

# LABORATORY MANUAL

## COMPUTER AIDED DESIGN LAB

ME-211-F

**List of Experiments:-**

Sl. No.	Name of experiment	Date	Signature
01	Setting up of drawing environment by setting drawing limits, drawing units, naming the drawing, naming layers, setting line types for different layers using various type of lines in engineering drawing, saving the file with .dwg extension.		
02	Layout drawing of a building using different layer and line colors indicating all Building details. Name the details using text commands, Make a Title Block.		
03	Make an Isometric dimensioned drawing of Connecting Rod using isometric grid and snap.		
04	Draw quarter sectional isometric view of a cotter joint.		
05	Draw different types of bolts and nuts with internal and external threading in Acme and Square threading standards. Save the bolts and nuts as blocks suitable for insertion.		
06	Draw 3D models by extruding simple 2D objects, dimension and name the objects.		
07	Draw a spiral by extruding a circle.		
08	To Draw Orthographic Projection Drawings (Front, Top, Side) of boiler safety valve giving name the various components of the valve.		

**Note:-**

- 1. At least seven experiments are to be performed in the semester.**
- 2. At least five experiments should be performed from the above list. Remaining two experiments may either be performed from the above list or designed & set by the concerned institute as per the scope of the syllabus**

**Exp. 1. Setting up of drawing environment by setting drawing limits, drawing units, naming the drawing, naming layers, setting line types for different layers using various type of lines in engineering drawing, saving the file with .dwg extension.**

### **Setting up drawing limits:-**

- i) menu bar :- Format > Drawing limits  
OR
- ii) Command:- limits

Set the drawing areas by **limits** command. First enter the limits command. It will display the prompt “to specify the lower left corner” and then the next prompt is “specify the upper right corner”.

The following is the prompt sequence of the limits command for setting limits.

Specify lower left corner or [ ON/OFF]: 0,0  
Specify upper right corner :11,9

### **Setting up drawing Units:-**

- i) menu bar :- Format > units  
OR
- ii) Command:- units

The **units** command is used to select a format for the unit of distance and angle measurement.

After entering the **units** command it will display the **Drawing Units** dialog box which is used to set the units and angles. Then specify the Precision for the units and angles from the corresponding precision drop-down list. After selecting all the set-up press **OK**.

### **Setting up drawing Layers:-**

- i) menu bar :- Format > layer  
OR
- ii) Command:- layer

After entering the Layer command it will display the Layer Properties Manager dialog box. You can create new layers with assign a new line-types, layers name and line colour.

#### **Procedure for create a new layer:-**

Choose the **new** button in the **Layer Properties Manager** dialog box. A new layer with name **Layer 1** is created. Now type the name of the new layer.

#### **Procedure for setting the line type:-**

To assign a new **line-type** to a layer, click in the current line type displayed with a particular layer in the **Layer Properties Manager** dialog box. After clicking in the line-type. Auto CAD will display the **select line-type** dialog box. Click on the **Load** button and select the new line-type form the **Load or Reload** dialog box and then choose the **OK** button.

**Procedure for save a file:-**

The drawing must be saved before exit from the file.

- i) menu bar :- File > save or save as  
OR
- ii) Command:- save or qsave.

After entering the save command it will display a **Save Drawing As** dialog box. Select the proper place and type the file name. Select file-types and then click **Save** button.

Note:- All the drawing files save with .dwg extension.

**Exp. 2. Layout drawing of a building using different layer and line colors indicating all Building details. Name the details using text commands, Make a Title Block..**

**Procedure:-** List of commands for drawing Building and make a title block.

- i) Layer
- ii) Line
- iii) Offset
- iv) Trim
- v) Dimlinear
- vi) Block

i) Layer

Draw > Layer

Command: layer (It will display a dialog box)

Select New button and create three (3) different layers with different colors

Layers	Color
1. Drawing	Blue
2. Title Block	Yellow
3. Dimension	Red

ii) Line

Draw > Line

Command: \_line

Specify first point:

Specify next point or [Undo]: <Ortho on> 60

iii) Offset

Modify > offset

Command: \_offset

Specify offset distance or [Through] <Through>: 2

Select object to offset or <exit>:

Specify point on side to offset:

iv) Trim

Modify > Trim

Command: \_trim

Select cutting edges ...

Select objects to trim:

v) Dimlinear

Dimension > linear

Command: `_dimlinear`

Specify first extension line origin or <select object>:

Specify second extension line origin:

Specify dimension line location:

vi) Block:-

Command: `_block`

Draw > Block > Make (it will display a Block Definition dialog box)

a) Specify the Block Name.

b) Specify the insertion Base Point (This point is used as a reference point to insert the block by choosing Pick Point)

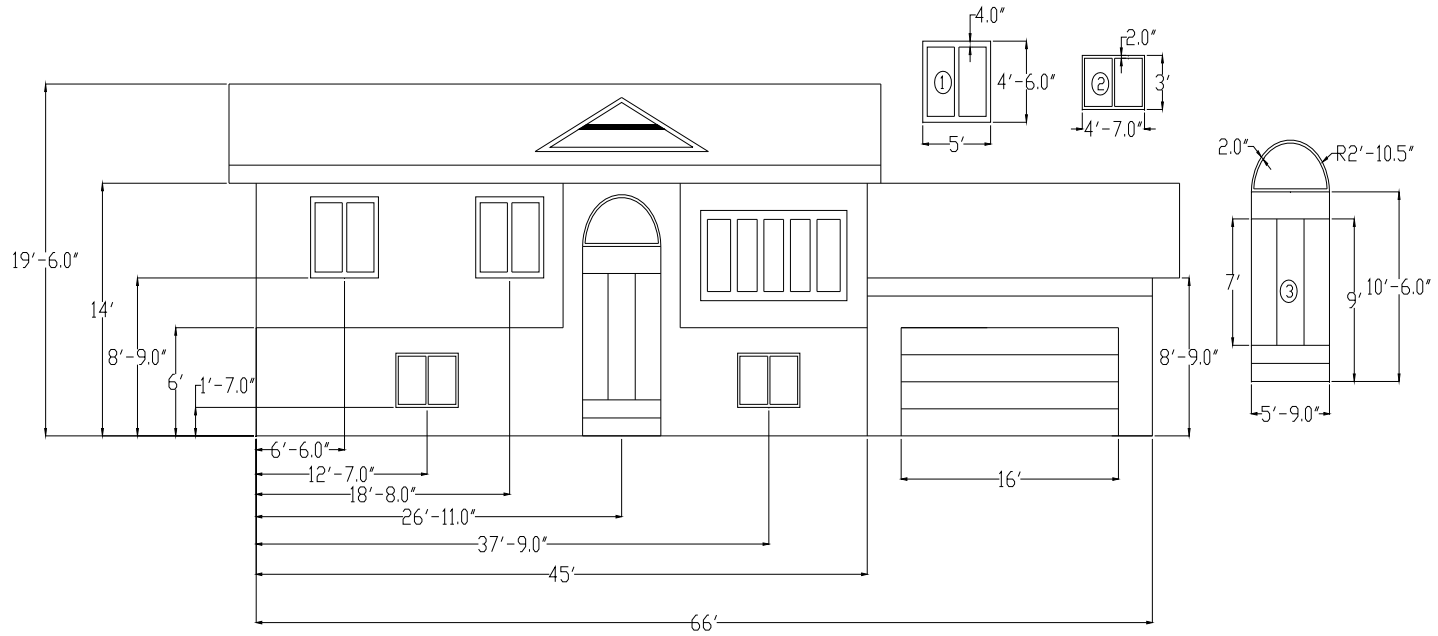
c) Select object.


d) OK.

Command: `Insert`

Insert > Block > Select the Block Name from the drop down list > Ok.

**BUILDING DRAWING**  
**(Exp.-02)**



Dronacharya College of Engineering		Name	Date
		Design	
		Drawn	
		Checked	
		Standard	
Scale-1:1		Approved	
 Drawing of a Building		Sheet No.-02	

**Exp. 3. To make an isometric dimensional drawing of a connecting rod using isometric grid and snap.**

The connecting rod in the connection b/w the piston and crank shaft. It joins the wrist pin of the piston with the throw or crank pin of the crankshaft. The lighter the connection rod and piston greater the resulting power and lesser the vibration because the reciprocation weight is less.

Isometric drawing are generally used to help visualize the shape of an object. It is much easier to conceive the shape of the object.

An Isometric drawing should not be confused with a three-dimensional (3D) drawing. An Isometric Drawing is just a two-dimensional representation of a three-dimensional drawing in 2D plane.

**PROCEDURE:-**

**GRID:-** This will make the whole screen into a graph paper

**SNAP:-** If you select snap or s, auto cad takes that you want the specification to grid spacing requirements.

**Tools > Drafting Setting > Select SNAP AND GRID tab > SNAP TYPE & STYLE > Select Isometric snap > OK.**

The crosshairs are displayed at an isometric angle, and their orientation depends on the current isoplane.

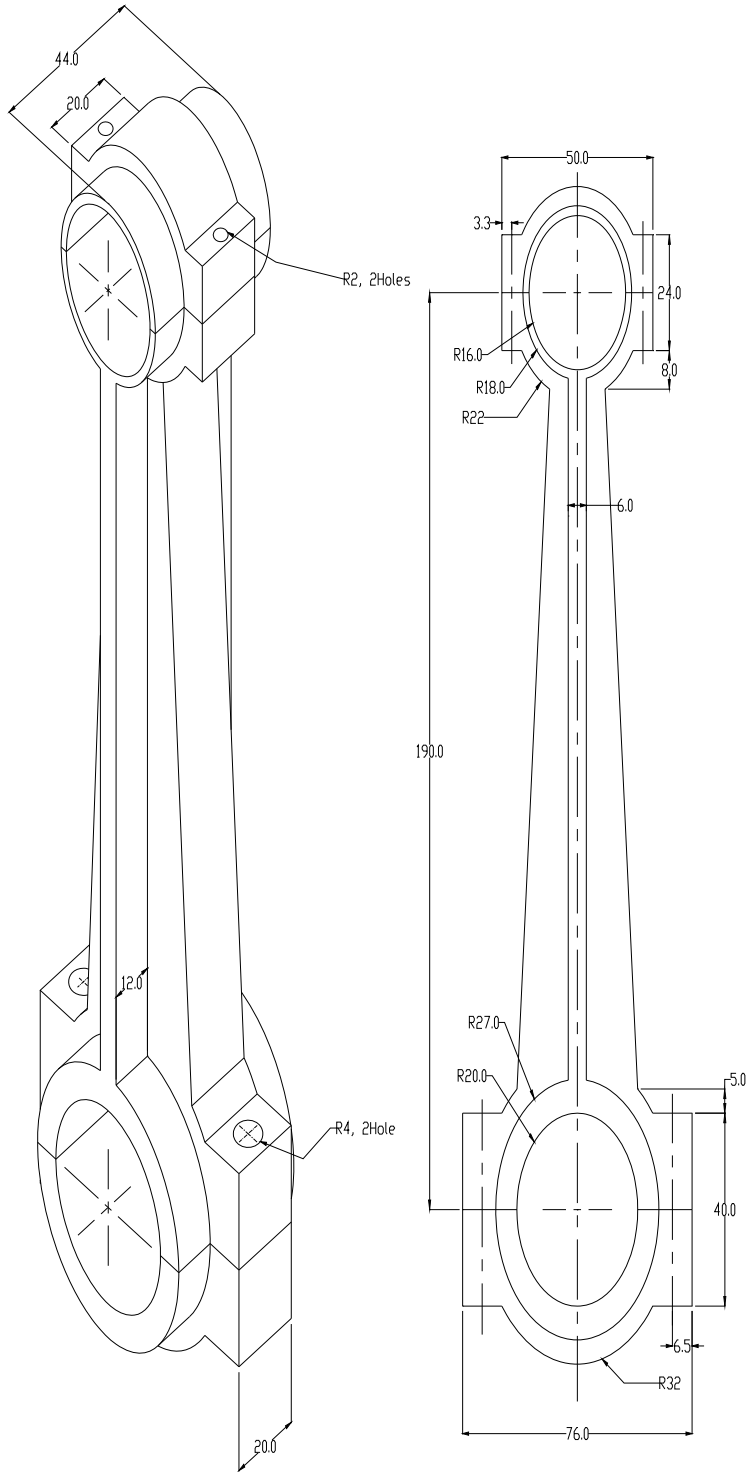
You can change the isoplane by pressing CTRL+E or F5.

**Drawing Isometric Circle:-**

Isometric circles are drawn by using the ELLIPS command and then selecting the Isocircle option. Before you enter the radius or diameter of the isometric circle, you must make sure you are in the required isoplane.



**CONNECTING ROD**  
**(Exp.-03)**



**Exp. 4. Draw quarter sectional isometric view of a cotter joint.**

Isometric drawings are generally used to help visualize the shape of an object. An isometric drawing is just a two-dimensional representation of a three-dimensional drawing in a 2D plane.

**Procedure for setting the snap and grid:-**

Command: snap

Specify snap spacing or [ON/OFF/Aspect/Rotate/Style/Type] <10.0000>: s

Enter snap grid style [Standard/Isometric] <S>: i

Command: \_line

Specify first point:

Specify next point or [Undo]:

Command: ellipse

Specify axis endpoint of ellipse or [Arc/Center/Isocircle]: i

Specify center of isocircle:

Specify radius of isocircle or [Diameter]: 12

Command: copy

Select objects: <Snap off> 1 found

Select objects:

Specify base point or displacement, or [Multiple]: Specify second point of displacement or <use first point as displacement>: <Ortho on> <Isoplane Top> 30

Command: \_line

Specify first point:

Specify next point or [Undo]:

Similarly draw the isocircles of radius 22, 15, 28 at their respective position as mentioned in drawing

Command: trim

Select objects:

Command: erase

Select objects:

Command: \_arc

Specify start point of arc or [Center]:

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

Specify end point of arc:

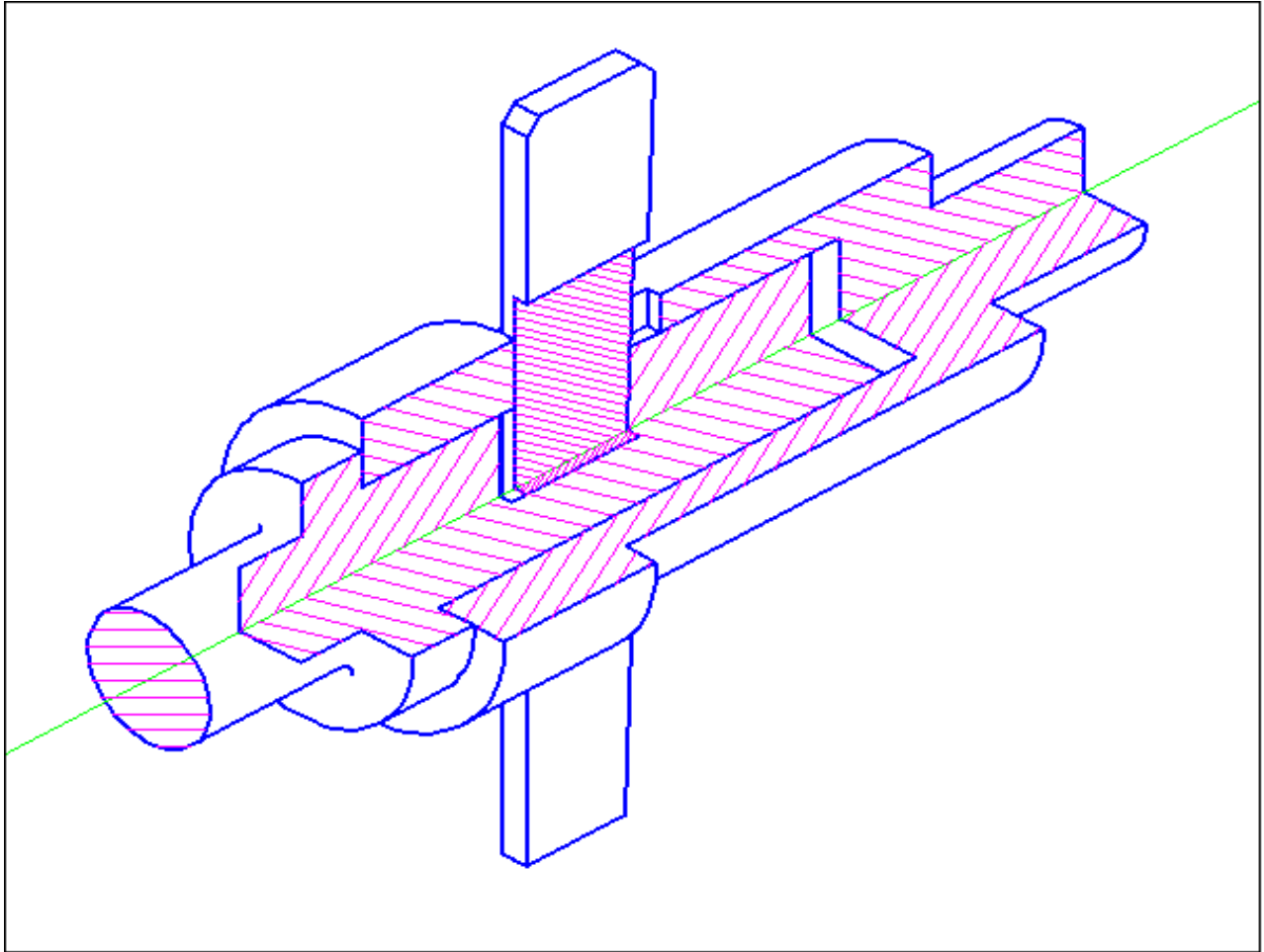
Command: hatch

Select internal point:

Specify angle

Command: save

**Quarter Sectional Isometric View of a Cotter-Joint  
(Exp.-04)**



**Exp. 5. Draw Different type's bolts and nuts with internal and external threading in Acme and Square threading standards. Save the bolts and nut as blocks suitable for insertion.**

**Procedure:-** List of commands for drawing bolts and nut.

- i) Polygon
- ii) Circle
- iii) Line
- iv) Line type (Center line, Hidden line etc.)
- v) Hatch
- vi) Block

- i) Polygon:- A regular polygon is a closed geometric figure with equal sides and equal angles. The number of sides varies from 3 to 1024.

Command: polygon

Command: Enter number of sides <4>: (Ex: 6)

Specify center of polygon or [Edge]: (Ex: 10, 10)

Enter an option [Inscribed in circle/Circumscribed about circle] <I>: I ( I or C )

Specify radius of circle: (Ex: 50)

- ii) Circle:-

Command: circle

Command: \_ Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: (Ex: 10, 10)

Specify radius of circle or [Diameter]: (Ex: 50 or d )

- iii) Line:-

Command: line

Command: Specify first point: (Ex: 10, 10)

Specify next point or [Undo]: (Ex: 100, 100)

- iv) Linetype:-

Command: linetype

Format > Linetype (It will display a Linetype Manager dialog box) > Load > select the required linetype from the given linetype list > ok.

v) Hatch:-

Command: bhatch

The Hatch command allows you to hatch a region enclosed within a boundary (closed area) by selecting a point inside the boundary or by selection the objects to be hatched.

Draw > Hatch (It will display a Boundary Hatch and Fill dialog box) > specify Hatch type > Hatch pattern > Angle > Scale > Choose pick point (select any point inside the object) or Select object > Choose Preview or Ok.

vii) Block:-

Command: \_block

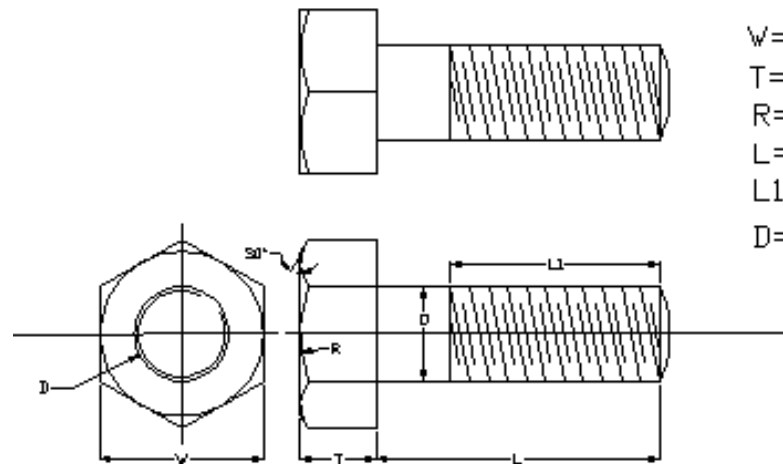
Draw > Block > Make (it will display a Block Definition dialog box)

- e) Specify the Block Name.
- f) Specify the insertion Base Point (This point is used as a reference point to insert the block by choosing Pick Point)
- g) Select object.
- h) OK.

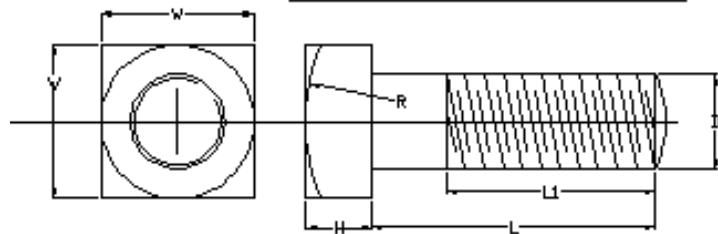
Command: Insert

Insert > Block > Select the Block Name from the drop down list > Ok.

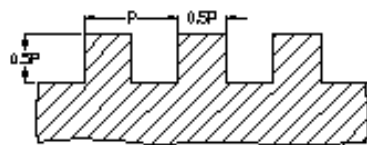
**Nut ,Bolt & Thread**  
**(Exp.-05)**



Hexagonal Headed bolt



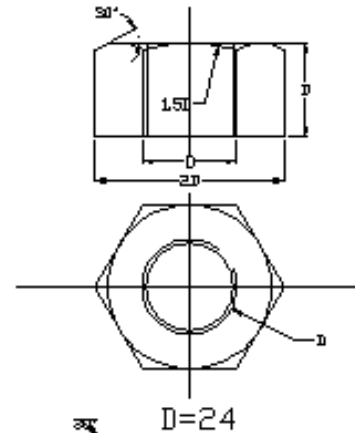
Square Headed bolt



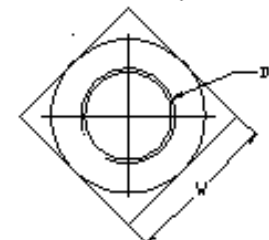
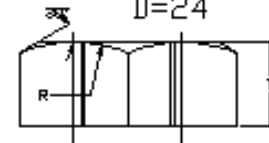
$P=24$   
Square Threads

$W=1.5D+3$   
 $T=0.8D$   
 $R=1.5D$   
 $L=3D$   
 $L_1=2D$   
 $D=24$

Hexagonal Nut

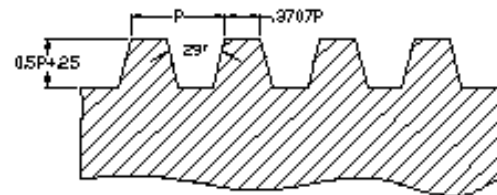


$D=24$



$T=0.9D$   
 $W=1.5D+3$   
 $R=1.5D$   
 $D=24$

Square Nut



$P=24$   
Acme Threads

**Exp. 6. Draw 3D models by extruding simple 2D objects, dimension and name the objects.**

**Procedure:-** List of commands for 3D models.

- i) 3D View
- ii) Region
- iii) Extrude
- iv) Move
- v) Text
- vi) Dimension
- vii) Shade

i) 3D View

View > 3D view > Choose any one of the view (Front/ Top/ Left etc.)

ii) Region

Draw > Region > Select object.

iii) Extrude

Draw > Solids > Extrude > Select object > Specify height > Specify angle.

iv) Move

Modify > Move > Select object > Specify base point > Specify next point.

v) Text

Draw > Text > Single Line Text > Specify Start Point > Specify height > Specify rotation angle > Enter Text.

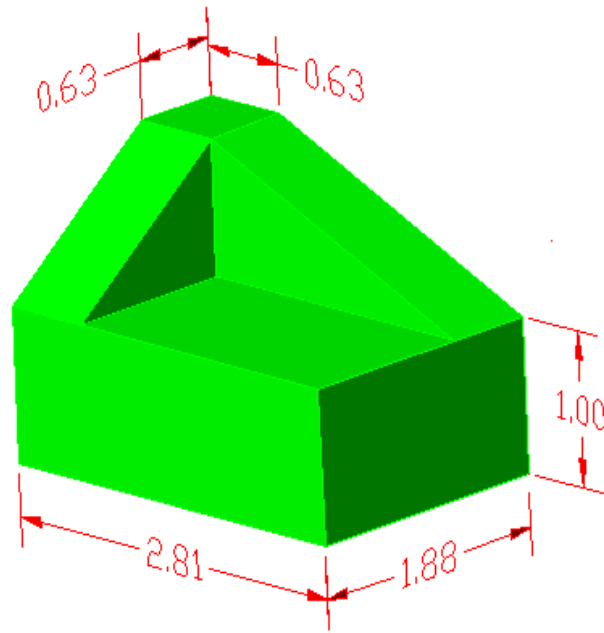
vi) Dimension

Dimension > Linear > Specify First Extension Line Origin > Specify Second Extension Line Origin.

vii) Shade

View > Shade > Select the required shade (Ex.- 2d wire frame/ Hidden/ Flat shaded/ Gouraud shaded, edges on etc.).

**3D Model**  
**(Exp.-06)**





**Exp. 7. Draw a Spiral by extruding a circle.**

**Procedure:-** List of commands for Draw the Spiral.

- i) Helix
- ii) Free Orbit
- iii) Circle
- iv) Sweep
- v) Visual Styles

i) Helix

Draw > Helix.

Command: Helix

Specify center point of base: 0,0

Specify base radius or [Diameter] <1.0000>: 10

Specify top radius or [Diameter] <10.0000>: 10

Specify helix height or [Axis endpoint/Turns/turn Height/tWist] <1.0000>: 50

ii) Free Orbit

View > Orbit > Free Orbit

iii) Circle

Draw > Circle > Center, Radius

Command: circle

Command: \_ Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: (Select any one end of the helix)

Command: Specify radius of circle or [Diameter]: (Ex: 50 or d )

iv) Sweep

Draw > Modeling > Sweep

Command: \_sweep

Select objects to sweep: (Select the circle)

Select sweep path or [Alignment/Base point/Scale/Twist]: (Select the helix)

v) Visual Styles

View > Visual Styles > Conceptual

Spiral  
(Exp.-07)

