



Dayananda Sagar College of Engineering

Shavige Malleshwara Hills, Kumaraswamy Layout, Bengaluru 560078
(An Autonomous Institute under VTU, Belagavi, Approved by AICTE & ISO 9001:2008 Certified)
Accredited by National Assessment & Accreditation Council (NAAC) with 'A' Grade

Department of Mechanical Engineering

Computer Aided Modelling and Analysis Laboratory Manual (CAMA Lab)

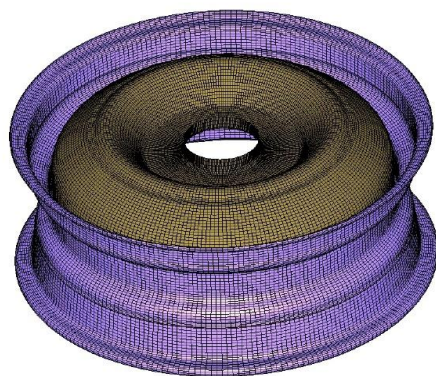
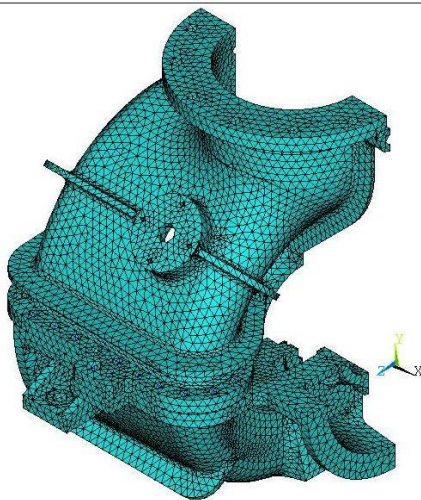
Subject Code: 17ME6DLCMA

Name of the Student:

USN:

Semester/Section:

Lab Batch:



Contents

1	Syllabus	2
2	Planner	3
3	Introduction to FEM	5
4	Introduction to ANSYS	6
5	Stress analysis of Bars	12
6	Stresses analysis of Trusses	19
7	Stresses analysis of Beams	22
8	Stresses analysis of Rectangular plate with circular hole	28
9	Thermal analysis	34
10	Fluid flow analysis	48
11	Dynamic analysis	56
12	Viva questions	63

Sub: COMPUTER AIDED MODELING AND ANALYSIS LABORATORY

Subject Code : 17ME6DLCMA

Hours/Week : 03

Total Hours : 40

CIE : 50

Exam Hours : 03

SEE : 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. **Fundamentals 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

-----**Total 50 Marks**

3.1.1. Course Outcomes (COs)

After completion of this course, student will be able to

CO1	Use finite element analysis software for structural and thermal analysis of engineering problems	3
CO 2	Choose suitable element type to solve structural and thermal engineering problems.	3
CO 3	Solve continuum problems by applying basics of Theory of Elasticity	3
CO 4	Evaluate safety of the component based on FEA software solutions	4
CO 5	Create FE models for 1D, 2D and axisymmetric problems	4
CO 6	Correlate the results of numerical analysis with analytical solutions	4

3.1.2. CO-PO matrices of courses selected in 3.1.1

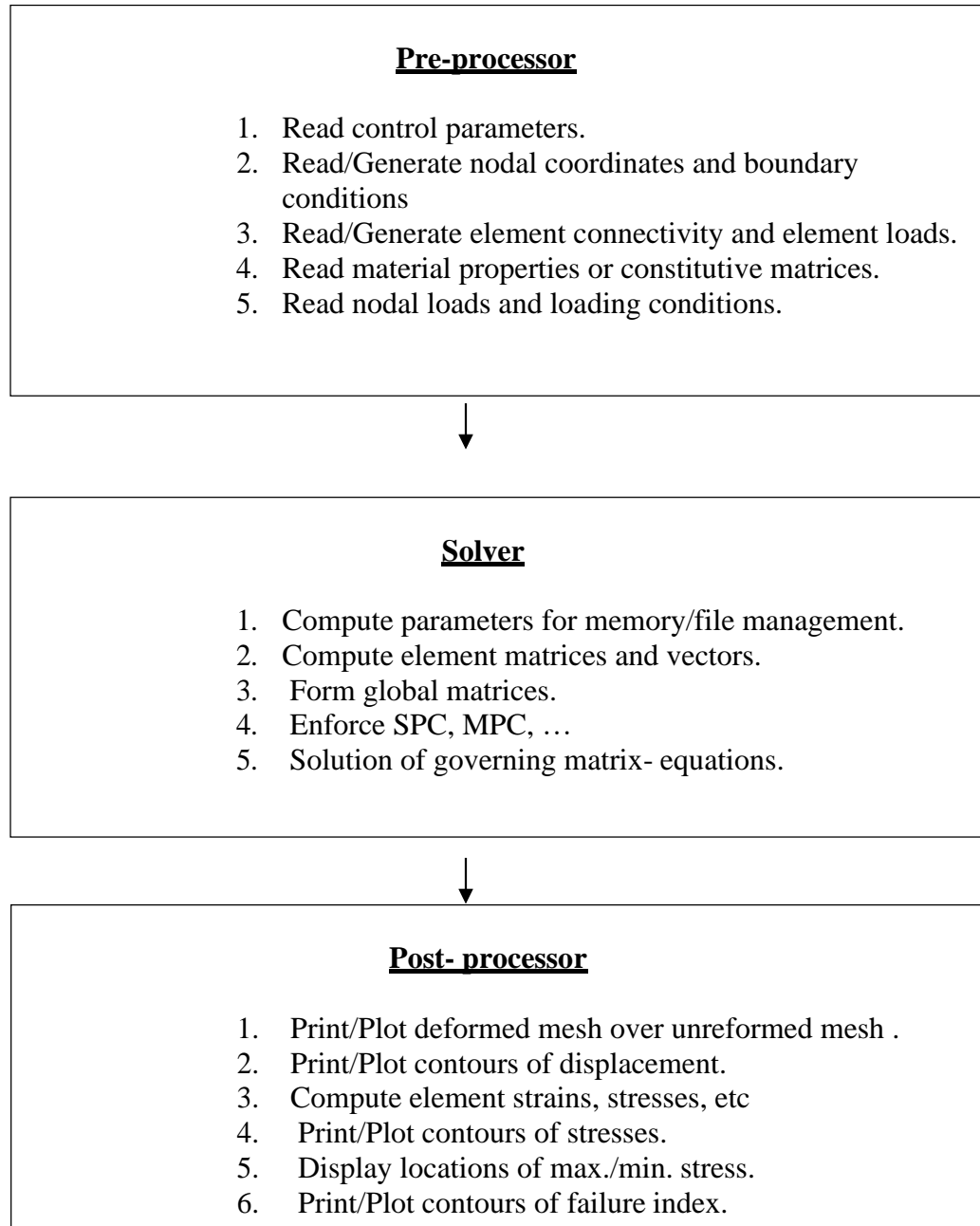
	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12	PSO1	PSO2
CO 1	3	3	3	3	3								3	3
CO 2	3	3	3	3	3									
CO 3	3	3	3	3	3									
CO 4	3	3	3	3	3	2								
CO 5	3	3	3	3	3									
CO 6	3	3	3	3	3									

Planner

1 st week	Part A/a	Introduction to FEM and Ansys 11 Bars of constant cross section area
2 nd week	Part A/a	Bars of tapered and stepped cross section area-1 Problem each
3 rd week	Part A/b	Trusses- 2 problems
4 th week	Part A/c	Beams: simply supported and cantilever - 2 Problems. (Minimum).
5 th week	Part A/c	Beams: UDL - 2 Problems (minimum).
6 th week	Part A/c	Beams: UVL - 2 Problems (minimum).
7 th week	Part B/a	Rectangular plate with circular hole – 2 Problems (minimum)..
8 th week	Part B/b	Thermal analysis: 1-D steady/ transient – 2Problems (minimum).
9 th week	Part B/b	Thermal analysis: 2-D steady/transient - 2 Problems. (Minimum).
10 th week	Part B/c	Fluid flow analysis: Flow over a cylinder and flat plate.- 2Problems
11 th week	Part B/d	Dynamic: natural frequency – 2 Problems (minimum).
12 th week	Part B/d	Dynamic: Harmonic frequency- 2 Problem. (Minimum).
13 th week	Part B/d	Dynamic Analysis (Revision.)
14 th week	Part B/d	Test and submission.

Introduction to FEM

A commercial FEM system consists of three basic modules: pre-processor; solver; and post-processor. These modules and their functions are illustrated in Fig. The pre-processor allows the user to create geometry or input CAD geometry, and provides the tools for meshing the geometry. The solver takes the finite element model provided by the pre-processor and computes the required response. The post processor takes the data from solver and presents it in a form that user can understand. Which is as shown below



A GENERAL PROCEDURE FOR FINITE ELEMENT ANALYSIS

Certain steps in formulating a finite element analysis of a physical are common to all such analysis, whether structural, heat transfer, fluid flow, or some other problem. These steps are embodied in commercial finite element software packages (some are mentioned in the following paragraphs) and are implicitly incorporated in this text, although we do not necessarily refer to the steps explicit in the following chapters. The steps are described as follows.

Preprocessing: -

The pre processing step is, quite generally, described as defining the model and includes

- Define the geometric domain of the problem.
- Define the element type(s) to be used.
- Define the material properties of the elements.
- Define the geometric properties of the elements (length, area and the like).
- Define the element connectivity's (mesh the model).
- Define the physical constraints (boundary conditions)
- Define the loadings.

The preprocessing (model definition) step is critical. In no case is there a better example of the computer of the computer-related axiom “garbage in, garbage out.” A perfectly computed finite element solution is of absolutely no value if it corresponds to the wrong problem.

Solution: -

During the solution phase, finite element software assembles the governing algebraic equations in matrix form and computes the unknown values of the primary field variable(s). The computed values are then used by back substitution to compute additional, derived variables, such as reaction forces, element stresses, and heat flow.

As it is not uncommon for a finite element model to be represented by tens of thousands of equations of equations, special solution techniques are used to reduce data storage requirements and computation time. For static, linear problems, a wave front solver, based on Gauss elimination (Appendix C), is commonly used. While a complete discussion of the various algorithms is beyond the scope of this text, the interested reader will find a thorough discussion in the Bathe book [1].

Post processing: -

Analysis and evaluation of the solution results is referred to as post processing. Post processor software contains sophisticated routines used for sorting, printing, and plotting selected result from a finite from a finite element solution. Examples of operations that can be accomplished include

- Sort element stresses in order of magnitude.
- Check equilibrium.
- Calculate factors of safety.
- Plot deformed structural shape.
- Animate dynamic model behavior.
- Produce color-coded temperature plots.

While solution data can be manipulated many ways in post processing, the most important objective is to apply sound engineering judgment in determining whether the solution results are physically reasonable.

INTRODUCTION to ANSYS:

ANSYS is an integrated design analysis tool based on FEM developed by ANSYS, Inc. It has its own tightly integrated pre-and post-processor. The ANSYS product documentation is excellent and it includes command references; operations guide; modeling and meshing guide; basic analysis procedure guide; advanced analysis guide; element reference; theory reference; structural analysis guide; thermal analysis guide; electromagnetic field guide; fluid dynamics guide; and coupled field analysis guide. Taken together, these manuals provide descriptions of procedures, commands, elements, and theoretical details needed to use the ANSYS program. All the above manuals except the ANSYS theory reference are available online through the ANSYS help system, which can be accessed either as a standalone system or from within ANSYS program. A brief description of the information found in each of the manuals follows.

Engineering capabilities of ANSYS products are: structural analysis (linear stress, nonlinear stress, dynamic, buckling); thermal analysis (steady state, transient, Conduction, radiation, and phase change); CFD analysis (steady state, transient, incompressible, compressible, laminar, turbulent); electromagnetic fields analysis (magneto static's, electrostatics); field and coupled field analysis (acoustics, fluid- structural, fluid- thermal, magnetic- fluid, magnetic- structural, magnetic- thermal, piezoelectric, thermal-electric, thermal-structural, electric-magnetic); sub-modeling; optimization and parametric design language.

Element library in ANSYS lists 189 finite elements. They are broadly grouped into: LINK, PLANE, BEAM, SOLID, CONTAC, COMBIM, PIPE, MASS, SHELL, FLUID, SOURCE, MATRIX, HYPER, VISCO, INFIN, INTER, SURF, etc. Under each type, different shapes and orders complete the list. Obviously, ANSYS has the best elements in its library.

Analysis procedures in ANSYS can be grouped into: static analysis; transient analysis; mode frequency analysis; harmonic response analysis; buckling analysis; sub-structuring analysis; and spectrum analysis.

In ANSYS, there are fundamentally different types of optimization. The first is referred to as design optimization; it works entirely with ANSYS parametric design language and is contained within its own module (ANSYS/OPT). The second is topology optimization, a form of shape optimization.

ANSYS finite element analysis software enables engineers to perform the following tasks:

- Build computer models or transfer CAD models of structure, products, components, or systems.
- Apply operating loads or other design performance conditions.
- Study physical response, such as stress levels, temperature distributions or electromagnetic fields.
- Optimize a design early in the product development process to reduce production cost.
- Do prototype testing in environments where it otherwise would be undesirable or impossible.

FEM Procedure:

1. Discretization of given domain into a collection of prescribed finite elements.
 - a. Construct the nodes and elements.
 - b. Number the nodes and elements.
 - c. Generate the geometric properties(e.g. Coordinates and cross sectional areas) needed for the problem.
2. Derivation of element equations for all typical elements in the mesh.
 - a. Construct the variational formulation of the given differential equations over the typical element
3. Assembly of element equations to obtain the equations of the whole problem.
 - a. Identify the inter element continuity conditions among the primary variables(relationship between the total degrees of freedom and global degrees of freedom connectivity of elements) by relating the element nodes to global nodes.
 - b. Identify the "equilibrium" conditions among the secondary variables.
 - c. Assemble the element equations using step 3a and 3b.
4. Imposition of boundary condition of the problem.
5. Solution of the assembled equations.
6. Post Processing of the results.

Aspects of general purpose finite computer programs

A general purpose finite element program should meet the requirements in general engineering application and should make use of the latest developments in numerical techniques. also, they should be capable of being adopted to fast changing computer hardware development.

Main characteristics of finite element programs are:

- ▶ range of application
- ▶ type of response
- ▶ material types
- ▶ material wall constructions
- ▶ loading conditions
- ▶ type of loads

Range of application

- structures/solids
- aerospace structures
- civil engineering structures
- mechanical systems
- nuclear reactors
- fluid dynamics
- piping systems
- manufacturing processes

Type of response

- linear static
- non-linear static
- fluid dynamics
- thermal analysis
- heat transfer
- electrostatic/electromagnetic etc

Material type

- isotropic
- linear elastic
- non-linear elastic
- an isotropic
- elasto-plastic
- visco-elastic
- temperature dependent
- layered composites etc

Material wall construction

- monocoque
- layered
- sandwich
- composite material

Loading conditions

- static
 - general variation
 - axisymmetric variation
- dynamic(time dependent)
- deformation dependent
- contact/friction
- temperature etc

Types of loads

- point load
- line load
- surface loads/traction force
- volume loads/body force
- initial stress/strains or residual stress/strain
- thermal loading etc

Types of analysis

- ▶ stress analysis
- ▶ dynamic response, vibration and sound
- ▶ stability analysis
- ▶ optimization
- ▶ manufacturing processes
- ▶ crash analysis
- ▶ fatigue, fracture and durability analysis etc

Stress analysis

- ▶ small/large deformation
- ▶ small/large strain
- ▶ elasto-plastic
- ▶ visco-elastic
- ▶ thermal effects etc

Dynamic response, vibration and sound

- ▶ free vibration/modal analysis
- ▶ forced vibration/transient/steady state response
- ▶ spectral density analysis/shock analysis
- ▶ dynamic response of linear/non-linear damping etc

Stability analysis

- ▶ buckling instability
 - geometrical non-linearity
 - material non-linearity

Optimization

- ▶ structural optimization
- ▶ shape optimization
- ▶ section optimization
- ▶ material optimization

Manufacturing process

- ▶ sheet metal forming
- ▶ hydro forming
- ▶ welding process etc

Crash analysis

- ▶ impact
- ▶ crash worthiness etc

Fatigue, fracture and durability analysis

- ▶ crack growth
- ▶ fatigue life durability prediction

Different analysis types and responses

static analysis

- ▶ deflection
- ▶ stresses
- ▶ strains
- ▶ forces
- ▶ energies

dynamic analysis

- ▶ frequencies
- ▶ deflection(mode shape)
- ▶ stresses
- ▶ strains
- ▶ forces
- ▶ energies

Heat transfer analysis

- ▶ temperature
- ▶ heat fluxes
- ▶ thermal gradients
- ▶ heat flow from convection faces

Fluid analysis

- ▶ pressures
- ▶ gas temperatures
- ▶ convection coefficients
- ▶ velocities

Automotive industry

- ▶ static analyses
- ▶ model analysis
- ▶ transient dynamics
- ▶ heat transfer
- ▶ mechanisms
- ▶ fracture mechanics
- ▶ metal forming
- ▶ crashworthiness

Architectural

- ▶ soil mechanics
- ▶ rock mechanics
- ▶ hydraulics
- ▶ fracture mechanics
- ▶ hydro elasticity

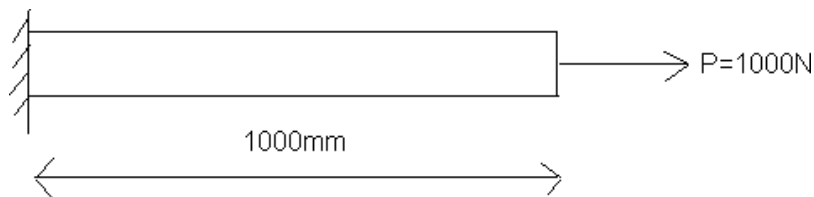
Aerospace industry

- ▶ static analyses
- ▶ model analyses
- ▶ aerodynamics
- ▶ transient dynamics
- ▶ heat transfer
- ▶ fracture mechanics
- ▶ creep and plasticity analysis
- ▶ composite materials
- ▶ aero elasticity
- ▶ metal forming
- ▶ crashworthiness

(Students are required to solve problems theoretically and compare the answer with Ansys results)

Bar

1) For the simple bar shown in the figure determine the displacements, stress and the section .the C/S of the bar is 500 mm^2 , length is 1000 mm , and the young's modulus is $E=2 \times 10^5 \text{ N/mm}^2$, Take load $P=1000 \text{ N}$.



Procedure:

1> Main Menu > Preferences

Click on structural

OK

2> Main Menu > Preprocessor > Element Type > Add/Edit/Delete

Click on Add.

In library of element types Menu

Select Link & 2DSPar1.

Click OK.

Elements type Menu .

Click close.

3> Main Menu > Preprocessor > Real Constants >Add/Edit/Delete.

Real Constant Menu.

Click on add.

OK.

Real Constant set Numbers 1,for Link 1.

C/S Area=500.

OK.

Close.

4> Main Menu > Preprocessor > Mat Prop > Material Models

Define Material Model Behavior Menu

Double click on structural.

Double click on linear.

Double click on elastic.

Double click on isotropic.

Linear isotropic properties for Mat No 1.

Ex = 2.e5

PRXY = 0.3

OK

5>Main Menu > Preprocessor > Modeling > Create > Key points > In active CS.

Create Key point in a CS menu.

Apply.

Enter X = 1000.

OK.

Modeling > Create > Lines > St.lines

Create straight line menu.

Click on end point of line.

OK.

6>Main Menu > Preprocessor > Meshing > Size cntrls >Manual size > lines > Picked Lines.

Element size on Picked Lines. Menu.

Click on line.

Click OK.

Element size on Picked Lines. Menu.

No of element Division = 10.

OK.

Main menu > Preprocessor > Meshing > Mesh tools.

Mesh tool Menu.

Click on Mesh.

Mesh line menu.

Pick the Line.

Press OK.

Press on save DB.

Main menu > Solutions > Define Loads > Apply > Structural > Displacement > On nodes.

Apply U,ROT on Nodes. Menu.

Select ALL DOF

OK

Solution > Define Loads > Apply > Structural > Force/moment > On nodes.

Pick the node where the load is to be applied.

OK.

Apply F/M on Nodes Menu.

Direction of Force FX.

Value 1000.

OK.

Save DB.

Solution > Solve > Current LS.

Solve current Load step Menu.

OK

Solution is done.

Close.

Main Menu > General Postprocessor > Plot Results > Contour Plot > Nodal Solution.

Contour Nodal Solution Data. Menu

Nodal Solution > DOF Solution > X-Comp of dispel.

OK.

General Post Processor >Element Table >Define Table.

Element Table Data Menu.

Add.

Define Additional Element Table Hems Menu.

User label for item [SAXL]

Select

By Sequence num, LS.

LS,1

OK.

Close .

GPP > Plot Results > contour Plot > Element table.

Contour Plot of Element Table Data Menu.

Item to be plotted SAXL

Yes – Average.

OK.

GPP > List Results > Reaction Solution.

List Reaction Solution.

Select All Items.

OK.

PRRSOL command.

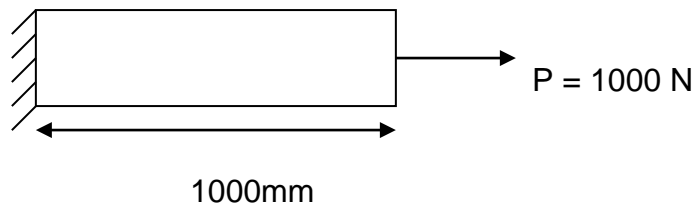
(note down the result)

.....XXXXXXXXXXXXX.....

(Students are required to solve problems theoretically and compare the answer with Ansys results)

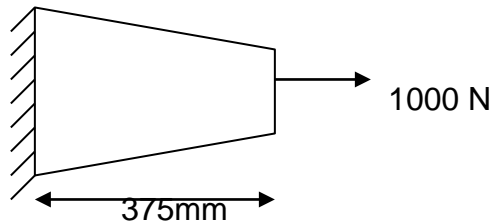
BAR PROBLEMS

1. For the simple bar shown in the figure determine the displacements, stress and the reaction. The cross section of the bar is 500mm^2 , length is 1000mm and the Young's Modulus is $E = 2 \times 10^5 \text{ N/mm}^2$. Take load $P = 1000\text{N}$.



2. For the tapered bar shown in the figure determine the displacement, stress and reaction in the bar.

Given: $A_1 = 1000\text{mm}^2$ and $A_2 = 500\text{mm}^2$. $E = 2 \times 10^5 \text{ N/mm}^2$.



3. Determine the nodal displacement and maximum stress in a 1D stepped bar made of steel and aluminum subjected to point load and thermal load with the boundary conditions as shown.

$$T_0 = 20^\circ\text{C} \text{ and } T_f = 80^\circ\text{C}$$

$$E_{\text{steel}} = 2 \times 10^5 \text{ N/mm}^2$$

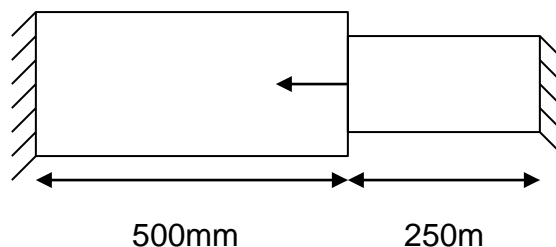
$$E_{\text{Al}} = 0.7 \times 10^5 \text{ N/mm}^2$$

$$\text{Area of steel, } A_{\text{steel}} = 1000\text{mm}^2$$

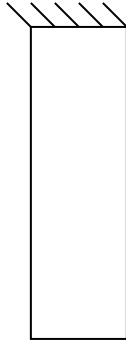
$$\text{Area of aluminum, } A_{\text{Al}} = 500\text{mm}^2$$

$$\alpha_{\text{steel}} = 11.7 \times 10^{-6}/^\circ\text{C}$$

$$\alpha_{\text{aluminum}} = 23 \times 10^{-6}/^\circ\text{C}$$

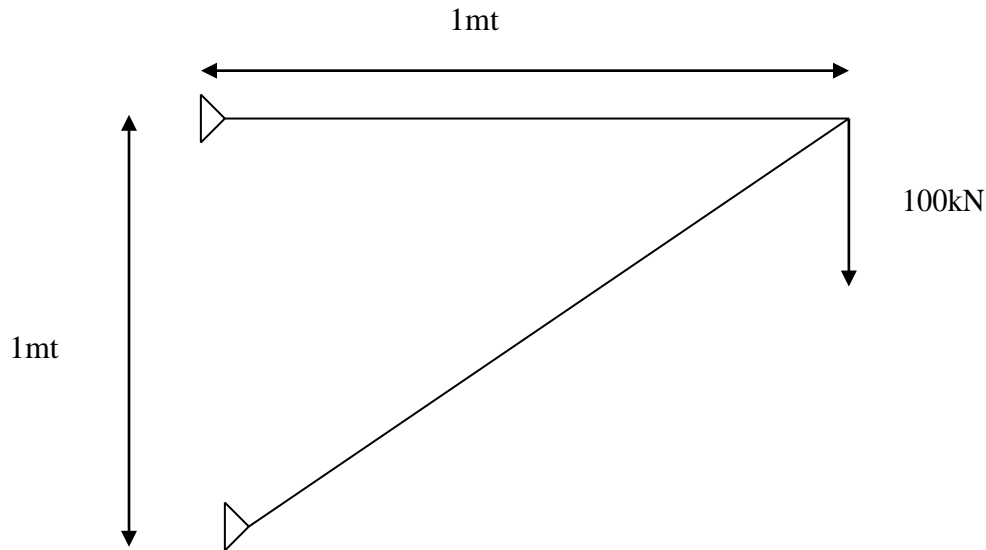


4) For the simple bar shown in the figure determine the displacement, strain, stress caused due to self weight.
Given length of the bar is 0.5m, cross-section area of the bar is 0.1m^2 , $P = 7848\text{ kg/m}^3$ and Young's Modulus $2 \times 10^{11}\text{ N/m}^2$.



TRUSSES

- 1) Two bar truss shown in Fig. Determine nodal displacement & the stress in each member.
Take $E = 210 \text{ GPa}$, Area = 0.01 sq.m .



Main Menu > Preferences > Structural
OK

Main Menu > Preprocessor > Element type > Add/Edit/Delete
Element types Menu
Click on Add
Library of Element types Menu
Select, Link, 2Dspar1
OK
Close

Preprocessor > Real Constants > Add/Edit/Delete
Real Constants Menu
Click on Add
OK
Real constant Set number 1, per Link 1
C/s area 0.01
OK
Real Constant Menu
Close

Preprocessor > Material Properties > Material Models
Define Material Model behavior Menu
Double Click on structural
Double Click on Linear

Double Click on Elastic

Double Click on Isotropic

Linear Isotropic.... Menu

Ex = 210e9

PRXY = 0.3

OK

Close the Menu

Preprocessor > Modeling > Create > Key points > in Active CS

Create Key points....Menu

XYZ location [0] [0] []

Apply

XYZ location [0] [1] []

Apply

XYZ location [1] [1] []

OK

Modeling > Create > Lines > Lines > Straight Line

Create Straight line Menu

(click on key points)

OK

Preprocessor > Meshing > Size Controls > Manual Size > Lines > All Lines

Element Sizes on All Selected Lines Menu

No. of element Division [1]

OK

Meshing > Mesh tool

Mesh tool Menu

Click on Mesh

Mesh Lines Menu

Pick the Lines to be meshed

Click OK

Solution > Define loads > Apply > Structural > Displacement > on Nodes

Apply V, RoT on nodes menu.

Pick Nodes for displacement constraints.

OK

Apply V, RoT on Nodes

All DOF

OK

Solution > Define Loads > Apply > Structural > Force/moment > on Nodes

Apply F/M on Nodes Menu

Pick Nodes for force/moment loading

OK

Apply F/M on Nodes Menu

Direction of Force/Moment Fy

Value -100000

OK

Solution > Solve > Current LS

Solve Current load setup

OK

Solution is Done !

Close

----- .

Main Menu > General Postprocessor > Plot Results > Contour Plot > Nodal solutions

Contour Nodal Solutions Data Menu

Nodal Solution > DOF Solution > Y-component of Displacement

OK

----- .

General Postprocessor > Element Table > Define Table

Element Table Data Menu

Click on Add

Define Additional Element table items

User label for item [] ANY

Select by Sequence number, LS

[LS, 1]

OK

Close

General Post processing > Plot Results > Contour Plot > Element Table

Contour Plot of Element Table Data

Item to be plotted [] Any

[Yes – average]

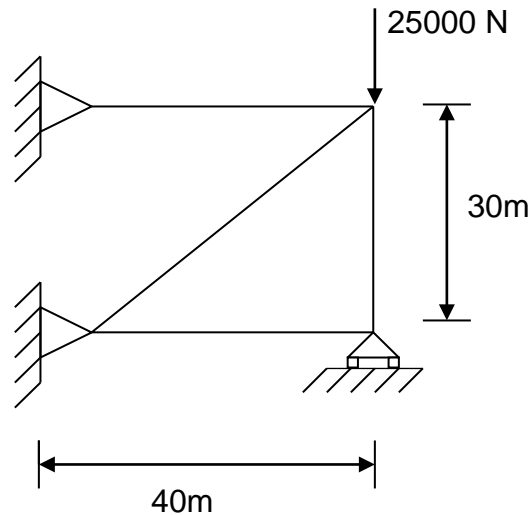
Note down : OK

DMX =

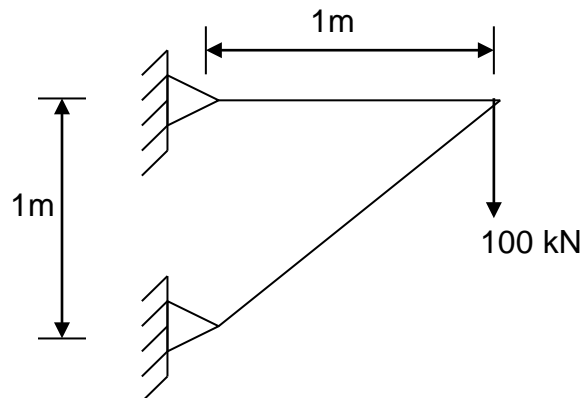
SMN =

TRUSSES

1. Consider the four bar truss shown in figure. It is given that $E = 29.5 \times 10^6 \text{ N/m}^2$ and $A = 1 \text{ m}^2$ for all the elements. Determine the deformation, maximum stress and the reaction forces in the truss system.



2. Two bar truss shown in figure. Determine the nodal displacements and stress in each member. Take $E = 210 \times 10^9 \text{ Pa}$, area $A = 0.01 \text{ m}^2$



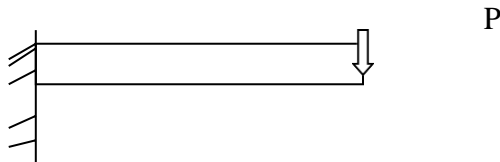
(Students are required to solve problems theoretically and compare the answer with Ansys results)

BEAM

- (1) A 2m long cantilever with rectangular section (60mm×90mm) is subjected to concentrated load of 4kN at the free end of the beam. Determine the deflection, maximum stress due to bending, Bending moment & Shear force, take

$$E=200\text{Gpa}$$

$$P=4\times 10^3\text{ N, Length }L=2\text{m}=2000\text{mm}$$



$$\text{Deflection}=y=PL^3/3EI=14.6\text{mm}$$

$$\text{Maximum stress}=\sigma=M/Z=6PL/BD^2=98.765\text{N/mm}^2$$

PROCEDURE:

- 1) Main Menu > Preferences

Select structural

OK.

- 2) Main Menu>Preprocessor>Element type>Add/Edit/Delete

Element types Menu

Click on Add

Library of Element Types Menu

Select Beam, 2D Elastics

Click OK

Element Types Menu

Close.

- 3) Main menu>Real constants>Add/Edit/Delete

Real constant Menu

Click on Add

Element Types for..... Menu

Type1. BEAM 3

Click OK

Real constants for BEAM 8Menu

C/S AREA 5400

Area moment of Inertia $I_{z2}=3645\text{ e}^8$

Total beam height HEIGHT=90

OK

Close.

4) Preprocessor>Material props>Material Models

Define Material Model Behavior

Structural-Linear-Elastic-Isotropic (Double click)

Linear Isotropic properties for Material Numb. Menu

$EX = 2e^5$

OK

Close the Model Behavior Menu.

5) Pre-processor>Modeling>Create>In Active CS

Create Key points in ACS Menu

(Manually, Enter the key point co-ordinates in the window)

KP 1 (0, 0)

KP 2 (2000, 0)

6) Pre-processor> Modeling>Create>Lines>Straight Lines

Create Straight Lines Menu

(Pick end key points of line)

OK.

7) Pre-processor>Meshing>Mesh Tool

Mesh Tool Menu

Click on Mesh

Mesh Lines Menu

(Pick lines to be Meshed)

Click OK

Click on SAVE_DB.

8) Solution>Define Loads> Apply>Structural>Displacement>On Nodes

Apply U, ROT on Nodes Menu

(Pick the node for displacement constraint)

OK

Apply U, ROT on Nodes Menu.

Select All DOF

VALUE Displacement value O.

OK.

Displacement>Force/Moment>On Key Points

Apply F/M on KPS Menu

(Pick Key points for force/moment loading)

OK

Apply F/M on Nodes Menu.

Direction of Force/Moment.FY

VALUE Force/Moment value..... 4e3

OK.

9) Solution>Solve>Current LS

Solve Current load step – Menu

(The window above shows the input that is used in pre-processor and solution part.

Make sure that the data given above are correct and then press OK)

OK.

10) General Post Processor>Plot Results>Contour Plot>Nodal Solution

(Contour Nodal Solution Data Menu)

Nodal Solution>DOF Solution>Y-Component of Displacement

OK.

11) Plot controls>Style>Size and Shape. (From Utility Menu).

Size and Shape Menu>Display of Element ON

OK.

12) General Post Processor>Contour Plot>Nodal Solution

Contour Nodal Solution Data Menu

Nodal Solution>Stress>Von Mises Stress

OK.

13) General Post Processor>Element Table>Define Table

Element Table Data Menu

Click on Add

Define Additional Element Table Items Menu

Use label for item. MMOMZI

Select: By sequence numSMISC

&

Apply

Define Additional Element Table items

User label for item.... MMOMZI

Select: By sequence num... SMISC

:

OK.

(Note the change in Element Table Data).

14) General Post Processor>Plot Results>Contour Plot>Line Element Results

Plot Line-Element Results Menu

Element table item at node I: MMOMZI

Element table item at node J: MMOMZI

OK.

(you see Bending Moment Diagram)

15) General Post Processor>Element Table>Define Table

Element Table Data Menu

Click on Add

Define Additional Element Table Items

User Label for item... MFORYI

Select: By Sequence num, SMISC

: SMISC,
2
APPLY

Define Additional Element Table items Menu

User label for item: MFORYJ

Select: By Sequence num, SMISC

: SMISC, 8
OK.

(Observe the change in element table data)

CLOSE.

16) Plot Line Element Results Menu

Element table item at node I: MFORYI

Element table item at node J: MFORYJ

OK.

(Note the changes on screen)

17) General Post Processor > List Results > Reaction Solution

List Reaction Solution Menu

Select All Items

OK.

(Note the changes on screen)

Result Comparison

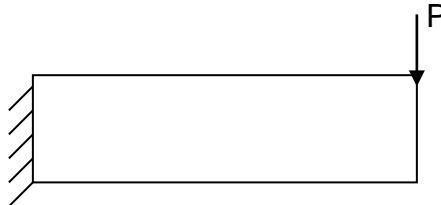
	Ansys	Theoretical
Deformation:	14.642mm	14.63mm
Stress:	98.765N/mm ²	98.765N/mm ²

.....XXXXXXXXXXXXXXXXXXXX.....

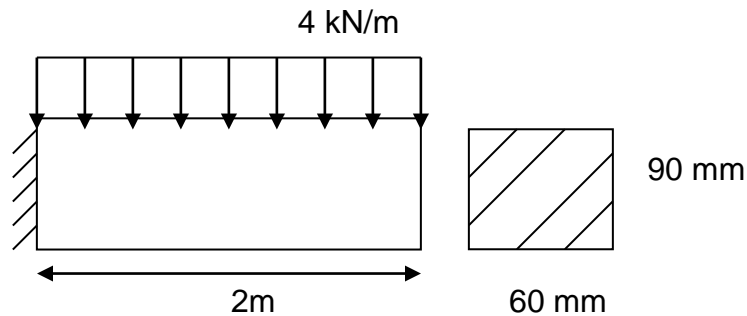
(Students are required to solve problems theoretically and compare the answer with Ansys results)

BEAM PROBLEMS

1. A 2 m long cantilever with rectangular section (60mm x 90 mm) is subjected to concentrated load of 4 kN at the free end of the beam. Determine the deflection, maximum stress due to bending, bending moment and shear force. Take $E = 200 \text{ GPa}$, $P = 4 \times 10^3 \text{ N}$, Length $L = 2 \text{ m} = 2000 \text{ mm}$.



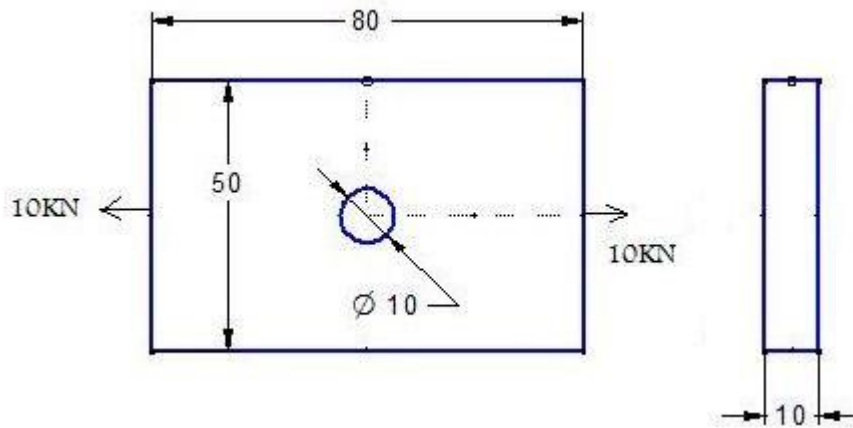
2. A 2m long cantilever with rectangular section (60mm x 90 mm) is subjected to uniformly distributed load 4 kN/m through out its length. Determine maximum bending stress and its deflection. Take $E = 200 \text{ GPa}$.



(Students are required to solve problems theoretically and compare the answer with Ansys results)

PLATE WITH A HOLE

- 1) Determine the maximum stress for a rectangular plate of 50mm×80mm with hole of 10mm diameter in the centre is loaded in axial tension. Thickness of the plate is 10mm. Take $E=100\text{Gpa}$. All dimensions in mm.



$$\sigma_{\text{nor}} = F/(w-a)t = 10000/(50-10)10 = 25\text{N/mm}^2$$

$$\text{Stress Concentration factor } K_{\sigma} = \sigma_{\text{max}}/\sigma_{\text{norm}}$$

$$2.5 = \sigma_{\text{max}}/25$$

$$\sigma_{\text{max}} = 2.5 \times 25 = 62.5\text{N/mm}^2$$

- 1) Main Menu > Preprocessor

Preferences for GUI Filtering Menu

Select: Structural

OK.

- 2) Preprocessor > Element Type > Add/Edit/Delete

Element Types Menu

Click on Add

Library of Element Types Menu

Select: Solid, Quad 4 node42

OK

Element Types Menu

Type1 PLANE42

Click on OPTIONS

PLANE42 element type options Menu

Element behavior K3: Plane stress W/thk

OK

CLOSE.

3) Preprocessor > Real Constants > Add/Edit/Delete

Real Constants Menu

Add

Element Type forMenu

Type1 PLANE42

OK

Real constant set Number 1, for PLANE42 Menu

Thickness THK : 10

OK

Real Constants Menu

Set 1

CLOSE

4) Preprocessor>Material Props>Material Models

Define Material Model Behavior Menu

Material Model Number 1

Structural-Linear-Elastic-Isotropic

Linear Isotropic properties for Material Num. Menu

EX : 200000

OK

Material Model Number 1

- Linear Isotropic

Enter the Key points in the coordinate box as shown below

KP	X	Y
1	0	0
2	40	0
3	40	25
4	0	25

Create Key points in Active Co-ordinate System Menu

(Enter all the Key points as usual in the Menu then press Ok)

OK.

5) Preprocessor>Modeling>Create>Areas>Through KPS

(Pick all KPS defining the area)

OK

(Observe the change in screen i.e. Rectangle)

Modeling>Create>Areas>Circle>Solid Circle

Solid circular Area Menu

Radius: 5

OK

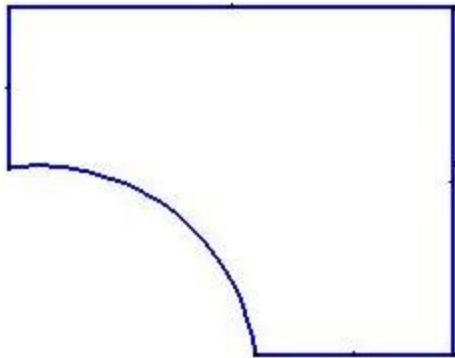
Modeling>Create>Operate>Booleans>Subtract>Areas

Subtract Areas Menu

(i) Pick base areas from which to subtract.

(ii) Pick areas to be subtracted.

OK.



6) Preprocessor>Meshing>Mesh Tools

Mesh Tool Menu

Click on Mesh

Mesh Areas Menu

(Pick areas to be Meshed)

OK

(Note the Meshing on the Screen)

Preprocessor>Meshing>Mesh Tool

Mesh Tool Menu

Click on Refine

Refine Mesh at elements Menu

Click on Pick All

Refine Mesh at Element Menu

Level of refinement: 3

OK

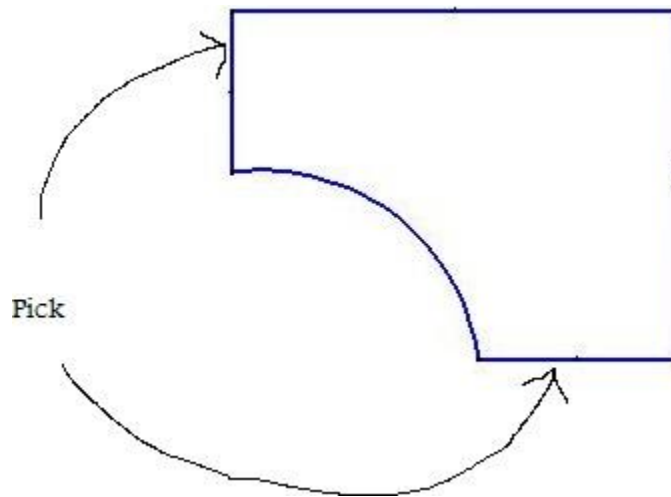
(Observe the change in Meshing on screen)

: SOLVER:

- 7) Preferences>Preprocessor>Solution>Define Loads>Apply>Structural>Displacement>Symmetry BC>On Lines.

Apply SYMM on Lines Menu

(Pick or enter lines for symmetry condition)

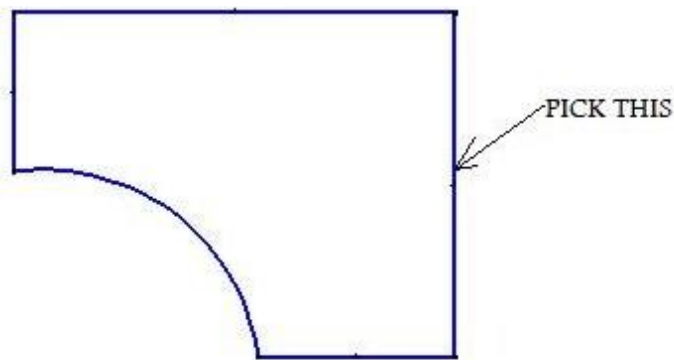


OK

- 8) Solution>Define Loads >Apply>Structural>Pressure>On Lines

Apply PRES on Lines Menu

(Pick the line for Pressure Loading)



OK

Apply PRES on Lines Menu

VALUE Load PRES Value: -20

OK.

9) Solution > Solve > Current LS

Solve Current Load Step Menu

(The window above shows the input that is used in preprocessor and solution part. Make sure the data given above are correct and then press OK)

Solution in done!

STATUS Command Menu

Press

General Post processor

Solution > General Post processor > Plot results > Contour plot > Nodal Solution

Contour Nodal Solution Data Menu

Stress

Select Von Mises stress

OK

(Observe the changes on screen)

Contour Plot > Nodal Solutions

Contour Nodal Solution Data Menu

Nodal Solution > DOF Solution > X-component of displacement

OK

(Note the changes on screen)

General Post processor > List Results > Reaction Solution

PRRSQL Command Menu

(Note down the values from the Menu)

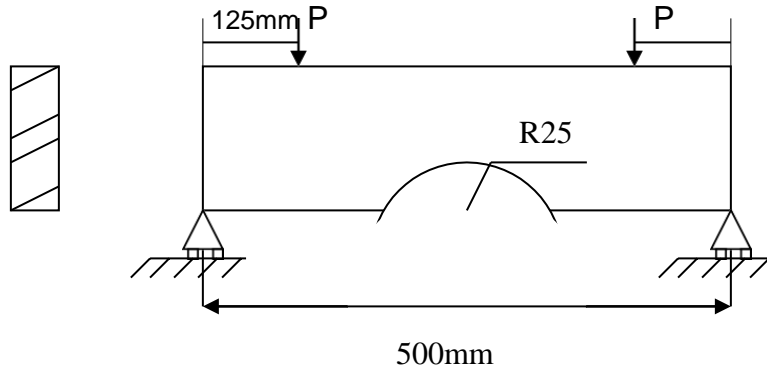
Results Comparison

	FEM	Theoretical
Deformation	0.004329mm	_____
Stress	62.244N/mm ²	6.25N/mm ²
XXXXXXXXXXXXXXXXXXXX		

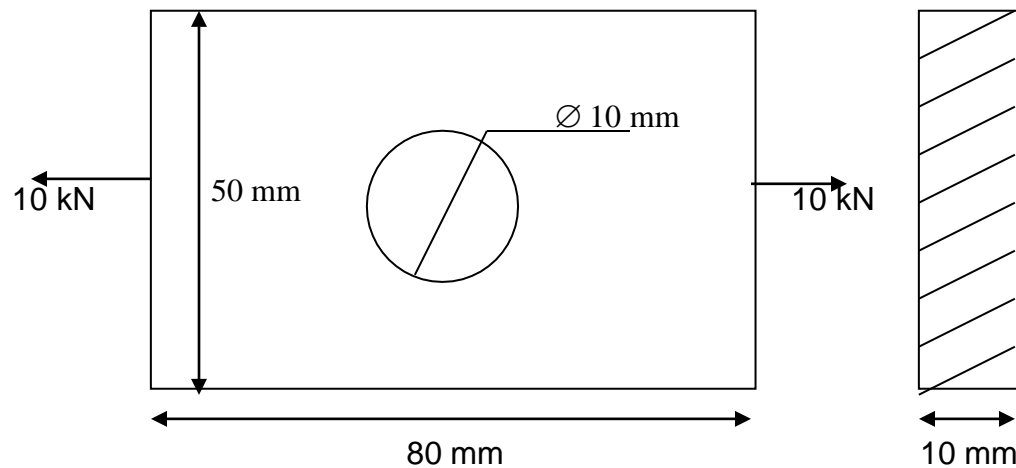
(Students are required to solve problems theoretically and compare the answer with Ansys results)

PLATE PROBLEMS

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



2. Determine the maximum stress for a rectangular plate of 50mm x 80mm with a hole of 10mm diameter in the center is loaded in axial tension. Thickness of the plate is 10mm. Take $E = 200\text{GPa}$.



Heat transfer analysis using ANSYS

The finite element solution performed via ANSYS calculates nodal temperatures. The nodal temperatures are then used to obtain thermal quantities of interest such as:

- . The temperature distributions
- . The amount of heat lost or gained
- . Thermal fluxes.

The heat balance equation obtained from the principle of conservation of energy is the basis for thermal analysis in ANSYS

The ANSYS program handles all three primary modes of heat transfer: conduction, convection, and radiation.

Types of thermal analysis:

1. Steady-state thermal analysis: Determines the temperature distribution and other thermal quantities in an object that are caused by thermal loads do not vary over time. Such loads include the following:

- . Convections
- . Radiation
- . Heat flow rates
- . Heat fluxes (heat flow per unit area)
- . Heat generation rates (heat flow per unit volume)
- . Constant temperatures boundaries

A steady-state thermal analysis may be either linear. With constant material properties; or nonlinear, with material properties that depend on temperature. The thermal properties of most material do vary with temperature, so the analysis usually is nonlinear.

2. Transient thermal analysis: determines the temperature distribution and other thermal quantities under conditions that vary over a period of time.

Element library: The ANSYS and ANSYS Professional programs include about 40 elements for performing steady-state thermal analyses. Commonly used element types are

LINK 31 – Radiation Link
LINK 32 – 2-D Conduction Bar
LINK 33 – 3-D Conduction Bar
LINK 34 – Convection Link
PLANE 35 – 2-D 6 – Node Triangular Thermal solid
PLANE 55 – 2 - D Thermal Solid
SHELL 57 – Thermal shell
SOLID 70 – 3-D Thermal solid
MASS 71 – Thermal Mass
PLANE77 – 2-D 8-Node Thermal Solid
SOLID 87 – 3-D 10 – Node Tetrahedral Thermal Solid
SOLID90 – 3-D 20- Node Thermal Solid

Typical Units in ANSYS (SI)

. Temperature:	C (or K)
. Heat Flow:	Watts
. Thermal Conductivity:	Watts/ (meter – C)
. Density:	Kilogram/ (meter ³)
. Specific Heat :	(Joule/ (Kilogram- C)
. Film Coefficient:	Watt/(meter)
. Heat flux:	Watt/(meter)
. Thermal Gradients:	C / meter
. Internal Heat Generation:	Watt / (meter)

Thermal loading conditions:

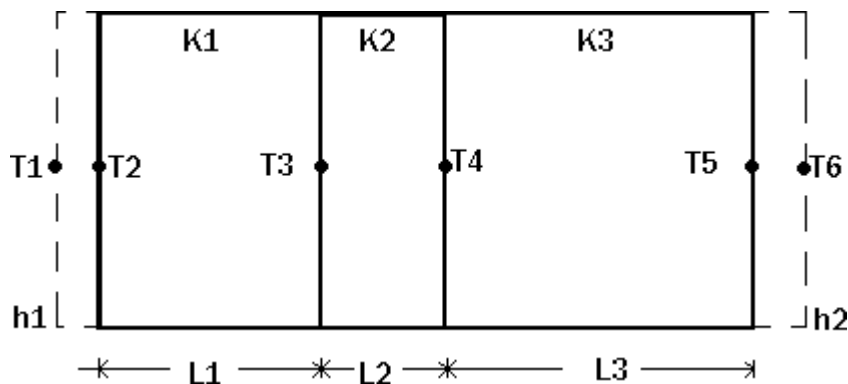
Temperatures	Regions of the model where temperatures are known.
Convections	Surfaces where heat is transferred to (or from) surroundings by means of convection. Input consists of film coefficient h and bulk temperature of the surrounding fluid Tb.
Heat flux	Surfaces where the heat flow rate per unit area is known.
Heat flow	Points where the heat flow rate is known.
Heat generation	Regions where the volumetric heat generation rate is known.
Radiation	Surfaces where heat transfer occurs by means of radiation. Input consists of emmissivity, Stefan-Bolt Mann constant, and optionally, temperature at a space node.
Adiabatic surfaces	Perfectly insulated surfaces where no heat transfer takes place .

Problem 1 :

Composite wall 1 :

A furnace wall is made of inside silica brick($k = 1.5 \text{ W/mK}$) and outside of magnesia brick($k = 4.9 \text{ W/mK}$), each 10cm thick. The inner and outer surfaces are exposed to fluids at temperatures at 820°C and 110°C respectively. The contact Resistance is $0.001 \text{ m}^2\text{K/W}$. The heat transfer co-efficient for the inner and outer surface is $35 \text{ W/m}^2\text{K}$. Find the heat flow through the wall per unit area per unit time and temperature distribution across the wall.

(The students are expected to solve this problem manually & compare the results with ANSYS.)



$$K_1 = 1.5 \text{ W/mK}$$

$$K_3 = 4.9 \text{ W/mK}$$

$$L_1 = L_2 = 100 \text{ mm}$$

$$T_1 = 820^\circ\text{C}$$

$$T_6 = 110^\circ\text{C}$$

$$R_c = 0.001 \text{ m}^2\text{K/W}$$

$$h_1 = h_2 = 35 \text{ W/m}^2\text{K}$$

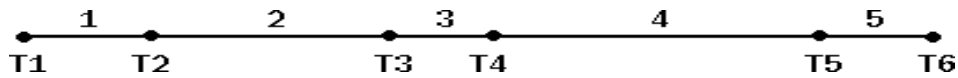
$$R_c = \frac{L_2}{K_2 A}$$

Take $L_2 = 1 \text{ mm}$

$$K_2 = 1 \text{ W/mK}$$

$$A = 1 \text{ m}^2$$

Procedure :



For FE Model :

General Procedure to Save :

Desktop > Program > ANSYS > Ansys product Launcher

Working Directory :

Job Name : .ht01

(1) Main Menu > Preferences

Select : Thermal

OK

(2) Preprocessor > Element Type > Add/Edit/Delete

Element Types Menu

Add

Library of element types Menu

Select : Link, 2D Conduction 32

Element type reference number : 1

Apply

Select : Link Convection34

Element type reference number : 2

OK

(3) Preprocessor > Real Constants >

Element type for Real constant Menu

Select Type 1 LINK 32

OK

Real Constant Set Number 1, for LINK 32

Real Constant Set No. 1

C/s area AREA 1

OK

In Element Type for Real Const. Menu

Select : Type 2 LINK 32

OK

Real Const. Set No.2, for LINK 34

Real Const. Set No. : 2

Convection Surface AREA : 1

OK

Real Constants Menu

Select Set 2

Add

(4) Preprocessor > Material Properties > Material Models

Define Material Model behaviour Menu

Material Model Number 1

Thermal > Conductivity > Isotropic

Conductivity for Material Number 1 Menu

KXX = 1.5

OK

Material Properties > Material Models

Define Material Model Behavior Menu

Material > New Model

Material Model Number 1

Define Material ID

Define Material ID : Z

OK

In Define Material Model behavior Menu

Material Model Number 2

Thermal > Conductivity > Isotropic

Conductivity for Material Number 2 Menu

KXX = 1

OK

Material Model Number 3

Conductivity for Material Number 3 Menu

KXX = 4.9

OK

Material Model Number 4

Material Model Available

Thermal/Convection or Film Co-ef..

Convection / Film co-efficient for material.....Menu

HF : 35

OK

(5) Preprocessor > Modeling > Create > Nodes > in Active CS

Create Nodes in Active co-ordinate System

Node Number : 1

XYZ Location in Active CS : 0, 0, 0

Apply

Node Number : 2

XYZ Location in Active CS : 0.001, 0, 0

Apply

Node Number : 3

XYZ Location in Active CS : 0.101, 0, 0

Apply

Node Number : 4

XYZ Location in Active CS : 0.102, 0, 0

Apply

Node Number : 5

XYZ Location in Active CS : 0.202, 0, 0

Apply

Node Number : 6

XYZ Location in Active CS : 0.203, 0, 0

Apply

(6) Modeling > Create > Elements > Element Attributes

Element Attributes Menu

Element type Number : 2 LINK34

Material Number 4

Real Constant Set No. : 2

OK

Modeling > Create > Elements > Auto Numbered > Through Nodes

Elements from Nodes Menu

(Pick the Nodes defining the Element)

OK

Create > Elements > Element Attributes

Element Attributes Menu

Element type Number : 1 LINK32

Material Number 1

Real Constant Set No. : 1

OK

Modeling > Create > Elements > Auto Numbered > Through Nodes

Elements from Nodes Menu

(Pick the Nodes defining the Element)

OK

Create > Elements > Element Attributes

Element Attributes Menu

Element type Number : 1 LINK32

Material Number 2

Real Constant Set No. : 1

OK

Modeling > Create > Elements > Auto Numbered > Through Nodes

Elements from Nodes Menu

(Pick the Nodes defining the Element)

OK

Create > Elements > Element Attributes

Element Attributes Menu

Element type Number : 1 LINK32

Material Number 3

Real Constant Set No. : 1

OK

Modeling > Create > Elements > Auto Numbered > Through Nodes

Elements from Nodes Menu

(Pick the Nodes defining the Element)

OK

Create > Elements > Element Attributes

Element Attributes Menu

Element type Number : 2 LINK34

Material Number 4

Real Constant Set No. : 2

OK

Modeling > Create > Elements > Auto Numbered > Through Nodes

Elements from Nodes Menu

(Pick the Nodes defining the Element)

OK

Preprocessor > Modeling

Click PlotCtrls.(Utility Menu)

Elem/Attributes numbering : Element Numbers

OK

(7) Solution > Analysis Type > New Analysis

New Analysis Menu

Select : Steady State

OK

(8) Solution > Define loads > Apply > Thermal > Temperature > on Nodes

Apply TEMP on Nodes Menu(Pick nodes for Temp. Specification)

(Select 1)

OK

Apply TEMP on Nodes Menu

DOF to be constrained : TEMP

VALUE Load TEMP value : 820

Apply

Apply TEMP on Nodes Menu(Pick nodes for Temp. Specification)

(Select : 5)

OK

Apply TEMP on Nodes Menu

Select TEMP

VALUE Load TEMP value : 110

OK

(9) Solution > Solve > Current LS

Solve Current load setup

OK

Solution is Done !

Close

(10) Main Menu > General Postprocessor > Read results > Last Set

General Postprocessor > List Results > Nodal Solution

List Nodal Solution Menu

Nodal Solution/DOF Solution/Temperature

OK

PRNSOL Command Menu

(Note down the Temp. value from the Menu)

Node Temp.

1 -

2 -

3 -

4 -

5 -

6 -

PlotCtrls > Styles > Size & Shape (Utility Menu)
Size & Shape Menu
Display of Element [] on
Real constant multiplier 0.2
OK

(11) General Postprocessing > Plot Results > Contour Plot > Nodal Solution
Contour Nodal Solution Data Menu
Nodal Solution/DOF Solution/Temperature
OK

(12) General Postprocessor > Element Table > Define Table
Element Table Data Menu
Click on Add
Define Additional Element table items
User label for item : htrans
Select : by Sequence number, SMISC
[SMISC 1]
OK
Close

Element Table > List Element Table
List Element Table data
Lab 1-9 items to be listed : HTRANS
OK
Element Table > List Element Table
PRETAB Command

(Note down the value from menu as shown below)

STAT	CURRENT
ELEM	HTRANS
1	
2	
3	
4	
5	

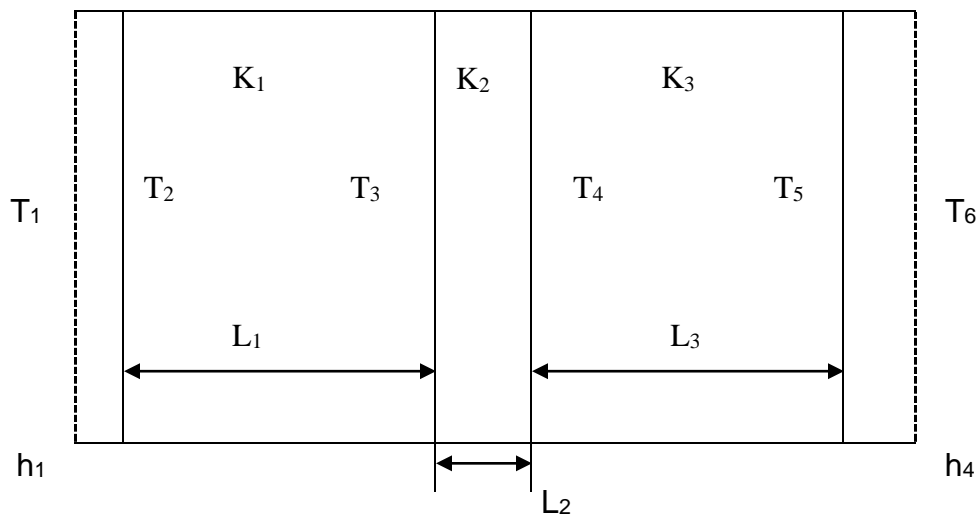
Comparison of Results

Quantity	ANSYS	Theory
T2	680.31°C	680.31°C
T3	354.36°C	354.36°C
T4	349.47°C	349.47°C
T5	249.69°C	249.69°C
Heat Flow	4889.2W/m ²	4889.21W/m ²

.....XXXXXXXXXXXXXXXXXXXXXXXXXXXX.....

HEAT TRANSFER PROBLEMS

1. A furnace wall is made of inside silica brick ($k = 1.5 \text{ W/mK}$) and outside magnesia brick ($k = 4.9 \text{ W/mK}$), each 10cm thick. The inner and outer surfaces are exposed to fluids at temperatures of 820°C and 110°C respectively. The contact resistance is $0.001\text{m}^2 \text{ K/W}$. the heat transfer coefficient for inner and outside surfaces is equal to $35 \text{ W/m}^2\text{K}$. Find the heat flow through the wall per unit area per unit time and temperature distribution across the wall. (March/April 2001)



$K_1 = 1.5 \text{ W/mK}$
 $K_3 = 4.9 \text{ W/mK}$
 $L_1 = L_2 = 100\text{mm}$
 $T_1 = 820^\circ\text{C}$

$$T_6 = 110^\circ\text{C}$$

$$R_c = 0.001 \text{ m}^2\text{K/W}$$

$$h_1 = h_4 = 35 \text{ W/m}^2\text{K}$$

$$R_c = L_2/K_2A$$

$$\text{Take } L_2 = 1\text{mm},$$

$$K_2 = 1 \text{ W/mK}$$

$$A = 1\text{m}^2$$

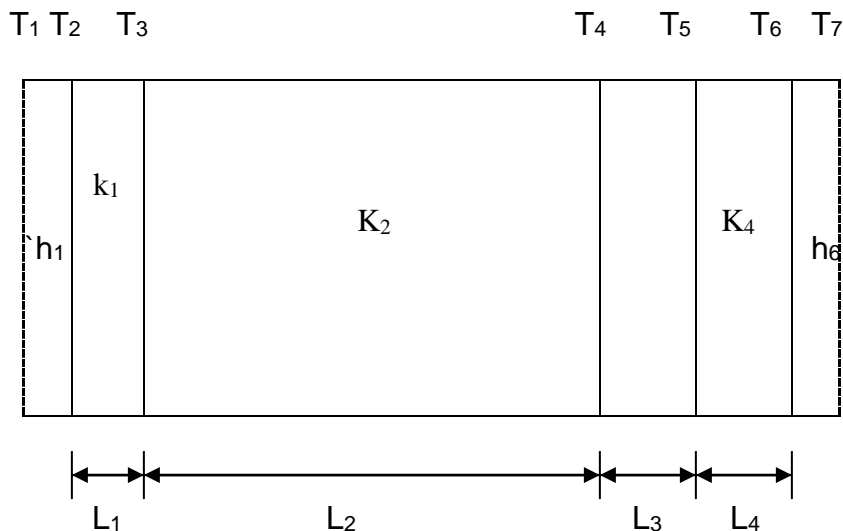
2. The exterior wall of a building is constructed of four materials: 12mm thick gypsum board, 75mm thick fiber glass insulation, 20mm thick plywood and 20mm thick hard board. The inside and outside air temperatures are 20°C and -10°C respectively. The convective heat transfer coefficients on the inner and outer surfaces of the wall are $6 \text{ W/m}^2\text{K}$ and $10 \text{ W/m}^2\text{K}$ respectively. Determine the heat flux and the temperature distribution. Take,

$$K \text{ for gypsum } 0.176 \text{ W/mK}$$

$$K \text{ for fiber glass } 0.036 \text{ W/mK}$$

$$K \text{ for plywood } 0.115 \text{ W/mK}$$

$$K \text{ for hard board } 0.215 \text{ W/mK}$$



$$h_1 = 6 \text{ W/m}^2\text{K}$$

$$K_1 = 0.176 \text{ W/mK}$$

$$K_2 = 0.036 \text{ W/mK}$$

$$K_3 = 0.115 \text{ W/mK}$$

$$K_4 = 0.215 \text{ W/mK}$$

$$h_6 = 10 \text{ W/m}^2\text{K}$$

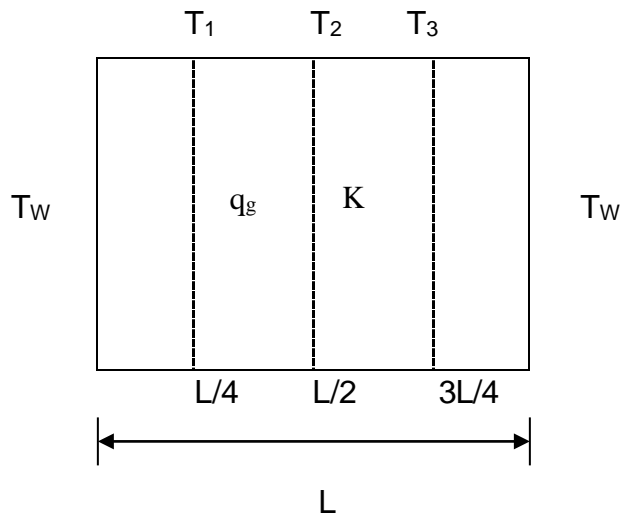
$$L_1 = 12\text{mm}$$

$$\begin{aligned} L_2 &= 75\text{mm} \\ L_3 &= 20\text{mm} \\ L_4 &= 20\text{mm} \end{aligned}$$

$$\begin{aligned} T_1 &= 20^\circ\text{C} \\ T_6 &= -10^\circ\text{C} \\ A &= 1\text{m}^2 \end{aligned}$$

3. The rate of heat generation in a slab of thickness 160mm ($K = 180 \text{ W/mK}$) is $1.2 \times 10^6 \text{ W/m}^3$. if the temperature of each surface of the solid is 120°C , compute and plot the temperature distribution across the wall and find
1. Temperatures at the mid and quarter planes.
 2. Temperature gradients at the mid and quarter planes.

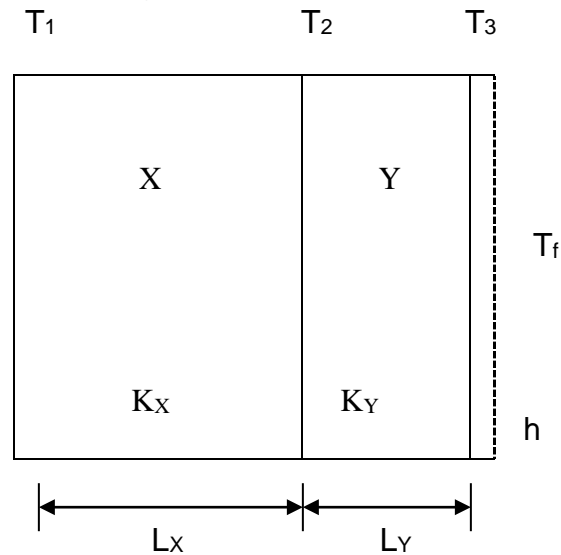
Plane wall with heat generation.



$$\begin{aligned} K_x &= 180 \text{ W/mK} \\ L &= 160\text{mm} \\ T_w &= 120^\circ\text{C} \\ q_g &= 1.2 \times 10^6 \text{ W/m}^3 \\ A &= 1\text{m}^2 \end{aligned}$$

4. A plane wall 'X' ($K = 75 \text{ W/mK}$) is 60mm thick and has volumetric heat generation of $1.5 \times 10^6 \text{ W/m}^3$. it is insulated on one side while the other side is in contact with surface of another wall 'Y' ($K = 150\text{W/mK}$) which is 30mm thick and has no heat generation. The free surface of wall 'Y' is exposed to a cooling fluid at 20°C with a convection coefficient of $950 \text{ W/m}^2\text{K}$. Find steady state temperatures at salient points across the composite wall.

Composite wall with heat generation.



$$K_X = 75 \text{ W/mK}$$

$$K_Y = 150 \text{ W/mK}$$

$$L_X = 60 \text{ mm}$$

$$L_Y = 30 \text{ mm}$$

$$T_f = 20^\circ\text{C}$$

$$q_g = 1.5 \times 10^6 \text{ W/m}^3$$

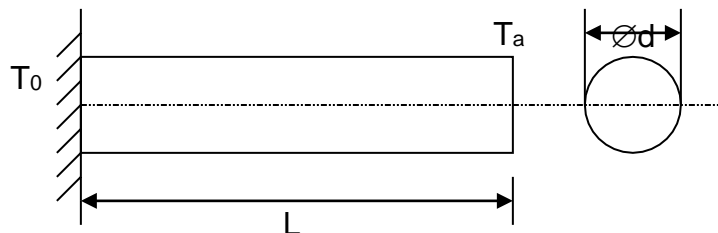
$$h = 950 \text{ W/m}^2\text{K}$$

$$A = 1 \text{ m}^2$$

5. Current passes through a stainless steel wire of 2.5mm diameter ($K = \text{W/mK}$) causing volumetric heat generation of $26.14 \times 10^8 \text{ W/m}^3$. The wire is submerged in a fluid maintained at 50°C and convective heat transfer coefficient at the wire surface is $4000 \text{ W/m}^2\text{K}$. find the steady state temperature at the centre and at the surface of the wire.

6. PIN FIN:

A rod of 6cm diameter with $K = 98 \text{ W/mK}$ and 125cm long is attached to an evaporation chamber maintained at -15°C . The film coefficient of heat transfer is $40 \text{ W/m}^2\text{K}$ and the ambient temperature is 28°C . Compute and plot the temperature distribution along the length of the fin and find the length up to which there will be ice formation.

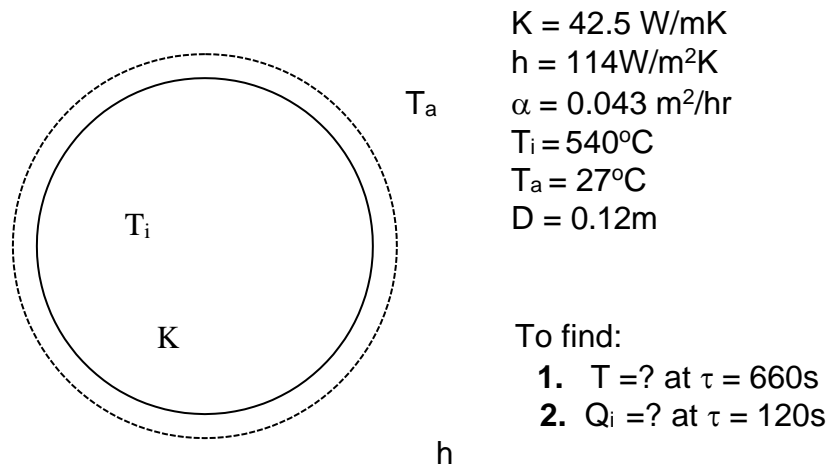


$K = 98 \text{ W/mK}$
 $L = 125 \text{ cm}$
 $T_0 = -15^\circ\text{C}$
 $h = 40 \text{ W/m}^2\text{K}$
 $d = 6 \text{ cm}$
 $T_a = 28^\circ\text{C}$

7. A 12cm diameter mild steel sphere ($K = 42.5 \text{ W/mK}$) at 540°C is placed in an air stream at 27°C and the corresponding heat transfer coefficient is $114 \text{ W/m}^2\text{K}$. Calculate:
1. Temperature of the sphere after 10 min and
 2. Instantaneous heat transfer rate two minutes after the commencement of cooling.

The properties of mild steel are:
Density = 7850 kg/m^3
Specific heat = 475 J/kgK
Thermal diffusivity = $0.043 \text{ m}^2/\text{hr}$.

Transient heat transfer (lumped system)



(Students are required to solve problems theoretically and compare the answer with Ansys results)

Fluid flow analysis using ANSYS

The ANSYS / FLOTTRAN CFD (Computational Fluid Dynamics) offers comprehensive tools for analyzing two-dimensional and three-dimensional fluid flow fields. ANSYS is capable of modeling a vast range of analysis types such as: airfoils for pressure analysis of airplane wings (lift and drag), flow in supersonic nozzles, and complex, three-dimensional flow patterns in a pipe bend. In addition, ANSYS/FLOTTRAN could be used to perform tasks including:

- . Calculating the gas pressure and temperature distributions in an engine manifold.
- . Studying the thermal stratification and breakup in piping systems
- . Using flow-mixing studies to evaluate potential for thermal shock
- . Doing natural convection analyses to evaluate the thermal performance of chips in electronic enclosures
- . Conducting heat exchanger studies involving different fluids separated by solid regions

FLOTTRAN analysis provides an accurate way to calculate the effects of fluid flows in complex solids without having to use the typical heat transfer analogy of heat flux as fluid flow.

Types of FLOTTRAN analysis that ANSYS is able to perform include:

- . Laminar or Turbulent Flows
- . Thermal Fluid Analysis
- . Adiabatic Conditions
- . Free surface Flow
- . Compressible or incompressible Flows
- . Newtonian or Non-Newtonian Fluids
- . Multiple species transport

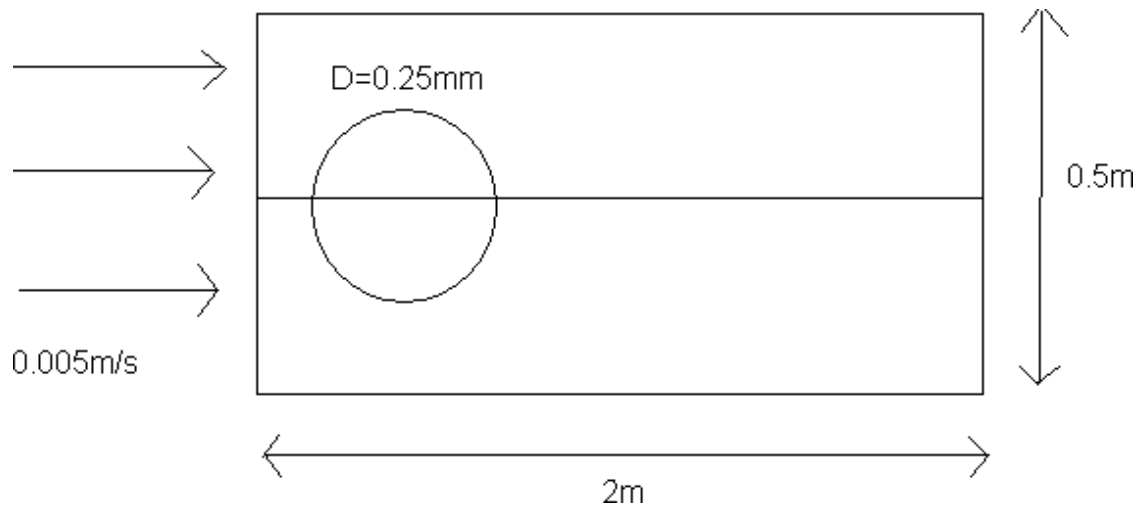
*NOTE: These types of analyses are not mutually exclusive. For example, a laminar analysis can be compressible or incompressible.

A typical FLOTTRAN analysis consists of seven main steps:

1. Determine the problem domain.
2. Determine the flow regime.
3. Create the finite element mesh.
4. Apply boundary conditions.
5. Set FLOTTRAN analysis parameters.
6. Solve the problem.
7. Examine the results.

FLOW OVER A CYLINDER;

1) Atmospheric air at 20DEG C flows with a velocity of 5mm/sec over a long horizontal cylinder of diameter 25cm. compute and plot the velocity distribution of air over the cylinder.



The cylinder is 0.25m in diameter.

Considering symmetry about horizontal, only the upper half of the cylinder is computed. The results are assumed to be the same below the X-axis. (axis of symmetry).

The arbitrary flow area considered is 2m by 0.5m.

The velocity of air at infinite distance from the plate is 5mm/sec (.005m/sec) (Laminar flow).

From utility menu.

File > change Jobname.

Change Jobname menu

Enter new Jobname: fluid001.

OK>

1. Preferences for GUI Filtering Menu.

Select FLOTRAN CFD.

h-Method.

OK.

2. Preprocessor > Element Type > Add/Edit/Delete.

Element Types Menu.

OK.

Select FLOTRAN CFD, @DFLOTRAN141.

OK.

3. Preprocessor > Modeling > Create > Areas > Rectangle > By 2 corners.

Rectangle by 2 corners.

WP X = 0.

WP Y = 0.

Width : 2

Height: 0.5

OK.

4. Modeling > Create > Areas > Circle > Solid Circle.

Solid Circular Area Menu.

WP X = 0.5

WP Y = 0

Radius = 0.125

OK.

5. Modeling > Operate > Boolean > Subtract > Areas.

Subtract Areas Menu.

(Pick areas from which to subtract).

Subtract Areas Menu.

(Pick areas to be subtracted).

OK.

6. Preprocessor > Meshing > Size Cntrl > Manual Size > Lines > All Lines.

Element sizes on all selected lines Menu.

Element edge length = 0.02.

OK.

7. Preprocessor > Meshing > mesh > Areas > Free.

Mesh Areas Menu.

(pick areas to be meshed)

OK.

(observe the meshing on screen).

8. Preprocessor > FLOTRAN set up > Fluid properties.

Fluid properties menu.

Density : AIR-SI.

Viscosity : AIR-SI.

OK.

CFD Flow properties Menu.

OK.

9. From Utility Menu.

Plot > Lines.

10. Preprocessor > Loads > Define Loads > Apply > Fluid/CFD > Velocity On lines.

Apply V on lines Menu.

(Pick lines for velocity specification)

OK.

Apply VELO load on lines Menu.

VX Load Value : 0.005

VY a Load Value : 0

OK.

Loads > define loads > Apply > Fluid/CFD > Velocity > onlines.

Apply V on lines.

(Pick lines for velocity specification).

OK.

Apply VELO load on lines Menu.

VX Load Value : 0

VY Load Value : 0

OK.

Apply V on Lines Menu.

Pick lines for vel. specifications.

OK.

Apply Velo Loads on lines menu.

VX Load Value : 0

VY Load value : 0

OK.

11. Preprocessor > Loads > define Loads > Apply > Fluid/CFD > Pressure DOF > On lines.

Apply PRES on lines Menu.

(Pick lines for pressure specification).

OK.

Apply PRES on lines Menu.

PRES pressure value : 0.

OK.

12. Solution > FLOTRAN set up > Execution Cntrl.

Steady State control settings.

EXEC Global iterations 1000.

OK.

13. Solution > Run FLOTRAN.

ANSYS Process Status Menu.

FLOTRAN solution.

(observe graph on the screen)

Solution is done!

Close.

14. General Postproc > read results > Last Set.

15. General Post proc > Plot Results > Contour Plot > Nodal Solution.

Contour Nodal Solution Data.

Nodal Solution/DOF Solution/Fluid Velocity.

OK.

(Observer the picture on the screen).

16. General Post Proc > Plot Results > Vector Plot > Predefined.

Vector Plot Of Predefined Vectors Menu.

Select ; DOF Solution, Velocity V.

OK.

(Observe the Screen).

17. General PP > Plot Results > Vector Plot > FLOTRAN > Defi Trace Pt.

Define Trace points menu.

(Pick WP location for trace points)

OK.

18. From utility Menu.

Plot ctrl's > animate > Particle Flow.

(Observe screen).

19. General PP > Plot Results > Vector Plot > Flow Trace > Defi Trace Pt.

Animate Flow trace Menu.

Select : DOF solution , Velocity VX.

OK.

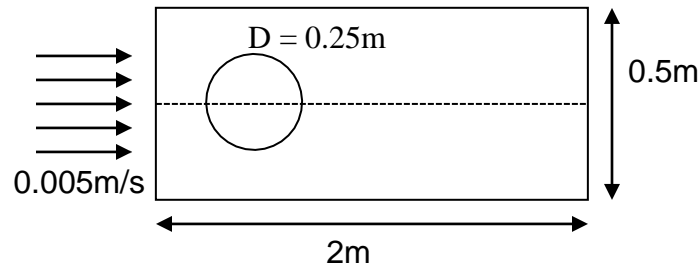
Animation Control Menu .

Close

(Students are required to solve problems theoretically and compare the answer with Ansys results)

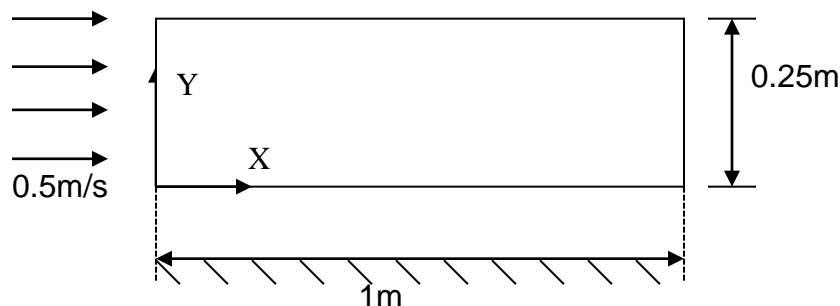
FLUID PROBLEMS

1. Atmospheric air at 20°C flows with a velocity of 5mm/s over a long horizontal cylinder of diameter 25cm. Compute and plot the velocity distribution of air over the cylinder.



The cylinder is 0.25m in diameter considering symmetry about the horizontal only the upper half of the cylinder is computed. The results are assumed to be the same below the x-axis (axis of symmetry). The arbitrary flow area considered is 2m by 0.5m. The velocity of the air at infinite distance from the plate is 5mm/s (0.005m/s) (laminar flow).

2. Atmospheric air flows over a flat plate with a velocity of 0.5m/s. Compute and plot the velocity boundary layer for the flow of air over the plate and find velocity distribution at a distance of 0.5m from the leading edge.



Assume the plate is 1m long.

The arbitrary flow area considered is 1m by 0.25m.

The free stream velocity of the air is 0.5m/s. atmospheric pressure is assumed on all faces except the face where the velocity is input into the system.

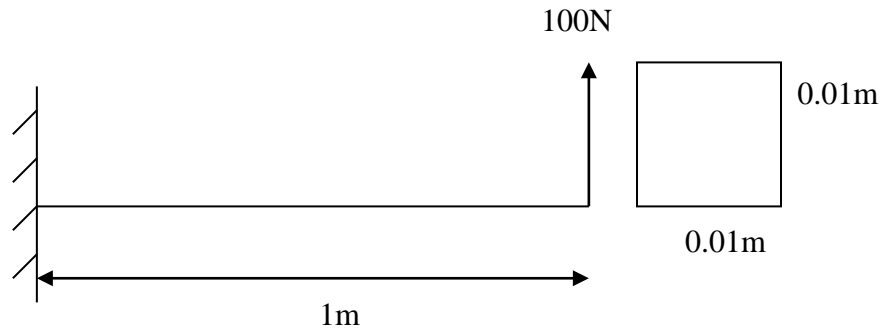
DYNAMIC ANALYSIS

Simple harmonic analysis of a cantilever beam as shown below

$$E = 206800 \text{e}6 \text{ N/m}^2$$

$$I = 8.33 \text{ e-}10 \text{ mm}^4$$

$$\rho = 7830 \text{ kg /m}^3$$



Procedure:

1. Preferences

Preferences for GUI filtering menu

Select ☒ structural

OK

2. Preprocessor > Element Type > Add / Edit / Delete

Element types menu

Click on Add

Library of element types

Select: Beam: 2D elastic 3

:

Element types menu

Type1 BEAM3

Close

3. Preprocessor > Real Constants > Add / Edit / Delete

Real constants menu

Add

Elements types for.....Menu

Type 1 BEAM 3

OK

Real constants for BEAM 3

C/s area : $1e-4$

I22 : $8.33e-10$

Height : 0.01

OK

Real constants menu

Set 1

Close

4. Preprocessor > Material Props > Material Models

Linear isotropic properties for materialMenu

Ex = 206800e6

OK

Define Material Model Behavior Menu

Material Model Number 1

Structural -> Linear -> Elastic -> Isotropic

Click on Density

Density for Material Number 1

DENS : 798.16

OK

The density 7830 kg/m^3 is to be converted to N/m^3

Define Material Model Behavior Menu

Material Model Number 1

Close it

5. Preprocessor > Modeling > Create > Key Points

Inactive CS

Create Key Points in Active Coordinate System Menu

(Enter the key points

KP1, 0, 0

KP2 1, 0)

OK

Modeling > Create > Lines > Straight Lines

Create Straight Lines Menu

(Pick end key points of lines)

OK

6. Preprocessor > Meshing > Mesh Tool

Mesh Tool Menu

Click on Mesh

7. Solution > Analysis Type > New Analysis

New Analysis Menu

Select ☒ harmonic

Analysis type > Analysis options

Harmonic Analysis Menu

Solution method : Full

DOF Printout format : Real + imaginary

OK

Analysis Type > analysis options

Full Harmonic Analysis Menu

Equation solver : Frontal solver

Tolerance : $1e - 008$

OK

8. Solution > Define loads > Apply > Displacement

On key points

Apply U, ROT on KPS menu

All DOF

(Pick the node)

OK

9. Solution > Define Loads > Apply Force / Moment / On Nodes

Apply F/M on Nodes Menu

(Pick the node for force / moment loading)

OK

Apply F/M on Nodes Menu

Direction : FY

Value : 100

OK

10. Solution > Load Step Options > Time / Frequency

Freq and sub steps

Harmonic Frequency and Sub step Options

Harmonic frequency : 0, 100

Number of sub steps : 100

☒ Stepped

OK

11. Solution > Solve > Current Ls

Solution is done!

Main Menu > Time Hist. Post Proc

Time history variables – Menu

Click on FREQ

Add Time History Variable Menu

Nodal solution → DOF Solution → Y Component of displacement

Node for Data Menu

Pick the Node

OK

Time History Variable – Menu

(Click on second icon)

Select : UY2

PRVAR Command Menu.

.....XXXXXXXXXXXXXXXXX.....

(Students are required to solve problems theoretically and compare the answer with Ansys results)

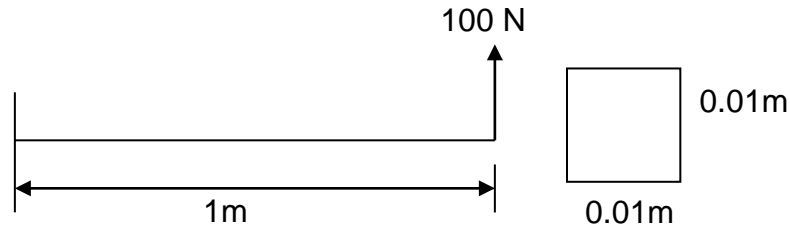
DYNAMIC ANALYSIS

- Simple harmonic analysis of a cantilever beam as shown below.

$$E = 206800 \text{e}6 \text{ N/m}^2$$

$$I = 8.33 \text{e} - 10 \text{ mm}^4$$

$$P = 7830 \text{ kg/m}^3$$



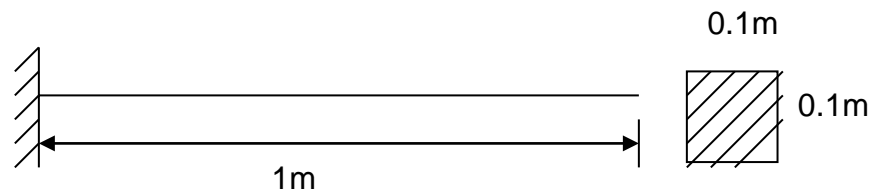
- Simple modal analysis of a cantilever beam as shown below.
Objective is to get the frequency and different modes of vibration.

$$E = 2 \text{e}11 \text{ N/m}^2$$

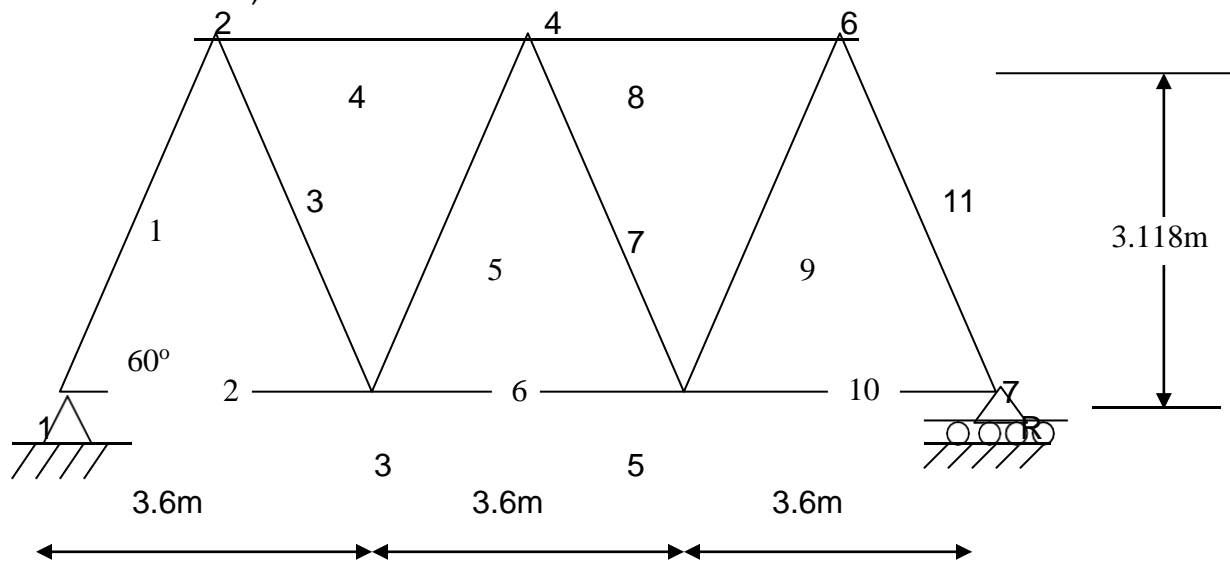
$$I = 8.33 \text{e} - 06 \text{ m}^4$$

$$\text{Area } A = 0.01 \text{m}^2$$

$$\text{Density} = 7830 \text{ kg/m}^3$$



- Determine the modal deflections for the truss system shown below ($E = 200 \text{GPa}$, $A = 3250 \text{mm}^2$)



Viva Questions

- 1) Explain (1) Transpose of a matrix (2) symmetric matrix (3) diagonal matrix (4) unit matrix.
- 2) State the fundamental differential equation in Cartesian and cylindrical coordinates.
- 3) What is a Cartesian tensor?
- 4) State the limitations of one-dimensional finite element analysis.
- 5) Define variation functions
- 6) What is an interpolation function?
- 7) Define shape functions.
- 8) What is stiffness matrix?
- 9) Define what is connectivity in finite element model?
- 10) Classify the boundary conditions.
- 11) What is a direct method?
- 12) Explain the method of node numbering for three- and four-node elements.
- 13) Define constant strain triangles.
- 14) State the three-dimensional strain-displacement equations of elasticity in cylindrical co-ordinates.
- 15) State the three-dimensional stress-strain equation for isotropic elasticity.
- 16) What is beam finite element?
- 17) State the governing equation for a beam and conditions for analysis.
- 18) List the conditions of displacement method for beam analysis
- 19) Why Galerkin method is preferred over variation method?
- 20) State the mathematical representation of variation principle.
- 21) Briefly write about the method of weighted residuals.
- 22) Write a short note on initial value problems.
- 23) State how isoperimetric elements are used in finite element analysis.
- 24) What is shape function? Write the shape function for triangular finite element.
- 25) Write a short notes on isoperimetric quadrilateral elements.
- 26) State and discuss the governing equations for axi symmetric elasticity problems.
- 27) State the method of solving a dynamic or wave propagation problem.
- 28) Name some Eigen value problems.
- 29) Define tetrahedral element and give an example for its use.
- 30) Write a short note on higher order finite elements.
- 31) What is element continuity in finite element analysis?
- 32) Name some Ansys commands that you have used . and explain.