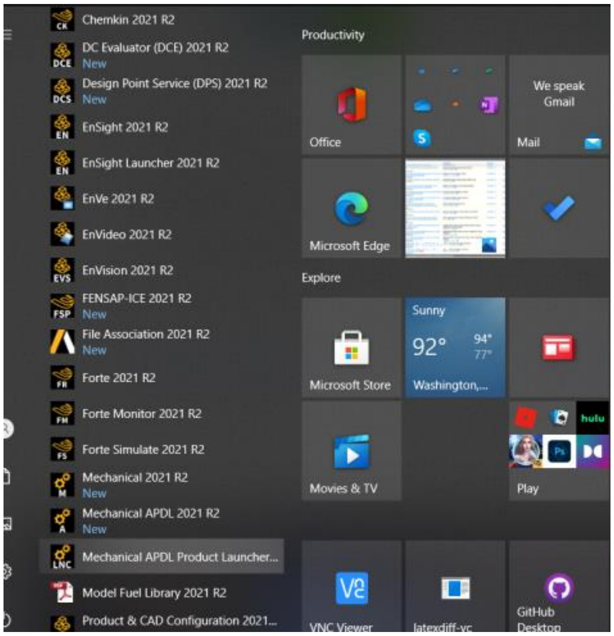
2021/08/24

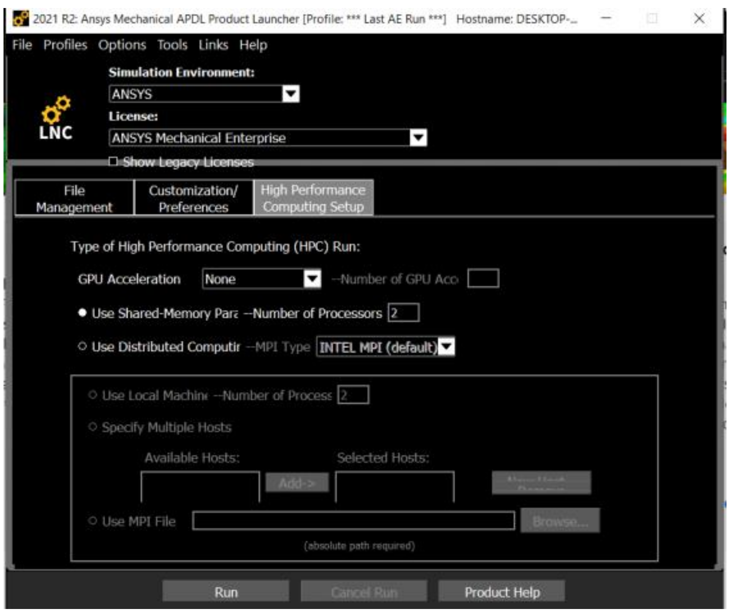
Tuesday, August 24, 2021 1:31 PM

If you need resources for learning C++, just contact me (for the second project)

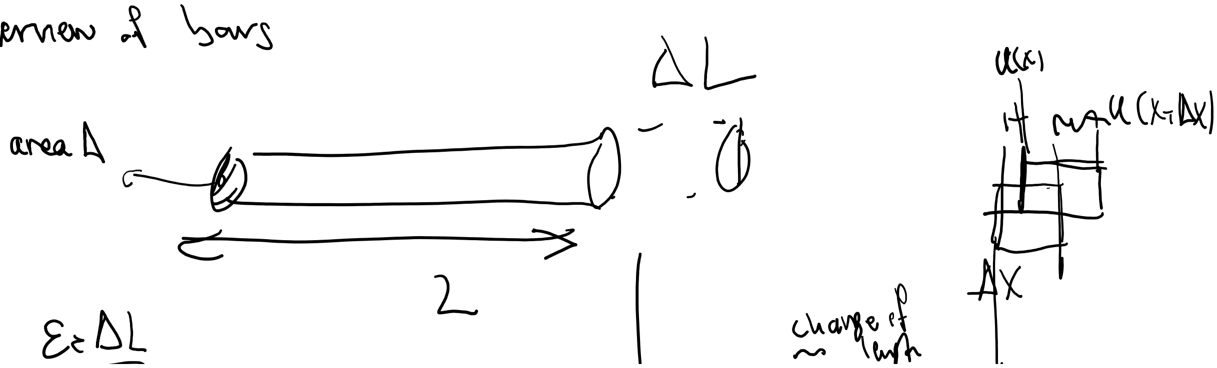
Running Ansys after the installation:




So choose shared memory option here before running it on your laptop:




Overview of bars



2

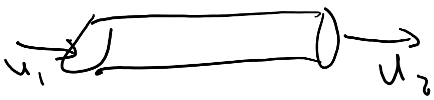
$\varepsilon = \frac{\Delta L}{L}$

$\sigma = E\varepsilon$

$E$: elastic modulus

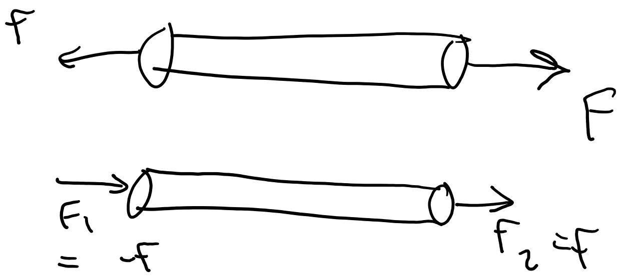
$F = \sigma A = EA\varepsilon$

$\Delta L =$




$\Delta L = u_2 - u_1$

$F = EA\left(\frac{u_2 - u_1}{L}\right) = \frac{EA}{L}(u_2 - u_1)$




$\Delta x$

change of length $\sim \Delta L$

$x$

$\varepsilon = \frac{u(x+\Delta x) - u(x)}{\Delta x} = \frac{du}{dx}$

$\Delta x$: initial length

$F = EA\frac{du}{dx}$

more general $E(x)$, $A(x)$

we'll do it later

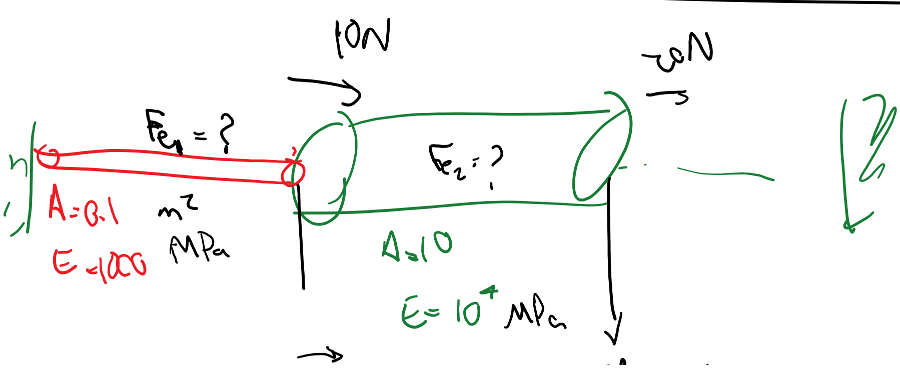
$$\begin{bmatrix} F_1 \\ F_2 \end{bmatrix} = \frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{bmatrix} u_1 \\ u_2 \end{bmatrix}$$

FEM

stiffness matrix for a bar for constant A constant E

Solid Mechanics version

$F = \frac{EA}{L}\Delta L$



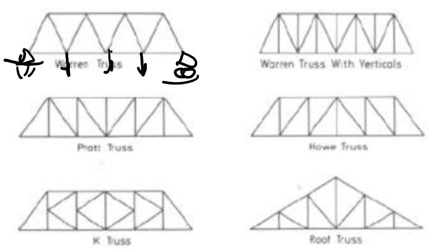
$E = 10^4$ MPa

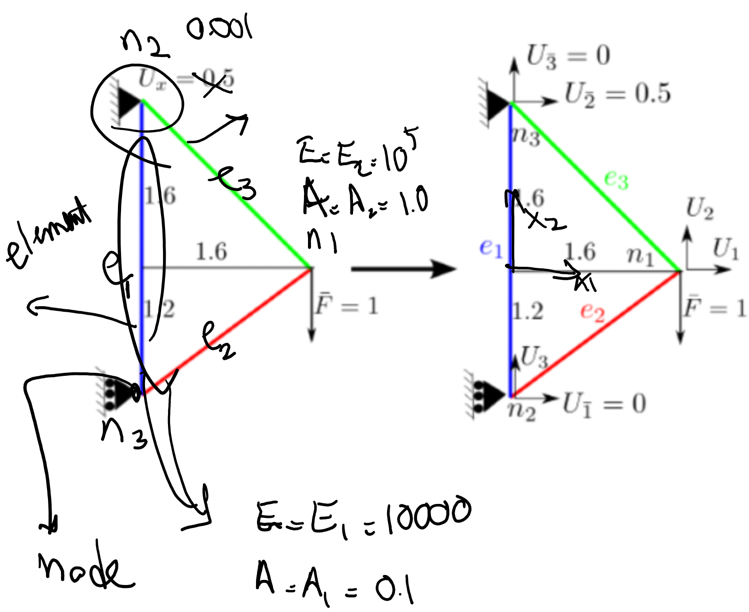
$\vec{u}_1 = ?$ $u_2 = ?$

want to solve this to obtain $u_1, u_2, f_{e_1}, f_{e_2} \ldots$

---

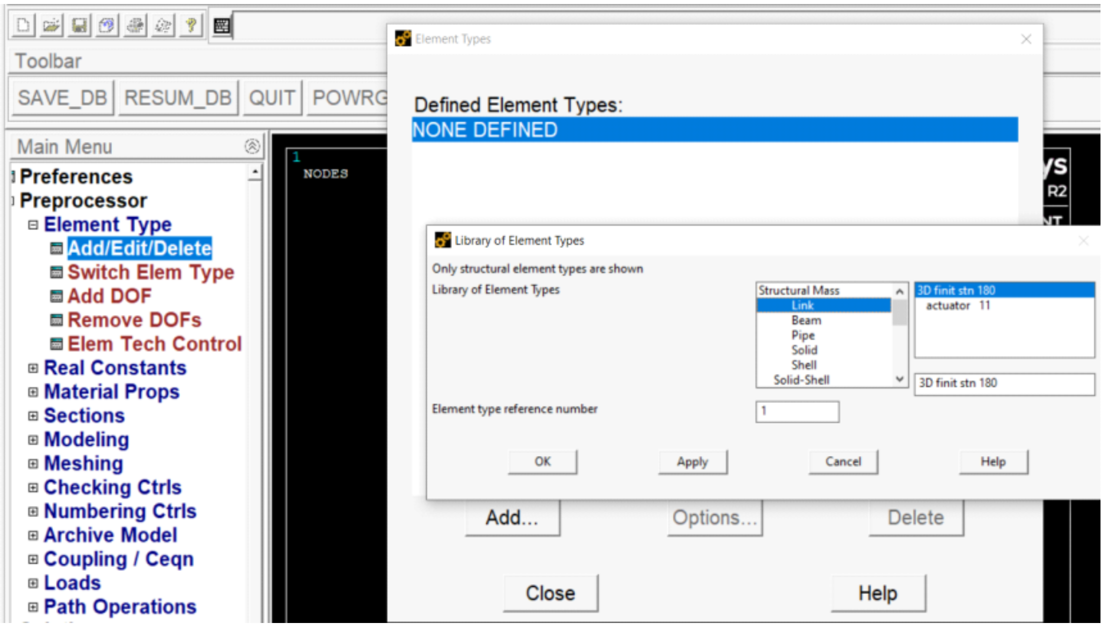
Trusses are simply rotated bars assembled in 2D & 3D



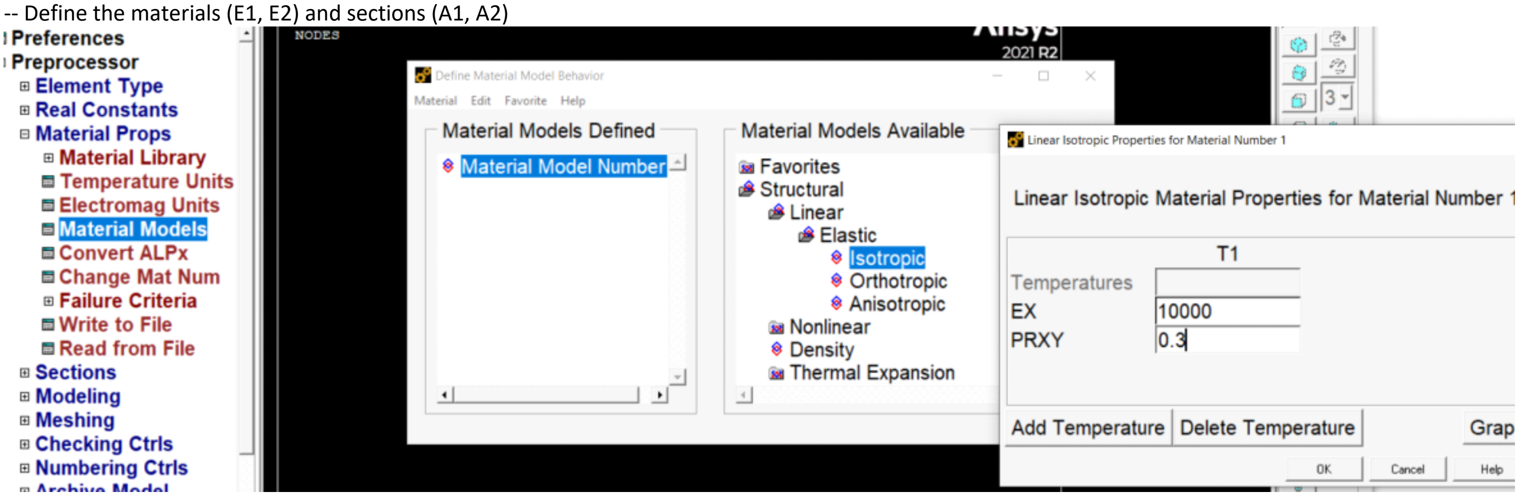




In Ansys truss elements are called links.

- Specify that we are using truss (link) elements.



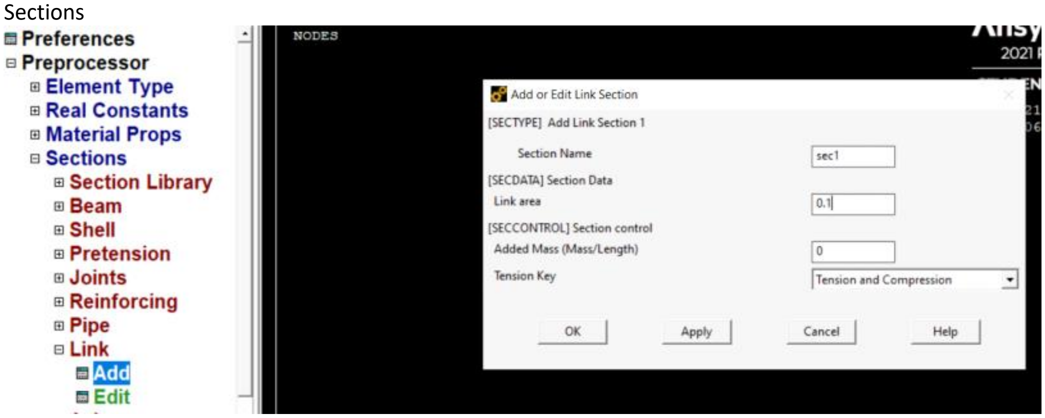

-- Define the materials (E1, E2) and sections (A1, A2)



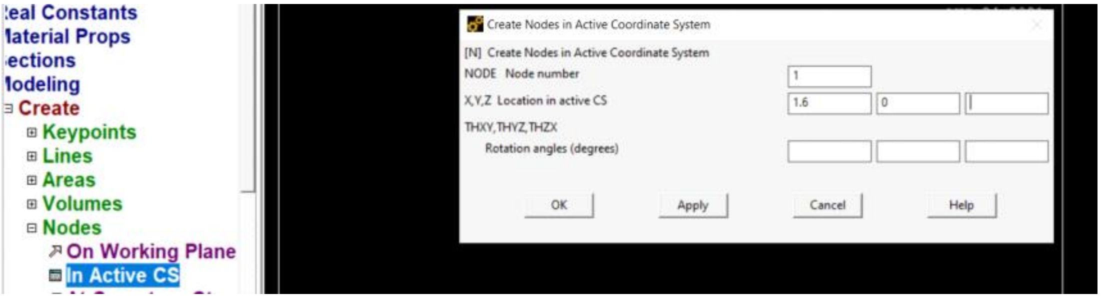

Copy and paste, and edit E = 10000 for material 2 (E2)

----

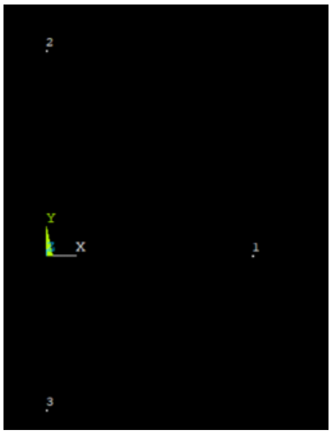
Sections



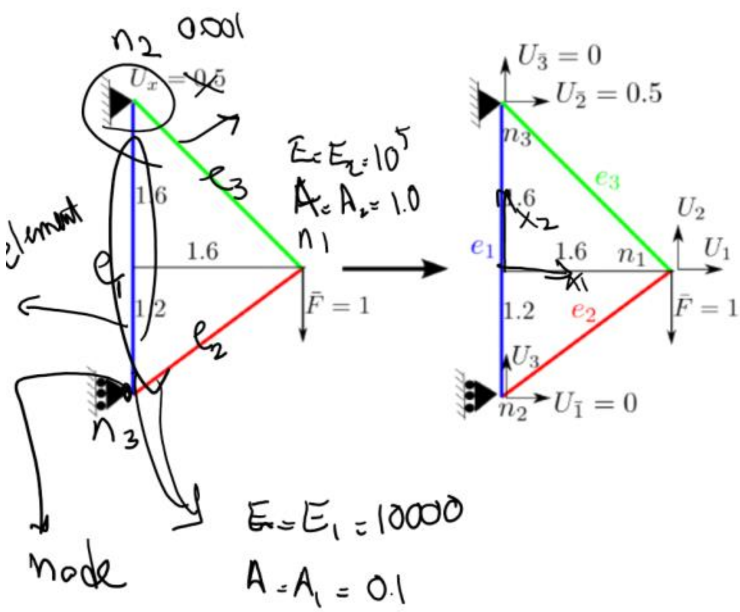

Create nodes



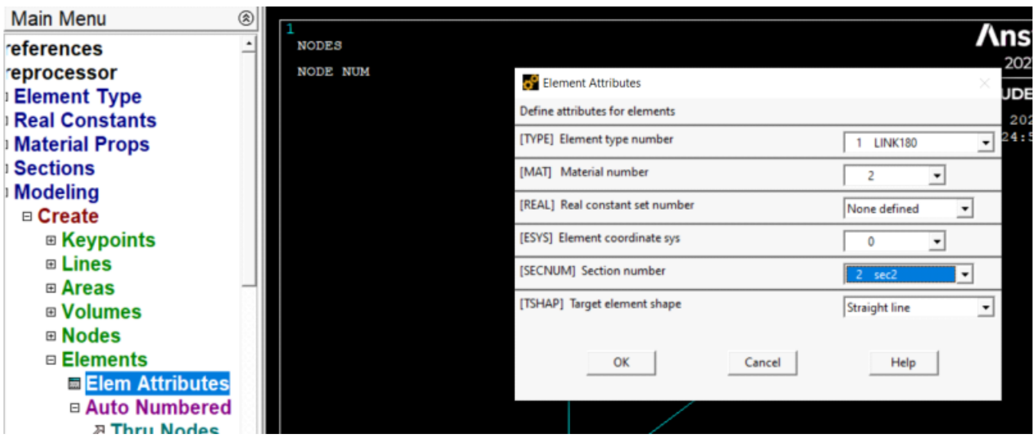

Apply and create nodes 2, 3

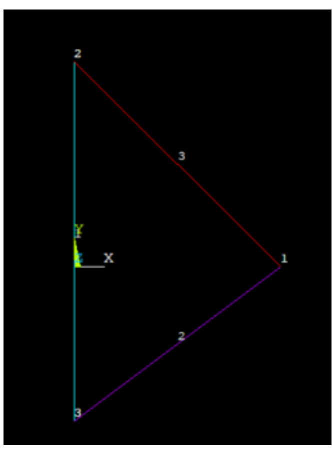



- Create elements



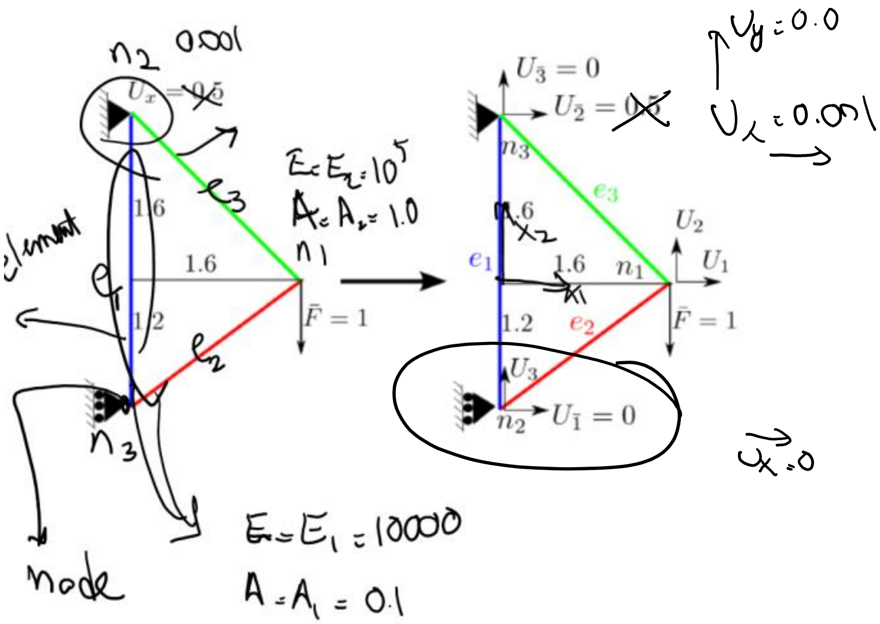

After creating element 2, we change to E2 and A2 to create element e3



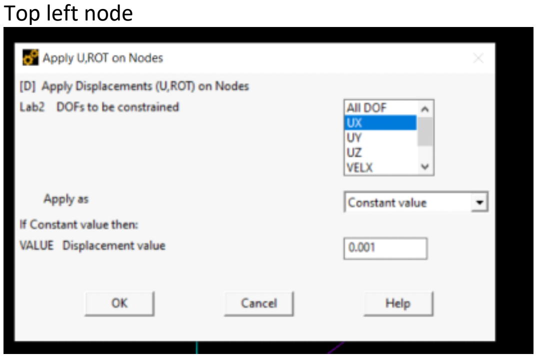
----

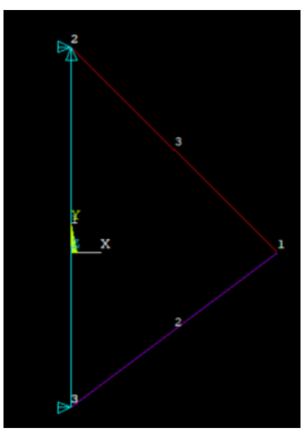
We apply supports (prescribed displacements)



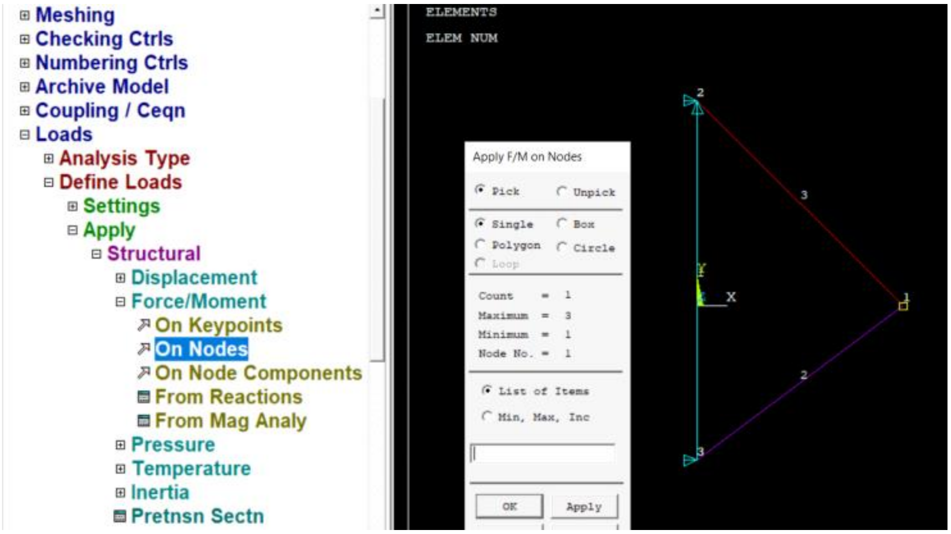

- ⊞ Numbering Ctrls
- ⊞ Archive Model
- ⊞ Coupling / Ceqn
- ⊟ Loads
  - ⊞ Analysis Type
  - ⊟ Define Loads
    - ⊞ Settings
    - ⊟ Apply
      - ⊟ Structural
        - ⊟ Displacement
          - On Lines
          - On Areas
          - On Keypoints
          - On Nodes

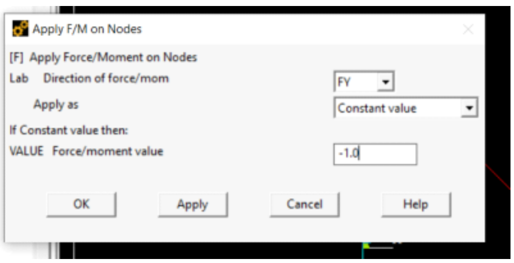
Top left node



--- apply the force on node 1



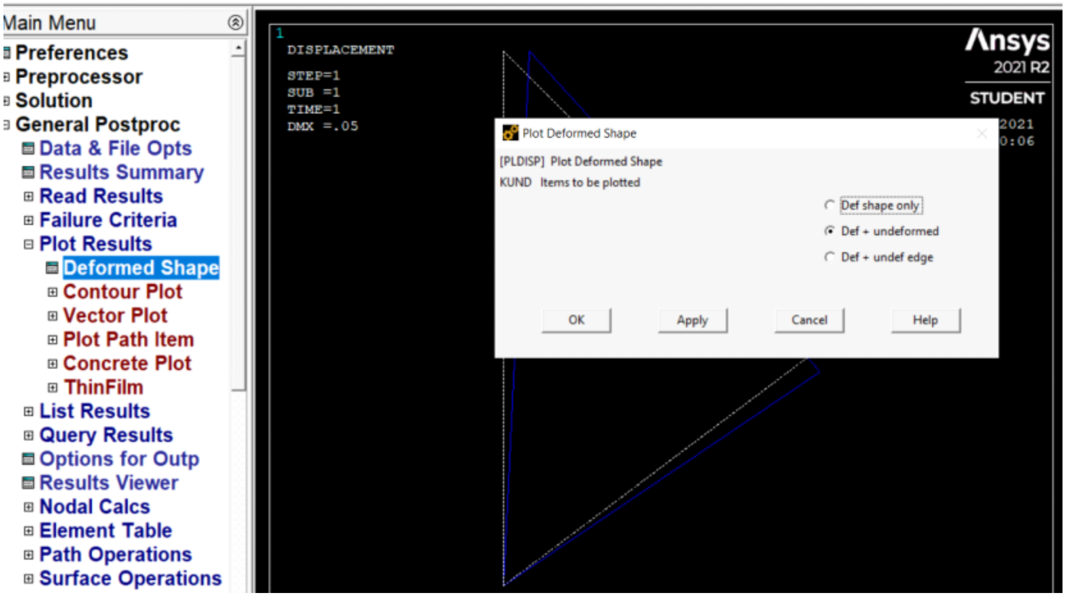




---

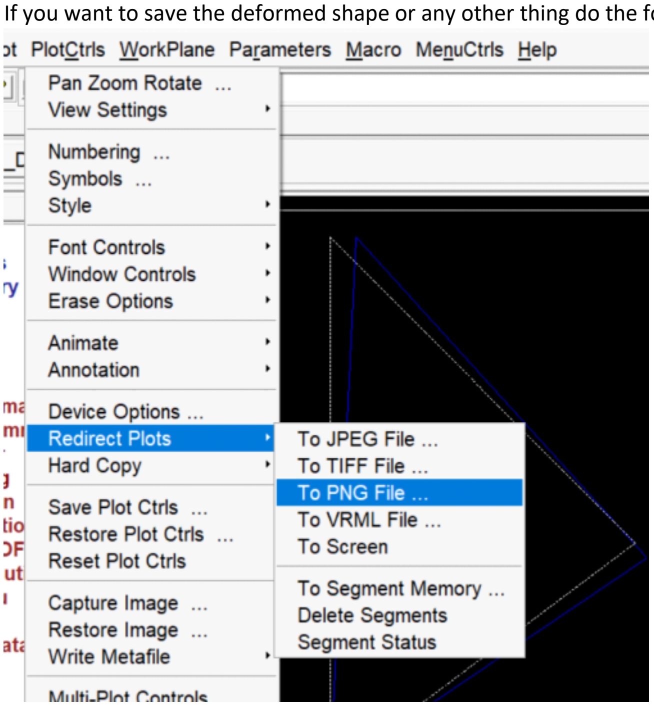

----

Postprocessing

Show the deformed shape:



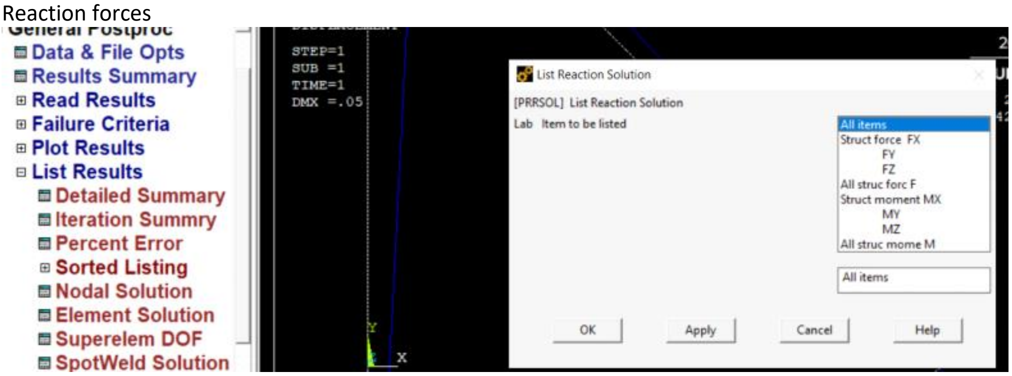

If you want to save the deformed shape or any other thing do the following




----

For 1D elements (bars, trusses, ...) we list the following:

A. Reaction forces

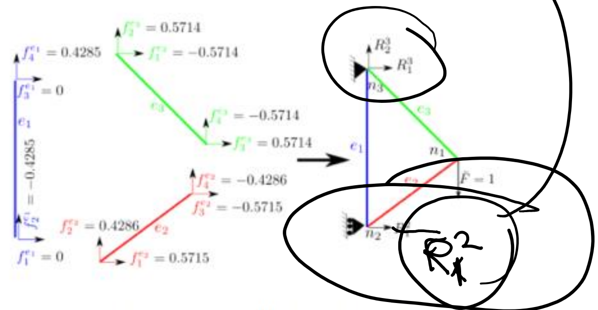



NODE FX FY FZ

2 0.0000 0.71511E-016

```
NODE      FX        FY         FZ
  2  0.0000    0.71511E-016
  3  0.94369E-016

TOTAL VALUES
VALUE   0.94369E-016 0.71511E-016  0.0000
```

When you do it at home, you get these values



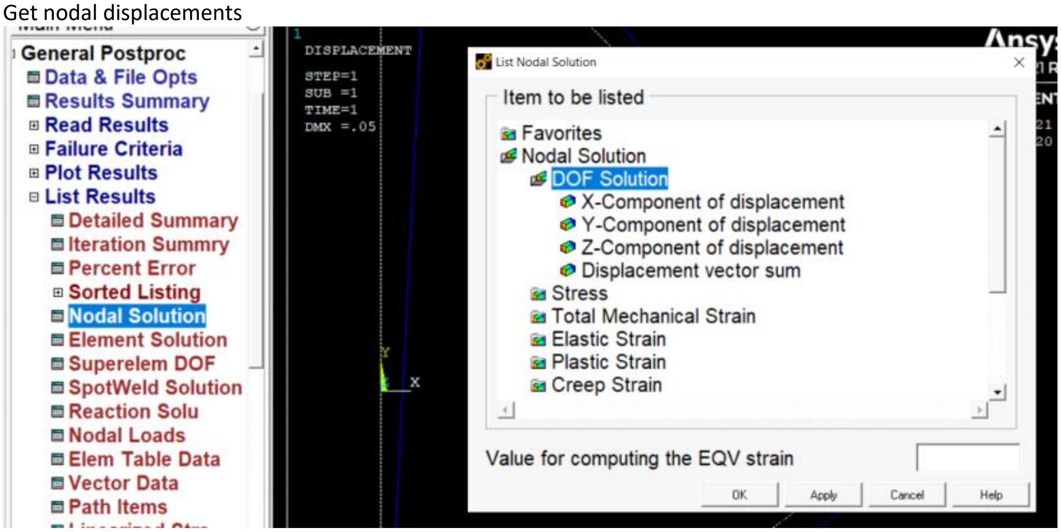
- First, we compute reaction forces by adding up forces from individual elements that contribute to reaction forces:

$$R_1^2 = f_1^{e1} + f_1^{e2} = 0 + 0.5715 = 0.5715 \qquad (397a)$$

$$R_1^3 = f_3^{e1} + f_1^{e3} = 0 + -0.5714 = -0.5714 \qquad (397b)$$

$$R_2^3 = f_4^{e1} + f_2^{e3} = 0.4285 + 0.5714 = 0.9999 \qquad (397c)$$

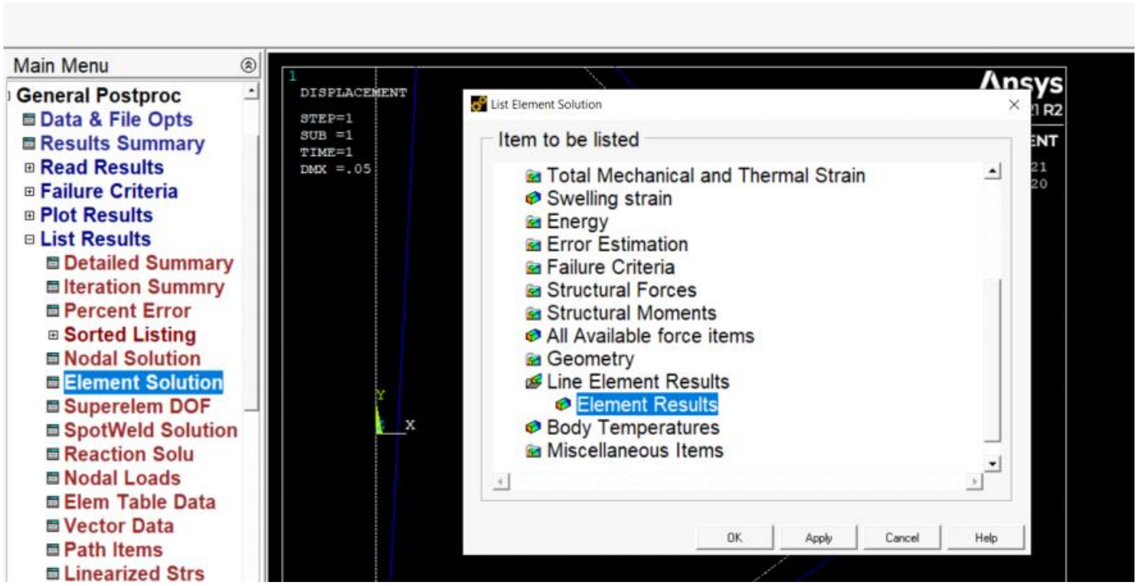
Get nodal displacements




```
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  0.21429E-001-0.28571E-001  0.0000      0.35714E-001
     2  0.50000E-001  0.0000       0.0000     0.50000E-001
     3   0.0000      -0.20023E-015  0.0000     0.20023E-015
```

MAXIMUM ABSOLUTE VALUES
NODE        2          1          0          2
VALUE   0.50000E-001-0.28571E-001  0.0000      0.50000E-001

To list element forces use




PRINT ELEM ELEMENT SOLUTION PER ELEMENT

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

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

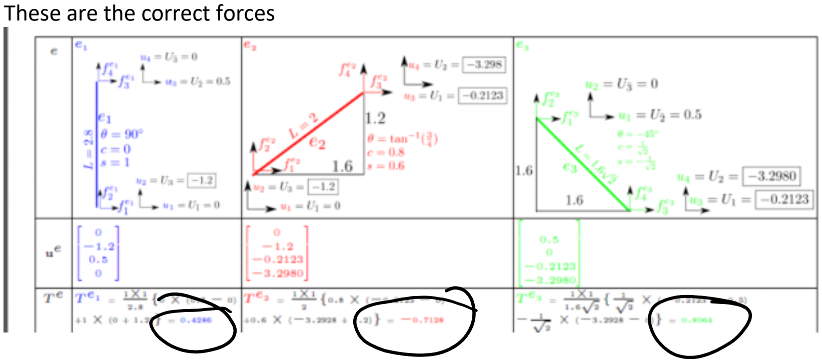
EL=     1 NODES=      2       3 MAT=     1  XC,YC,ZC=  0.000     0.2000      0.000     AREA= 0.10000E-01            LINK180
FORCE= 0.71511E-16 STRESS= 0.71511E-14 EPEL= 0.71511E-16
TEMP=    0.00    0.00  EPTH=  0.0000

Incorrect forces

EL=     2 NODES=      3       1 MAT=     1  XC,YC,ZC= 0.8000    -0.6000      0.000     AREA= 0.10000E-01            LINK180
FORCE=-0.11796E-15 STRESS=-0.11796E-13 EPEL=-0.11796E-15
TEMP=    0.00    0.00  EPTH=  0.0000

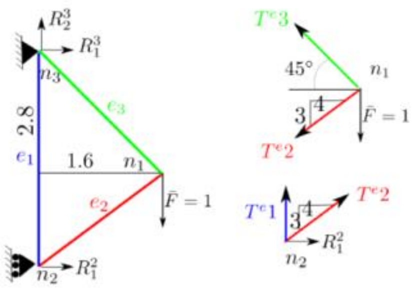
EL=     3 NODES=      2       1 MAT=     2  XC,YC,ZC= 0.8000     0.8000      0.000     AREA= 0.10000            LIN K180
FORCE=  0.0000     STRESS=  0.0000     EPEL=  0.0000
TEMP=    0.00    0.00  EPTH=  0.0000

These are the correct forces



The exact solution of this statistically-determinant structure is obtained here (forces do not depend on E and A)

## Truss Example: Direct solution method




- Since this is a statically determinate structure, we can easily solve the forces and verify our FEM forces.

$$\Sigma F_2 = 0 \;\Rightarrow\; R_2^3 - \bar{F} = 0 \;\Rightarrow\; R_2^3 = 1 \tag{399a}$$

$$\Sigma M_{n_3} = 0 \;\Rightarrow\; 2.8R_1^2 - 1.6\bar{F} = 0 \;\Rightarrow\; R_1^2 = \frac{4}{7} = 0.5714 \tag{399b}$$

$$\Sigma F_1 = 0 \;\Rightarrow\; R_1^2 + R_1^3 = 0 \;\Rightarrow\; R_1^3 = -\frac{4}{7} = -0.5714 \tag{399c}$$

$$\Sigma F_1 = 0(@n_2) \;\Rightarrow\; R_1^2 + \frac{4}{5}T^e2 = 0 \;\Rightarrow\; T^e2 = -\frac{5}{7} = -0.7143 \tag{399d}$$

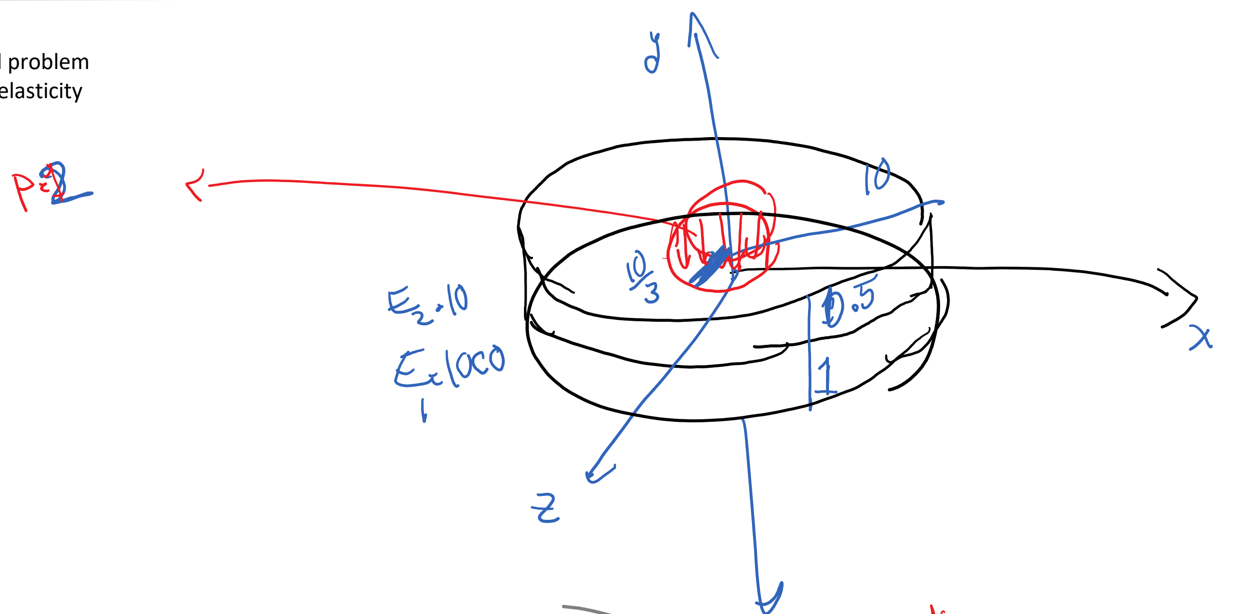
$$\Sigma F_2 = 0(@n_2) \;\Rightarrow\; T^e1 + \frac{3}{5}T^e2 = 0 \;\Rightarrow\; T^e1 = \frac{3}{7} = 0.4286 \tag{399e}$$

$$\Sigma F_1 = 0(@n_1) \;\Rightarrow\; -\frac{4}{5}T^e2 - \frac{1}{\sqrt{2}}T^e3 = 0 \;\Rightarrow\; T^e3 = \frac{4}{7}\sqrt{2} = 0.8081 \tag{399f}$$

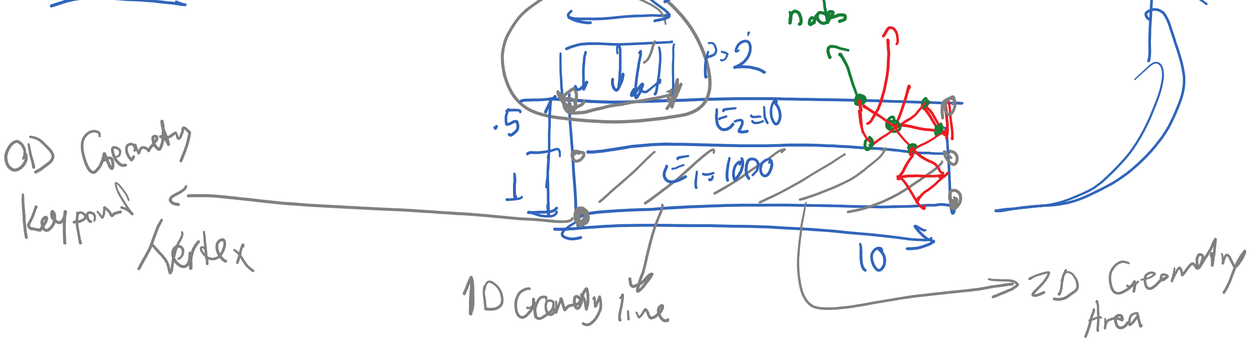
327 / 456

---

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