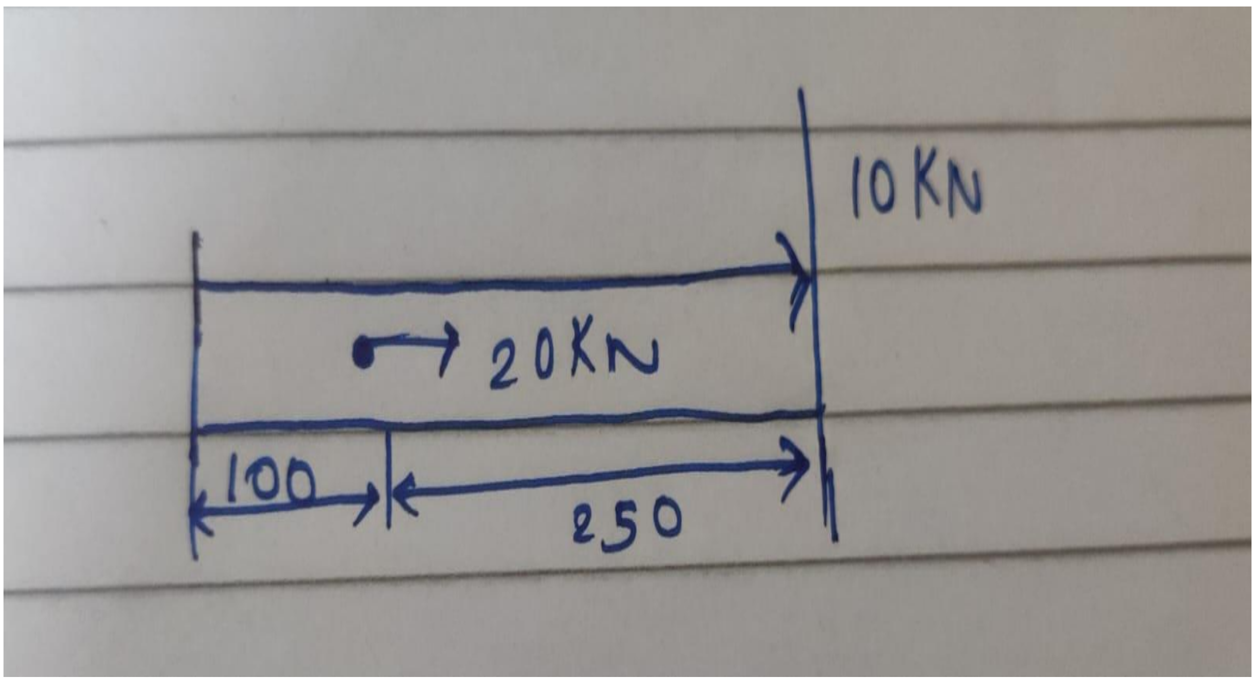
Name of Student: Aniket Patil	**Class: TE MECH 2**
Semester/ Year: 6^{th} / 3^{rd}	**Roll No: 29**
Date of performance:	**Date of Submission**
Examined by: Prof. B.R Pujari	**Expt No: 4**

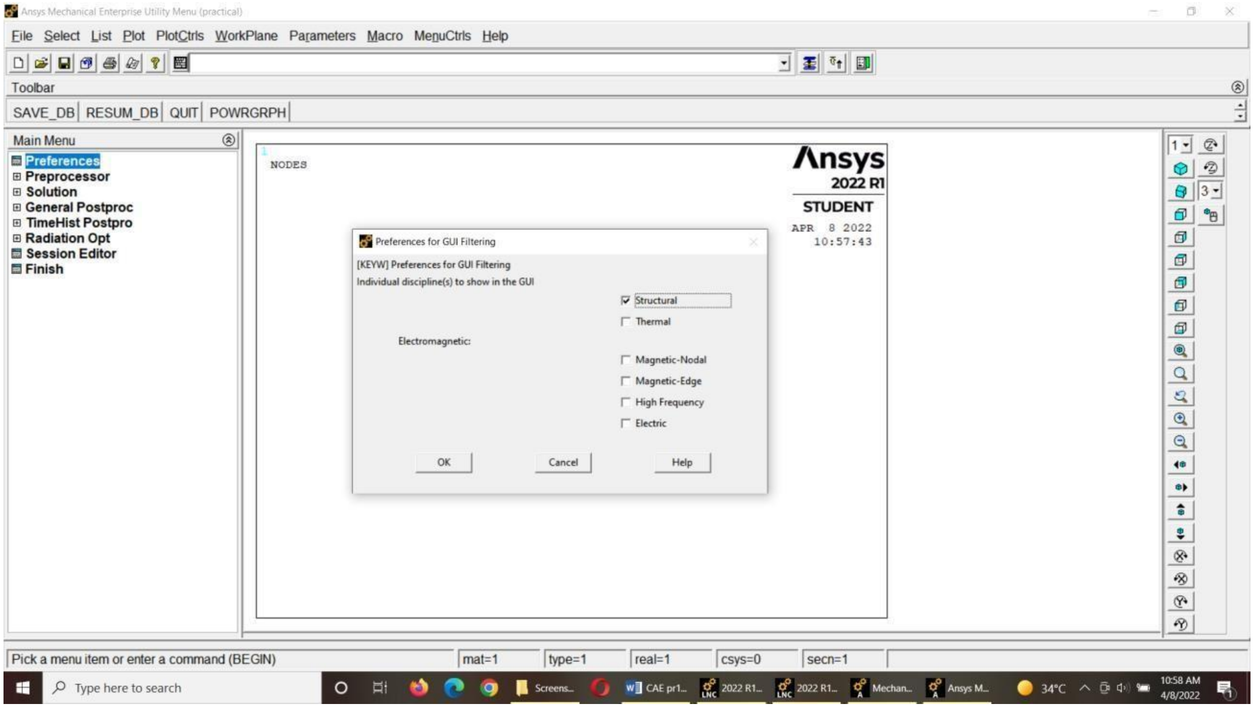
AIM OF EXPERIMENT:- Stress and deflection analysis of beam using finite element package. Finite Element Package: ANSYS 2022

Stress distribution in a beam with applied load.

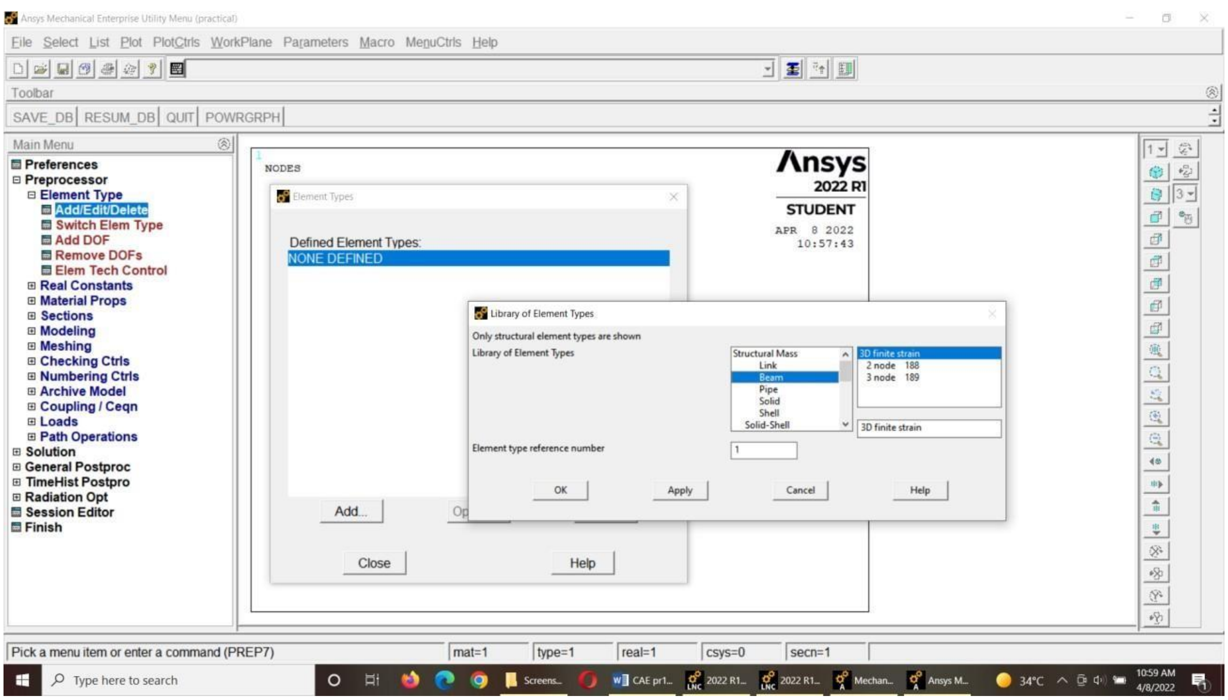
E= 2x10e5 Mpa R= 25mm **μ**= 0.3



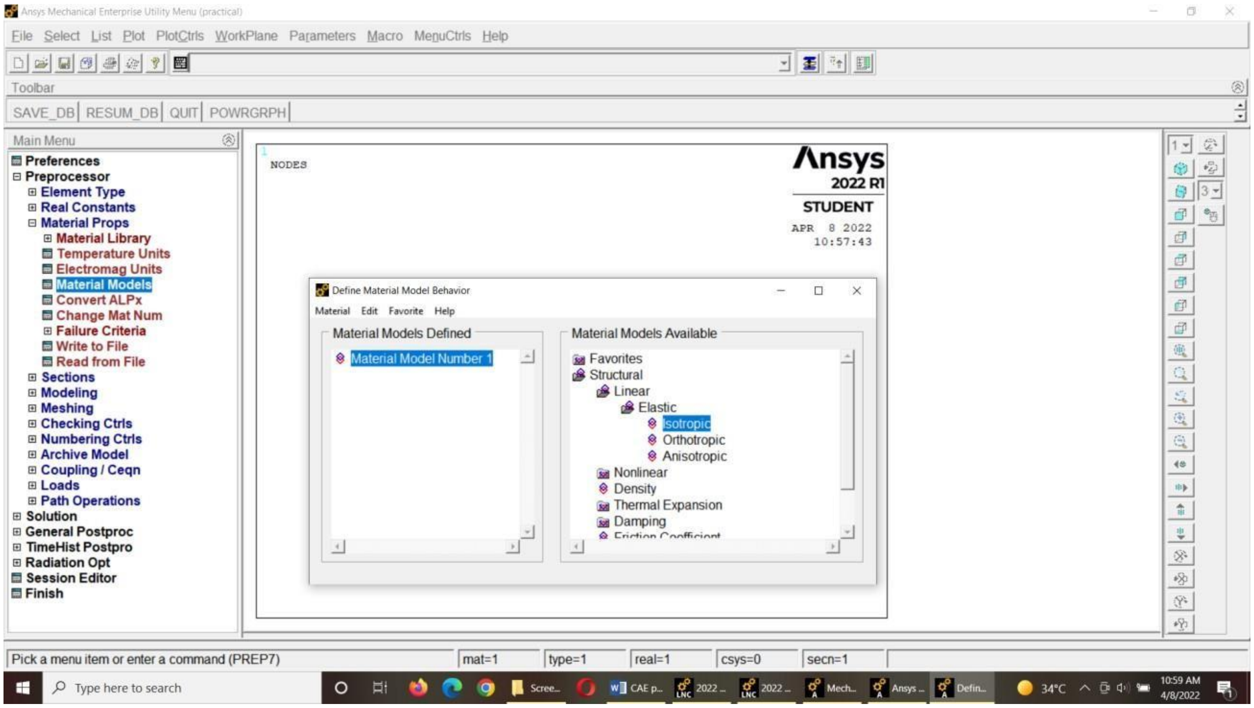
Step 1: Select type of Analysis ----------- Preferences> structural>Press Ok



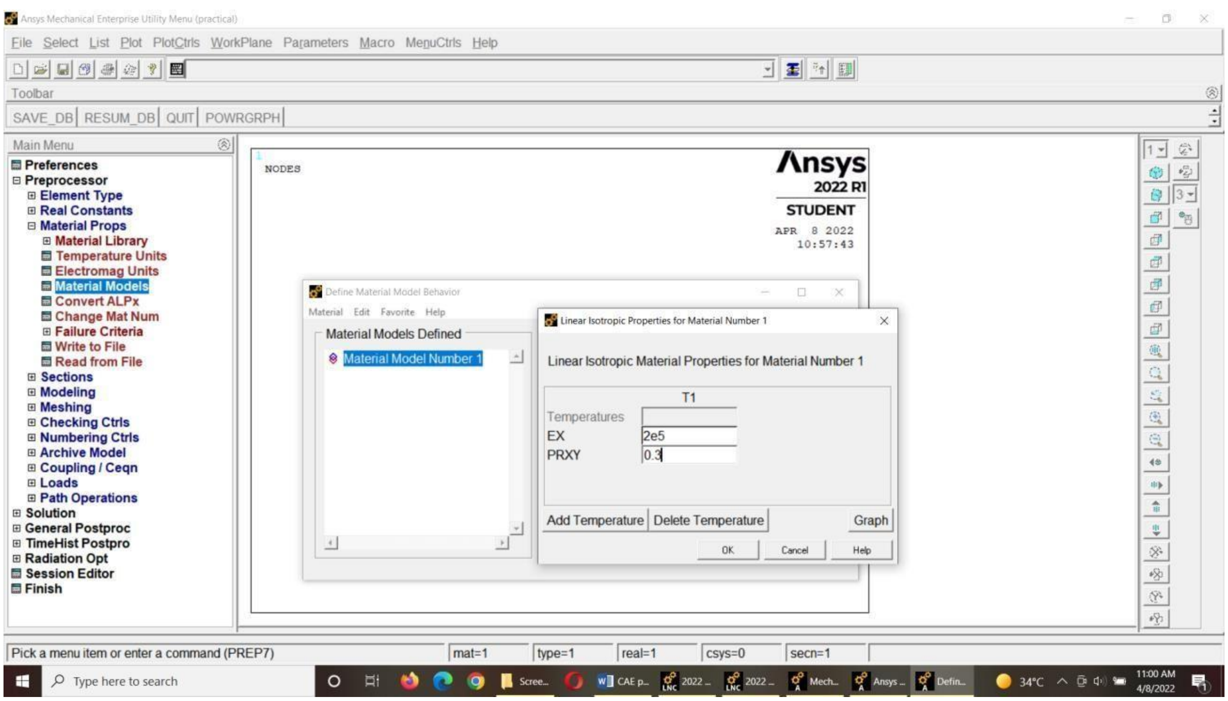
Step 2: Add the element type……preprocessor>element type>beam>2node 188> Press ok



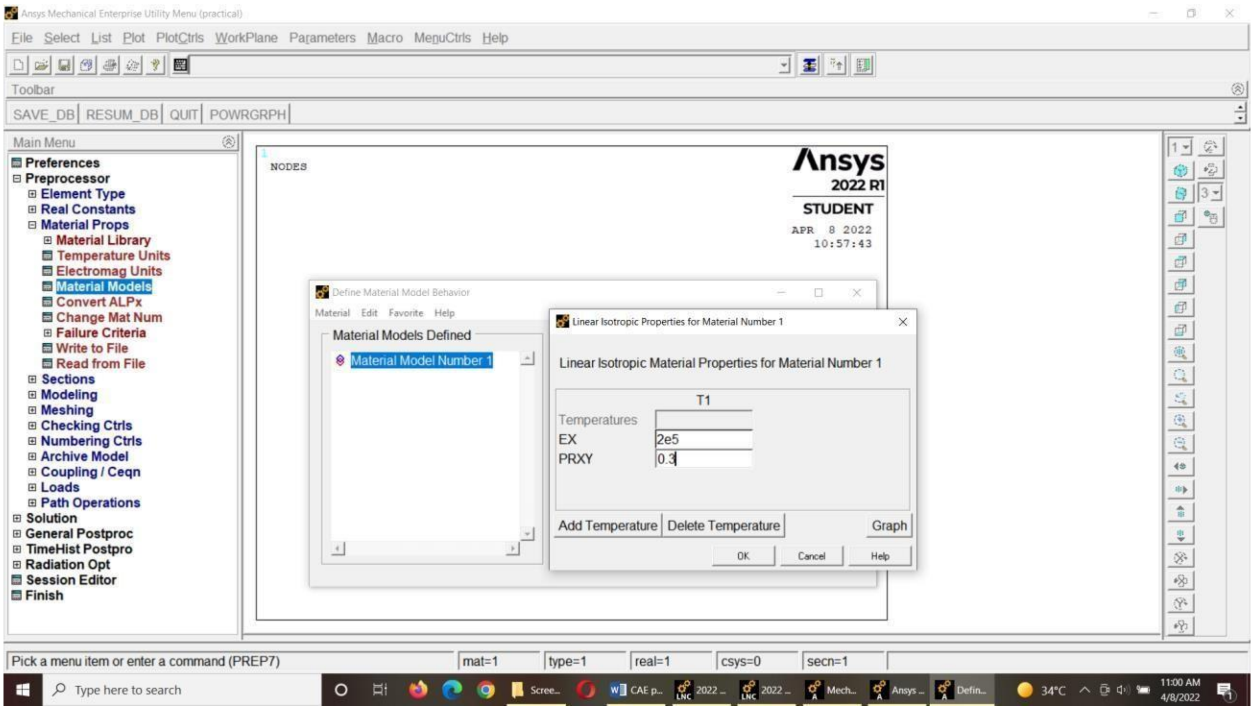
Step 3: Preprocessor>Material prop.>Material models>Material number1



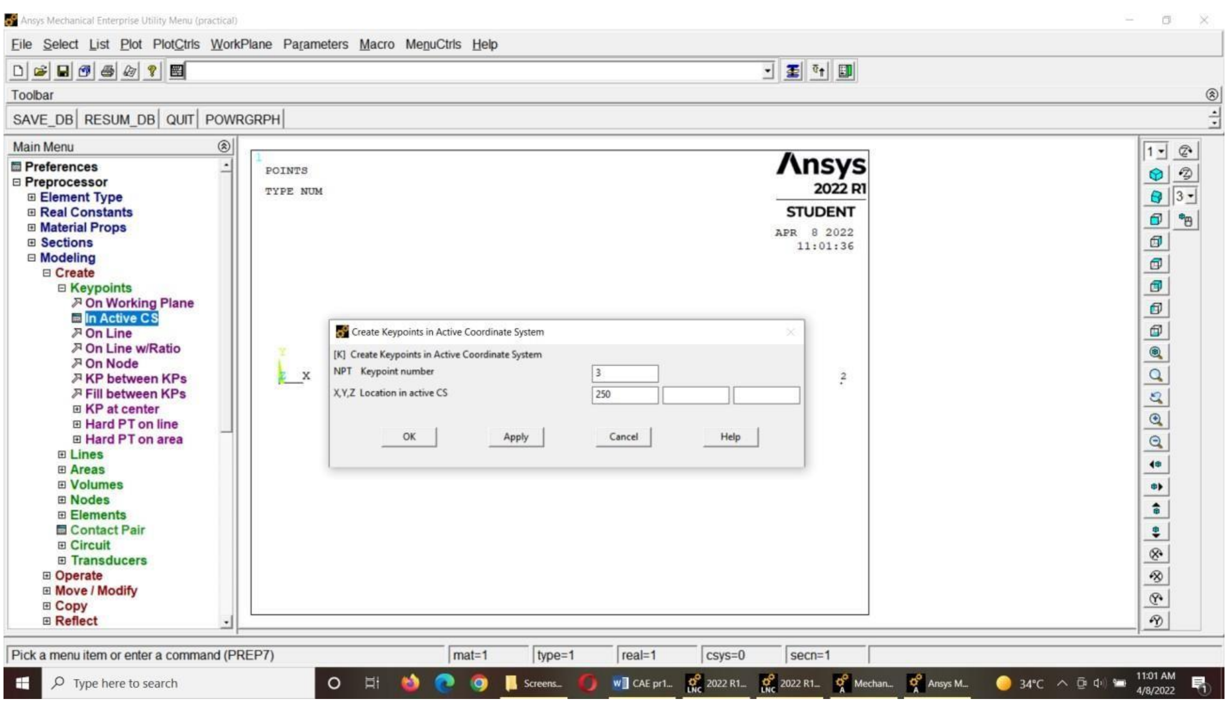
Step 4: Material models>structural>linear>elastic>isotropic.



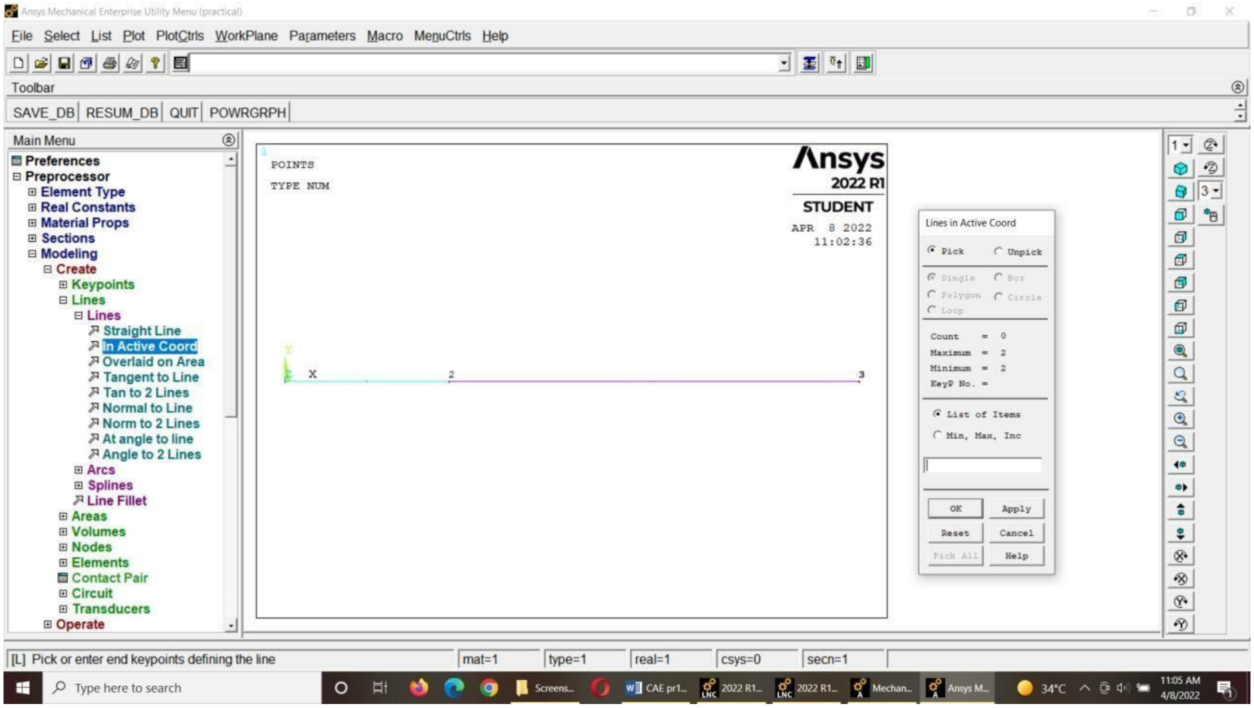
Step 5: selecting section of beam….section>beam>common section>select section.



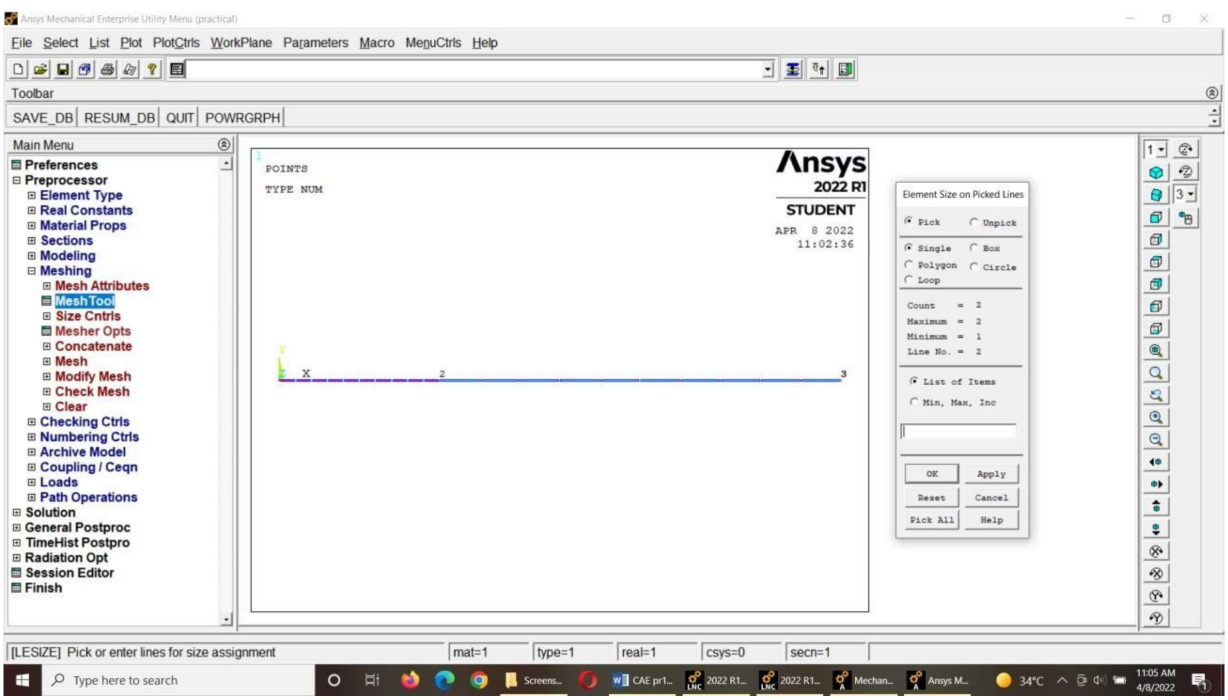
Step6: Creating Keypoints:- modeling>create>keypoint>in active cs>select co- ordinate.



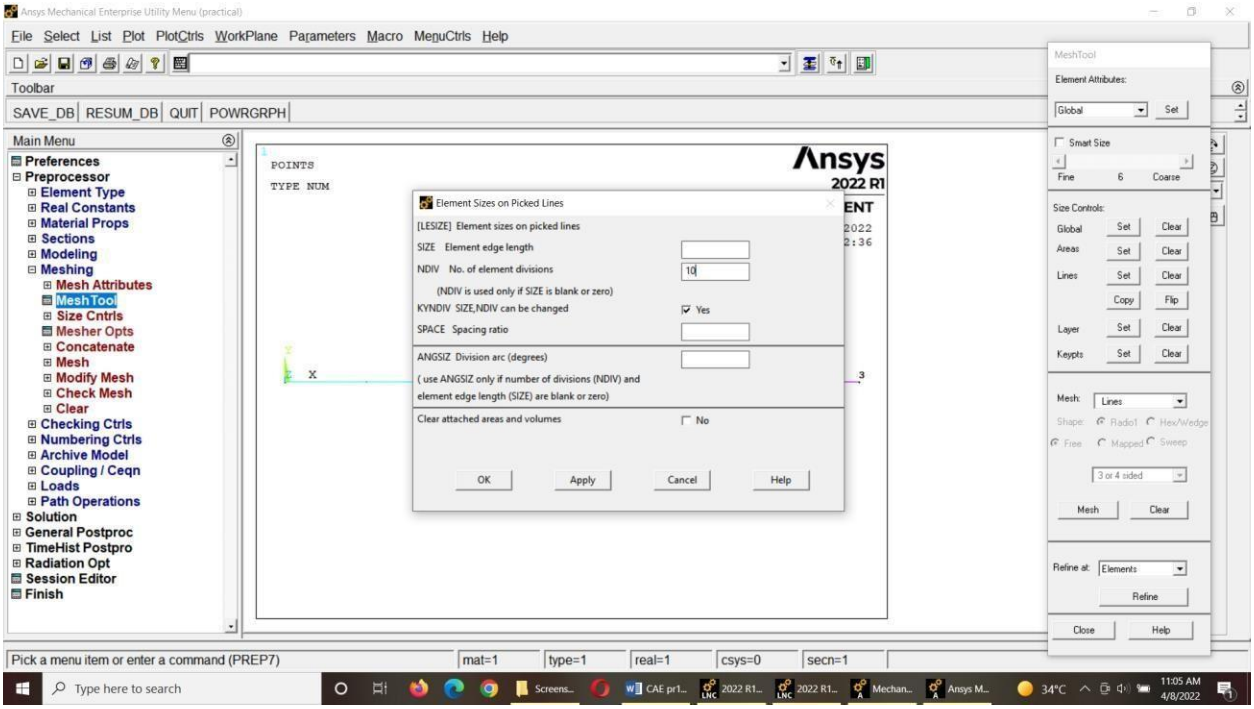
Step7: Lines>Lines>In active co-ordinate cs> Join Co Ordinates



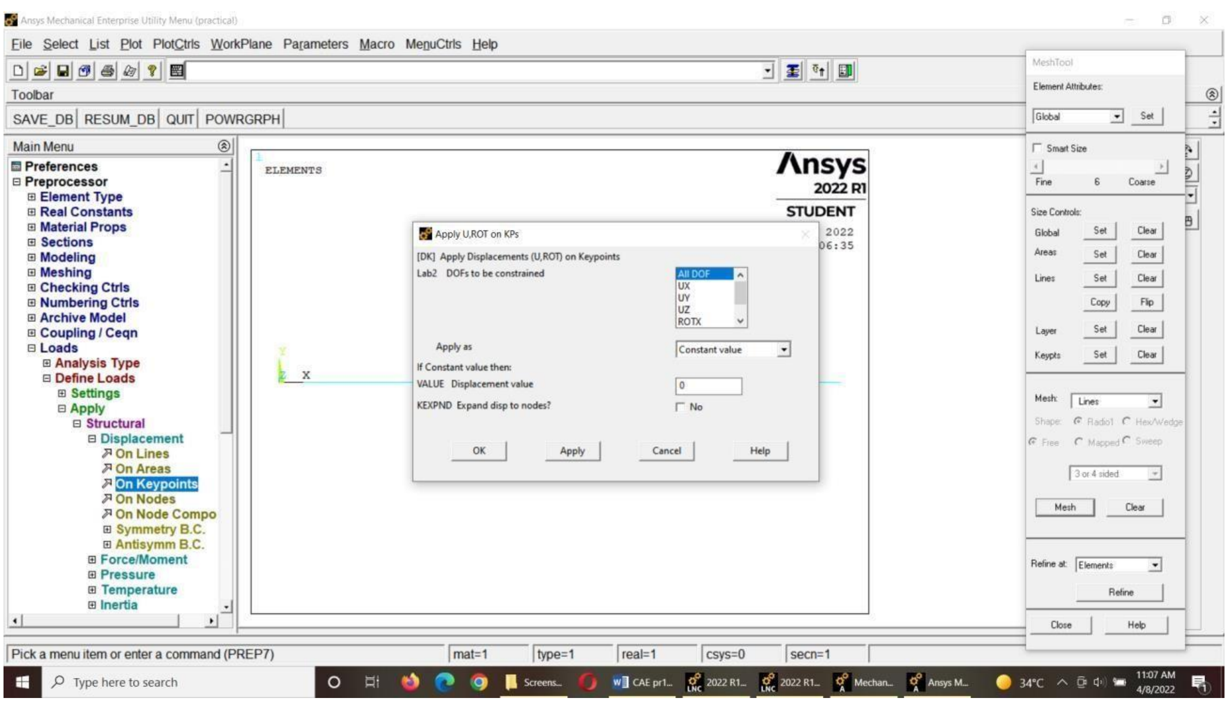
Step8: meshing:- Meshing> meshtools>lines>Set>Select Model>Apply



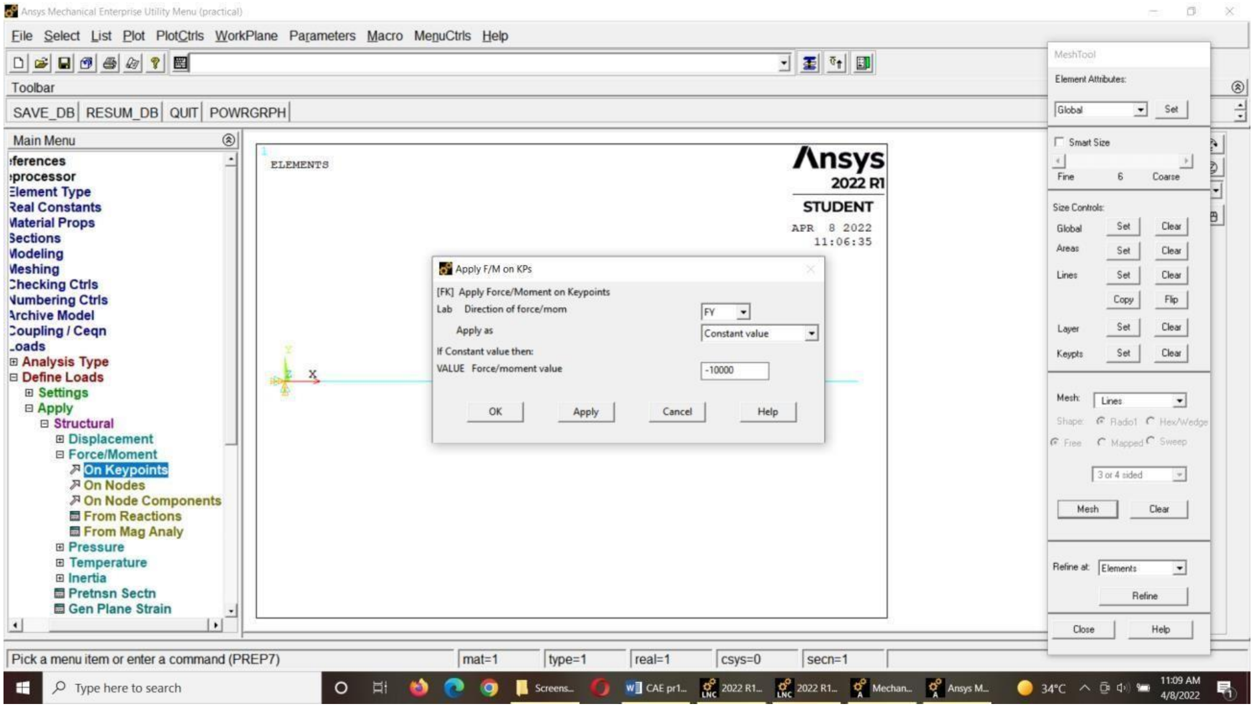
Step9: Enter no division>ok>mesh>selectmodel>ok



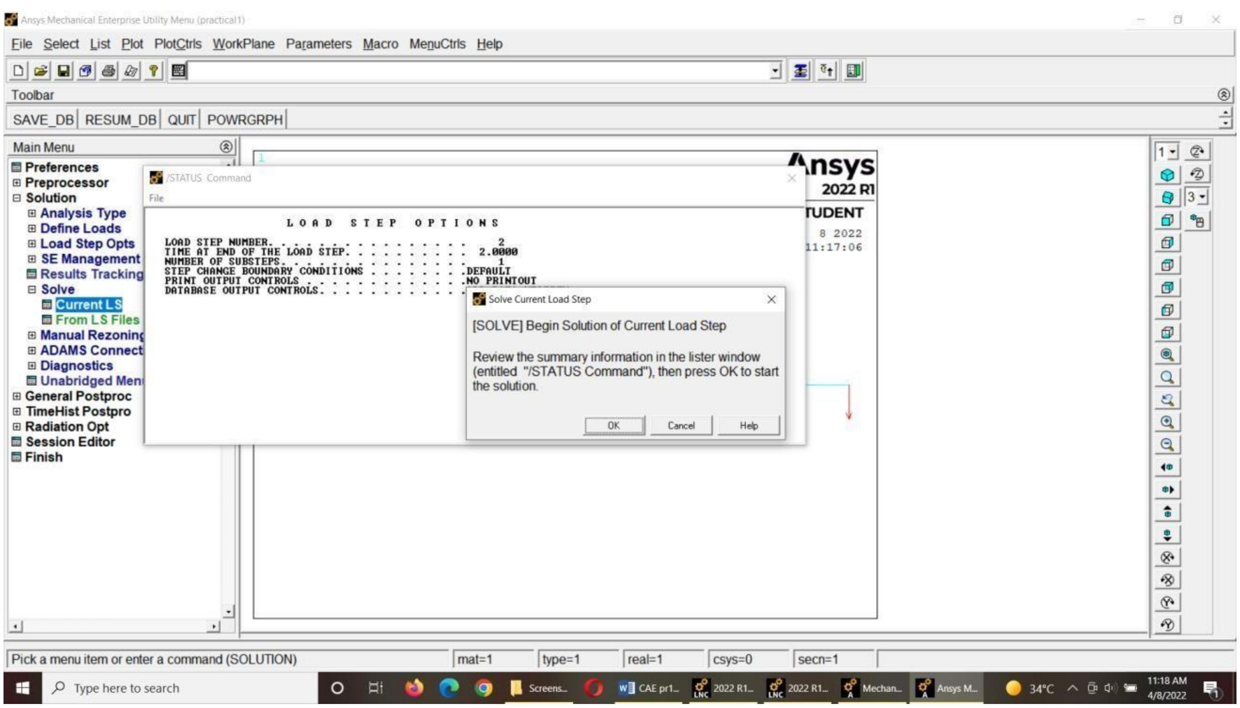
Step10: Apply loads: Laods>define loads>apply>structural>displacement>on keypoints> All Dof>0>ok



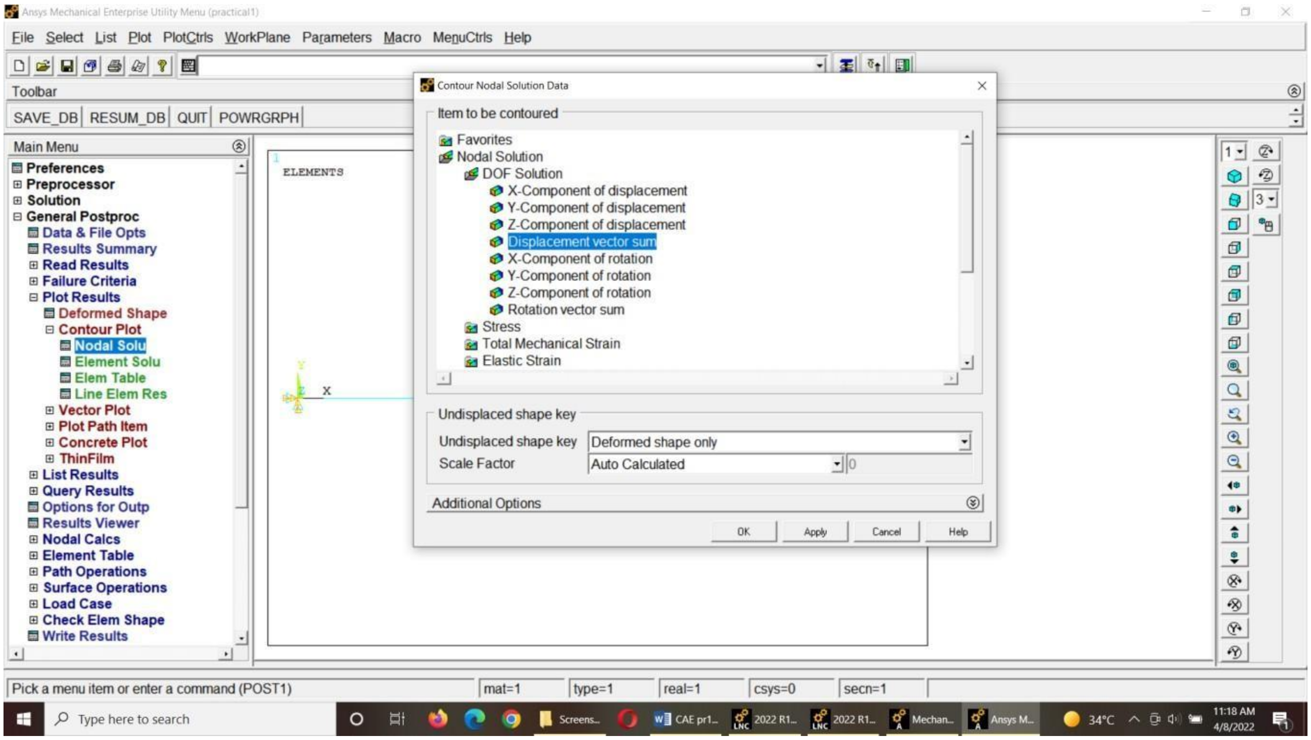
Step11: Loads>Define loads>apply>forces>on keypoints> selecting direction of forces (here FY)>ok



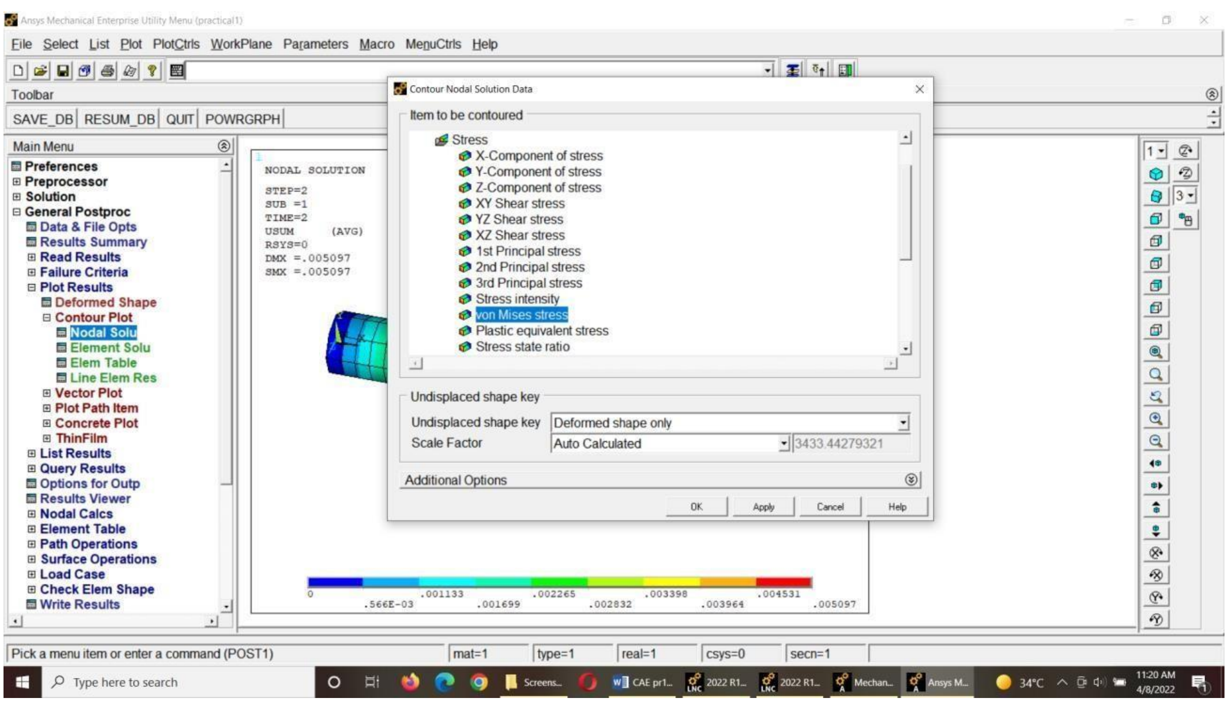
Step12:Solution:- solution>solve>currentls> done



Step13: General postproc> plot result> Nodal solution> Dof >Vector sum displacement> apply.

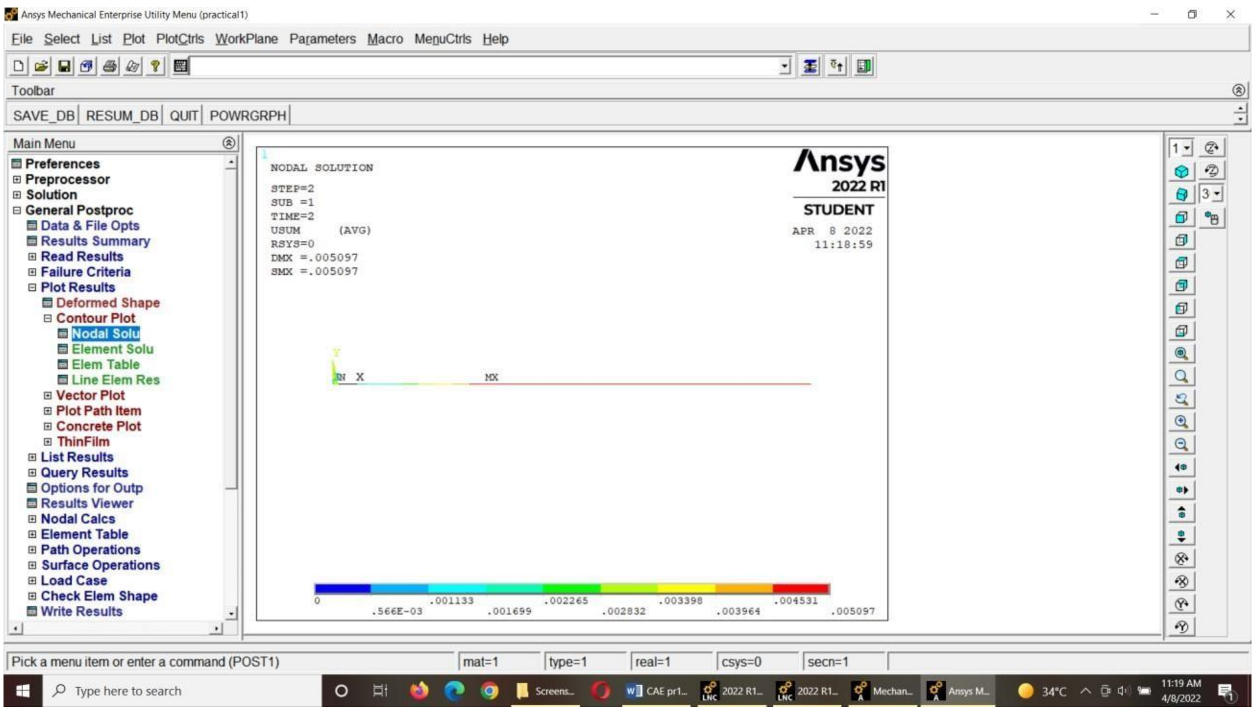


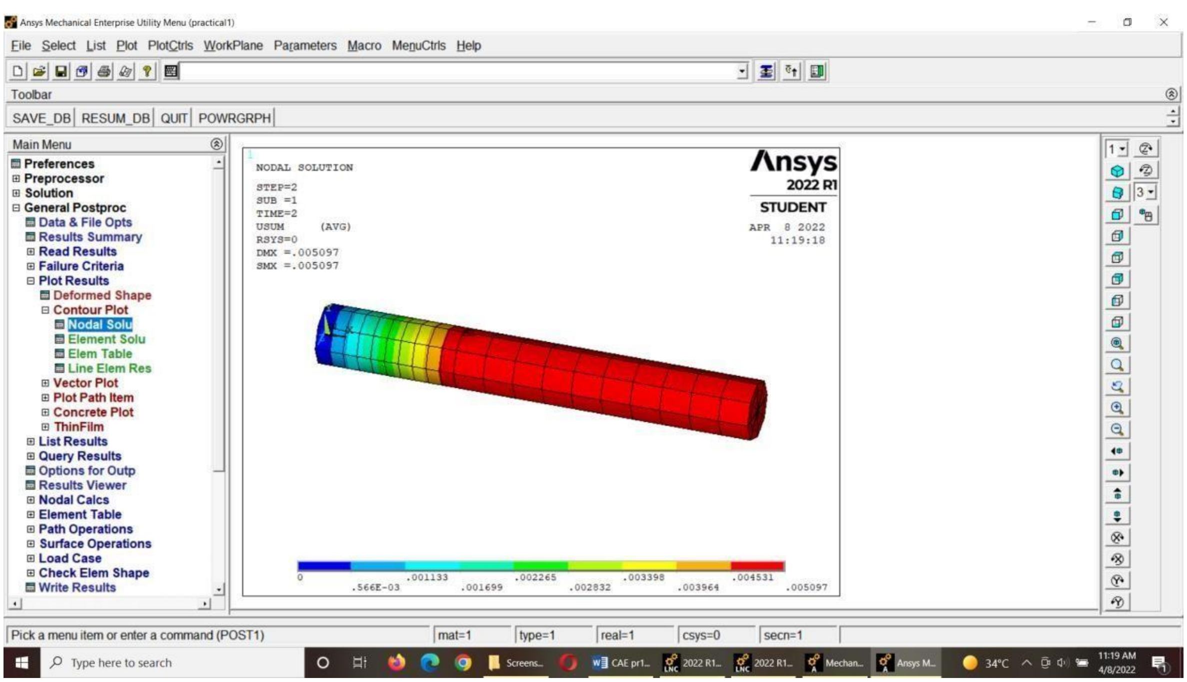
Step14: Nodal solution> stress> von misses stress> apply



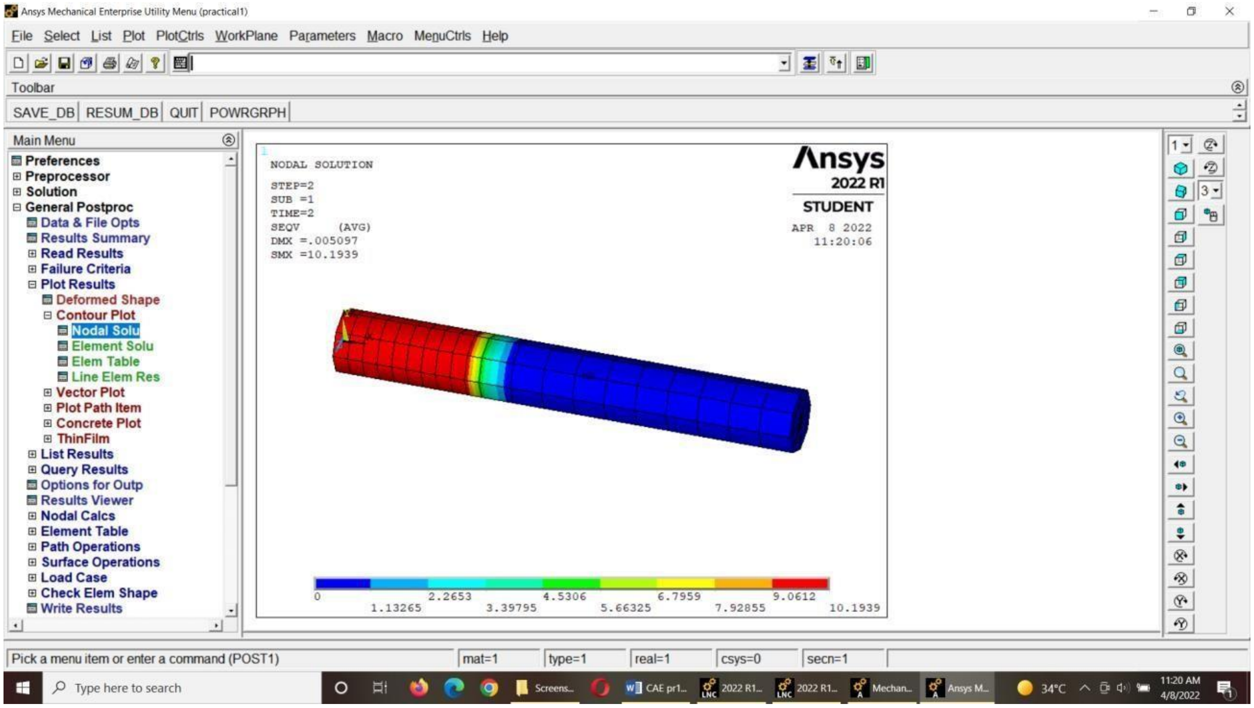
RESULTS:-

NODAL DISPLACEMENT:-





STRESSES:-

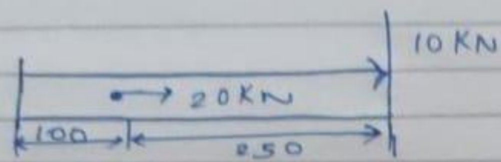


SO HERE BY ANALYSIS WE HAVE GOT MAX. INDUCED STRESS IS 10.913 N/MM2 MAX. DISPLACEMENT IS 0.005 MM

By analytical Solution:

Name: Aniket Patil
Roll no: 29
Exp No: 4

1)

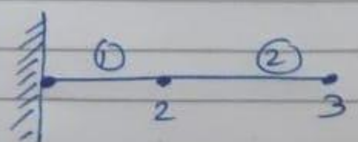


$E = 2 \times 10^5$

$\mu = 0.3$

$R = 25\,mm$

$A = \frac{\pi}{4} \times 25^2 = 490.87\,mm^2$



Element	Node
1	1-2
2	2-3

$$k_1 = \frac{A_1 E_1}{L_1}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

$$= \frac{490.87 \times 2 \times 10^5}{140}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

$$= 98.17 \times 10^4 \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

$$= 10^4 \begin{bmatrix} 98.17 & -98.17 \\ -98.17 & 98.17 \end{bmatrix}$$

$$K_2 = \frac{490.8 \times 2 \times 10^5}{250} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

$$= 39.26 \times 10^4 \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

$$= 10^4 \begin{bmatrix} 39.26 & -39.26 \\ -39.26 & 39.26 \end{bmatrix}$$

$$K = 10^4 \begin{bmatrix} 98.17 & -98.17 & 0 \\ -98.17 & 98.17+39.26 & -39.26 \\ 0 & -39.26 & 39.26 \end{bmatrix}$$

$$K = 10^4 \begin{bmatrix} 98.17 & -98.17 & 0 \\ -98.17 & 137.43 & -39.26 \\ 0 & -39.26 & 39.26 \end{bmatrix} \begin{bmatrix} q_1 \\ q_2 \\ q_3 \end{bmatrix} \begin{bmatrix} 0 \\ 20 \times 10^3 \\ 40 \times 10^3 \end{bmatrix}$$

$q_2 = 0.0101$ mm

$q_3 = -0.0152$ mm

$$\sigma_1 = \frac{E_1}{L_1} \begin{bmatrix} -1 & 1 \end{bmatrix} \begin{bmatrix} q_1 \\ q_2 \end{bmatrix}$$

$$= \frac{2 \times 10^5}{100} \begin{bmatrix} -1 & 1 \end{bmatrix} \begin{bmatrix} 0 \\ 0.0101 \end{bmatrix}$$

$\sigma_1 = 20.2\ N/mm^2$

$$\sigma_2 = \frac{2 \times 10^5}{250} \begin{bmatrix} -1 & 1 \end{bmatrix} \begin{bmatrix} 0.0101 \\ -0.0152 \end{bmatrix}$$

$$\boxed{\sigma_2 = 4.08\ N/mm^2}$$

CONCLUSION:-

Thus by comparing analytical and software solution we have got Max. stresses:-

By ansys solution:-10.139 N/mm2 By

analytical solution:- 20.2 N/mm2

Max. displacement:-

By ansys solution:- 0.00509 mm By

analytical solution:- 0.0101 mm

Thus we have got 50% error in finding displacement and stresses.