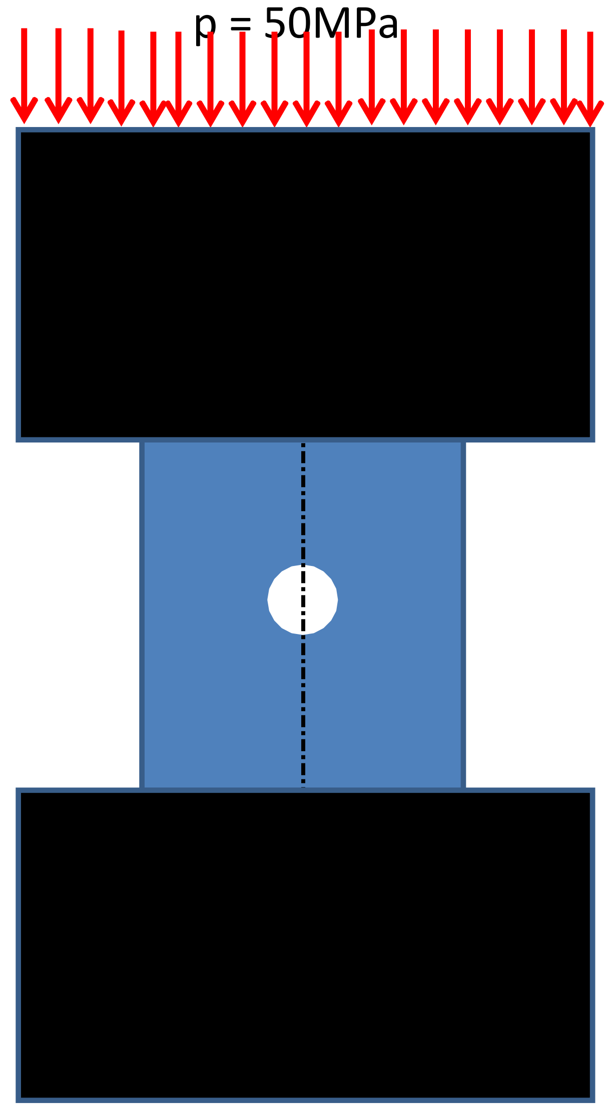
Problem Statement

Determine the stress/strain distribution in the Al alloy plate compressed in between two elastic platens of the hardened steel. Compare the cases when the Al alloy is elastic and elastic-plastic.



$0.1 \times 0.25 m^2$ with a center hole of diameter of 0.02m; plane stress analysis

Steel platens: $0.25 \times 0.25 mm^2$

E = 70GPa (Al alloy)

ν = 0.3

YS = 100 MPa

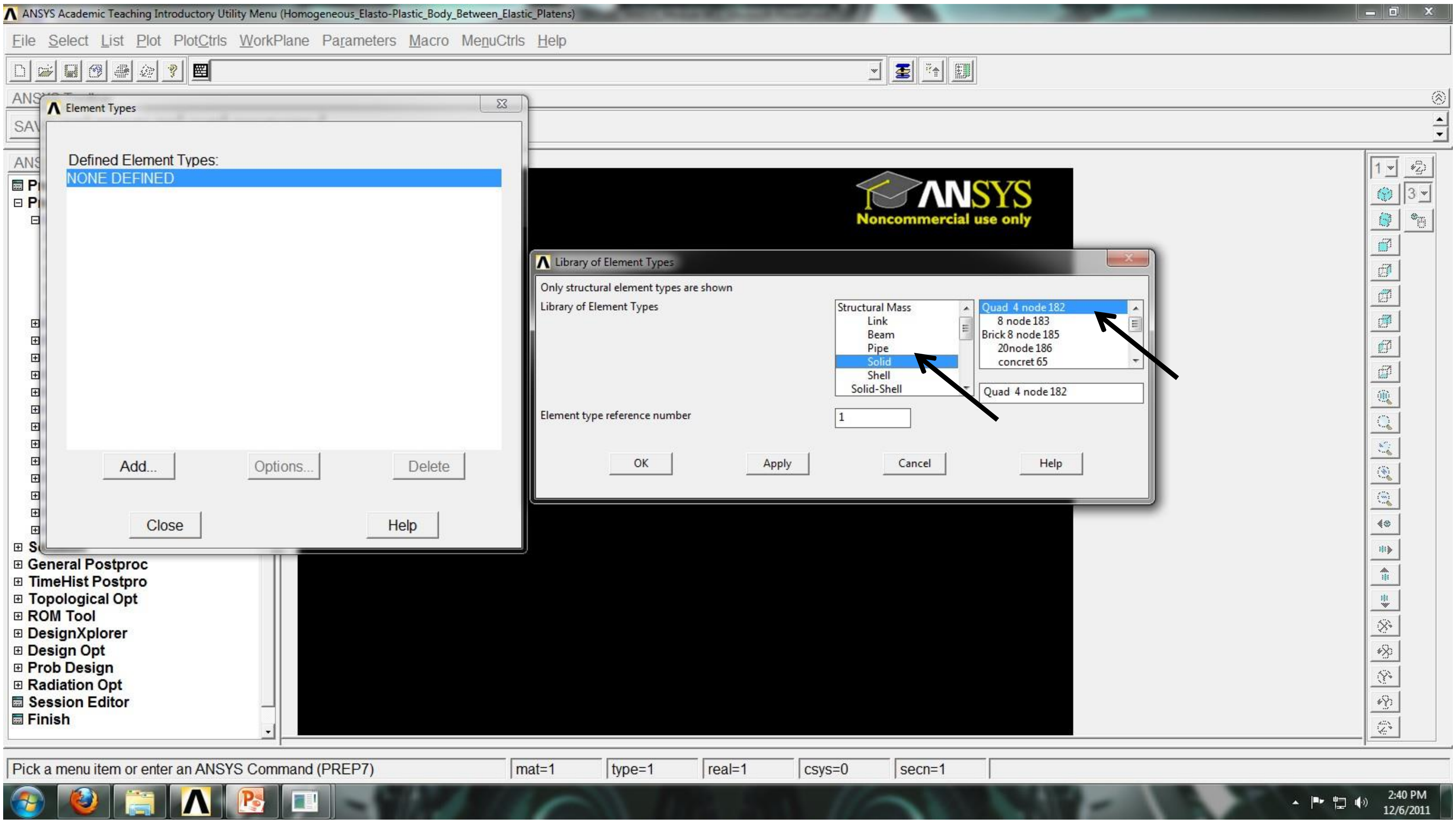
Tangent Modulus = 400MPa

E = 300GPa (Hardened Steel alloy)

ν= 0.3

Choose Element – Basic Element

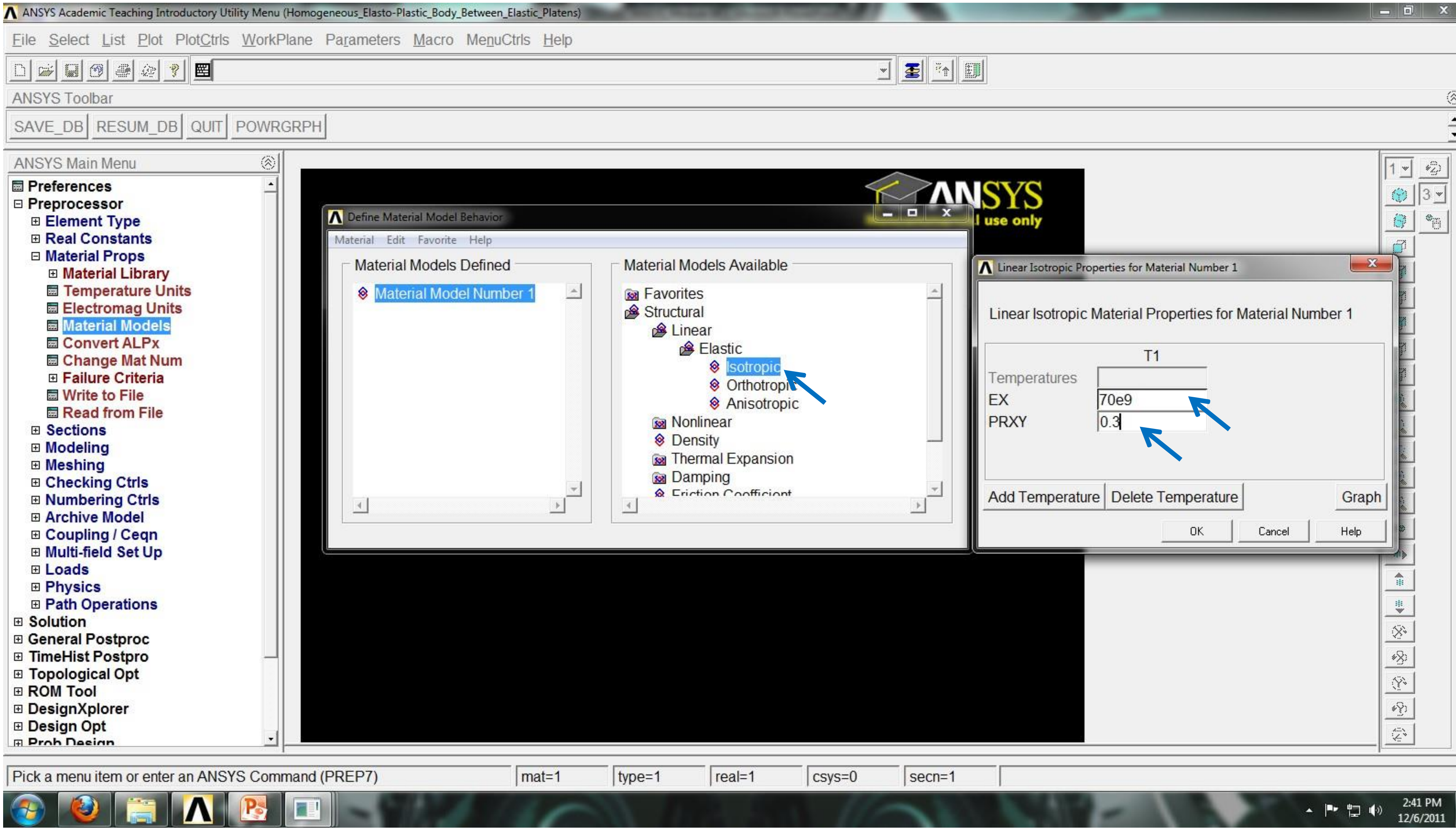
- Preprocessor → Element type → Add/Edit/Delete
- Add
- Solid → Quad 4node 182 (4 nodes, one at each corner)



- **Material Model**

Structural → Linear → Elastic → Isotropic

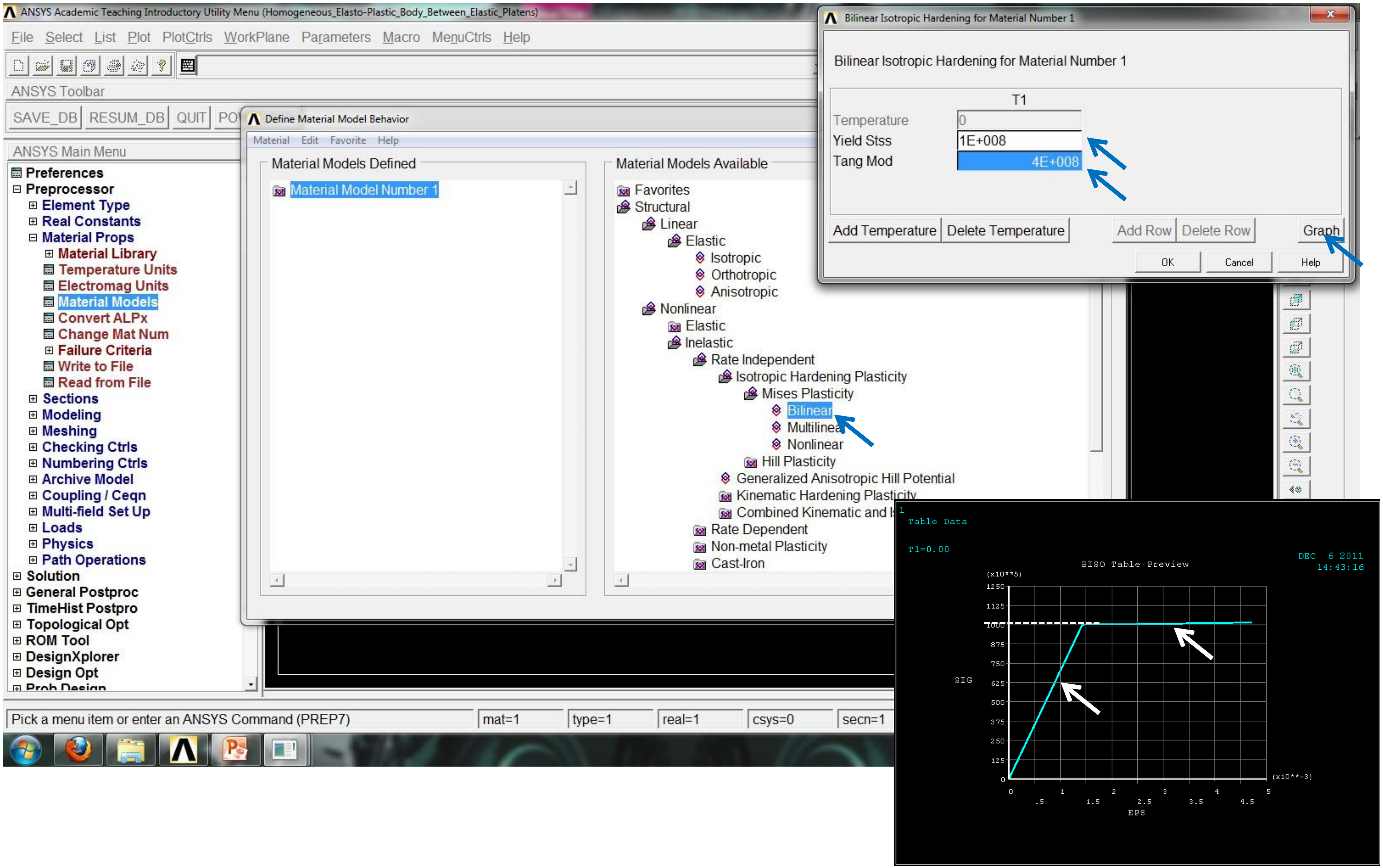
EX = 70GPa PRXY = 0.3



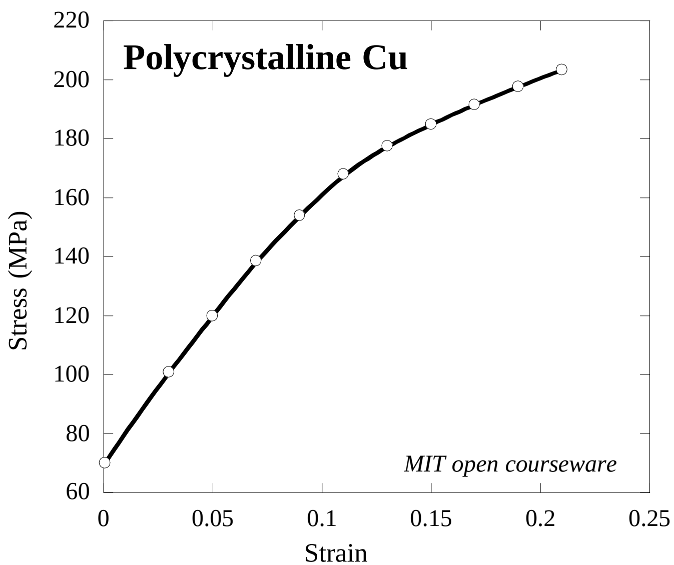
- **Material Model**

Structural → Nonlinear → Inelastic → Rate Independent → Isotropic Hardening Plasticity → Mises Plasticity → Bilinear

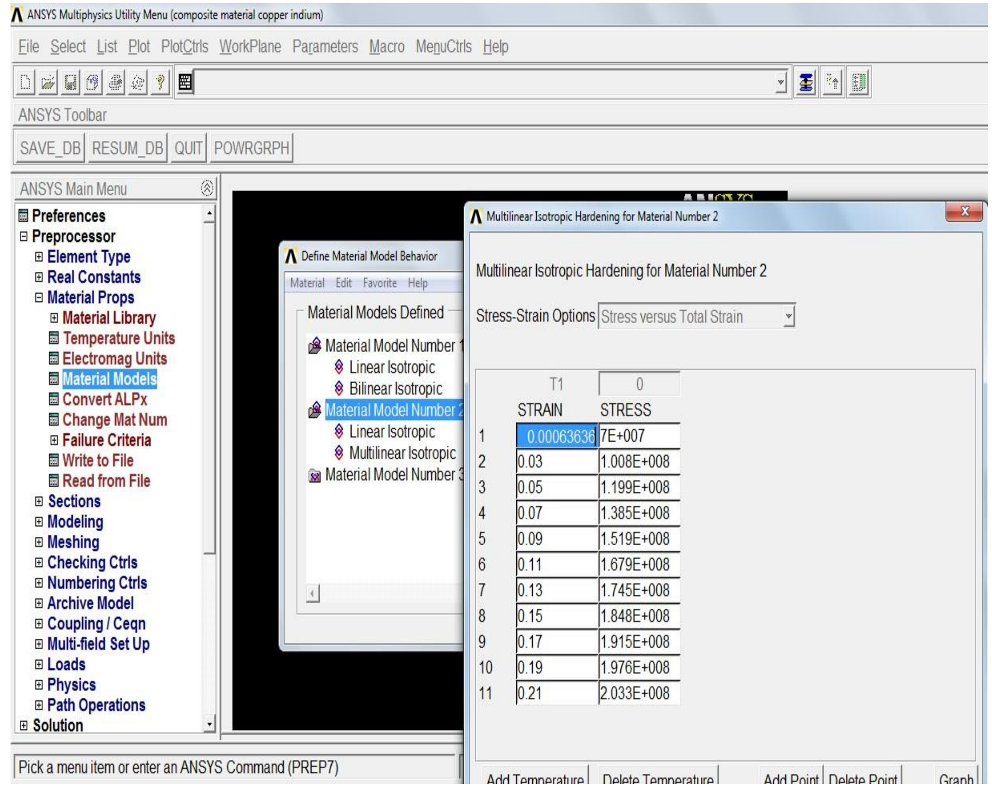
EX = 100MPa Tang Mod = 400MPa



If Other options of plasticity are chosen, then:



If Multilinear option is chosen, then:



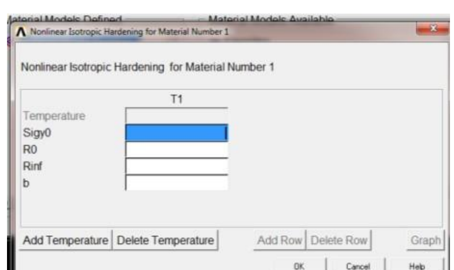
If Non-linear option is chosen, then:

- Conduct best fit curve analysis with following equation (a form of Voce's equation – rigid-plastic material)

$$\sigma = \sigma_{YS} + R_0 \varepsilon_{pl} + R_\infty \left(1 - e^{-b\varepsilon_{pl}}\right)$$

- Comparison with Voce's equation:

$$C = \sigma_{YS} + R_0 \varepsilon_{pl} + R_\infty \qquad m = \frac{R_\infty}{C} \qquad n = b$$

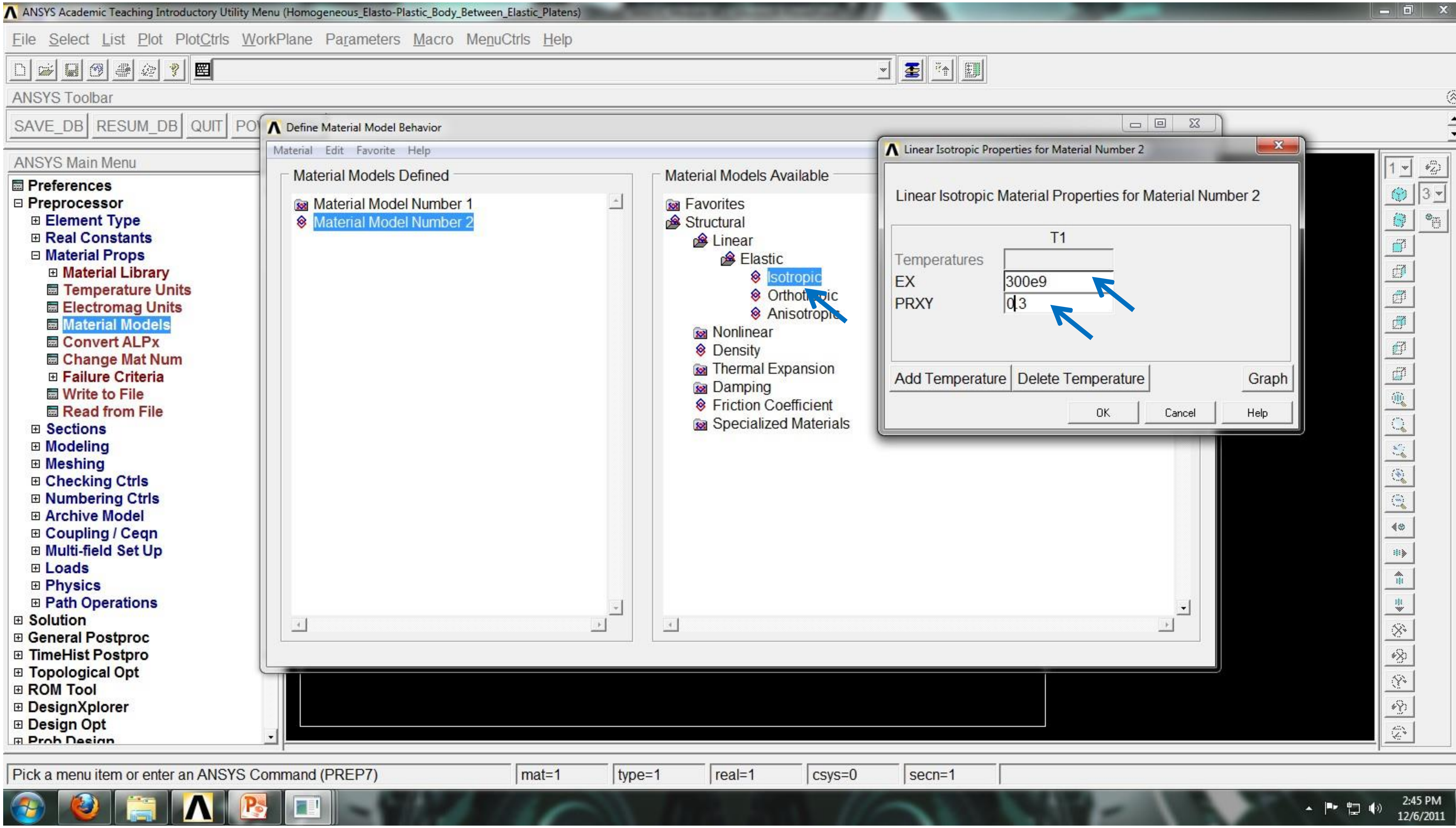


- **Material Model**

  File → New Model

  Structural → Linear → Elastic → Isotropic

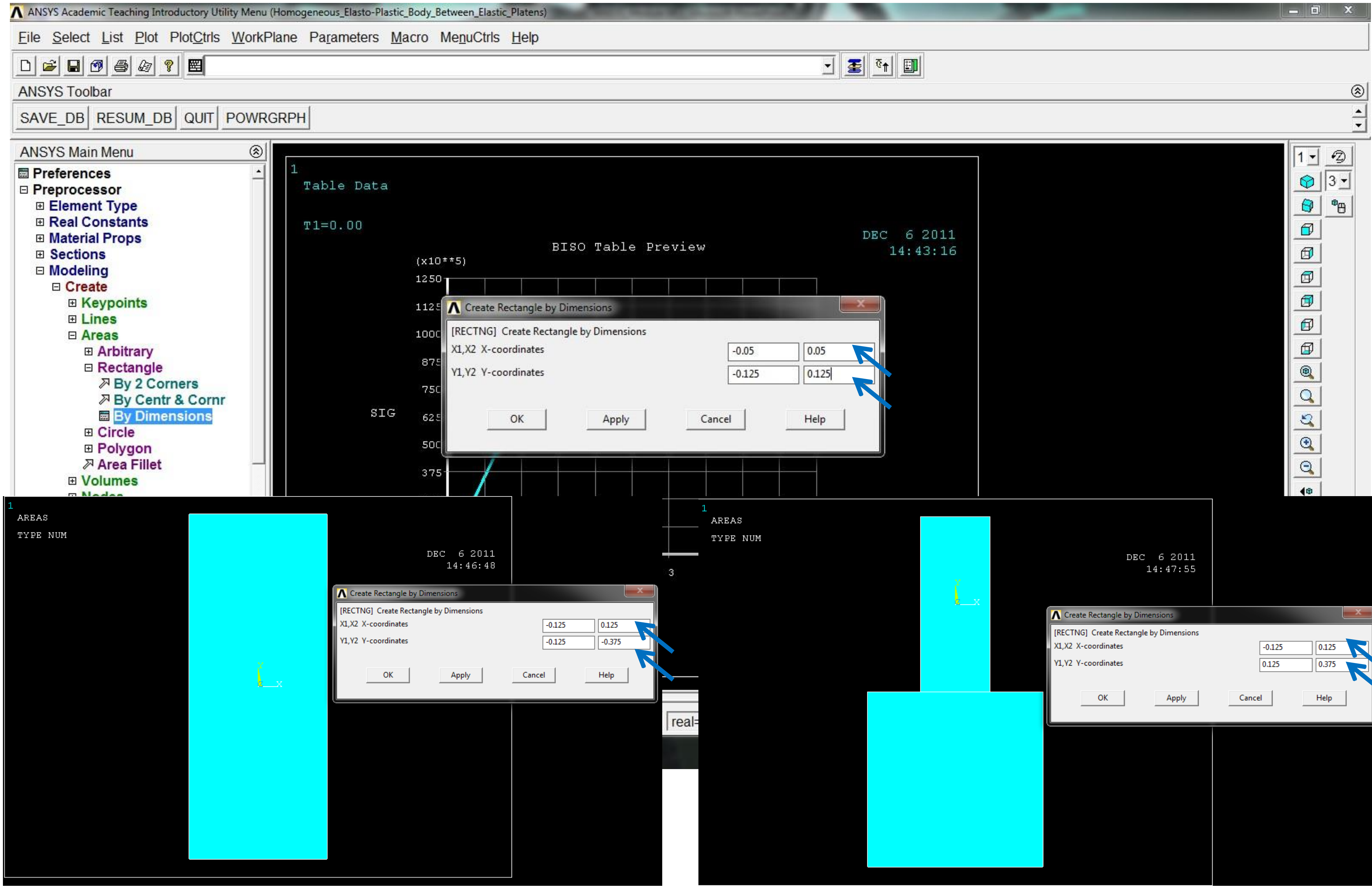
  EX = 300GPa PRXY = 0.3



Create Geometry (Modeling)

Create Rectangles

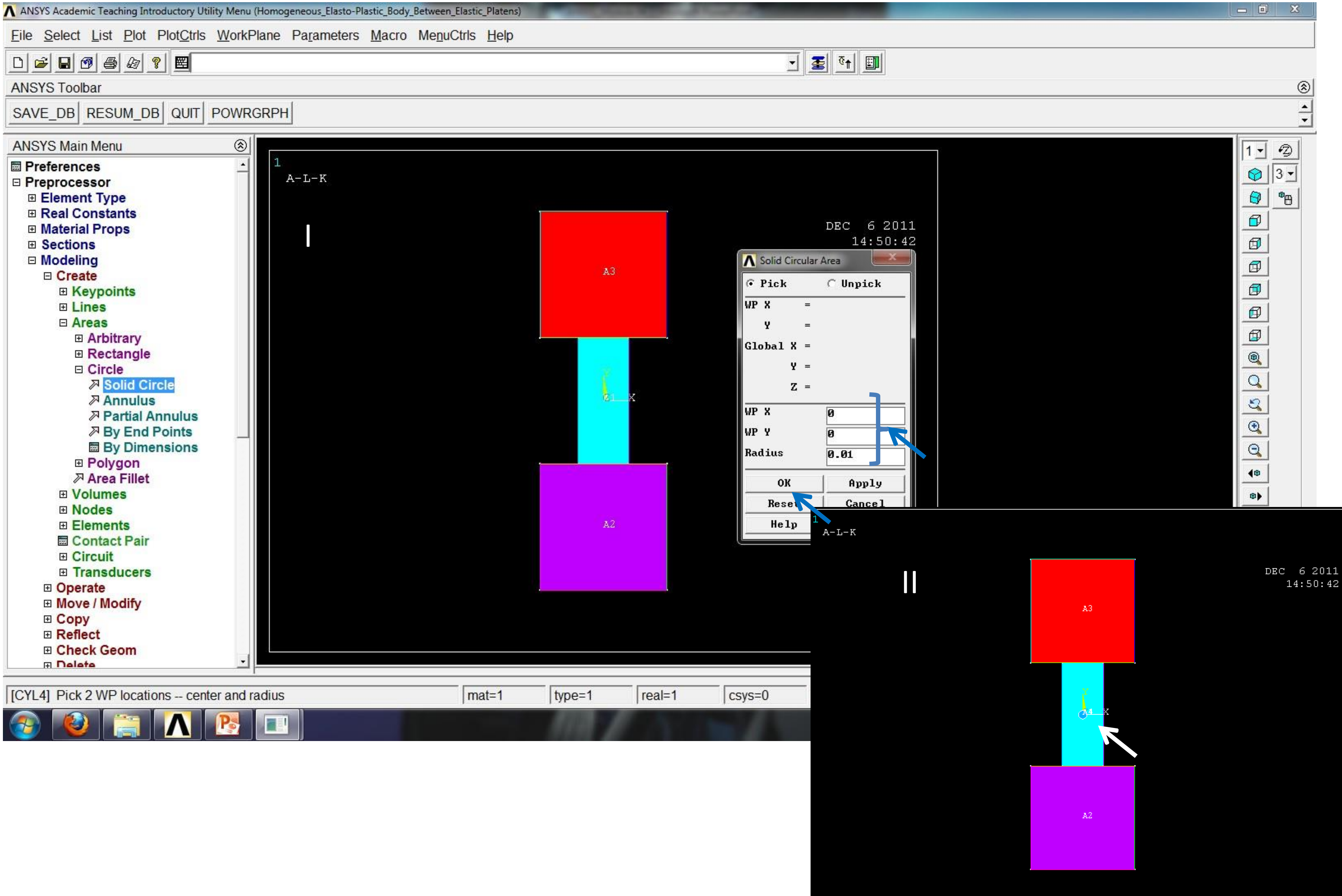
Preprocessor → Create → Areas → Rectangle → By Dimension



Create Geometry (Modeling)

Create Circular Area for Making Hole

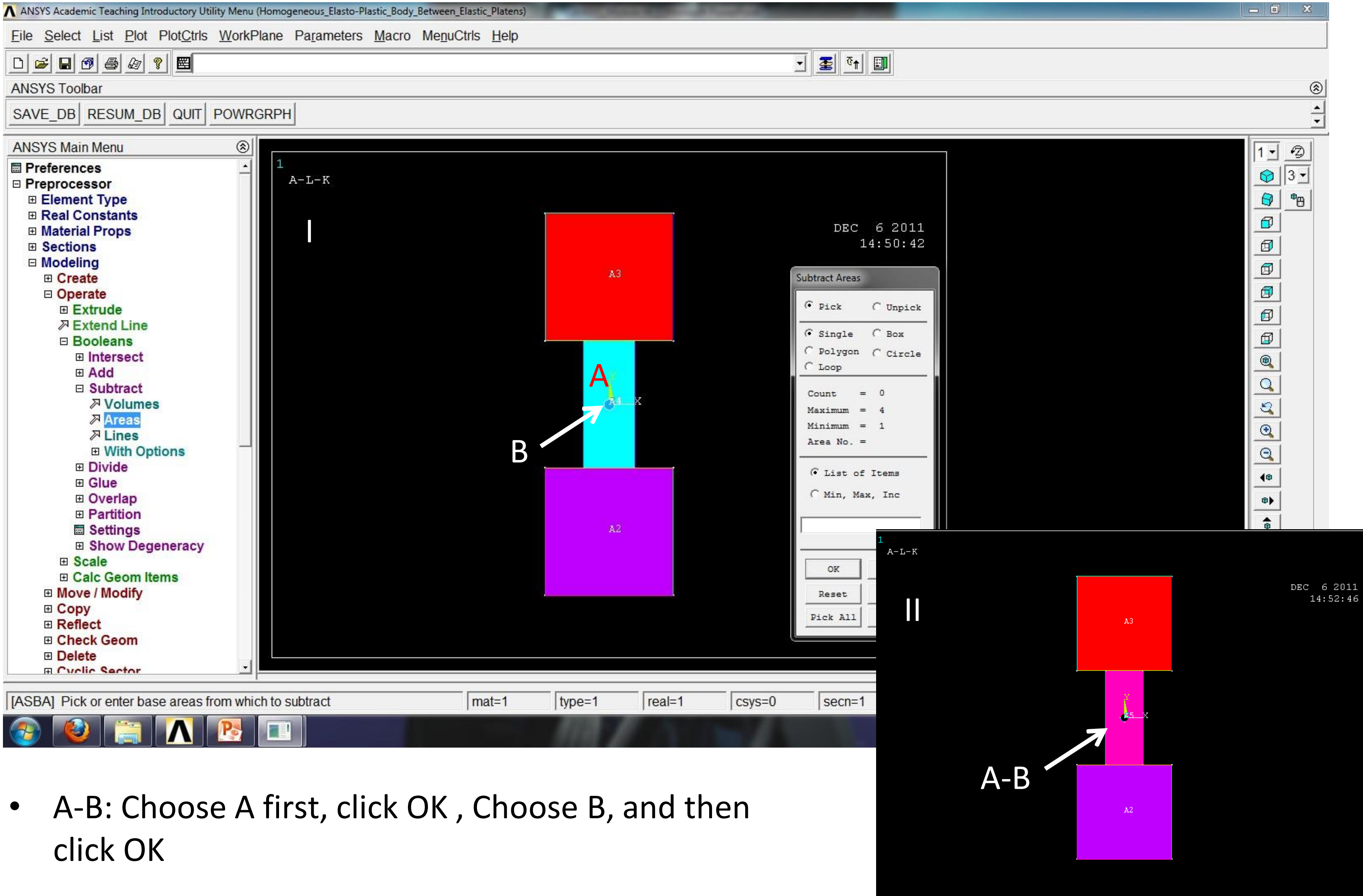
Preprocessor → Create → Areas → Circle → Solid Circle



Create Geometry (Modeling)

Creating a hole in the Al alloy sample

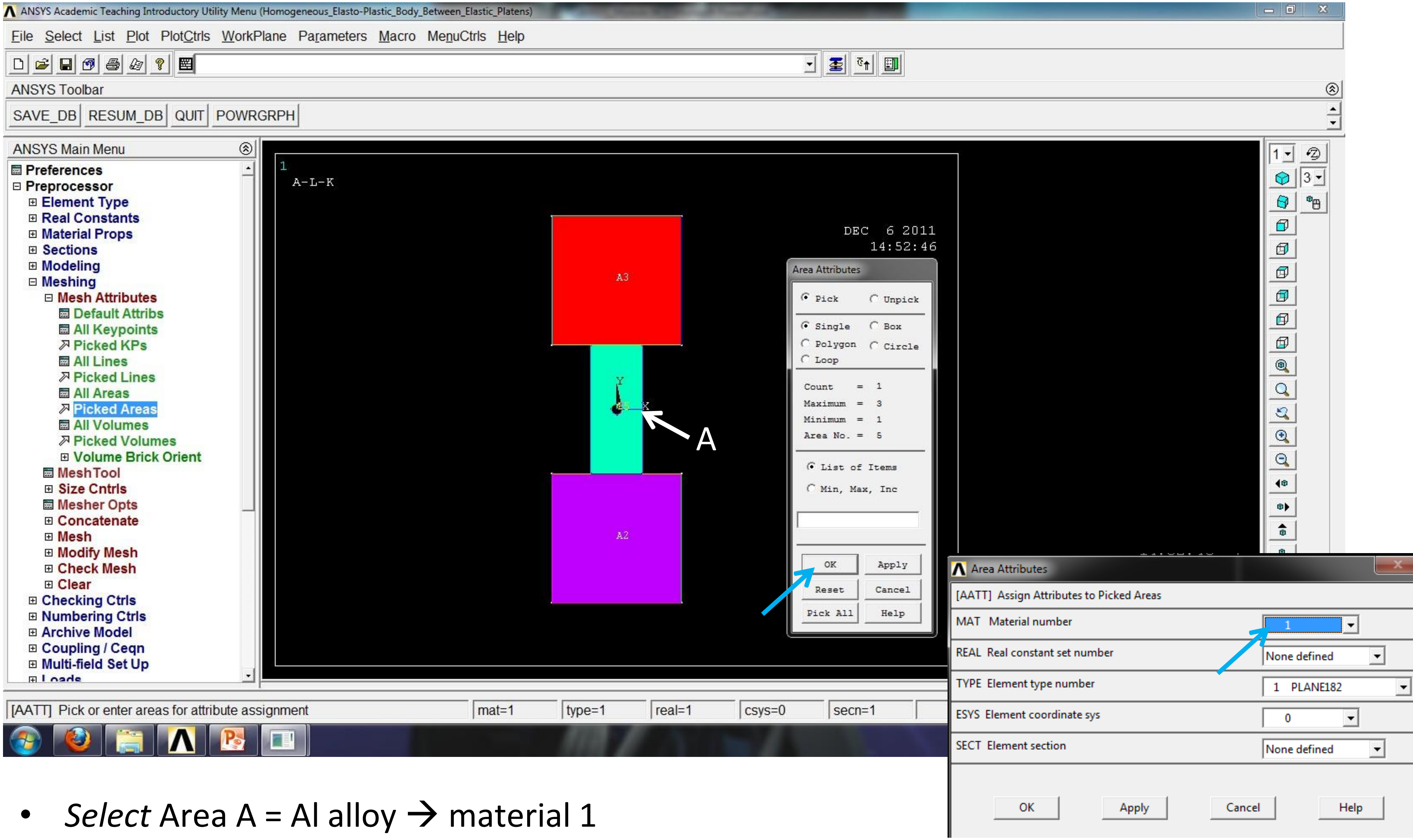
Preprocessor →Modeling → Operate → Booleans → Subtract → Area



- A-B: Choose A first, click OK , Choose B, and then click OK

Assigning Different Areas to Different Materials Attributes

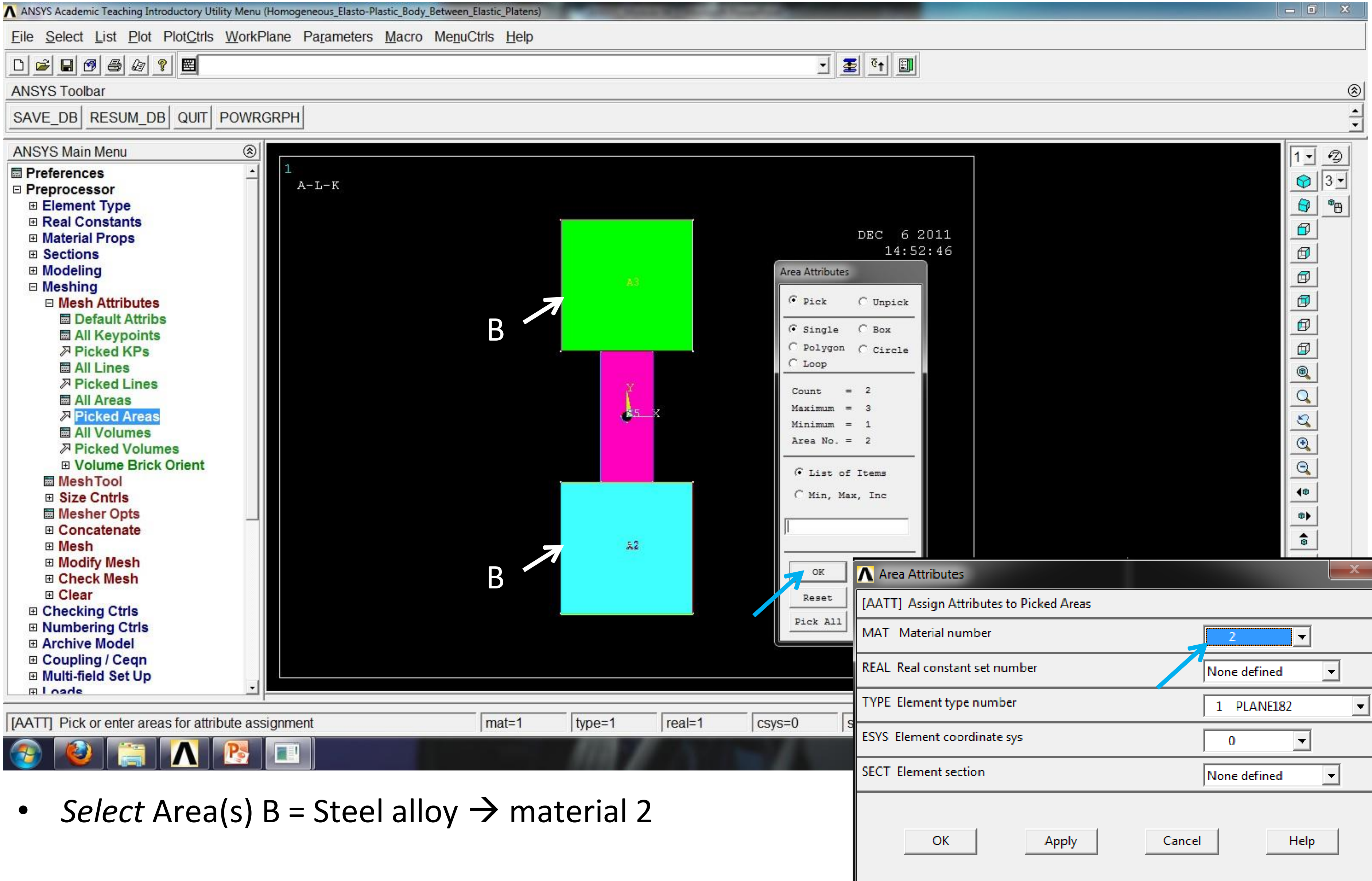
Preprocessor → Meshing → Mesh Attributes → Picked Areas



- *Select* Area A = Al alloy → material 1

Assigning Different Areas to Different Materials Attributes

Preprocessor → Meshing → Mesh Attributes → Picked Areas

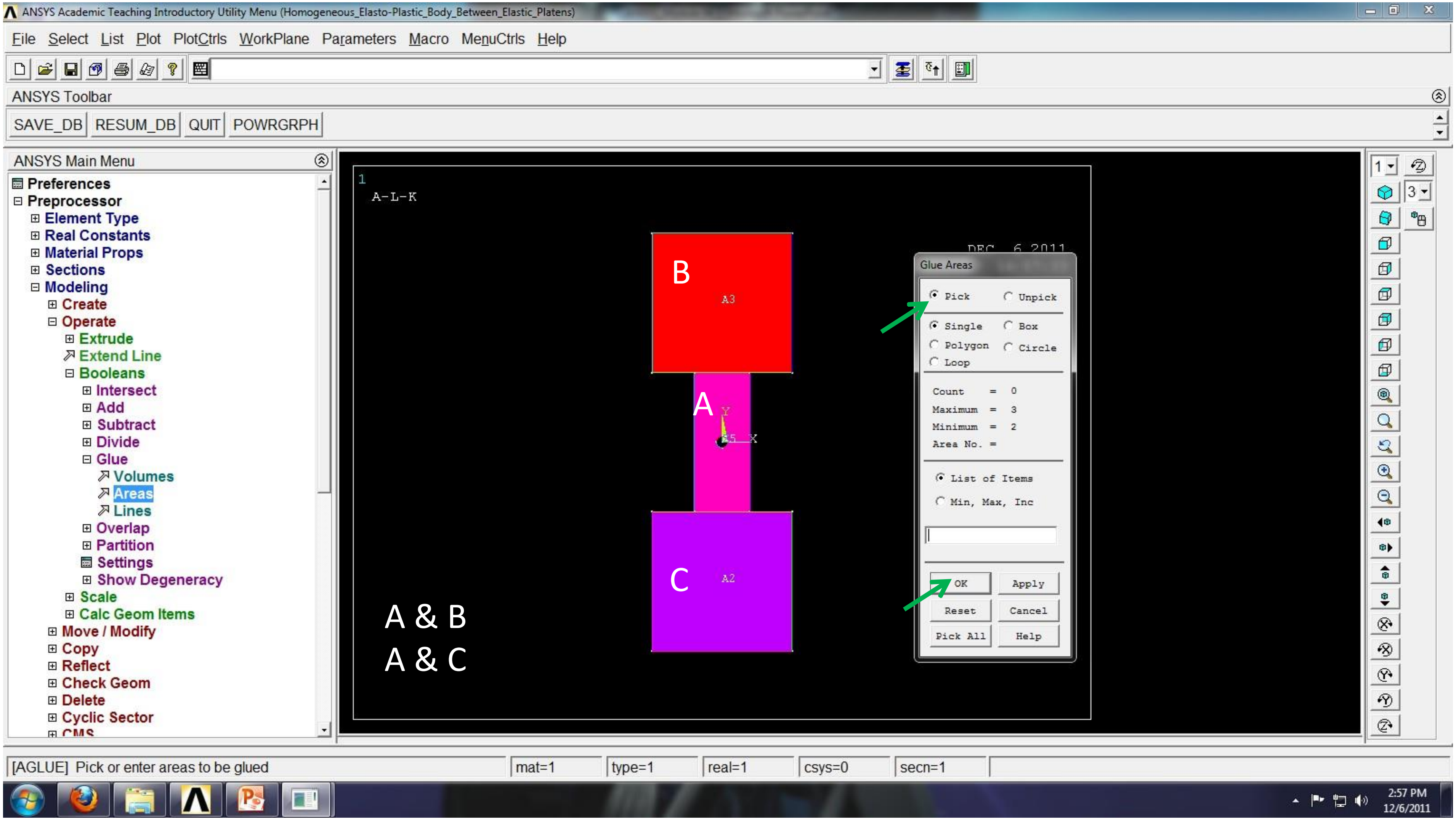


- *Select* Area(s) B = Steel alloy → material 2

Glue Areas (Al alloy and Steel platens)

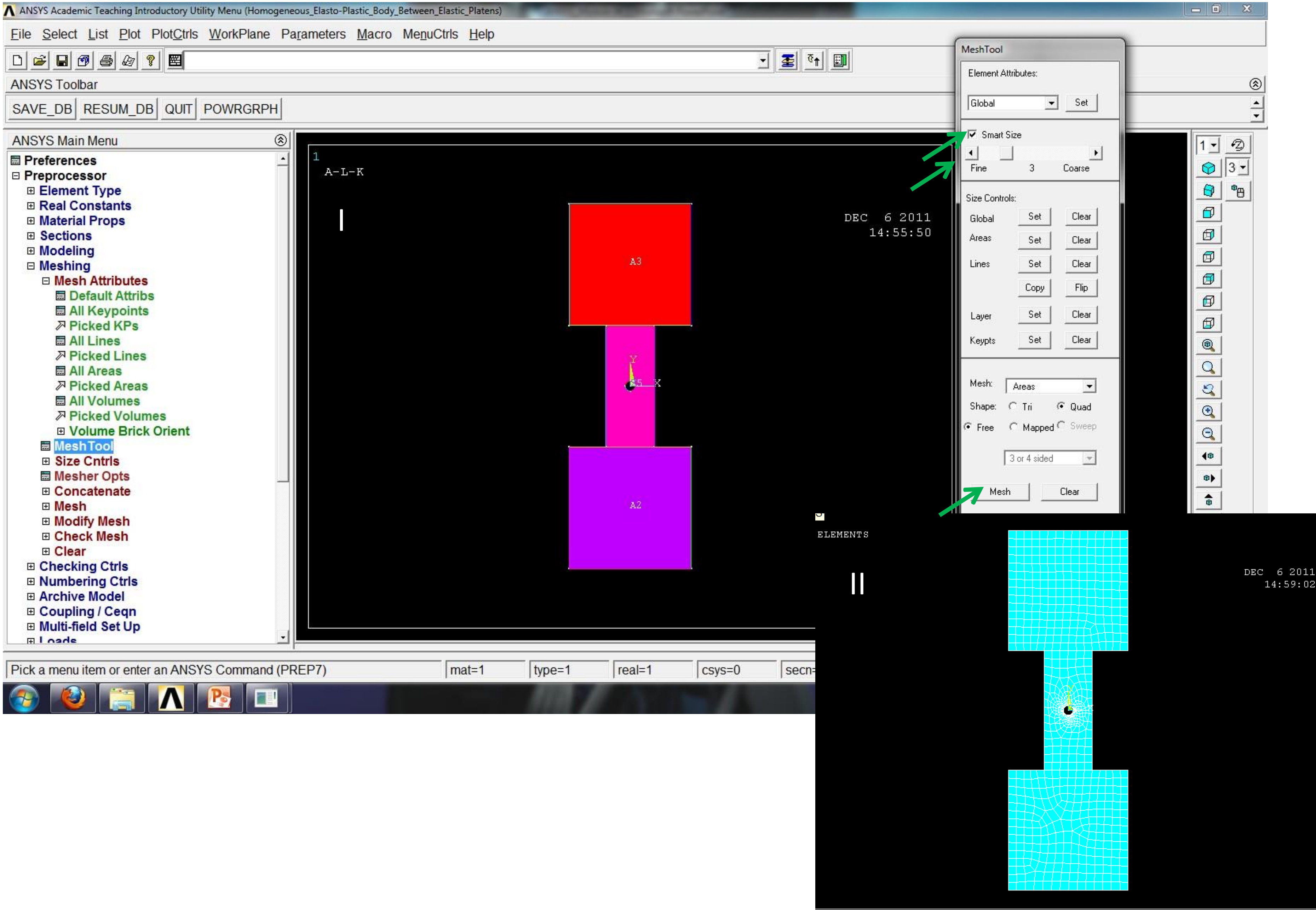
Preprocessor → Modeling → Operate → Glue → Areas

Glue a pair of areas at a time together!



Meshing (Initial)

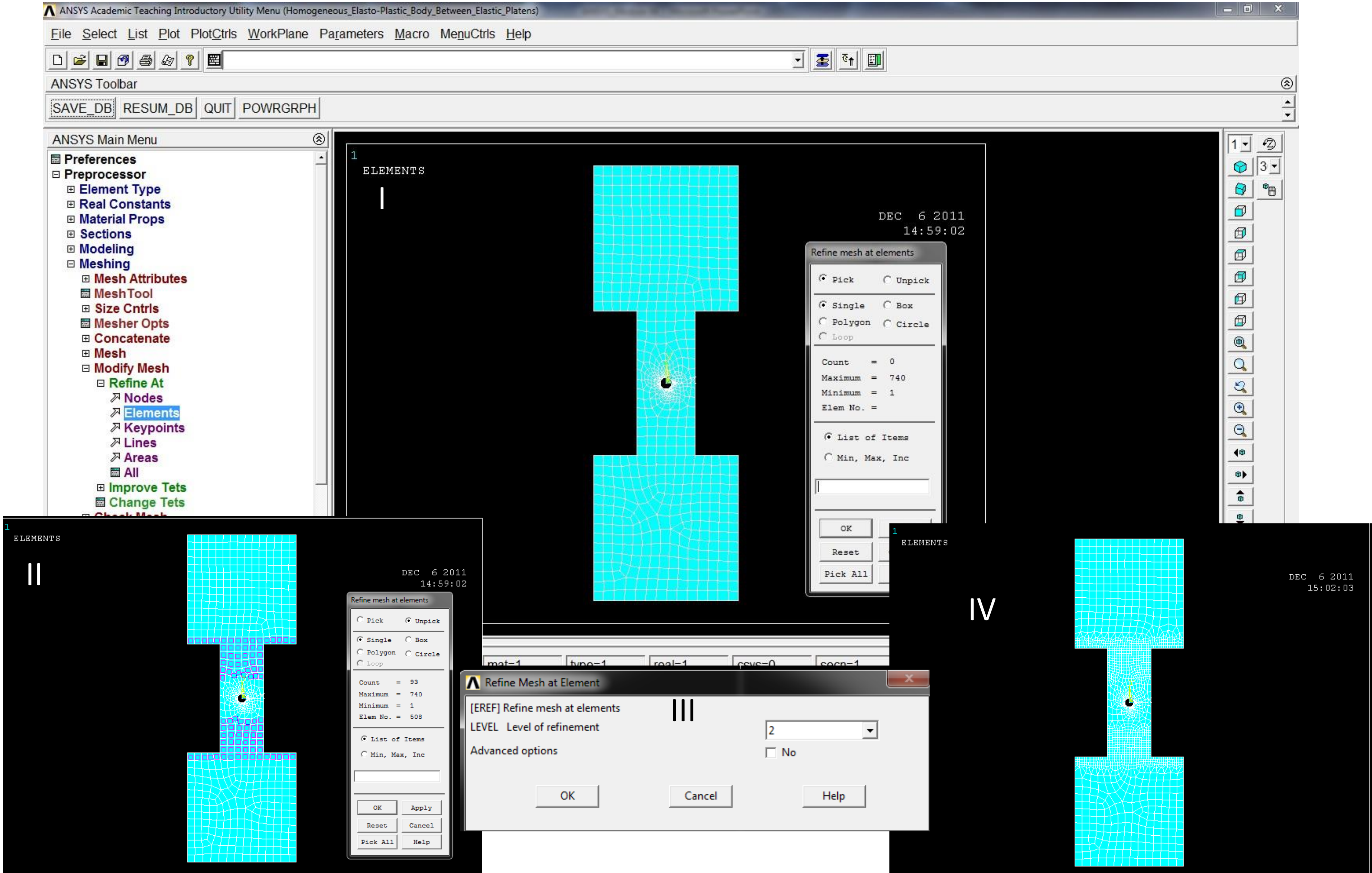
Preprocessor → Meshing→ Mesh Tool



Meshing → Refining meshes near the platens/sample interface

Preprocessor → Meshing→ Modify Mesh → Refine At → Elements

Pick elements *near platens/sample interface* → Select "2"

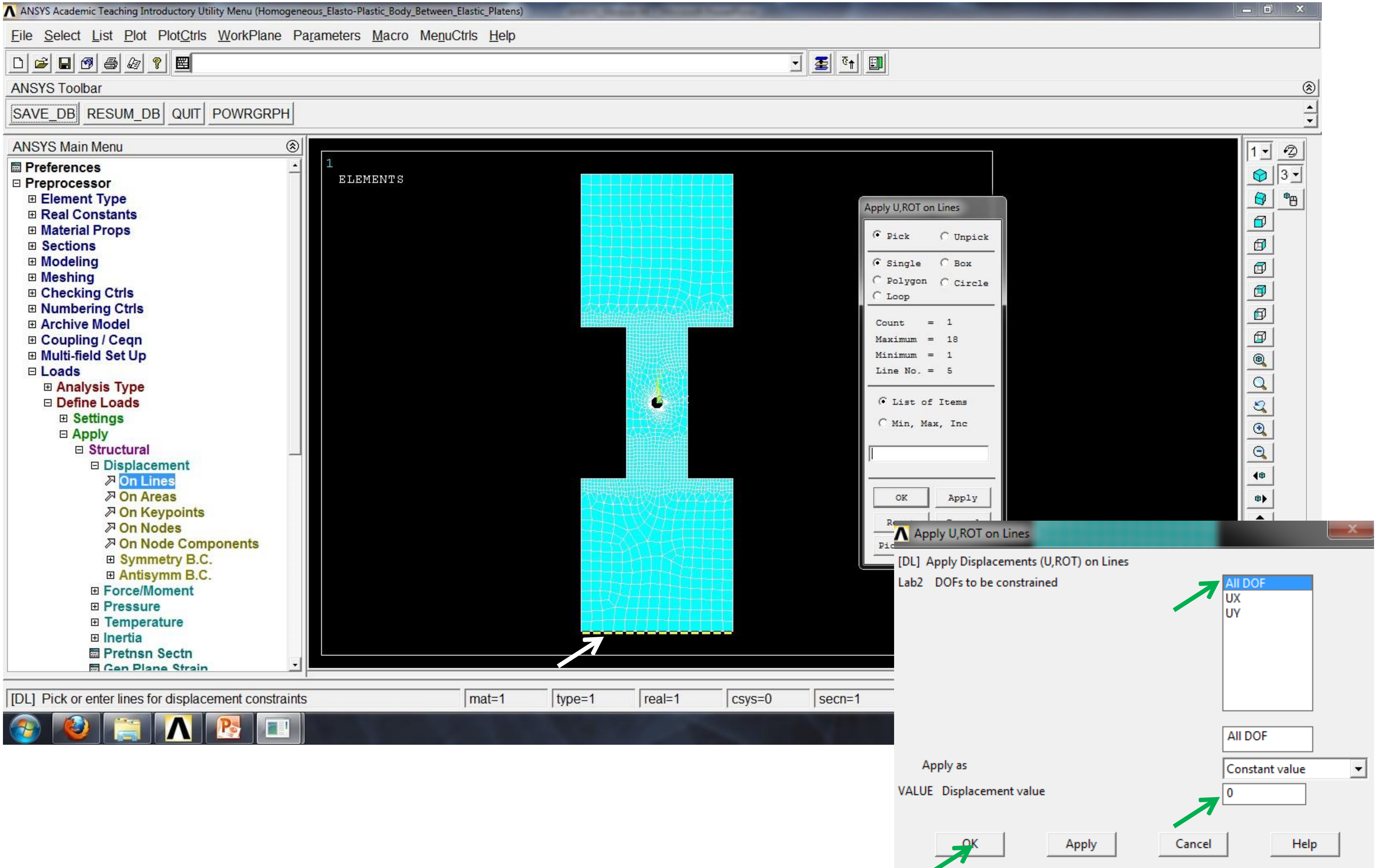


Applying Displacement Boundary Conditions

Preprocessor → Loads→ Apply → Structural → Displacement

Choose *on Line* → select *bottommost lines* (it is highlighted)→ Press *OK*

In the pop-up window: Select All DOF → Choose “0” as the given value

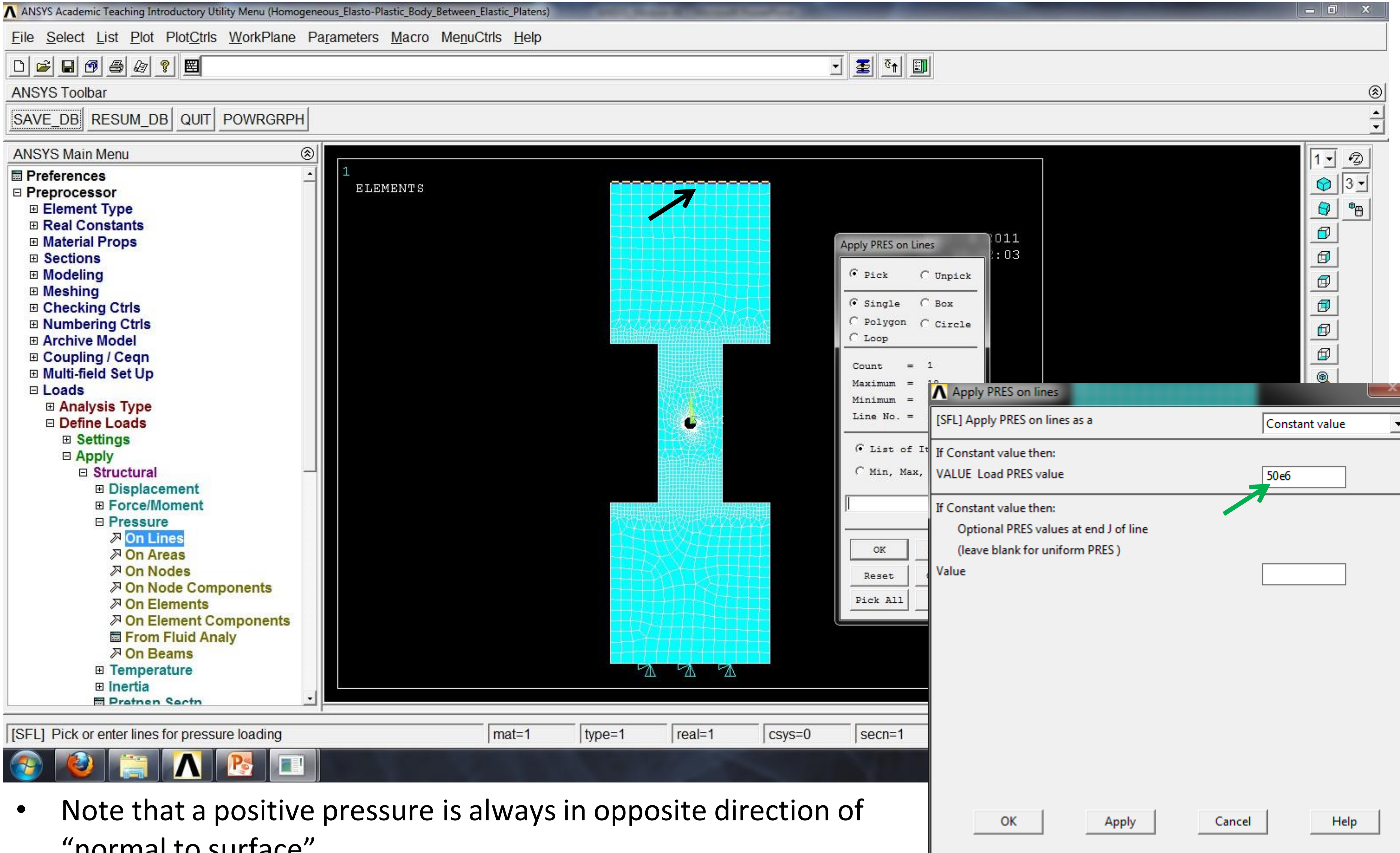


Applying Displacement Boundary Conditions

Preprocessor → Loads→ Apply → Pressure → On Line

Choose *Top most line*→ Press *OK*

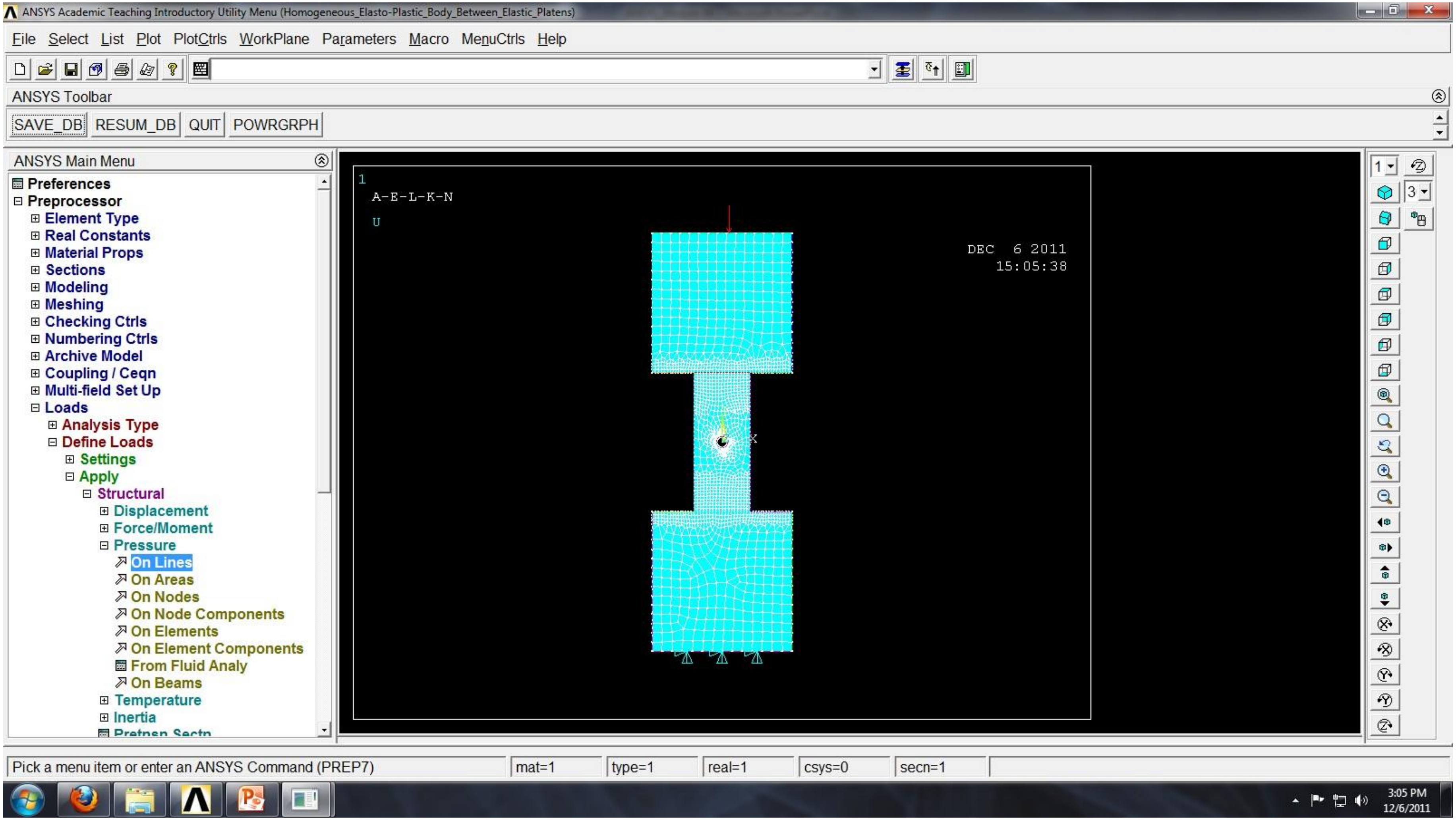
In the pop-up window: *Load PRES* = 50e6 (i.e. 50MPa)



- Note that a positive pressure is always in opposite direction of "normal to surface"

Plots → Multi Plots

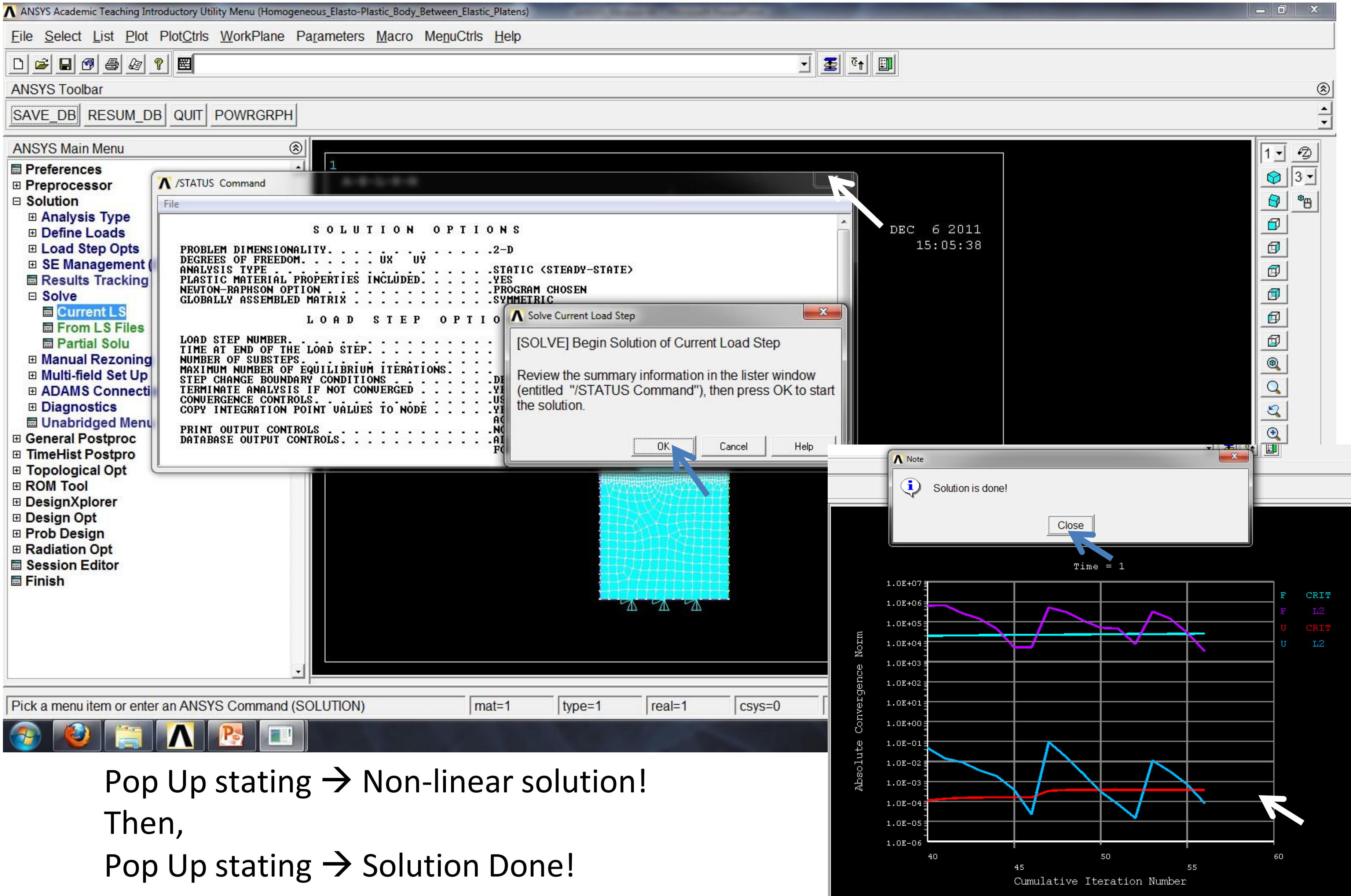
→ Check if all boundary conditions and loads are correct!



Solving the Model

Solution → Current LS

Choose *OK*



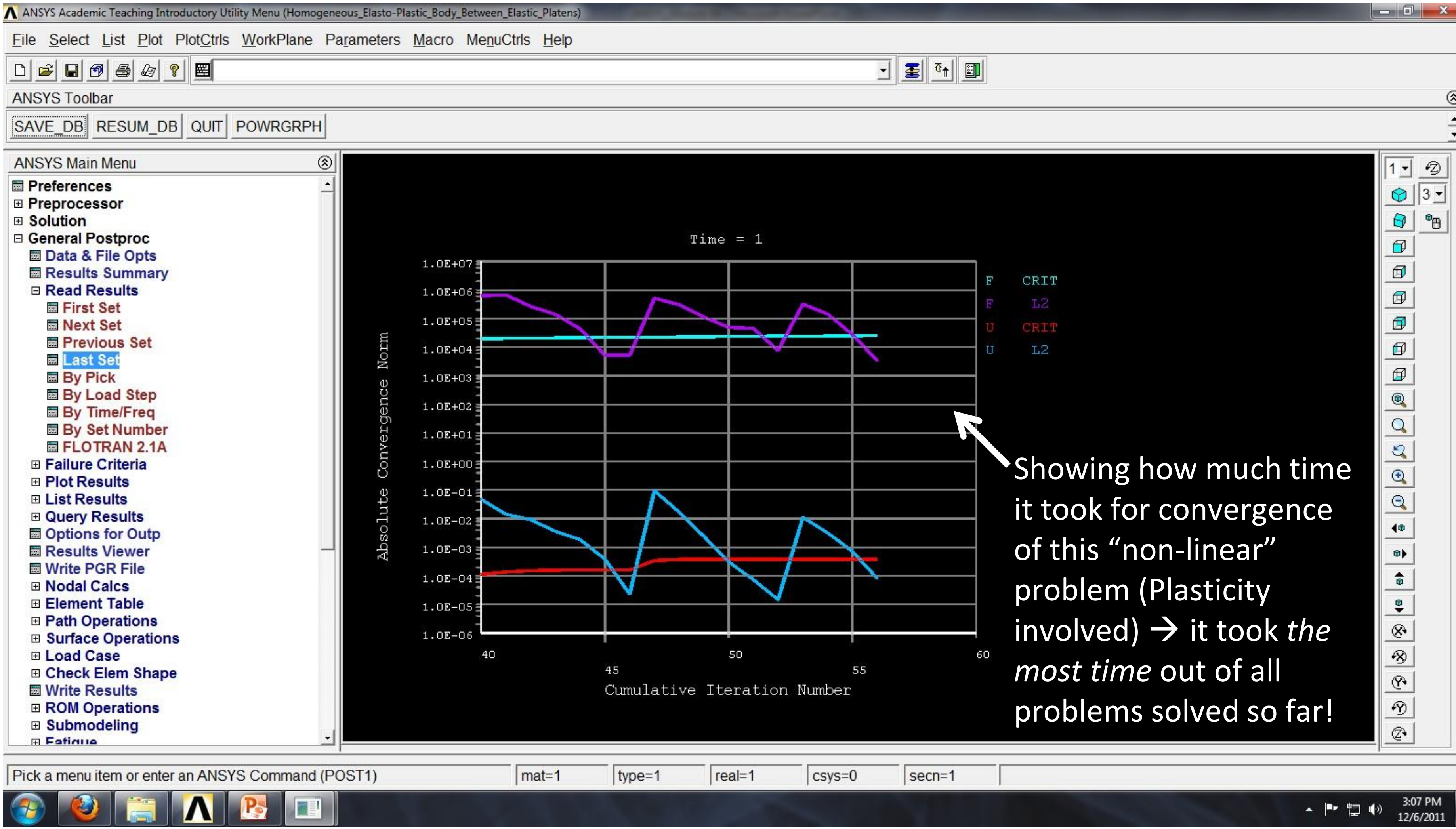
Pop Up stating → Non-linear solution!

Then,

Pop Up stating → Solution Done!

Reading Results

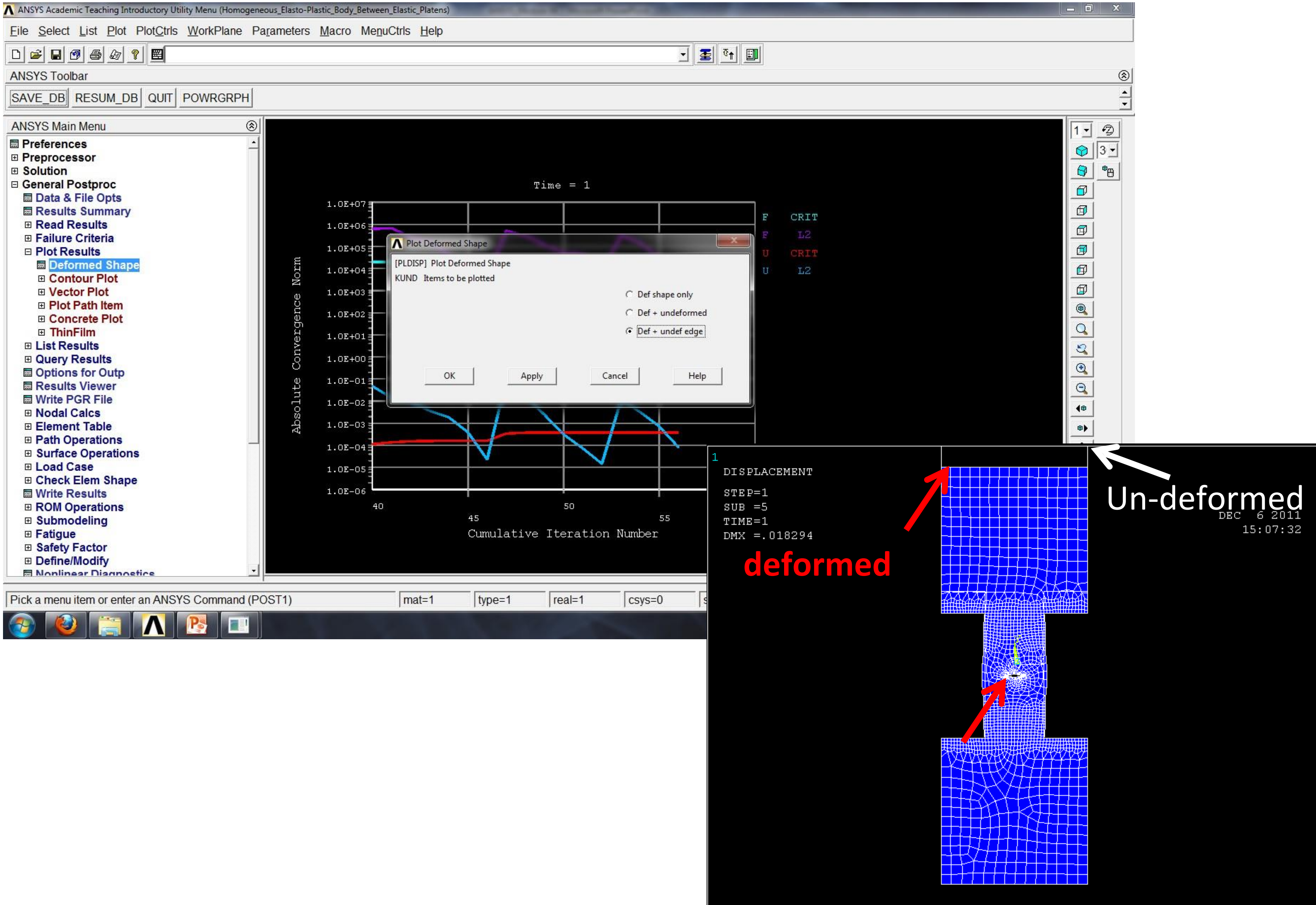
General PostProc → Read Results → Last Set



Plotting/Listing Results

General PostProc → Plot Results → Deformed Shape

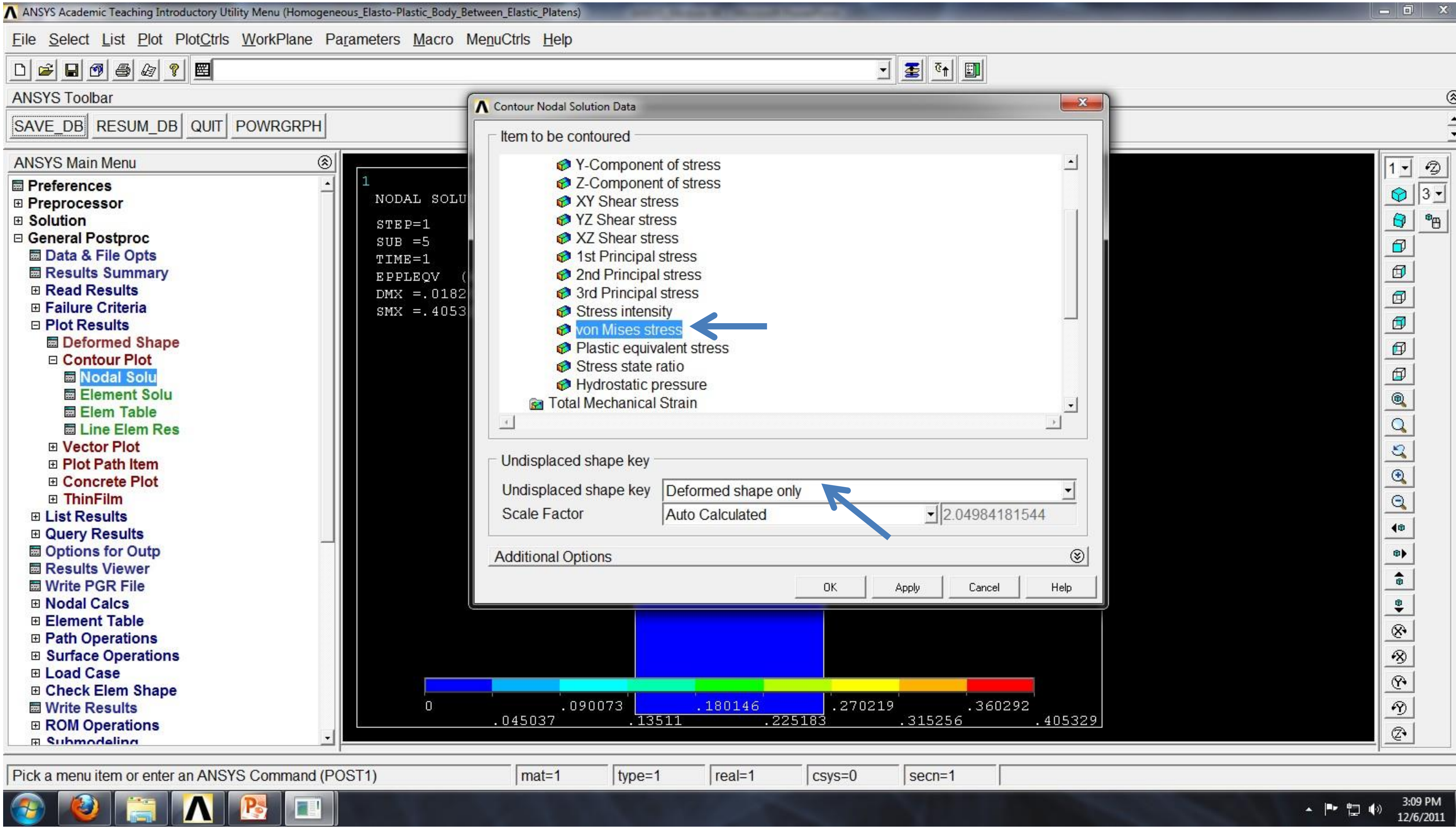
Choose *Def+ Undef Edge*



Plotting Results

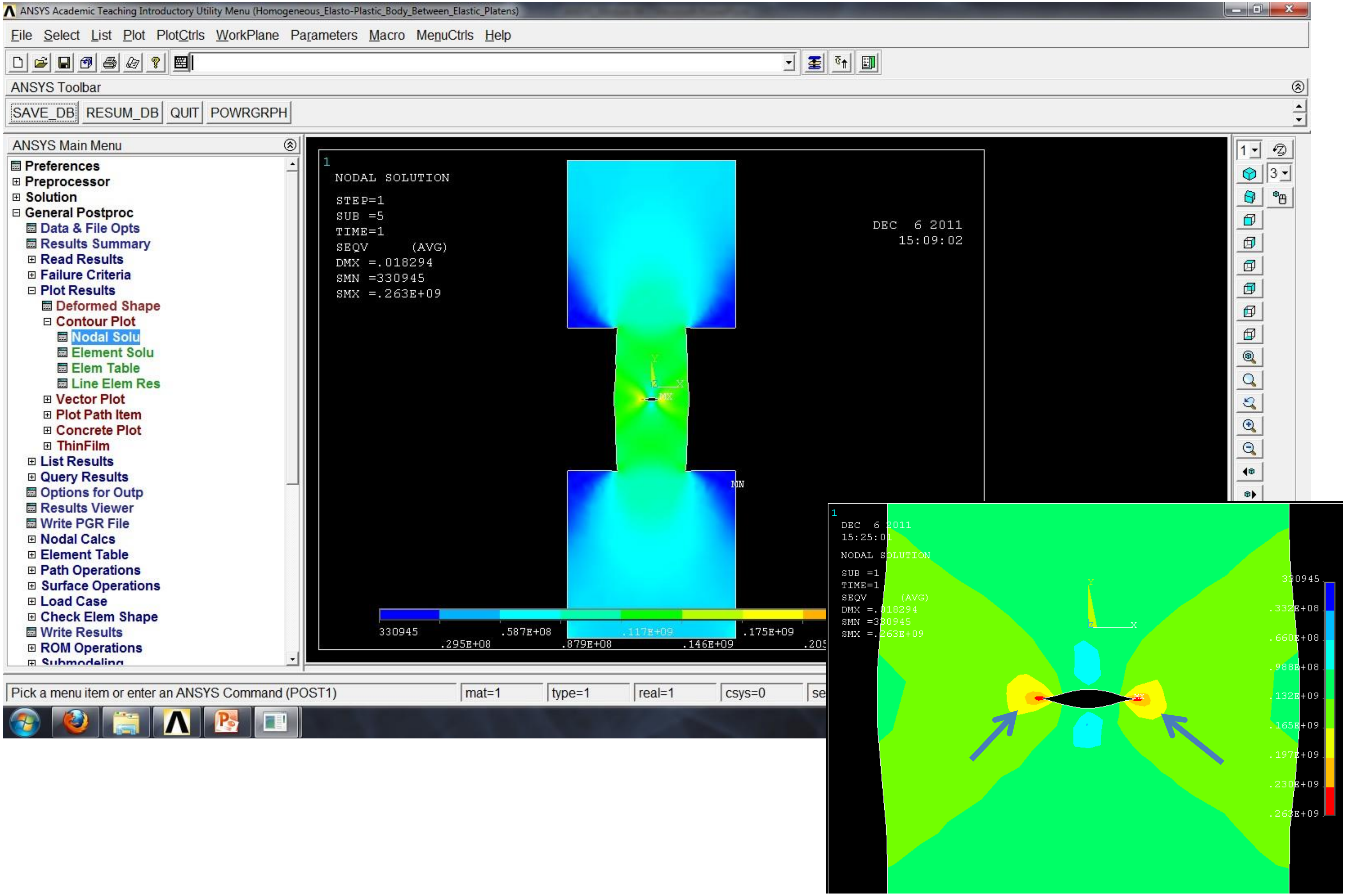
General PostProc → Plot Results → Contour Plot → Nodal Solu

Choose *Nodal Solutions* → *Stress* → *von Mises Stress (or others)*



- von Mises stress can be treated as the equivalent stress in the material → important quantity for assessing failure

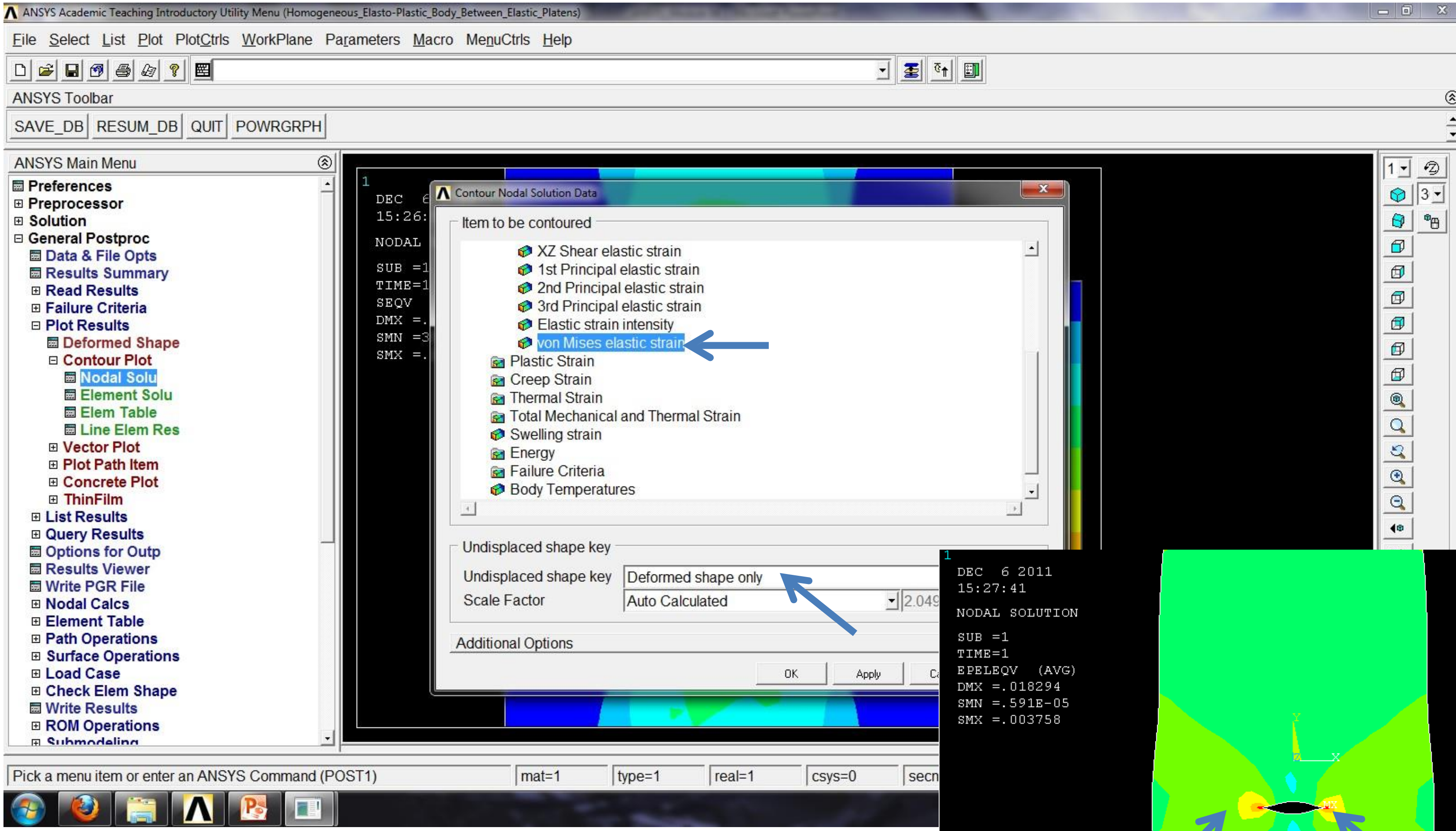
von Mises Stress: Axial Compression

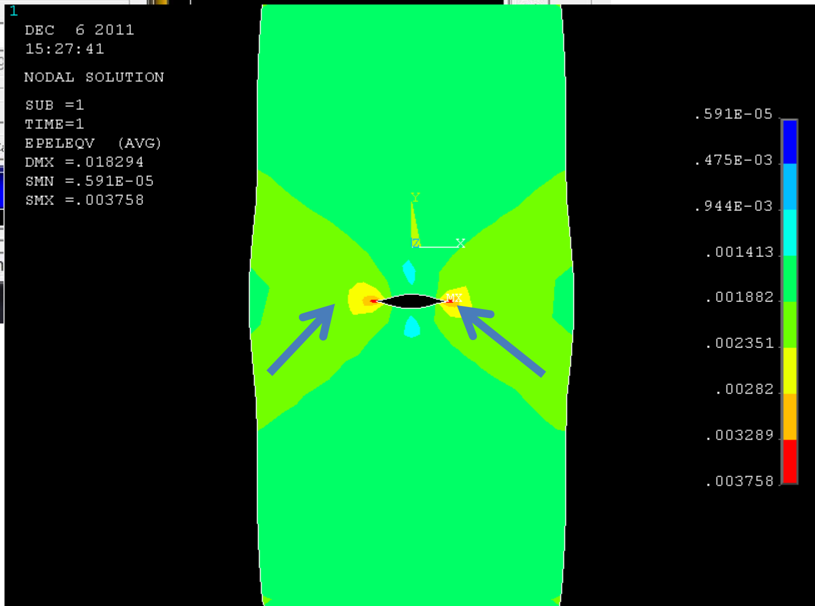


Plotting Results

General PostProc → Plot Results→ Contour Plot → Nodal Solu

Choose *Nodal Solutions* → Elastic Strain → *von Mises Strain*

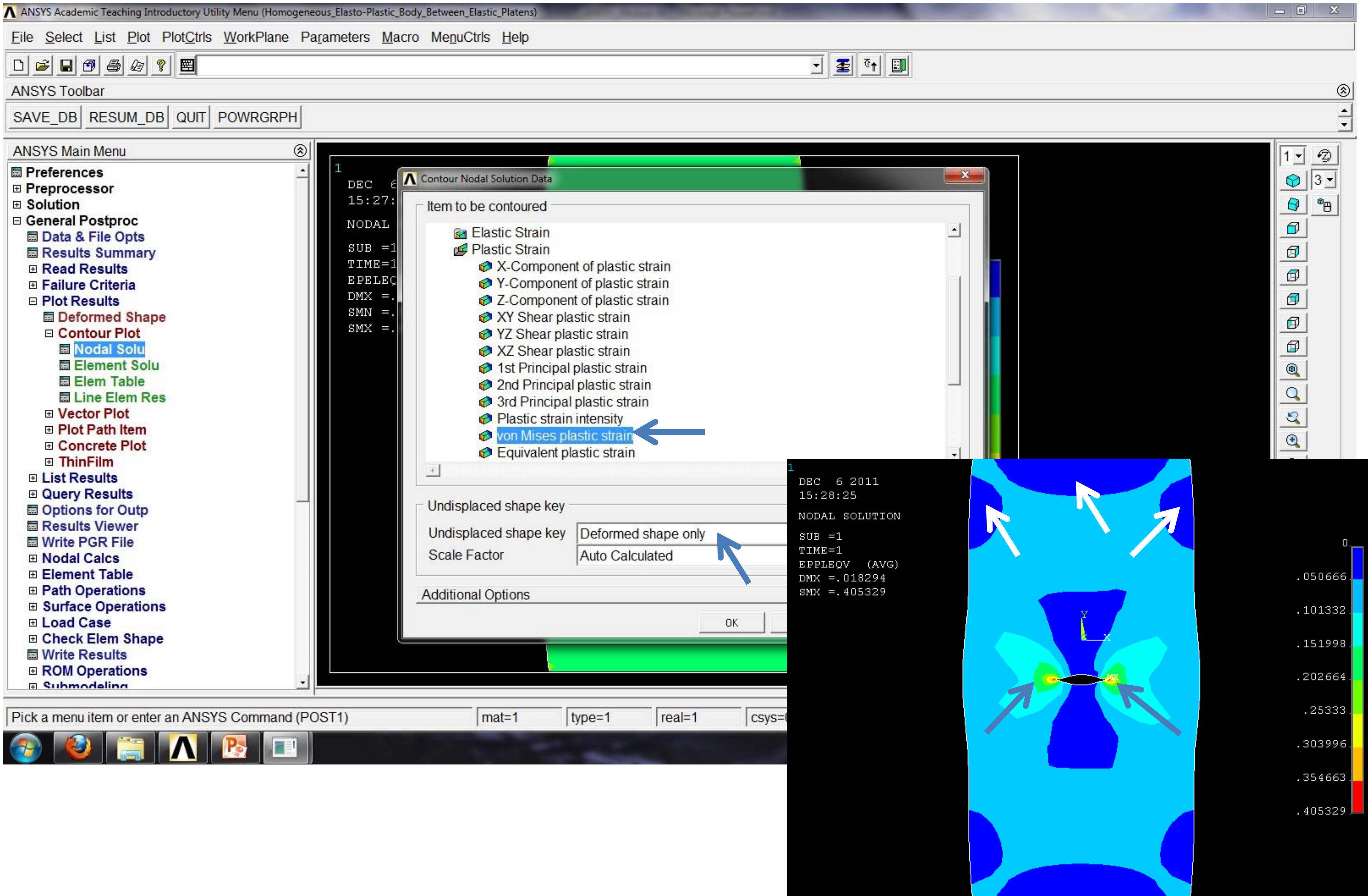




Plotting Results

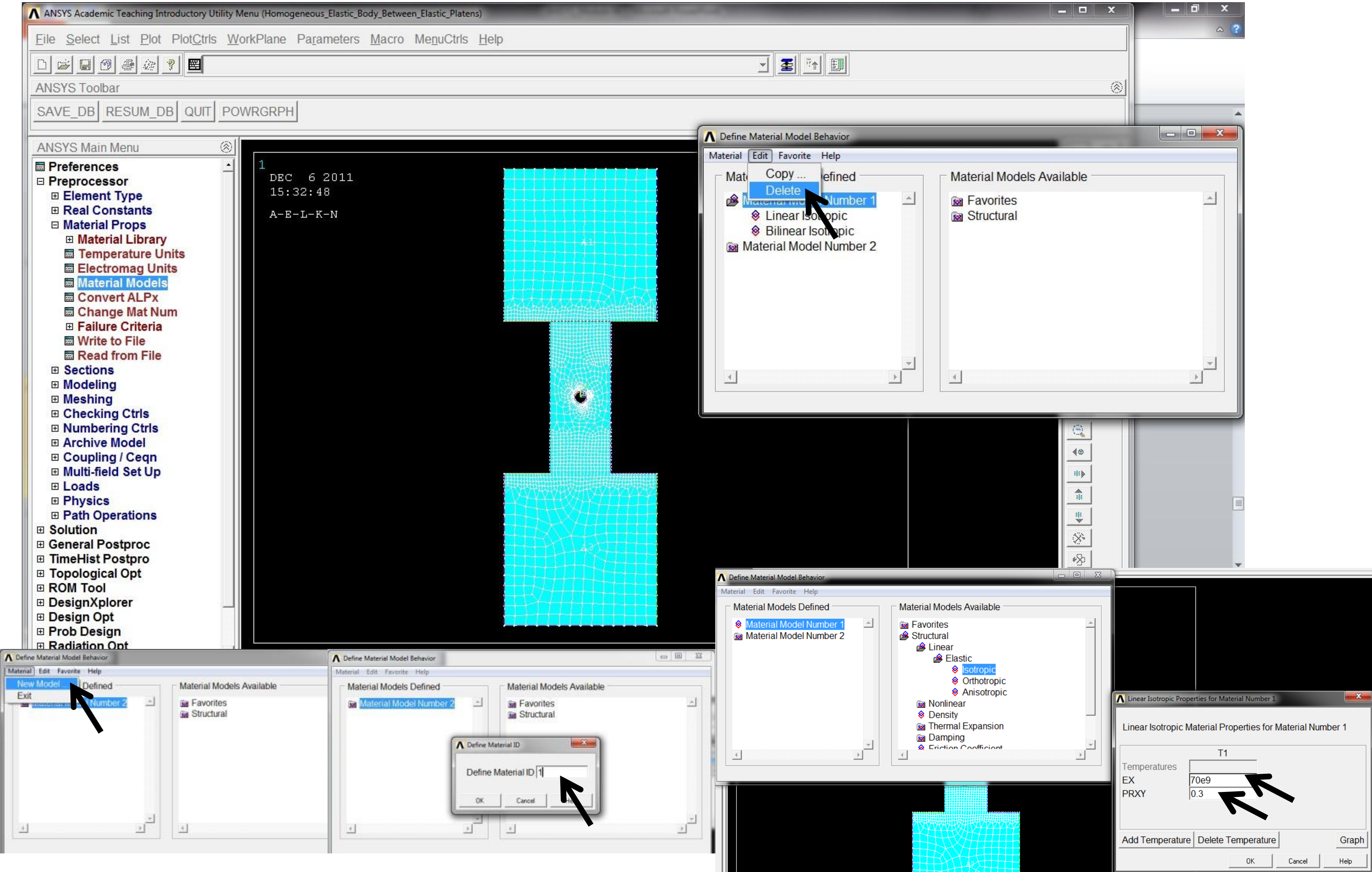
General PostProc → Plot Results→ Contour Plot → Nodal Solu

Choose *Nodal Solutions* → Plastic Strain → *von Mises Strain*



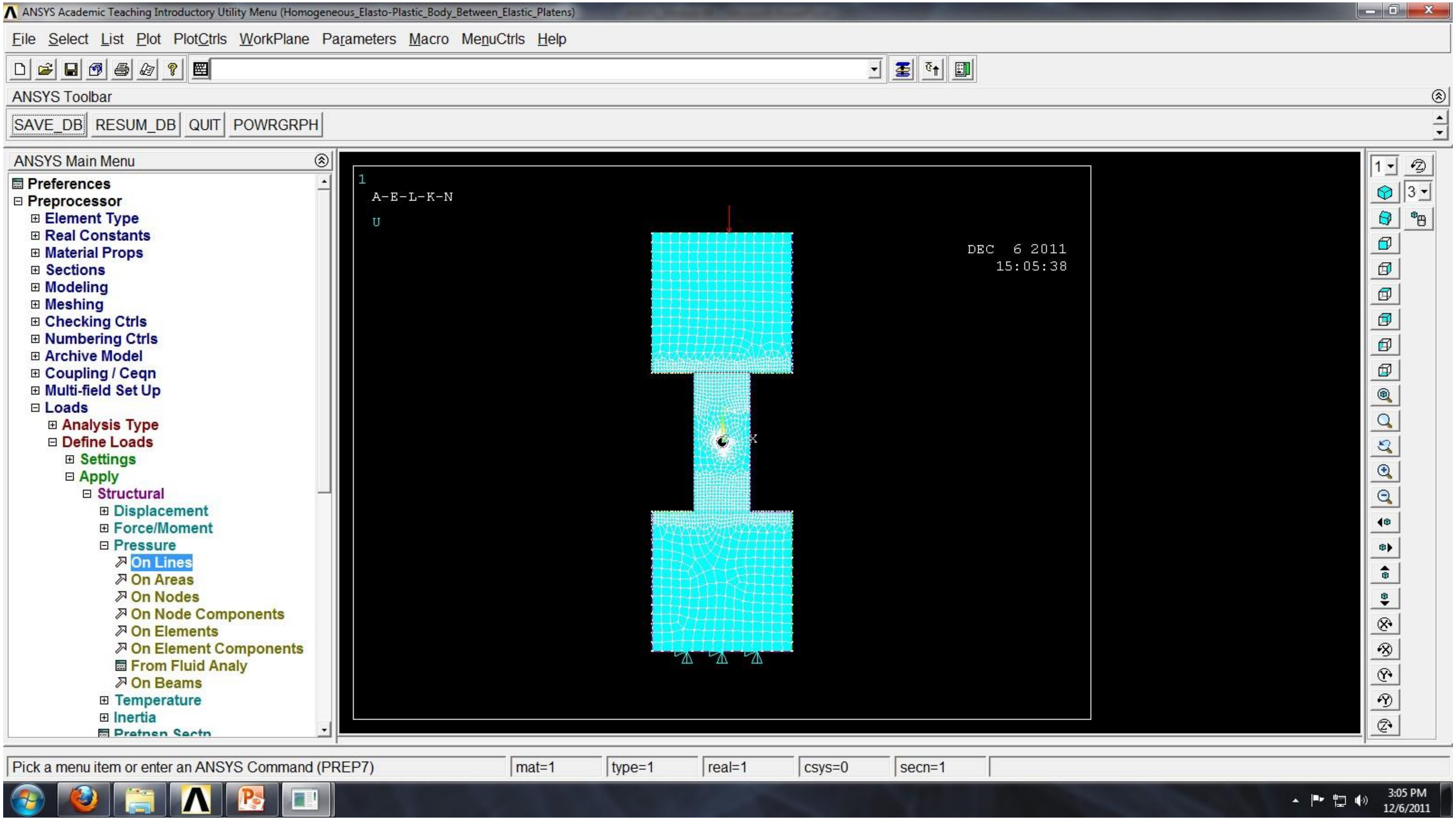
➢ Let's solve the earlier problem with no plasticity!

Delete earlier material model 1, add *new* material model 1 with *elasticity only*



Plots → Multi Plots

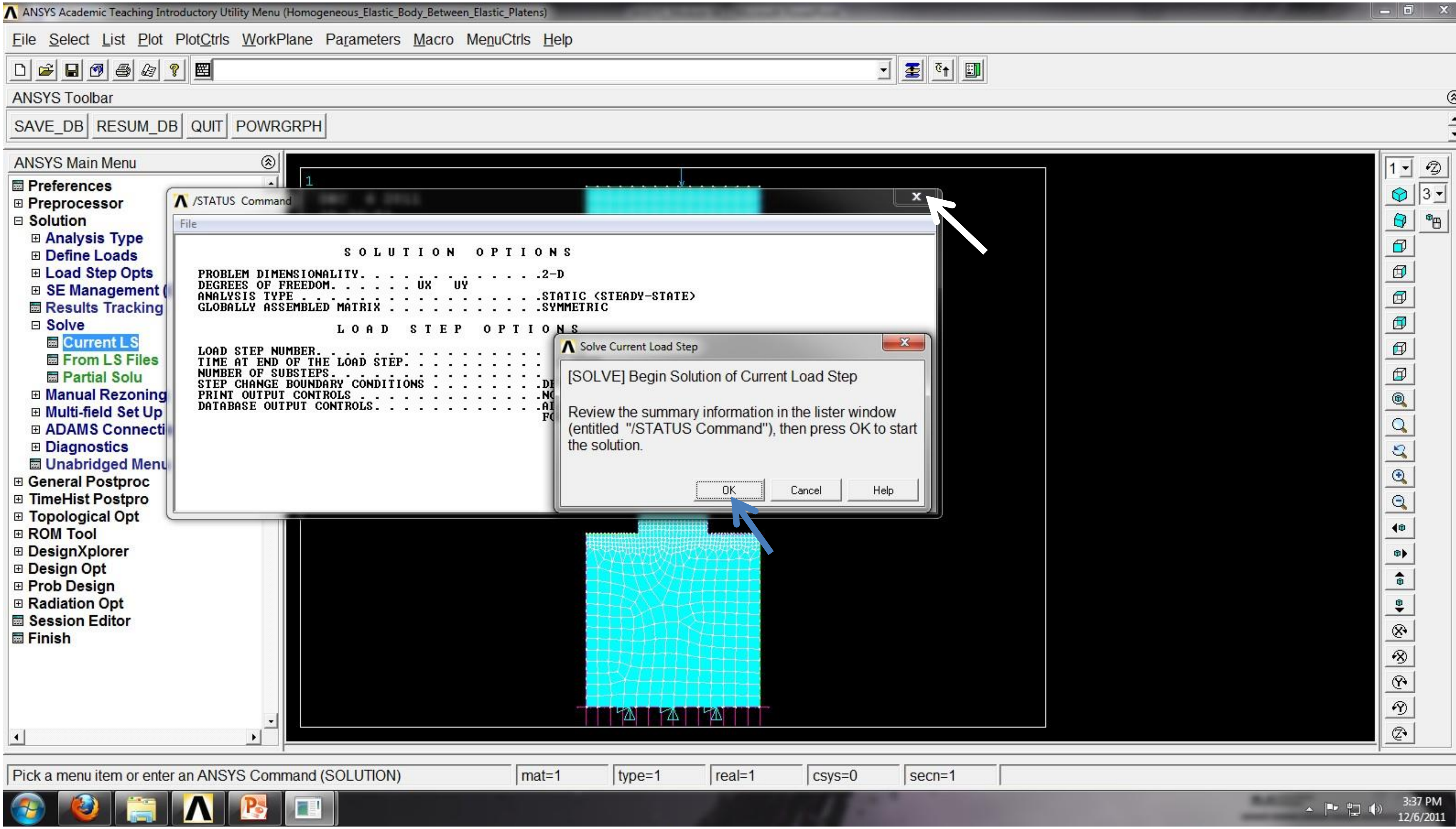
→ Check if all boundary conditions and loads are correct!



Solving the Model

Solution → Current LS

Choose *OK*



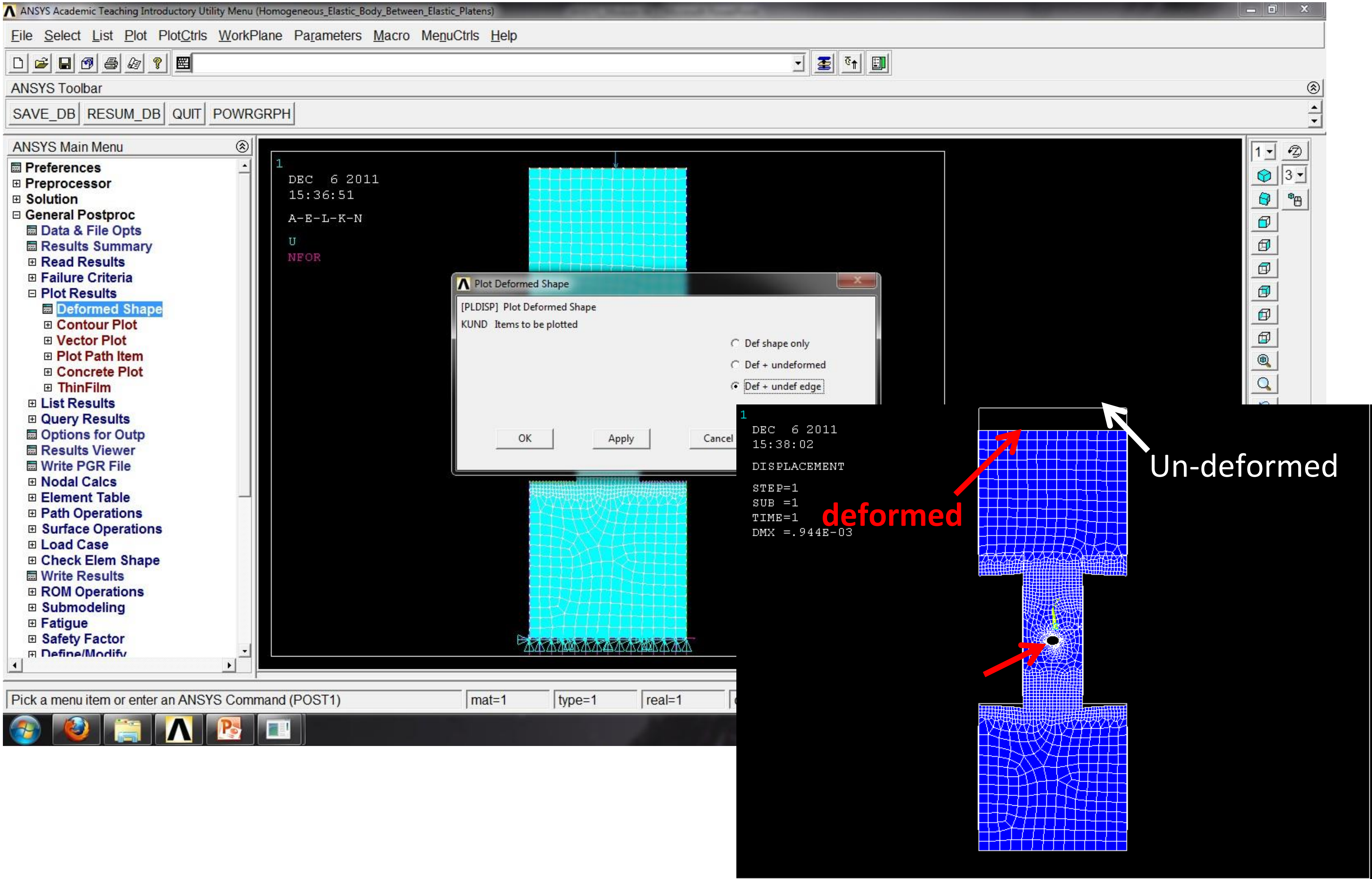
Pop Up stating → Solution Done!

very less time taken for solving this problem as compared to case when plasticity was involved

Plotting/Listing Results

General PostProc → Plot Results → Deformed Shape

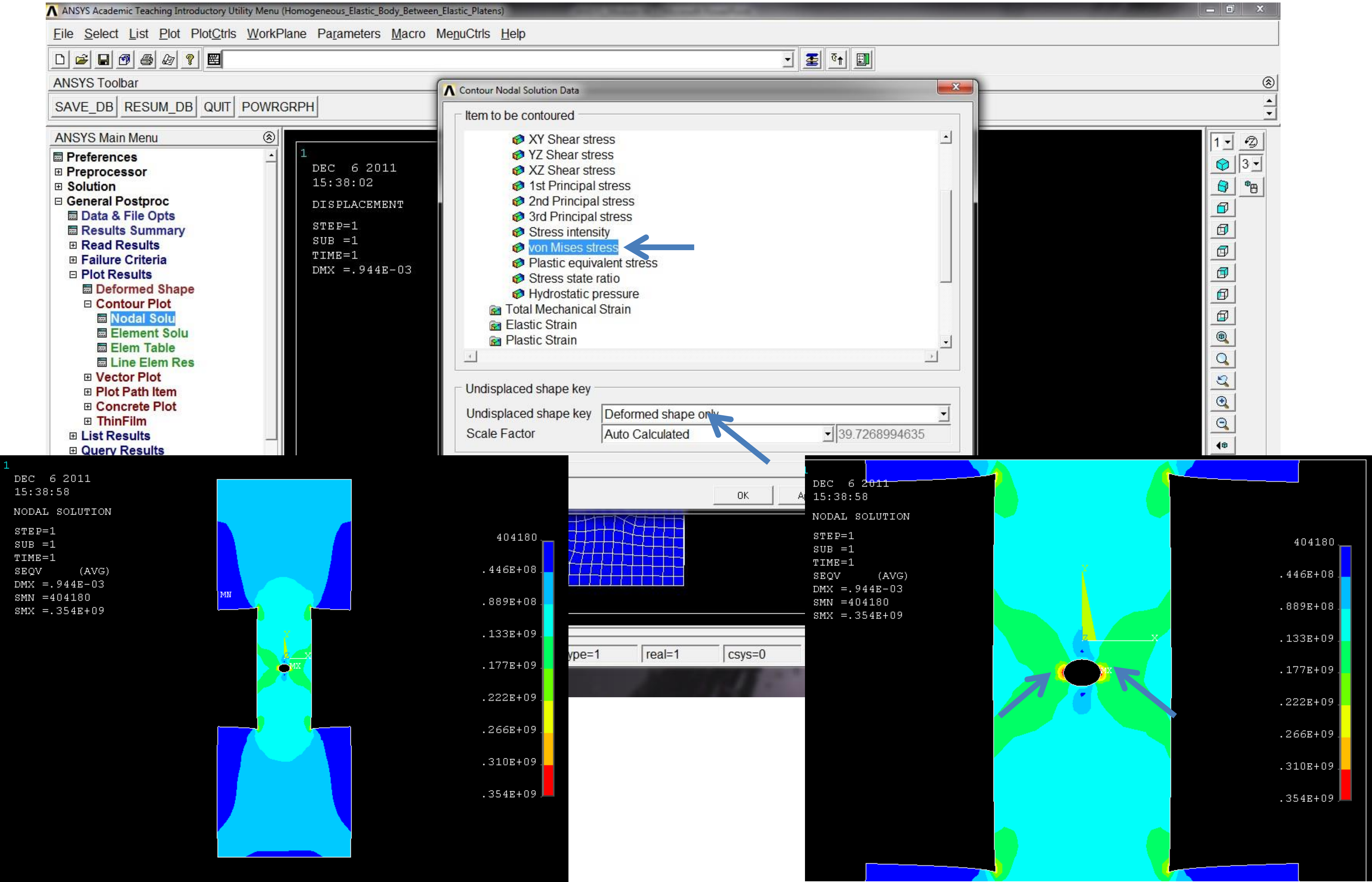
Choose *Def+ Undef Edge*



Plotting Results

General PostProc → Plot Results→ Contour Plot → Nodal Solu

Choose *Nodal Solutions* → *Stress* → *von Mises Stress (or others)*



Plotting Results

General PostProc → Plot Results→ Contour Plot → Nodal Solu

Choose *Nodal Solutions* → Elastic Strain → *von Mises Strain*