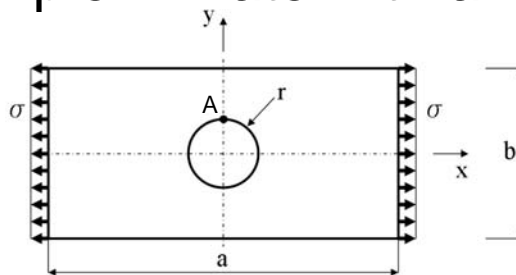


Course in ANSYS

Example0240

Example – Plate with a hole



Objective:

Determine the maximum stress in the x-direction for point A and display the deformation figure

Tasks:

How should this be modelled?

Compare results with results obtained from norm calculations?

Topics:

Element type, Real constants, modeling, plot results, output graphics

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

$$\nu = 0.3$$

$$a = 200\text{mm}$$

$$b = 100\text{mm}$$

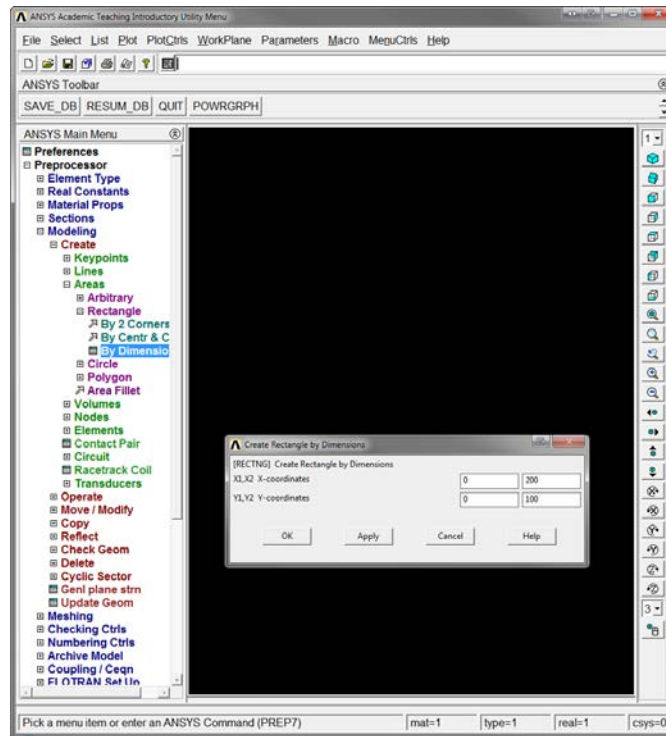
$$t = 10\text{mm}$$

$$r = 10\text{mm}$$

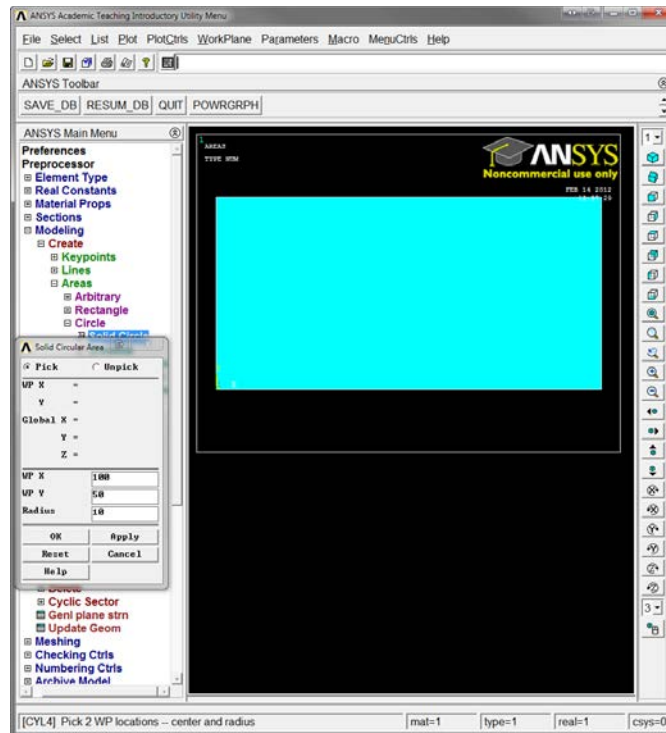
$$\sigma = 100\text{N/mm}^2$$

STEP-BY-BY PICTORAL GUIDE: for use with ANSYS 13

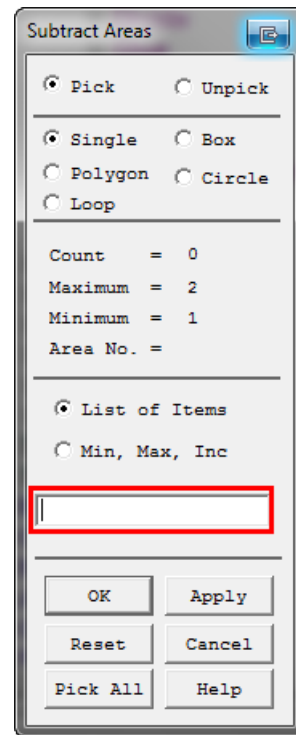
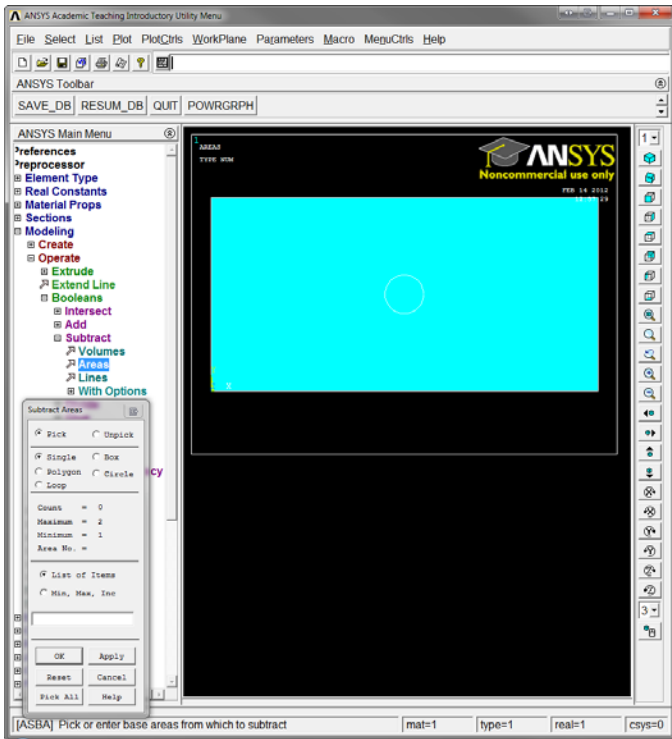
1. Preprocessor → Modeling → Create → Areas → Rectangle → By Dimensions



2. Preprocessor → Modeling → Create → Areas → Circle → Solid Circle

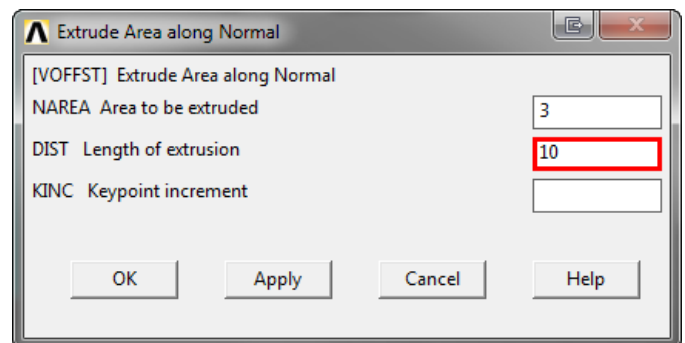
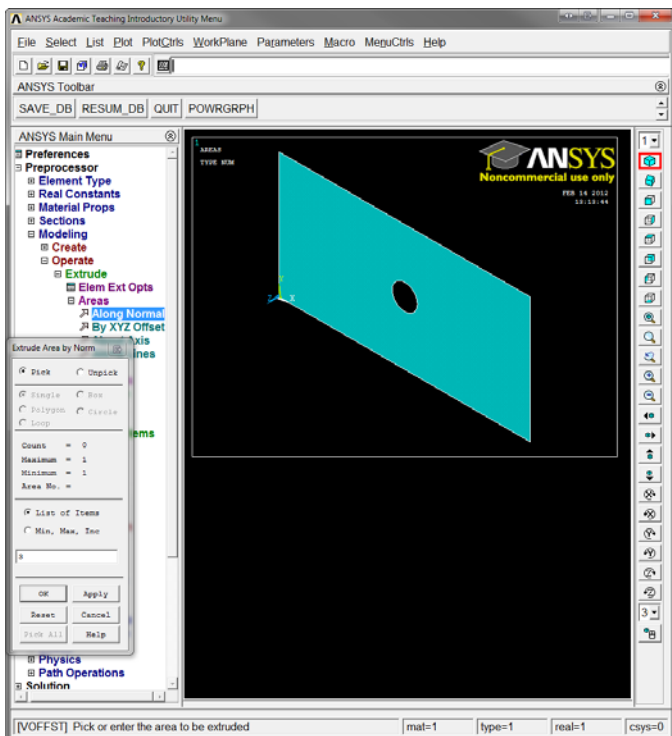


3. Preprocessor → Modeling → Operate → Booleans → Subtract → Areas



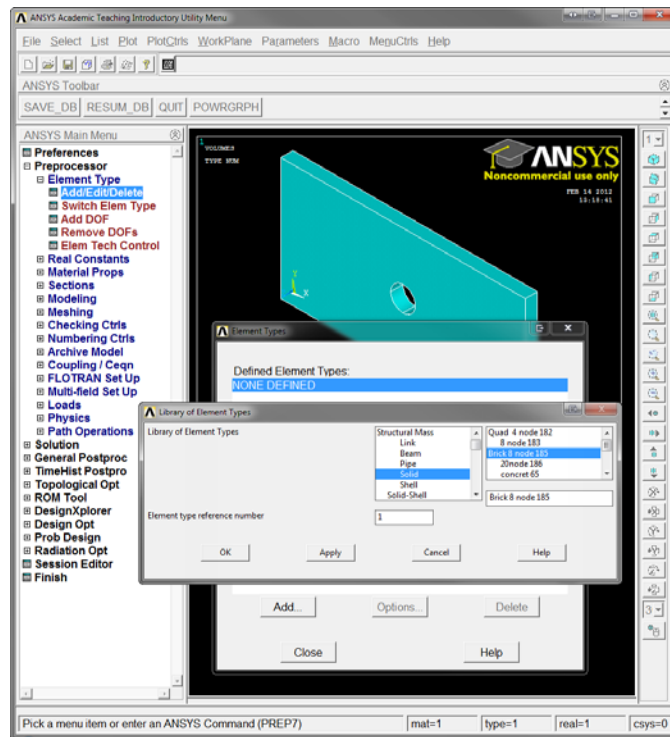
- Type "1" in the red box, as seen above, to select the first area (the rectangle) for the material that will be left behind and select "OK"
- Type "2" in the red box, as seen above, to select the second area (the circle) for the material that is to be removed and select "OK"

4. Preprocessor → Modeling → Operate → Extrude → Areas → Along Normal Axis



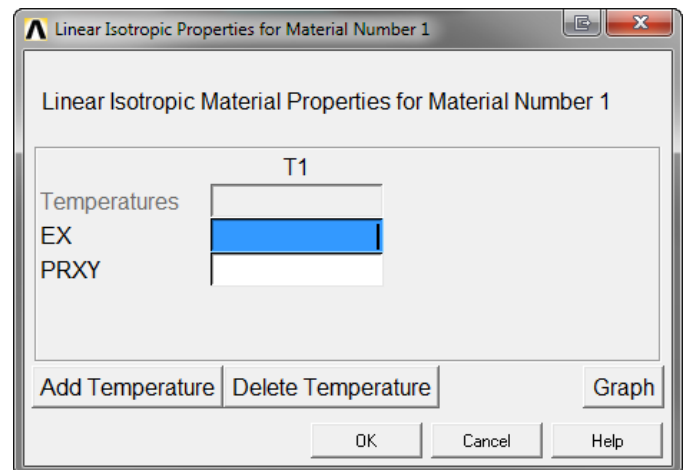
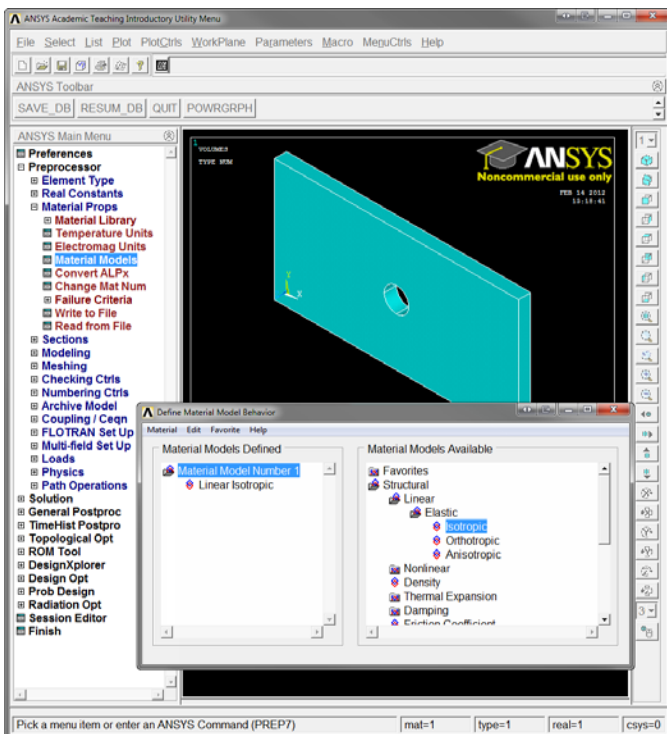
- Type "3" in the box to select the third area (the result of the subtraction above) and select "OK"
- Type "10" in the red box, as seen above, to set the length (or depth) of extrusion and select "OK"

5. Preprocessor → Element Type → Add/Edit/Delete → Add



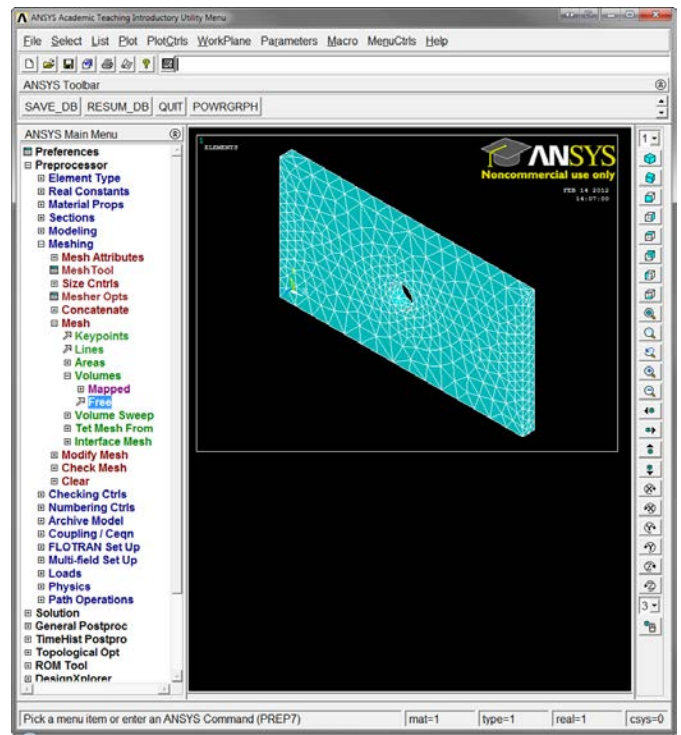
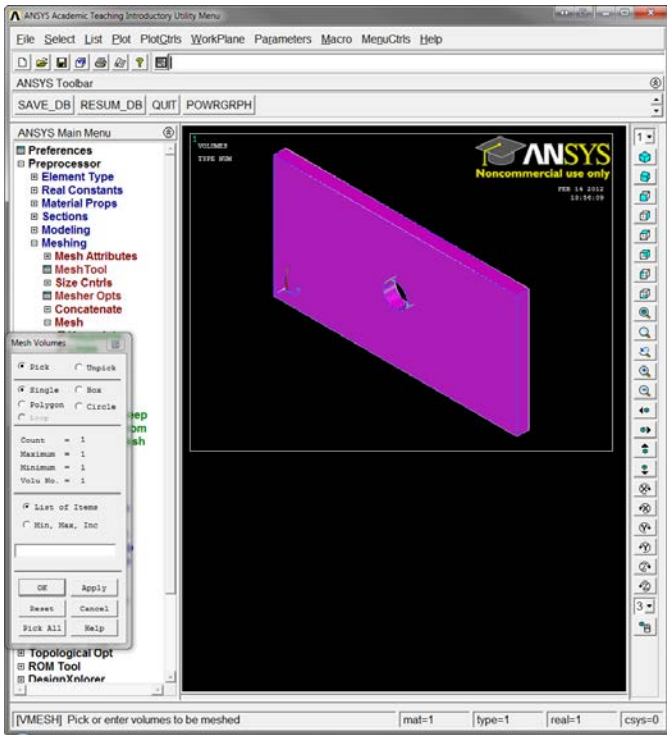
- Select "Add"
- From this window, select "Solid" then "20node 186" and select "OK"

6. Preprocessor → Material Props → Material Models



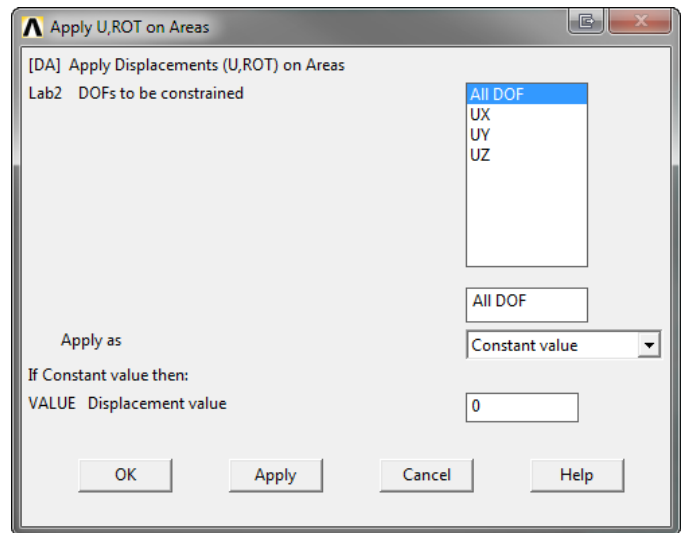
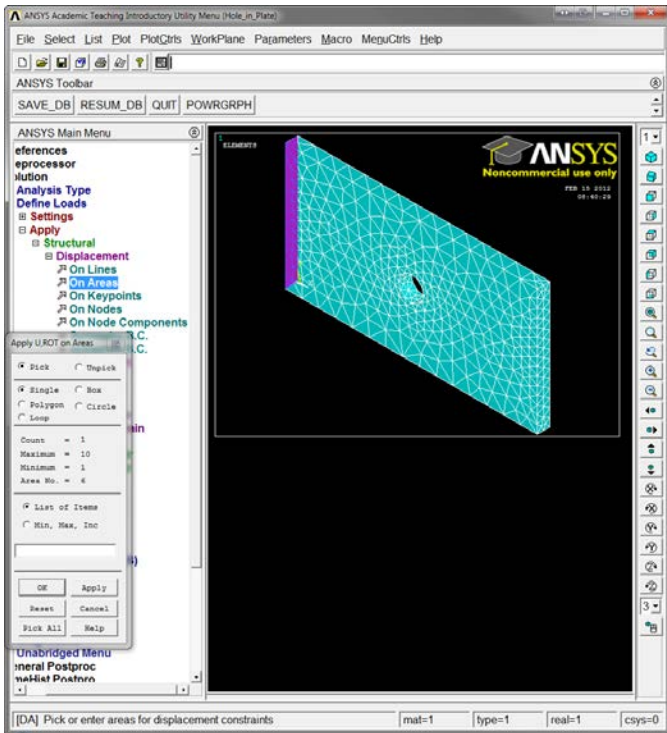
- Select Structural → Linear → Elastic → Isotropic, as seen in the picture above and to the left
- Input the appropriate values into the windows, as seen above: 'EX' is for the Modulus of Elasticity (120000) and 'PRXY' is for the Poisson's Ratio (.3).
- Close the "Define Material Model Behavior" window.

7. Preprocessor → Meshing → Mesh → Volumes → Free



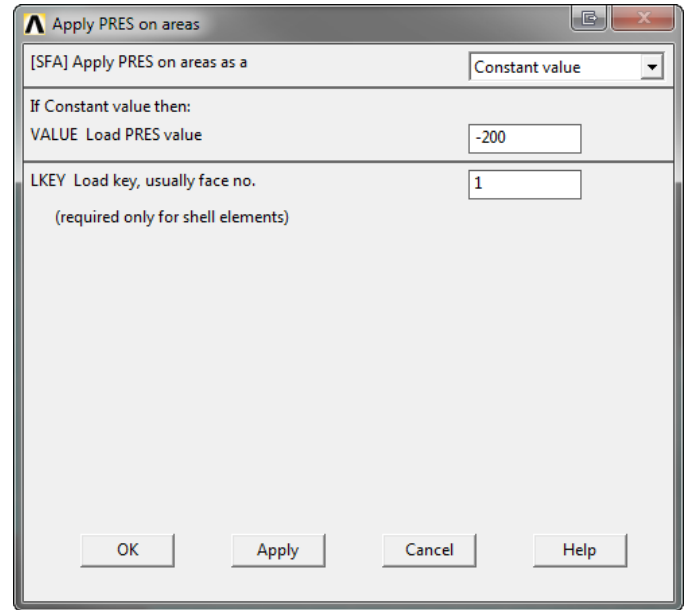
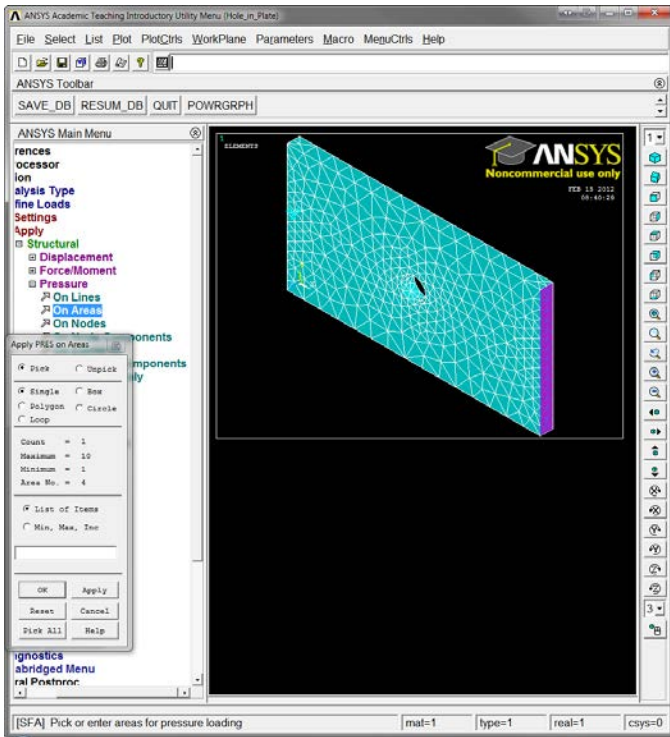
Note: Select the volume by clicking on it and click “OK”, the result should look like the picture above and to the right

8. Solution → Define Loads → Apply → Structural → Displacement → On Areas



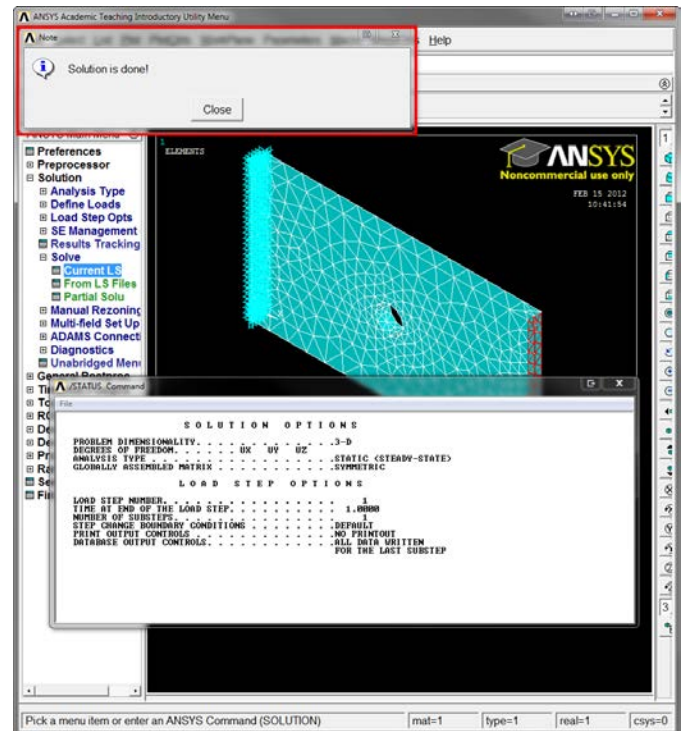
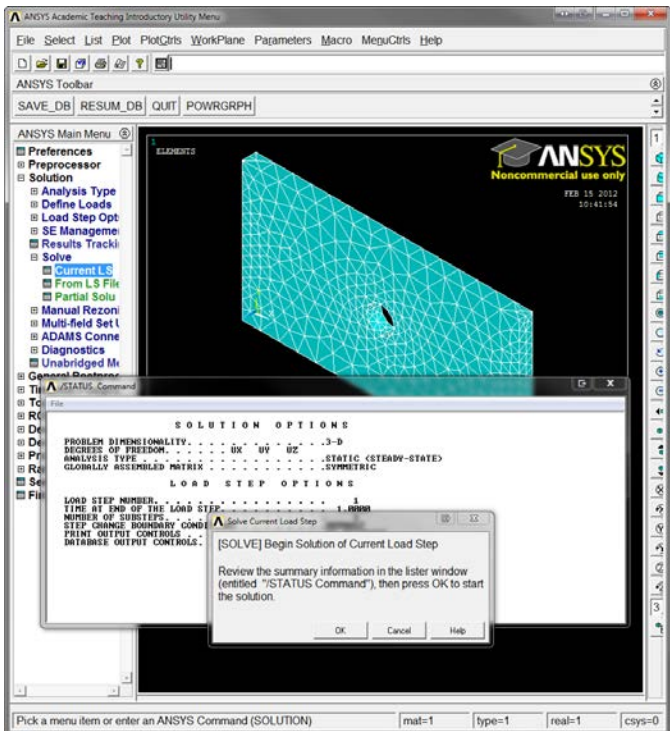
Note: The area can be selected by clicking on it and click “OK”, select the DOFs to be “ALL DOF” in order to lock that side at that position and put the displacement value to “0”

9. Solution → Define Loads → Apply → Structural → Pressure → On Areas



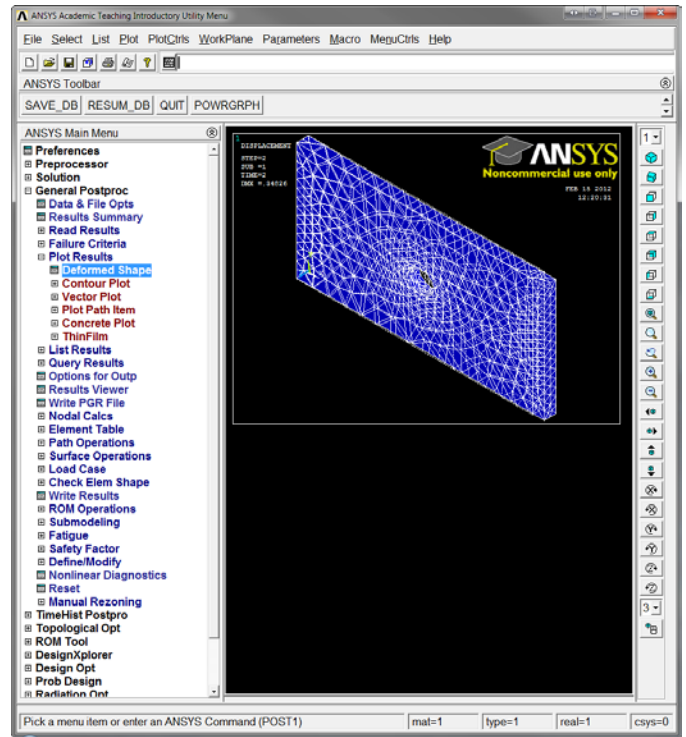
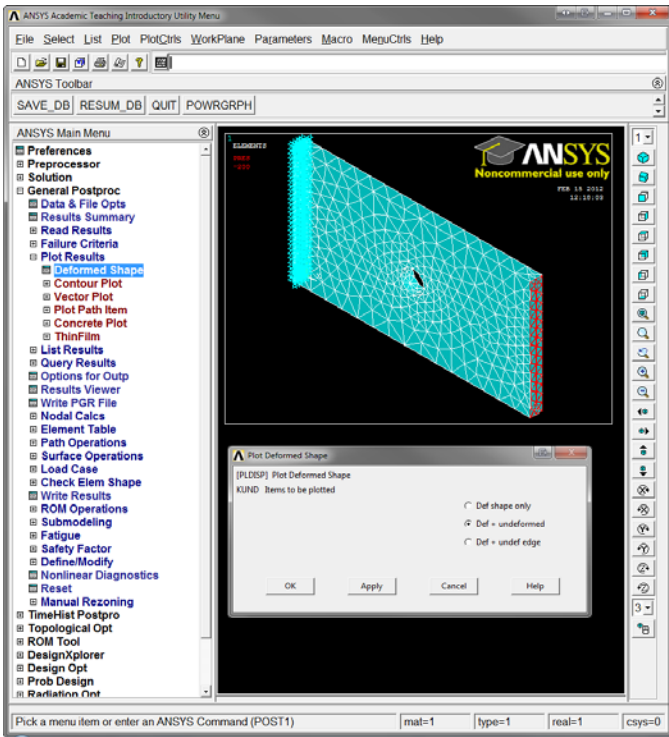
Note: The area can be selected by clicking on it and click “OK”, set the pressure value to “-200” in order to model the uniform pressure distribution found on both sides of the plate.

10. Solution → Solve → Current LS

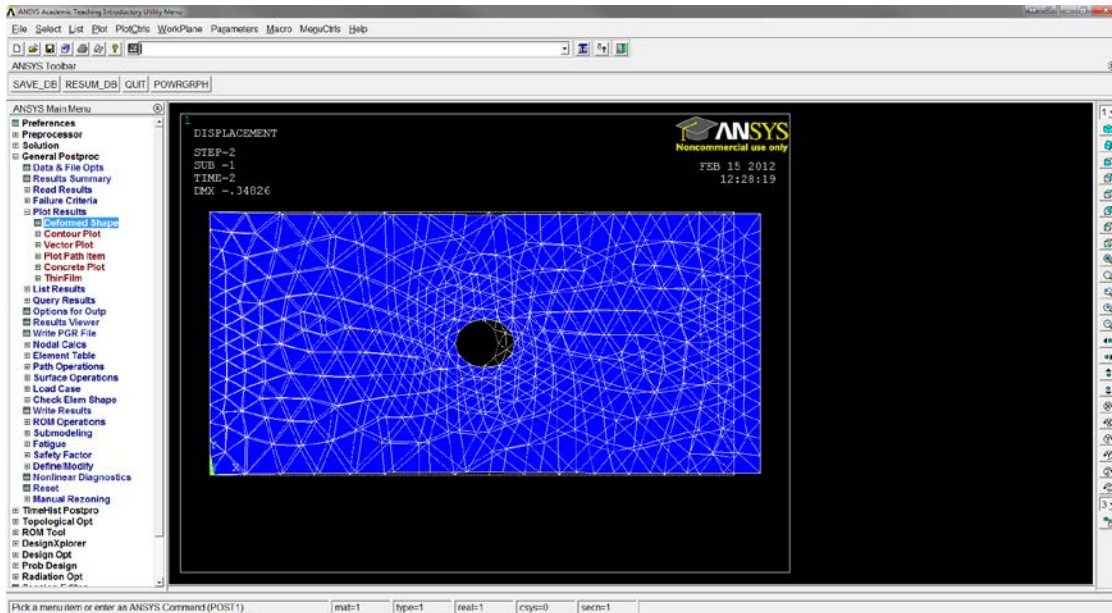


Note: Select “OK” in the box, seen above and to the left. The solution will then be solved. The solution has been successfully when, **and only when**, the window, as seen above and to the right, is displayed.

11. General Postproc → Plot Results → Deformed Shape



Note: Select “Def + undeformed” to see the effects of this applied load on the structure, as seen above and to the right. Look at the front view to see a clearer image of the deformation.



You are now finished!!!