DEPARTMENT MECHANICAL ENGINEERING

CAD & CAM
LAB MANUAL

SUBJECT NAME	CAD & CAM Lab
SUBJECT CODE	1960382
COURSE-BRANCH	B. Tech - Mechanical Engineering
YEAR-SEMESTER	IV - I
ACADEMIC YEAR	2021-2022
REGULATION	MLRS-R19

MARRI LAXAMAN REDDY

INSTITUTE OF TECHNOLOGY AND MANAGEMENT

MISSION AND VISION OF THE INSTITUTE:

Our Vision:

To establish as an ideal academic institution in the service of the nation the world and the humanity by graduating talented engineers to be ethically strong globally competent by conducting high quality research, developing breakthrough technologies and disseminating and preserving technical knowledge.

Our Mission:

To fulfill the promised vision through the following strategic characteristics and aspirations:

- Contemporary and rigorous educational experiences that develop the engineers and managers;

- An atmosphere that facilitates personal commitment to the educational success of students in an environment that values diversity and community;

- Prudent and accountable resource management;

- Undergraduate programs that integrate global awareness, communication skills and team building across the curriculum;

- Leadership and service to meet society's needs;

- Education and research partnerships with colleges, universities, and industries to graduate education and training that prepares students for interdisciplinary engineering research and advanced problem solving;

- Highly successful alumni who contribute to the profession in the global society.

Vision and Mission statements of the Department of Mechanical Engineering:

Vision Statement:

"The Mechanical Engineering Department strives immense success in the field of education, research and development by nurturing the budding minds of young engineers inventing sets of new designs and new products which may be envisaged as the modalities to bring about a green future for humanity"

Mission Statement:

1. Equipping the students with manifold technical knowledge to make them efficient and independent thinkers and designers in national and international arena.

2. Encouraging students and faculties to be creative and to develop analytical abilities and efficiency in applying theories into practice, to develop and disseminate new knowledge.

3. Pursuing collaborative work in research and development organizations, industrial enterprises, Research and academic institutions of national and international, to introduce new knowledge and methods in engineering teaching and research in order to orient young minds towards industrial development.

PROGRAM EDUCATIONAL OBJECTIVE

PEO 1: Graduates shall have knowledge and skills to succeed as Mechanical engineers for their career development.

PEO 2: Graduates will explore in research.

PEO 3: Mechanical Graduates shall have the ability to design products with various interdisciplinary skills

PEO 4: Graduates will serve the society with their professional skills

PROGRAM OUTCOMES

A. Engineering Knowledge: Apply the knowledge of mathematics, science, engineering fundamentals and an engineering specialization for the solution of complex engineering problems.

B. Problem Analysis: Identify, formulate, research, review the available literature and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural science and engineering sciences.

C. Design and development of solutions: Design solutions for complex engineering problems and design system components or processes that meet the specific needs with appropriate considerations for public health safety and cultural, societal and environmental considerations.

D. Conduct investigations of complex problems: Use research based knowledge and research methods including design of experiments, analysis and interpretation of data and synthesis of the information to provide valid conclusions.

E. Modern tool usage: Create, select and apply appropriate techniques, resources and modern engineering and IT tools including predictions and modeling to complex engineering activities with an understanding of the limitations.

F. The Engineer and society: Apply reasoning, informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practices.

G. Environment and sustainability: Understand the impact of the professional engineering solutions in society and environmental context and demonstrate the knowledge of and need for sustainable development.

H. Ethics: Apply ethical principles and commitment to professional ethics, responsibilities and norms of the engineering practice.

I. Individual and team work: Function effectively as an individual and as a member or leader in diverse teams and in multi-disciplinary settings.

J. Communication: Communicate effectively on complex engineering activities with the engineering community and with the society at large, such as being able to comprehend, write effective reports, design documentation, make effective presentations, give and receive clear instructions.

K. Project management and finance: Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects and in multidisciplinary environments.

L. Life – long learning: Recognize the need and have the preparation, ability to engage in independent and life – long learning in the broadest context of technological change.

PROGRAMME SPECIFIC OUTCOMES:

PS01: Students acquire necessary technical skills in mechanical engineering that make them employable graduate.

PSO2: An ability to impart technological inputs towards development of society by becoming an entrepreneur.

COURSE OBJECTIVES:

1. To be able to understand and handle design problems in a systematic manner.
2. To be able to apply CAD in real life applications.
3. To be understand the basic principles of different types of analysis.

COURSE OUTCOMES:

ME 383.1   Able to solve simple problems using FEA software.

ME 383.2   Generate freeform shapes in part mode to visualize components.

ME 383.3   Create complex engineering assemblies using appropriate assembly constraints.

ME 383.4   Develop G and M codes for turning components.

ME 383.5   Develop G and M codes for milling components.

ME 383.6   Generate automated tool paths for given engineering component.

INSTRUCTIONS TO THE STUDENTS

1. Every student should obtain a copy of the laboratory manual
2. It is important that all students arrive at each session on time.
3. Dress code: Students must come to the laboratory wearing:
   - Trousers.
   - half-sleeve tops.
   - Leather shoes.
   - Half pants, loosely hanging garments and slippers are not allowed.
4. Students should come with thorough preparation for the experiment to be conducted.
5. Students will not be permitted to attend the laboratory unless they bring the practical record fully completed in all respects pertaining to the experiment conducted in the previous class.
6. Experiment should be started only after the staff-in-charge has checked the experimental setup.
7. All the calculations should be made in the observation book. Specimen calculations for one set of readings have to be shown in the practical record.
8. Wherever graphs are to be drawn, A-4 size graphs only should be used and the same should be firmly attached to the practical record.
9. Practical record and observation should be neatly maintained.
10. They should obtain the signature of the staff-in-charge in the observation book after completing each experiment.
11. Theory regarding each experiment should be written in the practical record before procedure in your own words.

LABORATORY SAFETY PRECAUTIONS

1. Laboratory uniform, shoes & safety glasses are compulsory in the lab.
2. Do not touch anything with which you are not completely familiar. Carelessness may not only break the valuable equipment in the lab but may also cause serious injury to you and others in the lab.
3. Please follow instructions precisely as instructed by your supervisor. Do not start the experiment unless your setup is verified & approved by your supervisor.
4. Do not leave the experiments unattended while in progress.
5. Do not crowd around the equipment's & run inside the laboratory.
6. During experiments material may fail and disperse, please wear safety glasses and maintain a safe distance from the experiment.
7. If any part of the equipment fails while being used, report it immediately to your supervisor. Never try to fix the problem yourself because you could further damage the equipment and harm yourself and others in the lab.
8. Keep the work area clear of all materials except those needed for your work and cleanup after your work.

LIST OF EXPERIMENTS

1. Sketching: Development of part drawings for various components in the form of orthographic Representation of dimensioning and tolerances.

2. Part Modelling: Generation of various 3D Models through Protrusion, revolve, sweep. Design of simple components.

3. Determination of deflection and stresses in 2D and 3D trusses and beams.

4. Determination of deflections, principal and Von-mises stresses in plane stress.

5. Determination of stresses in 3D and shell structures (at least one example in each case)

6. Estimation of natural frequencies and mode shapes, Harmonic response of 2D beam.

7. Study state heat transfer analysis of plane and axi-symmetric components.

8. Development of process sheets for various components based on Tooling and Machines.

9. Development of manufacturing defects and tool management systems.

10. Study of various post processors used in NC Machines.

11. Development of NC code for free form and sculptured surfaces using CAM software.

12. Machining of simple components on NC lathe and Mill by transferring NC Code / from CAM software.

What is CAD/CAM?
CAD/CAM (computer-aided design and computer-aided manufacturing) refers to computer software that is used to both design and manufacture products.

CAD is the use of computer technology for design and design documentation. CAD/CAM applications are used to both design a product and program manufacturing processes, specifically, CNC machining. CAM software uses the models and assemblies created in CAD software to generate tool paths that drive the machines that turn the designs into physical parts. CAD/CAM software is most often used for machining of prototypes and finished parts.

Why CAD/CAM?
 Computer Aided Design and Computer Aided Manufacture is the way things are made these days. Without this technology we wouldn't have the range and quality of products available or, at least, they wouldn't be available at a price most of us can afford. Hand-building and manual techniques still very much have their place and Design Education needs to treasure and foster these skills so that future generations will have the „hands-on" skills to understand the man-made world and a solid modeling system is usually an interactive computer graphics system that is intended to create true three-dimensional components and assemblies. Recent advances in CAD software, computers, and graphical displays have made it possible to use solid representations of components being considered in the design process. These solid models can be employed in numerous ways.

<u>**Advantages of Solid Modeling**</u>
A realistic visual display: By producing a shaded visible surface image of the solid, solid modeling allows a designer to see exactly what has been created. Easy to deal with different views: Once a part has been created, we have the ability to rotate, shade, section, or produce almost any view required by a designer.
Single associated model database: The solid modeler provides the only database suitable for all CAD operations. Almost all information needed for part generation is contained in the solid model. The algorithm should be able to ensure that it represents physically possible shape that is complete and unambiguous Applications. e.g., automatic generation of a mesh for a finite element analysis. Provide the next generation of engineers, designers and technicians. All of these professionals will be using CAD/CAM techniques or CAD/CAM products in their work, alongside practical hands-on skill. Design and Technology education has to reflect modern practice so it is crucial that students have the opportunity to use real CAD/CAM tools in their designing and making.

<u>**DESIGN PROCESS AND ROLE OF CAD**</u>

1. Recognition of need
2. Definition of problem
3. Synthesis
4. Analysis and optimization
5. Evaluation
6. Presentation

SOLID MODELING

REQUIREMENTS FOR MODELING ASSEMBLING

1. Part modeling and analysis

The part analysis includes the material type, mass and inertial properties, functional properties of the faces, etc.

2. Hierarchical relationships
An assemble tree and assemble sequence must be given.

3. Mating conditions.
There are two methods for specifying mating conditions: Specify the location and orientation of each part in the assembly, together with the representation of the part itself, by providing a 4 x 4 homogeneous transformation matrix. (i.e., transformation from MCS to WCS).
Specify the spatial relationships between its individual parts as mating conditions.

CAD/CAE/CAM Data Exchange

Computer databases are now replacing paper blueprints in defining product geometry and non-geometry for all phases of product design, analysis, and manufacturing. It becomes increasingly important to find effective procedures for transferring data among CAD/CAE/CAM systems.

The need to exchange modeling data is directly motivated by the need to integrate and automate the design and manufacturing process to obtain the maximum benefits from CAD/CAE/CAM systems. Four Types of Modeling Data to be transferred:

(1)Shape
(2)Non shape
(3)Design
(4) Manufacturing

(1) Shape data consists of both geometrical and topological information as well as part features. Entity attributes such as font, color, and layer as well as annotation is considered part of the entity geometrical information. Topological information applies only to products described via solid modeling. Features allow high-level concept communication about parts.
Examples are hole, flange, web, pocket, chamfer, etc.

(2) Non shape data includes graphics data such as shaded images, and model global data as measuring units of the database and the resolution of storing the database numerical values.

(3) Design data has to do with the information that designers generate from geometric models for analysis purposes. e.g., mass property and finite element mesh data.

(4) Manufacturing data consists of i nformati on such as tooling, NC tool paths, tolerancing, process planning, tool design, and bill of materials.

Commonly Used CAD Data Exchange Format:
IGES (Initial Graphics Exchange Specification)
PDES (Product Data Exchange Using STEP)

IGES is focused on CAD-to-CAD exchange where primarily shape and non shape data were to be transferred from one system to another.

PDES is previous called Product Data Exchange Standard. It is for the exchange of complete product descriptions which covers the four types of modeling data (i.e., shape, non shape, design and manufacturing).

Other data exchange interfaces include: STL, Neutral, SET, ECAD, VDA, STEP, PDGS, CATIA, Render, CGM, VRML, PATRAN, TIFF, etc.

EXERCISES

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 30^0 line by use the vertical line

9. Then mirror the 30^0 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 30^0

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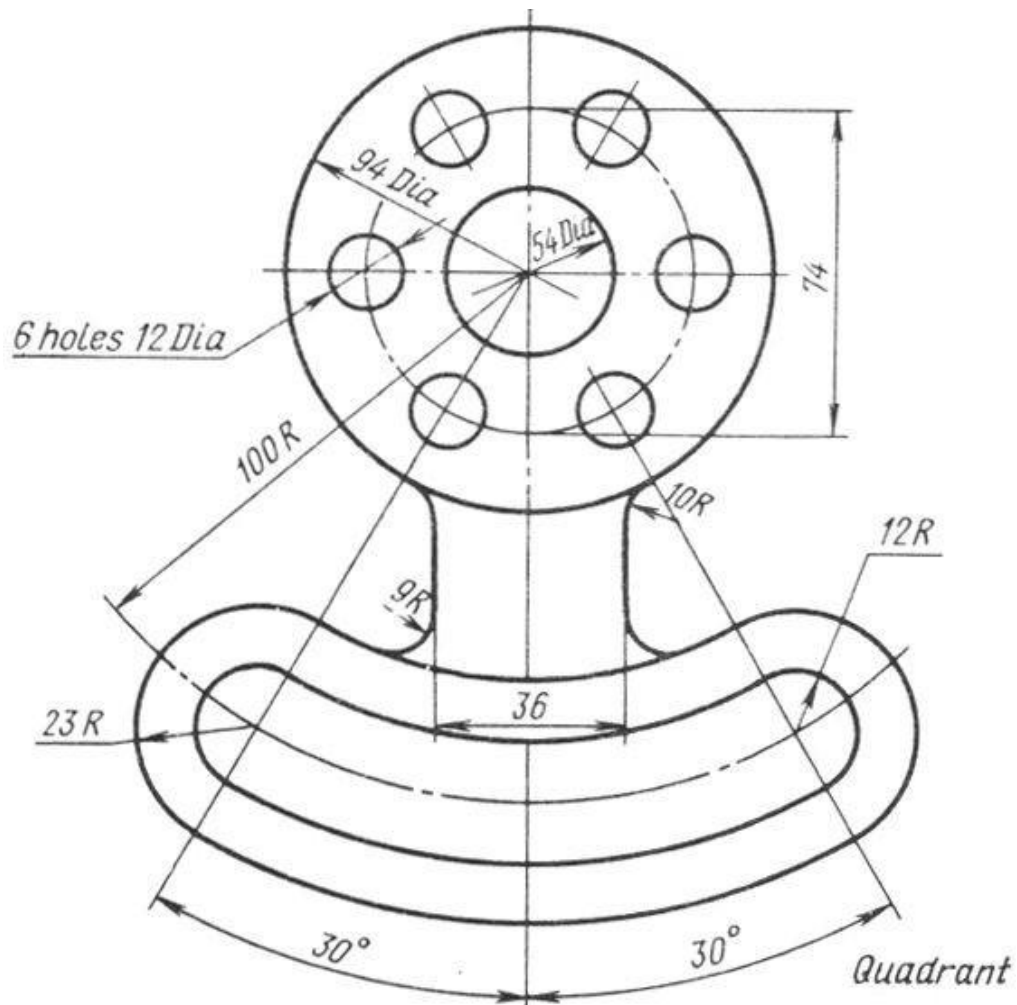
Result: Hence the required 2D diagram is created using Auto CAD.

<u>Viva Questions</u>

1. What is CAD?
2. What is the difference between Pan and Zoom?
3. What is the difference between line and spline?
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 30^0 angle to the +ve Y axis as shown in the figure and a ray of angle 60^0 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.
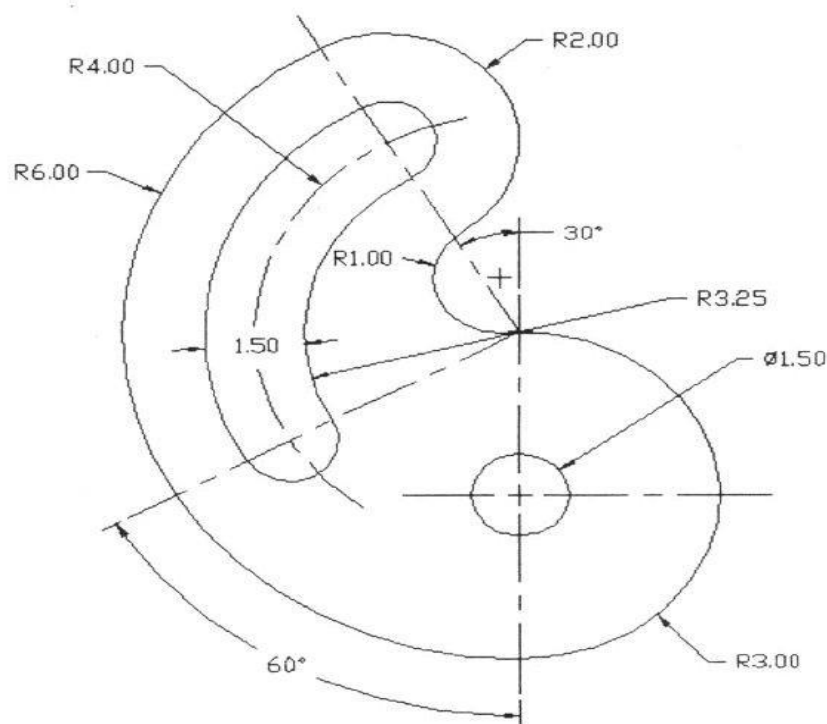
Result: Hence the required 2D diagram is created using Auto CAD.

<u>Viva Questions</u>

1. What is the difference between line and Spline?
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

12R

8

85 Dia

62 Dia

32

104

14

14 R

94

4 holes 12 Dia

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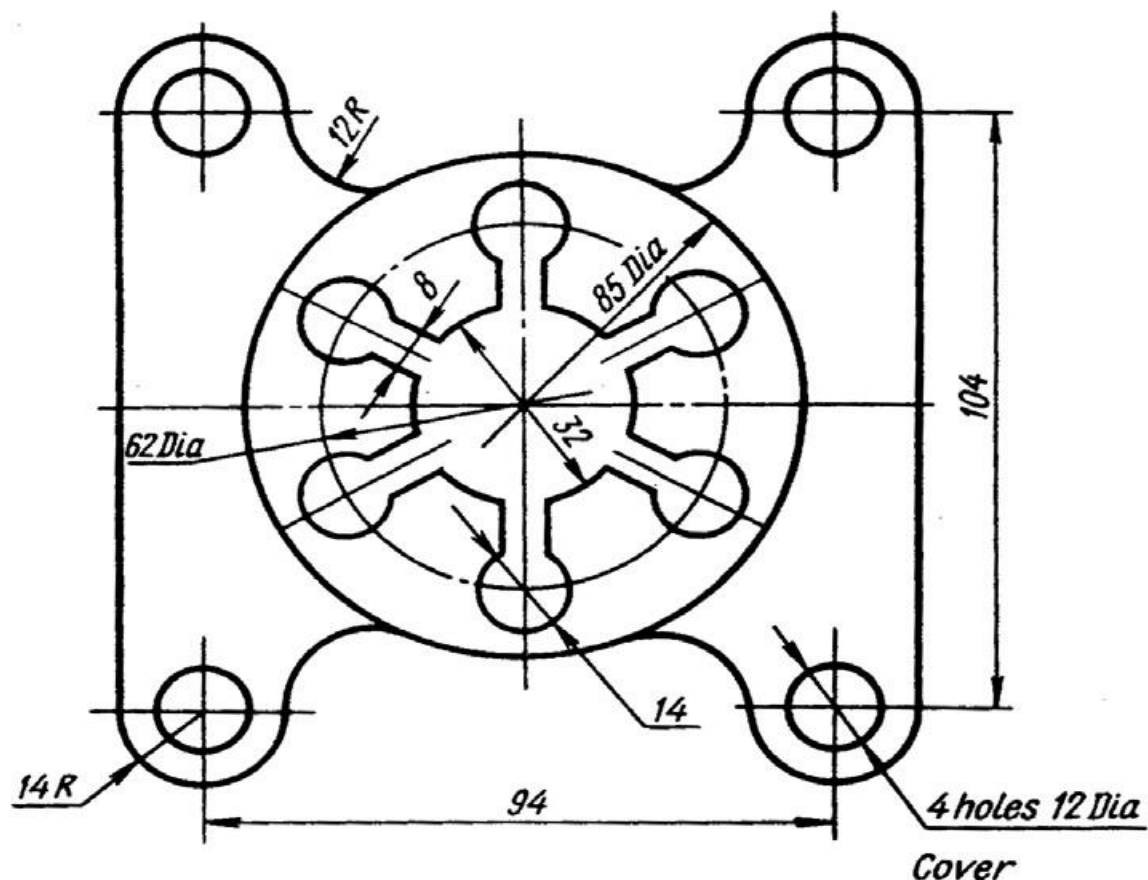
Result: Hence the required 2D diagram is created using Auto CAD.

<u>Viva Questions</u>

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.

4. Draw the 2 axes lines

5. Draw 2 concentric circles of diameter 58 and 40 above the axes

6. Draw the 10dia circle on the 40dia of the circle

7. And use the array command from modify tool bar to represent the 8 holes
   with 10dia from centre of the circles.

8. Draw the 2 concentric circles of diameter 58 & 36 below the axes

9. Draw the circles of the diameter 8 on the circle of 36dia

10. Use the array command from the modify tool bar to represent the 6 holes
    with 8dia

11. Draw the 2 concentric circles of diameter 22 and 20 radius at the right side
    of the vertical line from the vertical axes.

12. Fillet the circles of radius 20 with 15 radiuses.

13. And again draw the circles of diameter 16& 20R at left side of the vertical
    line from the vertical axes.

14. And chamfer the circles of radius 20 with 58dia circle

15. Then trim the unwanted lines to get the required 2D drawing.

58

8 holes 10 Dia

40 Dia

15 R

20 R

16

34

58 Dia

22

20 R

34

36 Dia

6 holes 8 Dia

40

40

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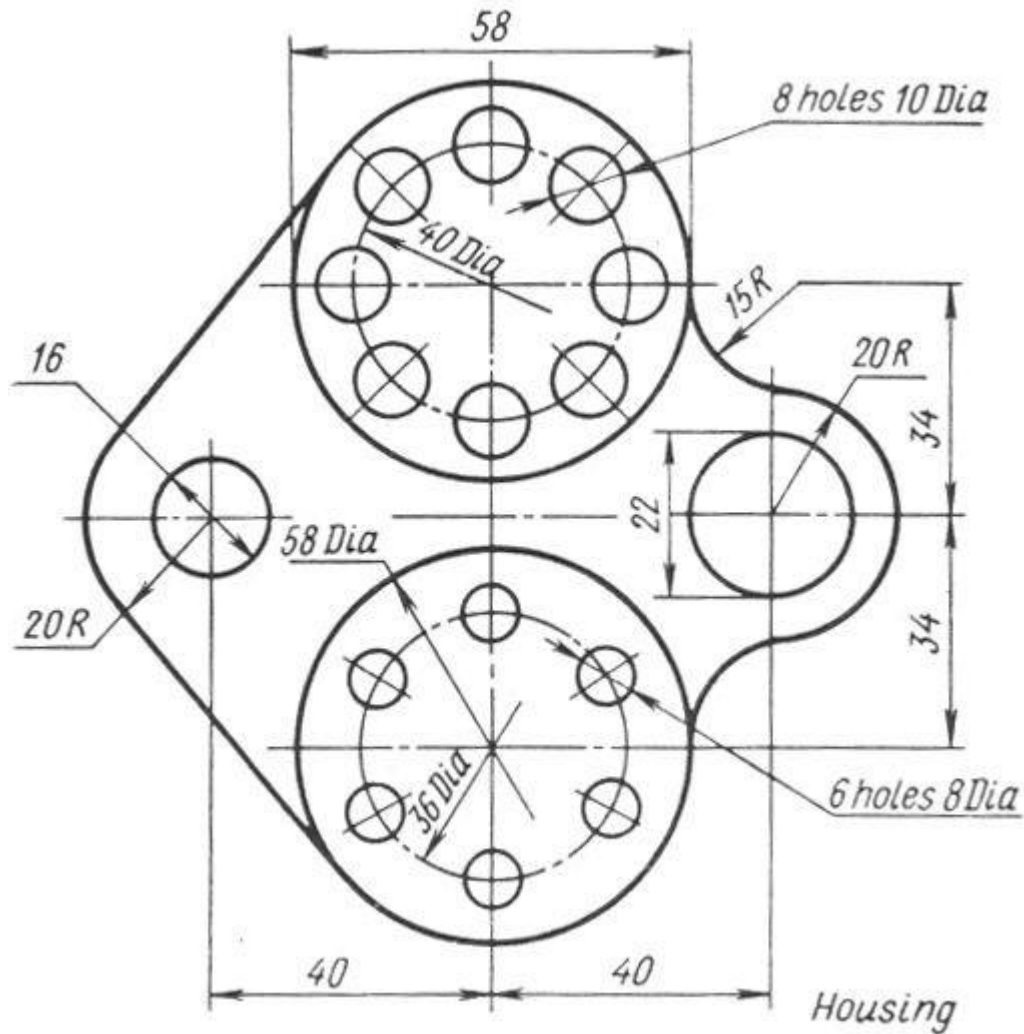
Result: Hence the required 2D diagram is created using Auto CAD.

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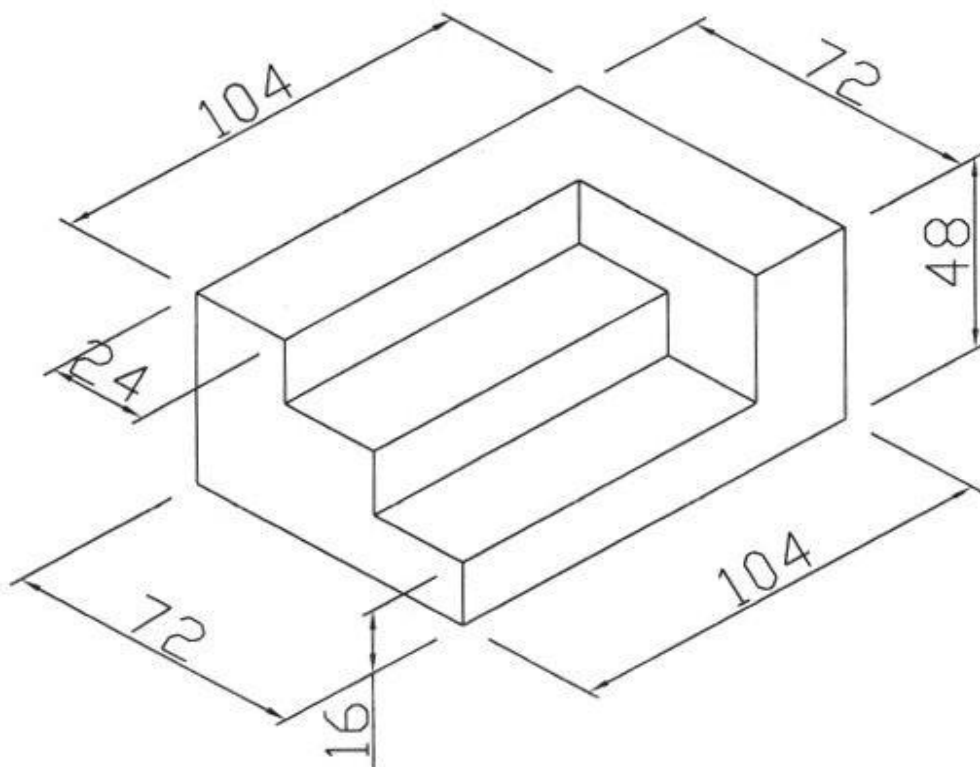
Result: Hence the required 2D isometric diagram is created using Auto CAD.

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.

1. CATIAV5

EXERCISE – 1

Drafting using CATIA V5

Objective:

- To develop the part drawings of given component using CATIA V5.

Out comes:

- Understand how to use sketcher commands to complete the given sketch.
- Understand the importance of using limits and view settings.

Scope of the experiment:

To provide an introduction to the use of sketcher commands in drawing 2D sketch and experience them how to apply and where to apply the various commands. Know the difference between draw and modify commands.

Description of software:

The software used for doing this sketch is catia v5 and it is developed by Dassault Systèmes. CATIA software(/kəˈtiːə/, an acronym of computer-aided three-dimensional interactive application) is a multi-platform software suite for computer-aided design (CAD), computer-aided manufacturing (CAM), computer-aided engineering (CAE), PLM and 3D, developed by the French company Dassault Systèmes. It is a three dimensional modeling tool. This tool is used in the design of various objects such as vehicles, buildings, components, etc.

Figure 1-3 *Initial screen that appears after starting CATIA V5R13*

Procedure

In this exercise, you will draw the sketch of the model shown in Figure 1-31. The sketch is shown in Figure 1-32. You will not dimension the sketch. The solid model and the dimensionsare given only for your reference.
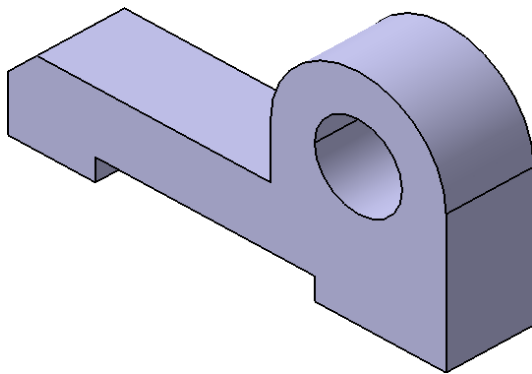
Figure 1-31 *Solid Model for Tutorial 1*   **Figure 1-32** *Sketch of the model*
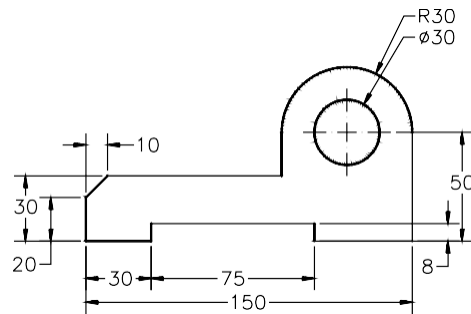
The following steps are required to complete this tutorial:

a.   Start CATIA V5 and then start a new CATpart file.
b.   Draw the sketch of the model using the **Line**, **Arc**, and **Circle** tools, refer to Figures 1-35 and 1-36.
c.   Save and close the file.

Starting CATIA V5 and Opening a New Part File

1.   Start CATIA V5 by choosing **Start > Programs (All Programs** if you are working with Windows XP **> CATIA > CATIA V5R13** or by double-clicking on the shortcut icon of CATIA V5R13 available on the desktop of your computer.

   A new **Product1** file is started.

2.   On choosing **Close** from **File** menu, the start screen of CATIA V5 is displayed. Choose **Start > Mechanical Design > Part Design** to make sure that you are in **Part Design** workbench. To open a new file in **Part Design** workbench, choose **File > New** from menu bar. The **New** dialog box is displayed, as shown in Figure 1-33.

*Figure 1-33 Selecting **Part** from the **New** dialog box*

3. Select **Part** from the **List of Types** list box from this dialog box and choose the **OK** button.

A new file in the **Part Design** workbench is opened.

4. Choose the **Sketcher** button from the **Sketcher** toolbar and then select the YZ plane as the sketching plane, to invoke the **Sketcher** workbench. The screen that is displayed by invoking the **Sketcher** workbench, is shown in Figure 1-34.
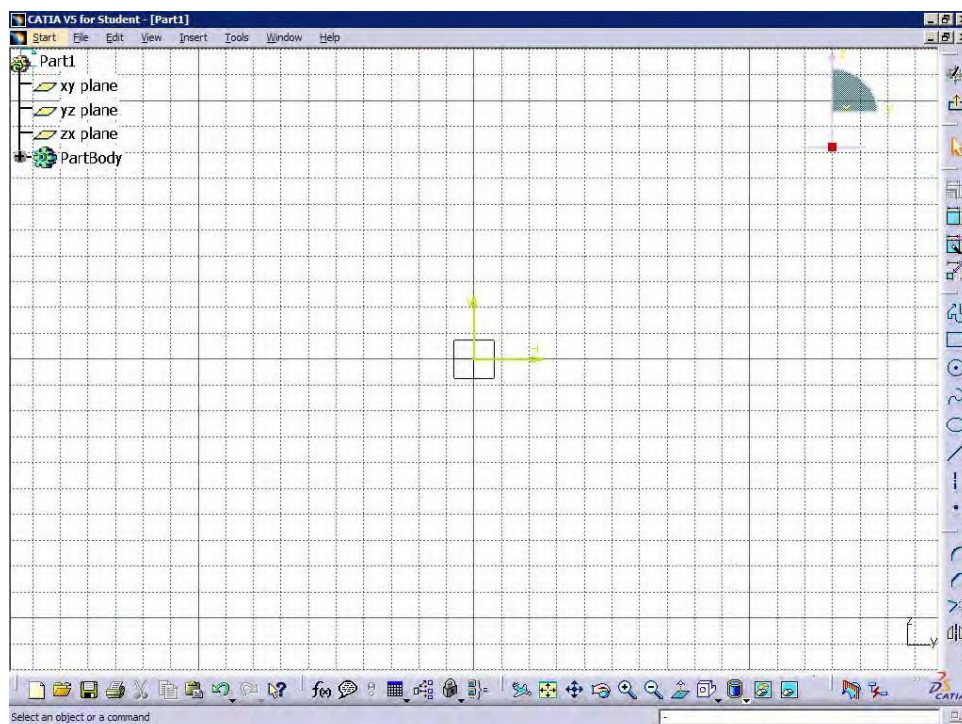
*Figure 1-34 **Sketcher** workbench screen*

You will draw the sketch in two sections, first as the outer loop and second as the inside circle.

Drawing the Outer Loop of the Sketch

It is recommended to create the sketch symmetrically around the origin. This will reduce the time required for constraining and dimensioning the sketches. The outer loop of sketch can be drawn using the **Line** and the **Arc** tools. You will start drawing the outer loop from the lower left corner of the sketch.

1. Invoke the **Line** tool by choosing the **Line** button from the **Profile** toolbar.

2. Choose the **Snap to Point** button from the Sketch tools toolbar, if not already chosen.

3. Move the cursor in the third quadrant. The coordinates of the point will be displayed along with the cursor.

4. Click at the point whose coordinates are -50mm, -30mm, and then move the cursor horizontally toward the right.

   You will notice that the color of line turns blue, when you move the cursor horizontally.

   Note
   Whenever a line turns blue while drawing, it means that the line is constrained. The constraint may be horizontal or vertical, depending on the direction in which the line is drawn.

   All the constraints that are applied to the drawn sketch will not be explained in this tutorial. You will learn about such constraints individually in later chapters.

   Refer to Figure 1-32. The length of the first horizontal line at the lower left corner of the sketch is 30mm. Therefore, move the cursor until the length of the line is shown as 30mm in the **L** edit box of the **Sketch tools** toolbar.

5. Press the left mouse button, when the length of the line in the **L** edit box of the **Sketch tools** toolbar displays a value of 30mm.

   The first horizontal line is drawn. You will notice a **Horizontal** constraint is applied on it. After the line is drawn, it is still active and is displayed in orange color. Left click in the geometry area to make sure it is no more selected.

   As soon as you specify the endpoint of line, the **Line** tool is terminated. Therefore, you need to choose this button again and again to draw multiple lines. You can avoid this by double-clicking on the **Line** button in the **Profile** toolbar. Now, the line tool will not be terminated until you terminate it by pressing ESC key twice on the keyboard.

6. Double-click on the **Line** button to invoke the **Line** tool and select the endpoint of the first horizontal line.

7. Press the TAB key thrice on the keyboard to highlight the value displayed in the **L** edit

box of the **Sketch tools** toolbar. Type **8** in this edit box and press the ENTER key.

8. Now move the cursor vertically upward and click when a vertical line is displayed.

    A vertical line of length 8mm will be drawn. You will notice that this line is no longer in the select mode and you are prompted to select the start point of the next line. This is because of double-clicking on the **Line** button. It makes the **Line** tool active till you invoke any other tool.

9. Select the endpoint of the vertical line as the start point of the second horizontal line. Enter **75** in the **L** edit box of the **Sketch tools** toolbar. Move the cursor horizontally toward the right and click when a horizontal line is displayed.

    This draws the second horizontal line of length 75mm.

10. Select the endpoint of the second horizontal line, as the start point of the second vertical line and move the cursor vertically downward. Click when the length of the line in the **L** edit box shows a value of 8mm.
    This draws the second vertical line of length 8mm.

    Note
    You will notice that while drawing the second vertical line, the inferencing line is displayed in the geometry area. The inferencing lines are often displayed whenever the endpoint of a line is constrained, with an element already available in the sketch.

11. Select the endpoint of the second vertical line as the start point of the third horizontal line and move the cursor horizontally toward the right. Click to draw the third horizontal line, when the length of the line in the **L** edit box shows a value of 45mm.

12. Select the endpoint of the previous line as the start point of the third vertical line and move the cursor vertically upwards. Click when the length of the line is 50mm.

    This draws the third vertical line of length 50mm. Next, you can draw the arc.

13. To draw the arc, first invoke the **Circle** toolbar, by choosing the down arrow available on the right of the **Circle** button, from the **Profile** toolbar. Choose the **Three Point Arc** button from to invoke the **Three Point Arc** tool.

14. Select the start point of the arc as the endpoint of the previous vertical line and click on it.

15. Move the cursor to a point whose coordinates are 70mm, 50mm. These are displayed in the **Sketch tools** toolbar and also on top of the cursor. Click on this point to define the second point.

16. Move the cursor to specify the third point of the arc. Click on the point when the cursor snaps a location 40mm, 20mm in the geometry area. The coordinate values are displayed on top of the cursor.

This draws the arc for the outer loop. As the arc is in selection mode, click anywhere in the geometry area to end the selection mode. Now, to continue drawing the outer loop, you need to invoke the **Line** tool again.

17. Double-click on the **Line** button from the **Profile** toolbar to invoke the **Line** tool.

18. Select the endpoint of the arc as the start point of the fourth vertical line. Move the cursor vertically downward to draw it. Click when the length value of the line is 20mm in the **L** edit box of the **Sketch tools** toolbar.

    This draws the fourth vertical line of length 20mm. The line is no longer in selection mode and you are prompted to enter the start point of the next line.

19. Select the endpoint of the previous line as the start point of the fourth horizontal line. Move the cursor horizontally toward left. Click when the length of the line in the **L** edit box of the **Sketch tools** toolbar shows a value of 80mm.

    This draws the fourth horizontal line of length 80mm. Note that the line is green in color, as it passes through the origin.

20. Select the endpoint of the previous line as the start point of the inclined line. Move the cursor such that the line is drawn at an angle of 225-degree. The current angle will be displayed in the **A** edit box of the **Sketch tools** toolbar. Click when a vertical inferencing line is displayed between the endpoint of the inclined line and the start point of the first horizontal line. This draws the inclined line of horizontal length values 10mm.

21. Select the endpoint of the inclined line as the start point of the next line. Move the cursor vertically downwards. Click when the length of the line in the **L** edit box shows a value of 20mm.

    This completes the sketch of the outer loop. It is recommended to modify the geometry area, such that the sketch fits in the screen. This is done using the **Fit All In** tool.

22. Choose the **Fit All In** button from the **View** toolbar to fit the current sketch on the screen.

    The outer loop of the sketch is completed and is shown in Figure 1-35. The display of the constraints is turned off using the **Hide/Show** tool.

Drawing Inner Circle

The circle will be drawn using the **Circle** tool.

1. Choose the **Circle** button from the **Circle** toolbar to invoke the **Circle** tool. You are prompted to define the center point of the circle.

2. Move the cursor to a point whose coordinates are 70mm, 20mm. Click when the cursor snaps to this point.
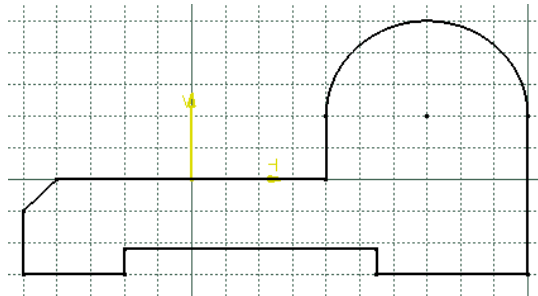
Figure 1-35 *Outer loop of the sketch*

3. Move the cursor horizontally toward the right and click when the radius of the circle in the **R** edit box of the **Sketch tools** toolbar shows a value of 15mm. Click anywhere to remove the circle from selection.

This completes the sketch for Tutorial 1. The final completed sketch for Tutorial 1 with the display of constraints turned on is shown in Figure 1-36.
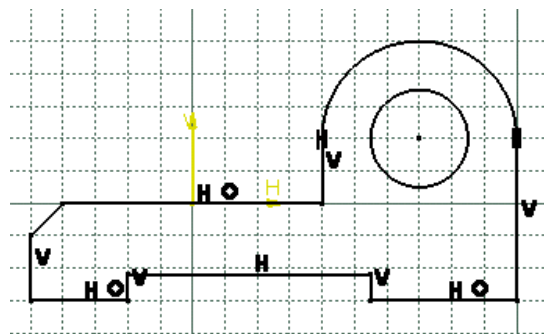

Figure 1-36 *Final sketch for Tutorial 1*

Conclusion:

After completing the sketch, you need to save it. You need to save each tutorial of this chapter in the *c01* folder, that is in the *CATIA* folder.

1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box. Create *CATIA* folder inside the *\My Documents* folder. Then create *c01* folder inside the *CATIA* folder.

2. Enter the name of the file as *c01tut1* in the **File name** edit box and choose the **Save** button. The file will be saved in the *\My Documents\CATIA\c01* folder.

3. Close the part file by choosing **File** > **Close** from the menu bar.

Additional Viva Questions:

1. Expand CATIAV5.

2. Write the function of workbench in catia v5.

3. What are sketch tools in sketcher workbench.

4. List the sketch tools used in catiav5.

5. Can you tell which tool is used to exit from sketcher workbench.

6. How to get into sketcher module from default catia window.

7. Write the use of geometrical constraints.

8. What are the steps required to go to part module from sketcher.

9. How do you measure arc length.

10. Mention the colors provided in catia.

Innovative Questions:

1. When to use geometric constraints in catia.

2. Can we generate 3D model with out sketcher.

3. On what machines catiav5 should be run.

4. Where can we use catia for?

5. How to edit existing dimension in catia?

6. Tell about catia applications used in design industry.

7. How do you make use of catiav5.

8. Categorize the catia versions developed by dassault system.

9. What type of programming language uses catia.

10. Where catia v5 modules can be applicable in design.

S.NO.	QUESTION	LEVEL
1.	Using the sketcher workbench design the given sketch using catia.	L5
2.	Choose the correct module for the given sketch to design in catia.	L6
3.	List the steps required and draw the given component using catia.	L4
4.	From the given sketch make use of correct module and complete it.	L3

EXERCISE – 2

Part Modelling using CATIA V5

Objective:

- To develop the part modeling of given component using CATIA V5.

Out comes:

- Understand how to use modeling commands to complete the given sketch.
- Understand the importance of using limits and view settings.

Scope of the experiment:

To provide an introduction to the use of sketcher commands in drawing 3D sketch and experience them how to apply and where to apply the various commands. Know the difference between draw and modify commands.

Description of software:

The software used for doing this sketch is catia v5 and it is developed by Dassault Systèmes. CATIA software(/kəˈtiːə/, an acronym of computer-aided three-dimensional interactive application) is a multi-platform software suite for computer-aided design (CAD), computer-aided manufacturing (CAM), computer-aided engineering (CAE), PLM and 3D, developed by the French company Dassault Systèmes. It is a three dimensional modeling tool. This tool is used in the design of various objects such as vehicles, buildings, components, etc.

Figure 1-3 *Initial screen that appears after starting CATIA V5R13*

Note

Even if you draw any sketch in the space containing the hidden elements, it will not be visible and will be displayed only after you return back to the visible geometry area.

You can change the standard element to a construction element in this space or vice-a-versa.

Procedure

In this exercise, you will draw the sketch of the model shown in Figure 1-37. The sketch is shown in Figure 1-38. You will not dimension the sketch. The solid model and the dimensionsare given only for your reference.
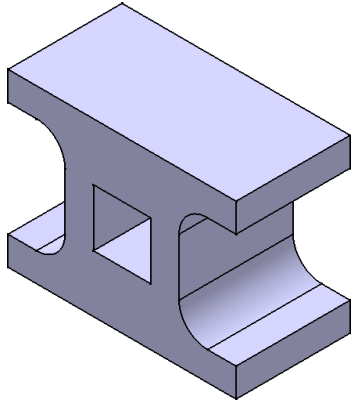
Figure 1-37 *Solid Model for Tutorial 2*
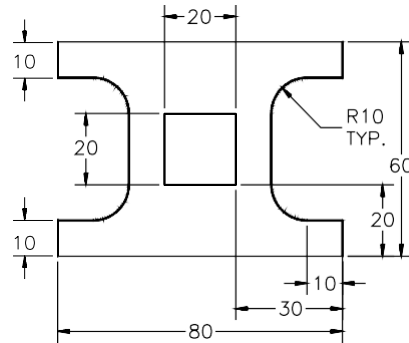
Figure 1-38 *Sketch of the model*

The following steps are required to complete this tutorial:

a. Start a new CATpart file.
b. Draw the sketch of the model using the **Profile** and **Rectangle** tool, refer to Figures 1-39 through 1-41.
c. Save and close the file.

Starting New Part File

1. Choose **File > New** from the menu bar. The **New** dialog box is displayed.

2. Select **Part** from the **List of Types** list box from this dialog box. Choose the **OK** button. A new file in **Part Design** workbench will open.

3. Choose the **Sketcher** button from the **Sketcher** toolbar and then select the YZ plane as the sketching plane, to invoke the **Sketcher** workbench.

    You will draw the sketch in two sections, first the outer loop and next the inner cavity.

Drawing the Outer Loop of the Sketch

You will draw the outer loop of the sketch using the **Line** and the **Arc** tool. Start drawing the outer loop from the left corner of the sketch. It is recommended to keep the origin in middle of the drawn sketch, as this will reduce the time required for constraining and dimensioning the sketches. This will also helps you to capture the design intent very easily.

1. Invoke the **Profile** tool from the **Profile** toolbar.

2. Move the cursor in the third quadrant. The coordinates of the point will be displayed above the cursor.

3. Specify the start point of the line at the point whose coordinates are -40, -30 and then move the cursor horizontally toward the right.

   You will notice that the color of line turns blue, when you move the cursor horizontally.

4. Move the cursor to a location whose coordinates are 40, -30. The coordinates of the point can be seen on top of the cursor.

5. Specify the endpoint of the line at this location. A rubber band line is attached to the cursor. Move the cursor vertically upward.

6. Specify the endpoint of the second line on the point whose coordinates are 40mm, -20mm.

   A rubber band line is attached to the cursor.

7. Move the cursor horizontally toward the left and specify the endpoint of the third line where the value of the coordinates is 30, -20.

   After drawing these three lines, draw a tangent arc using the **Tangent Arc** option available in the **Profile** tool.

8. Choose the **Tangent Arc** button available in the **Sketch tools** toolbar.

9. Move the cursor to a location whose coordinates are 20, -10 and specify the endpoint of the tangent arc. Figure 1-39 shows the sketch, after drawing three lines and the tangent arc. The system switches back to the **Line** mode.
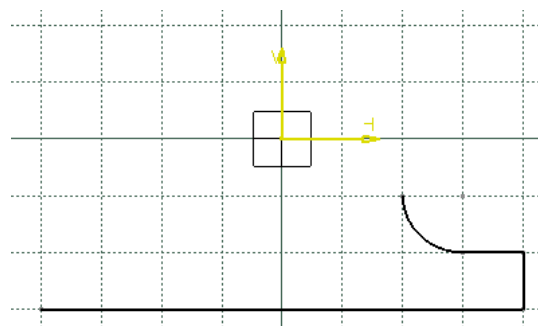
Figure 1-39 *Sketch after drawing three lines and a tangent arc*

10. Move the cursor vertically upward to a location whose coordinates are 20, 10.

11. Specify the endpoint of the line at this location.

Next, you need to draw a tangent arc by switching to the arc mode using the **Tangent Arc** option available in the **Profile** tool.

12. Choose the **Tangent Arc** button from the **Sketch tools** toolbar.

13. Move the cursor to a location whose coordinates are 30, 20 and specify the endpoint of the tangent arc.

    The system switches back to the **Line** mode.

14. Move the cursor horizontally toward the right and specify the endpoint of the line, when the value of the coordinates is 40, 20.

15. Move the cursor vertically upward and specify the endpoint of the line, when the value of the coordinates is 40, 30.

16. Move the cursor horizontally toward the left and specify the endpoint of the line, when the value of the coordinates is -40, 30.

17. Move the cursor vertically downward and specify the endpoint of the line, when the value of the coordinates is -40, 20.

18. Move the cursor horizontally toward the right and specify the endpoint of the line, when the value of the coordinates is -30, 20.

    Next, you need to draw a tangent arc by switching to the tangent arc mode.

19. Choose the **Tangent Arc** button from the **Sketch tools** toolbar to switch the tangent arc mode.

20. Move the cursor to a location whose coordinates are -20, 10 and specify the endpoint of arc at this location.

    The system switches back to the Line mode.

21. Move the cursor vertically downward and specify the endpoint of the line, where the value of the coordinates is -20, -10.

22. Switch to the Tangent mode and move the cursor to a location whose coordinates are -30mm, -20mm. Specify the endpoints of the tangent arc at this location.

23. Move the cursor horizontally toward the left and specify the endpoint of the line, when the value of coordinates is -40, -20.

24. Move the cursor vertically downward and specify the endpoint of the line when it snaps the start point of the outer loop. The sketch after completing the outer loop of the sketch, after hiding the constraints, is shown in Figure 1-40.
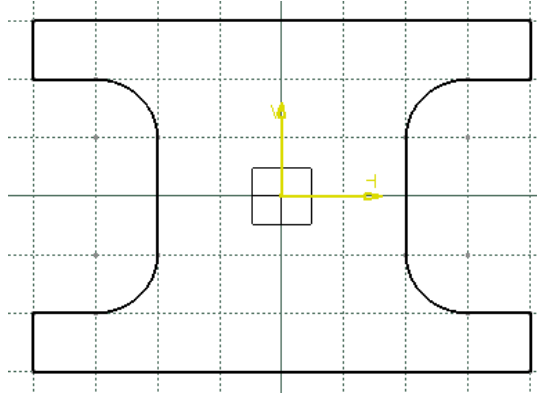
Figure 1-40 *Sketch after drawing outer loop of the sketch*

Drawing the Inner Cavity of the Sketch

After drawing the outer loop of the sketch, you need to draw its inner rectangular cavity. You will use the **Rectangle** tool to draw the inner cavity.

1. Choose the **Rectangle** tool from the **Profile** toolbar.

2. Move the cursor to a location whose coordinates are -10, 10. Specify the upper left corner of the rectangle at this location.

3. Move the cursor to a location whose coordinates are 10, -10. Specify the lower-right corner of the rectangle at this location.

4. Choose the Fit All in button from the View toolbar to fit the sketch in the geometry area.

    The final sketch, after drawing the inner loop, is shown in Figure 1-41. Note that the display of constraints has been turned on in this figure.
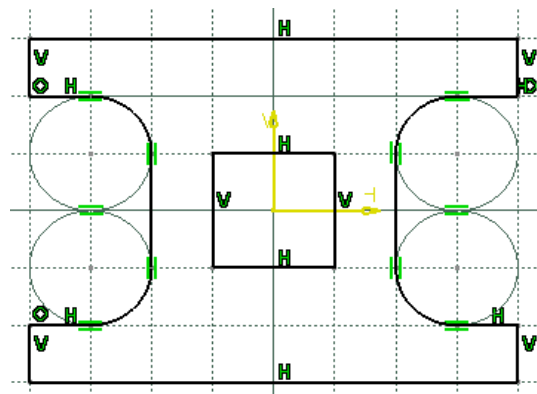
Figure 1-41 *Final sketch after drawing inner loop of the sketch*

Conclusion

After completing the sketch you need to save it. As mentioned earlier, you need to save each tutorial of this chapter in the *c01* folder in the *CATIA* folder.

1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box. Browse for the folder named *c01* that you created in the last tutorial.

2. Enter the name of the file as *c01tut2* in the **File name** edit box and choose the **Save** button. The file will be saved in the *\My Documents\CATIA\c01* folder.

3. Close the part file by choosing **File > Close** from the menu bar.

Additional Viva Questions:

1. Expand CATIAV6.

2. Write the function of part workbench in catia v5.

3. What are modeling tools in part workbench.

4. List the part tools used in catiav5.

5. Can you tell which tool is used to exit from part workbench.

6. How to get into part module from default catia window.

7. Write the use of dimensional constraints.

8. What are the steps required to go to sketch module from part.

9. How do you measure arc radius.

10. Write the use of colors provided in catia.

Innovative Questions:

1. When to use dimensional constraints in catia.

2. Can we generate 2D model with out sketcher.

3. On what machines catiav6 should be run.

4. Where can we use catia modules for?

5. How to edit part module in catia?

6. Tell about catia applications used in manufacturing industry.

7. How do you make use of catiav6.

8. Categorize the catia versions developed by dassault system.

9. What type of programming language uses catiav6.

10. Where catia v6 modules can be applicable in manufacturing.

S.NO.	QUESTION	LEVEL
1.	Using the part workbench design the given sketch using catia.	L5
2.	Choose the correct module for the given sketch to design in catia.	L6
3.	List the steps required and draw the given component using catia.	L4
4.	From the given sketch make use of correct module and complete it.	L3

EXERCISE – 3

Generation of 3D model using CATIA V5

Objective:

- To develop a 3D model of given component using CATIA V5.

Out comes:

- Understand how to use modeling commands to complete the given sketch.
- Understand the importance of using limits and view settings.

Scope of the experiment:

To provide an introduction to the use of modeling and sketcher commands in drawing 3D sketch and experience them how to apply and where to apply the various commands. Know the difference between draw and modify commands.

Description of software:

The software used for doing this sketch is catia v5 and it is developed by Dassault Systèmes. CATIA software(/kəˈtiːə/, an acronym of computer-aided three-dimensional interactive application) is a multi-platform software suite for computer-aided design (CAD), computer-aided manufacturing (CAM), computer-aided engineering (CAE), PLM and 3D, developed by the French company Dassault Systèmes. It is a three dimensional modeling tool. This tool is used in the design of various objects such as vehicles, buildings, components, etc.

Figure 1-3 *Initial screen that appears after starting CATIA V5R13*

Note

Even if you draw any sketch in the space containing the hidden elements, it will not be visible and will be displayed only after you return back to the visible geometry area.

You can change the standard element to a construction element in this space or vice-a-versa.

Procedure

In this exercise, you will draw the sketch of the model shown in Figure 1-42. The sketch is shown in Figure 1-43. You will not dimension the sketch. The solid model and the dimensionsare given only for your reference.
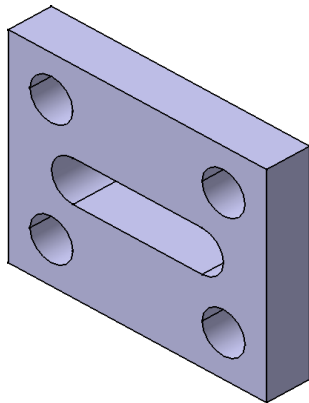
Figure 1-42 Solid model for Tutorial 3
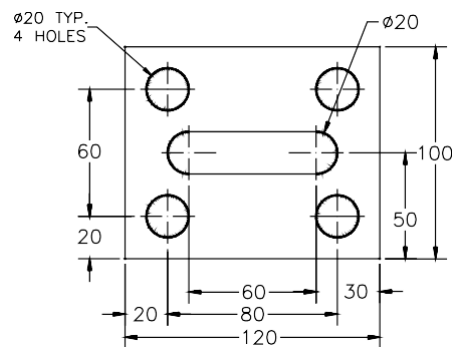
Figure 1-43 Sketch for the solid model

The following steps are required to complete this tutorial:

a. Start a new CATpart file.
b. Draw the sketch of the model using the **Rectangle**, **Profile**, and the **Circle** tools, refer to Figures 1-44 through 1-46.
c. Save the sketch and close the file.

Starting New Part File
1. Choose **File > New** from the menu bar; the **New** dialog box is displayed.

2. Select **Part** from the **List of Types** list box in this dialog box. Choose the **OK** button. A new file in the **Part Design** workbench will be opened.

3. Choose the **Sketcher** button from the **Sketcher** toolbar and then select the YZ plane as the sketching plane.

This sketch will be drawn in two parts. Initially, you will draw the outer loop of the sketch, that is, a rectangle. Next, you need to draw the inner loops of the sketch, which consists of multiple inner loops that are four holes and an elongated hole. First you will draw an elongated hole using the **Profile** tool and then the four holes using the **Circle** tool.

Drawing the Outer Loop of the Sketch
The outer loop of the sketch will be drawn using the **Rectangle** tool.

1. Choose the **Rectangle** button from the **Profile** toolbar.

2. Move the cursor to a location whose coordinates are -60, -50 and specify the lower left corner of the rectangle.

3. Move the cursor to the location whose coordinates are 60, 50 and specify the upper right corner of the rectangle. Figure 1-44 shows the outer loop of the sketch drawn using the **Rectangle** tool.
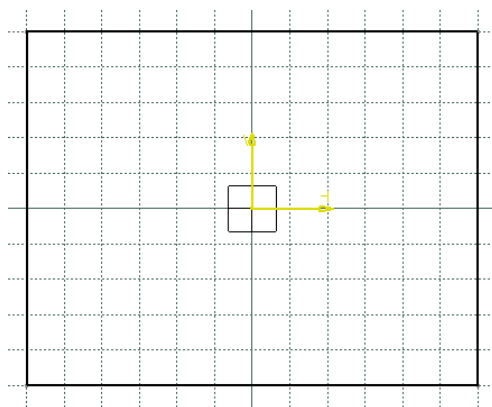
Figure 1-44 Outer loop of the sketch

Drawing the Inner Loop of the Sketch
After drawing the outer loop of the sketch, draw its inner loop.

1. Choose the **Profile** button from the **Profile** toolbar.

2. Move the cursor to a location whose coordinates are -30, 10 and specify the start point of the line.

3. Move the cursor horizontally toward the right and specify the endpoint of the line, when the value of the coordinates is 30, 10.

Next, you need to draw a tangent arc by switching over to the **Tangent Arc** option using the **Sketch tools** toolbar.

4. Choose the **Tangent Arc** button from **Sketch tools** toolbar to switch over to the arc mode.

5. Move the cursor to a location whose coordinates are 30, -10 and specify the endpoint of the tangent arc.

    The system switches over to the **Line** mode.

6. Move the cursor to the location whose coordinates are -30, -10 and specify the endpoint of the line.

7. Choose the **Tangent Arc** button from the **Sketch tools** toolbar to switch over to the arc mode.

8. Move the cursor to the start point of the first horizontal line of the elongated hole. Specify the endpoint of the arc when it snaps the start point.

    The sketch, after drawing the elongated hole, is shown in Figure 1-45.
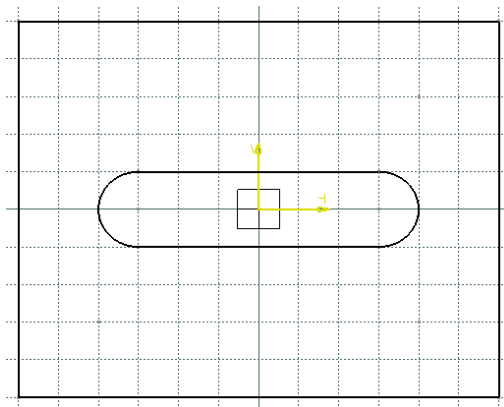
Figure 1-45 Sketch after drawing the elongated hole

9. Choose the **Circle** button from the **Circle** toolbar.

    Note
    If you have closed the Circle toolbar, you can use the Profile toolbar to invoke the Line tool.

10. Move the cursor to a location whose coordinates are 40, 30 and specify the center point of the circle.

11. Specify the value of **10** as the radius in the **Radius** edit box available in the **Sketch tools** toolbar.

    You will observe that a radius dimension is displayed attached to circle because you have specified the value of the radius in the **Radius** edit box available in the **Sketch tools** toolbar.

12. Choose the **Circle** button from the **Circle** toolbar.

13. Move the cursor to a location whose coordinates are 40, -30 and specify the center point of the circle.

14. Specify the value of 10 as the radius in the **Radius** edit box available in the **Sketch tools** toolbar.

15. Similarly, draw the other two circles. The coordinates of the center point of other two circles are -40, 30 and -40, -30 respectively. The final sketch, with the display of constraints turned on, is shown in Figure 1-46.
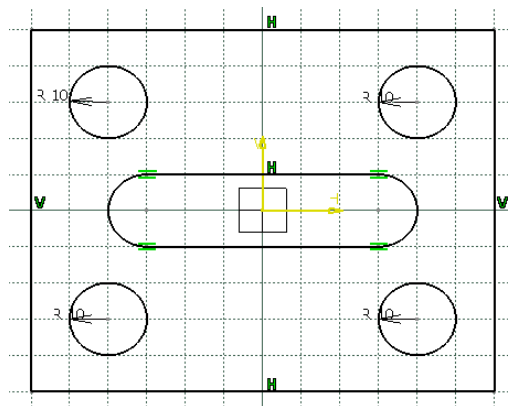
Figure 1-46 *Final sketch*

Conclusion

1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box. Browse for the folder named *c01* that you created in the first tutorial.

2. Enter the name of the file as *c01tut3* in the **File name** edit box and choose the **Save** button. The file will be saved in the *\My Documents\CATIA\c01* folder.

3. Close the part file by choosing **File > Close** from the menu bar.

1. Expand PLM.

2. Write the function of part workbench in catia v6.

3. What are modify tools in part workbench.

4. List the draw tools used in catiav5.

5. Can you tell which tool is used to exit from drafting workbench.

6. How to get into assembly module from default catia window.

7. Write the use of constraints.

8. What are the steps required to go to drafting module from part.

9. How do you measure a sketch in part module.

10. Mention the use of color code provided in catia.

Innovative Questions:

1. When to use constraints in catia.

2. Can we generate 3D model in drafting module directly.

3. On what machines catiav6 does not run.

4. Where can we use drafting module for?

5. How to edit a sketch in drafting module in catia?

6. Tell about surface module applications used in manufacturing industry.

7. How do you make use of solid modeling module.

8. Categorize the catia modules developed by dassault system.

9. What type of modules uses catiav6 for various purposes in manufacturing industry.

10. Where catia v5 modules can be applicable in manufacturing.

S.NO.	QUESTION	LEVEL
1.	Using the part workbench draft the given sketch using catia.	L5
2.	Choose the correct module for the given sketch to design in catia.	L6
3.	List the steps required and draw the given component using catia.	L4
4.	From the given sketch make use of correct module and complete it.	L3

2. ANSYS

EXERCISE – 4

Determination of deflections and stresses in 2D truss

Objective:

- To determine deflections and stresses in 2D truss using ansys software.

Out comes:

- Understand how to use ansys tools to analyse the given sketch.
- Understand the steps and menus used in ansys in various steps.

Scope of the experiment:

To provide an introduction to the use of ansys software in determining deflections and stresses of 2D truss and experience them how to apply and where to apply the various loads.

Description of software:

The software used for doing this exercise is ansys apdl and it is developed by Ansys, Inc. Regardless of the type of simulation, each model is represented by a powerful scripting language … the Ansys Parametric Design Language (APDL). APDL is the foundation for all sophisticated features, many of which are not exposed in the Workbench Mechanical user interface. It also offers many conveniences such as parameterization, macros, branching and looping, and complex math operations. All these benefits are accessible within the Ansys Mechanical APDL user interface.
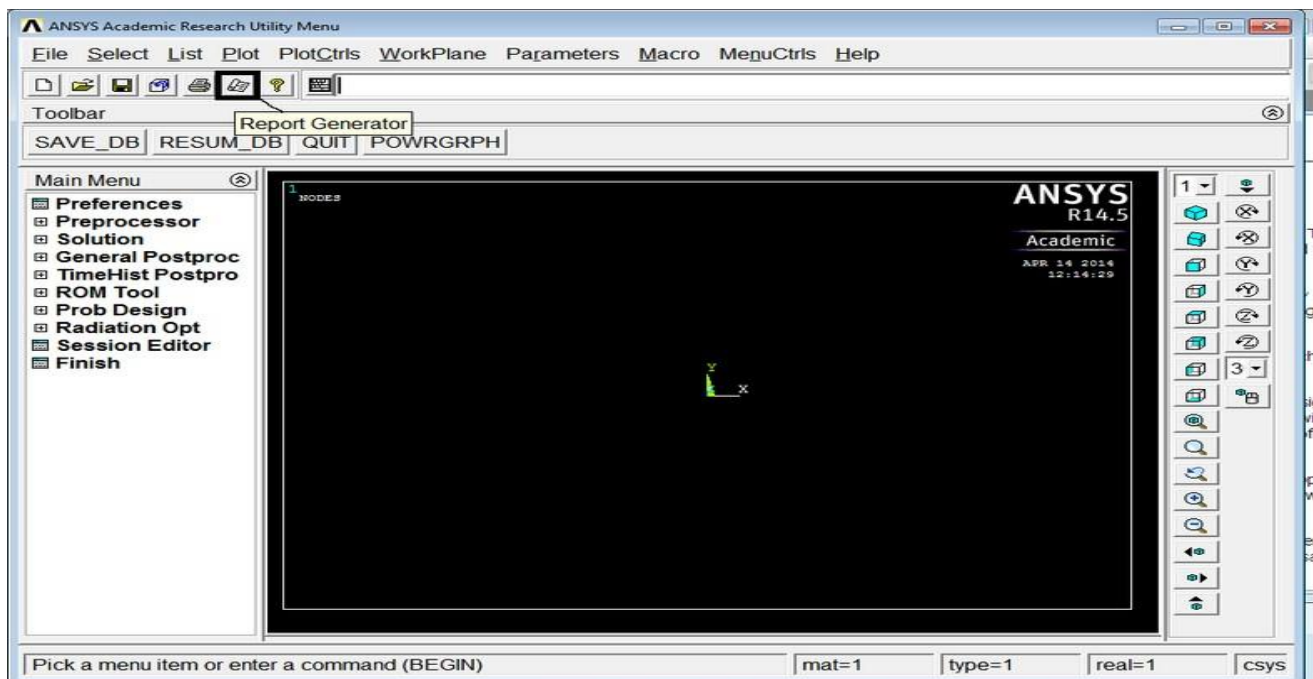
FORCE AND STRESS ANALYSIS USING TWO LINK ELEMENTS IN TRUSSES

GENERAL STEPS

Step 1: Ansys Utility Menu

      File – clear and start new – do not read file – ok

      File – change job name – enter new job name – xxxx – ok

      File – change title – enter new title – yyy – ok

Step 2: Ansys Main Menu – Preferences select – STRUCTURAL - ok

Step 3: Preprocessor

      Element type – select type of element from the table and the required options

      Real constants – give the details such as thickness, areas, moment of inertia, etc. required depending on the nature of the problem.

      Material Properties – give the details such as Young's modulus, Poisson's ratio etc. depending on the nature of the problem.

Step 4: Modeling – create the required geometry such as nodes elements, area, and volume by using

      the appropriate options.

Step 5: Generate – Elements/ nodes using Mesh Tool if necessary (in 2D and 3D problems)

Step 6: Apply boundary conditions/loads such as DOF constraints, Force/Momentum, Pressure etc.

Step 7: Solution – Solve the problem

Step 8: General Post Processor – plot / list the required results.

Step 9: Plot ctrls – animate – deformed shape – def+undeformed-ok

Step 10: to save the solution

PROCEDURE:

1. Ansys Main Menu – Preferences-select – STRUCTURAL- h method – ok

2. Element type – Add/Edit/Delete – Add – Link – 3D Finit stn 180 – ok – close.

3. Real constants – Add – ok – real constant set no – 1 – c/s area – 0.1 – ok – close.

4. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX
   1. 210e9– Ok – close.

5. Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS– 0.75 (x value w.r.t first node) – apply (second node is created) – x,y,z location in CS –(0, -0.5),(x, y value w.r.t first node) – ok (third node is created

6. Create–Elements–Elem Attributes – Material number – 1 – Real constant set number
   1. 1 – ok

7. Auto numbered – Thru Nodes – pick 1 & 2 – apply – pick 2 & 3–– ok (elements are created through nodes).

8.   Loads – Define loads – apply – Structural – Displacement – on Nodes – pick node 1 &3 – apply – DOFs to be constrained – All DOF – ok

9.   Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply – direction of For/Mom – FY – Force/Moment value – 5000 (-ve value)

10.   Solve – current LS – ok (Solution is done is displayed) – close.

11.   Element table – Define table – Add –'Results data item' – By Sequence num – LS – LS1 – ok.

12.   Plot results – contour plot –Element table – item to be plotted LS,1, avg common nodes- yes average- ok.

13.   List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).

14.   Plot results- nodal solution-ok-DOF solution- Y component of displacement-ok.

15.   Animation: PlotCtrls – Animate – Deformed shape – def+undeformed-ok.

CONCLUSION:

     Thus the force and stress analysis of two link element in trusses is done by using the ANSYS Software.

1. How to import data in ansys from an external file.

2. Write the function of preprocessor used in ansys.

3. What is resume in ansys.

4. List the steps involved in ansys.

5. Can you tell how to import data from cad with an example.

6. How to export data from ansys to cad.

7. Write about p- method.

8. What is optimization in ansys.

9. How to display results in ansys.

10. Categorize the different menus in ansys.

Innovative Questions:

1. How to open and save ansys apdl file?

2. Can we run apdl in ansys work bench .

3. On what machines ansys should be run.

4. Where can we use ansys apdl and workbench .

5. How to add noise to an image in ansys?

6. Tell about ansys application program software.

7. How do you make use of ansys function library?

8. Categorize the ansys menus used in various steps.

9. What type of programming language is ansys.

10. Where ansys can be applicable.

S.NO.	QUESTION	LEVEL
1.	From the given sketch modify the boundaries and calculate deflections and stresses.	L5
2.	Conclude the maximum stress and deflection from the given sketch.	L6
3.	Analyze the given sketch and generate results using ansys software.	L4
4.	From the given sketch how do you make use of ansys to determine stresses and deflection.	L3

EXERCISE – 5

Determination of deflections and stresses in beams

Objective:

- To determine deflections and stresses in beams using ansys software.

Out comes:

- Understand how to use ansys tools to analyse the given sketch.
- Understand the steps and menus used in ansys in various steps.

Scope of the experiment:

To provide an introduction to the use of ansys software in determining deflections and stresses of beams and experience them how to apply the various loads.

Description of software:

The software used for doing this exercise is ansys apdl and it is developed by Ansys, Inc. Regardless of the type of simulation, each model is represented by a powerful scripting language … the Ansys Parametric Design Language (APDL). APDL is the foundation for all sophisticated features, many of which are not exposed in the Workbench Mechanical user interface. It also offers many conveniences such as parameterization, macros, branching and looping, and complex math operations. All these benefits are accessible within the Ansys Mechanical APDL user interface.

STRESS AND DEFLECTION ANALYSIS IN CANTILEVER BEAM WITH POINT LOAD

GENERAL STEPS

Step 1: Ansys Utility Menu

  File – clear and start new – do not read file – ok

  File – change job name – enter new job name – xxxx – ok

  File – change title – enter new title – yyy – ok

Step 2: Ansys Main Menu – Preferences select – STRUCTURAL - ok

Step 3: Preprocessor

  Element type – select type of element from the table and the required options

  Real constants – give the details such as thickness, areas, moment of inertia, etc. required depending on the nature of the problem.

  Material Properties – give the details such as Young's modulus, Poisson's ratio etc. depending on the nature of the problem.

Step 4: Modeling – create the required geometry such as nodes elements, area, and volume by using

  the appropriate options.

Step 5: Generate – Elements/ nodes using Mesh Tool if necessary (in 2D and 3D problems)

Step 6: Apply boundary conditions/loads such as DOF constraints, Force/Momentum, Pressure etc.

Step 7: Solution – Solve the problem

Step 8: General Post Processor – plot / list the required results.

Step 9: Plot ctrls – animate – deformed shape – def+undeformed-ok

Step 10: to save the solution

PROBLEM:

Compute the Shear force and bending moment diagrams for the beam shown and find the maximum deflection. Assume rectangular c/s area of 0.2 m * 0.3 m, Young's modulus of 210 GPa, Poisson's ratio 0.27.
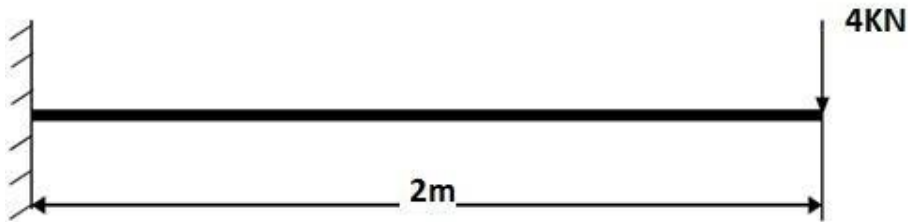
PROCEDURE:

1. Ansys Main Menu – Preferences-Select – STRUCTURAL- h method – ok

2. Element type – Add/Edit/Delete – Add – BEAM – 2 node Beam 188 – ok- close.

3. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX
   – 210e9– PRXY – 0.27 –ok – close.

4. Sections-Beams-common sections- sub type- rectangle (1st element) - enter b=200, h=300-
   preview-ok.

5. Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in
   CS– 2 (x value w.r.t first node) – ok (second node is created).

6. Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – ok (elements are created
   through nodes).

7. Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 – apply –

8. DOFs to be constrained – ALL DOF – ok.

9. Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply –
   direction of For/Mom – FY – Force/Moment value –(-40000) (-ve value) – ok.

10. Solve – current LS – ok (Solution is done is displayed) – close.

11. Displacement: Plot Results – Contour plot – Nodal solution – DOF solution – displacement vector sum – ok.

12. Stress: Plot Results – Contour plot – Nodal solution – stress – von mises stress – ok.

13. Element table – Define table – Add – 'Results data item' – By Sequence num – SMISC – SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 19 – apply, By Sequence num – SMISC – SMISC, 3 – apply, By Sequence num – SMISC – SMISC, 16 – ok – close.

14. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS19 – ok (Shear force diagram will be displayed).

15. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS3 – Elem table item at node J – SMIS16 – ok (bending moment diagram will be displayed).

16. Reaction forces: List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).

CONCLUSION:

Thus the stress and deflection analysis in cantilever beam with point load is done by using the ANSYS Software.

1. How to import file in ansys workbench from cad.

2. Write the function of post processor used in ansys.

3. What are different approximate solution methods .

4. List the steps involved in FEA.

5. Can you give example for higher order elements.

6. How to obtain stiffness matrix.

7. Write about h- method.

8. What is stiffness matrix in ansys.

9. How to identify order of elements.

10. Categorize the different menus in ansys workbench.

Innovative Questions:

1. How to open and save ansys workbench file?

2. Can we open cad model in ansys and write its procedure.

3. On what machines ansys workbench should be run.

4. Where can we use ansys software in industries.

5. How to add noise to an image in ansys workbench?

6. Tell about ansys workbench application program software.

7. How do you make use of ansys workbench function library?

8. Categorize the ansys workbench menus used in various steps.

9. What type of programming language in ansys workbench is used.

10. Where ansys workbench can be applicable.

S.NO.	QUESTION	LEVEL
1.	From the given sketch modify the boundaries and calculate deflections and stresses.	L5
2.	Conclude the maximum stress and deflection from the given sketch.	L6
3.	Analyze the given sketch and generate results using ansys software.	L4
4.	From the given sketch how do you make use of ansys to determine stresses and deflection.	L3

EXERCISE – 6

Determination of deflections and stresses in beams

Objective:

- To determine deflections and stresses in beams using ansys workbench software.

Out comes:

- Understand how to use ansys tools to analyse the given sketch.
- Understand the steps and menus used in ansys in various steps.

Scope of the experiment:

To provide an introduction to the use of ansys software in determining deflections and stresses of beams and experience them how to apply the various loads.

Description of software:

The software used for doing this exercise is ansys workbench and it is developed by Ansys, Inc. Regardless of the type of simulation, each model is represented by a powerful scripting language … the Ansys Parametric Design Language (APDL). ANSYS Workbench platform is the backbone for delivering a comprehensive and integrated simulation.It is used to perform various types of structural, thermal, fluid, and electromagnetic analyses.
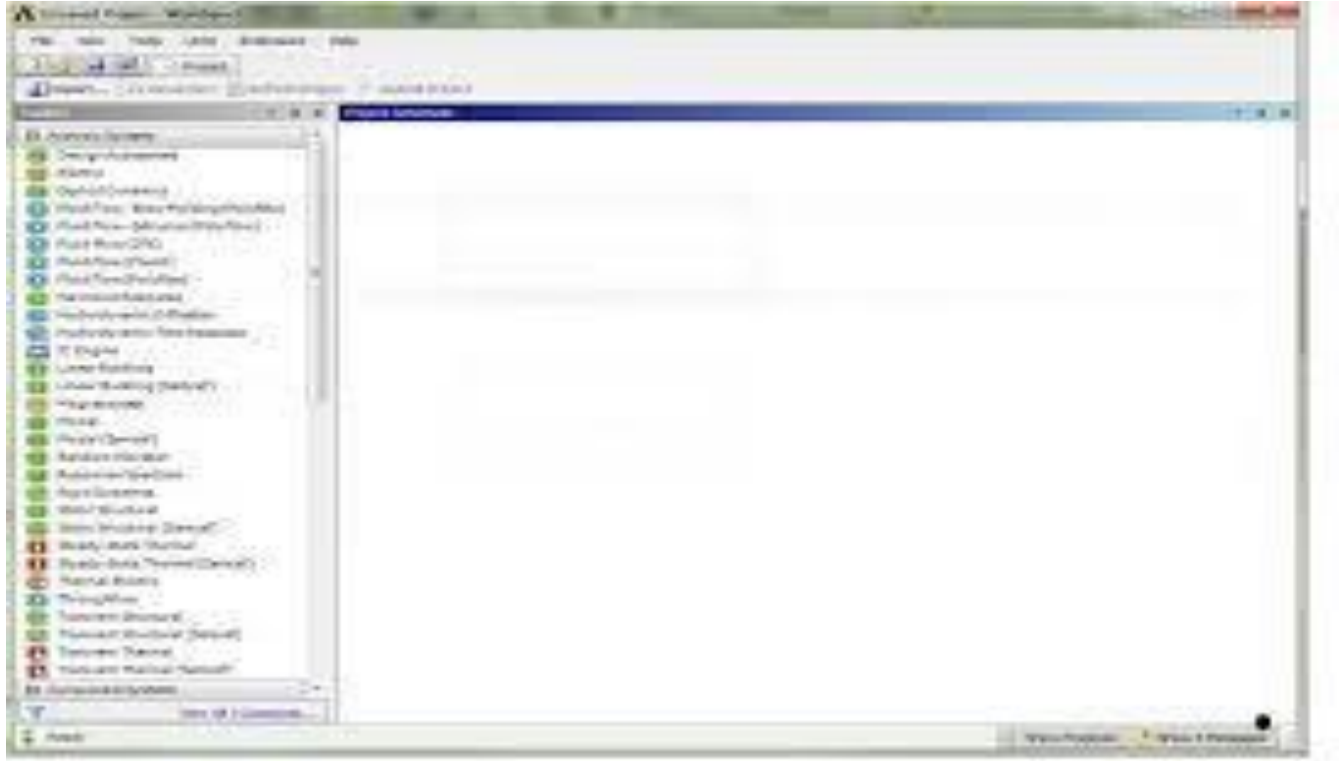
Given

Beam under 3-point bending with a centric applied force F as shown in Figure 1
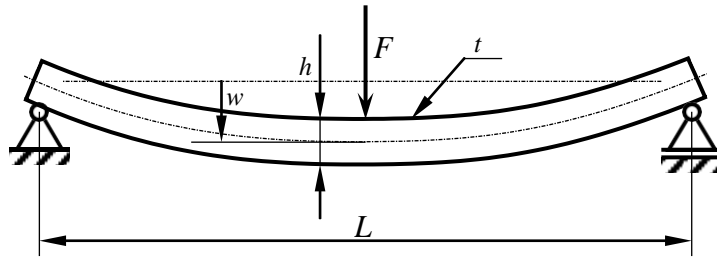
Figure 1: Beam under 3-point bending.

Relevant geometrical and material data for our problem are given in Table 1:

F	= 500,000 N	Applied force
L	= 2,000 mm	Length of the beam
h	= 60 mm	Height of the beam cross section
t	= 20 mm	Thickness of the beam cross section
E	= 210,000 N/mm²	Young's modulus
v	= 0.3	Poisson's ration
σ_{yield}	= 235 N/mm²	Allowable stress: yield stress of steel

Table 1: Geometry and material data.

Questions

1. Will the beam break and were would it start breaking?
2. If not, what would be the maximum deflection w?

Procedure:

Preprocessor (Setting up the Model)
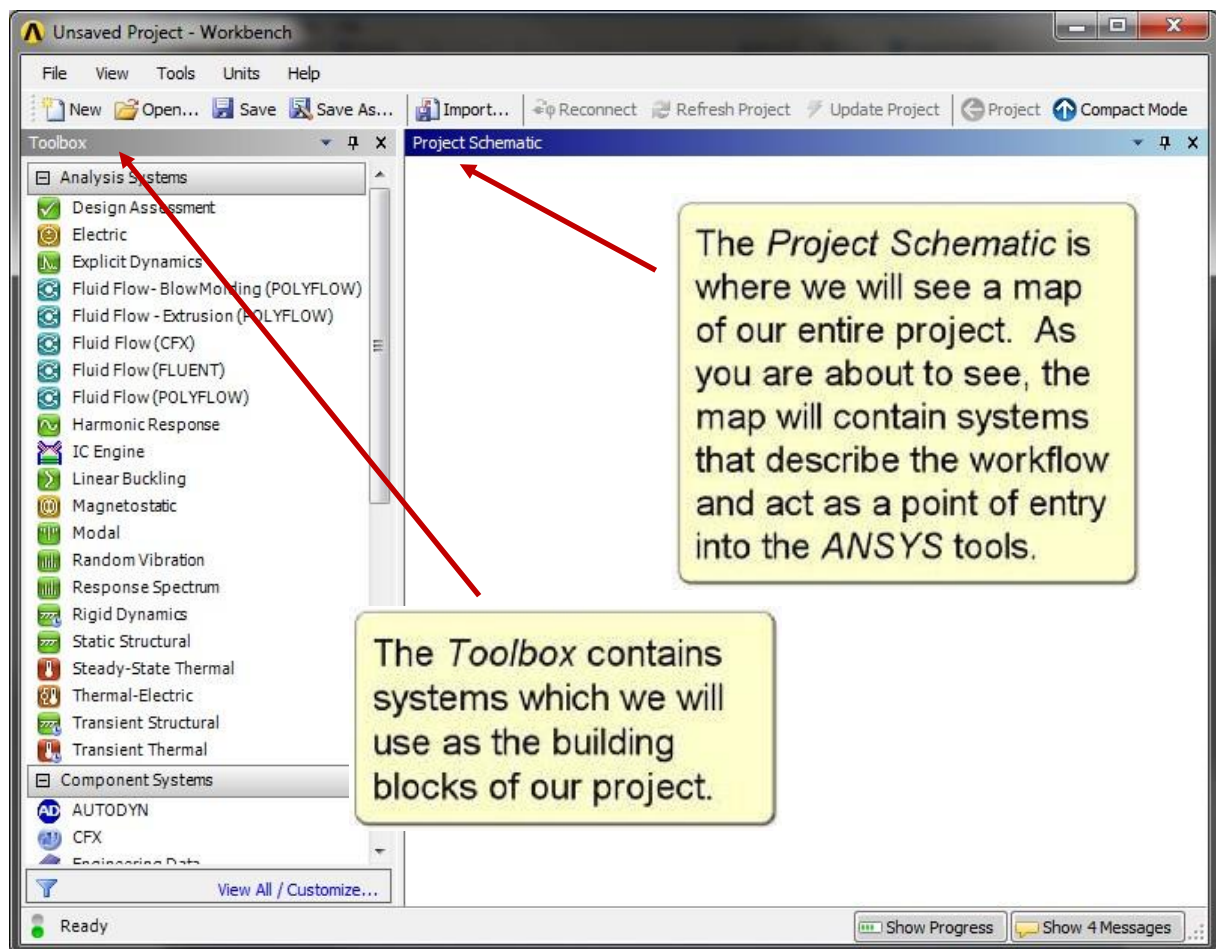
Figure 5: Interface of ANSYS Workbench 14.0

Build the Geometry Using DesignModeler Module

To begin; drag the **Static Structural Module** from the **Analysis Systems** toolbox and drop into the **Project Schematic** (Figure 6) and double click on the **Geometry** sub-module to open the **DesignModeler**. When asked, choose the desired unit system.

Choose **Create → Primitives → Box** from the main menu to create a beam by entering the desired dimensions in the **Details** box (Figure 7). Hit the **Generate** button to actually create the geometry.

When specifying the dimension of the beam, make sure that the origin of the coordinate system is located on the center of the beam's cross section. The beam axis must be oriented along the global x-axis.
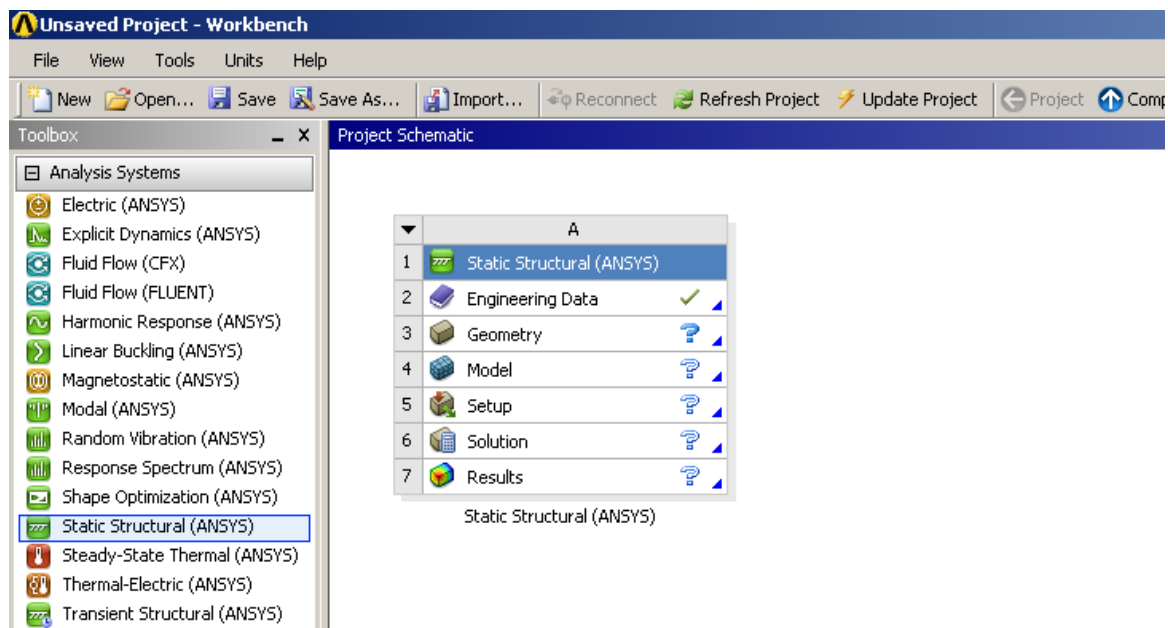
Figure 6: Drag-and-drop the Static Structural module to create a new analysis.
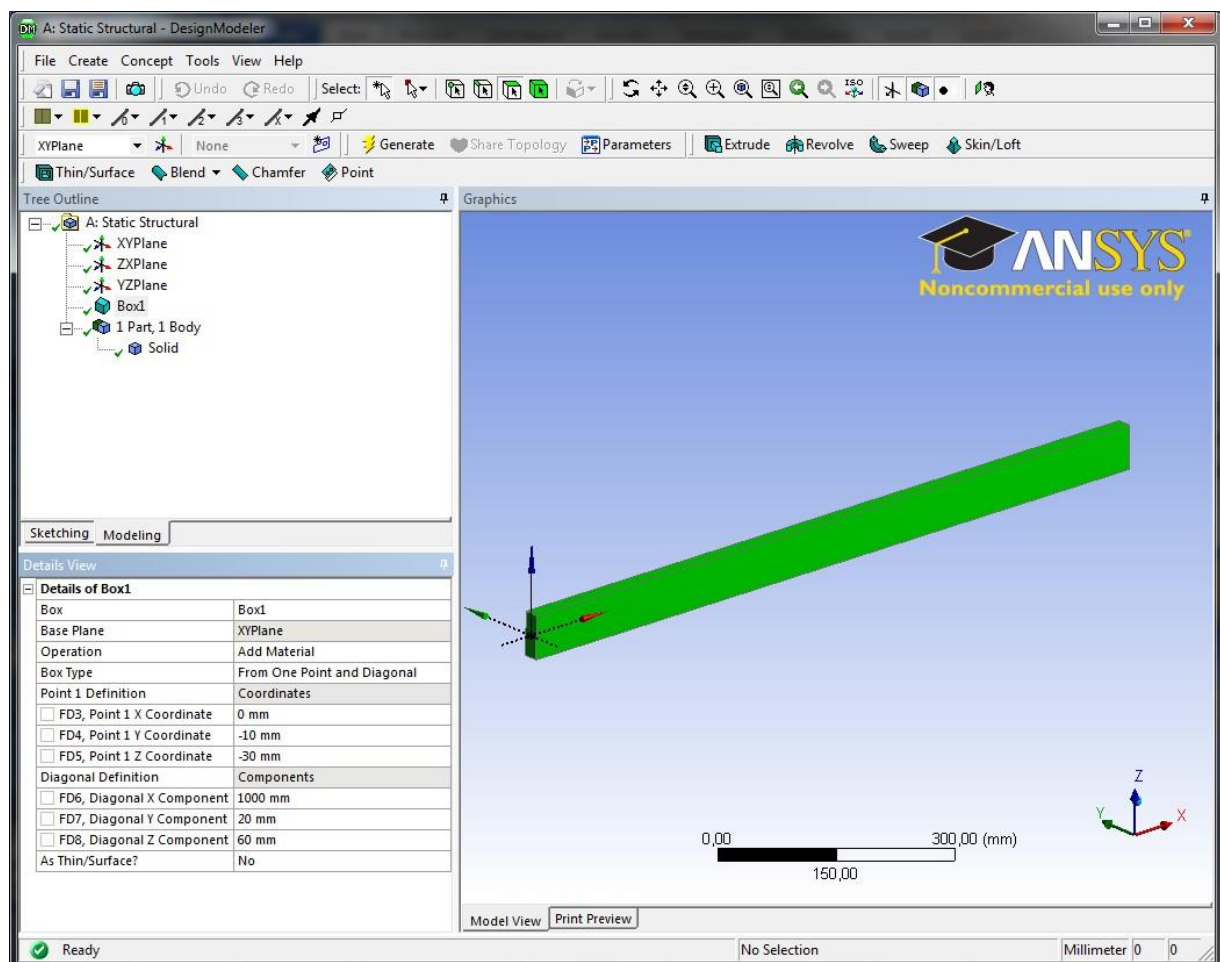
Figure 7: The beam on the DesignModeler

Note that ANSYS Workbench saves the model automatically; you can simply close **DesignModeler** now and continue with the next step.

Material Properties

i. Define Material Properties

Materials define the mechanical behavior of the FE model. We will use a simple linear-elastic, isotropic material model. In the project schematic, right click on **Engineering Data** to open a context menu and choose **Edit...** (Figure 8).
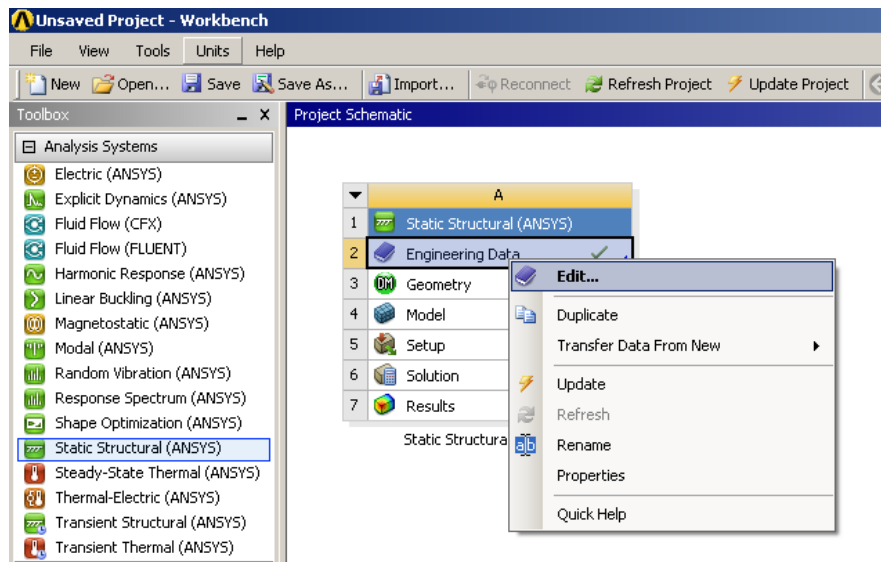
Figure 8: Edit engineering data

Enter a name for your new material in row 4 (Figure 9).
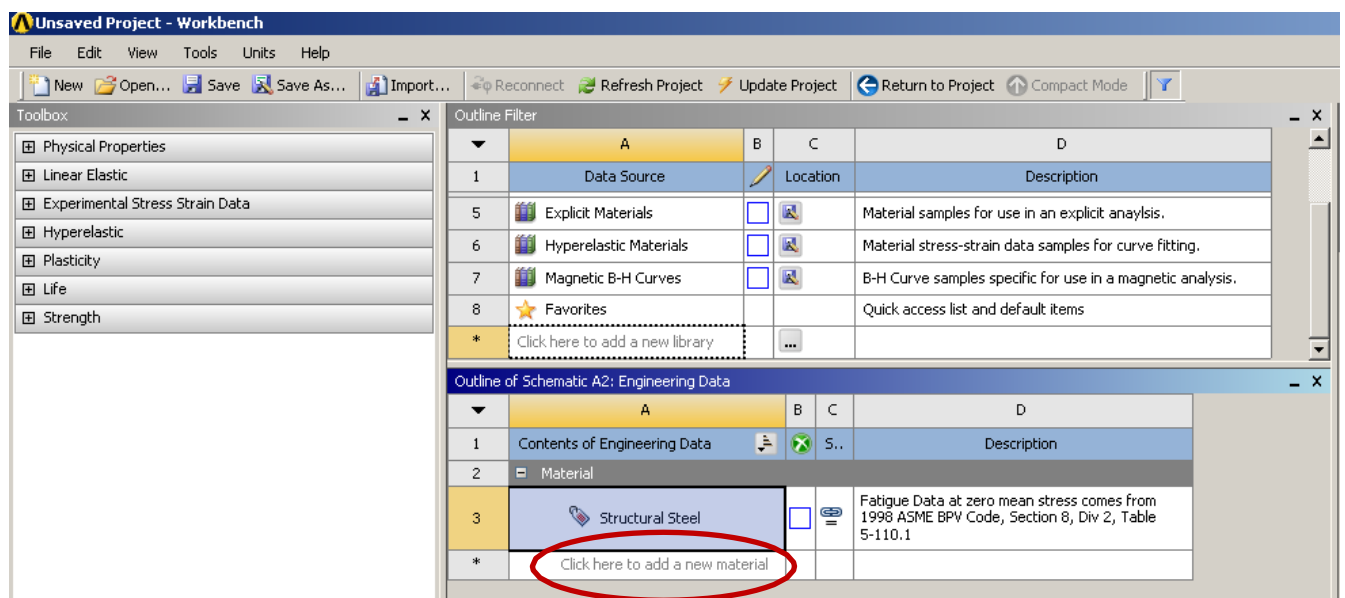
Figure 9: Naming the new material

Choose the simplest available material model by dragging the item **Isotropic Elasticity** from the **Toolbox** and dropping it onto the row with your newly defined material (Figure 10). **Isotropic Elasticity** requires certain material parameters: **Young's Modulus** and **Poisson's Ratio.**

After you entering the appropriate parameters do not forget to push **Return to Project** button and to update the project

Figure 10: Define the material properties.

Meshing

To mesh the solid body select double click the **Model** sub-module in the Project schematic to open the **Mechanical** module (Figure 11).
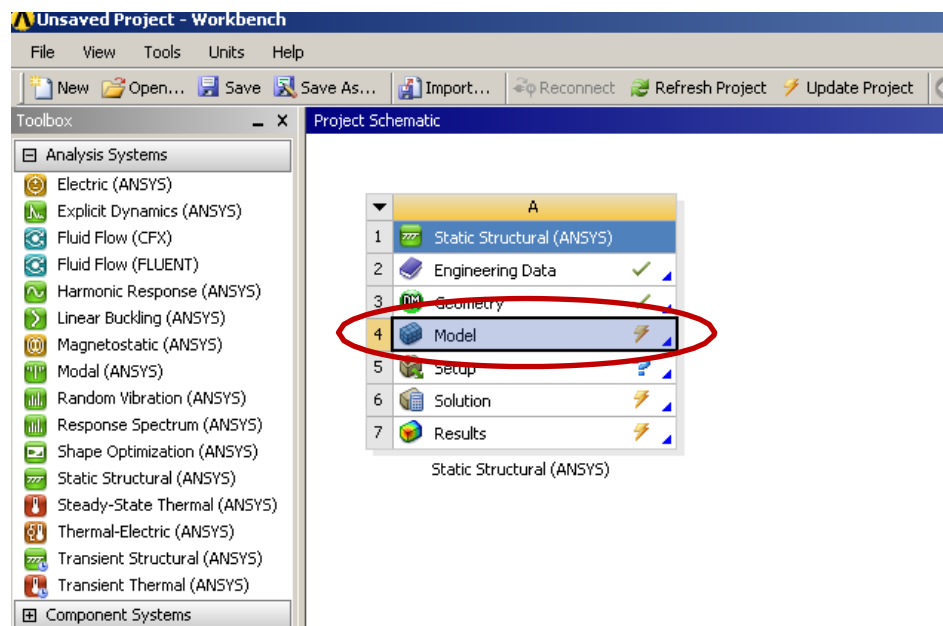

Figure 11: Starting the meshing and analysis module

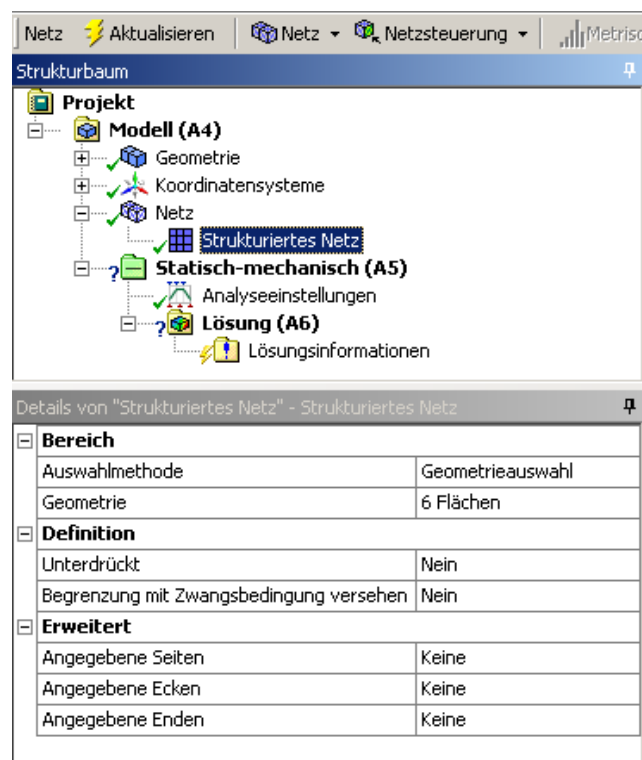Right click on **Mesh** in the **Structure Tree** and select a meshing method (Figure 12).

Mesh (right click) →Insert →Mapped Mesh

Figure 12: Setup meshing details

At this point focus on the **Details** box and select the geometry (all 6 faces) and do not forget to update the project; then click the **Generate** button.
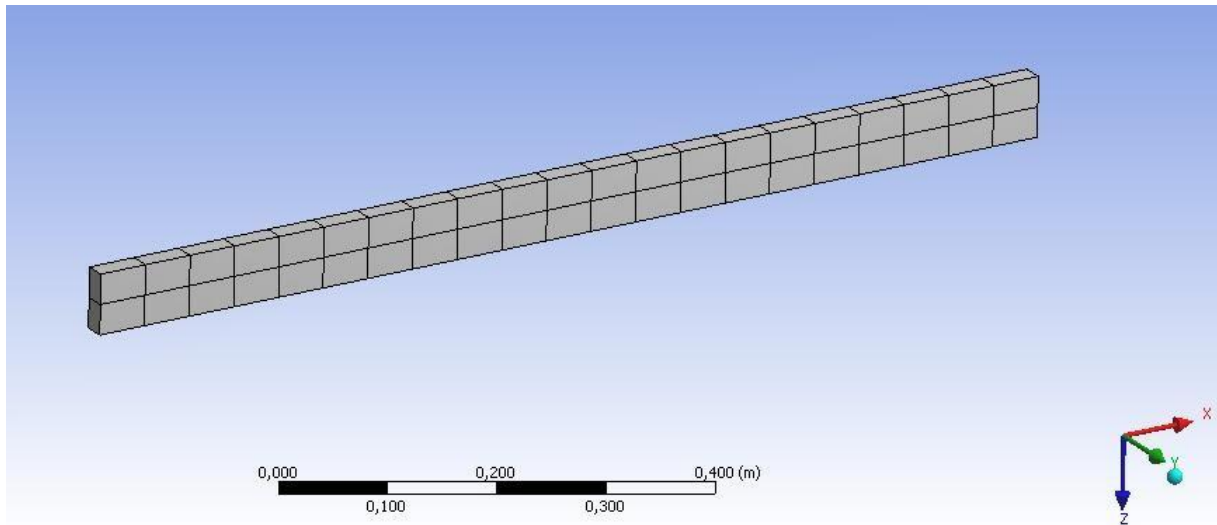
Figure 13: Meshed body

Applying Loads and Boundary Conditions

i. Boundary Conditions

To setup the necessary support:

Structure Tree → Static-Mechanic (right click) → Insert → Fixed Support

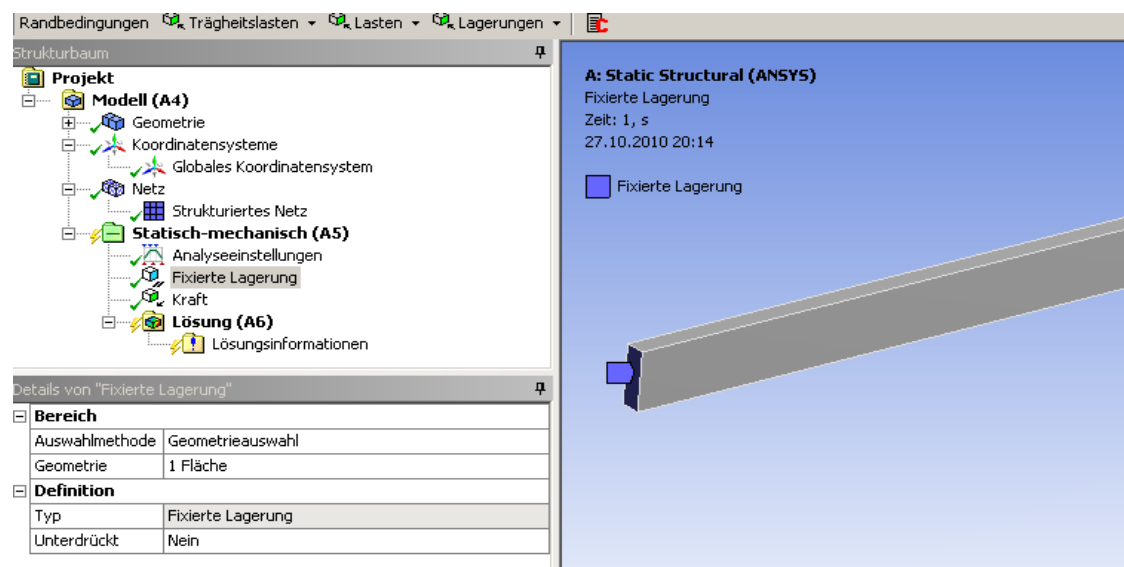**Select the Face** you want to assign as a fixed support.

Figure 14: Fixed Face of the body

ii.   Applying the Loads

Structure Tree → Static-Mechanic (right click) → Insert → Force

Select the Face you want to assign a force on this surface.

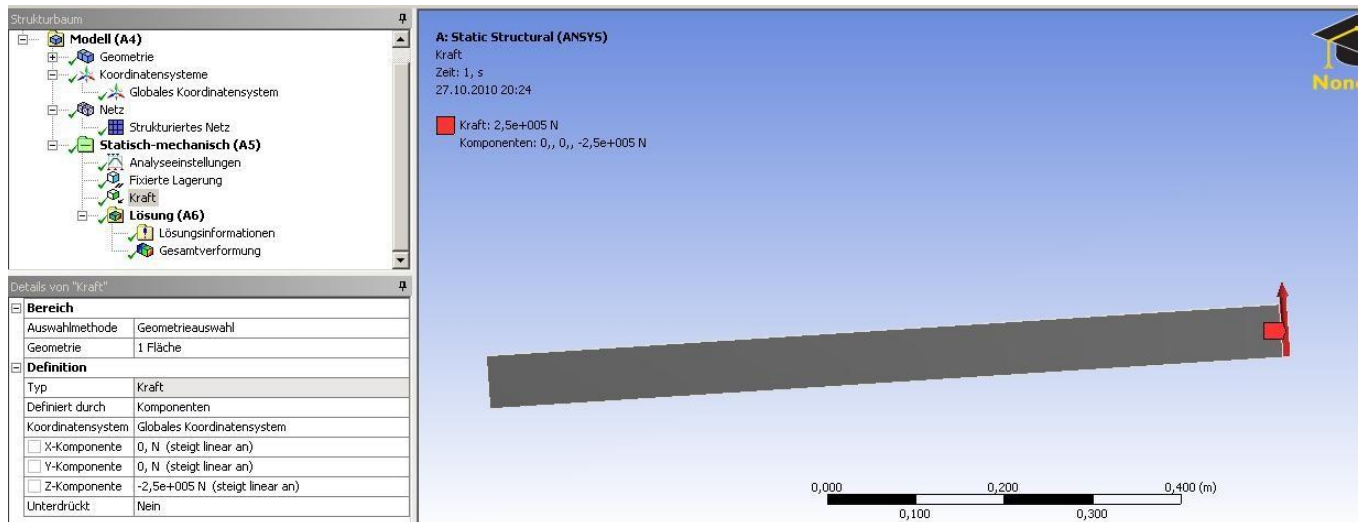Assign the components of the force vector. Now the problem is ready to be solved.
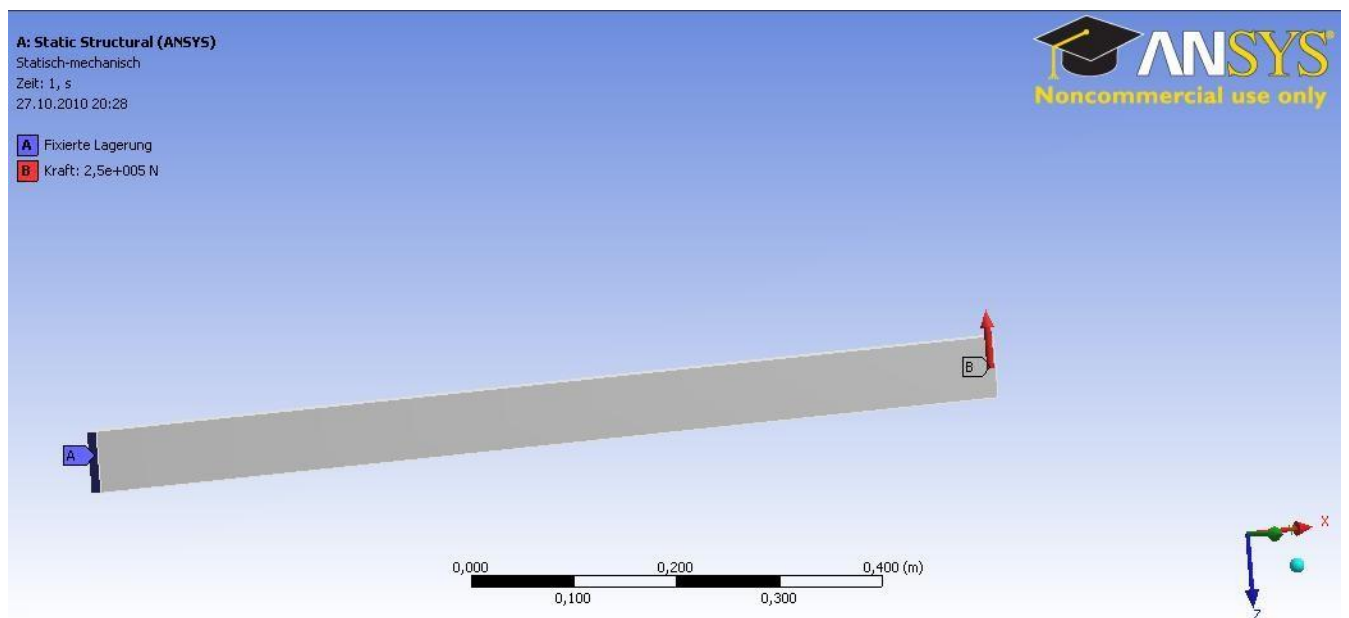

Figure 15: Applied force

Figure 16: Fixed face of the body and applied force

. Solving

To solve the problem add some solving tools to the **Solution** node which is part of the **Structure tree.**
Solution (right click)→ Total Deformation
Solution (right click)→ Total Strain
Solution (right click)→ Total Stress
Solution (right click)→ Von-Mises Equivalent Stress
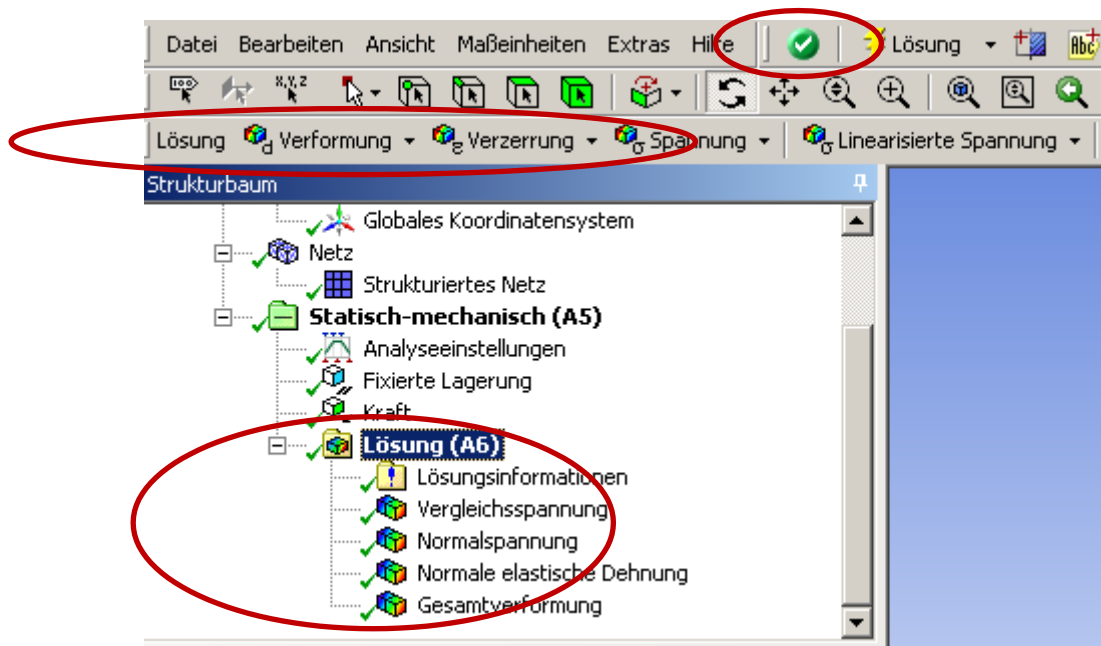
Click **Solve**

Figure 17: Required solving tools
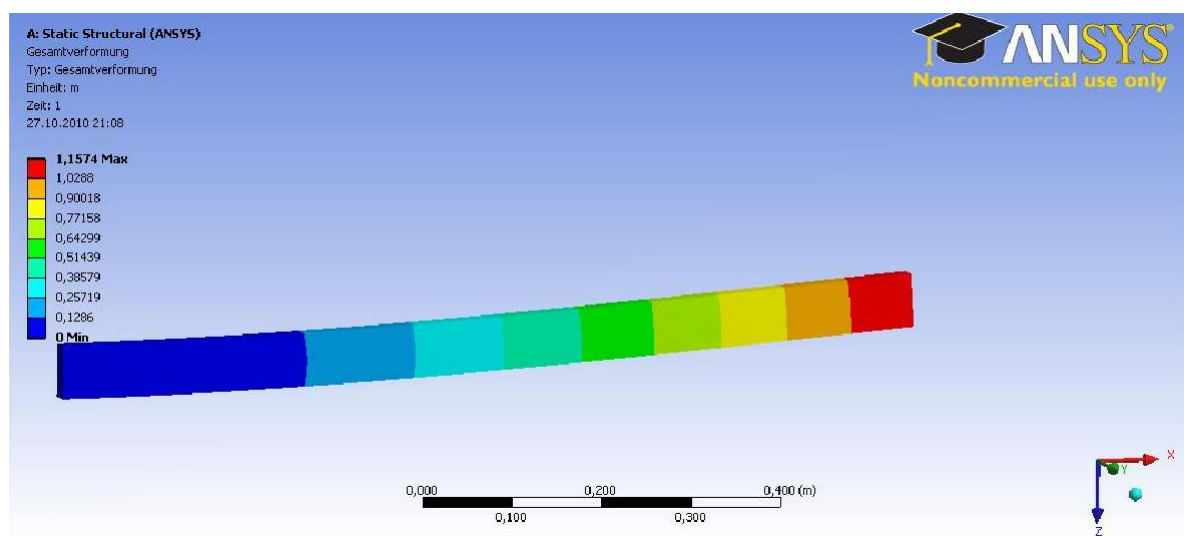
a. Contour Plot of Deformed Shape

Figure 17: <u>SCALED</u> Displacement solution (z direction) of the beam (with too high force).

1. What is the total DOF of ansys commercial package.

2. Write about workspace.

3. What is swap face.

4. List the scalar parameters in ansys.

5. Can you give example for low order elements.

6. How to obtain fine meshing in workbench.

7. Write about primary nodes.

8. What is mirror in ansys.

9. How to identify order of elements.

10. Categorize the different menus in ansys workbench.

Innovative Questions:

1. How to open geometry module in ansys workbench file?

2. Can we open cad model in ansys geometry directly.

3. On what machines ansys workbench 19.0 should be run.

4. Where can we use ansys 14.0 software in industries.

5. How to add noise to an image in ansys workbench 14.0?

6. Tell about ansys workbench application program software.

7. How do you make use of ansys workbench function library?

8. Categorize the ansys workbench menus used in various steps.

9. What type of programming language in ansys workbench 19.0 is used.

10. Where ansys workbench can be applicable.

S.NO.	QUESTION	LEVEL
1.	From the given sketch modify the boundaries and calculate deflections and stresses.	L5
2.	Conclude the maximum stress and deflection from the given sketch.	L6
3.	Analyze the given sketch and generate results using ansys software.	L4
4.	From the given sketch how do you make use of ansys to determine stresses and deflection.	L3

EXERCISE – 7

Harmonic analysis

Objective:

- To perform harmonic analysis for the given beam.

Out comes:

- Understand how to use ansys tools to analyse the given beam.
- Understand the steps and menus used in ansys in various steps.

Scope of the experiment:

To provide an introduction to the use of ansys software in performing harmonic analysis on the given beam using full method.

Description of software:

The software used for doing this exercise is ansys apdl and it is developed by Ansys, Inc. Regardless of the type of simulation, each model is represented by a powerful scripting language … the Ansys Parametric Design Language (APDL). APDL is the foundation for all sophisticated features, many of which are not exposed in the Workbench Mechanical user interface. It also offers many conveniences such as parameterization, macros, branching and looping, and complex math operations. All these benefits are accessible within the Ansys Mechanical APDL user interface.
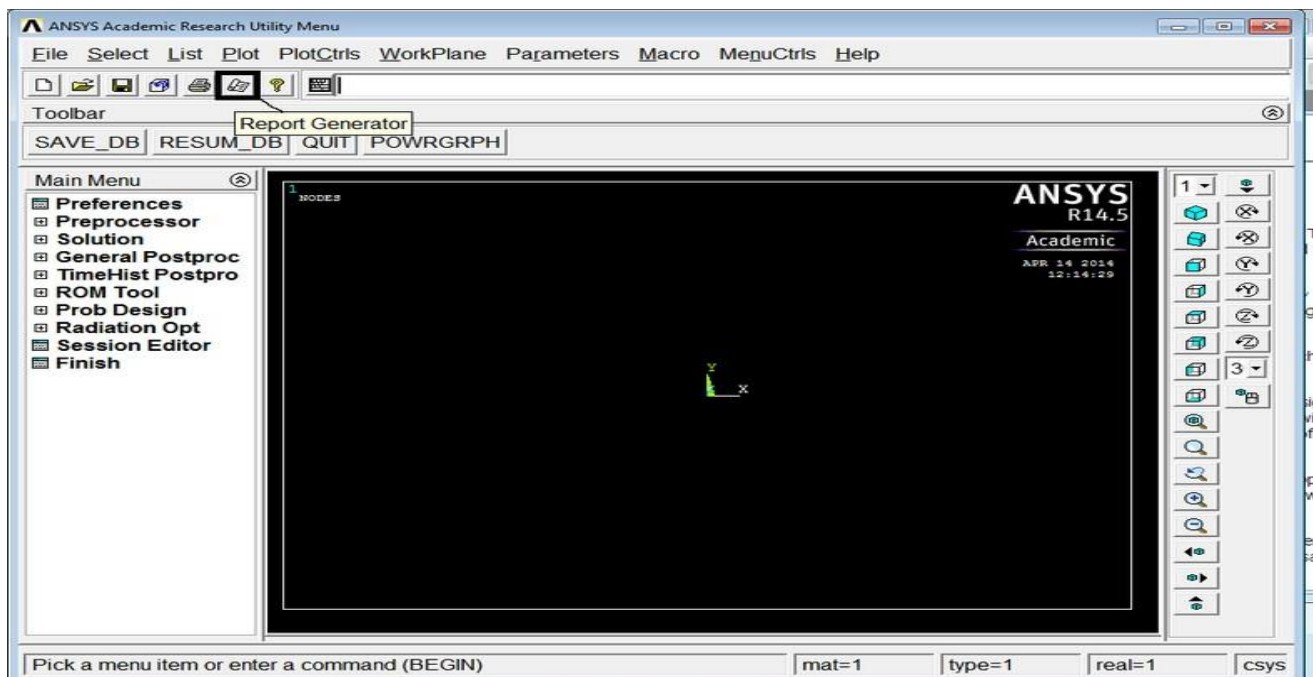
STRESS AND DEFLECTION ANALYSIS IN SIMPLY SUPPORTED BEAM WITH POINT LOAD

 PROCEDURE:

1. Ansys Main Menu – Preferences-Select – STRUCTURAL- h method – ok

2. Element type – Add/Edit/Delete – Add – BEAM – 2 node BEAM 188– ok- close.

3. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX
   – 2.10e5– PRXY – 0.27 – ok – close.

4. Sections-Beams-common sections- sub type- rectangle (1st element) -enter b=100, h=100-preview-ok.

5. Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS– 1000 (x value w.r.t first node) – apply (second node is created) – 2500 (x value w.r.t first node) – apply(third node is created)- x,y,z location in CS-3500 (x value w.r.t first node)-ok.

6. Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 apply – pick 2 & 3 apply – pick 3 & 4 – ok (elements are created through nodes).

7. Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 & 4 – apply –DOFs to be constrained – all DOF – ok.

8. Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply – direction of For/Mom – FY – Force/Moment value – -2000(-ve value) – ok-Force/Moment –

on Nodes- pick node 3 – apply –direction of For/Mom – FY – Force/Moment value – -4000(-ve value) – ok.

9. Solve – current LS – ok (Solution is done is displayed) – close.

10. Displacement: Plot Results – Contour plot – Nodal solution – DOF solution – displacement vector sum – ok.

11. Stress: Plot Results – Contour plot – Nodal solution – stress – vonmises stress – ok.

12. Element table – Define table – Add – 'Results data item' – By Sequence num – SMISC – SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 19 – apply, By Sequence num – SMISC – SMISC, 3 – apply, By Sequence num – SMISC – SMISC, 16 – ok – close.

13. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS19 – ok (Shear force diagram will be displayed).

14. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS3 – Elem table item at node J – SMIS16 – ok (bending moment diagram will be displayed).

15. Reaction forces: List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).

NOTE: For Shear Force Diagram use the combination SMISC 6 & SMISC 19, for Bending Moment Diagram use the combination SMISC 3 & SMISC 16.

16. Animation: PlotCtrls – Animate – Deformed results – DOF solution – USUM – ok.

RESULT:

Thus the stress and deflection analysis in simply supported beam with point load is done by using the ANSYS Software.
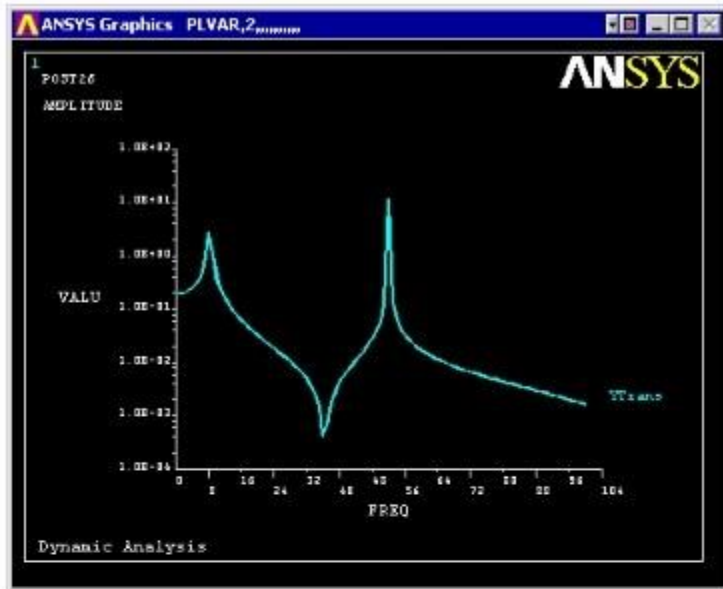
This is the response at node 2 for the cyclic load applied at this node from 0 - 100 Hz.

Conclusion: Using full method harmonic analysis is performed and results were taken on few nodes.

1. Define term node.

2. Write about element.

3. What is convergence.

4. List types of convergence used in ansys.

5. Can you tell use of geometric invariance in ansys.

6. How do we use pascal triangle in FEA.

7. Write about connectivity.

8. What are the methods to improve problem solution.

9. Define plane stress.

10. What is plane strain.

Innovative Questions:

1. Compare FEA with solid mechanics.

2. Write the packages available for FEA.

3. On what machines FEA packages should be run.

4. Where can we use model of physical system.

5. List the properties of shape function.

6. Tell about different coordinates involved in chain rule.

7. How do you make use of galerkin method in FEA?

8. Categorize the ansys procedure in doing FEA.

9. What type of programming language is NASTRAN.

10. Where weighted residual methods can be applicable.

S.NO.	QUESTION	LEVEL
1.	From the given sketch modify the boundaries and perform harmonic analysis.	L5
2.	Conclude the amplitude and frequency from the given beam.	L6
3.	Analyze the given sketch and generate results using ansys software.	L4
4.	From the given sketch how do you make use of ansys to perform harmonic analysis.	L3

EXERCISE – 8

Modal analysis

Objective:

- To perform modal analysis for the given beam.

Out comes:

- Understand how to use ansys tools to analyse the given beam.
- Understand the steps and menus used in ansys in various steps.

Scope of the experiment:

To provide an introduction to the use of ansys software in performing modal analysis on the given beam using full method.

Description of software:

The software used for doing this exercise is ansys apdl and it is developed by Ansys, Inc. Regardless of the type of simulation, each model is represented by a powerful scripting language … the Ansys Parametric Design Language (APDL). APDL is the foundation for all sophisticated features, many of which are not exposed in the Workbench Mechanical user interface. It also offers many conveniences such as parameterization, macros, branching and looping, and complex math operations. All these benefits are accessible within the Ansys Mechanical APDL user interface.
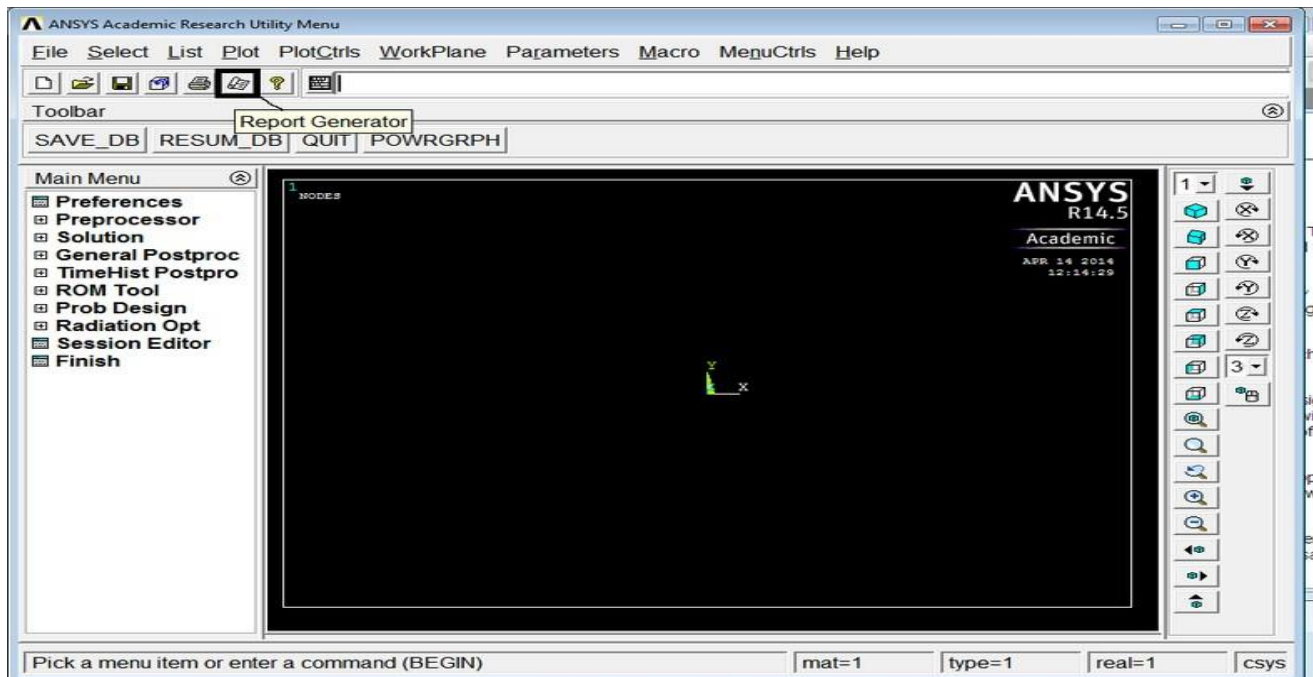
MODEL ANALYSIS OF CANTILEVER BEAM WITHOUT LOAD

AIM:

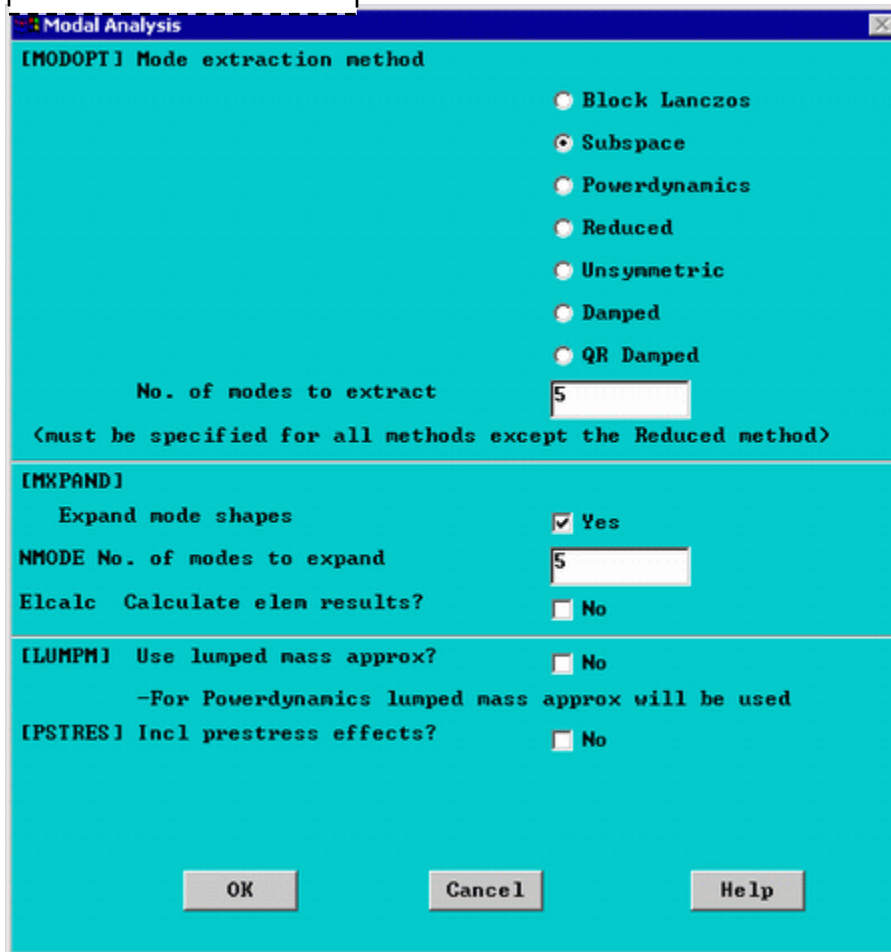To perform a Model Analysis of Cantilever beam without load using analysis software ANSYS.
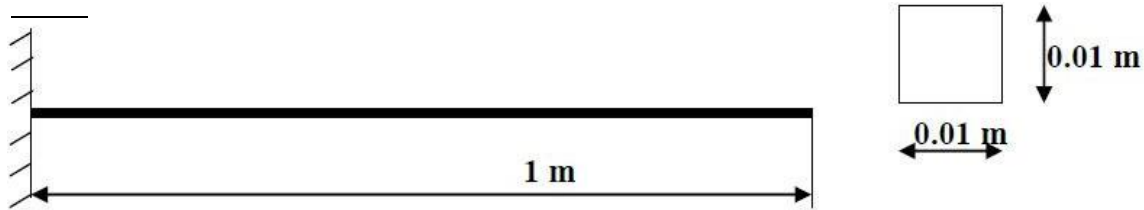
PROBLEM DESCRIPTION:

Model Analysis of Cantilever beam for natural frequency determination.

Modulus of elasticity = 200GPa, Density = 7800 Kg/m3.

○ As shown, select the **Subspace** method and enter 5 in the 'No. of modes to extract'

○ Check the box beside 'Expand mode shapes' and enter 5 in the 'No. of modes to expand'

○ Click 'OK'

  Note that the default mode extraction method chosen is the **Reduced Method**. This is the fastest method as it reduces the system matrices to only consider the Master Degrees of Freedom (see below). The **Subspace Method** extracts modes for all DOF's. It is therefore more exact but, it also takes longer to compute (especially when the complex geometries).

○ The following window will then appear
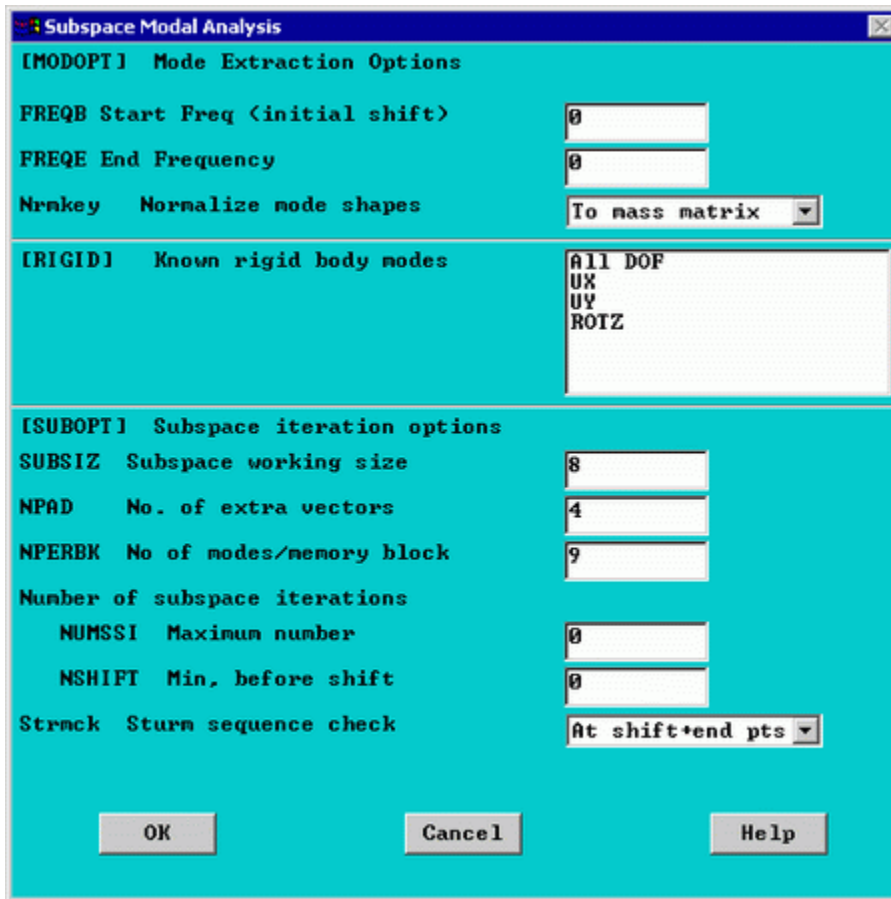
For a better understanding of these options see the *Commands* manual.

o For this problem, we will use the default options so click on OK.

3. **Apply Constraints**
   Solution > Define Loads > Apply > Structural > Displacement > On Keypoints

   Fix Keypoint 1 (ie all DOFs constrained).

4. **Solve the System**
   Solution > Solve > Current LS
   ```
   SOLVE
   ```

Postprocessing: Viewing the Results

1. **Verify extracted modes against theoretical predictions**

   o Select: **General Postproc > Results Summary...**

   The following window will appear

```
A SET,LIST Command                                    x

File

  xxxxx   INDEX OF DATA SETS ON RESULTS FILE   xxxxx

  SET   TIME/FREQ   LOAD STEP   SUBSTEP   CUMULATIVE
   1   8.3000           1          1          1
   2   52.011           1          2          2
   3   145.64           1          3          3
   4   285.51           1          4          4
   5   472.54           1          5          5
```
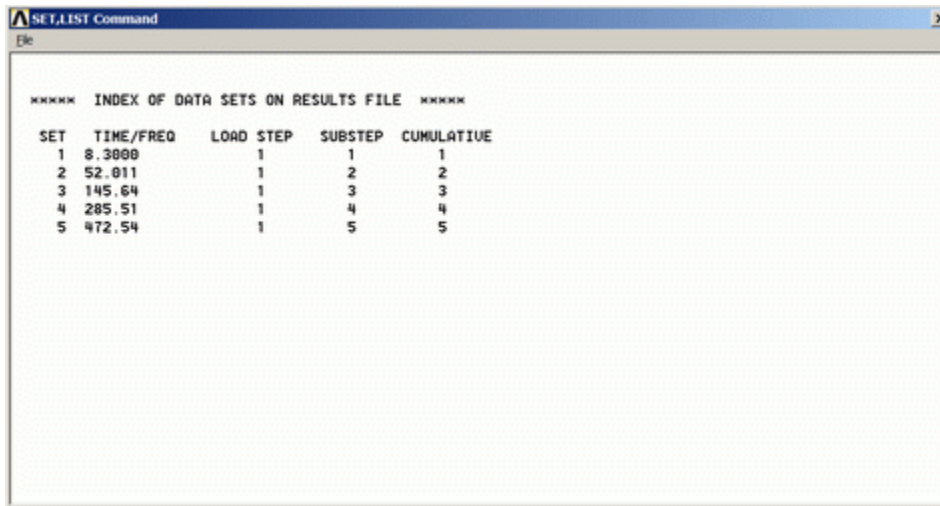
The following table compares the mode frequencies in Hz predicted by theory and ANSYS.

Mode	Theory	ANSYS	Percent Error
1	8.311	8.300	0.1
2	51.94	52.01	0.2
3	145.68	145.64	0.0
4	285.69	285.51	0.0
5	472.22	472.54	0.1

Note: To obtain accurate higher mode frequencies, this mesh would have to be refined even more (i.e. instead of 10 elements, we would have to model the cantilever using 15 or more elements depending upon the highest mode frequency of interest).

2. **View Mode Shapes**

○ Select: **General Postproc > Read Results > First Set**

This selects the results for the first mode shape

○ Select **General Postproc > Plot Results > Deformed shape** . Select 'Def + undef edge'

The first mode shape will now appear in the graphics window.

○ To view the next mode shape, select **General Postproc > Read Results > Next Set** . As above choose **General Postproc > Plot Results > Deformed shape** . Select 'Def + undef edge'.

○ The first four mode shapes should look like the following:
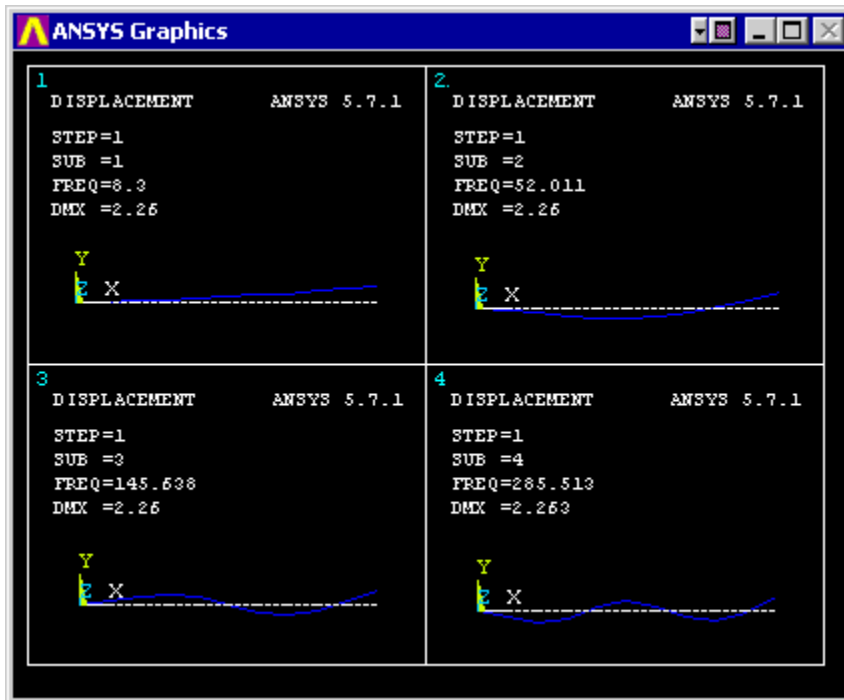
3. **Animate Mode Shapes**

   ○ Select **Utility Menu (Menu at the top) > Plot Ctrls > Animate > Mode Shape**

   The following window will appear
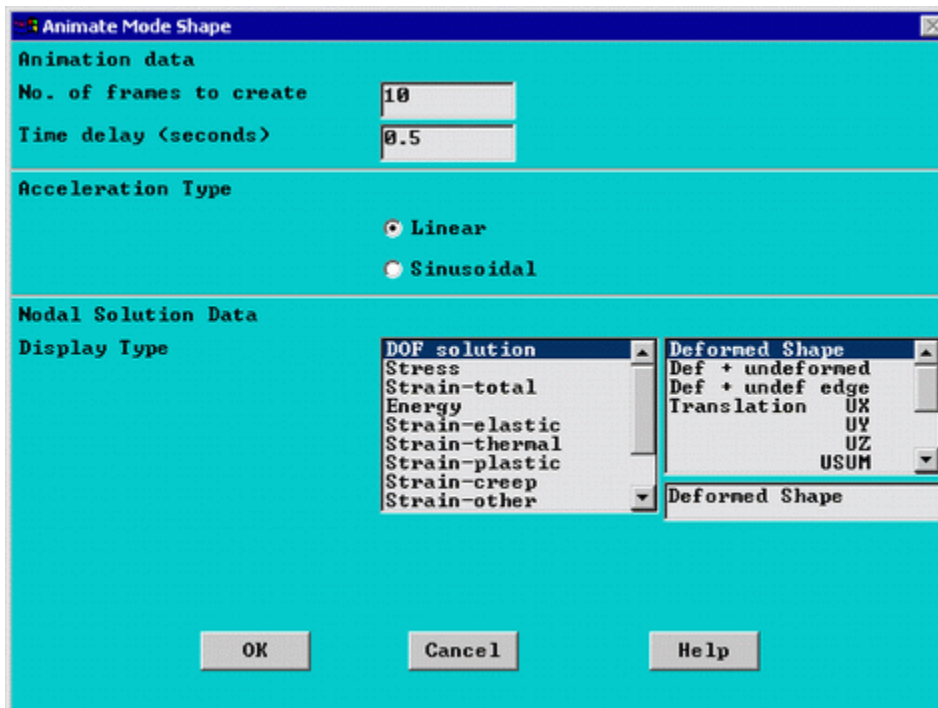
   ○ Keep the default setting and click 'OK'
   ○ The animated mode shapes are shown below.

- Mode 1
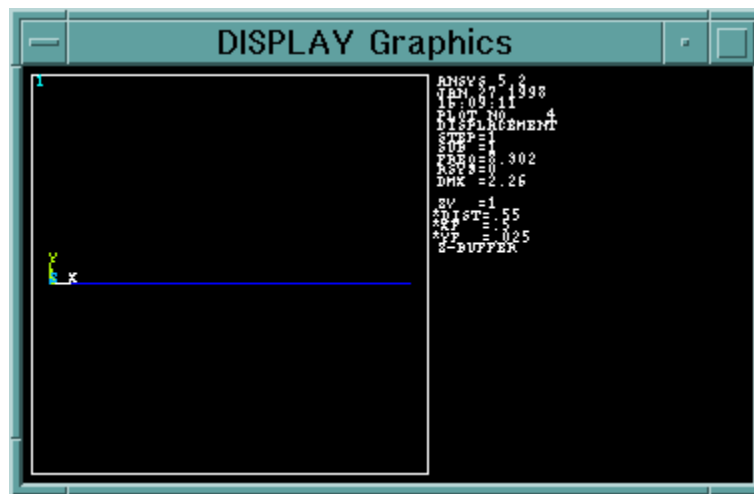
- Mode 2
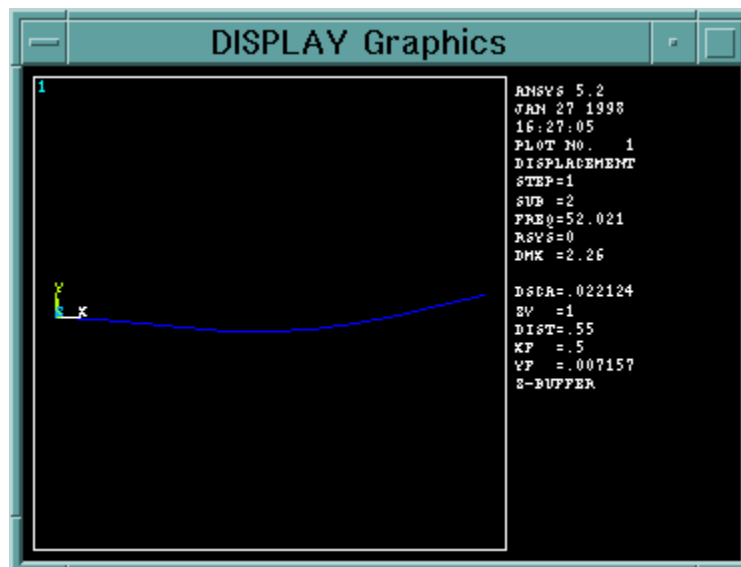
- Mode 3
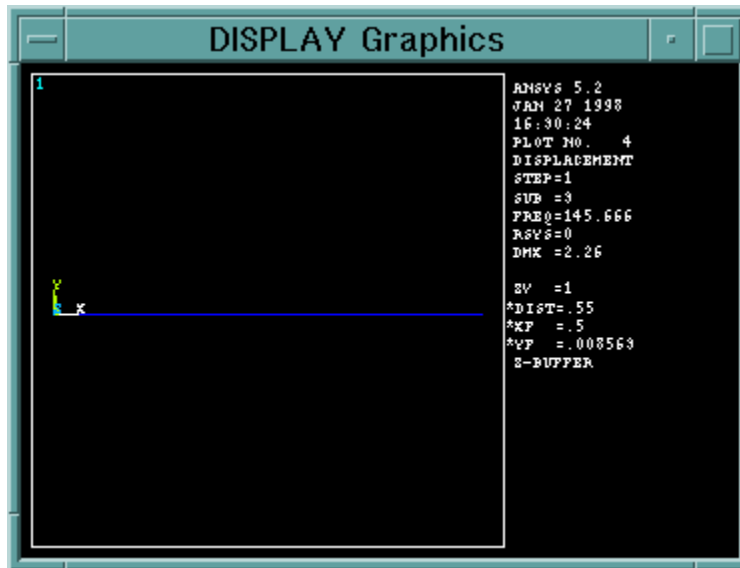
- Mode 4
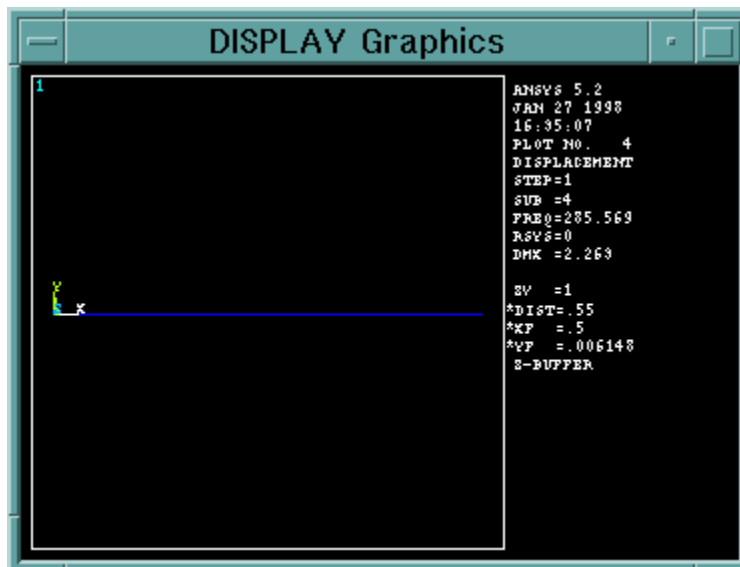
Conclusion: Mode shapes were generated for the given using modal analysis.

1. Define term element.

2. Write about node.

3. What is beam.

4. List types of beams used in ansys.

5. Can you tell use of beam analysis in ansys.

6. How do we use modal analysis in analyzing beams.

7. Write about cantilever beam.

8. What are the different loads acting on beam.

9. Define truss.

10. What is simply supported beam.

Innovative Questions:

1. Compare FEA with experimental method.

2. Write the packages available for FEA.

3. On what processors FEA packages should be run.

4. Where can we use modal analysis results of beam.

5. List the types of shape function.

6. Tell about principal of virtual work.

7. How do you make use of virtual displacement in FEA?

8. Categorize the advantages in doing FEA.

9. What type of programming language is IDEAS.

10. Specify the terms required to solve FEA problem.

Blooms Taxonomy questions for mid/End exam.

S.NO.	QUESTION	LEVEL
1.	From the given sketch modify the boundaries and perform modal analysis.	L5
2.	Determine the number of mode shapes from the given beam.	L6
3.	Analyze the given sketch and generate results using ansys software.	L4
4.	From the given sketch how do you make use of ansys to perform modal analysis.	L3

EXERCISE – 9

Thermal analysis

Objective:

- To perform heat transfer analysis for the given object.

Out comes:

- Understand how to use ansys tools to analyse the heat transfer analysis.
- Understand the steps and menus used in ansys in various steps.

Scope of the experiment:

To provide an introduction to the use of ansys software in performing thermal analysis on the given object using conduction.

Description of software:

The software used for doing this exercise is ansys apdl and it is developed by Ansys, Inc. Regardless of the type of simulation, each model is represented by a powerful scripting language … the Ansys Parametric Design Language (APDL). APDL is the foundation for all sophisticated features, many of which are not exposed in the Workbench Mechanical user interface. It also offers many conveniences such as parameterization, macros, branching and looping, and complex math operations. All these benefits are accessible within the Ansys Mechanical APDL user interface.
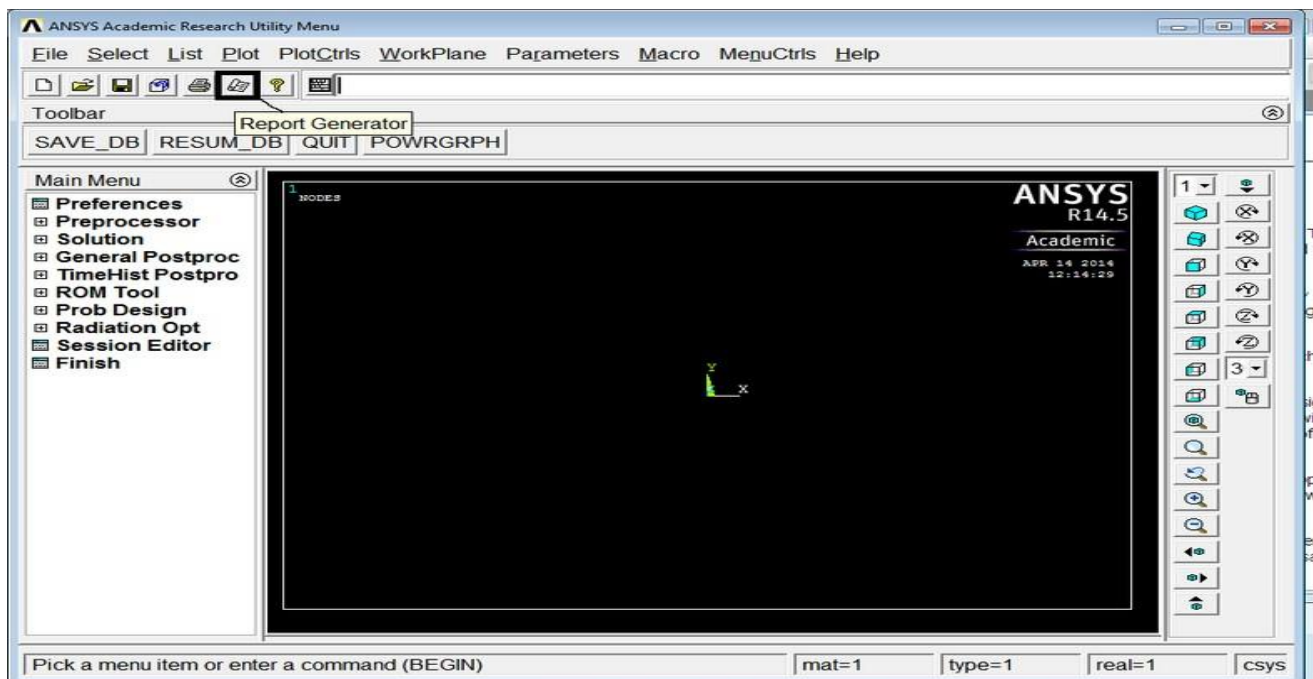
THERMAL STRESS ANALYSIS WITHIN THE RECTANGULAR PLATE

AIM:

   To perform a thermal stress analysis of a rectangular plate using analysis software ANSYS.

PROBLEM DESCRIPTION:

        2-D heat conduction problem for the temperature distribution within the rectangular plate. Thermal conductivity of the plate, KXX=401 W/(m-K).

 PROCEDURE:

1. Ansys Main Menu – Preferences-select – THERMAL- h method– ok

2. Element type – Add/Edit/Delete – Add – Solid – Quad 4 node – 55 – ok – option – elementbehavior K3 – Plane stress with thickness – ok – close.

3. Material Properties – material models – Thermal – Conductivity – Isotropic – KXX – 401.

4. Modeling – Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 0, 10, 0, 20 – ok.

5. Meshing – Mesh Tool – Mesh Areas – Quad – Free – Mesh – pick all – ok. Mesh Tool – Refine – pick all – Level of refinement – 3 – ok.

6. Loads – Define loads – apply – Thermal – Temperature – on Lines – select 1000 C lines – apply – DOFs to be constrained – TEMP – Temp value – 1000 C – ok.

7. Loads – Define loads – apply – Thermal – Temperature – on Lines – select 1000 C lines –

8. Solve – current LS – ok (Solution is done is displayed) – close.

9. Read results-last set-ok

10. List results-nodal solution-select temperature-ok

11. Observe the nodal solution per node.

12. From the menu bar-plot ctrls-style-size and shape-display of the element-click on real constant multiplier=0.2, don't change other values-ok.

13. Plot results-contour plot-nodal solution-temperature-deformed shape only-ok

14. Element table-define table-add-enter user label item=HTRANS, select by sequence no SMISC, 1-ok-close.

15. Element table-list table-select HTRANS-ok

conclusion

Thus the thermal stress analysis of a rectangular plate is done by using the ANSYS Software.

1. Define conduction.

2. Write about convection.

3. What are types of shape function.

4. List types of elastic constants used in ansys.

5. Can you tell use of young's modulus and poisson's ratio in ansys.

6. How do we use thermal analysis in heat transfer applications.

7. Write about radiation.

8. What are the different modes of heat transfer.

9. Define heat.

10. What is Stefan boltzman constant.

Innovative Questions:

1. Compare FEA with therotical method.

2. Write the packages available for FEA thermal analysis.

3. On what processors thermal analysis should be run.

4. Where can we use thermal analysis results in heat transfer applications.

5. List the types of thermal analysis.

6. Tell about transient thermal analysis.

7. How do you make use of virtual displacement in FEA?

8. Categorize the advantages of thermal analysis in doing fluent.

9. What type of programming language is fluent.

10. Specify the terms required to solve thermal problems.

Blooms Taxonomy questions for mid/End exam.

| S.NO.	QUESTION	LEVEL
1.	From the given object apply the boundaries and perform thermal analysis.	L5
2.	Write your opinion in conducting thermal analysis in convection mode.	L6
3.	Analyze the given sketch and generate results using ansys software.	L4
4.	From the given sketch how do you make use of ansys to perform thermal analysis.	L3

3. CAM

Introduction

The term numerical control is a widely accepted and commonly used term in the machine tool industry. Numerical control (NC) enables an operator to communicate with machine tools through a series of numbers and symbols. NC which quickly became Computer Numerical Control (CNC) has brought tremendous changes to the metalworking industry. New machine tools in CNC have enabled industry to consistently produce parts to accuracies undreamed of only a few years ago. The same part can be reproduced to the same degree of accuracy any number of times if the CNC program has been properly pre- pared and the computer properly programmed. The operating commands which control the machine tool are executed automati- cally with amazing speed, accuracy, efficiency, and repeatability.

The ever-increasing use of CNC in industry has created a need for personnel who are knowledgeable about and capable of preparing the programs which guide the machine tools to produce parts to the required shape and accuracy. With this in mind, the authors have prepared this textbook to take the mystery out of CNC - toput it into a logical sequence and express it in simple language that everyone can understand. The preparation of a program is explained in a logical step-by-step procedure, with practical ex- amples to guide the student. Almost everything that can be produced on a conventional ma- chine tool can be produced on a computer numerical control machine tool, with its many advantages. The machine tool move- ments used in producing a product are of two basic types: point- to-point (straight-line movements) and continuous path (contouring movements).

The Cartesian, or rectangular, coordinate system was devised by the French mathematician and philosopher Rene' Descartes. With this system, any specific point can be described in mathematical terms from any other point along three perpendicular axes. This concept fits machine tools perfectly since their construction is generally based on three axes of motion (X, Y, Z) plus an axis of rotation. On a plain vertical milling machine, the X axis is the horizontal movement (right or left) of the table, the Y axis is the table cross movement (toward or away from the column), and the Z axis is the vertical movement of the knee or the spindle. CNC systems rely heavily on the use of rectangular coordinates be- cause the programmer can locate every point on a job precisely.
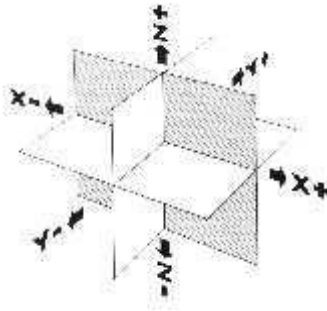
Block of Information:

CNC information is generally programmed in blocks of five words. Each word conforms to the EIA standards and they are written on a horizontal line. If five complete words are not included in each block, the machine control unit (MCU) will not recognize the information, therefore the control unit will not be activated.

Machines Using CNC

Early machine tools were designed so that the operator was standing in front of the machine while operating the controls. This design is no longer necessary, since in CNC the operator nolonger controls the machine tool movements. On conventional machine tools, only about 20 percent of the time was spent remov- ing material. With the addition of electronic controls, actual time spent removing metal has increased to 80 percent and even higher. It has also reduced the amount of time required to bringthe cutting tool into each machining position.

Machine types

Lathe

The engine lathe, one of the most productive machine tools, has always been an efficient means of producing round parts. Most lathes are programmed on two axes.
The X axis controls the cross motion of the cutting tool
Negative X (X-) moves the tool towards the spindle centerline; positive X moves the tool away from the spindle centerline.
The Z axis controls the carriage travel toward or away from the headstock.
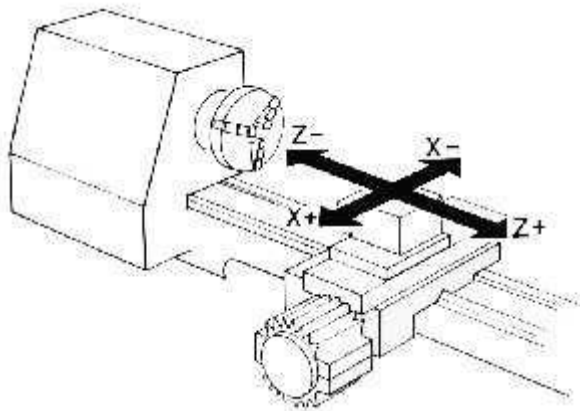
Milling Machine

The milling machine has always been one of the most versatile machine tools used in industry Operations such as milling, contouring, gear cutting, drilling, boring, and reaming are only a few of the many operations which can be performed on a milling machine. The milling machine can be programmed on three axes:

The X axis controls the table movement left or right.
The Y axis controls the table movement toward or away from the column.
The Z axis controls the vertical (up or down) movement of the knee or spindle.

Z AXIS
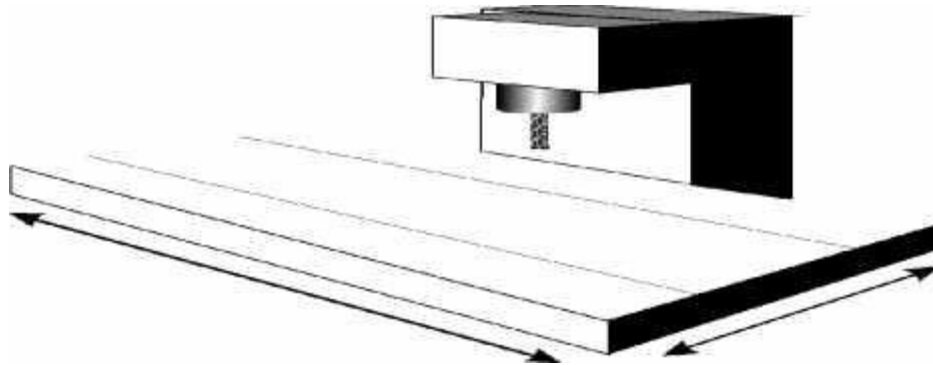
Y AXIS

X AXIS

Programming Systems

Two types of programming modes, the incremental system and the absolute system, are used for CNC. Both systems have applications in CNC programming, and no system is either right or wrong all the time. Most controls on machine tools today are capable of handling either incremental or absolute programming.

Incremental program locations are always given as the distance and direction from the immediately preceding point. Command codes which tell the machine to move the table, spindle, and knee are explained here using a vertical milling machine. as an example:
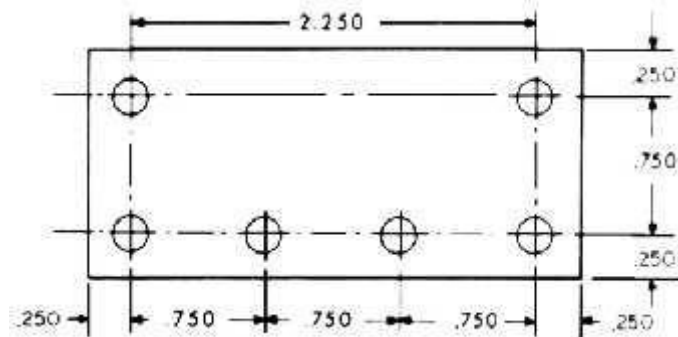
A "X plus" (X+) command will cause the cutting tool to be located to the right of the last point.
A "X plus" (X-) command will cause the cutting tool to be located to the right of the last point.
A "Y plus" (Y+) command will cause the cutting tool to be located toward the column.
A "Y minus" (Y-) will cause the cutting tool to be located away from the column.
A "Z plus" (Z+) command will cause the cutting tool or spindle to move up or away from the workpiece.

•A "Z minus" (Z-) moves the cutting tool down or into the work- piece.
In incremental programming, the G91 command indicates to the computer and MCU (Machine Control Unit) that programming is in the incremental mode.

Absolute program locations are always given from a single fixed zero or origin point (Fig. . The zero or origin point may be a position on the machine table, such as the corner of the worktable or at any specific point on the workpiece. In absolute dimensioning and programming, each point or location on the workpiece is given as a certain distance from the zero or reference point.
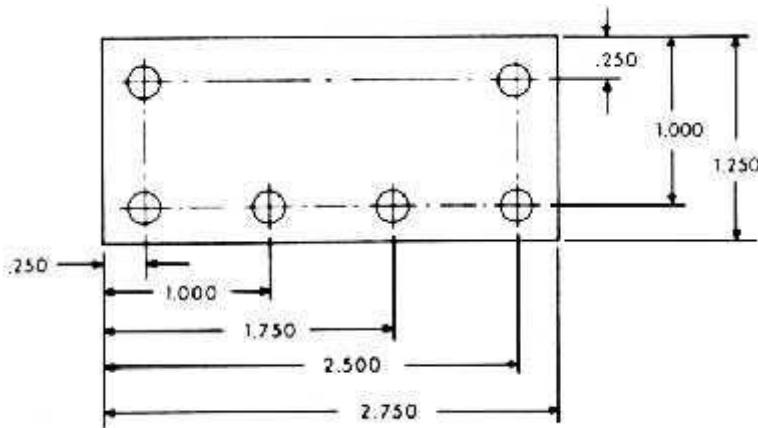
A "X plus" (X+) command will cause the cutting tool to be located to the right of the
    zero or origin point.
•   A "X minus" (X-) command will cause the cutting tool to be located to the left of the
    zero or origin point.
•   A "Y plus" (Y+) command will cause the cutting tool to be located toward the
    column.
•   A "Y minus" (Y-) command will cause the cutting tool to be located away from the
column.

In absolute programming, the G90 command indicates to the computer and MCU that the programming is in the absolute mode.

EXERCISE – 10

Development of NC code

Objective:

- To develop a NC code for given part manually and check using cam software.

Out comes:

- Understand how to write CNC code for basic process like turning, milling.
- Understand the steps and codes used to write a program.

Scope of the experiment:

To provide an introduction to develop NC code for given part manually.

Description of software:

The software used for doing this exercise is fusion 360 and it is developed by Autodesk. Fusion 360 is a cloud-based CAD/CAM tool for collaborative product development. Fusion 360 enables exploration and iteration on product ideas and collaboration within distributed product development team. Fusion 360 combines organic shapes modelling, mechanical design and manufacturing in one comprehensive package.
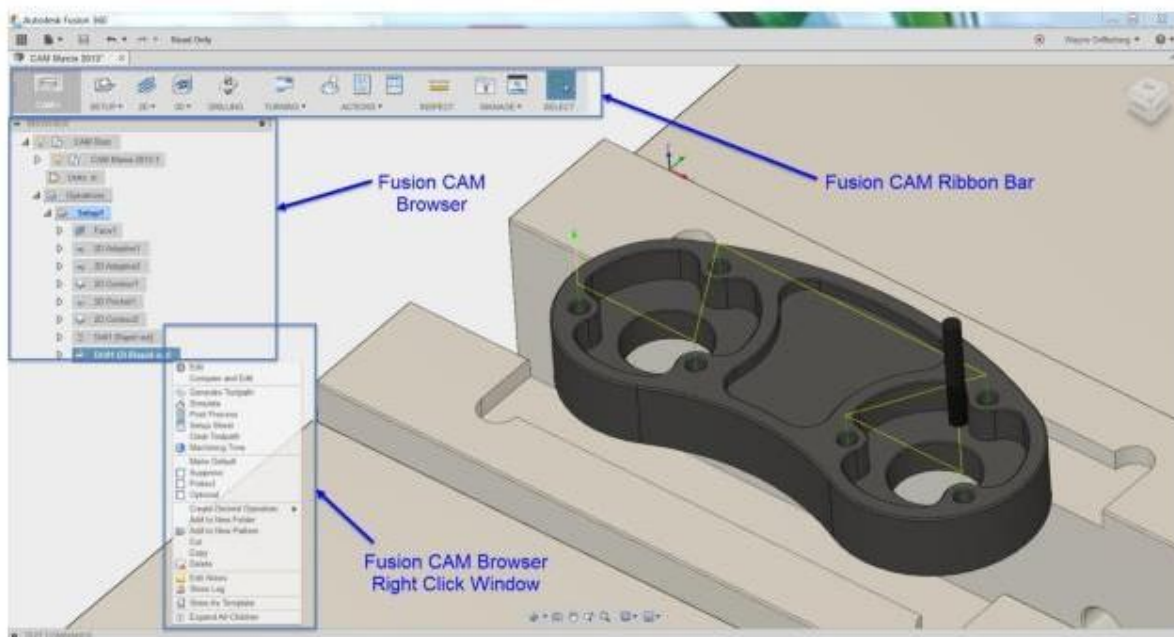
CNC Programming
Cylindrical Part

Raw Material — 70, 25

Finished Part — 17.5, 22.5, 20, 30
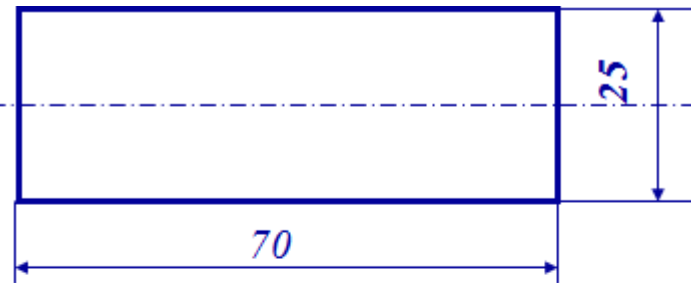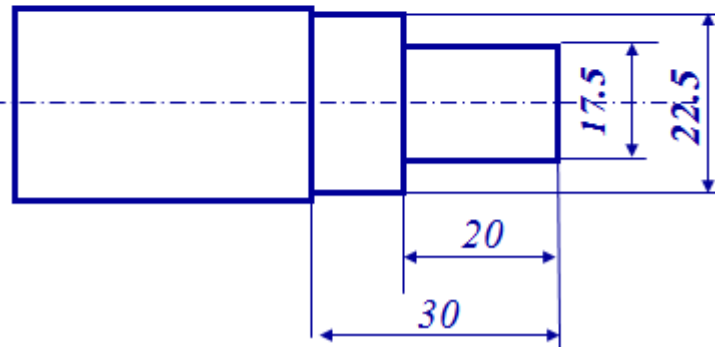
N0005 G53	**To cancel any previous working zero point**
N0010 T0404	**N0010 Sequence number**
	T0404 Select tool number 404
N0020 G57 G00 X26.0 Z0.0 S500 M04	**G57 To set the working zero point as saved**
	G00 Rapid movement (no cutting)
	X26.0 *X* location (as a diameter; 13 form zero)
	Z0.0 *Z* location
	S500 Spindle speed is 500 rpm
	M04 Rotate spindle counterclockwise

N0030 G01 X-0.20 F100	**G01 Linear interpolation (cutting)**
	X-0.20 Move only in x direction until you pass the center by 0.1 mm (facing)
	F100 Set feed rate to 100 mm/min.

N0040 G00 Z2.0	**G00 Move rapidly away from work piece (no cutting)**
	Z2.0 the movement is 2 mm away from the face.
N0050 X50.0 Z50.0	**Go to a safe location away from the workpiece [x = 50 (25 from zero), z = 50] to change the tool.**
N0060 T0404	**T0404 Select tool number 404**

N0070 G57 G00 X22.50 Z2.0 S500

G57 PS0
G00 Rapid movement (no cutting)
X22.50 *X* location (as a diameter; 11.25 form zero)
Z2.0 *Z* location
S500 Spindle speed is 500 rpm

N0080 G01 Z-30.0 F100

G01 Linear interpolation (cutting)
Z-30 Move only in z direction (external turning)
F100 Set feed rate to 100 mm/min.

N0090 G00 X23.0 Z2.0 S500

G00 Move rapidly away from work piece (no cutting) to location x= 23.0 (11.50 from zero) and z = 2.0.

N0100 G84 X17.5 Z-20.0 D0=200 D2=200 D3=650

G84 Turning cycle for machining the step
X17.5 final diameter
Z-20 length of step is 20 mm
D0=200 Finish allowance in X direction (0.2 mm) D2=200 Finish allowance in Z direction (0.2 mm)
D3=650 Depth of cut in each pass (0.65 mm)

N0110 G00 Z2.0

G00 Move rapidly away from workpiece (no cutting)
Z2.0 the movement is 2 mm away from the face.

N0120 X50.0 Z50.0

X50.0 Z50.0 Move to the tool changing location

N0130 M30

M30 Program End

.

Additional Viva Questions:

1. How to import data in fusion 360 from an external file.

2. Write the function of import data forms used in fusion 360.

3. What are the 2D turning operations available in fusion 360.

4. List the 2D milling operations available in fusion 360.

5. Can you tell how to import CAD data into cam module with an example.

6. How to export data from fusion 360 to NC machine.

7. Write the procedure to export NC code from fusion 360.

8. What are the steps required to do simple turning in fusion 360.

9. How to do simulation in fusion 360.

10. Categorize the tool paths provided in fusion 360.

Innovative Questions:

1. How to call cam module in fusion 360?

2. Can we open igs file in fusion 360 if so how ?

3. On what machines fusion 360 should be run.

4. Where can we use fusion 360 for?

5. How to add noise to a simulation in fusion 360?

6. Tell about fusion 360 applications.

7. How do you make use of fusion 360 in manufacturing industry.

8. Categorize the cam process used in fusion 360.

9. What type of programming language is used in fusion 360.

10. Where fusion 360 software can be applicable

S.NO.	QUESTION	LEVEL
1.	Using fusion 360 software generate a NC code for the given part.	L5
2.	Create a NC code for the given part using cam software.	L6
3.	List the cam procedure used in fusion 360 to generate NC code for given part.	L4
4.	For the given part how do you make use of fusion 360 for doing cam.	L3

EXERCISE – 11

CNC PROGRAMMING using Fusion 360

Objective:

- To develop a CNC code for the given simple part.

Out comes:

- Understand how to write CNC code for basic process like turning, milling.
- Understand the steps and codes used to write a program.

Scope of the experiment:

To provide an introduction to the use of cam software in developing NC code for given part

Description of software:

The software used for doing this exercise is fusion 360 and it is developed by Autodesk. Fusion 360 is a cloud-based CAD/CAM tool for collaborative product development. Fusion 360 enables exploration and iteration on product ideas and collaboration within distributed product development team. Fusion 360 combines organic shapes modelling, mechanical design and manufacturing in one comprehensive package.
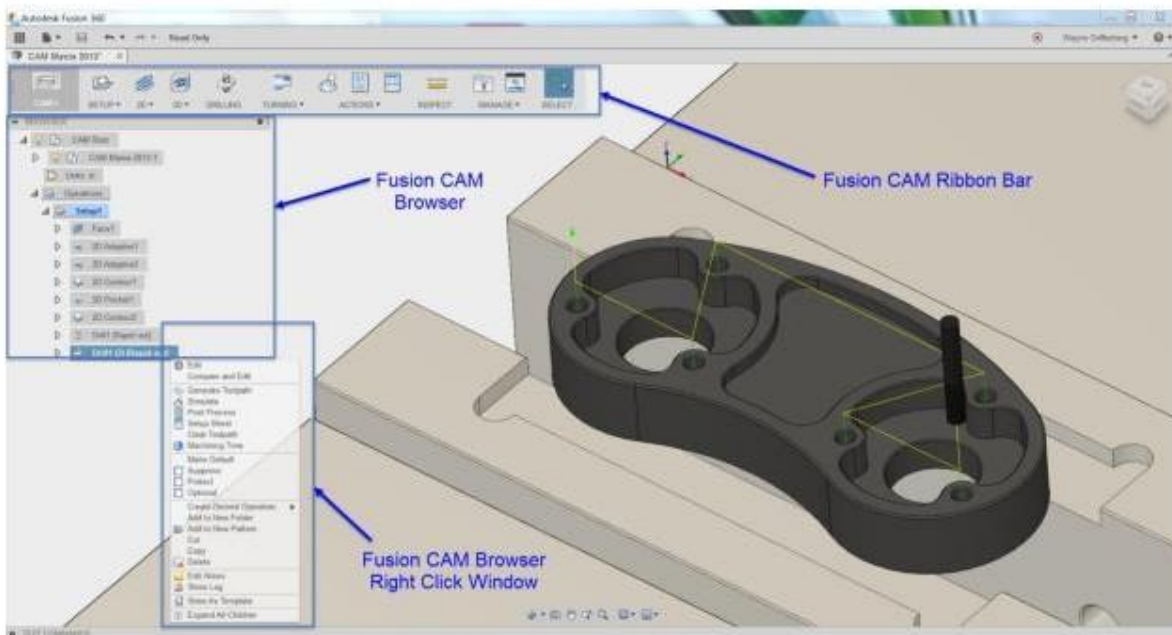
Procedure :

CAD Modeling in Autodesk Fusion 360

1.	Start Autodesk Fusion 360, a new file *'Untitled'* will be created automatically. Save it in a project folder and give it a describing name. Create a new sketch and select the plane to draw it in.	
2.	Draw the 2D geometry of the part. Use the drawing tools in the 'SKETCH' menu. When the sketch is done, return to 3D view by clicking 'STOP SKETCH'.	
3.	Use the tools in the 'MODIFY' menu to pull the sketch into a 3D geometry and modify it. Press Pull = Raise 2D into 3D Chamfer = Create a chamfered edge Fillet = Create an edge with a radius	
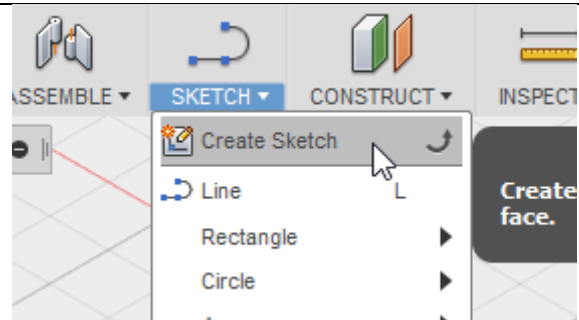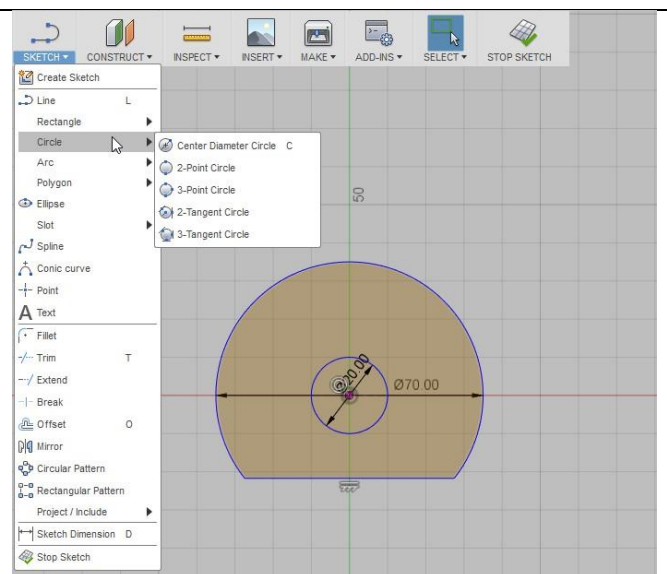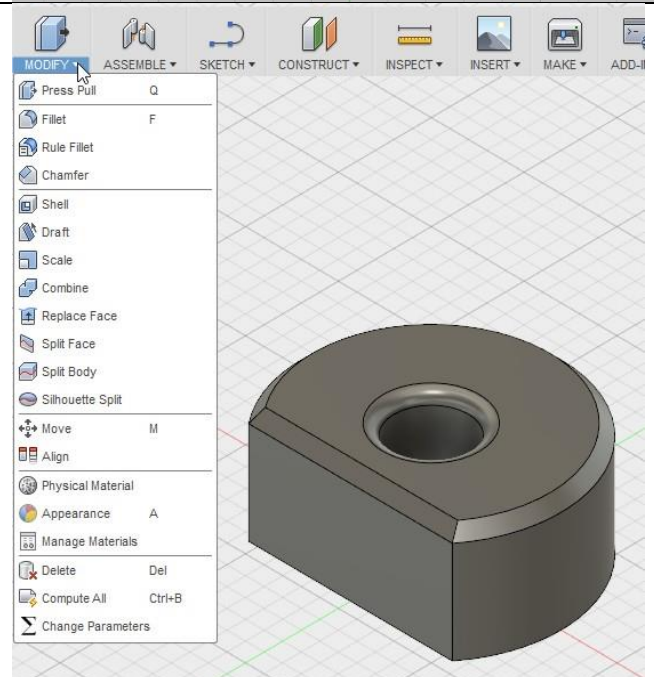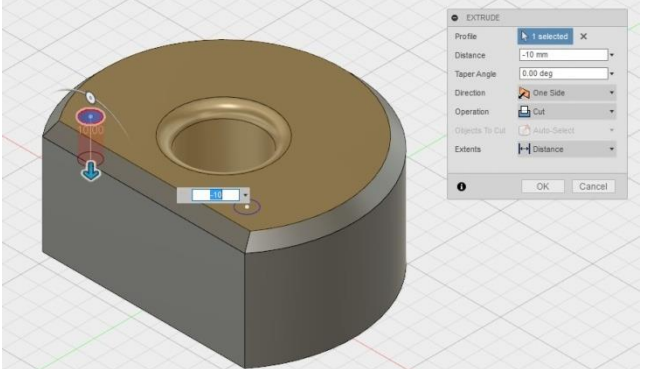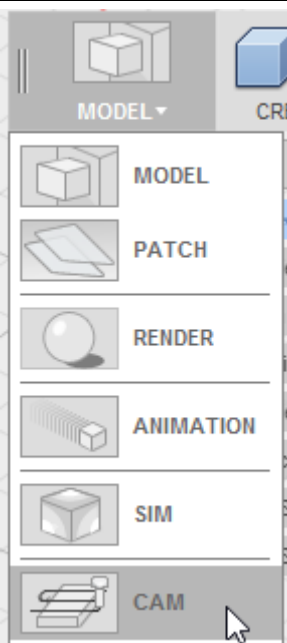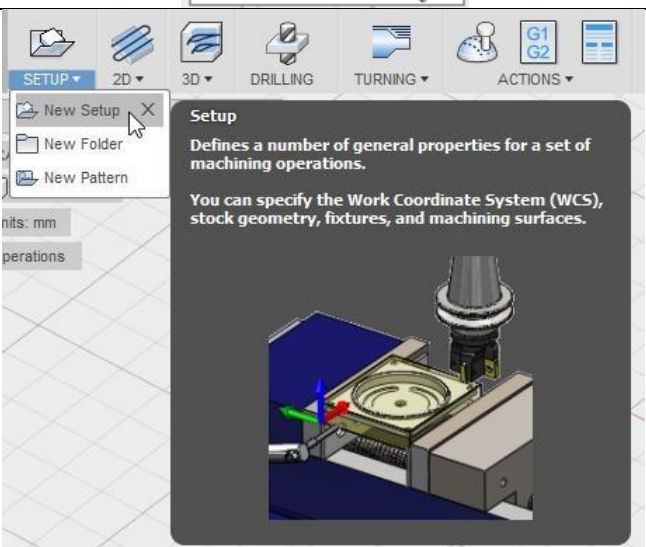
4.	Create additional sketches if necessary. Modify them into 3D geometry.	

CAM Process in Autodesk Fusion 360	

1.	Select CAM mode.	
2.	Create a new setup. Select the origin point under the 'Setup' tab. Specify the stock material shape and size under the 'Stock' tab.	

3.	Create the necessary cutting operations. Choose the cutting operation from the 2D menu. Select the cutting tool to be used. Fill in the Spindle Speed, Ramp Spindle Speed, Cutting Feedrate, Lead-In Feedrate, Lead-Out Feedrate, Ramp Feedrate and Plunge Feedrate according to the Speeds and Feeds Table. Select the geometry to be cut. Select the heights of the depth of the cut. Select the depth per pass under 'Passes' tab. **Keep in mind how the part should be fixtured / clamped to the table. It is common to machine it halfway and then flip it around in order to machine the second half.**	

2D CONTOUR : 2D CONTOUR1

Tool | Geometry | Heights | Passes | Linking

▼ Tool

Tool — Select...
#1 - Ø8 mm flat

Coolant — Flood

▼ Feed & Speed

Spindle Speed	1200
Surface Speed	30.1593 m/min
Ramp Spindle Speed	1200 rpm
Cutting Feedrate	190
Feed per Tooth	0.0791667 mm
Lead-In Feedrate	190
Lead-Out Feedrate	190
Ramp Feedrate	190
Plunge Feedrate	190
Feed per Revolution	0.158333 mm

OK   Cancel
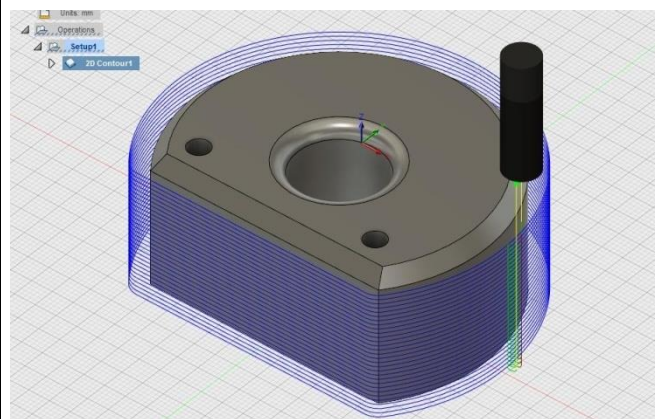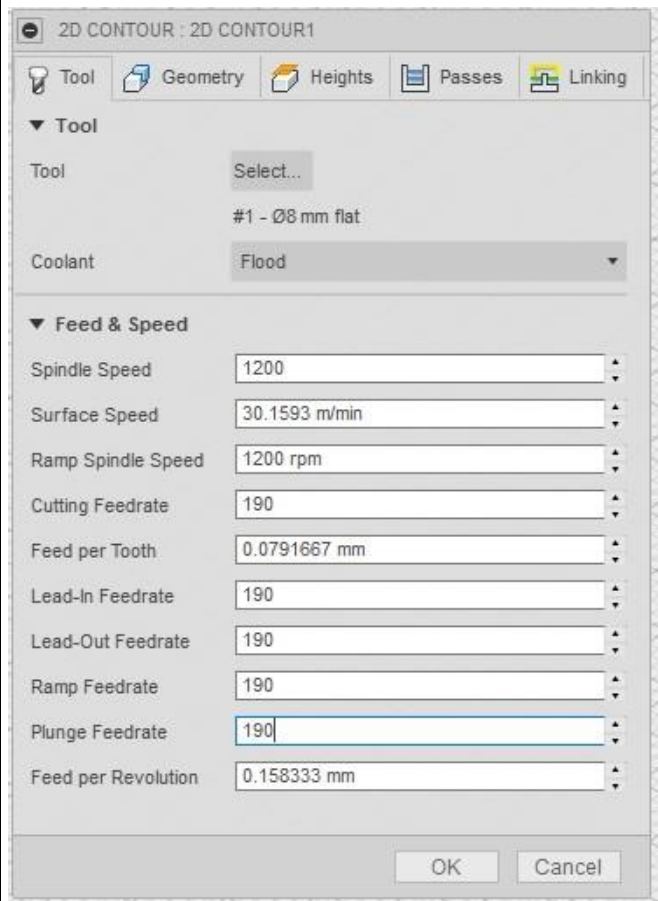
4.	Simulate to make sure that the toolpaths are correct.	
5.	Generate the G -Code by selecting 'Post Process' Select 'mach3mill.cps' under 'Post Configuration' Give the program a name under 'Program name or number' Hit OK , save the file and head to the CNC Mill!	
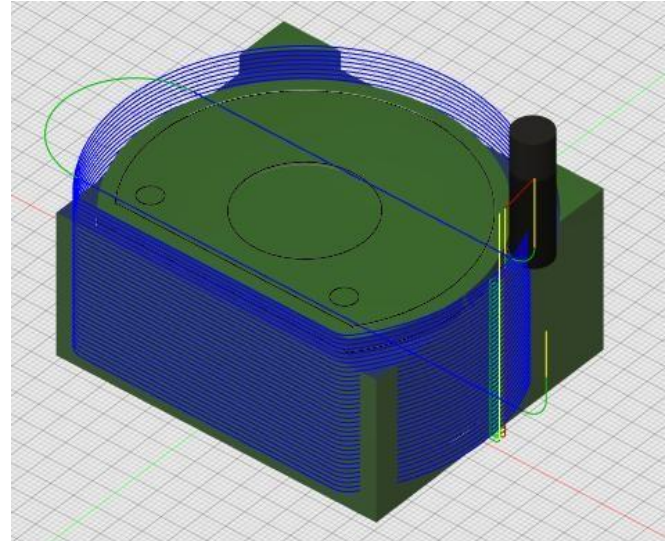
.

Additional Viva Questions:

1. How to import data in fusion 360 from an external file.

2. Write the function of import data forms used in fusion 360.

3. What are the 2D turning operations available in fusion 360.

4. List the 2D milling operations available in fusion 360.

5. Can you tell how to import CAD data into cam module with an example.

6. How to export data from fusion 360 to NC machine.

7. Write the procedure to export NC code from fusion 360.

8. What are the steps required to do simple turning in fusion 360.

9. How to do simulation in fusion 360.

10. Categorize the tool paths provided in fusion 360.

Innovative Questions:

1. How to call cam module in fusion 360?

2. Can we open igs file in fusion 360 if so how ?

3. On what machines fusion 360 should be run.

4. Where can we use fusion 360 for?

5. How to add noise to a simulation in fusion 360?

6. Tell about fusion 360 applications.

7. How do you make use of fusion 360 in manufacturing industry.

8. Categorize the cam process used in fusion 360.

9. What type of programming language is used in fusion 360.

10. Where fusion 360 software can be applicable

S.NO.	QUESTION	LEVEL
1.	Using fusion 360 software generate a NC code for the given part.	L5
2.	Create a NC code for the given part using cam software.	L6
3.	List the cam procedure used in fusion 360 to generate NC code for given part.	L4
4.	For the given part how do you make use of fusion 360 for doing cam.	L3

COMPUTER AIDED MODELING AND ANALYSIS LAB
VIVA QUESTIONS

1. What is the total degree of freedom of Ansys commercial package.
2. What are the different menus in ANSYS.
3. What is Work space and Swap space.
4. What is default value of worksp1ace and Swapspace?
5. What file format ANSYS can support
6. What is the current ANSYS version
7. What is optimization in ANSYS.
8. What is sub structuring and adaptive meshing.
9. Which package is better – ANSYS or NASTRAN.
10. What is Resume in ANSYS.
11. What is P-method and H-method
12. What is scalar parameters in ANSYS.
13. What are primary nodes and Secondary nodes.
14. What is Mirror (Reflection).
15. What is subroutines.

VIVA QUESTIONS ON FEM

What are the different approximate solution methods.
Different approximate solution methods are
1. Functional approximation
2. Finite Difference method
3. Finite Element method

What do you mean by continuum.
Structure which is considered for analysis is called continuum.

Define term node
The element which is connected with another element at junction is called node.

Define term element
Descritised structure is called an element .

What is convergence
Process of achieving value to actual solution.

What are the types convergence
p-convergence
h-convergence

What is p- convergence
Convergence by increasing the
elements

What is h convergence
Convergence by increasing
nodes

What is higher order elements
The element which contain more no. of nodes are called higher order element .

Give example for higher order elements
Higher order elements are CST, Quadrilateral element.

What do you mean by compatible elements.
Elements, which are compatible with adjacent element, like no discontinuity, overlap or sudden slope.

What is geometric invariance.
The property in which the shape of the element will not change with change in local coordinates is called geometric invariance.

Why do we use Pascal's triangle in FEA
If the displacement equation doesn't contain all the required terms then balancing is done by the Pascal's triangle.

What are the steps involved in FEA
Steps involved in FEA:
1.Modelling
2.Descritization of structure
3.Derivation of elemental stiffness matrix.
4.Assembly of elemental equation
5.Applying boundary conditions
6.Computation of stress and strain
7.Interpretaion of results

what is stiffness matrix
The matrix when contain parameters like E, A, displacement and applied force is stiffness matrix

.Q16) How to obtain stiffness matrix
Stiffness matrix can be obtained by applying condition of minimum potential energy to potential energy equation.

What is displacement function
: Displacement function is the assumed polynomial equation, which satisfies boundary conditions.

How to identify order of elements
Order of elements depends on the no. of nodes.

Mention different types of elements
Different types of elements are bar elements, beam element, truss element, shell element, axis symmetric.

Mention some application of FEA

Mechanical, Aerospace, Civil, structure analysis, biomedical, geo -mechanic, electromagnetic.

What is connectivity.

Relation between the connected elements is connectivity.

What are the methods to improve problem solution

Problem solution can be improved by increasing no. of elements or no. of nodes .

Define symmetry in matrix

It is the square matrix in which the element of the row are same as that of element of column.

What is plane stress Stress

acting on 2-D element.

What is plane strain Strain

occurring in 2-D element.

Compare FEA with solid mechanics

complicated irregular structures are difficult in solid mechanics but FEA it's easier with greater accuracy.

What are the packages available for FEA

packages available for FEA: ANSYS, I-DEAS,NASTRAN,ABAQUS, COSMOS,ALGOL,PATRAN.

Define potential energy

Energy possessed by the body due to it's position

Define minimum potential energy

for consertive system of all kinematically admissible displacement field those corresponding to equilibrium extremise the total potential energy if extreme condition is a minimum the equilibrium state is stable

Write potential energy equation for cantilever beam

$\Pi = EI/2 \int (d2y/dx2) dx - pie$

Mention 2 different methods to approach the model of physical system .

Discrete system , continuum

Difference between global co ordinate and local co ordinate

Global co-ordinate	Local co ordinate
1.Contain 2 degrees of freedom	Contain 1 degrees of freedom
2. Represented without prime	Represented with prime
3. Systematic schemes are used	No schemes are used

What is local coordinate

local coordinate contains 1-D.O.F. at each node.

What is global co ordinate
Global coordinate contains 2-D.O.F. at each node.

General assumption made in stress
Assumptions:
1.Truss elements are connected by fracture less pin.
2.Load is applied on the load.
3.Only two forces compressive and tensile are considered.

What is shape function
It is mathematical polynomial, which gives displacement within the element.

What are two general natural coordinate Two
general co-ordinates are η(eeta) and zeeta.

Mention the range of natural co-ordinate Range of
natural coordinates is – 0 to 1 and –1 to 1.

Number of shape function in CST Number
of shape function in CST are three.

Number of shape function in quadrilateral
Number of shape function in quadrilateral are 4.

Why we are using natural integration
we are using natural integration to simplify the problem.

Explain one point formula
1-point shape function contains one term $\int f(x)dx = w.f(x)$

Explain two point formula
2-point shape function contains two term $\int f(x).dx = w1f(x1)\ w2f(x2)$

Why we are using polynomial equation in FEA
Polynomial equation gives continuous solution & it is simple to solve problem.

What are the two important characteristics in stiffness matrix
Characteristics of stiffness matrix is symmetric and bonded

Mention two schemes to represent band width2-
Schemes are
Horizontal numbering
Vertical numbering

What are forces involved in work potential
Forces involved are Body force, Traction force and Point force

Q.48)What are the different methods to apply boundary condition

Elimination, multiconstraint, penalty method

What is isoparametric elements
These are those the S.F. used to define variables of displacement equal to S.F. used to represent geometry.

What is orthotropic elements
Material which has three orthogonal planes of symmetry said to be orthotropic elements. Only nine constants are required to describe constituent equation

What is anisotropic elements
The material which doesn't contain any plane of symmetry

What is isotropic elements
isotropic material is one in which every plane is plane of symmetry only two constants are enough to describe constituent equation

What is super parametric elements
GSF > DSF (geometric shape function, displacement shape function
Q.54) What is sub parametric elements
GSF < DSF

Different coordinates involved in chain rule.
Different coordinates involved in chain rule are normal , local and displacement coordinates .

What are the 2 different approaches to study elasticity ?
Strength of material , Theory of elasticity

Mention any two methods to solve continuum problems.
1. Raleighritz method
   2. Galerkin method

List the properties of shape functions. SF for
1D bar element N1=0, N2= 0 at node 1
N1=0 N2= 1 at node 2 . Diff. Of S.F. S.F are constant

Define truss.
Structural member which is subjected to either tensile or compression

What is weighted residual methods.
It's a method in FEA for accurate solution avoiding error
(residue = error)

Different methods to solve weighed residual problem.Point
allocation, Sub domain , Galerkein,least square.

Explain the principle of virtual work.
If the force and displacement are unrelated by cause effect relation then the work is said to be virtual work.

Explain the principle of virtual displacement.
Actual displacement is considered without bothering amount of force is called virtual displacement .

What are different types of shape functions.
Diff types are Lagarangian and Hermite Shape function

Differentiate two types of shape functions.
Lagarangian shape function only for variable
Hermite shape function is for both variable and its derivative

Mention some advantages of FEA over solid mechanics.
1.Applied for complicated structure
2.   Analysis is simple
   3. More accurate solution

Define Young's Modulus and Poisson's Ratio.
E - it is the ratio of stress and strain
μ- It is the ratio of lateral strain and longitudinal strain

Mention different types of elastic constants.
Young's modulus, shear modulus, bulk modulus

Specify the terms required to solve FEA problem.
Meshing , properties of material,boundry condition and initial condition.

What are the assumptions made in linear static problems.
all displacement is small, material is isotrpic,linear,elastic solid with   E and μ

Which is the most accepted form of numerical integration in FEM?
Gaussian Quadrature

List the different approaches to derive integral equation
Direct method , variation method, weighed residual method, energy method

What are the advantages of symmetrical matrix.
symmetrical matrix simplifies the calculation

What are the different types of errors in FEA?
Modeling error , Descritized and Numerical error

What are the advantages of isoparametric elements
Useful in modeling structure with curved edges
They are versatile & they are used in 2-D and 3-D elasticity problems

Q.76). Define frontal method for finite element matrices.
In 3- dimensional problems, the size of the stiffness matrix increases rapidly even with the banded method of modeling. An alternative direct method that results in considerable saving in the use of computer memory is called frontal method.

Q.77) What is the another name of the 3-dimensional frames
Space frames.

Q.78). Define beam elements.
Beam elements are slender members that are used for supporting transverses loading.

Explain preprocessor steps.
Determining the Nodal coordinates, connectivity, boundary condition, material information

Explain processing steps.
Stiffness generation, modification, solutions to the equation resulting in evolution.

Explain post processing steps.
Deformation confirmation, mode shapes, temperature and stress distribution, interpretation

What are the difference b/w beams and plane frames.
It is similar to beams expect that axial loads &axial deformations are present. The elements also have different orientation.

Mention some common material properties.
Isotropic, orthrotropic, ductility, brittleness

What are the different types of analysis.
Thermal, structural (load), fluid, electromagnetic analysis

Define steady state analysis.
The analysis carried out at constant temperature.

What is the advantage of subjecting solids to axisymmetric loading.
Axisymmetric loading reduces the 3-D problem into 2-D problems because of total symmetry about the z-axis.

Define CST elements.
The constant strain triangle is that where the displacement inside an element is represented by 3-nodal displacement (3-shape functions).

How to generate the data files for larger problems in FEA.Using
MESHGEN program generates data files.

Define mesh plotting
It is the convenient way of reviewing the coordinate and connectivity data is by plotting it using computer.

Explain lumped mass matrices.
It is the total element mass in each direction is distributed equally to the nodes of the element, and the masses are associated with translational degrees of freedom.

Briefly explain steps involved in Lagrangian method

- Formulation of potential energy function.
  assuming displacement function
- Checking displacement function considering boundary condition
- Substitute differential function in potential energy equation
- Potential energy function is minimized
- Unknown parameters are determined and substituted in assumed   equation

Explain steps involved in Gallerkins method.
Formulate differential equation of the equilibrium
Assume trial function, which satisfies boundary conditions
Substitute displacement function in differential equation then assume the difference due to approx. function be 'R' (residue)
Use gallerkin formula
Determine unknown terms and then substitute in differential function

   Define jacobian matrix.
The matrix which is defined explicitly in terms of the local coordinate is known as jacobian(J).

Mention six components of stress.3
linear stress along x, y, z direction
3 lateral stress along x, y, z direction

Mention six components of strain.3
linear strain along x, y, z direction
3 lateral strain along x, y, z direction

   Define Winkler foundations.
Large beams are supported on soil form a class of applications known as winkler foundations.

   Define variational principle
The problem which specifies a scalar quantity potential energy is defined in an integral form.

   How to solve the prismatic problems.
The coefficient of the ordinary differential equation are independent of one of the coordinate and the solution of the system can frequently carried out efficiently by standard analytical methods.

   Define stress & strain.
Stress is the ratio of applied load to its area.
Strain is the ratio of change in length to its original length.

Mention the two distinct procedures available for obtaining the approximation in theintegral forms.
Method weighed residuals.
Method of variation functional.