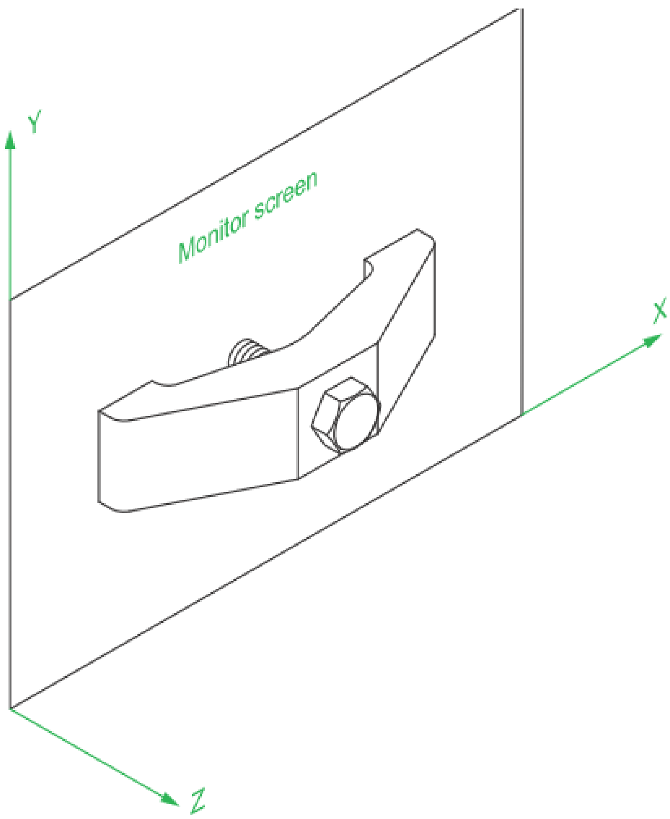
Northern Technical University

Technical College/ Kirkuk

Refrigeration and Air Conditioning Department



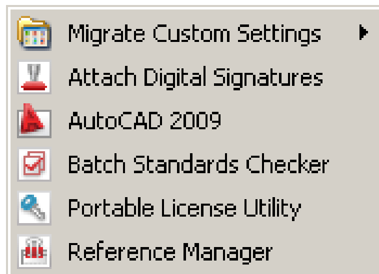
Engineering Drawing Using AutoCAD

Chapter 1

Introduction

1.1 Launching AutoCAD

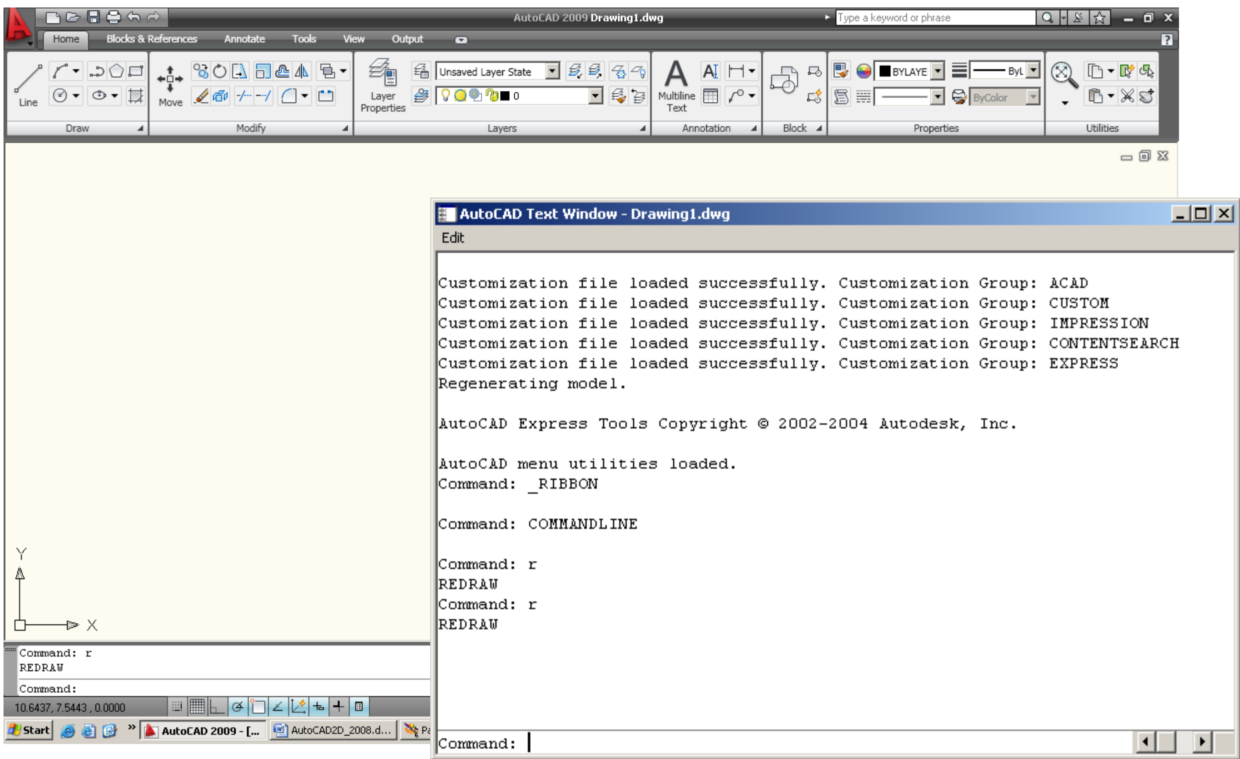
1. **Choose** Start from the Windows program manager.
2. **Choose** Programs, **Autodesk ,AutoCAD 2009**.
3. **Click** the AutoCAD 2009 for Windows icon.



1.2 Text and Graphics Screens

The graphics screen and the text screen are two different screens available in the drawing editor.

1. **Press** Function key **F2** on the keyboard.



1.3 Canceling a Command

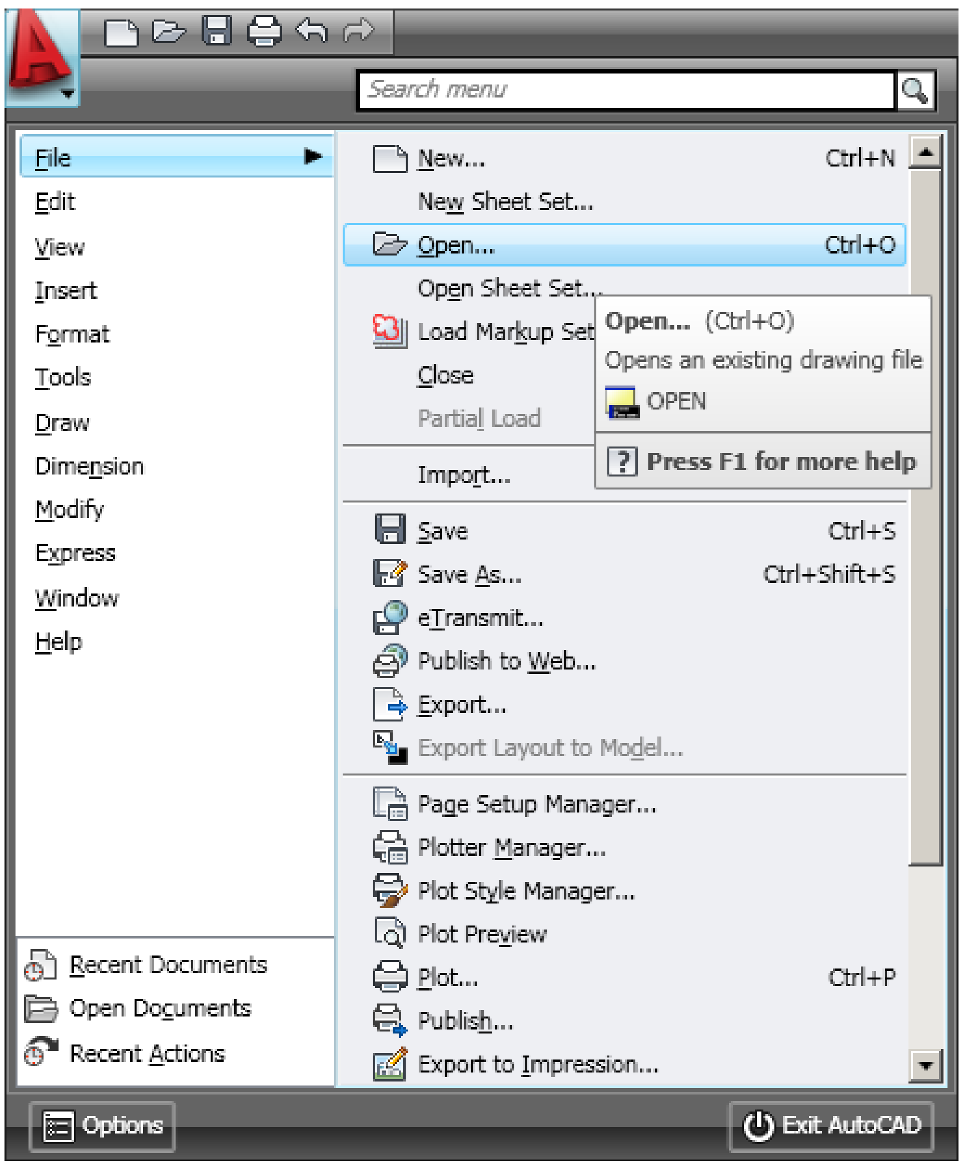
1. Press the **ESCAPE (ESC)** key on the keyboard.

TIP: Pressing ESC twice clears nested commands

1.4 Menus and Colors

Menu Browser

1. Click on the A icon in the upper left corner of the drawing area.
2. Click the desired pulldown menu.
3. Click on the command to be executed from the pulldown.



Quick Access Toolbar

1. Click on one of the following icons for quick access to commands QNEW, OPEN, SAVE, PLOT, and UNDO/REDO.

Right-click the Quick Access toolbar and click Customize Quick Access Toolbar. The Customize User Interface dialog opens and displays the list of commands available.

Drag commands you want to add from the command list pane in the Customize User Interface dialog box to the Quick Access toolbar.

Info Center

Quickly search for a variety of information sources, access product updates and announcements, and save topics with InfoCenter.

Ribbon

The ribbon provides a single, compact placement for operations that are relevant to the current workspace. It eliminates the need to display multiple toolbars, reducing clutter in the application window. The ribbon maximizes the area available for work using a single compact interface.

The ribbon can be displayed horizontally, vertically, or as a floating palette. The horizontal ribbon is displayed at the top of the drawing window by default when you create or open a drawing.

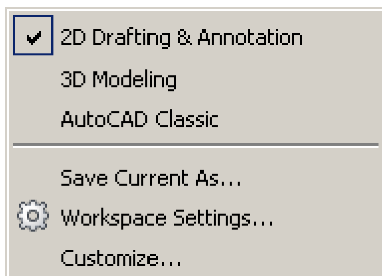
You can create your own panels to display on the ribbon; you can also modify the commands and controls on existing ribbon panels.

1.5 Workspaces

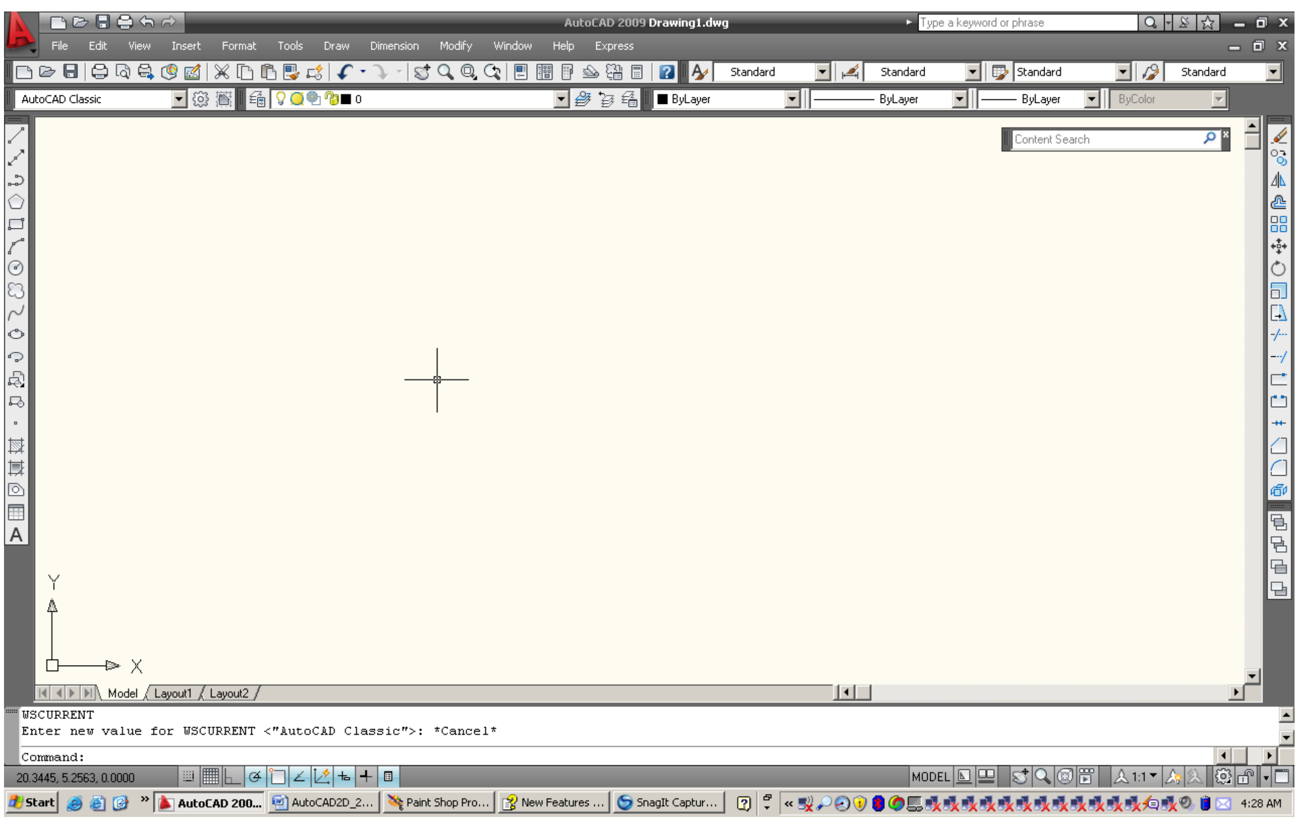
You can switch between the workspaces from the menu browser.

1. Click the Workspace switching icon in the lower left corner of the screen.

2. Click on one of the following workspace options



AutoCAD classic workspace



1.6 AutoCAD ClassicToolbars

Toolbars can be docked on the screen or they can float about the screen.

To Float a Toolbar:

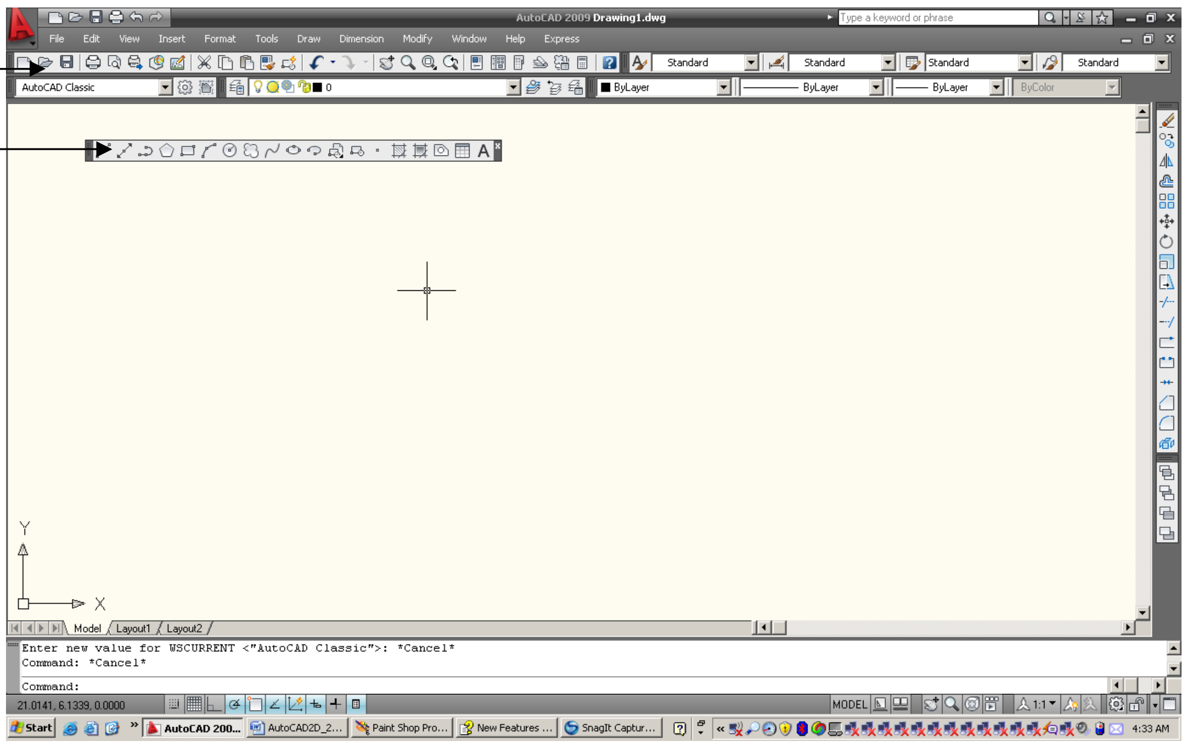
1. Choose the gray border surrounding each tool.
2. Drag the toolbar to any area on the screen.

To Dock a Toolbar:

1. Choose the title or gray border of the toolbar.
2. Drag the toolbar to the top, bottom, left, or right area of the graphics display.

Docked Toolbars

Floating Toolbars

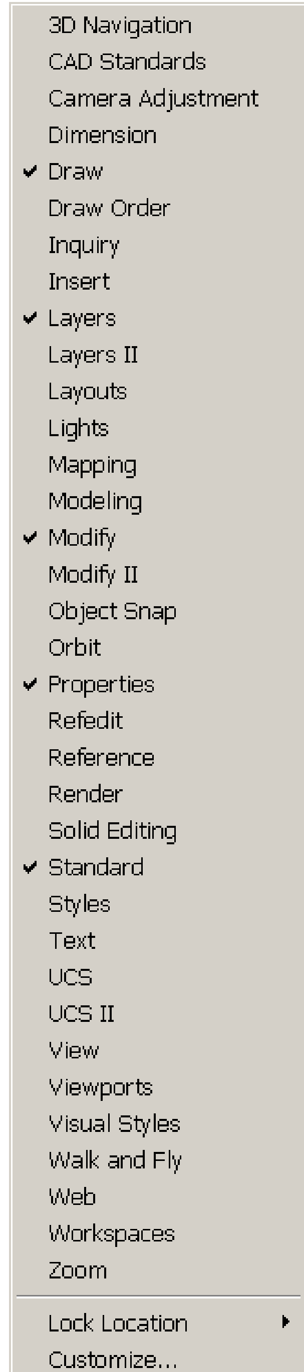


TIP: -Holding the CTRL key while dragging will prevent docking.

Loading Toolbars

Right-clicking on an icon in any toolbar

This will show a list of all available toolbars.



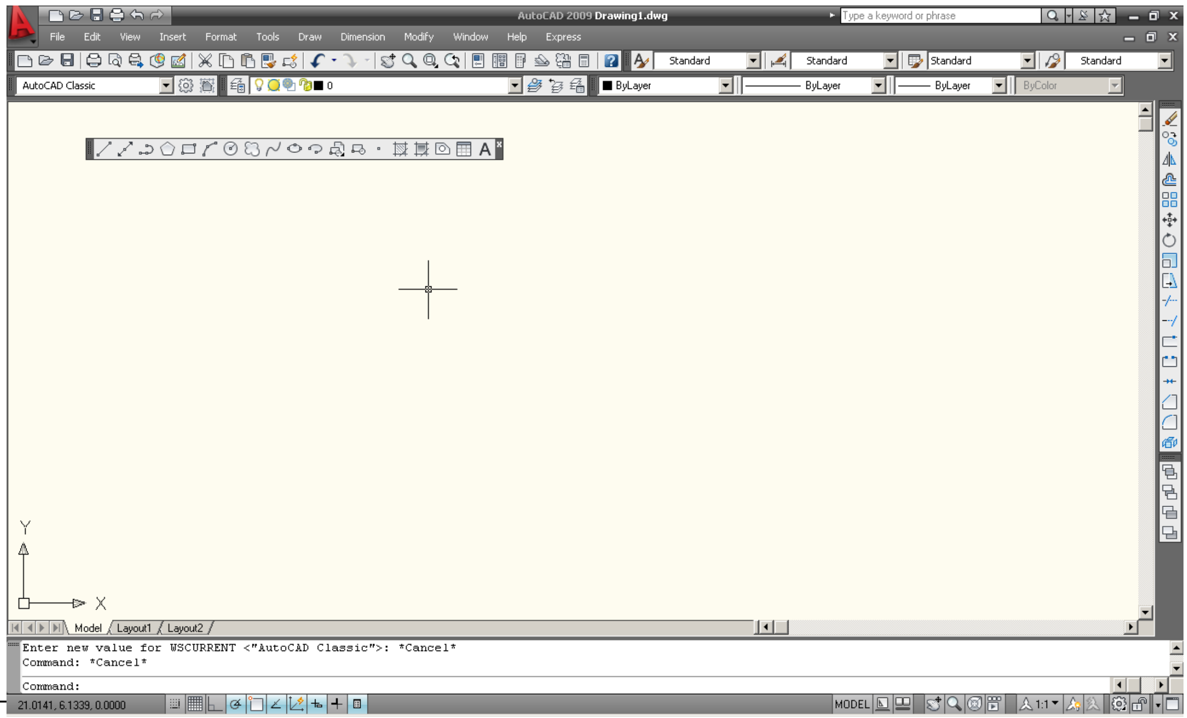
Help Tooltips

1. Move the mouse to the toolbar but do not pick the button.

1.7 Status Bar and Command Prompt

The Status Bar is the area below the command line that shows messages as well as coordinates, modes, and the current time.

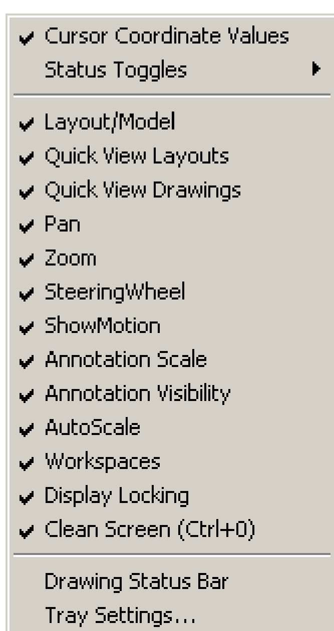
To activate SNAP, GRID, ORTHO, OSNAP, MSPACE, PSPACE, and TILE, you must double-click on the mode to change.



Status Bar

TIP:

- Right click on the blank area of the status bar to see the tools to turn off/on.



1.8 Typing Commands

Typing a Command

All AutoCAD commands can be typed in at the command line. Many commands also have one or two letter aliases that can also be typed as shortcuts to the commands.

1. Type the desired command at the command prompt.

   Command : **LINE**

   or

2. Type the command's alias.
   Command: **L**

3. Press **ENTER**.

4. Type an option at the command prompt.

TIP:Many AutoCAD commands require you to press ENTER to complete the command. You know you are no longer in an AutoCAD command when you see a blank command line.

Reissuing the Last Command

The last used AutoCAD command can be re-entered by one of the following three methods of ENTER. The ENTER key on the keyboard will always act as ENTER, the SPACEBAR and RIGHT MOUSE will act as enter most of the time .

1. Press the **ENTER** key on the keyboard

   or

2. Press the **Space bar** on the keyboard.

   or

3. Click the right mouse button.

1.9 Pointing Device (Mouse)

AutoCAD uses either a mouse or digitizing tablet to select objects in a drawing.

Left Mouse Button

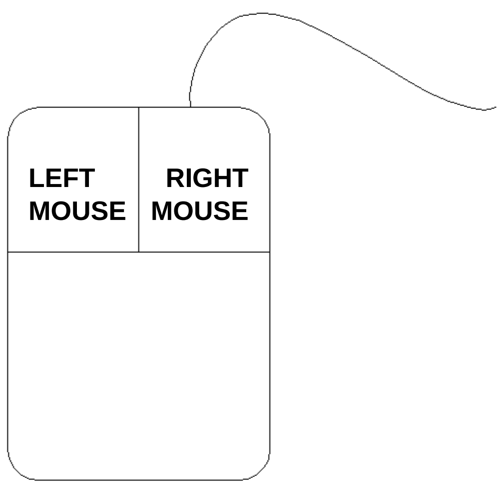
Used to pick or select objects

1. Click the left mouse button to select an object area in the drawing.
2. Press **ESC twice** to deselect an object (or to cancel a command).

Right Mouse Button

Used to enter a command, repeat last command, or access shortcut menus.

1. Click the right mouse button.



TIPS:

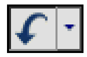
- SHIFT + the right mouse button brings up the object snap menus.

1.10 Undo and Redo

Reverses the last action.

1. Choose **Edit, Undo**.

   or

2. Click the Undo icon. 

   or

3. Press **CTRL + Z**.
4. Type U at the command prompt to undo the last command.

   Command: **U**

Redo

Reverses the effects of a single UNDO or U command.

1. Choose **Edit, Redo**.

   or

2. Click the Redo icon. 

   or

3. Type REDO at the command prompt to redo the last undo command.

   Command: **REDO**

1.11 Function Keys and Accelerator Keys

F1	Displays Help
F2	Toggles Text Window
F3	Toggles OSNAP
F4	Toggles TABMODE
F5	Toggles ISOPLANE
F6	Toggles UCSDETECT
F7	Toggles GRIDMODE
F8	Toggles ORTHOMODE
F9	Toggles SNAPMODE
F10	Toggles Polar Tracking
F11	Toggles Object Snap Tracking
F12	Toggles Dynamic Input

1.12 Open Existing Drawings

1. **Choose** File, OPEN.

   or

2. **Press** CTRL + O.

   or

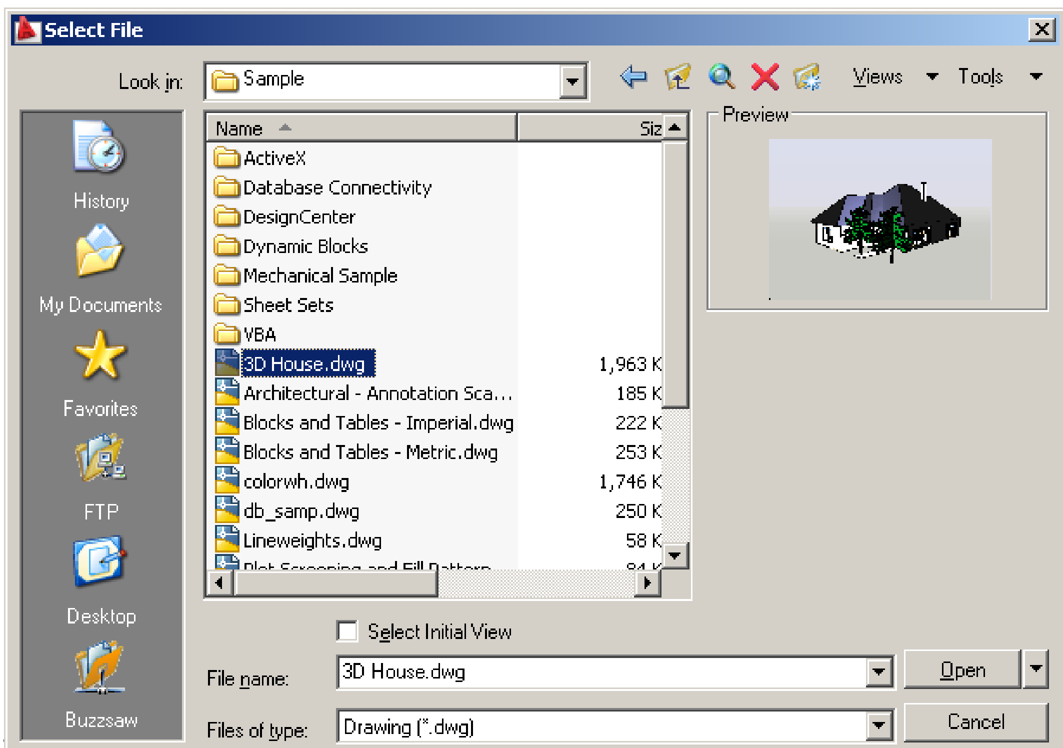
3. **Click** the OPEN icon.

   or

4. **Type** OPEN at the command prompt.

   Command: **OPEN**

5. **Press** ENTER
6. **Double Click** the desired directory to find the drawing to open.
7. **Click** the drawing name to open.
8. **Click** The OK button.

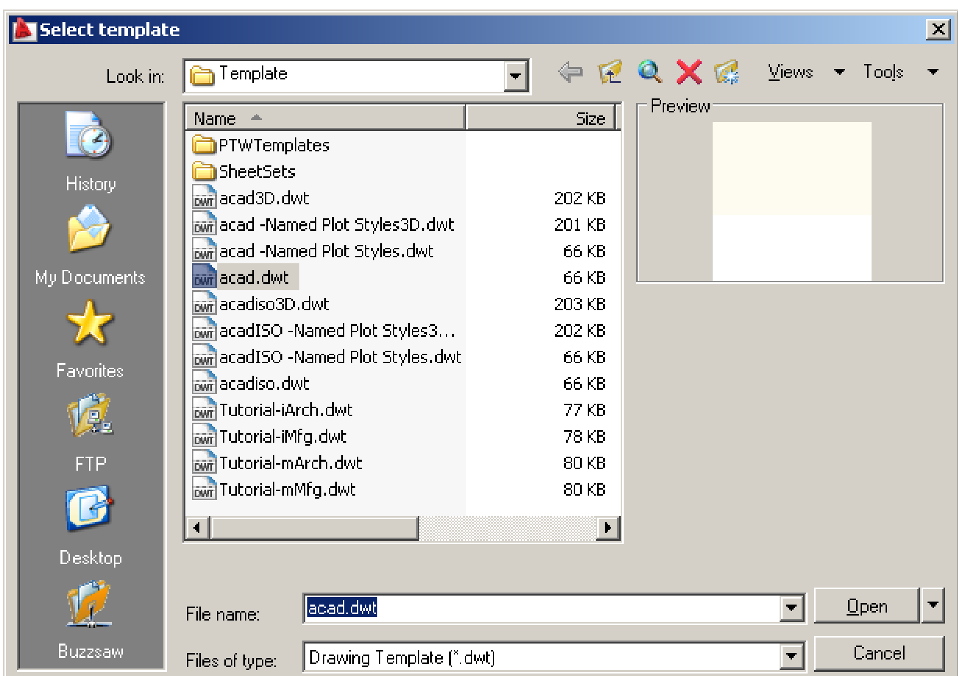


1.12 Creating a New Drawing

NEW Command

Creates a new drawing file.

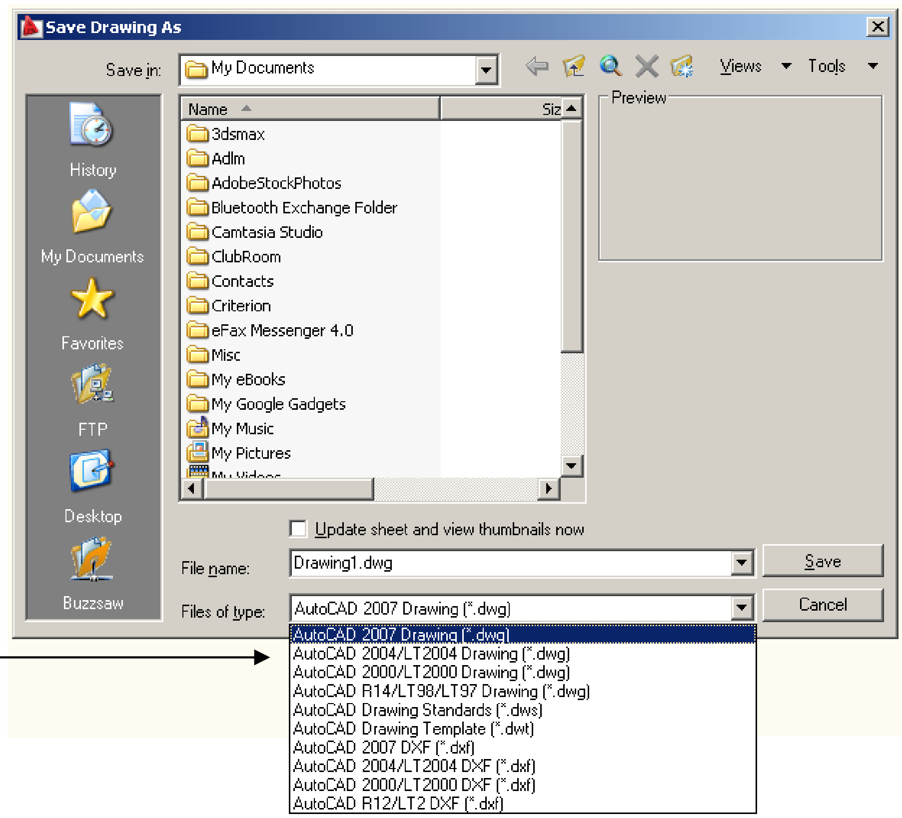
1.	**Choose**	File, New.
		or
2.	**Press**	CTRL + N
		or
3.	**Click**	the New icon.
		or
4.	**Type**	NEW at the Command prompt.
		Command: **NEW**
5.	**Choose**	One of the options for creating a new drawing.
6.	**Click**	The OK button.
7.	**Save**	the drawing as another name.



1.13 Saving Drawings

Saves the most recent changes to a drawing. The first time an unnamed drawing is saved the "Save As" dialog box appears. AutoCAD saves its drawings as files with extensions ending in .DWG.

1. **Choose** **File, Save or Saveas**.
   or
2. **Type** SAVE or SAVEAS at the command prompt.
   Command: **SAVE** or **SAVEAS**
3. **Press** **ENTER**
4. **Type** A new drawing name or keep the existing drawing name.
5. **Click** The OK button.



Various file types the drawings can be saved as

TIP:

Clicking the dropdown list for File type changes the format that the drawing can be saved in.

1.14 Exiting AutoCAD

1. **Choose** File, Exit.

   or

2. **Type** QUIT at the command prompt.

   Command: **QUIT**

3. **Press** ENTER
4. **Click** Yes to save changes or No to discard changes.

Chapter 2

PRINCIPLES OF DRAWING

PRINCIPLES OF DRAWING

2.1 INTRODUCTION

Engineering drawings are to be prepared on standard size drawing sheets. The correct shape and size of the object can be visualised from the understanding of not only the views of it but also from the various types of lines used, dimensions, notes, scale, etc. To provide the correct information about the drawings to all the people concerned

2.2 DRAWING SHEET

Engineering drawings are prepared on drawing sheets of standard sizes. The use of standard size sheet, saves paper and facilitates convenient storage of drawings.

2.2.1 Sheet Sizes

The basic principles involved in arriving at the sizes of drawing sheets are:

(*a*) $X : Y = 1 : \sqrt{2}$, (*b*) $XY = 1$

where X and Y are the sides of the sheet. For a reference size A0 (Table 2.1) having a surface area of 1 m^2, X = 841 mm and Y = 1189 mm. The successive format sizes are obtained either by halving along the length or doubling along the width, the areas being in the ratio 1:2 (Fig. 2.1).

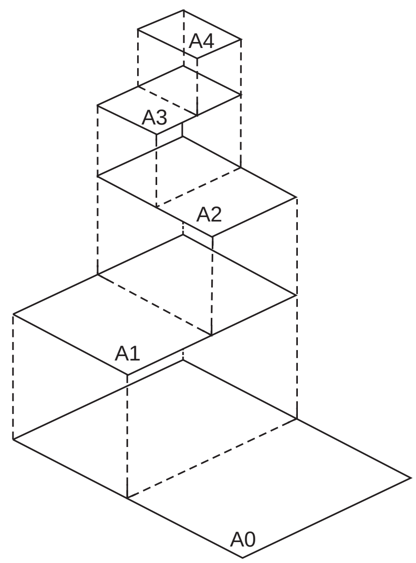


Fig. 2.1 Drawing sheet formats

2.2.2 Designation of Sizes

The original drawing should be made on the smallest sheet, permitting the necessary clarity and resolution. The preferred sizes according to ISO-A series (First choice) of the drawing sheets are given in Table 2.1. When sheets of greater length are needed, special elongated sizes (Second choice) are used (Table 2.2). These sizes are obtained by extending the shorter sides of format of the ISO-A series to lengths that are multiples of the shorter sides of the chosen basic format.

Table 2.1 Preferred drawing sheet sizes (First choice) ISO-A Series

Designation	*Dimensions* (mm)
A0	841 × 1189
A1	594 × 841
A2	420 × 594
A3	297 × 420
A4	210 × 297

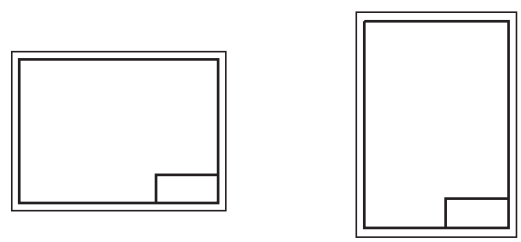
Table 2.2 Special elongated sizes (Second choice)

Designation	*Dimensions* (mm)
A3 × 3	420 × 891
A3 × 4	420 × 1188
A4 × 3	297 × 630
A4 × 4	297 × 840
A4 × 5	297 × 1050

2.2.3 Title Block

The title block should lie within the drawing space such that, the location of it, containing the identification of the drawing, is at the bottom right hand corner. This must be followed, both for sheets positioned horizontally or vertically (Fig. 2.2).

The direction of viewing of the title block should correspond in general with that of the drawing. The title block can have a maximum length of 170 mm. Figure 2.3 shows a typical title block, providing the following information:



(a) (b)

Fig. 2.2 Location of title block

(*i*) Title of the drawing
(*ii*) Sheet number
(*iii*) Scale
(*iv*) Symbol, denoting the method of projection
(*v*) Name of the firm
(*vi*) Initials of staff drawn, checked and approved.

NOTE According to Bureau of Indian Standards, SP-46:1998, "Engineering Drawing Practice for Schools and Colleges", First angle projection is preferred.

2.2.4 Borders and Frames

Borders enclosed by the edges of the trimmed sheet and the frame, limiting the drawing space, should be provided with all sheet sizes. It is recommended that these borders have a minimum width of 20 mm for the sizes A0 and A1 and a minimum width of 10 mm for the sizes A2, A3 and A4 (Fig. 2.4). A filing margin for taking perforations, may be provided on the edge, far left of the title block.

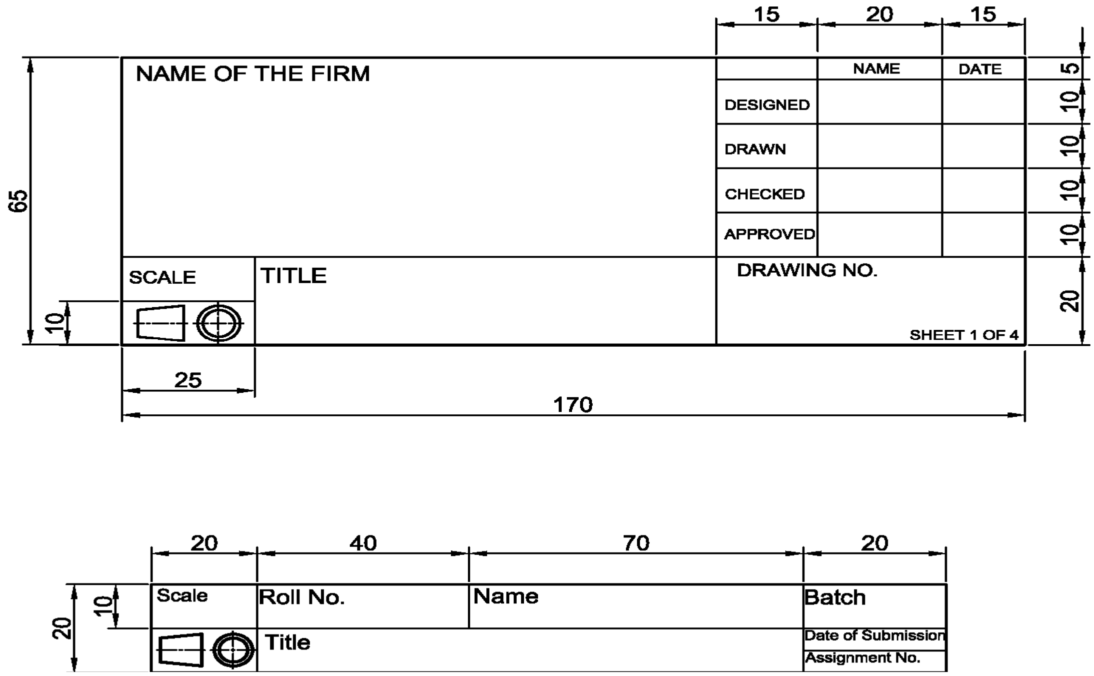


Fig. 2.3 Details in title block

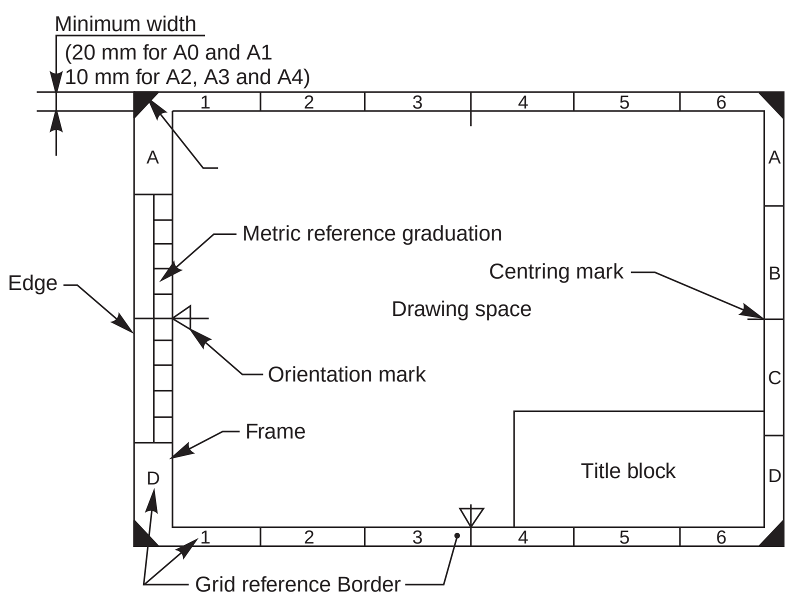


Fig. 2.4 Drawing sheet layout

2.3 SCALES

Scale is the ratio of the linear dimension of an element of an object as represented in the drawing, to the real linear dimension of the same element of the object itself. Wherever possible, it is desirable to make full size drawings, so as to represent true shapes and sizes. If this is not practicable, the largest possible scale should be used. While drawing very small objects, such as watch components and other similar objects, it is advisable to use enlarging scales.

2.3.1 Designation

The complete designation of a scale should consist of the word Scale, followed by the indication of its ratio as:

SCALE 1 : 1 for full size,

SCALE × : 1 for enlarged scales,

SCALE 1 : × for reduced scales.

The designation of the scale used on the drawing should be shown in the title block.

2.3.2 Recommended Scales

The recommended scales for use on technical drawings are given in Table 2.3. The scale and the size of the object in turn, will decide the size of the drawing.

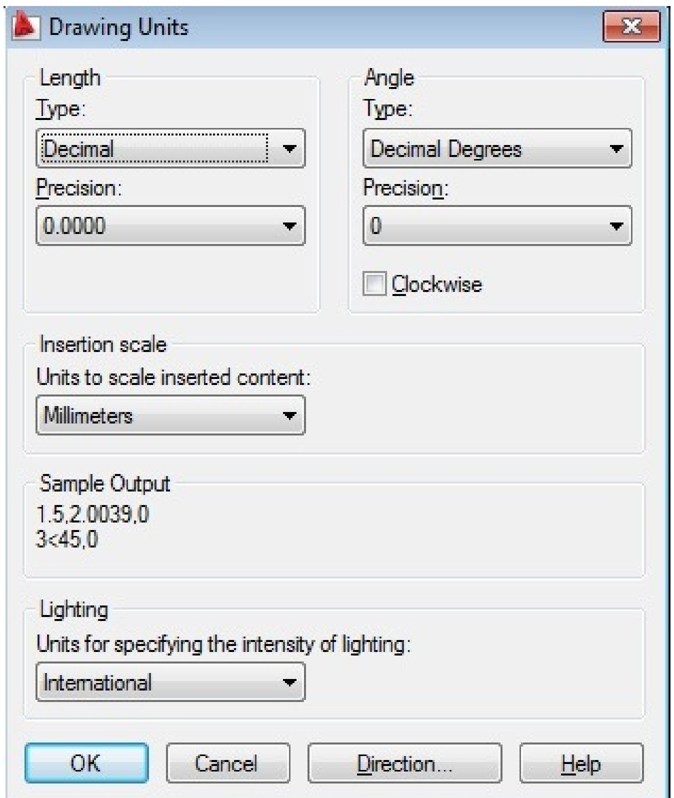
Table 2.3 Recommended scales

Category	*Recommended Scales*		
Enlarged scales	50:1	20:1	10:1
	5:1	2:1	
Full size			1:1
Reduced scales	1:2	1:5	1:10
	1:20	1:50	1:100
	1:200	1:500	1:1000
	1:2000	1:5000	1:10000

2.4 Setting up a drawing

2.4.1 UNITS Command

1. **Choose** Format, Units...
   or
2. **Type** DDUNITS at the command prompt.
   Command: **DDUNITS or UN**
3. **Choose** a units and angle setting.
4. **Choose** a precision setting.



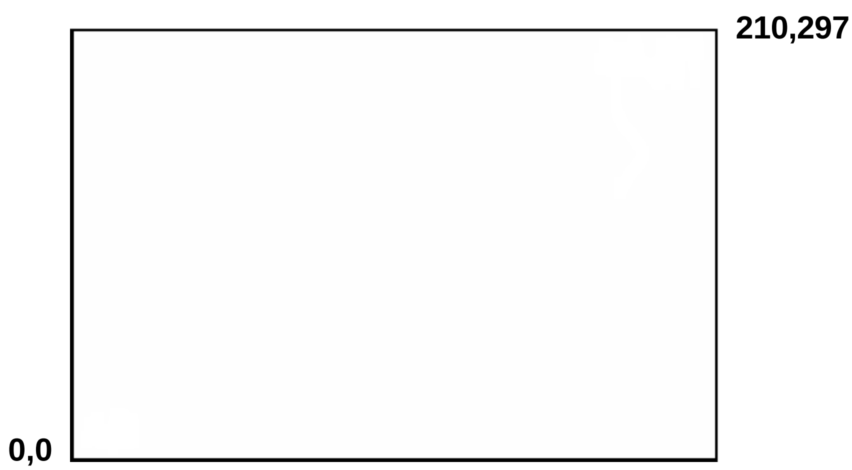
2.4.2 Drawing limits

The drawing limits are two-dimensional points in the World Coordinate System that represent a lower-left limit and an upper-right limit.

The drawing limits also govern the portion of the drawing covered by the visible grid and determine the minimum area a ZOOM All displays.

1.	**Choose**	Format, Drawing Limits.
		or
2.	**Type**	LIMITS at the command prompt
		Command: **LIMITS**
3.	**Type**	One of the following options On/Off/Lower left corner <.000,0.000>: **0,0**
4.	**Type**	One of the following options for the upper right limit:
		Upper right corner <420.00,297.00>: **210,297**

Drawing with lower left limit of 0,0 and upper right limit of **210,297**



TIPS:

You can also pick points to define the limits.

The limcheck variable controls whether or not you can draw outside the limits that are set. A setting of 0 (off) indicates that you can draw outside the limits and a setting of 1(on) indicates that you cannot.

Chapter 3

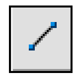
Draw Commands

3.1 Line Command

Creates single straight line segments

1. **Choose** Draw, Line.

or

2. **Click** the Line icon. 

or

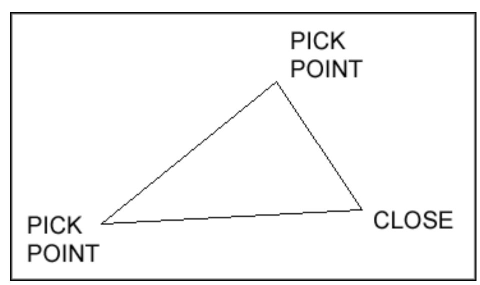
3. **Type** LINE from the command prompt
   Command: **LINE** or **L**
4. **Press** ENTER
5. **Pick** From point: (**point**)
6. **Pick** Specify next point or [Close/Undo]:(**point**)
7. **Pick** Specify next point or [Close/Undo]:(**point**)
8. **Press** ENTER to end line sequence

or

9. **Type** U to undo the last segment
   To point: **U** (undo)

or

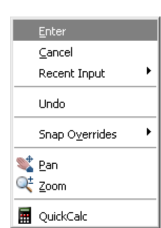
10. **Type** C to create a closed polygon
    To point : **C** (close)



TIPS:

• You can continue the previous line or arc by responding to the From point: prompt with a space or ENTER.

• Choose the right mouse button for the line pop-up menu to appear while in the line command

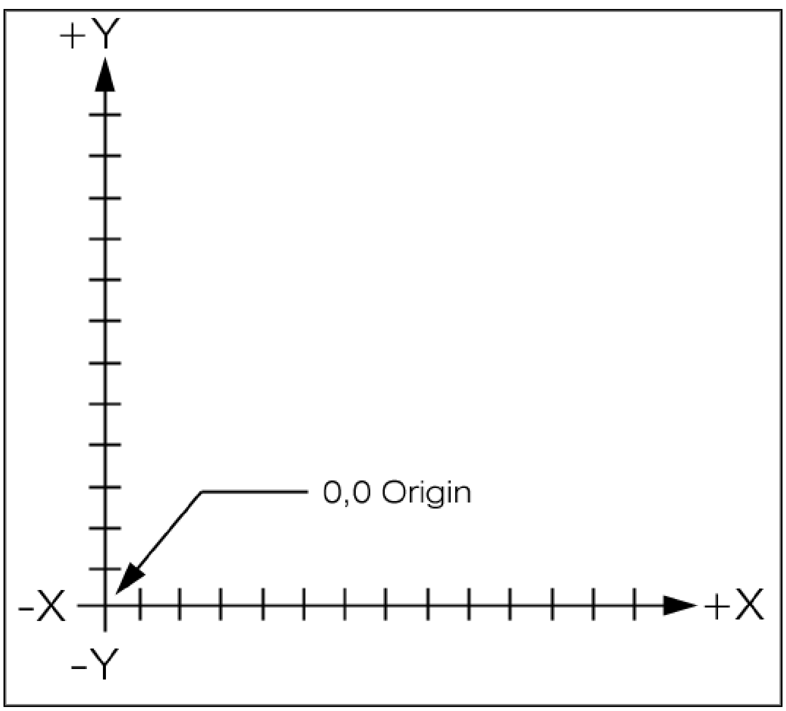


3.2 Cartesian Coordinate System

AutoCAD provides the user with an infinite two dimensional area to work with. Any entities place on the working two dimensional plane can be defined relative to the Cartesian coordinate system.

The Cartesian coordinate system divides a two dimensional plane with two perpendicular axis. The X axis runs horizontal across the bottom of the screen. The Y axis runs vertically along the left side of the screen. These two axis intersect at the bottom left corner of the screen.

Each of these axis is further divided into segments. Each segment is given a value. The X axis segments increase in value to the right. The positive X values are to the right of the intersection of the two axis. The negative X values are to the left. The positive Y values are above the intersection and increase up. The negative Y values are below.



Absolute Coordinates

1. **Type** x,y coordinate when AutoCAD asks for a point.

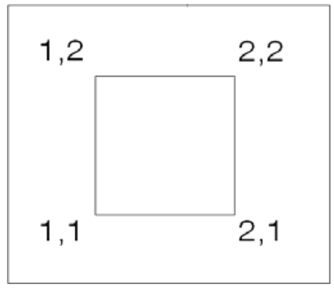
From point: **1,1**

To point: **2,1**

To point: **2,2**

To point: **1,2**

To point: **1,1**



NOTE: If dynamic input (F12) is on, you must type the # sign before entering absolute coordinates (e.g.#1,1).

Relative Coordinates

1. **Type** @deltax,deltay when AutoCAD asks for a point.

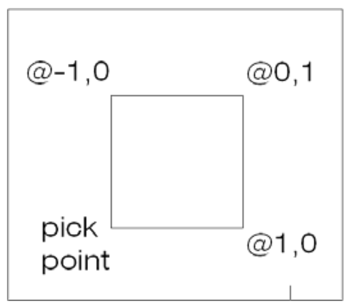
From point pick point

To point: **@1,0**

To point: **@0,1**

To point: **@-1,0**

To point: **@0,-1**



Polar Coordinates

1. **Type** @distance<angle when AutoCAD asks for a point.

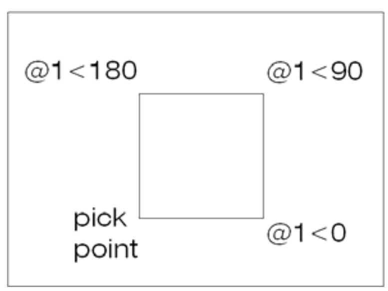
From point: pick point

To point:**@1<0**

To point:**@1<90**

To point:**@1<180**

To point:**@1<270**



3.3 Dynamic Input

Dynamic Input provides a command interface near the cursor to help you keep your focus in the drafting area.

When Dynamic Input is on, tooltips display information near the cursor that is dynamically updated as the cursor moves. When a command is active, the tooltips provide a place for user entry.

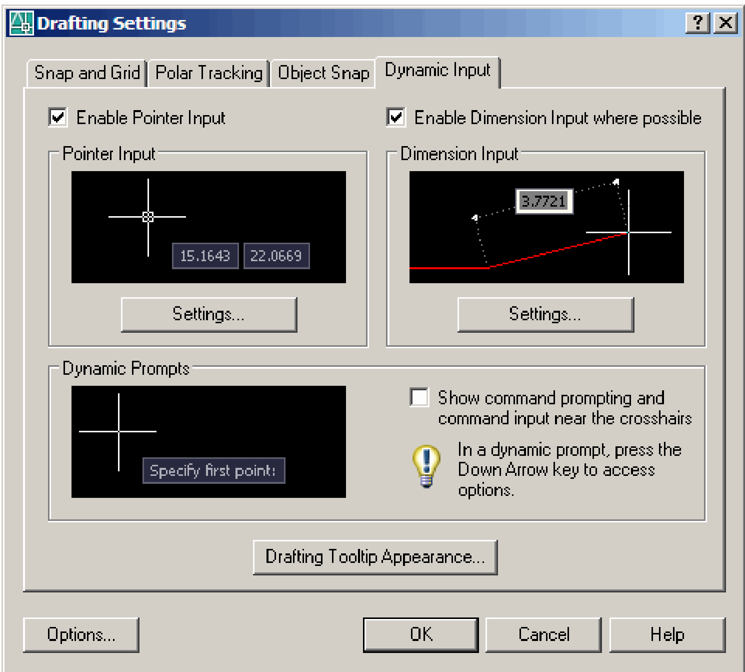
Turning Dynamic Input ON/OFF

1. Click **Dyn** on the status bar

   or

2. Press **F12**

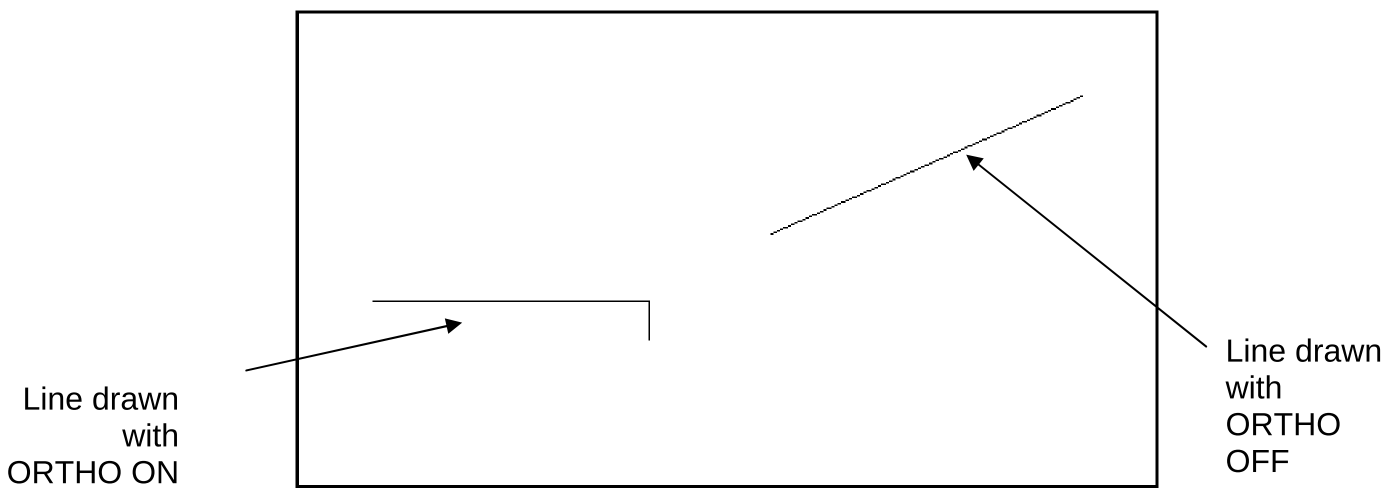
Tip: Right-click Dyn and click Settings to control what is displayed by each component when Dynamic Input is on.



3.4 Orthogonal Lines

Controls lines from being drawn at various angles to straight lines. When the snap grid is rotated, ortho mode rotates accordingly.

1.	**Press**	Function Key **F8**.
		or
2.	**Double Click**	ORTHO from the Status Bar.
		or
3.	**Press**	CTRL + L.



3.5 Polar Tracking

Polar Snaps work independently from snaps. With Polar Snaps on, AutoCAD shows the distances and angles being displayed as the cursor moves.

1. **Choose** Tools, Drafting Settings

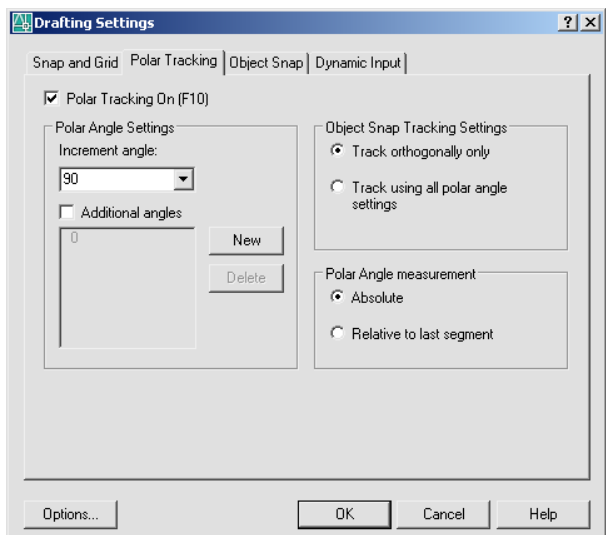
   or

2. **Type** DDSETTINGS at the command prompt.

   Command : **DDESTTINGS**

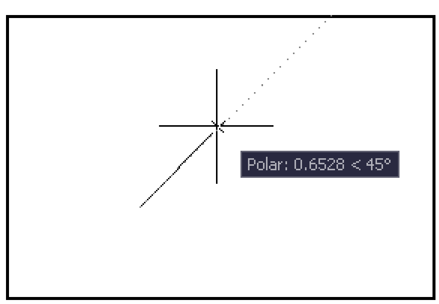
3. **Choose** the Polar trackingTAB from the dialog box.

4. **Select** the desired incremental angle from the dropdown list (or create a new angle).



5. **Pick** OK to exit the dialog box.

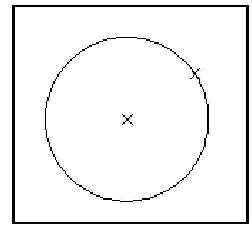
6. **Draw** a LINE using the Polar Snap references.



3.6 Circles

Circle Command

Circle, Center Radius



1. **Choose** Draw, Circle.

   or

2. **Click** the Circle icon. 

   or

3. **Type** CIRCLE at the command prompt.

   Command: **CIRCLE**

4. **Type** One of the following options: 3P/2P/TTR/<<center point>>:

   or

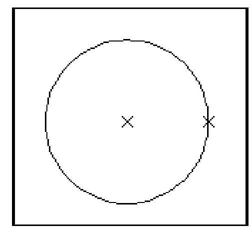
5. **Pick** A center point.
6. **Type** A radius or diameter.

   or

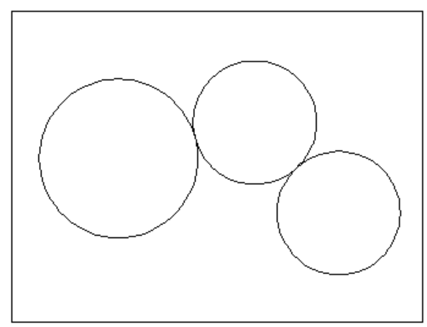
7. **Pick** A radius or diameter

   Diameter/<<radius>>:

Circle, Center Diameter

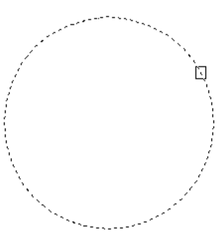


Circle, Tangent, Tangent Radius

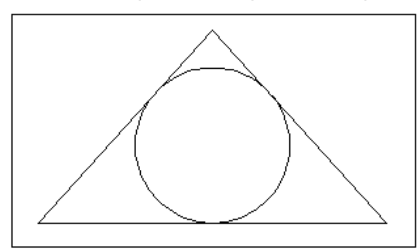


TIPS:

- To create circles that are the same size, press ENTER when asked for the circle radius.
- When selecting a circle with a pickbox, be sure to select the circumference of the circle.



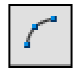
Circle, Tangent, Tangent, Tangent



3.7 Arc Command

1. **Choose** Draw, Arc.

   or

2. **Click** the Arc icon. 

   or

3. **Type** ARC at the command prompt

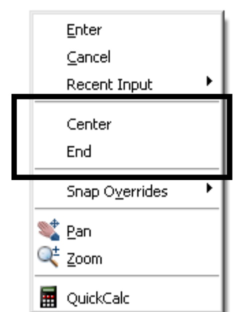
   Command: **ARC**

4. **Draw** One of the arcs.

TIPS:

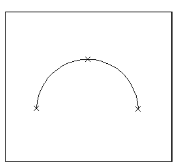
-Except for 3 point arcs, arcs are drawn in a COUNTERCLOCKWISE direction.

- While in the arc command, press the right mouse button to select the following options for arcs:

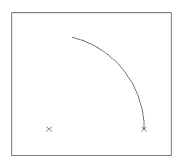


Arc Examples

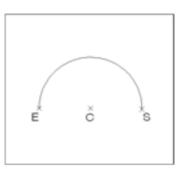
3 point arc



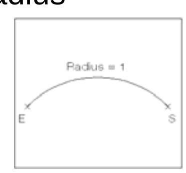
Start ,center, chord length



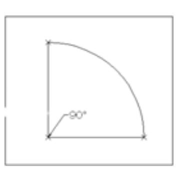
start, center, end



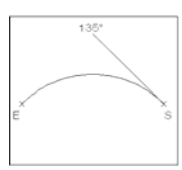
Start, end, radius



Start , center, included angle



Start, end, direction



Example (1):- Line tool

1. Open AutoCAD. The drawing area will show the settings of the **acadiso.dwt** template – **Limits** set to **420,297, Grid** set to **10, Snap** set to **5**

2. *Left-click* on the **Line** tool in the **2D Draw** control panel (Fig. 3.1).

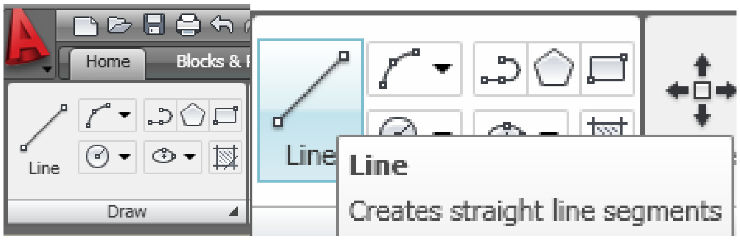


Fig. 3.1 The **Line** tool from the **2D Draw** control panel or from the **Draw** toolbar

3. Make sure **Snap** is on by either pressing the **F9** key or the **SNAP** button in the status bar. _**Snap on**_ will show in the command palette.

4. Move the mouse around the drawing area. The cursor's pick box will jump from point to point at 5 unit intervals. The position of the pick box will show as coordinate numbers in the status bar (left-hand end).

5. Move the mouse until the coordinate numbers show **60,240,0** and press the **Pick** button of the mouse (*left-click*).

6. Move the mouse until the coordinate numbers show **260,240,0** and *left-click*.

7. Move the mouse until the coordinate numbers show **260,110,0** and *left-click*.

8. Move the mouse until the coordinate numbers show **60,110,0** and *left click*.

9. Move the mouse until the coordinate numbers show **60,240,0** and *left click*.

Then press the **Return** button of the mouse (*right-click*).

Fig. 3.2 appears in the drawing area.

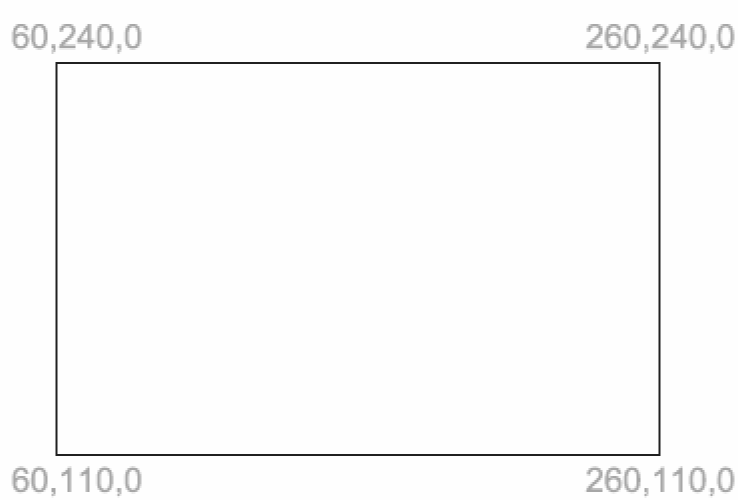


Fig. 3.2

Example (2):- Line tool

1. Clear the drawing from the screen with a *click* on the **Close** drawing button
Make sure it is not the AutoCAD 2009 window button.

2. The warning window appears in the centre of the screen. *Click* its **No** butt on.

3. *Left-click* on **New...** in the **File** drop-down menu and from the **Select template** dialog which appears *double-click* on **acadiso.dwt**.

4. *Left-click* on the **Line** tool icon and *enter* figures as follows at each prompt of the command line sequence:

Command:_line Specify first point: 80,235
Specify next point or [Undo]: 275,235
Specify next point or [Undo]: 295,210
Specify next point or [Close/Undo]: 295,100
Specify next point or [Close/Undo]: 230,100
Specify next point or [Close/Undo]: 230,70
Specify next point or [Close/Undo]: 120,70
Specify next point or [Close/Undo]: 120,100
Specify next point or [Close/Undo]: 55,100
Specify next point or [Close/Undo]: 55,210
Specify next point or [Close/Undo]: c (Close)

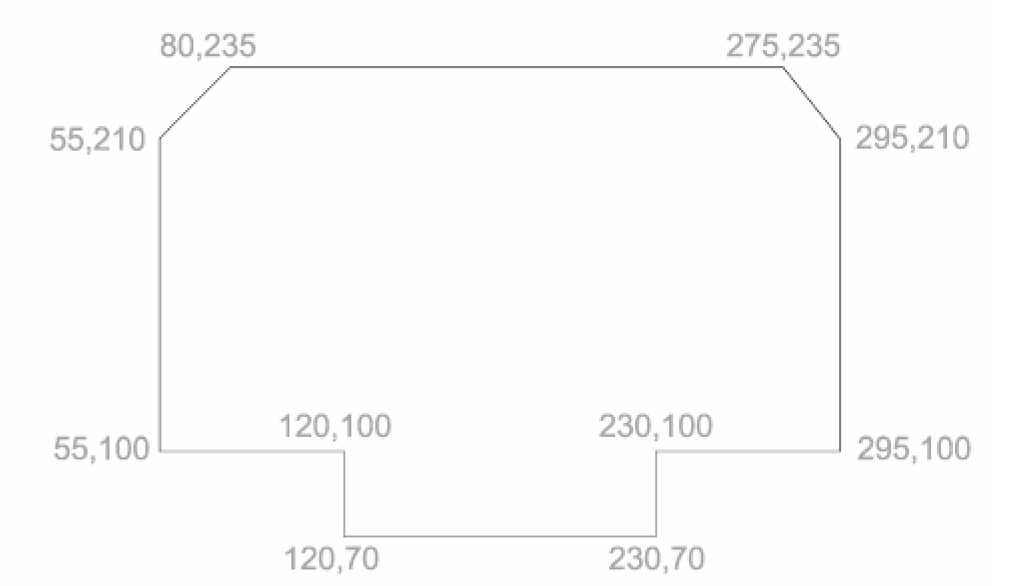


Fig.3.3

Example (3):- Line tool

1. Close the drawing and open a new **acadiso.dwt** window.

2. *Left-click* on the **Line** tool icon and *enter* figures as follows at each prompt of the command line sequence:

Command:_line Specify first point: 60,210 ↵
Specify next point or [Undo]: @50,0 ↵
Specify next point or [Undo]: @0,20 ↵
Specify next point or [Close/Undo]: @130,0 ↵
Specify next point or [Close/Undo]: @0,_20 ↵
Specify next point or [Close/Undo]: @50,0 ↵
Specify next point or [Close/Undo]: @0,_105 ↵
Specify next point or [Close/Undo]: @_50,0 ↵
Specify next point or [Close/Undo]: @0,_20 ↵
Specify next point or [Close/Undo]: @_130,0 ↵
Specify next point or [Close/Undo]: @0,20 ↵
Specify next point or [Close/Undo]: @_50,0 ↵
Specify next point or [Close/Undo]: c (Close) ↵

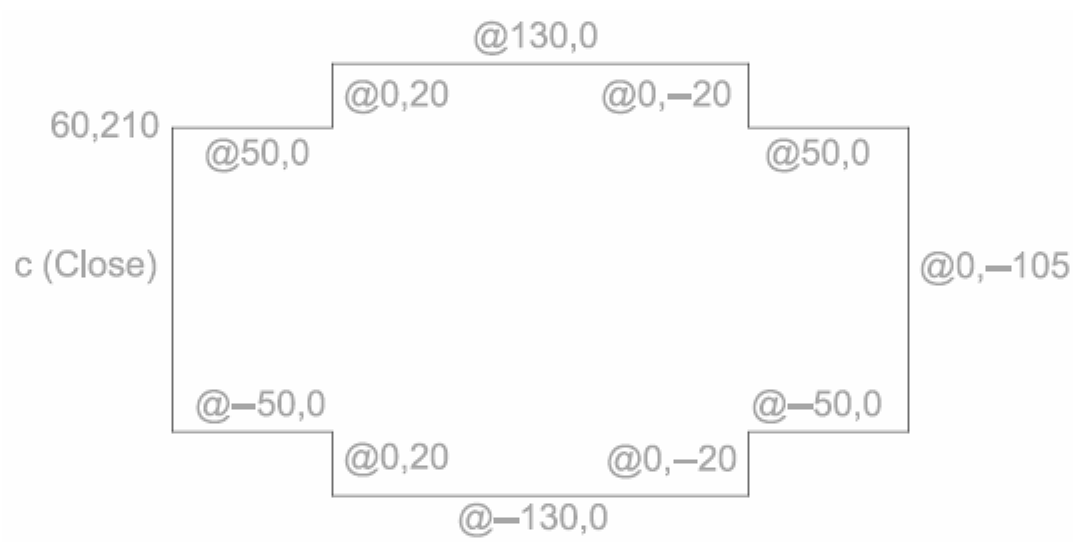


Fig. 3.4

Example (4):– Line tool

1. Close the drawing and open a new acadiso.dwt window.

2. *Left-click* on the **Line** tool icon and *enter* figures as follows at each prompt of the command line sequence:

Command:_line Specify first point: 70,230

Specify next point: @220,0

Specify next point: @0, –70

Specify next point or [Undo]: @115 <225

Specify next point or [Undo]: @ –60,0

Specify next point or [Close/Undo]: @115 <135

Specify next point or [Close/Undo]: @0,70

Specify next point or [Close/Undo]: c (Close)

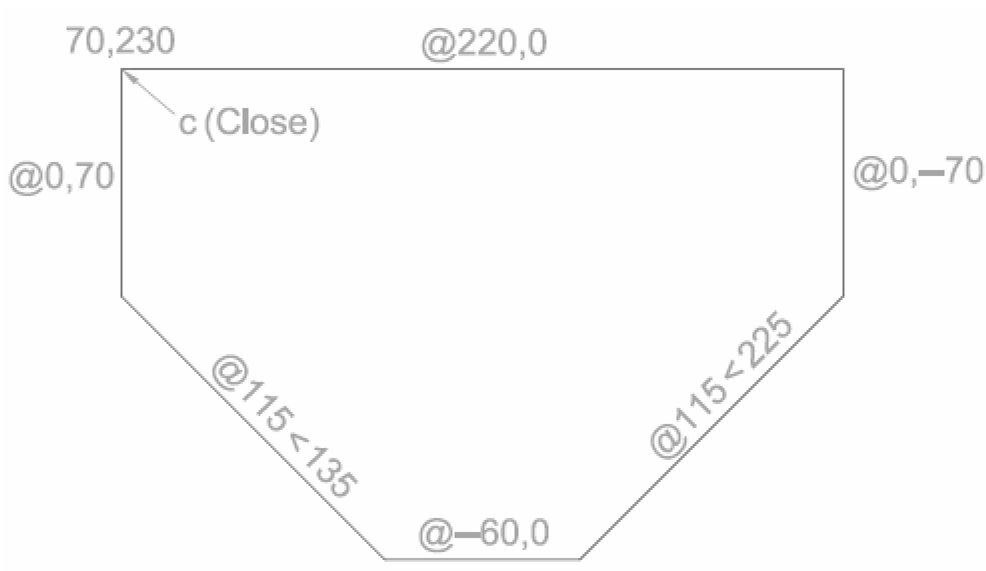


Fig. 3.5

Example (5):– Line tool

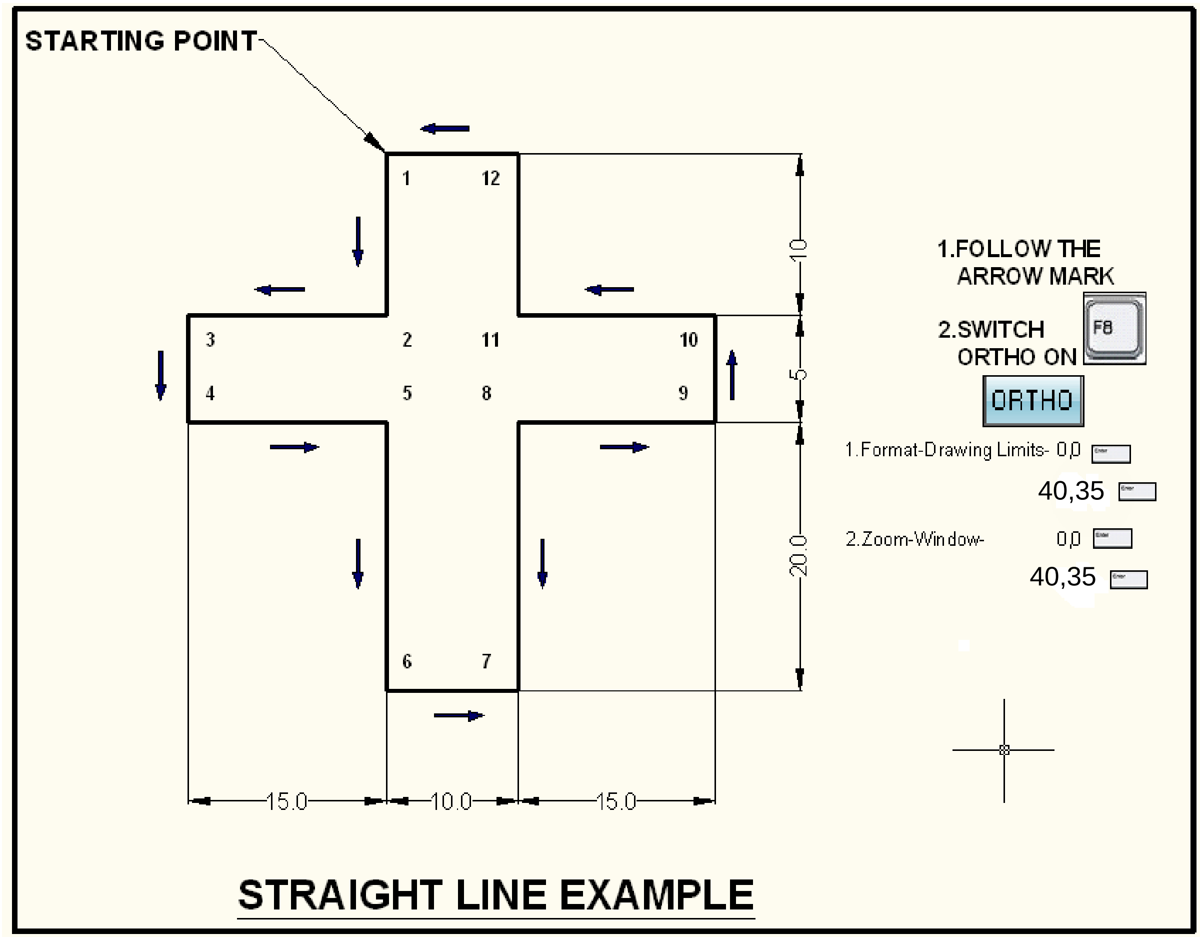


Fig. 3.6

Command:units

Command:limits
Reset Model space limits:
Specify lower left corner or [ON/OFF] <0.0000,0.0000>:

Specify upper right corner <12.0000,9.0000>: 40,35

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

Specify first corner: 0,0
Specify opposite corner: 40,35

Command: line

Specify first point: 15,35

Specify next point or [Undo]: 15,25

Specify next point or [Undo]: 0,25

Specify next point or [Close/Undo]: 0,20

Specify next point or [Close/Undo]: 15,20

Specify next point or [Close/Undo]: 15,0

Specify next point or [Close/Undo]: 25,0

Specify next point or [Close/Undo]: 25,20

Specify next point or [Close/Undo]: 40,20

Specify next point or [Close/Undo]: 40,25

Specify next point or [Close/Undo]: 25,25

Specify next point or [Close/Undo]: 25,35

Specify next point or [Close/Undo]: c

Example (6):– Line tool

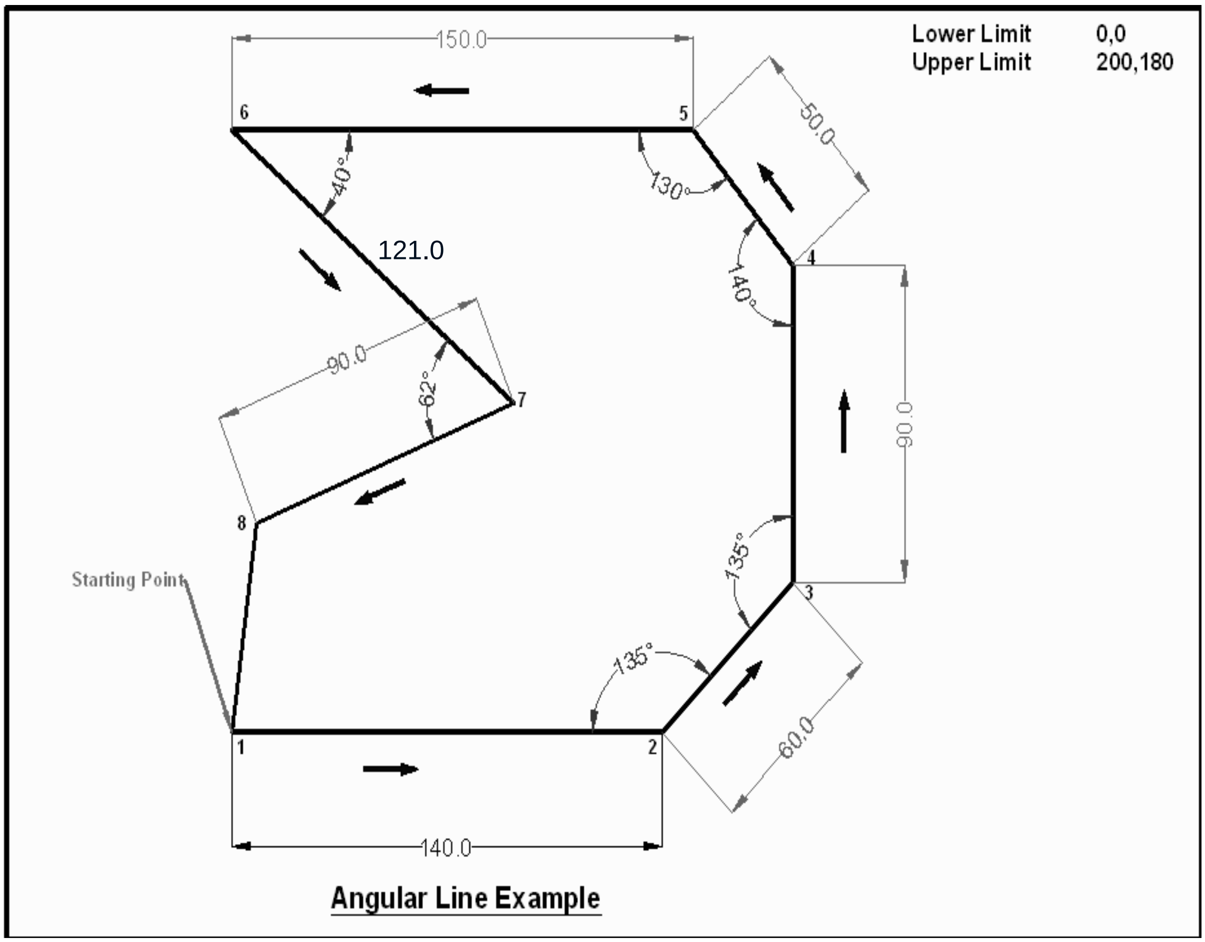


Fig. 3.7

Before starting the drawing solve for the unknown angles .

Angular Measurement

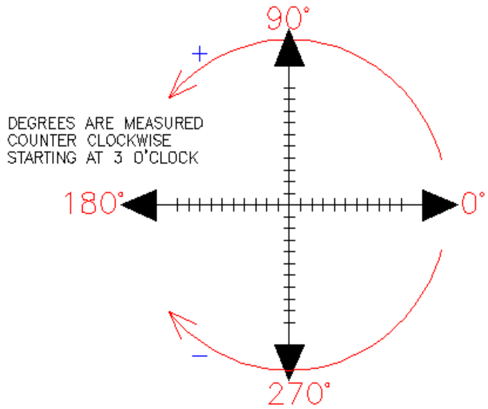


Fig. 3.8 Angular Measur ement

AutoCAD measures angles in a particular way also. Look at the diagram below and then place your mouse on it to see how this is done.

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

You might not always have an obvious reference point for 0 degrees. Look at the example below and place your mouse on the image to find out angle in question.

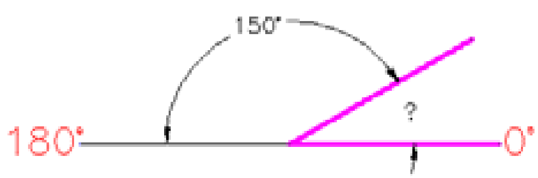


Fig. 3.9 Finding Angle

In this example, you are given information about the lines, but not the angle AutoCAD needs to draw the line from the start point. What you are given though, is (a) the knowledge that 0° is at the 3 o'clock position (b) the knowledge that 180° is at the 9 o'clock position and (c) the angle between 180° and the line you want to draw is 150°. With this information, you can figure out what angle you need. Here is a fool-proof way of getting the angle you need:

1.) Start at the 0° position and measure counter-clockwise (+) to 180°.

2.) From 180°, measure clockwise 150°.

3.) Consider that you just went +180-150 and use that as an equation: +180-150=30

If you solved the angles check your angles with drawing below

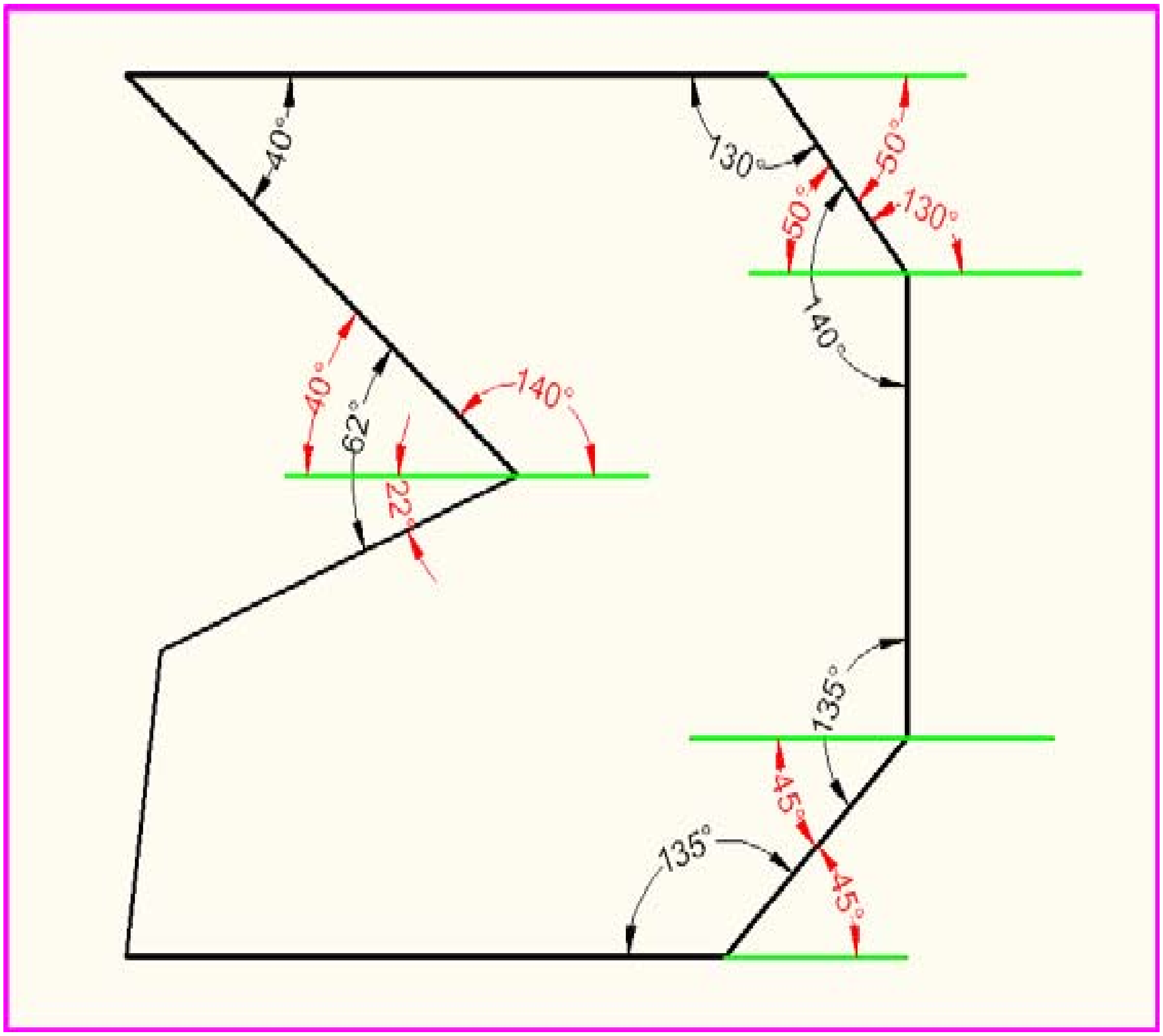


Fig. 3.10 After Solving Angles

Command: units

Command: limits

Reset Model space limits:
Specify lower left corner or [ON/OFF] <0.0000,0.0000>: 0,0

Specify upper right corner <12.0000,9.0000>: 200,180

Command: zoom
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: line

Specify first point: 0,0

Specify next point or [Undo]: @140<0

Specify next point or [Undo]: @60<45

Specify next point or [Close/Undo]: @90<90

Specify next point or [Close/Undo]: @50<130

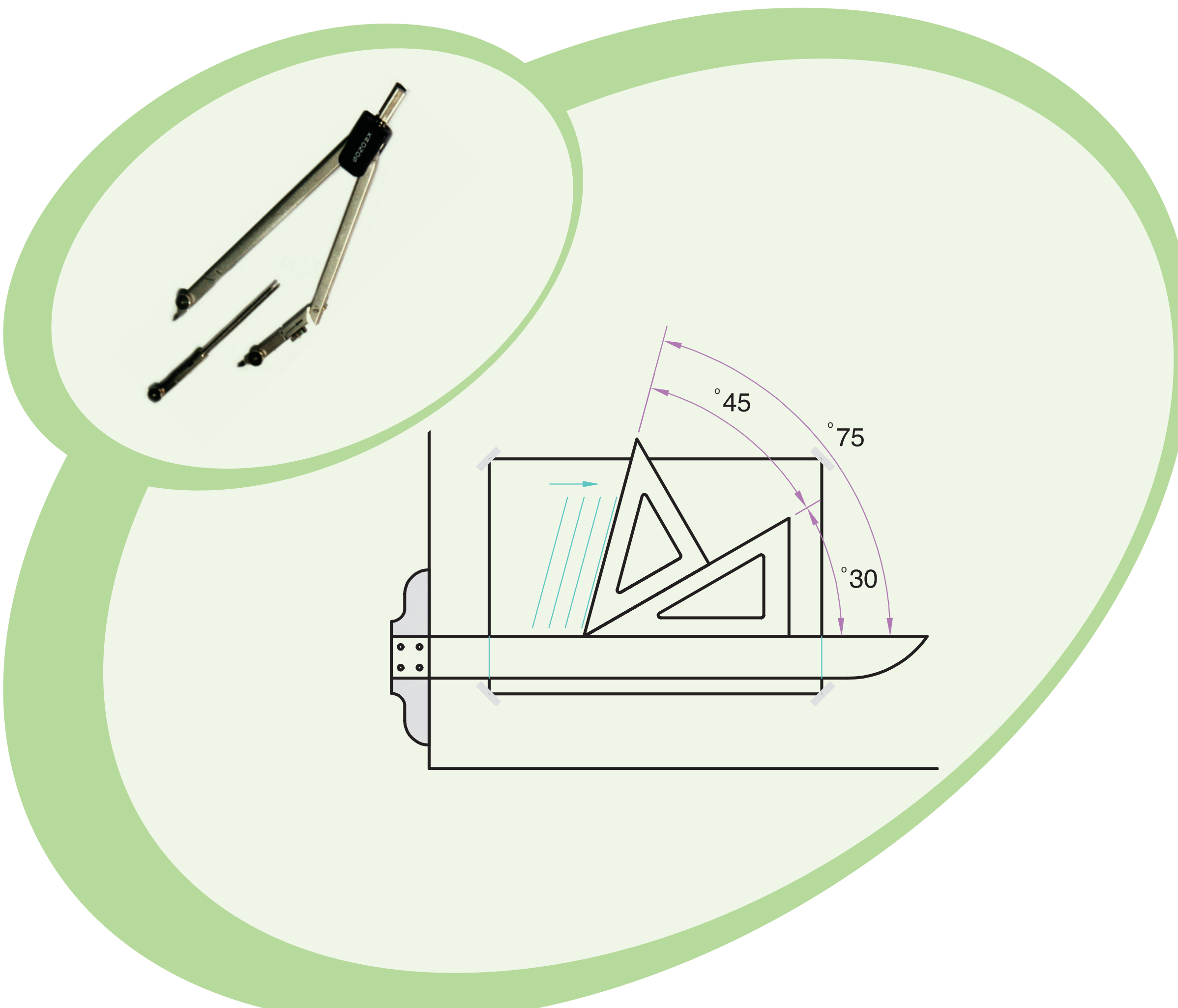
Specify next point or [Close/Undo]: @150<180

Specify next point or [Close/Undo]: @121<320

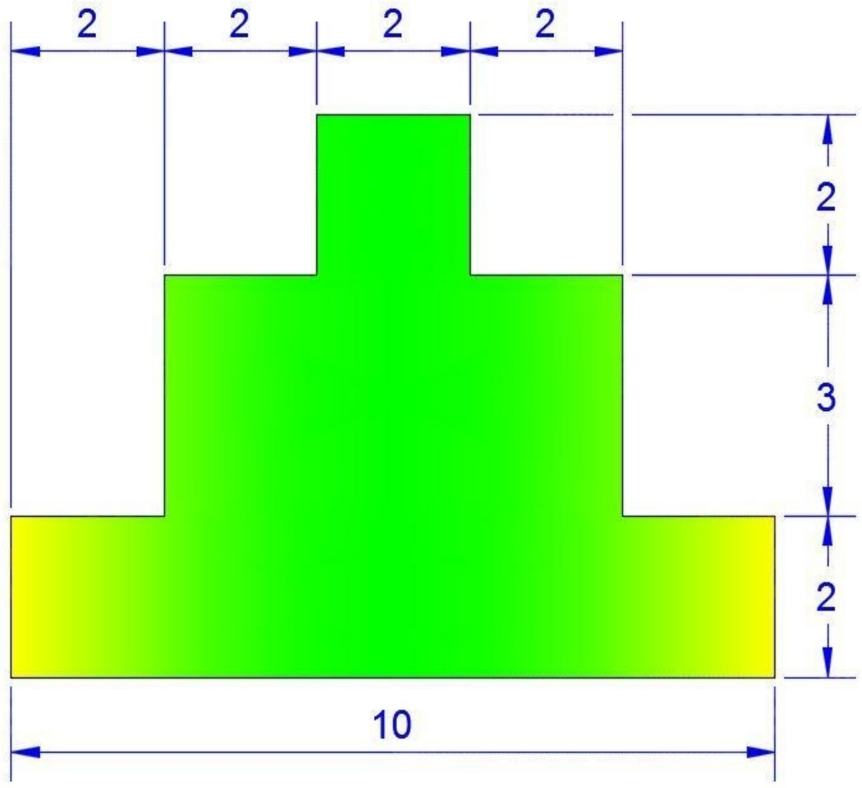
Specify next point or [Close/Undo]: @90<202

Specify next point or [Close/Undo]: c

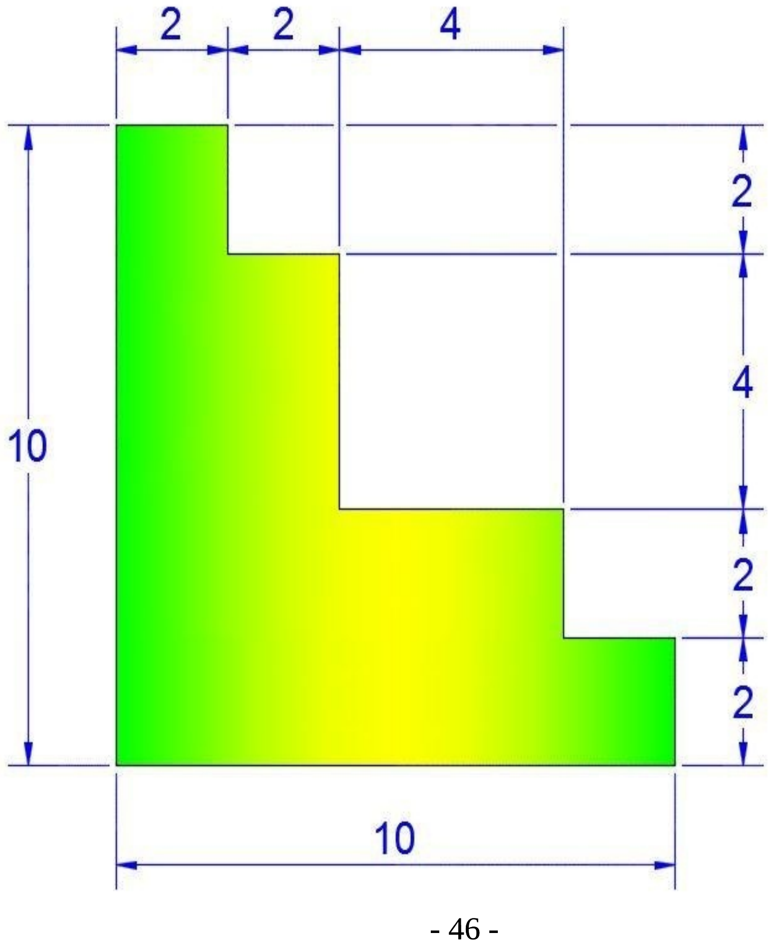
Exercises



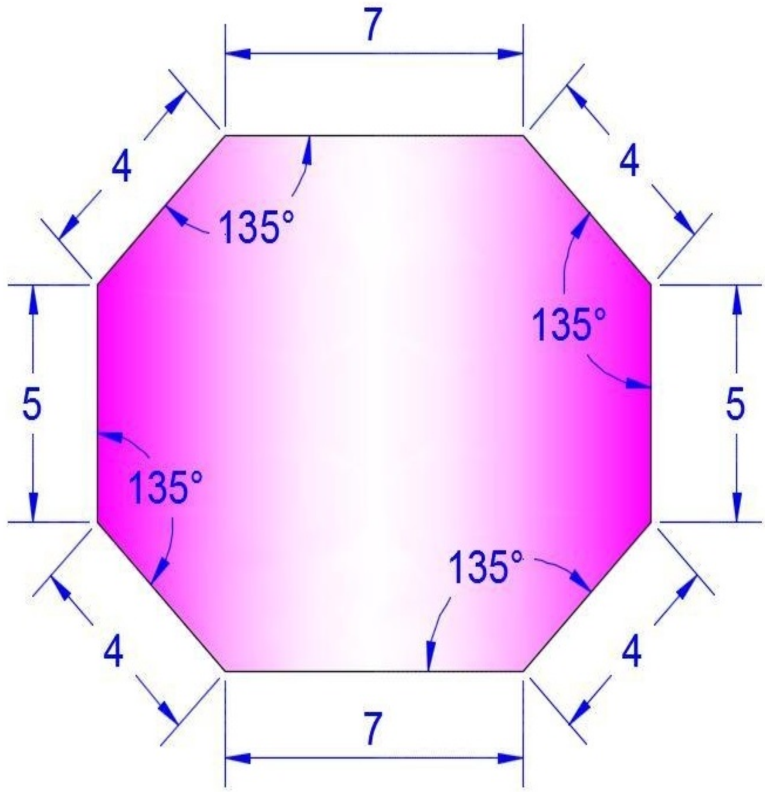
Exercise (1)



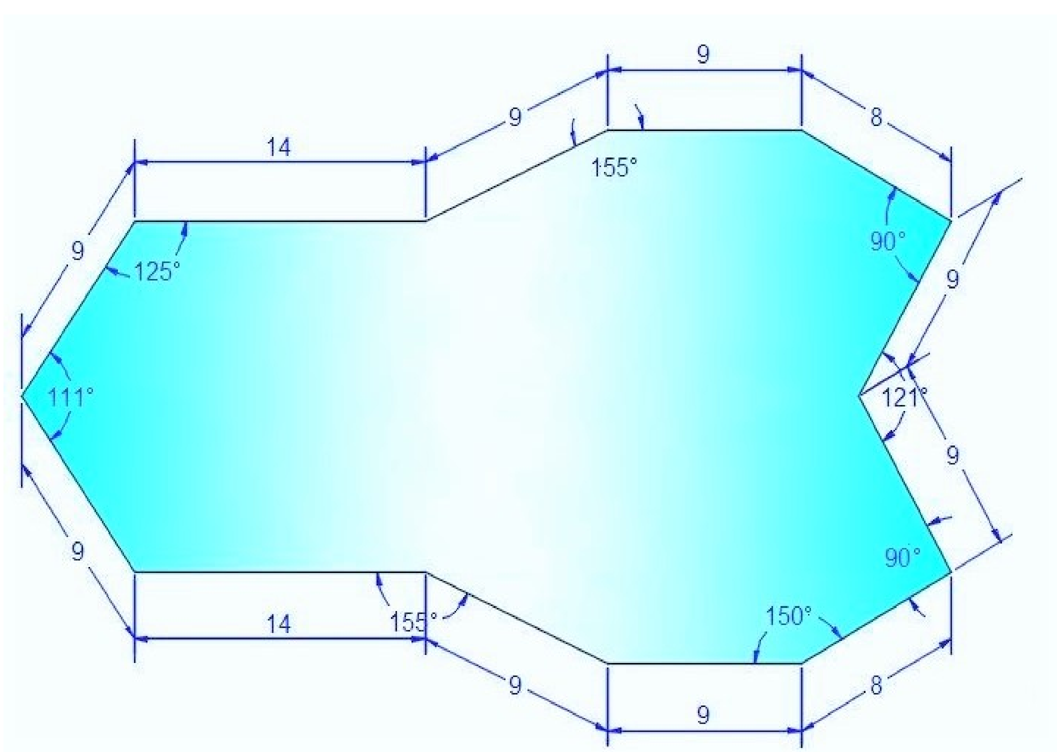
Exercise (2)



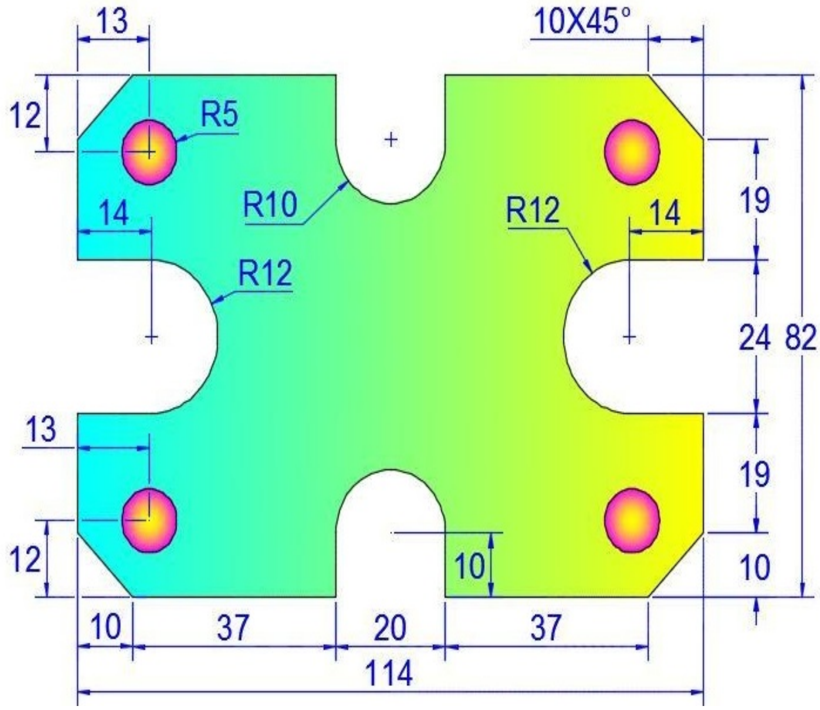
Exercise (3)



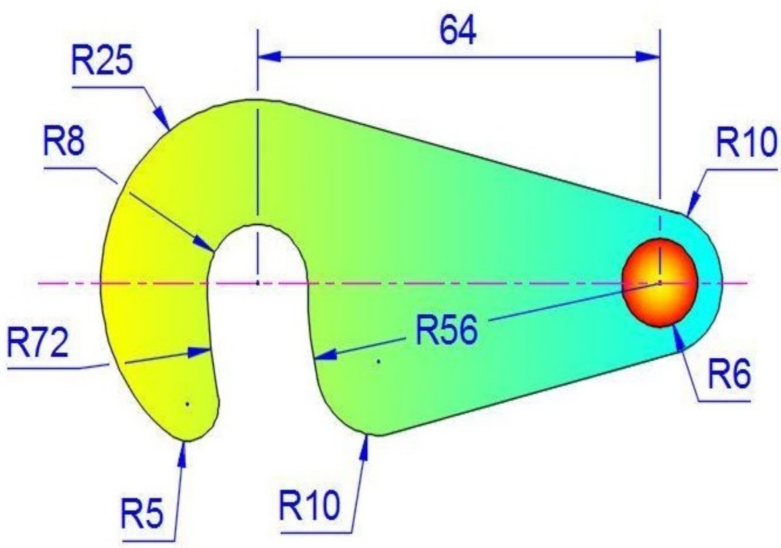
Exercise (4)



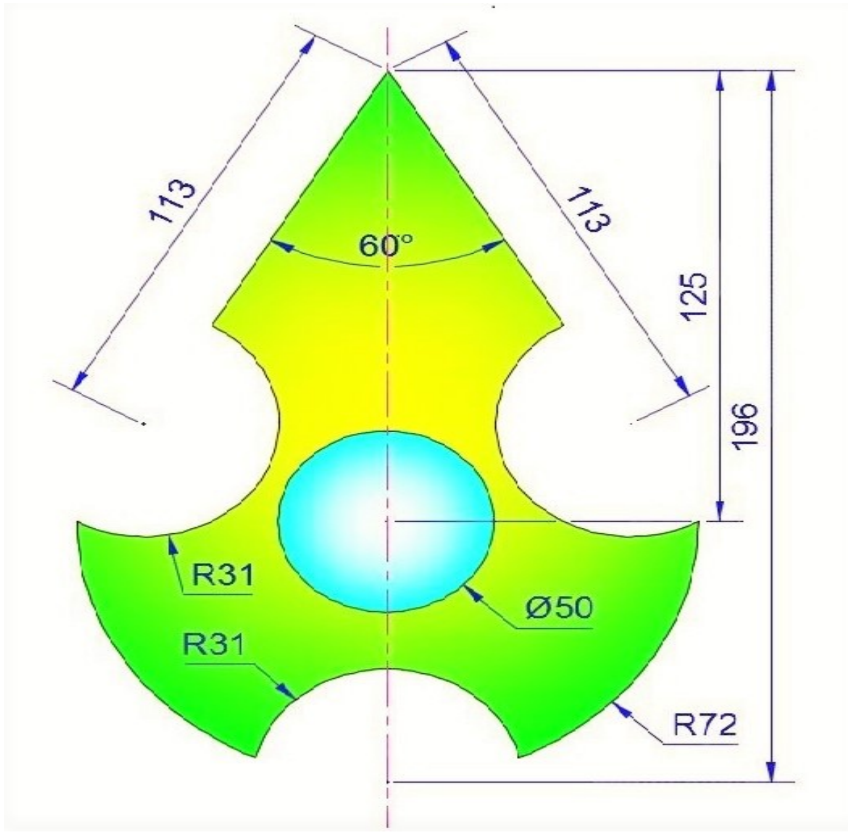
Exercise (5)



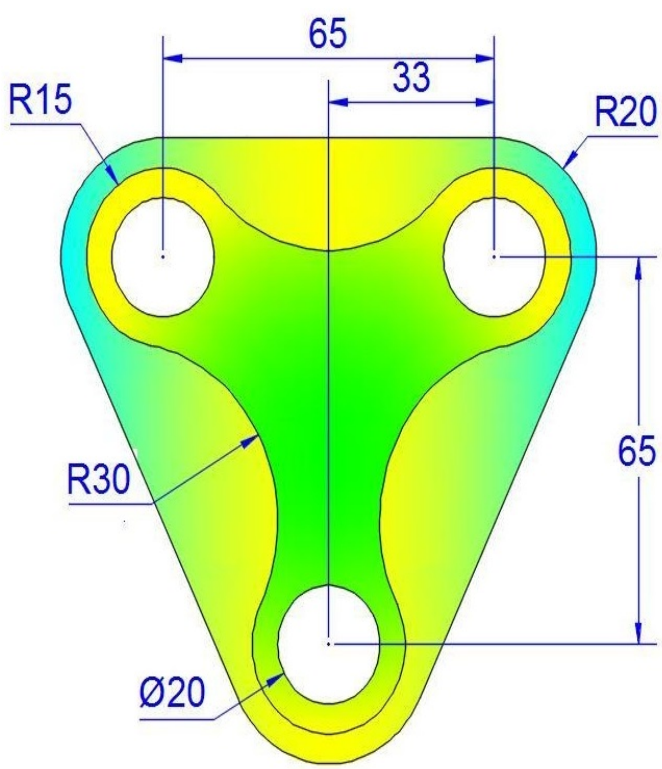
Exercise (6)



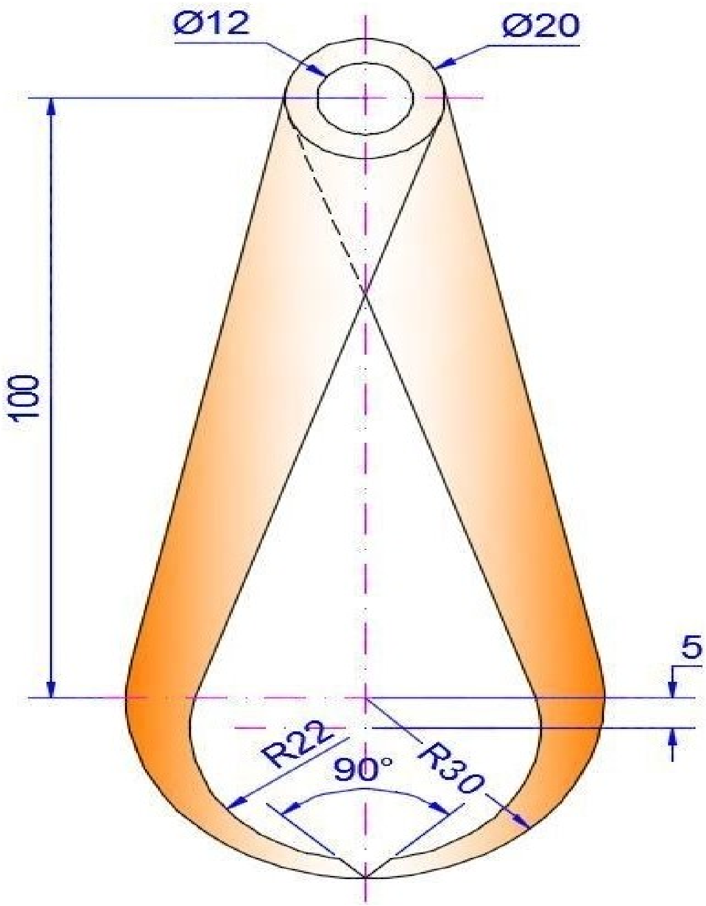
Exercise (8)



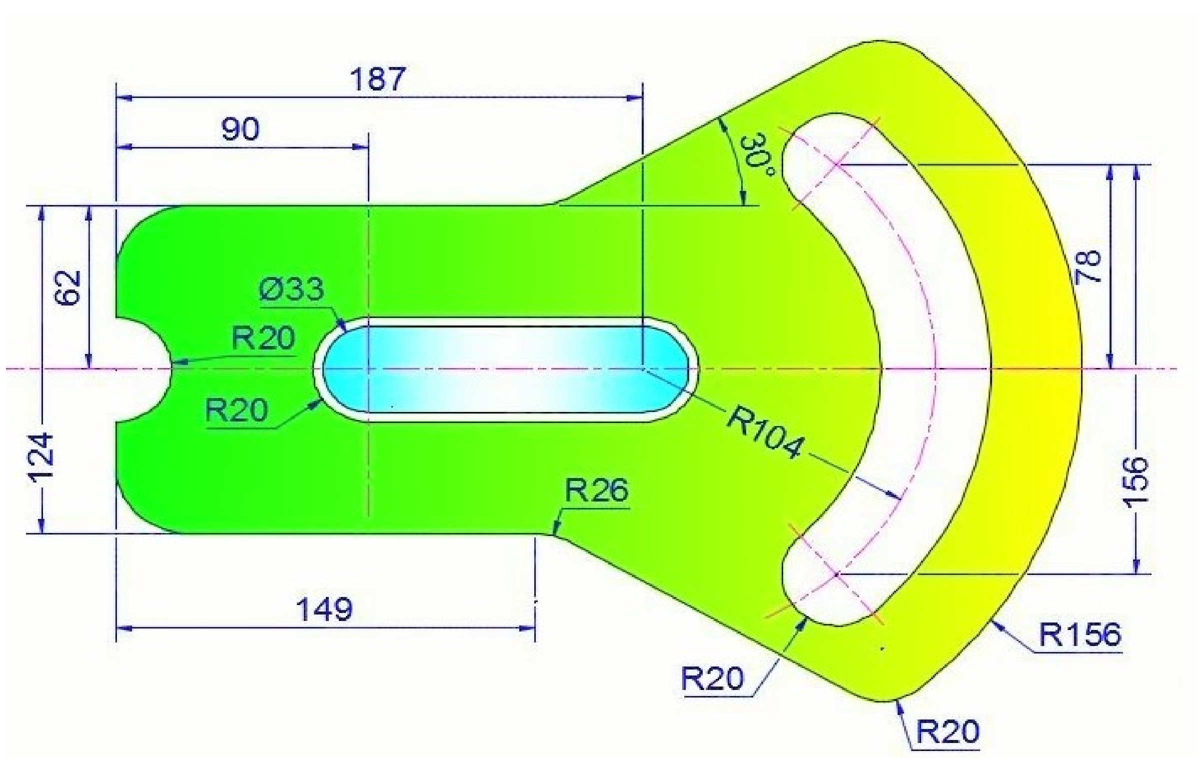
Exercise (9)



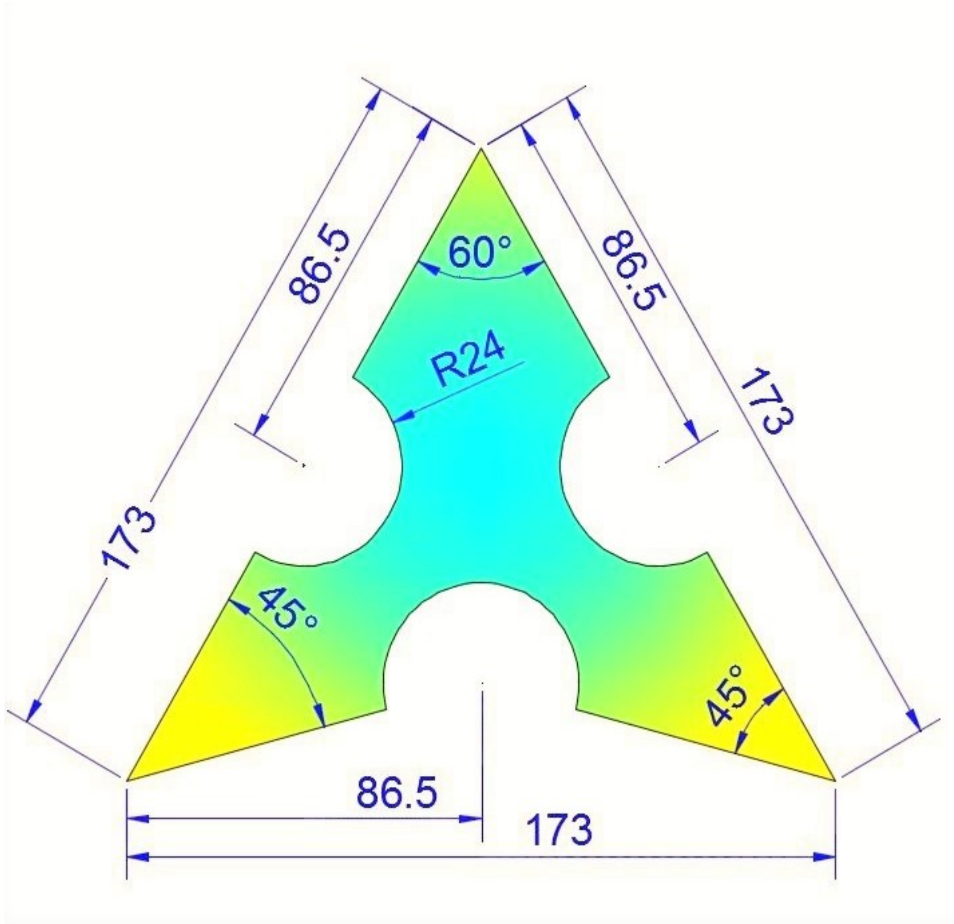
Exercise (10)



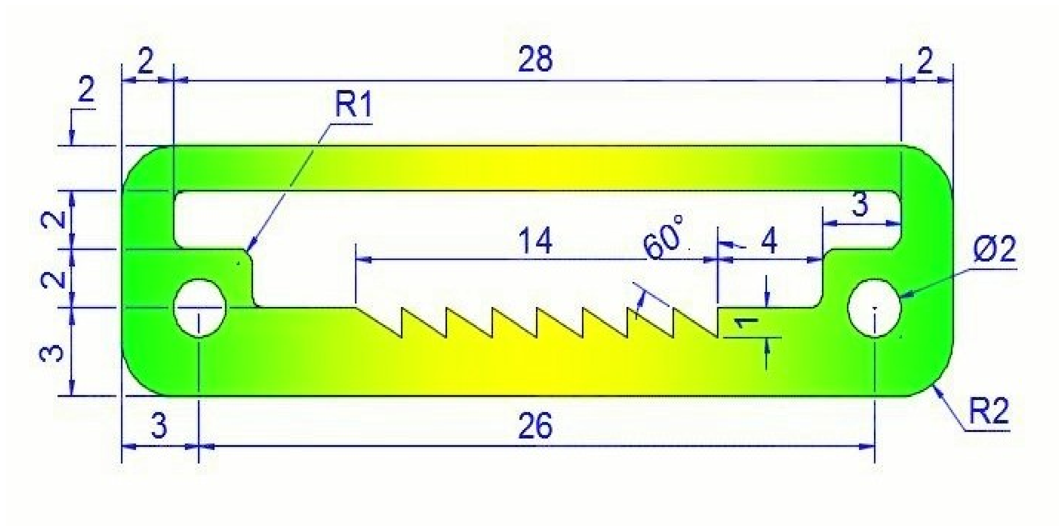
Exercise (11)



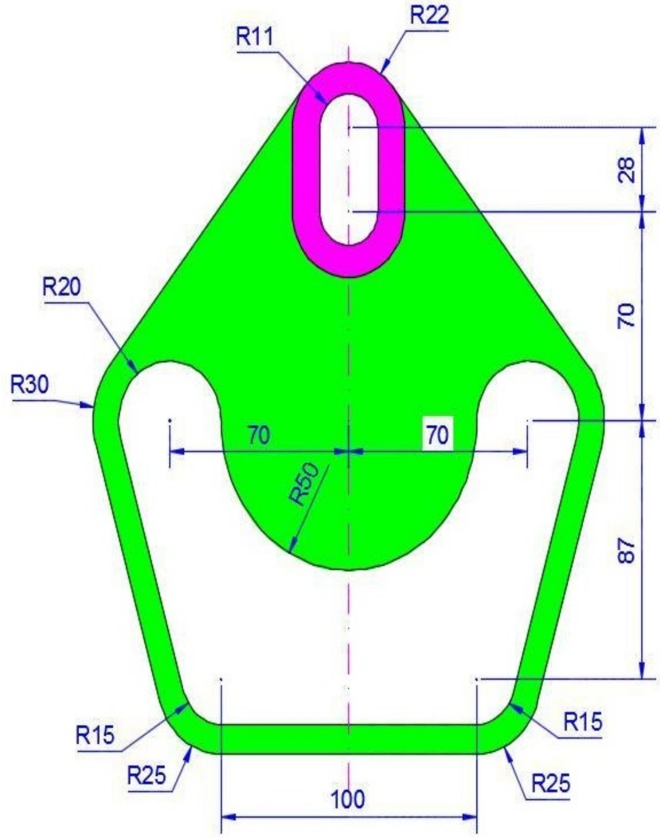
Exercise (12)



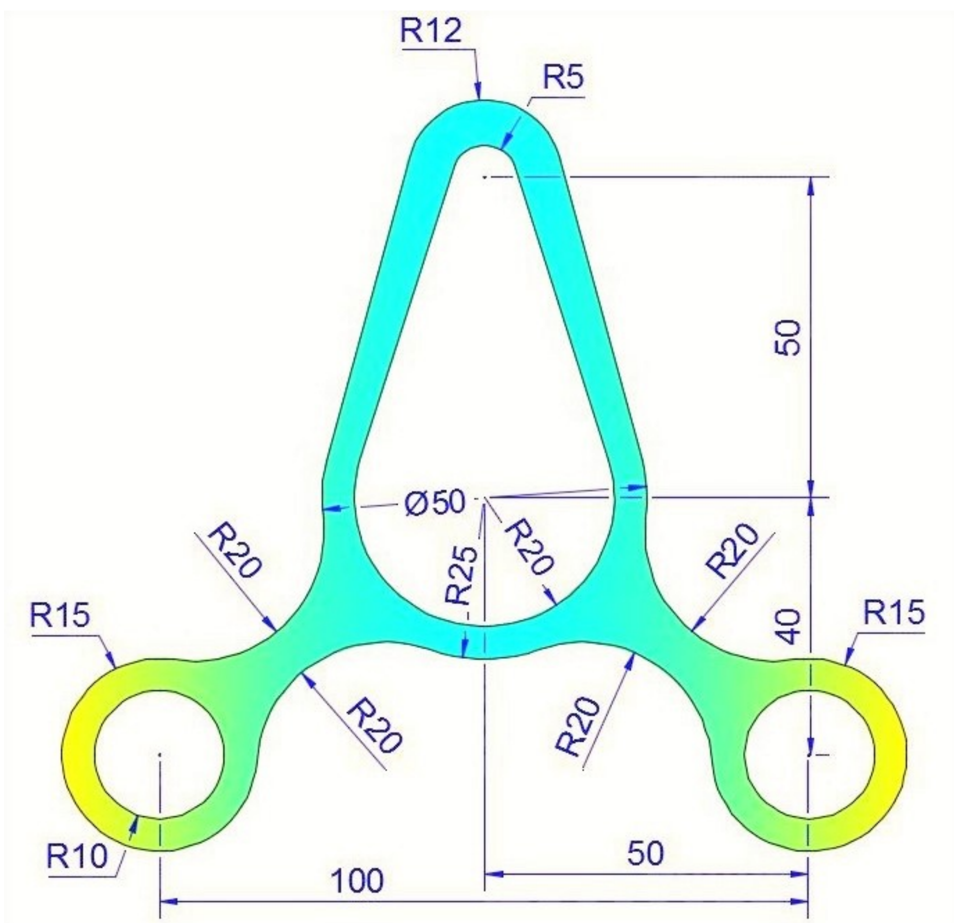
Exercise (13)



Exercise (14)



Exercise (15)



Chapter 4
More Draw Commands

4.1 Polygon

1. **Choose** Draw, Polygon.

   or

2. **Click** the Polygon icon. 

   or

3. **Type** Polygon at the command prompt. Command: **POLYGON**

4. **Type** The number of sides for the polygon (3-1024)
   Number of sides <default>: **number**

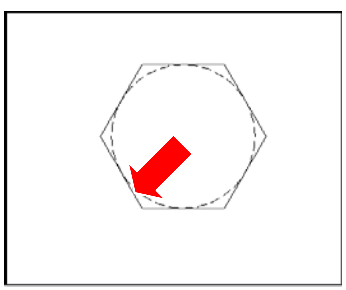
5. **Pick** The center of the polygon. Edge/<Center of polygon>:**pick**

   or

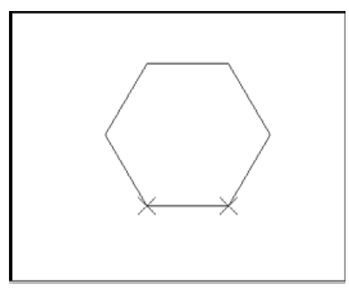
6. **Type** **E** to define the polygon by two edges.

7. **Type** **I** or **C** to place the polygon inside or outside of an imaginary circle. Inscribed in circle/Circumscribed about circle (I/C):

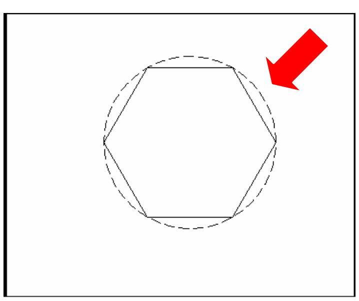
Polygon Inscribed in an imaginary circle



Polygon drawn with an edge



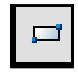
Polygon circumscribed around an imaginary circle



4.2 Rectangle

1. **Choose** Draw, Rectangle.

   or

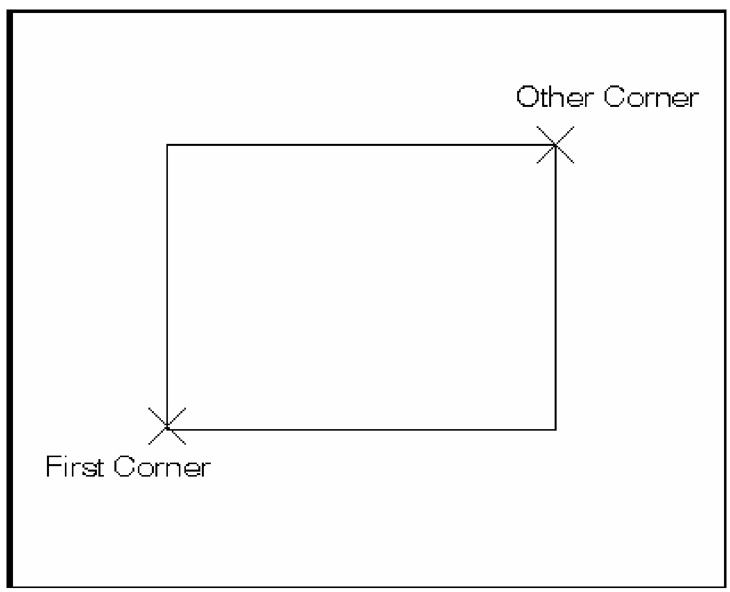
2. **Click** the Rectangle icon. 

   or

3. **Type** Rectang at the command prompt
   Command: **RECTANG** Chamfer/Elevation/Fillet/Thickness/Width/<First corner>:

4. **Pick** first corner.

5. **Pick** other corner or type coordinates **(i.e. @4,2)**.



4.3 Donut

Donuts are filled rings or solid-filled circles that actually are closed polylines with width.

1. **Choose** Draw, Donut.

or

2. **Type** Donut at the command prompt.
   Command: **DONUT**

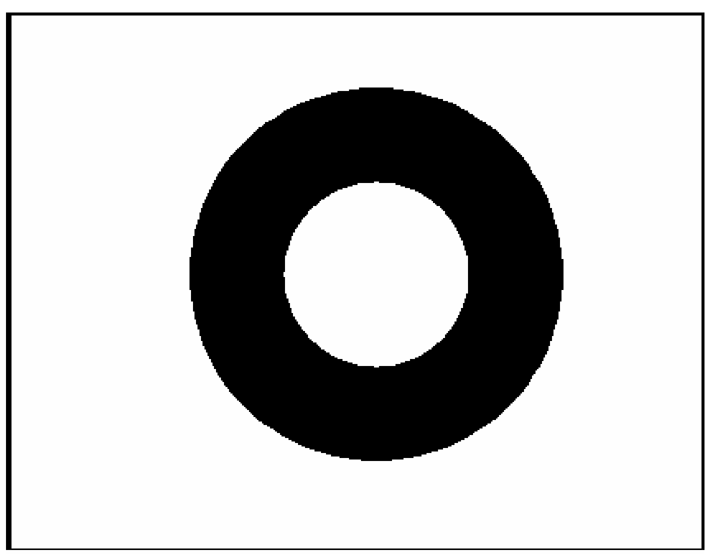
or

3. **Choose** the donut icon. 

4. **Type** A value for the inside diameter. Inside diameter <last>: **.5**

5. **Type** A value for the outside diameter. Outside diameter <last>: **1**

6. **Pick** A point for the center of the donut. Center of doughnut: (**point**)



4.4 Ellipse

Creates an ellipse or an elliptical arc.

1. **Choose** Draw, Ellipse.

   or

2. **Choose** the Ellipse icon. 

   or

3. **Type** ELLIPSE at the command prompt
   Command: **ELLIPSE**

4. **Type** One of the following options: Arc/Center/Isocircle
   /<Axis endpoint 1>:

Ellipse options:

Axis endpoint 1 Defines the first axis by two specified 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.

Axis endpoint 2: <Other axis distance> / Rotation: Specify a point or enter a distance

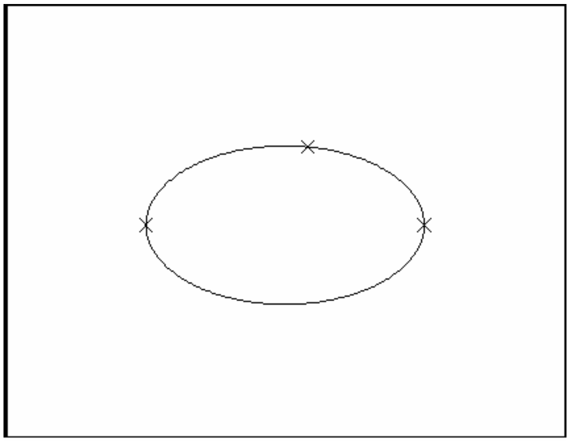
Arc Creates an elliptical arc. The angle of the first axis determines the angle of the elliptical arc. The first axis can define either the major or the minor axis of the elliptical arc.

Center Creates the ellipse by a specified center point.

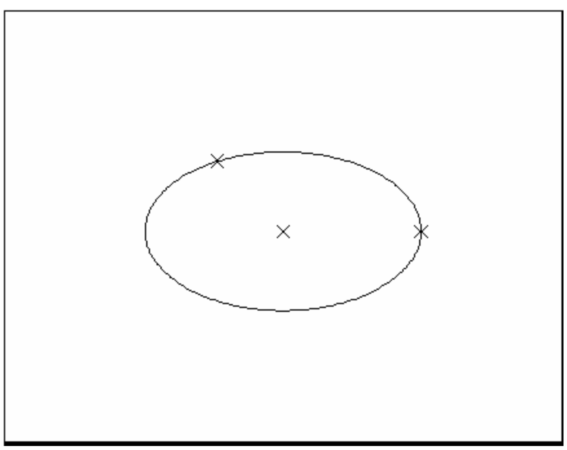
Isocircle Creates an isometric circle in the current isometric drawing plane.

Rotation The major axis is now treated as the diameter of a circle that will be rotated a specified amount around the axis. You enter an angle between 0 and 89.4 degrees.

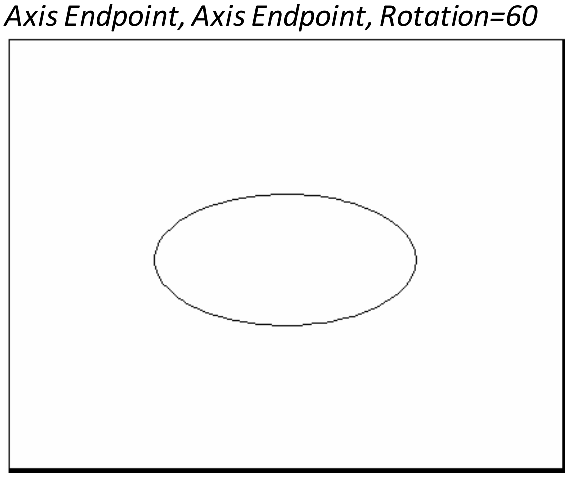
ELLIPSE,
Axis , Eccentricity (Axis Endpoint, Axis Endpoint, Other Axis Distance)



ELLIPSE,
Center, Axis, Axis



ELLIPSE,
Axis Endpoint, Axis Endpoint, Rotation=60

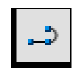


4.5 Pline Command

A polyline is a connected sequence of line segments created as a single object. You can create straight line segments, arc segments, or a combination of the two.

1. **Choose** Draw, Polyline.

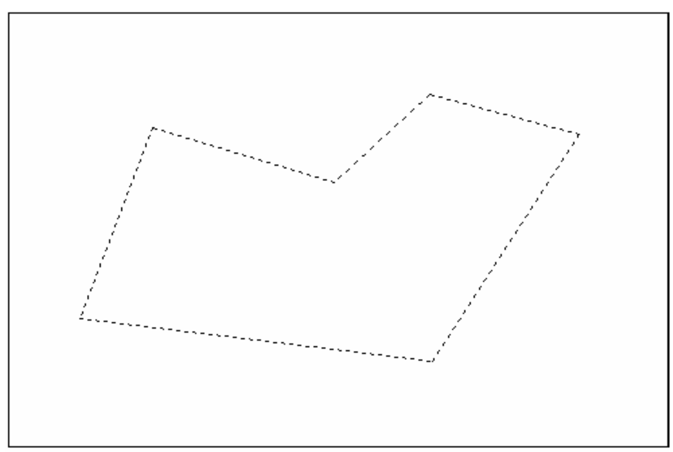
   or

2. **Pick** the Pline icon. 
3. **Type** PLINE at the command prompt
   Command : **PLINE** or **PL**
4. **Pick** A point on the drawing to start the polyline
   From point: (**select**)
5. **Type** One of the following options
   Arc/Close/Halfwidth/Length/Undo/Width

   or

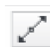
6. **Pick** A point to continue drawing
   Arc/Close/Halfwidth/Length/Undo/Width/
   <endpoint of line>: (**pick point**)

Polyline as one segment



4.6 Construction Line

Creates an infinite line.

1. **Choose** Draw, Construction Line
   or
2. **Choose** the XLINE icon. 

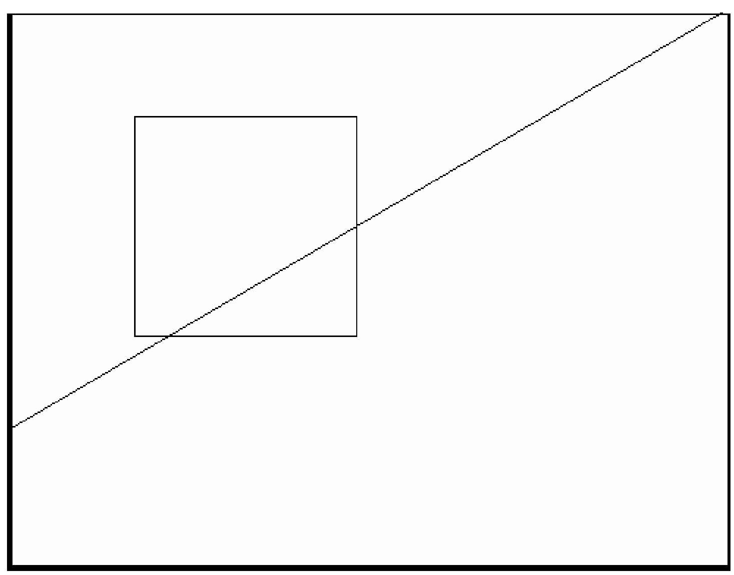
   or

3. **Type** XLINE at the command prompt.
   Command: **XLINE**
   Specify a point or [Hor/Ver/Ang/Bisect/Offset]:

XLINE Options

HOR Creates a horizontal xline passing through a specified point.
VER Creates a vertical xline passing through a specified point
ANG Creates an xline at a specified angle.
BISECT Creates an xline that passes through the selected angle vertex and bisects the angle between the first and second line

OFFSET Creates an xline parallel to another object.



4.7 Ray Command

Creates an infinite line in one direction.

1. **Choose** Draw, Ray

   or

2. **Choose** the Ray icon. 

   or

3. **Type** RAY at the command prompt.

   Command: **RAY**

   Specify a point : **(pick through point)**

4.8 Point Command

1. **Choose** Draw, Point, Single or Multiple Point.

   or

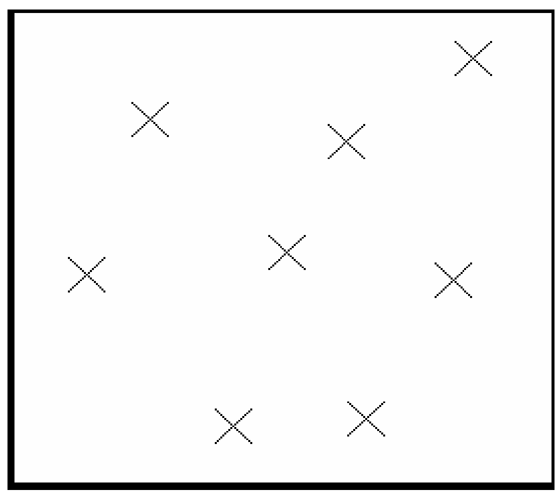
2. **Click** the Point icon.

   or

3. **Type** POINT at the command prompt.

   Command: **POINT**

4. **Pick** A point on the drawing. Point (**point**)



4.8.1 Point Styles

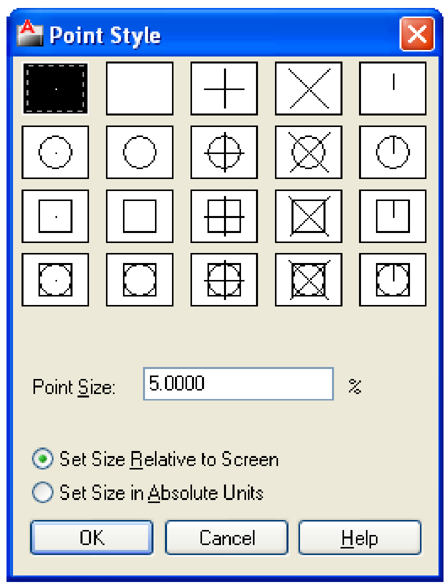
Changes the appearance of points and point sizes.

1. **Choose** Format, Point Style...

or

Type DDPTYPE at the command prompt.

Command : **DDPTYPE**



4.8.2 Divide

1. **Choose** Draw, Point, Divide.

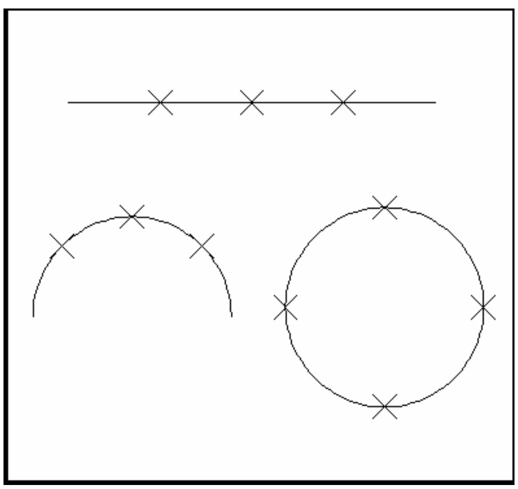
or

2. **Type** DIVIDE at the command prompt
   Command: **DIVIDE**

3. **Pick** Object to divide
   Select object to divide: (**pick one object**)

4. **Type** The number of equal segments to divide the object into<Number of segments>/Block: (**number**)

Objects divided using points



TIP:

- Block symbols can be used in place of a point. The Block must currently be defined within the drawing. If you answer yes to the Align block? prompt, the Block will be rotated round its insertion point so that it is drawn tan- gent to the object being divided.

4.8.3 Measure

1. **Choose** Draw, Point, Measure.

   or

2. **Type** MEASURE at the command prompt
   Command: **MEASURE**

3. **Pick** Object to measure: Select object to measure:
   (**pick one object**)

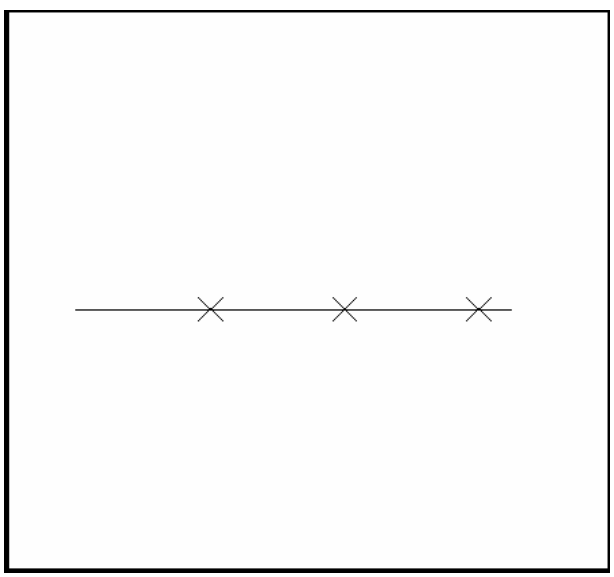
4. **Type** The length of each segment along the object.
   <Length of segment>/Block: (**number**)

   or

5. **Type** B to specify a block instead of a point to insert.

Points placed along measured distance

(remaining length is on the right side of the line)



Chapter 5

Drawing Aids

5.1 SNAP Command

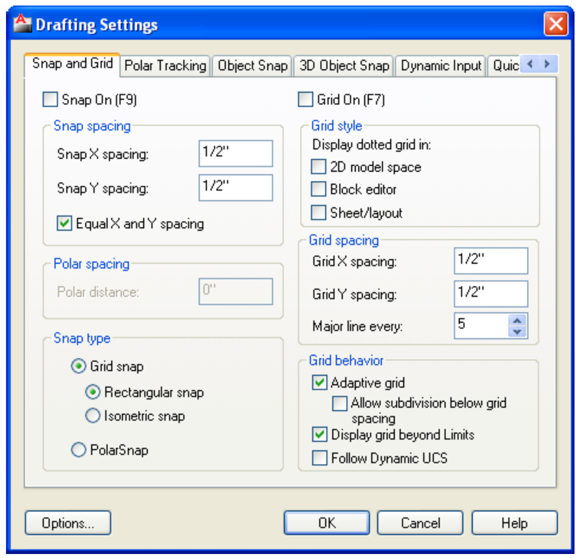
1. **Choose** Tools, Drafting Settings...

   or

2. **Type** SNAP at the command prompt.

   Command: **SNAP or SN**

3. **Type** One of the following options: Snap spacing or [ON/OFF/Aspect/Style/Type]:



Turn Snap On/OFF

3. **Press** Function Key **F9** to turn the snap ON/OFF.

   or

4. **Double Click** SNAP on the Status Bar.

   or

5. **Press** CTRL + B.

TIP:

- Click with the right mouse button on the SNAP option from the status bar as a shortcut to changing the snap settings

5.2 Grid Command

1. **Choose** Tools, Drafting Settings...

   or

2. **Type** DSETTINGS at the command prompt.

   Command : DSETTINGS (DS)

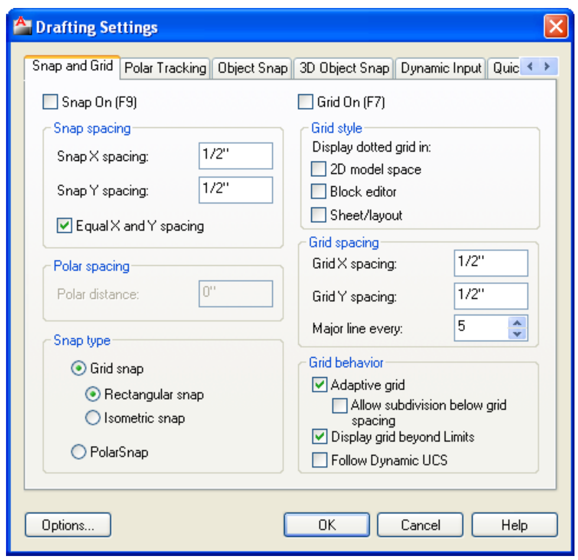
   or

3. **Type** GRID at the command prompt.

   Command: **GRID**

4. **Type** One of the following options:

   Grid spacing(X) or ON/OFF/Snap/Aspect <0000>:



Turn Grid On/Off

1. **Press** Function Key **F7** to turn the grid ON/OFF.

   or

2. **Double Click** GRID on the Status Bar.

   or

3. **Press** CTRL + G.

5.3 Running Object Snaps

An object snap mode specifies a snap point at an exact location on an object. OSNAP specifies running object snap modes, which remain active until you turn them off.

1. **Type** DDOSNAP at the command prompt

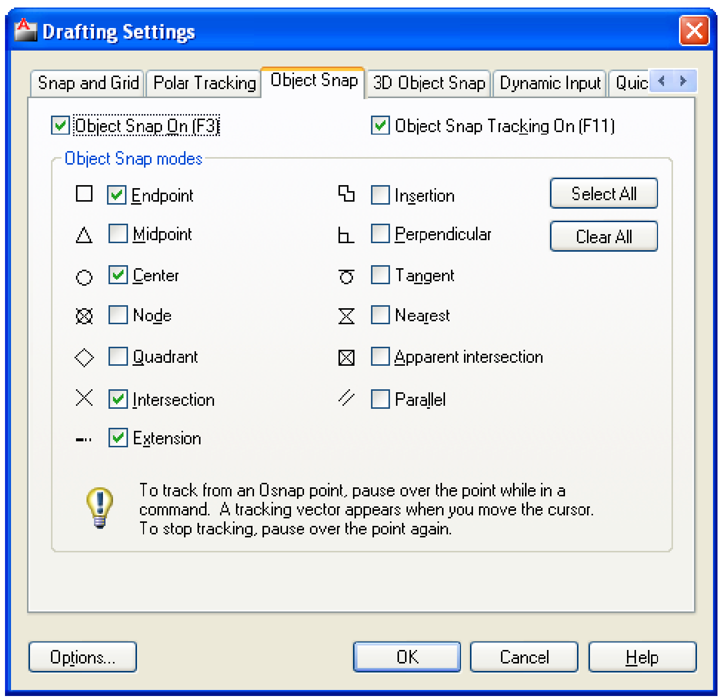
   Command: **DDOSNAP**

   or

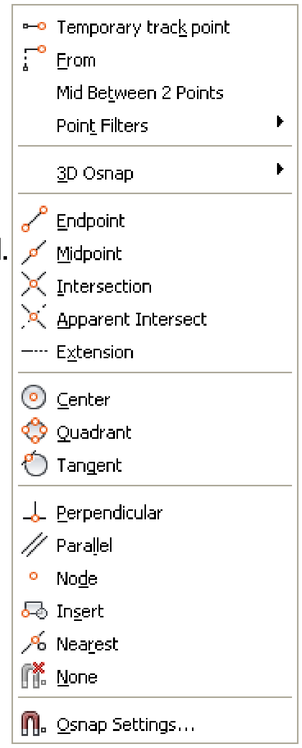
   Click OSNAP on the Status Bar.

   Right Click the Object Snap TAB.

2. **Choose** an object snap to turn ON/OFF from the dialog box.



5.4 Case by Case (Temporary Mode)



1. **Press** SHIFT + the RIGHT MOUSE BUTTON.

or

Click one of the object snaps located Object Snap toolbar icon.

or

Type The object snap at the prompt line.

Command: Line

From pt: **ENDP**

To pt: **MID**

To pt: **CEN**

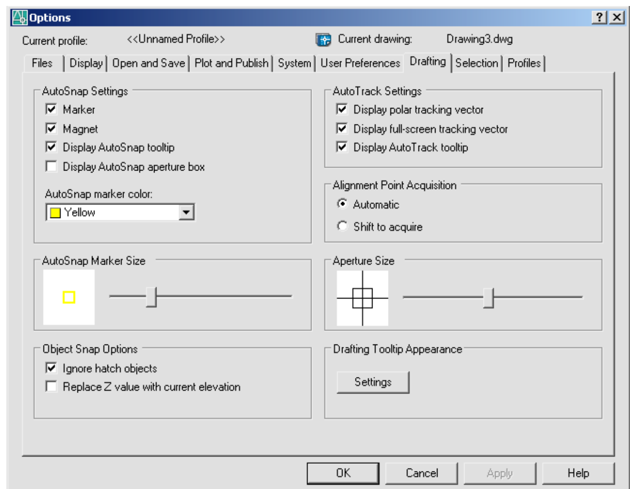
TIP:

- Case by Case objects snaps will override running mode object snaps.

5.5 Osnap Settings

When you use any of the object snap settings, AutoSnap displays a marker and a Snap tip when you move the cursor over a snap point.

1. **Type** Options at the command prompt.
   Command: **OPTIONS**
2. **Select** the Drafting tab.
3. **Change** settings and choose OK.



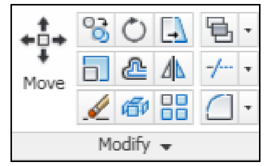
The following are object snap modes. In bold caps are shortcut abbreviations to type.

CENter	Center of Arc or Circle
ENDpoint	Closest endpoint of Line/Arc
INSertion	Insertion point of Text/Block/Shape/ Attribute
INTersection	Intersection of Lines/Arcs/Circles
MIDpoint	Midpoint of a line/Arc or midpoint
NEAerst	Nearest point on a Line/Arc/Circle/Point
APParent Int	Finds where two entities would intersect
NODe	Nearest point entity (or Dimension definition point)
NONe	None (off)
PERpendicular	Perpendicular to a Line/Arc/Circle
QUAdrant	Quadrant point on an Arc/Circle
QUIck	Quick mode (first find, not closest)
TANgent	Tangent to Arc or Circle

Chapter 6

Introduction to Edit Commands

6.1 Erase Command



Deletes objects from a drawing.

1. **Choose** Modify, Erase.

   or

   Click the Erase icon. 

   or

   Type ERASE at the command prompt.

   Command : **ERASE** or **E**

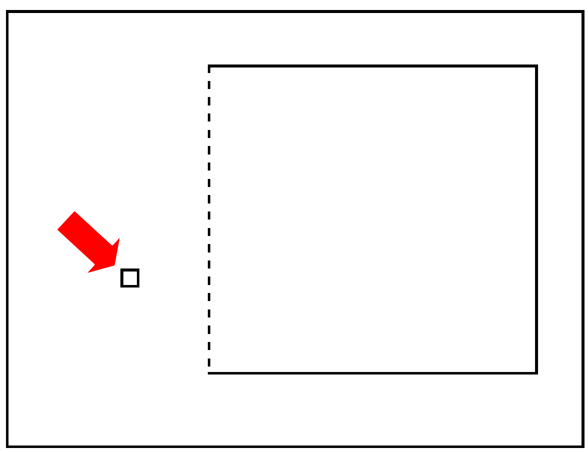
2. **Pick** Object at the select object prompt.

   Select objects: (**pick object**)

3. **Press** ENTER when you are done choosing objects.

   Select objects: **ENTER**

Select objects with pickbox



TIP:

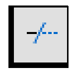
- If the cursor is not touching an object, AutoCAD will create a crossing or window selection as defined on the following pages.

6.2 TRIM

The TRIM command allows you to trim objects in a drawing so they end precisely at a cutting edge defined by one or more other objects in the drawing.

1. **Choose** Modify, Trim.

   or

2. **Click** the Trim icon. 

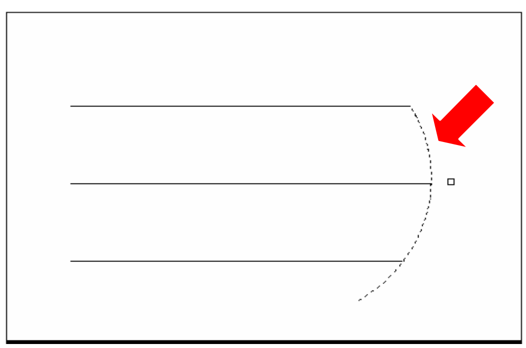
3. **Type** TRIM at the command prompt
   Command: **TRIM**
   Select cutting edge(s)...

4. **Pick** The CUTTING edge to extend to
   Select objects: (**select**)

5. **Press** ENTER to accept the cutting edge
   Select objects: (**press enter**)

6. **Pick** Objects to trim
   <Select object to trim> / Project / Edge / Undo:
   Select an object, enter an option, or press enter

7. **Press** ENTER when you are done choosing objects
   Select object to trim/Undo: (**press enter**)



Lines Trimmed to an Arc (Arc is cutting edge)

TIP:

- Hold the SHIFT key to interactively extend instead of trim.

Edgemode

Controls how the TRIM and EXTEND commands determine cutting and boundary edges.

6.3 Offset Command

Offset Distance

To offset a specified distance:

1. **Choose** Modify, Offset.

or

2. **Choose** the Offset icon. 

or

3. **Type** OFFSET at the command prompt. Command: **OFFSET or O**

4. **Type** The distance to offset.

   Offset distance or <Through point>: (**number**)

5. **Pick** The object to offset.
   Select object to offset: (**select object**)

6. **Pick** A side to offset object to. Side to offset: (**pick side**)

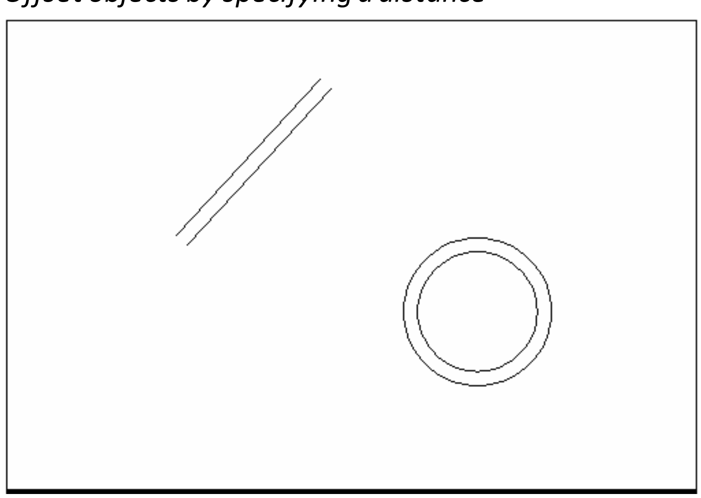
7. **Pick** Another object to offset

   Select object to offset: (**pick side**)

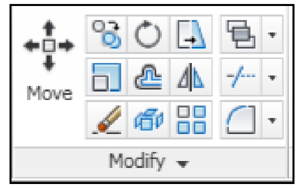
or

8. **Press** Enter to end the command.

Offset objects by specifying a distance

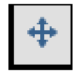


6.4 Move Command



1. **Choose** Modify, Move.

or

Click the Move icon. 

or

Type MOVE at the command prompt

Command: **MOVE or M**

2. **Pick** Objects to move
Select objects: (**select**)

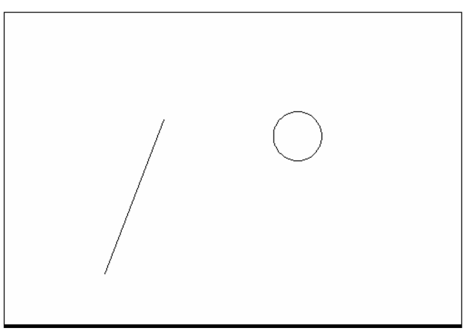
3. **Pick** A point to move from

Base point or displacement: (**pick point**)

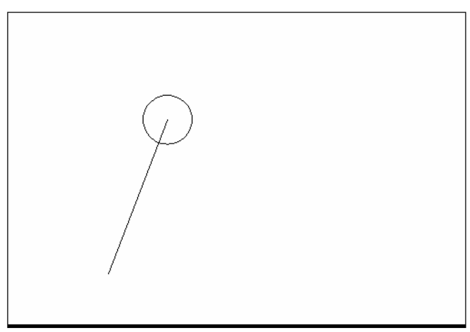
4. **Pick** A point to move to

Second point of displacement: (**pick point**)

Circle before move



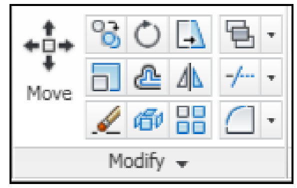
Circle after move



TIP:

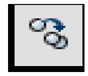
- To move an object a specified distance, type a distance at the second point of displacement prompt: **@1<0**

6.5 Copy Command



1. **Choose** Modify, Copy.

   or

   Click the Copy icon. 

   or

   Type COPY at the command prompt. Command: **COPY or CP**

2. **Pick** Objects to copy.

   Select objects: (**select**)

3. **Pick** A point to move from.

   Base point or displacement/Multiple: (**pick point**).

4. **Pick** A point to copy to.

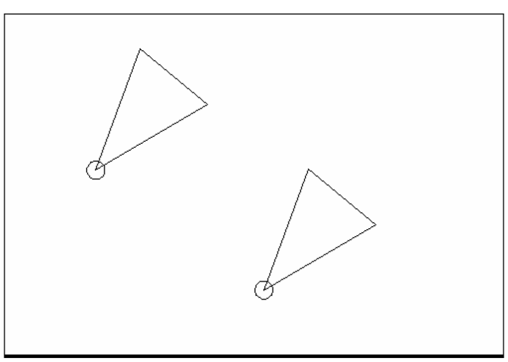
   Second point of displacement: (**pick point**)

   or

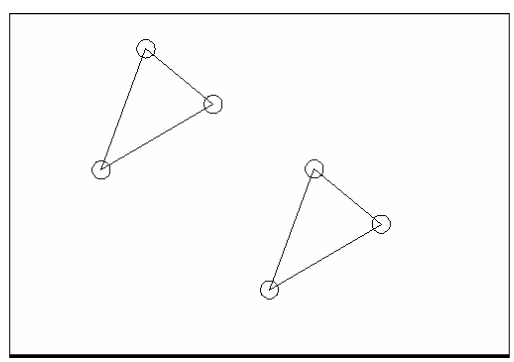
   Type A point to copy to.

   Second point of displacement: @ **1<0**

Duplicate objects copied



Multiple objects copied



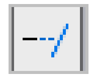
TIP:

- To copy many objects in the same copy command, type M for Multiple at the "Base point or displacement/Multiple" option

6.6 EXTEND

1. **Choose** Modify, Extend.

   or

2. **Click** the Extend icon. 

   or

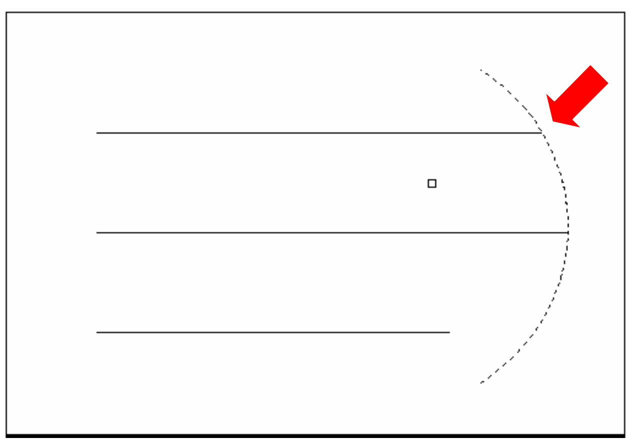
3. **Type** EXTEND at the command prompt
   Command: **EXTEND**
   Select boundary edge(s)...

4. **Pick** The BOUNDARY edge to extend to
   Select objects: (**select**)

5. **Press** ENTER to accept the boundary edge
   Select objects: (**press enter**)

6. **Pick** The objects to extend
   <Select object to extend> / Project / Edge / Undo: Select an object, enter an option, or press enter : (**select**)

7. **Press** ENTER when you are done choosing objects



Lines Extended to an Arc (Arc is boundary edge)

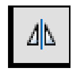
TIP:

- Use the object selection option FENCE to choose multiple objects

6.7 MIRROR

1. **Choose** Modify, Mirror.

   or

2. **Click** the Mirror icon. 

   or

3. **Type** MIRROR at the command prompt. Command:

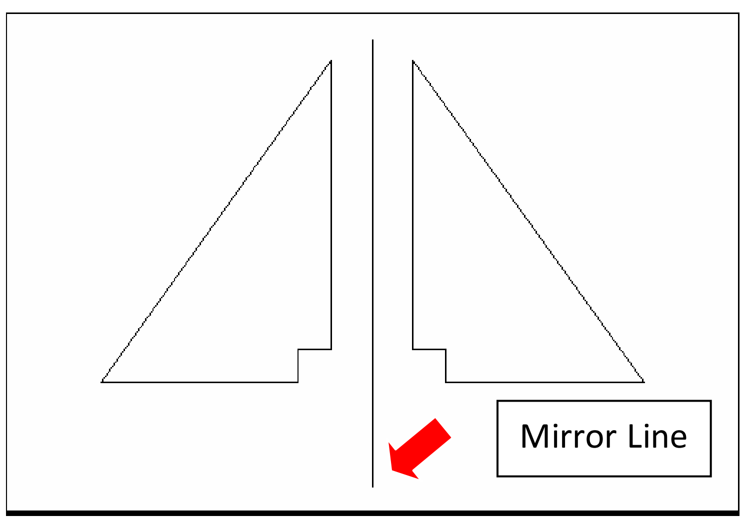
   MIRROR

4. **Pick** Objects to mirror.
   Select objects:(**select**)

5. **Pick** First point of mirror line: (**point**)

6. **Pick** Second point: (**point**)

7. **Type** Yes to delete the original objects and
   No to keep them.
   Delete old objects? **Y** or **N**



6.8 ROTATE

1. **Choose** Modify, Rotate.

   or

2. **Click** the Rotate icon. 

   or

3. **Type** ROTATE at the command prompt
   Command : **ROTATE**

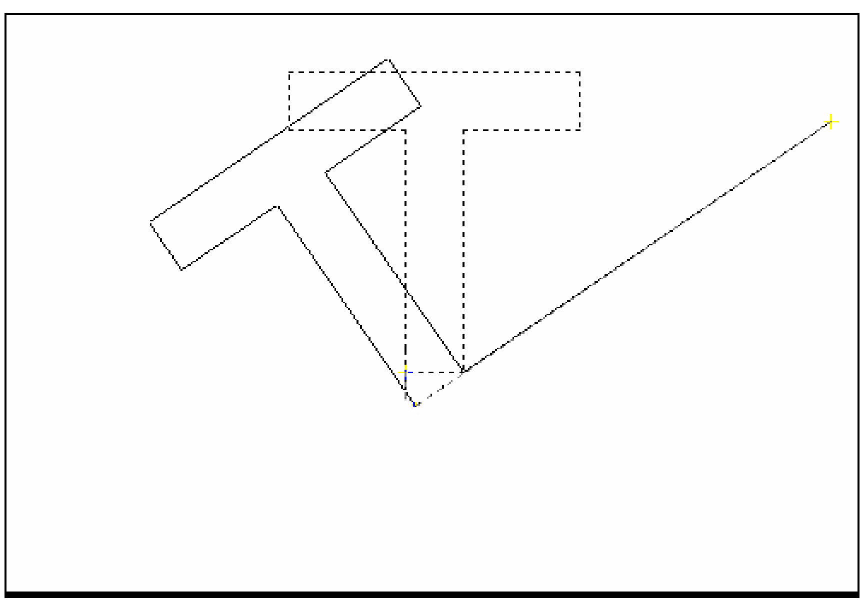
4. **Pick** Objects to rotate: Select objects:(**select**)

5. **Pick** A pivot point to rotate around
   Base point: (**point**)

6. **Type** A rotation angle<Rotation angle>/Reference: (**number**)

   or

7. **Pick** A rotation angle<Rotation angle>/Reference: (**point**)



Reference Angle Rotation

A positive angle causes counterclockwise rotation, and a negative angle produces clockwise rotation. If you respond to the last prompt with r, you can specify the current rotation and the new rotation you want. AutoCAD prompts:

1. **Type** R for a rotation angle<Rotation angle>/Reference: (**R**)
2. **Choose** An existing rotation angle Rotation angle: (**number or points**)
3. **Choose** A new rotation angle New angle: (**number or points**)

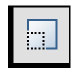
TIP:

- You can show AutoCAD the reference angle (by pointing to the two endpoints of a line to be rotated), and then specify the new angle. You can specify the new angle by pointing or by dragging the object.

6.9 SCALE

1. **Choose** Modify, Scale.

   or

2. **Click** the Scale icon. 

   or

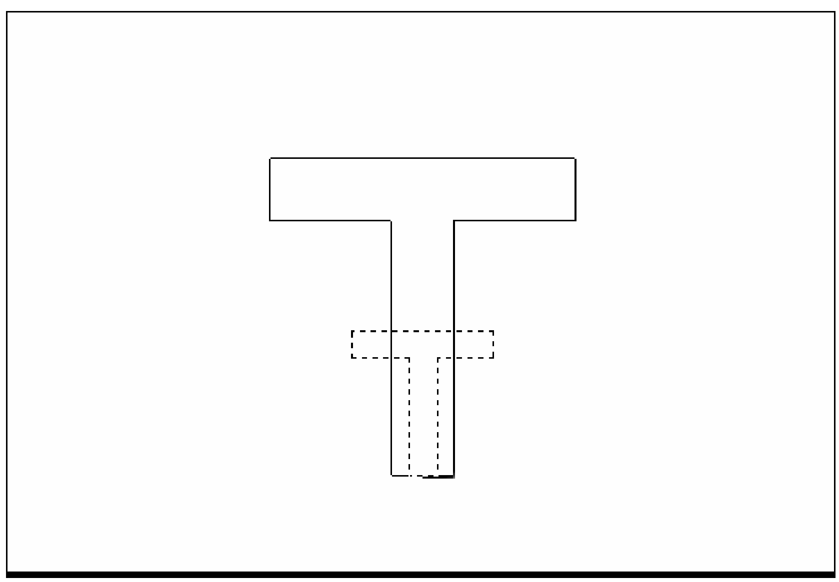
3. **Type** SCALE at the command prompt
   Command: **SCALE**
   Select objects: (**select objects**)

4. **Pick** A pivot point to scale about Base point: (**point**)

5. **Type** A rotation angle<Scale factor>/Reference:(**number**)

   or

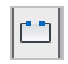
6. **Pick** A scale factor<Scale factor>/Reference: (**point**)
   Scale factor/Reference: (**points**)



6.10 Break

1. **Choose** Modify, Break.

Or

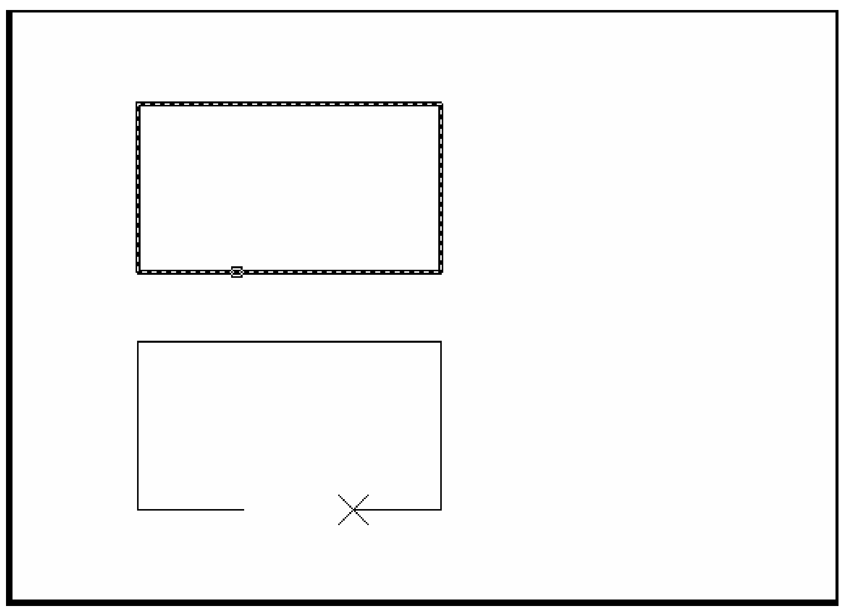
2. **Click** the Break icon. 

or

3. **Type** BREAK at the command prompt.
   Command: **BREAK**

4. **Pick** Object to break.
   Select object: (**select one object**)

5. **Pick** A second break point.
   Enter second point : (**point**)



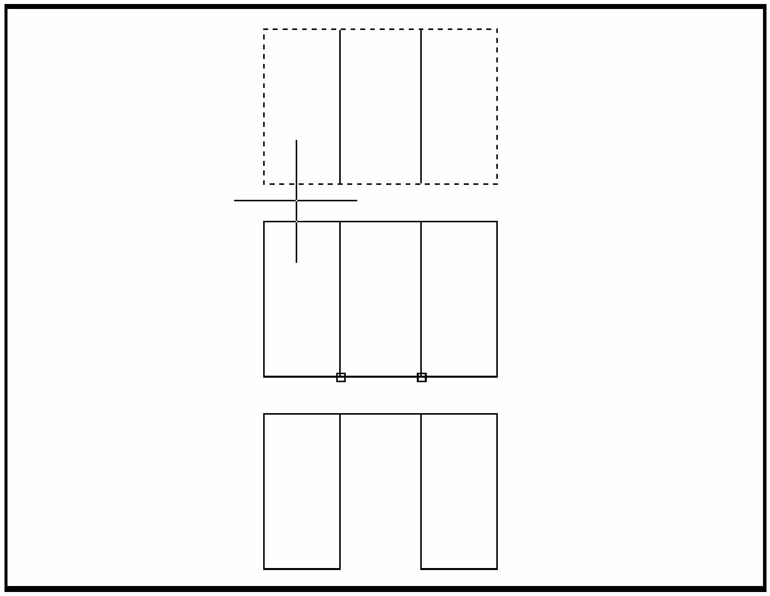
or

6. **Type** **F** to choose a different break point

   Enter second point (or F for first point

7. **Pick** The first break point on the object
   Enter first point: (**point**)

8. **Pick** A second break point



TIPS:

You can also type coordinates instead of picking a break point. Enter second point (or F for first point) @3'<0

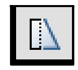
If you break a circle, it changes to an arc by deleting the portion from the first point to the second, going counterclockwise.

Breaking a Polyline with nonzero width will cause the ends to be cut square.

6.11 Stretch

1. **Choose** Modify, Stretch.

or

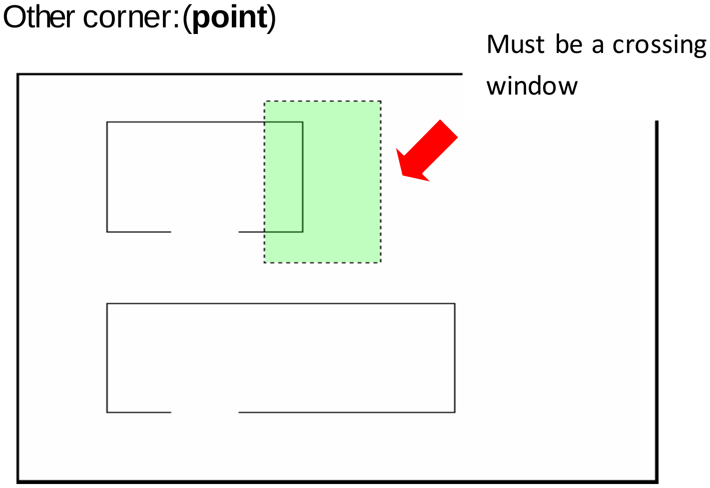
2. **Click** the Stretch icon. 

3. **Type** STRETCH at the command prompt.
Command : **STRETCH** Select objects to stretch by window...

4. **Type** C to choose CROSSING window
Select objects: **C**

5. **Pick** A first corner to stretch. First corner: (**point**)

6. **Pick** The opposite corner to window the objects to stretch.
Other corner: (**point**)



7. **Press** ENTER to accept objects to stretch.

8. **Pick** A base point to stretch from Base point: (**point**)

9. **Pick** A point to stretch to New point: (**point**)

or

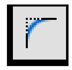
10. **Type** A distance to stretch. New point: **@1<0**

TIP:
The Stretch command must use a CROSSING window or a CROSSING POLYGON window.

6.12 Fillet

1. **Choose** Modify, Fillet.

   or

2. **Click** the Fillet icon. 

   or

3. **Type** FILLET at the command prompt.
   Command: **FILLET**

4. **Pick** First object to fillet. Polyline/Radius/Trim<Select two objects>: select first object.

5. **Pick** Second object to fillet.
   Select second object: select second object.

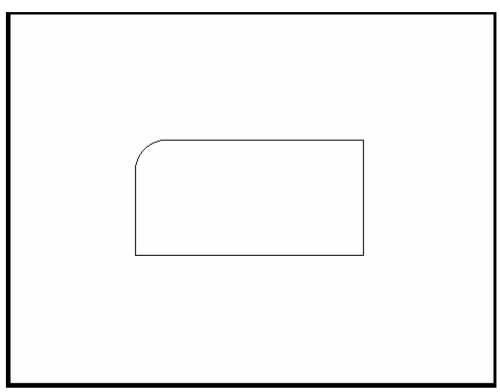
   or

6. **Type** One of the following options:

P Fillets an entire Polyline.

R Sets the fillet radius.

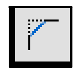
T Sets the trimmode (trim cuts the fillet corner and no trim keeps the fillet corner)



TIP:

- You can also fillet PARALLEL lines as well as PLINES with LINES
- Type a radius of zero (0) to create a clean 90 degree corner

6.13 Chamfer

1. **Choose** Modify, Chamfer.
   or
2. **Click** the Chamfer icon. 
   or
3. **Type** CHAMFER at the command prompt.
   Command: **CHAMFER**
4. **Pick** First object to chamfer.
   Polyline/Distance/Angle/Trim/Method<Select first line>: **select first object**
5. **Pick** Second object to chamfer.
   Select second object: select second object.
   or
6. **Type** One of the following options:

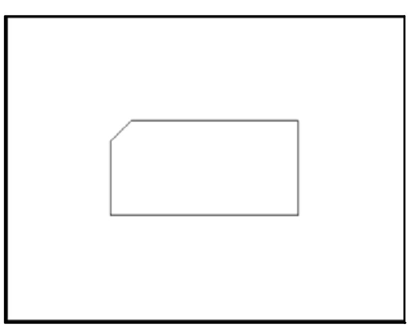
P Chamfers entire Polyline.

D Sets chamfer distances.
A Uses a distance and angle method instead of two distances.

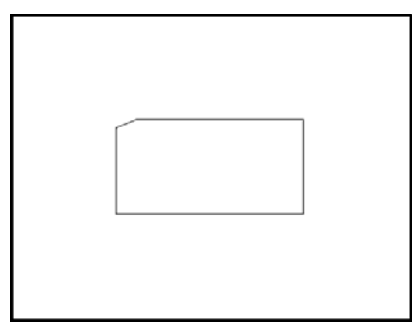
T Sets the trim mode

M Sets the method to distance or angle.

Chamfer with equal distances



Chamfer with different distances



6.14 Array

Rectangular Array

To draw rectangular array:

1. **Choose** Modify, Array.

   or

2. **Click** the Array icon. 

   or

3. **Type** ARRAY at the command prompt.
   Command : **ARRAY**

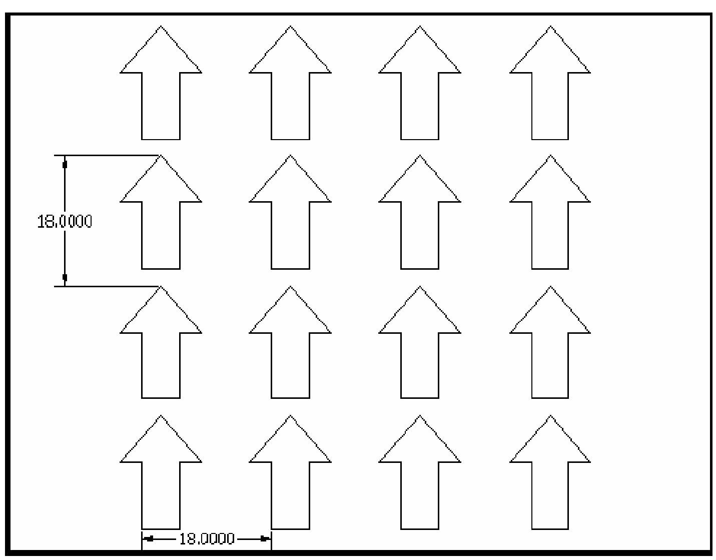
4. **Pick** Objects to array. Select
   objects : (**select**)

5. **Type** R for a rectangular array.
   Enter array type [Rectangular/PAth/POlar] <Rectangular>: **R**

   Type = Rectangular Associative = Yes

   Select grip to edit array or [ASsociative/Base point/COUnt/Spacing/COLumns/Rows/Levels/eXit]<eXit>:

6. **Type** **S** to change the spacing distance between each column and/or row.
7. **Type** **COL** to change the number of columns.
8. **Type** **R** to change the number of rows.

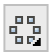


Polar Array

To draw a polar array:

1. **Choose** Modify, ARRAY.

   or

2. **Click** the Array icon. 

   or

3. **Type** ARRAY at the command prompt.

   Command:
   ARRAY

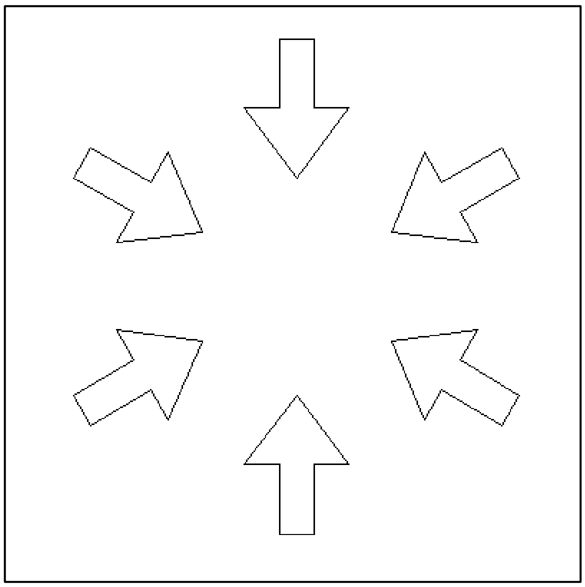
4. **Pick** Objects to array.
   Select objects: (**select**)

5. **Type** P to draw a polar array. Enter array type
   [Rectangular/PAth/POlar] <Rectangular>: PO
   (R/P): **PO**

6. **Pick** A center point for the array. Center point of array:
   pick point

7. **Type** one of the following options or press **ENTER**

   Select grip to edit array or [ASsociative/Base point/Items/Angle between/Fill angle/ROWs/Levels/ROTate items/eXit]<eXit>:



Chapter 7

Dimensions

7.1 Linear Dimensions

1. **Choose** Dimension, Linear.

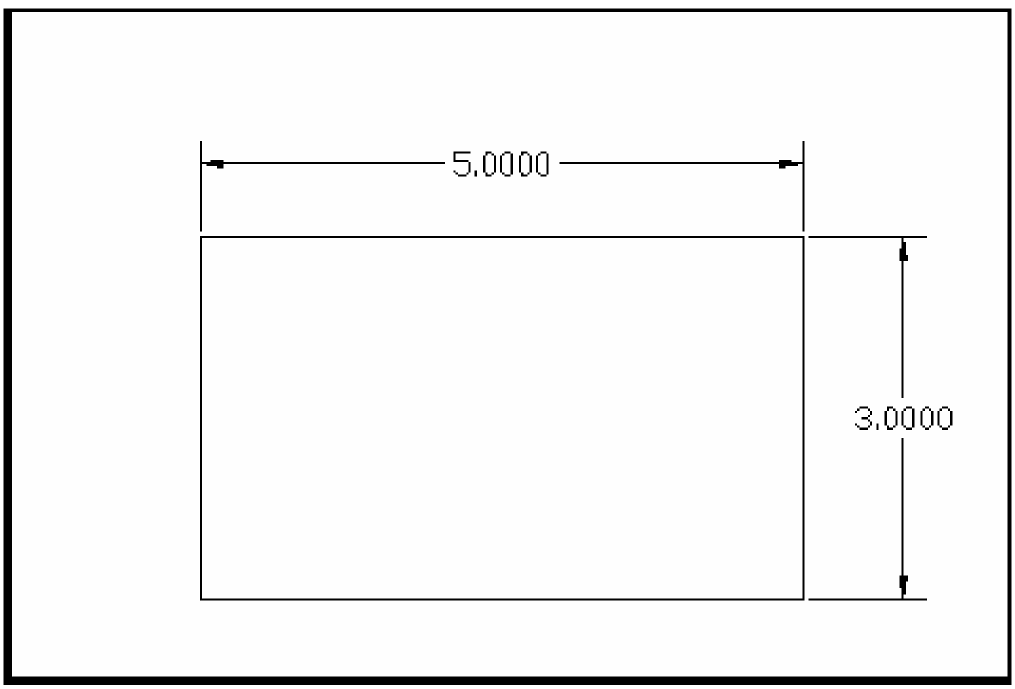
or

2. **Click** the Linear Dimension command from the toolbar.

or

3. **Type** DIM at the command prompt. Command: **DIM**

Dim: HOR or VER

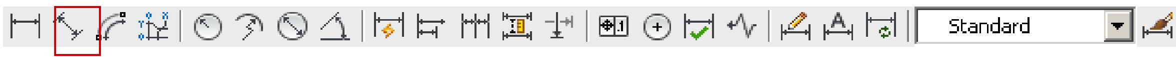


7.2 Aligned Dimensions

1. **Choose** Dimension, Aligned.

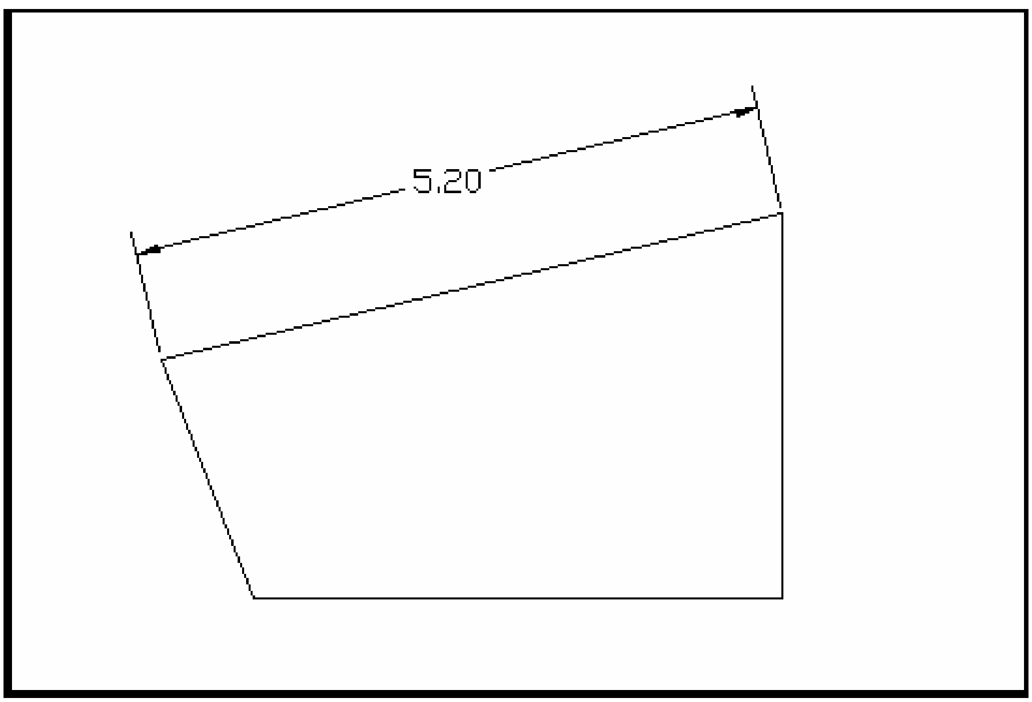
or

2. **Click** the Aligned Dimension command from the toolbar.



or

3. **Type** DIM at the command prompt. Command: **DIM**
   Dim: **ALIGNED**

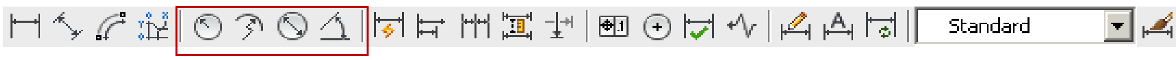


7.3 Radial Dimensions

1. **Choose** Dimension, Radius or Diameter.

or

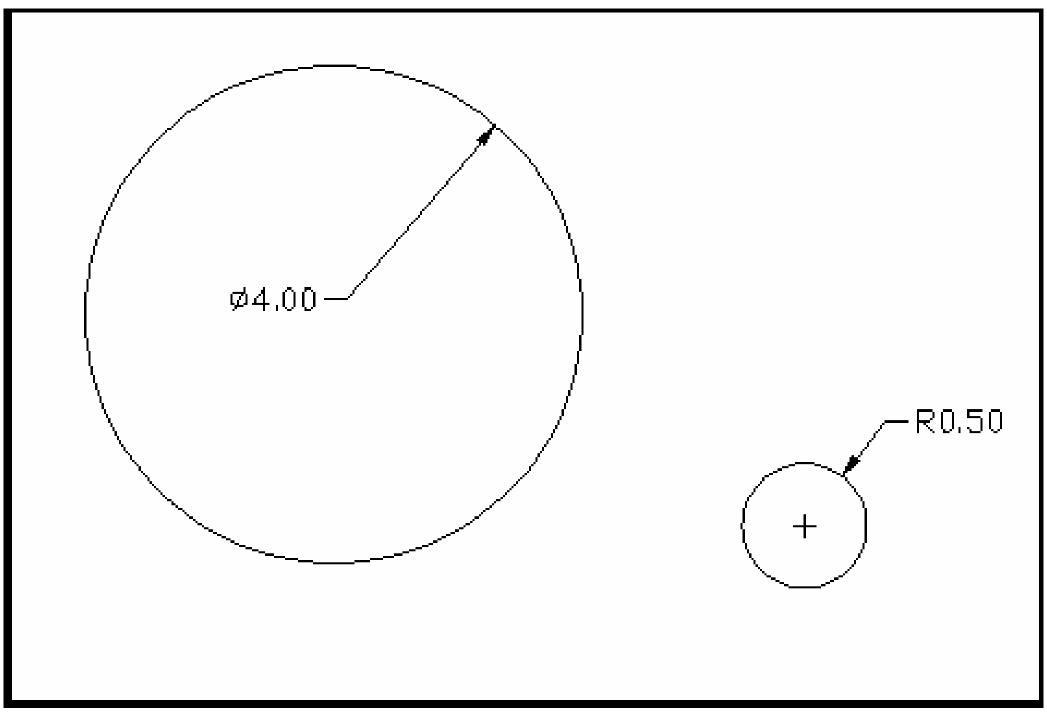
2. **Click** the Radial Dimensions command from the toolbar.



or

3. **Type** DIM at the command prompt. Command: **DIM**

Dim: **RADIUS or DIAMETER**

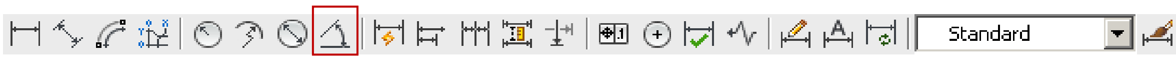


7.4 Angular Dimensions

1. **Choose** Dimension, Angular.

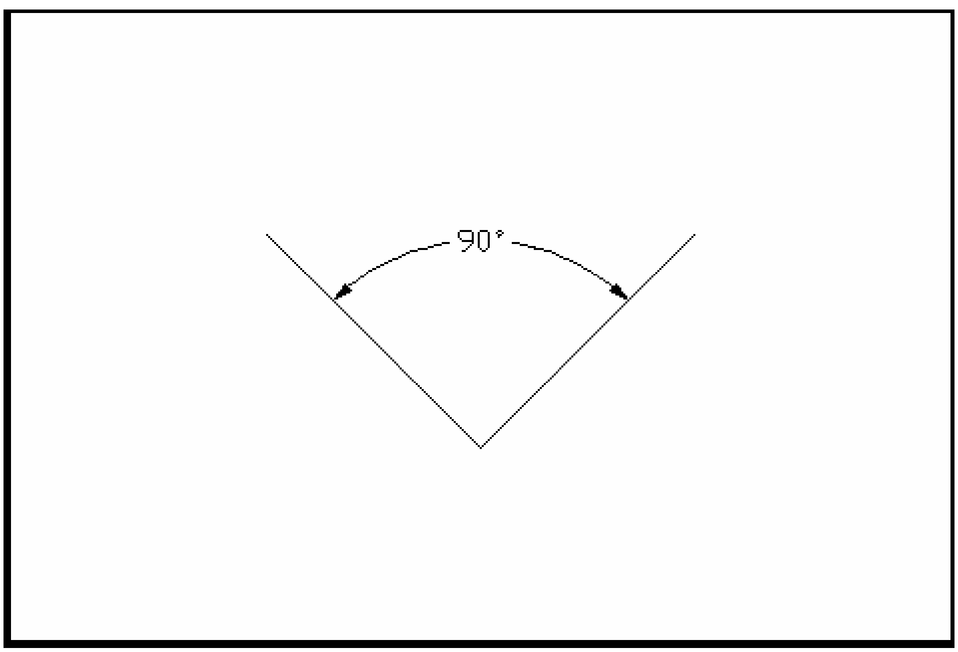
or

2. **Click** the Angular Dimensions command from the toolbar.



or

3. **Type** DIM at the command prompt. Command: **DIM**
Dim: **ANGULAR**

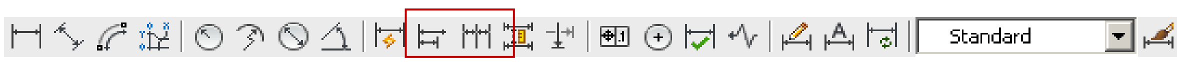


7.5 Continued and Baseline Dimensions

1. **Choose** Dimension, Continue or Baseline.

or

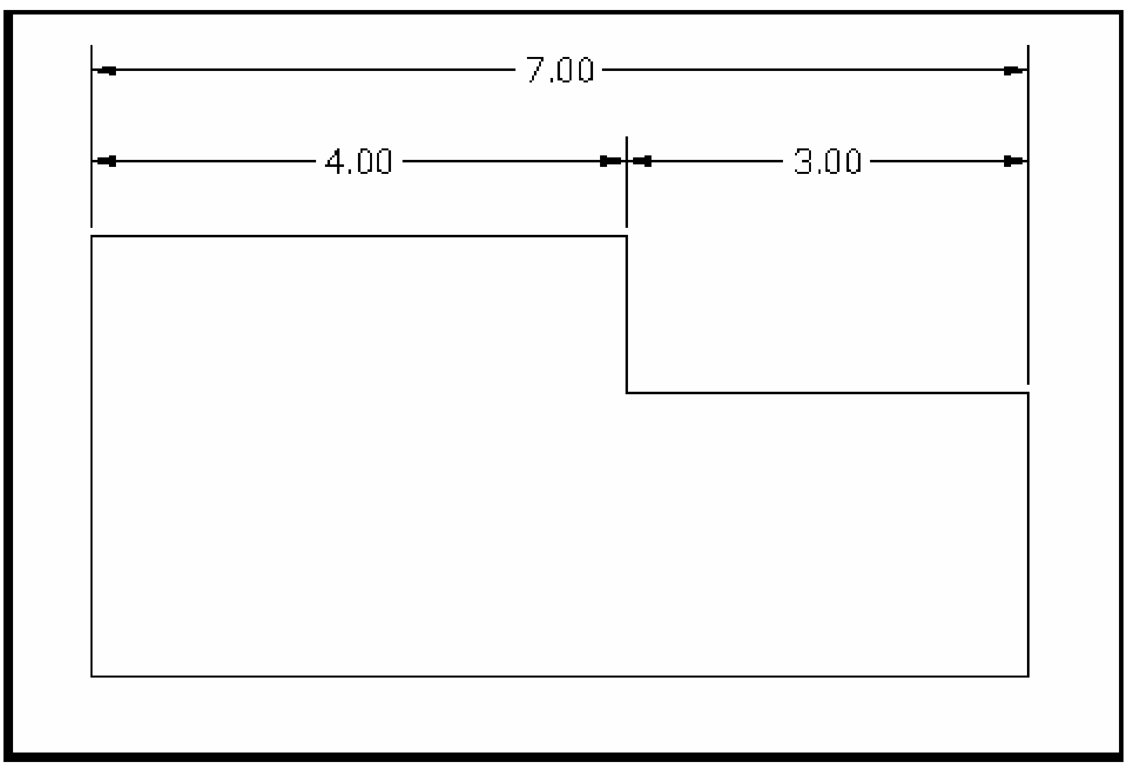
2. **Click** the Continue or Baseline Dimensions command from the toolbar.



or

3. **Type** DIM at the command prompt. Command: **DIM**

Dim: **CONTINUE or BASELINE**

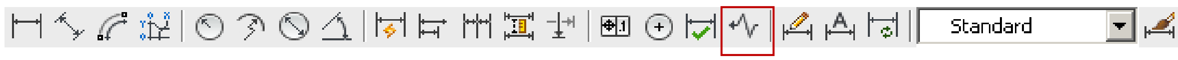


7.6 Leaders

1. **Choose** Dimension, Leader...

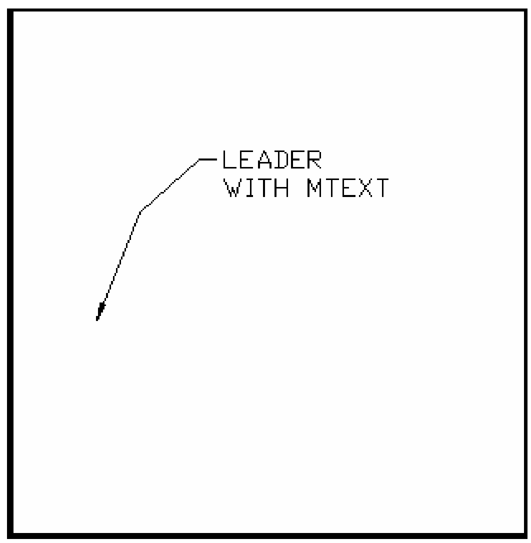
or

2. **Click** the Leader icon from the Dimension toolbar.



or

3. **Type** QLEADER at the command prompt. Command:
QLEADER



7.7 Quick Dimensions

Quickly creates dimension arrangements from the geometry you select.

1. **Choose** Dimension, QDIM.

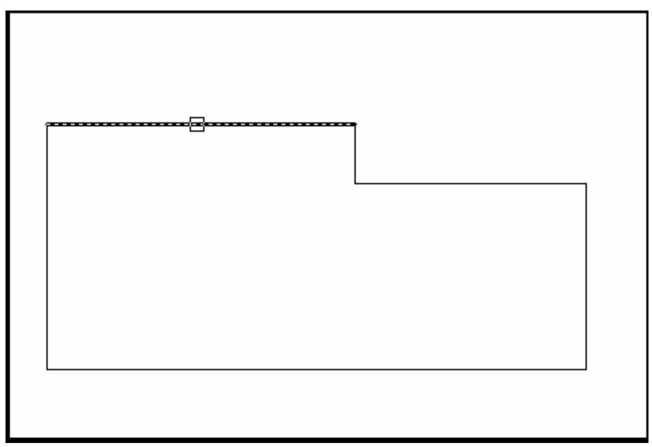
   or

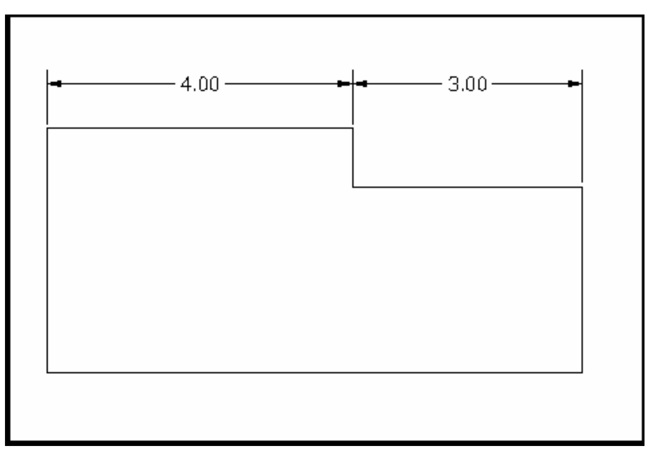
2. **Click** the Quick Dimension icon from the Dimensions toolbar.

   or

3. **Type** QDIM at the command prompt. Command: **QDIM**

4. **Pick** the objects to dimension.





7.8 Creating Dimension Styles

1. **Choose** Format, Dimension Style...

   or

2. **Choose** Dimension, Style.

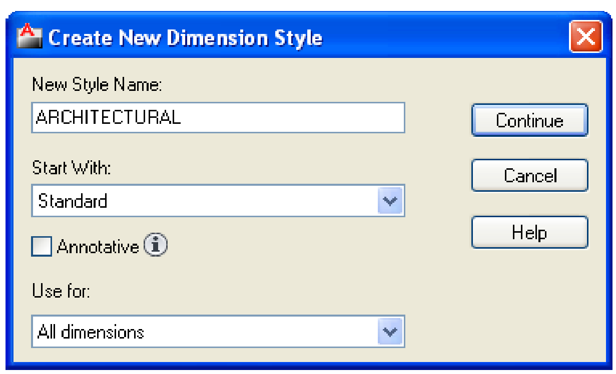
   or

3. **Choose** Dimension Style icon from the Dimension Style toolbar.

4. **Type** DDIM at the command prompt

   Command:**DDIM**

5. **Choose** New... from the dialog box.
6. **Create** a new style from the existing styles.



7. **Click** the Continue button.

TIP:

- All dimension variables except for DIMSHO and DIMASO can be saved as a style.

Chapter 8

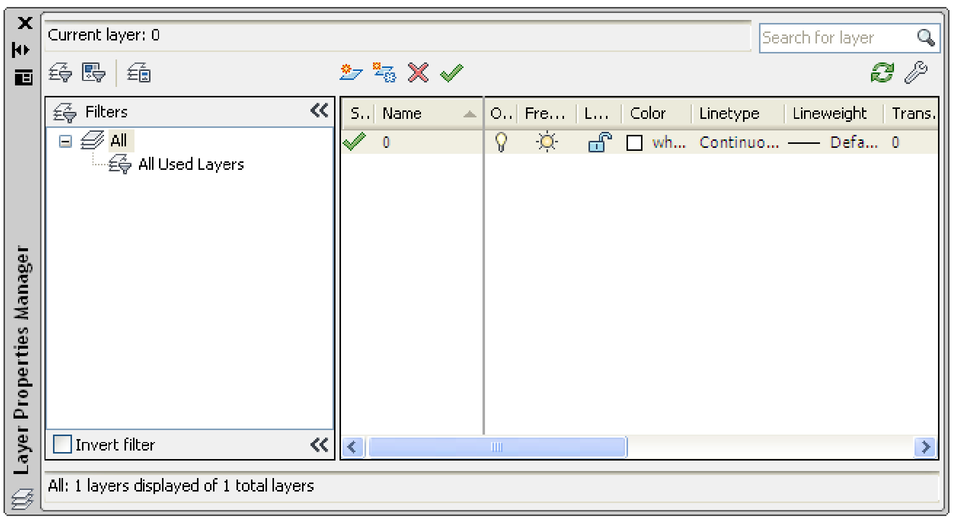
Layers, Linetypes, Colors

8.1 Introduction to Layers and Layer Dialog Box

1. **Choose** The Layer Properties icon. 

or

2. **Type** LAYER at the command prompt.
Command: **LAYER (or LA)**



8.2 Layer Options

?	Lists layers, with states, colors and linetypes
Make	Creates a new layer and makes it current
Set	Sets current layer
New	Creates new layers
ON	Turns on specified layers
OFF	Turns off specified layers
Color	Assigns color to specified layers
Ltype	Assigns linetype to specified layers
Freeze	Completely ignores layers during regeneration
Thaw	Unfreezes specified layers Ltype
Lock	Makes a layer read only preventing entities from being edited but available visual reference and osnap functions
Unlock	Places a layer in read write mode and available for edits
Plot	Turns a Layer On for Plotting
No Plot	Turns a Layer Off for Plotting
LWeight	Controls the line weight for each layer

TIP:

- Layers can be set using the command line prompts for layers. To use this, type
- LAYER or -LA at the command prompt

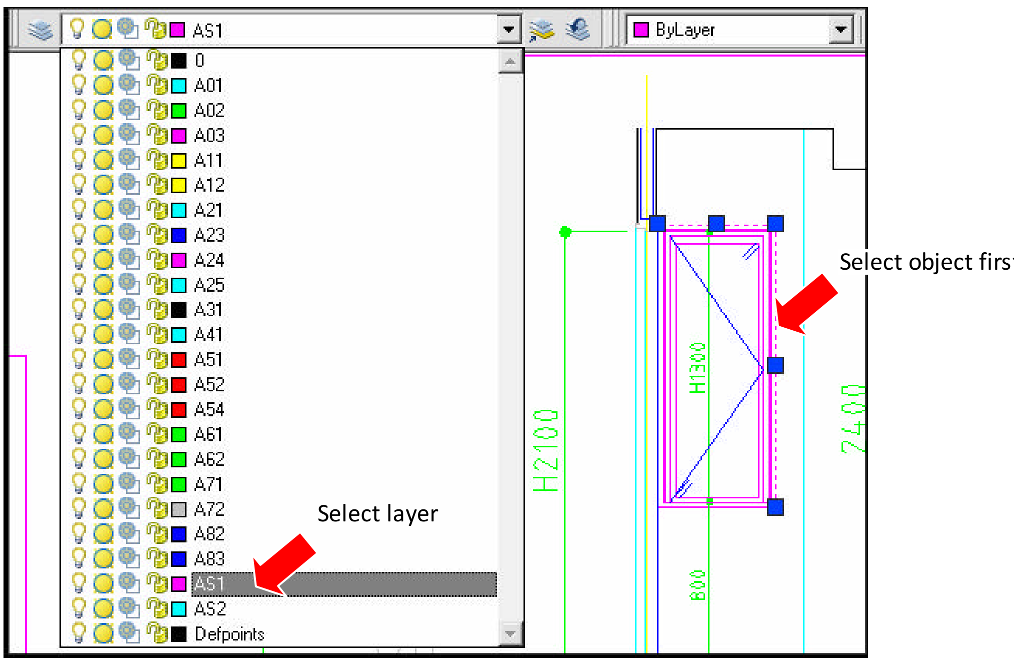
1. **Type** Command: -**LAYER** or **LA**
2. **Type** One of the following layer options
   ?/Make/Set/New/ON/OFF/Color/Ltype/Freeze/Thaw:

8.3 Layer Shortcuts

Changing the Layer of an Object

1. **Click** Once on the object to change.
2. **Select** the desired layer from the Layer Control Box dropdown.

AutoCAD will move the object to the new layer



8.4 Making a Layer Current

1. **Click** — once on the Make Object's Layer Current icon. 
2. **Select** — object whose layer will become current:

Match Properties

1. **Choose** — Modify, Match Properties.

   or

2. **Click** — the Match Properties Icon from the Layer toolbar. 

   or

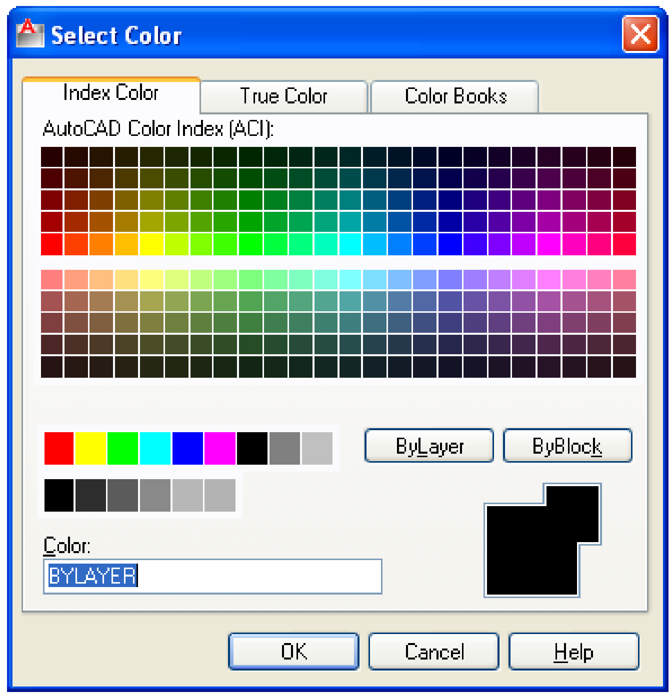
3. **Type** — Command : **MATCHPROP or MA**
4. **Select** — the object whose properties you want to copy (1).
5. **Select** — the objects to which you want to apply the properties (2).

8.5 Color Command

1. **Type** COLOR at the command prompt.
   Command: **COLOR or COL**

   or

2. **Choose** Color on the Object Properties toolbar and then select a color from the list or select Other to display the Select Color dialog box.



TIP:

- These settings ignore the current layer settings for color.

By Layer

If you enter bylayer, new objects assume the color of the layer upon which they are drawn.

By Block

If you enter byblock, AutoCAD draws new objects in the default color (white or black, depending on your configuration) until they are grouped into a block. When the block is inserted in the drawing, the objects in the bloc k inherit the current setting of the COLOR command.

8.6 Linetypes

Lines of different types and thicknesses are used for graphical representation of objects. The types of lines and their applications are shown in Table 8.1. Typical applications of different types of lines are shown in Figs. 8.1 and 8.2.

Table 8.1 Types of lines and their applications

Line	*Description*	*General Applications*
A	Continuous thick	A1 Visible outlines
B	Continuous thin (straight or curved)	B1 Imaginary lines of intersection B2 Dimension lines B3 Projection lines B4 Leader lines B5 Hatching lines B6 Outlines of revolved sections in place B7 Short centre lines
C	Continuous thin, free-hand	C1 Limits of partial or interrupted views and sections, if the limit is not a chain thin
D	Continuous thin (straight) with zigzags	D1 Line (see Fig. 2.5)
E	Dashed thick	E1 Hidden outlines
G	Chain thin	G1 Centre lines G2 Lines of symmetry G3 Trajectories
H	Chain thin, thick at ends and changes of direction	H1 Cutting planes
J	Chain thick	J1 Indication of lines or surfaces to which a special requirement applies
K	Chain thin, double-dashed	K1 Outlines of adjacent parts K2 Alternative and extreme positions of movable parts K3 Centroidal lines

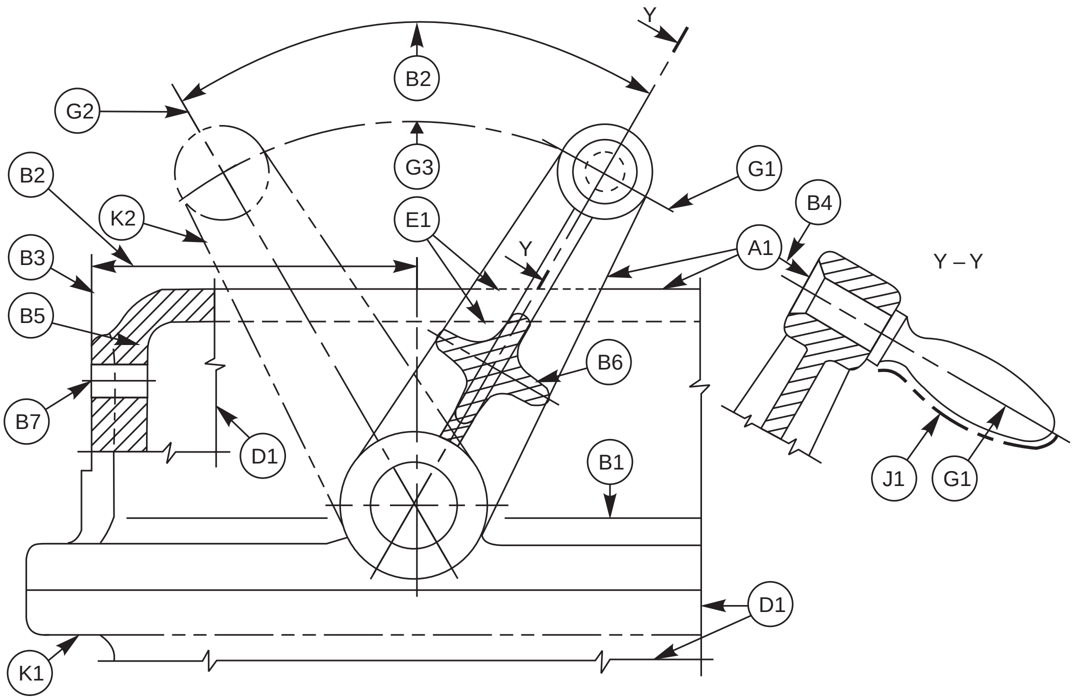


Fig. 8.1 Applications of lines

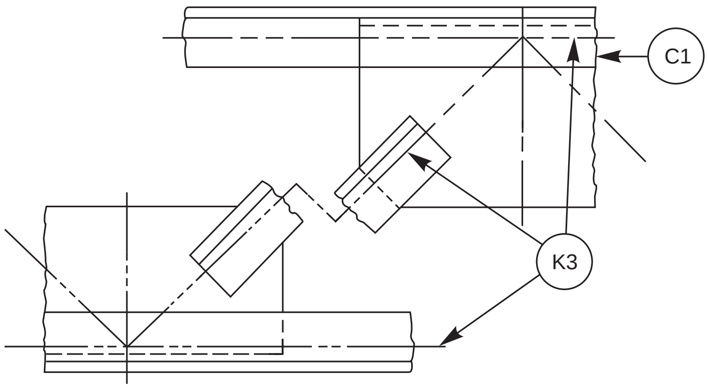
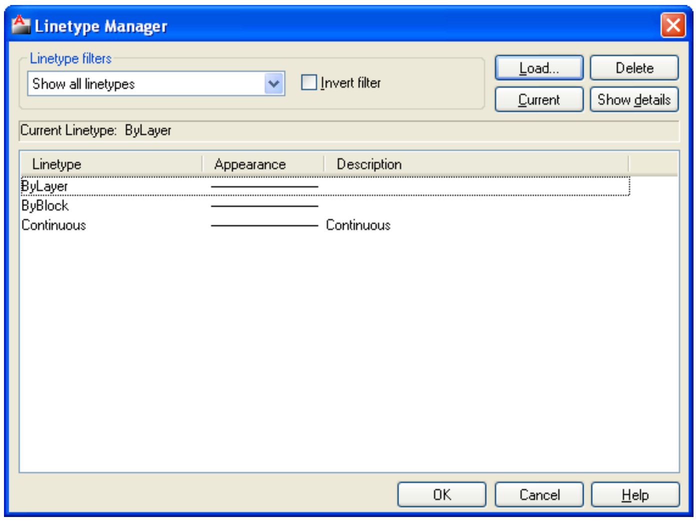


Fig. 8.2 Applications of lines

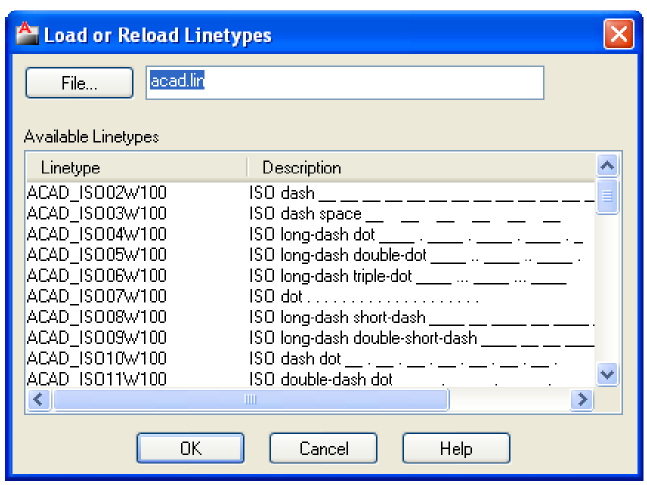
8.6.1 Loading and Changing Linetypes

1. **Type** LTYPE at the command prompt. Command: **LTYPE or LT**

2. **Choose** Load... to see a list of available linetypes.



3. **Choose** the desired linetype to assign.

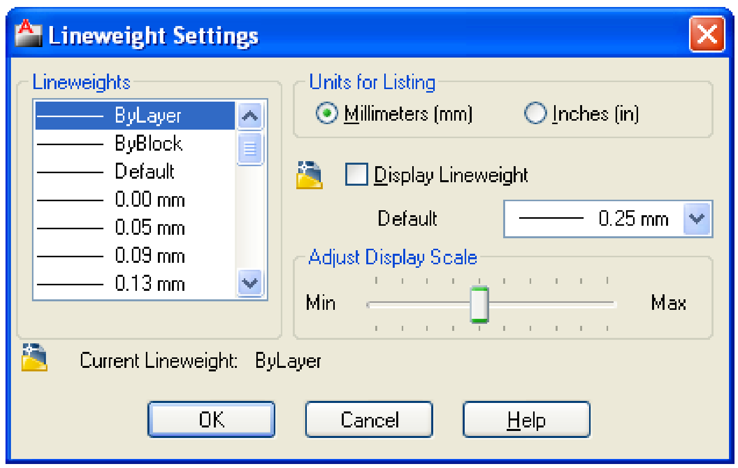


4. **Click** OK.

8.6.2 Lineweights

Loading and Changing Lineweights

1. **Type** LINEWEIGHT at the command prompt.
Command: **LINEWEIGHT or LWEIGHT**



TIPS:

- Lineweights can also be assigned to layers.
- The Display Lineweights feature can be turned on/off o n the status bar to show or not show lineweights in the drawing, thus making regenerations faster.

- Lineweights are displayed using a pixel width in proportion to the real-world unit value at which they plot. If you are using a high-resolution monitor, you can adjust the lineweight display scale to better display different lineweight widths.

Chapter 9
Text

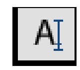
9.1 Text Command

Text

Creates a single-line text object

1. **Type** TEXT at the command prompt
   Command: **TEXT**

   or

2. **Pick** the Single Line Text icon from the Text Toolbar. 

3. **Pick** A start point
   Justify/Style/<Start Point>: (**point**)

   or

4. **Type** J to change the justification or S to change the text style.

5. **Type** A text height
   Height <default>: (**type value or pick two points**)

6. **Type** A rotation angle
   Rotation angle <default>: (**angle or point**)

7. **Type** A text string
   Text: (**type text string**)

8. **Press** enter to exit the Text: prompt.

DTEXT (Dynamic Text)

Creates a single-line text object, showing the text dynamically on the screen as it is entered.

1. **Choose** Draw, Text, Single Line Text.

   or

2. **Type** DTEXT at the command prompt
   Command : **DTEXT**

3. **Follow** the steps 3-8 from above.

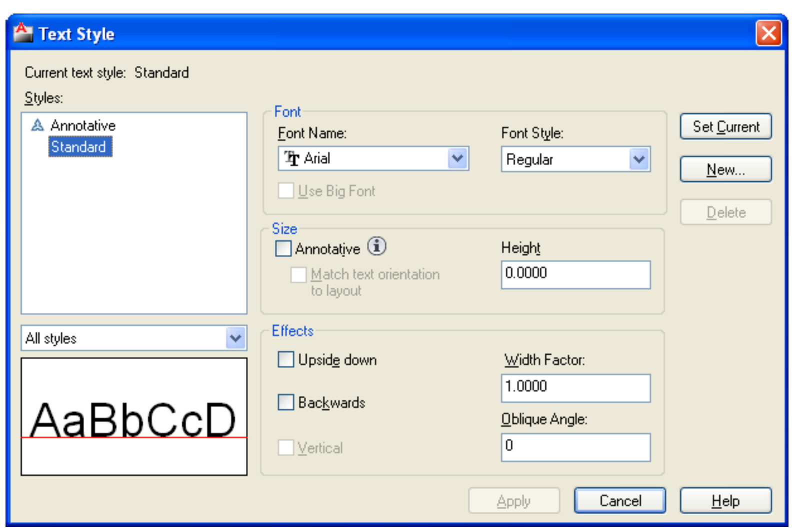
9.2 Text Styles

Style Command

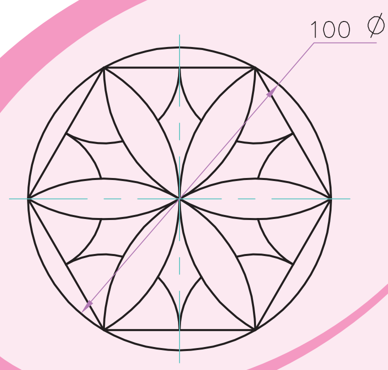
1. **Choose** Format, Text Style...

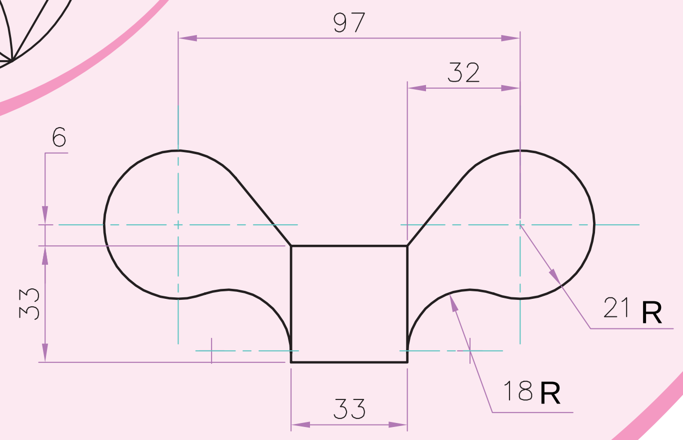
   or

2. **Type** STYLE at the command prompt. Command: **STYLE**
3. **Choose** a style from the menu or create a NEW style.
4. **Choose** a font file.
5. **Type** a height for the text (set to zero to vary heights)
6. **Type** a width factor for each character. Width factor <1>: (**enter**)
7. **Type** an obliquing (slant) angle. Obliquing angle <0>: (**angle or enter**)
8. **Type** Yes or No to place characters backwards. Backwards? (**Y** or **N**)
9. **Type** Yes or No to draw characters upside down. Upside down? (**Y** or **N**)
10. **Type** Yes or No to draw characters vertically

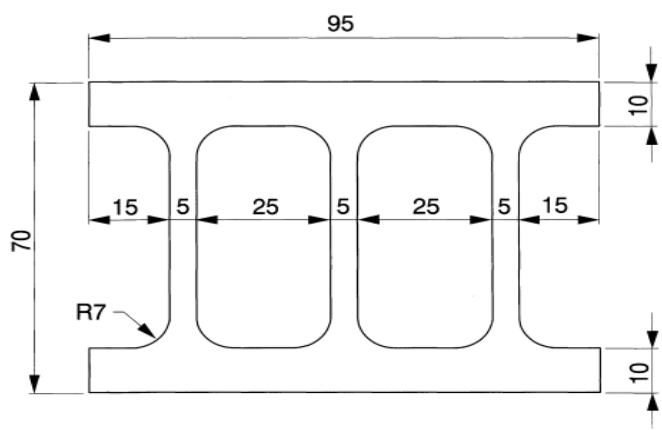


DRAWING EXERCISES

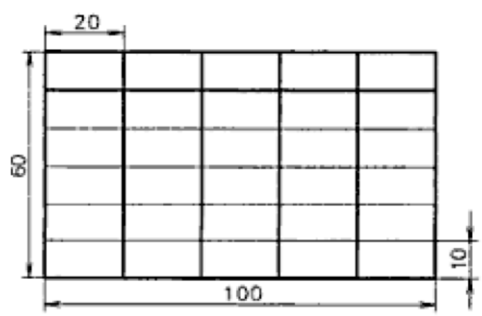




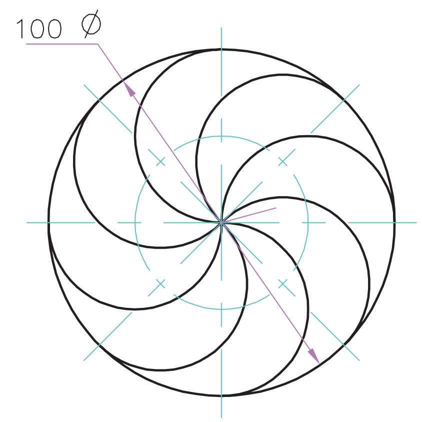
DRAWING EXERCISES



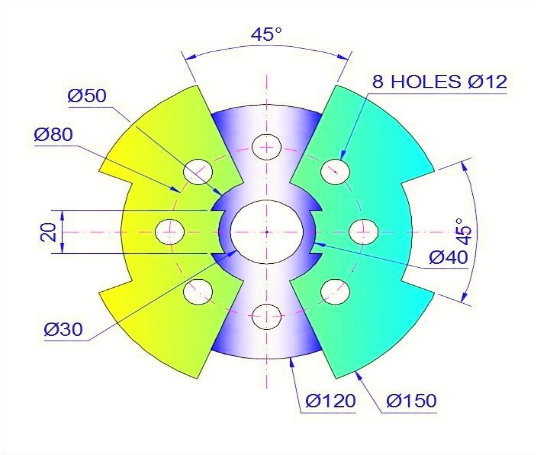
Exercise (14)



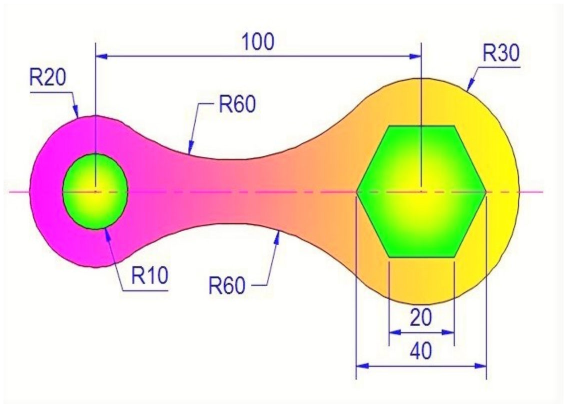
Exercise (15)



Exercise (16)

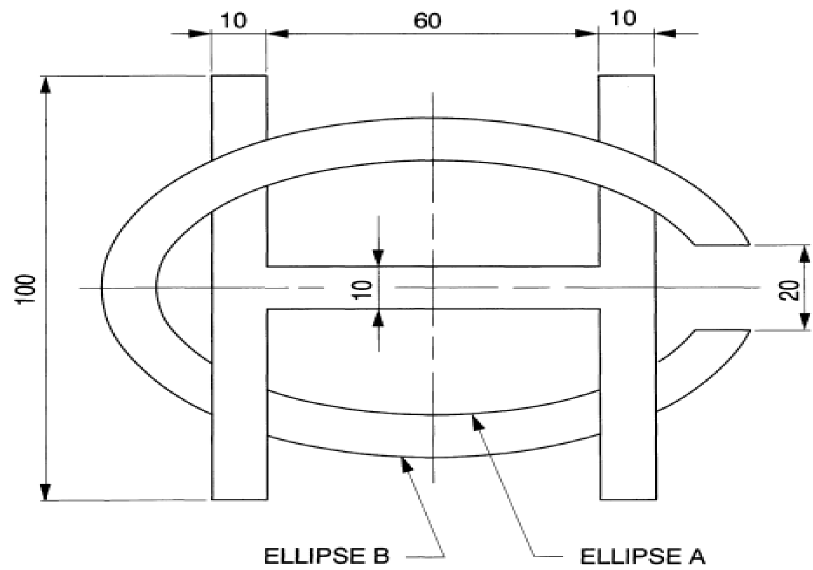


Exercise (17)



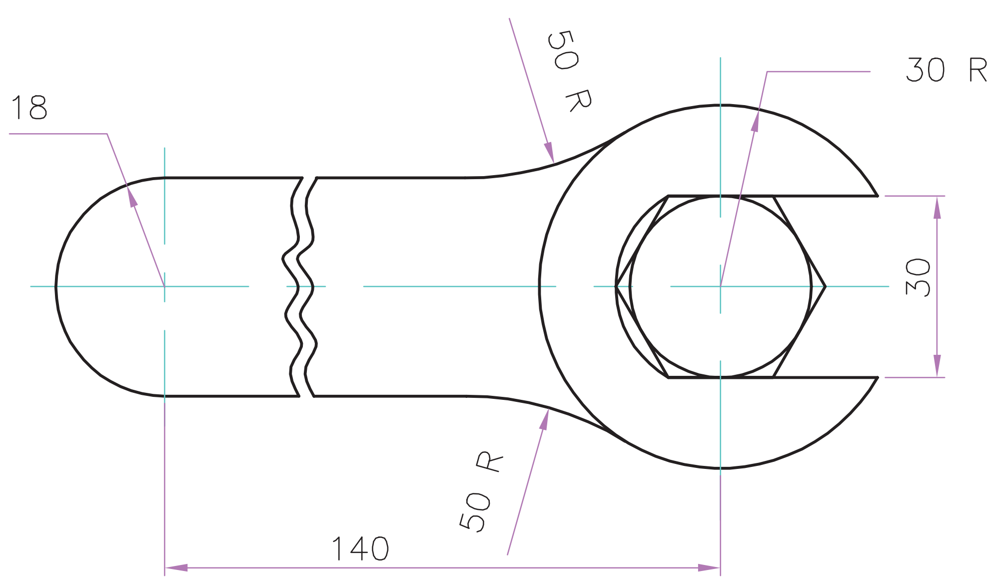
Exercise (18)

Exercise (19)

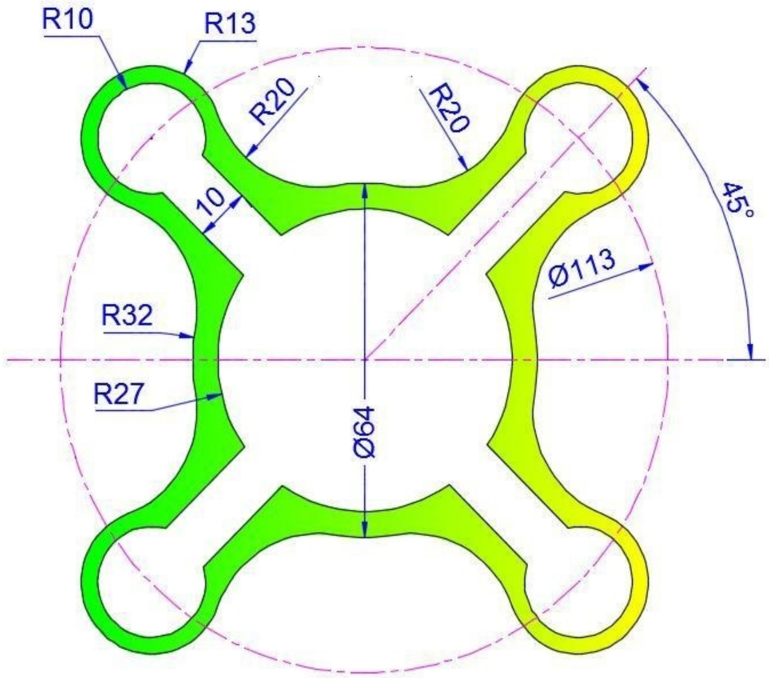


Ellipse A has a major axis of 100 mm and minor axis of 60 mm.
Ellipse B has a major axis of 120 mm and a minor axis of 80 mm.

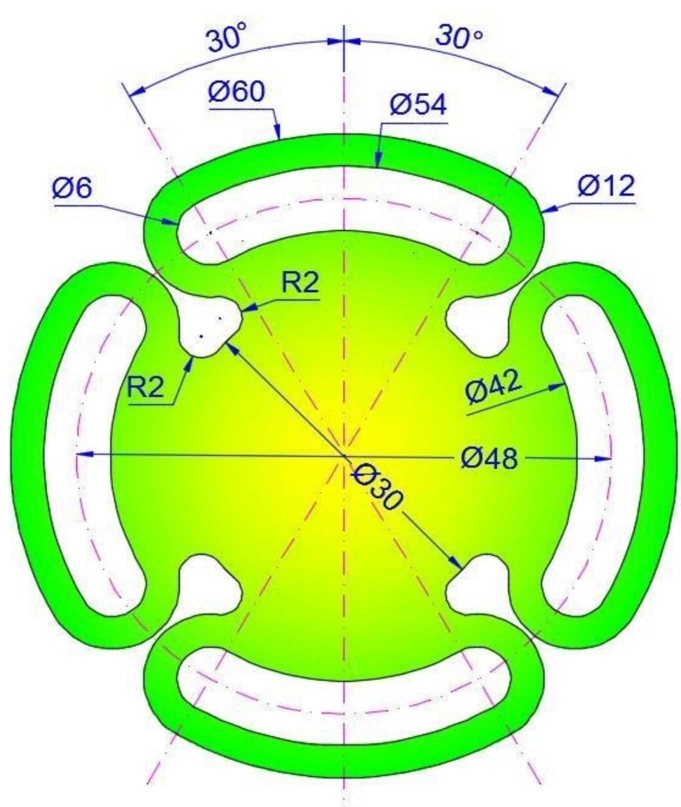
Exercise (20)



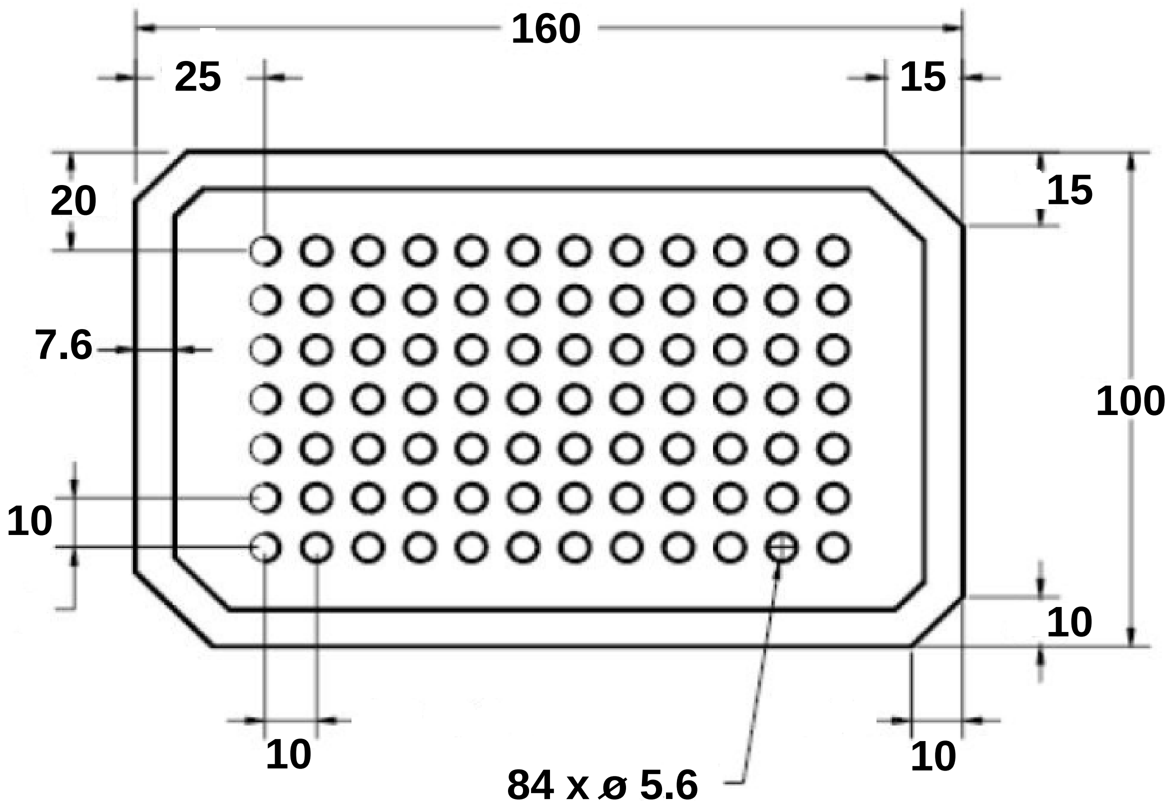
Exercise (21)



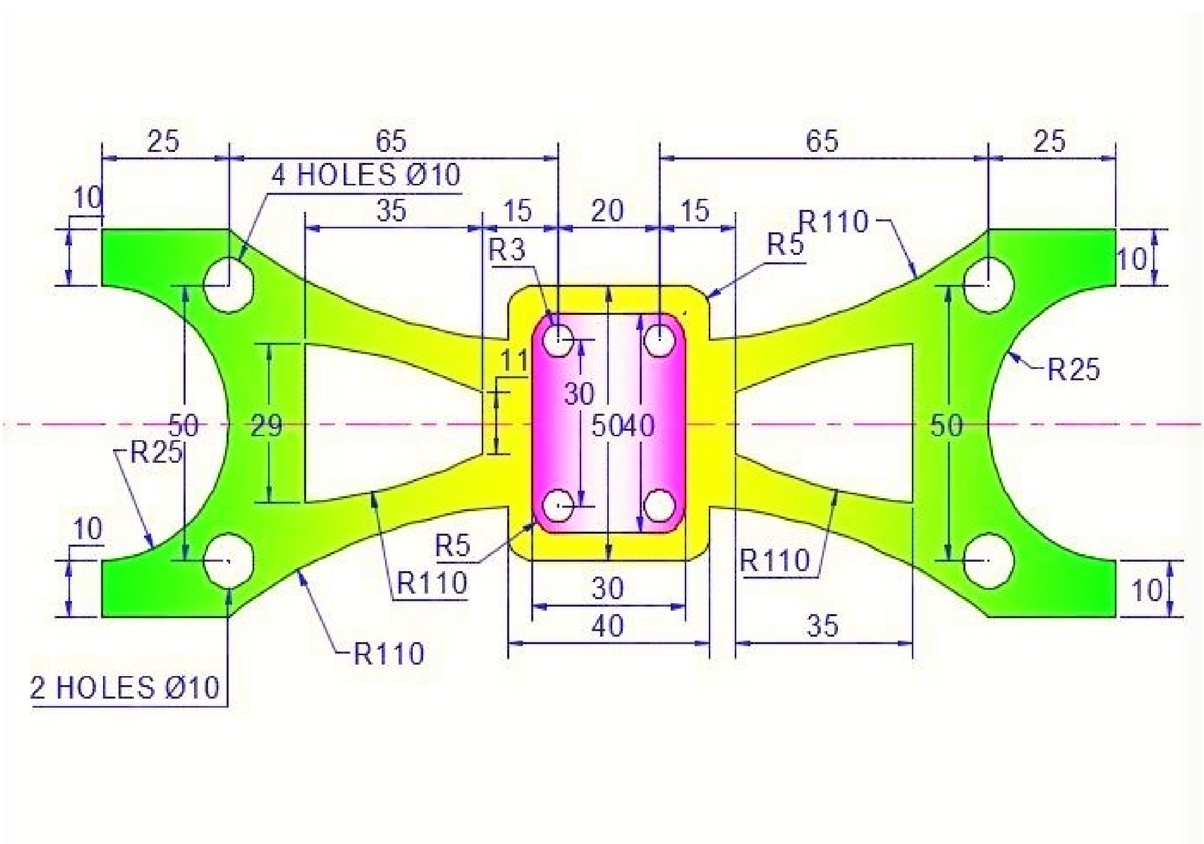
Exercise (22)



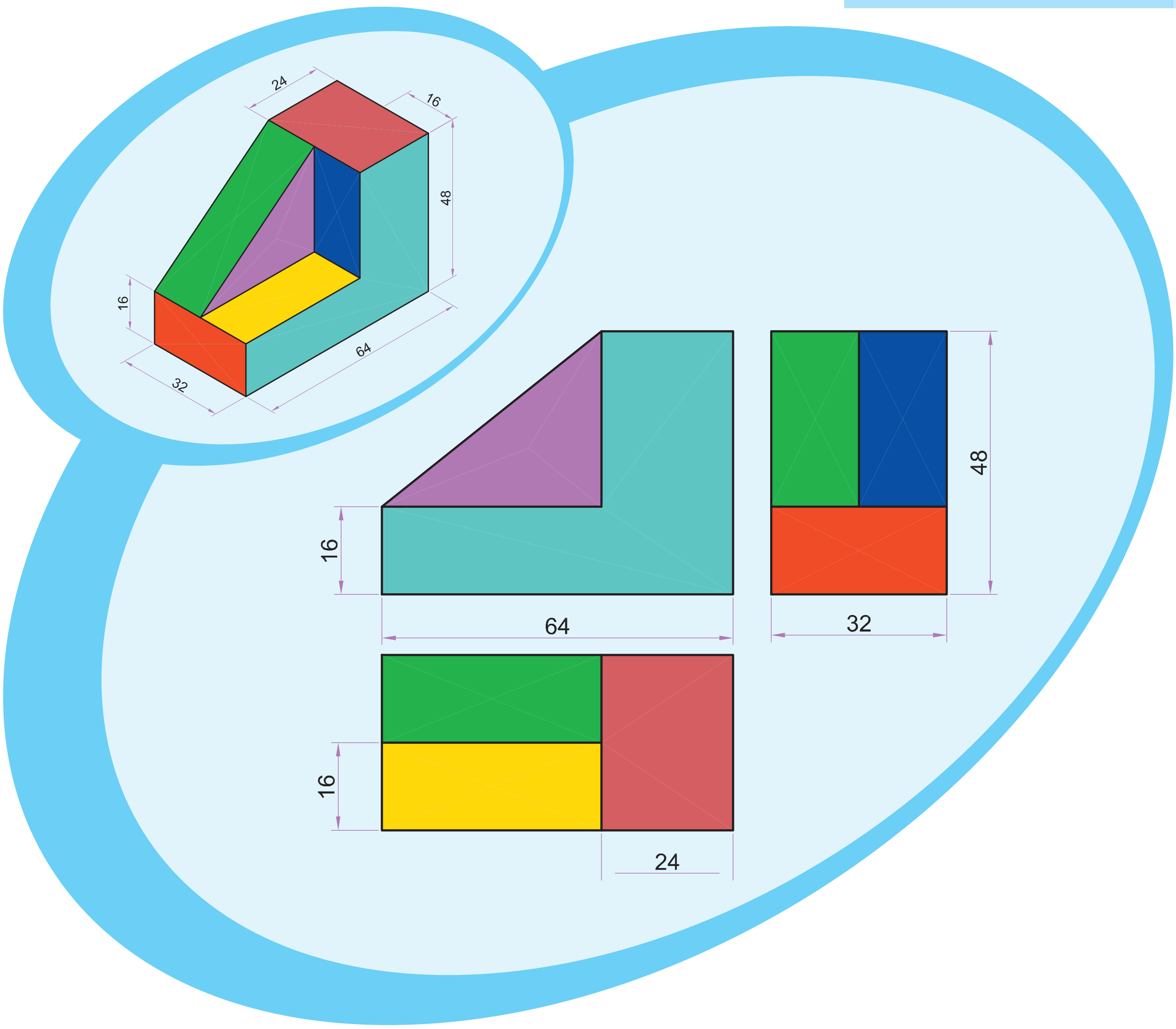
Exercise (23)



Exercise (24)



ORTHOGRAPHIC PROJECTIONS



ORTHOGRAPHIC PROJECTIONS

10

10.1 INTRODUCTION

Any object has three dimensions, viz., length, width and thickness. A projection is defined as a representation of an object on a two dimensional plane. The projections of an object should convey all the three dimensions, along with other details of the object on a sheet of paper. The elements to be considered while obtaining a projection are :

(*i*) The object

(*ii*) The plane of projection

(*iii*) The point of sight

(*iv*) The rays of sight

A projection may be obtained by viewing the object from the point of sight and tracing in correct sequence, the points of intersection between the rays of sight and the plane on to which the object is projected. A projection is called orthographic projection when the point of sight is imagined to be located at infinity so that the rays of sight are parallel to each other and intersect the plane of projection at right angle to it.

The principles of orthographic projection may be followed in four different angles or systems, viz., first, second, third and fourth angle projections. A projection is said to be first, second, third or fourth angle when the object is imagined to be in the first, second, third or fourth quadrant respectively. Throughout this book, first angle projection is followed.

10.2 PRINCIPLE OF FIRST ANGLE PROJECTION

In first angle projection, the object is imagined to be positioned in the first quadrant. The view from the front of the object is obtained by looking at the object from the right side of the quadrant and tracing in correct sequence, the points of intersection between the projection plane and the rays of sight extended. The object is between the observer and the plane of projection (vertical plane). Here, the object is imagined to be transparent and the projection lines are extended from various points of the object to intersect the projection plane. Hence, in first angle projection, any view is so placed that it represents the side of the object away from it.

10.3 MATHODS OF OBTAINING ORTHOGRAPHIC VIEWS

10.3.1 View From the Front

The view from the front of an object is defined as the view that is obtained as projection on the vertical plane by looking at the object normal to its front surface. It is the usual practice to position the object such that its view from the front reveals most of the important features. Figure 10.1 shows the method of obtaining the view from the front of an object.

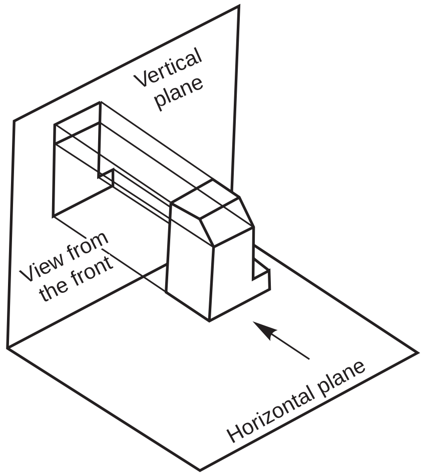


Fig. 10.1 Principle of obtaining the view from the front

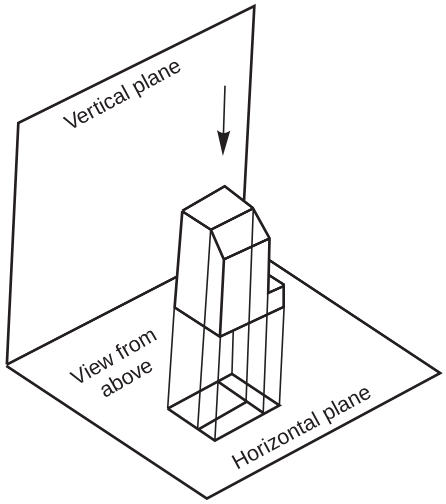


Fig. 10.2 Principle of obtaining the view from above

10.3.2 View From Above

The view from above of an object is defined as the view that is obtained as projection on the horizontal plane, by looking the object normal to its top surface. Figure 3.2 shows the method of obtaining the view from above of an object.

10.3.3 View From the Side

The view from the side of an object is defined as the view that is obtained as projection on the profile plane by looking the object, normal to its side surface. As there are two sides for an object, viz., left side and right side, two possible views from the side, viz., view from the left and view from the right may be obtained for any object. Figure 3.3 shows the method of obtaining the view from the left of an object.

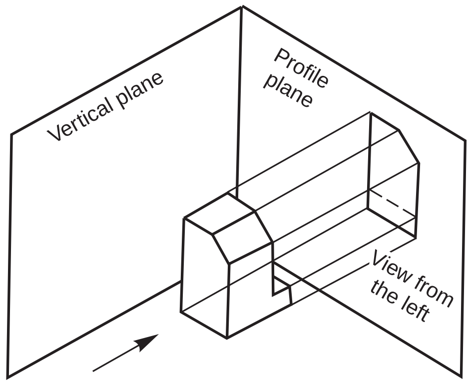


Fig. 10.3 Principle of obtaining the view from the left

10.4 PRESENTATION OF VIEWS

The different views of an object are placed on a drawing sheet which is a two dimensional one, to reveal all the three dimensions of the object. For this, the horizontal and profile planes are rotated till they coincide with the vertical plane. Figure 10.4 shows the relative positions of the views, viz., the view from the front, above and the left of an object.

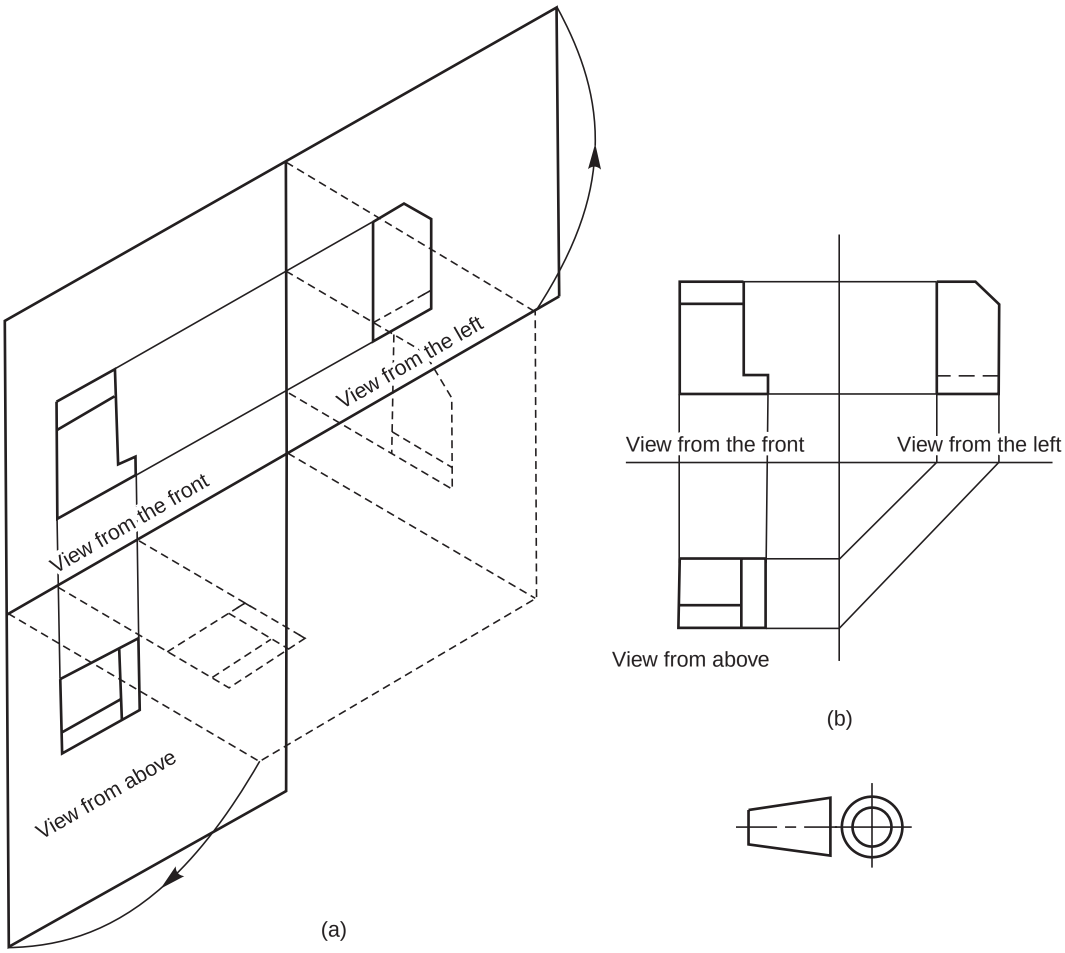


Fig. 10.4 Relative positions of the three views and the symbol

10.5 DESIGNATION AND RELATIVE POSITIONS OF VIEWS

An object positioned in space may be imagined as surrounded by six mutually perpendicular planes. So, for any object, six different views may be obtained by viewing at it along the six directions, normal to these planes. Figure 10.5 shows an object with six possible directions to obtain the different views which are designated as follows:

1. View in the direction **a** = view from the front
2. View in the direction **b** = view from above
3. View in the direction **c** = view from the left

4. View in the direction **d** = view from the right

5. View in the direction **e** = view from below

6. View in the direction **f** = view from the rear

Figure 3.6 *a* shows the relative positions of the above six views in the first angle projection and Fig.10.6b ,the distinguishing symbol of this method of projection. Figure 10.7 *a* shows the relative position of the views in the third angle projection and Fig.10.7*b*, the distinguishing symbol of this method of projection.

NOTE A comparison of Figs. 10.6 and 10.7 reveals that in both the methods of projection, the views are identical in shape and detail. Only their location with respect to the view from the front is different.

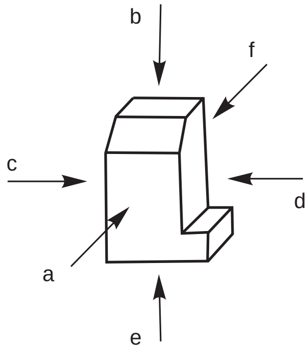
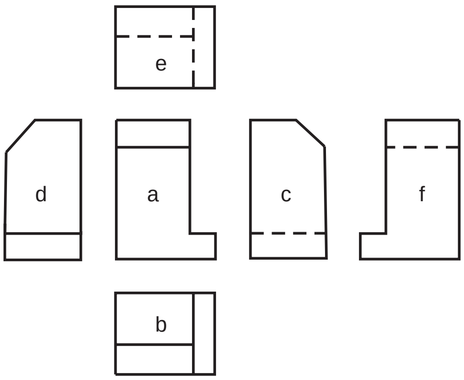
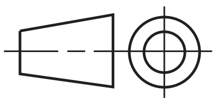


Fig.10.5 Designation of the views

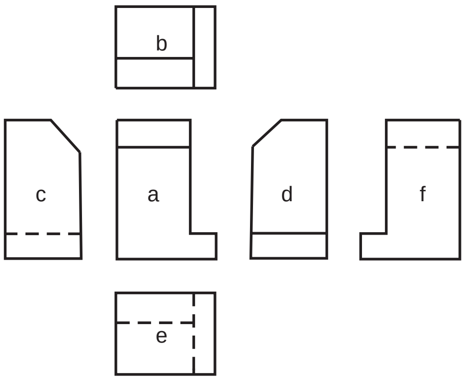


(a)

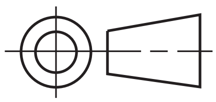


(b)

Fig. 10.6 Relative positions of six views in first angle projection



(a)



(b)

Fig. 10.7 Relative positions of six views in third angle projection

10.6 POSITION OF THE OBJECT

It is important to understand the significance of the position of the object relative to the planes of projection. To get useful information about the object in the orthographic projections, the object may be imagined to be positioned properly because of the following facts :

1. Any line on an object will show its true length, only when it is parallel to the plane of projection.

2. Any surface of an object will appear in its true shape, only when it is parallel to the plane of projection.

In the light of the above, it is necessary that the object is imagined to be positioned such that its principal surfaces are parallel to the planes of projection.

10.6.1 Hidden Lines

While obtaining the projection of an object on to any principal plane of projection, certain features of the object may not be visible. The invisible or hidden features are represented by short dashes of medium thickness. Figure 10.8 shows the application of hidden lines in the projection of an object.

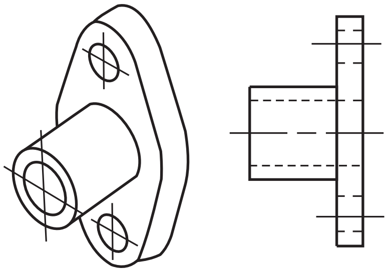


Fig. 10.8 Application of hidden lines

10.6.2 Curved Surfaces

Certain objects contain curved surfaces, tangential to other curved surfaces. The difficulty in representing the surfaces can be overcome if the following rule is observed. Wherever a tangential line drawn to the curved surface becomes a projector, a line should be drawn in the adjacent view. Figure 10.9 shows the representation of certain curved surfaces, tangential to other curved surfaces.

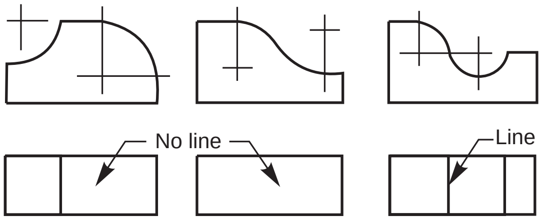


Fig. 10.9 Representation of tangential curved surfaces

Certain objects manufactured by casting technique, frequently contain corners filleted and the edges rounded. When the radius of a rounded corner is greater than 3 mm and the angle between the surfaces is more than 90°, no line is shown in the adjacent view. Figure 10.10 shows the application of the above principle.

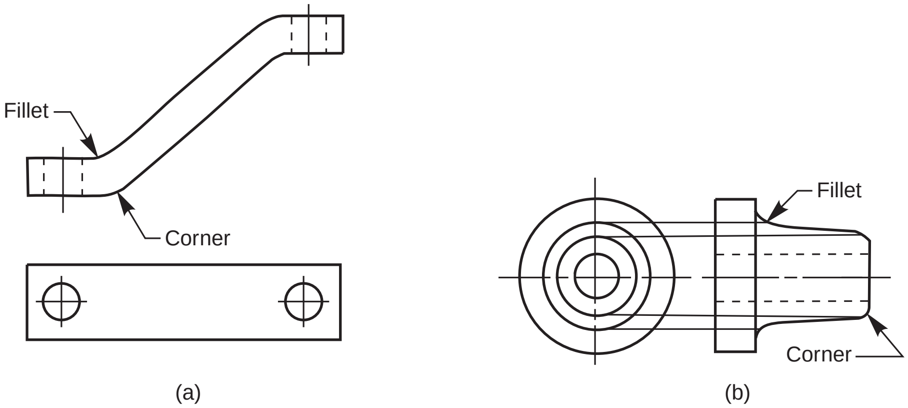


Fig. 10.10 Representation of corners and fillets

If true projection is followed in drawing the view of an object containing fillets and rounds; it will result in misleading impression. In conventional practice, fillets and rounds are represented by lines called runouts. The runouts are terminated at the point of tangency (Fig.10.11).

10.7 SELECTION OF VIEWS

For describing any object completely through its orthographic projections, it is important to select a number of views. The number of views required to describe any object will depend upon the extent of complexity involved in it. The higher the symmetry, the lesser the number of views required.

10.7.1 One - View Drawings

Some objects with cylindrical, square or hexagonal features or, plates of any size with any number of features in it may be represented by a single view. In such cases, the diameter of the cylinder, the side of the square, the side of the hexagon or the thickness of the plate may be expressed by a note or abbreviation. Square sections are indicated by light crossed diagonal lines. Figure 10.12 shows some objects which may be described by one-view drawings.

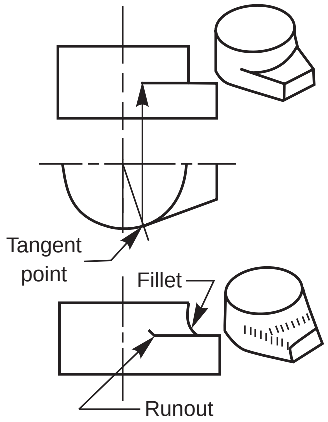


Fig. 10.11 Runouts

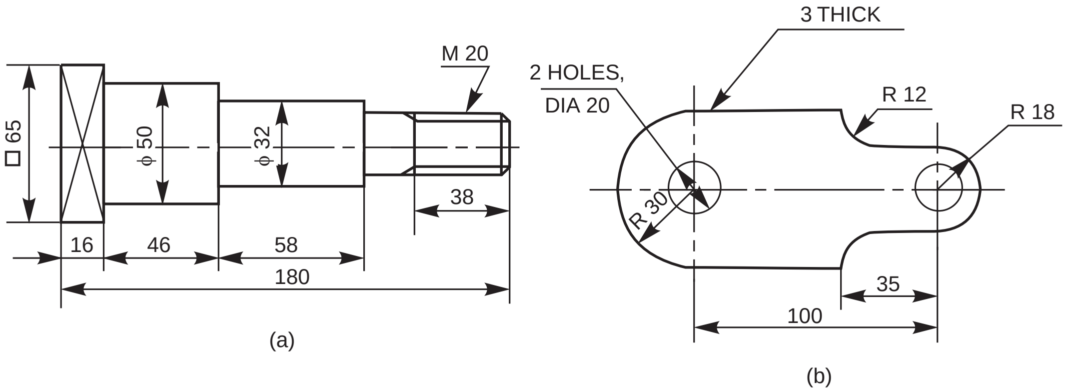


Fig. 10.12 One view drawings

10.7.2 Two - View Drawings

Some objects which are symmetrical about two axes may be represented completely by two views Normally, the largest face showing most of the details of the object is selected for drawing the view from the front. The shape of the object then determines whether the second view can be a view from above or a side view. Figure 10.13 shows the example of two-view drawings.

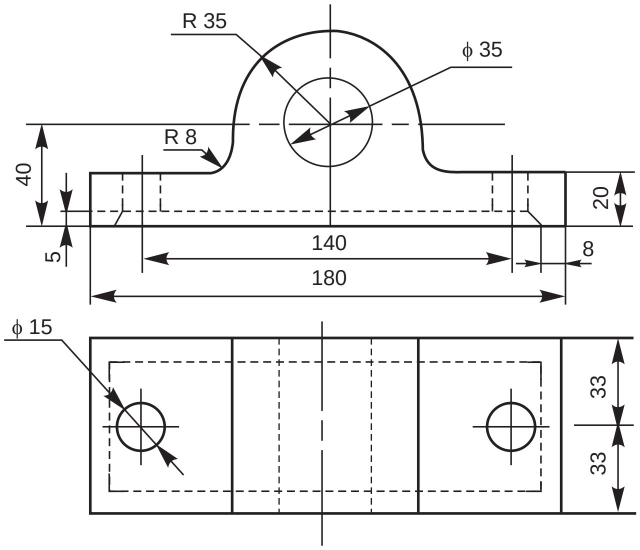
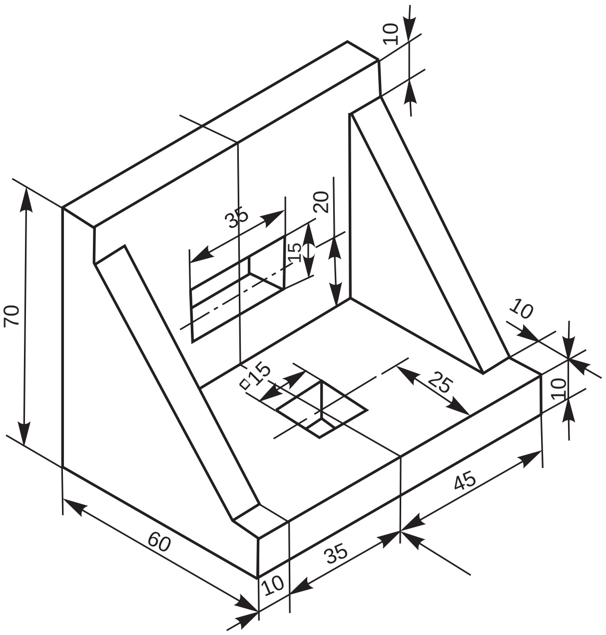


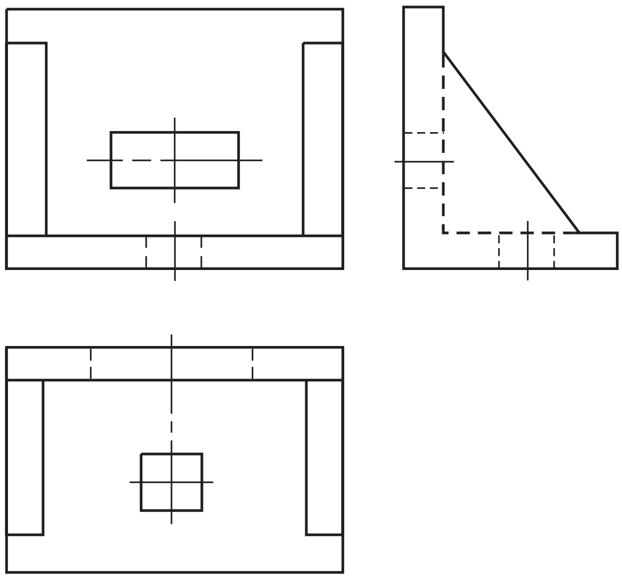
Fig. 10.13 Two view drawing

10.7.3 Three - View Drawings

In general, most of the objects consisting of either a single component or an assembly of a number of components, are described with the help of three views. In such cases, the views normally selected are the views from the front, above and left or right side. Figure 10.14 shows an object and its three necessary views.



(a)



(b)

Fig. 10.14 Three view drawing

10.8 DEVELOPMENT OF MISSING VIEWS

When two views of an object are given, the third view may be developed by the use of a mitre line.

10.8.1 To construct the view from the left , from the two given views

Construction (fig 10.15)

1. Draw the views from the front and above.

2. Draw the projection lines to the right of the view from above.

3. Decide the distance, D from the view from the front at which, the side view is to be drawn.

4. Construct a mitre line at 45°.

5. From the points of intersection between the mitre line and the projection lines, draw vertical projection lines.

6. Draw the horizontal projection lines from the view from the front to intersect the above lines. The figure obtained by joining the points of intersection in the order is the required view.

Figure 10.16 shows the steps to be followed in constructing the view from above of an object, from the given views from the front and left.

NOTE These exercises are aimed at improving the practice in reading and developing the imagination of the student.

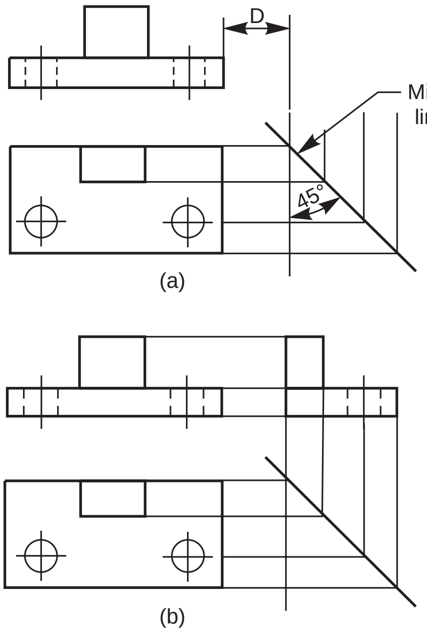


Fig.10.15 Construction of the view from the left

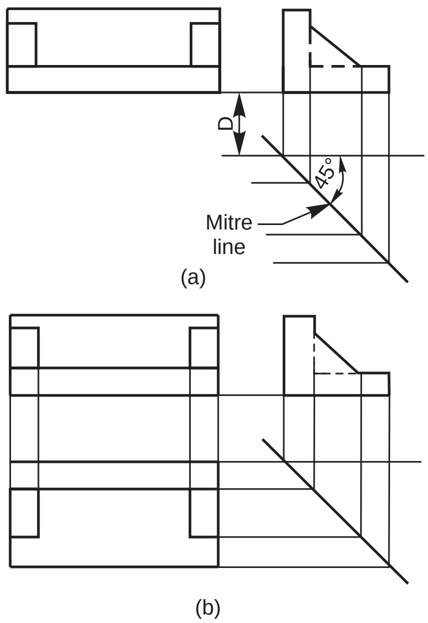


Fig. 10.16 Construction of the view from above

10.9 SPACING THE VIEWS

Most paper comes in standard sizes. The largest sheet you are likely to use is A0 and the smallest A4. If your drawing paper has no frame then draw one. A minimum of 20 mm is used on A0 and A1 from the edge of the paper to the frame line and a minimum of 10 mm on A2, A3 and A4.

In order to space out the views that you will draw on your paper use the following formulas (A, B and C are the maximum sizes of your views) and the p and q dimensions are the distances between the views.

You do not have to use exact dimensions which might complicate the sums; use sensible approximations for A, B and C (Fig. 17)

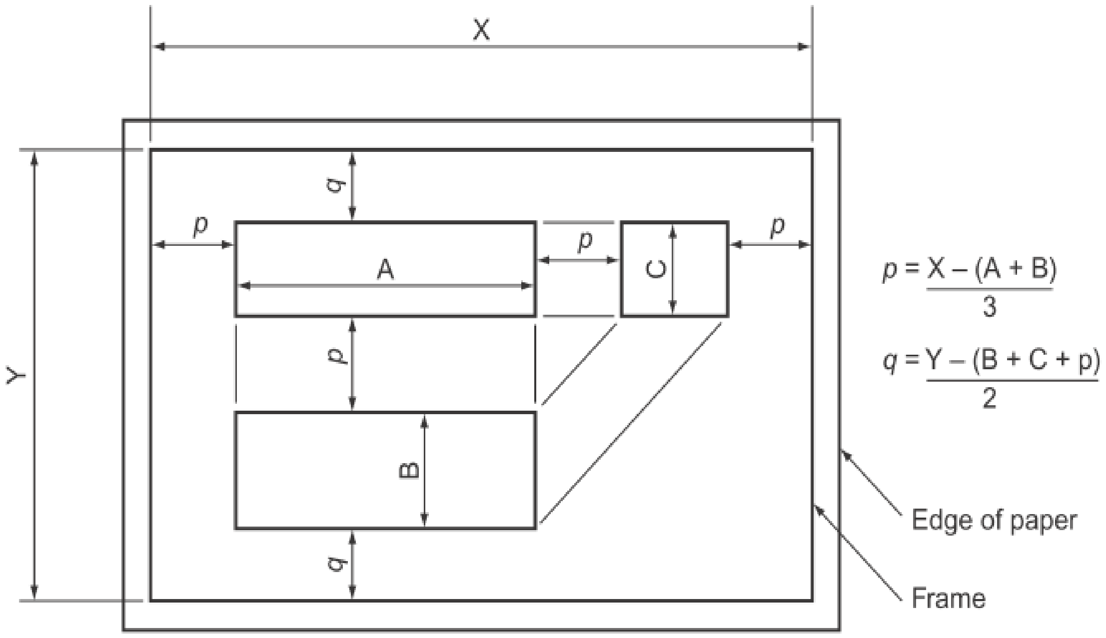
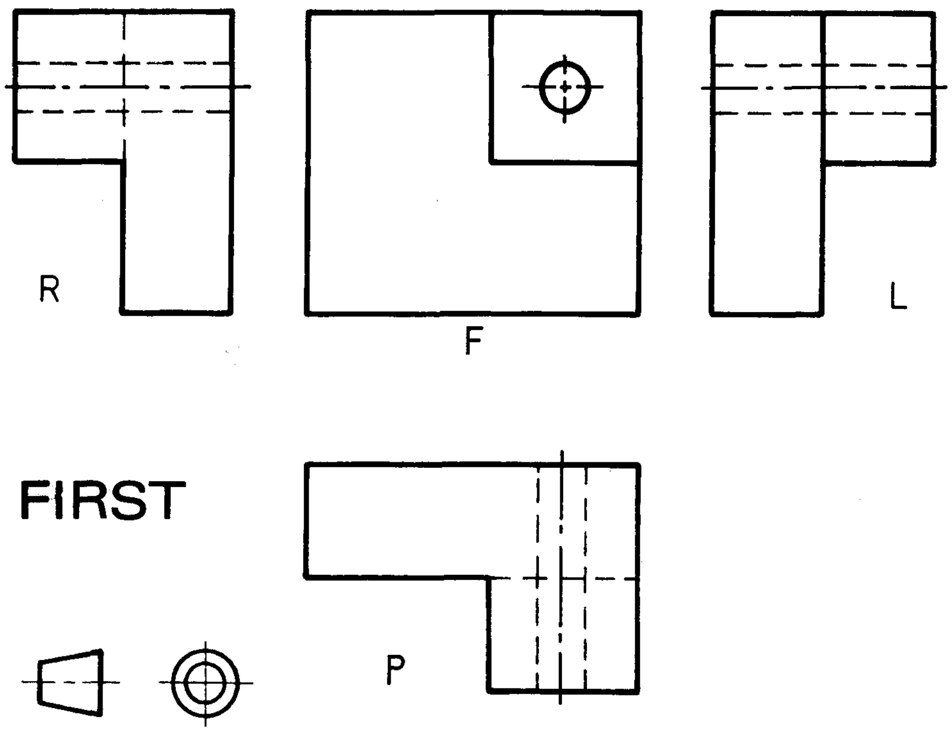
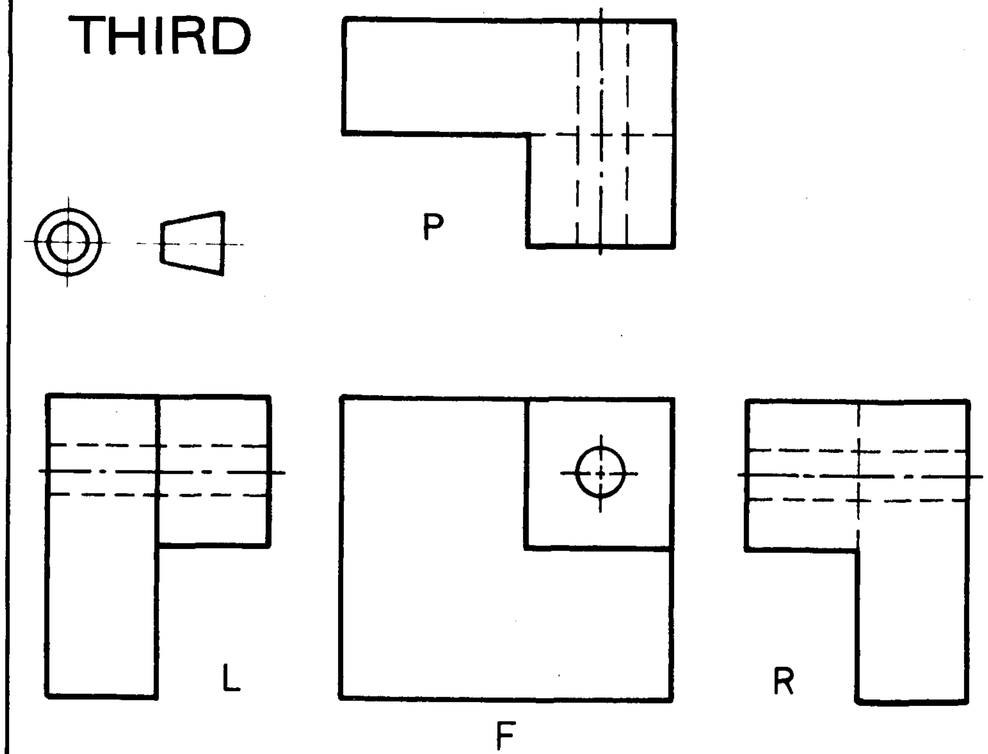


Figure (17) Positioning of views to be drawn.

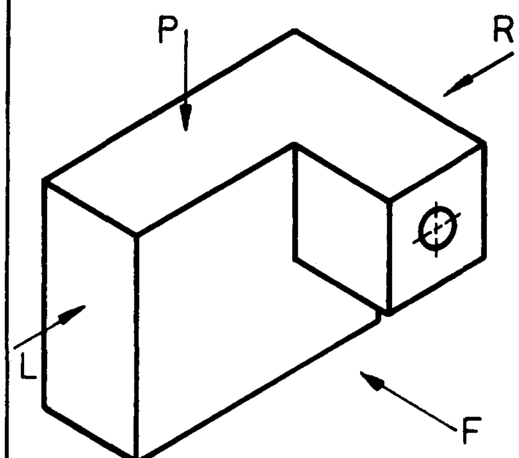
A comparison of first angle projection and third angle projection





NOTE:

(1) The plan is BELOW the front view.
(2) The right-hand side view is on the LEFT-hand side of the front view.
(3) The left-hand side view is on the RIGHT-hand side of the front view.

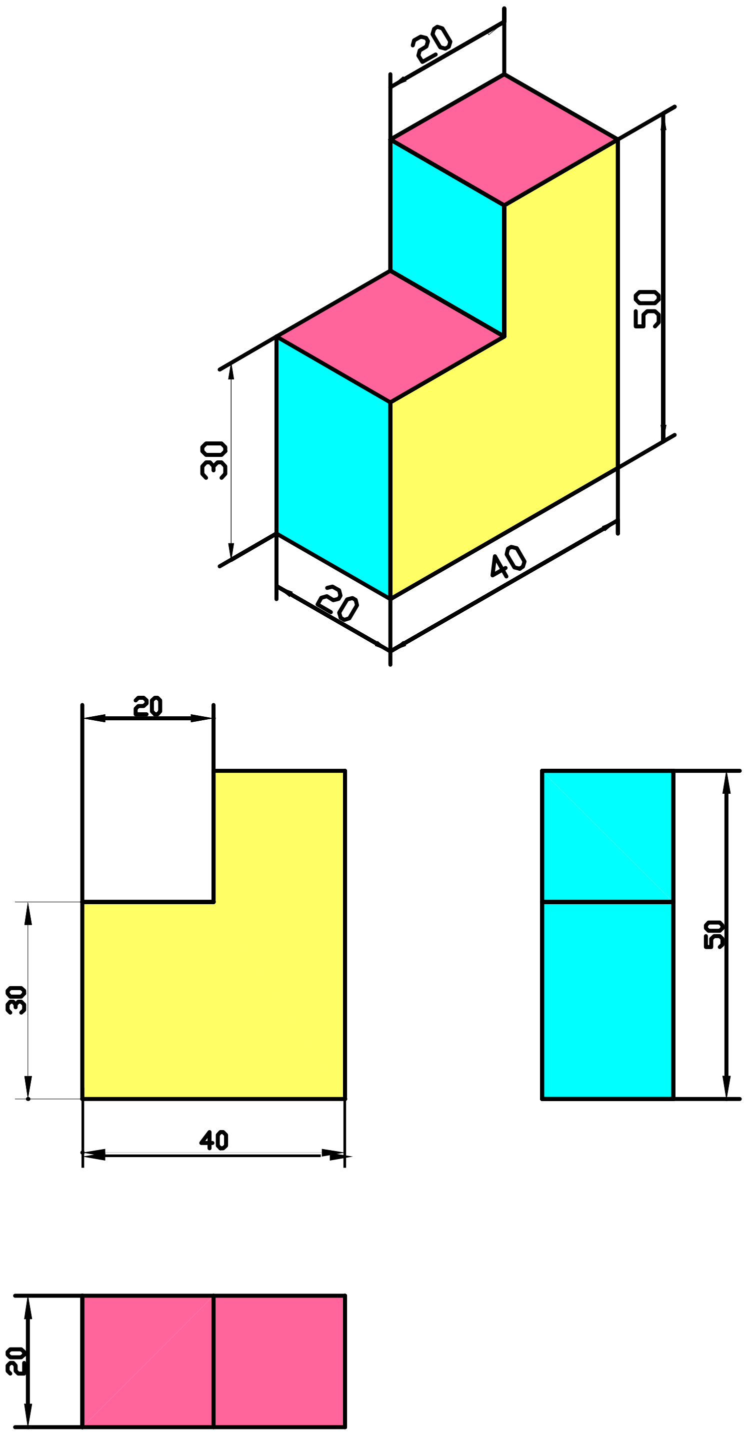


NOTE:

(1) The plan view is ABOVE the front view.
(2) The right-hand side view is on the RIGHT-hand side of the front view.
(3) The left-hand side view is on the LEFT-hand side of the front view.

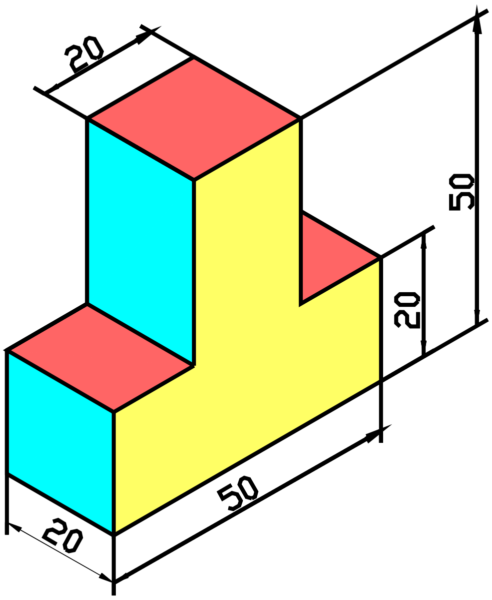
Example:1

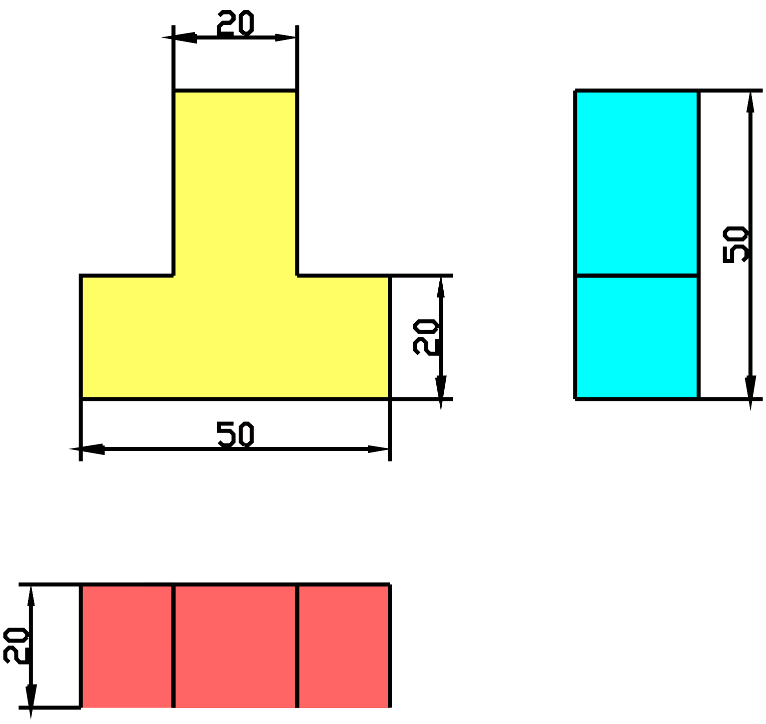
Use a AutoCAD package to create the view from the front ,the view from above and the view from the left from the given isometric view of a machine component.



Example:2

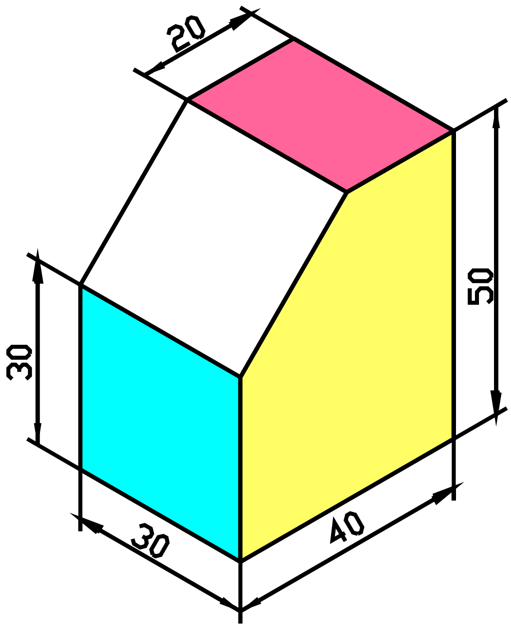
Use a AutoCAD package to create the view from the front ,the view from above and the view from the left from the given isometric view of a machine component.

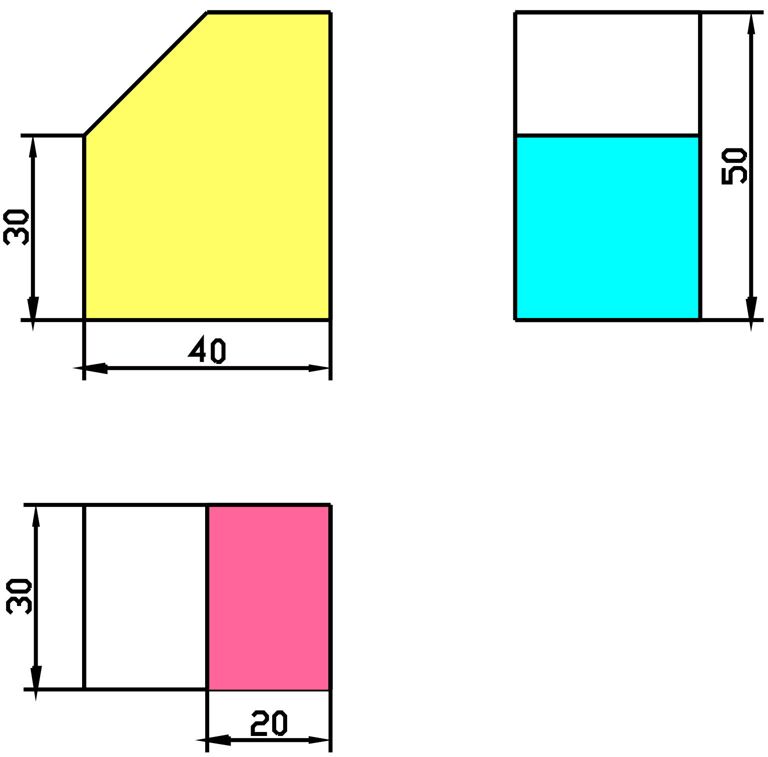




Example:3

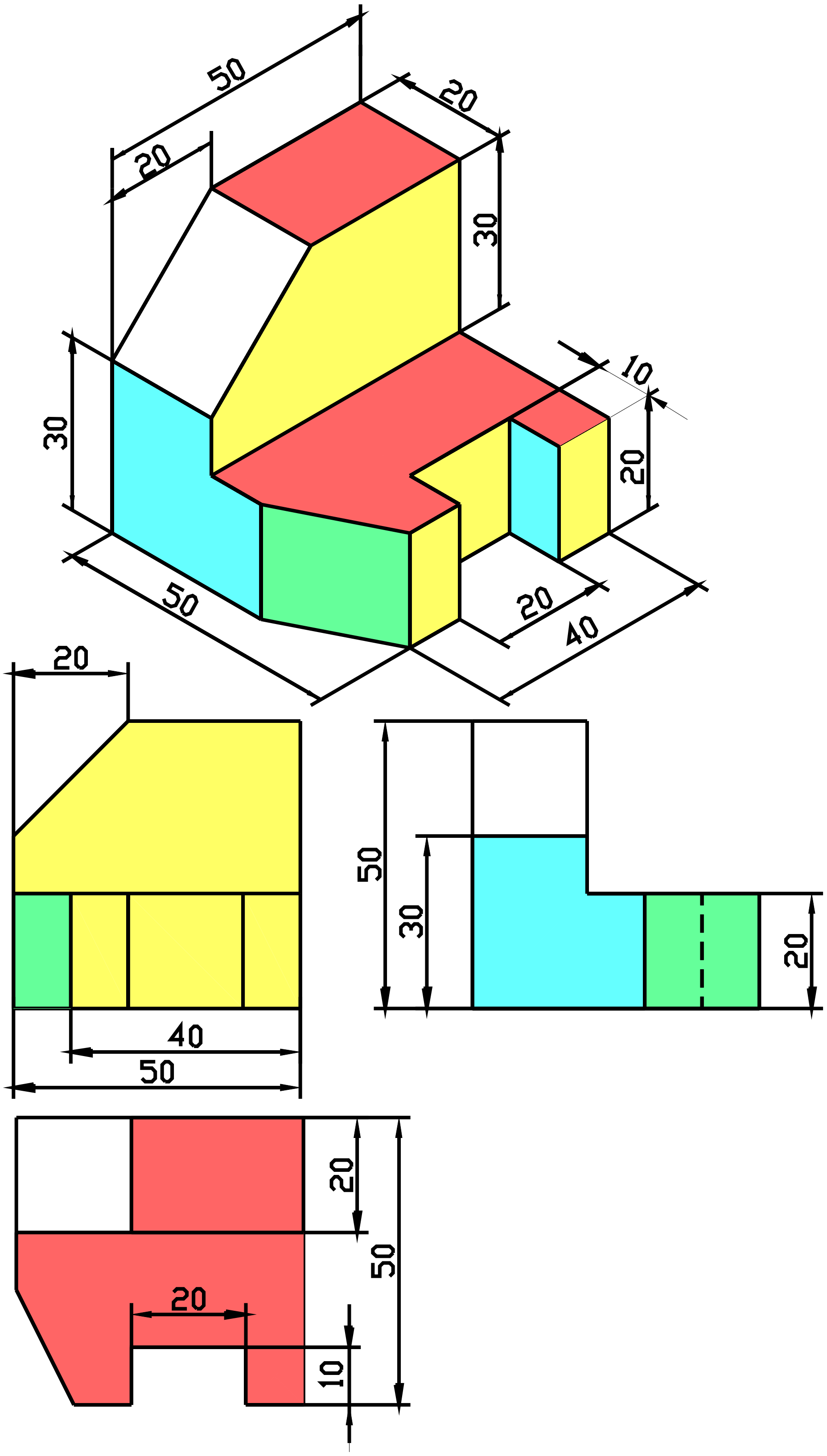
Use a AutoCAD package to create the view from the front ,the view from above and the view from the left from the given isometric view of a machine component.





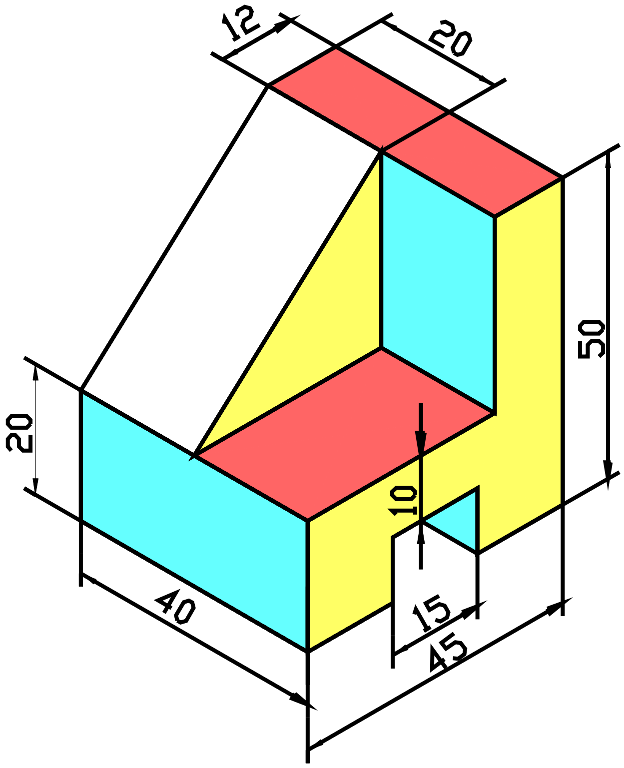
Example:4

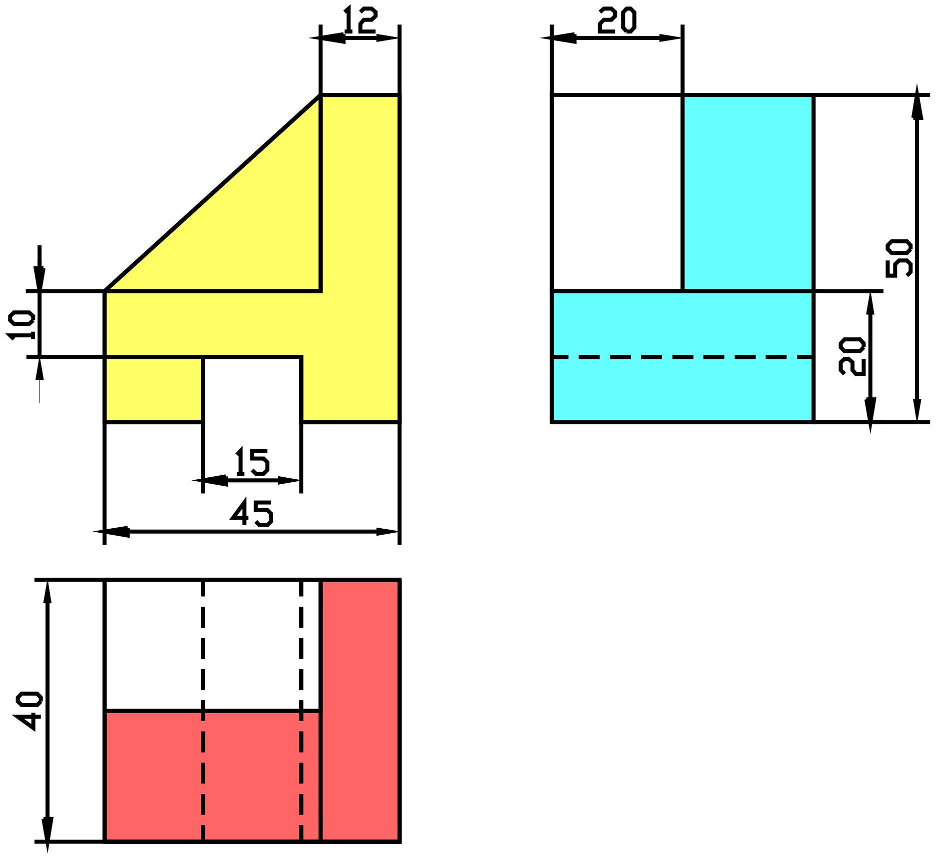
Use a AutoCAD package to create the view from the front ,the view from above and the view from the left from the given isometric view of a machine component.



Example:5

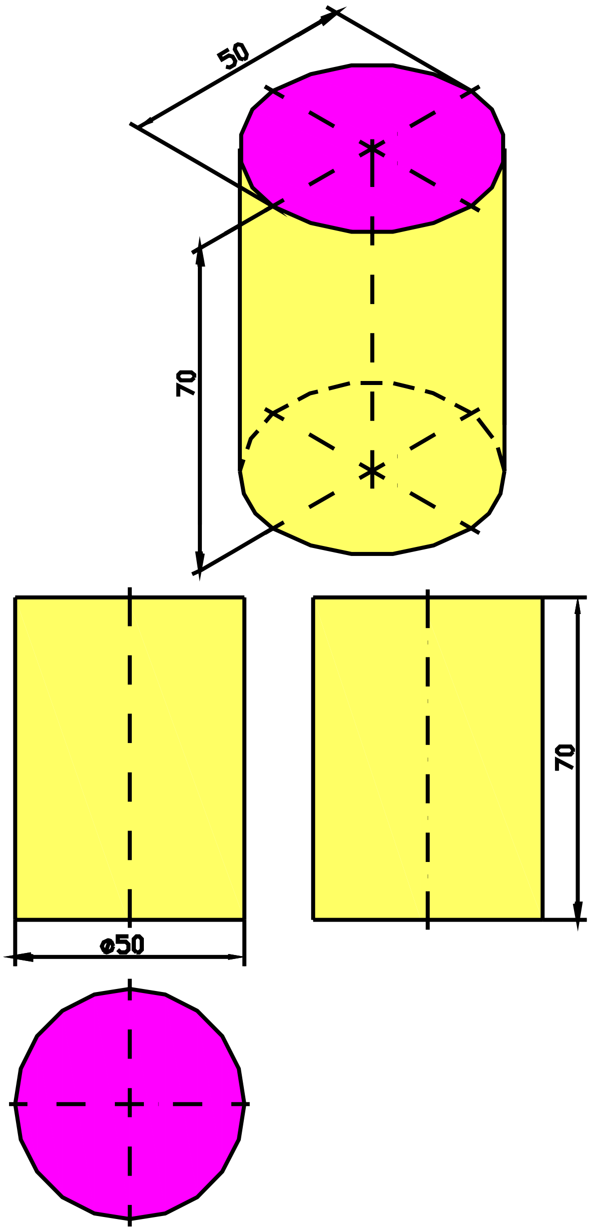
Use a AutoCAD package to create the view from the front ,the view from above and the view from the left from the given isometric view of a machine component.



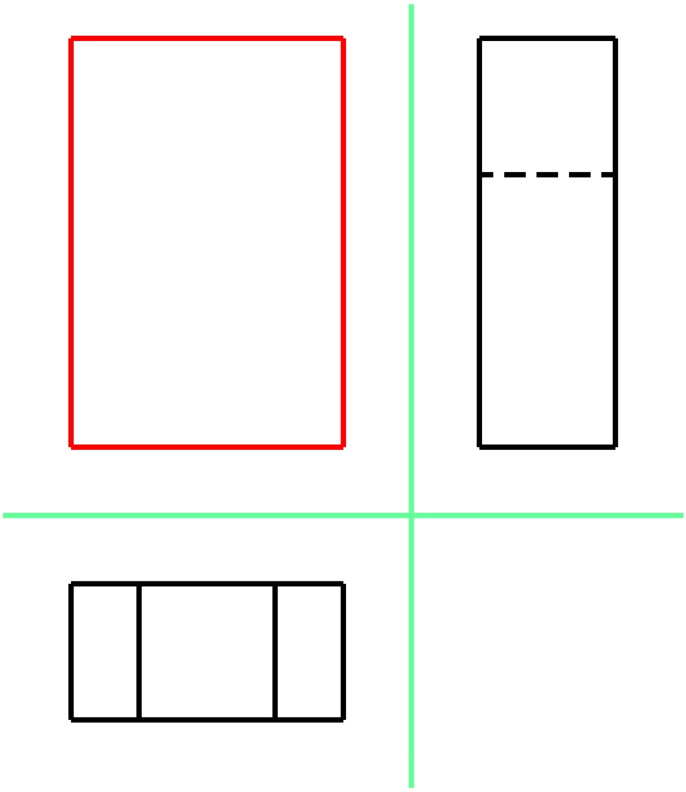


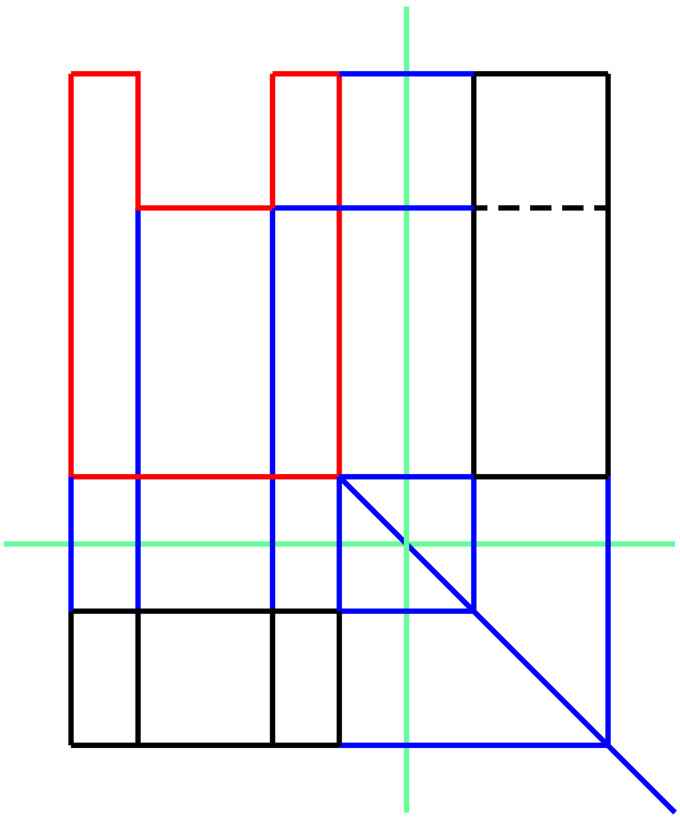
Example:6

Use a AutoCAD package to create the view from the front ,the view from above and the view from the left from the given isometric view of a machine component.

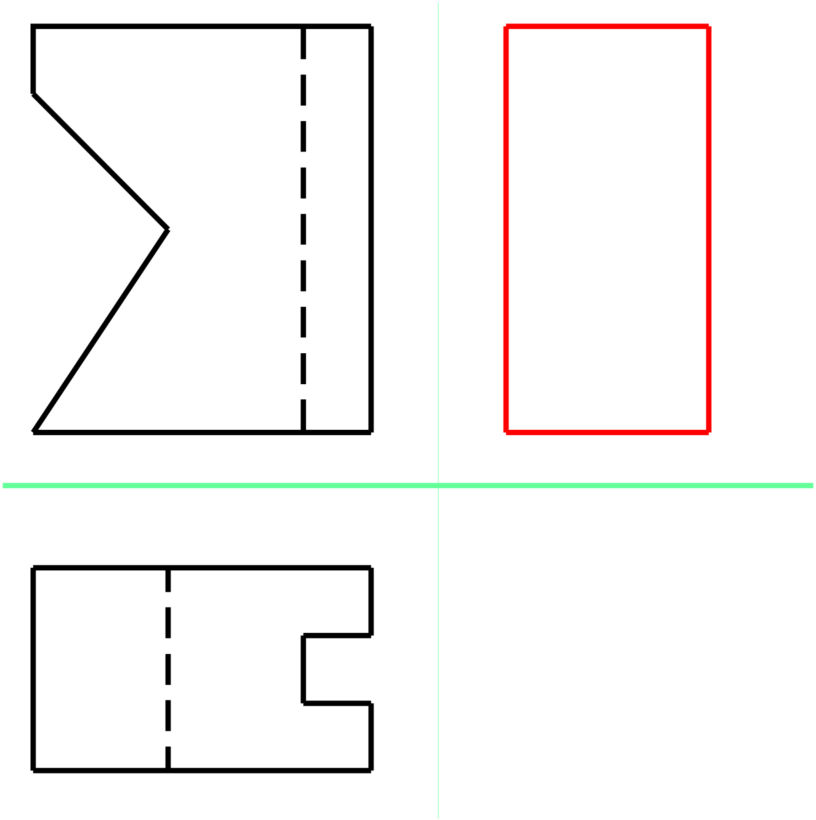


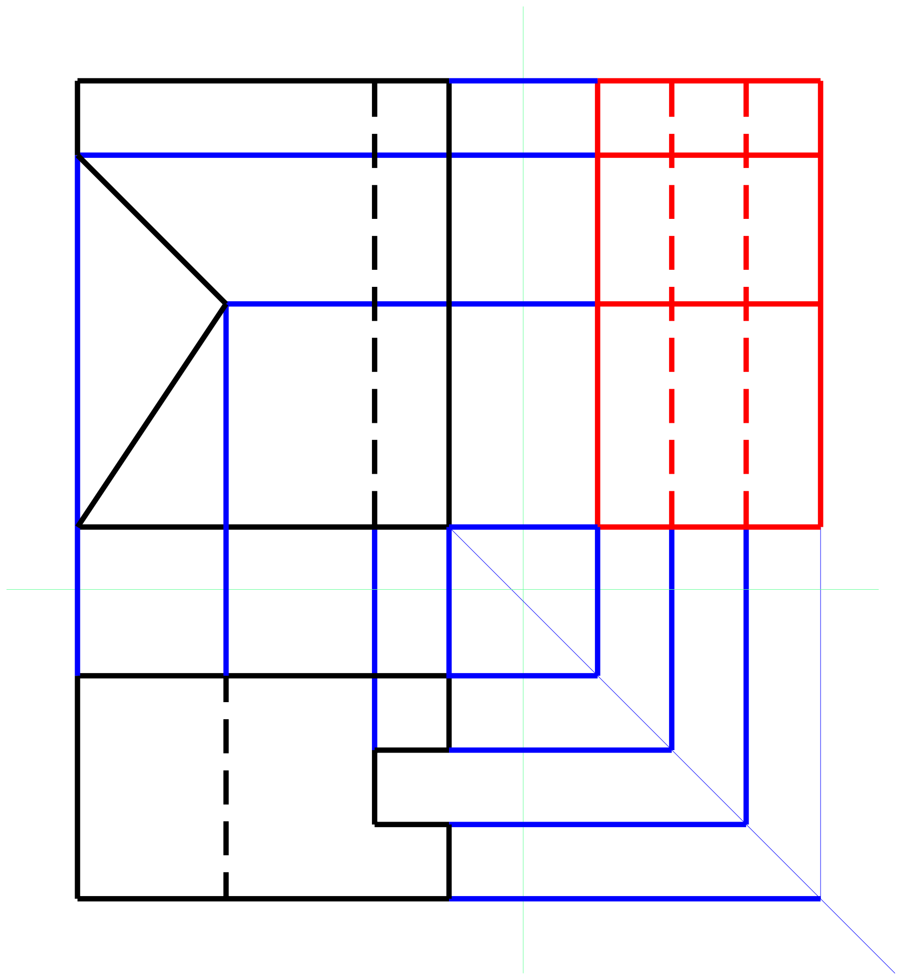
Example : 7   Given two complete views, sketch the missing view.



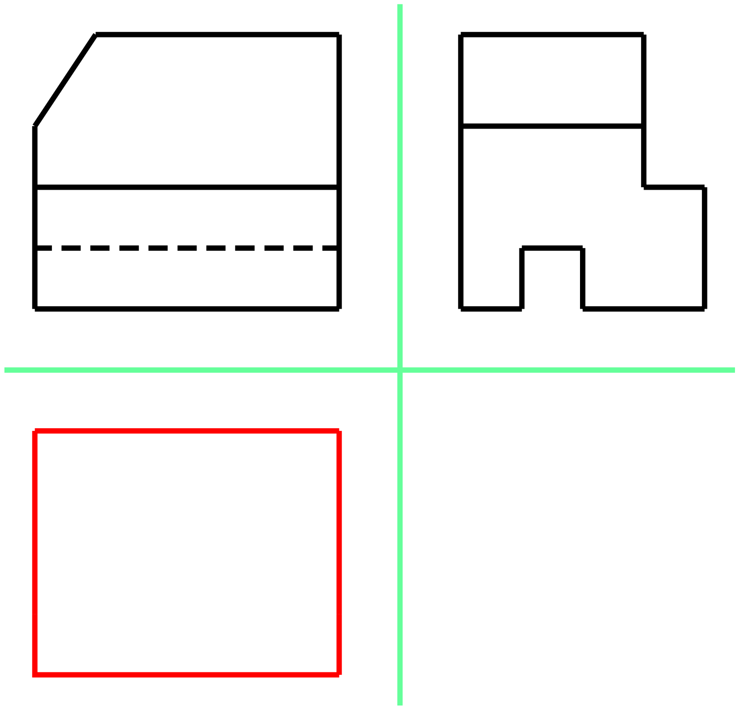


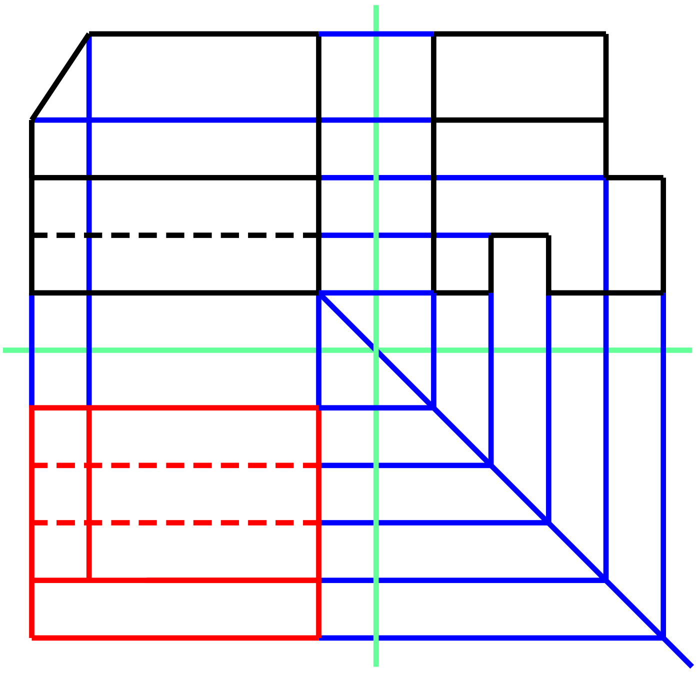
Example : 8   Given two complete views, sketch the missing view.





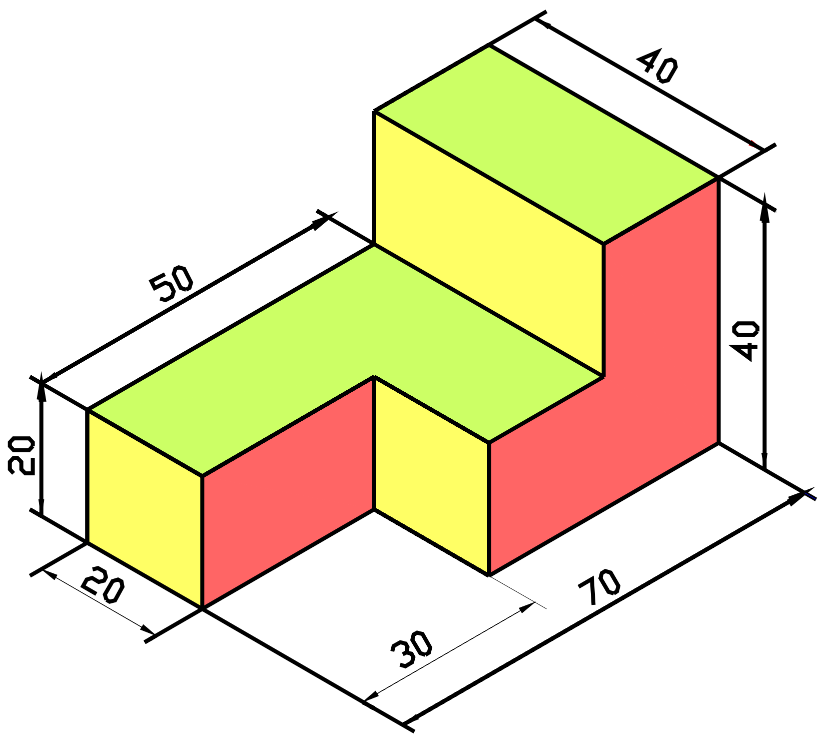
Example : 9   Given two complete views, sketch the missing view.



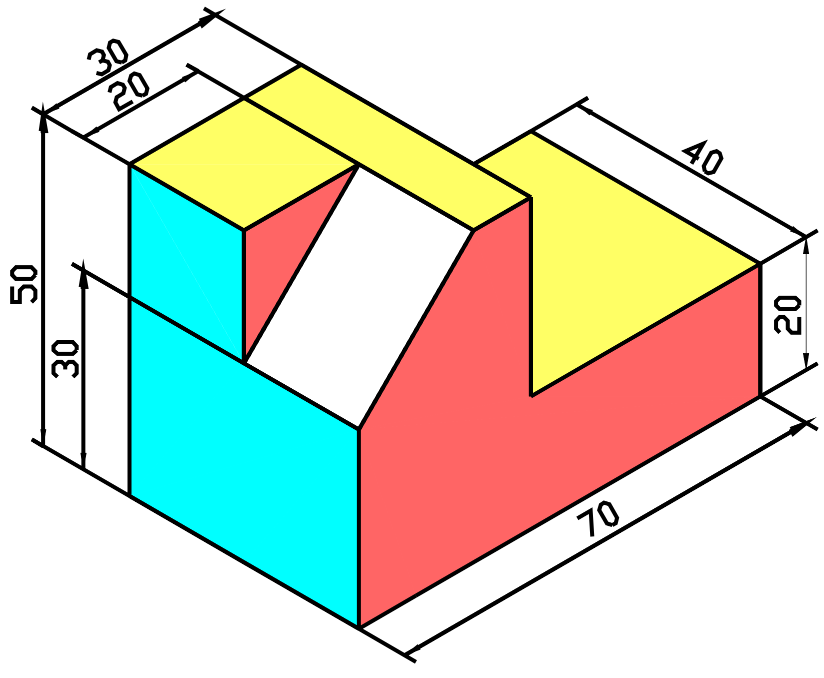


Exercise 1:

Use a AutoCAD package to create the view from the front ,the view from above and the view from the left from the given isometric view of a machine component.



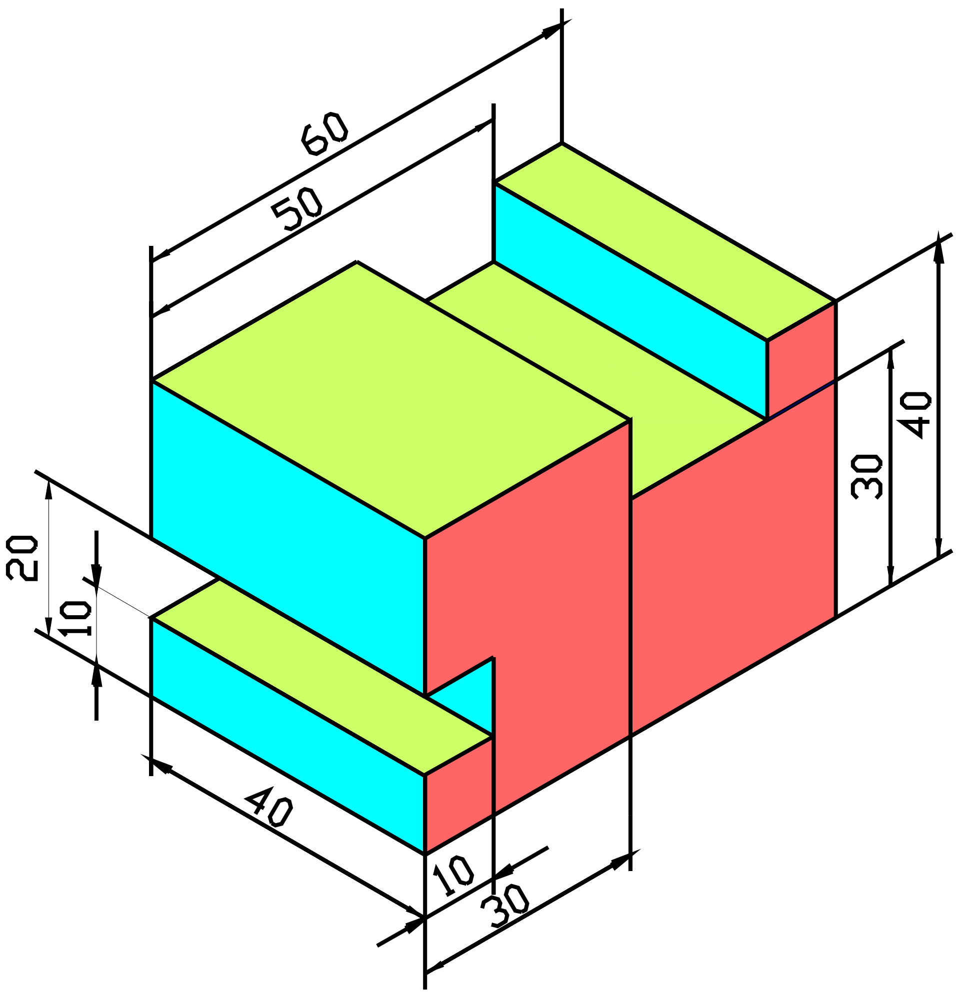
−1



−2

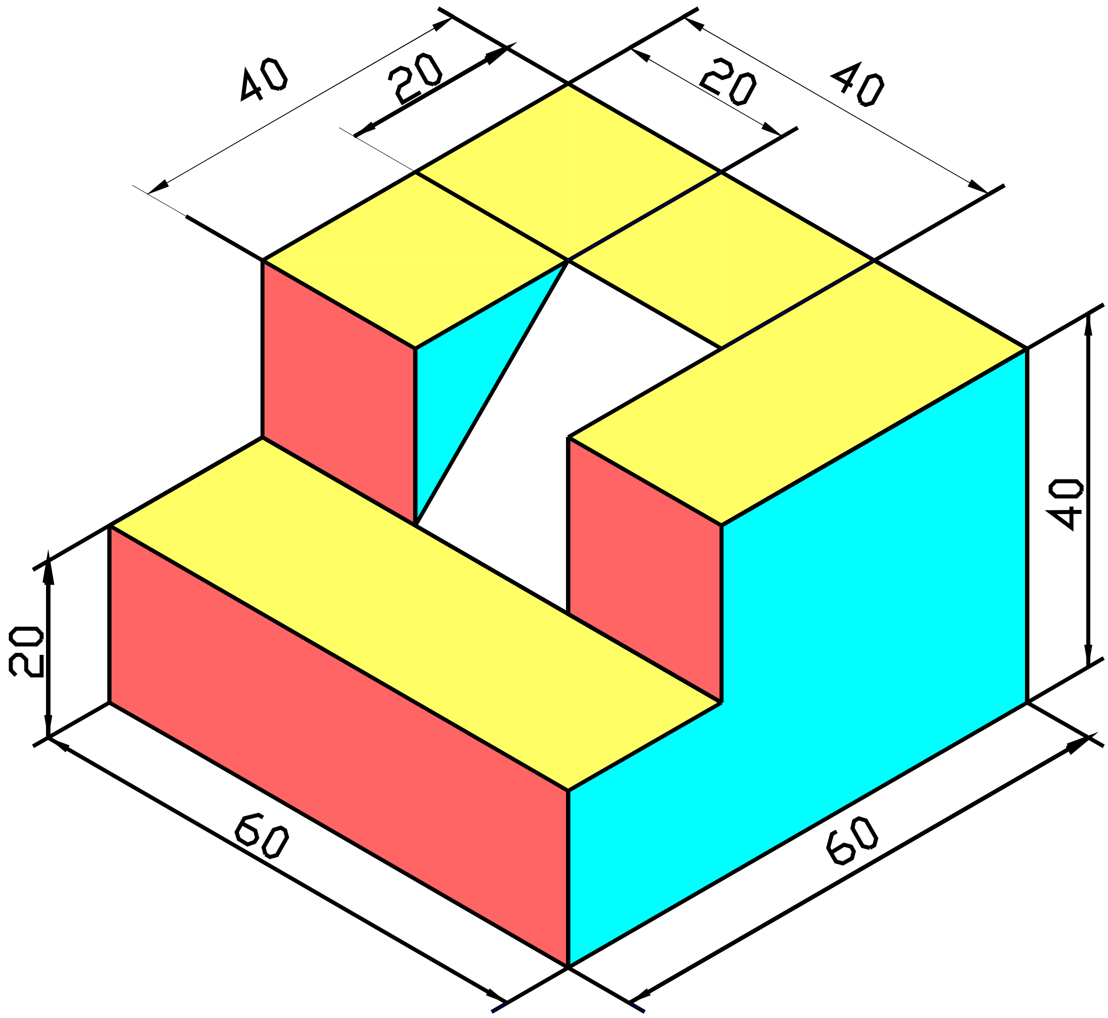
Exercise 2:

Use a AutoCAD package to create the view from the front ,the view from above and the view from the left from the given isometric view of a machine component.



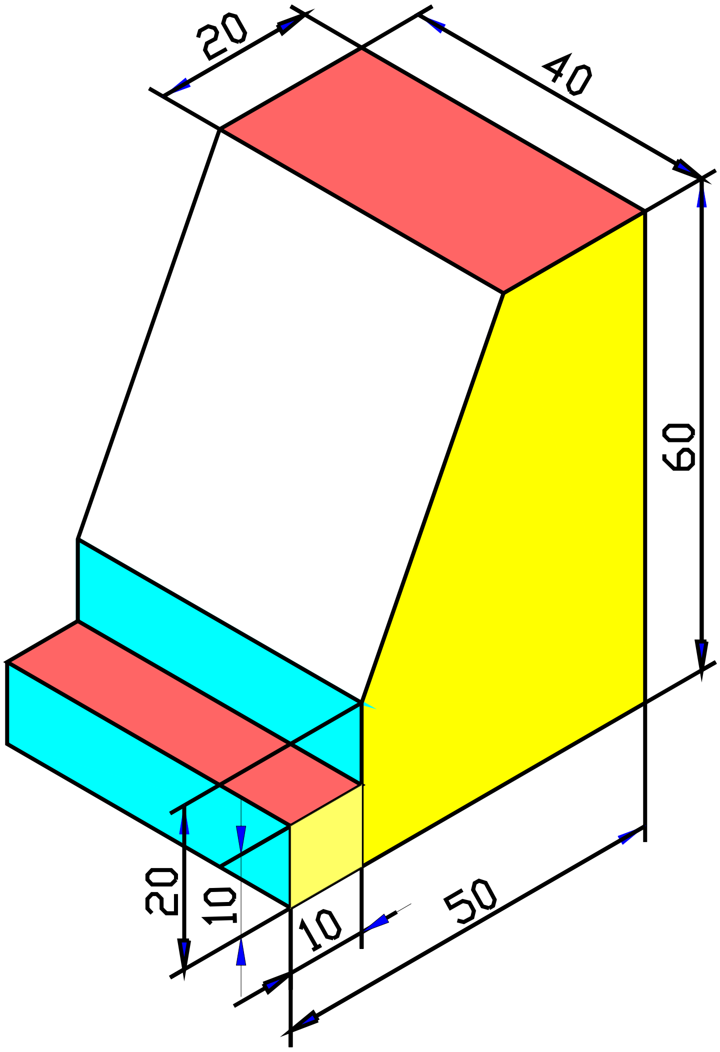
Exercise 3:

Use a AutoCAD package to create the view from the front ,the view from above and the view from the left from the given isometric view of a machine component.



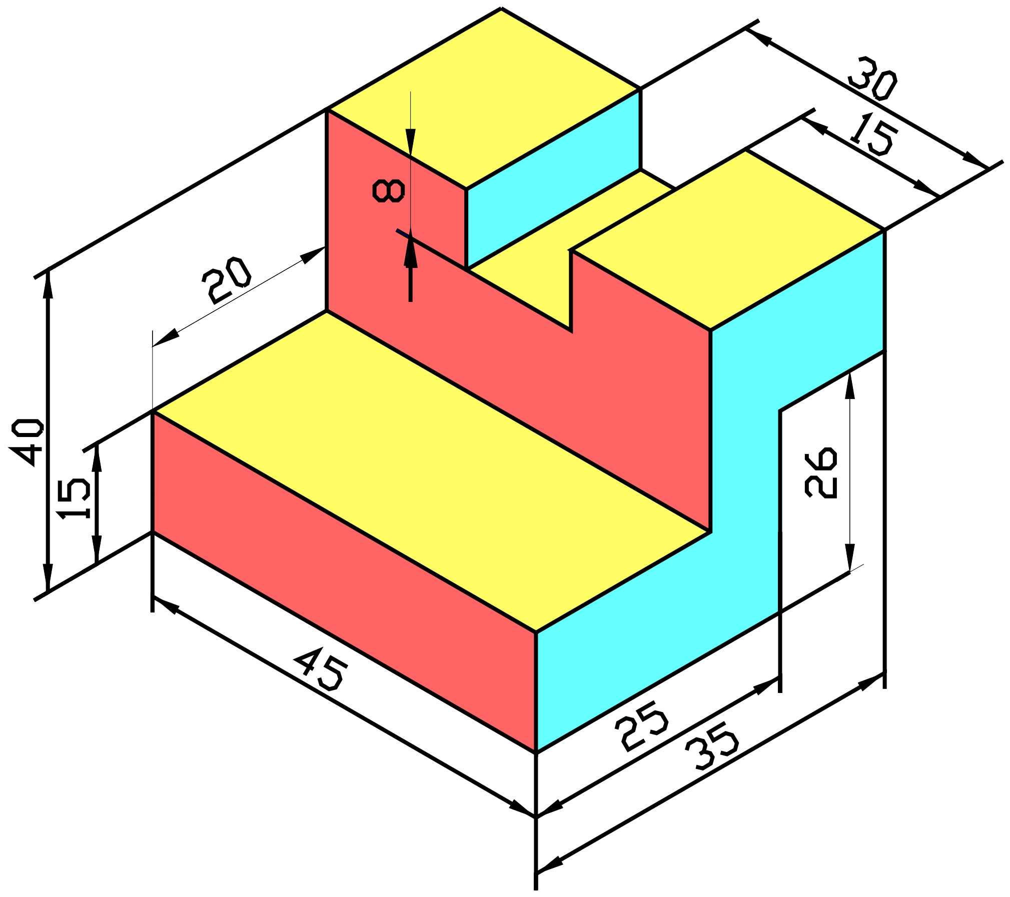
Exercise 4:

Use a AutoCAD package to create the view from the front ,the view from above and the view from the left from the given isometric view of a machine component.



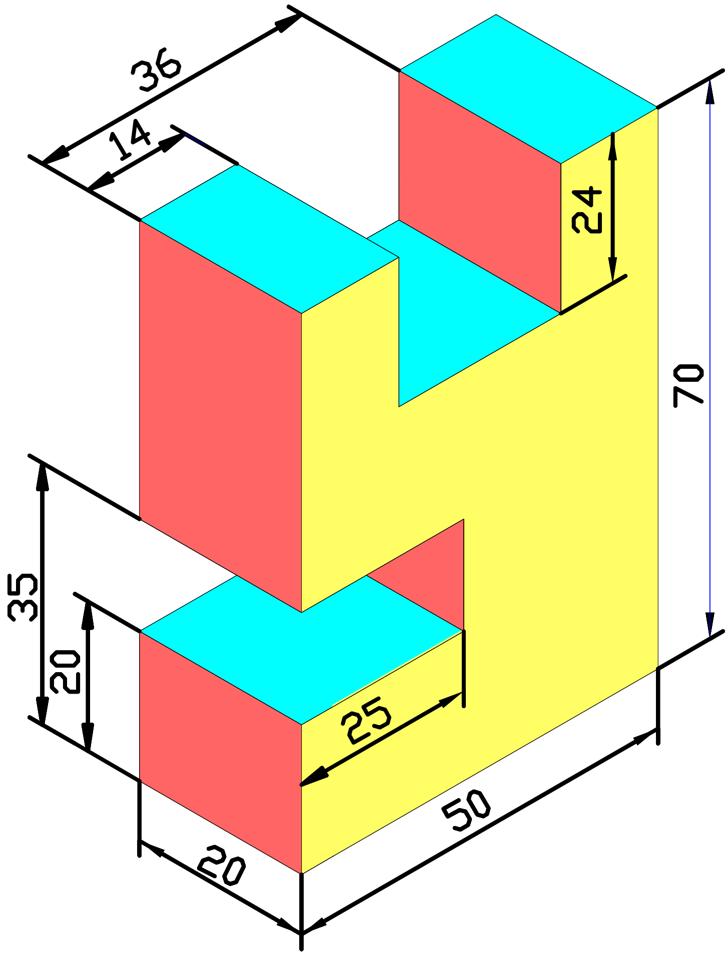
Exercise 5:

Use a AutoCAD package to create the view from the front ,the view from above and the view from the left from the given isometric view of a machine component.



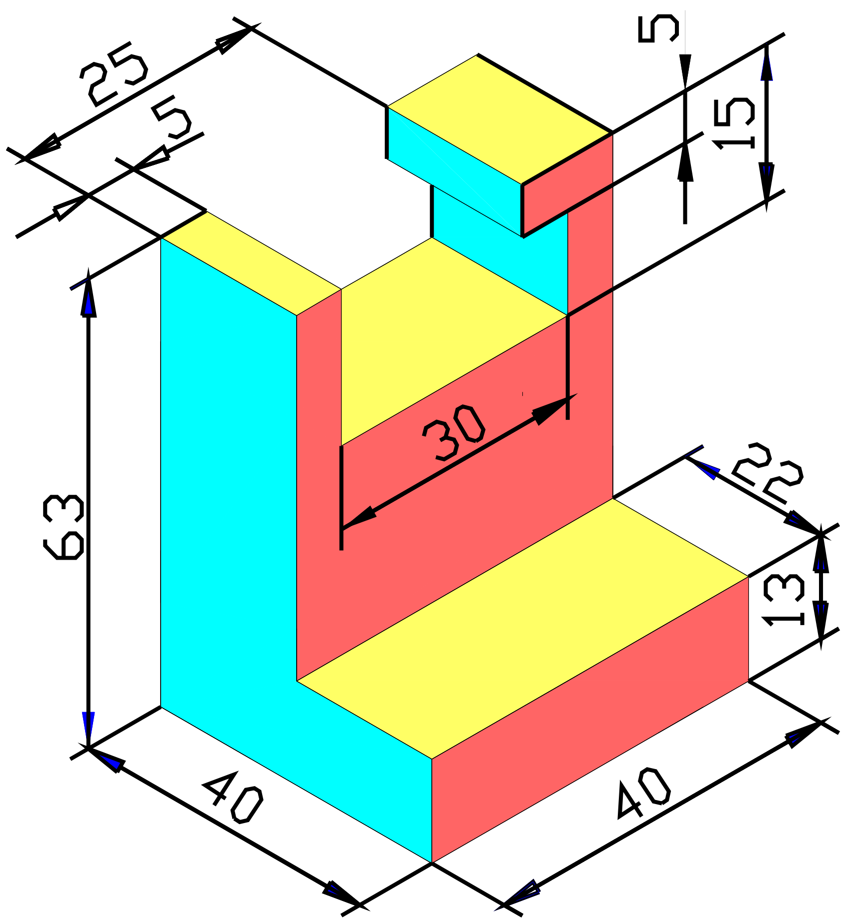
Exercise 6 :

Use a AutoCAD package to create the view from the front ,the view from above and the view from the left from the given isometric view of a machine component.



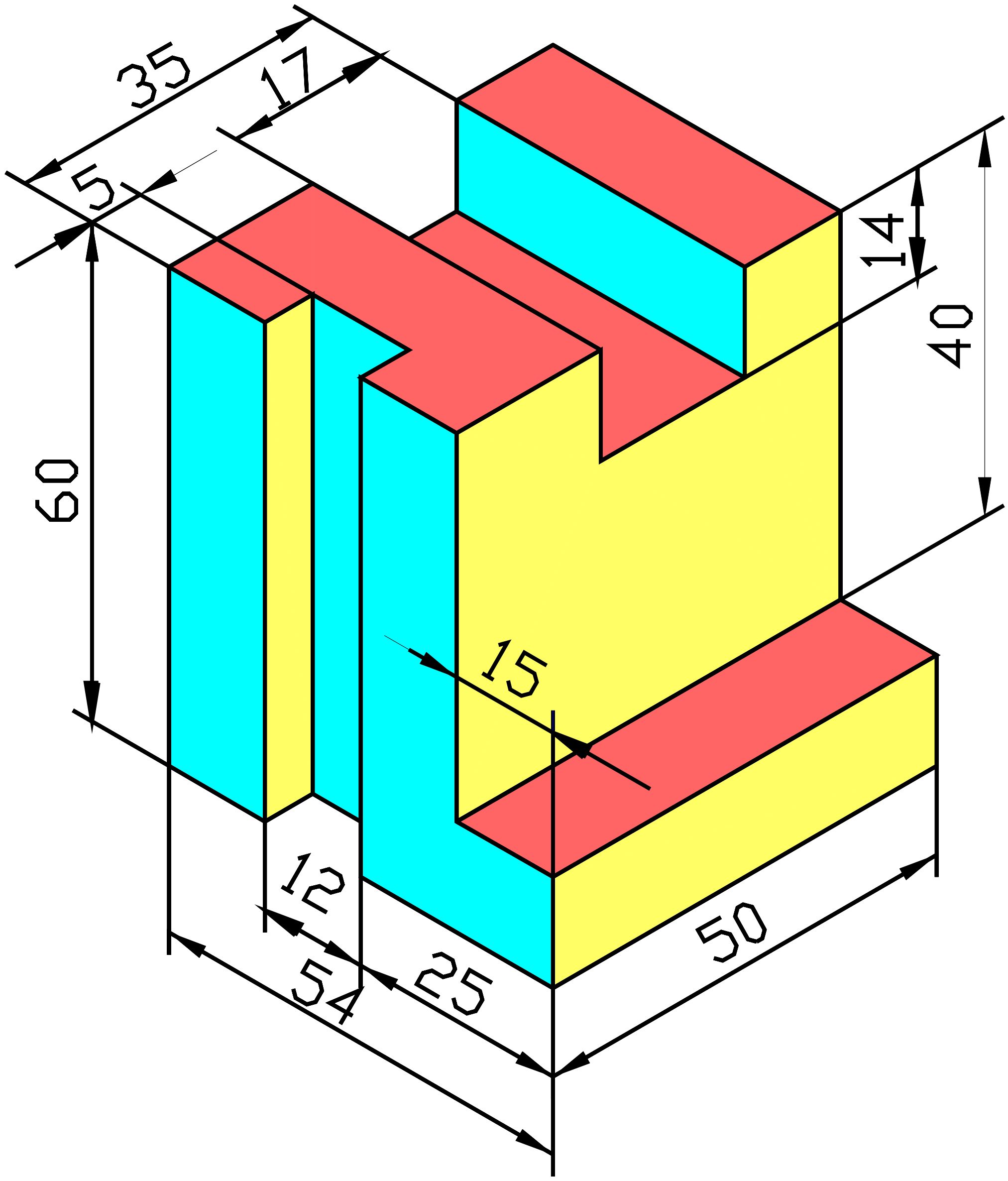
Exercise 7 :

Use a AutoCAD package to create the view from the front ,the view from above and the view from the left from the given isometric view of a machine component.



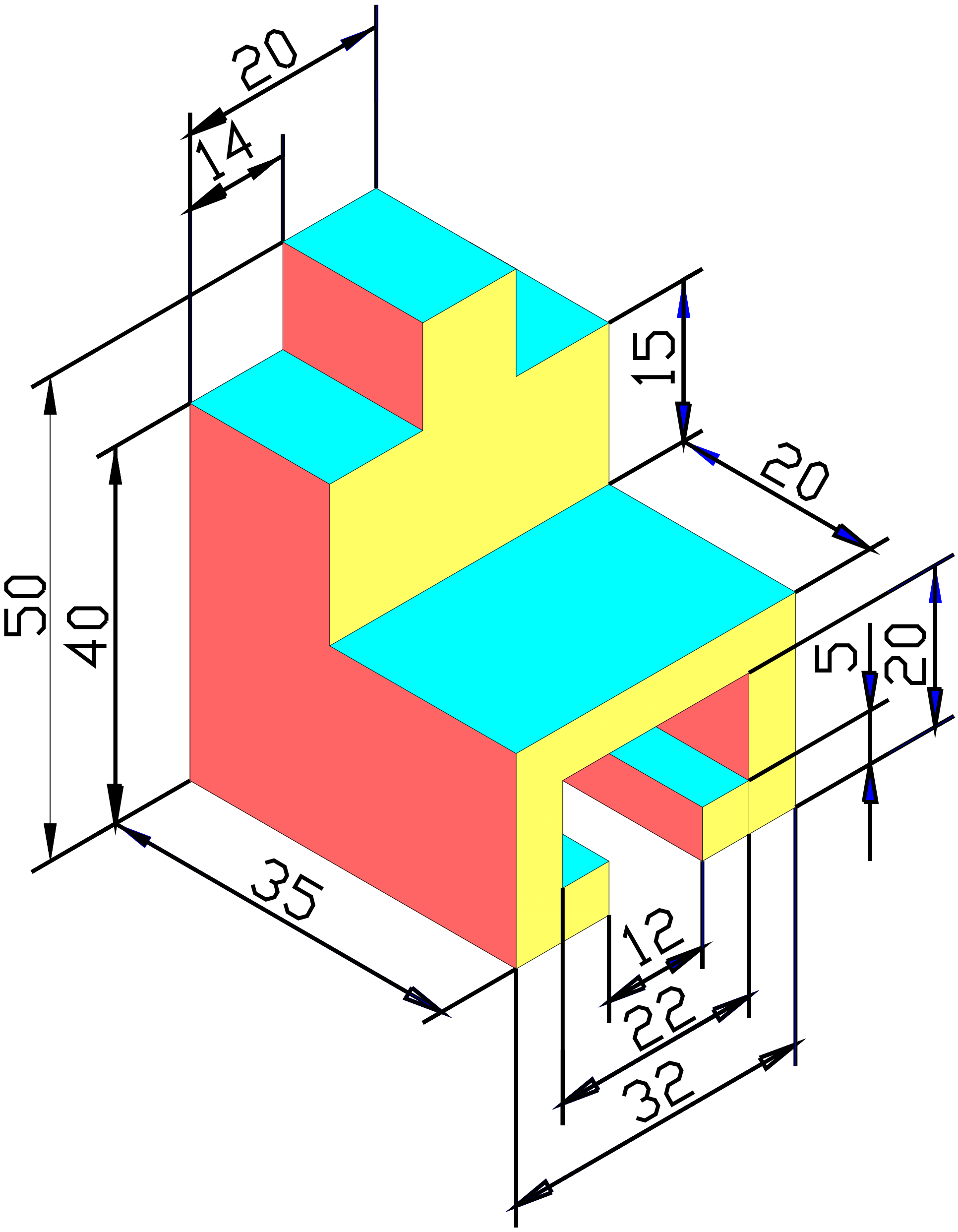
Exercise 8 :

Use a AutoCAD package to create the view from the front ,the view from above and the view from the left from the given isometric view of a machine component.



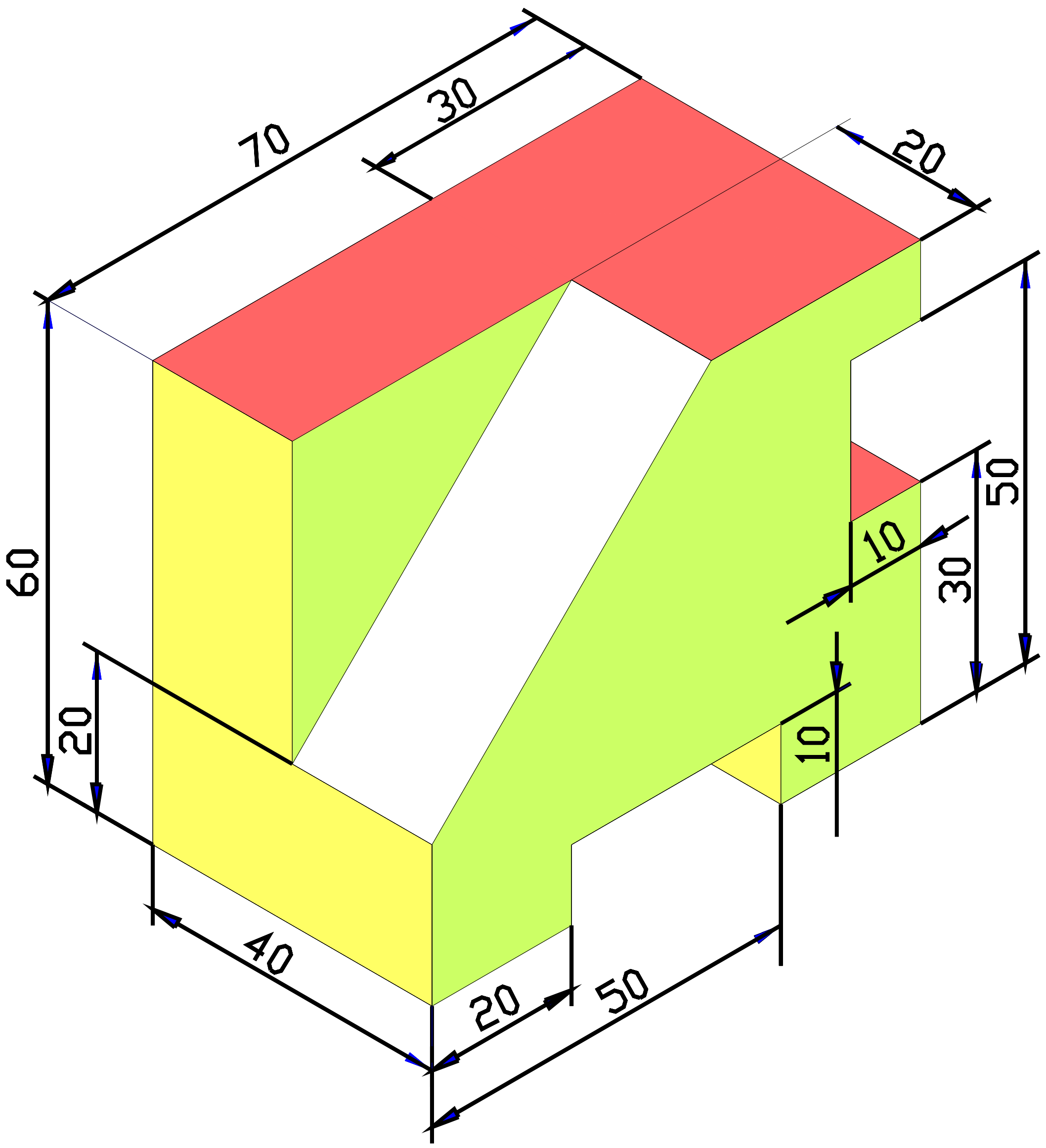
Exercise 9 :

Use a AutoCAD package to create the view from the front ,the view from above and the view from the left from the given isometric view of a machine component.



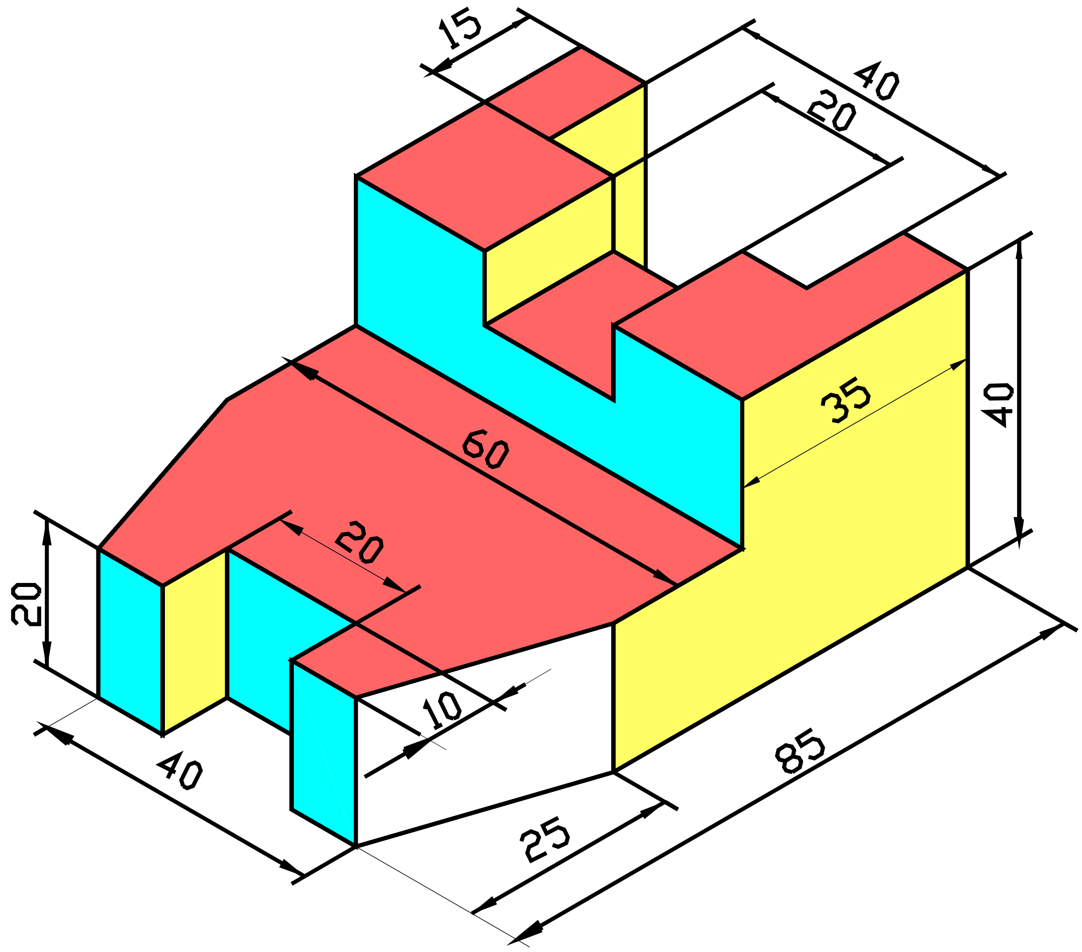
Exercise 10 :

Use a AutoCAD package to create the view from the front ,the view from above and the view from the left from the given isometric view of a machine component.



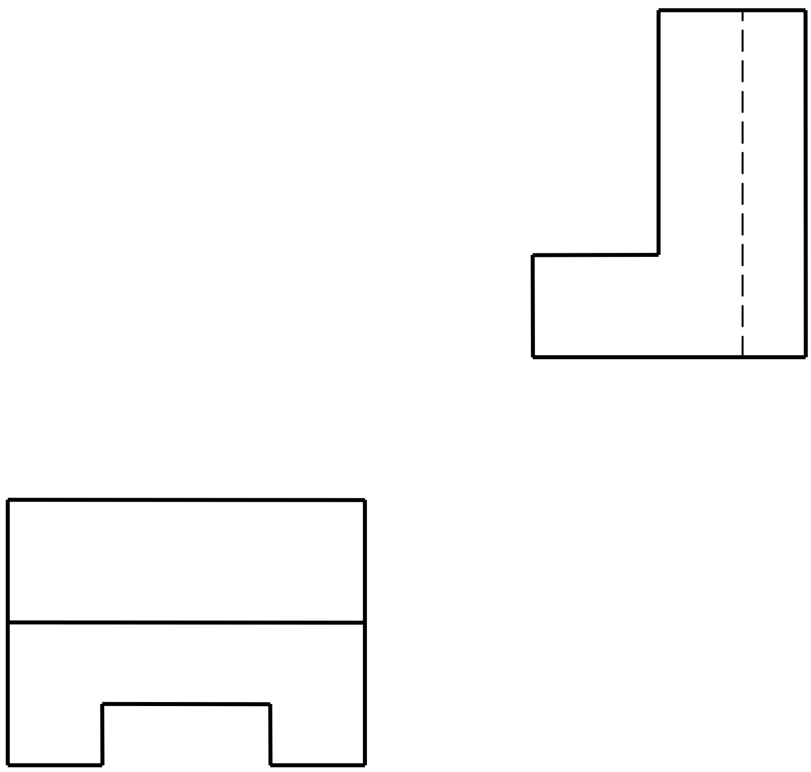
Exercise 11 :

Use a AutoCAD package to create the view from the front ,the view from above and the view from the left from the given isometric view of a machine component.



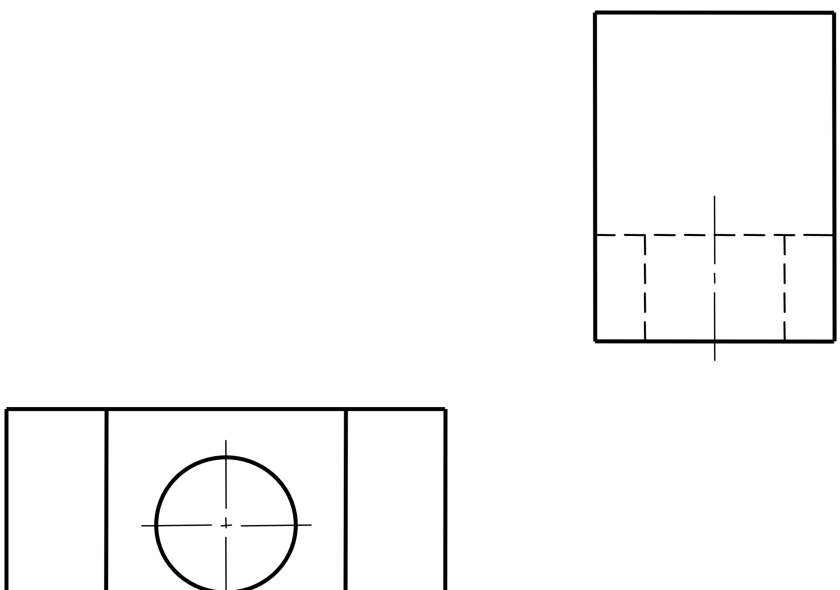
Exercise 12 :

Given two complete views, sketch the missing view.



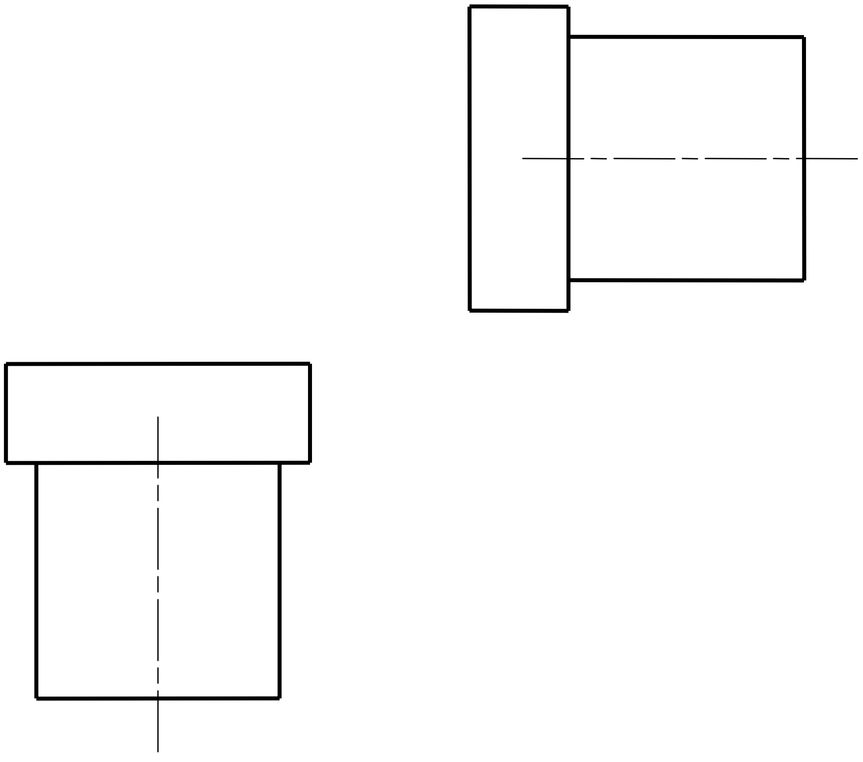
Exercise 13 :

Given two complete views, sketch the missing view.



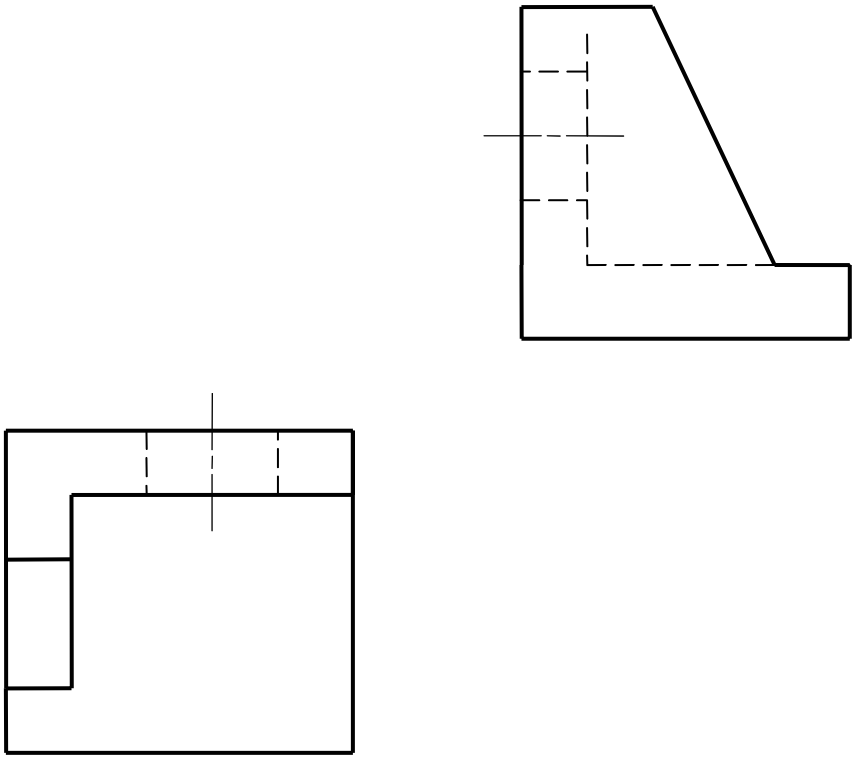
Exercise 14 :

Given two complete views, sketch the missing view.



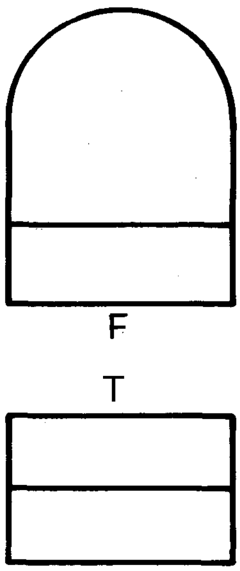
Exercise 15 :

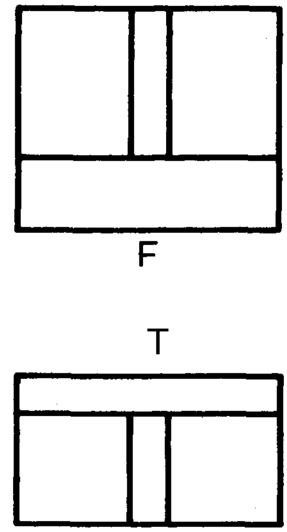
Given two complete views, sketch the missing view.

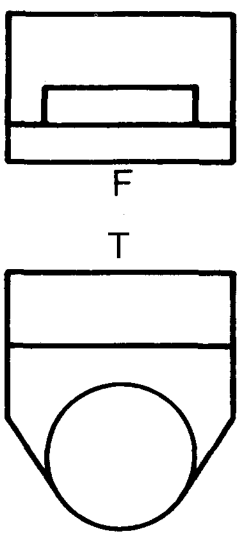


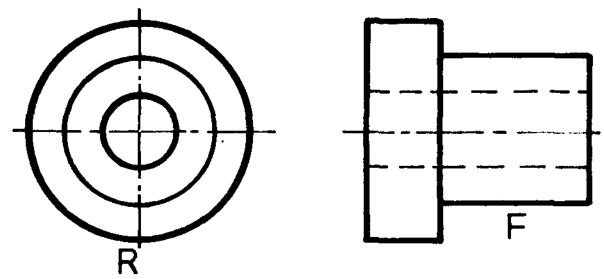
Exercise 16 :

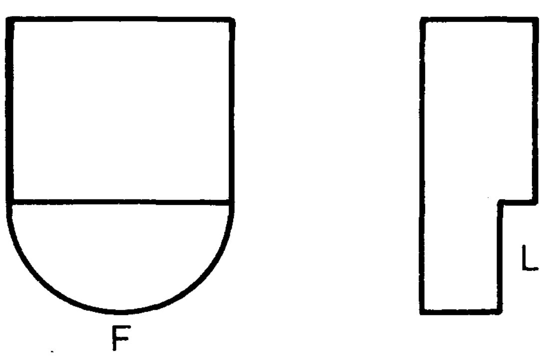
Given two complete views, sketch the missing view.

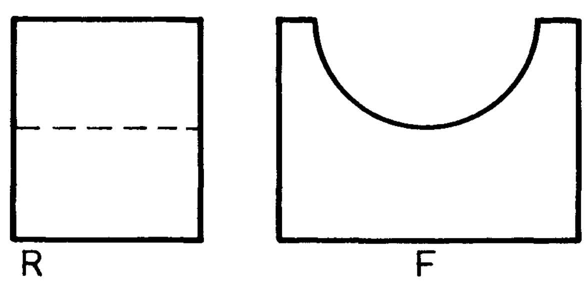


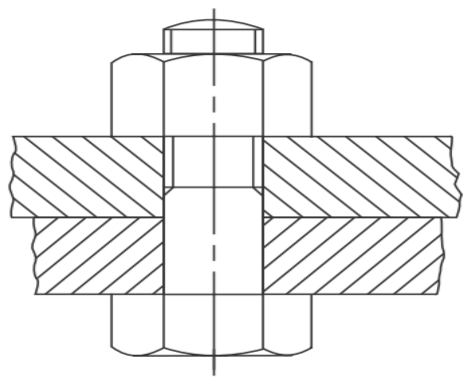


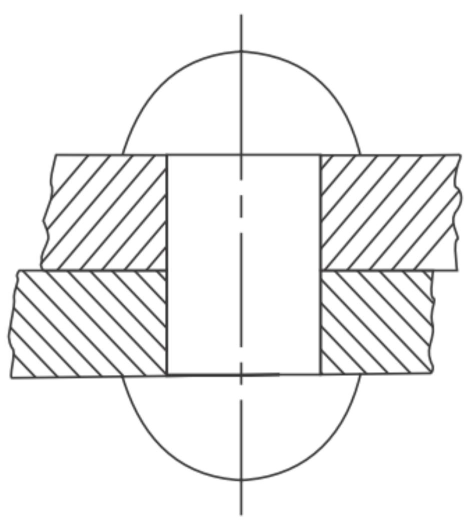


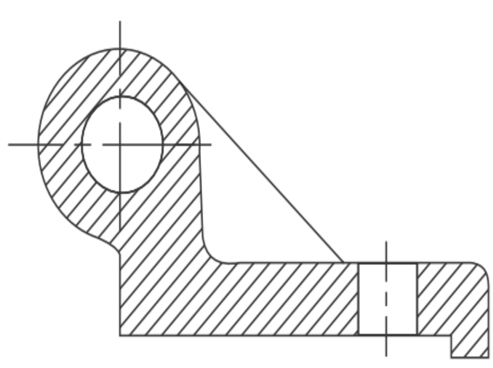












11

SECTIONAL VIEWS

11.1 INTRODUCTION

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

A sectional view is obtained by imagining the object, as if cut by a cutting plane and the portion between the observer and the section plane being removed. Figure 11.1 a shows an object, with the cutting plane passing through it and Fig. 11.1b, the two halves drawn apart, exposing the interior details.

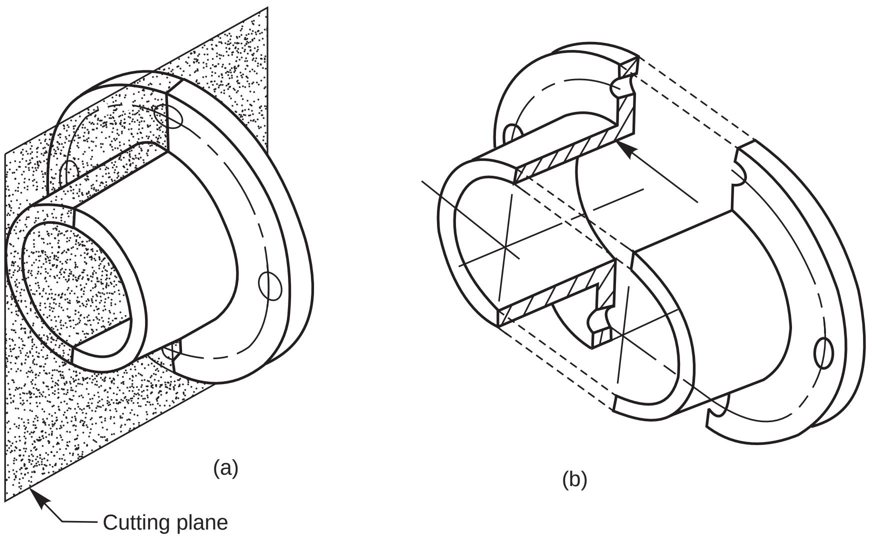


Fig. 11.1 Principles of sectioning

11.2 FULL SECTION

A sectional view obtained by assuming that the object is completely cut by a plane is called a full section or sectional view. Figure 11.2a shows the view from the right of the object shown in Fig.11.1 *a*, in full section. The sectioned view provides all the inner details, better than the unsectioned view with dotted lines for inner details (Fig.11.2 *b*). The cutting plane is represented by its trace (V.T) in the view from the front (Fig.11.2c) and the direction of sight to obtain the sectional view is represented by the arrows.

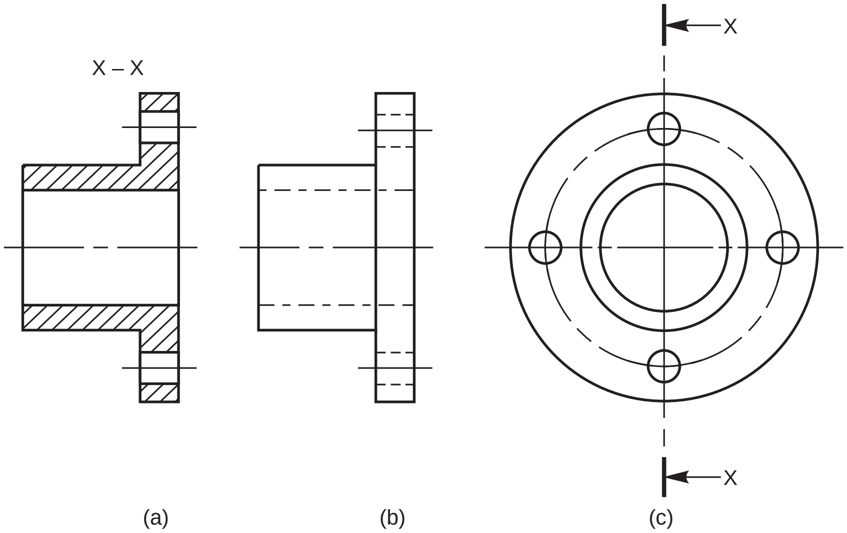


Fig.11.2 Sectioned and un-sectioned views

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

Figure11.3 represents the correct and incorrect ways of representing a sectional view. Sections are used primarily to replace hidden line representation, hence, as a rule, hidden lines are omitted in the sectional views.

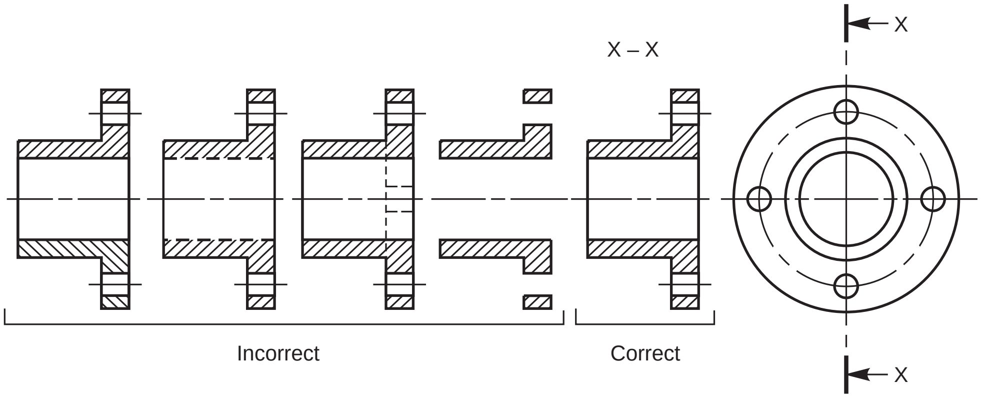
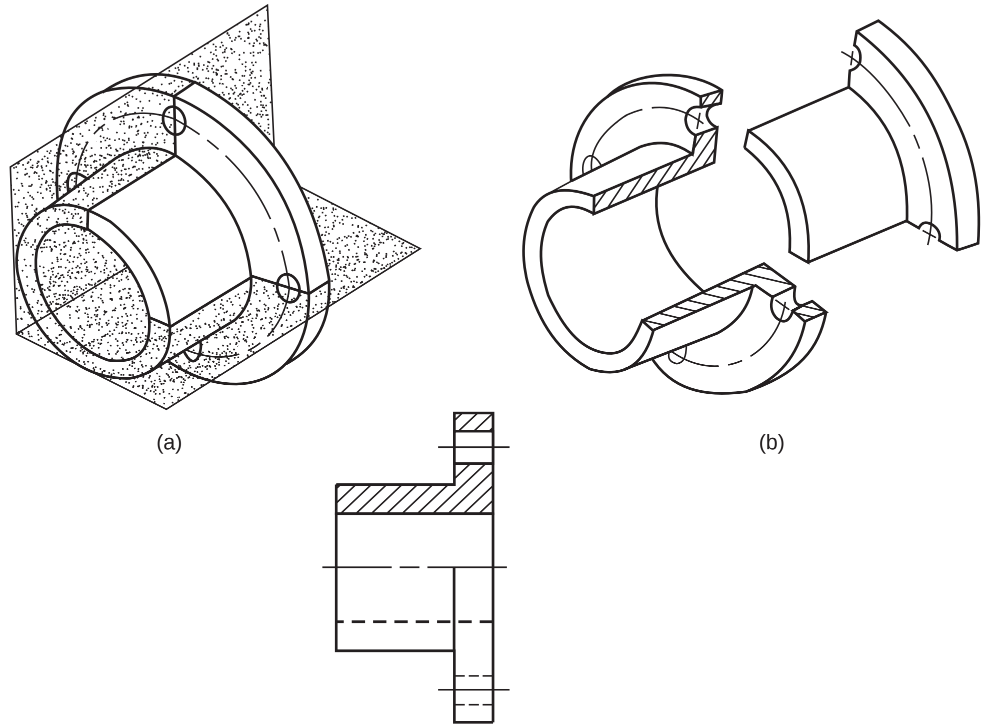


Fig.11.3 Incorrect and correct sections

11.3 HALF SECTION

A half sectional view is preferred for symmetrical objects. For a half section, the cutting plane removes only one quarter of an object. For a symmetrical object, a half sectional view is used to indicate both interior and exterior details in the same view. Even in half sectional views, it is a good practice to omit the hidden lines. Figure11.4*a* shows an object with the cutting plane in position for obtaining a half sectional view from the front, the top half being in section. Figure 11.4*b* shows two parts drawn apart, exposing the inner details in the sectioned portion. Figure 11.4c shows the half sectional view from the front. It may be noted that a centre line is used to separate the halves of the half section. Students are also advised to note the representation of the cutting plane in the view from above, for obtaining the half sectional view from the front.



(a) (b)

(c)

Fig.11.4 Method of obtaining half sectional view

11.4 AUXILIARY SECTIONS

Auxiliary sections may be used to supplement the principal views used in orthographic projections. A sectional view projected on an auxiliary plane, inclined to the principal planes of projection, shows the cross-sectional shapes of features such as arms, ribs and so on. In Fig.11.5, auxiliary cutting plane X-X is used to obtain the auxiliary section X-X.

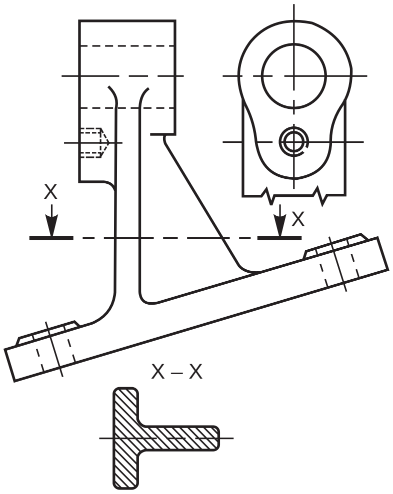
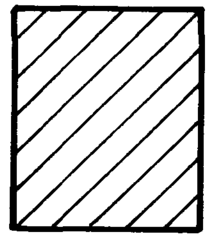


Fig.11.5 Auxiliary section

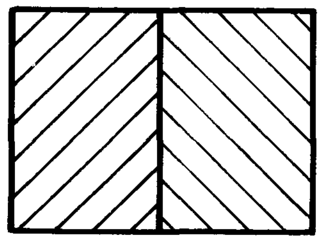
11.5 The rules of sectioning

1. A sectioned object is shown by lines drawn preferably at 45°. Thin lines touch the outline.



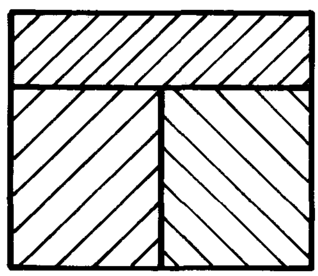
Size of sectioned part determines line spacing - preferably not less than 4 mm.

2. If two adjacent parts are sectioned, the section lines are drawn in opposite directions.



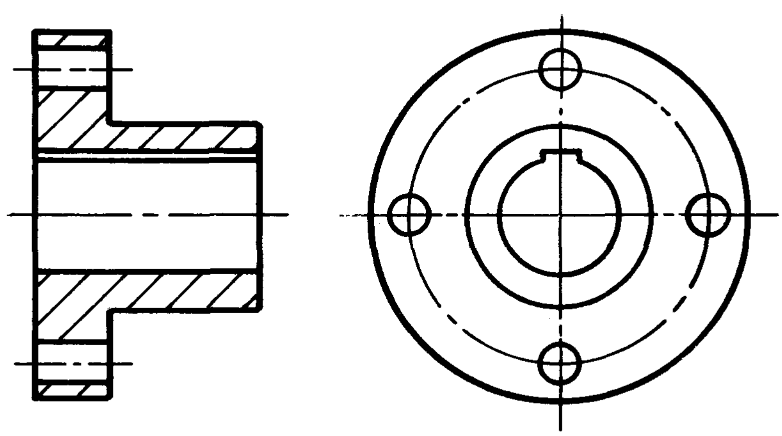
Lines are staggered where the parts are in contact.

3. Where more than two parts of an assembly are to be sectioned, the lines cannot all be opposite.



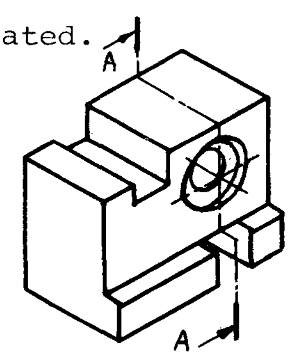
Section lines are closer together on the third area — usually the smallest

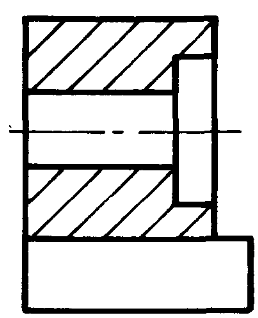
4. The sectional view of a symmetrical object is obtained when the section plane cuts through the obvious centre line.
Hatching may be omitted if the meaning is clear without it.



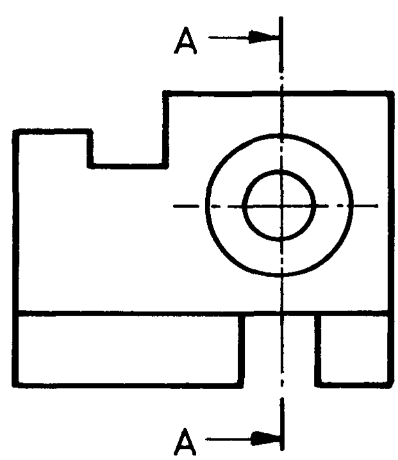
Sectional View

5. If an object is NOT symmetrical the section plane chosen should be clearly stated.



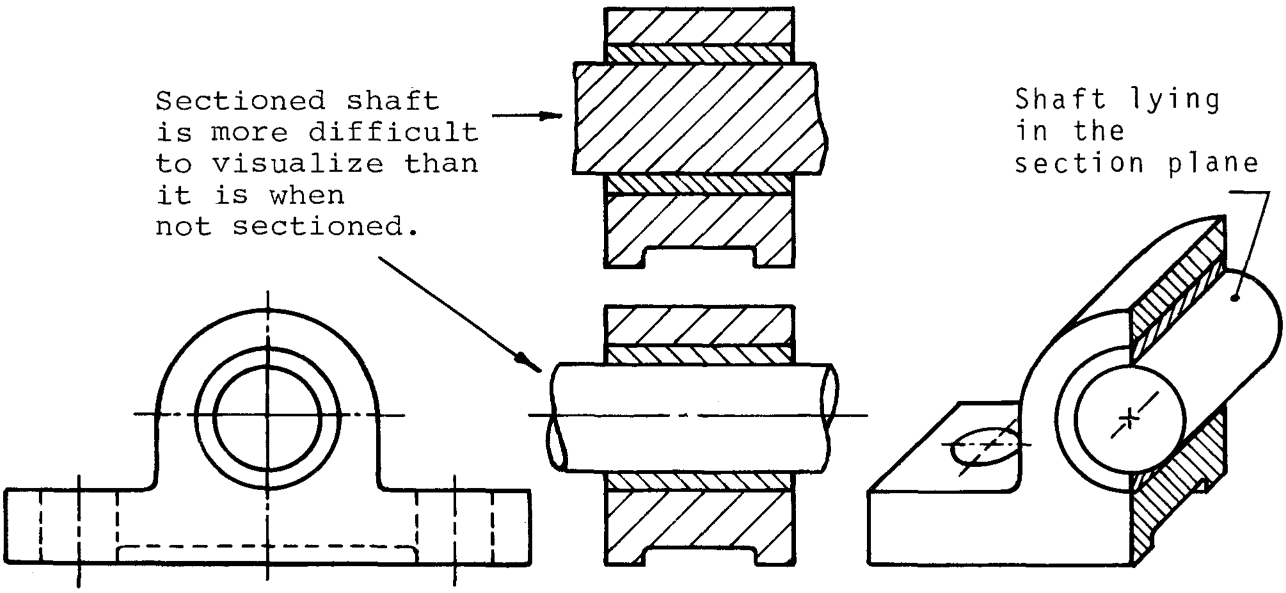


Section A-A



11.6 Sectioning Exceptions

There are a number of features and parts which are NOT normally sectioned even though they may lie in the section plane. A good way to accept these exceptions to the general rule is to imagine how complicated the drawing would look if they were sectioned. They are sectioned, however, when they lie ACROSS the section plane. See section D-D. ✱

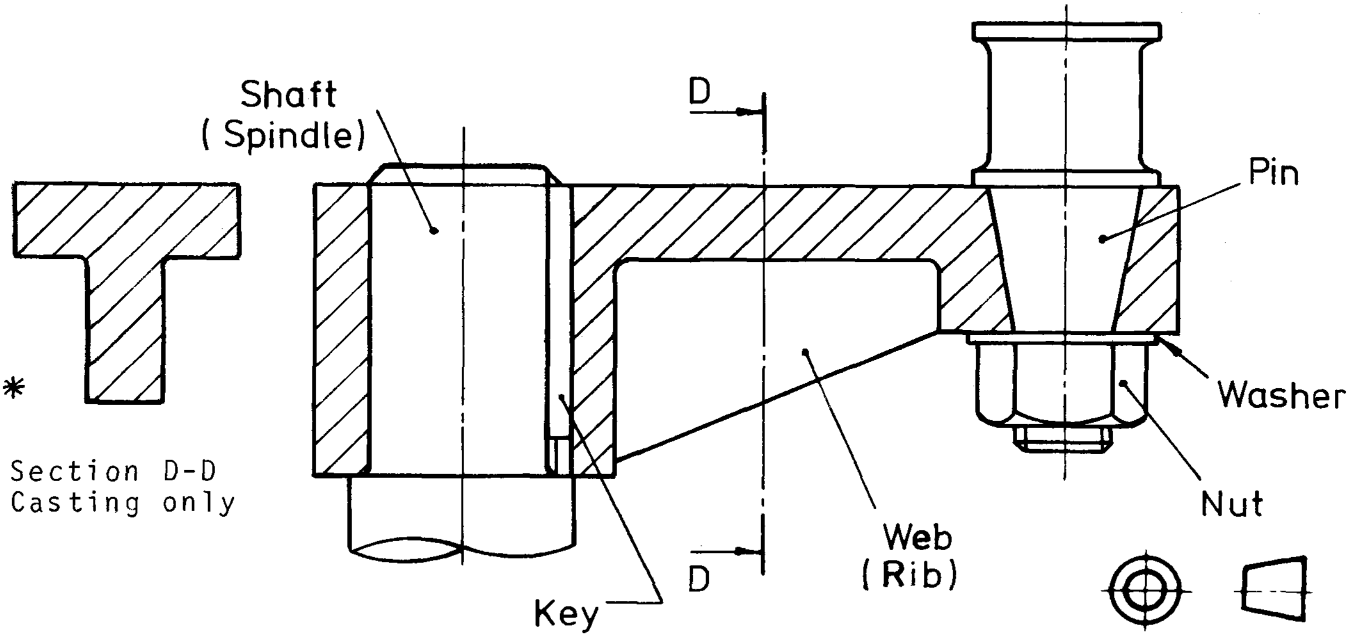


Sectional
End View

BS 308 states that the following are NOT sectioned even if they lie in a given section plane:

Shafts	Keys	Rivets	Ribs	Pins	Dowels
Cotters	Bolts	Gear Wheels	Nuts	Webs	Washers

Some of these are illustrated in BS 308 by this diagram:



11.7 CONVENTIONAL REPRESENTAION

Certain draughting conventions are used to represent materials in section and machine elements in engineering drawings.

11.7.1 Materials

As a variety of materials are used for machine components in engineering applications, it is preferable to have different conventions of section lining to differentiate between various materials. The recommended conventions in use are shown in Fig.11.6.

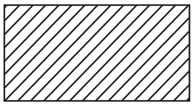
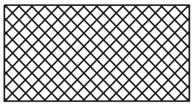
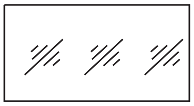
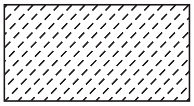
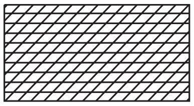
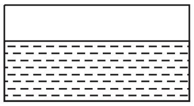
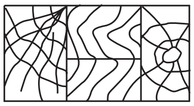
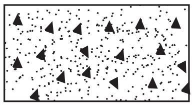
Type	Convention	Material
Metals		Steel, Cast Iron, Copper and its Alloys, Aluminium and its Alloys, etc.
		Lead, Zinc, Tin, White-metal, etc.
Glass		Glass
Packing and Insulating material		Porcelain, Stoneware, Marble, Slate, etc.
		Asbestos, Fibre, Felt, Synthetic resin products, Paper, Cork, Linoleum,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

Fig.11.6 Conventional representation of materials

11.8 Cross- hatching and Pattern Filling

It is common practice to fill an area with a pattern of some sort. The pattern can help differentiate between components, or it can signify the material composition of an object. This is accomplished by HATCH command. Hatching generates line entities for the chosen pattern and adds them to the drawing. AutoCAD normally groups these lines into a general block.

HATCH Command—performs hatching. The pattern filling is illustrated in Fig. 11.8 by selecting appropriate choice in response to HATCH command.

11.8.1 HATCH Command

Choose Draw, Hatch...

or

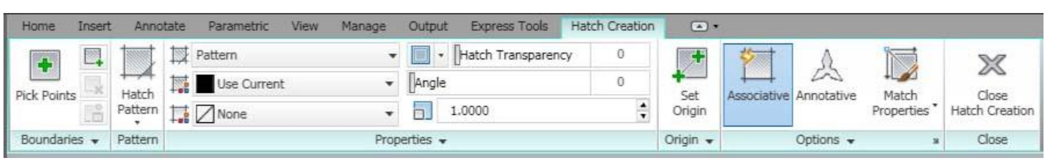
Click the Hatch icon 

or

Type HATCH at the command prompt

Command: **HATCH**

AutoCAD's ribbon changes to show hatch options



11.8.2 Hatch Patterns

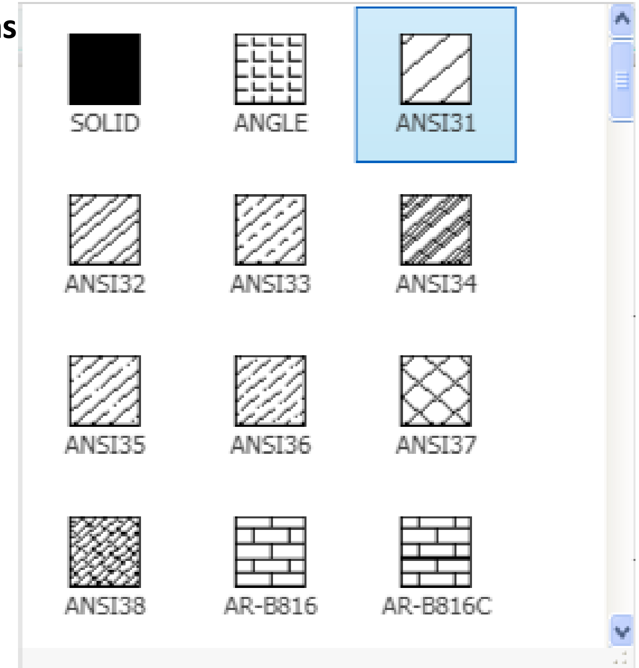
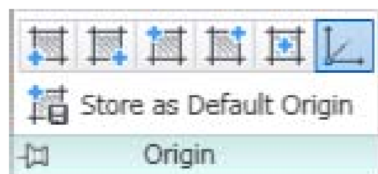


Fig 11.8

Boundaries

Origin



Hatch Options

Pattern Type Sets the current pattern type by using AutoCAD's predefined patterns or user defined patterns.

Pattern Properties Sets the current pattern, scale, angle, and spacing. Controls if hatch is double spaced or exploded.

Pick Points Constructs a boundary from existing objects that form an enclosed area.

Select Objects Selects specific objects for hatching. The Boundary Hatch dialog box disappears and AutoCAD prompts for object selection.

Inherit Properties Applies the properties of an existing associative hatch to the current Pattern Type and Pattern Properties options.

Preview Hatch Displays the hatching before applying it. AutoCAD removes the dialog box and hatches the selected areas.

Associative Controls associative hatching.

Apply Creates the crosshatching in the boundary.

11.8.3 HATCHEDIT

1. **Choose** Modify, Hatch...

   or

   Click the Hatch Edit icon. 

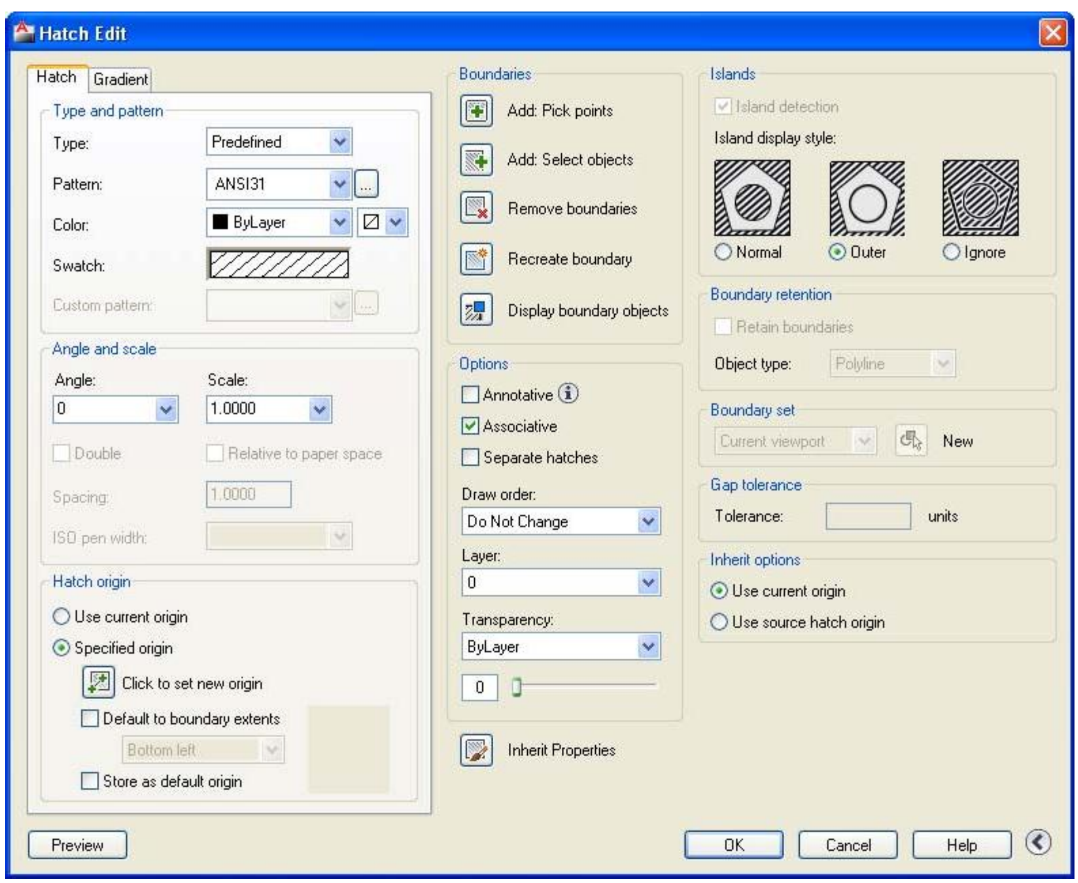
   or

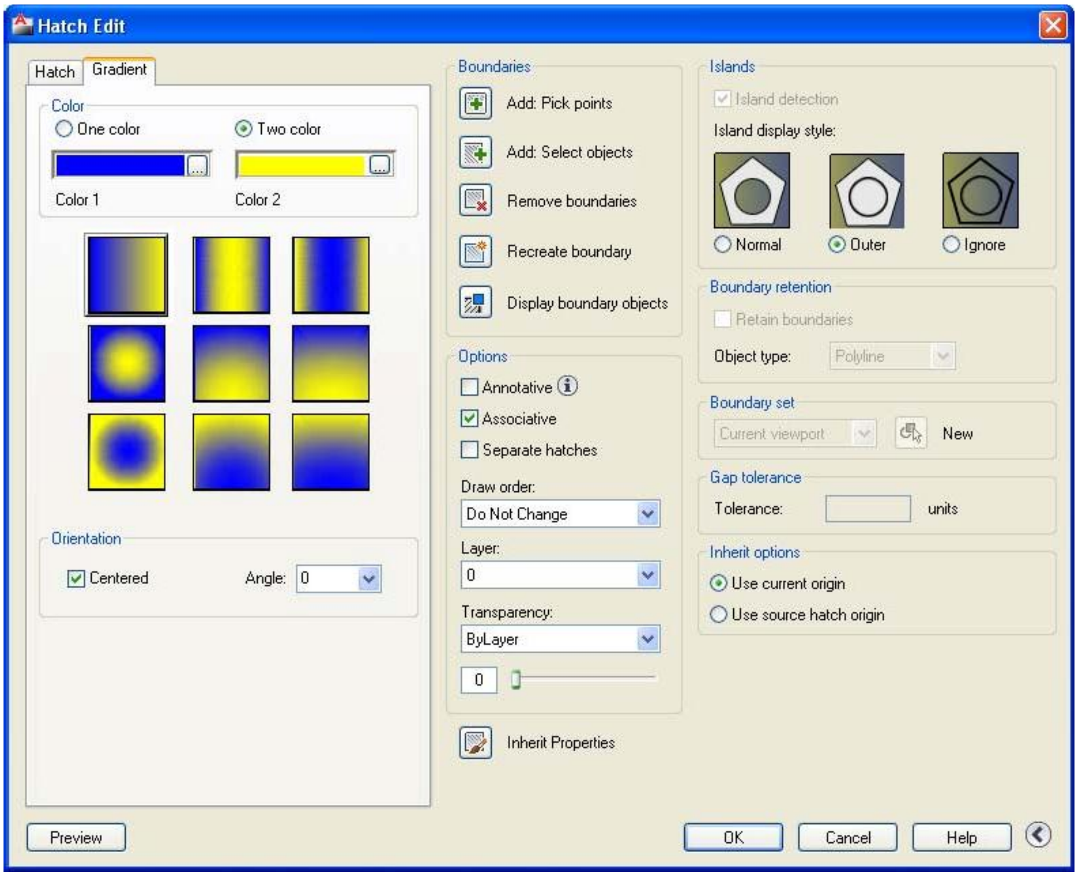
   Type HATCHEDIT at the command prompt.

   Command :**HATCHEDIT**

2. **Choose** One of the BHATCH options to modify.

3. **Pick** The OK button.




Hatch Edit
Hatch
Gradient
Color
One color
Two color
Color 1
Color 2
Orientation
Centered
Angle:
0
Preview
Boundaries
Add: Pick points
Add: Select objects
Remove boundaries
Recreate boundary
Display boundary objects
Options
Annotative
Associative
Separate hatches
Draw order:
Do Not Change
Layer:
0
Transparency:
ByLayer
0
Inherit Properties
Islands
Island detection
Island display style:
Normal
Outer
Ignore
Boundary retention
Retain boundaries
Object type:
Polyline
Boundary set
Current viewport
New
Gap tolerance
Tolerance:
units
Inherit options
Use current origin
Use source hatch origin
OK
Cancel
Help

EXAMPLES

EXAMPLE (1)

Figure11.9 shows the isometric view of a machine block and (*i*) the sectional view from the front, (*ii*) the view from above and (*iii*) the sectional view from the left.

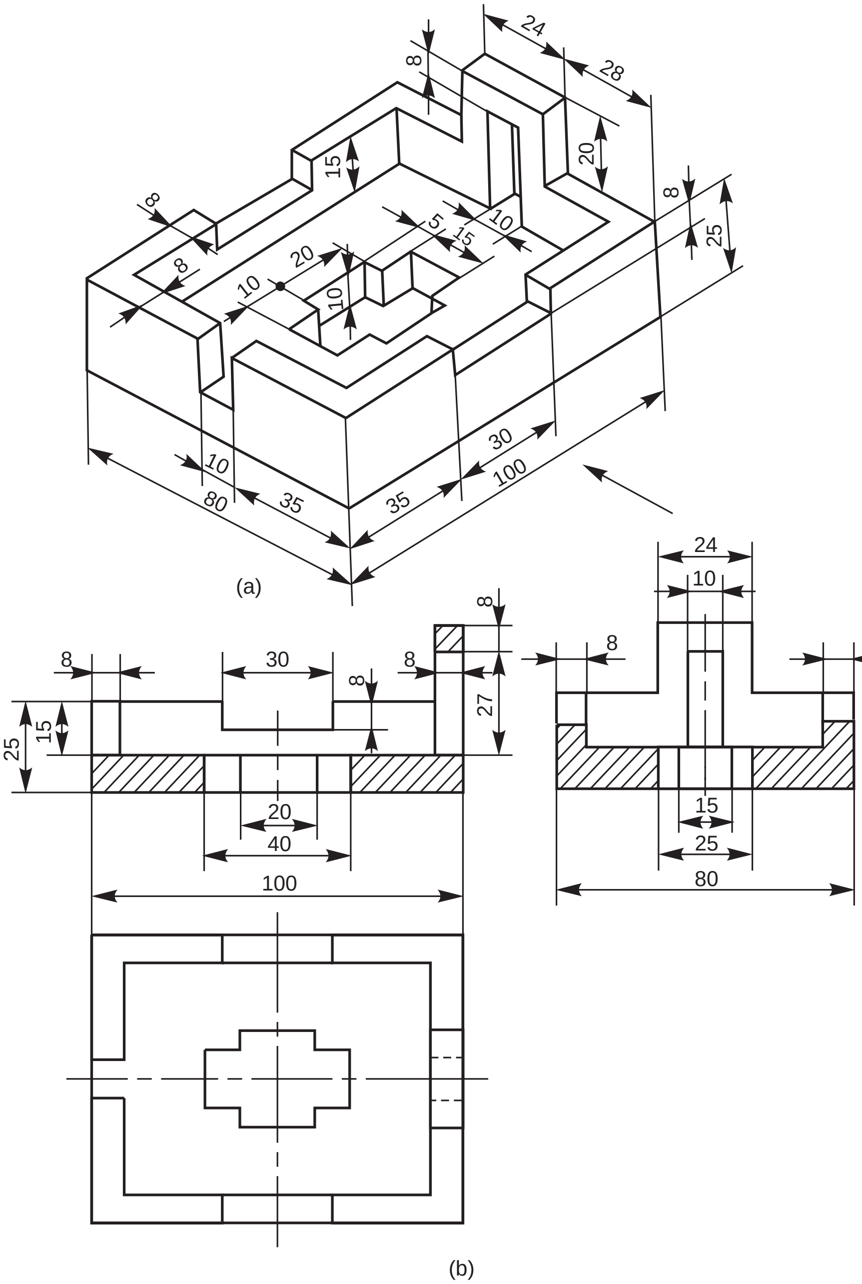
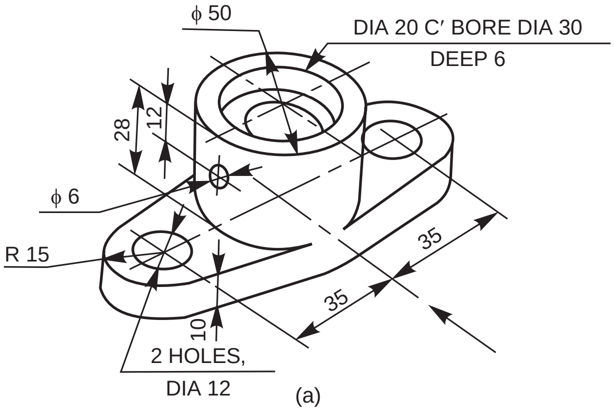


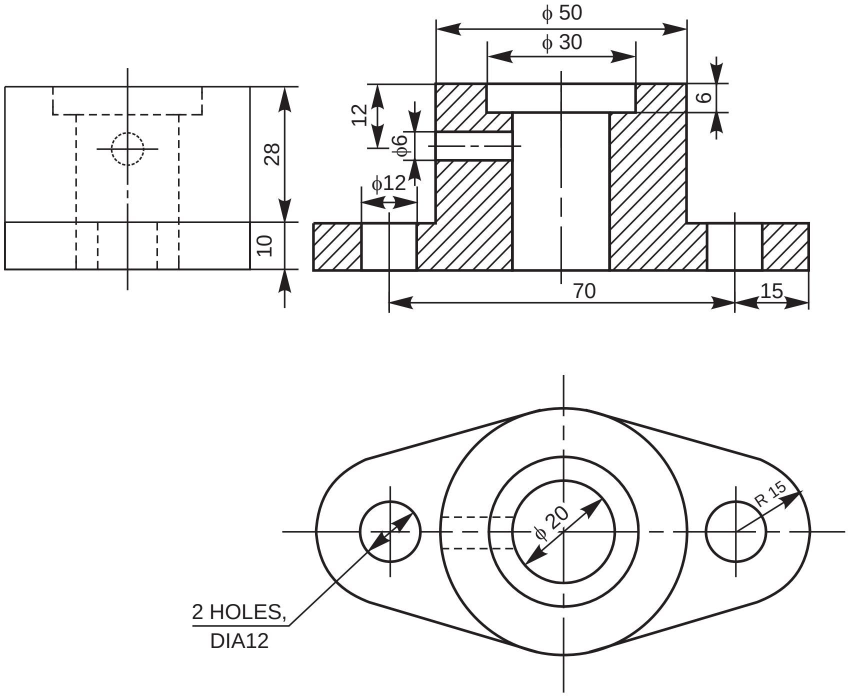
Fig.11.9 Machine block

EXAMPLE (2)

Figure11.10 shows the isometric view of a shaft support. Sectional view from the front, the view from above and the view from the right are also shown in the figure.



(a)



(b)

Fig.11.10 Shaft support

EXAMPLE (3)

Figure11.11 shows the isometric view of a machine component along with the sectional view from the front, the view from above and the view from the left.

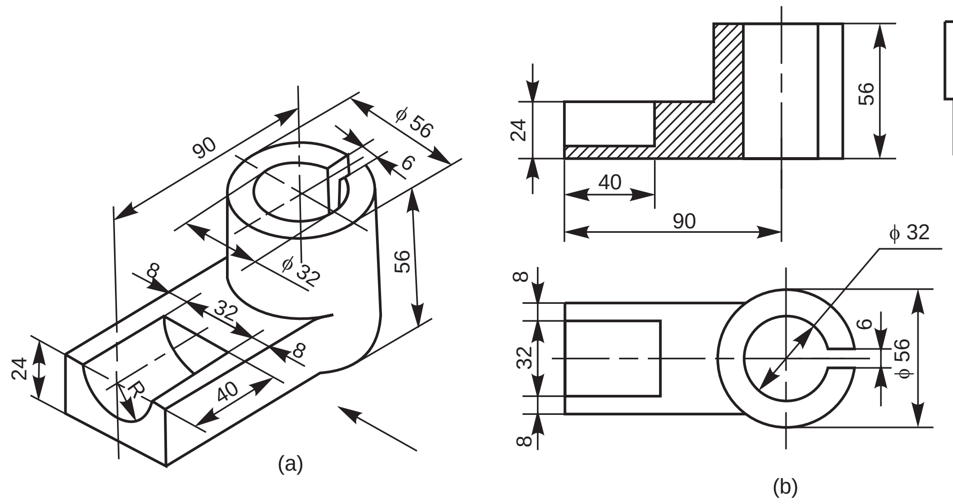


Fig.11.11 Machine component

EXAMPLE (4)

Figure11.12 shows a sliding block and (*i*) the view from the front, (*ii*) the view from above and (*iii*) the sectional view from the right.

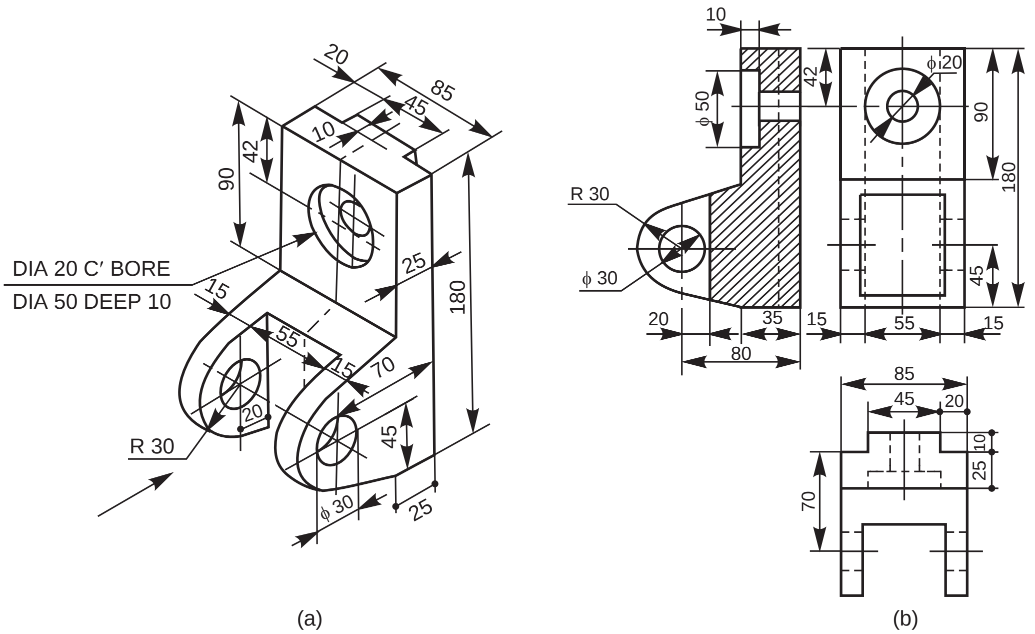


Fig.11.12 Sliding block

EXAMPLE (5)

Figure11.13 shows the orthographic views of a yoke. The figure also shows the sectional view from the front, the sectional view from the right and the view from above.

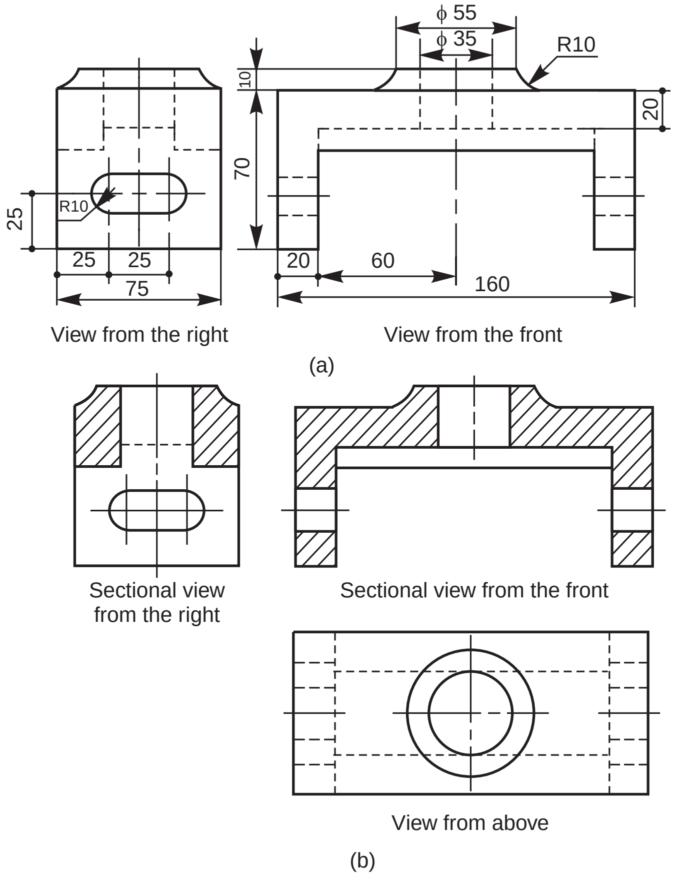
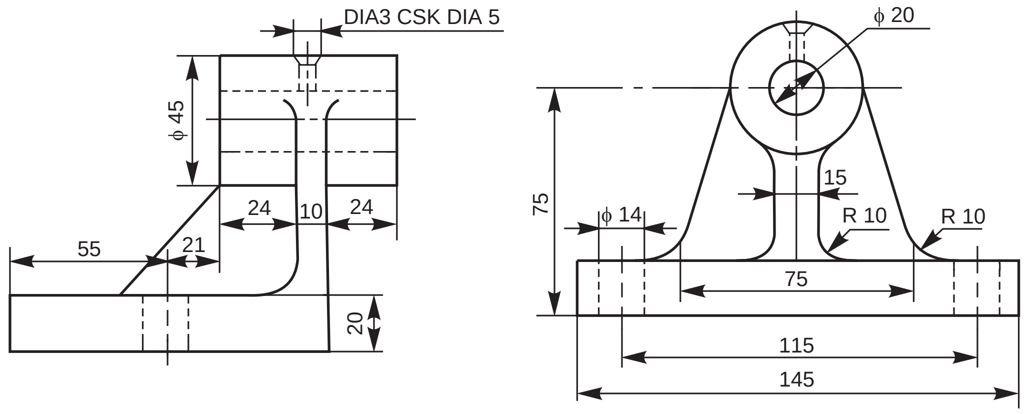


Fig.11.13

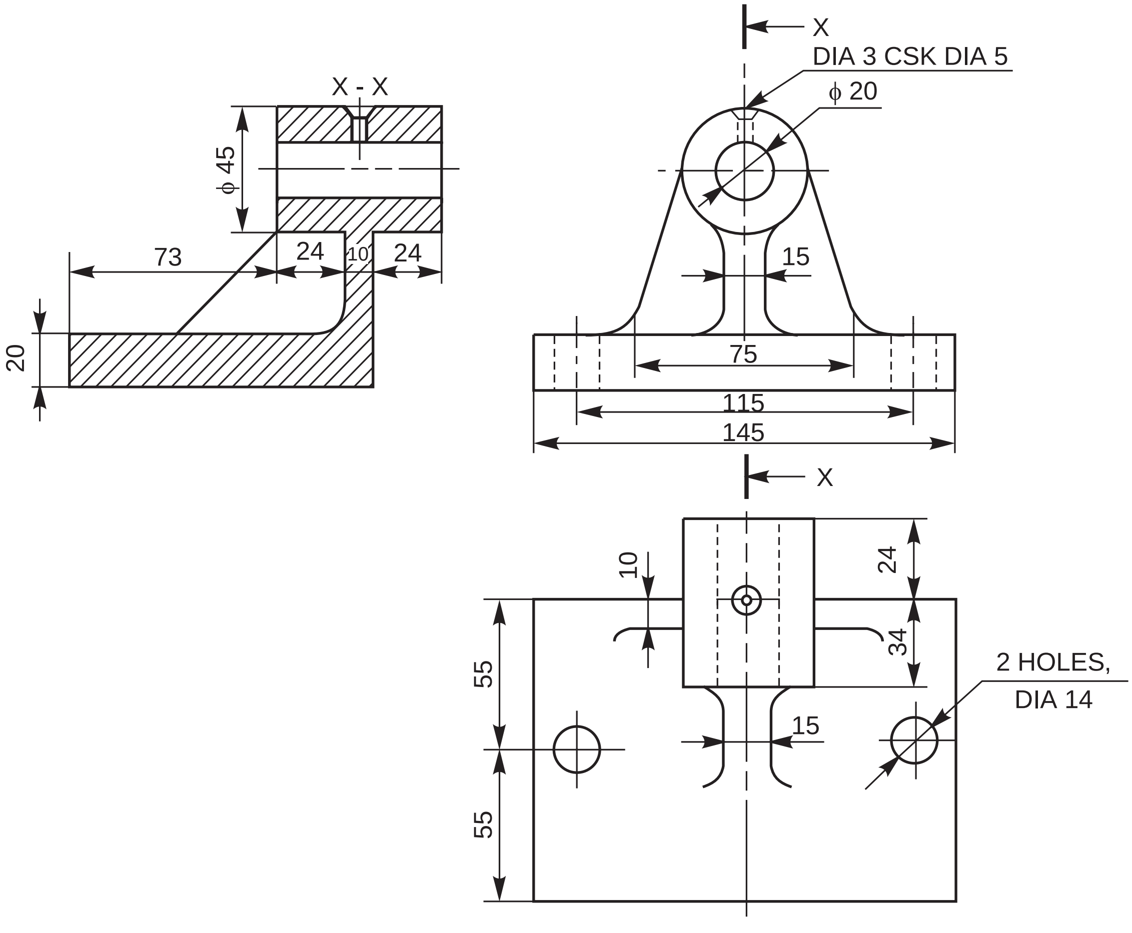
EXAMPLE (6)

Figure11.14 shows the orthographic views of a bearing bracket. The sectional view from the right and view from above are developed and shown in the figure.



(a)

Fig 11.14 Bearing bracket



(b)

Fig 11.14 Bearing bracket

Exercise 1:

Figure 11.15 shows the isometric view of a machine block. Draw (i) sectional view from the front, (ii) the view from above and (iii) the section view from the left .

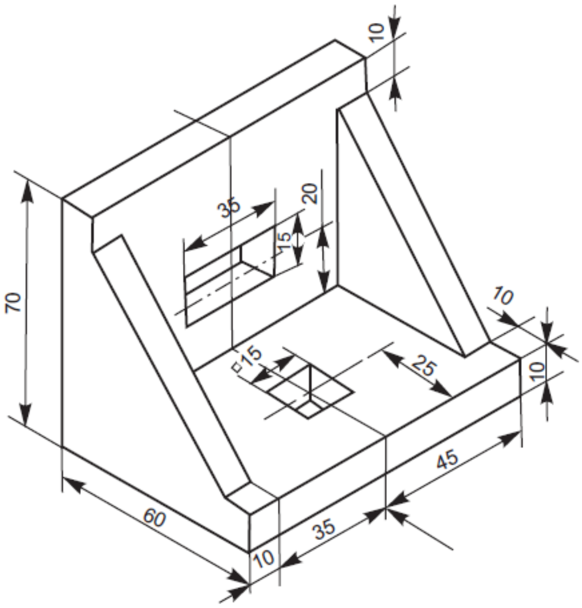
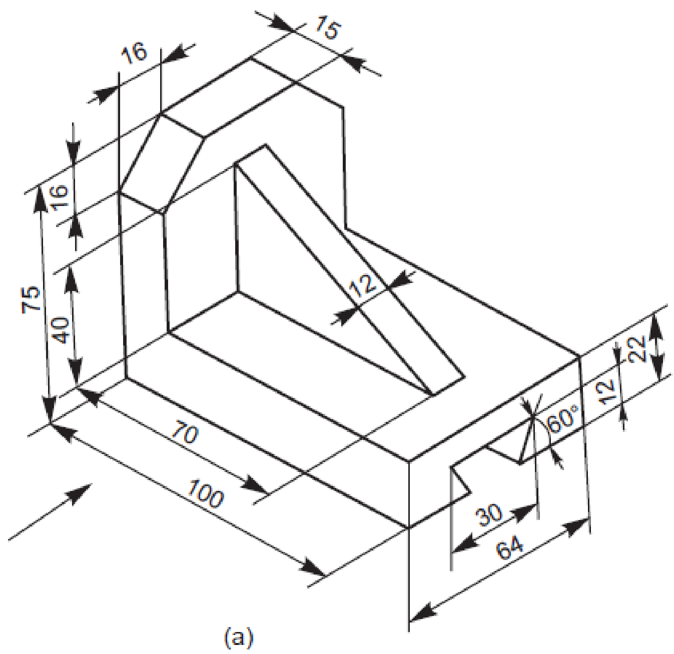


Fig 11.15

Exercise 2:

Figure 11.16 shows the isometric view of a machine block Draw (i) sectional view from the front, (ii) the view from above and (iii) the section view from the left .



(a)

Fig 11.16

Exercise 3:

Figure 11.17 shows the isometric view of a machine block Draw (i) the view from the front, (ii) sectional view from above and (iii) the view from the left.

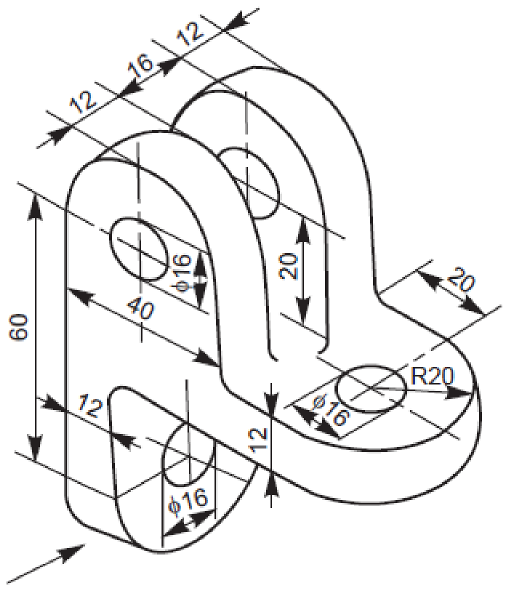


Fig 11.17

Exercise 4:

Figure 11.18 shows the isometric view of a machine block Draw (i) the view from the front, (ii) sectional view from above and (iii) the view from the left.

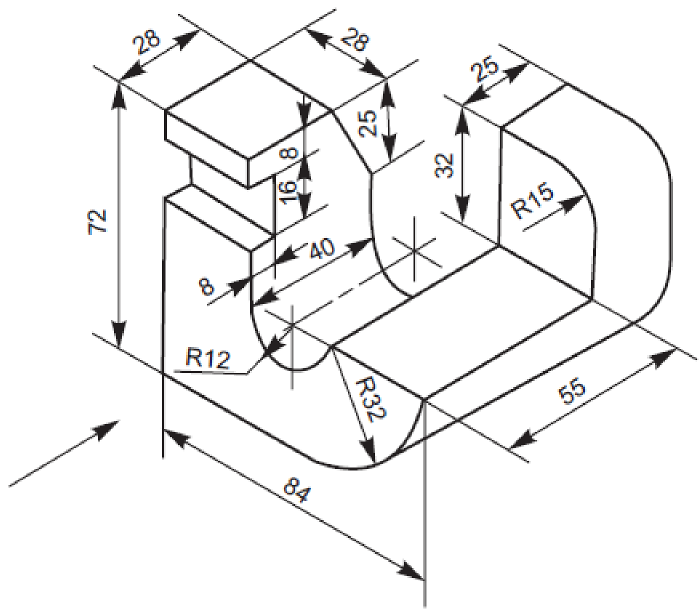


Fig 11.18

Exercise 5:

Figure 11.19 shows the isometric view of a machine block Draw (i) sectional view from the front, (ii) the view from above and (iii) the section view from the left .

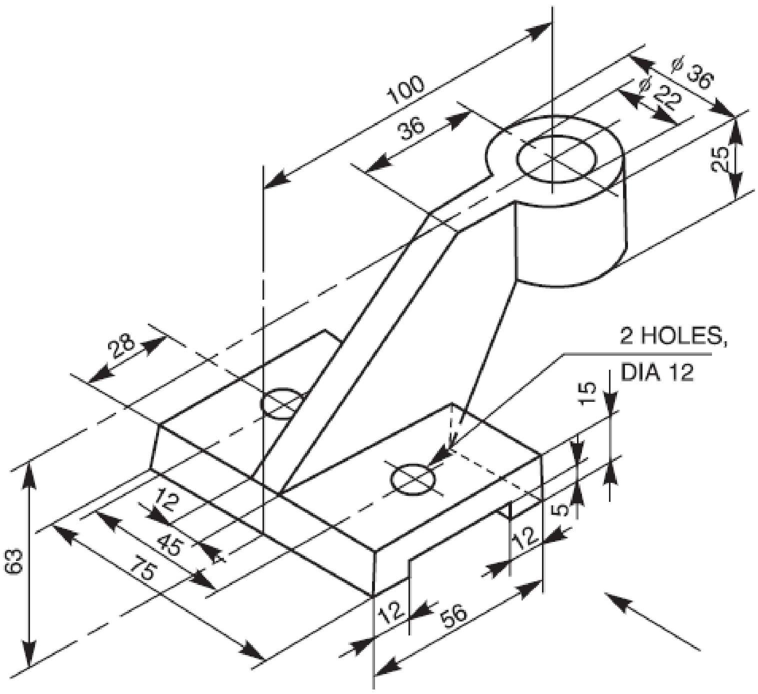


Fig 11.19

Exercise 6:

Figure 11.20 shows the isometric view of a machine block Draw (i) sectional view from the front, (ii) the view from above and (iii) the section view from the left .

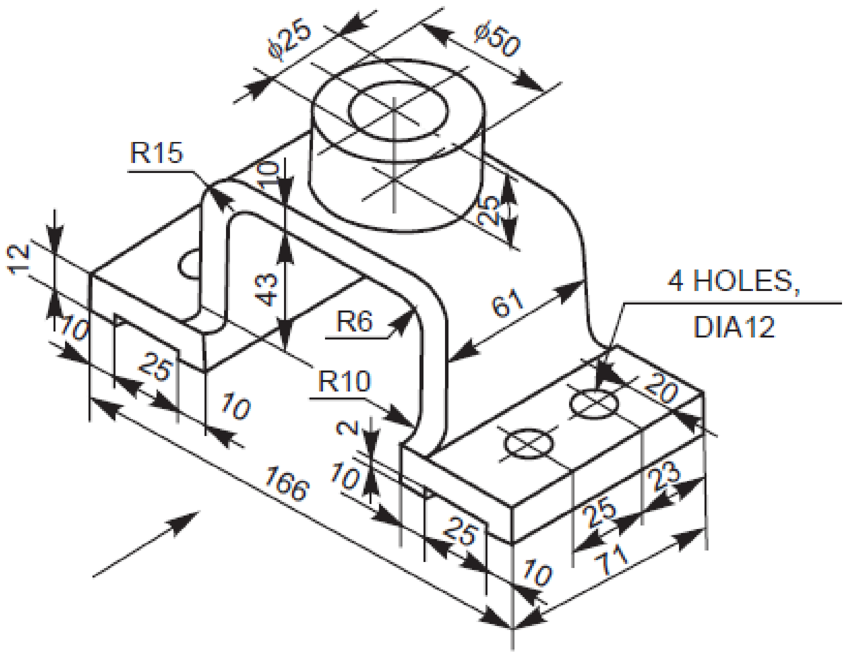


Fig 11.20

Exercise 7:

Figure 11.21 shows the isometric view of a machine block Draw (i) sectional view from the front, (ii) the view from above and (iii) the section view from the left .

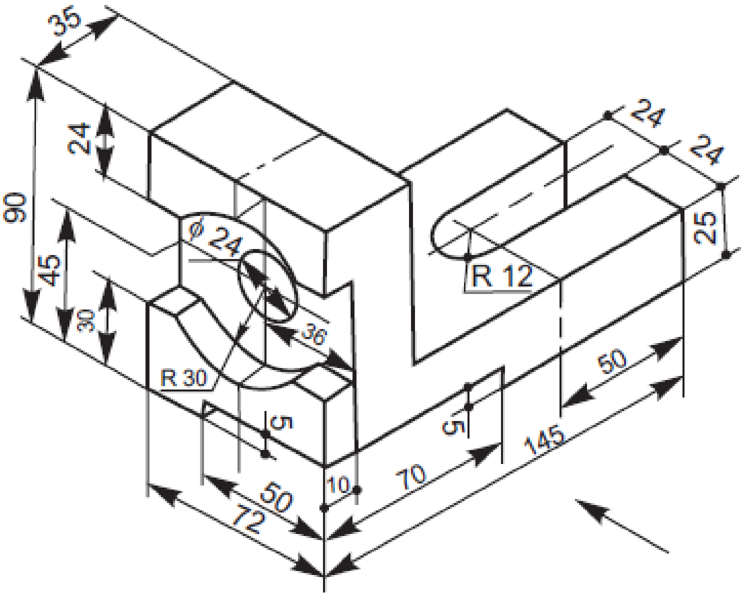


Fig 11.21

Exercise 8:

Figure 11.22 shows the isometric view of a machine block Draw (i) sectional view from the front, (ii) the view from above and (iii) the section view from the left .

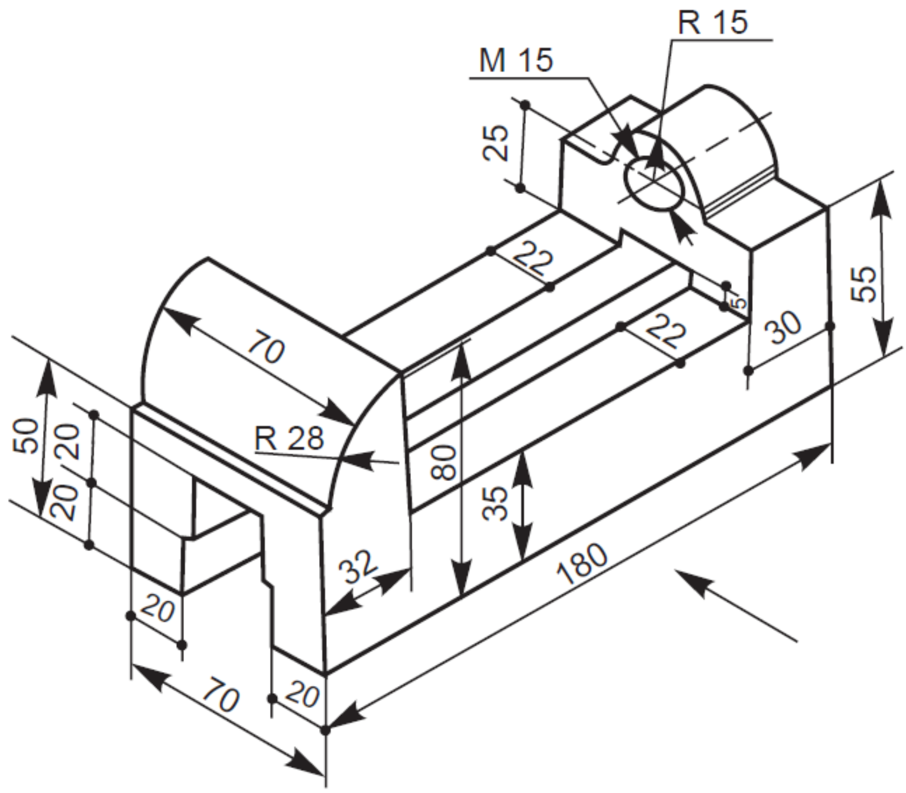


Fig 11.22