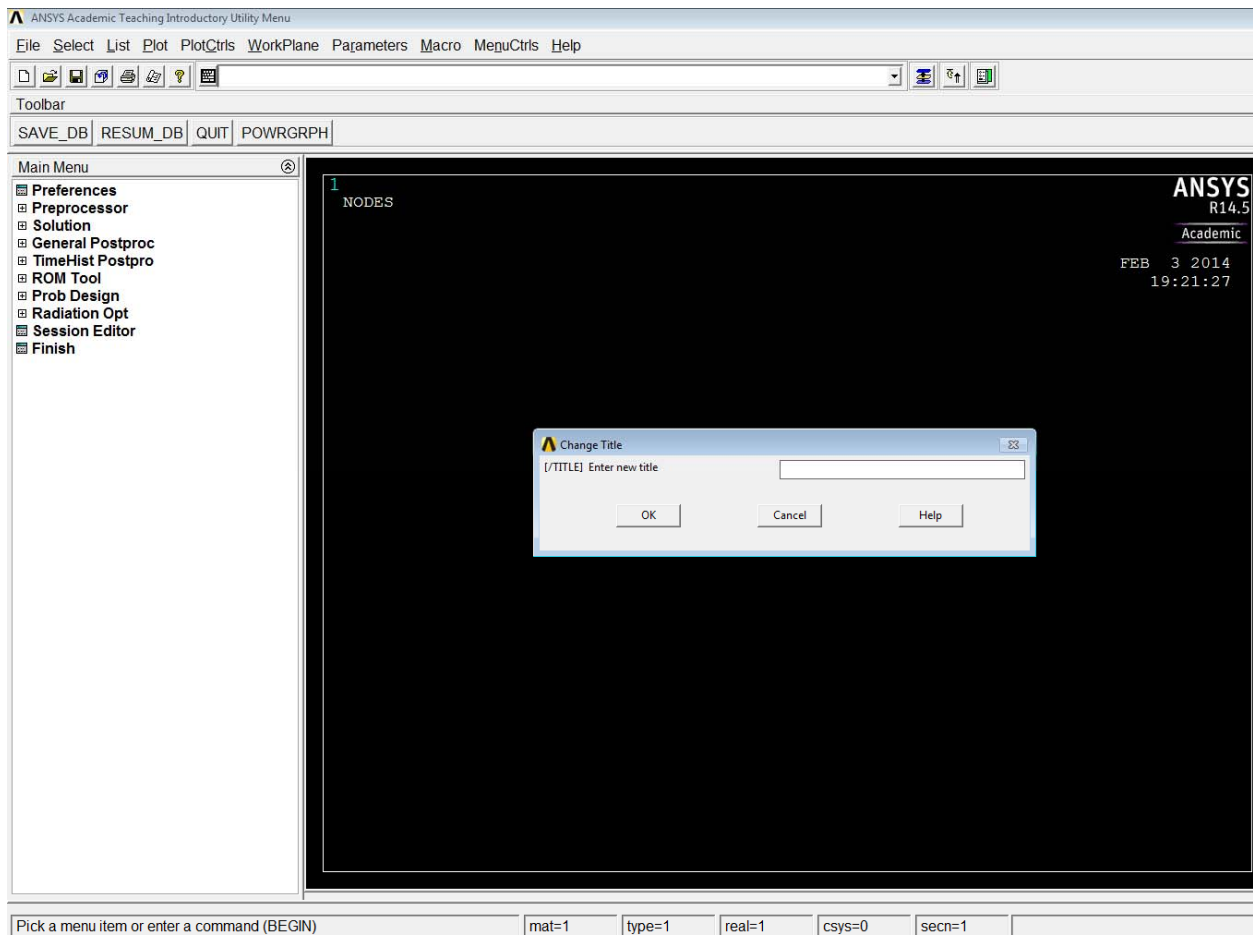
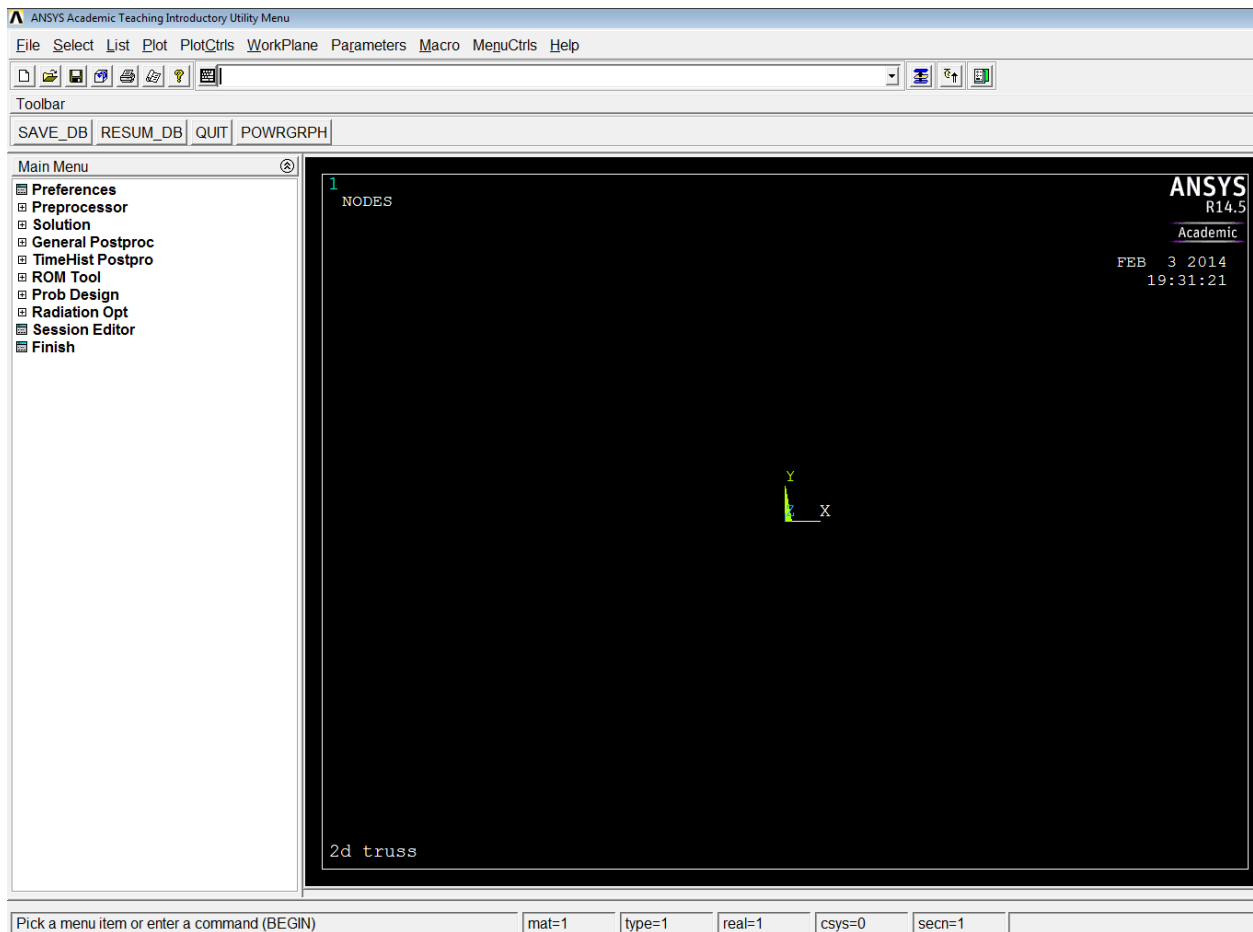


1. Title: Utility Menu> File> Change Title



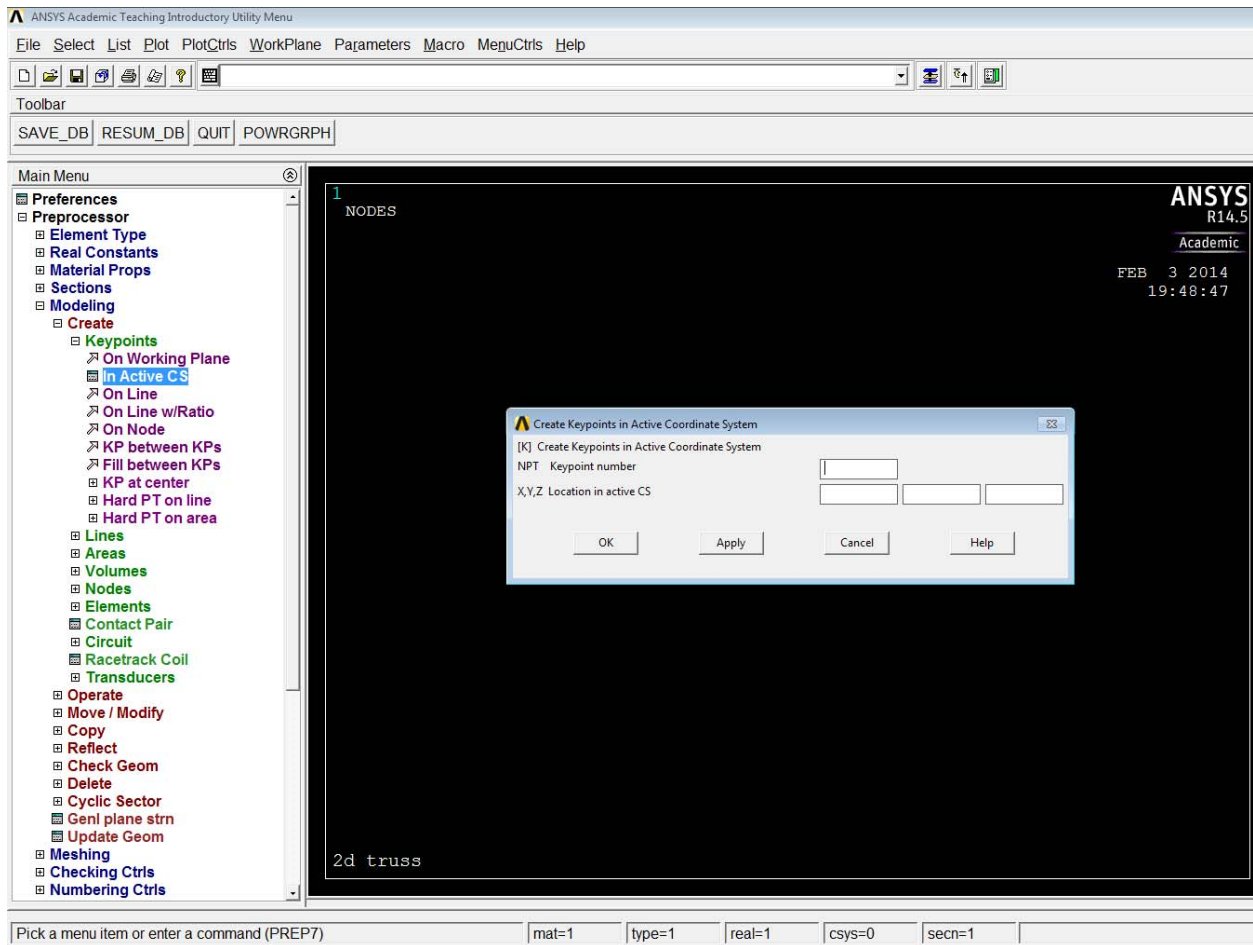
2. Enter title, Ok and then Utility Menu> Plot> Replot



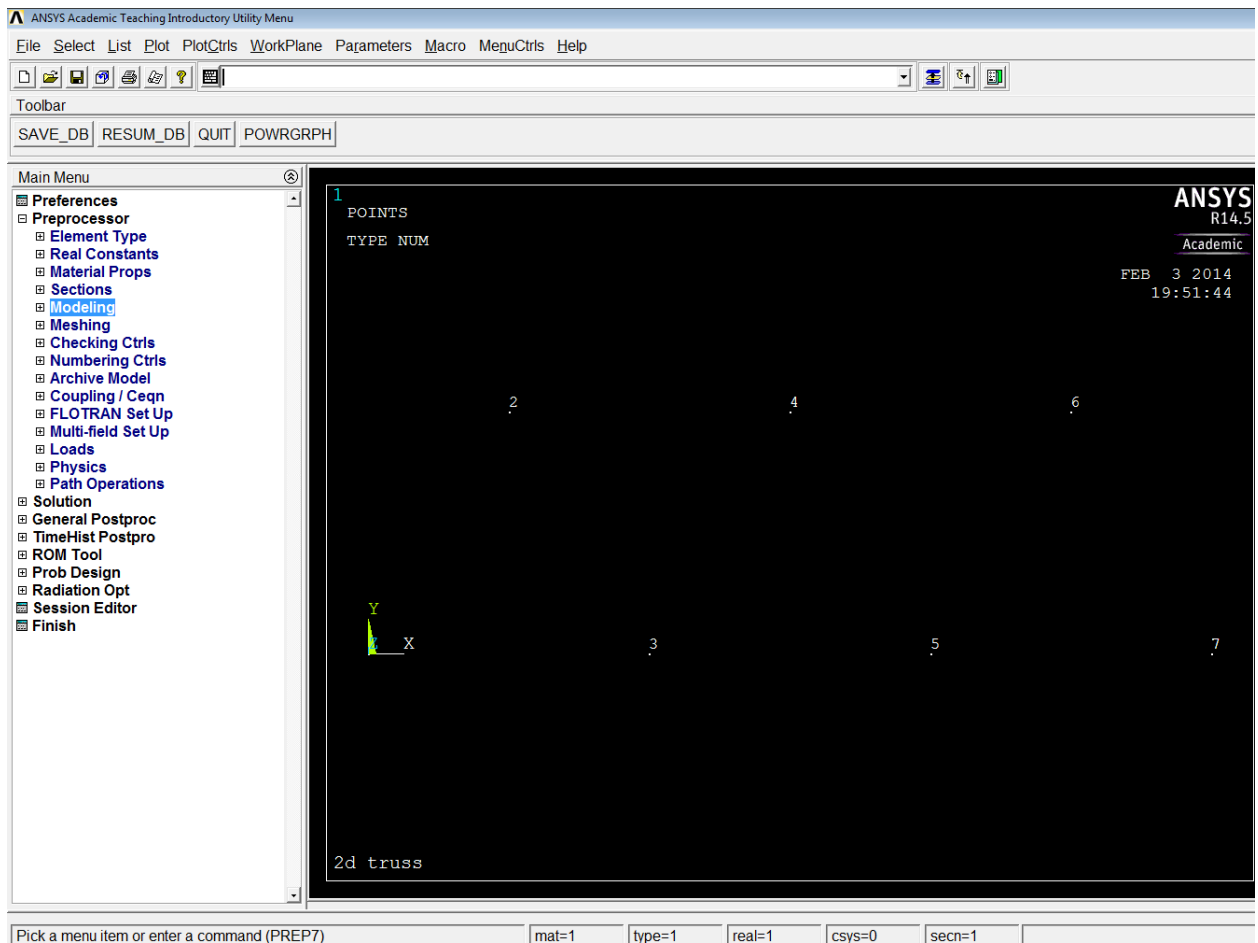
The following keypoints are to be entered next:

Keypoint No.	X Coordinate	Y Coordinate
1	0	0
2	1800	3118
3	3600	0
4	5400	3118
5	7200	0
6	9000	3118
7	10800	0

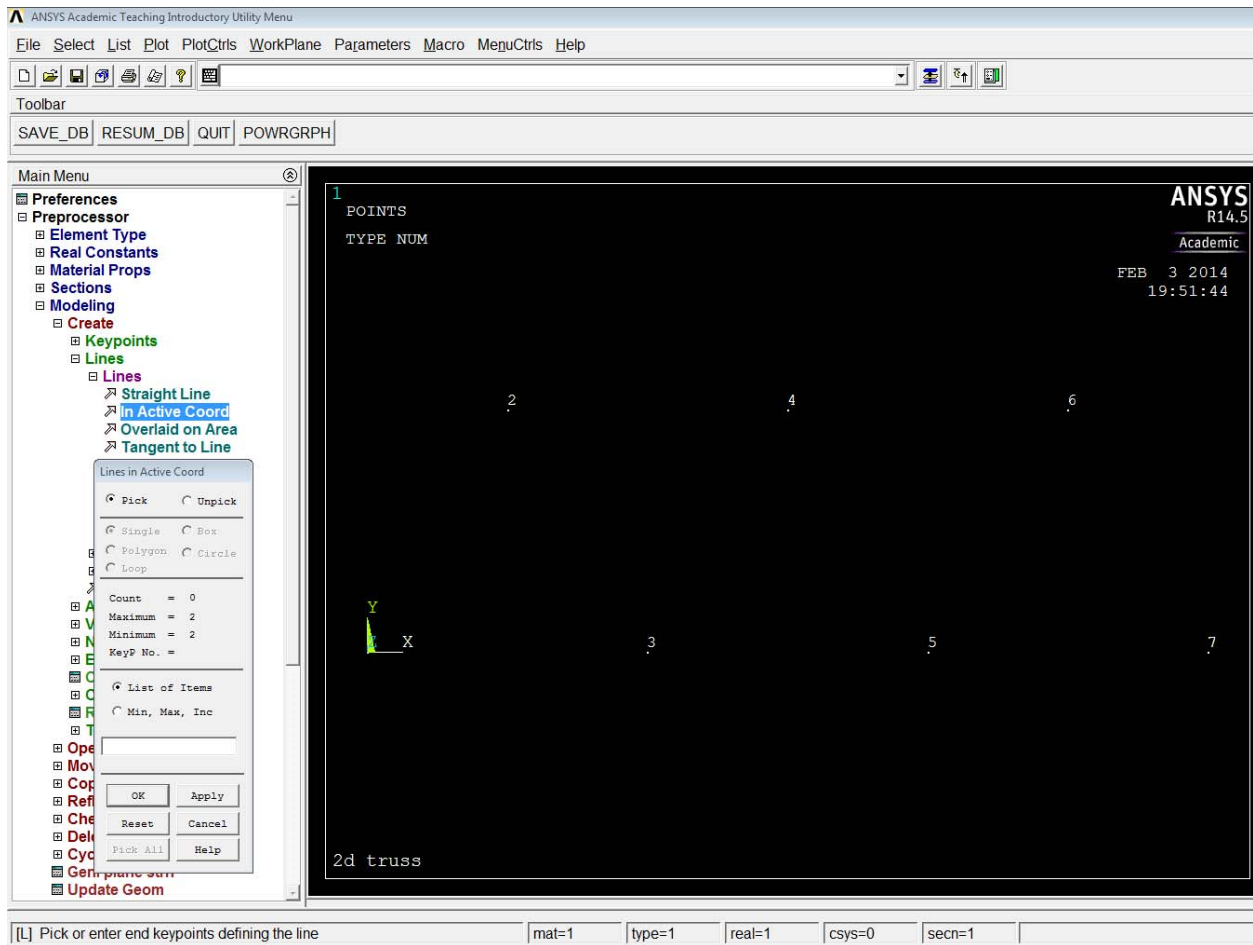
3. Keyword selection: Main menu> Preprocessor> Modeling> Create> Keypoints> In Active CS



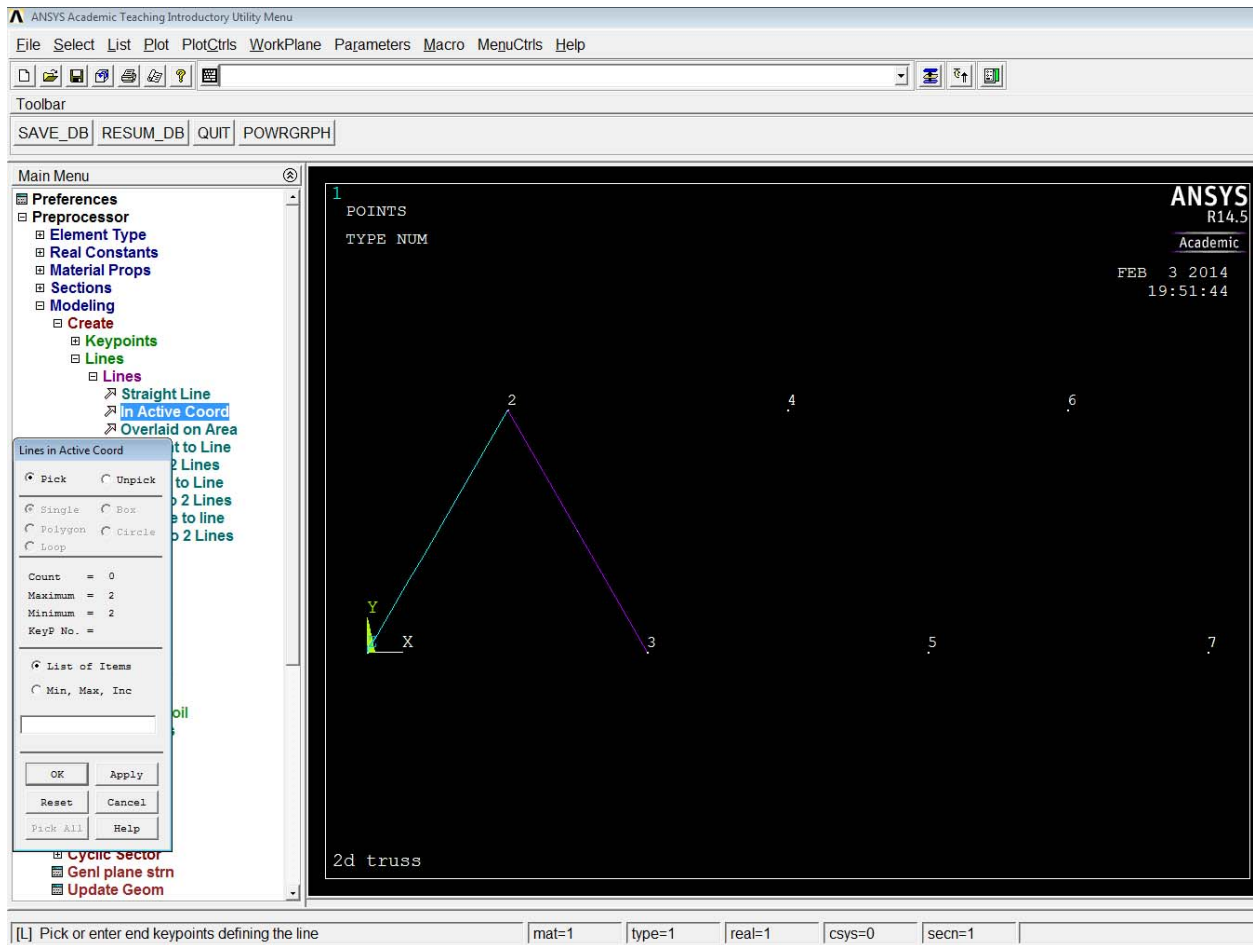
4. Enter the coordinates and press apply until done



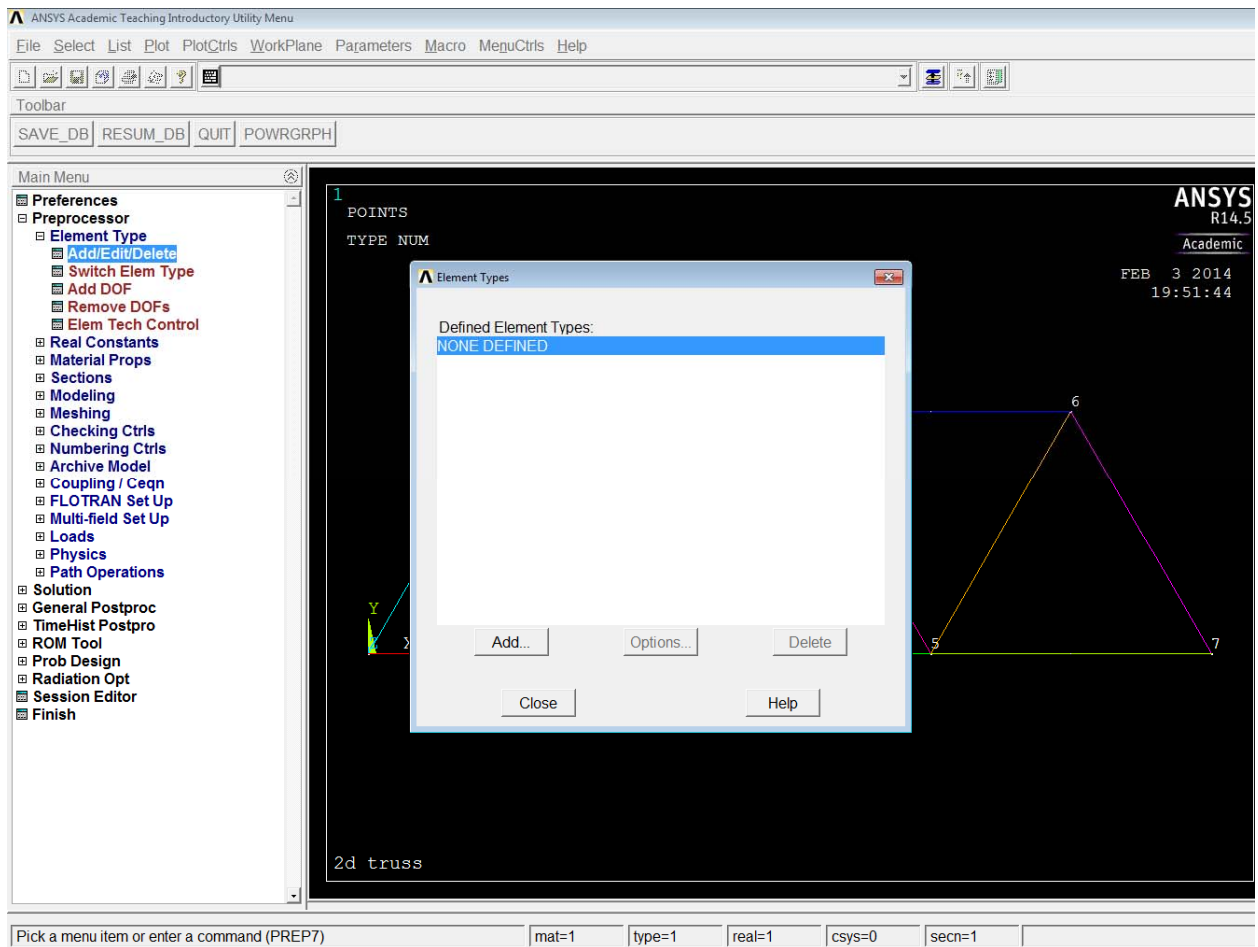
5. Drawing lines: Preprocessor> Modeling> Create> Lines> Lines> In Active Coord



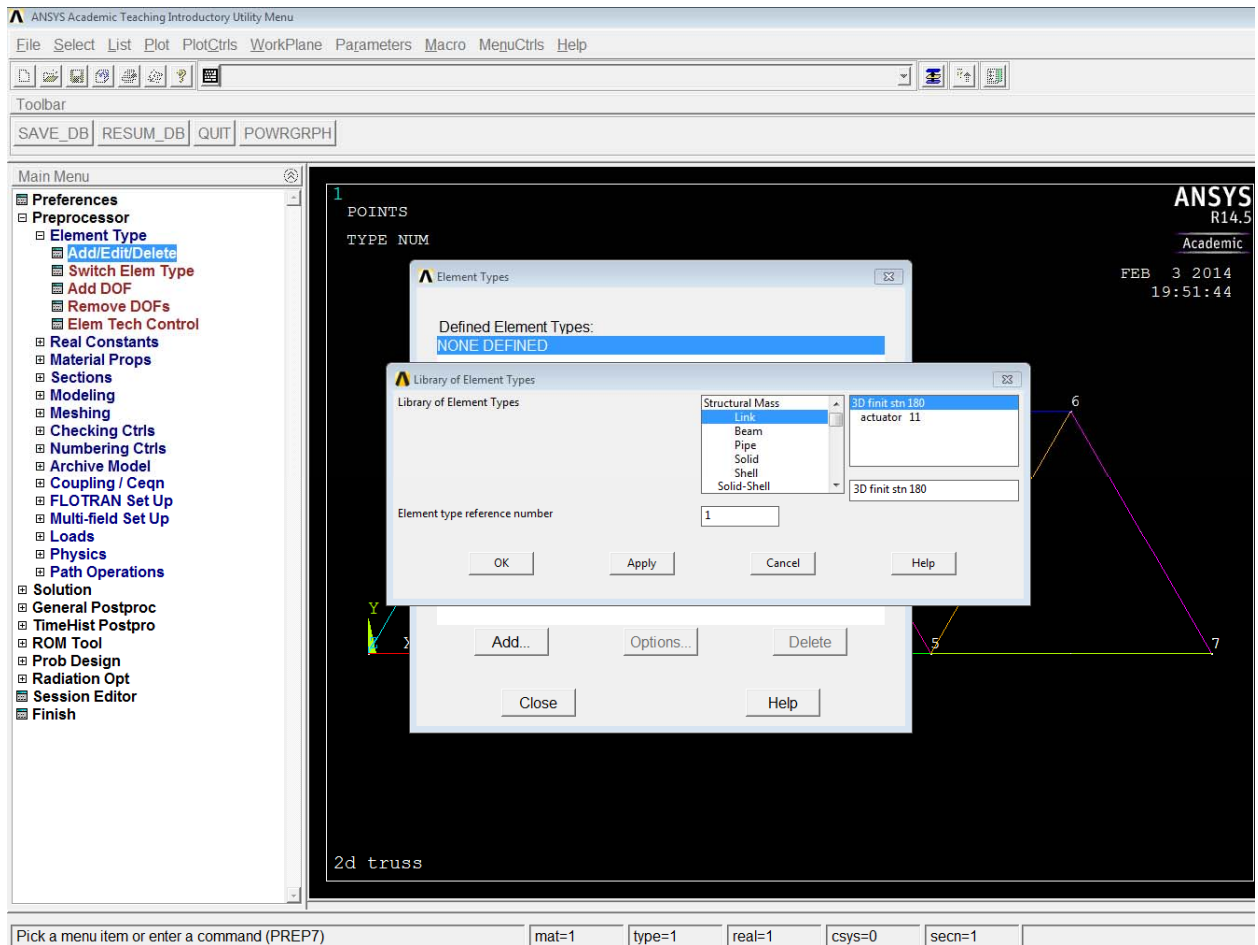
6. Start picking keypoints and press ok once done.



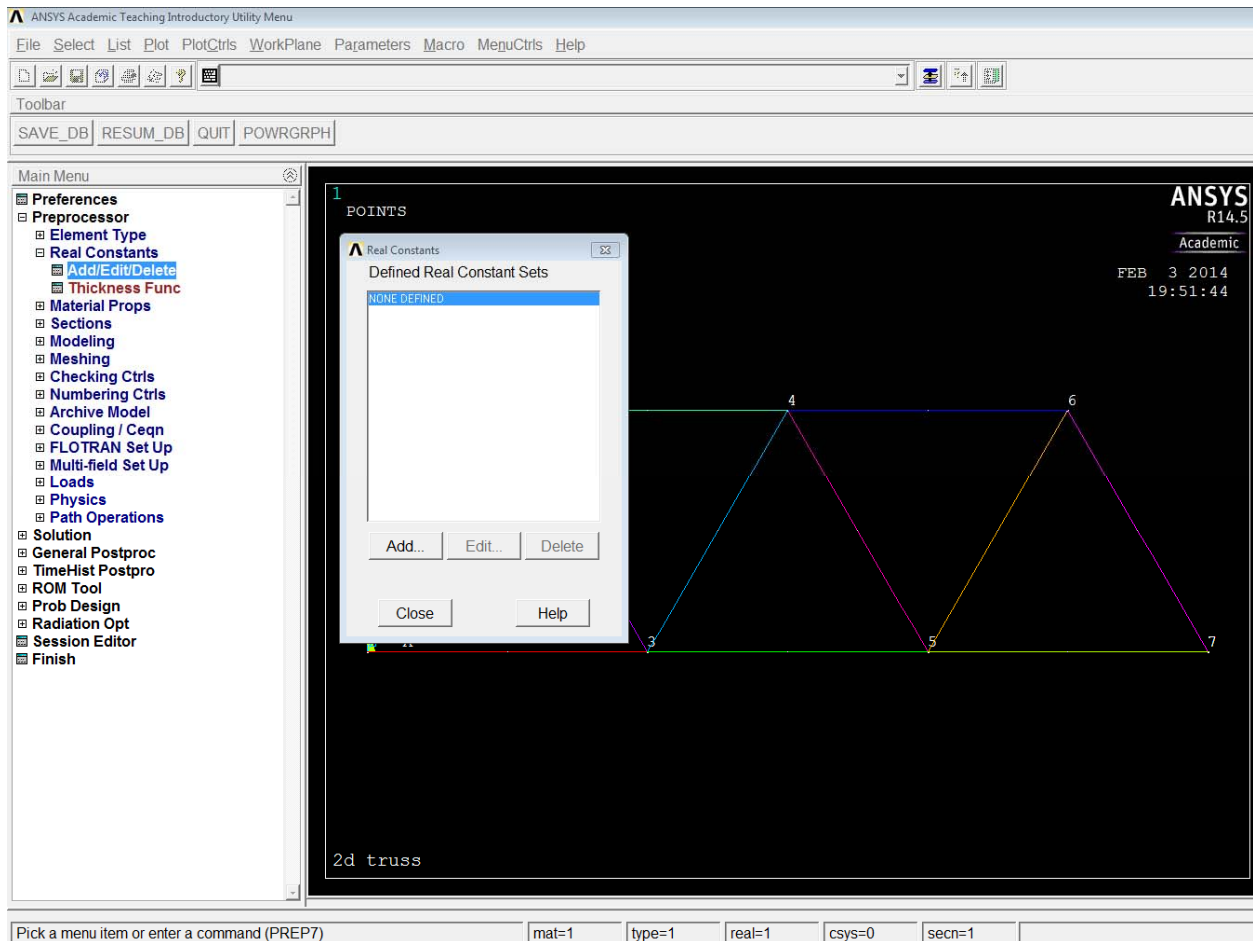
7. Defining elements: Preprocessor> Element type> Add/ Edit/ Delete



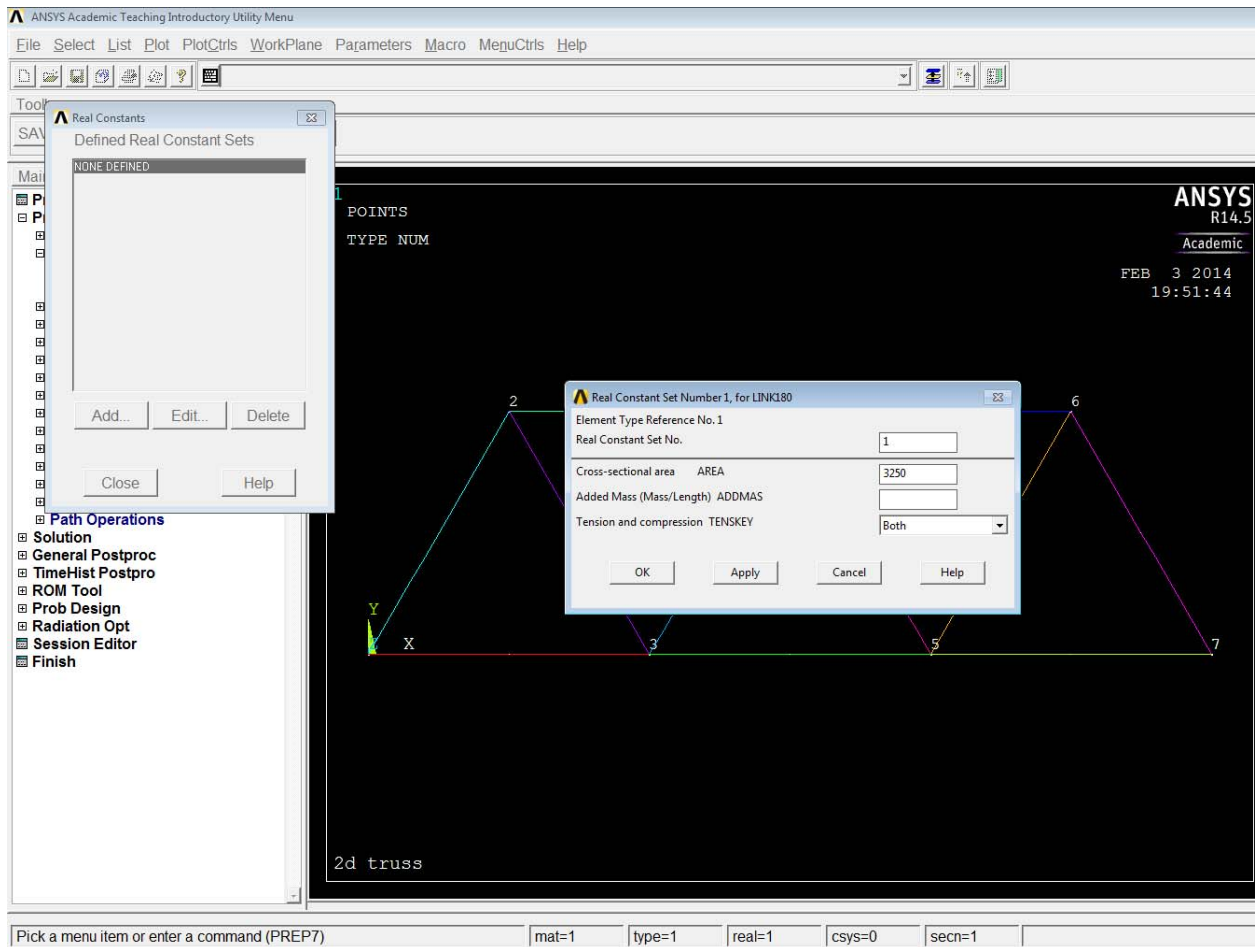
8. Choose Structural mass> Link> 3D finit stn 180 and then ok



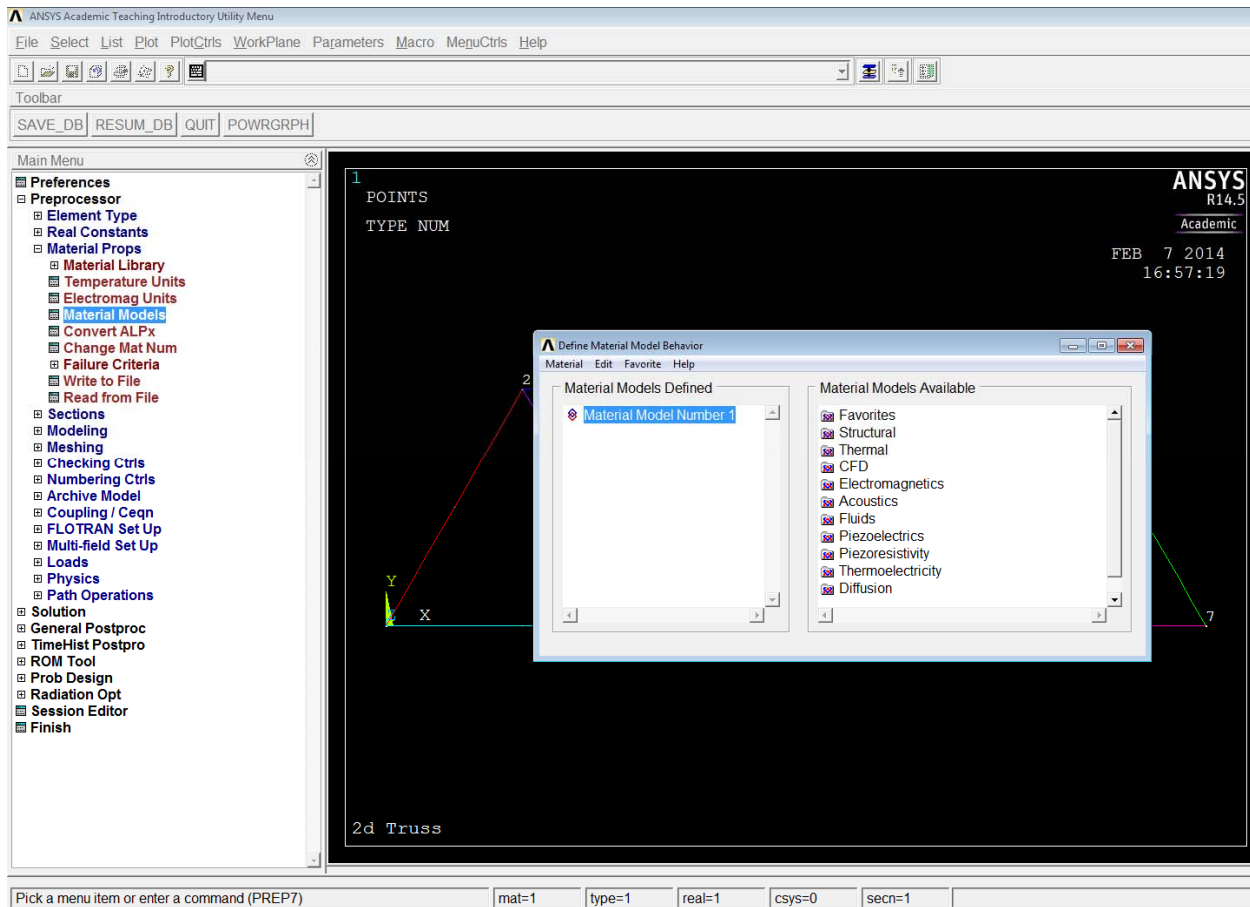
9. Real constants: Preprocessor> Real Constants> Add/ Edit/ Delete and then click Add



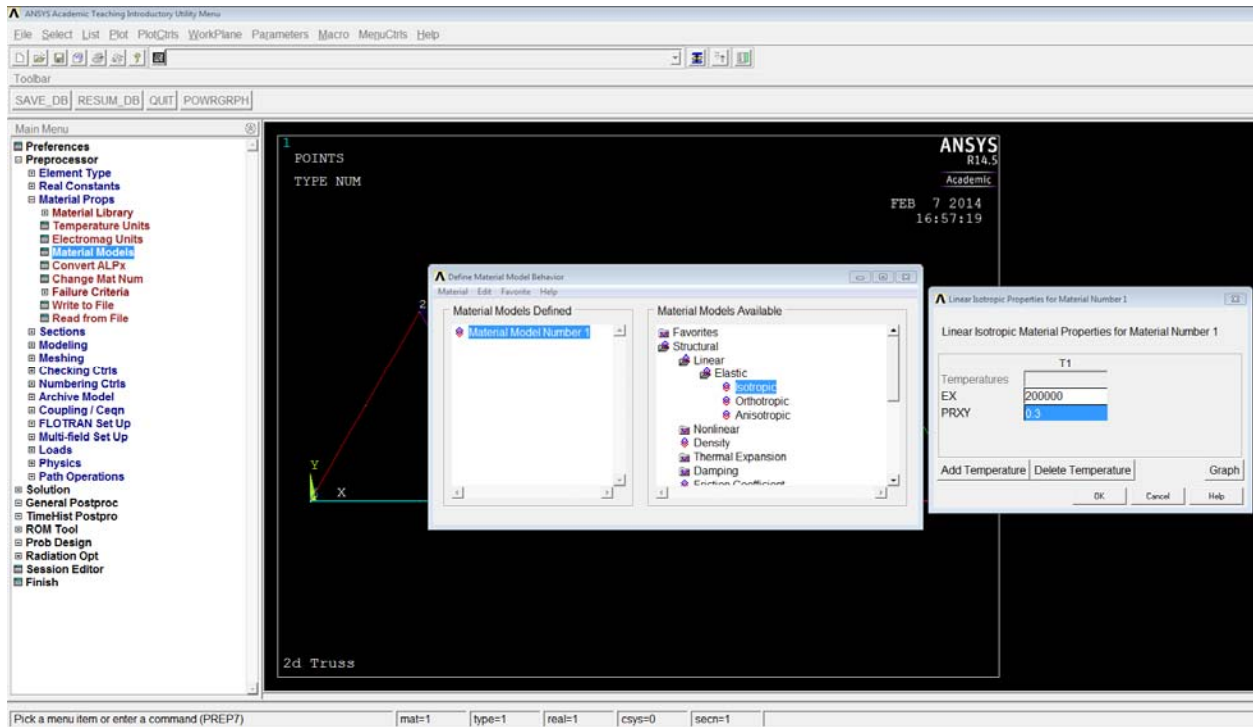
10. Then Ok. Enter 3250 as area and press Ok. After that close the real constant box.



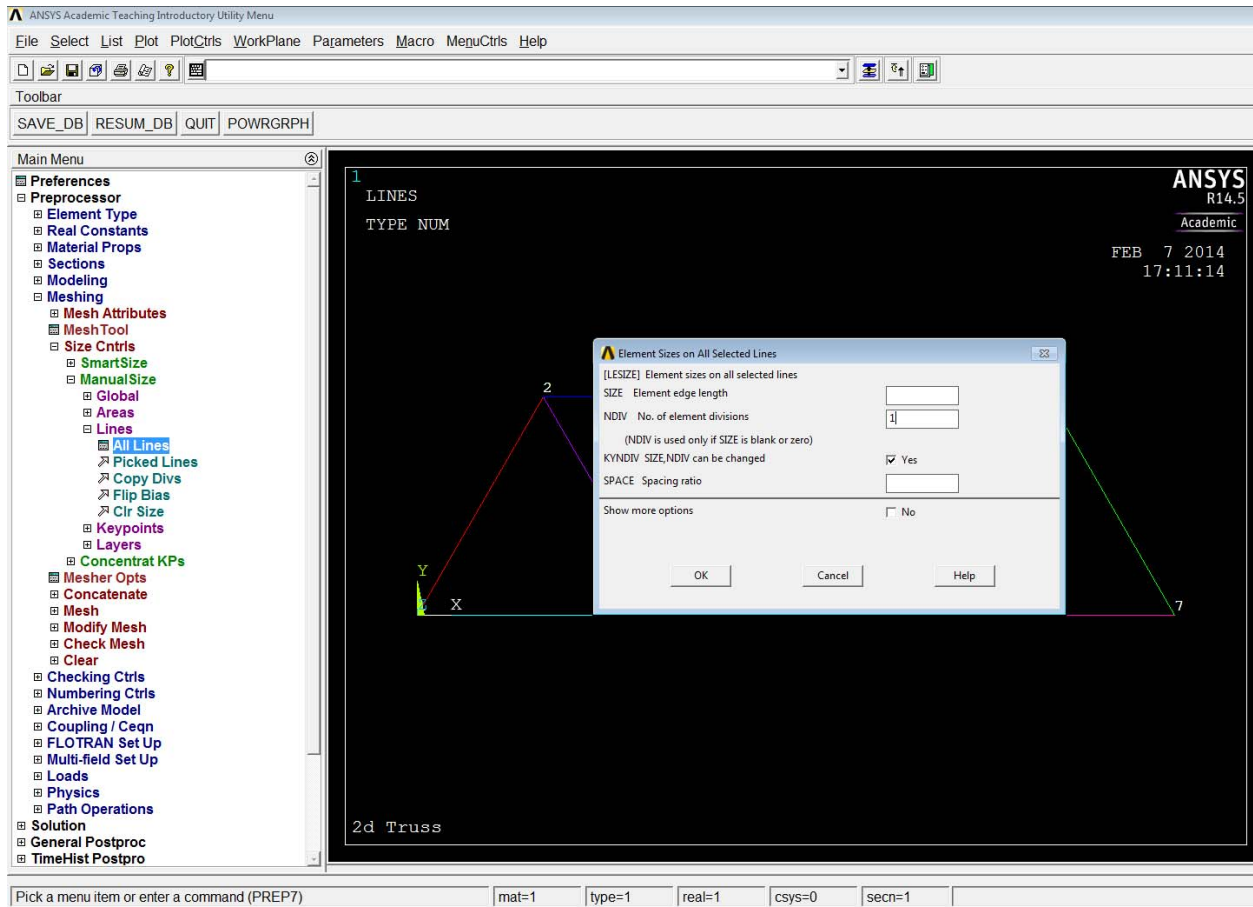
11. Material properties: Preprocessor> Material props> Material Models



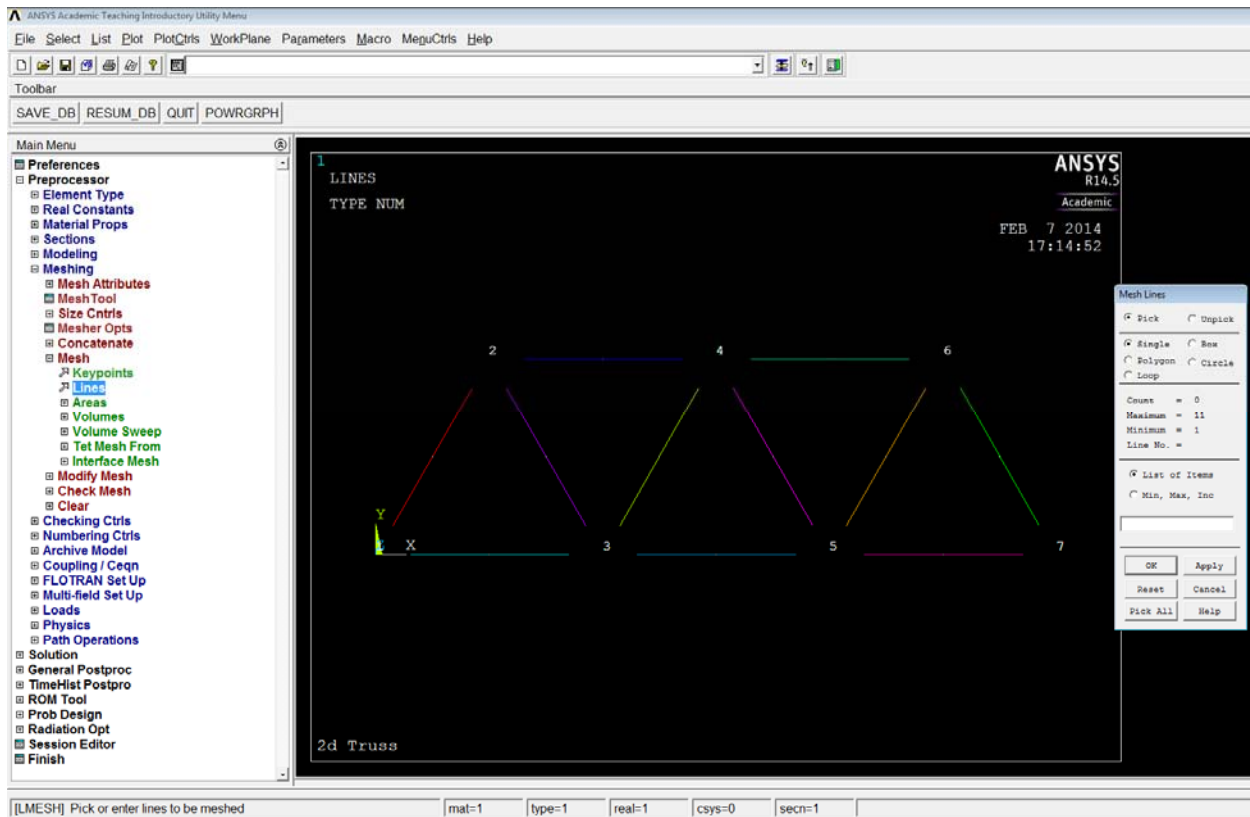
12. Structural> Linear> Elastic> Isotropic. Enter 200000 as Ex and 0.3 as PRxy. Then select ok and close material model window.



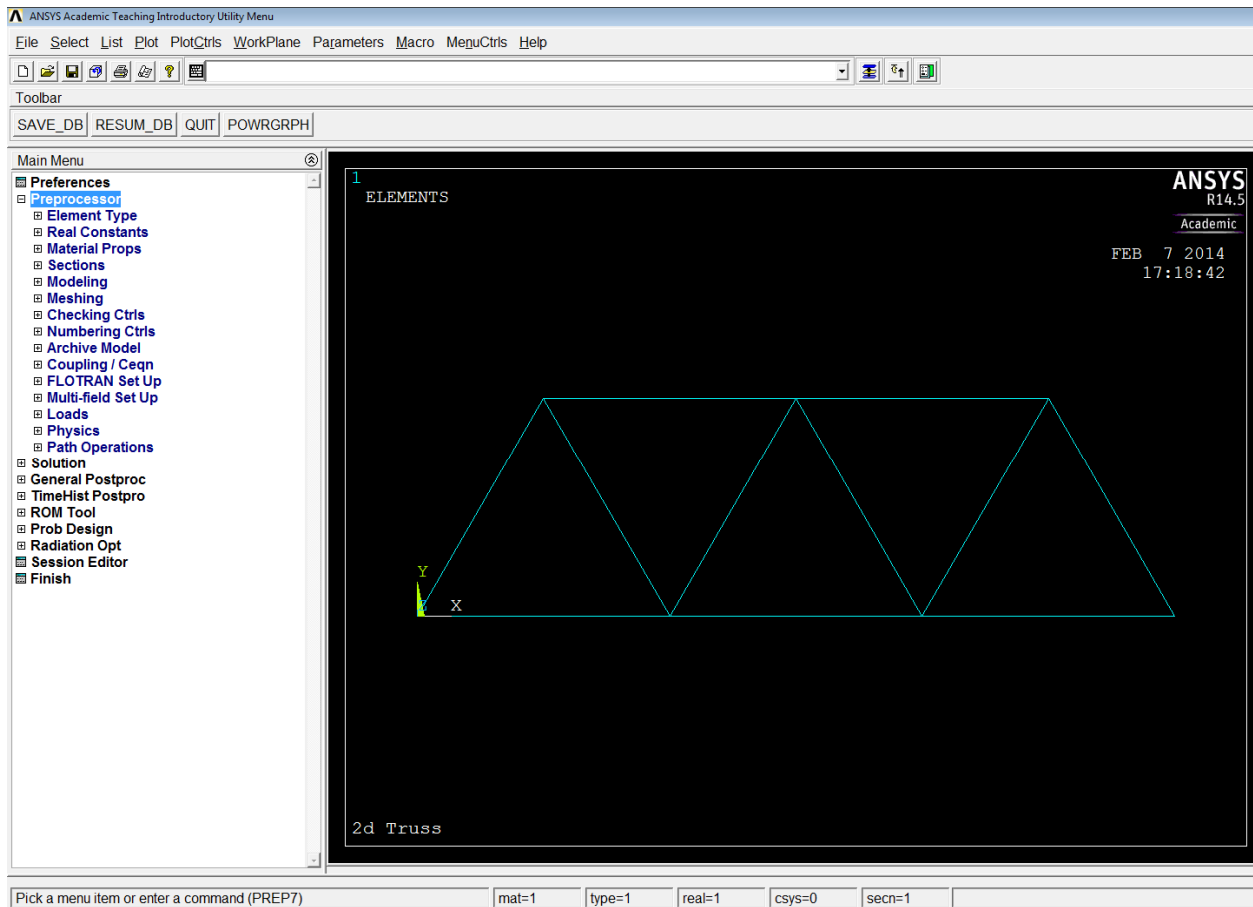
13. Meshing: Preprocessor> Meshing> Size Cntrls> ManualSize> Lines> All Lines. Enter 1 in the NDIV field and then select ok.



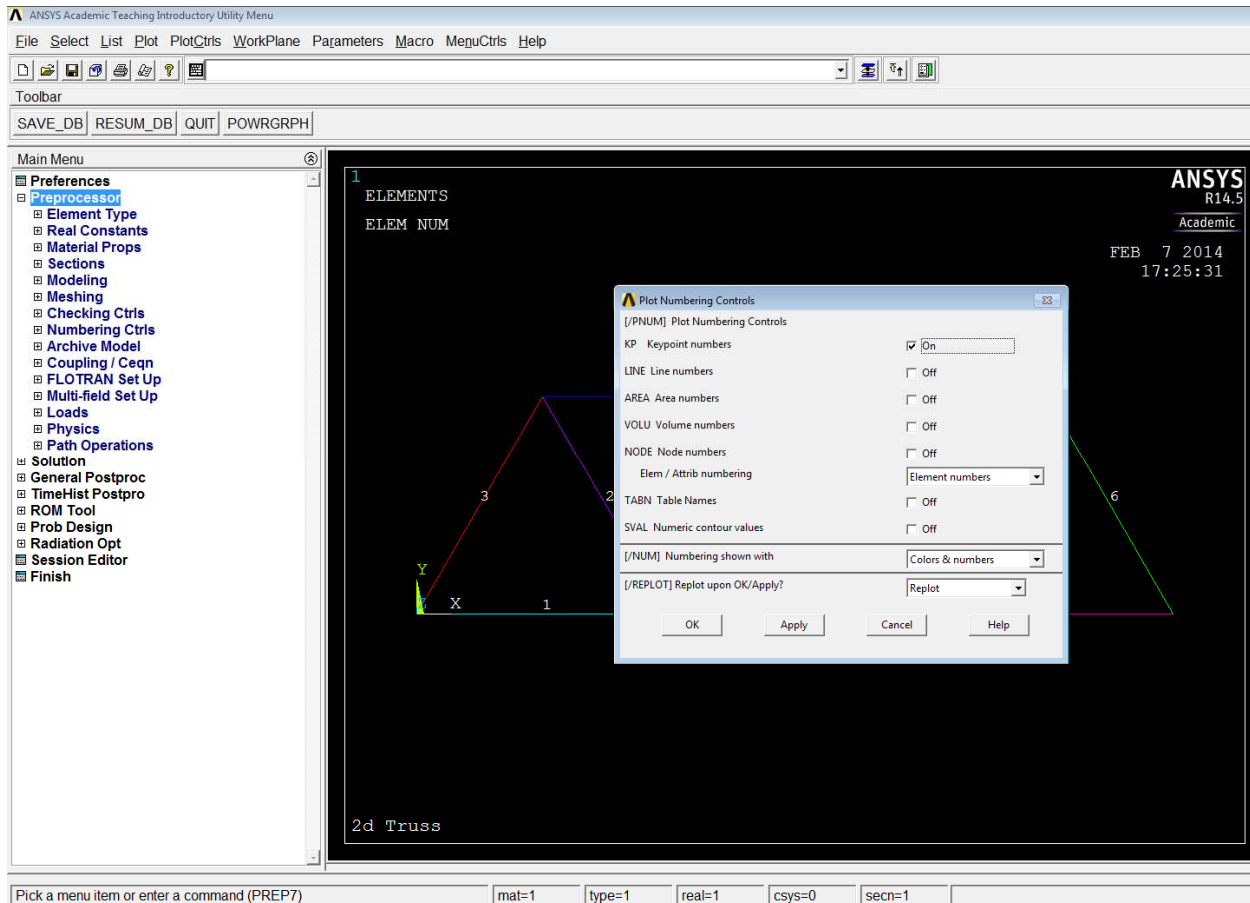
14. Preprocessor> Meshing> Mesh> Lines.



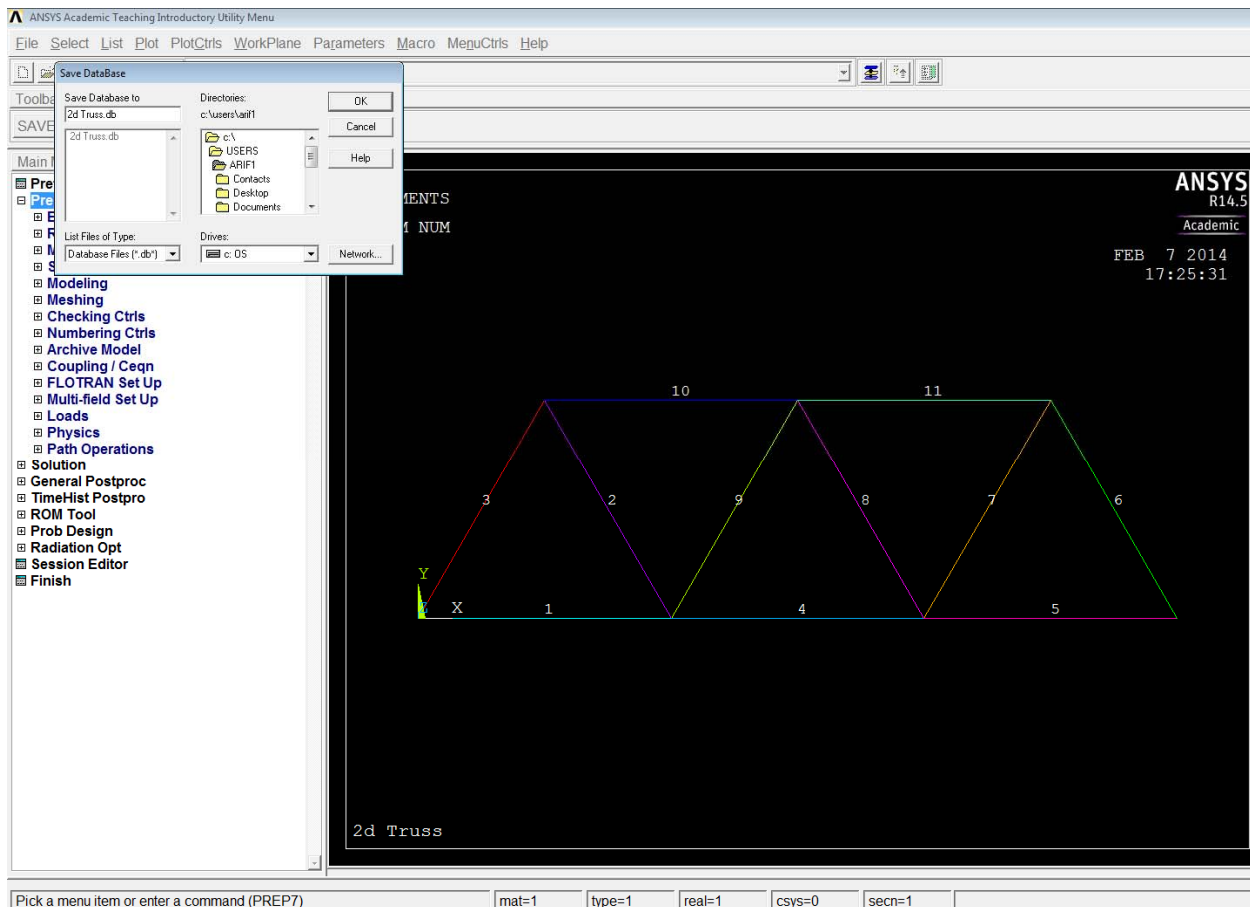
15. Select Pick All and the screen should look like this.



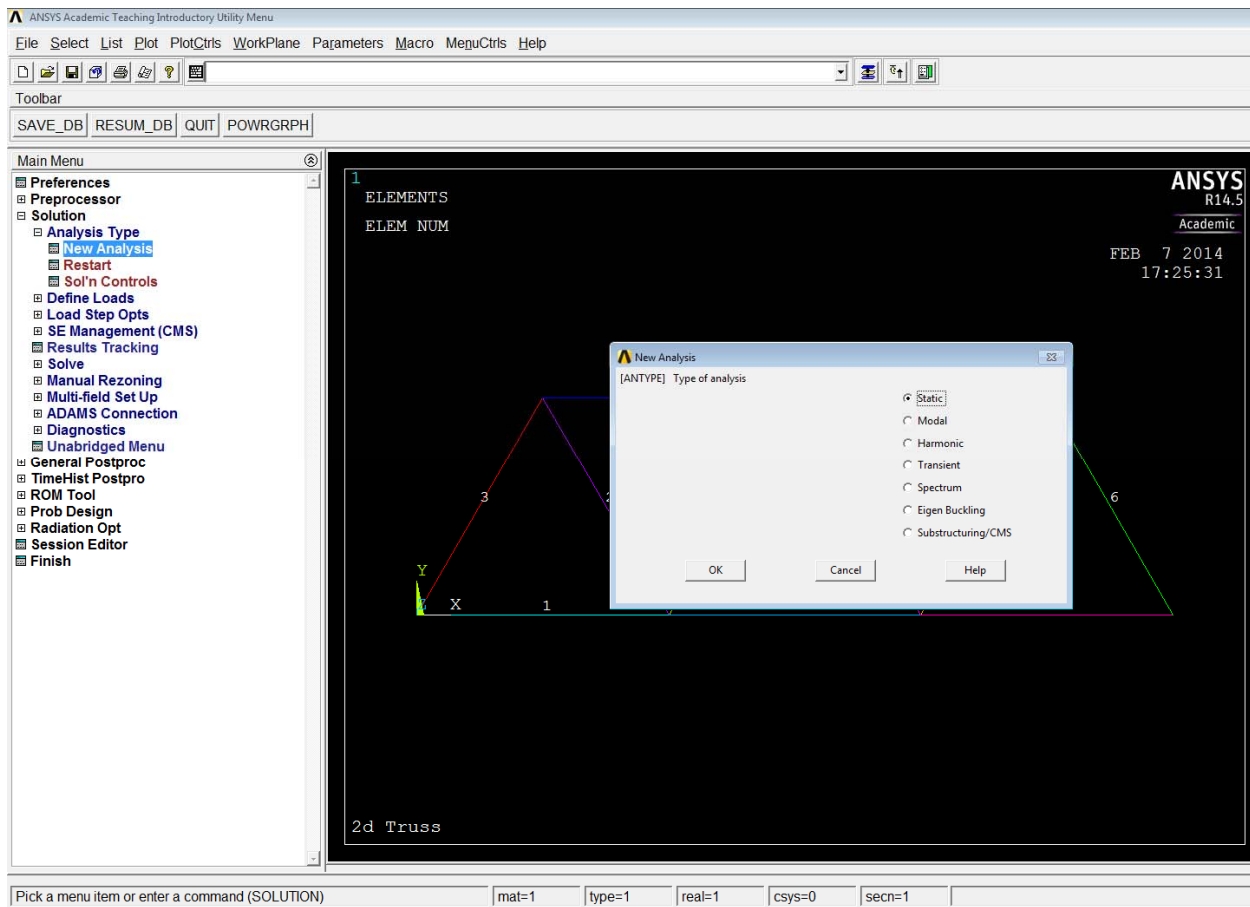
16. Keypoint numbering: Utility Menu> PlotCtrls> Numbering. Select as shown in the figure and select ok.



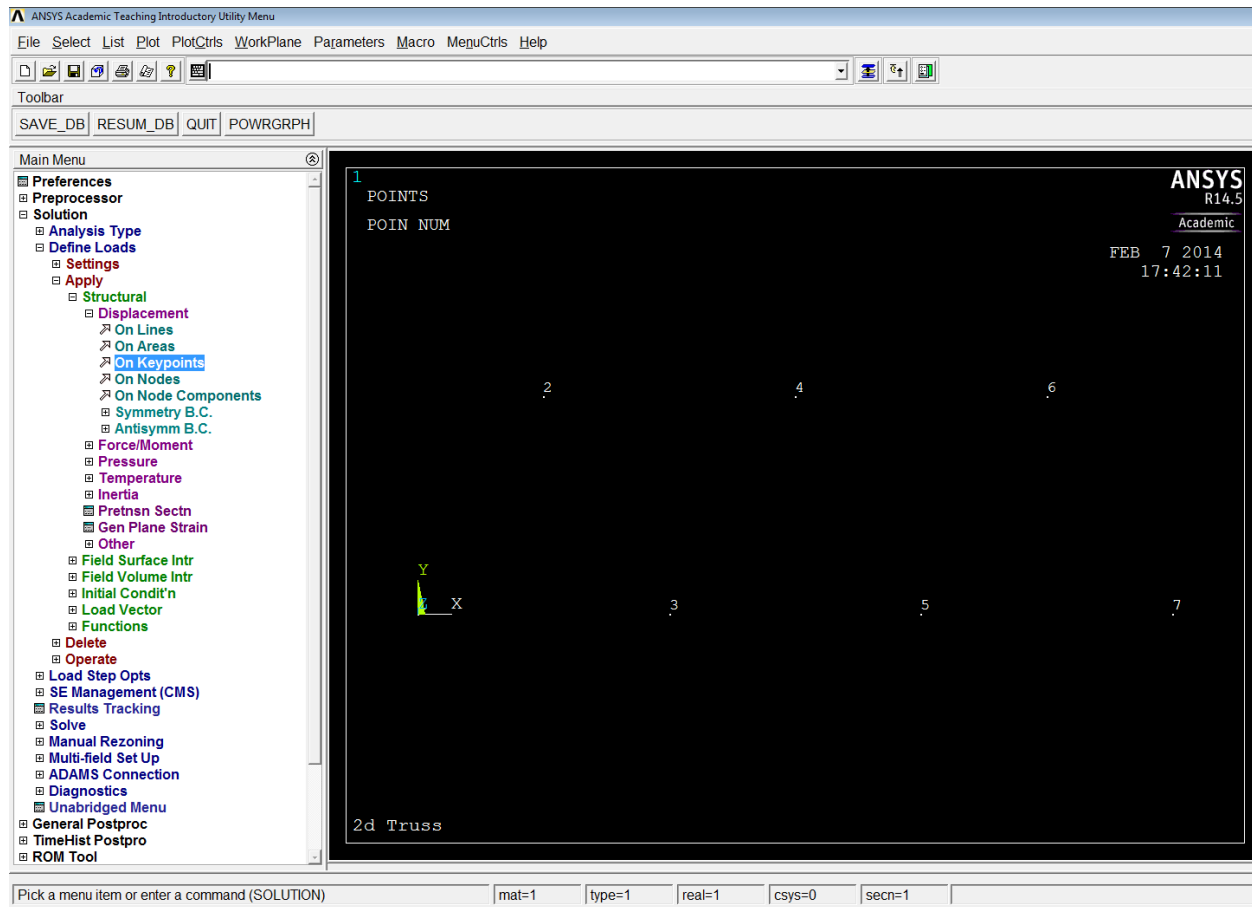
17. Saving: Utility Menu> File> Save as. Enter the file name and select the directory where you would like to save your file and then select ok.



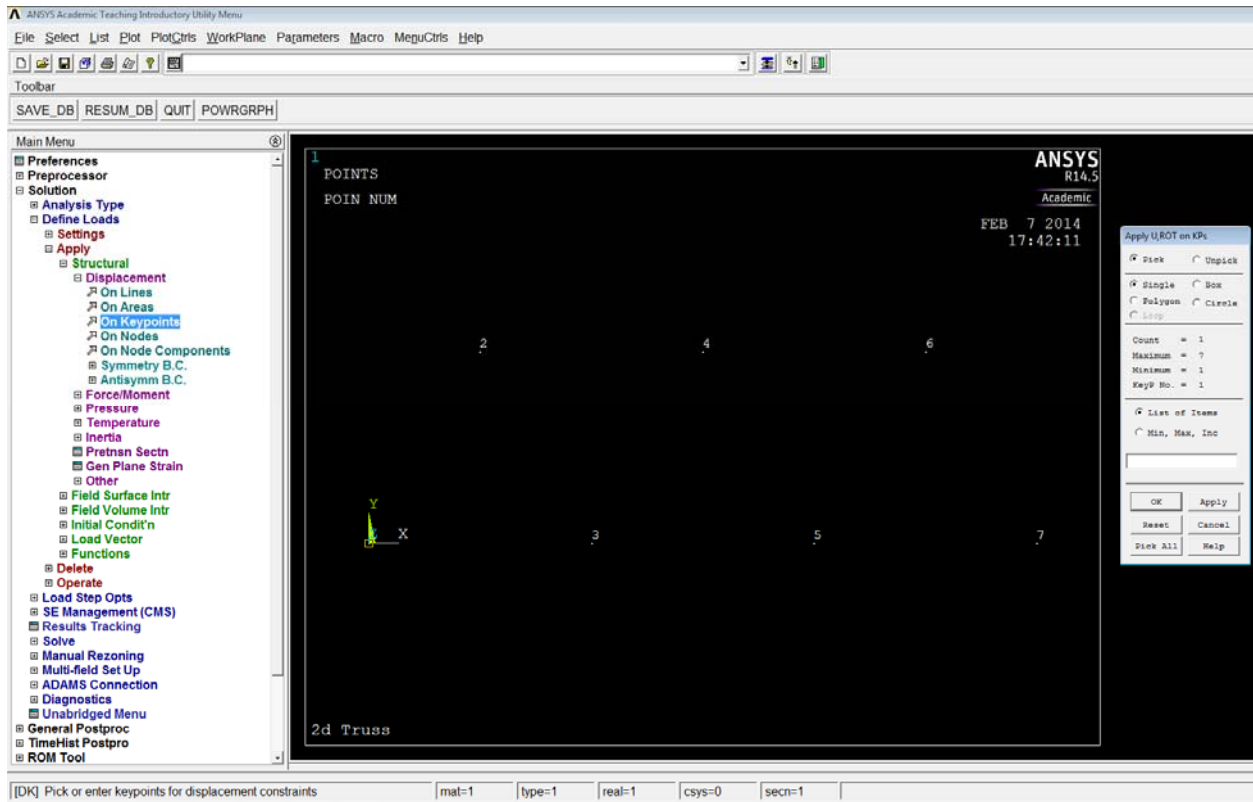
18. Solution type: Main Menu> Solution> Analysis Type> New Analysis. Select as shown and then ok.



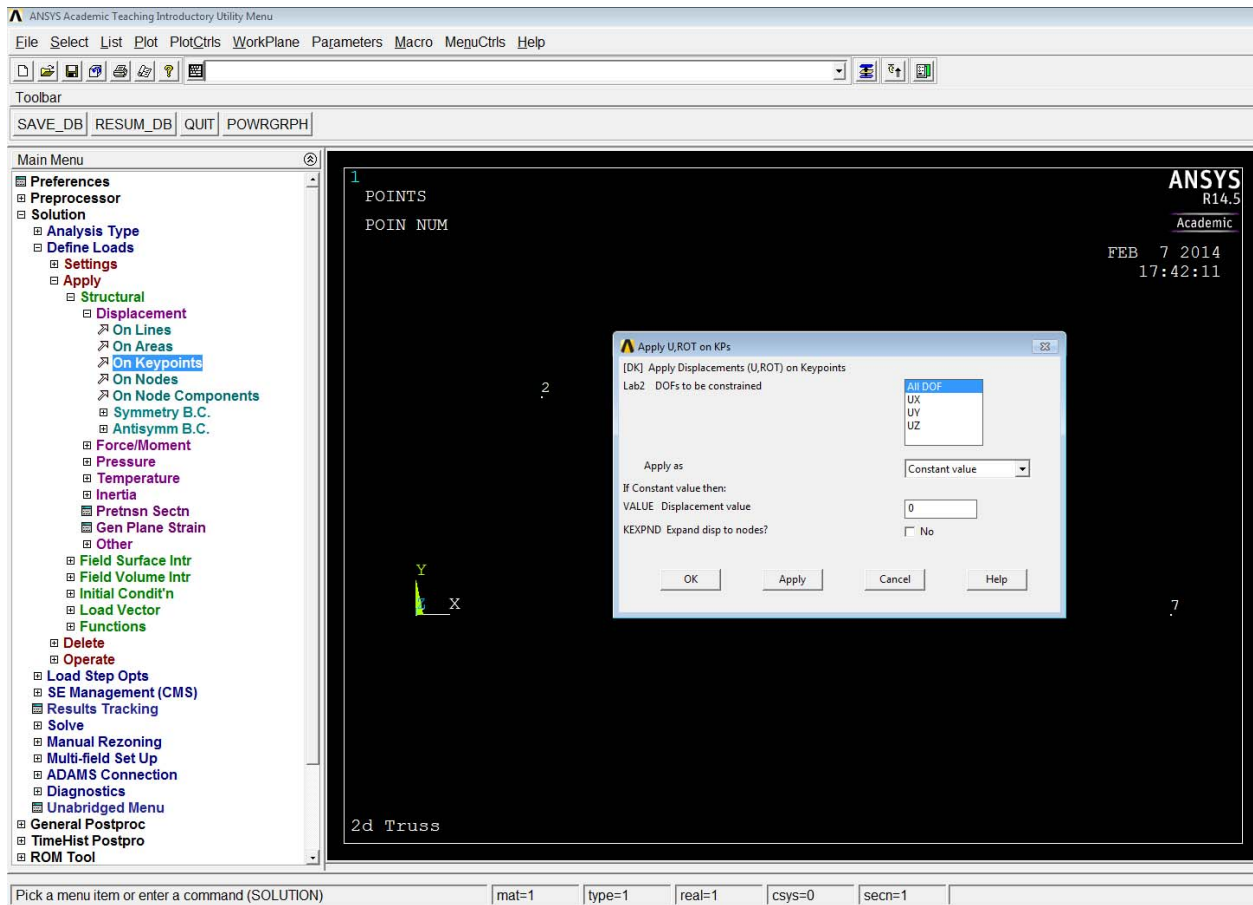
19. Utility Menu> Plot> Keypoints> Keypoints.



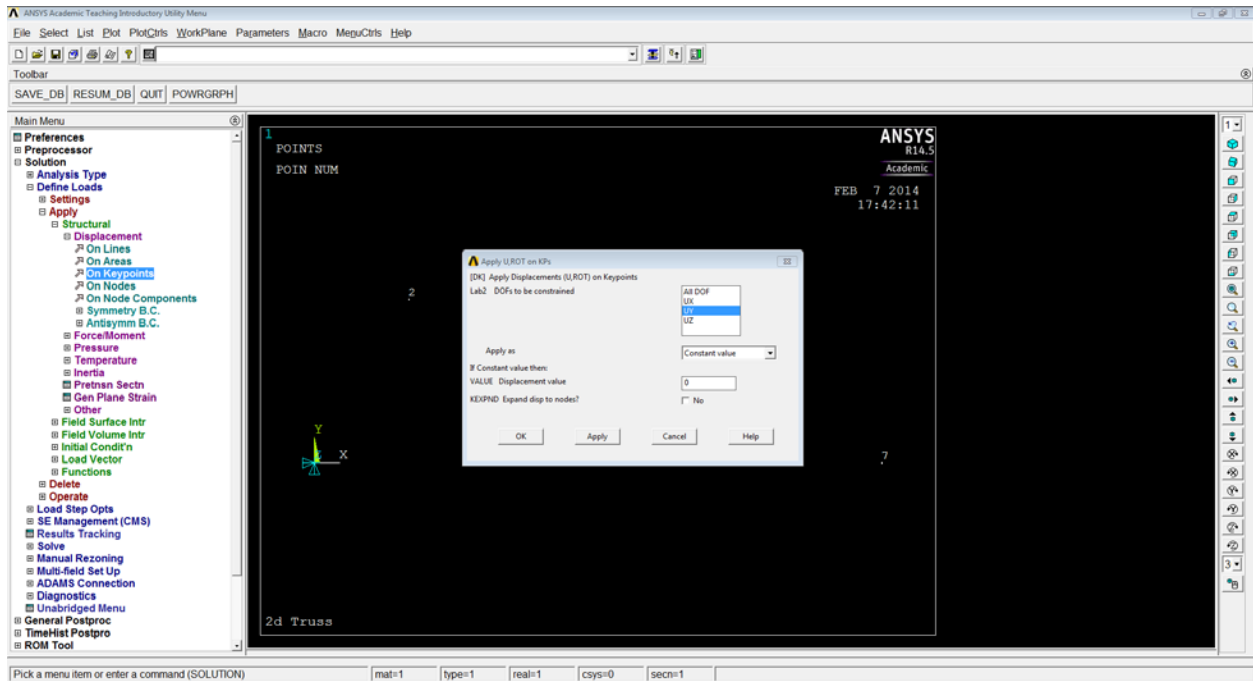
20. Constraints: Main Menu> Solution> Define Loads> Apply> Structural> Displacement> On Keypoints.
Select Keypoint 1 and select ok in the selection box.



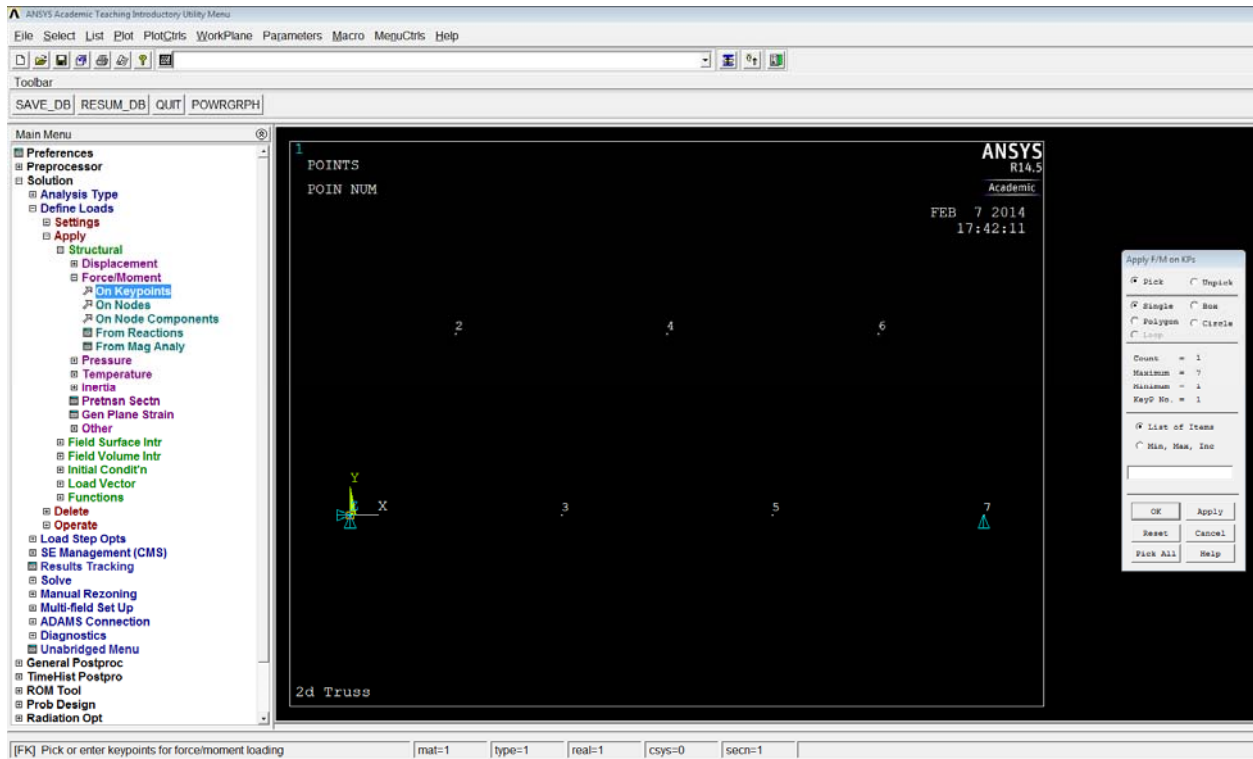
21. Select All DOF, enter 0 in VALUE and then select ok.



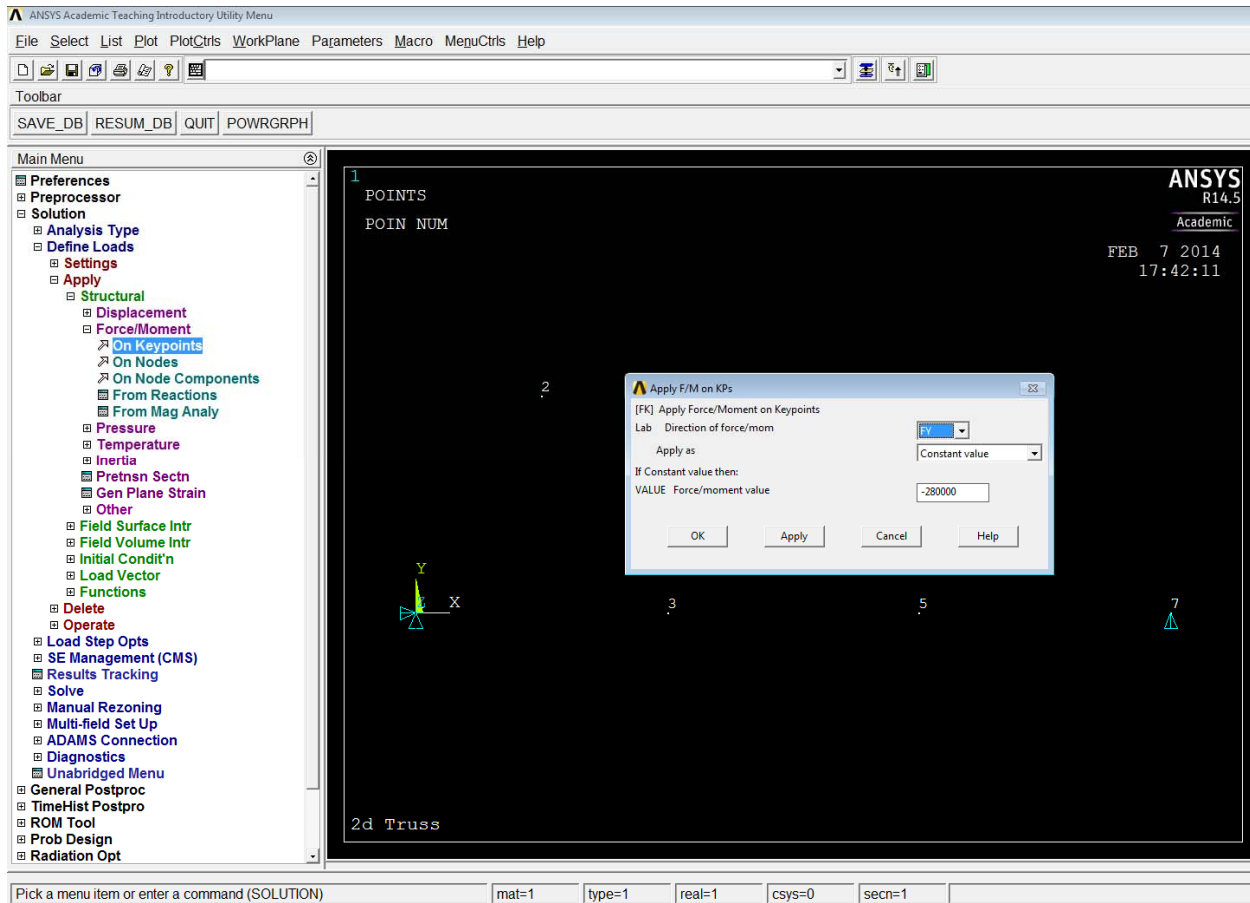
22. Similarly apply the constraint on the right end by deselecting All DOF, selecting UY and finally entering 0.



23. Loading: Solution> Define Loads> Apply> Structural> Force/ Moment> On Keypoints. Select Keypoint 1 and then select ok.

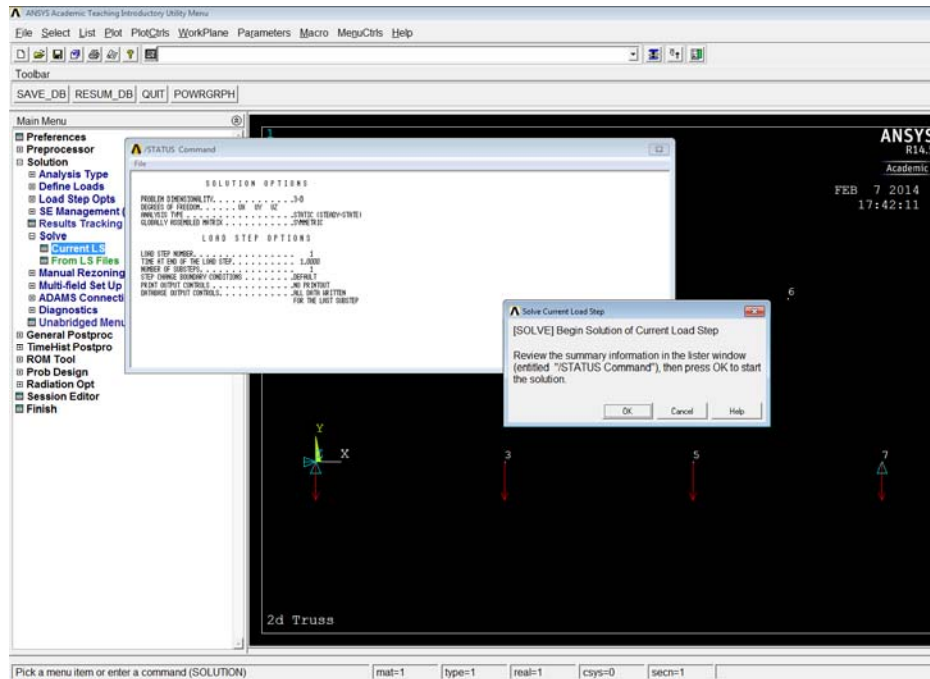


24. Select FY in Direction of force/ mom, Constant value in Apply as and enter -280000 in VALUE. Then select ok.

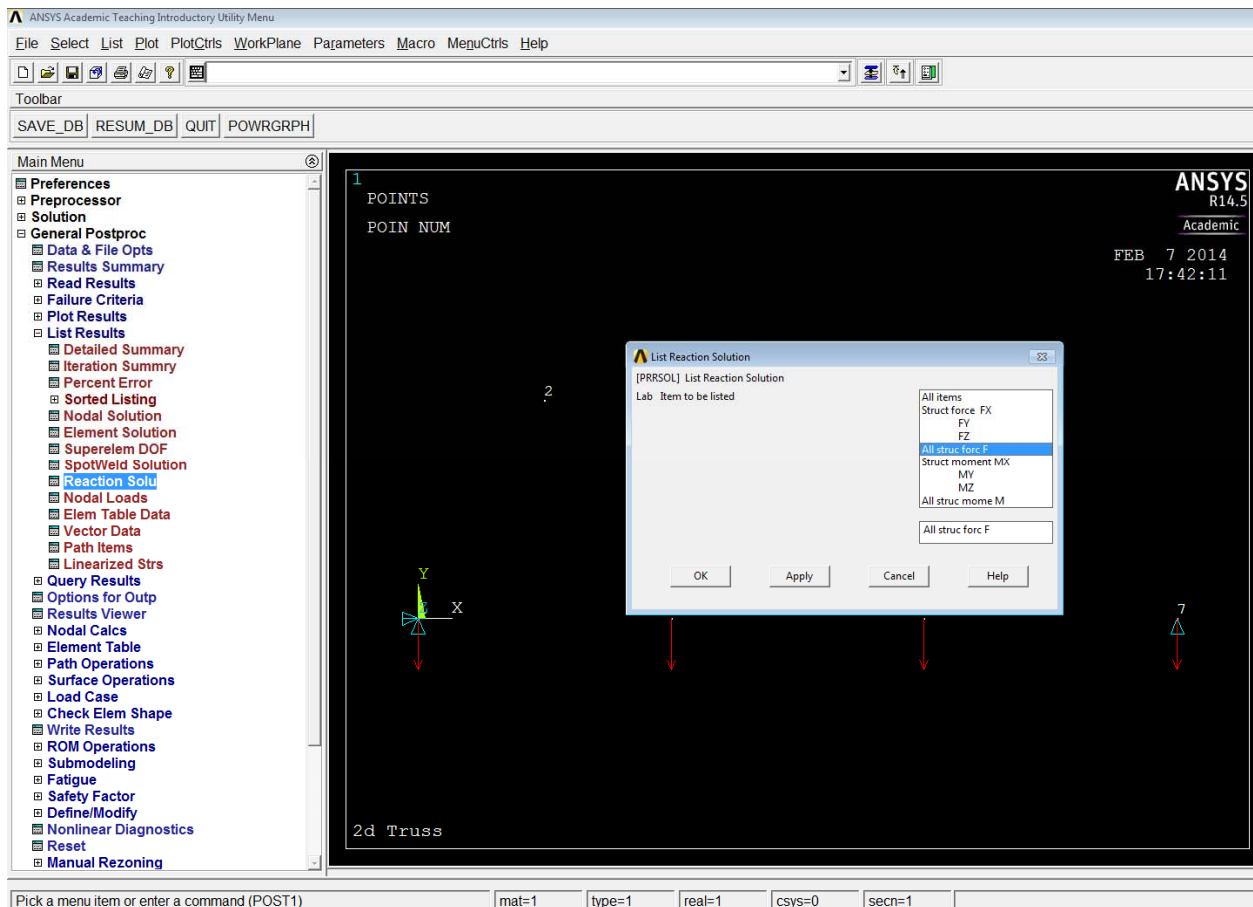


The screenshot displays the ANSYS Academic Teaching Introductory Utility Menu. The top menu bar includes File, Select, List, Plot, PlotCtrls, WorkPlane, Parameters, Macro, MenuCtrls, and Help. Below the menu bar is a toolbar with various icons for file operations and analysis. The Main Menu is open on the left, showing a tree structure of options: Preferences, Preprocessor, Solution, Analysis Type, Define Loads, Settings, Apply, Structural, Displacement, Force/Moment, On Keypoints, On Nodes, On Node Components, From Reactions, From Mag Analy, Pressure, Temperature, Inertia, Pretension Sectn, Gen Plane Strain, Other, Field Surface Intr, Field Volume Intr, Initial Condit'n, Load Vector, Functions, Delete, Operate, Load Step Opts, SE Management (CMS), Results Tracking, Solve, Manual Rezoneing, Multi-field Set Up, ADAMS Connection, Diagnostics, Unabridged Menu, General Postproc, TimeHist Postpro, ROM Tool, Prob Design, and Radiation Opt. The main window shows a 2D Truss model with 7 points and 5 elements. The points are labeled 1 through 7, and the elements are labeled 1 through 5. The model is a simple truss structure with a horizontal base and a vertical support. The text '2d Truss' is visible in the bottom left corner of the main window. The top right corner of the main window displays 'ANSYS R14.5 Academic FEB 7 2014 17:42:11'.

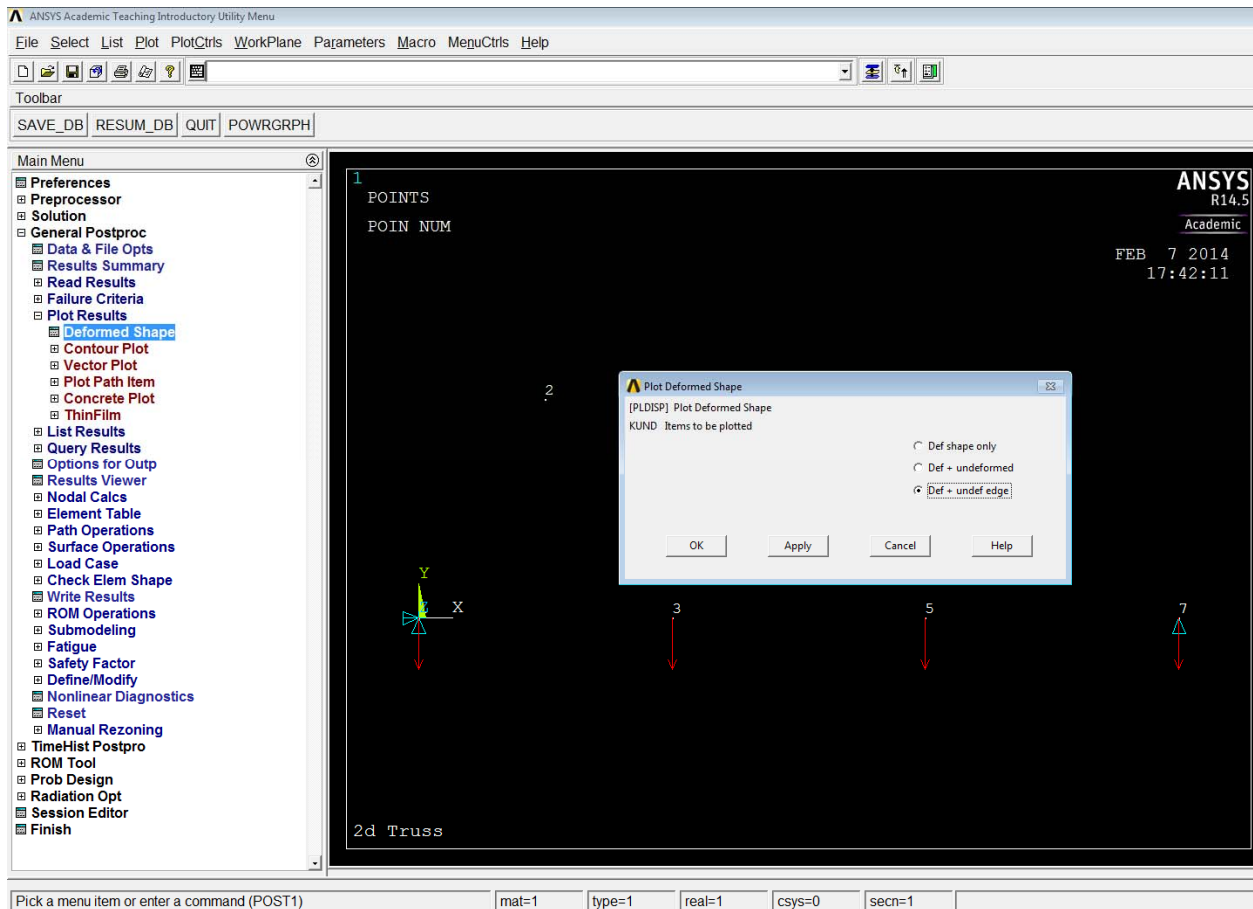
26. Solving: Main Menu> Solution> Solve> Current LS. Then select ok. Once solution is done close the box.



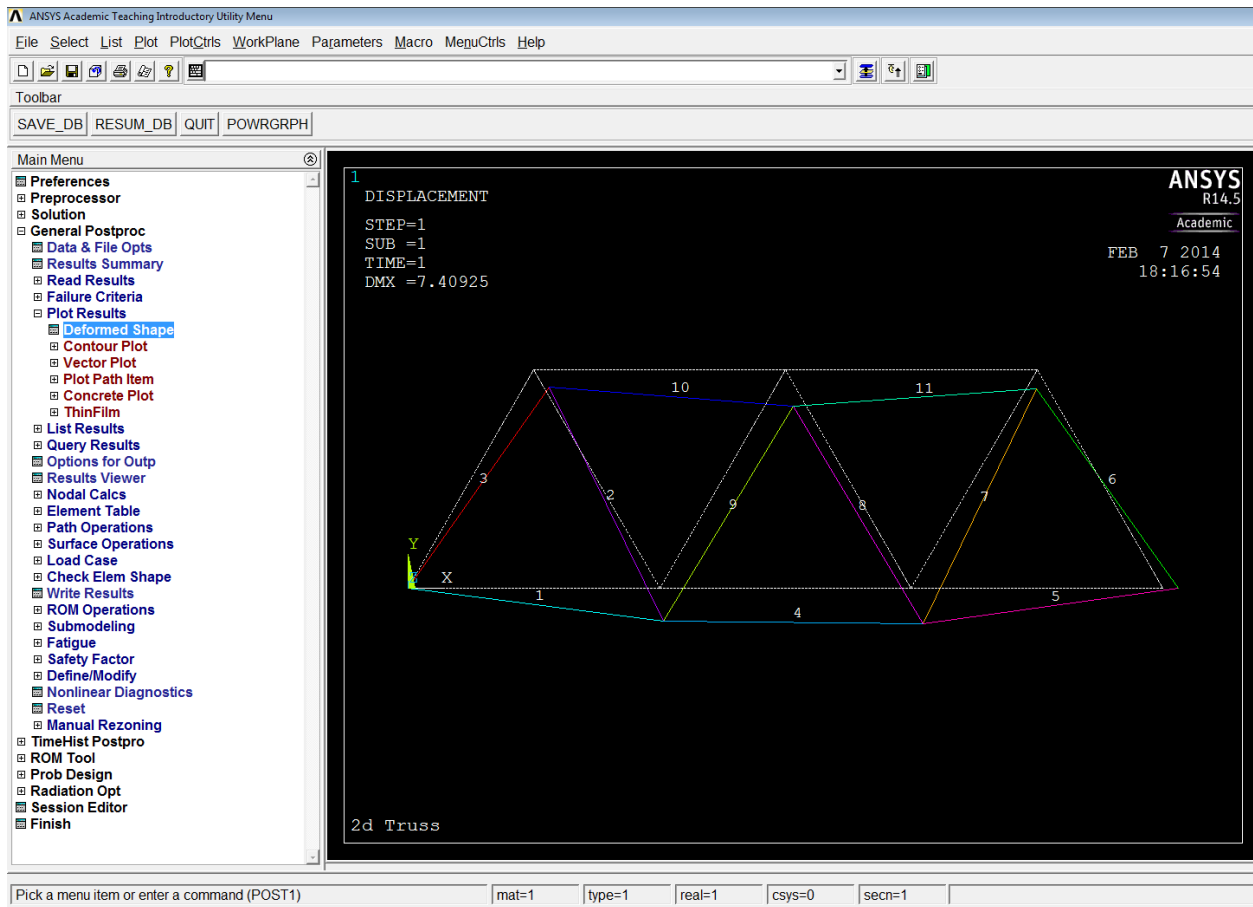
27. Reaction forces: General Postproc> List Results> Reaction Solu. Select All struc forc F and then select ok. The window will give the reaction force results.



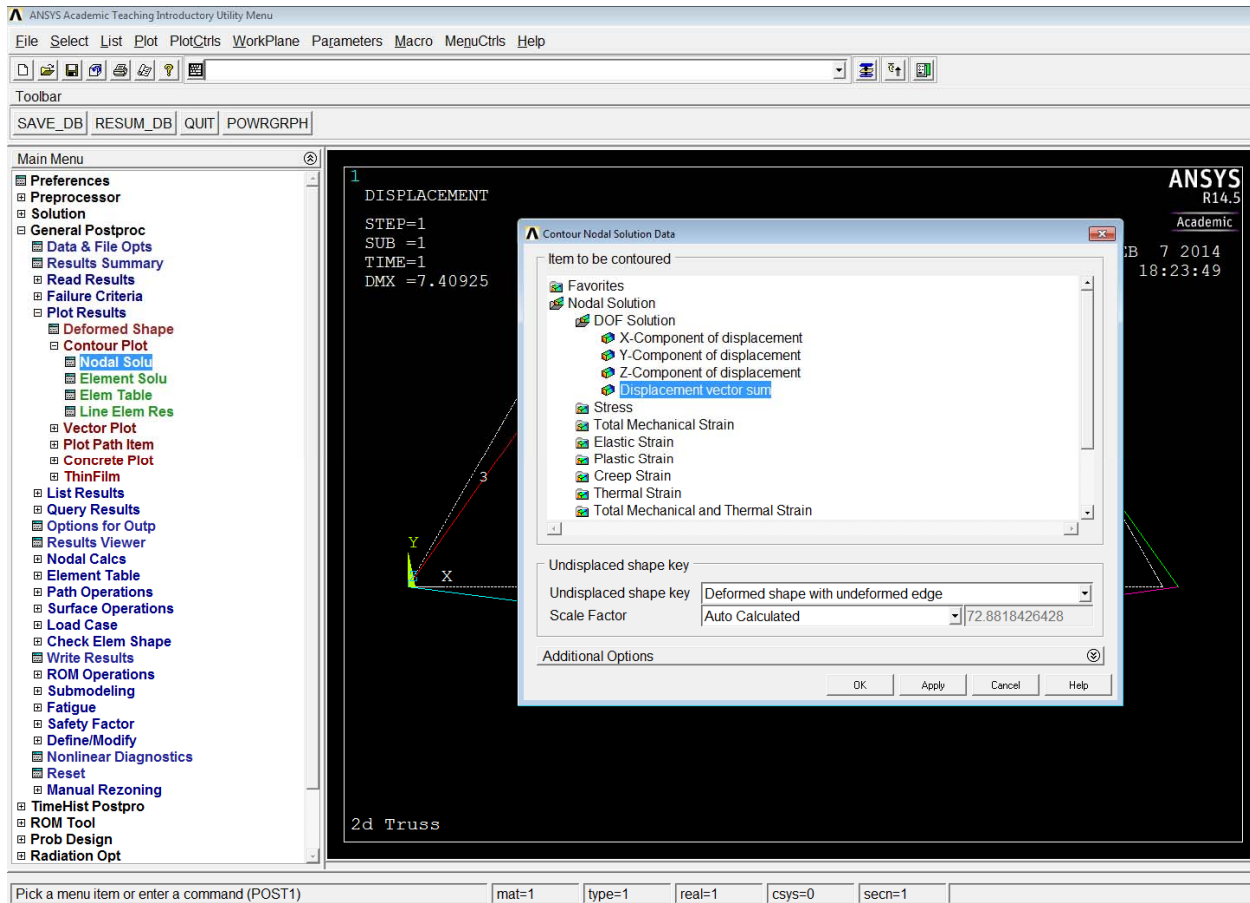
28. Deflection : General Postproc> Plot Results> Deformed Shape. Select Def + undef edge and then ok.



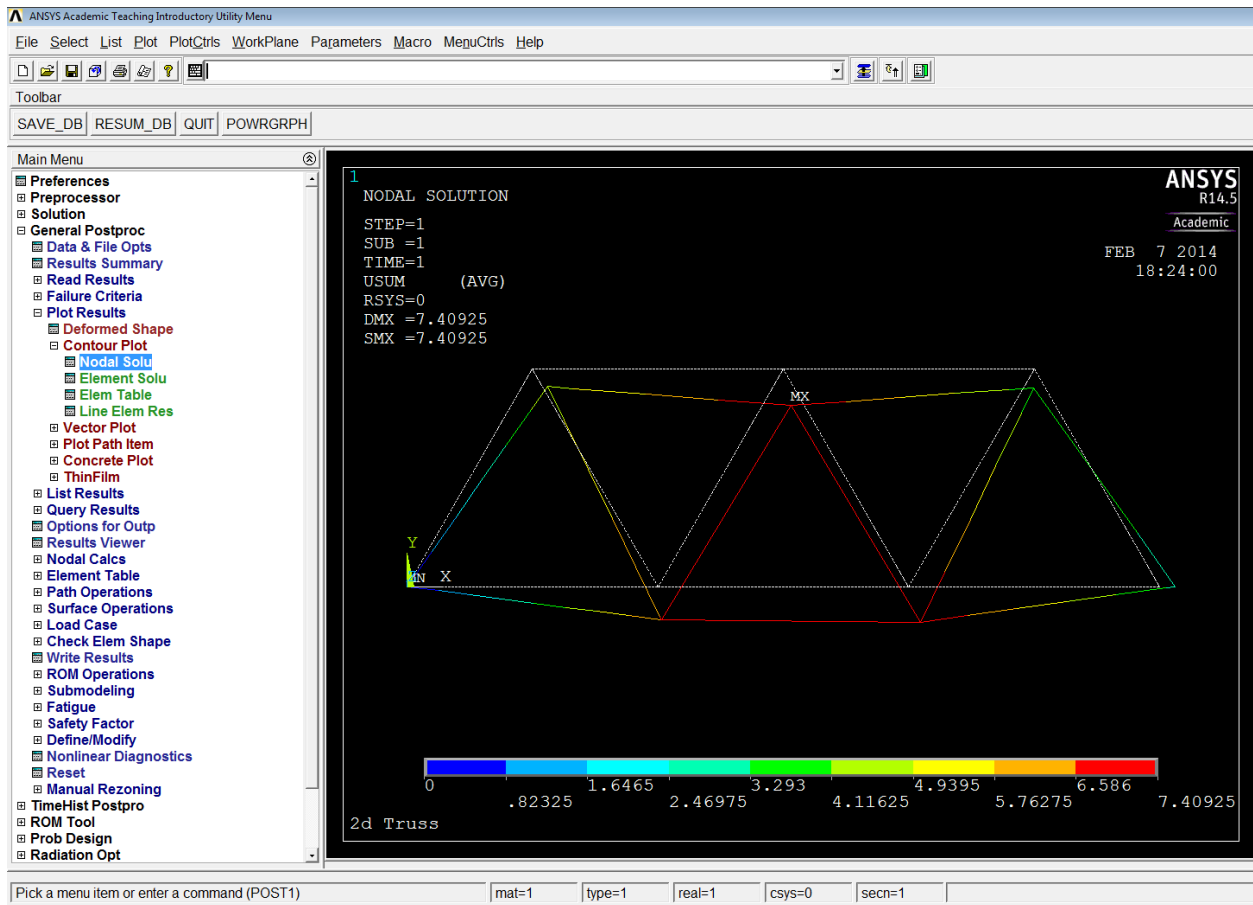
29. The screen should show the deflection plot.



30. Deflection contour: General Postproc> Plot results> Contour Plot> Nodal Solu. Then select as shown in the figure and select ok.



31. The screen should show the displacement contour plot.



The screenshot shows the ANSYS Academic Preprocessor Utility Menu. The 'List Results' path is selected in the left-hand menu. The main window displays the 'List Nodal Solution' dialog box, which is used to select the data to be listed. The 'Displacement vector sum' option is selected under the 'Nodal Solution' category. The background shows a 2D truss model with a color scale for stress, ranging from 0 to 7.40925.

ANSYS Academic Preprocessor Utility Menu

File Select List Plot PlotCtrls WorkPlane Parameters Macro MenuCtrls Help

Toolbar

SAVE_DB RESUM_DB QUIT POWRGRPH

Main Menu

- Preferences
- Preprocessor
- Solution
 - General Postproc
 - Data & File Opts
 - Results Summary
 - Read Results
 - Failure Criteria
 - Plot Results
 - List Results
 - Detailed Summary
 - Iteration Summary
 - Percent Error
 - Sorted Listing
 - Nodal Solution
 - Element Solution
 - Superelem DOF
 - SpotWeld Solution
 - Reaction Solu
 - Nodal Loads
 - Elem Table Data
 - Vector Data
 - Path Items
 - Linearized Strs
 - Query Results
 - Options for Outp
 - Results Viewer
 - Nodal Calcs
 - Element Table
 - Path Operations
 - Surface Operations
 - Load Case
 - Check Elem Shape
 - Write Results
 - ROM Operations
 - Submodeling
 - Fatigue
 - Safety Factor
 - Define/Modify
 - Nonlinear Diagnostics
 - Reset
 - Manual Rezonig

1 NODAL SOLUTION

STEP=1
SUB =1
TIME=1
USUM (AVG)
RSYS=0
DMX =7.40925
SMX =7.40925

ANSYS R14.5 Academic FEB 7 2014 18:27:25

List Nodal Solution

Item to be listed

- Favorites
- Nodal Solution
 - DOF Solution
 - X-Component of displacement
 - Y-Component of displacement
 - Z-Component of displacement
 - Displacement vector sum
 - Stress
 - Total Mechanical Strain
 - Elastic Strain
 - Plastic Strain
 - Creep Strain
 - Thermal Strain
 - Total Mechanical and Thermal Strain

Value for computing the EQV strain

OK Apply Cancel Help

2d Truss

0 .82325 1.6465 2.46975 3.293 4.11625 4.9395 5.76275 6.586 7.40925

33. The results are shown in terms of nodes. The node and Keypoint numbering may not be the same. The node numbering can be obtained by selecting Plot from the Utility menu.

