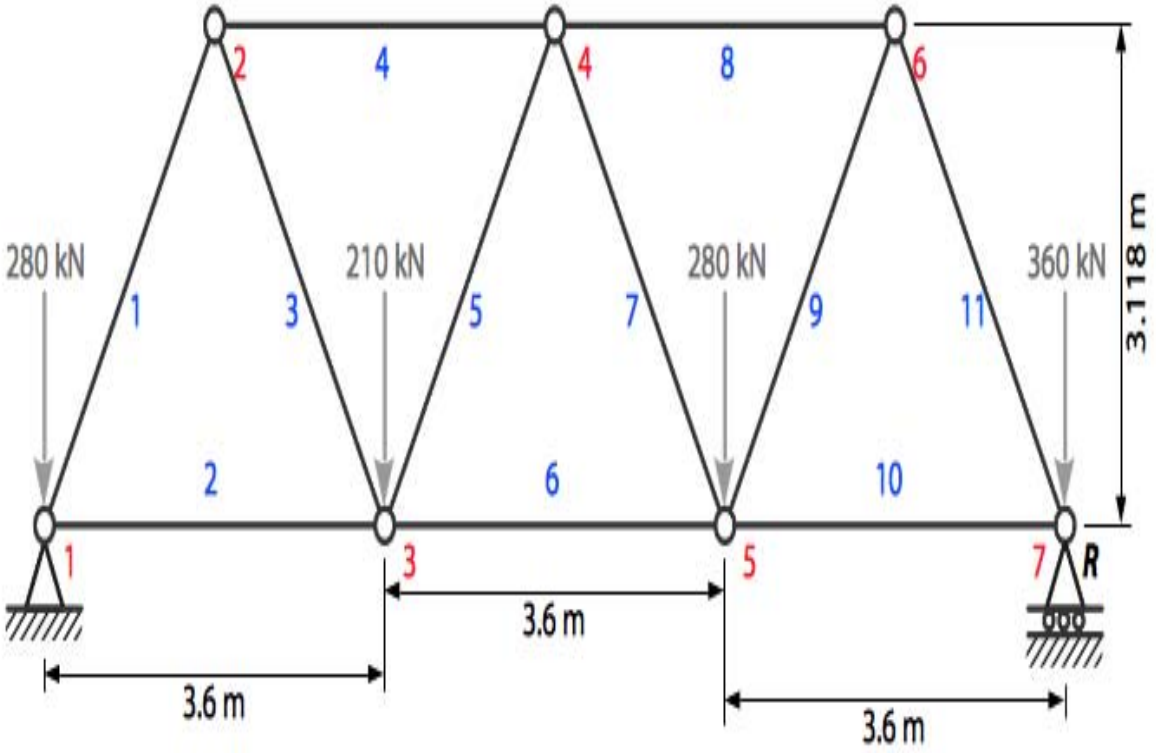
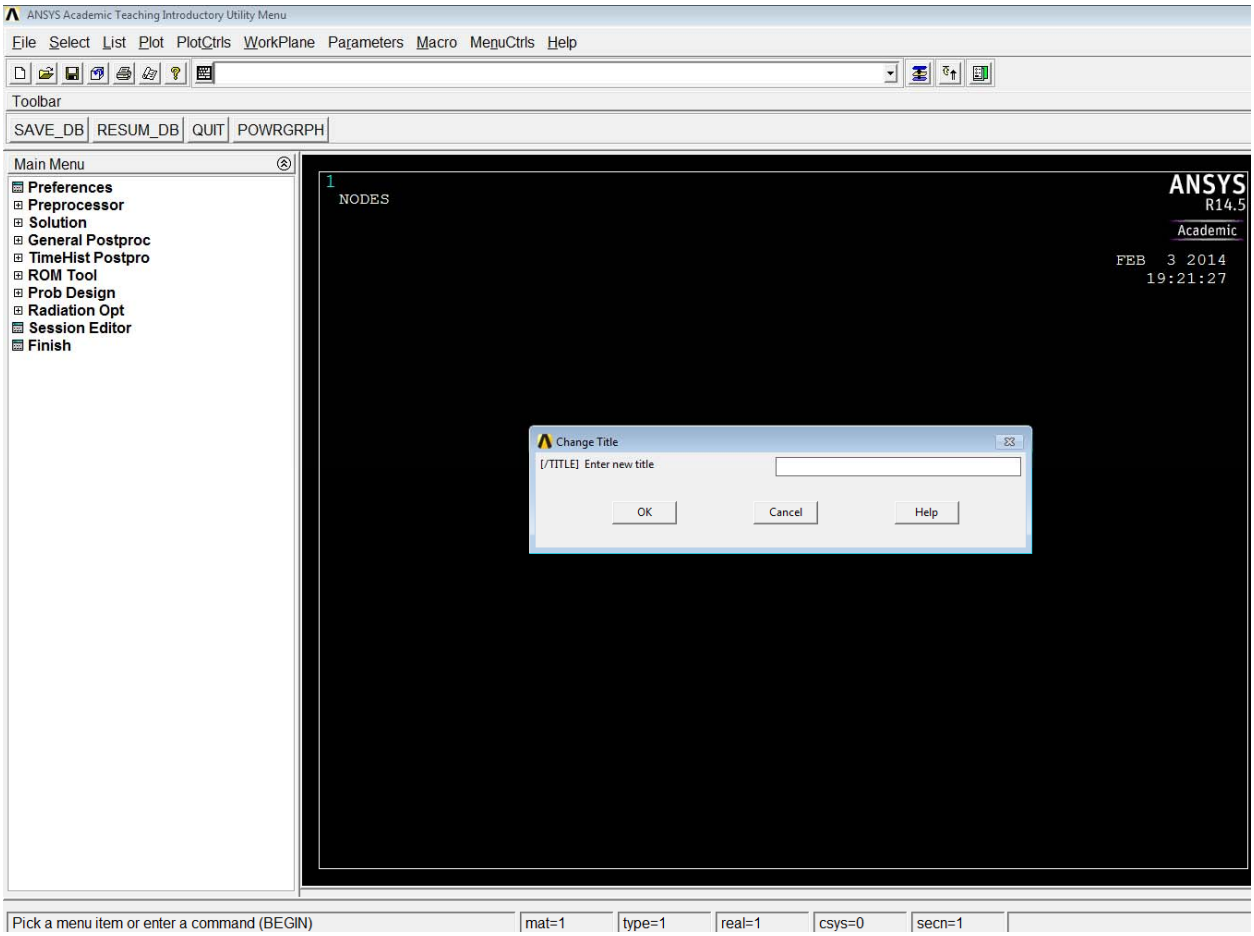
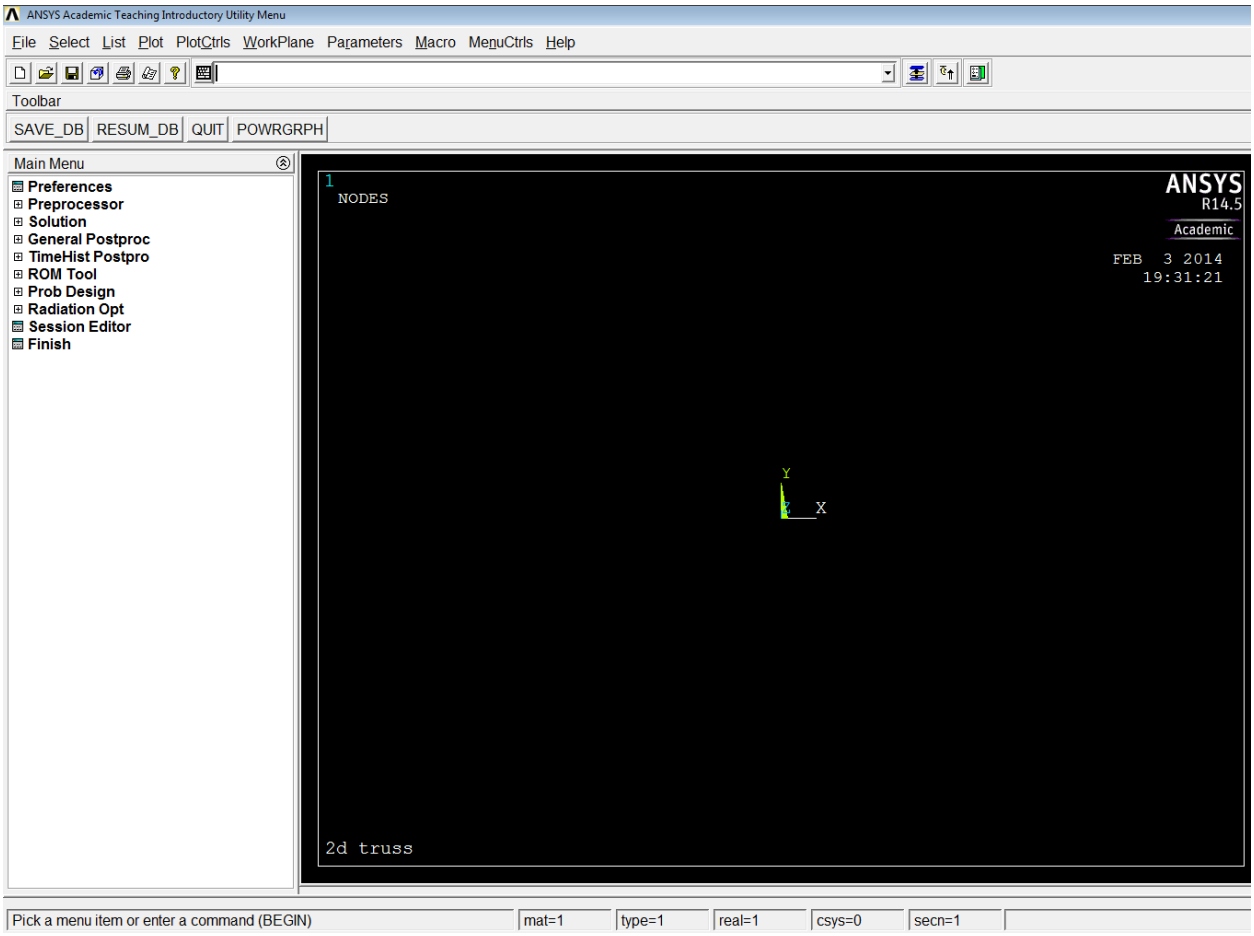

280 kN
210 kN
280 kN
360 kN
3.118 m
1
2
3
4
5
6
7
8
9
10
11
R
3.6 m
3.6 m
3.6 m

1. Title: Utility Menu> File> Change Title



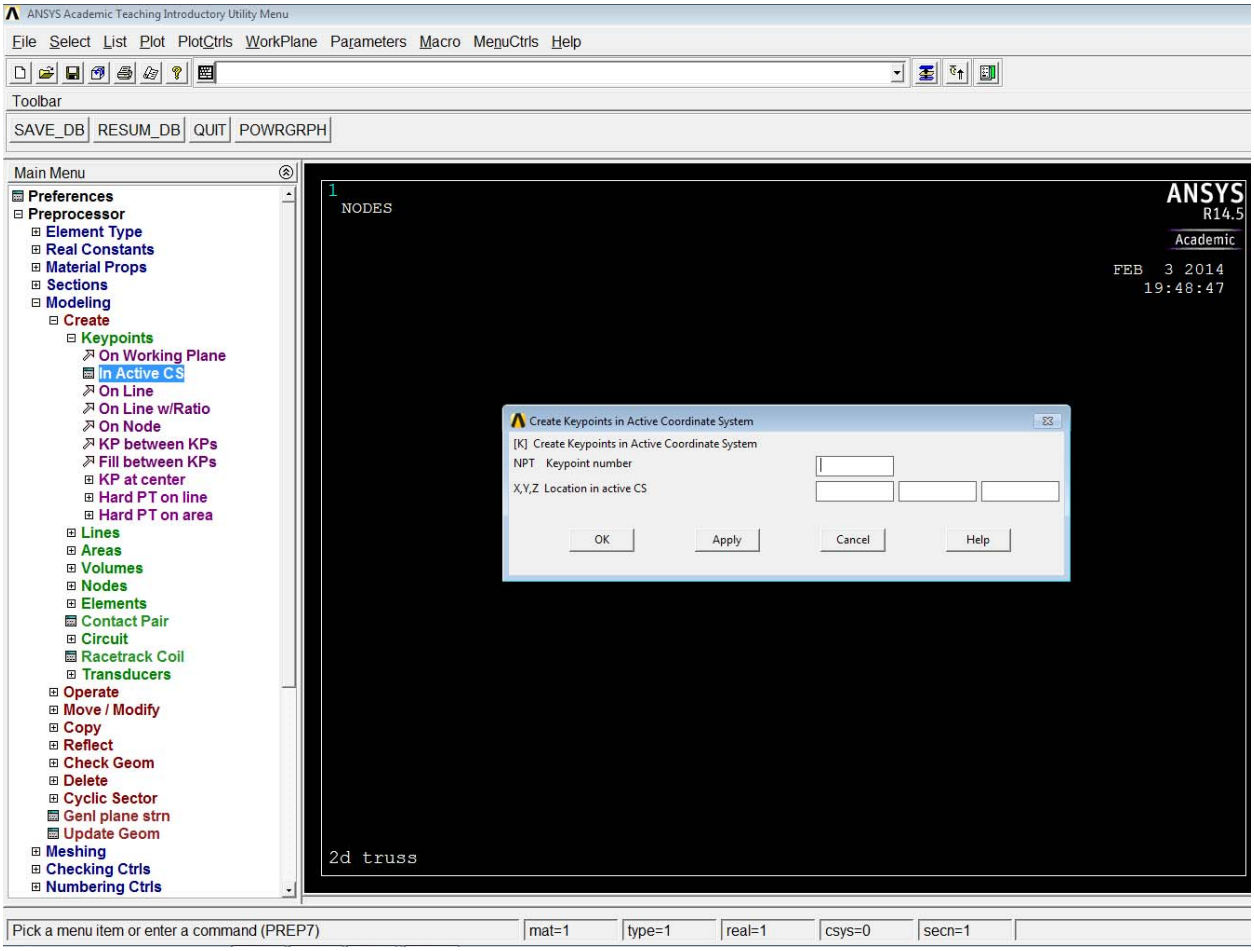
2. Enter title, Ok and then Utility Menu> Plot> Replot



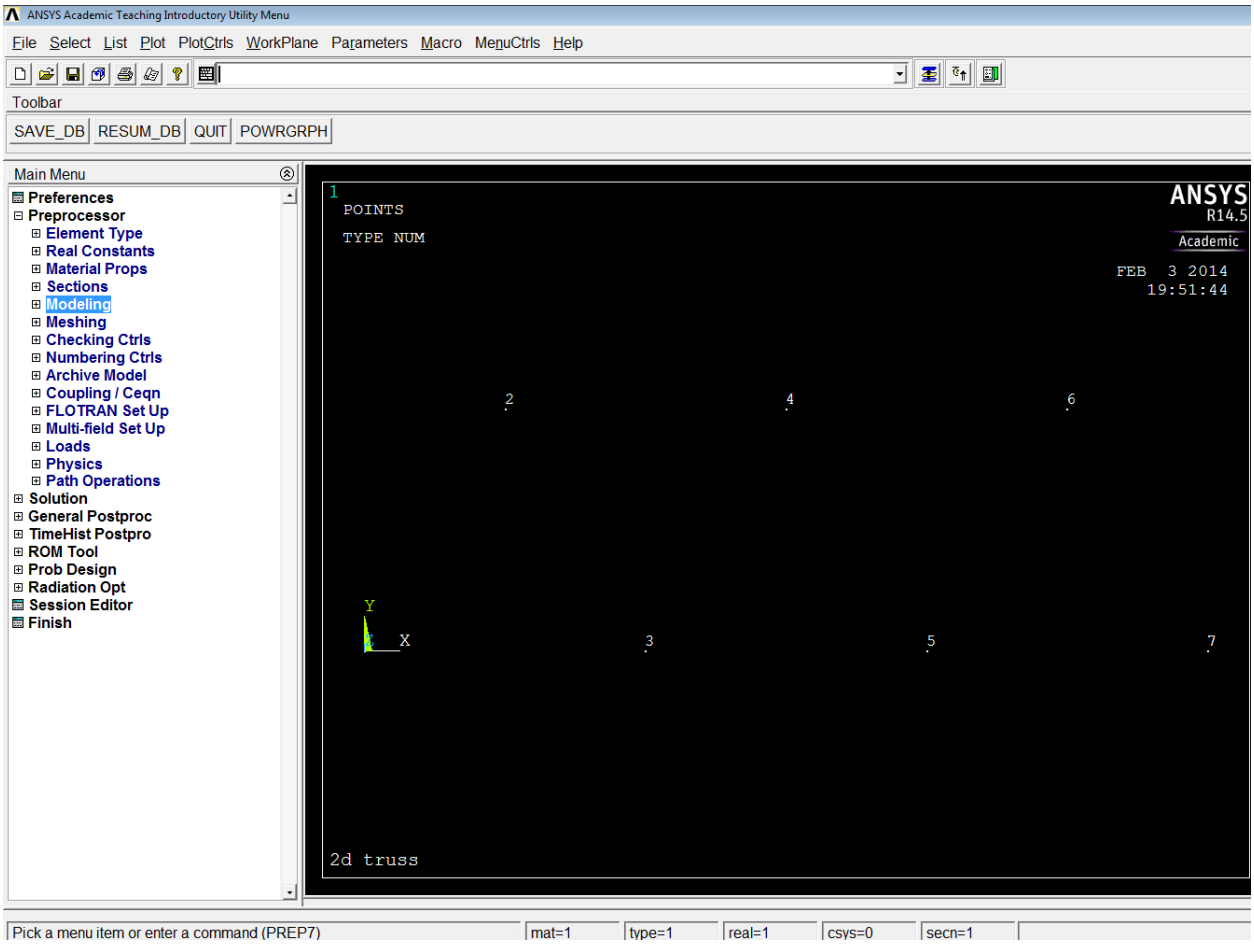
The following keypoints are to be entered next:

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

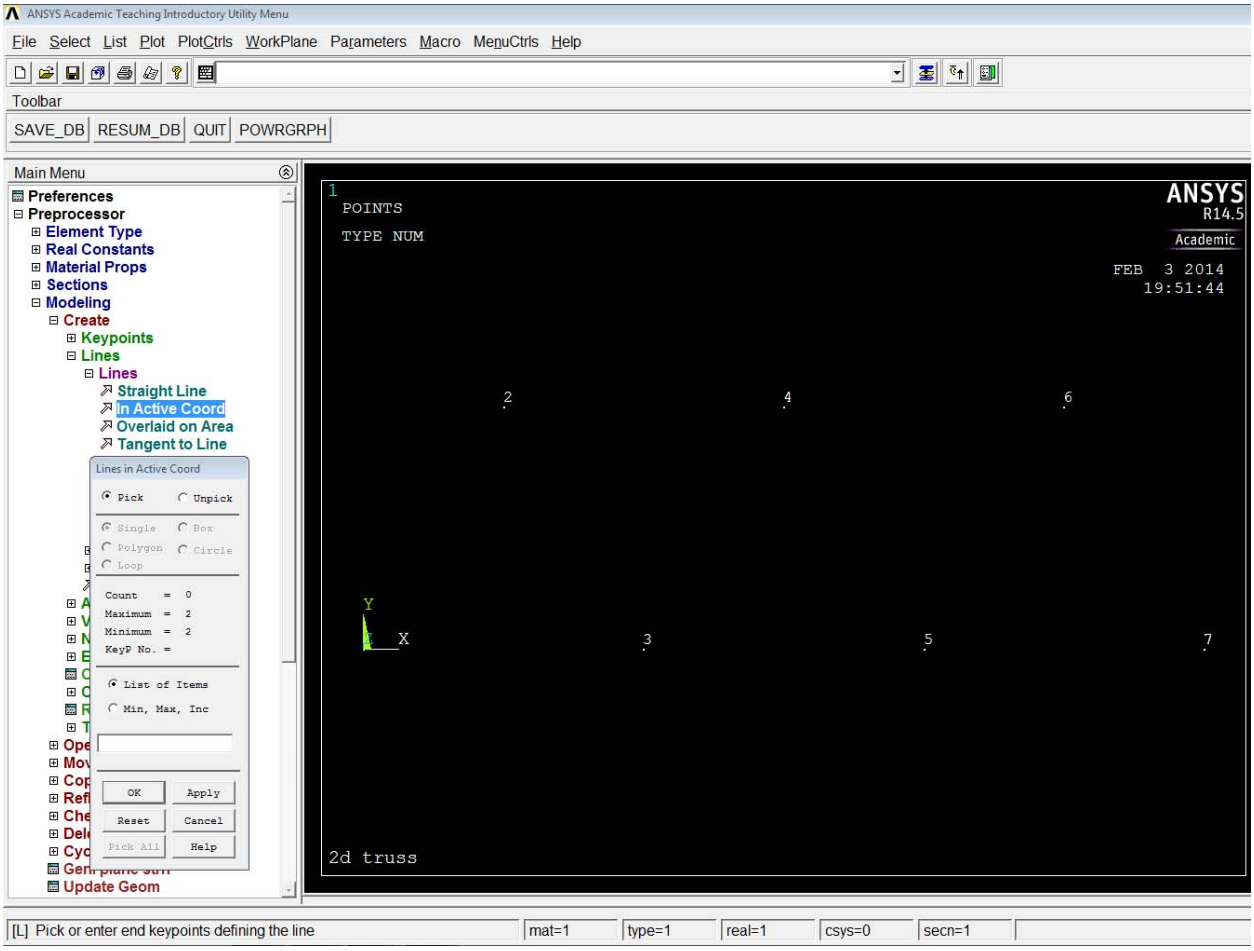
3. Keyword selection: Main menu> Preprocessor> Modeling> Create> Keypoints> In Active CS



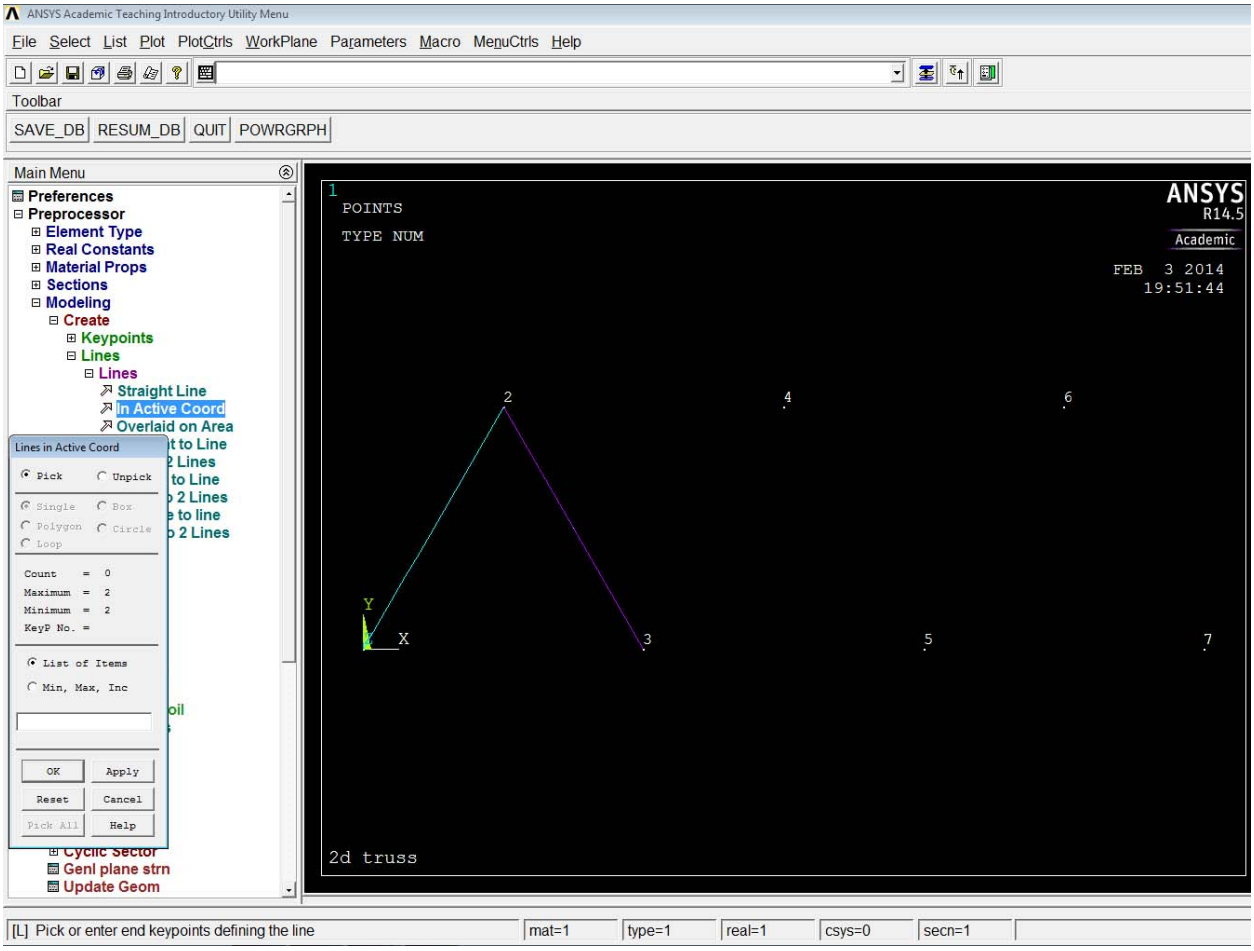
4. Enter the coordinates and press apply until done



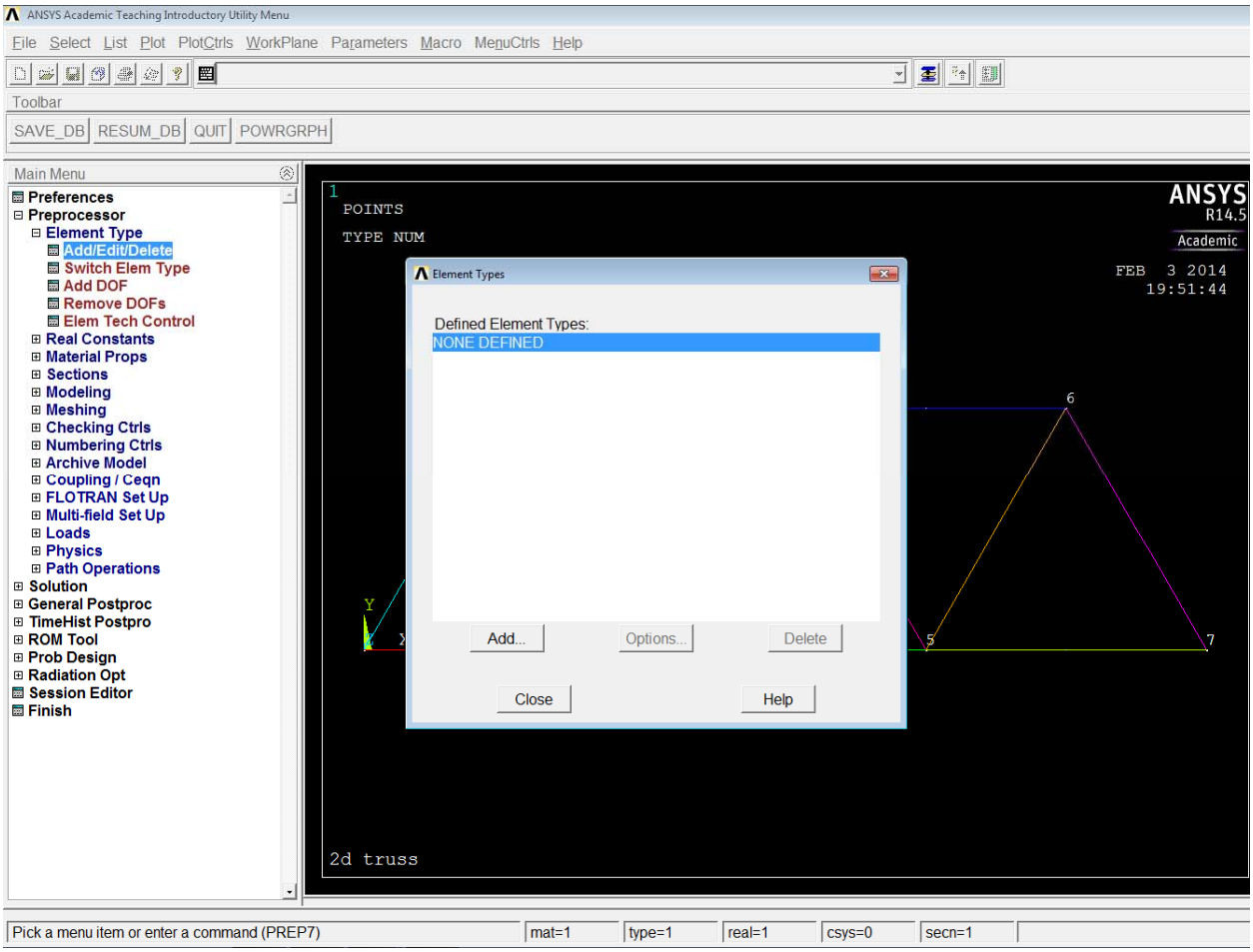
5. Drawing lines: Preprocessor> Modeling> Create> Lines> Lines> In Active Coord



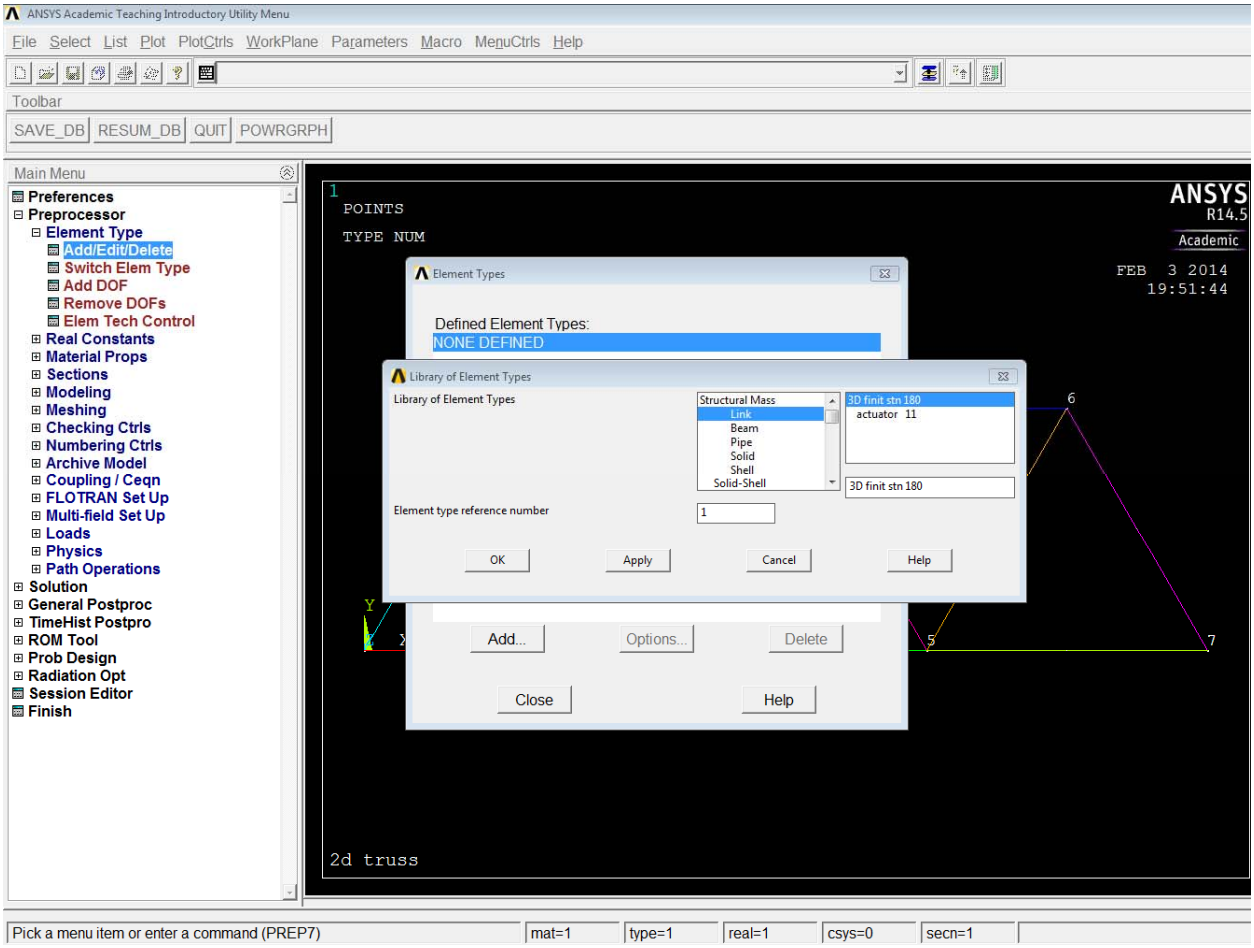
6. Start picking keypoints and press ok once done.



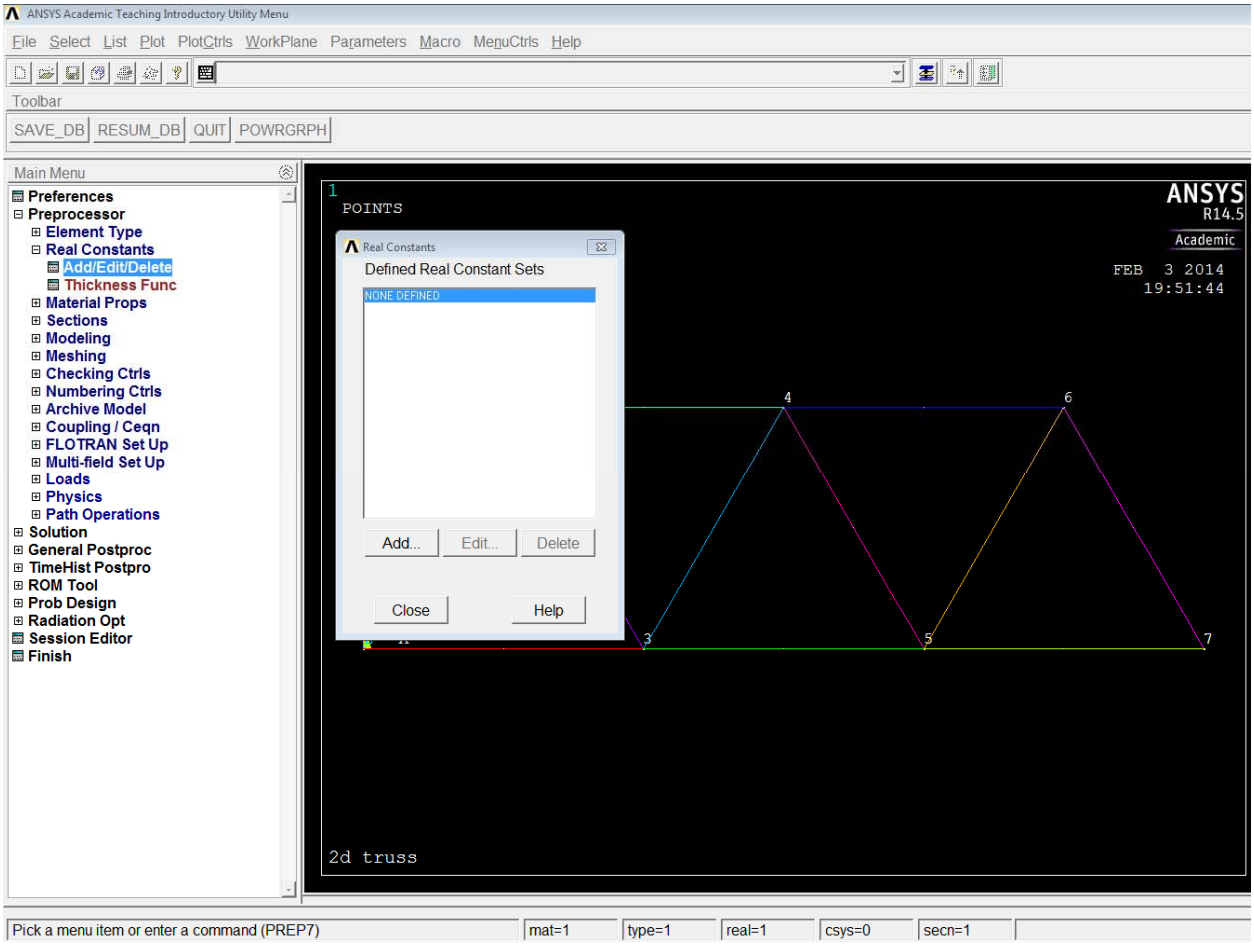
7. Defining elements: Preprocessor> Element type> Add/ Edit/ Delete



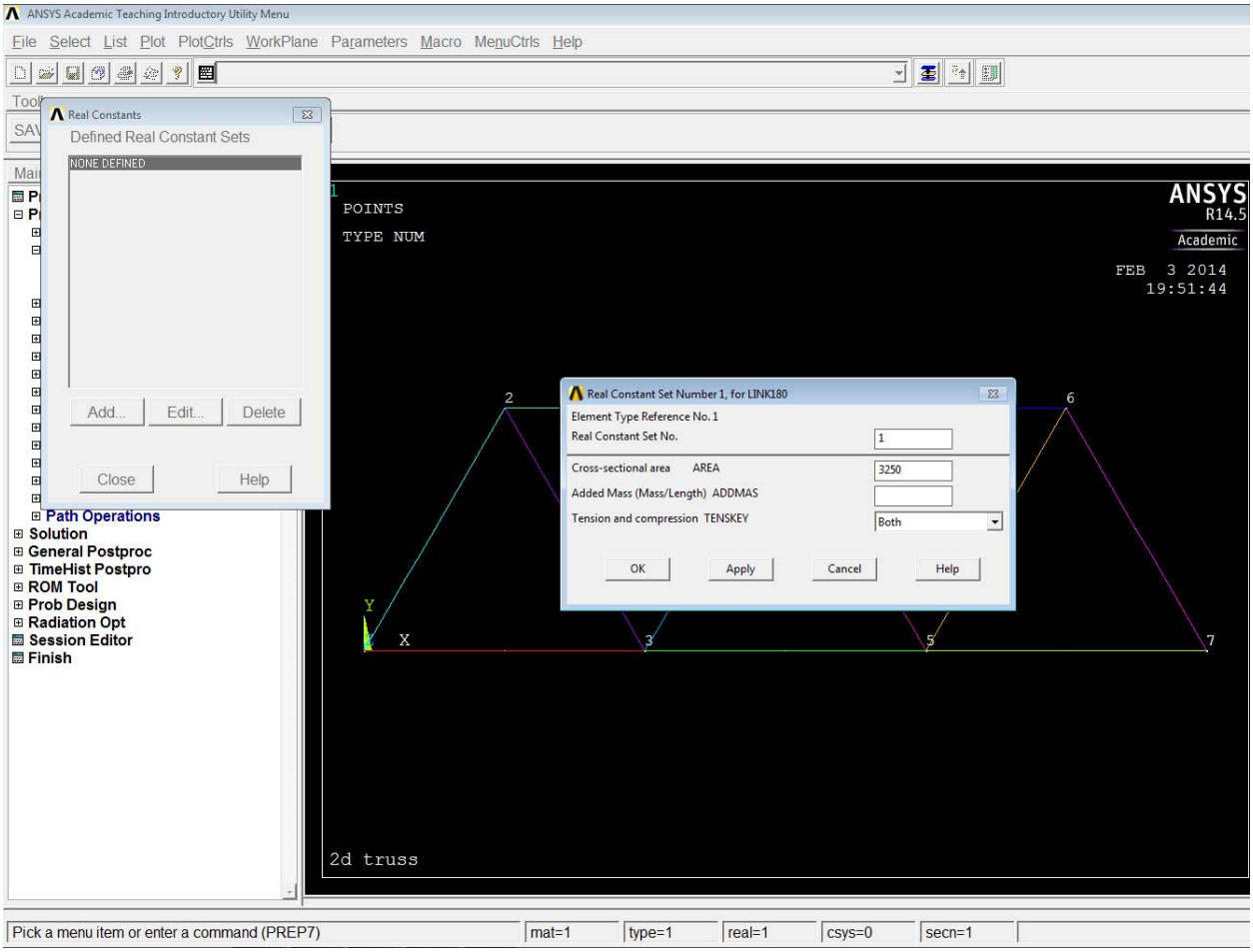
8. Choose Structural mass> Link> 3D finit stn 180 and then ok



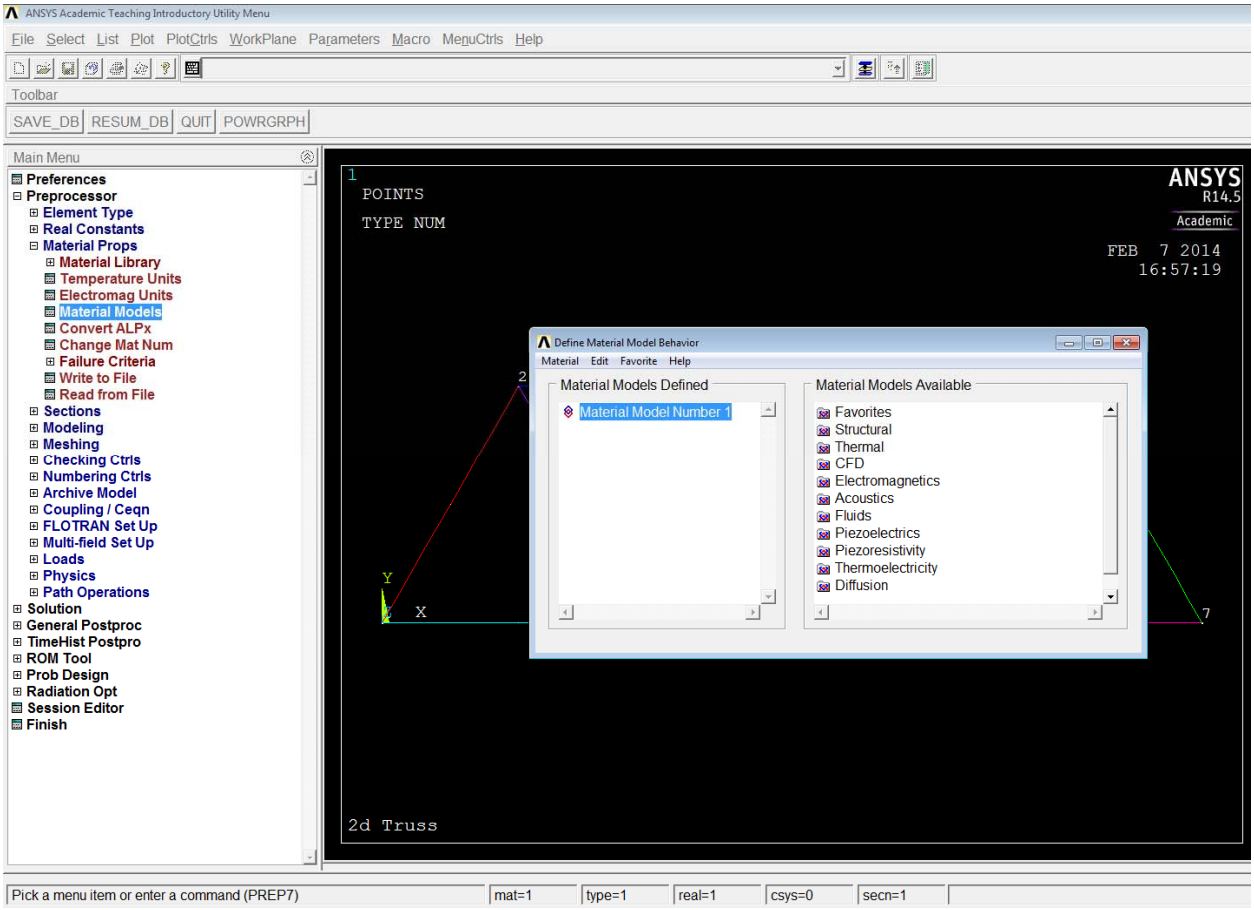
9. Real constants: Preprocessor> Real Constants> Add/ Edit/ Delete and then click Add



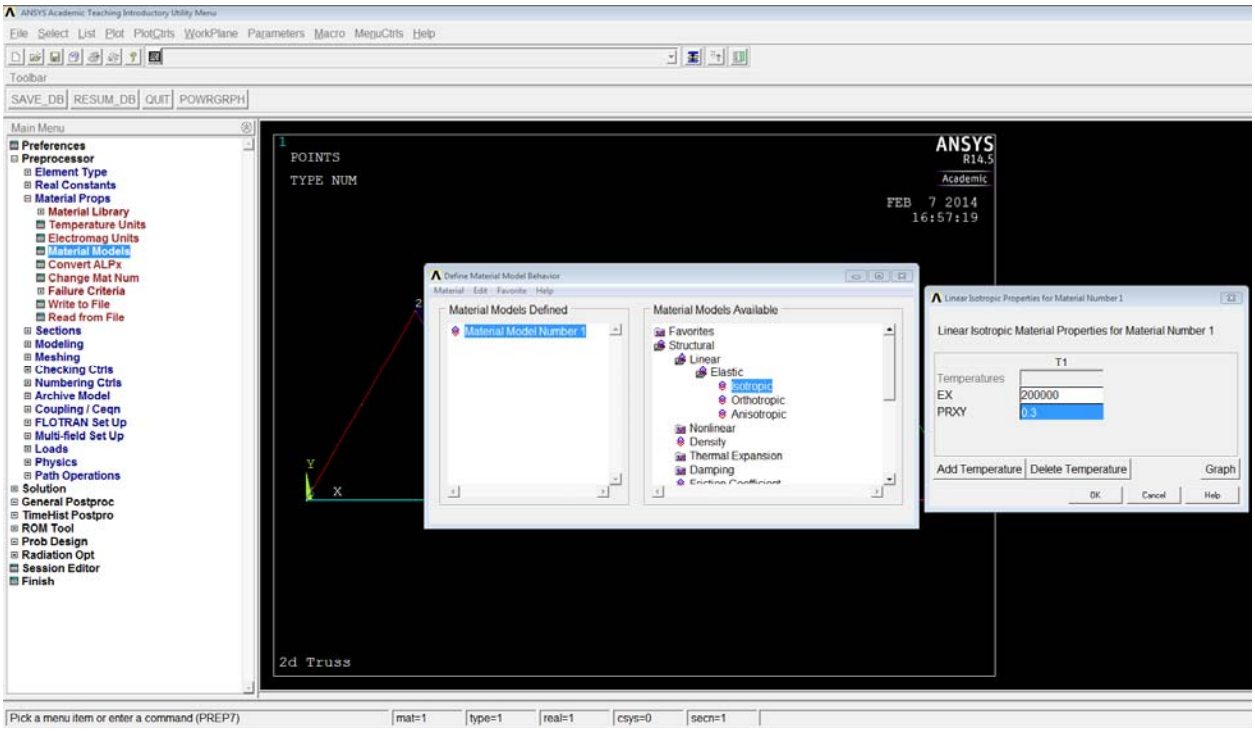
10. Then Ok. Enter 3250 as area and press Ok. After that close the real constant box.



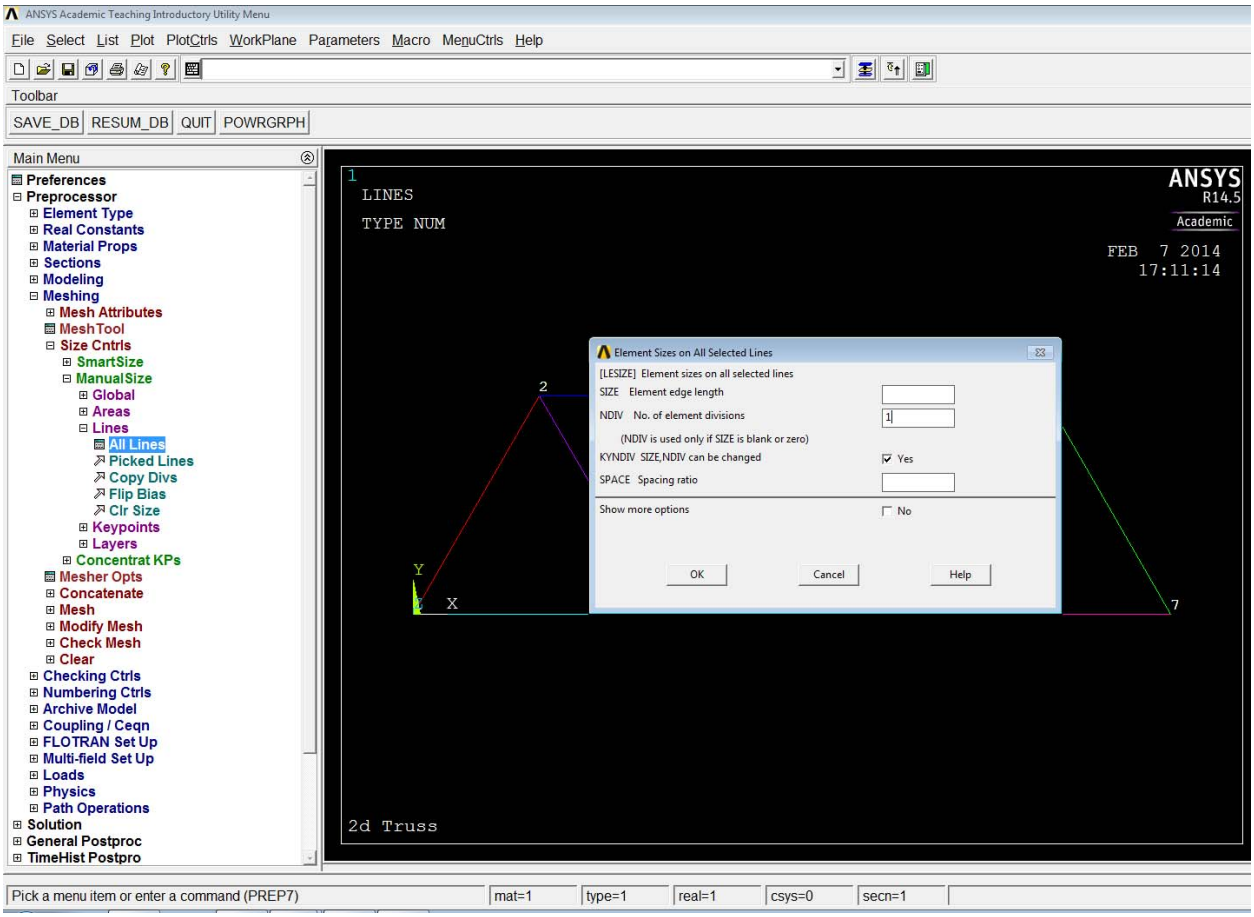
11. Material properties: Preprocessor> Material props> Material Models



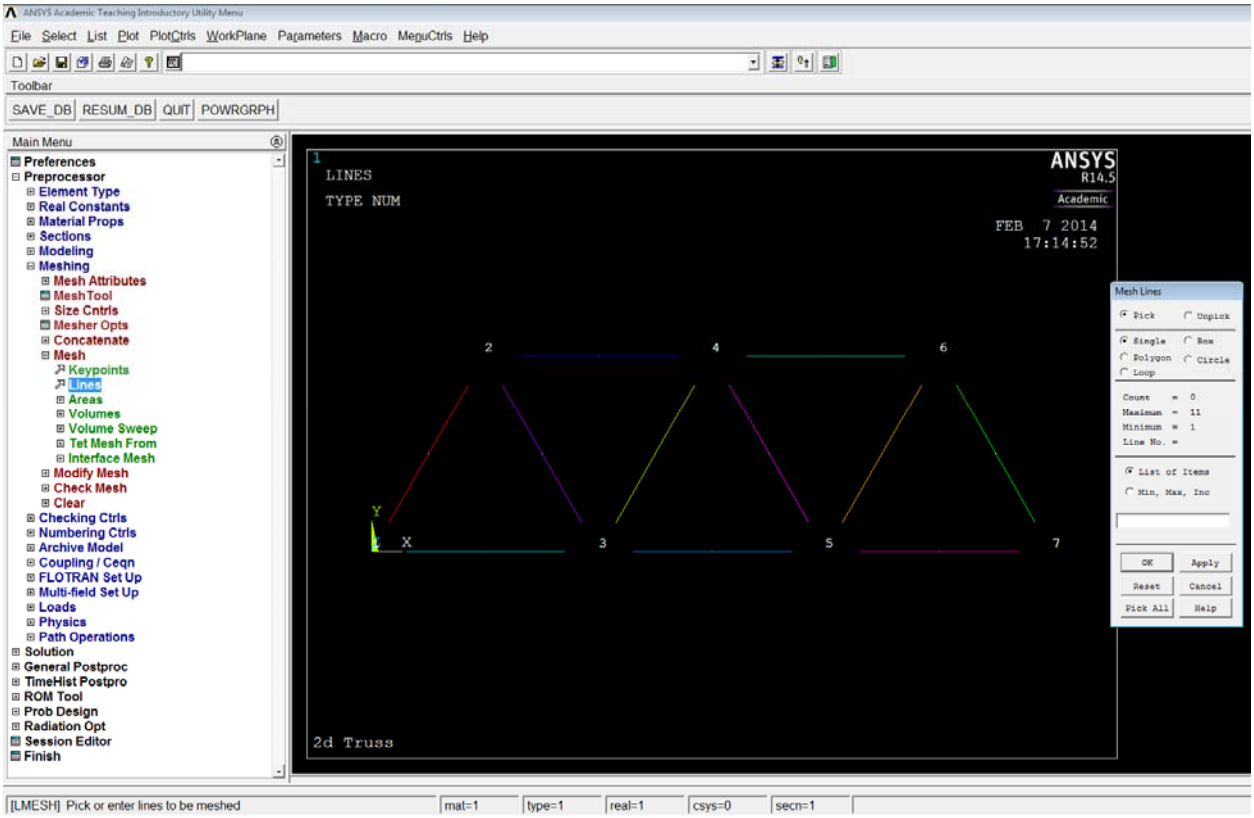
12. Structural> Linear> Elastic> Isotropic. Enter 200000 as Ex and 0.3 as PRxy. Then select ok and close material model window.



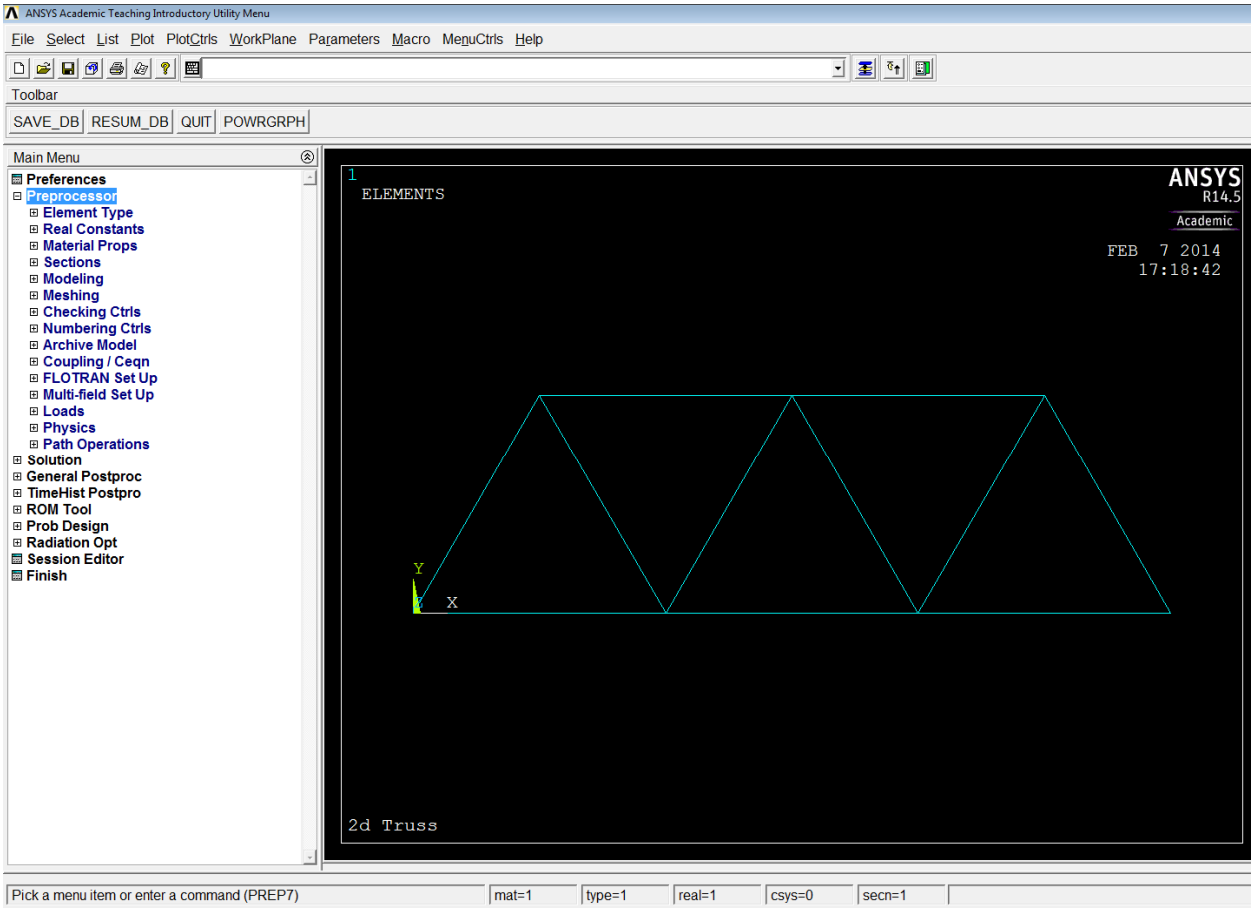
13. Meshing: Preprocessor> Meshing> Size Cntrls> ManualSize> Lines> All Lines. Enter 1 in the NDIV field and then select ok.



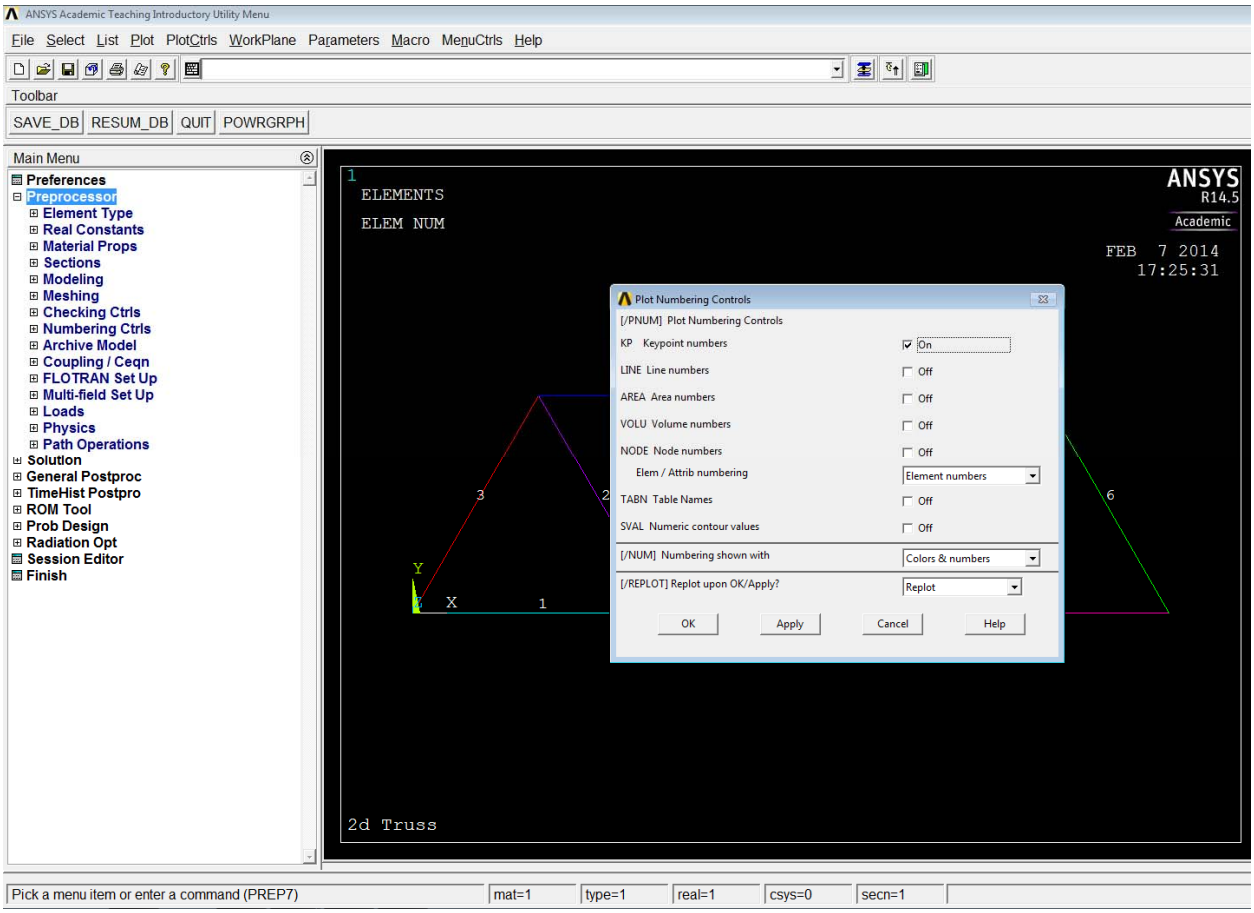
14. Preprocessor> Meshing> Mesh> Lines.



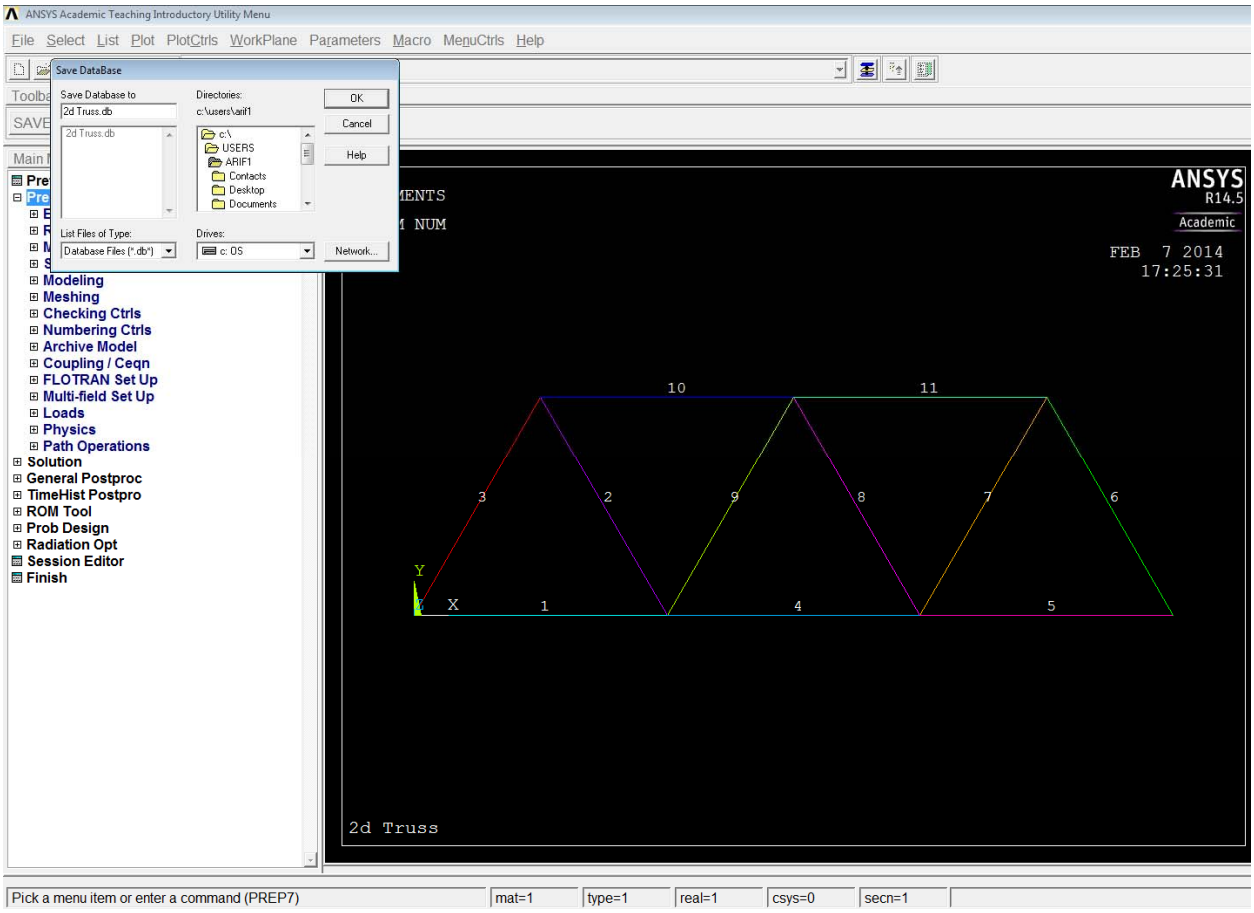
15. Select Pick All and the screen should look like this.



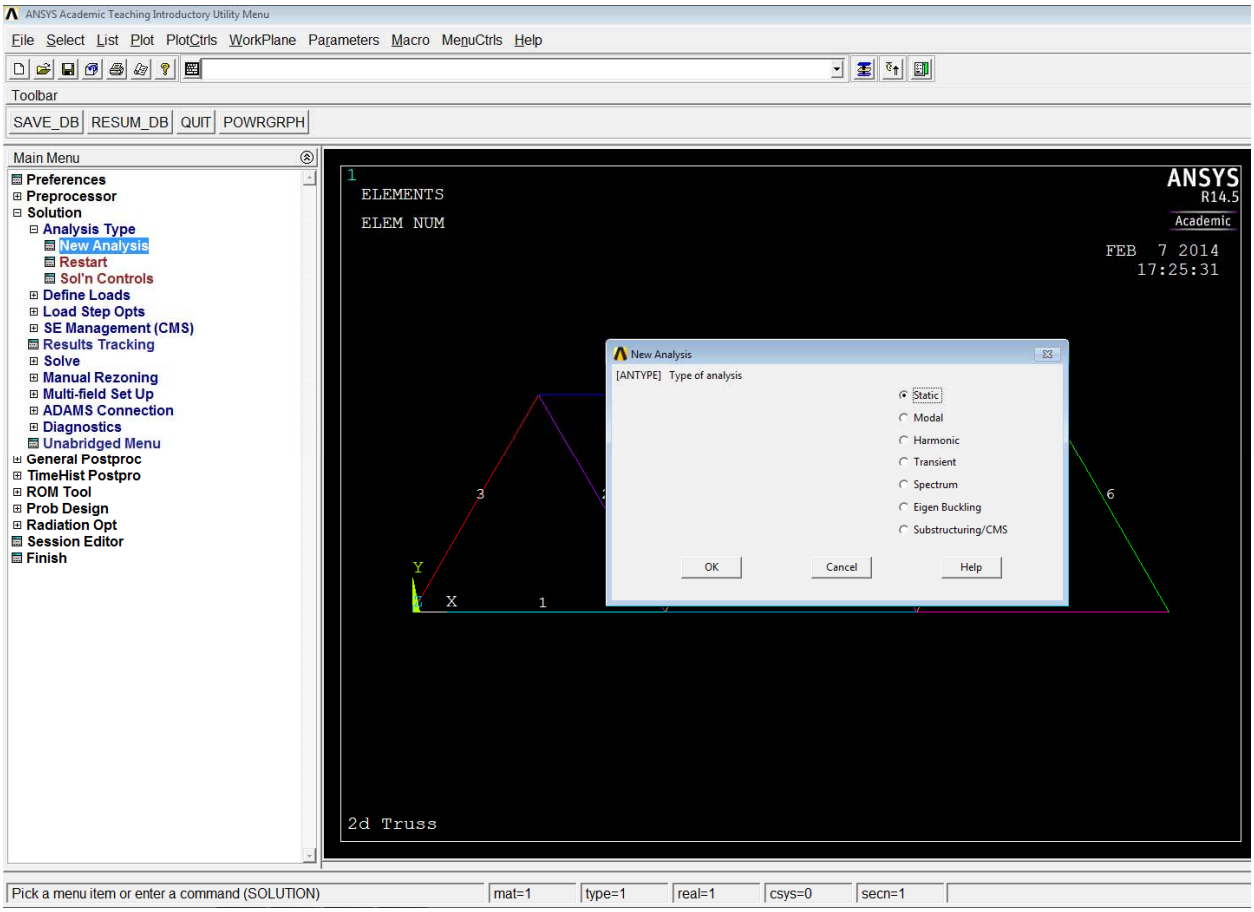
16. Keypoint numbering: Utility Menu> PlotCtrls> Numbering. Select as shown in the figure and select ok.



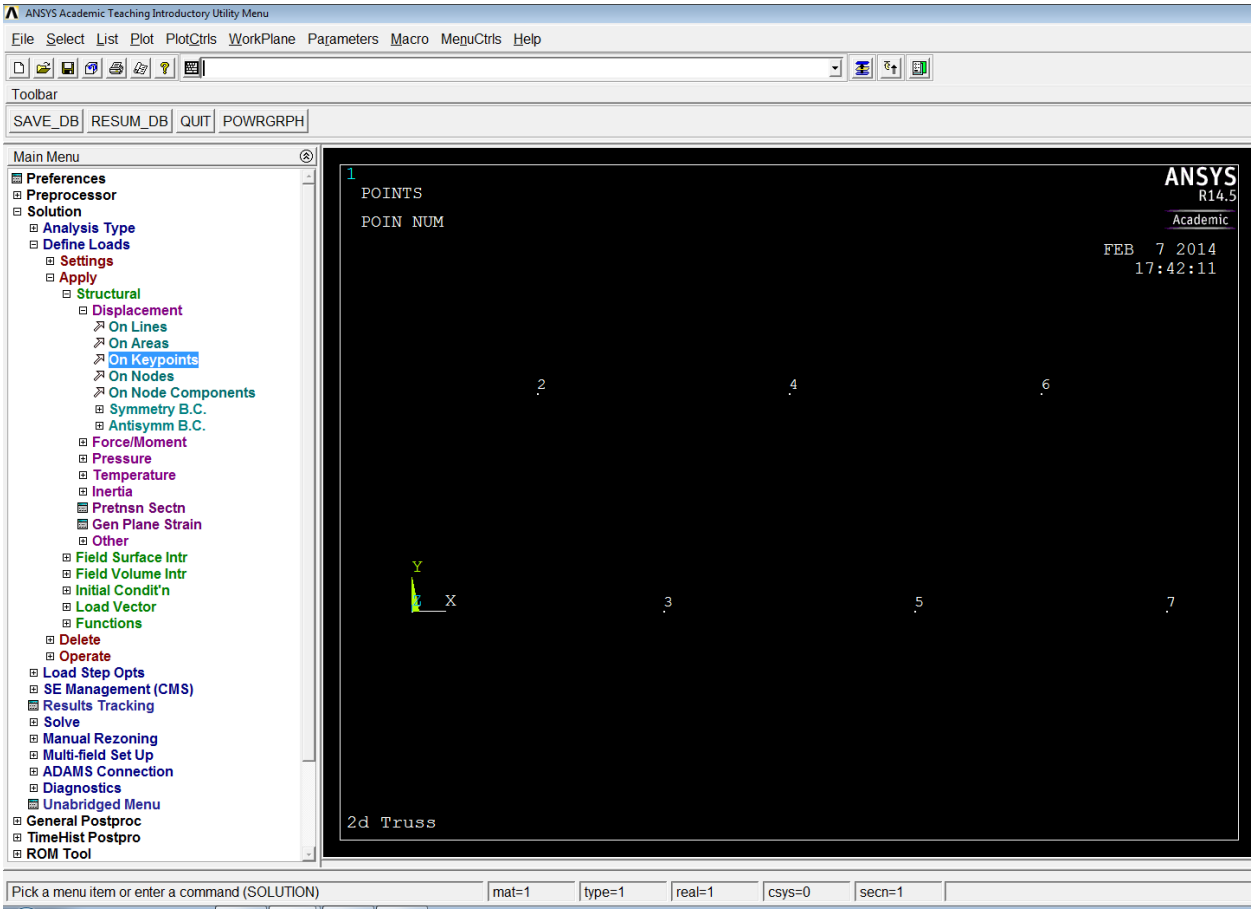
17. Saving: Utility Menu> File> Save as. Enter the file name and select the directory where you would like to save your file and then select ok.



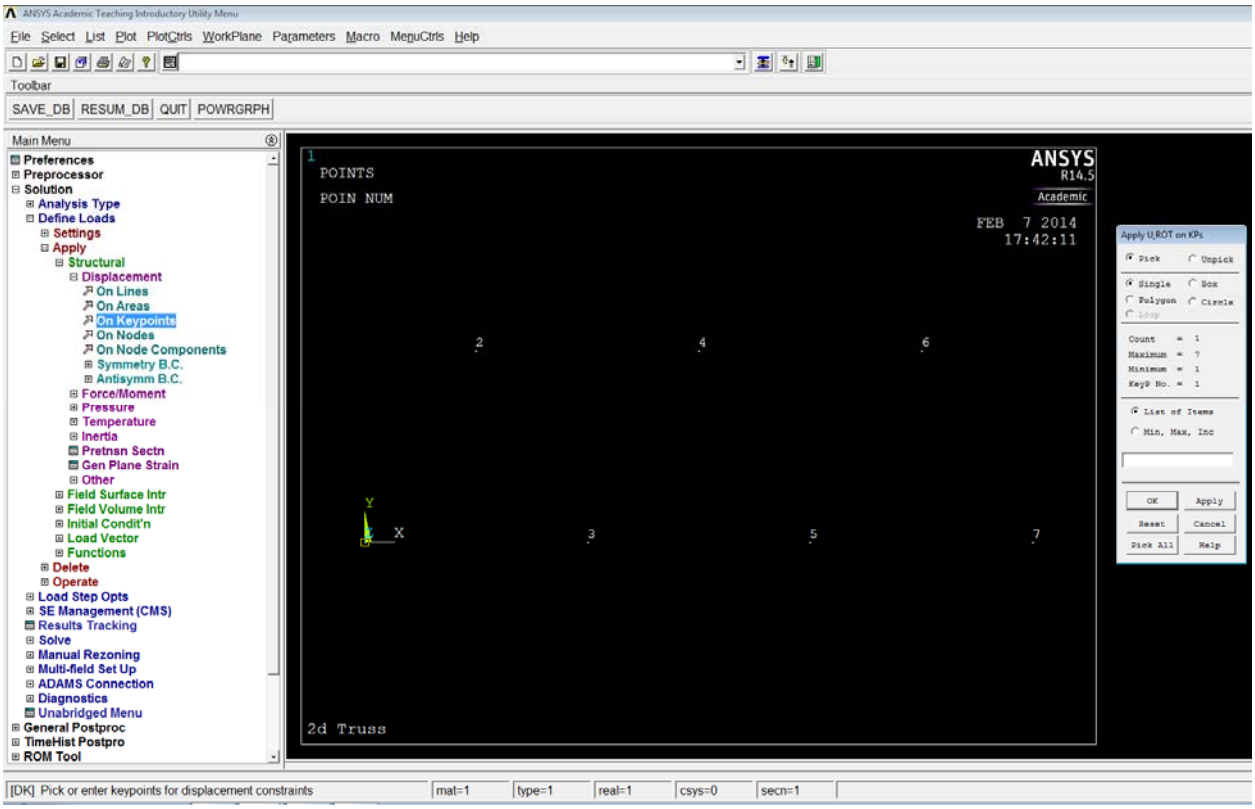
18. Solution type: Main Menu> Solution> Analysis Type> New Analysis. Select as shown and then ok.



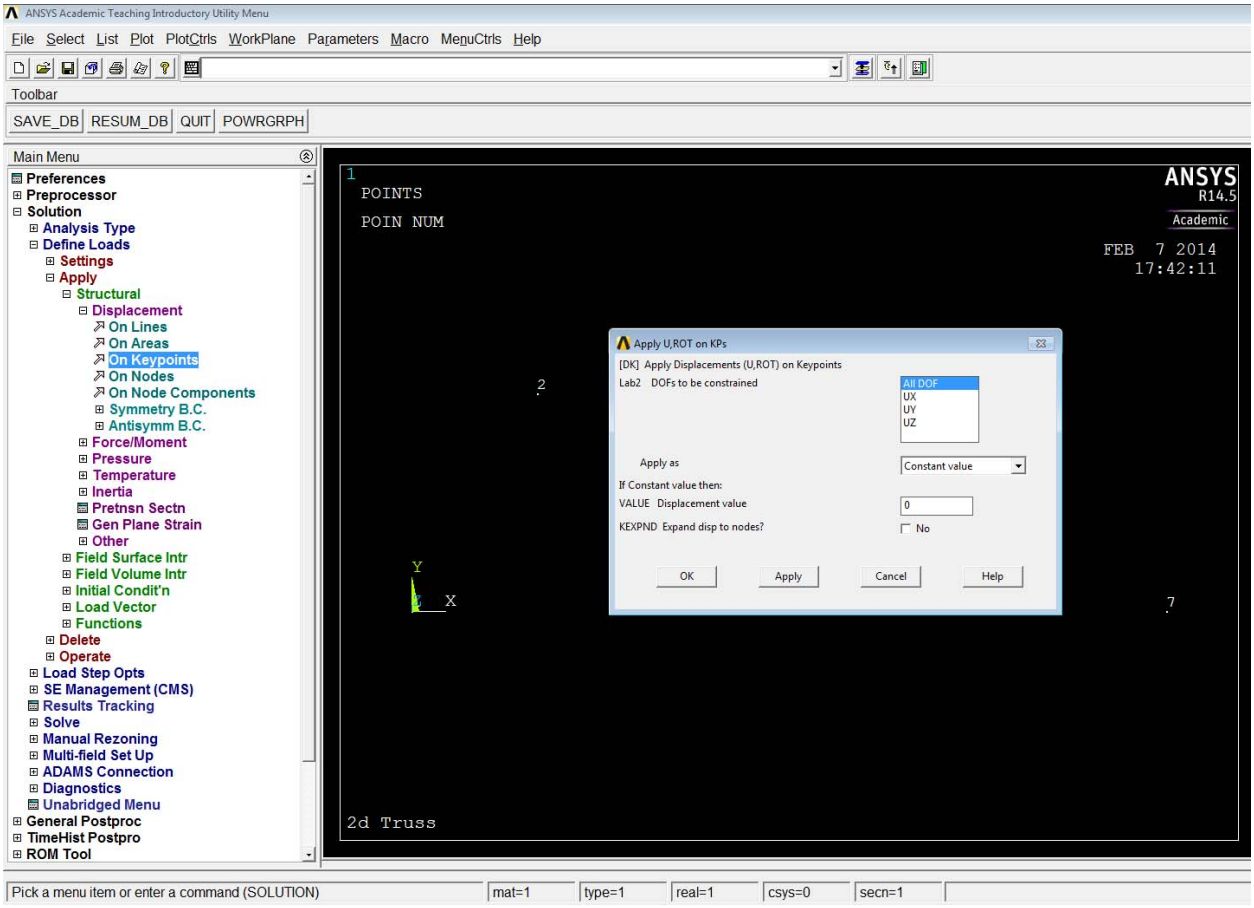
19. Utility Menu> Plot> Keypoints> Keypoints.



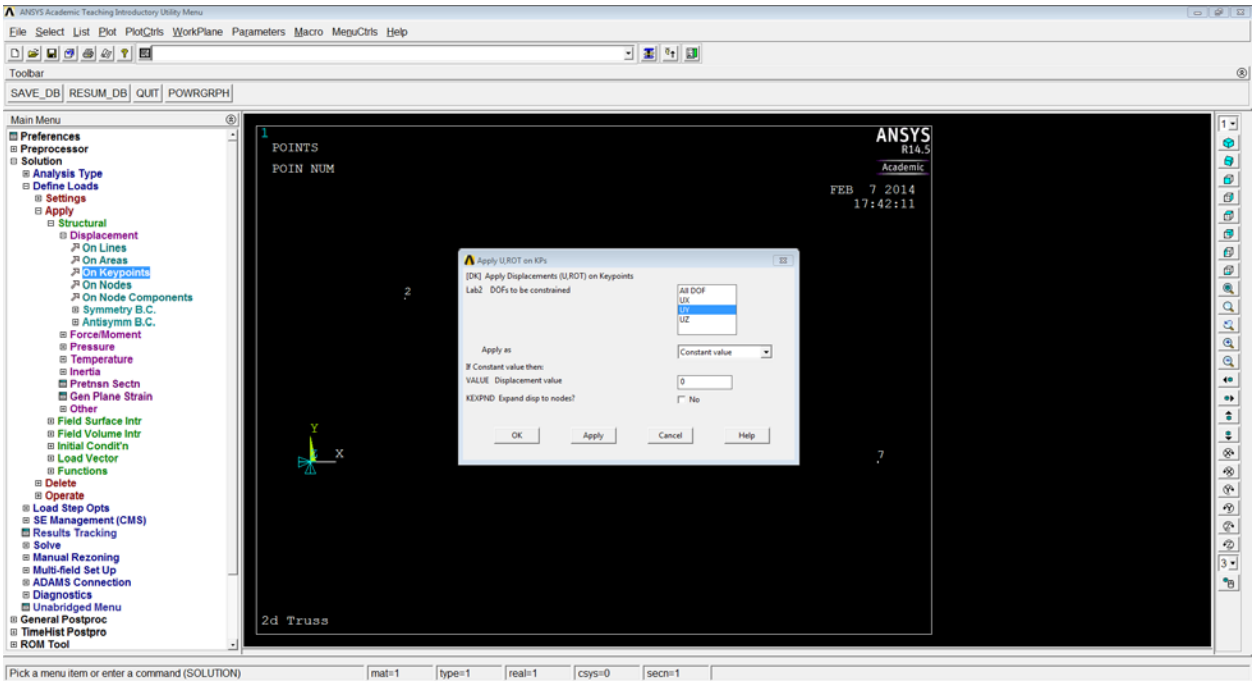
20. Constraints: Main Menu> Solution> Define Loads> Apply> Structural> Displacement> On Keypoints. Select Keypoint 1 and select ok in the selection box.



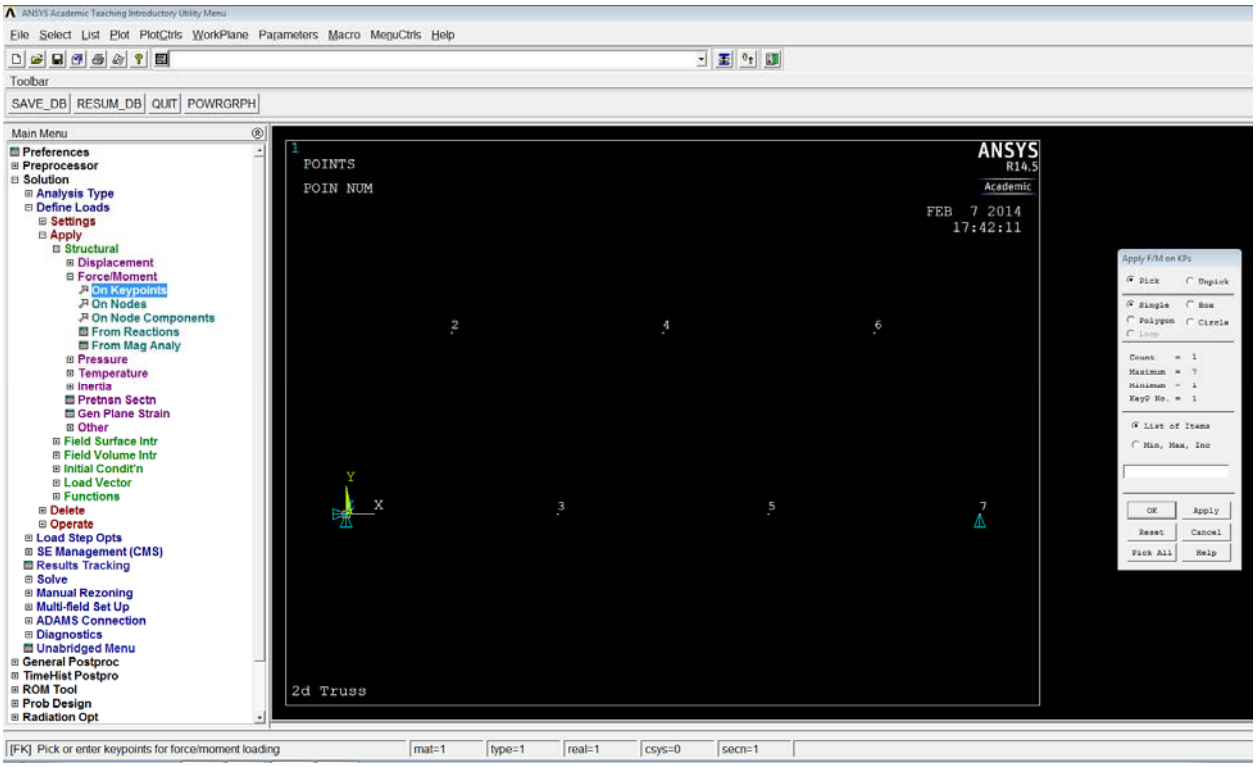
21. Select All DOF, enter 0 in VALUE and then select ok.



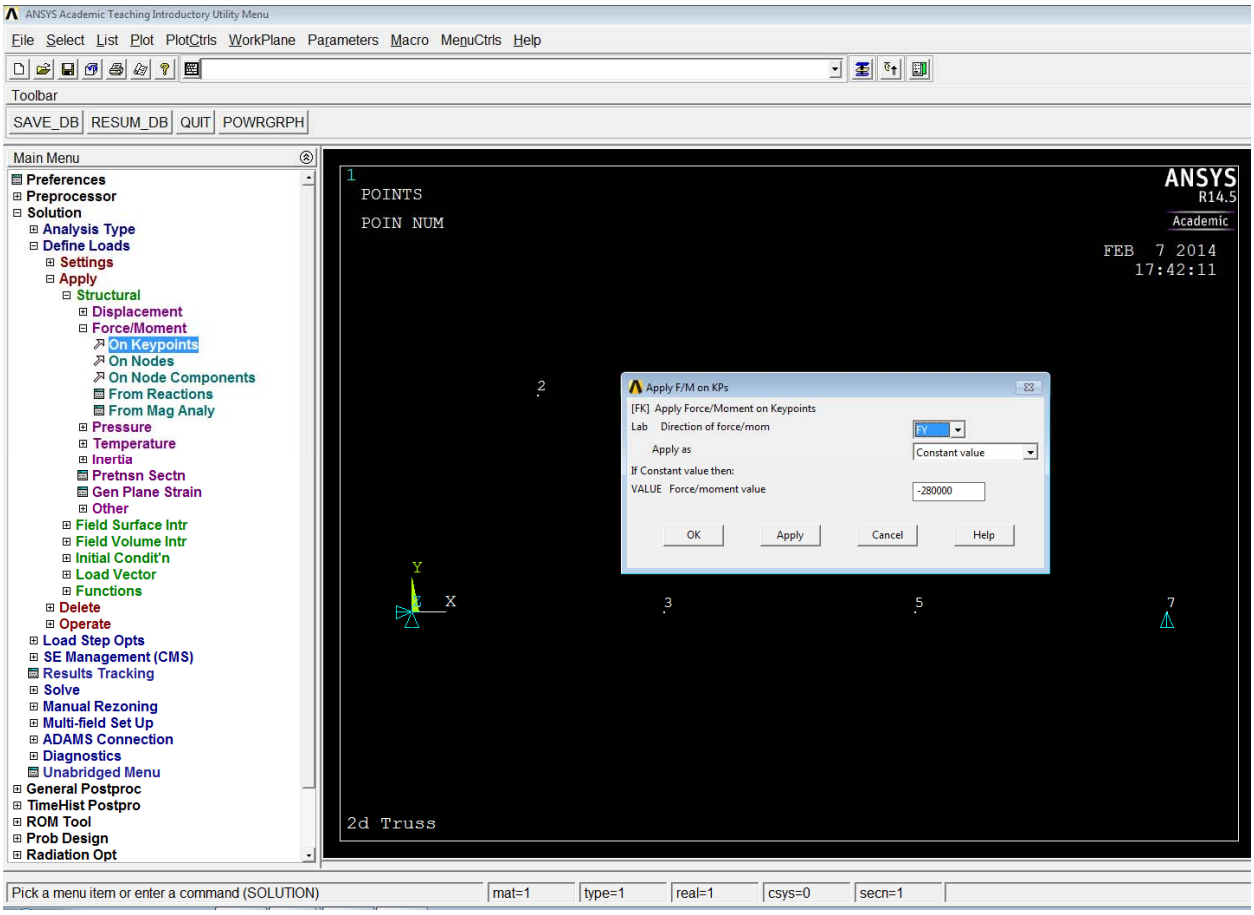
22. Similarly apply the constraint on the right end by deselecting All DOF, selecting UY and finally entering 0.



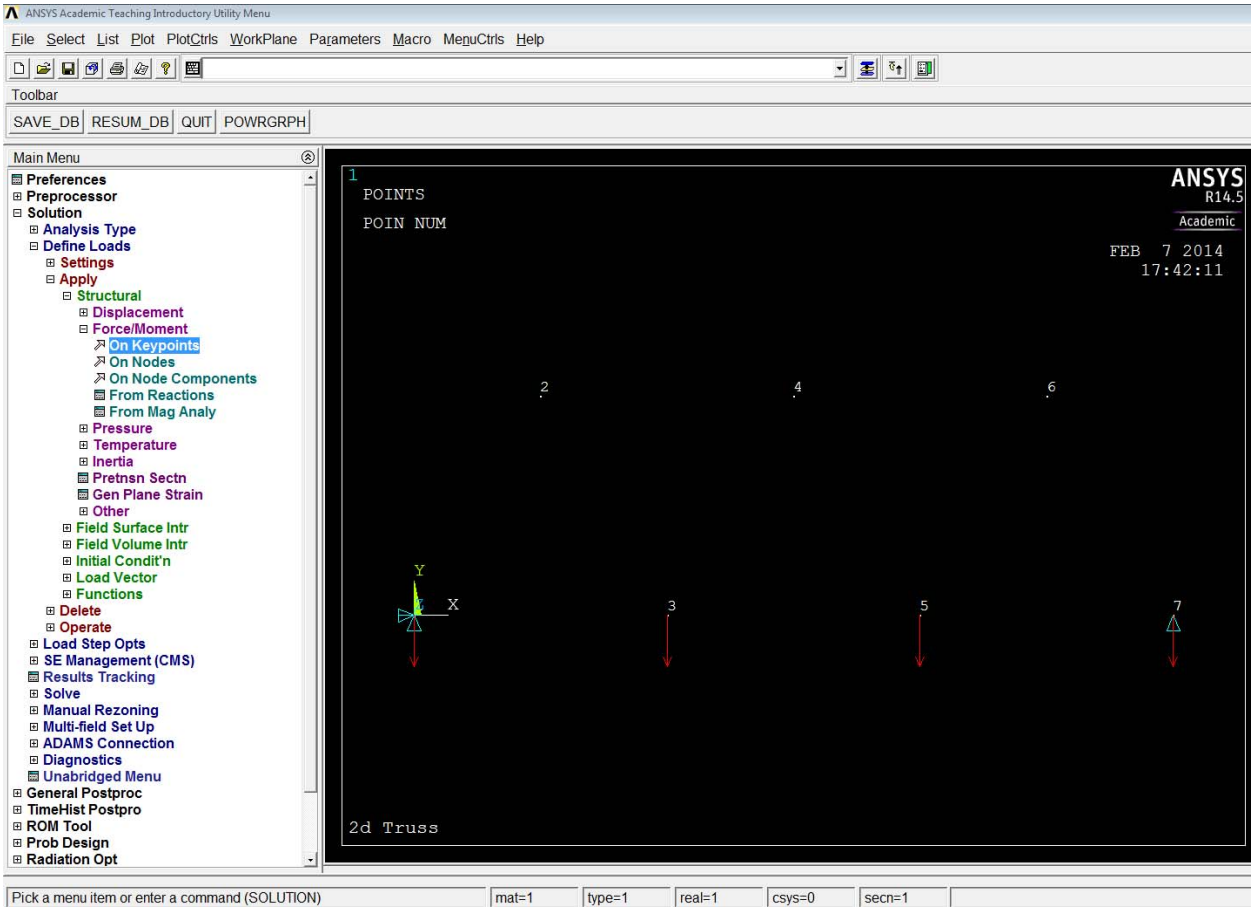
23. Loading: Solution> Define Loads> Apply> Structural> Force/ Moment> On Keypoints. Select Keypoint 1 and then select ok.



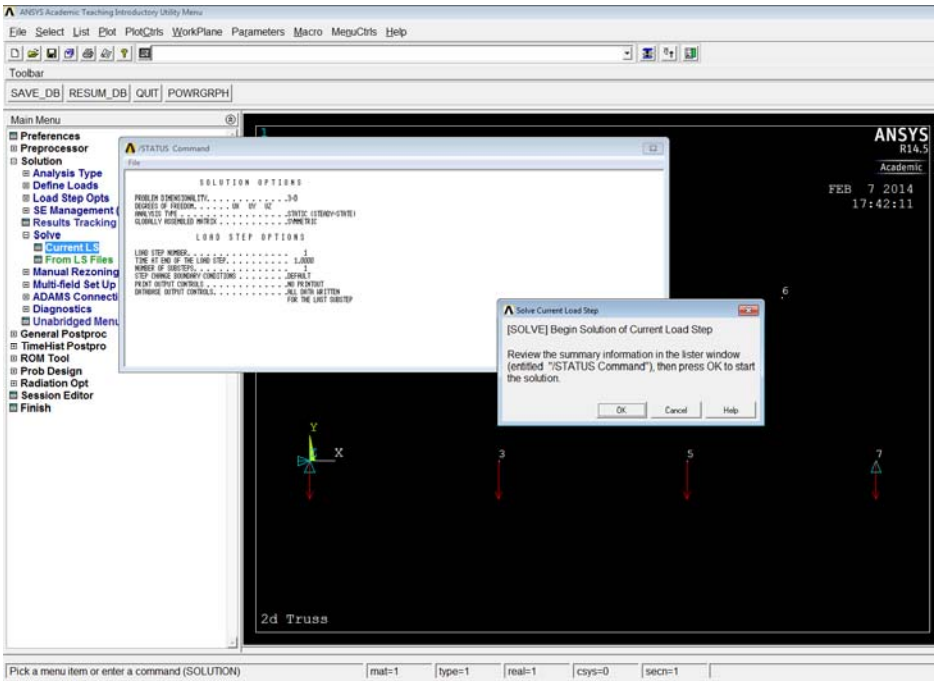
24. Select FY in Direction of force/ mom, Constant value in Apply as and enter -280000 in VALUE. Then select ok.



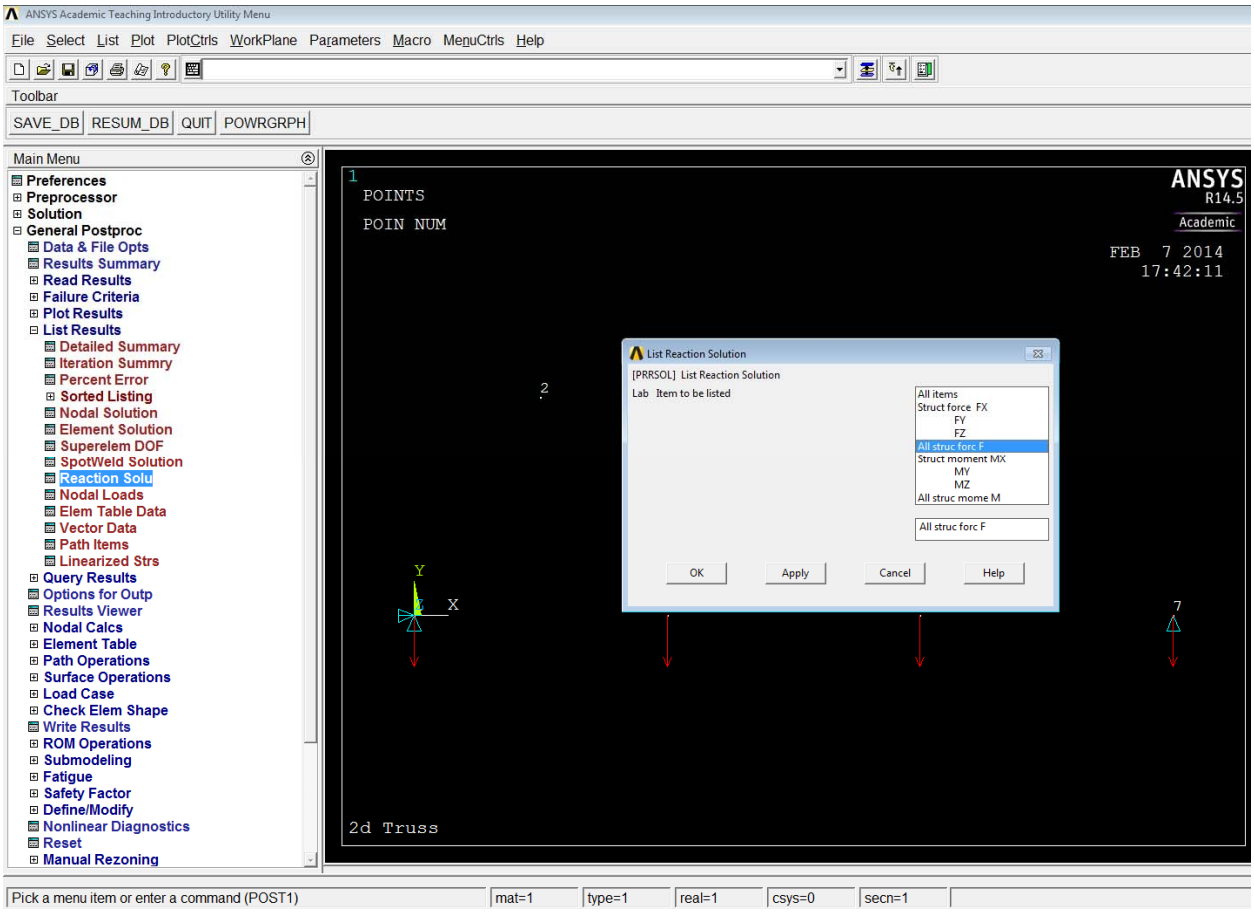
25. Apply the remaining loads similarly. At the end the structure will show the loads.



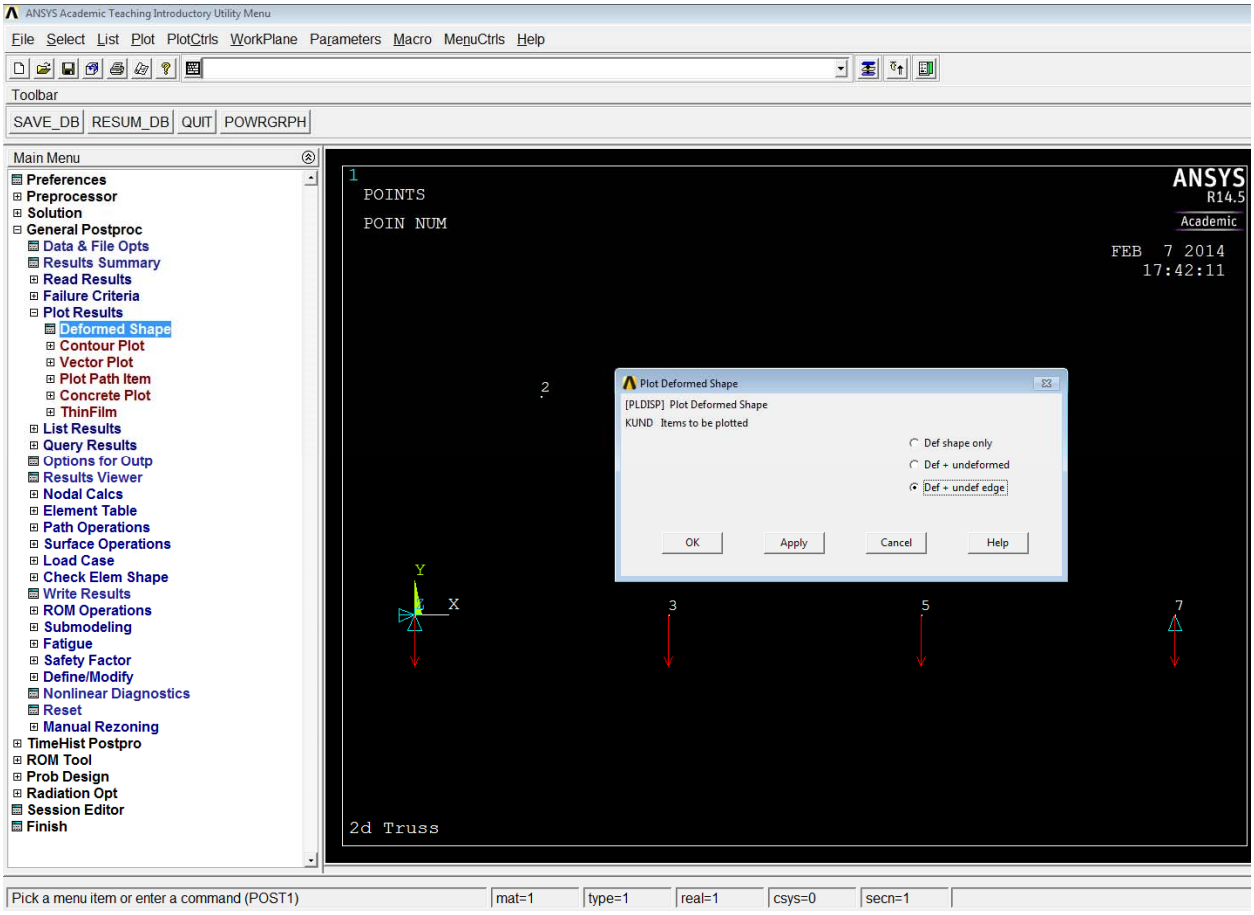
26. Solving: Main Menu> Solution> Solve> Current LS. Then select ok. Once solution is done close the box.



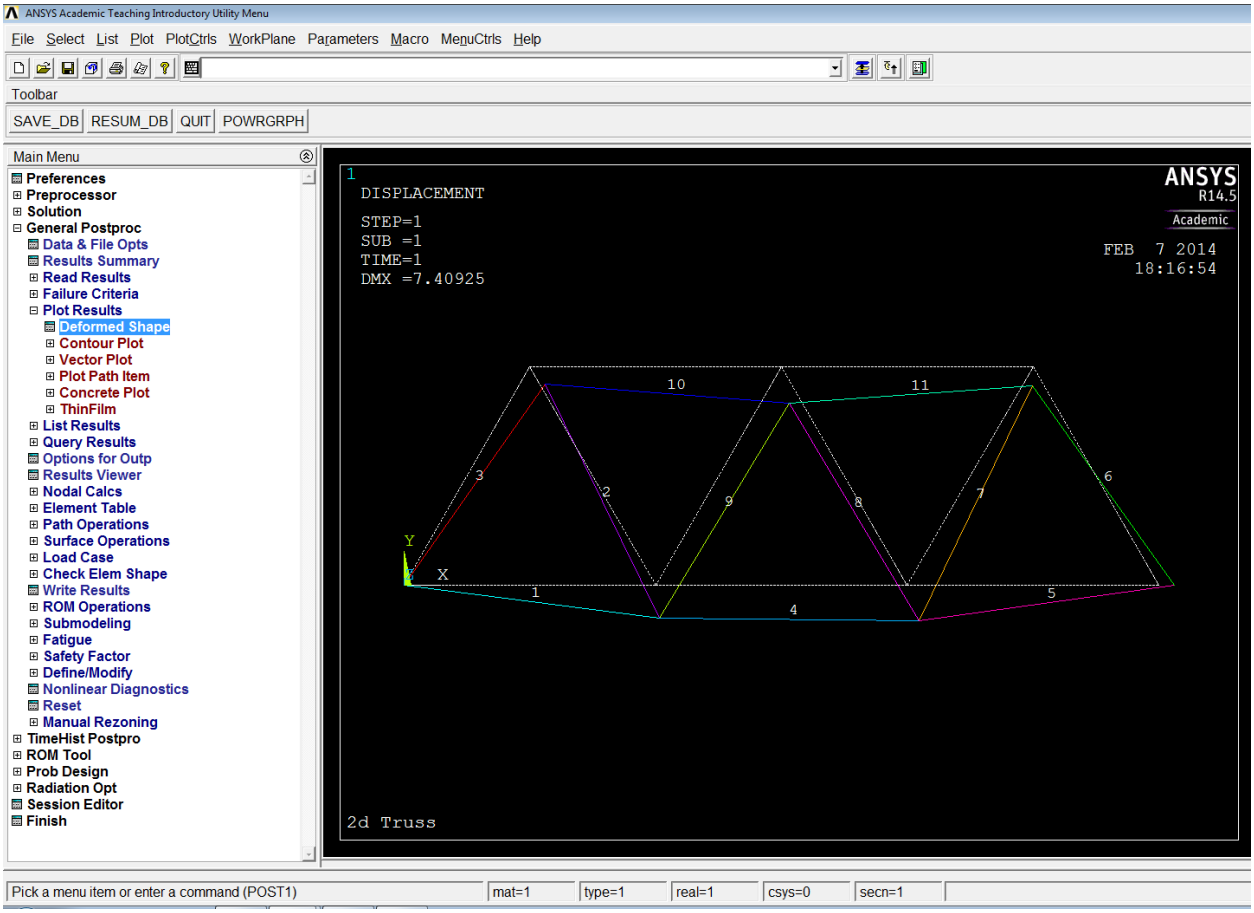
27. Reaction forces: General Postproc> List Results> Reaction Solu. Select All struc forc F and then select ok. The window will give the reaction force results.



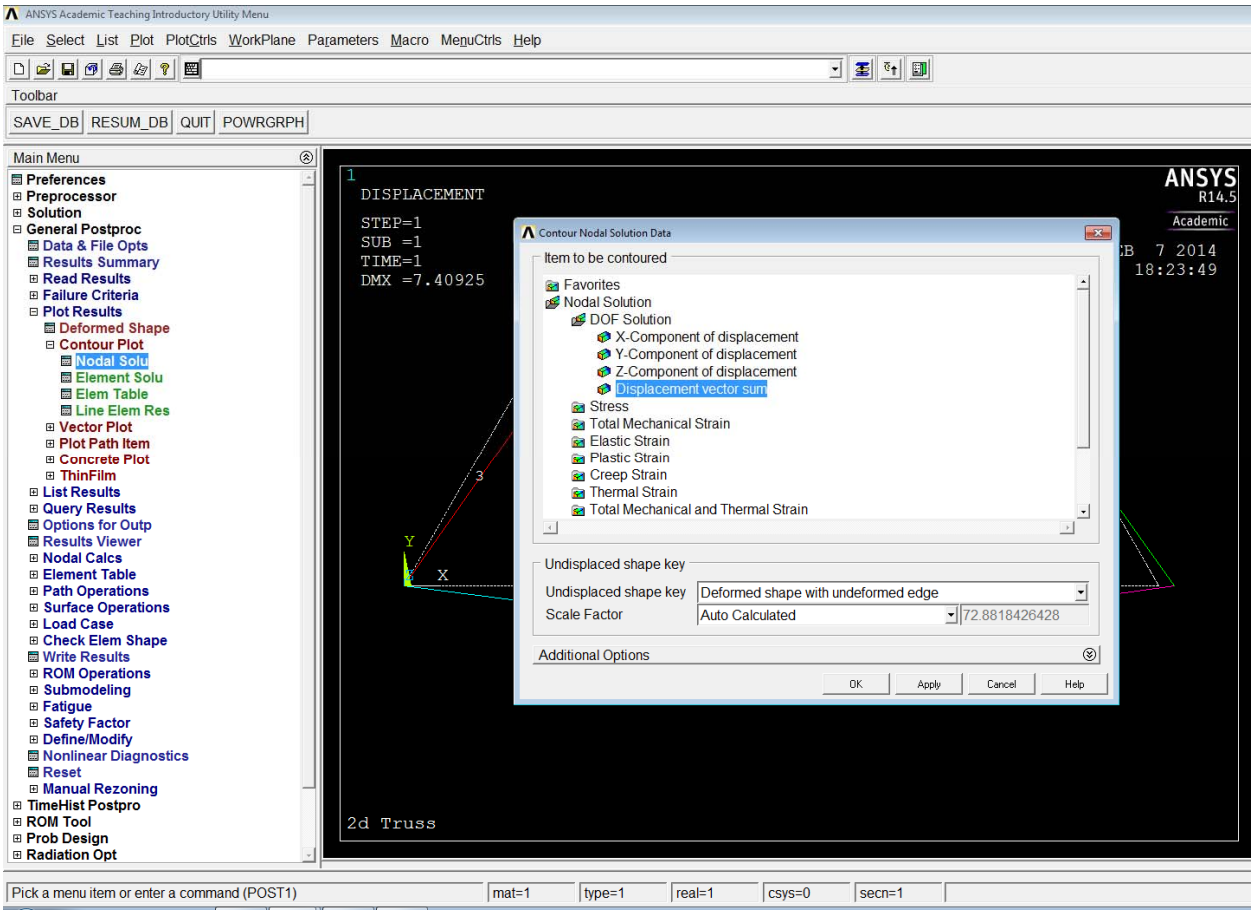
28. Deflection : General Postproc> Plot Results> Deformed Shape. Select Def + undef edge and then ok.



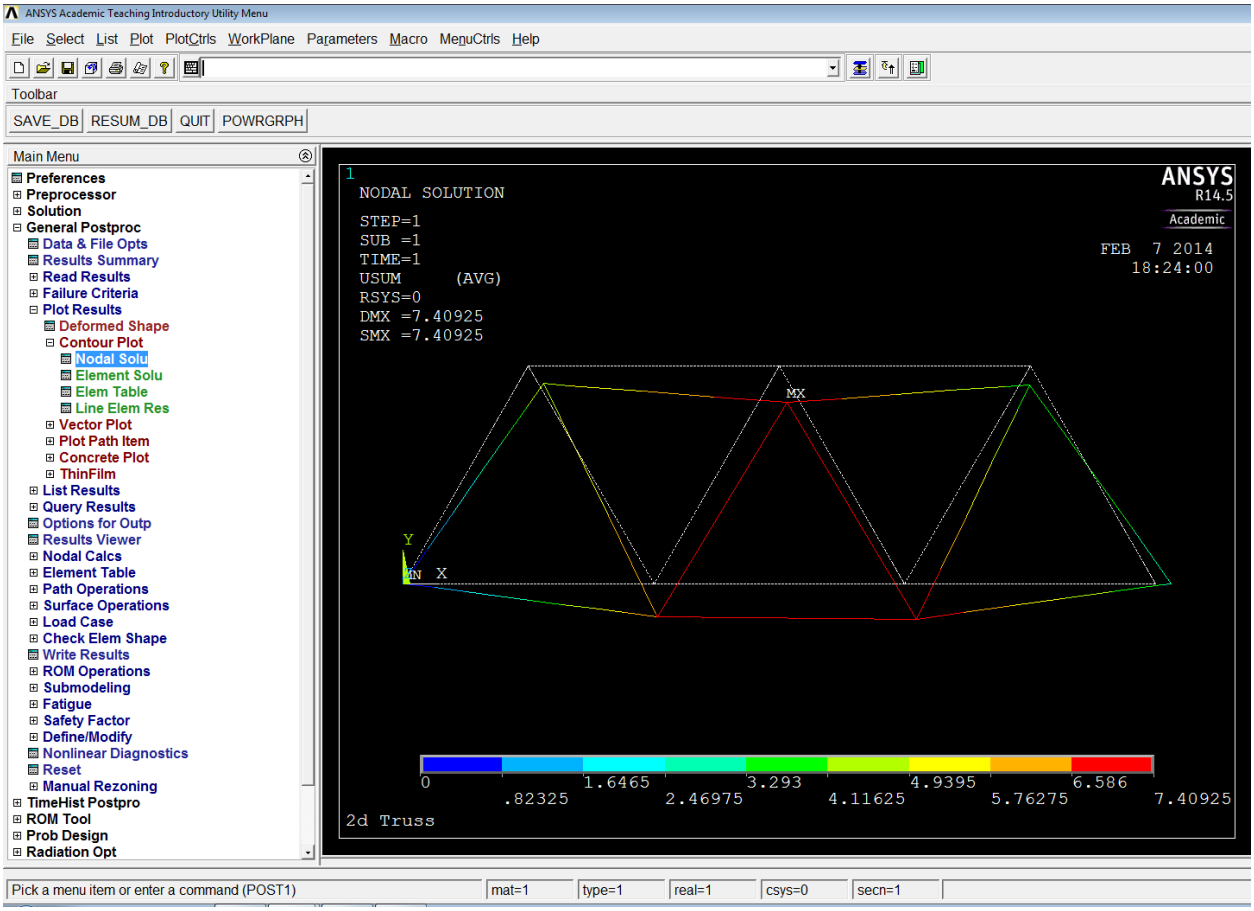
29. The screen should show the deflection plot.



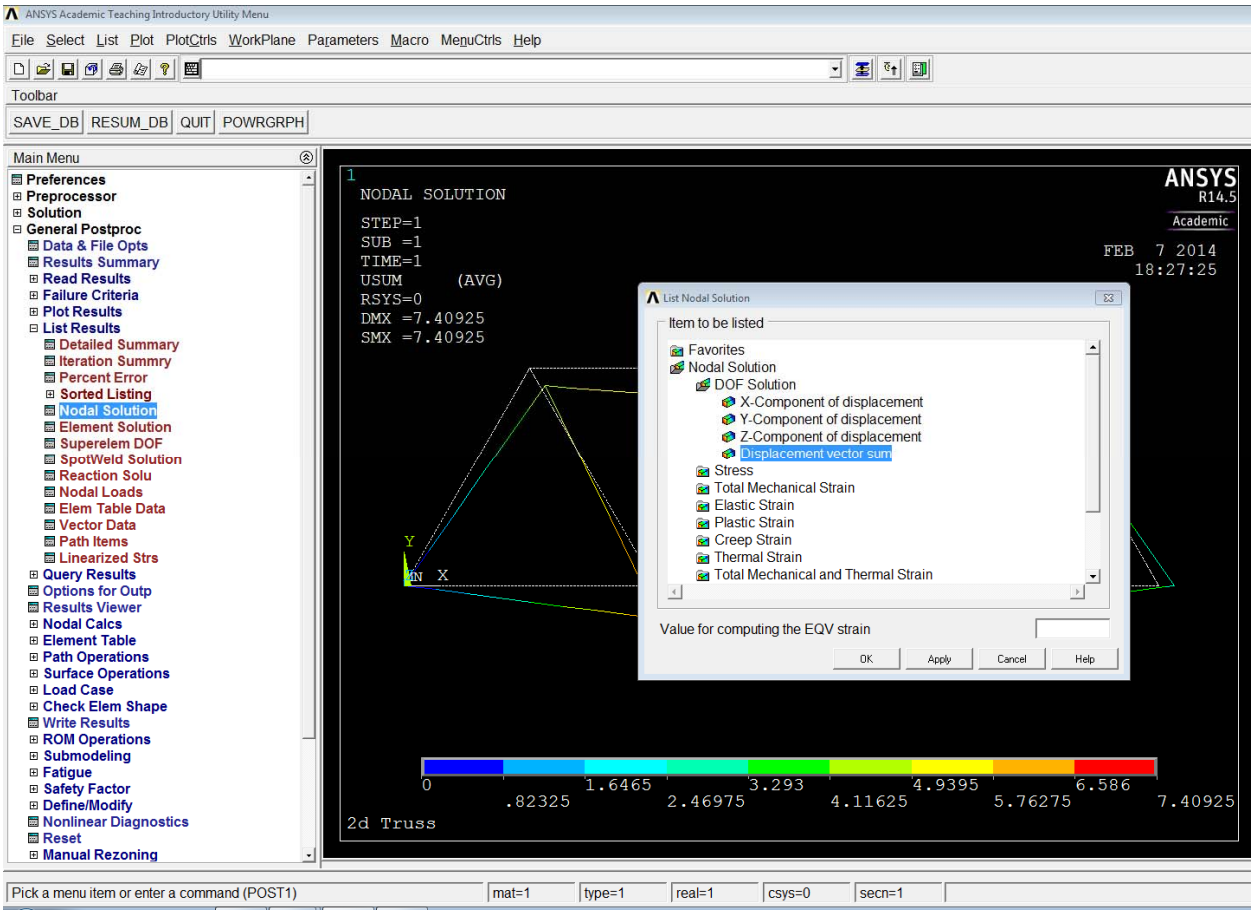
30. Deflection contour: General Postproc> Plot results> Contour Plot> Nodal Solu. Then select as shown in the figure and select ok.



31. The screen should show the displacement contour plot.



32. Listing deflection: General Postproc> List Results> Nodal Solution. Then select DOF solution and Displacement vector sum as before. Then select ok.



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.