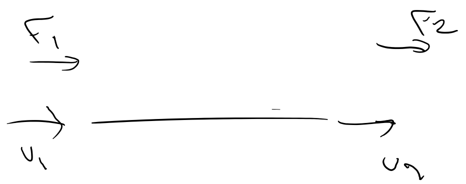
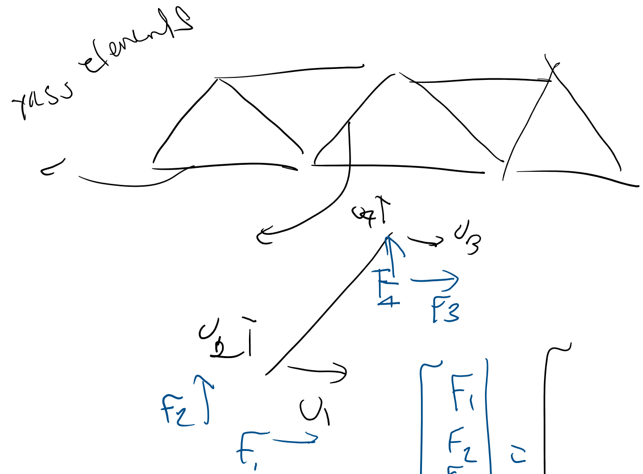
2023/08/28

Monday, August 28, 2023 11:23 AM

Finite element:

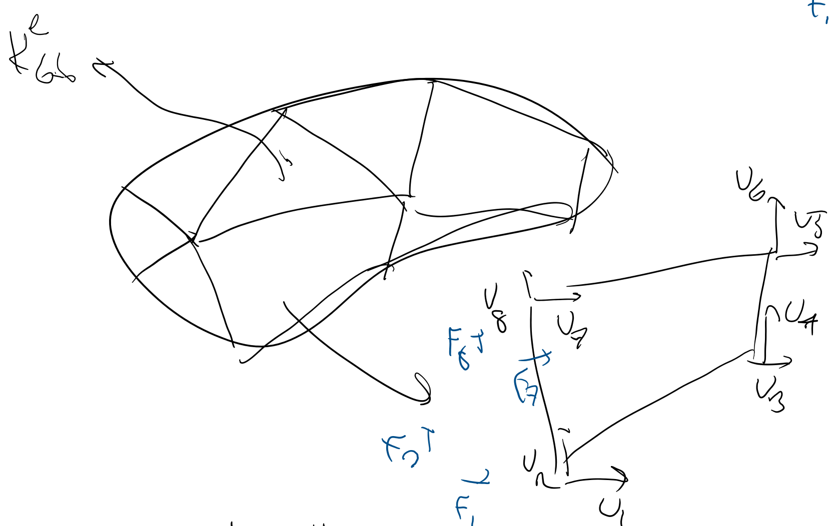
Engineering perspective: It relates nodal displacements (unknowns) to forces (force-like quantities that are additive)





$$\begin{bmatrix} F_1 \\ F_2 \end{bmatrix} = \begin{bmatrix} & \\ & \end{bmatrix}_{2\times 2} \begin{bmatrix} u_1 \\ u_2 \end{bmatrix}$$

$$\begin{bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \end{bmatrix} = \begin{bmatrix} & \\ & \end{bmatrix}_{4\times 4} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix}$$

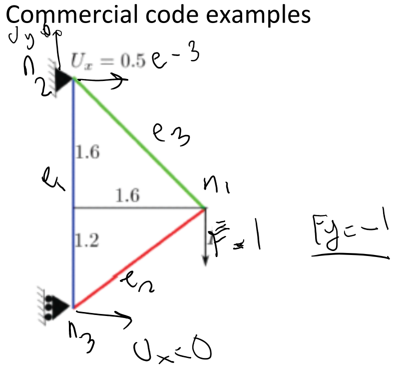


$$\begin{bmatrix} F_1 \\ \vdots \\ F_8 \end{bmatrix} = K_{8\times 8} \begin{bmatrix} u_1 \\ u_2 \\ \vdots \\ u_8 \end{bmatrix}$$

the assembly follows the same process

one place we use can provide specific element stiffness matrix to a program

Commercial code examples



$F_y = -1$

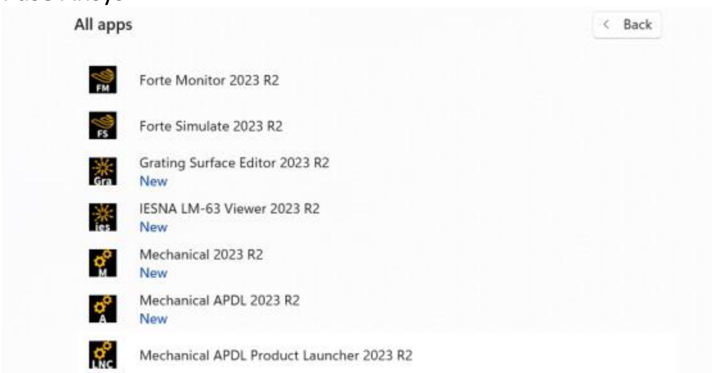
for each element we need Area(A) & elastic modulus E

	E	A
e_1	100 E_1	1 A_1
e_2	100 E_1	1 A_1
e_3	1000 E_2	10 A_2

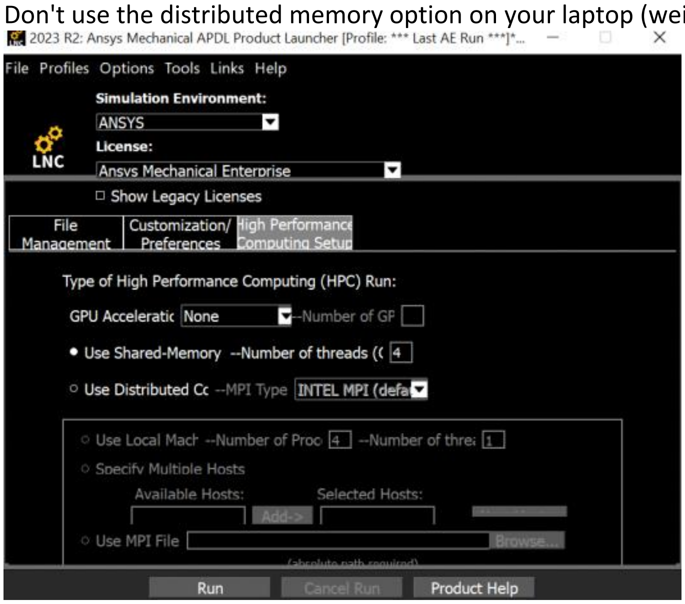
- Ansys:
  - Installation: (free academic version, link). While this is a limited version, it is sufficient for your project and is recommended due to the ease of installation
  - Make sure in ADPL launcher you use "Shared Memory" under High Performance Computing Setup.
  - Link to command lines (Acknowledgment: Matthew Carter).

OR use the program of your choice

I use Ansys

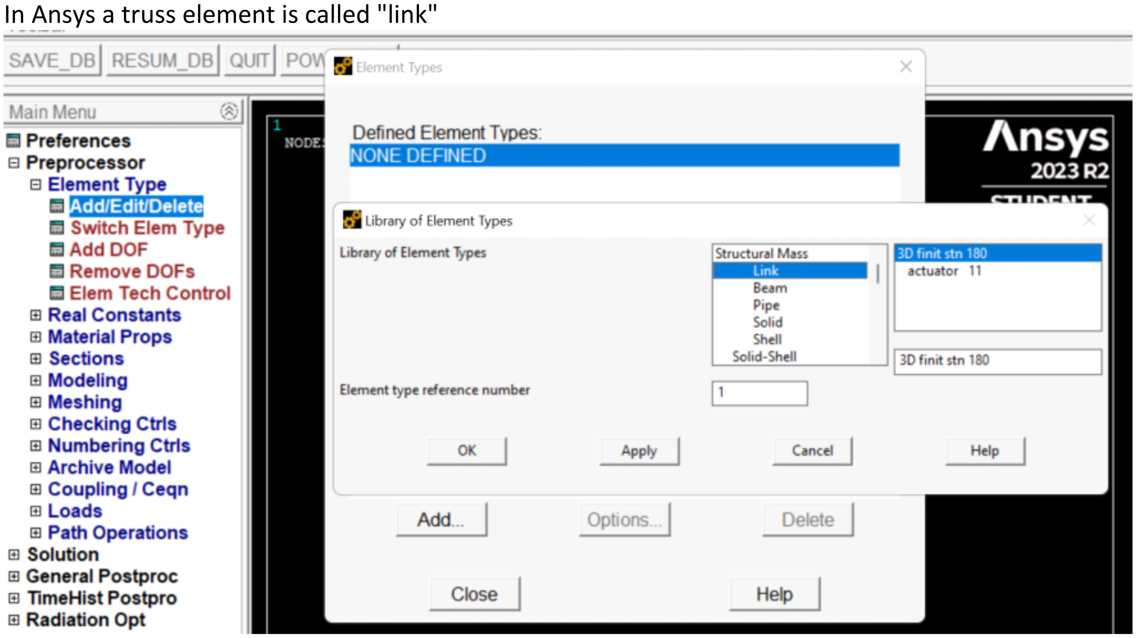


Don't use the distributed memory option on your laptop (weird crashes)



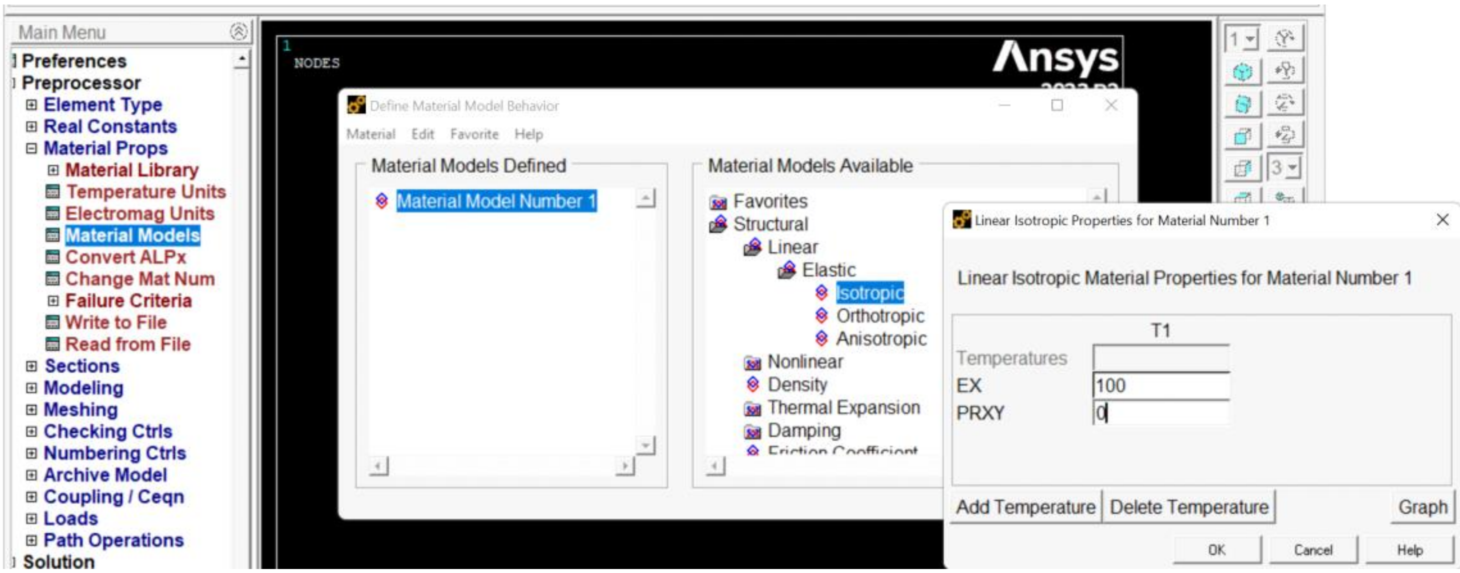
Define the element types we want to use.

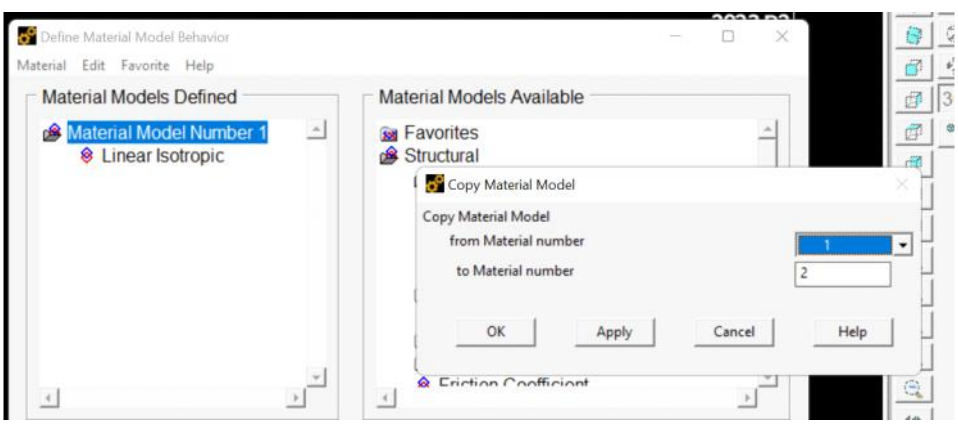
In Ansys a truss element is called "link"

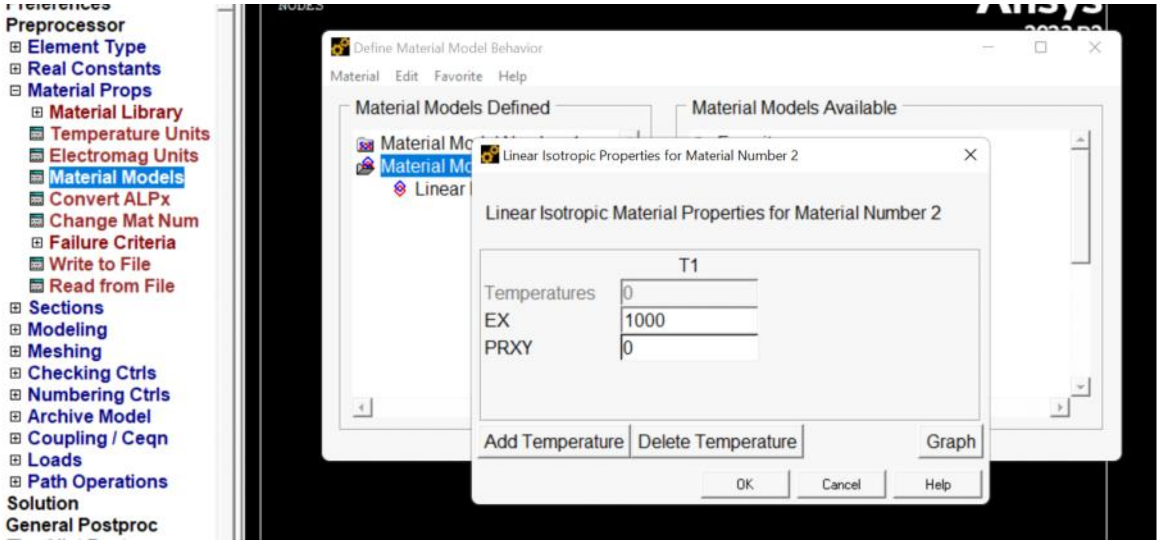


Let's define material properties

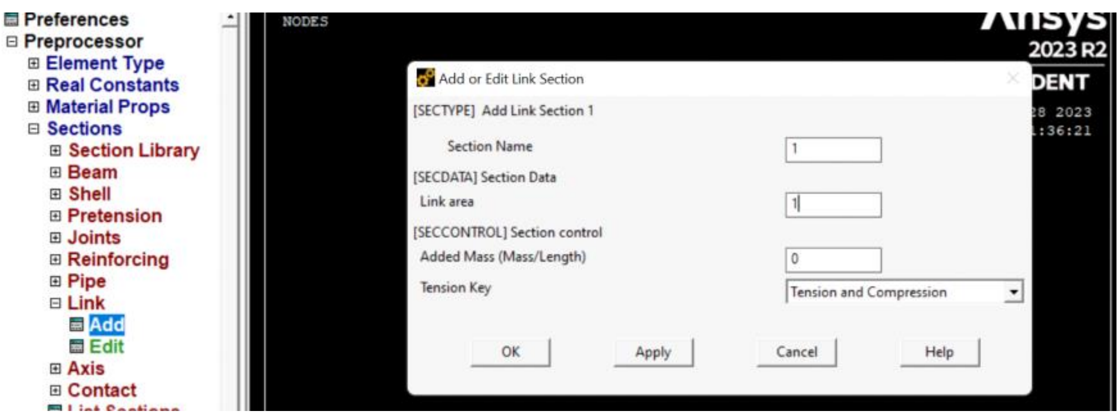
	E	A
e_1	100 E_1	1 A_1
e_2	100 E_1	1 A_1
e_3	1000 E_2	10 A_2

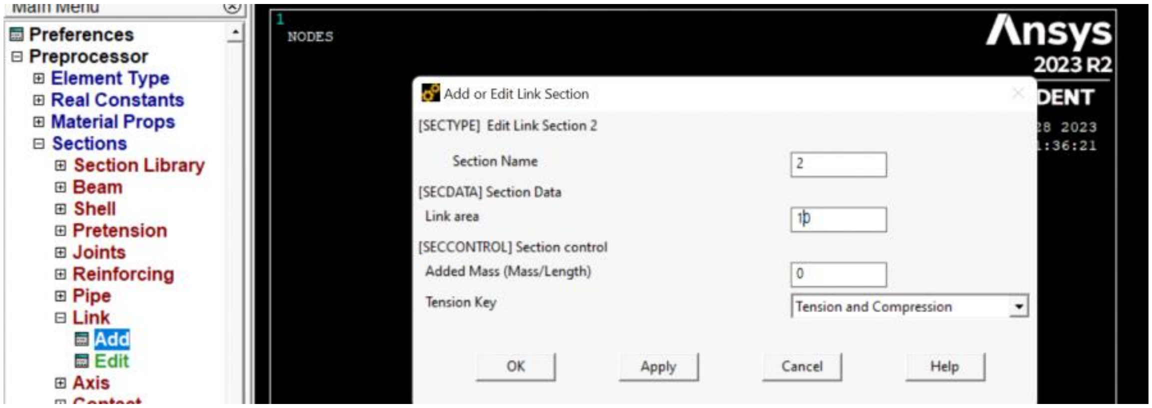




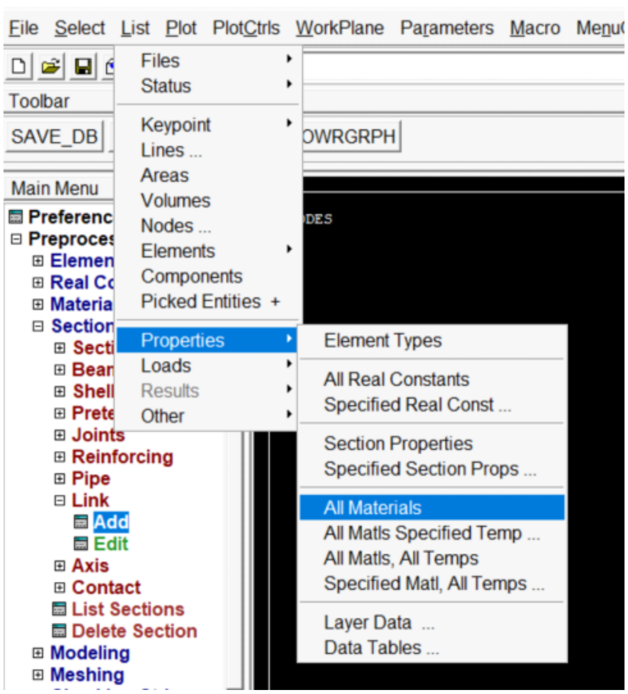


Define the section areas:





We can check our material and section properties (not necessary)



EVALUATE MATERIAL PROPERTIES FOR MATERIALS 1 TO 2
IN INCREMENTS OF 1

MATERIAL NUMBER = 1 EVALUATED AT TEMPERATURE OF
0.0000
EX = 100.00
PRXY = 0.0000

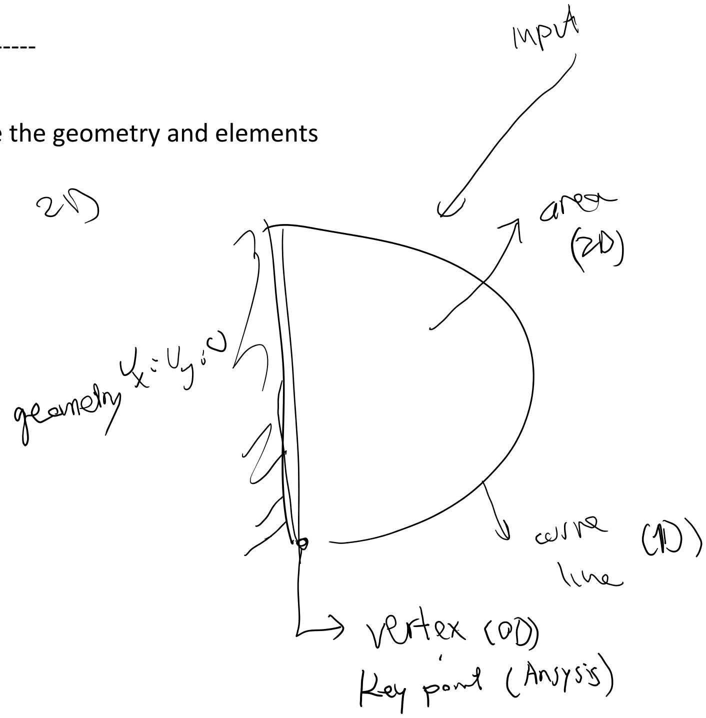
MATERIAL NUMBER = 2 EVALUATED AT TEMPERATURE OF
0.0000
EX = 1000.0
PRXY = 0.0000
------------- same with sections
LIST SECTION ID SETS 1 TO 2 BY 1

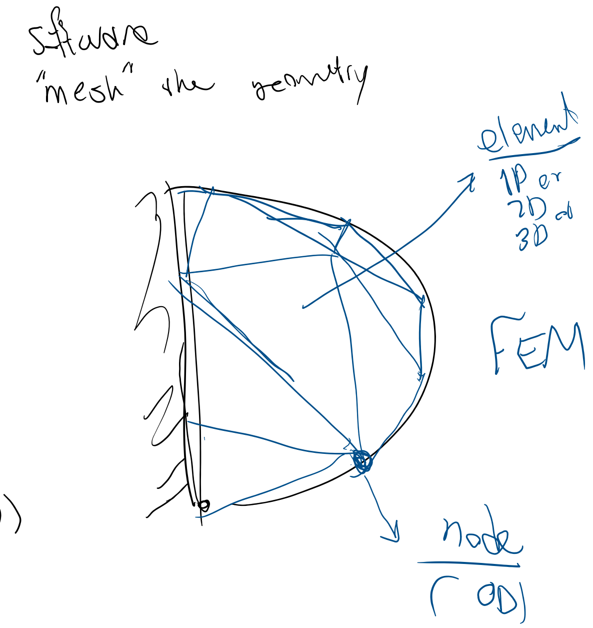
SECTION ID NUMBER: 1
LINK SECTION NAME IS: 1
Link Area = 1.0000

SECTION ID NUMBER: 2
LINK SECTION NAME IS: 2
Link Area = 10.000

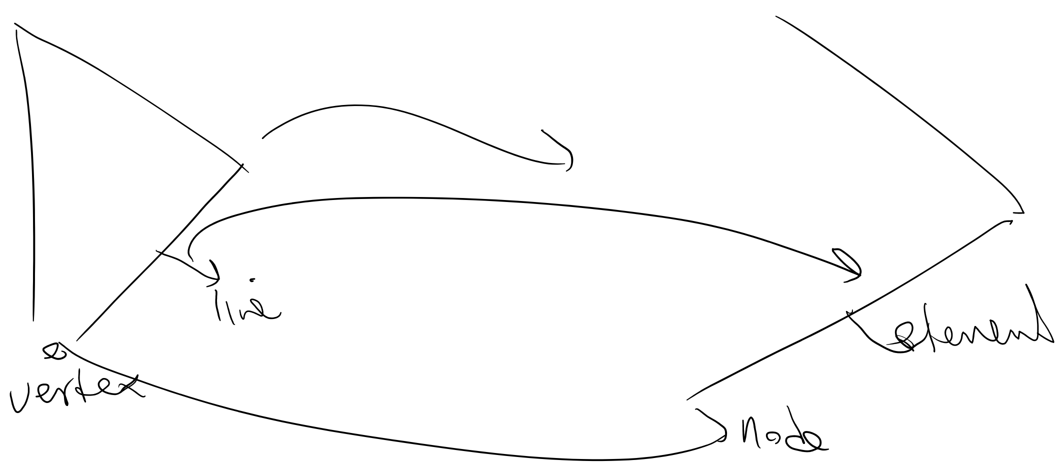
Create the geometry and elements

e.g. 2D



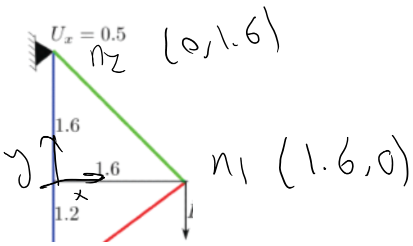


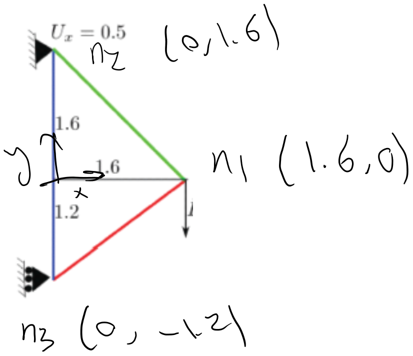
we prefer to provide geometry to the program & ask it to create FEM elements & nodes (mesh it) (2D & 3D)
— we can get closer to exact geometries

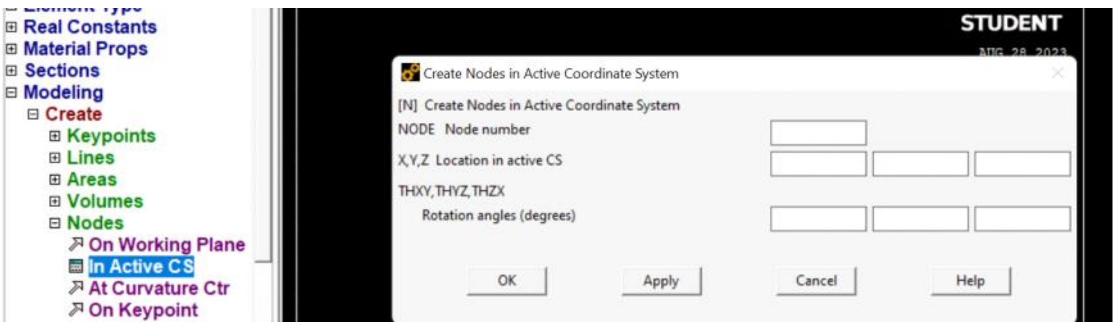


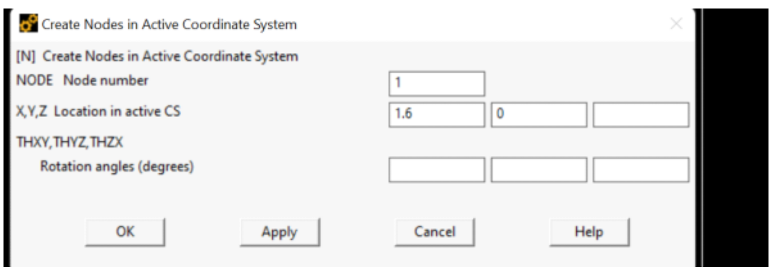
1D elements just directly create FEM objects (nodes & elements)

Create the nodes

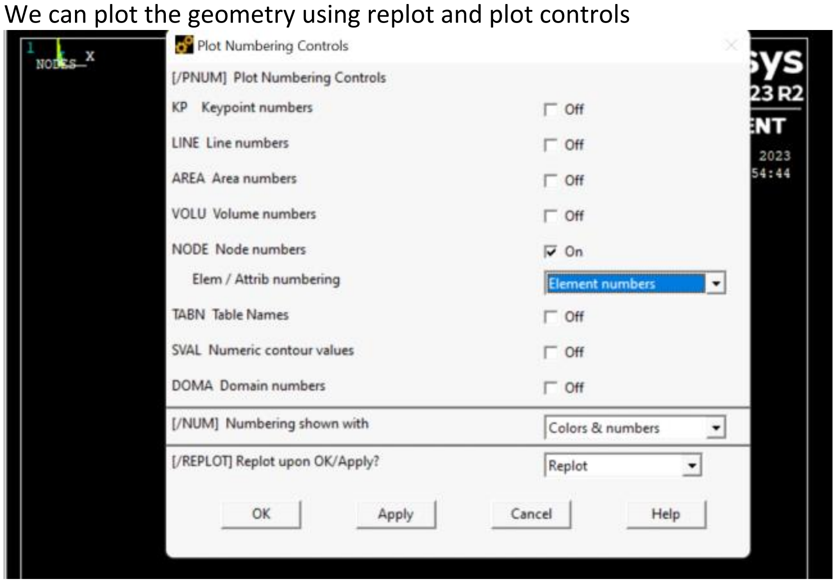




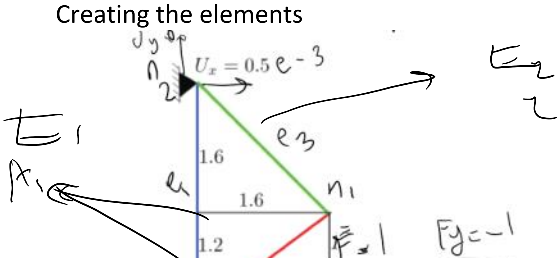




We can plot the geometry using replot and plot controls

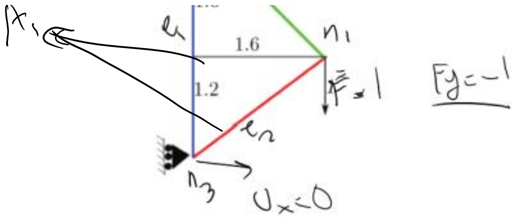


Creating the elements



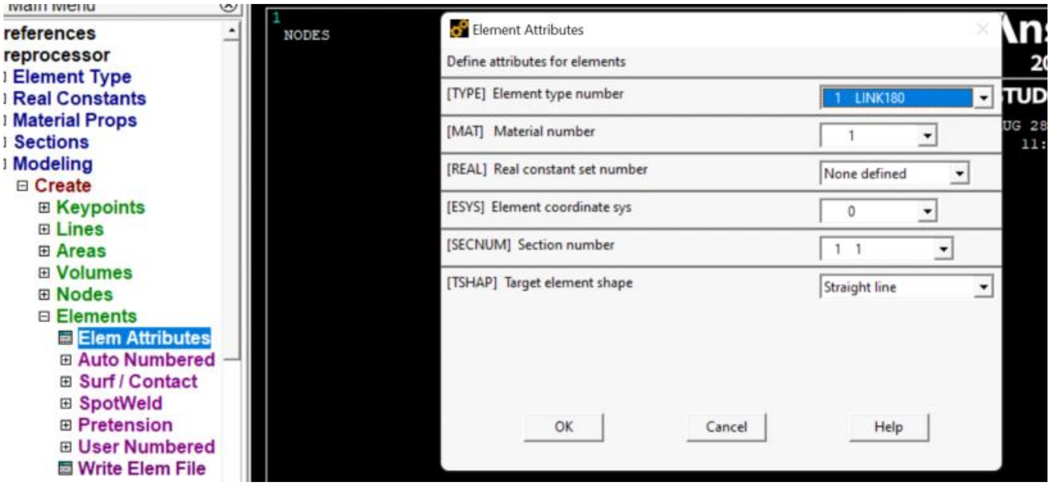
& elastic modulus E

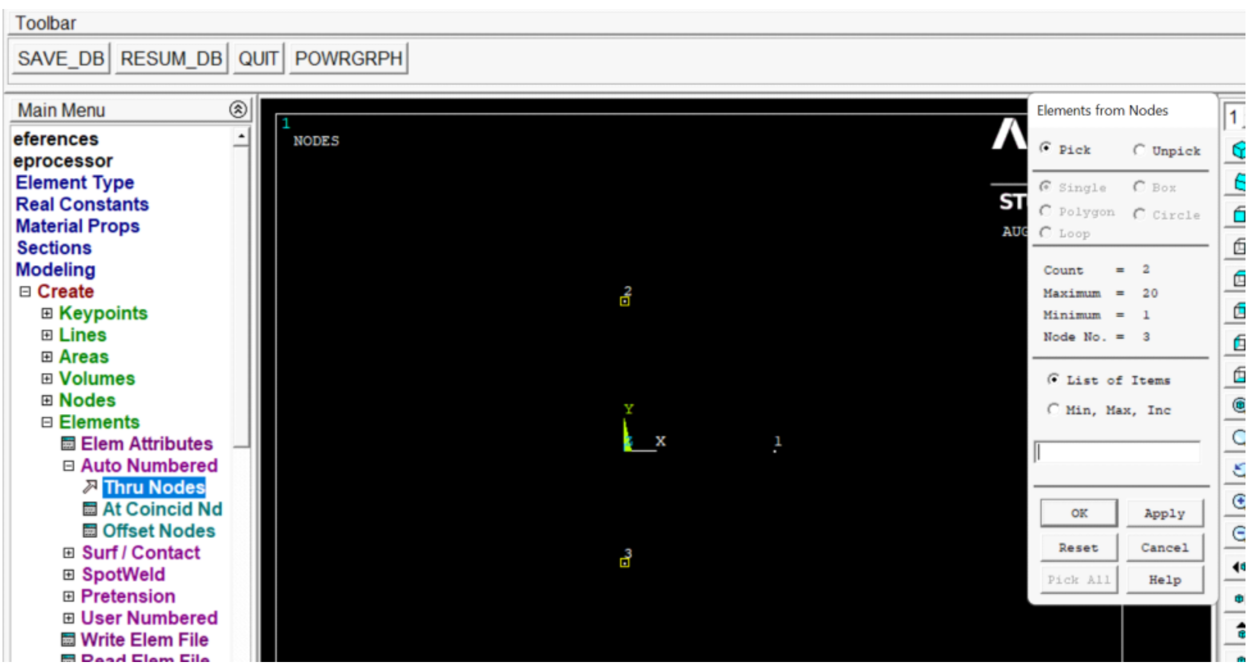
	E	A
e1	100 E1	1 A1
e2	100 E1	1 A1



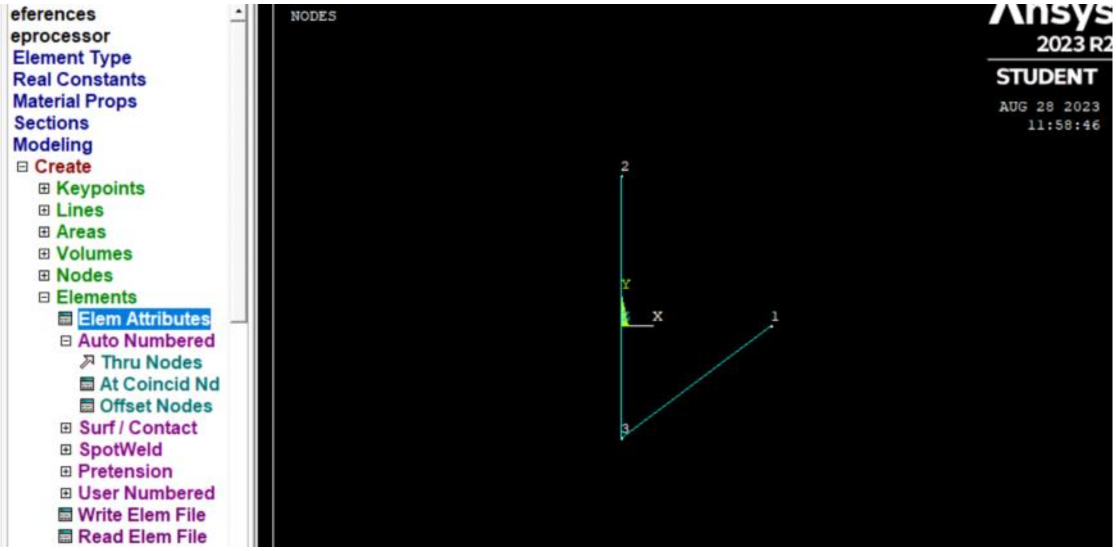
e_2	100 E_1	1	A_1
e_3	1000 E_2	10	A_2

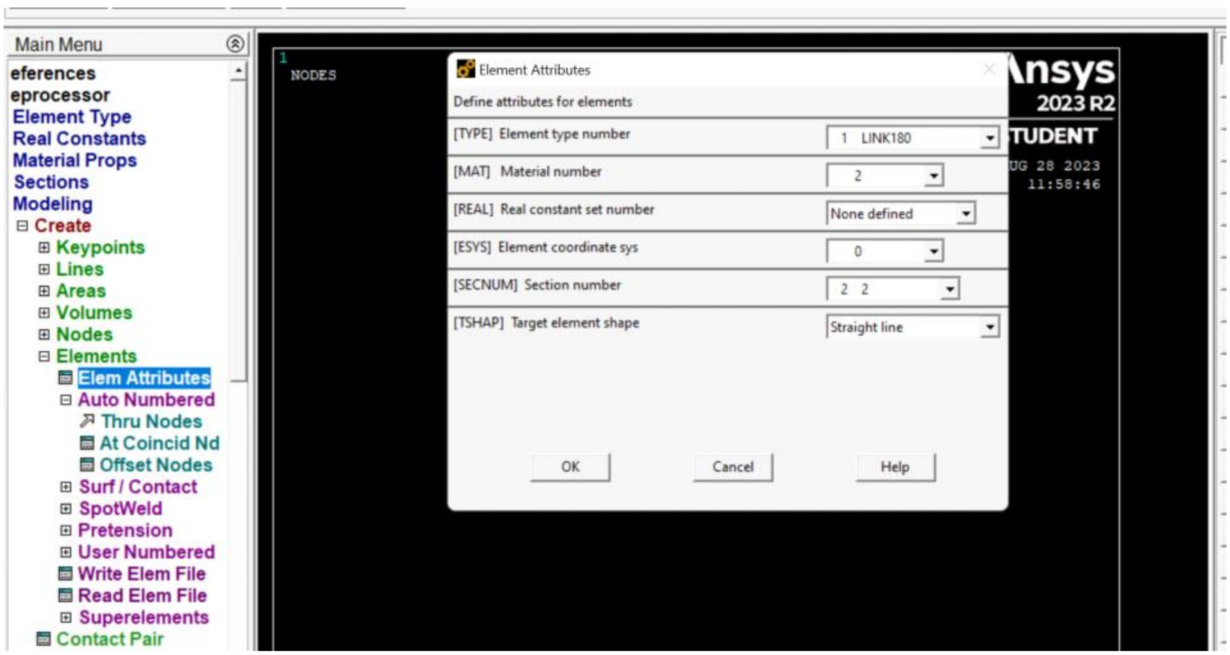
Check material and section number:

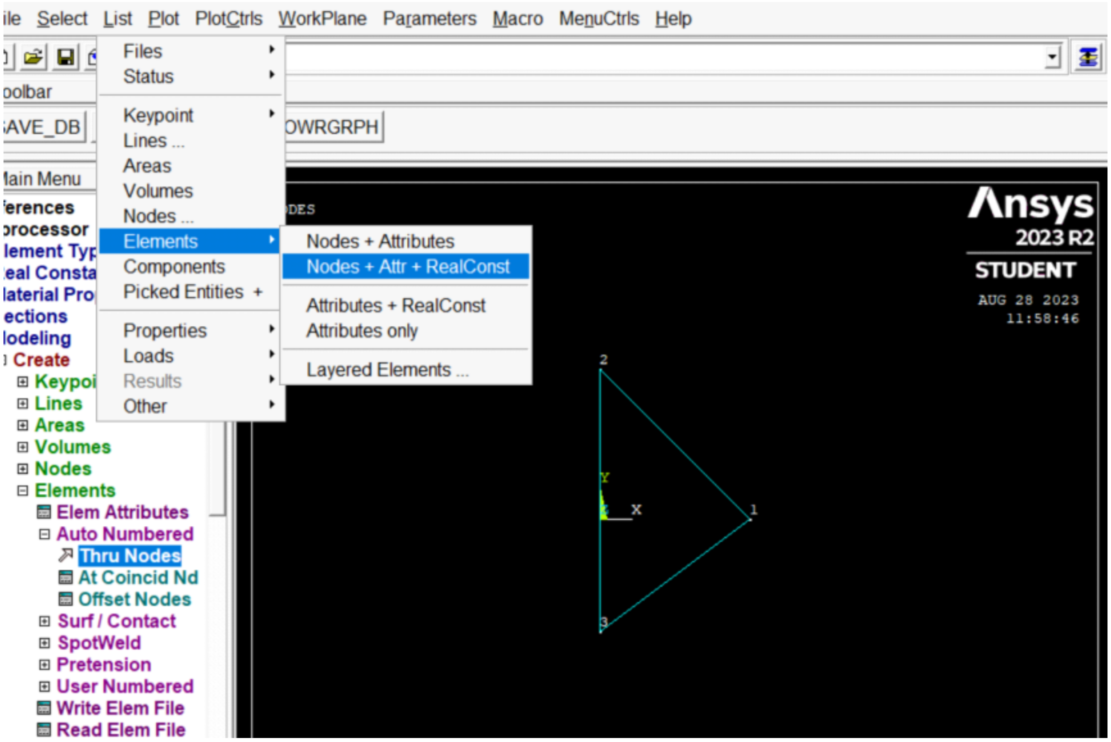




Change E and A for element 3:


eferences
eprocessor
Element Type
Real Constants
Material Props
Sections
Modeling
Create
Keypoints
Lines
Areas
Volumes
Nodes
Elements
Elem Attributes
Auto Numbered
Thru Nodes
At Coincid Nd
Offset Nodes
Surf / Contact
SpotWeld
Pretension
User Numbered
Write Elem File
Read Elem File
NODES
2023 R2
STUDENT
AUG 28 2023
11:58:46

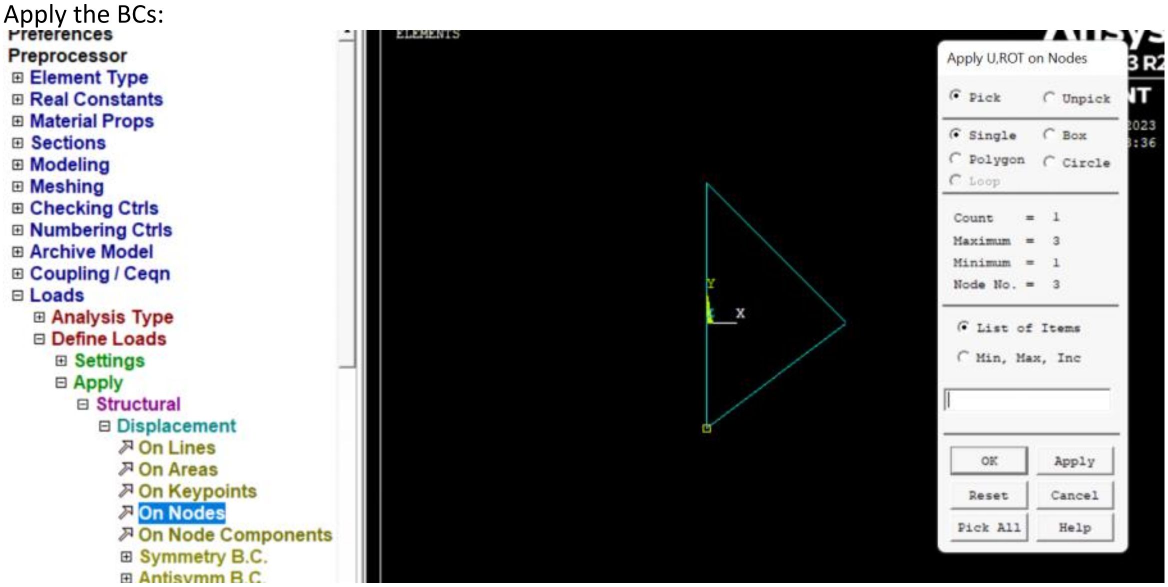

Main Menu
eferences
eprocessor
Element Type
Real Constants
Material Props
Sections
Modeling
Create
Keypoints
Lines
Areas
Volumes
Nodes
Elements
Elem Attributes
Auto Numbered
Thru Nodes
At Coincid Nd
Offset Nodes
Surf / Contact
SpotWeld
Pretension
User Numbered
Write Elem File
Read Elem File
Superelements
Contact Pair
NODES
Element Attributes
Define attributes for elements
[TYPE] Element type number
1 LINK180
[MAT] Material number
2
[REAL] Real constant set number
None defined
[ESYS] Element coordinate sys
0
[SECNUM] Section number
2 2
[TSHAP] Target element shape
Straight line
OK
Cancel
Help
Ansys
2023 R2
TUDENT
UG 28 2023
11:58:46

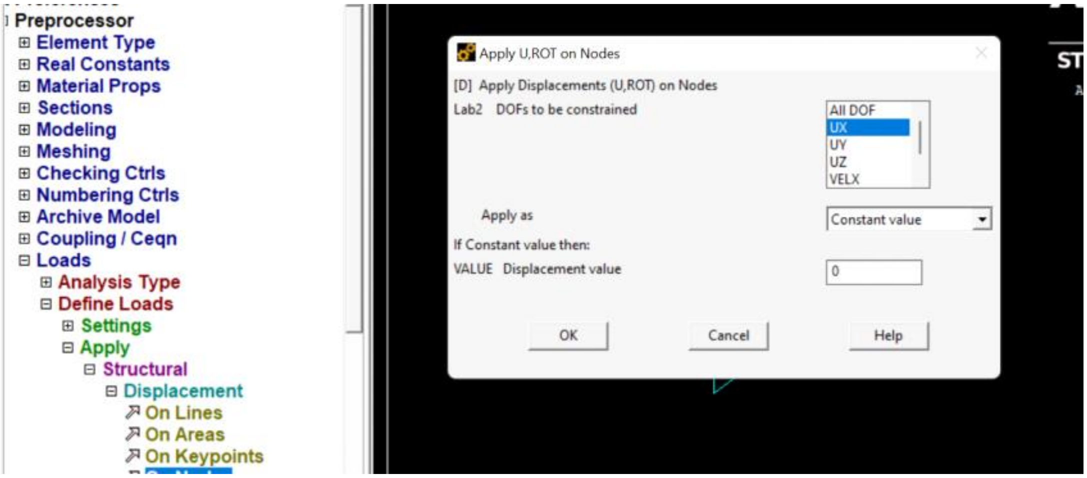


LIST ALL SELECTED ELEMENTS. (LIST NODES)

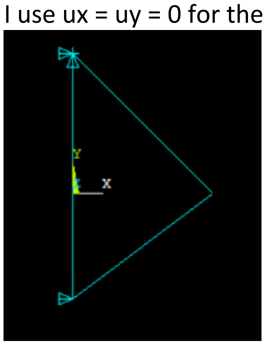
ELEM	MAT	TYP	REL	ESY	SEC	NODES	
1	1	1	1	0	1	2	3
2	1	1	1	0	1	3	1
3	2	1	1	0	2	2	1

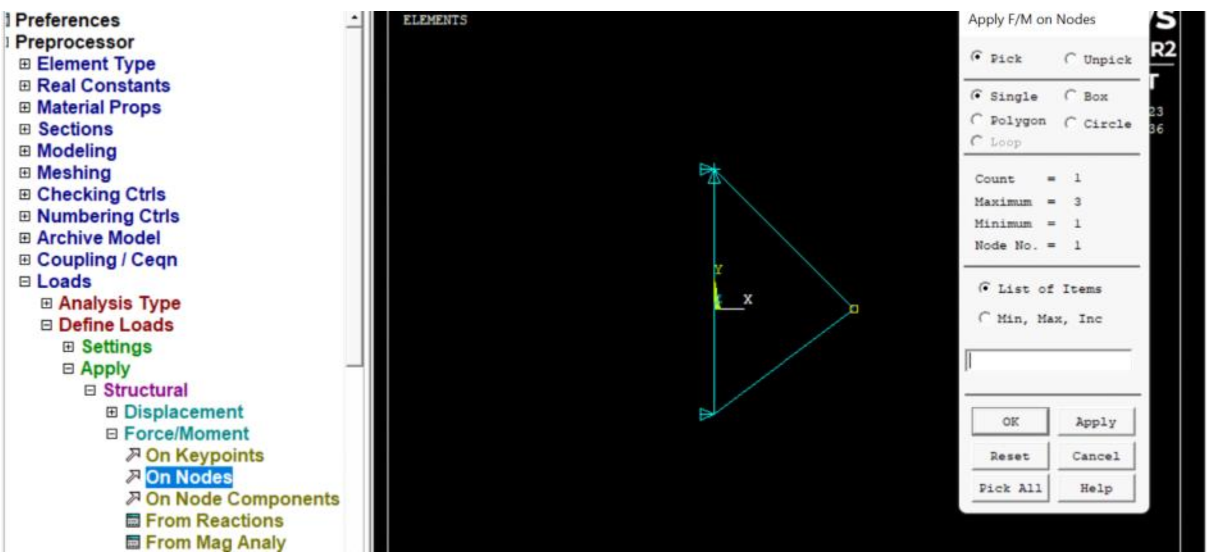
Apply the BCs:

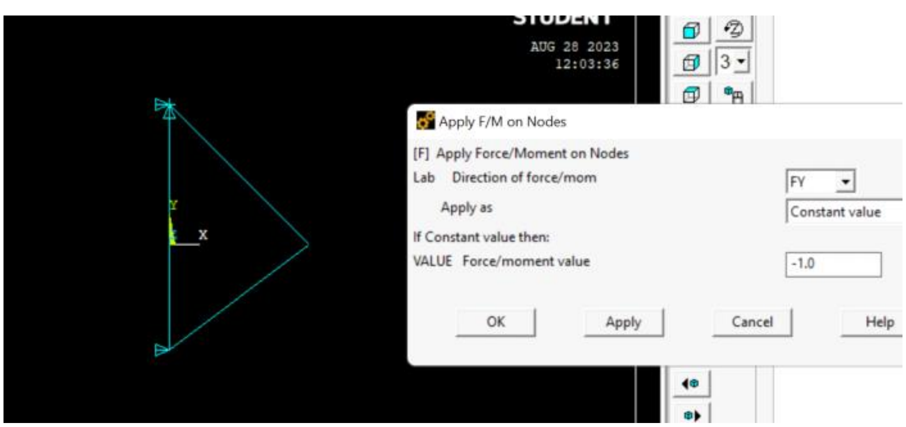




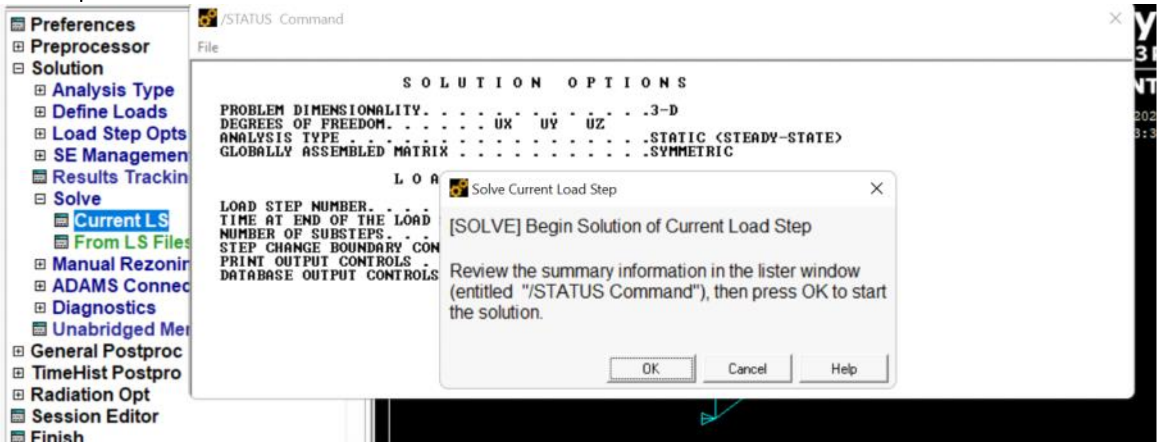
I use ux = uy = 0 for the top node





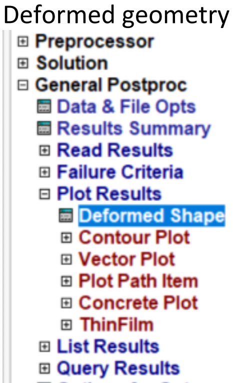


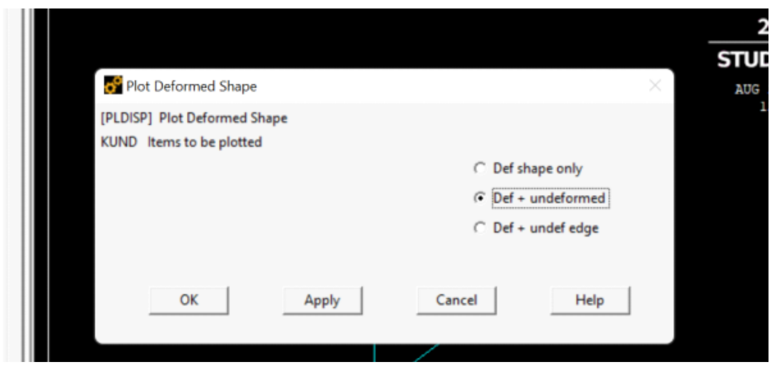
At this point we can solve it

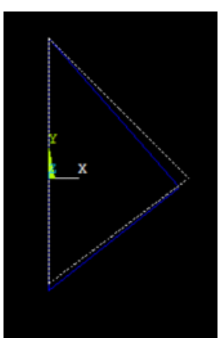


Postprocess (viewing / outputting the results):

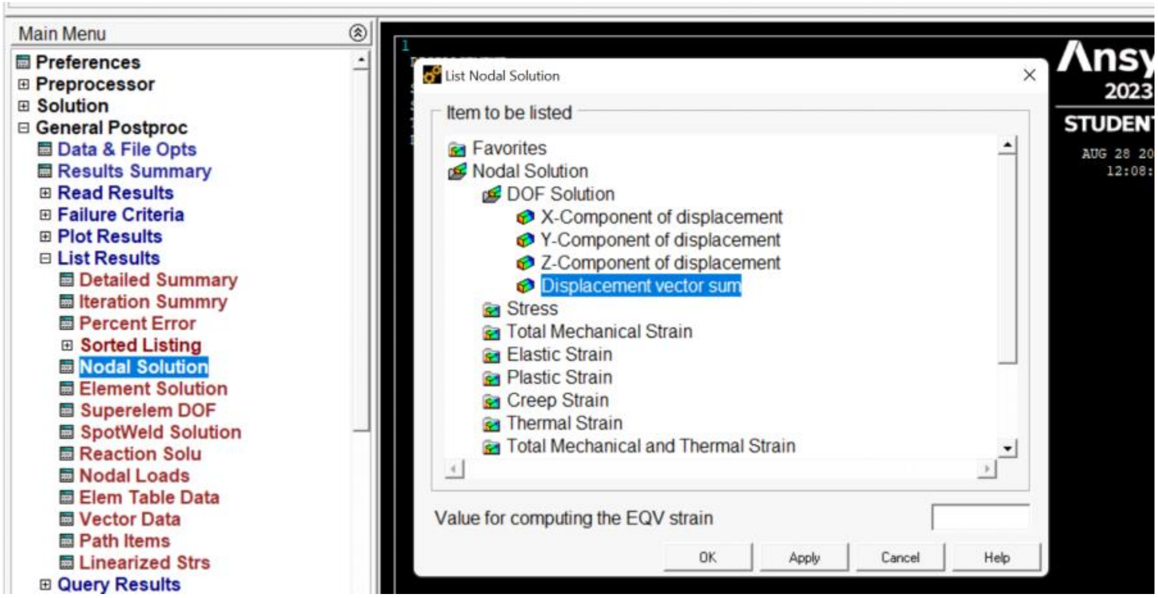
1. Deformed geometry







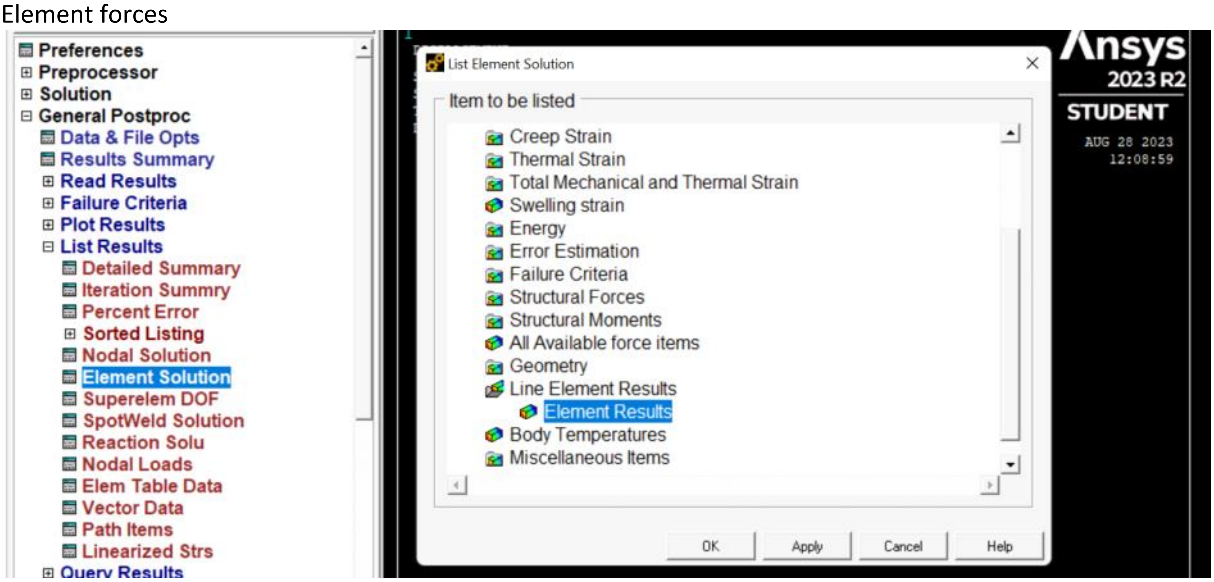
2. Nodal displacements



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

```
  NODE     UX         UY         UZ        USUM
    1 -0.15236E-001-0.15495E-001  0.0000    0.21731E-001
    2   0.0000      0.0000      0.0000      0.0000
    3   0.0000     -0.12000E-001  0.0000     0.12000E-001

 MAXIMUM ABSOLUTE VALUES
 NODE         1          1          0          1
 VALUE  -0.15236E-001-0.15495E-001  0.0000     0.21731E-001
```

3. Element forces



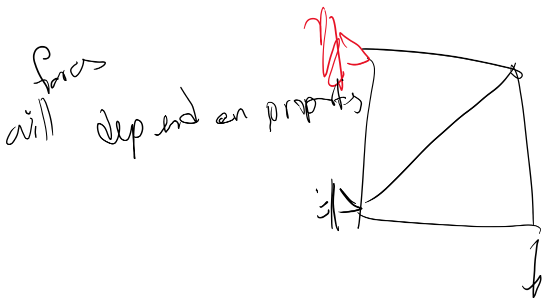

```
 LOAD STEP     1  SUBSTEP=     1
 TIME=    1.0000        LOAD CASE=  0

 EL=     1 NODES=      2      3 MAT=     1  XC,YC,ZC=  0.000     0.2000      0.000     AREA=  1.0000
LINK180
 FORCE= 0.42857      STRESS= 0.42857     EPEL= 0.42857E-02
 TEMP=    0.00    0.00  EPTH=  0.0000

 EL=     2 NODES=      3      1 MAT=     1  XC,YC,ZC= 0.8000    -0.6000      0.000     AREA=  1.0000
```

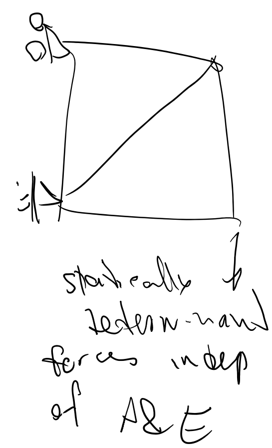
TEMP=   0.00   0.00 EPTH= 0.0000

EL=   2 NODES=   3   1 MAT=   1 XC,YC,ZC= 0.8000   -0.6000   0.000   AREA= 1.0000
LINK180
FORCE=-0.71429   STRESS=-0.71429   EPEL=-0.71429E-02
TEMP=   0.00   0.00 EPTH= 0.0000

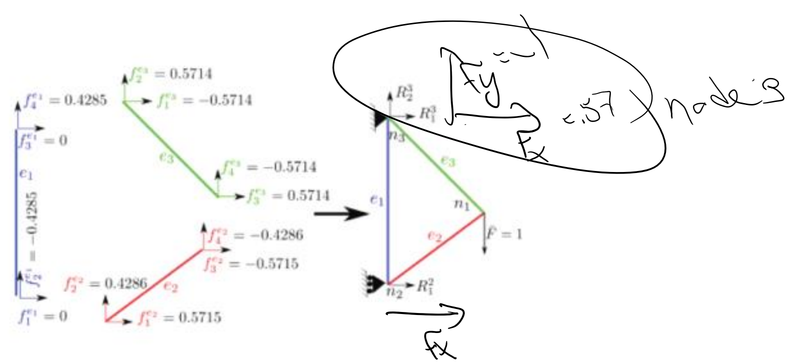
EL=   3 NODES=   2   1 MAT=   2 XC,YC,ZC= 0.8000   0.8000   0.000   AREA= 10.000
LINK180
FORCE= 0.80812   STRESS= 0.80812E-01 EPEL= 0.80812E-04
TEMP=   0.00   0.00 EPTH= 0.0000

T^e	$T^{e_1} = \frac{1\times 1}{2.8}\{0 \times (0.5 - 0) + 1 \times (0 + 1.2)\} = 0.4286$	$T^{e_2} = \frac{1\times 1}{2}\{0.8 \times (-0.2123 - 0) + 0.6 \times (-3.2928 + 1.2)\} = -0.7128$	$T^{e_3} = \frac{1\times 1}{1.6\sqrt{2}}\{\frac{1}{\sqrt{2}} \times (-0.2123 - 0.5) - \frac{1}{\sqrt{2}} \times (-3.2928 - 0)\} = 0.8064$





4. Reaction forces



- Also, if we want to double-check our calculations on **free dofs**. This step is not needed and it may be done as a verification for hand calculations:

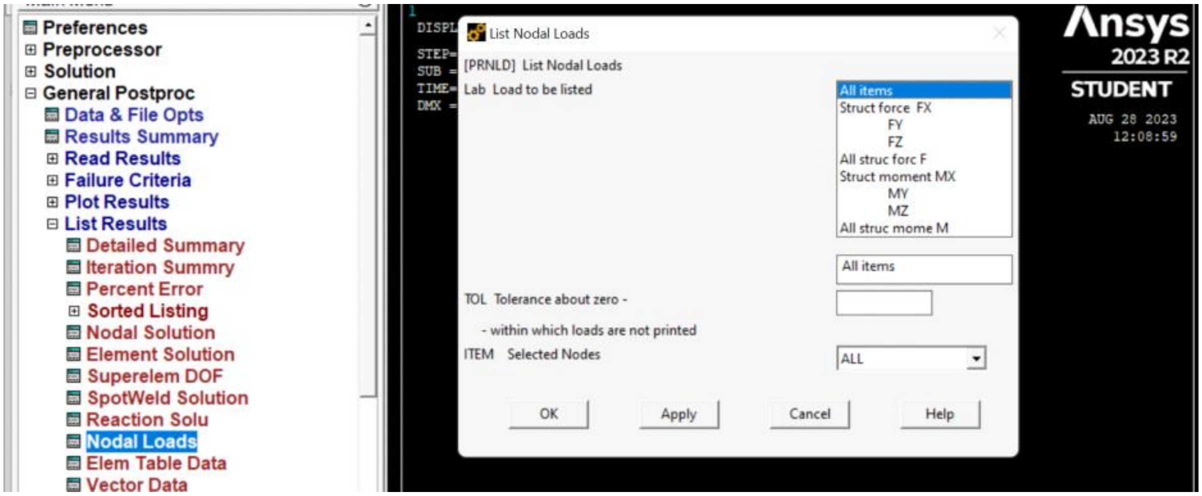
$$F_1^1 = f_3^{e_2} + f_3^{e_3} = -0.5715 + 0.5714 = -0.0001 \qquad (398a)$$

$$F_2^1 = f_4^{e_2} + f_4^{e_3} = -0.4286 + -0.5714 = -1 = \bar{F} \qquad (398b)$$

$$R_2^2 = f_2^{e_1} + f_2^{e_2} = -0.4285 + 0.4286 = 0.0001 \qquad (398c)$$

You are screen sharing   01:00:18   Stop Share

326 / 456



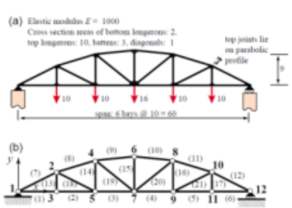
LOAD STEP= 1 SUBSTEP= 1
TIME= 1.0000 LOAD CASE= 0

THE FOLLOWING X,Y,Z SOLUTIONS ARE IN THE GLOBAL COORDINATE SYSTEM

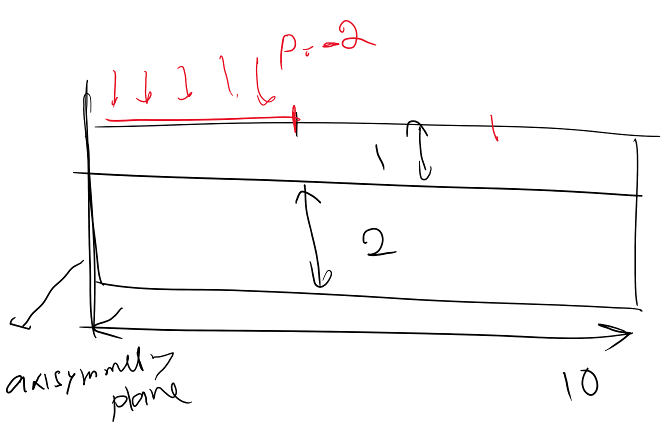
NODE	FX	FY	FZ
1		1.0000	
2	0.57143	-1.0000	
3	-0.57143		

TOTAL VALUES
VALUE -0.28866E-014 0.11102E-015 0.0000

References:
[1]. Introduction to Finite Element Methods (ASEN 5007), Fall 2013, Department of Aerospace Engineering Sciences, University of Colorado at Boulder: Part III: Computer Implementation of Finite Elements: Chapter 22 Index. FEM Programs for Trusses and Frames.



A 2D plate example

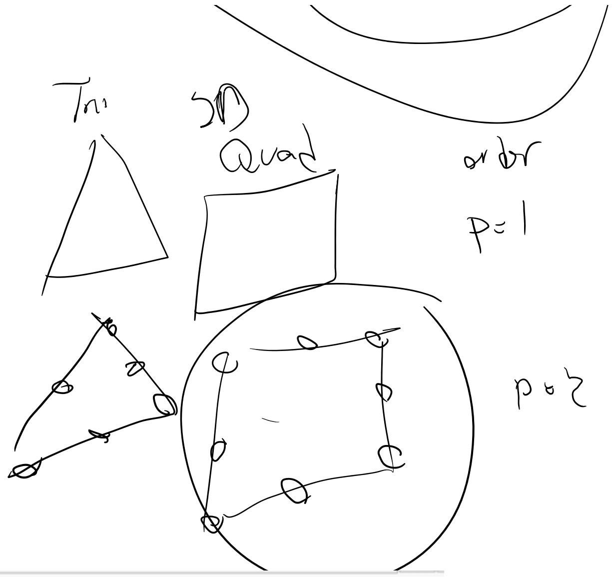


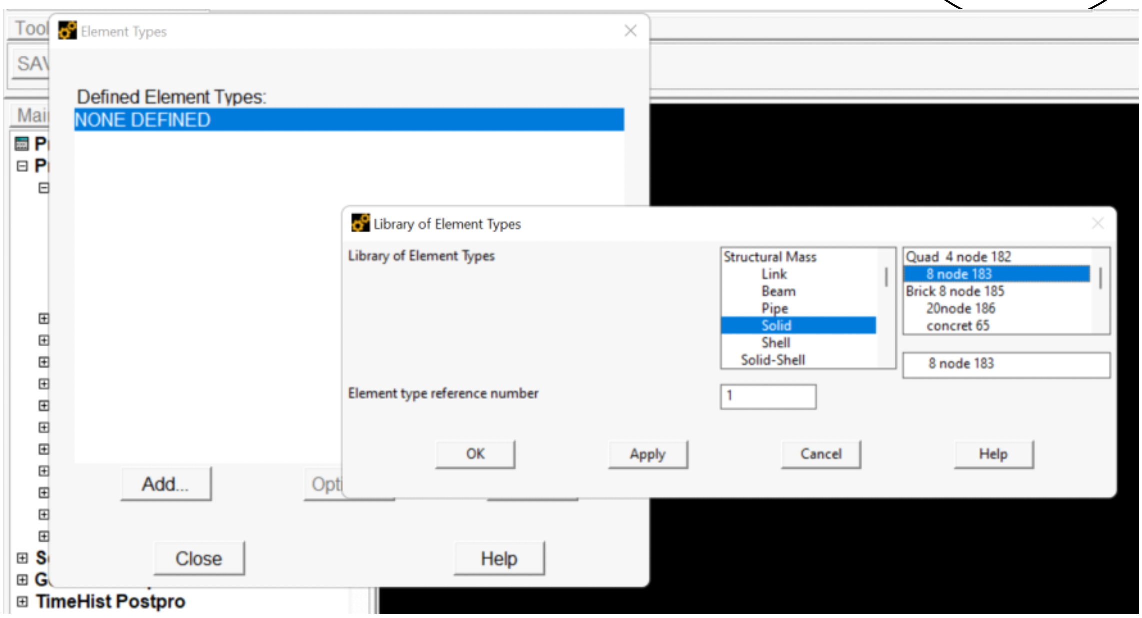
E = 100

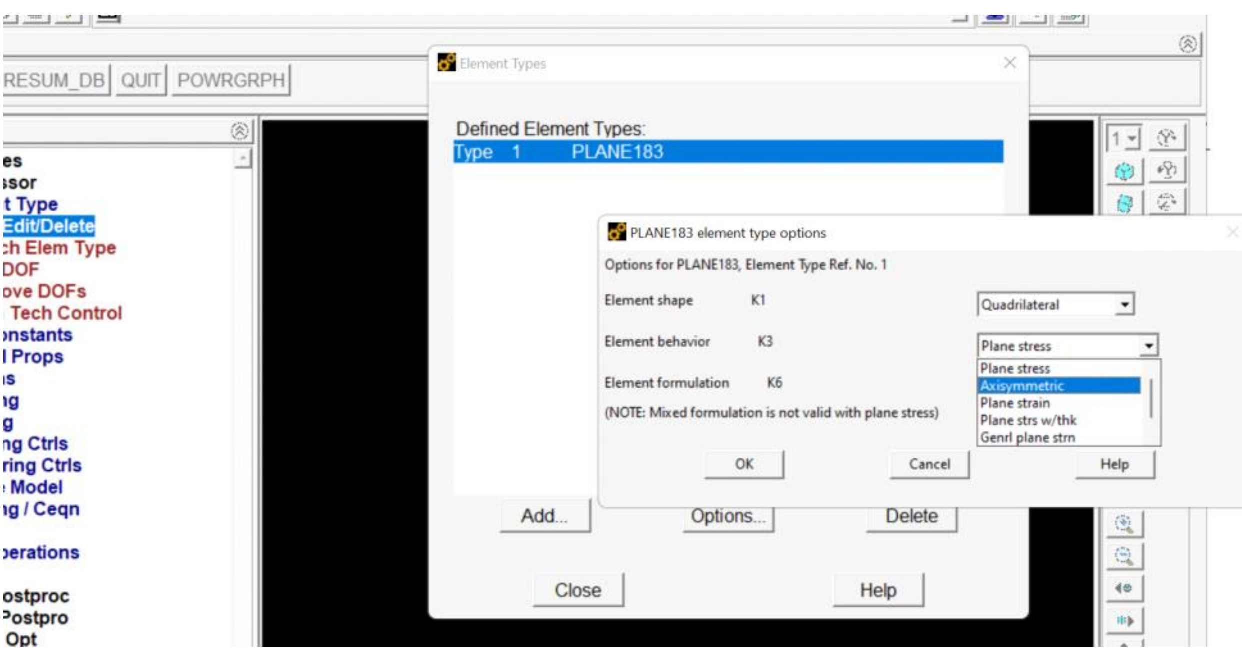
ν = .3

E = 1000

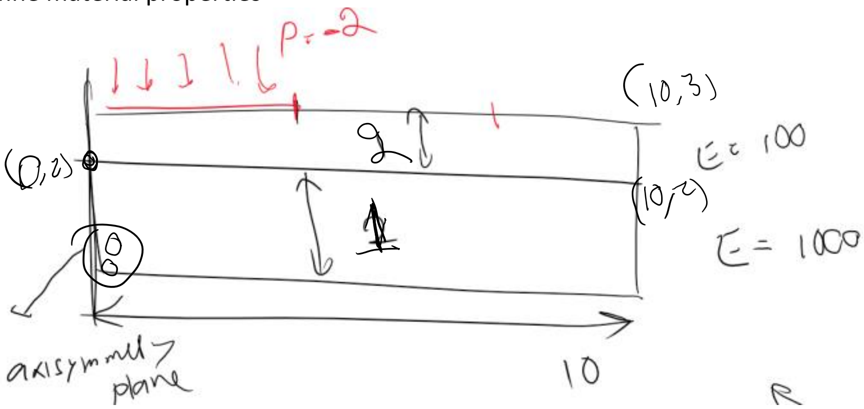
Define the element type



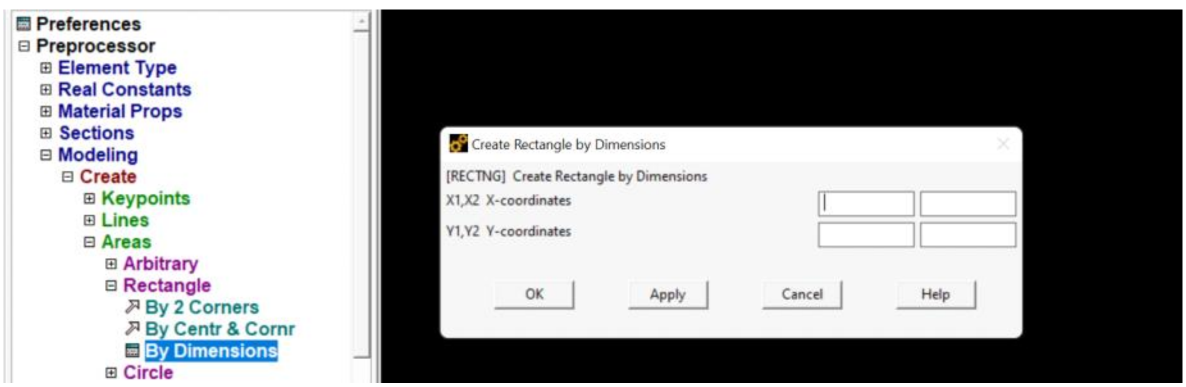


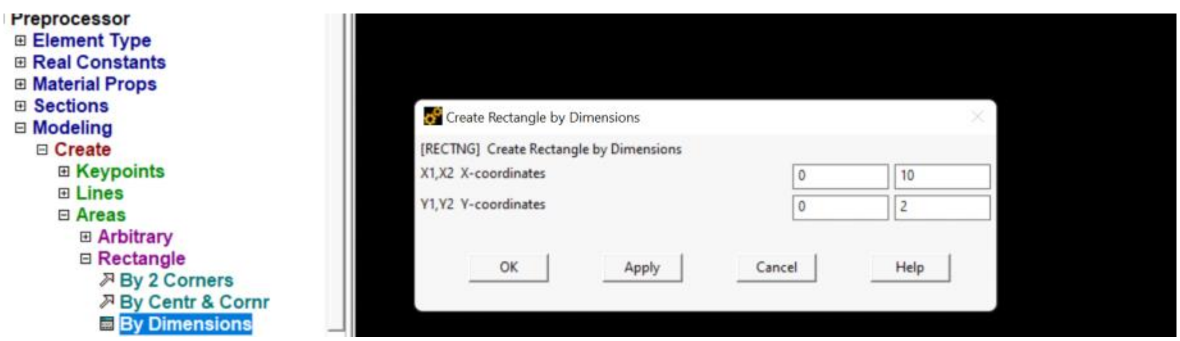


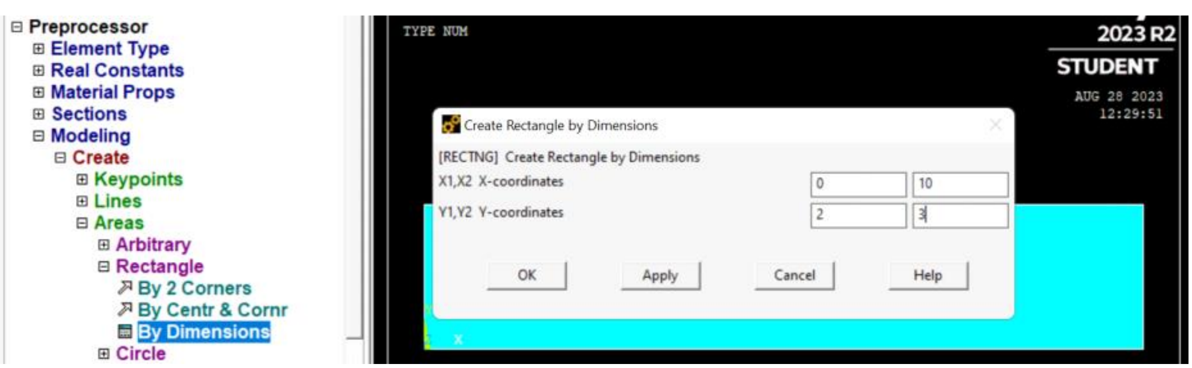
Define material properties



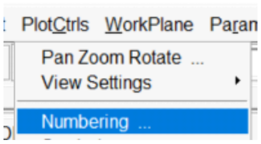
Define the geometry
I'm going to create areas and areas create lines and vertices (keypoints) associated with them

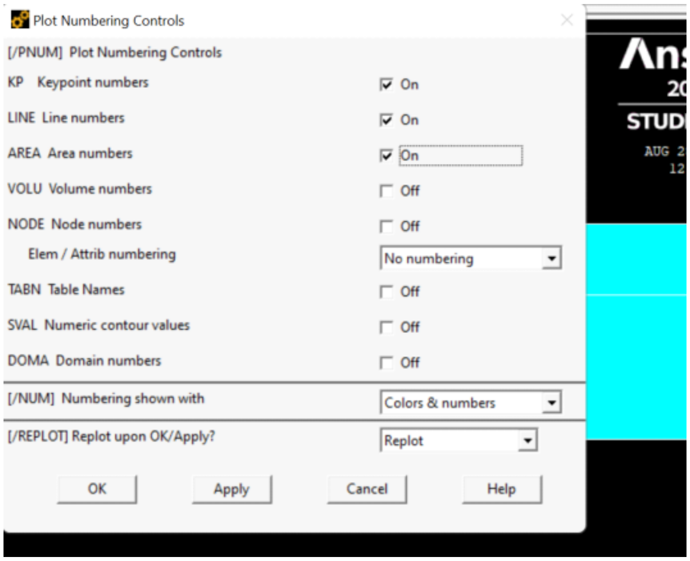


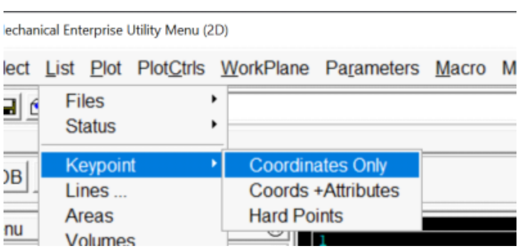




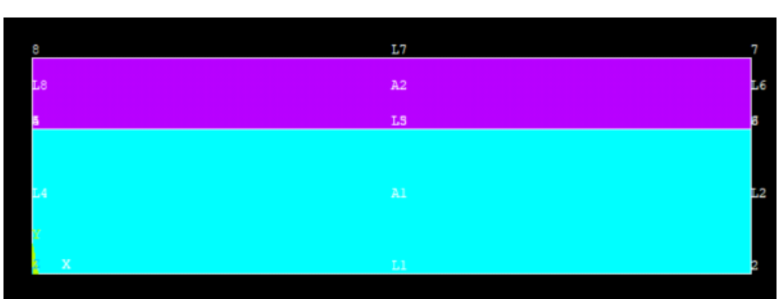
Show all geometry IDs



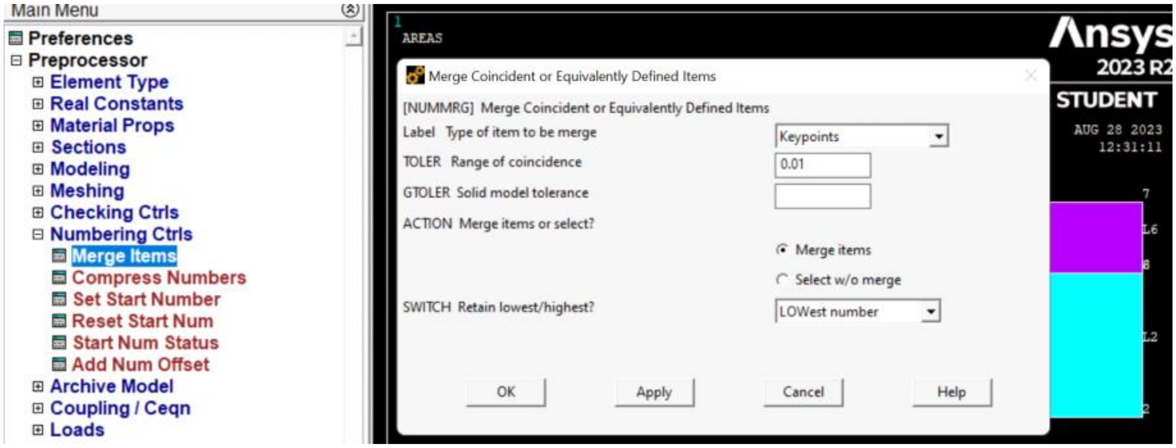


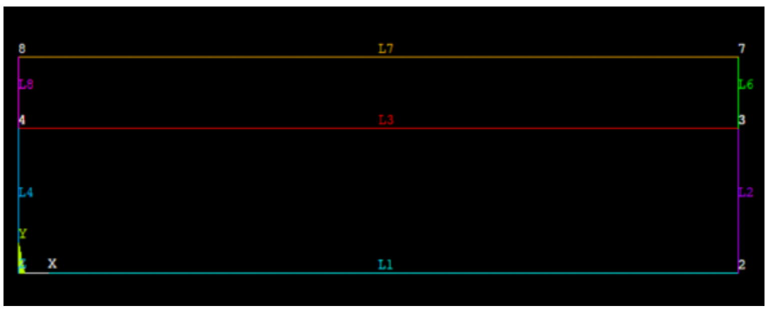


NO.	X,Y,Z LOCATION			THXY,THYZ,THZX ANGLES		
1	0.000000	0.000000	0.000000	0.0000	0.0000	0.0000
2	10.00000	0.000000	0.000000	0.0000	0.0000	0.0000
3	10.00000	2.000000	0.000000	0.0000	0.0000	0.0000
4	0.000000	2.000000	0.000000	0.0000	0.0000	0.0000
5	0.000000	2.000000	0.000000	0.0000	0.0000	0.0000
6	10.00000	2.000000	0.000000	0.0000	0.0000	0.0000
7	10.00000	3.000000	0.000000	0.0000	0.0000	0.0000
8	0.000000	3.000000	0.000000	0.0000	0.0000	0.0000



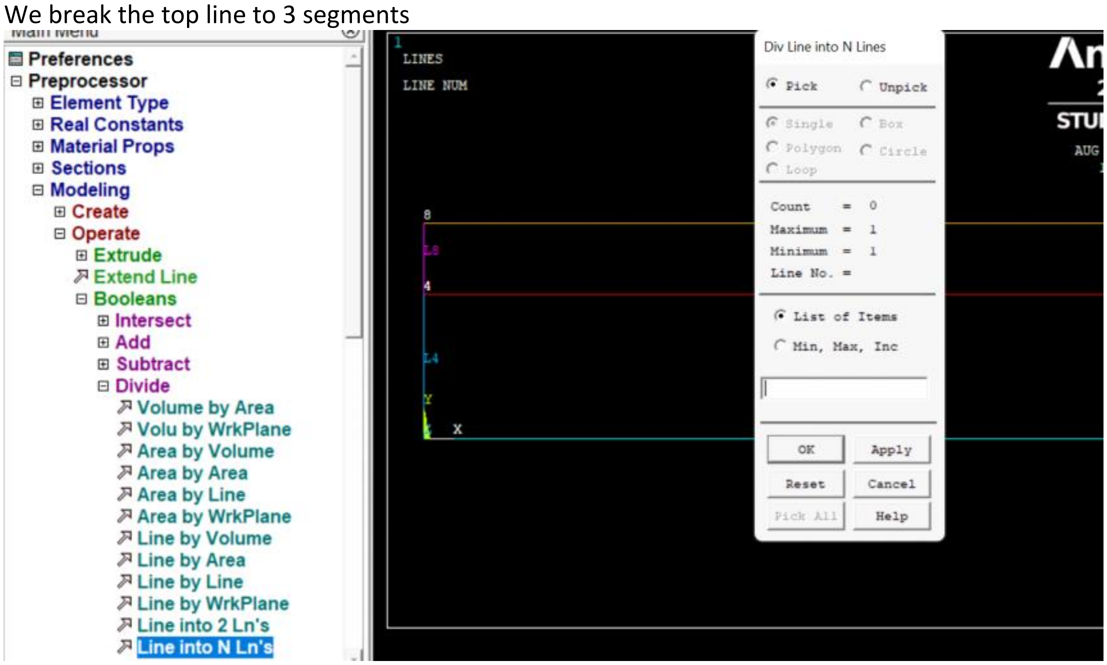
We need to merge the keypoints between the two rectangles so the rectangles are attached

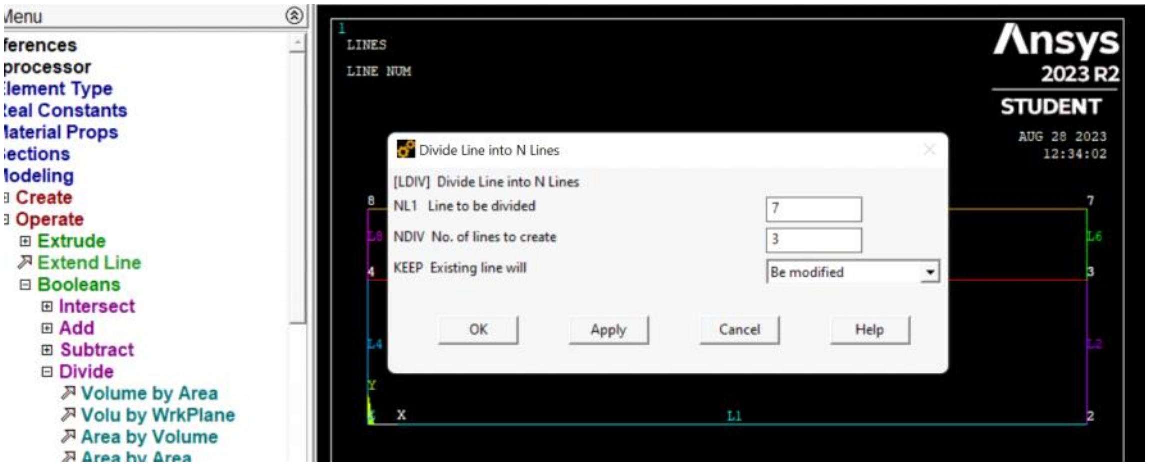




NO.	X,Y,Z LOCATION			THXY,THYZ,THZX ANGLES		
1	0.000000	0.000000	0.000000	0.0000	0.0000	0.0000
2	10.00000	0.000000	0.000000	0.0000	0.0000	0.0000
3	10.00000	2.000000	0.000000	0.0000	0.0000	0.0000
4	0.000000	2.000000	0.000000	0.0000	0.0000	0.0000
7	10.00000	3.000000	0.000000	0.0000	0.0000	0.0000
8	0.000000	3.000000	0.000000	0.0000	0.0000	0.0000

We break the top line to 3 segments




Menu
ferences
processor
Element Type
Real Constants
Material Props
Sections
Modeling
Create
Operate
Extrude
Extend Line
Booleans
Intersect
Add
Subtract
Divide
Volume by Area
Volu by WrkPlane
Area by Volume
LINES
LINE NUM
Ansys
2023 R2
STUDENT
AUG 28 2023
12:34:02
Divide Line into N Lines
[LDIV] Divide Line into N Lines
NL1 Line to be divided
7
NDIV No. of lines to create
3
KEEP Existing line will
Be modified
OK
Apply
Cancel
Help
L1