Department of Mechanical Engineering

LAB MANUAL
(2017-2018)

Modelling and Analysis Lab (FEA)
(FEA LAB/15MEL68)

VI SEMESTER

NAME: __________________________________________________________

USN : __________________________________________________________

BATCH: ________________ SECTION: __________________________
Channabasaveshwara Institute of Technology
(Affiliated to VTU, Belagavi & Approved by AICTE, New Delhi)
(NAAC Accredited & ISO 9001:2015 Certified Institution)

Department of Mechanical Engineering

Modelling and Analysis Lab (FEA)
(FEA LAB/15MEL68)

Version 1.1
February 2018

Prepared by:
Mr. Natesh C P
Mr. Harshith H S
Assistant Professor’s

Reviewed by:
Mr. Kiran Gowd M R
Assistant Professor

Approved by:
HOD
Dept. of Mechanical Engineering
College Vision
To create centers of excellence in education and to serve the society by enhancing the quality of life through value based professional leadership.

College Mission

- To provide high quality technical and professionally relevant education in a diverse learning environment.
- To provide the values that prepare students to lead their lives with personal integrity, professional ethics and civic responsibility in a global society.
- To prepare the next generation of skilled professionals to successfully compete in the diverse global market.
- To promote a campus environment that welcomes and honors women and men of all races, creeds and cultures, values and intellectual curiosity, pursuit of knowledge and academic integrity and freedom.
- To offer a wide variety of off-campus education and training programmes to individuals and groups.
- To stimulate collaborative efforts with industry, universities, government and professional societies.
- To facilitate public understanding of technical issues and achieve excellence in the operations of the institute.

Department Vision
To create a state of the art learning environment to nurture the learning, blending human values, academic professionalism and research process in the field of mechanical engineering for the betterment of society.

Department Mission

The mission of the department is to

- provide requisite foundation to our students in Mechanical Engineering
- provide cutting edge laboratory resources to bridge the gap between theoretical and practical concepts
- provide exposure to various mechanical industries through periodic industrial visits
- enhance our students skill set and to make them industry ready by systematic skill development program
DEPARTMENT OF MECHANICAL ENGINEERING
SYLLABUS
MODELLING AND ANALYSIS LAB (FEA)

<table>
<thead>
<tr>
<th>Course</th>
<th>Code</th>
<th>Credits</th>
<th>L-T-P</th>
<th>Assessment</th>
<th>Exam Duration</th>
</tr>
</thead>
<tbody>
<tr>
<td>Modeling and Analysis Lab</td>
<td>15MEL68</td>
<td>02</td>
<td>1-0-2</td>
<td>80 20</td>
<td>3Hrs</td>
</tr>
</tbody>
</table>

PART – A

**Study of a FEA package and modelling and stress analysis of:**
1. Bars of constant cross section area, tapered cross section area and stepped bar.
2. Trusses – *(Minimum 2 exercises of different types)*
3. Beams – Simply supported, cantilever, beams with point load, UDL, beams with varying load etc., *(Minimum 6 exercises different nature)*

PART - B

1. Thermal Analysis – 1D & 2D problem with conduction and convection boundary conditions *(Minimum 4 exercises of different types)*
2. Dynamic Analysis to find
   a) Fixed – fixed beam for natural frequency determination
   b) Bar subjected to forcing function
   c) Fixed – fixed beam subjected to forcing function

PART – C (only for demo and oral exam)

1. Demonstrate the use of graphics standards (IGES, STEP etc) to import the model from modeler to solver.
2. Demonstrate one example of contact analysis to learn the procedure to carry out contact analysis.
3. Demonstrate at least two different type of example to model and analyze bars or plates made from composite material.

REFERENCE BOOKS:
3. *Finite Element Analysis*, George R. Buchanan, Schaum Series

Scheme for Examination:
One Question from Part A - 32 Marks (08 Write up +24)
One Question from Part B - 32 Marks (08 Write up +24)
Viva-Voce - 16 Marks

Total 80 Marks
**Note:** If the student fails to attend the regular lab, the experiment has to be completed in the same week. Then the manual/observation and record will be evaluated for 50% of maximum marks.
OBJECTIVES

- The course is intended to provide basic understanding of Modelling and Analysis techniques.
- Students with following aspects:
  1. To acquire basic understanding of Modelling and Analysis software.
  2. To understand the different kinds of analysis and apply the basic principles to find out the stress and other related parameters of bars, beams loaded with loading conditions.
  3. To lean to apply the basic principles to carry out dynamic analysis to know the natural frequency of different kind of beams.

OUTCOMES

At the end of the course the students are able to:

1. Demonstrate the basic features of an analysis package.
2. Use the modern tools to formulate the problem, and able to create geometry, discretize, apply boundary condition to solve problems of bars, truss, beams, plate to find stress with different loading conditions.
3. Demonstrate the deflection of beams subjected to point, uniformly distributed and varying loads further to use the available results to draw shear force and bending moment diagrams.
4. Analyze the given problem by applying basic principle to solve and demonstrate 1D and 2D heat transfer with conduction and convection boundary conditions.
5. Carry out dynamic analysis and finding natural frequencies for various boundary conditions and also analyze with forcing function.
General Instructions to Students

- Students are informed to present 5 min before the commencement of lab.
- Students must enter their name in daily book before entering into lab.
- Students must leave Foot wares before entering lab.
- Students must not carry any valuable things inside the lab.
- Students must inform lab assistant before He/She uses any computer.
- 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.
- For any software/hardware/ Electrical failure of computer during working, 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.
- Students must submit Record book for evaluation before the commencement of lab.
- Students must keep observation book (if necessary).
- Students must keep silent near lab premises.
- Students are informed to follow safety rules.
- Students must obey lab rules and regulations.
- Students must maintain discipline in lab.
- Do not crowd around the computers and run inside the laboratory.
- Please follow instructions precisely as instructed by your supervisor. Do not start the experiment unless your setup is verified & approved by your supervisor.
<table>
<thead>
<tr>
<th>Sl. No.</th>
<th>Title</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>Performing a Typical ANSYS Analysis</td>
</tr>
<tr>
<td>2.</td>
<td>General Steps</td>
</tr>
<tr>
<td>3.</td>
<td>Bars of Constant Cross-section Area</td>
</tr>
<tr>
<td>4.</td>
<td>Bars of Tapered Cross section Area</td>
</tr>
<tr>
<td>5.</td>
<td>Stepped Bar</td>
</tr>
<tr>
<td>6.</td>
<td>Trusses</td>
</tr>
<tr>
<td>7.</td>
<td>Simply Supported Beam</td>
</tr>
<tr>
<td>8.</td>
<td>Simply Supported Beam with Uniformly varying load</td>
</tr>
<tr>
<td>9.</td>
<td>Simply Supported Beam with Uniformly distributed load</td>
</tr>
<tr>
<td>10.</td>
<td>Beam with moment and overhung</td>
</tr>
<tr>
<td>11.</td>
<td>Cantilever Beam</td>
</tr>
<tr>
<td>12.</td>
<td>Beam with angular loads, one end hinged and at other end roller support</td>
</tr>
<tr>
<td>13.</td>
<td>Stress analysis of a rectangular plate with a circular hole</td>
</tr>
<tr>
<td>14.</td>
<td>Corner angle bracket</td>
</tr>
<tr>
<td>15.</td>
<td>Thermal analysis</td>
</tr>
<tr>
<td>16.</td>
<td>Modal Analysis of Cantilever beam for natural frequency determination</td>
</tr>
<tr>
<td>17.</td>
<td>Fixed- fixed beam subjected to forcing function</td>
</tr>
<tr>
<td>18.</td>
<td>Bar subjected to forcing function and Part - C</td>
</tr>
<tr>
<td>19.</td>
<td>Additional problems</td>
</tr>
<tr>
<td>20.</td>
<td>Viva questions</td>
</tr>
</tbody>
</table>
Performing a Typical ANSYS Analysis

The ANSYS program has many finite element analysis capabilities, ranging from a simple, linear, static analysis to a complex, nonlinear, transient dynamic analysis. The analysis guide manuals in the ANSYS documentation set describe specific procedures for performing analyses for different engineering disciplines.

A typical ANSYS analysis has three distinct steps:

- Build the model.
- Apply loads and obtain the solution.
- Review the results.

Building a Model

Building a finite element model requires more of an ANSYS user's time than any other part of the analysis. First, you specify a job name and analysis title. Then, you use the PREP7 preprocessor to define the element types, element real constants, material properties, and the model geometry.

Specifying a Job name and Analysis Title

This task is not required for an analysis, but is recommended.

Defining the Job name

The job name is a name that identifies the ANSYS job. When you define a job name for an analysis, the job name becomes the first part of the name of all files the analysis creates. (The extension or suffix for these files' names is a file identifier such as .DB.) By using a job name for each analysis, you insure that no files are overwritten. If you do not specify a job name, all files receive the name *FILE* or *file*, depending on the operating system.

Command(s): /FILNAME

GUI: Utility Menu>File>Change Job name

Defining Element Types

The ANSYS element library contains more than 100 different element types. Each element type has a unique number and a prefix that identifies the element category: BEAM4, PLANE77, SOLID96, etc. The following element categories are available
The element type determines, among other things:

- The degree-of-freedom set (which in turn implies the discipline-structural, thermal, magnetic, electric, quadrilateral, brick, etc.)
- Whether the element lies in two-dimensional or three-dimensional space.

For example, BEAM4, has six structural degrees of freedom (UX, UY, UZ, ROTX, ROTY, ROTZ), is a line element, and can be modeled in 3-D space. PLANE77 has a thermal degree of freedom (TEMP), is an eight-node quadrilateral element, and can be modeled only in 2-D space.

**Defining Element Real Constants**

Element real constants are properties that depend on the element type, such as cross-sectional properties of a beam element. For example, real constants for BEAM3, the 2-D beam element, are area (AREA), moment of inertia (IZZ), height (HEIGHT), shear deflection constant (SHEARZ), initial strain (ISTRN), and added mass per unit length (ADDMAS). Not all element types require real constants, and different elements of the same type may have different real constant values.

As with element types, each set of real constants has a reference number, and the table of reference number versus real constant set is called the *real constant table*. While defining the elements, you point to the appropriate real constant reference number using the **REAL** command

(Main Menu> Preprocessor> Create> Elements> Elem Attributes).

**Defining Material Properties**

Most element types require material properties. Depending on the application, material properties may be:

- Linear or nonlinear
- Isotropic, orthotropic, or anisotropic
- Constant temperature or temperature-dependent.

As with element types and real constants, each set of material properties has a material reference number. The table of material reference numbers versus material property sets is called the *material table*. Within one analysis, you may have multiple material property sets (to correspond with multiple materials used in the model). ANSYS identifies each set with a unique reference number.
Main Menu > Preprocessor > Material Props > Material Models.

Creating the Model Geometry

Once you have defined material properties, the next step in an analysis is generating a finite element model-nodes and elements-that adequately describes the model geometry.

There are two methods to create the finite element model: solid modeling and direct generation.

With solid modeling, you describe the geometric shape of your model, and then instruct the ANSYS program to automatically mesh the geometry with nodes and elements. You can control the size and shape of the elements that the program creates. With *direct generation*, you "manually" define the location of each node and the connectivity of each element. Several convenience operations, such as copying patterns of existing nodes and elements, symmetry reflection, etc. are available.

Apply Loads and Obtain the Solution

In this step, you use the SOLUTION processor to define the analysis type and analysis options, apply loads, specify load step options, and initiate the finite element solution. You also can apply loads using the PREP7 preprocessor.

Applying Loads

The word loads as used in this manual includes boundary conditions (constraints, supports, or boundary field specifications) as well as other externally and internally applied loads. Loads in the ANSYS program are divided into six categories:

- DOF Constraints
- Forces
- Surface Loads
- Body Loads
- Inertia Loads
- Coupled-field Loads

You can apply most of these loads either on the solid model (keypoints, lines, and areas) or the finite element model (nodes and elements).

Two important load-related terms you need to know are load step and substep. A *load step* is simply a configuration of loads for which you obtain a solution. In a structural analysis, for example, you may apply wind loads in one load step and gravity in a second load step. Load steps are also useful in dividing a transient load history curve into several segments.
PART-A

Problem 1.1: Bar of Constant Cross-section Area

Consider the bar shown in figure below. Young’s modulus is 2.1×10^5 N/mm^2 and Area is 500mm^2. Determine the Nodal Displacement, Stress in each element, Reaction forces.

1. Ansys Main Menu – Preferences-Select – STRUCTURAL – h method- 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) – ok (second node is created).
6. Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – ok (elements are created through nodes).
9. Solve – current LS – ok (Solution is done is displayed) – close.
11. Plot results – contour plot –Element table – item to be plotted LS,1, avg common nodes- yes average- ok.
12. List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).
13. Plot results- nodal solution-ok-DOF solution- x component of displacement-ok.
RESULT:

**Analytical approach:**

Calculation:

Displacement: __________________

Stress: ______________________

Reaction force: ________________

**ANSYS Results:**

<table>
<thead>
<tr>
<th></th>
<th>ANSYS</th>
<th>Theoretical</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deformation</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Stress</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Reaction</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Problem 1.2: Bars of Tapered Cross section Area

Consider the Tapered bar shown in figure below. Determine the Nodal Displacement, Stress in each element, Reaction forces

\[ E = 2 \times 10^5 \text{ N/mm}^2, \text{ Area at root, } A_1 = 1000 \text{ mm}^2, \text{ Area at the end, } A_2 = 500 \text{ mm}^2. \]

Solution: The tapered bar is modified into 2 elements as shown below with modified area of cross section.

\[ (A_1 + A_2)/2 = (1000+500)/2 = 750 \text{ mm}^2 \]

\[ A_1 = (1000+750)/2 = 875 \text{ mm}^2 \]

\[ A_2 = (500+750)/2 = 625 \text{ mm}^2 \]

\[ L_1 = 187.5 \text{ mm} \text{ & } L_2 = 187.5 \text{ mm} \]

1. Ansys Main Menu – Preferences-Select – STRUCTURAL- h method– ok
3. Real constants – Add – ok – real constant set no – 1 – cross-sectional AREA1 – 875 – apply-ok
4. Add – ok – real constant set no – 2 – cross-sectional AREA 2 – 625-ok
6. Modeling – Create – keypoints- In Active CS, =0, Y=0 – Apply (first key point is created) – location in active CS, X= 187.5, Y=0, apply (second key point is created) -location in active CS X=375, Y=0(third key point is created) -ok.
7. Modeling-Create – lines-straight lines-pick key points 1 & 2-ok- pick key points 2 & 3-ok
8. Meshing-mesh attributes-picked lines (pick the lines)-ok-material no= 1, real constants set no = 1, element type no =1, link 1, element section= none defined-pick
the other line-ok-material number 2-define material id 2- real constants set no = 2, element type no =2-element section= none defined-ok.

9. Meshing-size controls-manual size-lines-all lines- no of element divisions=10(yes)-ok
10. Meshing-mesh tool-mesh-pick the lines-ok (the color changes to light blue)
11. Loads – Define loads – apply – Structural – Displacement – on key points- pick key point 1 – apply –DOFs to be constrained – ALL DOF, displacement value=0 – ok.
13. Solve – current LS – ok (Solution is done is displayed) – close.
15. Plot results – contour plot –Element table – item to be plotted LS,1, avg common nodes- yes average- ok.
16. List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).
17. Plot results- nodal solution-ok-DOF solution- x component of displacement-ok.

RESULT:

Analytical approach:

Calculation:
Displacement: _____________________
Stress: __________________________
Reaction force: ____________________

**ANSYS Results:**

<table>
<thead>
<tr>
<th></th>
<th>ANSYS</th>
<th>Theoretical</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deformation</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Stress</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Reaction</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Problem 1.3: Stepped Bar
Consider the stepped bar shown in figure below. Determine the Nodal Displacement, Stress in each element, Reaction forces.

1. Ansys Main Menu – Preferences-Select – STRUCTURAL- h method – ok
4. Add – ok – real constant set no – 2 – cross-sectional AREA 2 – 600-ok
6. Modeling – Create – key points– In Active CS, =0, Y=0 – Apply (first key point is created) – location in active CS, X= 600, Y=0, apply (second key point is created) -location in active CS X=1100, Y=0(third key point is created) -ok.
7. Modeling-Create – lines-straight lines-pick key points 1 & 2-ok- pick key points 2 & 3-ok
8. Meshing-mesh attributes-picked lines (pick the lines)-ok-material no= 1, real constants set no = 1, element type no =1, link 1, element section= none defined-pick the other line-ok-material number 2-define material id 2- real constants set no = 2, element type no =2-element section= none defined-ok.
9. Meshing-size controls-manual size-lines-all lines- no of element divisions=10(yes)-ok
10. Meshing-mesh tool-pick the lines-ok (the color changes to light blue)
11. Loads – Define loads – apply – Structural – Displacement – on key points- pick key point 1 – apply –DOFs to be constrained – ALL DOF, displacement value=0 – ok.
13. Solve – current LS – ok (Solution is done is displayed) – close.
15. Plot results – contour plot –Element table – item to be plotted LS,1, avg common nodes- yes average- ok.
16. List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).
17. Plot results- nodal solution-ok-DOF solution- x component of displacement-ok.
RESULT:

Analytical approach:

Calculation:

<table>
<thead>
<tr>
<th></th>
<th>ANSYS</th>
<th>Theoretical</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deformation</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Stress</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Reaction</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
2. TRUSSES

**Problem 2.1:** Consider the four bar truss shown in figure. For the given data, find Stress in each element, Reaction forces, Nodal displacement. E = 210 GPa, A = 0.1 m².

1. Ansys Main Menu – Preferences-select – STRUCTURAL- h method – ok
5. Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS – 4 (x value w.r.t first node) – apply (second node is created) – x,y,z location in CS – 4, 3 (x, y value w.r.t first node) – apply (third node is created) – 0, 3 (x, y value w.r.t first node) – ok (forth node is created).
6. Create–Elements–Elem Attributes – Material number – 1 – Real constant set number – 1 – ok
11. Solve – current LS – ok (Solution is done is displayed) – close.
13. Plot results – contour plot –Element table – item to be plotted LS,1, avg common nodes- yes average- ok.
14. Reaction forces: List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).
15. Plot results- nodal solution-ok-DOF solution- Y component of displacement-ok.
RESULT:

Analytical approach:

Calculation:
Displacement: ______________________
Stress: __________________________
Reaction force: ____________________

**ANSYS Results:**

<table>
<thead>
<tr>
<th></th>
<th>ANSYS</th>
<th>Theoretical</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deformation</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Stress</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Reaction</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
**Problem 2.2:** Consider the two bar truss shown in figure. For the given data, find Stress in each element, Reaction forces, Nodal displacement. $E = 210$ GPa, $A = 0.1 \text{ m}^2$.

1. Ansys Main Menu – Preferences-select – STRUCTURAL- h method – ok
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 – ok
7. Auto numbered – Thru Nodes – pick 1 & 2 – apply – pick 2 & 3— ok (elements are created through nodes).
10. Solve – current LS – ok (Solution is done is displayed) – close.
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).
RESULT:

Analytical approach:

Calculation:
Displacement: ____________________
Stress: _________________________
Reaction force: ________________

**ANSYS Results:**

<table>
<thead>
<tr>
<th></th>
<th>ANSYS</th>
<th>Theoretical</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deformation</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Stress</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Reaction</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Problem 2.3: Determine the nodal deflections, reaction forces, and stress for the truss system shown below (E = 200GPa, A = 3250mm²).

ANSYS Results:

<table>
<thead>
<tr>
<th></th>
<th>ANSYS</th>
<th>Theoretical</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deformation</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Stress</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Reaction</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
3. BEAMS

Problem 3.1: Simply Supported Beam

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 of 210 MPa, Poisson’s ratio 0.27.

1. Ansys Main Menu – Preferences-Select – STRUCTURAL- h method – ok
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).
9. Solve – current LS – ok (Solution is done is displayed) – 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).

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.


RESULT:

Analytical approach:

Calculation:
Displacement: ____________________
Shear force: ____________________
Bending moment: ________________
Stress:________________________

**ANSYS Results:**

<table>
<thead>
<tr>
<th></th>
<th>ANSYS</th>
<th>Theoretical</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deflection</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Shear force</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bending moment</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Stress</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Problem 3.2: Simply Supported Beam with uniformly varying load.

Compute the Shear force and bending moment diagrams for the beam shown and find the maximum deflection. Assume rectangular c/s area of 100mm * 100m, Young’s modulus of $2.1 \times 10^5$ N/mm$^2$, Poisson’s ratio = 0.27.

![Beam Diagram]

1. Ansys Main Menu – Preferences-Select – STRUCTURAL- h method- ok
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– 3000 (x value w.r.t first node) – apply (second node is created) – 4500 (x value w.r.t first node) –apply (third node is created) – 6000 (x value w.r.t first node) – ok (forth node is created).
10. Solve – current LS – ok (Solution is done is displayed) – 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).


RESULT:

Analytical approach:

Calculation:

Deflection: ______________________

Shear force: ______________________

Bending moment: ______________________

Stress: ______________________

ANSYS Results:

<table>
<thead>
<tr>
<th></th>
<th>ANSYS</th>
<th>Theoretical</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deflection</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Shear force</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bending moment</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Stress</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Problem 3.3: Simply Supported Beam with Uniformly distributed load.

Compute the Shear force and bending moment diagrams for the beam shown and find the maximum deflection. Assume rectangular c/s area of 0.1 m * 0.1 m, Young’s modulus of 210 GPa, Poisson’s ratio 0.27.

![Beam Diagram](image)

1. Ansys Main Menu – Preferences-select – STRUCTURAL – ok
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– 4 (x value w.r.t first node) – apply (second node is created) – 6 (x value w.r.t first node) – ok (third node is created).
10. Solve – current LS – ok (Solution is done is displayed) – close.
14. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS 6 – 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 – SMIS 3 – Elem table item at node J – SMIS 16 – 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).
RESULT:

Analytical approach:

Calculation:

Deflection:______________________
Shear force:______________________
Bending moment:__________________
Stress:__________________________

ANSYS results:

<table>
<thead>
<tr>
<th></th>
<th>ANSYS</th>
<th>Theoretical</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deflection</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Shear force</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bending moment</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Stress</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Problem 3.4: Beam with moment and overhung

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.

1. Ansys Main Menu – Preferences-Select – STRUCTURAL- h method- ok
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) – apply (second node is created) – 4 (x value w.r.t first node) –apply (third node is created) – 6 (x value w.r.t first node) – apply (forth node is created) – 7 (x value w.r.t first node) – ok (fifth node is created).
9. Solve – current LS – ok (Solution is done is displayed) – 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).
15. Reaction forces: List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).

**RESULT:**

**Analytical approach:**

Calculation:

<table>
<thead>
<tr>
<th></th>
<th>ANSYS</th>
<th>Theoretical</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deflection</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SFD</td>
<td></td>
<td></td>
</tr>
<tr>
<td>BMD</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Stress</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Problem 3.5: Cantilever Beam

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.

1. Ansys Main Menu – Preferences-Select – STRUCTURAL- h method – ok
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).
8. DOFs to be constrained – ALL DOF – ok.
10. Solve – current LS – ok (Solution is done is displayed) – 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).
RESULT:

Analytical approach:

Calculation:

<table>
<thead>
<tr>
<th></th>
<th>ANSYS</th>
<th>Theoretical</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deflection</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SFD</td>
<td></td>
<td></td>
</tr>
<tr>
<td>BMD</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Stress</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Problem 3.6: Beam with angular loads

Compute the Shear force and bending moment diagrams for the beam shown in fig such that one end hinged and at the other end is having roller support and find the maximum deflection. Assume rectangular c/s area of 0.2 m \times 0.3 m, Young’s modulus of 210 GPa, Poisson’s ratio 0.27.

![Beam diagram](image)

1. Ansys Main Menu – Preferences select STRUCTURAL – ok
4. Sections – Beams – common sections – sub type – rectangle (1\textsuperscript{st} 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 – 1 (x value w.r.t first node) – apply (second node is created) – 2 (x value w.r.t first node) – apply (third node is created) – 3 (x value w.r.t first node) – apply (forth node is created) – 4 (x value w.r.t first node) – ok (fifth node is created).
10. Solve – current LS – ok (Solution is done is displayed) – 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).

**RESULT:**

**Analytical approach:**

Calculation:
Deflection:______________________
Shear force:______________________
Bending moment:______________________
Stress:______________________

**ANSYS Results:**

<table>
<thead>
<tr>
<th></th>
<th>ANSYS</th>
<th>Theoretical</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deflection</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Shear force</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bending moment</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Stress</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Stress analysis of a rectangular plate with circular hole

Problem 4.1: 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). \(E = 210\text{GPa}, t = 1\text{ mm},\) Poisson’s ratio = 0.3, Dia. of the circle = 10 mm, Analysis assumption – plane stress with thickness is used.

1. Ansys Main Menu – Preferences–Select – STRUCTURAL-h method – ok
5. Modeling –Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 0, 60, 0, 40 – ok.
7. Operate – Booleans – Subtract – Areas – pick area which is not to be deleted (rectangle) – apply – pick area which is to be deleted (circle) – ok.
11. Solve – current LS – ok (Solution is done is displayed) – close.
12. Deformed shape-Plot Results – Deformed Shape – def+undeformed – ok.
13. Plot results – contour plot – Element solu – Stress – Von Mises Stress – ok (the stress distribution diagram will be displayed).
RESULT:

Analytical approach:

Calculation:

<table>
<thead>
<tr>
<th></th>
<th>ANSYS</th>
<th>Theoretical</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deformation</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Stress</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
**Problem 4.2:** 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.

![Image](image-url)

**Ansys Main Menu:**
1. Preferences select STRUCTURAL – ok
3. Real constants – Add – ok – real constant set no – 1 – Thickness – 0.5 – ok.
5. 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. Create – Area – Circle – solid circle – X, Y, radius – 0, 1, 1 – apply – X, Y, radius – 5, -2, 1 – ok.

**Operate – Booleans – Add – Areas – pick all.**
7. Operate – Booleans –
8. Add – Areas – pick all. Create – Area – Circle – solid circle – X, Y, radius – 0, 1, 0.4 – apply – X, Y, radius – 5, -2, 0.4 – ok.
9. 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.

**Material Properties:**
- E = 30 x 10^6 psi
- Poisson’s ratio = 0.27
- t = 0.5 in.
- Pressure load from 50 to 500 lb
13. Solve – current LS – ok (Solution is done is displayed) – close.
15. Plot results – contour plot – Element solu. – Stress – Von Mises Stress – ok (the stress distribution diagram will be displayed).

RESULT:
PART B
THERMAL ANALYSIS

Problem 5.1: Solve the 2-D heat conduction problem for the temperature distribution within the rectangular plate. Thermal conductivity of the plate, KXX=401 W/(m-K).

1. Ansys Main Menu – Preferences-select – THERMAL- h method– ok
4. Modeling – Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 0, 10, 0, 20 – ok.
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
RESULT:
Problem 5.2: A furnace wall is made up of silica brick (K=1.5 W/m°C) and outside magnesia brick (K= 4.9 W/m°C) each 10cm thick. The inner and outer surfaces are exposed to fluids at temperatures of 820°C and 110°C respectively. The contact resistance is 0.001 m²°C/W. The heat transfer coefficient for inner and outer surfaces is equal to 35 W/m²K. Find the heat flow through the wall per unit area per unit time and temperature distribution across the wall. Area= 1m².

Given: \( K_1 = 1.5 \text{ W/m}^\circ \text{C}, \ K_2 = 1 \text{ W/m}^\circ \text{C}, \ K_3 = 4.9 \text{ W/m}^\circ \text{C}, \ h_1 = h_4 = 35 \text{ W/m}^2\text{K}, \ T_1 = 820^\circ \text{C}, \ T_6 = 110^\circ \text{C}, \ L_1 = L_2 = 10 \text{ cm & } L = 1 \text{ mm}. \)

1. Preferences-thermal-h method-ok
2. Preprocessor-Element type-add/edit/delete-link, 3d conduction 33, element type reference N0.=1-apply-link, convection 34 element type reference no.2=2-ok-close
3. Real constant- add/edit/delete-real constant set no=1-C/S area =1-ok-close.
4. Real constant- add/edit/delete-real constant set no=2-C/S area =1-ok-close.
5. Material properties-material model-thermal conductivity-isotropic-KXX=1.5-ok.
   From the define material model behavior menu bar-material new model
   Enter define material id=2-ok
   Thermal-conductivity-isotropic-Kxx=1-ok
   Define material id=3-ok
   Thermal-conductivity-isotropic-Kxx=4.9-ok
   Define material id=4-ok-convection or film coefficient HF= 35, close

Modeling-create-nodes-in active CS
   Enter node no=1, x=0, y=0, z=0-apply
   Enter node no=2, X=0.001, Y=0, Z=0-apply
   Enter node no=3, X=0.101, Y=0, Z=0-apply
   Enter node no=4, X=0.102, Y=0, Z=0-apply
   Enter node no=5, X=0.202, Y=0, Z=0-apply
   Enter node no=6, X=0.203, Y=0, Z=0-ok.

Modeling-create-element-element attributes
   Enter element type no=2 LINK 34 (convection)
   Material no=4 (convection or film coefficient)
   Real constant set no=2 (convection)-ok
Modeling-create-element-auto numbered-through node-pick the nodes 1 & 2-ok
Modeling-create-element-element attributes
Enter element type no=1 LINK 33 (Conduction)
Material no=1 (conduction)
Real constant set no=1 (conduction)-ok
Modeling-create-element-auto numbered-through node-pick the nodes 2 & 3-ok
Modeling-create-element-element attributes
Enter element type no=1 LINK 33 (Conduction)
Material no=2 (conduction)
Real constant set no=1 (conduction)-ok
Modeling-create-element-auto numbered-through node-pick the nodes 3 & 4-ok
Modeling-create-element-element attributes
Enter element type no=1 LINK 33 (Conduction)
Material no=3 (conduction)
Real constant set no=1 (conduction)-ok
Modeling-create-element-auto numbered-through node-pick the nodes 4 & 5 ok.
Modeling-create-element-element attributes
Enter element type no=2 LINK 34 (Convection)
Material no=4 (convection or film coefficient)
Real constant set no=2 (convection)-ok
Modeling-create-element-auto numbered-through node-pick the nodes 5 & 6 ok.

Observe the straight line.
From the menu bar select plot controls-Numbering-Plot numbering control and select element/attributes numbering=element no and don’t change other attributes-ok

   • Solution-define loads-apply-thermal-temperature-on nodes-pick the first nodes-ok-
     temperature-load-temperature value=820°C-apply.
   • Define load-apply-thermal-temperature-on nodes-pick the last node-ok, select temperature-load temperature value=110°C-ok.
   • Solution- solve-current LS-ok.
     Solution is done-close.

7. Read results-last set-ok
8. List results-nodal solution-select temperature-ok
9. Observe the nodal solution per node.
10. 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.
11. Plot results-contour plot-nodal solution-temperature-deformed shape only-ok
12. Element table-define table-add-enter user label item=HTRANS, select by sequence no SMISC, 1-ok-close.
13. Element table-list table-select HTRANS-ok
RESULT:

Analytical approach:

Calculation:

ANSYS Results:

<table>
<thead>
<tr>
<th></th>
<th>ANSYS</th>
<th>Theoretical</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nodal temperature</td>
<td></td>
<td></td>
</tr>
<tr>
<td>T₁</td>
<td></td>
<td></td>
</tr>
<tr>
<td>T₂</td>
<td></td>
<td></td>
</tr>
<tr>
<td>T₃</td>
<td></td>
<td></td>
</tr>
<tr>
<td>T₄</td>
<td></td>
<td></td>
</tr>
<tr>
<td>T₅</td>
<td></td>
<td></td>
</tr>
<tr>
<td>T₆</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Heat flux</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
**Problem 5.3:** The exterior wall of a building is constructed of four materials, 12mm thick gypsum board, 75mm thick fibre glass insulation, 20mm thick plywood and 20mm thick hardboard. The inside and outside air temperatures are 20°C and -10°C respectively. The convective heat transfer coefficients on the inner and outer surfaces of the wall are 6W/m²°C and 10 W/m²°C respectively. Determine the heat flux and the temperature distribution. Take K for gypsum=0.176W/m°C, K for fibre glass=0.036W/m°C, K for plywood=0.115 W/m°C and K for hardboard=0.215 W/m°C. Area= 1m².

1. Preferences-thermal-h method-ok
2. Element type-add/edit/delete-add-link, 3d conduction 33,element type reference N0.=1-apply-link, convection 34 element type reference no.2=2-ok-close
3. Real constant- add/edit/delete-add-real constant set no=1-C/S area =1-ok-close.
4. Real constant- add/edit/delete-add-real constant set no=2-C/S area =1-ok-close.
5. Material properties-material model-convection film coefficient-enter HF=6-ok

From the define material model behaviour menu bar-material new model
Enter define material id=2-ok
Thermal-conductivity-isotropic-K_{xx}=0.176-ok
Define material id=3-ok
Thermal-conductivity-isotropic-K_{xx}=0.036-ok
Define material id=4-ok
Thermal-conductivity-isotropic-K_{xx}=0.115-ok
Define material id=5-ok
Thermal-conductivity-isotropic-K_{xx}=0.215-ok
Define material id=6-convection or film coefficient HF=10, close

Modeling-create-nodes-in active CS
Enter node no=1,x=0,y=0,z=0-apply
Enter node no=2, X=0.001, Y=0, Z=0-apply
Enter node no=3, X=0.013, Y=0, Z=0-apply
Enter node no=4, X=0.088, Y=0, Z=0-apply
Enter node no=5, X=0.108, Y=0, Z=0-apply
Enter node no=6, X=0.128, Y=0, Z=0-apply
Enter node no=7, X=0.129, Y=0, Z=0-ok

Modeling-create-element-element attributes
Enter element type no=2 LINK 34 (convection)
Material no=1 (convection or film coefficient)
Real constant set no=2 (convection)-ok
Modeling-create-element-auto numbered-through node-pick the nodes 1 & 2-ok

Modeling-create-element-element attributes
Enter element type no=1 LINK 33 (Conduction)
Material no=2 (conduction)
Real constant set no=1 (conduction)-ok
Modeling-create-element-auto numbered-through node-pick the nodes 2 & 3-ok

Modeling-create-element-element attributes
Enter element type no=1 LINK 33 (Conduction)
Material no=3 (conduction)
Real constant set no=1 (conduction)-ok
Modeling-create-element-auto numbered-through node-pick the nodes 3 & 4-ok

Modeling-create-element-element attributes
Enter element type no=1 LINK 33 (Conduction)
Material no=4 (conduction)
Real constant set no=1 (conduction)-ok
Modeling-create-element-auto numbered-through node-pick the nodes 4 & 5-ok.

Modeling-create-element-element attributes
Enter element type no=1 LINK 33 (Conduction)
Material no=5 (convection or film coefficient)
Real constant set no=1 (conduction)-ok
Modeling-create-element-auto numbered-through node-pick the nodes 5 & 6-ok.

Modeling-create-element-element attributes
Enter element type no=2 LINK 34 (convection)
Material no=6 (convection or film coefficient)
Real constant set no=2 (convection)-ok
Modeling-create-element-auto numbered-through node-pick the nodes 6 & 7-ok

Observe the straight line.
From the menu bar select plot controls-Numbering-Plot numbering control and select
element/attributes numbering=element no and don’t change other attributes-ok

   • Solution-define loads-apply-thermal-temperature-on nodes-pick the first nodes-ok-
     temperature-load-temperature value=20\(^\circ\) C-apply.
   • Define load-apply-thermal-temperature-on nodes-pick the last node-ok, select
temperature-load temperature value=-10\(^\circ\) C-ok.
   • Solution- solve-current LS-ok.
     Solution is done-close.
7. Read results-last set-ok
8. List results-nodal solution-select temperature-ok
9. Observe the nodal solution per node.
10. 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.
11. Plot results-contour plot-nodal solution-temperature-deformed shape only-ok
12. Element table-define table-add-enter user label item=HTRANS, select by sequence no
    SMISC, 1-ok-close.
13. Element table-list table-select HTRANS-ok
RESULT:

Analytical approach:

Calculation:

---

**ANSYS Results:**

<table>
<thead>
<tr>
<th>Nodal temperature</th>
<th>ANSYS</th>
<th>Theoretical</th>
</tr>
</thead>
<tbody>
<tr>
<td>( T_1 )</td>
<td></td>
<td></td>
</tr>
<tr>
<td>( T_2 )</td>
<td></td>
<td></td>
</tr>
<tr>
<td>( T_3 )</td>
<td></td>
<td></td>
</tr>
<tr>
<td>( T_4 )</td>
<td></td>
<td></td>
</tr>
<tr>
<td>( T_5 )</td>
<td></td>
<td></td>
</tr>
<tr>
<td>( T_6 )</td>
<td></td>
<td></td>
</tr>
<tr>
<td>( T_7 )</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Heat flux
Problem 5.4: A plane wall ‘X’ (K=75W/mK) is 60 mm thick and has volumetric heat generation of 1.5x10^6 W/m^3. It is insulated on one side while the other side is in contact with the surface of another wall ‘Y’ (K=150W/mK) which is 30mm thick and has no heat generation. The free surface of wall ‘Y’ is exposed to a cooling fluid at 20^0C with a convection coefficient of 950 W/m^2K. Find steady state temperatures at salient points across the composite wall. Area= 1m^2.

1. Preferences-Thermal-h method-ok
2. Element type-add/edit/delete- add-solid, quad 4node 55-enter reference number=1-select options-element behaviour,K_3= plane thickness-ok
3. Real constants- add/edit/delete-add enter real constant set no. 1, thickness=2-ok-close
4. Material properties-Material models-material number 1-thermal-conductivity-isotropic-KXX=75-ok.
5. From the menu bar select material-new model-enter material no. ID 2=2-select
6. Material model no.2- thermal-conductivity-isotropic-KXX=150-ok.
7. Modeling-create-areas-rectangles-by dimensions-X_1=0, X_2=0.06, Y_1=0,Y_2=0.03-apply- X_1=0.06, X_2=0.09, Y_1=0,Y_2=0.03-OK
   Modeling-operate-Boolean-glue-areas-pick the material-ok
8. Meshing-size controls-manual size-picked lines-pick the first vertical line, middle line and the last vertical line-ok-number of element divisions=2-apply
   Meshing-size controls-manual size-picked lines-pick the first rectangle top and bottom lines-number of element divisions=60-apply
   Meshing-size controls-manual size-picked lines-pick the top and bottom lines of second rectangle-number of element divisions=30-ok.
   Meshing-areas- free-pick all-ok.
9. Solution-Analysis type-new analysis-steady state-ok
   ➢ define loads-apply-thermal-heat generated on areas-pick the first rectangle-ok
   ➢ define loads-apply-thermal-heat generated on areas-apply Hgen on areas as constant value- load Hgen value=1.5e6
   ➢ define loads-apply-thermal-convection-on lines-pick the back corner line-ok-enter
   film coefficient=950-bulk temperature=20^0C (Don’t change other attributes)-ok
define loads-apply-thermal-heat flux-on lines-pick the front corner line-ok-enter heat flux=0-ok
10. Solution-solve-current LS-ok-close
11. Read results-last set-ok
12. List results-nodal solution-select temperature-ok
13. Observe the nodal solution per node.
14. 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.
15. Plot results-contour plot-nodal solution-temperature-deformed shape only-ok

RESULT:

Analytical approach:

Calculation:

<table>
<thead>
<tr>
<th>Nodal temperature</th>
<th>ANSYS</th>
<th>Theoretical</th>
</tr>
</thead>
<tbody>
<tr>
<td>T₁</td>
<td></td>
<td></td>
</tr>
<tr>
<td>T₂</td>
<td></td>
<td></td>
</tr>
<tr>
<td>T₃</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Problem 6.1: Modal Analysis of Cantilever beam for natural frequency determination. Modulus of elasticity = 200GPa, Density = 7800 Kg/m3.
RESULT:

Analytical solution:

ANSYS Results:
Problem 6.2: Fixed- fixed beam subjected to forcing function
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 = 200GPa, Poisson’s ratio = 0.3, Density = 7800 Kg/m3.

1. Ansys Main Menu – Preferences-select – STRUCTURAL- h method – ok
4. Sections-Beams-common sections- sub type- rectangle (1st element) - enter b=100, h=100- preview-ok.
5. Modeling – Create – Keypoints – in Active CS – x,y,z locations – 0,0 – apply – x,y,z locations –1.0 – ok (Keypoints created).
13. Solve – current LS – ok (Solution is done is displayed) – close.
14. TimeHistPostpro
   Select ‘Add’ (the green ‘+’ sign in the upper left corner) from this window – Nodal solution -DOF solution – Y component of Displacement – ok. Graphically select node 2 – ok.
   Select ‘List Data’ (3 buttons to the left of ‘Add’) from the window.
   Time History Variables’ window click the ‘Plot’ button, (2 buttons to the left of ‘Add’)
   This is the response at node 2 for the cyclic load applied at this node from 0 - 100 Hz.
RESULT:
PART – C (only for demo and oral exam)

Demonstrate the use of graphics standards (IGES, STEP etc) to import the model from modeler to solver.

The actual dimensions are referred to as 2D and 3D in a computer workspace. 2D is flat and uses the horizontal and vertical (X and Y) dimensions. Since the image has only two dimensions, when turning the image into one side, it becomes a flat image. On the other way, 3D adds the depth (Z) dimension that allows rotation and visualization from multiple perspectives. This shows how different a photo and a sculpture are.

3D modelling is widely used in different industries like films, animation and gaming, interior designing and architecture. Besides these, it is used in the medical industry for interactive representations of anatomy. In constructing digital representations of mechanical models or parts, the large numbers of 3D software are utilized before they are actually manufactured. With the software used in such fields as CAD/CAM related software, not only the mechanical models are constructed but also assembled and functionality is observed.

Moreover, 3D modelling is utilized in the industrial design in which 3D products are modelled before being represented to the customers. 3D modelling is used in media and event industries for the stage or set design.

Different programs support different functionalities and different tools are used for different types of projects. Hence, the models are shared and transferred between different programs for different stages of the design. The various 3D programs have been launched and used in the media industry. Hence, exchanging files between them is where the importing and exporting menu commands come in. These command options can be found in the file menu as it shows in figure below.

![Import command Menu](image-url)
The importing and exporting commands allow users to share 3D geometry with other 3D programs. There are many 3D programs used in the world nowadays. Depending on the purpose the software is chosen to meet the requirement. Autodesk 3ds Max is one of the programs widely used in the media and entertainment industry all over the world. It is also utilized in media education institutions such as, Metropolia US. It was also used in the project that this thesis describes. There are several file formats that can be imported and exported into 3ds Max. All acceptable files are automatically displayed in the file dialog box when selecting the import option.

The available import formats include the following: • Autodesk (FBX) • 3D Studio Mesh, Project and Shapes (3DS, PRJ; SHP) • Adobe Illustrator (AI) • Collada (DAE) • Land XML/DEM/DDF • AutoCAD and Legacy AutoCAD (DWG, DXF) • Flight Studio Open Flight (FLT) • Motion Analysis (HTR, TRC) • Initial Graphics Exchange Standard (IGE, IGS, IGES) • Autodesk Inventor (IPT, WIRE, IAM) • Lights cape (LS, VW, LP) • OBJ Material and Object (OBJ) • ACIS SAT (SAT) • Google Sketch Up (SKP) • Stereo Litho (STL) • VIZ Material XML Import (XML) • STEP (STP, STEP) • Rhino (3DM).

Many available options can be imported into 3ds Max, but not all of them are mentioned in this thesis. In the thesis, only those formats were utilized to import a 3D model for the project of the case study are analyzed. STEP, IGES and 3DS are file formats used to export the model. Though those format files are exported from the same CAD file, two of them are not compatible. In the case study, the model designed by the CAD software is utilized mostly in industries. The model cannot be used directly because it lacks textures and materials. In order for it to work in video commercials, it needs to be transferred to other software for more designs. Moreover, to give more options for using the model with fewer errors and the best result, the model is exported into three different formats IGES, STEP and 3DS.

**IGES File**

IGES (Initial Graphics Exchange Specification) was the first specification for CAD data exchange published in 1980 as a NBS (National Bureau of Standards) report in USA. IGES was originally created for exchanging the drafting data like 2D/3D wireframe models, text, dimensioning data, and a limited class of surfaces. Due to developing and requiring of users, IGES has been ongoing improvement and now it can support more capabilities such as entities, syntax, clarity and consistency.

The IGES specification describes the file format, language format, and the product definition data. In the product definition, geometric, topological, and non-geometric data are included. The geometric entities used to define the geometry are determined in the geometry part. The topology part describes the entities that identify the relationship between the geometric entities. There are three divisions such as annotation, definition, and organization in the non-geometric part. The annotation category contains dimensions, drafting notations and text. With the definition category, users can define specific properties of individuality or collections of entities.

The organization category describes groupings of geometry, annotation, or property elements. An IGES file has six sections, which are Flag, Start, Global, Directory Entry,
Parameter Data, and Terminate. A directory entry and parameter data entry are included in each entity instance. The directory entry has an index and attributes to describe the data. The parameter data defines the specific entity and is defined by fixed length records in accordance with the corresponding entity.

The size of the IGES files and the processing time are practical problems. As IGES files are composed of fixed format records and in both the directory entry section and the parameter data section each entry has to have record, errors occur in pre-and post-processor implementations.

**STEP File**

STEP (Standard for the Exchange of Product model data) is a new international standard (ISO 10303) for representing and exchanging product model information including an object-flavored data specification language called EXPRESS. STEP also describes implementation methods, for example, a physical transfer files, and offers different resources such as geometric and topological representation.

STEP was developed in 1984 as a worldwide collaboration with the purpose of defining a standard to cover all aspects of a product during its lifetime. It is a collection of standards to represent and exchange product information. While the main parts of STEP are already international standards, many parts remain to be under development. The development is performed under the control of the International Standards Organization (ISO), Technical Committee 184 (TC184, Industrial Automation Systems) and Subcommittee 4 (SC4, Industrial Data and Global Manufacturing Programming Languages).

STEP aims to offer system-independent mechanisms describe the product information in computer aided systems throughout its lifetime. It separates the representation of product information from the implementation methods that are used for data exchange. A basis for archiving product information and a methodology for the conformance testing of implementations are provided by STEP.

EXPRESS is used to specify the representation of product information. This facilitates development of implementation and enables consistency of representation. STEP does not only define the geometric sharpness of a product, but also includes topology, features, tolerance specifications, material properties, so that the goal of the design, manufacturing, testing, inspection and support of the product is completely defined. STEP covers the total product life cycle in terms of sharing, storage and exchange. It is said that STEP is the most important effort ever established in engineering and will take the position of current CAD exchange standards.

STEP is well known and widely used as an exchange data form. It is supported by many software tools such as ECAD or EDA (Altium Designer, Circuit Studio, Circuit Maker, Cadence, etc.), MCAD (IDA step and express Engine), Dassault Systemes (Catia, Solid Works and 3DVIA shape), PTC and Autodesk.
Demonstrate one example of contact analysis to learn the procedure to carry out contact analysis.

Contact stresses occur in the transmission of forces between two parts with punctiform contact: roller bearings or ball bearings, toothed gears, cam mechanisms, etc.

On basis of the general equations of the theory of elasticity, H. Hertz (1881-1882) laid the foundations of the theory of stresses and deformations of elastic bodies in contact points.

In the general case of two bodies in contact subjected to a perpendicular force on the contact surfaces the outline of the contact surface is an ellipsis, which transforms into a circular surface or a strip in extreme situations (the case of linear contact can be considered a particular case of the punctiform contact).

The distribution of the stress on the contact surface is given by the ordinates of the ellipsoid built on the contact surface with the maximum stress at the centre of the ellipsis (Fig.1).

![Fig. 1. Contact between two bodies [1]](image)

The value of the maximum contact stress in the center of the contact ellipsis is given by the relation [2]:

$$\sigma_{\text{max}} = 1.5 \frac{P}{\pi \cdot a \cdot b}$$

(1)

where:
- “a” and “b” are geometrical parameters, respectively, curvature radii of the two bodies in contact.
- The material in the vicinity of the contact ellipsis is compressed in all directions approximately identically and can withstand to high stresses without residual deformations. Thus, experiments show that for special steels for rolling bodies in punctiform contact in use, allowable value of the maximum stress is:

$$\sigma_a = 3500-5000 \text{ MPa}$$  – for steel

![Fig. 3. (a) Modeling and meshing contact between two spheres (Algor Screen Capture); (b) Modeling and meshing contact between the spheres on the flat surface (Algor Screen Capture);](image)
Demonstrate at least two different types of example to model and analyze bars or plates made from composite material.

1. \( A = 160 \text{ MM}, \ B = 80 \text{ MM} \)

![FEM Model](image)

**Figure 1** Composite profile dimensions and exemplary ply sequence

![FEM Model](image)

**Fig. FEM Model**
2.

**Fig.** Composite plate & Ripper
ADDITIONAL PROBLEMS

1. Calculate the stresses and displacement for the plate shown below. Let the load be $P = 100\text{N}$ applied at equal distance from both ends and $E = 3 \times 10^7 \text{N/mm}^2$.

![Plate Diagram]

2. Current passes through a stainless steel wire of 2.5 mm diameter ($k=200 \text{ W/mK}$) causing volumetric heat generation of $26.14 \times 10^8 \text{ W/m}^3$. The wire is submerged in a fluid maintained at $500 \text{ C}$ and convective heat transfer coefficient at the wire surface is $4000 \text{ W/m}^2 \text{K}$. Find the steady state temperature at the centre and at the surface of the wire.

3. Calculate the maximum value of Von-misses stresses in the stepped beam with a rounded plate as shown in the figure. Where Young’s modulus, $E=210 \text{Gpa}$, Poisson’s ratio is 0.3 and the beam thickness is $10\text{mm}$, the element size is $2\text{mm}$.

![Beam Diagram]

4. Loads will not be applied to the beam shown below in order to observe the deflection caused by the weight of the beam itself. The beam is to be made of steel with a Young’s modulus of elasticity of $200 \text{ Gpa}$.
5. A beam has a force of 1000N applied as shown below. The purpose of this optimization problem is to minimize the weight of the beam without exceeding the allowable stress. It is necessary to find the cross-sectional dimensions of the beam in order to minimize the weight of the beam. However, the width and height of the beam cannot be smaller than 10mm. The maximum stress anywhere in the beam cannot exceed 200 MPa. The beam is to be made of steel with a modulus of elasticity of 200 GPa.
VIVA QUESTIONS & SOLUTIONS

1. Theories of failure.
   a. Maximum Principal Stress Theory- A material in complex state of stress fails, when the maximum principal stress in it reaches the value of stress at elastic limit in simple tension.
   b. Maximum Shear Stress Theory- A material in complex state of stress fails when the maximum shearing stress in it reaches the value of shearing stress at elastic limit in uniaxial tension test.
   c. Maximum Principal Strain Theory- Failure in a complex system occurs when the maximum strain in it reaches the value of the strain in uniaxial stress at elastic limit.
   d. Maximum Strain Energy Theory- A material in complex state of stress fails when the maximum strain energy per unit volume at a point reaches the value of strain energy per unit volume at elastic limit in simple tension test.
   e. Maximum Distortion Energy Theory- This theory is also known as Von-Mises criteria for failure of elastic bodies. According to this theory part of strain energy causes only changes in volume of the material and rest of it causes distortion. At failure the energy causing distortion per unit volume is equal to the distortion energy per unit volume in uniaxial state of stress at elastic limit.

2. What is factor of safety?
   The maximum stress to which any member is designed is much less than the ultimate stress and this stress is called working stress. The ratio of ultimate stress to working stress is called factor of safety.

3. What is Endurance limit?
   The max stress at which even a billion reversal of stress cannot cause failure of the material is called endurance limit.

4. Define: Modulus of rigidity, Bulk modulus
   Modulus of rigidity: It is defined as the ratio of shearing stress to shearing strain within elastic limit.
   Bulk modulus: It is defined as the ratio of identical pressure ‘p’ acting in three mutually perpendicular directions to corresponding volumetric strain.

5. What is proof resilience?
   The maximum strain energy which can be stored by a body without undergoing permanent deformation is called proof resilience.
6. What is shear force diagram?
   A diagram in which ordinate represent shear force and abscissa represents the position of the section is called SFD.

7. What is bending moment diagram?
   A diagram in which ordinate represents bending moment and abscissa represents the position of the section is called BMD.

   a. The beam is initially straight and every layer of it is free to expand or contract.
   b. The material is homogeneous and isotropic.
   c. Young’s modulus is same in tension and compression.
   d. Stresses are within elastic limit.
   e. Plane section remains plane even after bending.
   f. The radius of curvature is large compared to depth of beam.

9. State the three phases of finite element method.
   Preprocessing, Analysis & Post processing

10. What are the h and p versions of finite element method?
    Both are used to improve the accuracy of the finite element method. In h version, the order of polynomial approximation for all elements is kept constant and the numbers of elements are increased. In p version, the numbers of elements are maintained constant and the order of polynomial approximation of element is increased.

11. What is the difference between static analysis and dynamic analysis?
    Static analysis: The solution of the problem does not vary with time is known as static analysis. E.g.: stress analysis on a beam.
    Dynamic analysis: The solution of the problem varies with time is known as dynamic analysis. E.g.: vibration analysis problem.

12. What are Global coordinates?
    The points in the entire structure are defined using coordinates system is known as global coordinate system.

13. What are natural coordinates?
    A natural coordinate system is used to define any point inside the element by a set of dimensionless number whose magnitude never exceeds unity. This system is very useful in assembling of stiffness matrices.
14. What is a CST element?

Three node triangular elements are known as constant strain triangular element. It has 6 unknown degrees of freedom called u1, v1, u2, v2, u3, v3. The element is called CST because it has constant strain throughout it.

15. Define shape function.

In finite element method, field variables within an element are generally expressed by the following approximate relation:

\[ \Phi (x,y) = N_1(x,y) \Phi_1 + N_2(x,y) \Phi_2 + N_3(x,y) \Phi_3 + N_4(x,y) \Phi_4 \]

where \( \Phi_1, \Phi_2, \Phi_3 \) and \( \Phi_4 \) are the values of the field variables at the nodes and \( N_1, N_2, N_3 \) and \( N_4 \) are interpolation function. \( N_1, N_2, N_3, N_4 \) are called shape functions because they are used to express the geometry or shape of the element.

16. What are the characteristics of shape function?

The characteristics of the shape functions are as follows:

- The shape function has unit value at one nodal point and zero value at the other nodes.
- The sum of shape functions is equal to one.

17. Why polynomials are generally used as shape function?

- Differentiation and integration of polynomials are quite easy.
- The accuracy of the results can be improved by increasing the order of the polynomial.
- It is easy to formulate and computerize the finite element equations.

18. State the properties of a stiffness matrix.

The properties of the stiffness matrix \([K]\) are:

- It is a symmetric matrix.
- The sum of the elements in any column must be equal to zero.
- It is an unstable element, so the determinant is equal to zero.

19. What are the difference between boundary value problem and initial value problem?

The solution of differential equation obtained for physical problems which satisfies some specified conditions known as boundary conditions. If the solution of differential equation is obtained together with initial conditions then it is known as initial value problem. If the solution of differential equation is obtained together with boundary conditions then it is known as boundary value problem.
20. What is meant by plane stress?
   Plane stress is defined as a state of stress in which the normal stress ($\sigma$) and the shear stress directed perpendicular to plane are zero.

21. Define plane strain.
   Plane strain is defined to be a state of strain in which the strain normal to the xy plane and the shear strains are assumed to be zero.

22. Define Quasi-static response.
   When the excitations are varying slowly with time then it is called quasi-static response.

23. What is a sub parametric element?
   If the number of nodes used for defining the geometry is less than the number of nodes used for defining the displacements is known as sub parametric element.

24. What is a super parametric element?
   If the number of nodes used for defining the geometry is more than the number of nodes used for defining the displacements is known as sub parametric element.

25. What is meant by isoparametric element?
   If the number of nodes used for defining the geometry is same as number of nodes used for defining the displacements then it is called parametric element.

26. What is the purpose of isoparametric element?
   It is difficult to represent the curved boundaries by straight edges finite elements. A large number of finite elements may be used to obtain reasonable resemblance between original body and assemblage. In order to overcome this drawback, iso parametric elements are used i.e for problems involving curved boundaries, a family of elements ‘isoparametric elements’ are used.

27. What are isotropic and orthotropic materials?
   A material is isotropic if its mechanical and thermal properties are the same in all directions. Isotropic materials can have homogeneous or non-homogeneous microscopic structures.
   Orthotropic materials: A material is orthotropic if its mechanical or thermal properties are unique and independent in three mutually perpendicular directions.

28. What is discretization?
   Discretization is the process of dividing given problem into several small elements, connected with nodes.
29. Steps in FEM

- Discretization
- Selection of the displacement models
- Deriving element stiffness matrices
- Assembly of overall equations/ matrices
- Solution for unknown displacements
- Computations for the strains/stresses