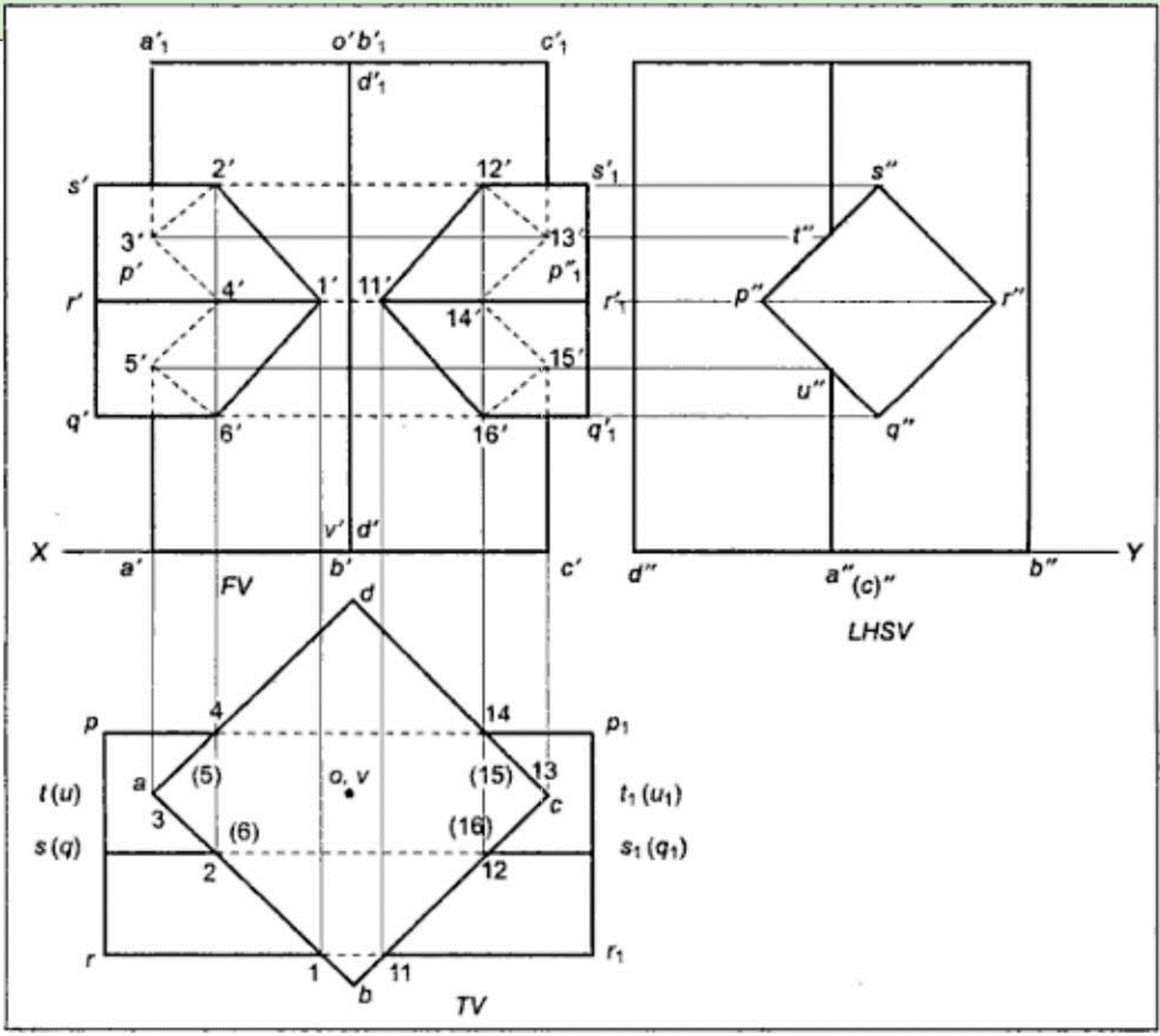
Example 8.2

A vertical square prism with 50 mm side at its base and an axis of 90 mm is standing on its base with side faces equally inclined to the VP. It is completely penetrated by another square prism of 35 mm sides at its base and axis 90 mm long. The axis of the penetrating prism is parallel to both, the HP and the VP, 8 mm in front of the axis of the vertical prism and 45 mm above the base of



CURVE OF INTERSECTION

the vertical. If side faces of the penetrating prism are equally inclined to the VP, draw the projections showing the curves of intersections.



Instructions

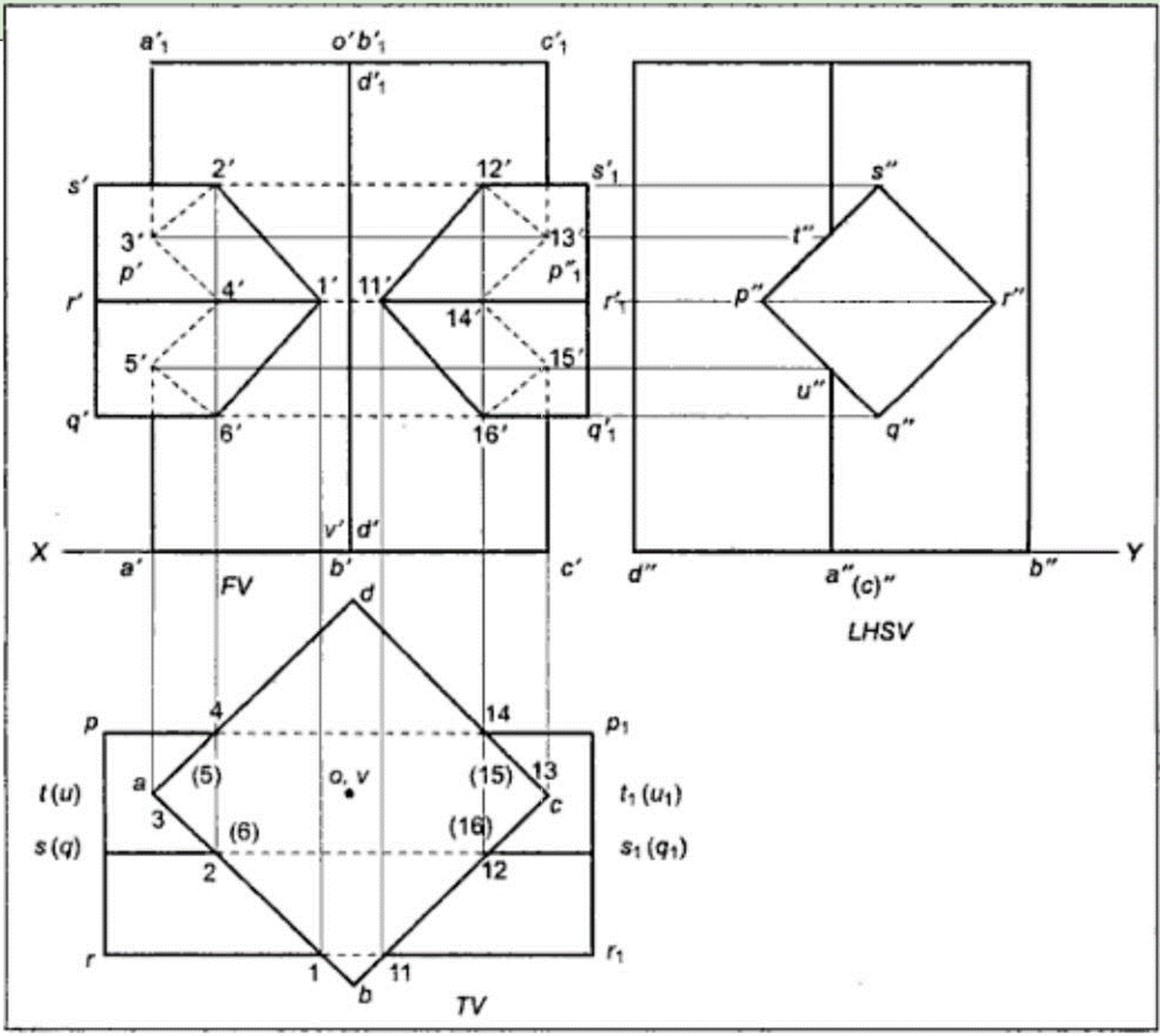
Step 1: The vertical prism will be projected as a square in the top view and as a

horizontal one in the side view. Draw these squares in the top view and side view and then draw rectangle views of both the solids in proper relative positions.

CURVE OF INTERSECTION

Step 2: A vertical prism has the axial view as a square in the top view. The portion of this square within the boundary of the other solid (i.e., within p-Pi-q-r) represents the curve of intersection in that view.

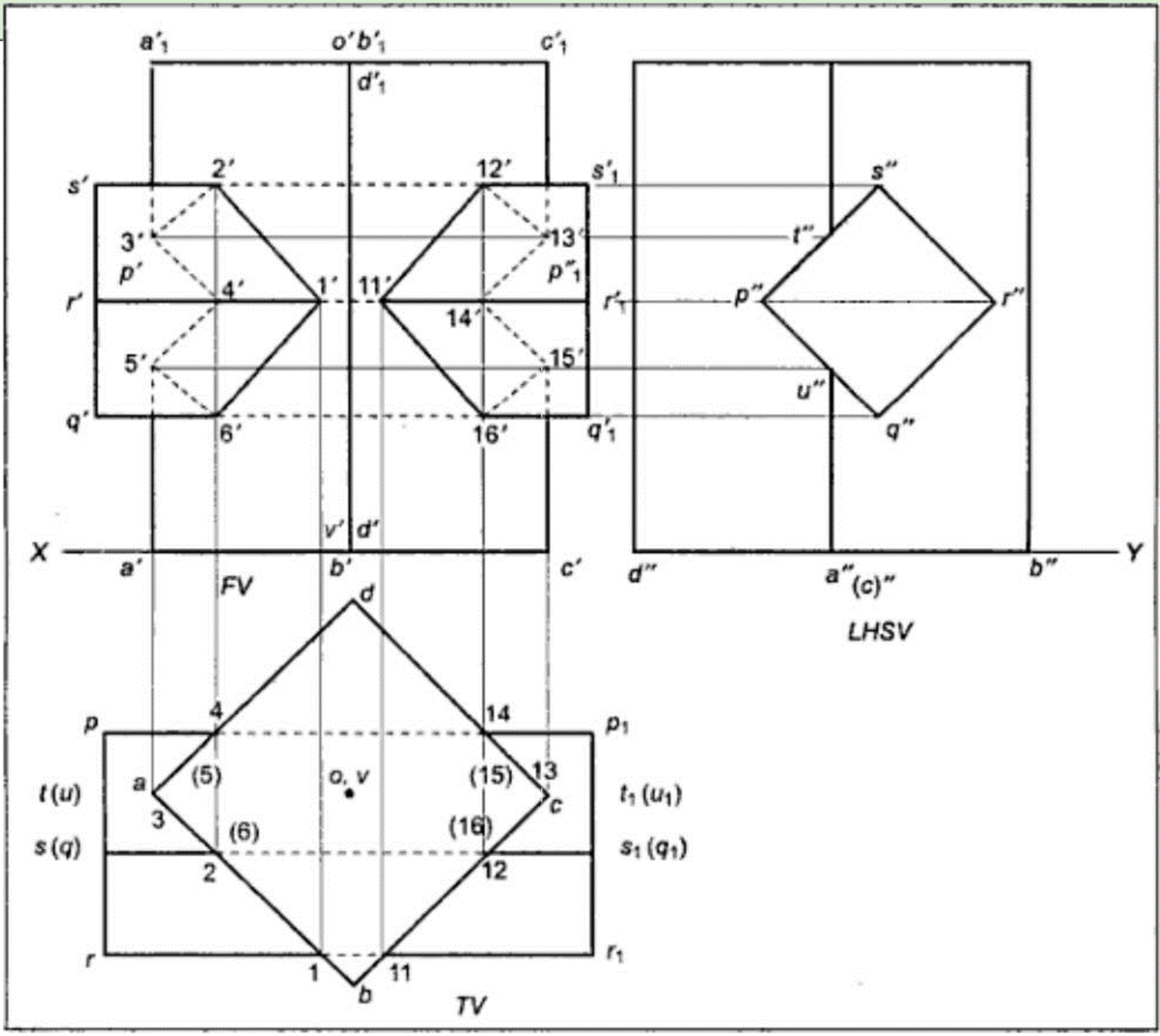
Step 3: The solids being plain solids, draw surface lines passing through critical points only. Points on edges pp_b qq_{lt} rr_v ss_y and the extreme left and right points at comers are critical points. As the edges are already drawn only coinciding lines tt and uu, are required to be drawn passing through



CURVE OF INTERSECTION

comer points. «j is assumed to be a visible surface line between pp, and ss, while uu_y is assumed to be a hidden surface line between pp} and qq_v

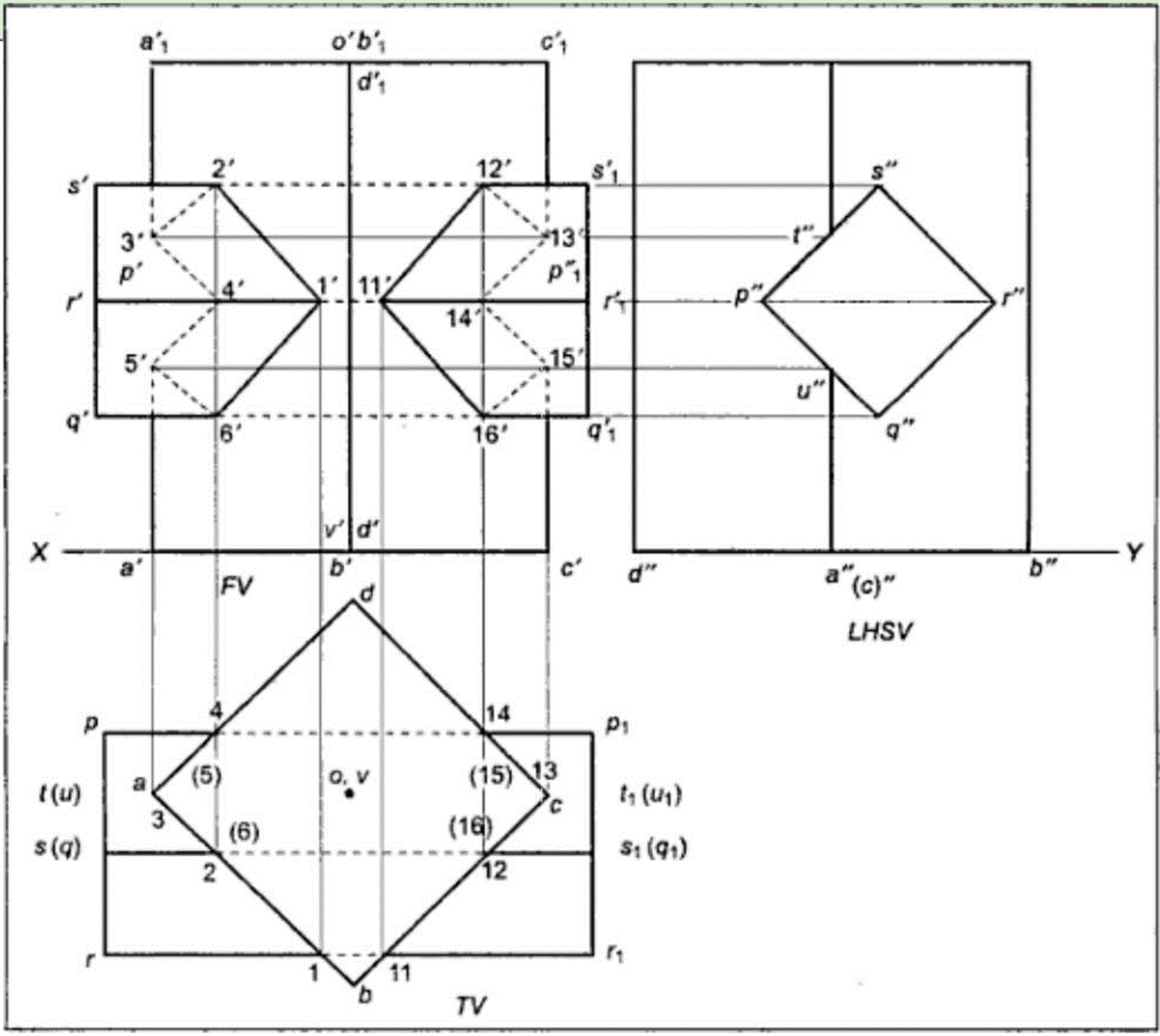
Step 4: Common points between the left part of the curve and the surface lines are numbered 1, 2, 3, 4 on visible lines rr, ss, and pp}, respectively, and as 5 and 6 on hidden surface lines uuy and qq_v. Similarly, points are numbered as 11, 12, 13, and 14 on visible surface lines and 15 and 16 on hidden surface lines. Note that 1-2-3-...6 is made up of straight lines but it is called the curve of intersection.



CURVE OF INTERSECTION

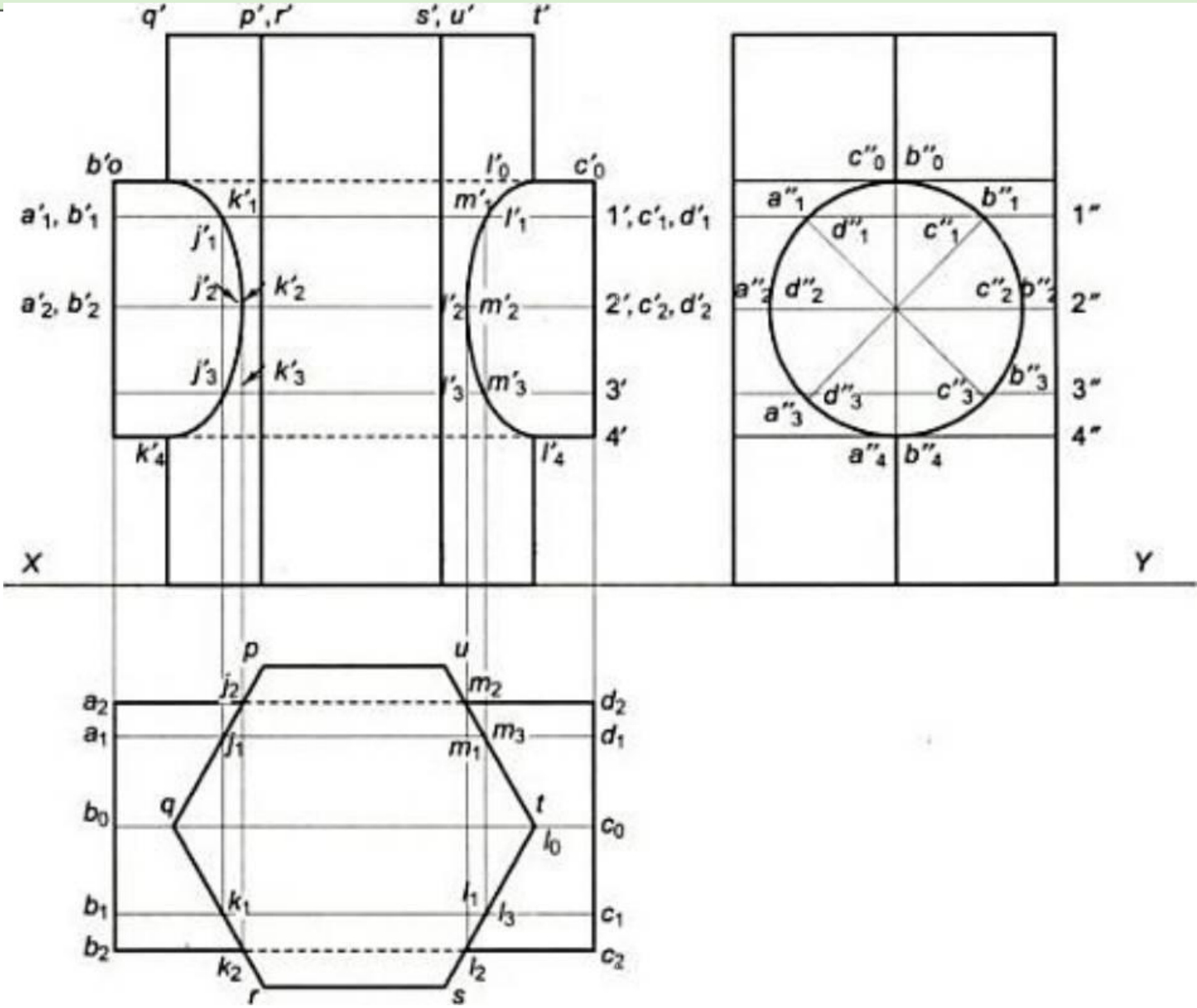
Step 5: The points from the top view are projected in the front view by drawing vertical projectors and intersecting the concerned surface lines. The points so obtained are joined by thin lines in serial cyclic order by straight lines as none of the solids is a curved one.

Step 6: Projections are completed by drawing proper conventional lines, taking due care of visibility.



<u>PROCEDURE FOR DRAWING THE CURVE OF INTERSECTION OF SURFACES IN ORTHOGRAPHIC PROJECTIONS USING THE CUTTING PLANE METHOD</u>

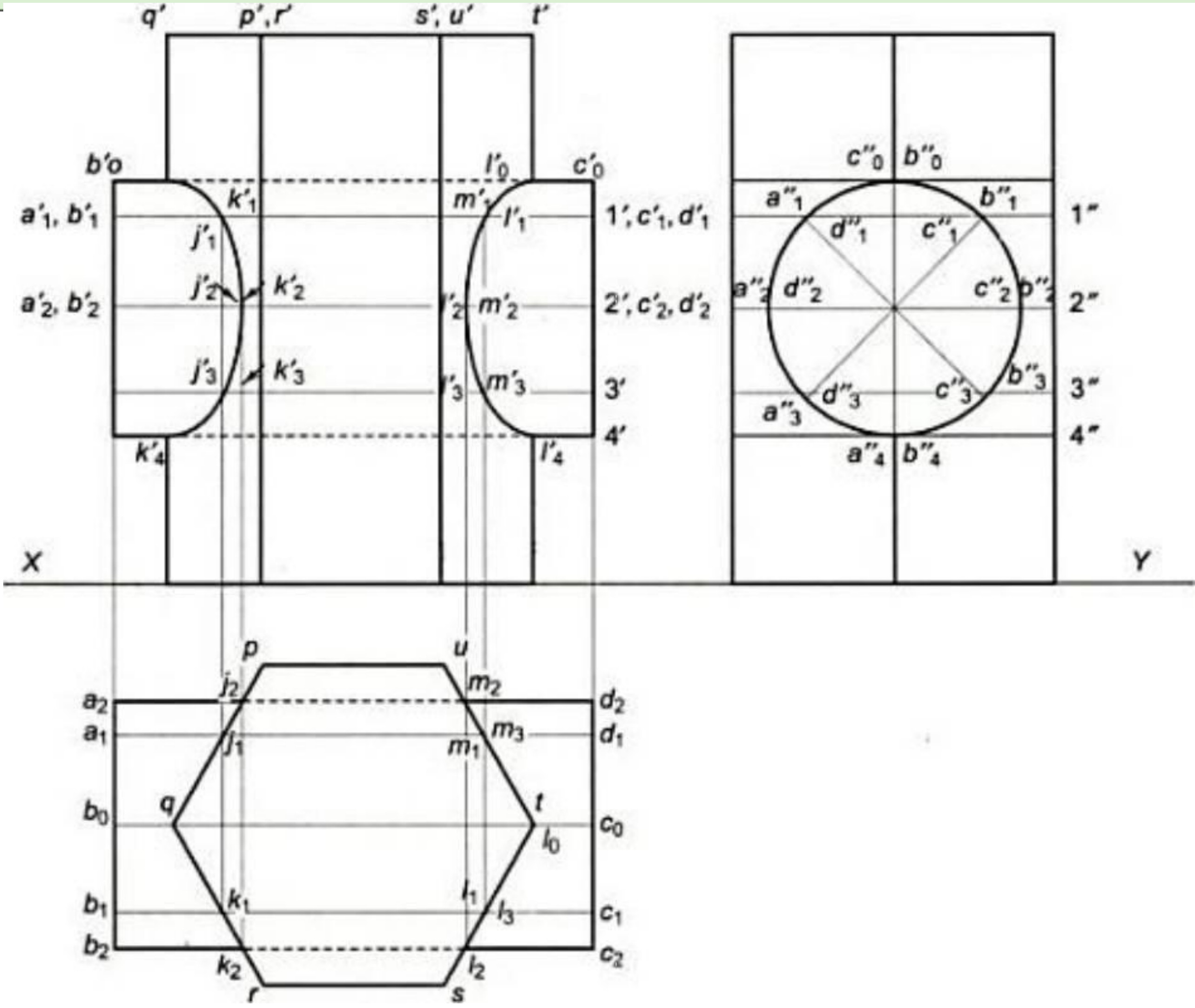
Step I: Draw, using thin lines, the projections of the two given solids in proper relative positions (See Figure). IA vertical hexagonal prism and a penetrating horizontal cylinder are shown as given solids in this figure



CURVE OF INTERSECTION

Step II: Select convenient positions for the cutting planes so that the shape of section for each of the solids could be imagined and easily drawn in at least one of the views. [In Figure, horizontal cutting planes are selected so that the shape of the section is a rectangle for the cylinder, and a hexagon for the prism, in the top view.)

Step III: Draw the shapes of sections of the two solids in any one of the views and locate the points in which these shapes of sections intersect each other. Project these points on the cutting



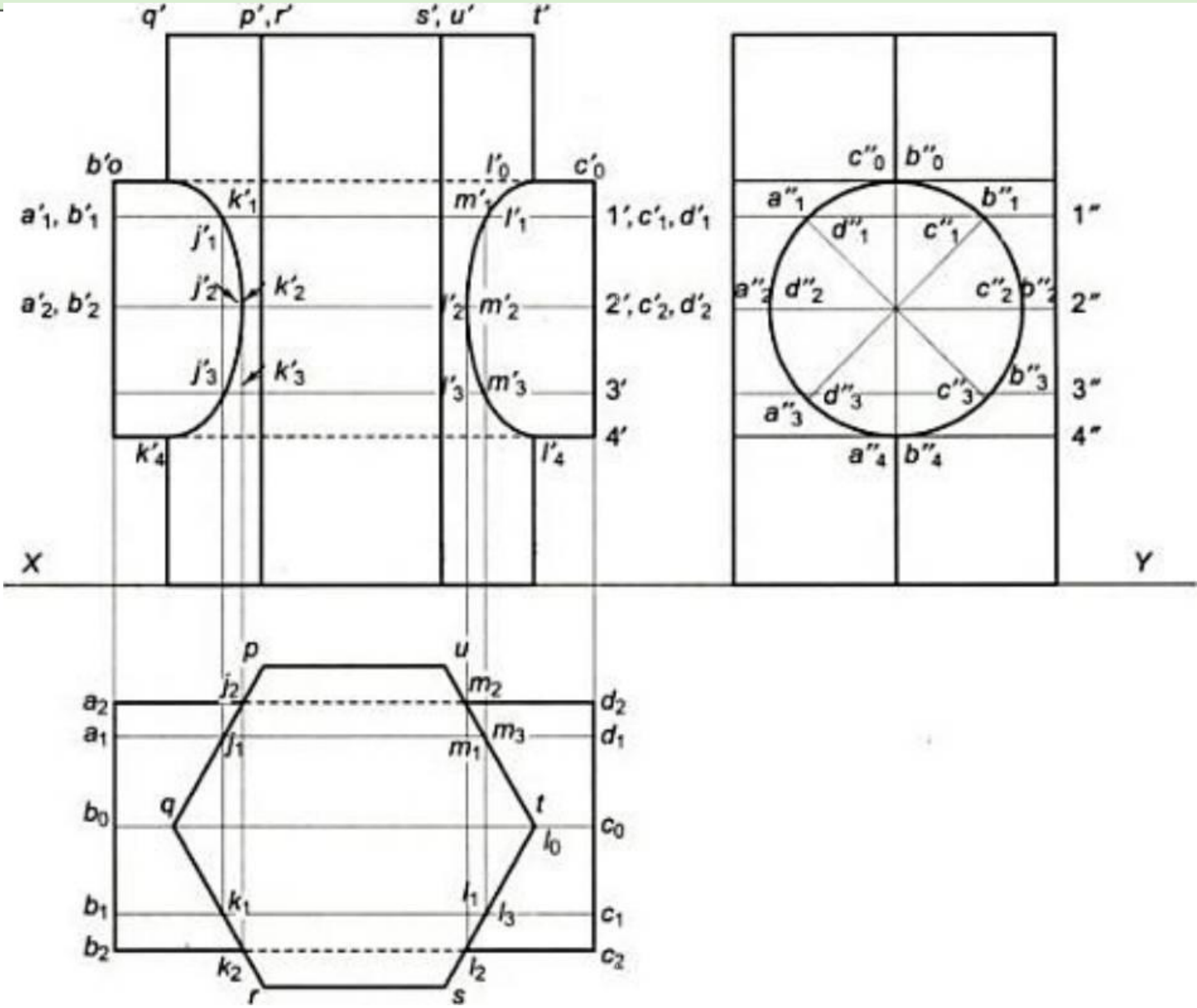
CURVE OF INTERSECTION

line. (In Figure, a number of horizontal cutting planes 1', 2', etc. are drawn in the FV and their shapes are drawn in section in the top view. The points of intersections of the rectangle of the cylinder and the hexagon of the prism, are projected back on the respective cutting pllane lines in the FV.l

Step IV: The points obtained in Step III are joined in proper sequence, which is top to

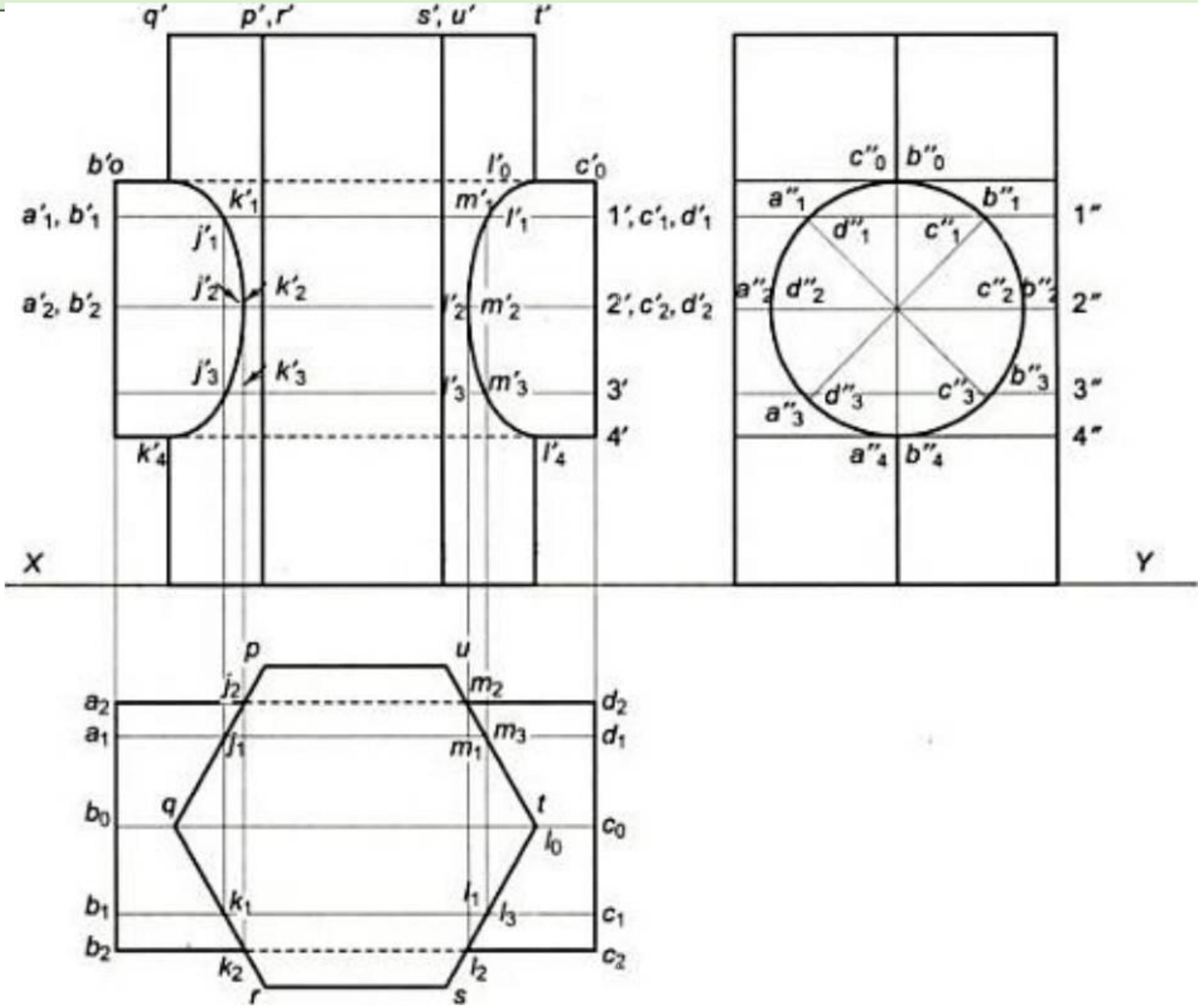
bottom for visible and hidden curve points.

Step V: Complete the projections by drawing proper conventional lines for all the existing edges and



CURVE OF INTERSECTION

Boundaries.



Thank You!

FOR YOUR ATTENTION,CO-OPERATION AND KNOWLEDGE, THANK YOU CLASS … WISH YOU THE BEST OUT THERE!