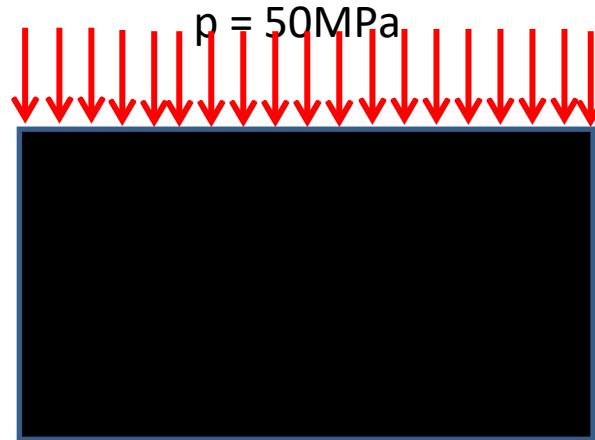


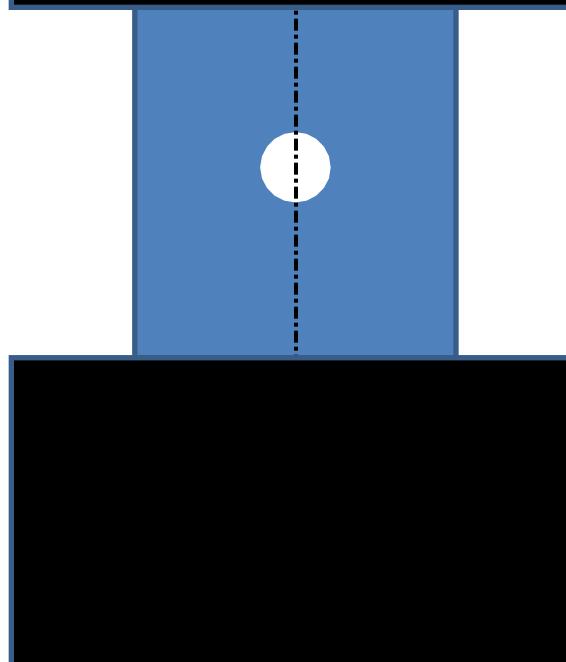
Problem Statement

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.



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

Steel platens: $0.25 \times 0.25 \text{mm}^2$



$$E = 70 \text{GPa} \text{ (Al alloy)}$$

$$\nu = 0.3$$

$$YS = 100 \text{ MPa}$$

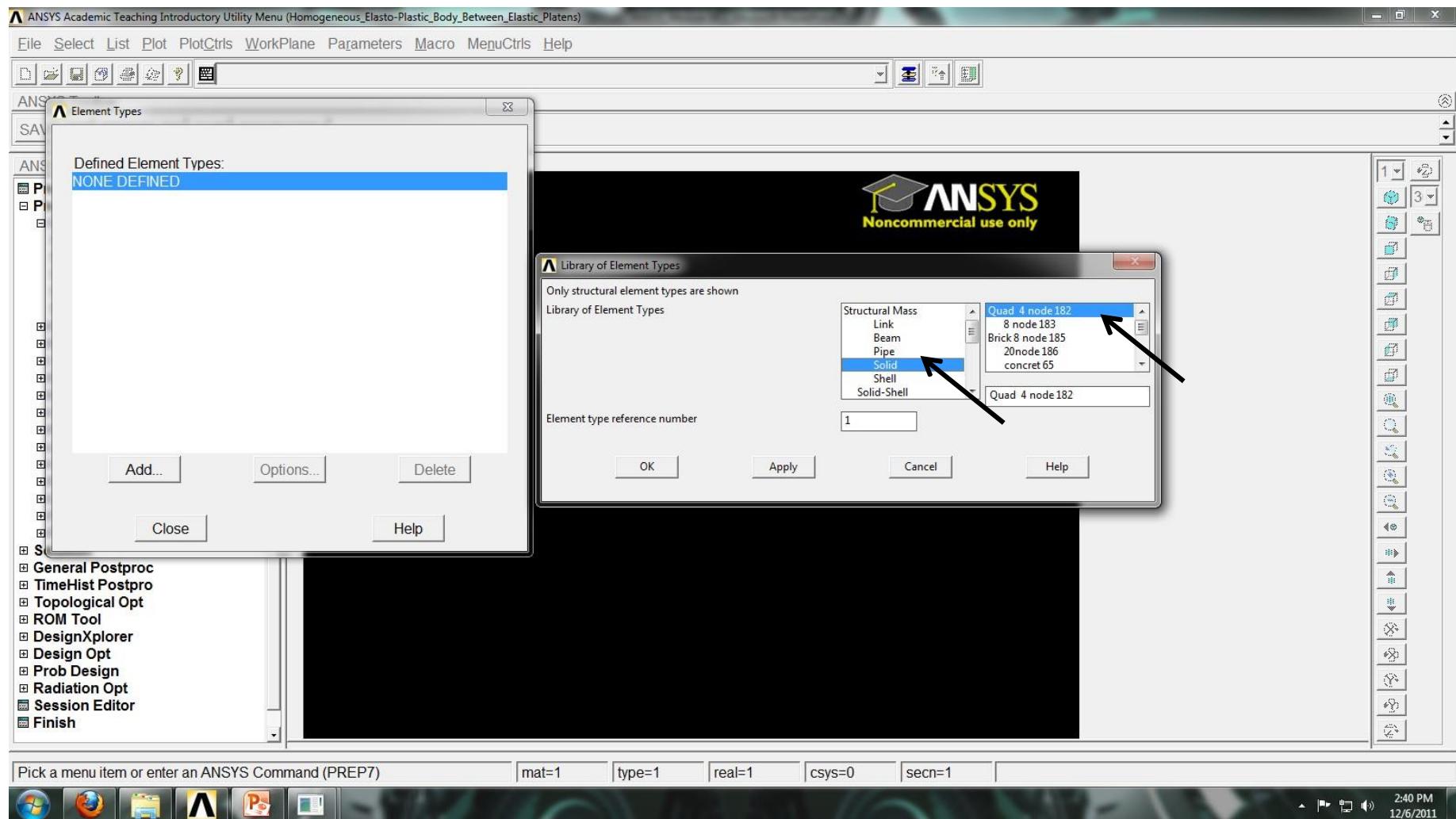
$$\text{Tangent Modulus} = 400 \text{MPa}$$

$$E = 300 \text{GPa} \text{ (Hardened Steel alloy)}$$

$$\nu = 0.3$$

Choose Element – Basic Element

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

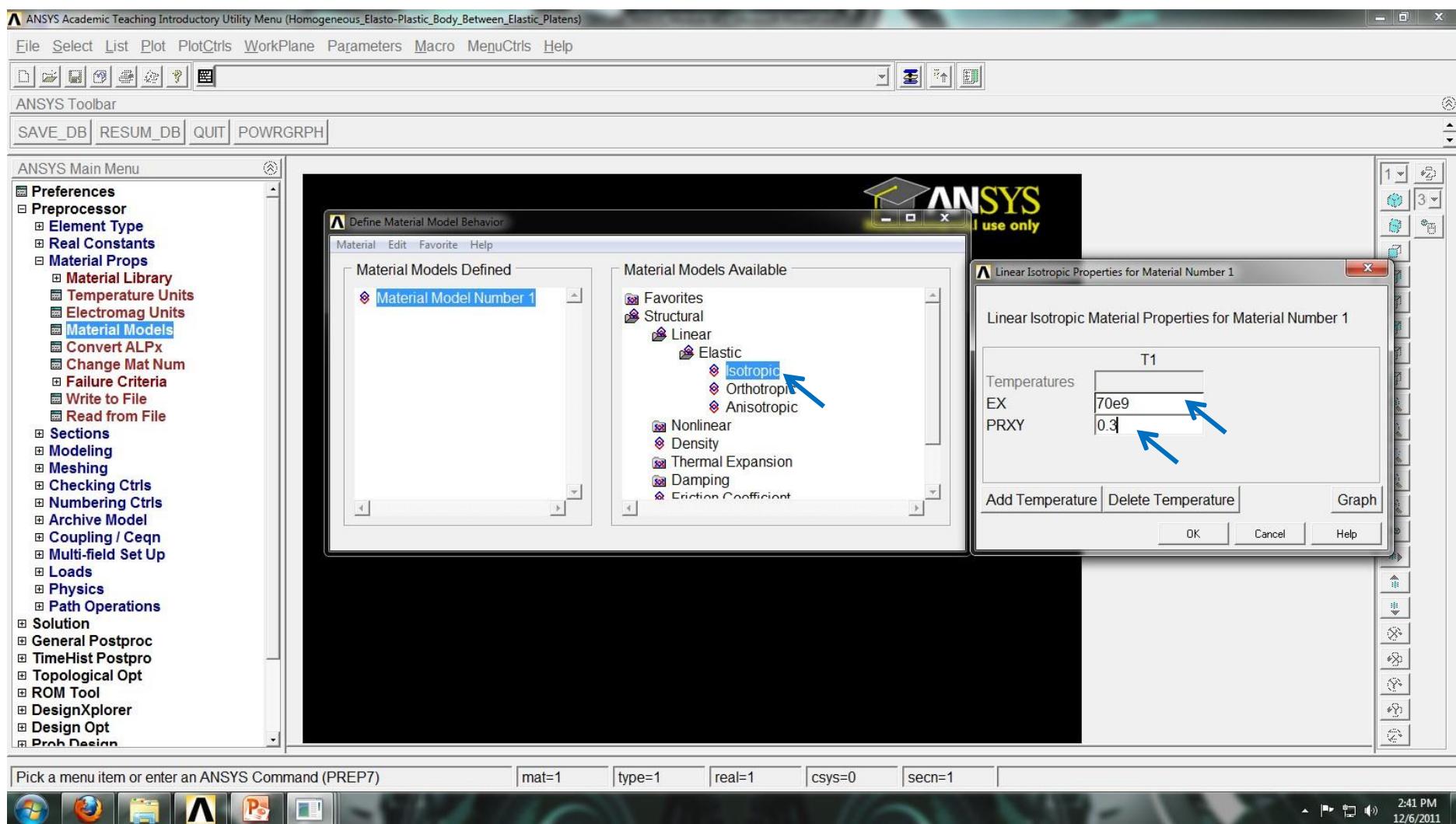


• Material Model

Structural → Linear → Elastic → Isotropic

$EX = 70\text{GPa}$

$\text{PRXY} = 0.3$



• Material Model

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

$E_x = 100 \text{ MPa}$

Tang Mod = 400MPa

The screenshot shows the ANSYS Academic Teaching Introductory Utility Menu interface. On the left, the main menu bar includes File, Select, List, Plot, PlotCtrls, WorkPlane, Parameters, Macro, MenuCtrls, and Help. Below the menu bar is the ANSYS Toolbar with various icons. The central part of the screen displays the "Define Material Model Behavior" dialog box. Under "Material Models Available", the "Bilinear" option under "Inelastic" is selected, highlighted with a blue arrow. To the right of this dialog is a sub-dialog titled "Bilinear Isotropic Hardening for Material Number 1". This sub-dialog contains fields for Temperature (T1), Yield Stss (1E+008), and Tang Mod (4E+008). Arrows point from the values in the sub-dialog back to their respective entries in the main dialog. At the bottom right, there is a "Graph" button and a preview window showing a stress-strain curve for the Bilinear model.

ANSYS Academic Teaching Introductory Utility Menu (Homogeneous_Elasto-Plastic_Body_Between_Elastic_Platten)

File Select List Plot PlotCtrls WorkPlane Parameters Macro MenuCtrls Help

ANSYS Toolbar

SAVE_DB RESUM_DB QUIT POK

ANSYS Main Menu

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
- Multi-field Set Up
- Loads
- Physics
- Path Operations

Solution

General Postproc

TimeHist Postpro

Topological Opt

ROM Tool

DesignXplorer

Design Opt

Prob Design

Pick a menu item or enter an ANSYS Command (PREP7) mat=1 type=1 real=1 csys=0 secn=1

ANSYS Main Menu

Define Material Model Behavior

Material Models Defined

Material Model Number 1

Material Models Available

- Favorites
- Structural
- Linear
 - Elastic
 - Isotropic
 - Orthotropic
 - Anisotropic
- Nonlinear
 - Elastic
 - Inelastic
 - Rate Independent
 - Isotropic Hardening Plasticity
 - Mises Plasticity**
 - Bilinear**
 - Multilinear
 - Nonlinear
 - Hill Plasticity
 - Generalized Anisotropic Hill Potential
 - Kinematic Hardening Plasticity
 - Combined Kinematic and L
 - Rate Dependent
 - Non-metal Plasticity
 - Cast-Iron

T1

Temperature 0

Yield Stss 1E+008

Tang Mod 4E+008

Add Temperature Delete Temperature Add Row Delete Row Graph OK Cancel Help

Table Data

T1=0.00

BISO Table Preview

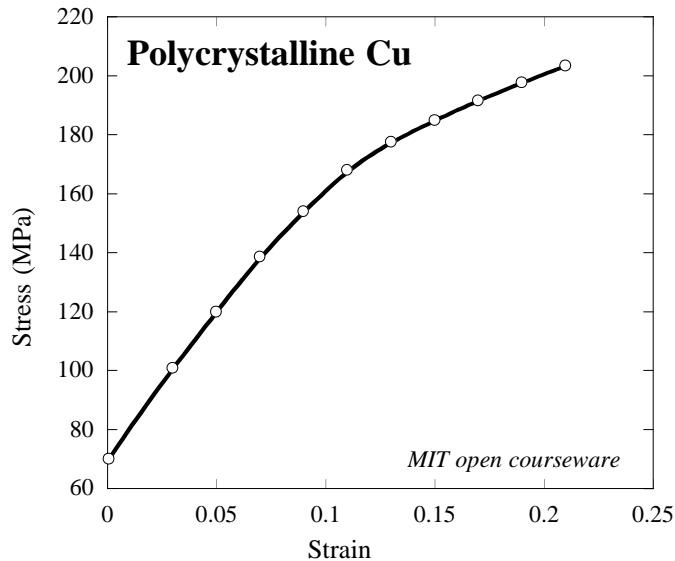
DEC 6 2011 14:43:16

SIG (x10⁻⁵)

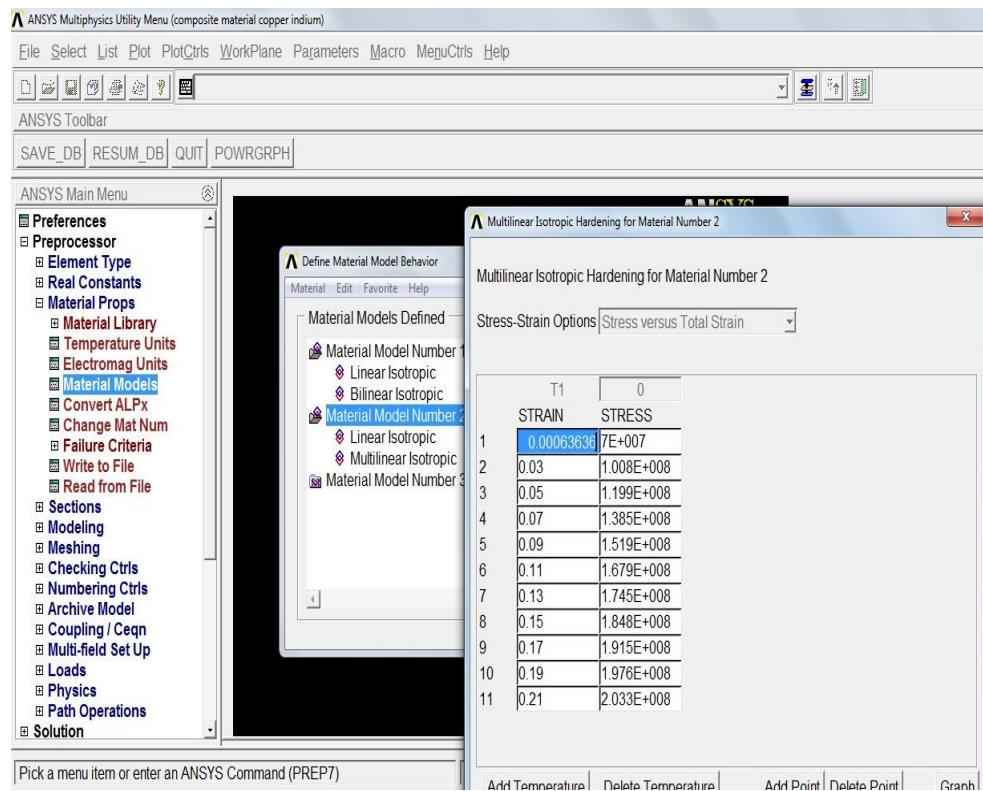
EPS (x10⁻³)

EPS (x10 ⁻³)	SIG (x10 ⁻⁵)
0	0
1.5	1000
4.5	1000

If Other options of plasticity are chosen, then:



If Multilinear option is chosen, then:

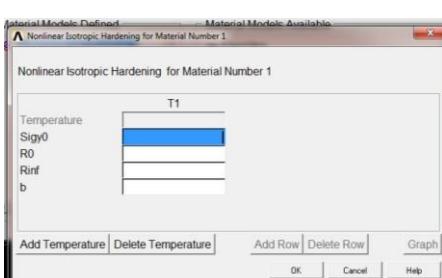


If Non-linear option is chosen, then:

- 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:
- $$C = \frac{\sigma}{\sigma_{YS}} + \frac{R}{R_\infty} \varepsilon_{pl} + \frac{R_\infty}{R_{YS}}$$
- $$m = \frac{R_\infty}{C}$$
- $$n = b$$

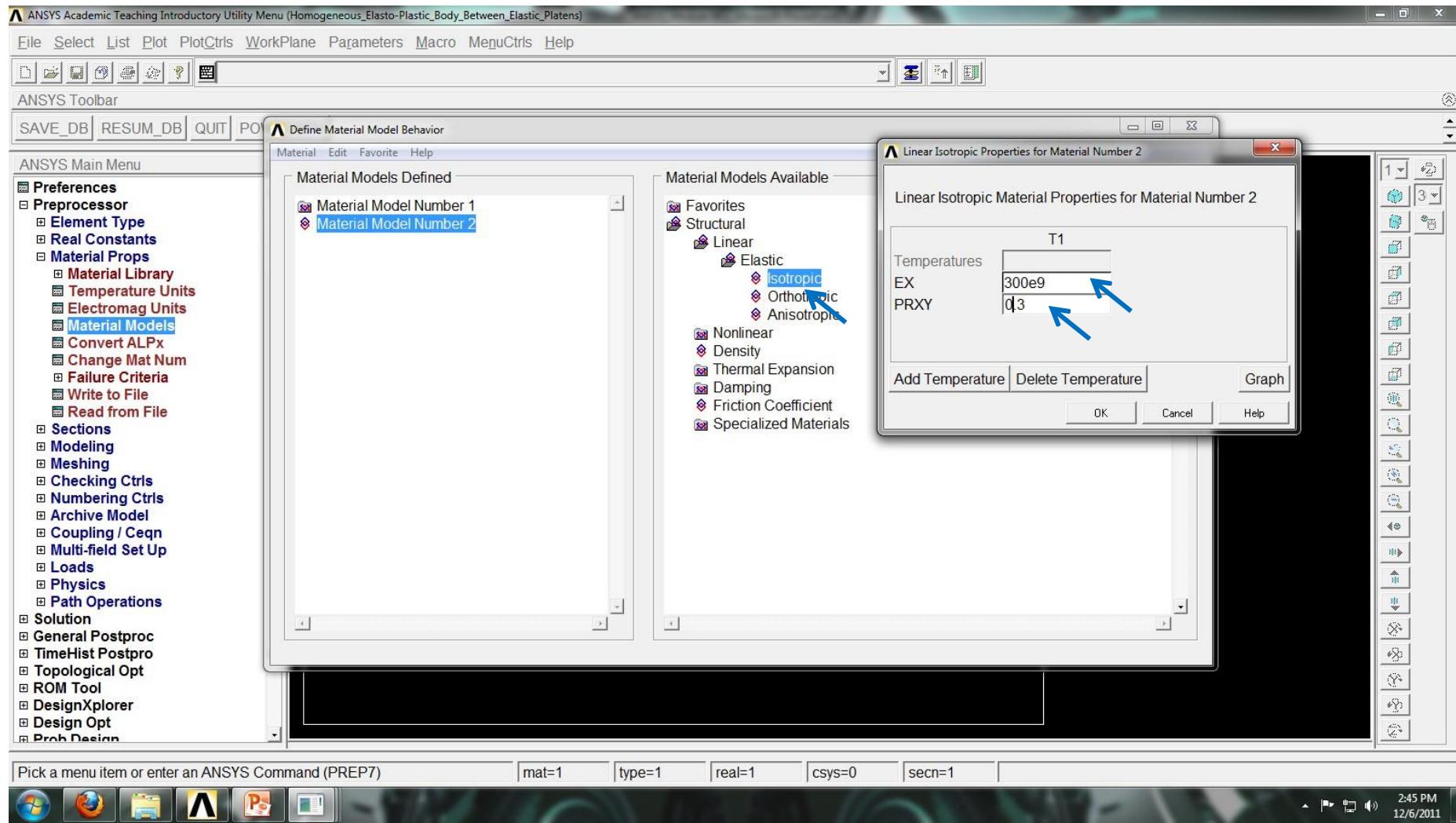


- Material Model

File → New Model

Structural → Linear → Elastic → Isotropic

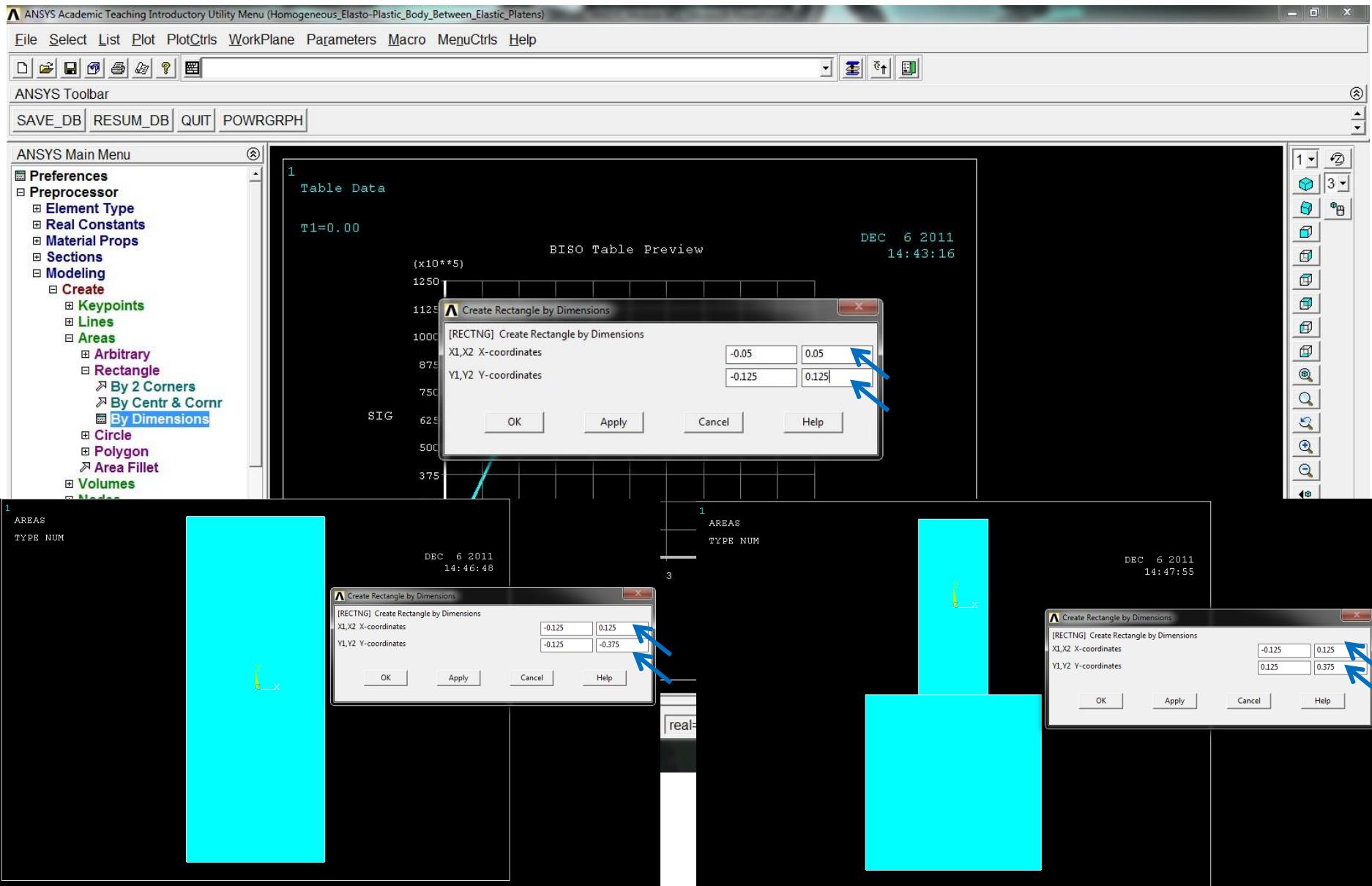
$E_x = 300\text{GPa}$ $\text{PRXY} = 0.3$



Create Geometry (Modeling)

Create Rectangles

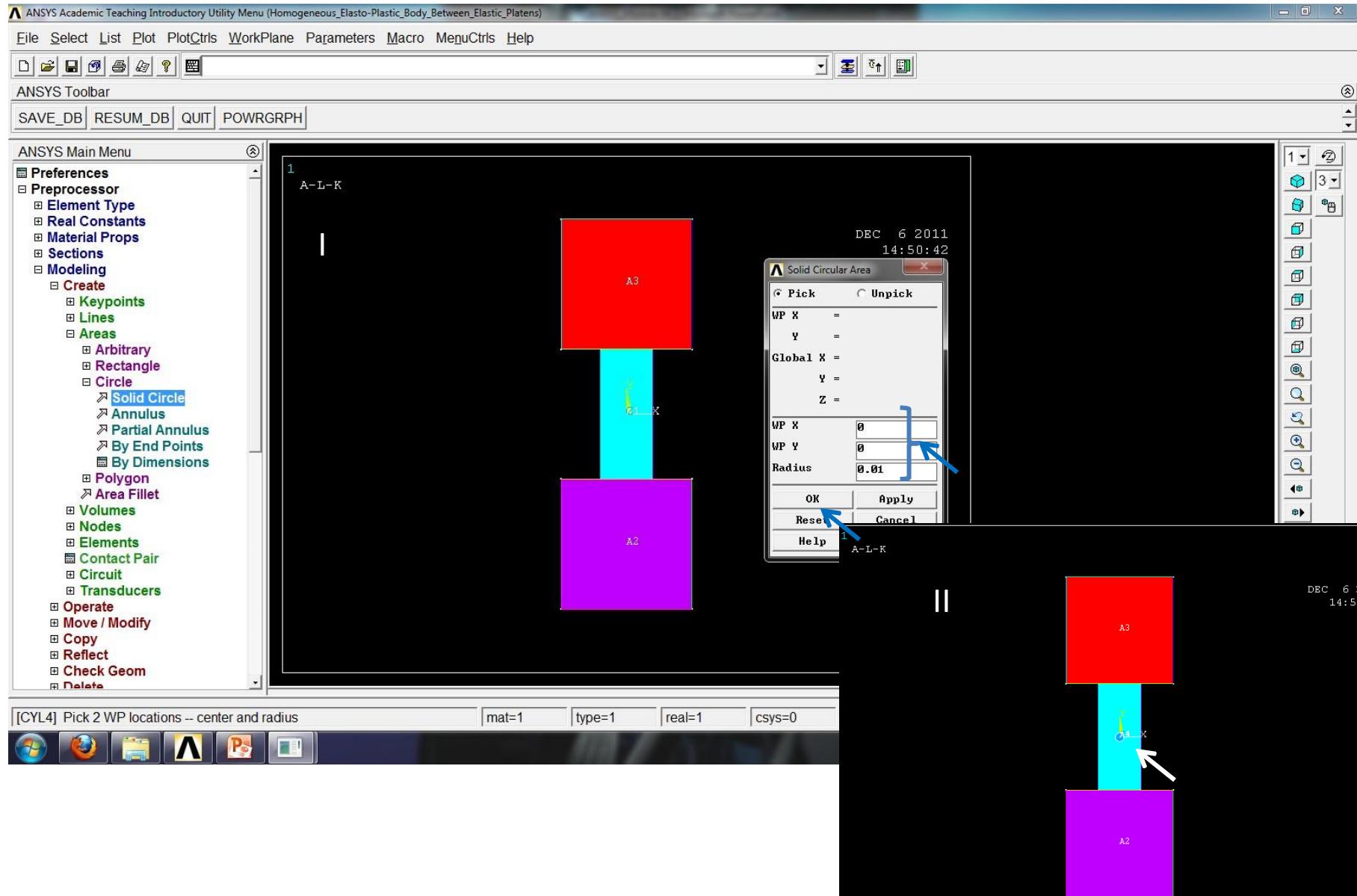
Preprocessor → Create → Areas → Rectangle → By Dimension



Create Geometry (Modeling)

Create Circular Area for Making Hole

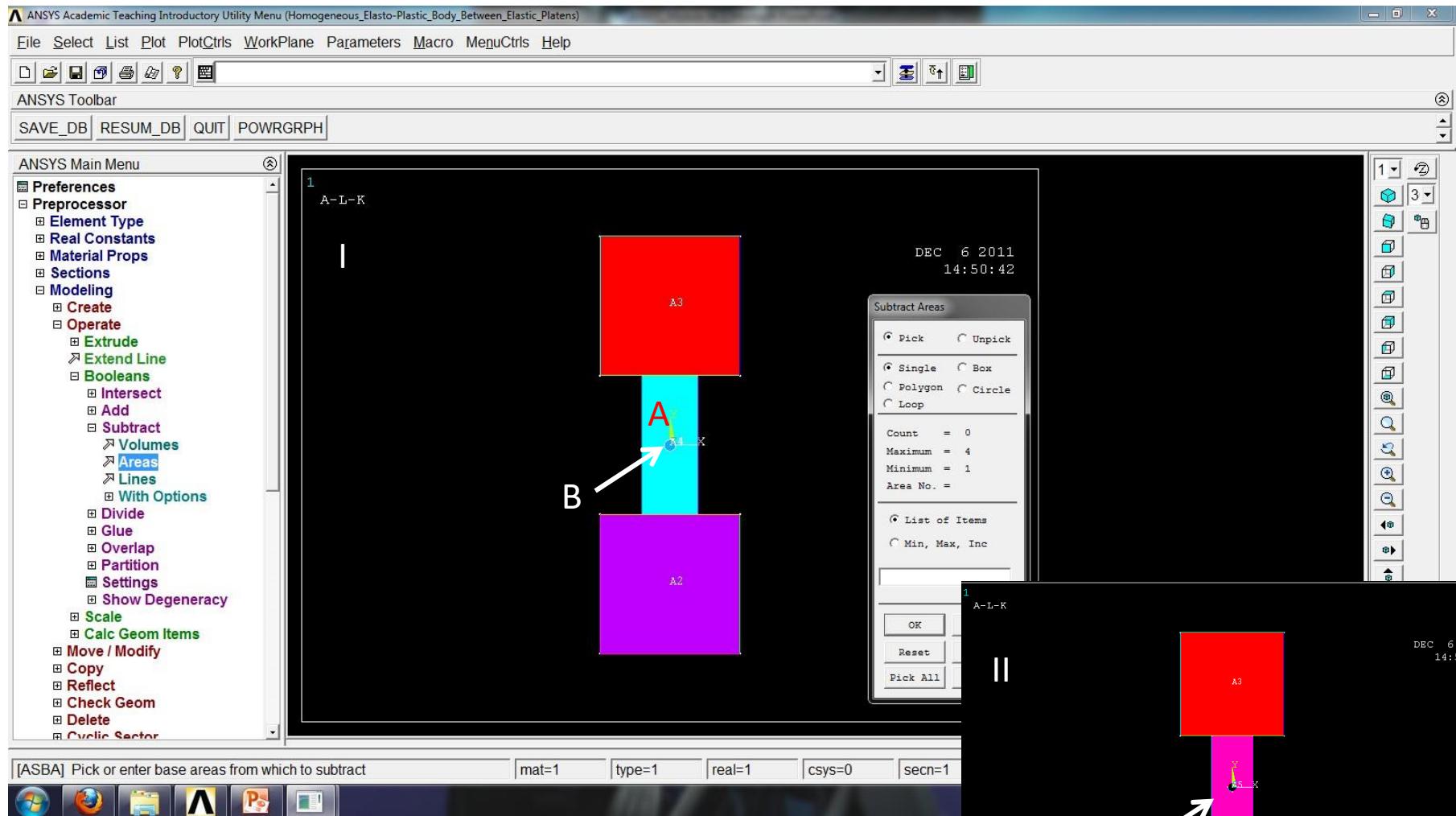
Preprocessor → Create → Areas → Circle → Solid Circle



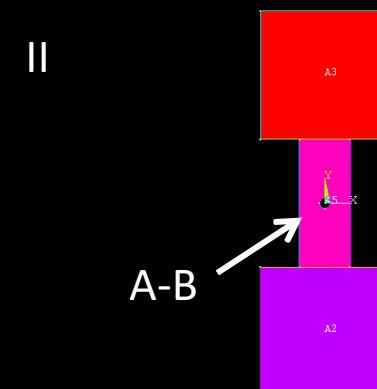
Create Geometry (Modeling)

Creating a hole in the Al alloy sample

Preprocessor → Modeling → Operate → Booleans → Subtract → Area

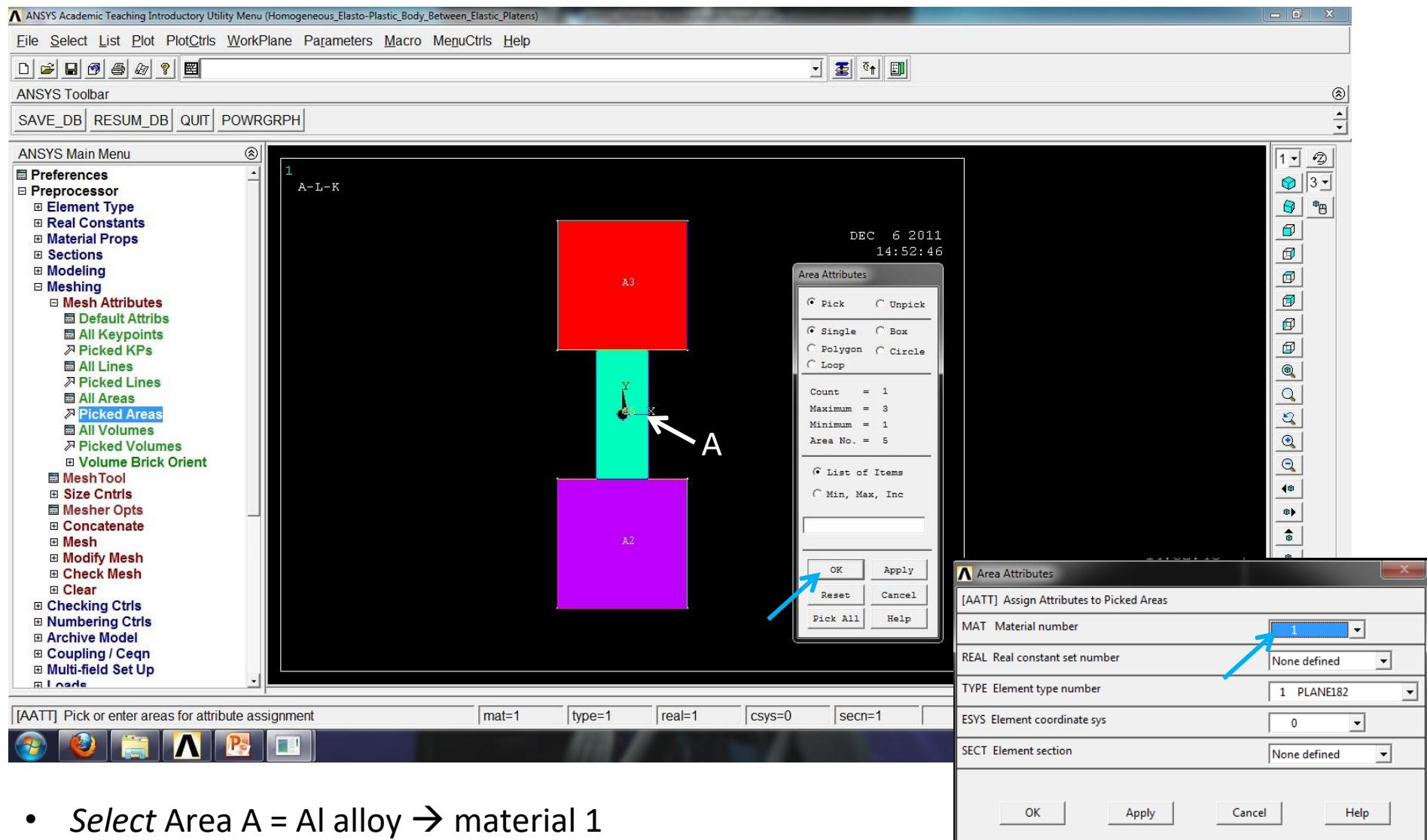


- A-B: Choose A first, click OK , Choose B, and then click OK



Assigning Different Areas to Different Materials Attributes

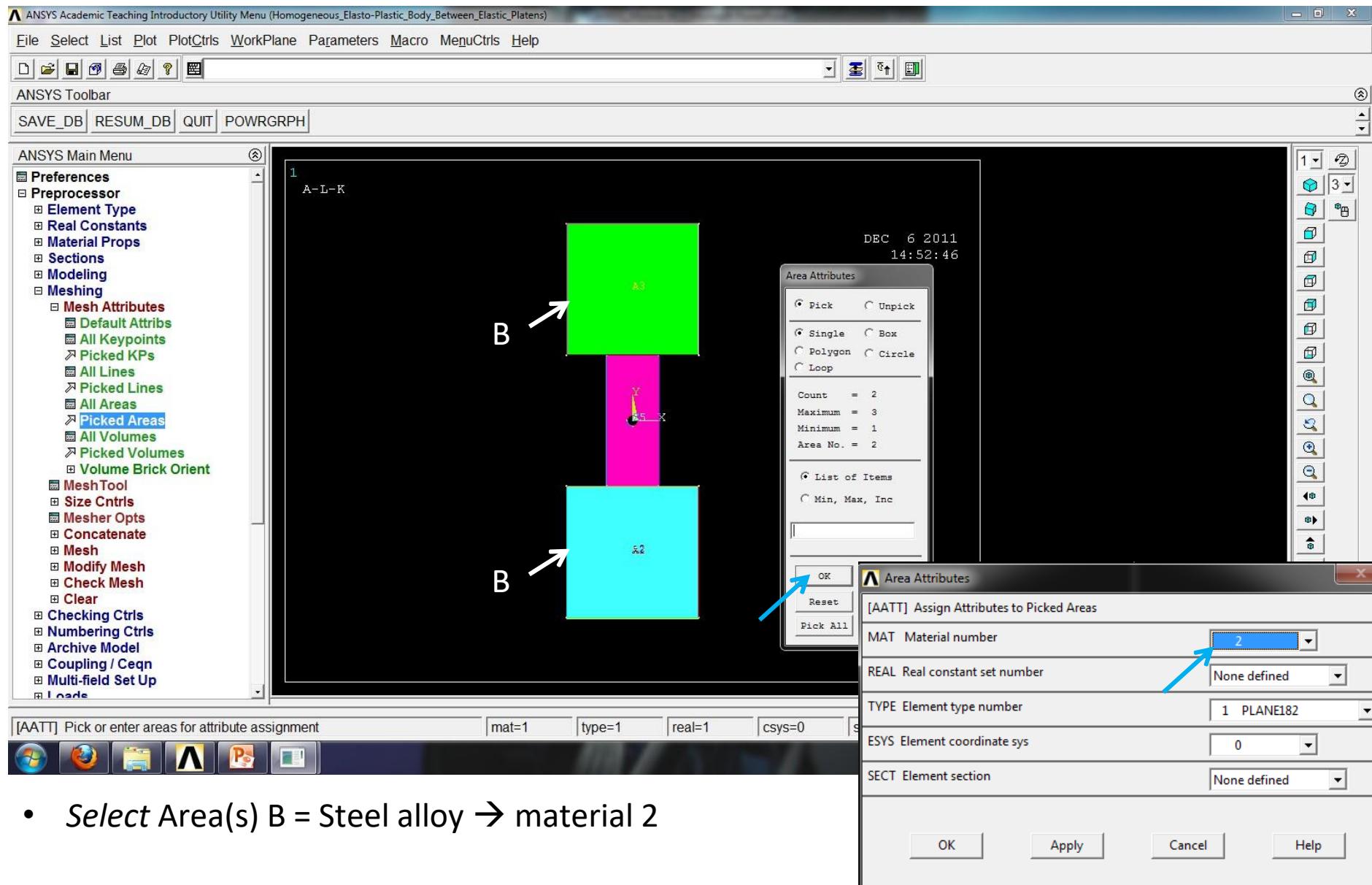
Preprocessor → Meshing → Mesh Attributes → Picked Areas



- Select Area A = Al alloy → material 1

Assigning Different Areas to Different Materials Attributes

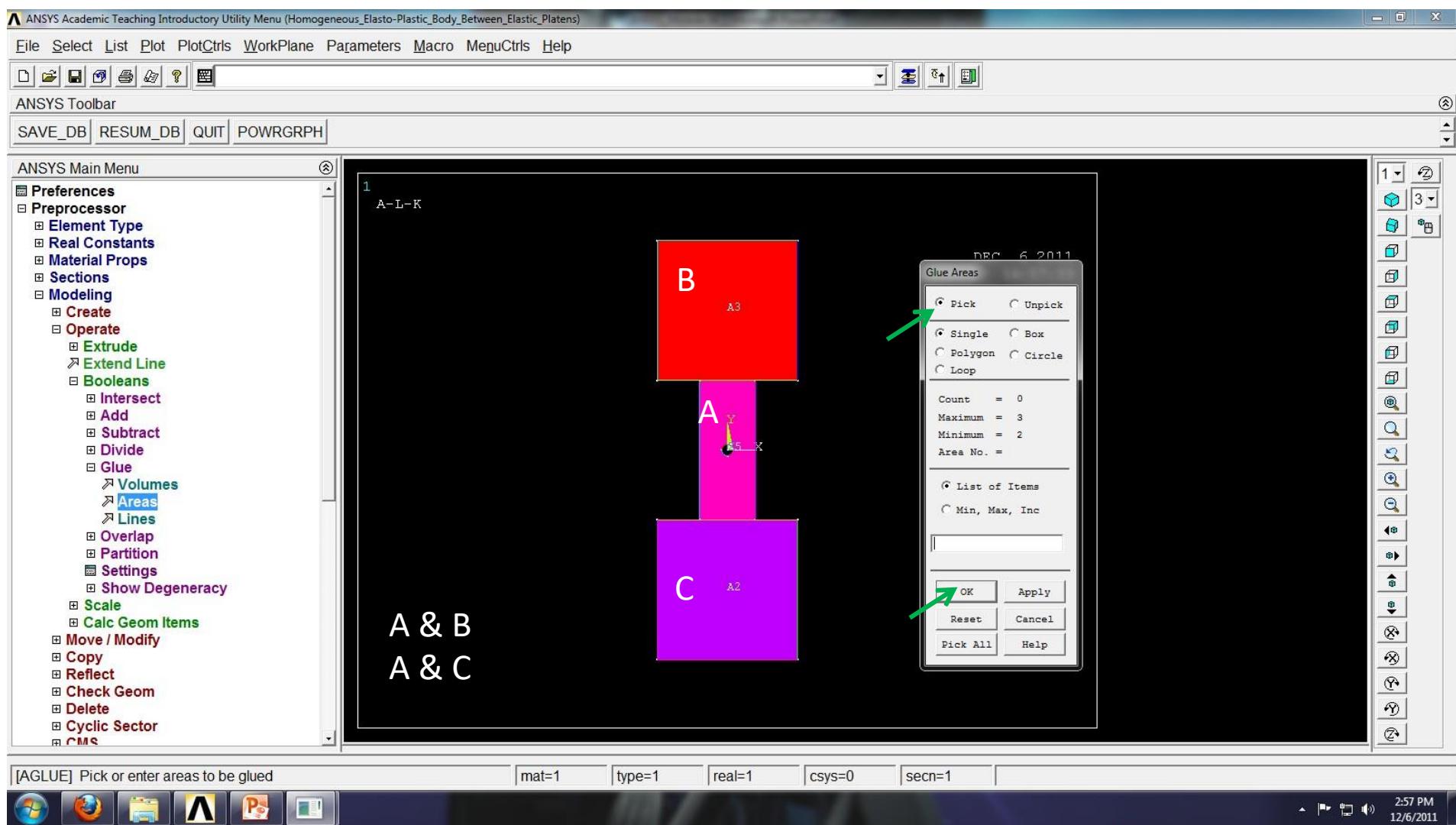
Preprocessor → Meshing → Mesh Attributes → Picked Areas



Glue Areas (Al alloy and Steel platens)

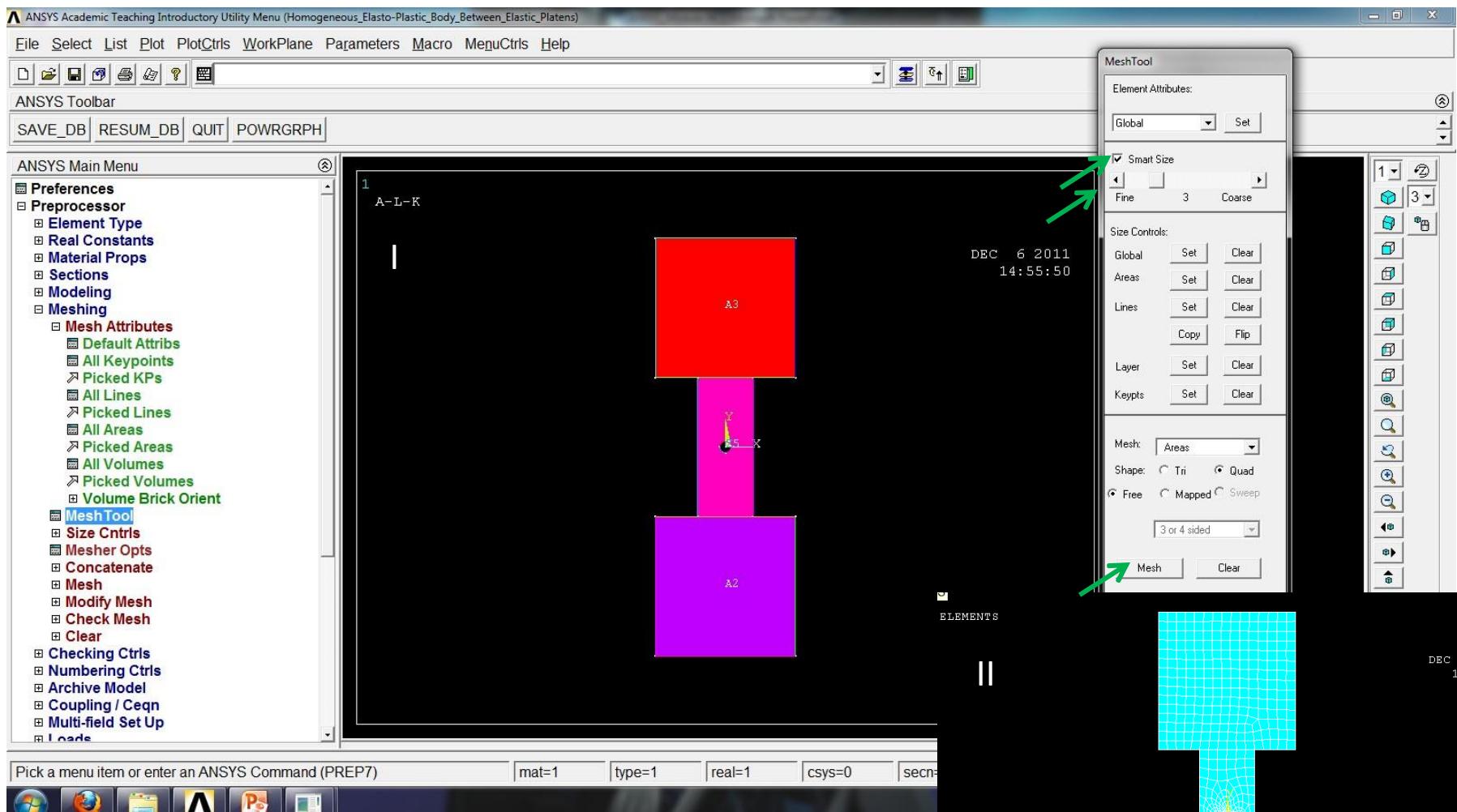
Preprocessor → Modeling → Operate → Glue → Areas

Glue a pair of areas at a time together!



Meshering (Initial)

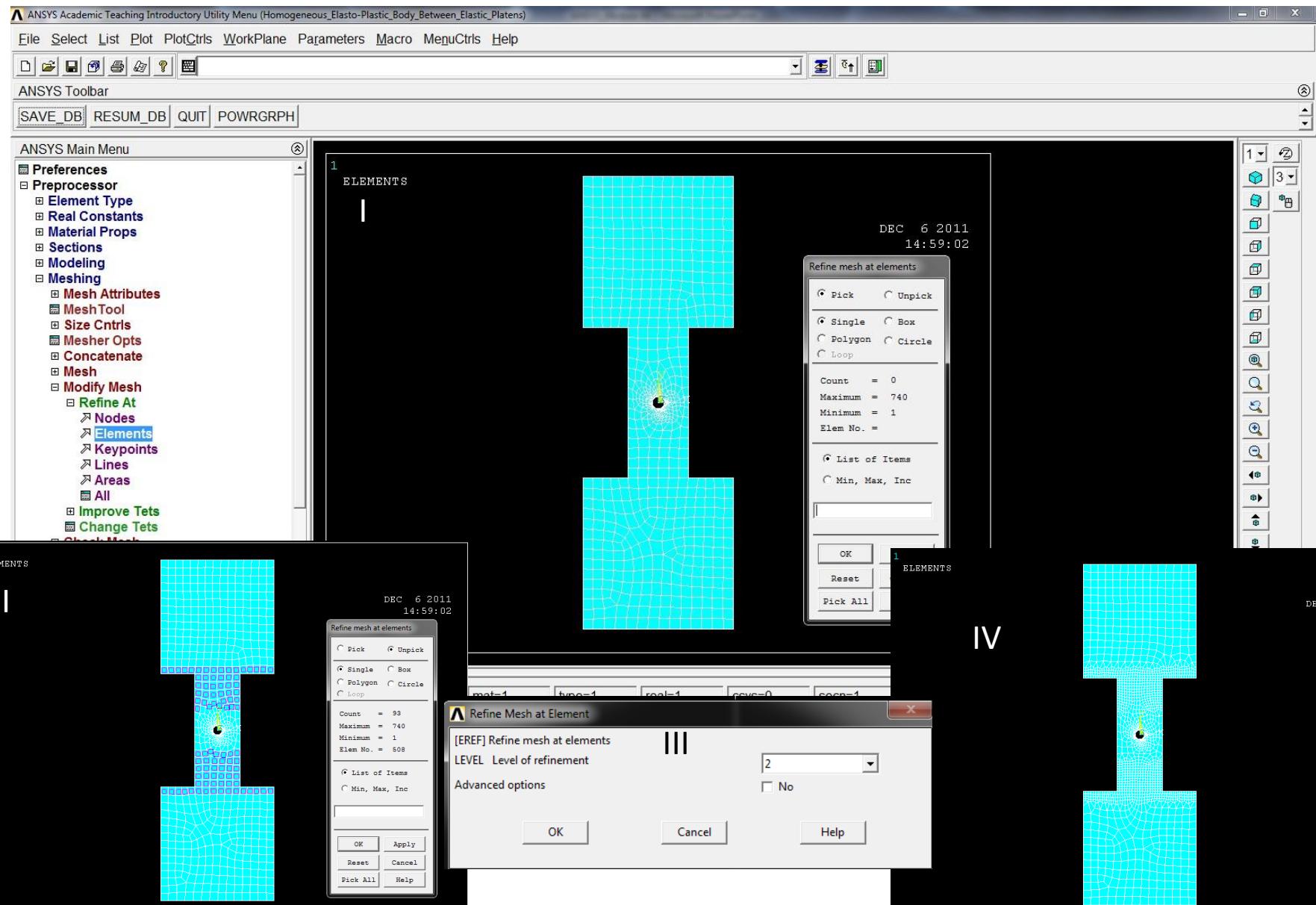
Preprocessor → Meshing → Mesh Tool



Meshering → Refining meshes near the platens/sample interface

Preprocessor → Meshing → Modify Mesh → Refine At → Elements

Pick elements near platens/sample interface → Select “2”

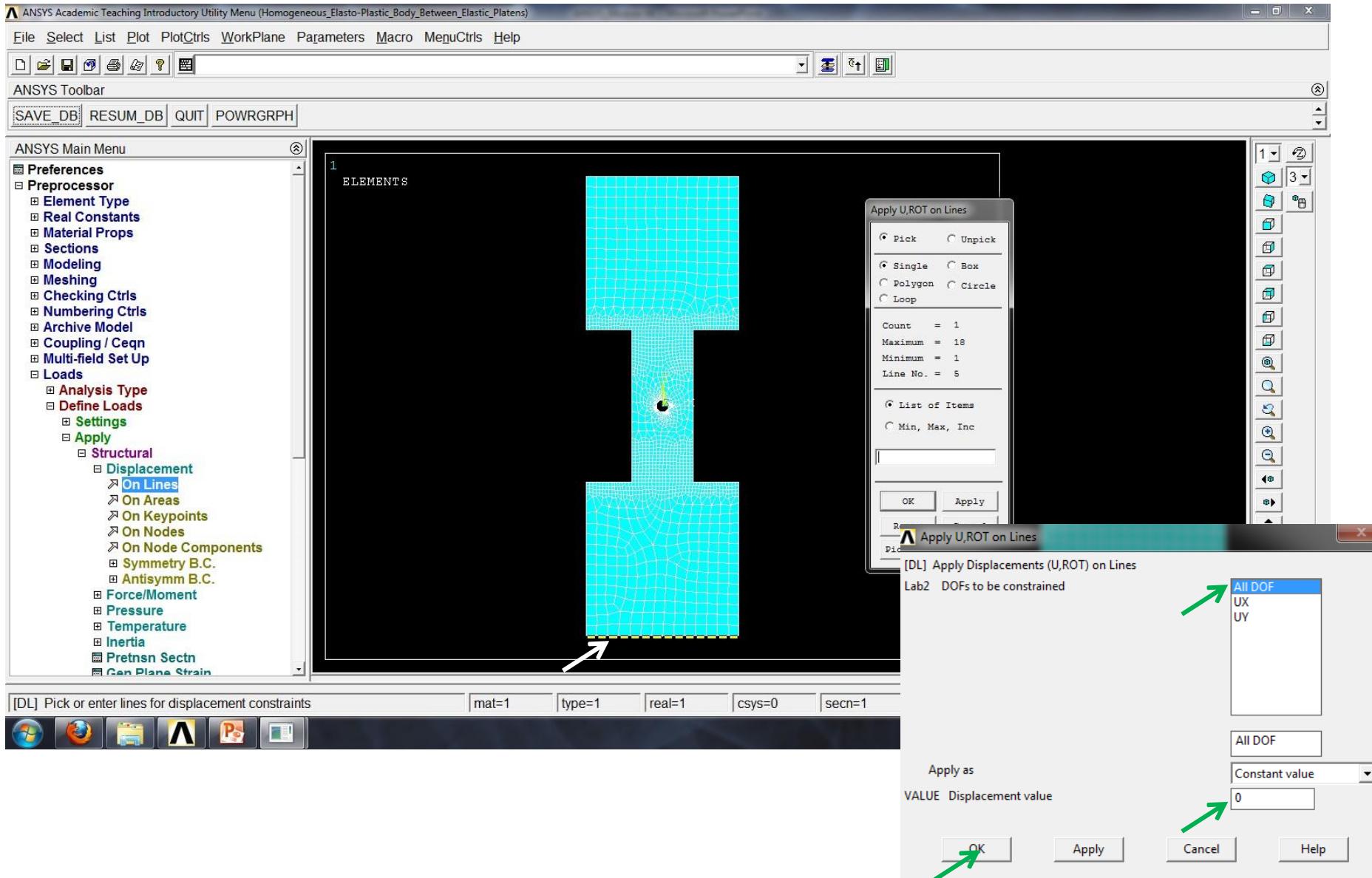


Applying Displacement Boundary Conditions

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

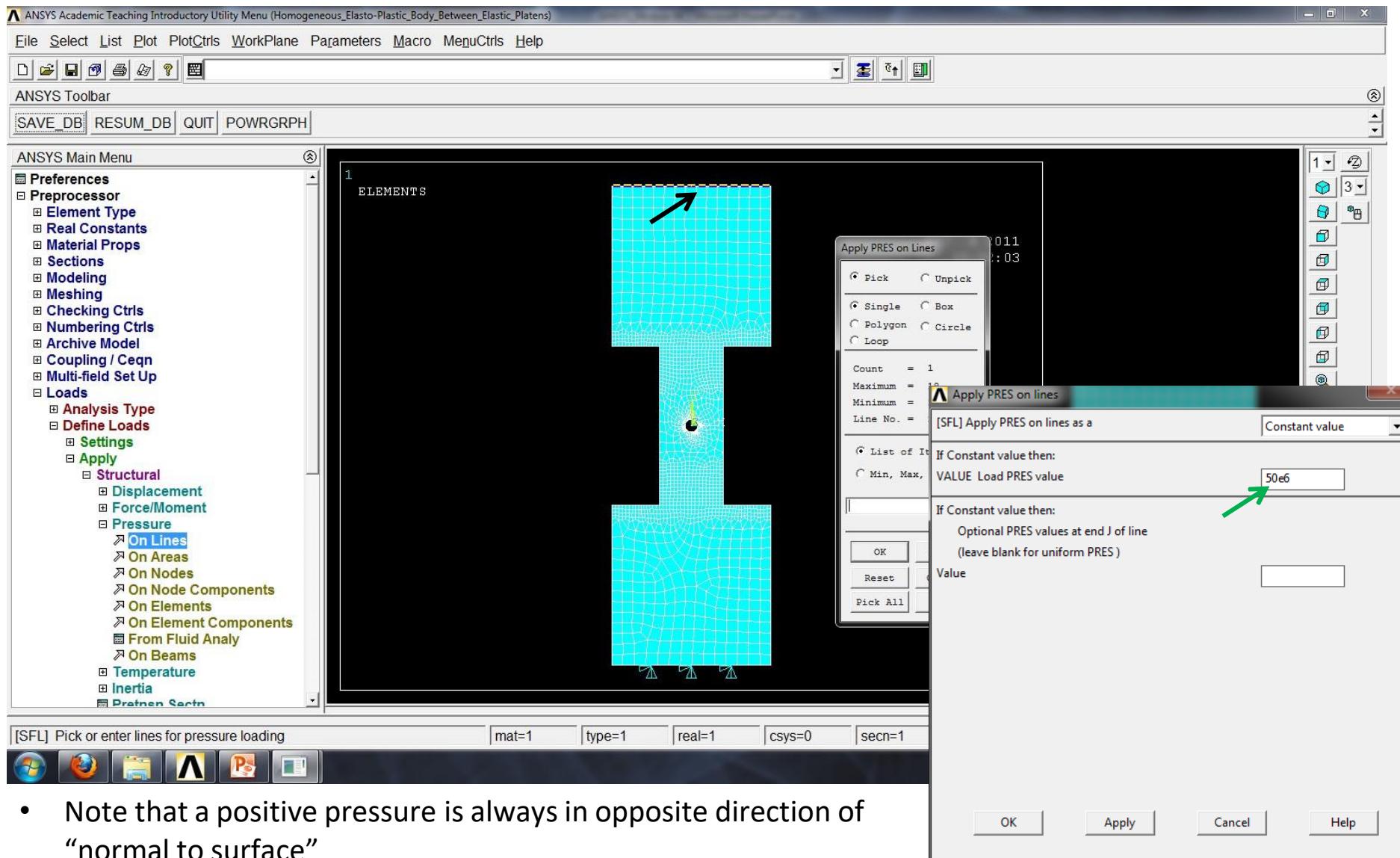


Applying Displacement Boundary Conditions

Preprocessor → Loads → Apply → Pressure → On Line

Choose *Top most line* → Press OK

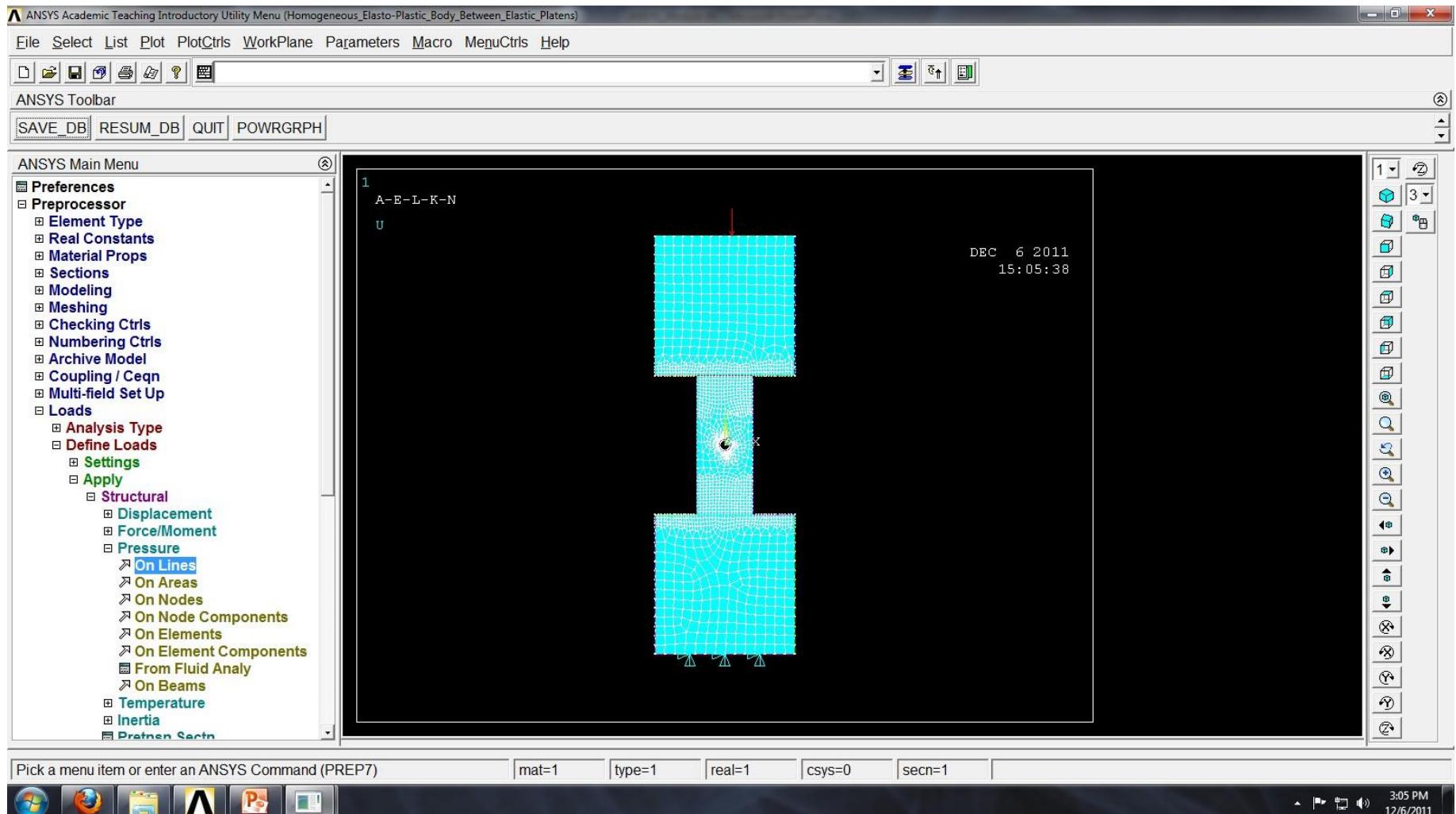
In the pop-up window: *Load PRES = 50e6* (i.e. 50MPa)



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

Plots → Multi Plots

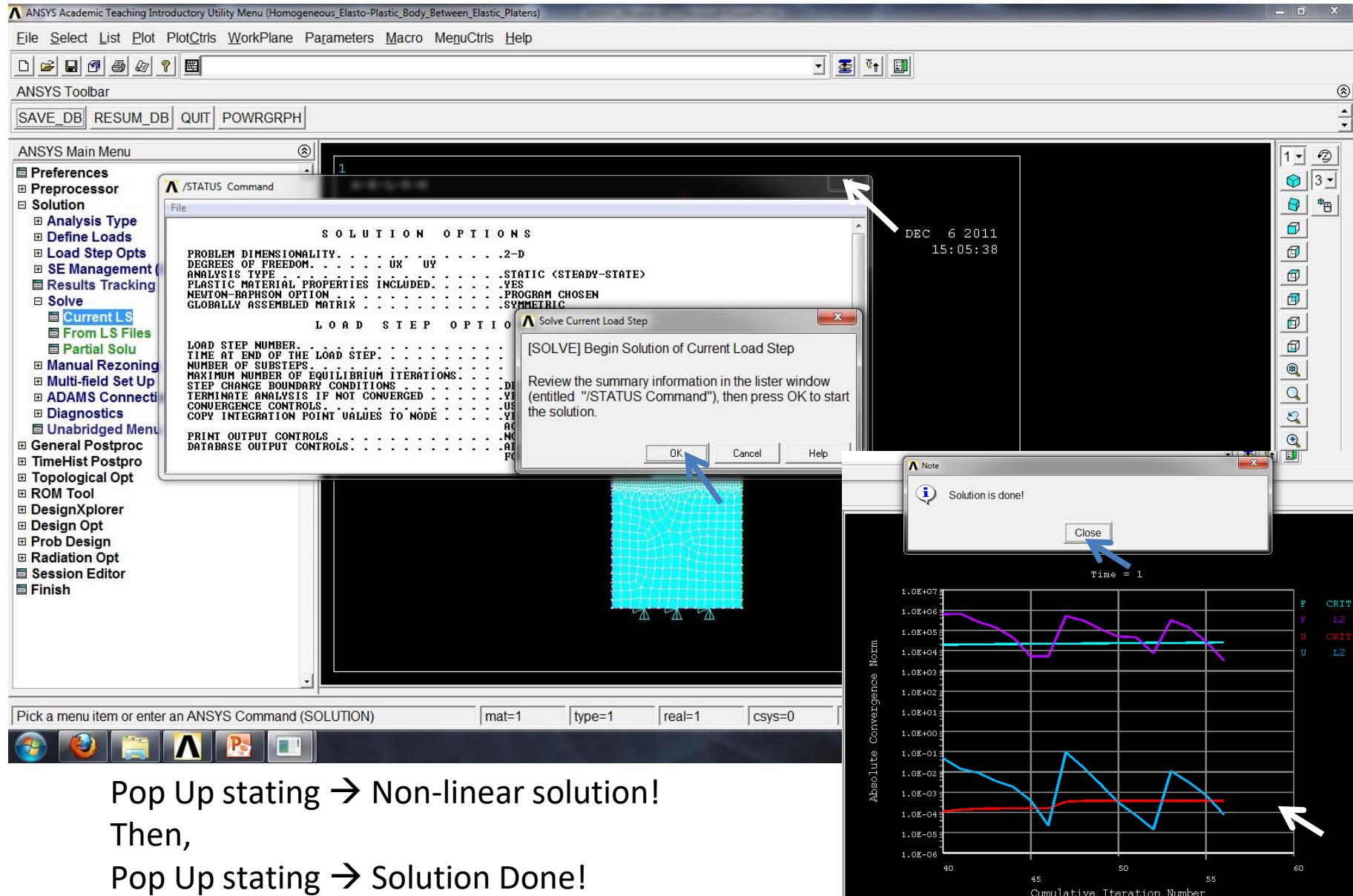
→ Check if all boundary conditions and loads are correct!



Solving the Model

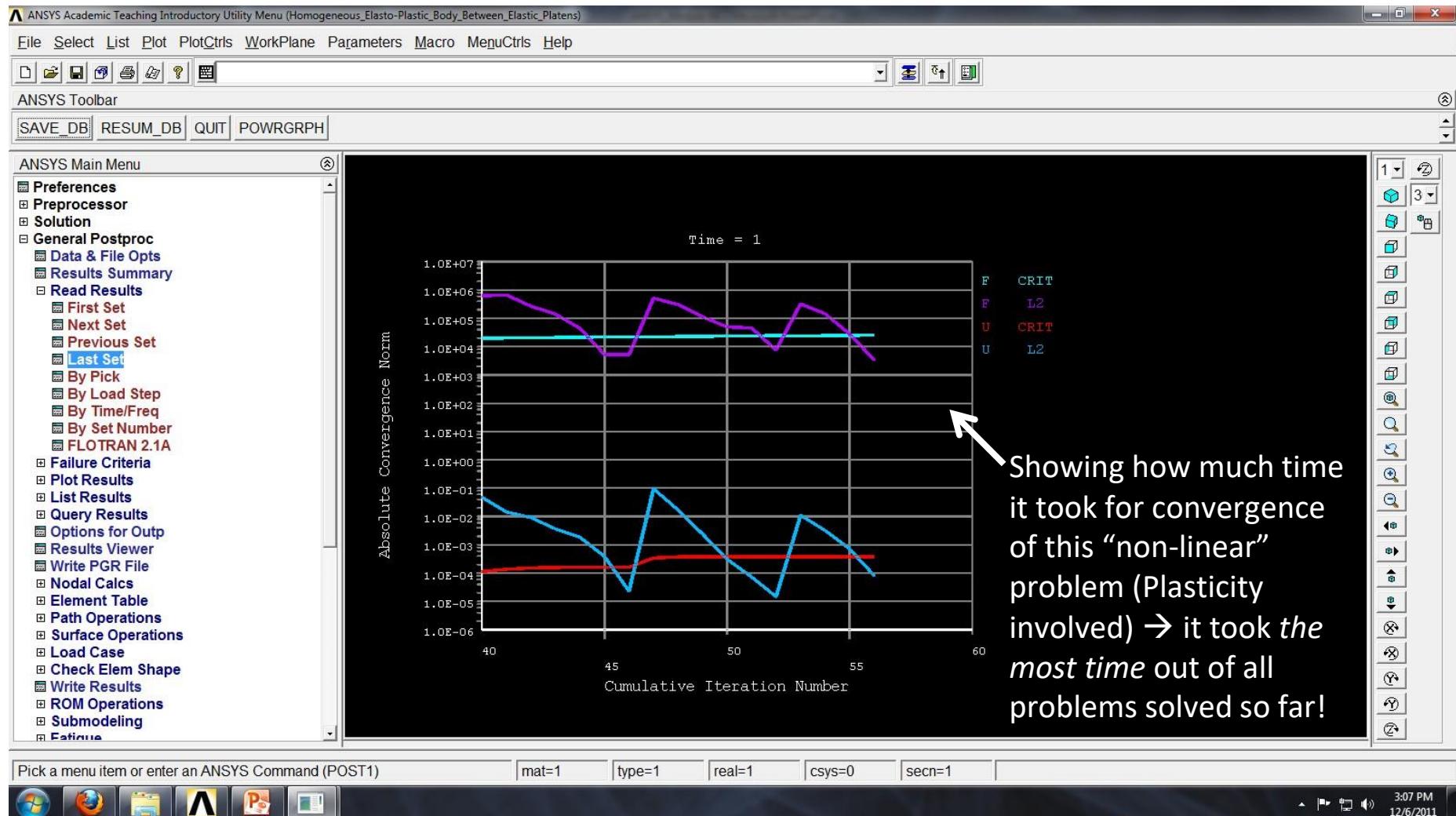
Solution → Current LS

Choose OK



Reading Results

General PostProc → Read Results → Last Set

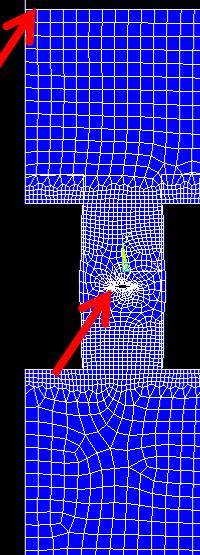
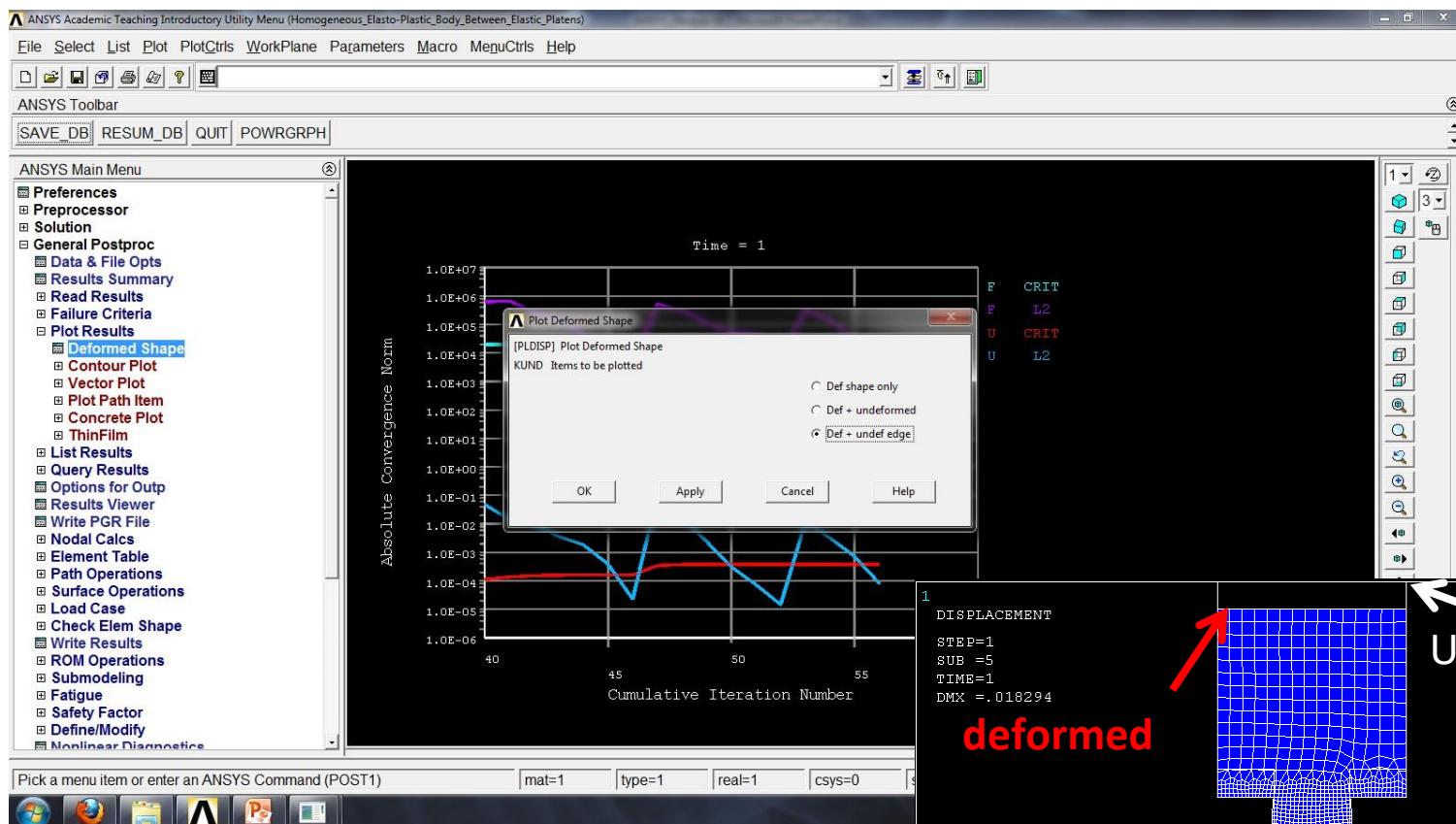


Pick a menu item or enter an ANSYS Command (POST1)

mat=1 type=1 real=1 csys=0 secn=1

Plotting/Listing Results

General PostProc → Plot Results → Deformed Shape
Choose *Def+ Undef Edge*

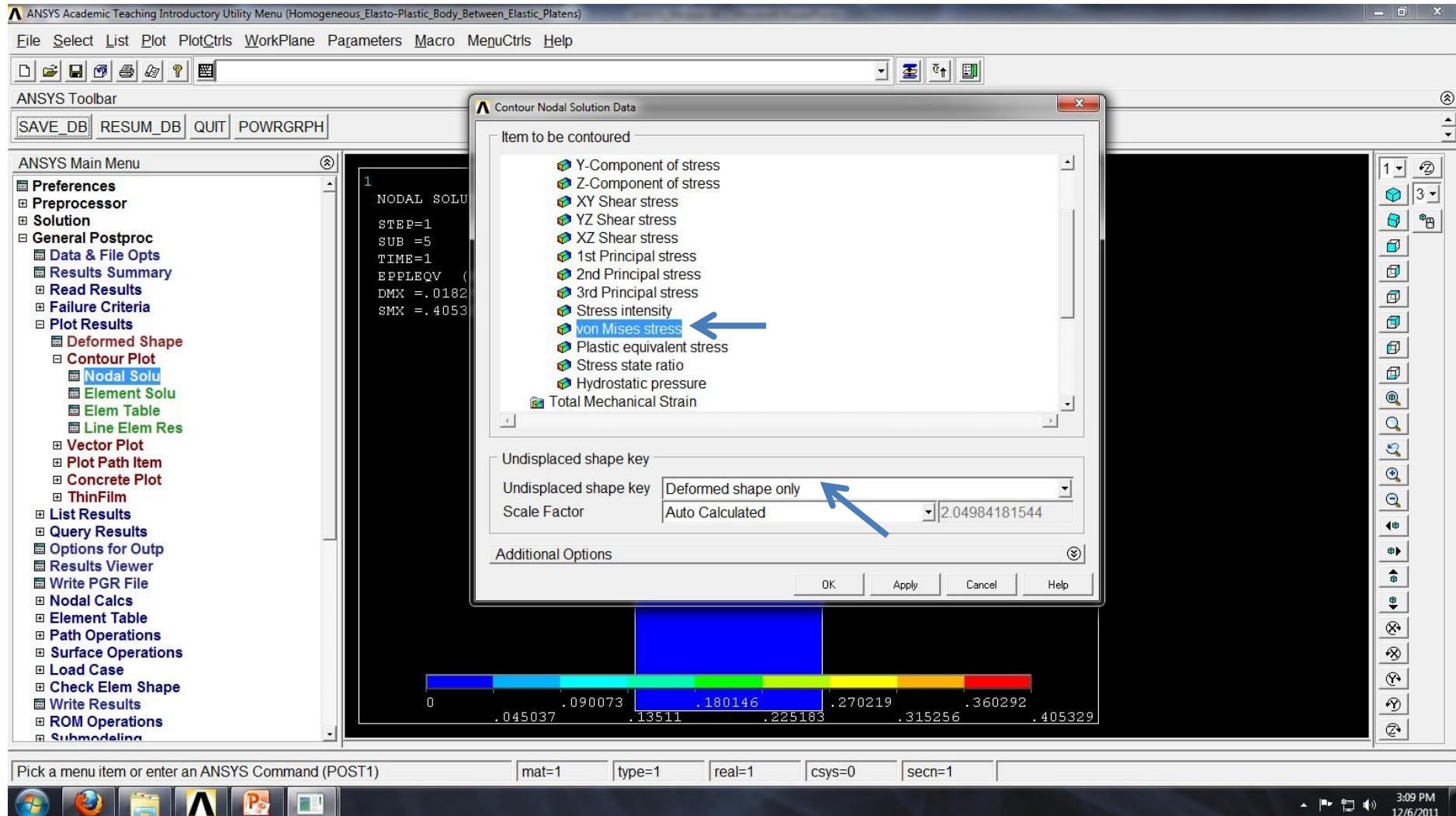


Un-deformed
DEC 6 2011
15:07:32

Plotting Results

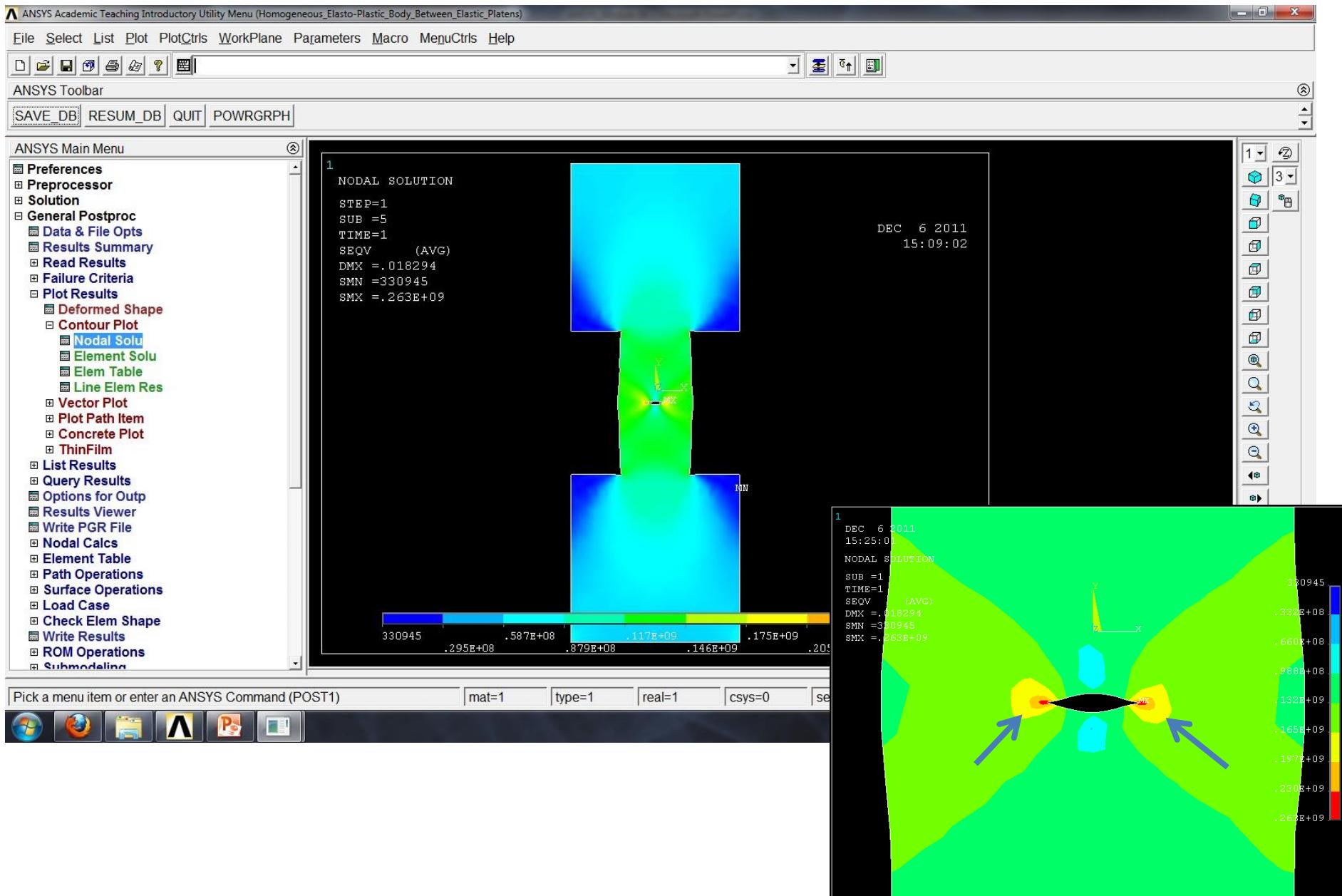
General PostProc → Plot Results → Contour Plot → Nodal Solu

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

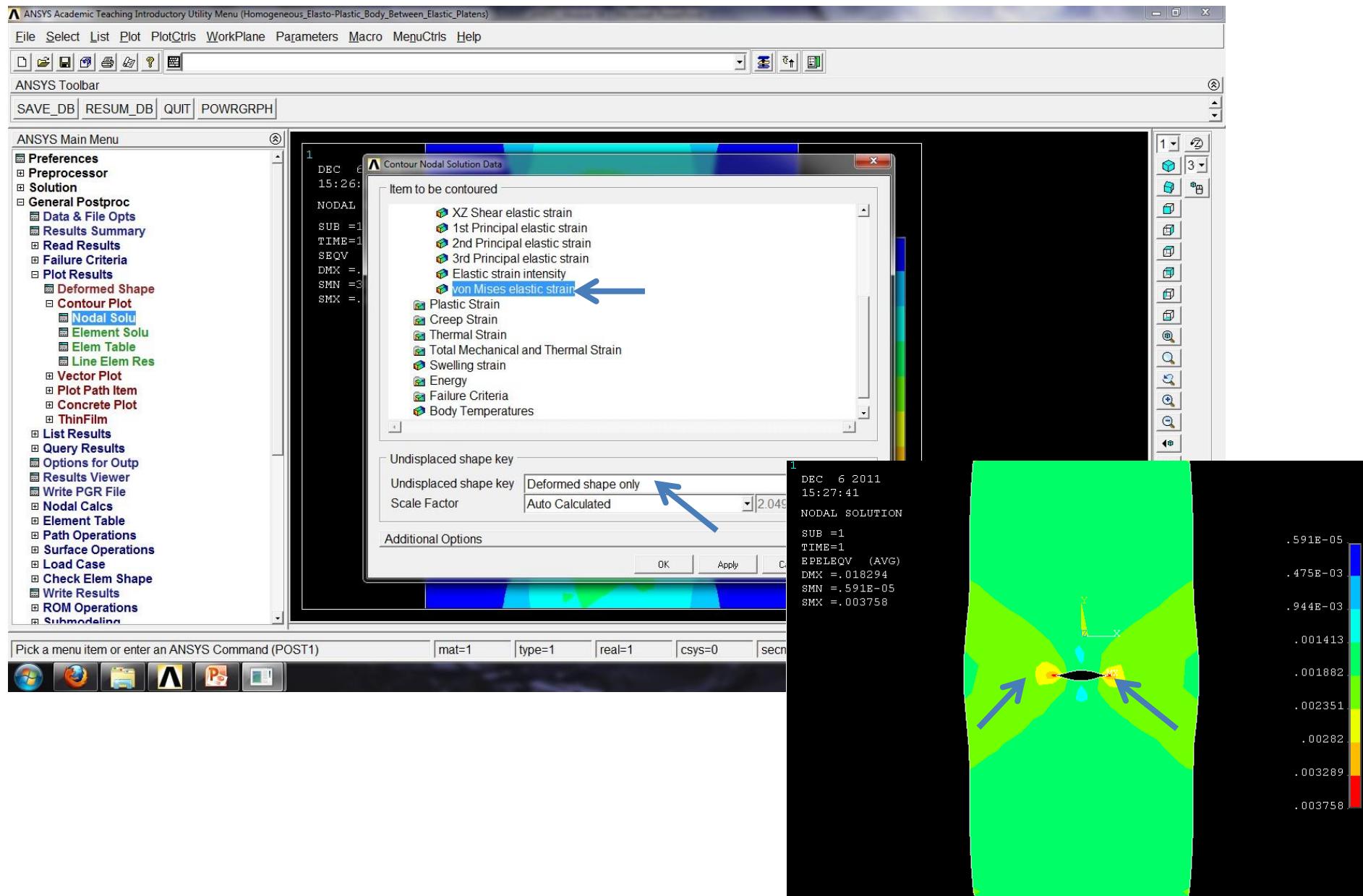
von Mises Stress: Axial Compression



Plotting Results

General PostProc → Plot Results → Contour Plot → Nodal Solu

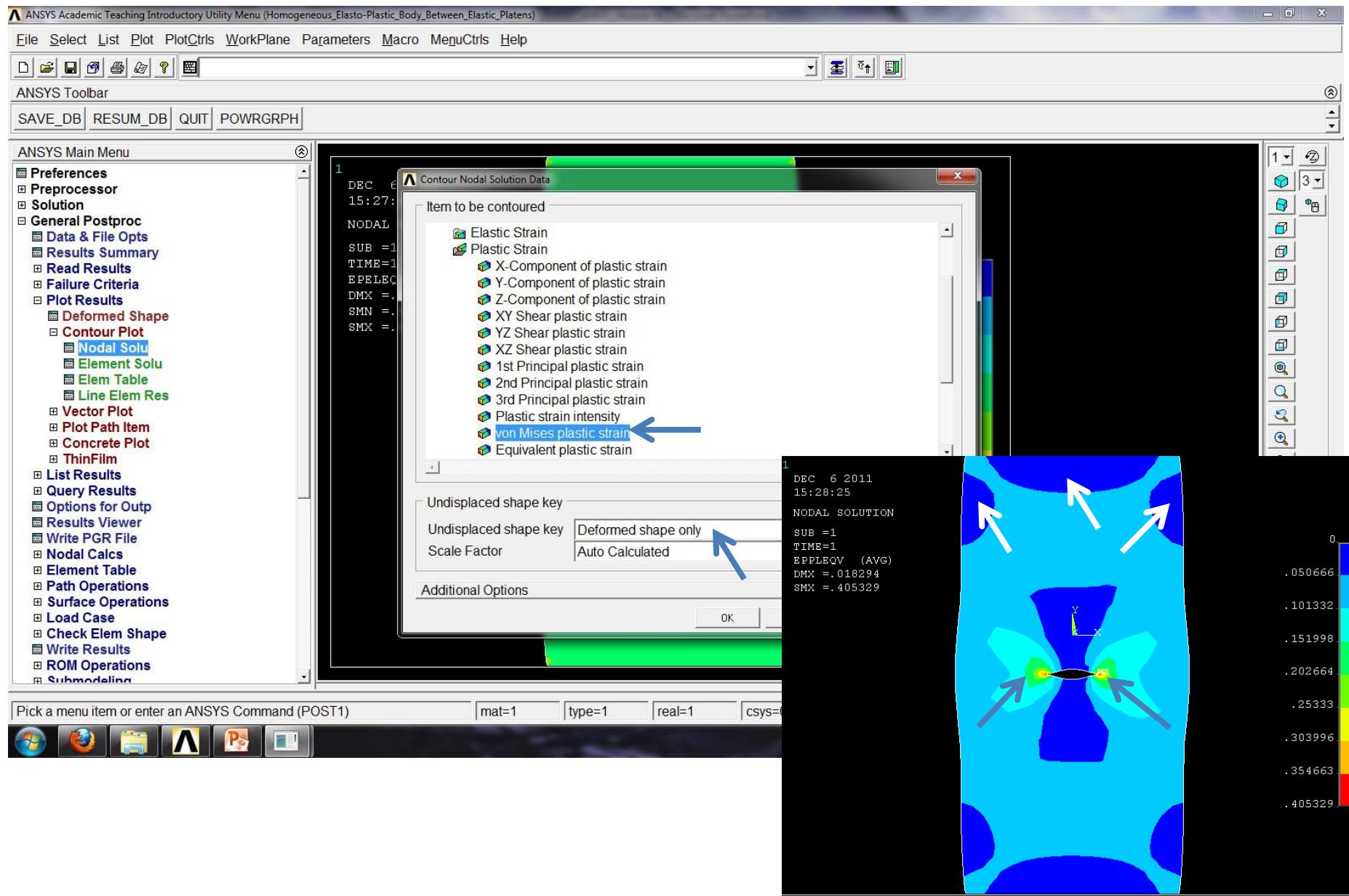
Choose *Nodal Solutions* → Elastic Strain → *von Mises Strain*



Plotting Results

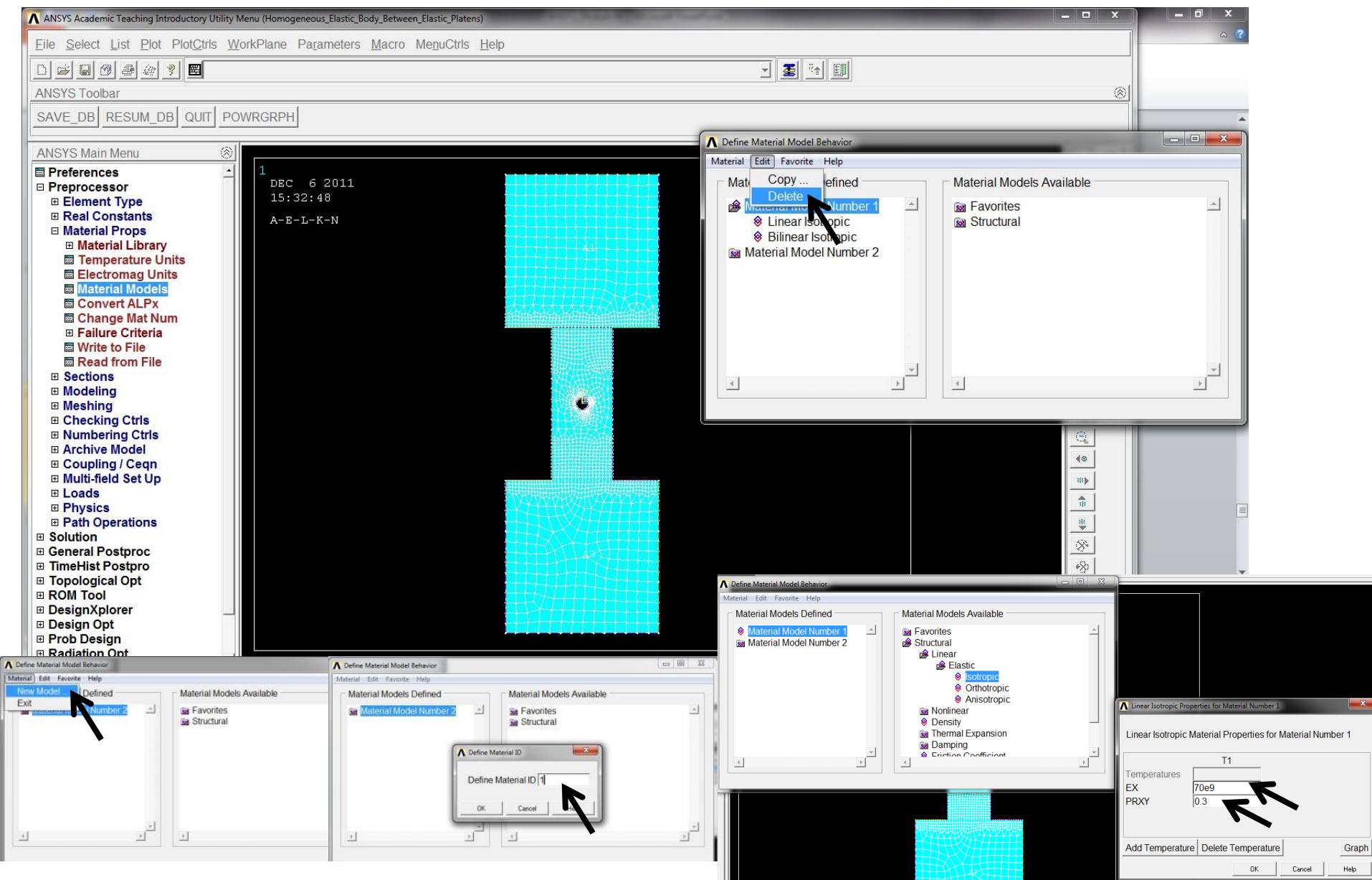
General PostProc → Plot Results → Contour Plot → Nodal Solu

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



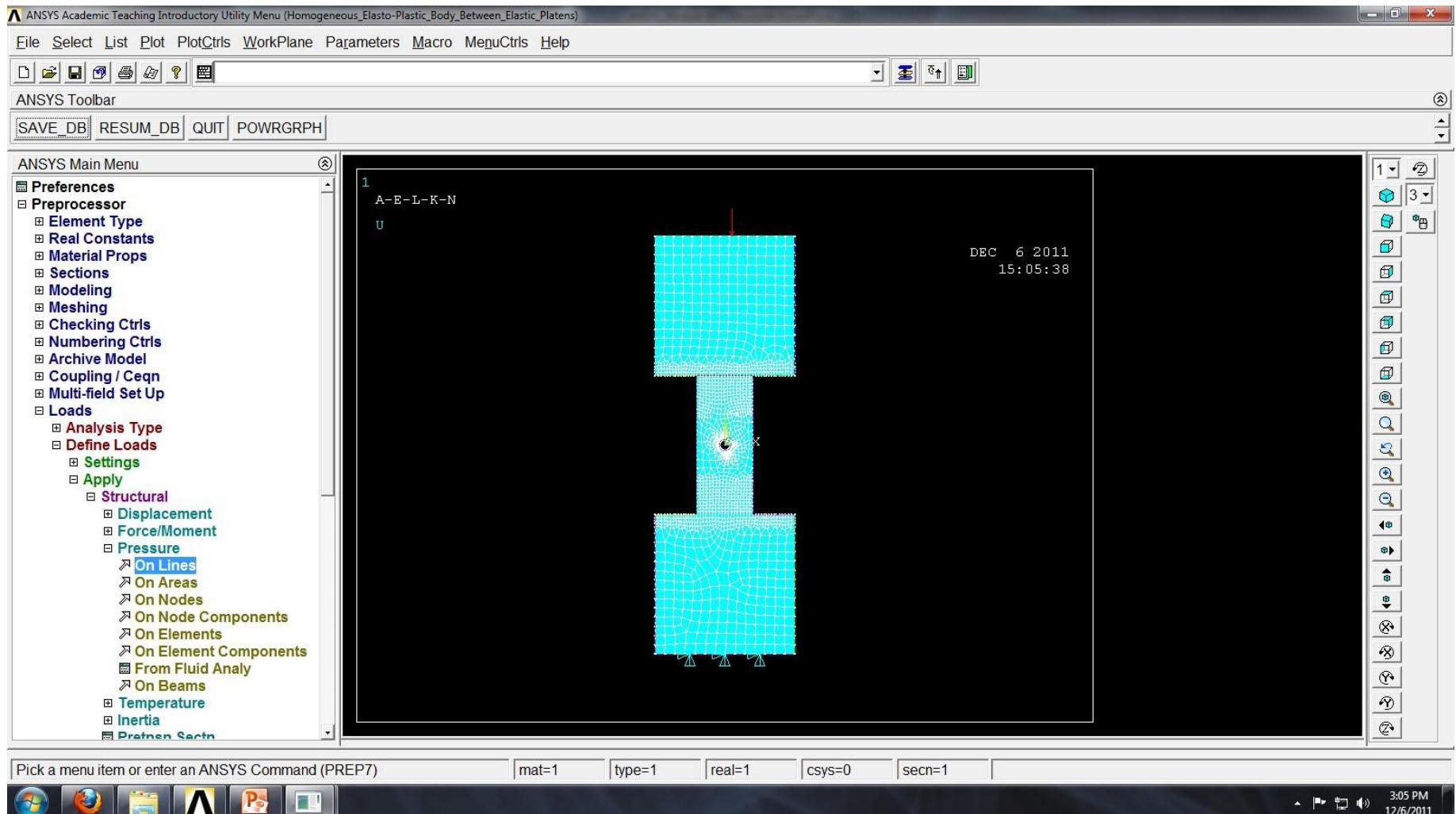
➤ Let's solve the earlier problem with no plasticity!

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



Plots → Multi Plots

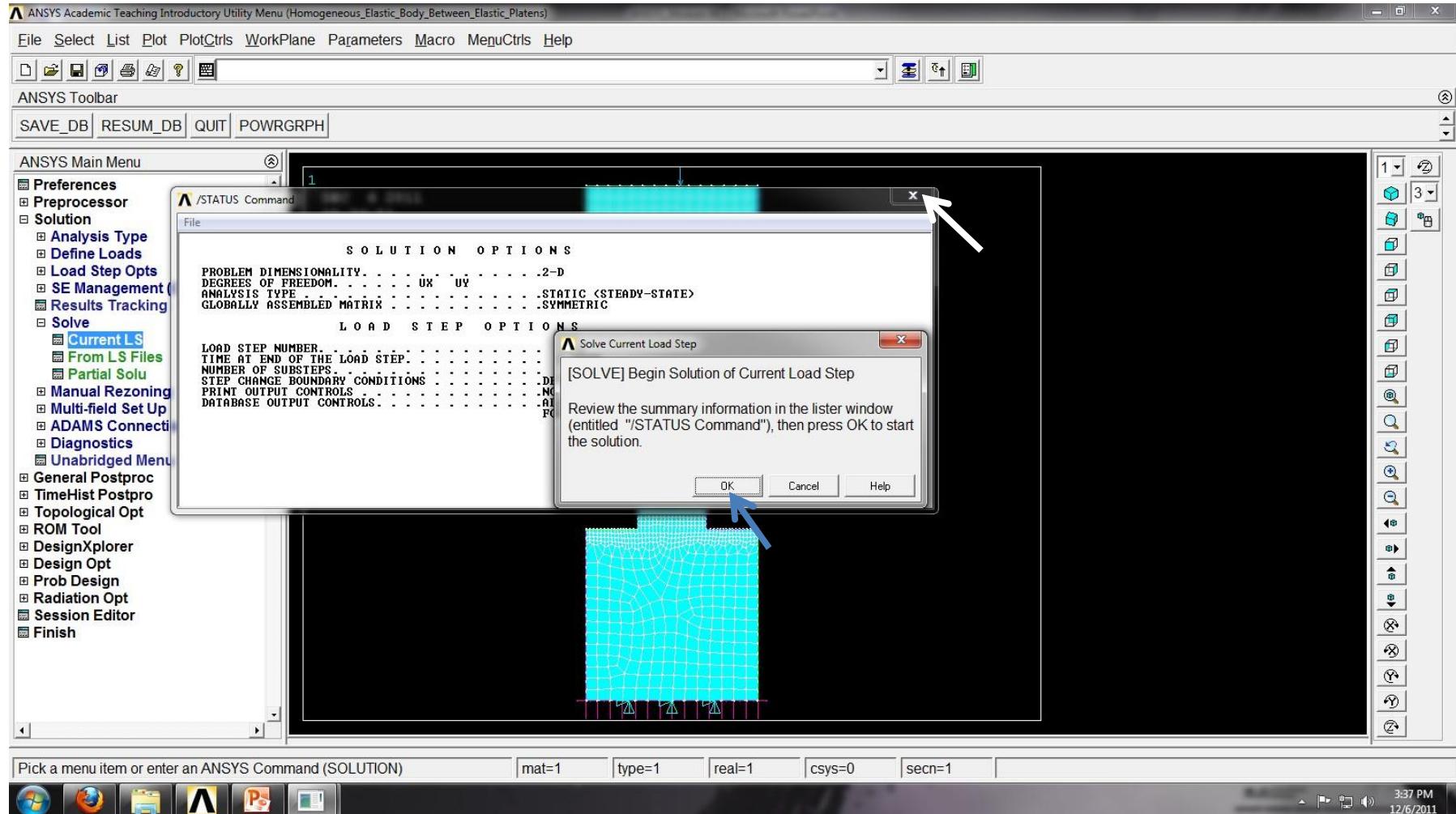
→ Check if all boundary conditions and loads are correct!



Solving the Model

Solution → Current LS

Choose OK

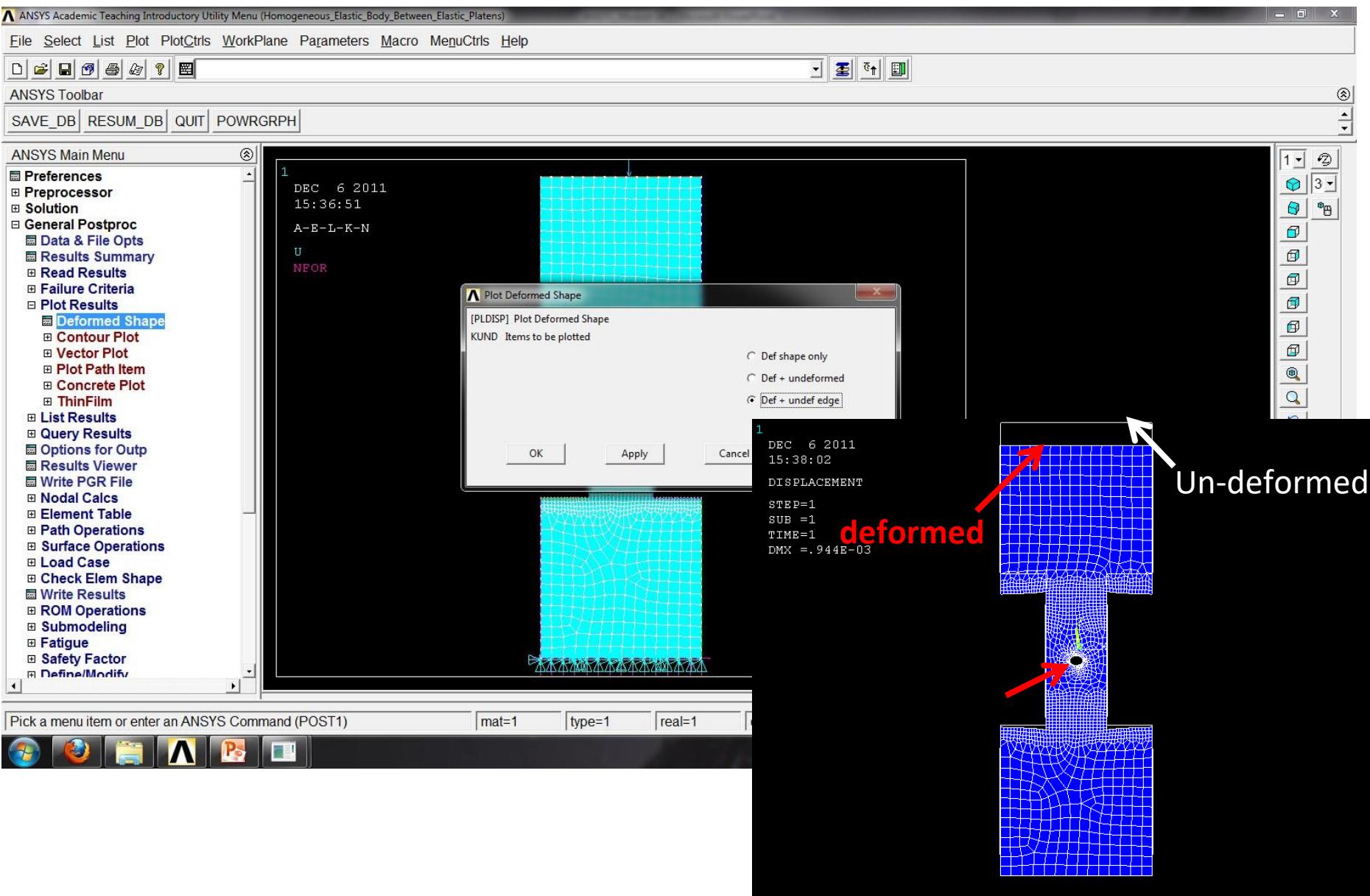


Pop Up stating → Solution Done!

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

Plotting/Listing Results

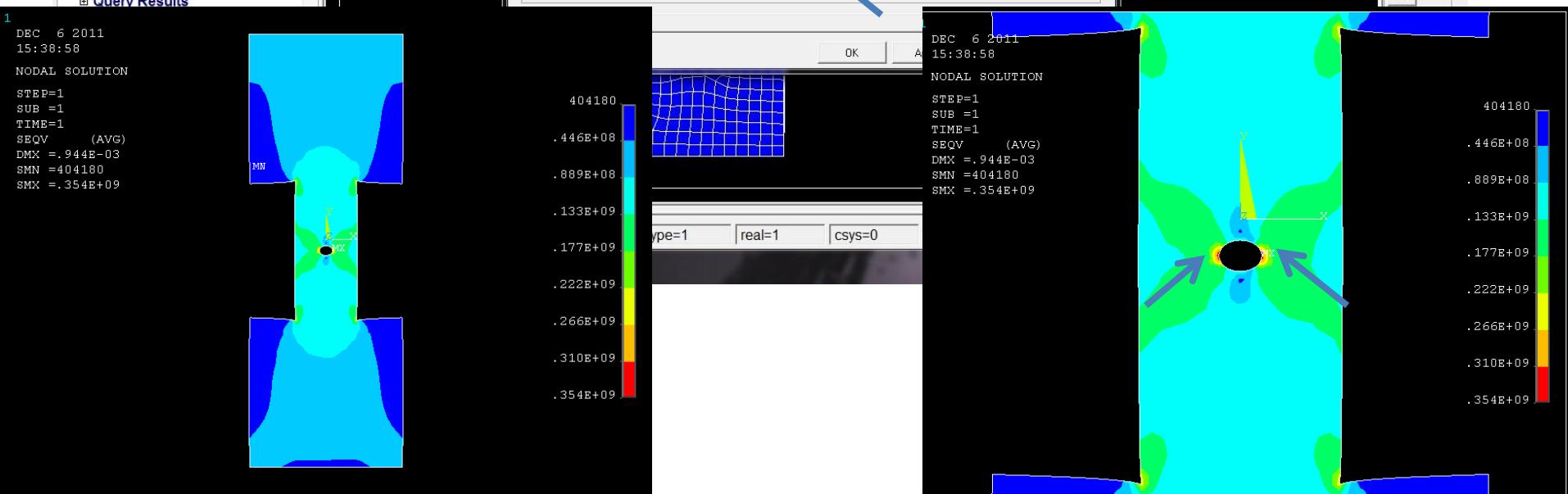
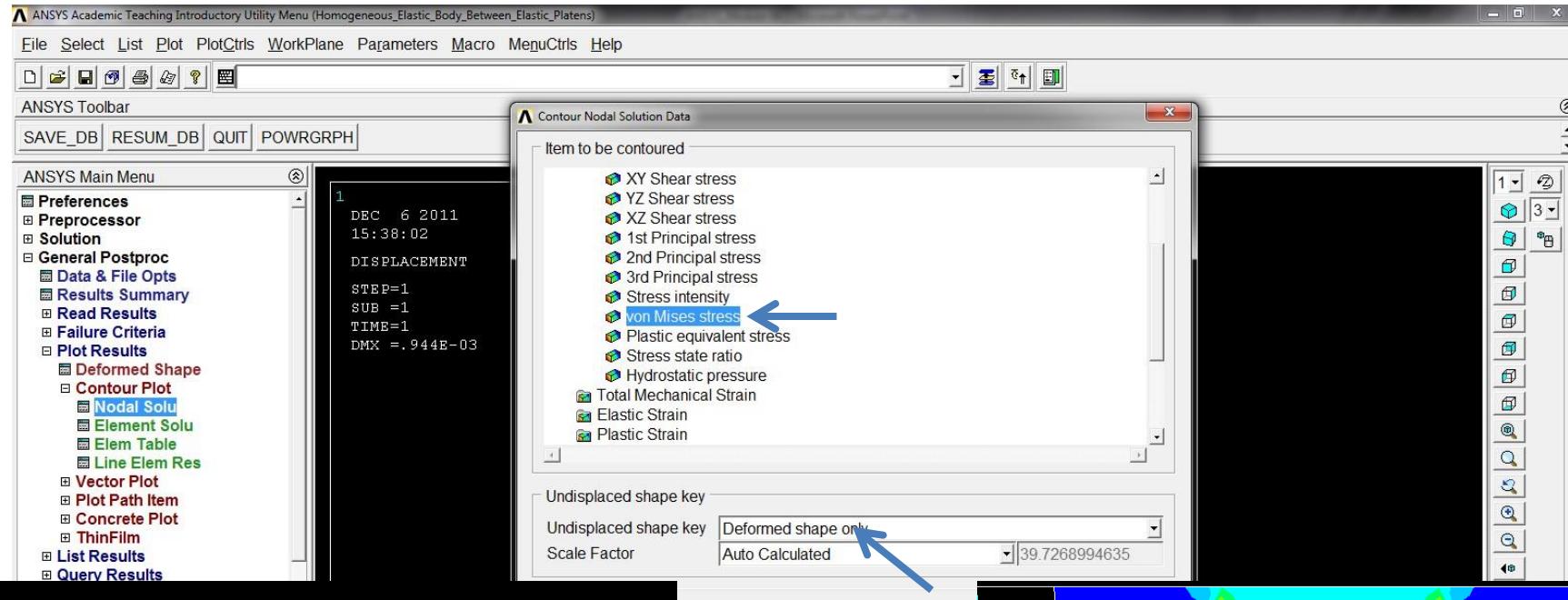
General PostProc → Plot Results → Deformed Shape
Choose *Def+ Undef Edge*



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*

