Pre-Processor

1. Read control Parameters
2. Read / Generate nodal coordinates and boundary Conditions
3. Read / Generate element connectivity and element Loads.
4. Read material properties or constitutive matrices.
5. Read nodal loads and loadings 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.

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

04/08/2021

COMPUTER AIDED MODELLING AND ANALYSIS

Introduction to FEM :-

A commercial FEM system consists of three basics modules - preprocessor, solver and post processor. These modules and their functions are illustrated. 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 form that user can understand.

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. 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 types to be used.

→ Define the material properties of the elements.

→ Define the geometric properties of elements.

→ Define the element connectivities. (mesh the model).

D D M M Y Y Y Y

→ Define the physical constraints (Boundary Conditions).

→ Define the loadings.

The preprocessor step is critical. In no case is there a better example of the computer of the computer-related axion "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 variables. 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 is commonly used.

* Post Processing:-

Analysis and evaluation of the solution results is reffered to as post processing. Post processor software contains sophgsticated routines used for sorting, pointing and plotting selected result from a finite from a finite element solution.

D D M M Y Y Y Y

While solution data can be manipulated many ways in post processing, the most important objective is to apply sound engineering judgement 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 pocesor. The ANSYS product documentation is excellent and it includes command references, operations guide, modelling and meshing guide, basic analysis procedure guide, etc. Taken together, these manuals provide descriptions of procedures, commands, elements and theoritical details needed to use the ANSYS program. All the above except the ANSYS theory reference are available online through ANSYS help system.

Engineering capabilities of ANSYS products are structural analysis, thermal analysis, CFD analysis, electromagnetic field analysis, field and coupled field analysis and much more. Engineering Library in ANSYS lists 189 finite elements. They are broadly grouped into LINK, PLANE, BEAM, SOLID, CONTAC, COMBIM, PIPE, MASS, SHELL, FLUID, SOURCE, MATRIX, etc. Under each type, different shapes and orders complete the list.

In ANSYS, there are fundamentally different types of optimization. The first is referred to as design optimization, second is topology optimization.

D D MM Y Y Y Y

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

→ Build computer models or transfer CAD models of structure, products, components or system.

→ Apply operating loads or other design performance conditions.

→ Study physical response, such as stress levels, temperature distributions or electromagnetic fiels.

→ Optimize a design early in the product development process to reduce production cost.

FEM Procedure:-

1. Descritization 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 needed for the problem.

2. Derivation of element equations for all typical elements in the mesh.

(a) Construct the variational formulation of 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 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 eng. structures, mechanical systems, nuclear reactors, fluid dynamics, piping system, manufacturing processes.

→ Type of response

Liner static, non-linear static, fluid dynamics, thermal analysis, heat transfer, etc.

→ Material type

Isotropic, linear elastic, non-linear elastic, an isotropic, elasto-plastic, visco-elastic, temperature dependent, etc.

→ Material wall construction

monocoque, layered, sandwich, composed material.

→ Loading conditions

Static, dynamic, deformation dependent, contact / friction, temperature, etc.

→ Types of loads

Point load, line load, surface loads / traction force, thermal loading, etc.

→ Type of analysis

Stress analysis, dynamic response, vibration, stability analysis, optimization, manufacturing process, etc.

→ Stress analysis

Small / large deformation, elasto-plastic, visco-elastic, thermal effects, etc.

→ Dynamic response, vibration and sound

Free vibration | modal analysis, dynamic response of linear | non-linear damping, etc.

→ Stability analysis

Buckling instability

→ Optimization

Structural optimization, shape optimization, section optimization, material optimization.

→ Crash analysis

Impact, crash worthiness, etc.

→ Fluid analysis

Pressures, gas temperatures, convection coefficients, velocities

→ Automotive industry

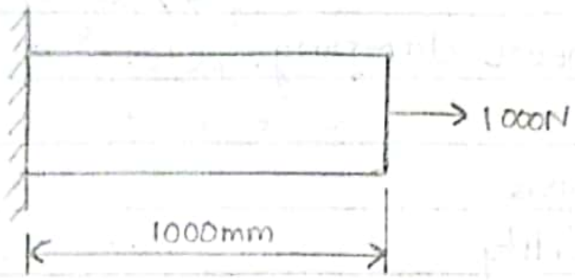
Static analysis, model analysis, transient dynamics, heat transfer, mechanics, etc.

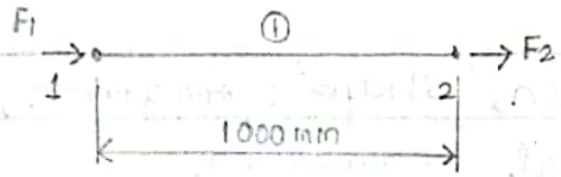
→ Architectural

Soil mechanics, rock mechanics, hydraulics, etc.

→ Aerospace industry

Static analysis, modal analysis, aerodynamics, etc.





FEM MODEL

Theoretical Solution:-

1) Stress, $\sigma = \dfrac{P}{A} = \dfrac{1000}{500} = 2\,N/mm^2$

(2). Deformation, $\delta l = \dfrac{Pl}{AE} = \dfrac{1000 \times 1000}{500 \times 2 \times 10^5}$

$\delta l = 0.01\,mm$

DD MM YYYY
04 08 2021

PROBLEM NO:- 1 BAR

(1) For the simple bar shown in the figure. Determine the displacement, Stress and the section the c/s of the bar is $500 mm^2$, length is 1000mm and yang's Modulus is $E = 2\times10^5 N/mm^2$. Take load P = 1000N.

Sol:- 1. Preferences ⇒ structural

2. Preprocessor ⇒ Element type ⇒ link ⇒ 2D Spar 1

3. Real Constants → Set 1 → $500 mm^2$

4. Material properties ⇒ structural ⇒ linear ⇒ elastic ⇒ isotropic ⇒ E = 2e5 Pa

5. Modelling → Create ⇒ keypoints ⇒ Active C.S ⇒ Point 1 - 0 ⇒ Point 2 - 1000

6. Create ⇒ Lines ⇒ Straight line ⇒ Click on required nodes.

7. Meshing ⇒ Size control ⇒ Manual size ⇒ All Lines ⇒ No. of divisions = 10

8. Mesh tool ⇒ Mesh ⇒ Pickall.

9. Define loads ⇒ Apply ⇒ Structural ⇒ Displacement on nodes 1 ⇒ All DOF
Force / moment ⇒ On nodes ⇒ 2nd Node ⇒ Magnitude 1000

10. Solve ⇒ Current L.S

11. General Post Processing ⇒ Plot results ⇒ Contour Plot ⇒ Nodal solution ⇒ X or Y component.

12. Element table ⇒ Add table ⇒ By sequence ⇒ L.S 1

13. Plot results ⇒ Contour plot ⇒ Element table ⇒ Yes, average

14. List results ⇒ Reaction solution.

Results:-

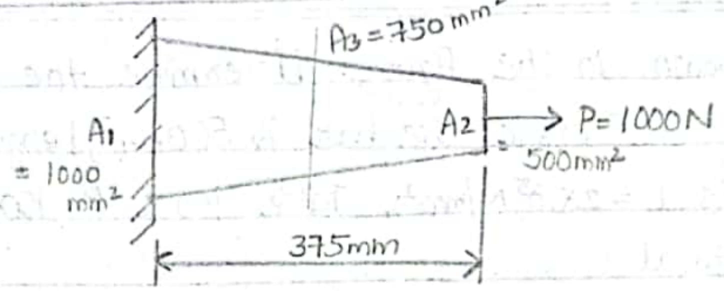
Solution are in global co-ordinate system

Node 1 : Fx = -1000 Fy = 0

Max. displacement = 0.01mm

Max. stress = 2 N/mm^2

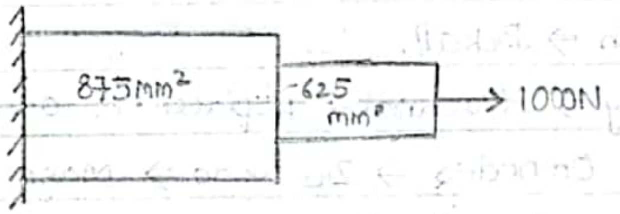
Theoritical Calculations:-

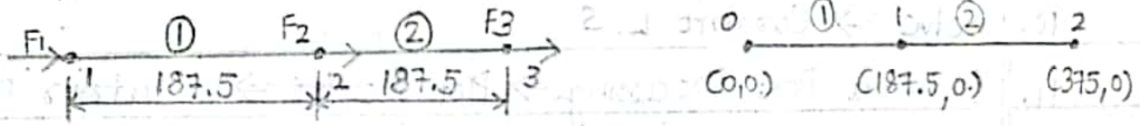


* Stepped bar 1 = $\frac{1000+750}{2}$ = $875 mm^2$

* Stepped bar 2 = $\frac{1000+500}{2}$ = $625 mm^2$

Modified bar:-





FEM MODEL

$$\delta l = \frac{4Pl}{\pi d_1 d_2 E}$$

$A_1 = (\pi/4) \times d_1^2$; $d_1 = \sqrt{(A_1 \times 4)/\pi} = 35.68 mm$

$A_2 = (\pi/4) \times d_2^2$; $d_2 = \sqrt{(A_2 \times 4)/\pi} = 25.23 mm$

$\delta l = (4 \times 1000 \times 375) / (\pi \times 35.68 \times 25.23 \times 2 \times 10^5)$

$\delta l = 2.65 \times 10^{-3} mm$

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

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

DSCE

PROBLEM NO. 2 TAPERED BAR

DD MM YYYY
04 08 2021

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

Sol: 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

5. Modelling ⇒ create ⇒ Keypoints ⇒ Active C.S ⇒ 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

8. Size control ⇒ Manual size ⇒ all line ⇒ No. of division = 10.

9. Mesh tool ⇒ Mesh ⇒ Pickall

10. Define loads ⇒ apply ⇒ structural ⇒ Displacement ⇒ On Nodes ⇒ All DOF

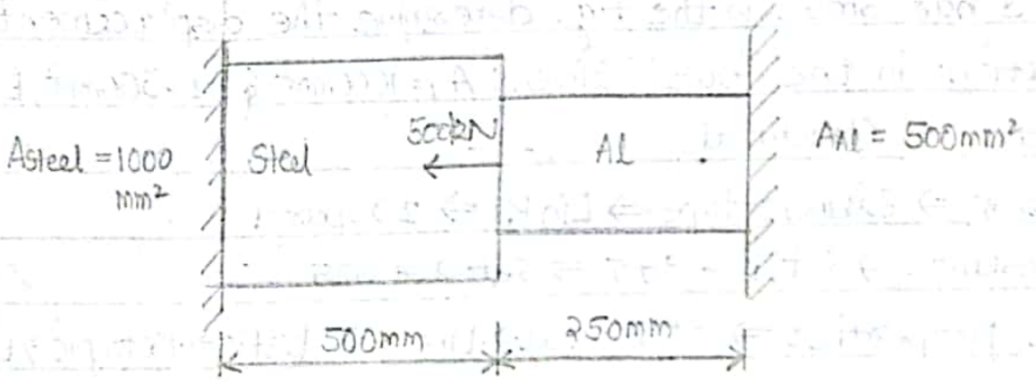
11. Force/Moment ⇒ on nodes ⇒ 2nd node ⇒ Magnitude = 1000

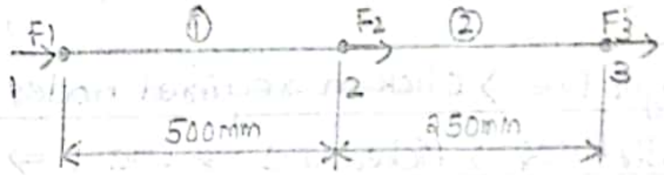
12. Solve ⇒ Current L.S

13. General post Processing ⇒ Plot results ⇒ contour plot ⇒ Nodal solution ⇒ X component.

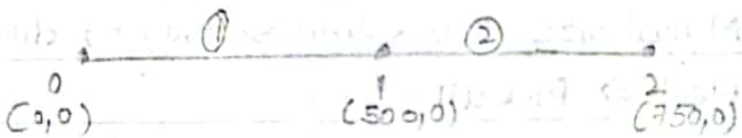
14. Element table ⇒ Add Table ⇒ By sequence ⇒ LS 1
Plot results ⇒ Contour Plot ⇒ Element table ⇒ Yes, average
List results ⇒ reaction solutions.

Results:-
Max. Stress = $1.6 N/mm^2$.
Max. displacement = 0.00257 mm.





FEM MODEL



Coordinates

Theoritical calculations:

$\delta L = \alpha \Delta T L$

$\delta l_1 = 11.7 \times 10^{-6} \times 500 \times 60 = 0.351 mm$

$\delta l_2 = 23 \times 10^{-6} \times 250 \times 60 = 0.345 mm$

$\delta L_T = 0.351 + 0.345 = 0.696 mm$

PROBLEM NO-3 STEPPED BAR OF TWO FIXED ENDS

D D M M Y Y Y Y
04 08 2021

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

Given :- $T_0 = 20°C$ and $T_1 = 80°C$, Area of steel, $A_{steel} = 1000mm^2$, $E_{steel} = 2e5 N/mm^2$, $A_{Al} = 500mm^2$, $E_{Al} = 0.7e5$, α steel $= 11.7e-6 \, 1/°C$, $\alpha_{Al} = 23e-6 \, 1/°C$.

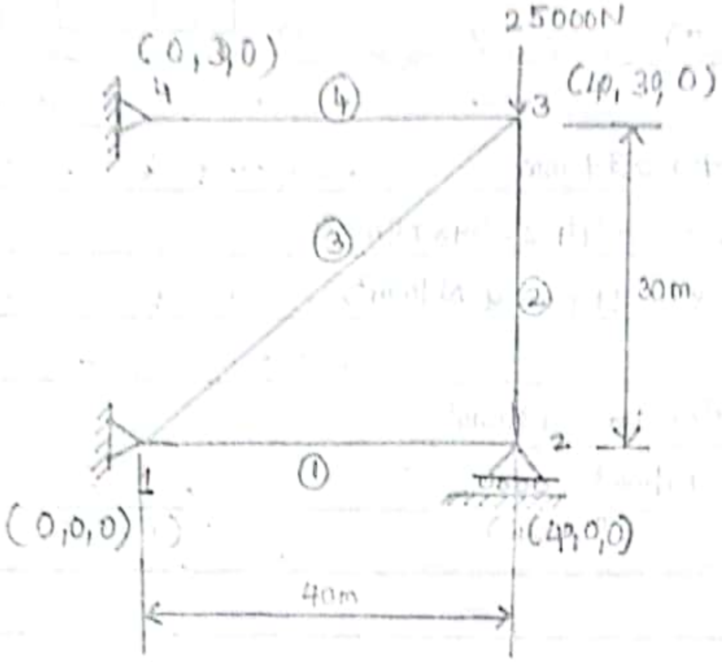
Sol:
1. Preference ⇒ Structural
2. Pre-processor ⇒ element type ⇒ link ⇒ 2D spar
3. Real Constant ⇒ Set 1 - 875 ⇒ set 2 - 625
4. Material properties ⇒ structural ⇒ Linear ⇒ Elastic ⇒ isotropic ⇒ E=2e5 ⇒ Px = 0.3 ⇒ secant coefficient ⇒ Isotropic ⇒ ALPX - 11.7e-6
5. Material ⇒ New Model ⇒ Structural ⇒ Linear ⇒ Elastic ⇒ isotropic ⇒ E = 0.7e5 ⇒ Secant coefficient ⇒ Isotropic ⇒ ALPX - 23e-6 °/c.
6. Modelling ⇒ Create ⇒ Key Points ⇒ Active C.S ⇒ Point 1-0 ⇒ Point 2-500 ⇒ Point 3 - 750
7. Create ⇒ Lines ⇒ straight line ⇒ select required nodes.
8. Meshing ⇒ Mesh attributes ⇒ Picked Lines ⇒ Line 1 ⇒ Linear Count No=2 Repeat for line 2 ⇒ Real constant No.2
9. Size control ⇒ manual size ⇒ all line ⇒ No. of divisions =10 Mesh tool ⇒ Mesh ⇒ Pickall.
10. Define loads ⇒ apply ⇒ Structural ⇒ displacement ⇒ on nodes Select Node 1 & Node 3 ⇒ All DOF.
11. Force / moment ⇒ On nodes ⇒ select Node 2 ⇒ -1000 ⇒ Temperature ⇒ on element ⇒ 60 ⇒ Pick all ⇒ 60
12. Solve ⇒ Current L.S
13. General Post Processing ⇒ Plot results ⇒ Contour Plot ⇒ Nodal Solution ⇒ Define element table ⇒ By sequence ⇒ LS, 1
14. Plot results ⇒ Contour Plot ⇒ Element table ⇒ Yes, average List results ⇒ reaction solution.

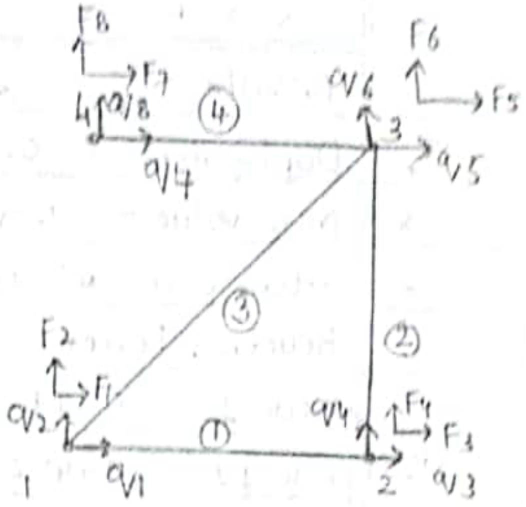
D D M M Y Y Y Y

Results:-

* Displacement = 0.75537 mm
* Min. value of stress = $-442.548\ N/mm^2$
* Max. value of stress = $114.904\ N/mm^2$
* Reaction Forces

- Node 1 = 0.44255 e+6 N/mm^2
- Node 12 = 57452 N/mm^2





FEM MODEL

D D MM YYYY
04 08 2021

PROBLEM-4 TRUSSES

(4.) Two bar truss shown in fig. Determine nodal displacements and the stress in each member. Take E = 2×10^5 MPa, Area=0.01m^2

Sol: 1. Preference ⇒ element type → line ⇒ 2D Spar 1

2. Real constant ⇒ Link 1 ⇒ Area = 0.01

3. Material Properties ⇒ Material Models ⇒ Structural ⇒ Linear ⇒ elastic → isotropic ⇒ Exx = 2e5

4. Modelling ⇒ Create ⇒ Keypoints ⇒ In Active CS

Nodes	X	Y	Z
1	0	0	0
2	40	0	0
3	40	30	0
4	0	30	0

5. Modelling ⇒ create ⇒ lines ⇒ Straight Line ⇒ On Key Points

6. Meshing ⇒ Size controls ⇒ Manual size ⇒ Lines ⇒ All Lines ⇒ No. of div=1

7. Meshing ⇒ Mesh tool ⇒ Mesh ⇒ Pick the line

8. Solutions ⇒ Define loads ⇒ Apply ⇒ Structural ⇒ Displacement ⇒ ON nodes ⇒ ALL DOF

9. Define loads ⇒ Force/moment ⇒ on nodes ⇒ Fy ⇒ on 3rd node - 200N

10. Solve ⇒ Current L.S

11. General Post Processor ⇒ Plot results ⇒ Contour Plot ⇒ Nodal Solution ⇒ DOF ⇒ X or Y component

12. Element table ⇒ Define table ⇒ By sequence ⇒ L.S1

12. Plot results ⇒ contour plot ⇒ Element table ⇒ Yes average

Results:-

* Deformation:-

DMX = 3.38542 m

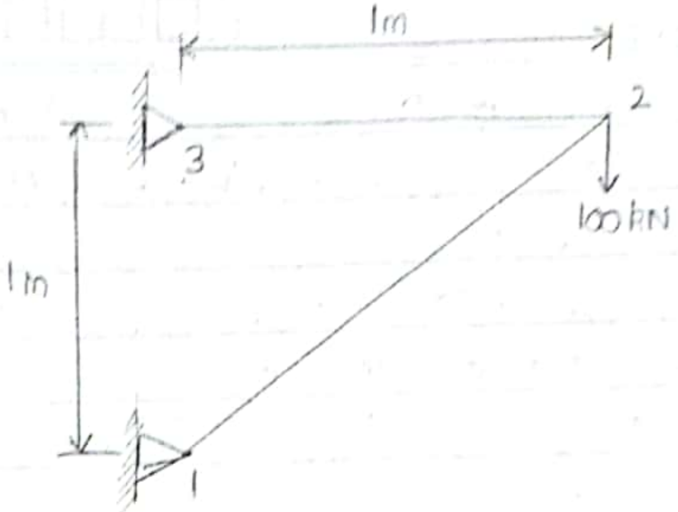
SMN = -3.28125 mm

* Stress:

DMX = 3.38542

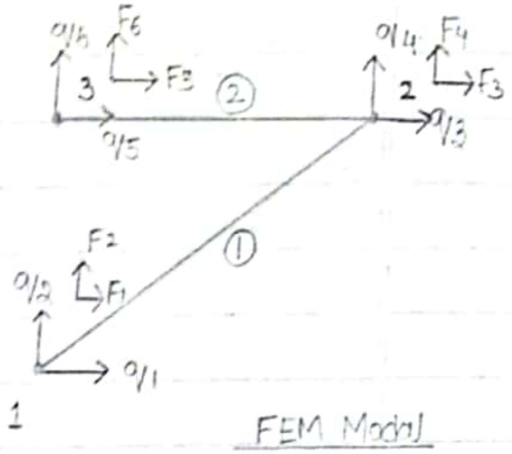
SMN = -10937.5.

SMX = 4166.67



Coordinates

Node	X	Y	Z
1	0	0	0
2	1	1	0
3	0	1	0



FEM Model

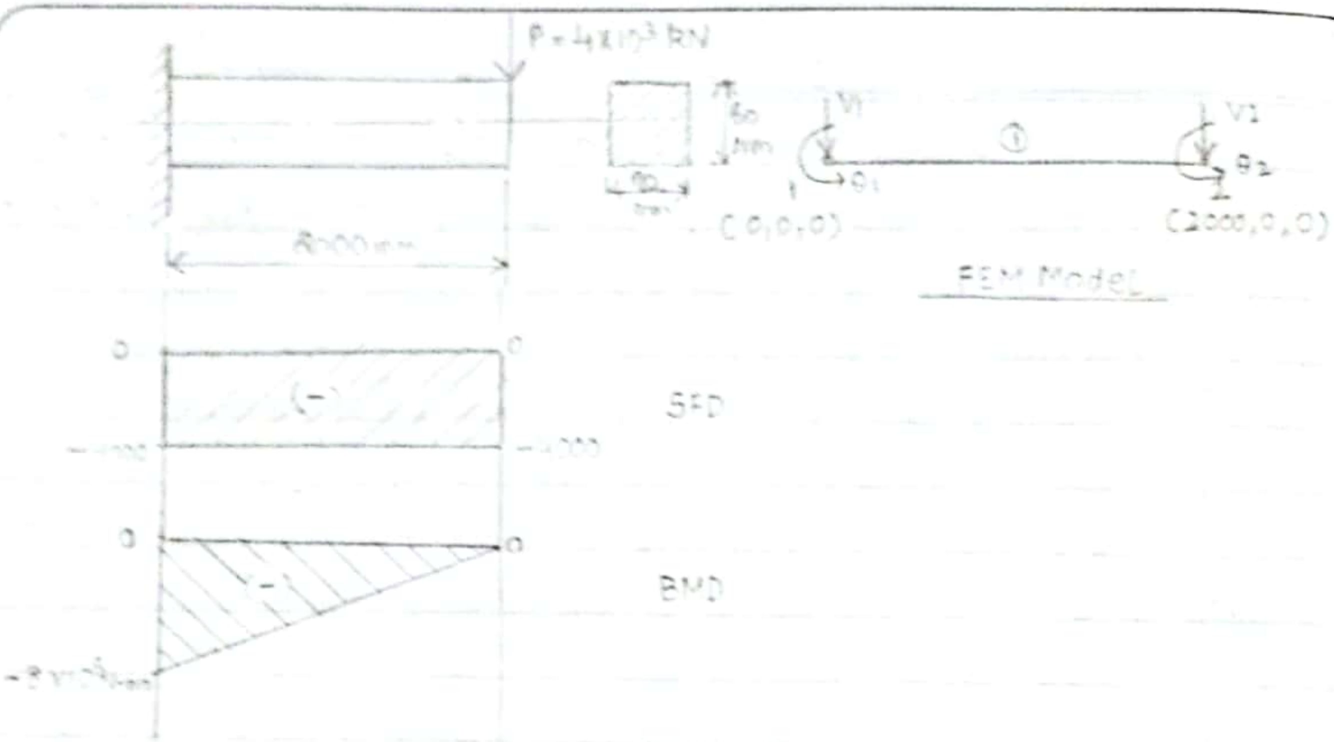
DD MM YYYY
04 08 2021

PROBLEM-5 TRUSSES

(5) Two bar truss shown in Fig. Determine total displacement & the Stress in each member. Take E = 210GPa, Area = 0.01 sq.m

Sol:- Steps:-

1. Preferences ⇒ Structural
2. Pre-processor ⇒ Element type ⇒ Add/Edit/Delete ⇒ Link ⇒ 2D spar1
3. Real constants ⇒ Set number 1 ⇒ C/s area ⇒ 0.01
4. Material properties ⇒ Material Models ⇒ Structural ⇒ Linear ⇒ Elastic ⇒ Isotropic ⇒ Ex = 210e3
5. Modelling ⇒ Create ⇒ Keypoints ⇒ InActive CS ⇒ Create Key points. [Coordinates on left side]
6. Modelling ⇒ Create lines ⇒ straight line ⇒ click on Key points.
7. Meshing ⇒ Size controls ⇒ Manual size ⇒ Lines ⇒ All lines = No. of element Division = 1
8. Mesh tool ⇒ Mesh ⇒ Pick the lines to be meshed.
9. Solution ⇒ Define Loads ⇒ Apply ⇒ Structural ⇒ Displacement ⇒ on nodes ⇒ Select node 1 & 3 ⇒ All DOF
10. Loads ⇒ Apply ⇒ Structural ⇒ Force/moment ⇒ on Nodes ⇒ Apply F/M on node 2 ⇒ Direction of Force/Moment Fy ⇒ Value = -100e3.
11. Solve ⇒ current L.S
12. General Post processor ⇒ Plot results ⇒ Contour plot ⇒ Nodal solution ⇒ DOF solution ⇒ Y-component of Displacement
13. Element table ⇒ Define table ⇒ Select by Sequence no., LS1
14. Plot result ⇒ Contour plot ⇒ Element Table ⇒ Yes-average.



Calculations:

$A = 60 \times 90 = 5400\,mm^2$

$\frac{d}{2} = \frac{90}{2} = 45\,mm$

$I = \frac{bd^3}{12} = \frac{60 \times (90)^3}{12} = 3645000\,mm^4$

$Z = \frac{I}{d/2} = \frac{bd^2}{6} = 81000\,mm^3$

$\sigma_{max} = \frac{M}{Z} = \frac{4000 \times 2000}{81000} = 98.765\,N/mm^2$

Deflection $= \frac{PL^3}{3EI} = 14.63\,mm$

D D M M Y Y Y Y
11 08 2021

PROBLEM-6 BEAMS

(6) 2m Long cantilever with rectanglar section (90mm x 60mm) 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= 200GPa, P= 4×10^3N.

Sol:- Procedure:-

1. Preferences ⇒ Structural
2. Preprocessor ⇒ Element type ⇒ Beam ⇒ 2D Elastics.
3. Real Constants ⇒ C/S Area - 5400 ⇒ Area moment of inertia ⇒ 3.45e8
4. Material Properties ⇒ Material models ⇒ Structural ⇒ Linear ⇒ Elastic ⇒ Isotropic ⇒ E_{xx} = 2e5
5. Modelling ⇒ Create ⇒ In Active C.S ⇒ Create Key points
   KP 1 (0,0)
   KP2 (2000,0)
6. Modelling ⇒ create → Lines ⇒ Straight Lines ⇒ Pick Key Points.
7. Meshing ⇒ Mesh Tool ⇒ Mesh ⇒ Pick the lines
8. Solution ⇒ Define loads ⇒ Apply ⇒ Structural ⇒ Displacement ⇒ on nodes ⇒ Node 1 ⇒ All DOF
9. Displacement ⇒ Force / moment ⇒ on Key Points ⇒ Node 2 ⇒ FY = -4×10^3
10. Solution → Solve ⇒ Current L.S
11. General Post Processor ⇒ Plot Results ⇒ Contour plot ⇒ Nodal Solution ⇒ DOF Solution ⇒ Y- Component of Displacement
12. Counter Plot ⇒ Nodal solution ⇒ Stress → Von-mises stress
13. Element Table ⇒ Define table ⇒ Add
   Use lablel - SFD1 ⇒ SMIC,6
   Use Lablel- SFD2 ⇒ SMIC, 12
   Use lablel - BMD1 ⇒ SMIC, 2
   Use Label - BMD 2 ⇒ SMIC, 8
14. Plot Line Element Results
   Element table item at node I - SFD1

D D M M Y Y Y Y

Element table item at node J :- SFD2

Same step for Bending moment Diagram.

15. List Results ⇒ Reaction solution ⇒ select all items.

Results:-

From ANSYS

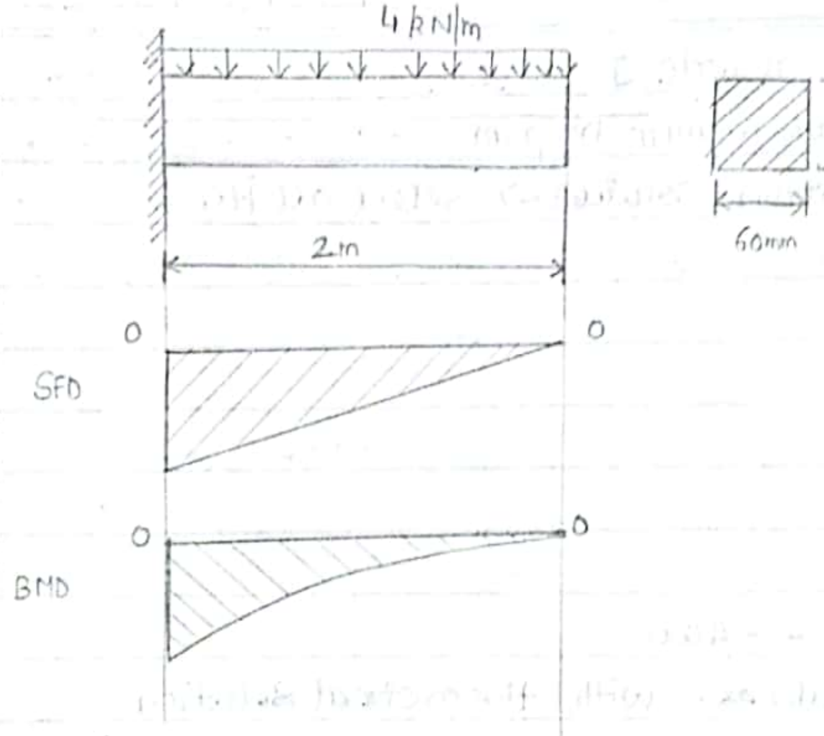
DMX = 14.632

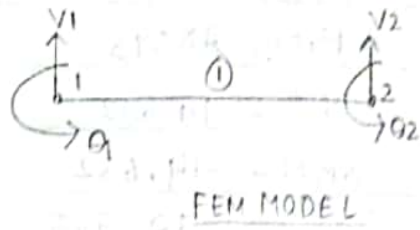
SMN = −14.632

SMX = 98.765

Max. Shear force = − 4000

ANSYS solution matches with theoretical solution.





Calculations:-

Moment of Inertia $= I = bd^3/12 = 60\times(90)^3/12 = 3645000\ mm^4$

$y = d/2 = 90/2 = 45\ mm$

$$\sigma_{max} = \frac{M \times y}{I}$$

$$= \frac{8\times10^6 \times 45}{364.5\times10^4} = 98.76\ N/mm^2$$

$$\text{deflection} = \frac{PL^4}{8EI} = \frac{4000\times(2000)^4}{2(200\times10^5\times364\times10^4)}$$

$$= 10.989\ mm$$

DD MM YYYY
11 | 08 | 2021

PROBLEM-7 BEAMS

(7). A 2m Long Cantilever with rectangular section (60mm x 90mm) is subjected to uniformly distributed load 4 kN/m through out its length. Determine maximum bending stress and its deflection. Take E= 200GPa

Sol: Procedure:

1. Preferences ⇒ Structural
2. Preprocesser → Element type ⇒ Add/Delete ⇒ Beam ⇒ 2D Elastics.
3. Material Properties ⇒ Material model ⇒ Structural ⇒ Elastic ⇒ isotropic ⇒ Exx = 200e3.
4. Sections ⇒ Beam ⇒ Common sections ⇒ Rectangle ⇒ b=60, d=90
5. Modelling ⇒ Create ⇒ Keypoints ⇒ In Active C.S ⇒ Eenter coordinates
   Node 1 - (0,0,0) ; Node 2-(2000,0,0)
6. Modelling ⇒ Lines ⇒ straight line ⇒ Pick the Keypoints 1 & 2
7. Meshing ⇒ Mesh tool ⇒ Lines set ⇒ choose Line ⇒ No.of div=1
   Mesh tool ⇒ Mesh ⇒ select the Line.
8. Solution ⇒ Define load ⇒ Apply ⇒ Structural ⇒ Displacement ⇒ on node ⇒ select Node 1 ⇒ ALL DOF
9. Pressure ⇒ on Line ⇒ Select line ⇒ Value -4
10. Solve ⇒ Current L.S ⇒ OK
11. General - Post Processor ⇒ Plot result ⇒ Contour Plot ⇒ Nodal solution ⇒ DOF Solution ⇒ Y component of Displacement.
12. Element Table ⇒ Define table ⇒ Add
    Use Lablel - SFD1 ⇒ SMIC, 6
    Use Lablel - SFD2 ⇒ SMIC, 12
    Use Lablel - BMD1 ⇒ SMIC, 2
    Use Label - BMD2 ⇒ SMIC, 8
13. Plot Line Element Results.
    Element Table item at node I - SFD1
    Element Table item at node J - SFD2

D D M M Y Y Y Y

Same step for Bending moment Diagram

14. List Results ⇒ Reaction solution ⇒ Select all items.

Results:-

From ANSYS

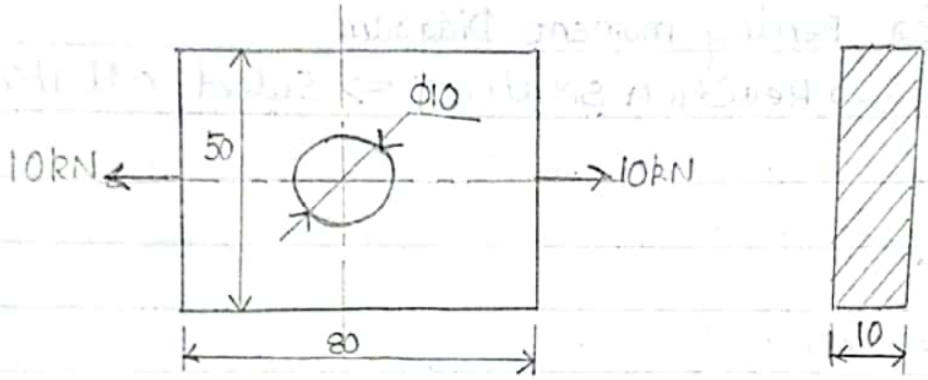
Deformation = 10mm

Stress = 98.76 N/mm^2

From Theory

Deformation = 10.98mm

Stress = 98.76 N/mm^2



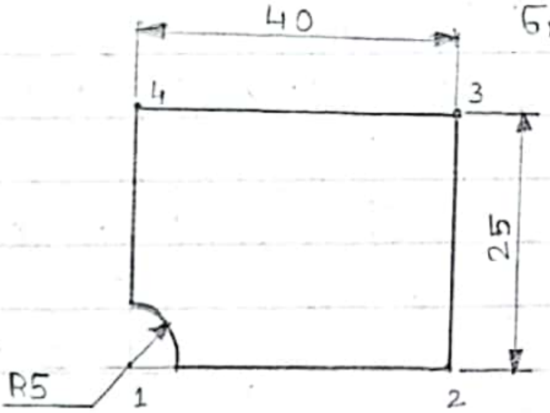
Calculations :-

$$\sigma_{nor.} = \frac{F}{(b-d)\,t} \quad \frac{10\times10^3}{(50-10)\times10} = 25\,N/mm^2$$

Stress concentration factor , $K_\sigma = \dfrac{\sigma_{max}}{\sigma_{norm.}}$

$$2.5 = \frac{\sigma_{max}}{25}$$

$$\sigma_{max} = 2.5\times25 = 62.5\,N/mm^2$$



FEM MODEL

Coordinates

Node	X	Y	Z
1	0	0	0
2	40	0	0
3	40	25	0
4	0	25	0

DSCE

D D M M Y Y Y Y
11 08 2021

PROBLEM-8 PLATE WITH A HOLE

(8) Determine the maximum stress for a rectangular plate of 50mm X 80mm with hole of 10mm diameter in the centre is loaded is axial tension. Thickness of the plate is 10mm. Take E = 200GPa. All dimensions in mm.

Sol:- Procedure:-

1. Pre-Processor ⇒ Structural
2. Pre-Processor ⇒ Element type ⇒ solid, Quad 4 node 42 ⇒ ok
3. Element type ⇒ Type PLane 42 ⇒ Element behavior K3 ⇒ plane stress w/thk.
4. Preprocessor ⇒ Real constants ⇒ Type PLANE 42 ⇒ Thickness TH 10 ⇒ close
5. Preprocessor ⇒ Material Property ⇒ Material Model ⇒ structure ⇒ Linear ⇒ Elastic ⇒ Isotropic ⇒ Exx = 2e5
6. Modelling ⇒ create ⇒ key Point ⇒ In active C.S

Nodes	X	Y	Z
1	0	0	0
2	40	0	0
3	40	25	0
4	0	25	0

7. Modelling ⇒ create ⇒ Areas ⇒ Through Key Points.
8. Modelling ⇒ Create ⇒ Areas ⇒ Circle ⇒ Solid circle ⇒ X=0 Y= R=5
9. Modelling ⇒ operate ⇒ Booleans ⇒ Substract ⇒ Areas
(i) Pick base area from which to Sbstract
(ii) Pick area to be substracted.
10. Meshing ⇒ Mesh tools ⇒ Mesh ⇒ Mesh area ⇒ Mesh at element ⇒ Pickall ⇒ Refine Mesh ⇒ level of refinement=3.
11. Solution ⇒ Define load ⇒ Apply load ⇒ structural ⇒ Displacement ⇒ Symmetry B.C ⇒ On lines
12. Define Load ⇒ Apply ⇒ structural ⇒ Pressure ⇒ on lines

D D M M Y Y Y Y

⇒ Load Pressure value = 20

13. Solution ⇒ Solve ⇒ Current L.S

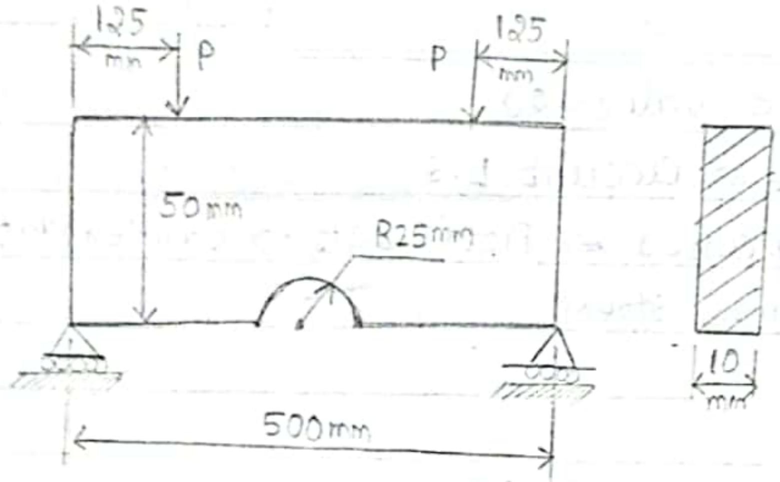
14. General Post processor ⇒ Plot results ⇒ Counter Plot ⇒ nodal Solution ⇒ Von mises stress.

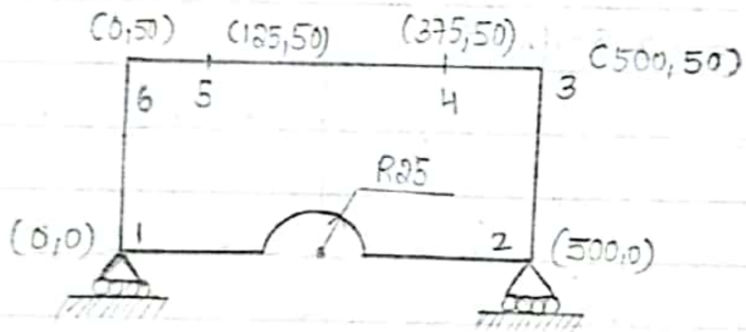
Results:-

Max. deformation = 0.008657 mm

Max. stress = 62.077 N/mm^2

Theoritical stress = 62.5 N/mm^2





Nodal solution :-

X -direction

DMX = 0.026615

SMN = 0.013025

SMX = -0.004877

Y-Direction

DMX = 0.026615

SMN = -0.024916

SMX = -0.502 E-03

PROBLEM-9 PLATE PROBLEM

DD MM YYYY
11 08 2021

9. Calculate the stress and displacement for the plate shown below. Let the load be P=100N applied at equal distance from both ends and E= 3e5 N/mm²

Sol: Procedure:

1. Preprocessor ⇒ Element type ⇒ Solid ⇒ Quad 4 node 42 ⇒ OK
2. Element type ⇒ Type PLANE 42 ⇒ Element behavior K3 ⇒ Plane stress w/ thk.
3. Real Constants ⇒ TYPE PLANE 42 ⇒ Thickness THK: 10
4. Material Property ⇒ Material Model ⇒ Structural ⇒ Linear ⇒ Elastic ⇒ Isotropic ⇒ Exx = 3e5.
5. Modelling ⇒ Create ⇒ Key points ⇒ In active C.S

Nodes	X	Y	Z
1	0	0	0
2	500	0	0
3	500	50	0
4	375	50	0
5	125	50	0
6	0	50	0

6. Modelling ⇒ Create ⇒ Areas ⇒ Through KPS
7. Modelling ⇒ Create ⇒ Area ⇒ Circle ⇒ solid Circle ⇒ X=250; Y=0; Radius = 25
8. Modelling ⇒ Operate ⇒ Booleans ⇒ Substract ⇒ Area
(i) Pitch box area from which to substract.
(ii) Pitch area to be substracted.
9. Meshing ⇒ Mesh tool ⇒ Mesh ⇒ Mesh area ⇒ Mesh at Element ⇒ Pick all ⇒ Refine mesh ⇒ level of refinement: 3
10. Loads ⇒ Define loads ⇒ Structural ⇒ Force/moment ⇒ No.node ⇒ Pick node 1 and 2 ⇒ apply ⇒ DOF - UY - OK
11. Load ⇒ Define Load ⇒ apply ⇒ Structural ⇒ Force/moment

Nodal Solution (Stress)

Y component

DMX = 0.026615

SMN = -3.159

SMX = 3.404

von Mises stress

DMX = 0.026615

SMN = 0.004078

SMX = 14.131

DSCE

D D M M Y Y Y Y

⇒ on key point ⇒ select Keypoint 4 and 5 → 100

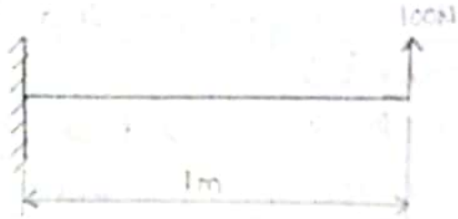
12. Solution ⇒ Solve ⇒ current L.S

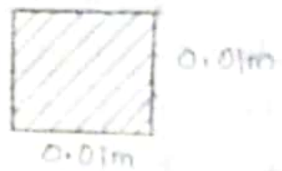
13. Solution ⇒ General Post-Processor ⇒ Plot results ⇒ Counter plot ⇒ Nodal solution ⇒ Von mises solution.

Result:-

Max. deformation = 0.026615 mm

Max. Stress = 14.131 N/mm²





$E = 206800e6 \; N/m^2$

$I = 8.33e\text{-}10 \; N/m^4$

$\rho = 7830 \; Kg/m^3$

$A = 0.01 \times 0.01 \; m^2$

D D	M M	Y Y Y Y
11	08	2021

PROBLEM - 10 DYNAMIC ANALYSIS

(10). Simple harmonic analysis of a cantilever beam as shown below

$E = 206800e6\ N/m^2$, $I = 8.33e-10\ mm^4$, $\rho = 7830\ Kg/m^3$

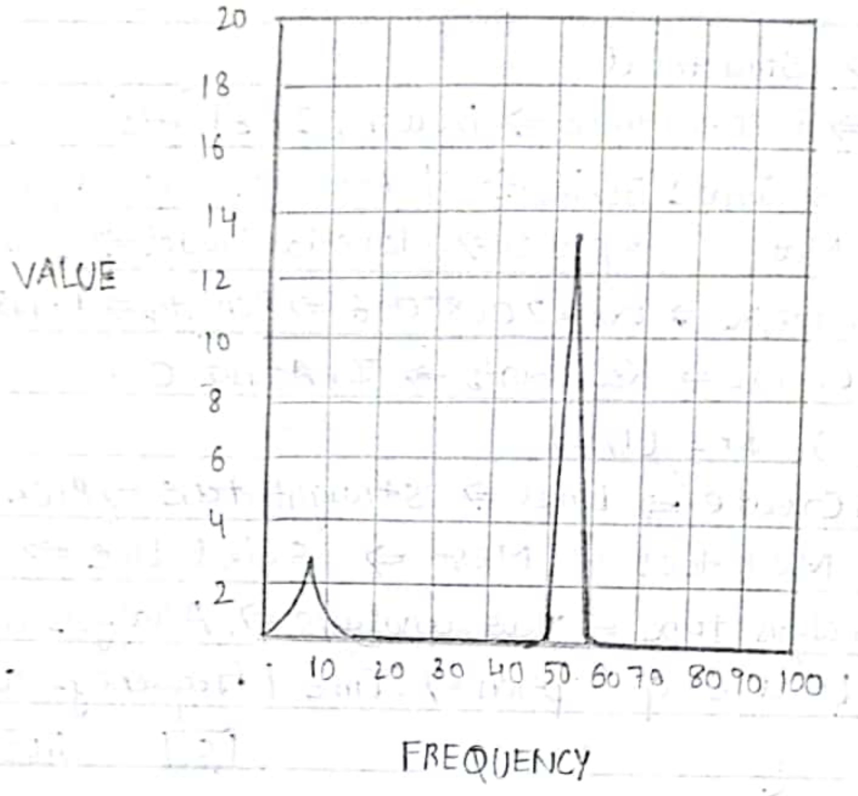
Sol:- Procedure:-

1. Preferences ⇒ Structural
2. Pre-processor ⇒ Element type ⇒ Beam, 2D Elastic
3. Real constants ⇒ Type 1 BEAMS ⇒ C/S Area = 0.01x0.01 ⇒ I = 8.33e-10
4. Preprocessor ⇒ Material properties ⇒ Material Model ⇒ Linear ⇒ elastic ⇒ Isotropic ⇒ Exx = 206800e6 ⇒ Density ⇒ DENS = 7830
5. Modelling ⇒ Create ⇒ Key Points ⇒ In Active C.S
   KP 1 (0,0) ; KP 2 (1,0)
6. Modelling ⇒ Create ⇒ Lines ⇒ straight line ⇒ Pick end points.
7. Meshing ⇒ Mesh tool ⇒ Mesh ⇒ Select line ⇒ OK
8. Solution ⇒ Analysis type ⇒ New analysis ⇒ Analysis option ⇒ Harmonic ⇒ Load step option ⇒ Time / frequency and substeps.

   | 0 | | 100 |

   | 100 |

   ☑ stepped
9. Solution ⇒ Define loads ⇒ Apply ⇒ Structural ⇒ Displacement ⇒ on Keypoints ⇒ Pick left end Key point ⇒ ALL DOF
10. Solution ⇒ Define loads ⇒ Apply ⇒ Structural ⇒ Force / moment ⇒ on keypoints ⇒ Pick right end point ⇒ Fy ⇒ Value of force / moment = -100
11. Solution ⇒ Solve ⇒ current L.S
12. Time List Post Processor ⇒ A window opens ⇒ click on given clover ⊞ on left corner of window ⇒ click on element shown in window ⇒ | VU2 | 2 | Y-Com dis |
13. DOF solution ⇒ Y component ⇒ Select a point where load time is actual (Right end K.P). ⇒ Graph appears

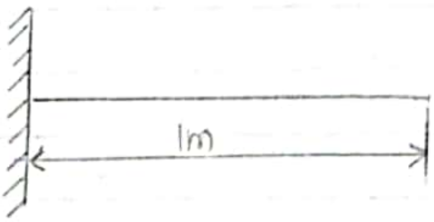


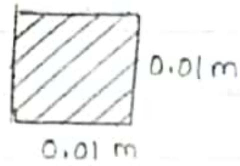
VALUE
20
18
16
14
12
10
8
6
4
2
10 20 30 40 50 60 70 80 90 100
FREQUENCY

DD MM YYYY

Result:

The graph of frequency v/s values is plotted.





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

$I = 8.33e{-6} m^4$

$A = 0.01 m^2$

$\rho = 7830 \ kg/m^3$

D D M M Y Y Y Y
11 08 2021

PROBLEM NO-11 MODAL ANALYSIS

(11) Simple modal analysis of a cantilever beam is as shown in fig. Objective is to get the frequency and different modes of vibration.

E = 2e11 N/m2 ; I = 8.33e-06 m4 ; A = 0.01 m² ; ρ = 7830 Kg/m³

Sol: Procedure:-

1. Preferences ⇒ Structural
2. Pre-processor ⇒ element type ⇒ Add ⇒ Select Beam, 2D elastic
3. Main menu ⇒ Real constant ⇒ Add ⇒ Type 1 Beam 3 ⇒ C/S Area = 0.1 × 0.1
   I = 8.33e-06 ⇒ Beam Height = 0.1
4. Preprocessor ⇒ Material properties ⇒ Material Models ⇒ Structural ⇒ Linear ⇒ Elastic ⇒ Isotropic ⇒ Exx = 2e11 ⇒ Density ⇒ DEN ⇒ 7830
5. Pre Processor ⇒ Modelling ⇒ Create ⇒ Key Points ⇒ Inactive C.S
   → K.P 1 (0,0)
   → K.P 2 (1,0)
6. Pre Processor ⇒ Modelling ⇒ Create ⇒ Lines ⇒ Straight lines ⇒ Pick end key points of Lines.
7. PreProcessor ⇒ Meshing ⇒ Mesh tool ⇒ Mesh ⇒ select line ⇒ OK
8. Solution ⇒ Analysis ⇒ New analysis ⇒ • Metal
   ⇒ Analysis option
   ⇒ No. of mods to extract 10
   ⇒ No. of much to expand 10
9. Solution ⇒ Define loads ⇒ Apply ⇒ Structural ⇒ Displacement ⇒ On Key Points ⇒ Pick left end Key Point ⇒ ALLDOF ⇒ OK
10. Solution ⇒ Solve ⇒ Current LS
11. General Post Processar ⇒ Result Summary ⇒ Read result - First set ; next set.
   ⇒ Plot result ⇒ Contour Plot ⇒ Nodal Solution ⇒ Note Down
   ⇒ Read result ⇒ Next set ⇒ Plot result ⇒ Counterplot ⇒ Nodal solution.

Nodal Solution for first set

STEP -1

Sub= 1

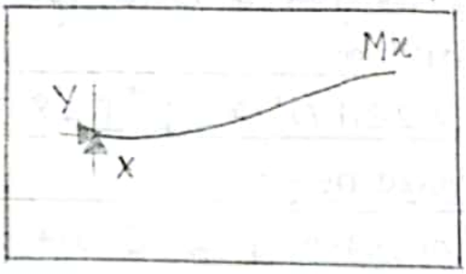
FREQ -81.468

UV (AUg)

RSUS =0

DMX = 0.225625

SMX = 0.225625



Nodal solution for second set

STEP=1

Sub =2

FREQ =504.76

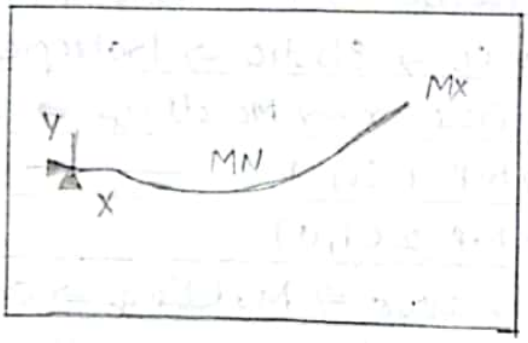
UV (AUg)

RSUS =0

DMX= 0.222385

SMX = 0.222388

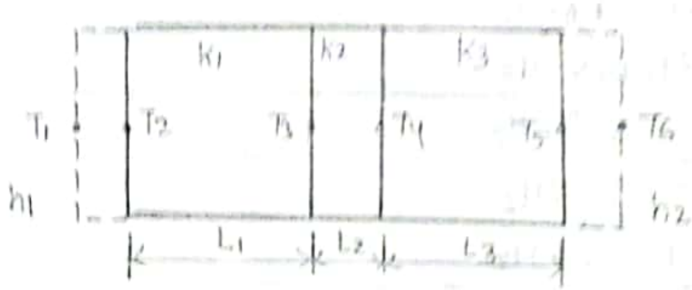
SMN = -0.160909



DD MM YYYY

Result Summary:-

Set	Time/Freq.
1	81.468 Hz
2	504.76 Hz
3	1263.5 Hz
4	1388.3 Hz
5	2653.3 Hz



$K_1 = 1.5$ W/m.K ; $K_3 = 4.9$ W/m.K

$L_1 = L_2 = 100$mm

$T_1 = 820°C$

$T_6 = 110°C$

$R_c = R_2 = 0.001\ m^2 K/W$

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

$$R_2 = \frac{L_2}{K_2 A_2}$$

Take $L_2 = 1$ mm

$K_2 = 1$ W/m.K

$A = 1\ m^2$

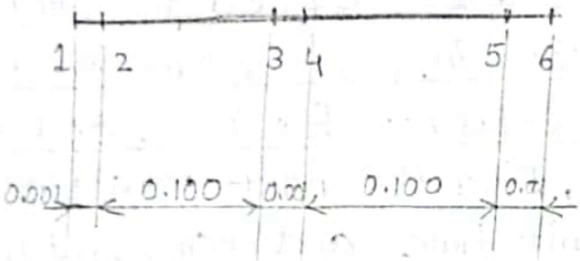
PROBLEM NO-12 HEAT TRANSFER

D D M M Y Y Y Y

(12) A furnace wall is made of inside silica brick ($k = 1.5 W/m.K$) and outside magnesia brick ($K = 4.9 W/mK$), 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 m²K/W; the heat transfer coefficient for inner and outside surfaces is equal to 35 W/m^2K. Find the heat flow through the wall per unit area per unit time and temperature distribution across the wall.

Sol:- Procedure:-

1. Preference ⇒ Thermal
2. Preprocessor ⇒ Element type ⇒ Add/Delete ⇒ Element type ⇒ Add
   Library of element types menu
   Select link 2D conduction 32
   Element type reference number : 1
   Apply
   Select: Link Convection 34
   Element type reference number : 2
3. Preprocessor ⇒ Real constant ⇒ 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
4. In element type for real constant Menu.
   Select: Type 2 Link 32 ⇒ OK
5. Real Constant ⇒ Set No. 2
   Convection Surface Area : 1 ⇒ OK
6. Real constant ⇒ select Set 2 ⇒ Add



FEM Model

Coordinates

1 → (0, 0, 0)
2 → (0.001, 0, 0)
3 → (0.101, 0, 0)
4 → (0.102, 0, 0)
5 → (0.202, 0, 0)
6 → (0.203, 0, 0)

DD MM YYYY

7. Preprocessor ⇒ Material properties ⇒ Material model ⇒ Material model Number 1 → Thermal → Conductivity ⇒ Isotropic ⇒ Conductivity for material number 1 menu ⇒ Kxx = 1.5 ⇒ OK.
Material property ⇒ Material model ⇒ Define material ⇒ Material ⇒ New model ⇒ Material model number 1 ⇒ Define material ID: 2 ⇒ OK.

In Define material model behaviour Menu ⇒ Material model number 2 ⇒ Thermal ⇒ Conductivity ⇒ Isotropic ⇒ Conductivity for material number 2 Menu ⇒ Kxx = 1 ⇒ OK

Do the same for remaining material.

8. Preprocessor ⇒ Modelling ⇒ Create ⇒ Nodes ⇒ In active C.S

Nodes	x	y	z
1	0	0	0
2	0.001	0	0
3	0.101	0	0
4	0.102	0	0
5	0.202	0	0
6	0.203	0	0

9. Modelling ⇒ Create ⇒ Elements ⇒ Element Attributes
Element type number : 2 Link34
Material number : 1
real constant set No. : 2
Modelling ⇒ Create ⇒ Element ⇒ Auto Numbered ⇒ Through Nodes ⇒ Element from Nodes menu (Pick the nodes defining the element) ⇒ OK

D D M M Y Y Y Y

Create ⇒ Element ⇒ Element Attributes ⇒ Element Attribute.
menu ⇒ Element type number : 1 link 32
Material number : 2
Real constant set No: 1 ⇒ OK.

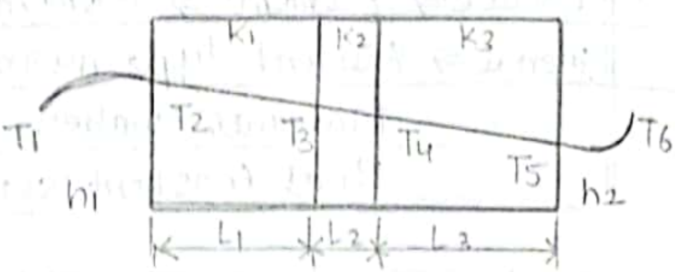
Modelling ⇒ Create ⇒ Element ⇒ Auto numbered ⇒ Through Nodes ⇒ Element from Nodes Menu (Pick the nodes defying the element) ⇒ OK

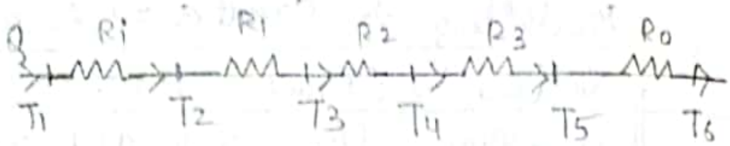
Create ⇒ Elements ⇒ Element attributes.
Element type number : 1 link 32
Material Number : 3
Real constant Set no: 1
Modelling ⇒ Create ⇒ Element ⇒ Auto Number ⇒ Through Nodes ⇒ Element from Nodes Menu (Pick the node defining element).

Create ⇒ Element ⇒ Element attributes.
Element type No: 1 Link 32
Material number : 4
Real constant Set No: 1
Modelling ⇒ Create ⇒ Elements ⇒ Auto Numbered ⇒ Through Nodes. Element from Notes Menu (Pick the nodes defining the element).

Create ⇒ Element ⇒ Element Attributes.
Element Attributes Menu.
Element type No: 2 link 34
Material Number : 4

Theoritical calculations





$L_1 = 10cm = 10 \times 10^{-2} m$

$K_1 = 1.5\ W/m.K$

$L_3 = 10cm = 10 \times 10^{-2} m$

$K_3 = 4.9\ W/m.K$

$R_2 = 0.001\ m^2K/W$

$T_1 = 820°C$

$T_6 = 110°C$

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

$$\frac{Q}{A} = \frac{T_1 - T_6}{\frac{1}{h_1} + \frac{L_1}{K_1} + \frac{L_2}{K_2} + \frac{1}{h_2} + \frac{L_3}{K_3}} = \frac{820 - 110}{\frac{1}{35} + \frac{10 \times 10^{-2}}{1.5} + 0.001 + \frac{10 \times 10^{-2}}{4.9} + \frac{1}{35}}$$

$$\frac{Q}{A} = 4889.21\ W/m^2$$

$A = 1m^2 \quad \therefore \quad \boxed{Q = 4889.21\ W}$

Intermediate Temperatures:-

$$Q = \frac{T_1 - T_2}{1/h_1 A_i} = \frac{820 - T_2}{1/35} \; ; \quad \boxed{T_2 = 680.3°C}$$

$$Q = \frac{T_2 - T_3}{L_1/K_1 A_1} \qquad T_3 = T_2 - \left(Q \times \frac{L_1}{K_1 A_1}\right)$$

$$= 680.3 - \left(\frac{4889.21 \times 10 \times 10^{-2}}{1.5 \times 1}\right)$$

$$\boxed{T_3 = 354.36°C}$$

$$Q = \frac{T_5 - T_6}{\frac{1}{h_2 A_o}} \qquad T_5 = T_6 + \left(Q \times \frac{1}{h_2 A_o}\right)$$

Real constant Set no:2
OK
Modelling ⇒ Create ⇒ element ⇒ Auto numbered ⇒ Through Nodes element from Nodes Menu (Pick the nodes defining the element)

Preprocessor ⇒ Modelling
click Plot controls.
Element |Attributes numbering : Element number
10. Solution ⇒ Analysis type ⇒ New analysis
New analyses Menu ⇒ select : steady state.
11. Solution ⇒ Define Loads ⇒ Apply ⇒ Thermal ⇒ Temp. ⇒ on Nodes
Apply Temp. on Nodes Menu (Pick nodes for Temp. specification
(Select 1) ⇒ OK
Apply Temperatures on Nodes menu.
DOF to be constrained : TEMP
Valve load Temp. value : 820 ⇒ Apply.
Apply Temp. on nodes menu (Pick node for Temp. Specifications). (Select 5) ⇒ OK.
Apply Temp on Nodes Menu ⇒ Select Temp ⇒ Value Load.
Temp. value :110 > OK.
12. Solution ⇒ Solve ⇒ Current L.S ⇒ Solve ⇒ Solution ⇒ Done.
13. Main Menu ⇒ General Post processor ⇒ Read results ⇒ Last Set ⇒ General post processer ⇒ Last results ⇒ Nodal Solutions
List Nodal Solution Menu ⇒ Nodal Solution ⇒ DOF Sol.
PRNSOL Command Menu (Note down the temp. value from the menu.)
14. General post Processing ⇒ Plot results ⇒ Counter plot ⇒ Nodal Solution Counter Nodal Solution Data Menu

$$= 110 + \left(4889.21 \times \frac{1}{35 \times 1}\right)$$

$$\boxed{T5 = 249.69°C}$$

$$Q = \frac{T_4 - T_5}{\frac{L_3}{K_3 A_3}} \quad ; \quad T_4 = T_5 + \left(Q \times \frac{L_3}{K_3 A_3}\right)$$

$$= 249.69 + \left(\frac{4889.21 \times 10 \times 10^{-2}}{4.9 \times 1}\right)$$

$$\boxed{T_4 = 349.47°C}$$

DSCE

Scanned with CamScanner

D D M M Y Y Y Y

15. General post processor ⇒ element table → Define table ⇒ Add ⇒ Define additional element table item ⇒ SMIC I.

Result:-

Node	Temperature °c
1.	820
2.	680.31
3.	354.36
4.	349.47
5.	249.69
6.	110

Maximum absolute value at Node 1 and its value is 820 °C

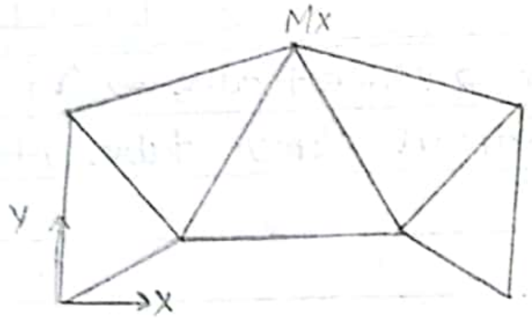
STAT Element	Current SMIS1
1	4889.2
2	4889.2
3	4889.2
4	4889.2
5	-4889.2

Minimum value
Element 5
Value = -4889.2

Maximum value
Element 4
Value 4889.2

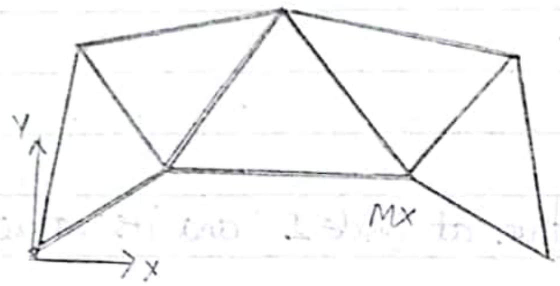
Scanned with CamScanner

Set(1):-



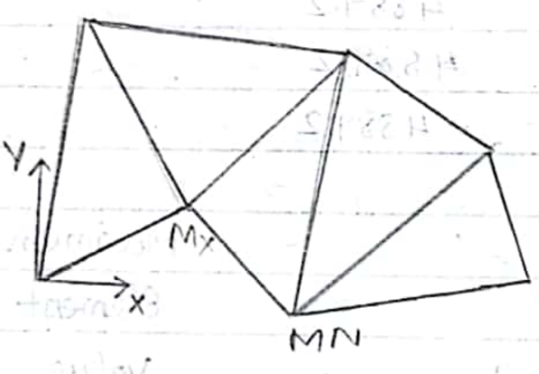
Freq:- 48.394 Hz
DMX:- 0.041534
SMX:- 0.036817

Set(2):-

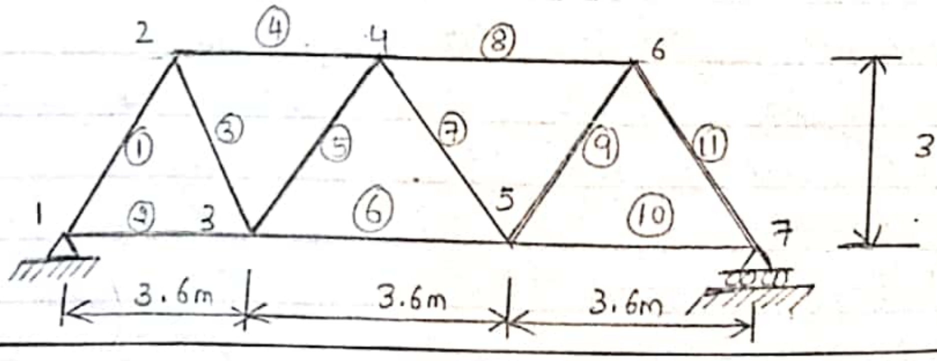


Freq:- 72.864
DMX:- 0.042565
DMX:- 0.026339

Set(3):-



Freq:- 152.89
DMX:- 0.049504
SMX:- -0.04612
SMN:- 0.048317



FEM MODAL

DSCE

DD MM YYYY: 11 08 2021

PROBLEM-13 MODAL ANALYSIS OF TRUSS

(13). Determine the modal deflection for the truss system shown in fig. E = 200GPa , A = 3250 mm^2

Sol:- Procedure:-

1. Preferences → structural
2. Preprocessor → Element type → Link 2D spar 1
3. Real Constant → c/s Area = 3250×10^{6}
4. Material Properties → Material Model → Isotropic → Exx 2e5, Pxy = 0.3
5. Modelling → Create → Keypoints → In Active C.S

Nodes	X	Y	Z
1	0	0	0
2	1.8	3.118	0
3	3.6	0	0
4	5.4	3.118	0
5	7.2	0	0
6	9	3.118	0
7	10.8	0	0

6. Modelling → create → Lines → straight line → Pick the KPs.
7. Meshing → Mesh tool → Mesh → Select Lines → OK
8. Solution → Analysis type → New analysis → Modal → Analysis option
   No. of modes to extract = 10
   No. of modes to expand = 10
9. Solution → Define Loads → Apply → Structural → Displacement → on key point → pick left end and right end keypoint → ALL DOF UY
10. Solution → Solve → Current L.S

D D M M Y Y Y Y

11. General Post-Processor → Result summary → read result
→ First set
→ Next set.

12. Plot results → contour plot → Nodal solution → Notedocon

13. Read result → Next set → Plot result → contour plot → nodal solution.

Result Summary.

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

11/8/21