LIST OF EXPERIMENTS

Sl. No.	Date	Name of the Experiment	Page No.	Marks
1	04/08/2021	FEM Introduction	01	
2	04/08/2021	2D - Bar	09	
3	04/08/2021	2D - Tapered Bar	10	
4	04/08/2021	2D - stepped Bar.	12	
5	04/08/2021	Trusses (2 Bar)	14	
6	04/08/2021	Trusses (4 Bar)	16	
7	11/08/2021	Beams (single Load)	18	
8	11/08/2021	Beams (UDL)	20	
9	11/08/2021	plate with hole	22	
10	11/08/2021	plate problems	24	

LIST OF EXPERIMENTS

Sl. No.	Date	Name of the Experiment	Page No.	Marks
11	11/08/2021	Heat transfer.	26	
12	11/08/2021	Dynamic Analysis (Harmonic Analysis)	28	
13	11/08/2021	Dynamic Analysis - (Model Analysis)	30	
14		Dynamic Analysis	32	
15				
16				
17				
18				
19				
20				

R → 22
T → 20
V → 03
045

DD MM YYYY
04 08 2021

Introduction to FEM

A commercial FEM system consists of three basics 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 front that user can understand. which is as shown below.

pre-processor

1. Read control parameters.
2. Read/Generate nodal co-ordinates and boundary conditions.
3. Read/Generate element connectivity and element Loads.
4. Read materials properties or constitutive matrices.
5. Read nodal Loads and loading conditions.

↓

Solver

1. Compute parameters for memory/file management.
2. Compute element matrices and vectors.
3. Form global matrices.
4. Enforce SPC, MPC,....
5. Solution of governing matrix-equations.

↓

DD MM YYYY
04 08 2021

post - processor

1. print/plot deformed mesh over unreformed mesh.
2. print/plot contours of displacement.
3. Compute element strains, stresses, etc.
4. print/plot contours of stresses.
5. Display locations of max/min. stress.
6. print/plot contours of failure index.

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 struc-tural, heat transfer, fluid flow, or some other problem These steps are embodied in commercial finite element software packages (some are mentioned in the following para-graphs) and are implicitly incorporated in the text, alth-ough we do not necessarily refer to the steps explicits 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 problems.
* 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).

DD MM YYYY
04 08 2021

* Define the element connectivity's (mesh the model).
* Define the physical constraints (boundary conditions).
* Define the loadings.

Solution:-

During the solution phase, finite element software assemble 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 ba-ck substitution to compute addition, derived variables, such as reaction forces, element stresses, & heat flow.

post processing:-

Analysis & evaluation of the solution results is refered to as post processing. post processor software contains sop-histicated routines used for sorting printing & plotting selected result from a finite from a finite element solu-tion. Examples of operations that can be accomplished inclu-de.

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.

DD MM YYYY
04 08 2021

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 & it includes command references; operations guide; modeling and meshing guide; basic analysis procedure guide; advance analysis guide; element reference; theory reference; structural analysis; thermal analysis guide; electromagnetic field guide; fluid dynamics guide; and coupled field analysis guide.

Element library in ANSYS lists 189 finite elements. They are broadly grouped into: LINK, PLANE, BEAM, SOLID, CONTAC, COMBIN, PIPE, MASS, SHELL, FLUID, SOURCE, MATRIX, HYPER, VISCO, INFIN, INTER, SURF etc.

FEM procedure

1. Descretization of given domain into a collection of of prescribed finite elements.
   a. Construct the nodes and elements.
   b. Number the nodes and elements.
   c. Generate the geometric properties (eg: coordinates and cross sectional area) needed for the problem.

2. Derivation of element equation for all typical element 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 equati-ons of the whole problem.
ⓐ. Identify the inter element continuity conditions among the primary variables by relating the element nodes to global nodes
ⓑ. Identify the "equilibrium" conditions among the secondary variables.
ⓒ. Assemble the element equation using step 3a & 3b.
4. Imposition of boundary condition of 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 appli-cation and should make use of the latest development in numerical techniques.

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.
→ Structure / solids.
→ aerospace structures.

→ civil engineering structures
→ Nuclear reactions.
→ fluid dynamics.
→ piping systems.
→ manufacturing processes.

Type of response.
→ linear static.
→ Non-linear static.
→ Thermal analysis.
→ Fluid dynamics.
→ Heat transfer.
→ Electrostatic /electromagnetic etc.

Material wall construction
→ monocoque.
→ Layered.
→ sandwich.
→ composite material

Types of loads
→ point load
→ line load
→ Surface loads.
→ volume loads.
→ initial stress
→ Thermal loadings etc.

D	D	M	M	Y	Y	Y	Y
0	4	0	8	2	0	2	1

Stress analysis
→ small/large deformation.
→ small/large strain.
→ elasto-plastic.
→ visco-elastic.
→ Thermal effects etc.

Stability analysis
→ Buckling instability
→ geometric non-linearity.
→ material non-linearity.

Crash analysis
→ Impact.
→ crash worthiness etc.

Static analysis
→ Deflection.
→ stresses.
→ strains.
→ forces
→ energies.

Heat transfer analysis
→ Temperature.
→ Heat fluxes.
→ Thermal gradients.
→ Heat flow from convection faces.

8

DD MM YYYY
04 08 2021

Fluid analysis

→ pressure

→ gas temperatures

→ convection co efficients

→ velocities

Automotive industry

→ static analysis.

→ 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 analysis.

→ Model analysis.

→ aerodynamics.

→ transient dynamics

→ Heat transfer.

→ Fracture Mechanics

→ Creep & plasticity analysis.

→ Composite materials.

→ Aero elasticity.

→ metal forming.

→ crashworthiness.

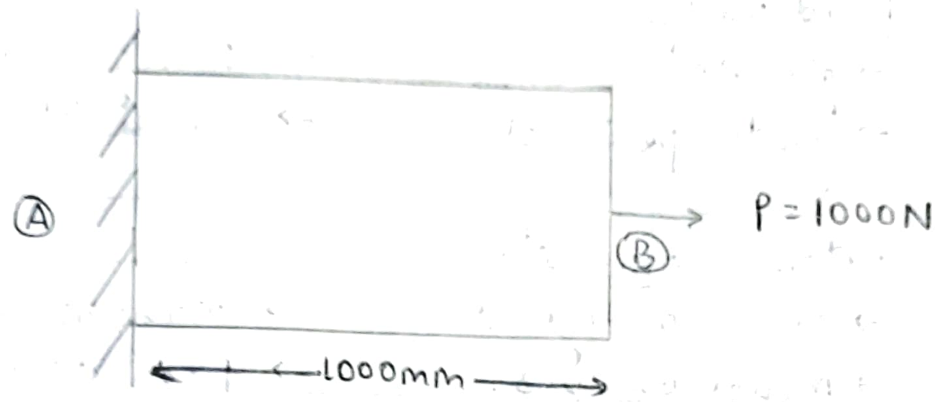
DD MM YYYY
04 08 2021

Bar

The simple bar shown in the fig. determine the displacement, stress, and the sections. the c/s of the bar is 500 mm², length is 1000 mm, and the youngs Modulus is E = 2*10^5 N/mm², Take load P = 1000 N.

1. preferences → structural.
2. preprocessor → Element type → link → 2D spar 1
3. Real constants → set 1 → 500 mm²
4. Material properties → structural → linear → elastic → isotropic → E = 2e5, P_{xy} = 0.3
5. Modelling → create → key points → active CS → point 1 - 0 → point 2 - 1000.
6. Create → line (straight line) → click on required nodes
7. meshing → Size control → manual size → all lines → No of division → 10.
8. Mesh tool → Mesh → pick all.
9. Define loads → Apply → structural → displacement → on nodes 1 → All DOF → Force → on node ⇒ 2 Node → 1000
10. Solve → current LS
11. General postprocessing ⇒ Plot results → contour plot → Nodal soln → X or Y compounents.
12. Element table → Add table → By sequence → LS, 1
13. plot results → contour plot → element table → yes average.
14. List results ⇒ reaction soln

Result: The comparision is shown b/w theoritical & Ansys software.



Solution :

$A = 500\,mm^2$

$P = 1000\,N$

$E = 2\times10^5\,N/mm^2$

→ Axial load $\sigma = \dfrac{F}{A} = \dfrac{1000}{500} = 2\,N/mm^2$

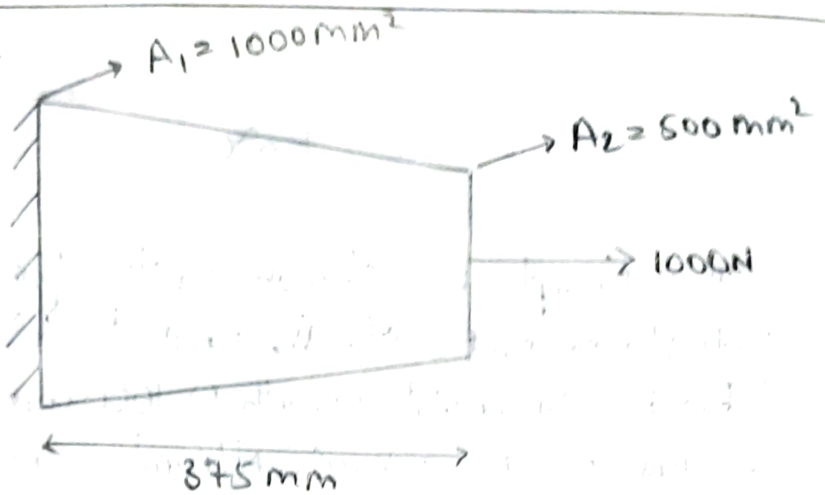
→ Deformation $\Delta l = \dfrac{\sigma l}{E} = \dfrac{2\times1000}{2\times10^5}$

$\boxed{\Delta L = 10^{-2} = 0.01\,mm}$

→ Reaction $= 10^{-3}\,N$ at A.

Comparision.

Results	Theoretical	Ansys.
Δl	10^{-2} mm	10^{-2} mm
σ	$2\,N/mm^2$	$2\,N/mm^2$
Reaction at A	-10^{3} N	-10^{-3} N
Reaction at B	0 N	0 N



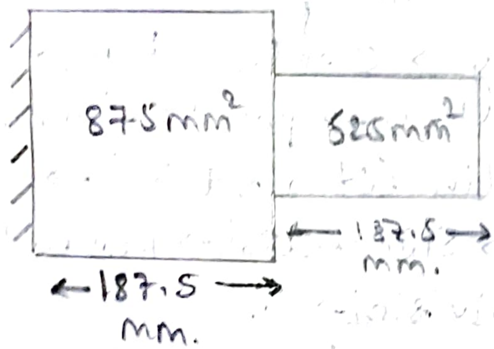
stepped bar 1 $= \frac{1000+750}{2} = 875\ mm^2$

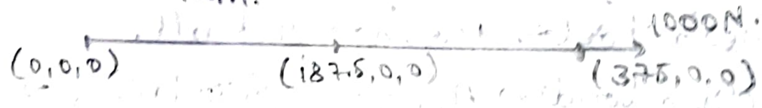
stepped bar 2 $= \frac{725+500}{2} = 625\ mm^2$

$A = \pi/4\, d^2$

$A_1 = 1000 mm^2 = D_1 = 35.7mm$

$A_2 = 500 mm^2 = D_2 = 25.23mm$





$\sigma_1 = \frac{P}{A_1} = \frac{1000}{875} = 1.143$ Mpa.

$\sigma_2 = \frac{P}{A_2} = \frac{1000}{625} = 1.6$ mpa.

Displacement $= \Delta l = \frac{4PL}{E\pi d_1 d_2} = \frac{4 * 1000 \times 375}{2\times10^5 \times \pi \times 35.7 \times 25.23}$

D D M M Y Y Y Y
04 08 2021

Tapered Bar

Q For the tapered bar shown in the figure determine the displacement, stress and reaction in the bar.
Given: $A_1 = 1000mm^2$ & $A_2 = 500mm^2$ $E = 2\times10^5 N/mm^2$

Soln:
1. preference → structural.
2. preprocessor → element type → link → 2D spar 1
3. Real constants → set 1 → 875 → set 2 → 625.
4. material properties → structural → linear → elastic → isotropic → E = 2e5 P_{xy} = 0.3
5. Modeling → Create → keypoints → active CS → point 1 – 0 → point 2 → 187.5 → point 3 – 375.
6. Create → line → straight line → click on required nodes.
7. Meshing → mesh attributes. → picked line → line 1 → Real constant set 1 → line 2 → Real constant set 2
Size control → manual size → all line → No of division 10.
8. Mesh tool → Mesh → pick all.
9. Define loads → apply → structural → displacement on Nodes → All DOF → force/Moment → on nodes → 2nd nodes → Magnitude 1000

$\boxed{\Delta L = 2.65 \times 10^{-3}\ mm}$

Ansys :

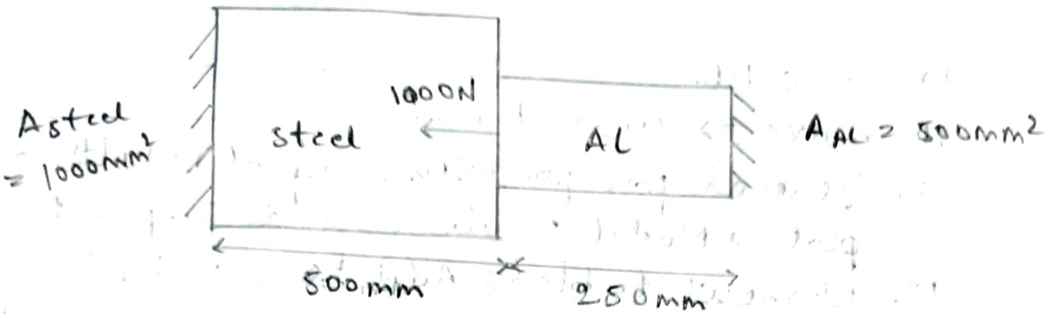
DMX = 0.002573 mm.
SMN = 1.143 N/mm^2
SMX = 1.6
Reaction = - 1000

Comparision

	Theoretical	Ansys.
ΔL	2.65×10^{-3} mm	2.573×10^{-3} mm.
Max stress (σ)	1.6 N/mm^2	1.6 N/mm^2
Reaction At A	-10^3 N	-10^{-3} N
Reaction at B	0	0

10. Save DB.
11. Solve ⇒ current LS.
12. General post processing → plot results → contour plot → Nodal solution → X compount.
13. Element table → Add table → By Sequence → LS,1 plot results → Contour plot → Element table → Yes, avgrage.
    List results → Reaction Solutions.

Result: The comparision is shown b/w the theoretical and Ansys Software.



$\Delta T = 60^\circ C$ $\quad T_0 = 20^\circ C$ $\quad T_f = 80^\circ C$ $\quad E_{steel} = 2e5$

$E_{al} = 0.7e5$

$\alpha_{steel} = 11.7\,e-6$

$\alpha_{al} = 23e-3$

$\Delta = \alpha_a t L_a + \alpha_s t L_s$

$= 0.345 + 0.351$

$\boxed{\Delta = 0.695\,mm}$

$\delta_{T(st)} - \delta_{ST} = \delta_{AL} - \delta_{T(AL)}$

$0.351 - \left(\frac{PL}{AE}\right)_{st} = \left(\frac{PL}{AE}\right)_{AL} - 0.345$

$0.351 - \left(\frac{(5\times10^5 + R)\,500}{1000\times2\times10^5}\right) = \frac{R\times250}{500\times7\times10^4} - 0.345$

$0.696 = \frac{50R}{7\times10^6} + \frac{5\times10^5 + R}{4\times10^5}$

$\Rightarrow R = 57.768\times10^3\,N \longrightarrow$ Reaction.

<u>stress at AL</u> $\quad \sigma = \frac{P}{A} = \frac{57.708\times10^3}{500}$

$\boxed{\sigma_{AL} = 115.416\,N/mm^2}$

D D M M Y Y Y Y
|0|4|0|8|2|0|2|1|

Bar (Stepped bar).

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

$T_0 = 20°c$ $T_f = 80°c$ $E_{steel} = 2e5\ N/mm^2$

Area of steel $A_{steel} = 1000mm^2$ $E_{Al} = 0.7e5\ N/mm^2$

Area of aluminium $A_{al} = 500mm^2$.

$\alpha_{steel} = 11.7e-6/°c$

$\alpha_{alu} = 23e-6/°c$

Soln:

1. preference → Structural
2. preprocessor → element type → Link → 2D spar
3. Real constant → $A_1 = 1000mm^2$ → $A_2 = 500mm^2$.
4. material Model 1 → linear → elastic → isotropic
   $E_x = 2e5$ $P_{xy} = 0.3$.
   Secant co-efficient for thermal expansion is 11.7e-6.
5. Modelling → Repeat for material 2.
6. Modelling → Create → key points → active CS → point 1 - 0 → point 2 → 500 → point 3 - 750.
7. Create → line → straight line → select required nodes.
8. Meshing → mesh attribute → picked line → line 1 → linear const No = 1
   Repeat for line 2 ⇒ Real const No 2.

stress at steel : $5 \times 10^5 = 57.708 \times 10^3 / 100$

$\sigma = 142.292 \ N/mm^2$

Displacement in steel | Displacement in Al

$E = \frac{142.292}{2 \times 10^5} =$

$\boxed{E = 7.115 \times 10^{-4}}$

$E = \frac{y'}{y}$

$= 1.6488 \times 10^{-3} \times 250$

$\boxed{y' = 0.41}$

$= 0.341 + 0.41$

$= 0.741 mm.$

$\frac{y'}{y} = E$

$y'_2 = 7.115 \times 10^{-4} \times 500 = 0.3557.$

$y_{total} = 0.41 + 0.3557$

$= 0.7657 mm$

Result comparision

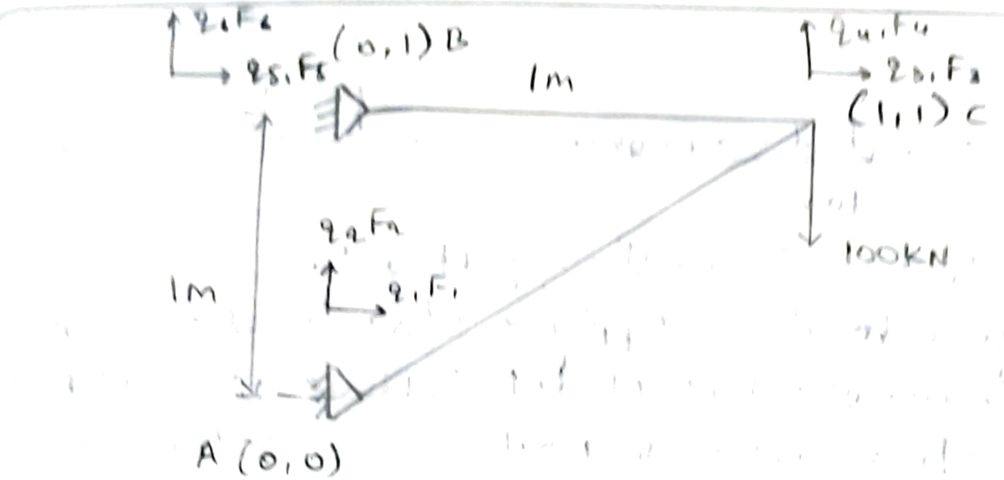
	Ansys	theoretical.
Deformation	0.753 mm	0.754 mm
Stress	114.90 N/mm^2	114.92 N/mm^2

D D M M Y Y Y Y
04 08 2021

Size control → Manual size → all line → No of divisi -on → 10.

9. Mesh tool → mesh → pick all.
10. Define loads → Apply → structural → displacement. → on nodes select Node1 & Node 3 → All DOF
11. Force/moment → on nodes ⇒ Select Node 2 → -10000
→ temperature on element → 60
12. Solve ⇒ current LS
13. General post processing → plot Results → contour plate → Nodal solution → Define element table → By sequence → LS1
plot result → countour plot → element table → yes, avg.
List result → Reaction solution.

Result: The comparision of result is shown.



Solution

$q_1 = q_2 = q_5 = q_6 = 0$ $\quad\quad F_1 = F_2 = F_6 = F_5 = F_3 = 0$

$q_3 = 0.048 \times 10^{-3}\,m$ $\quad\quad F_4 = -100KN$

$q_4 = -0.182 \times 10^{-3}\,m$

$\sigma = E.\epsilon = 210 \times 10^9 \times 0.048 \times 10^{-3}$

$\boxed{\sigma = 10.08 \times 10^8\,N/m^2}$

using $k = \frac{AE}{l}\begin{bmatrix} l^2 & ml & -l^2 & -ml \\ ml & m^2 & -ml & -m^2 \\ -l^2 & -ml & l^2 & ml \\ -ml & -m^2 & ml & m^2 \end{bmatrix}$ & $kq = F$.

$\sigma = \frac{E}{L}\begin{bmatrix} -l & -m & l & m \end{bmatrix}$

Nodal co-ordinate data

Node No	x	y
1	0	0
2	1	1
3	0	1

Element connecting data

Element	IN	FN	le	l	m
1	1	2	1.414	0.707	0.707
2	2	3	1	-1	0

$k_1 = 10^9\begin{bmatrix} 0.74 & -0.74 & -0.74 & 0.74 \\ -0.74 & 0.74 & 0.74 & -0.74 \\ -0.74 & 0.74 & 0.74 & -0.74 \\ 0.74 & -0.74 & -0.74 & 0.74 \end{bmatrix}$

$k_2 = 10^9\begin{bmatrix} 2.1 & 0 & -2.1 & 0 \\ 0 & 0 & 0 & 0 \\ -2.1 & 0 & 2.1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$

DSCE

D D M M Y Y Y Y
04 08 2021

Trusses

1 Two bar truss shown in Fig. Determine nodal displacem-ent & the stress in each member. Take E = 210 Gpa. Area = 0.01 sq.m.

Soln:

1. Preference → Element type → link → 2D Spar.
2. Real constant → Link1 → Area → 0.01
3. Material properties → Material models → structural → linear → elastic → isotropic → E = 2e5 → Pxy = 0.3
4. Modeling → Create → keypoints → in active CS.

XYZ location	0	0	0	A
XYZ location	0	1	0	B
XYZ location	1	1	0	C

6. Modelling → Create → lines → straight lines → on keypoints
7. Meshing → Mesh tool → size controls → Manual Size → lines → All lines → No of div → 1
8. Meshing → Mesh tool → mesh → pick the lines.
9. Solutions → Define loads → Apply → structural → ON nodes → All DOF.
10. Define loads → Force/moment → ON nodes at C 100KN along y direction.
11. Solve the current load setup.

$$K = 10^{9}\begin{bmatrix} 0.74 & 0.74 & -0.74 & -0.74 & 0 & 0 \\ 0.74 & 0.74 & -0.74 & -0.74 & 0 & 0 \\ -0.74 & -0.74 & 2.84 & 0.74 & -2.1 & 0 \\ -0.74 & -0.74 & 0.74 & 0.74 & 0 & 0 \\ 0 & 0 & -2.1 & 0 & 2.1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$$

$Kq = F$

$q_3 = -1.824 \times 10^{-4}$ mm.

$q_8 = 0.477 \times 10^{-4}$ mm.

$$\sigma_1 = \frac{E}{l}\begin{bmatrix} -l & -m & l & m \end{bmatrix}\begin{bmatrix} q_1 \\ q_2 \\ q_3 \\ q_4 \end{bmatrix}$$

$\sigma_1 = 1 - 1.418 \times 10^{7}$

$\sigma_2 = 1001700$ Mpa

<u>Ansys solution</u>

DMX = 0.188E-03

SMN = 0.182E-03

SMX = 0.100E+08

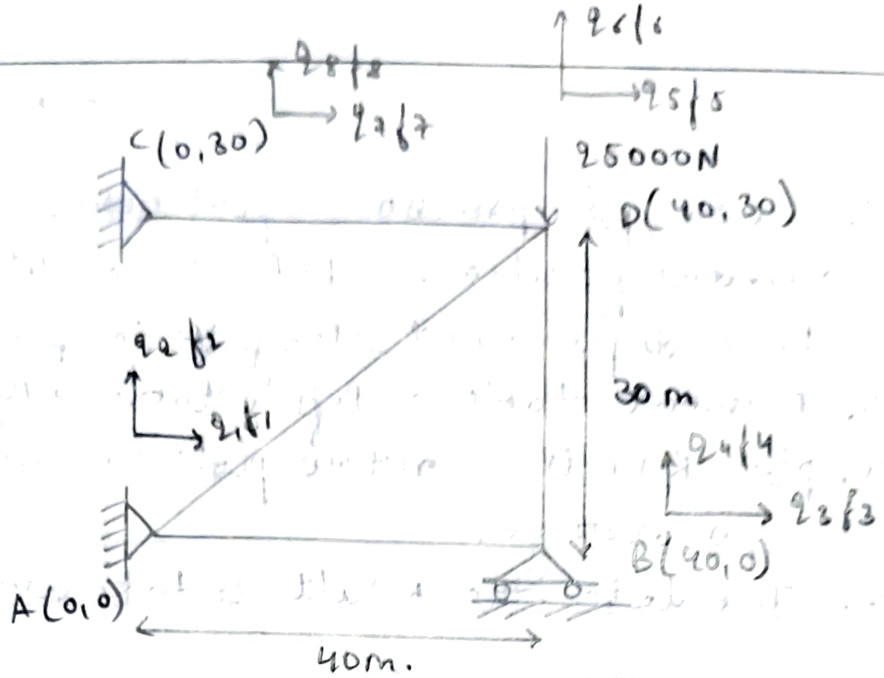
<u>Result</u>

	Ansys	theoretical
Displacement	0.188×10^{-3} m	0.182×10^{-3} m
Stress.	10×10^{6} N/m^2	10.08×10^{6} N/m^2

DD MM YYYY
04 08 2021

12. General post processor → plot results → contour plot → Nodal solution. → DOF → X & Y components.
13. Find displacement along Y-component.
14. Element table → Define table → By sequence → LS.
15. plot Results → contour plot → Element table → yes average.
16. Tabulate the Result & take down values.

Result: The comparision is shown Below.



where

$$q_1 = q_2 = q_4 = q_5 = q_8 = q_7 = 0.$$

$$f_1 = f_2 = f_8 = f_4 = f_7 = f_8 = 0.$$

Using formula we get.

q_3 = Deformation $= -22.25 \times 10^{-3}$ m

stress $= 4167$ N/m²

Reaction $=$ Fx $= 4167$ N

Fy $= 3126$ N

Ansys solution.

DMX = 0.022243 m

SMN = -1.0937

SMX = 4,167 N/mm²

16

DD MM YYYY
04 08 2021

Trusses

2 Consider that four bar truss shown in figure. it is given that $E = 29.5 \times 10^6$ N/m² and $A = 1m^2$ for all the elements. Determine the deformation, Maximum stress & the reaction forces in the truss system.

1. preference → Element → link → 2D spar.1.
2. Real constants → link1 → Area = 1m²
3. Material properties → Material models → Structural → linear → elastic → isotropic → Ex = 2e5 → PRxy = 0.3.
4. Modelling → Create → Keypoints → In active CS.

XYZ location	0	0 0
XYZ location	40	0 0
XYZ location	0	30 0
XYZ location	40	30 0

5. Modeling → Create → lines → straight lines → on key points.
6. Meshing → size controls → manual size → lines → All lines → No of div. 1.
7. Meshing → Mesh tool → Mesh → pick the lines.
8. Solutions → Define loads → Apply → structural → Displacement → On Nodes → All DOF.
9. Define loads → Force/moment ⇒ ON nodes Fy ⇒ on D node 25000N.
10. Solve → current LS.

Result

	Ansys	Theoretical
Deformation	0.022952 m	0.0225 m.
stress	4167 N/mm^2	4167 N/mm^2
Reaction.	F_x = 4167 N F_y = 3126 N	F_x = 4167 N F_y = 3126 N

D D M M Y Y Y Y
04 08 2021

11. General plot processor → plot results → contour plot → Nodal solution. → All DOF → X or Y component.
12. Element table → Define table → By sequence → LS,1
13. plot results → Contour plot → Element table → yes average.

Result

The comparision b/w Ansys & Theoretical value is shown.

$E = 200\ \text{Gpa} \quad P = 4\times10^3\ N \quad L = 2000\ mm$

<u>Solution</u>

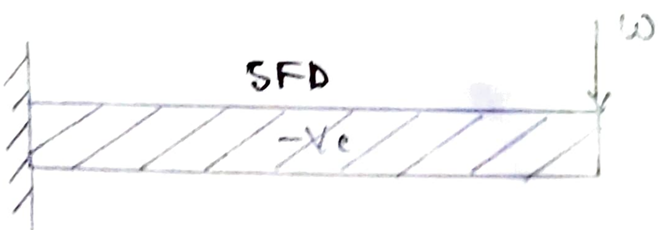
Deflection $y = \dfrac{PL^3}{3EI}$

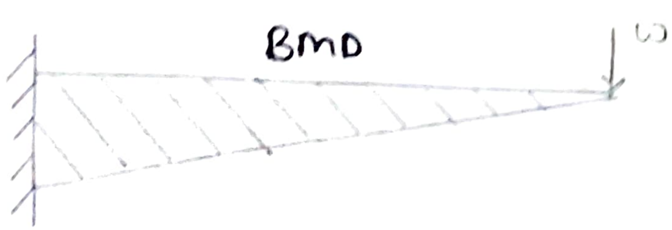
$I = \dfrac{bh^3}{12} = 3645\times10^3\ mm^4$

$A = 60\times90 = 5400\ mm^2$

$\boxed{y = 14.6\ mm}$

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





DD MM YYYY
11 08 2021

Beam

1 A 2m long catilever with rectangular section (60*90mm) is subjects 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 = 200Gpa P = 4X10^4 N.
Length L=2m = 2000mm.

Soln
1. preferences → structural
2. preprocessor → element type → Beam → 2D elastic.
3. Real constant A = 5400mm²
   Izz = 3645e3
   Height = 90
4. material properties → linear → elastic → isotropic
   Ex = 200e3 PRxy = 0.3
5. preprocessor → modeling → create → Inactive CS
   A (0, 0) B(2000, 0).
6. modeling → create → lines → straight lines → pick keypoints.
7. meshing → mesh tool → mesh line.
8. solution → Define loads → Apply → structural → Displac-ement → on nodes.
9. solution → solve → current LS
10. General processor → plot results → contour plot → Nodal solution → DOF solution → Y component
11. General post processor → Element table → Define table
Element table data menu → By sequence num → SMISC

Ansys Solution

Deformation $y = 14.642$ mm

Stress $\sigma = 98.765$ N/mm^2.

Result composision

	Ansys	theoretical
Deformation	14.642mm	14.63mm.
Stress	98.765N/mm^2	98.76N/mm^2

DD MM YYYY
01 08 2021

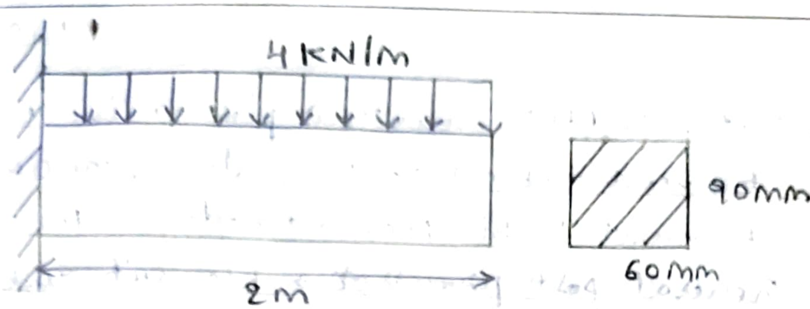
12. Plot Results → Countour plot → line Element Results
Element table item at node I: MMOMZI
Element table item at node J: MMOMZJ.

13. General post processor → Element table → Define table
→ By Sequence num, SMISC (6,8) Apply → ok → close

14. plot line Element Results Menu.
Element table item at node I: MFORYI
Element table item at node J: MFORYJ.

15. List Results → Reaction Solution.
List Reaction Solution Menu.
Select all items → ok.

Result: comparision of result is shown.



$E = 200\,GPa \quad P = 4KN/m \quad l = 2000\,mm$

$A = 5400\,mm^2 \quad I = 3645 \times 10^3\,mm^4$

$H = 90,$

Deflection $= \frac{WL^4}{8EI} = \frac{4\times10^3 \times (2000)^4}{8 \times 200\times10^3 \times 3645\times10^3}$

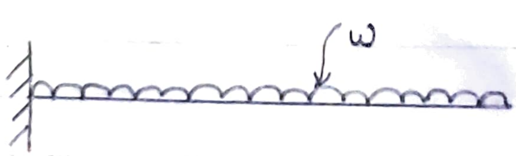
$\boxed{y = 0.0109\,m}$

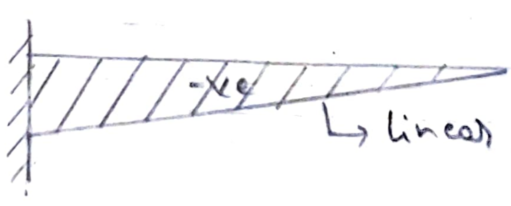
Bending moment $= M = \frac{WL^2}{2} = \frac{4\times1000\times(2000)^2}{2}$

$= 8\times10^6\ N\,mm$

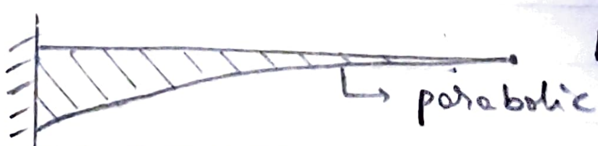
Bending stress $= \frac{Mb}{I} \cdot C = \frac{8\times10^6}{3645\times10^3} \times \frac{90}{2}$

$= 98.7\ N/mm^2$





SFD



BMD.

DSCE

DD MM YYYY
11 08 2021

Beam.

2 A 2m long cantilever with rectangular section (60x90) mm is subjected to uniformly distributed load 4KN/M through out its length. Determine maximum bending stress and its deflection take E=200GPa.

soln

1. preference → structural.
2. preprocessor → element type → beam → 2D elastic.
3. Real constant → A = 5400 mm^2

   I_{zz} = 3645e3

   height = 90.
4. Material properties → Linear → elastic → isotropic

   Ex = 200e3 . PRxy = 0.3
5. pre processor → Modeling → create → Inactive CS.

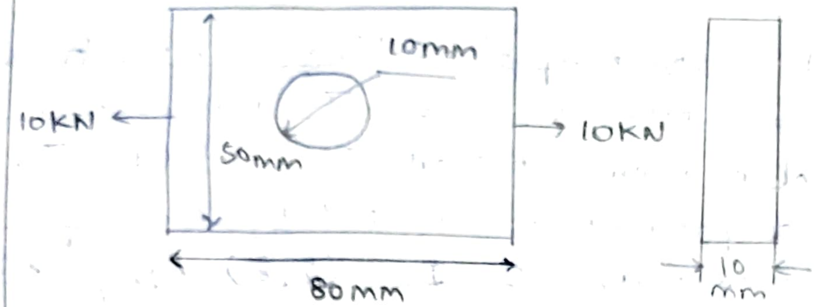
   A(0,0) B(2000,0).
6. Using Modeling → create → lines → straight line → pick keypoints.
7. Meshing → mesh tool → mesh line.
8. Solution → Define loads → Apply → structural → Displacement → on nodes.
9. Solution → solve → current LS.
10. General preprocessor → plot results → contour plot → Nodal solution → DOF soln → Y component.
11. General post processor → Element type table → Define table → By sequence number → SMISC.

Result

	Ansys	theoretical.
Deformation	10mm	10.34mm
Stress	98.76 N/mm^2	98.76 N/mm^2

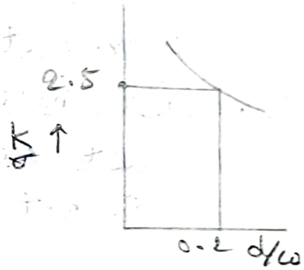
12. Plot Results → Contour plot → Line Element Results
Element table item at node I : MMOMZI.
Element table item at node J : MMOMZJ.
13. General post processor → Element table → Define table
→ By sequence num, SMISC (2,8) Apply → OK →
Close.
14. plot line Element Results menu.
Element table item at node I : MFORYI
Element table item at node J : MFORYJ.
15. List Result → Reaction Soln
List Reaction Soln menu.
Select all items → OK

Result: Comparision of result is shown.



$F = 10KN \quad h = 10mm \quad w = 50mm \quad d = 10mm$

$\frac{d}{w} = \frac{10}{50} = 0.2 \quad \therefore K_r = 2.5$



$\sigma_{avg} = \frac{F}{A} = \frac{10 \times 10^3}{(b-d)t}$

$= \frac{10 \times 10^3}{(50-10)\,10}$

$\boxed{\sigma_{avg} = 25 N/mm^2}$

pressure $= \frac{F}{A}$

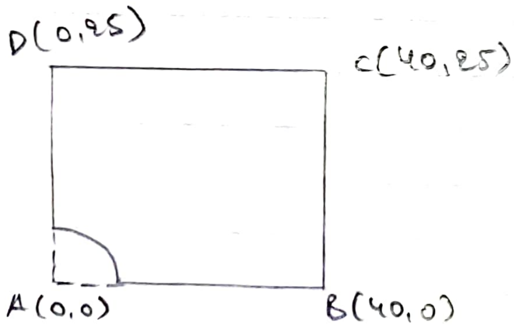
$= \frac{10 \times 10^3}{50 \times 10}$

$\boxed{P = 20 N/mm^2}$

$\sigma_{max} = \sigma_{avg} * K_t$

$= 25 \times 2.5$

$\boxed{\sigma_{max} = 62.5 N/mm^2}$



D D M M Y Y Y Y
11 08 2021

plate with hole

1 Determine the max 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=200Gpa.

Soln: 1. preference → structural → ok.
2. preprocessor → element type → solid → Quad 4 → ok.
3. preprocessor → element type → option → K3 → plane stress with thickness.
4. Real constant → Add → select → ok → thickness (10mm) → ok.
5. Material properties → Model → structural → linear → elastic → isotropic → E=2e5 μ=0.3 value.
6. Model → Create → Areas → Rectangular → By 2 Corners → x, y (0, 0) → width (40) Height (25)
7. Model → create → circle → Solid circle → (x, y) (0, 0) → Radius (5).
8. operate → Booleans → Substract → Areas → Select (Rectangle) ok then (circle) ok
9. Meshing → Mesh tool → Area (Set) → Apply → (0.1) → ok.
10. Rise hidden → mesh → ok.

11. Load → Define → Apply → structural → Displacement → symmetric B.C → on lines → select lines → ok.

Ansys results

Deformation · 0.004329 mm.

stress 62.244 N/mm².

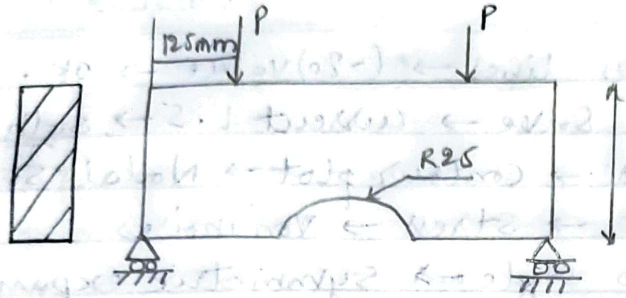
Result:

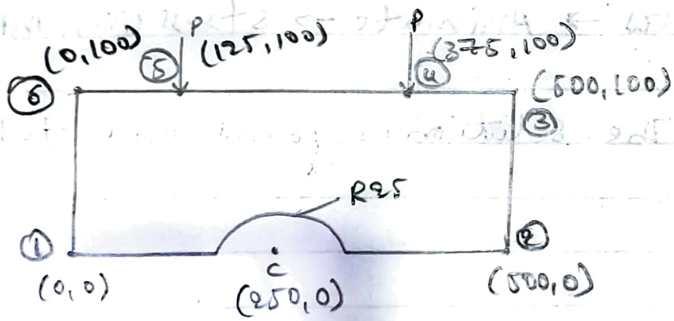
	Ansys	Theoretical
Deformation.	0.004329mm	–
stress	62.244 N/mm²	62.5 N/mm².

D D M M Y Y Y Y
11 08 2021

12. pressure → on lines → (-20) value → ok.
13. Solution → Solve → current L.S → soln done
14. plot results → contour plot → Nodal·soln → displacement → stress → von moises.
15. plot ctls → style → symmetric expansion → periodic → 1/4 → ok.
16. plot ctls → Animate → stress (von mises) → ok.

Result: The solution is found and tabulated.


125mm
P
P
R25
100mm


(0,100)
P
(125,100)
P
(375,100)
(500,100)
R25
(0,0)
(250,0)
(500,0)
C

D	D	M	M	Y	Y	Y	Y
1	1	0	8	2	0	2	1

plate problems

2 Calculate the stresses and displacement for the plate shown below. Let the load be P = 100N applied at equal distance from both ends and E = 3e7 N/mm^2.

Soln:
1. preference → structural
2. Element type → Solid → Quad 4 → OK.
3. Element type → Type plane stress with thickness → Element behaviour K3 → plane stress w/thickness.
4. Real constant → Type PLANE42 → Thickness → 10mm.
5. material properties → material model → structural → linear → elastic → isotropic → Ex: 3e5
6. Modeling → Create → keypoints → In active CS

A	location	0	0	0
B	location	500	0	0
C	location	500	100	0
D	location	375	100	0
E	location	125	100	0
D	location	0	100	0

7. modeling → Create → Area → Through keypoint
8. modeling → Create → Area → circle → solid circle → X axis: 250 → Y axis: 0 → Radius: 25
9. Modeling → Create → operate → Boolean → Substract → Area → pick Rectangle first → pick area to be substract-ed (circle) → OK.
10. Meshing → Mesh tool → ~~Mesh~~ → Mesh area → mesh at element → ~~pick all~~ → Refine Mesh → level of refinement : 3.

Ansys Solution

DMX = 0.477E-04 mm

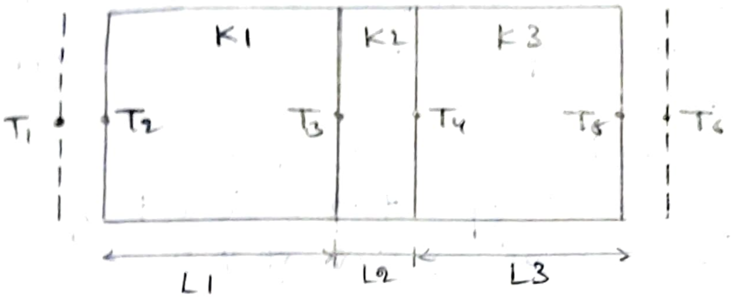
SMN = 0.005404

SMX = 7.879 N/mm^2

DD	MM	YYYY
11	08	2021

11. loads → Define loads → structural → Force / moment → on Nodes → pitch node → 1 & 2 → apply → DOF VY → OK.
12. load → Define loads → apply → structural → Force / moment → on Keypoint → Select Keypoint 4 & 5 → -100
13. Solution → Solve → Current LS
14. Solution → General soln → plot results → countour plot → Nodal soln → Von moises solution.
15. List the Reaction Solution.

<u>Result</u>: The Ansys solution is found and shown.



$T_1 = 820°C \quad T_5 = 110°C$

$R_c = \dfrac{L_2}{A_2 k_2} = \dfrac{1}{k_2 \times 1}$

$\boxed{k_2 = 1\, w/mk}$

$A = 1m^2.$

$k_1 = 1.5\, w/mk.$

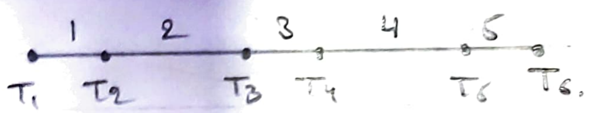
$k_2 = 1\, w/mk$

$k_3 = 4.9\, w/mk.$

$h_1 = h_2 = 35\, w/m^2K$

$L_1 = L_3 = 100mm$

$L_4 = L_2 = L_5 = 1mm$



$R = \dfrac{L}{kA.}$

$R_c = 0.001\, m^2 k/w$

$R_1 = \dfrac{1}{h_1} = 0.029 = R_5$

$R_2 = \dfrac{L_1}{A_1 k_1} = 0.067\, m^2 k/w$

$R_3 = \dfrac{L_2}{A_2 k_2} = 10^{-3}\, m^2 k/w$

$R_4 = \dfrac{L_3}{A_3 k_3} = 0.02\, m^2 k/w$

$R_{eq} = R_1 + R_2 + R_3 + R_c + R_4 + R_5$

$\boxed{R_{eq} = 0.148\, m^2 k/w}$

$Q = \dfrac{T_1 - T_6}{R_{eq}} = \dfrac{820 - 110}{0.148} = \boxed{4797.3 = Q}$

DSCE

26

DD MM YYYY 11 08 2021

Heat transfer

A furnance wall in made of inside silica brick (K=1.5 w/mK) and outside of magnesia brick (K=4.9 w/mK), each 10cm thick. The inner and outer surfaces are exposed to fluids at temperature at 820°c and 110°c respectively. The contact Resistance is 0.001 m²K/w. The heat transfer co-efficient for the inner and outer surface is 35 w/m²K. Find the heat flow throught the wall per unit area per unit time and temperature distribution across the wall.

soln:

1. preference → Thermal → OK.
2. preprocessor → Element type → link → conduction (link33) Apply → convection (link 34) → ok.
3. Real constant → Add → link 33 (Area 1) → OK
   link34 (Area 1) → ok.
4. Material properties → Material models → Thermal → conduction → isotropic (K_1 = 1.5)
   material → New model → isotropic → (k_2 = 1)
   material → New Model → istropic → (K_3 = 4.9)
   Material → New Model → convection → (HF = 35).
   Material → exit.
5. Modeling → Create → Nodes → In active CS → Apply (0,0) (0.001m) Apply; (0.101m) Apply, (0.102m) Apply, (0.202m) Apply, (0.203m) Apply.
6. Element → ~~Element~~ Attributes → values of convection (1-2) 34. → Auto Numbered → Thru nodes → Select Nodes.

$$4797.3 = \frac{T_1 - T_2}{0.029}$$

$$\boxed{T_2 = 670.878^\circ C}$$

$$4797.3 = \frac{T_2 - T_3}{0.067}$$

$$\boxed{T_3 = 349.46^\circ C}$$

$$4797.3 = \frac{T_3 - T_4}{10^{-3}}$$

$$\boxed{T_4 = 344.66^\circ C}$$

$$4797.3 = \frac{T_4 - T_5}{0.02}$$

$$\boxed{T_5 = 248.7^\circ C}$$

Result:

Quantity	Ansys	Theory.
T_2	680.31°C	680.31°C
T_3	354.36°C	354.36°C
T_4	349.47°C	349.47°C
T_5	249.69°C	249.69°C
Heat flow	4889.2 W/m²	4797.3 W/m²

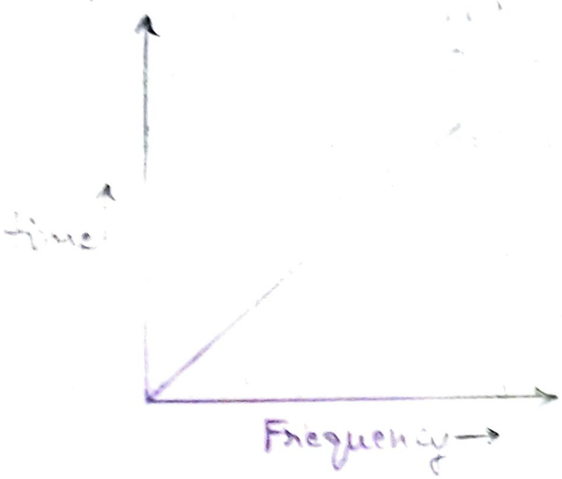
D	D	M	M	Y	Y	Y	Y
1	1	0	8	2	0	2	1

node 2-3 → link33 → set1 → material1
node 3-4 → link33 → set1 → material2.
node 4-5 → link32 → set2 → material3.
node 5-6 → link34 → set2 → material4.

7. Solution → Ansys tool → New Analysis → steady state → ok
8. Define loads → Apply → Thermal → Temperature → on Nodes
   Node (1) select (Apply) → Temp (820°).
   Node (6) select (Apply) → Temp (110°)
9. Solve → current LS
10. General post processor → plot results → contour plot → DOF Soln → Nodal soln.
11. List Results → For listing table → Reaction Soln → Heat flow.
12. Define table → Add → By sequence num → SMISC1

Result: The comparision is shown between ansys & theory.

Nature of graph



Scale

x-axis
1cm = 10unit

y-axis
1cm = 10units

DD MM YYYY
11 08 2021

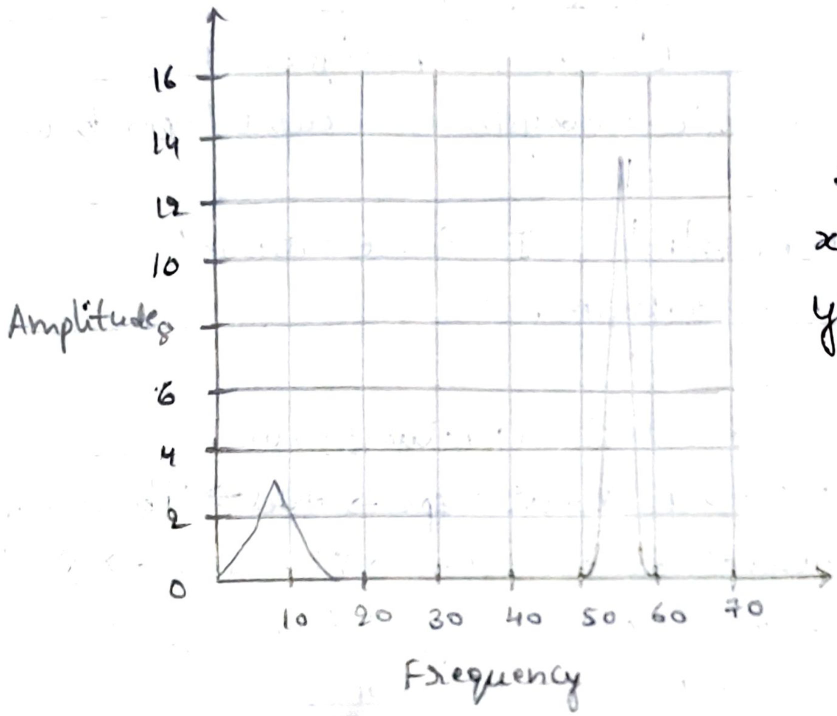
Dynamic Analysis
(Hormonic Analysis)

1 Simple hormonic analysis of a cantilever beam as shown below.

E = 206800e6 N/m² I = 8.33e-10 mm⁴
ρ = 7830 Kg/m³.

Soln :-

1. preference → Select structural → ok.
2. preference → Element type → Add → library of element types menu → Select Beam → 2D statics → ok
3. Real constants → Add → Type1 BEAMS → OK.
   c/s Area = 0.01 X 0.01
   I = 8.33e-10
   Height = 0.0K → ok.
4. preprocessor → Material properties → Material model → structural → linear → elastic → isotropic → Ex = 206800 e6 · ν = 0.3.
   Density → DENS → 7830 → OK
5. preprocessor → Modeling → Create → Key points → Inactive CS → Kp1 (0,0) Kp2 (1,0).
6. preprocessor → Modeling Create → lines → Straight lines → pick lines → OK.
7. preprocessor → Meshing → Mesh tool → Global size = 100 → ok.
8. Solution → Analysis type → New analysis → Analysis option → Hormonic load step option → Time / Frequency → frequency & sub steps. 0, 100, 100
   ☐ stepped.



Scale
x axis 1cm = 10units
y axis 1cm = 10units

DSCE

DD MM YYYY
11 08 2021

9. Solution → Define loads → Apply → structural → Displacement → on keys → pick left end key points → All DOF → OK

10. Solution → Define loads → Apply → structural → Force/ moment → on keys.
⇒ pick right end point → FY → value of force → -100.

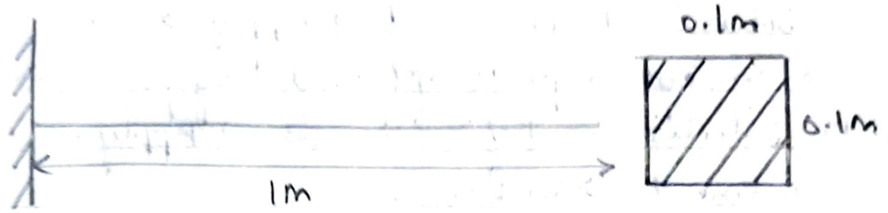
11. Solution → Solve → current LS → Solution → done → OK.

12. Time list post processor → A window opens → click on element shown in window

UV2	2	Y-com dis

→ DOF Soln
→ Y - Component.
→ Select a point where load / time is acted (Right end KP).
→ Graph appears.

<u>Result</u>: The graph of frequency v/s values is plotted.



$g = 9.81$ $\quad (\beta_1 I)^2 = 3.52$ $\quad (\beta_2 I)^2 = 22$ $\quad \beta_3 I^2 = 61.7$

$\omega_1 = \dfrac{(\beta I)^2}{\sqrt{EI/\rho I^4}}$ $\qquad I = 8.33E-06$

$\rho = 7{,}830\ kg/m^3$

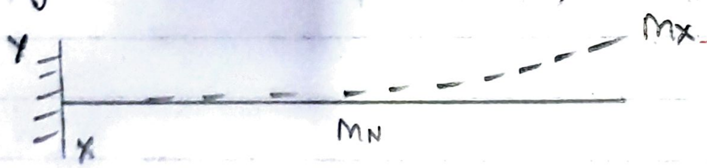
Natural frequency $= \dfrac{\omega}{2\pi}$

$\omega_1 = 1.608E3$ $\qquad f_1 = 255.89$

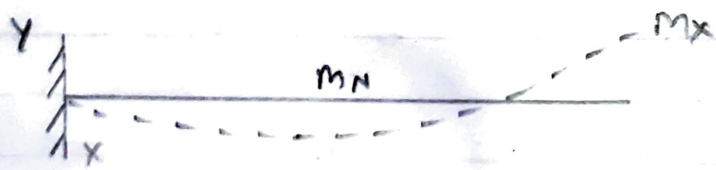
$\omega_2 = 1.005E4$ $\qquad f_2 = 1599.35$

$\omega_3 = 2.819E4$ $\qquad f_3 = 4486.47$

Ansys results



FREQ = 81.476



FBEQ = 506.37

FREQ = 1278

D D M M Y Y Y Y
11 08 2021

Dynamic Analysis
(Model Analysis)

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

$E = 2e11\ N/m^2$

$I = 8.33e-06\ m^4$

Area $A = 0.01\ m^2$

Density $= 7830\ kg/m^3$.

Soln

1. preference → structural → OK.
2. preprocessor → Elemental type → Add → select Beam → 2D elastic → OK.
3. Real constant → Add → Type1 → Beam 3 → OK
   - c/s Area = 0.1 × 0.1
   - I = 8.33e-06
   - H = 0.1
4. preprocessor → material property → Material models → structural → linear → elastic → Isotropic.
   - Ex = 2e11
   - ν = 0.5

   Density → DENS → 7830 → OK.
5. preprocessor → Modeling → Create → Keypoints → In active CS → KP1 (0,0) KP2 (4,0).
6. preprocessor → Modeling → Create → Lines → straight lines pick end key points of lines → ok.
7. preprocessor → Meshing → mesh tool → Global sizes → no of division - 100 → click on mesh → ok.

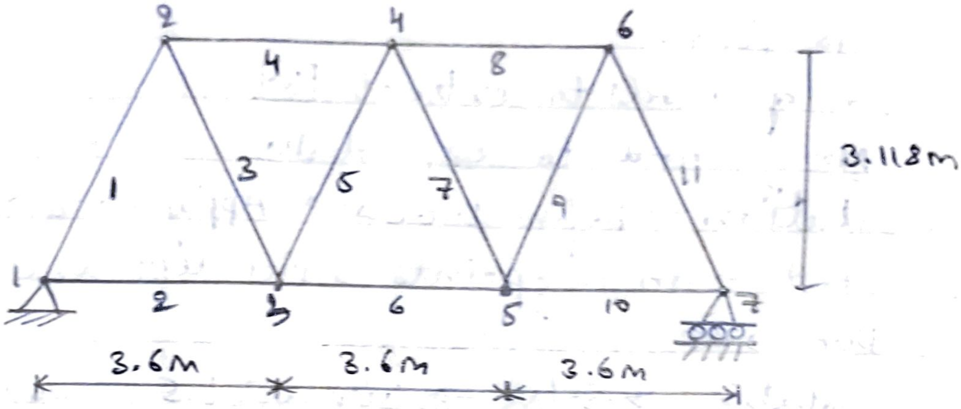
Result :

Mode	Ansys (Hz) Frequency	theory (Hz) Frequency
1	81.476	81.7
2	506.37	510.63
3	1278	1432.1

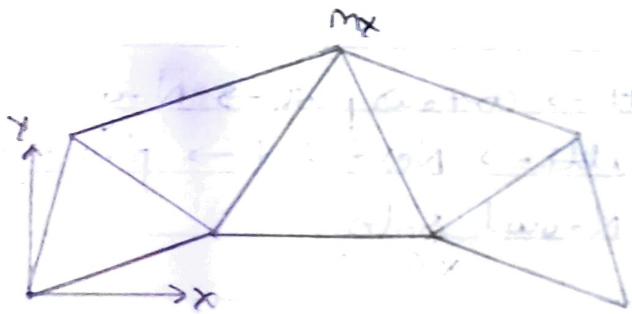
DD MM YYYY
11 08 2021

8. Solution → Analysis → New analysis → $\boxed{\cdot}$ model → Analy-sis option.
No of mods to extract $\boxed{10}$
No of mods to expand $\boxed{10}$
9. Solution → Define loads → Apply → structural → Displace-ment → on keypoints → pick left and key points → All DOF → ok.
10. Solution → Solve → current L.S → Solutions is done → ok
11. General post processor → Result symmery → Read result → first set.
next set.
→ plot result → contour plot → Nodal soln → Note down
→ Read results → Next set → plot results → counter plot → Nodal soln.

Result: The comparision is shown in table.

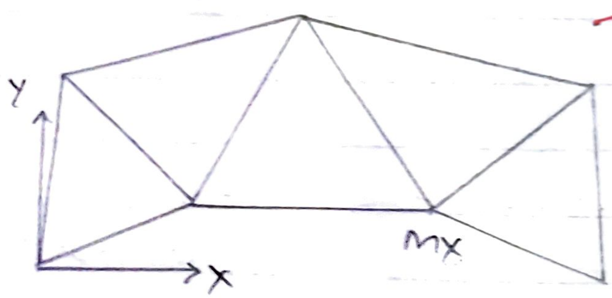


Set 1 :



Freq = 48.394 Hz

Set 2 :



Freq = 72.86 Hz

DD MM YYYY
11 08 2021

Dynamic analysis

3 Determine the modal deflections for the truss system shown below ($E = 200\,Gpa$, $A = 3250\,mm^2$).

Soln:- 1. preference → Select structural → ok.

2. preprocessor → Element type → Add → library of element → link 2D spar → ok.

3. Real constant → link1 → Area $= 3250 \cdot \times 10^{6}$

4. preprocessor → Material property → Material Model → structural → linear → elastic → Isotropic → Ex = 2e5 → PRXY = 0.3

5. Modeling → create → keypoints → inactive CS.

XYZ location	[0]	[0]
XYZ location	[1.8]	[3.118]
XYZ location	[3.6]	[0]
XYZ location	[5.4]	[3.118]
XYZ location	[7.2]	[0]
XYZ location	[9]	[3.118]
XYZ location	[10.8]	[0]

6. Modeling → create → lines → straight line → on key points.

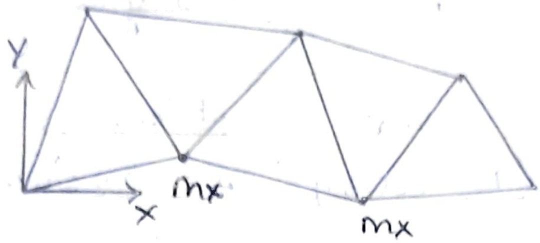
7. preprocessor → Meshing → Mesh tool → Global size → No of division - 1 → click on mesh → ok.

8. solution → Analysis type → New Analysis → [• Modal] → Analysis option.

No of Modes to extract [10]

No of Modes to expand [10]

Set 3 :



Freq = 152.89

Result :

Set	Time/Freq
1	48.384
2	72.864
3	152.89
4	195.24
5	246.76.

DD MM YYYY
11 08 2021

9. Solution → Define loads → Apply → structural → Displace -ment → on keypoints → pick left end and right end keypoint (1 & 7) DOF → UY → OK
10. Solution → Solve → current L.S → Solution → OK.
11. General postprocessor → Result summary → read result
→ first set.
→ Next set.
→ plot results → countour plot → Nodal soln → Note down
→ Read results → Next set → plot results → contour plots
→ Nodal soln.

Result: The result are tabulted.

11/8/21