Basic FEA and Savin Solution for Elliptical Holes

Notes prepared by:

Mr. Daniel Sasser
MSME Candidate

April 2015

Accessing Ansys for ME Students

- https://www.me.washington.edu/remotedesktop
- Then follow the access instructions, using your Netid and password (same as MyUW)
- Open Ansys 16.0 (folder on desktop)
- Open Workbench or Mechanical APDL
- AA students should speak with Brian Leverson about access

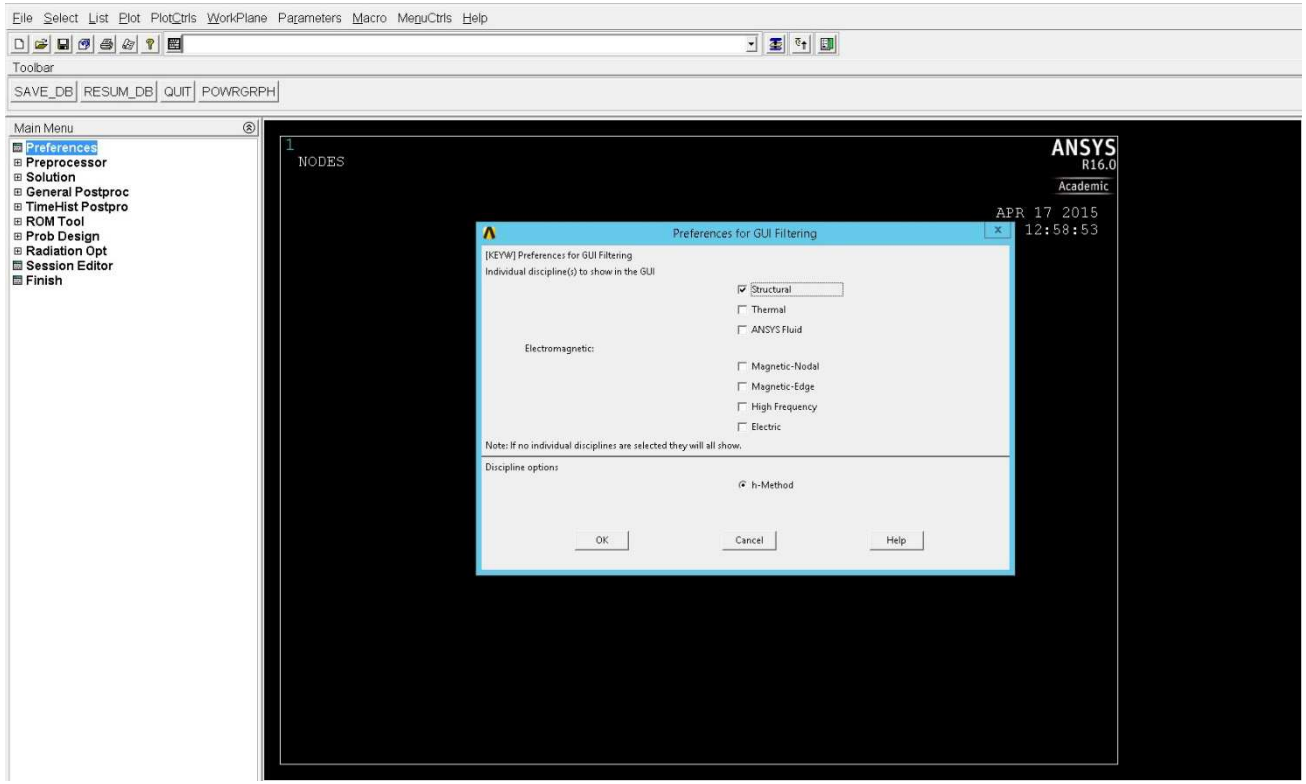
Workbench Setup, Square Panel Example using ACP

- Add ACP (pre) – drag and drop
  - Engineering Data - add material as needed (orthotropic elasticity, etc) (units)
  - Geometry – Sketch desired shape (12” square) (units)
    - Be careful with placement
    - Import CAD file
    - Create>Surface from sketch>Select sketch
    - Generate
  - Model (units)
    - Select surface body, apply thickness
    - Mesh (can be refined, mapped, etc)
    - Generate
  - Setup (units)
    - Fabric – ply thickness (0.007161 in)
    - Stackup – stacking sequence and order (top down or bottom up, symmetry or not)
    - Oriented Selection Set – elements to apply lay-up to and which direction to stack
    - Modeling Groups – number of plies and orientation (12 plies, 0 deg)

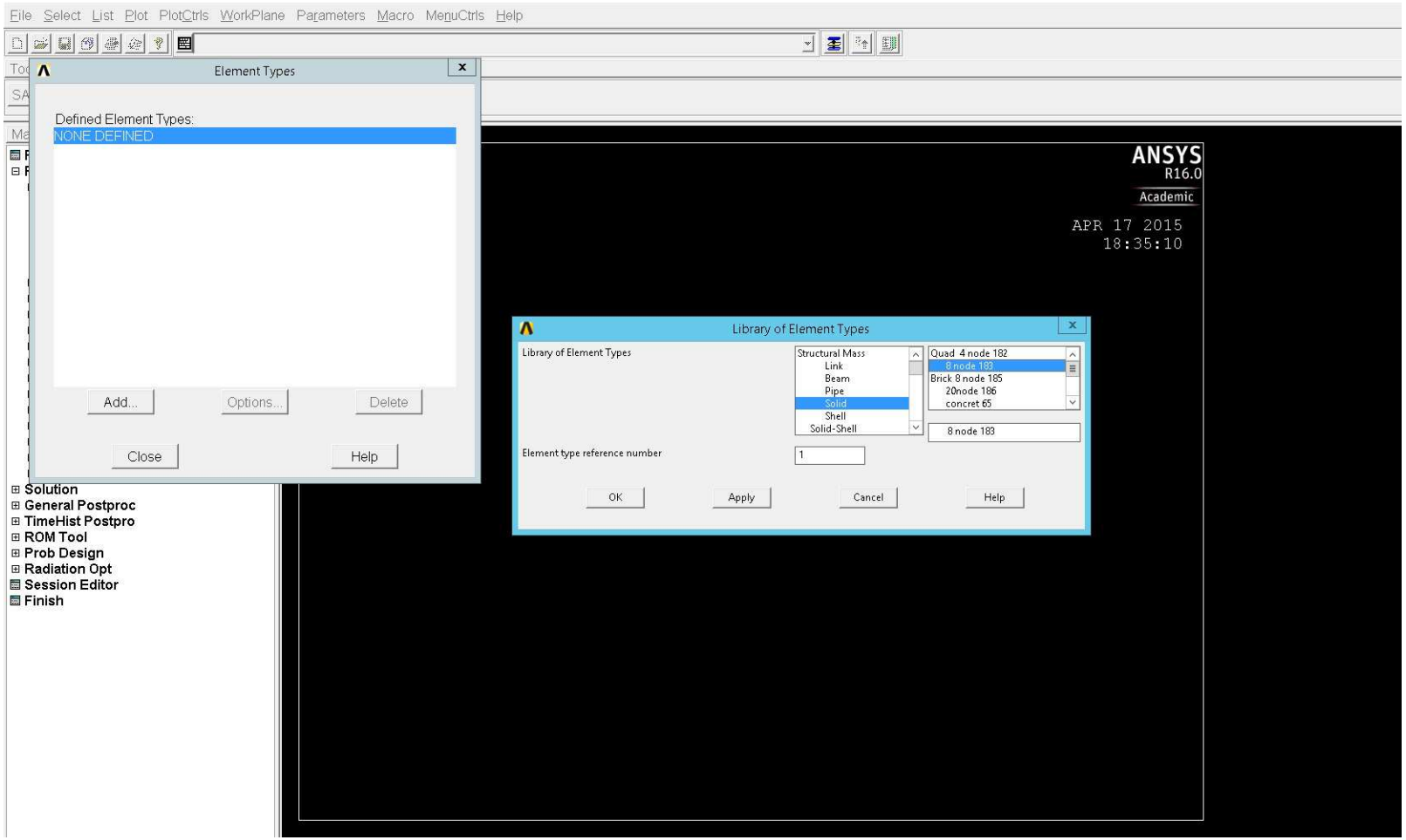
Workbench Setup, Square Panel Example using ACP

- Static Structural – drag and drop on ACP (pre) through Setup
  - BC's, Loads, Output geometry (paths, probes, etc)
    - Line pressure along top (stress resultant, 100 lb/in)
    - Displacement along bottom (y = 0)
  - Outputs (normal stress, strain, deformation, etc)
  - Solve
- Compare to CLT (Professor Tuttle's code)
  - Unidirectional panel – constant strain and stress
    - $\varepsilon_{11} = 46 \times 10^{-6}$ $\varepsilon_{22} = -15 \times 10^{-6}$
    - $\sigma_{11} = 1163.7$ psi $\sigma_{22} = -7.88 \times 10^{-7}$ psi

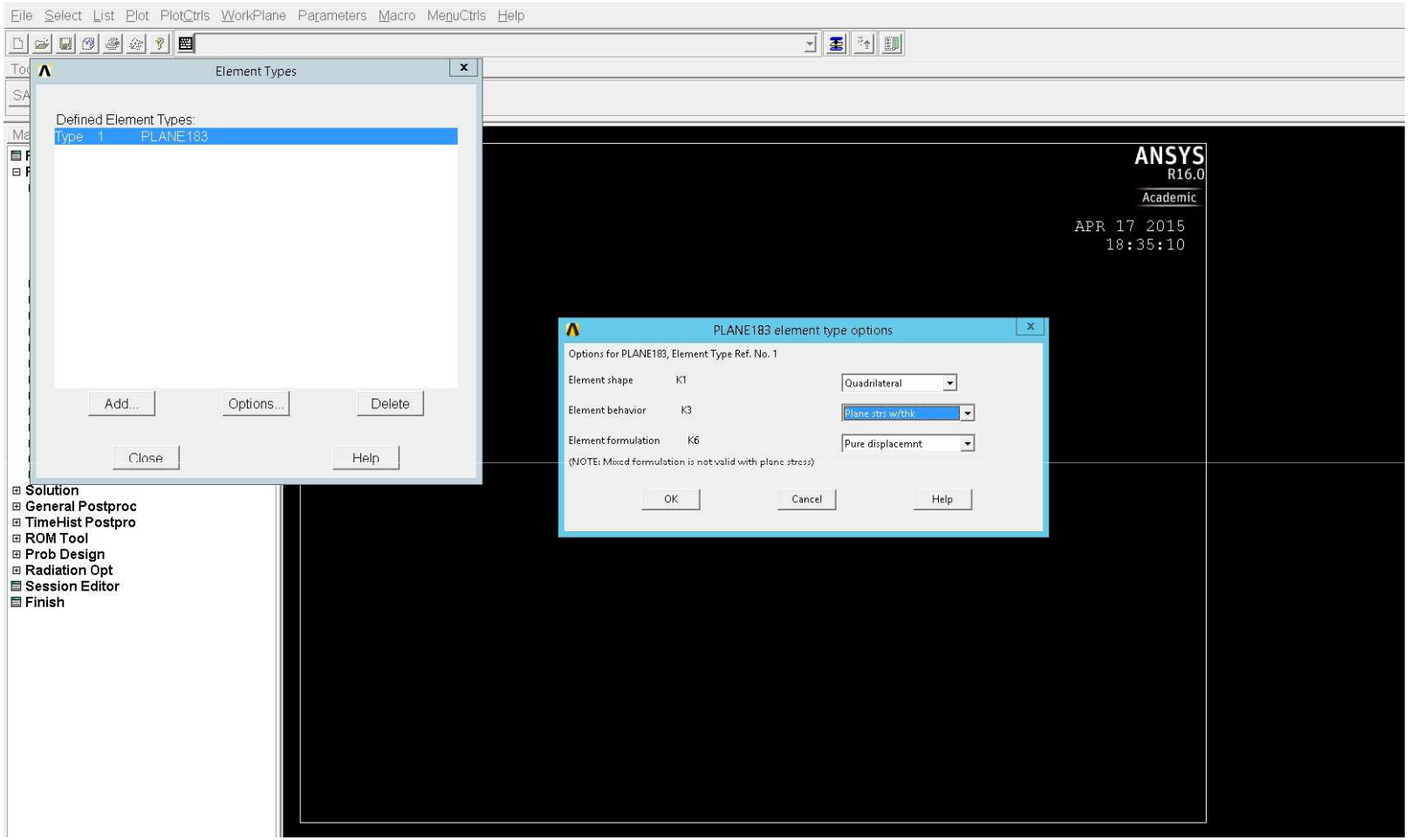
MAPDL Setup, Square Panel



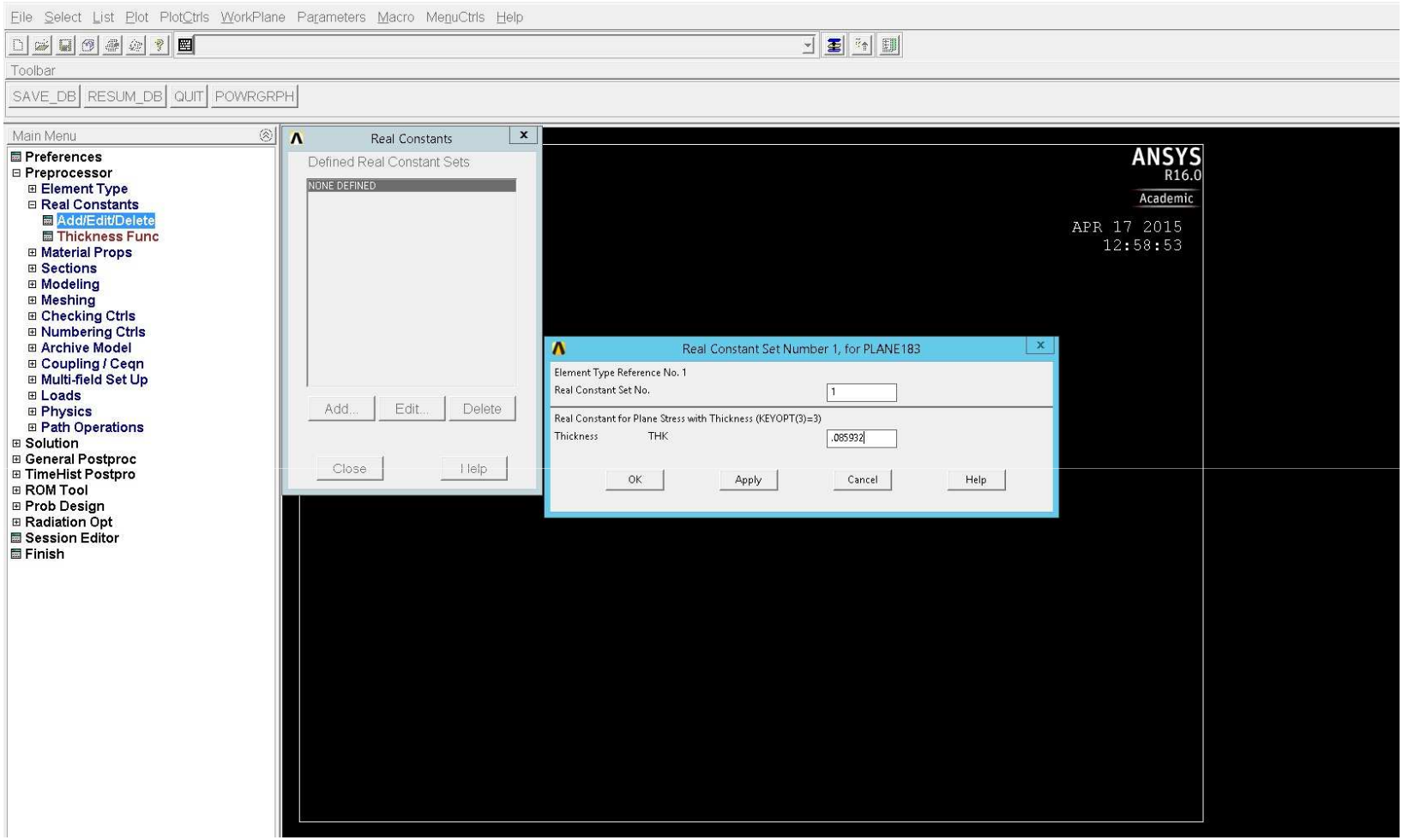
- Open Ansys Mechanical APDL
- Double click preferences
  - Select Structural Analysis



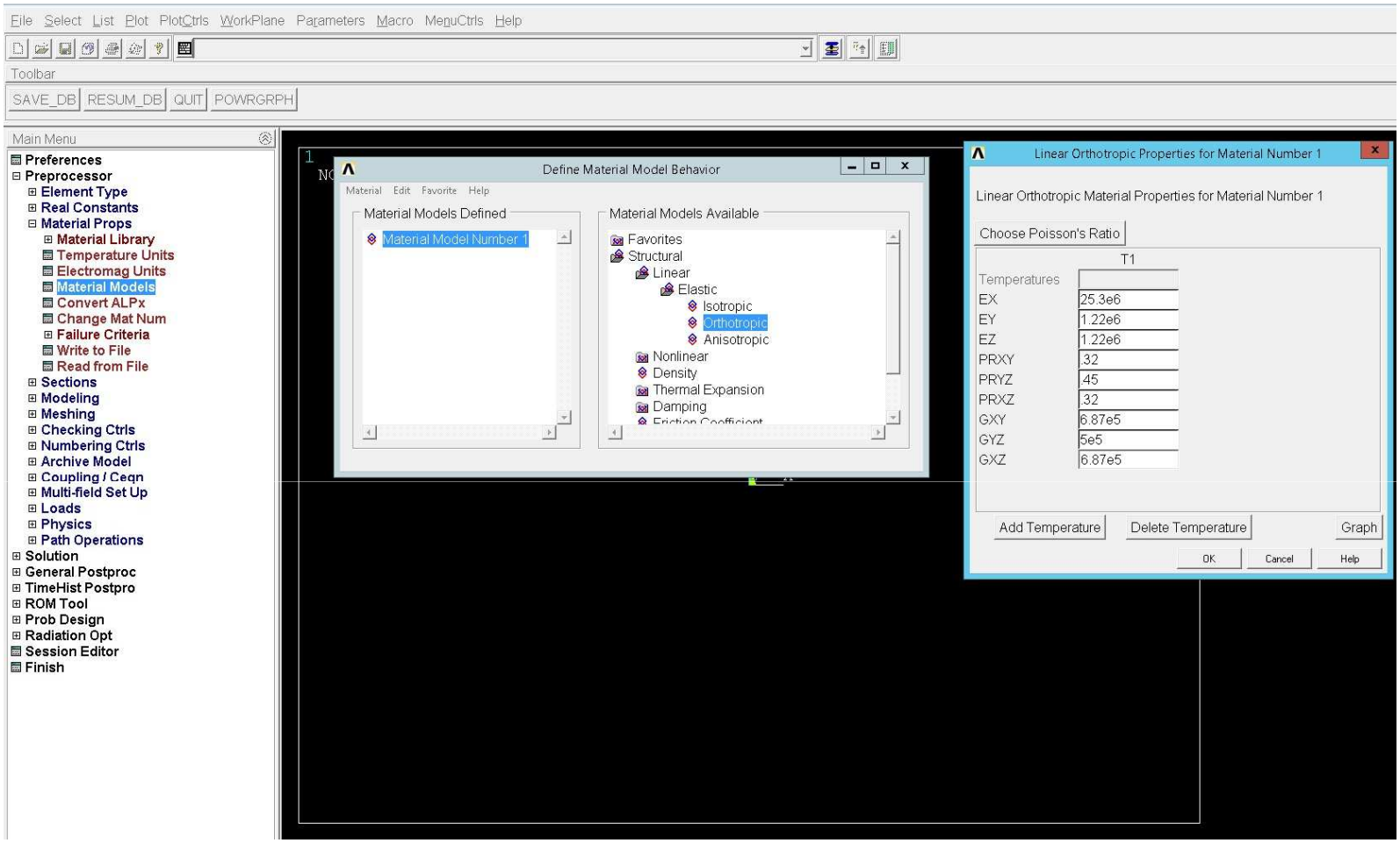
- Select Preprocessor>Element Type>Add/Edit/Delete
  - Add type Solid>Quad 183 (adds element type PLANE 183)



- Select Type 1>Options
- Change element behavior to ‘Plane stress with thickness’

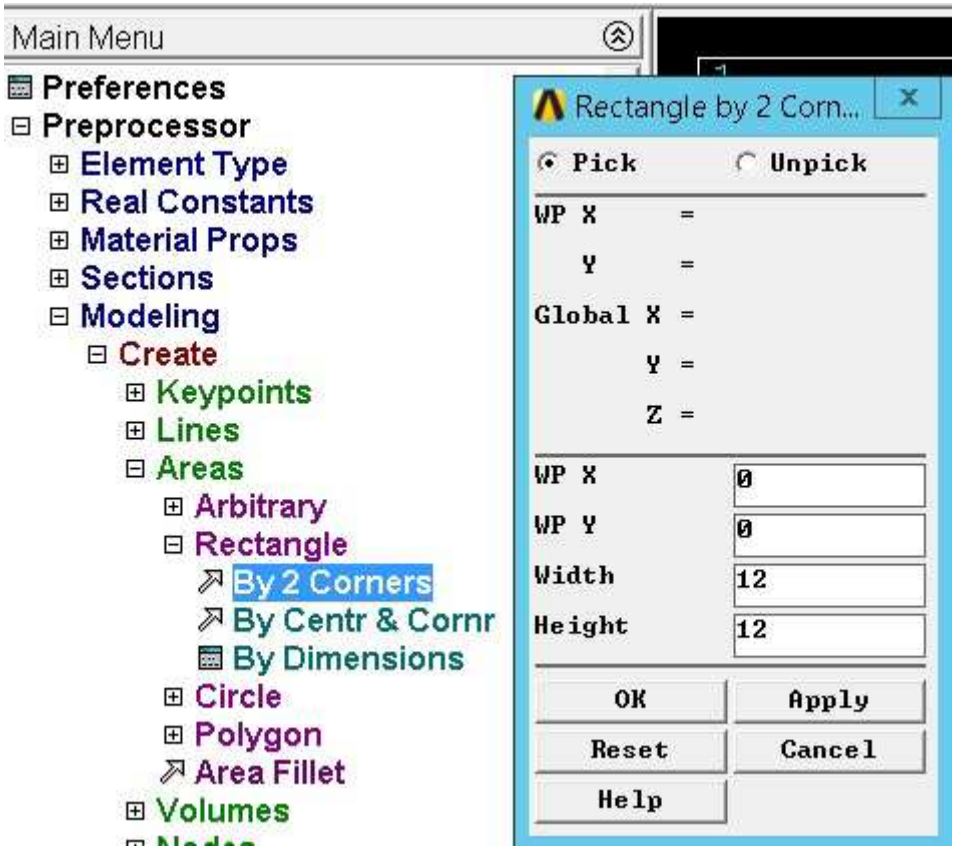


- Real Constants>Add/Edit/Delete>Add – Thickness
  - Created a surface model, then applied thickness to it, but the model is still just a surface



- Material Props>Material Models
  - Structural>Linear>Elastic>Orthotropic
    - Set material properties

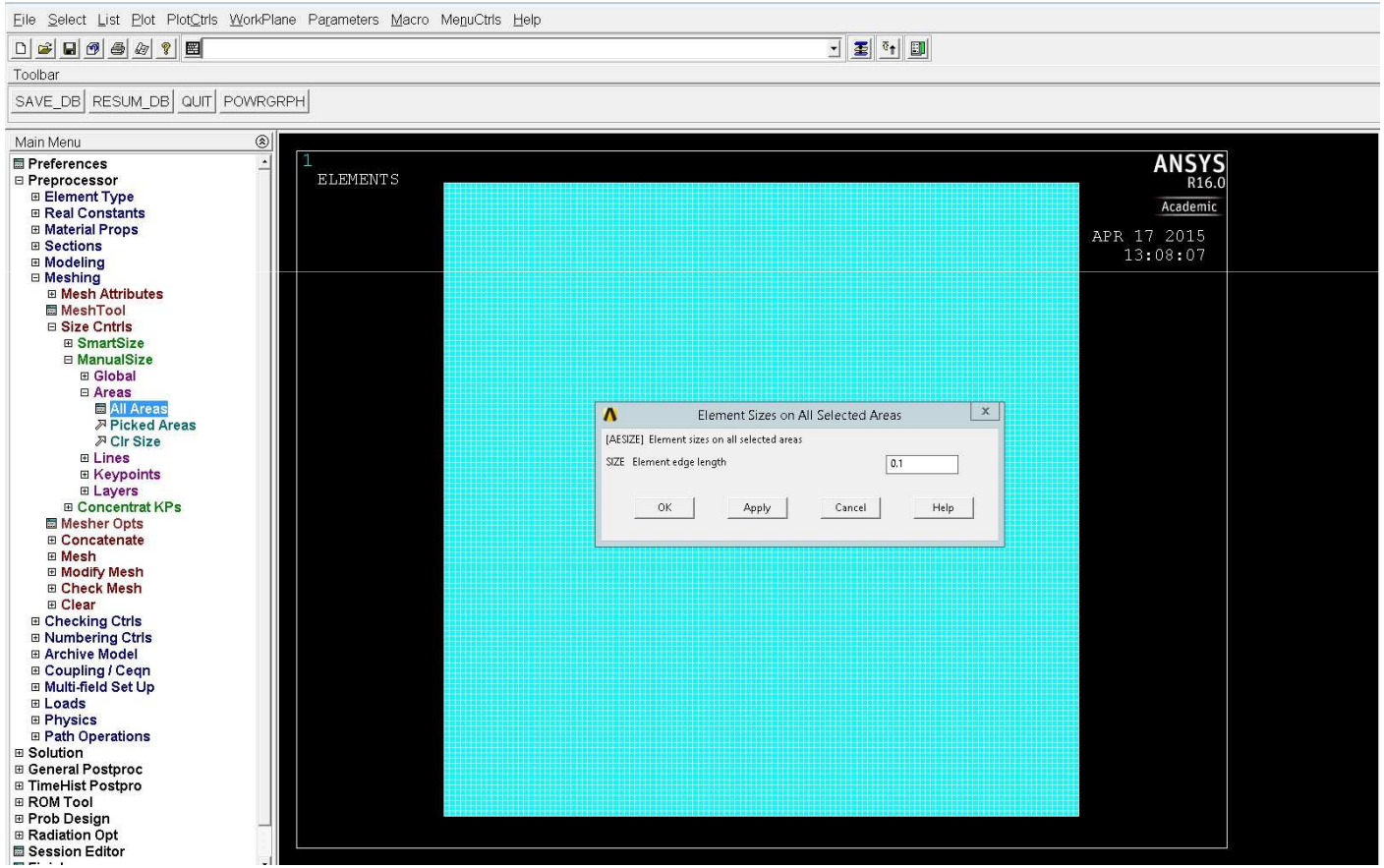
Create Geometry



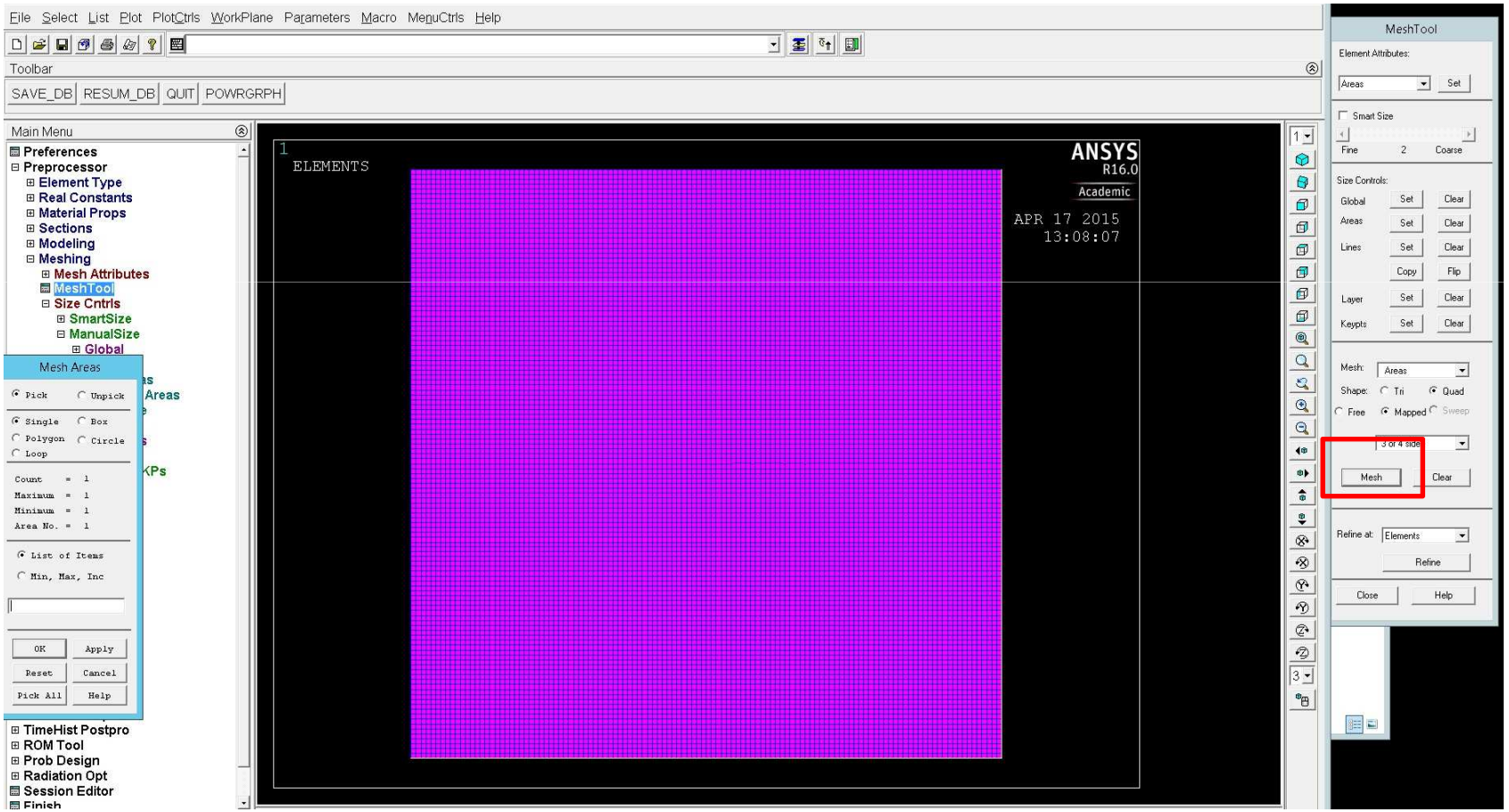
- Modeling>Create>Areas>Rectangle
  - By 2 Corners at (0,0) and (12,12)

Meshing

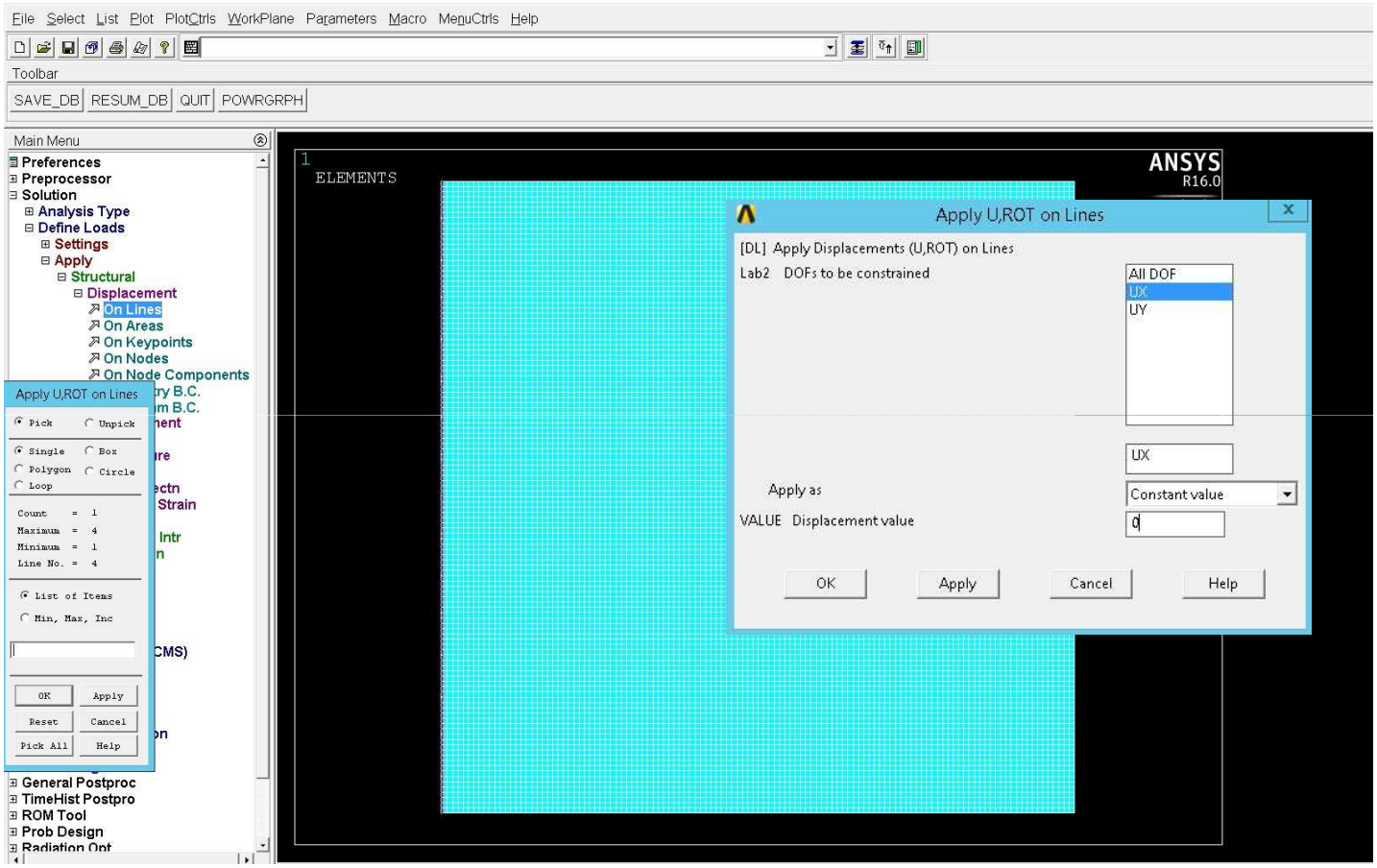
- Meshing>Size Cntrls>ManualSize>Areas>All Areas
  - Set edge length (0.1 results in mesh seen in blue)
  - This sets the size of the mesh, the next step will create it



- Meshing>MeshTool (window on far right)
  - Select ‘Mesh’ button in red box
- Select the area, then click ok (window on left)

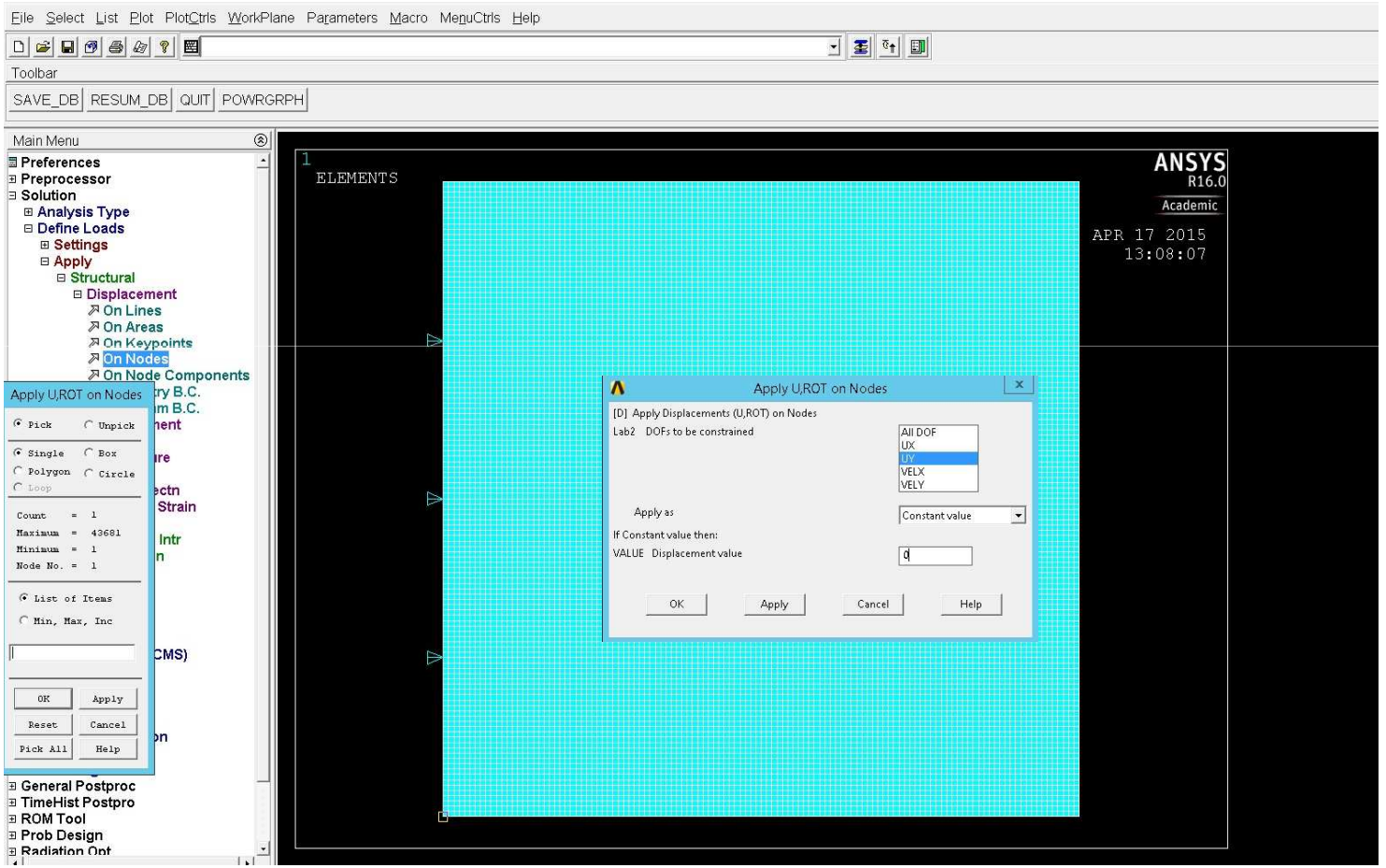


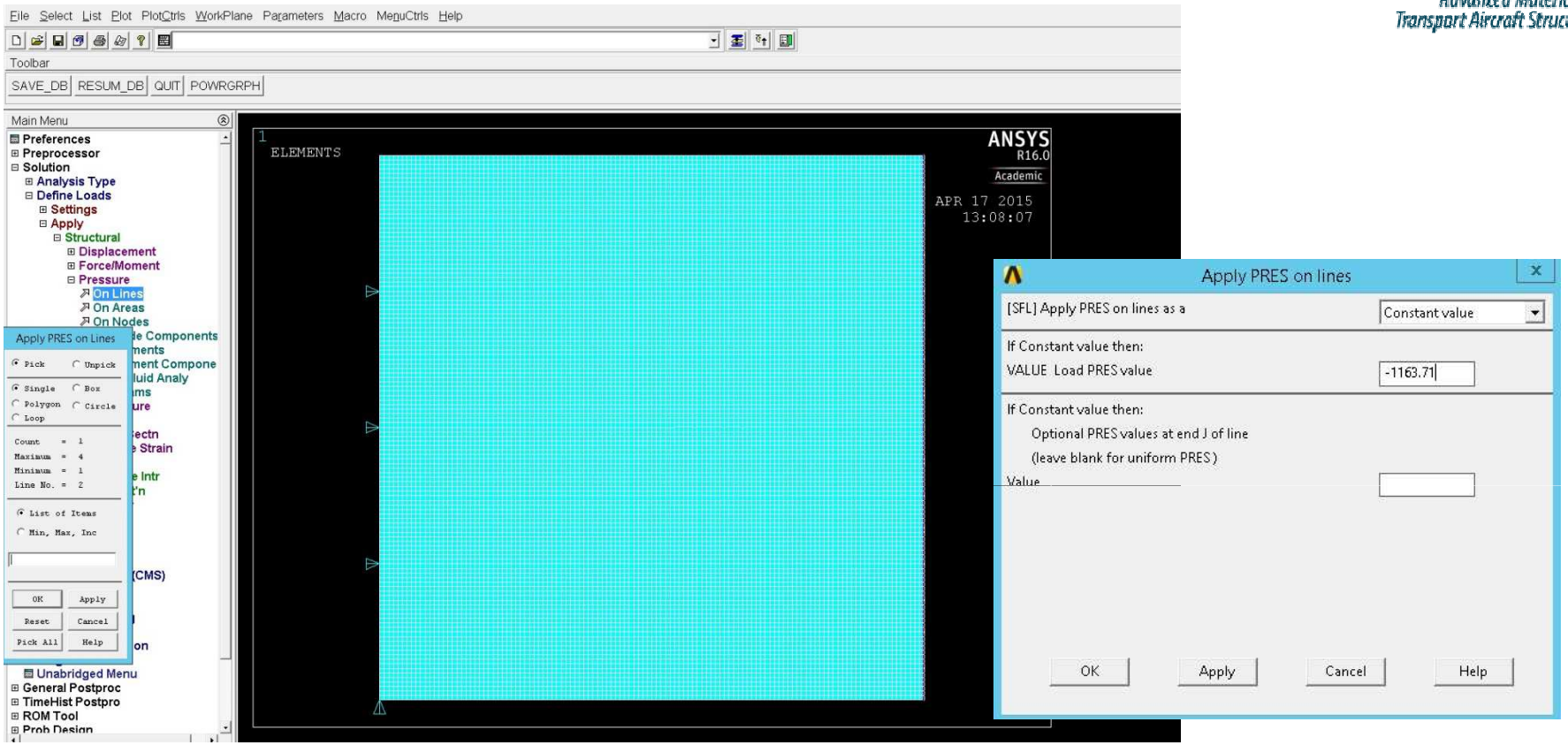
Boundary Conditions and Loads



- Solution>Define Loads>Apply>Structural>Displacement>On Lines
  - Select left edge (highlighted in purple), click ok
  - Constrain UX = 0 (cannot move to the left or right)

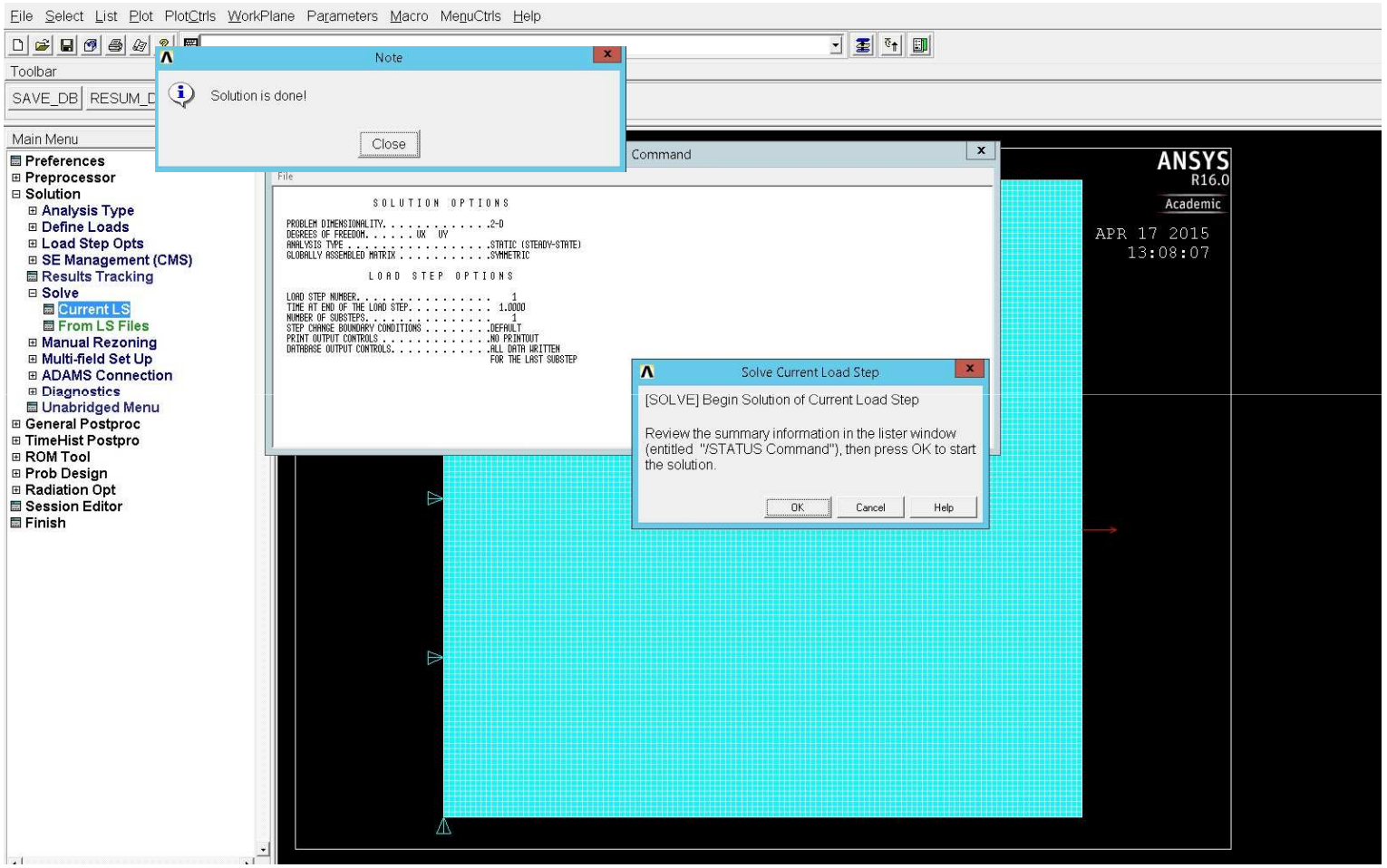
- Apply>Structural>Displacement>On Nodes
  - Select lower left corner, click ok, set UY = 0
  - Will not solve unless you do this due to rigid body motion





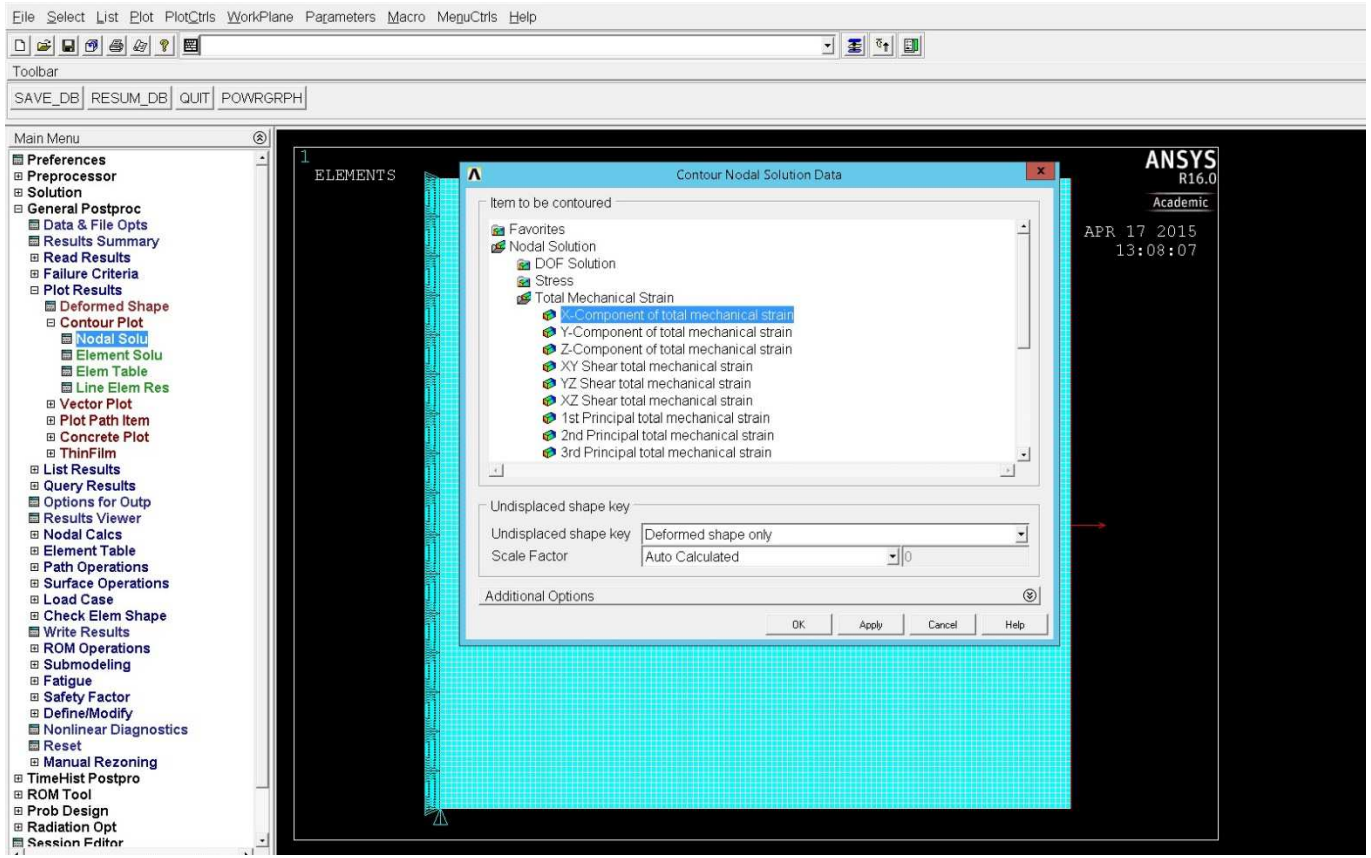
- Apply>Structural>Pressure>On Lines
  - Select right edge (in purple), click ok, load is N_{xx}/thickness
    - Negative because pressure is towards the object

Solve

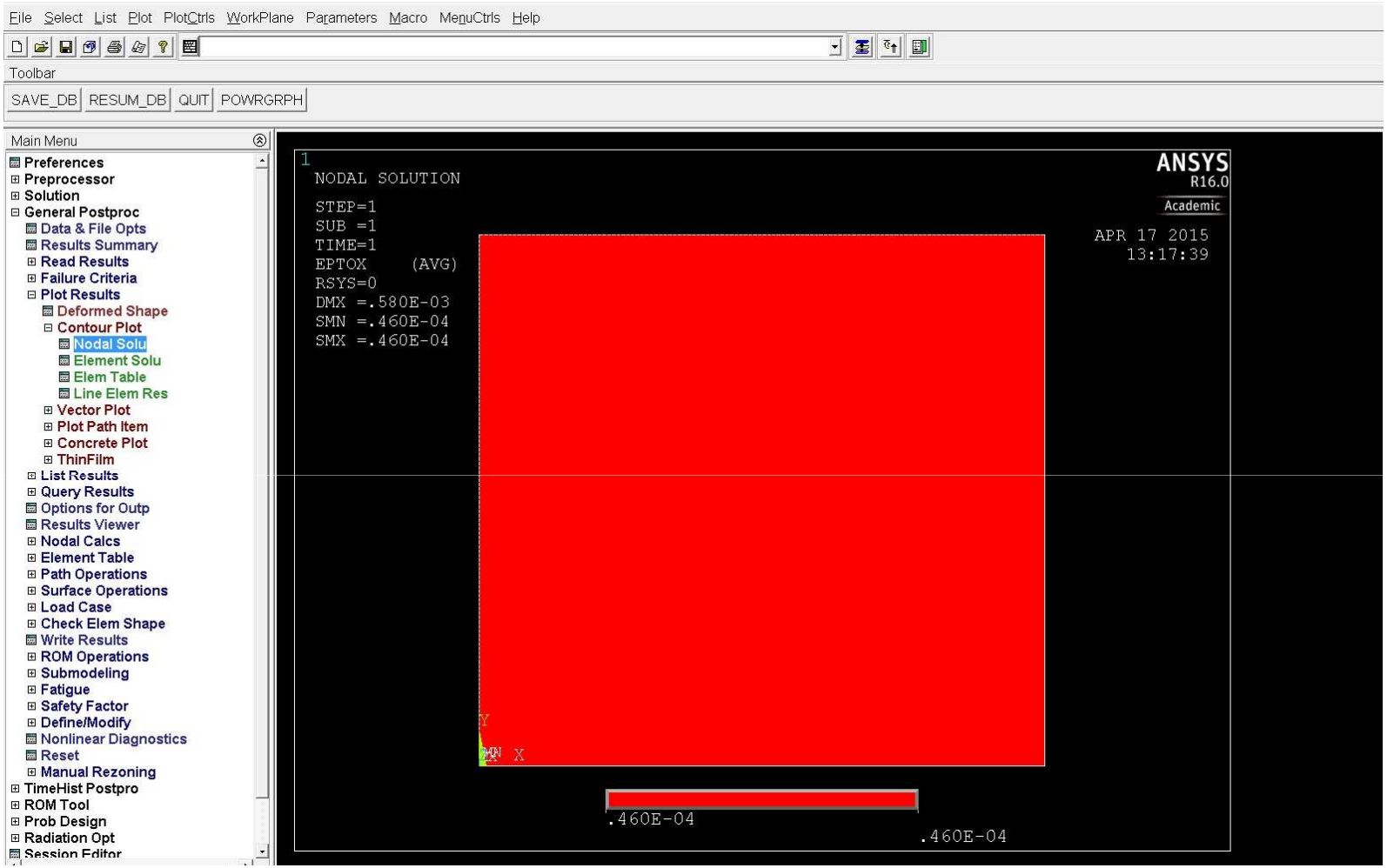


- Solution>Solve>Current LS, click ok
  - 'Solution is done!' Window opens, click close

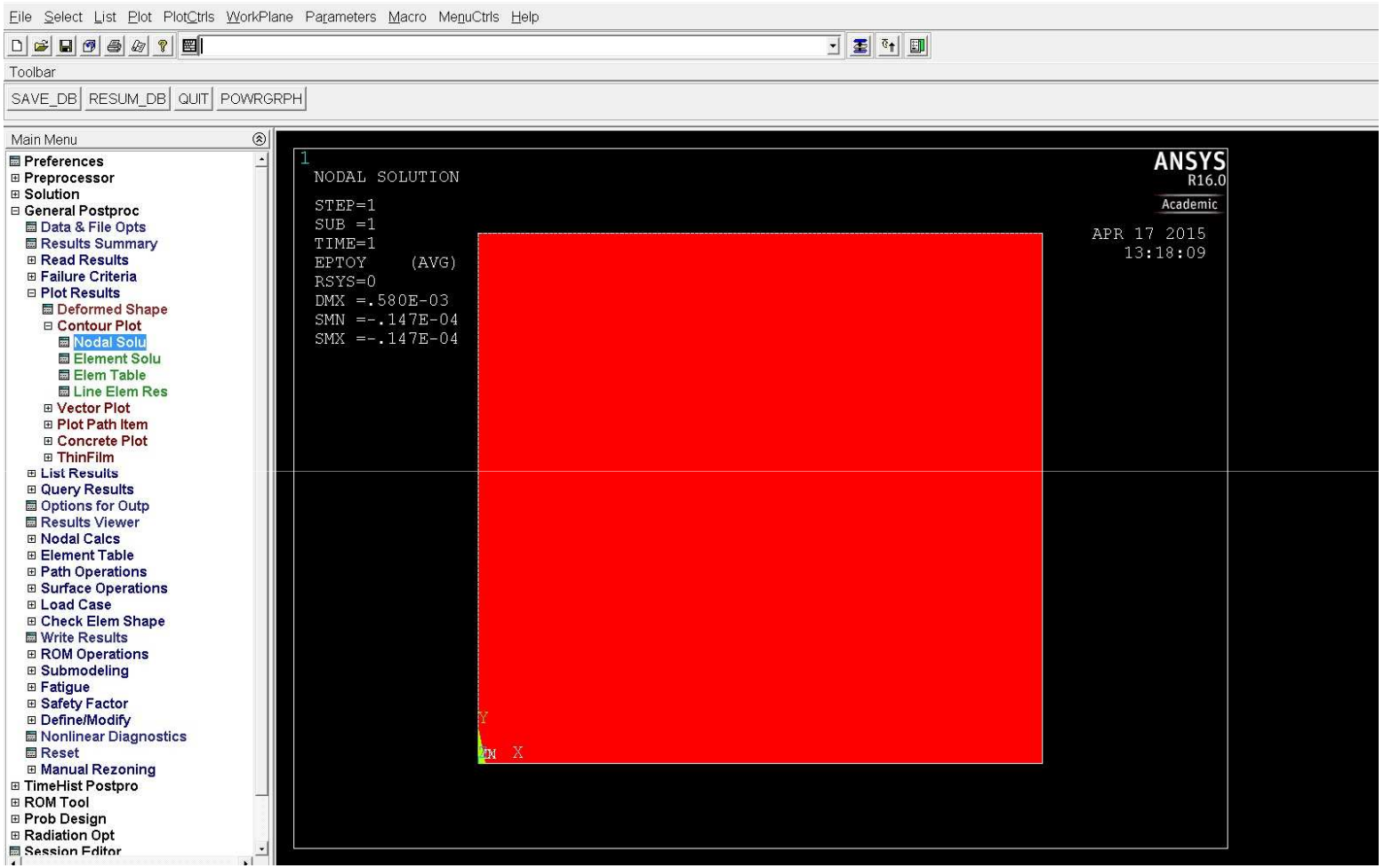
Post Processing



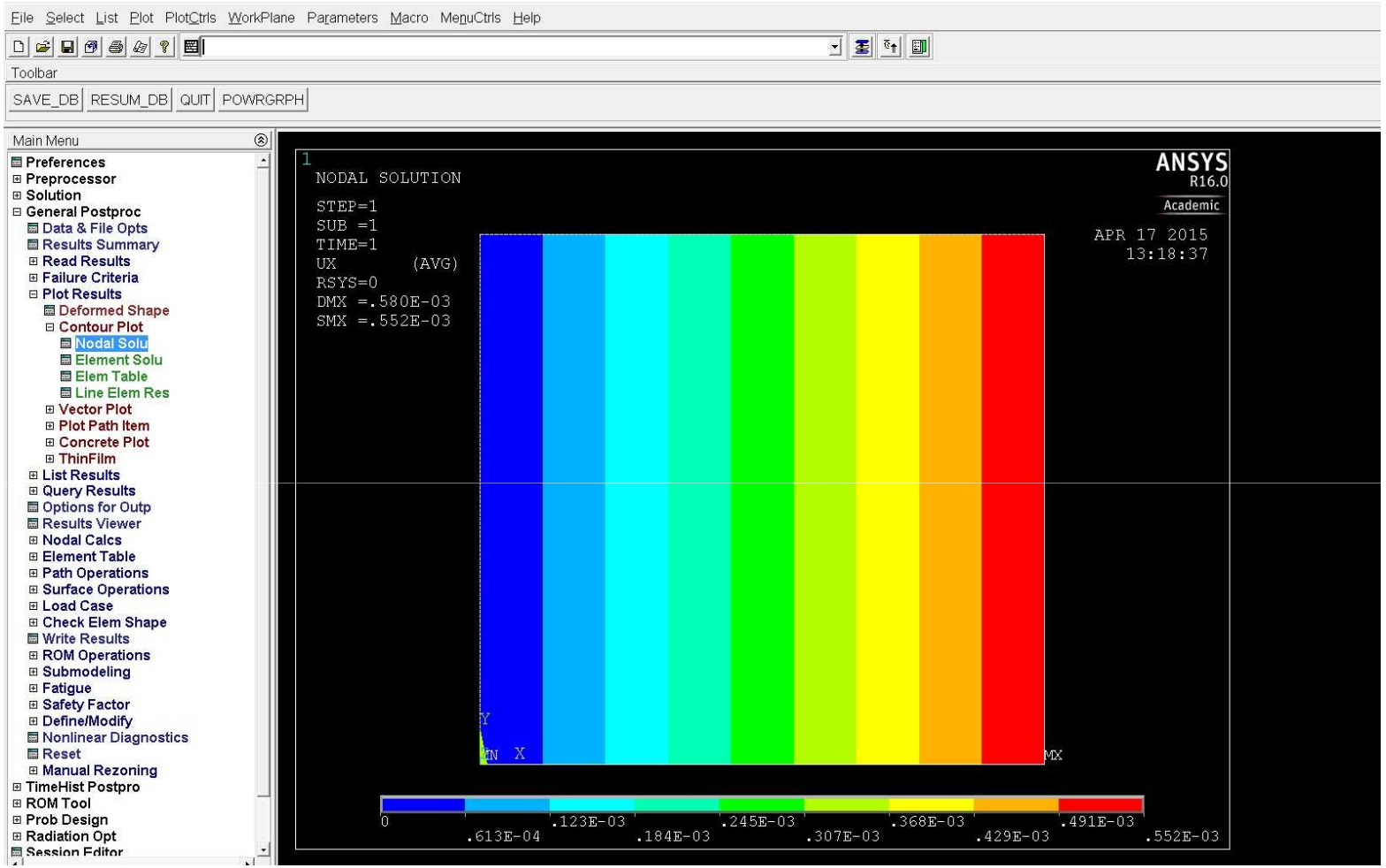
- General Postproc>Plot results>Contour Plot>Nodal Solu>Total Mechanical Strain>X-component



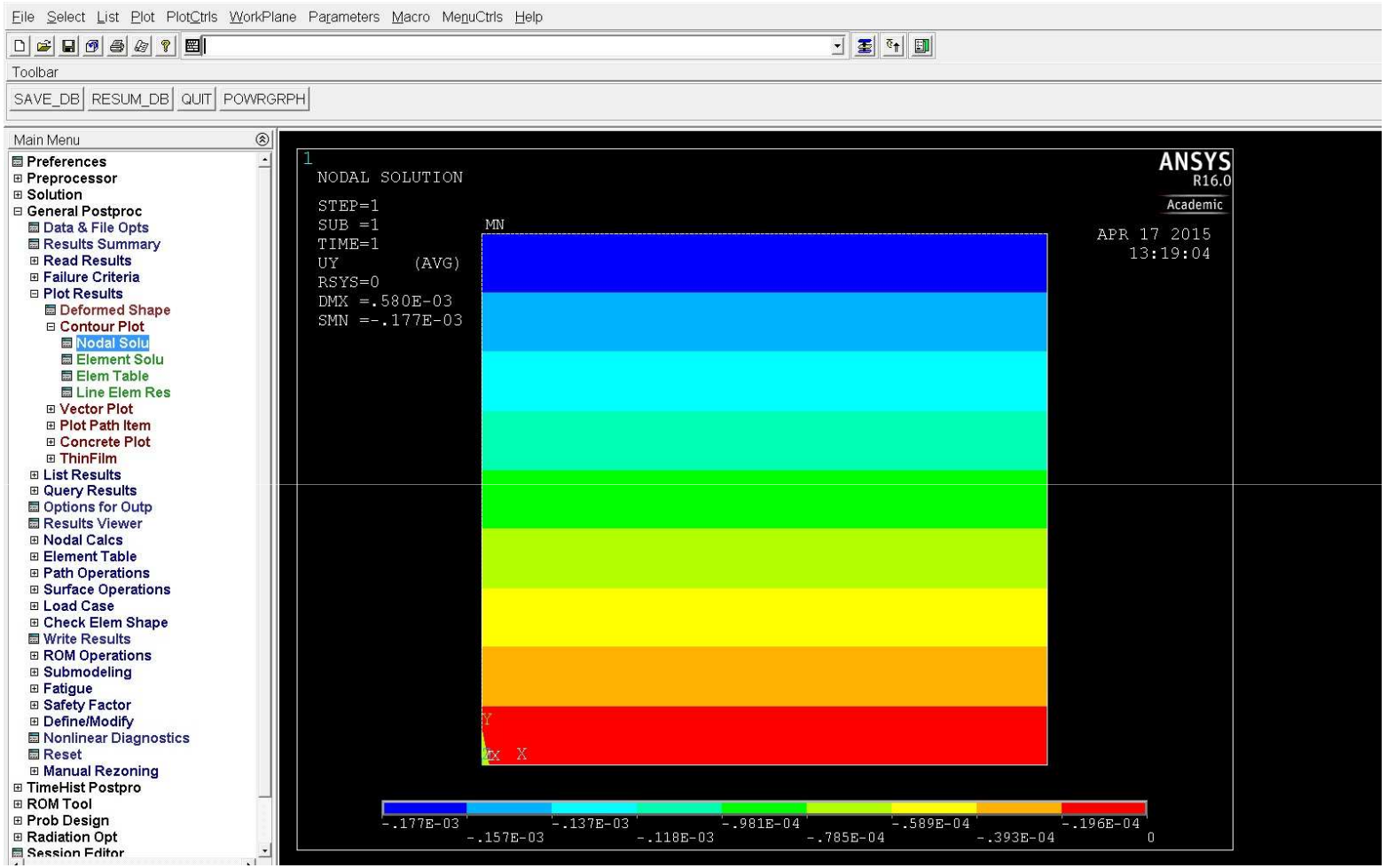
- Strain in x-direction = 46 microstrain
- Why is it constant?



- Strain in y-direction = -15 microstrain
- Does this agree with CLT and Workbench?



- Deformation in x-direction



- Deformation in y-direction

Savin Solution Ansys Workbench

Tangential stresses

20 mm hole (.787402 in)

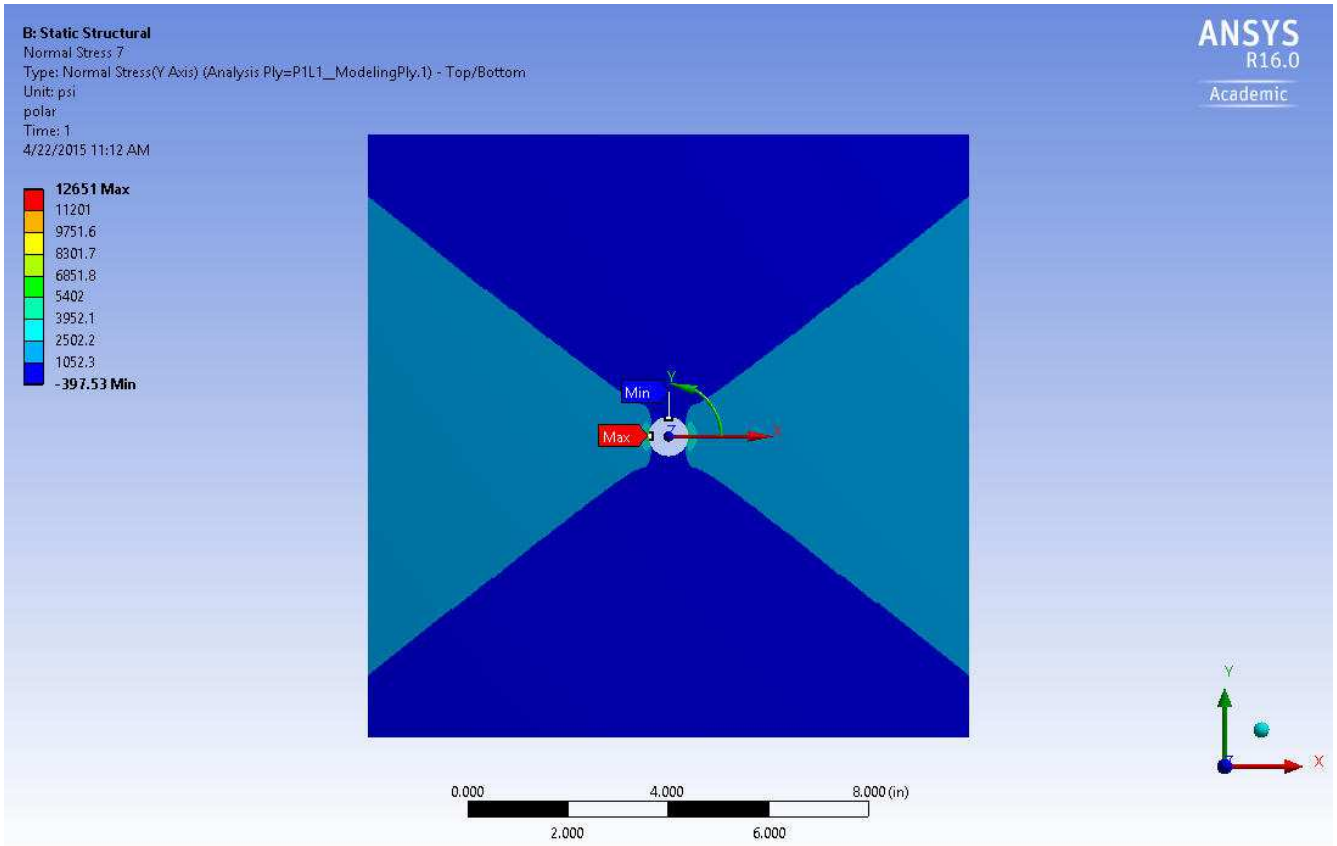
Polar Coordinate System

σ_{max} = 12651 psi

(0° defined by Ansys, 90° defined by text)

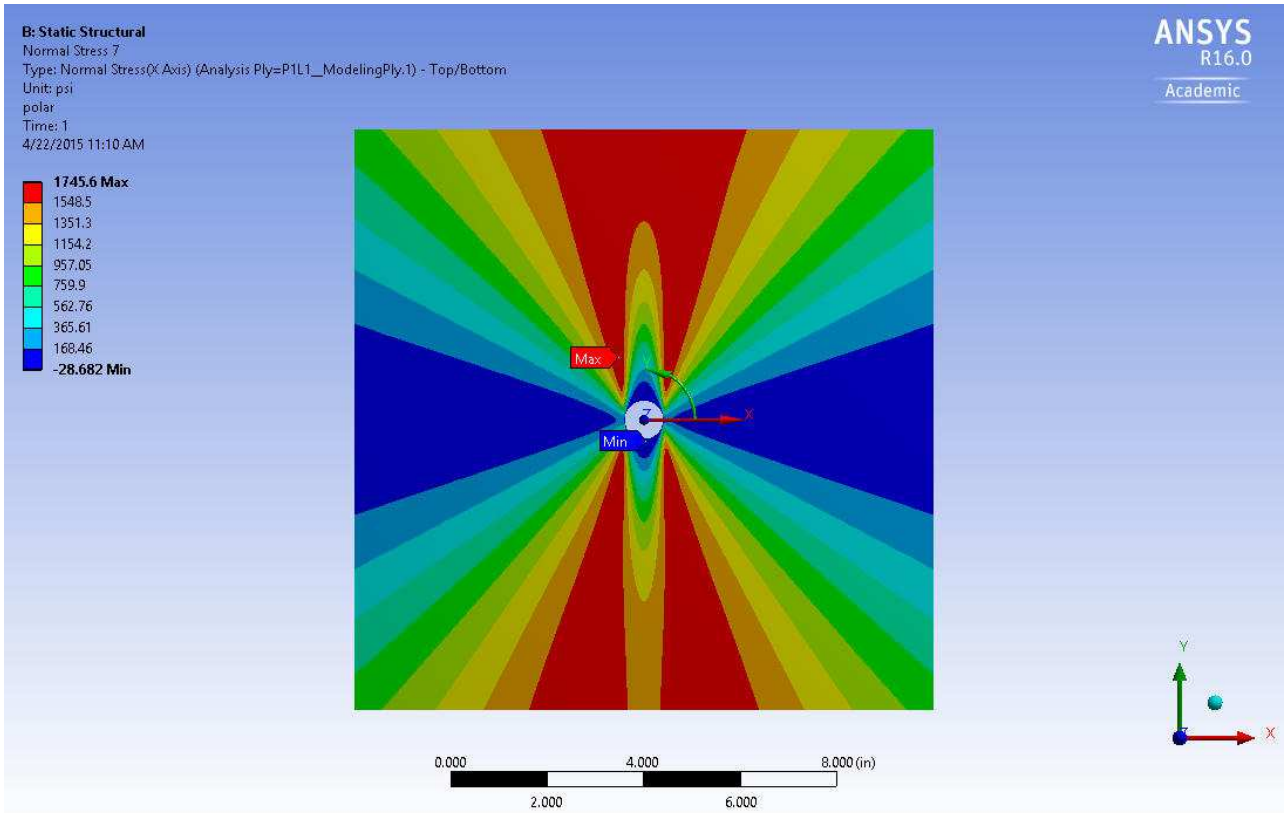
8 ply, unidirectional, fiber direction is global y-axis (top to bottom)

Alternative CS (solution, local, fiber, global)



Stresses normal to the hole

σ_{max} = 1745.6 psi



Savin Solution, Circular Hole

E_{11} = 25.3 msi

E_{22} = 1.22 msi

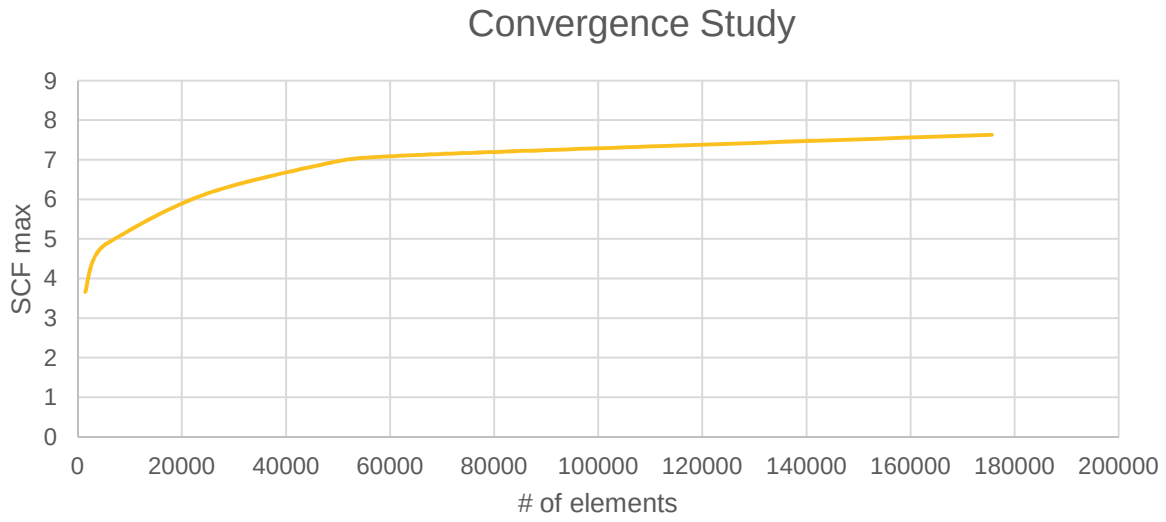
G_{12} = .687 msi

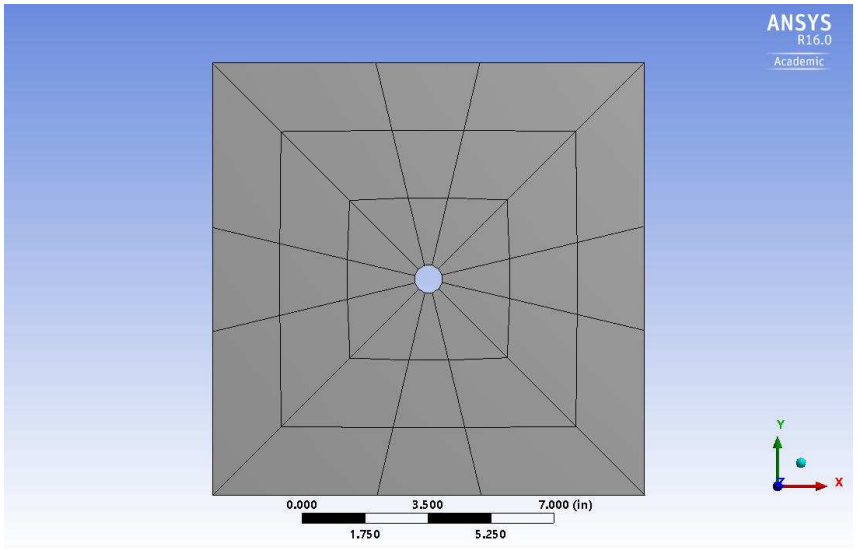
v_{12} = 0.32

Time taken to run also increases, topping out at 30 minutes with 175,000 elements

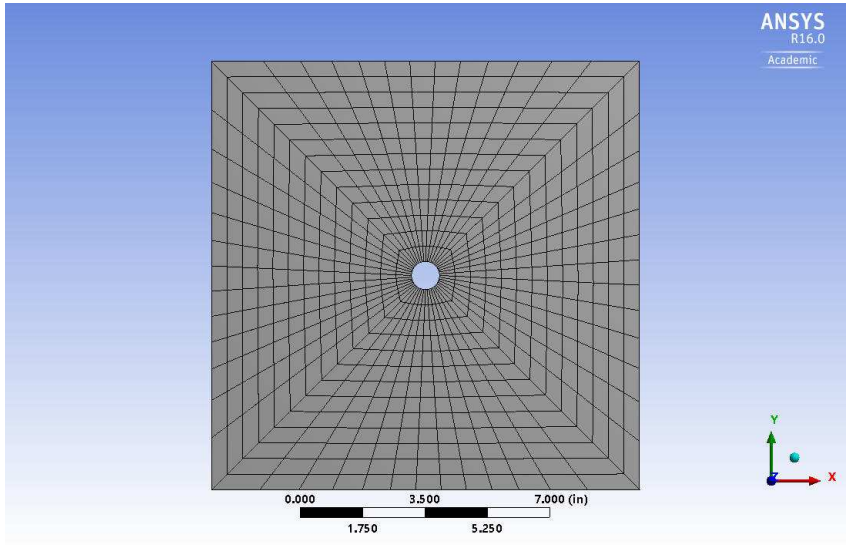
SCF_{max} = 7.63

Holes program predicts 7.7 for these properties





- Default mesh size from Ansys
  - σ_{max} = 2457
  - SCF = 1.4



- First mesh I used
  - σ_{max} = 6393
  - SCF = 3.7

90000 elements
~5 minutes to mesh
~5 minutes to solve

SCF_{max} = 7.2

1500 elements around circumference of hole