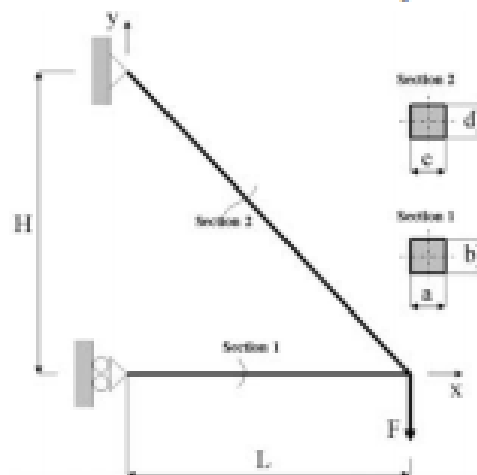


Course in Ansys

Example 0150

Example – Truss 2D



Objective:

Compute the maximum deflection

Tasks:

Display the deflection figure?

Topics:

Topics: Start of analysis, Element type, Real constants, Material, modeling, element size for beam models, saving/restoring

$E = 210000\text{N/mm}^2$

$\nu = 0.3$

$L = 100\text{mm}$

$H = 120\text{mm}$

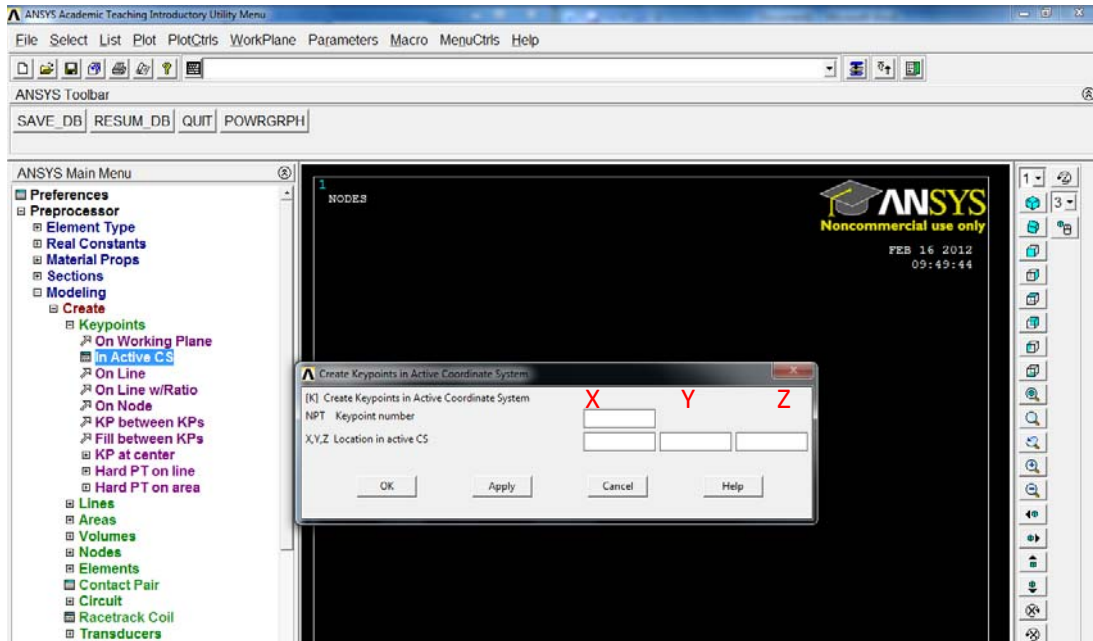
$a = b = 20\text{mm}$

$c = d = 10\text{mm}$

$F = 100\text{N}$

Step by Step guide for Ansys 13

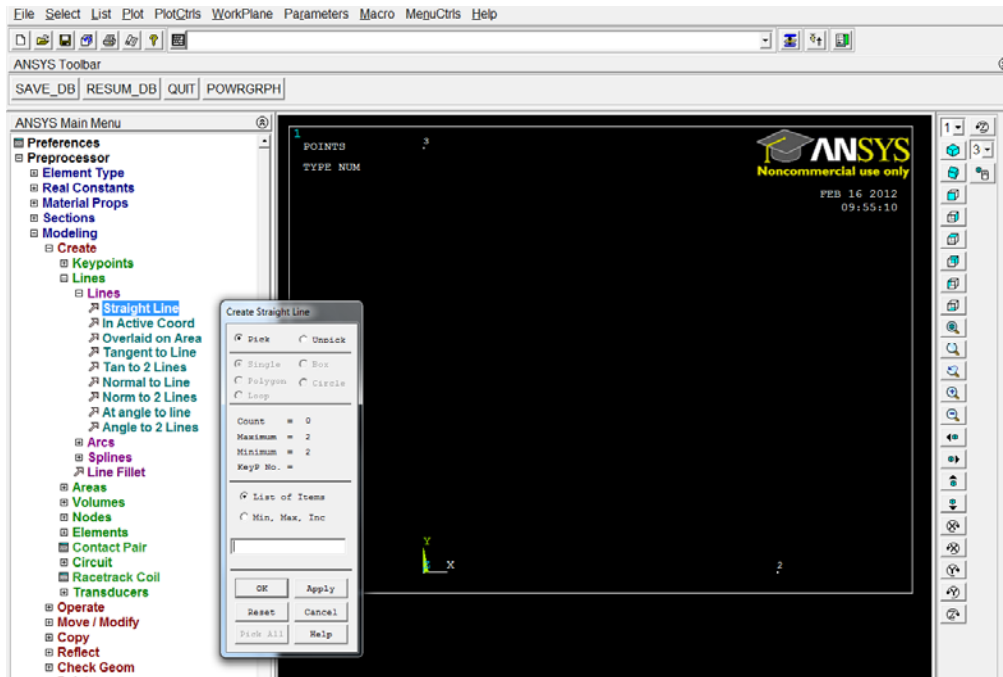
1. Preprocessor > Modeling > Create > Keypoints In Active CS



3 Points (click apply for 1 and 2 and ok for 3)

X	Y	Z
0	0	0
100	0	0
0	120	0

2. Preprocessor > Modeling > Create > Lines > Lines>Straight Line

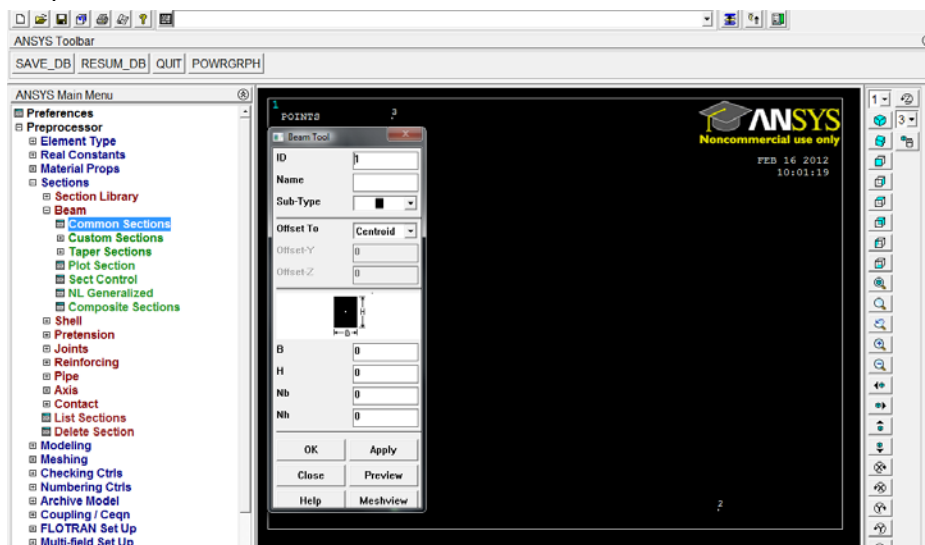


Click on point 1, ok, point 2, ok

Create lines between points 1-2; points 2-3;

Alternatively type the point numbers into the popup box and press enter to create lines

3. Preprocessor>Sections>Beam>Common Sections



Section 1: B=20 H=20

Section 2: B=10 H=10

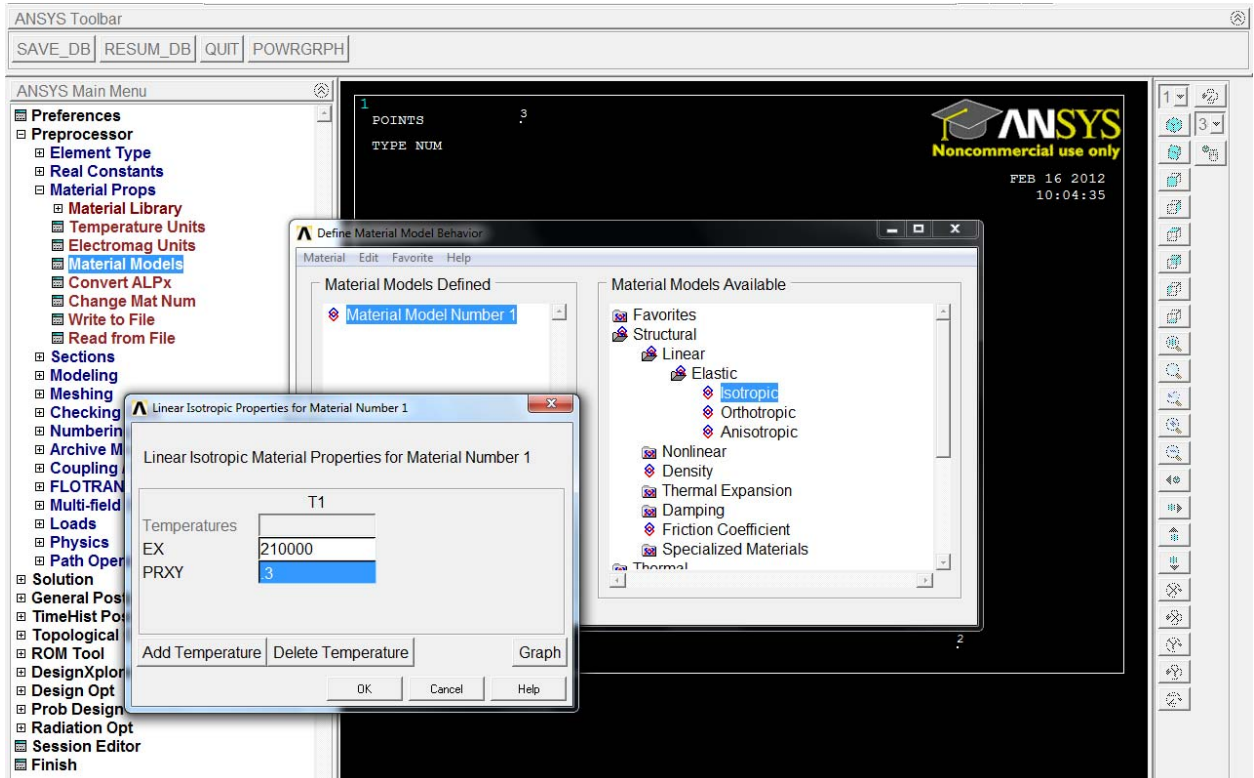
4. Preprocessor>Materials>Material Props>Material Model

Inside the resulting window

Structural>Linear>Elastic>Isotropic

EX = 210000

PRXY = .3



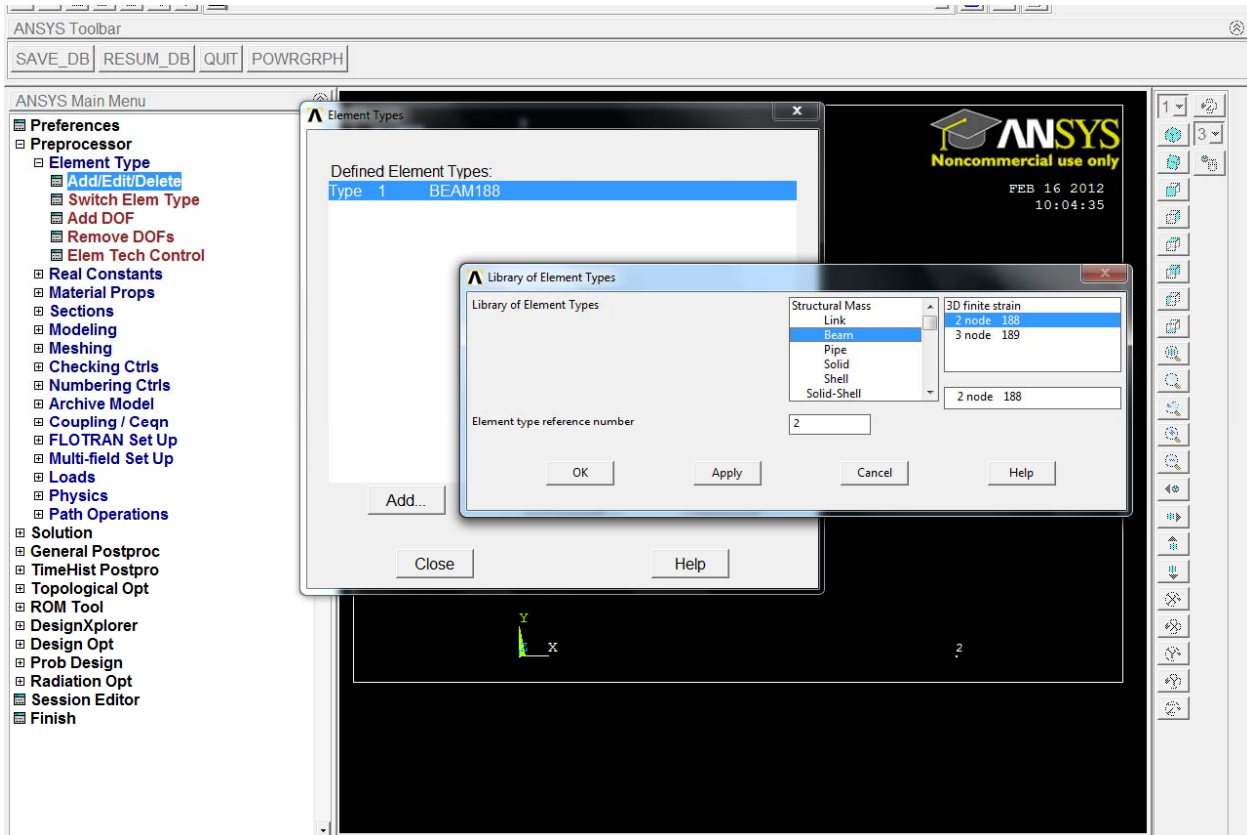
5. Close popup window

6. Preprocessor>Element Type>Add/Edit/Delete

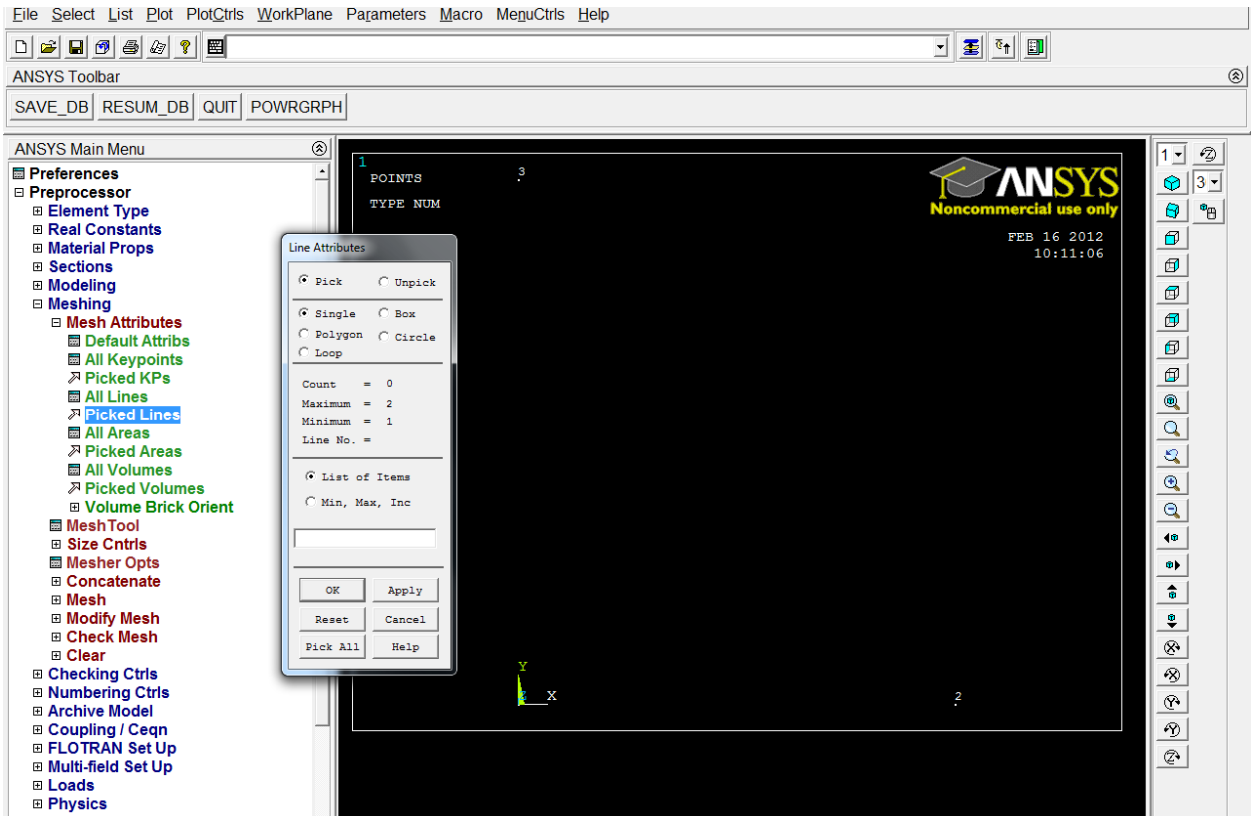
Click Add

Select Beam> 2 node 188

Click OK, Click Close

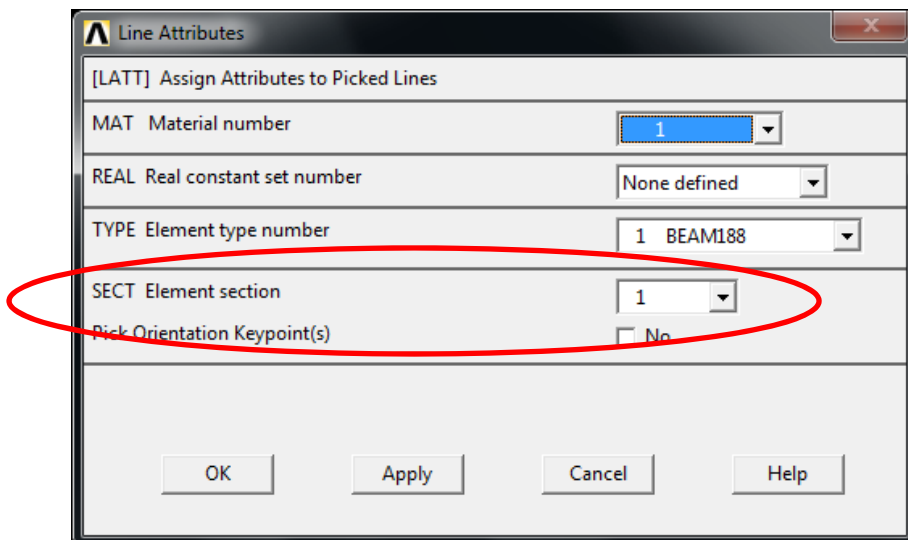


7. Preprocessor>Meshing>Mesh Attributes>Picked Lines



Select the line between points 1 and 2 (horizontal line) > Click OK

Under Element Type select 1

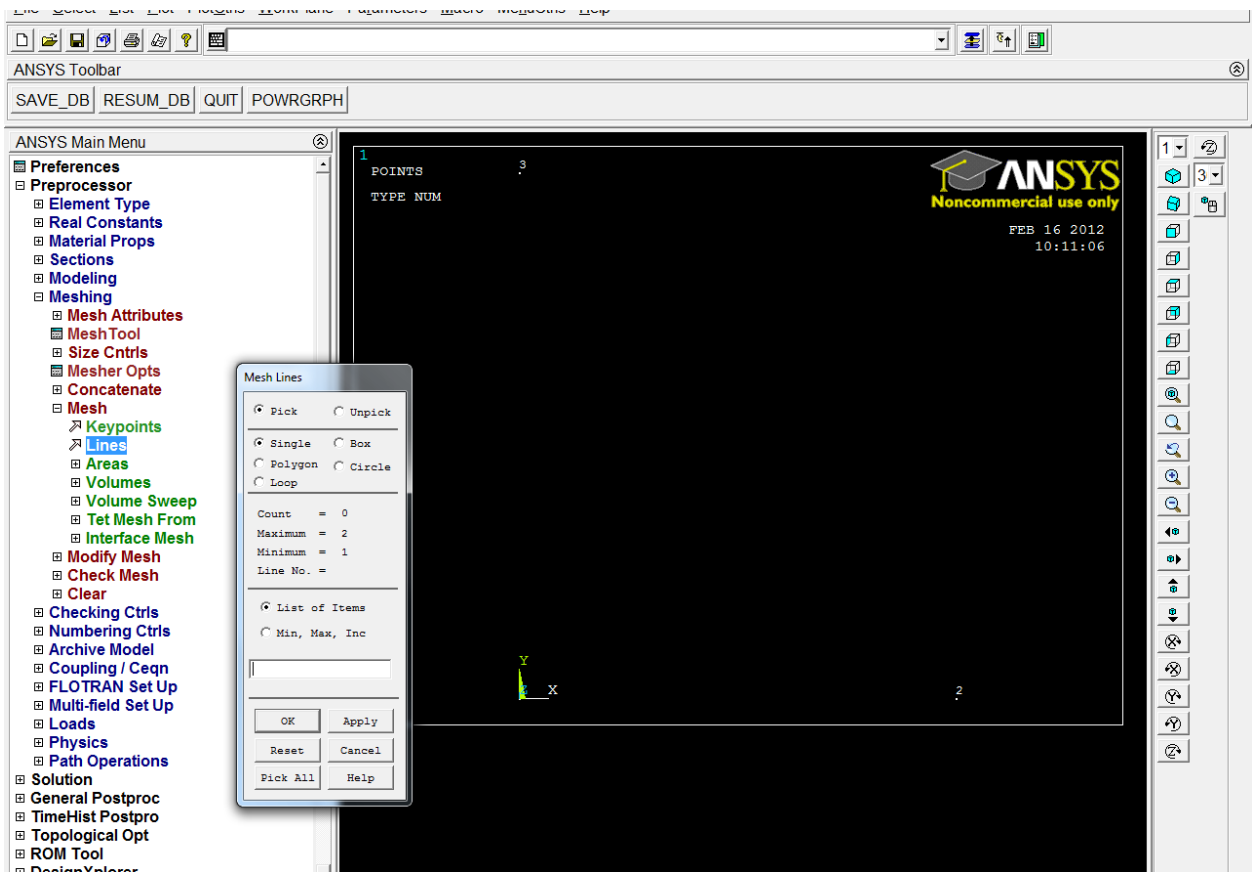


Select the Line between points 2 and 3 (diagonal line) > click OK

Under Element Type change to 2

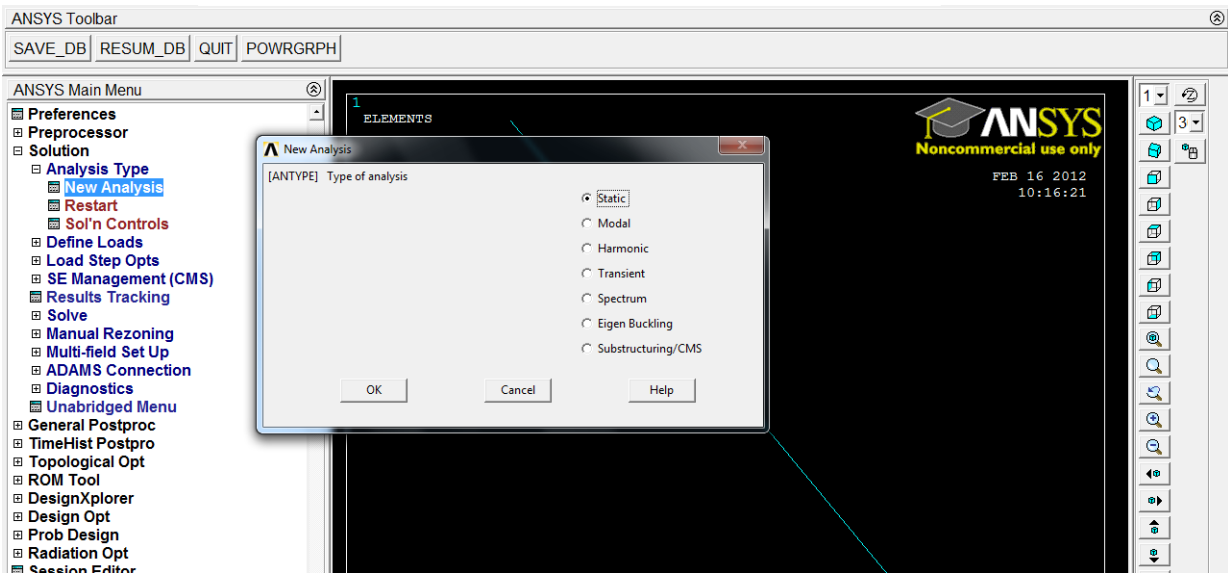
8. Preprocessor>Meshing>Mesh>lines

a. Pick All



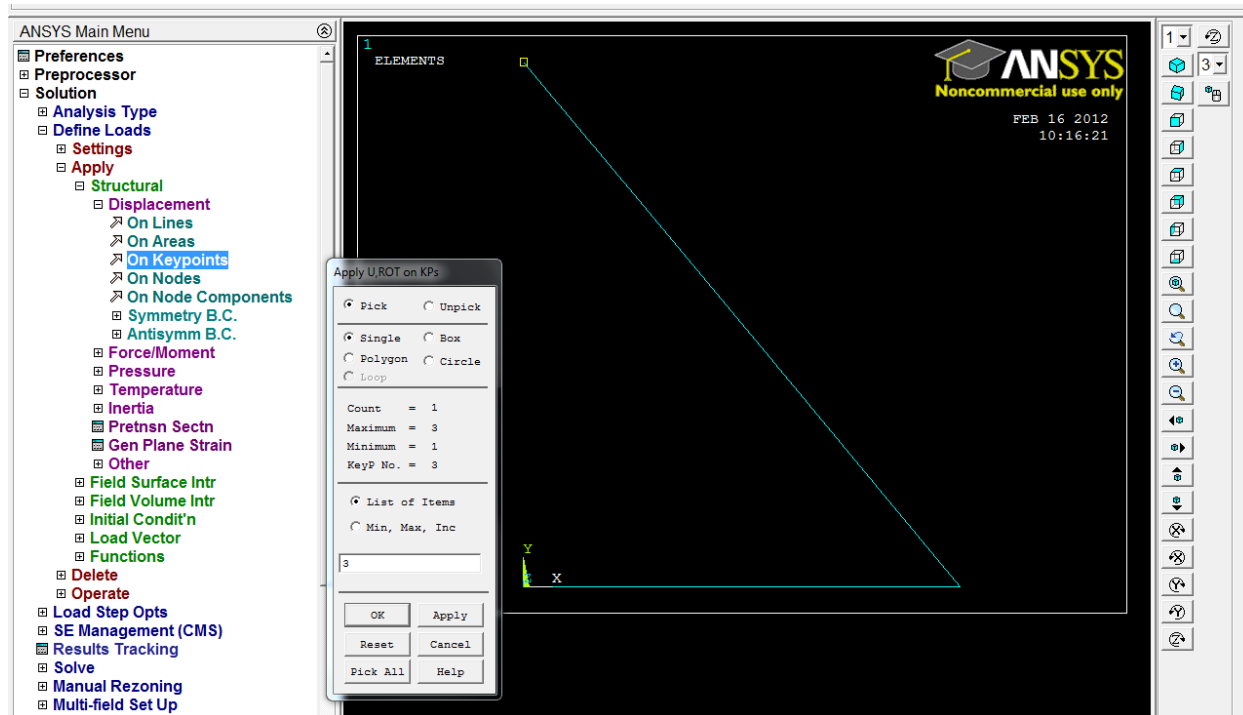
9. Solution > Analysis Type> New Analysis

a. Select static > click OK

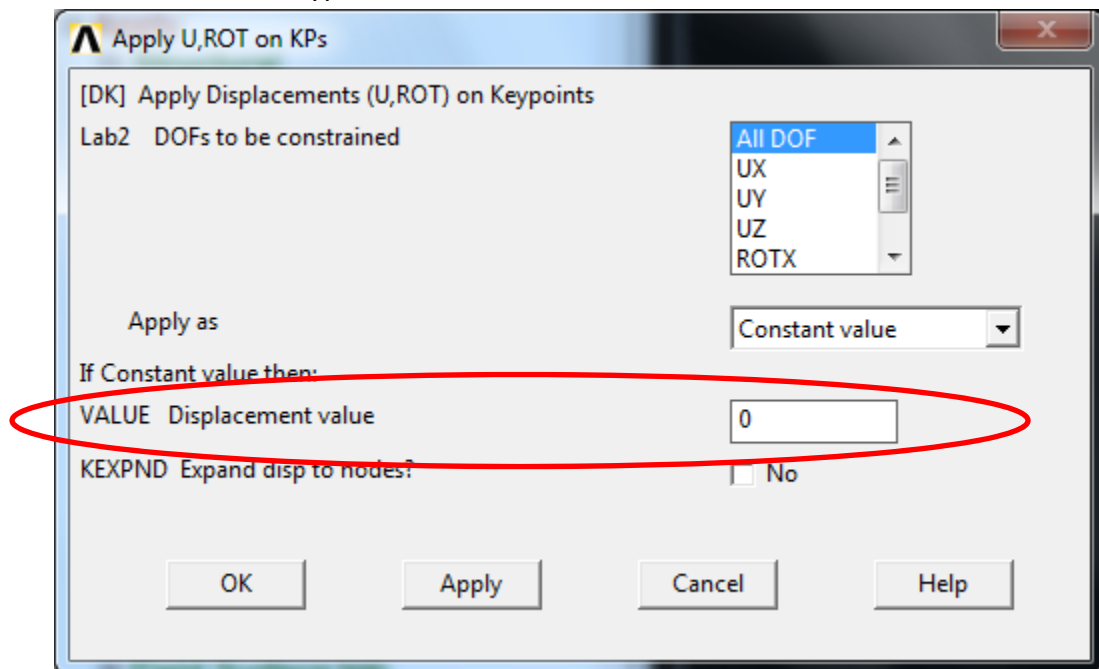


10. Solution > Define Loads > Apply > Structural > Displacement > On Keypoints

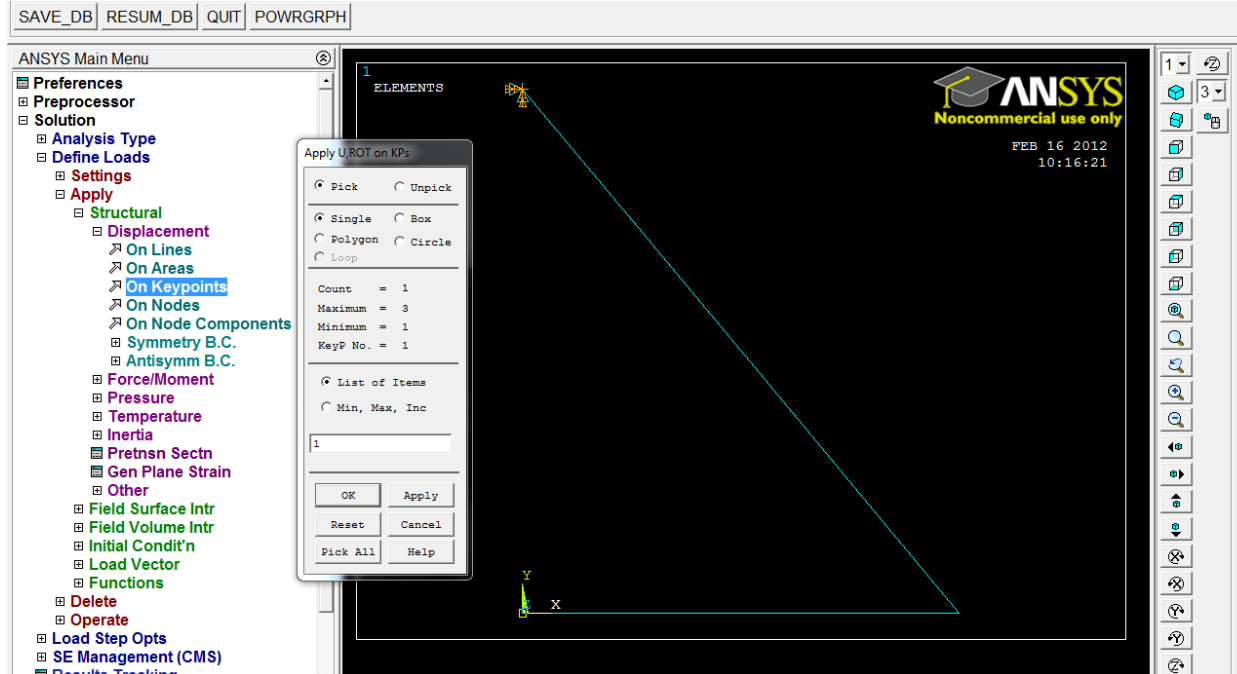
- a. Select keypoint 3 (or type 3 into the box) > click ok



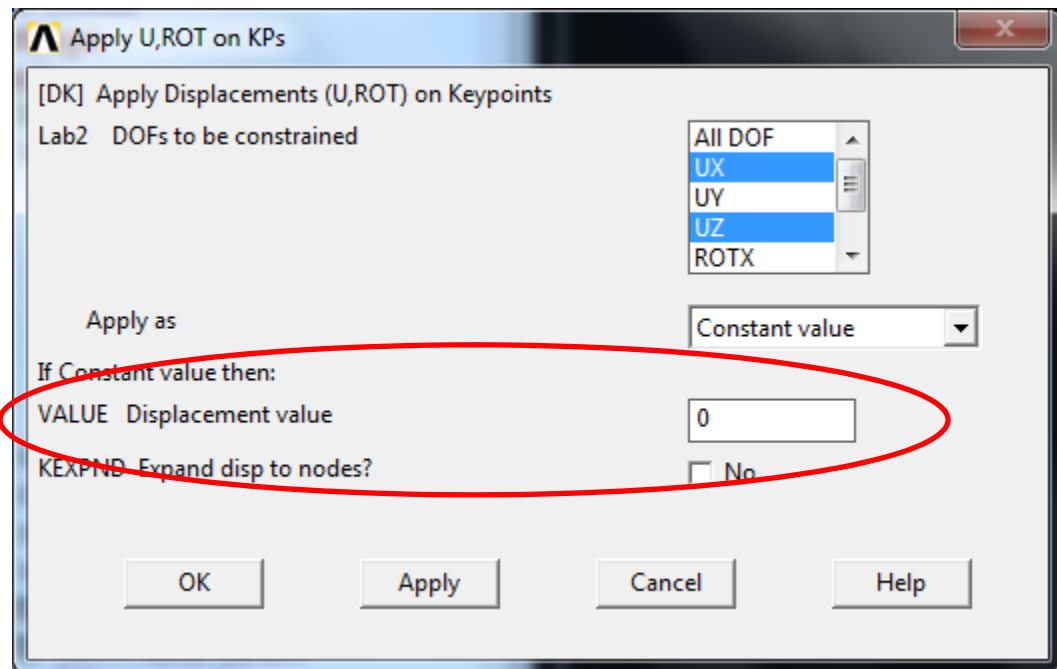
- b. Select All DOF and type in 0 for Value > click OK



11. Solution > Define Loads > Apply> Structural> Displacement> On Keypoints
a. Select Keypoint 1 > click OK

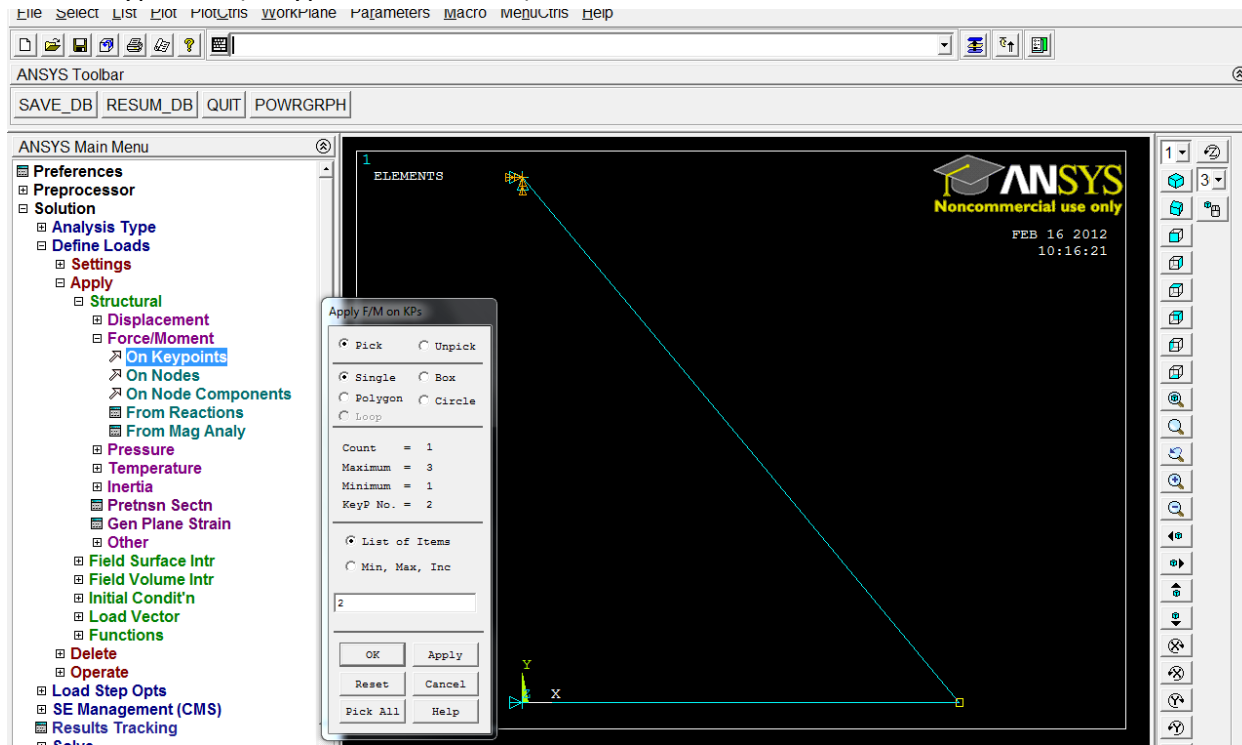


- b. In the next box select UX and UZ and for Value select 0 (this creates a roller fixture which allows movement in the Y direction ONLY) >Click OK

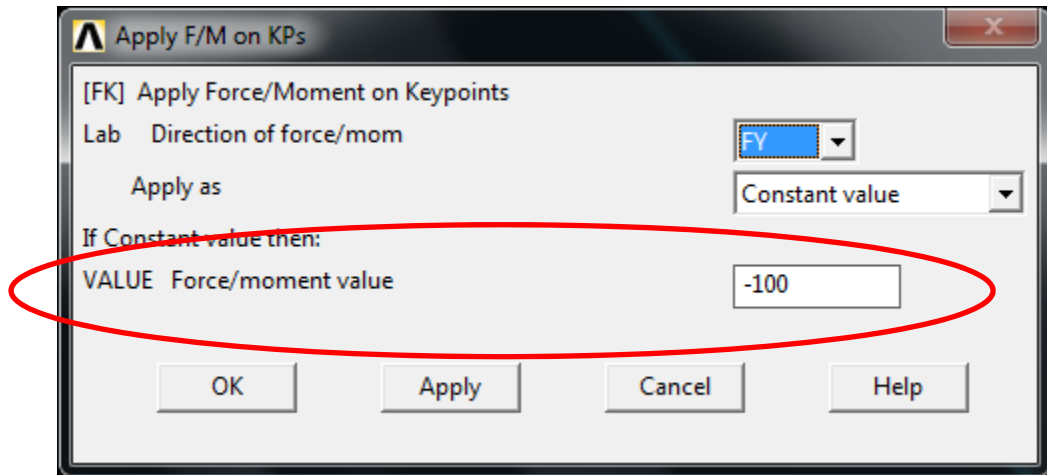


12. Solution > Define Loads > Apply> Structural> Force/Moment>On Keypoints

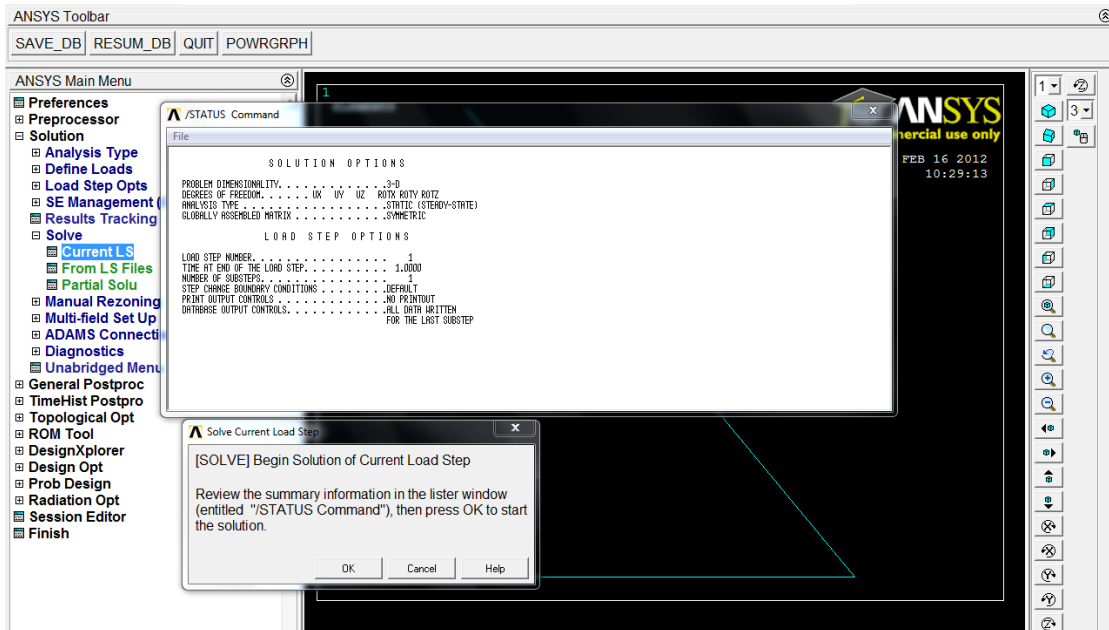
a. Select keypoint 2 (or type the number 2) > OK



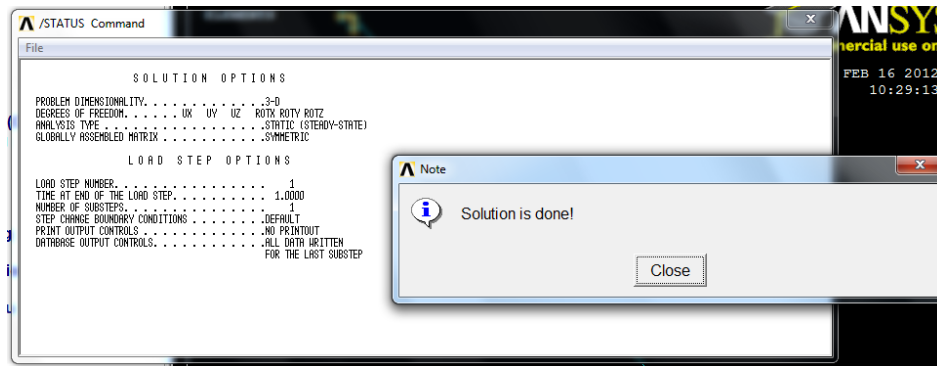
b. Select FY for direction, Constant Value, and enter -100 for the VALUE box (the force is directed down, hence the negative)> click OK



13. Solution > Solve >Current LS

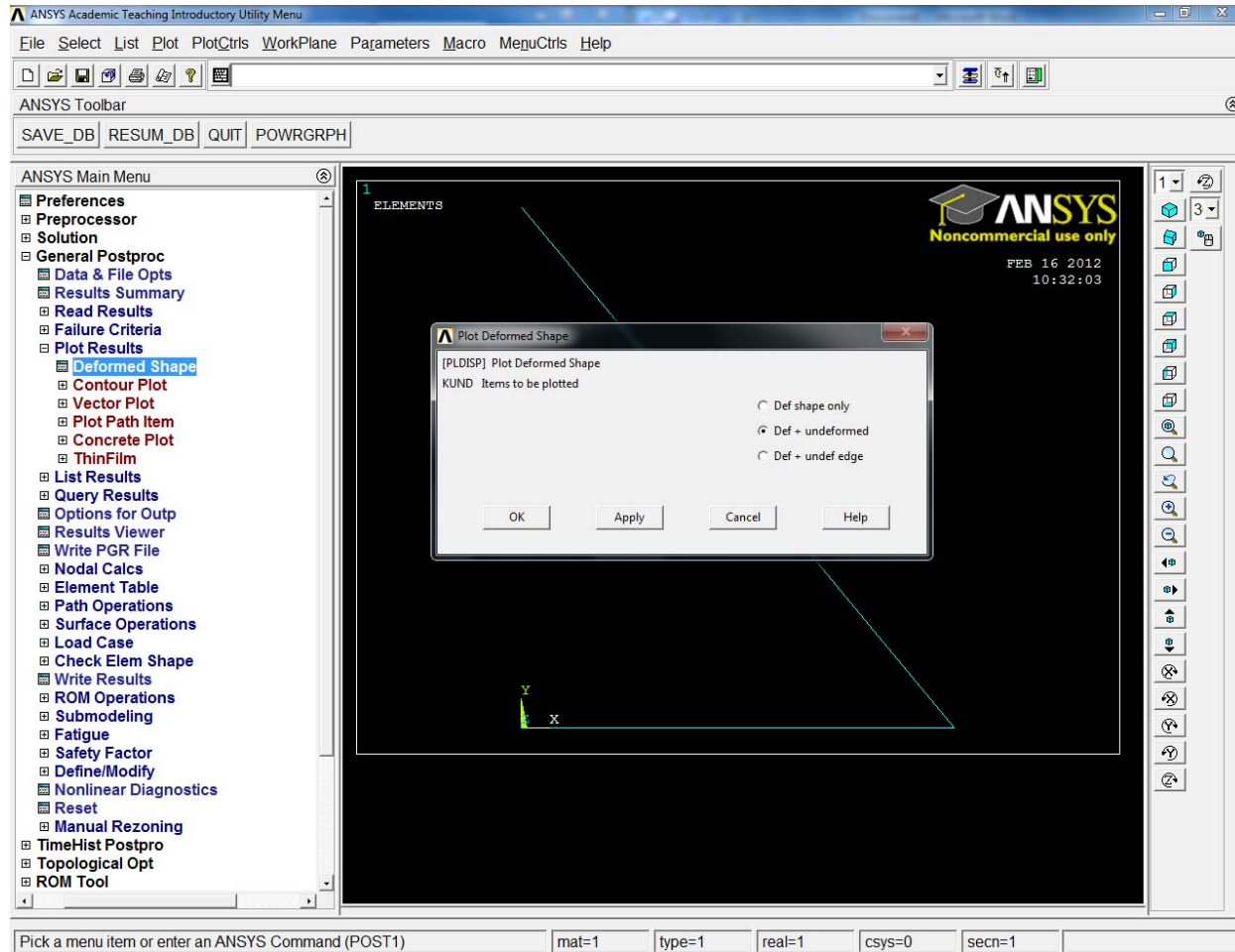


a. Click OK



14. General Postproc>Plot Results>Deformed Shape

- Select Def + undeformed> OK



15. General Postproc>Plot Results> Contour plot>Nodal Solu

- In Popup window Nodal Solution>Stress>con Mises stress
- Click OK

