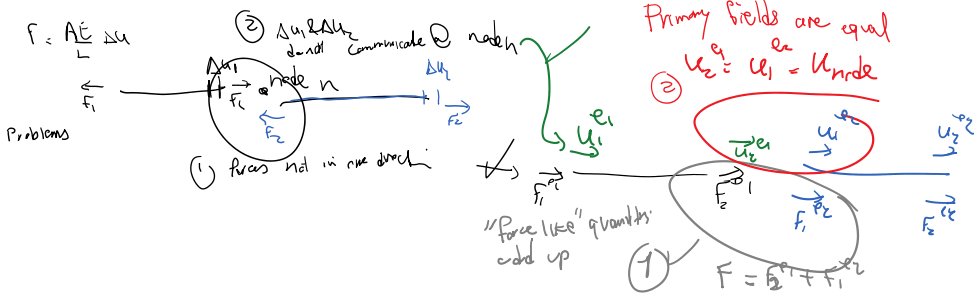
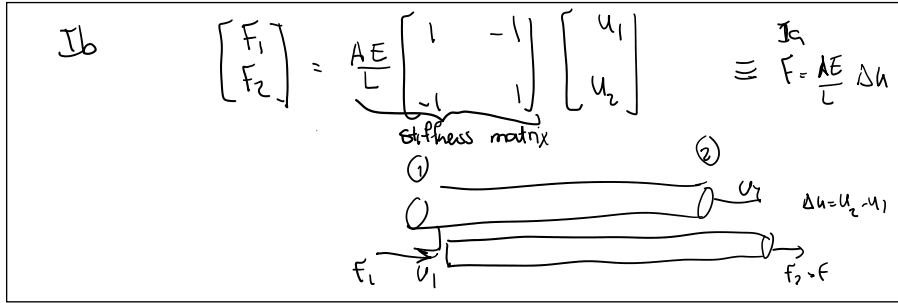
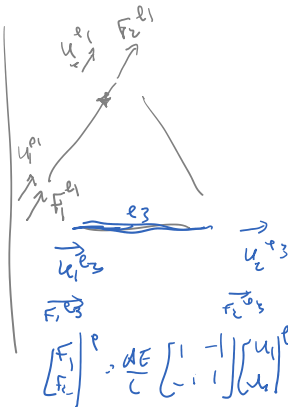
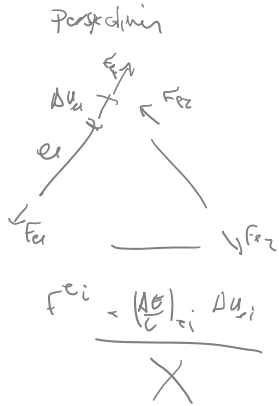
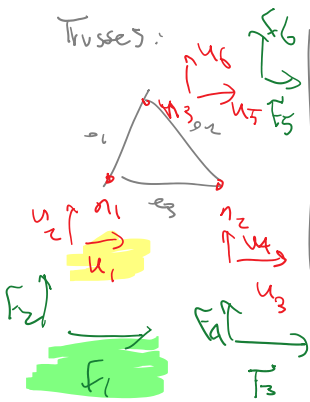


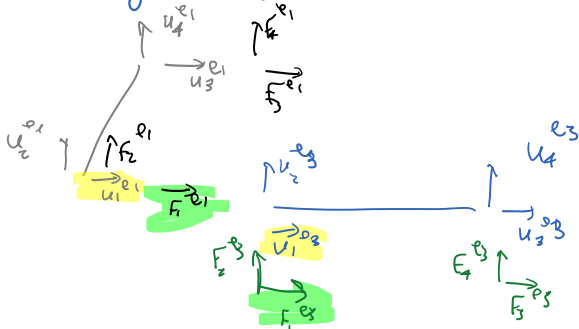
From last time:



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



break to global x,y components

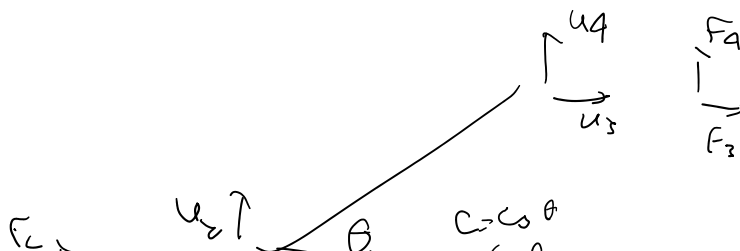


$u_1^e = u_1^g = u_1$

$F_1^e + F_1^g = F_1$

displacements are equal

Forces add

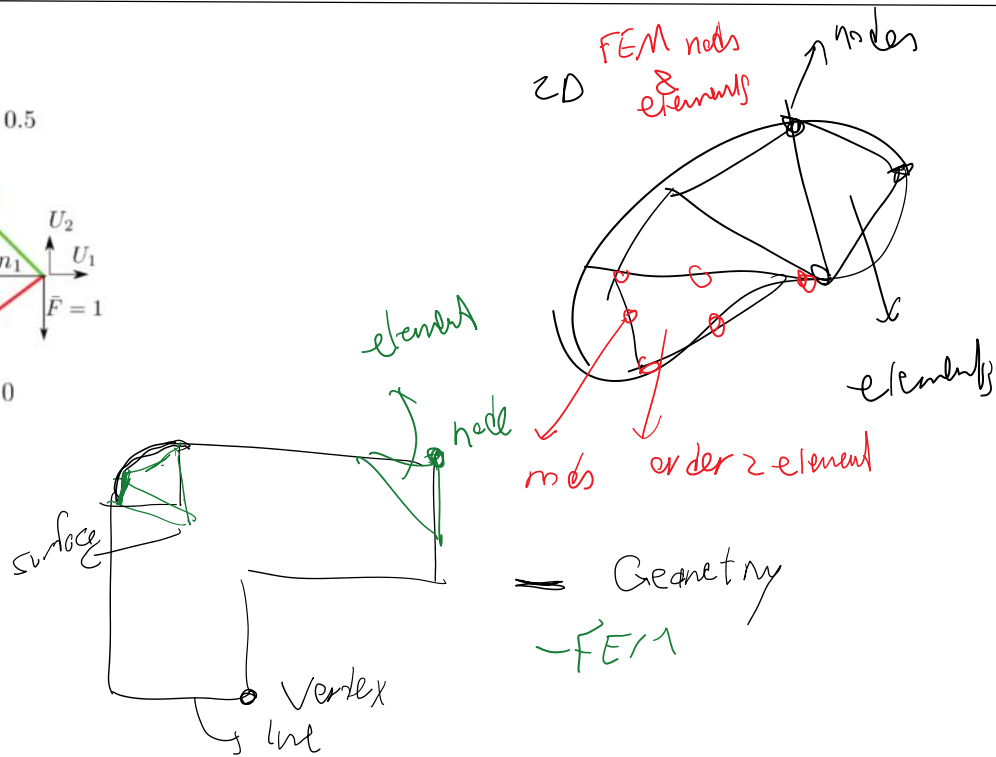
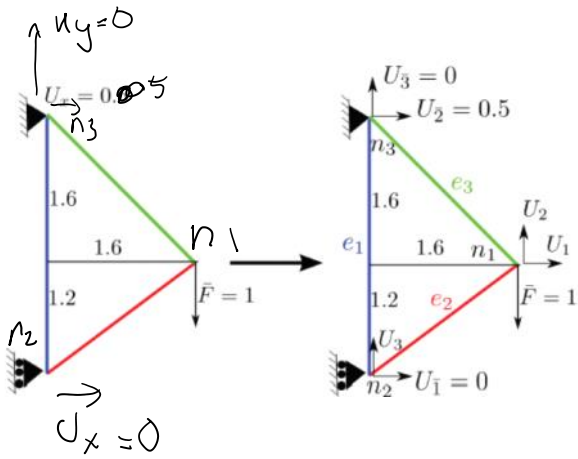


$$\begin{bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \end{bmatrix} = \frac{AE}{L} \begin{bmatrix} k_b & -k_b \\ -k_b & k_b \\ -k_1 & k_1 \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix}$$

$c = \cos \theta$
 $s = \sin \theta$

$$k_b = \begin{bmatrix} c^2 & cs \\ cs & s^2 \end{bmatrix}$$

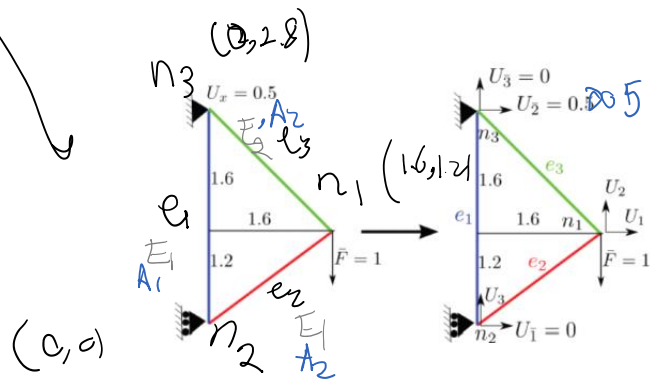
$$\begin{bmatrix} F_x \\ F_y \\ F_z \end{bmatrix} = \frac{AE}{L} \begin{bmatrix} -k_b & k_b \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \end{bmatrix}$$



FEM packages:

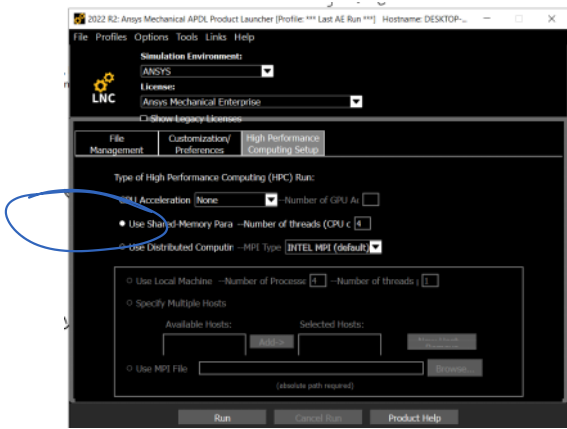
- We always work with geometry objects (vertices, lines, surfaces, volumes). For example, we even apply the loads and other BCs on geometry. Finally, we mesh the geometry.
- The only exception is for domains with 1D elements. In this case, we directly specify FEM objects (nodes and elements)

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



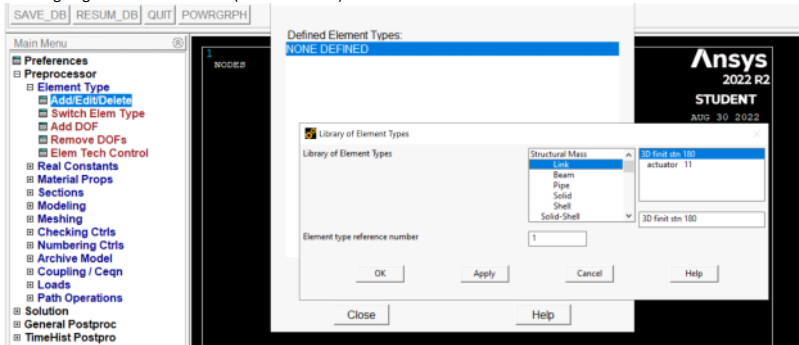
$$k = \frac{AE}{L}$$

← material property
 ↓ geometry
 Ansys calculates it
 Section property



1. Define elements to be used

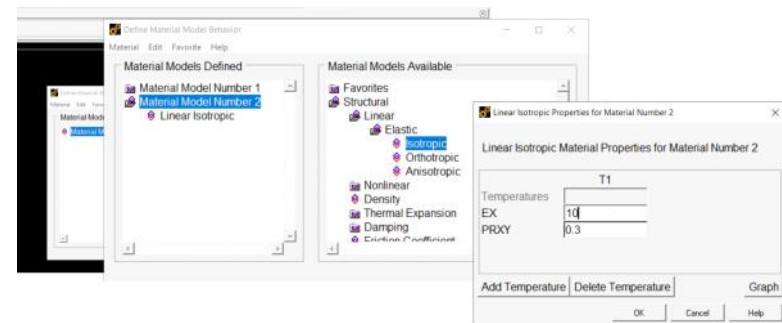
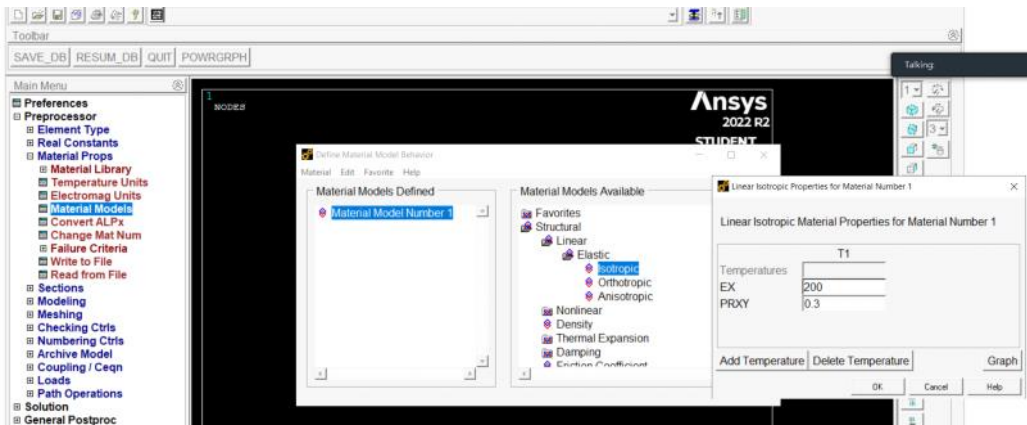
We are going to use link elements (truss elements)



2. Add material properties

E1 = 200

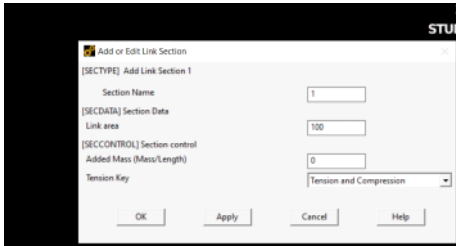
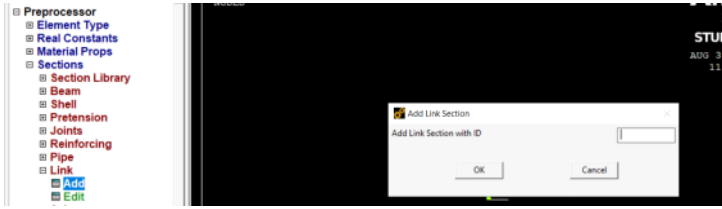
E2 = 10



3. Define section properties

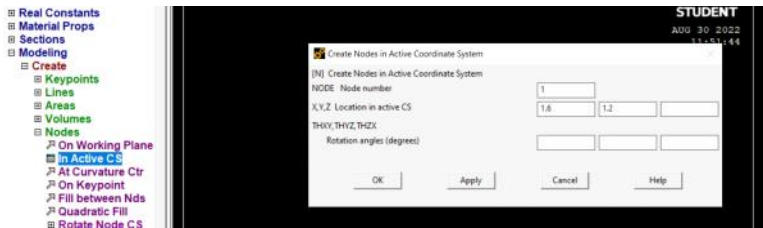
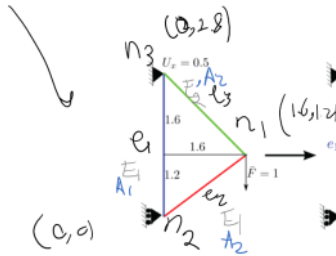
$$A_c = 10$$

$A_2 = 100$



Add section 2

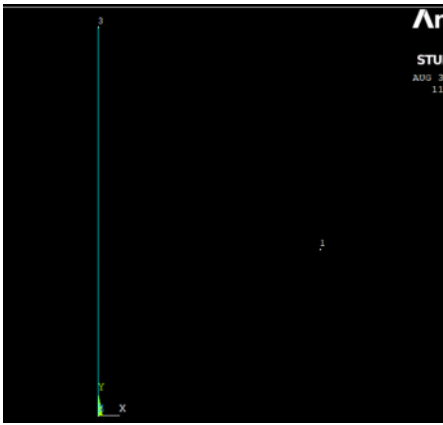
You can list materials and sections
4. Define nodes:



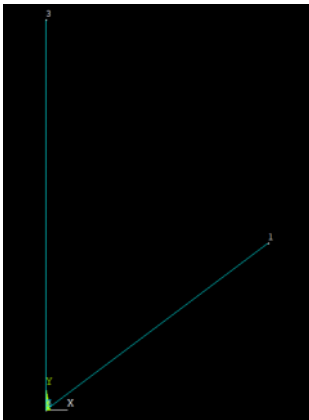
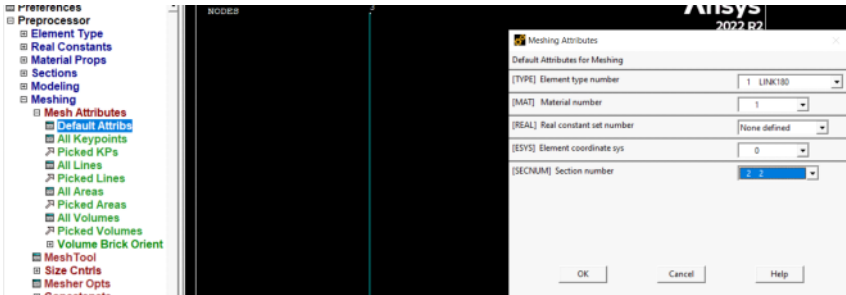
And nodes 2, 3

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

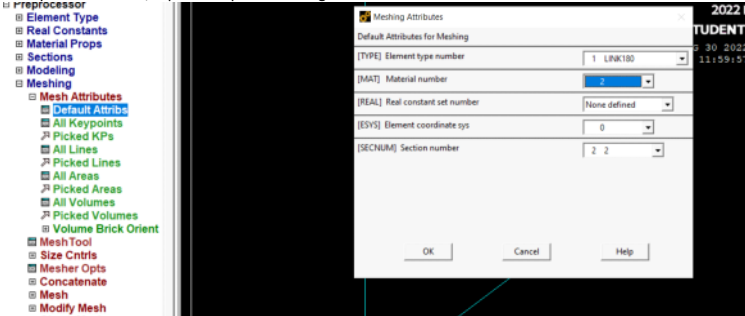


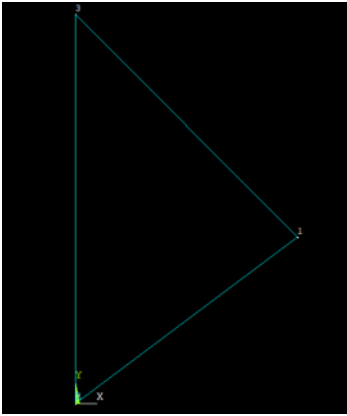


For element 2: change A from A1 to A2



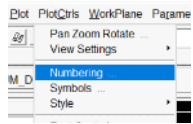
Finally for element 3, repeat this process change E from E1 to E2



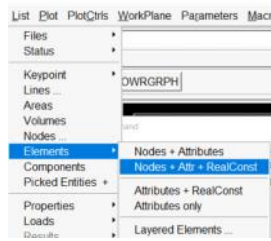


Show the element numbering

ip:file:lab1by:ME:en



If want to check elements are formed correctly



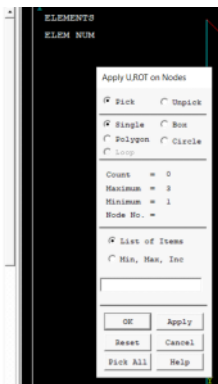
LIST ALL SELECTED ELEMENTS. (LIST NODES)

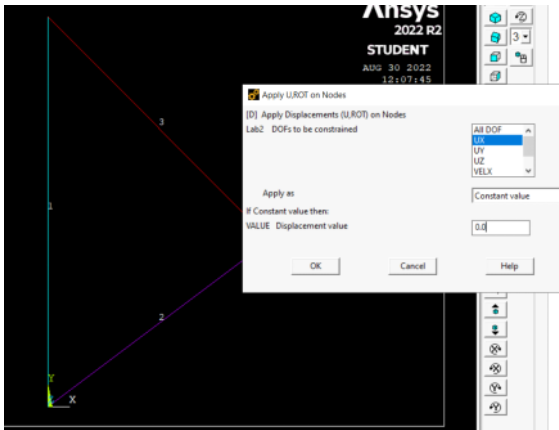
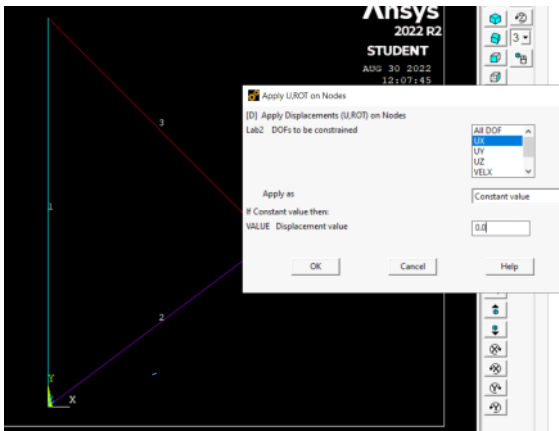
ELEM	MAT	TYP	REL	ESY	SEC	NODES
1	1	1	1	0	1	2 3
2	1	1	1	0	2	2 1
3	2	1	1	0	2	3 1

6. BCs:

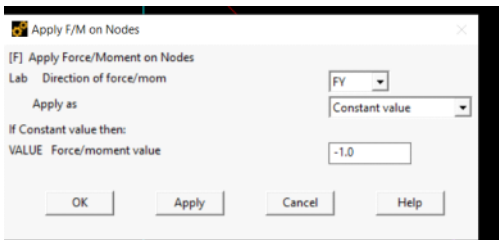
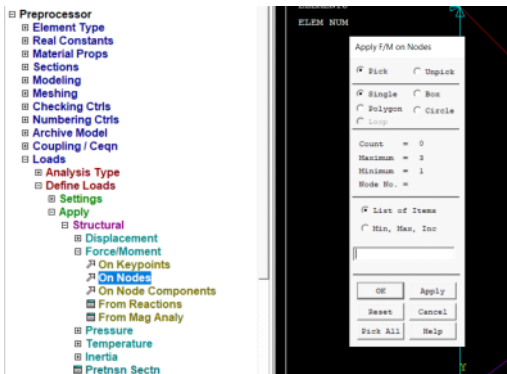
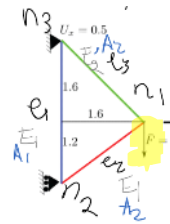
a. Displacements:

- Preferences
- Preprocessor
 - Element Type
 - Real Constants
 - Material Props
 - Sections
 - Modeling
 - Meshing
 - CheckingCtrls
 - NumberingCtrls
 - Archive Model
 - Coupling / Ceqn
 - Loads
 - Analysis Type
 - Define Loads
 - Settings
 - Apply
 - Structural
 - Displacement
 - On Lines
 - On Areas
 - On Keypoints
 - On Nodes
 - On Node Components
 - Symmetry B.C.
 - Antisymm B.C.
 - Force/Moment
 - Pressure
 - Temperature





b. Forces

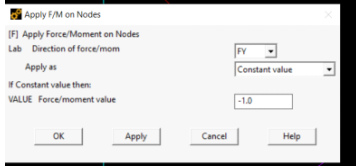


Preprocessor stage is finished.
Solving the problem:

prescribed

Preprocessor stage is finished.

Solving the problem:



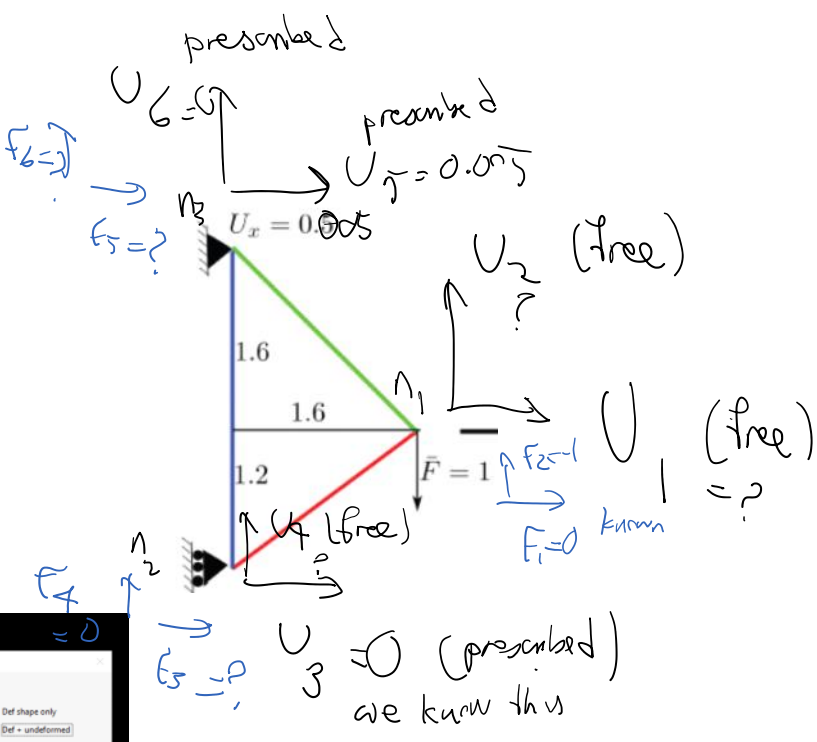
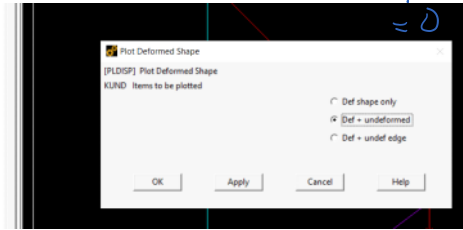
Postprocess:

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

- General Postproc
- Data & File Optns
- Results Summary
- Read Results
- Failure Criteria
- Plot Results
 - Deformed Shape
 - Contour Plot
 - Vector Plot
 - Plot Path Item
 - Concrete Plot
 - ThinFilm
- List Results
- Query Results
- Options for Outp
- Results Viewer
- Nodal Calcs

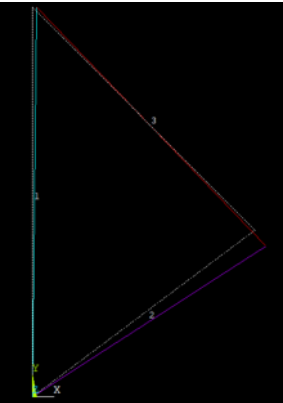


Prescribed, Dirichlet dof:

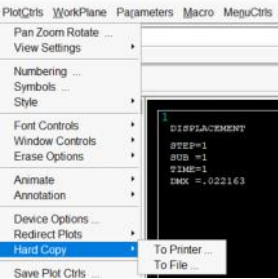
- we know the displacement (primary field)
- We don't know the force

Free (Neumann) dof

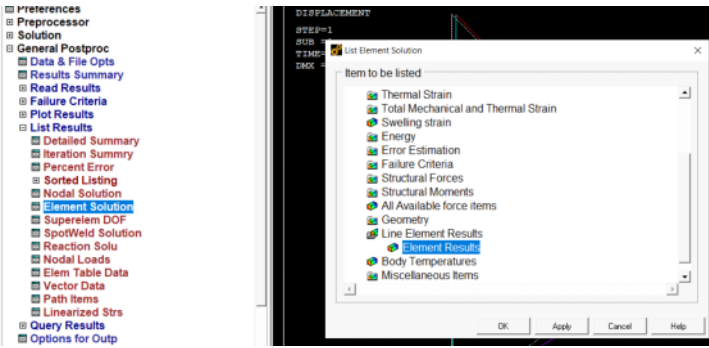
- We don't know the displacement (primary field)
- We know the force



2.



2. Elements solutions (axial force and stress)



PRINT ELEM ELEMENT SOLUTION PER ELEMENT

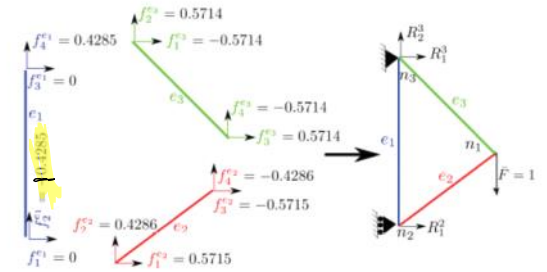
***** POST1 ELEMENT SOLUTION LISTING *****

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

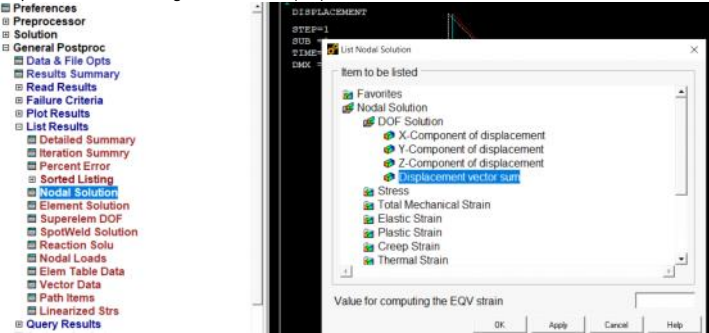
EL= 1 NODES= 2 3 MAT= 1 XC,YC,ZC= 0.000 1.400 0.000 AREA= 100.00 LIN K180
 FORCE=0.42857 STRESS=0.42857E-02 EPEL=0.21429E-04
 TEMP= 0.00 0.00 EPTH= 0.00000

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= 0.00000

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= 0.00000



3. Displacements for free degrees of freedom (dof)



PRINT U NODAL SOLUTION PER NODE

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

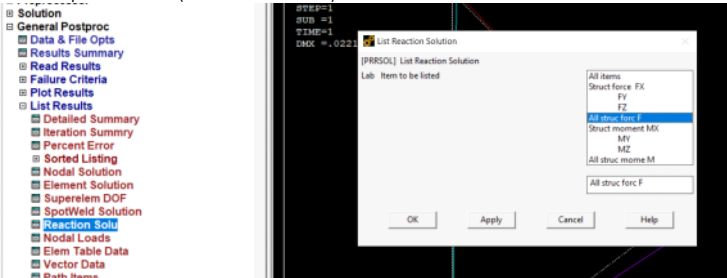
LOAD STEP= 1 SUBSTEP= 1
 TIME= 1.0000 LOAD CASE= 0

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

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

free dof (handwritten above UX, UY, UZ)
prescribed (handwritten below UX, UY, UZ)

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
------	----	----	----



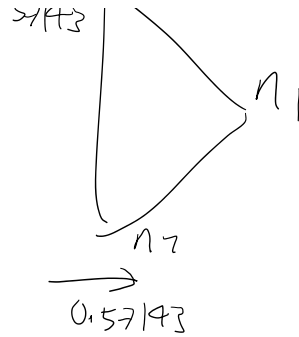
TIME= 1.0000 LOAD CASE= 0

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

NODE	FX	FY	FZ
2	0.57143		
3	-0.57143	1.0000	

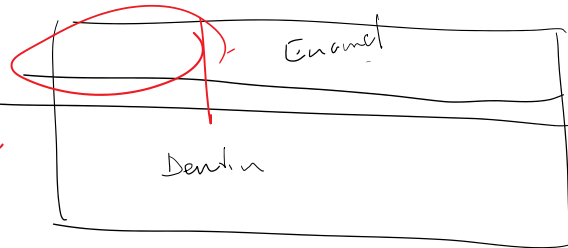
TOTAL VALUES
 VALUE -0.77716E-015 1.0000 0.0000

HW1 and final project is a truss problem

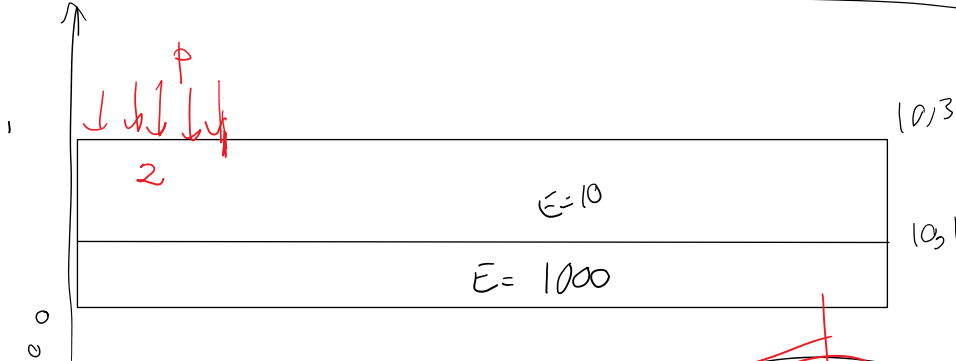


2D example
Project 1, a dental crown problem

what's the best gradient for the filling



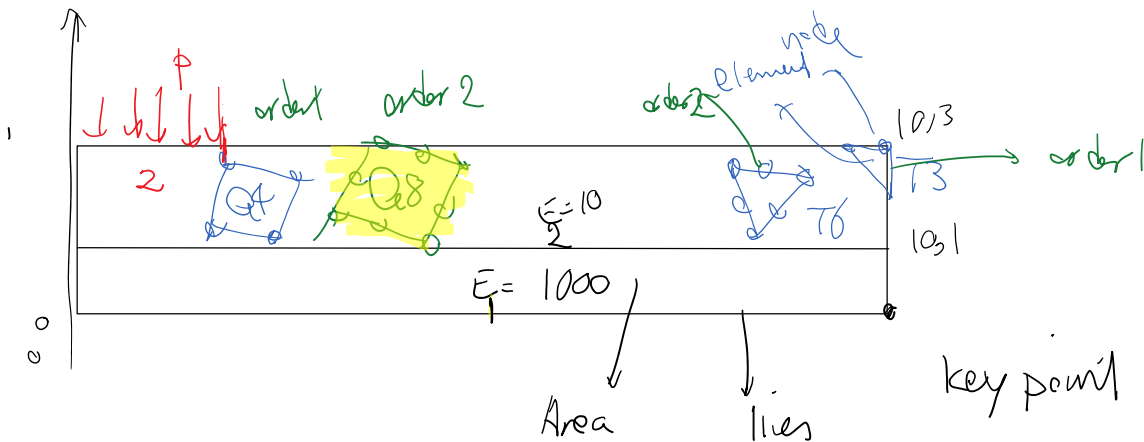
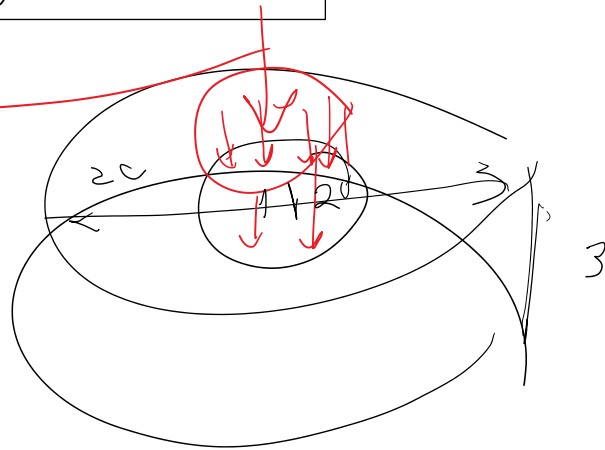
axis of rotation



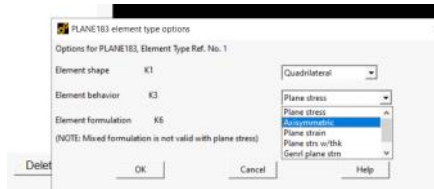
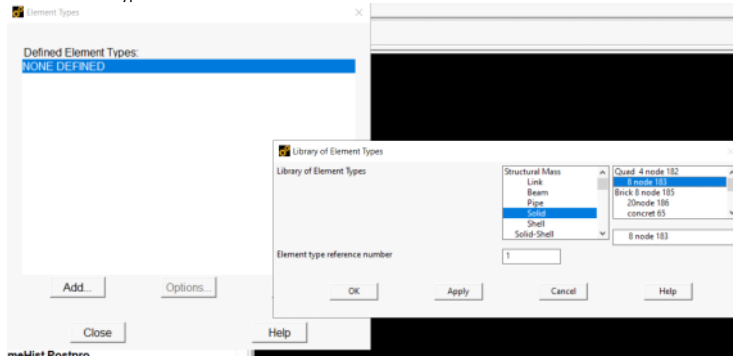
resultant force C

$$F = \pi r^2 p$$

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

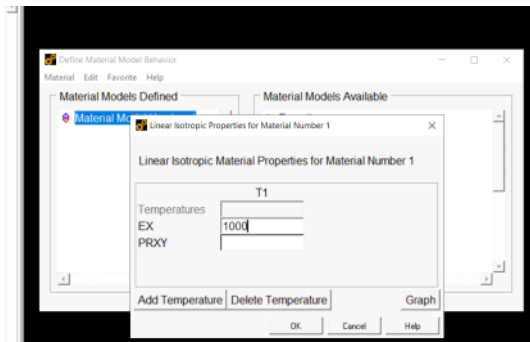


1. Define element type



2. Materials

- ▣ Preferences
- ▣ Preprocessor
 - ▣ Element Type
 - ▣ Real Constants
 - ▣ Material Props
 - ▣ Material Library
 - ▣ Temperature Units
 - ▣ Electromag Units
 - ▣ Material Models
 - ▣ Convert ALPx
 - ▣ Change Mat Num
 - ▣ Failure Criteria
 - ▣ Write to File
 - ▣ Read from File
 - ▣ Sections
 - ▣ Modeling
 - ▣ Meshing
 - ▣ Checking Ctrls
 - ▣ Numbering Ctrls
 - ▣ Archive Model
 - ▣ Coupling / Ceqn
 - ▣ Loads
 - ▣ Path Operations
- ▣ Solution
- ▣ General Postproc
- ▣ TimeHist Postpro



And then 2