

MANUAL
For
AutoCAD Laboratory

B.E Mechanical and Production



Mechanical Engineering Department

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Course Outcomes

At the end of the course, the student

1. Will be able to draw isometric and orthogonal projections and sectional views of various mechanical components.
2. Will be able to draw free hand sketches of various mechanical components
3. Will be able to understand the shape and structure of different types of joints, screws, keys and Couplings
4. Will be sufficiently knowledgeable to use both the software and drafter to produce assembly views of various mechanical components from part drawings.

List of Experiments:

I. Machine Drawing (AutoCAD):

1. Format of drawing sheet & title block,
2. Conventions of drawing lines and dimensions,
3. Convention for sectional views.
4. Simple machine elements.
5. Riveted and screwed fastenings.
6. Joints and coupling.

INTRODUCTION TO AUTOCAD

Introduction to Design Cycle

Any engineering idea takes the final shape only after completing a series of design process. Engineering design process involves a number of steps which need not be in a definite order. Its approach varies with individual requirements. A logical common to all design projects are shown in figure 1.

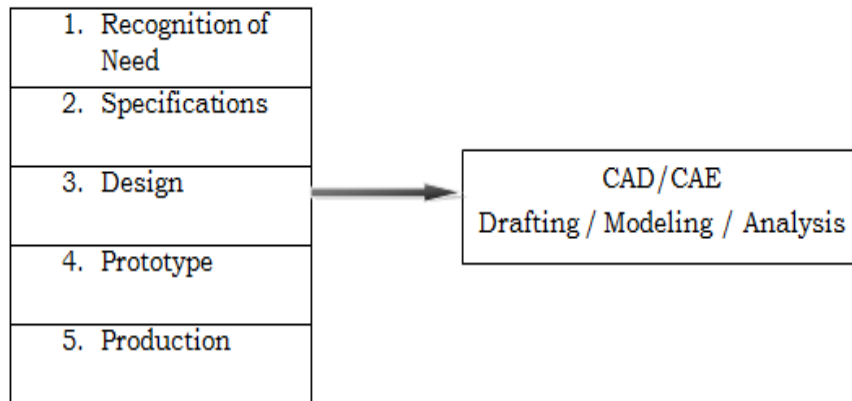


Figure 1: Design Process.

Computer Aided Design and Computer Aided Engineering (CAD/CAE) can be defined as the use of computer systems to assist in the creation, modification, analysis or optimization of a design. The computer systems consist of the hardware and software to perform specialized design functions.

Drafting

Drafting is the technique of generating 2D drawings called sketches. The sketches convey all the information needed to fabricate the design. It is called the language of the engineers.

Modeling

Modeling defines the building of 3D models on the computer screen. 3D models, pictures the design that you see or imagine.

Analysis

Analysis is the process of validating your design. In CAE, we validate the design using the popular Finite Element Analysis (FEA) technique. Various programs are available to check whether your design meets the requirements.

Introduction to AutoCAD

AutoCAD is a PC based 2D and 3D CAD program which greatly reduces the time and effort to create and revise drawings and can also produce accurate drawings faster, and also reuse the information in your drawings.

With its compact and industry specific tools AutoCAD is gaining importance in engineering industries worldwide.

System Requirements

Listed below are the hardware and operating system requirements for running AutoCAD on any system

- Intel i3 processor
- 4GB Ram
- 500GB Hard Disk
- Microsoft Windows 7
- 17'' Monitor
- Keyboard and Optical Mouse

First Look at AutoCAD

Launching AutoCAD is simple. Double click on shortcut icon from the windows desktop. Within seconds you could see the AutoCAD desktop (Graphic User Interface) as shown in figure 2.

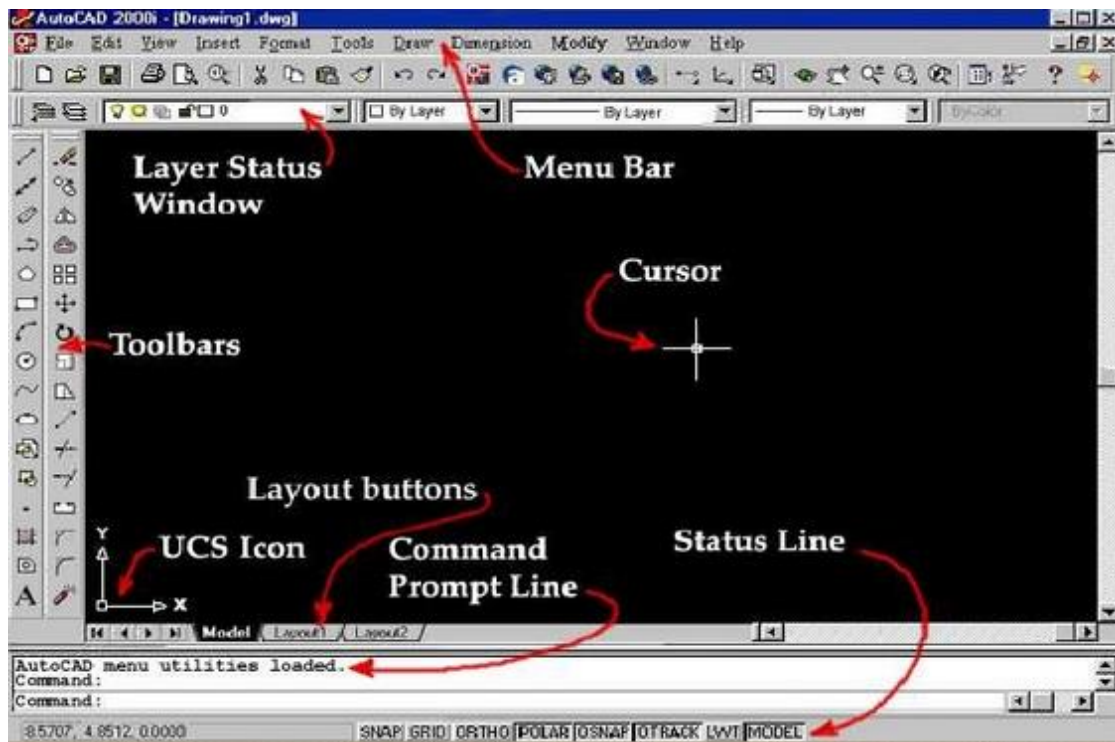


Figure 2: Graphics User Interface in AutoCAD

Command Prompt Line

You can invoke any command by typing it in the command prompt line space provided as shown in figure 3. The message in the command tells you what to do next or they offer a lot of options.

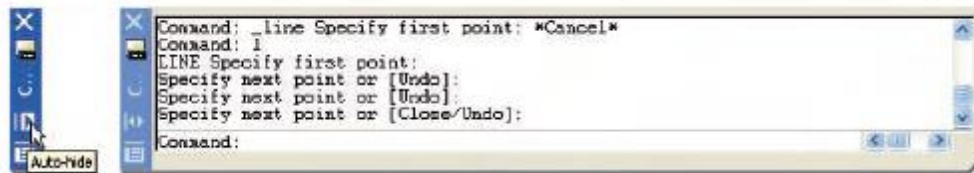


Figure 3: Command Prompt Line / bar

Toolbar

Toolbar menu as shown in figure 4, supplies commands as picture type icons, so you can quickly execute the commands and edit the location and content of commands. Once press the right button of mouse on any toolbar, toolbar list is displayed. Then select the required toolbar that you want to display on the screen.

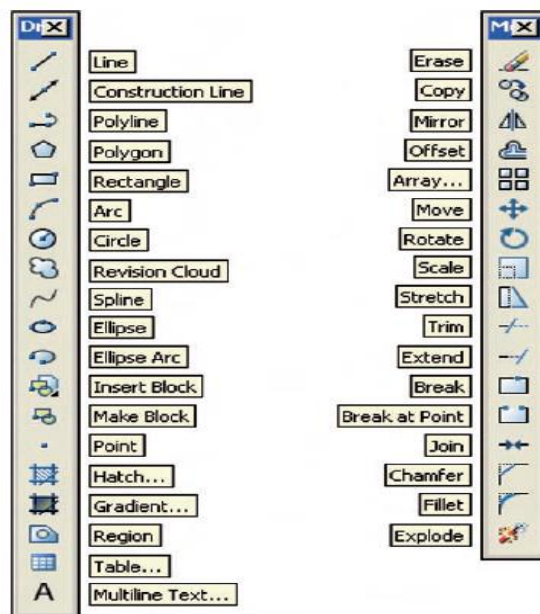


Figure 4: Toolbar

Menu Bar

The menu bar is situated under the title bar and contains names of menus from which commands can be selected. Figure 5 shows the View drop-down menu which appears with a *left click* on the name. *Left-click* 3D Views in the drop-down menu and a sub-menu appear, from which other sub-menus can be selected if required.

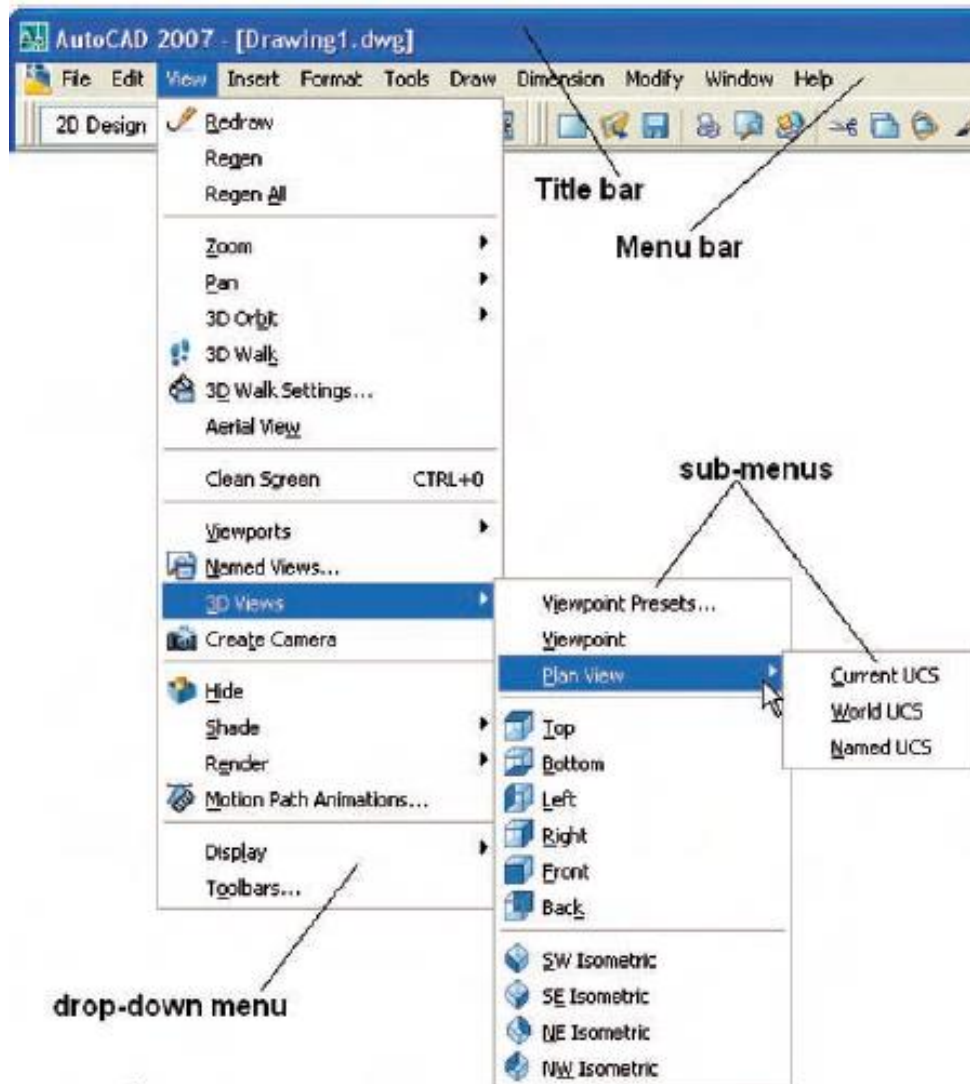


Figure 5: Menu Bar

Status Bar

Status bar as shown in figure 6 shows current working state, SNAP, GRID, ORTHO and coordinate values.



Figure 6: Status Bar

Starting commands using command bar

To invoke a command through the command bar, type a command and press enter. For example, to create circle:

Command: Circle press enter

2point / 3point / PTR / RadTanTan / Arc / Multiple / <Center of circle>:

You can also shorten your command just by entering few characters called hotkeys. Example for circle press ‘c’

Command: ‘c’ press enter

Starting commands using functional keys

Functional keys are assigned to certain commands for quick access. Listed below are the function keys used most often

Function Keys	Function defined in AutoCAD
F1	Online Help
F2	Toggles between command window On and Off
F3	Hide / Unhide the command Bar
F4	Toggles between tablet On and Off
F5	Switches among Isoplane Top, Right and Left
F6	Toggles between Coordinates On and Off
F7	Toggles between Grid On and Off
F8	Toggles between Ortho Mode On and Off
F9	Toggles between Snap Mode On and Off
F10	Hide / Unhide the Status Bar
F11	Toggles between Object Snap Tracking On and Off

Setting up a drawing

The general template settings are

- Setting drawing units
- Setting drawing limits
- Setting layer properties
- Setting line types
- Setting text style

- Setting grid and snap spacing

Setting drawing units

With AutoCAD you can typically draw at full size (1:1) scale, and then set a scale factor when you print or plot your drawing. However, you have to define whether one linear drawing unit represents an inch, a meter, or a mile. For both linear and angular units, you can also set the degree of precision, such as the number of decimal places etc. the precision setting affect only the display of the distances, angles and coordinates.

Units

Format > Drawing Units... the below dial box will open as shown in figure 7.

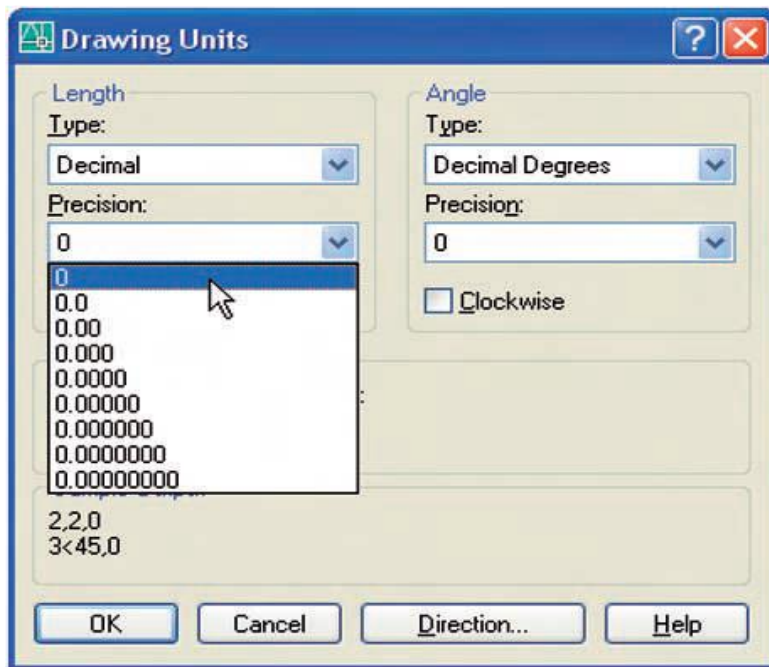


Figure 7: Drawing units dialog box

Setting Drawing limits

Drawing limits form an invincible boundary around the drawing. Setting the drawing limits ensures that you cannot create a drawing larger than what can fit on a specific sheet of paper when printed at a specified scale.

Limits

Format > Drawing limits

Drawing limits are two dimensional points in the world coordinates system (WCS) that represent a lower left limit and an upper right limit. You cannot impose limit on the Z direction. It also determines the area of the drawing that can display grid dots, and the minimum area displayed by ZOOM All.

Command: limits press enter

Limits are off: ON / <lower left corner><0.0000, 0.0000>: use one of the points fixing method or enter an option

Upper right corner <12.0000, 9.0000>: use one of the point fixing methods.

```
Command: limits
Reset Model space limits:
Specify lower left corner or [ON/OFF] <0,0>:
Specify upper right corner <420,297>:
Command:
```

Command: Zoom

```
Command: zoom
Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: a
Regenerating model.
Command:
```

Paper

Setting > Sheet Size

Select the desired paper size and press OK to insert the sheet in the drawing file. To select a custom sheet size, select the user's defined option from the list box and then uncheck the insert sheet check box to specify parameters for size and scale of sheet.

ISO paper sizes (plus rounded inch values)

Format	A series ^[2]		B series ^[3]		C series ^[4]	
	mm × mm	in × in	mm × mm	in × in	mm × mm	in × in
0	841 × 1189	33.11 × 46.81	1000 × 1414	39.37 × 55.67	917 × 1297	36.10 × 51.06
1	594 × 841	23.39 × 33.11	707 × 1000	27.83 × 39.37	648 × 917	25.51 × 36.10
2	420 × 594	16.54 × 23.39	500 × 707	19.69 × 27.83	458 × 648	18.03 × 25.51
3	297 × 420	11.69 × 16.54	353 × 500	13.90 × 19.69	324 × 458	12.76 × 18.03
4	210 × 297	8.27 × 11.69	250 × 353	9.84 × 13.90	229 × 324	9.02 × 12.76
5	148 × 210	5.83 × 8.27	176 × 250	6.93 × 9.84	162 × 229	6.38 × 9.02
6	105 × 148	4.13 × 5.83	125 × 176	4.92 × 6.93	114 × 162	4.49 × 6.38
7	74 × 105	2.91 × 4.13	88 × 125	3.46 × 4.92	81 × 114	3.19 × 4.49
8	52 × 74	2.05 × 2.91	62 × 88	2.44 × 3.46	57 × 81	2.24 × 3.19
9	37 × 52	1.46 × 2.05	44 × 62	1.73 × 2.44	40 × 57	1.57 × 2.24
10	26 × 37	1.02 × 1.46	31 × 44	1.22 × 1.73	28 × 40	1.10 × 1.57

Commands used in AutoCAD

AutoCAD shortcut Keys	
Character	Command
AA	Area
AL	Align
AR	Array
B	Block
BC	BClose
BE	BEdit
BH	Hatch
BO	Boundary
BR	Break

BS	BSave
C	Circle
CH	Properties
-CH	Change
CHA	Chamfer
CHK	Check Standards
COL	Color
CO	Copy
CYL	Cylinder
D	DimStyle

DAL	DimAligned
DAN	DimAngular
DAR	DimArc
DBA	DimBaseLine
DI	Dist
DO	DONUT
DS	Dsettings
DST	DimStyle
DT	Text
DV	DView
E	Erase
EL	Ellipse
ER	External Reference
EX	Extend
Exit	Quit
EXP	Export
EXT	Extrude
F	Fillet
FI	Filter
FSHOT	FlatShot
G	Group
H	Hatch
HE	Hatch Edit

HI	Hide
I	Insert
IM	Image
IMP	Import
IN	Intersect
INF	Interfere
J	Join
L	Line
LA	Layer
LI	List
LO	Layout
LT	Line Type
M	Move
MA	Match Prop
MAT	Materials
ME	Measure
MI	Mirror
ML	MLine
MT	MText
O	Offset
OP	Options
ORBIT	3D Orbit
OS	OSnap

P	Pan
PL	PLine
PO	Point
POL	Polygon
PR	Properties
PRE	Preview
PRINT	Plot
PSOLID	Poly Solid
PYR	Pyramid
R	Redraw
RA	Redraw All
RC	Render Crop
RE	Regen
REA	Regen All
REC	Rectangle
REG	Region
REN	Rename
REV	Revolve
RO	Rotate
RR	Render
S	Stretch
SC	Scale
SEC	Section

SL	Slice
SN	Snap
SO	Solid
SPL	Spline
SPLANE	Section Plane
ST	Style
STA	Standards
SU	Substract
TA	Table
TH	Thickness
TO	Tool Bar
TOL	Tolerance
TOR	Torus
TR	Trim
UC	UCSMAN
UN	Units
UNI	Union
V	View
VP	VPoint
W	WBlock
WE	Wedge
X	Explode

TYPES OF COMMANDS AND ITS PROCEDURE TO USE

1. LINE:

Toolbar: Draw

Command entry: LINE or 'L'

Click Home tab → Draw panel → Line.

Specify the start point.

→ You can use the pointing device or enter coordinate values at the command prompt.

→ Complete the first line segment by specifying the endpoint.

→ To undo the previous line segment during the LINE command, enter 'u' or click 'Undo'.

→ Specify the endpoints of any additional line segments.

→ Press ENTER to end or 'c' to close a series of line segments.

→ To start a new line at the endpoint of the last line drawn, start the LINE command again and press ENTER at the Specify Start Point prompt.

2. CONSTRUCTION LINE (XL):

Menu: Draw → Construction Line

Toolbar: Draw

Command entry: xline or 'xl'.

Specify a point or [Hor/Ver/Ang/Bisect/Offset]: Specify a point or enter an option.

Lines that extend to infinity, such as xlines, can be used to create, construction and reference lines, and for trimming boundaries.

* **Hor:** Creates a horizontal xline passing through a specified point.

Specify through point: Specify the point (1) through which you want the xline to pass, or press ENTER to end the common. The xline is created parallel to the X axis.

* **Ver:** Creates a vertical xline passing through a specified point.

* **Ang:** Creates an xline at a specified angle.

Enter angle of xline (0) or [Reference]: Specify an angle or enter Angle of Xline, Specifies the angle at which to place the line.

* **Specify through point:** Specify the point through which you want the xline to pass.

Enter angle of xline <45>: 'xl' is created through the specified angle.

An xline is created through the specified point, using the specified angle.

* **Reference:** Specifies the angle from a selected reference line.

The angle is measured counterclockwise from the reference line.

Select a line object: Select a line, polyline, ray, or xline

3. POLYGON:

Ribbon: Home tab → Draw panel → PolygonMenu → Draw → Polygon

Toolbar: Draw

Command entry: polygon

→ Enter number of sides <current>: Enter a value between 4 and 1024 or press ENTER

→ Specify center of polygon or [Edge]: Specify a point (1) or enter 'e'
Center of Polygon: Define the center of the polygon.

→ Enter an option [Inscribed in circle/Circumscribed about circle]
<current>: Enter 'I' or 'c' or

* **Inscribed in Circle:** Specifies the radius of a circle on which all vertices of the polygon lie.

→ Specify radius of circle: Specify a point (2) or enter a value specifying the radius with your pointing device determines the rotation and size of the polygon. Specifying the radius with a value draws the bottom edge of the polygon at the current snap rotation angle

* **Edge:** Defines a polygon by specifying the endpoints of the first edge.

→ Specify first endpoint of edge: Specify a point (1)

→ Specify second endpoint of edge: Specify a point (2)

You can specify the different parameters of the polygon including the number of sides. The difference between the inscribed and circumscribed options is shown in the figure.

3. RECTANGLE (REC):

Menu: Draw → Rectangle

Toolbar: Draw

Command entry: **rec**, **rectang** or **rectangle**

Current settings: Rotation = 0,

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]:

Specify a point or enter an option, with this command, you can specify the rectangle parameters (length, width, rotation) and control the type of corners (fillet, chamfer, or square).

→ First Corner Point (0, 0): Specifies a corner point of the rectangle.

→ Specify other corner point or [Area/Dimensions/Rotation]:

→ Specify a point or enter an option

→ Other Corner Point: @100,75. (eg)

Creates a rectangle using the specified points as diagonally opposite corners.

* **Area:** Creates a rectangle using the area and either a length or a width.

If the Chamfer or Fillet option is active, the area includes the effect of the chamfers or fillets on the corners of the rectangle.

→ Enter area of rectangle in current units <100>: Enter a positive value

Calculate rectangle dimensions based on [Length/Width] <Length>: Enter 'L' or 'w'

→ Enter rectangle length <20>: Enter a non-zero value or

→ Enter rectangle width <10>: Enter a non-zero value

→ Specify other corner point or [Area/Dimensions/Rotation]:

→ Move the cursor to display one of four possible locations for the rectangle and click the one that you want

* **Dimensions:** Creates a rectangle using length and width values.

→ Specify length for rectangles <x>: Enter a non-zero value

→ Specify width for rectangles <y>: Enter a non-zero value

→ Specify other corner point or [Area/Dimensions/Rotation]:

→ Move the cursor to display one of four possible locations for the rectangle

And click the one that you want

* **Rotation:** Creates a rectangle at a specified rotation angle.

→ Specify rotation angle or [Points] <0> Specify an angle by entering a value, specifying a point, or entering 'p' and specifying two points

→ Specify other corner point or [Area/Dimensions/Rotation]:

→ Move the cursor to display one of four possible locations for the rectangle

and click the one that you want Chamfer Sets the chamfer distances for the rectangle.

→ Specify first chamfer dist for rectangles <current>: Specify a dist (1) or press ENTER

→ Specify second chamfer dist for rectangles <current>: Specify a dist (2) or press ENTER

The values become the current chamfer distances for subsequent RECTANG commands.

* **Elevation:** Specifies the elevation of the rectangle.

→ Specify the elevation for rectangles <current>: Specify a distance or press ENTER

The value becomes the current elevation for subsequent RECTANG commands.

* **Fillet:** Specifies the fillet radius of the rectangle.

→ Specify fillet radius for rectangles <current>: Specify a distance or press ENTER

The value becomes the current fillet radius for subsequent RECTANG commands.

* **Thickness:** Specifies the thickness of the rectangle.

→ Specify thickness for rectangles <current>: Specify a distance or press ENTER

The value becomes the current thickness for subsequent RECTANG commands.

* **Width:** Specifies the polyline width of the rectangle to be drawn.

→ Specify line width for rectangles <current>: Specify a distance or press ENTER

The value becomes the current polyline width for subsequent RECTANG commands.

4. ARC (A):

Ribbon: Home tab Draw panel → Arc drop-down → 3-Point.

Menu: Draw → Arc → 3 Points

Toolbar: Draw,

Command entry: arc or 'a'

Specify start point of arc or [Center]: Specify a point,

enter c, or press ENTER to start tangent to last line, arc, or polyline

To create an arc, you can specify combinations of

center, endpoint, start point, radius, angle, chord length, and direction values.

→ Start Point, Specifies the starting point of the arc.

→ The endpoint of the last drawn line or arc is used and you are

Immediately prompted to specify the endpoint of the new arc.

This creates an arc tangent to the last drawn line, arc, or polyline.

→ Specify second point of arc or [Center/End]:

→ Second Point draws an arc using three specified points on the arc's circumference.

→ Specify end point of arc: Specify a point (3)

The first point is the start point (1).

The second point (2) is a point on the circumference of the arc

The third point is the endpoint (3).

Note: You can specify a three-point arc either clockwise or counterclockwise.

* **Center:** Specifies the center of the circle of which the arc is a part.

→ Specify center point of arc:

→ Specify end point of arc or [Angle/chord Length]:

* **End Point:** Using the center point (2), draws an arc counterclockwise from the start point (1) to an endpoint that falls on an imaginary ray drawn from the center point through the third point (3). The arc does not necessarily pass through this third point.

* **Angle:** Draws an arc counterclockwise from the start point (1), using a center point (2) with a specified included angle. If the angle is negative, a clockwise arc is drawn.

Specify included angle: →Specify an angle (45⁰)

* **Chord Length:** Draws either a minor or a major arc based on the distance of a straight line between the start point and endpoint.

Note: If the chord length is positive, the minor arc is drawn counterclockwise from the start point, If the chord length is negative, the major arc is drawn counterclockwise.

Specify length of chord: Specify a length.

* **End:** Specifies the endpoint of the arc.

Specify end point of arc:

→ Specify center point of arc or [Angle/Direction/Radius]:

* **Center Point:** Draws an arc counterclockwise from the start point (1).

to an endpoint that falls on an imaginary ray drawn from the center point (3), through the second point specified (2).

* **Angle:** Draws an arc counterclockwise from the start point (1) to an endpoint (2), with a specified included angle. If the angle is negative, a clockwise arc is drawn.

Specify included angle: Enter an angle in degrees or specify an angle by moving the pointing device counterclockwise

* **Direction:** Begins the arc tangent to a specified direction. It creates any arc, major or minor, clockwise or counterclockwise, beginning with the start point (1), and ending at an endpoint (2). The direction is determined from the start point.

* **Radius:** Draws the minor arc counterclockwise from the start point (1) to the endpoint (2). If the radius is negative, the major arc is drawn.

→ Specify radius of arc:

* **Center, Start, and End:** Specifies the center of the circle of which the arc is a part.

→ Specify center point of arc:

→ Specify start point of arc:

→ Specify end point of arc or [Angle/chord Length]:

* **End Point:** Draws an arc counterclockwise from the start point (2), to an endpoint that falls on an imaginary ray drawn from the center point (1) through a specified point (3).

* **Angle:** Draws an arc counterclockwise from the start point (2), using a

center point (1) with a specified included angle. If the angle is negative, a clockwise arc is drawn.

→ Specify included angle:

* **Chord Length:** Specify length of chord: Draws either a minor or a major arc based on the distance of a straight line between the start point and endpoint.

5. CIRCLE:

Ribbon: Home tab → Draw panel → Circle drop-down, (Center, Radius

Menu: Draw → Circle → Center, Radius

Toolbar: Draw

Command entry: circle or 'C'

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: Specify a point or enter an option

→ Center Point draws a circle based on a center point and a diameter or a radius.

→ Specify radius of circle or [Diameter]: Specify a point, enter a value, enter d, or press ↵

→ Radius, defines the radius of the circle. Enter a value, or specify a point.

→ Diameter, defines the diameter of the circle. Enter a value, or specify a second point.

→ Specify diameter of circle <current>: Specify a point (2), enter a value, or press Enter

* **3P (Three Points):3P:** Draws a circle based on three points on the circumference.

→ Specify first point on circle: Specify a point (1)

→ Specify second point on circle: Specify a point (2)

→ Specify third point on circle: Specify a point (3)

* **2P (Two Points): 2P:** Draws a circle based on two endpoints of the diameter.

→ Specify first endpoint of circle's diameter: Specify a point (1)

→ Specify second endpoint of circle's diameter: Specify a point (2)

* **TTR (Tan, Tan, Rad) Ttr:** Draws a circle with a specified radius tangent to two objects.

→ Specify point on object for first tangent of circle: Select a circle, arc, or line

→ Specify point on object for second tangent of circle: Select a circle, arc, or line

→ Specify radius of circle <current>:

6. REVISION CLOUD:

Ribbon: Home tab → Draw panel → Revision Cloud

Menu: Draw → Revision Cloud

Toolbar: Draw

Command entry: revcloud

Minimum arc length: 0.5000, Maximum arc length: 0.5000

Specify st point or [Arc length/Object/Style] <Object>: To draw, enter an option, or press ↵

→ Guide crosshairs along cloud path, when the start and end lines meet, the following message is displayed at the Command prompt. Revision cloud finished, the resulting object is a polyline.

* **Arc Length:** Specifies the length of the arcs in a revision cloud.

→ Specify minimum length of arc <0.5000>: Specify a minimum arc length

→ Specify maximum length of arc <0.5000>: Specify a maximum arc length

→ Guide crosshairs along cloud path... Revision cloud finished

7. SPLINE:

Ribbon: Home tab → Draw panel → Spline

Menu: Draw → Spline **Toolbar:** Draw

Command entry: Spline

Specify first point or [Object]: Specify a point , You can control the maximum distance between the B-spline curve and the fit points.

* **First Point:** Creates a spline using points you specify.

→ Specify next point: Specify a point

→ Enter points until you have defined the spline curve.

After you enter two points, the following prompt is displayed:

→ Specify next point or [Close/Fit Tolerance] <Start tangent>:

Specify a point, enter an option, or press Enter

* **Next Point:** Continuing to enter points adds additional spline curve segments until you press Enter. Enter undo to remove the last point specified. Once you press Enter, you are prompted to specify the start tangent for the spline curve.

* **Close:** Closes the spline curve by defining the last point as co-incident with the first and making it tangent to the joint.

8. ELLIPSE:

Ribbon: Home tab

→ Draw panel → Ellipse drop-down → Center

Menu: Draw → Ellipse → Center

Toolbar: Draw

Command entry: Ellipse

Specify axis endpoint of ellipse or [Arc/Center/Isocircle]: Specify a point or enter an option

→ The first two points of the ellipse determine the location and length of the first axis.

→ The third point determines the dist between the center and the end point of the IInd axis.

* **Axis Endpoint:** Defines the first axis by its two endpoints. The angle of the first axis determines the angle of the ellipse.

→ The first axis can define either the major or the minor axis of the ellipse.

→ Specify other endpoint of axis: Specify a point (2)

→ Specify distance to other axis or [Rotation]: Specify a distance by entering a value or locating a point (3), or enter r

* **Distance to Other Axis:** Defines the second axis using the dist from the midpoint of the first axis to the endpoint of the second axis (3).

* **Rotation:** Creates the ellipse by appearing to rotate a circle about the first axis.

Specify rotation around major axis: Specify a point (3), or enter an positive angle value < 90⁰

* **Ellipse Arc:** Creates an elliptical arc. And the procedure is same as ellipse.

→ Axis Endpoint: Defines the start point of the first axis.

→ Specify other endpoint of axis:

→ Specify distance to other axis or [Rotation]: Specify a distance or enter 'r'

* **Center:** Creates the elliptical arc using a center point you specify.

→ Specify center of elliptical arc:

→ Specify endpoint of axis: Specify dist to other axis or [Rotation]: Specify a dist or enter r

* **Distance to Other Axis:** Defines the second axis as the distance from the Center of the elliptical arc, or midpoint of the first axis, to the point you specify.

→ Specify start angle or [Parameter]: Specify a point (1), enter a value, or enter p

* **Isocircle:** Creates an isometric circle in the current isometric drawing plane.

→ Specify center of isocircle: → Specify radius of isocircle or [Diameter]:

Specify a distance or enter 'd'

Radius: Creates a circle using a radius you specify.

Diameter: Creates a circle using a diameter you specify.

→ Specify diameter of isocircle: Specify a distance

Note: The Isocircle option is available only when you set the Style option of SNAP to Isometric.

9. MAKE OR CREATE BLOCK:

Ribbon: Home tab → Block panel → Create

Menu: Draw → Make Block

Toolbar: Draw

Command entry: Block

Defines and names a block.

* **Name:** Names the block. The name can have up to 255 characters and can include letters, numbers, blank spaces, and any special character not used.

→ The block name and definition are saved in the current drawing.

Note: You cannot use **DIRECT, LIGHT, AVE_RENDER, RM_SDB, SH_SPOT, and OVERHEAD** as valid names.

* **Preview:** If an existing block is selected under Name, displays a preview of the block.

* **Base Point:** Specifies an insertion base point for the block. The default value is 0, 0 and 0.

* **Specify On-Screen:** Prompts you to specify the base point when the dialog box is closed.

* **Pick Insertion Base Point:**

X: Specifies the X coordinate value.

Y: Specifies the Y coordinate value.

Z: Specifies the Z coordinate value.

* **Objects:** Specifies the objects to include in the new block and whether to retain or delete the selected objects or convert them to a block.

* **Specify On-Screen:** Prompts you to specify the objects when the dialog box is closed.

- * **Select Objects:** Closes the Block Definition dialog box temporarily while you select the objects for the block. When you finish selecting objects, press Enter to redisplay the Block Definition dialog box.
- * **Quick Select:** Displays the Quick Select dialog box, which defines a selection set.
- * **Retain:** Retains the selected objects as distinct objects in the drawing after you create.
- * **Convert to Block:** Converts the selected objects to a block instance in the drawing after you create the block.
- * **Delete:** Deletes the selected objects from the drawing after you create the block.
- * **Objects Selected:** Displays the number of selected objects.

10 . INSERT BLOCK (I):

Ribbon: Home tab → Block panel → Insert

Menu: Insert Block

Toolbar: Insert

Command entry: insert or 'I'

Specifies the name and position of the block or drawing to insert.

The last block you insert during the current editing session becomes the default block for subsequent uses of INSERT.

Note: The position of the inserted block depends on the orientation of the UCS.

- * **Name:** Specifies the name of a block to insert, or the name of a file to insert as a block.
- * **Browse:** Opens the Select Drawing File dialog box (a standard file selection dialog box) where you can select a block or drawing file to insert.
- * **Path:** Specifies the path to the block.
- * **Preview:** Displays a preview of the specified block to insert.
 - A lightning bolt icon in the lower-right corner of the preview indicates that the block is dynamic. And a icon indicates that the block is annotative.
- * **Insertion Point:** Specifies the insertion point for the block.
 - Specify On-Screen: Specifies the insertion point of the block using the pointing device.
 - X:Sets the X coordinate value.

→ Y:Sets the Y coordinate value.

→ Z:Sets the Z coordinate value.

* **Scale:** Specifies the scale for the inserted block. Specifying negative values for the X, Y, and Z scale factors inserts a mirror image of a block.

* **Specify On-Screen:** Specifies the scale of the block using the pointing device.

→ X:Sets the X scale factor.

→ Y:Sets the Y scale factor.

→ Z:Sets the Z scale factor.

* **Uniform Scale:** Specifies a single scale value for X, Y, and Z coordinates.

A value specified for X is also reflected in the Y and Z values.

* **Rotation:** Specifies the rotation angle for the inserted block in the current UCS.

Specify On-Screen: Specifies the rotation of the block using the pointing device.

* **Angle:** Sets a rotation angle for the inserted block.

* **Block Unit:** Displays information about the block units.

* **Unit:** Specifies the INSUNITS value for the inserted block.

* **Factor:** Displays the unit scale factor, which is calculated based on the INSUNITS value of the block and the drawing units.

* **Explode:** Explodes the block and inserts the individual parts of the block.

When Explode is selected, you can specify a uniform scale factor only.

11 . POINT:

Ribbon: Home tab → Draw panel → Point drop-down → Multiple Points

Toolbar: Draw

Menu: Draw → Point → Single Point

Command entry: point or 'P'

Specify a point: Points can act as nodes to which you can snap objects.

→ You can specify a full three-dimensional location for a point. The current elevation is assumed if you omit the Z coordinate value.

→ Specifying the value 32, 64, or 96 selects a shape to draw around the point, in addition to the figure drawn through it:

12 . HATCH & GRADIENT:**Toolbar:** Draw**Command entry:** Hatch or 'h'

Defines the boundaries, pattern type, pattern properties, and other parameters for hatch and gradient fill objects.

* **Add Pick Points:** Determines a boundary from existing objects that form an enclosed area around the specified point Pick internal point or [Select objects/remove Boundaries]:(Click within the area to be hatched or filled, specify an option, enter u or undo to undo the last selection, or press ENTER to return to the dialog box, While picking internal points, you can right-click in the drawing area at any time to display a shortcut menu that contains several options. If you turn on Island Detection, objects that enclose areas within the outermost boundary are detected as islands. How HATCH detects objects using this option depends on which island detection method you select in the More Options area of the dialog box.)

Note: Displays red circles on unconnected endpoints when a boundary cannot be determined.

* **Add Select Objects:** Determines a boundary from selected objects that form an enclosed area. The dialog box closes temporarily, and you are prompted to select objects.

Select objects or [pick internal point/remove Boundaries]:

Select objects that define the area to be hatched or filled, specify an option, →enter 'u' or undo to undo the last selection, or press ENTER to return to the dialog box.

When you use the select Objects option, HATCH does not detect interior objects automatically. You must select the objects within the selected boundary to hatch or fill those objects according to the current island detection style. Each time you click Select Objects, HATCH clears the previous selection set. While selecting objects, you can right-click at any time in the drawing area to display a shortcut menu.

You can undo the last selection or all selections, change the selection method, change the island detection style, or preview the hatch or gradient fill.

* **Remove Boundaries:** Removes from the boundary definition any of the objects that were added previously.

When you click Remove Boundaries, the dialog box closes temporarily, and displays a Command prompt.

Select Objects or [Add boundaries]: Select objects to be removed from the boundary definition, specify an option, or press ENTER to return to the dialog box

* **Select Objects:** Removes temporary boundary objects for hatching as you select them.

* **Add Boundaries:** Adds temporary boundary objects for hatching as you select them.

* **Recreate Boundary:** Creates a polyline or region around the selected hatch or fill, and optionally associates the hatch object with it.

When you click Recreate Boundary, the dialog box closes temporarily, and displays a Command prompt.

Enter type of boundary object [Region/Polyline] <current>: Enter 'r' to create a region or 'p' to create a polyline, Re associate hatch with new boundary? [Yes/No] <current>: Enter 'y' or 'n'

* **Annotative:** Specifies that the hatch is annotative. Click the information Icon to learn more about annotative objects.

* **Associative:** Controls whether the hatch or fill is associative or non associative. A hatch or fill that is associative is updated when you modify its boundaries.

* **Create Separate Hatches:** Controls whether a single hatch object or multiple hatch objects are created when several separate closed boundaries are specified.

* **Draw Order:** Assigns the draw order to a hatch or fill. You can place a hatch or fill behind all other objects, in front of all other objects, behind the hatch boundary, or in front of the hatch boundary.

* **Inherit Properties:** Hatches or fills specified boundaries using the hatch or fill properties of a selected hatch object. HPINHERIT controls whether the hatch origin of the resulting hatch is determined by HPORIGIN or by the source object. After selecting the hatch object whose properties you want the hatch to inherit, you can right-click in the drawing area and use the options on the shortcut menu to switch between the Select Objects and Pick Internal Point options to create boundaries.

When you click Inherit Properties, the dialog box closes temporarily, and displays a Command prompt.

* **Select hatch object:** Click within a hatched or filled area to select the hatch whose properties are to be used for the new hatch object

* **Preview:** Dismisses the dialog box and displays the currently defined boundaries with the current hatch or fill settings. Click in the drawing or press ESC to return to the dialog box. Right-click or press, ENTER to accept the hatch or fill. This option is not available when you have not yet specified points or selected objects to define your boundaries.

13 .REGION:

Ribbon: Home tab→ Draw panel→ Region**Menu:** Draw → Region

Toolbar: Draw

Command

entry: Region or 'r' or 'reg'

→ Select objects: Use an object selection method and press ENTER when you finishRegions are two-dimensional areas you create from closed shapes or loops. Closed polylines, lines, and curves are valid selections. Curves include circular arcs, circles, elliptical arcs, ellipses, and splines. You can combine several regions into a single, complex region.

14 .TABLE:

Ribbon: Home tab→ Annotation panel→ Table

Menu: Draw → Table

Toolbar: Draw

Command entry: Table

Summary: A table is a compound object that contains data in rows and columns.

Note: It can be created from an empty table or a table style. A table can also be linked to data in a Microsoft Excel spreadsheet.

The Insert Table dialog box is displayed. If you select a table cell when the ribbon is active, the Table ribbon contextual tabdisplays. If you enter -table at the Command prompt, options are displayed.

15 .MULTILINE TEXT:

Ribbon: Home tab,Annotation panel → Multiline Text drop-down→ Single Line

Menu: Draw → Text → Single Line Text

Command entry: text or 't'

Current text style: <current> Current text height: <current>Annotative: <current>

Specify start point of text or [Justify/Style]: Specify a point or enter an option

Start Point: Specifies a start point for the text object.

Specify height <current>: Specify a point (1), enter a value, or press ENTER

The Specify Height prompt is displayed only if the current text style is not annotative and does not have a fixed height.

Specify paper text height <current>: Specify a height, or press ENTER

The Specify Paper Text Height prompt is displayed only if the current text style is annotative.

Specify rotation angle of text <current>: Specify an angle or press ENTER

Enter text in the In-Place Text Editor for single-line text.

* **Justify**: Controls justification of the text.

Enter an option [Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR]:

You can also enter any of these options at the Specify Start Point of Text prompt.

* **Align**: Specifies both text's height and orientation by designating the endpoints of the baseline.

→ Specify first endpoint of text baseline: Specify a point (1)

→ Specify second endpoint of text baseline: Specify a point (2)

→ Enter text in the In-Place Text Editor for single-line text.

The size of the characters adjusts in proportion to their height. The longer the text string, the shorter the characters.

* **Fit**: Specifies that text fits within an area and at an orientation defined with two points and a height. Available for horizontally oriented text only.

→ Specify first endpoint of text baseline: Specify a point (1)

→ Specify second endpoint of text baseline: Specify a point (2)

→ Specify height <current>: Enter text in the In-Place Text Editor for single-line text.

* **Center**: Aligns text from the horizontal center of the baseline, which you specify with a point.

→ Specify center point of text: Specify a point (1)

→ Specify height <current>:

→ Specify rotation angle of text <current>:

Enter text in the In-Place Text Editor for single-line text.

* **Style:** Specifies the text style, which determines the appearance of the text characters. Text you create uses the current text style.

→ Enter style name or <current>: Enter a text style name or enter to list all text styles

→ Entering lists the current text styles, associated font files, height, and other parameters.

16 . ERASER (E):→ Ribbon

Home tab→ Modify panel → Erase :

Modify → Erase

Toolbar: Modify

Shortcut menu: Select the objects to erase, right-click in the drawing area, and click erase

Command entry: erase or 'e'

Instead of selecting objects to erase, you can enter an option, such as 'L' to erase the last object drawn, 'p' to erase the previous selection set, or ALL to erase all objects. You can also enter? to get a list of all options.

17 . COPY : (CO)→ Ribbon

Home tab → Modify panel → Copy

Menu: Modify → Copy

Toolbar→ Ribbon → Modify

Shortcut menu: Select the objects to copy, and right-click in the drawing area. Click Copy Selection.

Command entry: copy or 'co'

Select objects: Use an object selection method and press Enter when you finish

Current settings: Copy mode = current

- Specify base point or [Displacement/mode/Multiple] <Displacement>:
Specify a base point or enter an option.
- The two points you specify define a vector that indicates how far the copied objects are to be moved and in what direction.
- If you press Enter at the Specify Second Point prompt, the first point is interpreted as a relative X, Y, Z displacement. For example, if you specify 2,3 for the base point and press Enter at the next prompt, the objects are copied 2 units in the X direction and 3 units in the Y direction from their current position.
- The COPY command repeats automatically by default. To exit the command, press Enter.
- With the COPYMODE system variable, you can control whether multiple copies are created automatically.
- Displacement: Specifies a relative distance and direction using coordinates.
Specify displacement <last value>: Enter coordinates to represent a vector
- * **Mode:** Controls whether the command repeats automatically. This setting is controlled by the COPYMODE system variable.
Enter a copy mode option [Single/Multiple] <current>:Enter 's' or 'm'
- * **Multiple:** Overrides the Single mode setting. The COPY command is set to repeat automatically for the duration of the command. This setting is controlled by the COPYMODE system variable.
- * **Note:** This option is displayed only when copy mode is set to Single

18 . MIRROR (MI):Ribbon → Home tab → Modify

Menu → Modify → Mirror Not available in menus in the current workspace

Toolbar → Modify

Command entry → mirror or 'mi'

Select objects: Use an object selection method and press Enter to finish

- * **Specify first point of mirror line:** Specify a point
- * **Specify second point of mirror line:** Specify a point
- * The two specified points become the endpoints of a line about which the selected objects are mirrored. For mirroring in 3D, this line defines a mirroring plane perpendicular to the XY

plane of the user coordinate system (UCS) containing the mirror line.

* **Erase source objects?** [Yes/No] <N>: Enter 'y' or 'n', or press Enter

Yes: Places the mirrored image into the drawing and erases the original objects.

No: Places the mirrored image into the drawing and retains the original objects.

You can create objects that represent half of a drawing, select them, and mirror them across a specified line to create the other half.

***Note:**By default, when you mirror a text object, the direction of the text is not changed. Set the MIRRTEXT system variable to 1 if you do want the text to be reversed.

19 . OFFSET (O) :

Ribbon →Home tab

Modify panel → Offset

Menu: Modify → Offset

Toolbar → Modify

Command entry: offset or 'O'

Current settings: Erase source = current Layer = current OFFSETGAPTYPE = current

Specify offset distance or [Through/Erase/Layer] <current>: Specify a distance, enter an option, or press Enter

- You can offset an object at a specified distance or through a point. After you offset objects, you can trim and extend them as an efficient method to create drawings containing many parallel lines and curves.

* **NOTE:** The OFFSET command repeats for convenience. To exit the command, press **Enter**.

* Creates an object at a specified distance from an existing object.

* **Select object to offset or [Exit/Undo] <exit>:** Select one object, enter an option, or press Enter to end the command.

* Specify point on side to offset or [Exit/Multiple/Undo] <exit or next object>: Specify a point (1) on the side of the object you want to offset or enter an option

* **Exit:** Exits the OFFSET command.

- * **Multiple:** Enters the Multiple offset mode, which repeats the offset operation using the current offset distance.
- * **Undo:** Reverses the previous offset.
- * **Through:** Creates an object passing through a specified point.
- * **Select object to offset or <exit>:** Select one object or press Enter to end
- * **Note:** For best results when you offset a polyline with corners, specify the through point near the midpoint of a line segment, not near a corner.
- * **Erase:** Erases the source object after it is offset.
Erase source object after offsetting? [Yes/No] <Current>: Enter 'y' or 'n'
- * **Layer:** Determines whether offset objects are created on the current layer or on the layer of the source object.
- * **Enter layer option for offset objects [Current/Source] <current>:** Enter an option

20 . ARRAY (AR) : Ribbon → Home tab

Modify panel → Array.

Toolbar → Modify

Menu → Modify → Array

Command entry: array or 'ar'

Summary: You can create copies of objects in a regularly spaced rectangular or polar array.

The Array dialog box is displayed. You can create rectangular or polar arrays by choosing the appropriate option. Each object in an array can be manipulated independently. If you select multiple objects, the objects are treated as one item to be copied and arrayed.

If you enter -array at the Command prompt, options are displayed.

21 . MOVE (M) :

Ribbon → Home tab

Modify panel → Move

Menu: Modify → Move

Toolbar: Modify

Command entry → move or 'm'

* **Shortcut menu:** Select the objects to move, and right-click in the drawing area. Click Move.

* **Select objects:** Use an object selection method and press Enter when you finish

* Specify base point or [Displacement]<Displacement>: Specify a base point or enter 'd'

* **Specify second point or <use first point as displacement>:** Specify a point or press enter

* The two points you specify define a vector that indicates how far the selected objects are to be moved and in what direction.

If you press Enter at the Specify Second Point prompt, the first point is interpreted as a relative X, Y, Z displacement. For example, if you specify 2, 3 for the base point and press Enter at the next prompt, the objects move 2 units in the X direction and 3 units in the Y direction from their current position. Use coordinates, grid snaps, object snaps, and other tools to move objects with precision.

* **Displacement:** Specify displacement <last value>: Enter coordinates to represent a vector. The coordinate values that you enter specify a relative distance and direction

22 . ROTATE (RO) :

Ribbon → Home tab

Modify panel → Rotate

Menu: Modify → Rotate

Toolbar → Modify

Command entry: rotate or 'ro' **Shortcut menu:** Select the objects to rotate, and right-click in the drawing area. Click Rotate.

Current positive angle in UCS: ANGDIR=current ANGBASE=current

* **Select objects:** Use an object selection method and press ENTER when you finish

* Specify base point: Specify a point. Specify rotation angle or [Copy/Reference]: Enter an angle, specify a point, enter 'c', or enter 'r'

* **Rotation Angle :** Determines how far an object rotates around the base point.

The axis of rotation passes through the specified base point and is parallel to the Z axis of the current UCS.

- * **Copy:**Creates a copy of the selected objects for rotation.
- * **Reference:**Rotates objects from a specified angle to a new, absolute angle.
- * **Specify the reference angle <last reference angle>:**Specify an angle by entering a value or by specifying two points
- * **Specify the new angle or [Points] <last new angle>:** Specify the new absolute angle by entering a value or by specifying two points* When you rotate a viewport object, the borders of the viewport remain parallel to the edges of the drawing area. You can rotate selected objects around a base point to an absolute angle.

23 . SCALE (SC):

Ribbon → Home tab

Modify panel → Scale

Menu → Modify Scale Not available in menus in the current workspace

Toolbar: Modify

Shortcut menu → Select the objects to scale, and right-click in the drawing area. Click Scale.

Command entry: scale or 'sc'

* To scale an object, specify a base point and a scale factor. The base point acts as the center of the scaling operation and remains stationary. A scale factor greater than 1 enlarges the object. A scale factor between 0 and 1 shrinks the object.

* **Select objects:** Use an object selection method and press Enter when you finish

* **Specify base point:** Specify a point

The base point you specify identifies the point that remains in the same location as the selected objects change size (and thus move away from the stationary base point).

* **Note:**When you use the SCALE command with annotative objects, the position or location of the object is scaled relative to the base point of the scale operation, but the size of the object is not changed.

* **Specify scale factor or [Copy/Reference]:**Specify a scale, enter 'c', or enter 'r'

* **Scale Factor:** Multiplies the dimensions of the selected objects by the specified scale. A scale factor greater than '1' enlarges the objects. A scale factor between 0 and 1 shrinks the objects. You can also drag the cursor to make the object larger or smaller.

* **Copy:** Creates a copy of the selected objects for scaling.

* **Reference:** Scales the selected objects based on a reference length and a specified new length.

Specify reference length <1>: Specify a beginning length from which to scale the selected objects

Specify new length or [Points]: Specify a final length to which to scale the selected objects, or enter 'p' to define a length with two points.

24 . STRETCH ():

Ribbon → Home tab → Modify panel → Stretch

Menu → Modify → Stretch

Toolbar → Modify

Command entry: stretch

Objects that are partially enclosed by a crossing window are stretched. Objects that are completely enclosed within the crossing window, or that are selected individually, are moved rather than stretched. Several objects such as circles, ellipses, and blocks, cannot be stretched.

* Select objects to stretch by crossing-window or crossing-polygon...

* **Select objects:** Use the cpolygon option or the crossing object selection method, and press Enter. Individually selected objects and objects that are completely enclosed by the crossing selection are moved rather than stretched. STRETCH moves only the vertices and endpoints that lie inside the crossing selection, leaving those outside unchanged. STRETCH does not modify 3D solids, polyline width, tangent, or curve-fitting information.

* **Base Point:** Specify base point or [Displacement] <last displacement>:

Specify a base point or enter displacement coordinates

* **Specify second point or <use first point as displacement>:**

Specify a second point, or press Enter to use the previous coordinates as a displacement

* **Displacement:** Specify displacement <last value>: Enter displacement values for X, Y (and optionally Z)

If you enter a second point, the objects are stretched the vector distance from the base point to the second point. If you press Enter at the Specify Second Point of Displacement prompt, the first point is treated as an X,Y,Z displacement.

25 . TRIM (TR) :

Home tab → Modify panel → Trim and Extend drop-down **Menu** → Modify

Toolbar → Modify

Command entry: trim or 'tr'

To trim objects, select the boundaries. Then press ENTER and select the objects that you want to trim. To use all objects as boundaries, press ENTER at the first Select Objects prompt.

Current settings: Projection = current, Edge = current

Select cutting edges...

* **Select objects or <select all>**: Select one or more objects and press ENTER, or press ENTER to select all displayed objects* Select the objects that define the cutting edges to which you want to trim an object, or

Press ENTER to select all displayed objects as potential cutting edges. TRIM projects the cutting edges and the objects to be trimmed onto the XY plane of the current user coordinate system (UCS).

* **Note:**To select cutting edges that include blocks, you can use only the single selection, Crossing, Fence, and Select all options.

* Select object to trim or shift-select to extend or

[Fence/Crossing/Project/Edge/erase/Undo]: Select an object to trim, hold down SHIFT and select an object to extend it instead, or enter an option

* Specify an object selection method to select the objects to trim. If more than one trim result is possible, the location of the first selection point determines the result.

* **Object to Trim:** Specifies the object to trim. The prompt for selecting the object to trim repeats so you can trim multiple objects. Press ENTER to exit the command. Shift-Select to Extend

- * Extends the selected objects rather than trimming them. This option provides an easy method to switch between trimming and extending.
- * **Fence:** Selects all objects that cross the selection fence. The selection fence is a series of temporary line segments that you specify with two or more fence points. The selection fence does not form a closed loop.
- * **Specify first fence point:** Specify the starting point of the selection fence
- * **Specify next fence point or [Undo]:** Specify the next point of the selection fence or enter u
- * **Specify next fence point or [Undo]:** Specify the next point of the selection fence, enter U, or ENTER
- * **Crossing:** Selects objects within and crossing a rectangular area defined by two points.
- * **Specify first corner:** Specify a point
- * **Specify opposite corner:** Specify a point at a diagonal from the first point
- * **Note:** **Some crossing selections of objects to be trimmed are ambiguous. TRIM resolves the selection by following along the rectangular crossing window in a clockwise direction from the first point to the first object encountered.**
- * **Project:** Specifies the projection method used when trimming objects.
Enter a projection option [None/UCS/View] <current>: Enter an option or press ENTER
- * **None:** Specifies no projection. The command trims only objects that intersect with the cutting edge in 3D space.
- * **UCS:** Specifies projection onto the XY plane of the current UCS. The command trims objects that do not intersect with the cutting edge in 3D space.
- * **View:** Specifies projection along the current view direction. The command trims objects that intersect the boundary in the current view.
- * **Edge:** Determines whether an object is trimmed at another object's extrapolated edge or only to an object that intersects it in 3D space.
Enter an implied edge extension mode [Extend/No extend] <current>: Enter an option or press ENTER
- * **Extend:** Extends the cutting edge along its natural path to intersect an object in 3D space.
- * **No Extend:** Specifies that the object is trimmed only at a cutting edge that intersects it in 3D space.
- * **Note:** **When trimming hatches, do not set Edge to Extend. If you do, gaps in the trim**

boundaries will not be bridged when trimming hatches, even when the gap tolerance is set to a correct value.

- * **Erase:** Deletes selected objects. This option provides a convenient method to erase unneeded objects without leaving the TRIM command.
- * **Select objects to erase or <exit>:** Use an object selection method and press ENTER to return to the previous prompt
- * **Undo:** Reverses the most recent change made by TRIM.

26 . EXTEND (EX) :

Ribbon → Home tab

Modify panel → Trim and Extend drop-down

Menu → Modify → Extend

Toolbar → Modify

Command entry: extend or 'ex'

* **Summary:** To extend objects, first select the boundaries. Then press Enter and select the objects that you want to extend. To use all objects as boundaries, press Enter at the first Select Objects prompt.

Current settings: Projection = current, Edge = current

Select boundary edges...

Select objects or <select all>: Select one or more objects and press Enter, or press Enter to select all displayed objects

Select object to extend or shift-select to trim or [Fence/Crossing/Project/Edge/Undo]: Select objects to extend, or hold down SHIFT and select an object to trim, or enter an option.

Boundary Object Selection Uses the selected objects to define the boundary edges to which you want to extend

- * **Object to Extend:** Specifies the objects to extend. Press Enter to end the command.
- * **Shift-Select to Trim:** Trims the selected objects to the nearest boundary rather than extending them. This is an easy method to switch between trimming and extending.
- * **Fence:** Selects all objects that cross the selection fence. The selection fence is a series of temporary line segments that you specify with two or more fence points. The selection fence does not form a closed loop.

Specify first fence point: Specify the starting point of the selection fence

Specify next fence point or [Undo]: Specify the next point of the selection fence or enter u

Specify next fence point or [Undo]: Specify the next point of the selection fence enter 'u', or press Enter

* **Crossing:** Selects objects within and crossing a rectangular area defined by two points.

Specify first corner: Specify a point

Specify opposite corner: Specify a point at a diagonal from the first point

* **Note:** Some crossing selections of objects to be extended are ambiguous. **EXTEND** resolves the selection by following along the rectangular crossing window in a clockwise direction from the first point to the first object encountered.

* **Project:** Specifies the projection method used when extending objects.

Enter a projection option [None/Ucs/View] <current>: Enter an option or press Enter

* **Edge:** Extends the object to another object's implied edge, or only to an object that actually intersects it in 3D space.

Enter an implied edge extension mode [Extend/No extend] <current>: Enter an option or press Enter

* **Extend:** Extends the boundary object along its natural path to intersect another object or its implied edge in 3D space.

* **No Extend:** Specifies that the object is to extend only to a boundary object that actually intersects it in 3D space.

* **Undo:** Reverses the most recent changes made by EXTEND.

27 . JOIN (J) :

Ribbon → Home tab → Modify panel → Join

Menu: Modify → Join

Toolbar: Modify

Command entry: join or 'j'

* Objects to be joined must be located in the same plane. Each type of object has additional restrictions.

Select source object: Select a line, polyline, arc, elliptical arc, spline, or helix. Depending on the source object selected, one of the following prompts is displayed:

- * **Line:** Select lines to join to source: Select one or more lines and press ENTER, the line objects must be collinear (lying on the same infinite line), but can have gaps between them.
- * **Polyline:** Select objects to join to source: Select one or more objects and press ENTER the objects can be lines, polylines, or arcs. The objects cannot have gaps between them, and must lie on the same plane parallel to the UCS XY plane.
- * **Arc:** Select arcs to join to source or [close]: Select one or more arcs and press ENTER,
Or ↶ 'L'

The arc objects must lie on the same imaginary circle, but can have gaps between them. The Close option converts the source arc into a circle.

*** Note:** When joining two or more arcs, the arcs are joined counterclockwise beginning from the source object.

28 .CHAMFER OR (CH) :

Ribbon: Home tab → Modify panel → Chamfer and Fillet drop-down

Menu: Modify → Chamfer at the Command prompt, enter chamfer.

Toolbar: Modify

Command entry: chamfer or 'ch'

Summary: The distances and angles that you specify are applied in the order that you select the objects.

You can chamfer lines, polylines, rays, and xlines.

You can also chamfer 3D solids and surfaces. If you select a mesh to chamfer, you can choose to convert it to a solid or surface before completing the operation.

* List of Prompts

The following prompts are displayed.

(TRIM mode) Current chamfer Dist1 = current, Dist2 = current

→ Select first line or Undo/Polyline/Distance/Angle/Trim/mEthod/Multiple]:

Use an object selection method or enter an option

→ First Line: Specifies the first of two edges required to define a 2D chamfer, or the edge of 3D solid to chamfer.

→ Select second line or shift-select to apply corner: Use an object selection method or hold down Shift and select an object to create a sharp corner

- If you select lines or polylines, their lengths adjust to accommodate the chamfer line. You can hold down Shift while selecting the objects to override the current chamfer distances with a value of 0.
- If the selected objects are line segments of a 2D polyline, they must be adjacent or separated by no more than one segment. If they're separated by another polyline segment, CHAMFER deletes the segment that separates them and replaces it with the chamfer.
- If you select an edge on a 3D solid, you must indicate which one of the two surfaces adjacent to the edge is the base surface.

Base surface selection...

- Enter surface selection option [Next/OK (current)] <OK>: Enter n or o, or press Enter
- Entering o or pressing Enter sets the selected surface as the base surface. Entering n selects either of the two surfaces adjacent to the selected edge.
- Specify base surface chamfer distance <current>:
- Specify other surface chamfer distance <current>:

After you select the base surface and the chamfer distances, select the edges of the base surface to chamfer. You can select edges individually or all the edges at once.

Select an edge or [Loop]: Select an edge, enter L, or press Enter

- Edge: Selects an individual edge to chamfer.
- Loop: Switches to Edge Loop mode.
Select an edge loop or [Edge]: Select an edge, enter e, or press Enter
- Edge Loop: Selects all edges on the base surface.

- Polyline : Chamfers an entire 2D polyline.
- Select 2D polyline: The intersecting polyline segments are chamfered at each vertex of the polyline. Chamfers become new segments of the polyline.
- If the polyline includes segments that are too short to accommodate the chamfer distance, those segments are not chamfered.
- Distance: Sets the distance of the chamfer from the endpoint of the selected edge.
- Specify first chamfer distance <current>:
- Specify second chamfer distance <current>:

If you set both distances to zero, CHAMFER extends or trims the two lines so they end at the same point.

- Angle: Sets the chamfer distances using a chamfer distance for the first line and an angle for the second line.
- Specify chamfer length on the first line <current>:
Specify chamfer angle from the first line <current>:
- Trim: Controls whether CHAMFER trims the selected edges to the chamfer line endpoints.
- Enter Trim mode option [Trim/No trim] <current>:

Note : Trim sets the TRIMMODE system variable to 1; No Trim sets TRIMMODE to 0.

- If the TRIMMODE system variable is set to 1, CHAMFER trims the intersecting lines to the endpoints of the chamfer line. If the selected lines do not intersect, CHAMFER extends or trims them so that they do. If TRIMMODE is set to 0, the chamfer is created without trimming the selected lines.
- Method: Controls whether CHAMFER uses two distances or a distance and an angle to create the chamfer.
- Enter trim method [Distance/Angle] <current>:
- Multiple: Chamfers the edges of more than one set of objects. CHAMFER displays the main prompt and the Select Second Object prompt repeatedly until you press Enter to end the command.

29 . FILLET (F):

→ **Modify panel** → Chamfer and Fillet drop-down

Toolbar: Modify

Menu: Modify

Command entry: fillet or 'f'

→ Summary: In the example, an arc is created that is tangent to both of the selected lines. The lines are trimmed to the ends of the arc. To create a sharp corner instead, enter a radius of zero.

→ You can fillet arcs, circles, ellipses, elliptical arcs, lines, polylines, rays, splines, and xlines.

→ You can also fillet 3D solids and surfaces. If you select a mesh object for filleting, you can choose to convert the mesh to a solid or surface and continue the operation.

→ List of Prompts

Current settings: Mode = current, Radius = current

Select first object or [Undo/Polyline/Radius/Trim/Multiple]: Use an object selection method or enter an option

→ First Object: Selects the first of two objects required to define a 2D fillet or selects the edge of a 3D solid to round or fillet the edge.

→ Select second object or shift-select to apply corner: Use an object selection method or hold down Shift and select an object to create a sharp corner

→ If you select lines, arcs, or polylines, their lengths adjust to accommodate the fillet arc. You can hold down Shift while selecting the objects to override the current fillet radius with a value of 0.

→ If the selected objects are straight line segments of a 2D polyline, they can be adjacent or separated by one other segment. If they are separated by another polyline segment, FILLET deletes the segment that separates them and replaces it with the fillet.

→ More than one fillet can exist between arcs and circles. Select the objects close to where you want the endpoints of the fillet.

* **Enter fillet radius <current>**: Specify a distance or press Enter

Select an edge or [Chain/Radius]: Select edge(s), enter 'c', or enter 'r'

* **Edge**: Selects a single edge. You can continue to select single edges until you press Enter.

If you select three or more edges that converge at a vertex to form the corner of a box, FILLET computes a vertex blend that is part of a sphere if the three incident fillets have the same radii.

* **Chain:** Changes from selection of single edges to selection of sequential tangent edges, called a chain selection.

Select edge chain or <Edge/Radius>: Select an edge chain, enter e, or enter r

* **Edge Chain:** Selects a tangential sequence of edges when you select a single edge.

For example, if you select an edge on the top of a 3D solid box, FILLET also selects the other tangential edges on the top.

* **Edge:** Switches to a single-edge selection mode.

* **Radius:** Defines the radius of the rounded edge.

* **Enter fillet radius <current>:** Specify a distance or press Enter

The previous prompt is displayed:

Select an edge or [Chain/Radius]: Select one or more edges, or enter 'c' or 'r'

* **Undo:** Reverses the previous action in the command.

* **Polyline:** Inserts fillet arcs at each vertex of a 2D polyline where two line segments meet.

* **Select 2D polyline:** If one arc segment separates two line segments that converge as they approach the arc segment, FILLET removes the arc segment and replaces it with a fillet arc.

Radius: Defines the radius of the fillet arc.

→ Specify fillet radius <current>: Specify a distance or press Enter

→ The value you enter becomes the current radius for subsequent FILLET commands.

Changing this value does not affect existing fillet arcs.

* **Trim:** Controls whether FILLET trims the selected edges to the fillet arc endpoints.

Enter Trim mode option [Trim/No trim] <current>: Enter an option or press Enter

Trim: Trims the selected edges to the fillet arc endpoints.

No Trim: Does not trim the selected edges.

* **Multiple:** Rounds the edges of more than one set of objects. FILLET displays the main prompt and the Select Second Object prompt repeatedly until you press Enter to end the command.

30 . EXPLODE () :

Ribbon → Home tab

Modify panel → Explode

Menu → Modify

Toolbar → Modify

Command entry → explode

Summary: Explodes a compound object when you want to modify its components separately.

Objects that can be exploded include blocks, polylines, and regions, among others.

→ The color, line type, and line weight of any exploded object might change. Other results differ depending on the type of compound object you're exploding. See the following list of objects that can be exploded and the results for each.

→ To explode objects and change their properties at the same time, use XPLODE.

***Note:** If you're using a script or an ObjectARX® function, you can explode only one object at a time.

* **2D and Lightweight Polyline:** Discards any associated width or tangent information.

For wide polylines, the resulting lines and arcs are placed along the center of the polyline.

* **3D Polyline:**Explodes into line segments. Any line type assigned to the 3D polyline is applied to each resulting line segment.

* **3D Solid:**Explodes planar faces into regions. Non-planar faces explode into surfaces.

Annotative Objects Explodes the current scale representation into its constituent parts which are no longer annotative. Other scale representations are removed.

* **Arc:**If within a nonuniformly scaled block, explodes into elliptical arcs.

* **Block:**Removes one grouping level at a time. If a block contains a polyline or a nested block, exploding the block exposes the polyline or nested block object, which must then be exploded to expose its individual objects.

-Blocks with equal X, Y, and Z scales explode into their component objects. Blocks with unequal X, Y, and Z scales (nonuniformly scaled blocks) might explode into unexpected objects.

-When nonuniformly scaled blocks contain objects that cannot be exploded, they are collected into an anonymous block (named with a “*E” prefix) and referenced with the nonuniform scaling. If all the objects in such a block cannot be exploded, the selected block reference will not be exploded. Body, 3D Solid, and Region entities in a nonuniformly scaled block cannot be exploded.

-Exploding a block that contains attributes deletes the attribute values and redisplay the attribute definitions

-Blocks inserted with MINSERT and external references (xrefs) and their dependent blocks cannot be exploded.

***Body:**Explodes into a single-surface body (nonplanar surfaces), regions, or curves.

***Circle:**If within a non-uniformly scaled block, explodes into ellipses.

* **Leaders:** Explodes into lines, splines, solids (arrow heads), block inserts (arrow heads, annotation blocks), multiline text, or tolerance objects, depending on the leader.

* **Mesh Objects:** Explodes each face into a separate 3D face object. Color and materials assignments are retained.

* **Multiline Text:** Explodes into text objects.

* **Multiline:**Explodes into lines and arcs.

* **Polyface Mesh:**Explodes one-vertex meshes into a point object. Two-vertex meshes explode into a line. Three-vertex meshes explode into 3D faces.

* **Region:**Explodes into lines, arcs, or splines.

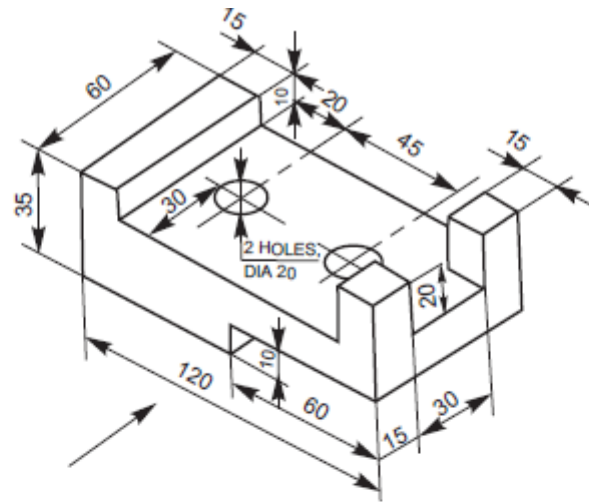
* **List of Prompts:** The following prompt is displayed.

-Select objects: Specifies which objects to explode. Select the objects and press Enter.

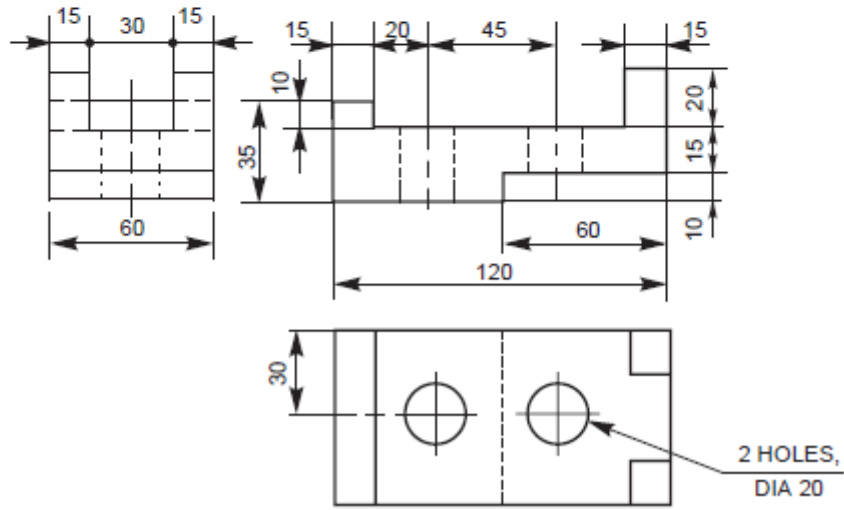
Exercise No. 1

Aim: To draw the orthographic projections (i.e. front view, top view and side view) of given solids.

i)

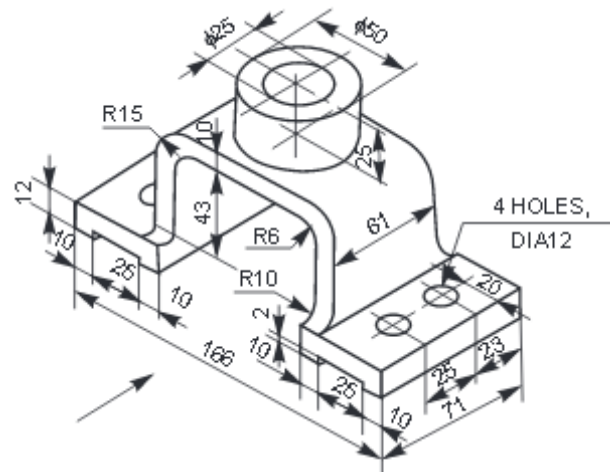


(a)

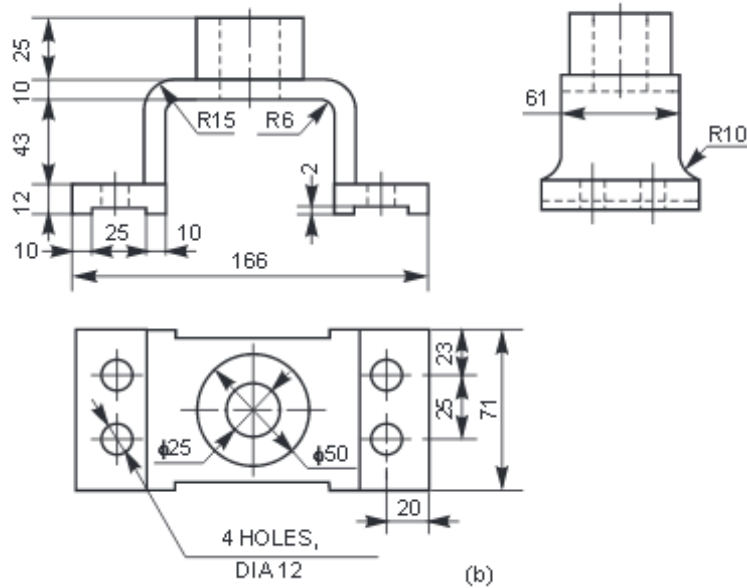


(b)

ii)



(a)



(b)

Commands Used:

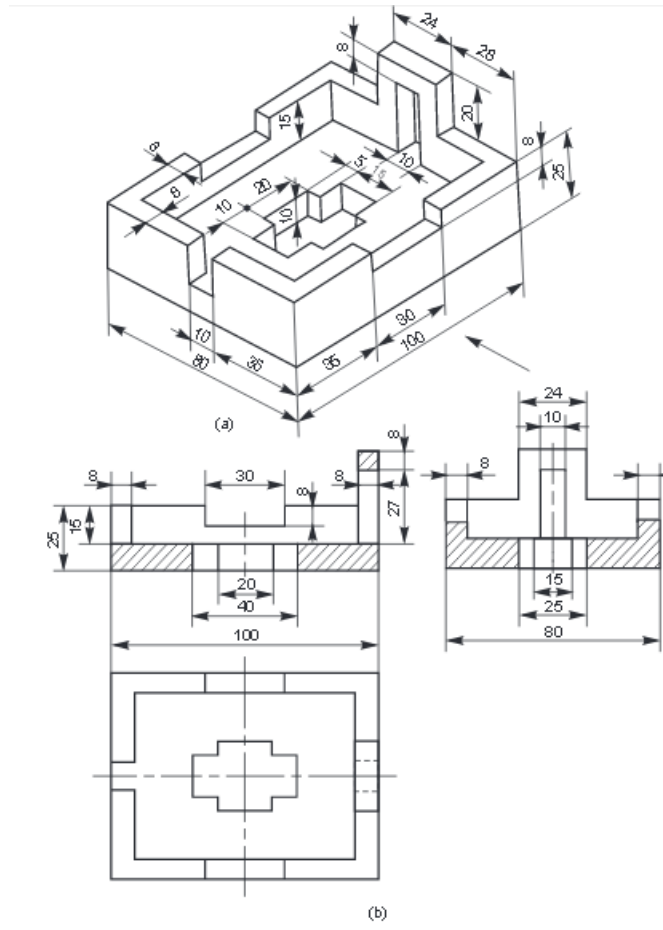
1. Line
2. Circle
3. Arc
4. Dimensions
5. Trim
6. Fillet

Procedure:

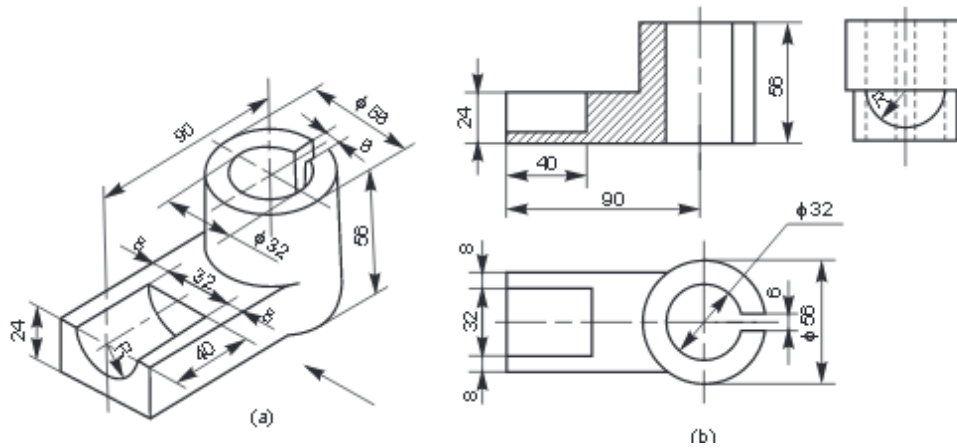
1. First draw the reference line and front view using the line, offset, Arc, Trim, Extend, Fillet commands as per the given dimensions and give dimension from Dimension Tool Bar.
2. Draw the top view by using line, circle, trim, extend commands as per given dimensions.
3. Draw the side view by using line, arc commands as per the given dimensions.

Exercise No. 2

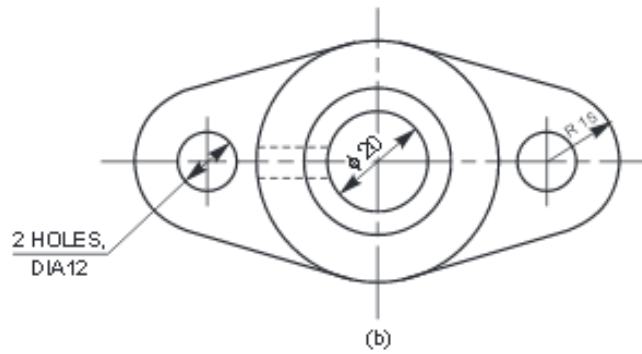
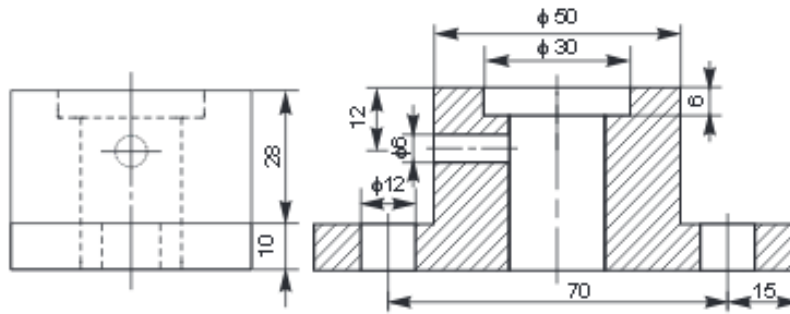
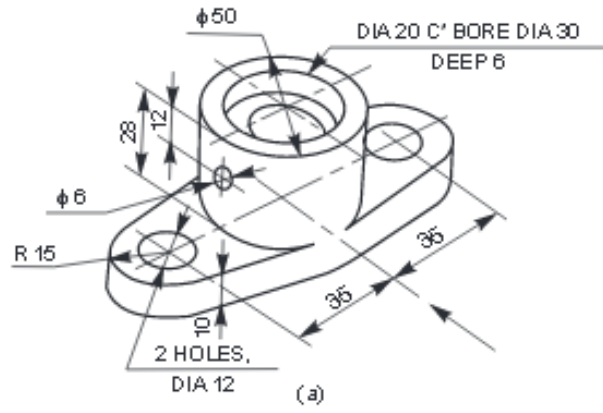
Aim: To draw the sectional view of given orthogonal solid components



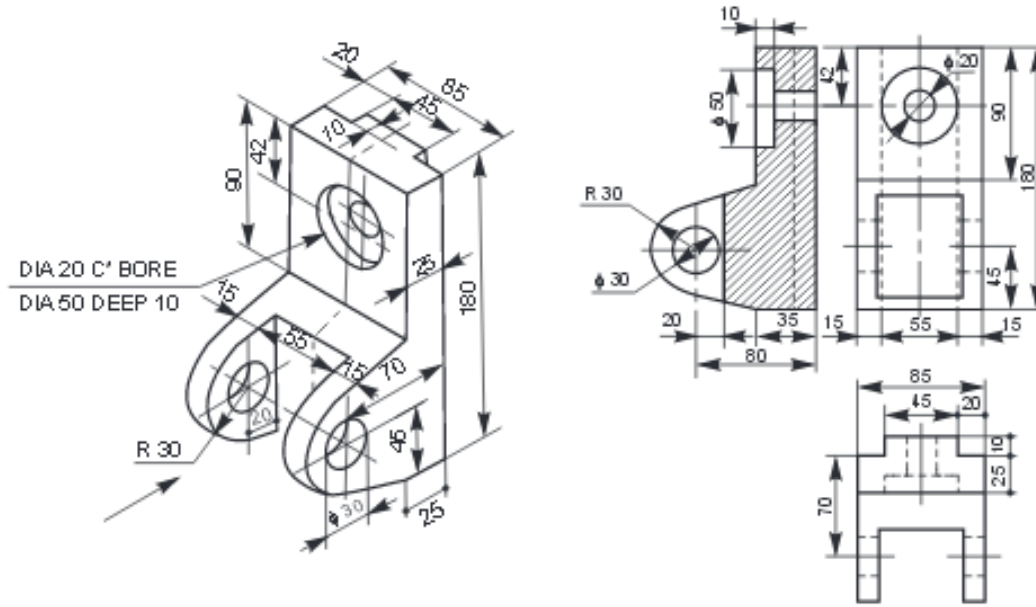
i) Machine Block



ii) Machine Component



iii) Shaft Support



iv) Sliding Block

Commands Used:

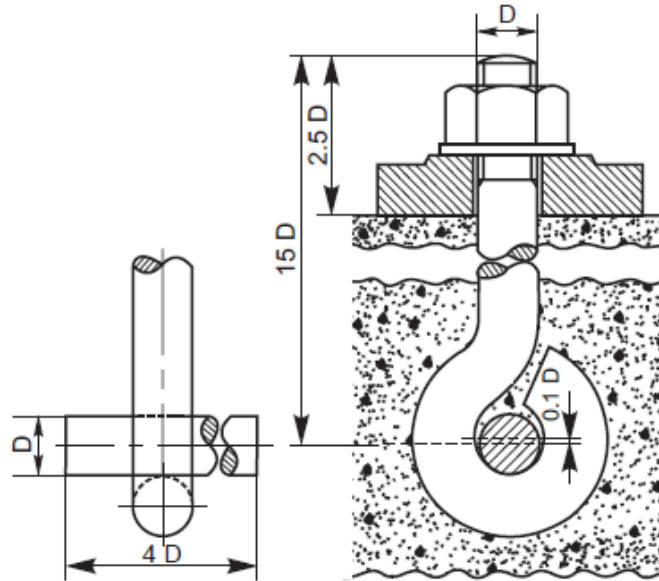
1. Line
2. Circle
3. Arc
4. Dimensions
5. Trim
6. Fillet
7. Hatch

Procedure:

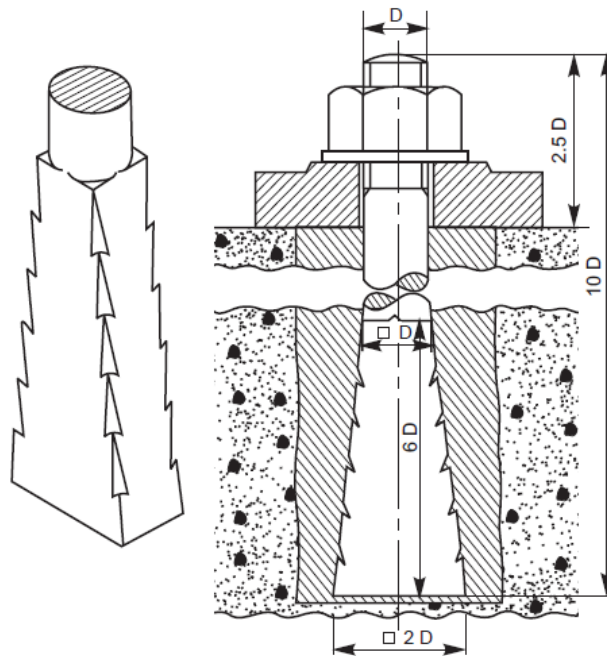
1. First draw the reference line and front view using the line, offset, Arc, Trim, Extend, Fillet commands as per the given dimensions.
2. Draw the top view by using line, circle commands as per given dimensions.
3. Draw the side view by using line, arc commands as per the given dimensions.
4. Hatch the required region to determine the section.

Exercise No.3

Aim: To draw the orthogonal Front and Side View of Eye Foundation Bolt and Rag Foundation Bolt



i) Eye Foundation Bolt



ii) Rag Foundation Bolt

Commands used:

1. Line
2. Circle
3. Arc
4. Dimensions
5. Trim
6. Fillet

Procedure:

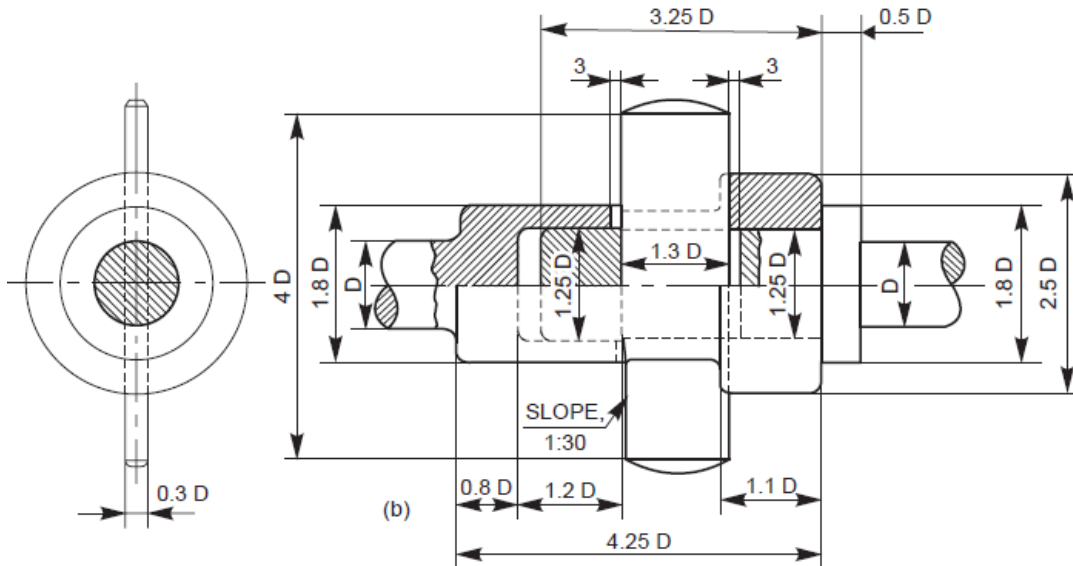
1. First draw the reference line and front view using the line, circle, arc, dimensions, trim, fillet commands as per the given dimensions.
2. Draw the side view by using line, trim, arc commands as per the given dimensions.

Applications:

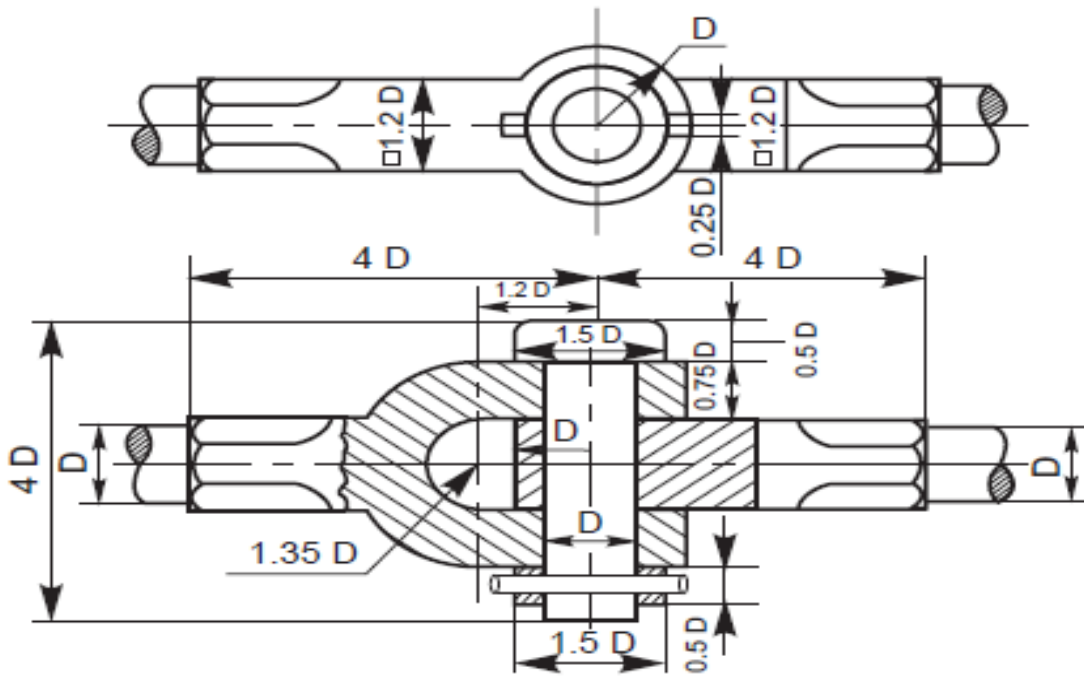
It is used to make a temporary joint of any two or more machine elements.

Exercise No. 4

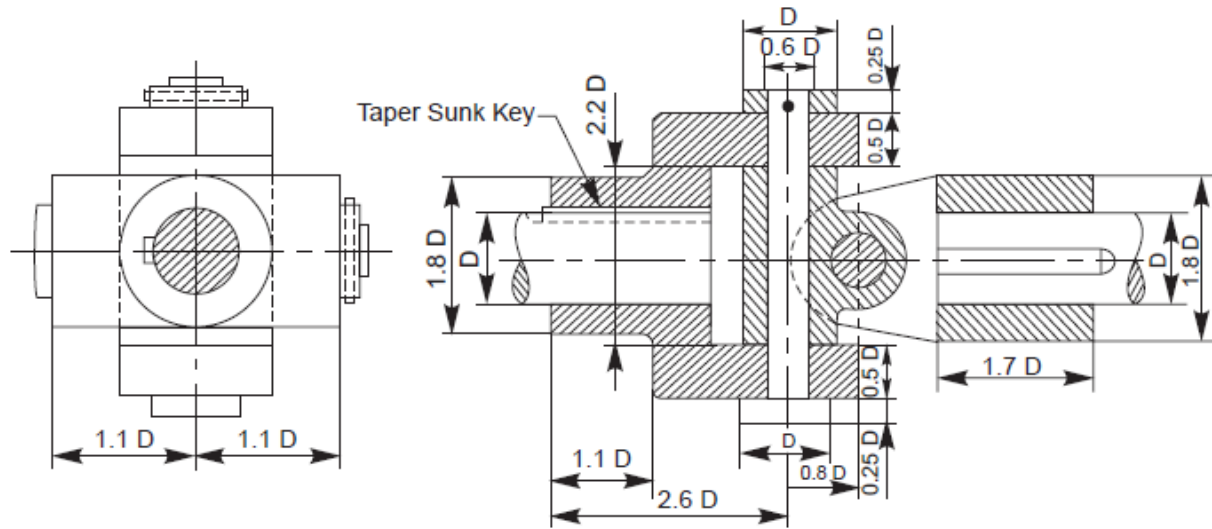
Aim: To draw the orthogonal sectional front view of cotter joint with socket and spigot end, Knuckle joint, and Universal Coupling.



i) cotter joint with socket and spigot end



ii) Knuckle joint



iii) Universal Coupling

Commands used:

1. Line
2. Circle
3. Arc
4. Dimensions
5. Trim
6. Fillet
7. Offset
8. Hatch

Procedure:

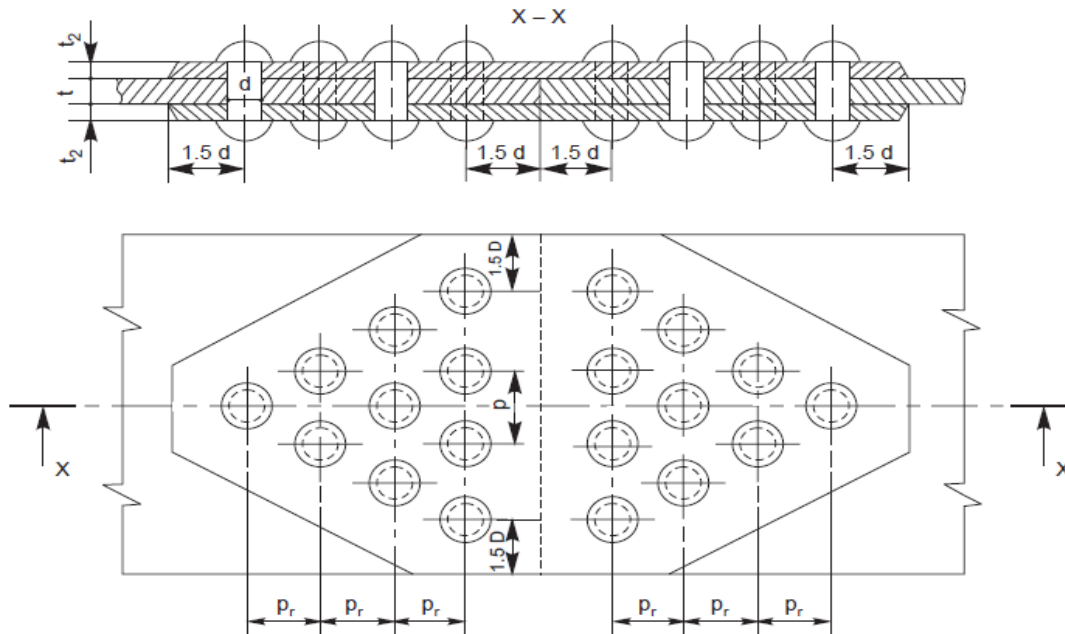
1. First draw the reference line and sectional front view using the line, circle, arc, dimensions, trim, fillet, offset, hatch commands as per the given dimensions.
2. Draw the side view by using line, arc and circle commands as per the given dimensions.

Applications:

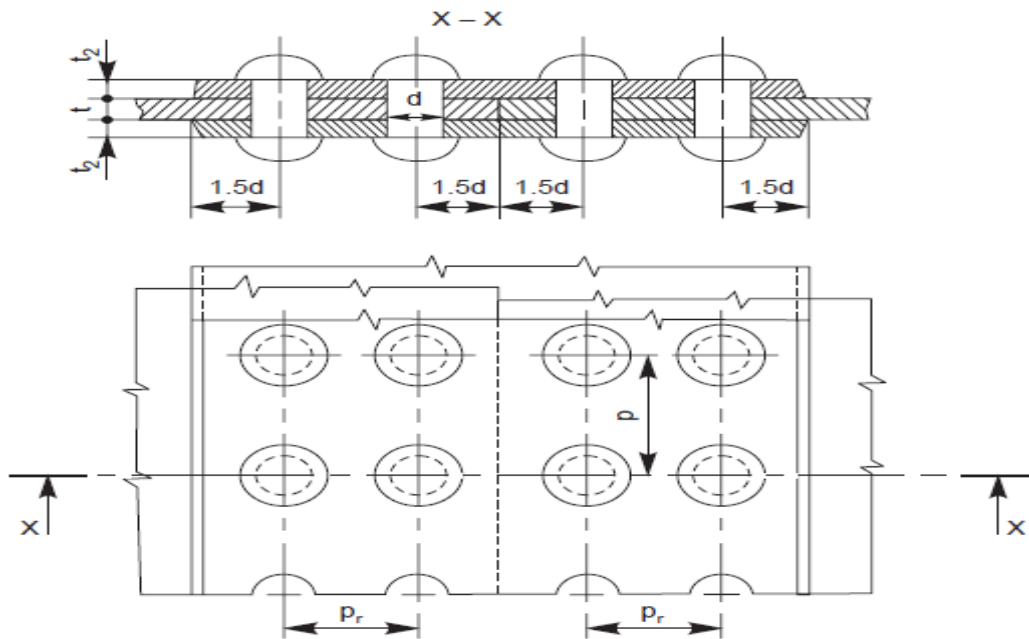
It is used to connect two rods, shafts and shaft & pulley to transmit forces and power from one member to other member.

Exercise No. 5

Aim: To draw the orthogonal sectional front view of double strap diamond butt joint and double riveted double strap chain butt joint.



i) double strap diamond butt joint



ii) double riveted double strap chain butt joint.

Commands used:

1. Line
2. Circle
3. Arc
4. Dimensions
5. Trim
6. Fillet
7. Offset
8. Hatch

Procedure:

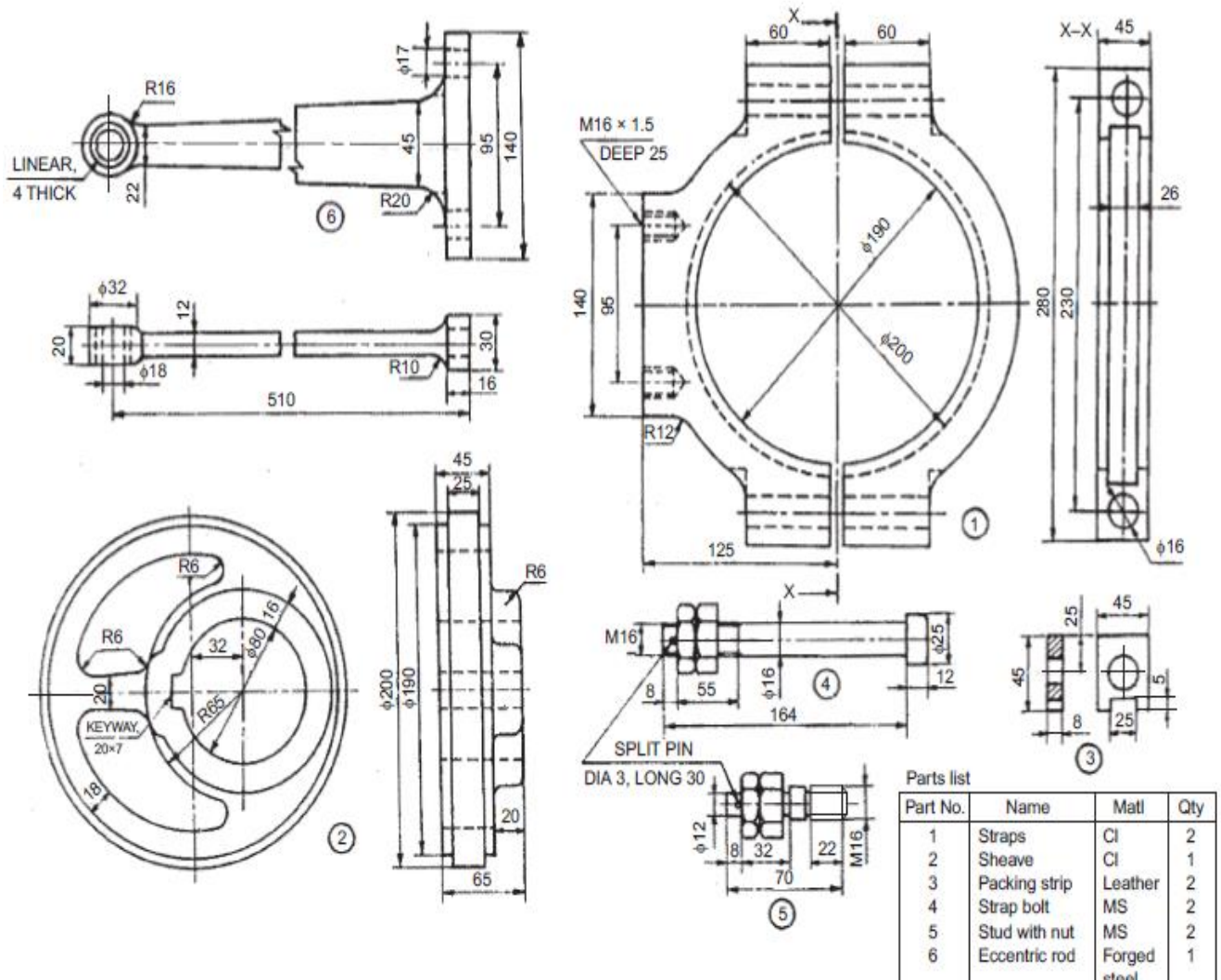
1. First draw the reference line and sectional front view using the line, circle, arc, dimensions, trim, fillet, offset, hatch commands as per the given dimensions.
2. Draw the side view by using line, arc, and circle commands as per the given dimensions.

Applications:

It is used to build boiler shell, pressure vessels and to join two parts semi-permanently and permanently and also to hold and transport the fluid from one place to other place.

Exercise No. 6

Aim: To assemble the parts and draw half sectional view from the front, with top half in section of the Eccentric.



Eccentric

Commands Used:

1. Line
2. Circle
3. Arc
4. Dimensions
5. Trim
6. Fillet
7. Offset
8. Hatch

Procedure:

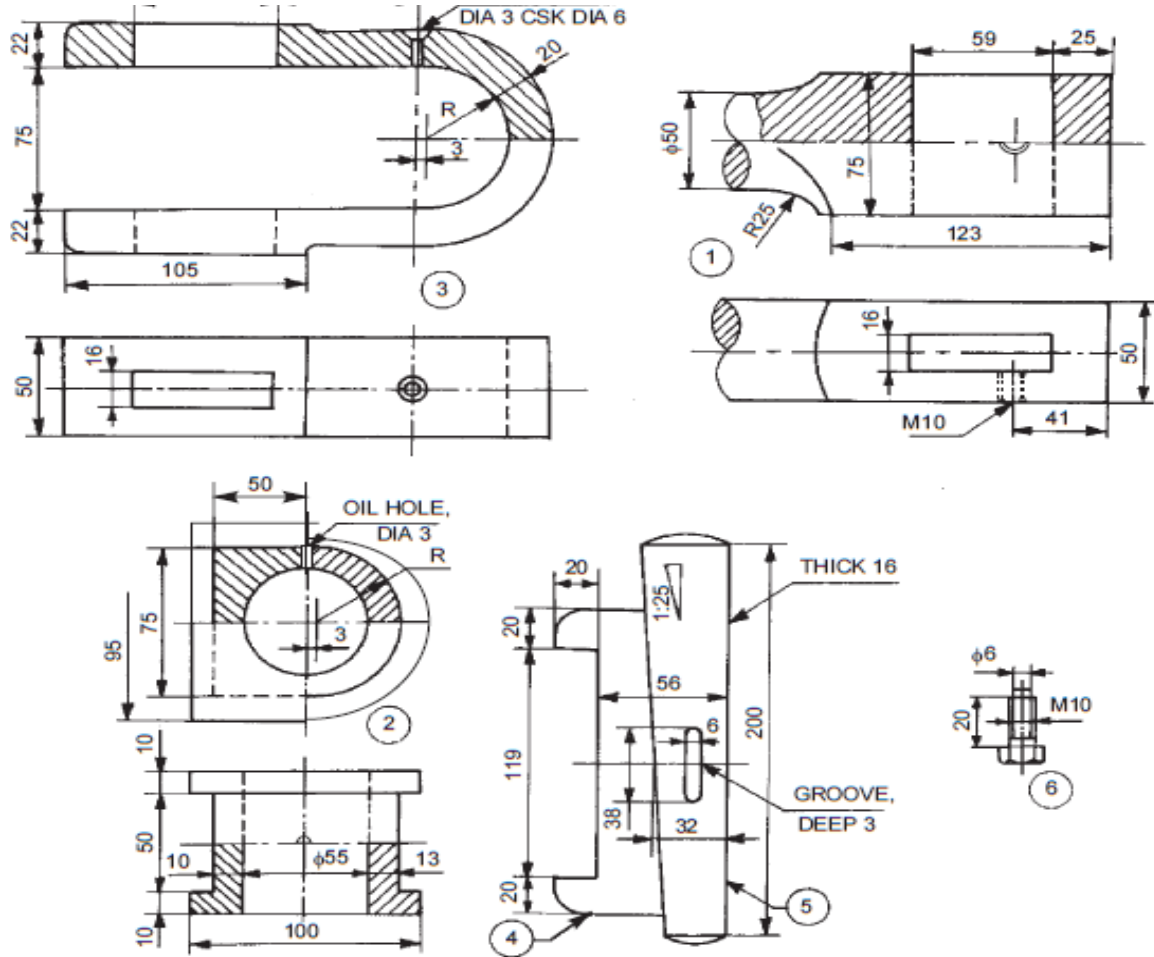
1. First draw the axis of first part which is the strap, then draw the strap using the commands as per the given dimensions.
2. Move the strap apart and draw the next part using some different color.
3. Draw all the parts using different colors according to the given dimensions.
4. Assemble all the parts by placing the centre of the axis of the parts in a manner by which they coincide.
5. Give the assembled figure dimensions to complete the figure.

Applications of Eccentric:

It is used to provide a short reciprocating motion, actuated by the rotation of a shaft. Eccentrics are used for operating steam valves, small pump plungers, shaking screens, etc. Rotary motion can be converted into reciprocating motion with an eccentric, but the reverse conversion is not possible due to excessive friction between the sheave and the strap. The crank arrangement, in a slider crank mechanism however, allows conversion in either direction.

Exercise No. 7

Aim: To assemble the parts and draw half sectional view from the front, with top half in section of the Steam Engine Connecting Rod End



Parts list

Sl. No.	Name	Matl.	Qty.
1	Connecting rod	FS	1
2	Brasses	GM	2
3	Strap	MS	1
4	Jib	MS	1
5	Cotter	MCS	1
6	Set screw	MCS	1

Steam Engine Connecting Rod End

Commands Used:

1. Line

2. Circle
3. Arc
4. Dimensions
5. Trim
6. Fillet
7. Offset
8. Hatch

Procedure:

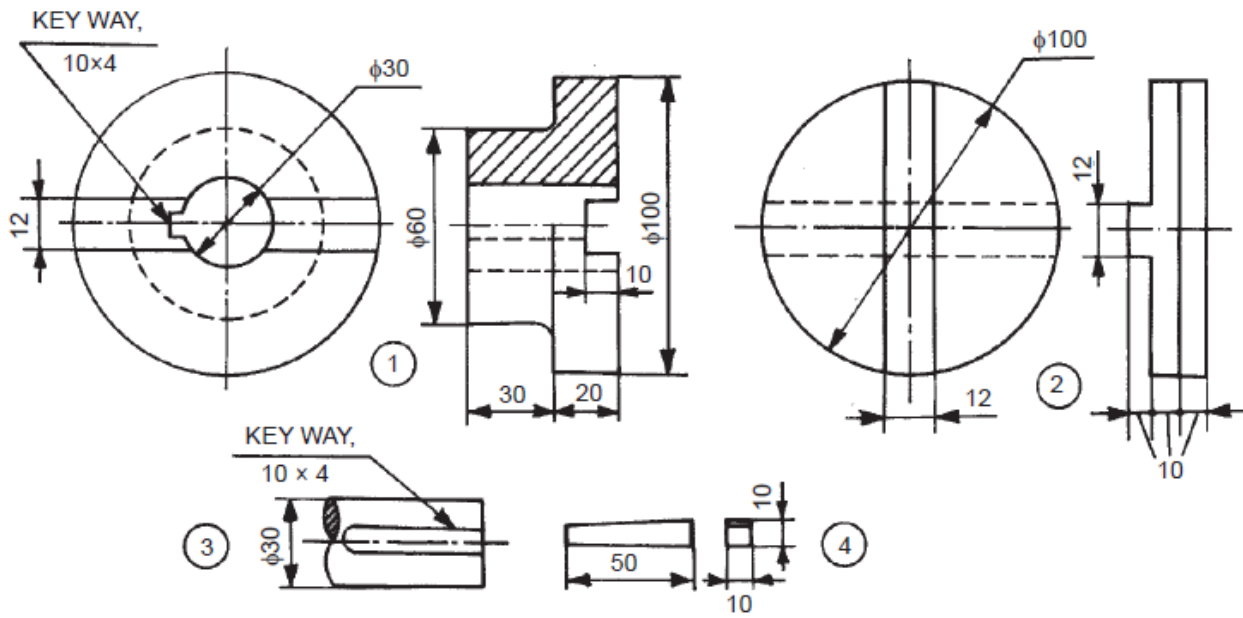
1. First draw the axis of first part which is the body, then draw the body using the commands as per the given dimensions.
2. Move the body apart and draw the next part using some different color.
3. Draw all the parts using different colors according to the given dimensions.
4. Assemble all the parts by placing the center of the axis of the parts in a manner by which they coincide.
5. Give the assembled figure dimensions to complete the figure.

Applications of Steam Engine Connecting Rod End:

Connecting rod in a steam engine connects the crosshead at one end (small end) and the crank at the other end (big end). The cross-section of the connecting rod can be square/circular in shape.

Exercise No. 8

Aim: To assemble the parts and draw (i) half sectional view from the front and (ii) the side view of the Oldham Coupling.



Parts list

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

Oldham Coupling.

Commands Used:

1. Line
2. Circle
3. Arc
4. Dimensions
5. Trim
6. Offset
7. Hatch

Procedure:

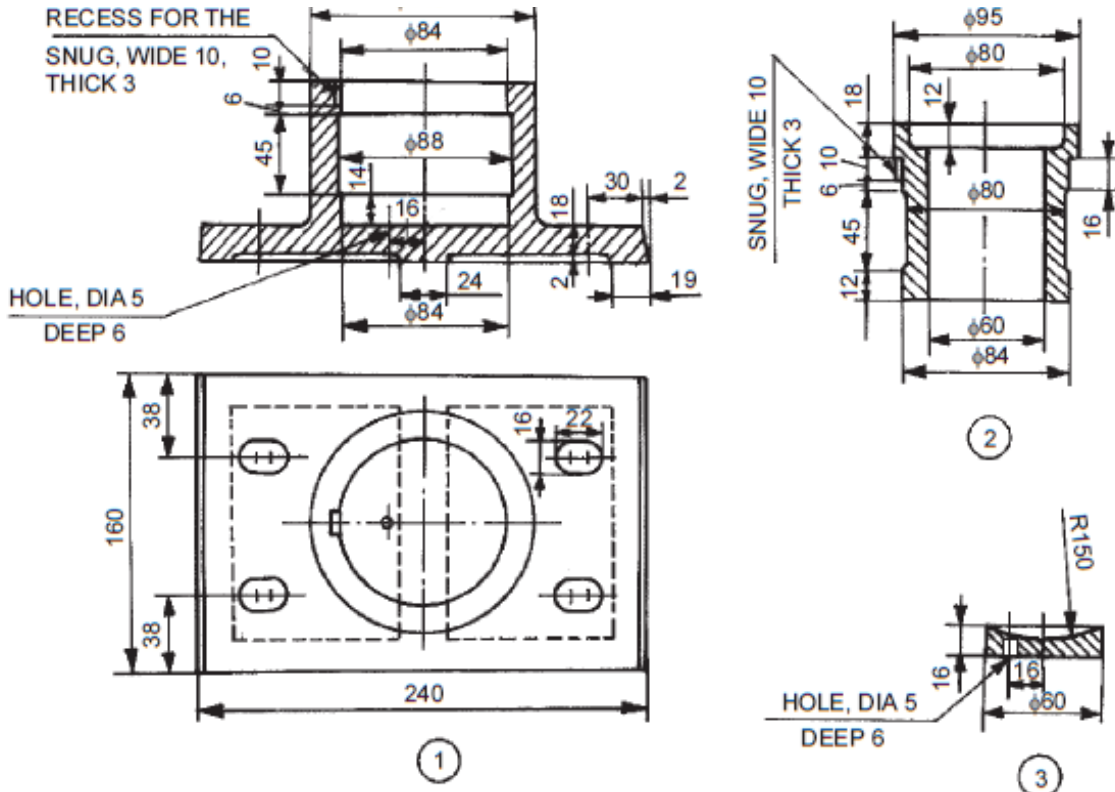
1. First draw the axis of first part of the Oldham coupling, then draw the part using the commands as per the given dimensions.
2. Move the part apart and draw the next part using some different color.
3. Draw all the parts using different colors according to the given dimensions.
4. Assemble all the parts by placing the centre of the axis of the parts in a manner by which they coincide.
5. Give the assembled figure dimensions to complete the figure.

Applications of Oldham Coupling:

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

Exercise No. 9

Aim: To assemble the parts and draw half sectional view from the front and top view of the Foot step bearing.



Parts list

Sl. No.	Name	Matl.	Qty.
1	Body	Cast iron	1
2	Bush	Brass	1
3	Disc	P Bronze	1
4	Shaft	Mild steel	1
5	Pin	Mild steel	1

Foot step bearing.

Commands Used:

1. Line
2. Circle
3. Arc
4. Dimensions
5. Trim
6. Fillet
7. Offset
8. Hatch

Procedure:

1. First draw the axis of first part of the Foot step bearing, then draw the first part using the commands as per the given dimensions.
2. Move the part apart and draw the next part using some different color.
3. Draw all the parts using different colors according to the given dimensions.
4. Assemble all the parts by placing the centre of the axis of the parts in a manner by which they coincide.
5. Give the assembled figure dimensions to complete the figure.

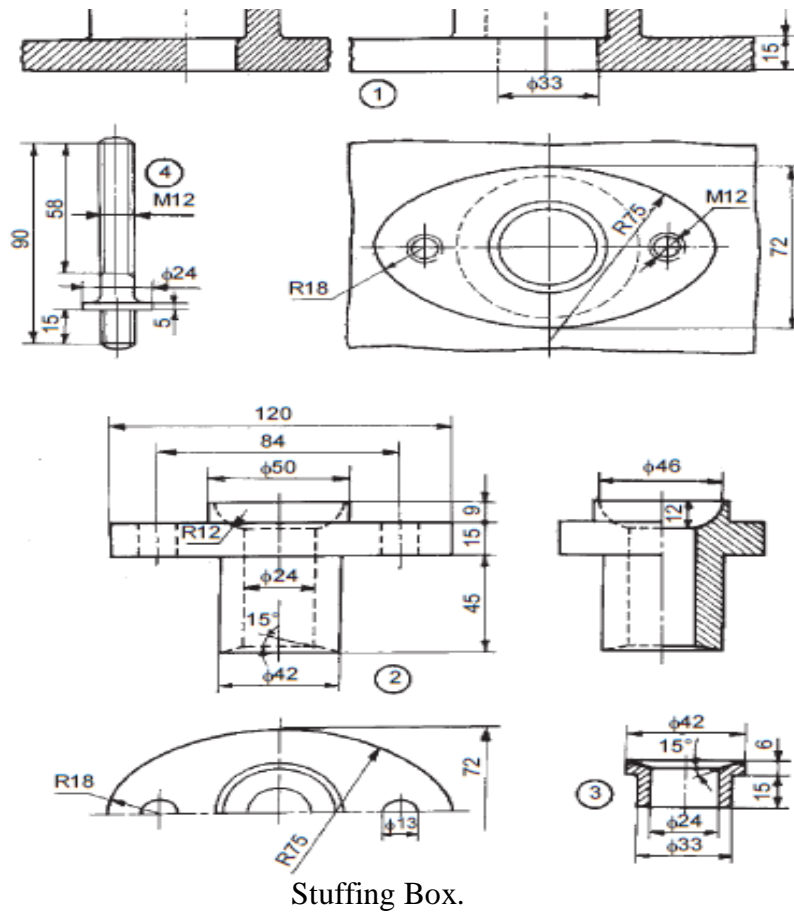
Applications of Foot step bearing:

This bearing is used to support a vertical shaft under axial load. Further, in this, the shaft is terminated at the bearing. The bottom surface of the shaft rests on the surface of the bearing which is in the form of a disc.

The disc is prevented from rotation by a pin inserted through the body, away from the centre. The bush is also prevented from rotation by a snug, provided at its neck, below the collar.

Exercise No. 10

Aim: To assemble the parts and draw half sectional view from the front, with right half in section of the Stuffing Box.



Command Used:

1. Line
2. Circle
3. Arc
4. Dimensions
5. Trim
6. Fillet

7. Offset
8. Hatch

Procedure:

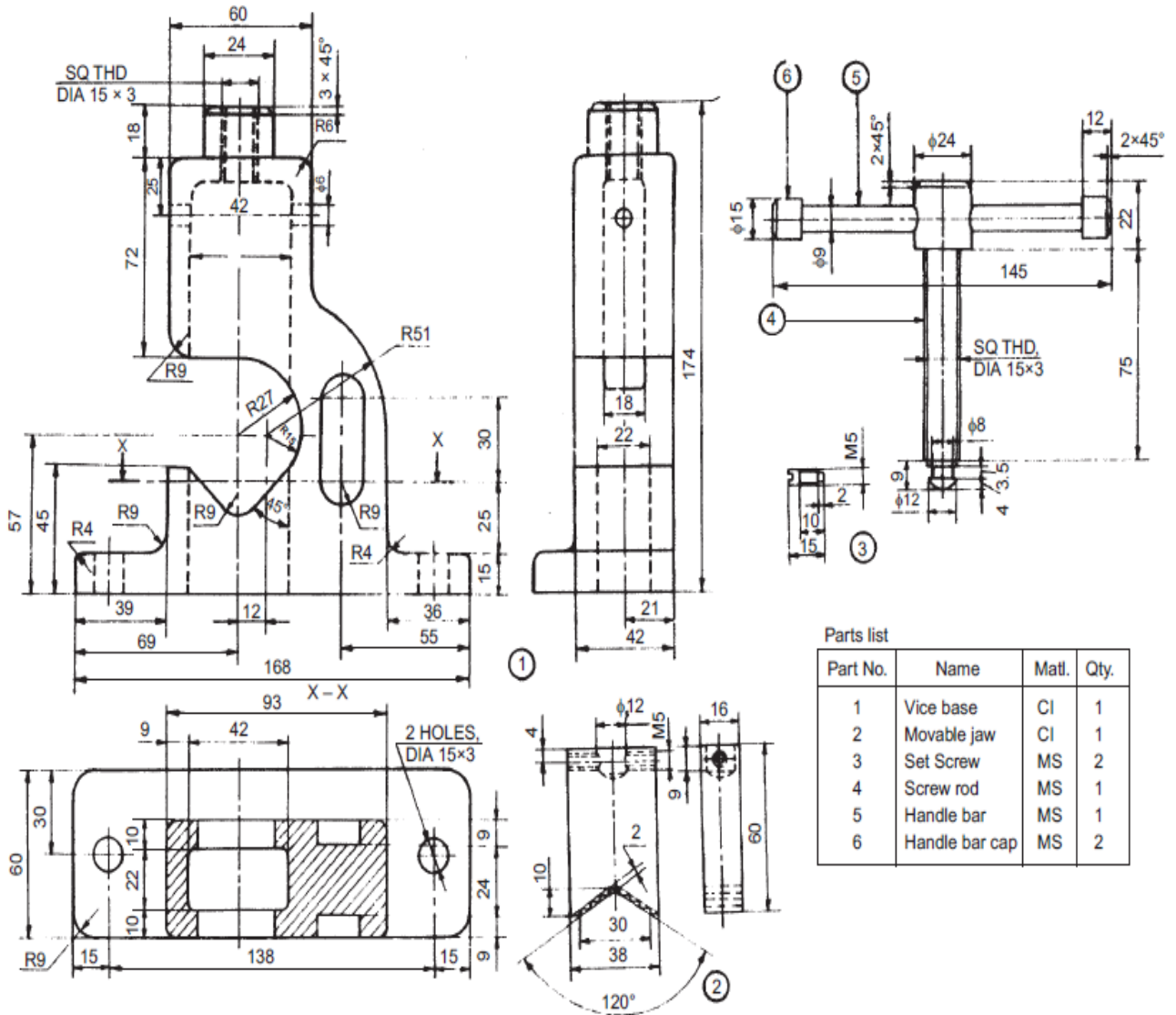
1. First draw the axis of first part which is the body, then draw the body using the commands as per the given dimensions.
2. Move the body apart and draw the next part using some different color.
3. Draw all the parts using different colors according to the given dimensions.
4. Assemble all the parts by placing the centre of the axis of the parts in a manner by which they coincide.
5. Give the assembled figure dimensions to complete the figure.

Applications of Stuffing Box:

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

Exercise No. 11

Aim: To assemble the parts and draw view from the front, with top half in section of the pipe vice



Pipe vice

Commands Used:

1. Line
2. Circle
3. Arc
4. Dimensions
5. Trim
6. Fillet
7. Offset
8. Hatch

Procedure:

1. First draw the axis of first part which is the body, then draw the body using the commands as per the given dimensions.
2. Move the body apart and draw the next part using some different color.
3. Draw all the parts using different colors according to the given dimensions.
4. Assemble all the parts by placing the centre of the axis of the parts in a manner by which they coincide.
5. Give the assembled figure dimensions to complete the figure.

Applications of pipe vice:

Pipe vice is a most commonly used tool. The V-shaped base can accommodate pipes of various diameters. The serrations provided on the end of the jaw provide grip on the pipe surface. Therefore the pipe vice is mainly used for performing operations on the pipes.