



Dr. M.G.R.
EDUCATIONAL AND RESEARCH INSTITUTE
DEEMED TO BE UNIVERSITY



University with Graded Autonomy Status

(An ISO 21001 : 2018 Certified Institution)

Periyar E.V.R. High Road, Maduravoyal, Chennai-95. Tamilnadu, India.

RECORD NOTEBOOK

COMPUTER AIDED DESIGN AND ANALYSIS LAB
(BME18OL2)

2024-2025(ODD SEMESTER)

DEPARTMENT

OF

MECHANICAL ENGINEERING

NAME : SUGUMARAN S

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COURSE/BRANCH : B. TECH CSE – DS(AI)

YEAR/SEM/SEC : IV / VII / C



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BONAFIDE CERTIFICATE

Register No: 211191101150

**Name of Lab: COMPUTER AIDED DESIGN AND ANALYSIS
LAB – (BME18OL2)**

Department: MECHANICAL ENGINEERING

Certified that this is the Bonafide record of work done By Mr. SUGUMARAN S of IV Year B. Tech
CSE – DS(AI) **COMPUTER AIDED DESIGN AND ANALYSIS LAB (BME18OL2)** during the
year 2024-2025.

Signature of Lab-in-Charge

Signature of Head of Dept.

Submitted for the Practical Examination held on _____

Internal Examiner

External Examiner

Table Of Contents

EXP.NO	DATE	TITLE	PAGE.NO	STAFF SIGNATURE
1		INTRODUCTION TO CAD 2DDRAWING	3	
2		BASIC COMMANDS IN CAD DRAW, MODIFY AND DISPLAY	10	
3		DIMENSION ANNOTATION SYMBOLS WELDING, FINISHING, THREADING, TITLE BLOCK	13	
4		KNUCKLE JOINT 2D DRAWING	18	
5		GIB AND COTTER JOINT 2DDRAWING	19	
6		SCREW JACK 2D DRAWING	20	
7		FOOT STEP BEARING 2D DRAWING	21	
8		KNUCKLE JOINT 3D DRAWING	22	
9		PLUMMER BLOCK	23	
10		BOOLEAN OPERATIONS USING CAD	25	
11		STRUCTURAL ANALYSIS OF BEAM USING DIFFERENT BOUNDARY	26	

Introduction

CATIA V5 is a powerful software package yet has a relatively short learning curve. One of the reasons for the short learning curve is that it is fully Windows compatible and the processes are consistent across the workbenches, toolbars and tools. If you learn the basics of a particular workbench the same process can be used for more complex problems. Several tools are used in more than one workbench.

The lessons in this workbook present basic real life design problems (when possible) and the workbenches, toolbars, and tools required to solve the problem. The lessons present this information with step-by-step instructions. For every step there are numerous possible methods of accomplishing the same thing. The steps are to be used as guides to solving the design problem. You are encouraged to try new and different methods, find the method that works for you.

The CATIA Acronym

The CATIA acronym stands for: “**Computer Aided Three Dimensional Interactive Application**”. V5 stands for Version 5.

A Brief History of CATIA

CATIA was developed by Dassault Systemes in the early 1980's and quickly emerged as an industry leader. CATIA V5 is different from V4 and earlier versions because it was programmed to take advantage of the MS Windows capability while maintaining the UNIX power and stability. As a user, making the transition to CATIA V5 from CATIA V4 is a huge step. CATIA V5 does not resemble CATIA V4 at all; it is new. For Experienced CATIA V4 users often seem to struggle with the change at first, until they can let go of the old and embrace the new. The learning curve has been greatly reduced, while maintaining the power and significantly improving the flexibility and integration.

With CATIA V5 the designer is not limited by the software but is limited only by his/her own ability to put the software to work. CATIA V5 is the engineering package that will lead industry into the world of Product Life Cycle Management (PLM).

CATIA Version 5 Basic Concepts

Upon completion of this course the student should have a full understanding of the following topics:

- Managing models (opening, closing, saving, etc)
- Solutions, Workbenches and Toolbars
- Tools, Options
- Tools, Customize
- Manipulation
- View Toolbar

Setting Up

1. Start-up is usually very slow....wait patiently.
2. **Menu bar:** Tools/Options/Infrastructure/Part Infrastructure/Display Display in Specification Tree, check all buttons **ON**.
3. **Menu bar:** Tools/Options/Display/Performance/3D Accuracy and 2D Accuracy set to **0.01**. PressOK to save.
4. **Menu bar:** Tools/Options/Mechanical Design/Sketcher/Constraint Make sure 'Creates the geometrical constraints' and 'Creates the dimensional constraints' are checked **ON**.
5. *Create a Folder named as your student (Catia_zID) and put all your files for the tutorial in it which will be collected at the end of each class.*

6. Setting up steps 1-4 need to be done in every Tutorial Class.



Fig. 2: The screen that appears after closing the initial product file while starting CATIA

INTRODUCTION

1. Part Design: Creating positive features from a sketch
2. Part Design: Creating negative features from a sketch
3. Part Design: Dress Up features including Threads
4. Plumbing Part
5. Examination of Complex Part
6. Assembly: Manipulation and Constraints
7. Structural Joint
8. Mechanical Assembly

Background: CATIA, Version 5, Release 20, Dassault Systems

Acronym for: Computer Aided Three dimensional Interactive Application

Workbenches

1. Part Design workbench
2. Wireframe and Surface Design workbench
3. Assembly Design Workbench
4. Drafting workbench

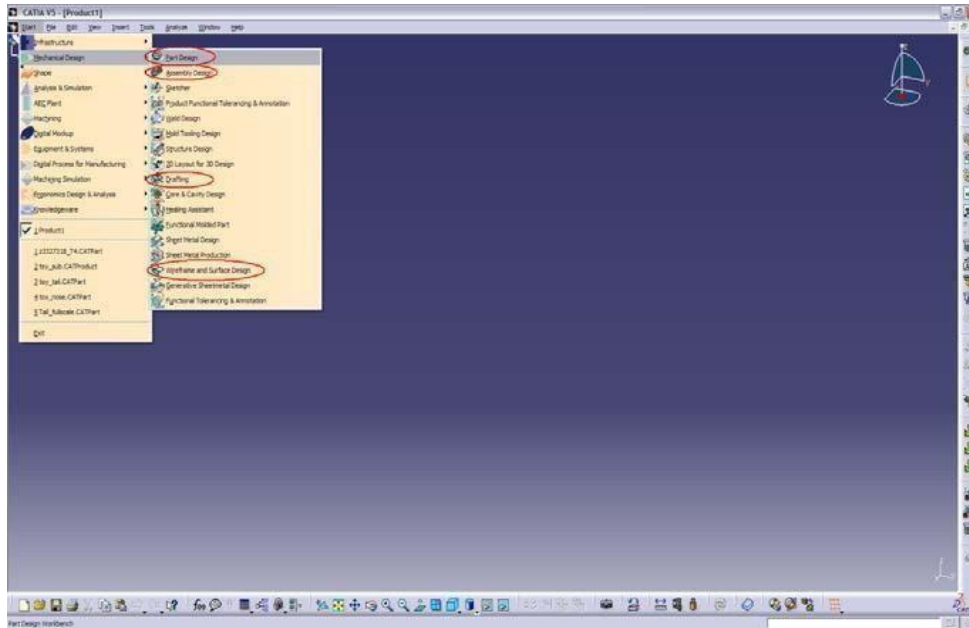


Fig. 1: Basic workbenches in CATIA

CAT Part

CAT Part is a file extension associated with all the files that are created in the **Sketcher**, **PartDesign**, and **Wireframe and Surface Design** workbenches.

CAT Product

CAT Product is a file extension associated with all the files that are created in the **Assembly Design** workbench.

CAT Drawing

CAT Drawing is a file extension associated with all the files that are created in the **Drafting** workbench.

Mouse Operations

1. **Select:** Left (L) Button Click. For multiple selections hold the CTRL key along with the Left Button
2. **Contextual menu:** Right (R) Button Click
3. **Object Manipulation:** Mouse wheel, alone or in combination with L or R button
 - Hold the middle button and move right or left to pan the object
 - Press the centre and the right and hold to rotate the object
 - To zoom hold the middle button and the CTRL key

Toolbars

1. Unpack and repack specific toolbars by dragging

2. Expand tools with sub-bars by selecting the black triangle (v)
3. If you accidentally close, restore the toolbars with **Menu bar:**
Tools/Customize/Toolbars/Restore All Contents and Restore position.

1. Standard Toolbar

This toolbar is common to all workbenches of CATIAV5.

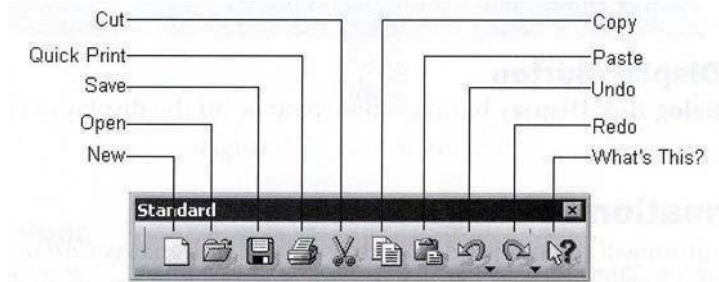


Fig. 3: The Standard toolbar

2. View Toolbar

The buttons in the View toolbar are used for manipulating the view of the model.

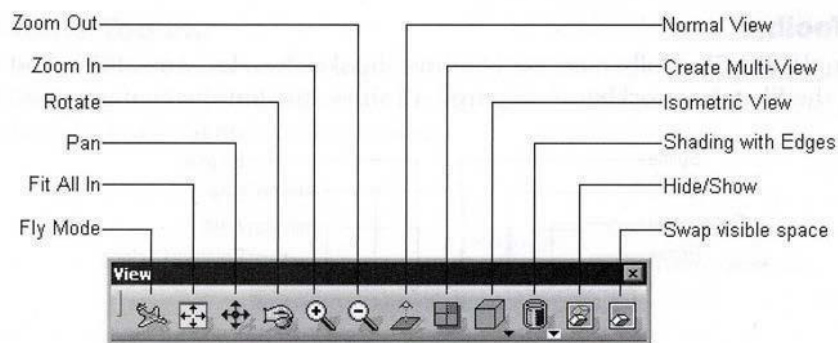


Fig. 4: The View toolbar

3. Sketcher Toolbar

The Sketcher button in the sketcher toolbar is used to invoke the **Sketcher** workbench. *After choosing the Sketcher button, select a plane, or a planar face to invoke the **Sketcher** workbench. You can select the plane and invoke the Sketcher or vice-versa.*



Fig. 5: The Sketcher toolbar

4. Profile Toolbar

The tools in the Profile toolbar are used to draw the sketches. It is one of the most important toolbars in the **Sketcher** workbench.

Fig. 6: The Profile toolbar

5. Constraint Toolbar

The tools in the Constraint toolbar are used to apply constraints to the geometric entities and assign *dimensions* to a drawn sketch.

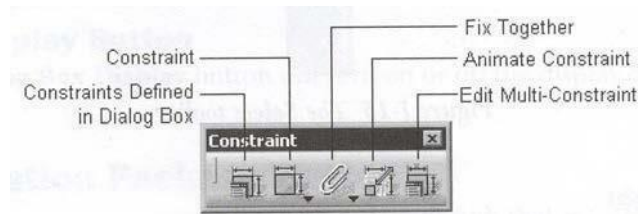


Fig. 7: The Constraint toolbar

6. Operation Toolbar

The tools in the Operation toolbar are used to *edit* the drawn sketches.

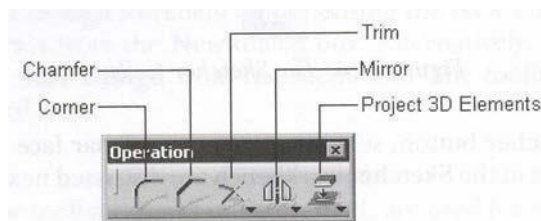


Fig. 8: The Operation toolbar

7. Sketch Tools Toolbar

The tools in the Sketch Tools toolbar are used to set the sketcher settings such as setting the snap, switching between the standard and construction elements, and so on.

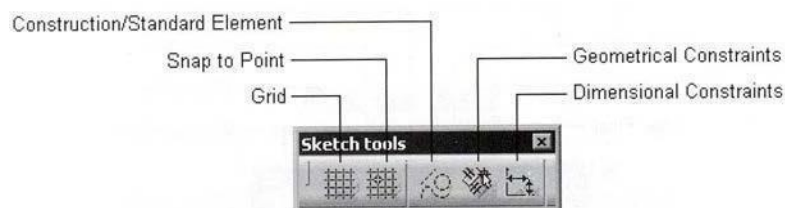


Fig. 9: The Sketch Tools toolbar

Once the basic sketch is complete, you need to convert it into a feature. Choose the 'Exit Workbench (icon on the right side of the toolbar, an arrow pointing up)' button from the

Workbench toolbar and switch back to the Part Design workbench.

The following toolbars of the **Part Design** workbench are discussed next.

1. Sketch-Based Features Toolbar

The tools in the Sketch-Based Features toolbar are used to *convert* a sketch drawn in the Sketcher workbench into a feature.

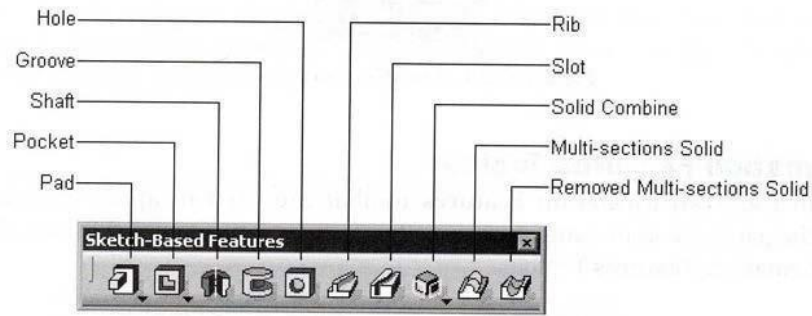


Fig. 10: The Sketch-Based Features toolbar

2. Dress-Up Features Toolbar

The tools in the Dress-Up Features toolbar are used to apply the dress-up features such as fillet, chamfer, shell, and so on.

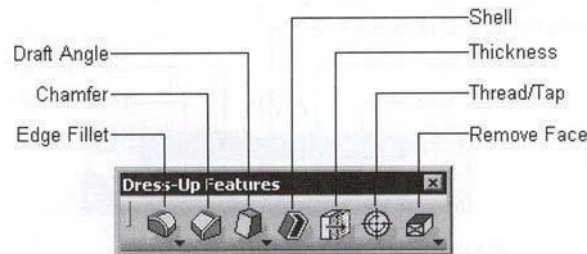


Fig. 11: The Dress-Up Features toolbar

3. Transformation Features Toolbar

The tools in the Transformation Features toolbar are used to apply the transformation features to the part such as moving, mirror, pattern, and so on.

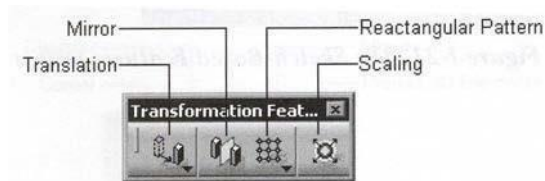


Fig. 12: The Transformation Features toolbar

Result: Hence the 2d drawing has been successfully implemented using Computer Aided Design Lab.

Aim: To implement draw, modify and display basic commands using CAD.

DRAW COMMANDS:

These AutoCAD commands are essential for creating precise geometric shapes. Mastering them can significantly increase your productivity in CAD design.

LINE: It is used to draw a straight line between two points. Simply type “LINE” into the command line and select the starting point. Then select the endpoint. You can also specify the length and angle of the line by typing in specific values.

CIRCLE: It draws a circle or an arc. To use this command, type “CIRCLE” into the command line and specify the center point and radius. You can also create an arc by specifying the start and end angles.

ARC: It draws an arc of a circle. To use this command, type “ARC” into the command line. Then specify the center point, radius, and start and end angles.

OFFSET: This creates a parallel line or curve at a specific distance from an existing line or curve. To use this command, type “OFFSET” into the command line and select the object you want to offset. Lastly, specify the distance.

TRIM: The TRIM command trims off parts of an object that extend beyond another object. To use this command, type “TRIM” into the command line and select the objects you want to trim. Then select the object that will be used as the cutting edge.

EXTEND: The EXTEND command extends an object to meet another object. To use this command, type “EXTEND” into the command line and select the object you want to extend. Then select the object that will be used as the boundary edge.

FILLET: The FILLET command rounds off the corners of two intersecting objects. To use this command, type “FILLET” into the command line, select the first object, and select the second object. Then specify the radius of the fillet.

CHAMFER: It creates a beveled edge between two intersecting objects. To use this command, type “CHAMFER” into the command line, select the first object, and select the second object. Then specify the distance and angle of the chamfer.

MIRROR: This command is used to create a mirror image of an object. To use this command, type “MIRROR” into the command line and select the object you want to mirror. Then select the mirror line.

ARRAY: The ARRAY command creates a pattern of multiple objects. To use this command, type “ARRAY” into the command line, select the object you want to duplicate, and specify the number of rows and columns. Then specify the spacing between the objects.

RECTANGLE: The RECTANGLE command draws rectangles. To use this command, type “RECTANGLE” in the command line. Specify the first corner point and then specify the opposite corner point.

POLYLINE: The POLYLINE command draws a series of connected lines and arcs. To use this command, type “POLYLINE” in the command line and specify the first point. Then continue adding points.

HATCH: The HATCH command fills an enclosed area with a pattern. To use this command, type “HATCH” in the command line and select the area to be hatched. Then specify the hatch pattern and scale.

DIMENSION: The DIMENSION command adds dimensions to your drawing. To use this command, type “DIMENSION” in the command line and select the objects to be dimensioned. Then specify the location and type of dimension.

MTEXT: The MTEXT command adds multiline text to your drawing. To use this command, type “MTEXT” in the command line and specify the insertion point. Then enter your text.

MODIFY COMMANDS:

ADDCENTER/ADC: This command allows you to insert or manage content such as blocks, hatch patterns, and xrefs. You can access the content from a single dialog box, making it easier to find and use.

ALIGN/AL: This command allows you to align objects with other objects in 2D and 3D. You can align objects based on different criteria such as endpoints, centers, and edges.

BASE: This command allows you to alter the base point of a drawing without changing its origin. You can move the base point to a different location. It can be useful when working with blocks or other objects.

B PARAMETER/PARAM: This command allows you to add a parameter with grips to a dynamic block definition. You can use this command to make a block more flexible.

BREAK: This command allows you to create a break (or gaps) in objects at one or two points. You can use this command to break up a line or object into smaller parts.

BSAVE: This command allows you to save the current block definition. You can use this command to save a block you created or modified.

BURST: This command allows you to explode a block but retain its attribute settings and layer definition. You can use this command to convert a block back into individual objects.

CHA/CHAMFER: This command allows you to add slanted edges to sharp corners of objects. You can use this command to add a bevel or chamfer the corner of a shape.

COPYBASE: This command allows you to copy an object according to a base point. You can use this command to make a copy of an object.

DIVIDE: This command allows you to divide objects into multiple equal parts. You can use this command to create evenly-spaced points or lines.

F/FILLET: This command allows you to add rounded corners to the sharp edges of objects. You can use this command to make the corners of a shape smoother.

LA/LAYERS: This command allows you to open the Layer Properties Manager Palette. It allows you to tweak the settings for layers. You can use this command to manage the properties of different layers.

MA/MATCHPROPERTIES: This command allows you to copy the properties of one object onto another. You can use this command to quickly apply the same properties to multiple objects.

MOCORO: This command allows you to move, copy, rotate, and scale an object in one command. You can use this command to perform multiple modifications to an object at once.

OVERKILL: This command allows you to remove overlapping or unnecessary objects from your work. You can use this command to clean up your drawing and improve its performance.

PURGE: This command allows you to remove unused objects from a drawing. You can use this command to remove any objects that are no longer needed in your drawing.

SC/SCALE: This command allows you to change the scale of an object. You can use this command to resize an object without changing its proportions.

SCALETEXT: This command allows you to change the scale of the text. You can use it to resize text to fit a specific space.

TEXTFIT: This command allows you to alter the size of a piece of text. You can use it to resize text to fit a specific area.

TEXTTOFRONT: Brings annotations to the front.

TORIENT: Changes the orientation of the text.

TR/TRIM: Trims a shape or line.

UNITS: Alters the unit settings of your drawing.

X/EXPLODE: Breaks up an object into its individual components—i.e., a polyline into simple lines.

XBIND/XB: Binds the definitions of named objects in an xref.

DISPLAY COMMANDS:

ZOOM: Increases or decreases the apparent size of objects in the current viewport. **Choose, Click, Type – All, Extents, Previous, Window, Number, Number X, Center, Dynamic.**

PAN: Shifts the location of a view. **Choose, Click, Type. REDRAW AND REGEN:** Redraw refreshes the current view

BLIPMODE: Controls the display of marker blips. When Blip mode is ON, a temporary mark in the shape of a plus sign (+) appears where points are specified. BLIPMODE is OFF by default.

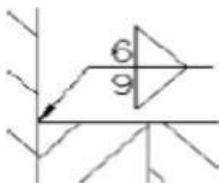
Result: Hence the basic commands DRAW, MODIFY and DISPLAY have been successfully implemented using CAD.

DIMENSION ANNOTATION SYMBOLS WELDING, FINISHING, THREADING, TEXT BILL OF MATERIALS, TITLE BLOCK

Aim: To implement annotation symbols using CAD.

WELDING SYMBOL:

1. Click Annotate tab Symbols panel Welding Symbol.
2. Select an object to attach the weld symbol to.
3. If you attached the symbol to a line, in the drawing area, specify the start point for the leader. If you attached the object to an arc, circle, ellipse or spline, skip to the next step.
4. Specify one or more points to define the vertices of the leader and press ENTER.
5. In the Symbol tab, specify the general appearance of the symbol as required.
 To align the position of the arrow side data, the identification line, or direction of the symbol tail to the left, in the Options section, select Flip symbol.
 To stagger the alignment of intermittent fillet welds (or ANSI edge welds) on both sides, in the Options section, select Stagger.
 To add the all-around designation, click the All-Around button.
 To add a tail to the welding symbol, click the Add Note Tail button. To add a process from a predefined list:
 - a. Click Add Process.
 - b. To prefix process numbers with the statement *Process ISO 4063*, select the Prefix Process number with "Process ISO 4063-" check box.
Note: This option is not available if the current drafting standard is ANSI.
 - c. Double-click the processes to add.
 - d. Click Close.
 To edit processes by entering details using the keyboard:
 - a. In the box beneath Add Process, click at the position you want to edit.
 - b. Enter the process details.
 To delete a process, select it and press the DELETE key. To draw a box around the note, select Add note frame.
Note: This button is not available if the current drafting standard is ANSI.
6. To specify arrow side data:
 - a. Click the Weld Type button on the arrow side and select a weld symbol. The available options change to match the symbol you select.
 - b. Specify parameters.
7. To specify other side data:
 - a. Click the Weld Type button on the other side and select a weld symbol. The available options change to match the symbol you select.
 - b. Specify parameters.
8. Click OK.
Note: To clear all data in the Weld Symbol dialog box, click Clear.



FINISHING SYMBOL:

1. Click Insert > Surface Finish. The GET SYMBOL menu appears.
2. Select one of the following:
 - Name—Selects a symbol from the SYMBOL NAMES menu containing a list of symbols that are currently in the drawing.
 - Pick Inst—Selects a symbol by picking an instance of the symbol in the drawing. Retrieve—

Retrieves a symbol definition from disk by picking the name from the name list

menu. Move along the directory tree, when needed, to select the symbol.

3. Select one of these from the INST ATTACH menu:

Leader—Creates the symbol with a leader. Select a command from the ATTACH TYPE menu. Entity—Attaches the symbol to an entity.

Normal—Places the symbol normal to an entity. The symbol is positioned with the vertical reference pointing up. A yellow arrow, normal to the entity, indicates the symbol positioning.

No Leader—Creates a symbol without a leader.

4. Respond to the prompts for the commands you have selected.

5. Specify a value for the surface roughness. The symbol is placed on the drawing with the size relative to the default text height.

To modify the value of a surface finish, use Value on the MODIFY DRAW menu.

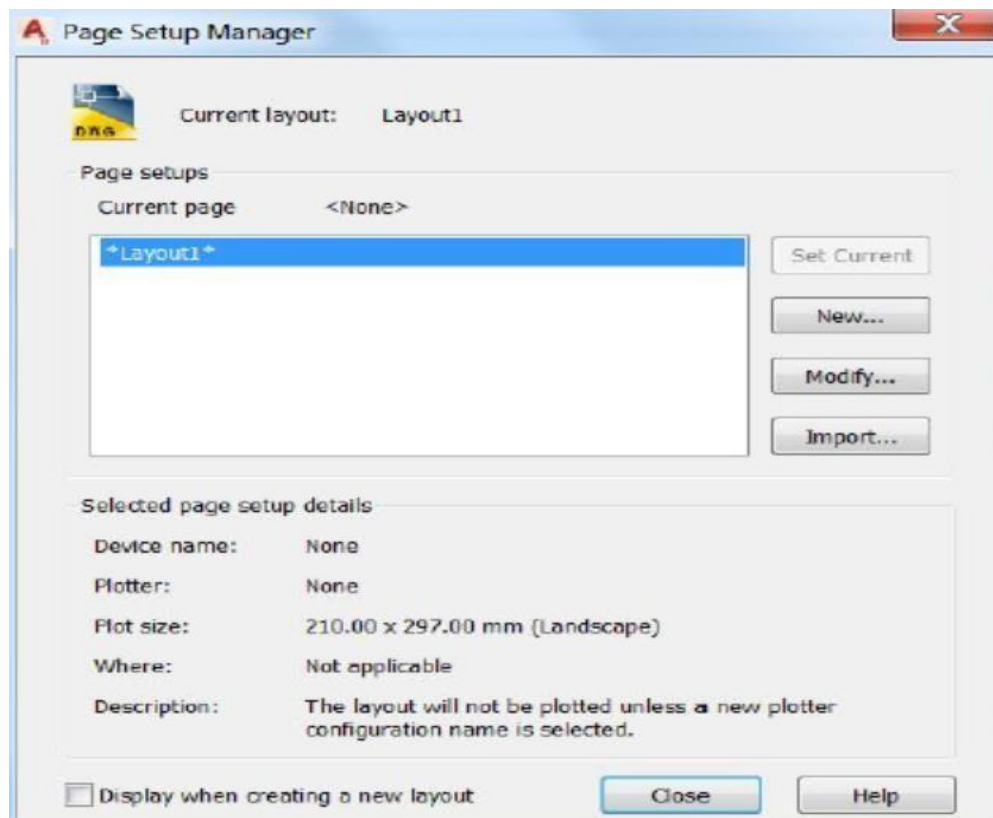
THREADING SYMBOL:

1. Open the file named Tutorial 4 YourProjectName.dwg. We are going to cut M10 x 1.5 threads. Check the Machinery's Handbook for complete information on thread forms.
2. Turn off all layers except the Rod Sketch and Rod Solid layers and make the Rod Solid layer the active layer. Revolve the Rod Sketch.
3. Turn on the External Threading Tool and Helix Path Layers, Make the Helix Path layer the active layer. Select the Helix tool and then select the Center snap point of the end of the rod.
4. Then select the intersection of the corner of the cutting tool as the base radius. Press enter to accept the same distance at the top radius.
5. RMB (Right Mouse Button) select turn Height and enter a height of 1.5 for the pitch of the thread.
6. And then track along the axis going a couple of turns beyond the end of the rod.
7. Start the sweep tool and select the cutting tool sketch. Then RMB and select Alignment and No. Select the Helical path.
8. If you are going to rapid prototype the part via an stl file you might want to make a copy of the thread cut shifted along the axis the desired clearance distance.
9. Subtract the Sweep and make sure the rod is on the Rod Solid Layer and turn off the other layers.
10. Turn off the layers except the Nut Sketch and Nut Solid Layers and make the Nut, Solid layer the active layer.
11. Extrude the hexagon to the center of the second circle.
12. Press pull the hole through the part.
13. Extrude the larger circle that is tangent to the hexagon with a taper of 45 deg.
14. Use the intersection tool to derive the intersection between the two solids. This cuts a chamfer on the nut.
15. Repeat the same procedure on the other side.
16. Add a 0.75mm chamfer to both ends of the hole in the nut.
17. Turn the internal threading tool layer on. Create a new helix path for the internal threading tool.
18. Sweep the cut tool and subtract as was done with the external thread.
19. That completes the internal and external cur threads or the use of threading symbol.

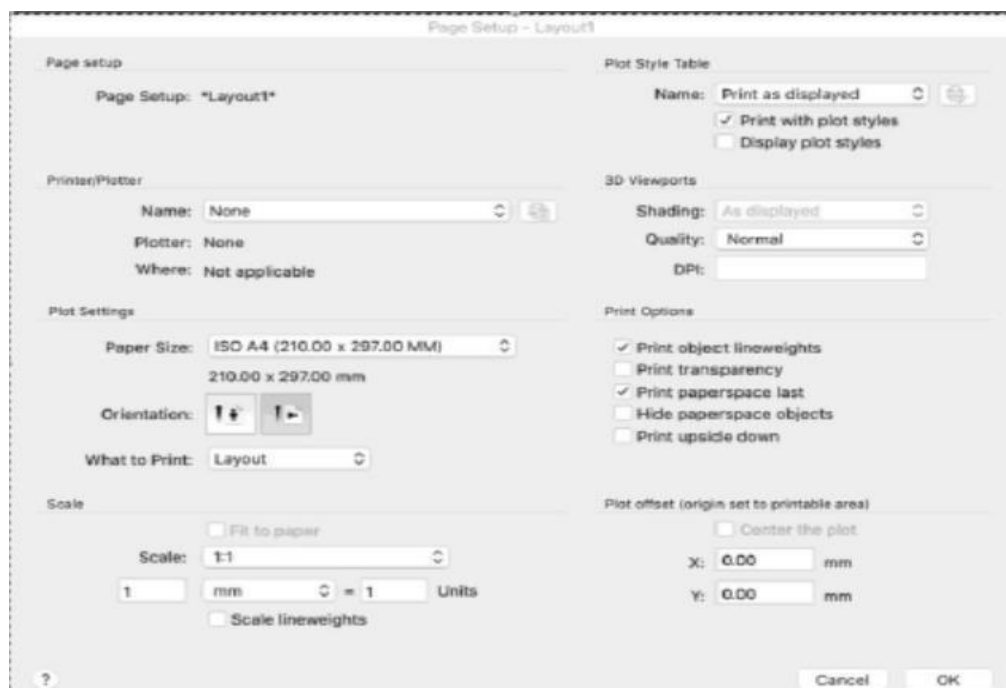
TITLE BLOCK:

1. First, locate or download a Title Block template. There are some basic templates that are already included in the AutoCAD or from the Download Finder Page.
2. Open up a blank drawing and click on the "Layout" tab or Right-click on the Layout 1 and select Page Setup Manager to enter your plot settings.

For Windows:

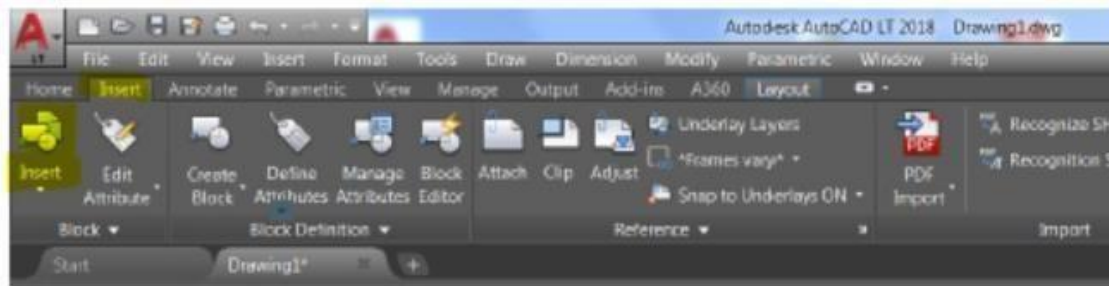


For Mac:

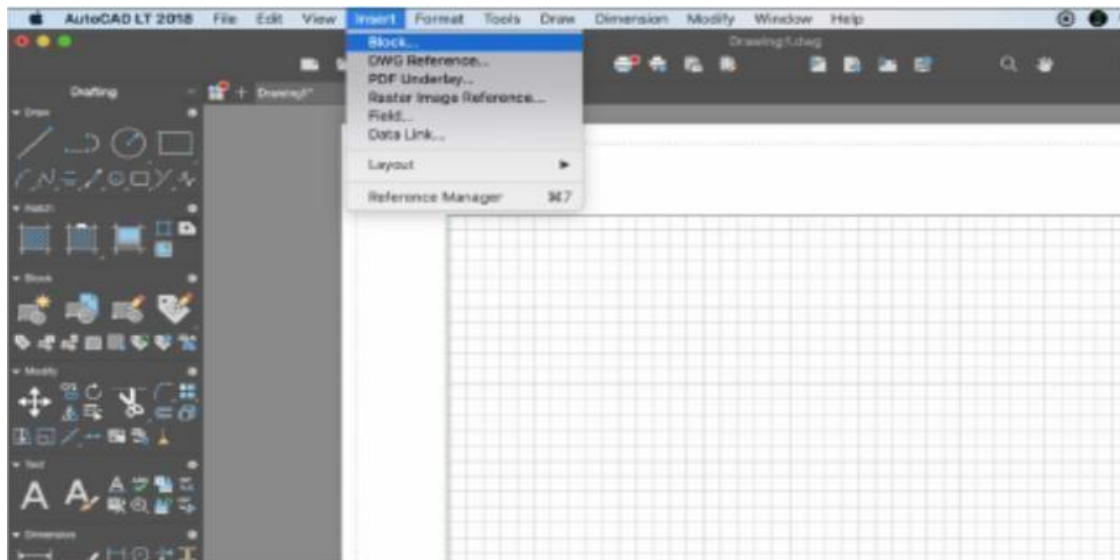


3. Click “Modify” to open the dialog box. The most important thing to pay attention to is the paper size, so make sure to set it up so that it matches the size of the Title Block template.
Select “Landscape” on the Drawing Orientation panel and hit “OK”.
4. Click on the “Insert” tab and then on the “Insert” button on the far left side of the ribbon.

5. For Windows:

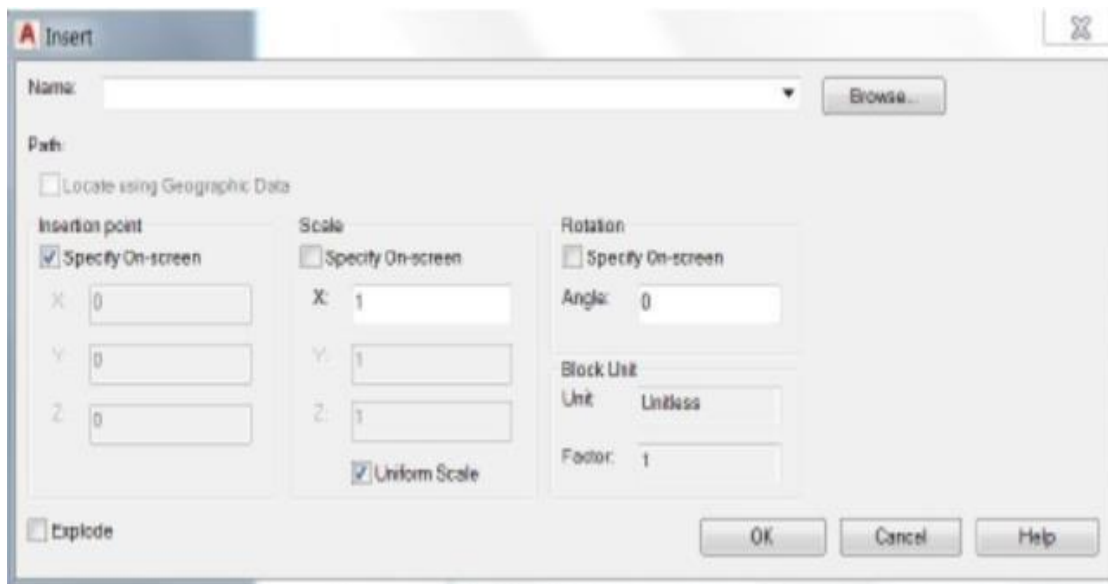


For Mac:



From the pop up menu, click on Browse and select your template. Leave the check boxes as they are and only check the “Specify on screen” box. Click “OK”.

For Windows:



For Mac:



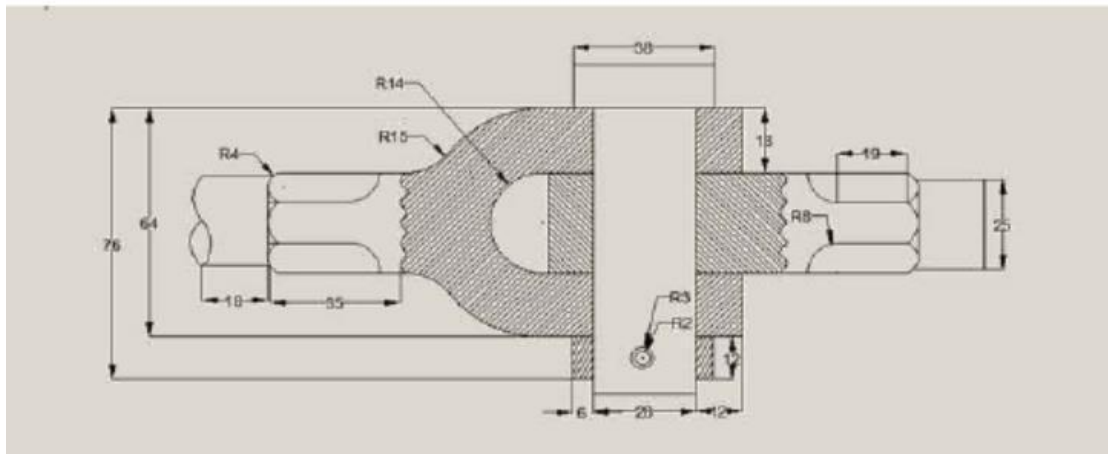
6. Position the Title Block. Note that the dashed line represents the print border, so position the Title Block within it.
7. Save the layout template. See the following documentation To Save a Layout

Result: Thus annotation symbols have been implanted using CAD successfully.

Exp No: 4

KNUCKLE JOINT 2D DRAWING

Aim: To develop the part drawing of Knuckle Joint in the orthographic representation.



LIST OF COMMANDS:

Line - To draw a line of required dimension.

Rectangle - To draw rectangle shape with specified length and width

Circle - To draw a circle of required radius.

Poly line - To draw multiple lines of required dimensions.

Arc - To draw arc of required dimensions.

Trim - To remove unwanted or excess dimensions of the element.

Zoom - To enlarge or reduce the view of component.

Fillet - To join sharp corners with a curve.

Mirror - To reflect the image on other side of the object.

Erase - To erase any object.

Hatch - Used to hatch enclosed area.

Join - To join two objects.

Dimension - To specify the product size using the annotations of dimension tool

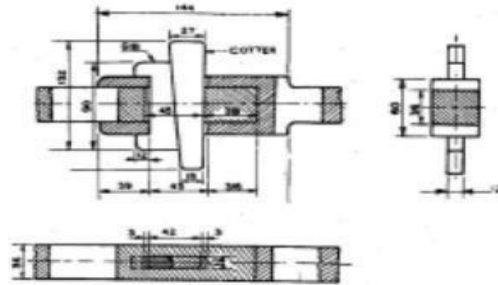
Result: Thus the part drawing of the knuckle joint is drawing in orthographic view.

Exp No: 5

GIB AND COTTER JOINT 2D DRAWING

Aim: To develop the part drawing of Gib and Cotter Joint in the orthographic representation.

Gib and cotter joint or rectangular rods



LIST OF COMMANDS:

Line - To draw a line of required dimension.

Rectangle - To draw rectangle shape with specified length and width

Poly line - To draw multiple lines of required dimensions.

Trim - To remove unwanted or excess dimensions of the element.

Zoom - To enlarge or reduce the view of component.

Fillet - To join sharp corners with a curve.

Mirror - To reflect the image on other side of the object.

Erase - To erase any object.

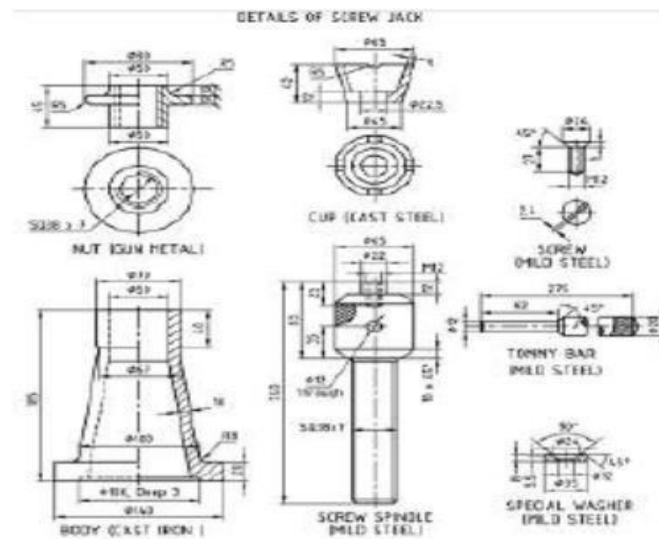
Hatch - Used to hatch enclosed area.

Join - To join two objects.

Dimension - To specify the product size using the annotations of dimension tool

Result: Thus the part drawing of the Gib and Cotter joint is drawing in orthographic view.

Aim: To develop the part drawing of Screw Jack in the orthographic representation.



LIST OF COMMANDS:

Line - To draw a line of required dimension.

Rectangle - To draw rectangle shape with specified length and width

Circle - To draw a circle of required radius.

Poly line - To draw multiple lines of required dimensions.

Arc - To draw arc of required dimensions.

Trim - To remove unwanted or excess dimensions of the element.

Zoom - To enlarge or reduce the view of component.

Fillet - To join sharp corners with a curve.

Mirror - To reflect the image on other side of the object.

Erase - To erase any object.

Hatch - Used to hatch enclosed area.

Join - To join two objects.

Dimension - To specify the product size using the annotations of dimension tool

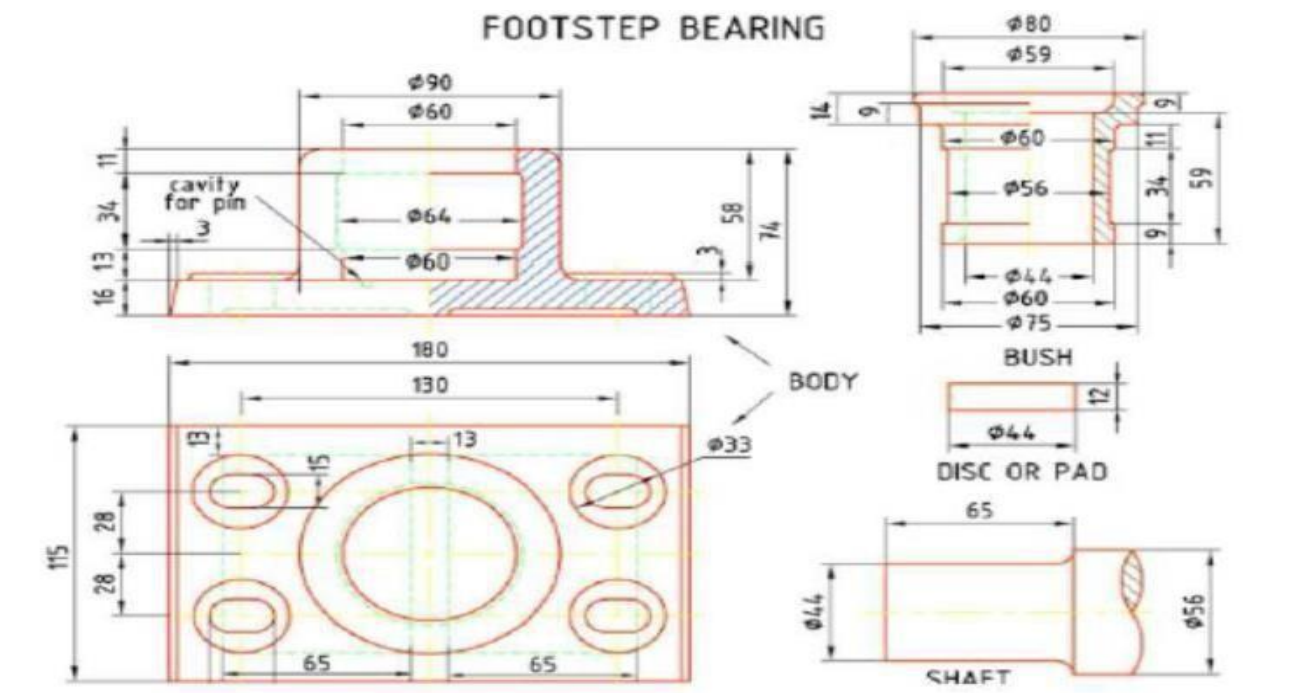
Offset - To draw the image of the object at required distance.

Break - To cut the object to required dimensions.

Result: Thus the part drawing of the Screw Jack is drawing in orthographic view.

FOOT STEP BEARING 2D DRAWING

Aim: To develop the part drawing of Foot Step Bearing in the orthographic representation.



LIST OF COMMANDS:

Line - To draw a line of required dimension.

Rectangle - To draw rectangle shape with specified length and width

Poly line - To draw multiple lines of required dimensions.

Trim - To remove unwanted or excess dimensions of the element.

Zoom - To enlarge or reduce the view of component.

Fillet - To join sharp corners with a curve.

Mirror - To reflect the image on other side of the object.

Erase - To erase any object.

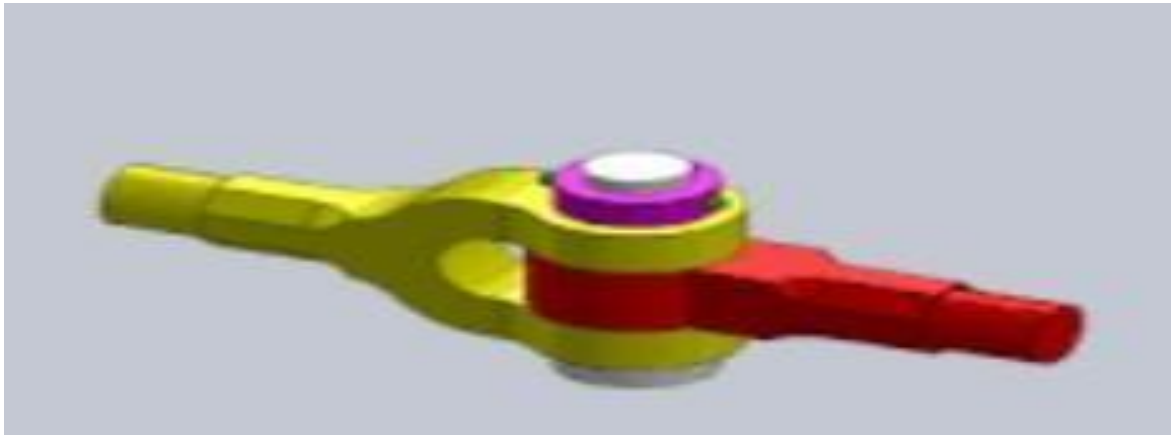
Hatch - Used to hatch enclosed area.

Join - To join two objects.

Dimension - To specify the product size using the annotations of dimension tool

Result: Thus the part drawing of the Screw Jack is drawing in orthographic view.

Aim: To develop the part drawing of Knuckle Joint from orthographic to isometric representation.



PROCEDURE:

1. Create the parts using Extrude and Revolve commands
2. For constructing holes and cutout, use hole command and cutout commands
3. After constructing each part save it as a separate part file with extension. par.
4. Assembly the various parts using the various assembly constrains (planer, design,mate, axial align, connect etc).
5. Save the assembly as a file with extension .asm.

Result: Thus the part drawing of the Knuckle Joint from Orthographic to Isometric representation is drawn successfully.

Aim: To develop the part drawing of Plummer Block from isometric to orthographic representation.

PROCEDURE:

- The drawings of Body, Cap, Bearing top & Bottom half, Nuts and shaft are studied
- 3D models of Body, Cap, Bearing top & Bottom half Nuts and shaft are created using Pro-E software.
- The Assembly of Plummer block was created as per the drawing specification.
- Detail all the components of the assembly as per the drawing standards.

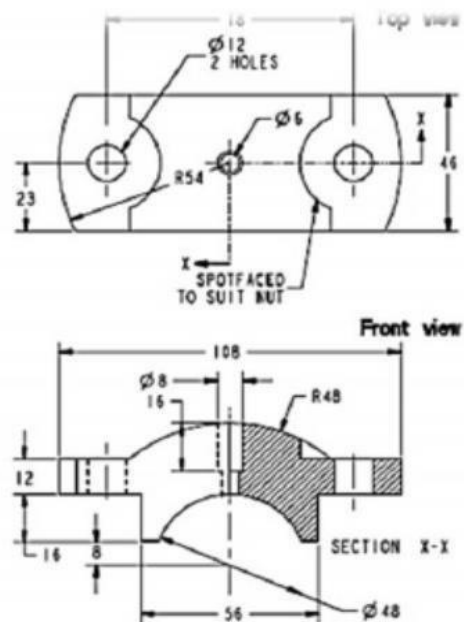
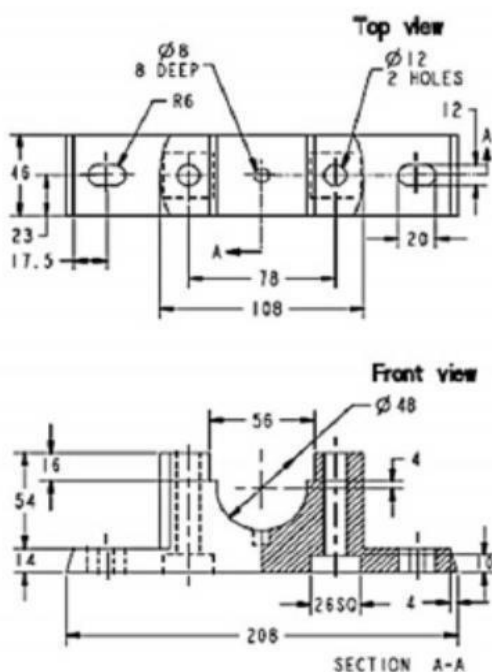
COMMANDS USED:

Sketcher Commands: Line, Circle, Arc, Fillet, Trim, Smart Dimension, Relations, Show, and View

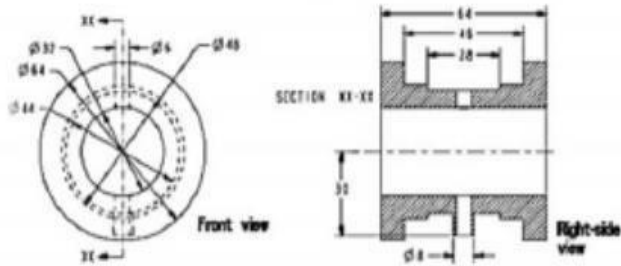
Features Commands: Extrude and Cut, Revolve, Fillet/Round, Chamfer, Hole - Simple, Pattern Fastening Features

Assembly Commands: Insert, Component, Existing Part/Assembly

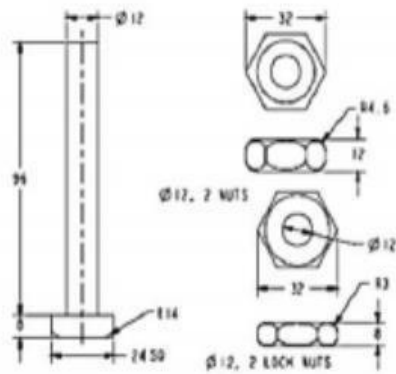
Mating Commands: Coincident, Concentric, Distance



Dimensions for Casting



Dimensions for the Brasses



Dimensions for the components of the assembly

Dimensions for Cap

S. NO.	NO. OFF	PART'S NAME	MATERIAL
1	1	BODY	CAST IRON
2	1	CAP	CAST IRON
3	1	BRASSES	CUN NITAL
4	2	SQUARE HEADED BOLT	MILD STEEL
5	2	NUT	MILD STEEL
6	2	LOCK NUT	MILD STEEL

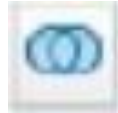




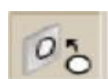
The Bill Of Material for Pedestal Bearing

Result: Thus the part drawing of the Plummer Block from isometric to orthographic representation is drawn successfully.

Exp No: 10**BOOLEAN OPERATIONS USING CAD**

Aim: To implement Boolean Commands using CAD. Boolean operation was named after George Boole, who first defined an algebraic system of logic in the mid-19th century.

Working in 3D usually involves the use of solid objects. At times you may need to combine multiple parts into one, or remove sections from a solid. AutoCAD has some commands that make this easy for you. These are the Boolean operations as well as some other helpful commands for solids editing.

CONTENT	INPUT	ICON	DESCRIPTION
UNION	UNION/UNI		Joins two or more fields into creating one based on the total geometry of all
SUBTRACT	SUBTRACT//SU		Subtracts one or more solids from another creating a solid based on the remaining geometry.
INTERSECT	INTERSECT/IN		Creates a single solid from one or more solids based on the intersected geometry
EXTRUDE FACE	SOLID EDIT		Allows you to increase the size of a solid by extruding out one of its faces
SLICE	SLICE		Slices a solid along a cutting plane
3D ALIGN	3D ALIGN		Aligns two 3D Objects in 3D Space.

Result: Thus Boolean Commands have been successfully implemented using CAD.

Exp No: 11

STRUCTURAL ANALYSIS OF BEAM WITH DIFFERENT BOUNDARY

Aim: To perform analysis of Beam structure with different boundary.

The following sequence lists the steps involved in defining a continuous span line beam, and analyzing it to produce dead load effects, temperature and shrinkage effects and envelopes of moving load effects. Links are included to the relevant help topics. More complete step by step guidance is contained in the [Examples](#) documentation.

Step 1: Start new Project

In the [Main Menu](#) select **File | New | Blank Project**

In the Main Menu select **Data | Structure Type | Line Beam**

Step 2: Add Design Beams

In the [Navigation Pane](#) select the **Design Beams** group.

Click on the **Add** button and either define a new [Design Beam](#) of required type or add an existing Design Beam. If adding an existing beam, this beam can remain as a linked file (useful if used in more than one project) or embedded in the project by clicking on the [Break Link](#) button.

For new Design Beam creation refer to the geometry steps of the Data Preparation Overview for [Steel Composite beams](#), [Prestressed beams](#), or [Reinforced Concrete beams](#).

Step 3: Define the Span details

In the Navigation Pane select the **Structure Definition** group.

Click on the **Structure Geometry** icon and populate the [Line Beam Geometry](#) form.

Step 4: Assign the Design Beams to the relevant Line Beam Spans

Click on the [Active Group Function](#) **Create Section and Beam Groups** to create one Structure Property group for each design beam.

Click on a **Structure Property** beam group item.

In the graphics window select a gray (unassigned) span (Note the 'Select' drop-down should be at its default Create setting). Green spans are already assigned to other beams.

Warnings are given if appropriate and the Design Beam is assigned to the span (which then becomes red).

Make any edits to the [Structure Properties: Beam](#) form and click **OK** to save the assignment.

Continue until Design Beams are assigned to all spans.

Step 5: Generate Loads and Analyze

In the Main Menu select **Data | [Automated Loading](#)**, and click on Analyze

Step 6: Analyze for Dead Loads, Temperature effects and Shrinkage / Creep effects

On the same form, click on the Dead and SDL Loading tab, and Analyze

Step 7: Transfer Results to the Design Beams for design / code checking

On the same form, click on the **Transfer Beam Load** button, and select a beam by pointing and clicking on the associated span. Then complete the Transfer Mapped Results form for assigning Analysis results to Design Beam effects, and click the **Transfer to Design Beam** button (or the **Save to Design Beam file** button if the Design Beam is linked to the project).

Result: Thus analysis of structure have been performed using different boundary on CAD.