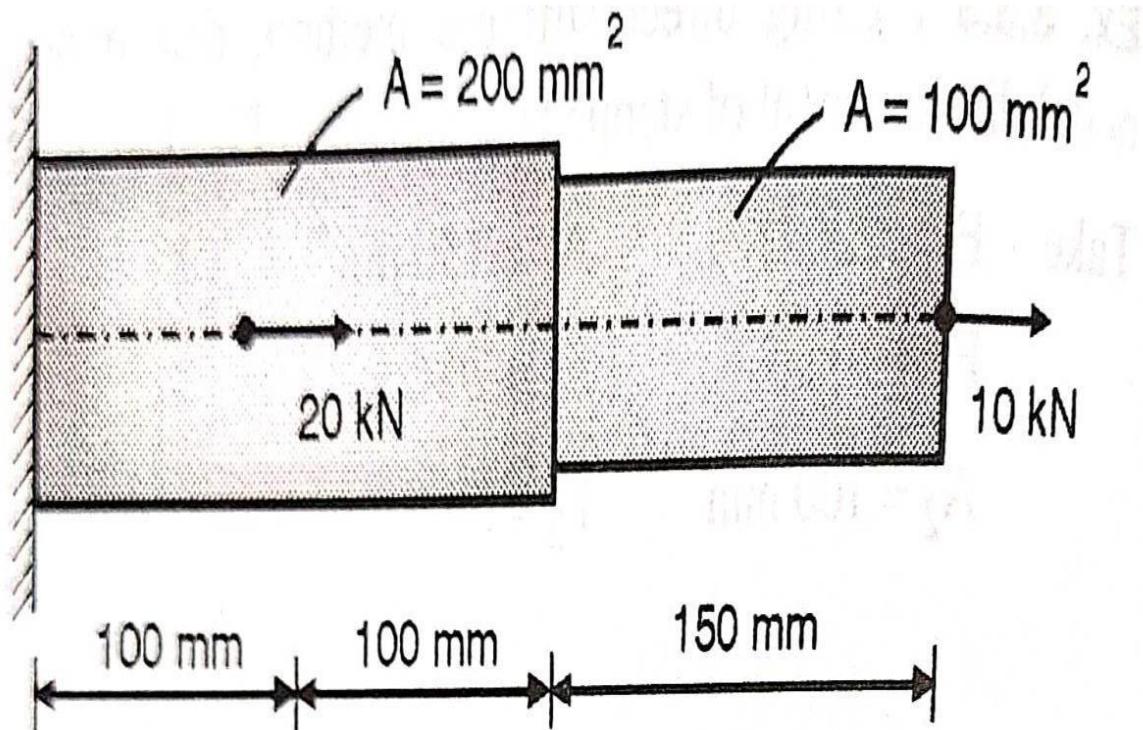


Name of Student:-Sourav Gujale	Class:TE Mech 1
Semester/Year:-6 th SEM/3 rd YEAR	Roll No:-61
Date of Performance:-	Date of Submission:-
Examined By:-Prof.B.R.Pujari	Experiment No:-1

Aim of Experiment:-1D Bar Element-Structual Linear Analysis

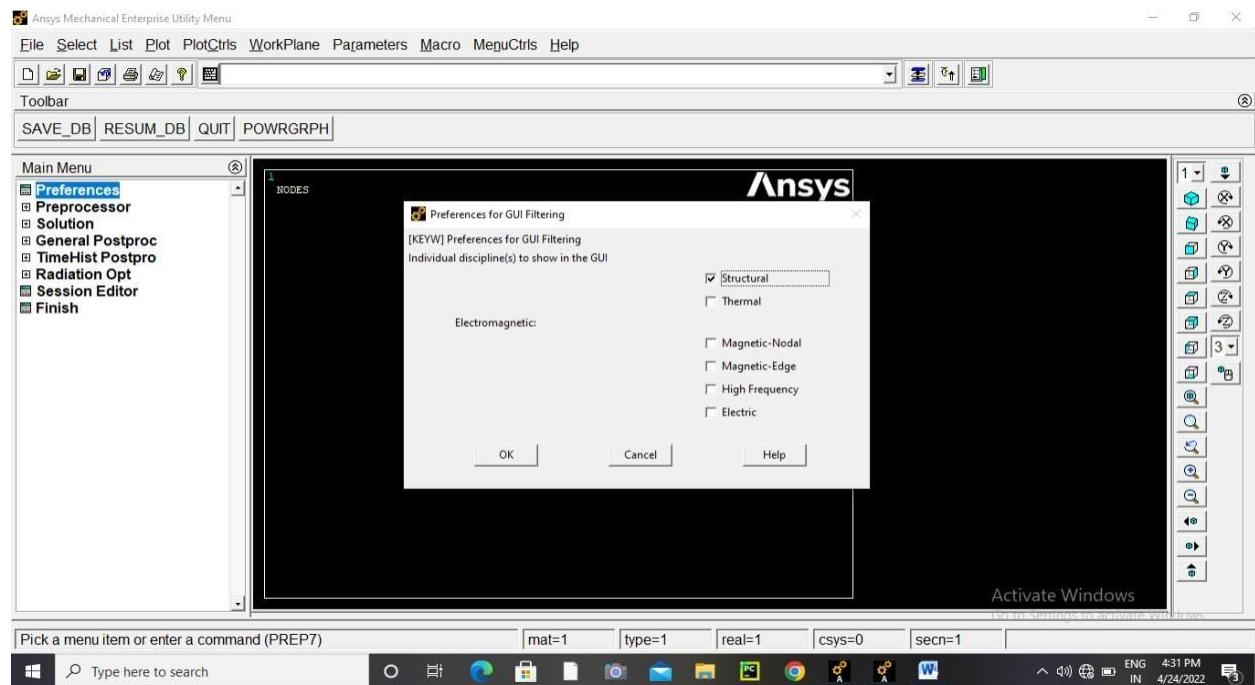
Finite Element Package used:-ANSYS 2022 R2

Problem Statement:-Analyze completely the problem given below. Assume $E=200\text{GPa}$.

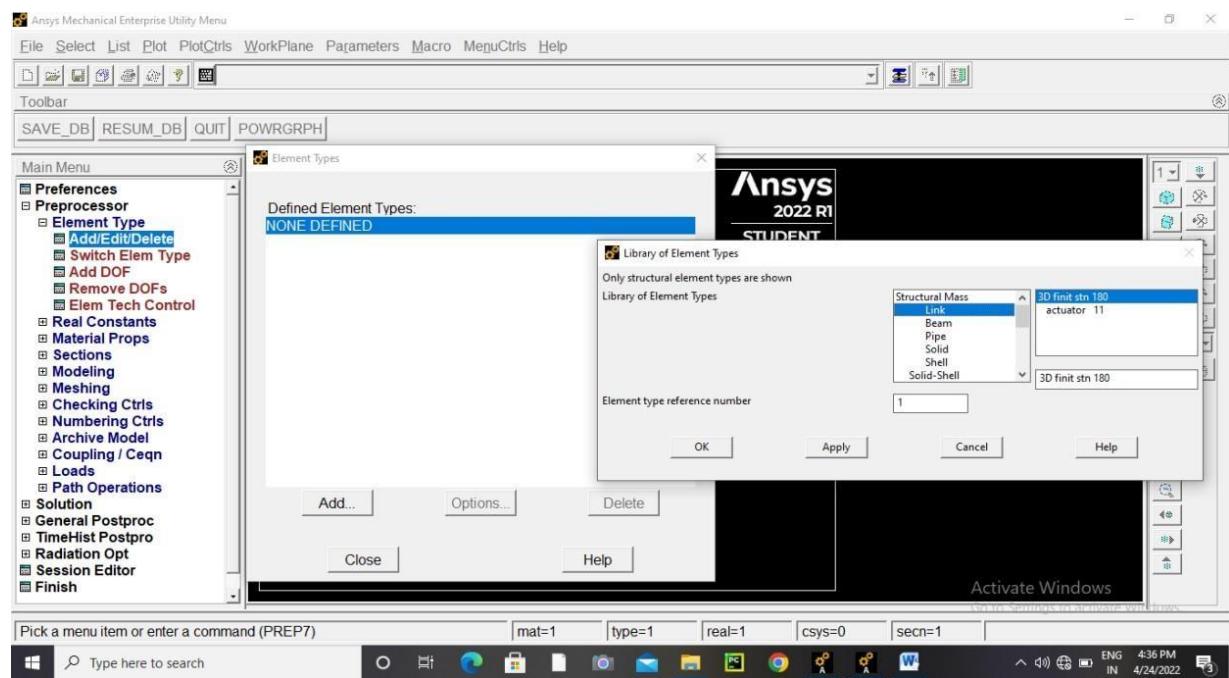


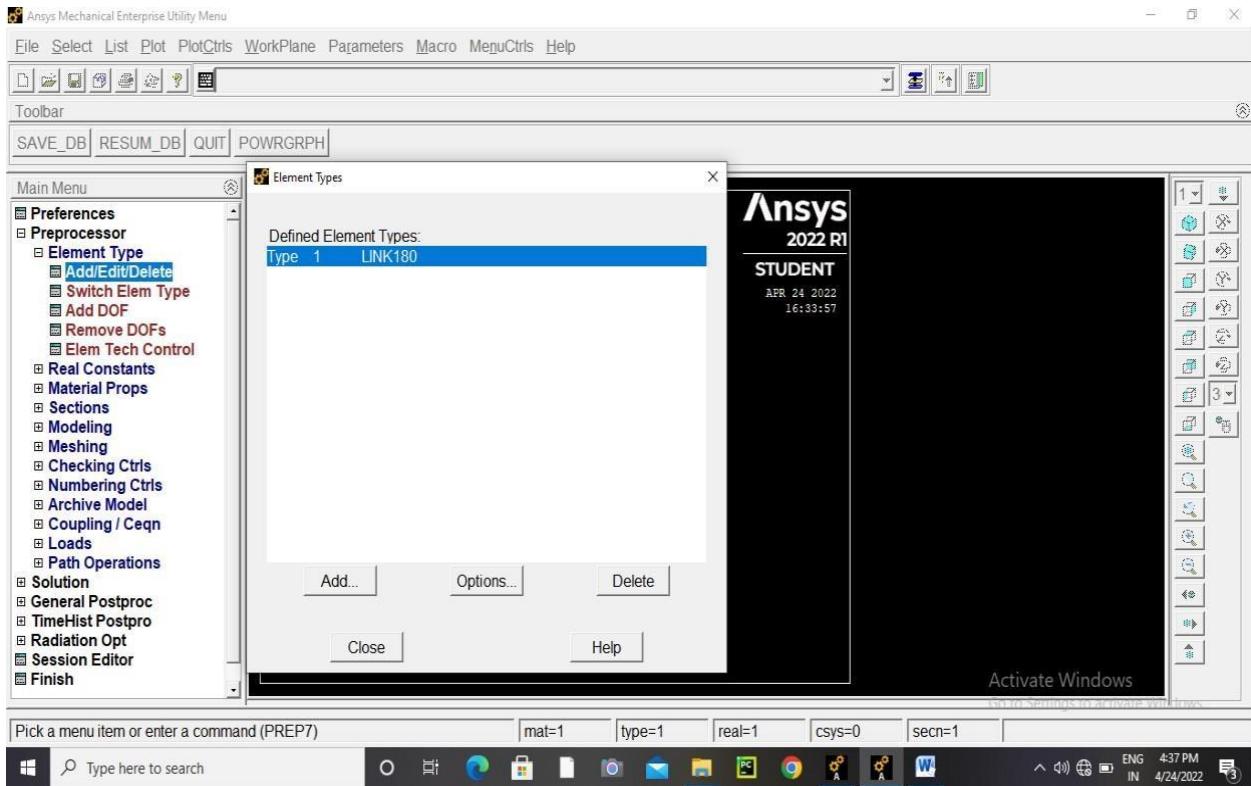
ANSYS(APDL) Solution:-

1. Open Ansys 2021 R2(Student)APDL>Preference>Structural>ok



2. Preprocessor>Element Type>Add/Edit/Delete>Add>link(3D finit stn 180)>ok

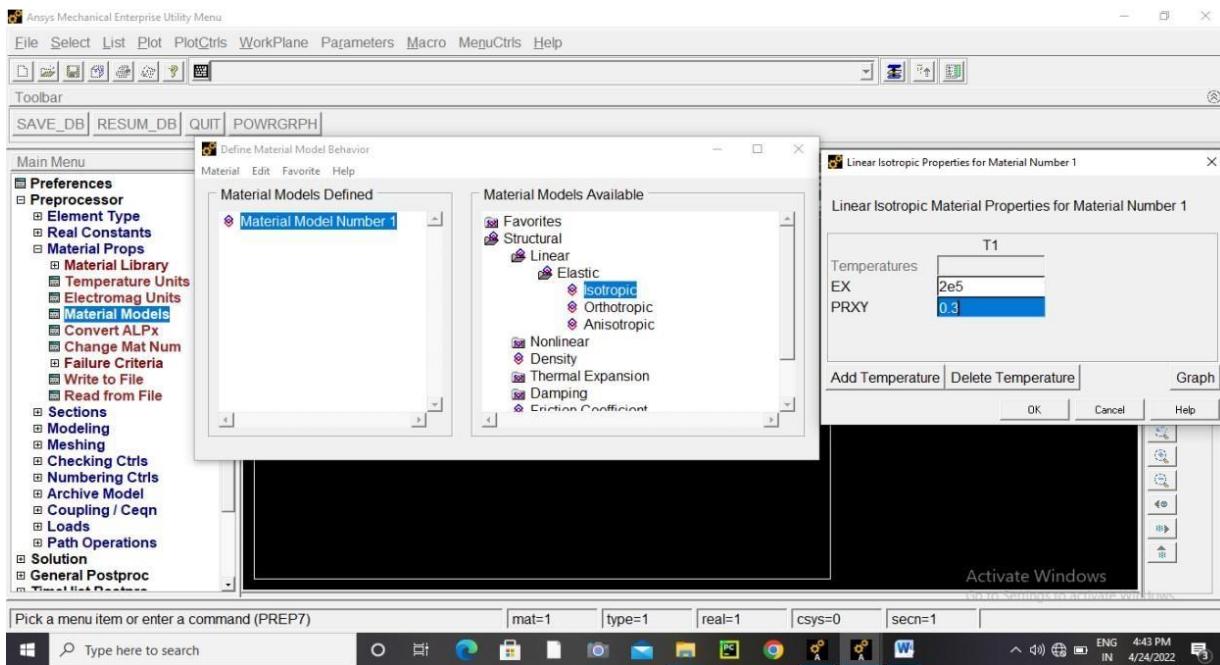




3. Preprocessor>Material Props>Material Models>

a) Material Model Number

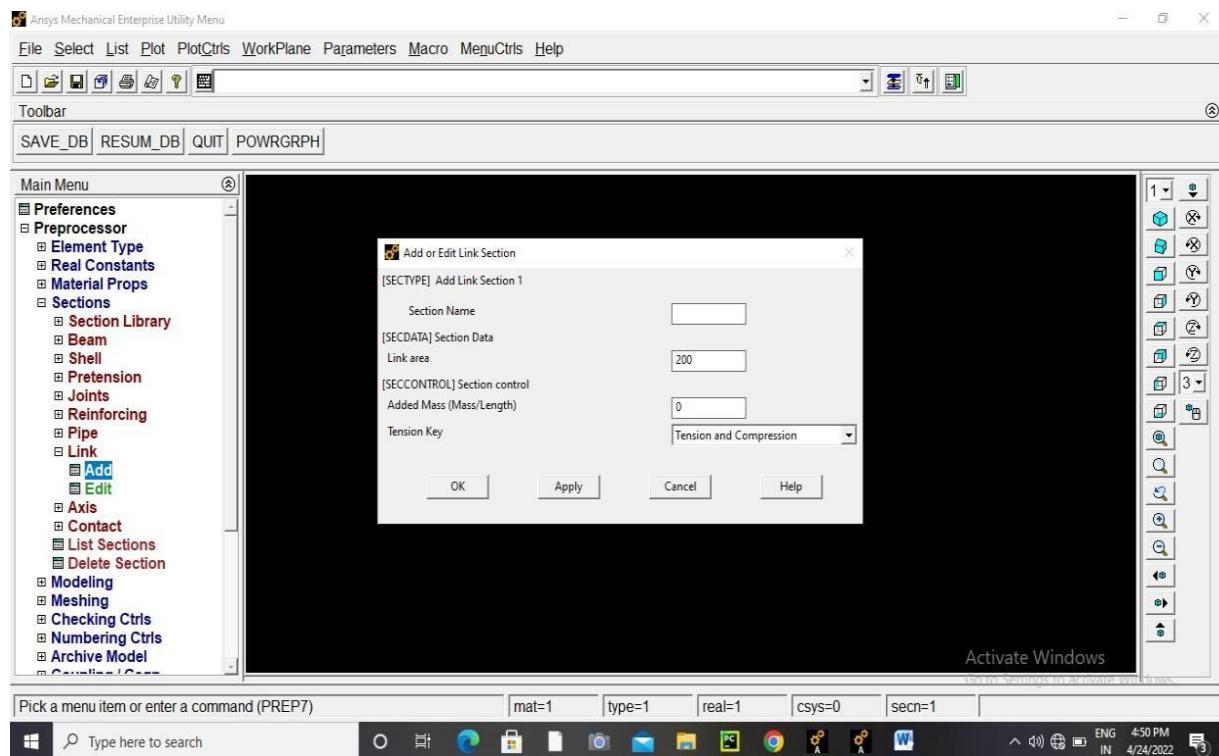
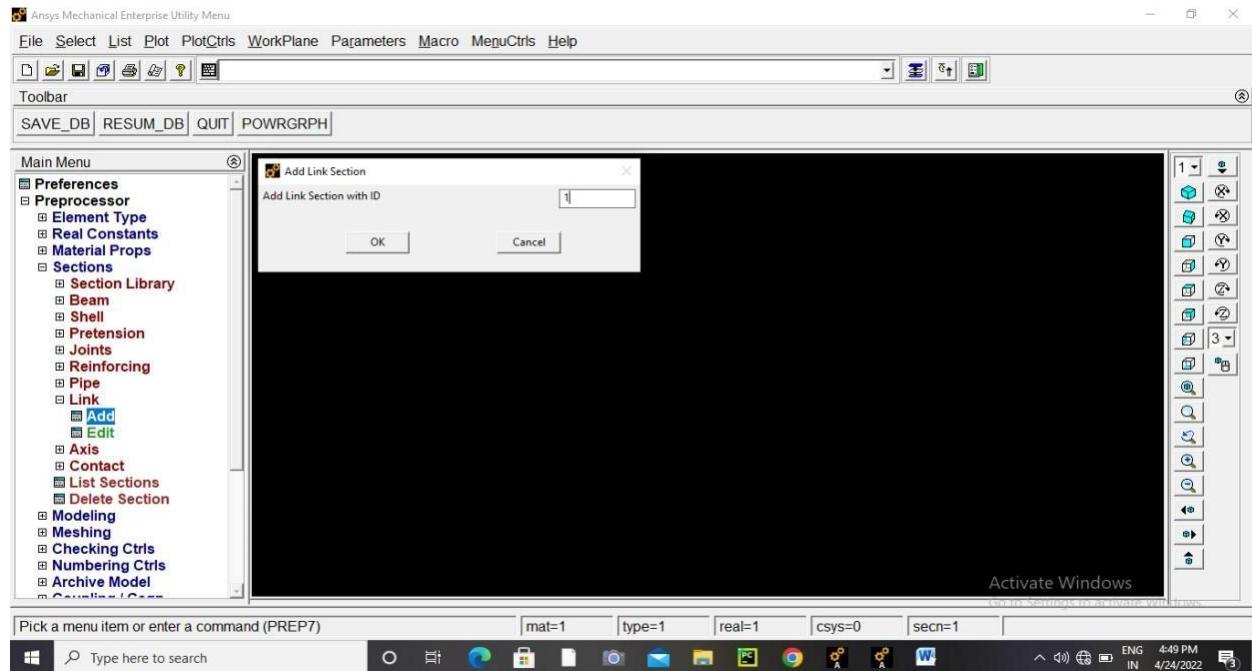
1>Structural>Linear>Elastic>Isotropic>EX=2e5>PRXY=0.3>ok



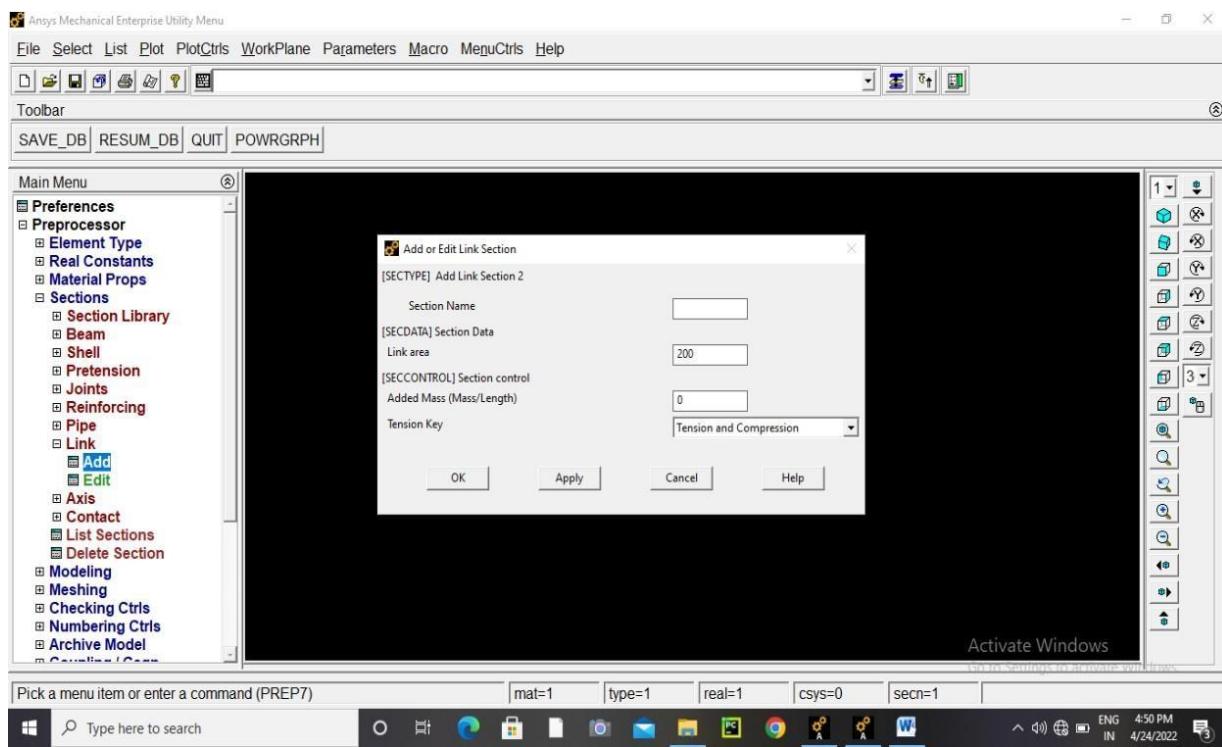
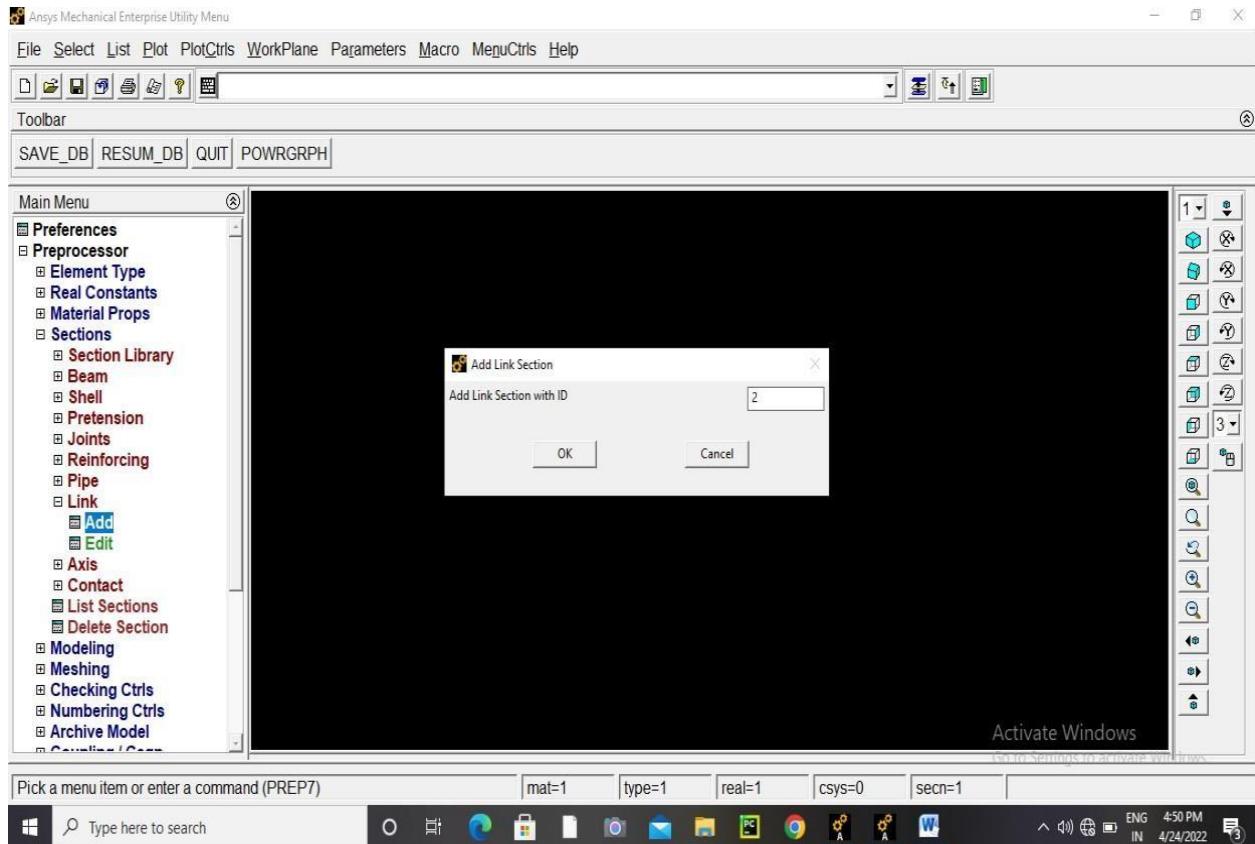
4. Preprocessor>>Section>>Link>>Add

a) Add link Section with ID

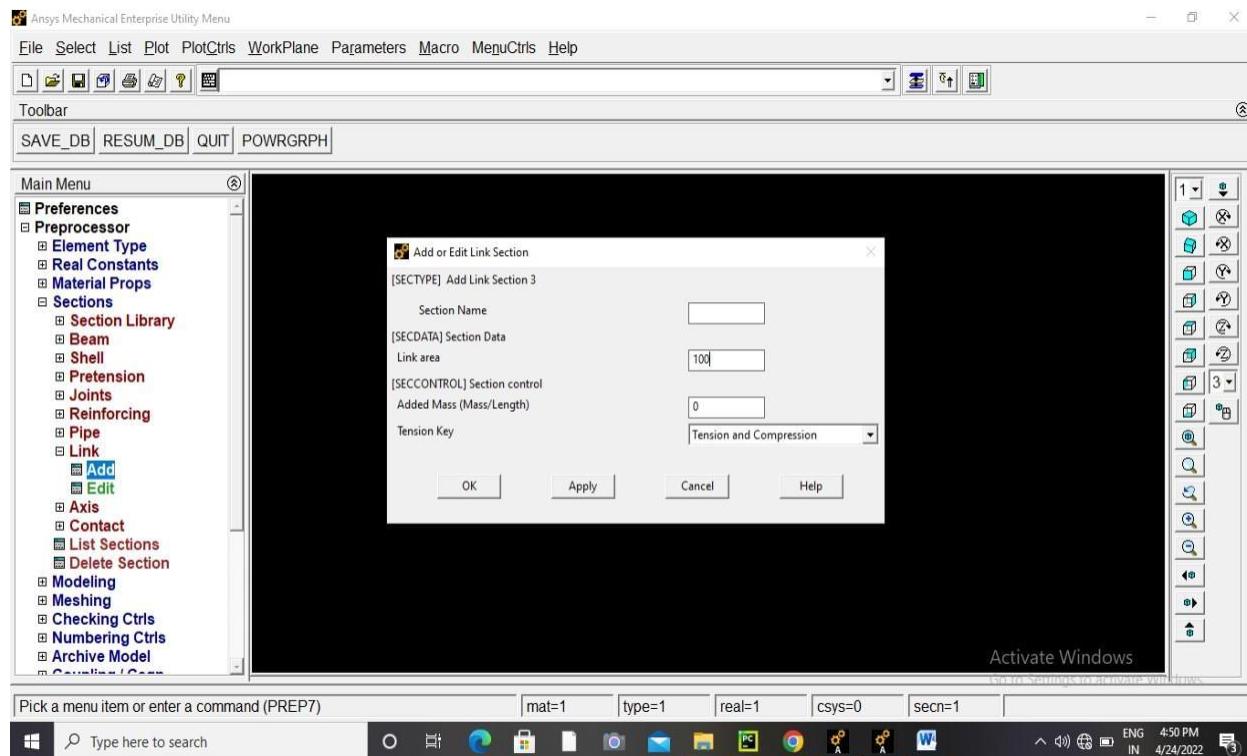
1>>ok>>link area>>200>Apply



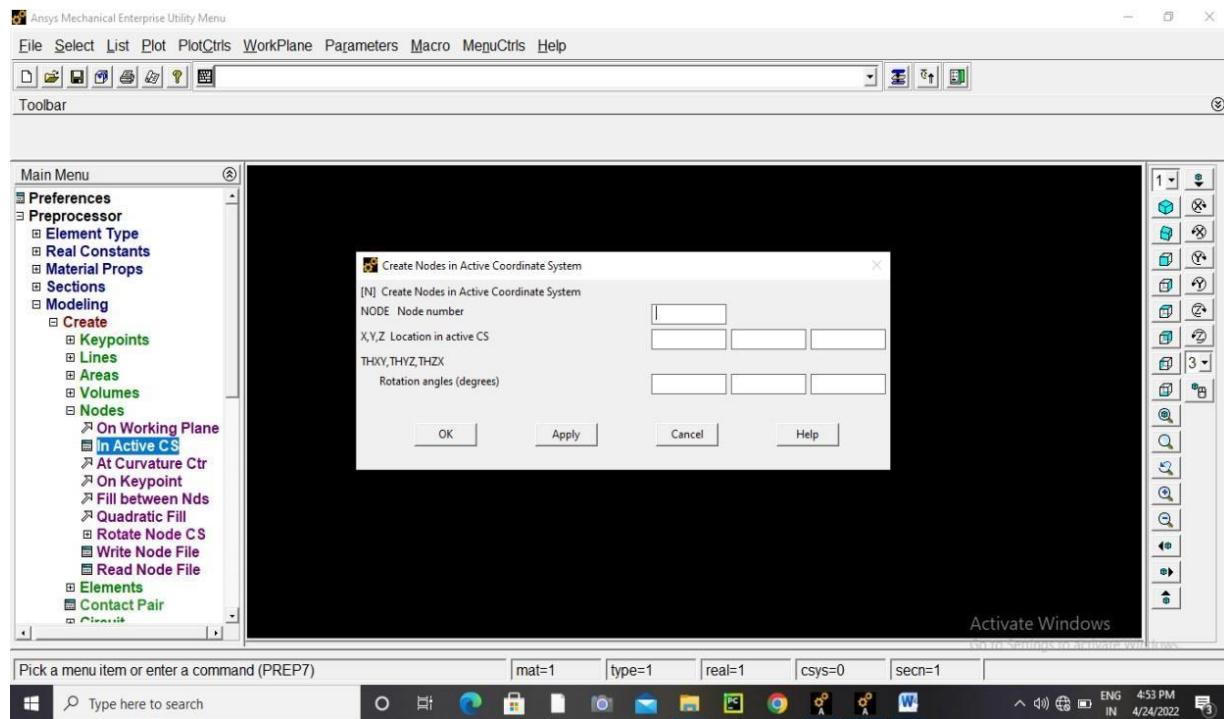
2>>ok>>link area>>200>>Apply



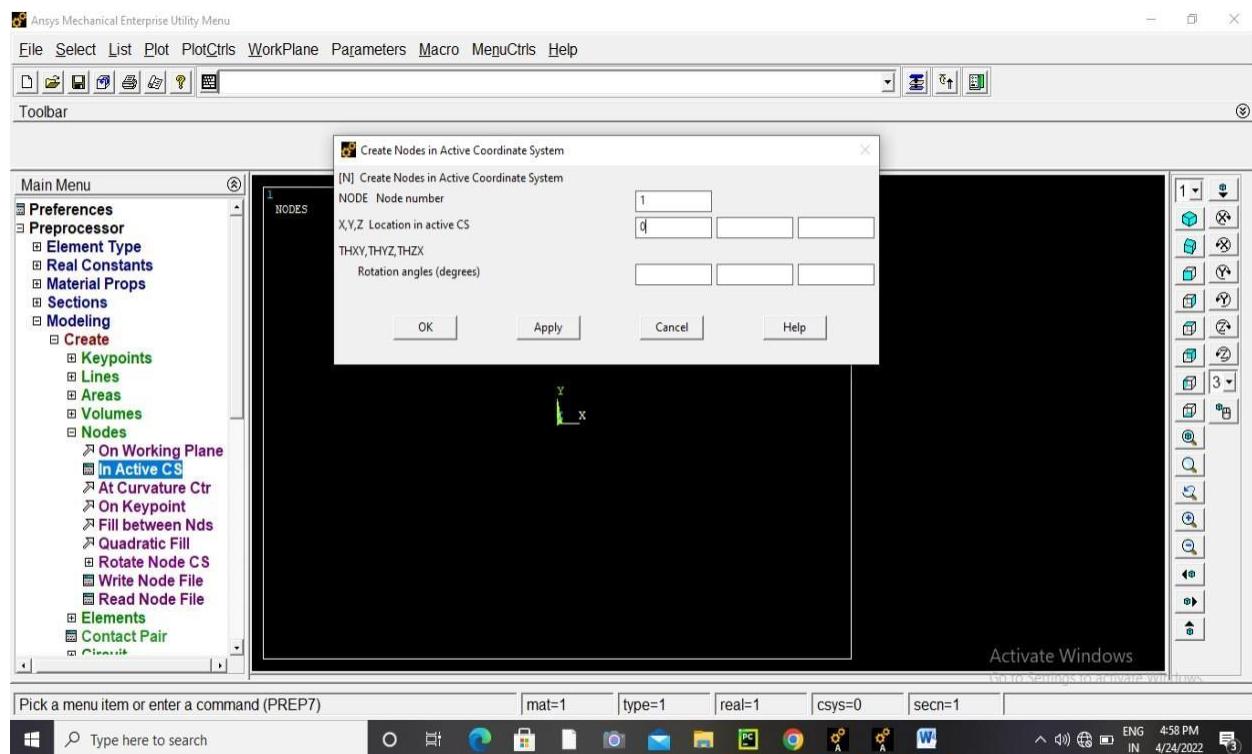
3>ok>link area>100>ok



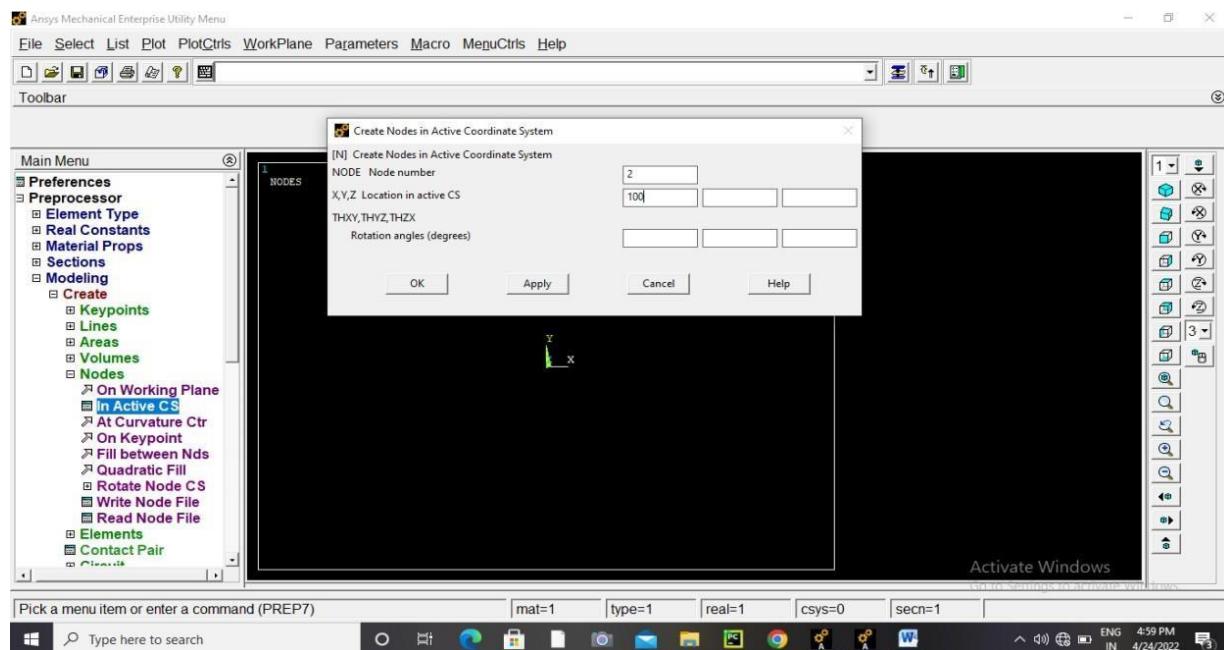
5. Preprocessor>Modeling>Create>Nodes



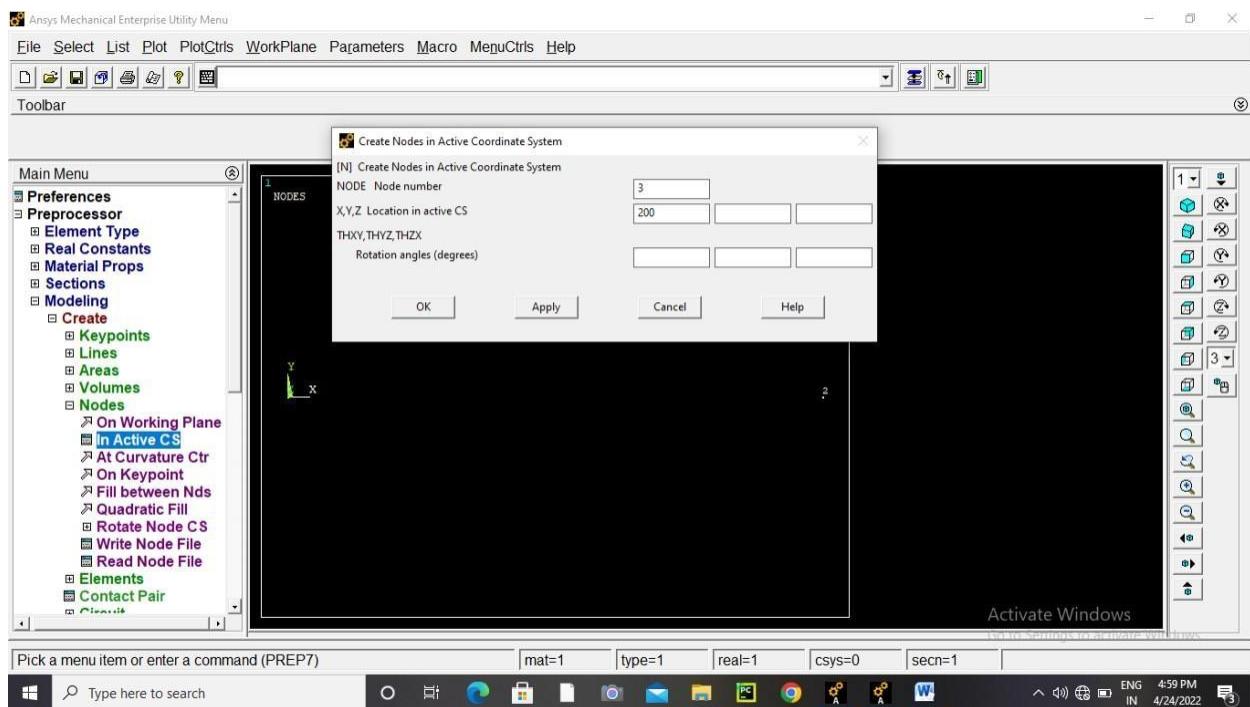
**a) Node Number>1
X,Y,Z Location in active CS>X(0mm)>Apply**



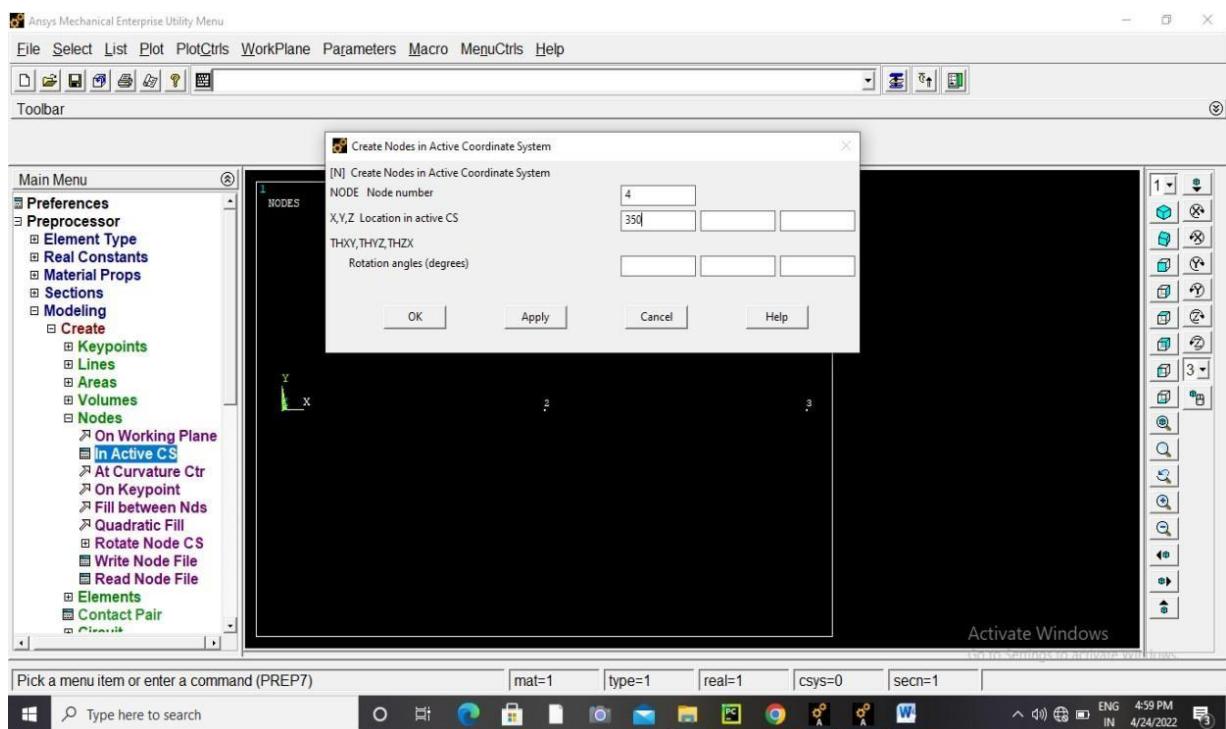
**b) Node Number>2
X,Y,Z Location in active CS>X(100mm)>Apply**



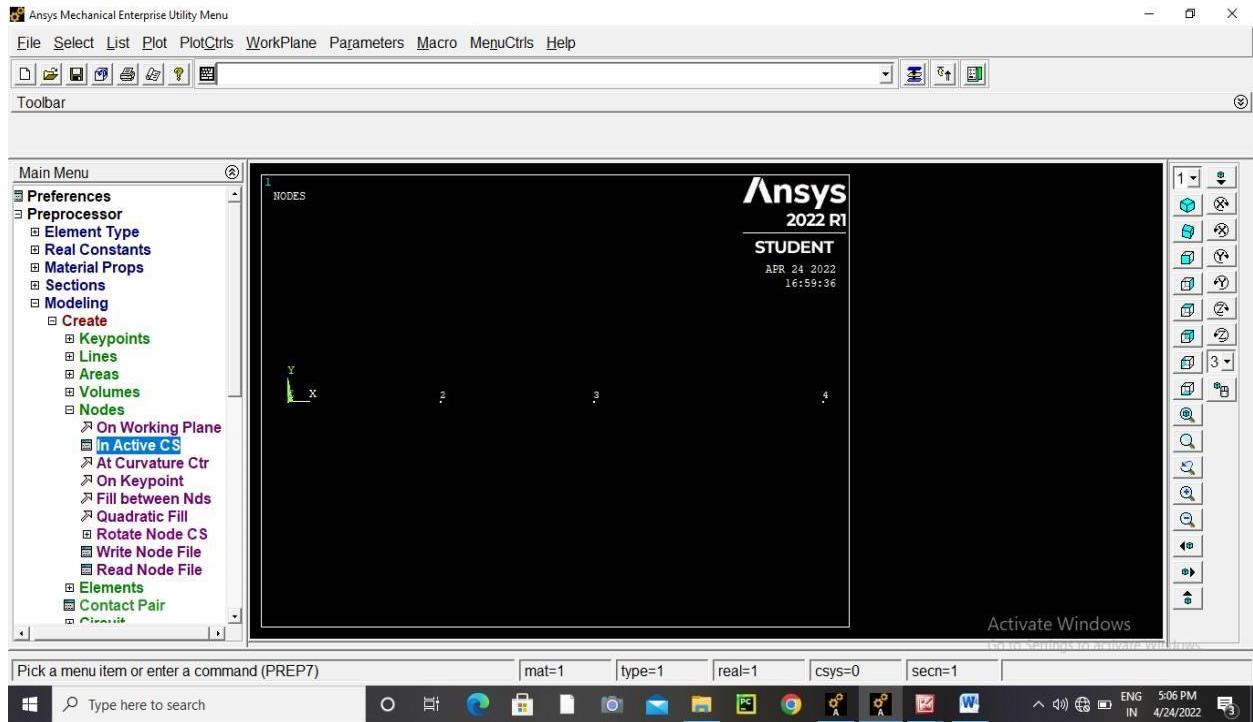
**c) Node Number>3
X,Y,Z Location in active CS>X(200mm)>Apply**



**d) Node Number>4
X,Y,Z Location in active CS>X(350mm)>OK**

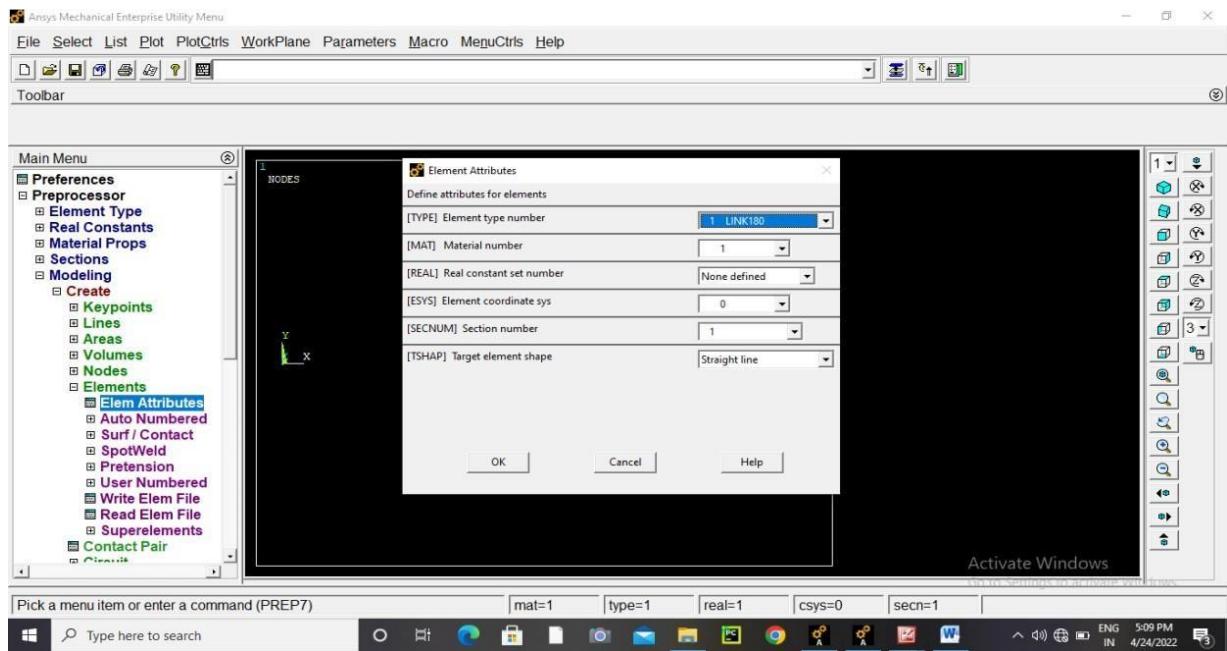


e) All Node Point are generated



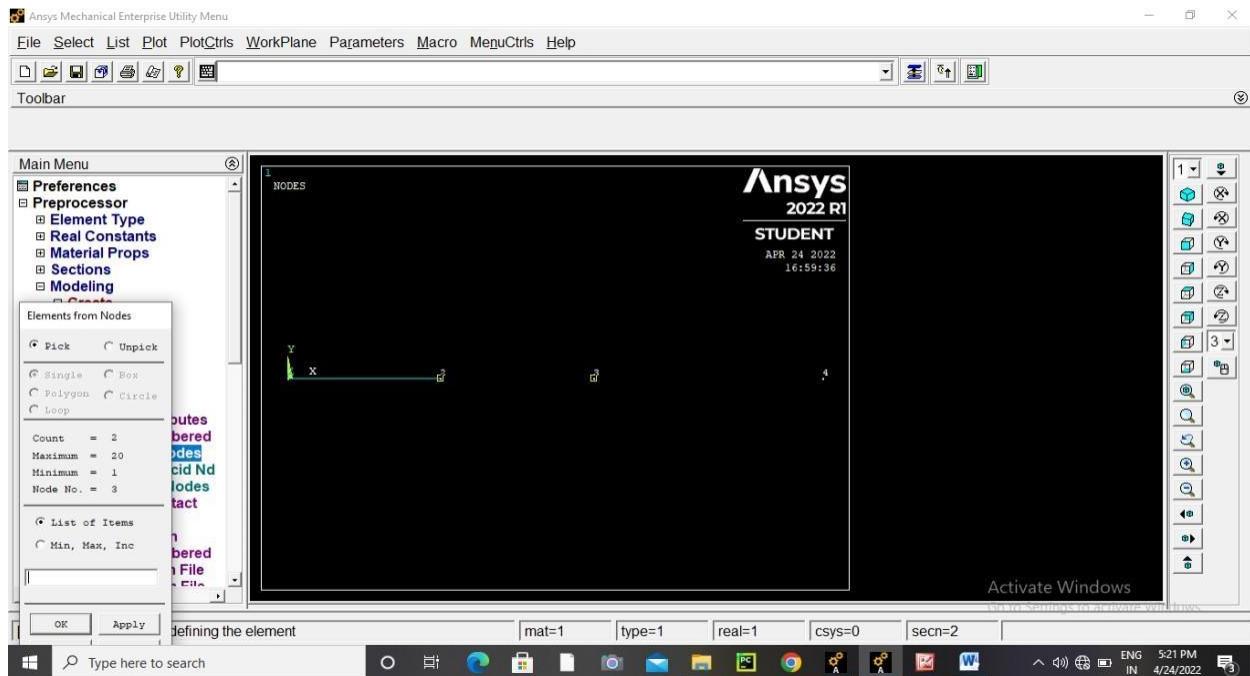
6. Preprocessor>Modeling>Create>Element>

1. Elem Attributes> 1.Material Number>1 2.Section Number>1



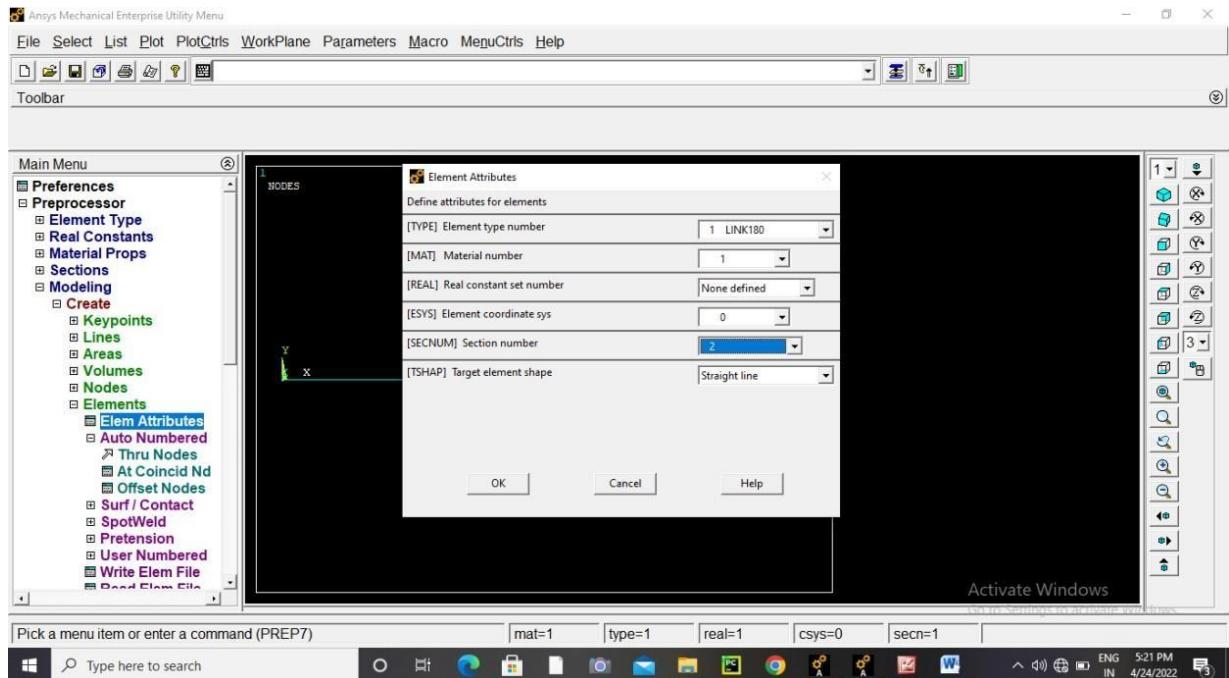
a.1 Auto Numbered>Thru Numbered>(Select Two point e.g=1 &2)>Ok

(Line Will be Generated from Point 1 & 2 as shown in figure)



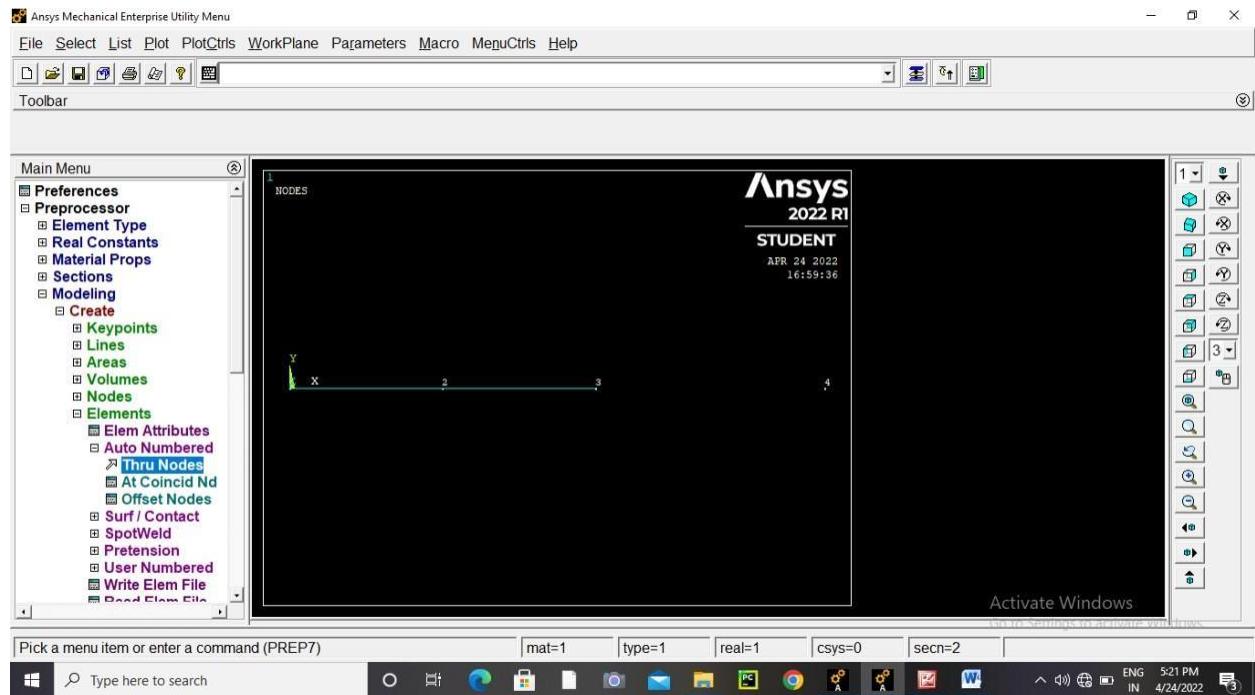
2. Elem Attributes> 1.Material Number>1

2. Section Number>2

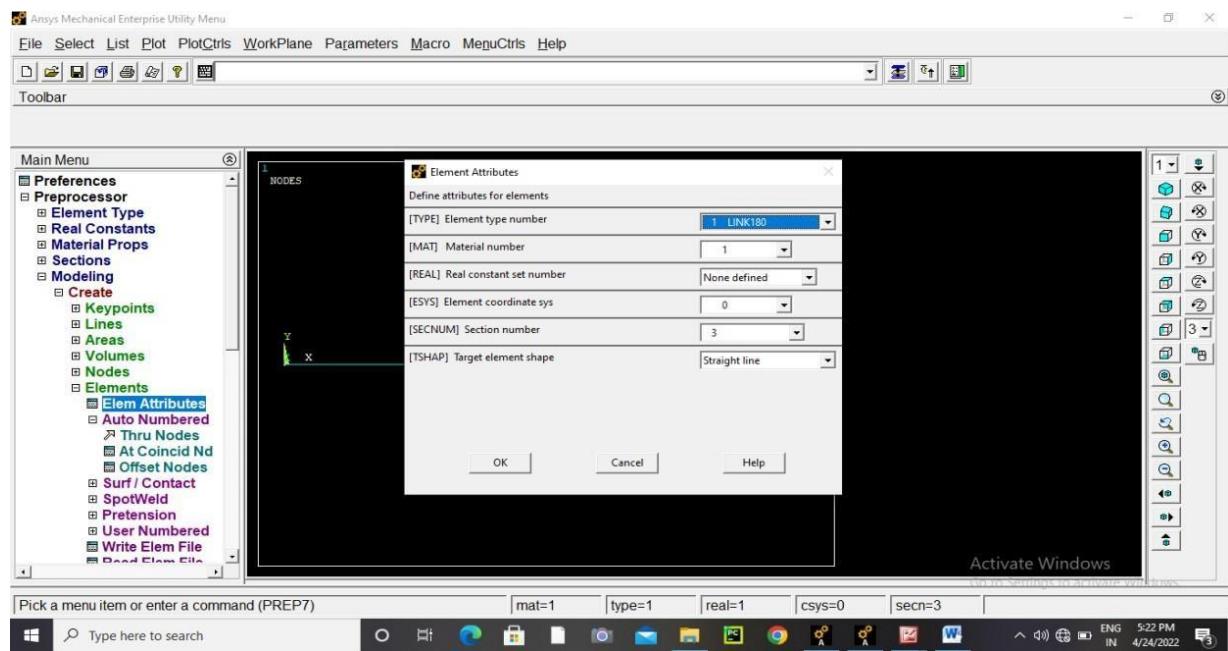


b).1 Auto Numbered>Thru Numbered>(Select Two point e.g=2 & 3)>Ok

(Line Will be Generated from Point 2 & 3as shown in figure)

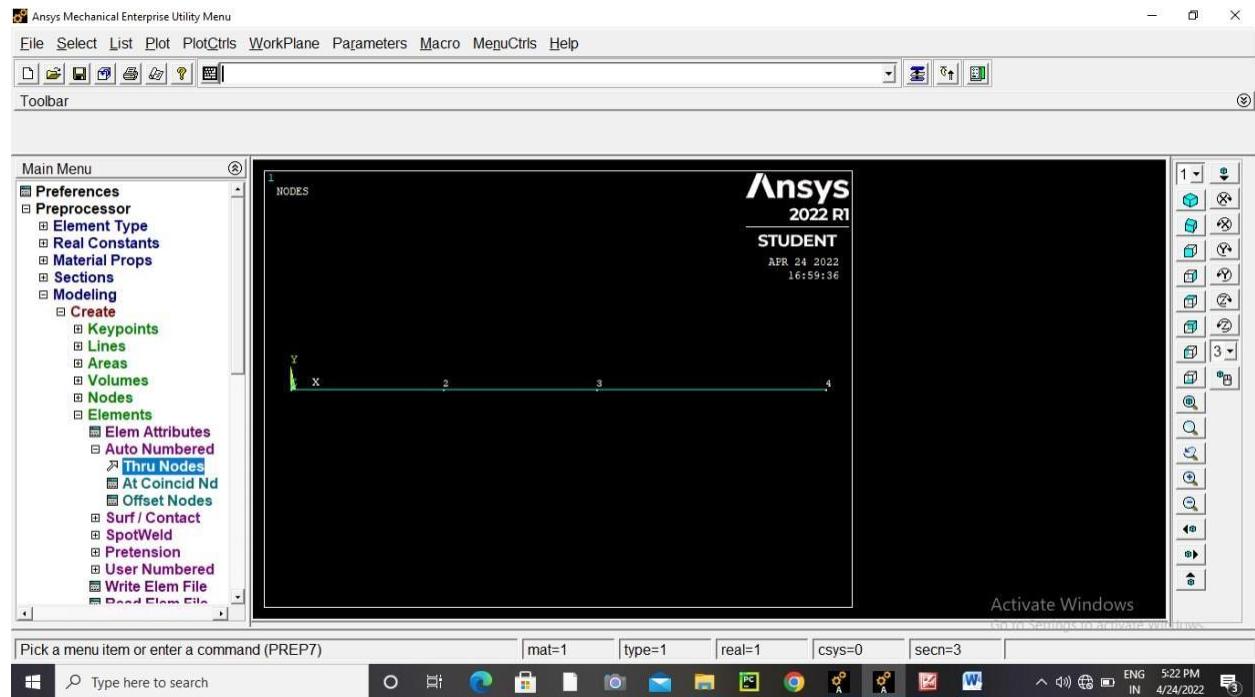


3. Elem Attributes>1.Material Number>1 2.Section Number>3



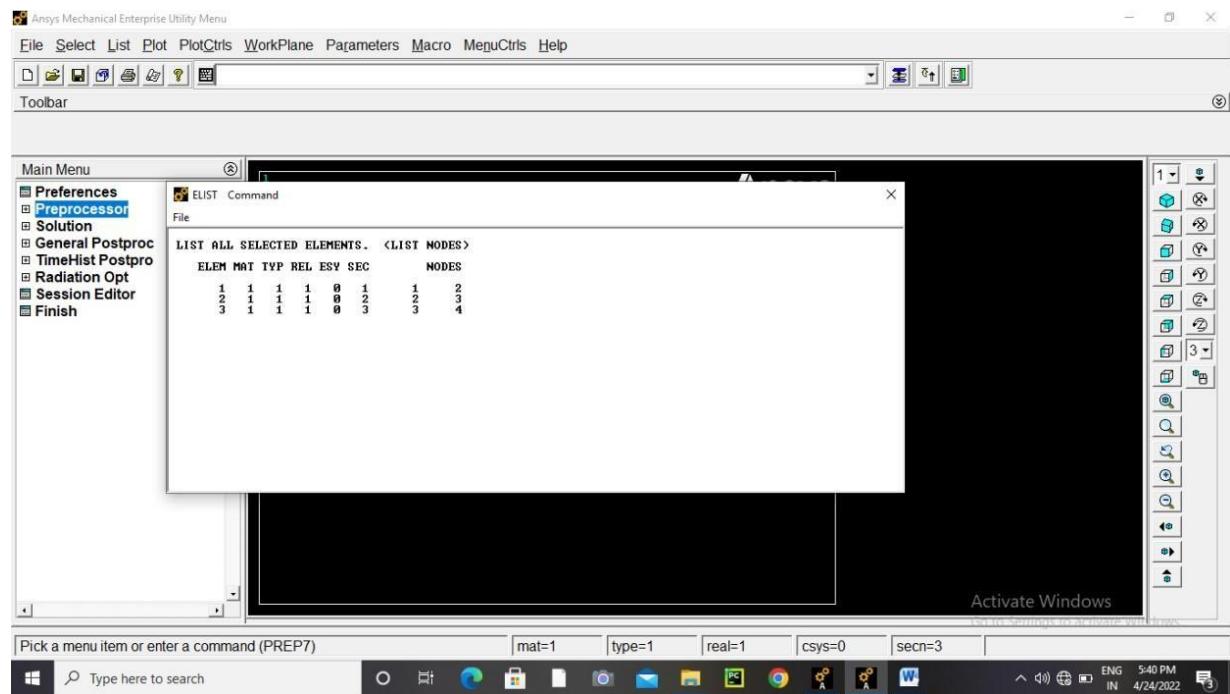
c.1 Auto Numbered>Thru Numbered>(Select Two point e.g=3 & 4)>Ok

(Line Will be Generated from Point 3 & 4 as shown in figure)

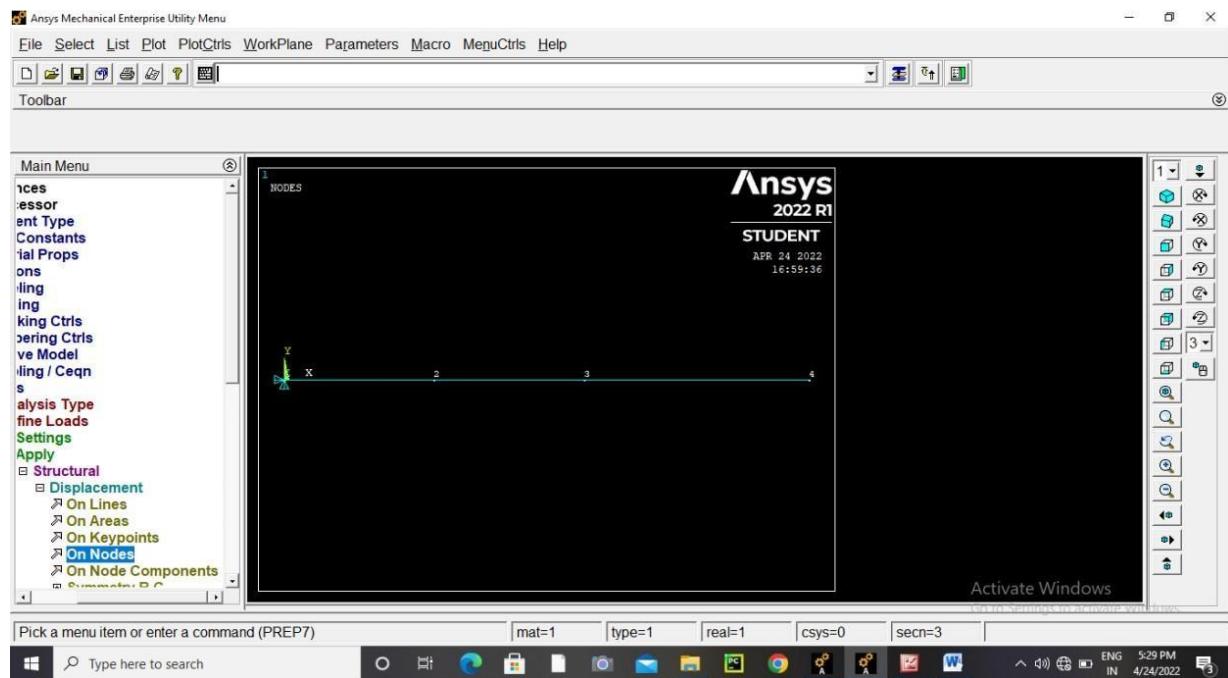
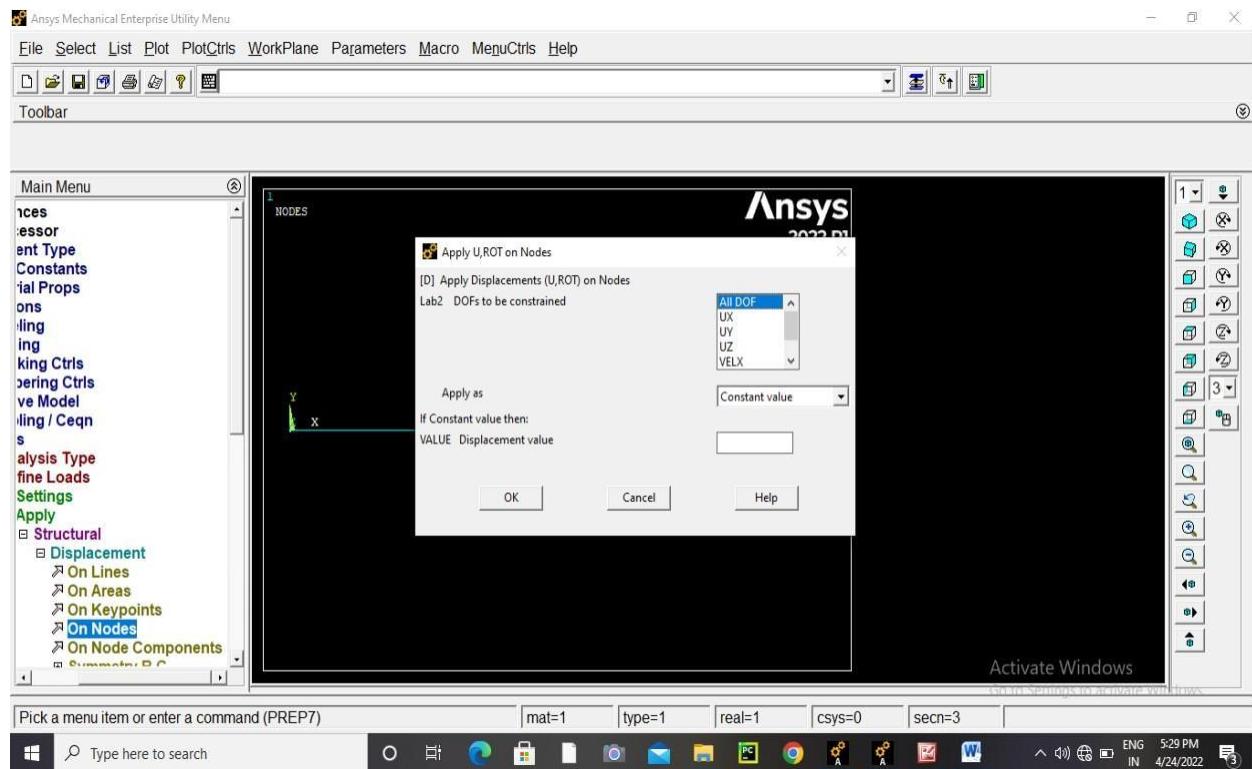


Note .1(Check all material & section id are applied properly are not)

List>>Element>>Nodes + Attributes>>Chart will show up. Check the chart.

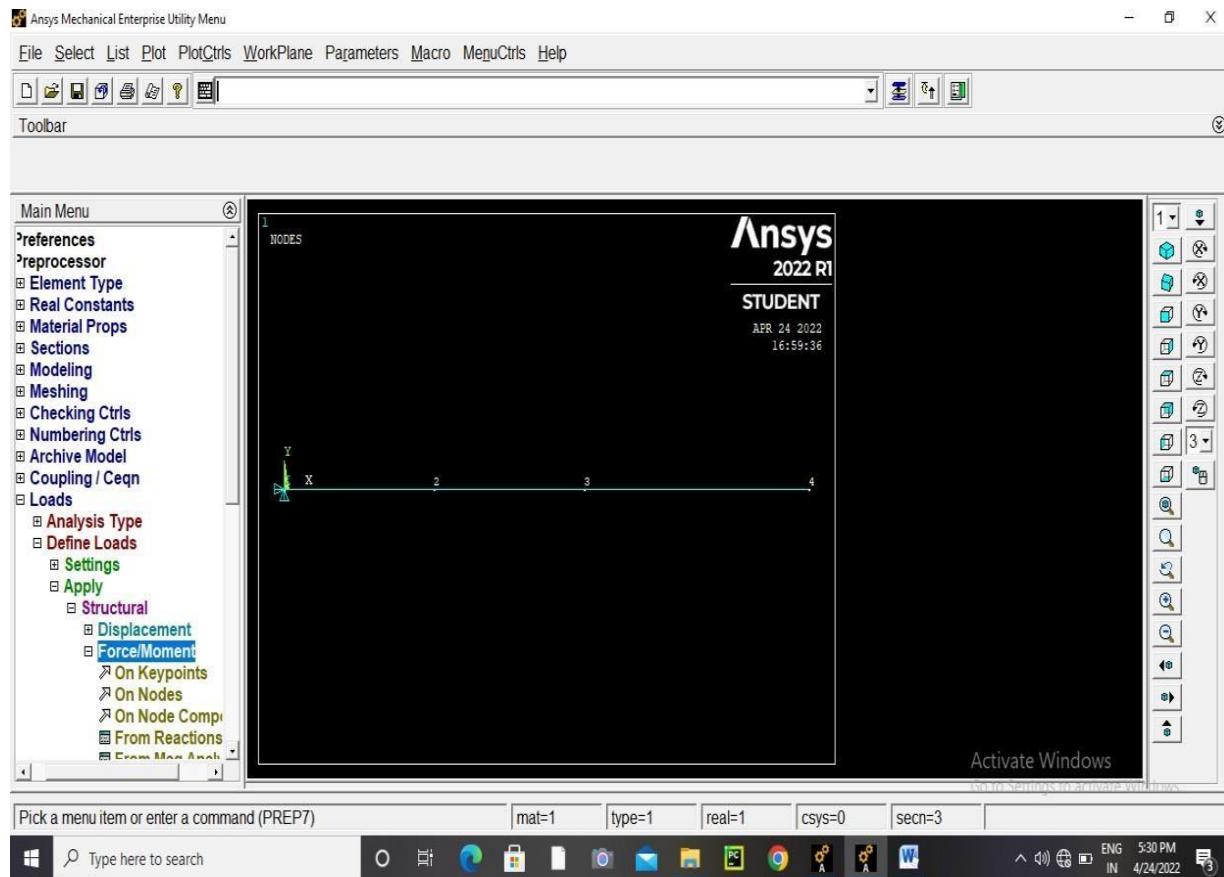


7. Preprocessor>Load>Define Load>Apply>Structural>Displacement>on Nodes>(Select Node Number 1)>ok>All dof>ok

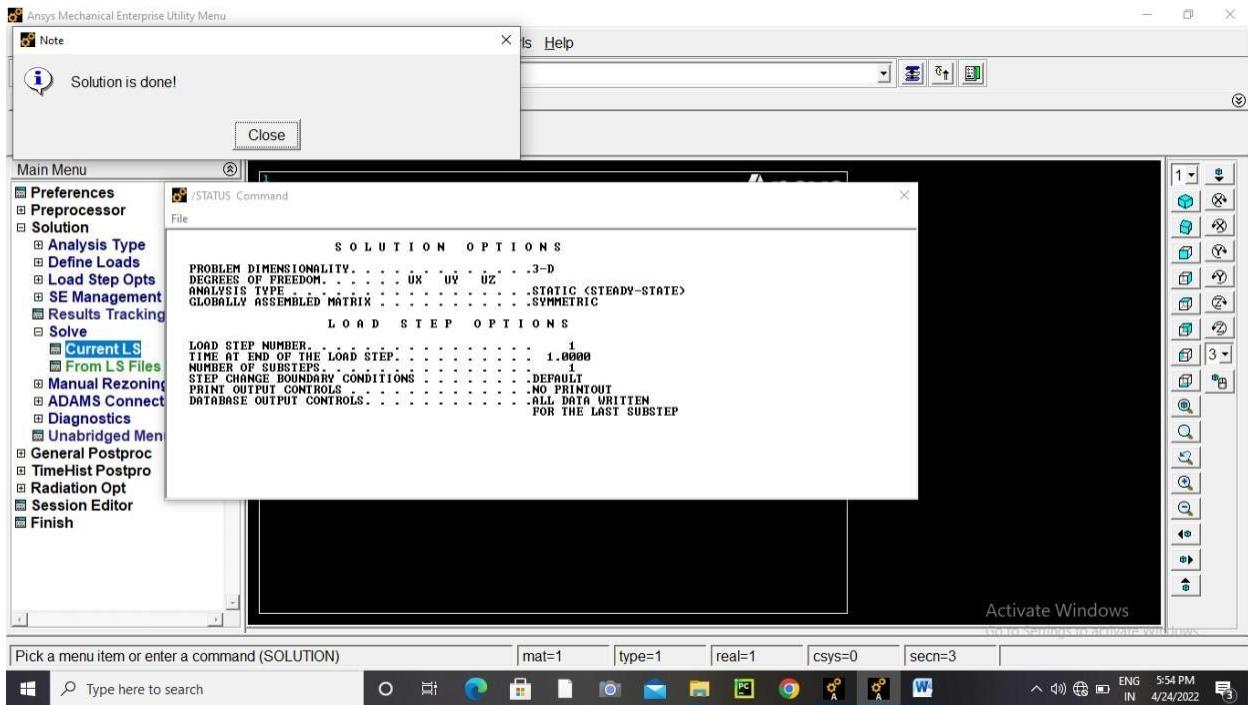


**8. Preprocessor>>Load>>DefineLoad>>Apply>>Structural>>Force/Moment>>on Nodes>>(Select Node Number 2)>>ok>>Direction of force/moment=(FX)>>VALUE Force/Moment value=20000>>OK
Preprocessor>>Load>>Define**

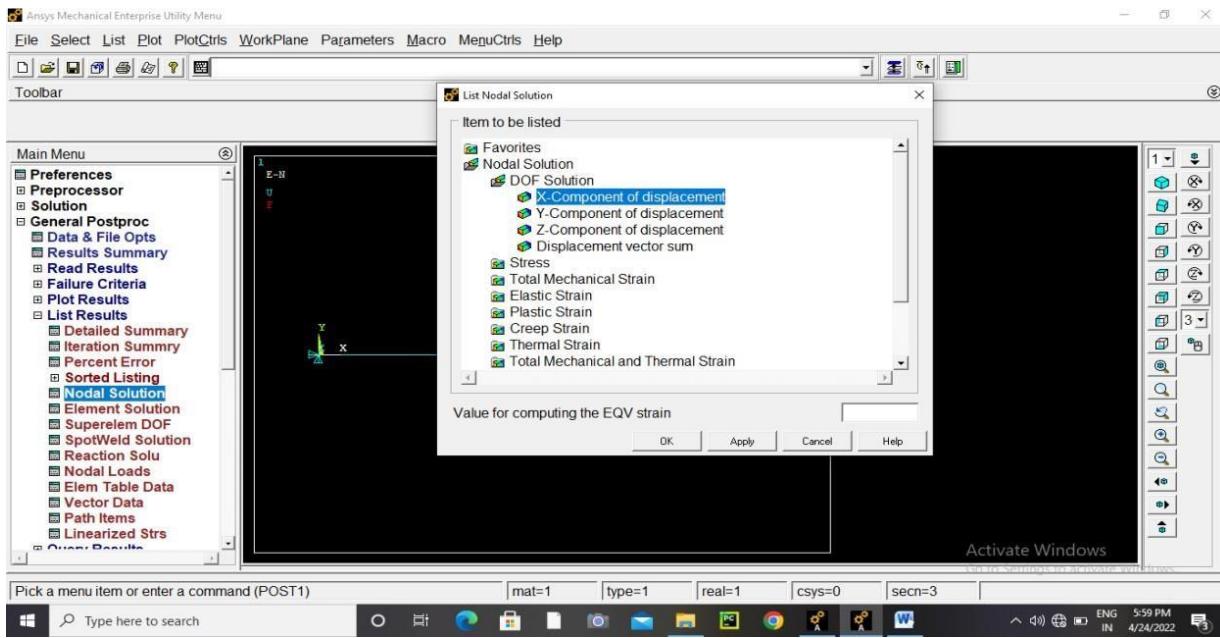
Load>>Apply>>Structural>>Force/Moment>>on Nodes>>(Select Node Number 4)>>ok>>Direction of force/moment=(FX)>>VALUE Force/Moment value=10000>>OK

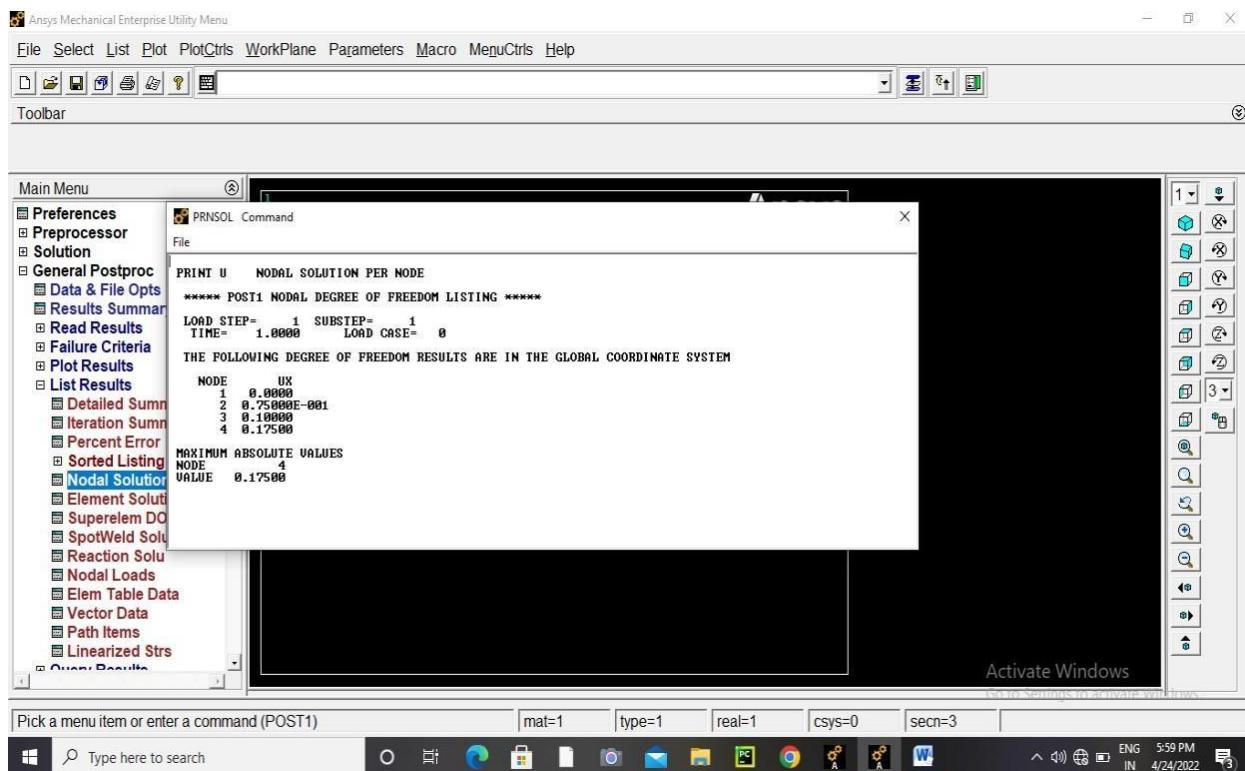


9. Solution>Solve Current LS>ok>Message will appear solution done (If message not appear when you have done some mistake you have to follow the step again)>close



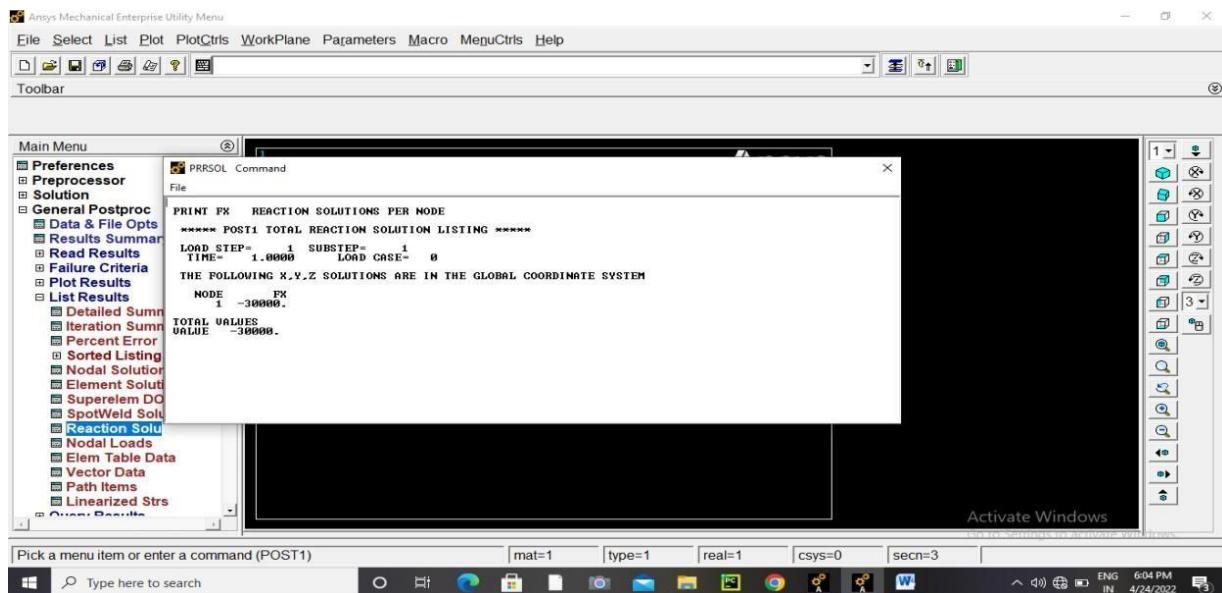
10. General Postproc>List Results>Nodal Solution>Window will appear >Nodal Solution>>DOF Solution>X Component of Displacement>ok

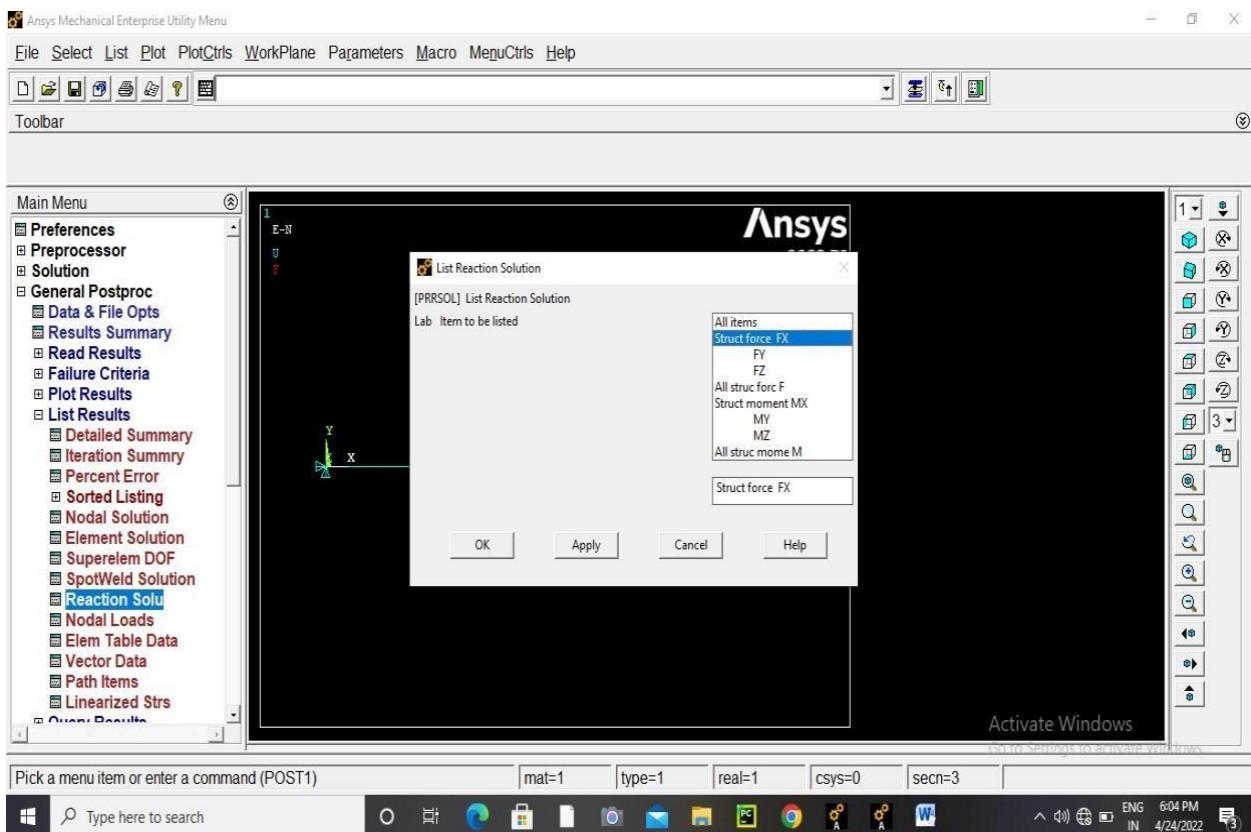




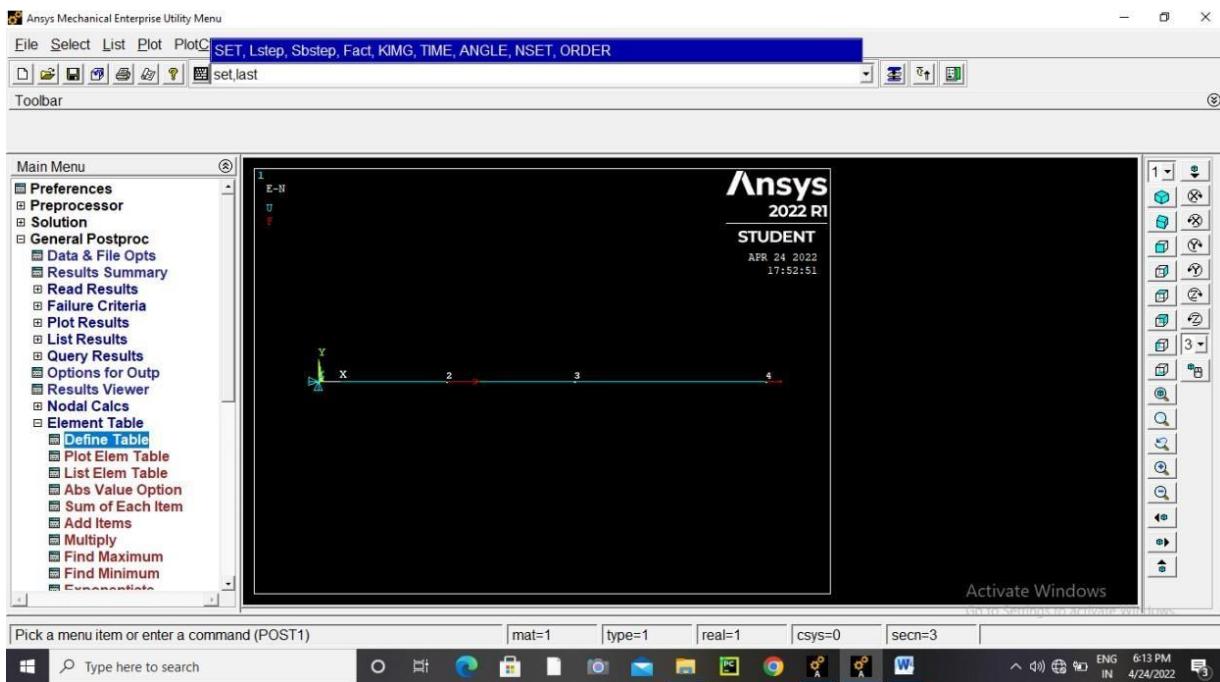
**11.General Postproc>>List Results>>Reaction Sol>>Window will appear
>>Struct force FX>>ok**

Result will appear

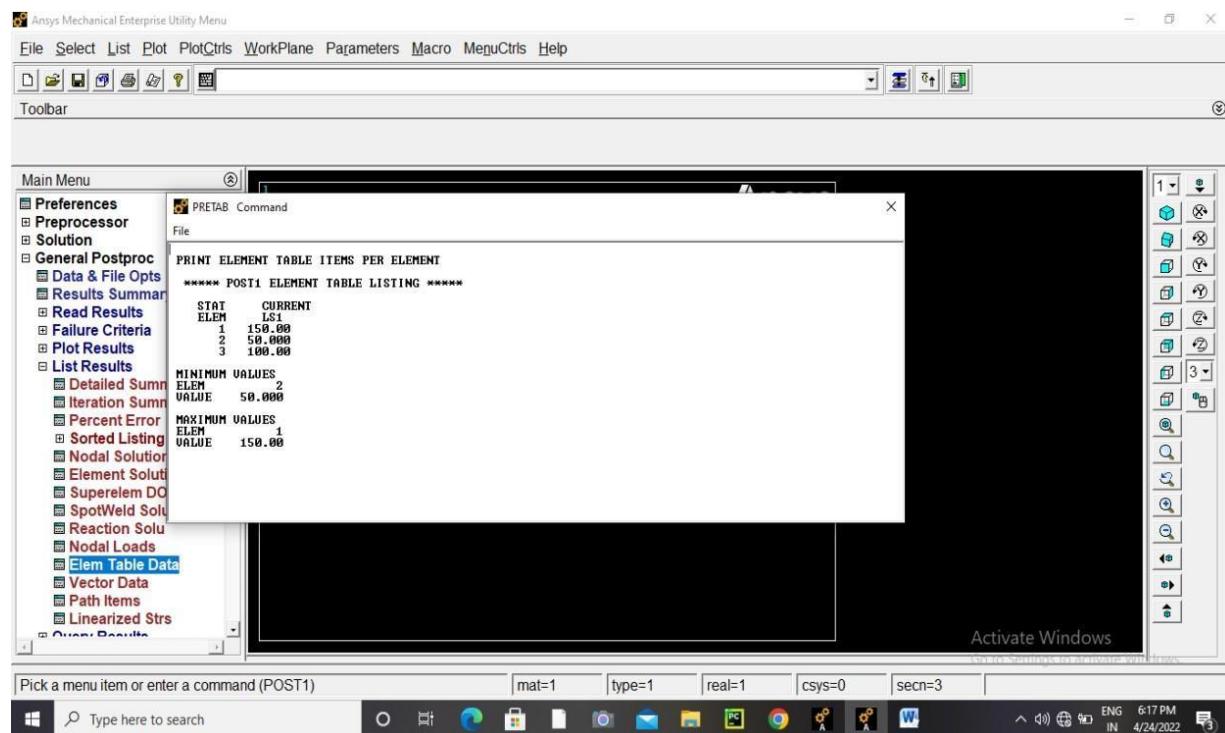
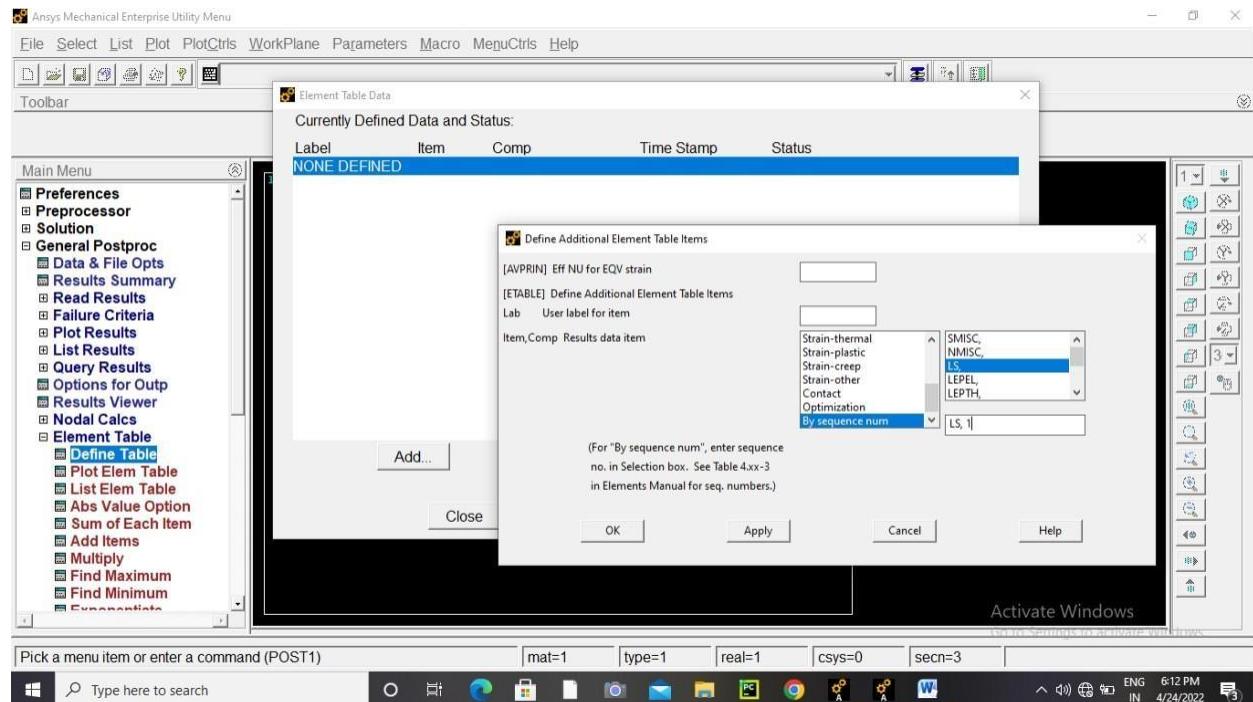




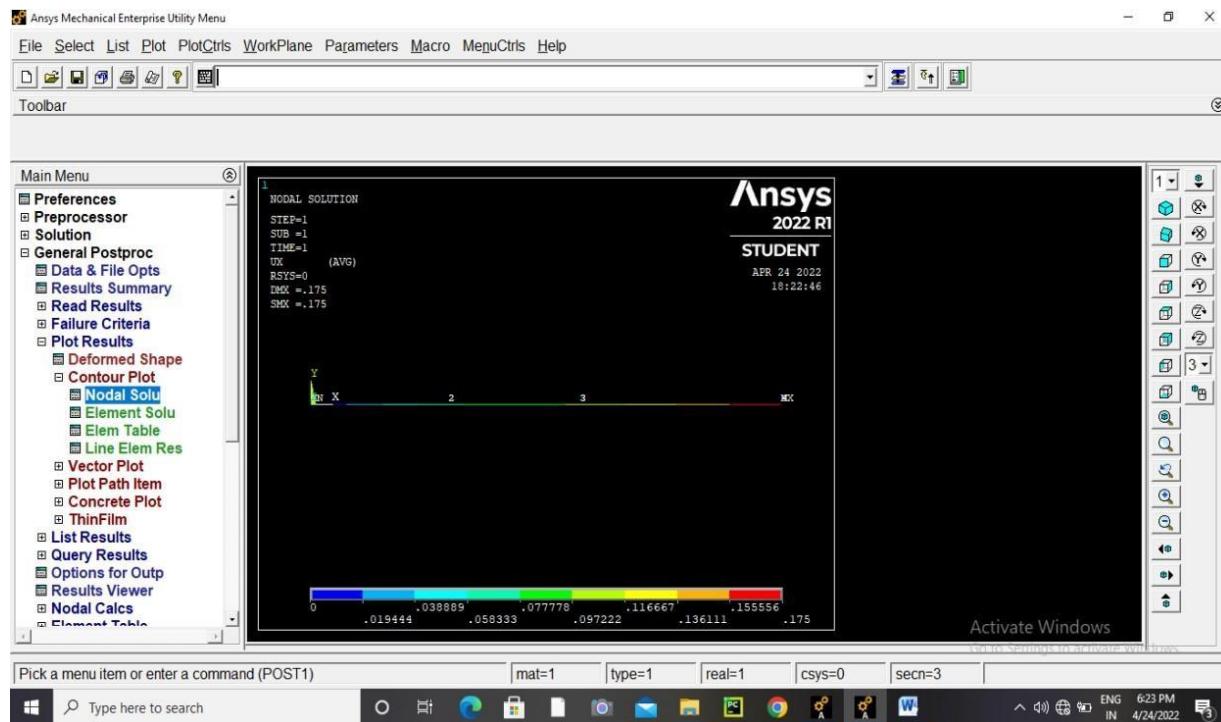
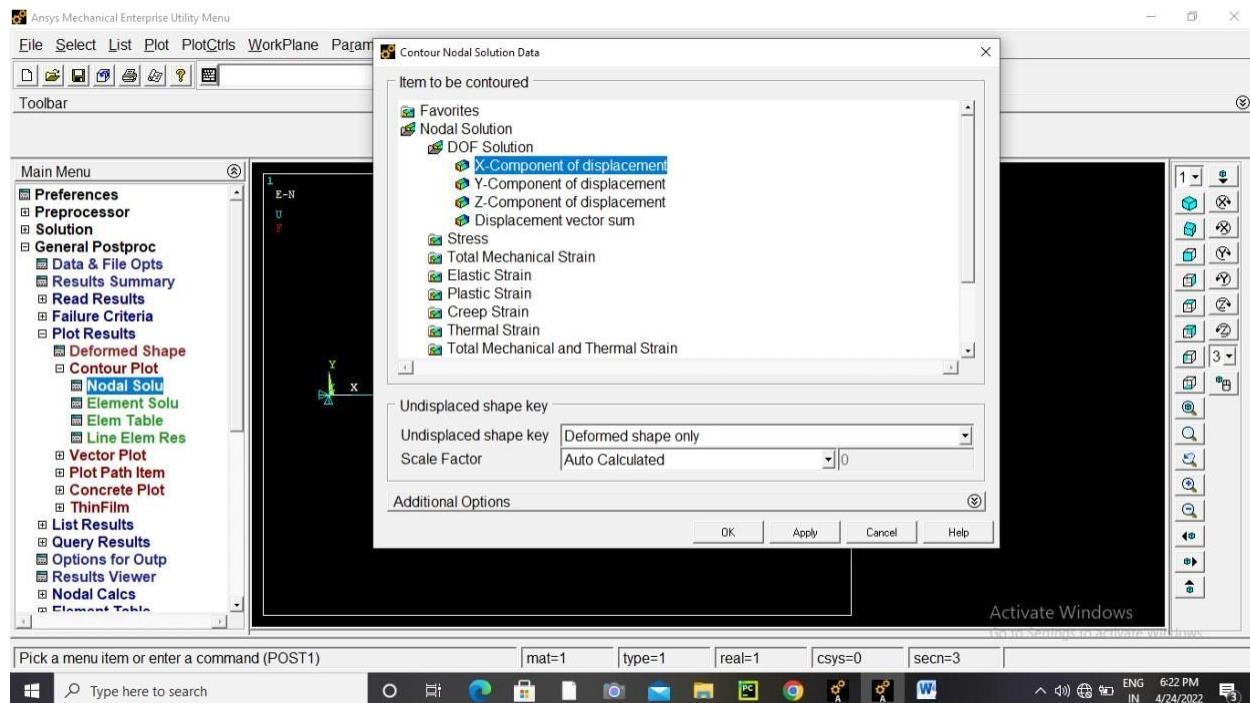
12. Before Calculation of stress we have write 1 command in command Prompt=Set,last



General PostProc>Element table>Define Table>Window will appear>ADD...>(Comp Result data item)>By sequence num>LS,1>ok>close)

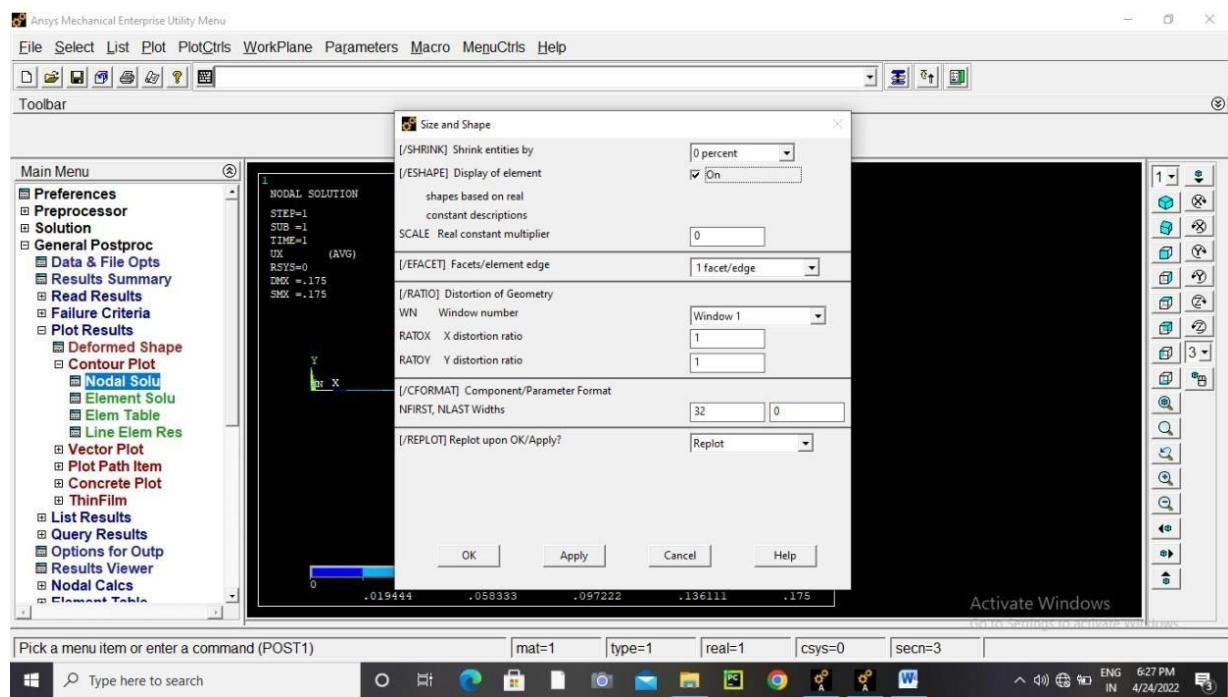
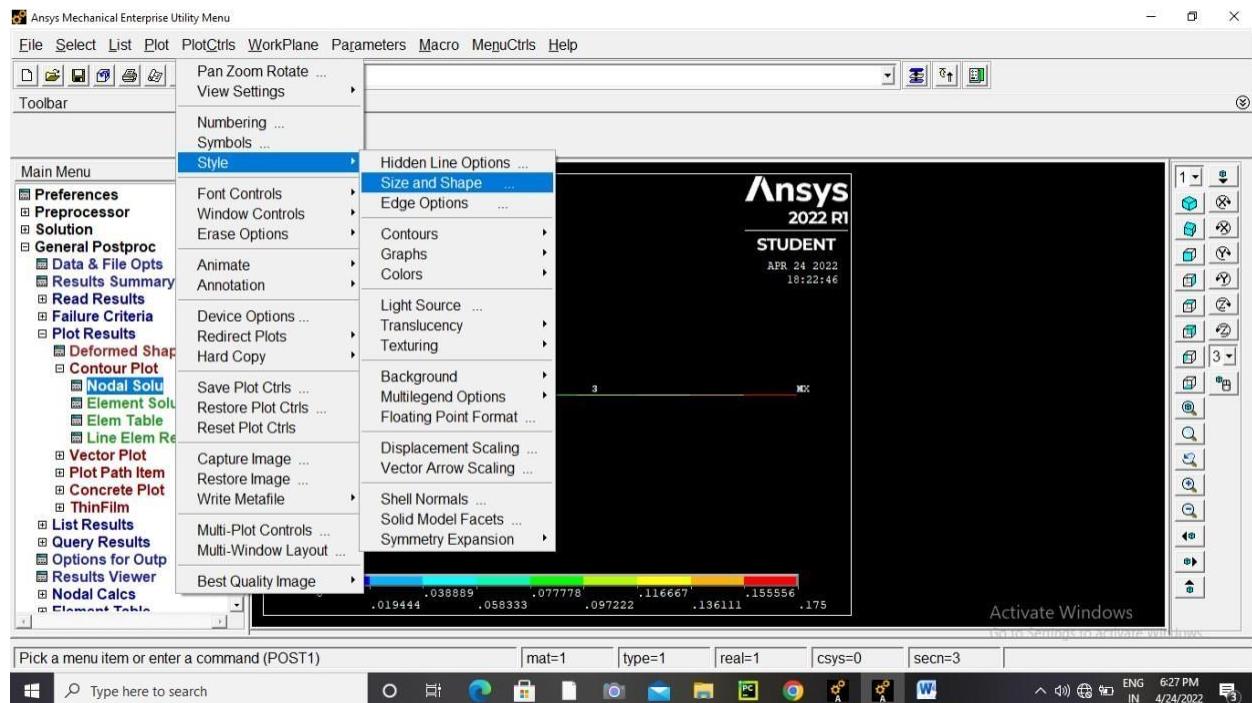


13.General Postproc>Plot Result>Contour Plot>Nodal Solu>Window will appear>Nodal Solution>DOF Solution>X Component of displacement

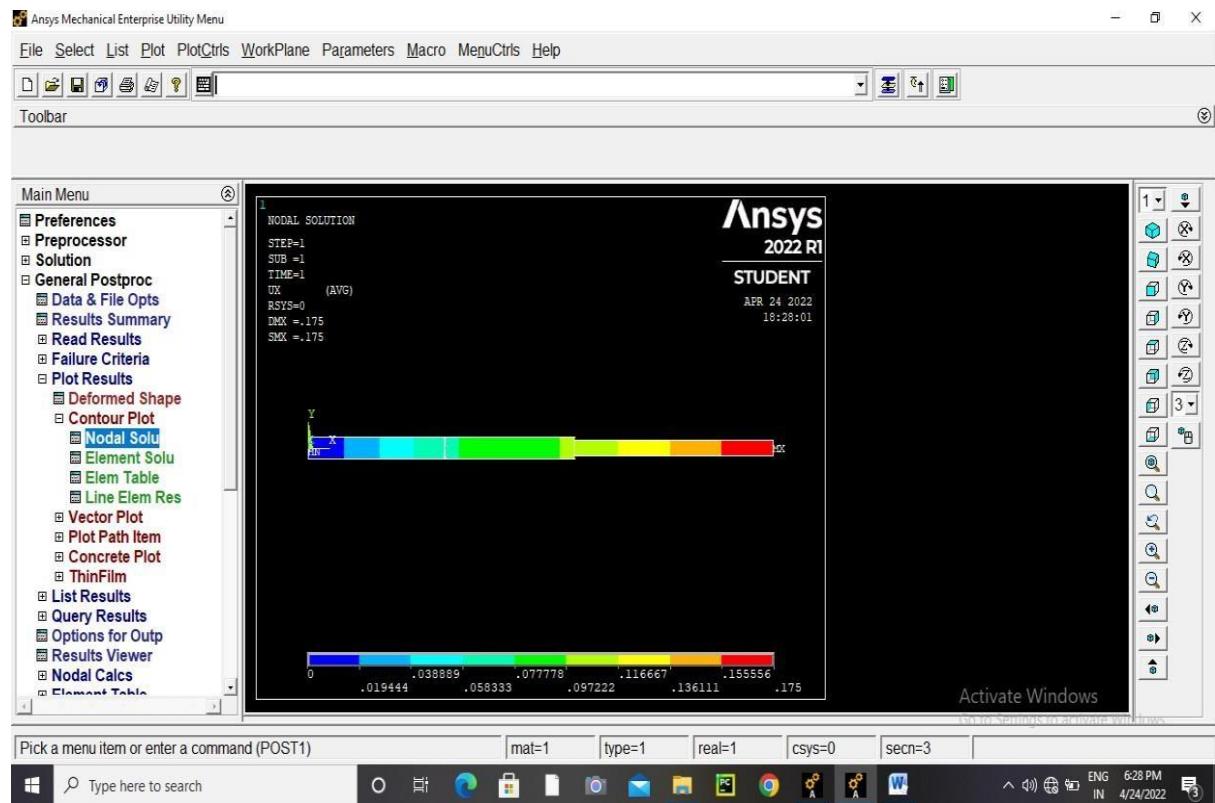


14. To See Actual step bar

PlotCtrls>Style>Size&Shape>Window will appear>Display Of element>>on



Result:-



Observation:-

	Displacement (mm)			Reaction (kN)	Stress (MPa)		
	d2	d3	d4	R1	In Element 1	In Element 2	In Element 3
Analytical Solution	0.075	0.1	0.1752	-30	150	50	100
Ansys Solution	0.075	0.1	0.1752	-30	150	50	100
Error(%)	00	00	00	00	00	00	00

Conclusion:-Thus,we have solve 1d bar element problem by analytical as well as by Ansys Software(APDL).By solving the problem with both the method it was observed that all the values are coming same. So, no error found.

NAME : SOURAV SANTAJI GUJALE

CLASS: TE MECH 1

SEMESTER/YEAR: 6

ROLL NO.: 61

DATE OF PERFORMANCE:

DATE OF SUBMISSION:

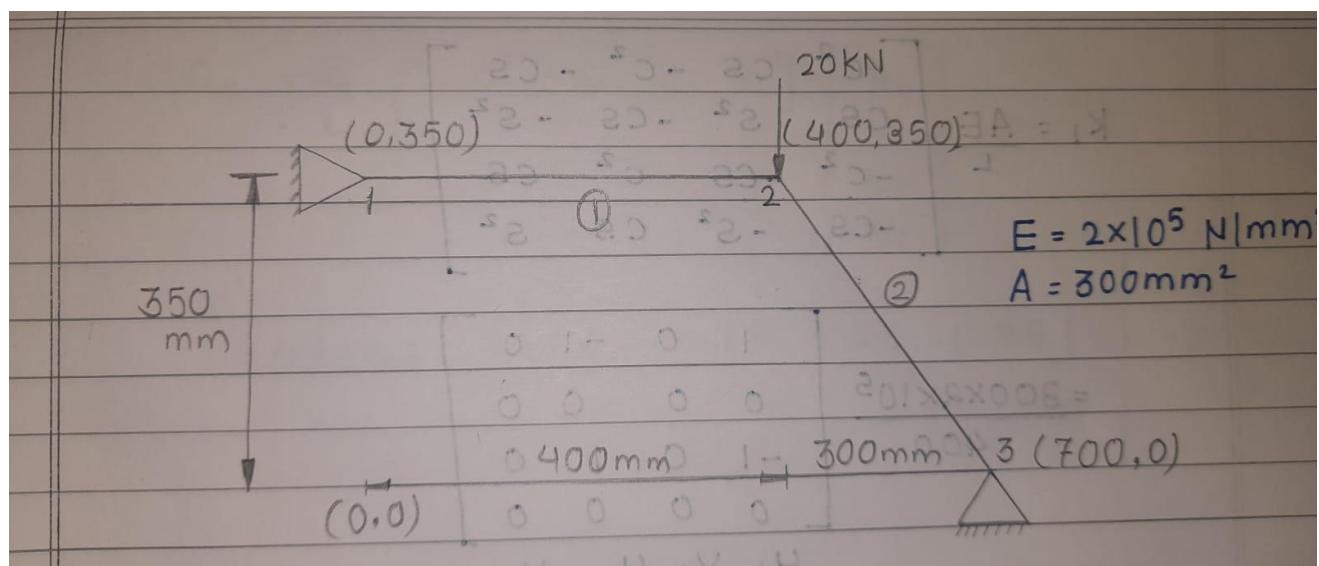
EXAMINED BY:

EXPERIMENT NO: 2

AIM OF EXPERIMENT:-Stress and deflection analysis of truss using finite element package.

Finite Element Package: ANSYS 2022

Stress distribution in truss with applied load.

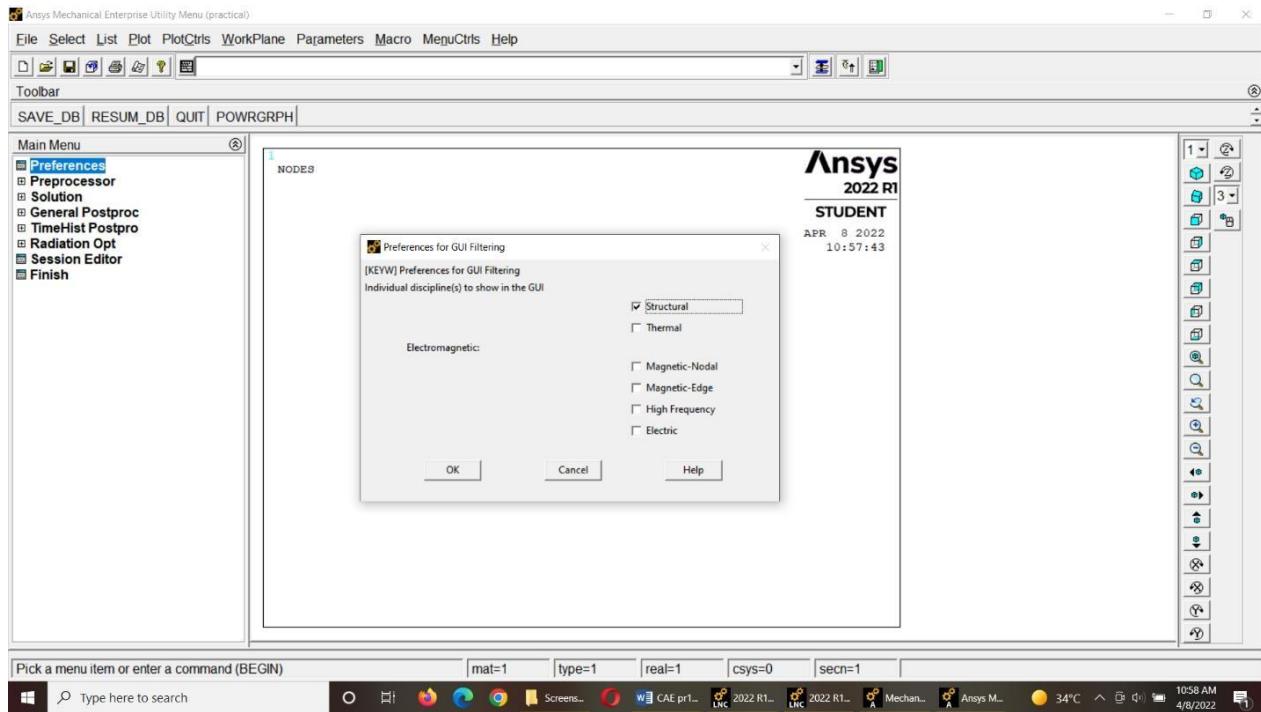


$$E = 2 \times 10^5 \text{ MPa}$$

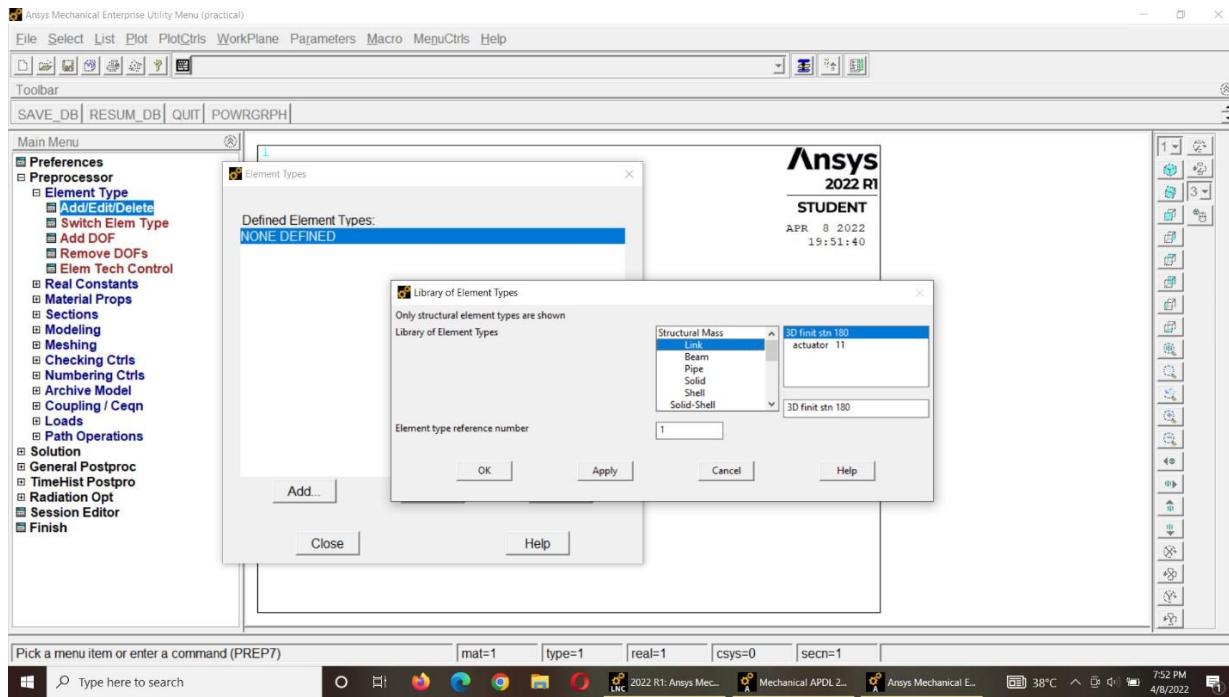
$$R = 25 \text{ mm}$$

$$u = 0.3$$

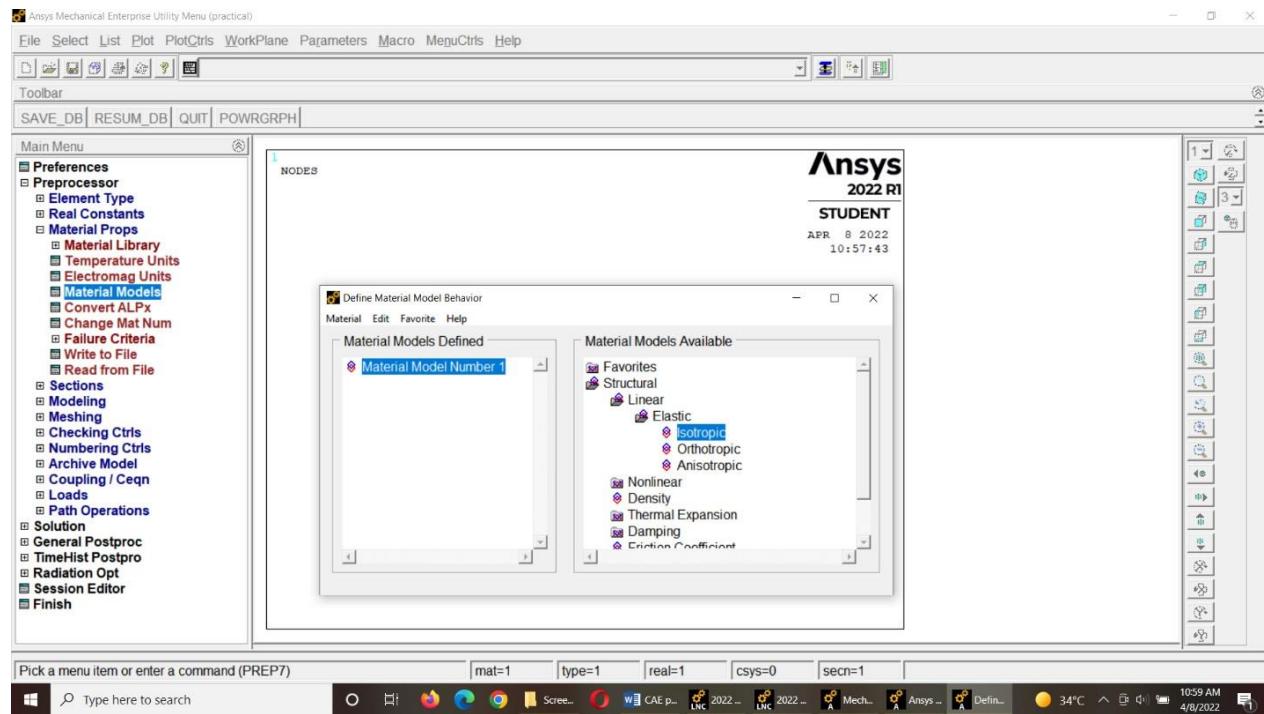
Step 1: Select type of Analysis---- Preferences> structural>Press Ok



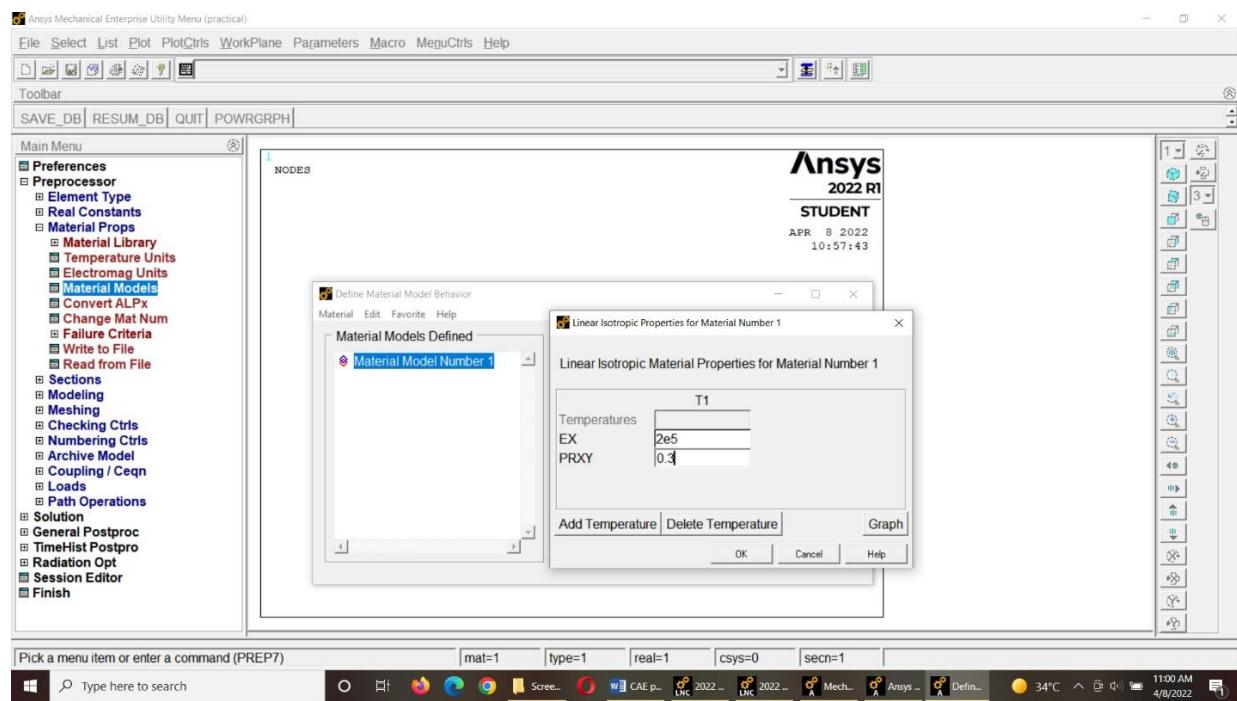
Step 2: Add the element type.....preprocessor>element type>link>3D finit 180> Press ok



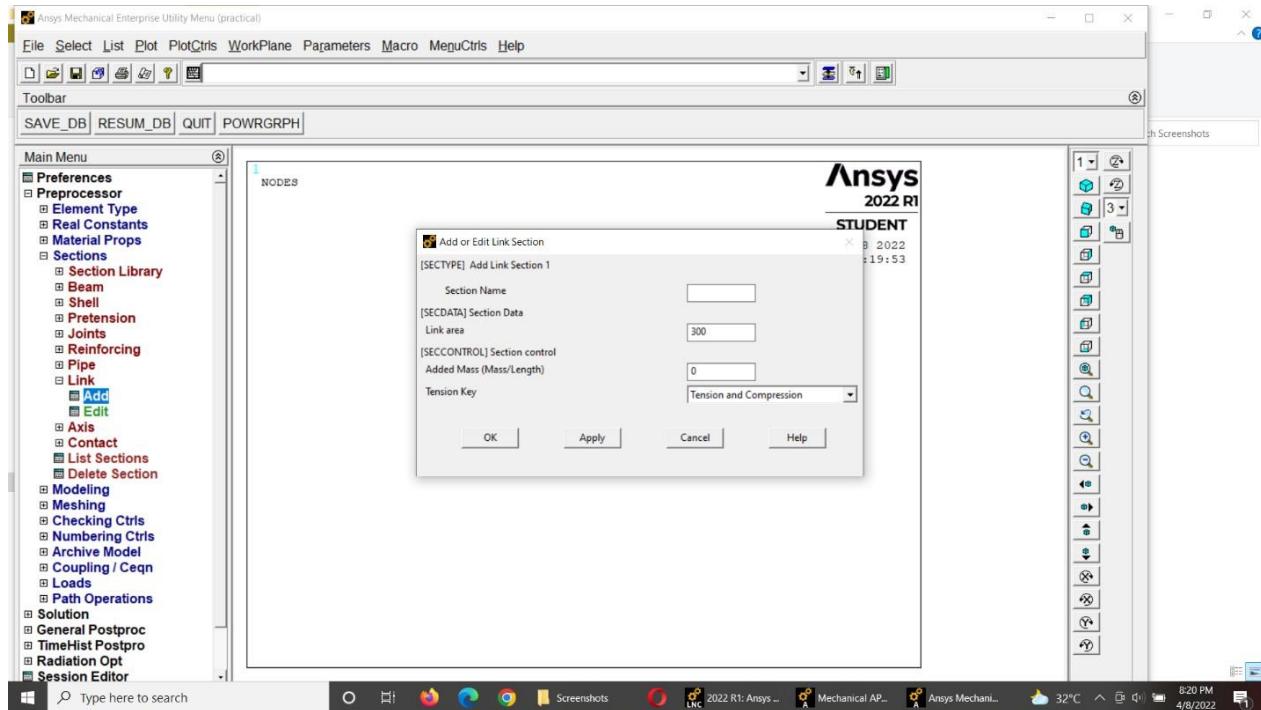
Step 3: Preprocessor>Material prop.>Material models>Material number1



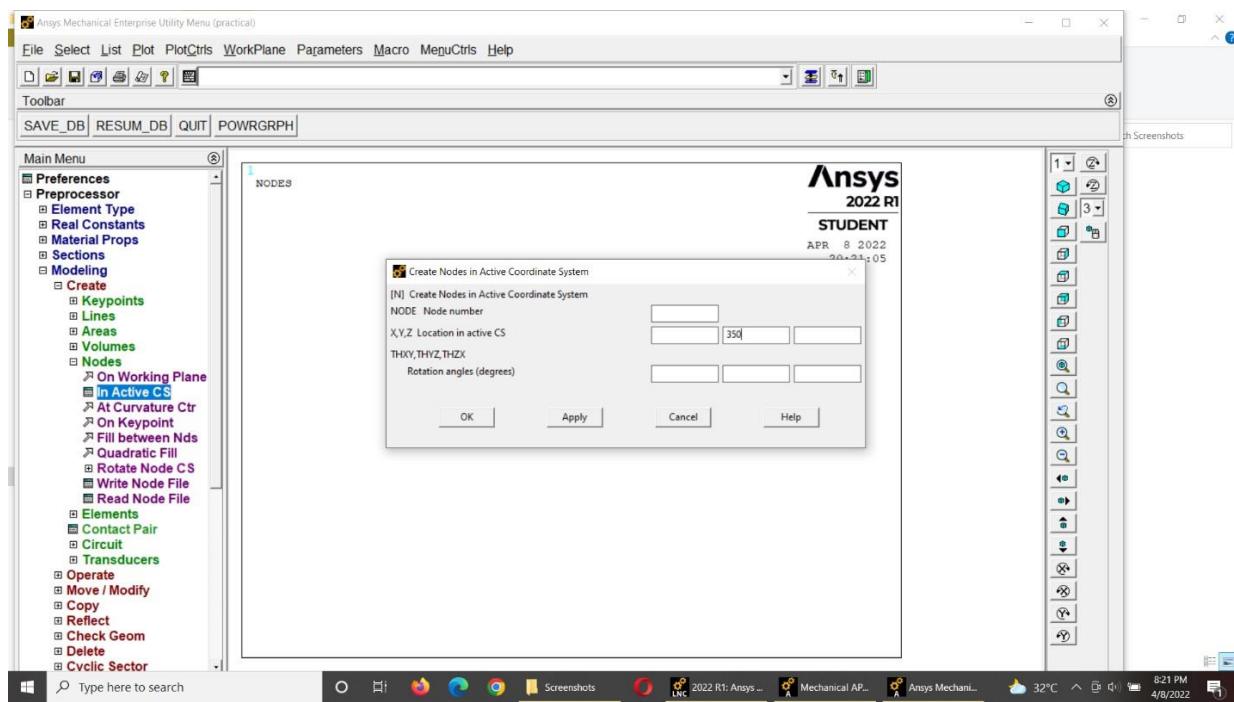
Step 4: Material models>structural>linear>elastic>isotropic.



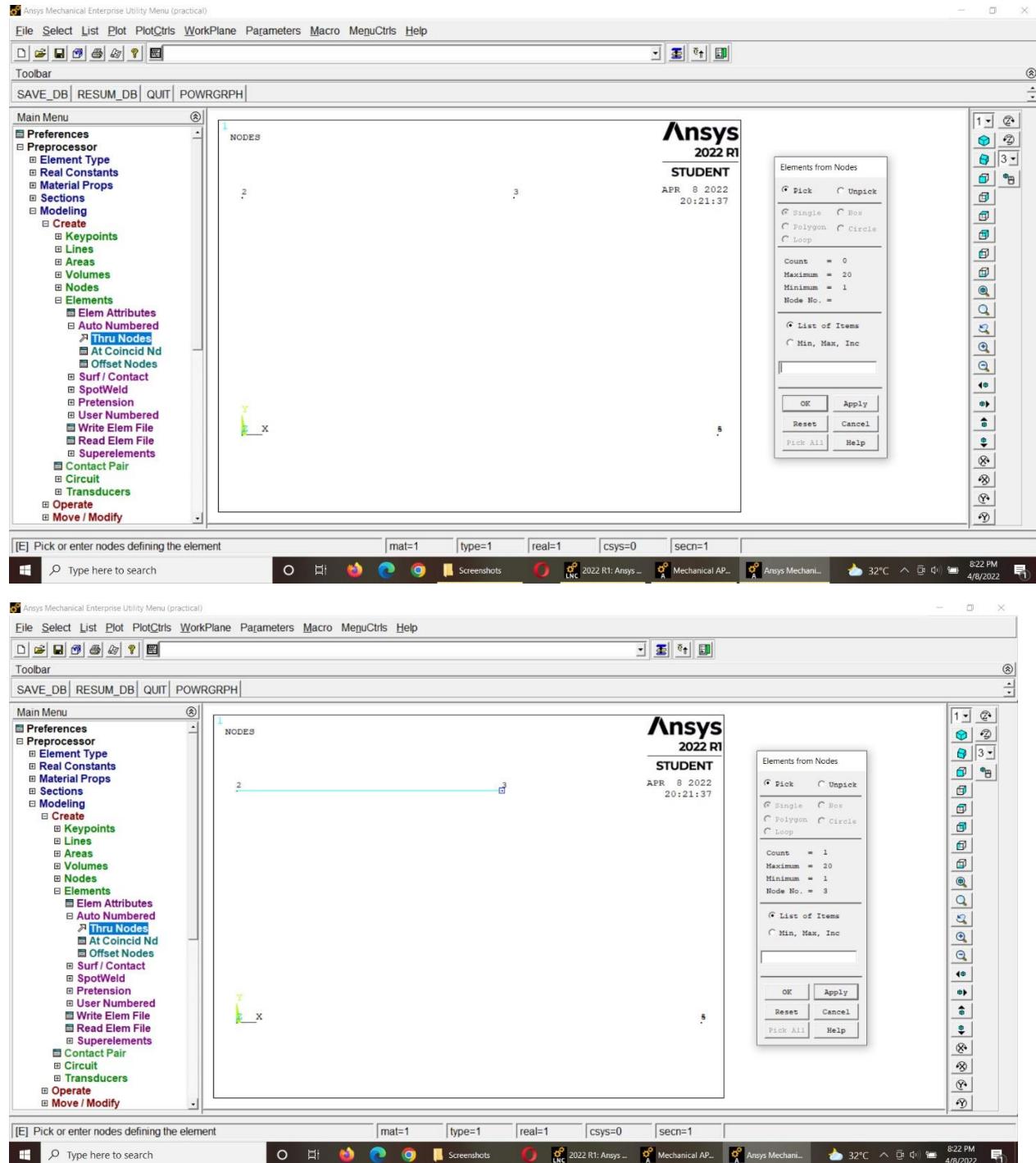
Step 5: selecting section of link....section>link>add>apply> type area of link> ok.



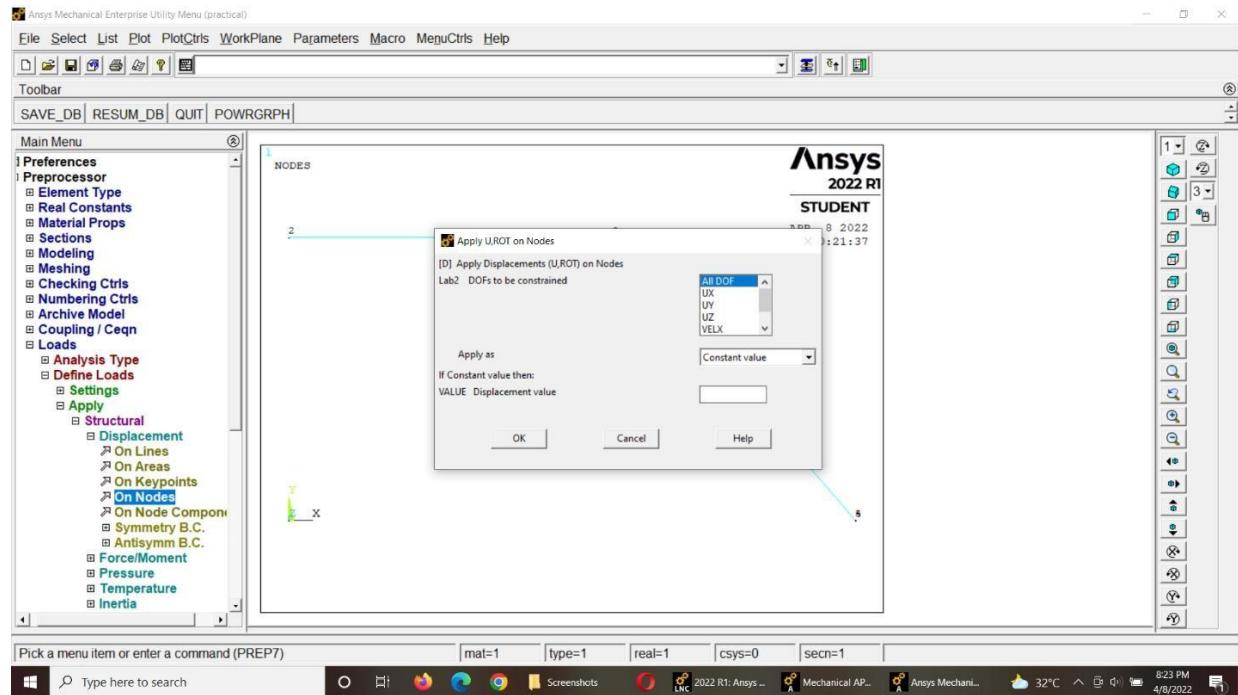
Step6: Creating Keypoints:- modeling>create>nodes>in active cs>select coordinate.



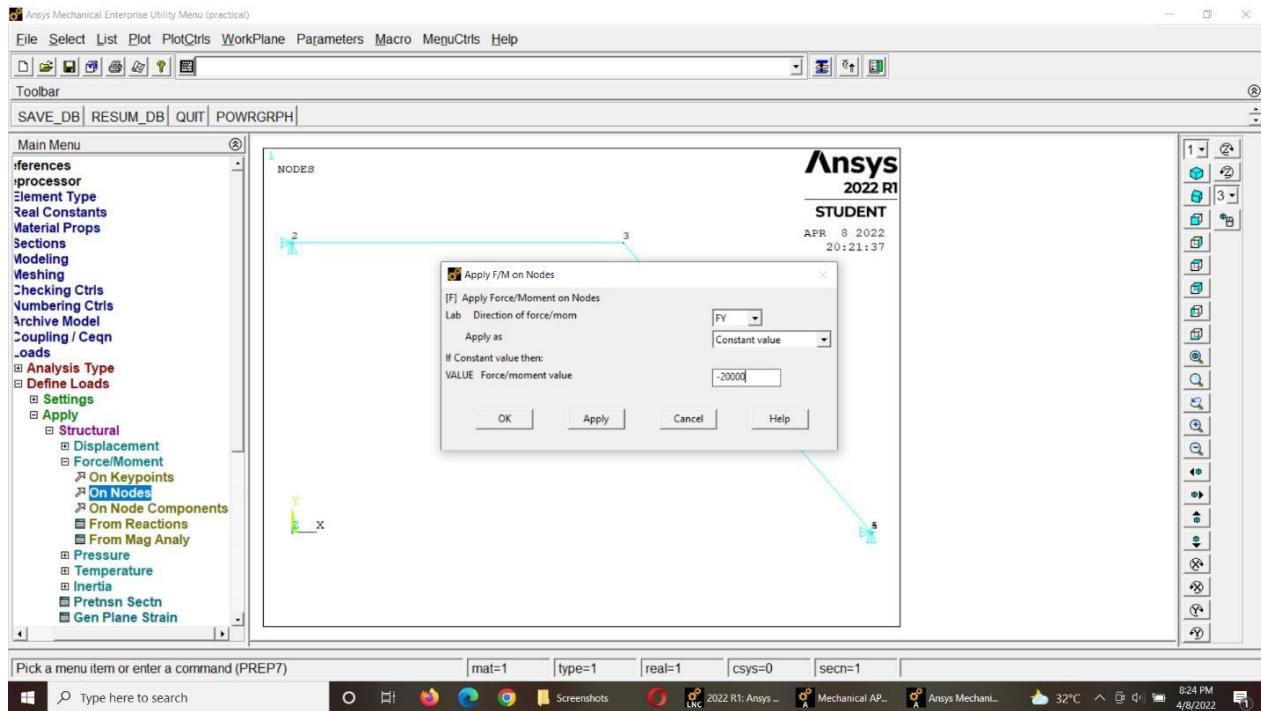
Step7: modeling>create>elements>auto numbered>thru nodes>select nodes one by one joining.



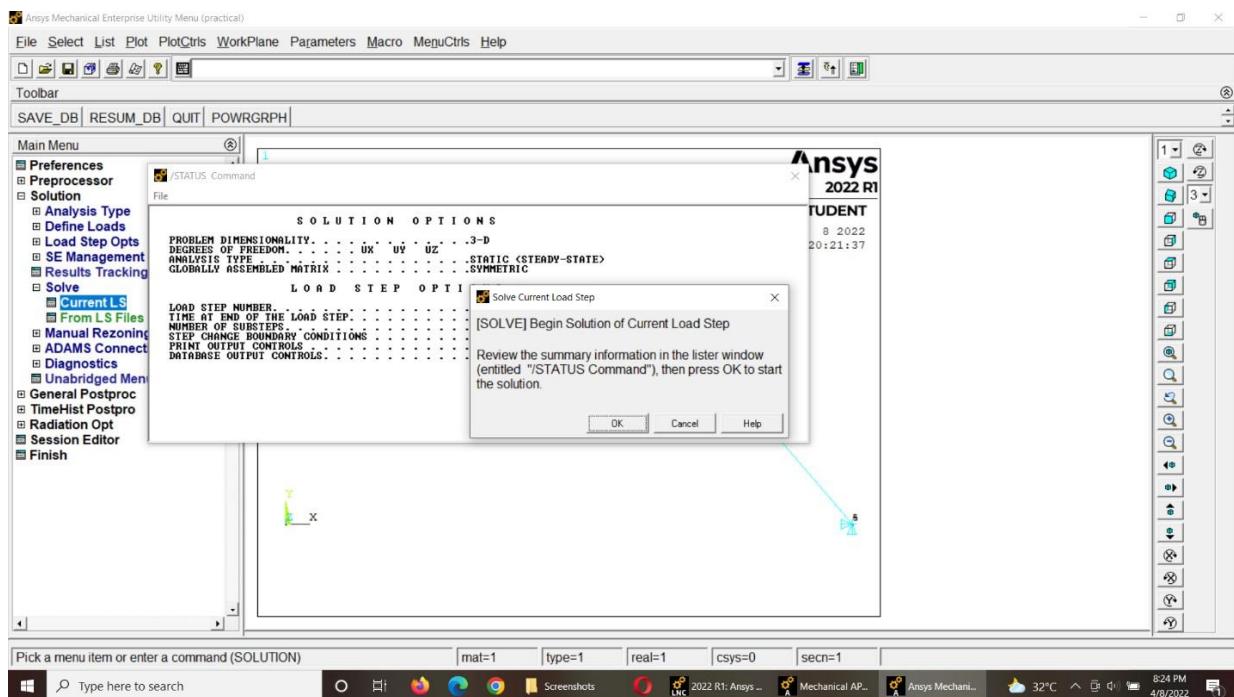
Step8: Apply loads: Laods>define loads>apply>structural>displacement>on nodes> All Dof>0>ok



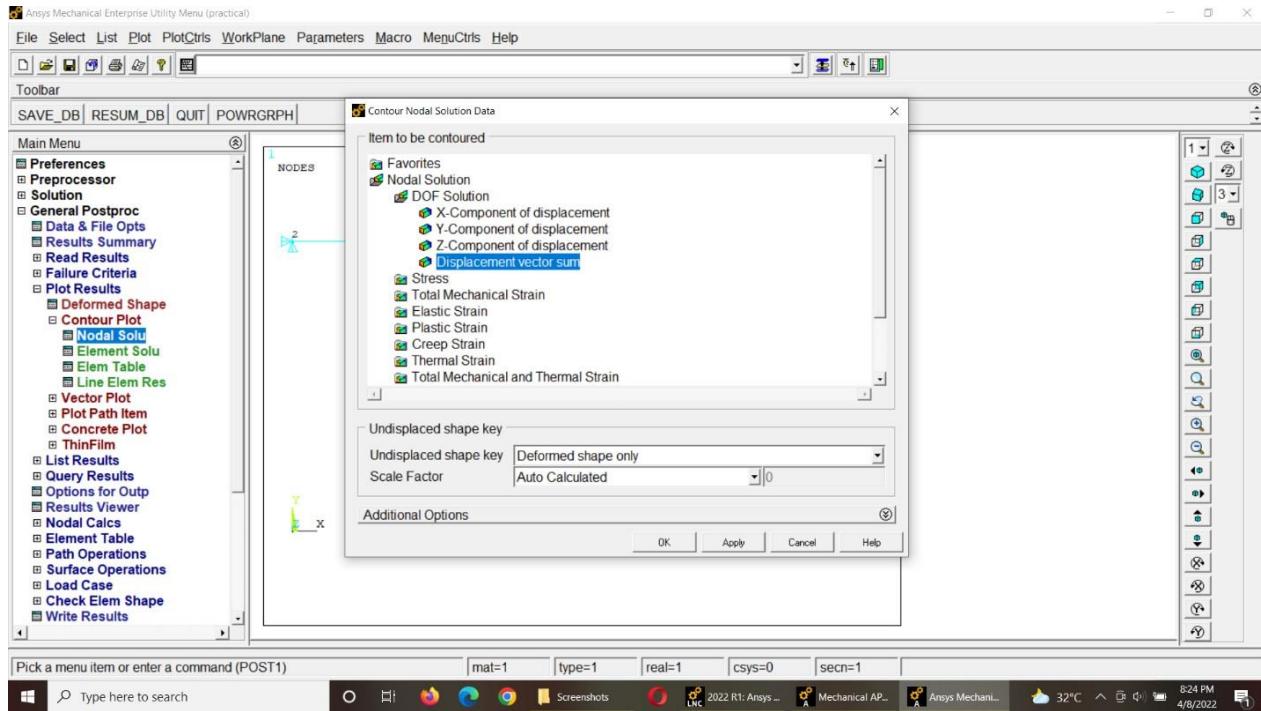
Step11: Loads>Define loads>apply>forces>on keypoints> selecting direction of forces (here FY in -ve)>ok



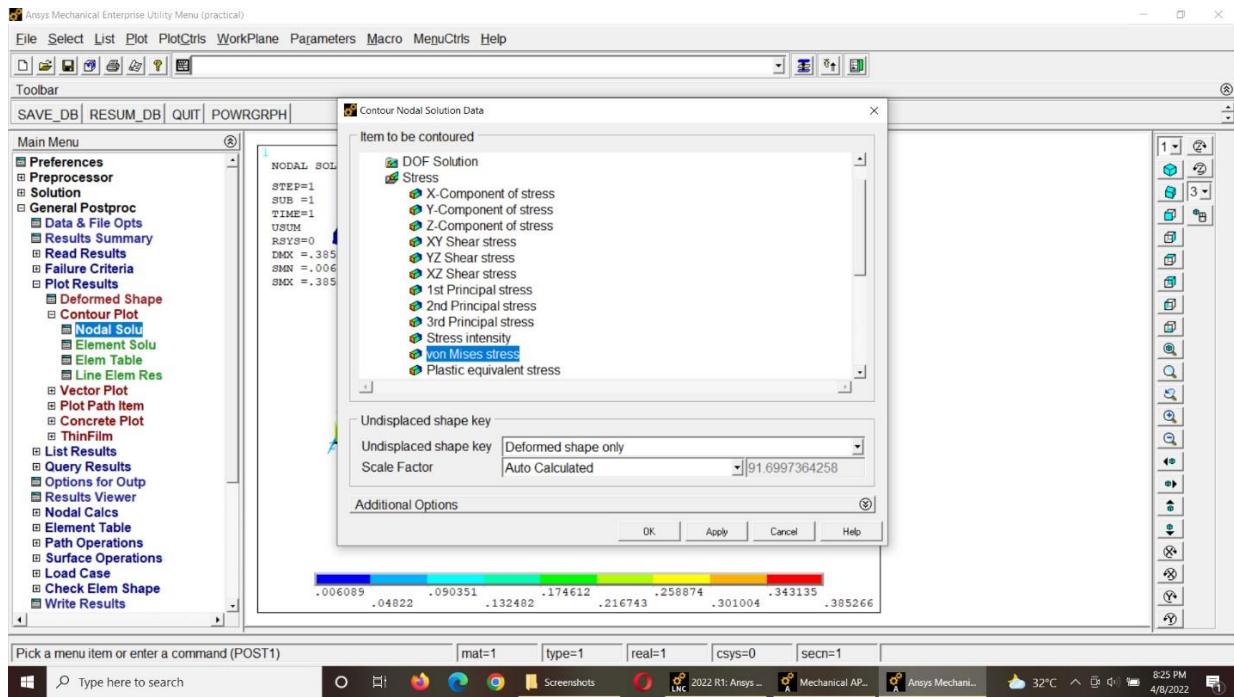
Step12:Solution:- solution>solve>currentls> done



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

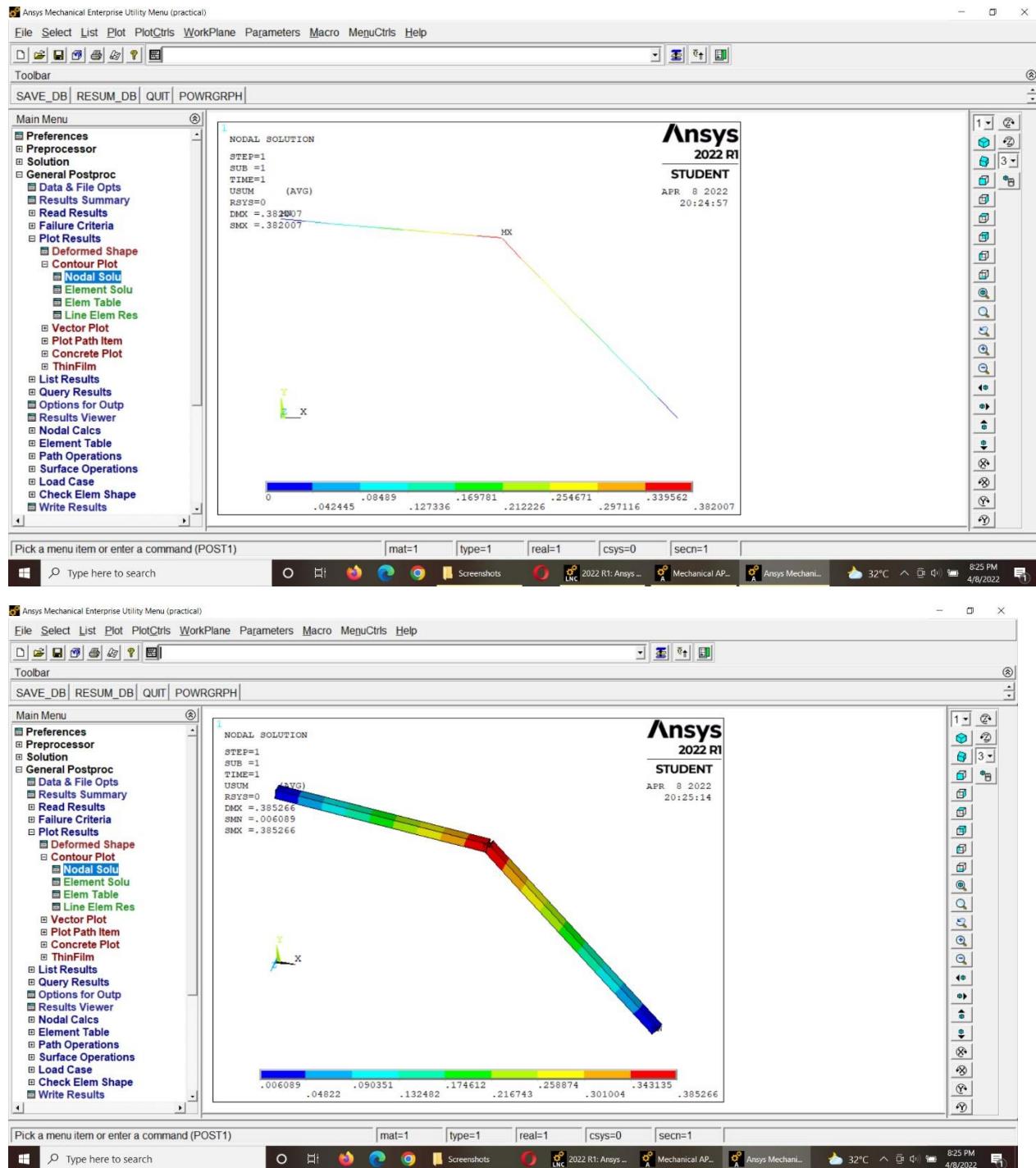


Step14: Nodal solution> stress> von misses stress> apply

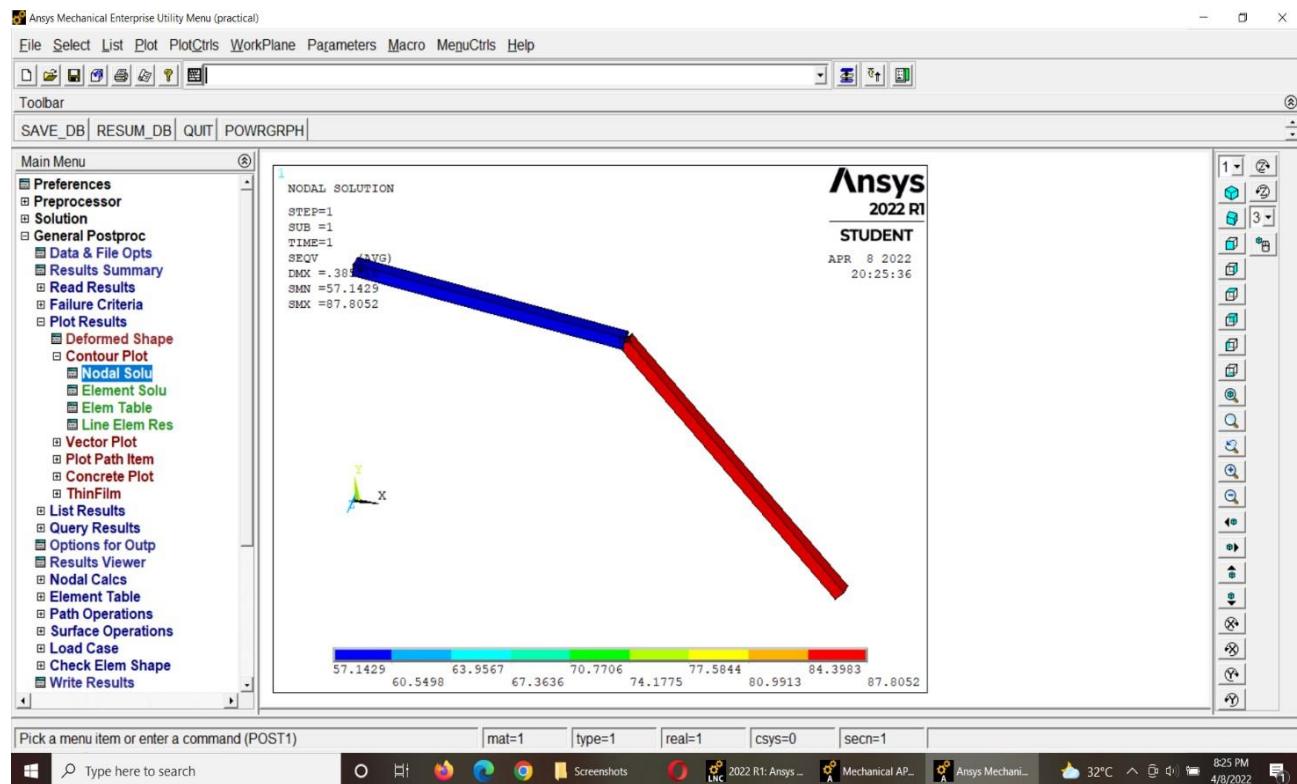


RESULTS:-

NODAL DISPLACEMENT:-



STRESSES:-

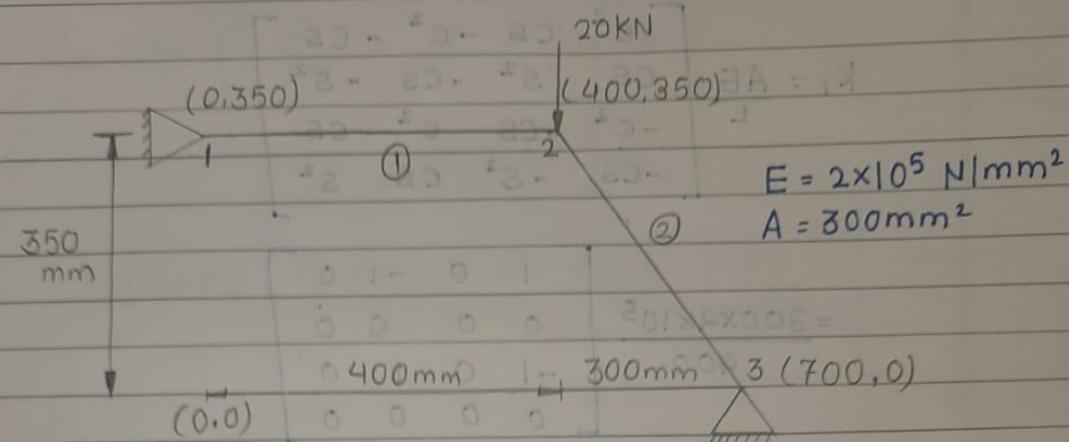


SO HERE BY ANALYSIS WE HAVE GOT MAX. INDUCED STRESS IS **87.8052N/MM²**

MIN. STRESSES ARE **57.14N/MM²** AND MAXIMUM DEFORMATION IS **0.3852MM**

BY analytical solution:-

Page No.	
Date	



Element	Node 1	length	$\cos\theta$	$\sin\theta$	$\cos\theta \cdot \sin\theta$
1	1-2	400	1	0	0
2	2-3				

$$C_1 = \frac{x_2 - x_1}{l_e} = \frac{400 - 0}{400} = 1$$

$$S_1 = \frac{y_2 - y_1}{l_e} = \frac{350 - 350}{400} = 0$$

$$C_2 = \frac{x_3 - x_2}{l_e} = \frac{700 - 400}{461} = 0.65$$

$$S_2 = \frac{y_3 - y_2}{l_e} = \frac{0 - 350}{461} = -0.75$$

$$C_1^2 = 1, S_1^2 = 0, CS = 0$$

$$C_2^2 = 0.42, S_2 = 0.56, CS = 0.48$$

$$K_1 = \frac{AE}{L} \begin{bmatrix} C^2 & CS & -C^2 & -CS \\ CS & S^2 & -CS & -S^2 \\ -C^2 & -CS & C^2 & CS \\ -CS & -S^2 & CS & S^2 \end{bmatrix}$$

$$= \frac{300 \times 2 \times 10^5}{400} \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

u_1, v_1, u_2, v_2

$$= 10^4 \begin{bmatrix} 15 & 0 & -15 & 0 \\ 0 & 0 & 0 & 0 \\ -15 & 0 & 15 & 0 \\ 1 & 0 & 0 & 0 \end{bmatrix} \begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{bmatrix}$$

$$= 10^4 \begin{bmatrix} 20.46 & 6.24 \\ 6.24 & 7.28 \end{bmatrix} \begin{bmatrix} u_2 \\ v_2 \end{bmatrix} = \begin{bmatrix} 0 \\ -20 \times 10^3 \end{bmatrix}$$

$$u_2 = -0.1121 \text{ mm}$$

$$v_2 = -0.370 \text{ mm}$$

$$G_1 = \frac{E}{L} \begin{bmatrix} -C & -S & CS \end{bmatrix} \begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{bmatrix}$$

$$= \frac{2 \times 10^5}{400} \begin{bmatrix} -1 & 0 & 16.0 \\ 0 & 0 & 0 \end{bmatrix} \begin{bmatrix} 0 \\ 0 \\ -0.1121 \\ -0.370 \end{bmatrix}$$

$$= -56.05 \text{ N/mm}^2$$

Page No.	
Date	

$$G_2 = \frac{E}{L} \begin{bmatrix} -c & -s & c & s \end{bmatrix} \begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{bmatrix}$$

$$= 433.83 \begin{bmatrix} -0.65 & -0.75 & 0.65 & 0.75 \end{bmatrix} \begin{bmatrix} -0.1121 \\ -0.1890 \\ 0 \\ 0 \end{bmatrix}$$

$$= -87.47 \text{ N/mm}^2$$

CONCLUSION:-

Thus by comparing analytical and software solution we have got

Max. stresses:-

By ansys solution:- 87.8052N/MM²

By analytical solution:- 87.47 N/mm²

Max. displacement:-

By ansys solution:- 0.3852MM

By analytical solution:- 0.370 mm

Thus we have got 1% & 4% error respectively in stress and displacement.

NAME: SOURAV SANTAJI GUJALE

CLASS: TE MECH 1

SEMESTER/YEAR: 6

ROLL NO.: 61

DATE OF PERFORMANCE:

DATE OF SUBMISSION:

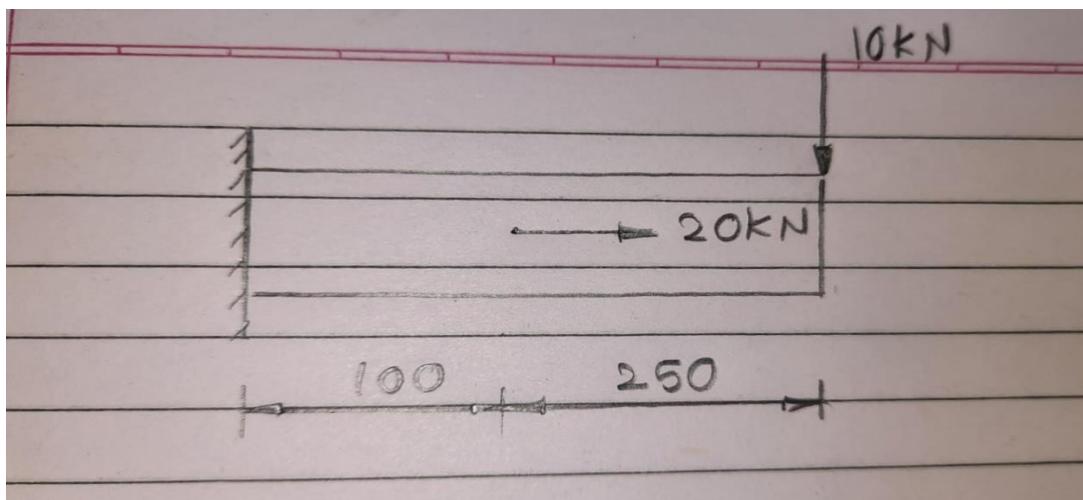
EXAMINED BY:

EXPERIMENT NO: 3

AIM OF EXPERIMENT: -Stress and deformation analysis of shell element using finite element package.

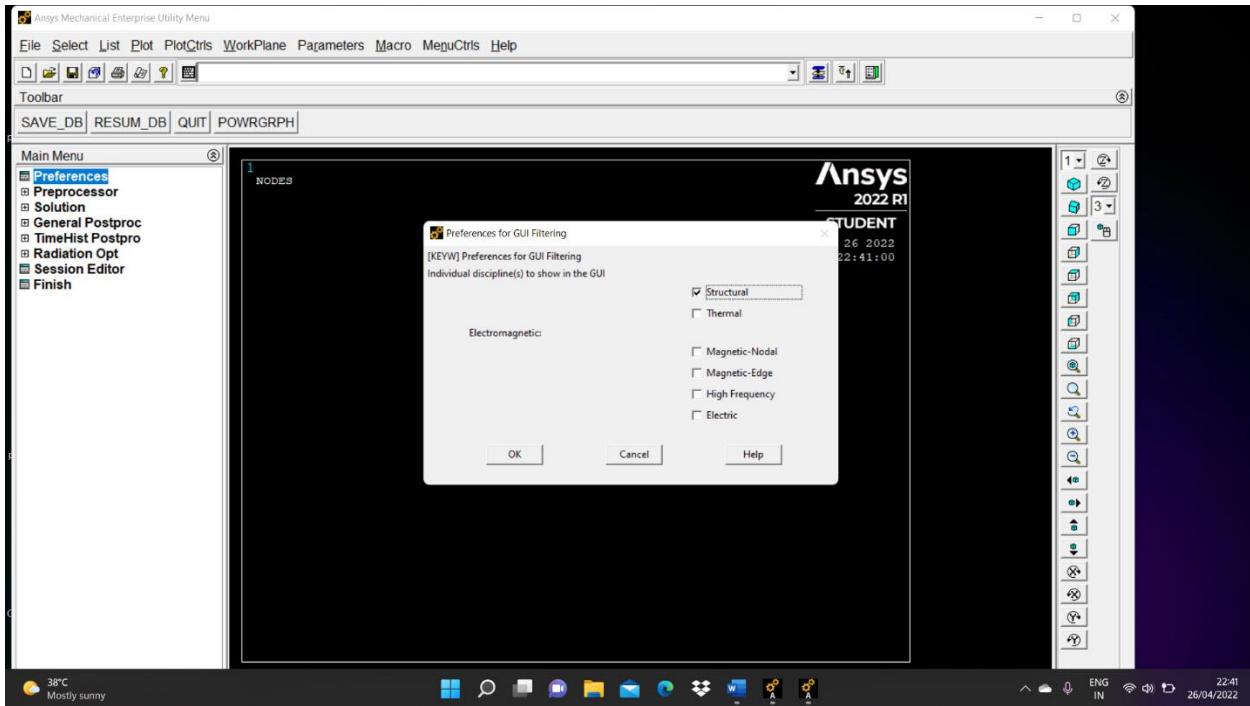
Finite Element Package: ANSYS 2022

Stress distribution in a shell with applied load.

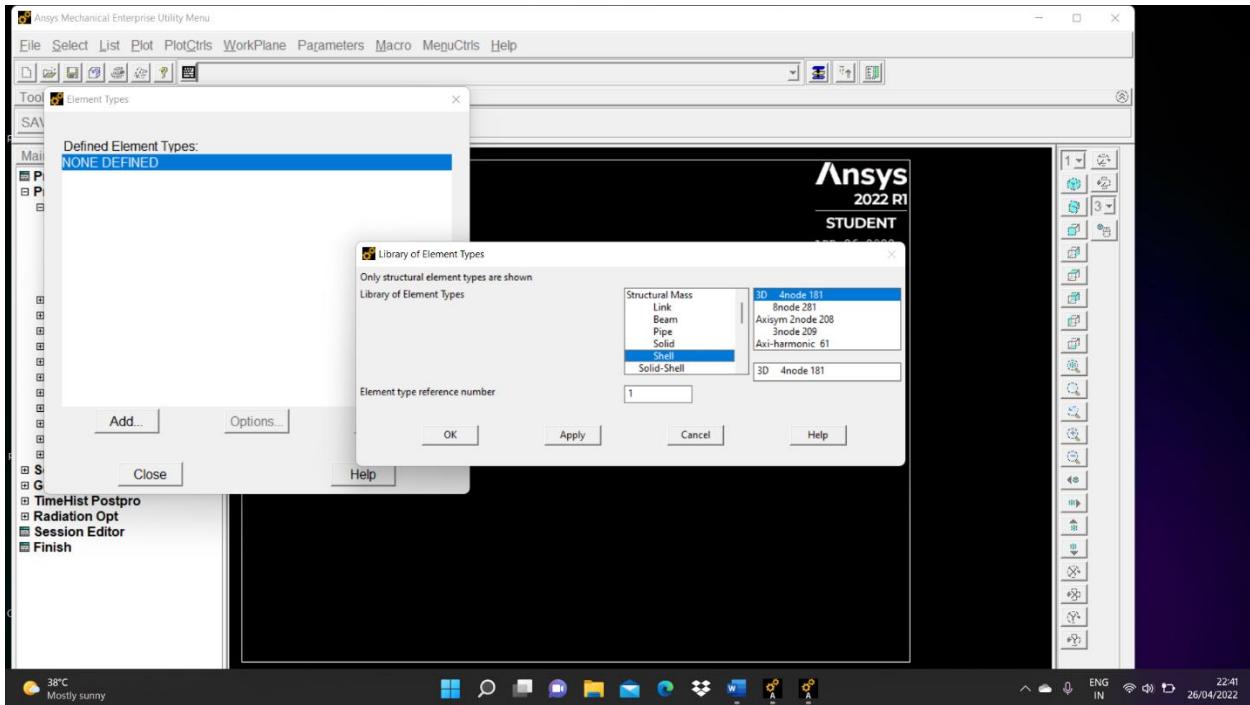


$$E = 2 \times 10^5 \text{ MPa} \quad u = 0.3$$

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

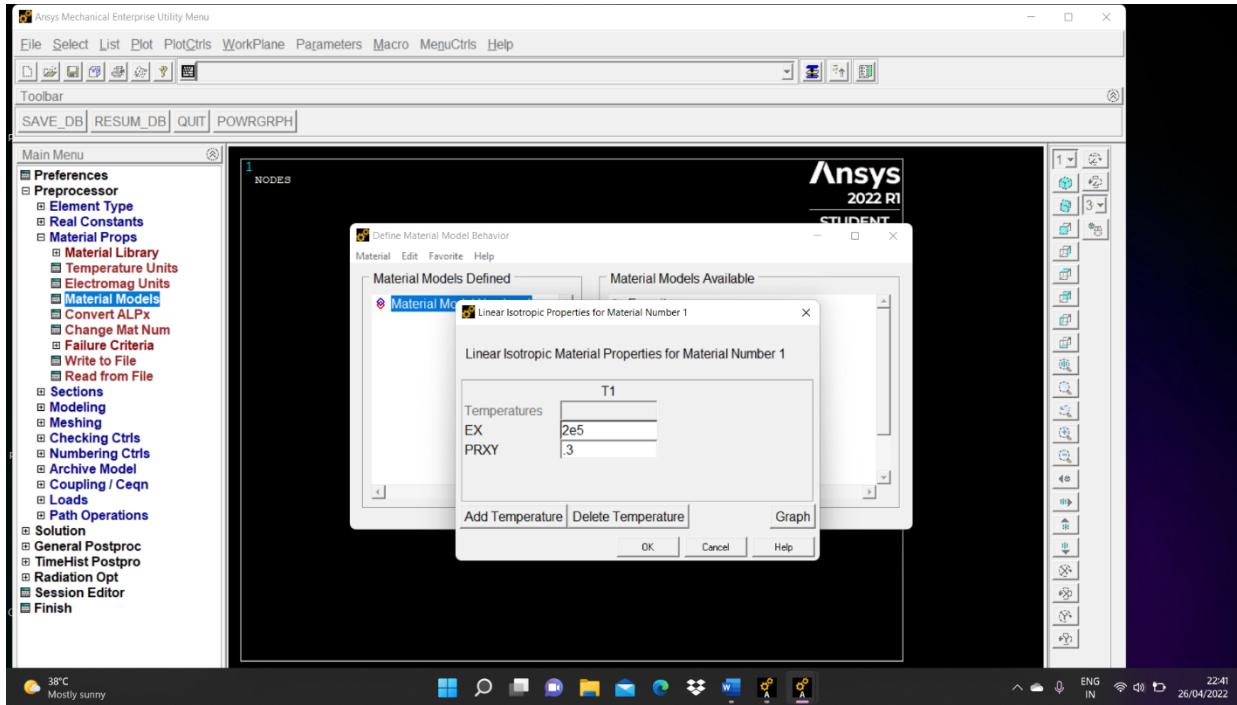


Step 2: Add the element type.....preprocessor>element type>shell>4node 181> Press ok

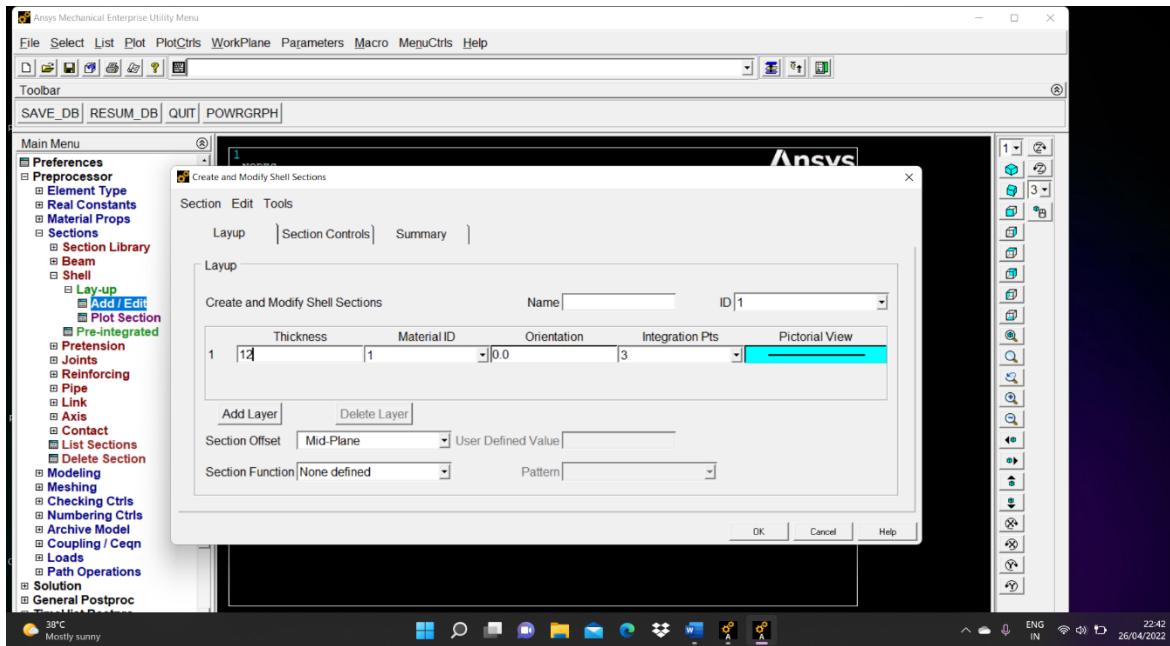


Step 3: Preprocessor>Material prop.>Material models>Material number1

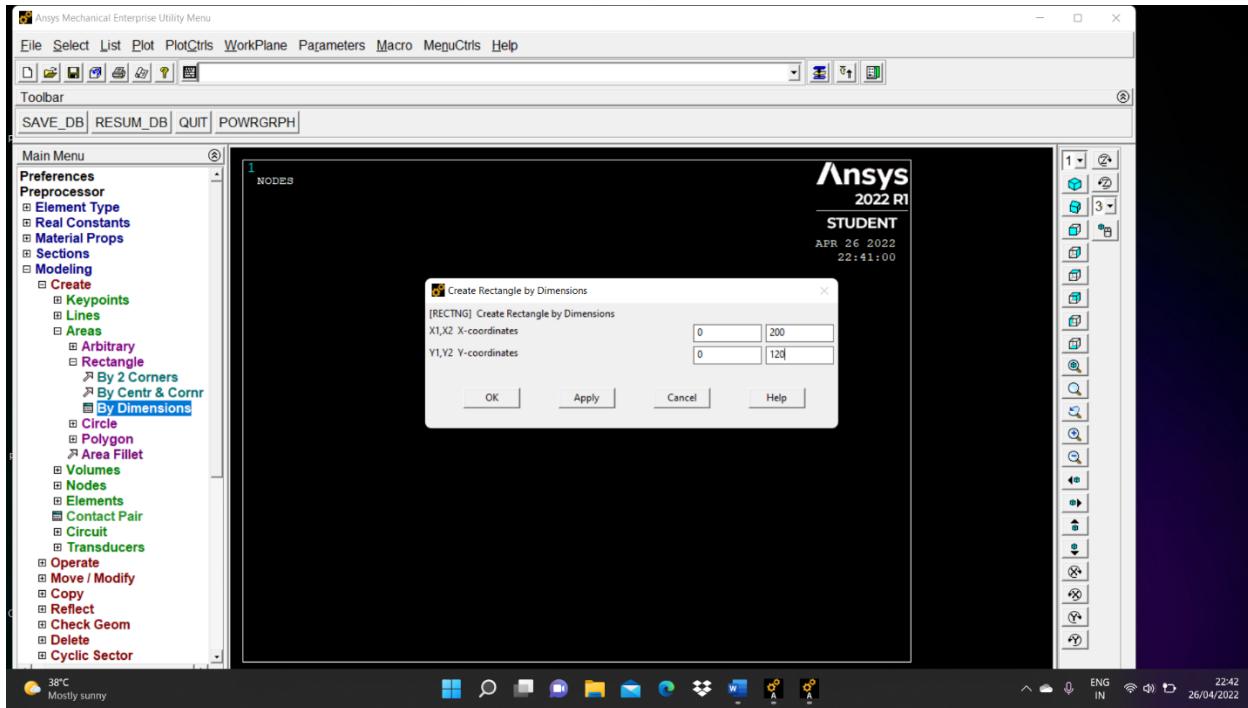
Step 4: Material models>structural>linear>elastic>isotropic.



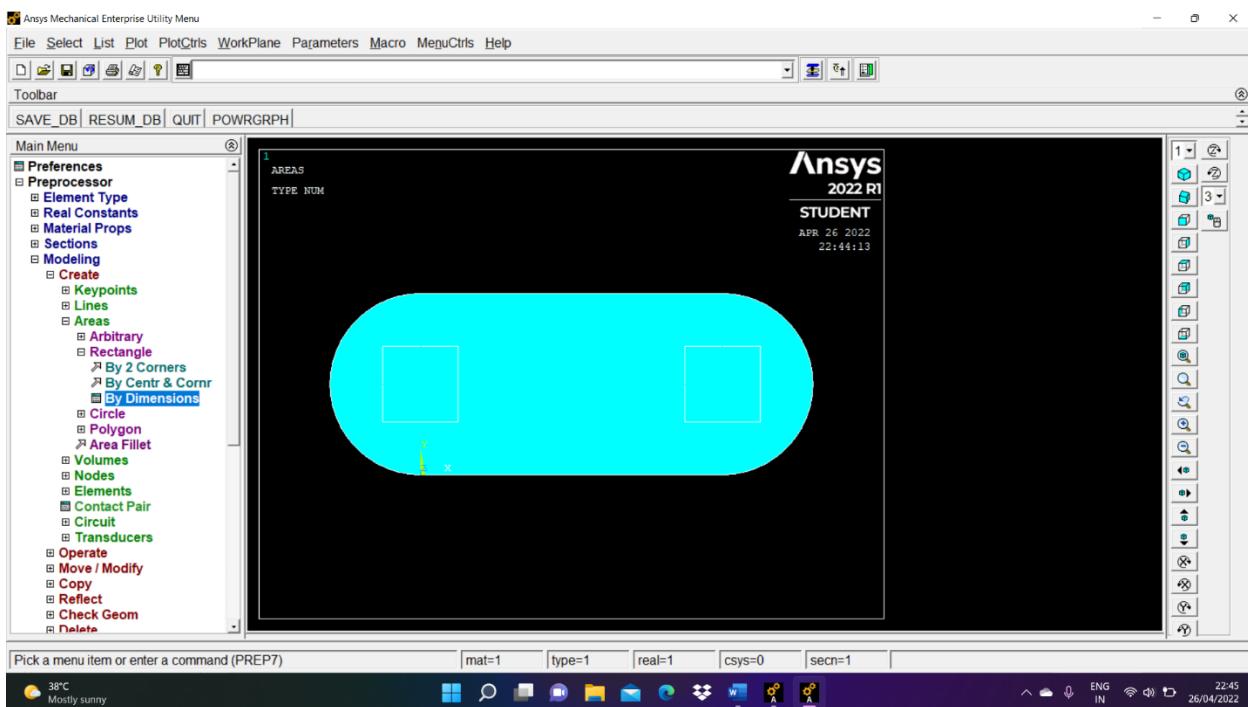
Step 5: selecting section of shell....section>shell>lay up>add /Edit>select thickness of section.



Step6: Creating areas :- modeling>create>areas>rectangle>by dimension.

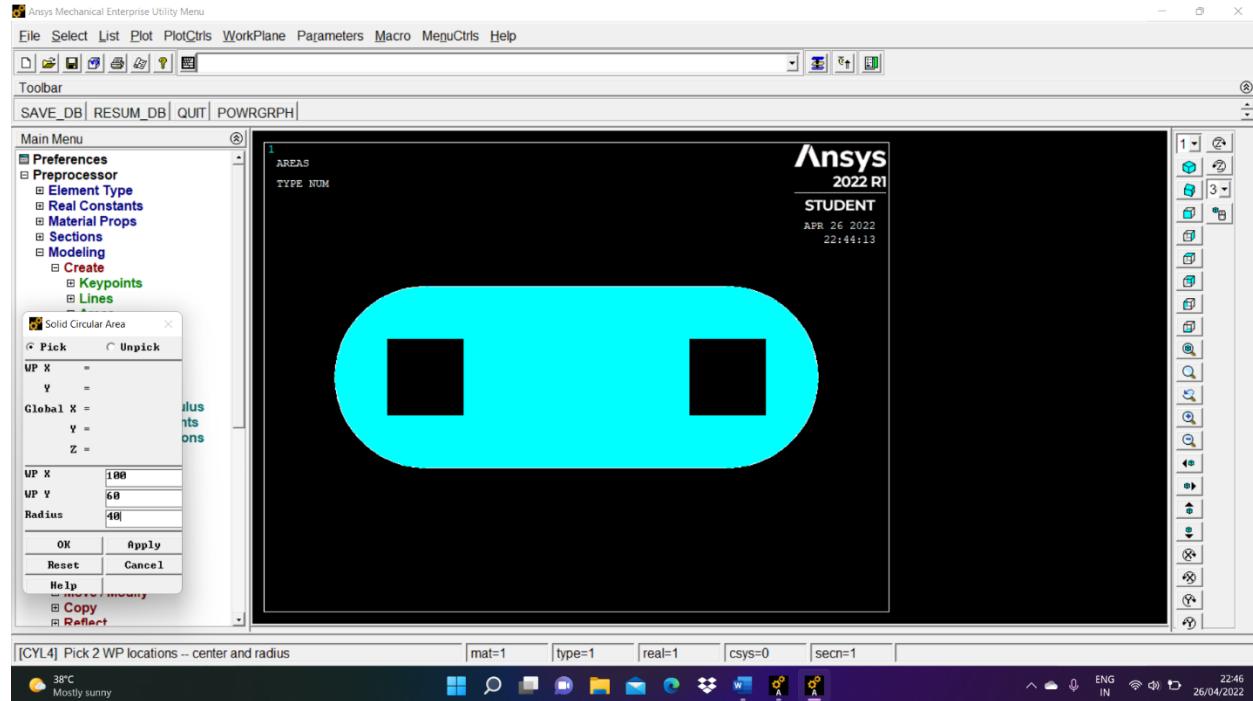


Step7: Creating areas:- modeling>create>areas>circle >solid circle> input co-ordinates and data

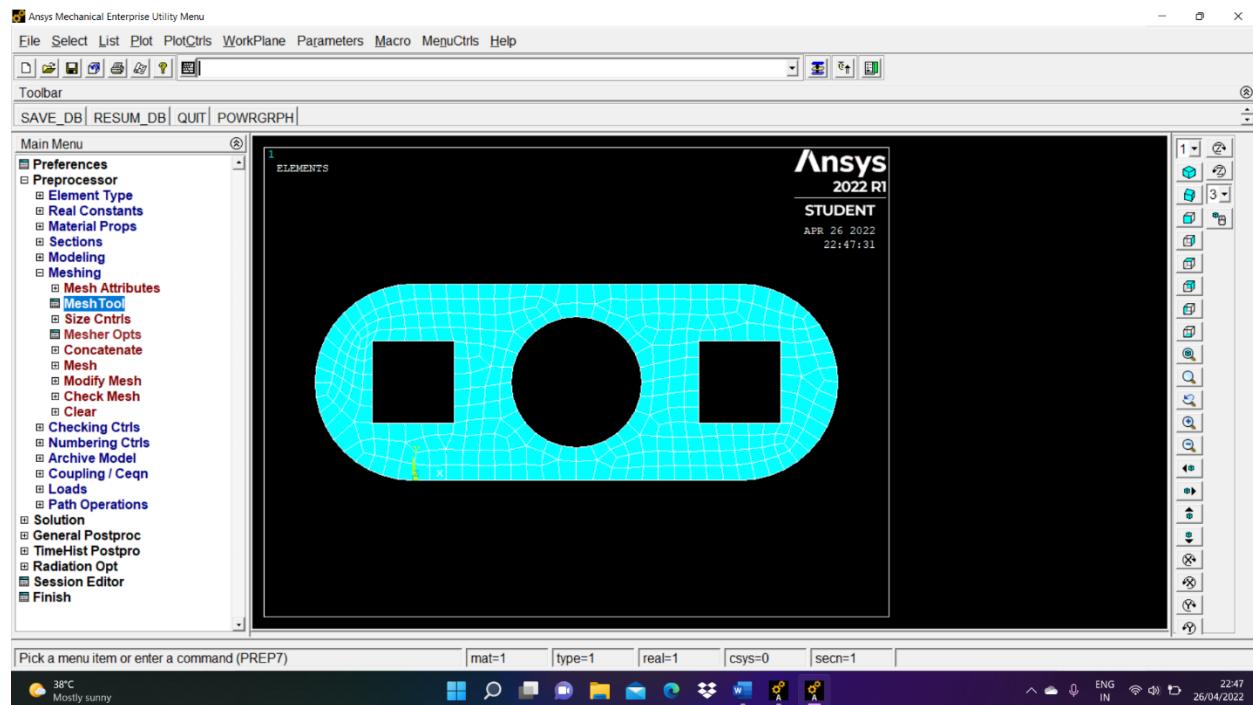


Step8: Creating holes:- modeling>create>areas>rectangle>by dimension.

Step9: subtracting holes: Modeling>operate>subtract>areas> select base area>ok> select subtracting area> ok> done

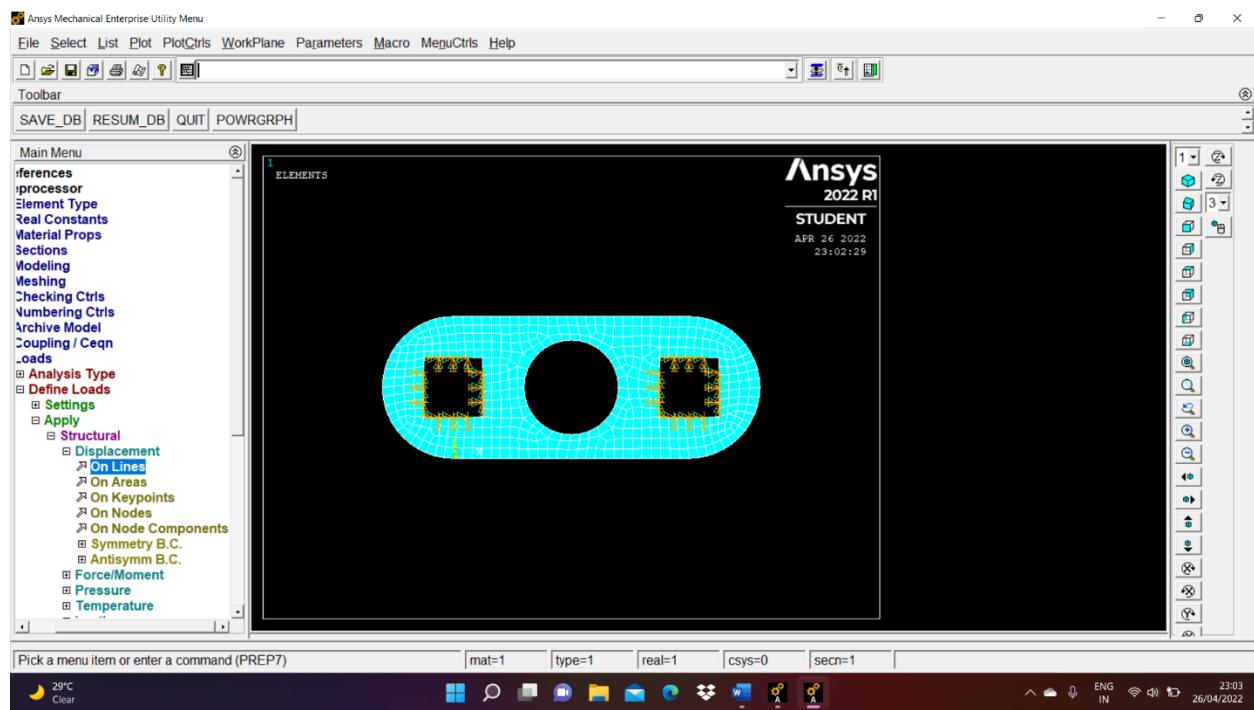


Step10: Same procedure for circular hole

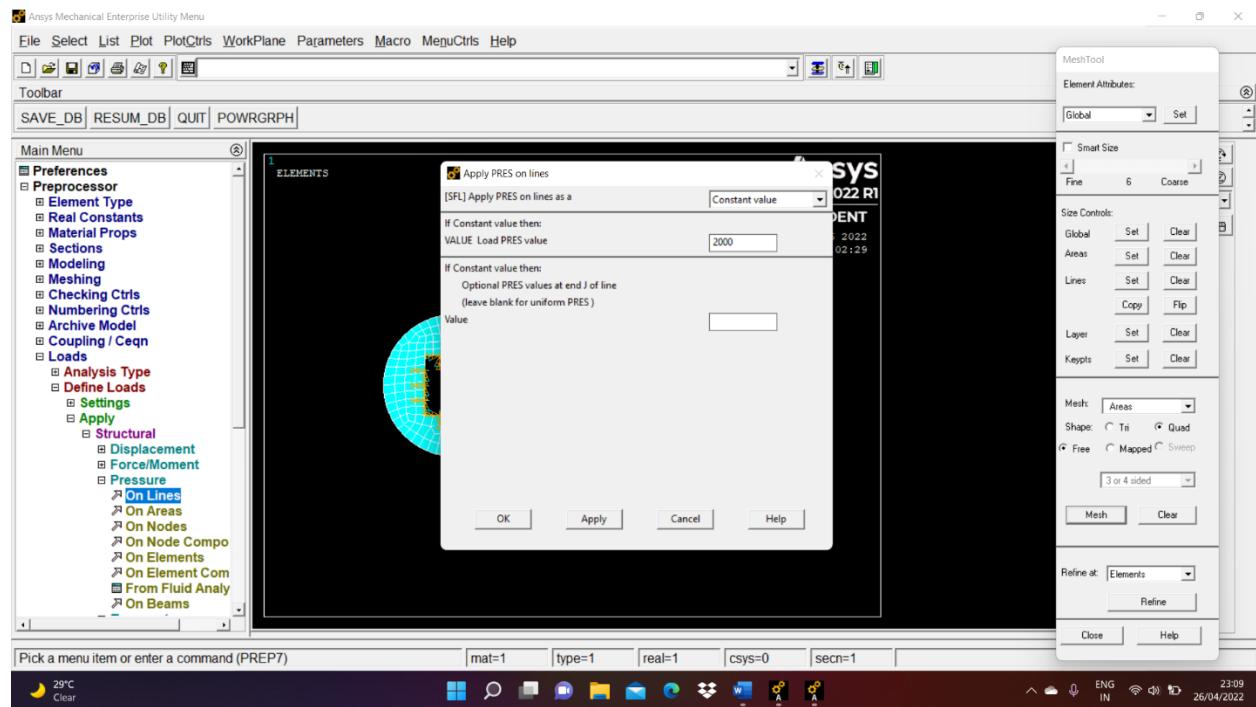


Step11: meshing the element: meshing>mesh tool> select areas>click on area>apply> slect mesh length>ok> mesh>ok

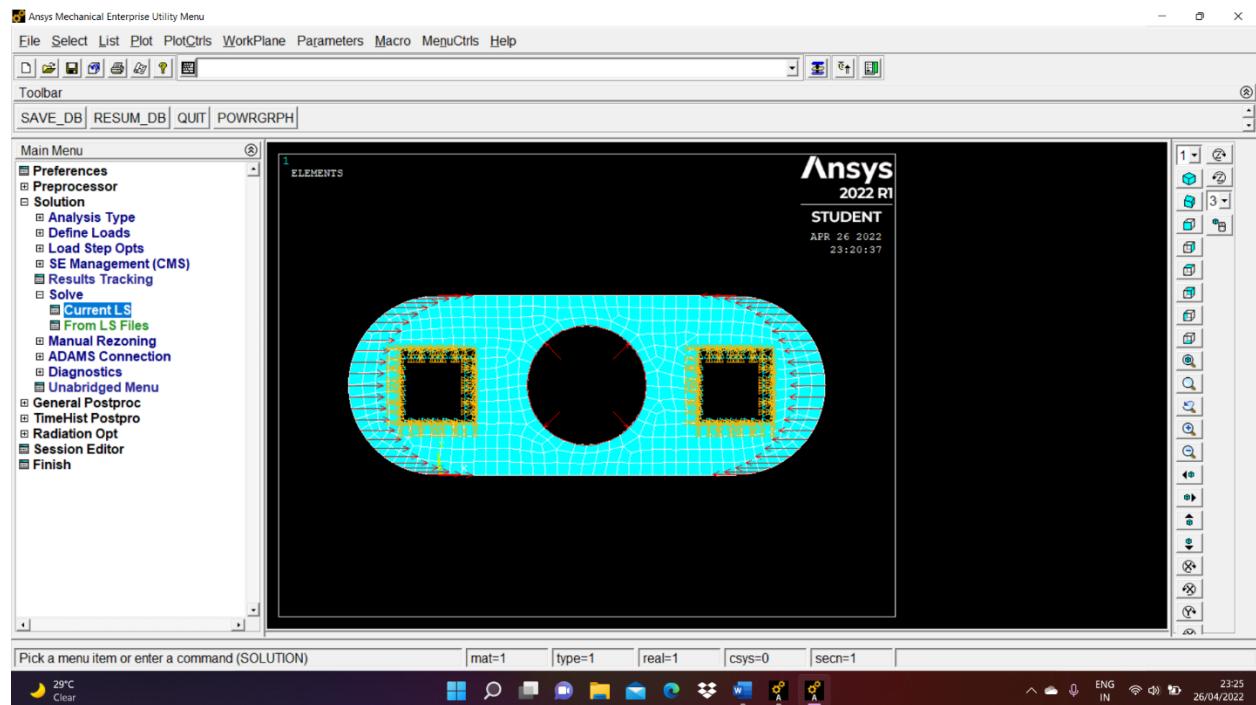
Step12:Loads: Define loads> apply>structural>displacement>on lines>All Dof>0>ok



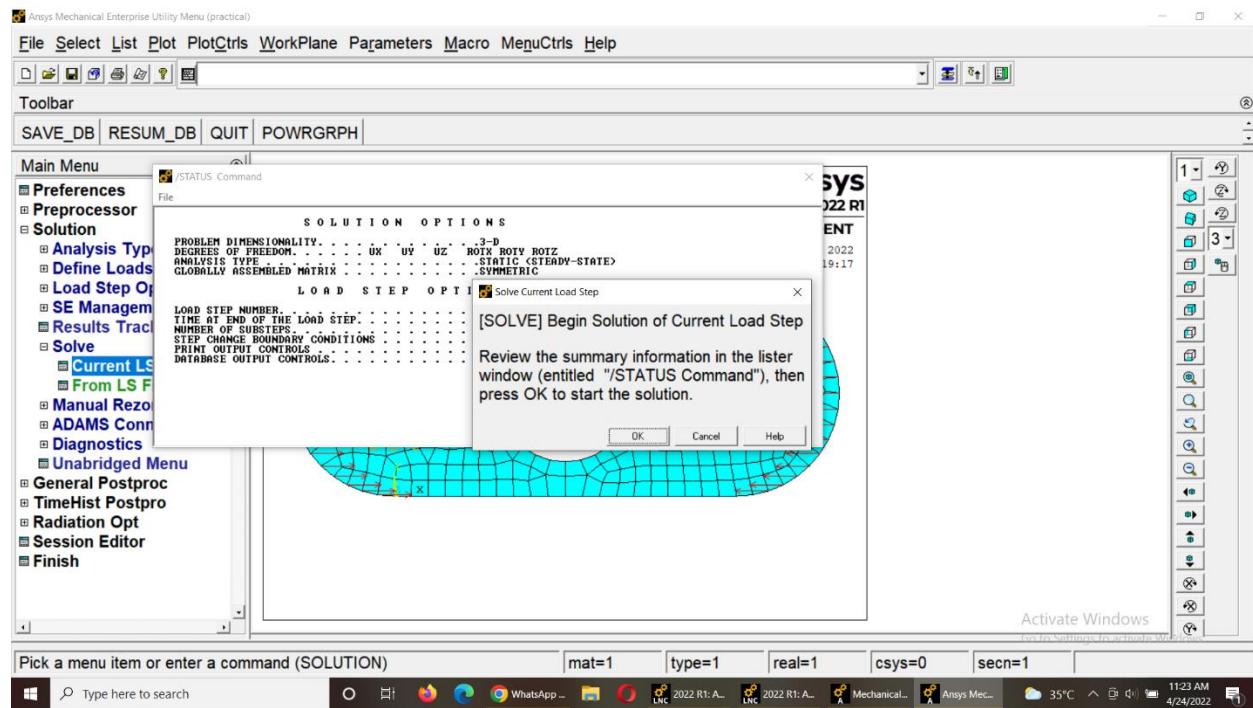
Step13: Loads: Define loads> apply>structural>pressure>on lines>put pressure value>2000>ok



Step14: Loads: Define loads> apply>structural>force>on nodes>FX>put force value>-1500>ok

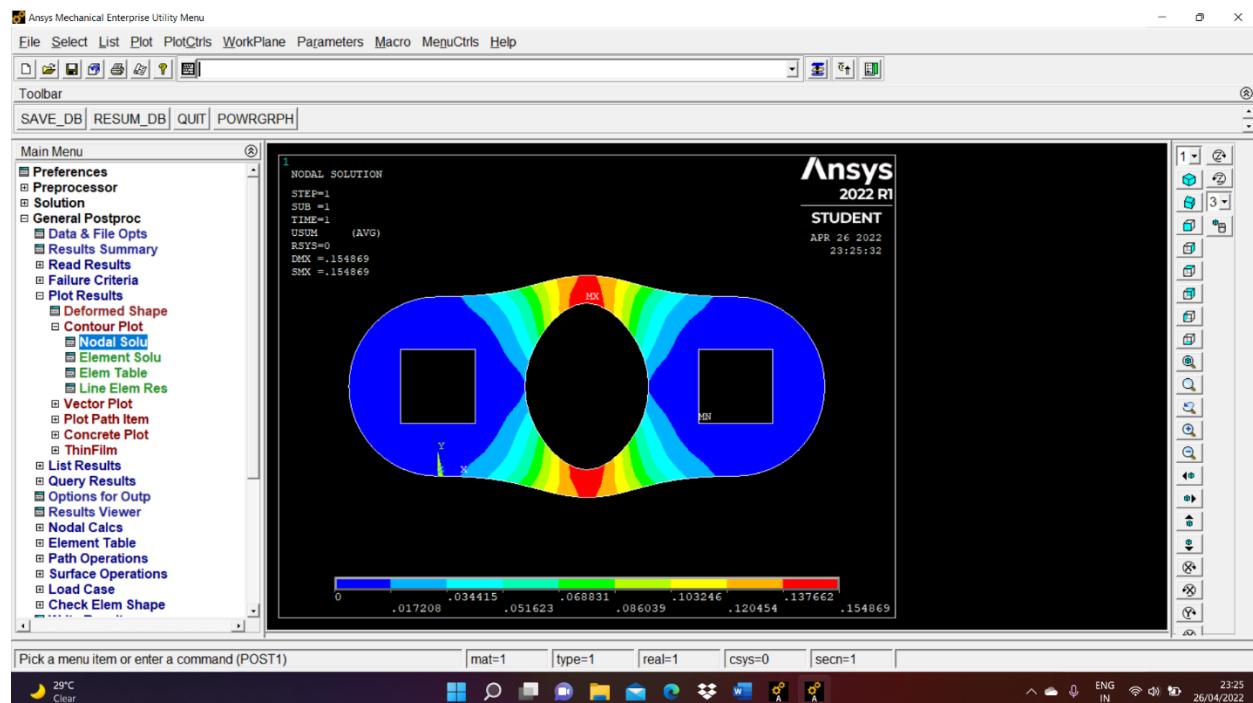


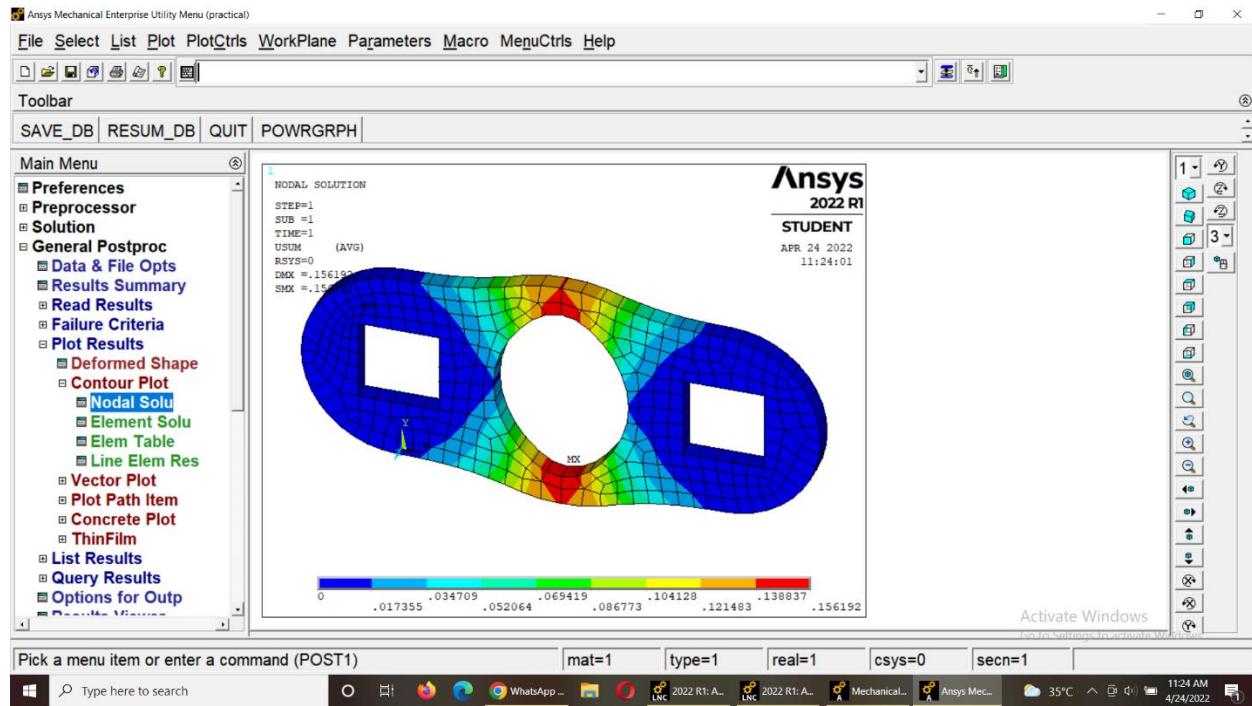
Step14: Solving solution: solution> solve>current LS>ok



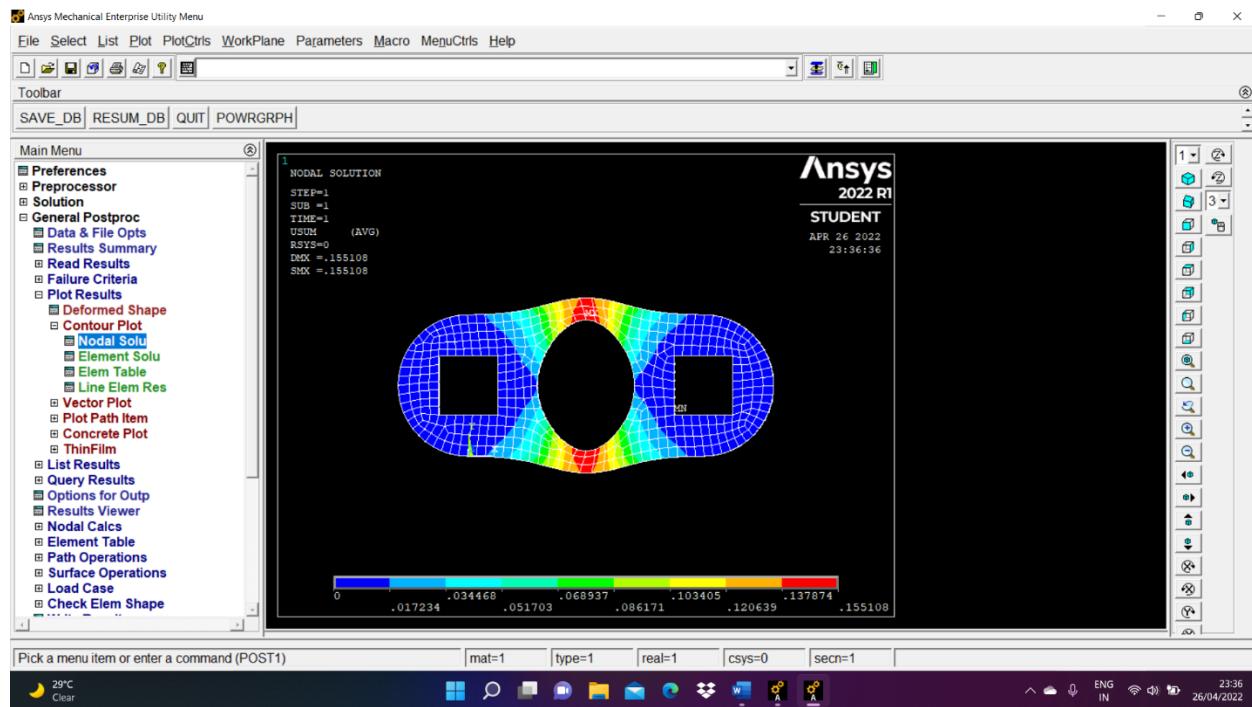
RESULTS:-

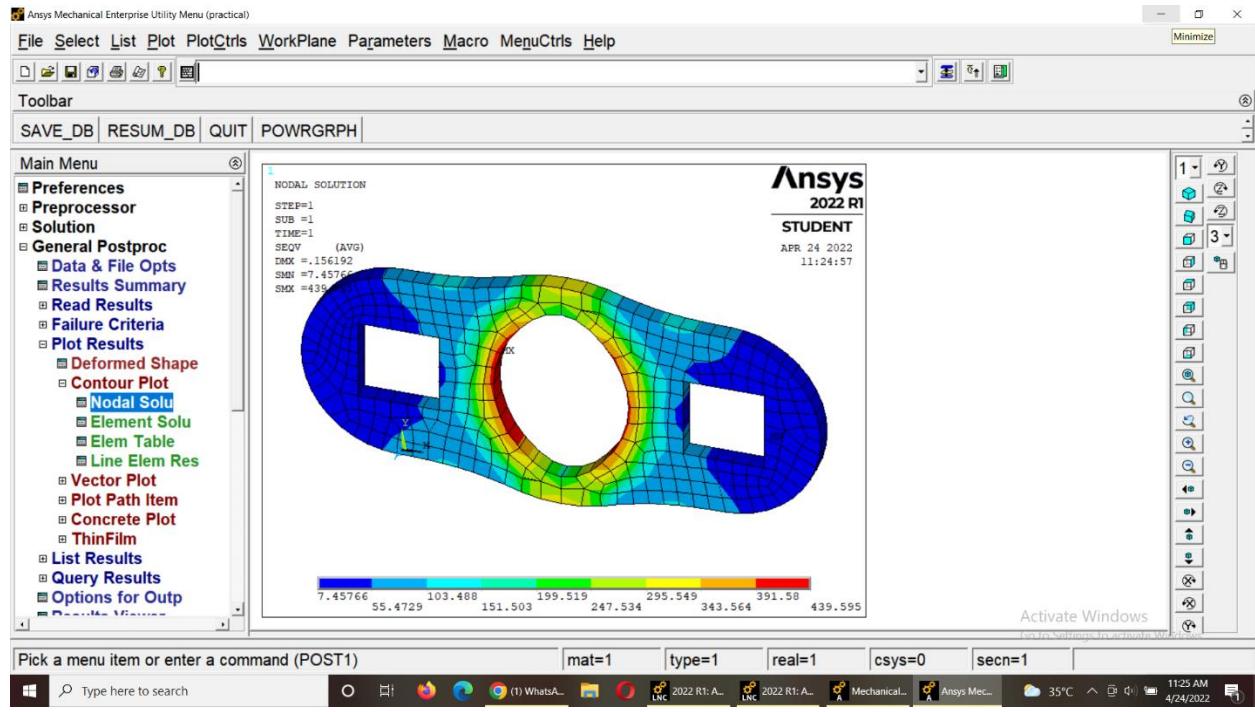
NODAL DISPLACEMENT:-





STRESSES:-

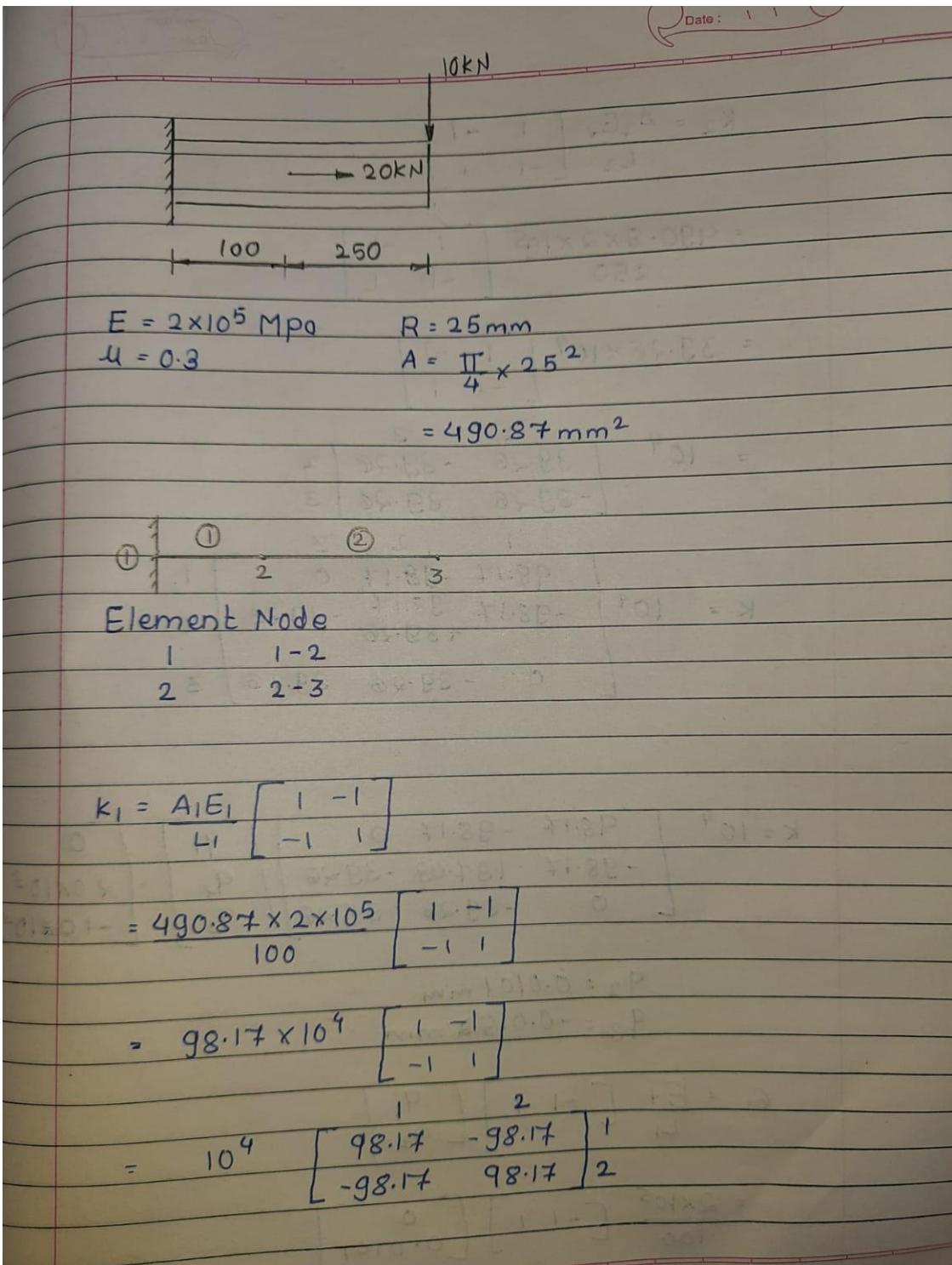




SO HERE BY ANALYSIS WE HAVE GOT MAX. INDUCED STRESS IS 439.895 N/MM²

MAX. DISPLACEMENT IS 0.155 MM

BY analytical solution:-



$$K_2 = \frac{A_2 E_2}{L_2} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

$$= \frac{490 \cdot 8 \times 2 \times 10^5}{250} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

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

$$= 10^4 \begin{bmatrix} 39.26 & -39.26 \\ -39.26 & 39.26 \end{bmatrix} \begin{matrix} 2 \\ 3 \end{matrix}$$

$$K = 10^4 \begin{bmatrix} 1 & 2 & 3 \\ 98.17 & -98.17 & 0 \\ -98.17 & 98.17 & -39.26 \\ 0 & -39.26 & 39.26 \end{bmatrix} \begin{matrix} 1 \\ 2 \\ 3 \end{matrix}$$

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

$$q_2 = 0.0101 \text{ mm}$$

$$q_3 = -0.0158 \text{ mm}$$

$$G_1 = \frac{EI}{L_1} \begin{bmatrix} -1 & 1 \end{bmatrix} \begin{bmatrix} q_1 \\ q_2 \end{bmatrix}$$

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

$$= 20.2 \text{ N/mm}^2$$

$$G_2 = \frac{E_2}{L_2} [-1 \ 1] \begin{bmatrix} 0.0101 \\ -0.0152 \end{bmatrix}$$

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

$$= 4.08 \text{ N/mm}^2$$

CONCLUSION:-

Thus by comparing analytical and software solution we have got

Max. stresses:-

By ansys solution:- 7.4576 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.

NAME : SOURAV SANTAJI GUJALE

CLASS: TE MECH 1

SEMESTER/YEAR: 6

ROLL NO.: 61

DATE OF PERFORMANCE:

DATE OF SUBMISSION:

EXAMINED BY:

EXPERIMENT NO:4

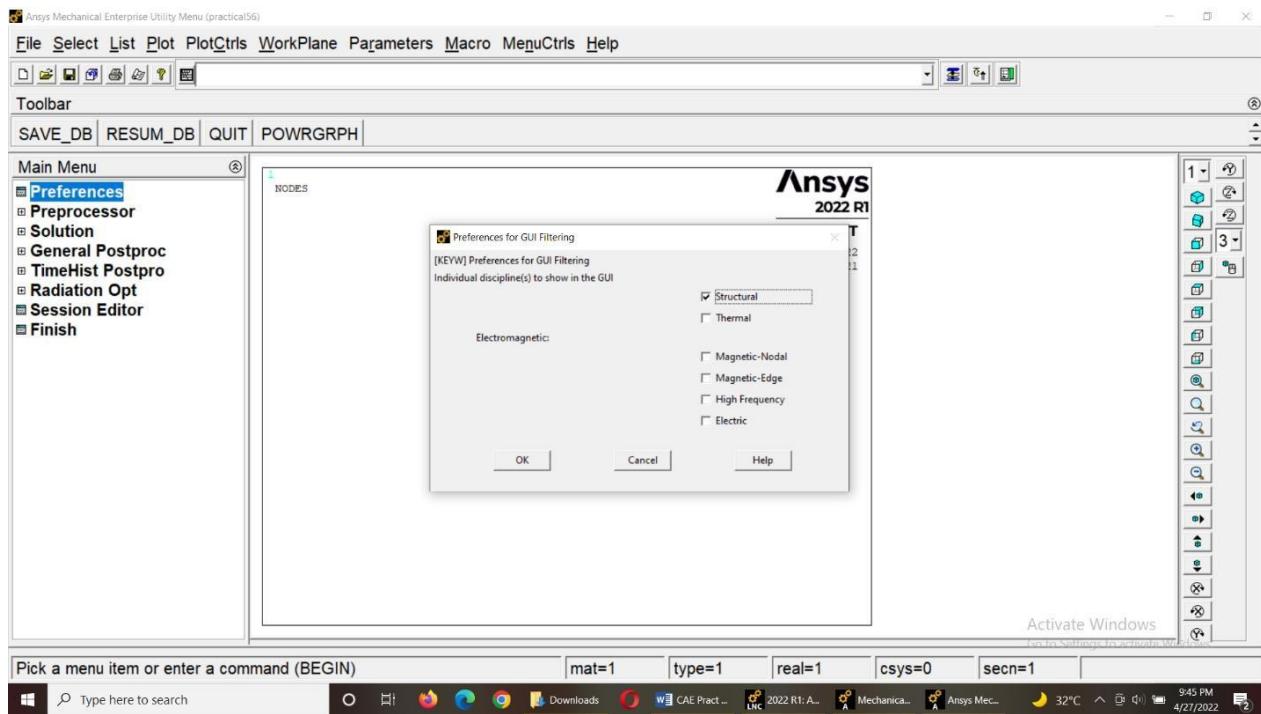
AIM OF EXPERIMENT:-BUCKLING analysis of beam using finite element package.

Finite Element Package: ANSYS 2022

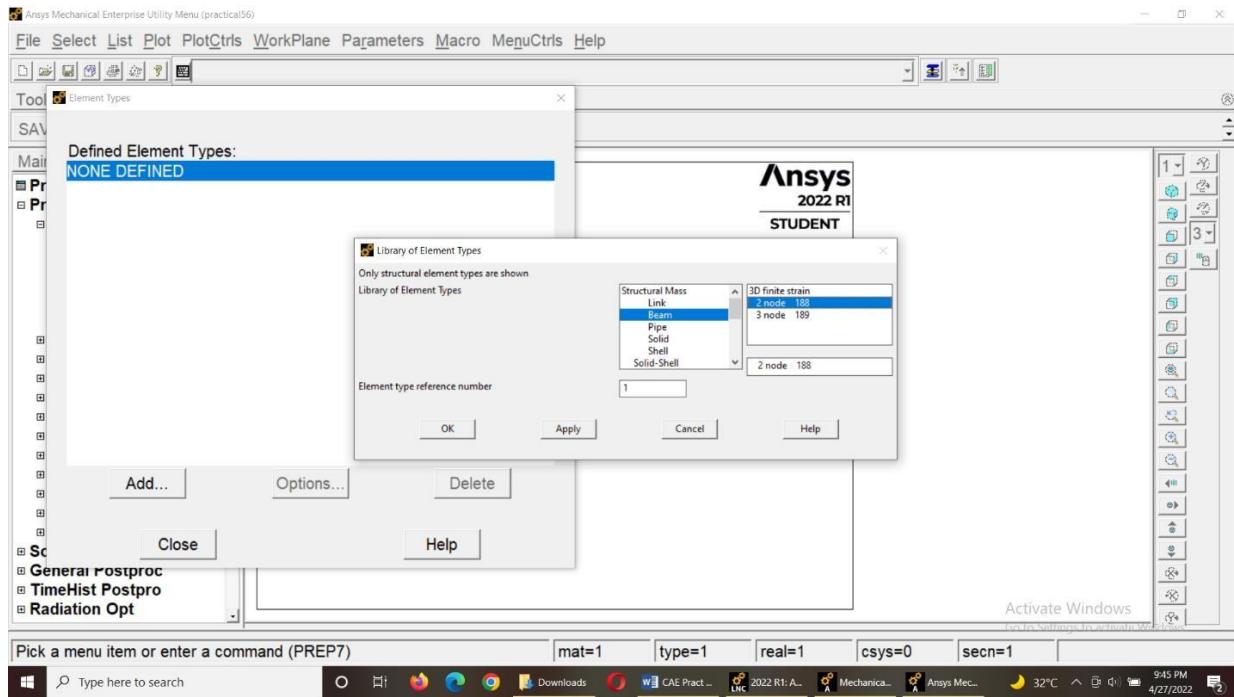
Stress distribution in a beam with applied load.

$$E = 2 \times 10^5 \text{ MPa} \quad R = 25 \text{ mm} \quad u = 0.3$$

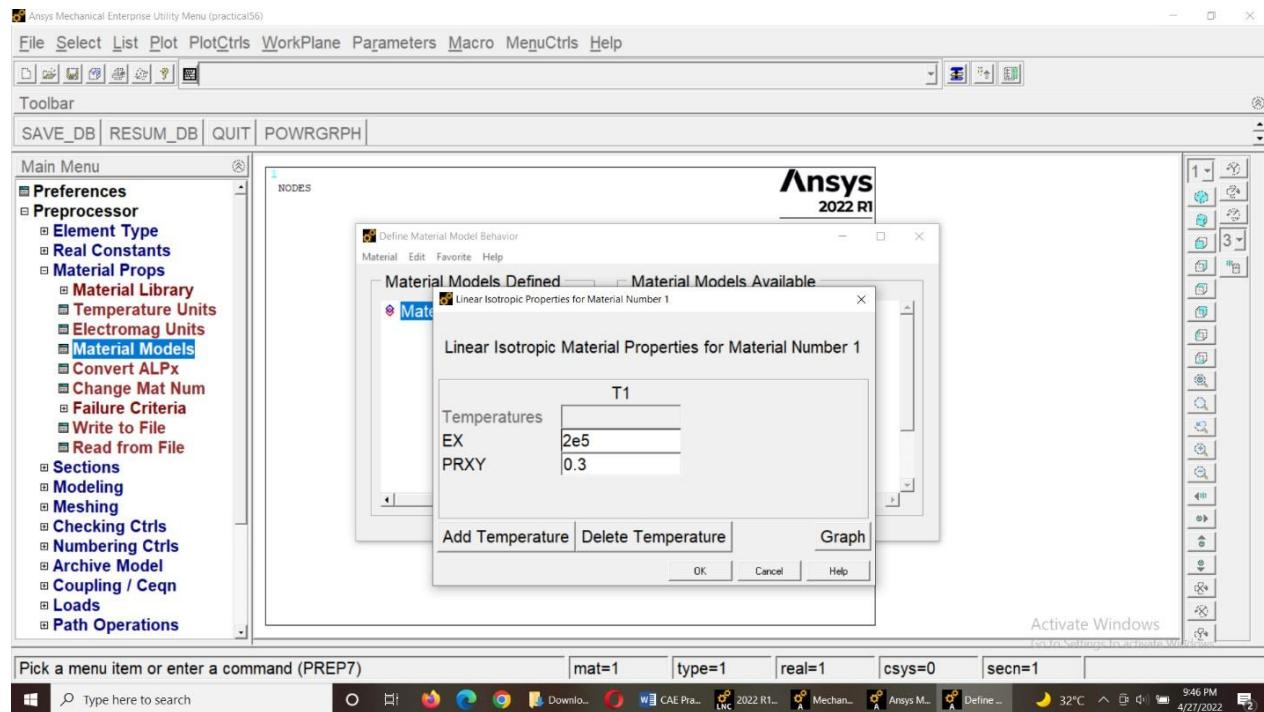
Step 1: Select type of Analysis---- Preferences> structural>Press Ok



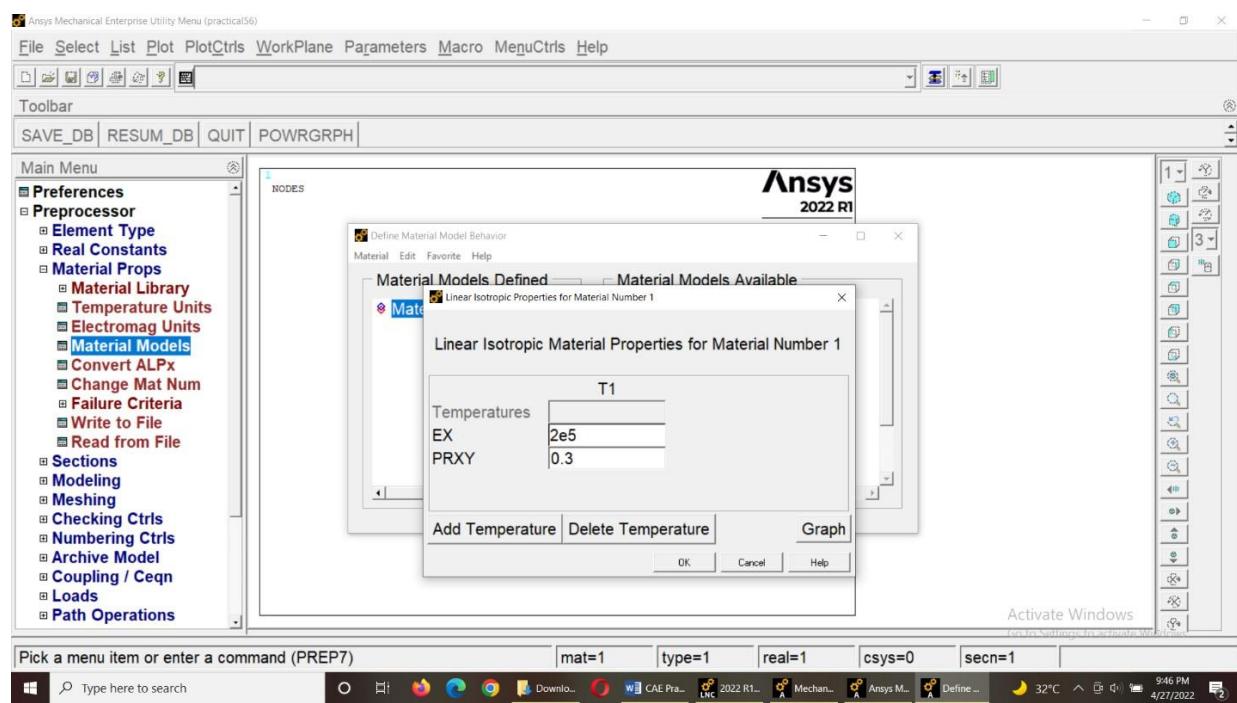
Step 2: Add the element type.....preprocessor>element type>beam>2node 188> Press ok



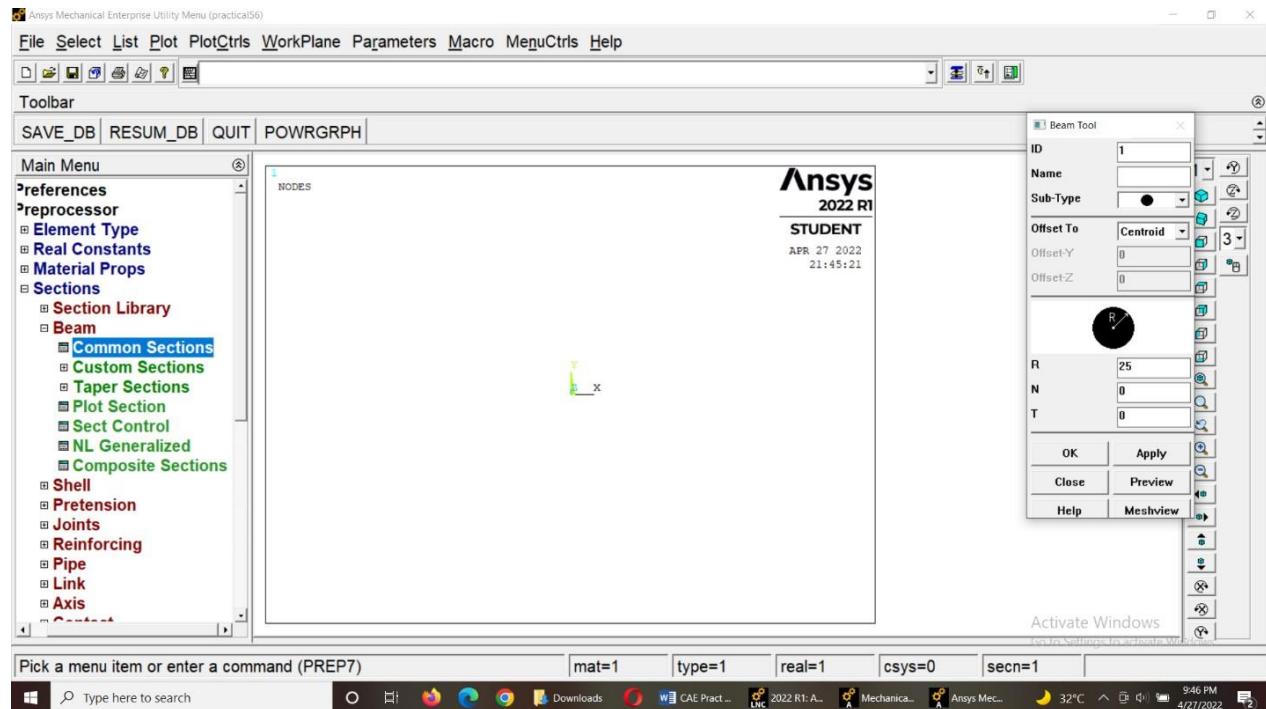
Step 3: Preprocessor>Material prop.>Material models>Material number1



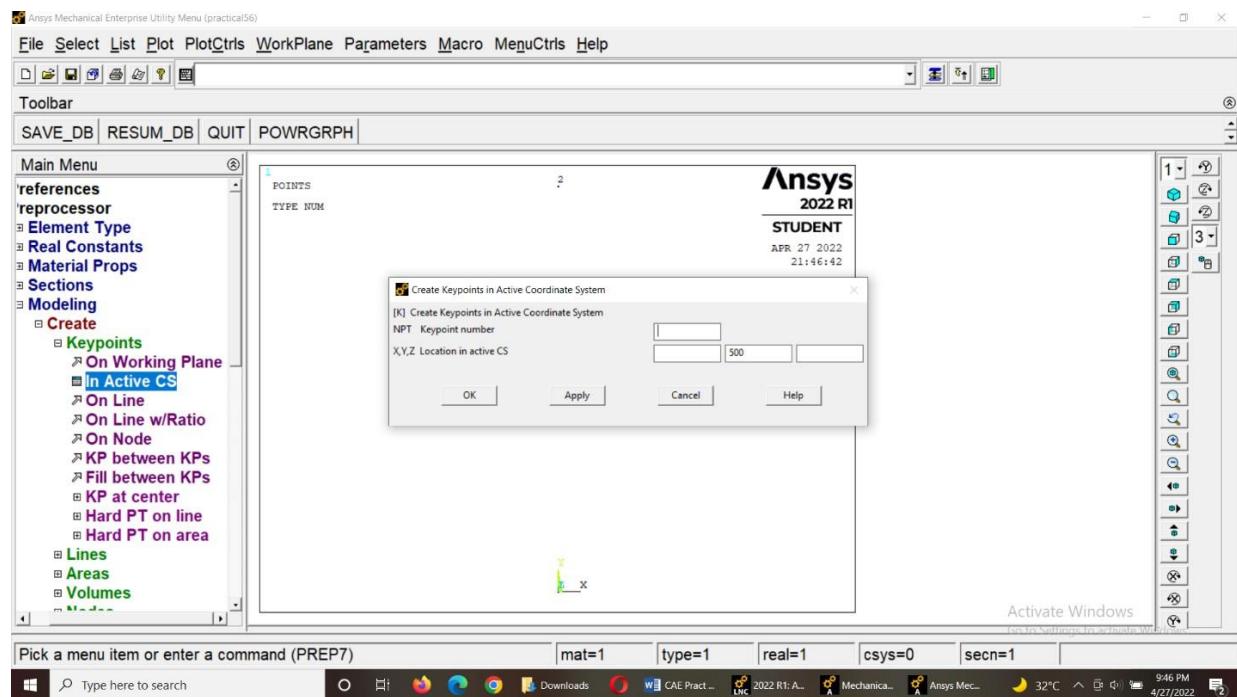
Step 4: Material models>structural>linear>elastic>isotropic.



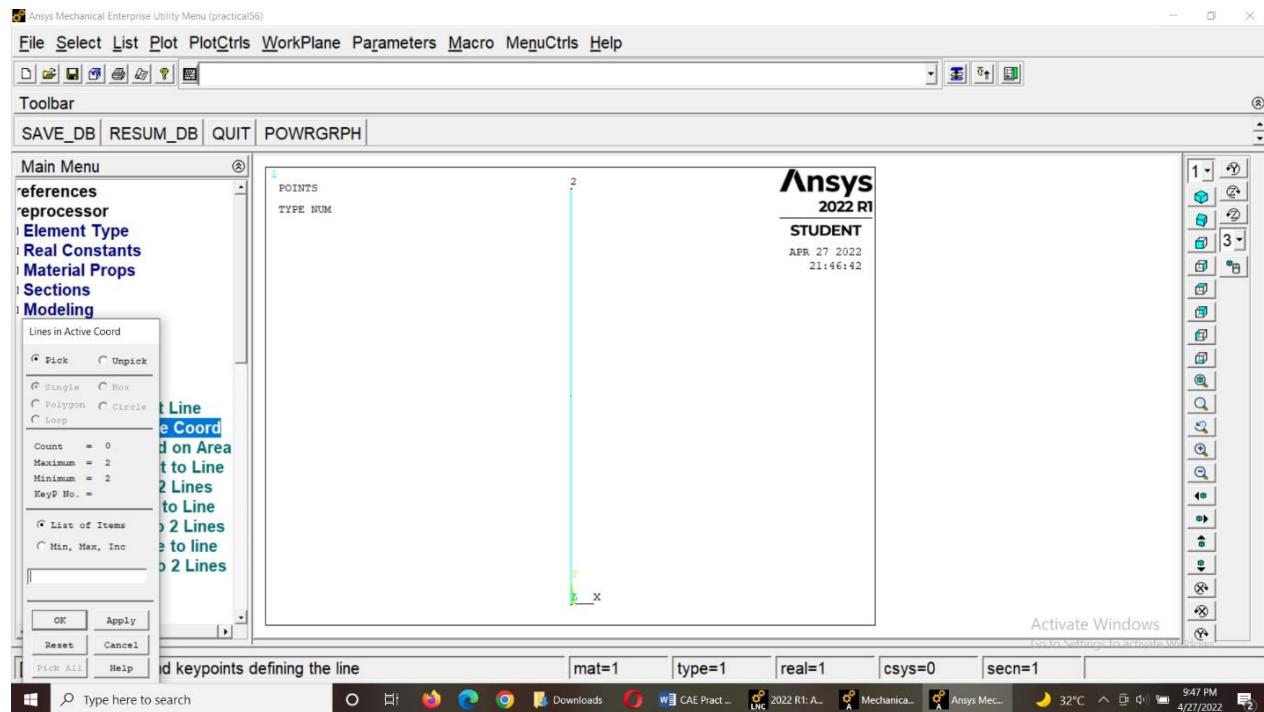
Step 5: selecting section of beam....section>beam>common section>select section.



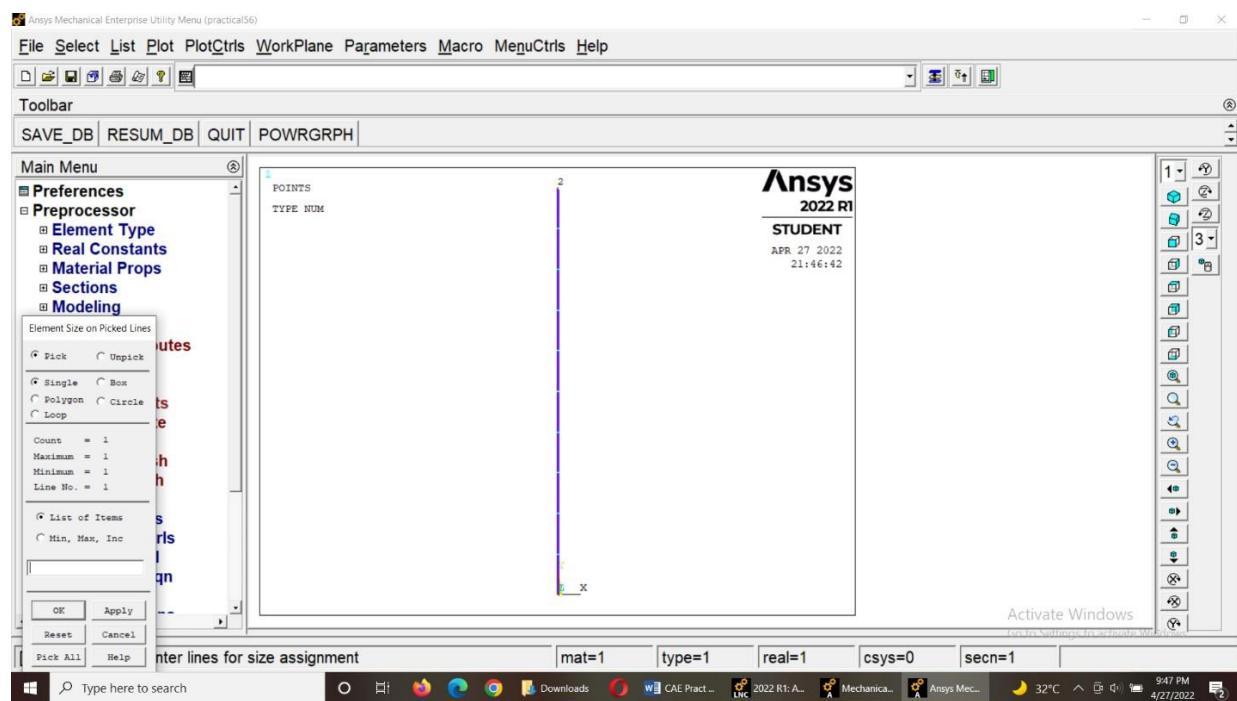
Step6: Creating Keypoints:- modeling>create>keypoint>in active cs>select coordinate.



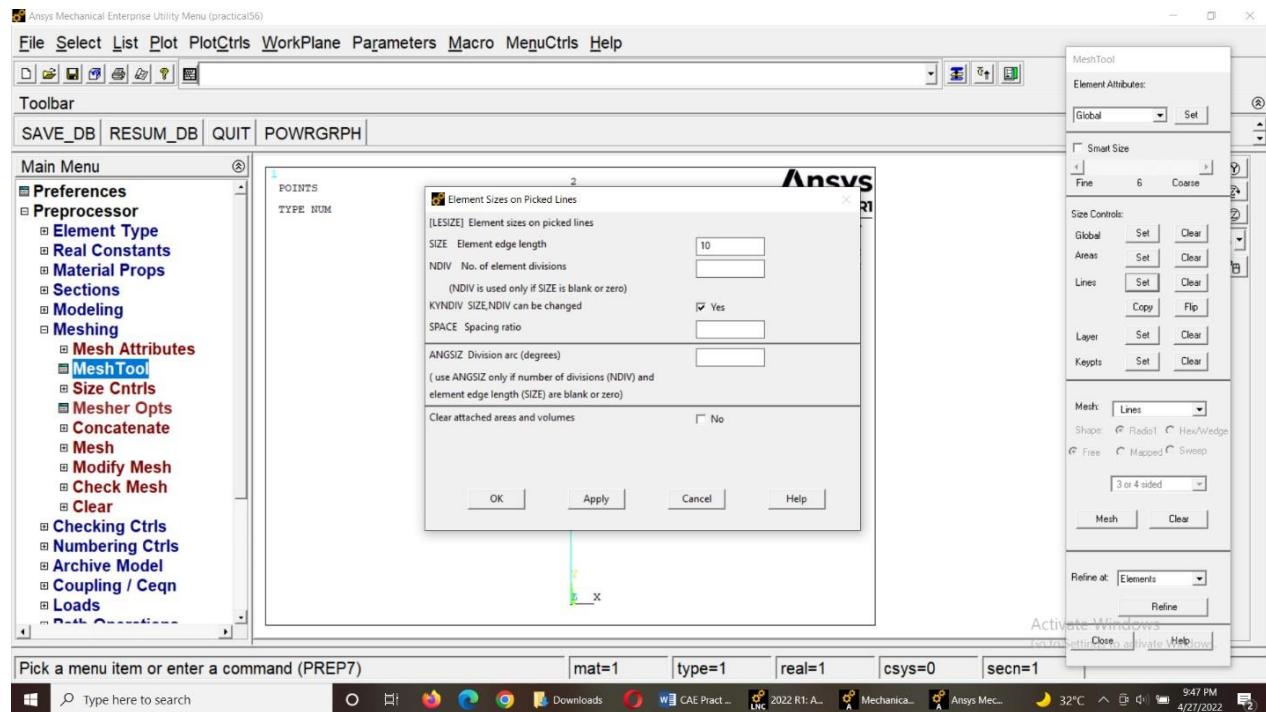
Step7: Lines>Lines>In active co-ordinate cs> Join Co Ordinates



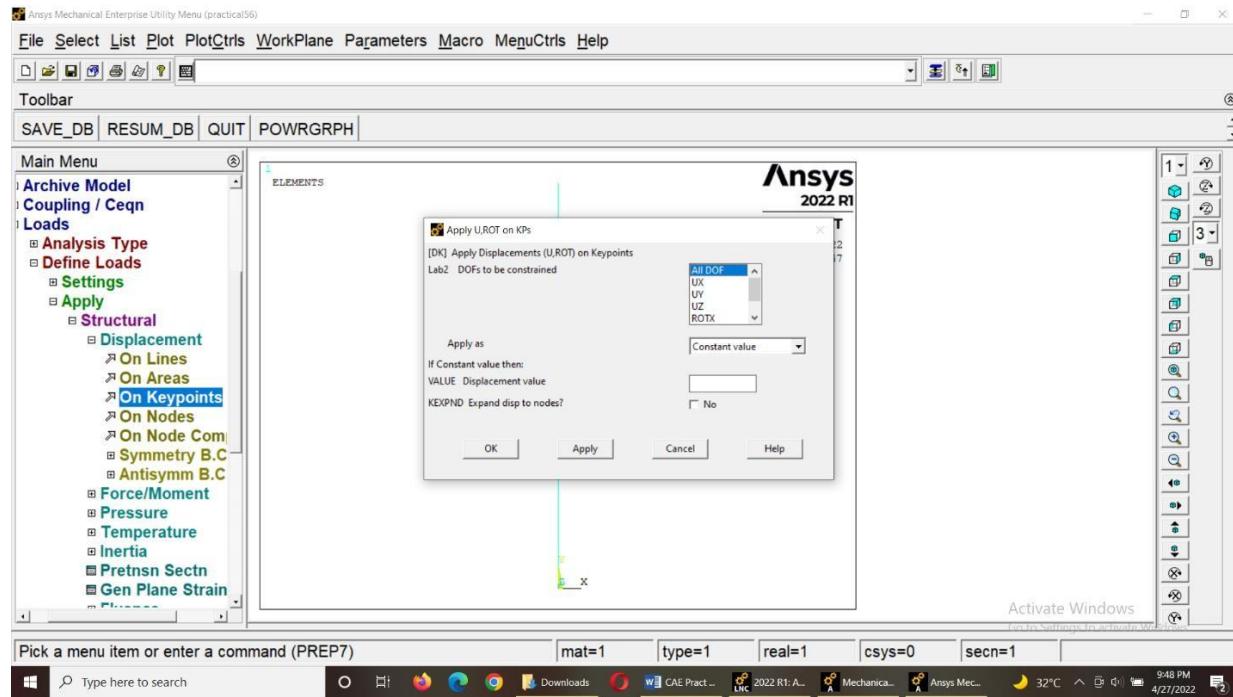
Step8: meshing:- Meshing> meshtools>lines>Set>Select Model>Apply



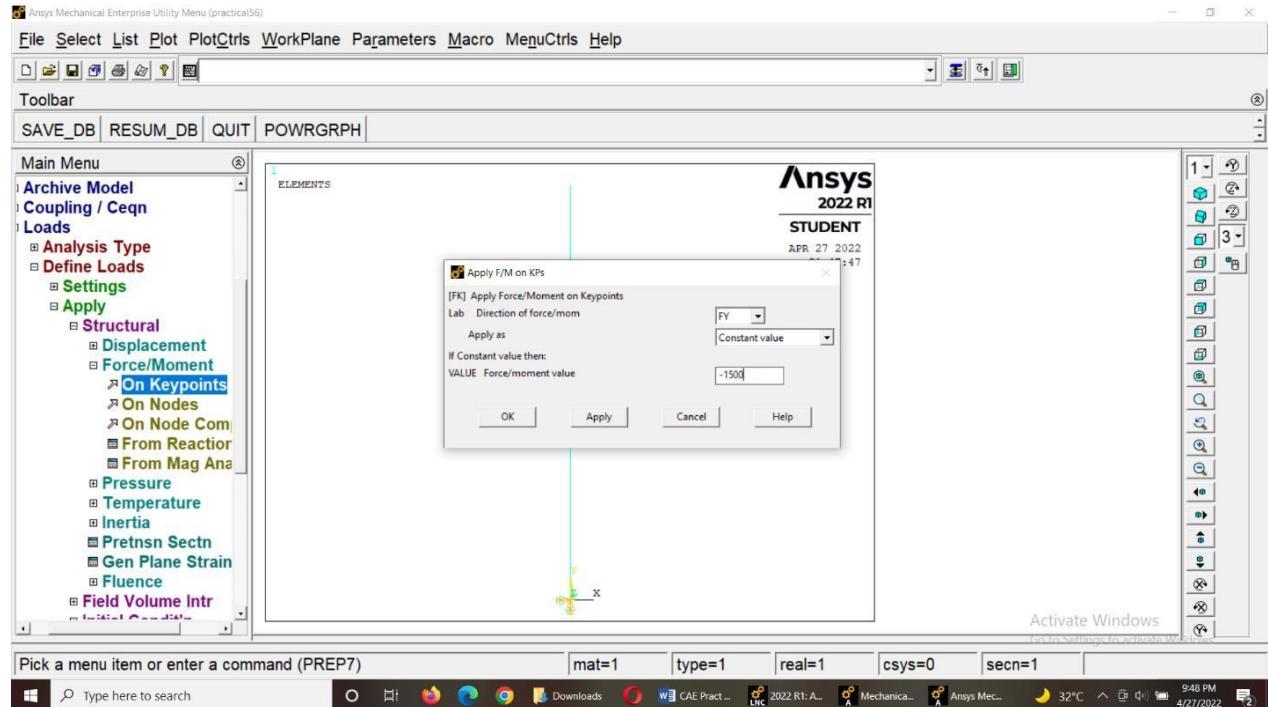
Step9: Enter no division>ok>mesh>selectmodel>ok



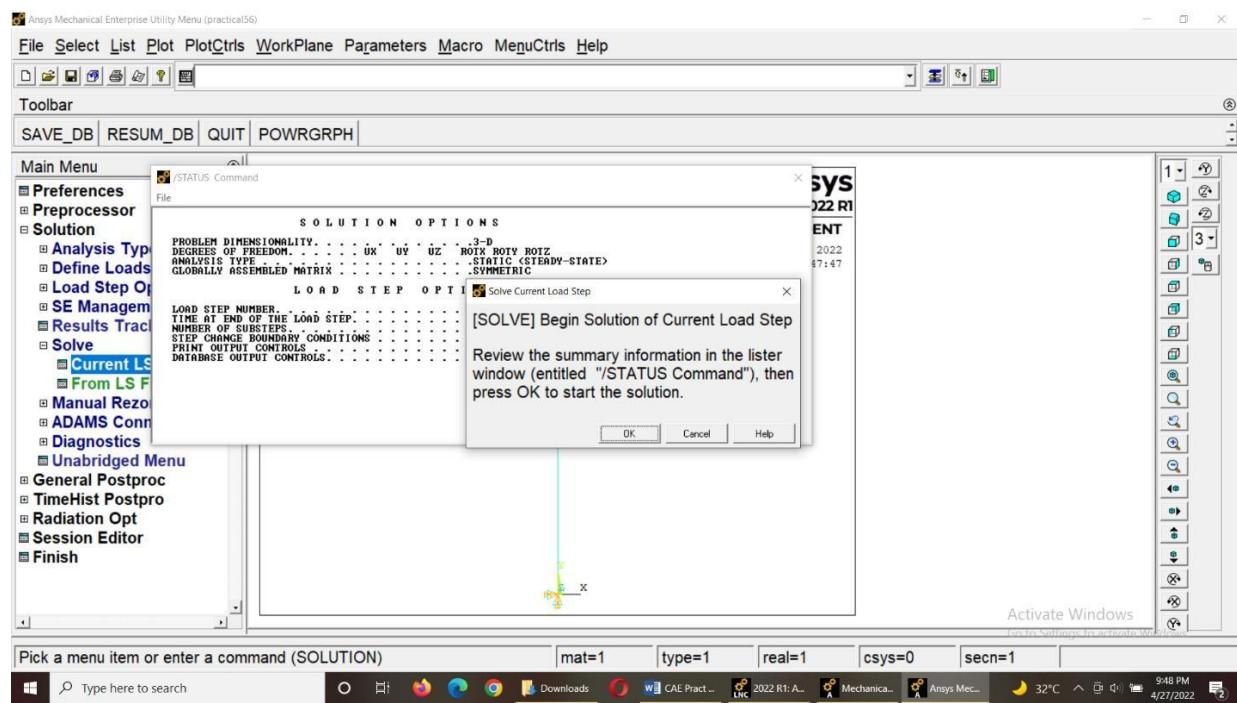
Step10: Apply loads: Laods>define loads>apply>structural>displacement>on keypoints> All Dof>0>ok



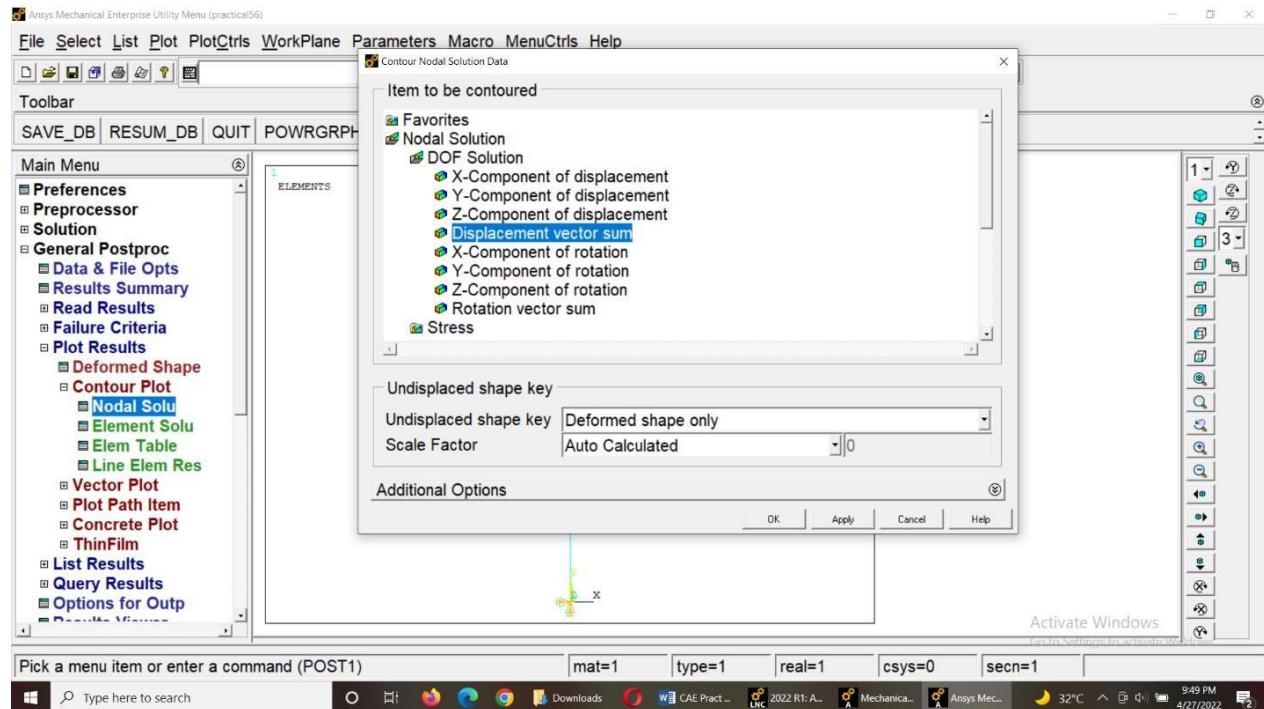
Step11: Loads>Define loads>apply>forces>on keypoints> selecting direction of forces (here FY)>ok



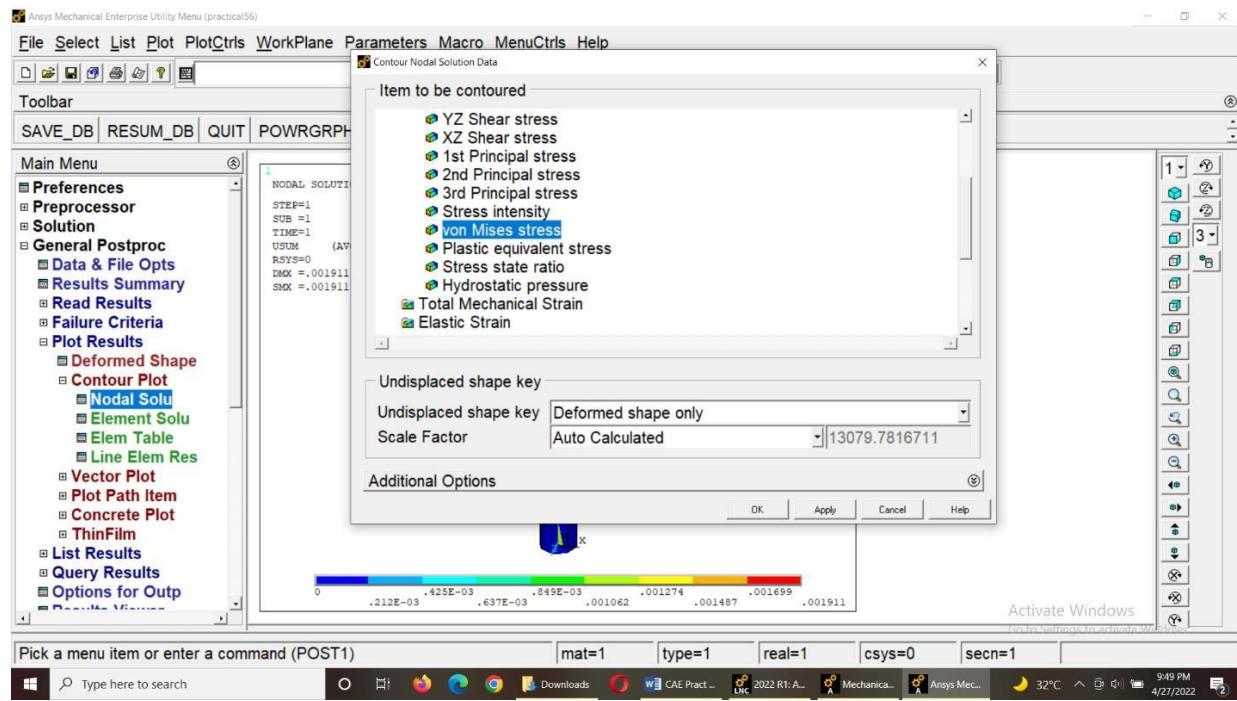
Step12:Solution:- solution>solve>currentls> done



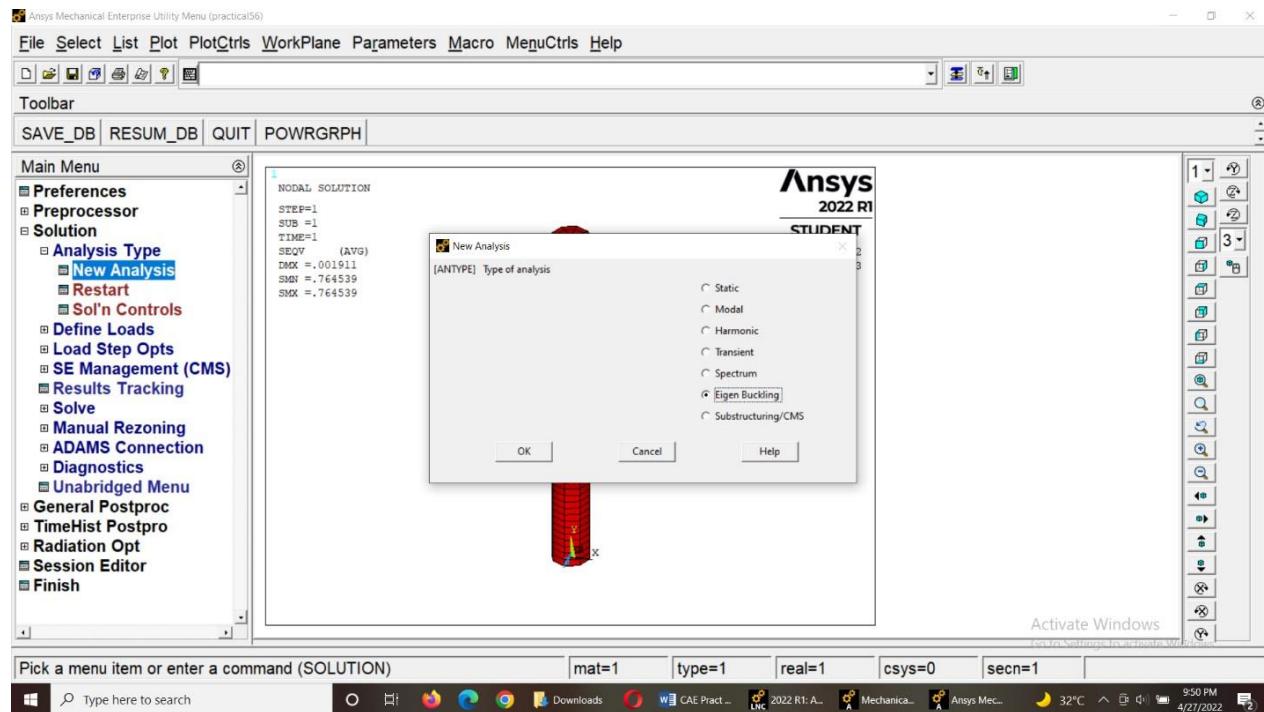
Step13: General postproc> plot result> Nodal solution> Dof >Vector sum displacement> apply.



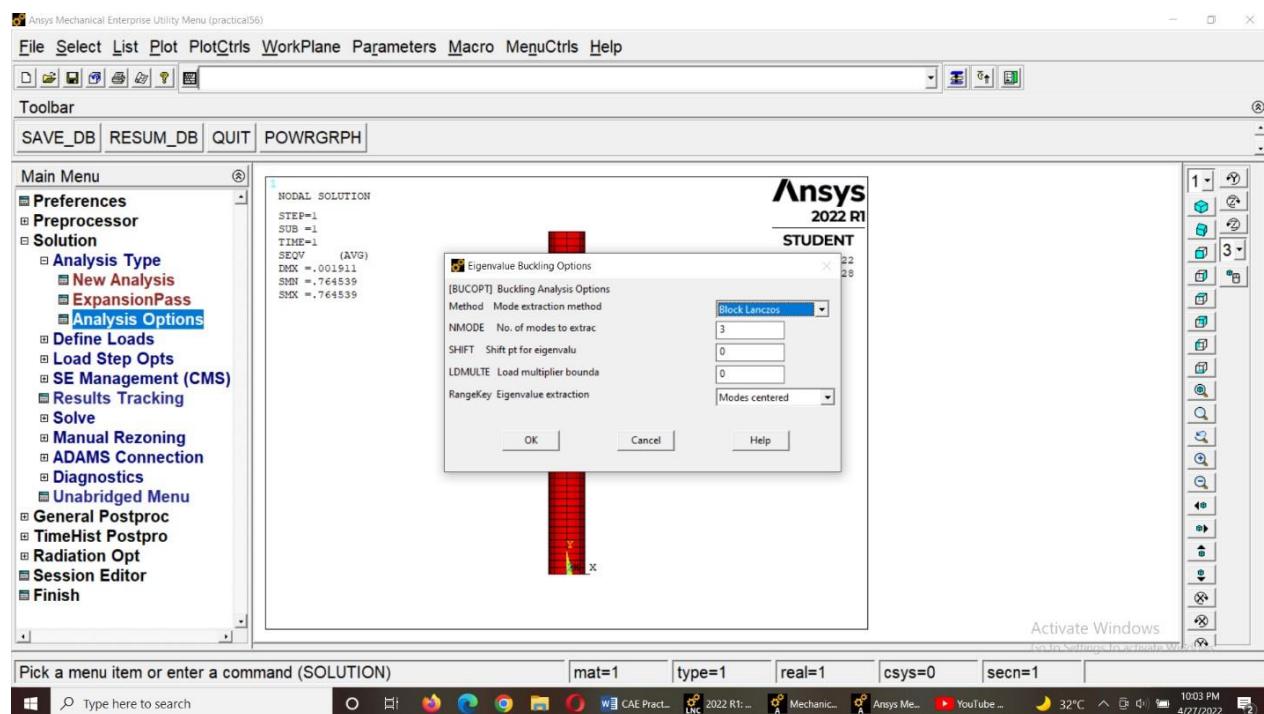
Step14: Nodal solution> stress> von misses stress> apply



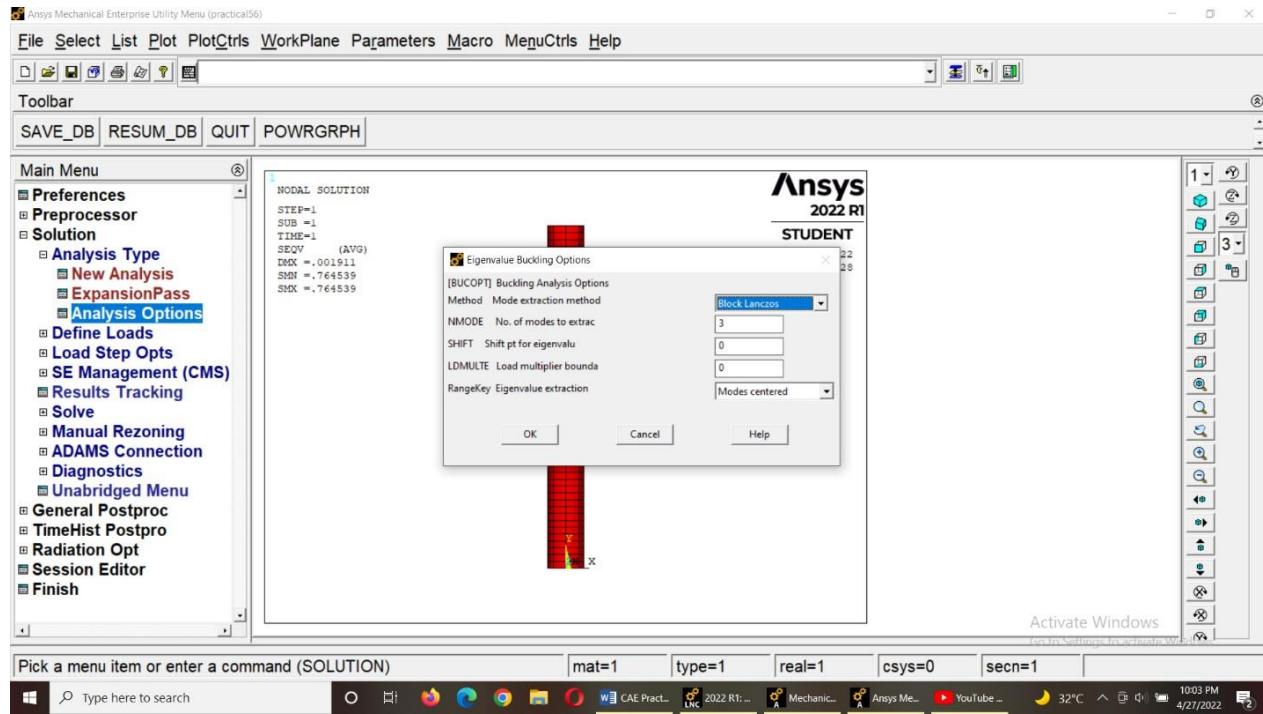
Step15: solution> Analysis Type>New Analysis> select Eigen Buckling.



Step 16: solution> Analysis option> select modes to extrac 3.

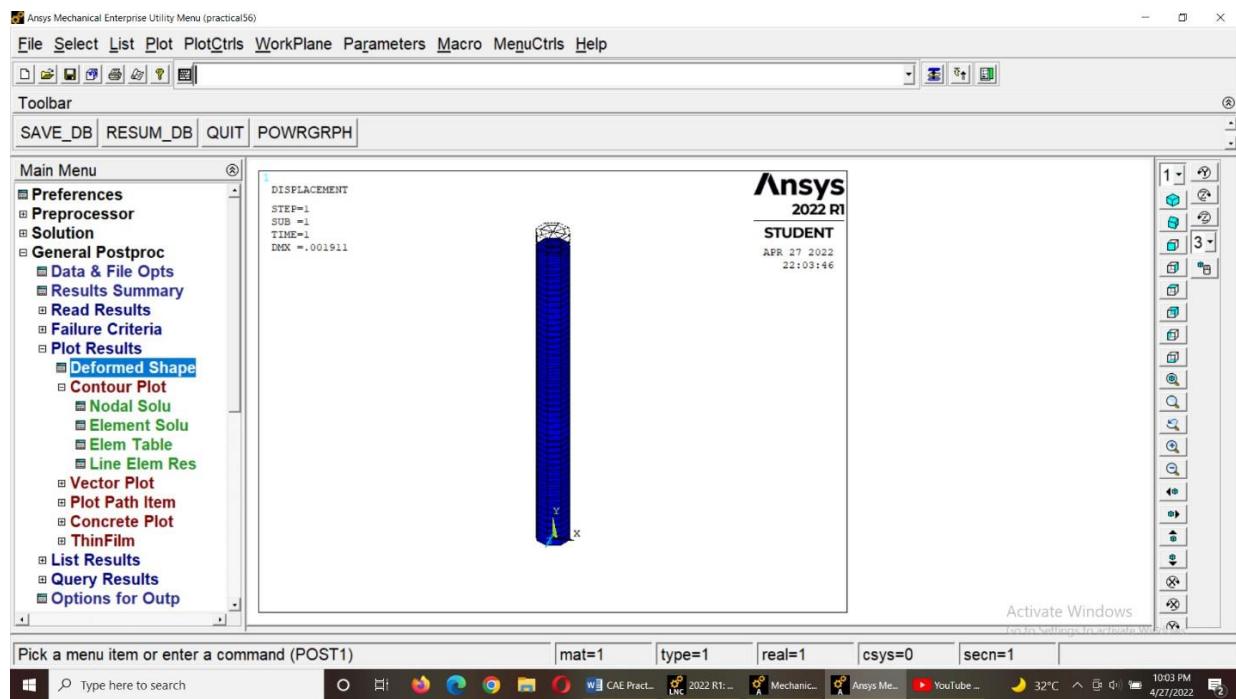


Step 17: Load step opts>Expansion> single Expansion> select extrac no> ok

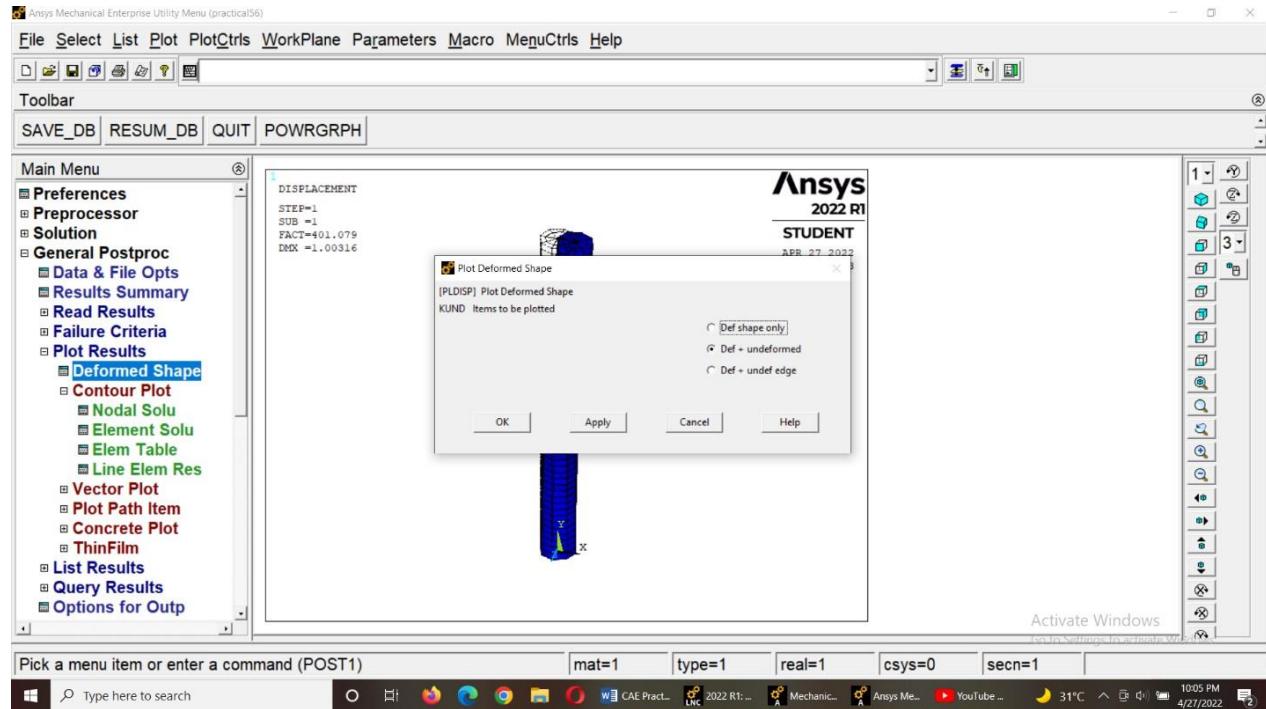


Step 18: solution> current Ls> solve

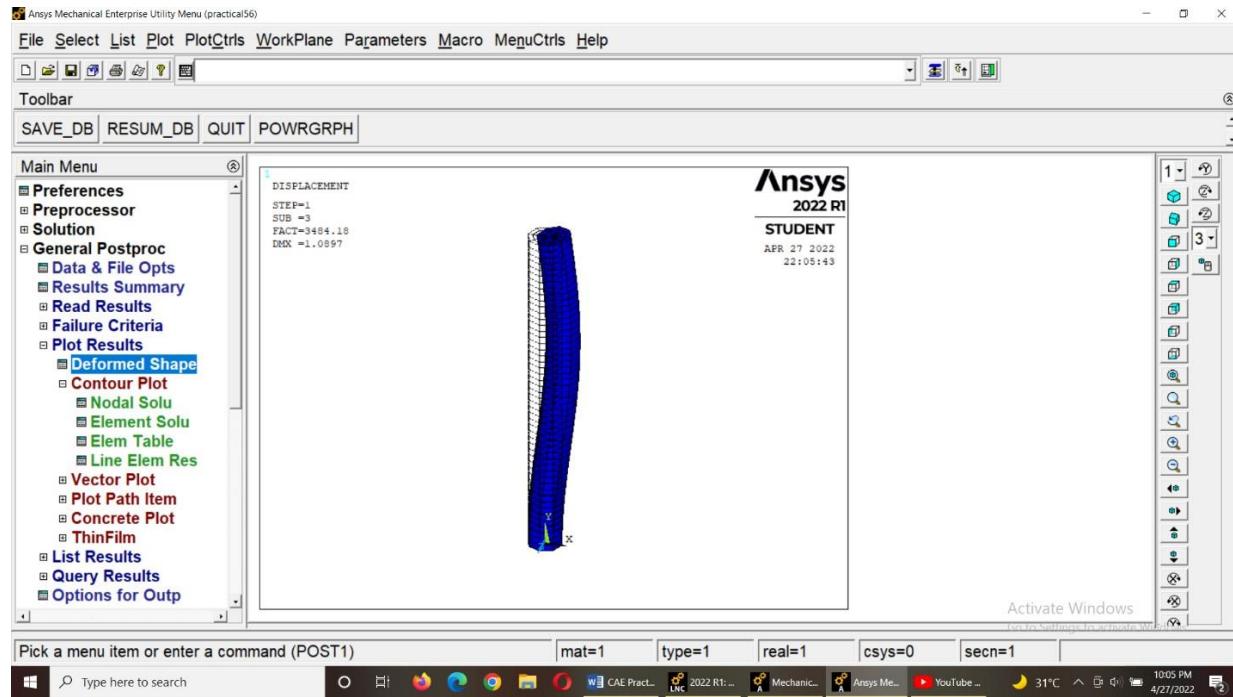
Step19: General postproc> read result> first step> plot result> deformed shape> Def+undeformed> ok



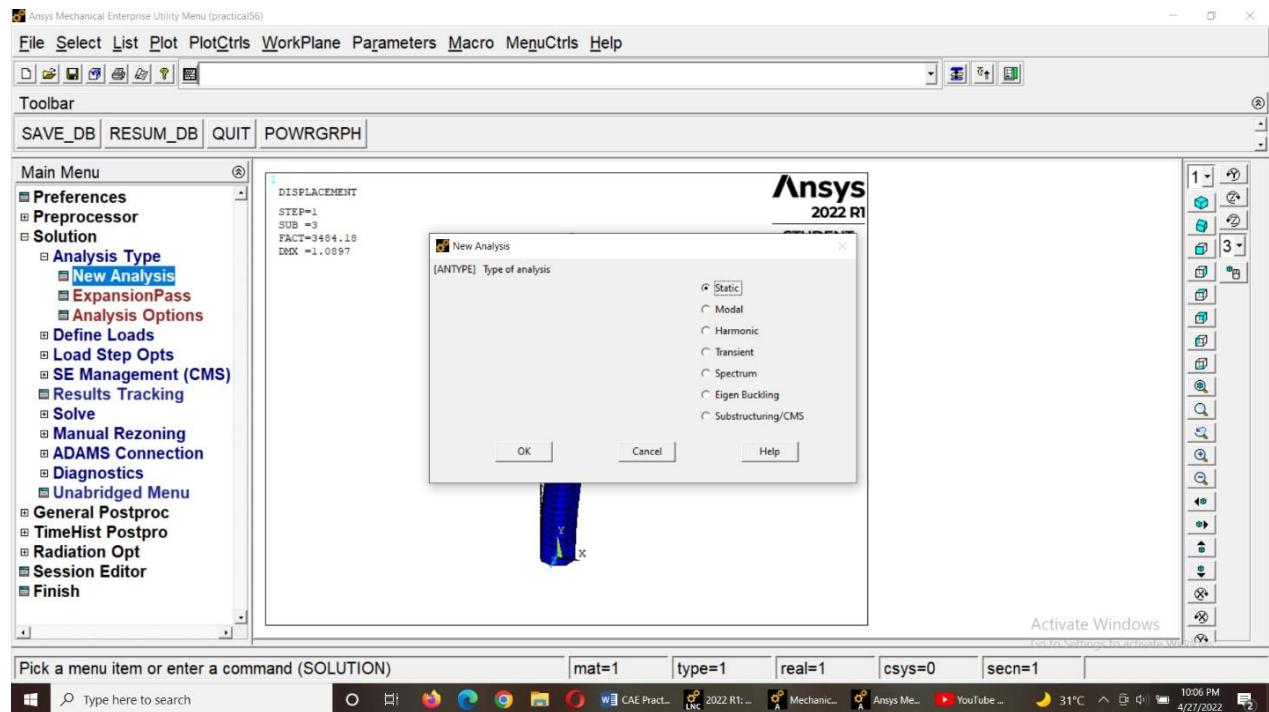
Step 20: General postproc> read result> next step> plot result> deformed shape> Def+undeformed> ok



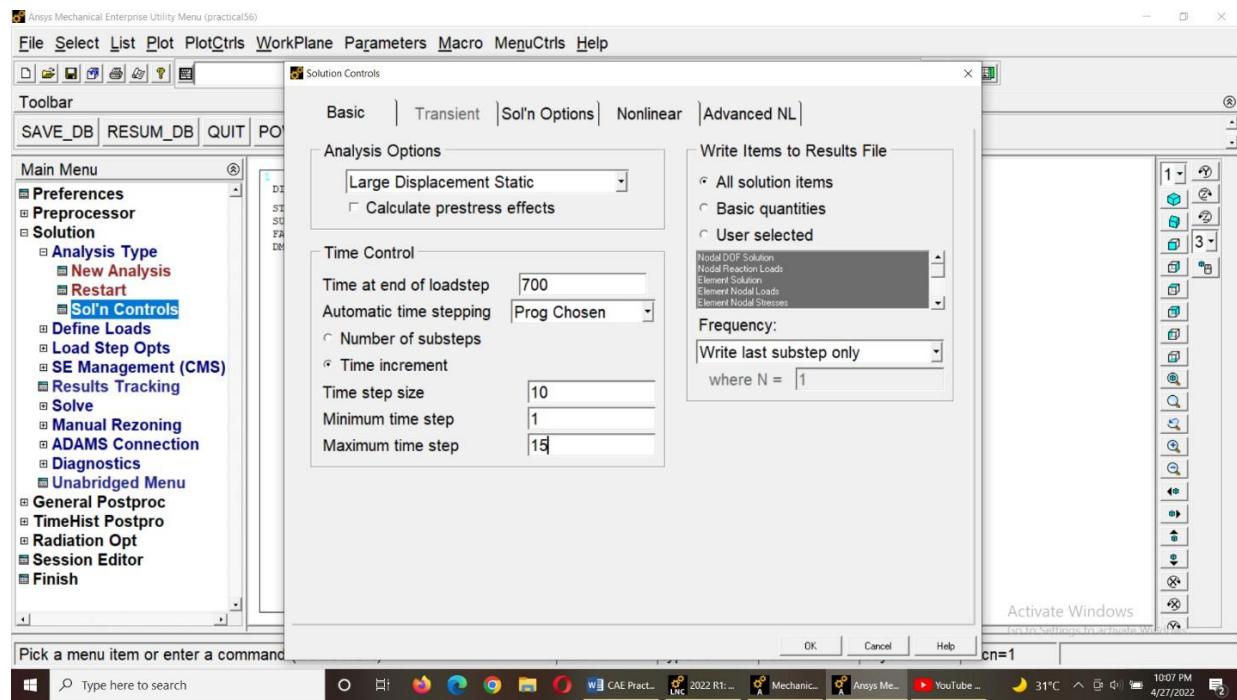
Step 21: General postproc> read result> next step> plot result> deformed shape> Def+undeformed> ok



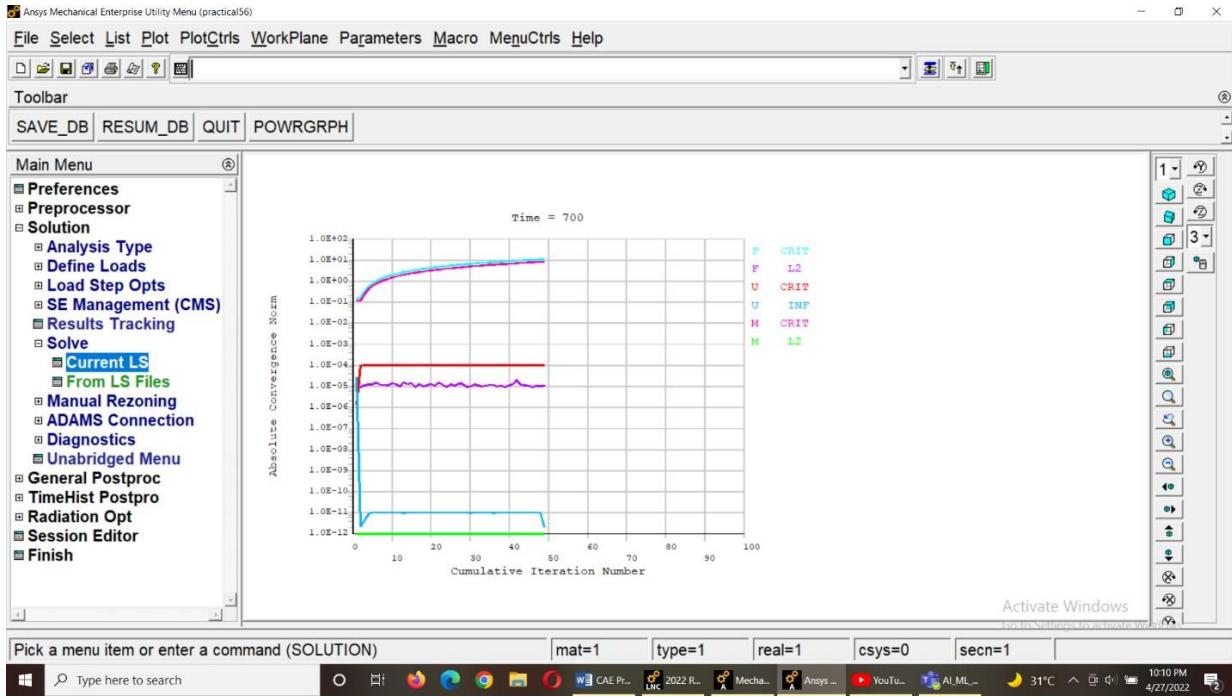
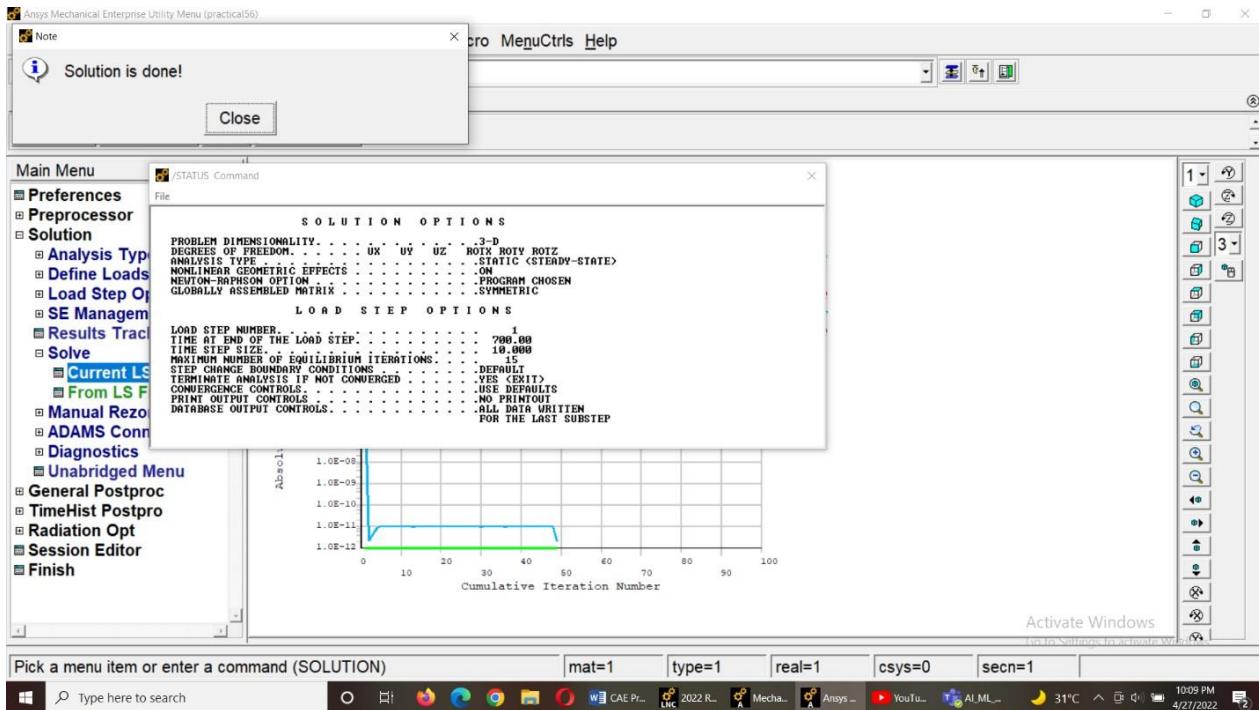
Step 22: solution> analysis type>new analysis>static> ok



Step 23: solution> analysis type>sol'n controls>select Large displacement static> in analysis options>input Time at end of loadstep> select Time increment>input Time step size, min time step, max time step> ok

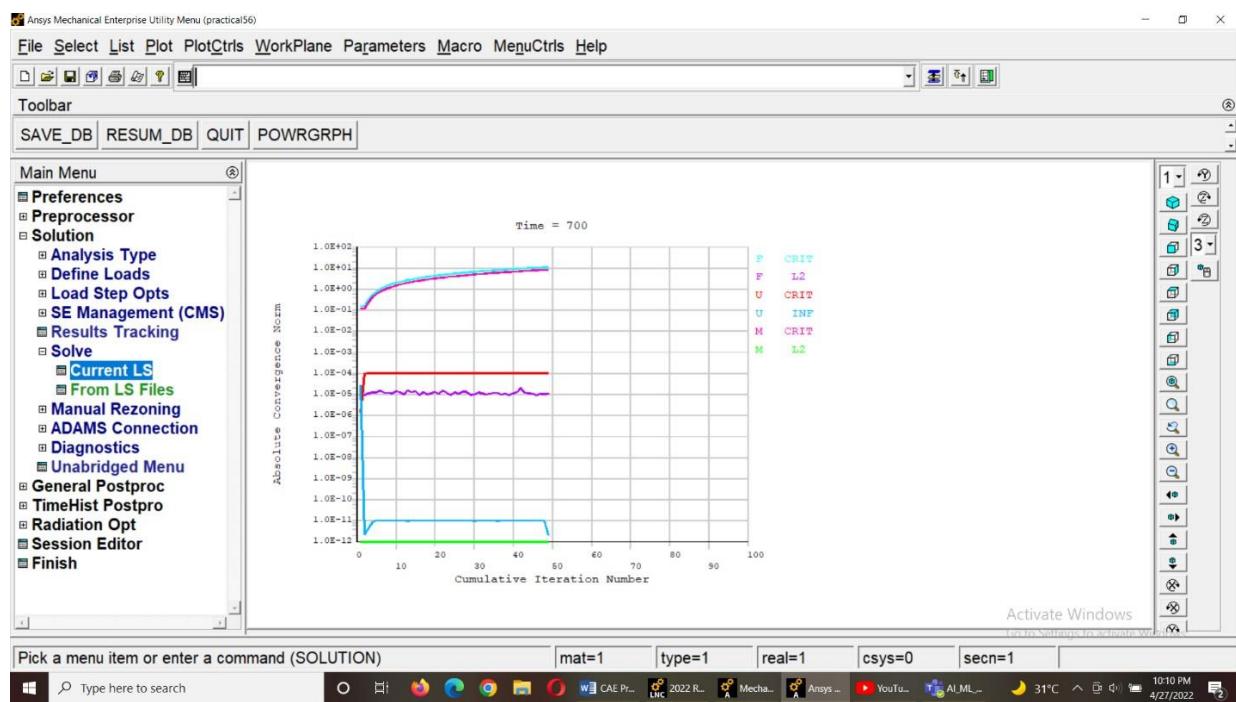
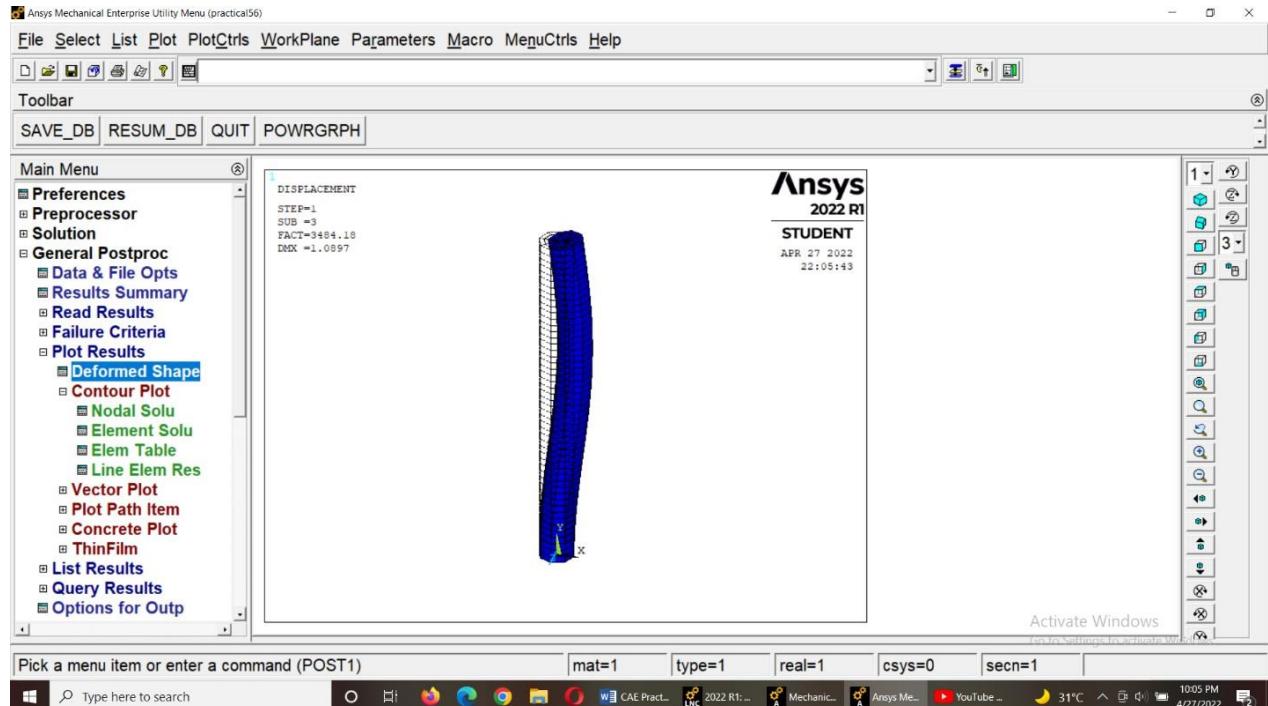


Step24: Solution: solution>solve>currentls> done



RESULTS:-

BUCKLING DISPLACEMENT:-



CONCLUSION

SO HERE BY ANALYSIS WE HAVE GOT MAX. DISPLACEMENT IS 1.0897 MM

NAME : SOURAV SANTAJI GUJALE

CLASS: TE MECH 1

SEMESTER/YEAR: 6

ROLL NO.: 61

DATE OF PERFORMANCE:

DATE OF SUBMISSION:

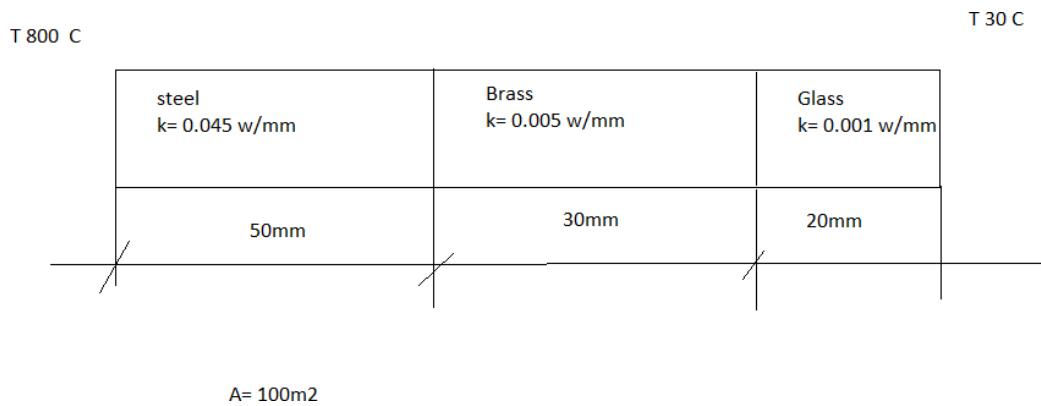
EXAMINED BY:

EXPERIMENT NO:5

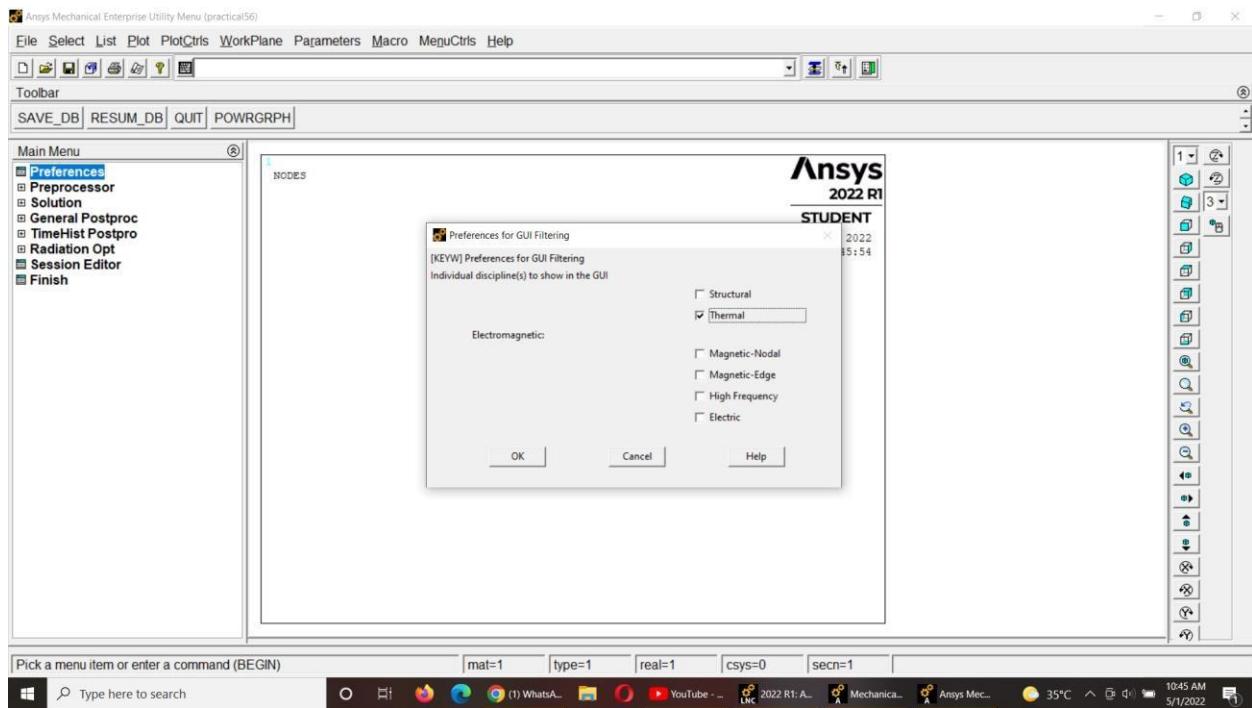
AIM OF EXPERIMENT:- Thermal analysis of beam using finite element package.

Finite Element Package: ANSYS 2022

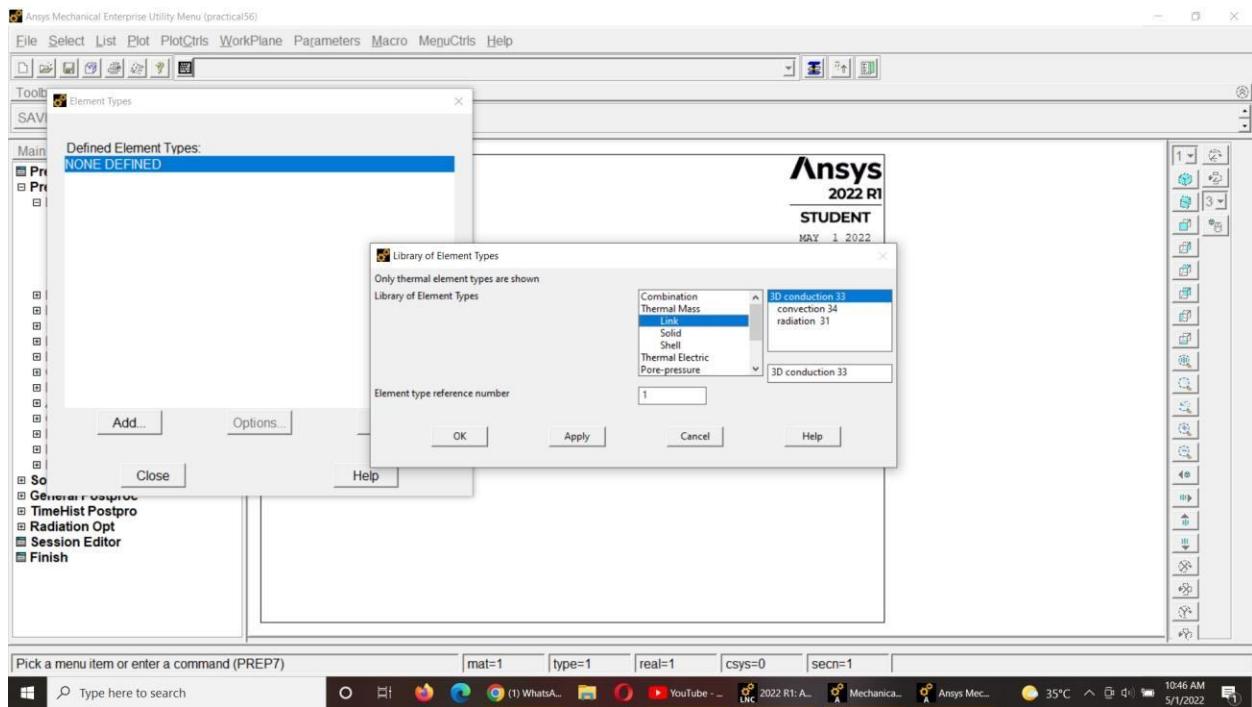
Stress distribution in a beam with applied load.



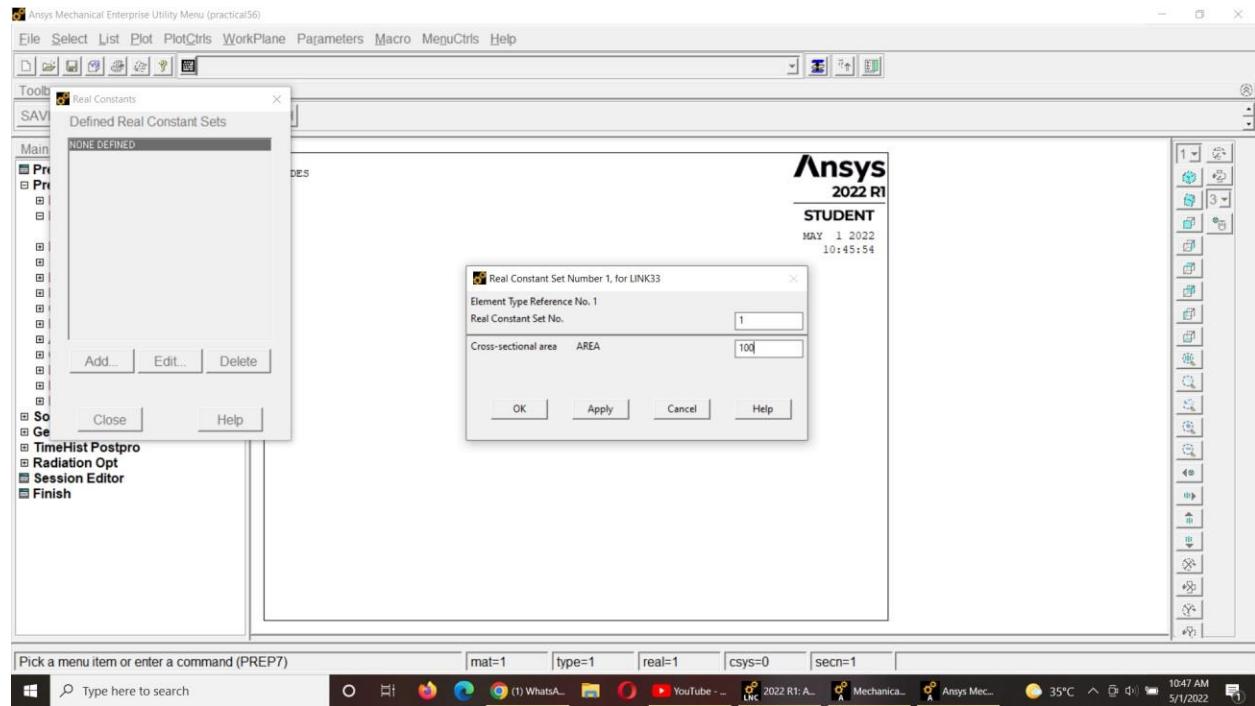
Step 1: Select type of Analysis---- Preferences> Thermal>Press Ok



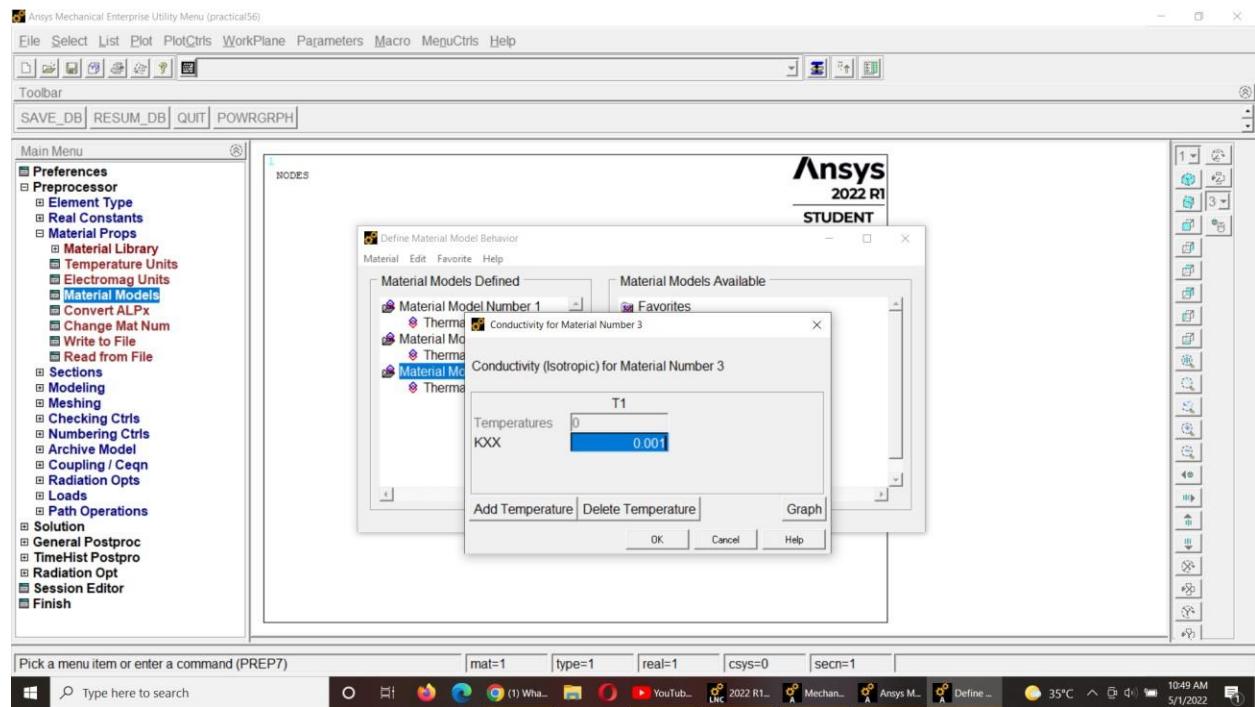
Step 2: Add the element type.....preprocessor>element type>link>3D conduction 33> Press ok



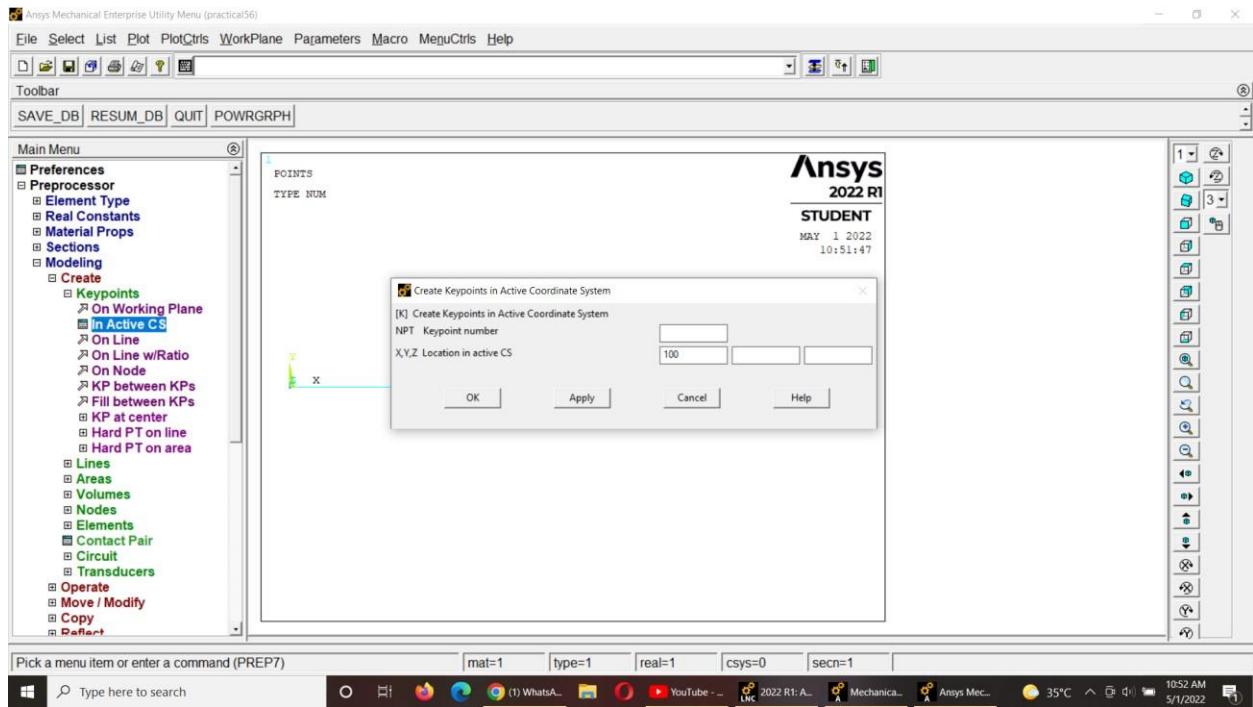
Step 3: Preprocessor>real constrains>Add>input cross section areas>ok



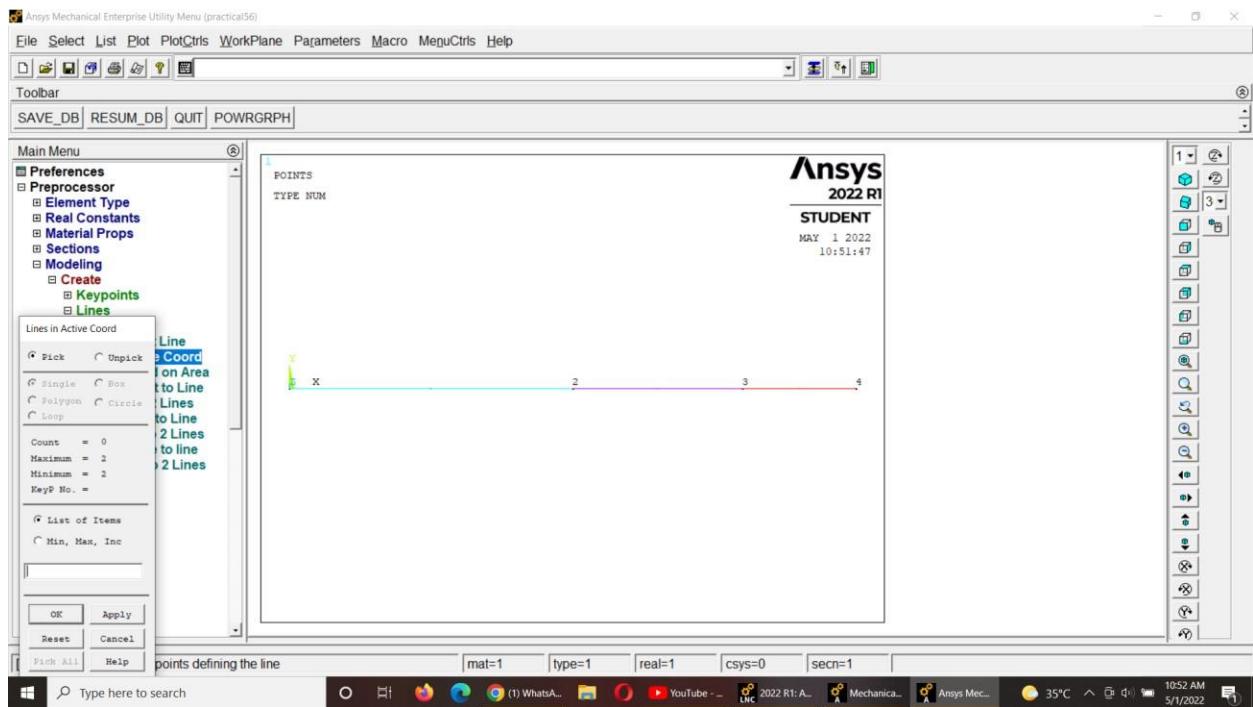
Step 4: material properties>Material models>Thermal>linearr>isotropic>conduction>put Thermal conductivity>ok.



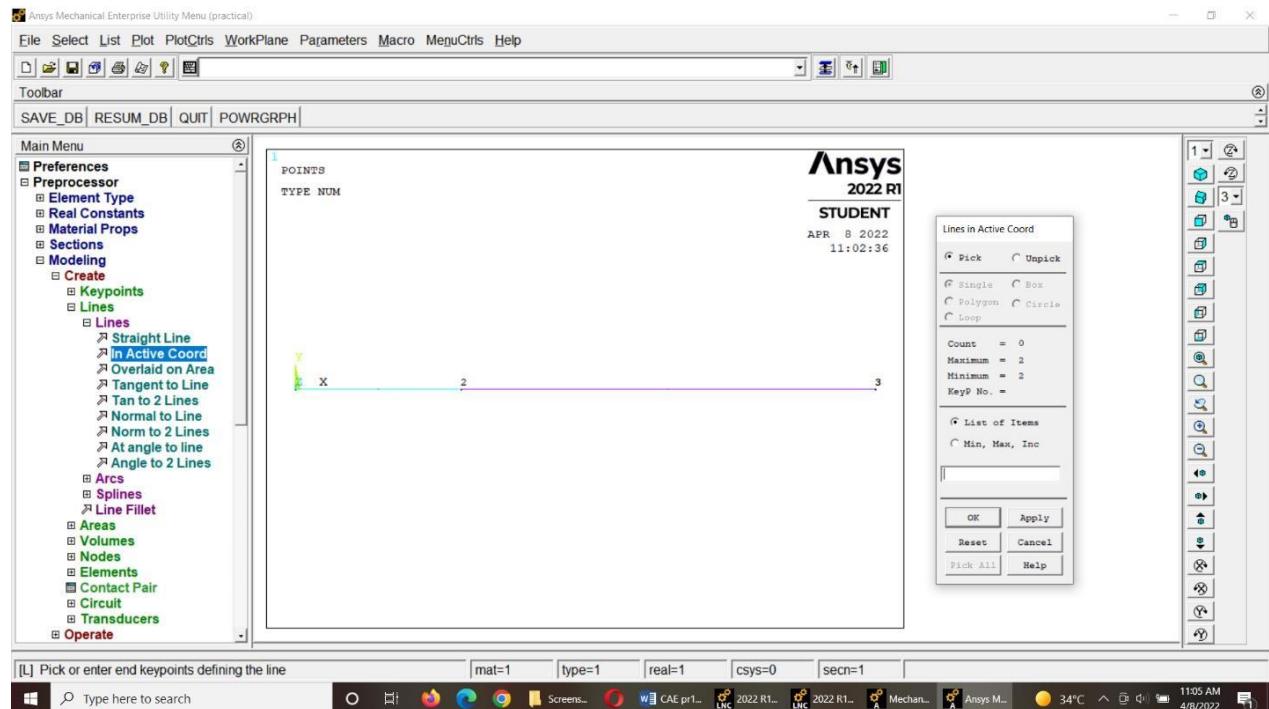
Step 5: Creating Key points:- modeling>create>key point>in active cs>select co-ordinate.



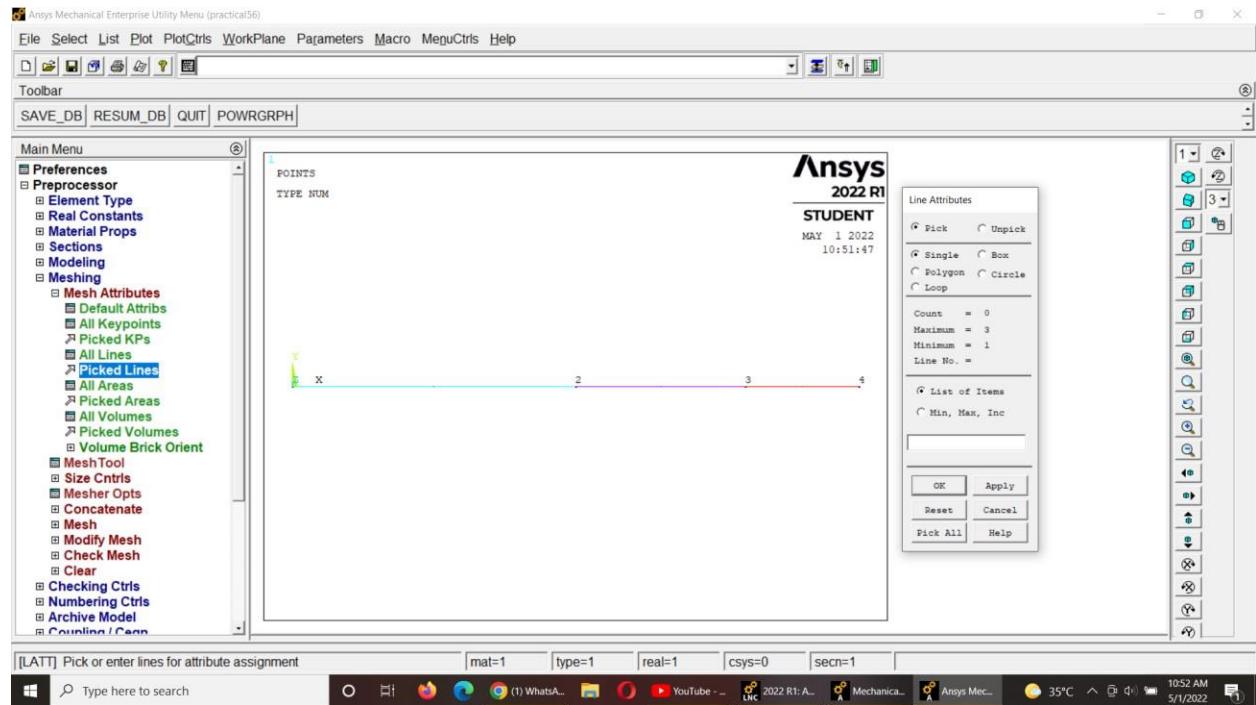
Step6: C Lines>Lines>In active co-ordinate cs> Join Co Ordinates

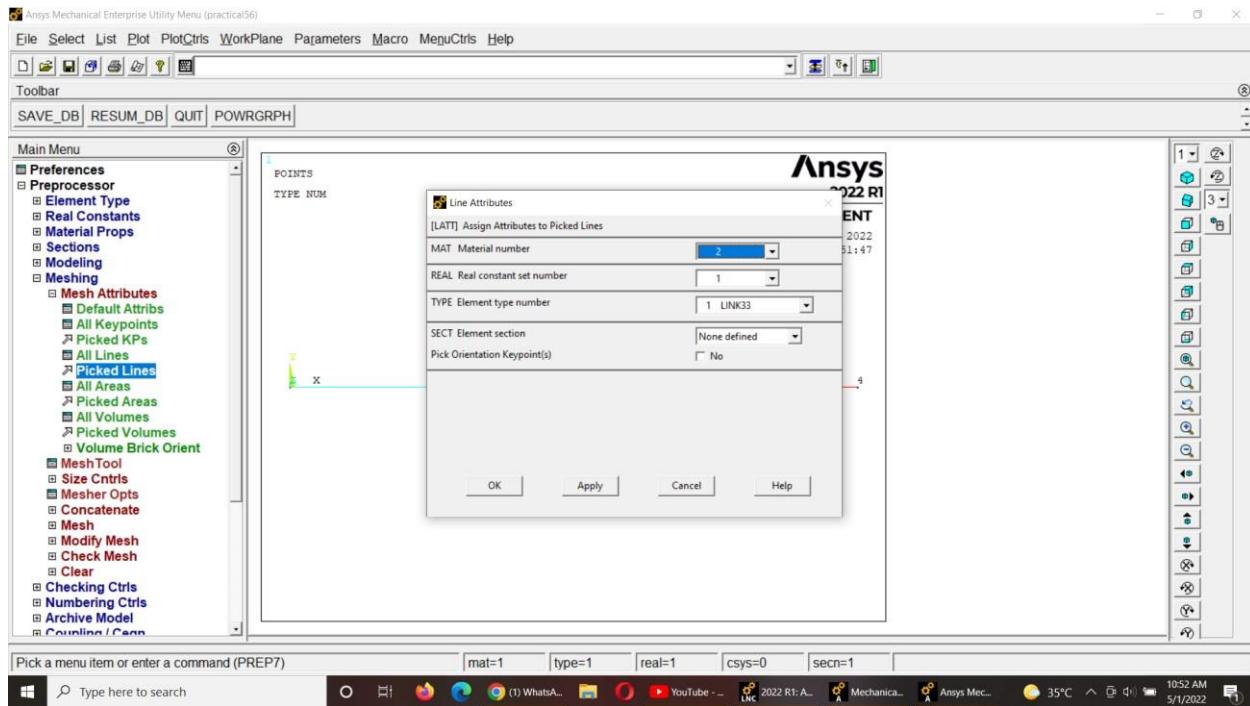


Step7: Lines>Lines>In active co-ordinate cs> Join Co Ordinates

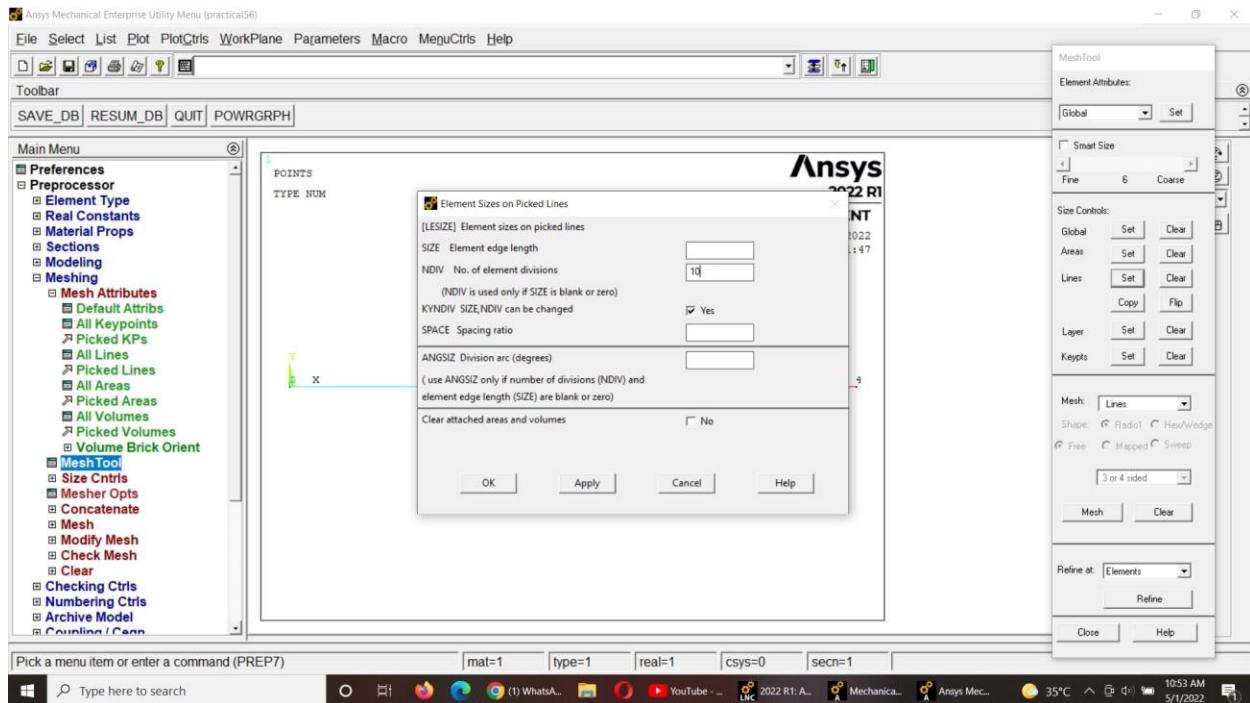


Step8: meshing:- Meshing> mesh attributes>picked lines>click on link>Select Material number>Apply

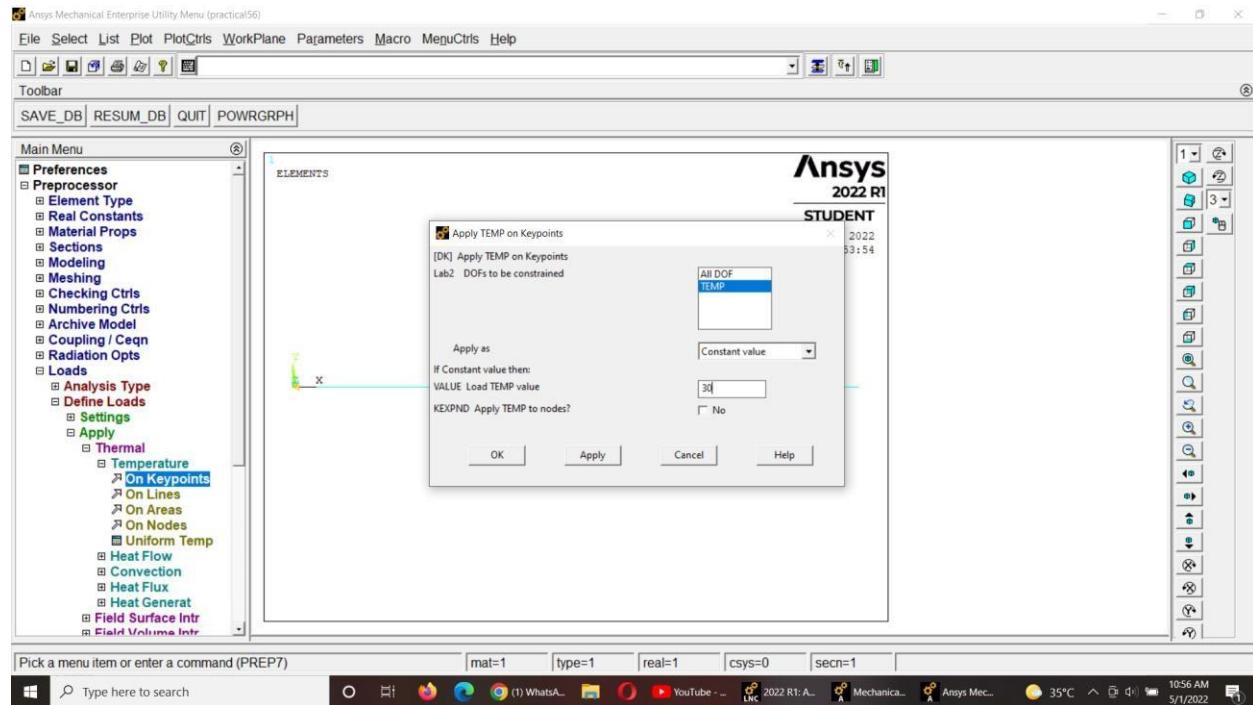




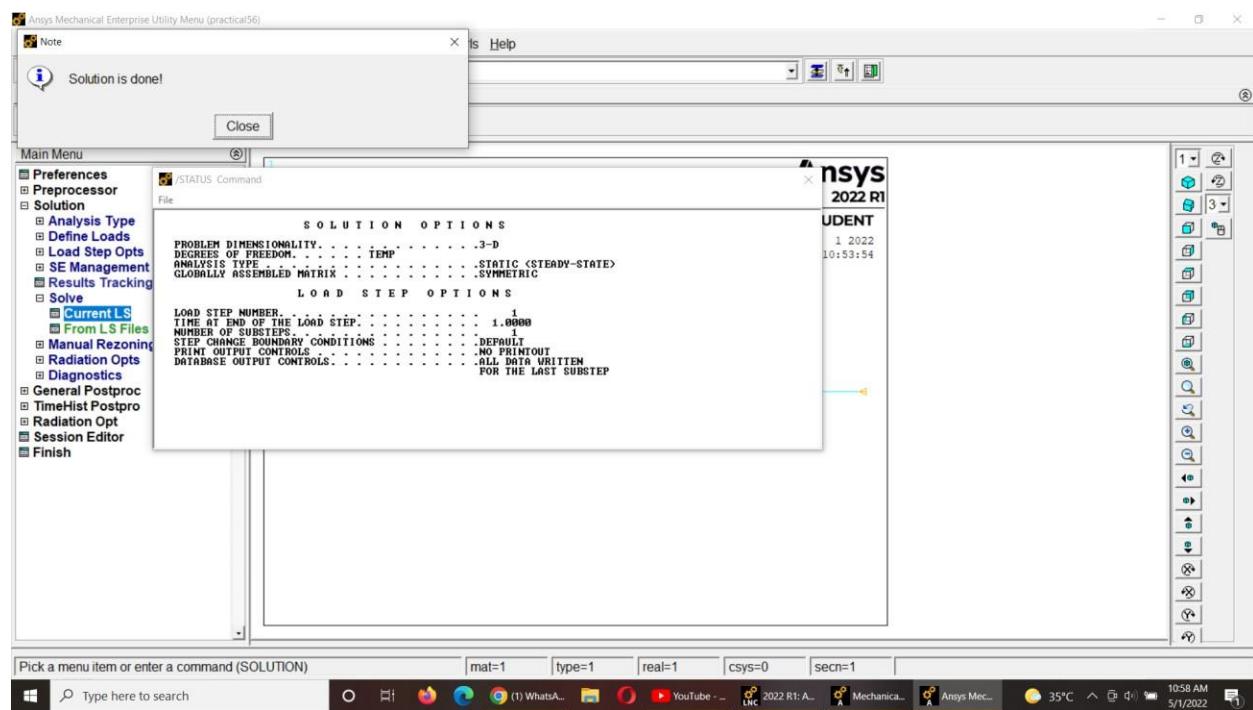
Step9: meshing:- Mesh Tools> select lines>lines>set>mesh>click on line>Apply



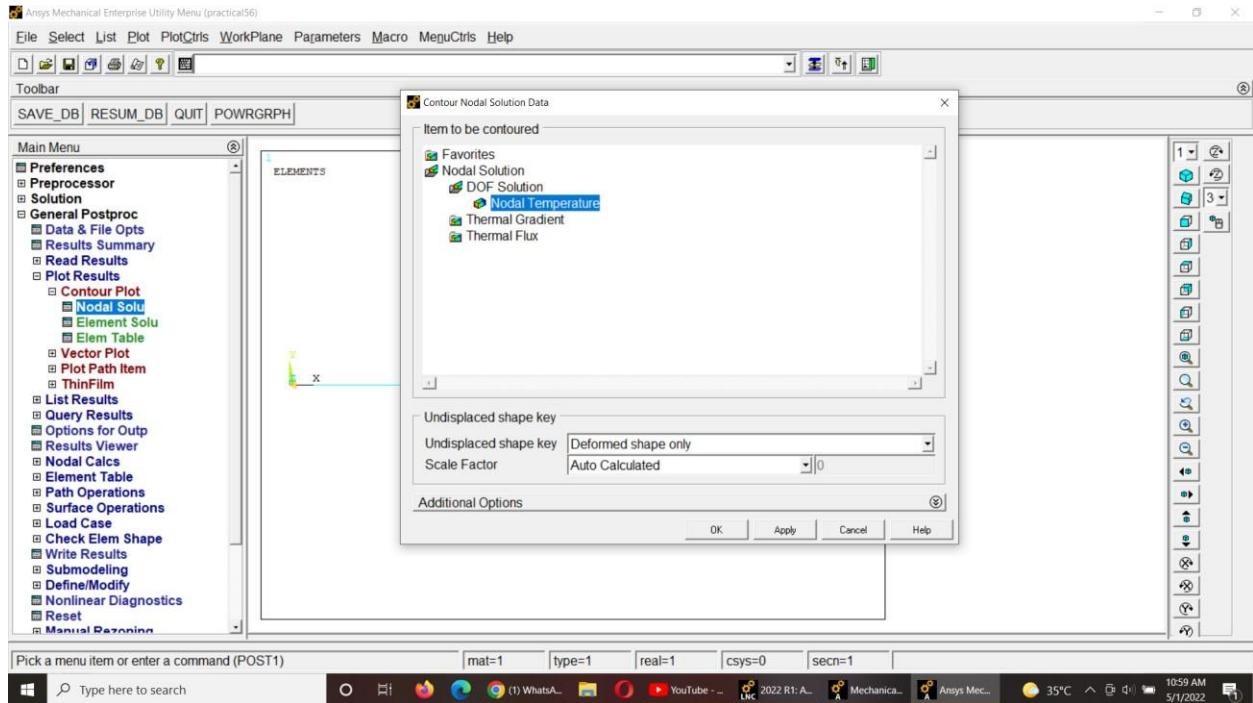
Step10: Apply loads: Loads>define loads>apply>Thermal>Temperature>on keypoints> select keypoints>Temp>Input temp>ok



Step11: Solution:- solution>solve>currentls> done

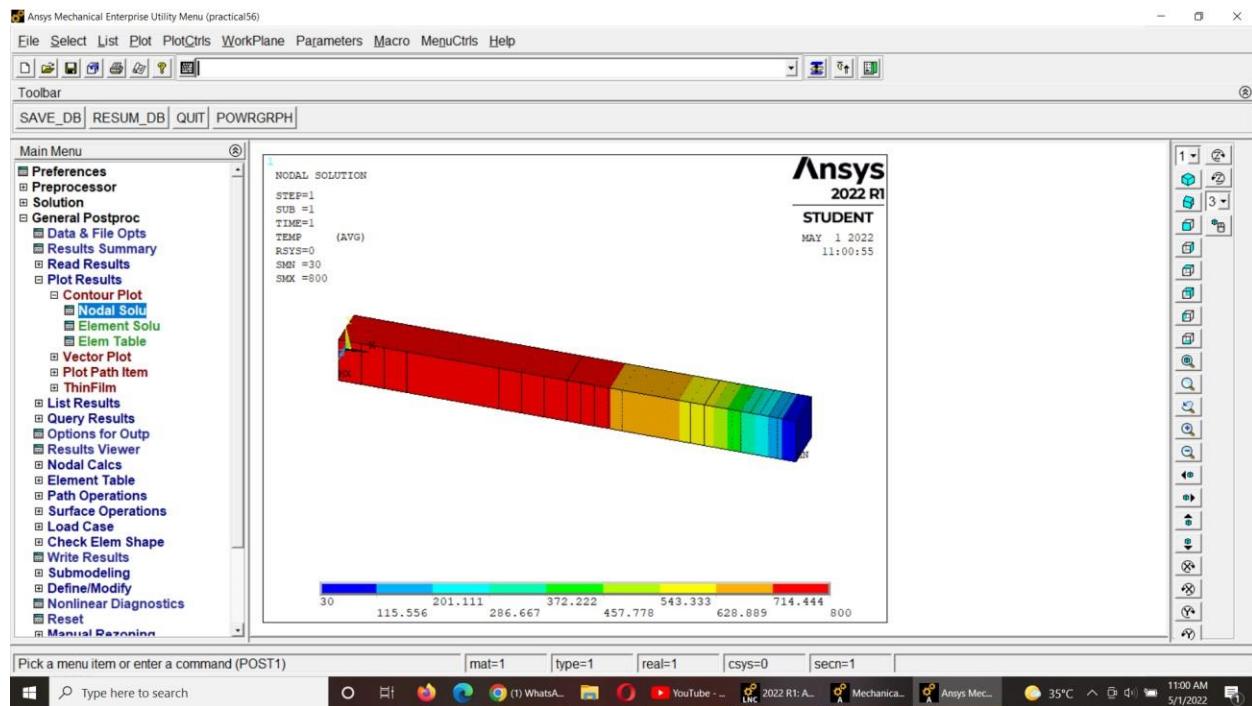
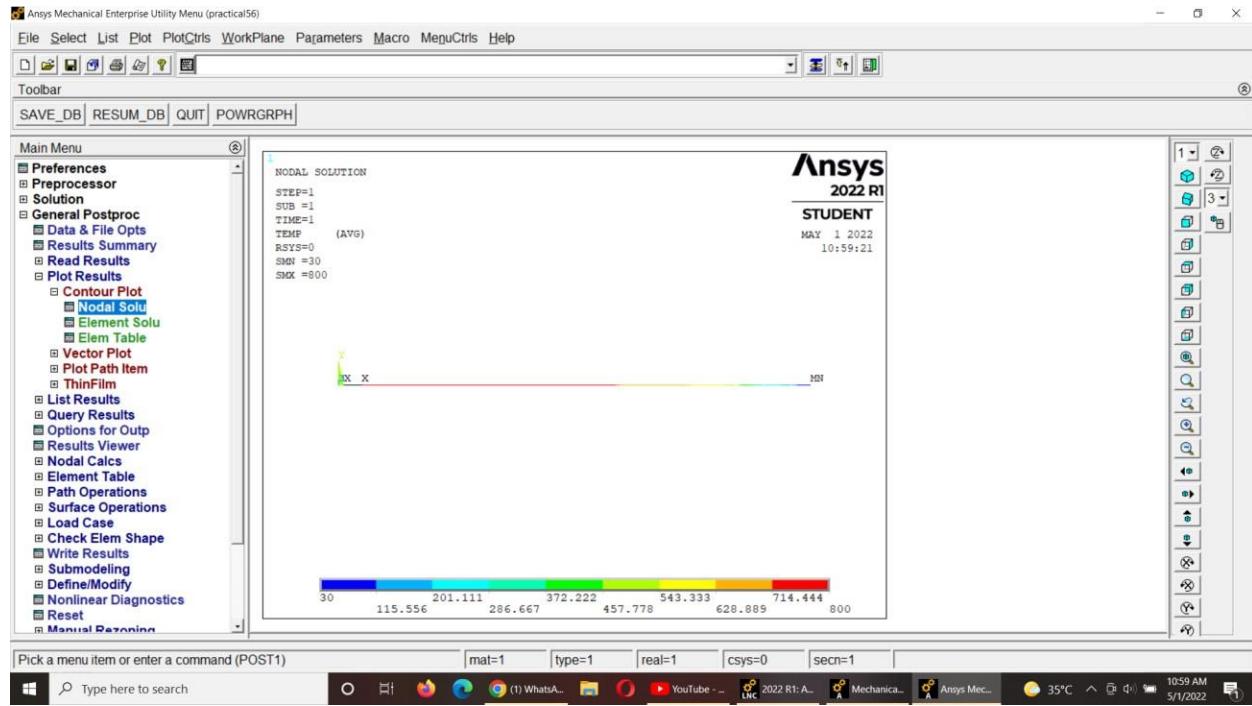


Step12: General postproc> read result> first set> plot result> Nodal solution> Dof solution >nodal temperature> apply.



RESULTS:-

Nodal Temp :-



PRINT TEMP NODAL SOLUTION PER NODE

******* POST1 NODAL DEGREE OF FREEDOM LISTING *******

LOAD STEP= 1 SUBSTEP= 1

TIME= 1.0000 LOAD CASE= 0

NODE TEMP

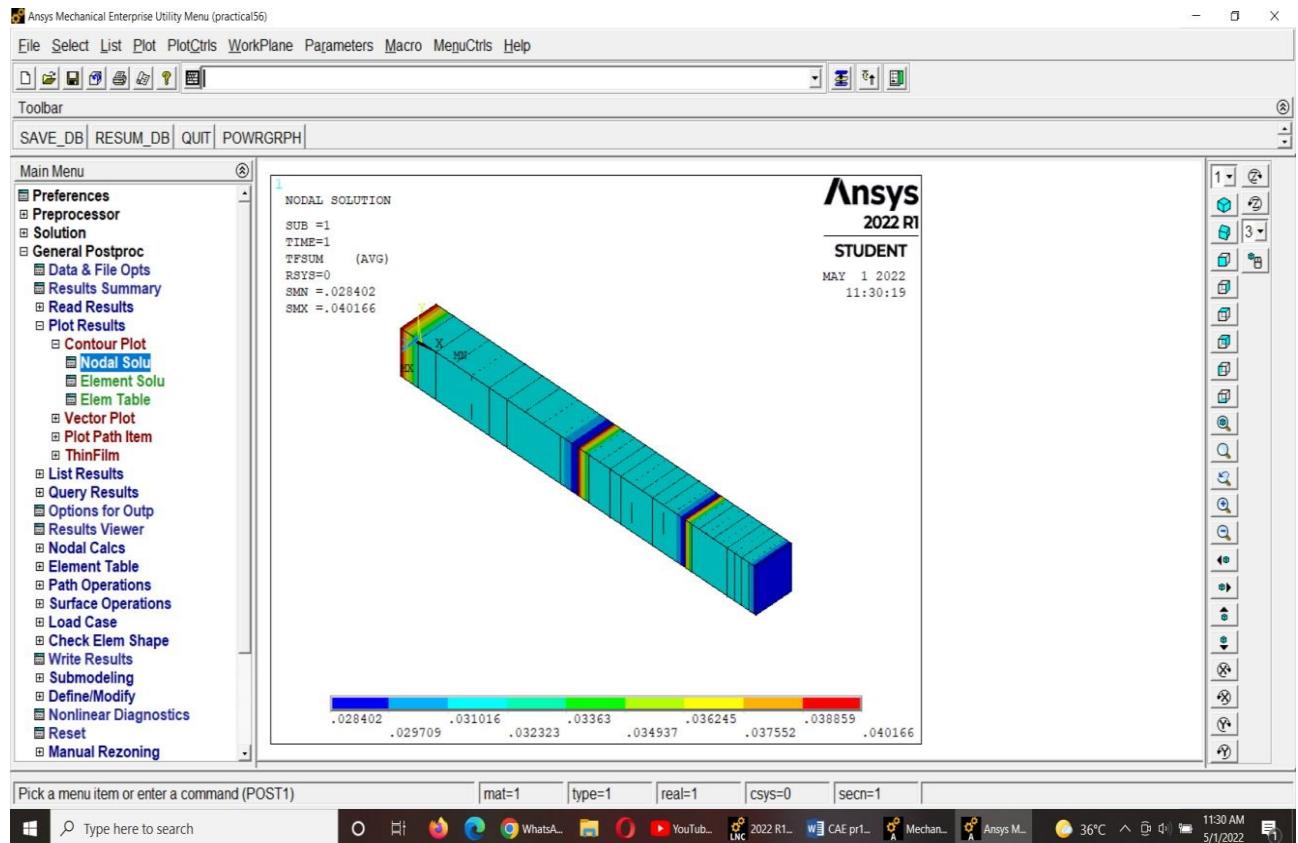
1	800.00
2	768.44
3	796.84
4	793.69
5	790.53
6	787.38
7	784.22
8	781.07
9	777.91
10	774.75
11	771.60
12	598.03
13	751.40
14	734.36
15	717.32
16	700.28

17	683.24
18	666.20
19	649.16
20	632.11
21	615.07
22	30.000
23	541.23
24	484.43
25	427.62
26	370.82
27	314.02
28	257.21
29	200.41
30	143.61
31	86.803

MAXIMUM ABSOLUTE VALUES

NODE	1
VALUE	800.00

Thermal flux :-



CONCLUSION:-

Thus by comparing analytical and software solution we have got

Max thermal flux occurs is **.040166 w/m²**

NAME : SOURAV SANTAJI GUJALE

CLASS: TE MECH 1

SEMESTER/YEAR: 6

ROLL NO.: 61

DATE OF PERFORMANCE:

DATE OF SUBMISSION:

EXAMINED BY:

EXPERIMENT NO:6

AIM OF EXPERIMENT:- Modal analysis – Cantilever beam. Aim: To determine natural frequency of cantilever beam using FEA package.

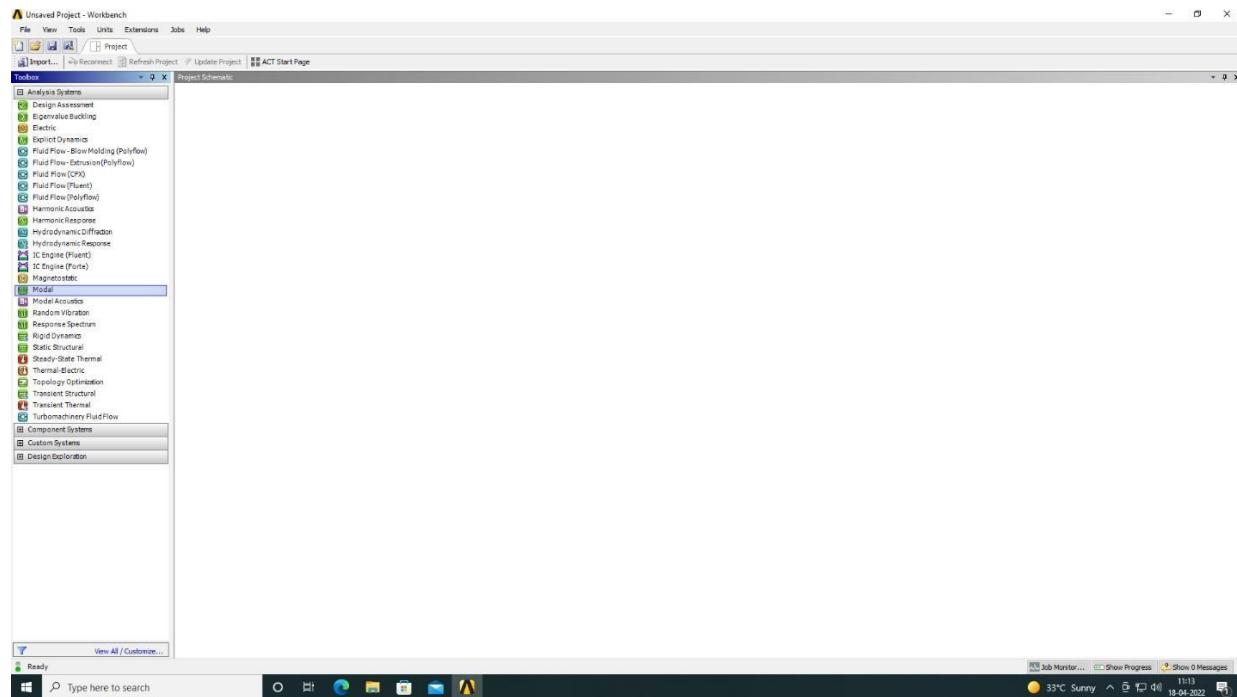
Consider a Beam



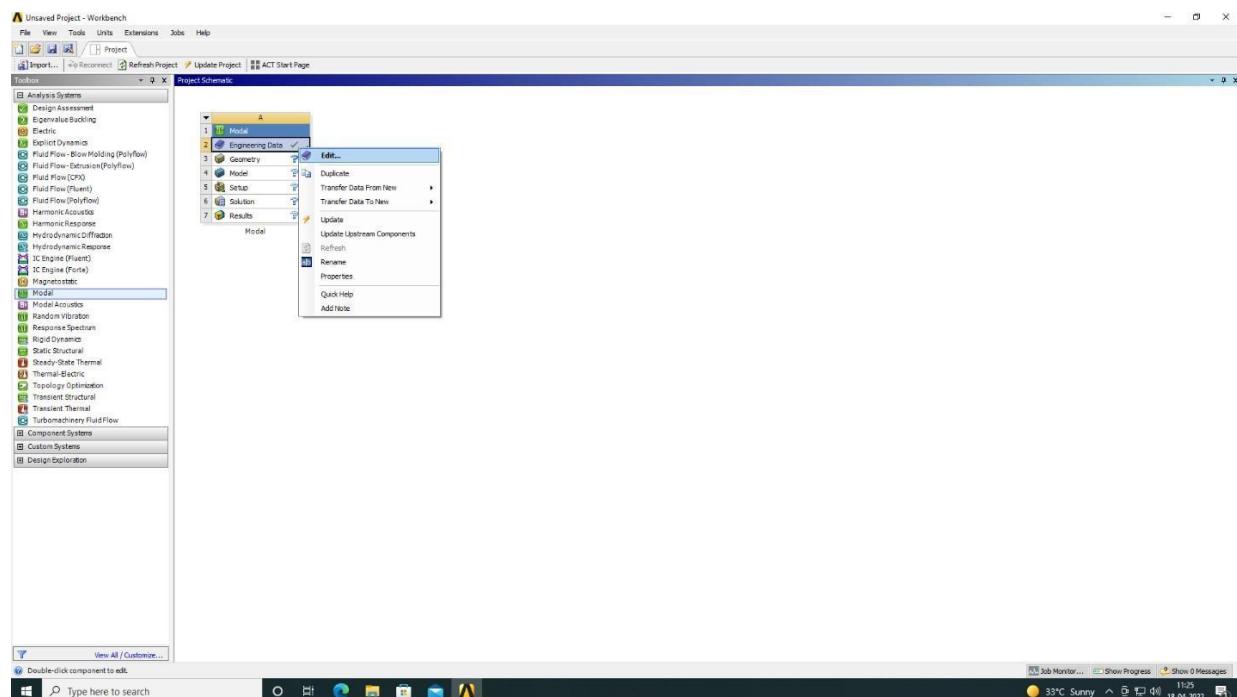
$$E = 2 \times 10^5 \text{ MPa}$$

$$A = 10 \times 10 \text{ mm} \quad u = 0.3$$

Step 1: Open Ansys 2022 Workbench > Analysis Systems > Modal > Ok.



Step 2: Engineering data>Edit>ok



Step 3: Check data > Close

The screenshot shows the A2i Engineering Data interface with several windows open:

- Outline of Schematic A2i: Engineering Data**: Shows a tree structure with 'Material' selected.
- Table of Properties Row 2: Structural Steel Field Variables**: A table with columns A, B, C, D, and E. It lists 'Temperature' with unit 'C' and value '22'.
- Properties of Outline Row 3: Structural Steel**: A detailed table of material properties. Key entries include:

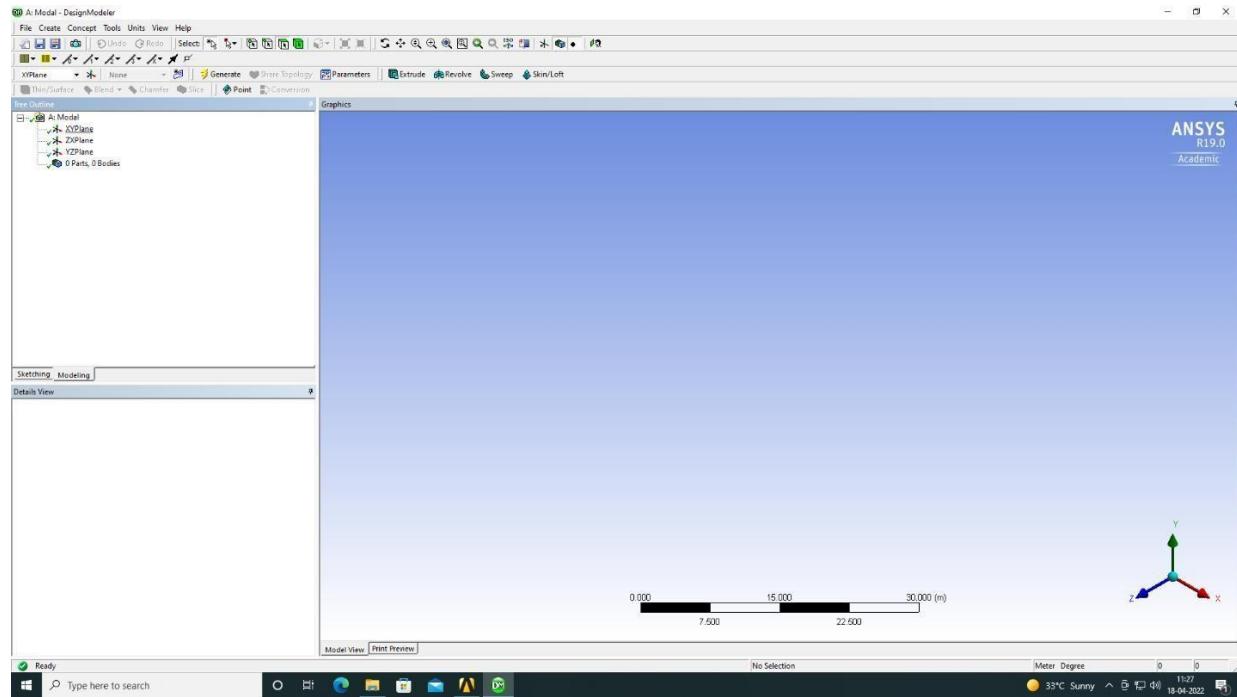
A	B	C
Property	Value	Unit
Density	7850	kg m ⁻³
Modulus of Elasticity	1.2E-05	C ⁻¹
Young's Modulus	2E+11	Pa
Poisson's Ratio	0.3	
Bulk Modulus	1.6667E+11	Pa
Shear Modulus	7.692E+10	Pa
- Chart: No data**: An empty chart window.

Step 4: Geometry> New design modeler Geometry>ok

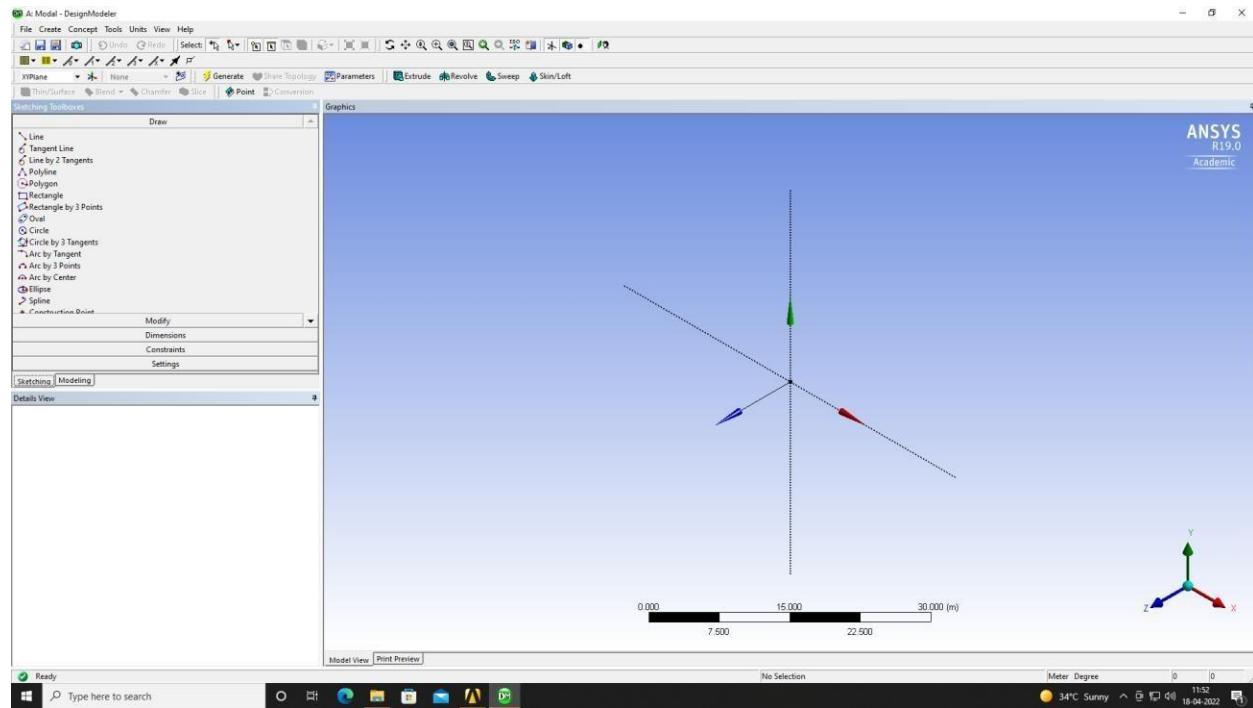
The screenshot shows the Project Schematic interface with the following details:

- Project Schematic**: A tree view under the 'Model' node. The 'Geometry' node has a context menu open, with the option 'New DesignModeler Geometry...' highlighted.
- Toolbox**: Shows various analysis and component systems.
- Task List**: Shows tasks like 'Import...', 'Refresh Project', and 'ACT Start Page'.
- Windows Taskbar**: Shows system icons and the date/time (18-04-2022).

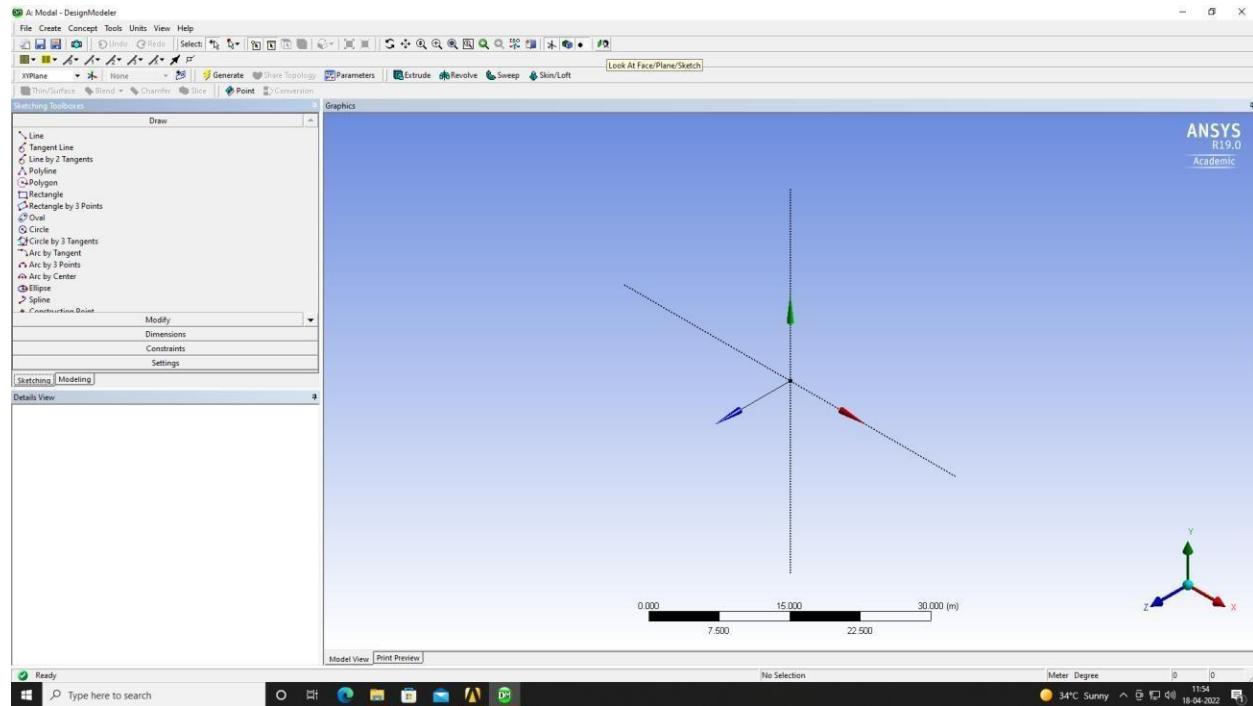
Step 5: Select XY Plane.



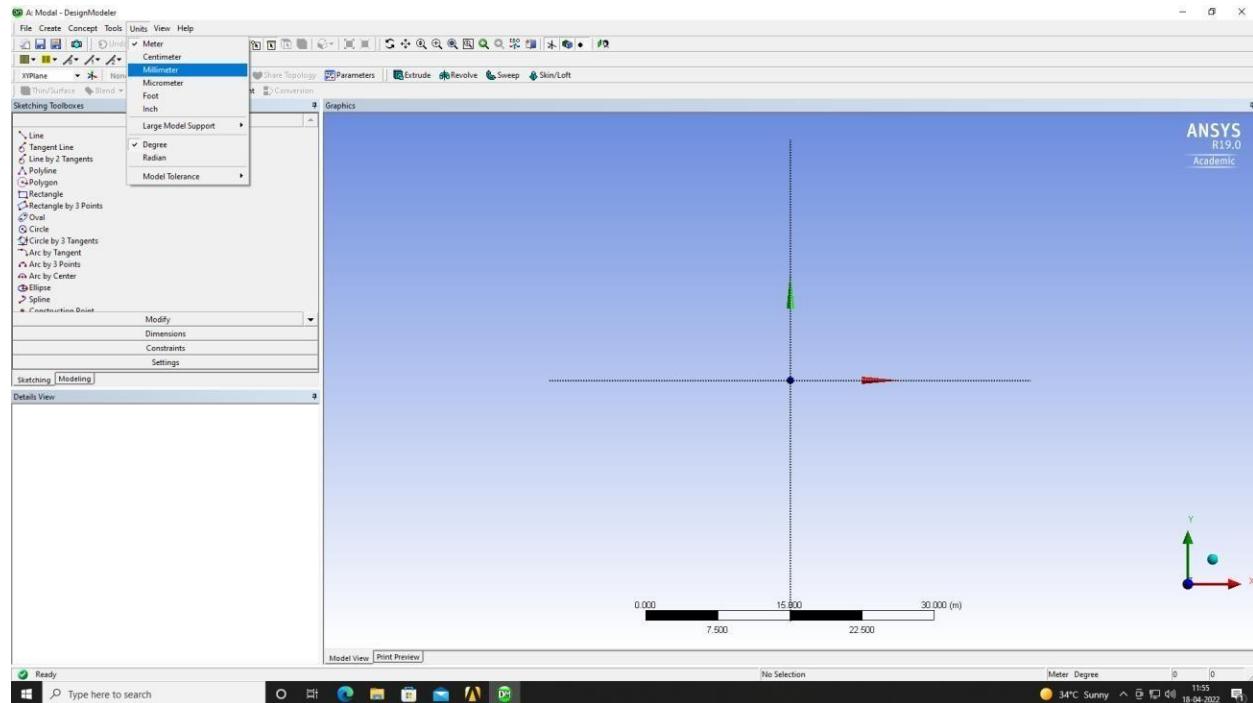
Step6: Select sketching



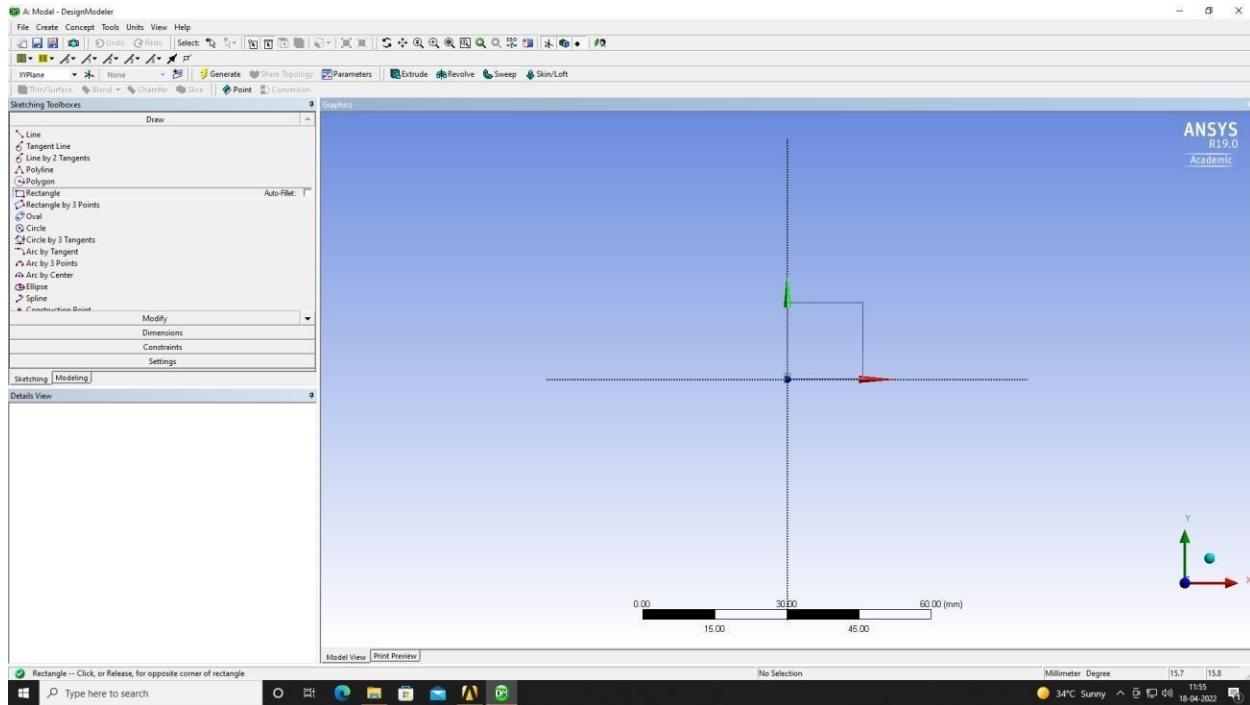
Step 7: Select Look At Face



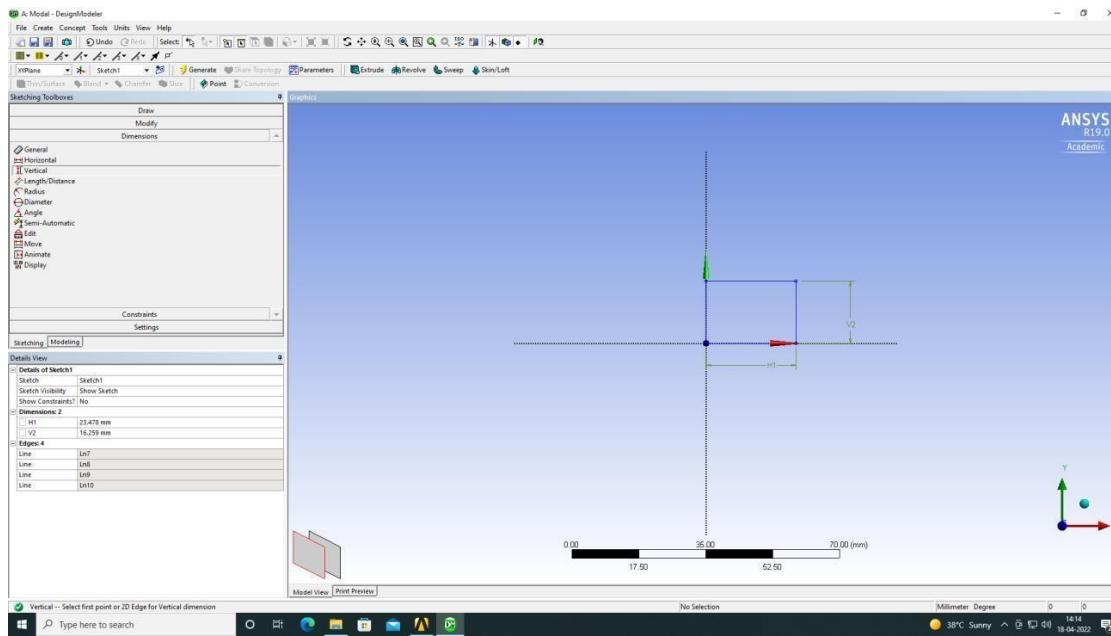
Step8: change unit to millimeter



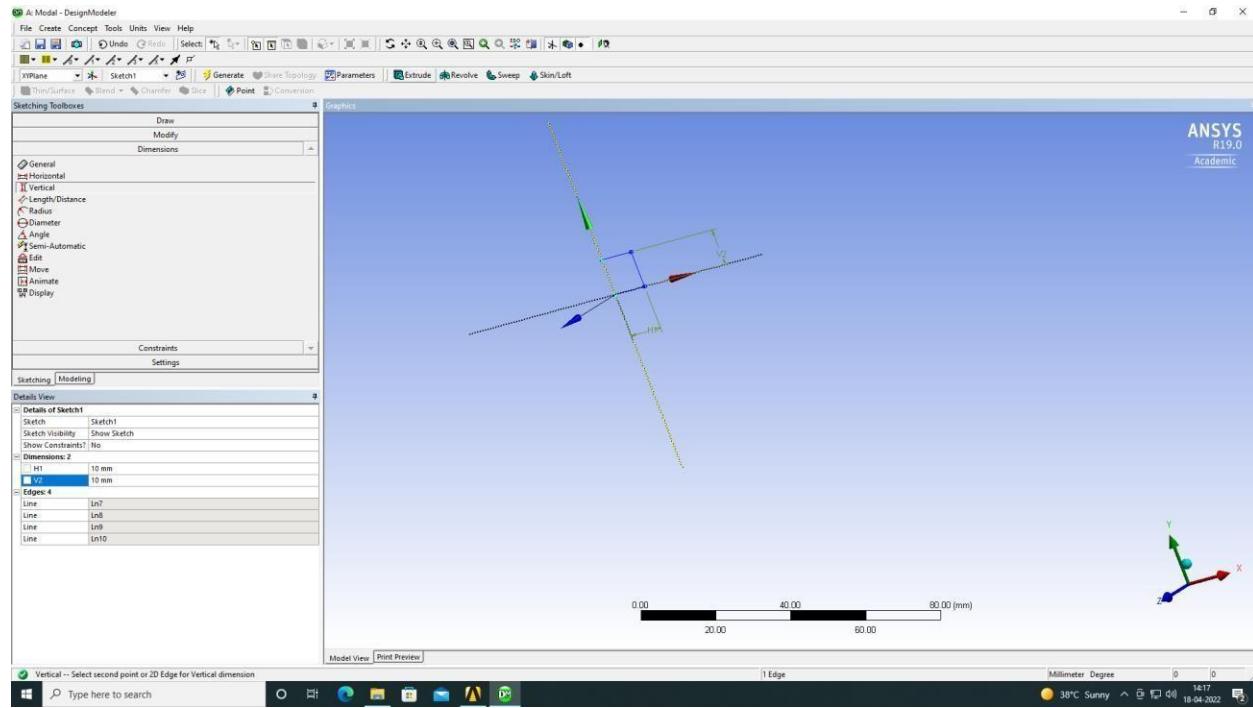
Step 9: Select and draw rectangle from origin.



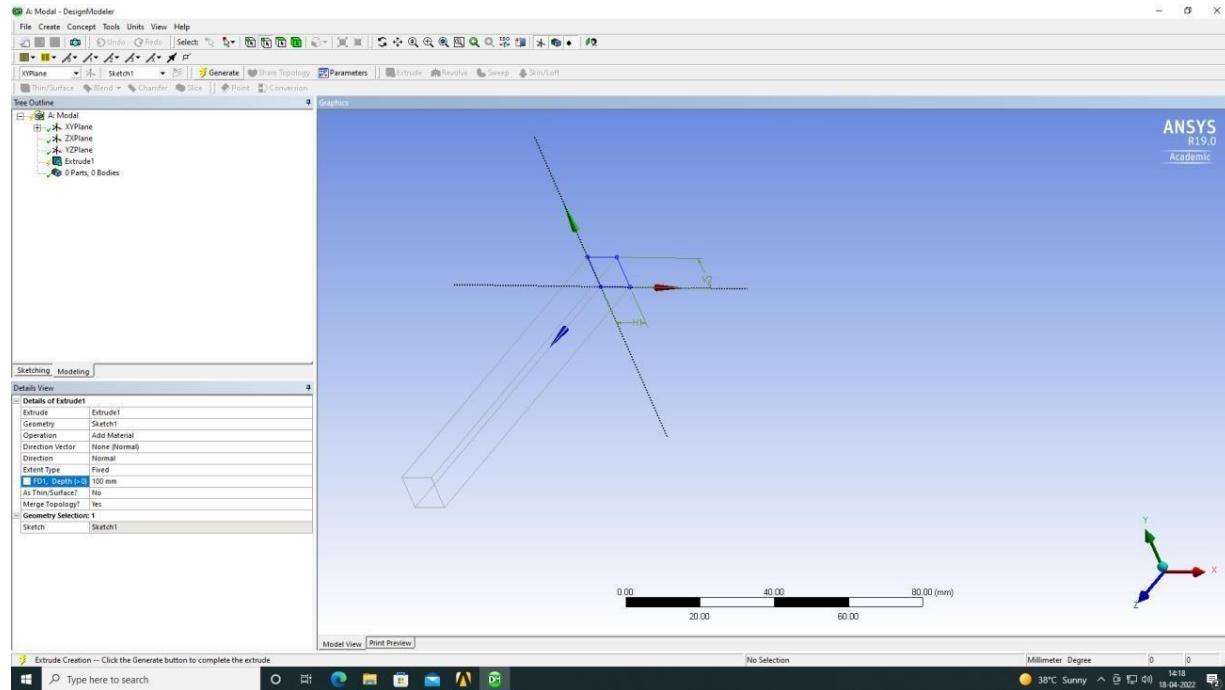
Step10: slect dimension> allot dimension to rectangle



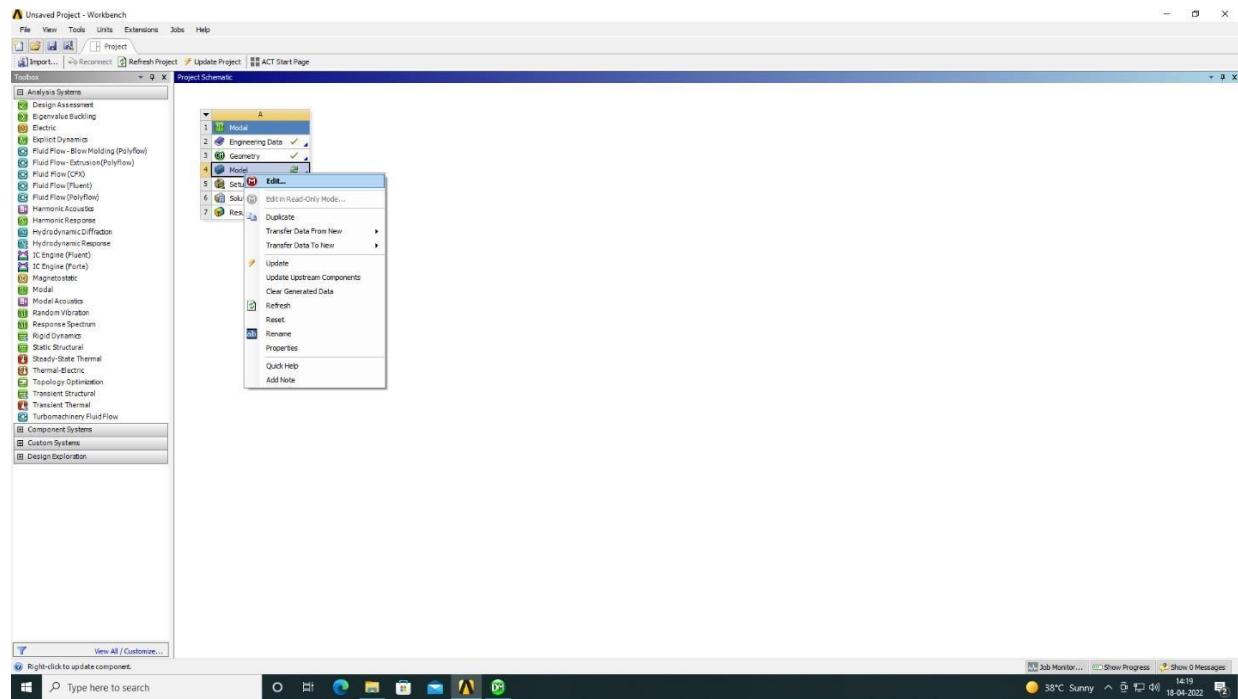
Step 11: Change dimensions from detail panel.



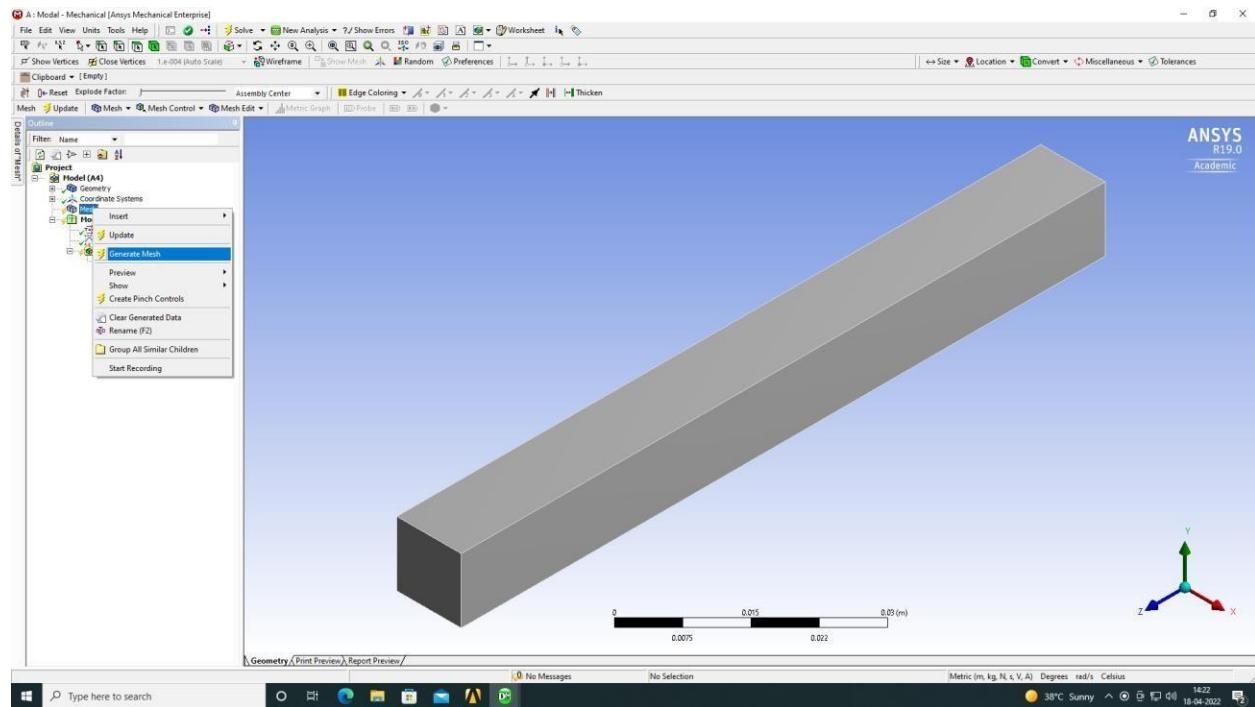
Step12: select extrude from command panel>select depth from detail panel>Generate

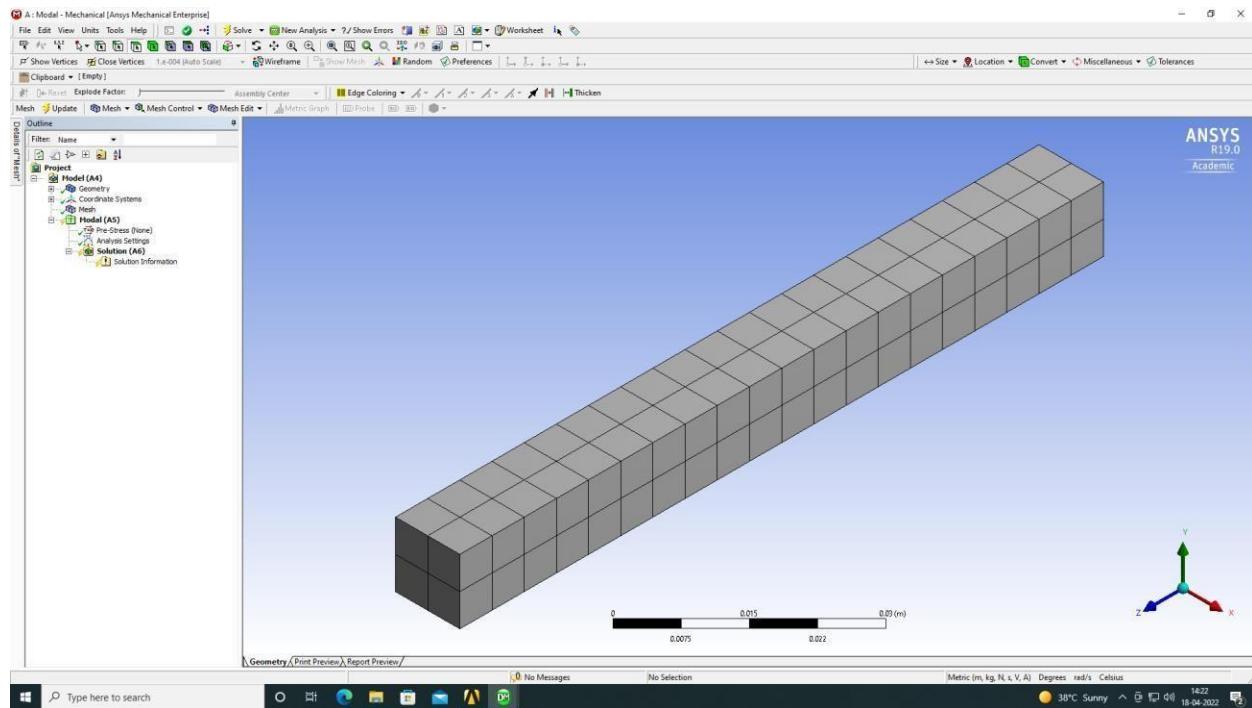


Step 13: Workbench > Modal > Edit > Ok.

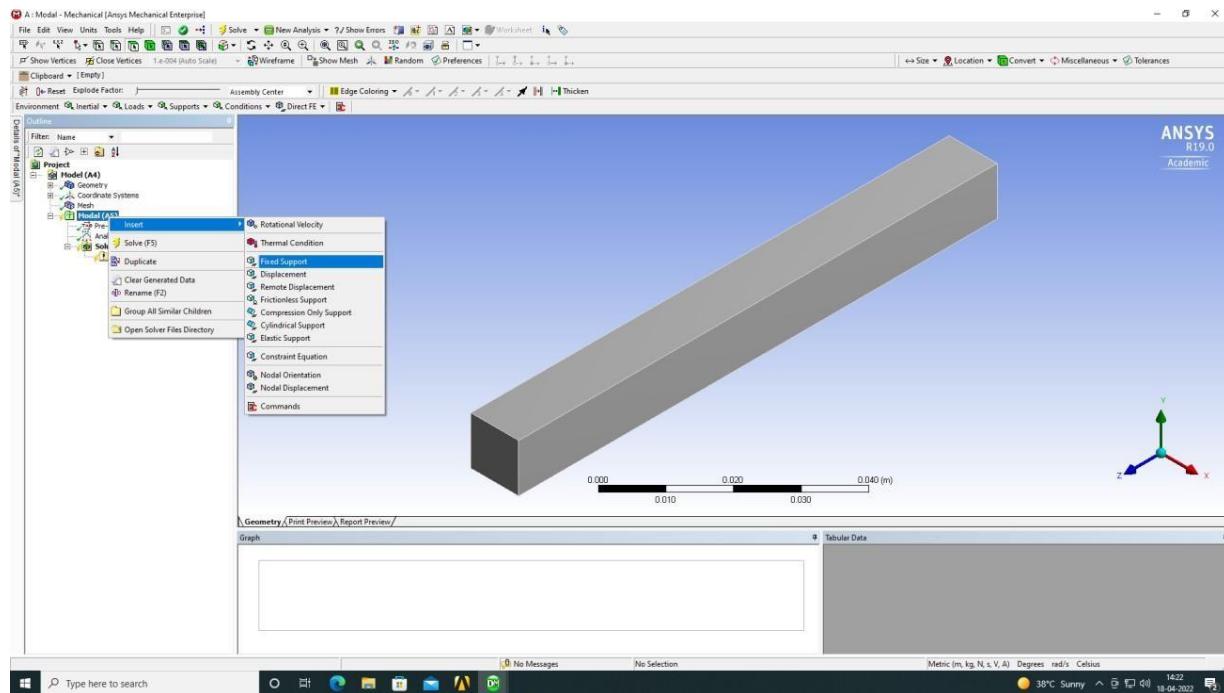


Step14: Mesh> Generate> ok

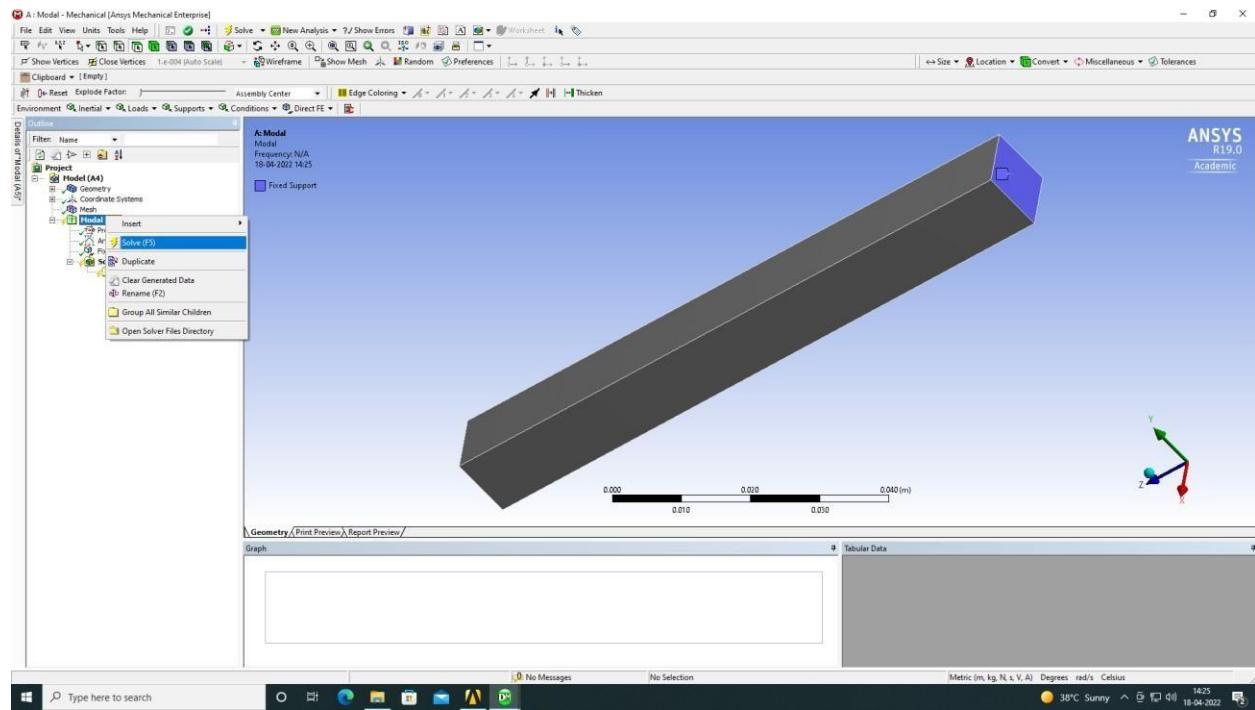




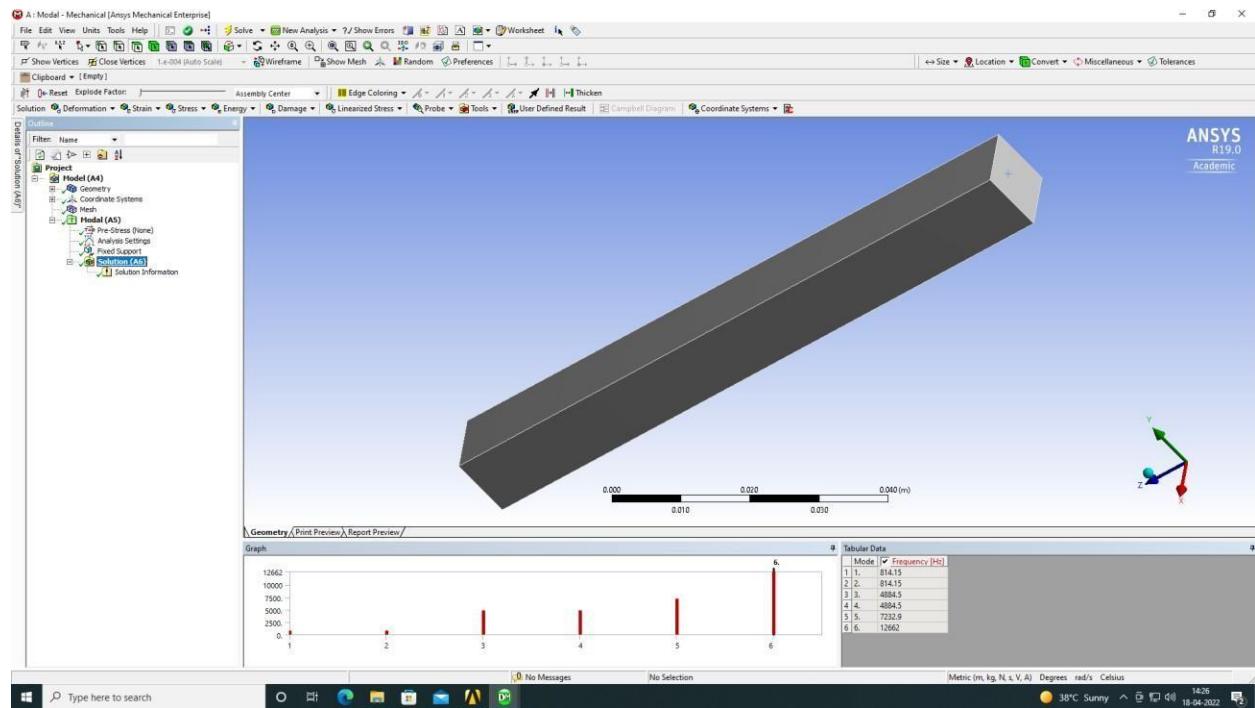
Step15: Step 16: Modal > Insert > Fixed support > Select one end of beam > click on apply detail panel.



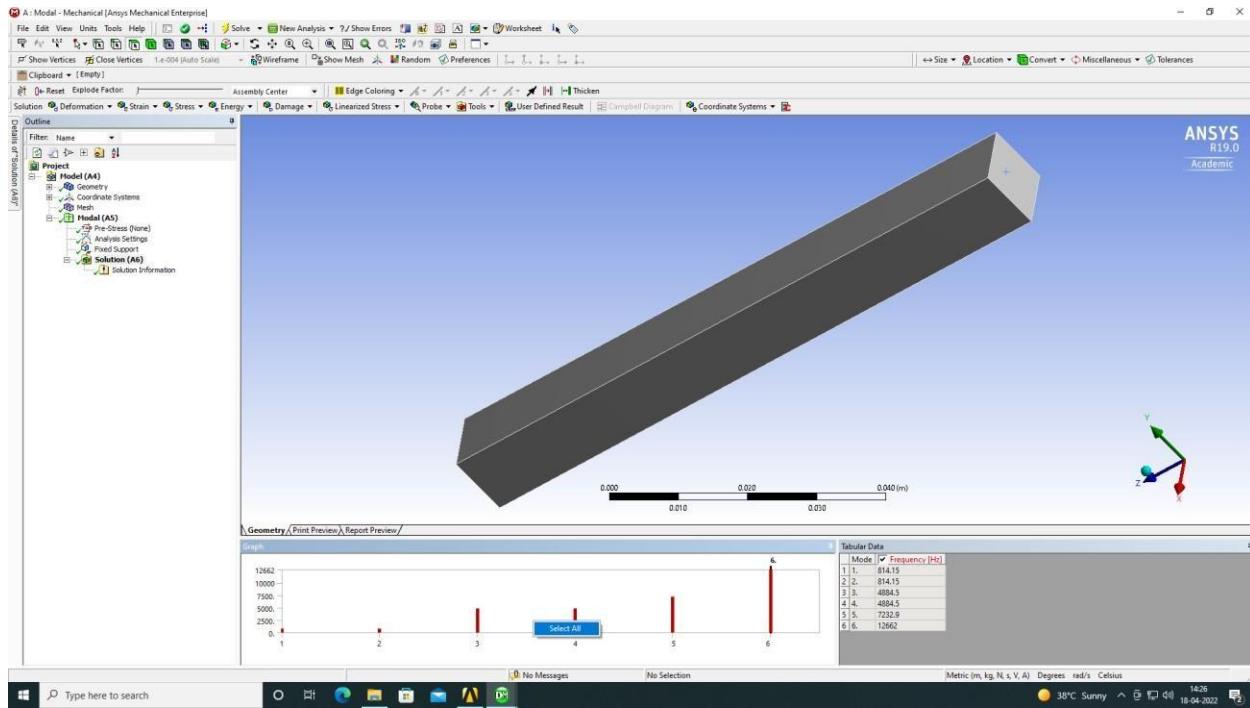
Step 17: Modal > Solve > Ok.



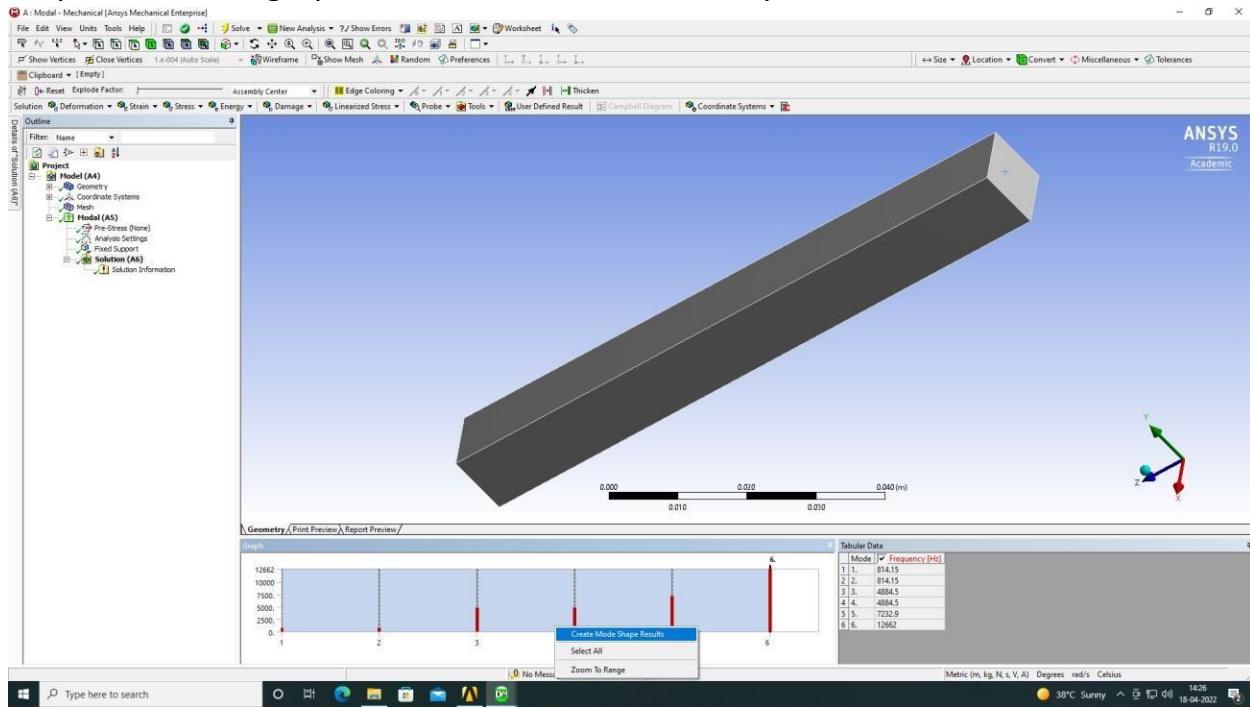
Step 18: Click on solution.



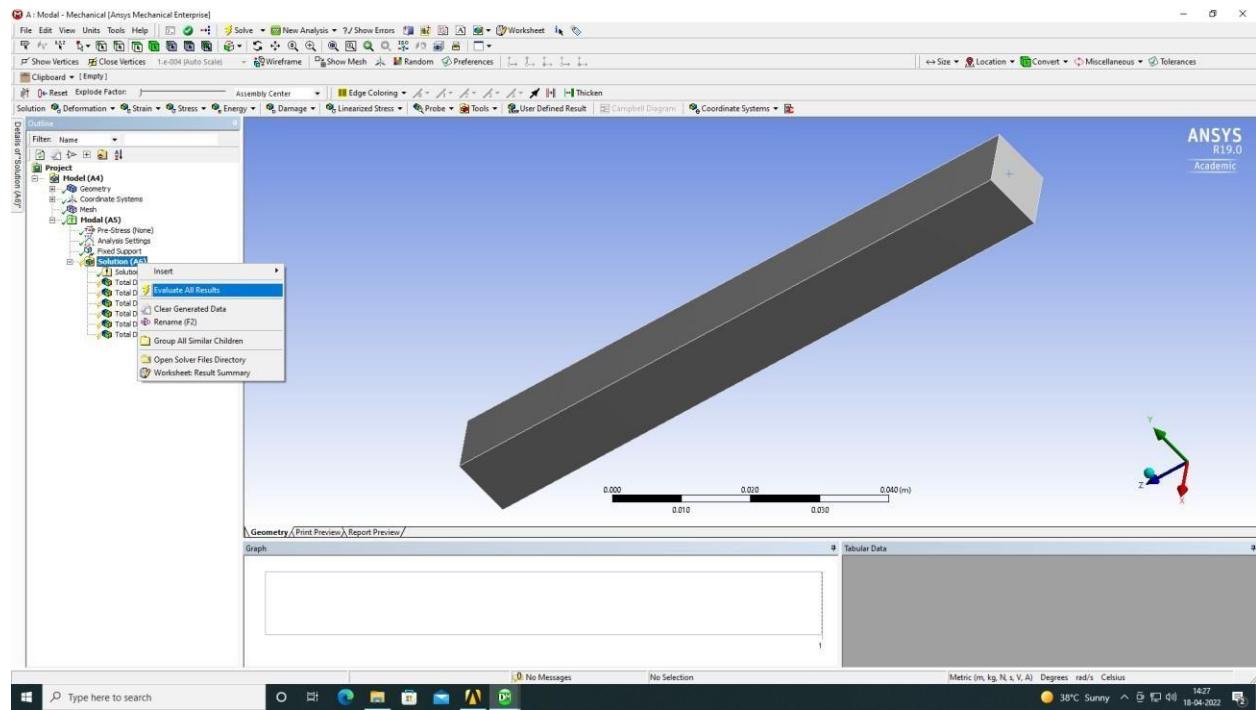
Step 19: Click on graph > click on select all.



Step 20: Click on graph > click on create mode shape results.



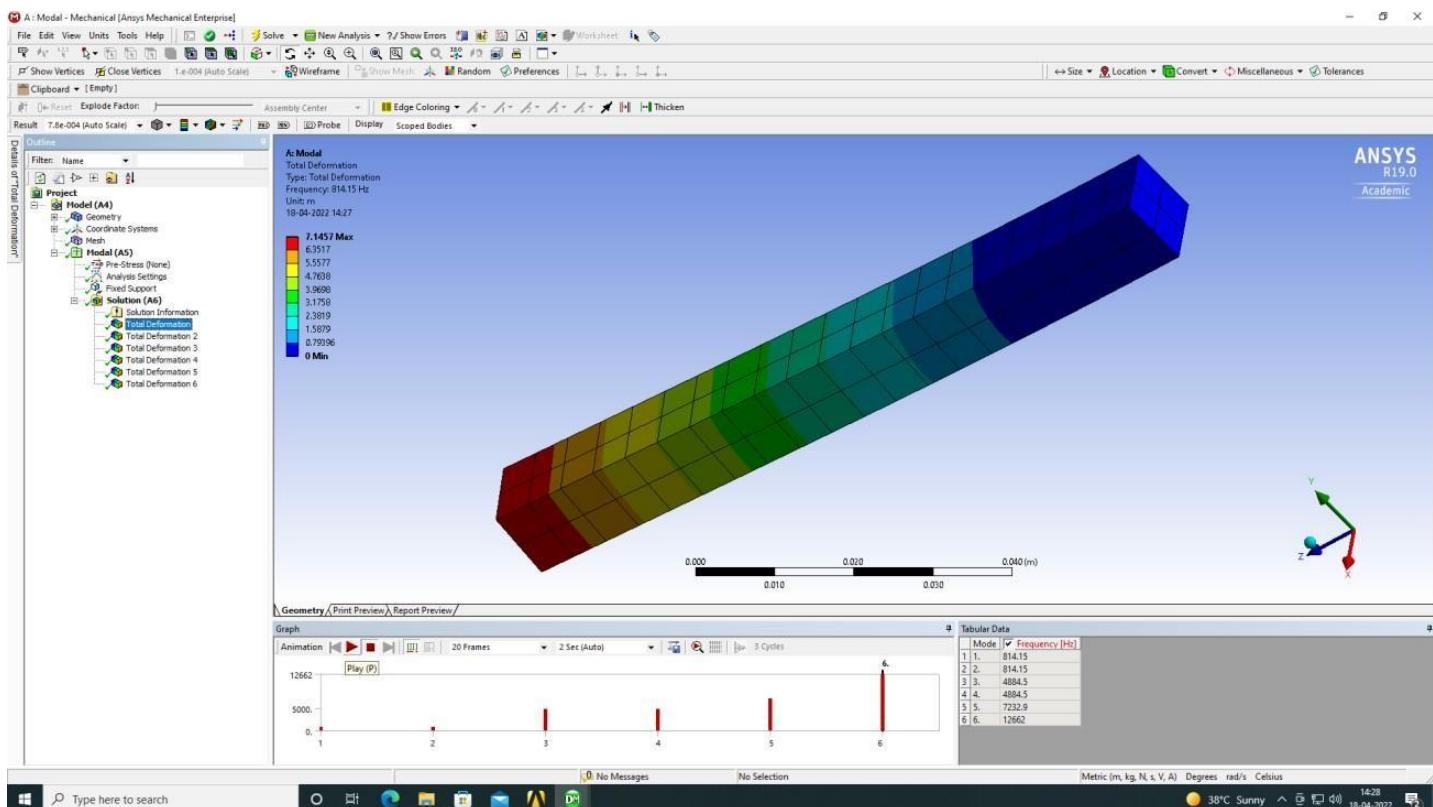
Step 21: Solution > Evaluate all result > Ok.



Step 22: Solution >Total deformation.

RESULTS:-

NODAL DISPLACEMENT:-



CONCLUSION:-

SO HERE BY ANALYSIS WE HAVE GOT MAX. DISPLACEMENT IS **2.1457 MM**
AT FREQUENCY OF **814.15 Hz**

NAME : SOURAV SANTAJI GUJALE

CLASS: TE MECH 1

SEMESTER/YEAR: 6

ROLL NO.: 61

DATE OF PERFORMANCE:

DATE OF SUBMISSION:

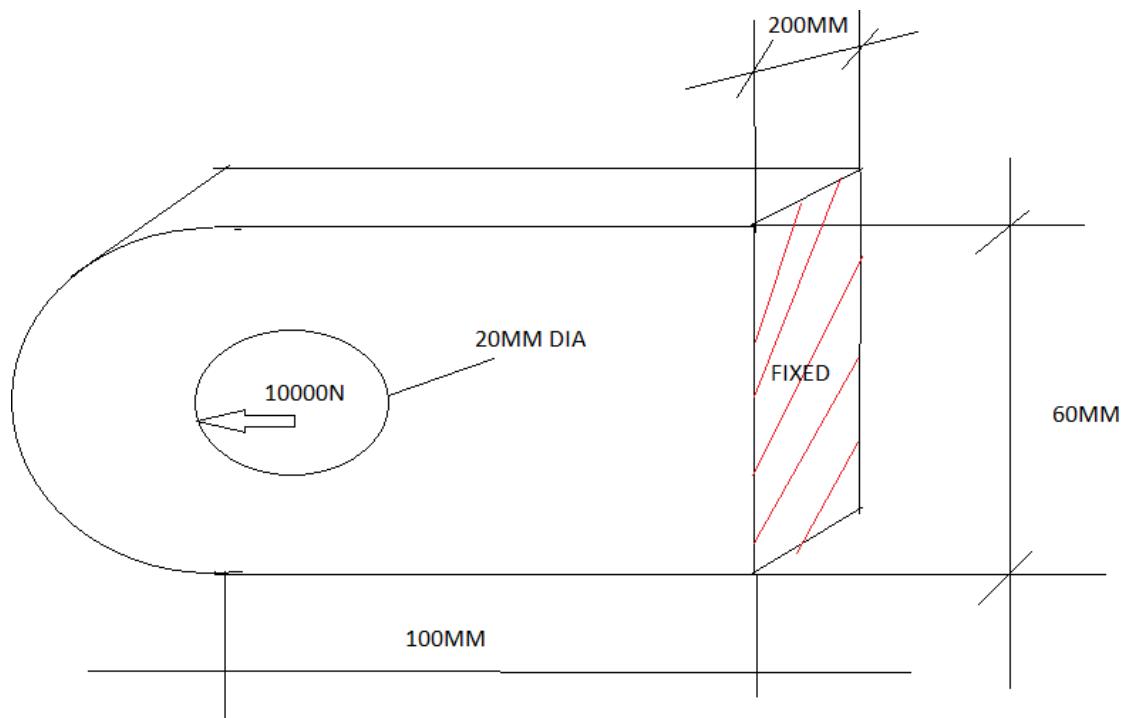
EXAMINED BY:

EXPERIMENT NO: 7

AIM OF EXPERIMENT:-Stress and deformation analysis of 3D element using finite element package.

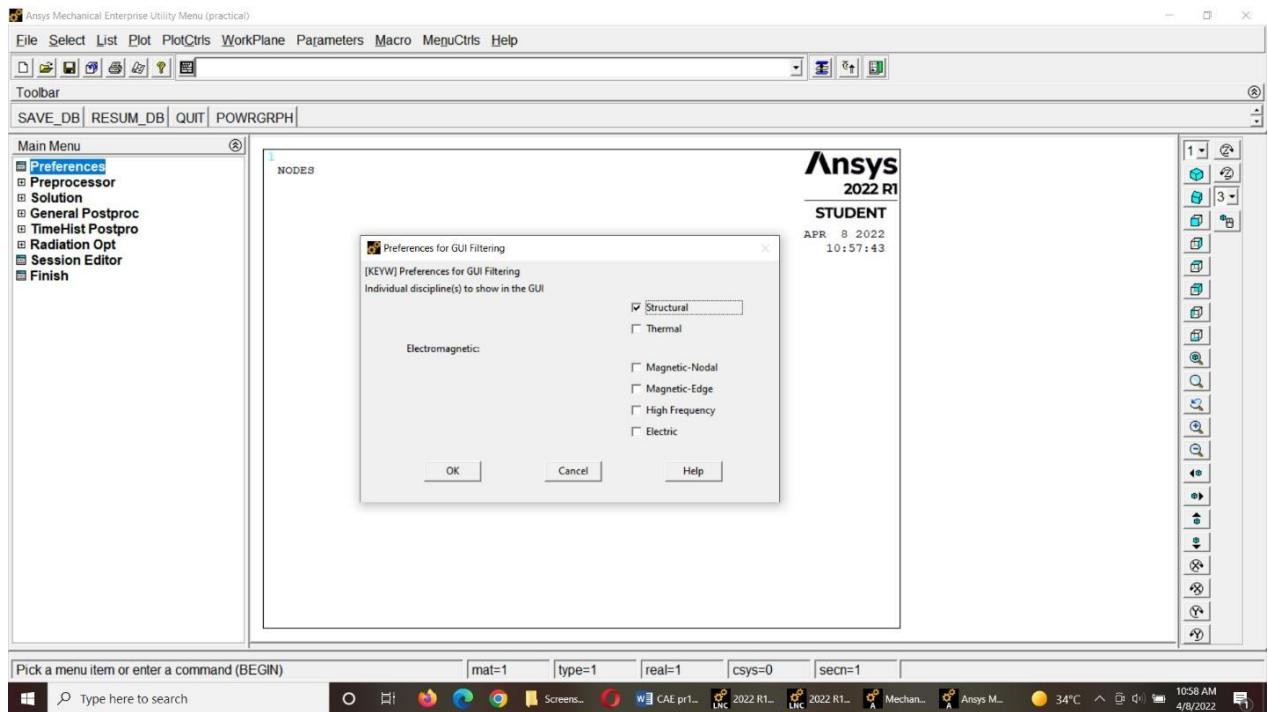
Finite Element Package: ANSYS 2022

Stress distribution in a shell with applied load.

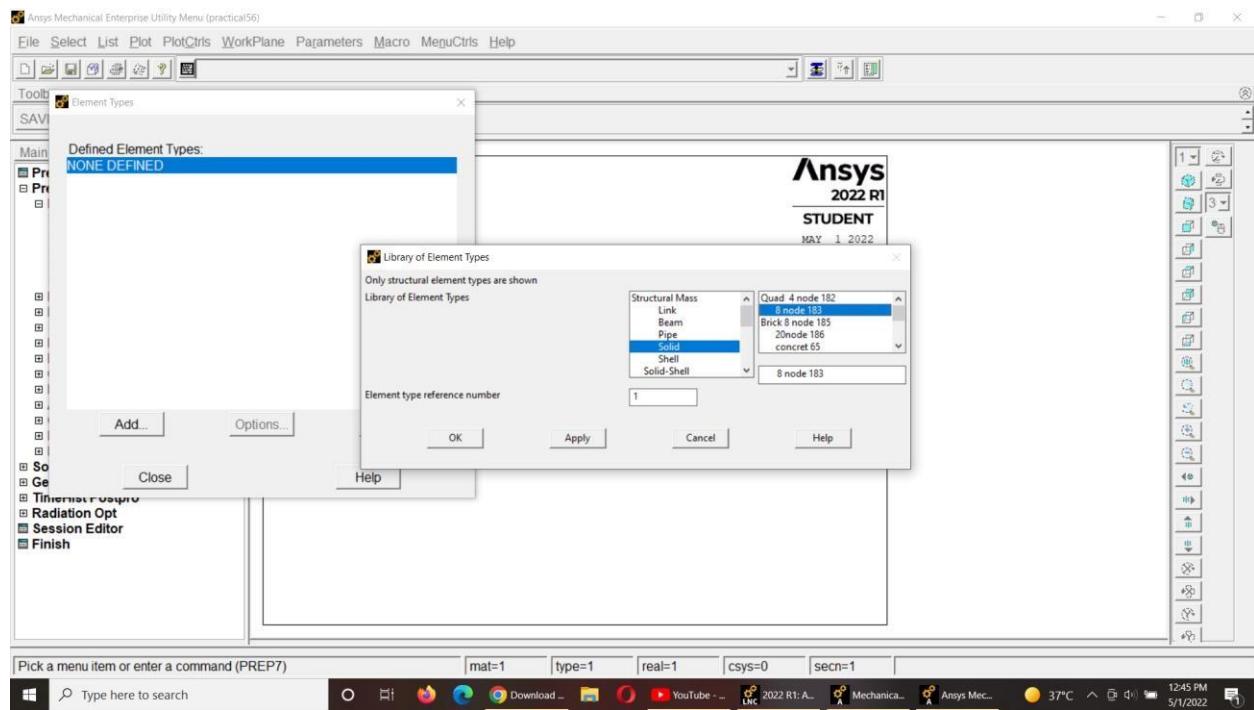


$$E = 2 \times 10^5 \text{ MPa} \quad u = 0.3$$

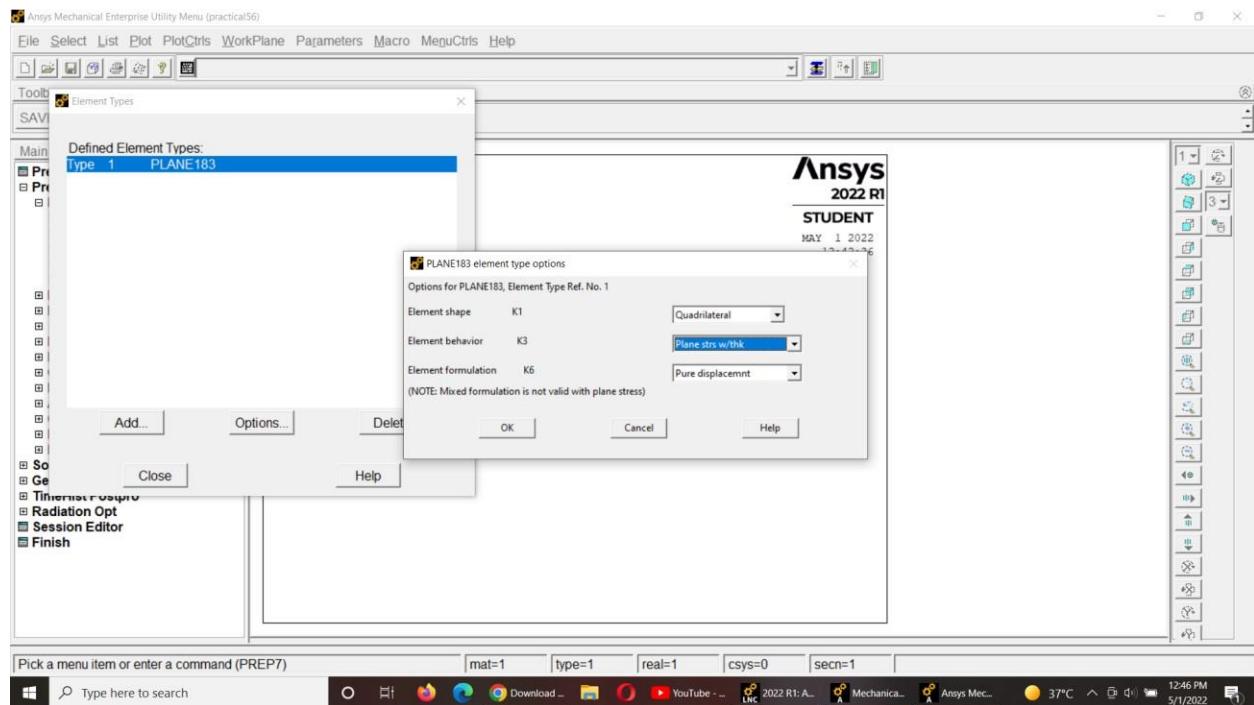
Step 1: Select type of Analysis---- Preferences> structural>Press Ok



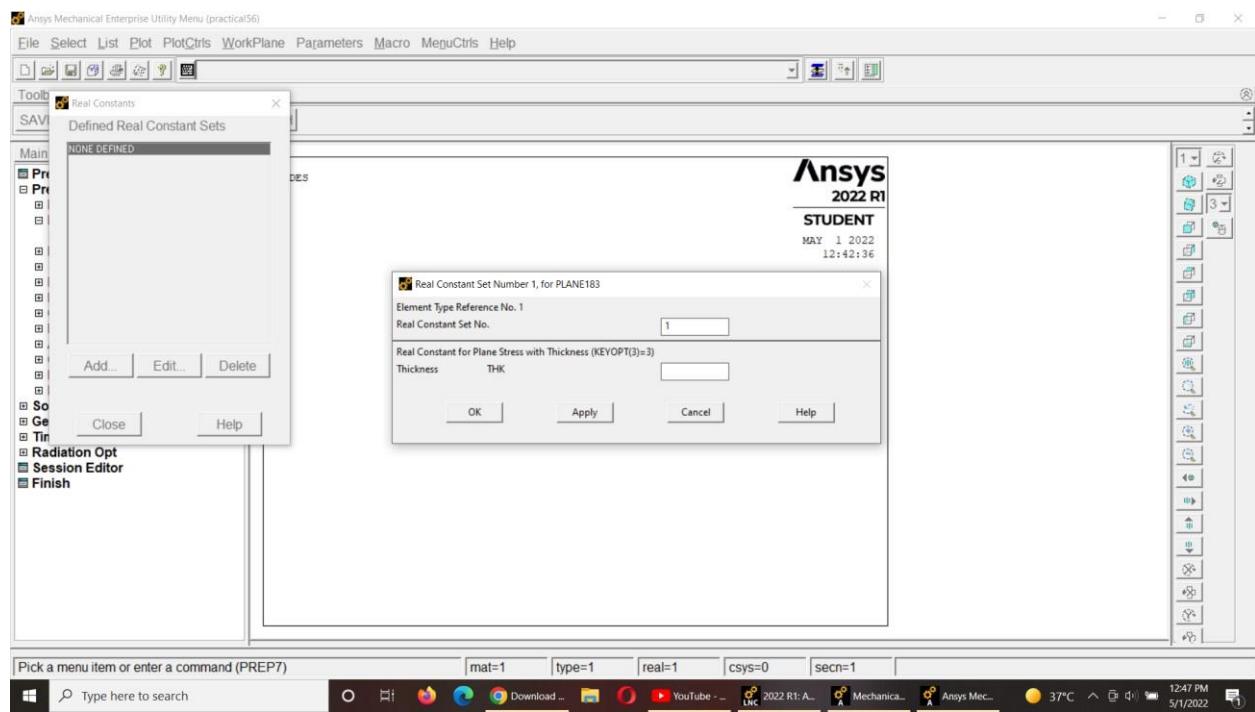
Step 2: Add the element type.....preprocessor>element type>SOLID>8node 183> Press ok



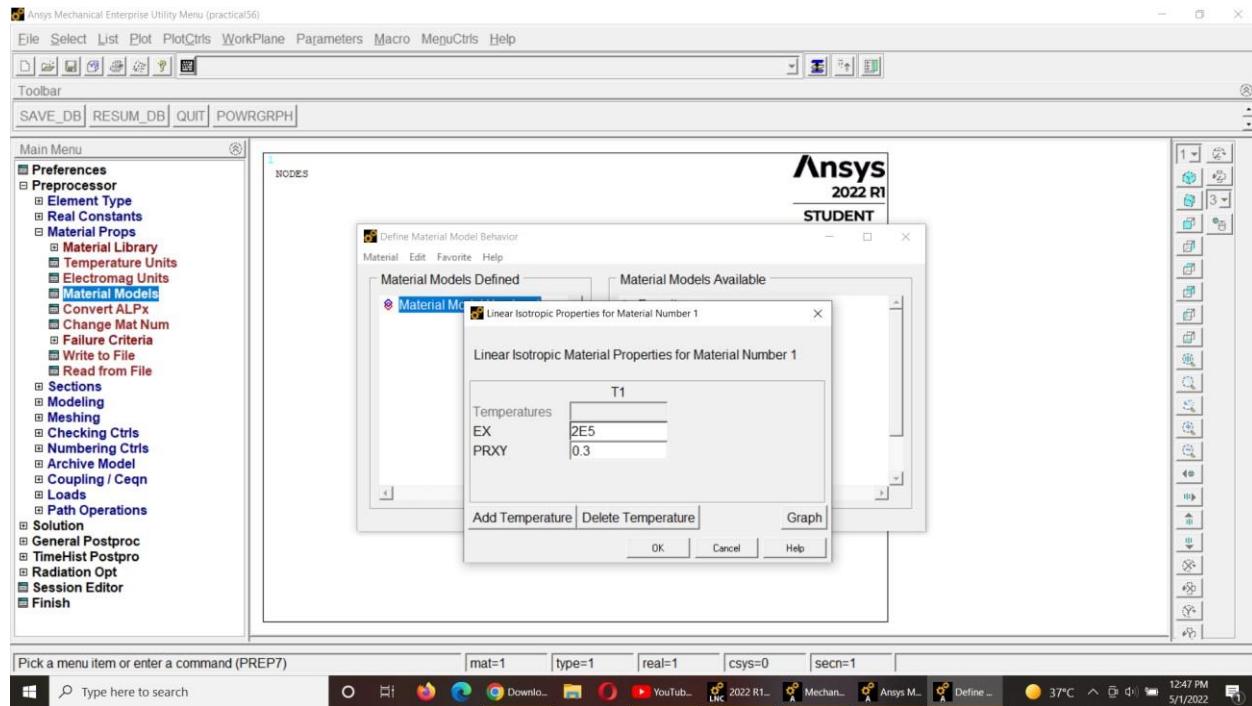
Step 3: options> Element behaviour>plane strss w/thck>ok



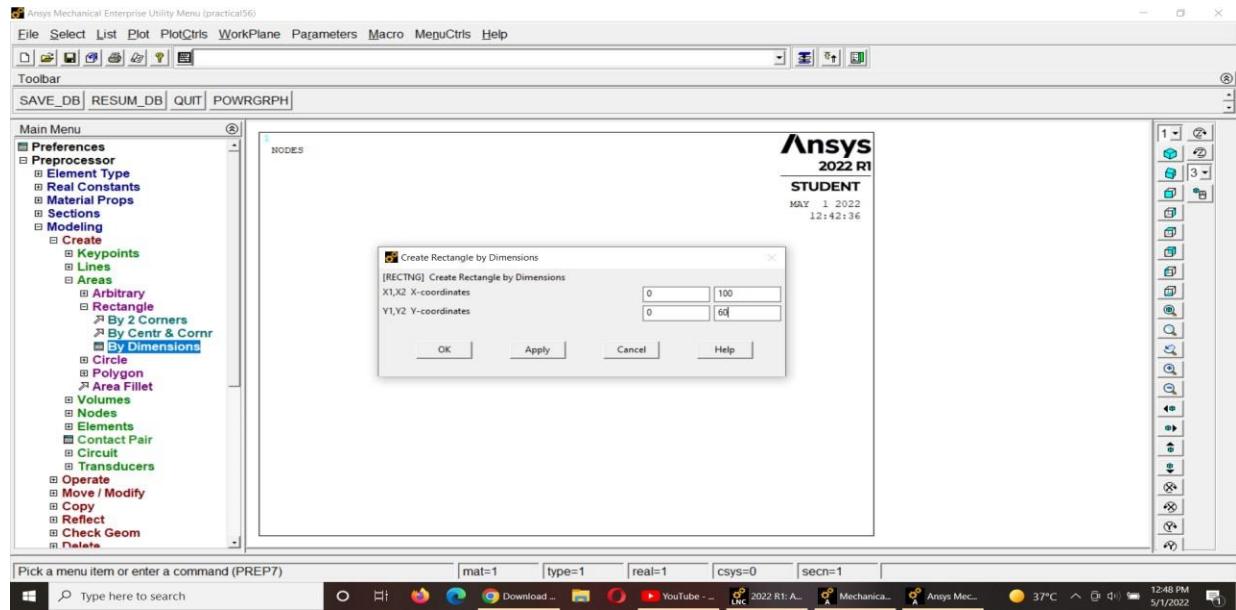
Step 4: Real constrains> Add>thickness>15>ok

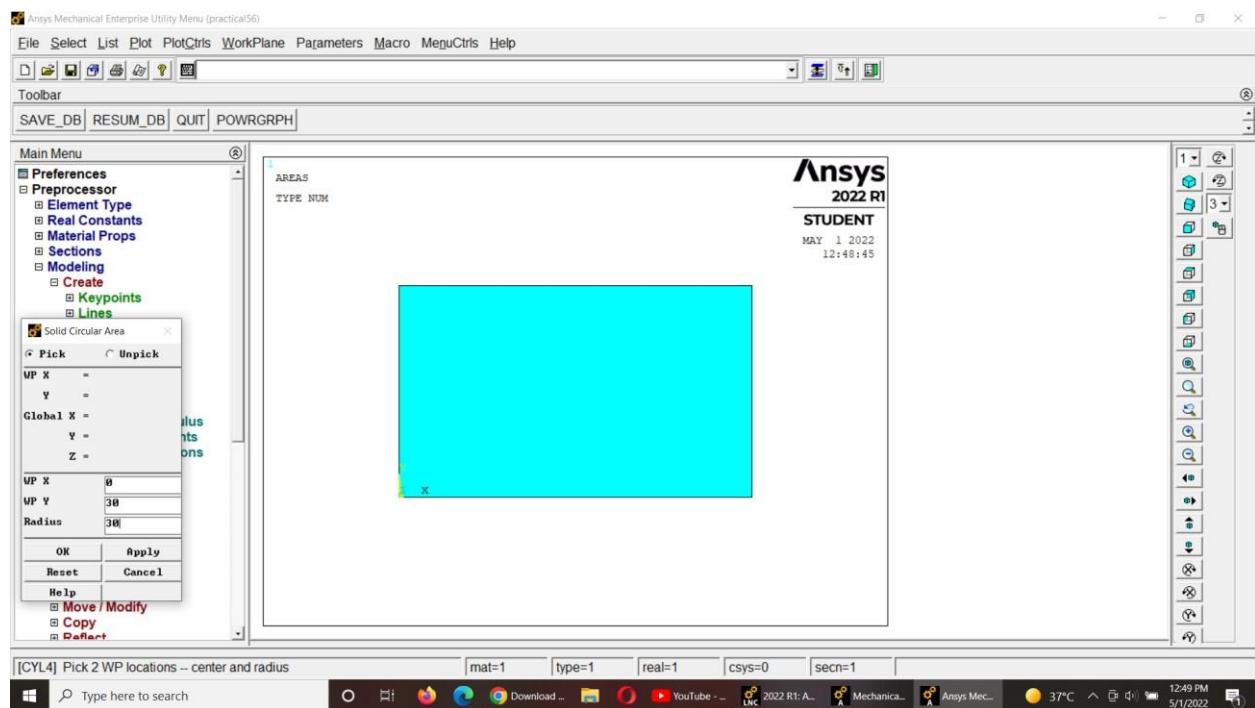


Step 5: Preprocessor>Material prop.>Material models> Material models>structural>linear>elastic>isotropic.

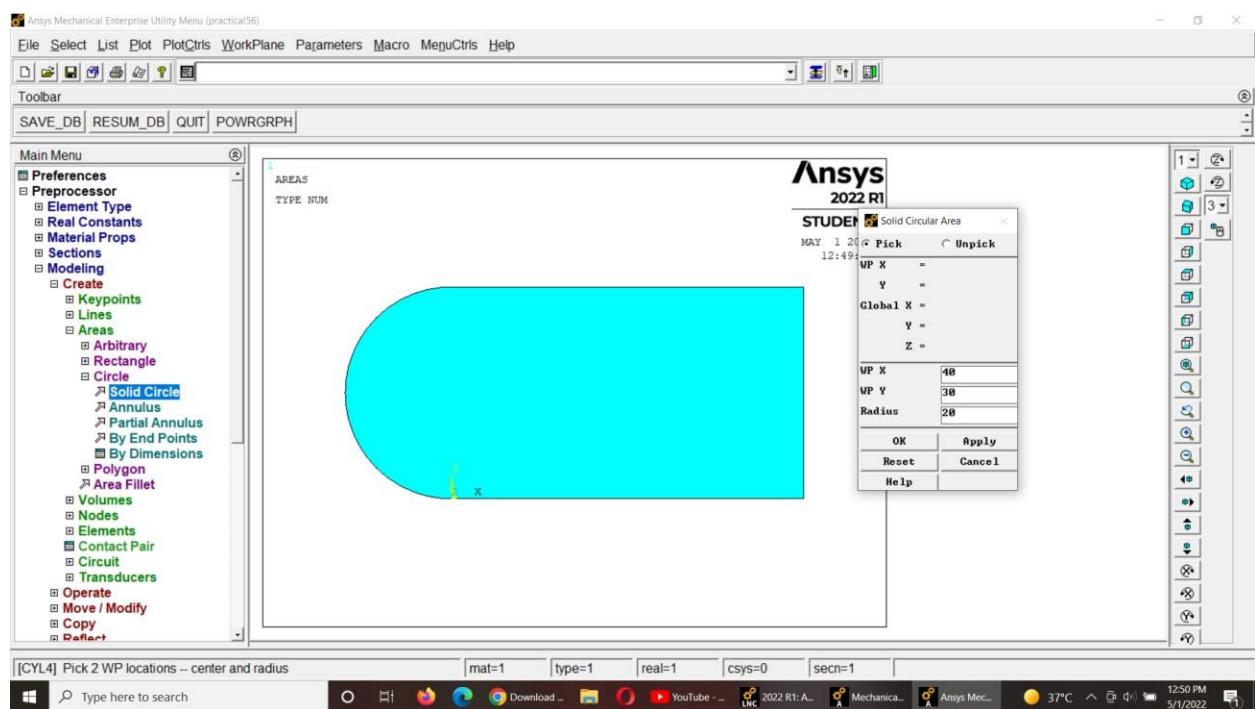


Step6: Creating areas :- modeling>create>areas>rectangle>by dimension.

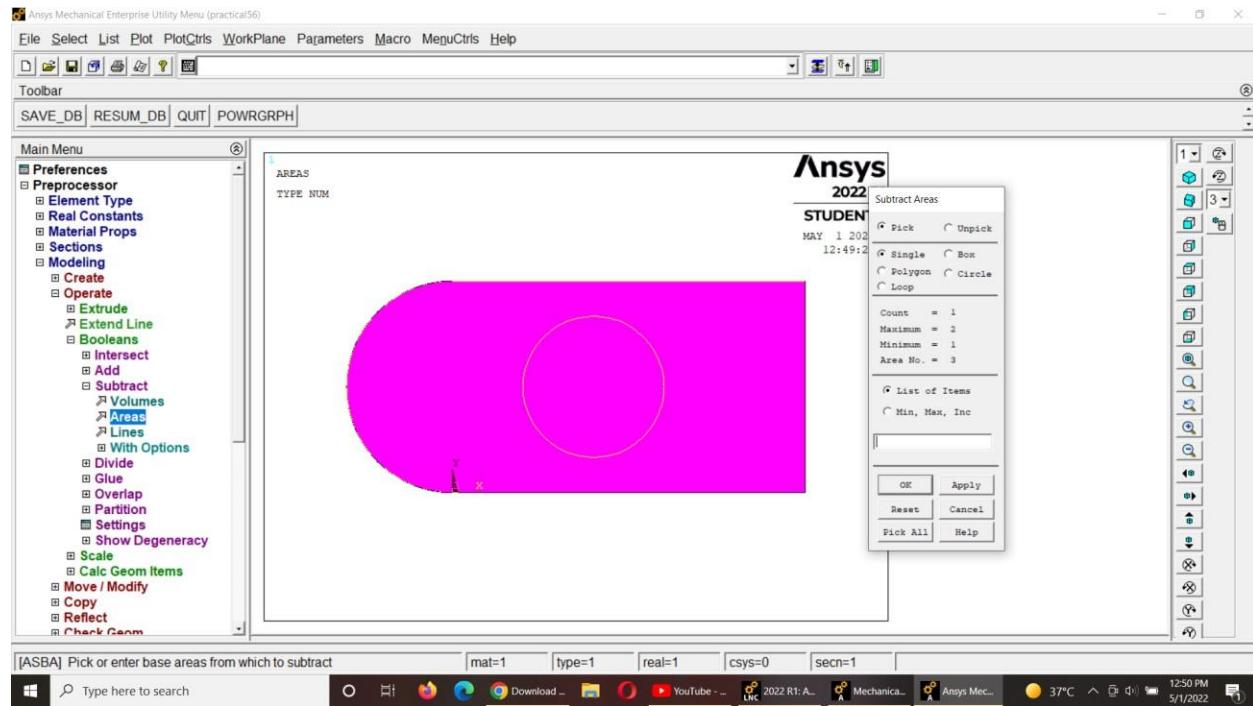




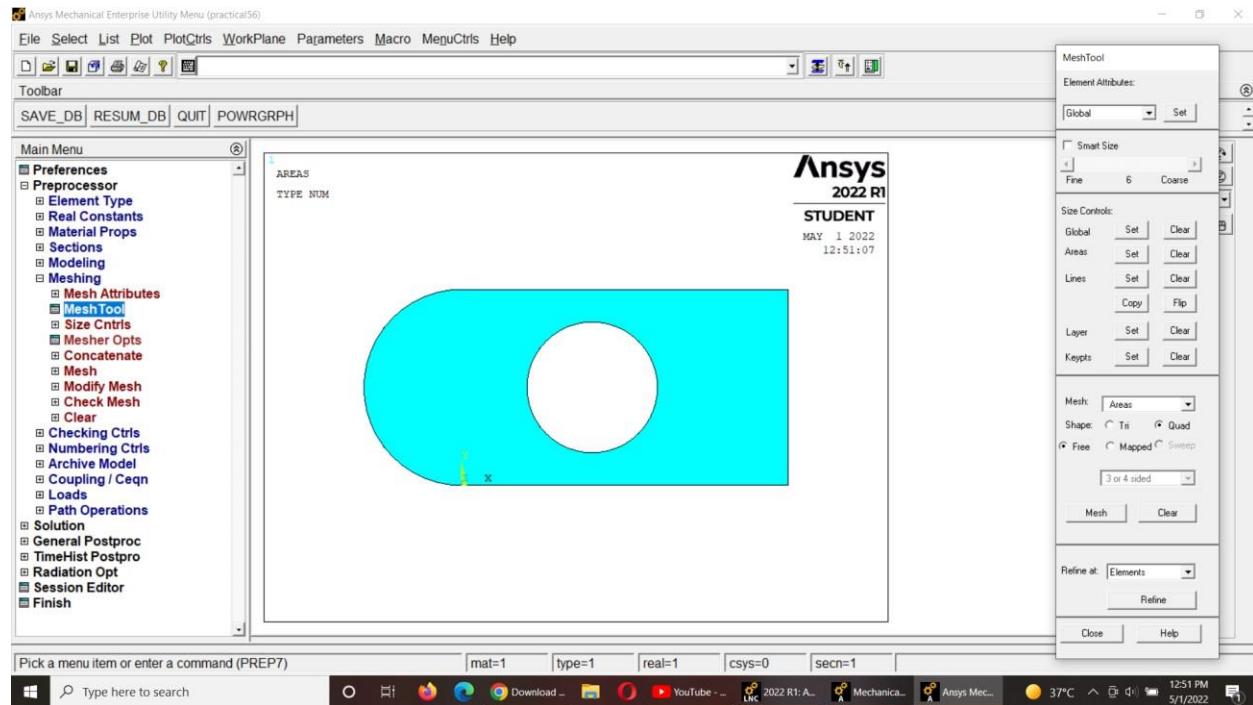
Step7: Creating areas :- modeling>create>areas>circle >solid circle> input coordinates and data



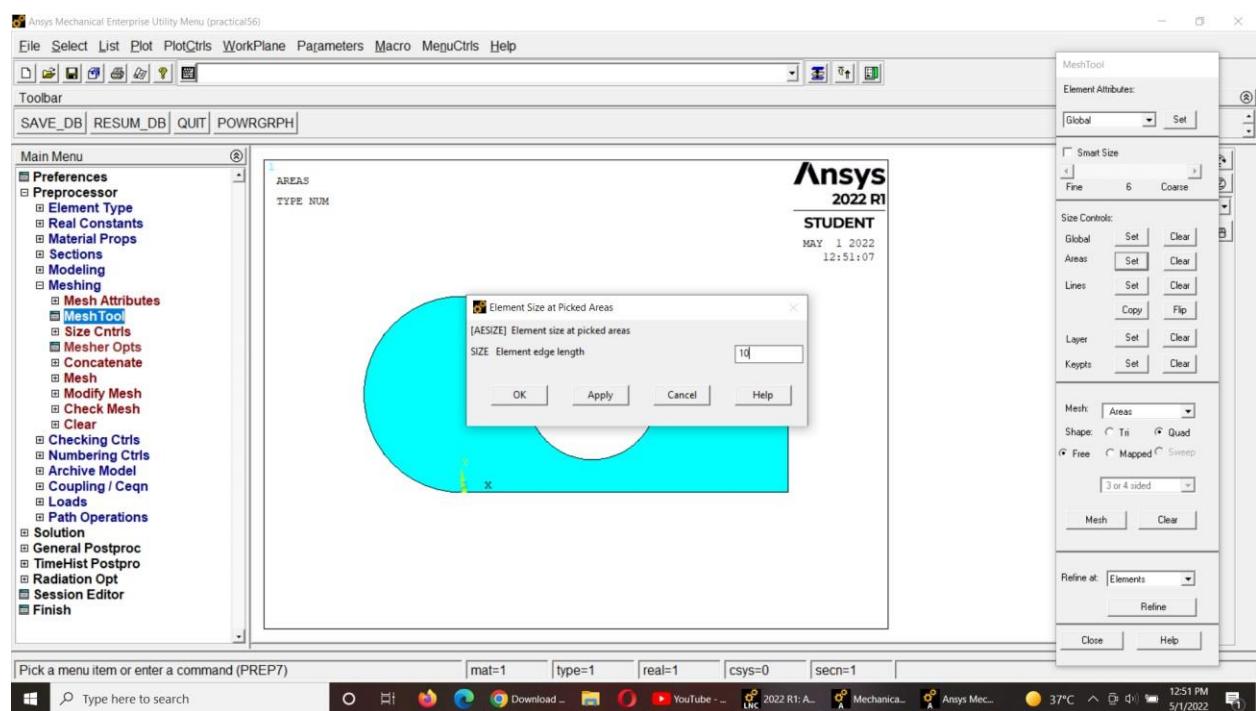
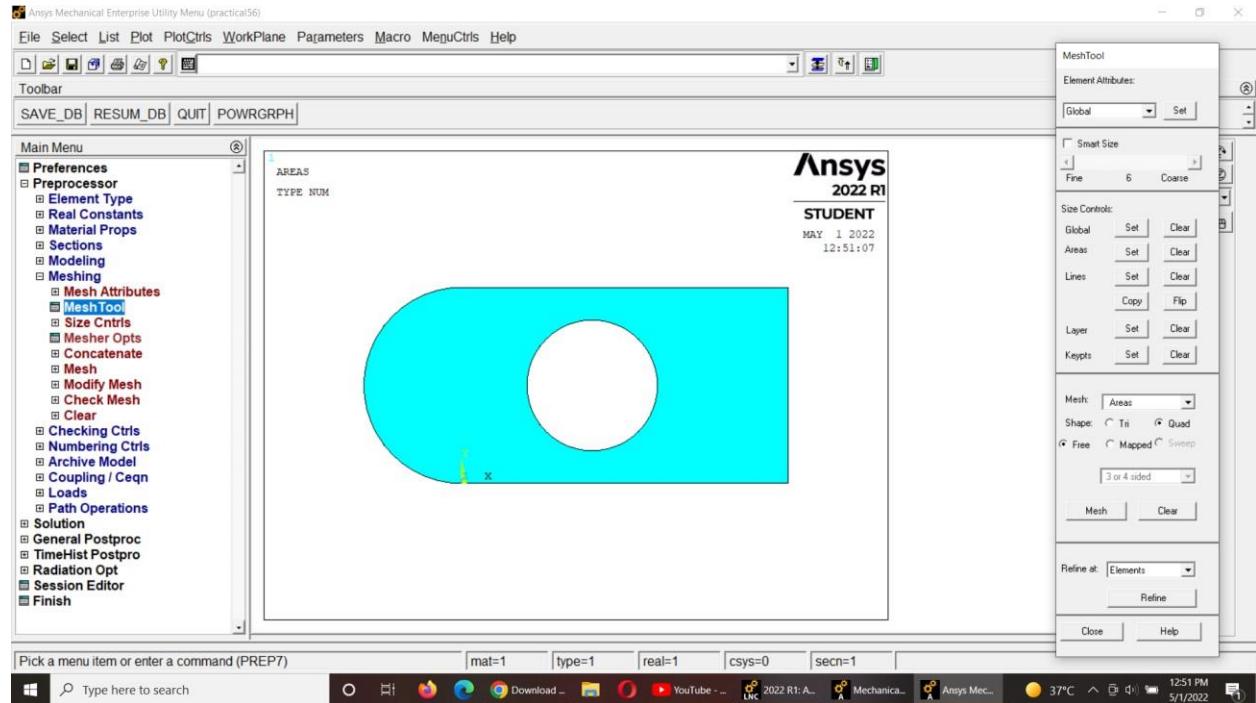
Step8: Creating holes :- modeling>create>areas>solid circle>by dimension.

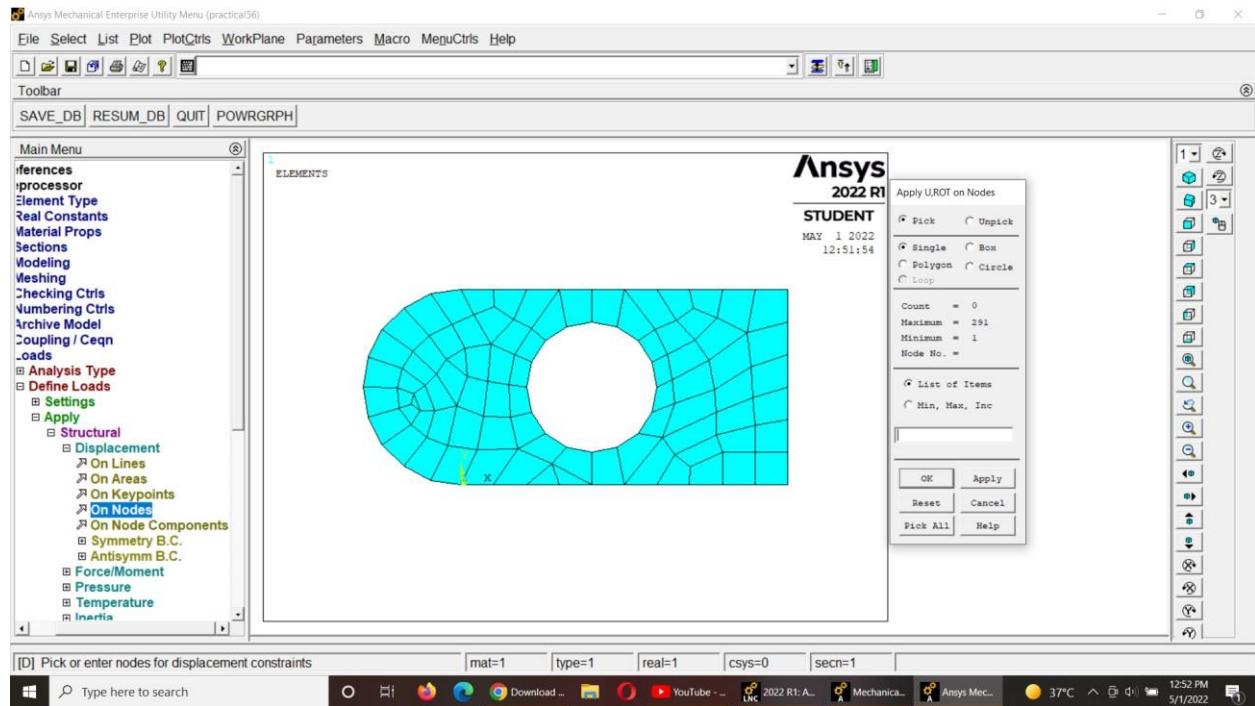


Step9: subtracting holes: Modeling>operate>subtract>areas> select base area>ok> select subtracting area> ok> done

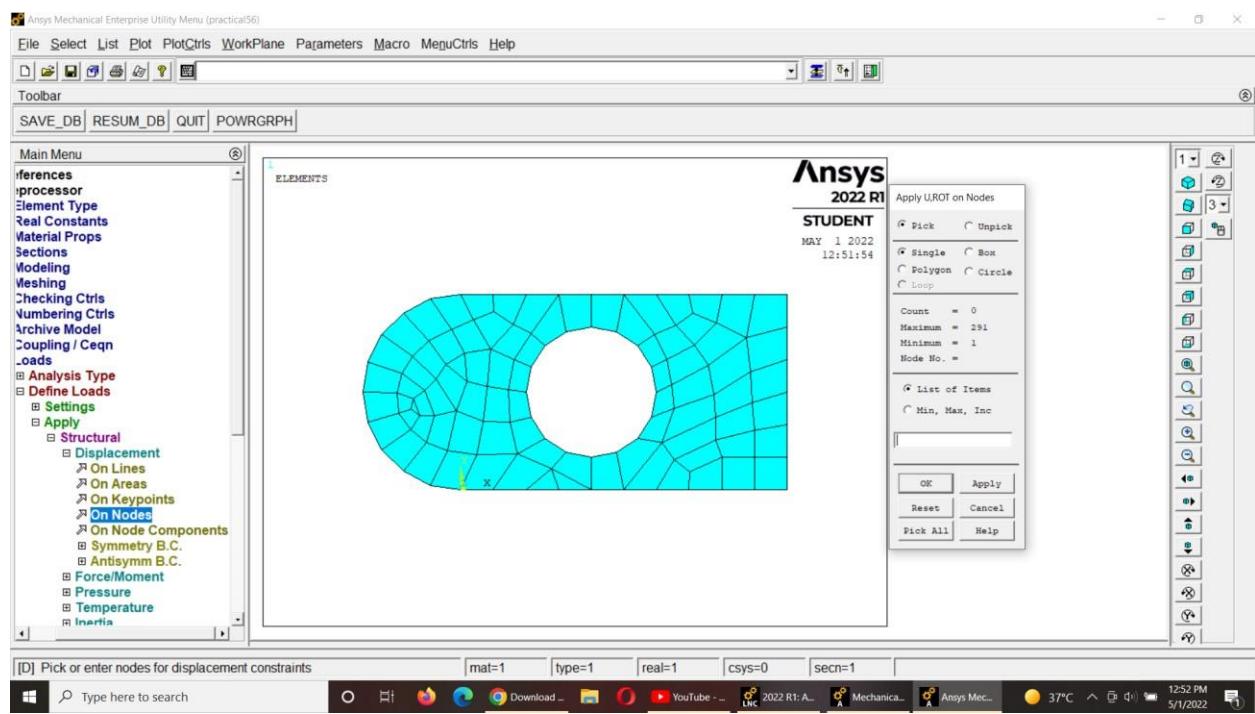


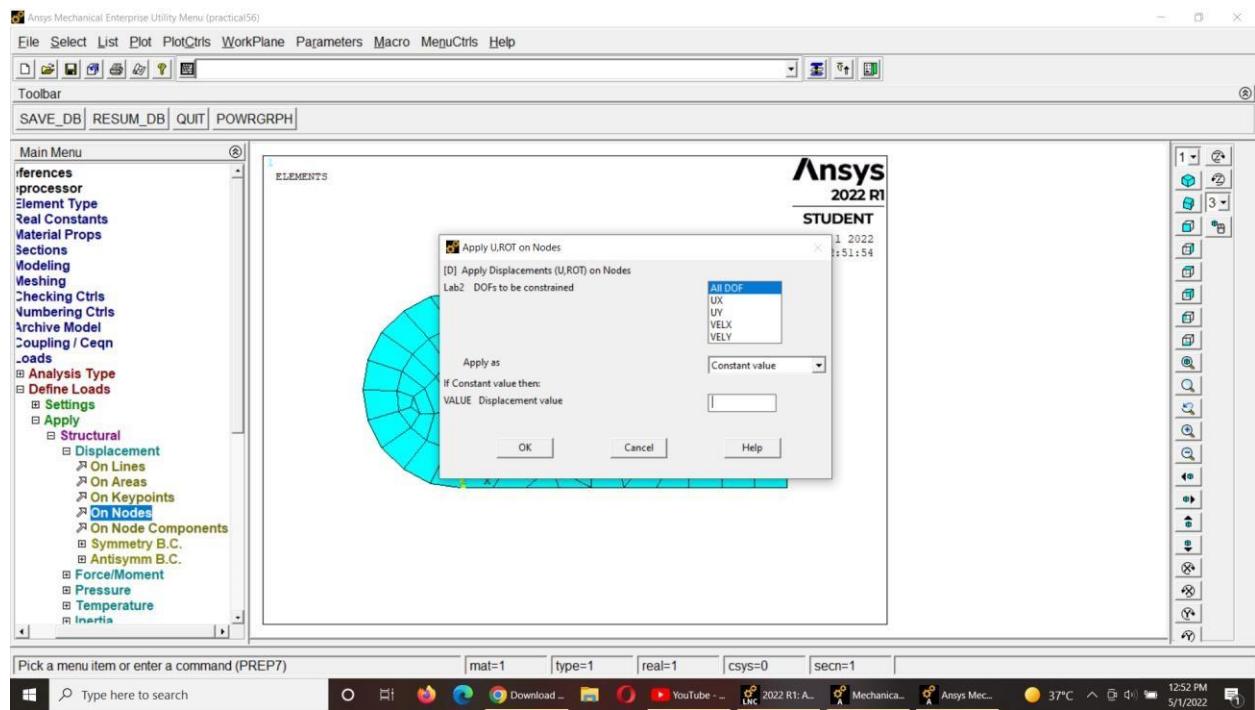
Step10: meshing the element: meshing>mesh tool> select areas>click on area>apply> slect mesh length>ok> mesh>ok



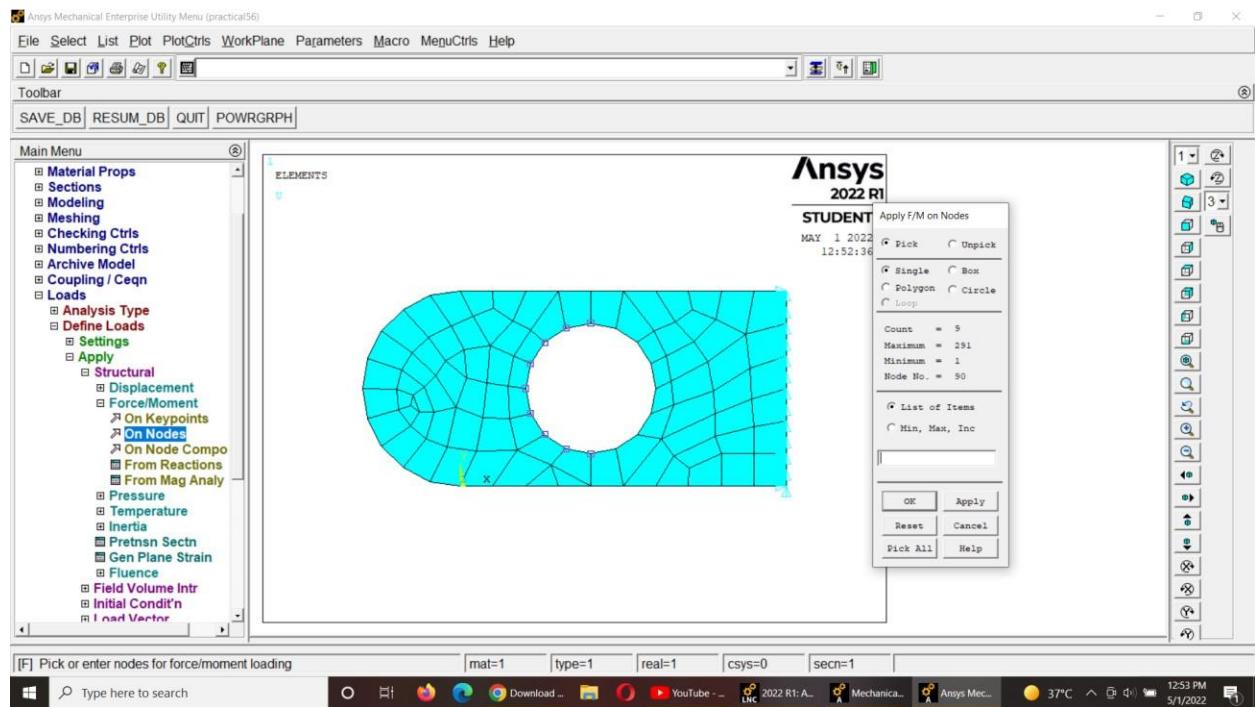


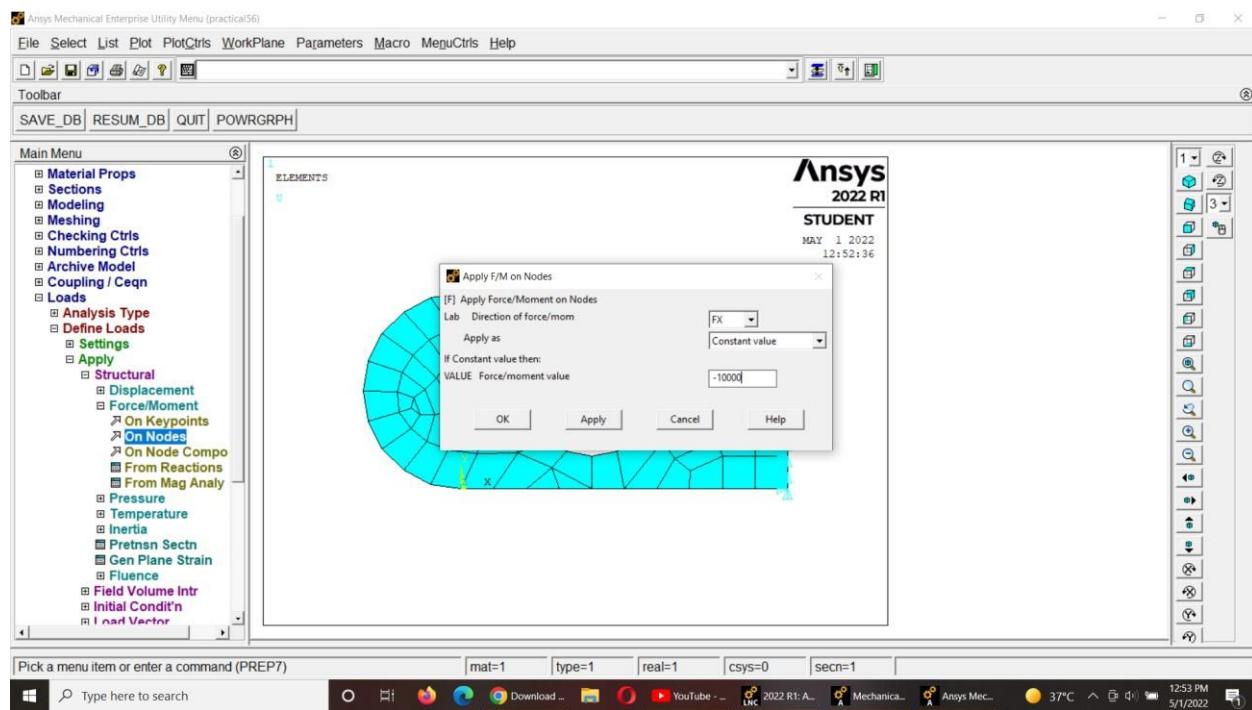
Step11: Loads: Define loads > apply > structural > displacement > on nodes > All Dof > 0 > ok



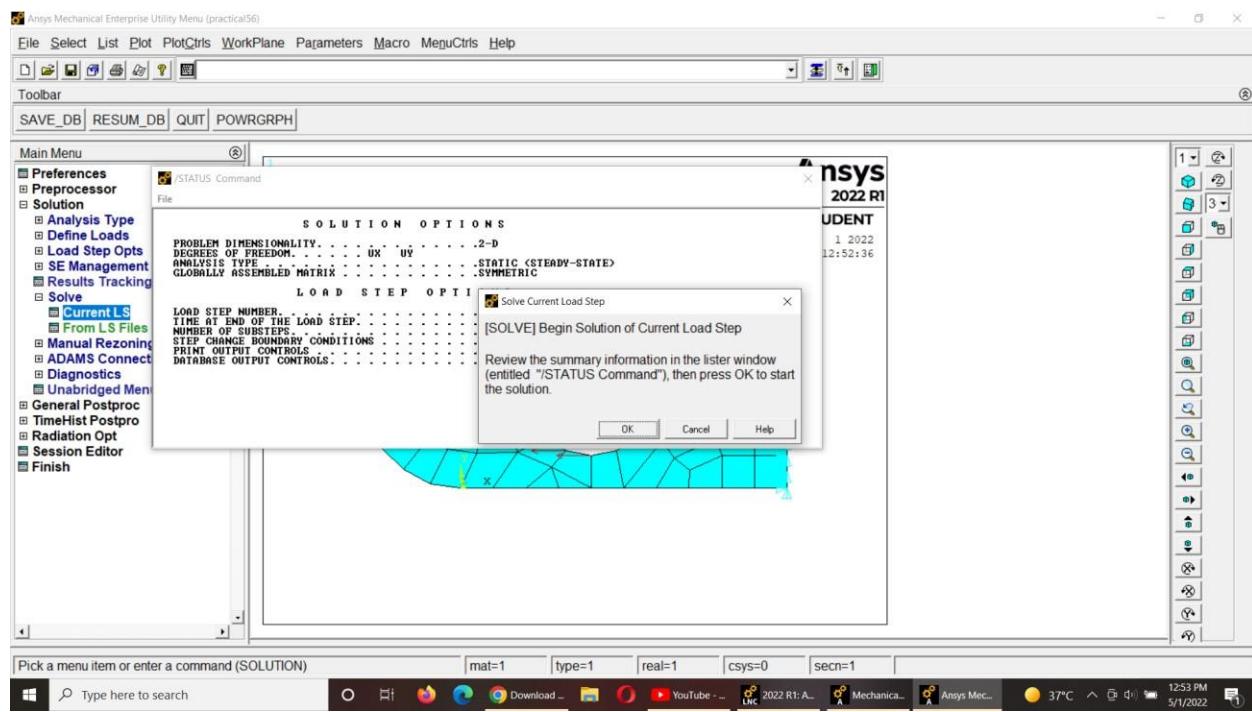


Step12: Loads: Define loads> apply>structural>force>on nodes>put force value>-10000>ok

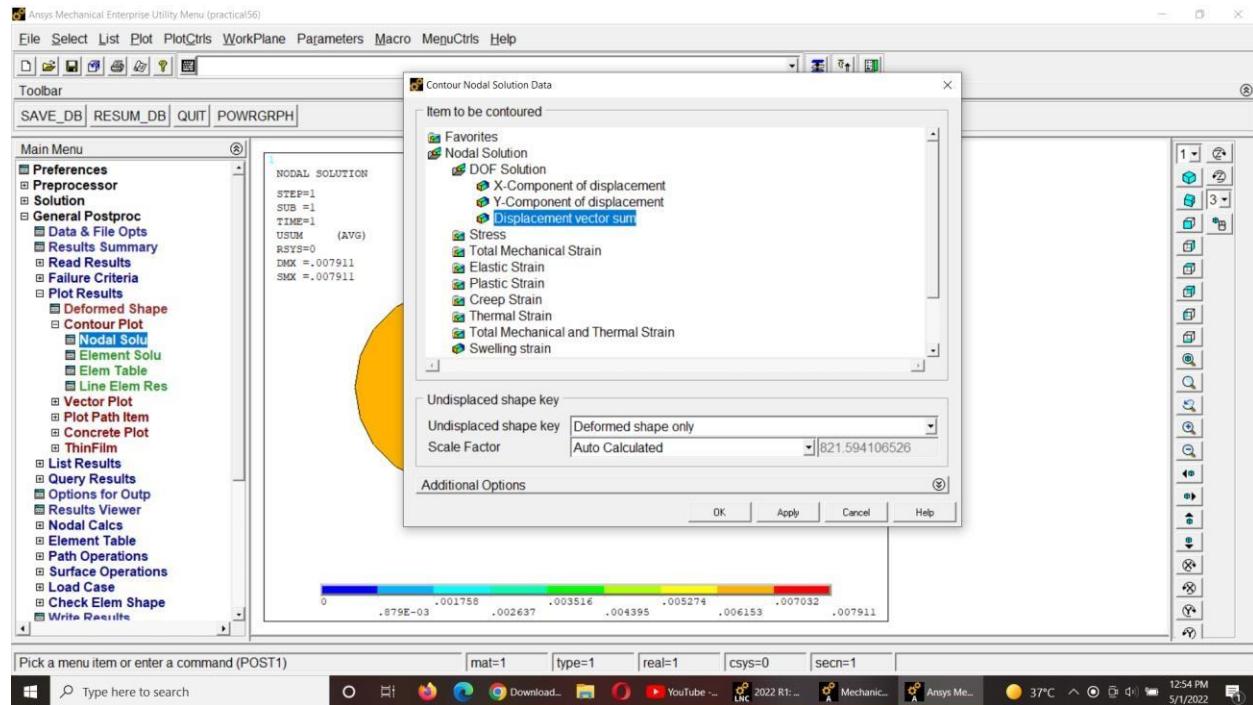




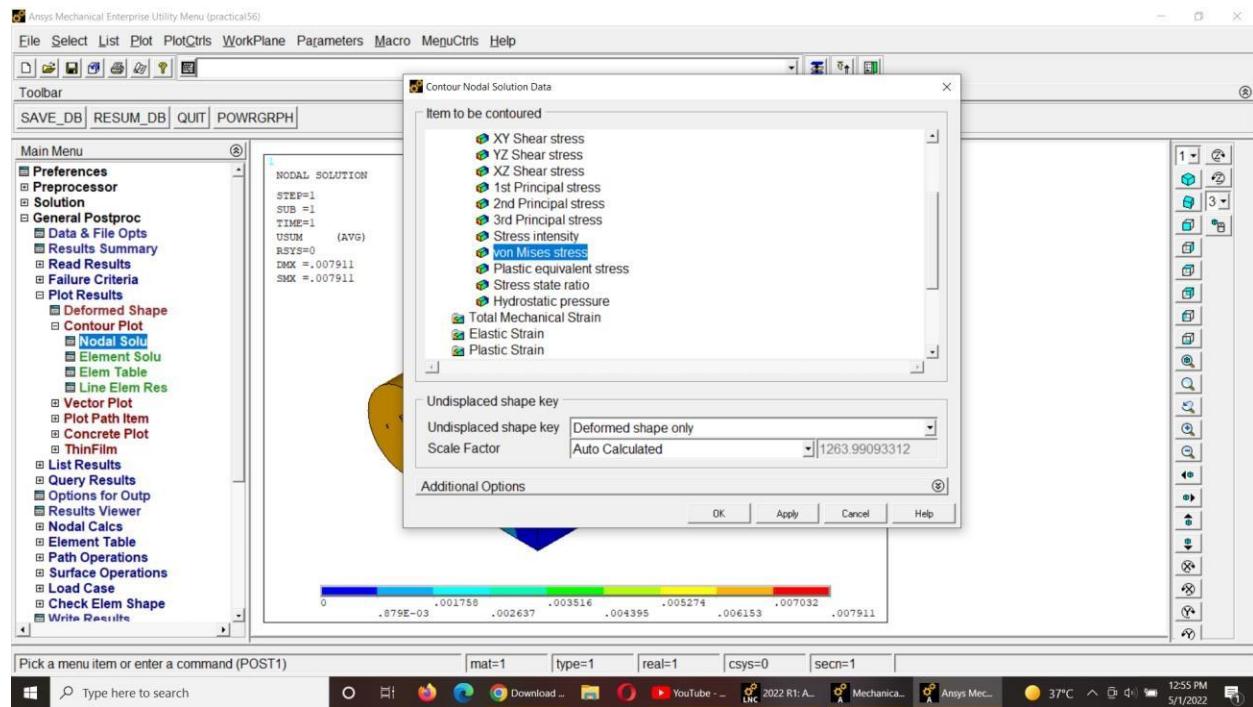
Step13: Solving solution: solution> solve>current LS>ok



Step14: General postproc> plot result> Nodal solution> Dof >Vector sum displacement> apply.

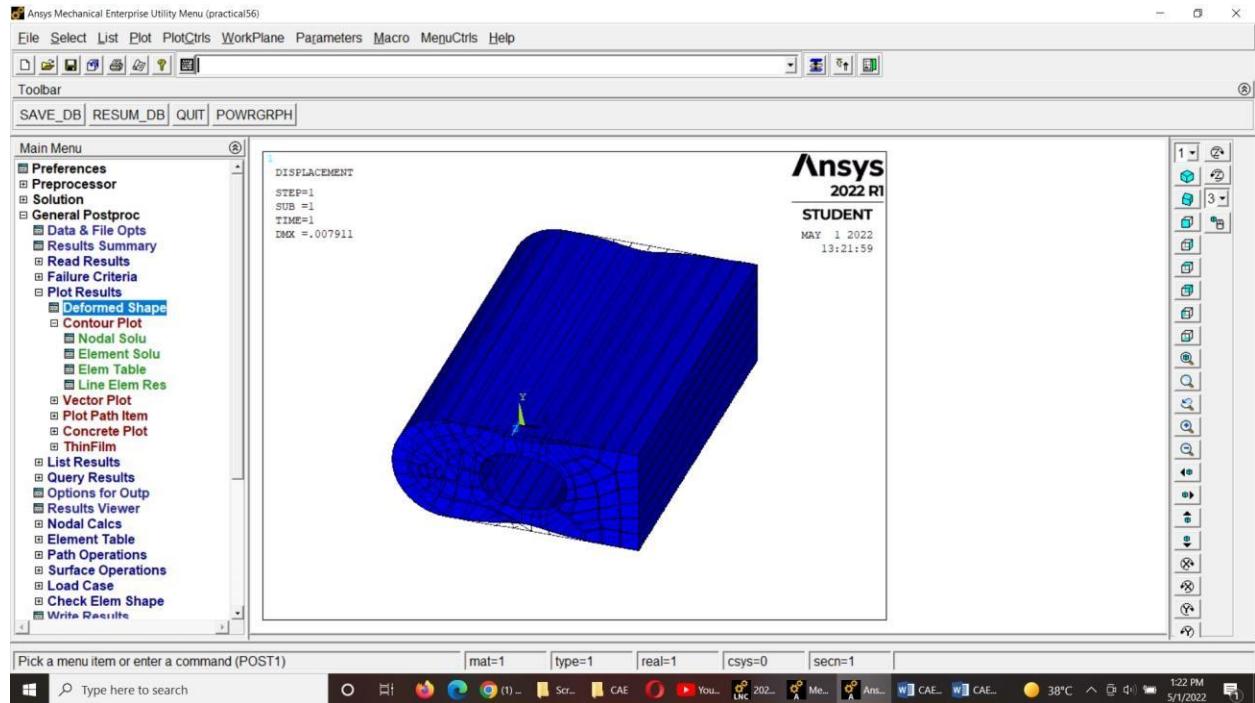
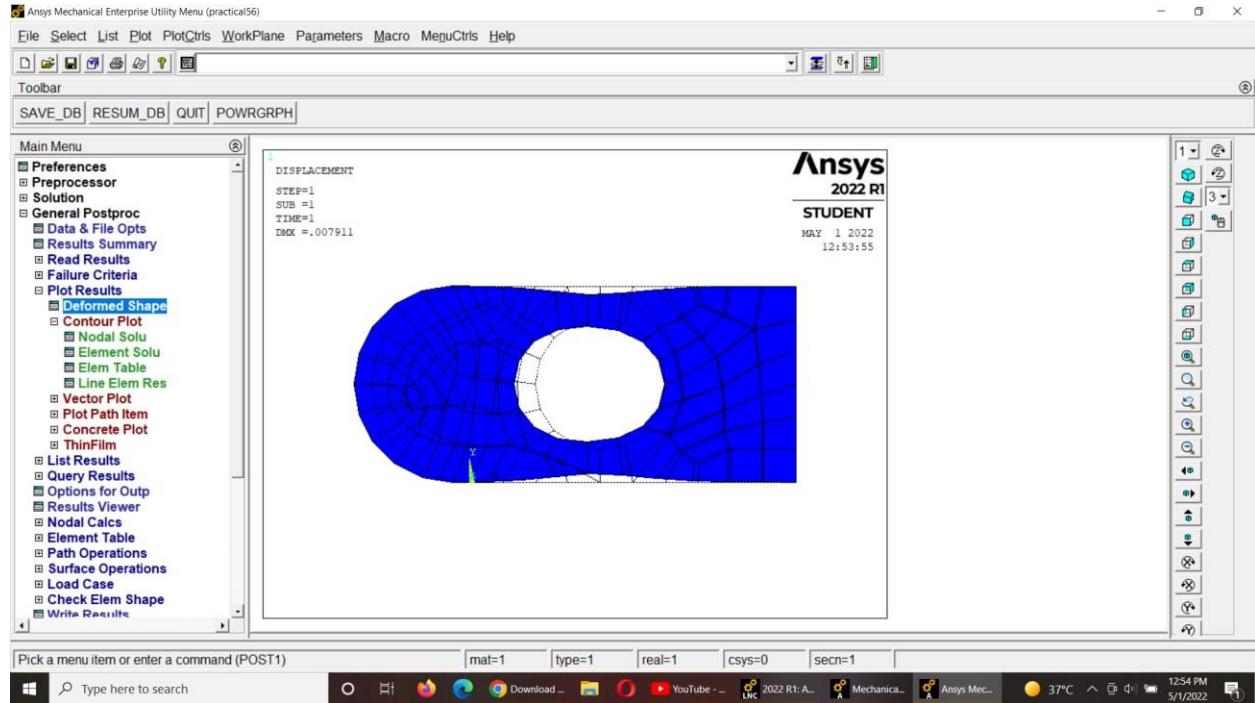


Step15: Nodal solution> stress> von misses stress> apply

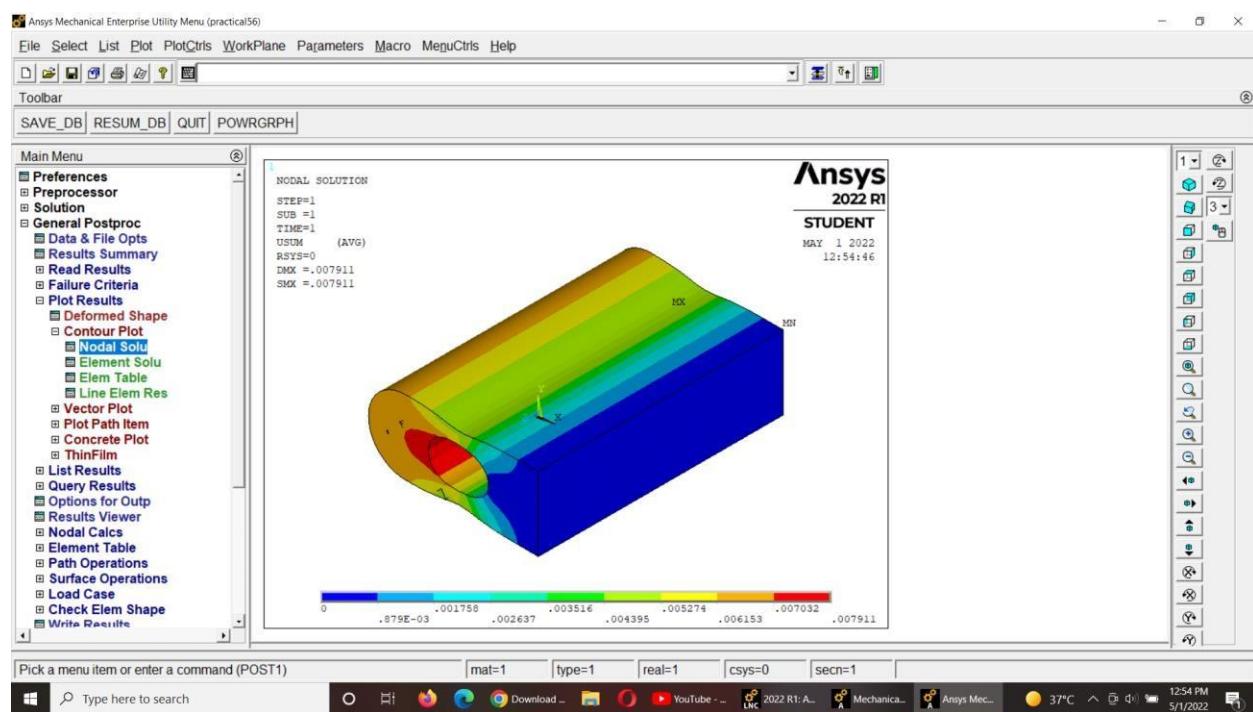
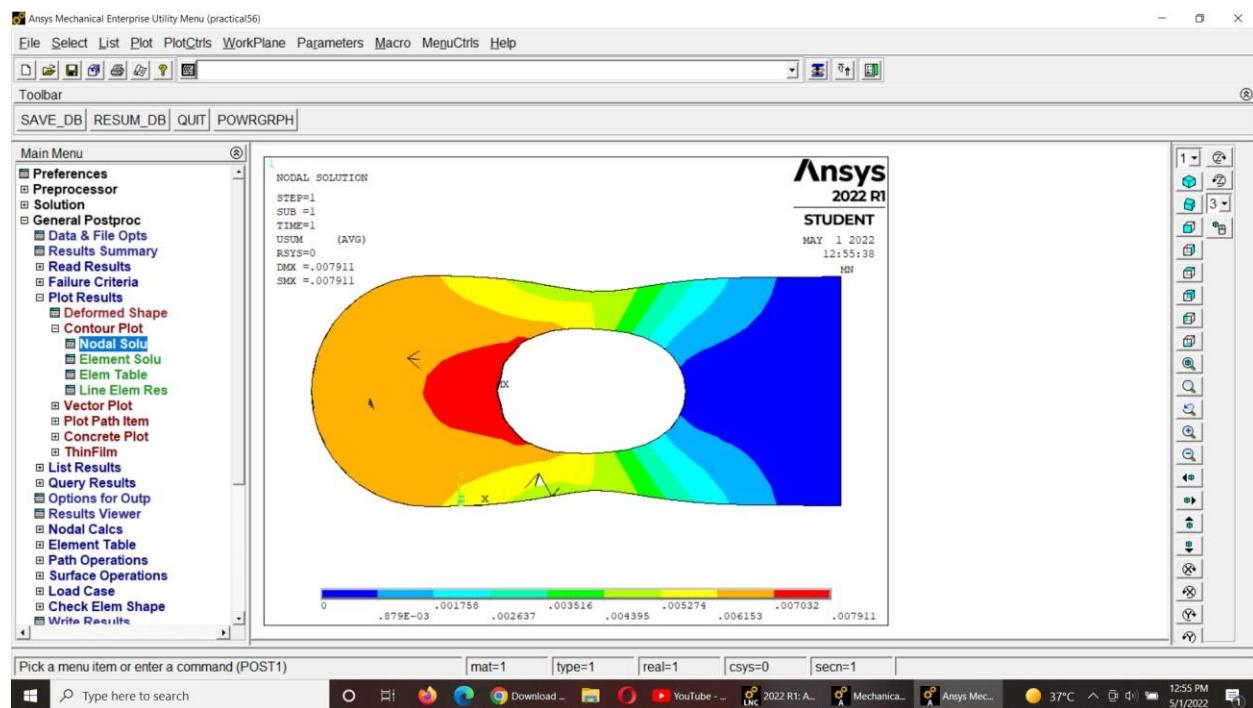


RESULTS:-

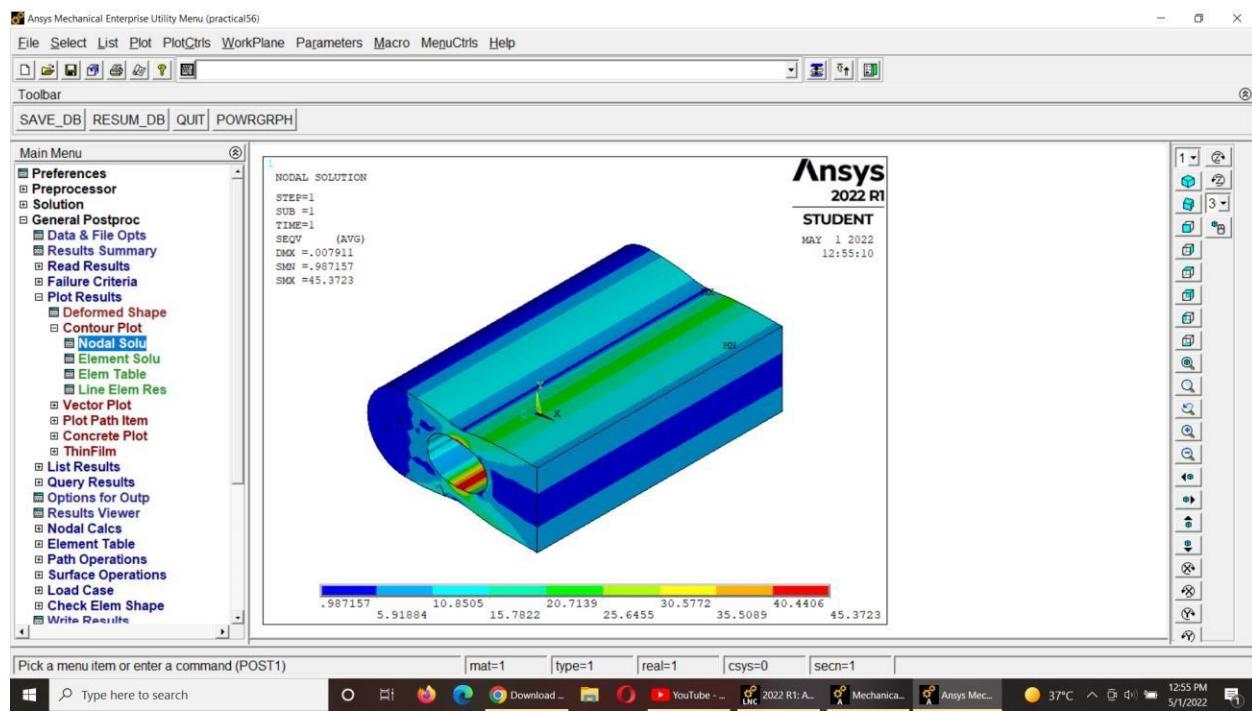
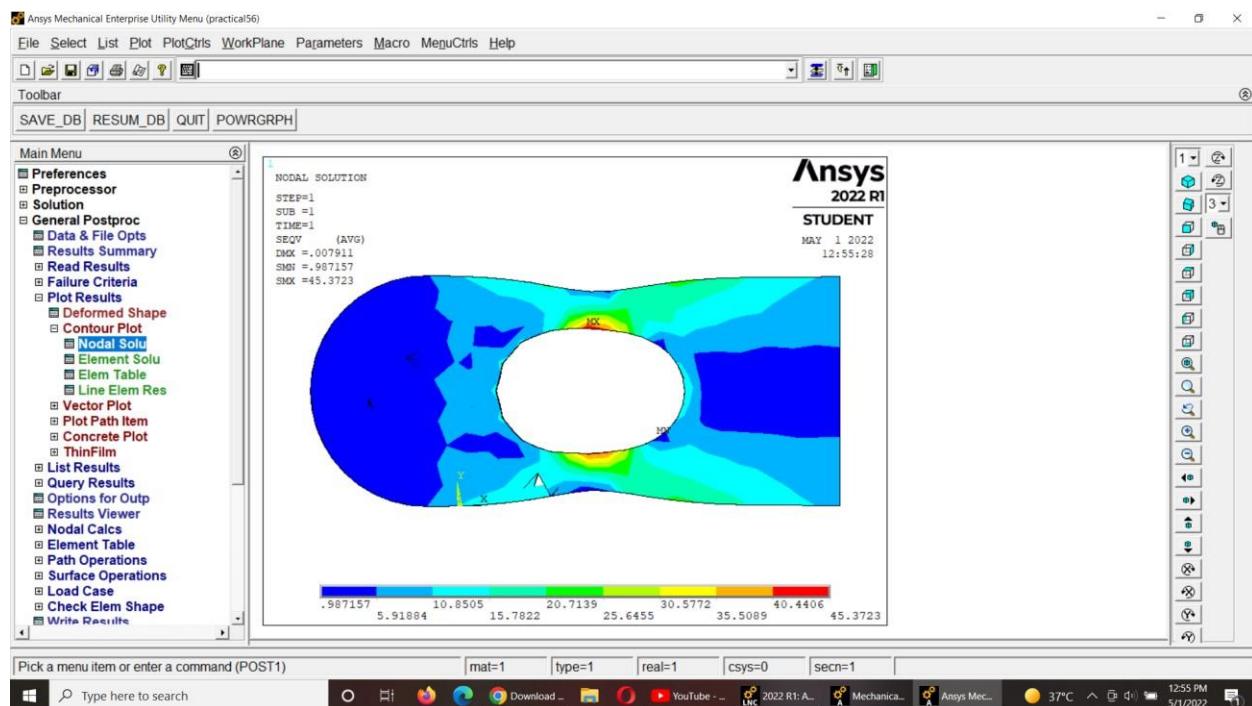
DEFORMED + UNDEFORMED:-



NODAL DISPLACEMENT:-



STRESSES:-



CONCLUSION:-

SO HERE BY ANALYSIS WE HAVE GOT MAX. INDUCED STRESS IS **45.3723 N/MM²**
MAX. DISPLACEMENT IS **0.007911MM**

