Channabasaveshwara Institute of Technology
(An ISO 9001:2008 Certified Institution)
NH 206 (B.H. Road), Gubbi, Tumkur – 572 216. Karnataka.

Department of Mechanical Engineering

```
LAB MANUAL
(2015–2016)
```

COMPUTER AIDED MODELING AND ANALYSIS LAB
(CAMA LAB-10MEL68)

```
VI SEMESTER
```

NAME: _____

USN   : _____

BATCH: _____SECTION:_____

CONTENTS

INDEX PAGE

Sl. No	Name of the Experiment	Date			Manual Marks (Max . 25)	Record Marks (Max. 10)	Signature (Student)	Signature (Faculty)
		Conduction	Repetition	Submission of Record				
Average								

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.**

Channabasaveshwara Institute of Technology
(An ISO 9001:2008 Certified Institution)
NH 206 (B.H. Road), Gubbi, Tumkur – 572 216. Karnataka.

DEPARTMENT OF MECHANICAL ENGINEERING

SYLLABUS

COMPUTER AIDED MODELING AND ANALYSIS LAB

Sub Code: 10MEL68	**IA Marks : 25**
Hrs/ Week: 03	**Exam Hours: 03**
Total Hrs. 42	**Exam Marks: 50**

PART - A
Study of a FEA package and modeling stress analysis of

a. Bars of constant cross section area, tapered cross section area and stepped bar **6 Hours**

b. Trusses – (Minimum 2 exercises) **3 Hours**

c. Beams – Simply supported, cantilever, beams with UDL, beams with varying load etc

(Minimum 6 exercises) **12 Hours**

PART - B

a) Stress analysis of a rectangular plate with a circular hole. **3 Hours**

b) Thermal Analysis – 1D & 2D problem with conduction and convection boundary conditions. (Minimum 4 exercises) **9 Hours**

c) Dynamic Analysis

1) Fixed – fixed beam for natural frequency determination

2) Bar subjected to forcing function

3) Fixed – fixed beam subjected to forcing function **9 Hours**

REFERENCE BOOKS:

1. **A first course in the Finite element method,** Daryl L Logan, Thomason, Third Edition

2. **Fundaments of FEM,** Hutton – McGraw Hill, 2004

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

Scheme for Examination:

One Question from Part A - 20 Marks (05 Write up +15)

One Question from Part B - 20 Marks (05 Write up +15)

Viva-Voce - 10 Marks

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 $2.1 \times 10^5 \text{N/mm}^2$ and Area is 500mm^2. Determine the Nodal Displacement, Stress in each element, Reaction forces.
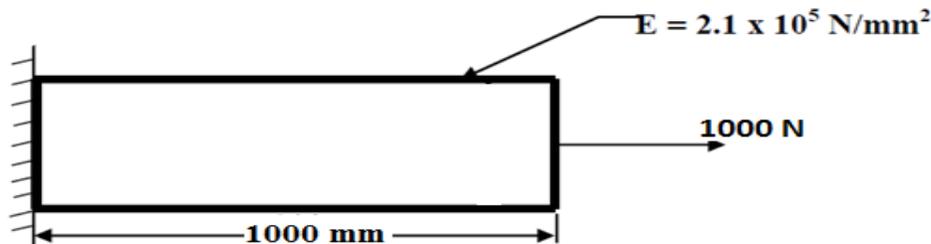
$E = 2.1 \times 10^5 \text{ N/mm}^2$

1000 N

1000 mm

1. Ansys Main Menu – Preferences-Select – STRUCTURAL – h method- ok
2. Element type – Add/Edit/Delete – Add – Link – 3D Finit stn 180 – ok – close.
3. Real constants – Add – ok – real constant set no 1 – c/s area – 500 – ok.
4. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 2.1e5 – PRXY – 0.27 – ok – close.
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).
7. Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 – apply –DOFs to be constrained – All DOF – ok.Loads –
8. Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply – direction of For/Mom – FX – Force/Moment value – 1000 (+ve value) – ok.
9. Solve – current LS – ok (Solution is done is displayed) – close.
10. Element table – Define table – Add –'Results data item' – By Sequence num – LS – LS1 – ok.
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.
14. Animation: PlotCtrls – Animate – Deformed shape – def+undeformed-ok.

RESULT:
Analytical approach:

Calculation:
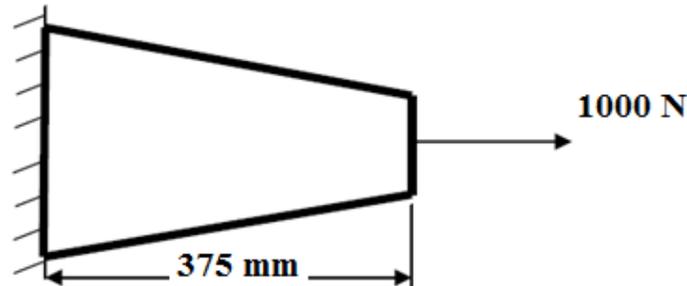
Displacement: _____

Stress: _____

Reaction force: _____

Ansys results:

	Ansys	Theoretical
Deformation		
Stress		
Reaction		

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

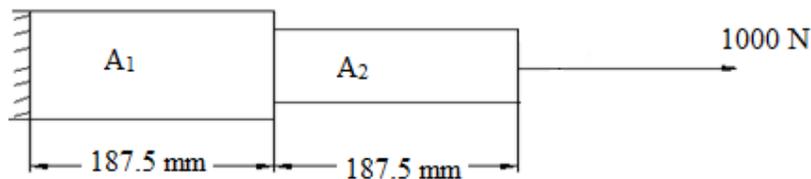$E = 2 \times 10^5$ N/mm^2, Area at root, $A_1 = 1000$ mm^2, Area at the end, $A_2 = 500$ mm^2.

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

$(A_1 + A_2)/2 = (1000+500)/2 = 750$ mm^2

$A_1 = (1000+750)/2 = 875$ mm^2

$A_2 = (500+750)/2 = 625$ mm^2

$L_1 = 187.5$ mm & $L_2 = 187.5$ mm

1. Ansys Main Menu – Preferences-Select – STRUCTURAL- h method– ok
2. Element type – Add/Edit/Delete – Add – link, 3D Finit stn 180 – ok- close.
3. Real constants – Add – ok – real constant set no – 1 – cross-sectional AREA1 – 875 – apply-ok
4. Add – ok – real constant set no – 2 – cross-sectional AREA 2 – 625-ok
5. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 2e5 –PRXY – 0.3 – ok – close.
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.
12. Loads – Define loads – apply – Structural – Force/Moment – on key points- pick last key point – apply – direction of For/Mom – FX – Force/Moment value – 1000 (+ve value) – ok.
13. Solve – current LS – ok (Solution is done is displayed) – close.
14. Element table – Define table – Add –'Results data item' – By Sequence num – LS – LS1 – ok.
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.
18. Animation: PlotCtrls – Animate – Deformed shape – def+undeformed-ok.

RESULT:

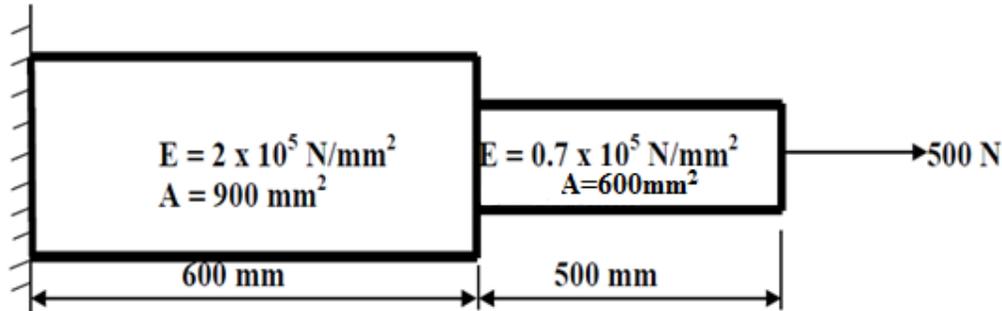### Analytical approach:

Calculation:

Displacement: _____

Stress: _____

Reaction force: _____

Ansys results:

	Ansys	Theoretical
Deformation		
Stress		
Reaction		

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
2. Element type – Add/Edit/Delete – Add – link, 3D Finit stn 180 – ok- close.
3. Real constants – Add – ok – real constant set no – 1 – cross-sectional AREA 1 – 900 – apply-ok
4. Add – ok – real constant set no – 2 – cross-sectional AREA 2 – 600-ok
5. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 2e5 –PRXY – 0.3- material- new material-define material id=2- Structural – Linear – Elastic – Isotropic – EX – 0.7e5 –PRXY – 0.3– ok – close.
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-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.
12. Loads – Define loads – apply – Structural – Force/Moment – on key points- pick last key point – apply – direction of For/Mom – FX – Force/Moment value – 500 (+ve value) – ok.
13. Solve – current LS – ok (Solution is done is displayed) – close.
14. Element table – Define table – Add –'Results data item' – By Sequence num – LS – LS1 – ok.
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.
18. Animation: PlotCtrls – Animate – Deformed shape – def+undeformed-ok.

RESULT:

Analytical approach:
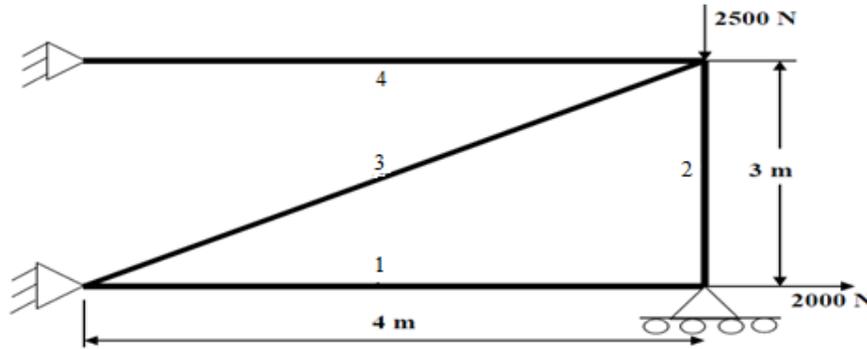
Calculation:

Displacement: _____

Stress in each element:_____

Reaction force: _____

Ansys results:

	Ansys	Theoretical
Deformation		
Stress		
Reaction		

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 m^2.

1. Ansys Main Menu – Preferences-select – STRUCTURAL- h method – ok
2. Element type – Add/Edit/Delete – Add – Link – 3D Finit stn 180 – ok – close.
3. Real constants – Add – ok – real constant set no – 1 – c/s area – 0.1 – ok – close.
4. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 210e9– Ok – close.
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
7. 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).
8. 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.
9. 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 – Structural –
10. Force/Moment – on Nodes- pick node 3 – apply – direction of For/Mom – FY – Force/Moment value – -2500 (-ve value) – ok.
11. Solve – current LS – ok (Solution is done is displayed) – close.
12. Element table – Define table – Add –'Results data item' – By Sequence num – LS – LS1 – ok.
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.
16. Animation: PlotCtrls – Animate – Deformed shape – def+undeformed-ok.

RESULT:

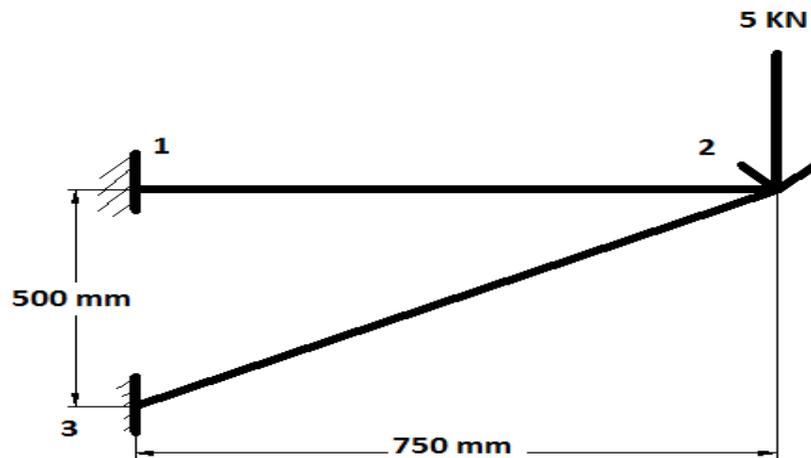**Analytical approach:**

Calculation:

Displacement: _____

Stress: _____

Reaction force: _____

Ansys results:

	Ansys	Theoretical
Deformation		
Stress		
Reaction		

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 m².

1. Ansys Main Menu – Preferences-select – STRUCTURAL- h method – ok
2. Element type – Add/Edit/Delete – Add – Link – 3D Finit stn 180 – ok – close.
3. Real constants – Add – ok – real constant set no – 1 – c/s area – 0.1 – ok – close.
4. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 210e9– Ok – close.
5. Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS– 0.75 (x value w.r.t first node) – apply (second node is created) – x,y,z location in CS –(0, -0.5),(x, y value w.r.t first node) – ok (third node is created
6. Create–Elements–Elem Attributes – Material number – 1 – Real constant set number – 1 – ok
7. Auto numbered – Thru Nodes – pick 1 & 2 – apply – pick 2 & 3— ok (elements are created through nodes).
8. Loads – Define loads – apply – Structural – Displacement – on Nodes – pick node 1 &3 – apply – DOFs to be constrained – All DOF – ok
9. Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply – direction of For/Mom – FY – Force/Moment value – 5000 (-ve value)
10. Solve – current LS – ok (Solution is done is displayed) – close.
11. Element table – Define table – Add –'Results data item' – By Sequence num – LS – LS1 – ok.
12. Plot results – contour plot –Element table – item to be plotted LS,1, avg common nodes- yes average- ok.
13. List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).
14. Plot results- nodal solution-ok-DOF solution- Y component of displacement-ok.
15. Animation: PlotCtrls – Animate – Deformed shape – def+undeformed-ok.

RESULT:

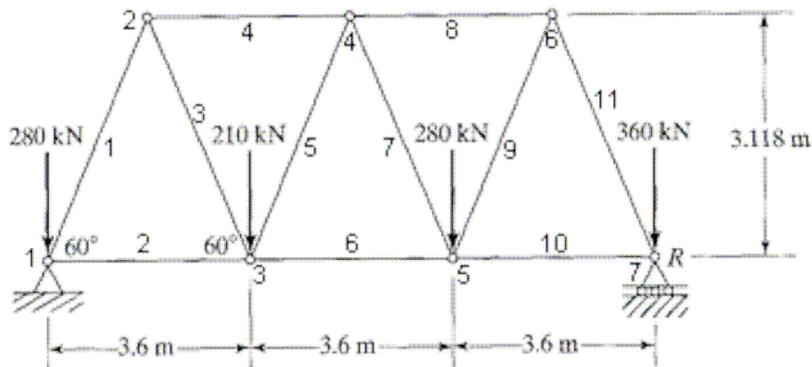**Analytical approach:**

Calculation:

Displacement: _____

Stress: _____

Reaction force: _____

Ansys results:

	Ansys	Theoretical
Deformation		
Stress		
Reaction		

Problem 2.3: Determine the nodal deflections, reaction forces, and stress for the truss system shown below(E = 200GPa, A = 3250mm2).
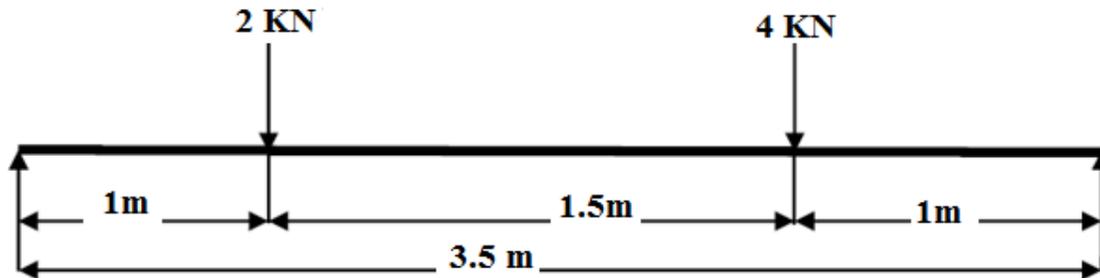

Ansys results:

	Ansys	Theoretical
Deformation		
Stress		
Reaction		

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
2. Element type – Add/Edit/Delete – Add – BEAM – 2 node BEAM 188– ok- close.
3. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 2.10e5– PRXY – 0.27 – ok – close.
4. Sections-Beams-common sections- sub type- rectangle (1st element) -enter b=100, h=100- preview-ok.
5. Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS– 1000 (x value w.r.t first node) – apply (second node is created) – 2500 (x value w.r.t first node) – apply(third node is created)- x,y,z location in CS-3500 (x value w.r.t first node)-ok.
6. Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 apply – pick 2 & 3 apply – pick 3 & 4 – ok (elements are created through nodes).
7. Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 & 4 – apply –DOFs to be constrained – all DOF – ok.
8. Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply –direction of For/Mom – FY – Force/Moment value – -2000(-ve value) – ok- Force/Moment – on Nodes- pick node 3 – apply –direction of For/Mom – FY – Force/Moment value – -4000(-ve value) – ok.
9. Solve – current LS – ok (Solution is done is displayed) – close.
10. Displacement: Plot Results – Contour plot – Nodal solution – DOF solution – displacement vector sum – ok.
11. Stress: Plot Results – Contour plot – Nodal solution – stress – vonmises stress – ok.
12. Element table – Define table – Add – 'Results data item' – By Sequence num – SMISC –SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 19 – apply, By Sequence num –SMISC – SMISC, 3 – apply, By Sequence num – SMISC – SMISC, 16 – ok – close.

13. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS19 – ok (Shear force diagram will be displayed).
14. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS3 – Elem table item at node J – SMIS16 – ok (bending moment diagram will be displayed).
15. Reaction forces: List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).
❖ NOTE: For Shear Force Diagram use the combination SMISC 6 & SMISC 19, for Bending Moment Diagram use the combination SMISC 3 & SMISC 16.
16. Animation: PlotCtrls – Animate – Deformed results – DOF solution – USUM – ok.

RESULT:

Analytical approach:

Calculation:

Displacement: _____

Shear force: _____

Bending moment: _____

Stress:_____

Ansys results:

	Ansys	Theoretical
Deflection		
Shear force		
Bending moment		
Stress		

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 m, Young's modulus of 2.1×10^5 N/mm^2, Poisson's ratio= 0.27.
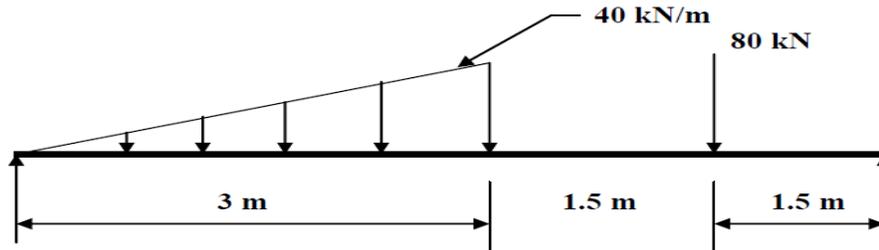
1. Ansys Main Menu – Preferences-Select – STRUCTURAL- h method- ok
2. Element type – Add/Edit/Delete – Add – BEAM – 2 nodes Beam 188 – ok – close.
3. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 2.1e5– PRXY – 0.27 –ok – close.
4. Sections-Beams-common sections- sub type- rectangle (1st element) - enter b=100, h=100- preview-ok.
5. Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS– 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).
6. Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – apply – pick 2 & 3 – apply –pick 3 & 4 – ok (elements are created through nodes).
7. Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 & 4 – apply –DOFs to be constrained – all DOF – ok.
8. Loads – Define loads – apply – Structural – Pressure – on Beams – pick element between nodes 1 & 2–apply–pressure value at node I– 0 (value)– pressure value at node J – 40000–ok.
9. Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 3 – apply – direction of For/Mom – FY – Force/Moment value – (-80000) (-ve value) – ok.
10. Solve – current LS – ok (Solution is done is displayed) – close.
11. Displacement: Plot Results – Contour plot – Nodal solution – DOF solution – displacement vector sum – ok.
12. Stress: Plot Results – Contour plot – Nodal solution – stress – von mises stress – ok.
13. Element table – Define table – Add – 'Results data item' – By Sequence num – SMISC –SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 19 – apply, By Sequence num –SMISC – SMISC, 3 – apply, By Sequence num – SMISC – SMISC, 16 – ok – close.
14. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS19 – ok (Shear force diagram will be displayed).

15. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS3 – Elem table item at node J – SMIS16 – ok (bending moment diagram will be displayed).
16. Reaction forces: List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).
17. Animation: PlotCtrls – Animate – Deformed results – DOF solution – deformed + undeformed – ok.

RESULT:

Analytical approach:

Calculation:

Deflection:_____

Shear force: _____

Bending moment:_____

Stress:_____

Ansys results:

	Ansys	Theoretical
Deflection		
Shear force		
Bending moment		
Stress		

Problem 3.3: Simply Supported Beam with Uniformally 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.
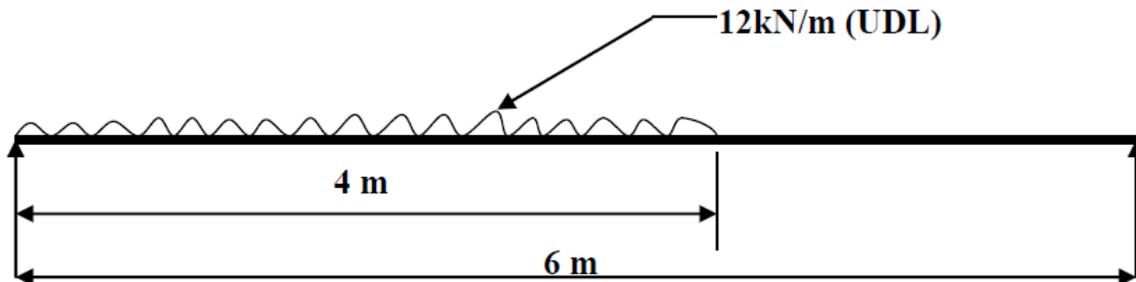
12kN/m (UDL)

4 m

6 m

1. Ansys Main Menu – Preferences-select – STRUCTURAL – ok
2. Element type – Add/Edit/Delete – Add – BEAM – 2 node 188 – ok- close.
3. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 210e9– PRXY – 0.27 –ok – close.
4. Sections-Beams-common sections- sub type- rectangle (1st element) - enter b=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).
6. Create – Nodes – Fill between Nds – pick 1 & 2 – apply – number of nodes to fill 7 – startingnode no – 4 – ok.
7. Create – Elements – Auto numbered – Thru Nodes – pick 1 & 4 apply– pick 4 & 5 apply– pick 5 & 6 apply– pick 6 & 7 apply– pick 7 & 8 apply– pick 8 & 9 apply – pick 9 & 10 apply– pick 10 & 2 apply – pick 2 & 3 – ok (elements are created through nodes).
8. Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 & 3 – apply – DOFs to be constrained – UY – ok.
9. Loads – Define loads – apply – Structural – Pressure – on Beams – pick all elements betweennodes 1 & 2 – apply – pressure value at node I – 12000 – pressure value at node J – 12000 –ok.
10. Solve – current LS – ok (Solution is done is displayed) – close.
11. Displacement: Plot Results – Contour plot – Nodal solution – DOF solution – displacement vector sum – ok.
12. Stress: Plot Results – Contour plot – Nodal solution – stress – von mises stress – ok.
13. Element table – Define table – Add – 'Results data item' – By Sequence num – SMISC –SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 19 – apply, By Sequence num –SMISC – SMISC, 3 – apply, By Sequence num – SMISC – SMISC, 16 – ok – close.
14. Plot results – contour plot – Line Element Results – Elem table item at node I – 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).

17. PlotCtrls – Animate – Deformed results – DOF solution – USUM – ok.

RESULT:

Analytical approach:

Calculation:

Deflection:_____
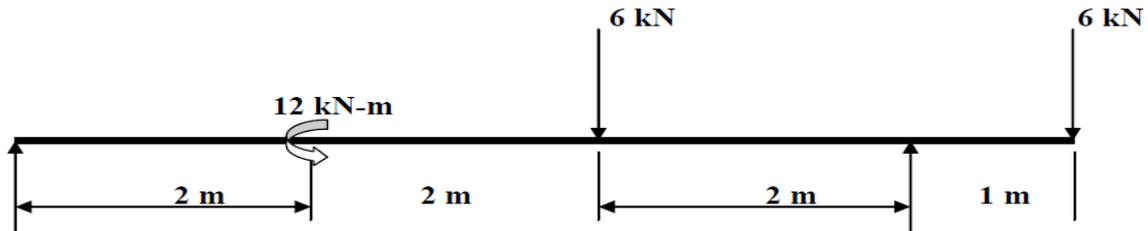
Shear force: _____

Bending moment: _____

Stress:_____

Ansys results:

	Ansys	Theoretical
Deflection		
Shear force		
Bending moment		
Stress		

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
2. Element type – Add/Edit/Delete – Add – BEAM – 2 node 188 – ok – close.
3. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 210e9– PRXY – 0.27 –ok – close.
4. Sections-Beams-common sections- sub type- rectangle (1st element) - enter b=200, h=300- preview-ok.
5. Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS– 2 (x value w.r.t first node) – 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).
6. Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – apply – pick 2 & 3 – apply –pick 3 & 4 – apply – pick 4 & 5 – ok (elements are created through nodes).
7. Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 & 4 –apply –DOFs to be constrained – UY – ok.
8. Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply direction of For/Mom – MZ – Force/Moment value - 12000 (anticlockwise, +ve value) – apply –pick node 3 – apply – direction of For/Mom – FY – Force/Moment value - -6000 (-ve value) –apply – pick node 5 – apply – direction of For/Mom – FY – Force/Moment value - -6000 (-ve value) – ok.
9. Solve – current LS – ok (Solution is done is displayed) – close.
10. Displacement: Plot Results – Contour plot – Nodal solution – DOF solution – displacement vector sum – ok.
11. Stress: Plot Results – Contour plot – Nodal solution – stress – von mises stress – ok.
12. Element table – Define table – Add – 'Results data item' – By Sequence num – SMISC –SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 19 – apply, By Sequence num –SMISC – SMISC, 3 – apply, By Sequence num – SMISC – SMISC, 16 – ok – close.
13. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS19 – ok (Shear force diagram will be displayed).
14. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS3 – Elem table item at node J – SMIS16 – ok (bending moment diagram will be displayed).

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

16. Animation: PlotCtrls – Animate – Deformed results – DOF solution – deformed + undeformed – ok.

RESULT:

Analytical approach:

Calculation:

Deflection: _____
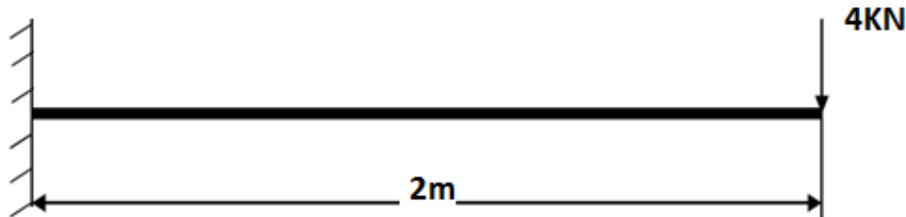
Shear force: _____

Bending moment:_____

Stress: _____

Ansys results:

	Ansys	Theoretical
Deflection		
SFD		
BMD		
Stress		

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
2. Element type – Add/Edit/Delete – Add – BEAM – 2 node Beam 188 – ok- close.
3. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 210e9– PRXY – 0.27 –ok – close.
4. Sections-Beams-common sections- sub type- rectangle (1ˢᵗ element) - enter b=200, h=300- preview-ok.
5. Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS– 2 (x value w.r.t first node) – ok (second node is created).
6. Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – ok (elements are created through nodes).
7. Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 – apply –
8. DOFs to be constrained – ALL DOF – ok.
9. Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply – direction of For/Mom – FY – Force/Moment value –(-40000) (-ve value) – ok.
10. Solve – current LS – ok (Solution is done is displayed) – close.
11. Displacement: Plot Results – Contour plot – Nodal solution – DOF solution – displacement vector sum – ok.
12. Stress: Plot Results – Contour plot – Nodal solution – stress – von mises stress – ok.
13. Element table – Define table – Add – 'Results data item' – By Sequence num – SMISC –SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 19 – apply, By Sequence num –SMISC – SMISC, 3 – apply, By Sequence num – SMISC – SMISC, 16 – ok – close.
14. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS19 – ok (Shear force diagram will be displayed).
15. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS3 – Elem table item at node J – SMIS16 – ok (bending moment diagram will be displayed).
16. Reaction forces: List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).

RESULT:

Analytical approach:

Calculation:

Deflection: _____
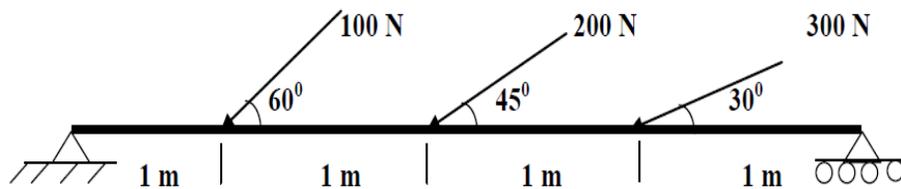
Shear force: _____

Bending moment: _____

Stress: _____

Ansys results:

	Ansys	Theoretical
Deflection		
SFD		
BMD		
Stress		

Problem 3.6: Beam with angular loads

Compute the Shear force and bending moment diagrams for the beam shown in fig such thatone 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 * 0.3 m, Young's modulus of 210 GPa, Poisson's ratio 0.27.

1. Ansys Main Menu – Preferencesselect – STRUCTURAL – ok
2. Element type – Add/Edit/Delete – Add – BEAM – 2node 188 – ok – close.
3. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 210e9– PRXY – 0.27 –ok – close.
4. Sections-Beams-common sections- sub type- rectangle (1st element) - enter b=200, h=300- preview-ok.
5. Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS– 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).
6. Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – apply – pick 2 & 3 – apply – pick 3 & 4 – apply – pick 4 & 5 – ok (elements are created through nodes).
7. Create – Nodes – Rotate nodes CS – by angles – pick node 2 – apply – about nodal z-axis – 60 – apply – pick node 3 – apply about nodal z- axis – 45 – apply – pick node 4 – apply – about nodal z –axis – 30 – ok.
8. Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 – apply –DOFs to be constrained – UX & UY – apply – pick node 5 – apply – DOFs to be constrained –UY – ok.
9. Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply direction of For/Mom – FX – Force/Moment value - -100 (-ve value) – apply – pick node 3 – apply – direction of For/Mom – FX – Force/Moment value - -200 (-ve value) – apply – pick node 4 – apply – direction of For/Mom – FX – Force/Moment value - 300 (-ve value) – ok.
10. Solve – current LS – ok (Solution is done is displayed) – close.
11. Displacement: Plot Results – Contour plot – Nodal solution – DOF solution – displacement vector sum – ok.
12. Stress: Plot Results – Contour plot – Nodal solution – stress – von mises stress – ok.
13. Element table – Define table – Add – 'Results data item' – By Sequence num – SMISC –SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 19 – apply, By Sequence num –SMISC – SMISC, 3 – apply, By Sequence num – SMISC – SMISC, 16 – ok – close.
14. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS19 – ok (Shear force diagram will be displayed).
15. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS3 – Elem table item at node J – SMIS16 – ok (bending moment diagram will be displayed).

RESULT:

Analytical approach:

Calculation:

Deflection:_____

Shear force: _____

Bending moment: _____

Stress:_____

Ansys results:

	Ansys	Theoretical
Deflection		
Shear force		
Bending moment		
Stress		

PART B

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 = 210GPa, t = 1 mm, Poisson's ratio = 0.3, Dia of the circle = 10 mm, Analysis assumption – plane stress with thickness is used.
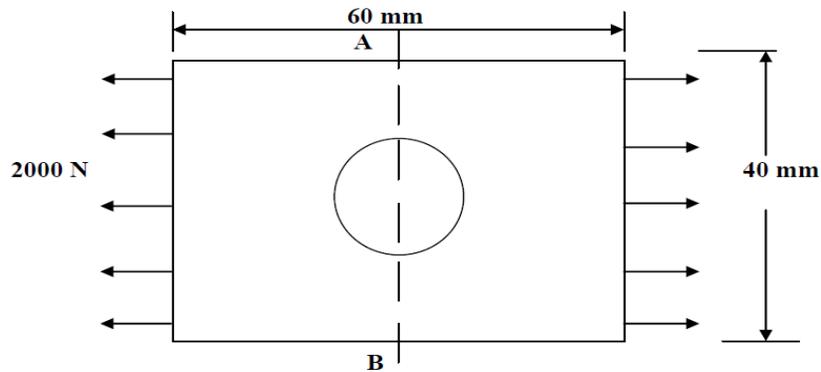
1. Ansys Main Menu – Preferences-Select – STRUCTURAL-h method – ok
2. Element type – Add/Edit/Delete – Add – Solid – Quad 4 node – 42 – ok – option – element behavior K3 – Plane stress with thickness – ok – close.
3. Real constants – Add – ok – real constant set no – 1 – Thickness – 1 – ok.
4. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 2.1e5 –PRXY – 0.3 – ok – close.
5. Modeling –Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 0, 60, 0, 40 – ok.
6. Create – Area – Circle – solid circle – X, Y, radius – 30, 20, 5 – 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.
8. Meshing – Mesh Tool – Mesh Areas – Quad – Free – Mesh – pick all – ok. Mesh Tool – Refine – pick all – Level of refinement – 3 – ok.
9. Loads – Define loads – apply – Structural – Displacement – on Nodes – select box – drag the left side of the area – apply – DOFs to be constrained – ALL DOF – ok.
10. Loads – Define loads – apply – Structural – Force/Moment – on Nodes – select box – drag the right side of the area – apply – direction of For/Mom – FX – Force/Moment value – 2000 (+ve value) – 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).
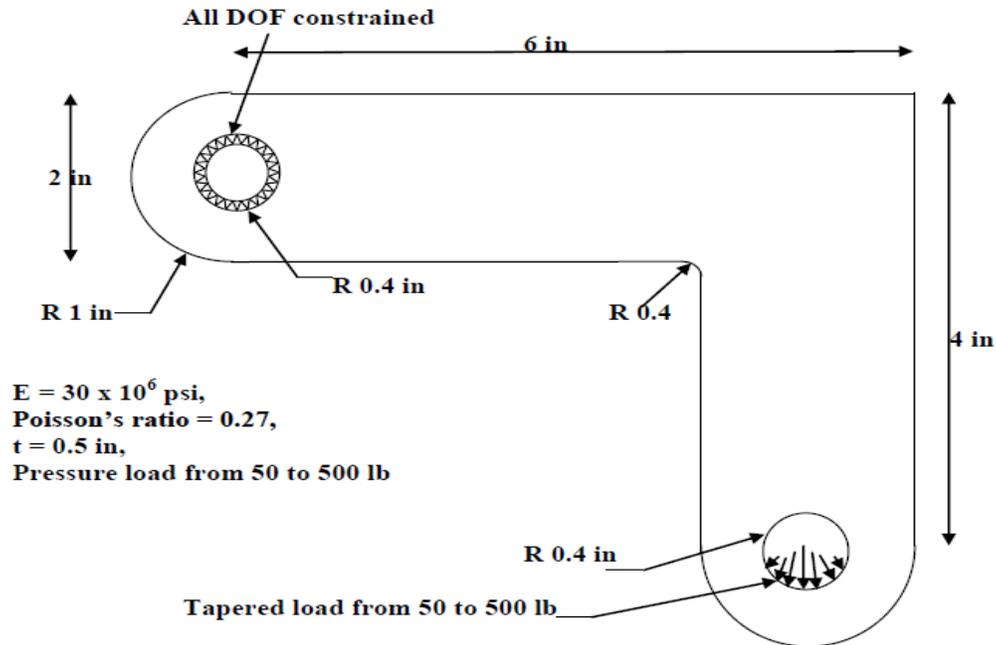
RESULT:

Analytical approach:

Calculation:

Ansys results:

	Ansys	Theoretical
Deformation		
Stress		

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.
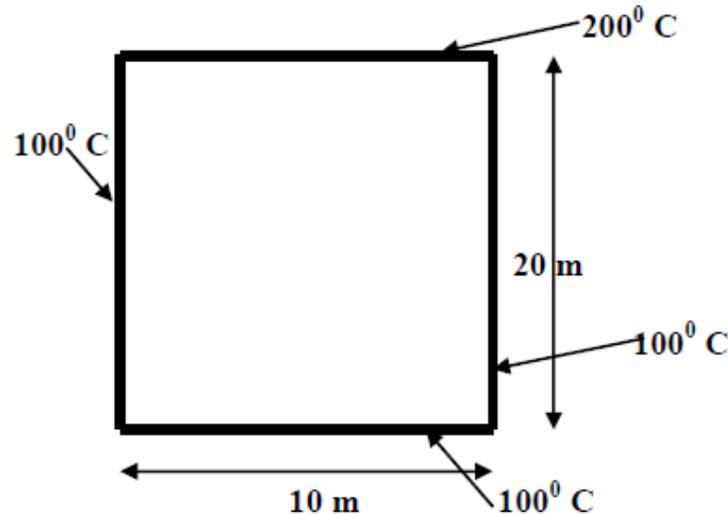
1. Ansys Main Menu – Preferencesselect – STRUCTURAL – ok
2. Element type – Add/Edit/Delete – Add – Solid – Quad 8 node – 82 – ok – option – element behavior K3 – Plane stress with thickness – ok – close.
3. Real constants – Add – ok – real constant set no – 1 – Thickness – 0.5 – ok.
4. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 30e6 –PRXY – 0.27 – ok – close.
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.
6. Operate – Booleans – Add – Areas – pick all.
7. Create – Lines – Line fillet – pick the two lines where fillet is required – apply – fillet radius – 0.4 – ok. Create – Areas – Arbitrary – by lines – pick filleted lines – ok. 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.
10. Meshing – Mesh Tool – Mesh Areas – Quad – Free – Mesh – pick all – ok. Mesh Tool – Refine– pick all – Level of refinement – 3 – ok.
11. Loads – Define loads – apply – Structural – Displacement – on Lines – select the inner lines of the upper circle – apply – DOFs to be constrained – ALL DOF – ok.
12. Loads – Define loads – apply – Structural – Pressure – on Lines – Pick line defining bottom left part of the circle – apply – load PRES value – 50 – optional PRES value – 500 – ok. Structural – Pressure – on Lines – Pick line defining bottom right part of the circle – apply – load PRES value – 500 – optional PRES value – 50 – ok.
13. Solve – current LS – ok (Solution is done is displayed) – close.
14. Plot Results – Deformed Shape – def+undeformed – ok.

15. Plot results – contour plot – Element solu – Stress – Von Mises Stress – ok (the stress distribution diagram will be displayed).
16. PlotCtrls – Animate – Deformed shape – def+undeformed-ok.

## RESULT:

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
2. Element type – Add/Edit/Delete – Add – Solid – Quad 4 node – 55 – ok – option – elementbehavior K3 – Plane stress with thickness – ok – close.
3. Material Properties – material models – Thermal – Conductivity – Isotropic – KXX – 401.
4. Modeling – Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 0, 10, 0, 20 – ok.
5. Meshing – Mesh Tool – Mesh Areas – Quad – Free – Mesh – pick all – ok. Mesh Tool – Refine – pick all – Level of refinement – 3 – ok.
6. Loads – Define loads – apply – Thermal – Temperature – on Lines – select 100_0 C lines – apply – DOFs to be constrained – TEMP – Temp value – 100_0 C – ok.
7. Loads – Define loads – apply – Thermal – Temperature – on Lines – select 100_0 C lines –
8. Solve – current LS – ok (Solution is done is displayed) – close.
9. Read results-last set-ok
10. List results-nodal solution-select temperature-ok
11. Observe the nodal solution per node.
12. From the menu bar-plot ctrls-style-size and shape-display of the element-click on real constant multiplier=0.2, don't change other values-ok.
13. Plot results-contour plot-nodal solution-temperature-deformed shape only-ok
14. Element table-define table-add-enter user label item=HTRANS, select by sequence no SMISC, 1-ok-close.
15. Element table-list table-select HTRANS-ok

RESULT:

RESULT:

Problem 5.2: A furnace wall is made up of silica brick (K=1.5W/m^{o}C) and outside magnesia brick (K= 4.9 W/m^{o}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 0.001m^{2o}C/W. The heat transfer coefficient for inner and outer surfaces is equal to 35 W/m^{2}K. Find the heat flow through the wall per unit area per unit time and temperature distribution across the wall. Area= 1m^2.
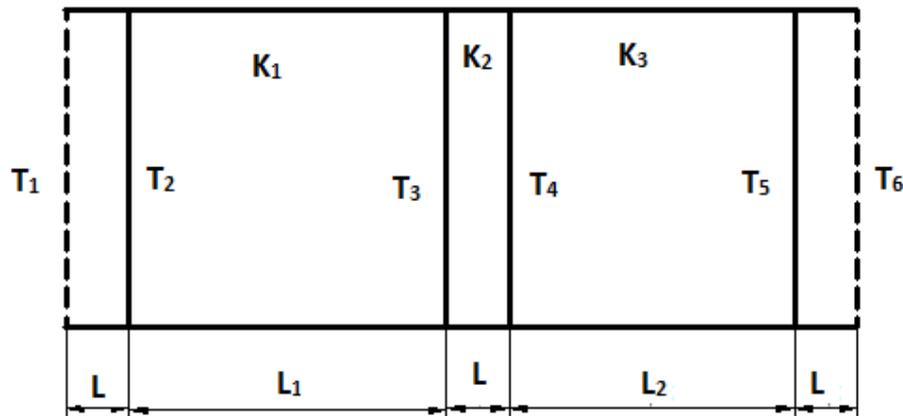
Given: K_1=1.5W/m^{o}C, K_2= 1 W/m^{o}C, K_3= 4.9 W/m^{o}C, h_1=h_4=35 W/m^{2}K, T1=820°C, T6=110 ^{0}C, L1=L$_2$= 10 cm & L= 1mm.

1. Preferences-thermal-h method-ok
2. Preprocessor-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-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-K$_{xx}$=1-ok
   Define material id=3-ok
   Thermal-conductivity-isotropic-K$_{xx}$=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 <u>don't change other attributes</u>-ok

6. Solution- Analysis type-new analysis-steady state-ok.

- Solution-define loads-apply-thermal-temperature-on nodes-pick the first nodes-ok-temperature-load-temperature value=820^0 C-apply.

- Define load-apply-thermal-temperature-on nodes-pick the last node-ok, select temperature-load temperature value=110^0 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:

	Ansys	Theoretical
Nodal temperature		
T_1		
T_2		
T_3		
T_4		
T_5		
T_6		
Heat flux		

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/m^2 ^{o}C and 10 W/m^2 ^{o}C respectively. Determine the heat flux and the temperature distribution. Take K for gypsum=0.176W/m^{o}C, K for fibre glass=0.036W/m^{o}C, K for plywood=0.115 W/m^{o}C and K for hardboard=0.215 W/m^{o}C. Area= 1m^2.
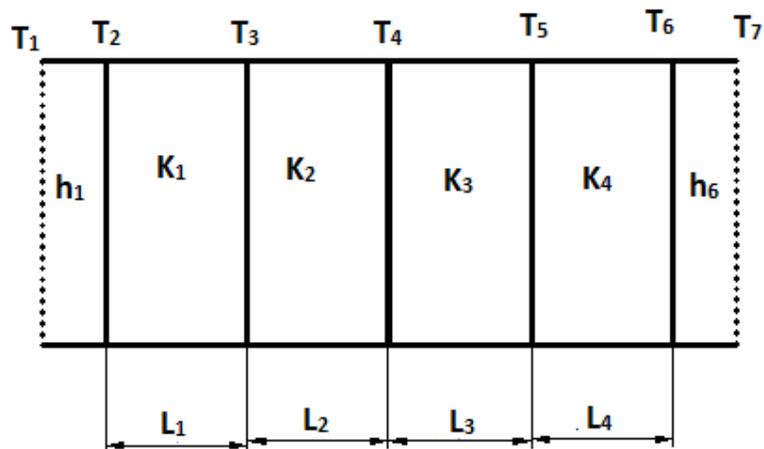
1. Preferences-thermal-h method-ok
2. Element type-add/edit/delete-add-link, 3d conduction 33,element type reference N0.=1-apply-link, convection 34 element type reference no.2=2-ok-close
3. Real constant- add/edit/delete-add-real constant set no=1-C/S area =1-ok-close.
4. Real constant- add/edit/delete-add-real 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 <u>don't change other attributes</u>-ok

6. Solution- Analysis type-new analysis-steady state-ok.

• Solution-define loads-apply-thermal-temperature-on nodes-pick the first nodes-ok-temperature-load-temperature value=20^0 C-apply.

• Define load-apply-thermal-temperature-on nodes-pick the last node-ok, select temperature-load temperature value=-10^0 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:

	Ansys	Theoretical
Nodal temperature		
T_1		
T_2		
T_3		
T_4		
T_5		
T_6		
T_7		
Heat flux		

Problem 5.4: A plane wall 'X' (K=75W/mK) is 60 mm thick and has volumetric heat generation of $1.5 \times 10^6 W/m^3$. 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= $1m^2$.
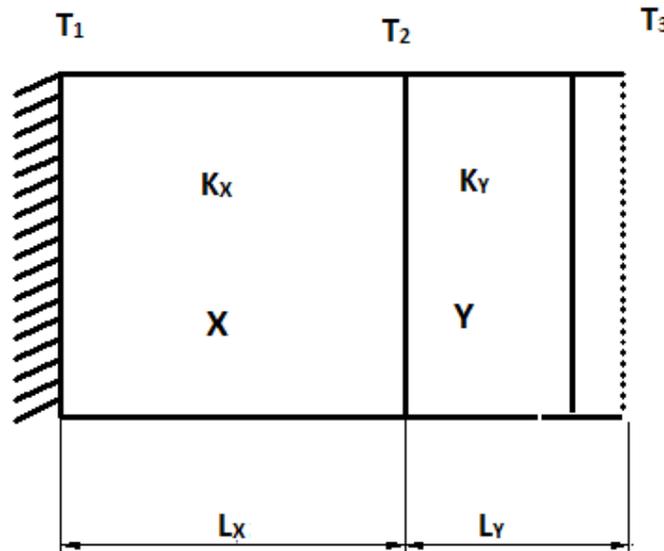
1. Preferences-Thermal-h method-ok
2. Element type-add/edit/delete- add-solid, quad 4node 55-enter reference number=1-select options-element behaviour,K_3= plane thickness-ok
3. Real constants- add/edit/delete-add- enter real constant set no. 1, thickness=2-ok-close
4. Material properties-Material models-material 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-mesh 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
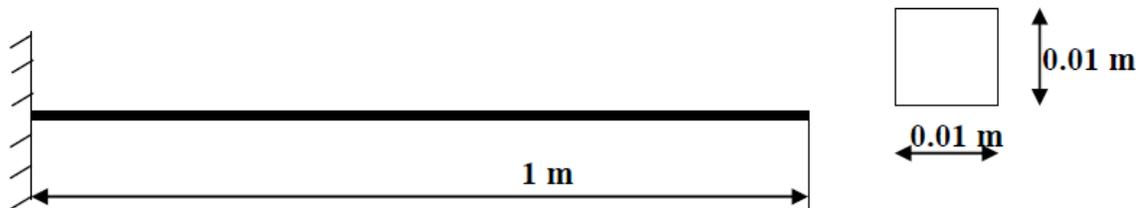
RESULT:

Analytical approach:

Calculation:

Ansys results:

	Ansys	Theoretical
Nodal temperature		
T_1		
T_2		
T_3		

Problem 6.1: Modal Analysis of Cantilever beam for natural frequency determination.

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

1. Ansys Main Menu – Preferences-select – STRUCTURAL- h method – ok
2. Element type – Add/Edit/Delete – Add – BEAM – 2 node 188– ok- close.
3. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 200e9– PRXY – 0.27 – Density – 7800 – ok – close.
4. Sections-Beams-common sections- sub type- rectangle (1st element) - enter b=0.01, h=0.01- 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).
6. Create – Lines – lines – in Active Coord – pick keypoints 1 and 2 – ok.
7. Meshing – Size Cntrls – ManualSize – Lines – All Lines – element edge length – 0.1 – ok. Mesh– Lines – Pick All – ok.
8. Solution – Analysis Type – New Analysis – Modal – ok.
9. Solution – Analysis Type – Subspace – Analysis options – no of modes to extract – 5 – no of modes to expand – 5 – ok – (use default values) – ok.
10. Solution – Define Loads – Apply – Structural – Displacement – On Keypoints – Pick firstkeypoint – apply – DOFs to be constrained – ALL DOF – ok.
11. Solve – current LS – ok (Solution is done is displayed) – close.
12. Result Summary
13. Read Results – First Set
14. Plot Results – Deformed Shape – def+undeformed – ok.
15. PlotCtrls – Animate – Deformed shape – def+undeformed-ok.
16. Read Results – Next Set
17. Plot Results – Deformed Shape – def+undeformed – ok.
18. PlotCtrls – Animate – Deformed shape – def+undeformed-ok

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.
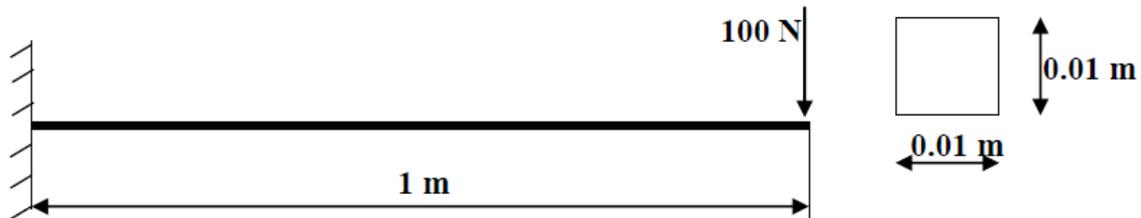

1. Ansys Main Menu – Preferences-select – STRUCTURAL- h method – ok
2. Element type – Add/Edit/Delete – Add – BEAM – 2 node BEAM 188 – ok – close.
3. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 200e9– PRXY – 0.3 – Density – 7800 – ok.
4. Sections-Beams-common 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).
6. Create – Lines – lines – in Active Coord – pick keypoints 1 and 2 – ok.
7. Meshing – Size Cntrls – ManualSize – Lines – All Lines – element edge length – 0.1 – ok. Mesh– Lines – Pick All – ok.
8. Solution – Analysis Type – New Analysis – Harmonic – ok.
9. Solution – Analysis Type – Subspace – Analysis options – Solution method – FULL – DOF printout format – Real + imaginary – ok – (use default values) – ok.
10. Solution – Define Loads – Apply – Structural – Displacement – On Keypoints – Pick firstkeypoint – apply – DOFs to be constrained – ALL DOF – ok.
11. Solution – Define Loads – Apply – Structural – Force/Moment – On Keypoints – Pick secondnode – apply – direction of force/mom – FY – Real part of force/mom – 100 – imaginary part of force/mom – 0 – ok.
12. Solution – Load Step Opts – Time/Frequency – Freq and Substps... – Harmonic frequency range– 0 – 100 – number of substeps – 100 – B.C – stepped – ok.
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')
    Utility Menu – PlotCtrls – 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.

RESULT:

ADDITIONAL PROBLEMS

1. Calculate the stresses and displacement for the plate shown below. Let the load be P = 100N applied at equal distance from both ends and E = 3e7 N/mm2.
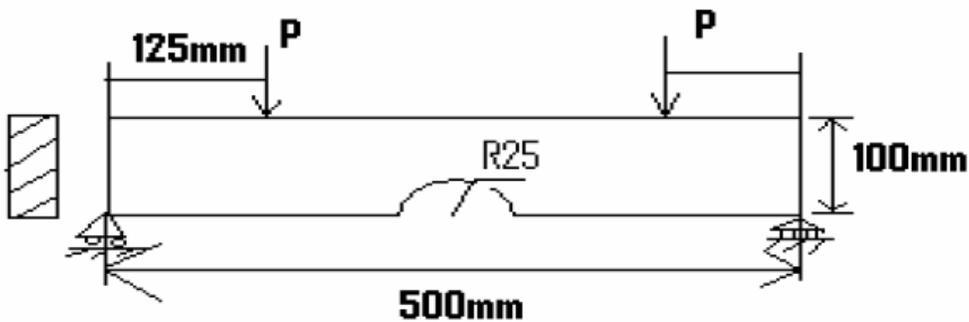
2. Current passes through a stainless steel wire of 2.5 mm diameter (k=200 W/mK) causing volumetric heat generation of 26.14X108 W/m3 .the wire is submerged in a fluid maintained at 500 C and convective heat transfer coefficient at the wire surface is 4000W/m2 K . Find the steady state temperature at 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=210Gpa, Poisson's ratio is 0.3 and the beam thickness is 10mm, the element size is 2mm
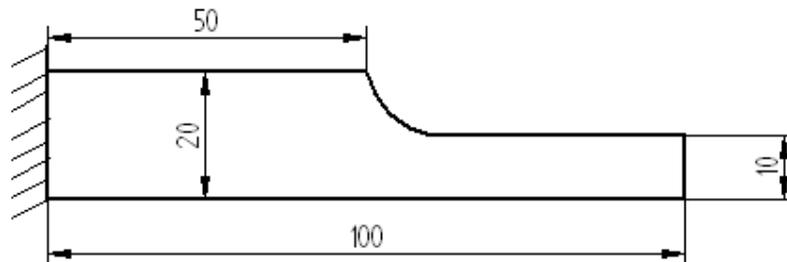
4. Loads will not be applied to the beam shown below in order to observe the deflection caused by the weight of the beam itself. The beam is to be made of steel with a Young's modulus of elasticity of 200 Gpa.
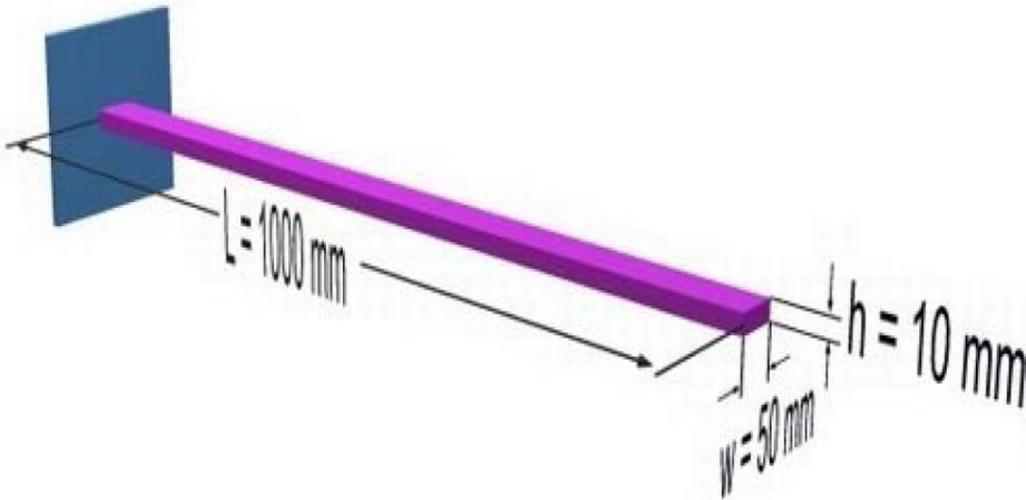
5. A beam has a force of 1000N applied as shown below. The purpose of this optimization problem is to minimize the weight of the beam without exceeding the allowable stress. It is necessary to find the cross sectional dimensions of the beam in order to minimize the weight of the beam. However, the width 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
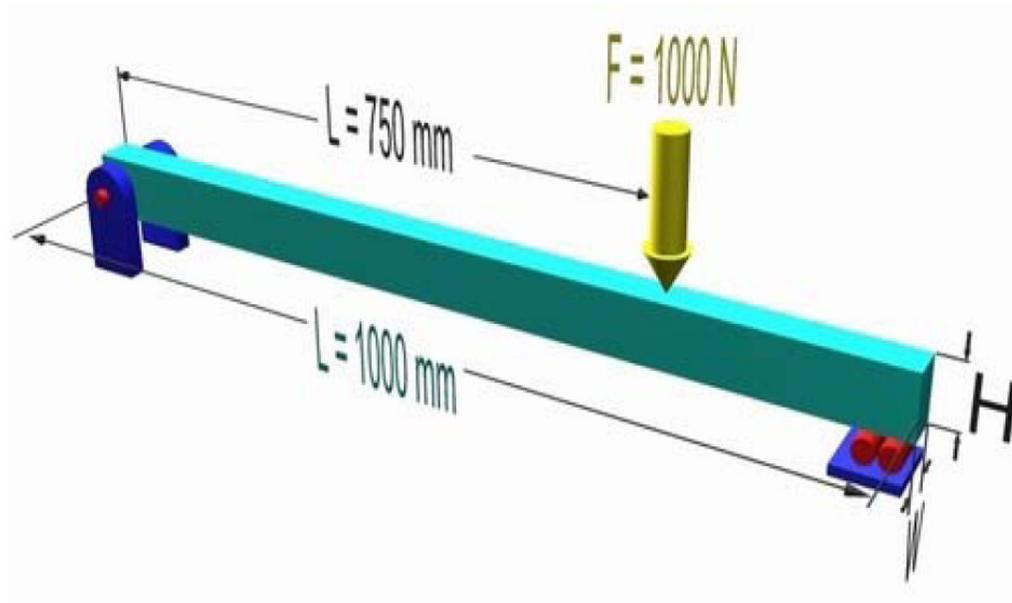
VIVA QUESTIONS

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.

8. Assumptions in simple theory of bending.

   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 N1, N2, N3 and N4 are interpolation function. N1, N2, N3, N4 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 (α) 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