Name of Student: Aniket Patil	Class: TE MECH 2
Semester/ Year: 6^{th} / 3^{rd}	Roll No: 29
Date of performance:	Date of Submission
Examined by: Prof. B.R Pujari	Expt No: 1

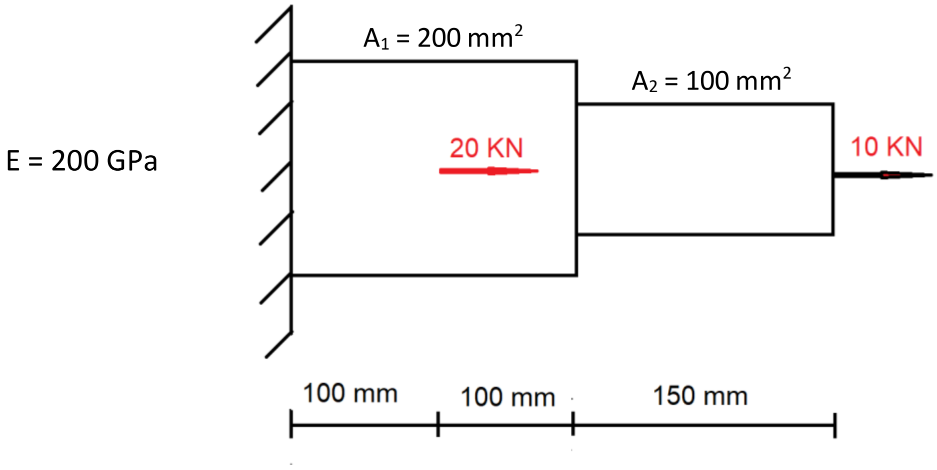
Experiment 1: 1D Bar Element – Structural Linear Analysis

Aim: To do the Structural Linear Analysis of a 1D Bar Element

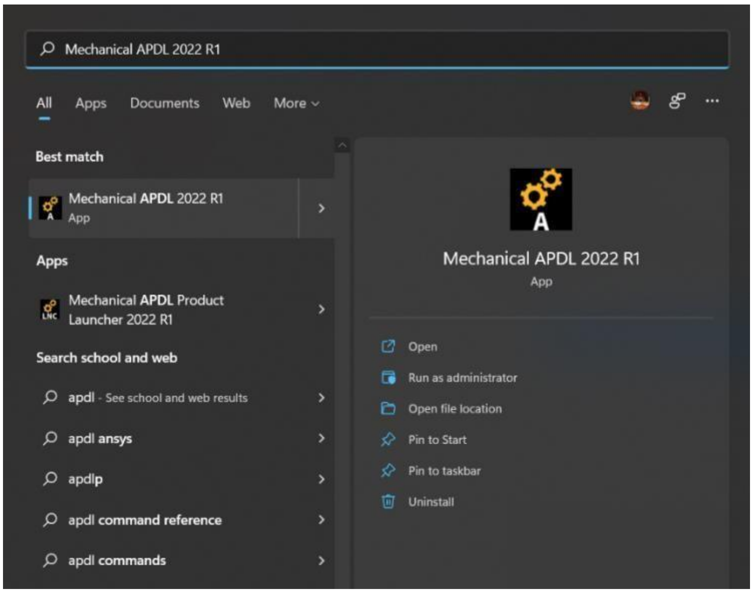
Objective: Run the simulation Using Ansys 2022 R1 and compare the analysisand analytical results to compute the error.

Package: Ansys 2022 R1

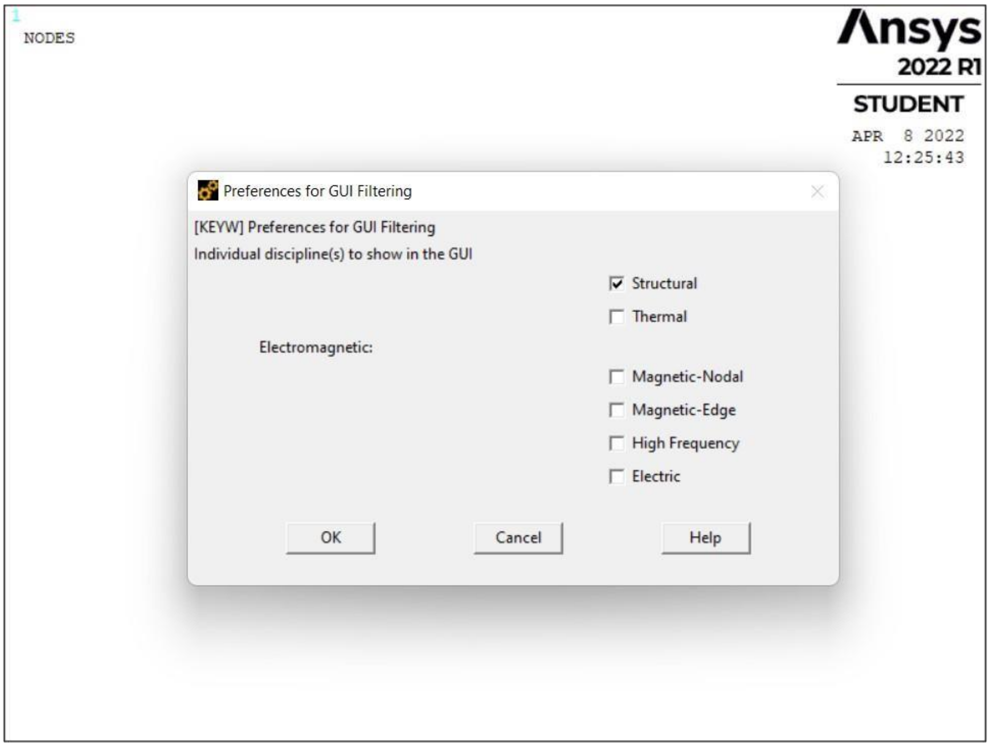
Problem:



Step 1: Run Ansys Mechanical APDL 2022 R1

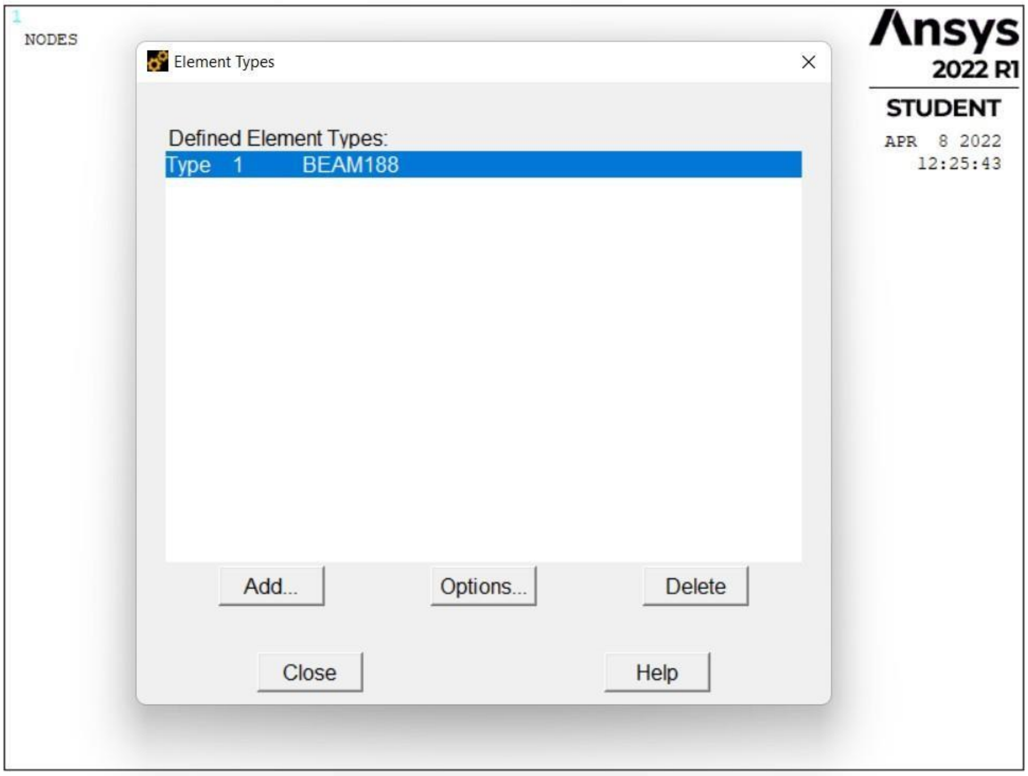


Step 2: Selecting Preference, Preferences → Structural



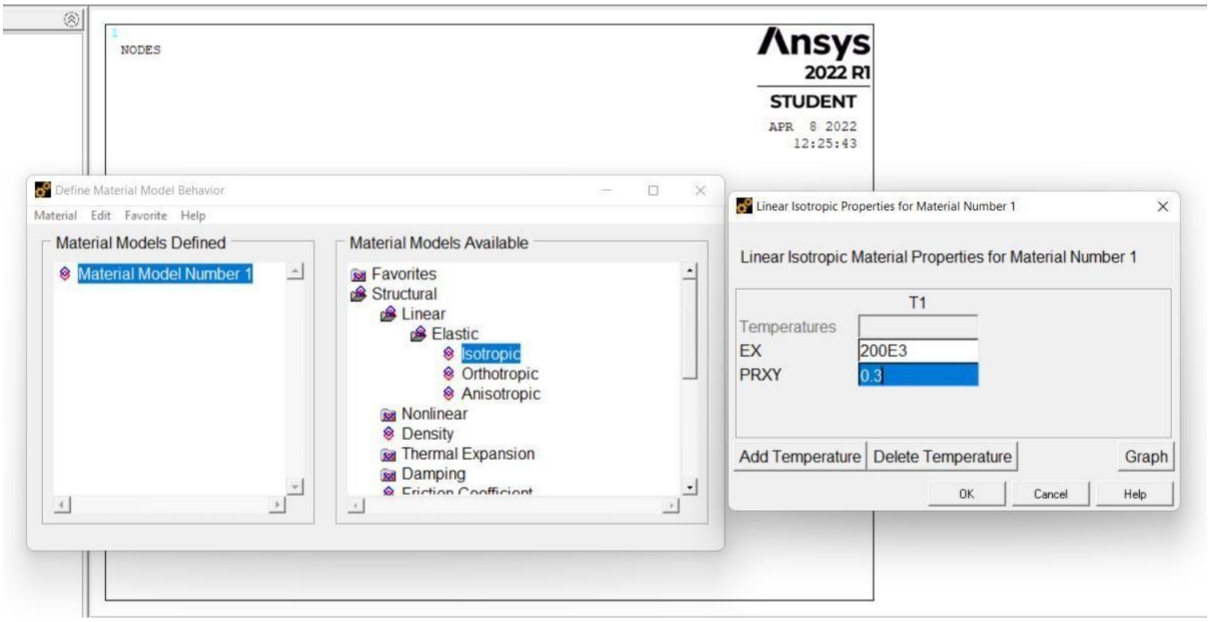
Step 3: Defining the Element Type

Pre-processor → Element Type → Add/Delete Element → Add → Beam → 2Node 188 → OK



Step 4: Defining Material Properties

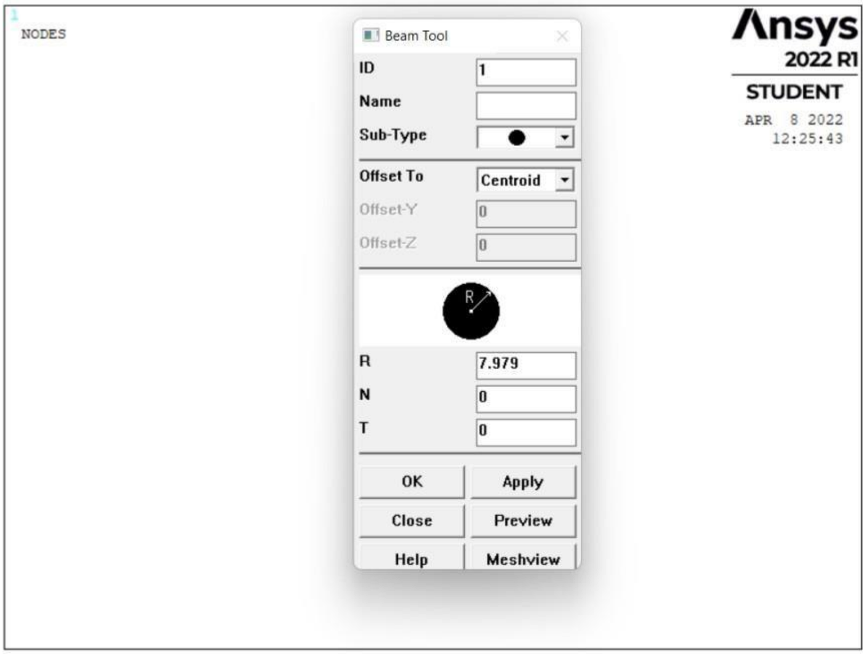
Material Props → Material Models → Material Models Available → Structural → Linear → Elastic → Isotropic



The Material Model is successfully defined. As in our problem the material Properties are same throughout, we will not define another Material Model.

Step 5: Defining Section Of our Beam

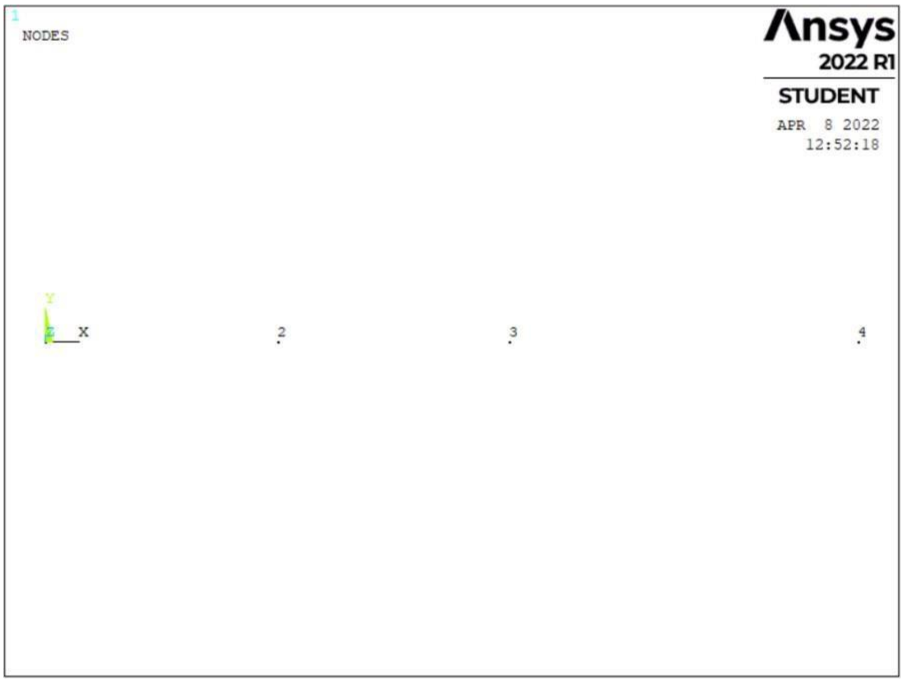
Section → Beam → Common Sections →Beam Tool → Sub type → R



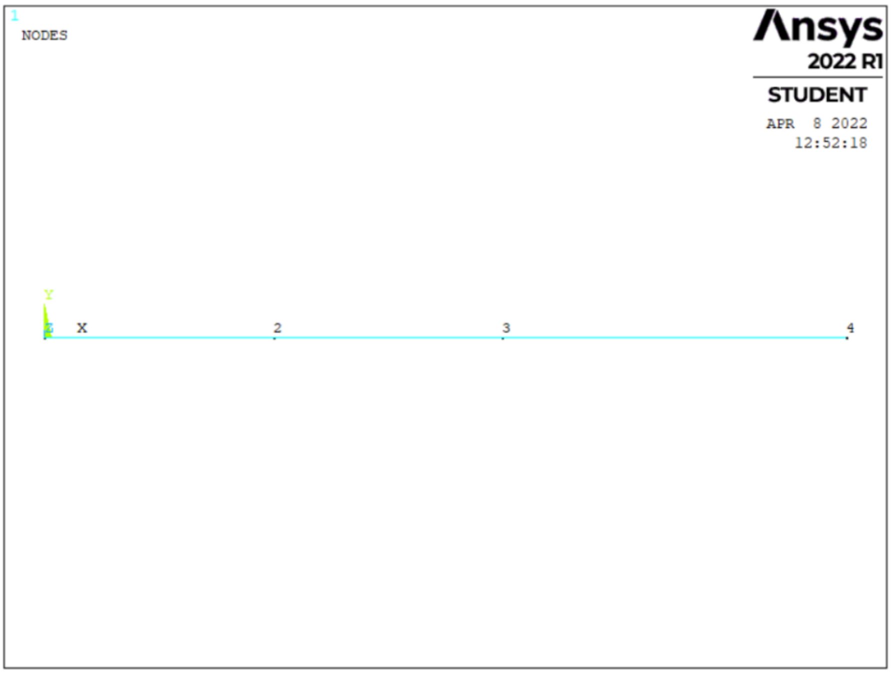
As we have a stepped beam, repeat step 2 times while entering the respectiveRadius Values

Step 6: Modelling the problem

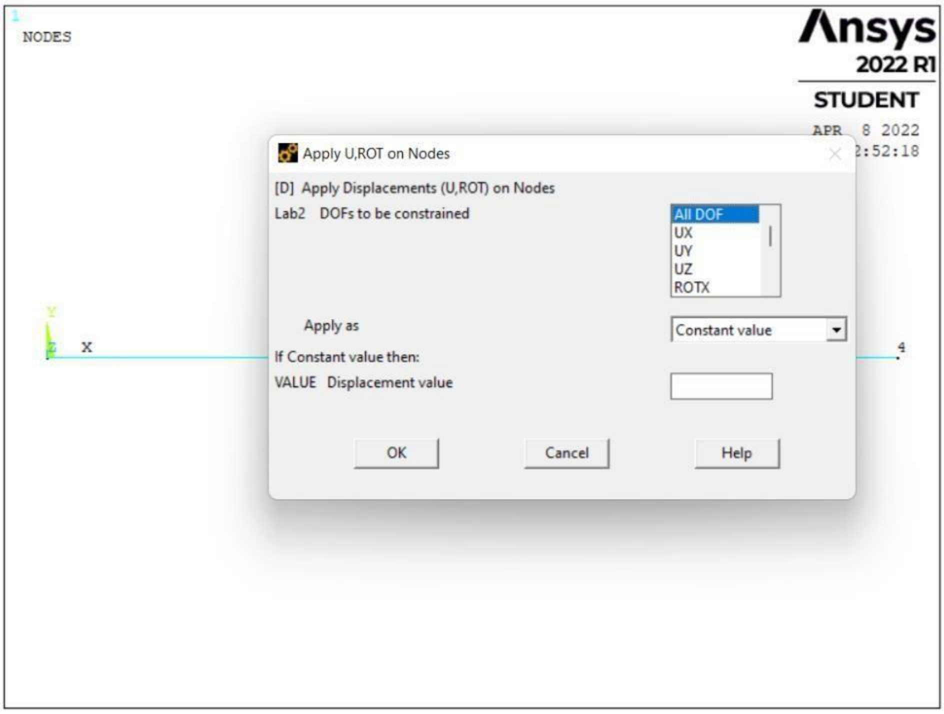
Modeling → Create → Nodes → In Active CS → Create the nodes using the Xvalues as (0,100,200,350)



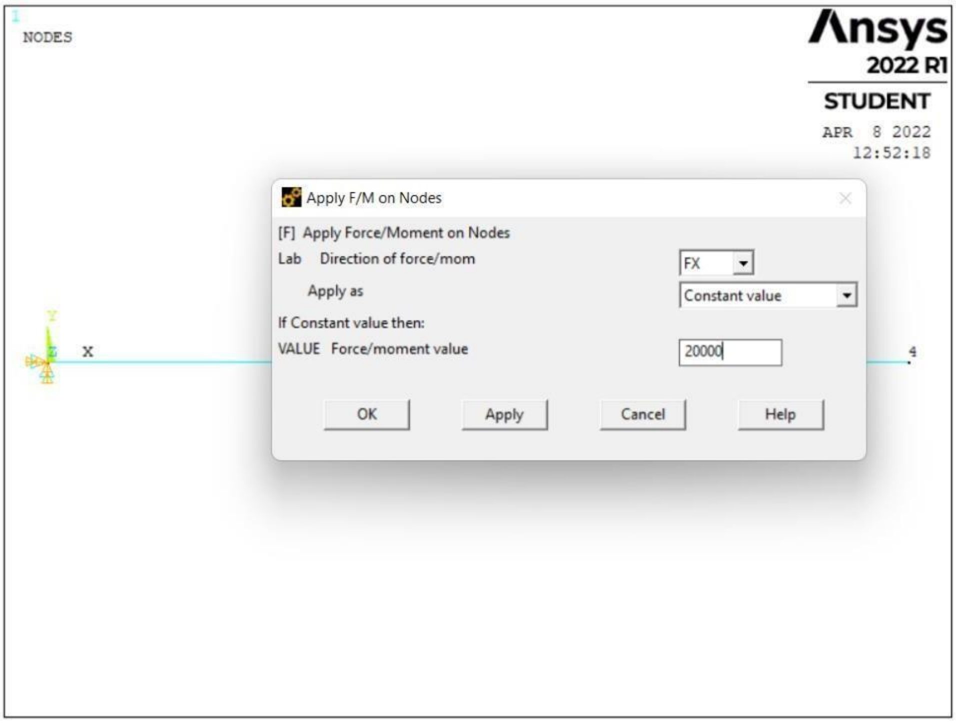
Step 7: Creating Element | Modeling → Create → Element → Element Attributes → Section 1 → OK → Auto Numbered → Thru Nodes → OK → Select Node 1,2. → OK → Repeat and select Node 2 and 3 → OK | As the problem involves a stepped beam and we have defined 2 sections, in the element attr. Select Section Number 2 and repeat the procedure for Nodes 3and 4 to give you the Following result.



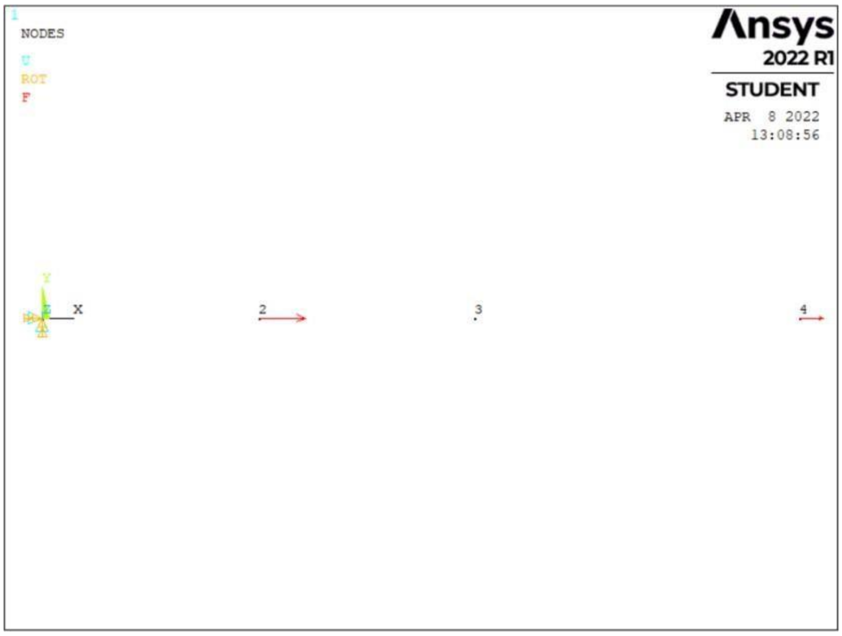
Step 8: Defining Constraints and Forces on the beam | Loads → Analysis type → New Analysis → Static → OK | Define Loads → Apply → Structural → Displacement → On Node → Select node 1 and give ALL DOF.



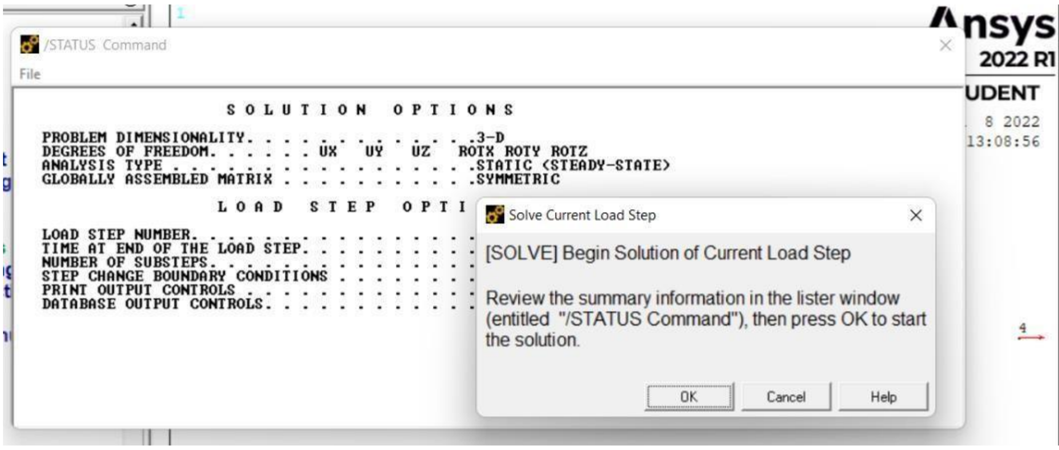
Force/moment → On Nodes → Select node 2 → 20000N | Similarly for node 4 Repeat this step and add 10000N.

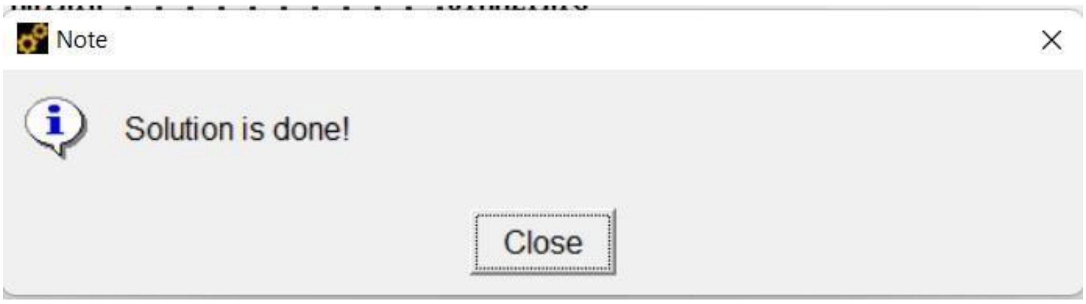


Result would be somewhat like this after you Right Click → Fit

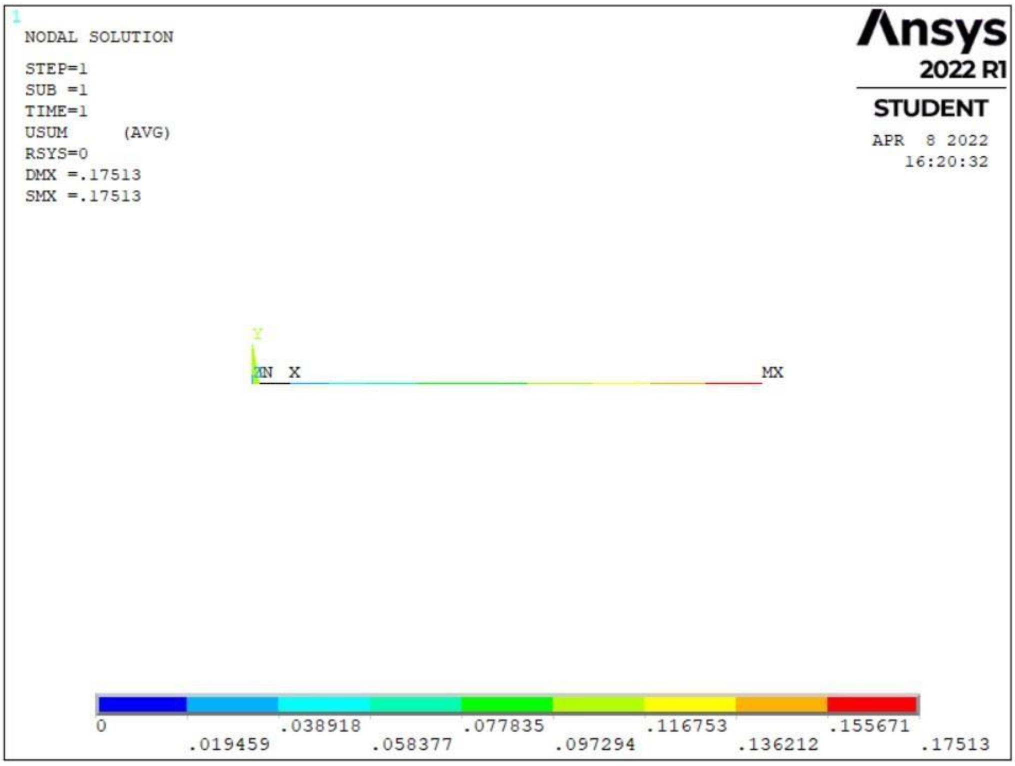


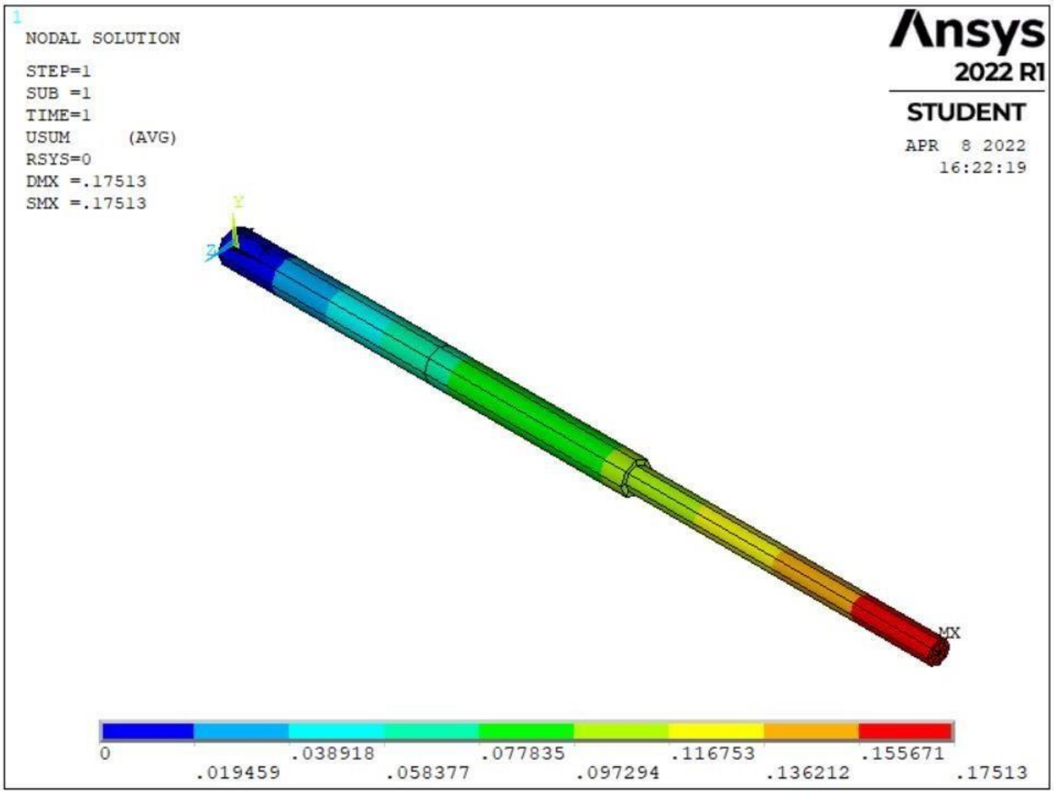
Step 9: Solution of our Problem | Solution → Solve → Current LS → OK



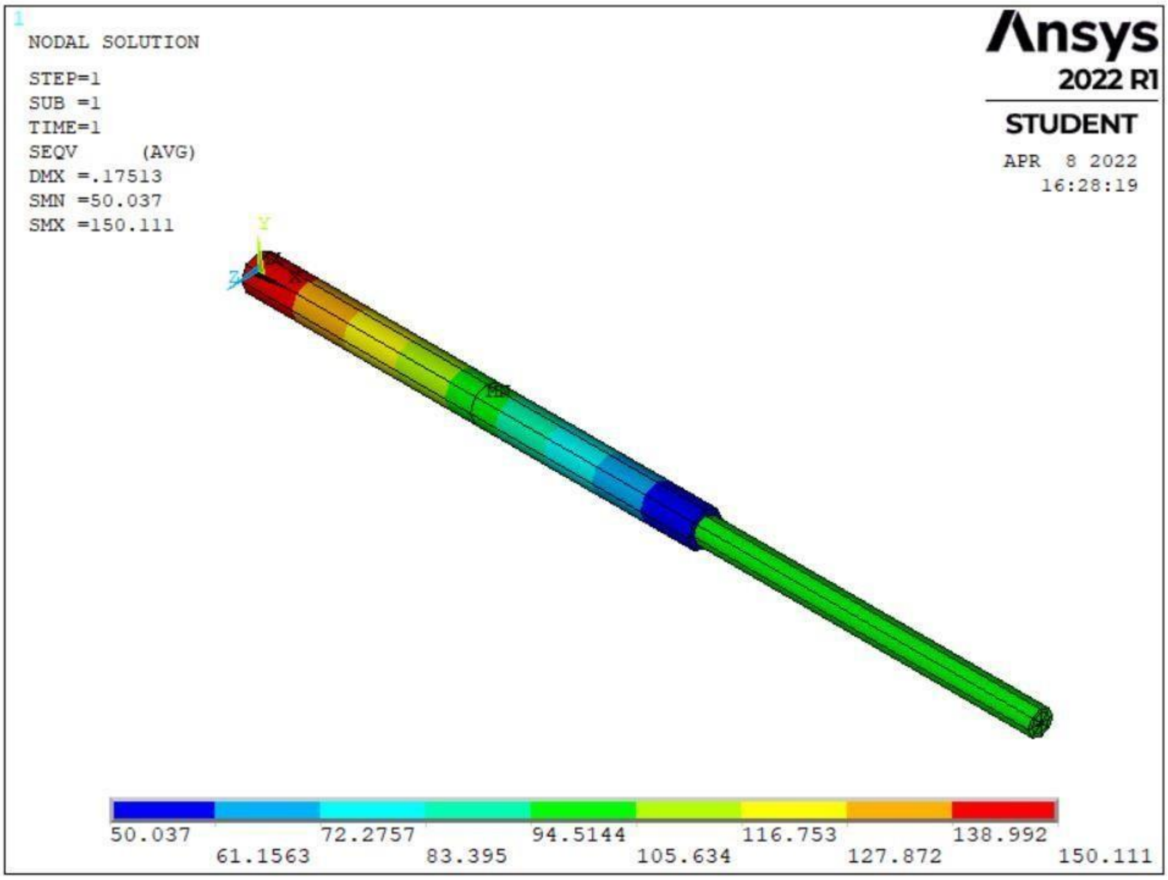


Step 10: Displaying the results with the post processor | General PostPro →Plot Results →Contour plot → Nodal Solution → DOF Solution → **Displacement Vector Sum**





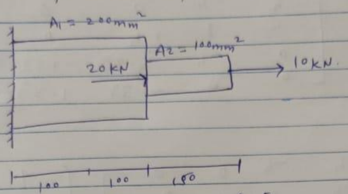
Step 11: For Stress Intensity go to General PostPro →Plot Results →Contour plot → Nodal Solution → Stress → Von Mises Stress → Apply → OK



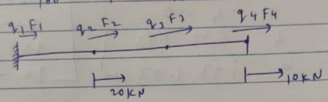
Aniket Patil

(1) Exp. no. (1)
Topic :

Q. Analyse the problems completely given below, Assume $E = 200\ GPa$.



Soln :-



Connectivity Table.

Element	Node	
1	1	2
2	2	3
3	3	4

$$k_1 = \frac{AE}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

$$= \frac{200 \times 2 \times 10^5}{100} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

$$k_1 = 10^5 \begin{bmatrix} 4 & -4 \\ -4 & 4 \end{bmatrix}$$

$$k_2 = \frac{200 \times 200 \times 10^3}{100} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

$$k_2 = 10^5 \begin{bmatrix} 4 & -4 \\ -4 & 4 \end{bmatrix}$$

$$k_3 = \frac{200 \times 10^3 \times 100}{150} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

$$k_3 = 10^5 \begin{bmatrix} 1.33 & -1.33 \\ -1.33 & 1.33 \end{bmatrix}$$

Reactions!

Page No. :
Date. : / /

Topic :

Global matrix.

$$k = 10^5 \begin{bmatrix} 4 & -4 & 0 & 0 \\ -4 & 8 & -4 & 0 \\ 0 & -4 & 5.33 & -1.33 \\ 0 & 0 & -1.33 & 1.33 \end{bmatrix}$$

$kq = F$.

$$10^5 \begin{bmatrix} 4 & -4 & 0 & 0 \\ -4 & 8 & -4 & 0 \\ 0 & 4 & 5.33 & -1.33 \\ 0 & 0 & -1.33 & 1.33 \end{bmatrix} \begin{bmatrix} q_1 \\ q_2 \\ q_3 \\ q_4 \end{bmatrix} = \begin{bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \end{bmatrix}$$

$$10^5 \begin{bmatrix} 8 & 4 & 0 \\ -4 & 5.33 & -1.33 \\ 0 & -1.33 & 1.33 \end{bmatrix} \begin{bmatrix} q_2 \\ q_3 \\ q_4 \end{bmatrix} = \begin{bmatrix} 20\times10^4 \\ 0 \\ 1\times10^4 \end{bmatrix}$$

$q_2 = 0.075mm$ $\quad q_3 = 0.1mm$.

$$\sigma_1 = \frac{200\times10^3}{100} [-1, 1] \begin{bmatrix} 0 \\ 0.075 \end{bmatrix}$$

$\sigma_1 = 150\ N/mm^2$.

$$\sigma_2 = \frac{200\times10^3}{100} [-1 \ 1] \begin{bmatrix} 0.075 \\ 0.1 \end{bmatrix}$$

$\sigma_2 = 50\ N/mm^2$.

$$\sigma_3 = \frac{200\times10^3}{150} [-1 \ 1] \begin{bmatrix} 0.1 \\ 0.175 \end{bmatrix}$$

$\sigma_3 = 100\ N/mm^2$.

Topic :

Reactions :

$R_1 = kq = F_1$

$$= 10^5 \begin{bmatrix} 4 & -4 & 0 & 0 \end{bmatrix} \begin{bmatrix} 0 \\ 0.075 \\ 0.1 \\ 0.175 \end{bmatrix} - 0$$

$R_1 = -30N$.

Error calculation

stress reading

By ansys solution :-

$= 150\ N/mm^2$

By Theory

$= 150\ N/mm^2$

Deformation / Deflection :-

From ansys $= 0.175$

From Theory $= 0.175$

error obtained from simulation and analytical is 0%.