COURSE MATERIAL

IV Year B. Tech I- Semester

MECHANICAL ENGINEERING

AY: 2024-25

COMPUTER AIDED DESIGN AND SIMULATION LAB

R20A0389

Prepared by:
S. Deepthi
Assistant Professor

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

DEPARTMENT OF MECHANICAL ENGINEERING

(Autonomous Institution-UGC, Govt. of India)
Secunderabad-500100, Telangana State, India.
www.mrcet.ac.in

MRCET CAMPUS

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

(AUTONOMOUS INSTITUTION - UGC, GOVT. OF INDIA)

Affiliated to JNTUH; Approved by AICTE, NBA-Tier 1 & NAAC with A-GRADE | ISO 9001:2015

Maisammaguda, Dhulapally, Komaplly, Secunderabad - 500100, Telangana State, India

LABORATORY MANUAL & RECORD

Name:..

Roll No:.................Branch:...

Year:..................Sem:...

MRCET CAMPUS

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

(AUTONOMOUS INSTITUTION - UGC, GOVT. OF INDIA)

Affiliated to JNTUH; Approved by AICTE, NBA-Tier 1 & NAAC with A-GRADE | ISO 9001:2015

Maisammaguda, Dhulapally, Komaplly, Secunderabad - 500100, Telangana State, India

Certificate

Certified that this is the Bonafide Record of the Work Done by

Mr./Ms...Roll.No.................of

B.Tech year .... Semester for Academic year **2024-2025**

in...Laboratory.

Date: Faculty Incharge HOD

Internal Examiner External Examiner

INDEX

S.No	Date	Name of the Activity/Experiment	Grade/ Marks	Faculty Signature

INDEX

S.No	Date	Name of the Activity/Experiment	Grade/ Marks	Faculty Signature

CONTENTS

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

(Autonomous Institution – UGC, Govt. of India)

VISION

- To establish a pedestal for the integral innovation, team spirit, originality and competence in the students, expose them to face the global challenges and become technology leaders of Indian vision of modern society.

MISSION

- To become a model institution in the fields of Engineering, Technology and Management.
- To impart holistic education to the students to render them as industry ready engineers.
- To ensure synchronization of MRCET ideologies with challenging demands of International Pioneering Organizations.

QUALITY POLICY

- To implement best practices in Teaching and Learning process for both UG and PG courses meticulously.
- To provide state of art infrastructure and expertise to impart quality education.
- To groom the students to become intellectually creative and professionally competitive.
- To channelize the activities and tune them in heights of commitment and sincerity, the requisites to claim the never – ending ladder of **SUCCESS** year after year.

For more information: www.mrcet.ac.in

VISION

To become an innovative knowledge center in mechanical engineering through state-of-the-art teaching-learning and research practices, promoting creative thinking professionals.

MISSION

The Department of Mechanical Engineering is dedicated for transforming the students into highly competent Mechanical engineers to meet the needs of the industry, in a changing and challenging technical environment, by strongly focusing in the fundamentals of engineering sciences for achieving excellent results in their professional pursuits.

Quality Policy

- ✓ To pursuit global Standards of excellence in all our endeavors namely teaching, research and continuing education and to remain accountable in our core and support functions, through processes of self-evaluation and continuous improvement.
- ✓ To create a midst of excellence for imparting state of art education, industry-oriented training research in the field of technical education.

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

(Autonomous Institution – UGC, Govt. of India)

www.mrcet.ac.in

Department of Mechanical Engineering

PROGRAM OUTCOMES

Engineering Graduates will be able to:

1. **Engineering knowledge:** Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.
2. **Problem analysis**: Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.
3. **Design/development of solutions**: Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.
4. **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.
5. **Modern tool usage**: Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.
6. **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 practice.
7. **Environment and sustainability**: Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.
8. **Ethics**: Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.
9. **Individual and teamwork**: Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.
10. **Communication**: Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.
11. **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.

12. **Life-long learning**: Recognize the need for and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.

PROGRAM SPECIFIC OUTCOMES (PSOs)

PSO1 Ability to analyze, design and develop Machine learning systems to solve the Engineering problems by integrating design and manufacturing Domains.

PSO2 Ability to succeed in competitive examinations or to pursue higher studies or research.

PSO3 Ability to apply the learned Mechanical Engineering knowledge for the Development of society and self.

Program Educational Objectives (PEOs)

The Program Educational Objectives of the program offered by the department are broadly listed below:

PEO1: PREPARATION

To provide sound foundation in mathematical, scientific and engineering fundamentals necessary to analyze, formulate and solve engineering problems.

PEO2: CORE COMPETANCE

To provide thorough knowledge in Mechanical Engineering subjects including theoretical knowledge and practical training for preparing Artificial models pertaining to Automobile Engineering, Element Analysis,Production Technology,Mechatronics etc.,

PEO3: INVENTION, INNOVATION AND CREATIVITY

To make the students to design, experiment, analyze, interpret in the core field with the help of other inter disciplinary concepts wherever applicable.

PEO4: CAREER DEVELOPMENT

To inculcate the habit of lifelong learning for career development through successful completion of advanced degrees, professional development courses, industrial training etc.

PEO5: PROFESSIONALISM

To impart technical knowledge, ethical values for professional development of the student to solve complex problems and to work in multi-disciplinary ambience, whose solutions lead to significant societal benefits.

GENERAL LABORATORY INSTRUCTIONS

1. Students are advised to come to the laboratory at-least 5 minutes before (to the starting time), those who come after 5minutes will not be allowed into the lab.
2. Plan your task properly much before to the commencement, come prepared to the lab with the synopsis / program / experiment details.
3. Student should enter into the laboratory with:

   a) Laboratory observation notes with all the details (Problem statement, Aim, Algorithm, Procedure, Program, Expected Output, etc.,) filled in for the lab session.

   b) Laboratory Record updated upto the last session experiments and other utensils (if any) needed in the lab.

   c) Proper Dress code and Identity card.
4. Sign in the laboratory login register, write the TIME-IN, and occupy the computer system allotted to you by the faculty.
5. Execute your task in the laboratory, and record the results/output in the lab observation notebook, and get certified by the concerned faculty.
6. All the students should be polite and cooperative with the laboratory staff, must maintain the discipline and decency in the laboratory.
7. Computer labs are established with sophisticated and high end branded systems, which should be utilized properly.
8. Students / Faculty must keep their mobile phones in SWITCHED OFF mode during the lab sessions. Misuse of the equipment, misbehaviors with the staff and systems etc., will attract severe punishment.
9. Students must take the permission of the faculty in case of any urgency to go out; if anybody found loitering outside the lab / class without permission during working hours will be treated seriously and punished appropriately.
10. Students should LOG OFF/ SHUT DOWN the computer system before he/she leaves the lab after completing the task (experiment) in all aspects. He/she must ensure the system / seat is kept properly.

IV Year B.Tech – ME - I Semester

(R20A0389) COMPUTER AIDED DESIGN AND SIMULATION LAB

COURSE OBJECTIVES:

1. To analyze the various mechanical components in both static conditions

2. To impart the students with necessary computer aided analysis skills.

3. To analyze the various mechanical components in the dynamic conditions.

4. Simulation of mechanical components by visualization software's.

5. To impart the knowledge on program-based simulation for solving the problems.

LIST OF EXPERIMENTS

1. To determine the stresses acting on a cantilever beam with a point load of -10000 N acting at one of its ends and perpendicular to the axis of the beam.

2. To determine the nodal deflections, reaction forces, and stress of the indeterminate truss system when subjected to point loads, E = 210 GPa, A = 0.1 m^2.

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

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

5. Determination of deflections component and principal and Von-mises stresses in plane stress, plane strain and Axisymmetric components.

6. The corner angle bracket is shown below. The upper left hand pin-hole is constrained around its entire circumference and a tapered pressure load is applied to the bottom of lower right hand pin-hole. Compute Maximum displacement, Von-Mises stress.

7. To perform a Modal Analysis of Cantilever beam for natural frequency determination. Modulus of elasticity = 200GPa, Density = 7800 Kg/m^3, Poison ratio = 0.27

8. To conduct the harmonic forced response test by applying a cyclic load (harmonic) at the end of the beam. The frequency of the load will be varied from 1 - 100 Hz. Modulus of elasticity = 200GPa, Poisson's ratio = 0.3, Density = 7800 Kg/m^3.

9. To perform a thermal stress analysis of a rectangular plate by using ANSYS software. Assume Thermal conductivity of the plate, KXX=401 W/(m-K).

10. To conduct the convective heat transfer analysis of a 2D component using ANSYS software. Thermal conductivity of the plate, KXX=16 W/(m-K).

11. Determining density of air using Scilab

12. Reynolds number calculation using Scilab

13. Fluids in Motion Bernoulli equation: Pitot Static Tube calculation using Scilab

Note: At least 10 experiments are to be conducted.

Any Three Software Packages from the following:

Use of Auto CAD, CATIA, Creo, ANSYS, SCILAB, Open FOAM, Matlab, NISA, CAEFEM, etc.

Course Outcomes

1. Understand the various types of analysis in Ansys.

2. Able to do the Simulation of mechanical components by visualization software

3. Understand the dynamic analysis of the various components and their properties.

4. Understand the concept of simulation using the program-based software.

5. Understand the solving and analyzing the mechanical components using empirical

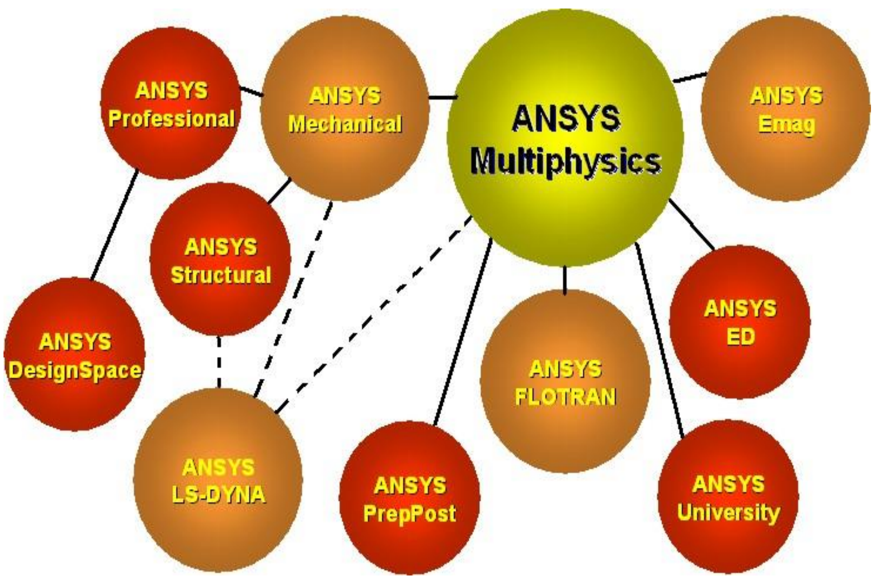
equations.

INTRODUCTION TO ANSYS

ANSYS is a general-purpose finite element-modeling package for numerically solving a wide variety of mechanical problems. These problems include: static/dynamic structural analysis (both linear and non-linear), heat transfer and fluid problems, as well as acoustic and electro-magnetic problems.

Why Ansys?

- ANSYS is a complete FEA software package used by engineers worldwide in virtually all fields of engineering:
  - ✓ Structural
  - ✓ Thermal
  - ✓ Fluid (CFD, Acoustics, and other fluid analyses)
  - ✓ Low- and High-Frequency Electromagnetics
- A partial list of industries in which ANSYS is used:
  - ✓ Aerospace--- Electronics & Appliances
  - ✓ Automotive--- Heavy Equipment & Machinery
  - ✓ Biomedical--- MEMS - Micro Electromechanical Systems
  - ✓ Bridges & Buildings--- Sporting Goods
- ANSYS Multiphysics is the flagship ANSYS product which includes all capabilities in all engineering disciplines.
  - ✓ ANSYS Classic Environment for exposure to all ANSYS functionality



- There are three main component products derived from ANSYS Multiphysics:
    - ✓ ANSYS Mechanical – structural & thermal capabilities
    - ✓ ANSYS Emag – electromagnetics
    - ✓ ANSYS FLOTRAN – CFD capabilities
- Other product lines:
    - ✓ ANSYS LS-DYNA – for highly nonlinear structural problems
    - ✓ ANSYS Professional – linear structural and thermal analyses, a subset of ANSYS Mechanical capabilities
    - ✓ ANSYSDesign Space – linear structural and steady state thermal analyses, a subset of ANSYS Mechanical capabilities in the Workbench Environment.

Structural analysis: is used to determine deformations, strains, stresses, and Reaction forces.

- **Static analysis:**
    - ✓ Used for static loading conditions.
    - ✓ Nonlinear behavior such as large deflections, large strain, contact, plasticity, hyper elasticity, and creep can be simulated.
- **Dynamic analysis:**
    - ✓ Includes mass and damping effects.
    - ✓ Modal analysis calculates natural frequencies and mode shapes.
    - ✓ Harmonic analysis determines a structure's response to sinusoidal loads of known amplitude and frequency.
    - ✓ Transient Dynamic analysis determines a structure's response to time-varying loads and can include nonlinear behavior.
- **Other structural capabilities**
    - ✓ Spectrum analysis
    - ✓ Random vibrations
    - ✓ Eigen value buckling
    - ✓ Substructuring, submodeling
- **Explicit Dynamics with ANSYS/LS-DYNA**
    - ✓ Intended for very large deformation simulations where inertia forces are dominant.
    - ✓ Used to simulate Impact, crushing, rapid forming, etc.

Thermal analysis: is used to determine the temperature distribution in an object. Other quantities of interest include amount of heat lost or gained, thermal gradients, and thermal flux. All three primary **heat transfer modes** can be simulated:

Conduction, convection, radiation.

- Steady-State – Time dependent effects are ignored.
- Transient – To determine temperatures, etc. as a function of time.
  – Allows phase change (melting or freezing) to be simulated.

Electromagnetic analysis: is used to calculate magnetic fields in electromagnetic devices.

Static and low-frequency electromagnetics:

– To simulate devices operating with DC power sources, low-frequency AC, or low-frequency transient signals.

Example: solenoid actuators, motors, transformers

– Quantities of interest include magnetic flux density, field intensity, magnetic forces and torques, impedance, inductance, eddy currents, power loss, and flux leakage.

Computational Fluid Dynamics (CFD): To determine the flow distributions and temperatures in a fluid.

– ANSYS/FLOTRAN can simulate laminar and turbulent flow, compressible and incompressible flow, and multiple species.

Applications: aerospace, electronic packaging, automotive design.

– Typical quantities of interest are velocities, pressures, temperatures, and film coefficients.

Working in ANSYS

1.1. Opening ANSYS SESSION:

Ansys can be opened in Windows Operating System through

- ❖ Start>programs>Ansys18>Interactive
- ❖ Start>programs>Ansys18>Run Interactive
- ❖ Start>programs>Ansys18>Batch

The Interactive Option is used in the very beginning of Ansys Session to set

- ❖ Working Directory
- ❖ Default File Name
- ❖ Graphics driver
- ❖ Data Space
- ❖ Workspace

- ❖ Menus to be visible
- ❖ Command Line Arguments

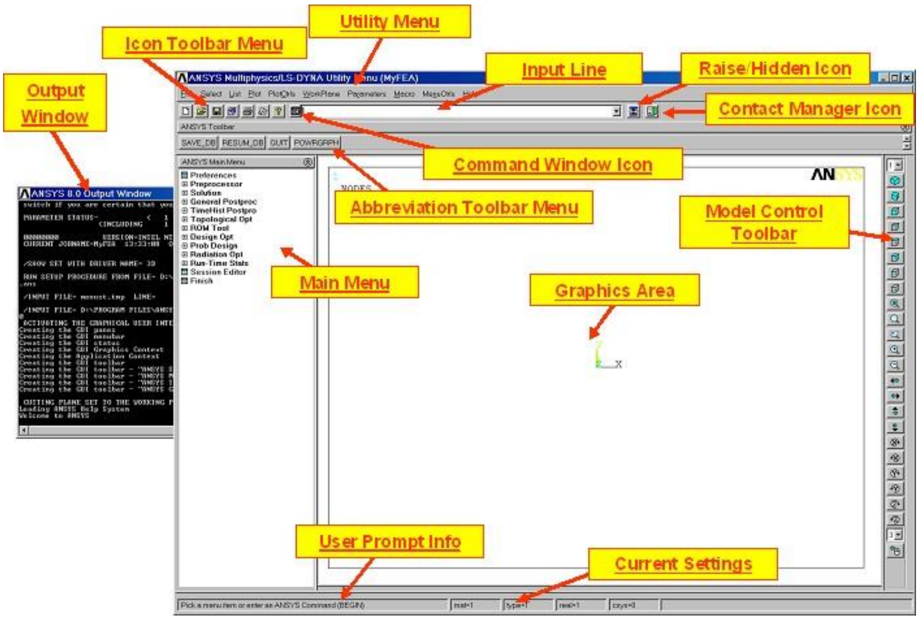
Run-Interactive directly opens the Ansys Graphical user Interface (GUI)

Batch Utility is used to run the Programs Background.

MODELLING APPROACH

- **Bottom-up approach:** Creation of model by defining the geometry of the structure with nodes and elements
- **Top-down approach:** Building a solid model using a 3D CAD program and then dividing the model into nodes and elements

1.2. ANSYS Menu:

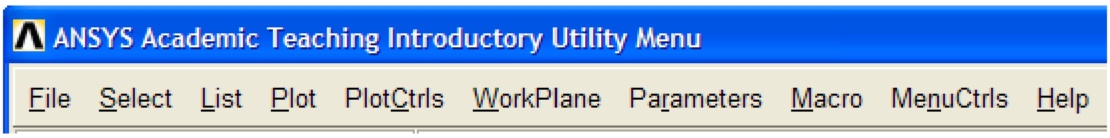


By Default, ANSYS opens 6 Menus. They are

1. Utility Menu
2. Main Menu
3. Input Window
4. Tool Bar
5. Graphics Window
6. Output Window

Utility Menu

This menu contains all important options as follows



A. File: The file contains

- **Clear & Start:** To clear the database &Start a new job
- **Resume from:** To resume the previously stored job
- **Save as:** Save the database as filename.db
- **Read Input from:** if input is taken from Outside programmed file
- **Switch Output:** To external file or by default files in *.iges format is supported without any additional software. BY CATIA, UG, PRO-E you can import the geometry
- **Export:** To export to use in other software's.
- **Exit:** To close the Ansys Session.

B. **Select:** This is very important option for viewing the results or applying the boundary conditions. The parts of the model can be selected and can manipulate for data. This option contains

- **Entities:** Entities to be selected like key points, lines, nodes, elements, areas, volumes, etc
- **Components:** Naming and grouping the selected components.
- **Everything:** Selecting only that part
- **Everything below:** Selecting the entities below that.

C. **List:** This option can be used to listing the elements, nodes, volumes, forces, displacements etc.

D. **Plot:** This option is used to plot the areas, volumes, nodes, elements etc.

E. **Plot Controls:** This option is very important and contains

- **Pan Zoom Rotate:** It opens another menu through which zooming and rotation of the model is possible.
- **View Setting:** By default Z plane is perpendicular to the viewer. By this view option, view settings can be changed.
- **Numbering:** this is useful for setting on/off the entity numbering
- **Symbols:** to view the applied translations, forces, pressures, etc. this option should be used to set them on.
- **Style:** Sectioning, vector arrow sizing and real structural appearances is possible through this.
- **Window Controls:** Window positioning (Layout) is possible with this.
- **Animate:** Animation can be done for the output data using this.
- **Device Options:** Wireframe models can be observed through this.
- **Hard Copy:** data can be sent either to printer or any external file.

- **Capture Image:** To capture the graphics window output to a *.bmp image.
- **Multiplot Window Layout:** To view the results in more than one window.
-

F. **Work plane:** By default, Z Plane is perpendicular for data input. For any changes in the global X,Y & Z planes, the work plane should be rotated to create the model or view

G. **Parameters:** These are the scalar parameters represented with values.

Eg: b=10

H. **Macros:** These are grouping of Ansys commands to fulfill particular work. These can be taken equivalent to C, C++ & Java Functions.

I. **Menu Controls:** This can be used to set on/off the menus.

J. **Help:** For all the help files related to commands and topics

Main Menu: This menu contains

- **Pre-processor:** This sub option can be used to build and mesh the model through proper element selection and boundary conditions.
- **Solution:** this option can be used solve the matrix equation through proper solver.
- **Post Processor:** This option is used to interpret the results.
- **Design Optimization:** This option is used to optimize the structure.
- **Time History Processor:** For dynamic problems, results can be viewed through this option.
- **Run Stats:** This option can be used to find the status of the model, time it
- take s for execution, computer processor capabilities, wave front size etc.

Input Window: This can be used to input commands or named selection.

Tool Bar: This contains options like saving the file, resuming the file database, Quitting the Ansys session and Graphics Type.

Graphics Window: This is where the model creation and plotting of results carried out.

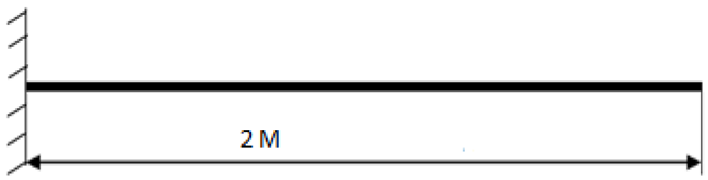
Output Window: This shows the status of the work being carried out.

EXPERIMENT NO. 1

Determination of deflection and stresses in 2D beams

Two-Dimensional static linear analysis of a cantilever beam

AIM: To determine the stresses acting on a cantilever beam with a point load of -10000 N acting at one of its ends and perpendicular to the axis of the beam.



Young's modulus = 2e5

Poisson's ratio = 0.3

Length of the beam = 2m = 2000mm

Breadth of the beam = 10 cm = 100mm

Height of the beam = 50mm

SYSTEM CONFIGURATION:

- Ram: 2 GB
- Processor: Intel CORE i3
- Operating system: Window XP Service Pack 3
- Software: ANSYS (Version19.0)

PROCEDURE:

Step 1: File – clear and start new – do not read file – ok

Preferences

Step 2: Ansys Main Menu – Preferences select – STRUCTURAL – h method – ok

Preprocessor

Step 3: Element type – Add/Edit/Delete – Add – BEAM – 2 node 188– ok- close.

Step 4: Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX– 2e5– PRXY – 0.3 – ok – close.

Step 5: Sections-Beams-common sections- sub type- rectangle (1St element) - enter B=100, H = 50- preview-ok.

Step 6: Modeling – Create – Nodes – In Active CS – Node Number =1 – X, Y, Z Locations = 0,0,0, - Apply (first node is created) – Node Number =2 – X, Y, Z Locations = 2000,0,0, - ok (second node is created)

Step 7: Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – ok (elements are created through nodes).

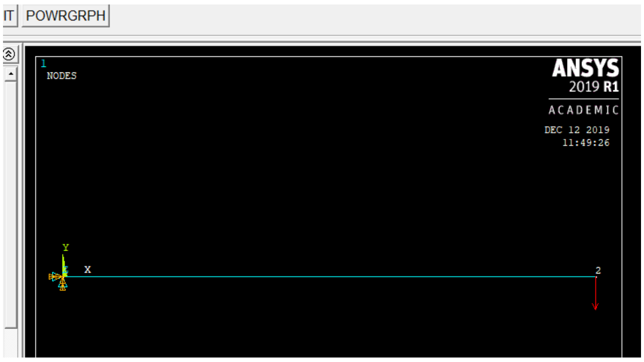
Solution

Step 8: Solution – Analysis Type – New Analysis – Static – ok.

Step 9: Solution – Define Loads – Apply – Structural – Displacement – On Nodes – Pick 1st node – apply – DOFs to be constrained – ALL DOF – ok**.**

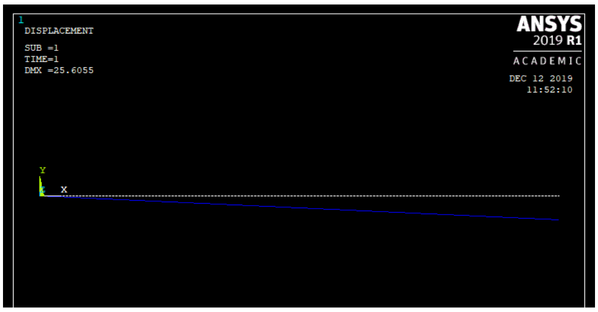
Step 10: Solution – Define Loads – Apply – Structural – Force/Moment – On Nodes – Pick 2nd node – apply – direction of force/mom – FY – Real part of force/mom – -10000 – ok.

Step 11: Solve – current LS – ok (Solution is done is displayed) – close.



General Post Processor

Step 12: Plot Results - Deformed Shape – Select-Def + undef edge' - click 'OK'



Step 13: Plot Results – Contour plot – Nodal solution – DOF solution – Y component of displacement – OK

RESULT:

Maximum Nodal Displacement (DMX) =

Maximum Stress (SMN) =

VIVA-VOCE QUESTIONS

1. What is meant by beam?
2. Explain the types of beams?
3. Define term element
4. Define term node
5. Give example for higher order elements

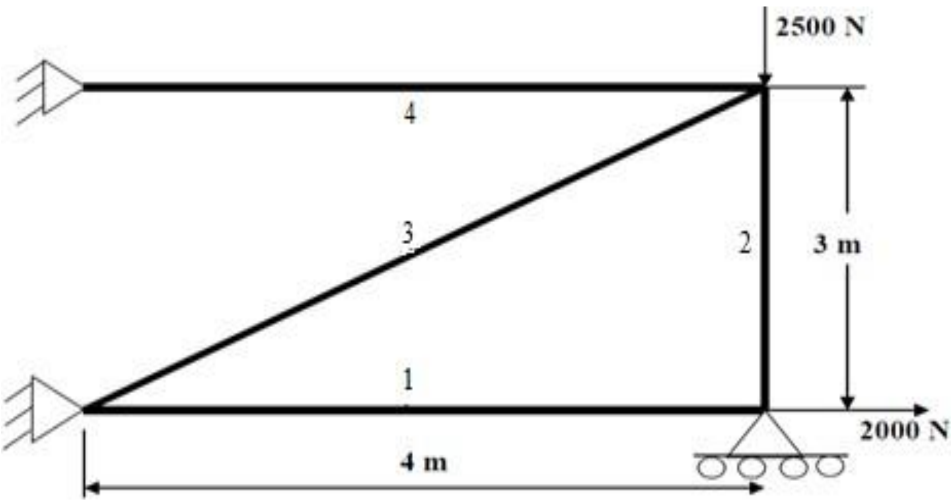
OBSERVATIONS

EXPERIMENT NO. 2

Determination of deflection and stresses in 2D trusses

2-D Static linear analysis of a truss structure

AIM: To determine the nodal deflections, reaction forces, and stress of the indeterminate truss system when subjected to point loads, E = 210 GPa, A = 0.1 m^2



.

SYSTEM CONFIGURATION:

- Ram: 2 GB
- Processor: Intel CORE i3
- Operating system: Window XP Service Pack 3
- Software: ANSYS (Version19.0)

PROCEDURE:

Step 1: File – clear and start new – do not read file – ok

Preferences

Step 2: Ansys Main Menu – Preferences select – STRUCTURAL – h method – ok

Preprocessor

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

Step 4: Real constants – Add – ok – real constant set no – 1 – c/s area – 0.1 – ok – close.

Step 5: Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX– 210e9 – ok – close.

Step 6: Modeling – Create – Nodes – In Active CS – Node Number =1 – X, Y, Z Locations = 0,0,0, - Apply (first node is created) – Node Number =2 – X, Y, Z Locations

= 4,0,0, - ok (second node is created) - Node Number =3 – X, Y, Z Locations = 4,3,0,

- ok (Third node is created) - Node Number =4 – X, Y, Z Locations = 0,3,0, - ok (fourth node is created)

Step 7: Create–Elements–Elem Attributes – Material number – 1 – Real constant set number – 1 – ok

Step 8: Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – apply- pick 2 & 3 – apply – pick 3 & 1 – apply –pick 3 & 4 – ok (elements are created through nodes).

Solution

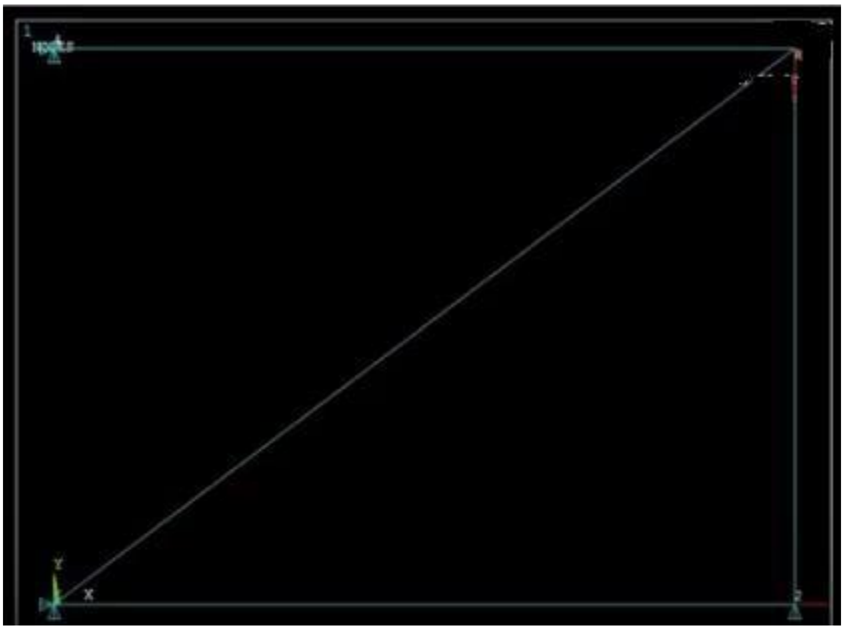
Step 9: Solution – Analysis Type – New Analysis – Static – ok.

Step 10: Loads – Define loads – apply – Structural – Displacement – on Nodes – pick node 1 & 4 – apply – DOFs to be constrained – All DOF – ok – on Nodes – pick node 2 – apply – DOFs to be constrained – UY – ok.

Step 11: Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply – direction of For/Mom – FX – Force/Moment value – 2000 (+ve value) – ok

Step 12: Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 3 – apply – direction of For/Mom – FY – Force/Moment value – -2500 (-ve value) – ok.

Step 13: Solve – current LS – ok (Solution is done is displayed) – close.



General Post Processor

Step 14: Plot Results - Deformed Shape – Select-Def + undef edge' - click 'OK'

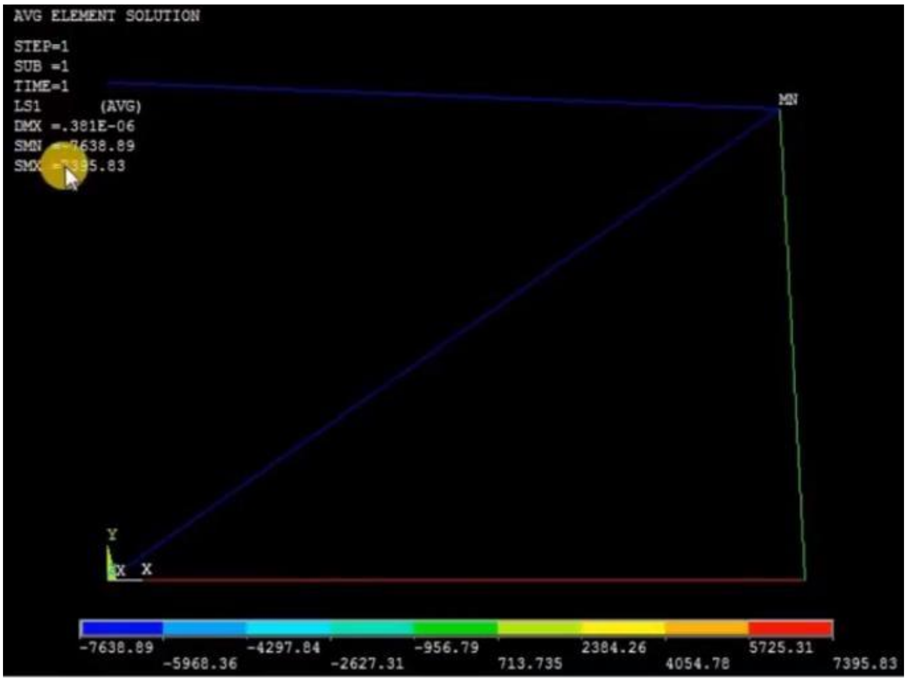
Step 15: Element table – Define table – Add –'Results data item' – By Sequence num – LS – LS1 – ok.

Step 16: Plot results – contour plot –Element table – item to be plotted LS,1, avg common nodes- yes average- ok.

Step 17: Reaction forces: List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).

Step 18: Plot results- nodal solution-ok-DOF solution- Y component of displacement-ok.

Step 19: Animation: PlotCtrls – Animate – Deformed shape – def+undeformed-ok.



RESULT:

Maximum Nodal Displacement (DMX) =

Minimum Stress (SMN) =

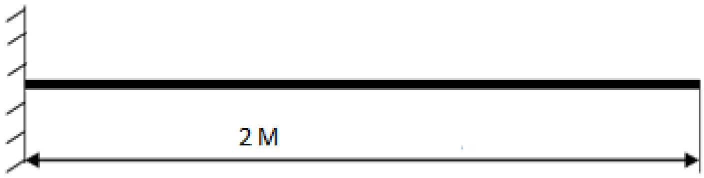
VIVA-VOCE QUESTIONS

1. What is the difference between beam and truss?
2. Explain shear force and bending moment
3. What is meant by point of contraflexure?
4. Element attributes must be set before meshing the solid model. (T/F)
5. In a plane strain, the strain in the direction of thickness is assumed to be zero.(T/F)

OBSERVATIONS

EXPERIMENT NO. 3

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



Young's modulus = 2e5

Poisson's ratio = 0.3

Length of the beam = 2m = 2000mm

Breadth of the beam = 10 cm = 100mm

Height of the beam = 50mm

Point load of -10000 N acting at one of its ends and perpendicular to the axis of the beam.

SYSTEM CONFIGURATION:

- Ram: 2 GB
- Processor: Intel CORE i3
- Operating system: Window XP Service Pack 3
- Software: ANSYS (Version19.0)

PROCEDURE:

Step 1: File – clear and start new – do not read file – ok

Preferences

Step 2: Ansys Main Menu – Preferences select – STRUCTURAL – h method – ok

Preprocessor

Step 3: Element type – Add/Edit/Delete – Add – BEAM – 2 node 188– ok- close.

Step 4: Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX– 2e5– PRXY – 0.3 – ok – close.

Step 5: Sections-Beams-common sections- sub type- rectangle (1st element) - enter B=100, H = 50- preview-ok.

Step 6: Modeling – Create – Nodes – In Active CS – Node Number =1 – X, Y, Z Locations = 0,0,0, - Apply (first node is created) – Node Number =2 – X, Y, Z Locations = 2000,0,0, - ok (second node is created)

Step 7: Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – ok (elements are created through nodes).

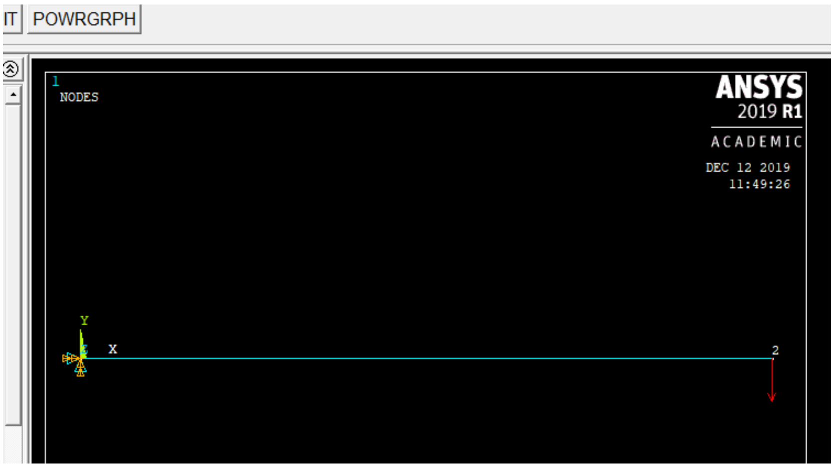
Solution

Step 8: Solution – Analysis Type – New Analysis – Static – ok.

Step 9: Solution – Define Loads – Apply – Structural – Displacement – On Nodes – Pick 1st node – apply – DOFs to be constrained – ALL DOF – ok**.**

Step 10: Solution – Define Loads – Apply – Structural – Force/Moment – On Nodes – Pick 2nd node – apply – direction of force/mom – FY – Real part of force/mom – -10000 – ok.

Step 11: Solve – current LS – ok (Solution is done is displayed) – close.



General Post Processor

Step 12: Plot Results - Deformed Shape – Select-Def + undef edge' - click 'OK'

Step 13: Element Table – Define Table – Add – By sequence num – SMISC, 2 – Apply

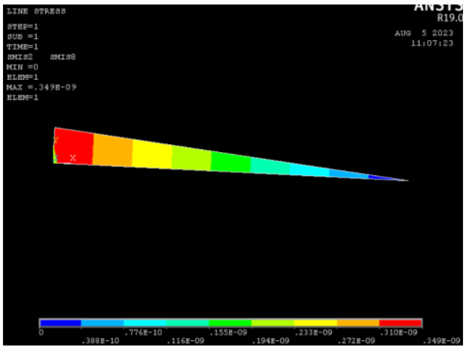
By sequence num – SMISC, 8 – Apply

By sequence num – SMISC, 6 – Apply

By sequence num – SMISC, 12 - ok

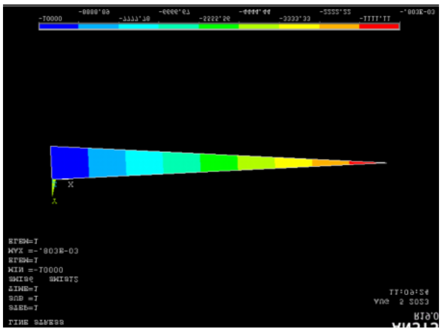
Step 14: Shear Force Diagram

Plot Results – Contour plot – Line Elem Res – Select SMIS2, SMIS8



Step 15: Bending Moment Diagram

Plot Results – Contour plot – Line Elem Res – Select SMIS6, SMIS12



RESULT:

Maximum Nodal Displacement (DMX) =

Maximum Stress (SMN) =

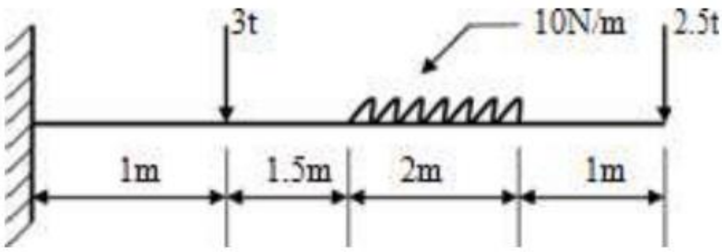
VIVA-VOCE QUESTIONS

1. What is meant by beam?
2. Explain the types of beams?
3. What is difference between node and element?
4. What is shear force?
5. What is bending moment?

OBSERVATIONS

EXPERIMENT NO. 4

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



SYSTEM CONFIGURATION:

- Ram: 2 GB
- Processor: Intel CORE i3
- Operating system: Window XP Service Pack 3
- Software: ANSYS (Version19.0)

PROCEDURE:

Step 1: File – clear and start new – do not read file – ok

Preferences

Step 2: Ansys Main Menu – Preferences select – STRUCTURAL – h method – ok

Preprocessor

Step 3: Element type – Add/Edit/Delete – Add – BEAM – 2 node 188– ok- close.

Step 4: Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX– 2e5– PRXY – 0.3 – ok – close.

Step 5: Sections-Beams-common sections- sub type- rectangle (1st element) - enter B=1, H = 1- preview-ok.

Step 6: Modelling > create > nodes > Inactive CS > (Enter x,y,z values) as shown in the figure:

Node 1:0,0,0

Node 2: 1,0,0

Node 3:2.5,0,0

Node 4:4.5,0,0

Node 5:5.5,0,0

Step 7: Create – Elements – Auto numbered – Thru Nodes – pick all the nodes one by one – ok (elements are created through nodes).

Solution

Step 8: Solution – Analysis Type – New Analysis – Static – ok.

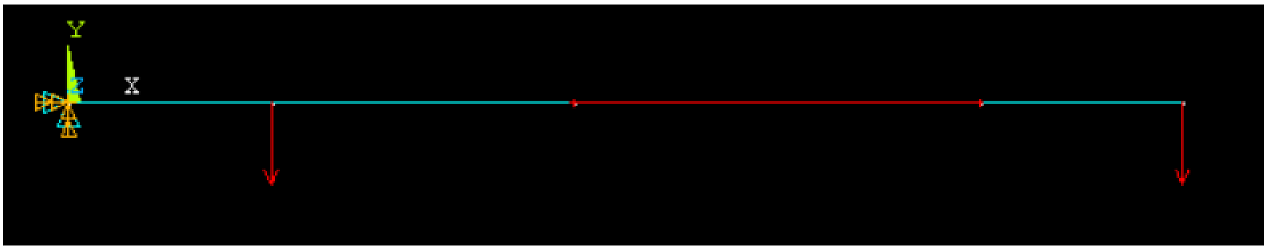
Step 9: Solution – Define Loads – Apply – Structural – Displacement – On Nodes – Pick 1st node – apply – DOFs to be constrained – ALL DOF – ok**.**

Step 10: Solution – Define Loads – Apply – Structural – Force/Moment – On Nodes – Pick 2nd node – apply – direction of force/mom – FY = -3 – ok.

Step 11: Solution – Define Loads – Apply – Structural – Force/Moment – On Nodes – Pick 5th node – apply – direction of force/mom – FY = -2.5 – ok

Step 12: Solution – Define Loads – Apply – Structural – Pressure – On Beam – Select beam between nodes 3, 4 – Enter pressure values as 0.001 at both I & J - ok

Step 13: Solve – current LS – ok (Solution is done is displayed) – close.



General Post Processor

Step 14: Plot Results - Deformed Shape – Select-Def + undef edge' - click 'OK'

Step 15: Element Table – Define Table – Add – By sequence num – SMISC, 2 – Apply

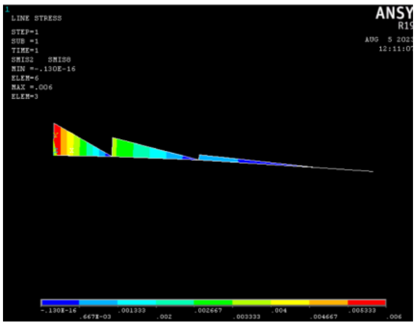
By sequence num – SMISC, 8 – Apply

By sequence num – SMISC, 6 – Apply

By sequence num – SMISC, 12 - ok

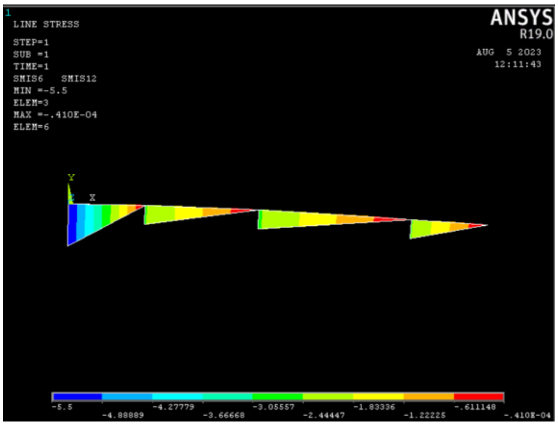
Step 16: Shear Force Diagram

Plot Results – Contour plot – Line Elem Res – Select SMIS2, SMIS8



Step 17: Bending Moment Diagram

Plot Results – Contour plot – Line Elem Res – Select SMIS6, SMIS12



RESULT:

Maximum Nodal Displacement (DMX) =

Maximum Stress (SMN) =

VIVA-VOCE QUESTIONS

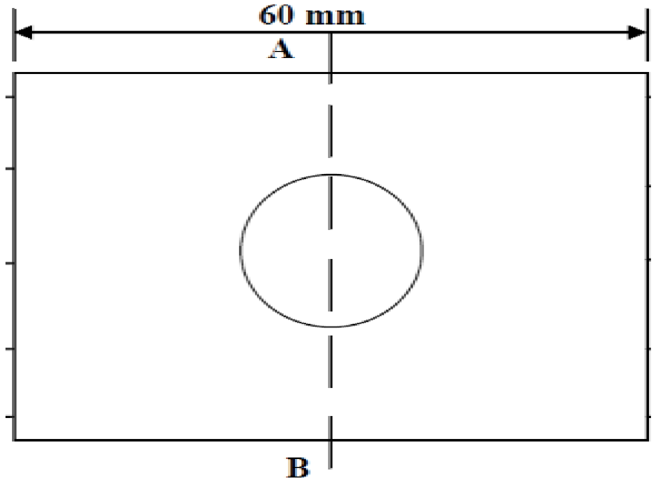
1. What is nodal solution?
2. What is pre processor?
3. What is post processor?
4. What type of options we use in preferences?
5. What is DOF?

OBSERVATIONS

EXPERIMENT NO. 5

AIM: Determination of deflections component and principal and Von-mises stresses in plane stress, plane strain and Axisymmetric components.

In the plate with a hole under plane stress, find deformed shape of the hole and determine the maximum stress distribution along A-B (you may use t = 1 mm). E = 210GPa, t = 1 mm, Poisson's ratio = 0.3, Dia of the circle = 20 mm, Density = 7800 Kg/m3. Analysis assumption – plane stress with thickness is used and do with axisymmetric



AIM: To perform a stress analysis of a rectangular plate with circular hole using analysis software ANSYS.

SYSTEM CONFIGURATION:

- Ram : 2 GB
- Processor : Intel CORE i3
- Operating system : Window XP Service Pack 3
- Software : ANSYS (Version19.0)

PROCEDURE:

Step 1: File – clear and start new – do not read file – ok

Preferences

Step 2: Ansys Main Menu – Preferences select – STRUCTURAL – h method – ok

Preprocessor

Step 3: Element type – Add/Edit/Delete – Add – Solid – Quad 4 node182 – ok – option – element behavior K3 – Plane stress with thickness – ok – close.

Step 4: Real constants – Add – ok – real constant set no – 1 – Thickness – 1 – ok.

Step 5: Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX– 210e9– PRXY – 0.3– ok – close.

Step 6: Modeling –Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 0, 30, 0,20 ok.

Step 7: Create – Area – Circle – solid circle – X, Y, radius – 0, 0, 10 – ok.

Step 8: Operate – Booleans – Subtract – Areas – pick area which is not to be deleted (rectangle) – apply – pick area which is to be deleted (circle) – ok.

Step 9: Meshing – Mesh Tool – Mesh Areas – Quad – Free – Mesh – pick all – ok.

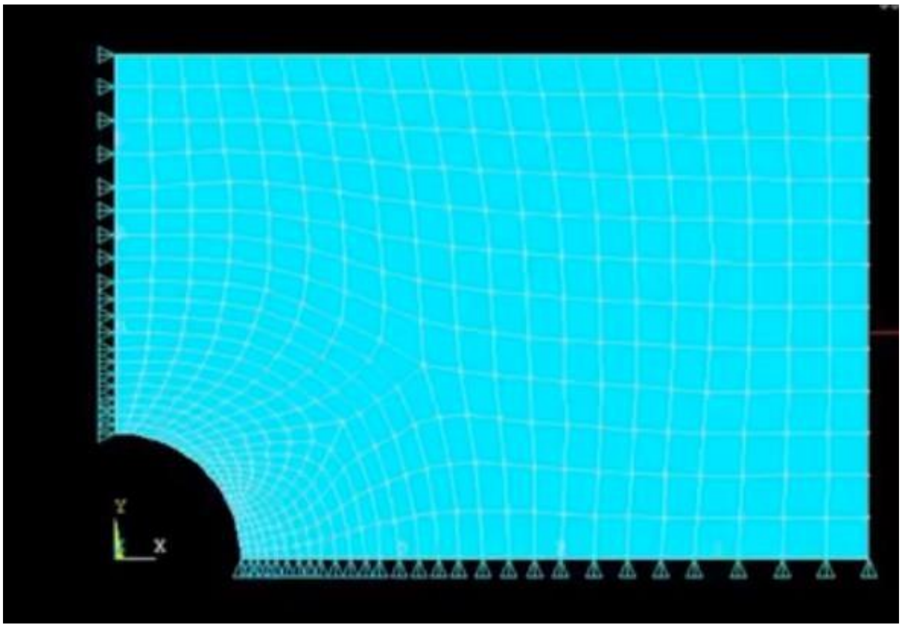
Mesh Tool – Refine – pick all – Level of refinement – 3 – ok.

Solution

Step 10: Solution – Define Loads –loads – apply – Structural – Displacement – Axisymm B.C – on lines – select bottom and left side line – ok.

Step 11: Solution – Define loads – apply – Structural – pressure – on lines – select the right-side line – apply – enter pressure value – 50 (-ve value) –ok.

Step 12: Solve – current LS – ok (Solution is done is displayed) – close.

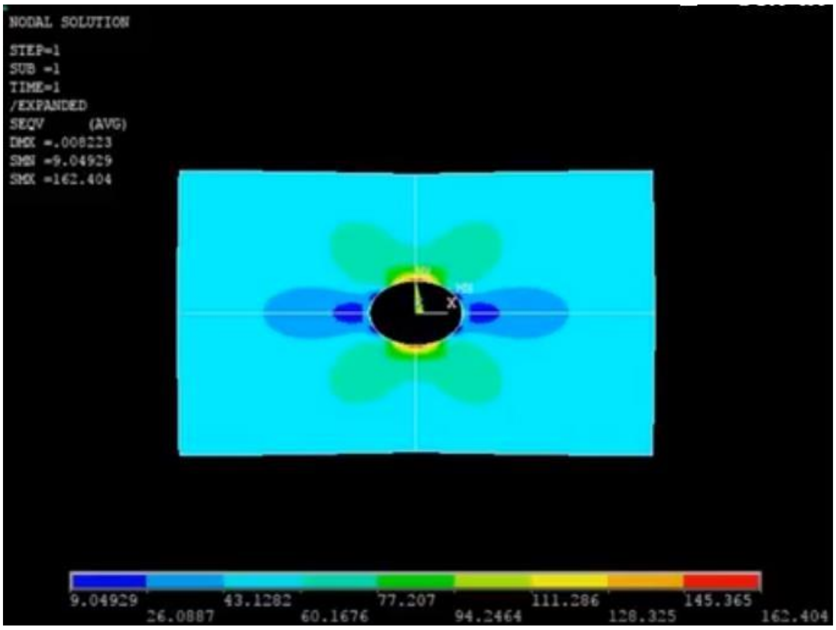


General Post Processor

Step 15: Plot results – contour plot – Element solution – Stress – Von Mises Stress – ok (the stress distribution diagram will be displayed).

Step 16: Plot Ctrls – Animate – Deformed shape – def + undeformed-ok.

Step 17: utility menu – plotctrls – styles- symmetric expansion – periodic/cyclic symmetry – select (1/4) dihedral sym - ok.



RESULT:

Maximum Nodal Displacement (DMX) =

Maximum Stress (SMN) =

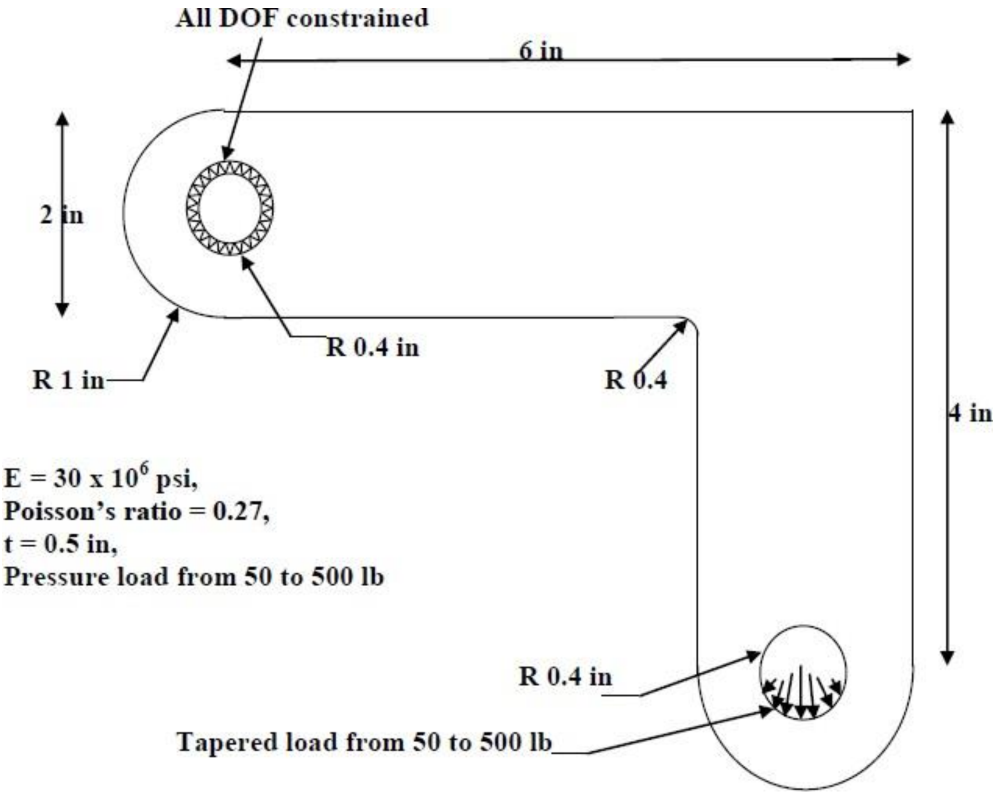
VIVA QUESTIONS:

1. What are the advantages of axisymmetric elements?
2. Define frontal method for finite element matrices
3. Define mesh plotting
4. Define stress and strain with units
5. Explain the boundary conditions for above problem

OBSERVATIONS

EXPERIMENT NO. 6

AIM: The corner angle bracket is shown below. The upper left hand pin-hole is constrained around its entire circumference and a tapered pressure load is applied to the bottom of lower right hand pin-hole. Compute Maximum displacement, Von-Mises stress.



SYSTEM CONFIGURATION:

- Ram : 2 GB
- Processor : Intel CORE i3
- Operating system : Window XP Service Pack 3
- Software : ANSYS (Version19.0)

PROCEDURE:

Step 1: File – clear and start new – do not read file – ok

Preferences

Step 2: Ansys Main Menu – Preferences select – STRUCTURAL – h method – ok

Preprocessor

Step 3: Element type – Add/Edit/Delete – Add – Solid – Quad 8 node 183– ok – option – element behavior K3 – Plane stress with thickness – ok – close.

Step 4: Real constants – Add – ok – real constant set no – 1 – Thickness – 0.5 – ok.

Step 5: Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX– 30e6– PRXY – 0.27– ok – close.

Step 6: Modeling –Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 0, 6, 0,2 – apply –Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 4, 6, -2, 2 – ok.

Step 7: Create – Area –Circle – solid circle – X, Y, radius – 0, 1, 1 – apply – X, Y, radius – 5, -2, 1 – ok.

Step 8: Operate – Booleans – Add – Areas – pick all.

Step 9: Create – Lines – Line fillet – pick the two lines where fillet is required – apply – fillet radius – 0.4 – ok.

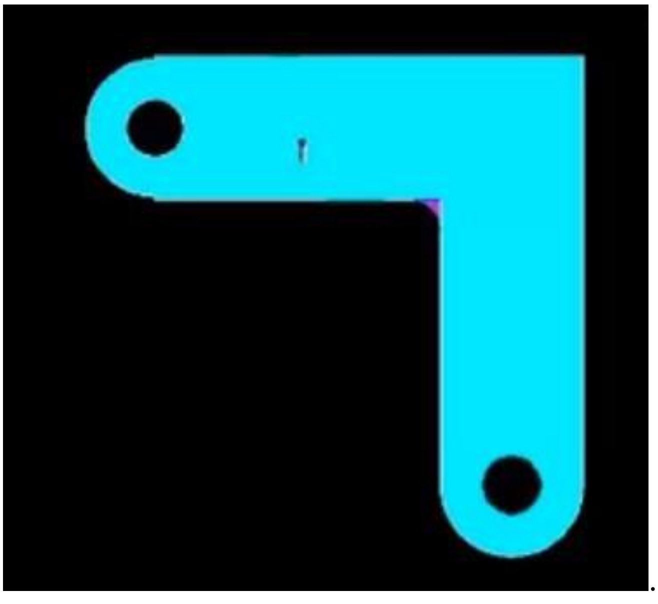
Step 10: Create – Areas – Arbitrary – by lines – pick filleted lines – ok.

Step 11: Operate – Booleans –Add – Areas – pick all

Step 12: Create – Area – Circle – solid circle – X, Y, radius – 0, 1, 0.4 – apply –X, Y, radius – 5, -2, 0.4 – ok.

Step 13: Operate – Booleans – Subtract – Areas – pick area which is not to be deleted (bracket) – apply – pick areas which is to be deleted (pick two circles) – ok.

Step 14: Meshing – Mesh Tool – Mesh Areas – Quad – Free – Mesh – pick all – ok. Mesh Tool – Refine – pick all – Level of refinement – 3 – ok.



Solution

Step 15: Loads – Define loads – apply – Structural – Displacement – on Lines – select the inner lines of the upper circle – apply – DOFs to be constrained – ALL DOF – ok.

Step 16: Loads – Define loads – apply – Structural – Pressure – on Lines – Pick line defining bottom left part ofthe circle – apply – load PRES value – 50 – optional PRES

Step 17: Loads – Define loads – apply – Structural – Pressure – on Lines – Pick line defining bottom right part of the circle – apply – load PRES value – 500 – optional PRES value – 50 – ok.

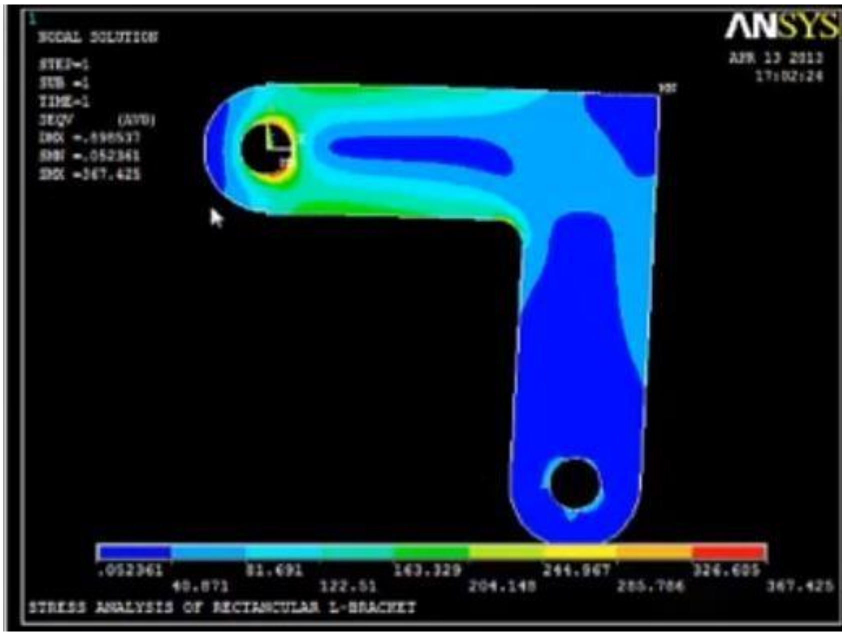
Step 18: Solve – current LS – ok (Solution is done is displayed) – close.

General Post Processor

Step 19: Plot Results – Deformed Shape – def+undeformed – ok.

Step 20: Plot results – contour plot – Element solu – Stress – Von Mises Stress – ok (the stress distribution diagram will be displayed).

Step 21: PlotCtrls – Animate – Deformed shape – def+undeformed-ok.



RESULT:

Maximum Nodal Displacement (DMX) =

Maximum Stress (SMN) =

VIVA QUESTIONS:

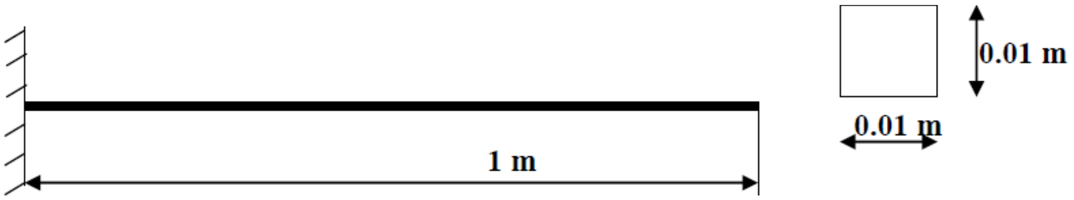
1. What is connectivity?
2. What is plane stress
3. Difference between global coordinate and local co ordinate
4. General assumption made in stress
5. Explain one-point formula

OBSERVATIONS

Experiment 7

Estimation of natural frequencies and mode shapes

AIM: To perform a Modal Analysis of Cantilever beam for natural frequency determination. Modulus of elasticity = 200GPa, Density = 7800 Kg/m^3, Poison ratio = 0.27



SYSTEM CONFIGURATION:

- Ram: 2 GB
- Processor: Intel CORE i3
- Operating system: Window XP Service Pack 3
- Software: ANSYS (Version19.0)

PROCEDURE:

Step 1: File – clear and start new – do not read file – ok

Preferences

Step 2: Ansys Main Menu – Preferences select – STRUCTURAL – h method – ok

Preprocessor

Step 3: Element type – Add/Edit/Delete – Add – BEAM – 2 node 188– ok- close.

Step 4: Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX– 200e9– PRXY – 0.27– Density – 7800 – ok – close.

Step 5: Sections-Beams-common sections- sub type- rectangle (1St element) - enter B=10, H = 10- preview-ok.

Step 6: Modeling – Create – Nodes – In Active CS – Node Number =1 – X, Y, Z Locations = 0,0,0, - Apply (first node is created) – Node Number =2 – X, Y, Z Locations = 1000,0,0, - ok (second node is created)

Step 7: Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – ok (elements are created through nodes).

Solution

Step 8: Solution – Analysis Type – New Analysis – Modal – ok.

Step 9: Solution – Analysis Type – Analysis options – no of modes to extract – 5 – no of modes to expand – 5 – ok – (use default values) – ok.

Step 10: Solution – Define Loads – Apply – Structural – Displacement – On Nodes – Pick 1st node apply – DOFs to be constrained – ALL DOF – ok.

Step 11: Solve – current LS – ok (Solution is done is displayed) – close.

General Post Processor

Step 12: Result Summary – close.

Step 13: Read Results – First Set

Step 14: Plot Results – Deformed Shape – def + undeformed – ok.

Step 15: Plot Ctrls – Animate – Deformed shape – def + undeformed-ok.

Step 16: Read Results – Next Set

Step 17: Plot Results – Deformed Shape – def + undeformed – ok.

Step 18: Plot Ctrls – Animate – Deformed shape – def + undeformed-ok.

RESULT:

Thus, the Model Analysis of Cantilever beam with load for natural frequency is done by using the ANSYS Software.

VIVA QUESTIONS:

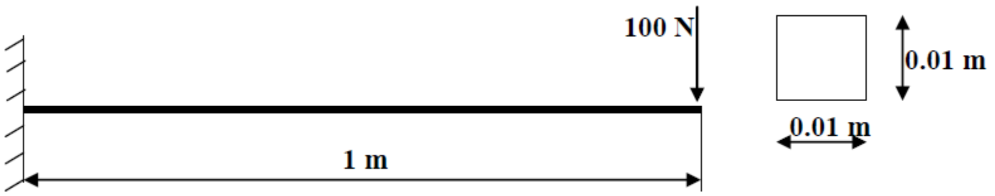
1. What is the total degree of freedom of Ansys commercial package?
2. What are the different menus in ANSYS?
3. What is Work space and Swap space.
4. What is default value of worksp1ace and Swap space?
5. What file format ANSYS can support

OBSERVATIONS

Experiment 8

Estimation of Harmonic response of 2D beam

AIM: To conduct the harmonic forced response test by applying a cyclic load (harmonic) at the end of the beam. The frequency of the load will be varied from 1 - 100 Hz. Modulus of elasticity = 200GPa, Poisson's ratio = 0.3, Density = 7800 Kg/m^3.



SYSTEM CONFIGURATION:

- Ram : 2 GB
- Processor : Intel CORE i3
- Operating system : Window XP Service Pack 3
- Software : ANSYS (Version19.0)

PROCEDURE:

Step 1: File – clear and start new – do not read file – ok

Preferences

Step 2: Ansys Main Menu – Preferences select – STRUCTURAL – h method – ok

Preprocessor

Step 3: Element type – Add/Edit/Delete – Add – BEAM – 2 node 188– ok- close.

Step 4: Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX– 200e9– PRXY – 0.3 – Density – 7800 – ok – close.

Step 5: Sections-Beams-common sections- sub type- rectangle (1St element) - enter b=10, h = 10- preview-ok.

Step 6: Modeling – Create – Nodes – In Active CS – Node Number =1 – X, Y, Z Locations = 0,0,0, - Apply (first node is created) – Node Number =2 – X, Y, Z Locations = 1000,0,0, - ok (second node is created)

Step 7: Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – ok (elements are created through nodes).

Solution

Step 8: Solution – Analysis Type – New Analysis – Harmonic – ok.

Step 9: Solution – Analysis Type – Analysis options – Solution method – FULL – DOF printout format – Real + imaginary – ok – (use default values) – ok.

Step 10: Solution – Define Loads – Apply – Structural – Displacement – On Nodes – Pick1st node - apply – DOFs to be constrained – ALL DOF – ok.

Step 11: Solution – Define Loads – Apply – Structural – Force/Moment – On Nodes – Pick2nd node – apply – direction of force/mom – FY – Real part of force/mom – -100 – imaginary part of force/mom – 0 – ok.

Step 12: Solution – Load Step Opts – Time/Frequency – Freq and Sub stps...
– Harmonic frequency range– 0 – 100 – number of substeps – 100 – B.C – stepped – ok.

Step 13: Solve – current LS – ok (Solution is done is displayed) – close.

Time Hist Post pro

Step 14: Time Hist post pro – Variable Viewer – Click "Add" icon – Nodal Solution – DOF Solution –Y-Component of displacement – ok – Enter 2 – ok. Click "List data" icon and view the amplitude list. Click "Graph" icon and view the graph. To get a better view of the response.

Step 15: Utility Menu – Plot Ctrls – Style – Graphs – Modify Axis – Y axis scale – Logarithmic –ok. Utility Menu – Plot – Replot.

This is the response at node 2 for the cyclic load applied at this node from 0 - 100 Hz.

File–Report Generator–Choose Append–ok–Image Capture–ok - close

RESULT:

Thus the harmonic analysis of 2D component is done by using the ANSYS Software.

VIVA QUESTION:

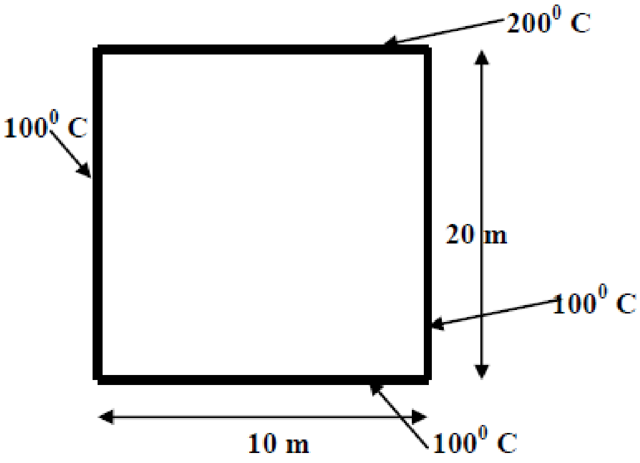
1. What is P-method and H-method
2. What is scalar parameters in ANSYS.
3. What are primary nodes and Secondary nodes.
4. What is Mirror (Reflection).
5. What is subroutines.

OBSERVATIONS

Experiment 9

Steady state heat transfer Analysis

AIM: To perform a thermal stress analysis of a rectangular plate by using ANSYS software. Assume Thermal conductivity of the plate, KXX=401 W/(m-K).



SYSTEM CONFIGURATION:

Ram	: 2 GB
Processor	: Intel CORE i3
Operating system	: Window XP Service Pack 3
Software	: ANSYS (Version19.0)

PROCEDURE:

Step 1: File – clear and start new – do not read file – ok

Preferences

Step 2: Ansys Main Menu – Preferences select – Thermal – h method – ok

Preprocessor

Step 3: Element type – Add/Edit/Delete – Add – Solid – Quad 4 node – 55 – ok – option – Element behavior K3 – Plane stress with thickness – ok – close.

Step 4: Real constants - Add/Edit/Delete – Add – Ok – THK 0.5 – Ok – Close.

Step 5: Material Properties – material models – Thermal – Conductivity – Isotropic – KXX – 401 - ok – close.

Step 6: Modeling – Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 0, 10, 0, 20 – ok

Step 7: Meshing – Mesh tool – Areas, set – select the object – Ok – Element edge length 0.05 - Ok – Mesh tool- Tri, free - mesh – Select the object –Ok.

Solution

Step 8: Solution – Define Loads – Apply – Thermal – Temperature – on Lines – select bottom, left, right lines –apply – DOFs to be constrained – TEMP – Temp value – 100^0 C – ok.

Step 9: Solution – Define Loads – Apply – Thermal – Temperature – on Lines – select top lines – apply – DOFs to be constrained – TEMP – Temp value – 200^0 C – ok.

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

General Post Processor

Step 11: Plot results – Contour plot – Nodal solution – DOF solution – Nodal Temperature – Ok.

RESULT:

Thus, the conductive heat transfer analysis of a 2D component is done by using the ANSYS Software

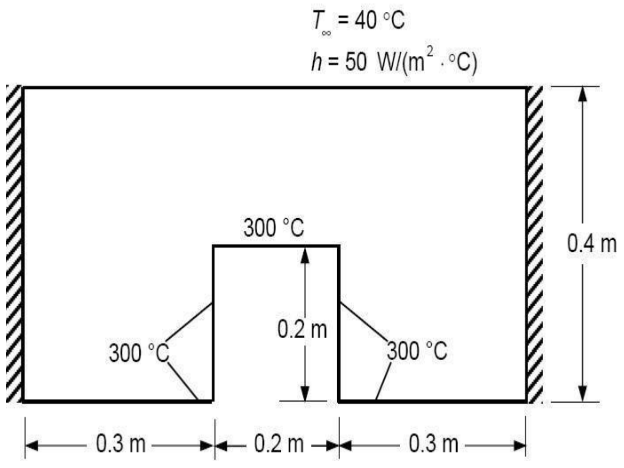
VIVA QUESTION:

1. What are the types of boundary conditions?
2. What are the three phases of finite element method.
3. What is meant by DOF?
4. Explain the general steps in FEA with the help of a flowchart?
5. Give examples for essential (forced or geometric) and non-essential (natural) boundary conditions.

OBSERVATIONS

Experiment 10

AIM: To conduct the convective heat transfer analysis of a 2D component using ANSYS software. Thermal conductivity of the plate, KXX=16 W/(m-K).



SYSTEM CONFIGURATION:

Ram	: 2 GB
Processor	: Intel CORE i3
Operating system	: Window XP Service Pack 3
Software	: ANSYS (Version19.0)

PROCEDURE:

Step 1: File – clear and start new – do not read file – ok

Preferences

Step 2: Ansys Main Menu – Preferences select – Thermal – h method – ok

Preprocessor

Step 3: Element type – Add/Edit/Delete – Add – Solid – Quad 4 node – 55 – ok – option – Element behavior K3 – Plane stress with thickness – ok – close.

Step 4: Real constants - Add/Edit/Delete – Add – Ok – THK 0.5 – Ok – Close.

Step 5: Material Properties – material models – Thermal – Conductivity – Isotropic – KXX – 16 - ok – close.

Step 6: Modeling – Create – Key Points – In Active CS

– Key Points Number =1 – X, Y, Z Locations = 0,0,0, - Apply (1st node is created)

– Key Points Number =2 – X, Y, Z Locations = 300,0,0, - Apply (2nd node is created)

– Key Points Number =3 – X, Y, Z Locations = 300,200,0, - Apply (3rd node is created)

– Key Points Number =4 – X, Y, Z Locations = 500,200,0, - Apply (4th node is

– Key Points Number =5 – X, Y, Z Locations = 500,0,0, - Apply (5th node iscreated)

– Key Points Number =6 – X, Y, Z Locations = 800,0,0, - Apply (6th node iscreated)

– Key Points Number =7 – X, Y, Z Locations = 800,400,0, - Apply (7th node is created)

– Key Points Number =8 – X, Y, Z Locations = 0,400,0, - ok (8th node is created)

Step 7: Modeling – Create – line – lines- straight lines-pick 1 &2, pick 2&3, pick 3 & 4, pick 4 & 5, pick 5 & 6, pick 6 & 7, pick 7 & 8, pick 8 & 0- ok

Step 8: Modeling – Create-Areas- Arbitrary –By lines- pick 1,2,3,4,5,6,7& 8 lines - ok

Step 9: Meshing – Mesh tool – Areas, set – select the object – Ok – Element edge length 0.05 - Ok – Mesh tool- Tri, free - mesh – Select the object –Ok.

Solution

Step 10: Solution – Define Loads – Apply – Thermal – Temperature – on Lines – select lines –apply – DOFs to be constrained – TEMP – Temp value – 300^{0}C – ok.

Step 11: Solution – Define Loads – Apply – Thermal – Convection – On lines – select the top line – Ok – Enter the values of film coefficient 50, bulk temperature 40 – Ok.

Step 12: Solve – current LS – ok (Solution is done is displayed) – close.

General Post Processor

Step 13: Plot results – Contour plot – Nodal solution – DOF solution – Nodal Temperature – Ok.

RESULT:

Thus, the convective heat transfer analysis of a 2D component is done by using the ANSYS Software.

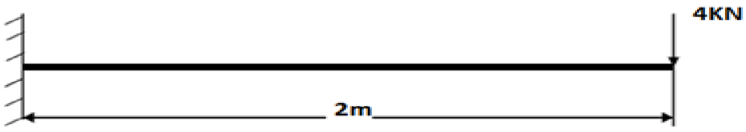
VIVA QUESTION:

1. What is H-method
2. Explain the types of analysis
3. What is meant by iso parametric element?
4. Is beam element an isoparametric element?
5. What is the difference between natural co-ordinates and simple natural co-ordinate?

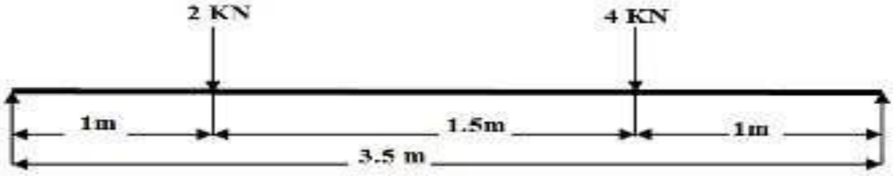
OBSERVATIONS

PRACTICE PROBLEMS

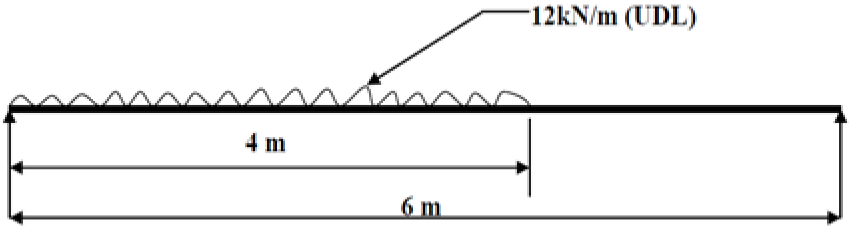
1. Compute the Shear force and bending moment diagrams for the beam shown and find the maximum deflection. Assume rectangular c/s area of 0.2 m * 0.3 m, Young's modulus of 210 GPa, Poisson's ratio 0.27.



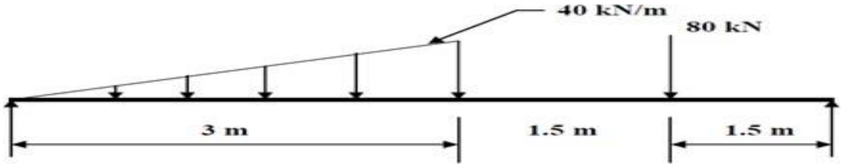
2. Compute the Shear force and bending moment diagrams for the beam shown and find the maximum deflection. Assume rectangular c/s area of 100 mm * 100mm, Young's modulus of210 MPa, Poisson's ratio 0.27.



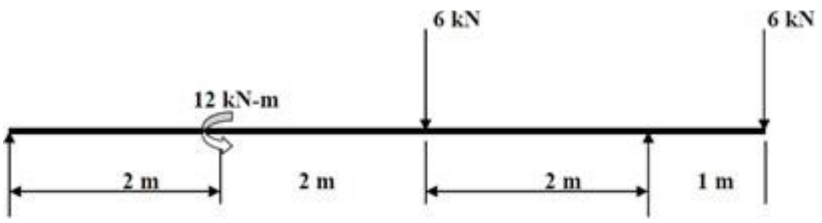
3. Compute the Shear force and bending moment diagrams for the beam shown and find the maximum deflection. Assume rectangular c/s area of 100 mm * 100mm, Young's modulus of210 MPa, Poisson's ratio 0.27.



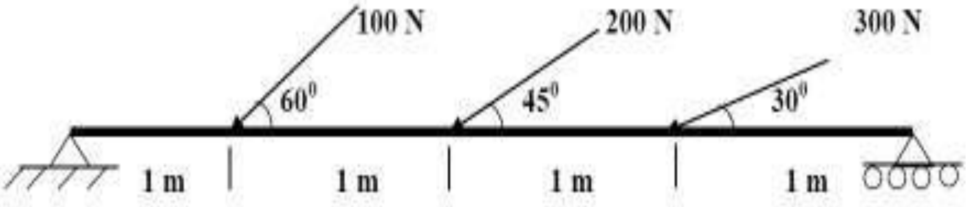
4. Compute the Shear force and bending moment diagrams for the beam shown and find the maximum deflection. Assume rectangular c/s area of 100 mm * 100mm, Young's modulus of210 MPa, Poisson's ratio 0.27.



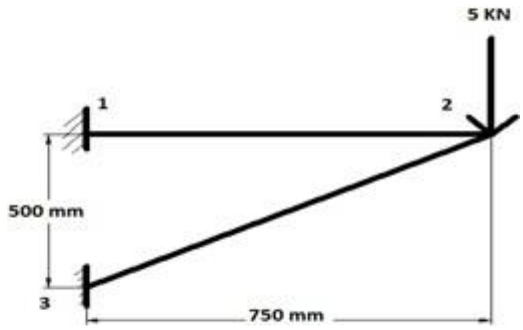
5. Compute the Shear force and bending moment diagrams for the beam shown and find the maximum deflection. Assume rectangular c/s area of 0.2 m * 0.3 m, Young's modulus of 210 GPa, Poisson's ratio 0.27.



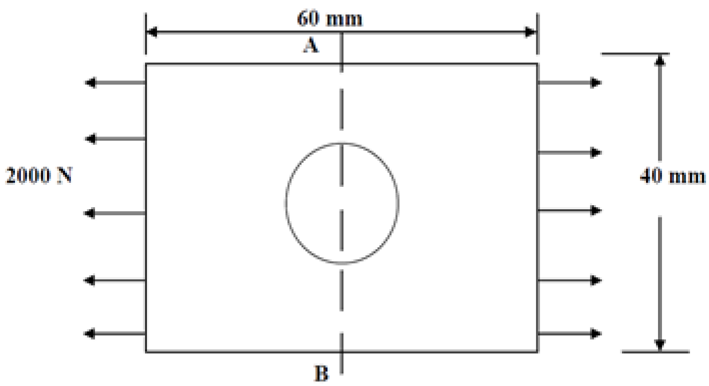
6. Compute the Shear force and bending moment diagrams for the beam shown and find the maximum deflection. Assume rectangular c/s area of 0.2 m * 0.3 m, Young's modulus of 210 GPa, Poisson's ratio 0.27.



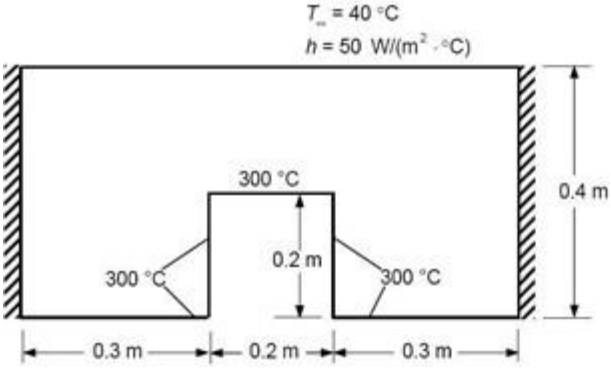
7. Consider the two bar truss shown in figure. For the given data, find Stress in each element, Reaction forces, Nodal displacement. E = 210 GPa, A = 0.1 m^2



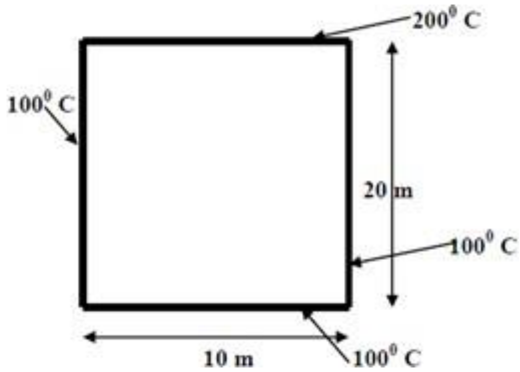
8. In the plate with a hole under plane stress, find deformed shape of the hole and determine the maximum stress distribution along A-B (you may use t = 1 mm). E = 210GPa, t = 1 mm, Poisson's ratio = 0.3, Dia of the circle = 10 mm, Analysis assumption – plane stress with thickness is used.



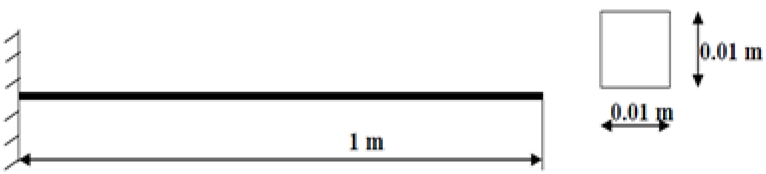
9. To conduct the convective heat transfer analysis of a 2D component using ANSYS software. Thermal conductivity of the plate, KXX=28 W/(m-K).



10. Solve the 2-D heat conduction problem for the temperature distribution within the rectangular plate. Thermal conductivity of the plate, KXX=510 W/(m-K).



11. Modal Analysis of Cantilever beam for natural frequency determination. Modulus of elasticity = 20GPa, Density = 6800 Kg/m^3.



12. Conduct a harmonic forced response test by applying a cyclic load (harmonic) at the end of the beam. The frequency of the load will be varied from 1 - 100 Hz. Modulus of elasticity = 20GPa, Poisson's ratio = 0.3, Density = 6800 Kg/m^3.