

CMR COLLEGE OF ENGINEERING & TECHNOLOGY

Approved by AICTE, New Delhi

And

Affiliated by JNTU, Hyderabad

MECHANICAL ENGINEERING DEPARTMENT



LAB MANUAL

CAD/CAM LAB

Prepared by

Faculty Members of Mechanical Engineering Department

(2011 – 12)



CMR COLLEGE OF ENGINEERING & TECHNOLOGY

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CMR COLLEGE OF ENGINEERING AND TECHNOLOGY
KANDLAKOYA (v), MEDCHAL ROAD, HYDERABAD

DEPARTMENT OF MECHANICAL ENGINEERING

List of experiments

CAD/CAM Lab

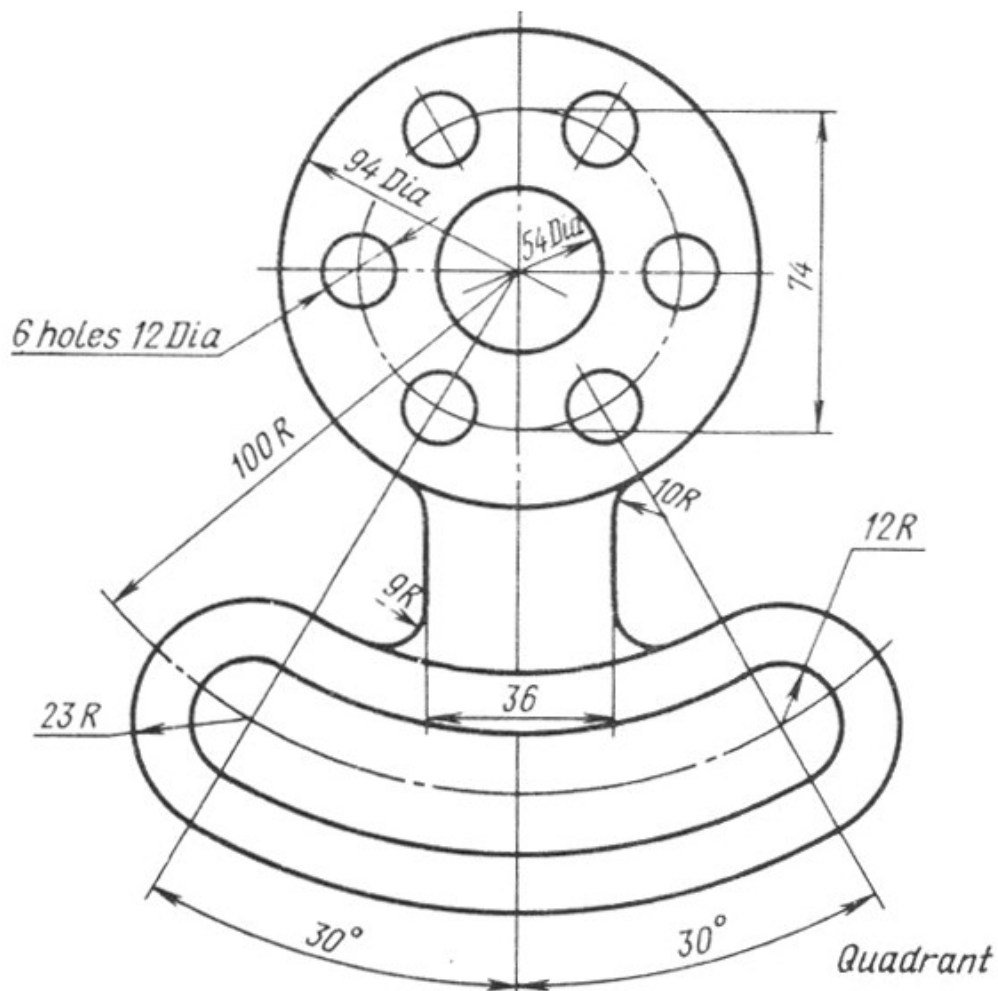
1. AUTOCAD-2007
2. SOLID WORKS-2006
3. EDGE CAM 11.0
4. CATIA V5 R19
5. ANSYS 10.0
6. CNC TURNING
7. CNC MILLING
8. CONTENT BEYOND SYLLABUS

Auto CAD Exercise – 1

Aim: To create a 2D view of the given diagram using Auto CAD.

Procedure:

1. Type limits in command menu & set value to 297,290.
2. Change the units to millimeters from inches and also precision to 0 by clicking format -> units ->ok.
3. To set the paper size type zoom -> enter and type a -> enter in Command bar.
4. Draw the 3 concentric circles with diameters 94, 74 & 54
5. Draw the two axis lines from centre of circles
6. Draw the vertical line from the centre of circle
7. From the modify tool bar, use the array command to draw the 6holes with 12dia from centre of circles
8. Now draw the 30degree line by use the vertical line
9. Then mirror the 30degree line, with vertical line
10. Again draw the concentric of radius 100 from centre of circle
11. From the modify toolbar, use the offset command to draw the 12 & 23 distance circle.
12. Draw 2 circles. With radius 23 & 12 on the 100R circle where the 30degree line co-inside.
13. From modify toolbar, mirror these circle to represent the another side.
14. And offset vertical line from centre of circle with a distance both side of vertical line.
15. From modify toolbar, use the fillet command to represent fillet of radius 10 & 9 to the offset line
16. Trim the unwanted lines to get required 2Ddraw



Result: Hence the required 2D diagram is created using Auto CAD.

Viva Questions

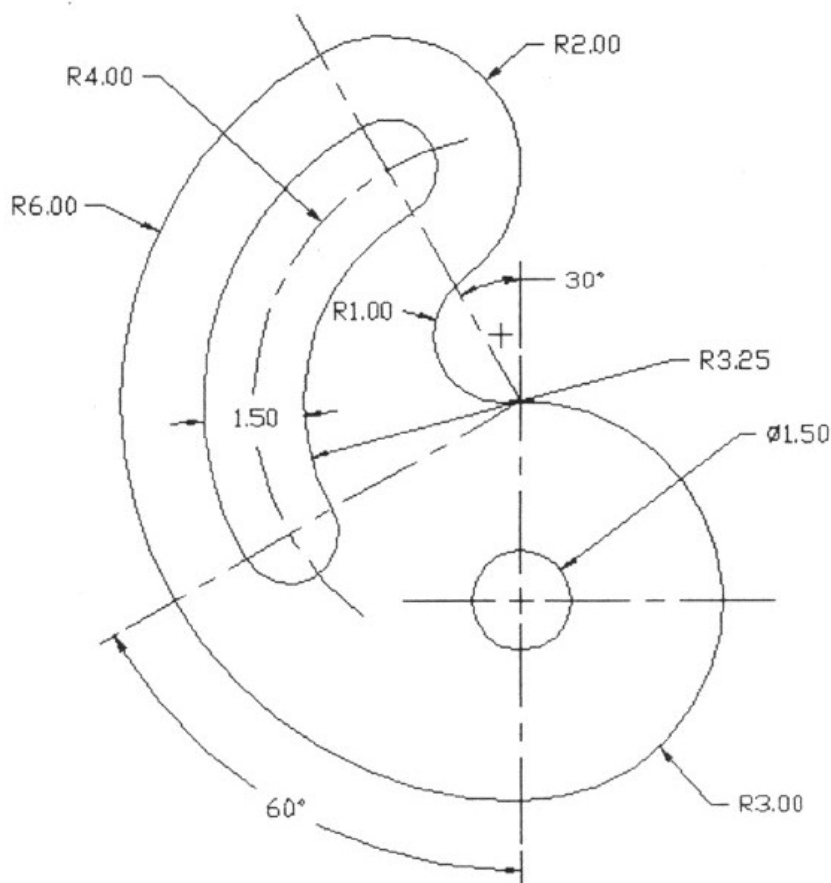
1. What is CAD?
2. What is the difference between Pan and Zoom?
3. What is the difference between line and pline?
4. What is the difference between chamfer and fillet?
5. By how many ways can you draw a circle?

Exercise – 2

Aim: To create a 2D view of the given diagram using Auto CAD.

Procedure:

1. Type limits in command menu & set values to 200,200.
2. Change the units to millimeters from inches and also precision to 0 by clicking format -> units -> ok.
3. To set the paper size type zoom -> enter and type a -> enter in command bar
4. Draw 2 lines as the axes and draw concentric circles of specified diameter.
5. Draw a ray of 30 degrees angle to the +ve Y axis as shown in the figure and a ray of angle 60 degrees to the -ve Y axis as shown in the figure.
6. Draw concentric circles from the point where the circle meets the positive Y axis as show.
7. Now trim the circles to get appropriate shape.
8. Continue with the design until the AutoCAD drawing his complete.
9. Give the dimensions from the dimension tool bar as in diagram.



Result: Hence the required 2D diagram is created using Auto CAD.

Viva Questions

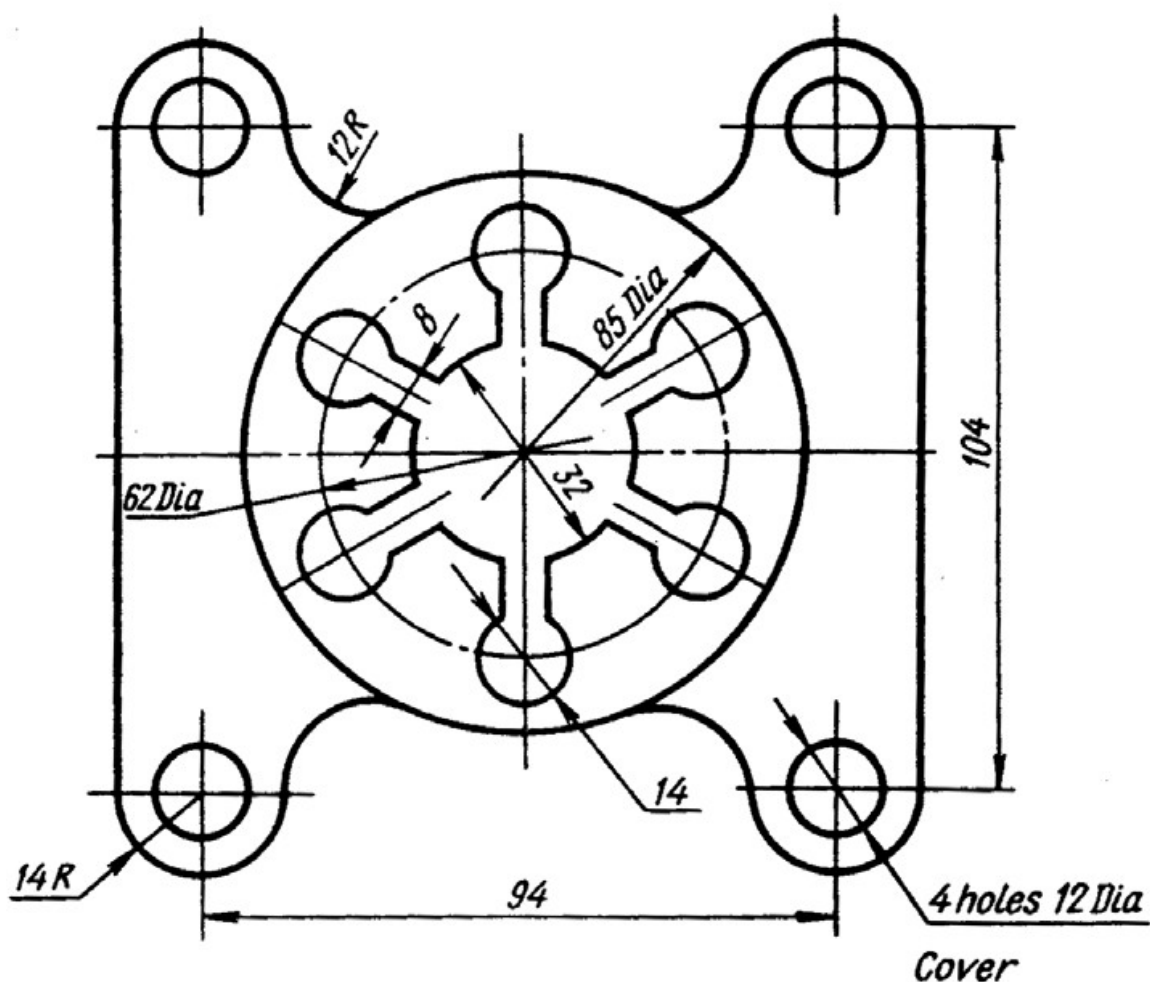
1. What is the difference between line and pline?
2. What is the difference between chamfer and fillet?
3. What are different ways to draw a circle?
4. What are the possible ways to draw an arc?
5. How to extend a line?
6. What is the procedure to enter into AutoCAD?

Exercise – 3

Aim: To create a 2D view of the given diagram using Auto CAD.

Procedure:

1. Type limits in command menu & set values to 45,45
2. Change the units to millimeters from inches and also precision to 0 by clicking format -> units -> ok.
3. To set the paper size type zoom -> enter and type a -> enter in command bar
4. Draw the 3 concentric circles of diameters 85 ,62,32
5. Draw the 2 axes lines from the centre of the circles
6. Draw the circle with 14dia on 62dia of circle and offset of the vertical line with distance 4 to both sides of the vertical line
7. Then trim the unwanted lines
8. Use the array command from modify tool bar to represent the 6 holes with 14 dia of centre of the circles
9. Offset the vertical and horizontal axes with 47 and 52 distance
10. And draw the 2 circles with 14 radius and 12 dia at coincide of the offset axes
11. From the modify tool bar select the fillet command to represent the 12R fillet
12. Then mirror this to require the 2D drawing
13. Finally trim the unwanted lines and circles



Result: Hence the required 2D diagram is created using Auto CAD.

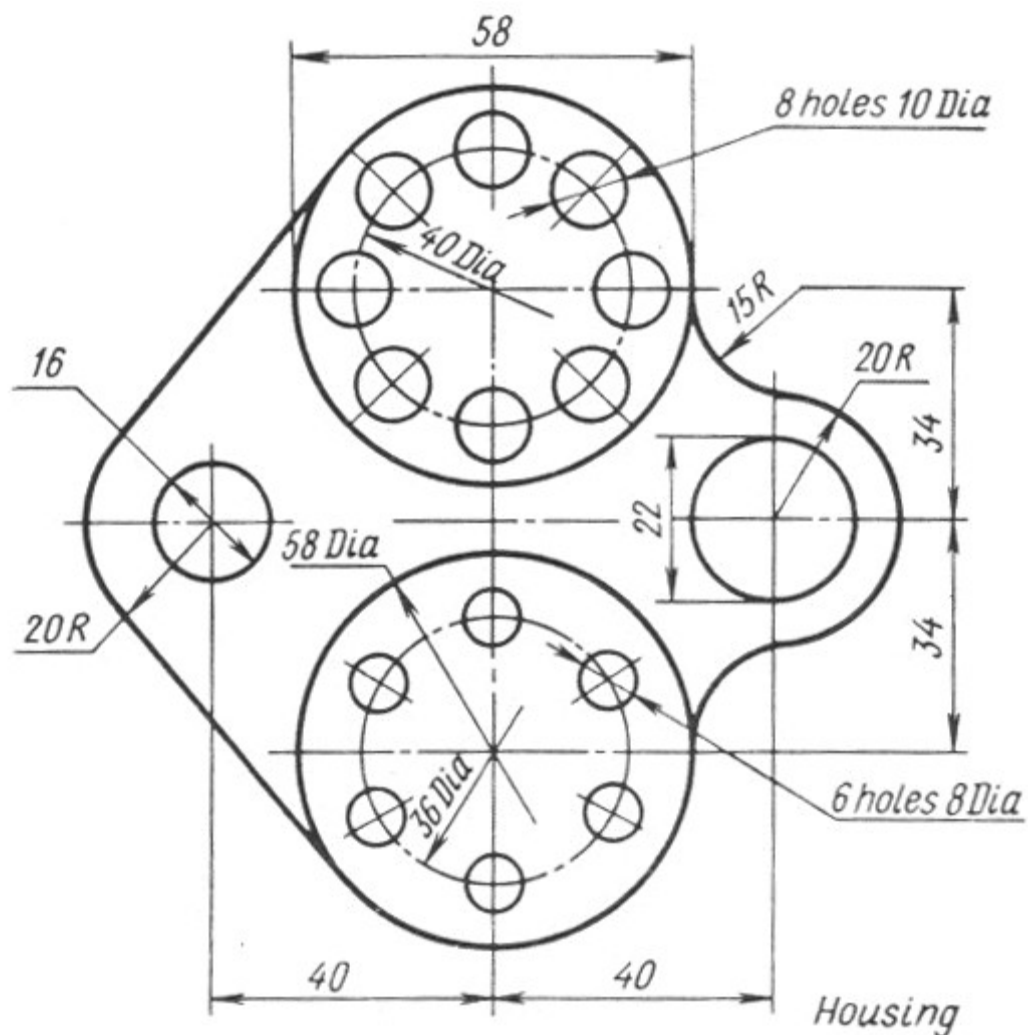
1. What are the possible ways to draw arc?
2. How to extend line?
3. What is the purpose of command trim?
4. How to convert 2D drawings to 3D drawings?
5. How to give dimension?

Exercise – 4

Aim: To create a 2D view of the given diagram using Auto CAD.

Procedure:

1. Type limits in command menu & set value to 297,290.
2. Change the units to millimeters from inches and also precision to 0 by clicking format -> units -> ok.
3. To set the paper size type zoom -> enter and type a -> enter in command bar.
1. Draw the 2 axes lines
2. Draw 2 concentric circles of diameter 58 and 40 above the axes
3. Draw the 10dia circle on the 40dia of the circle
4. And use the array command from modify tool bar to represent the 8 holes with 10dia from centre of the circles.
5. Draw the 2 concentric circles of diameter 58 & 36 below the axes
6. Draw the circles of the diameter 8 on the circle of 36dia
7. Use the array command from the modify tool bar to represent the 6 holes with 8dia
8. Draw the 2 concentric circles of diameter 22 and 20 radius at the right side of the vertical line from the vertical axes.
9. Fillet the circles of radius 20 with 15 radius.
10. And again draw the circles of diameter 16& 20R at left side of the vertical line from the vertical axes.
11. And chamfer the circles of radius 20 with 58dia circle
12. Then trim the unwanted lines to get the required 2D drawing



Result: Hence the required 2D diagram is created using Auto CAD.

Viva questions

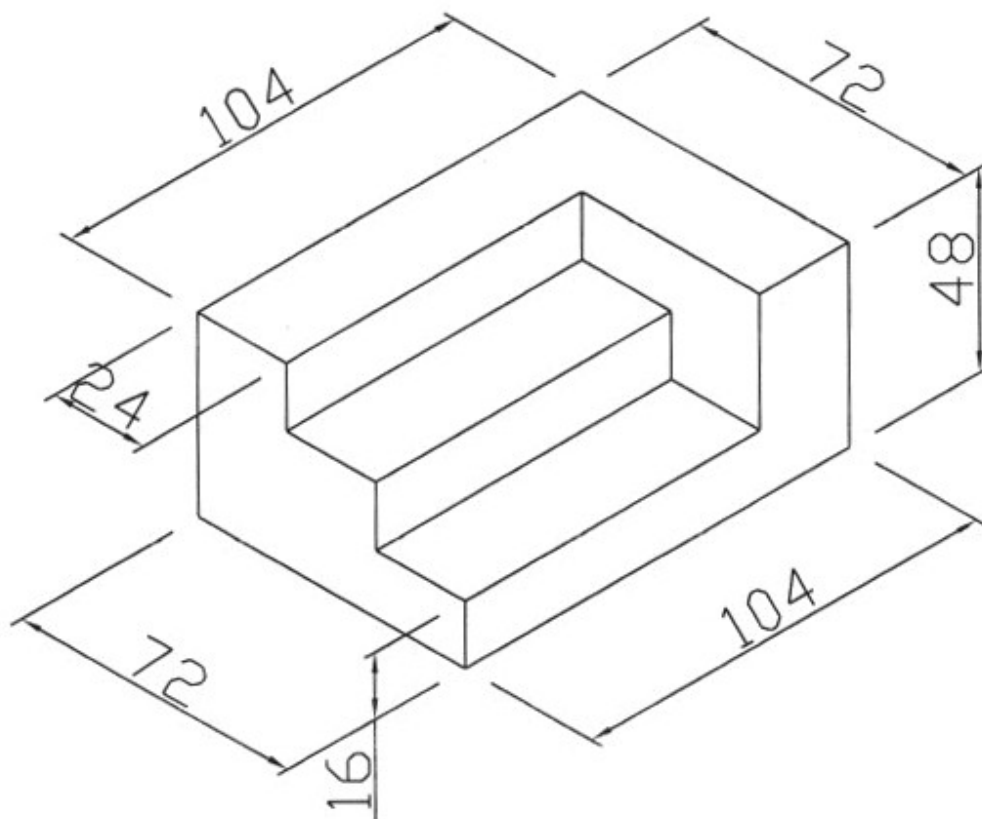
1. What is the purpose of command trim?
2. How to convert 2D drawings to 3D drawings?
3. What is the difference between Pan and Zoom?
4. What is difference between circle and ellipse?
5. What is the difference between rectangle and polygon?

Exercise – 5

Aim: To create a 2D isometric view of the given diagram using Auto CAD.

Procedure:

1. Type limits in command menu & set value to 297,290.
2. Change the units to millimeters from inches and also precision to 0 by clicking format -> units -> ok.
3. To set the paper size type zoom -> enter and type a -> enter in command bar.
4. Go to drafting settings and turn on isometric snap..
5. Use the F5 key to change between the views of isometric planes.
6. Start from the front view and draw the the line of length of line 104 using the F8 key (O snap key) and continue with the 48 length line.
7. Change to top plane and draw the 72mm line.
8. Continue in the same fashion to complete the whole figure.
9. Give the dimensions from the dimension tool bar as in diagram.



Result: Hence the required 2D isometric diagram is created using Auto CAD.

Viva questions

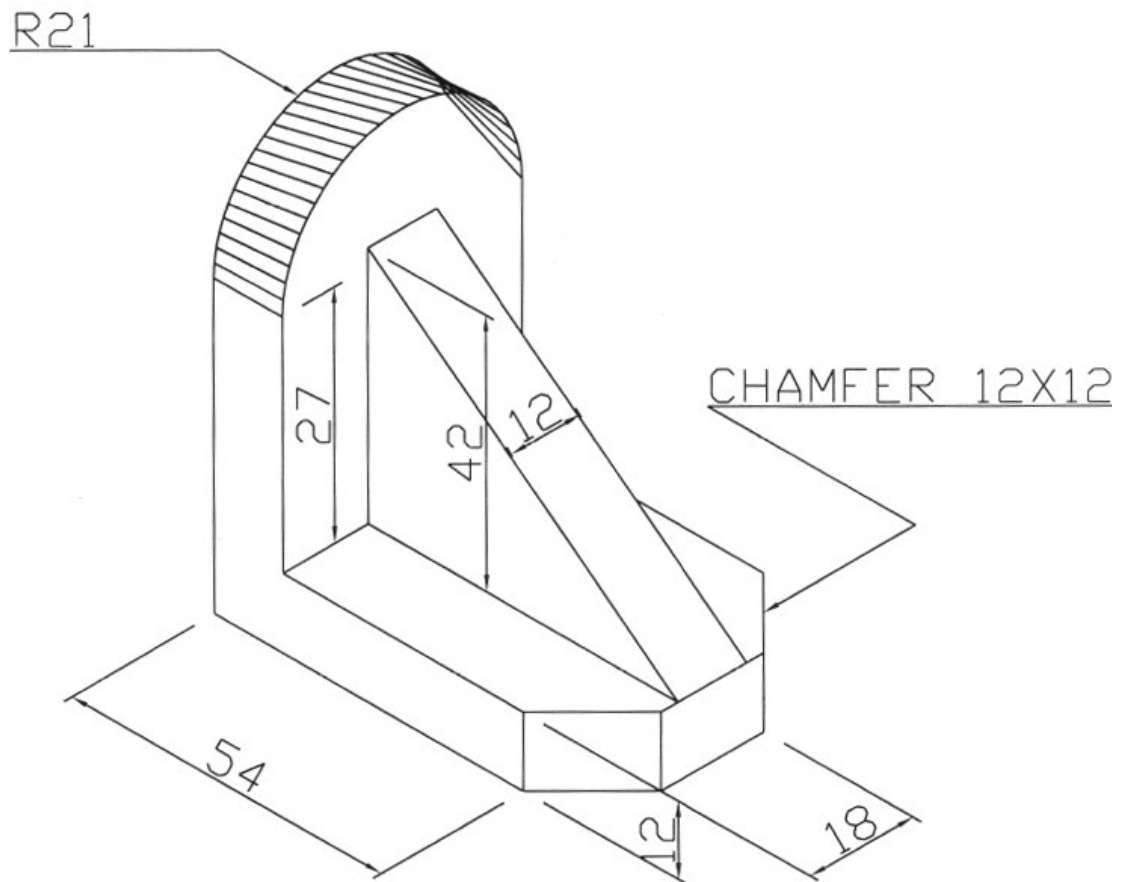
1. What is the difference between metric unit and English unit?
2. What is the shortcut key for ortho ON/OFF ?
3. What is the shortcut key for help?
4. What are the various ways to draw a measured line?
5. .What are object snap command.

Exercise – 6

Aim: To create a 2D isometric view of the given diagram using Auto CAD

Procedure:

1. Type limits in command menu & set value to 297,290.
2. Change the units to millimeters from inches and also precision to 0 by clicking format -> units ->ok.
3. To set the paper size type zoom -> enter and type a -> enter in Command bar.
4. Go to drafting settings and turn on isometric snap. Use the F5 key to change between the views of isometric planes.
5. Start from the front view and draw the the line of length of line 54.
6. Draw the semi circle using the iso circle option from the ellipse command.
7. Continue drawing using F5 and F8 snap keys.
8. Give proper dimensions to the figure and practice at home.



Result: Hence the required 2D isometric diagram is created using Auto CAD.

Viva questions

1. How do you select the isometric snap?
2. How to shift between different planes while using 0-snap?
3. How to give chamfer for a closed figure?
4. How to draw a circle in isometric diagram
5. How to dimension the diagram in isometric view.

Solid works

Exercise -1

Aim: draw the sketch of the model shown in Figure A. The sketch is shown in Figure B. You will not dimension the sketch. The solid model and the dimensions are given only for your reference.

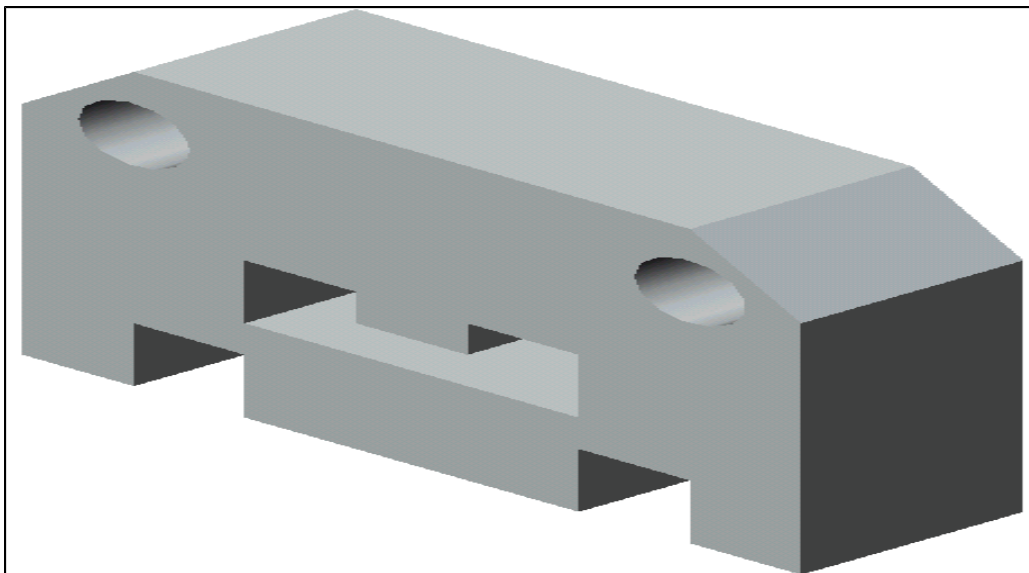


Figure A Solid model for exercise 1

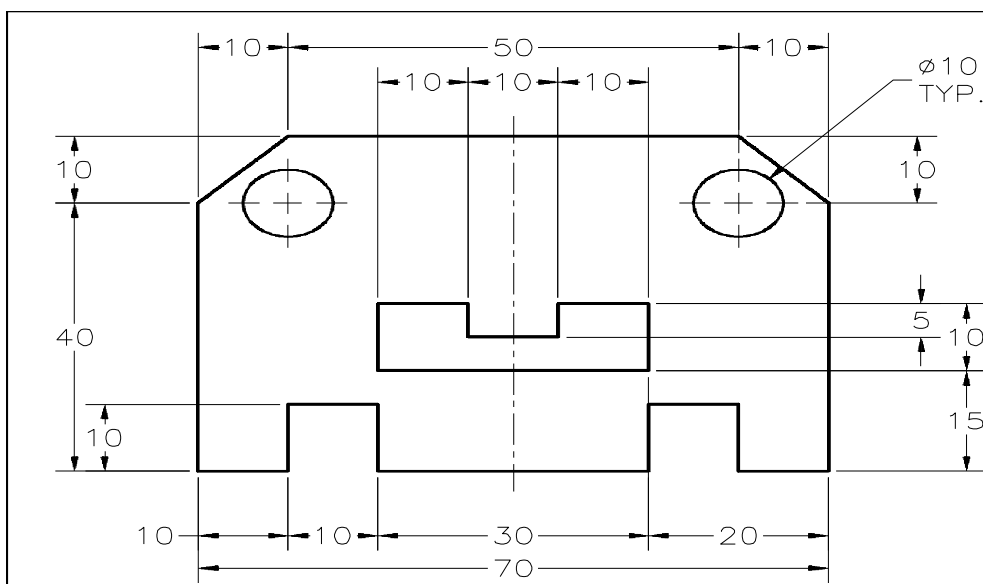


Figure B Sketch of the model

Procedure:

1. Start SolidWorks by choosing **Start > Programs > SolidWorks 2006 > SolidWorks 2006**.
2. The **SolidWorks Resource Task Pane** is displayed on the right of the **SolidWorks** window.
3. Choose the **New** button from the **Standard** toolbar.
4. The **New SolidWorks Document** dialog box is displayed.
5. The **Part** button is chosen by default. Choose the **OK** button from the **New SolidWorks Document** dialog box as shown in **Figure C**

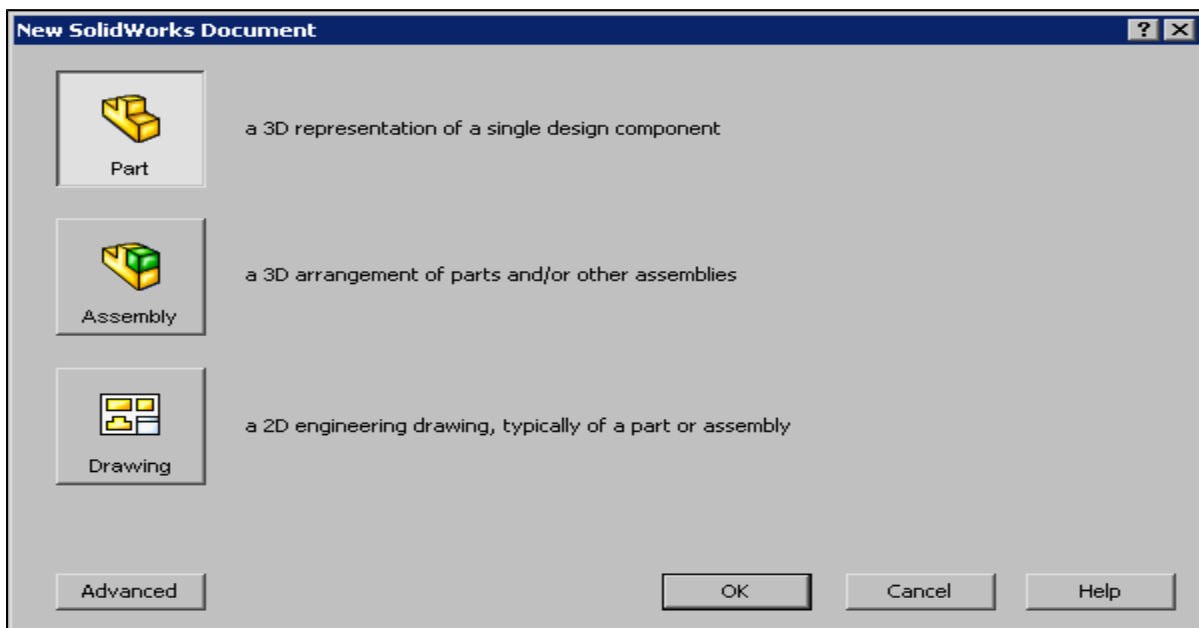
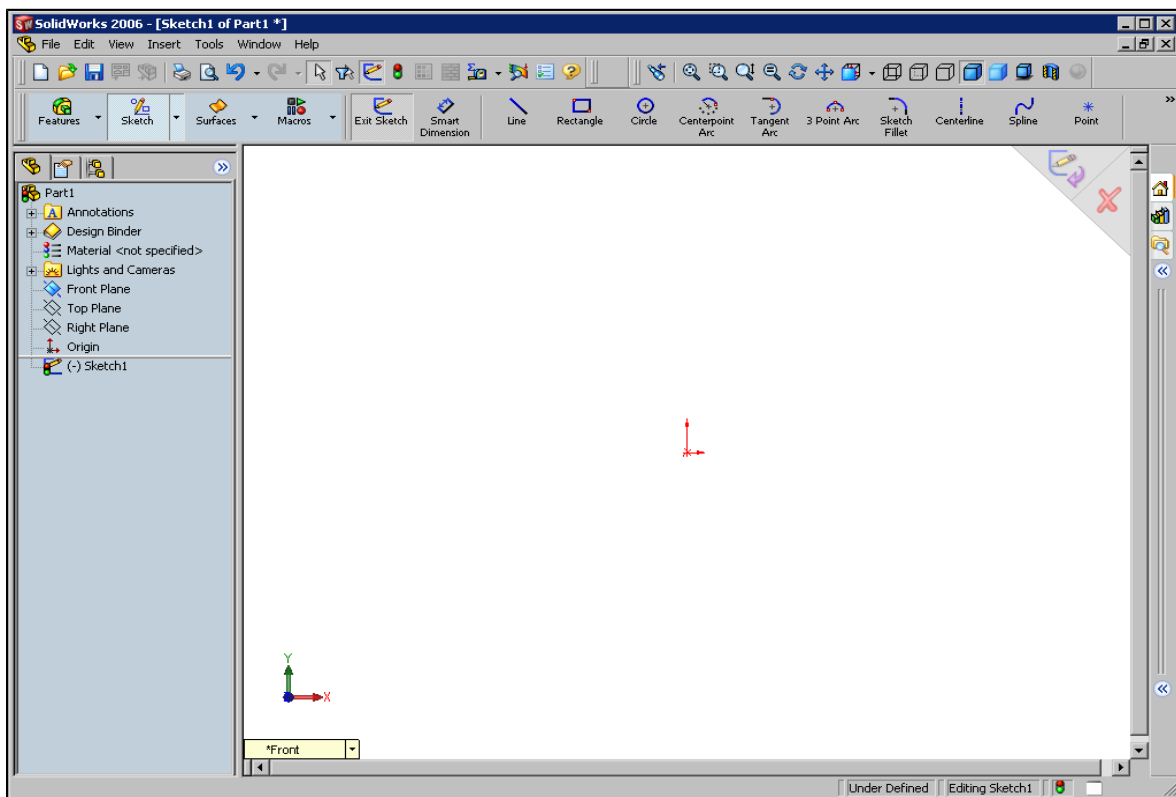


Figure C New SolidWorks Document dialog box

6. Choose the **Maximize** button available on the upper right corner of the part document window to maximize the document window.
7. Choose the **Sketch** button from the **Standard** toolbar. The **Edit Sketch PropertyManager** is displayed and you are prompted to select a plane on which you want to create the sketch.
8. Select the **Front** plane. The sketching environment is invoked and the plane is oriented normal to the view. You will notice that a red color origin is displayed in the center of the screen, indicating the sketching environment. The default screen appearance of the sketching environment of SolidWorks is shown in **Figure D**.

**Figure D Screen display in sketcher environment**

9. Choose **Tools > Options** from the menu bar to invoke the **System Options - General** dialog box.
10. Choose the **Document Properties** tab. The name of the dialog box is changed to the **Document Properties - Detailing** dialog box.
11. Select the **Units** option from the area on the left to display the options related to linear and angular units.

12. Select the **MMGS (millimeter, gram, second)** radio button from the **Unit system** area if it is not selected by default. Also, select the **Degrees** option from the drop-down list provided in the **Angular** units area.
13. Select **Grid/Snap** from the area on the left. Set the value of the **Major grid spacing** spinner to **100** and the value of the **Minor-lines per major** spinner to **20**.
14. Now, choose the **Go To System Snaps** button; the system options related to relations and snap are displayed. Select the **Grid** check box from the **Sketch Snaps** area and make sure that you clear the **Snap only when grid is displayed** check box.

The sketch of the model consists of an outer loop, two circles inside the outer loop, and a cavity. Therefore, it will be drawn using the **Line** and the **Circle** tools. You will first draw the outer loop and then the inner entities.

15. Choose the **Line** button from the **Sketch CommandManager** to invoke the **Line** tool. The arrow cursor will be replaced by the line cursor.
16. Move the cursor in the first quadrant close to the origin. Draw the outer loop of the sketch which is shown in **Figure E**.

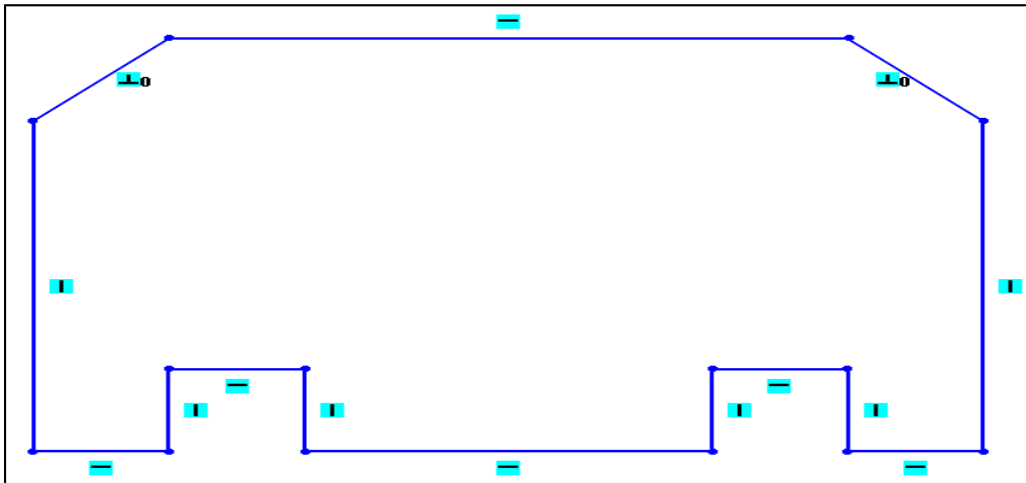


Figure E Outer loop of the sketch

17. The circles will be drawn using the **Circle** tool. You will use the inferencing line originating from the start points and endpoints of the inclined lines to specify the center point of the circles. As mentioned earlier, the cursor in the sketching environment jumps through a distance of 10 mm by default. But the radius of the circle is 5 mm and to specify this value, the cursor needs to jump only through a distance of 5 mm. This is the reason you need to modify the document settings so that the cursor jumps through a distance of 5 mm.
18. The sketch after drawing the two circles inside the outer loop is shown in **Figure F**.

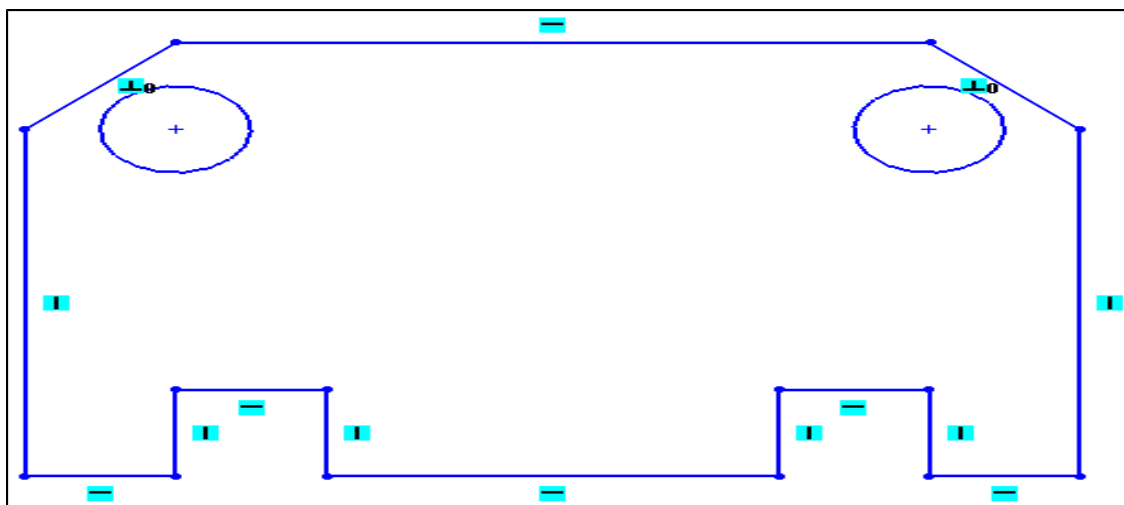


Figure F Sketch after drawing the two inner circles

19. Draw the sketch of the inner cavity. The final completed sketch for **Exercise 1** is shown in **Figure G**.

Figure B Sketch of the model

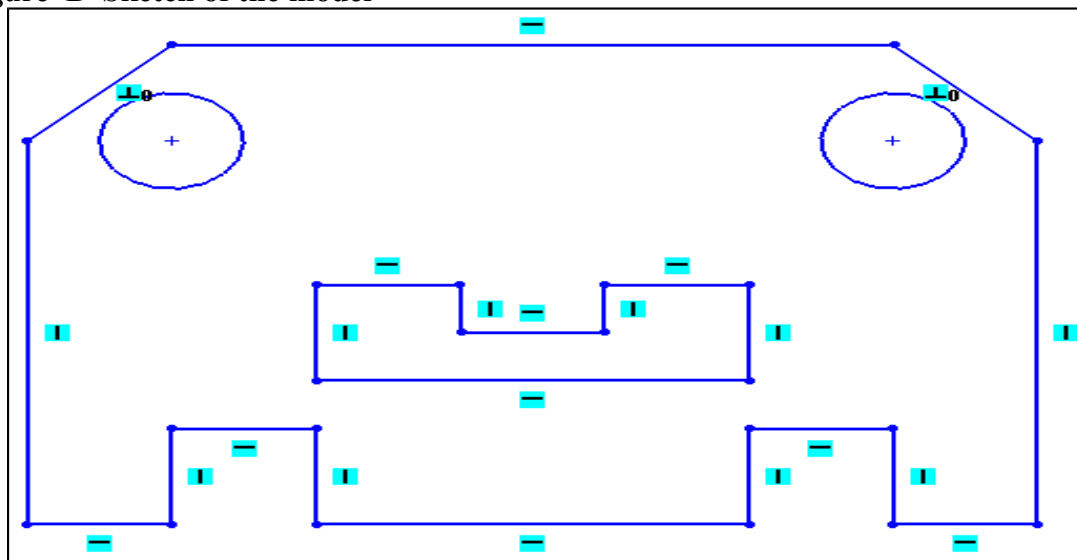


Figure G Final sketch for Exercise 1

20. Choose the **Save** button from the **Standard** toolbar

Result: Hence the required diagram is created using solid works

Exercise -2

Aim: To draw the basic sketch for the revolved solid model shown in Figure A. The sketch for the revolved solid model is shown in Figure B. Do not dimension the sketch as the solid model and the dimensions are given only for your reference.

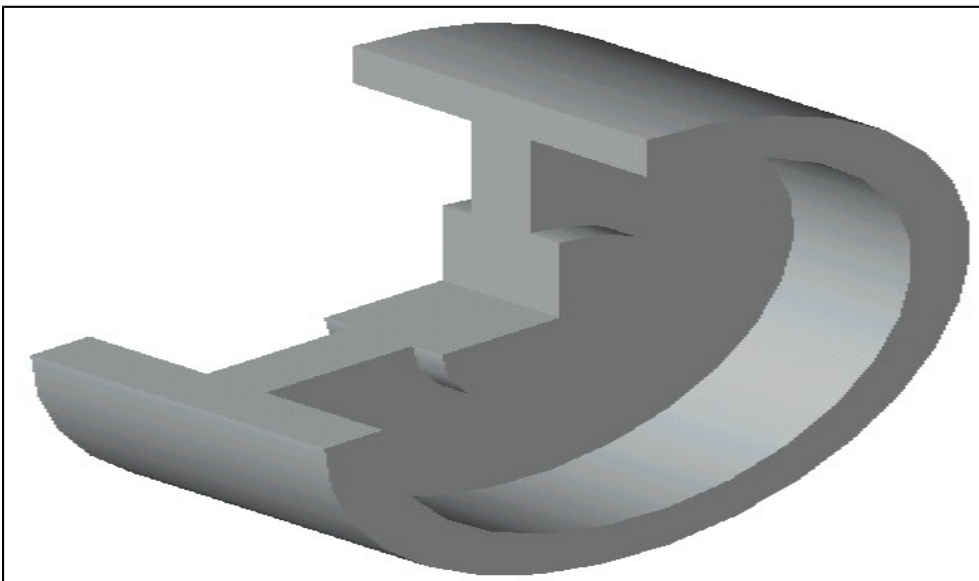


Figure A Revolved model for Exercise 2

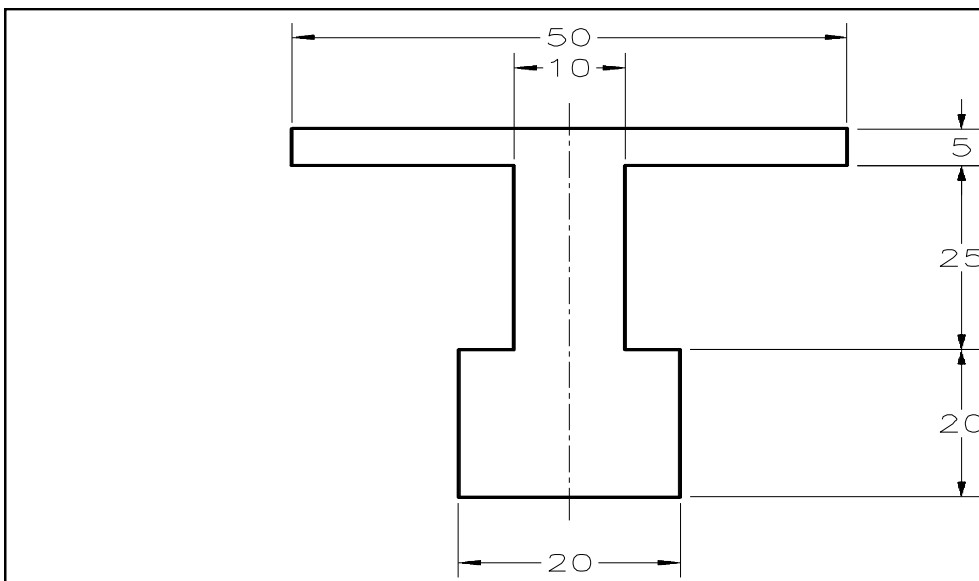


Figure B Sketch of the revolved model

Procedure:

1. Choose the **New** button from the **Standard** toolbar to invoke the **New SolidWorks Document** dialog box.
2. The **Part** button is chosen by default in the **New SolidWorks Document** dialog box. Choose **OK**. A new SolidWorks part document is started.
3. Choose the **Sketch** button from the **Standard** toolbar. The **Edit Sketch PropertyManager** is displayed. Select the **Front** plane to invoke the sketching environment.
4. Set the value of the **Major grid spacing** spinner to **50** and the value of the **Minor-lines per major** spinner to **10**.
5. The sketch consists of horizontal and vertical lines. Therefore, this sketch will be drawn using the **Line** tool. You will start drawing from the lower left corner of the sketch.
6. Choose the **Zoom to Fit** button from the **View** toolbar to fit the sketch on the screen. The completed sketch for **Tutorial 2** is shown in **Figure C**.

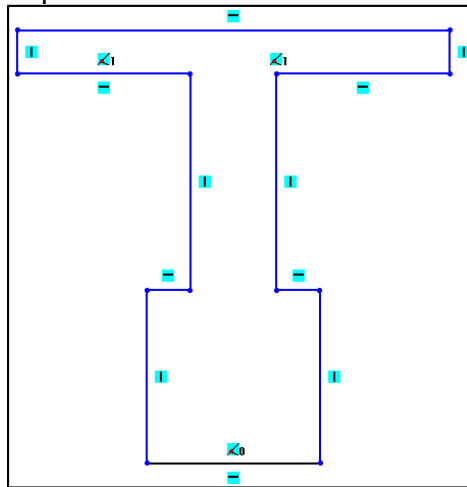


Figure C Final sketch for Exercise 2

7. Choose the **Save** button from the **Standard** toolbar
- Result:** Hence the required diagram is created using solid works

Exercise -3

Aim: To draw the basic sketch of the model shown in Figure A. The sketch to be drawn is shown in Figure B. Do not dimension the sketch as the solid model and the dimensions are given only for your reference.

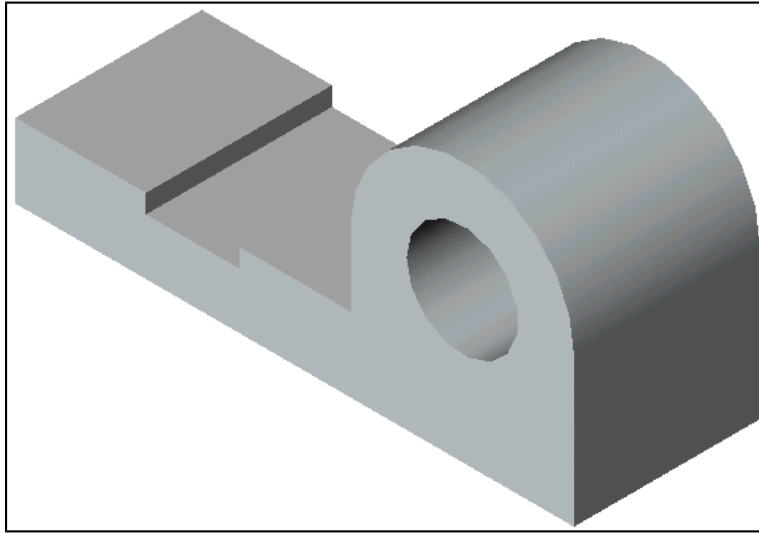


Figure A Solid model for Exercise 3

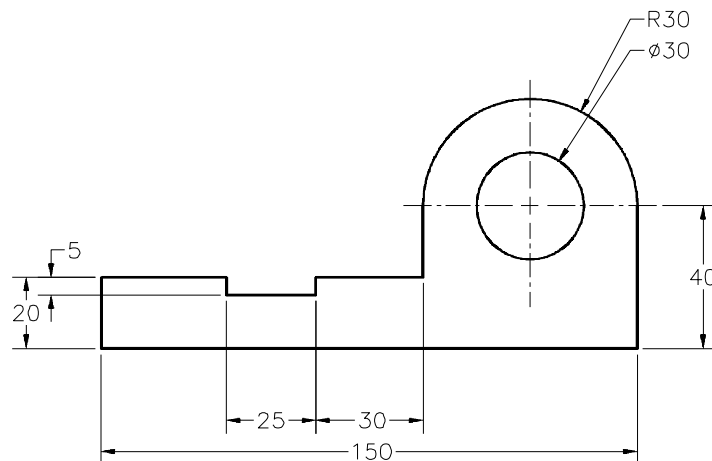


Figure B Sketch of the model

1. Choose the **New** button from the **Standard** toolbar to invoke the **New SolidWorks Document** dialog box.

2. The **Part** button is chosen by default in the **New SolidWorks Document** dialog box. Choose **OK**. A new SolidWorks part document is started.
3. Choose the **Sketch** button from the **Standard** toolbar. The **Edit Sketch PropertyManager** is displayed. Select the **Front** plane to invoke the sketching environment.
4. Set the value of the **Major grid spacing** spinner to **50** and the value of the **Minor-lines per major** spinner to **10**.
5. The sketch consists of an outer loop and an inner circle. Therefore, this sketch will be drawn using the **Line** and the **Circle** tools. You will start drawing from the lower left corner of the sketch.
6. Draw the outer loop of the sketch as shown in **Figure C**.

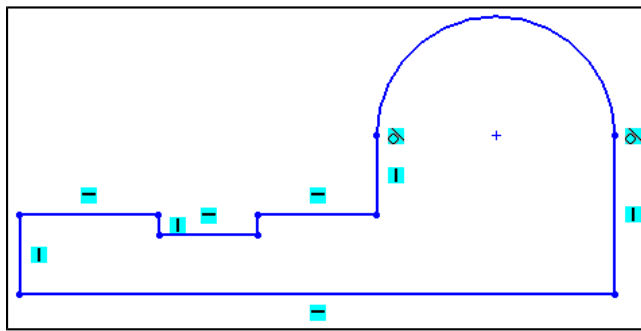


Figure C Sketch after drawing the outer loop

7. Draw the circle using **Circle** button **Figure D** shows the final sketch for **Tutorial3**.

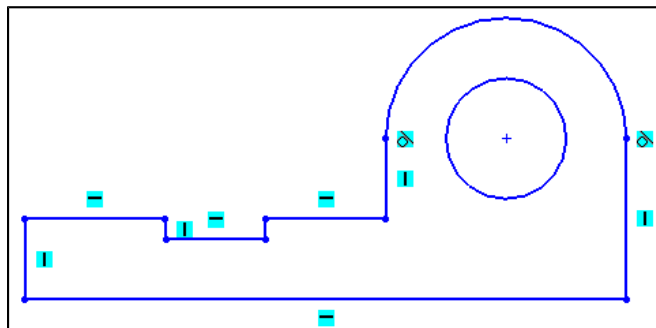


Figure D Final sketch for Exercise 3

8. Choose the **Save** button from the **Standard** toolbar
Result : Hence the required diagram is created using solid works

Exercise -4

Aim: To draw the sketch of the model shown in Figure A. The sketch for the model is shown in Figure B. Do not dimension the sketch as the dimensions and the solid model are given only for your reference.

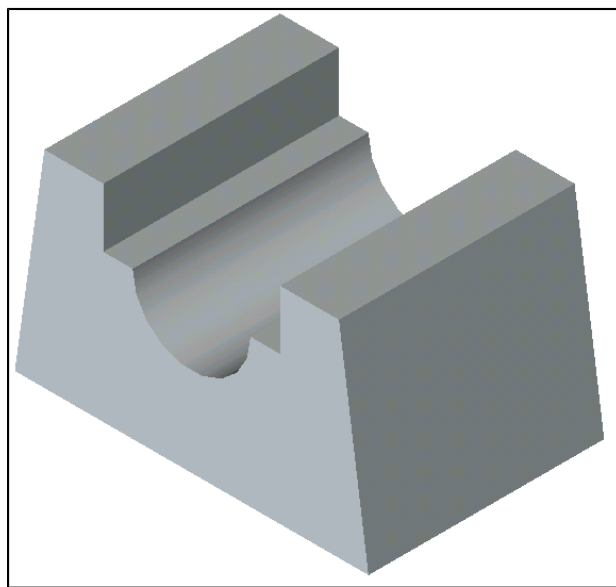


Figure A Model for Exercise 4

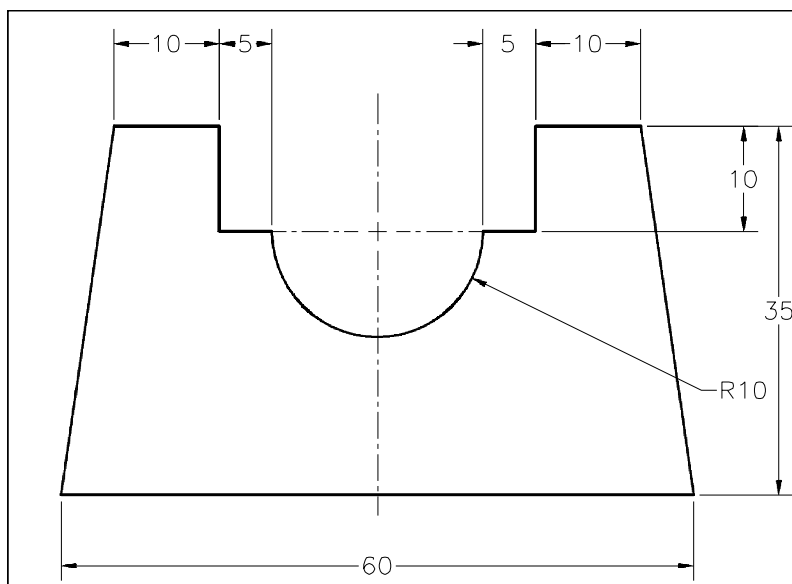


Figure B Sketch for Exercise 4

Procedure:

1. Choose the **New** button from the **Standard** toolbar to invoke the **New SolidWorks Document** dialog box.
2. The **Part** button is chosen by default in the **New SolidWorks Document** dialog box. Choose **OK**. A new SolidWorks part document is started.
3. Choose the **Sketch** button from the **Standard** toolbar. The **Edit Sketch PropertyManager** is displayed. Select the **Front** plane to invoke the sketching environment.
4. Set the value of the **Major grid spacing** spinner to **50** and the value of the **Minor-lines per major** spinner to **10**.
5. The sketch will be drawn using the **Line** tool. The arc in the sketch will also be drawn using the same tool. You will start drawing from the lower left corner of the sketch.
6. Choose the **Zoom to Fit** button from the **View** toolbar to modify the drawing display area. The final sketch for **Tutorial 4** is shown in **Figure C**.

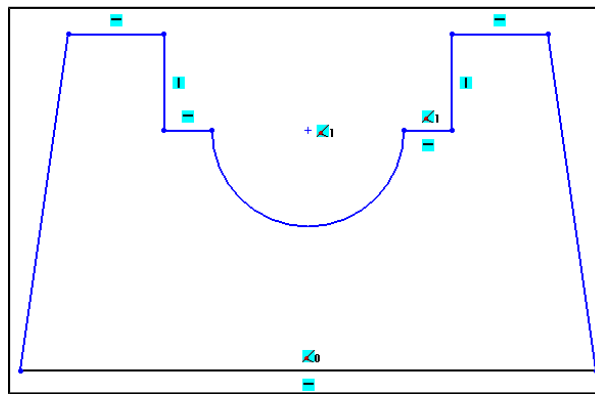


Figure C Final sketch for Exercise 4

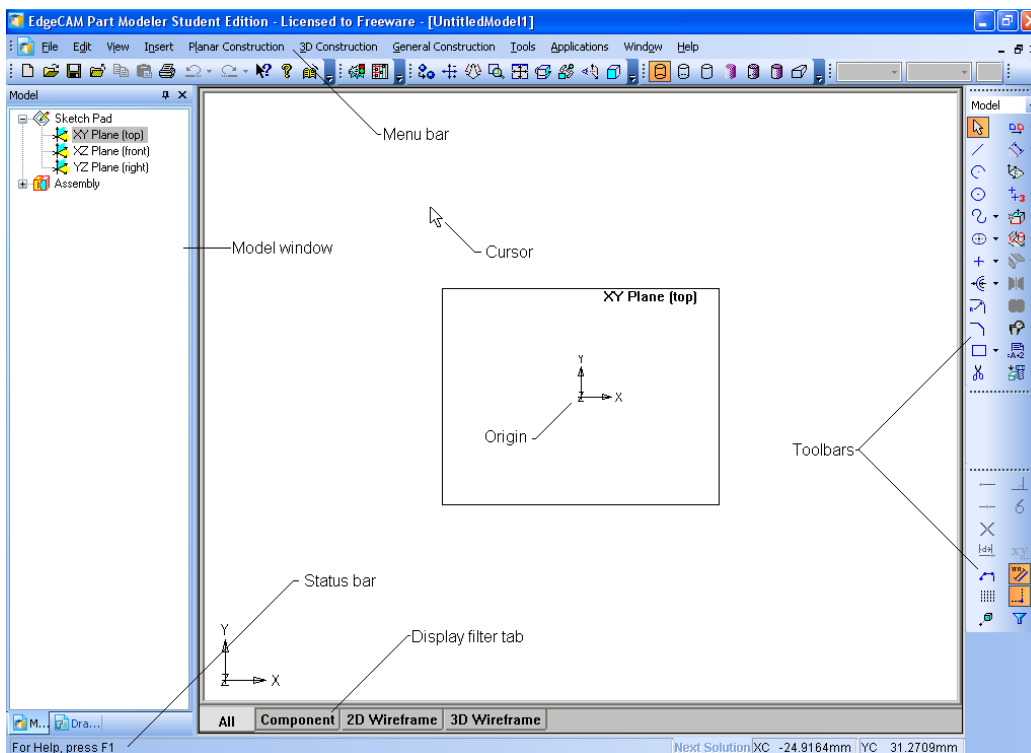
7. Choose the **Save** button from the **Standard** toolbar

Result: Hence the required diagram is created using solid works

Introduction EdgeCAM 11.0

1. EdgeCAM, developed by Pathtrace Engineering Systems, is one of the world's leading computer aided manufacturing software.
2. EdgeCAM provides you two modules, EdgeCAM Part Modeler and EdgeCAM.
3. The various machining capabilities of EdgeCAM include 2 to 3 axis machining, 5 axis trimming and deflashing, 2 to 4 axis (C and Y) turning and 2 to 4 axis wire erosion.
4. EdgeCAM provides simulator that is used to verify the machining operations in order to prevent damages to the tools and the machine tool.

The initial screen, after starting the EdgeCAM Part Modeler in the **Model** mode, is shown in the figure.



Initial screen display of the EdgeCAM Part Modeler in the Model mode

OPERATING MODES OF EdgeCAM PART MODELER

There are two operating modes in the EdgeCAM Part Modeler.

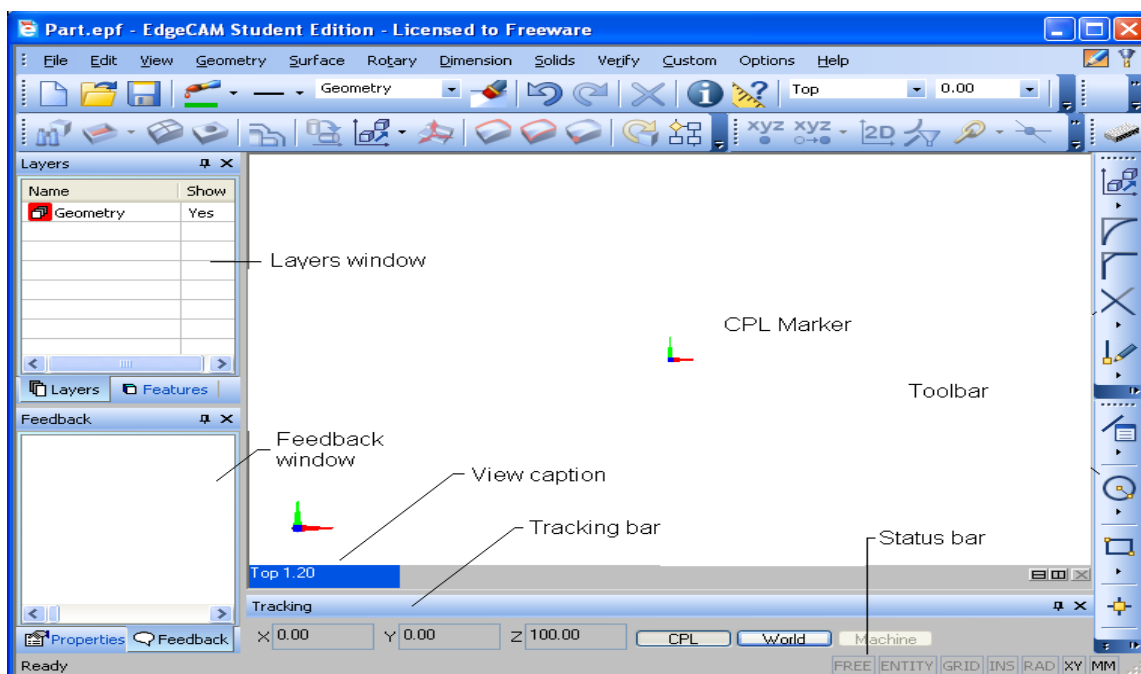
Model

The **Model** mode of the EdgeCAM Part Modeler is a feature-based parametric environment in which you can create solid models.

Drawing Mode

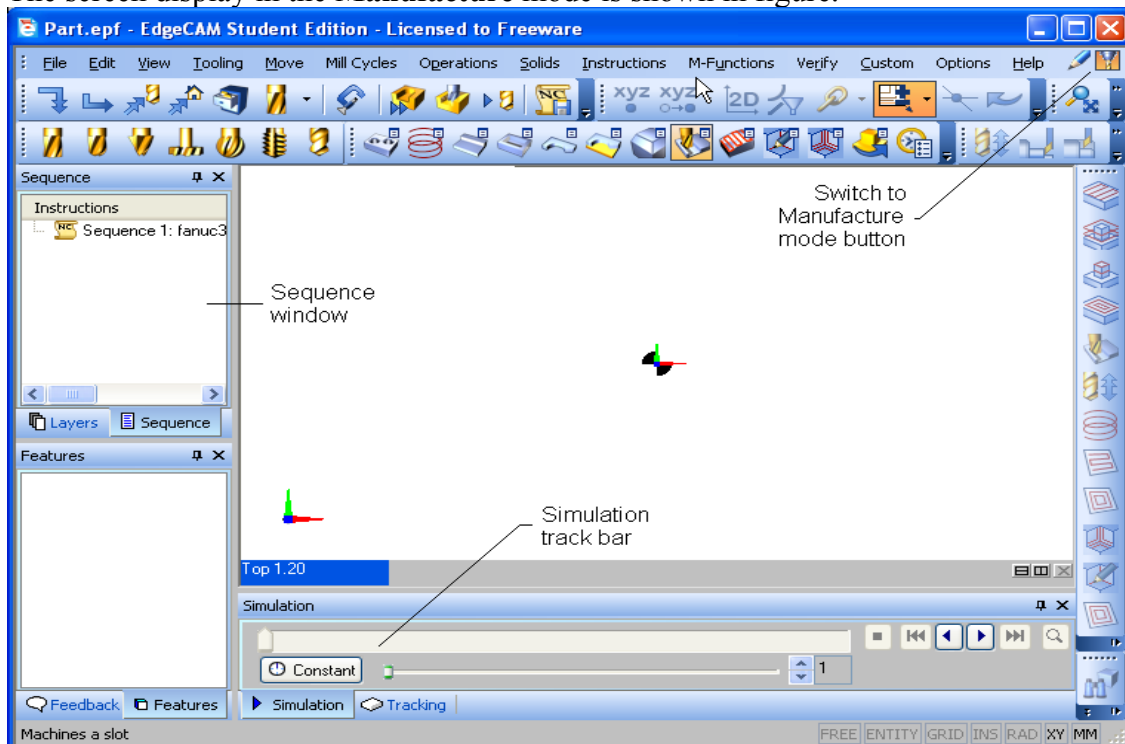
The **Drawing** mode is used for the documentation of the parts created earlier in the form of drawing views and their detailing.

The initial screen after starting EdgeCAM is shown in figure.



Screen display of EdgeCAM in the Design mode

Choose the **Switch to Manufacture Mode** button from the top right corner of the screen. The screen display in the **Manufacture** mode is shown in figure.



Screen display of EdgeCAM in the Manufacture mode

There are two operating modes of EdgeCAM.

1. Design Mode :

This mode offers a comprehensive set of tools for creating 2D and 3D geometries and complex surfaces, which are later used in the Manufacturing mode for generating toolpaths.

The Design mode offers two working environments, XY and ZX.

2. Manufacture Mode

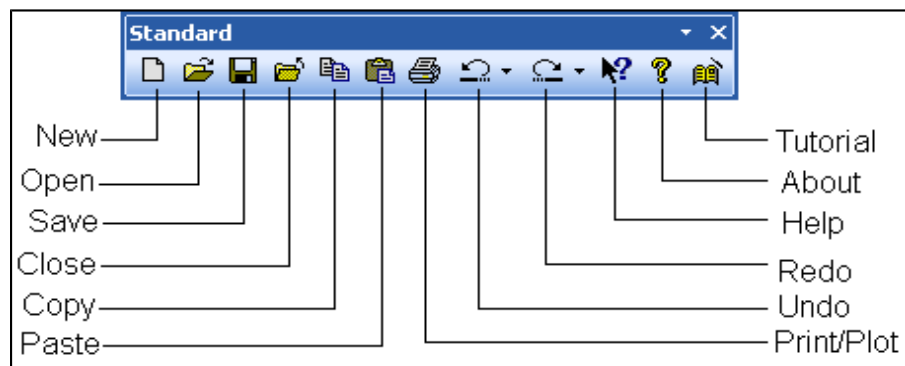
The Manufacture mode offers the tools for generating the toolpaths for various milling, turning, and wire EDM operations. You can enter the Manufacture mode by selecting Options > Manufacture from the menu bar or by selecting the Switch to Manufacture Mode button available at the top right corner of the EdgeCAM screen.

SYSTEM REQUIREMENTS FOR EdgeCAM 11.0

The following system requirements are needed to ensure a smooth running of EdgeCAM:

1. **System unit:** Current Intel Pentium compatible processor such as Pentium 4, Intel XEON, or AMD Athlon running Windows 2000 SP4 or Windows XP Professional.
2. **Memory:** 512 MB RAM minimum (1 GB for advanced systems).
3. **Disk drive:** 1GB free disk space.
4. **Internal/External drives:** A CD-ROM drive is required for program installation.
5. **Display:** Graphics Card capable of 1280 x 1024 high color setting & 64 Mb memory
6. 17" color monitor (minimum screen resolution of 1280 x 1024).
7. Wheel mouse.
8. Parallel Port for security key.

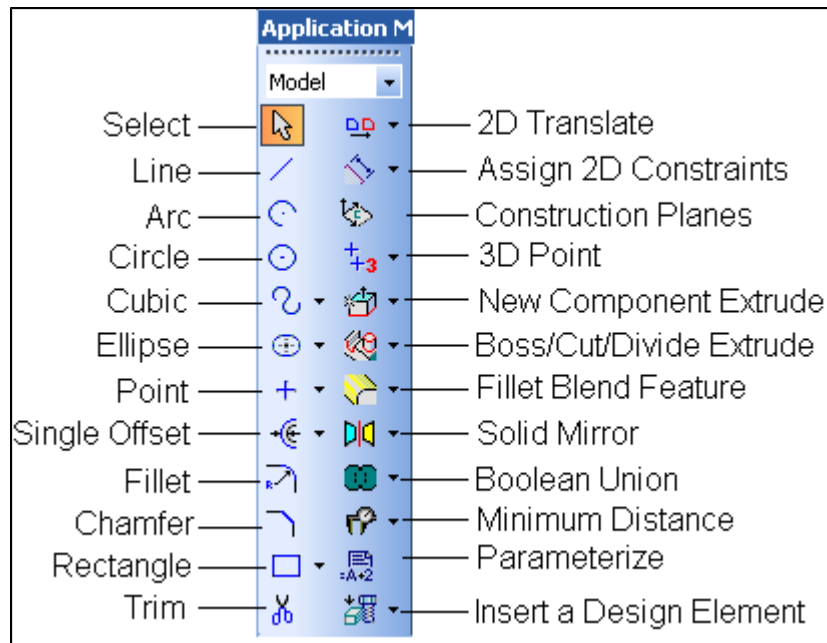
EdgeCAM PartModeler Toolbars



The Standard toolbar

Application Menu Toolbar

The tools in this toolbar are used for sketching a profile and creating features. Some tools are grouped together. Also, the tool used last will be displayed in the toolbar. Figure shows the **Application Menu** toolbar.



The Application Menu toolbar

HOT KEYS

Hot Keys

Alt+Backspace

CTRL+Y

CTRL+S

CTRL+W

CTRL+E

CTRL+D

CTRL+M

CTRL+L

CTRL+SHIFT+L

Function

Invoke the Undo tool

Invoke the Redo tool

Save the current document

Invoke the Zoom Window tool

Invoke the Zoom Extents tool

Switch to the Design mode

Switch to the Manufacture mode

Invoke the Line Dialog Box

Display/Hide the Layers window

Exercise 1

Aim: Draw the sketch of the model shown in **Figure A**. The sketch is shown in **Figure B**.

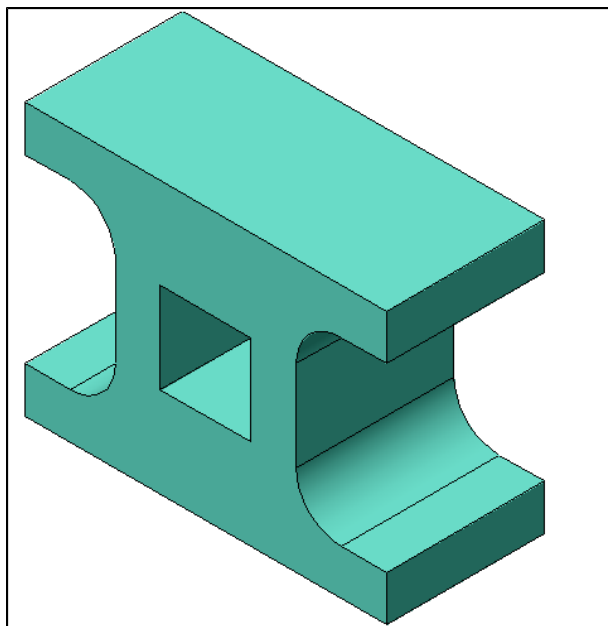


Figure A Model for Exercise 1

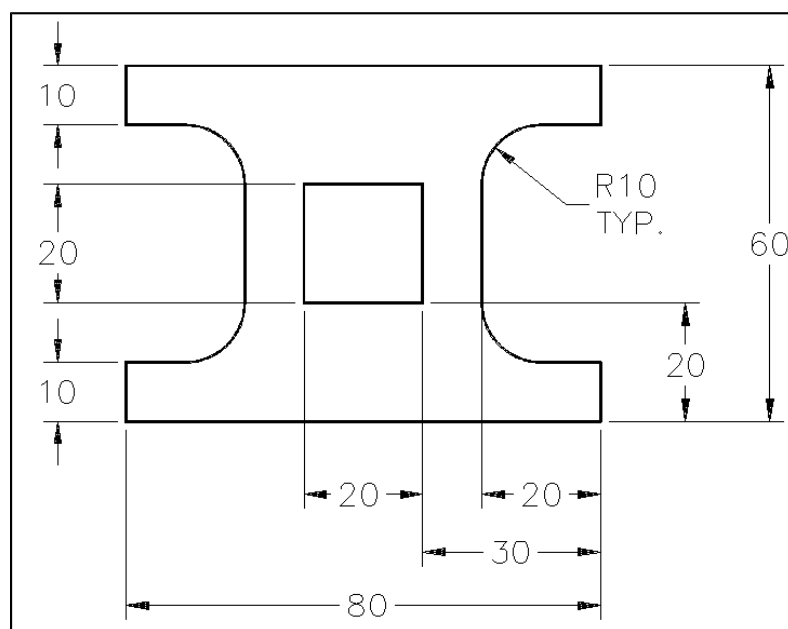
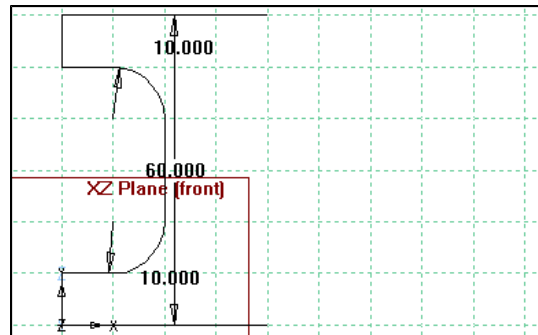
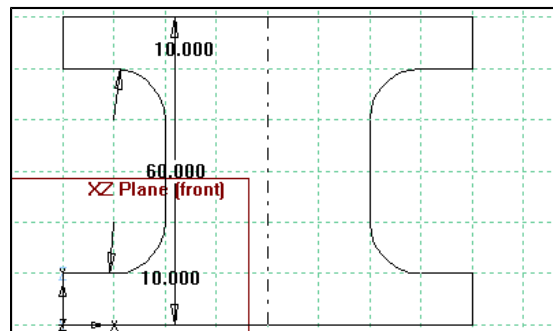


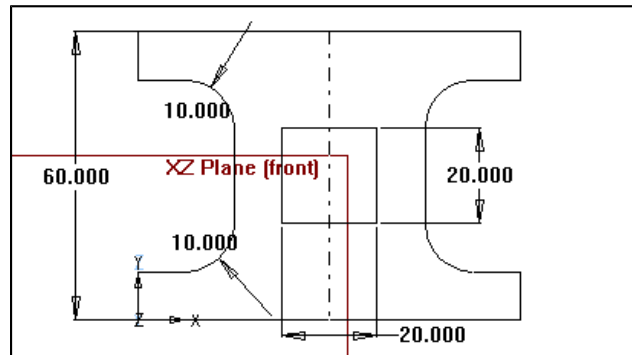
Figure B Sketch for Exercise 1

Procedure:

1. Start a new file in the EdgeCAM Part Modeler. Select the **XZ Plane (front)** from the **Model** window to make it the active construction plane.
2. Choose **Tools > Options > Document** from the menu bar to invoke the **Document options** dialog box. Set the value of the distance increment of the rectangular snap as **10, 10** in the **Document options** dialog box.
3. Draw one half of the sketch using the **Line** and **Arc** tool, as shown in **Figure C**.
4. The other half of the sketch is completed by mirroring the entities, as shown in **Figure D**.

**Figure C One half of the sketch****Figure D Sketch after mirroring**

5. Draw the inner rectangular cavity using the **Rectangle** tool. The final sketch is shown in **Figure E**.

**Figure E The final sketch**

6. Save the file

Result: Hence the required diagram is created using EDGE CAM

Exercise 2

Aim: draw the sketch of the model shown in **Figure A**. The sketch is shown in **Figure B**.

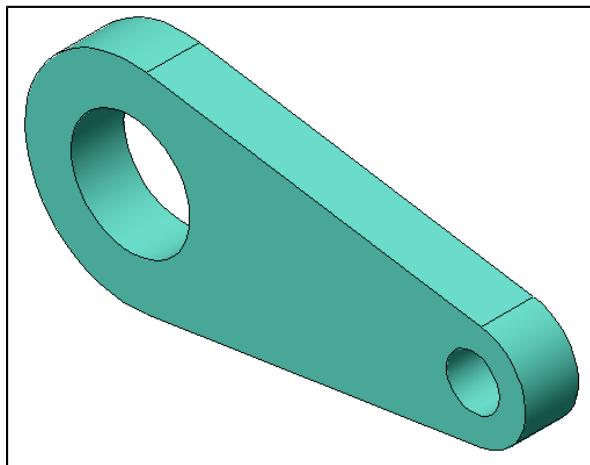


Figure A Model for Exercise 2

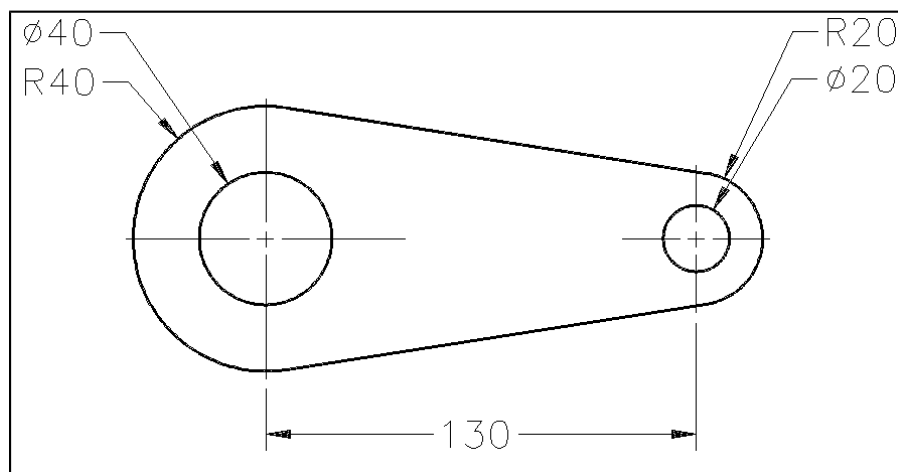


Figure B Sketch for Exercise 2

Procedure:

1. Start a new file in the EdgeCAM Part Modeler. Select the **XZ Plane (front)** from the **Model** window to make it the active construction plane.
2. Draw the outer profile using the **Line** and **Circle** tool. The sketch is shown in **Figure C**.
3. After drawing the outer profile of the sketch, trim its unwanted portion using the **Quick trimming** tool. The sketch after trimming is shown in **Figure D**.

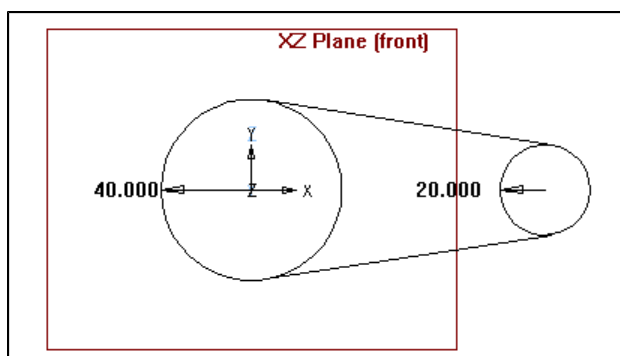


Figure C Sketch after drawing the two tangent lines

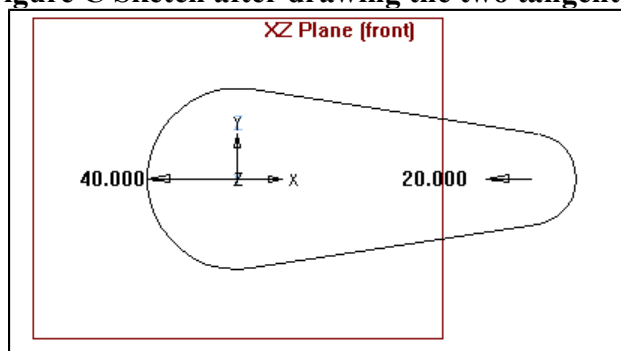


Figure D Sketch after trimming the unwanted portion

4. Draw its inner loops, that consist of two circles that will be drawn using the Circle tool. The final sketch is shown in Figure E.

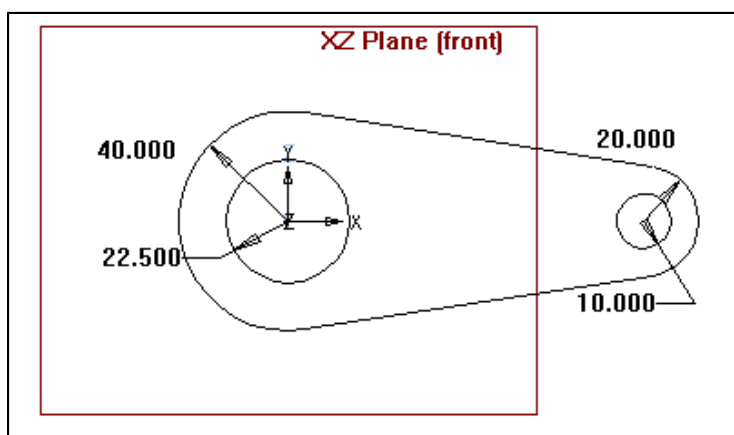


Figure E The final sketch

5. Save the file

Result : Hence the required diagram is created using EDGE CAM

INTRODUCTION TO CATIA V5R19

1. **CATIA** Stands for **Computer Aided Three Dimensional Interactive Application**
2. CATIA V5, developed by Dassault Systems, France, is a completely re-engineered, next-generation family of CAD/CAM/CAE software solutions for Product Lifecycle Management.
3. CATIA V5 serves the basic design tasks by providing different workbenches.
4. A workbench is defined as a specified environment consisting of a set of tools, which allow the user to perform the specific design tasks in a particular area.

The basic workbenches available in CATIA V5 are :

Part Design Workbench

The Part Design workbench is a parametric and feature-based environment, in which you can create solid models.

Wireframe and Surface Design Workbench

1. The Wireframe and Surface Design workbench is also a parametric and feature-based environment, in which you can create wireframe or surface models.
2. The tools in this workbench are similar to those in the Part Design workbench, with the only difference that the tools in this environment are used to create basic and advanced surfaces.

Assembly Design Workbench

1. The **Assembly Design** workbench is used to assemble the components using the assembly constraints available in this workbench.
2. There are two types of assembly design approaches:
 1. Bottom-up
 2. Top-down

Drafting Workbench

1. The **Drafting** workbench is used for the documentation of the parts or the assemblies created earlier in the form of drawing views and their detailing.
2. There are two types of drafting techniques:
 1. Generative drafting
 2. Interactive drafting

SYSTEM REQUIREMENTS

The following are the system requirements to ensure smooth running of CATIA V5R16 on your system:

- System unit: An Intel Pentium III or Pentium 4 based workstation running Microsoft 2000 Professional Edition or Windows XP Professional Edition.
- Memory: 256 MB of RAM is the minimum recommended for all applications. 512 MB of RAM is recommended for DMU applications.
- Disk drive: 4 GB Disk Drive space (Minimum recommended size)
- Internal/External drives: A CD-ROM drive is required for program installation.
- Display: A graphic color display compatible with the selected platform-specific graphic adapter. The minimum recommended monitor size is 17 inches.
- Graphics adapter: A graphics adapter with a 3D OpenGL accelerator is required with minimum resolution of 1024x768 for Microsoft Windows workstations and 1280x1024 for UNIX workstations.

IMPORTANT TERMS AND DEFINITIONS

Feature-based Modeling

- A feature is defined as the smallest building block that can be modified individually.
- A model created in CATIA V5 is a combination of a number of individual features and each feature is related to the other directly or indirectly.

Parametric Modeling

The parametric nature of a software package is defined as its ability to use the Standard properties or parameters in defining the shape and size of a geometry.

CATPart

CATPart is a file extension associated with all the files that are created in Sketcher, Part Design, and Wireframe and Surface Design workbenches of CATIA V5.

CATProduct

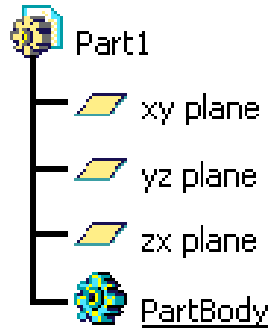
CATProduct is a file extension associated with all the files that are created in Assembly Design workbench of CATIA V5.

CATDrawing

CATDrawing is a file extension associated with all the files that are created in Drafting workbench of CATIA V5.

Specification Tree

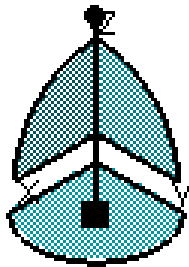
1. The specification tree keeps a track of all the operations that are carried on the part, as shown in the figure.
2. The specification tree that appears when you start a new file under the Part Design workbench, is as shown in the figure.



The specification tree that appears on starting a new CATPart file

Compass

1. It is a tool that is used to manipulate the orientation of parts, assemblies, or sketches.
2. You can also orient the view of the parts and assemblies.
3. By default, it appears on the top right corner of the geometry area.



The Compass

Constraints

1. Constraints are logical operations that are performed on the selected element to define its size and location with respect to other elements or reference geometries.
2. The constraints in **Sketcher** workbench are called geometric constraints and the constraints available in the **Assembly Design** workbench are called assembly constraints.

Geometric Constraints

These are the logical operations performed on sketched elements to define their size and position with respect to other elements.

Exercise 1

Aim: To draw the sketch of the model shown in Figure A. The sketch is shown in Figure B. You will not dimension the sketch. The solid model and the dimensions are given only for your reference.

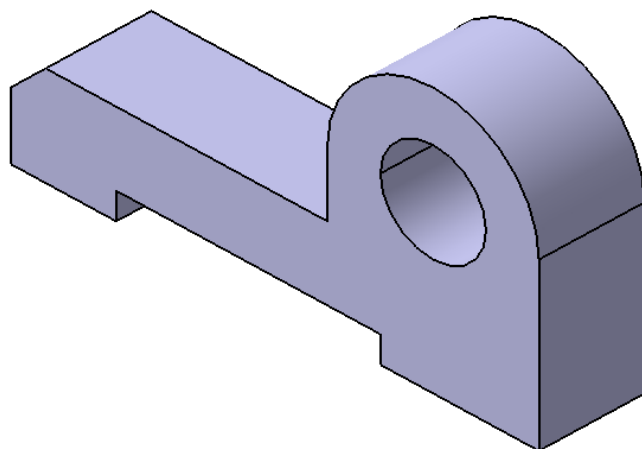


Figure A The solid model for Exercise 1

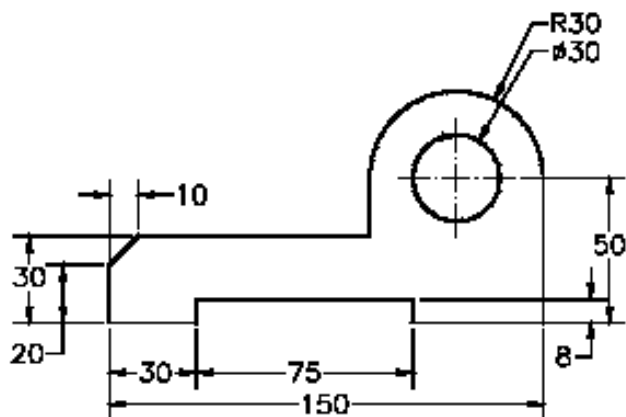


Figure B The sketch of the model

1. Start CATIA V5 and then start a new CAT part file.
2. Draw the sketch of the model using the **Line**, **Arc**, and **Circle** tools, as shown in **Figure C** and **Figure D**.

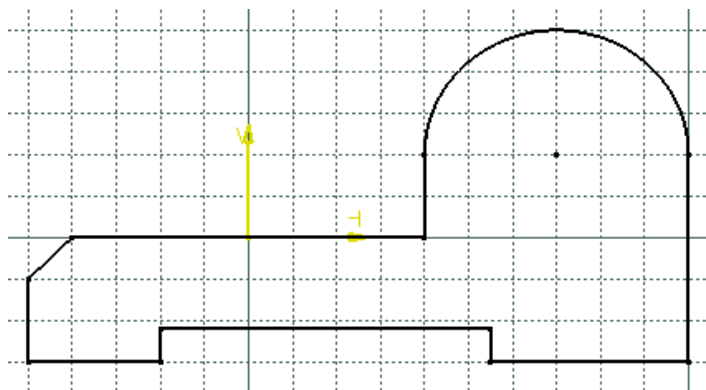


Figure C The outer loop of the sketch

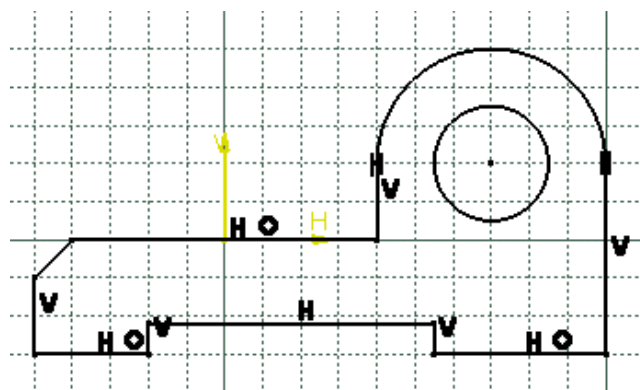


Figure D The final sketch for Exercise 1

3. Save the file

Result: Hence the required diagram is created using solid works

Exercise 2

Aim: To draw the sketch of the model shown in Figure A. The sketch is shown in Figure B. You will not dimension the sketch. The solid model and the dimensions are given only for your reference.

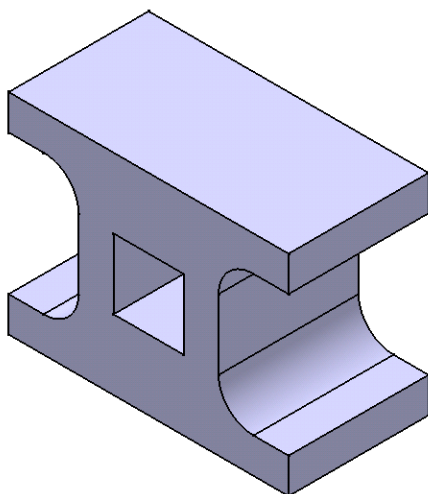


Figure A The solid model for Exercise 2

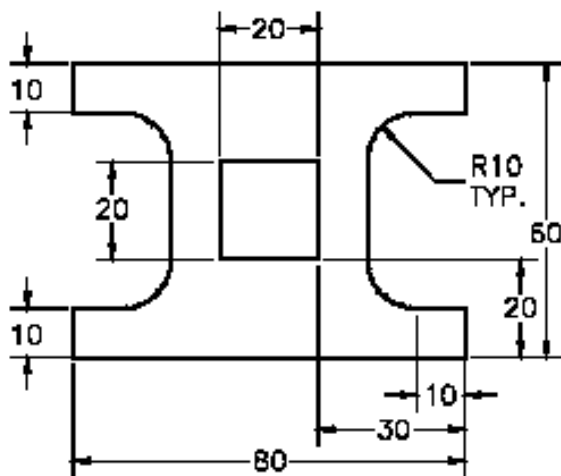


Figure B The sketch of the model

Procedure:

1. Start a new **CATpart** file.
2. Draw the sketch of the model using the **Profile** and **Rectangle** tool, as shown in **Figure C**, **Figure D** and **Figure E**.

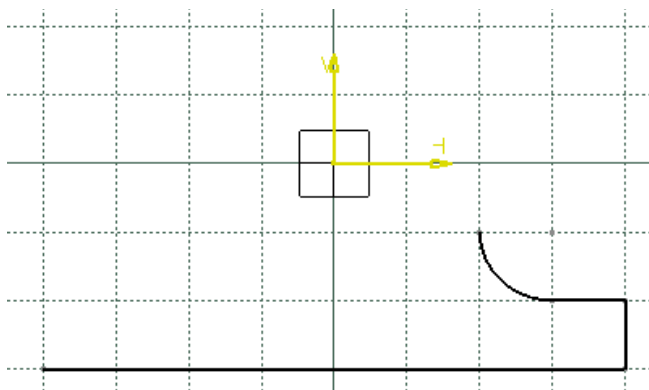


Figure C The sketch after drawing the three lines and a tangent arc

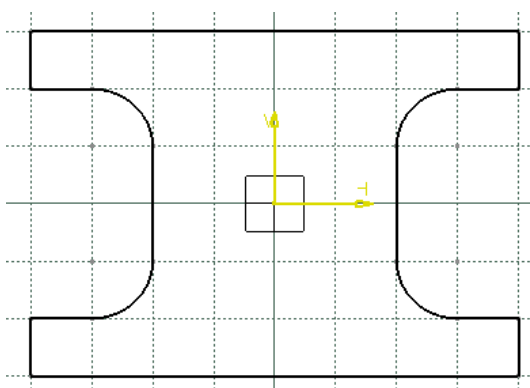


Figure D The sketch after drawing outer loop of the sketch

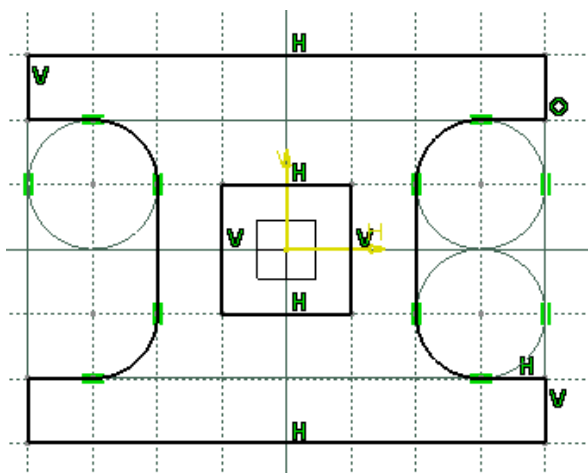


Figure E The final sketch after drawing the inner loop of the sketch

3. Save the file

Result: Hence the required diagram is created using solid works

Exercise 3

Aim: To draw the sketch of the model, as shown in **Figure A**. The sketch is shown in **Figure B**. You will not dimension the sketch. The solid model and dimensions are given for your reference.

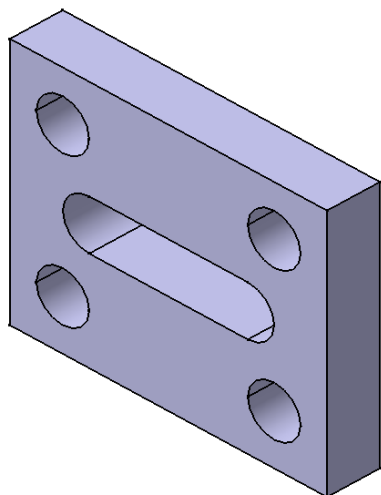


Figure A The solid model for Exercise 3

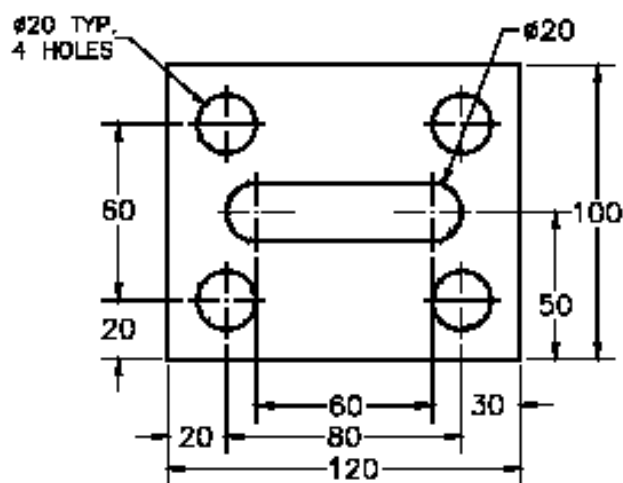


Figure B The sketch for the solid model

Procedure :

1. Start a new **CATpart** file.
2. Draw the sketch of the model using the **Rectangle**, **Profile**, and **Circle** tools, as shown in **Figure C**, **Figure D** and **Figure E**.

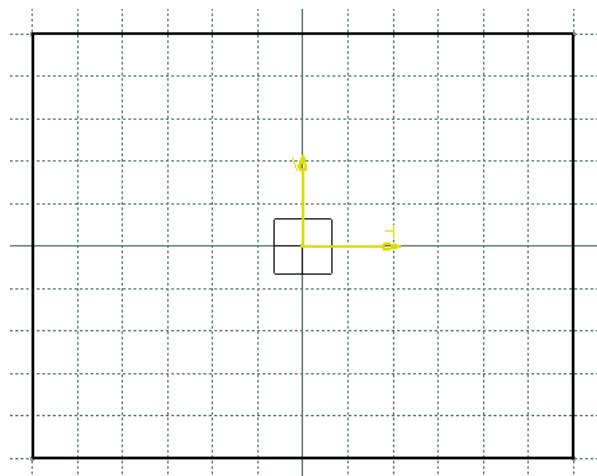


Figure C The outer loop of the sketch

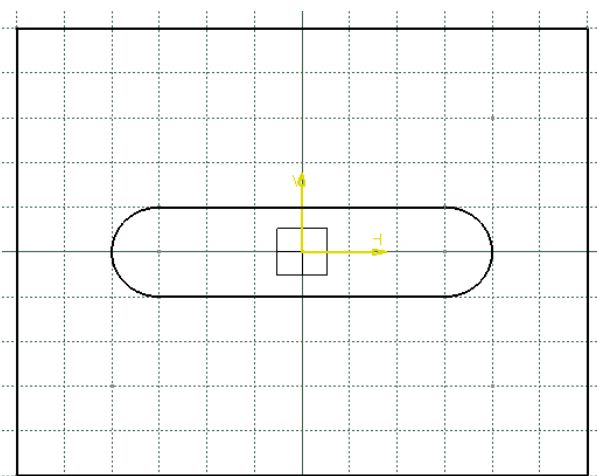


Figure D The sketch after drawing the elongated hole

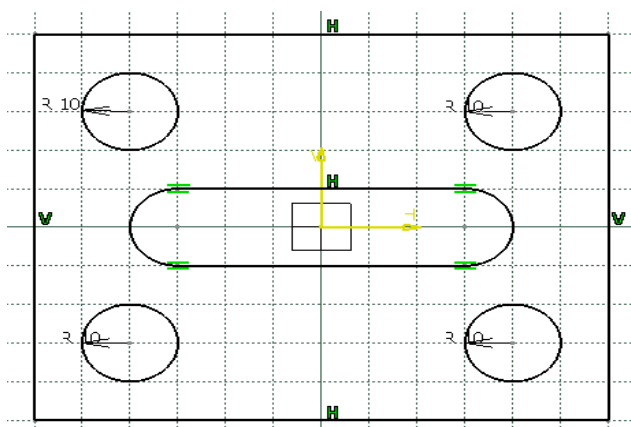


Figure E The final sketch

3. Save the file

Result: Hence the required diagram is created using solid works

Exercise 4

Aim: To draw the sketch of the model shown in Figure A. The sketch is shown in Figure B. Do not dimension the sketch. The solid model and the dimensions are given only for your reference.

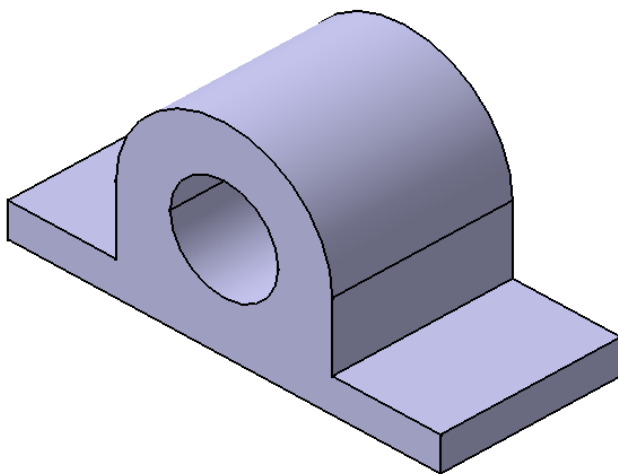


Figure A The solid model for Exercise 4

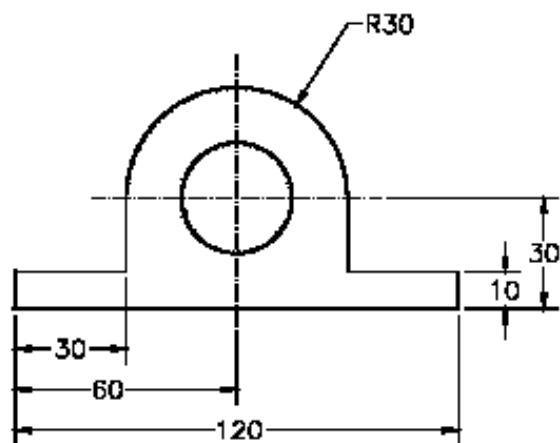


Figure B The sketch for the solid model

Procedure:

1. Start a new **CATpart** file.
2. Draw the sketch of the model using the **Profile** and the **Circle** tools, as shown in **Figure C** and **Figure D**.

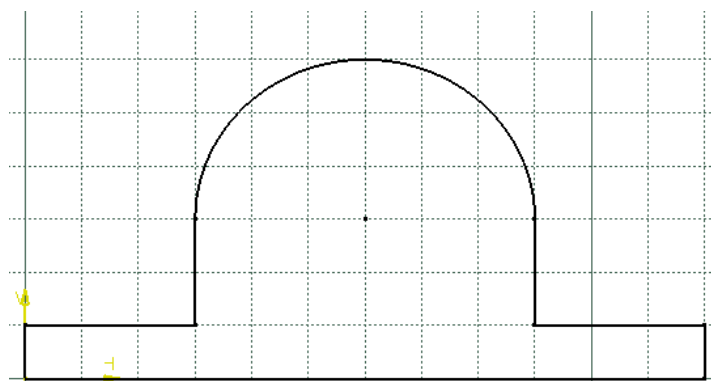


Figure C The sketch after drawing the outer loop

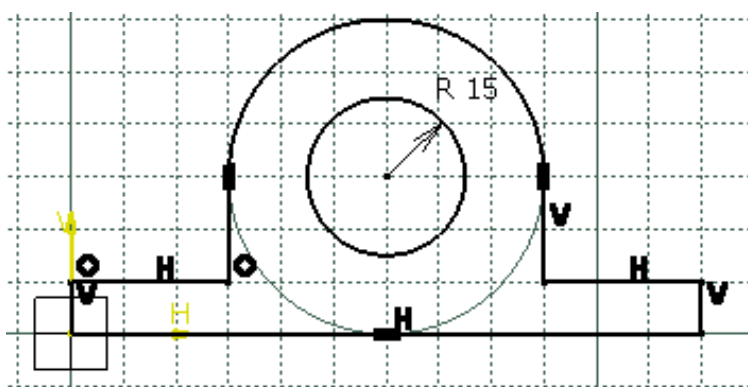


Figure D The final sketch for Tutorial 4

3. Save the file

Result: Hence the required diagram is created using solid works

Exercise 5

Aim: To draw the sketch of the model shown in **Figure A**. The sketch is shown in **Figure B**. You will not dimension the sketch. The solid model and the dimensions are given only for your reference.

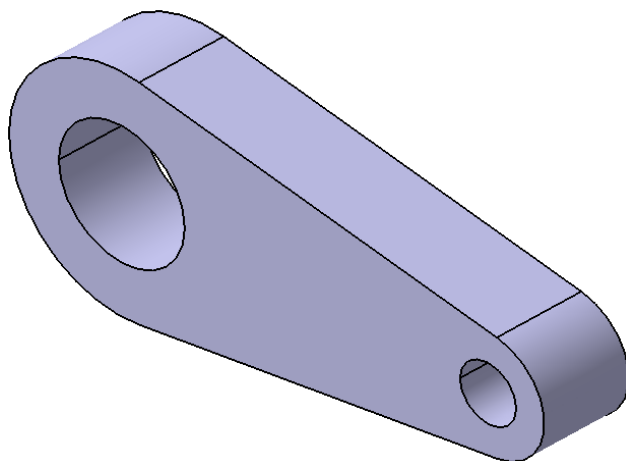


Figure A The Model for Exercise 5

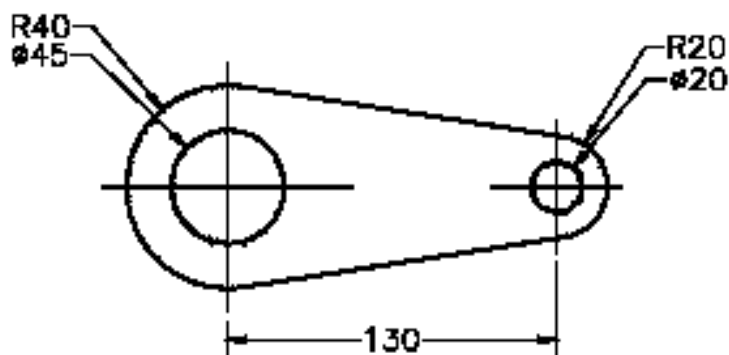


Figure B The sketch for Tutorial 1

Procedure:

1. Start a new file in the **Part** workbench and draw the outer loop of the sketch using the **Circle** and **By-Tangent Line** tool, as shown in **Figure C**, **Figure D**, **Figure E** and **Figure F**.

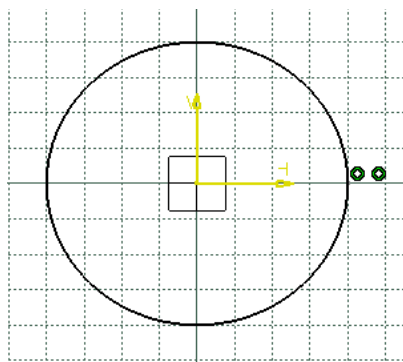


Figure C The Sketch after drawing the first circle

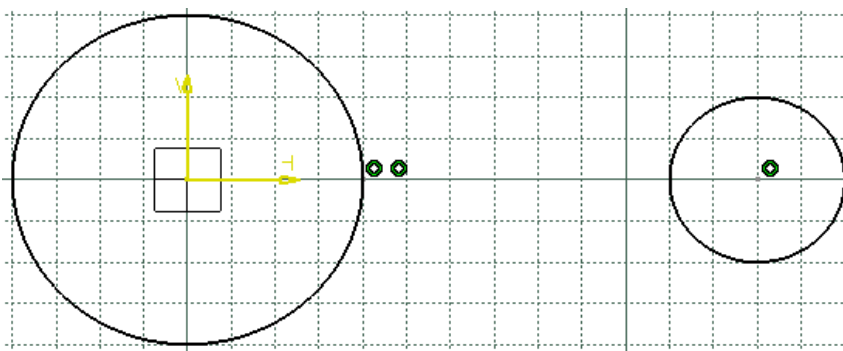


Figure D The Sketch after drawing the second circle

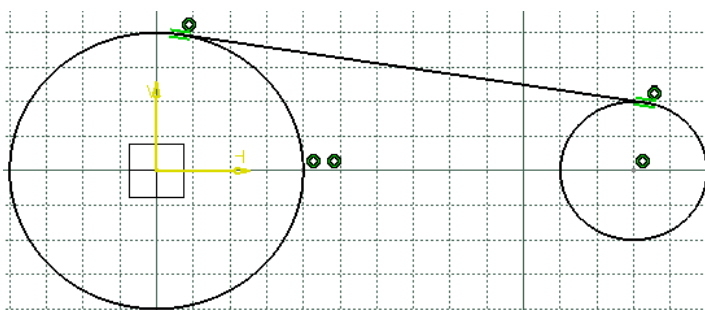


Figure E The sketch after drawing the first tangent line

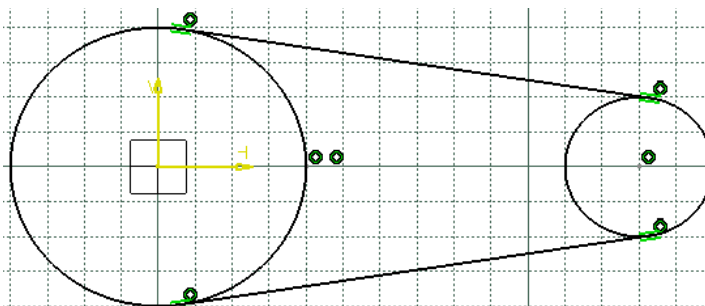


Figure F The sketch, after drawing the second tangent line

2. Trim the unwanted portion of the outer loop of the sketch using the **Quick Trim** tool, as shown in **Figure G** and **Figure H**.

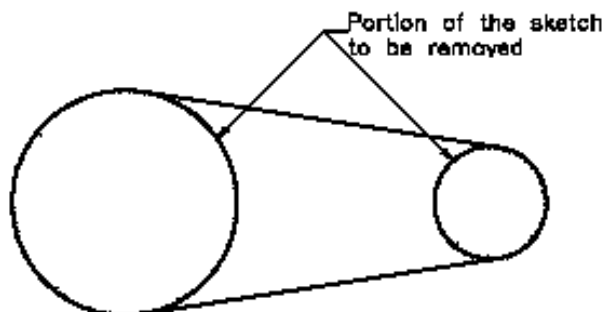


Figure G The unwanted portion of the sketch to be trimmed

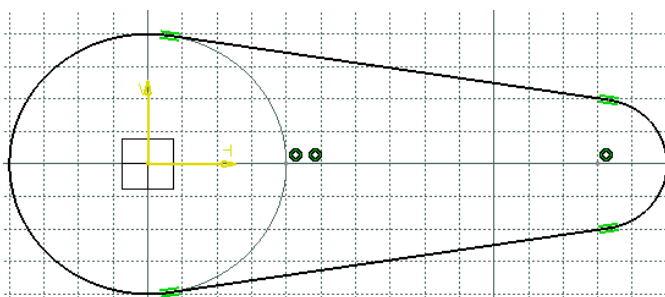


Figure H The sketch after trimming the unwanted portion

3. Draw the inner loops of the sketch using the **Circle** tool, as shown in **Figure I**.

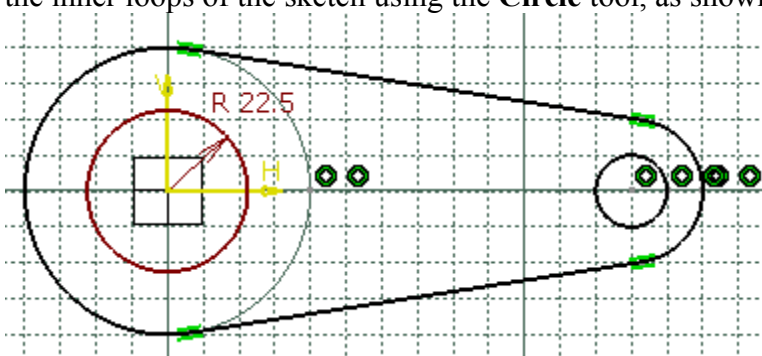


Figure I Final sketch

4. Save the file

Result: Hence the required diagram is created using solid works

ANSYS 10.0

INTRODUCTION TO FEA

1. The finite element analysis (FEA) is a computing technique that is used to obtain Approximate solutions to the boundary value problems in engineering.
2. It uses a numerical technique called the finite element method (FEM) to solve boundary value problems.
3. FEA involves a computer model of a design that is loaded and analyzed for specific results
4. The concept of FEA can be explained with a small example of measuring the perimeter of a circle, refer to Figure 1.

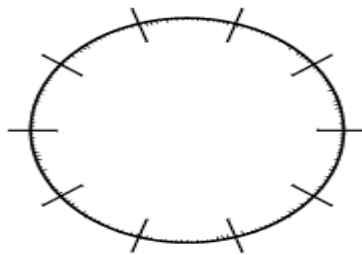


Figure 1 The Circle divided into equal small segments

General Working of FEA

Better knowledge of FEA will help you build more accurate models. It will also help you understand the backend working of ANSYS. Figure 2 shows a spring assembly that represents a simple two-spring element model.

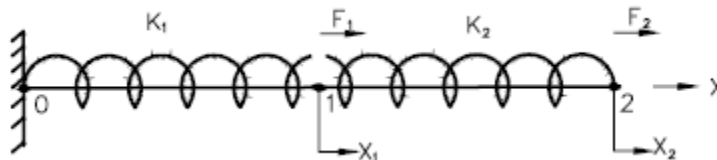


Figure 2 Representation of a two-spring assembly

These FEA terminologies are listed next.

Stiffness Matrix

The following part represents the stiffness matrix (K):

$$\begin{bmatrix} K_1 + K_2 & -K_2 \\ -K_2 & K_2 \end{bmatrix}$$

Degrees of Freedom

Degrees of freedom is defined as the ability of a node to translate or transmit the load.

Boundary Conditions

The boundary conditions are used to eliminate the unknowns in the system.

Nodes, Elements, and Element Shapes

These concepts are discussed next.

Nodes

Nodes are similar to the points in geometry and represent the corner points of an element, refer to Figure 3.

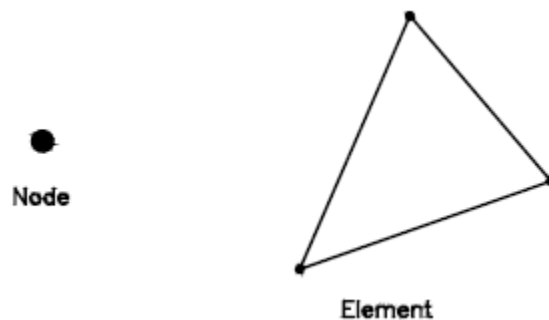


Figure 3 A node and an element

Elements

An element shape is specified by nodes. An element (triangular shaped) is shown in Figure 3.

Element Shapes

The following are the basic shapes of the elements:

Point Element

A point element is in the form of a point and therefore has only one node

Line Element

A line element has the shape of a line or curve, therefore a minimum of two nodes are required to define it, refer to Figure 4.

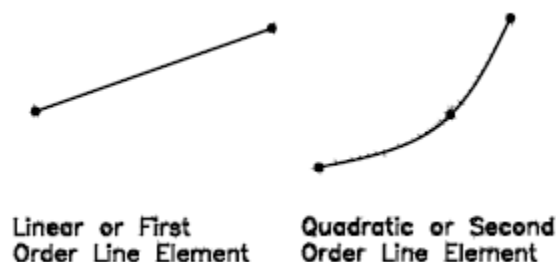


Figure 4 The line elements

Area Element

An area element has the shape of a quadrilateral or a triangle, therefore it requires a minimum of three or four nodes to define it. Some area elements are shown in Figure 5.

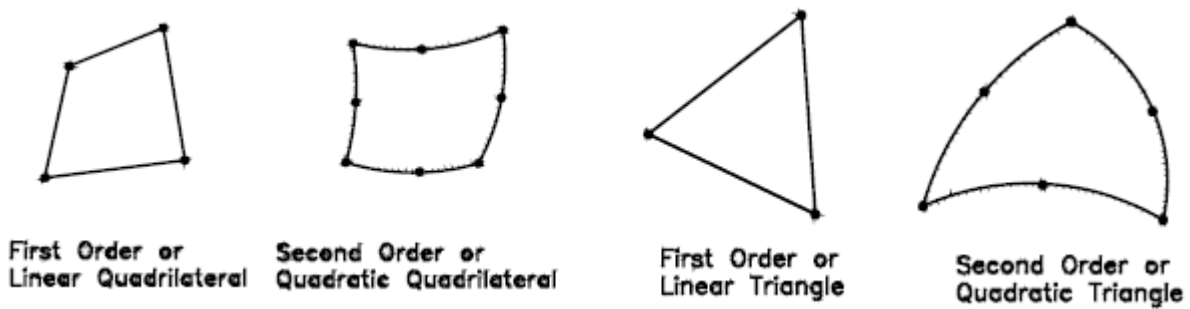


Figure 5 The area elements

Volume Element

A volume element has the shape of a hexahedron (8 nodes), wedge (6 nodes), tetrahedron (4 nodes), or a pyramid (5 nodes). Some of the volume elements are shown in Figure 6.

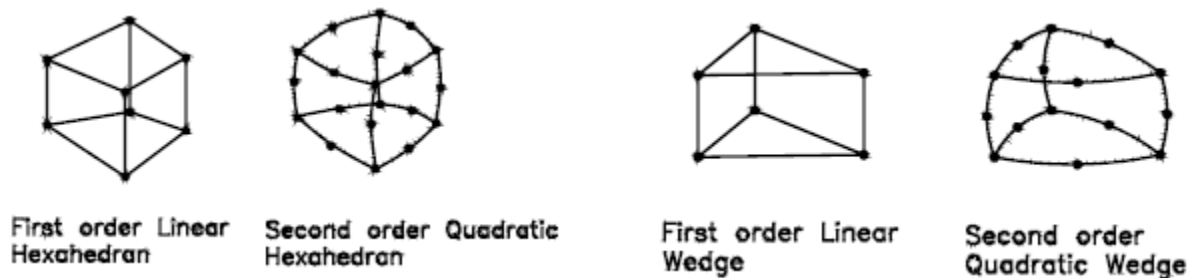


Figure 6 The volume elements

General Procedure of Conducting Finite Element Analysis

1. Set the type of analysis to be used.
2. Create the model.
3. Define the element type.
4. Divide the given problem into nodes and elements (mesh the model).
5. Apply material properties and boundary conditions.
6. Derive the element matrices and equations.
7. Assemble the element equations.
8. Solve the unknown quantities at nodes.
9. Interpret the results.

FEA through ANSYS

In ANSYS, the general process of finite element analysis is divided into three main phases, preprocessor, solution, and postprocessor, refer to Figure 7.

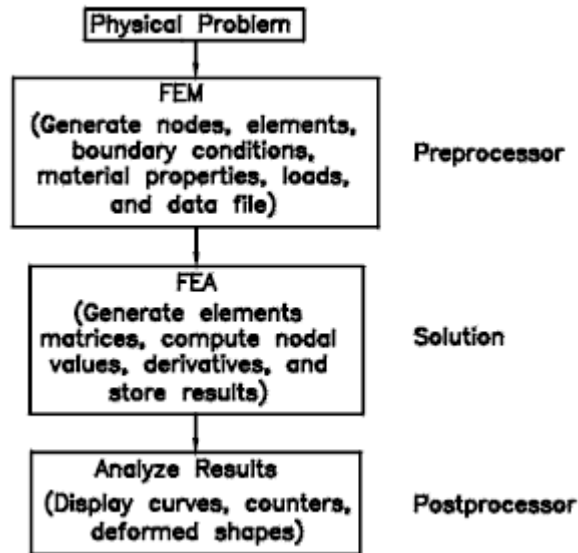


Figure 7 FEA through ANSYS

Preprocessor

The preprocessor is a program that processes the input data to produce the output that is used as input to the subsequent phase (solution), refer to Figure 7.

Solution

Solution phase is completely automatic. The FEA software generates the element matrices, computes nodal values and derivatives, and stores the result data in files.

Postprocessor

The output from the solution phase (result data files) is in the numerical form and consists of nodal values of the field variable and its derivatives.

Effective Utilization of FEA

Some prerequisites for effective utilization of FEA from engineers and FEA software are discussed next.

Engineers

An engineer who wants to work with this tool should have sound knowledge of Strength of Materials (for structural analysis), Heat Transfer, Thermodynamics (for thermal analysis), and a good analytical/designing skill.

Software

The FEA software should be selected based on the following considerations:

1. Analysis type to be performed.
2. Flexibility and accuracy of the tool.
3. Hardware configuration of your system.

FEA Software

There are variety of commercial FEA software packages available in market. Some of the most popular and commercially available FEA software are as follows:

1. ANSYS
2. Adina
3. Abaqus
4. Hypermesh
5. NX Nastran

6. Cosmos
7. NISA
8. Marc
9. Ls-Dyna
10. MSC/Dytran
11. Star-CD

ANSYS, developed by ANSYS, Inc., USA, is a dedicated Computer Aided Finite Element Modeling and Finite Element Analysis tool. ANSYS is known as the standard in the field of Computer Aided Engineering. The Graphical User Interface (GUI) of ANSYS enables the user to work with 3-dimensional (3D) models and also generate results from them.

SYSTEM REQUIREMENTS

1. System unit: An Intel Pentium 4 or AMD 64, running Microsoft 2000 Professional Edition, Windows XP 32-bit, or Windows XP x64 on it.
2. Memory: 512 MB of RAM is the minimum requirement for all applications. However, for 64-bit systems, 1 GB of RAM is the minimum requirement.
3. Disk drive: 2.2 GB Disk Drive space (Minimum recommended size).
4. A DVD drive is required for the program installation.
5. Graphics adapter: Graphics card compatible with the supported operating systems, capable of supporting 1024x768 High Color (16-bit), and a 17-inch monitor compatible with this type of graphics card.

Exercise 1

Aim: To create the beam shown in Figure 1. The dimensions of the beam are shown in the same figure. Save it with the name mentioned below.

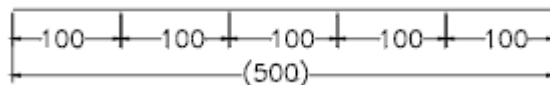


Figure 1 The beam

Procedure:

1. Invoke the **ANSYS Product Launcher** window and then start a new job.
2. Create keypoints using the **In Active CS** option, refer to Figure 2 and 3.

Keypoint Number	coordinate		
	X	Y	Z
2	100	0	0
3	200	0	0
4	300	0	0
5	400	0	0
6	500	0	0

Figure 2 The coordinate values for keypoints



Figure 3 Keypoints created

3. Create lines using the **Straight Line** option, refer to Figure 3 and 4
4. Save the job and exit the ANSYS session.

Line No.	Keypoint No.	
	From	To
2	2	3
3	3	4
4	4	5
5	5	6

Figure 3 The keypoints for creating lines

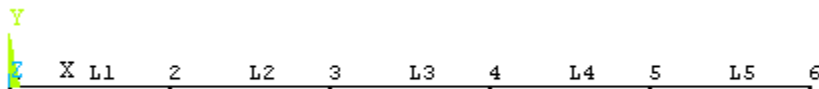


Figure 4 The beam created

Result: Hence the required beam is created using ANSYS

Exercise 2

Aim: to create the truss shown in Figure 1. Save it with the name mentioned below.

Procedure:

1. Invoke the **ANSYS Product Launcher** window and then start a new job.

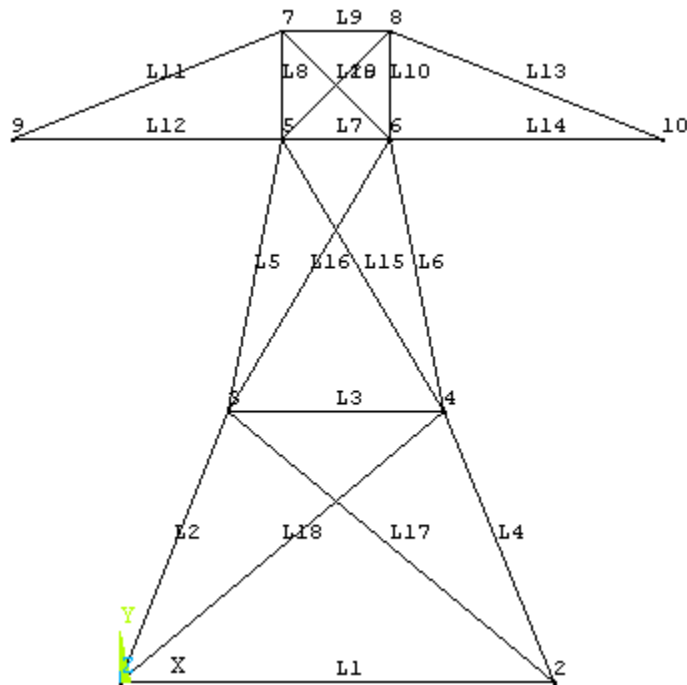


Figure 1 The truss

2. Create keypoints using the **In Active CS** option, refer to Figure 2 and 3

Keypoint Number	coordinate		
	X	Y	Z
2	400	0	0
3	100	250	0
4	300	250	0
5	150	500	0
6	250	500	0
7	150	600	0
8	250	600	0
9	-100	500	0
10	500	500	0

Figure 2 The coordinate values for keypoints

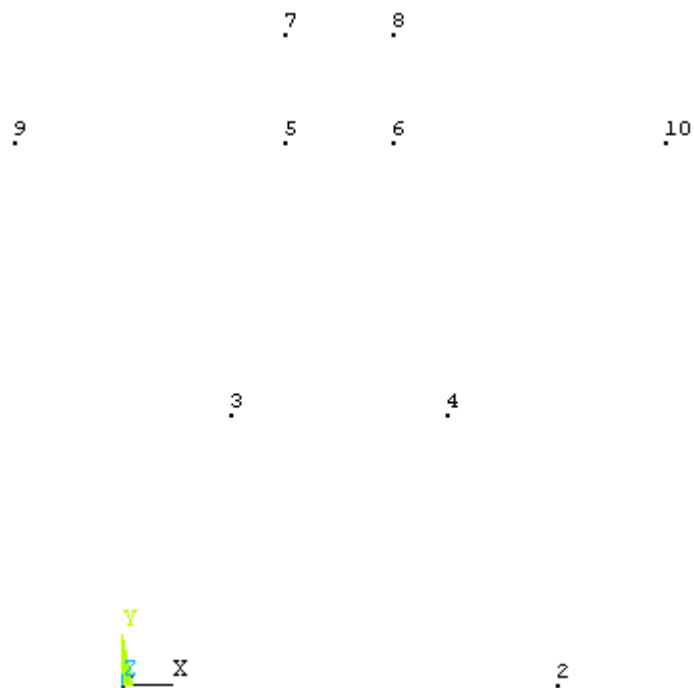


Figure 3 Keypoints created

Create lines using the Straight Line option, refer to Figure 4 and 5.

Line No.	Keypoint No.	
	From	To
2	1	3
3	3	4
4	4	2
5	3	5
6	4	6
7	5	6
8	5	7
9	7	8
10	6	8
11	7	9
12	5	9
13	8	10
14	6	10
15	4	5
16	3	6
17	2	3
18	1	4
19	5	8
20	6	7

Figure 3 Keypoints for creating lines

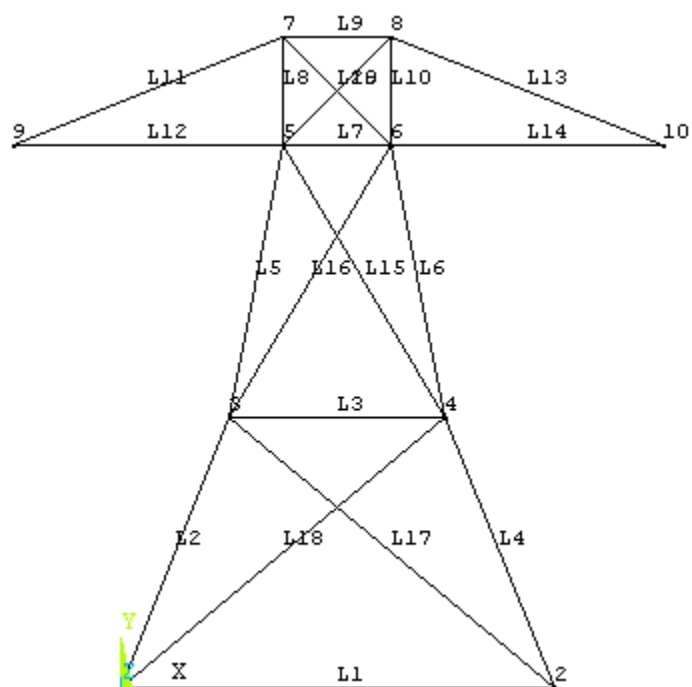


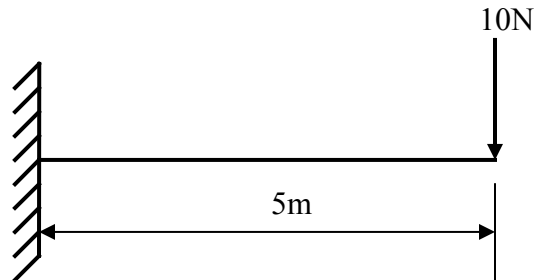
Figure 4 The truss created

Result: Hence the required truss is created using ANSYS

Structural Analysis

Exercise 3

Aim: To find the displacement, maximum, minimum stresses induced in a given cantilever beam and draw the shear force and bending moment diagrams by using ANSYS tool, also list the results according to the given loads.



Procedure:

Preferences:

Main menu > preferences > check structural

Preprocessor:

1. Main menu > preprocessor > element type > add/edit/delete > Add > select Beam -2D elastic3 > click ok > close.
2. Main menu > preprocessor > Real constraints > Add/Edit/Delete > Add > Click ok > Enter the following values Area=1, Area moment of Inertia (IZZ)=1/12, Height=1, > click ok > close.
3. Main menu > preprocessor > Material properties > Material model > structural > linear > elastic > isotropic > enter Ex as 2E5 & PRXY as 0.3 > click ok > close.
4. Main menu > preprocessor > modeling > create > nodes > Inactive CS > (Enter X<Y<Z location values) as shown in the figure starting from node-1 i.e., 0,0,0 and 10,0,0 thus two nodes are created.
5. Main menu > preprocessor > modeling > create > element > auto numbered > through nodes (select node by node and middle click) with this elements are created.

Solution:

1. Solution > define loads > Apply > structural > displacement > pick on nodes (Pick the node which are to be constrained with direction) > middle click > select DOF or UX or UY or UZ as required > click ok.
Pick node-1 > middle click > select all DOF since to constrain the beam.
2. Solution > define loads > Apply > structural > force/moment > on nodes > select node > middle click > select FX and FY or MZ depending on the problem > click ok. In the above problem load acting downwards i.e. in negative Y-direction at node -2 of about 10N.

General post processor:

- 1) General post processor > element table > define table > add select by sequence number > enter SMISC , 2,4,6,8,12 one after the other > click ok > close.
- 2) General post processor > plot results > deformed shape > click ok.
- 3) General post processor > plot results > contour plot > nodal solution > click ok.
- 4) General post processor > plot results > contour plot > element solution > click ok.
- 5) General post processor > plot results > contour plot > line element results > use the
- 6) Combination 2,8 for shear force diagram and 6,12 for bending moment diagram.
- 7) General post processor > plot results > vector plot > predefined > click ok.
- 8) General post processor > list results > nodal solution > click ok.
- 9) General post processor > list results > element solution > click ok.

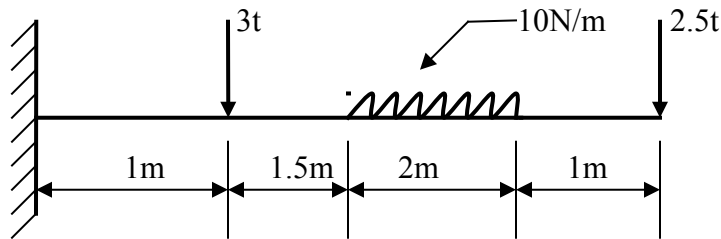
Result: Thus the analysis on the given beam is performed using ANSYS tool, Shear force ,bending moment diagrams are captured and values are noted.

Viva Questions:

1. What is ANSYS?
2. Why ANSYS is used?
3. What is Structural Analysis?
4. What is Thermal Analysis?
5. What is the difference between Structural Analysis and Thermal Analysis?
6. What is node?
7. What is element?
8. What is difference between node and element?
9. What are the minimum requirements to develop a drawing ANSYS?
10. What is general post processor?

Exercise 4

Aim: To find the displacement, maximum, minimum stresses induced in a given cantilever beam with uniformly distributed load and point loads and draw the shear force and bending moment diagrams by using ANSYS tool, also list the results according to the given loads.



Procedure:

Preferences:

Main menu > preferences > check structural

Preprocessor:

- 1) Main menu > preprocessor > element type > add/edit/delete > Add > select Beam -2D elastic3 > click ok > close.
- 2) Main menu > preprocessor > Real constraints > Add/Edit/Delete > Add > Click ok > Enter the following values Area=1, Area moment of Inertia (IZZ)=1/12, Height=1, > click ok > close.
- 3) Main menu > preprocessor > Material properties > Material model > structural > linear > elastic > isotropic > enter Ex as 2E5 & PRXY as 0.3 > click ok > close.
- 4) Main menu > preprocessor > modeling > create > nodes > Inactive CS > (Enter X<Y<Z location values) as shown in the figure starting from node-1 i.e., 0,0,0 and 1,0,0, 2.5,0,0, 4.5,0,0, 5,0,0 thus five nodes are created.
- 5) Main menu > preprocessor > modeling > create > element > auto numbered > through nodes (select node by node and middle click) with this elements are created.

Solution:

1. Solution > define loads > Apply > structural > displacement > pick on nodes (Pick the node which are to be constrained with direction) > middle click > select DOF or UX or UY or UZ as required > click ok.
Pick node-1 > middle click > select all DOF since to constrain the beam.

2. Solution > define loads > Apply > structural > force/moment > on nodes > select node > middle click > select FX and FY or MZ depending on the problem > click ok. In the above problem load acting downwards i.e. in negative Y-direction at node -2 and 5 of about 3tones and 2.5 tones.
3. Solution > define loads > apply > structural > pressure (applied or uniformly distributed load) > on beams > select beam middle click(Enter pressure value at the both ends) i.e., at I & J , enter 1 and 1 > click ok
Solution > solve > click ok.

General post processor:

1. General post processor > element table > define table > add select by sequence number > enter SMISC , 2,4,6,8,12 one after the other > click ok > close.
2. General post processor > plot results > deformed shape > click ok.
3. General post processor > plot results > contour plot > nodal solution > click ok.
4. General post processor > plot results > contour plot > element solution > click ok.
5. General post processor > plot results > contour plot > line element results > use the
6. Combination 2,8 for shear force diagram and 6,12 for bending moment diagram.
7. General post processor > plot results > vector plot > predefined > click ok.
8. General post processor > list results > nodal solution > click ok.
9. General post processor > list results > element solution > click ok.

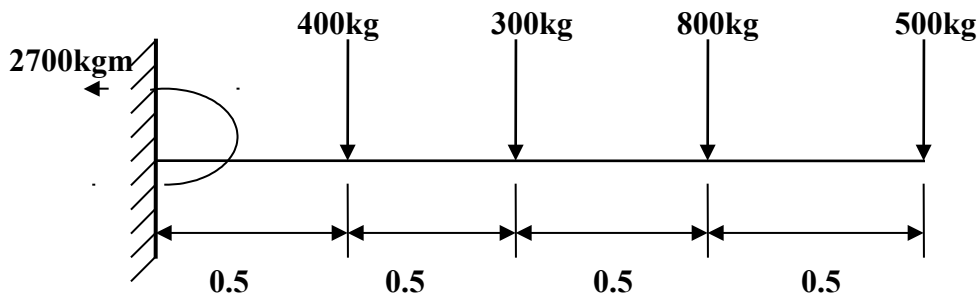
Result: Thus the analysis on the given beam is performed using ANSYS tool, Shear force, bending moment diagrams are captured and values are noted.

Viva Questions:

1. What is nodal solution?
2. What is pre processor?
3. What is post processor?
4. What is modeling?
5. What type of options we use in preferences?
6. ANSYS is it a tool or software?
7. What is the difference between Animation and Modeling?
8. What is DOF?
9. How to create a Node?
10. How to create an Element?

Exercise 5

Aim: To find the displacement, maximum, minimum stresses induced in a given cantilever beam with uniformly distributed load and point loads and draw the shear force and bending moment diagrams by using ANSYS tool, also list the results according to the given loads.



Procedure:

Preferences:

Main menu > preferences > check structural

Preprocessor:

1. Main menu > preprocessor > element type > add/edit/delete > Add > select Beam -2D elastic3 > click ok > close.
2. Main menu > preprocessor > Real constraints > Add/Edit/Delete > Add > Click ok > Enter the following values Area=1, Area moment of Inertia (IZZ)=1/12, Height=1, > click ok > close.
3. Main menu > preprocessor > Material properties > Material model > structural > linear > elastic > isotropic > enter Ex as 2E5 & PRXY as 0.3 > click ok > close.
4. Main menu > preprocessor > modeling > create > nodes > Inactive CS > (Enter X<Y<Z location values) as shown in the figure starting from node-1 i.e., 0,0,0 and 1,0,0, 0.5,0,0, 1,0,0, 1.5,0,0 , 2,0,0 thus six nodes are created.
5. Main menu > preprocessor > modeling > create > element > auto numbered > through nodes (select node by node and middle click) with this elements are created.

Solution:

1. Solution > define loads > Apply > structural > displacement > pick on nodes (Pick the node which are to be constrained with direction) > middle click > select DOF or UX or UY or UZ as required > click ok.
Pick node-1 > middle click > select all DOF since to constrain the beam.

2. Solution > define loads > Apply > structural > force/moment > on nodes > select node > middle click > select FX and FY or MZ depending on the problem > click ok. In the above problem load acting downwards i.e. in negative Y-direction at node -2 ,3,4,5 and 5 of about 400kg.300kg.800kg.500kg and at node-1 a force in MZ direction of about 2700 kgm is acting , pick node-1 > middle click > select MZ > enter value as 2700.

General post processor:

1. General post processor > element table > define table > add select by sequence number > enter SMISC , 2,4,6,8,12 one after the other > click ok > close.
2. General post processor > plot results > deformed shape > click ok.
3. General post processor > plot results > contour plot > nodal solution > click ok.
4. General post processor > plot results > contour plot > element solution > click ok.
5. General post processor > plot results > contour plot > line element results > use the
6. Combination 2,8 for shear force diagram and 6,12 for bending moment diagram.
7. General post processor > plot results > vector plot > predefined > click ok.
8. General post processor > list results > nodal solution > click ok.
9. General post processor > list results > element solution > click ok.

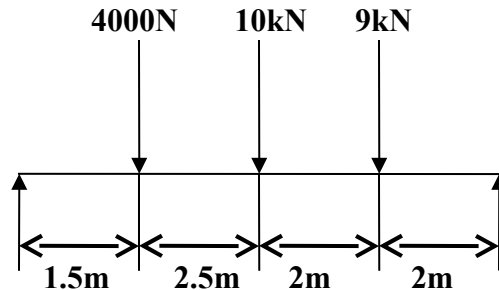
Result: Thus the analysis on the given beam is performed using ANSYS tool, Shear force, bending moment diagrams are captured and values are noted.

Viva Questions:

1. What is steady state Analysis?
2. What is unsteady state Analysis?
3. What are the types of Thermal Analysis?
4. What is task in Thermal Analysis?
5. How to apply loads in Thermal Analysis?
6. How to apply loads in Structural Analysis?
7. What is load?
8. What is structure?
9. What is beam?
10. What is Shear force?

Exercise 6

Aim: To find the displacement, maximum, minimum stresses induced in a given simply supported beam and draw the shear force and bending moment diagrams by using ANSYS tool, also list the results according to the given loads.



Procedure:

Preferences:

Main menu > preferences > check structural

Preprocessor:

1. Main menu > preprocessor > element type > add/edit/delete > Add > select Beam -2D elastic3 > click ok > close.
2. Main menu > preprocessor > Real constraints > Add/Edit/Delete > Add > Click ok > Enter the following values Area=1, Area moment of Inertia (IZZ)=1/12, Height=1, > click ok > close.
3. Main menu > preprocessor > Material properties > Material model > structural > linear > elastic > isotropic > enter Ex as 2E5 & PRXY as 0.3 > click ok > close.
4. Main menu > preprocessor > modeling > create > nodes > Inactive CS > (Enter X<Y<Z location values) as shown in the figure starting from node-1 i.e., 0,0,0 and 1.5,0,0, 3.5,0,0, 5.5,0,0, 7.5,0,0 , 2,0,0 thus five nodes are created.
5. Main menu > preprocessor > modeling > create > element > auto numbered > through nodes (select node by node and middle click) with this elements are created.

Solution:

1. Solution > define loads > Apply > structural > displacement > pick on nodes (Pick the node which are to be constrained with direction) > middle click > select DOF or UX or UY or UZ as required > click ok
Pick node-1 > middle click > select all DOF since to constrain the beam.
2. Solution > define loads > Apply > structural > force/moment > on nodes > select node > middle click > select FX and FY or MZ depending on the problem > click ok. In the above problem load acting downwards i.e. in negative Y-direction at node -2, 3,4 of about 4KN, 10KN, 7KN.

General post processor:

1. General post processor > element table > define table > add select by sequence number > enter SMISC , 2,4,6,8,12 one after the other > click ok > close.
2. General post processor > plot results > deformed shape > click ok.
3. General post processor > plot results > contour plot > nodal solution > click ok.
4. General post processor > plot results > contour plot > element solution > click ok.
5. General post processor > plot results > contour plot > line element results > use the
6. Combination 2,8 for shear force diagram and 6,12 for bending moment diagram.
7. General post processor > plot results > vector plot > predefined > click ok.
8. General post processor > list results > nodal solution > click ok.
9. General post processor > list results > element solution > click ok.

Result: Thus the analysis on the given beam is performed using ANSYS tool, Shear force, bending moment diagrams are captured and values are noted

Viva Questions:

1. What is Bending Moment?
2. What is Cantilever Beam?
3. What is Simply Supported Beam?
4. What are the types of Beams?
5. What are the types of Loads?
6. What is overhanging Beam?
7. What is UDL?
8. What is Work?
9. What are the units for Work?
10. What is General post processor?

Exercise 7

Aim:

To find the displacement, maximum, minimum stresses induced in a given cantilever beam with uniformly distributed load and point loads and draw the shear force and bending moment diagrams by using ANSYS tool, also list the results according to the given loads.

Procedure:
Preferences:

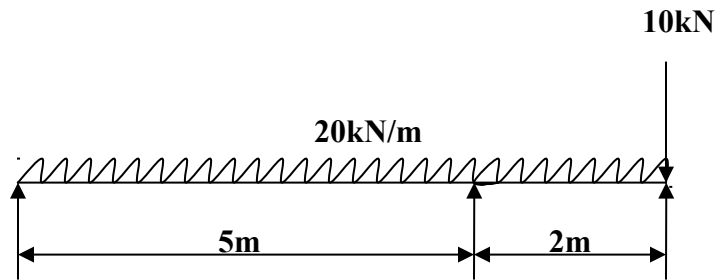
Main menu > preferences > check structural

Preprocessor:

2. Main menu > preprocessor > element type > add/edit/delete > Add > select Beam -2D elastic3 > click ok > close.
3. Main menu > preprocessor > Real constraints > Add/Edit/Delete > Add > Click ok > Enter the following values Area=1, Area moment of Inertia (IZZ)=1/12, Height=1, > click ok > close.
4. Main menu > preprocessor > Material properties > Material model > structural > linear > elastic > isotropic > enter Ex as 2E5 & PRXY as 0.3 > click ok > close.
5. Main menu > preprocessor > modeling > create > nodes > Inactive CS > (Enter X<Y<Z location values) as shown in the figure starting from node-1 i.e., 0,0,0 and 5,0,0, 1,0,0, 7,0,0 thus three nodes are created.
6. Main menu > preprocessor > modeling > create > element > auto numbered > through nodes (select node by node and middle click) with this elements are created.

Solution:

1. Solution > define loads > Apply > structural > displacement > pick on nodes (Pick the node which are to be constrained with direction) > middle click > select DOF or UX or UY or UZ as required > click ok.
Pick node-1 > middle click > select all DOF since to constrain the beam.
2. Solution > define loads > Apply > structural > force/moment > on nodes > select node > middle click > select FX and FY or MZ depending on the problem > click ok. In the above problem load acting downwards i.e. in negative Y-direction at node -2 and 5 of about 3tones and 2.5tones.
3. Solution > define loads > Apply > structural > pressure (applied or uniformly distributed load) > on beams > select beam middle click (Enter pressure value at the both ends) i.e. at I & J enter 20 and 20 > click ok.



General post processor:

1. General post processor > element table > define table > add select by sequence number > enter SMISC , 2,4,6,8,12 one after the other > click ok > close.
2. General post processor > plot results > deformed shape > click ok.
3. General post processor > plot results > contour plot > nodal solution > click ok.
4. General post processor > plot results > contour plot > element solution > click ok.
5. General post processor > plot results > contour plot > line element results > use the
6. Combination 2, 8 for shear force diagram and 6, 12 for bending moment diagram.
7. General post processor > plot results > vector plot > predefined > click ok.
8. General post processor > list results > nodal solution > click ok.
9. General post processor > list results > element solution > click ok.

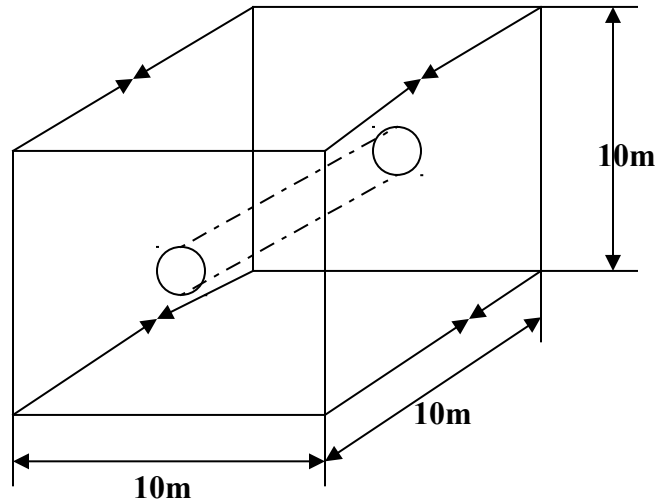
Result: Thus the analysis on the given beam is performed using ANSYS tool, Shear force, bending moment diagrams are captured and values are noted

Viva Questions:

1. What is Temperature?
2. How to convert degree Celsius's to Fahrenheit Heat?
3. What are the units of Temperature?
4. What is Zero Temperature?
5. What is UVL?
6. What is Heat?
7. What are the units of Heat?
8. Draw the B.M of U.D.L?
9. What is point of contraflexure?
10. In what beams point of contraflexure occurs?

Structural Analysis Solid Modeling

Aim: To find the displacement, maximum, minimum stresses induced in a given solid structure with uniformly distributed load and point loads by using ANSYS tool, also list the results according to the given loads.



Procedure:

Preferences:

Main menu > preferences > check structural

Preprocessor:

1. Main menu > preprocessor > element type > add/edit/delete > Add > select Brick 8 node 45 > ok > close.
2. Main menu > preprocessor > Material properties > Material model > structural > linear > elastic > isotropic > enter Ex as 2E5 & PRXY as 0.3 > click ok > close.
3. Main menu > preprocessor > modeling > create > key points > enter values as follows i.e., 0,0,0 , 10,0,0, 10,0,0, 0,10.0, 5,5,0 thus key points are generated.
4. Main menu > preprocessor > modeling > create > lines > straight lines> pick each key point individually starting from 1 to 1 > click ok(with this lines square is created).
5. Main menu > preprocessor > modeling > create > lines > arcs > full circle > pick the key point 5 draw a circle with radius nearly 2.
6. Main menu > preprocessor > modeling > create > areas > arbitrary > by lines > pick straight lines of a square thus area of square is created > plot > lines.

7. Main menu > preprocessor > modeling > create > areas > arbitrary > by lines > pick circle lines, thus circle area is created.
8. Main menu > preprocessor > modeling > operate > Booleans > subtract > areas pick square area first > click ok > pick circle area > click ok , then circle area is subtracted from the square area.
9. Main menu > preprocessor > modeling > operate > extrude > areas > along normal > pick the structure on the screen > click ok > enter length of extrude value 10 > click ok.
10. Main menu > preprocessor > meshing > mesh tool > click on global set > enter element edge length value as 0.1 > click ok > click. Mesh > select solid structure on the screen.

Solution:

- 17 Solution > define loads > Apply > structural > displacement > pick on nodes (Pick the node which are to be constrained with direction) > middle click > select DOF or UX or UY or UZ as required > click ok In this problem make constraint All DOF inside the circle, pick the whole area with mouse.
- 18 Solution > define loads > Apply > structural > force/moment > on nodes > select node > middle click > select FX and FY or MZ depending on the problem > click ok. Here 4 key points are in Z – direction and 4 key point are in +Z direction with load 20.
- 19 Solution > define loads > Apply > structural > pressure (applied or uniformly distributed load) > areas > select areas middle click (Enter pressure value at the both ends) i.e.20 > click ok.

Solution > solve > ok.

General post processor:

1. General post processor > plot results > deformed shape > ok.
2. General post processor > plot results > counter plot > nodal solution > ok.
3. General post processor > plot results > counter plot > element solution > ok.
4. General post processor > plot results > vector plot > predefined > ok.
5. General post processor > linear results > nodal solution > ok.
6. General post processor > list results > element solution > ok.

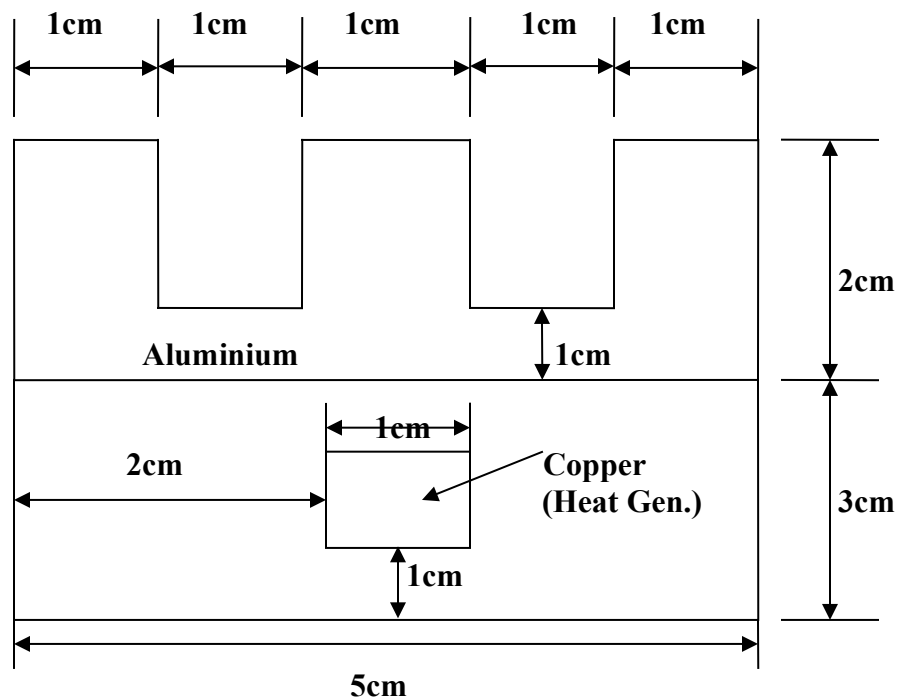
Result: Thus the analysis on the given solid structure is performed using ANSYS tool, Maximum, minimum stresses are found with respective to the given loads and values are noted.

Viva Questions:

1. How to apply loads in Structural Analysis?
2. What is load?
3. What is Cantilever Beam?
4. What is Simply Supported Beam?
5. How to create a Node?
6. How to create an Element?
7. What is Thermal Analysis?
8. What is the difference between Structural Analysis and Thermal Analysis?
9. What is DOF for structure?
10. What are types of loads?

Thermal Analysis

Aim: To determine the nodal temperature distribution, to determine the maximum value of temperature in the given fin structured component by using ANSYS tool, also list the results according to the given thermal conditions.



Procedure:

13. Click workplace > WP settings > check the Cartesian and grid only buttons > enter the following values > snapiner = 0.01, spacing = 0.01, minimum = 0, maximum = 0.2, tolerance = 0.003 > ok.
14. Go to the ANSYS Main menu > preprocessor > modeling > create > areas > rectangle > 2 corners.
15. Now we will pick the end points of the rectangle. First make the steel rectangle of dimensions 5cm X 3cm i.e. 5 units by 3 units on the grid > next make the copper square of dimensions 1cm X 1cm > next make the aluminium part by making a rectangle of dimensions

5cm X 2cm and then creating two smaller rectangles, which can then be subtracted from the main part to make the fins > from preprocessor, choose modeling > operate > Boolean > overlap > areas. Choose the steel area and then the aluminium area, and then click ok. The reason why we don't give the copper and the steel is that they overlap. Picture a copper plate resting on the steel area. The steel and aluminium are connected more intimately, and must be glued together > if you cannot see the complete work plane then go to utility menu > plot controls > pan, zoom, rotate and zoom out to see the entire work plane > the model should look like the one below.

Material Properties

Define material properties separately for steel aluminium and copper > click preprocessor > material props > material models. In the window that comes up choose Thermal > Conductivity > Isotropic > enter 1 for the material property number and click ok > fill in 20 for Thermal conductivity Click ok > Now the material has the properties defined in the above table. This represents the material properties for steel. Repeat the above steps to create material properties for aluminium (k=180, material number 2), and copper (k=386, material number 3). Do this by selecting material > new model in the "Define Material Model Behavior" window.

Element Properties:

Selecting Element Type:

Click processor > element Type > Add/Edit > Delete the 'Element Types' window that opens Click on Add> Type1 in the Element type reference number > Click on Thermal mass solid and select Quad 8 nodes 77. Click ok. Close the 'Element Types' window > so now we have selected Element Type1 to be a thermal solid 8 node element. The component will now be modeled with thermal solid 8 node element. This finishes the selection of element type.

Meshing

Go to preprocessor > Meshing > size controls > manual size > lines > All lines. In the menu that comes up type 0.01 in the Field for Element edge length > Click on ok. Now when you mesh the figure ANSYS will automatically create meshes that have an edge length of 0.01 m along the lines you selected > First we will mesh the steel area. Go to preprocessor > meshing > mesh attributes > Default Attributes > Now go to preprocessor > Meshing > mesh > Areas > Free pick the steel area and click ok > Repeat the same process for the aluminum and copper areas. Make sure you use the correct material number (2 and 3 respectively) for both areas. Also since the steel and the copper areas overlap make sure you pick the right area. If you choose the wrong area use preprocessor > meshing > Clear to undo the previous mesh and then repeat the previous steps.

Boundary conditions and Constraints:

Go to preprocessor > Loads > Define loads > Apply > Thermal > Heat Generate > on key points > select the corners of the copper square Click ok. > Enter $10e^6$ for the

HGEN value and click ok > Go to preprocessor > Loads > Define loads > Apply > Thermal > Convection > on lines. Pick all the lines on the outside of the object except the bottom one where the object is considered insulated. Click ok > Enter 50 for “Film coefficient” and 20 for “Bulk Temperature” click ok > Now the modeling of the problem is above .

Solution:

Go to ANSYS Main menu > solution > Analysis type > New Analysis > select steady state and click on ok > Go to solution > solve > currents LS > an error window may appear. Click ok on that window and ignore it > wait for ANSYS to solve the problem > click on Ok and close the ‘Information’ window.

Post Processor:

Go to ANSYS Main menu General post processing > list > Result > Nodal solution.

Select DOF solution and temperature. Click ok

Go to General post processing > plot results > counter plot > Nodal solution.

Select DOF solution and temperature to be plotted and click ok.

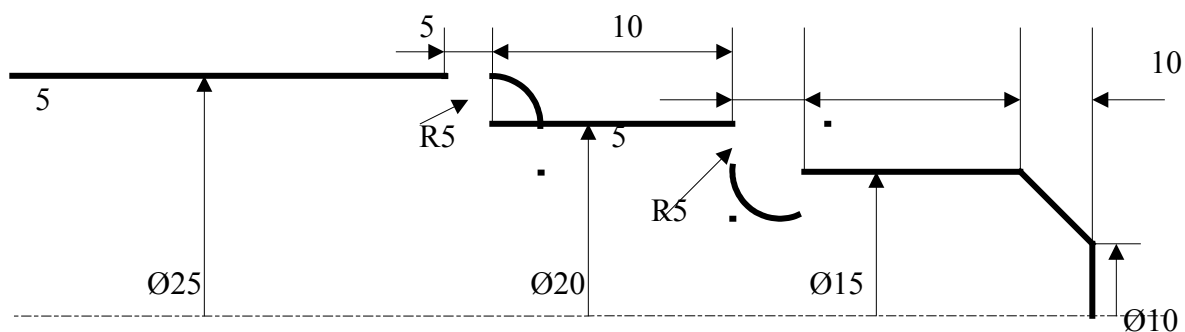
Result: Thus the Thermal analysis on the given structures performed using ANSYS tool and nodal distribution temperature is captured.

Viva Questions:

1. What is Temperature?
2. How to convert degree Celsius's to Fahrenheit Heat?
3. What are the units of Temperature?
4. What is Zero Temperature?
5. What are the types of Thermal Analysis?
6. What is task in Thermal Analysis?
7. How to apply loads in Thermal Analysis?
8. How to apply loads in Structural Analysis?
9. What is load?

CNC Exercise 1

Aim: To manufacture the given model.



Procedure:

```

G21
G98
G28 U0 W0
M06 T1
M03 S1500
G00 X26 Z1
G71 V0.5 R1
G71 P1 Q2 U0.1 W0.1 F50
N1 G01 X10 Z20
    G01 X15 Z2-5
    G01 X15 Z2-15
    G02 X15 Z2-20 R5
    G01 X20 Z2-30
N2 G03 X25 Z2-35 R5
G70 P1 Q2 S2000 F30
G28 U0 W0
M05
M30
  
```

Result:

Thus the given job is manufactured.

Viva Questions:

1. What is mean by M05?
2. What is mean by U00?
3. What is mean by W00?
4. What is mean by M30?
5. What is N series?
6. What is the difference between Manufacturing and Production?
7. What is Counter?
8. What is ASP?
9. What is F30 command?
10. How you fix the tool?

CONTENTS BEYOND SYLLABUS

3D Transformation

```
#include<stdio.h>
#include<conio.h>
#include<graphics.h>
#include<math.h>
int maxx,maxy,midx,midy;
void axis()
{
    getch();
    cleardevice();
    line(midx,0,midx,maxy);
    line(0,midy,maxx,midy);
}
void main()
{
    int gd,gm,x,y,z,o,x1,x2,y1,y2;
    detectgraph(&gd,&gm);
    initgraph(&gd,&gm,"e:\\tc\\bgi");

    setfillstyle(0,getmaxcolor());
    maxx=getmaxx();
    maxy=getmaxy();
    midx=maxx/2;
    midy=maxy/2;

    axis();

    bar3d(midx+50,midy-100,midx+60,midy-90,5,1);

    printf("Enter translation factor <BR>);
    scanf("%d%d",&x,&y);
    axis();
    printf("After translation:<BR>);
    bar3d(midx+x+50,midy-(y+100),midx+x+60,midy-(y+90),5,1);

    axis();
    bar3d(midx+50,midy-100,midx+60,midy-90,5,1);
    printf("Enter scaling factors<BR>);
```

```

scanf("%d%d%d", &x,&y,&z);

axis();
printf("After scaling<BR>");
bar3d(midx+(x*50),midy-(y*100),midx+(x*60),midy-(y*90),5*z,1);
axis();

bar3d(midx+50,midy-100,midx+60,midy-90,5,1);

printf("Enter rotating angle<BR> ");
scanf("%d",&o);
x1=50*cos(o*3.14/180)-100*sin(o*3.14/180);
y1=50*sin(o*3.14/180)+100*cos(o*3.14/180);
x2=60*cos(o*3.14/180)-90*sin(o*3.14/180);
y2=60*sin(o*3.14/180)+90*cos(o*3.14/180);

axis();
printf("After rotation about z axis<BR> ");
bar3d(midx+x1,midy-y1,midx+x2,midy-y2,5,1);
axis();

printf("After rotation about x axis <BR>");
bar3d(midx+50,midy-x1,midx+60,midy-x2,5,1);

axis();
printf("After rotation about yaxis<BR>");
bar3d(midx+x1,midy-100,midx+x2,midy-90,5,1);
getch();
closegraph();
}

```

BEZIERCURVE

```

#include<stdio.h>
#include<conio.h>
#include<graphics.h>

int x,y,z;

void main()
{
float u;
int gd,gm,ymax,i,n,c[4][3];

for(i=0;i<4;i++) { c[i][0]=0; c[i][1]=0; }

printf("\n\n Enter four points : \n\n");

for(i=0; i<4; i++)
{
printf("\t X%d Y%d : ",i,i);
scanf("%d %d",&c[i][0],&c[i][1]);
}

c[4][0]=c[0][0];
c[4][1]=c[0][1];

detectgraph(&gd,&gm);
initgraph(&gd,&gm,"e:\\tc\\bgi");

ymax = 480;

setcolor(13);
for(i=0;i<3;i++)
{
line(c[i][0],ymax-c[i][1],c[i+1][0],ymax-c[i+1][1]);
}

```

```

setcolor(3);
n=3;

for(i=0;i<=40;i++)
{
u=(float)i/40.0;
bezier(u,n,c);
if(i==0)
{ moveto(x,ymax-y);}
else
{ lineto(x,ymax-y); }
getch();
}
getch();
}
bezier(u,n,p)
float u;int n; int p[4][3];
{
int j;
float v,b;
float blend(int,int,float);
x=0;y=0;z=0;
for(j=0;j<=n;j++)
{
b=blend(j,n,u);
x=x+(p[j][0]*b);
y=y+(p[j][1]*b);
z=z+(p[j][2]*b);
}
}
float blend(int j,int n,float u)
{
int k;
float v,blend;
v=C(n,j);
for(k=0;k<j;k++)
{ v*=u; }
for(k=1;k<=(n-j);k++)
{ v *= (1-u); }
blend=v;
return(blend);
}
C(int n,int j)
{
int k,a,c;
a=1;

```

```
for(k=j+1;k<=n;k++) { a*=k; }  
for(k=1;k<=(n-j);k++) { a=a/k; }  
c=a;  
return(c);}
```