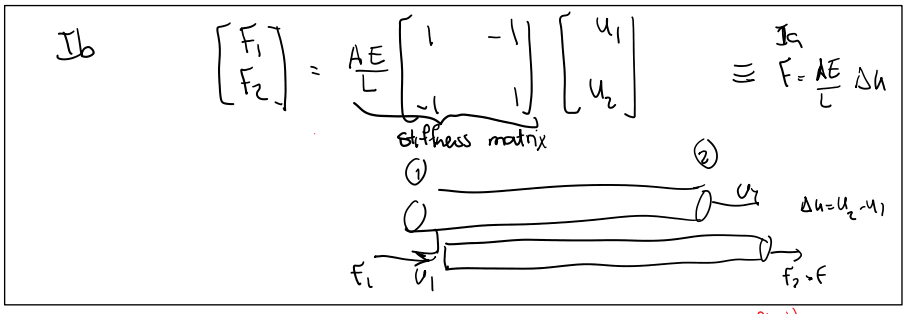
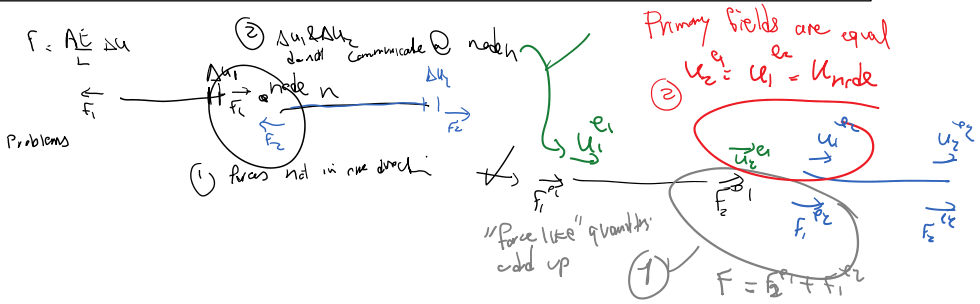
2022/08/30

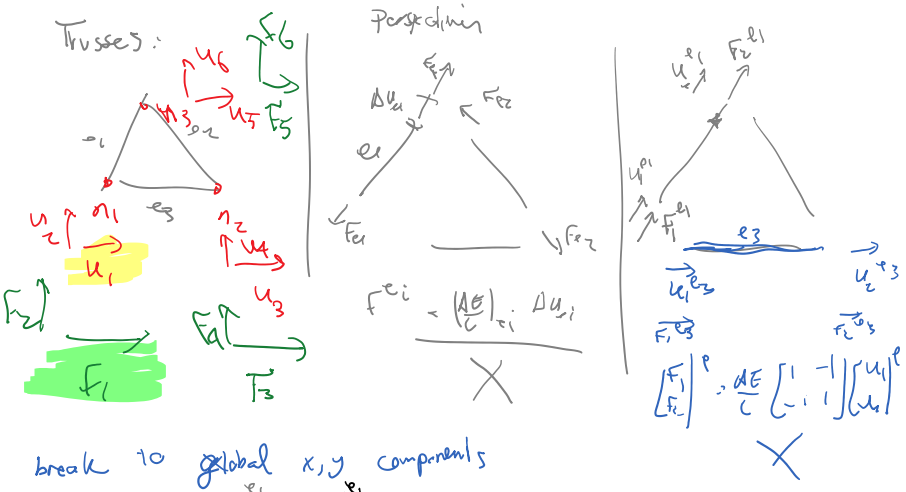
Tuesday, August 30, 2022 11:16 AM

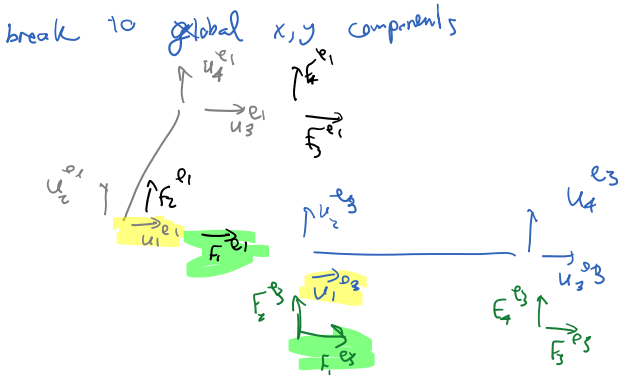
From last time:





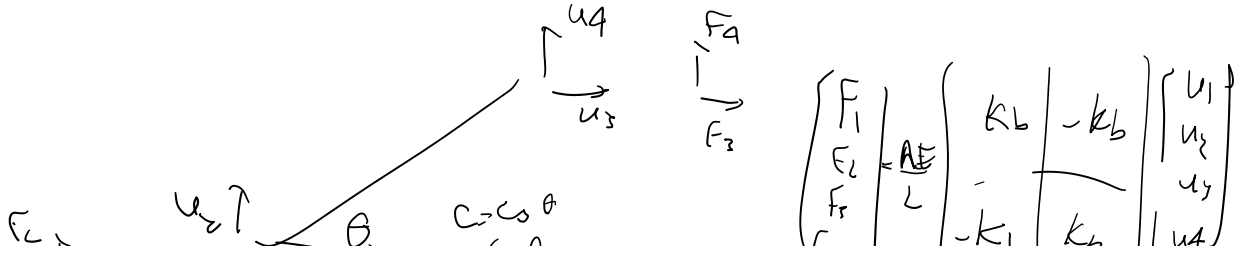
$$\therefore \begin{bmatrix} F_1^e \\ F_2^e \end{bmatrix} = \frac{AE}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{bmatrix} u_1^e \\ u_2^e \end{bmatrix}$$



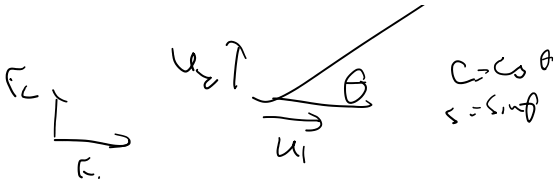


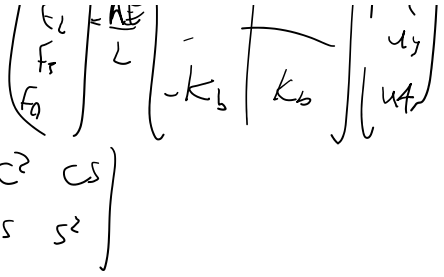
$u_1^{e_1} = u_1^{e_2} = u_1$ displacements are equal

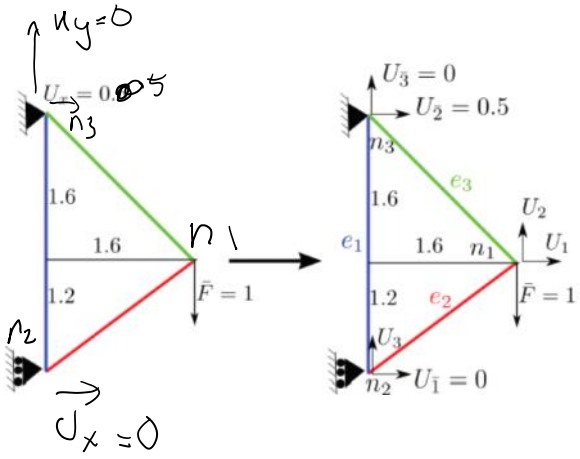
$F_1^{e_1} + F_1^{e_2} = F_1$ Forces add

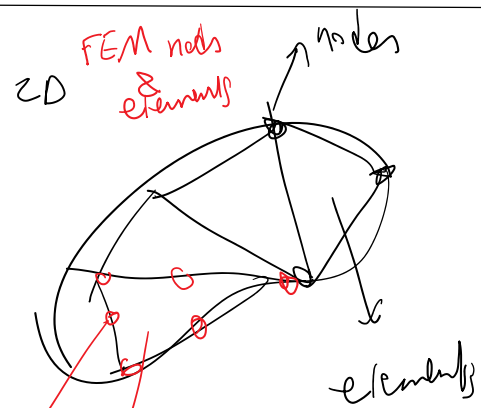


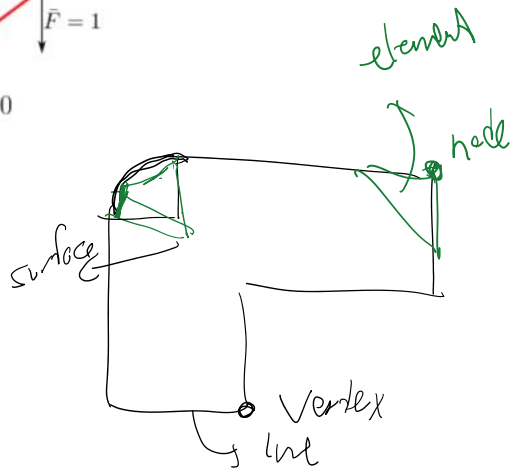
$$\begin{bmatrix} F_1 \\ F_2 \\ F_3 \end{bmatrix} = \frac{AE}{L}\begin{bmatrix} k_b & & -k_b \\ & & \\ -k_1 & & k_b \end{bmatrix}\begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix}$$







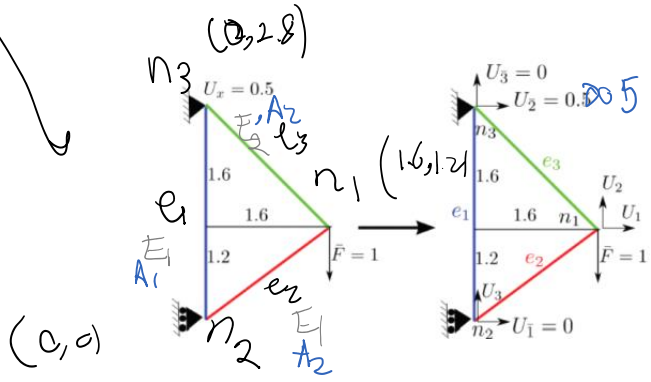


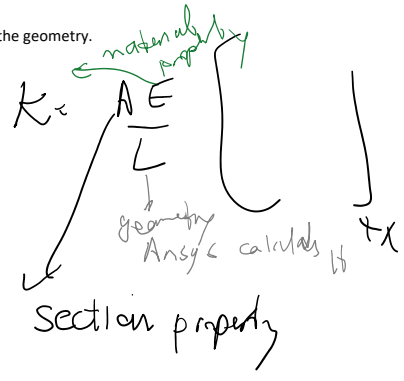


= Geometry
-FEM

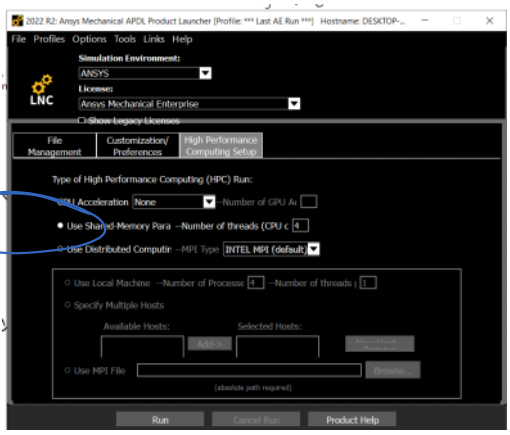
FEM packages:
- We always work with geometry objects (vertices, lines, surfaces, volumes). For example, we even apply the loads and other BCs on geometry. Finally, we mesh the geometry.
- The only exception is for domains with 1D elements. In this case, we directly specify FEM objects (nodes and elements)

materials
$E_1 = 200$
$E_2 = 10$
section
2 section
$A_1 = 10$
$A_2 = 100$



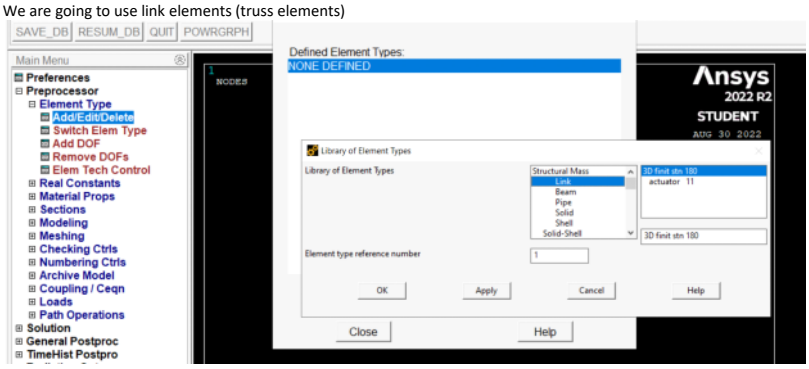


Launch Ansys Product Launcher



1. Define elements to be used

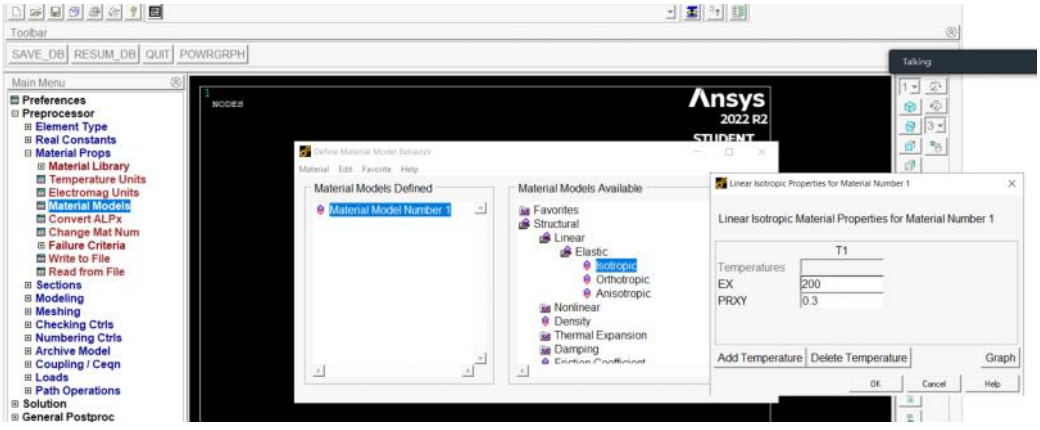
We are going to use link elements (truss elements)

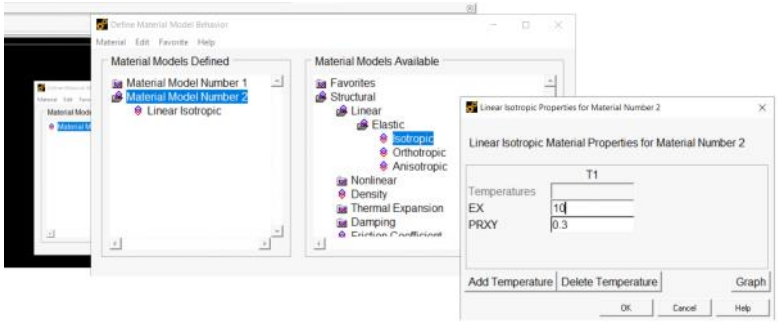


2. Add material properties

E1 = 200

E2 = 10

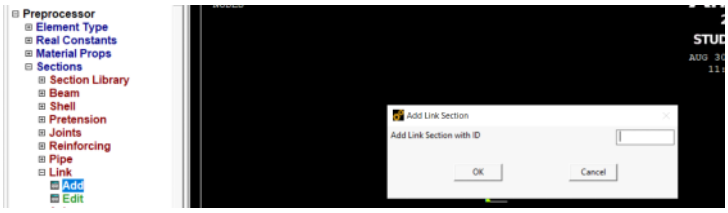


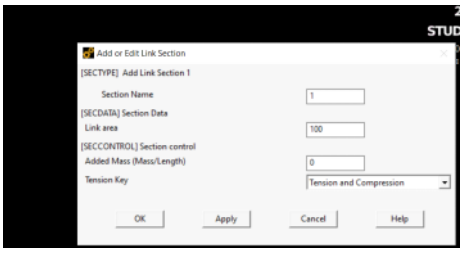


3. Define section properties

$A_1 = 10$

$A_2 = 100$

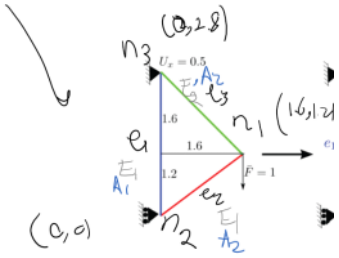


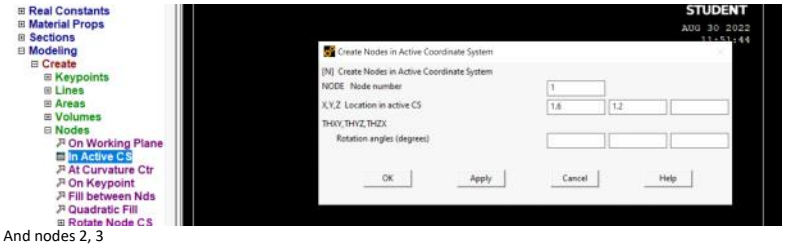


Add section 2

You can list materials and sections

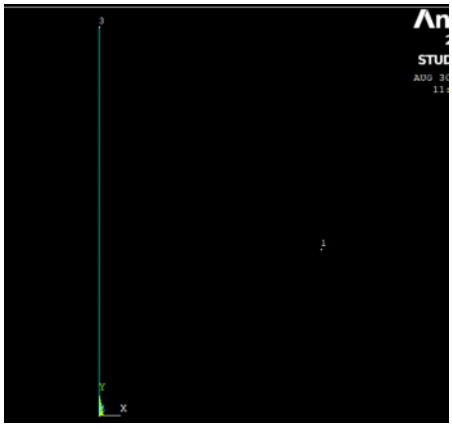
4. Define nodes:



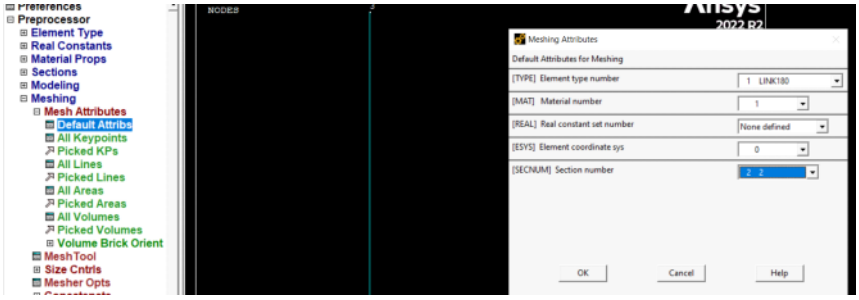


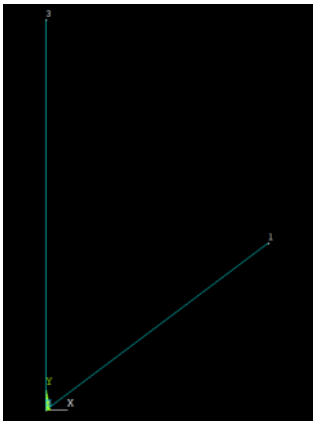
And nodes 2, 3

5. Step 4: define elements
   - Choose default material number
   - Choose default section number
   - Define element passing through nodes

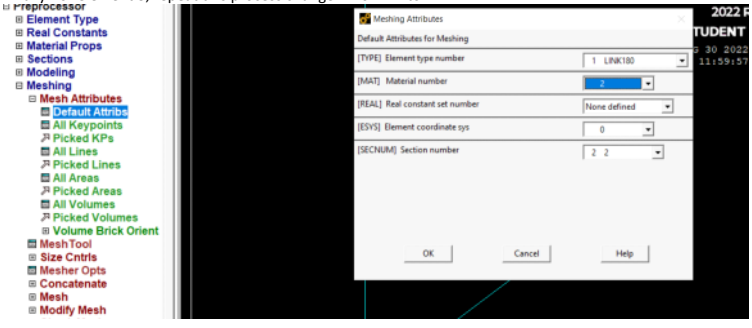


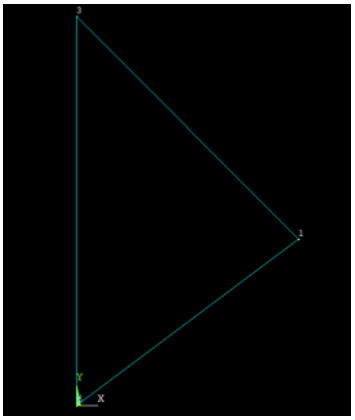
For element 2: change A from A1 to A2



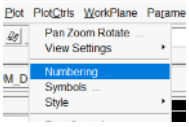


Finally for element 3, repeat this process change E from E1 to E2

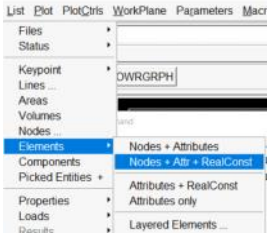




Show the element numbering



If want to check elements are formed correctly

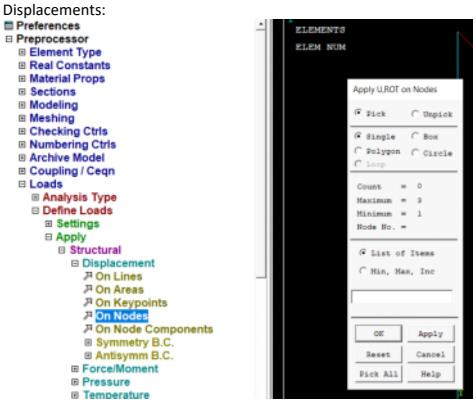


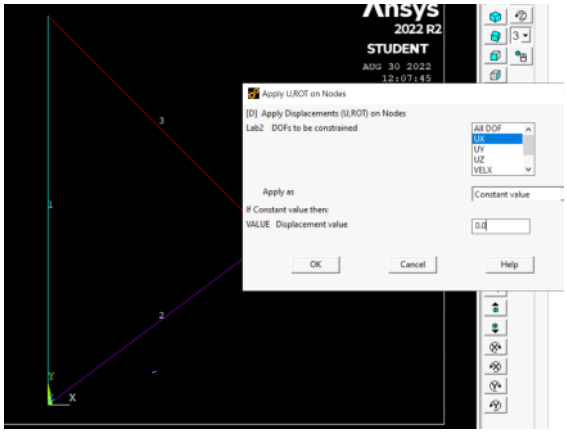
LIST ALL SELECTED ELEMENTS. (LIST NODES)

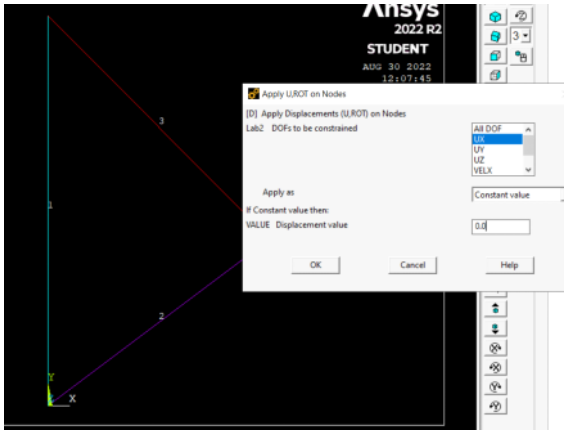
ELEM MAT TYP REL ESY SEC NODES

1 1 1 1 0 1 2 3
2 1 1 1 0 2 2 1
3 2 1 1 0 2 3 1

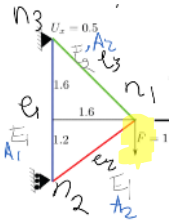
6. BCs:
   a. Displacements:

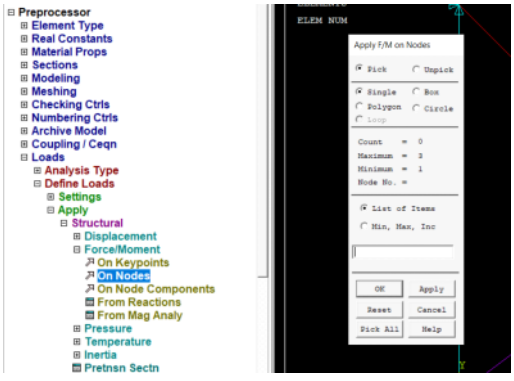


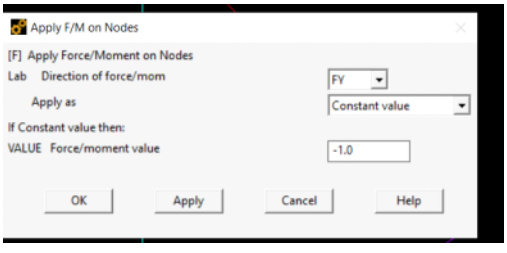




b. Forces



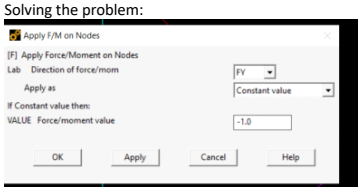




Preprocessor stage is finished.
Solving the problem:

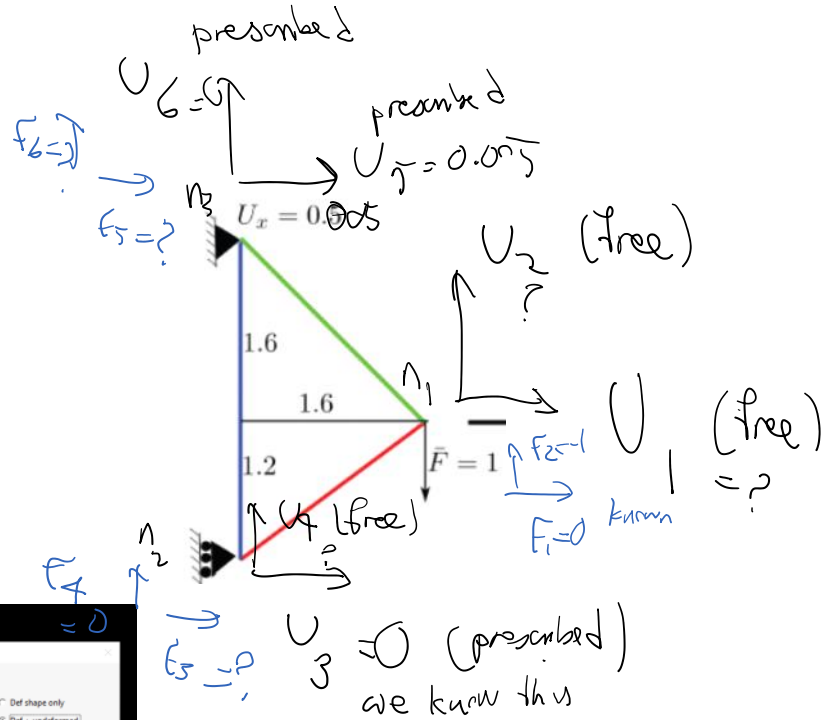
presented

Preprocessor stage is finished.
Solving the problem:

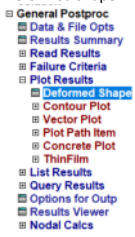


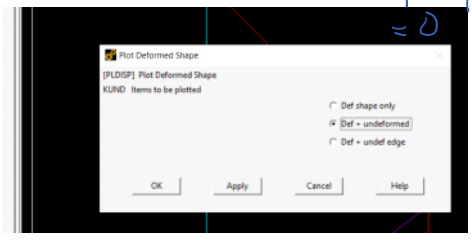
Postprocess:
View and list the results

1. Deformed shape
2. Elements solutions (axial force and stress)
3. Displacements for free degrees of freedom (dof)
4. Forces for prescribed dofs (called Reaction forces)



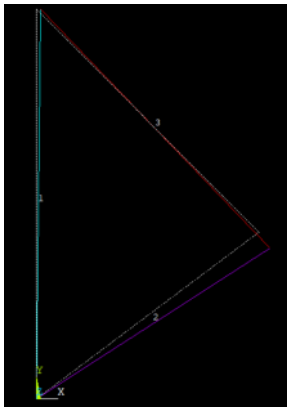
1. Deformed shape



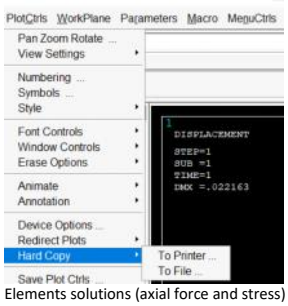


Prescribed, Dirichlet dof:
- we know the displacement (primary field)
- We don't know the force

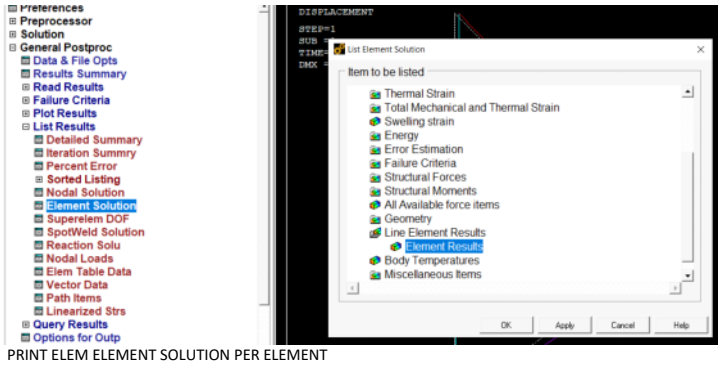
Free (Neumann) dof
- We don't know the displacement (primary field)
- We know the force



2.



2. Elements solutions (axial force and stress)



PRINT ELEM ELEMENT SOLUTION PER ELEMENT

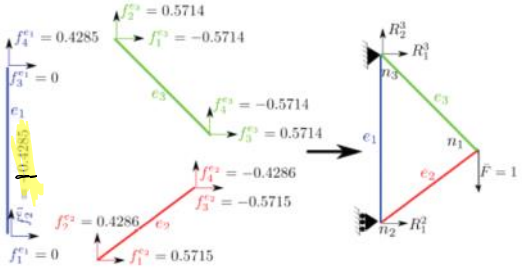
***** POST1 ELEMENT SOLUTION LISTING *****

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

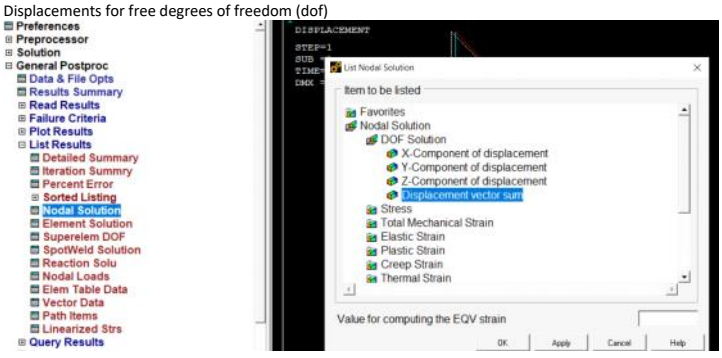
EL= 1 NODES= 2 3 MAT= 1 XC,YC,ZC= 0.000 1.400 0.000 AREA= 100.00 LIN K180
FORCE= 0.42857 STRESS= 0.42857E-02 EPEL= 0.21429E-04
TEMP= 0.00 0.00 EPTH= 0.0000

EL= 2 NODES= 2 1 MAT= 1 XC,YC,ZC= 0.8000 0.6000 0.000 AREA= 10.000 LIN K180
FORCE=-0.71429 STRESS=-0.71429E-01 EPEL=-0.35714E-03
TEMP= 0.00 0.00 EPTH= 0.0000

EL= 3 NODES= 3 1 MAT= 2 XC,YC,ZC= 0.8000 2.000 0.000 AREA= 10.000 LIN K180
FORCE= 0.80812 STRESS= 0.80812E-01 EPEL= 0.80812E-02
TEMP= 0.00 0.00 EPTH= 0.0000



3. Displacements for free degrees of freedom (dof)



PRINT U NODAL SOLUTION PER NODE

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

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

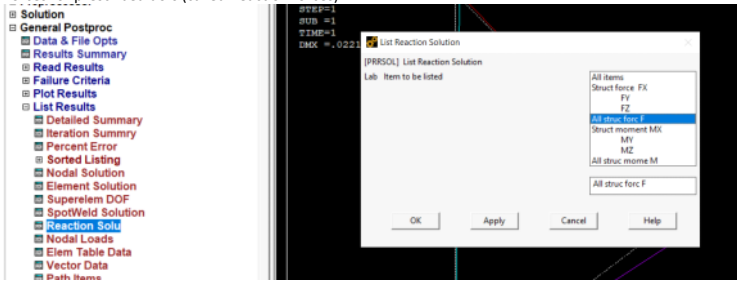
THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN THE GLOBAL COORDINATE SYSTEM

free dofs

NODE	UX	UY	UZ	USUM
1	0.12690E-001	-0.18170E-001	0.0000	0.22163E-001
2	0.0000	-0.60000E-004	0.0000	0.60000E-004
3	0.50000E-002	0.0000	0.0000	0.50000E-002

prescribed

4. Forces for prescribed dofs (called Reaction forces)



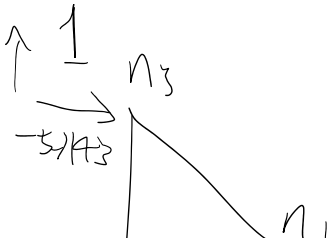
PRINT F REACTION SOLUTIONS PER NODE

***** 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	FY	FZ

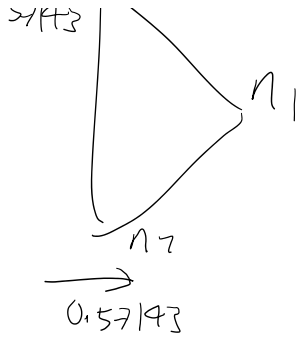


TIME= 1.0000 LOAD CASE= 0

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

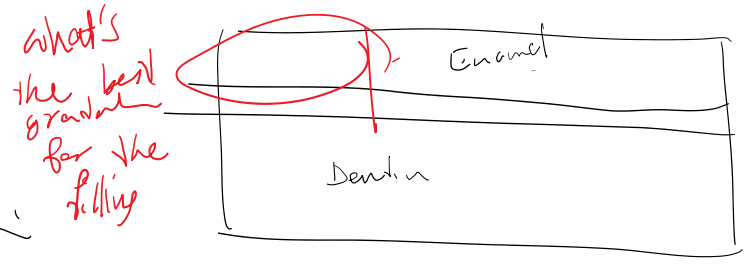
NODE	FX	FY	FZ
2	0.57143		
3	-0.57143	1.0000	

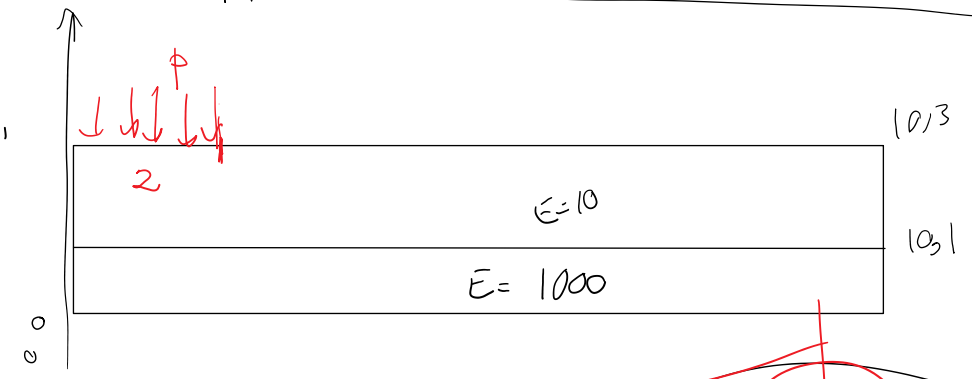
TOTAL VALUES
VALUE -0.77716E-015 1.0000 0.0000



HW1 and final project is a truss problem

2D example
Project 1, a dental crown problem

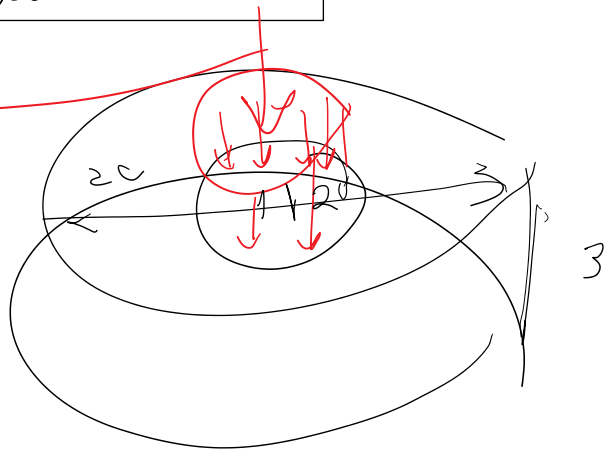


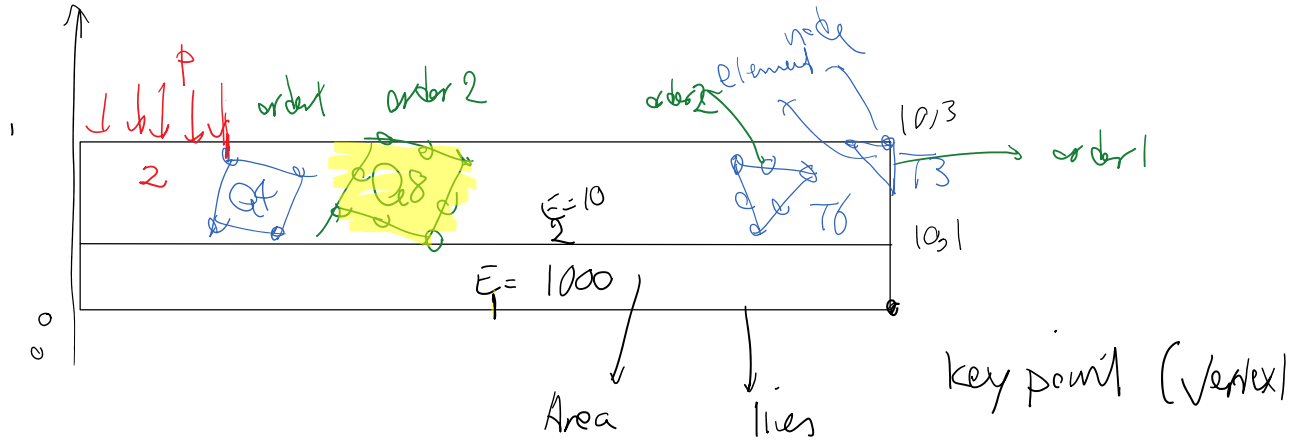


resultant force

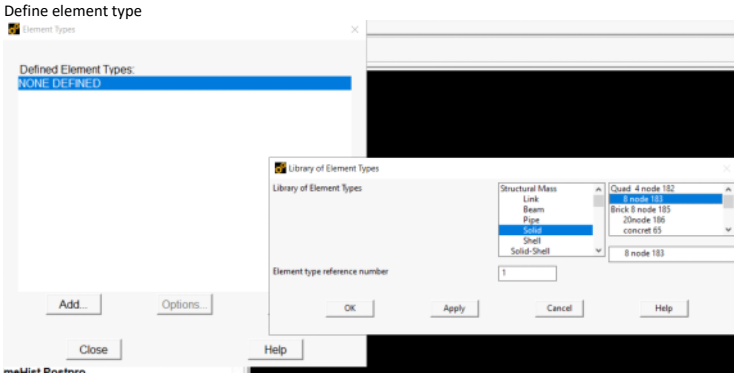
$F = \pi r^2 p$

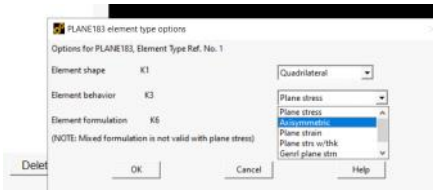
$p = \frac{F}{\pi r^2}$



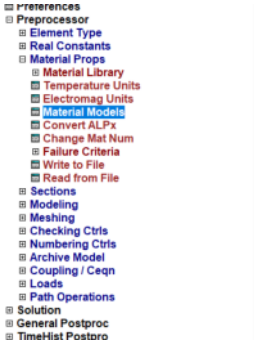


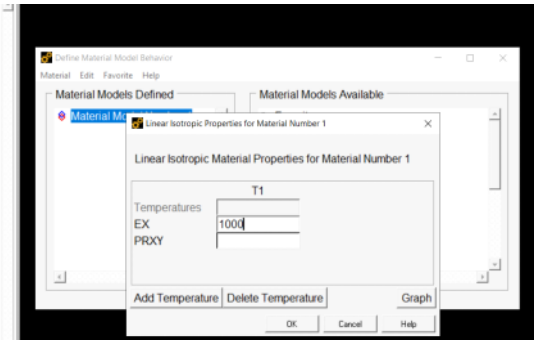
1. Define element type





2. Materials





And then 2