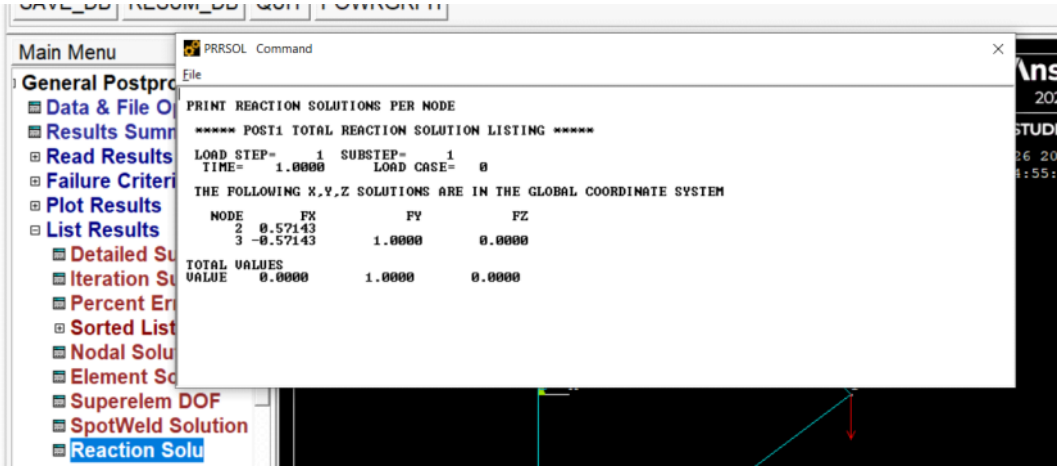
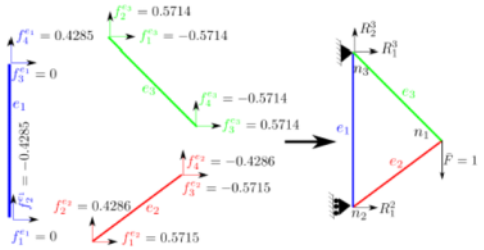
2021/08/26Ansys

Thursday, August 26, 2021 2:56 PM

Command line for truss forces, example in the class
F,1,FY,-1.0



Truss Example: Reaction Forces



- First, we compute reaction forces by adding up forces from individual elements that contribute to reaction forces:

$$R_1^2 = f_1^{e_1} + f_1^{e_2} = 0 + 0.5715 = 0.5715 \quad (397a)$$
$$R_1^3 = f_3^{e_1} + f_1^{e_3} = 0 + -0.5714 = -0.5714 \quad (397b)$$
$$R_2^3 = f_4^{e_1} + f_2^{e_3} = 0.4285 + 0.5714 = 0.9999 \quad (397c)$$

325 / 456

Nodal displacements

PRINT U NODAL SOLUTION PER NODE

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

LOAD STEP= 1 SUBSTEP= 1
TIME= 1.0000 LOAD CASE= 0

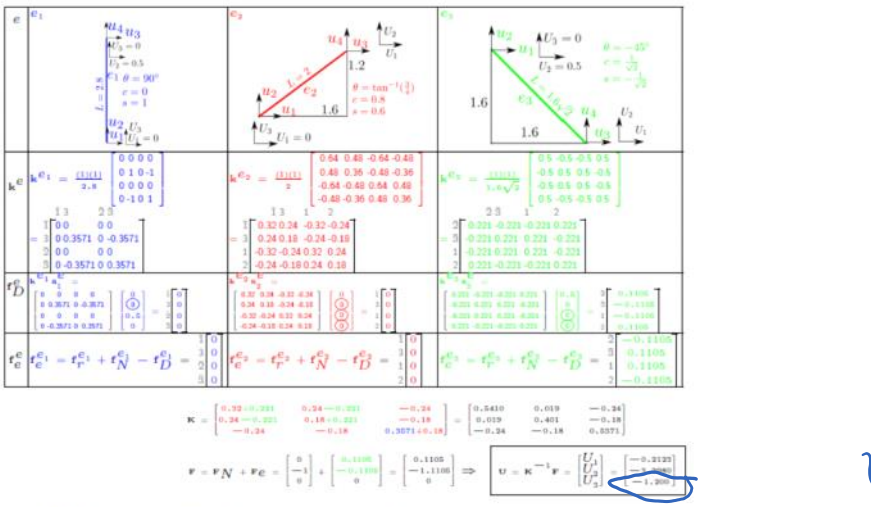
THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN THE GLOBAL COORDINATE SYSTEM

NODE	UX	UY	UZ	USUM
1	-0.42641E-004	-0.30124E-003	0.0000	0.30424E-003
2	0.0000	-0.12000E-003	0.0000	0.12000E-003
3	0.0000	0.0000	0.0000	0.0000

MAXIMUM ABSOLUTE VALUES

3 0.0000 0.0000 0.0000 0.0000

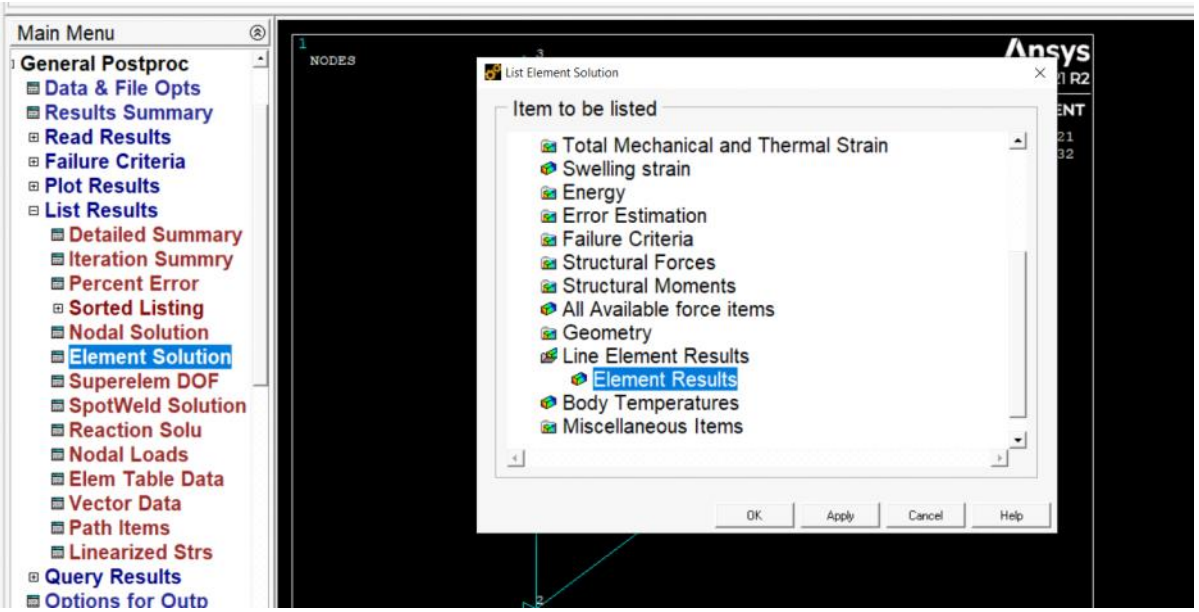
MAXIMUM ABSOLUTE VALUES
NODE 1 1 0 1
VALUE -0.42641E-004-0.30124E-003 0.0000 0.30424E-003



- Numbers encircled in the computation of essential BC force are displacements corresponding to free dofs. As mentioned before, in reality we do not consider them in computation of this force, but in hand calculation we just put zero for those values.

323 / 456

Element forces



PRINT ELEM ELEMENT SOLUTION PER ELEMENT

***** POST1 ELEMENT SOLUTION LISTING *****

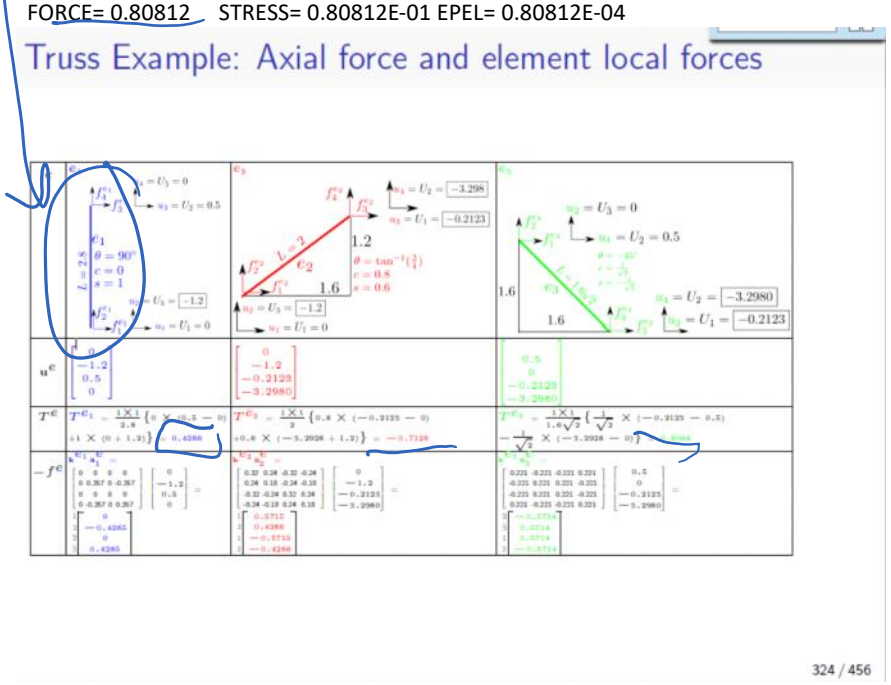
LOAD STEP 1 SUBSTEP= 1
TIME= 1.0000 LOAD CASE= 0

EL= 1 NODES= 3 2 MAT= 1 XC,YC,ZC= 0.000 0.2000 0.000 AREA= 10.000 LINK180
FORCE= 0.42857 STRESS= 0.42857E-01 EPEL= 0.42857E-04
TEMP= 0.00 0.00 EPTH= 0.0000

EL= 2 NODES= 2 1 MAT= 1 XC,YC,ZC= 0.8000 -0.6000 0.000 AREA= 10.000 LINK180
FORCE=-0.71429 STRESS=-0.71429E-01 EPEL=-0.71429E-04
TEMP= 0.00 0.00 EPTH= 0.0000

EL= 2 NODES= 2 1 MAT= 1 XC,YC,ZC= 0.8000 -0.6000 0.000 AREA= 10.000 LINK180
FORCE=-0.71429 STRESS=-0.71429E-01 EPEL=-0.71429E-04
TEMP= 0.00 0.00 EPTH= 0.0000

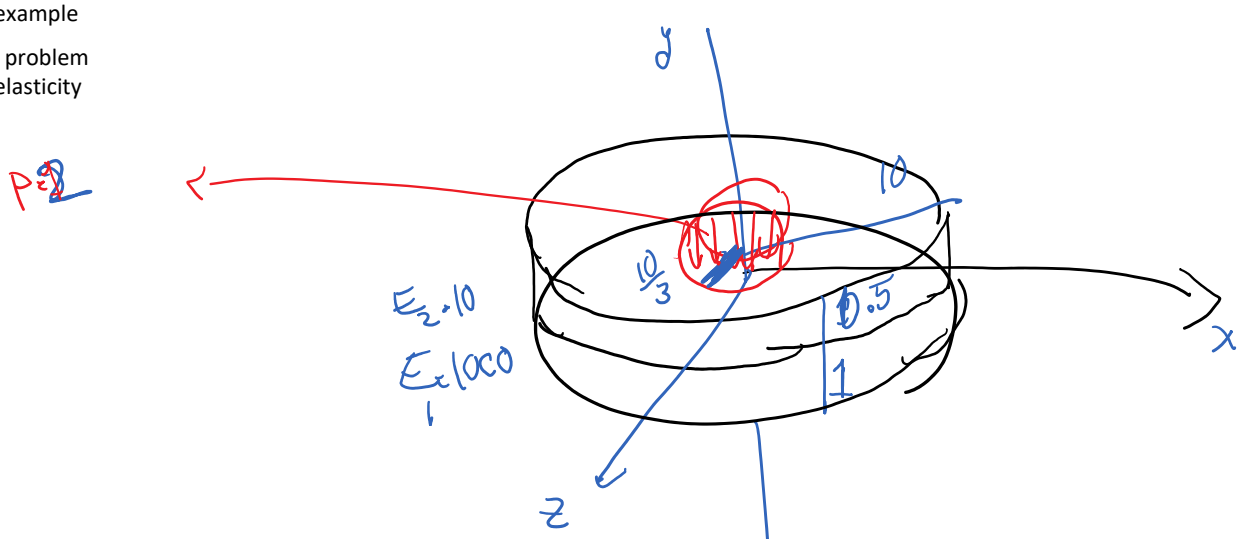
EL= 3 NODES= 1 3 MAT= 1 XC,YC,ZC= 0.8000 0.8000 0.000 AREA= 10.000 LINK180
FORCE= 0.80812 STRESS= 0.80812E-01 EPEL= 0.80812E-04



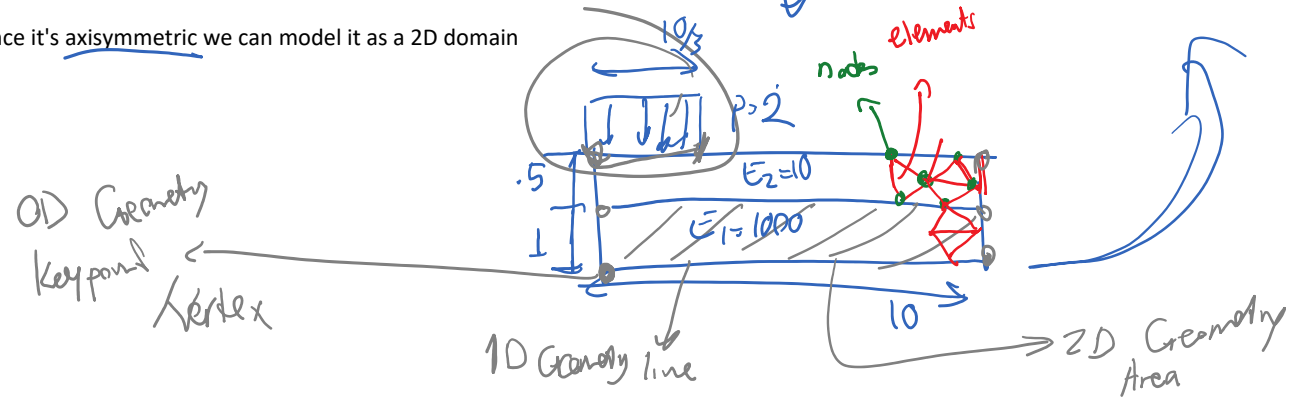
TEMP= 0.00 0.00 EPTH= 0.0000

2D example

2nd problem
2D elasticity

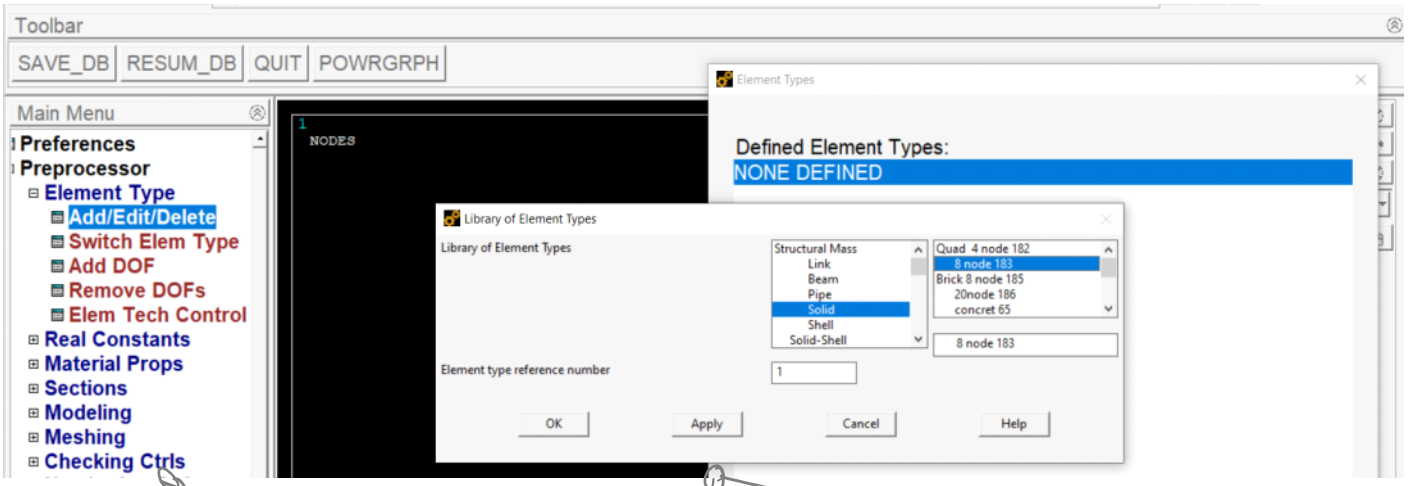


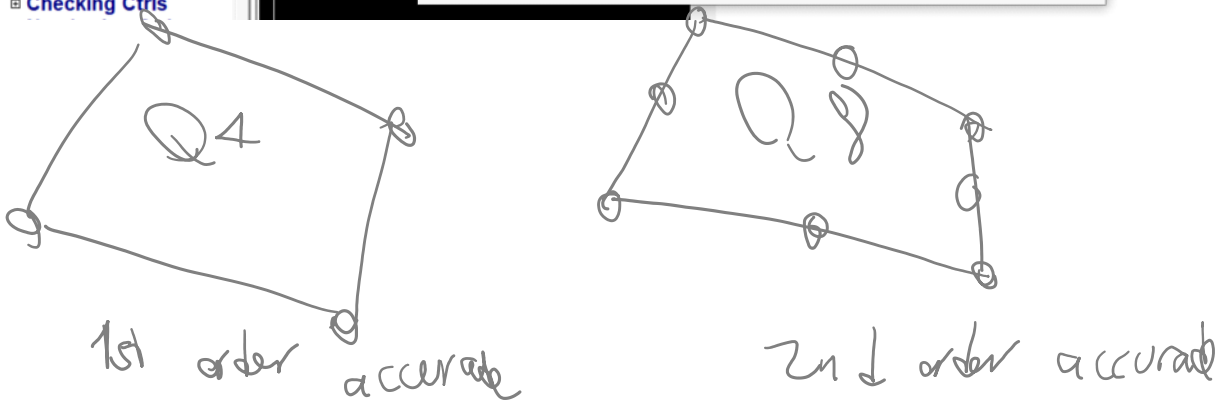
Since it's axisymmetric we can model it as a 2D domain

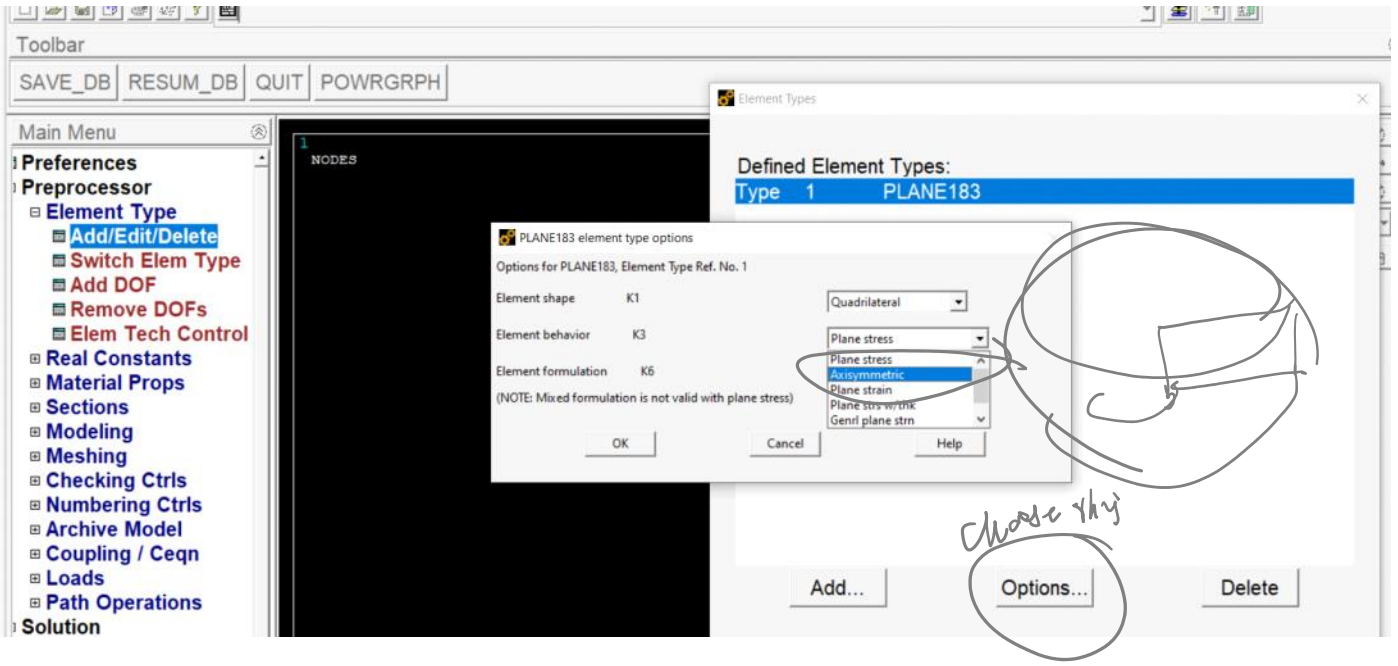


Problems with 2D / 3D elements:
We create the geometry and apply BC (supports, loads), … on geometry not on element nodes

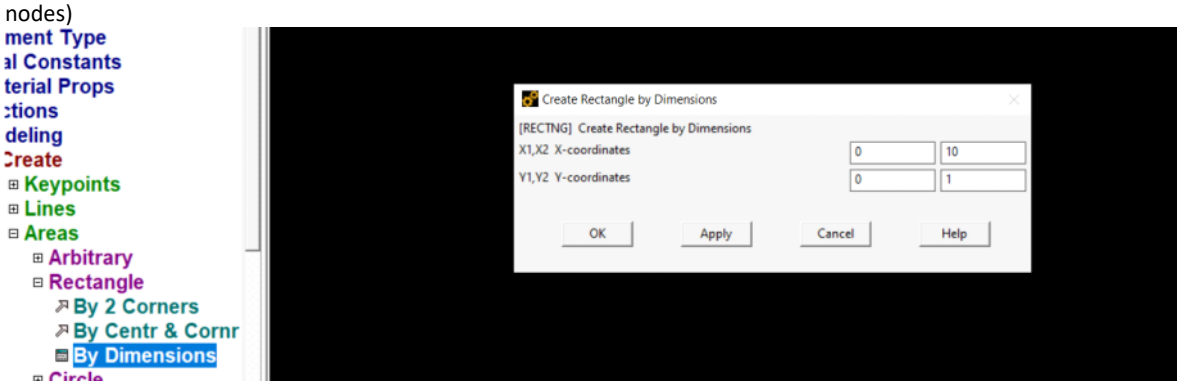
Define the element

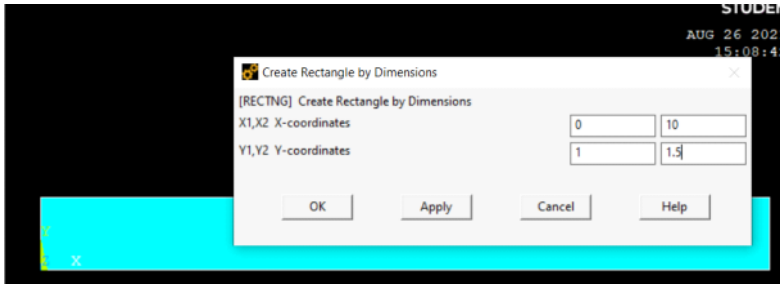




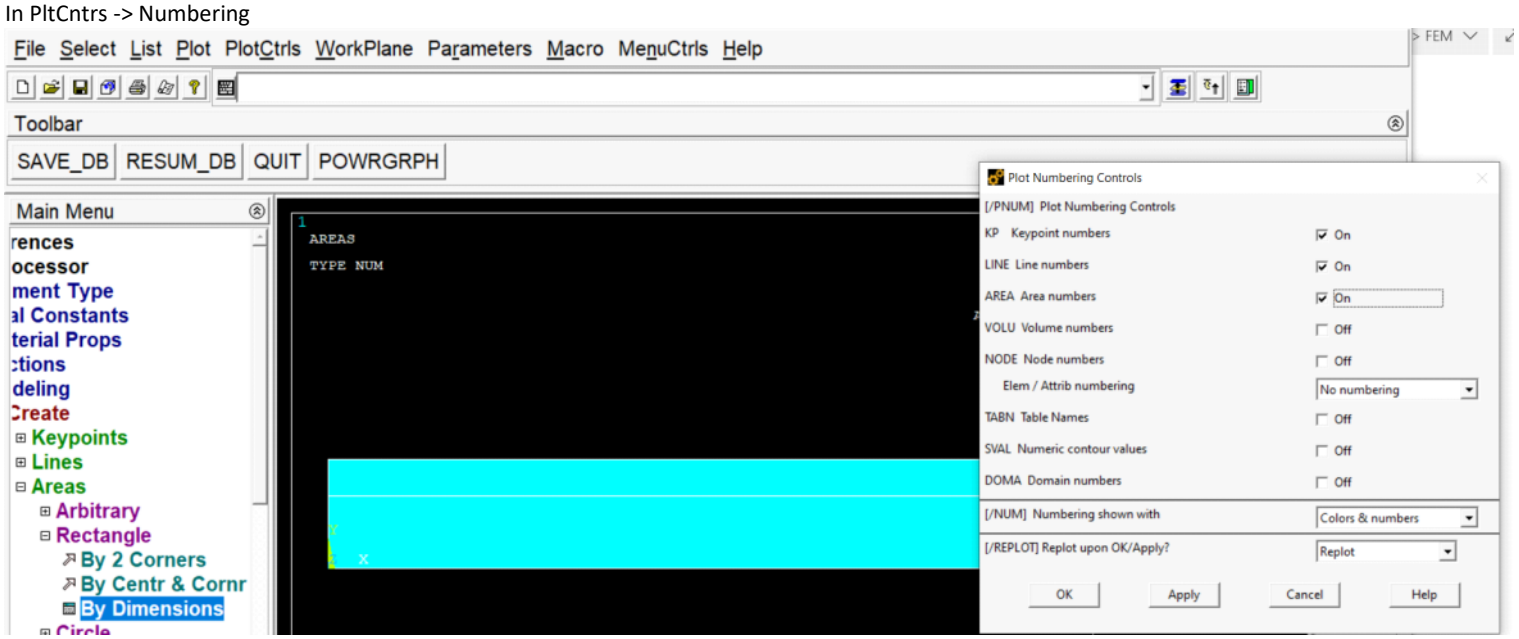


Create the geometry (in 2D, 3D create keypoints (0D), lines (1D), areas (2D), and volumes (3D) geometry objects and then mesh it (create elements and nodes)





In PltCntrs -> Numbering



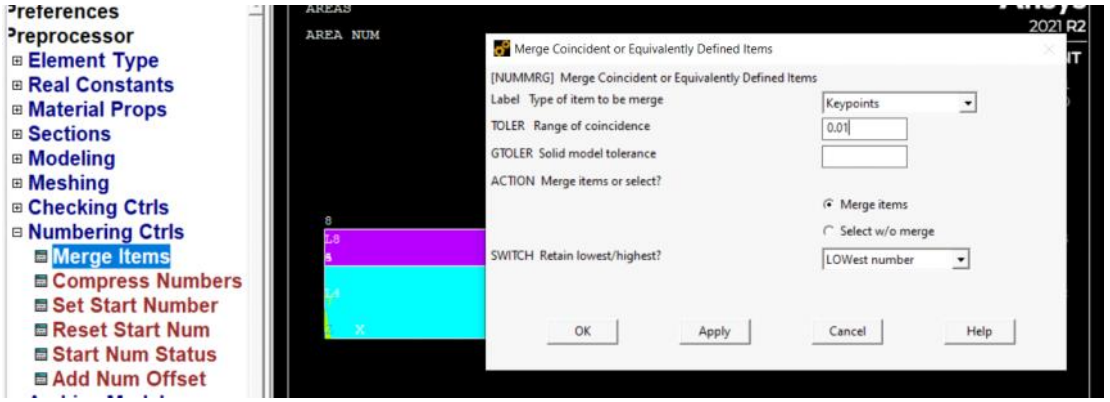
We end up with 8 rather than 6 keypoints

LIST ALL SELECTED KEYPOINTS. DSYS= 0

NO.	X,Y,Z LOCATION			THXY,THYZ,THZX ANGLES		
1	0.000000	0.000000	0.000000	0.0000	0.0000	0.0000
2	10.00000	0.000000	0.000000	0.0000	0.0000	0.0000
3	10.00000	1.000000	0.000000	0.0000	0.0000	0.0000
4	0.000000	1.000000	0.000000	0.0000	0.0000	0.0000
5	0.000000	1.000000	0.000000	0.0000	0.0000	0.0000
6	10.00000	1.000000	0.000000	0.0000	0.0000	0.0000

7	10.00000	1.500000	0.000000	0.0000	0.0000	0.0000
8	0.000000	1.500000	0.000000	0.0000	0.0000	0.0000

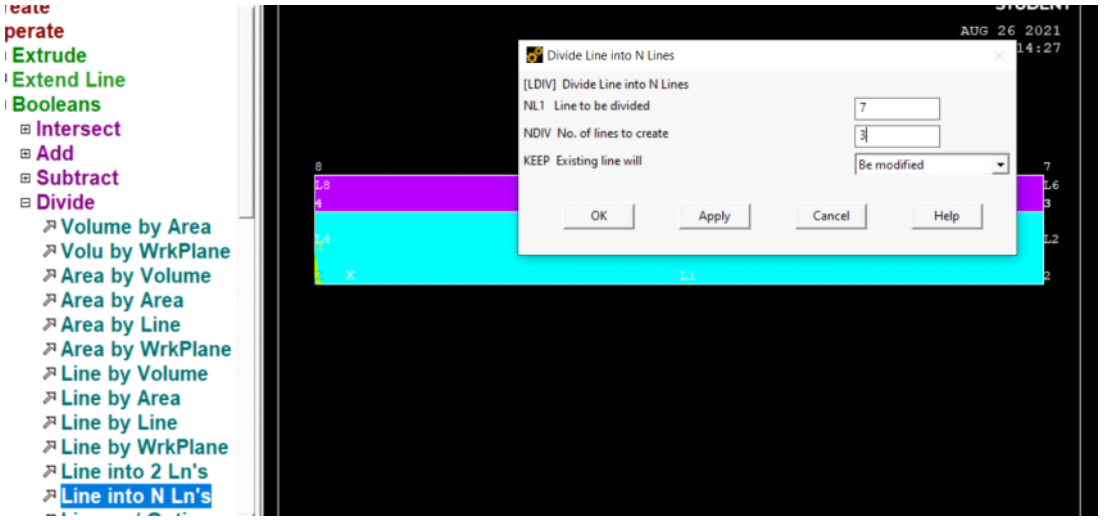
I am going to merge coinciding keypoints

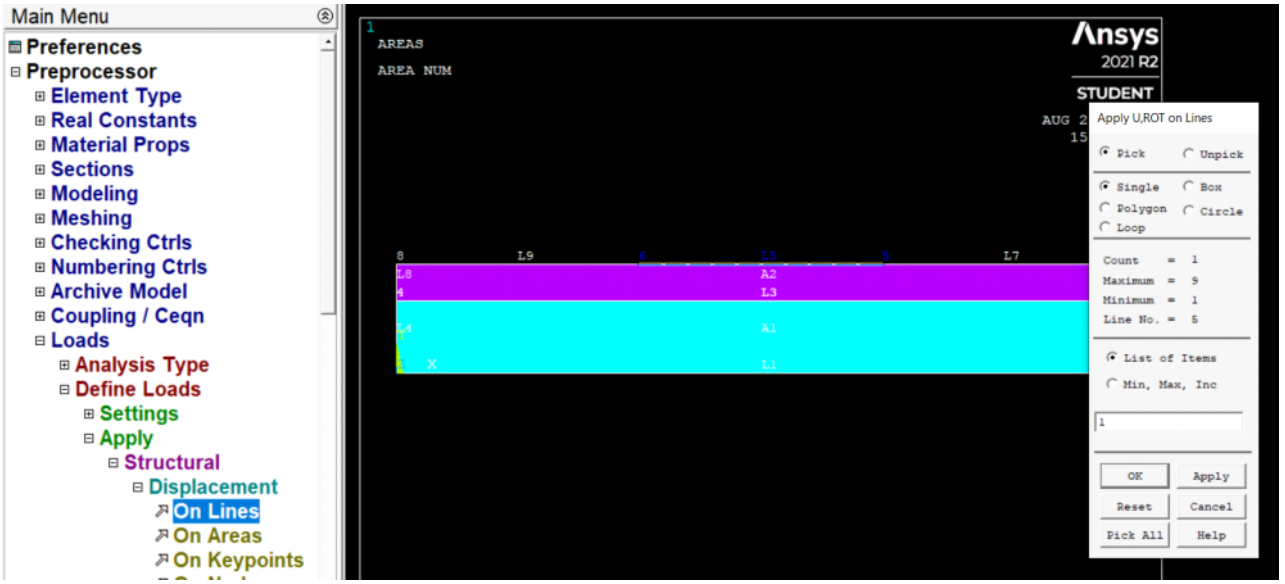


LIST ALL SELECTED KEYPOINTS. DSYS= 0

NO.	X,Y,Z LOCATION			THXY,THYZ,THZX ANGLES		
1	0.000000	0.000000	0.000000	0.0000	0.0000	0.0000
2	10.00000	0.000000	0.000000	0.0000	0.0000	0.0000
3	10.00000	1.000000	0.000000	0.0000	0.0000	0.0000
4	0.000000	1.000000	0.000000	0.0000	0.0000	0.0000
7	10.00000	1.500000	0.000000	0.0000	0.0000	0.0000
8	0.000000	1.500000	0.000000	0.0000	0.0000	0.0000

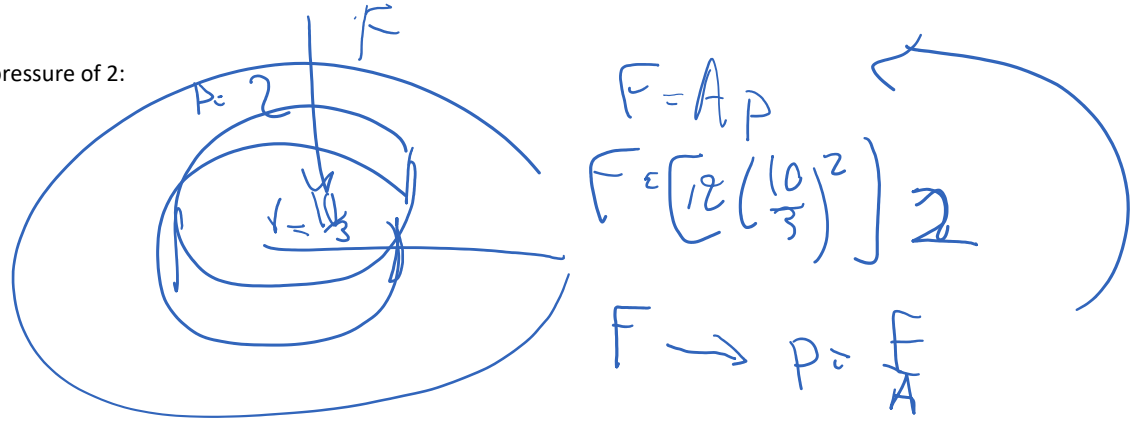
Breaking the top line into 3 segments so that we can apply the pressure on the left one

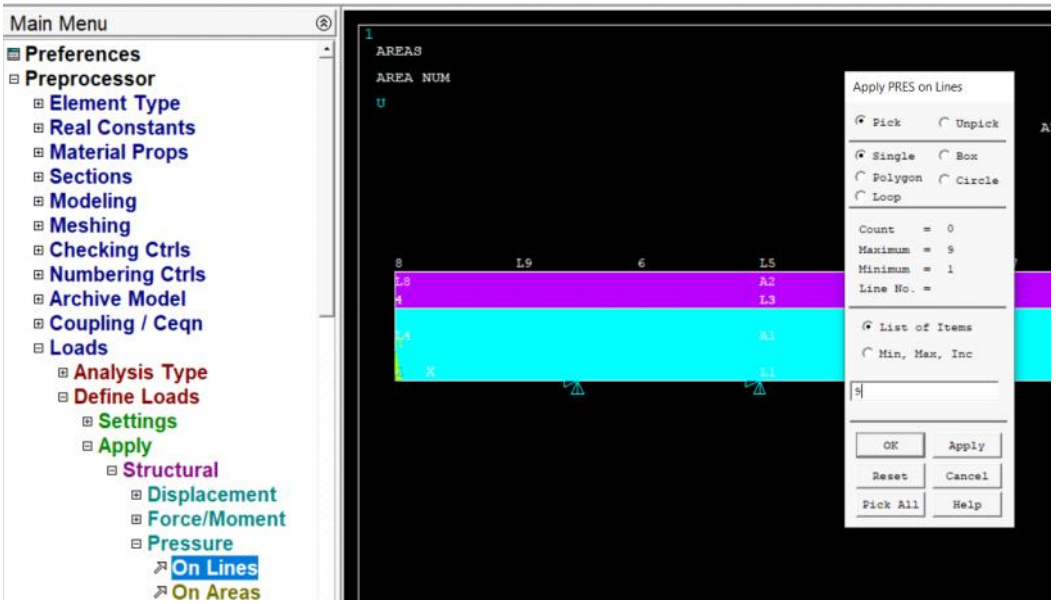


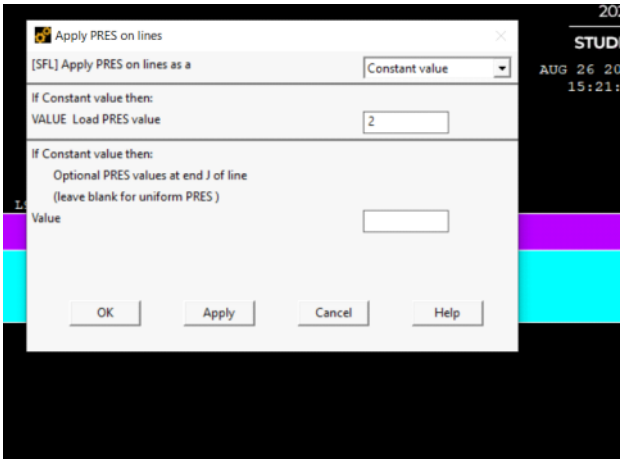


All surfaces (here lines) are by default traction free (no load), so we only need to apply load on line 9

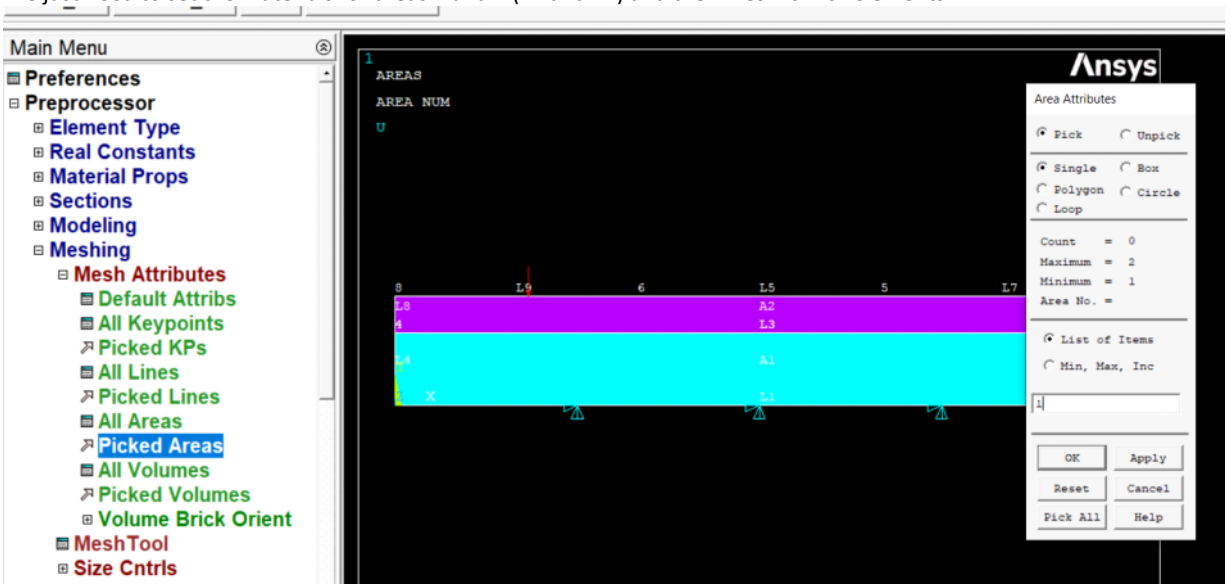
We are applying the pressure of 2:



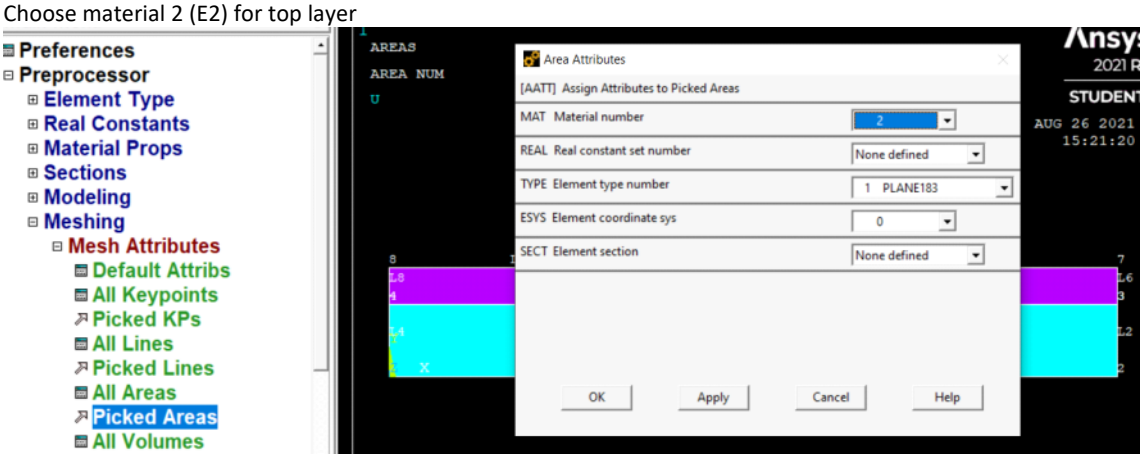




We just need to set the materials for areas 1 and 2 (E1 and E2) and then mesh it with elements



Choose material 2 (E2) for top layer

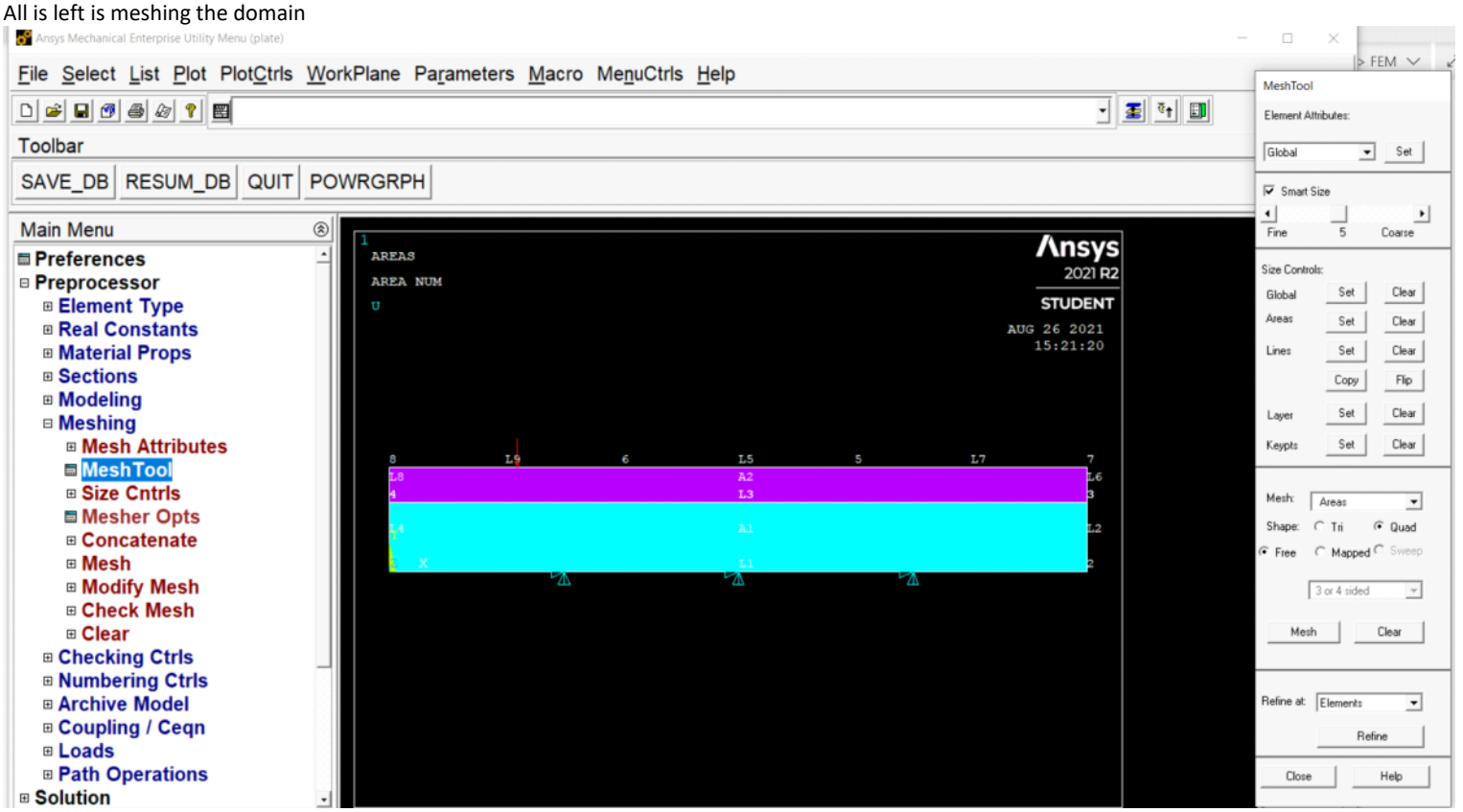


LIST ALL SELECTED AREAS.

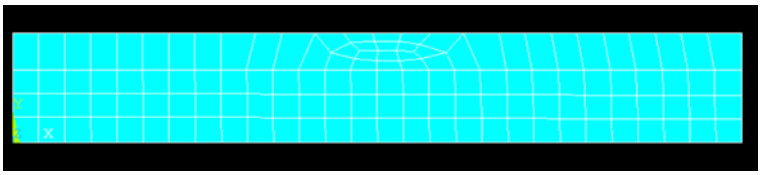
NUMBER	LOOP	LINES					AREA	ELEM	SIZE	#NODES	#ELEM	MAT	REAL	TYP	ESYS	SECN
1	1	1	2	3	4	N/A	0.000	0	0	1	0	1	0	0		
2	1	3	6	7	5	N/A	0.000	0	0	2	0	1	0	0		

9 8

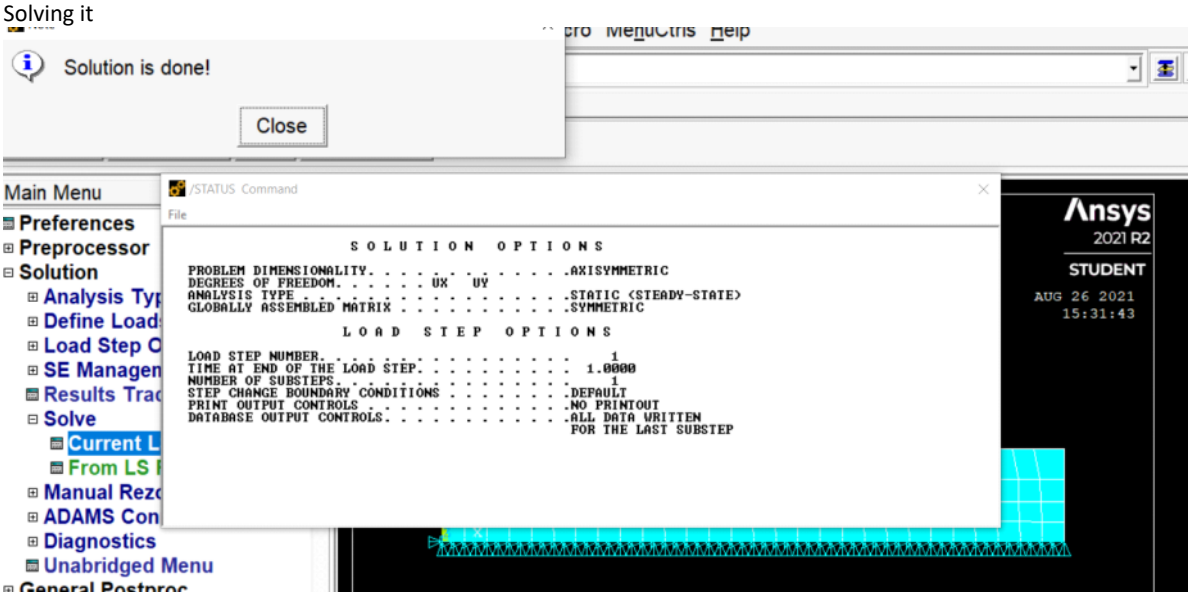
All is left is meshing the domain

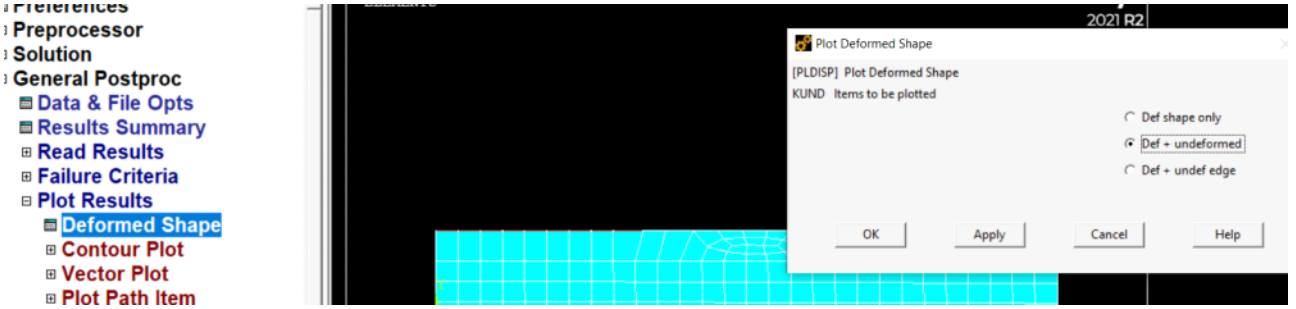


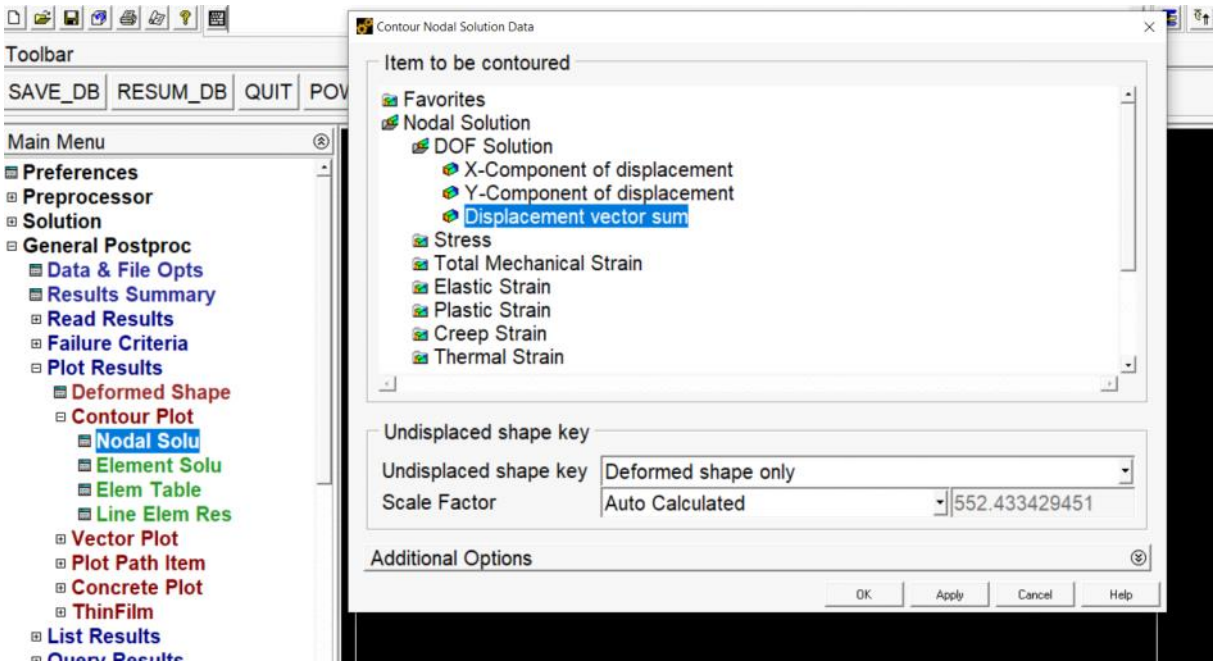
Press mesh
Select all

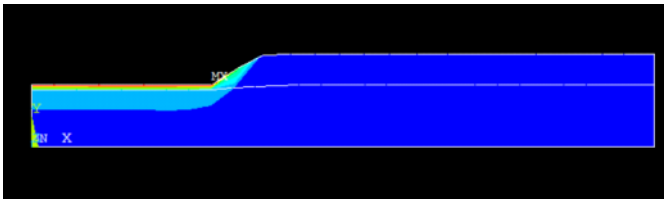


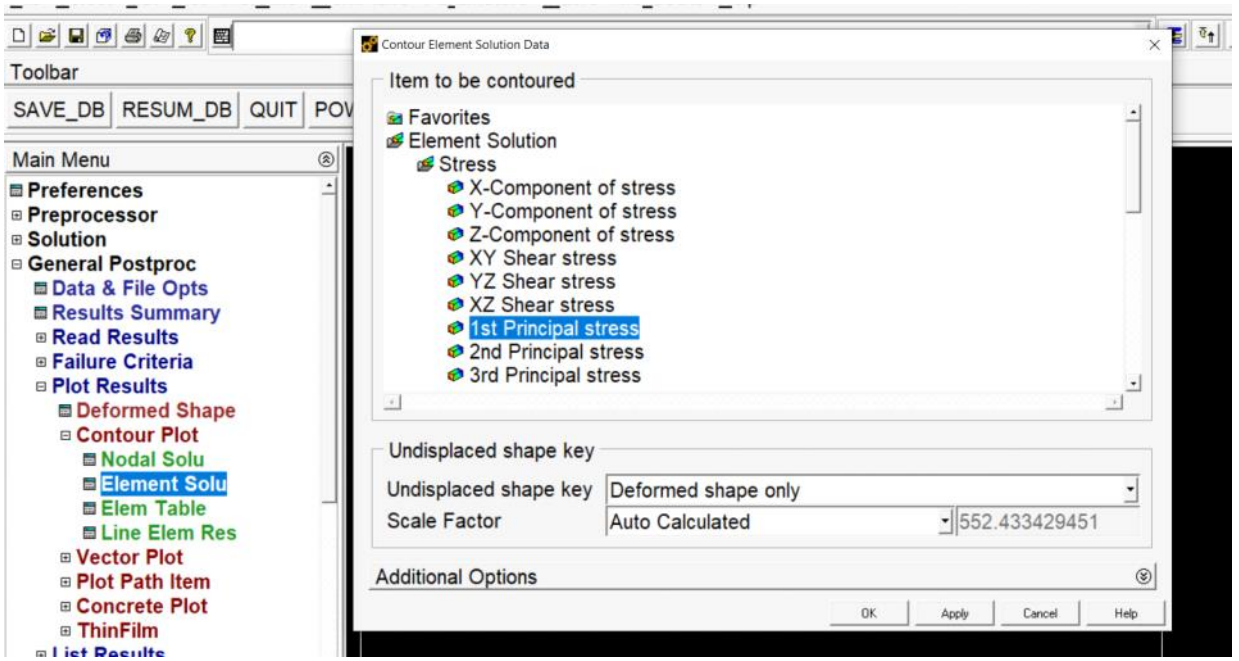
Solving it



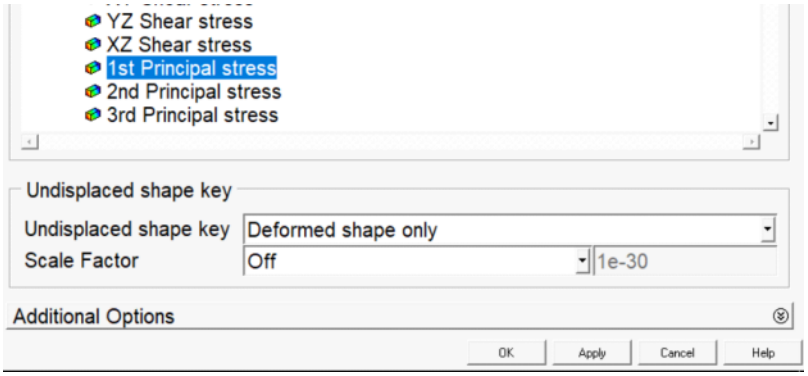

Preprocessor
Solution
General Postproc
Data & File Opts
Results Summary
Read Results
Failure Criteria
Plot Results
Deformed Shape
Contour Plot
Vector Plot
Plot Path Item
2021 R2
Plot Deformed Shape
[PLDISP] Plot Deformed Shape
KUND Items to be plotted
Def shape only
Def + undeformed
Def + undef edge
OK
Apply
Cancel
Help


Toolbar
SAVE_DB
RESUM_DB
QUIT
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
Contour Nodal Solution Data
Item to be contoured
Favorites
Nodal Solution
DOF Solution
X-Component of displacement
Y-Component of displacement
Displacement vector sum
Stress
Total Mechanical Strain
Elastic Strain
Plastic Strain
Creep Strain
Thermal Strain
Undisplaced shape key
Undisplaced shape key
Deformed shape only
Scale Factor
Auto Calculated
552.433429451
Additional Options
OK
Apply
Cancel
Help



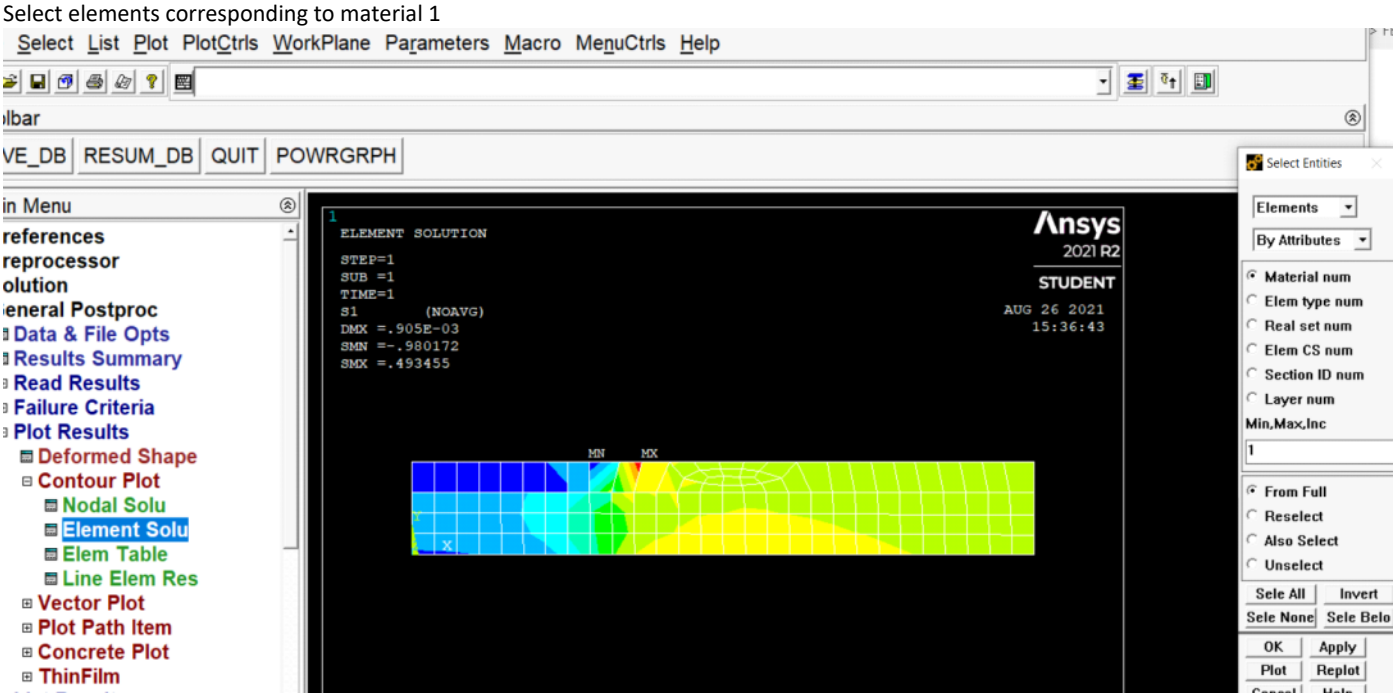

Toolbar
SAVE_DB
RESUM_DB
QUIT
Main Menu
Preferences
Preprocessor
Solution
General Postproc
Data & File Opts
Results Summary
Read Results
Failure Criteria
Plot Results
Deformed Shape
Contour Plot
Nodal Solu
Element Solu
Elem Table
Line Elem Res
Vector Plot
Plot Path Item
Concrete Plot
ThinFilm
List Results
Contour Element Solution Data
Item to be contoured
Favorites
Element Solution
Stress
X-Component of stress
Y-Component of stress
Z-Component of stress
XY Shear stress
YZ Shear stress
XZ Shear stress
1st Principal stress
2nd Principal stress
3rd Principal stress
Undisplaced shape key
Undisplaced shape key
Deformed shape only
Scale Factor
Auto Calculated
552.433429451
Additional Options
OK
Apply
Cancel
Help

1. How to plot it on undeformed geometry

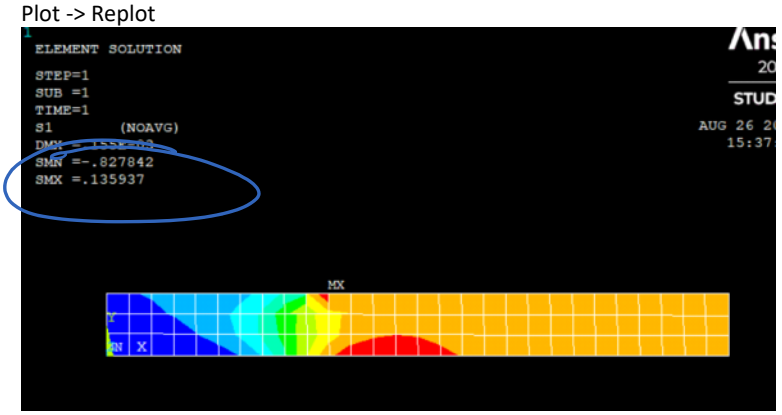


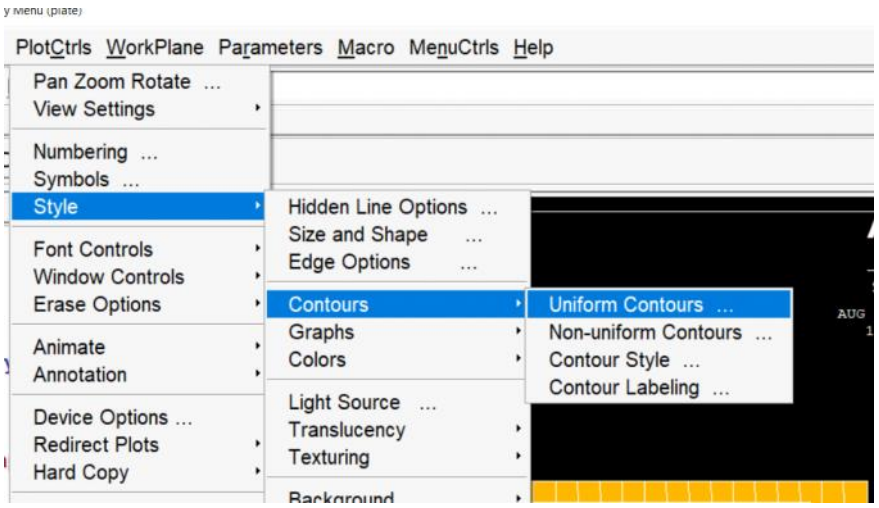
2. How to make the plot only for certain materials

Select elements corresponding to material 1



Plot -> Replot





Min and max for bottom layer

3. How to specify the range of contour plot