Modelling and Analysis Lab (FEA)

17MEL68

DEPARTMENT OF MECHANICAL ENGINEERING

BAPUJI INSTITUTE OF ENGINEERING AND TECHNOLOGY

DAVANGERE- 577 004

DEPARTMENT OF

MECHANICAL ENGINEERING

Modelling and Analysis Lab (FEA)

17MEL68

As per VTU Syllabus (CBCS) for VI Semester

Name : ....

USN : ....

Semester : ....Batch No.......................

Ashoka E
Faculty Incharge

V.K. Mallikarjun
Instructor

BAPUJI INSTITUTE OF ENGINEERING AND TECHNOLOGY

DAVANGERE- 577 004

VISION OF THE INSTITUTE

To be center of excellence recognized nationally and internationally, in distinctive areas of engineering education and research, based on a culture of innovation and invention.

MISSION OF THE INSTITUTE

BIET contributes to the growth and development of its students by imparting a broad based engineering education and empowering them to be successful in their chosen field by inculcating in them positive approach, leadership qualities and ethical values

VISION OF THE DEPARTMENT

The department endeavors to be a center of excellence, to provide quality education leading the students to become professional mechanical engineers with ethics, contributing to the society through research, innovation, entrepreneurial and leadership qualities.

MISSION OF THE DEPARTMENT

l. To impart quality technical education through effective teaching- learning process leading to development of professional skills and attitude to excel in Mechanical Engineering.

2. To interact with institutes of repute, to enhance academic and research activities.

3. To inculcate creative thinking abilities among students and develop entrepreneurial skills.

4. To imbibe ethical, environmental friendly and moral values amongst students through broad based education

PROGRAM EDUCATIONAL OBJECTIVES (PEO'S)

1. Enable to understand mechanical engineering systems those are technically viable, economically feasible and socially acceptable to enhance quality of life.
2. Apply modern tools and techniques to solve problems in mechanical and allied engineering streams.
3. Communicate effectively using innovative tools, to demonstrate leadership and entrepreneurial skills.
4. Be a professional having ethical attitude with multidisciplinary approach to achieve self and organizational goals.
5. Utilize the best academic environment to create opportunity to cultivate lifelong learning skills needed to succeed in profession.

PROGRAM SPECIFIC OUTCOMES (PSO'S)

PS01:-Apply the acquired knowledge in design, thermal, manufacturing and interdisciplinary areas for solving industry and socially relevant problems.

PS02:-To enhance the abilities of students by imparting knowledge in emerging technologies to make them confident mechanical engineers.

Modeling and Analysis Lab (FEA)

Course Code	17MEL68	CIE Marks	40
Teaching Hours/Week (L:T:P)	03 (1 Hour Instruction+ 2 Hours Laboratory	SEE Marks	60
RBT Levels	L1, L2, L3		
	Credits-02	Exam Hours	03

Course objectives:

- To acquire basic understanding of Modeling and Analysis software
- 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.
- To lean to apply the basic principles to carry out dynamic analysis to know the natural frequency

Sl. No.	Experiments
	PART A
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)**
4	Stress analysis of a rectangular plate with a circular hole
	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
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

Course outcomes: At the end of the course, the student will be able to:

CO1.Utilize the analysis software to create geometry, discretize, apply boundary condition to solve stress related problems on bars, Trusses and plate for different loading conditions

CO2.Demonstrate the deflection of beams subjected to point, uniformly distributed and varying loads further to use the available of results to draw shear force and bending moment diagrams.

CO3. Analyze the given problem by applying basic principle to solve and demonstrate 1D and 2D heat transfer with conduction and convection boundary conditions

CO4.Predict the dynamic Characteristics and nature frequency of 2D components for various boundary condition and also analyze with forcing function.

DO's

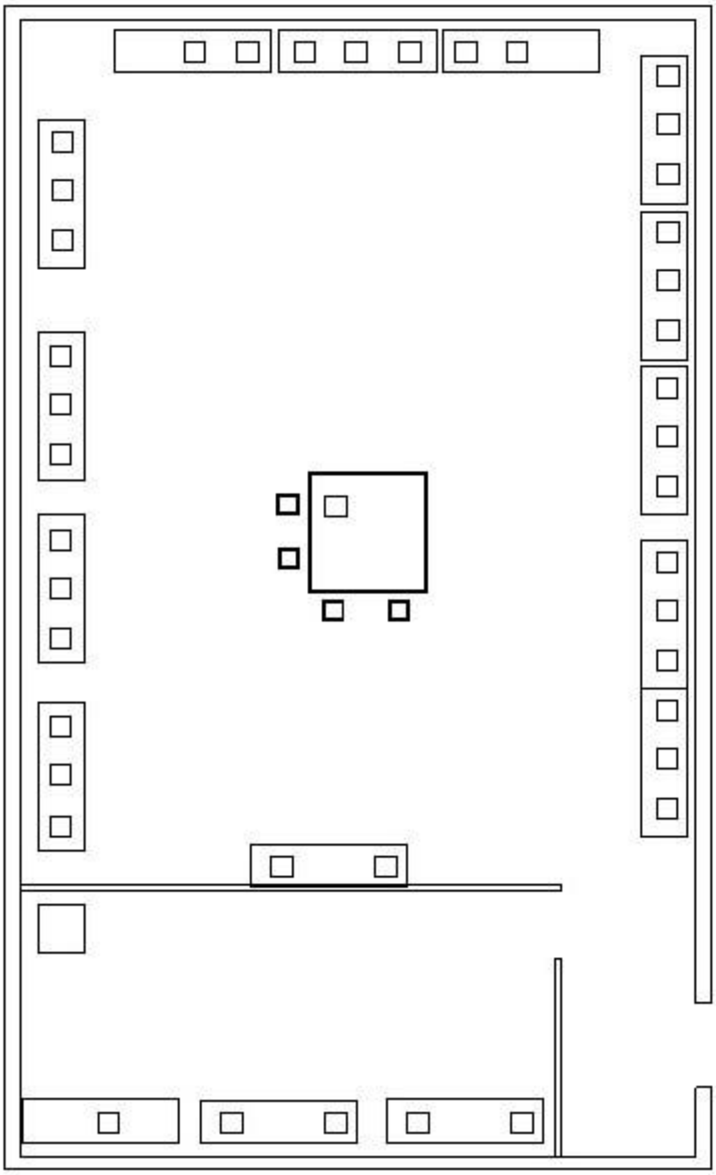
1. Students must always wear uniform and shoes before entering the lab.
2. Proper code of conduct and ethics must be followed in the lab.
3. Windows and doors to be kept open for proper ventilation and air circulation.
4. Check for the electrical connections and inform if any discrepancy found to the attention of lecturer/lab instructor.
5. Perform the experiment under the supervision/guidance of a lecturer/lab instructor only.
6. In case of fire use fire extinguisher/throw the sand provided in the lab.
7. Any unsafe conditions prevailing in the lab can be brought to the notice of the lab in charge.

DONT's

1. Do not touch any system without their prior knowledge,
2. Never overcrowd the laboratory Leave sufficient space for the person to operate the equipment's.
3. Never rest your hands on the system and the display board

CONTENTS

Lay out of Modelling and Analysis Lab



Getting Started with ANSYS

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 jobname 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 Jobname and Analysis Title
This task is not required for an analysis, but is *recommended*.

Defining the Jobname
The *jobname* is a name that identifies the ANSYS job. When you define a jobname for an analysis, the jobname 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 jobname for each analysis, you insure that no files are overwritten.

If you do not specify a jobname, all files receive the name *FILE* or *file*, depending on the operating system.

Command(s): **/FILNAME**
GUI: **Utility Menu>File>Change Jobname**

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,

BEAM	PLANE
COMBINation	SHELL
CONTACt	SOLID
FLUID HYPE	SOURCe
Relastic	SURFace
INFINite	TARGEt
LINK	USER
MASS	INTERface
MATRIX	VISCOelastic (or viscoplastic)
PIPE	

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, 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.

Substeps are incremental steps taken within a load step. You use them mainly for accuracy and convergence purposes in transient and nonlinear analyses. Substeps are also known as *time steps*-steps taken over a period of time.

Initiating the Solution

To initiate solution calculations, use either of the following:

Command(s): **SOLVE**

GUI: **Main Menu>Solution>Current LS**

When you issue this command, the ANSYS program takes model and loading information from the database and calculates the results. Results are written to the results file (*Jobname*.RST, *Jobname*.RTH, *Jobname*.RMG, or *Jobname*.RFL) and also to the database. The only difference is that only one set of results can reside in the database at one time, while you can write all sets of results (for all substeps) to the results file.

Review the Results

Once the solution has been calculated, you can use the ANSYS postprocessors to review the results.

General Steps

Step 1: Ansys Utility Menu

File – clear and start new – do not read file – ok
File – change job name – enter new job name – xxxx – ok
File – change title – enter new title – yyy – ok

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

Step 3: Preprocessor

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

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

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

Step 4: Modeling – create the required geometry such as nodes elements, area, volume by using the appropriate options**.**

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

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

Step 7: Solution – Solve the problem

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

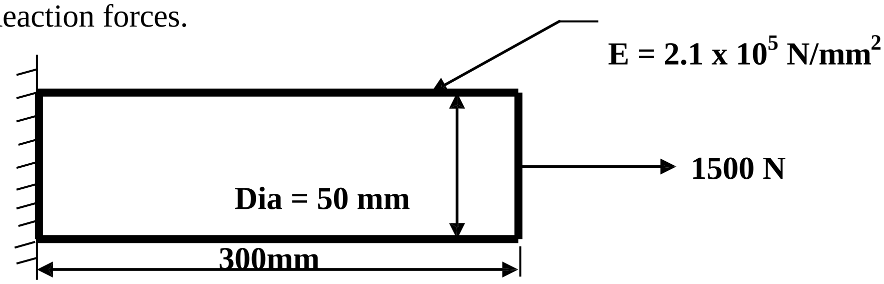
Step 9: Plot ctrls – animate – deformed shape – def+undeformed-ok

Step 10: to save the solution ansys tool bar- save model

PART A

Bars of Constant Cross-section Area

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



Step 1: Ansys Utility Menu

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

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

Step 3: Preprocessor

Element type – Add/Edit/Delete – Add – Link – 2D spar 1 – ok – close.

Real constants – Add – ok – real constant set no – 1 – c/s area – 22/7*50**2/4 – ok.

Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 2.1e5 – PRXY – 0.27 – ok – close.

Step 4: Preprocessor

Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS – 300 (x value w.r.t first node) – ok (second node is created).
Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – ok (elements are created through nodes).

Step 5: Preprocessor
Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 – apply – DOFs to be constrained – All DOF – ok.
Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply – direction of For/Mom – FX – Force/Moment value – 1500 (+ve value) – ok.

Step 6: Solution
Solve – current LS – ok (Solution is done is displayed) – close.

Step 7: General Post Processor
Element table – Define table – Add –‘ Results data item’ – By Sequence num – LS – LS1 – ok.

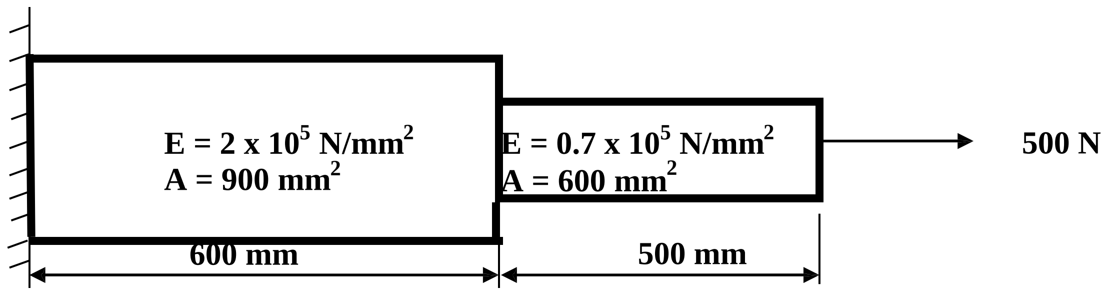
Step 8: General Post Processor
Plot Results – Deformed Shape – def+undeformed – ok.
Plot results – contour plot – Line Element Results – Elem table item at node I – LS1 – Elem table item at node J – LS1 – ok (Line Stress diagram will be displayed).

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

Step 9: PlotCtrls – Animate – Deformed shape – def+undeformed-ok

Stepped Bars

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



Step 1: Ansys Utility Menu

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

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

Step 3: Preprocessor

Element type – Add/Edit/Delete – Add – Link – 2D spar 1 – ok – close.

Real constants – Add – ok – real constant set no – 1 – c/s area – 900 – apply – real constant set no – 2 – c/s area – 600 – ok – close.

Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 2e5 – ok, – Material – New model – Define material ID – 2 – ok – Structural – Linear – Elastic – Isotropic – EX – 0.7e5 – ok – close.

Step 4: Preprocessor

Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS – 600 (x value w.r.t first node) – apply (second node is created) – x,y,z location in CS – 1100 (x value w.r.t first node) – ok (third node is created).

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

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

Create – Elements – Elem Attributes – Material number – 2 – Real constant set number – 2 – ok

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

Step 5: Preprocessor

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

Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 3 – apply – direction of For/Mom – FX – Force/Moment value – 500 (+ve value) – ok.

Step 6: Solution
Solve – current LS – ok (Solution is done is displayed) – close.

Step 7: General Post Processor
Element table – Define table – Add –' Results data item' – By Sequence num – LS – LS1 – ok.

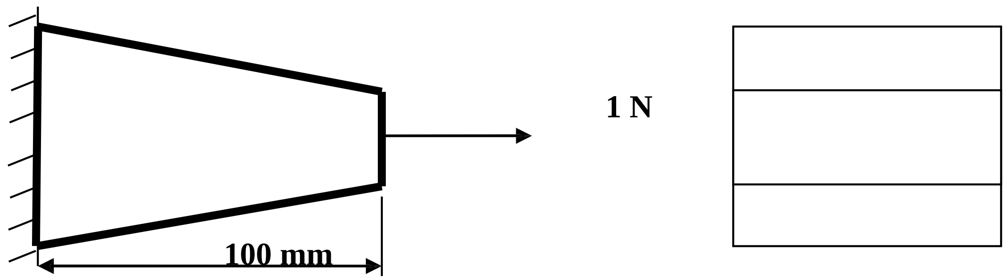
Step 8: General Post Processor
Plot Results – Deformed Shape – def+undeformed – ok.
Plot results – contour plot – Line Element Results – Elem table item at node I – LS1 – Elem table item at node J – LS1 – ok (Line Stress diagram will be displayed).

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

Step 9: PlotCtrls – Animate – Deformed shape – def+undeformed-ok

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 x 10^5 N/mm^2, Area at root = 20 x 20 = 400 mm^2, Area at the end = 20 x 10 = 200 mm^2.

Step 1: Ansys Utility Menu

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

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

Step 3: Preprocessor

Element type – Add/Edit/Delete – Add – BEAM – tapered 54 – ok- close.

Real constants – Add – ok – real constant set no – 1 – cross-sectional AREA1 – 400 – moment of inertia about Z IZ1 – 20*20**3/12 – cross-sectional AREA2 – 200 – ok.

Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 2e5 – PRXY – 0.27 – ok – close.

Step 4: Preprocessor

Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS – 100 (x value w.r.t first node) – ok (second node is created).
Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – ok (elements are created through nodes).

Step 5: Preprocessor
Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 – apply – DOFs to be constrained – ALL DOF – ok.
Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply – direction of For/Mom – FX – Force/Moment value – 1 (+ve value) – ok.

Step 6: Solution
Solve – current LS – ok (Solution is done is displayed) – close.

Step 7: General Post Processor

Element table – Define table – Add –' Results data item' – By Sequence num – SMISC – SMISC, 2 – apply, By Sequence num – SMISC – SMISC, 8 – apply, By Sequence num – SMISC – SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 12 – ok – close.
Element table – define table – add – 'Results data item' – By Sequence num – NMISC – NMISC, 1 – apply, 'results data item' – By Sequence num – NMISC – NMISC, 3 – ok.

Step 8: General Post Processor

Plot Results – Deformed Shape – def+undeformed – ok.
Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS2 – Elem table item at node J – SMIS8 – ok (Shear force diagram will be displayed).
Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS12 – ok (bending moment diagram will be displayed).

NOTE: For Shear Force Diagram use the combination SMISC 2 & SMISC 8, for Bending Moment Diagram use the combination SMISC 6 & SMISC 12. For Maximum Stress diagram use the combination NMISC 1 & NMISC 3.

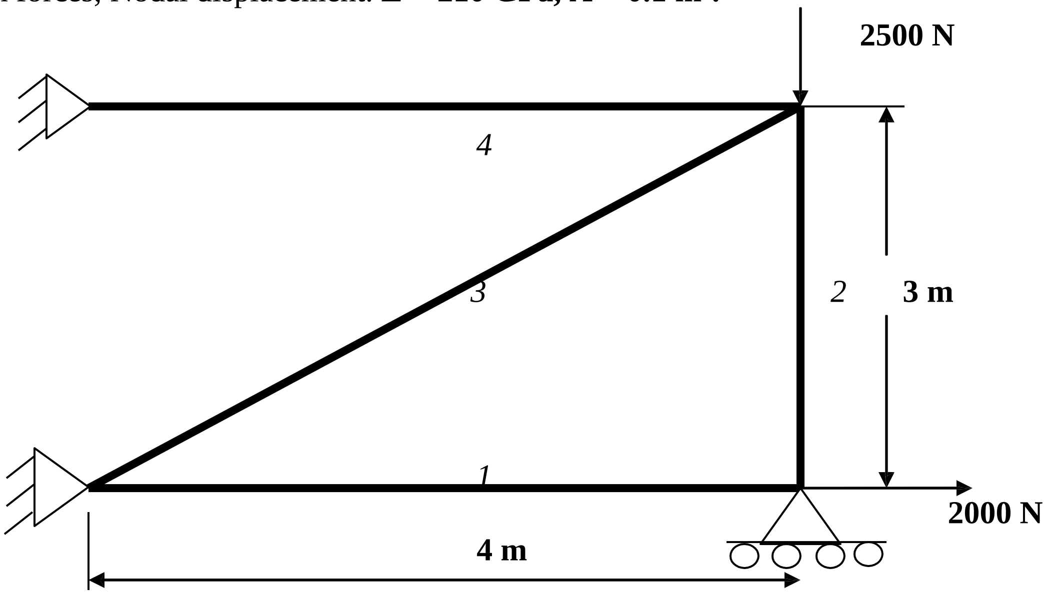
Plot results – contour plot – Line Element Results – Elem table item at node I – NMIS1 – Elem table item at node J – NMIS3– ok (the maximum stress value will be displayed).

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

Step 9: PlotCtrls – Animate – Deformed shape – def+undeformed-ok.

TRUSSES

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.**



Step 1: Ansys Utility Menu

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

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

Step 3: Preprocessor

Element type – Add/Edit/Delete – Add – Link – 2D spar 1 – ok – close.
Real constants – Add – ok – real constant set no – 1 – c/s area – 0.1 – ok – close.
Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 210e9 – ok – close.

Step 4: Preprocessor

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

Step 5: Preprocessor
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.

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 – Force/Moment – on Nodes- pick node 3 – apply – direction of For/Mom – FY – Force/Moment value – -2500 (-ve value) – ok.

Step 6: Solution
Solve – current LS – ok (Solution is done is displayed) – close.

Step 7: General Post Processor
Element table – Define table – Add – 'Results data item' – By Sequence num – LS – LS1 – ok.

Step 8: General Post Processor
Plot Results – Deformed Shape – def+undeformed – ok.

Plot results – contour plot – Line Element Results – Elem table item at node I – LS1 – Elem table item at node J – LS1 – ok (Line Stress diagram will be displayed).

Plot results – contour plot – Nodal solution – DOF solution – displacement vector sum – ok.

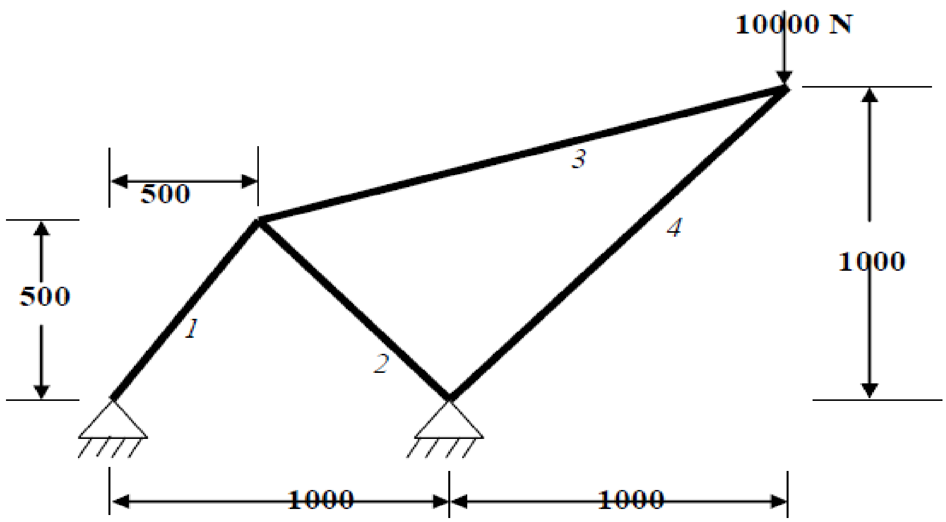
List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).

List Results – Nodal loads – items to be listed – All items – ok (Nodal loads will be displayed with the node numbers).

Step 9: PlotCtrls – Animate – Deformed shape – def+undeformed-ok

2. For the given data, find internal stresses developed, Nodal displacement in the planar truss shown in figure when a vertically downward load of 10000 N is applied as shown.

Member	C/s area mm^2	E N/mm^2
1	200	2×10^5
2	200	
3	100	
4	100	



Step 1: Ansys Utility Menu

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

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

Step 3: Preprocessor

Element type – Add/Edit/Delete – Add – Link – 2D spar 1 – ok – close.
Real constants – Add – ok – real constant set no – 1 – c/s area – 200 – apply – real constant set no – 2 – c/s area – 100 – ok – close.
Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 2e5 – PRXY – 0.27 – ok – close.

Step 4: Preprocessor

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) – 500, 500 (x, y value w.r.t first node) – apply (third node is created) – 2000, 1000 (x, y value w.r.t first node) – ok (forth node is created).

Create – Elements – Elem Attributes – Material number – 1 – Real constant set number – 1 – ok – Auto numbered – Thru Nodes – pick 1 & 3 – apply – pick 2 & 3 – ok – Elem Attributes – Material number – 1 – Real constant set number – 2 – ok – Auto numbered – Thru Nodes – pick 3 & 4 – apply – pick 2 & 4 – ok (elements are created through nodes).

Step 5: Preprocessor

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

Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 4 – apply – direction of For/Mom – FY – Force/Moment value – -10000 (-ve value) – ok.

Step 6: Solution

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

Step 7: General Post Processor

Element table – Define table – Add – ‘Results data item’ – By Sequence num – LS – LS1 – ok.

Step 8: General Post Processor

Plot Results – Deformed Shape – def+undeformed – ok.

Plot results – contour plot – Line Element Results – Elem table item at node I – LS1 – Elem table item at node J – LS1 – ok (Line Stress diagram will be displayed).

Plot results – contour plot – Nodal solution – DOF solution – displacement vector sum – ok.

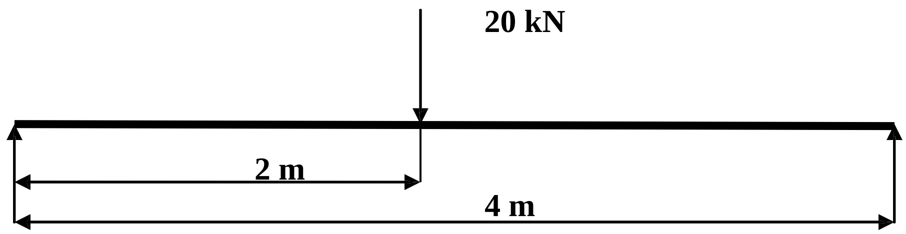
List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).

Step 9: PlotCtrls – Animate – Deformed shape – def+undeformed-ok

BEAMS

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 0.2 m * 0.3 m, Young's modulus of 210 GPa, Poisson's ratio 0.27.



Step 1: Ansys Utility Menu

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

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

Step 3: Preprocessor

Element type – Add/Edit/Delete – Add – BEAM – 2D elastic 3 – ok- close.

Real constants – Add – ok – real constant set no – 1 – c/s area – 0.2*0.3 moment of inertia – 0.2*0.3**3/12 – total beam height – 0.3 – ok.

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

Step 4: Preprocessor

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

Step 5: Preprocessor
Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 & 3 – apply – DOFs to be constrained – UY – ok.
Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply – direction of For/Mom – FY – Force/Moment value – -20000 (-ve value) – ok.

Step 6: Solution
Solve – current LS – ok (Solution is done is displayed) – close.

Step 7: General Post Processor
Plot Results – Deformed Shape – def+undeformed – ok.
Plot Results – Contour plot – Nodal solu – DOF solution – displacement vector sum – ok.
Element table – Define table – Add – 'Results data item' – By Sequence num – SMISC – SMISC, 2 – apply, By Sequence num – SMISC – SMISC, 8 – apply, By Sequence num – SMISC – SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 12 – ok – close.

NOTE: For Shear Force Diagram use the combination SMISC 2 & SMISC 8, for Bending Moment Diagram use the combination SMISC 6 & SMISC 12.

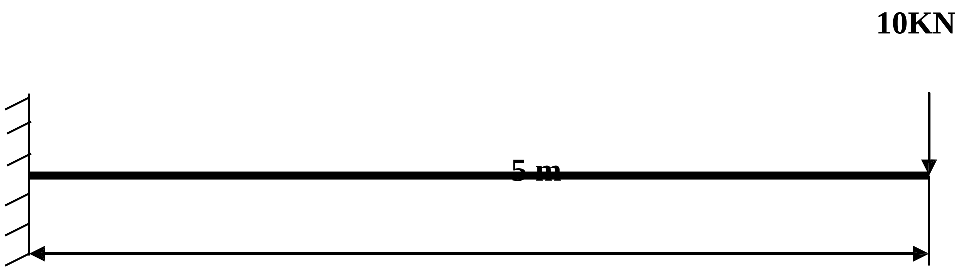
Step 8: General Post Processor
Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS2 – Elem table item at node J – SMIS8 – ok (Shear force diagram will be displayed).
Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS12 – ok (bending moment diagram will be displayed).

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

Step 9: PlotCtrls – Animate – Deformed results – DOF solution – USUM – ok.

2. 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.



Step 1: Ansys Utility Menu

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

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

Step 3: Preprocessor

Element type – Add/Edit/Delete – Add – BEAM – 2D elastic 3 – ok- close.

Real constants – Add – ok – real constant set no – 1 – c/s area – 0.2*0.3 moment of inertia – 0.2*0.3**3/12 – total beam height – 0.3 – ok.

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

Step 4: Preprocessor

Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS – 5 (x value w.r.t first node) – ok (second node is created).
Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – ok (elements are created through nodes).

Step 5: Preprocessor
Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 – apply – DOFs to be constrained – ALL DOF – ok.
Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply – direction of For/Mom – FY – Force/Moment value - -10000 (-ve value) – ok.

Step 6: Solution
Solve – current LS – ok (Solution is done is displayed) – close.

Step 7: General Post Processor
Plot Results – Deformed Shape – def+undeformed – ok.
Plot Results – Contour plot – Nodal solu – DOF solution – displacement vector sum – ok.
Element table – Define table – Add – ‘Results data item’ – By Sequence num – SMISC – SMISC, 2 – apply, By Sequence num – SMISC – SMISC, 8 – apply, By Sequence num – SMISC – SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 12 – ok – close.

NOTE: For Shear Force Diagram use the combination SMISC 2 & SMISC 8, for Bending Moment Diagram use the combination SMISC 6 & SMISC 12.

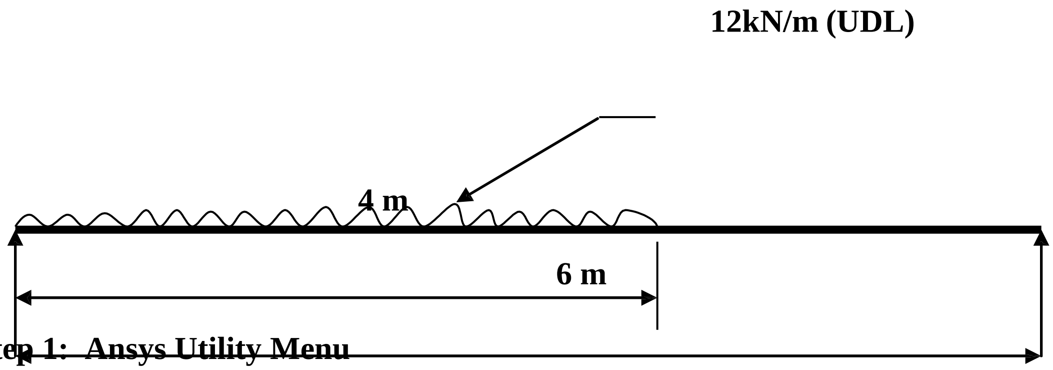
Step 8: General Post Processor
Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS2 – Elem table item at node J – SMIS8 – ok (Shear force diagram will be displayed).
Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS12 – ok (bending moment diagram will be displayed).

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

Step 9: PlotCtrls – Animate – Deformed results – DOF solution – USUM – ok.

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 0.2 m * 0.3 m, Young's modulus of 210 GPa, Poisson's ratio 0.27.



Step 1: Ansys Utility Menu

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

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

Step 3: Preprocessor

Element type – Add/Edit/Delete – Add – BEAM – 2D elastic 3 – ok- close.

Real constants – Add – ok – real constant set no – 1 – c/s area – 0.161*.322 moment of inertia – 0.161*.322 **3/12 – total beam height – 0.32 – ok.

Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 200e9 – PRXY – 0.3 –ok – close.

Step 4: Preprocessor

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) – 4 (x value w.r.t first node) – apply (third node is created)- 6(x value w.r.t fourth node) -ok
Create – Nodes – Fill between Nodes – pick 2 & 3 – apply – number of nodes to fill 7 – starting node no – 5 – ok.

Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 apply– pick 2 &5 apply– pick5 & 6 apply– pick 6 &7 apply– pick 7 &8 apply– pick8 &9 apply – pick 9 & 3 apply– pick3 &4 apply -ok (elements are created through nodes).

Step 5: Preprocessor
Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 &4– apply – DOFs to be constrained – UY – ok.

Loads – Define loads – apply – Structural – Pressure – on Beams – pick all elements between nodes 2 & 3 – apply – pressure value at node I – 12000 – pressure value at node J – 12000 – ok.

Step 6: Solution
Solve – current LS – ok (Solution is done is displayed) – close.

Step 7: General Post Processor
Plot Results – Deformed Shape – def+undeformed – ok.
Plot Results – Contour plot – Nodal solu – DOF solution – displacement vector sum – ok.
Element table – Define table – Add – 'Results data item' – By Sequence num – SMISC – SMISC, 2 – apply, By Sequence num – SMISC – SMISC, 8 – apply, By Sequence num – SMISC – SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 12 – ok – close.

NOTE: For Shear Force Diagram use the combination SMISC 2 & SMISC 8, for Bending Moment Diagram use the combination SMISC 6 & SMISC 12.

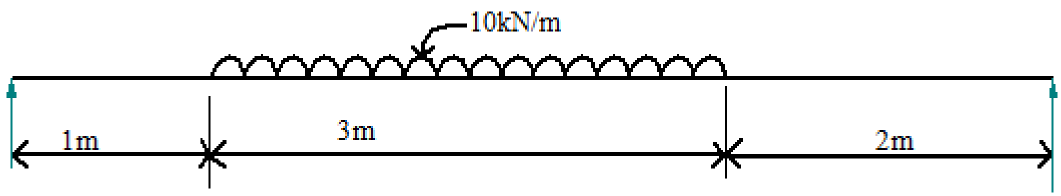
Step 8: General Post Processor
Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS2 – Elem table item at node J – SMIS8 – ok (Shear force diagram will be displayed).
Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS12 – ok (bending moment diagram will be displayed).

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

Step 9: PlotCtrls – Animate – Deformed results – DOF solution – USUM – ok.

4. 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 0.2 m * 0.3 m, Young's modulus of 210 GPa, Poisson's ratio 0.27.



Step 1: Ansys Utility Menu

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

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

Step 3: Preprocessor

Element type – Add/Edit/Delete – Add – BEAM – 2D elastic 3 – ok- close.

Real constants – Add – ok – real constant set no – 1 – c/s area – 0.2*0.3 moment of inertia – 0.2*0.3**3/12 – total beam height – 0.3 – ok.

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

Step 4: Preprocessor

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).
Create – Nodes – Fill between Nds – pick 1 & 2 – apply – number of nodes to fill 7 – starting node no – 4 – ok.
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).

Step 5: Preprocessor
Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 & 3 – apply – DOFs to be constrained – UY – ok.

Loads – Define loads – apply – Structural – Pressure – on Beams – pick all elements between nodes 1 & 2 – apply – pressure value at node I – 10000 – pressure value at node J – 10000 – ok.

Step 6: Solution
Solve – current LS – ok (Solution is done is displayed) – close.

Step 7: General Post Processor
Plot Results – Deformed Shape – def+undeformed – ok.
Plot Results – Contour plot – Nodal solu – DOF solution – displacement vector sum – ok.
Element table – Define table – Add – 'Results data item' – By Sequence num – SMISC – SMISC, 2 – apply, By Sequence num – SMISC – SMISC, 8 – apply, By Sequence num – SMISC – SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 12 – ok – close.

NOTE: For Shear Force Diagram use the combination SMISC 2 & SMISC 8, for Bending Moment Diagram use the combination SMISC 6 & SMISC 12.

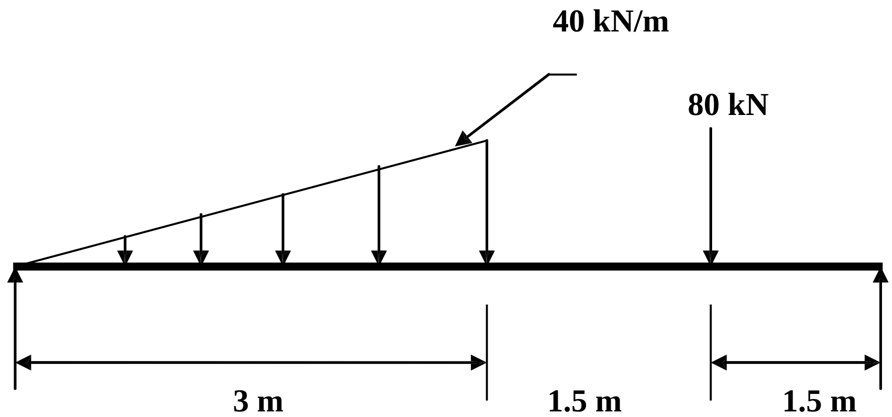
Step 8: General Post Processor
Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS2 – Elem table item at node J – SMIS8 – ok (Shear force diagram will be displayed).
Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS12 – ok (bending moment diagram will be displayed).

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

Step 9: PlotCtrls – Animate – Deformed results – DOF solution – USUM – ok.

5. 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 0.2 m * 0.3 m, Young's modulus of 210 GPa, Poisson's ratio 0.27.



Step 1: Ansys Utility Menu

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

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

Step 3: Preprocessor

Element type – Add/Edit/Delete – Add – BEAM – 2D elastic 3 – ok – close.

Real constants – Add – ok – real constant set no – 1 – c/s area – 0.2*0.3 moment of inertia – 0.2*0.3**3/12 – total beam height – 0.3 – ok.

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

Step 4: Preprocessor

Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS – 3 (x value w.r.t first node) – apply (second node is created) – 4.5 (x value w.r.t first node) – apply (third node is created) – 6 (x value w.r.t first node) – ok (forth node is created).
Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – apply – pick 2 & 3 – apply – pick 3 & 4 – ok (elements are created through nodes).

Step 5: Preprocessor

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

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.

Step 6: Solution
Solve – current LS – ok (Solution is done is displayed) – close.

Step 7: General Post Processor
Plot Results – Deformed Shape – def+undeformed – ok.
Plot Results – Contour plot – Nodal solu – DOF solution – displacement vector sum – ok.
Element table – Define table – Add – 'Results data item' – By Sequence num – SMISC – SMISC, 2 – apply, By Sequence num – SMISC – SMISC, 8 – apply, By Sequence num – SMISC – SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 12 – ok – close.

NOTE: For Shear Force Diagram use the combination SMISC 2 & SMISC 8, for Bending Moment Diagram use the combination SMISC 6 & SMISC 12.

Step 8: General Post Processor
Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS2 – Elem table item at node J – SMIS8 – ok (Shear force diagram will be displayed).
Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS12 – ok (bending moment diagram will be displayed).

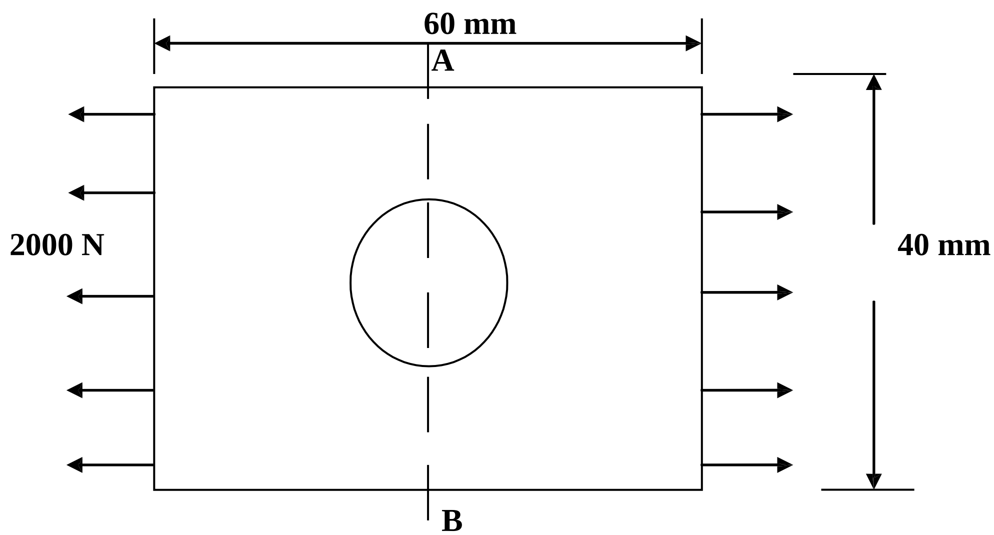
List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).
List Results – Nodal loads – items to be listed – All items – ok (Nodal loads will be displayed with the node numbers).

Step 9: PlotCtrls – Animate – Deformed results – DOF solution – USUM – ok.

Stress analysis of a rectangular plate with a circular hole

**** For 2D and 3D problems, after the geometry has been created meshing is to be done (elements/ nodes are created) ****

Problem 1. In the plate with a hole under plane stress, find deformed shape of the hole and determine the maximum stress distribution alone 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.



Step 1: Ansys Utility Menu

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

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

Step 3: Preprocessor

Element type – Add/Edit/Delete – Add – Solid – Quad 4 node – 42 – ok – option – element behavior K3 – Plane stress with thickness – ok – close.

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

Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 2.1e5 – PRXY – 0.3 – ok – close.

Step 4: Preprocessor

Modeling – Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 0, 60, 0, 40 – ok. Create – Area – Circle – solid circle – X, Y, radius – 30, 20, 5 – ok.
Operate – Booleans – Subtract – Areas – pick area which is not to be deleted (rectangle) – apply – pick area which is to be deleted (circle) – ok.
Meshing – Mesh Tool – Mesh Areas – Quad – Free – Mesh – pick all – ok. Mesh Tool – Refine – pick all – Level of refinement – 3 – ok.

Step 5: Preprocessor
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.
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.

Step 6: Solution
Solve – current LS – ok (Solution is done is displayed) – close.

Step 7: General Post Processor
Plot Results – Deformed Shape – def+undeformed – ok.
Plot results – contour plot – Element solu – Stress – Von Mises Stress – ok (the stress distribution diagram will be displayed).

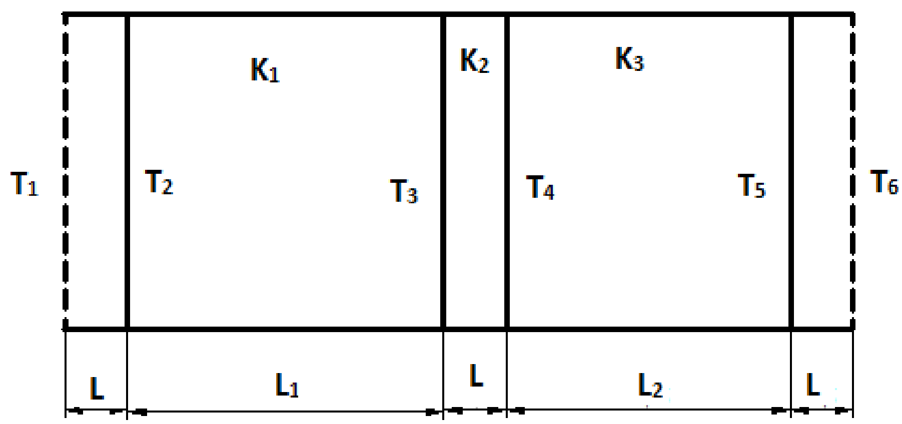
Step 8: Plotctrls – Animate – Deformed shape – def+undeformed-ok

PART B

Thermal Analysis

Problem 1:

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^{O}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 don't change other attributes-ok

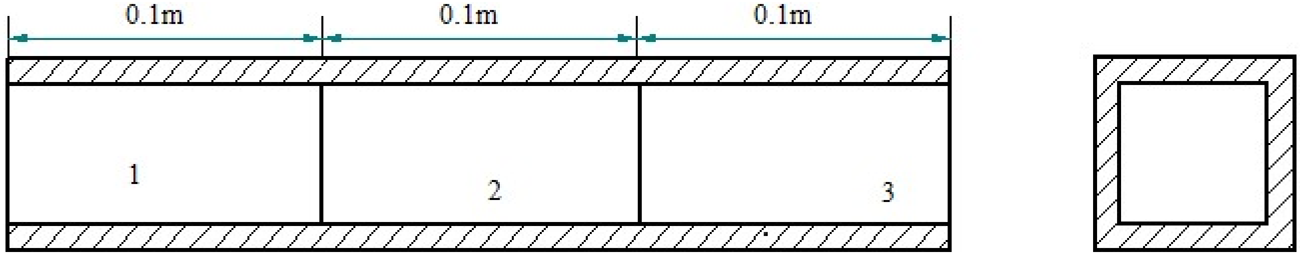
6. Solution- Analysis type-new analysis-steady state-ok.

- 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

2. For the composite wall idealized by the 1-D model shown in fig below, determine the interface temperature for element, K_1=5 W/m^{0}C, K_2=10 W/m^{0}C, and for element K_3=15 W/m^{0}C, Area=0.1m^2.The left end has a constant temperature of 200^{0}C and the right end has a constant temperature of 600^{0}C.



1. Preferences-Thermal-h method-ok
2. Element type-add/edit/delete-add-Link-20 conduction-ok-close.
3. Real constants- add/edit/delete-link-ok-add-area as 0.1-ok.
4. Material properties-material model 1 -thermal conductivity-isotropic-KXX=20-ok.

   From the define material model behavior menu bar-material new model

   Enter define material id=2-ok

   Thermal-conductivity-isotropic-K_{XX}=30-ok

   Define material id=3 ok

   Thermal-conductivity-isotropic-K_{XX}=50-ok

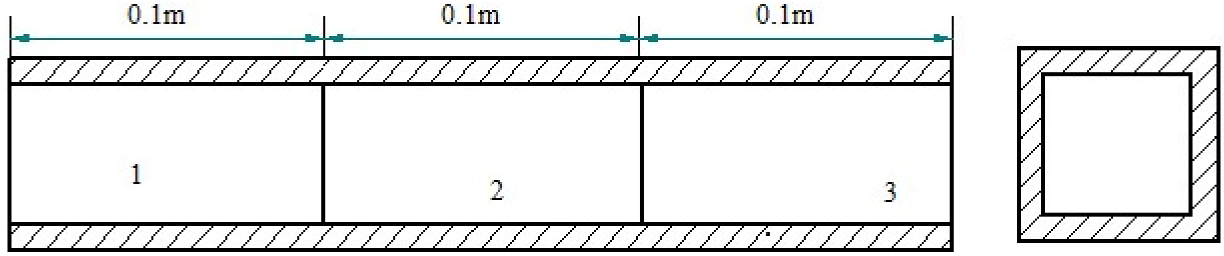
   Modeling – Create – Nodes – In Active CS – Apply -Create – Elements – Auto numbered - Thru Nodes – Select node-apply -ok
5. Solution-Analysis type-new analysis-steady state-ok

   define loads-apply-thermal-temp on nodes –select node 1 –ok-200^{0}c-ok-simillarly for node 4 –Apply a temp of 600^{0}C

6.Solution-solve-current LS-ok-close

7. General Post Processor –Result-List result-Nodal solution-DOF-node temp –plot control-animate-Deformed result-DOF solution-temp-ok.

3. For the composite wall idealized by the 1-D model shown in fig below, determine the interface temperature for element, K_1=20W/m^{0}C, K_2=30W/m^{0}C, and for element K_3=50 W/m^{0}C Area=1m^2,The left end has a constant temperature of 800^{0}C and the right end has a constant temperature of 20^{0}C.



1. Preferences-Thermal-h method-ok
2. Element type-add/edit/delete-add-Link-20 conduction-ok-close.
3. Real constants- add/edit/delete-link-ok-add-area as 1-ok.
4. Material properties-material model 1 -thermal conductivity-isotropic-KXX=20-ok.

   From the define material model behavior menu bar-material new model

   Enter define material id=2-ok

   Thermal-conductivity-isotropic-K_{XX}=30-ok

   Define material id=3 ok

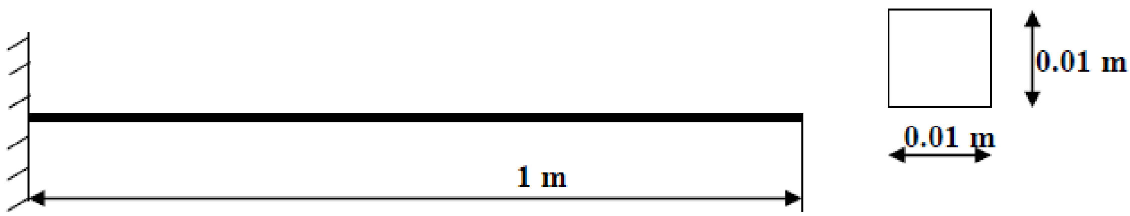
   Thermal-conductivity-isotropic-K_{XX}=50-ok

   Modeling – Create – Nodes – In Active CS – Apply -Create – Elements – Auto numbered -Thru Nodes – Select node-apply -ok
5. Solution-Analysis type-new analysis-steady state-ok

   define loads-apply-thermal-temp on nodes –select node 1 –ok-800^{0}c-ok-simillarly for node 4 –Apply a temp of 20^{0}C
6. Solution-solve-current LS-ok-close.
7. General Post Processor –Result-List result-Nodal solution-DOF-node temp –plot control-animate-Deformed result-DOF solution-temp-ok.

DYNAMIC ANALYSIS

1. Modal Analysis of Cantilever beam for natural frequency determination. Modulus of elasticity = 200GPa, Density = 7800 Kg/m^3



Step 1: Ansys Utility Menu

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

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

Step 3: Preprocessor

Element type – Add/Edit/Delete – Add – BEAM – 2D elastic 3 – ok- close.

Real constants – Add – ok – real constant set no – 1 – c/s area – 0.01*0.01 moment of inertia – 0.01*0.01**3/12 – total beam height – 0.01 – ok.

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

Step 4: Preprocessor

Modeling – Create – Keypoints – in Active CS – x,y,z locations – 0,0 – apply – x,y,z locations – 1,0 – ok (Keypoints created).
Create – Lines – lines – in Active Coord – pick keypoints 1 and 2 – ok.
Meshing – Size Cntrls – ManualSize – Lines – All Lines – element edge length – 0.1 – ok. Mesh – Lines – Pick All – ok.

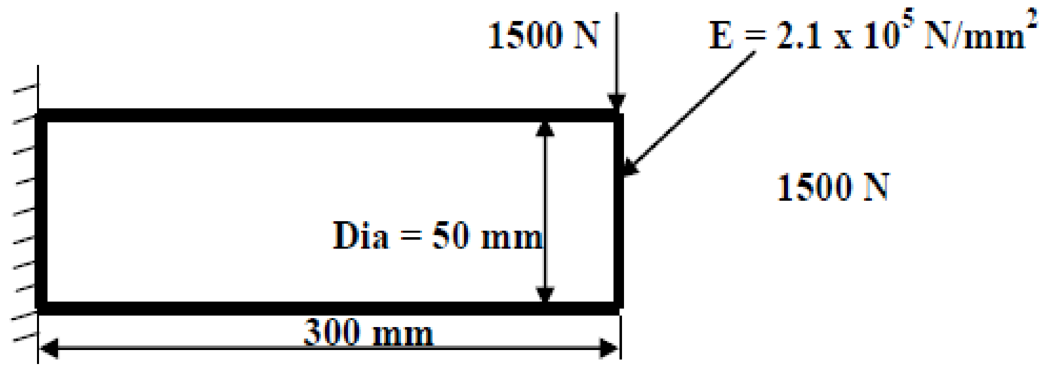
Step 5: Solution
Solution – Analysis Type – New Analysis – Modal – ok.
Solution – Analysis Type – Subspace – Analysis options – no of modes to extract – 5 – no of modes to expand – 5 – ok – (use default values) – ok.
Solution – Define Loads – Apply – Structural – Displacement – On Keypoints – Pick first keypoint – apply – DOFs to be constrained – ALL DOF – ok.
Solve – current LS – ok (Solution is done is displayed) – close.

Step 7: General Post Processor
Result Summary

Step 8: General Post Processor
Read Results – First Set
Plot Results – Deformed Shape – def+undeformed – ok.
PlotCtrls – Animate – Deformed shape – def+undeformed-ok.
Read Results – Next Set
Plot Results – Deformed Shape – def+undeformed – ok.
PlotCtrls – Animate – Deformed shape – def+undeformed-ok.

2. Bar subjected to forcing function

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



Step 1: Ansys Utility Menu

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

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

Step 3: Preprocessor

Element type – Add/Edit/Delete – Add – Link – 2D spar 1 – ok – close.

Real constants – Add – ok – real constant set no – 1 – c/s area – 22/7*50**2/4 – ok.

Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 2.1e5 – PRXY – 0.27 – Density – 7.8e-6 – ok – close.

Step 4: Preprocessor

Modeling – Create – Keypoints – in Active CS – x,y,z locations – 0,0 – apply – x,y,z locations – 300,0 – ok (Keypoints created).
Create – Lines – lines – in Active Coord – pick keypoints 1 and 2 – ok.
Meshing – Size Cntrls – ManualSize – Lines – All Lines – element edge length – 0.1 – ok. Mesh – Lines – Pick All – ok.

Step 5: Solution
Solution – Analysis Type – New Analysis – Harmonic – ok.

Solution – Analysis Type – Subspace – Analysis options – Solution method – FULL – DOF printout format – Real + imaginary – ok – (use default values) – ok.
Solution – Define Loads – Apply – Structural – Displacement – On Keypoints – Pick first keypoint – apply – DOFs to be constrained – ALL DOF – ok.

Solution – Define Loads – Apply – Structural – Force/Moment – On Keypoints – Pick second node – apply – direction of force/mom – FY – Real part of force/mom – 1500 – imaginary part of force/mom – 0 – ok.
Solution – Load Step Opts – Time/Frequency – Freq and Substps... – Harmonic frequency range – 0 – 100 – number of substeps – 100 – B.C – stepped – ok. Solve
– current LS – ok (Solution is done is displayed) – close.

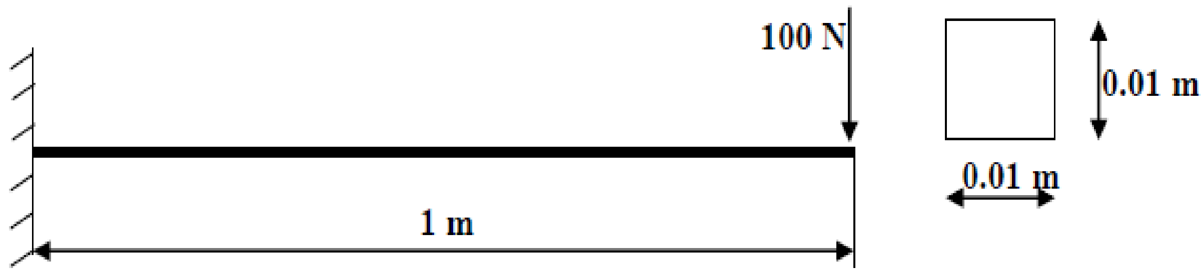
Step 6: TimeHist Postpro
Select 'Add' (the green '+' sign in the upper left corner) from this window – Nodal solution - DOF solution – Y component of Displacement – ok. Graphically select node 2 – ok.
Select 'List Data' (3 buttons to the left of 'Add') from the window.
'Time History Variables' window click the 'Plot' button, (2 buttons to the left of 'Add')

Step 7: 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.

3. 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/m^3.



Step 1: Ansys Utility Menu

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

Step 2: Ansys Main Menu – Preferences
select – STRUCTURAL - ok
Step 3: Preprocessor

Element type – Add/Edit/Delete – Add – BEAM – 2D elastic 3 – ok – close.

Real constants – Add – ok – real constant set no – 1 – c/s area – 0.01*0.01 moment of inertia – 0.01*0.01**3/12 – total beam height – 0.01 – ok.

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

Step 4: Preprocessor

Modeling – Create – Keypoints – in Active CS – x,y,z locations – 0,0 – apply – x,y,z locations – 1,0 – ok (Keypoints created).
Create – Lines – lines – in Active Coord – pick keypoints 1 and 2 – ok.

Meshing – Size Cntrls – ManualSize – Lines – All Lines – element edge length – 0.1 – ok. Mesh – Lines – Pick All – ok.

Step 5: Solution
Solution – Analysis Type – New Analysis – Harmonic – ok.
Solution – Analysis Type – Subspace – Analysis options – Solution method – FULL – DOF printout format – Real + imaginary – ok – (use default values) – ok.
Solution – Define Loads – Apply – Structural – Displacement – On Keypoints – Pick first keypoint – apply – DOFs to be constrained – ALL DOF – ok.
Solution – Define Loads – Apply – Structural – Force/Moment – On Keypoints – Pick second node – apply – direction of force/mom – FY – Real part of force/mom – 100 – imaginary part of

force/mom – 0 – ok.
Solution – Load Step Opts – Time/Frequency – Freq and Substps... – Harmonic frequency range – 0 – 100 – number of substeps – 100 – B.C – stepped – ok. Solve – current LS – ok (Solution is done is displayed) – close.

Step 6: Time Hist Post processor
Select 'Add' (the green '+' sign in cthe 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')

Step 7: 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.

VIVA QUESTIONS

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 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 Φ1, Φ2, Φ3 and Φ4 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:

19. What are Global coordinates?

The points in the entire structure are defined using coordinates system is known as global

- 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.

20. 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 valueproblem. If the solution of differential equation is obtained together with boundary conditions then it is known as boundary value problem.

21.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.

22.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.

23.Define Quasi-static response.

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

24.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.

25. 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.

26. 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.

27.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, isoparametric elements are used i.e for problems involving curved boundaries, a family of elements ‘isoparametric elements’ are used.

28.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.

29.What is discretization?

Discretization is the process of dividing given problem into several small elements, connected with nodes.

30. Steps in FEM

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