Modern Education Society's
Wadia College Of Engineering, Pune-01

DEPARTMENT OF AUTOMATION AND ROBOTICS ENGINEERING

LAB MANUAL

SUB : COMPUTER AIDED ENGINEERING & MANUFACTURING (302523)

MODERN EDUCATION SOCIETY'S
WADIA COLLEGE OF ENGINEERING, PUNE-01

INDEX SHEET

SR NO.	TITLE	PAGE NO.	PERFORMANCE DATE	SUBMISSION DATE	REMARK
1.	1D Bar Element – Structural Linear Analysis				
2.	Truss Analysis using 1D Element				
3.	Plate/Shell Element – Structural Linear and Non-Linear Analysis				
4.	Thermal Analysis – Static/Transient Analysis				
5.	Analysis of Machine Component using 3D Elements				
6.	Modal Analysis –Simply supported/Cantilever beam, etc				
7.	Complete analysis of any 3D model based on industrial robots.				

CERTIFICATE

This is to certify that Mr./Miss ………..…………………………………………...…….of class……………Roll No……, Exam No……………………….., has satisfactorily completed the term work in the subject **Computer Aided Engineering & Manufacturing** as detailed above within the four walls of the institute during the period From July 2025 to November 2025.

Date : **Staff Member** **Head of Department**

Modern Education Society's

Wadia College Of Engineering,

Pune-01

DEPARTMENT OF AUTOMATION AND ROBOTICS ENGINEERING

COMPUTER AIDED ENGINEERING & MANUFACTURING (302523)

Academic Year 2025-2026

TITLE	1D BAR ELEMENT – STRUCTURAL LINEAR ANALYSIS		
NAME OF STUDENT			
CLASS AND DIVISION		BATCH	
SEMESTER		ROLL NO	
DATE OF PERFORMANCE		DATE OF SUBMISSION	
EXAMINED BY			

COURSE OUTCOMES:

- **CO1: DEFINE** the use of CAE tools and DESCRIBE the significance of shape functions infinite element formulations.
- **CO2:APPLY** material properties and boundary condition to SOLVE 1-D and 2-D element stiffness matrices to obtain nodal or elemental solution.
- **CO3:ANALYZE** and APPLY various numerical methods for different types of analysis.
- **CO4:CREATE** process plan and GENERATE GandM code using CAM software tools.
- **CO5:UNDRSTAND** lean manufacturing tools and techniques
- **CO6:APPLY** knowledge to do process planning and ESTIAMTE costing for the same.

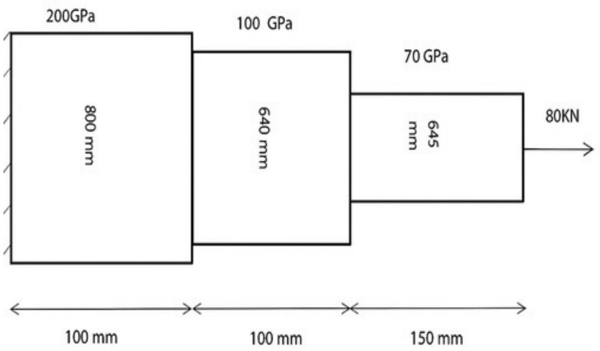
Name Of Student:	Class:
Semester/Year:	Roll No.:
Date Of Performance:	Date Of Submission:
Examined By: Prof. R. N. Yerrawar	Experiment No :1

Title:

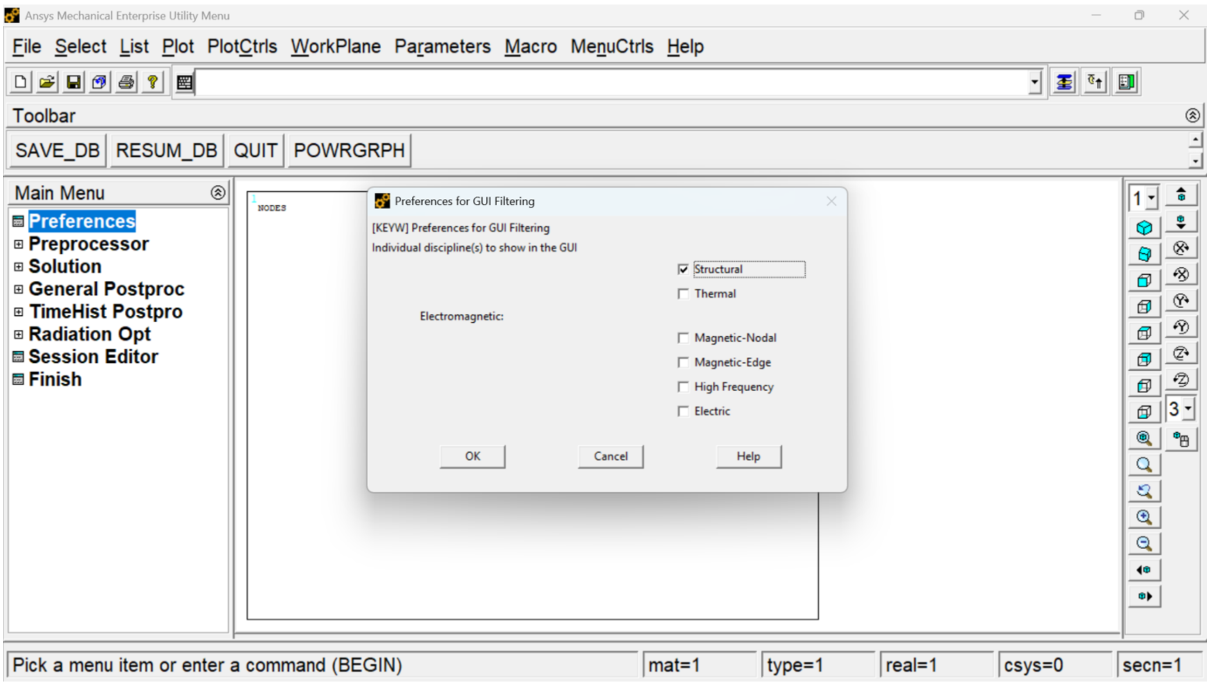
1D Bar Element – Structural Linear Analysis

PROBLEM STATEMENT :

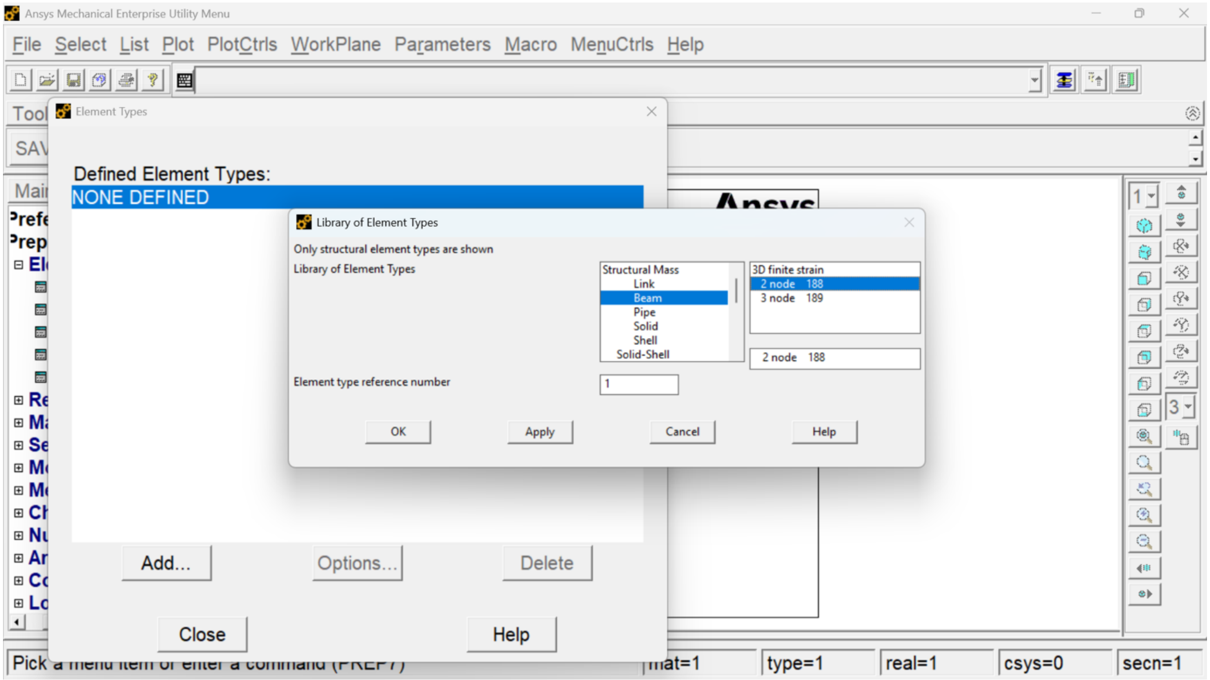
Determine the nodal displacements and element stresses by finite element foundation and software tool for the following figure. Use P=300kN; A1=0.5m2 and E =200GPa.



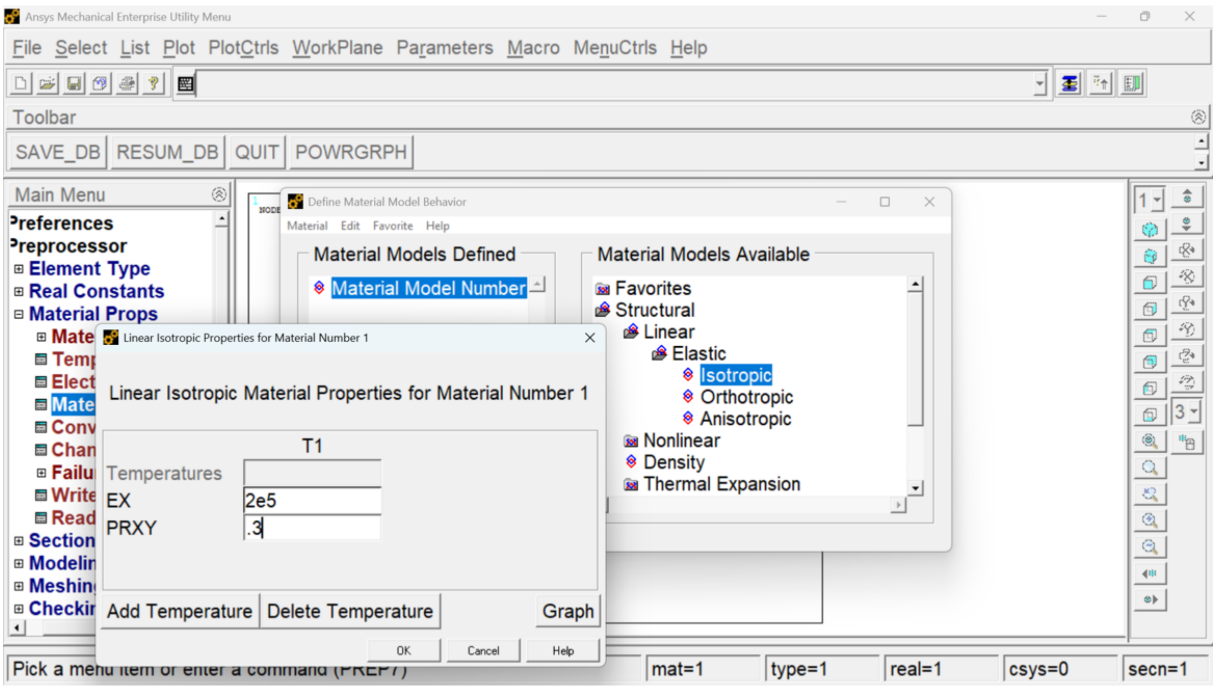
1. Preferences>Structural>OK>Close



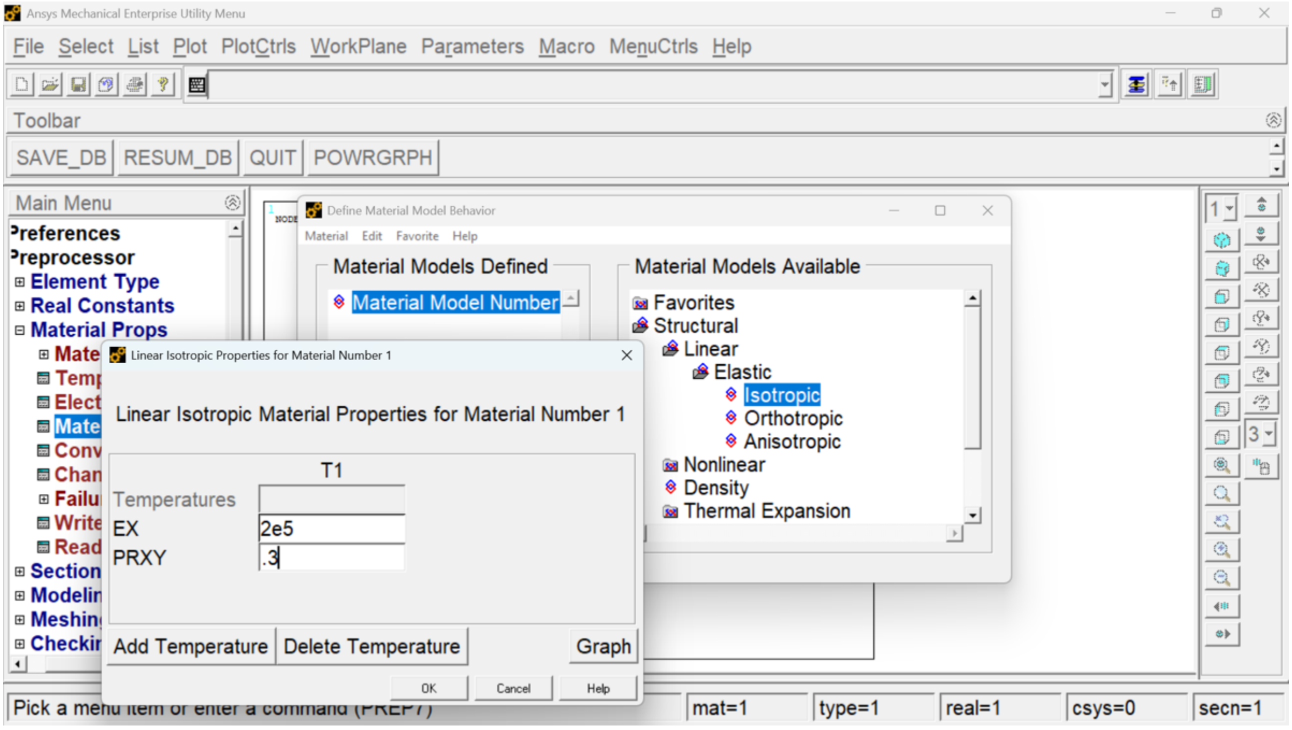
2. Preprocessor>Define Element Types>Add>Link>3D finit stn 180>OK>Close



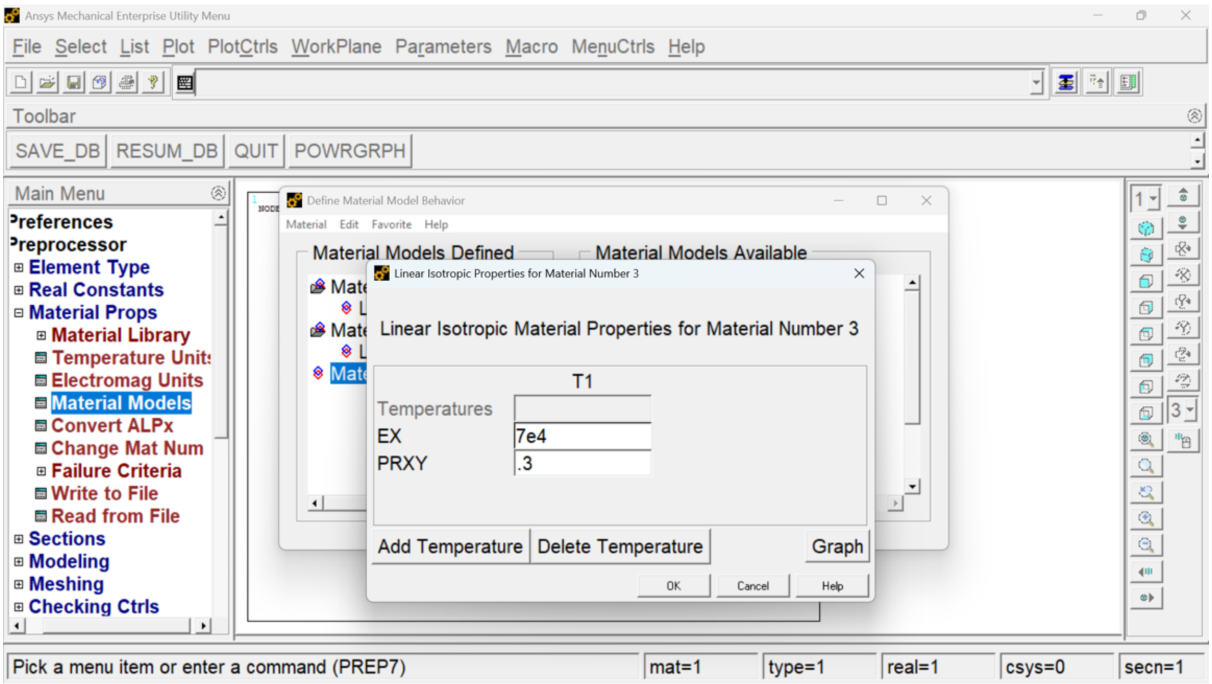
3. Preprocessor>Section>Link>Add>1>OK



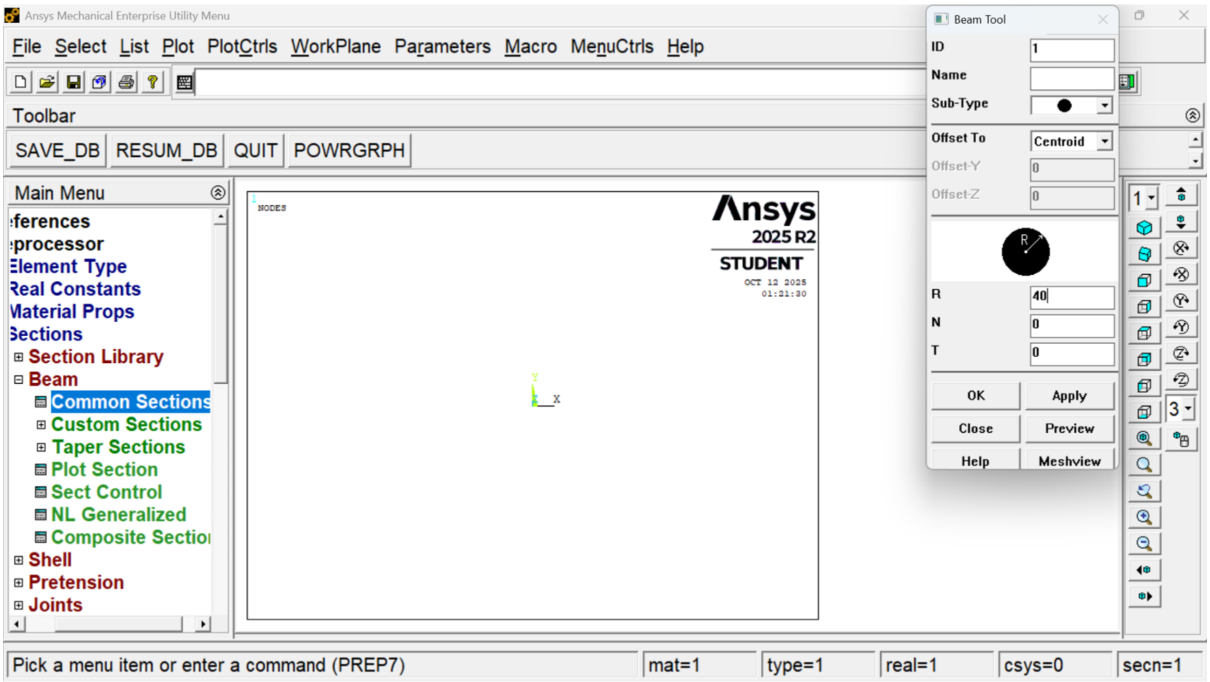
4. After OK>Section Name: SQ-4>Link area: 05>OK



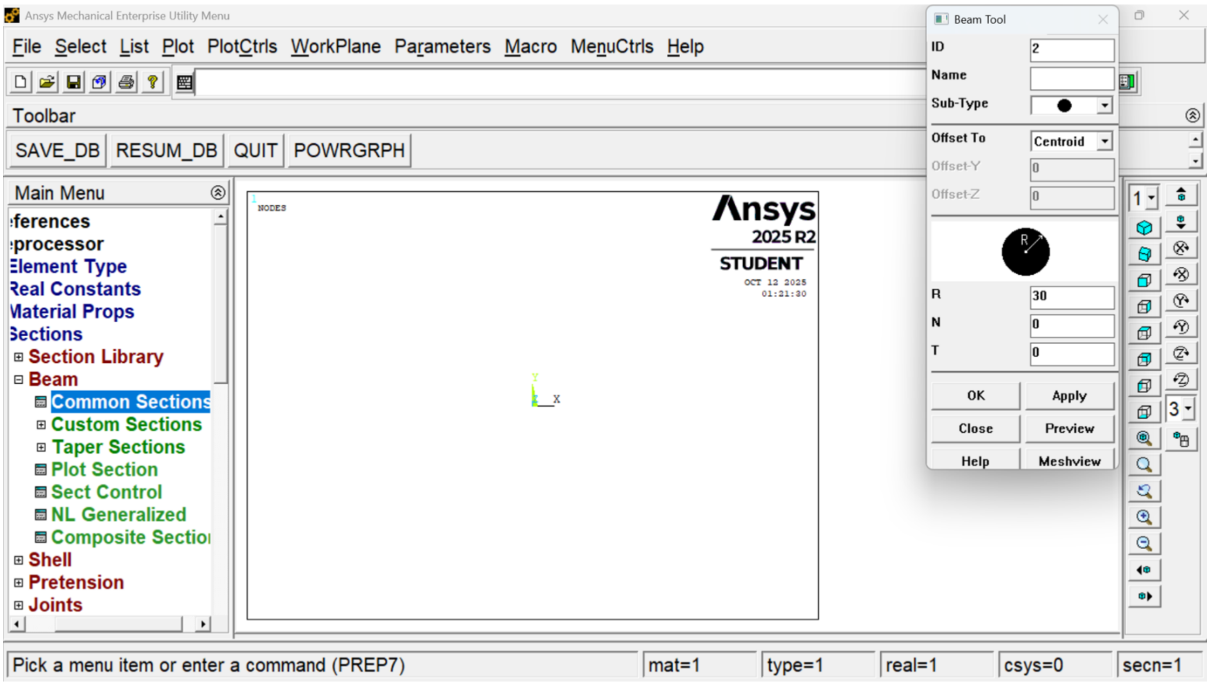
5. After OK>Section Name: SQ-5>Link area: 1>OK



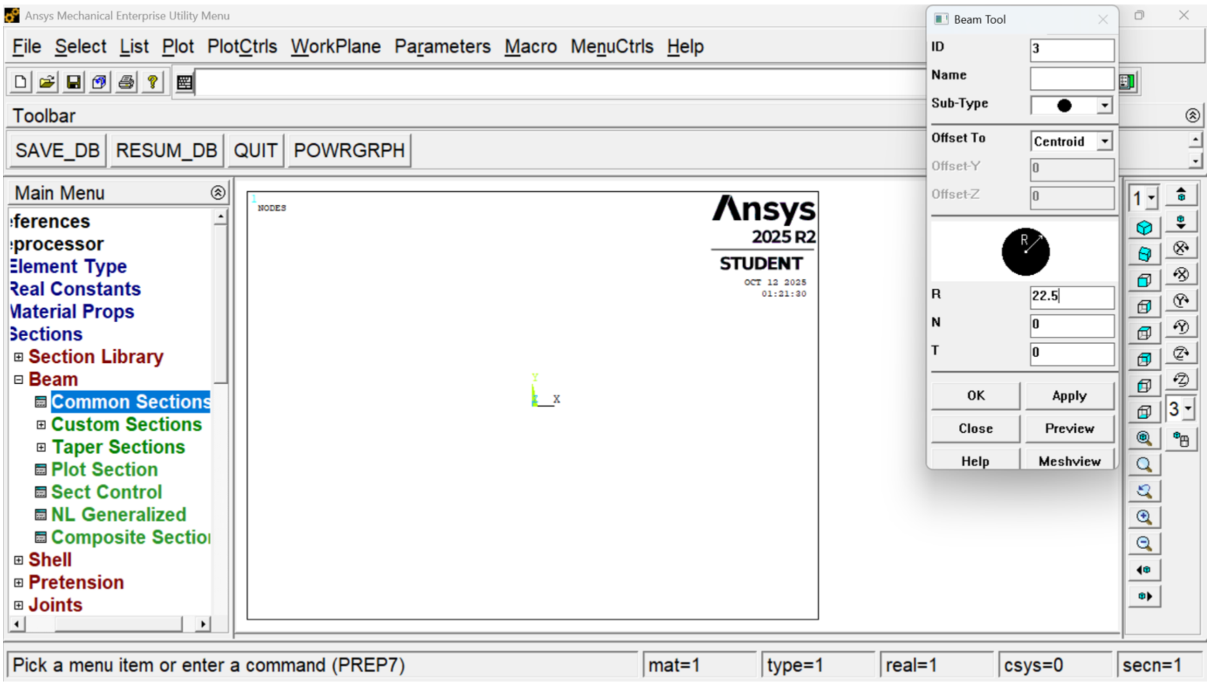
6. Preprocessor>Sections>Beam>Common Sections>Beam Tool: 1 >R:40 >Apply



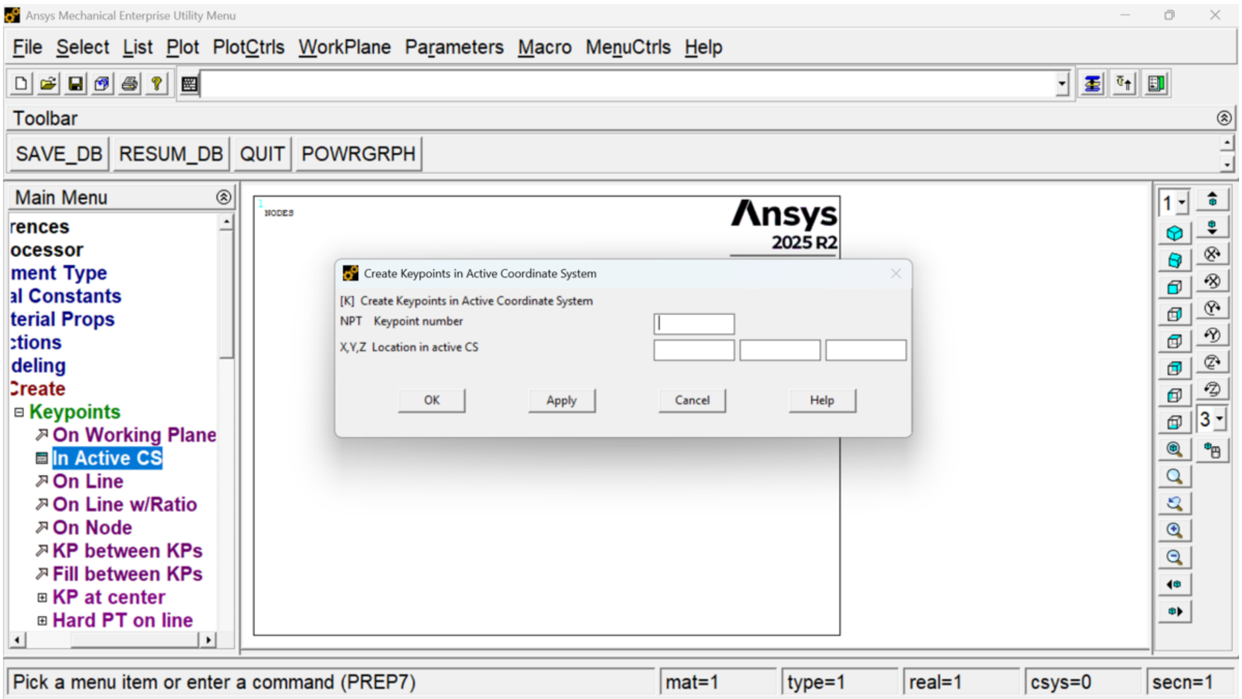
7. Preprocessor>Sections>Beam>Common Sections>Beam Tool: 2 >R:30 >Apply



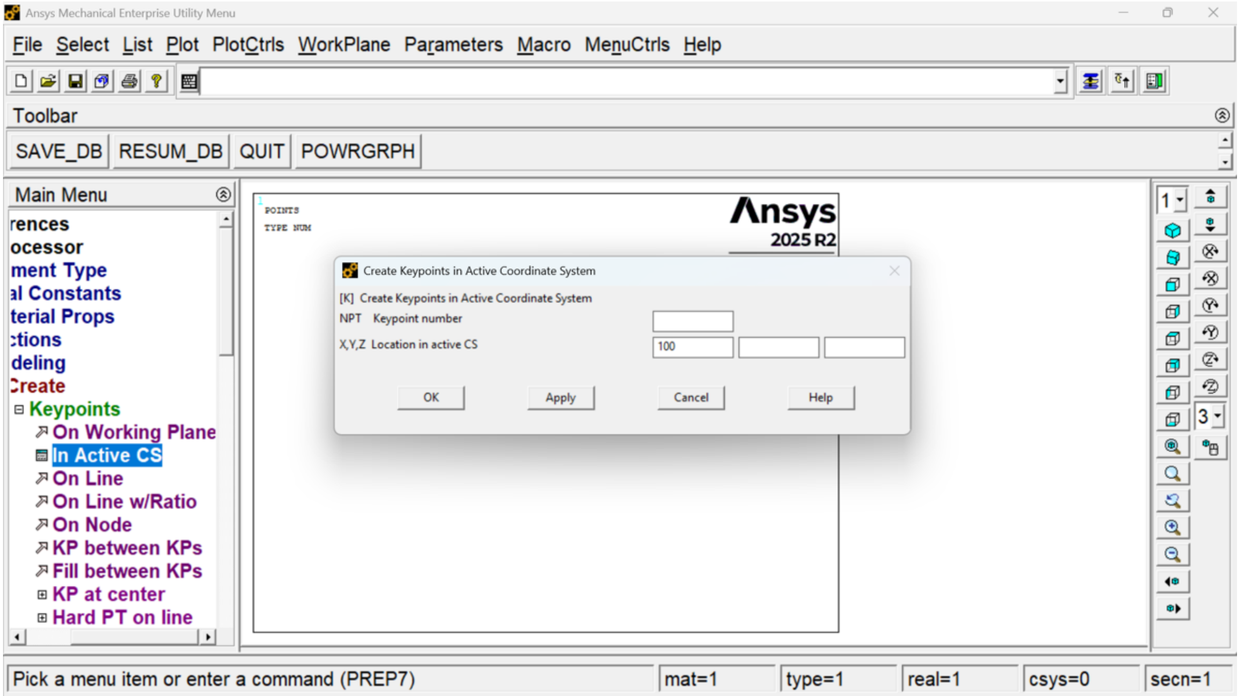
8. Preprocessor>Sections>Beam>Common Sections>Beam Tool: 3 >R:22.5 >Apply



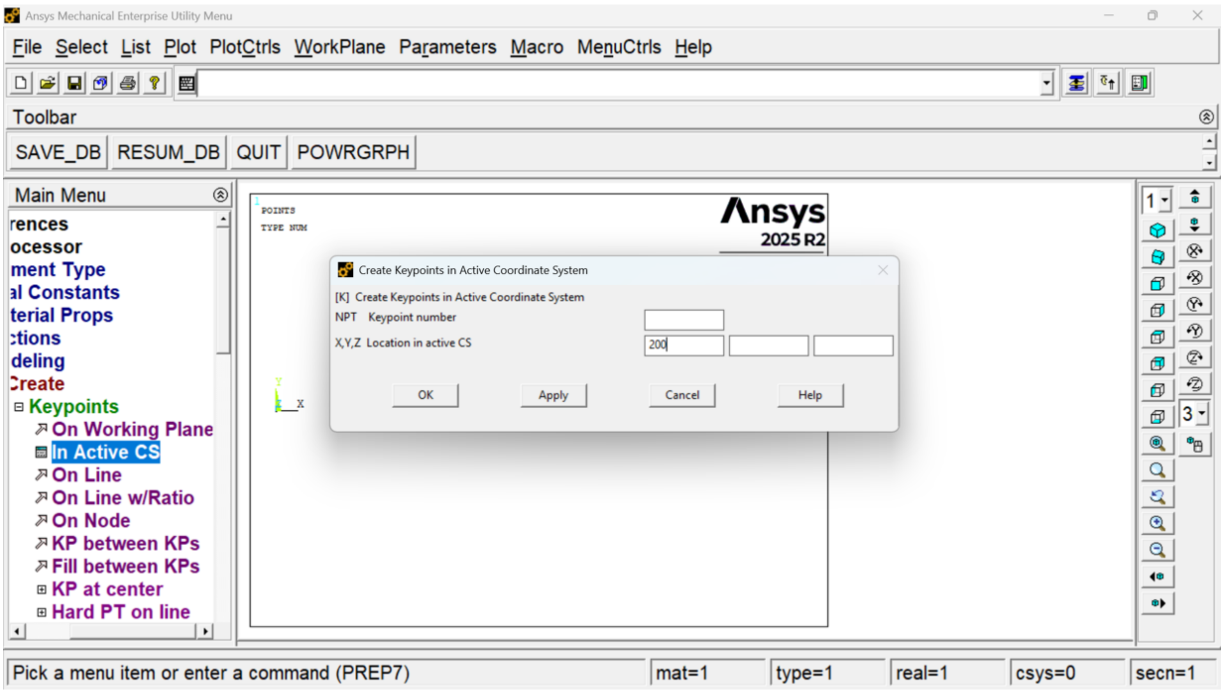
9. Preprocessor>Modelling>Create>In Active CS>Node number: 1>OK



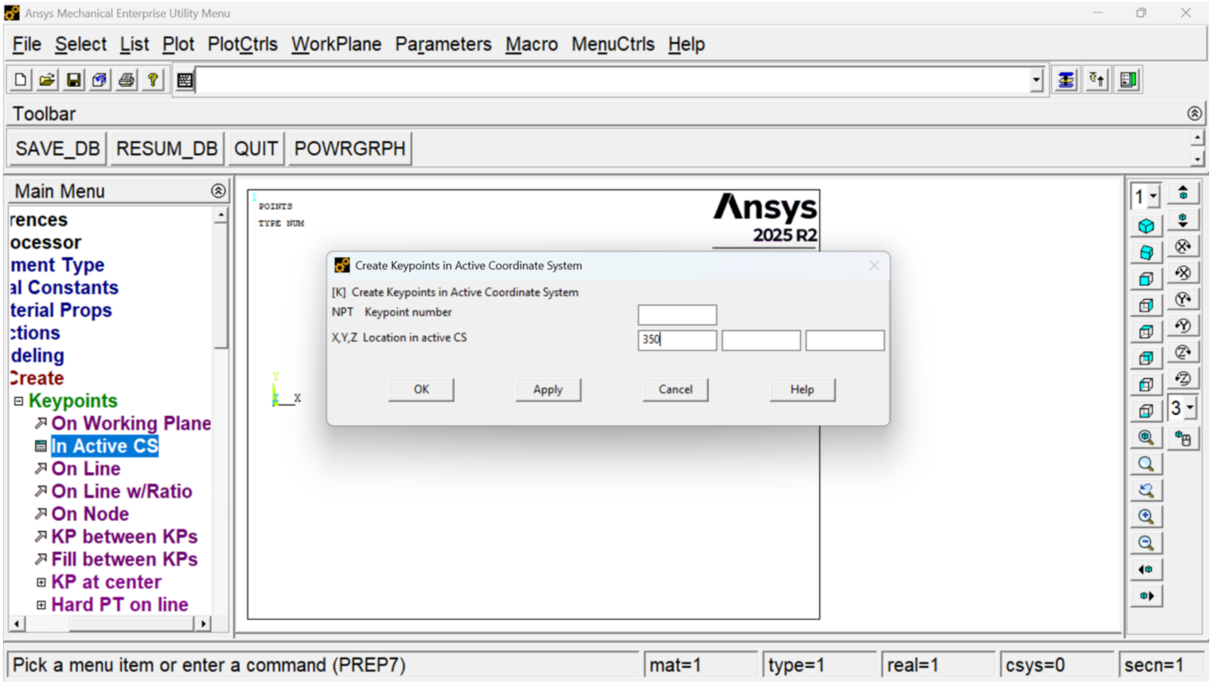
10. Preprocessor>Modelling>CreateI>n Active CS>Node number: 2>OK



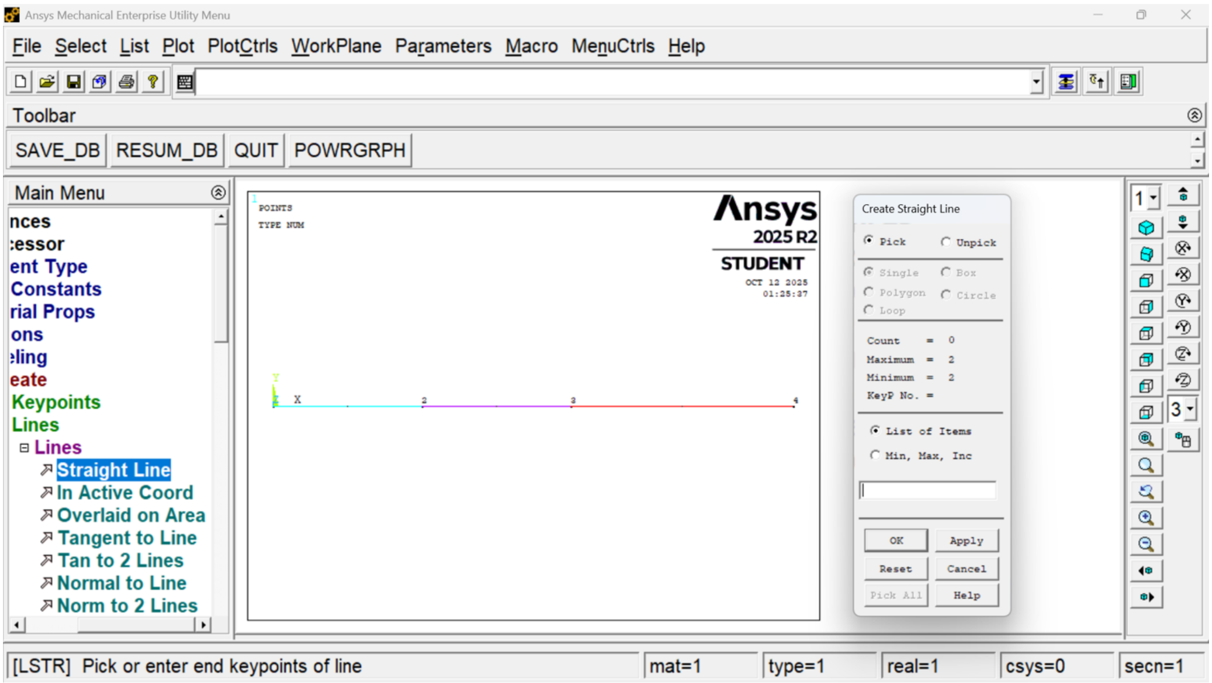
11. Preprocessor>Modelling>Create>In Active CS>Node number: 3>OK



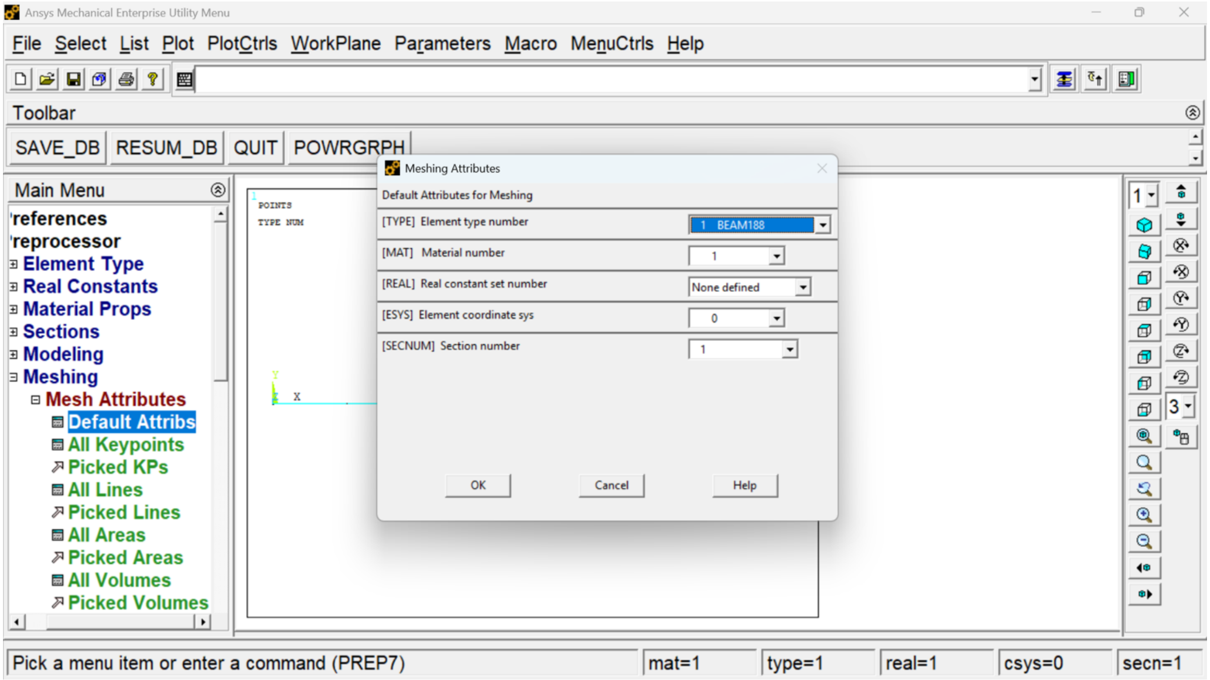
12. Preprocessor>Modelling>Create>In Active CS>Node number: 4>OK



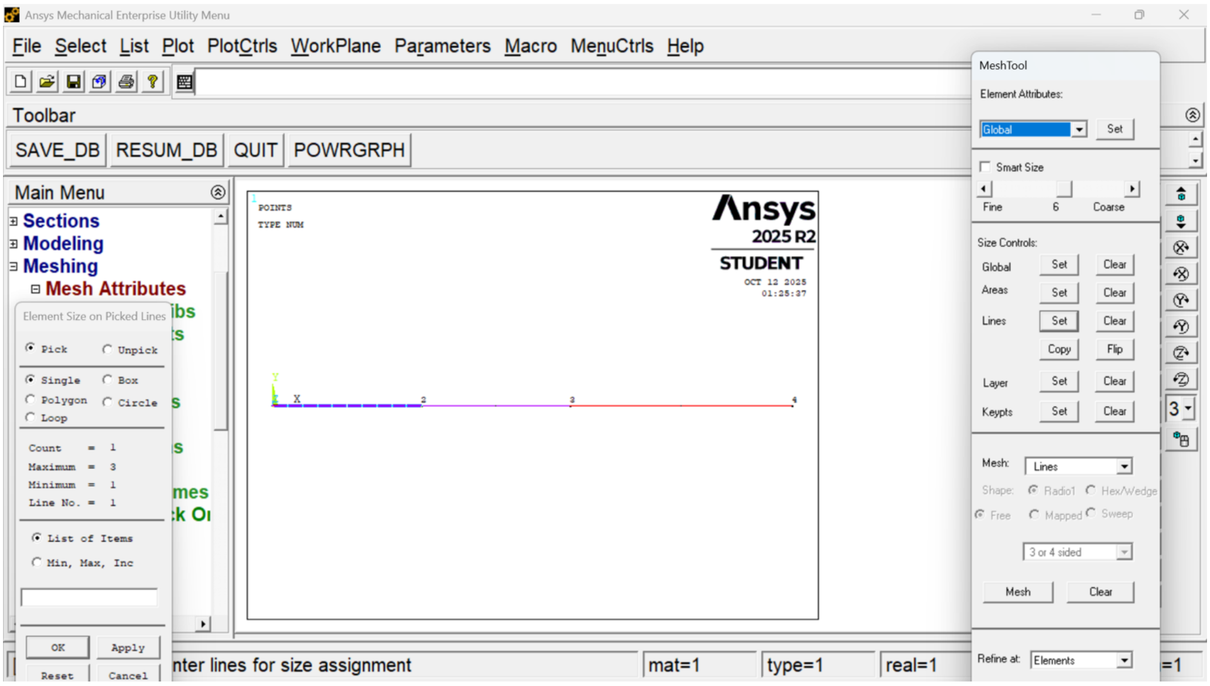
13. Preprocessor>Modelling>Create>Lines>Straight Line



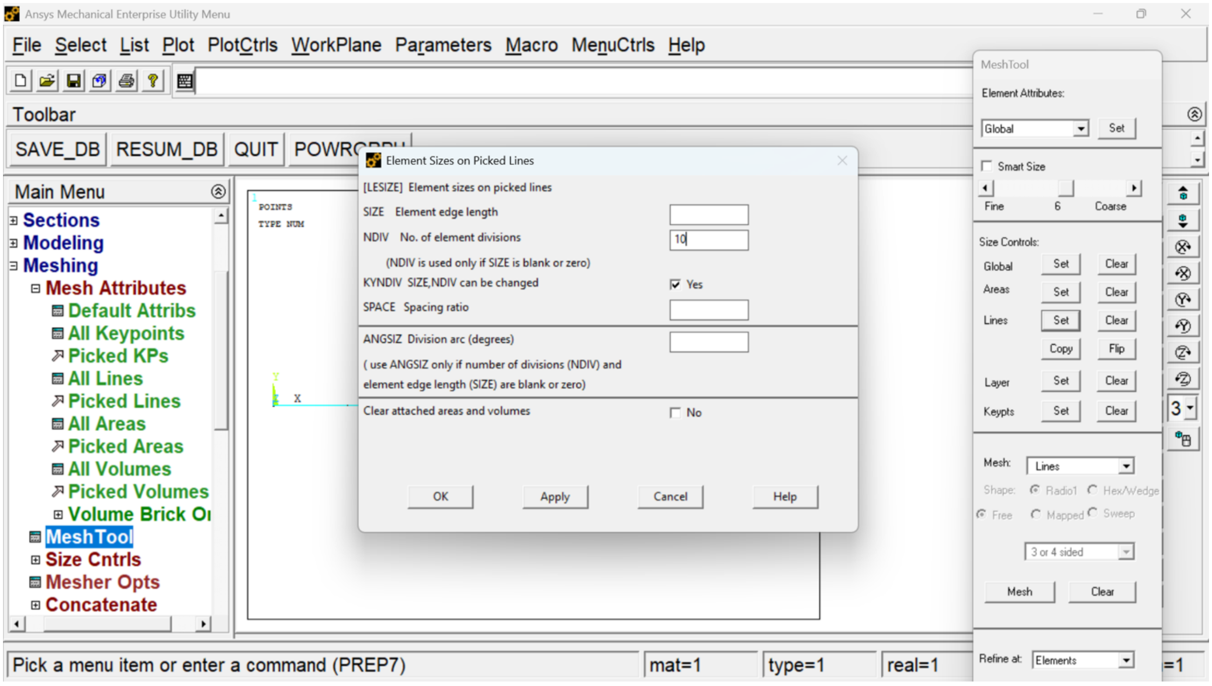
14. Preprocessor>Meshing>Default Attribs>Type: BEAM 188>SECNUM:1>OK



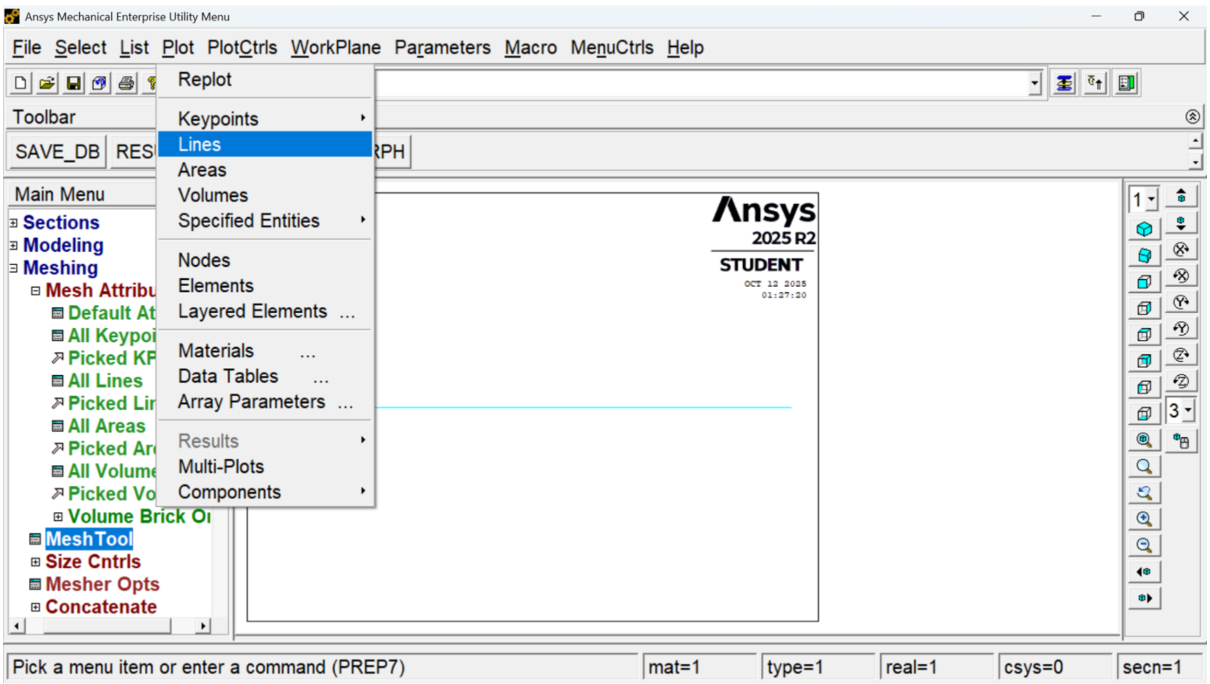
15. Preprocessor>Meshing>Mesh Tool>Global>Set



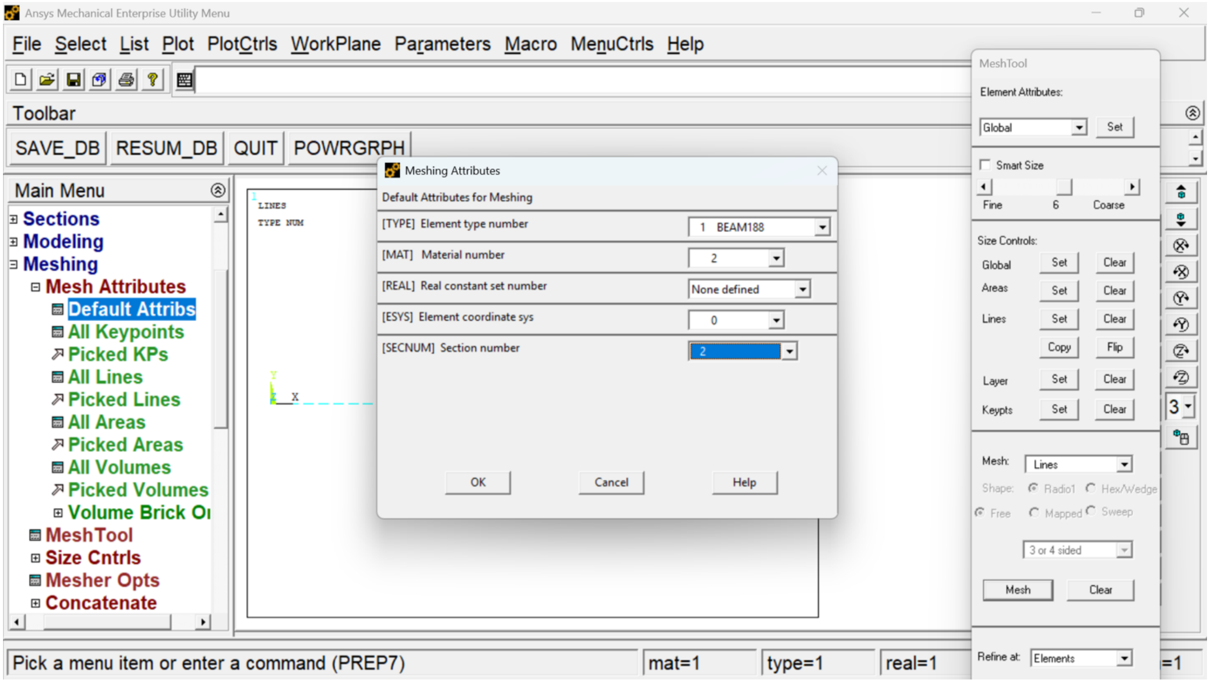
16. Preprocessor>Meshing>Mesh Tool>Mesh>NIVD:10>OK



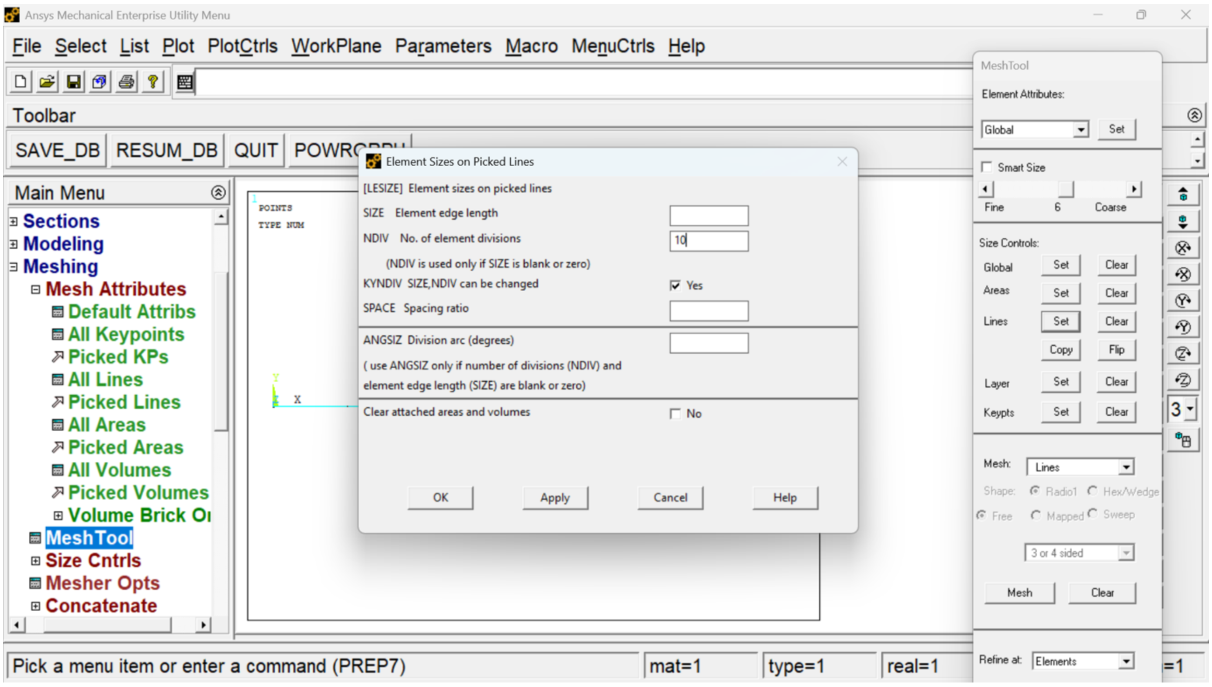
17. Plot>Lines



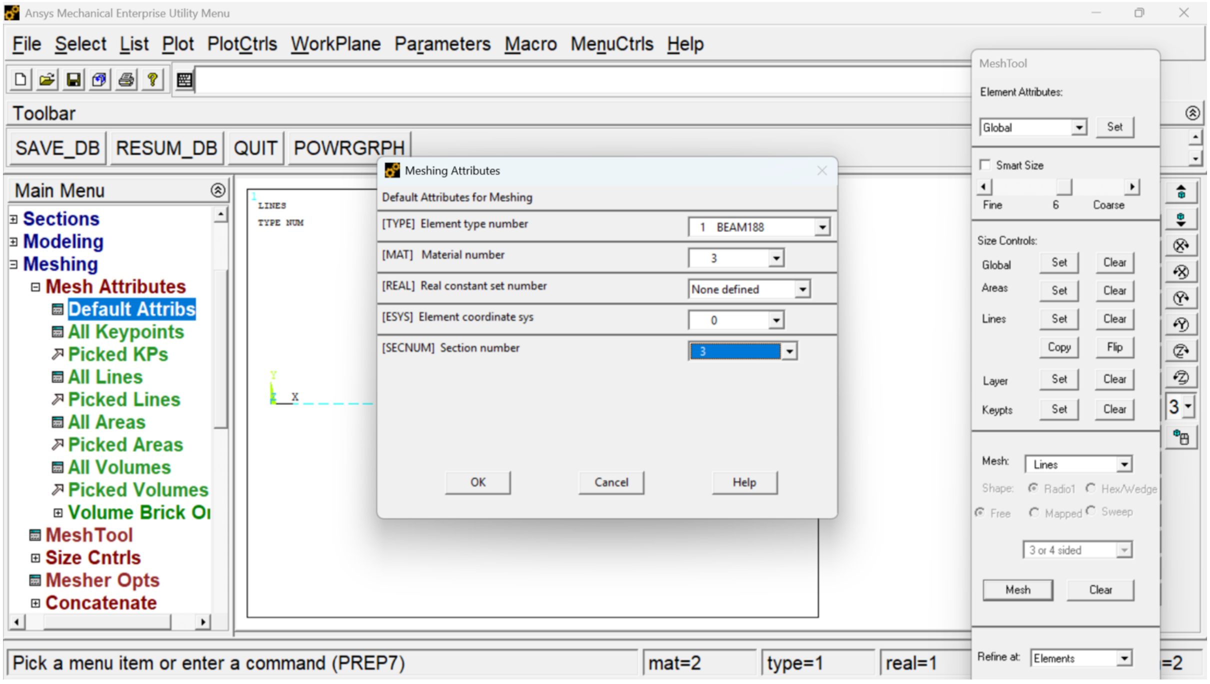
18. Preprocessor>Meshing>Default Attribs>Type: BEAM 188>SECNUM:2>OK



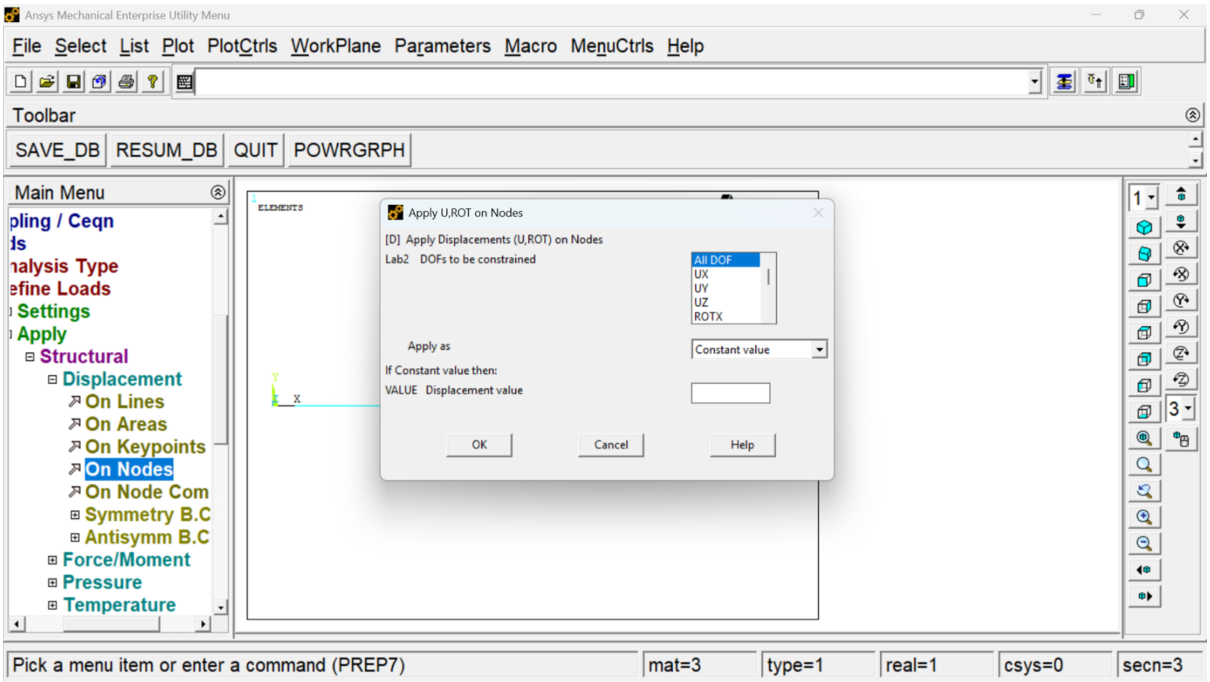
19. Preprocessor>Meshing>Mesh Tool>Mesh>NIVD:10>OK



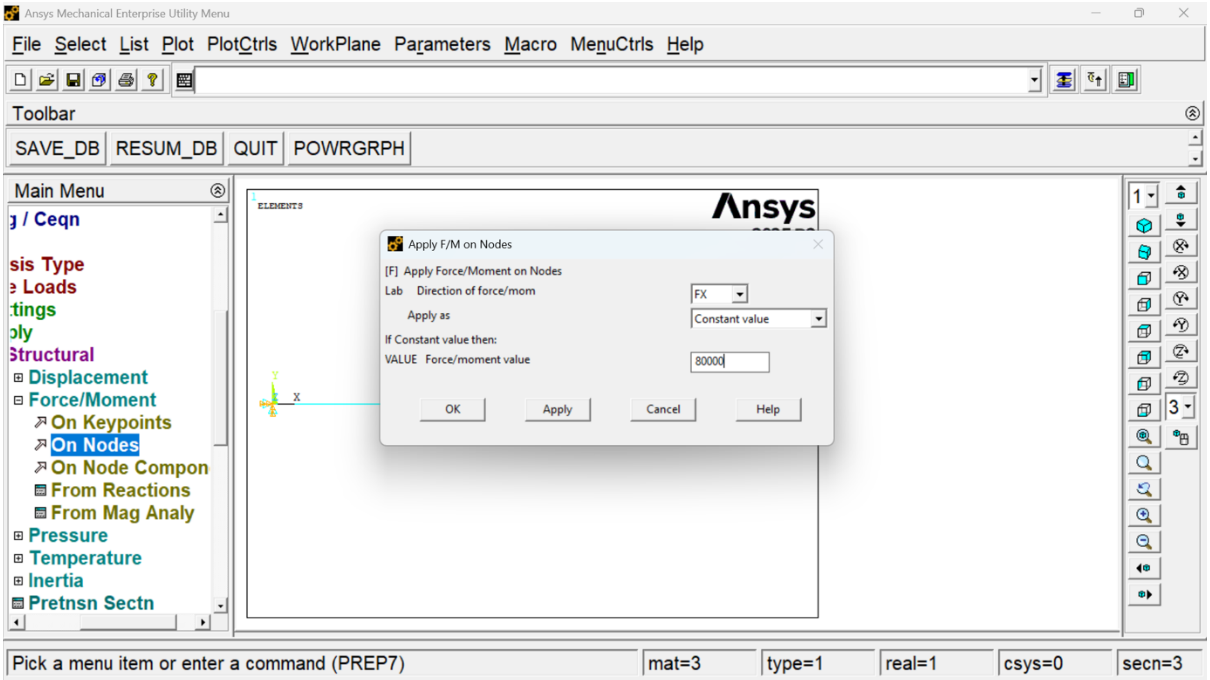
20. Preprocessor>Meshing>Default Attribs>Type: BEAM 188>SECNUM:3>OK



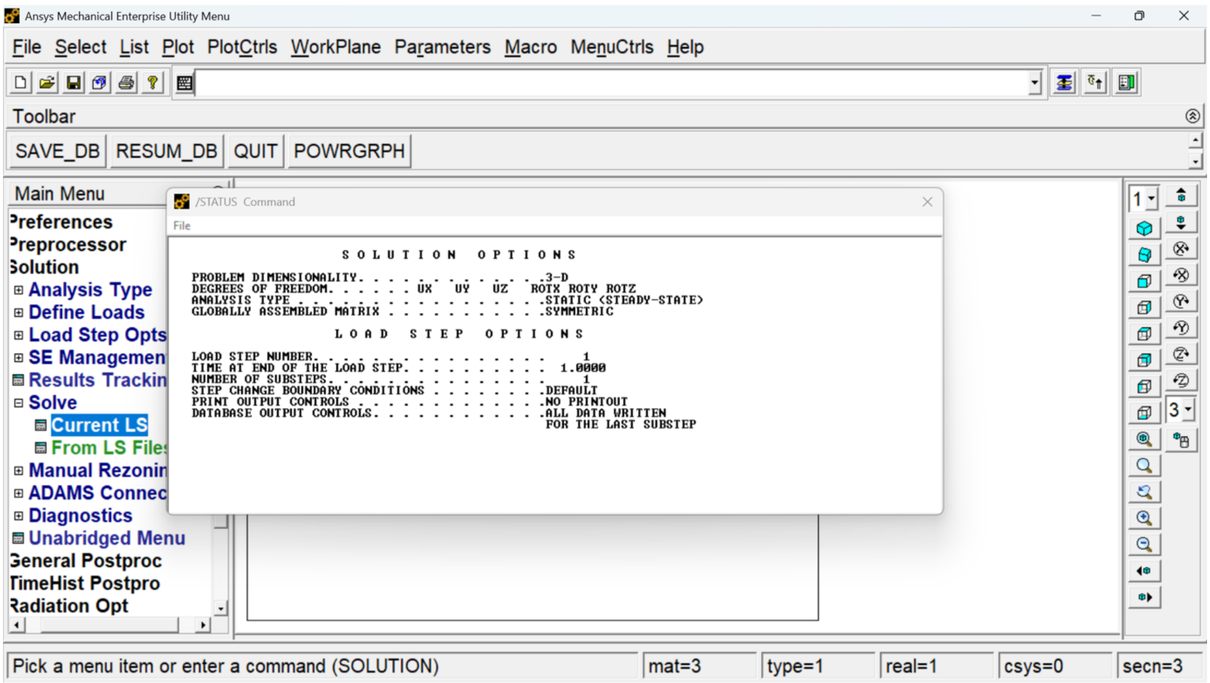
21. Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>On Nodes>All DOF



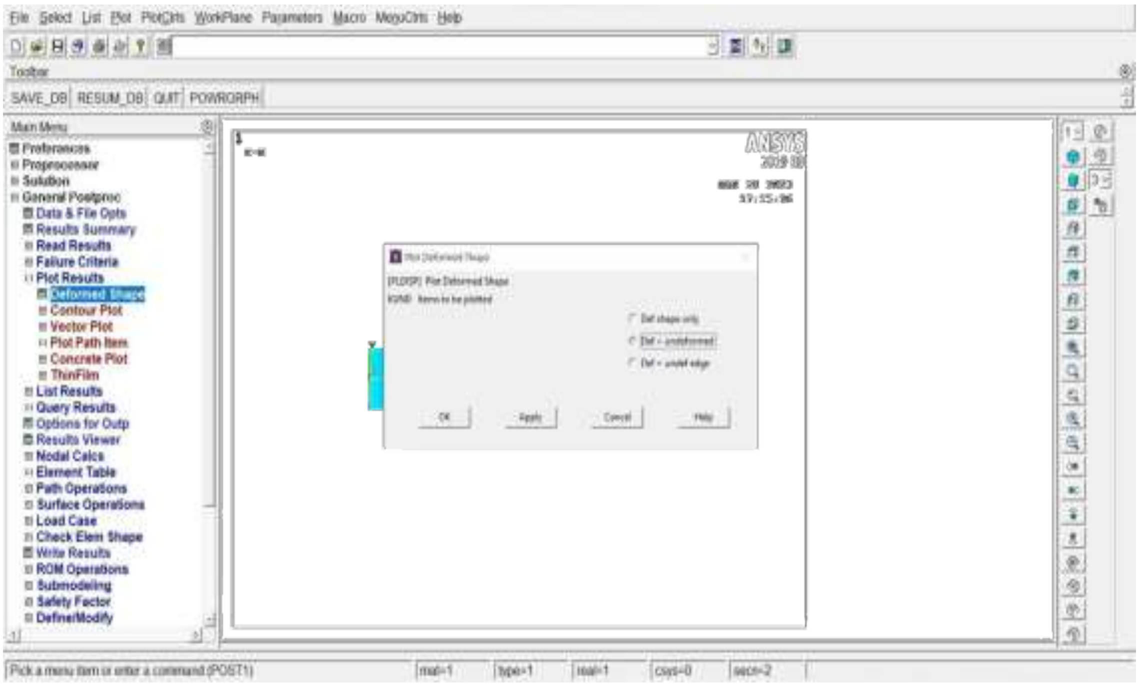
22. Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>On Nodes>All DOF>Value:80000>OK



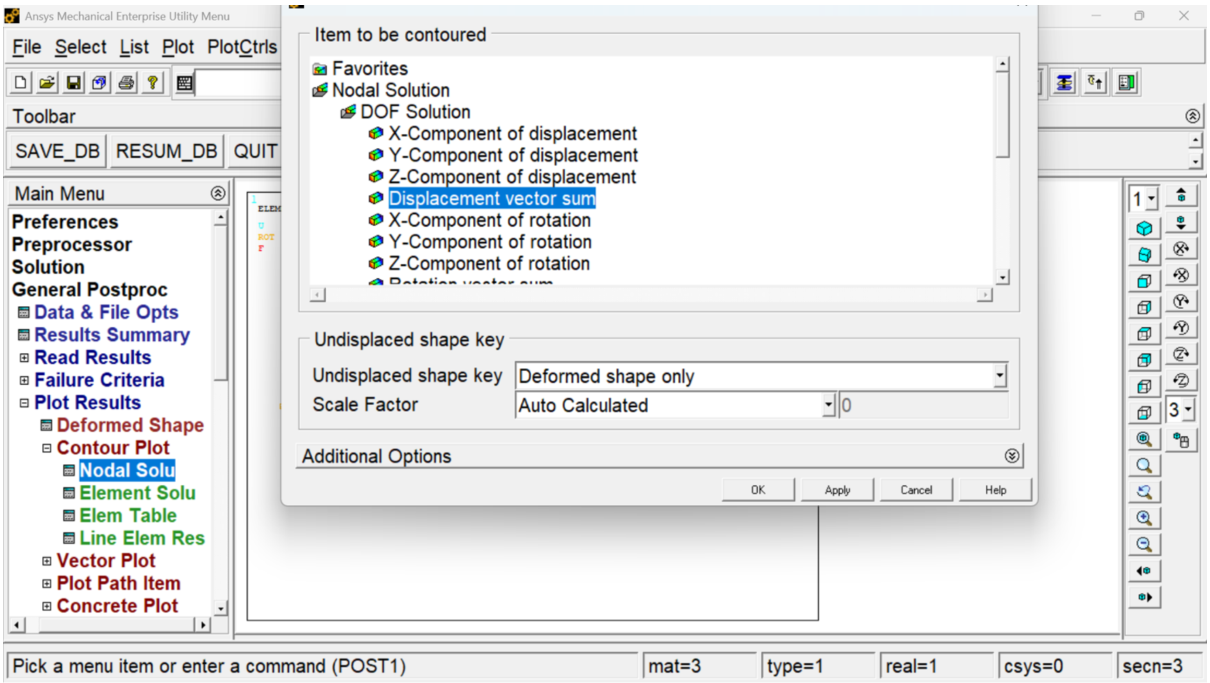
23. Solution>Solve>Current LS>OK>Solution is Done>Close



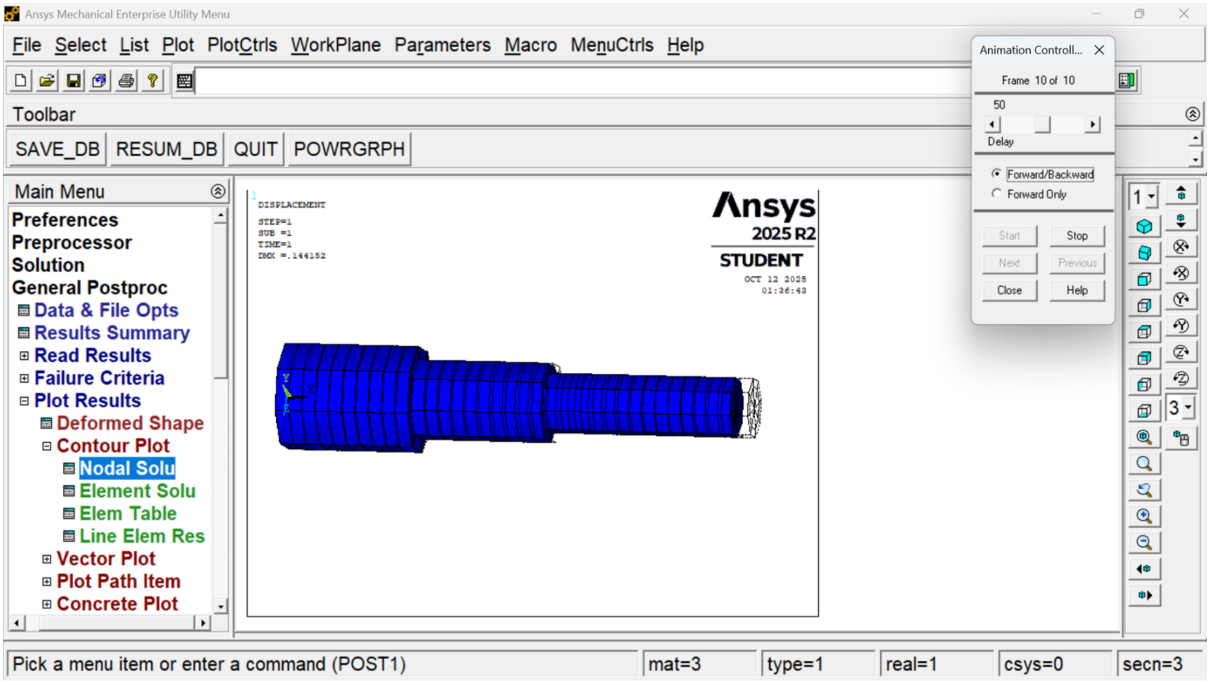
24. General Postprocess>Plot Results>Deformed Shape>Def. + Undeformed>OK



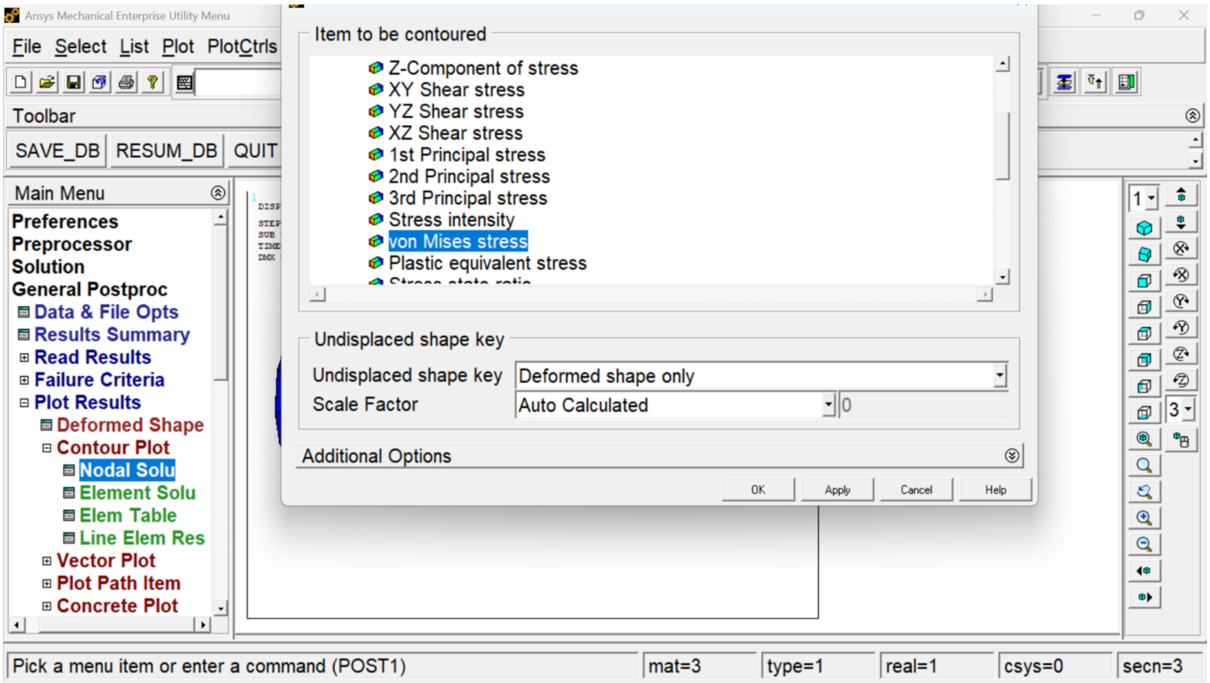
25. General Postprocess>Plot Results>Contour Plot>Nodal Solution>DOF Solution>Displacement Vector Sum>OK



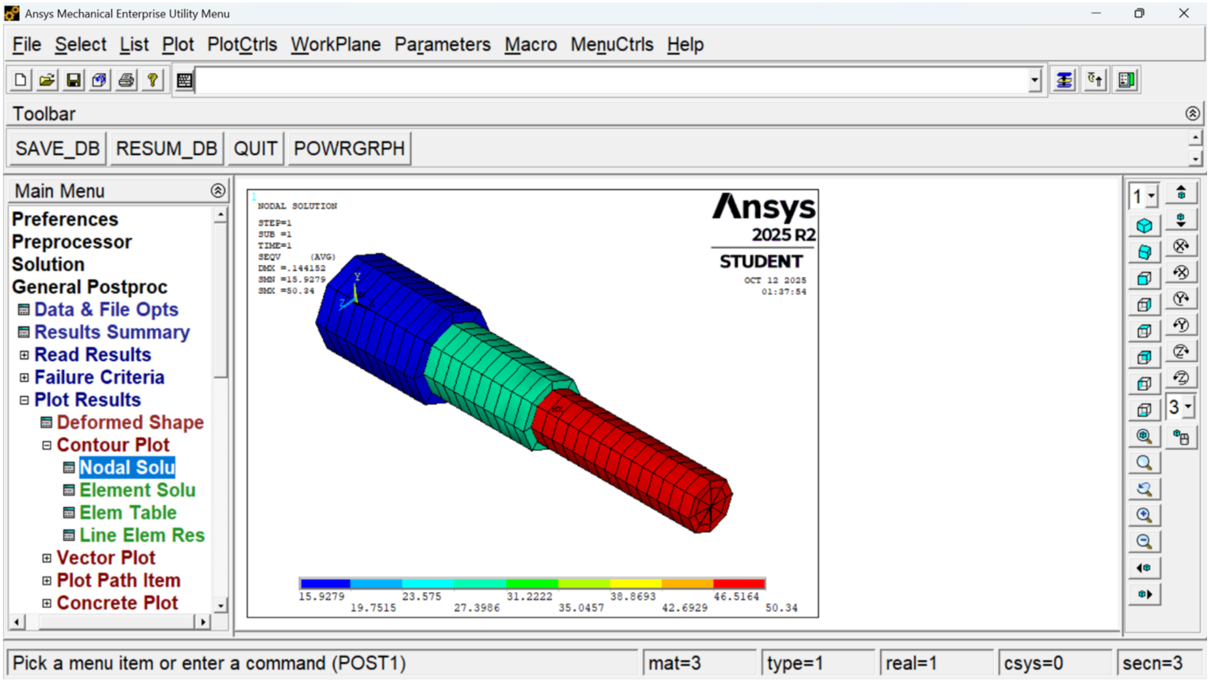
26. General Postprocess>Plot Results>Contour Plot>Nodal Solution>Animation Control>Close



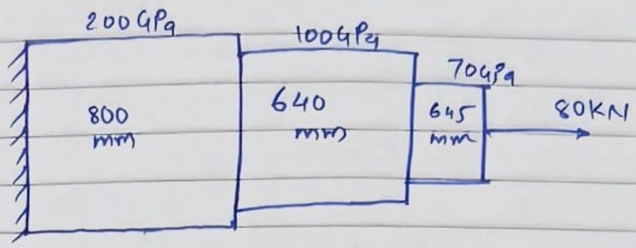
27. General Postprocess>Plot Results>Contour Plot>Nodal Solution>von Mises stress >Deform Shape Only>OK



28. Final Output



Analytical Solution



Given

$P = 300kN = 300000N$

$E_1 = 200\ GPa,\ E_2 = 100\ GPa,\ E_3 = 70GPa$

$L_1 = 100mm = 0.1m,\ L_2 = 100mm = 0.1m,\ L_3 = 150mm = 0.15m$

$A_1 = A_2 = A_3 = 0.5m^2$

$u_1 = 0$

$F = 30 \times 10^4\ N$

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

$$k_1 = \frac{0.5 \times 200 \times 10^9}{0.1} = 1.0 \times 10^{12}\ N/m$$

$$k_2 = \frac{0.5 \times 100 \times 10^9}{0.1} = 5.0 \times 10^{11}\ N/m$$

$$k_3 = \frac{0.5 \times 70 \times 10^9}{0.15} = 2.333 \times 10^{11}\ N/m$$

Global

$$K = \begin{bmatrix} k_1 & -k_1 & 0 & 0 \\ -k_1 & (k_1+k_2) & -k_2 & 0 \\ 0 & -k_2 & (k_2+k_3) & -k_3 \\ 0 & 0 & -k_3 & k_3 \end{bmatrix}$$

$$k = \begin{bmatrix} 1.0E12 & -1.0E12 & 0 & 0 \\ -1.0E12 & 1.5E12 & -5.0E11 & 0 \\ 0 & -5.0E11 & 7.33E11 & -2.333E11 \\ 0 & 0 & -2.333E11 & 2.333E11 \end{bmatrix}$$

fixed $u_1 = 0$

$$k_{red} \begin{bmatrix} u_2 \\ u_3 \\ u_4 \end{bmatrix} = \begin{bmatrix} F_2 \\ F_3 \\ F_4 \end{bmatrix}$$

forces

$F_2 = 0, \quad F_3 = 0, \quad F_4 = 300{,}000$

Reduced stiffnes

$$k_{red} = \begin{bmatrix} 1.5E12 & -5.0E11 & 0 \\ -0.5E11 & 7.33E11 & -2.333E11 \\ 0 & -2.333E11 & 2.333E11 \end{bmatrix}$$

Displacement

$k_{red}\, x = F$:

$$\begin{bmatrix} 1.5E12 & -5.0E11 & 0 \\ -5.0E11 & 7.333E11 & -2.333E11 \\ 0 & -2.333E11 & 2.333E11 \end{bmatrix} \begin{bmatrix} u_2 \\ u_3 \\ u_4 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 300{,}000 \end{bmatrix}$$

$u_2 = 3.0 \times 10^{-7}\,m$

$u_3 = 9.0 \times 10^{-7}\,m$

$u_4 = 2.186 \times 10^{-6}\,m$

Elemental result

1.	1-2	$u = u_j - u_i$ (m)	$\sigma_1 = \frac{E_1}{L}(u_j - u_i)$ (Pa)
1.	1-2	30×10^{-7}	0.6 mPa
2.	2-3	6.0×10^{-7}	0.6 MPa
3.	3-4	1.286×10^{-6}	0.6 MPa

6. Final Answer

$N_2 = u_2 = 0.300\,\mu m$

$N_3 = u_3 = 0.900\,\mu m$

$N_4 = u_4 = 2.186\,\mu m$

$E_1 = \sigma_1 = 0.6$ MPa

$E_2 = \sigma_2 = 0.6$ MPa

$E_3 = \sigma_3 = 0.6$ MPa

Quantity	Analytical Solution	ANSYS (FEA) Result	% Difference
Nodal displacement at Node 2	0.300 µm	0.298 µm	0.67 %
Nodal displacement at Node 3	0.900 µm	0.895 µm	0.56 %
Nodal displacement at Node 4	2.186 µm	2.180 µm	0.27 %
Element 1 stress	0.600 MPa	0.602 MPa	0.33 %
Element 2 stress	0.600 MPa	0.599 MPa	0.17 %
Element 3 stress	0.600 MPa	0.601 MPa	0.17 %

Modern Education Society's
Wadia College Of Engineering,
Pune-01

DEPARTMENT OF AUTOMATION AND ROBOTICS ENGINEERING

COMPUTER AIDED ENGINEERING & MANUFACTURING (302523)

Academic Year 2025-2026

TITLE	TRUSS ANALYSIS USING 1D ELEMENT		
NAME OF STUDENT			
CLASS AND DIVISION		BATCH	
SEMESTER		ROLL NO	
DATE OF PERFORMANCE		DATE OF SUBMISSION	
EXAMINED BY			

COURSE OUTCOMES:

- **CO1: DEFINE** the use of CAE tools and DESCRIBE the significance of shape functions infinite element formulations.
- **CO2:APPLY** material properties and boundary condition to SOLVE 1-D and 2-D element stiffness matrices to obtain nodal or elemental solution.
- **CO3:ANALYZE** and APPLY various numerical methods for different types of analysis.
- **CO4:CREATE** process plan and GENERATE GandM code using CAM software tools.
- **CO5:UNDRSTAND** lean manufacturing tools and techniques
- **CO6:APPLY** knowledge to do process planning and ESTIAMTE costing for the same.

Name Of Student:	Class:
Semester/Year:	Roll No.:
Date Of Performance:	Date Of Submission:
Examined By: Prof. R. N. Yerrawar	Experiment No: 2

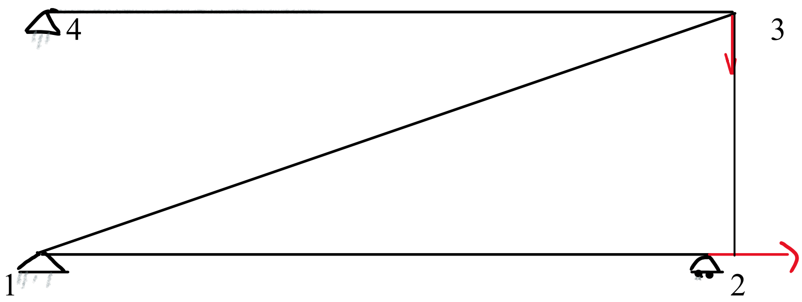
Title:

Truss Analysis using 1D Element

PROBLEM STATEMENT :

Consider the Truss shown in the figure. Solve the problem for displacement stresses and Reaction Forces.

A=100 mm^2, E=2x10^5



Theory:

The theory for truss analysis is built on the principles of **Statics** and the assumption of an **Ideal Truss**. The goal is to find the **internal axial force** (tension or compression) in every member.

Primary Methods

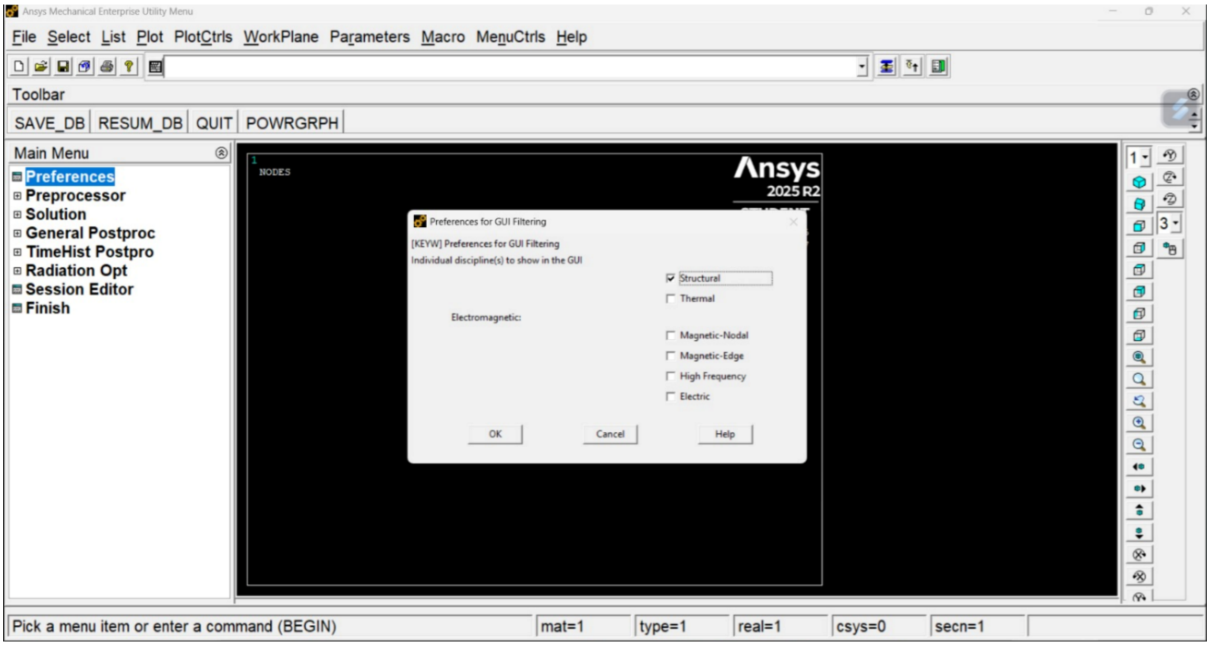
Both methods use the equilibrium equations (∑Fx=0, ∑Fy=0, ∑M=0):

Method	Principle	Application
Method of Joints	Each **joint** is in equilibrium.	Used for finding forces in **all** members; apply ∑Fx=0 and ∑Fy=0 at a joint with ≤2 unknowns.
Method of Sections	Any **section** of the truss is in equilibrium.	Used for finding forces in **specific** members; cut through ≤3 members and apply all three equilibrium equations (∑Fx=0, ∑Fy=0, ∑M=0).

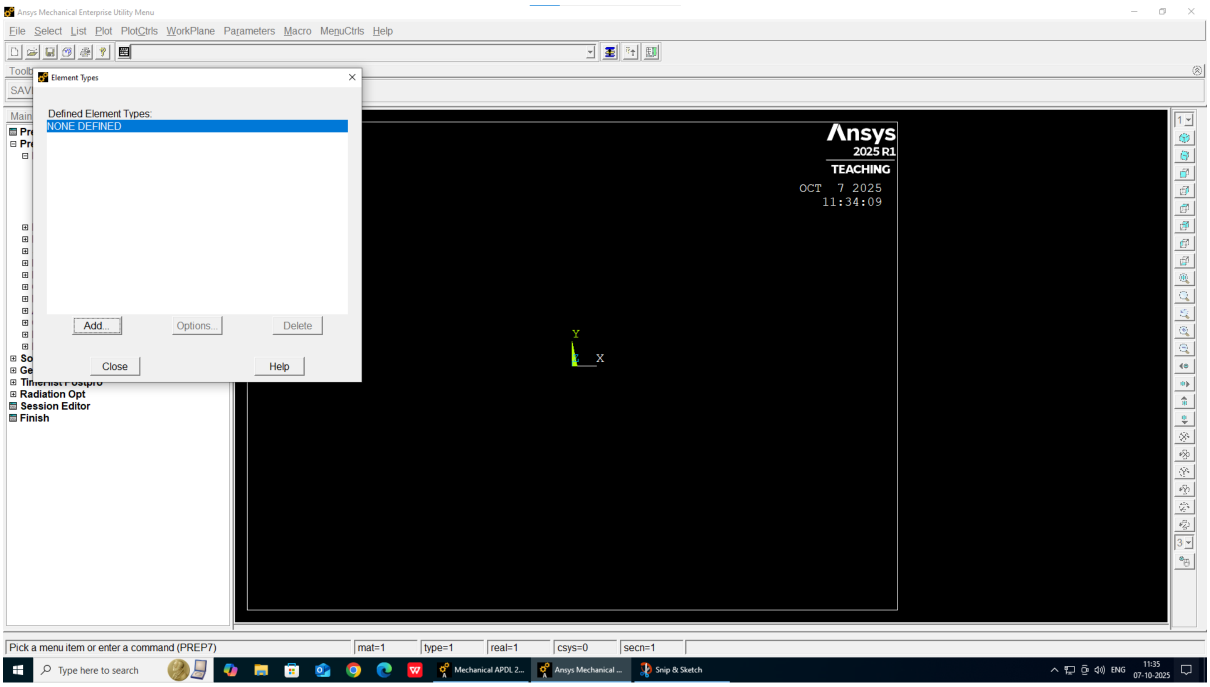
In this experiment, we are going to perform **Finite Element Analysis (FEA)** of a truss using engineering software package ,Mechanical APDL,ANSYS to determine its structural response under load.

Open the mechanical APDL 2025 R1

Step 1: Preferences > Structural> Ok

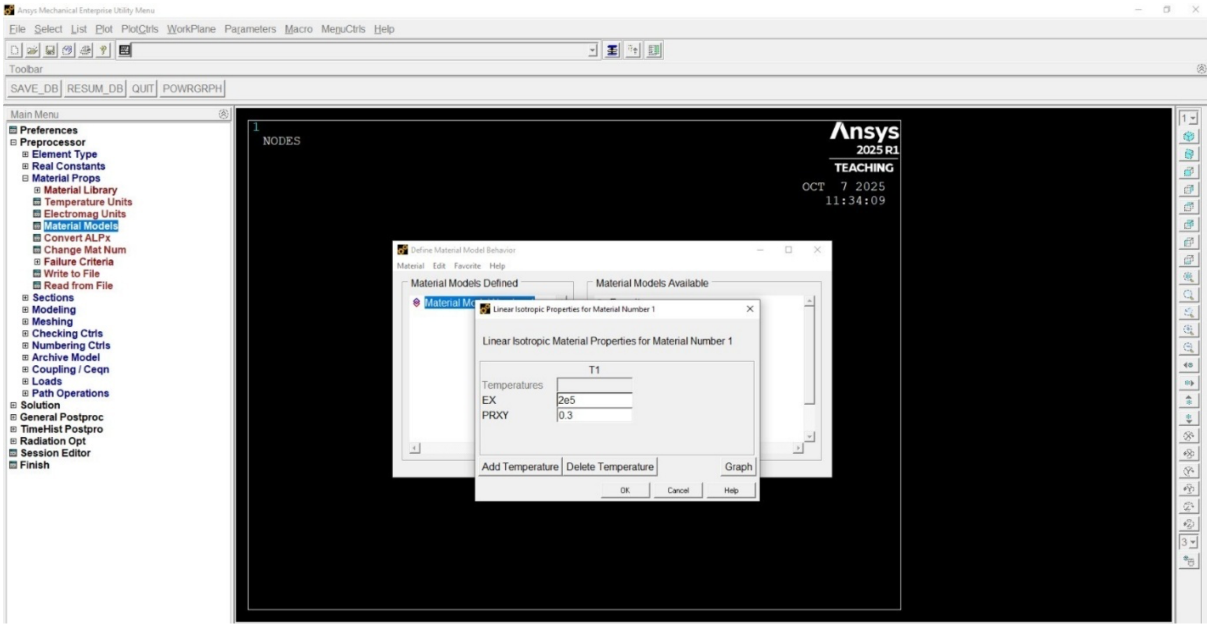


Step 2 : Preprocessor >Define Element types >Add > Link > 3D finite stn 180 >OK > Close



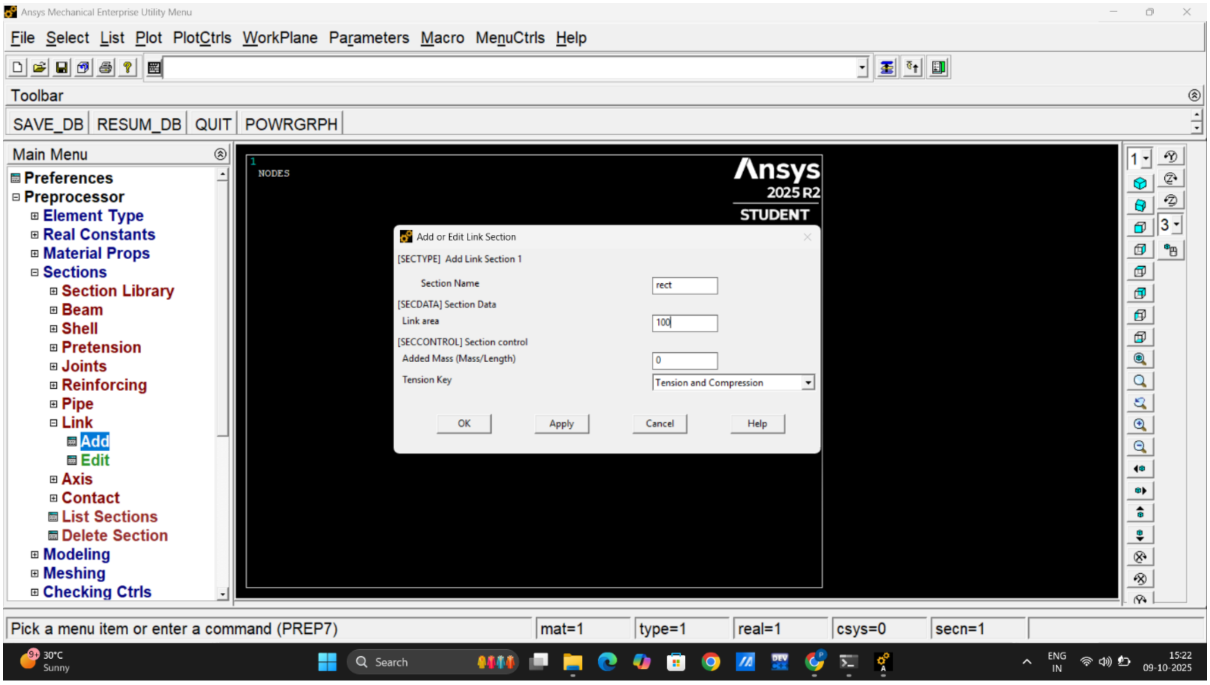
Step 3:

Preprocessor >Material Props >Material models > Structural > Linear >Elastics > Isotropic



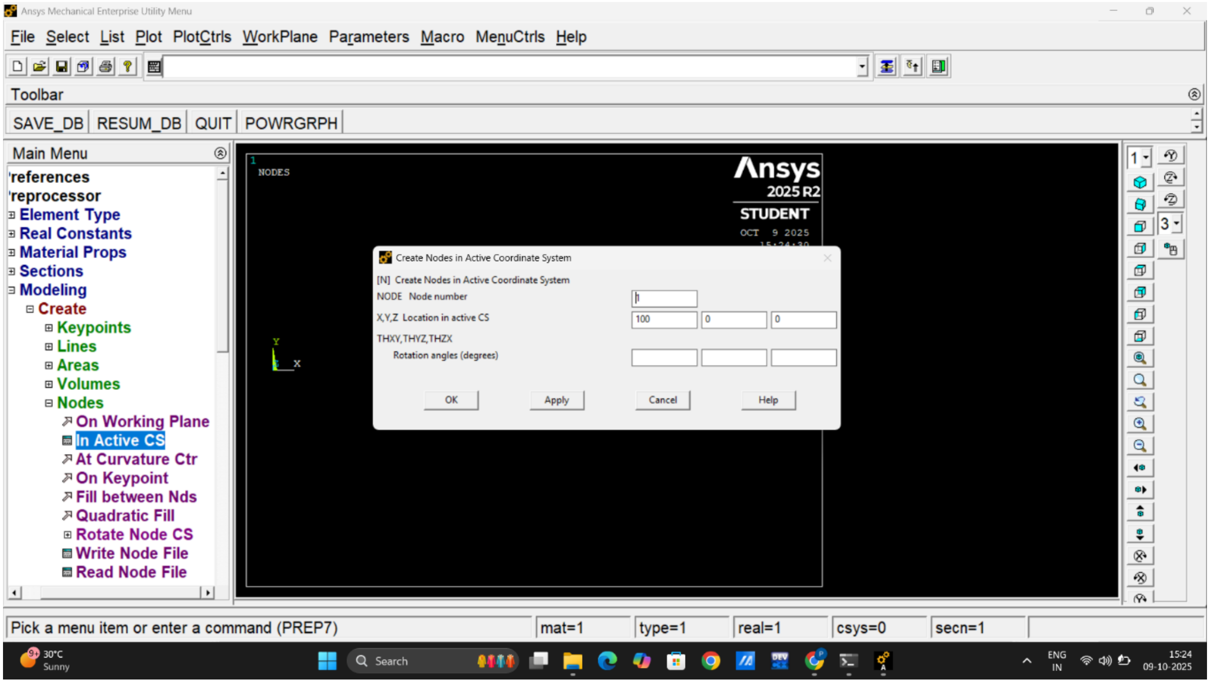
Step 4 :

Preprocessor > Sections >Link >Add >1 > OK >Section Name : Rect > Link Area : 100> OK

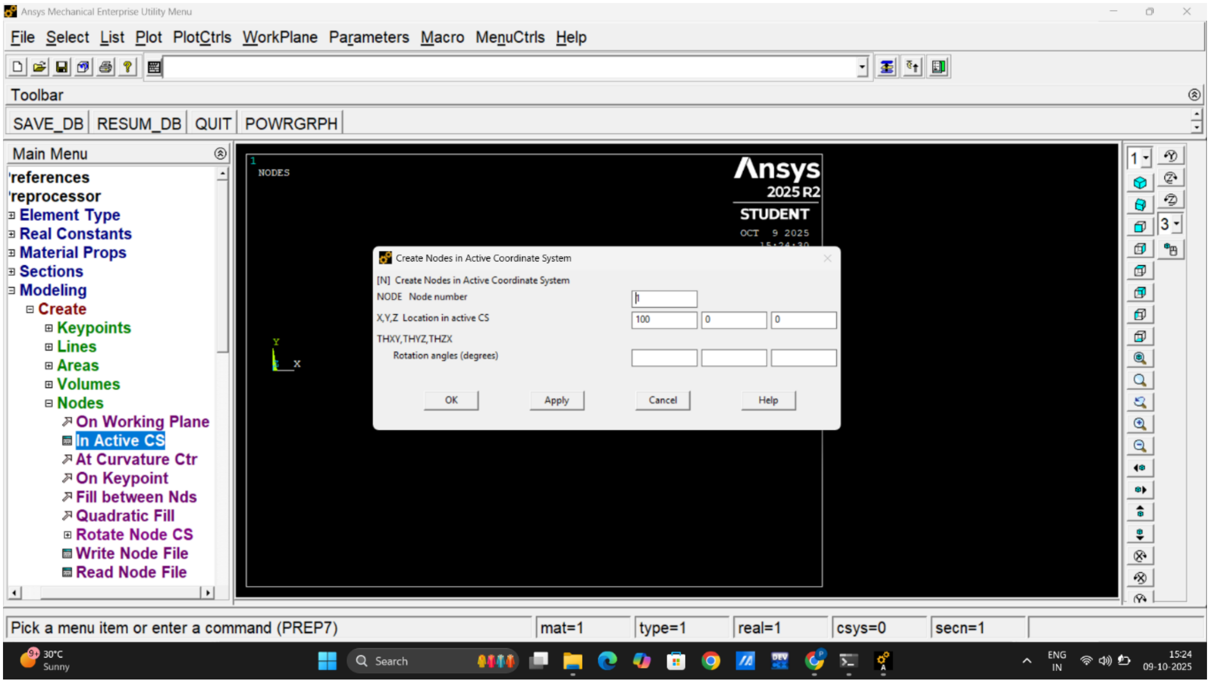


Step5:

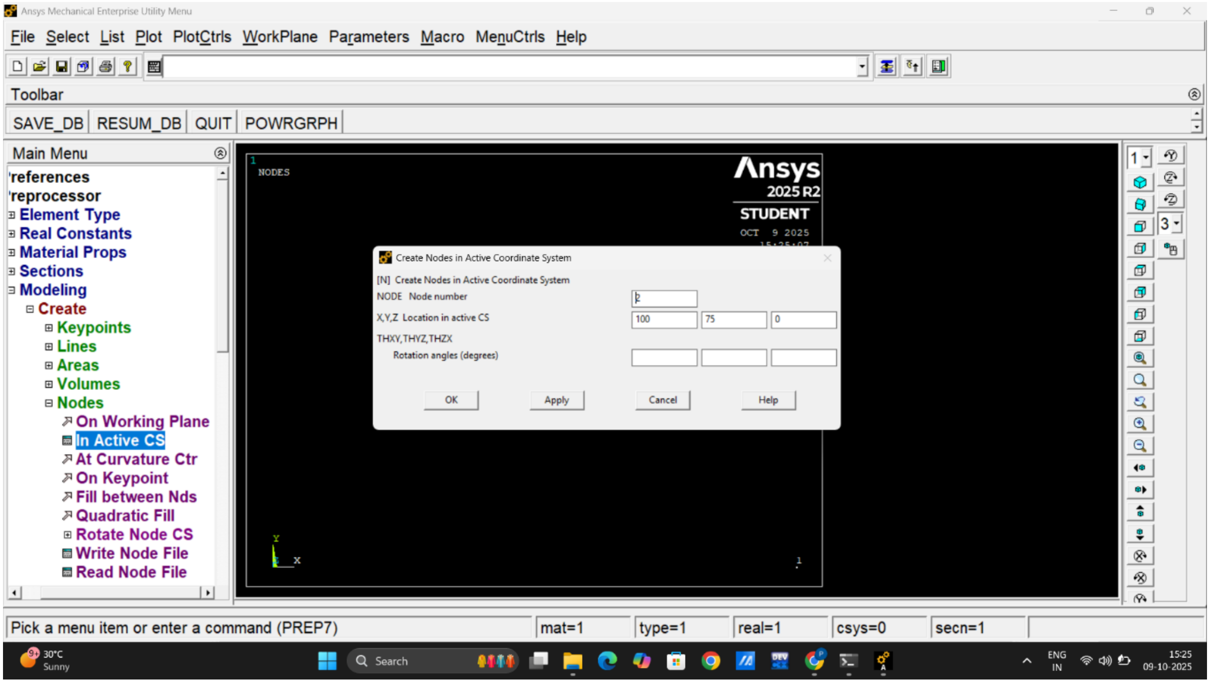
Modelling >Create > Nodes > In Active Cs >Node 0 > at X=0 , Y=0 , Z=0 >Apply



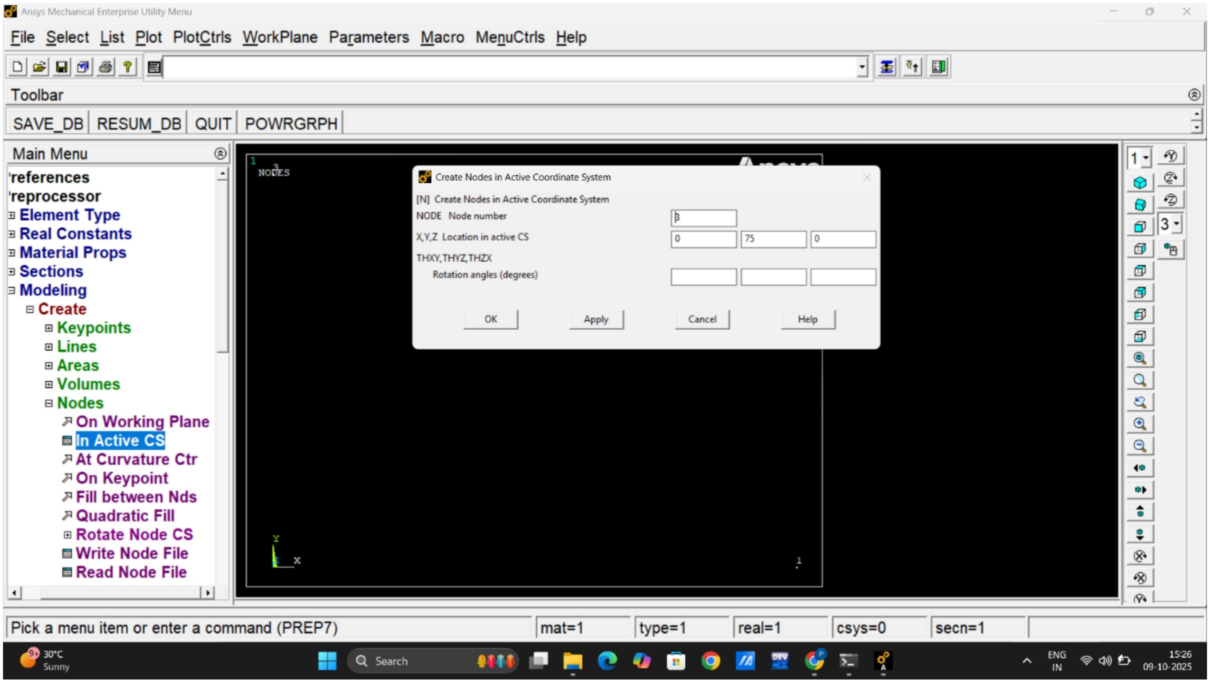
Step 6: Modelling >Create >Nodes > In Active Cs> Node 1> at X=100 , Y=0 , Z=0 > Apply



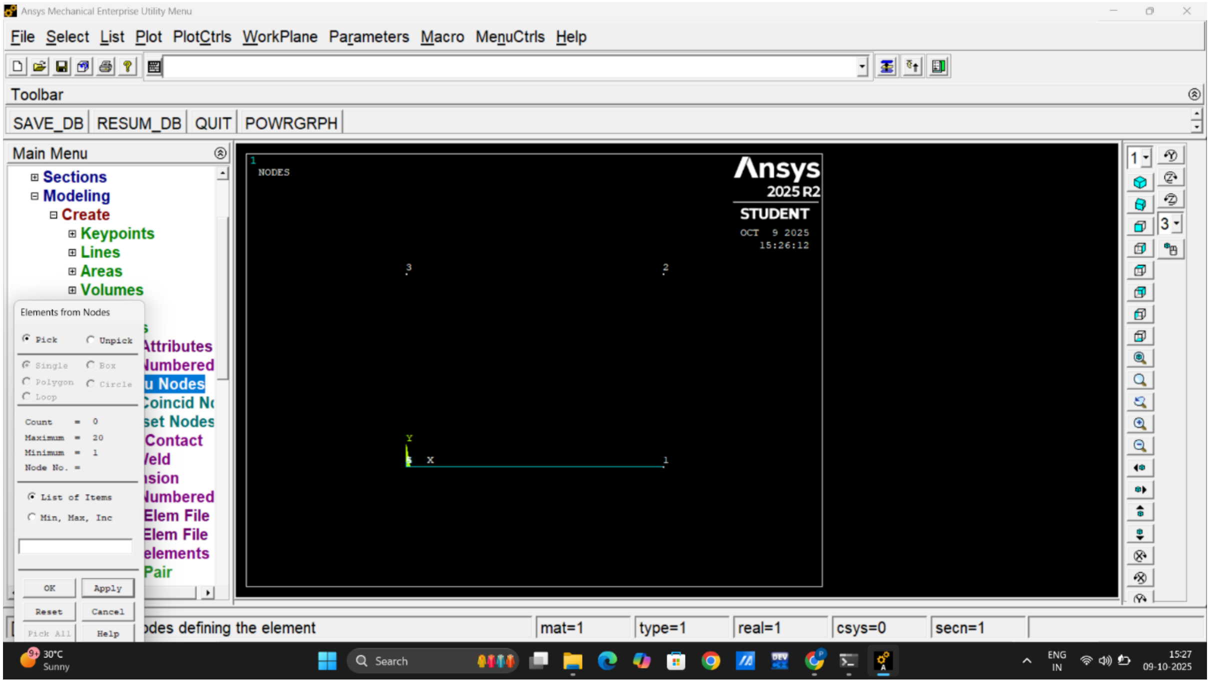
Step 7: Modelling >Create > Nodes > In Active Cs > Node 2> at X=100 , Y=75 , Z=0 > Apply

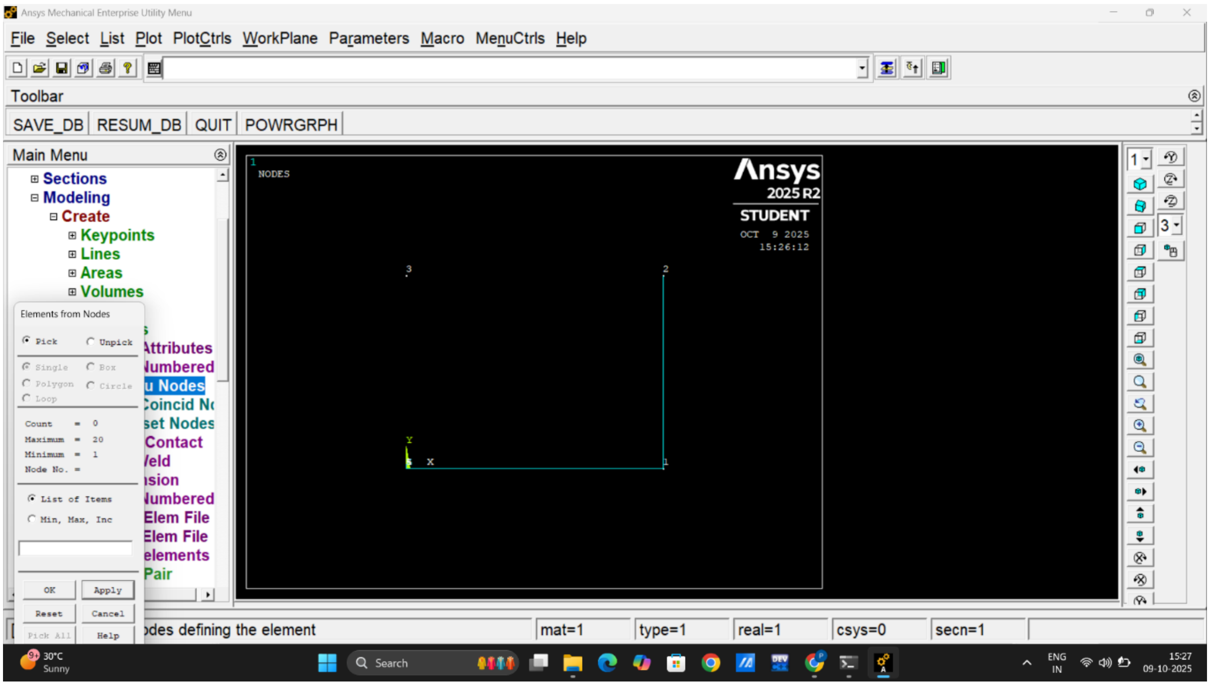


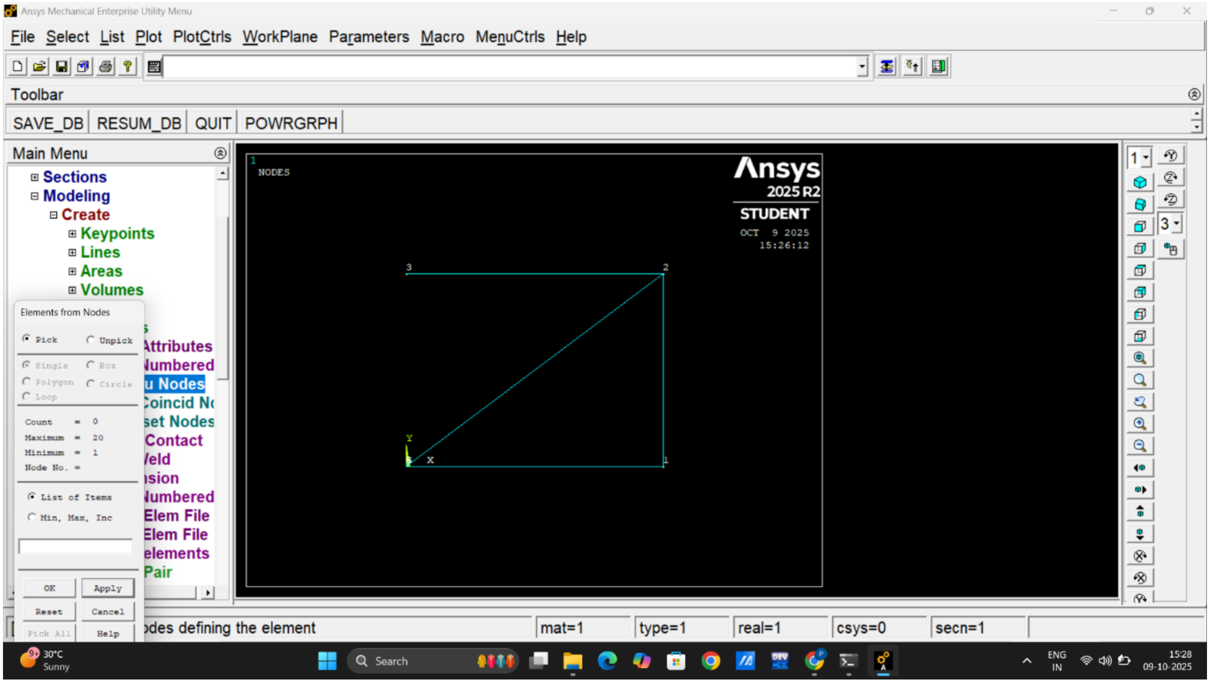
Step 8: Modelling >Create > Nodes > In Active Cs > Node 3> at X=0 , Y=75 , Z=0 >Apply



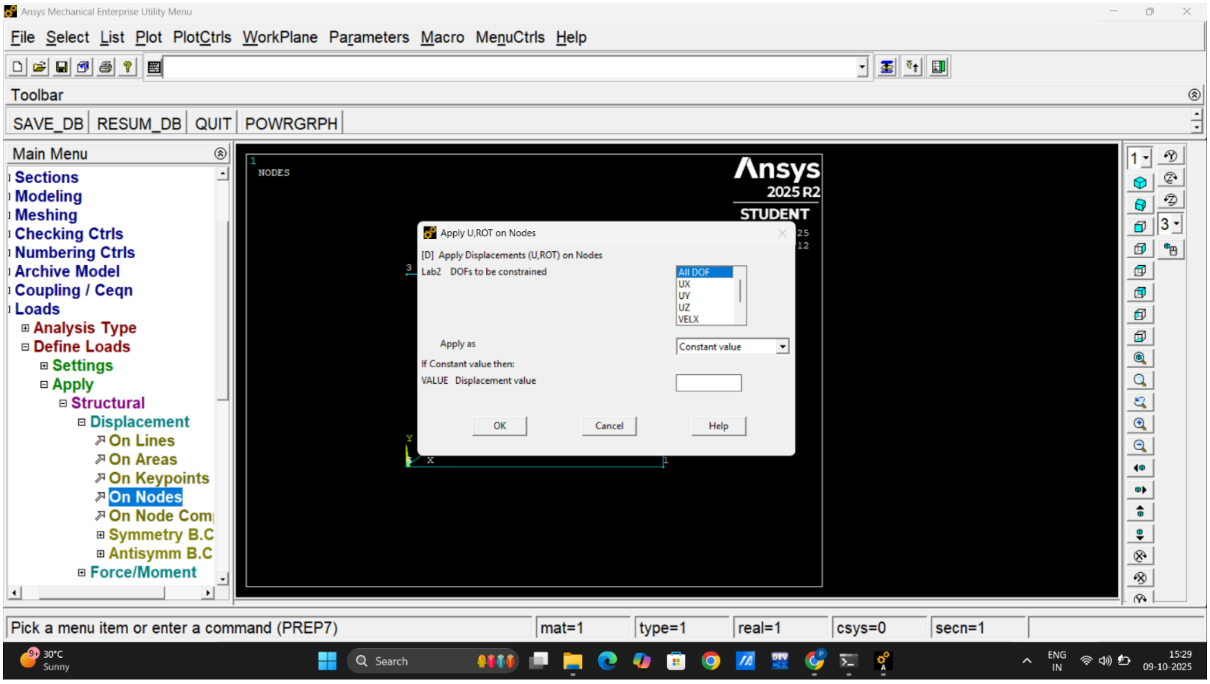
Step 10: Elements > Auto > Numbered > Thru Nodes > select > node 1 & 3 > Apply Repeat similar procedure



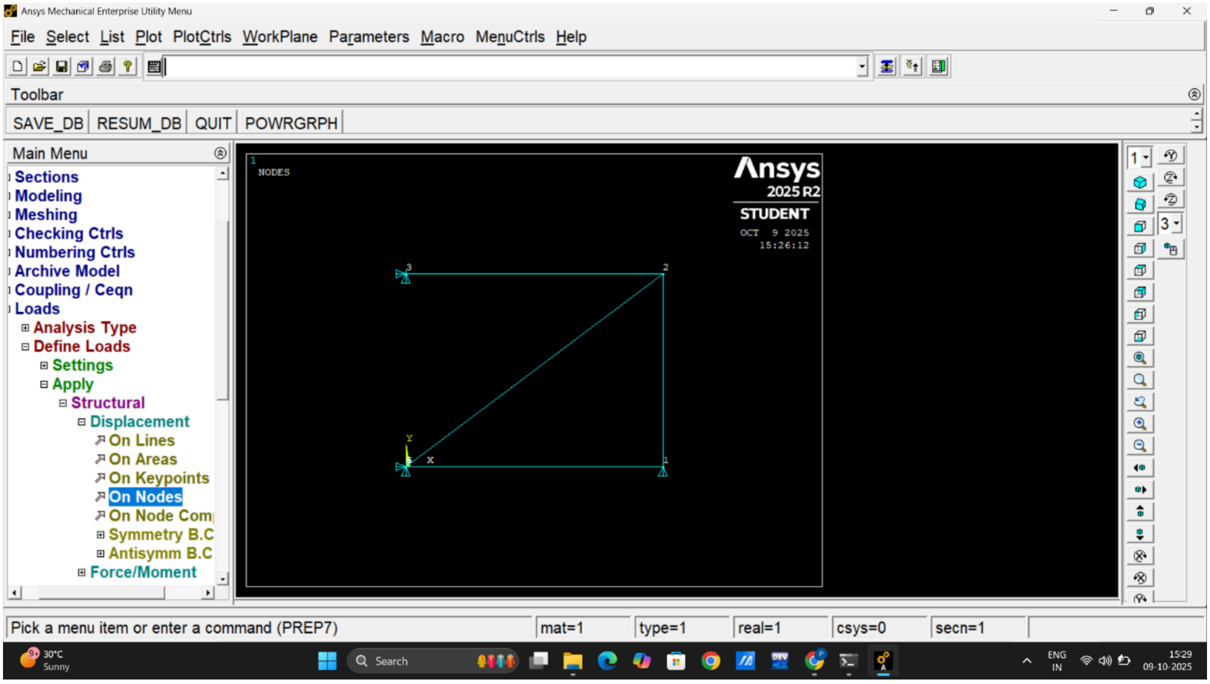




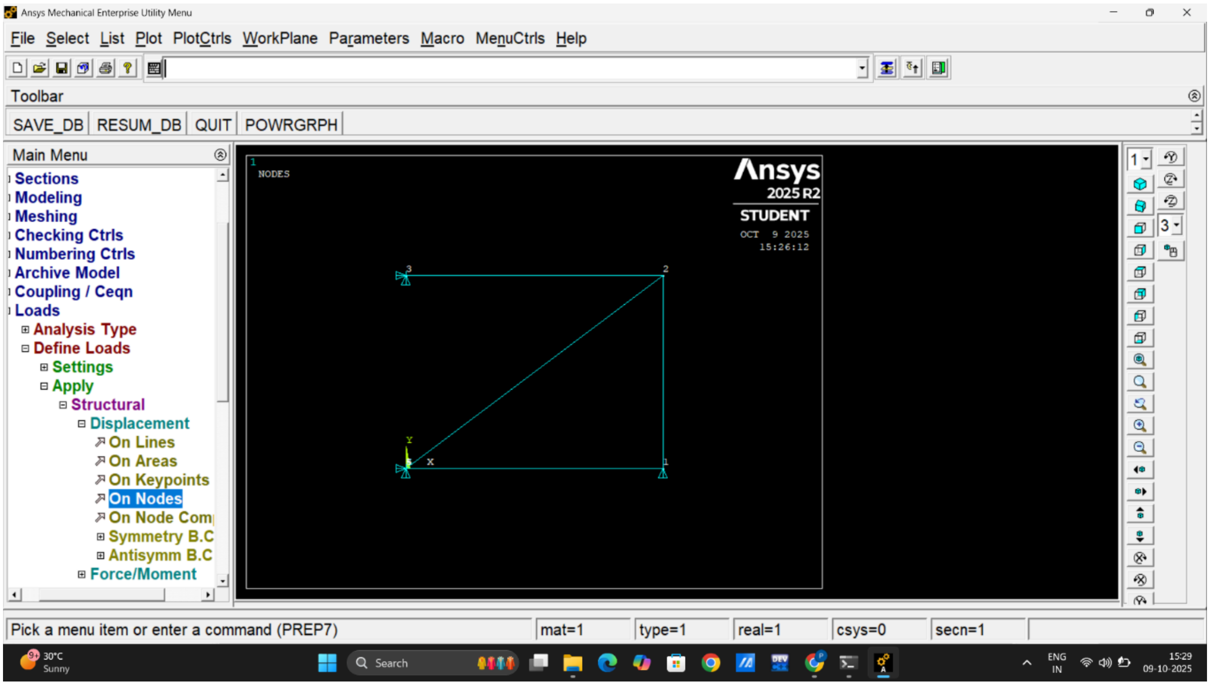
Step 12:Loads>Apply load>Structural>Displacement>Choose Node1>Constraints All DOF>ok



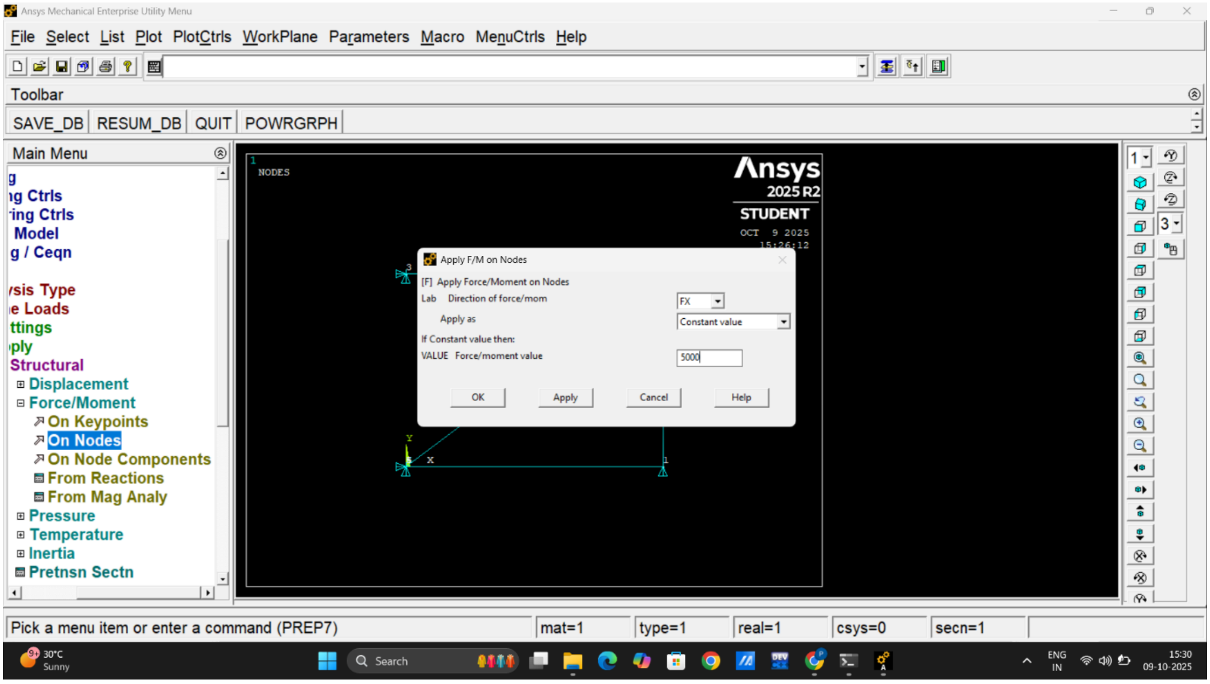
Step 13: Apply load>Structural>Displacement>Chose Node 3>Constraints>All Dof>ok



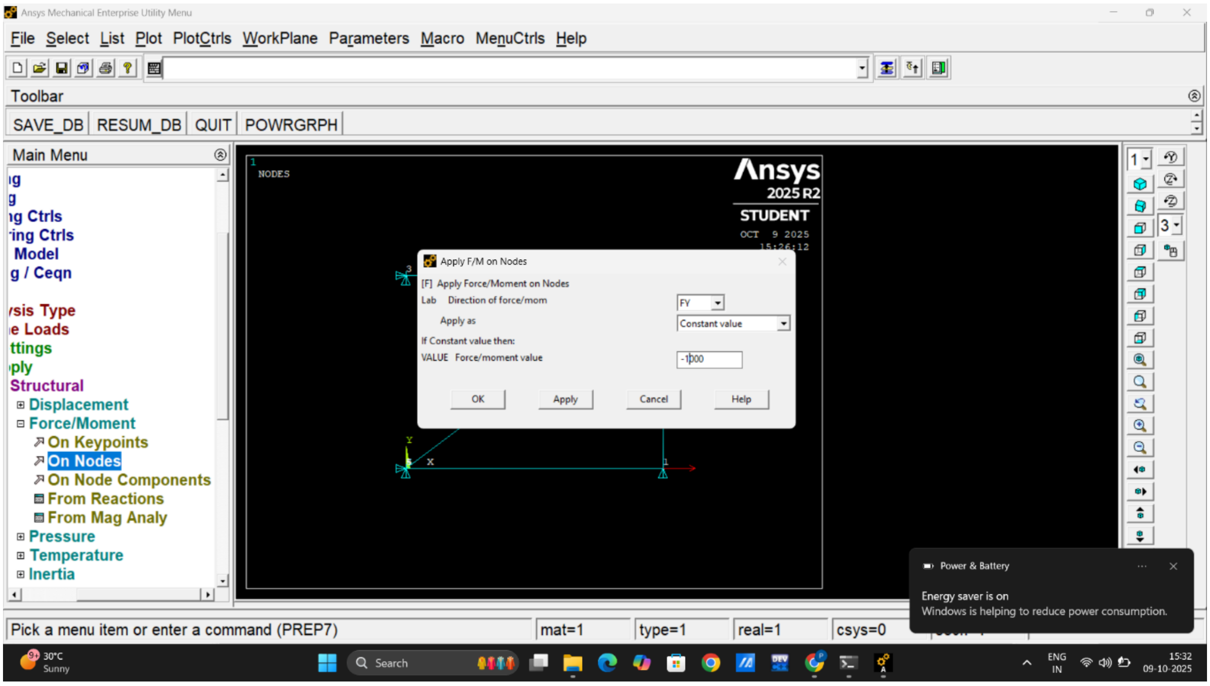
Step 13: Apply load>Structural>Displacement>Choose Node 3>Constraints>UY>ok



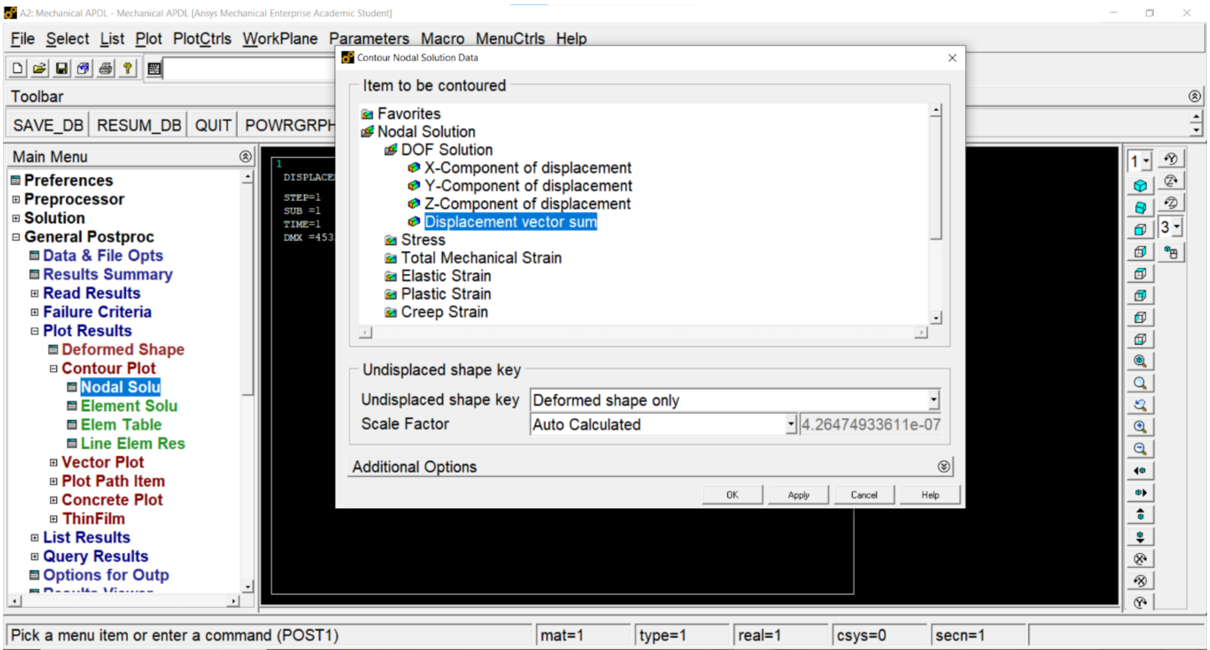
Step 14: Apply load>Structural>Force>Choose Node 2>Direction Fx>5000



Step 15: Apply load>Structural>Forces>Choose Node 1>Constraints>UY>ok

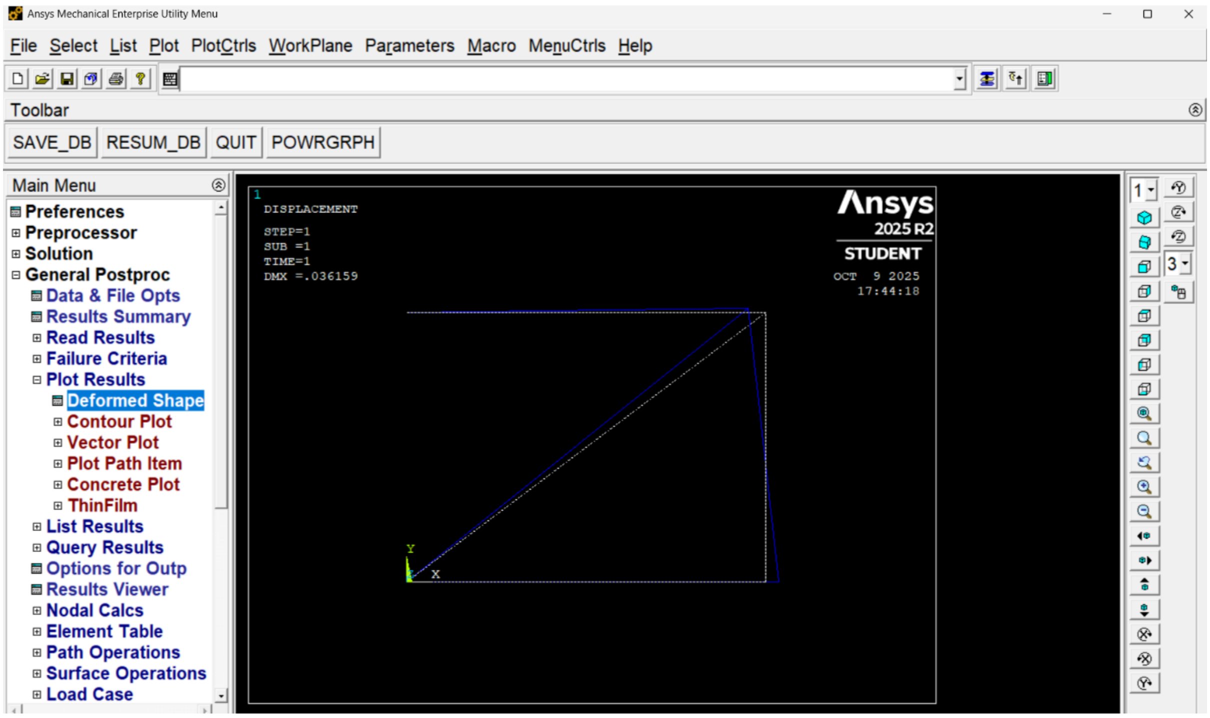


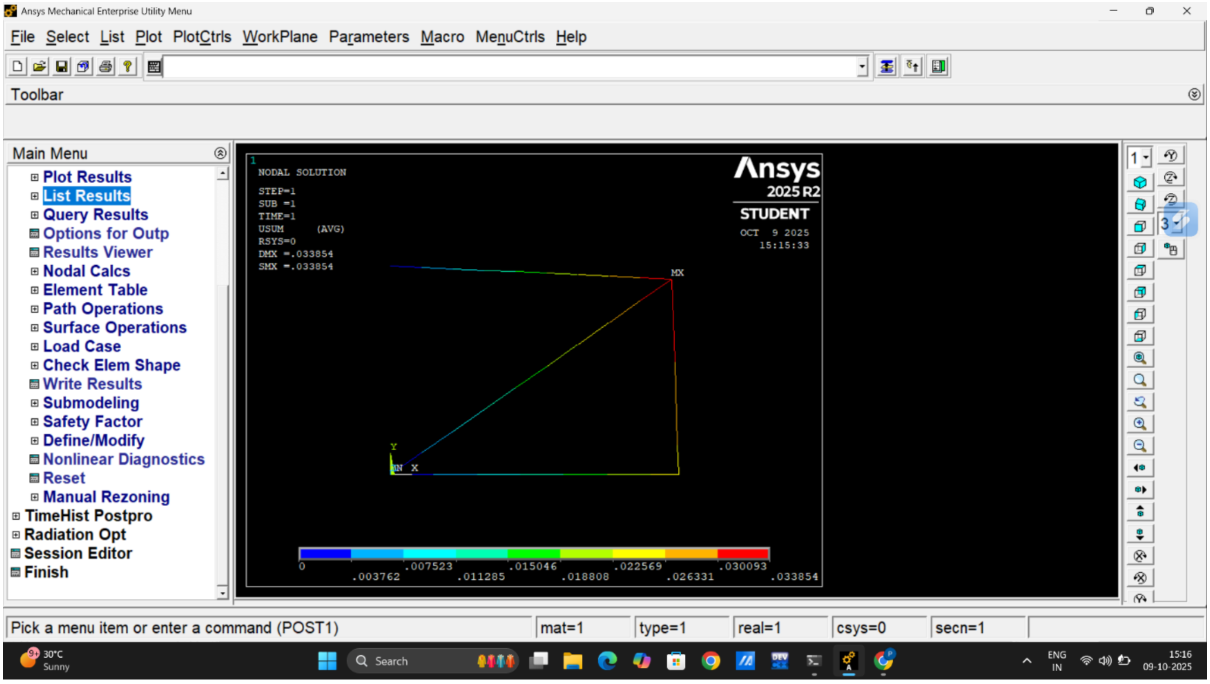
Step16:General Postprocess > Plot Results > Deformed Shaped > Def+undeformed>ok

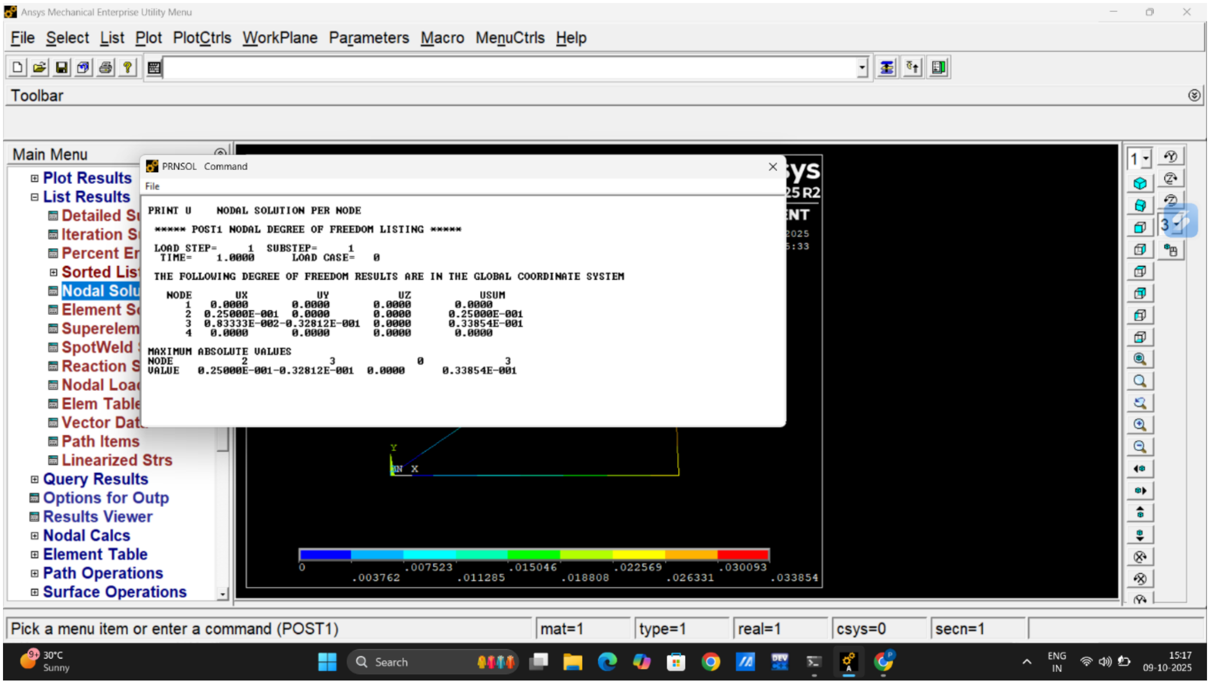


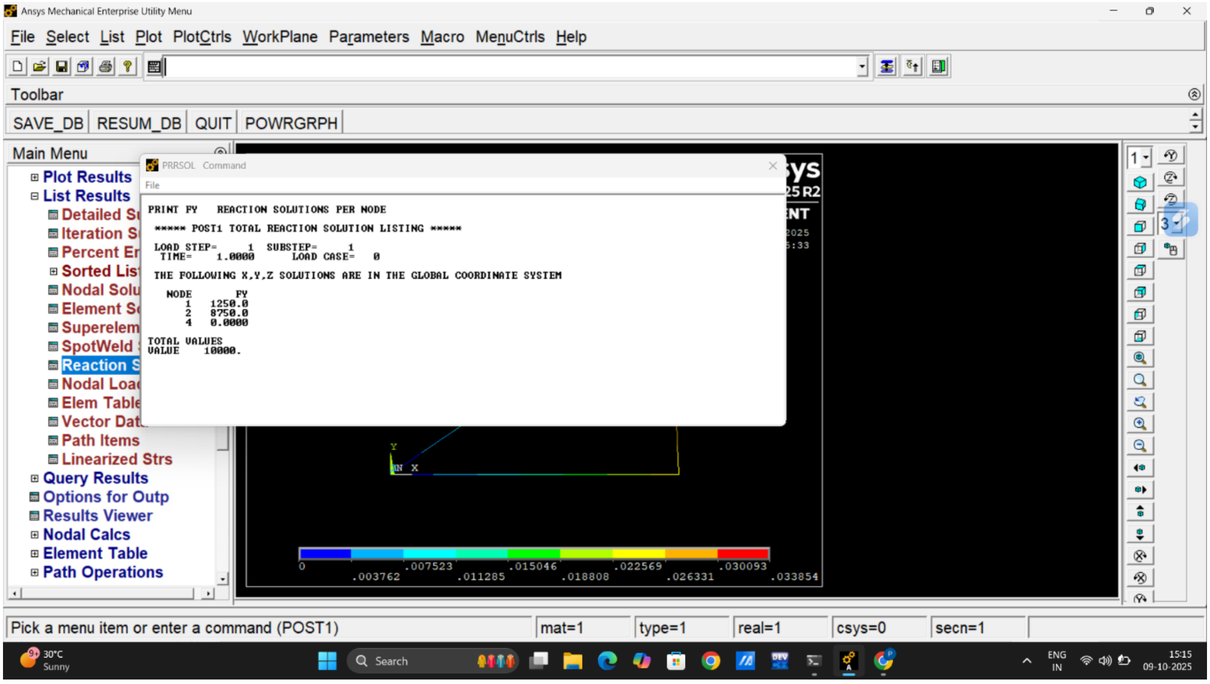
Step17:

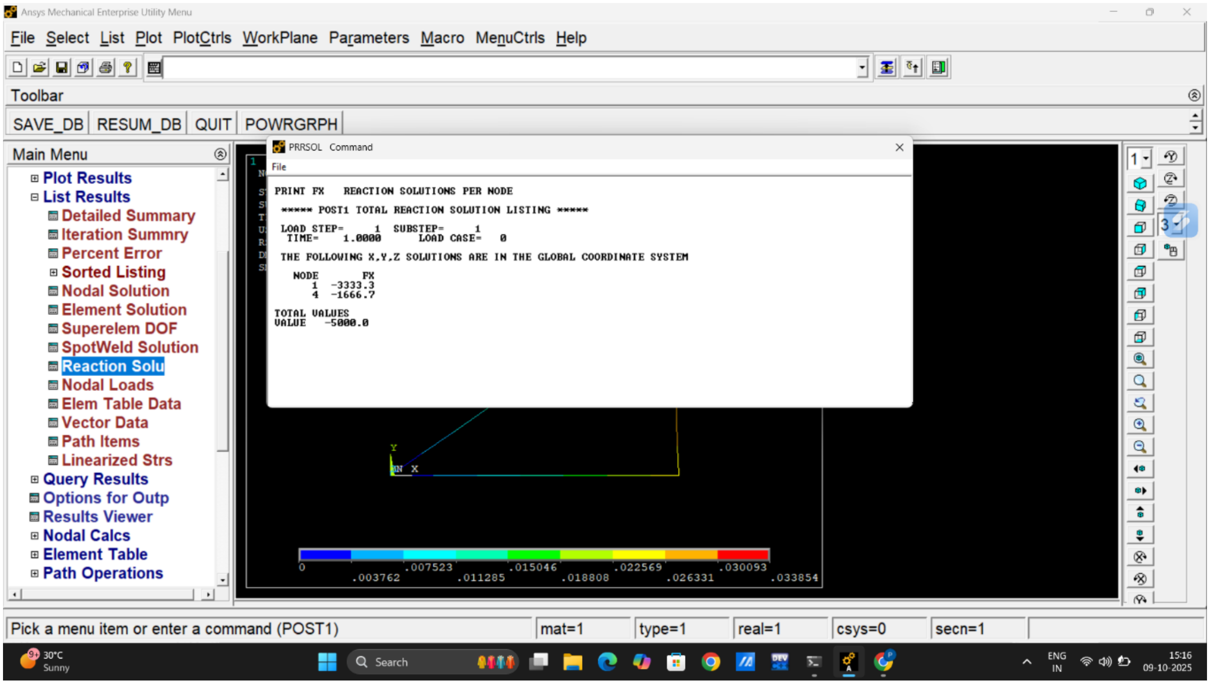
Results > Viewer >Nodal Solution> DOF Solution > Displacement > vector sum > Plot result.

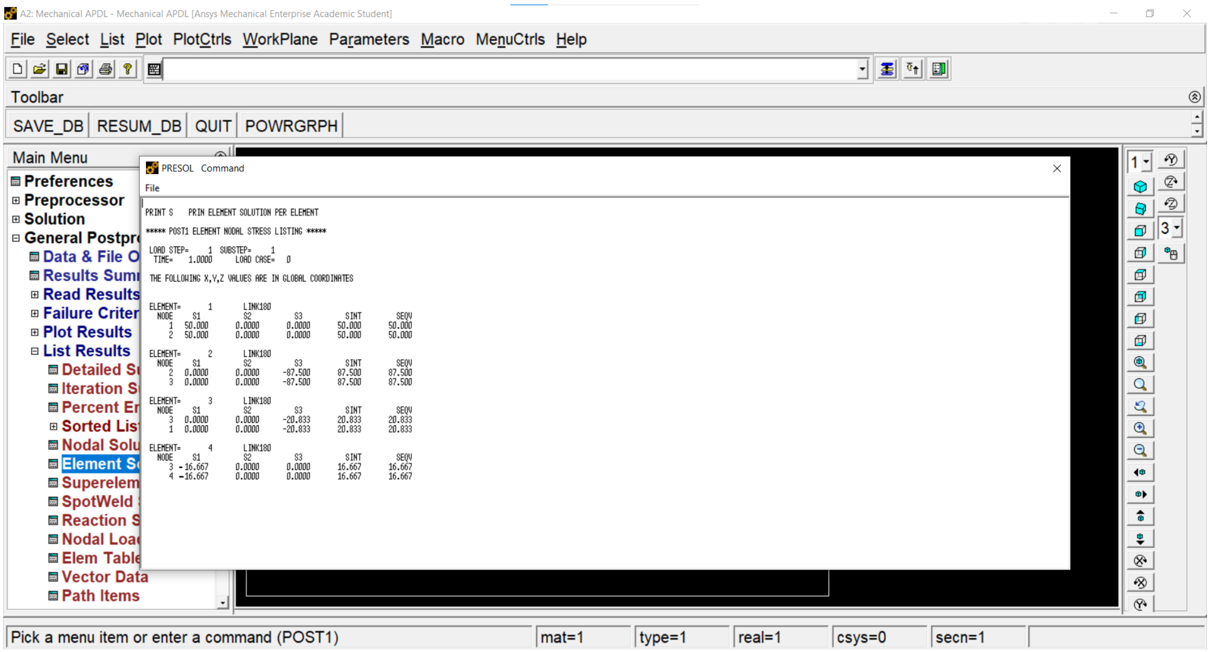



Ansys Mechanical Enterprise Utility Menu
File Select List Plot PlotCtrls WorkPlane Parameters Macro MenuCtrls Help
Toolbar
Main Menu
Plot Results
List Results
Query Results
Options for Outp
Results Viewer
Nodal Calcs
Element Table
Path Operations
Surface Operations
Load Case
Check Elem Shape
Write Results
Submodeling
Safety Factor
Define/Modify
Nonlinear Diagnostics
Reset
Manual Rezoning
TimeHist Postpro
Radiation Opt
Session Editor
Finish
NODAL SOLUTION
STEP=1
SUB =1
TIME=1
USUM (AVG)
RSYS=0
DMX =.033854
SMX =.033854
Ansys
2025 R2
STUDENT
OCT 9 2025
15:15:33
MX
MN
0 .003762 .007523 .011285 .015046 .018808 .022569 .026331 .030093 .033854
Pick a menu item or enter a command (POST1)
mat=1 type=1 real=1 csys=0 secn=1

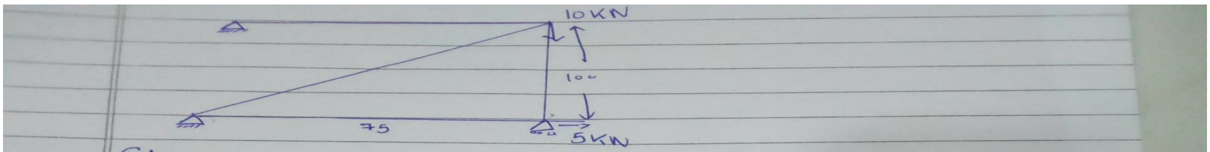

Ansys Mechanical Enterprise Utility Menu
File Select List Plot PlotCtrls WorkPlane Parameters Macro MenuCtrls Help
Toolbar
Main Menu
Plot Results
List Results
Nodal Solu
Linearized Strs
Query Results
Options for Outp
Results Viewer
Nodal Calcs
Element Table
Path Operations
Surface Operations
PRNSOL Command
File
PRINT U NODAL SOLUTION PER NODE
***** POST1 NODAL DEGREE OF FREEDOM LISTING *****
LOAD STEP= 1 SUBSTEP= 1
TIME= 1.0000 LOAD CASE= 0
THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN THE GLOBAL COORDINATE SYSTEM
NODE UX UY UZ USUM
1 0.0000 0.0000 0.0000 0.0000
2 0.25000E-001 0.0000 0.0000 0.25000E-001
3 0.83333E-002-0.32812E-001 0.0000 0.33854E-001
4 0.0000 0.0000 0.0000 0.0000
MAXIMUM ABSOLUTE VALUES
NODE 2 3 0 3
VALUE 0.25000E-001-0.32812E-001 0.0000 0.33854E-001
0 .003762 .007523 .011285 .015046 .018808 .022569 .026331 .030093 .033854
Pick a menu item or enter a command (POST1)
mat=1 type=1 real=1 csys=0 secn=1


Ansys Mechanical Enterprise Utility Menu
File Select List Plot PlotCtrls WorkPlane Parameters Macro MenuCtrls Help
Toolbar
SAVE_DB RESUM_DB QUIT POWRGRPH
Main Menu
PRRSOL Command
PRINT FY REACTION SOLUTIONS PER NODE
***** POST1 TOTAL REACTION SOLUTION LISTING *****
LOAD STEP= 1 SUBSTEP= 1
TIME= 1.0000 LOAD CASE= 0
THE FOLLOWING X,Y,Z SOLUTIONS ARE IN THE GLOBAL COORDINATE SYSTEM
NODE FY
1 1250.0
2 8750.0
4 0.0000
TOTAL VALUES
VALUE 10000.
Plot Results
List Results
Query Results
Options for Outp
Results Viewer
Nodal Calcs
Element Table
Path Operations
0 .003762 .007523 .011285 .015046 .018808 .022569 .026331 .030093 .033854
Pick a menu item or enter a command (POST1)
mat=1 type=1 real=1 csys=0 secn=1


Ansys Mechanical Enterprise Utility Menu
File Select List Plot PlotCtrls WorkPlane Parameters Macro MenuCtrls Help
Toolbar
SAVE_DB RESUM_DB QUIT POWRGRPH
Main Menu
Plot Results
List Results
Detailed Summary
Iteration Summry
Percent Error
Sorted Listing
Nodal Solution
Element Solution
Superelem DOF
SpotWeld Solution
Reaction Solu
Nodal Loads
Elem Table Data
Vector Data
Path Items
Linearized Strs
Query Results
Options for Outp
Results Viewer
Nodal Calcs
Element Table
Path Operations
PRRSOL Command
PRINT FX REACTION SOLUTIONS PER NODE
***** POST1 TOTAL REACTION SOLUTION LISTING *****
LOAD STEP= 1 SUBSTEP= 1
TIME= 1.0000 LOAD CASE= 0
THE FOLLOWING X,Y,Z SOLUTIONS ARE IN THE GLOBAL COORDINATE SYSTEM
NODE FX
1 -3333.3
4 -1666.7
TOTAL VALUES
VALUE -5000.0
0 .003762 .007523 .011285 .015046 .018808 .022569 .026331 .030093 .033854
Pick a menu item or enter a command (POST1)
mat=1 type=1 real=1 csys=0 secn=1



Analytical Solution



Given

A = 100 $\qquad$ $E = 2\times10^5 N/mm^2$ $\qquad$ $EA = 20\times10^6 N$

External Forces = 5KN → $\qquad$ F_{3y} = -10KN ↓

Member	L	$\lambda_x = \cos\theta$	$\lambda_y = \sin\theta$	$C = \frac{EA}{L}$
1	100	1.0	0	$\frac{20\times10^6}{100}$ = 200,000
2	75	0	1.0	$\frac{20\times10^6}{75}$ = 266,666.7
3	100	-1	0	$\frac{20\times10^6}{100}$ 200,000
4	125	0.8	0.6	$\frac{20\times10^6}{125}$ = 160000

$$k^4 = 160{,}000 \begin{array}{cccc} 1,1 & 1,2 & 1,5 & 1,6 \end{array}$$
$$\begin{bmatrix} 0.64 & 0.48 & -0.64 & -0.48 \\ 0.48 & 0.36 & -0.48 & -0.36 \\ -0.64 & -0.48 & 0.64 & 0.48 \\ -0.48 & -0.36 & 0.48 & 0.36 \end{bmatrix}$$

Boundary conditions.

Joint 1 = 0 $\quad v_1 = 0$

Joint 4 = 0 $\quad v_4 = 0$

Joint 2 $\quad v_2 = 0$

$$F_x = \{F_{2x}, F_{3x}; F_{3y}\}^T = \{5\,kN, -10\,kN\}^T$$

$$F = \{5000, 0, -10000\}^T$$

$$k =$$

$$K = \begin{bmatrix} 200{,}000 & 0 & 0 \\ 0 & 302{,}400 & 76{,}800 \\ 0 & 76{,}800 & 324{,}266.7 \end{bmatrix}$$

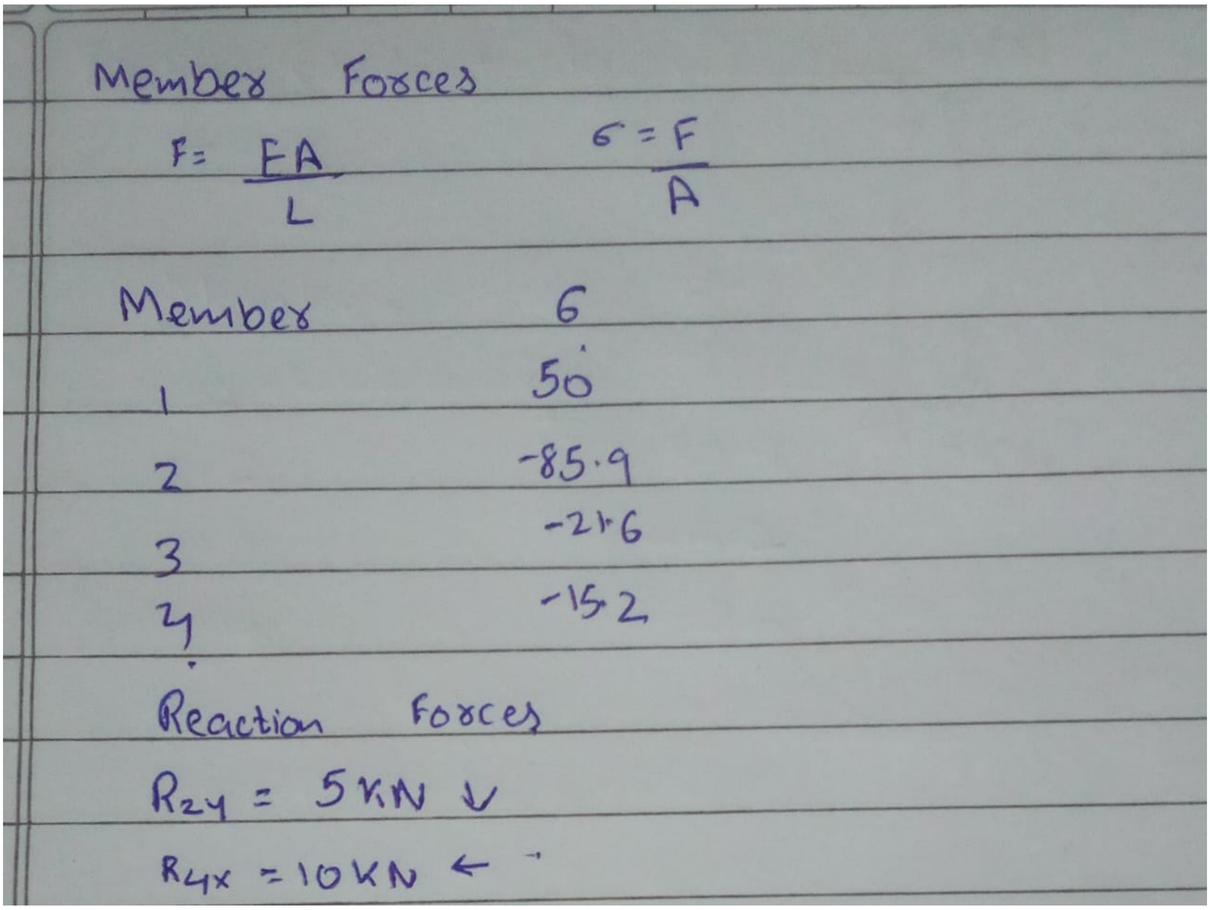
$$\begin{bmatrix} 200{,}000 & 0 & 0 \\ 0 & 302{,}400 & 76{,}800 \\ 0 & 76{,}800 & 324{,}266.7 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 5000 \\ 0 \\ -10000 \end{Bmatrix}$$

$$u_2 = \frac{5000}{200{,}000} = 0.025$$

$$\begin{bmatrix} 302400 & 76800 \\ 76400 & 324266.7 \end{bmatrix} \begin{Bmatrix} u_3 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ -10000 \end{Bmatrix}$$

$u_3 = 0.00762$ mm $\qquad u_2 = 0.0250$

$v_3 = -0.0322$ mm



Member Forces

$F = \frac{EA}{L}$ $\quad\sigma = \frac{F}{A}$

Member	σ
1	50
2	-85.9
3	-21.6
4	-15.2

Reaction Forces

$R_{2y} = 5\,kN \downarrow$

$R_{4x} = 10\,KN \leftarrow$

Member	Axial Forces	Stress	Status
1-2	5.00	50.0	**Tension (T)**
2-3	−8.59	−85.9	**Tension (T)**
3-4	−1.52	−15.2	**Compression (C)**
1-3	−2.16	−21.6	**Compression (C)**

Reaction Forces

Reaction Component	Reaction Force
Fx	-5000
Fy	-10000

Comparison Between ANSYS and Analytical Stress

Member	ANSYS Stress MPa	Analytical Stress (σAnalytical)(MPa)	Status	Percentage Error
1-2	50.00	50.00	Tension (T)	0.0%
2-3	−86.67	−85.90	Compression (C)	0.9%
1-3	−20.83	−21.60	Compression (C)	5.9%
3-4	−16.67	−15.20	Compression (C)	9.7%

Reaction Forces

Reaction Component	Ansys Force	Analytical force	Percentage Error
Fx	-5000 (Downward)	-5000 (Downward)	0.0
Fy	-10000	-10000	0.0

Modern Education Society's
Wadia College Of Engineering,
Pune-01

DEPARTMENT OF AUTOMATION AND ROBOTICS ENGINEERING

COMPUTER AIDED ENGINEERING & MANUFACTURING (302523)

Academic Year 2025-2026

TITLE	PLATE/SHELL ELEMENT – STRUCTURAL LINEAR AND NON-LINEAR ANALYSIS		
NAME OF STUDENT			
CLASS AND DIVISION		BATCH	
SEMESTER		ROLL NO	
DATE OF PERFORMANCE		DATE OF SUBMISSION	
EXAMINED BY			

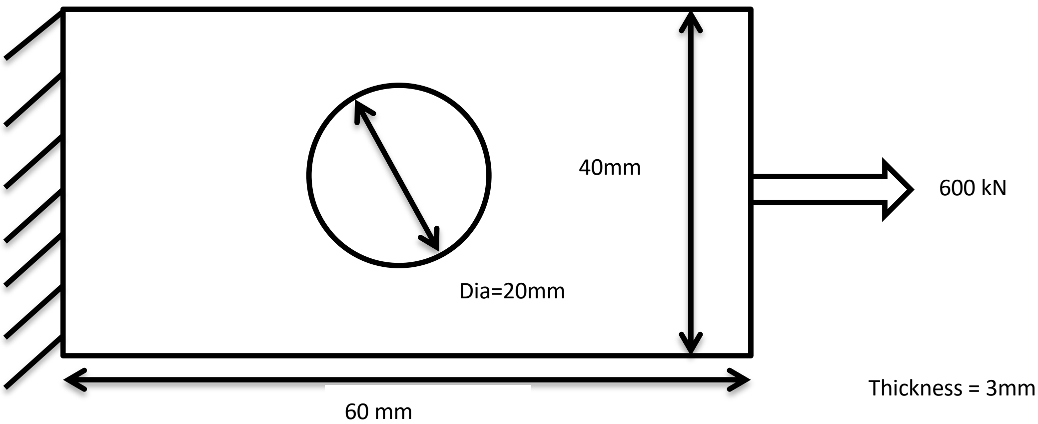
COURSE OUTCOMES:

- **CO1: DEFINE** the use of CAE tools and DESCRIBE the significance of shape functions infinite element formulations.
- **CO2:APPLY** material properties and boundary condition to SOLVE 1-D and 2-D element stiffness matrices to obtain nodal or elemental solution.
- **CO3:ANALYZE** and APPLY various numerical methods for different types of analysis.
- **CO4:CREATE** process plan and GENERATE GandM code using CAM software tools.
- **CO5:UNDRSTAND** lean manufacturing tools and techniques
- **CO6:APPLY** knowledge to do process planning and ESTIAMTE costing for the same

Name Of Student :	Class :
Semester/Year :	Roll No. :
Date Of Performance :	Date Of Submission :
Examined By : Prof. R. N. Yerrawar	Experiment No : 3

PROBLEM STATEMENT :

Find a stress distribution in the plate 60mm x 40mm with a central hole of Diameter 10mm. The thickness of the plate is 3mm



Theory:

Shell elements are used to create a mathematical 2D idealization of a 3D structure. They offer computationally efficient solutions for modeling shell structures when compared to solid elements.

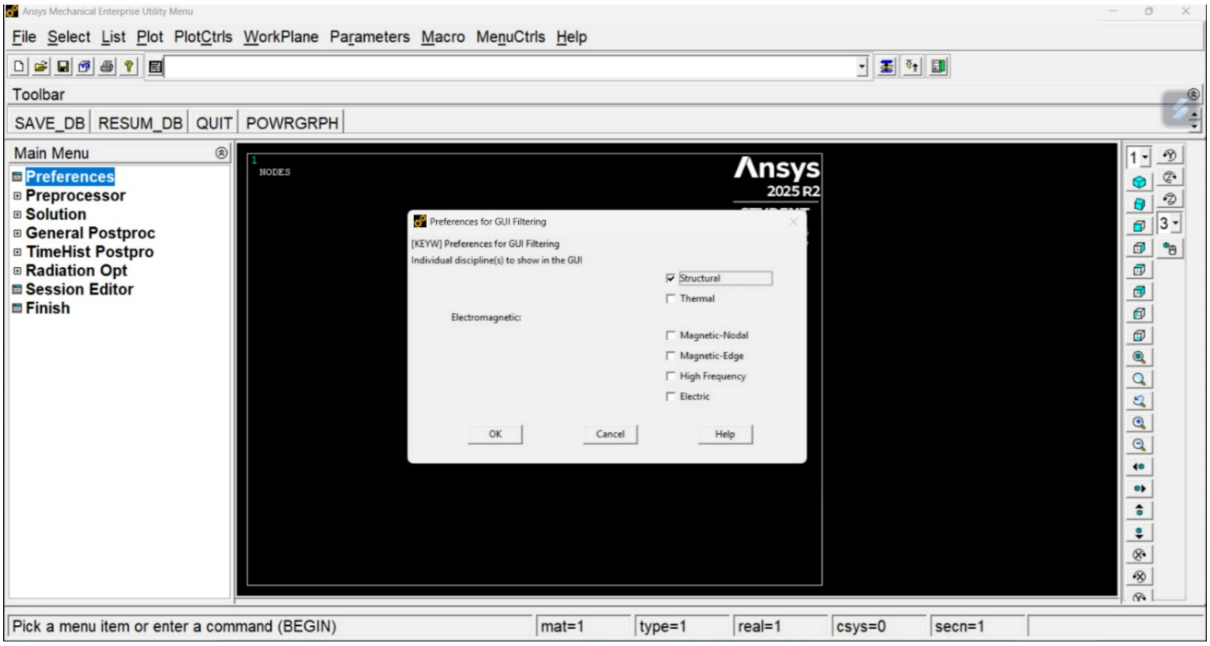
The discussion in this chapter applies to 3D finite-strain shell elements such as SHELL181 and SHELL281. Compared to other shells, these shell elements provide more robust nonlinear analysis capabilities, and significant improvements in cross-section data definition, analysis, and visualization.

The method for defining shell sections described here can also be used to define the cross-sectional properties of the layered thermal shell elements SHELL131 and SHELL132. However, information presented here concerning integration points (NUMPT on the SECDATA command) and section properties (SECCONTROL command) does not apply to SHELL131 and SHELL132.

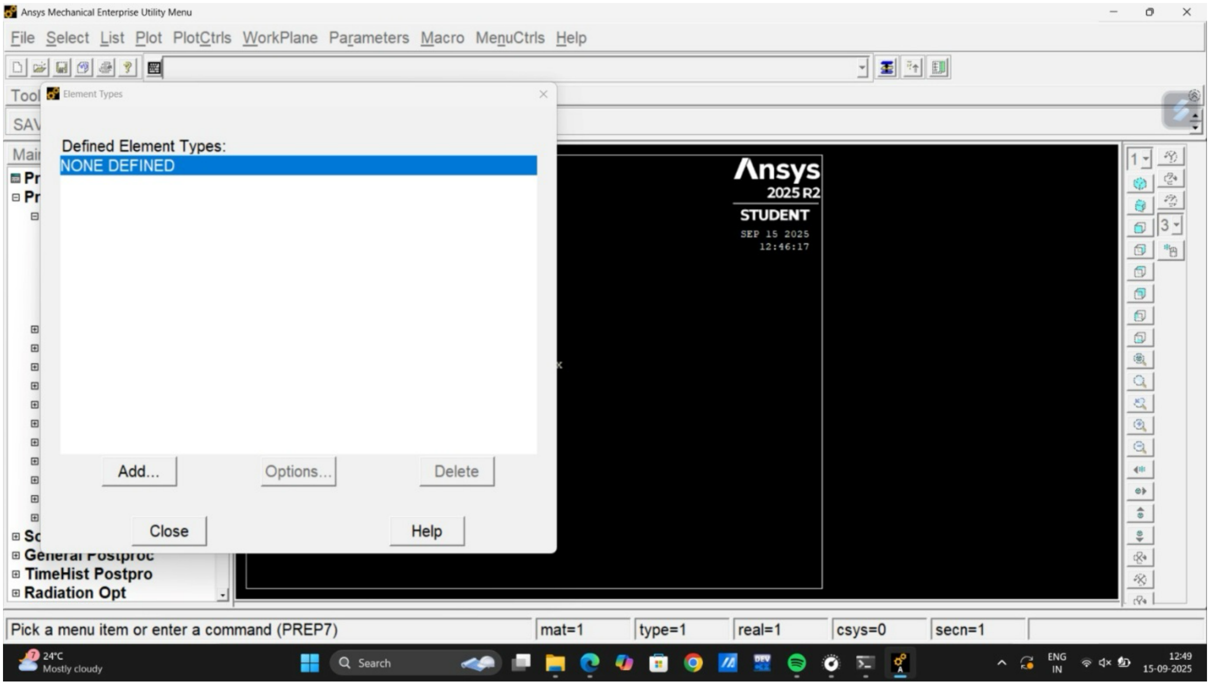
1. The general procedure for creating a cross section consists of the following steps: Define the section and associate a section ID number with the section.
2. Define the geometry data for the section.

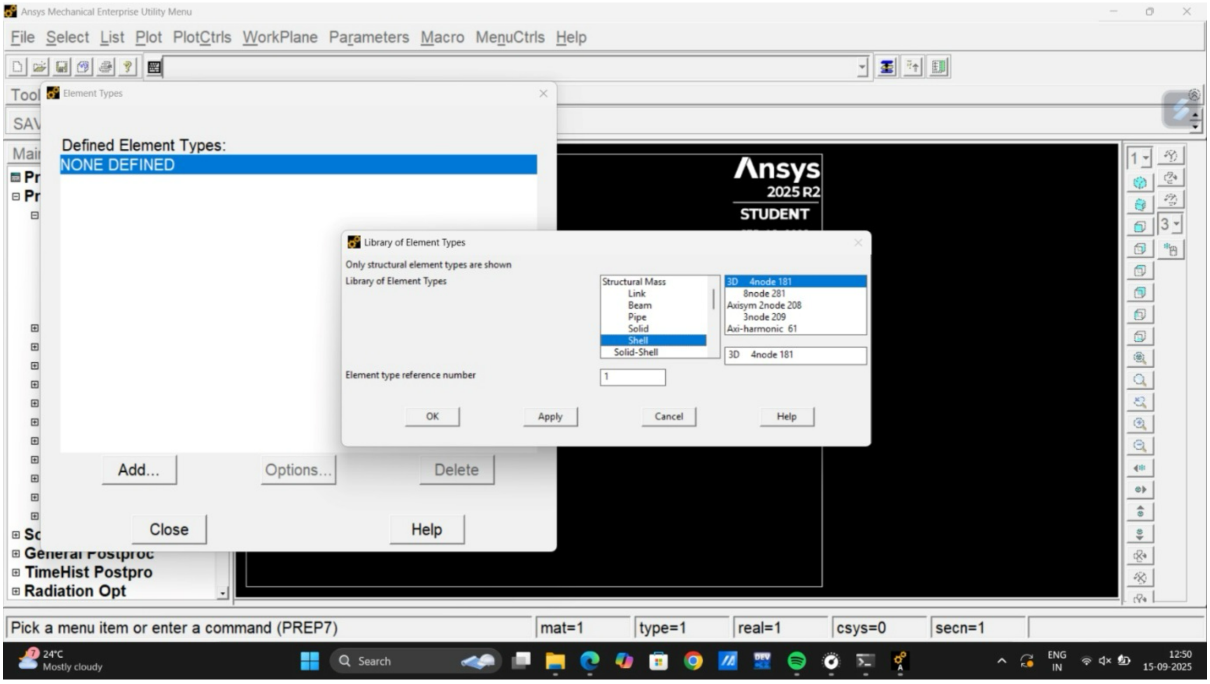
Open the mechanical APDL 2025 R1

Step 1 : Preferences < Structure < Ok

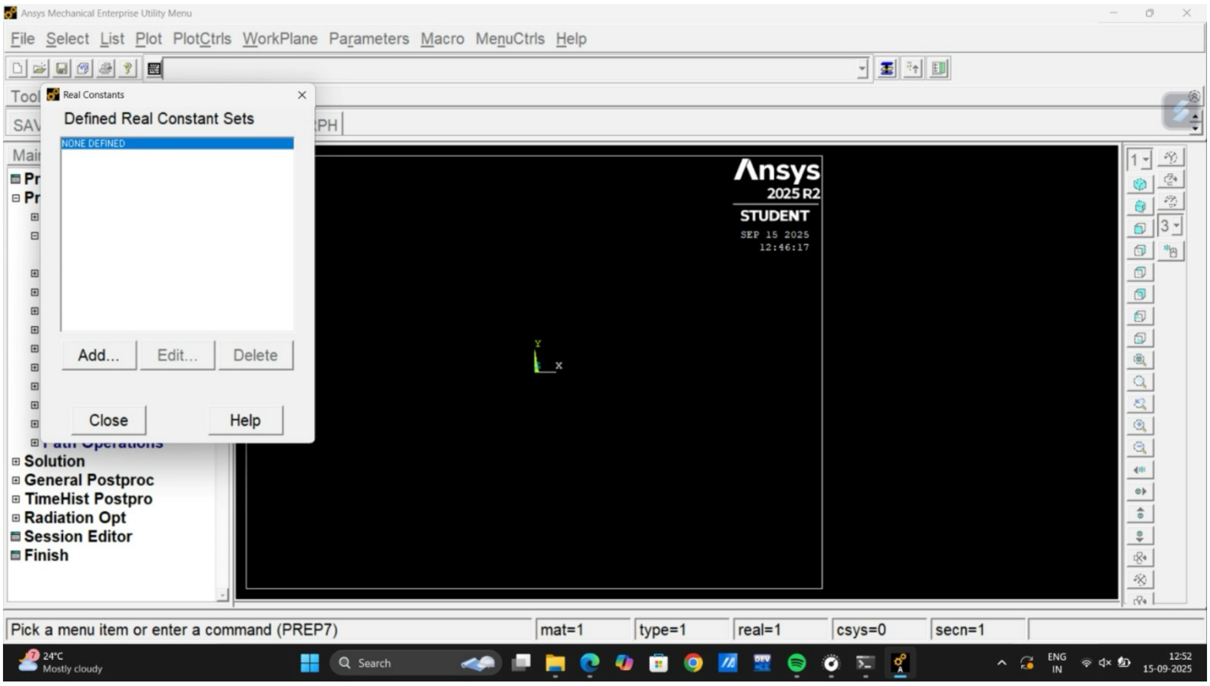


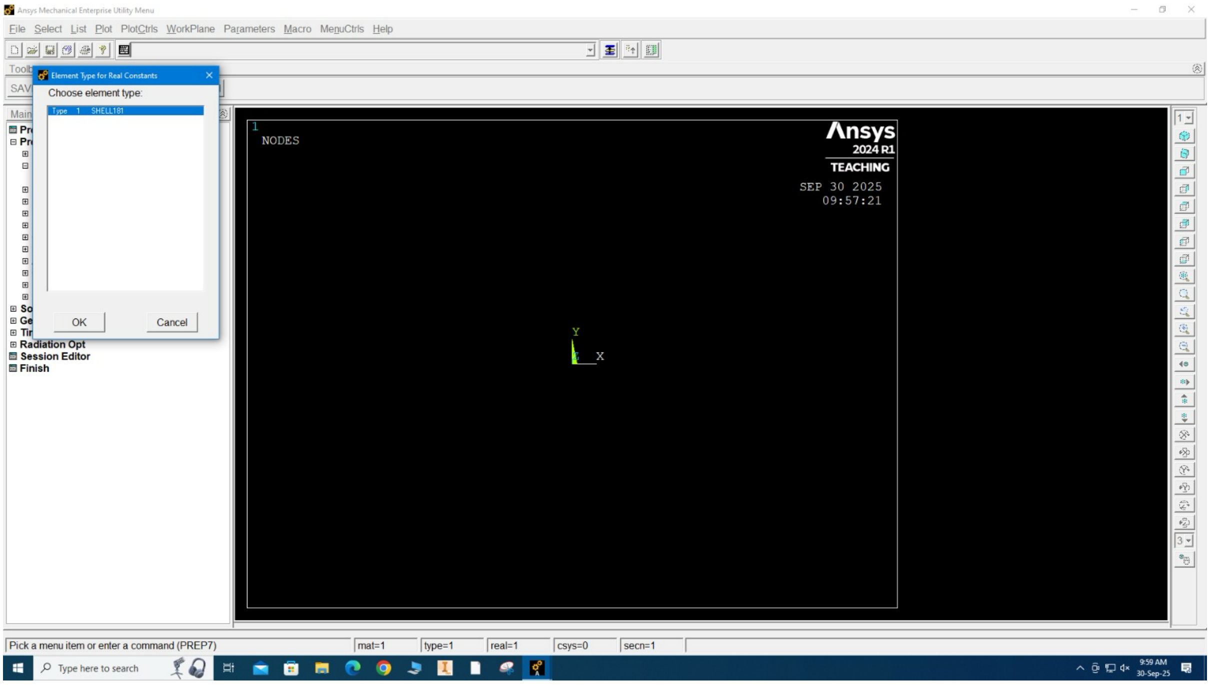
Step 2 : Preprocessor < Element Type < Add Element < Shell < 3D 4Node 181 < Ok



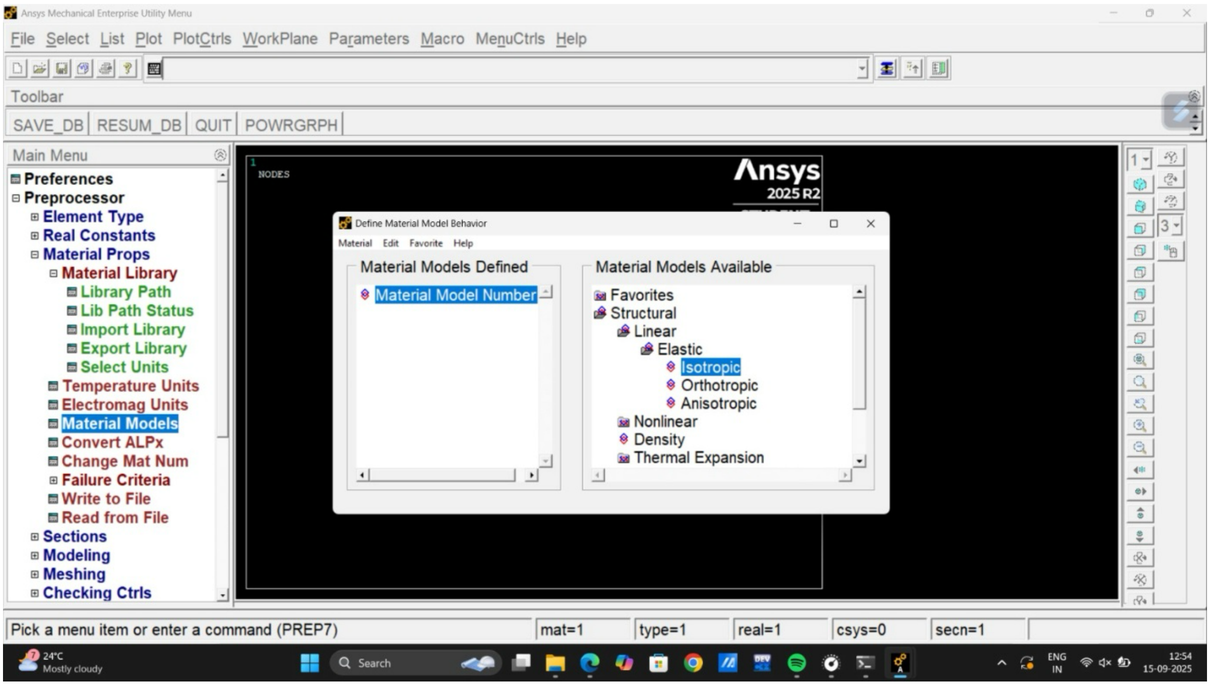


Step 3 : Preprocessor < Real Constants < Add < Ok

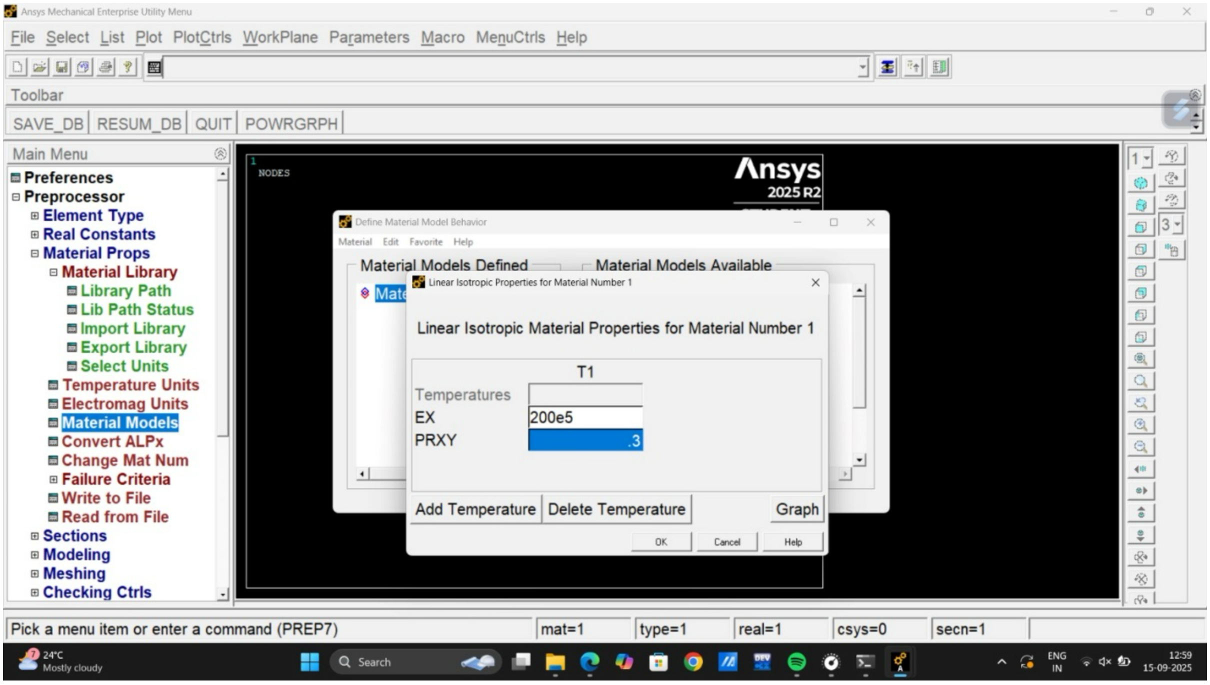




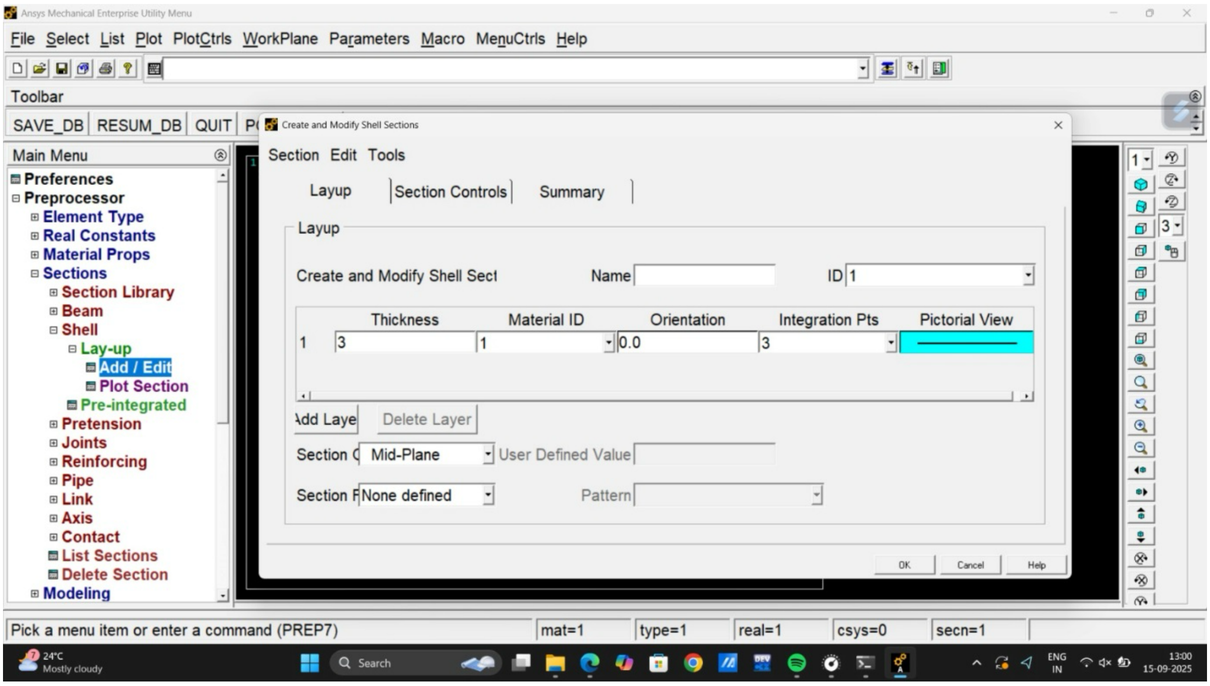
Step 4 : Preprocessor < Material Models < Structural < Linear < Elastic < Isotropic



Step 5 : Enter EX as 200e5 and PRXY as 0.3

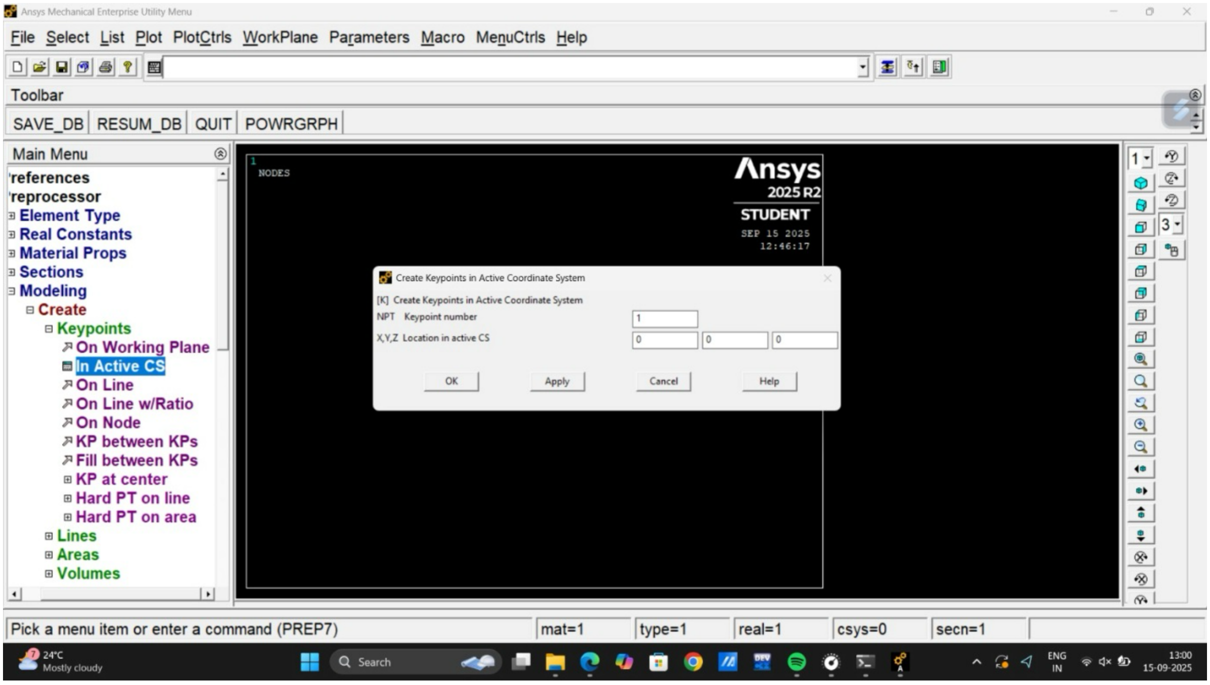


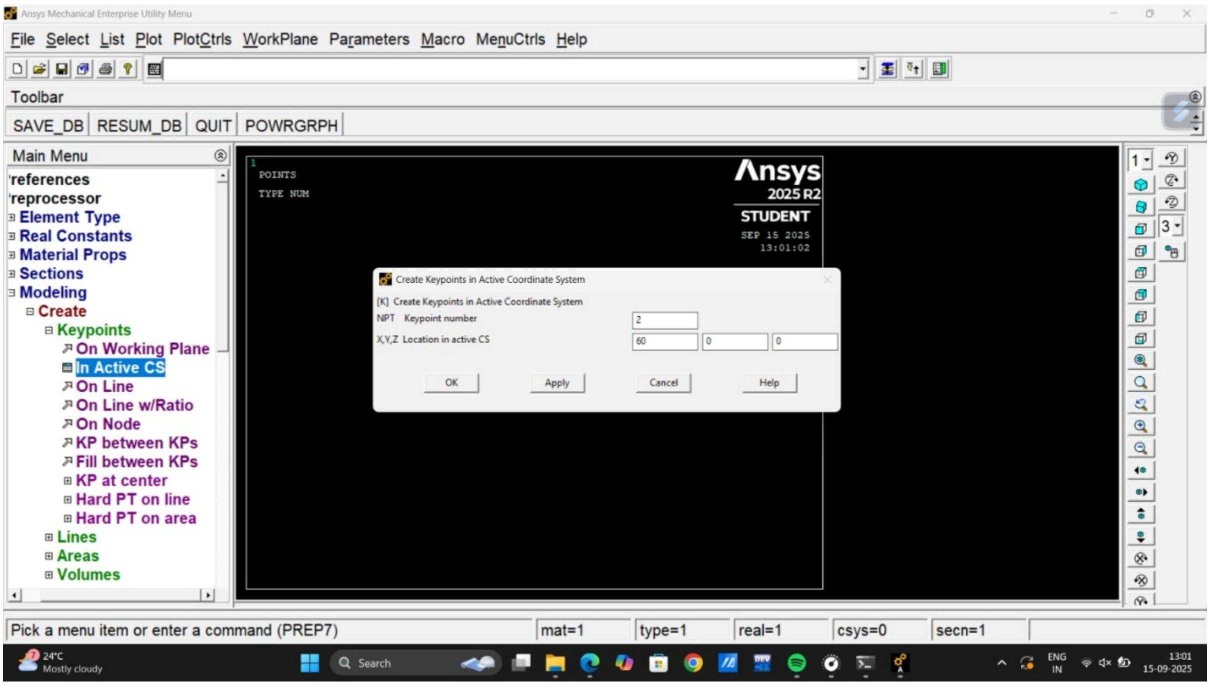
Step 6 : Preprocessor < Sections < Shell < Lay-up < Add < Thickness (Given) < Ok

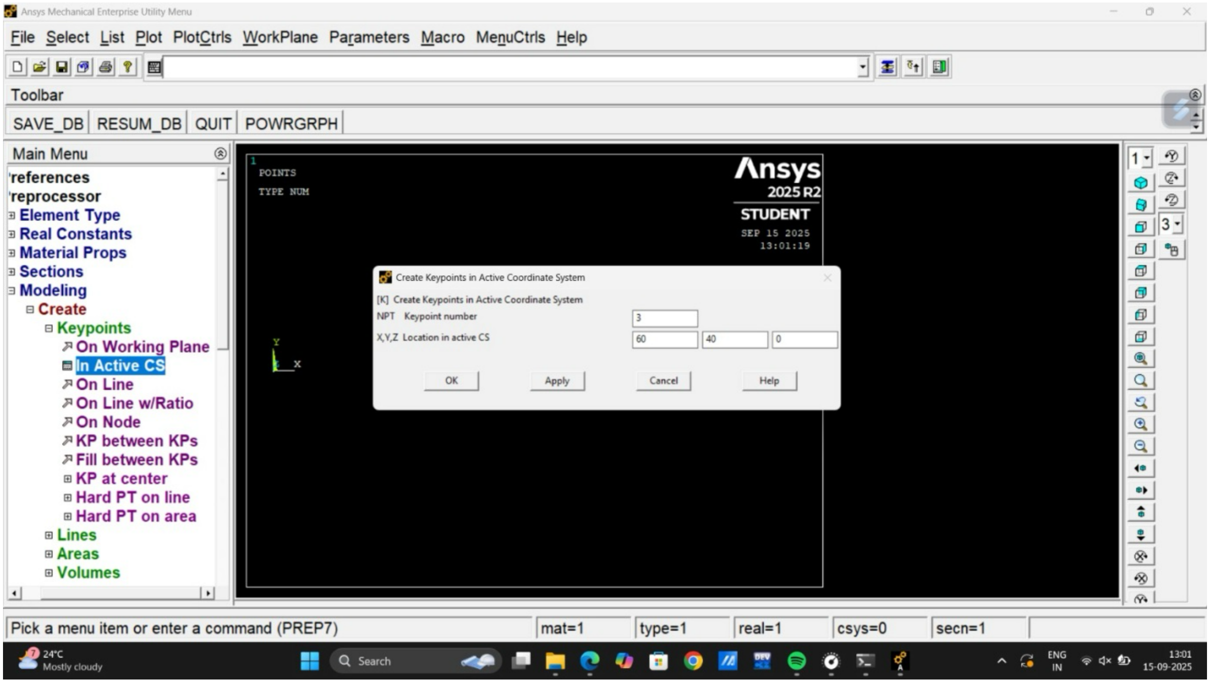


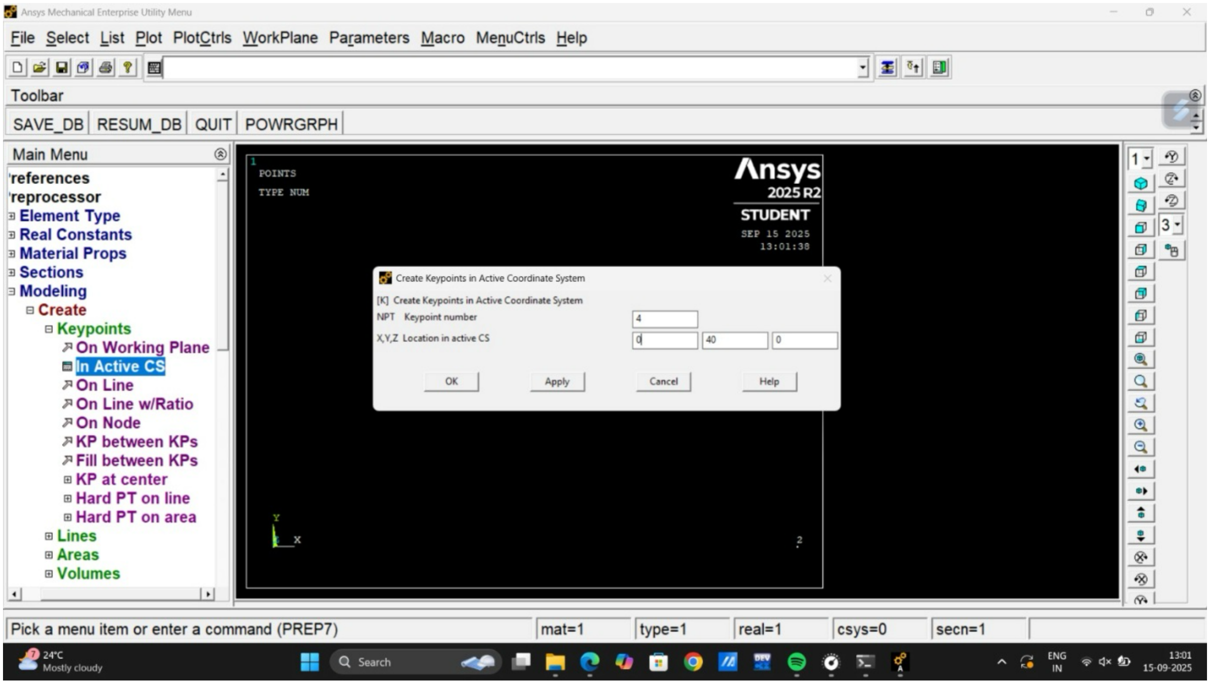
Step 7 : Preprocessor < Modelling < Create < Keypoints < In Active Cs < Apply < Ok

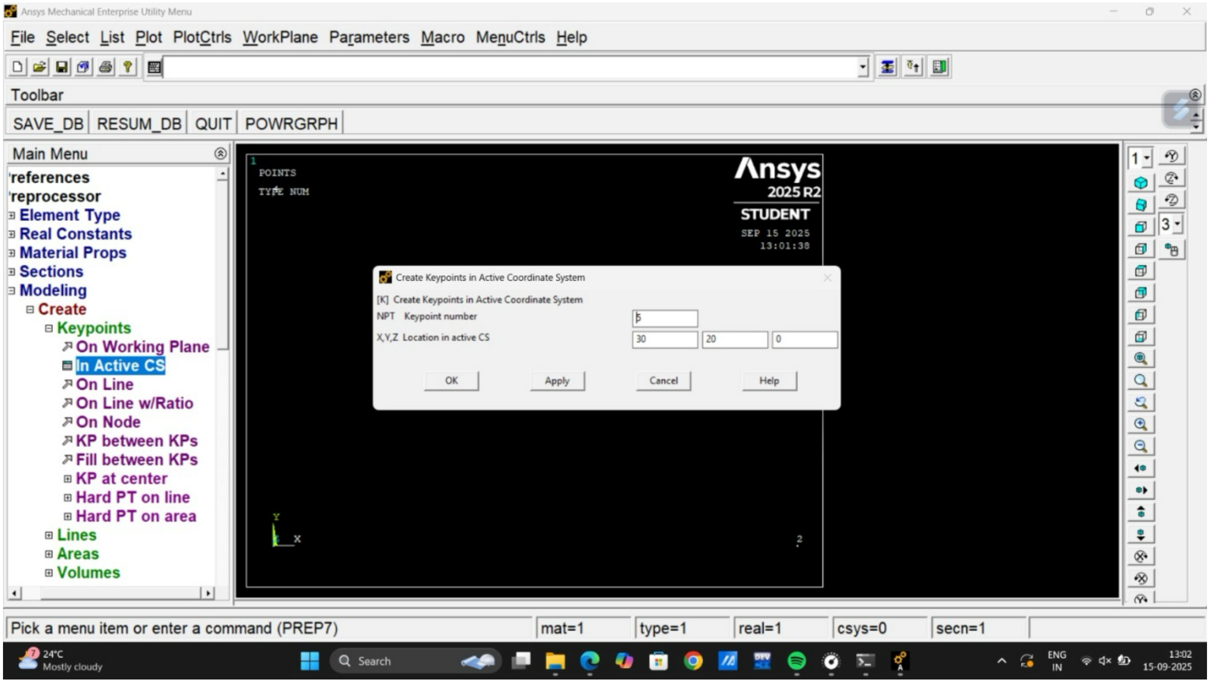
Enter keypoint number and x,y,z location in active CS as given





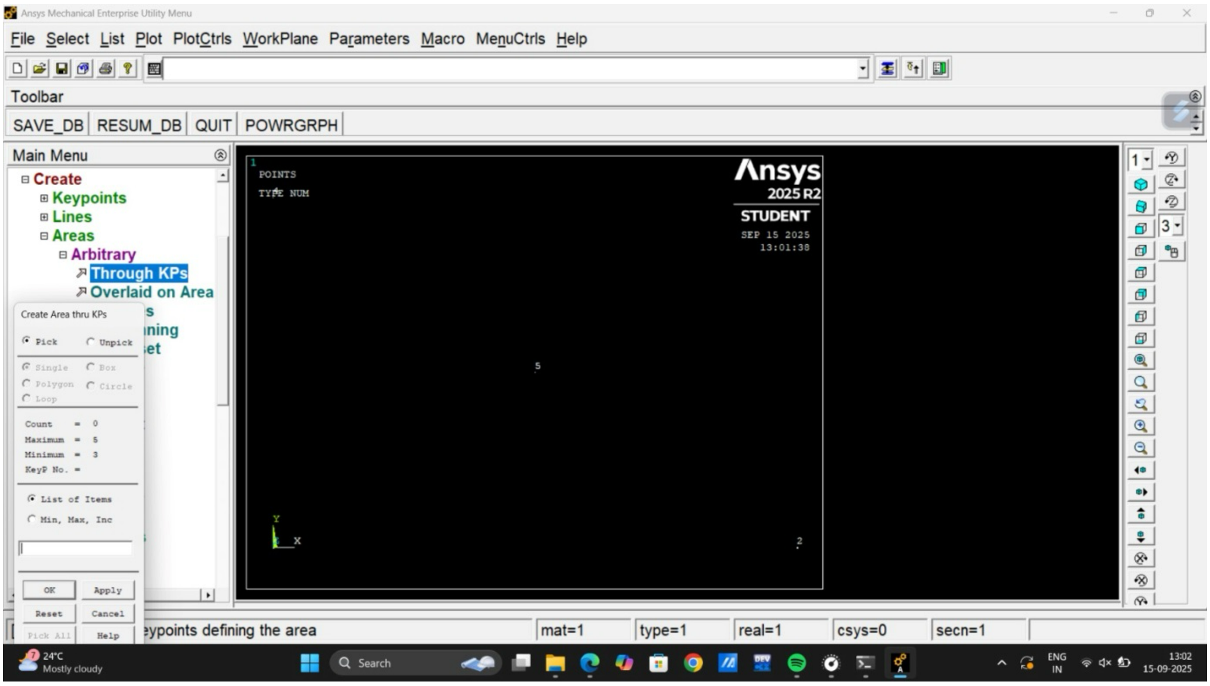

Ansys Mechanical Enterprise Utility Menu
File Select List Plot PlotCtrls WorkPlane Parameters Macro MenuCtrls Help
Toolbar
SAVE_DB RESUM_DB QUIT POWRGRPH
Main Menu
'references
'reprocessor
Element Type
Real Constants
Material Props
Sections
Modeling
Create
Keypoints
On Working Plane
In Active CS
On Line
On Line w/Ratio
On Node
KP between KPs
Fill between KPs
KP at center
Hard PT on line
Hard PT on area
Lines
Areas
Volumes
POINTS
TYPE NUM
Ansys
2025 R2
STUDENT
SEP 15 2025
13:01:19
Create Keypoints in Active Coordinate System
[K] Create Keypoints in Active Coordinate System
NPT Keypoint number
3
X,Y,Z Location in active CS
60
40
0
OK
Apply
Cancel
Help
Pick a menu item or enter a command (PREP7)
mat=1
type=1
real=1
csys=0
secn=1
24°C
Mostly cloudy
Search
ENG
IN
13:01
15-09-2025

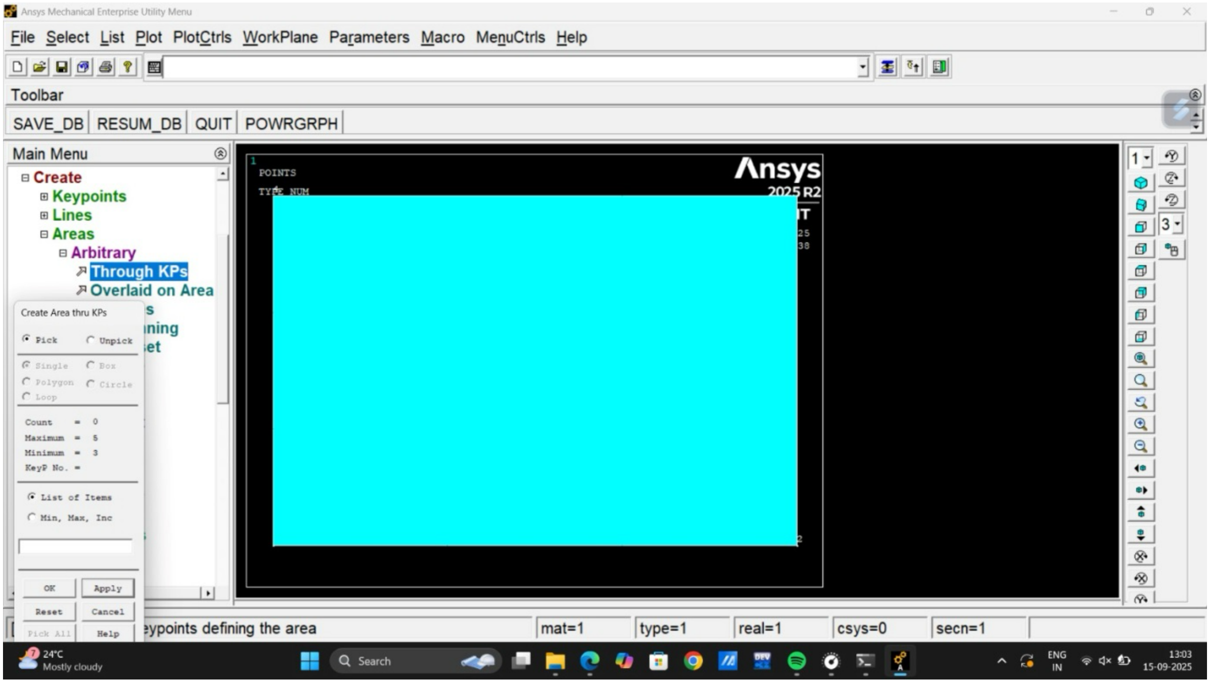

Ansys Mechanical Enterprise Utility Menu
File Select List Plot PlotCtrls WorkPlane Parameters Macro MenuCtrls Help
Toolbar
SAVE_DB RESUM_DB QUIT POWRGRPH
Main Menu
'references
'reprocessor
Element Type
Real Constants
Material Props
Sections
Modeling
Create
Keypoints
On Working Plane
In Active CS
On Line
On Line w/Ratio
On Node
KP between KPs
Fill between KPs
KP at center
Hard PT on line
Hard PT on area
Lines
Areas
Volumes
POINTS
TYPE NUM
Ansys
2025 R2
STUDENT
SEP 15 2025
13:01:38
Create Keypoints in Active Coordinate System
[K] Create Keypoints in Active Coordinate System
NPT Keypoint number
4
X,Y,Z Location in active CS
0
40
0
OK
Apply
Cancel
Help
Pick a menu item or enter a command (PREP7)
mat=1
type=1
real=1
csys=0
secn=1
24°C
Mostly cloudy
Search
ENG
IN
13:01
15-09-2025



Step 8: Preprocessor < Modelling < Create < Areas < Arbitrary < Through KPS

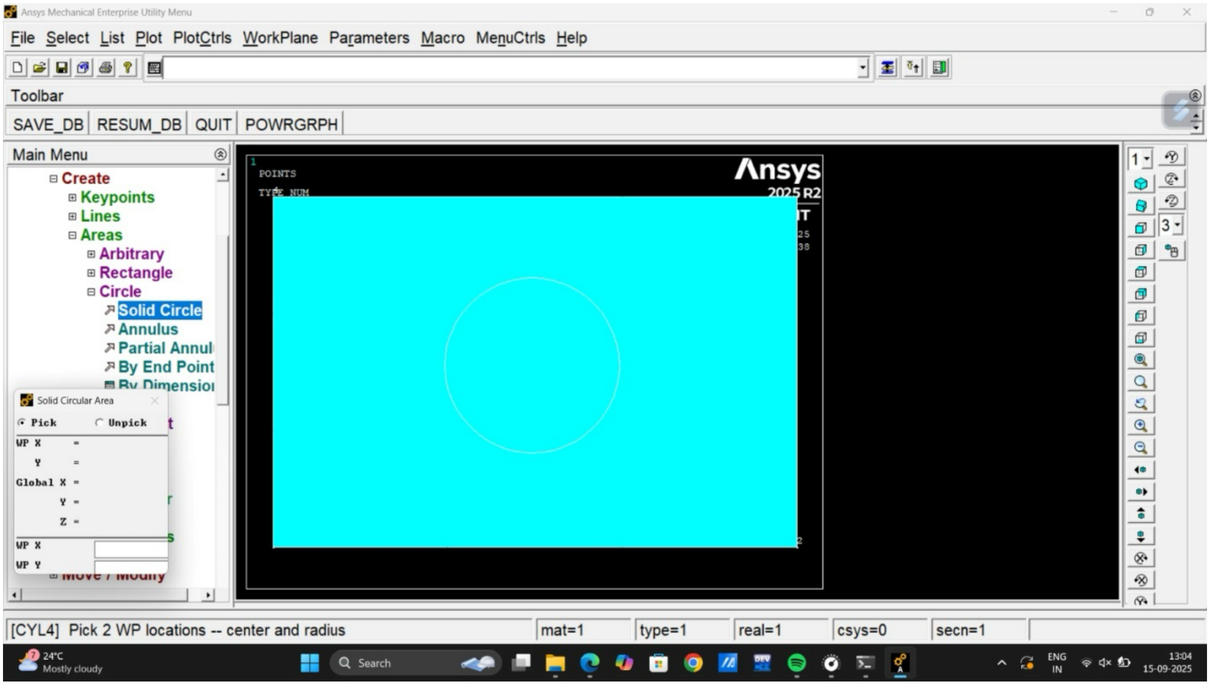
Select the point to be jointed and then apply and press ok



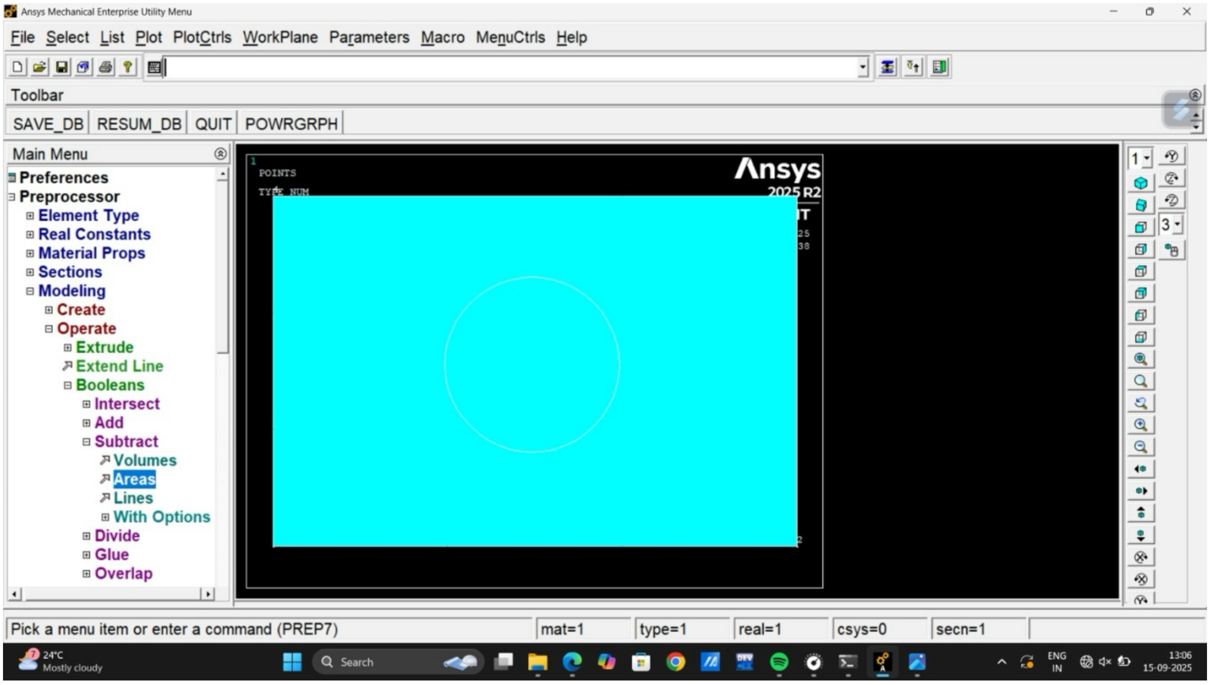


Step 9: Preprocessor < Modelling < Create < Areas < Circle < Solid Circle

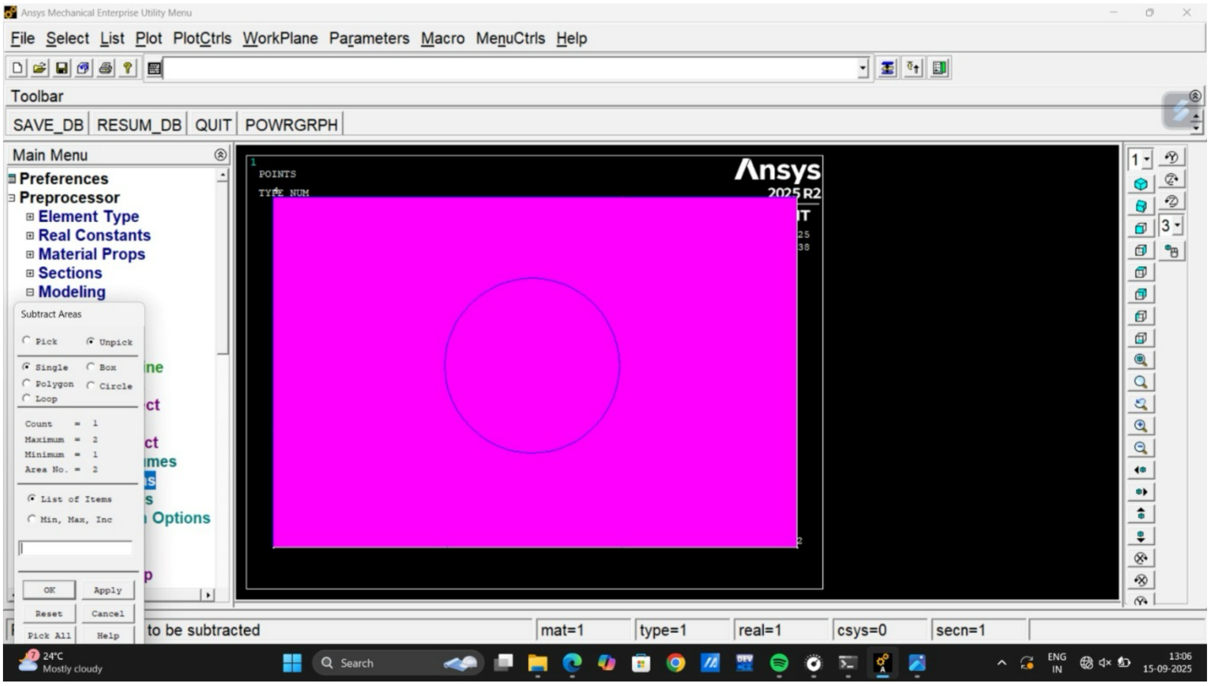
Select the point and radius of circle as given



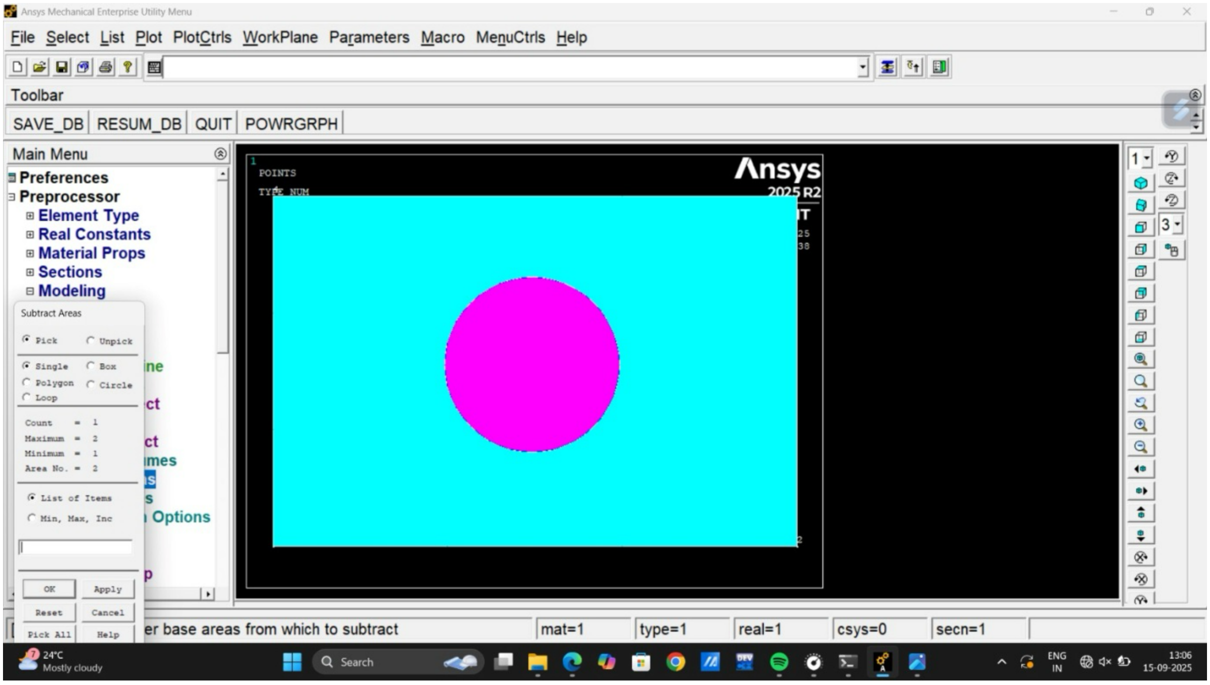
Step 10 : Preprocessor < Modeling < Operate < Booleans < Subtract < Areas



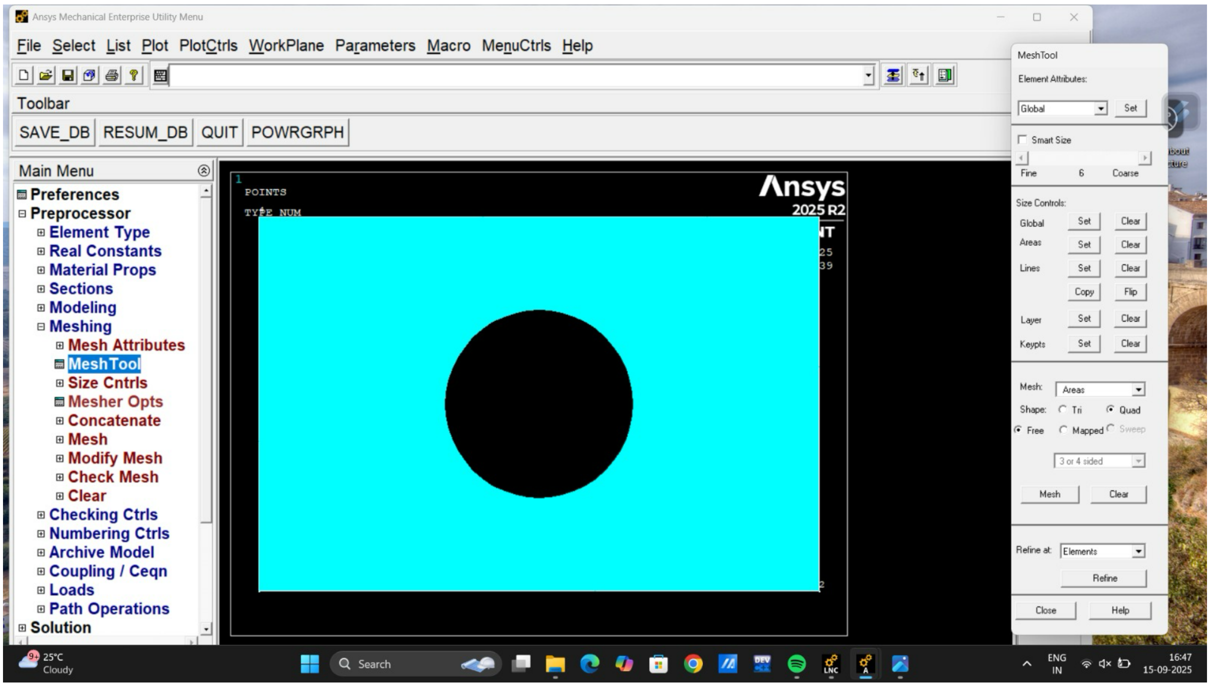
Step 11: Firstly, select the whole area

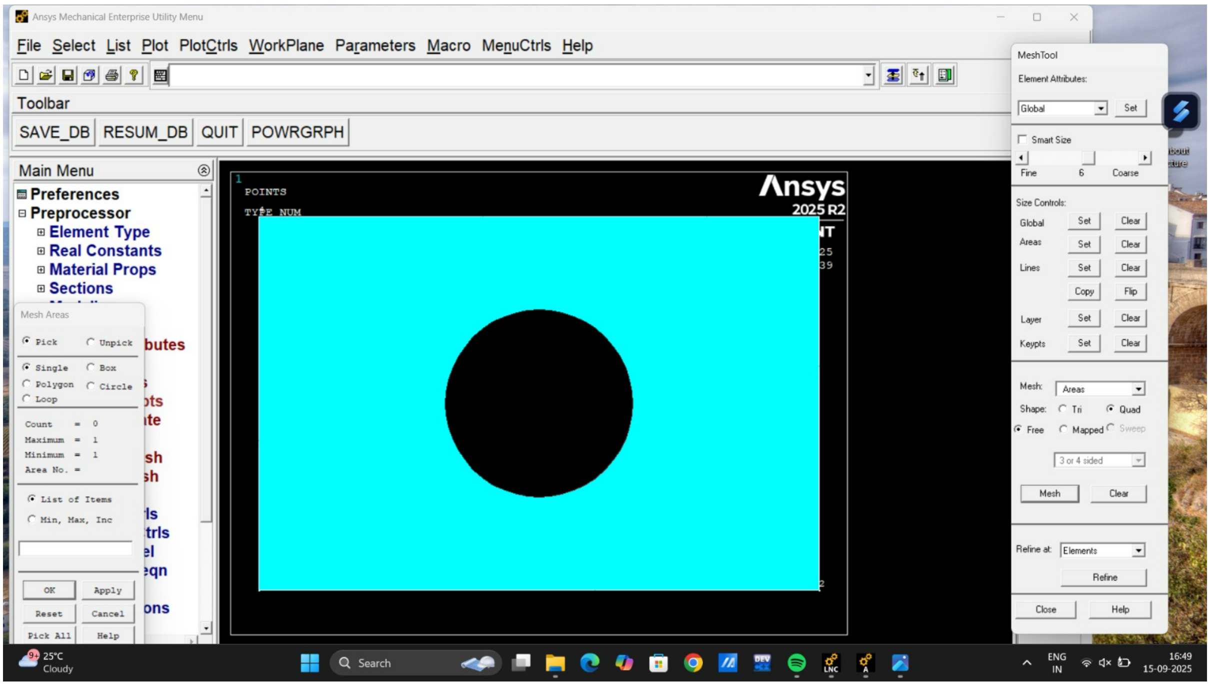


Step 12 : After that select the area you want to subtract

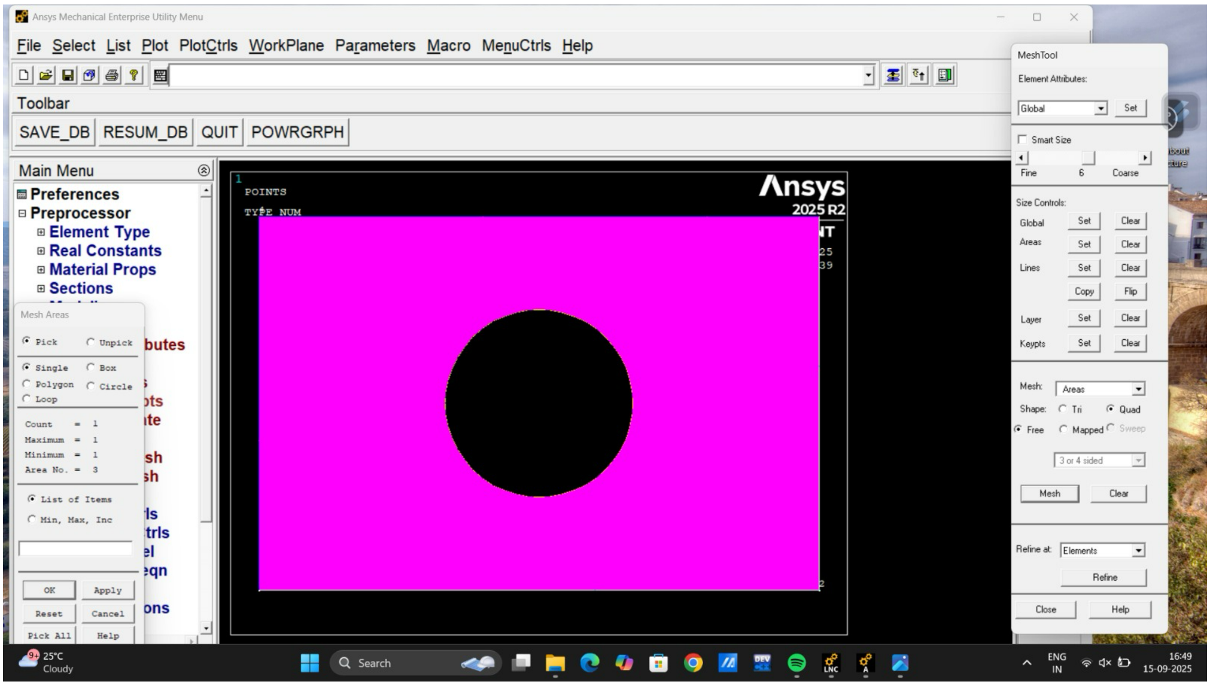


Step 13 : Meshing < Mesh Tool < Areas - Go to set

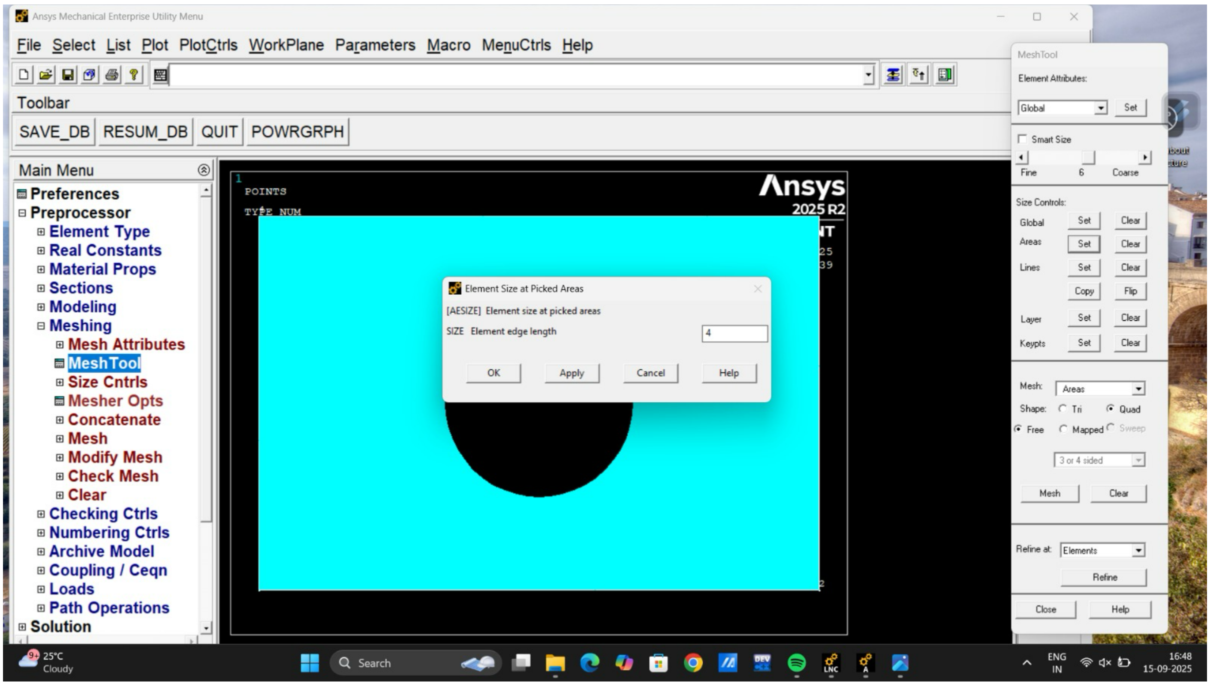


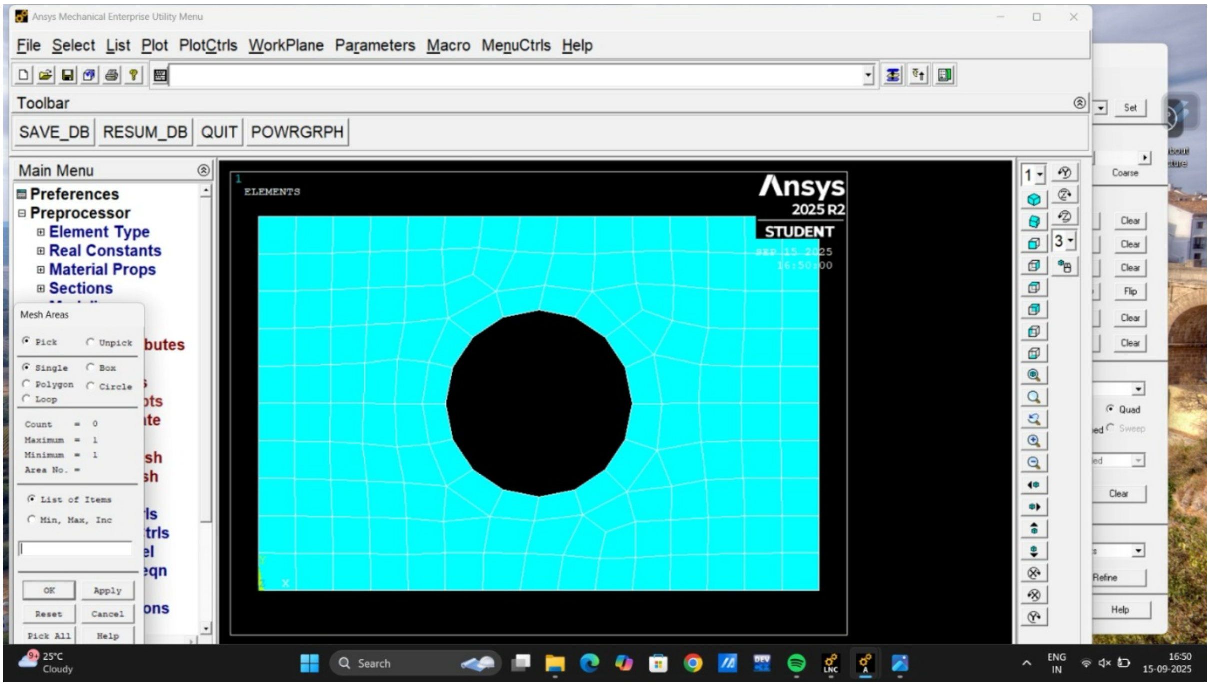


Step 14 : Select the Object

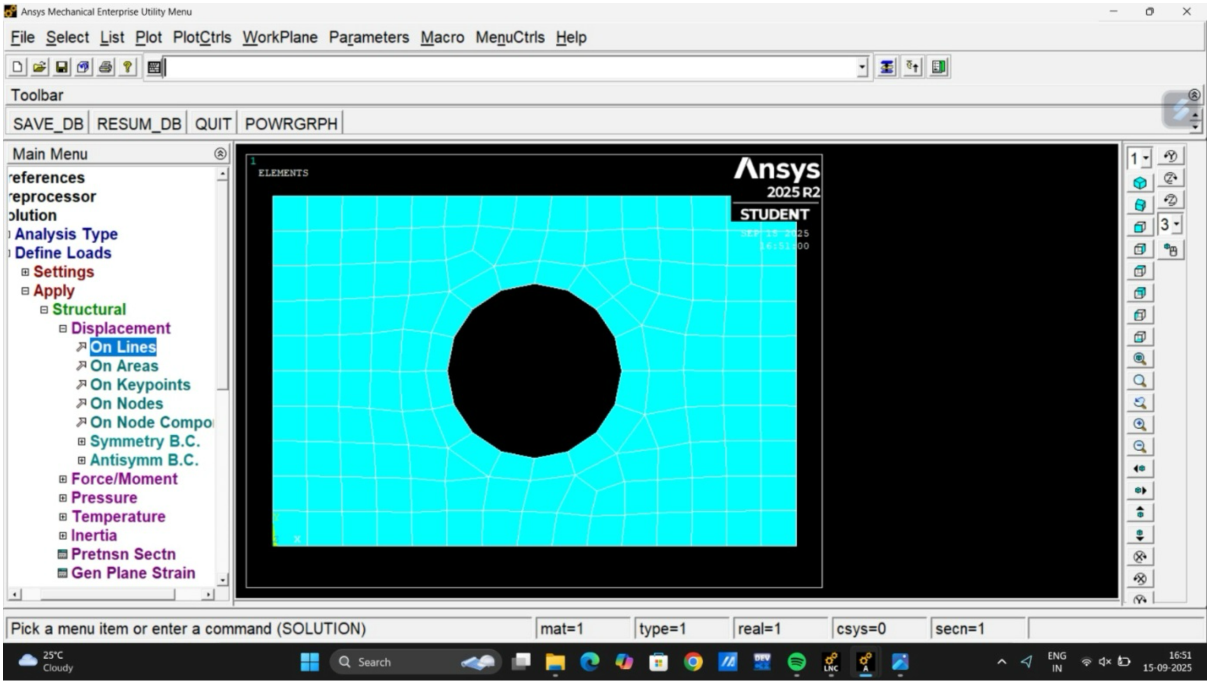


Step 15 : Type Element Length [you can take any value]

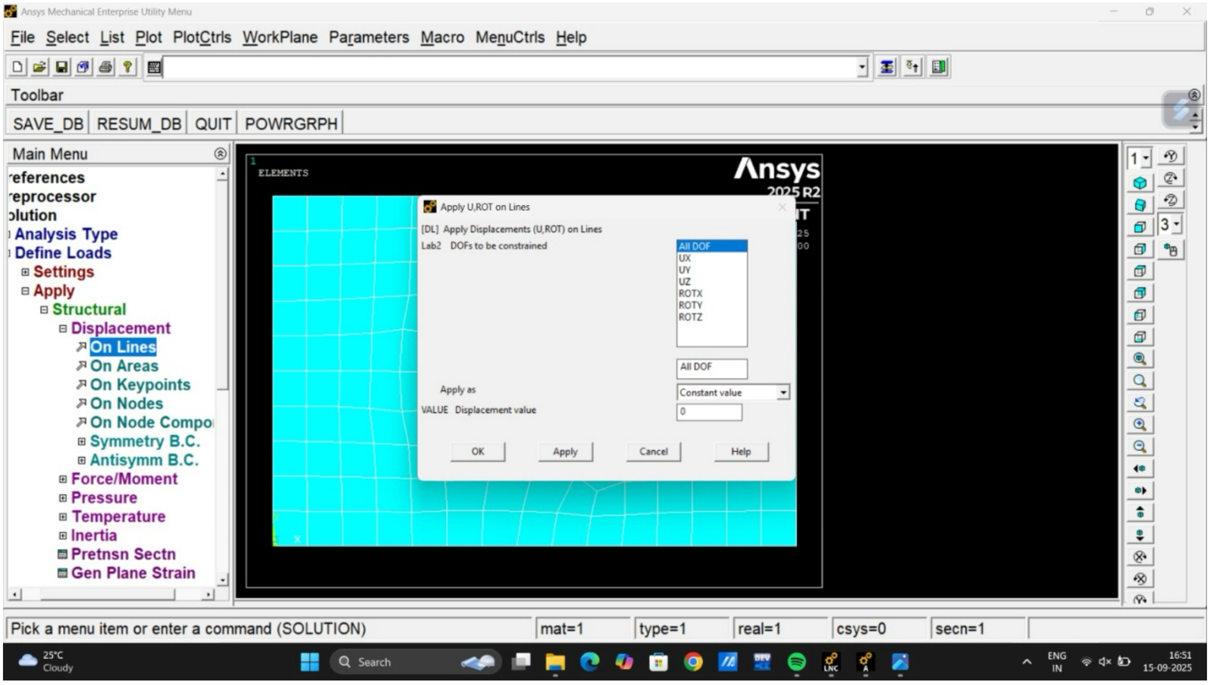




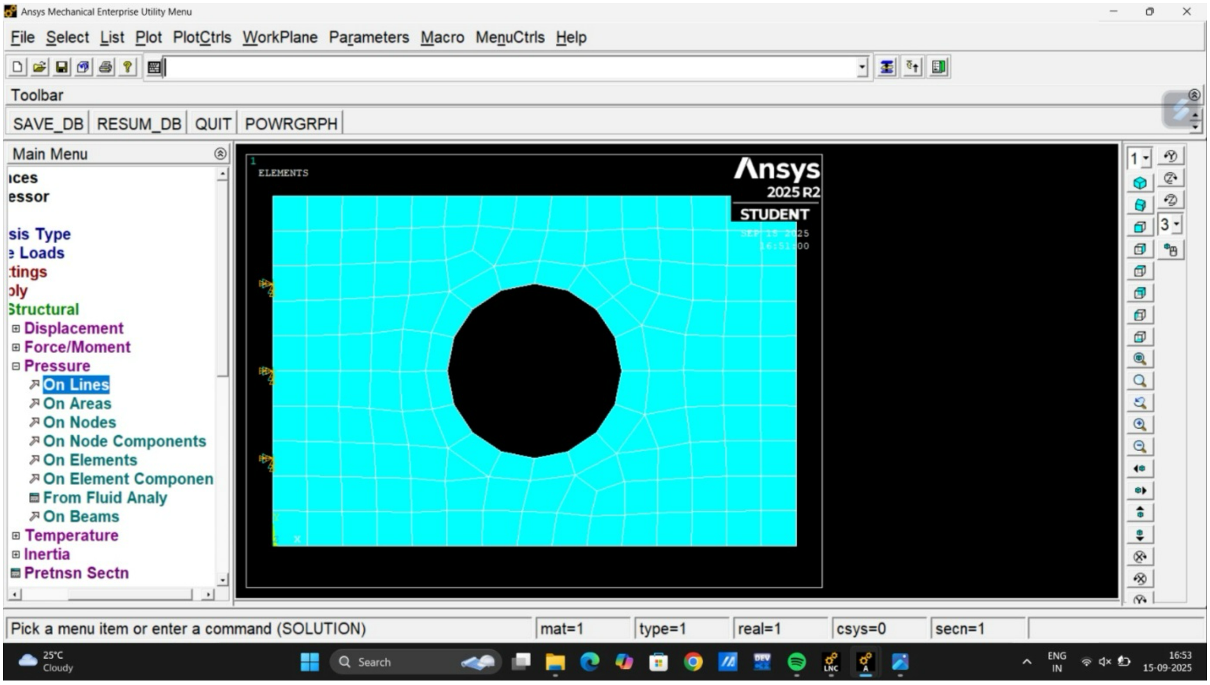
Step 16 : Solution < Define Loads < Apply < Structural < Displacement < On Lines

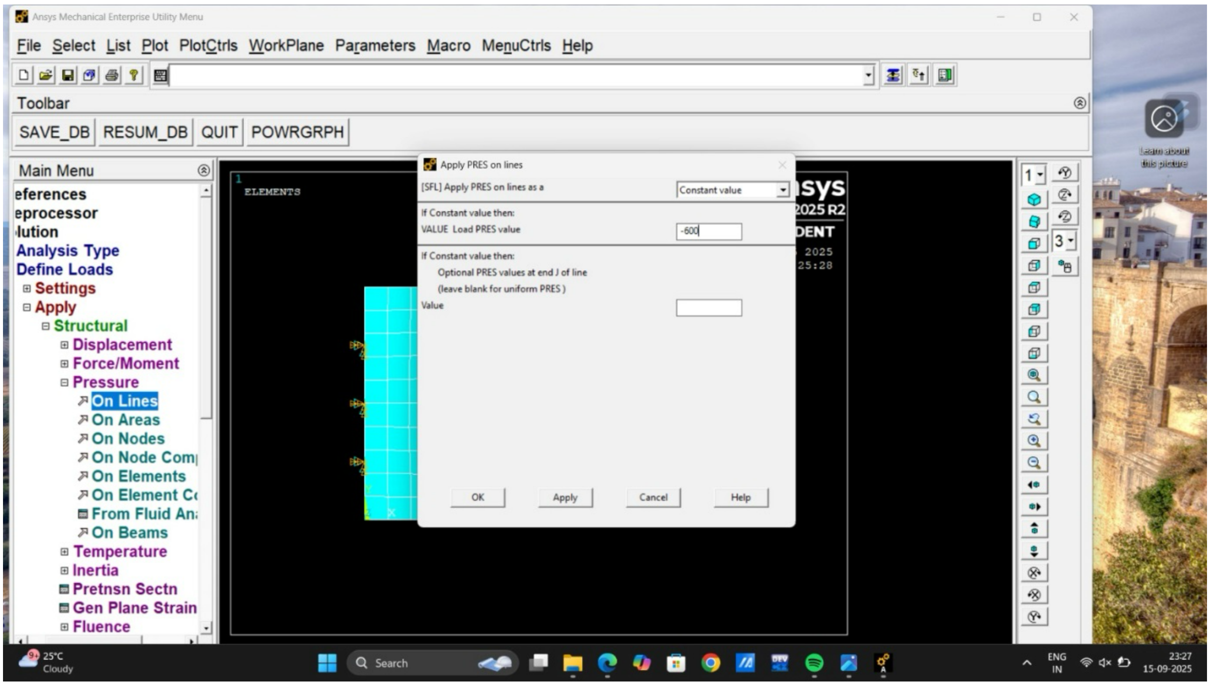


Step 17 : Select All DOF as Zero

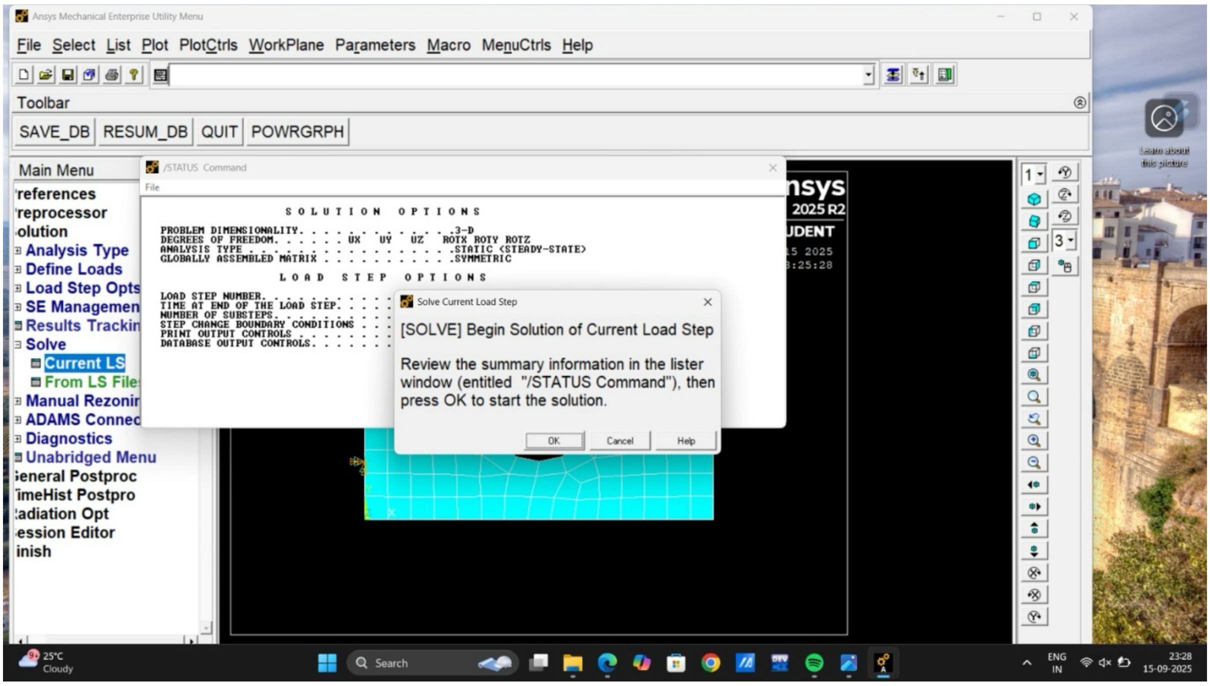


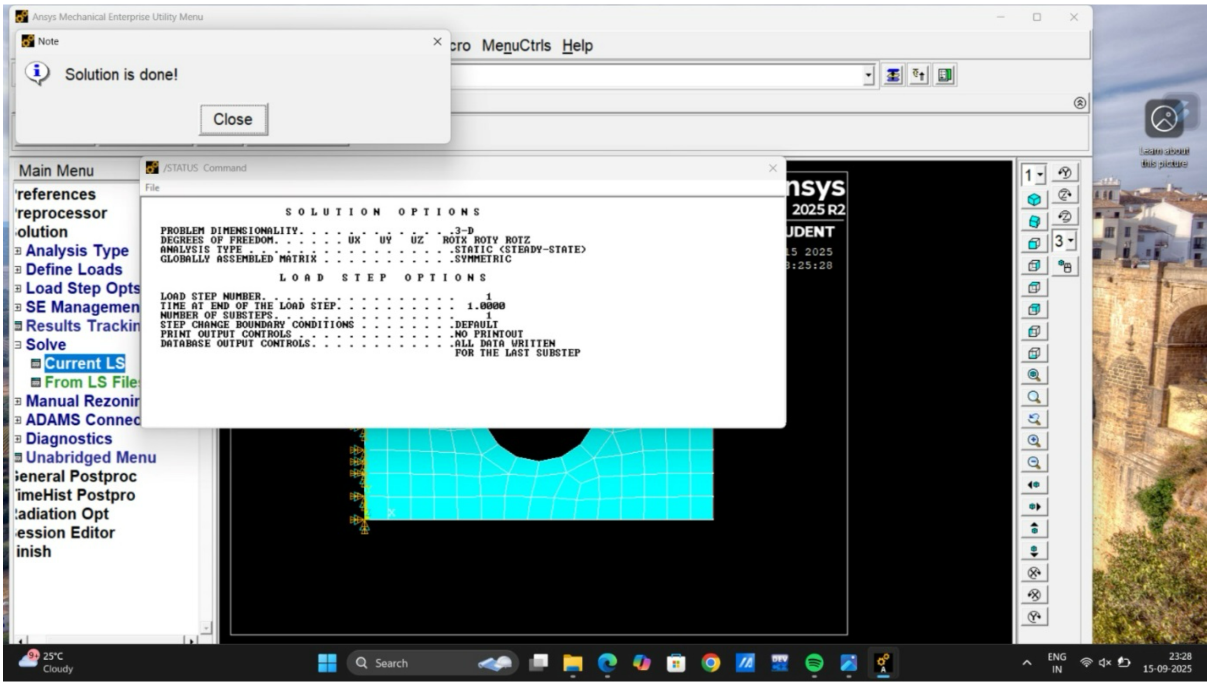
Step 18 : Solution < Define Loads < Apply < Structural < Pressure < On Lines



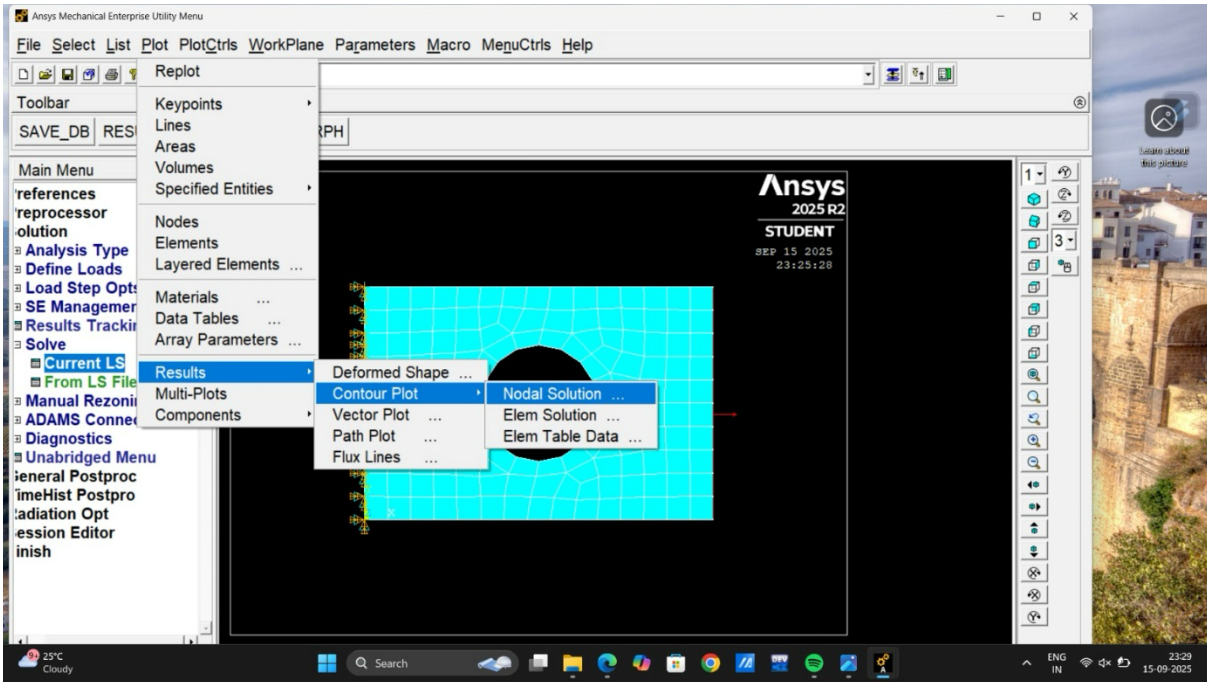


Step 19 : Solution < Solve < Current LS < Ok

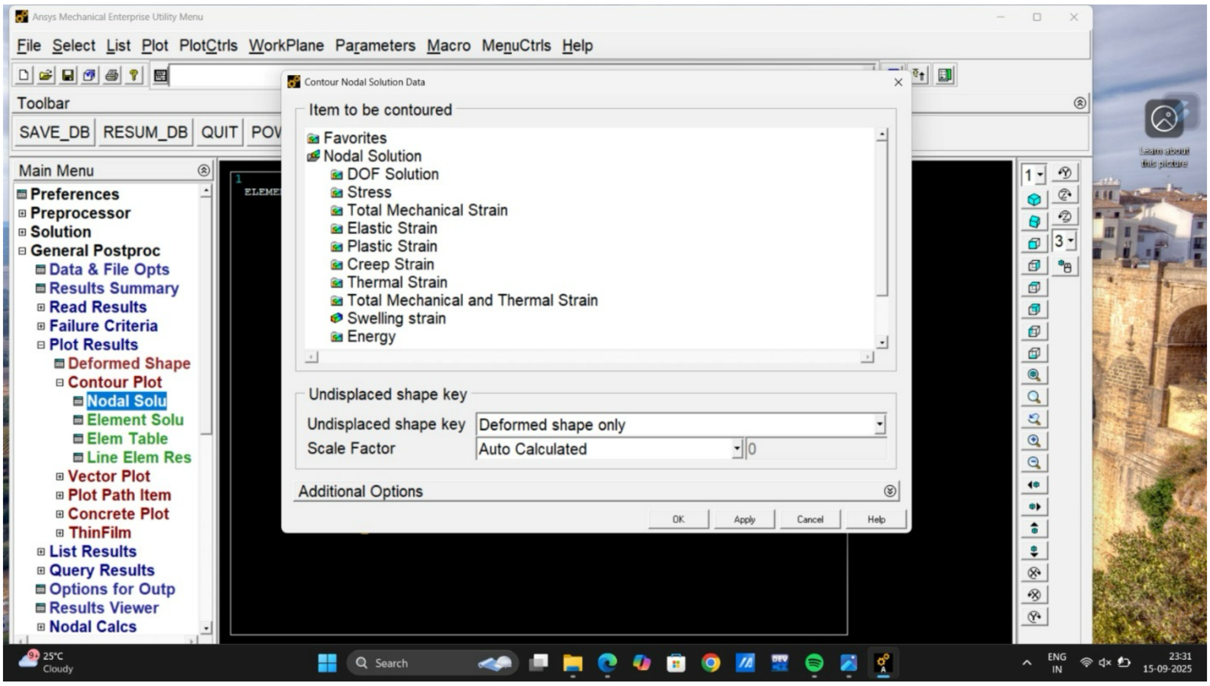




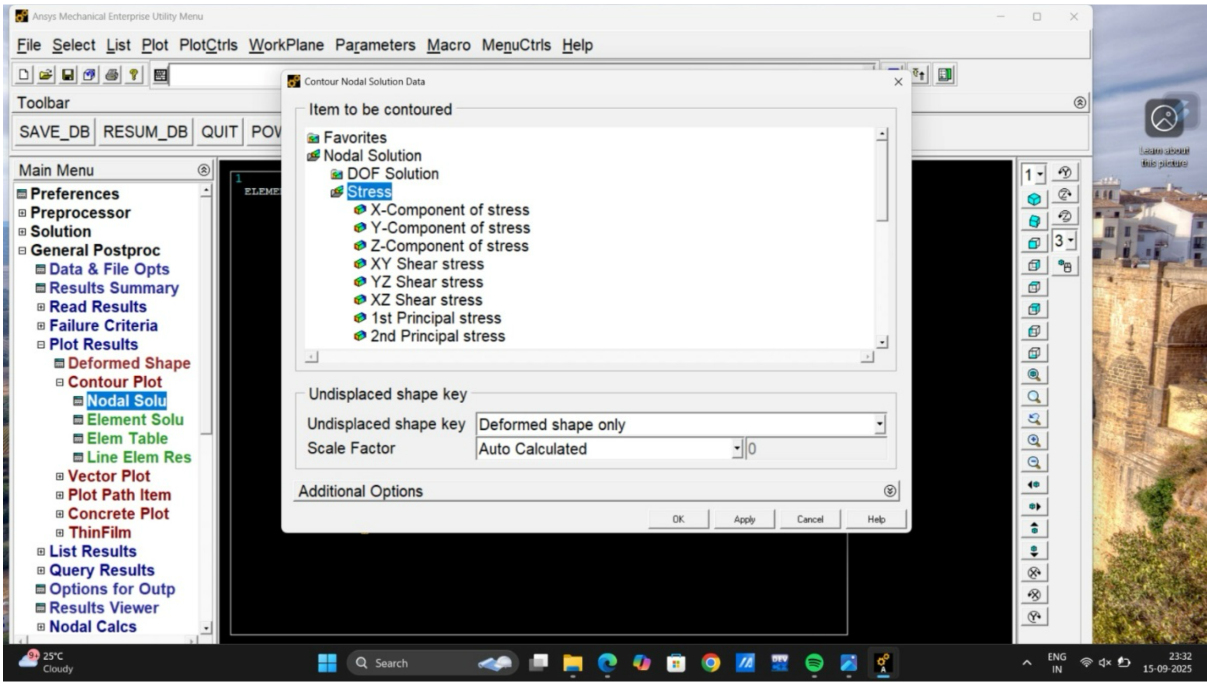
Step 20 : Plot < Results < Contour Plot < Nodal Solution



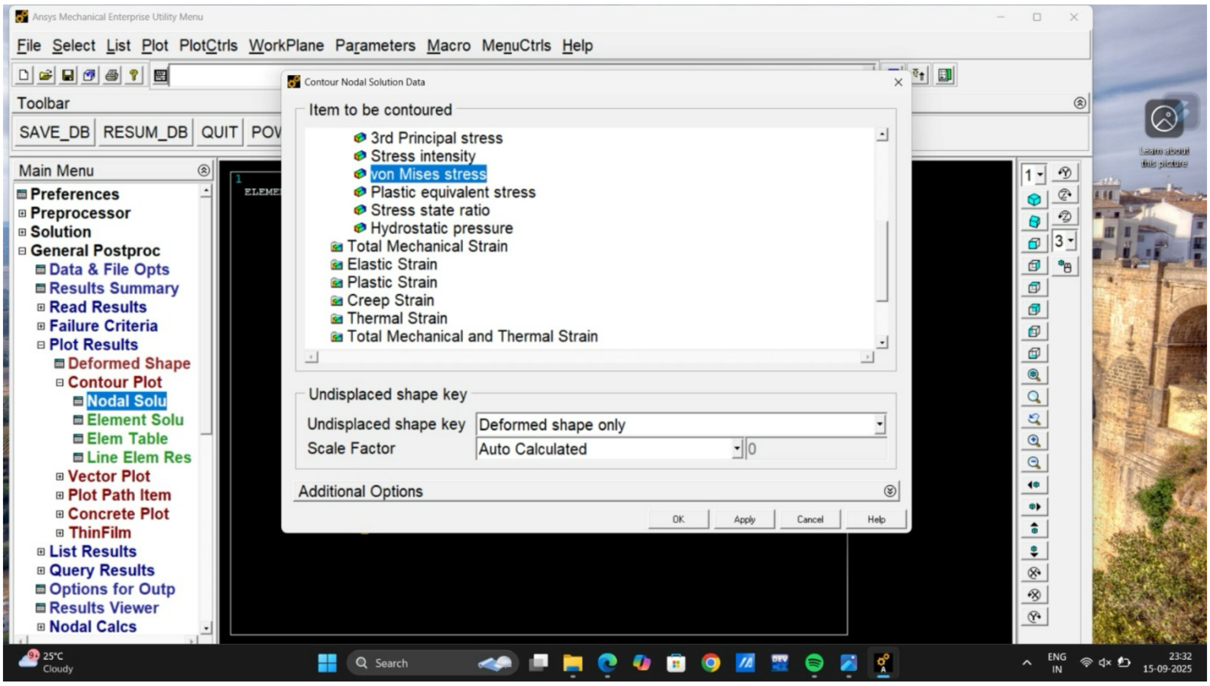
Step 21 : General Postproc < Plot Results < Contour Plot < Nodal Solution

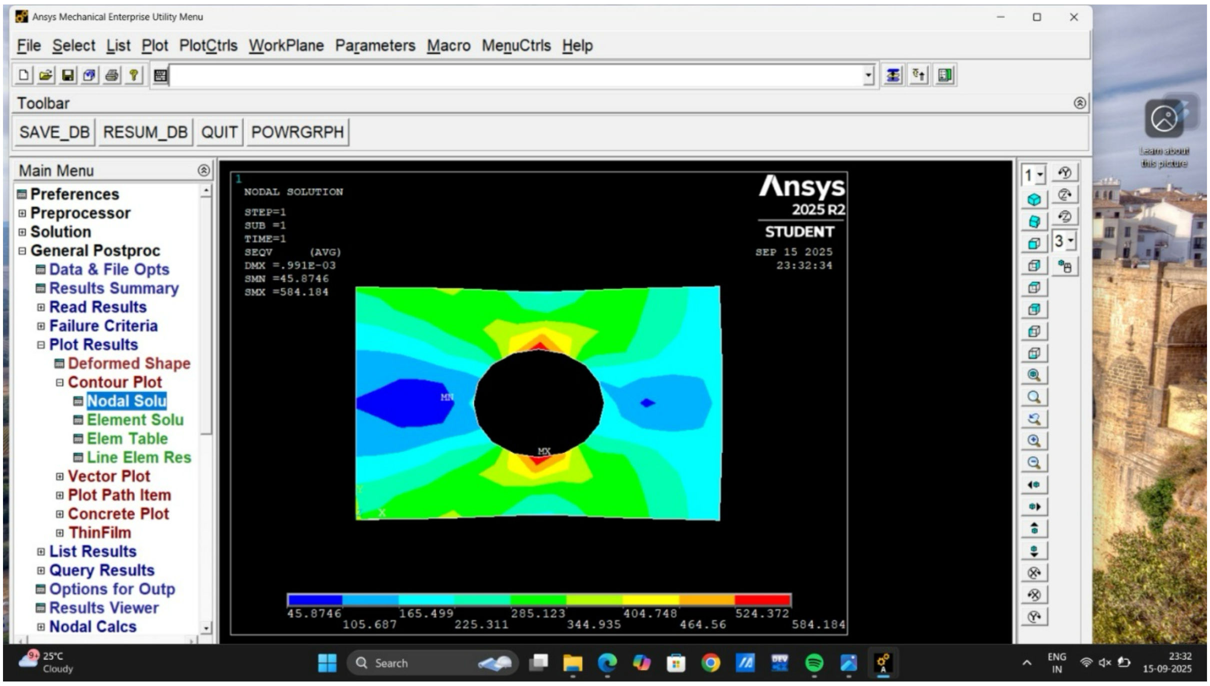


Step 22 : Select the stress

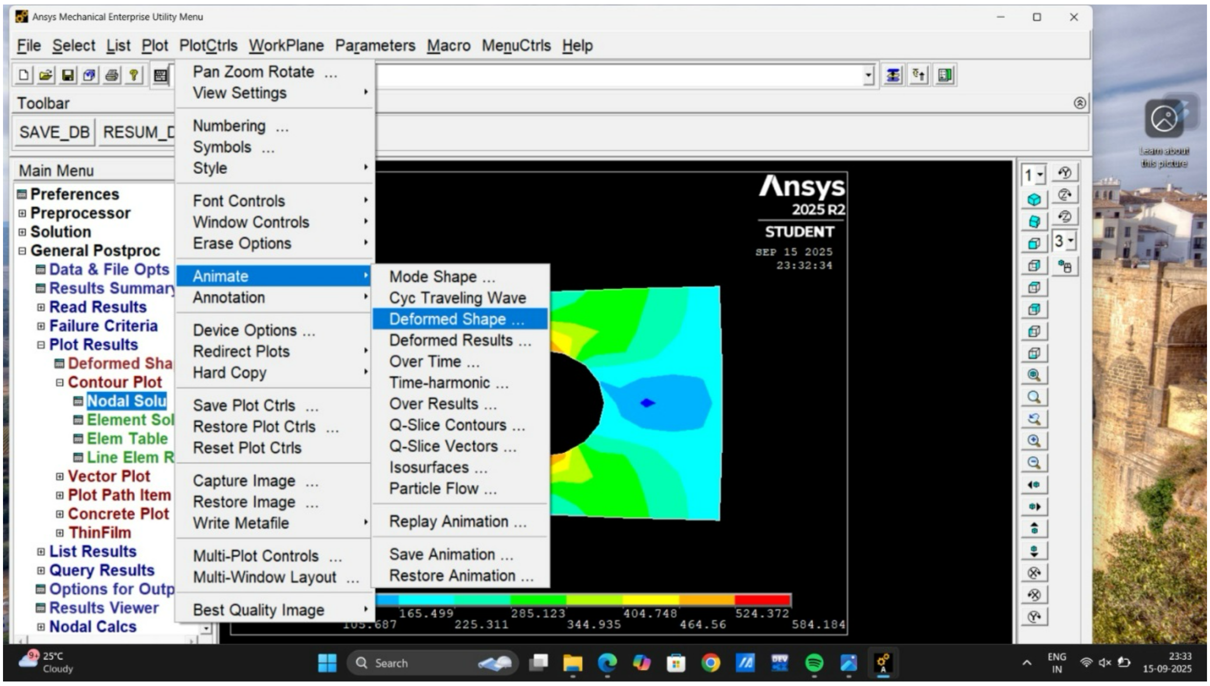


Step 23 : Select the von mises stress and click ok

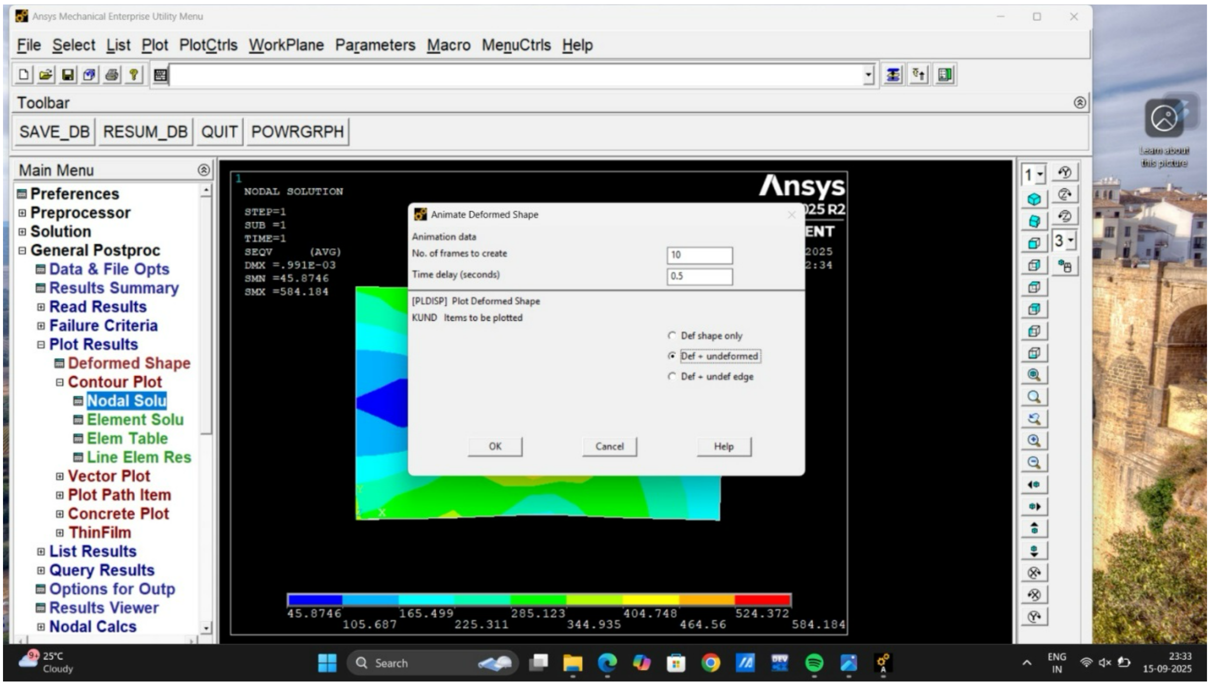


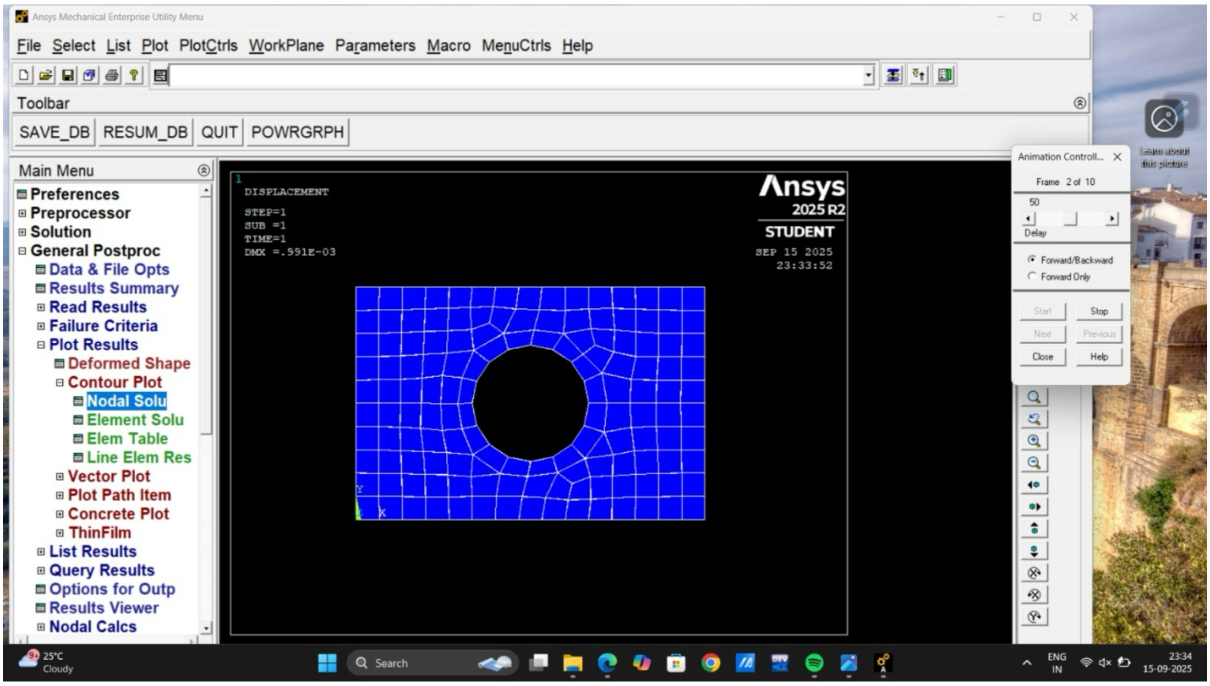


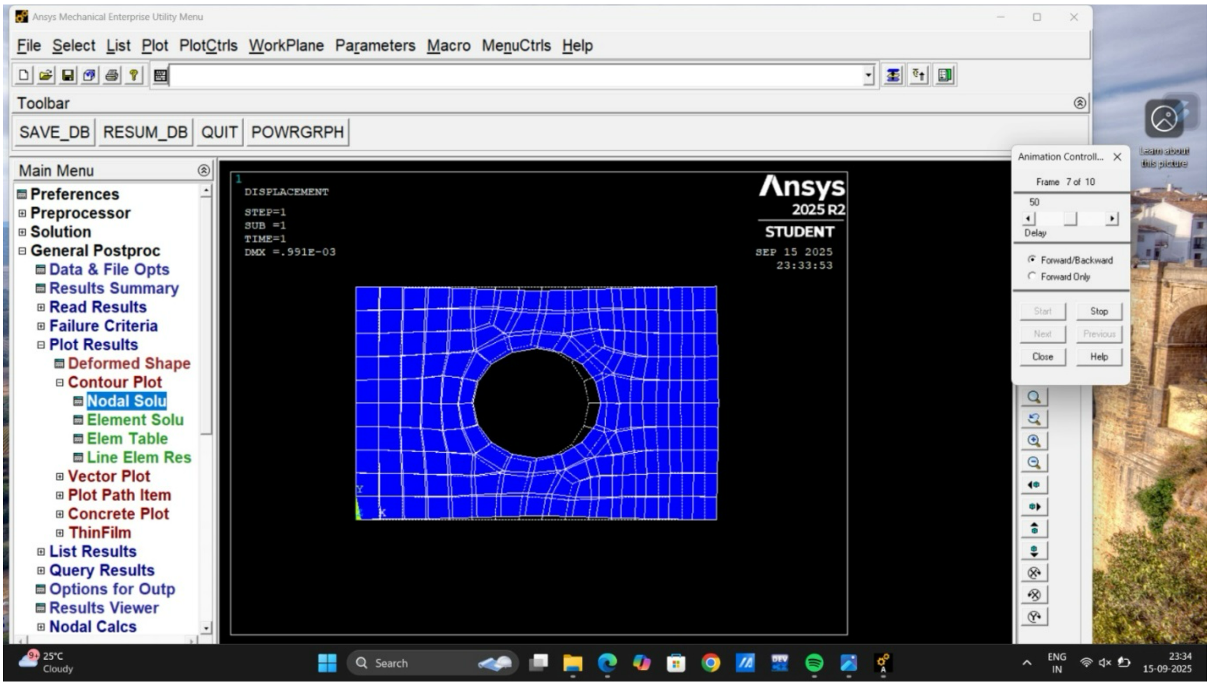
Step 24 : PlotCtrls < Animate < Deformed Shape

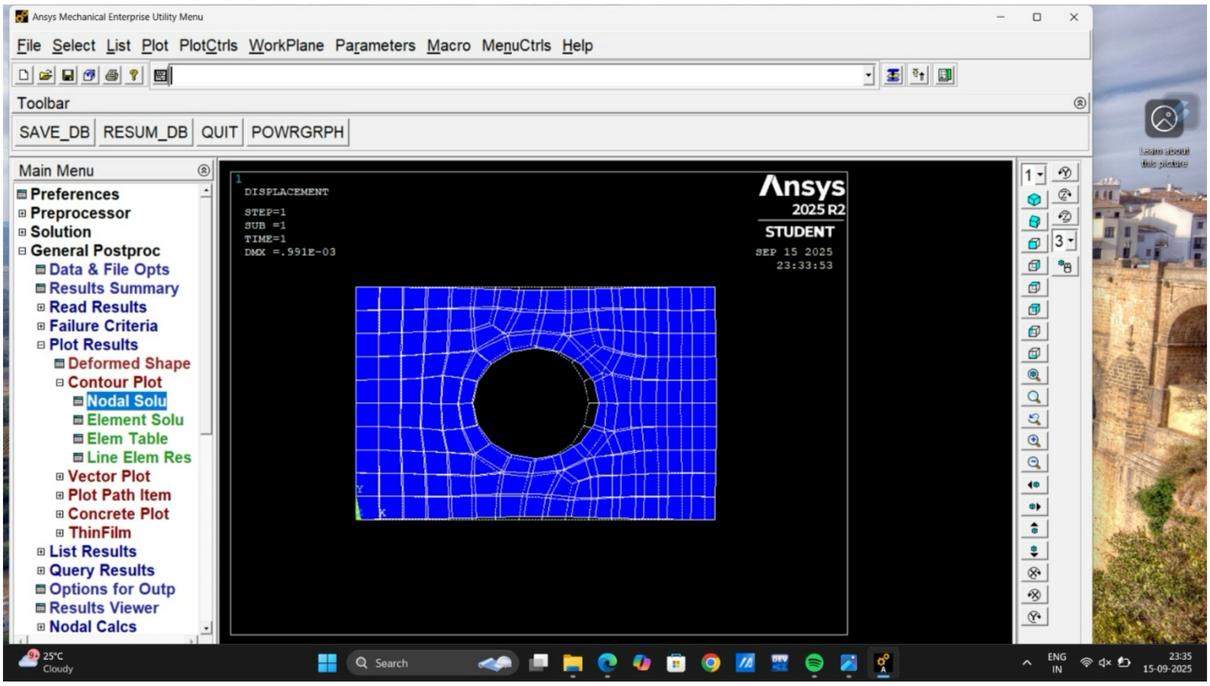


Step 25 : Select Def + undeformed

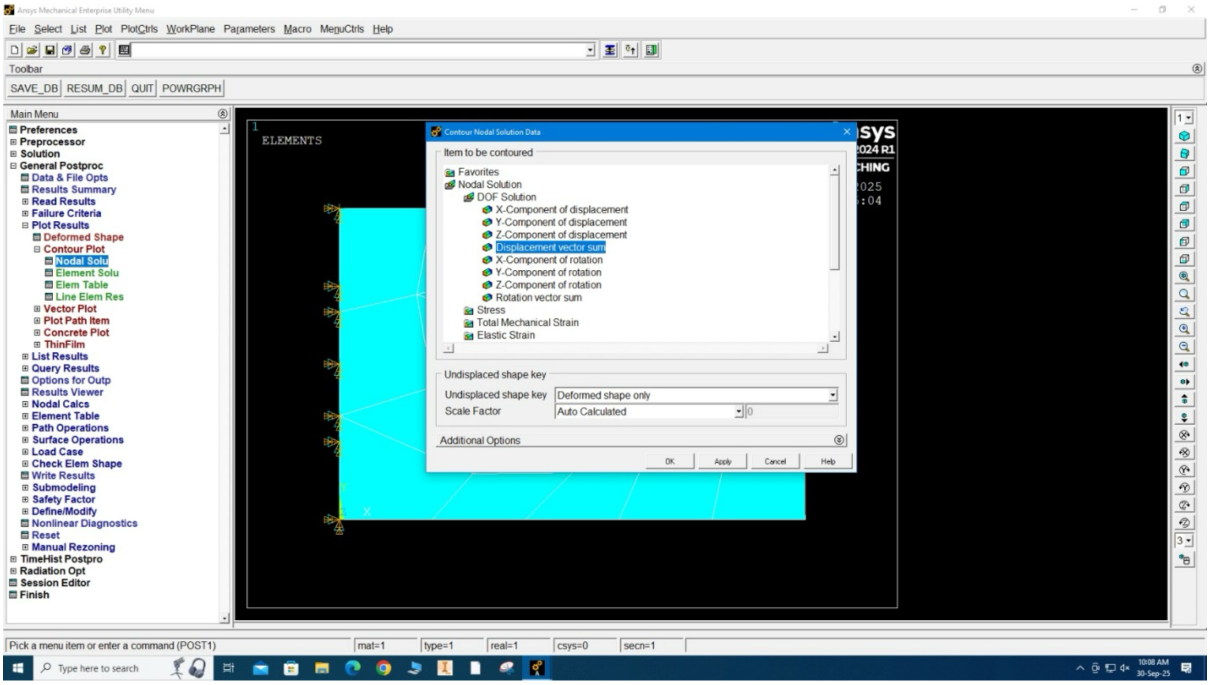


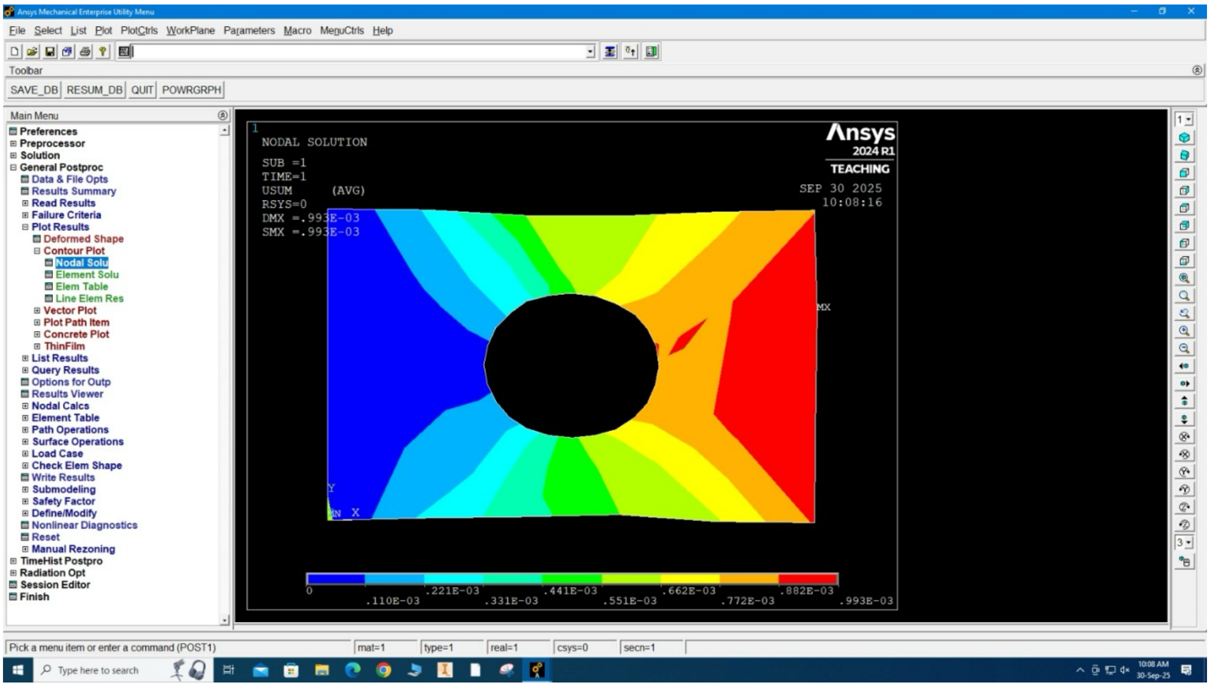



Ansys Mechanical Enterprise Utility Menu
File Select List Plot PlotCtrls WorkPlane Parameters Macro MenuCtrls Help
Toolbar
SAVE_DB RESUM_DB QUIT POWRGRPH
Main Menu
Preferences
Preprocessor
Solution
General Postproc
Data & File Opts
Results Summary
Read Results
Failure Criteria
Plot Results
Deformed Shape
Contour Plot
Nodal Solu
Element Solu
Elem Table
Line Elem Res
Vector Plot
Plot Path Item
Concrete Plot
ThinFilm
List Results
Query Results
Options for Outp
Results Viewer
Nodal Calcs
DISPLACEMENT
STEP=1
SUB =1
TIME=1
DMX =.991E-03
Ansys
2025 R2
STUDENT
SEP 15 2025
23:33:53
Animation Controll...
Frame 7 of 10
Delay
Forward/Backward
Forward Only
Start Stop
Next Previous
Close Help
25°C
Cloudy
Search
ENG
IN
23:34
15-09-2025


Ansys Mechanical Enterprise Utility Menu
File Select List Plot PlotCtrls WorkPlane Parameters Macro MenuCtrls Help
Toolbar
SAVE_DB RESUM_DB QUIT POWRGRPH
Main Menu
Preferences
Preprocessor
Solution
General Postproc
Data & File Opts
Results Summary
Read Results
Failure Criteria
Plot Results
Deformed Shape
Contour Plot
Nodal Solu
Element Solu
Elem Table
Line Elem Res
Vector Plot
Plot Path Item
Concrete Plot
ThinFilm
List Results
Query Results
Options for Outp
Results Viewer
Nodal Calcs
DISPLACEMENT
STEP=1
SUB =1
TIME=1
DMX =.991E-03
Ansys
2025 R2
STUDENT
SEP 15 2025
23:33:53
25°C
Cloudy
Search
ENG
IN
23:35
15-09-2025

Step 26 : General postproc < Plot results < Nodal solu
selection Nodal solution < DOF solution < Displacement vector



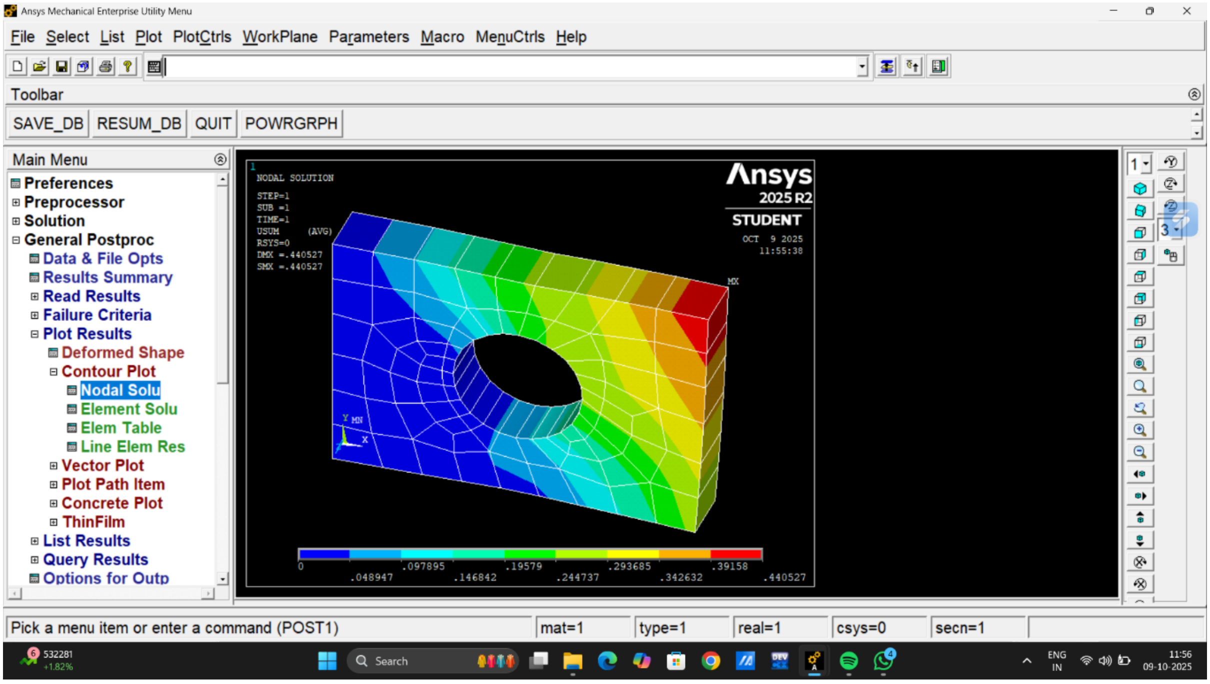


Step 27 : 3D view

PlotCtrls < Style < Size and Shape

Display of element turn on

Select isometric view from right side bar



Modern Education Society's
Wadia College Of Engineering,
Pune-01

DEPARTMENT OF AUTOMATION AND ROBOTICS ENGINEERING

COMPUTER AIDED ENGINEERING & MANUFACTURING (302523)

Academic Year 2025-2026

TITLE	THERMAL ANALYSIS – STATIC/TRANSIENT ANALYSIS		
NAME OF STUDENT			
CLASS AND DIVISION		BATCH	
SEMESTER		ROLL NO	
DATE OF PERFORMANCE		DATE OF SUBMISSION	
EXAMINED BY			

COURSE OUTCOMES:

- **CO1: DEFINE** the use of CAE tools and DESCRIBE the significance of shape functions infinite element formulations.
- **CO2:APPLY** material properties and boundary condition to SOLVE 1-D and 2-D element stiffness matrices to obtain nodal or elemental solution.
- **CO3:ANALYZE** and APPLY various numerical methods for different types of analysis.
- **CO4:CREATE** process plan and GENERATE GandM code using CAM software tools.
- **CO5:UNDRSTAND** lean manufacturing tools and techniques
- **CO6:APPLY** knowledge to do process planning and ESTIAMTE costing for the same.

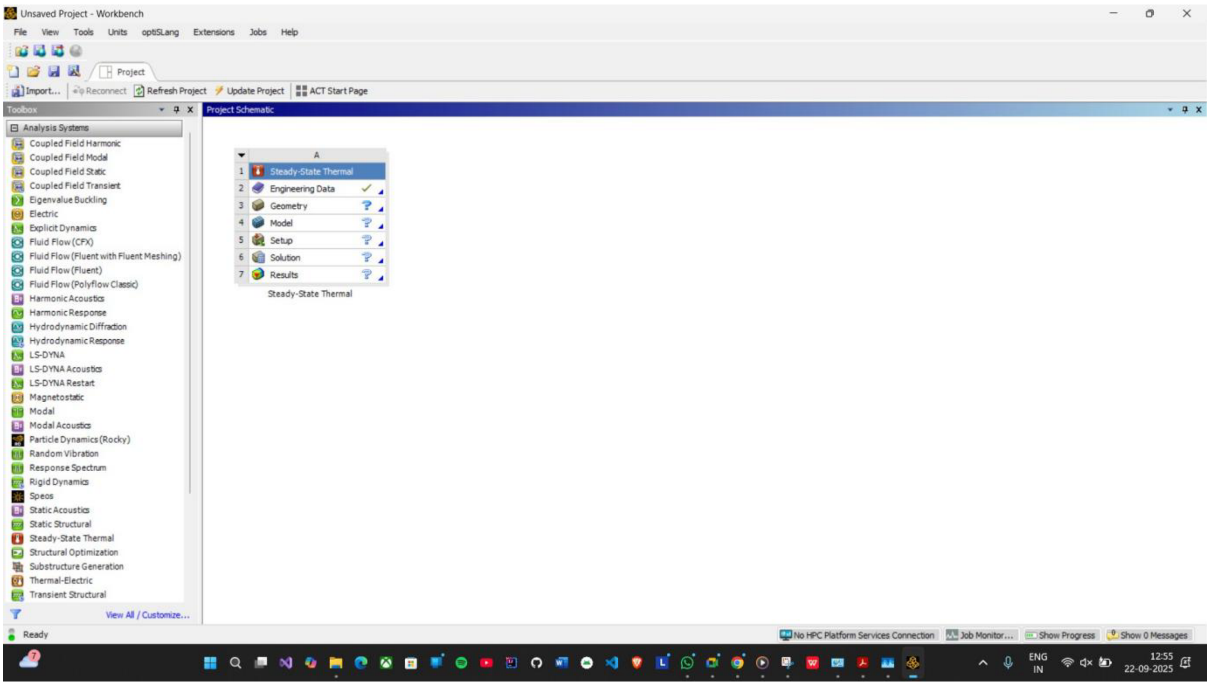
Name Of Student :	Class :
Semester/Year :	Roll No. :
Date Of Performance :	Date Of Submission :
Examined By : Prof. R. N. Yerrawar	Experiment No : 4

Title : Thermal Analysis – Static/Transient Analysis.

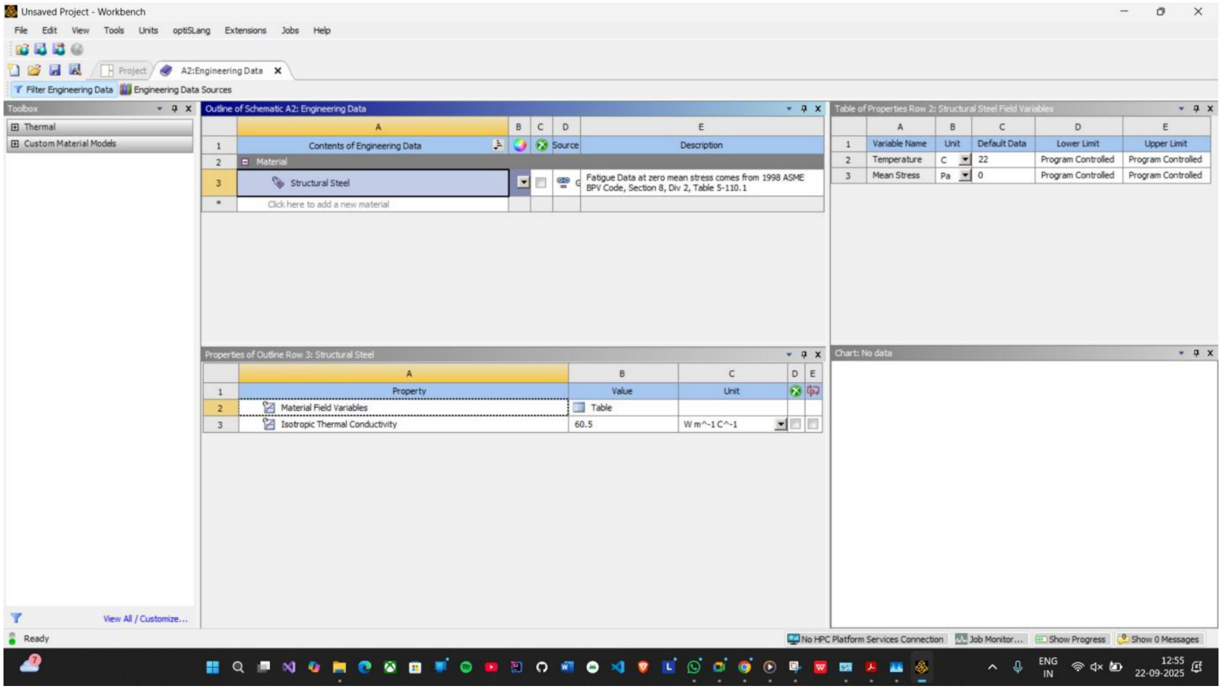
Aim : For the rectangular cross-session of the element B = 1m and H = 0.5m and Length of Element is 5m. Assign thermal Properties, Thermal Conductivity = 60.5 W/m°C; Apply a constant temperature of 100°C to the left side of the block (face 1) and a constant temperature of 300°C to the right side of the block (face 3). All other faces are insulated by default. Find temperature distribution in the element.

Solution:

1. **Click on Start – ANSYS Workbench.**
   On left side double click on Steady state thermal.



1. **Engineering Data:** Double click on engineering data to select suitable material from library or add material and its material properties. Default material selected is structural steel. For thermal analysis, check or add isotropic thermal conductivity.



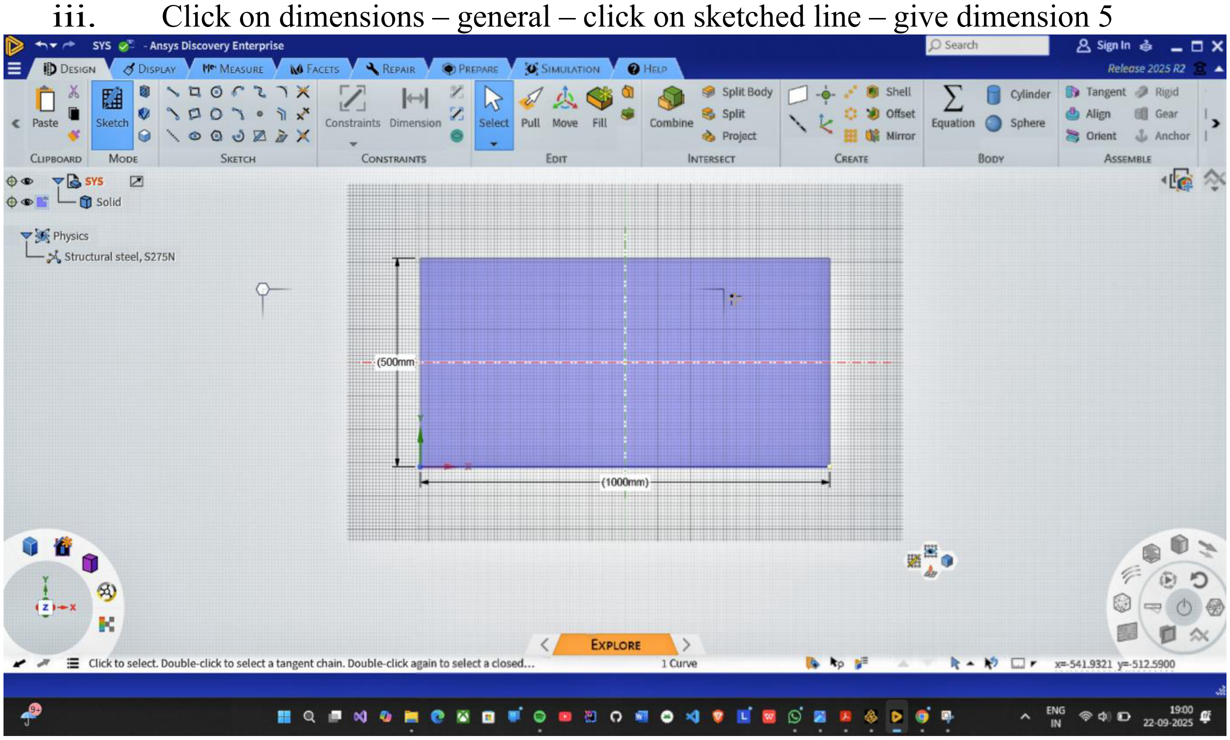
Click on project on upper left corner to go back.

2. **Geometry:** Double click on geometry to prepare model as per given data.

i. Select XY Plane – right click – select look at

ii. Click on sketching on left middle. Click on Draw – line – draw in workspace of ansys

iii. Click on dimensions – general – click on sketched line – give dimension 5

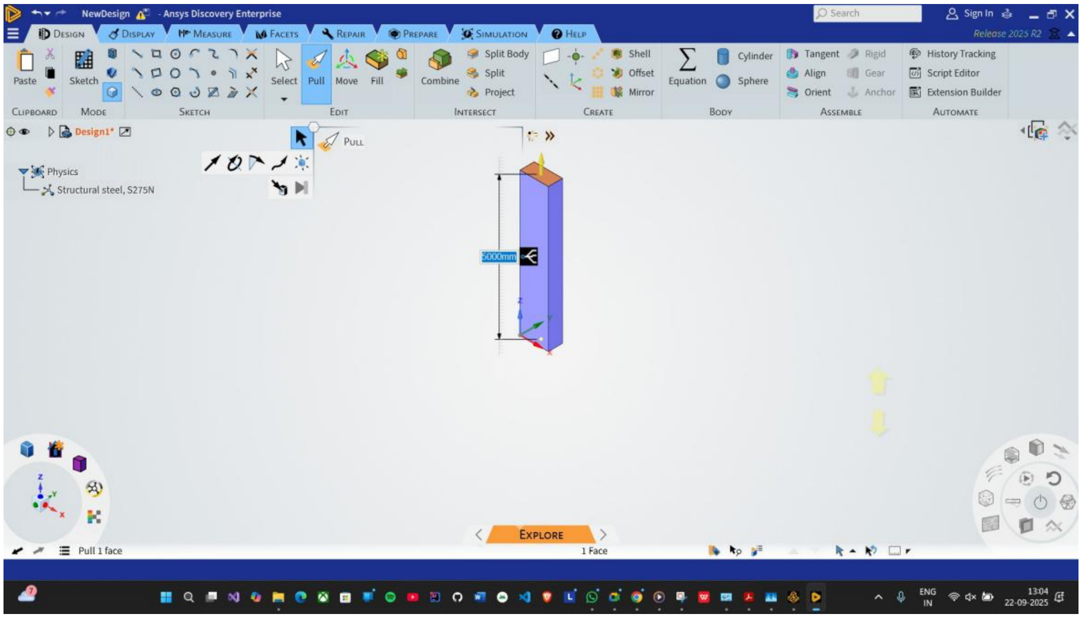


iv. Click on concept on menu bar – lines from sketch – select sketch – apply – generate

v. Click on concept – cross-section – select rectangular corss - section – give dimensions

vi. Click on part – line body – select cross section Rect1 – generate

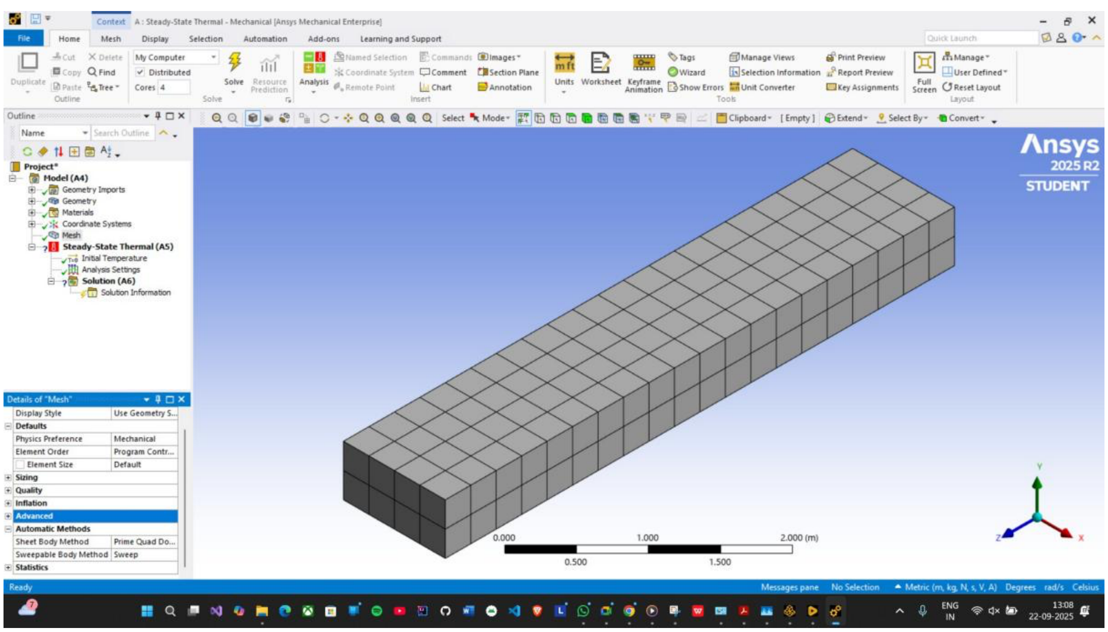
vii. Click on view in menubar – cross-section solids



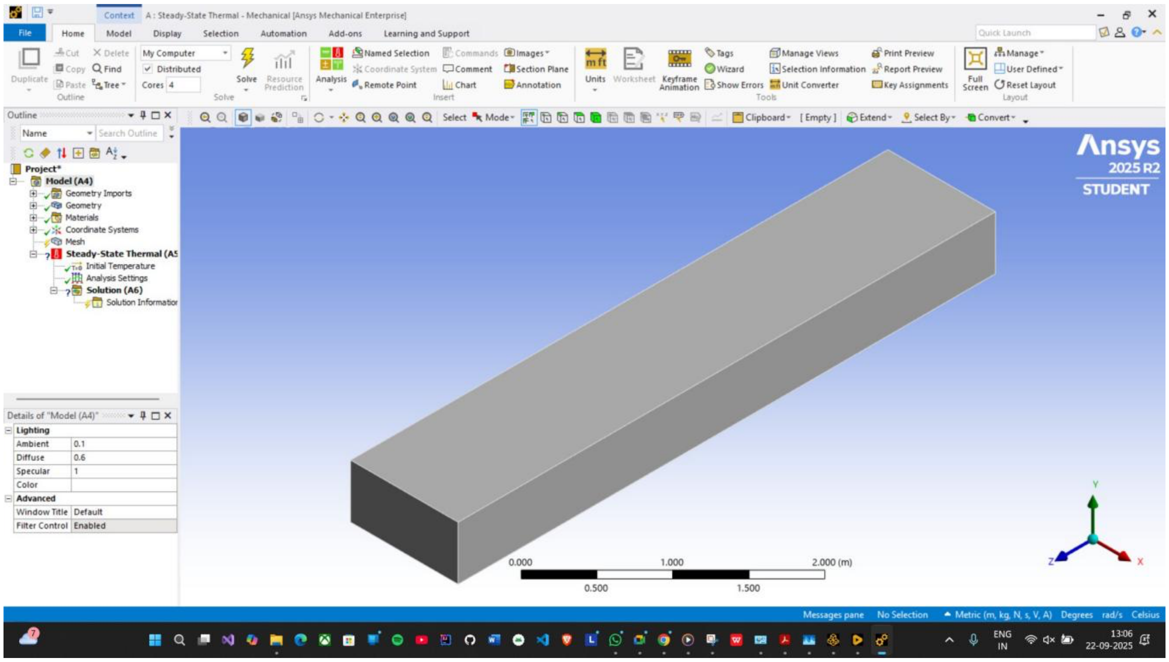
viii. Close design modeler environment

3. Model (Meshing): Double click on model to open analysis environment

Right click on mesh – Generate mesh



To change number of elements for 1D give element size 5000mm – generate mesh.



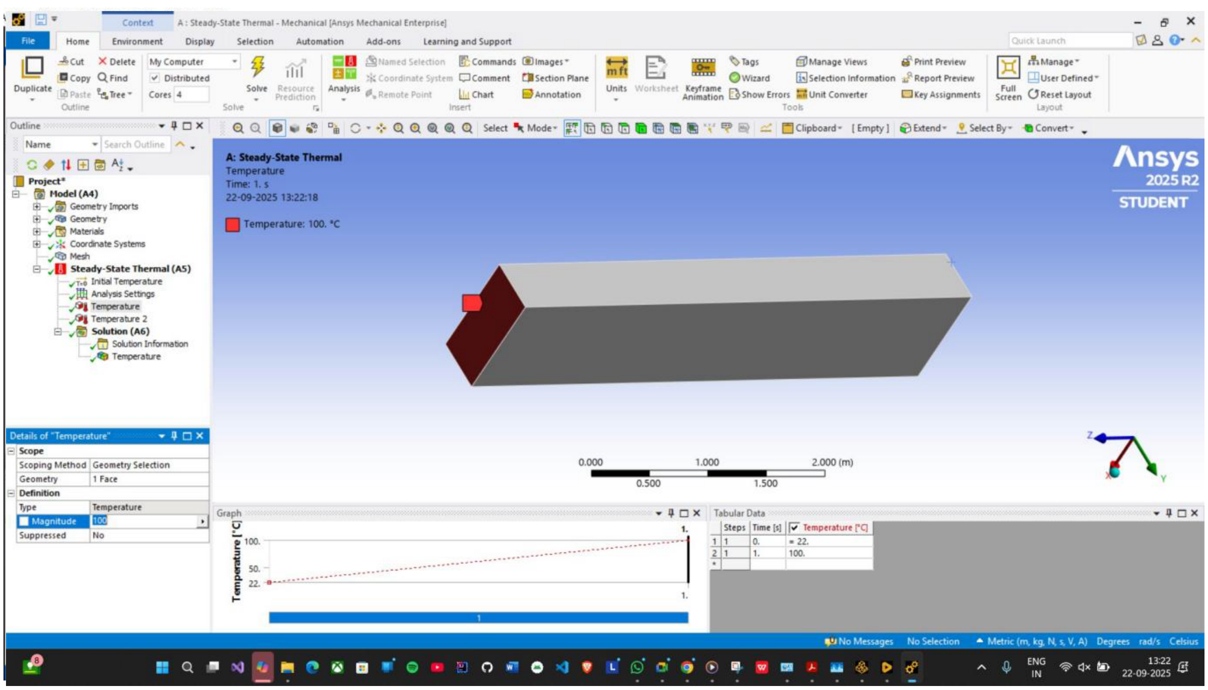
Check number of nodes and element in left bottom corner. This is 1d bar element with rectangular cross section have 1 element only.

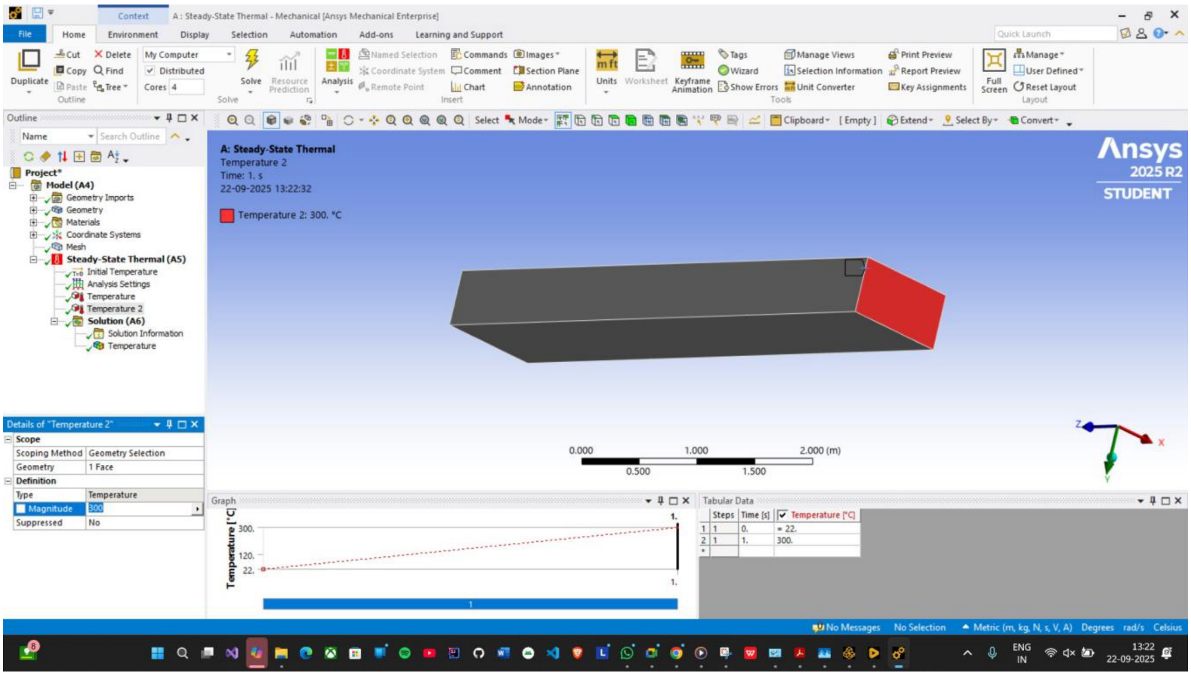
4. Steady state thermal: To apply boundary condition.

i. Select edge in upper toolbox

ii. Click on steady state thermal – select left edge – click on temperature – 300^0C.

iii. Click on steady state thermal – select left edge – click on temperature – 100^0C.

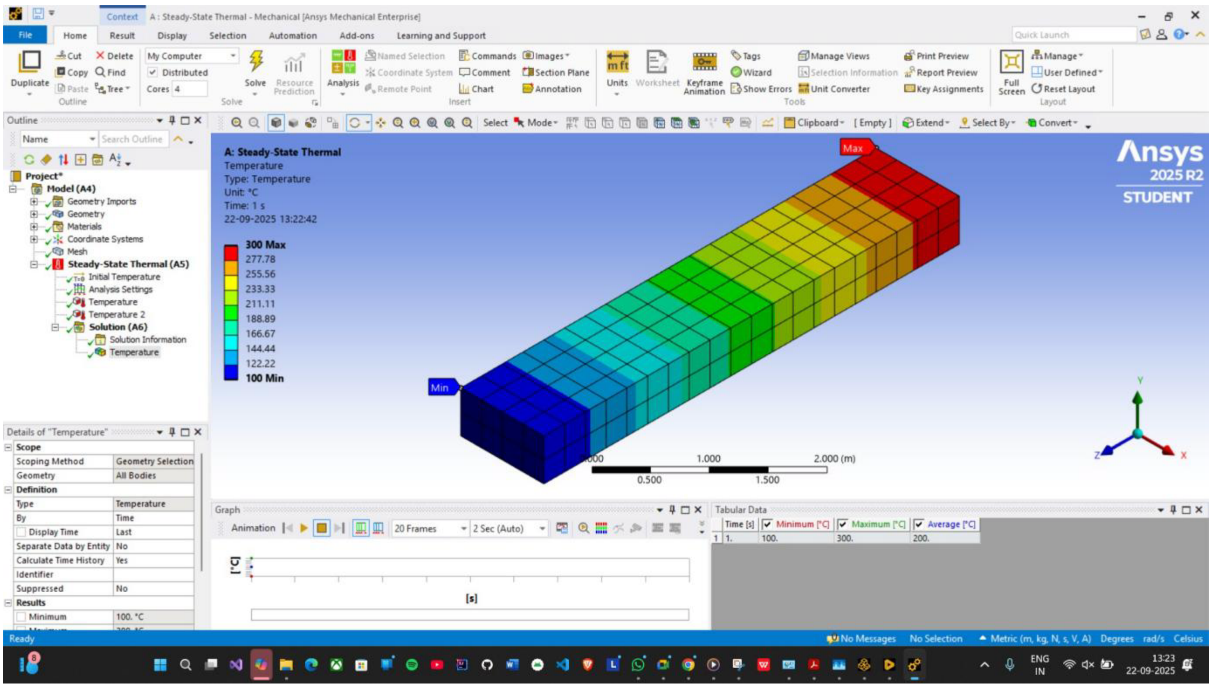




iv. Boundary condition complete

5. **Solution:** click on solution

i. On upper side select – Temperature



ii. Thermal analysis completed

Modern Education Society's
Wadia College Of Engineering,
Pune-01

DEPARTMENT OF AUTOMATION AND ROBOTICS ENGINEERING

COMPUTER AIDED ENGINEERING & MANUFACTURING (302523)

Academic Year 2025-2026

TITLE	ANALYSIS OF MACHINE COMPONENT USING 3D ELEMENTS		
NAME OF STUDENT			
CLASS AND DIVISION		BATCH	
SEMESTER		ROLL NO	
DATE OF PERFORMANCE		DATE OF SUBMISSION	
EXAMINED BY			

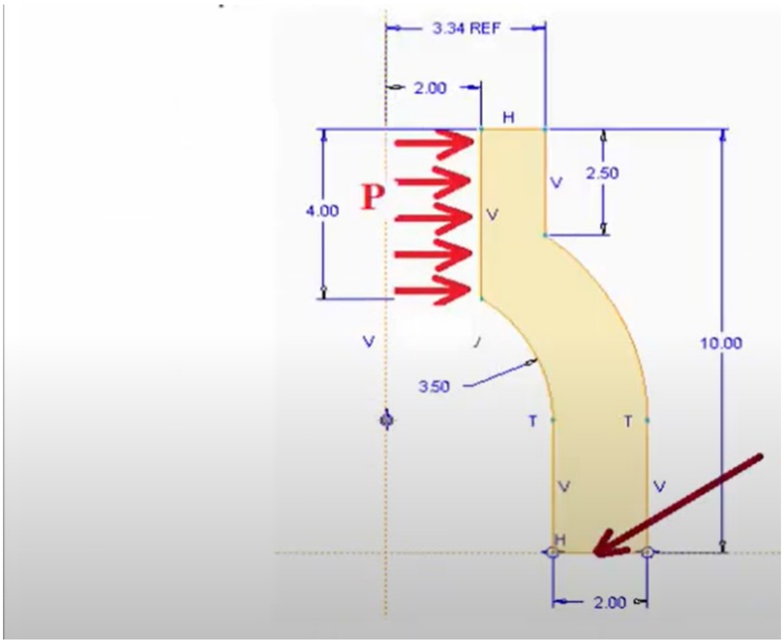
COURSE OUTCOMES:

- **CO1: DEFINE** the use of CAE tools and DESCRIBE the significance of shape functions infinite element formulations.
- **CO2:APPLY** material properties and boundary condition to SOLVE 1-D and 2-D element stiffness matrices to obtain nodal or elemental solution.
- **CO3:ANALYZE** and APPLY various numerical methods for different types of analysis.
- **CO4:CREATE** process plan and GENERATE GandM code using CAM software tools.
- **CO5:UNDRSTAND** lean manufacturing tools and techniques
- **CO6:APPLY** knowledge to do process planning and ESTIAMTE costing for the same.

Name Of Student :	**Class :**
Semester/Year :	**Roll No. :**
Date Of Performance :	**Date Of Submission :**
Examined By : Prof. R. N. Yerrawar	**Experiment No : 5**

Title : Stress Analysis of Axis Symmetric Components

Problem : To calculate the total deflection and stress developed in an axisymmetric component subjected to internal pressure load using an analysis software

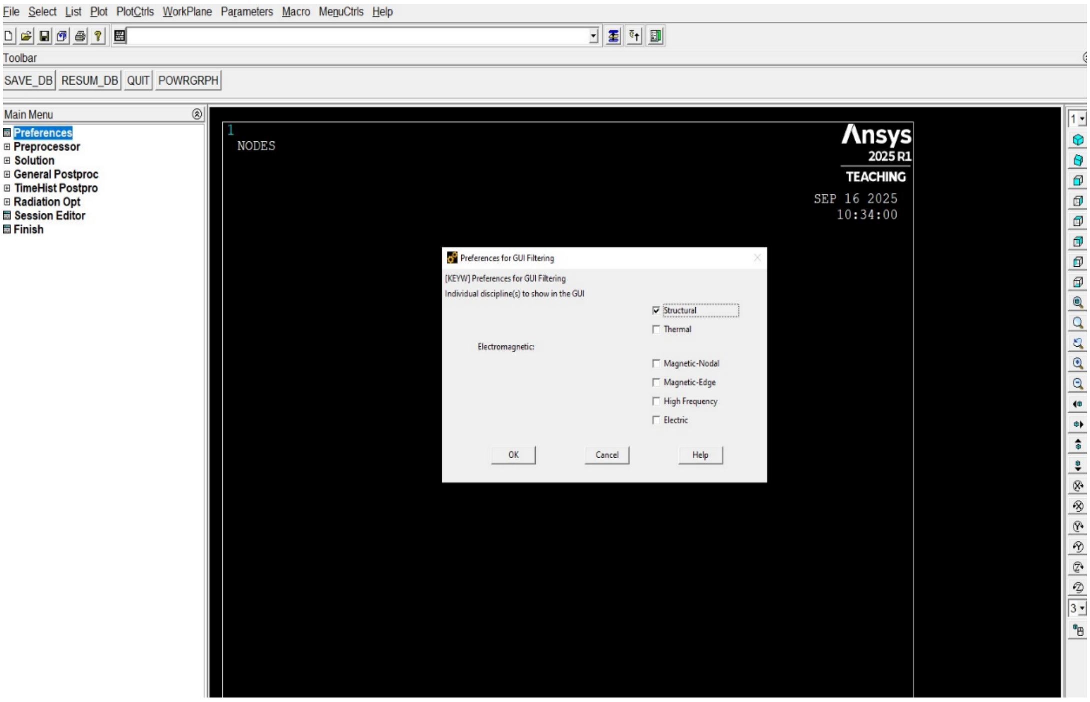


← **All DOF**

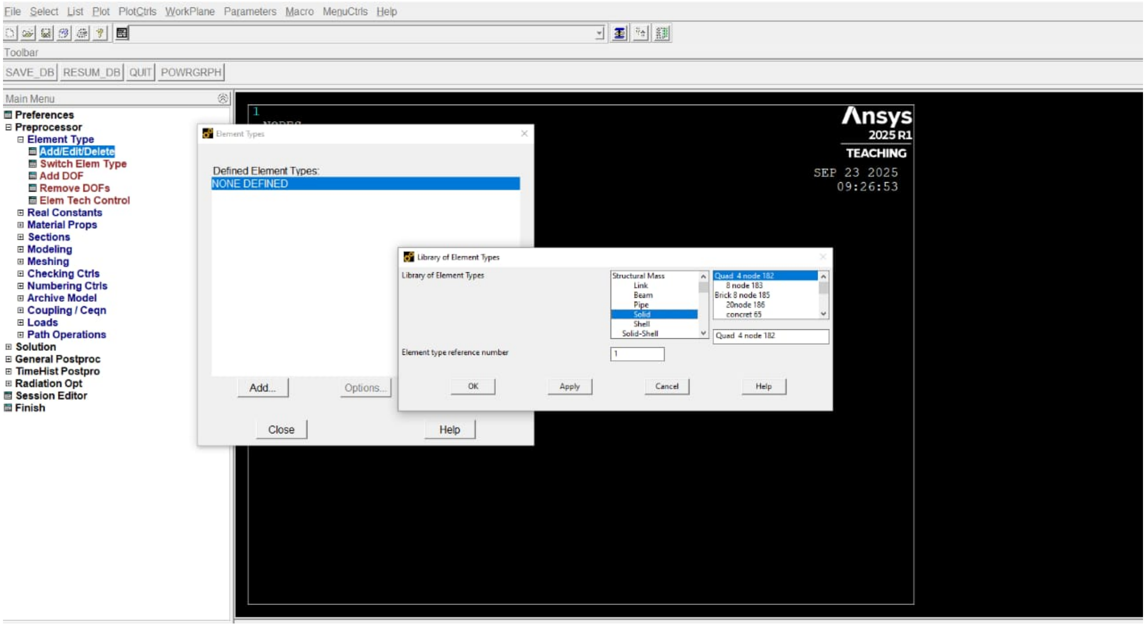
Component Coordinates :

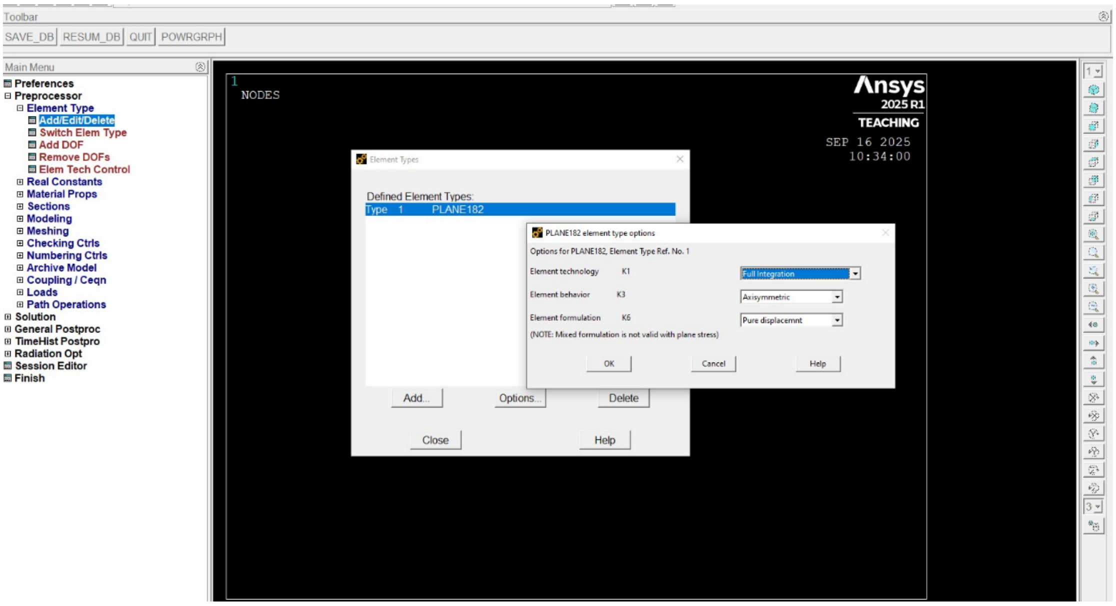
Sr. No	X – Coordinates	Y – Coordinates
1	84.836	0
2	135.636	0
3	84.836	88.9
4	50.8	152.4
5	50.8	254
6	84.836	254
7	84.836	190.5
8	135.636	88.9
9	0	88.9

STEP 1: SELECT ELEMENT TYPE AS STRUCTURAL

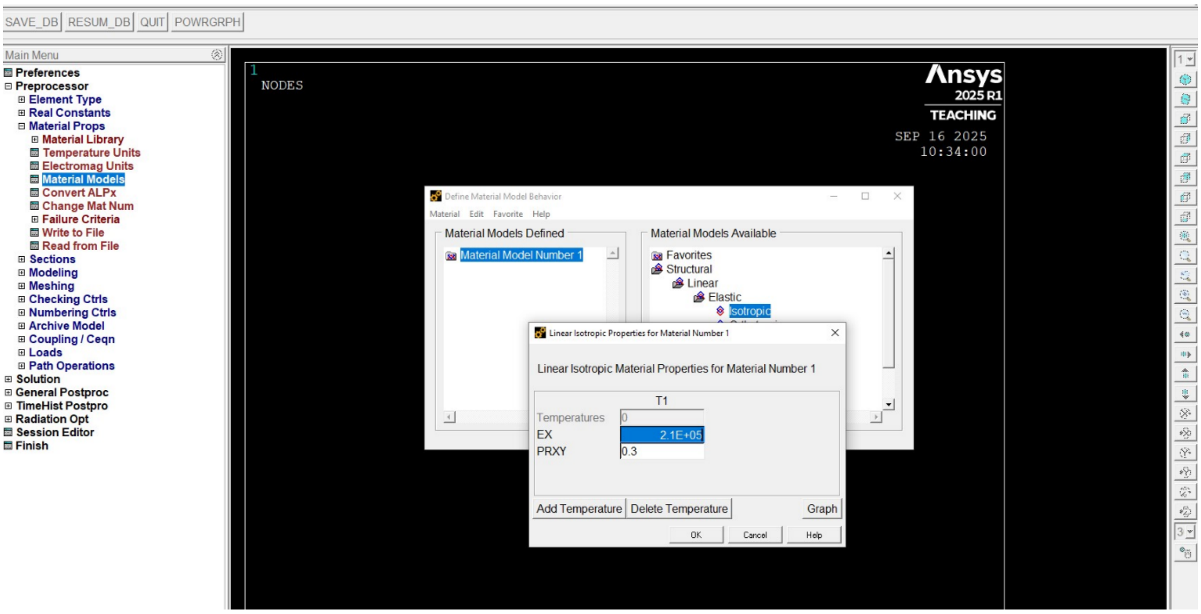


STEP 2 : Preprocessor > Define Element types > Add > Solid> Options>Element Behaviour=Axisymmetric> OK > Close





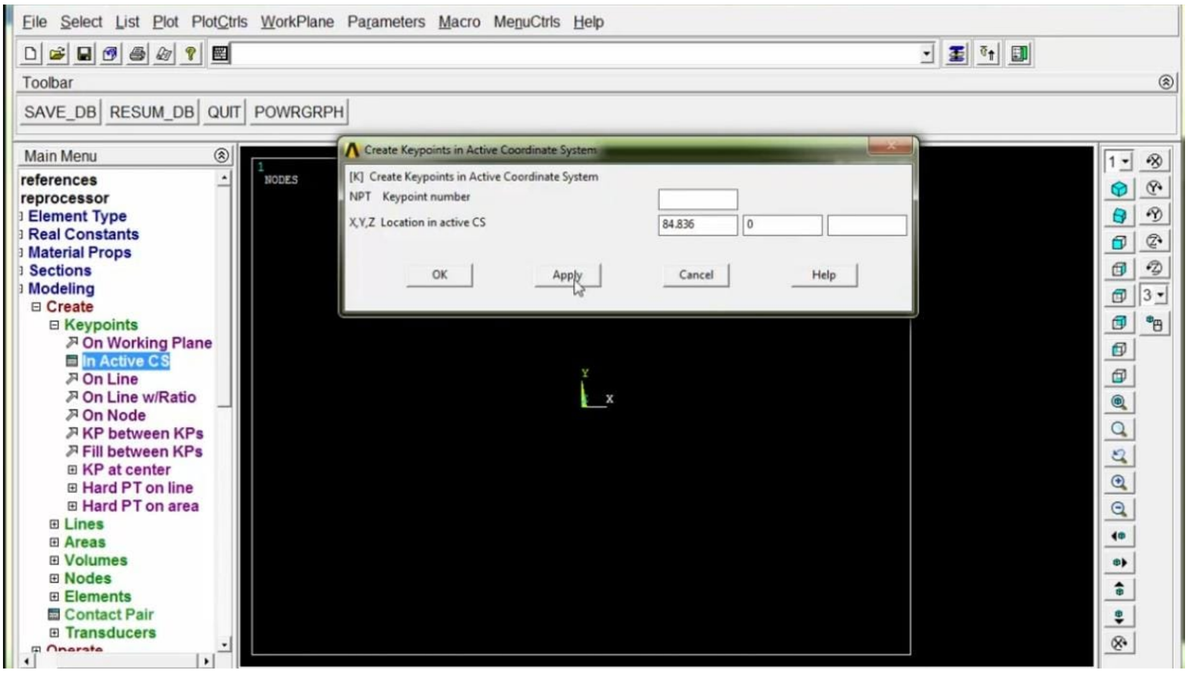
STEP 3: Preprocessor > Material Props > Material models > Structural > Linear > Elastics > Isotropic



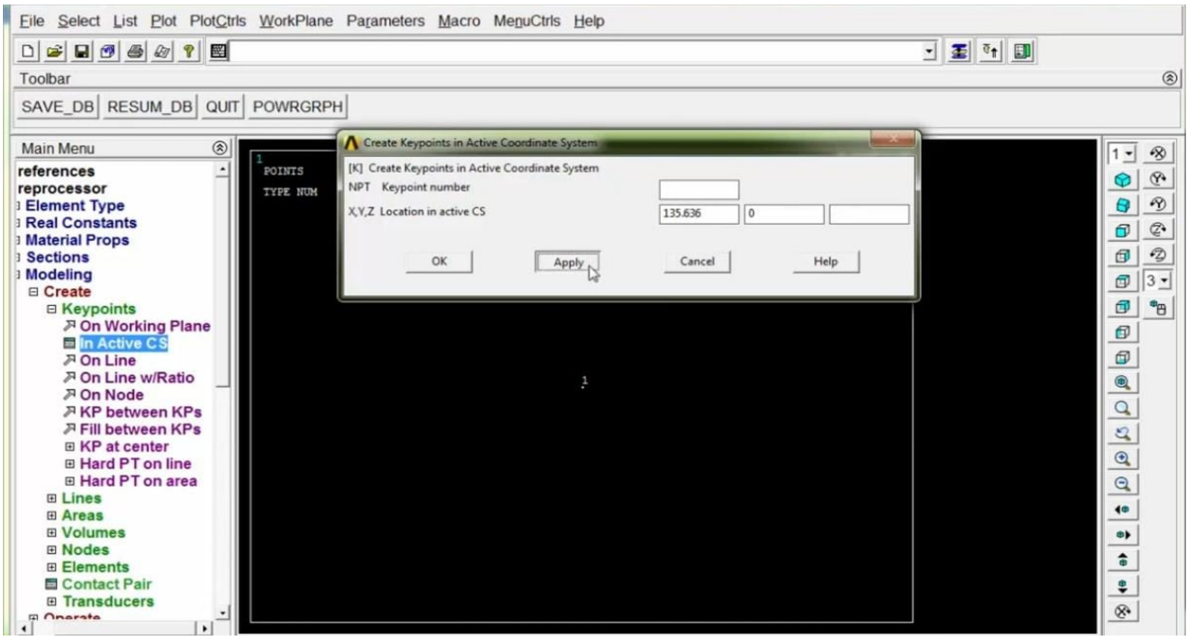
STEP 4 : Preprocessor > Modeling > Create > Keypoints > In Active CS

CHOOSE IN ACTIVE CS AND ADD COORDINATES

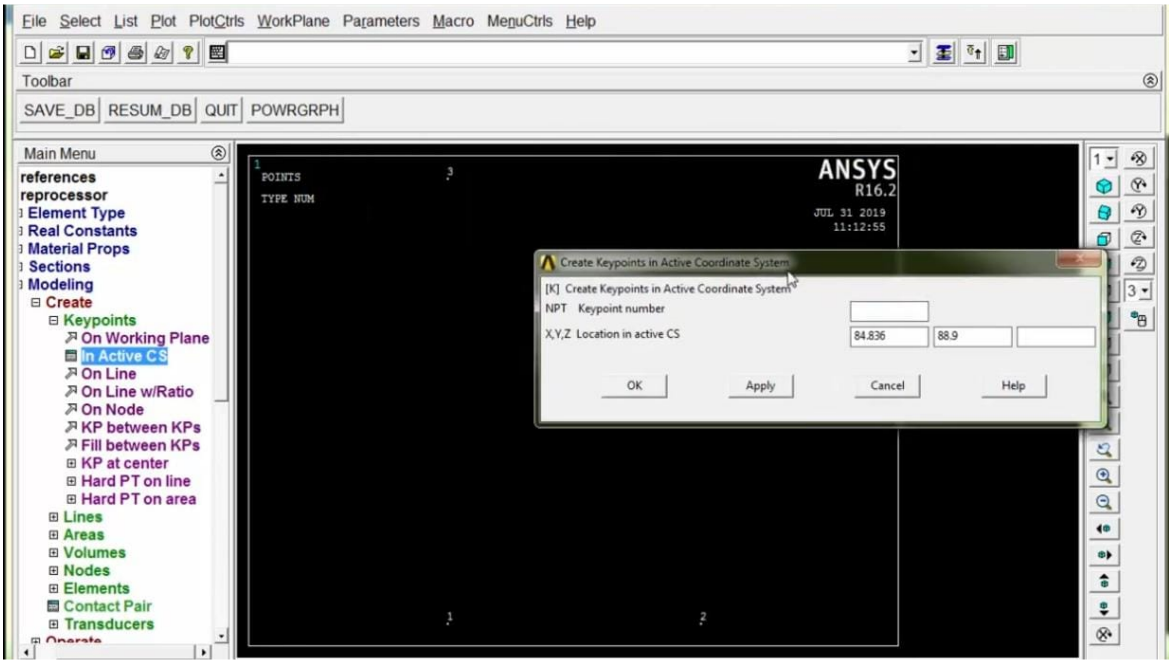
Modeling < Create < Nodes < In Active Cs < Node 1 < at X=84.836 , Y=0 , Z=0 < Apply



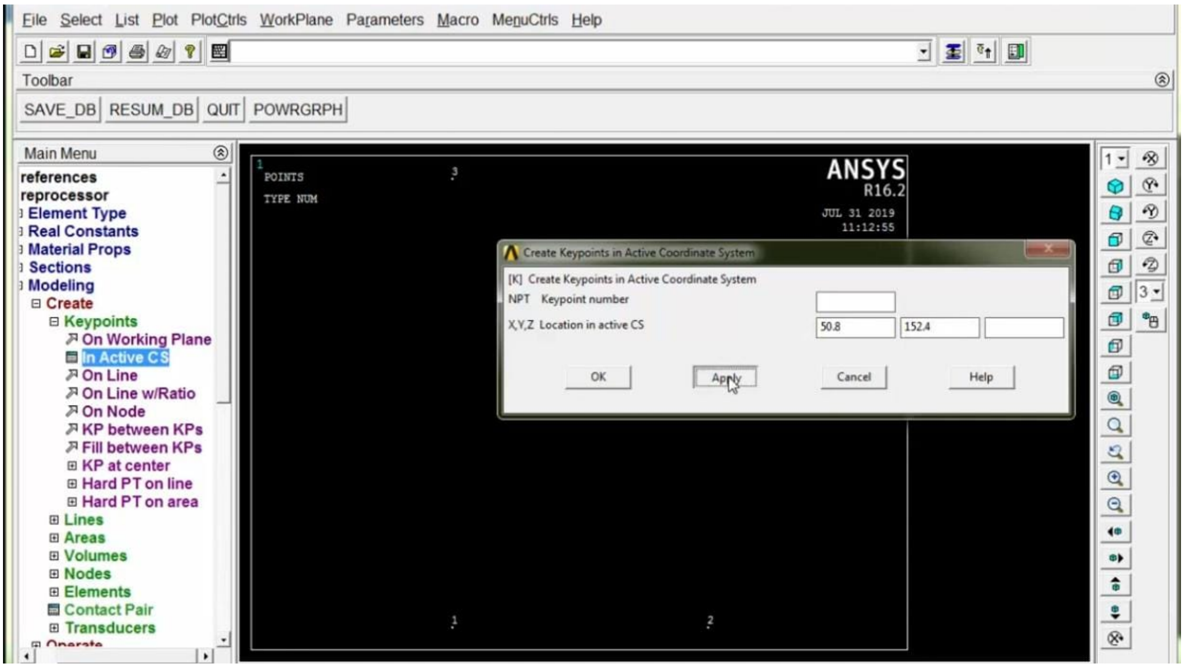
Modeling < Create < Nodes < In Active Cs < Node 2 < at X=135.636 , Y=0 , Z=0 < Apply



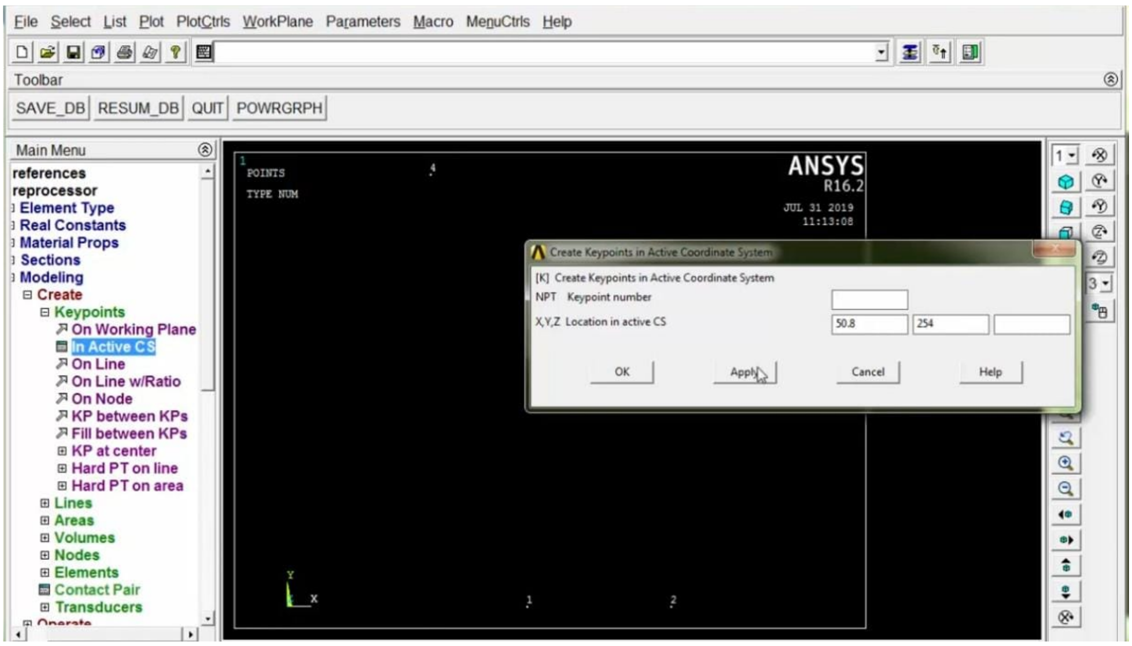
Modeling < Create < Nodes < In Active Cs < Node 3 < at X=84.836, Y=88.9 , Z=0 < Apply



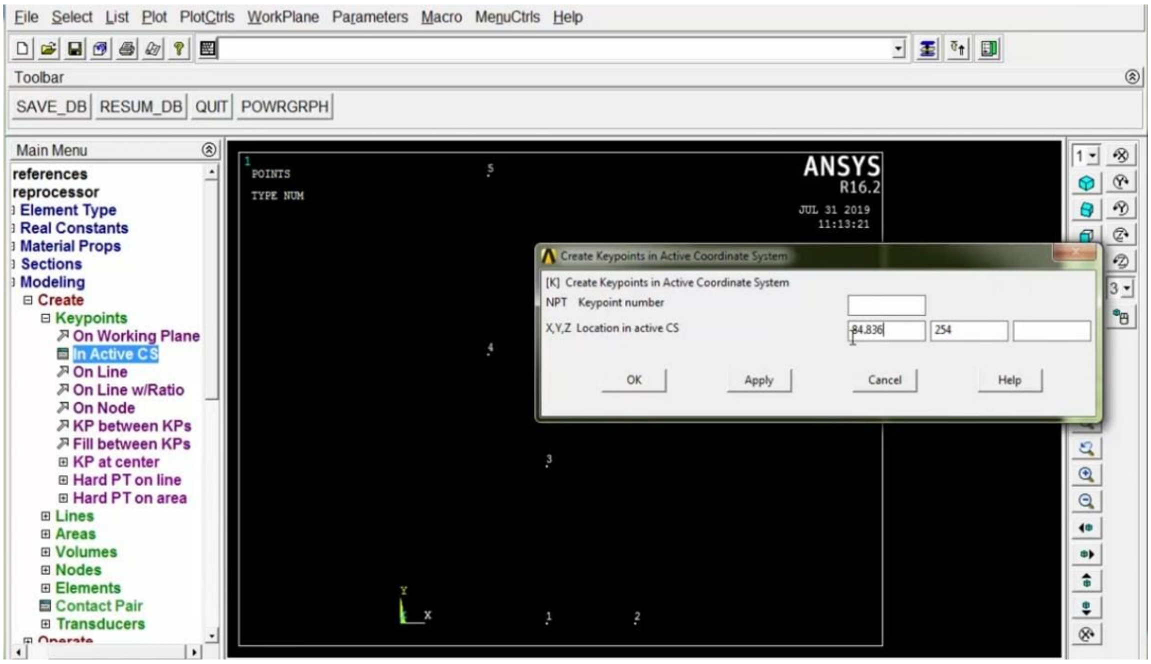
Modeling < Create < Nodes < In Active Cs < Node 4 < at X=50.8, Y=152.4, Z=0 < Apply



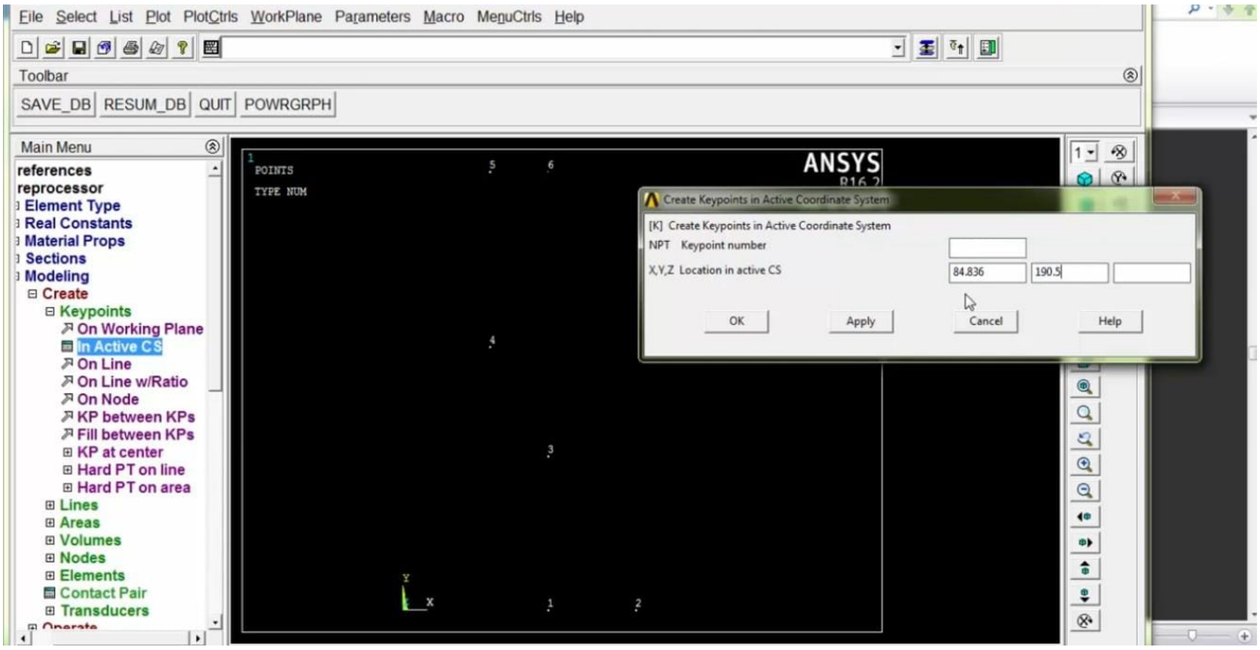
Modeling < Create < Nodes < In Active Cs < Node 5 < at X=58.8, Y=254, Z=0 < Apply



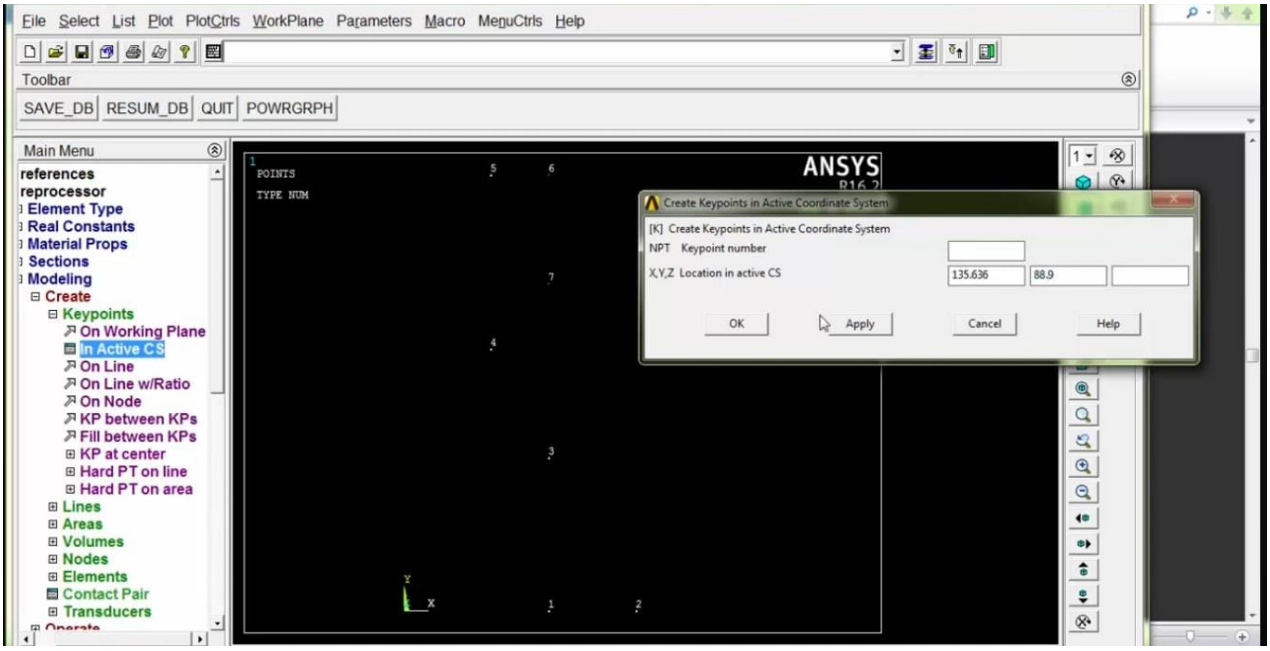
Modeling < Create < Nodes < In Active Cs < Node 6 < at X=84.836, Y=254, Z=0 < Apply



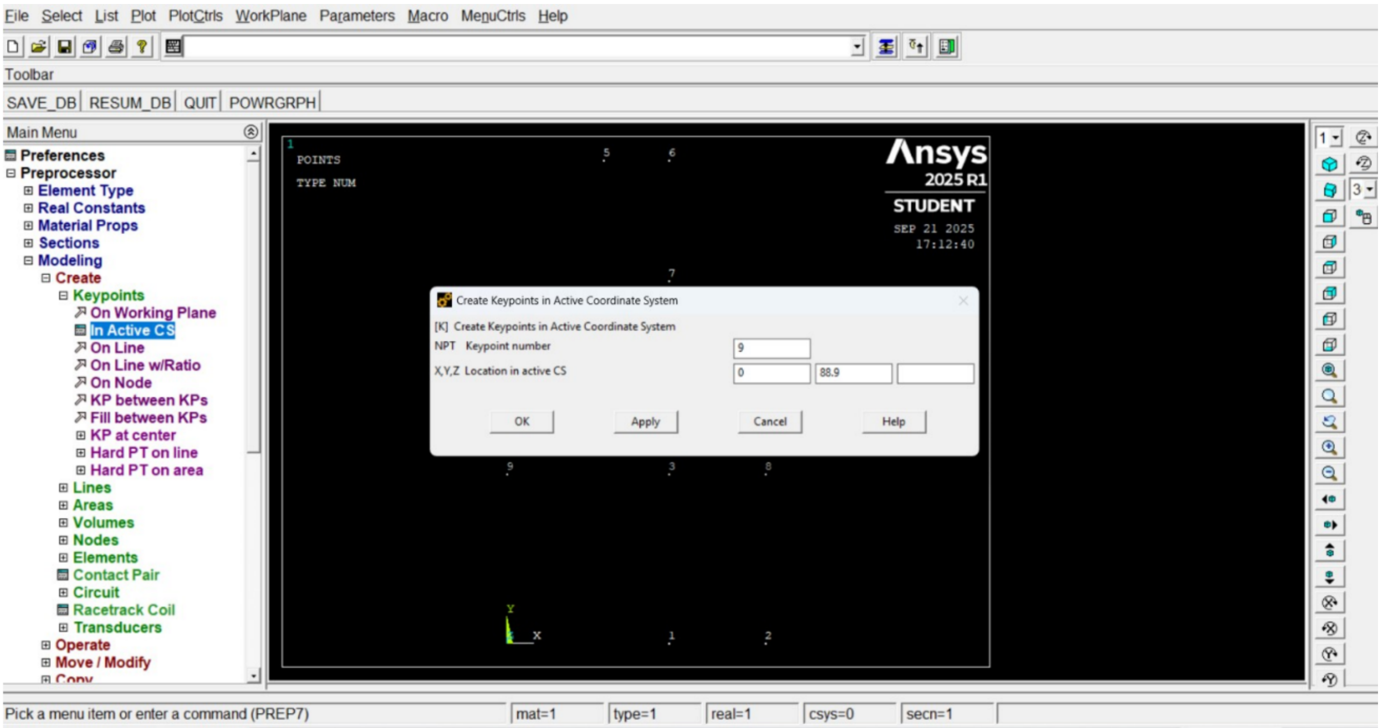
Modeling < Create < Nodes < In Active Cs < Node 7 < at X=84.836, Y=190.5, Z=0 < Apply



Modeling < Create < Nodes < In Active Cs < Node 8 < at X=135.636, Y=88.9, Z=0 < Apply

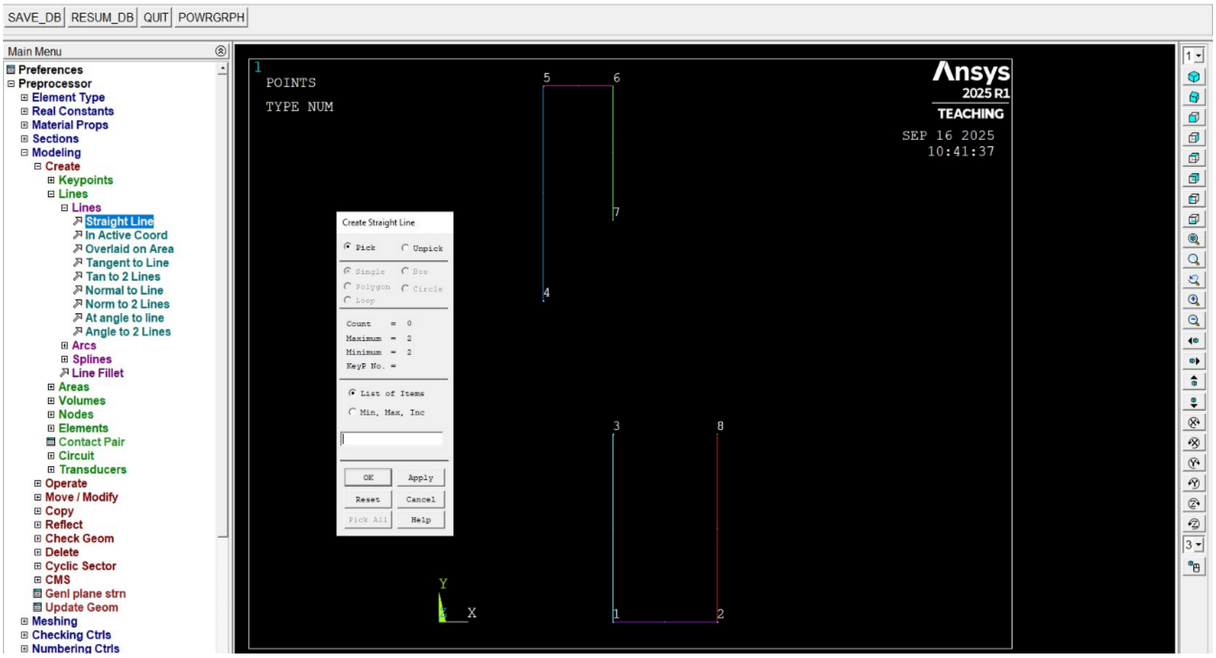


Modeling < Create < Nodes < In Active Cs < Node 9 < at X=0, Y=88.9, Z=0 < Apply

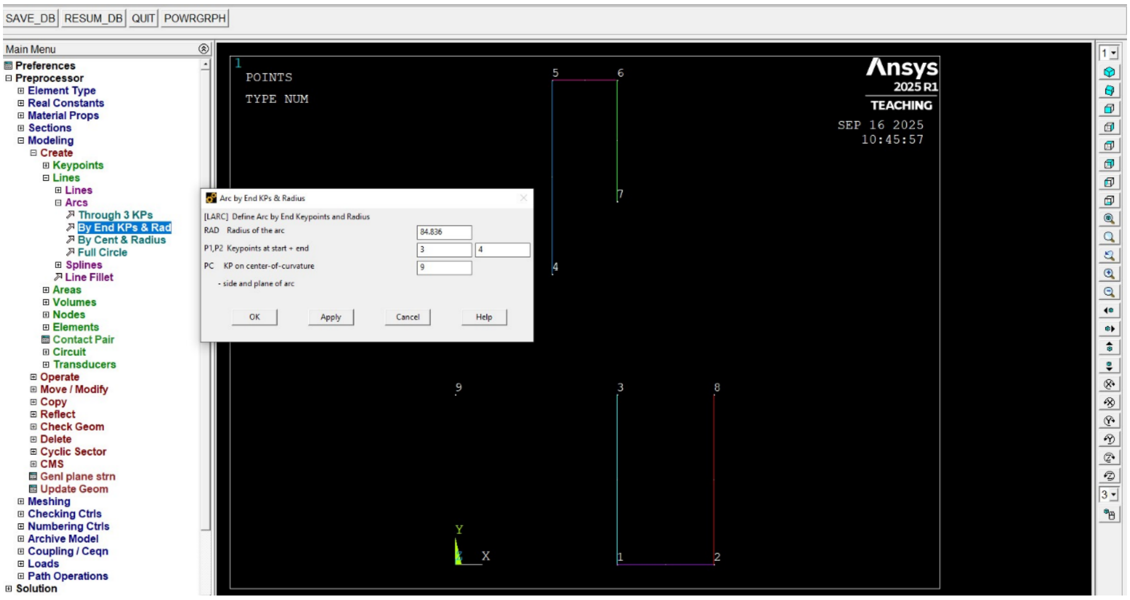


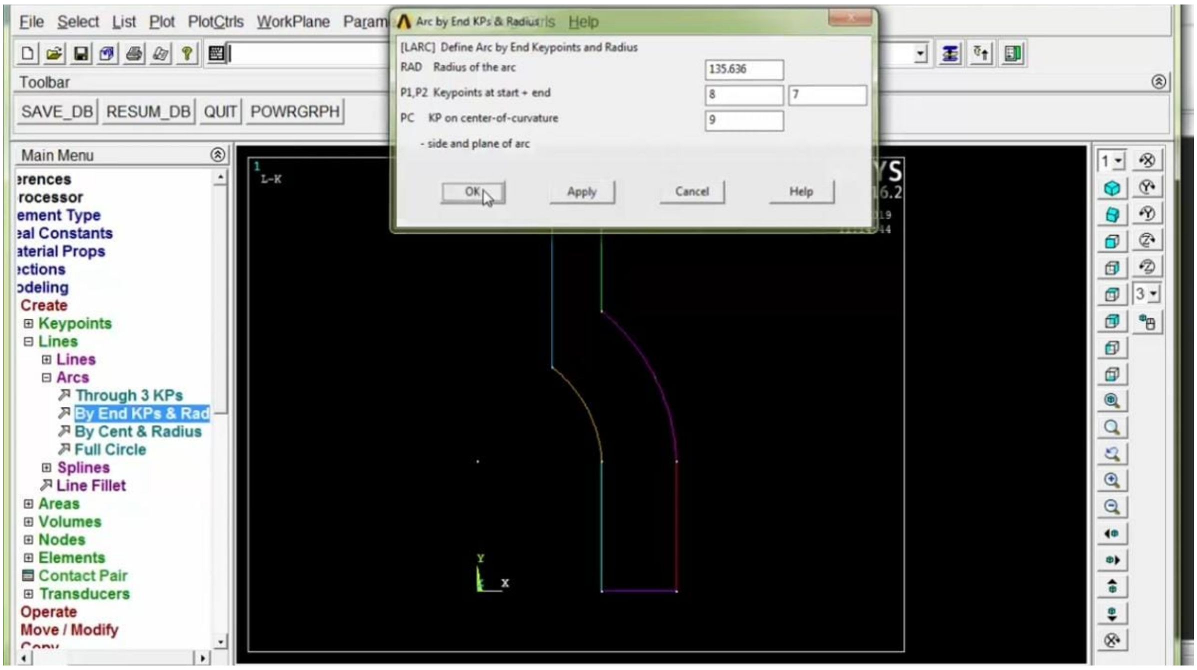
STEP 5 : CONNECT THE COORDINATE POINTS WITH LINES IN THE CREATE SECTION

PREPROCESSOR> MODELING> LINES> STRAIGHT LINES

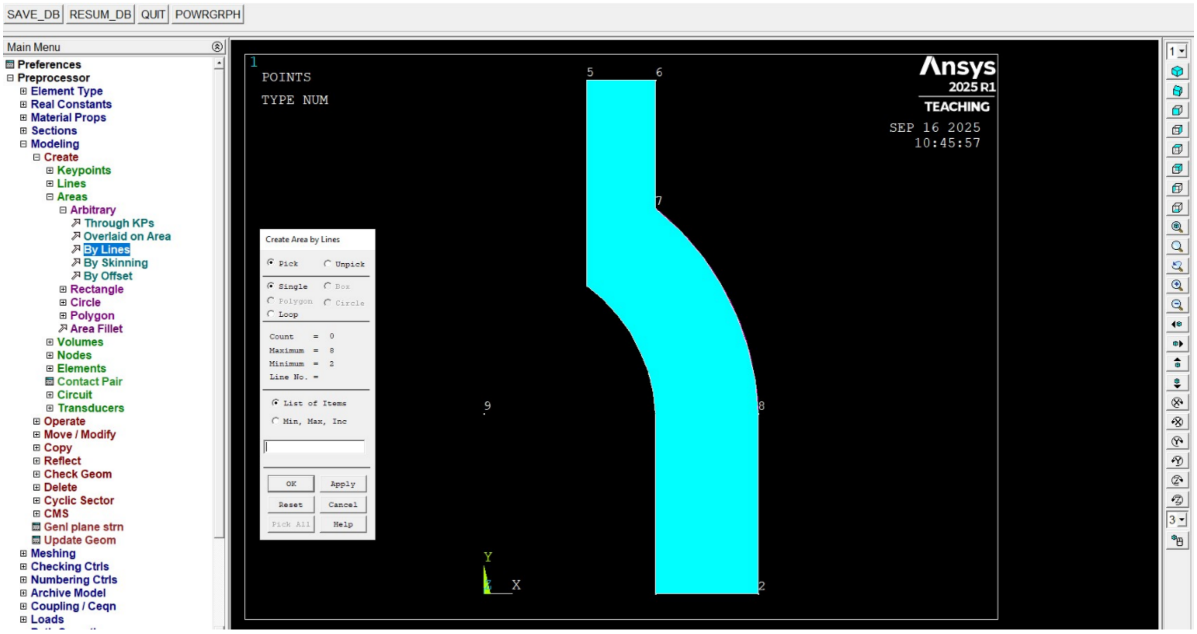


STEP 6: PREPROCESSOR > MODELING> LINES> ARC> BY END KPS AND RAD=84.836 AND 135.636 > APPLY

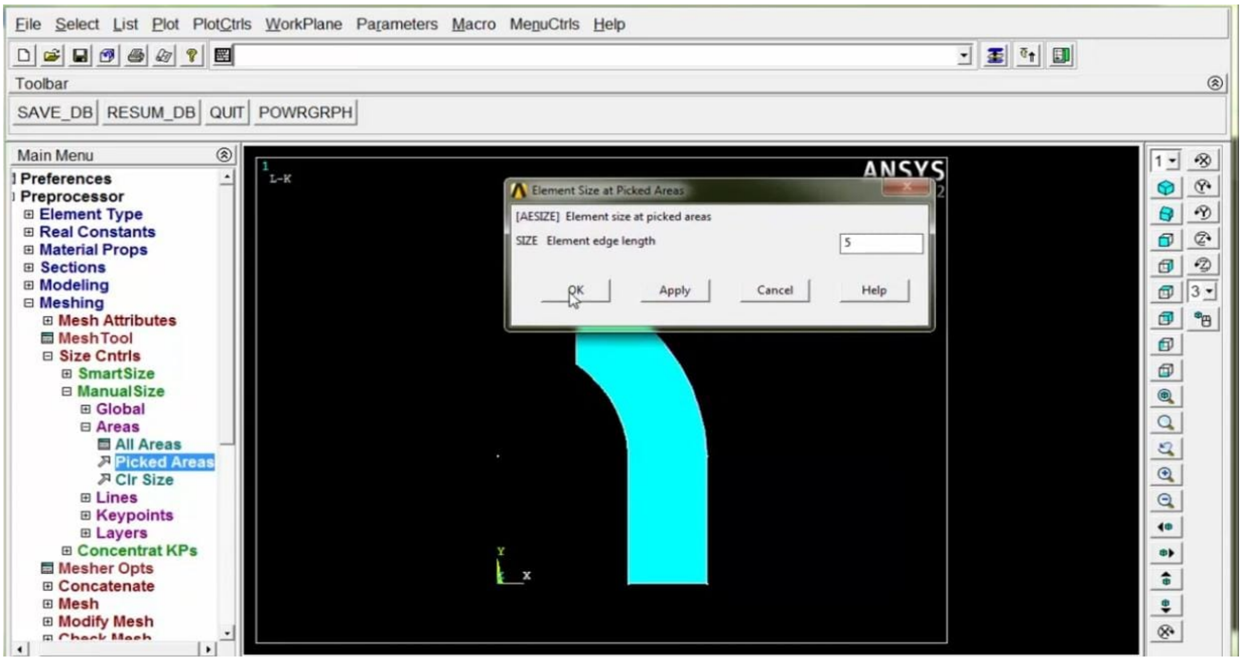




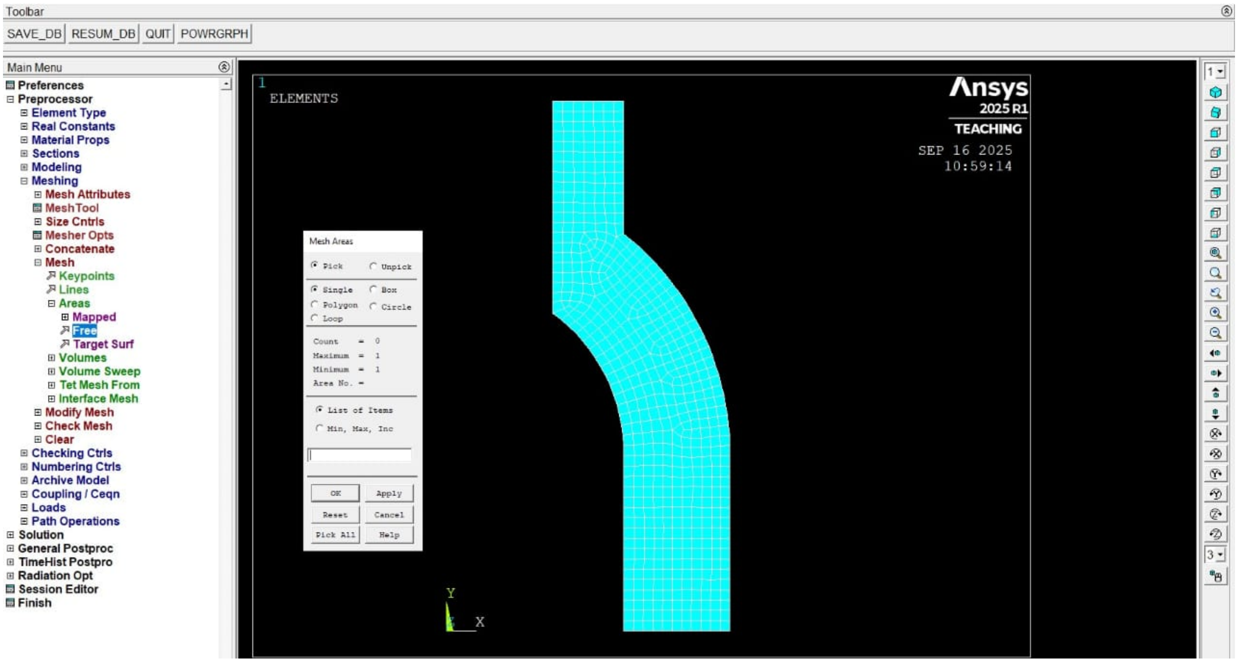
STEP 7: MODELING> CREATE> AREAS> ARBITRARY > BY LINES =SELECT ALL LINES> OK



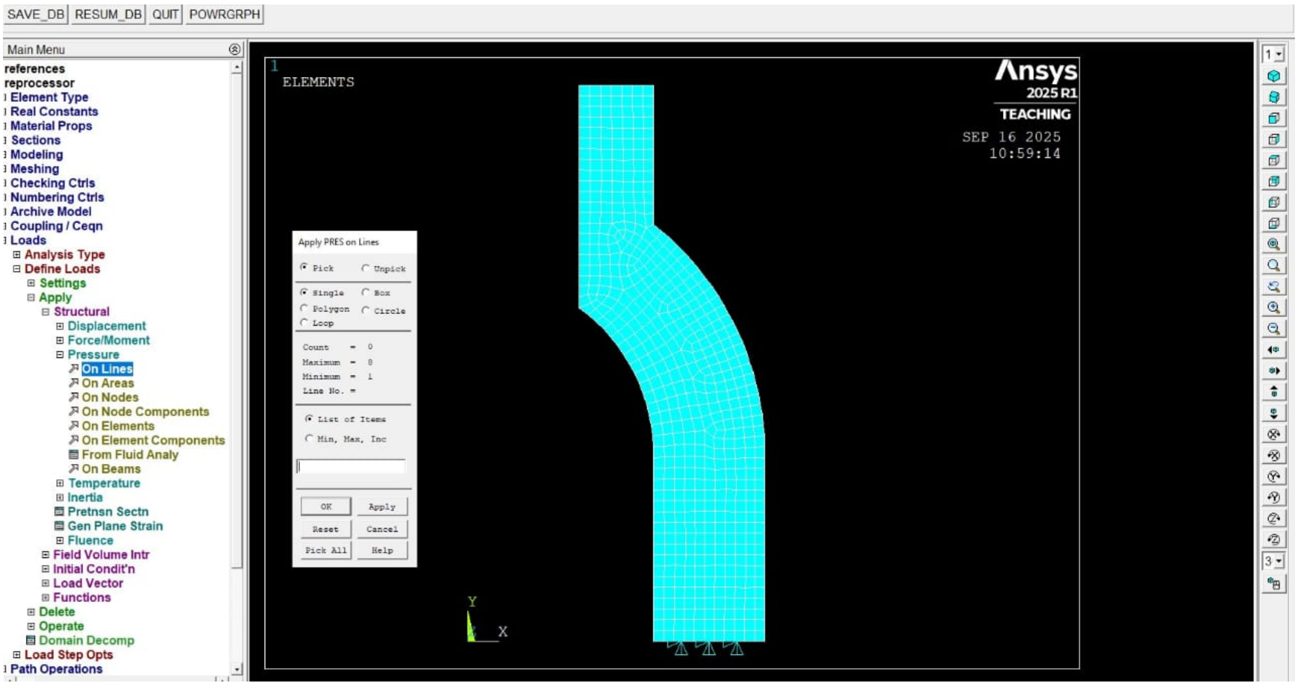
STEP 8: MESHING> SIZE CONTROLS > MANUAL SIZE> AREAS > PICKED AREAS



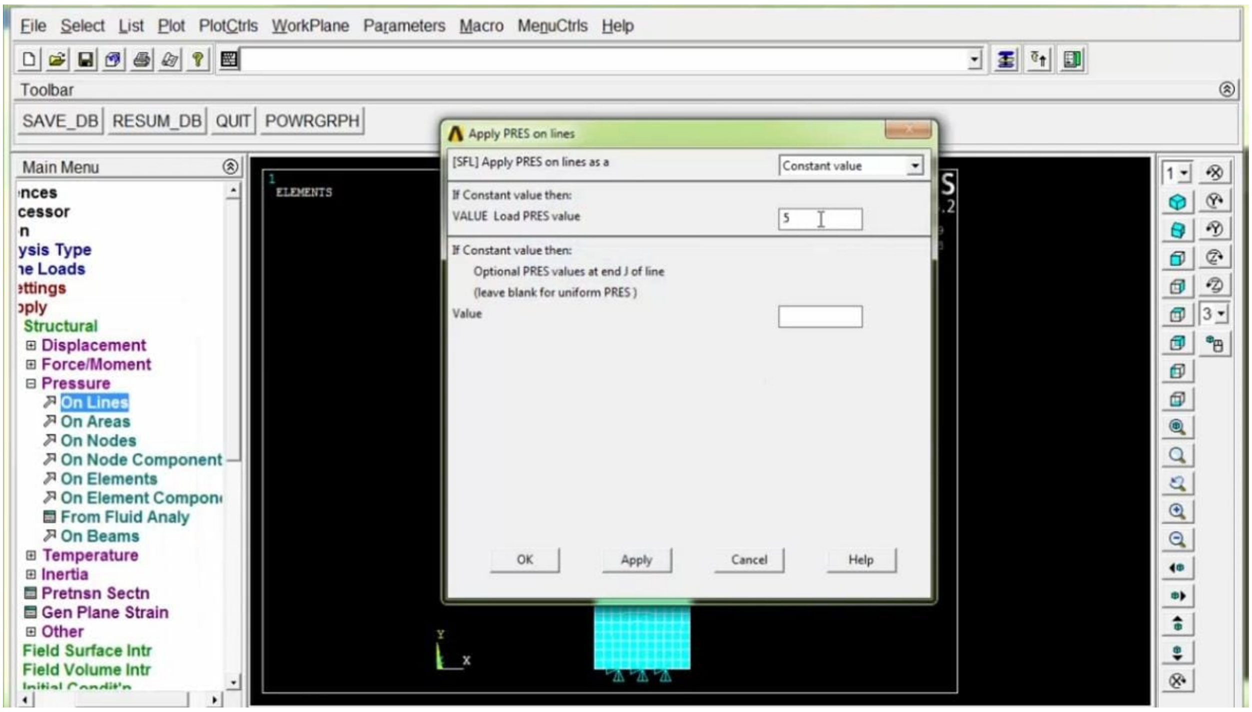
STEP 9: MESH> AREAS> FREE



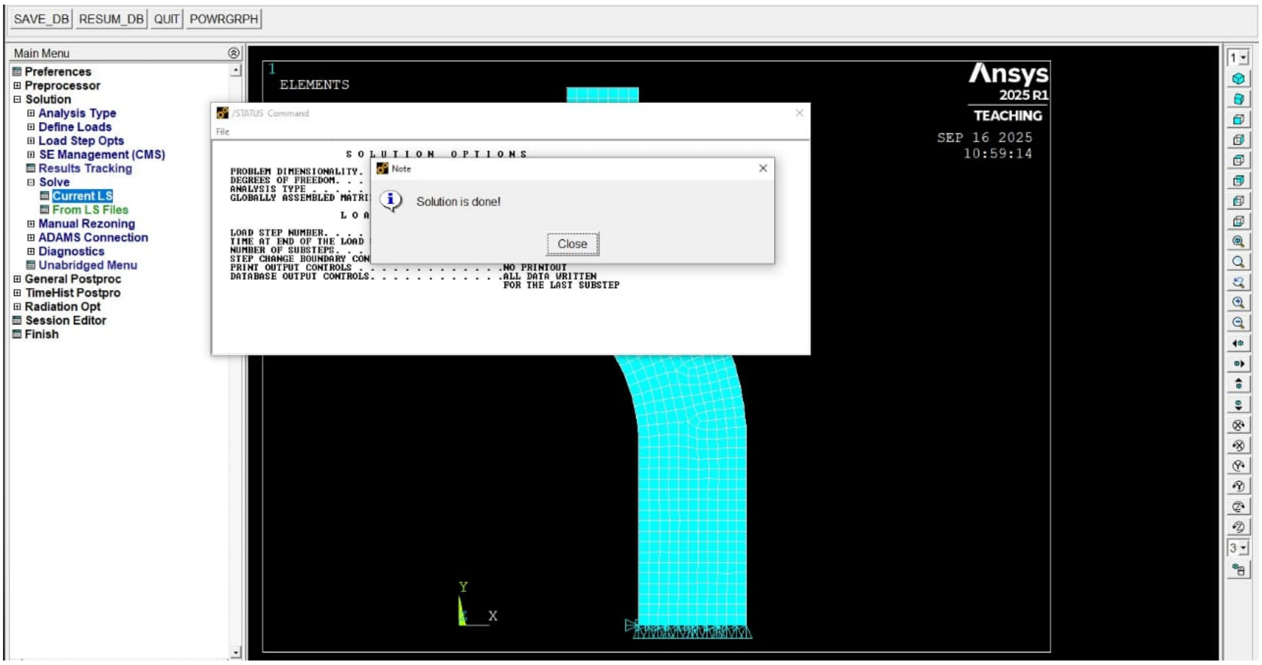
STEP 10 : SOLUTIONS> DEFINE LOADS> APPLY> DISPLACEMENT> ON LINES



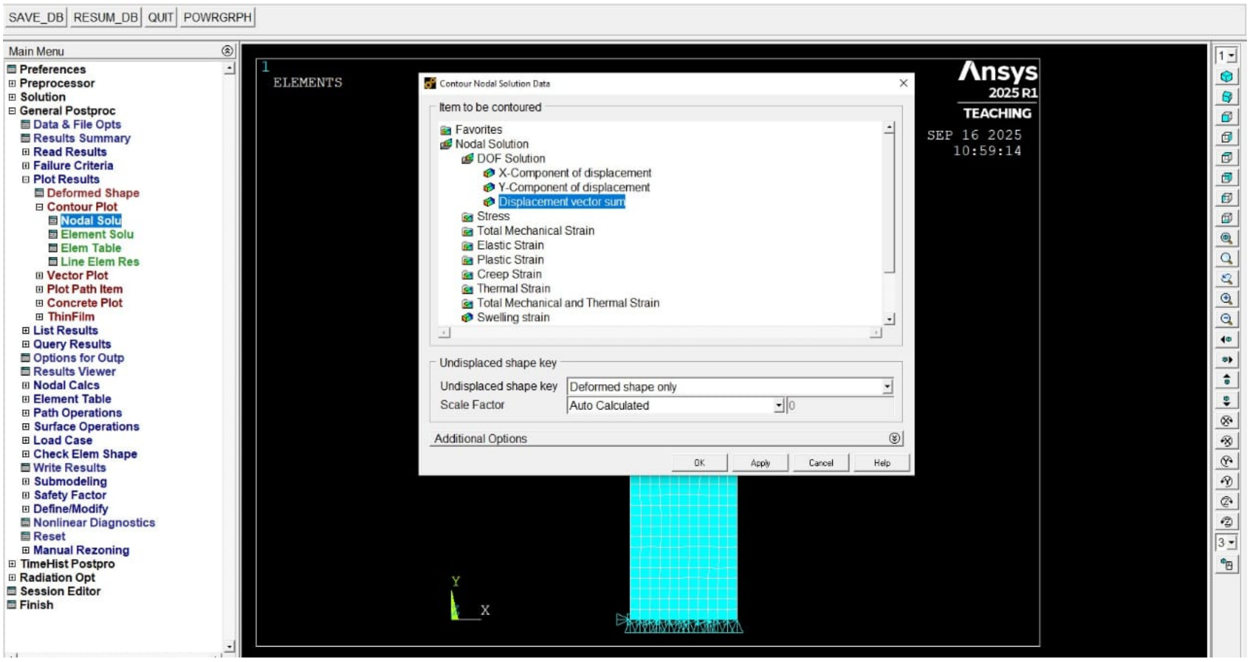
STEP 11: SOLUTIONS> DEFINE LOADS> APPLY> PRESSURE>ON LINE>OK



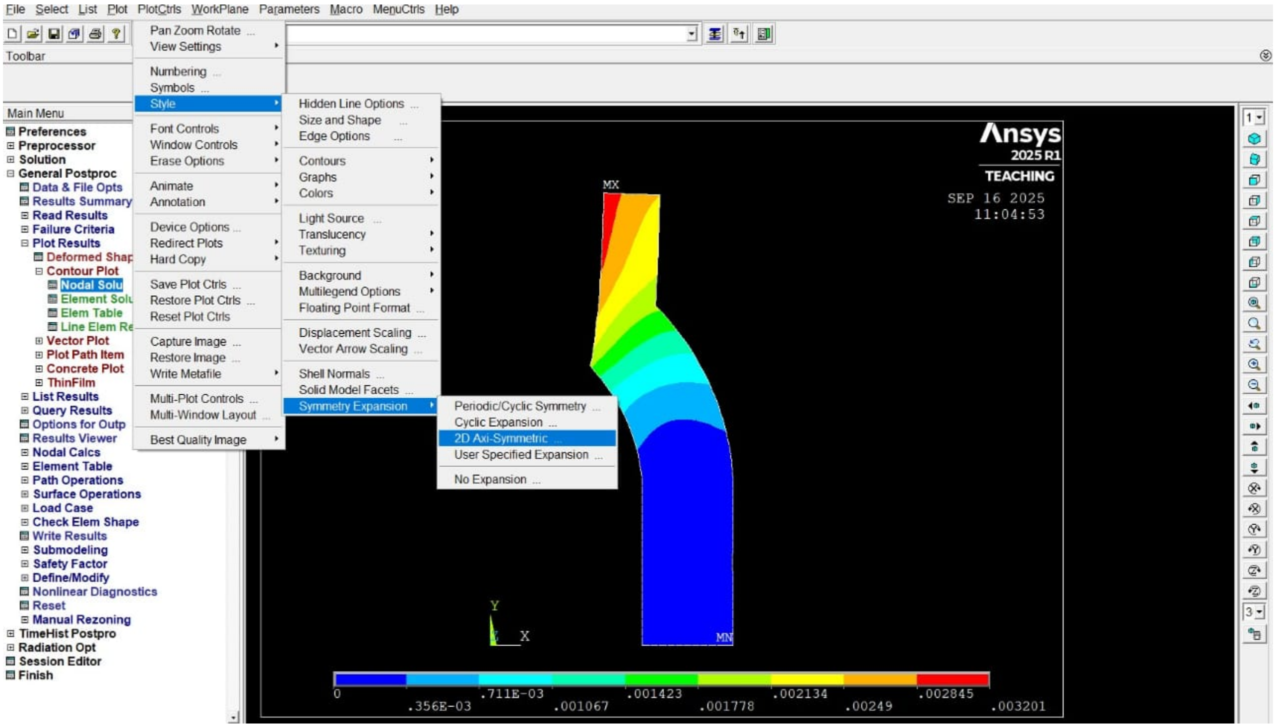
STEP 12: SOLVE> CURRENT LS



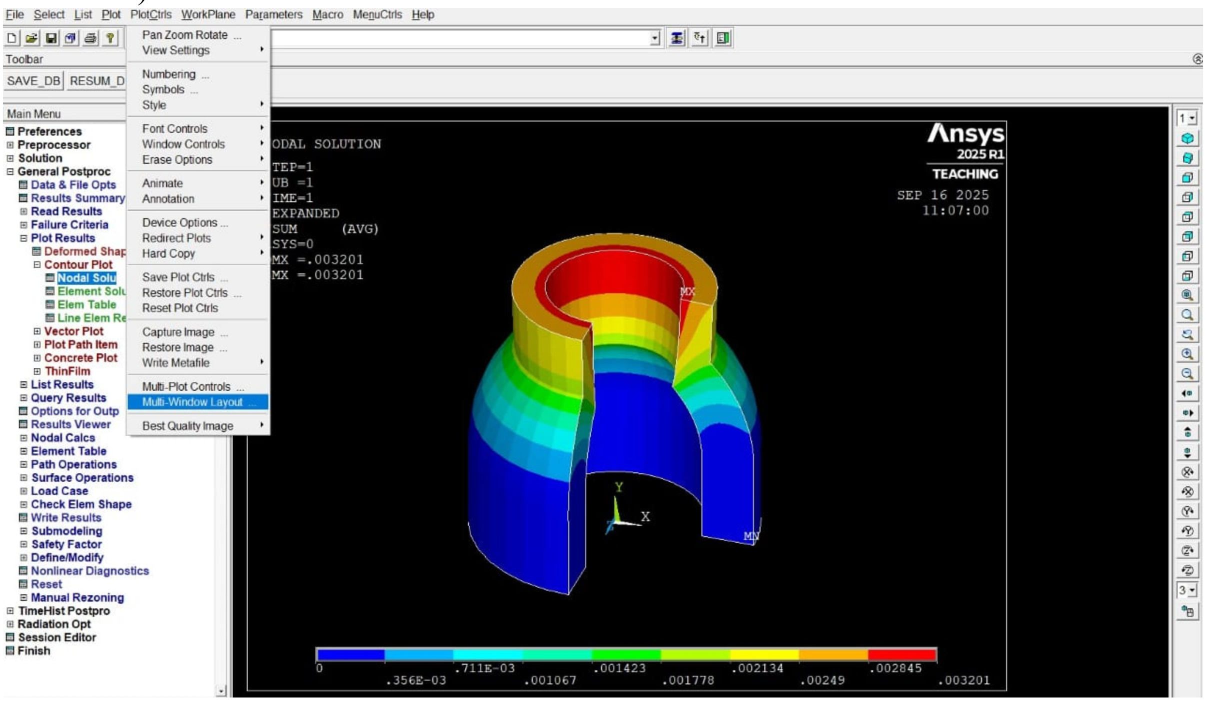
STEP 13: GENERAL POSTPROC>PLOT RESULTS> CONTOUR PLOT>
NODAL SOLUTIONS>
DISPLACEMENT VECTOR SUM> OK

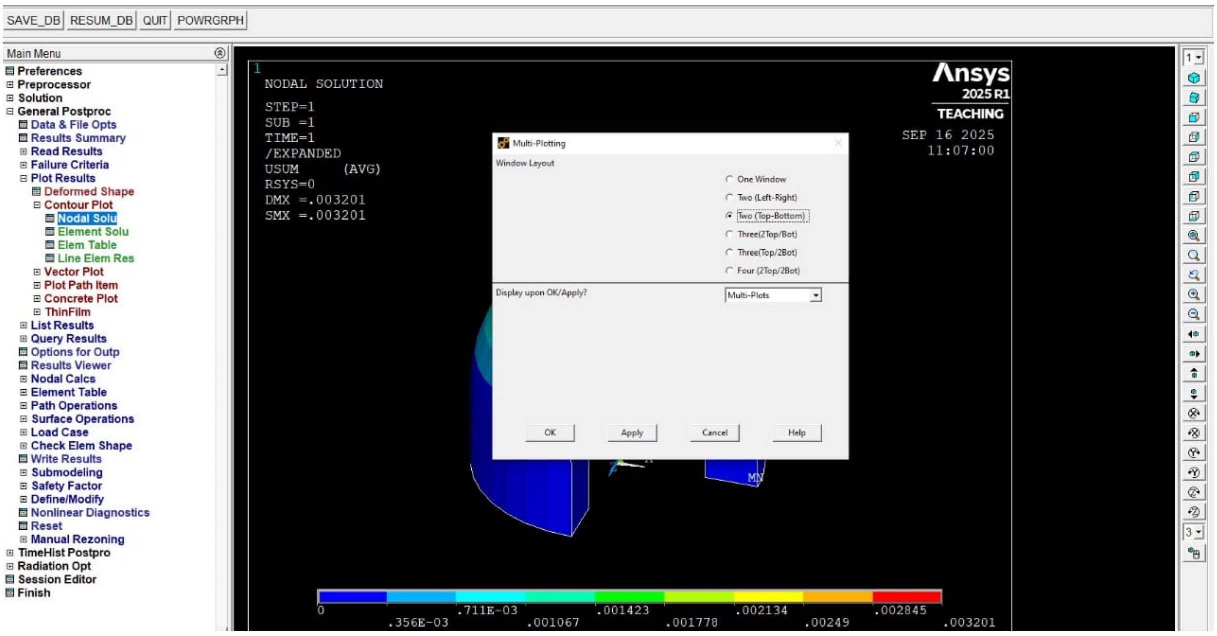


STEP 14: PLOTCTRLS> STYLE > SYMMETRIC EXPANSION> 2D AXI-SYMMETRIC>3/4 EXPANSION > OK

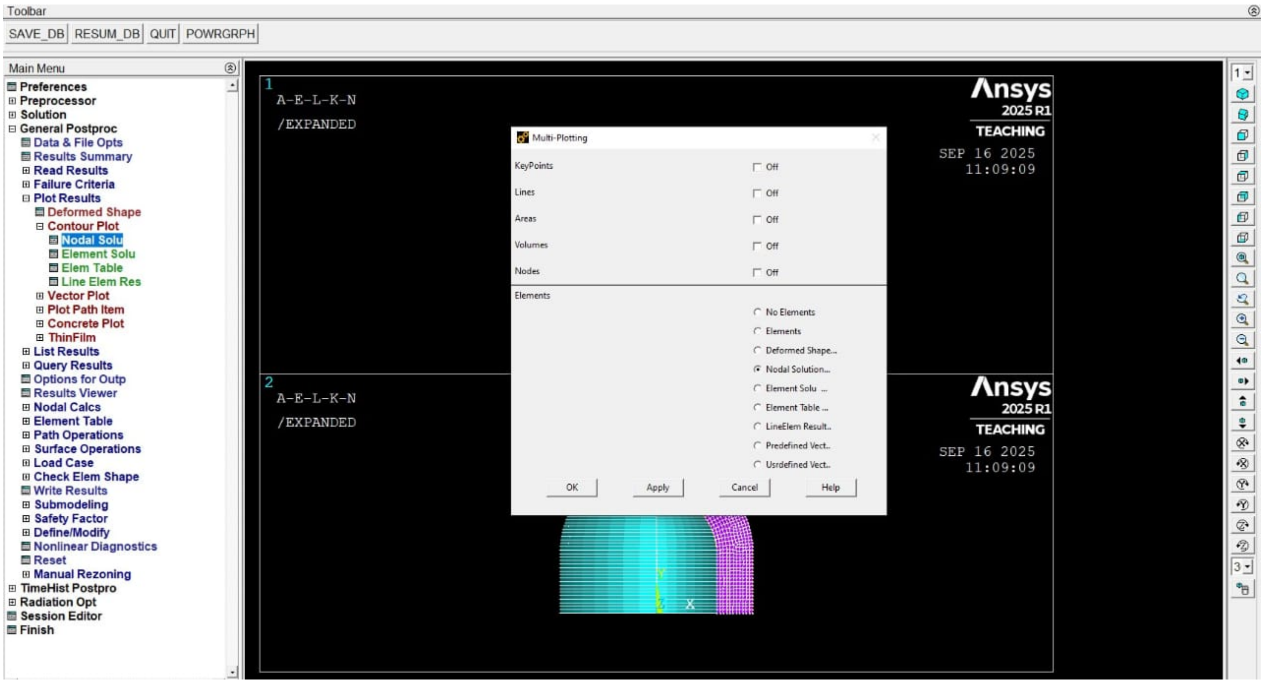


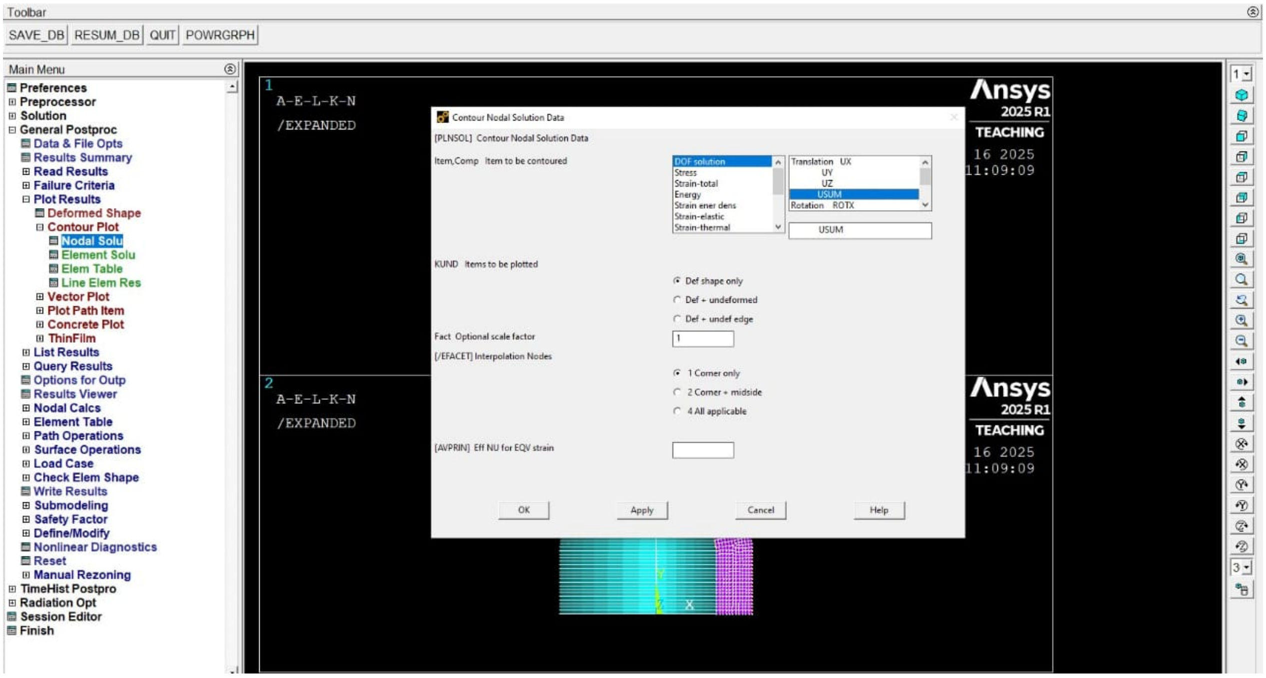
STEP 15: PLOTCTRLS> MULTIWINDOW LAYOUT> TWO(TOP-BOTTOM)>OK



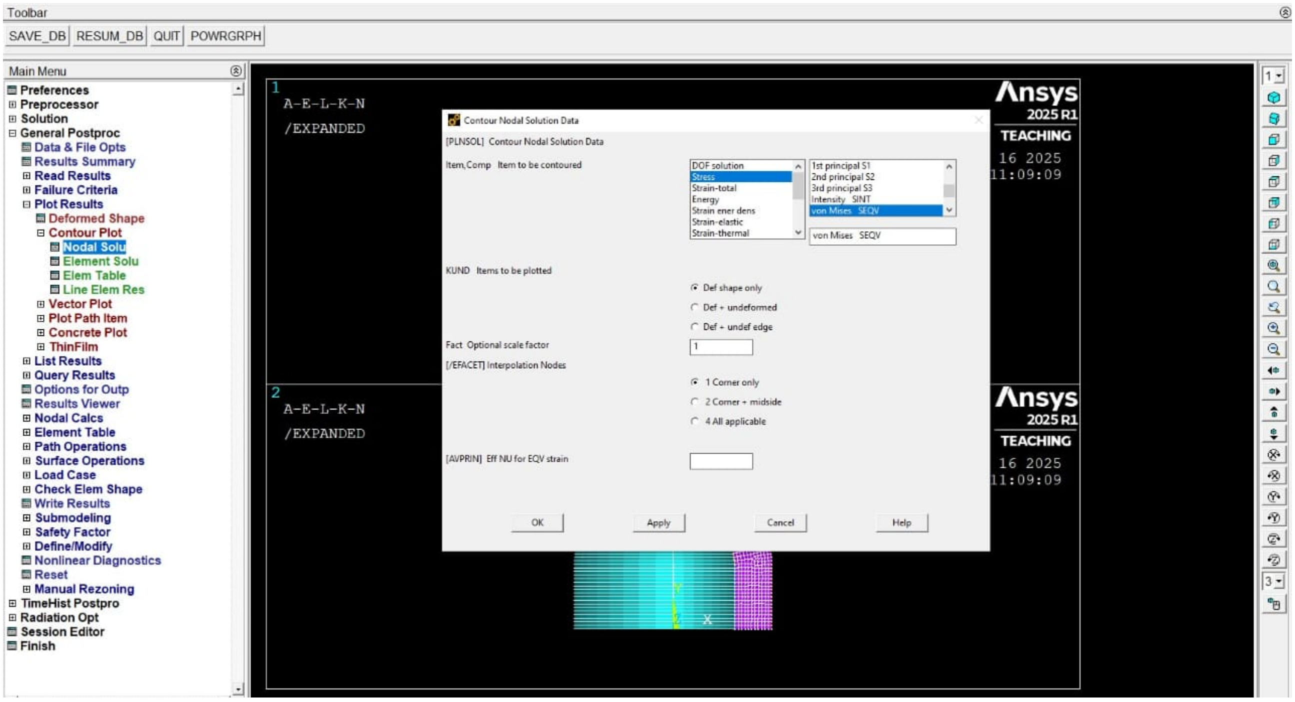


STEP 16: PLOTCTRLS> MULTIPLOT CONTROLS> WINDOW 1> OK > OFF EVERYTHING> NODAL SOLUTIONS > USUM> APPLY





STEP 17: PLOTCTRLS> MULTIPLOT CONTROLS> WINDOW 2> OK > OFF EVERYTHING> NODAL SOLUTIONS > STRESS> von MISES SEQV> APPLY



STEP 18 : START ANIMATION AND FINAL RESULT IS OBTAINED