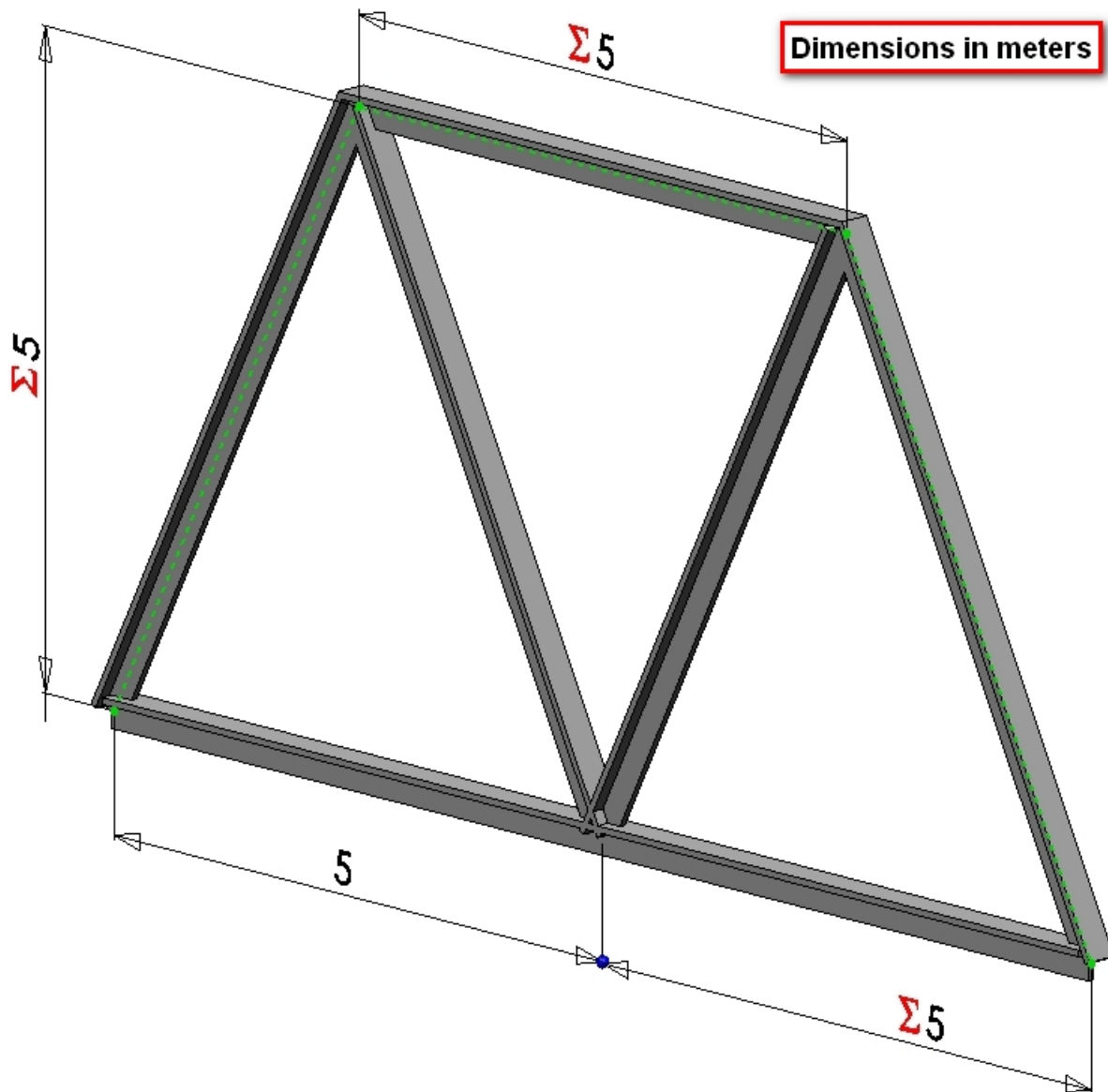


# Abaqus Beam Tutorial (ver. 6.12)

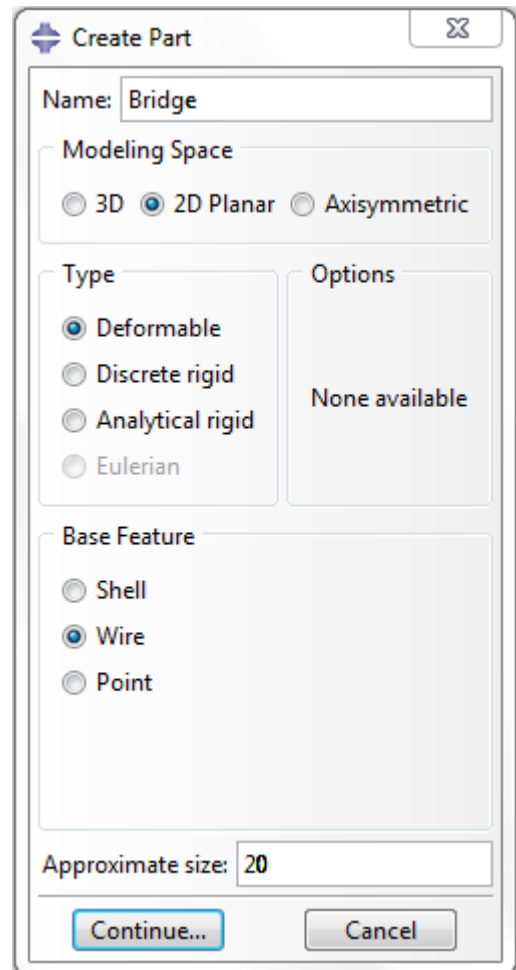
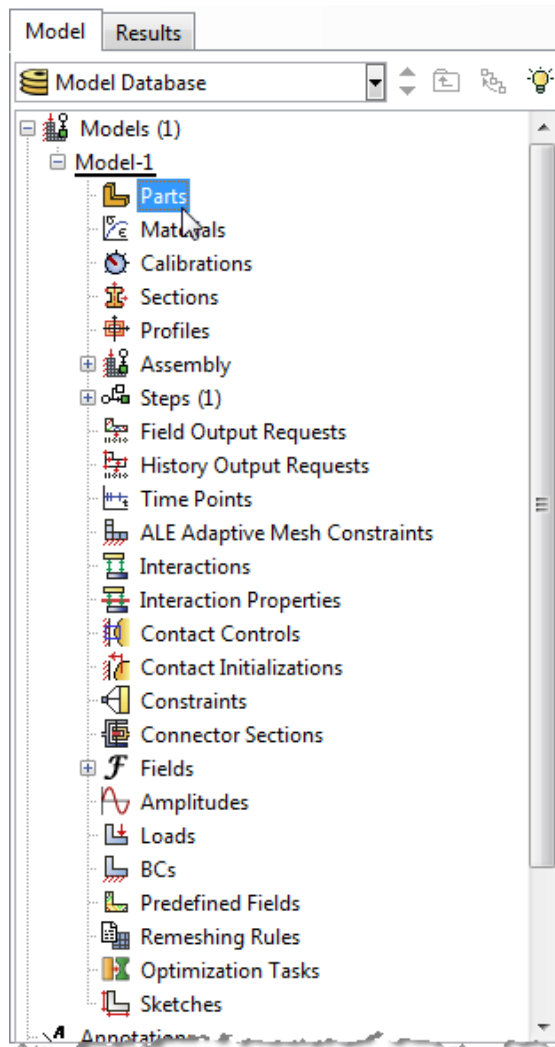
## Problem Description

The two-dimensional bridge structure is simply supported at its lower corners. The structure is composed of steel T-sections ( $E = 210 \text{ GPa}$ ,  $\nu = 0.25$ ) oriented as shown below. A uniform distributed load of  $1000 \text{ N/m}$  is applied to the lower horizontal members in the vertical downward direction. Determine the stresses and the vertical displacements.



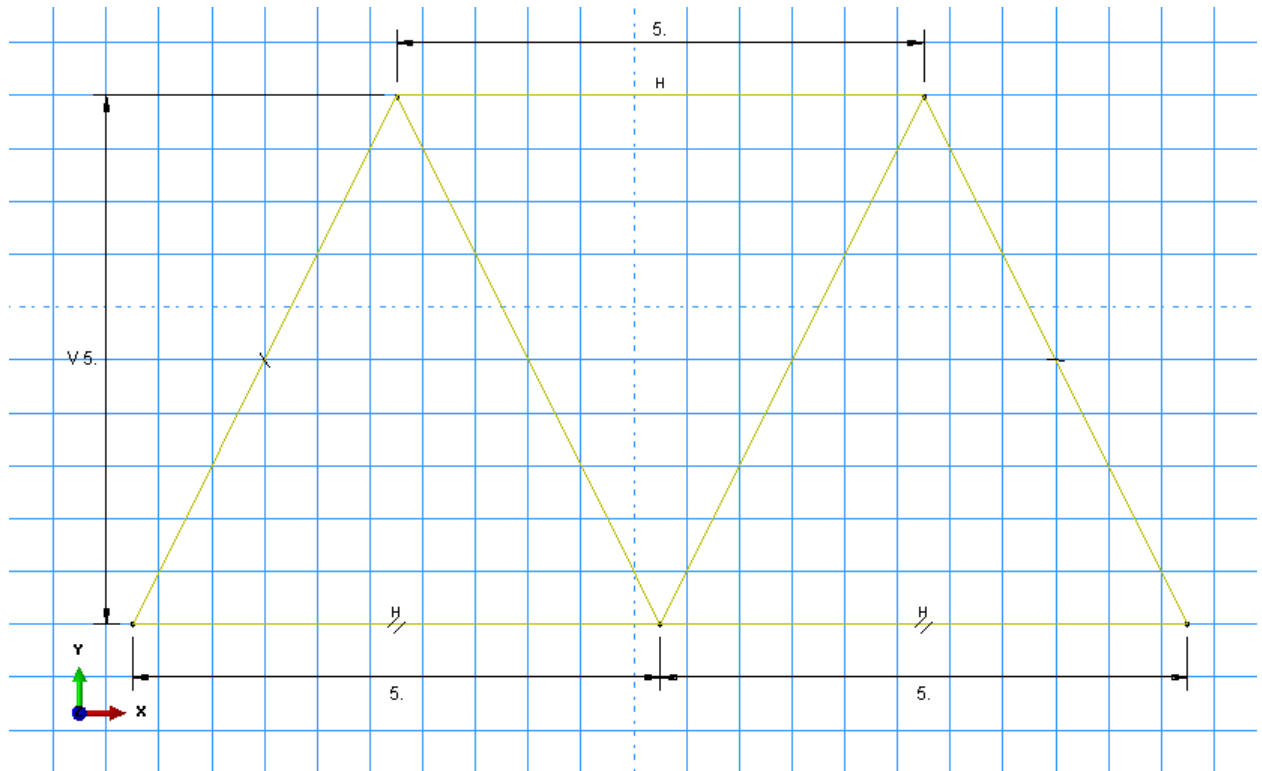
## Analysis Steps

1. Start Abaqus and choose to create a new model database
2. In the model tree double click on the “Parts” node (or right click on “parts” and select Create)



3. In the Create Part dialog box (shown above)
  - a. name the part
  - b. Select “2D Planar”
  - c. Select “Deformable”
  - d. Select “Wire”
  - e. Set approximate size = 20
  - f. Click “Continue...”

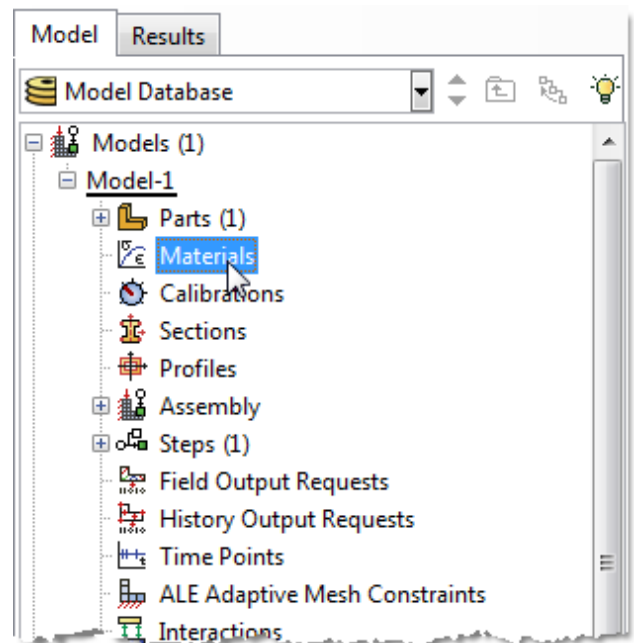
4. Create the geometry shown below (not discussed here)

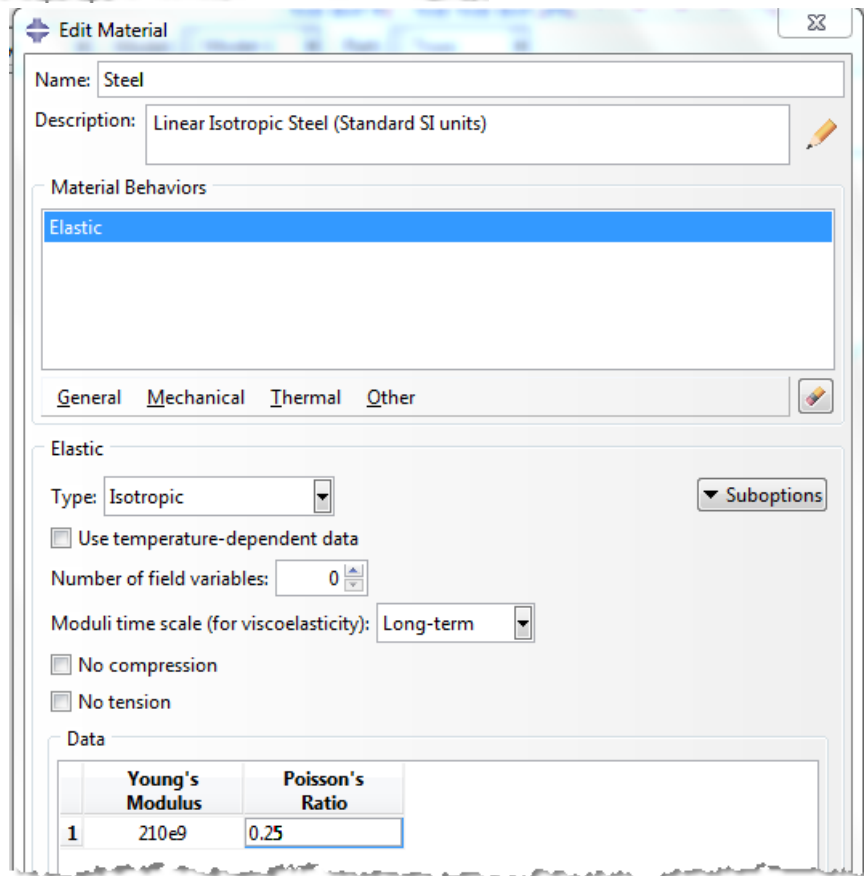
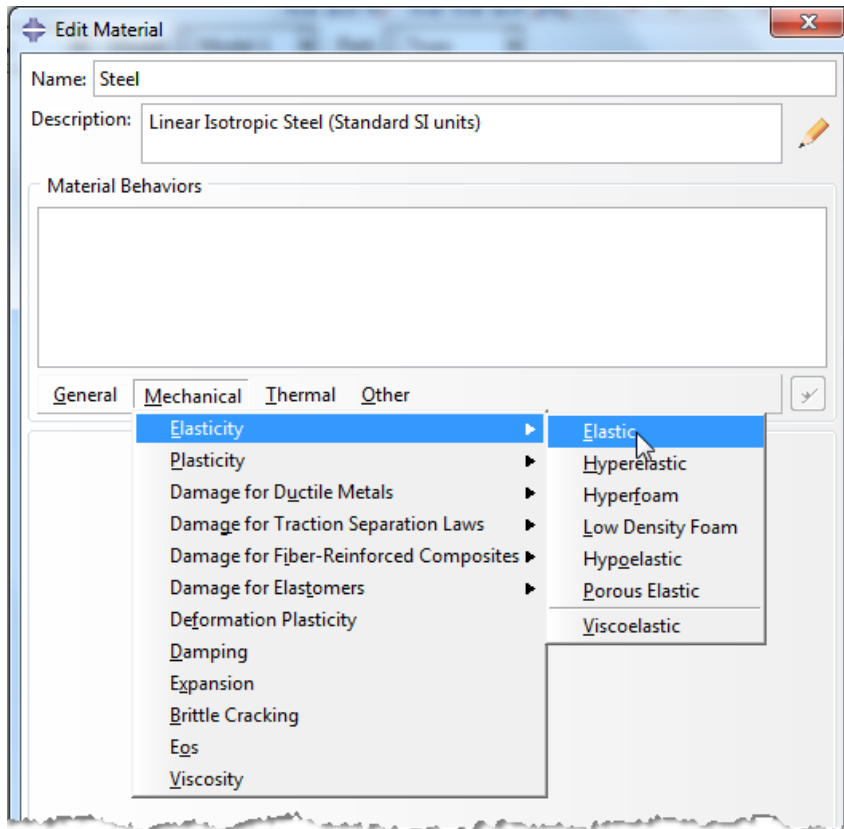


5. Double click on the “Materials” node in the model tree
  - a. Name the new material and give it a description
  - b. Click on the “Mechanical” tab → Elasticity → Elastic
  - c. Define Young’s Modulus and Poisson’s Ratio (use SI units)

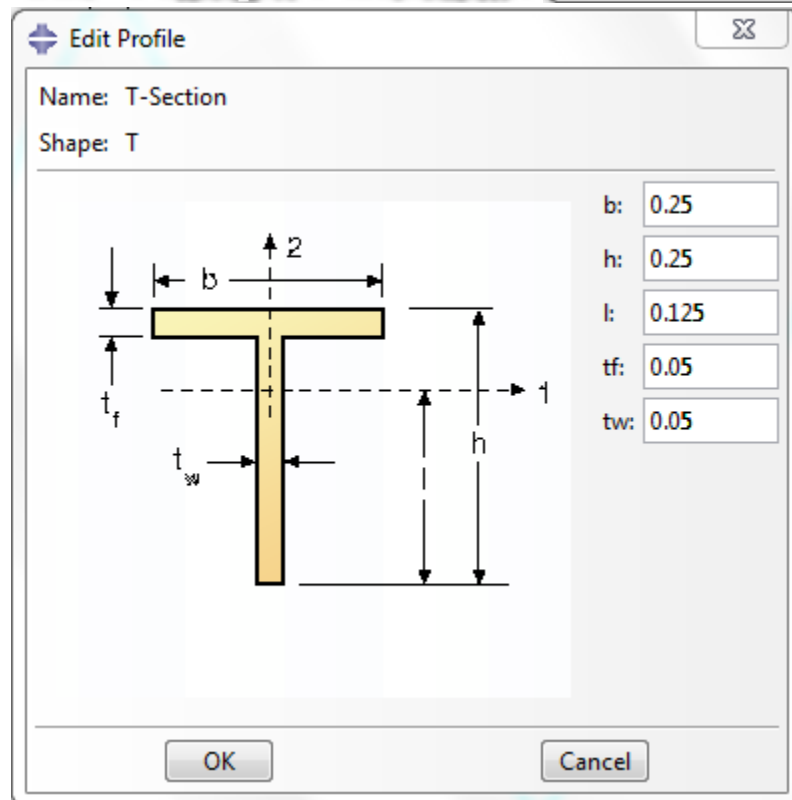
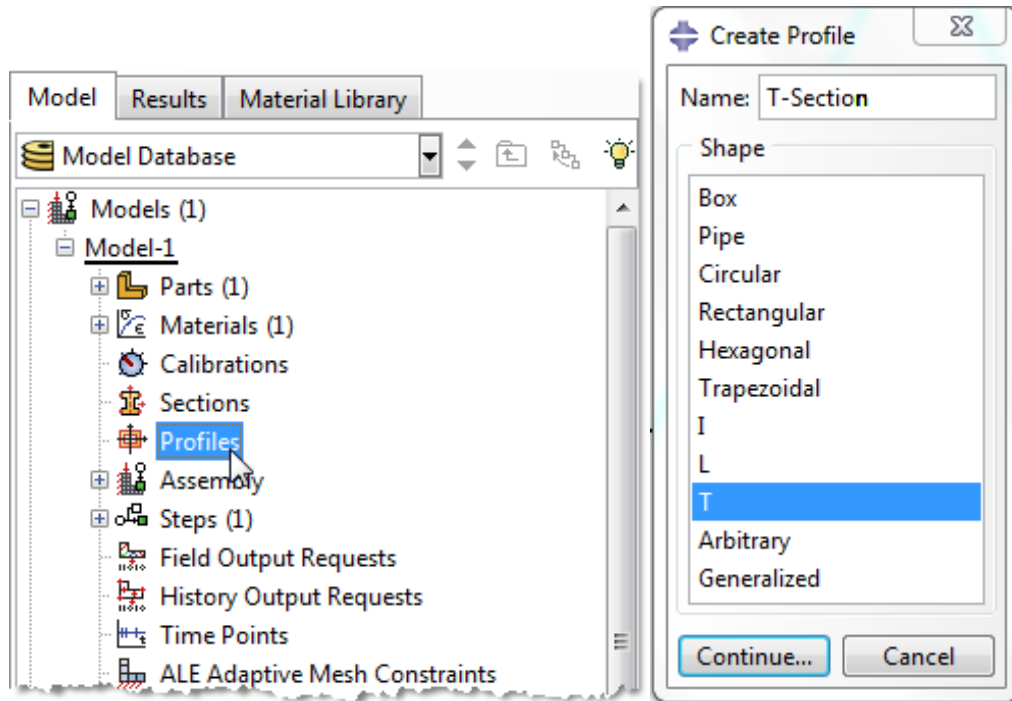
**WARNING: There are no predefined systems of units within Abaqus, so the user is responsible for ensuring that the correct values are specified**

- d. Click “OK”

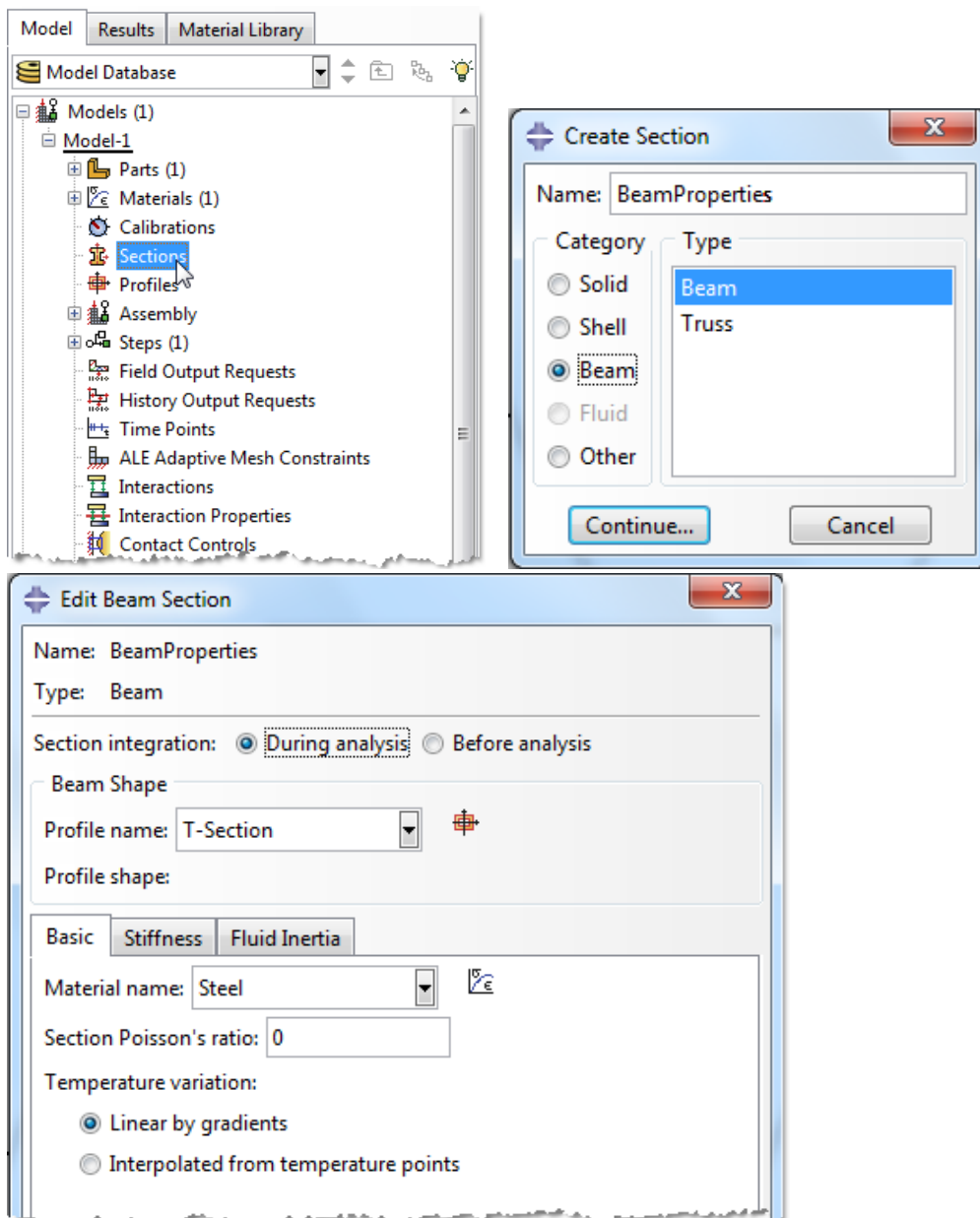




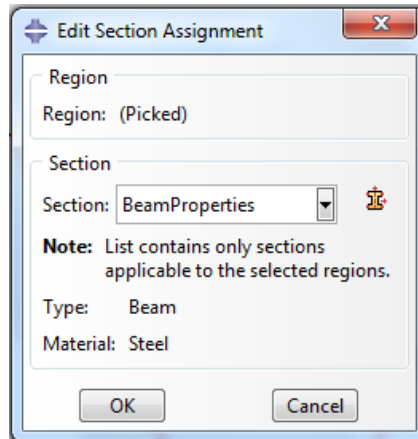
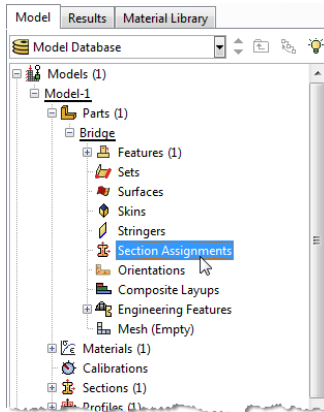
- 6. Double click on the “Profiles” node in the model tree
  - a. Name the profile and select “T” for the shape
  - b. Note that the “T” shape is one of several predefined cross-sections
  - c. Click “Continue...”
  - d. Enter the values for the profile shown below
  - e. Click “OK”



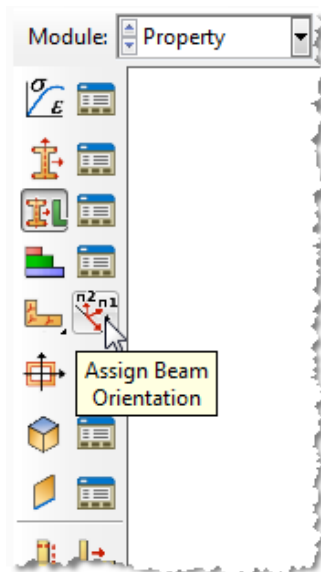
7. Double click on the “Sections” node in the model tree
  - a. Name the section “BeamProperties”
  - b. Select “Beam” for both the category and the type
  - c. Click “Continue...”
  - d. Leave the section integration set to “During Analysis”
  - e. Select the profile created above (T-Section)
  - f. Select the material created above (Steel)
  - g. Click “OK”



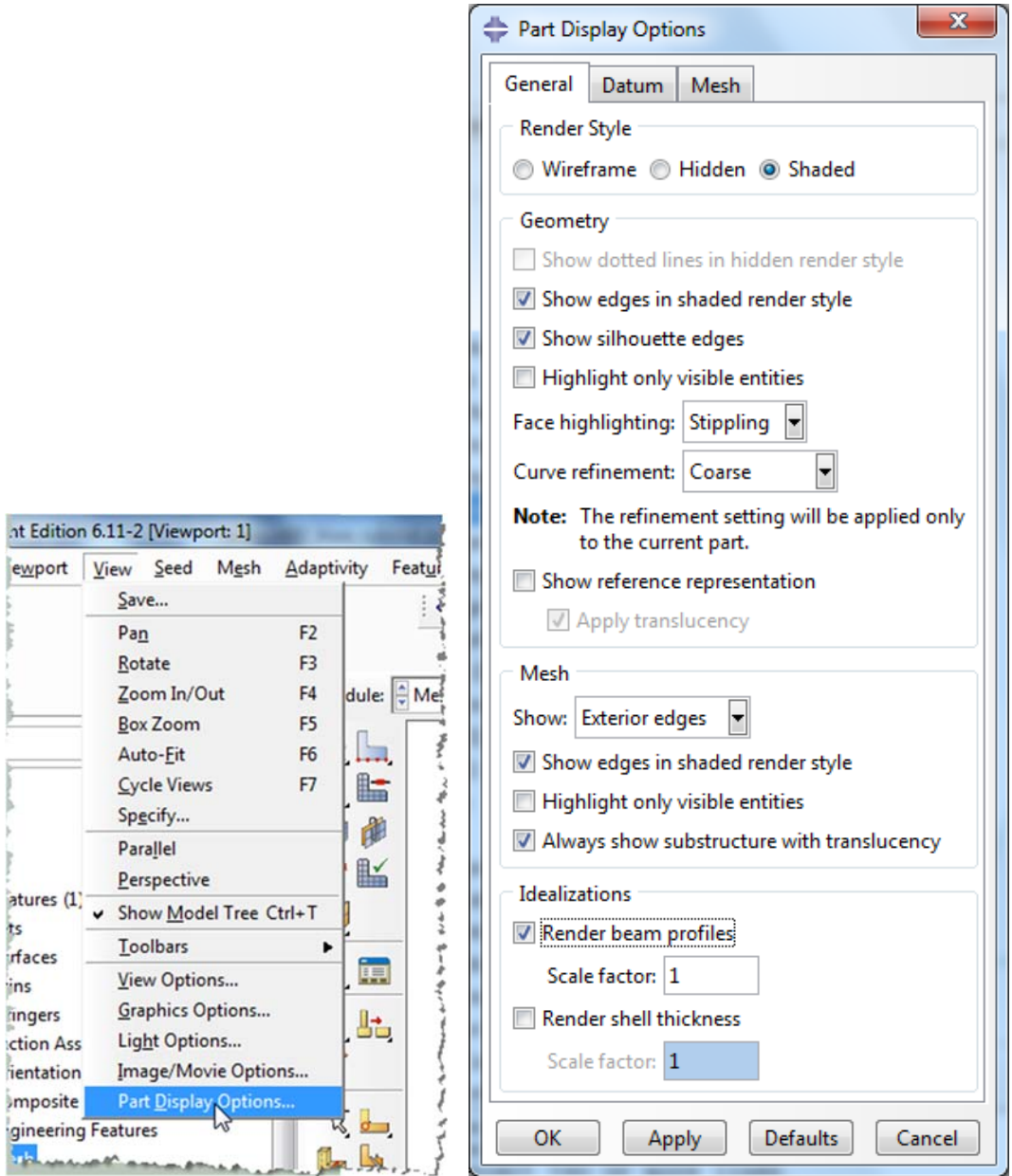
8. Expand the “Parts” node in the model tree, expand the node of the part just created, and double click on “Section Assignments”
  - a. Select the entire geometry in the viewport and press Enter
  - b. Select the section created above (BeamProperties)
  - c. Click “OK”



9. Click on the “Assign Beam Orientation” icon
  - a. Select the entire geometry from the viewport
  - b. Click “Done” in the prompt area
  - c. Accept the default value of the approximate n1 direction by pressing “Enter”
  - d. Click “OK”

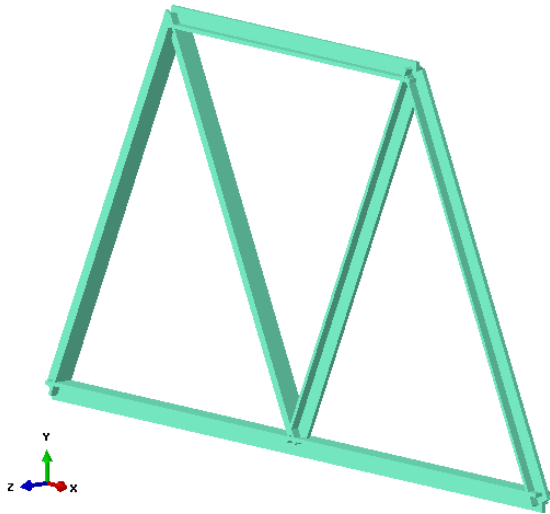


- 10. In the menu bar select View → Part Display Options
  - a. Check the Render beam profiles option on the General tab
  - b. Click “OK”

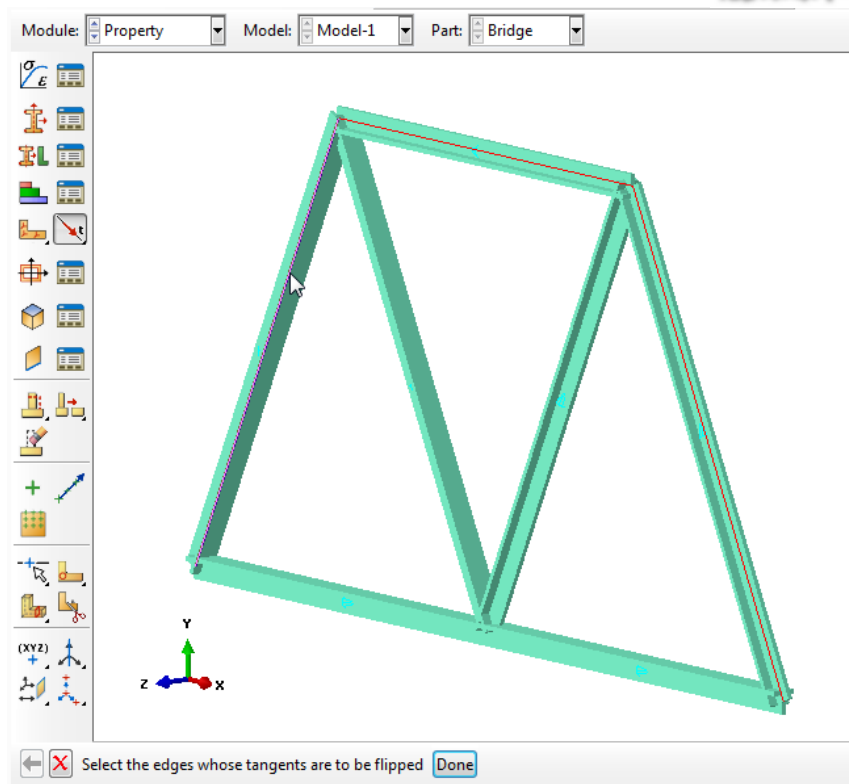
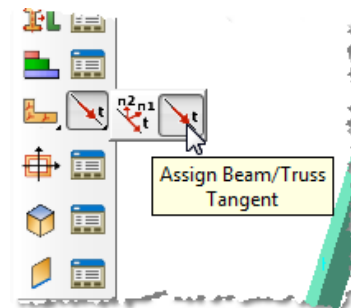




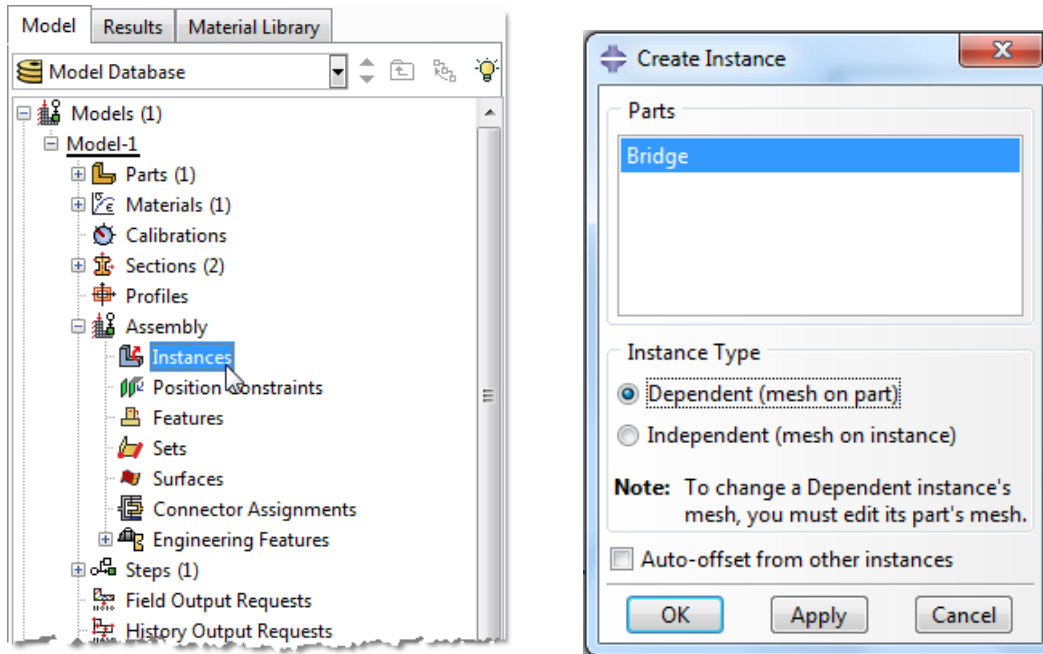
11. Note that the preview shows that the beam cross sections are not all orientated as desired (see Problem Description)



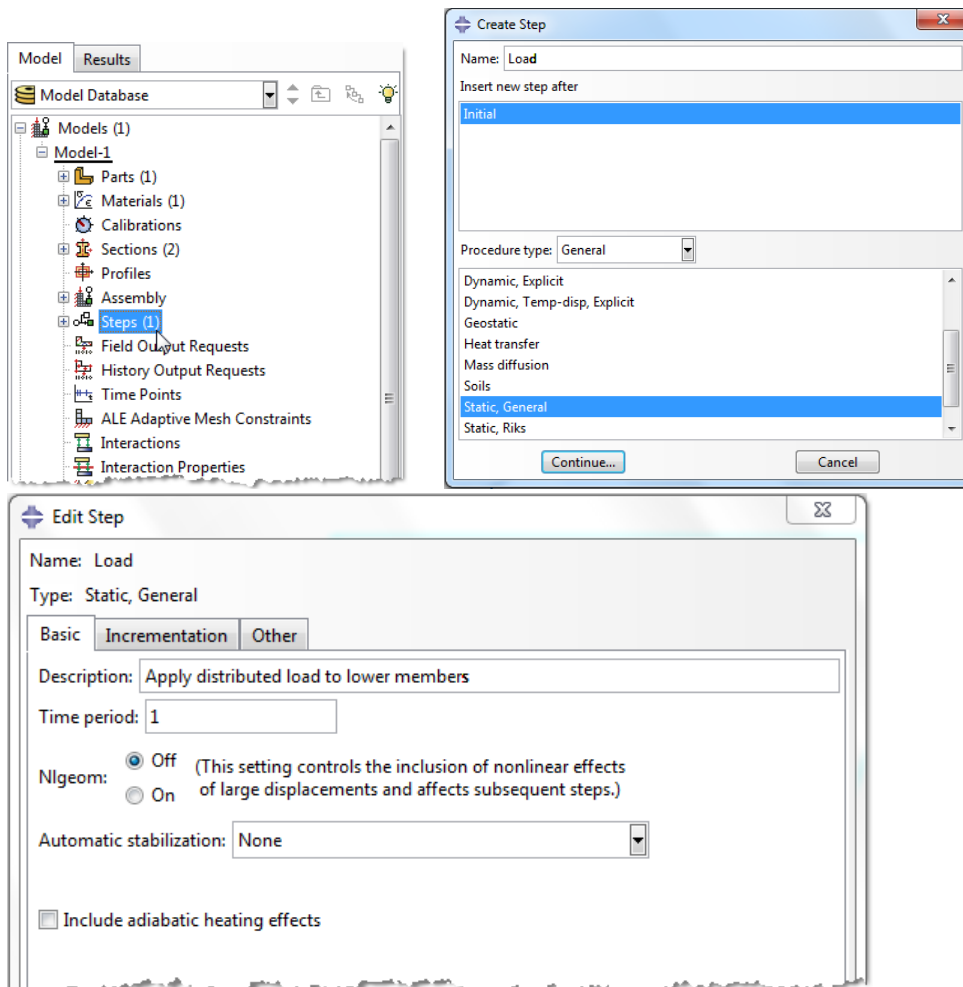
- a. In the toolbox area click on the “Assign Beam/Truss Tangent” icon
- b. Select the sections of the geometry that are off by 180 degrees
- c. Click “Done”



12. Expand the “Assembly” node in the model tree and then double click on “Instances”
  - a. Select “Dependent” for the instance type
  - b. Click “OK”

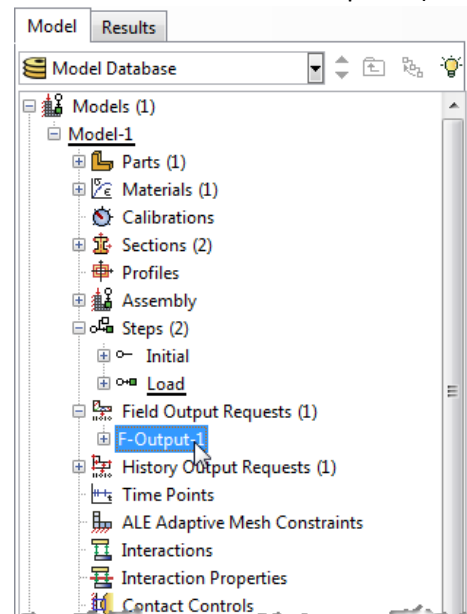


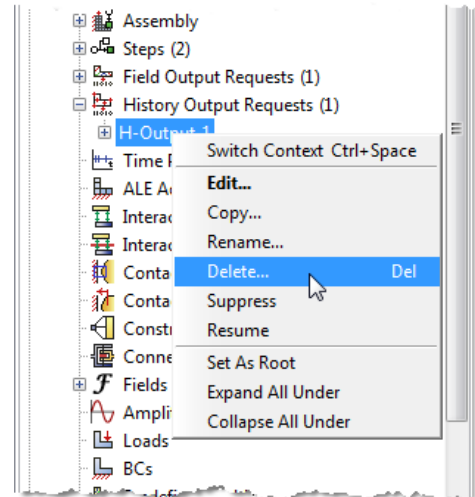
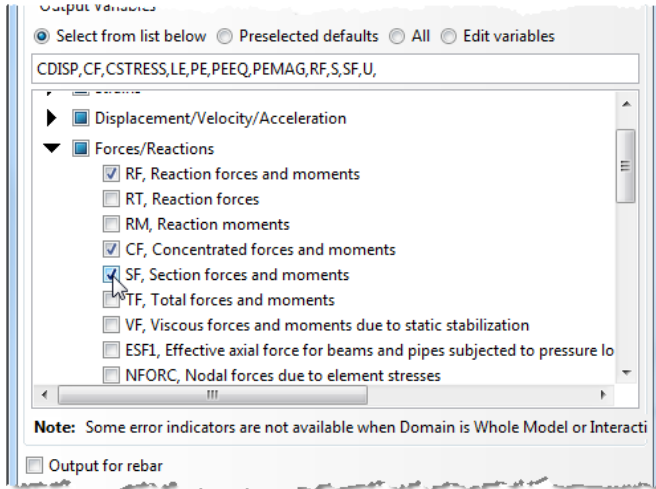
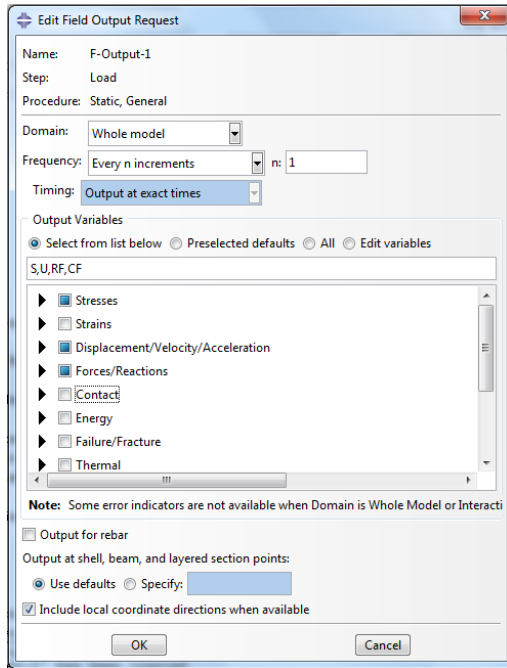
13. Double click on the “Steps” node in the model tree
  - a. Name the step, set the procedure to “General”
  - b. Select “Static, General”
  - c. Click “Continue...”
  - d. Give the step a description
  - e. Click “OK”



14. Expand the Field Output Requests node in the model tree, and then double click on F-Output-1 (F-Output-1 was automatically generated when creating the step)

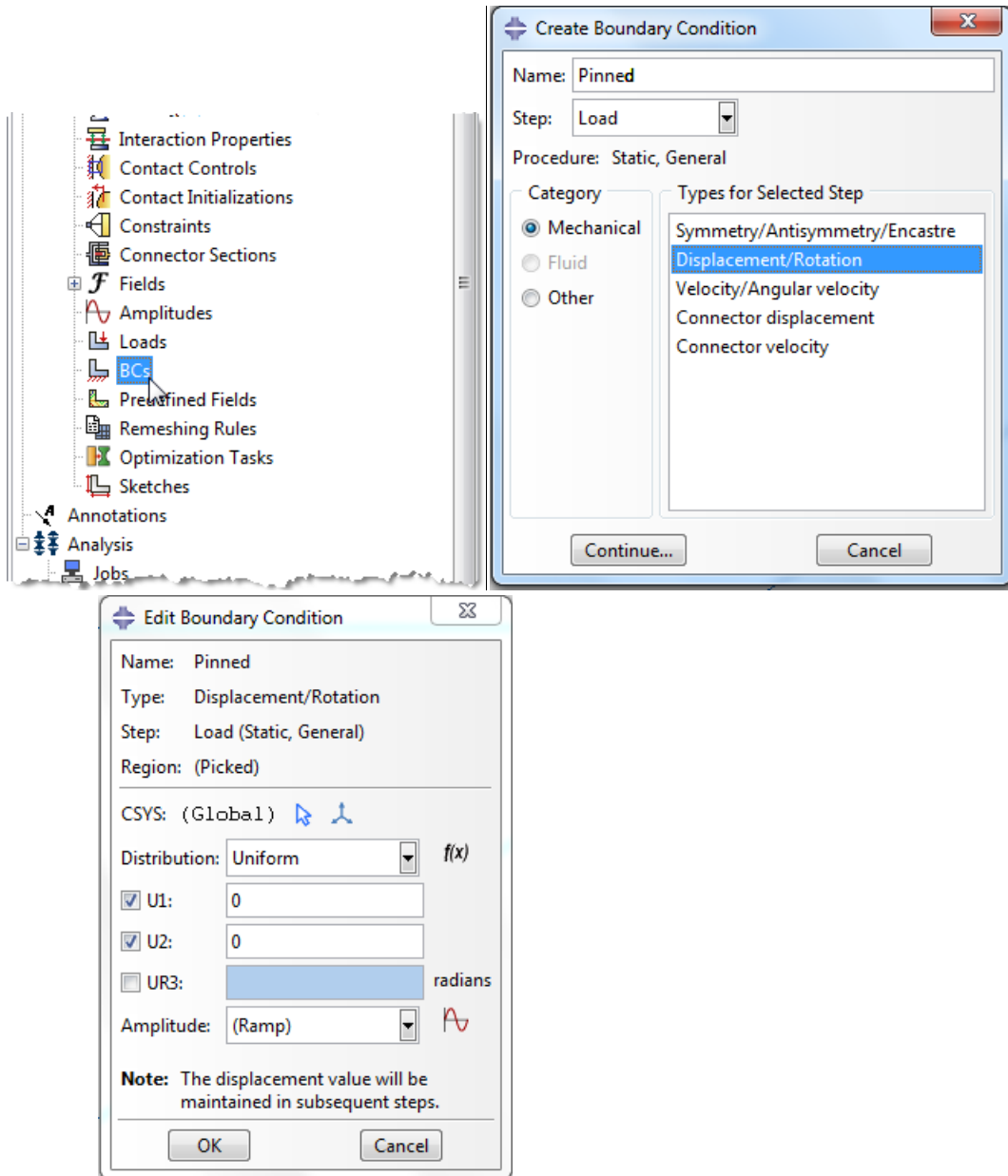
- a. As they are not needed for the current analysis, uncheck the variables “Strains” and “Contact”
- b. To view results for beam stress, and shear and moment diagrams open “Forces/Reactions”
- c. Click “Section Forces and Moments”
- d. Click “OK”



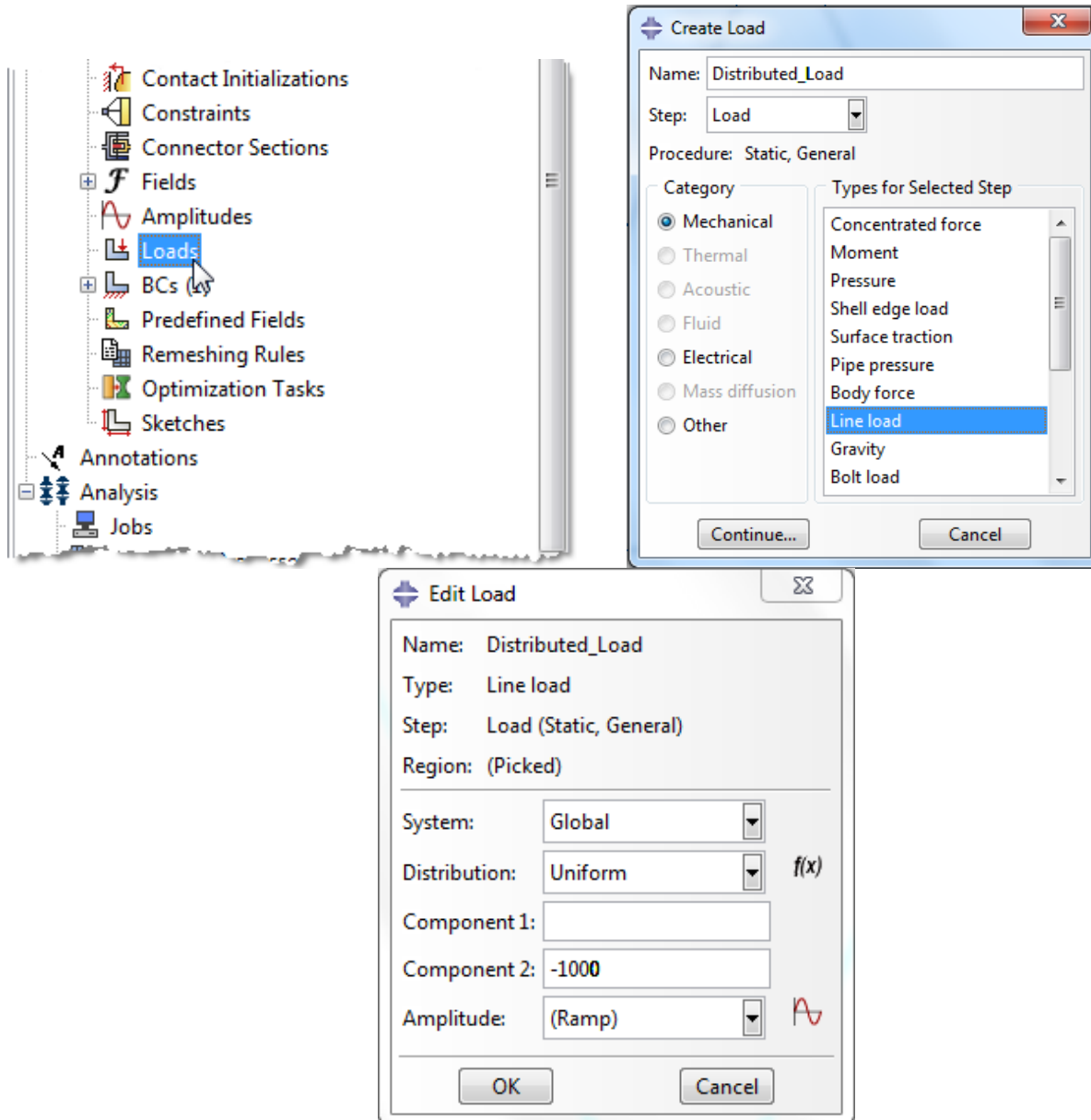


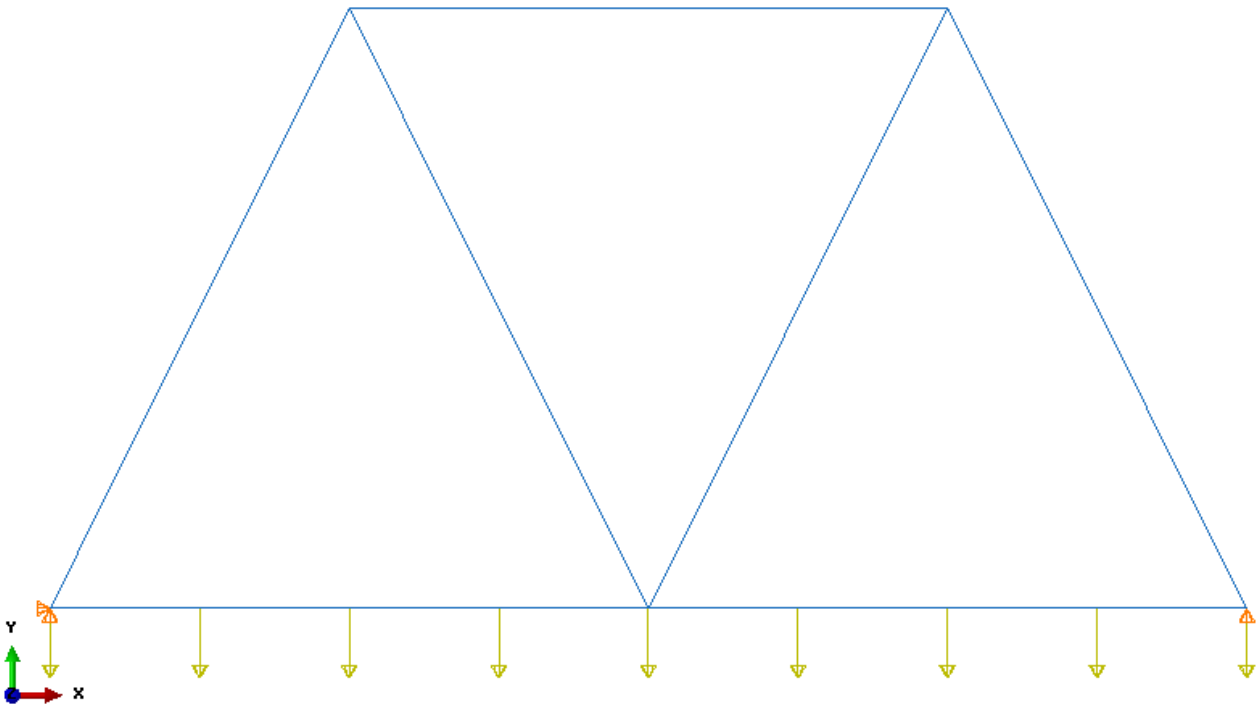
- 15. Expand the History Output Requests node in the model tree
  - a. Right click on H-Output-1 (H-Output-1 was automatically generated when creating the step)
  - b. Select Delete

16. Double click on the “BCs” node in the model tree
  - a. Name the boundary conditioned “Pinned” and select “Displacement/Rotation” for the type
  - b. Click “Continue...”
  - c. Select the lower-left vertex of the geometry and press “Done” in the prompt area
  - d. Check the U1 and U2 displacements and set them to 0
  - e. Click “OK”
  - f. Repeat for the lower-right vertex, but model a roller restraint (only U2 fixed) instead

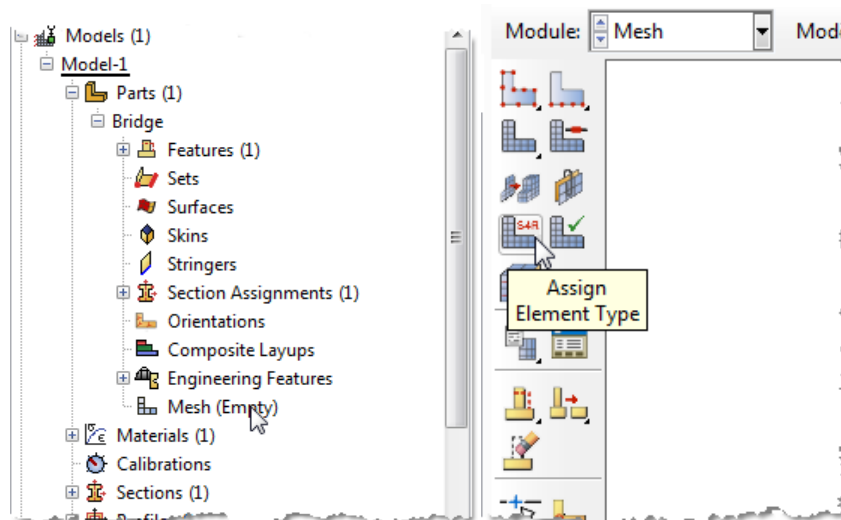


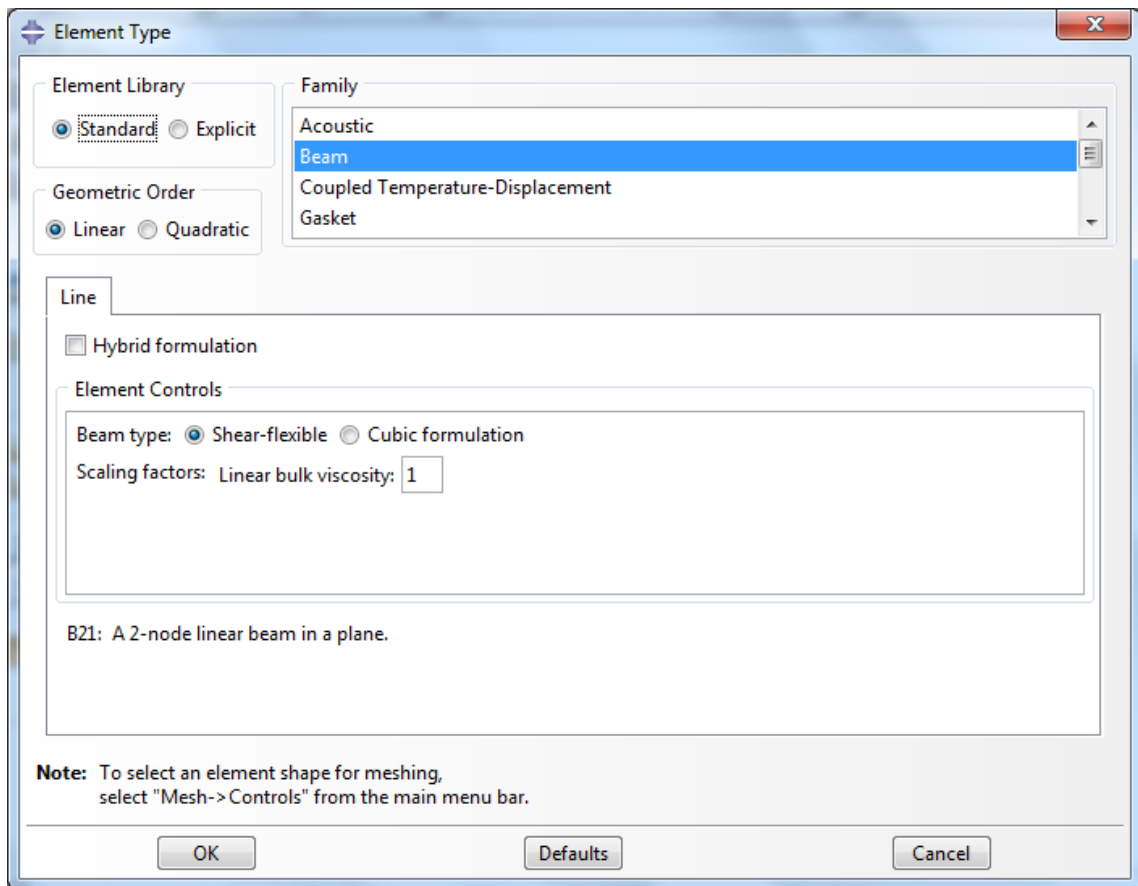
17. Double click on the “Loads” node in the model tree
  - a. Name the load “Distributed load” and select “Line load” as the type
  - b. Click “Continue...”
  - c. Select the lower horizontal edges of the geometry press “Done” in the prompt area
  - d. Specify component 2 = -1000 \*Note that because we have been using standard SI units the load applied is -1000 N/m, which is a total of -10,000 N distributed across the lower horizontal members
  - e. Click “OK”



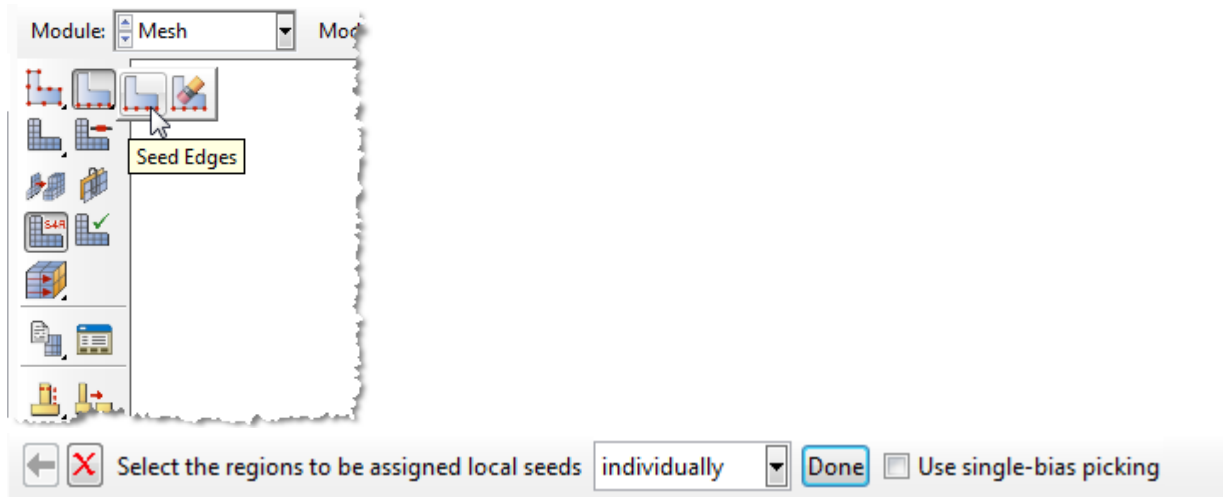


18. In the model tree double click on “Mesh” for the Bridge part
- In the toolbox area click on the “Assign Element Type” icon
  - Highlight all members in the viewport and select Done
  - Select “Standard” for element type
  - Select “Linear” for geometric order
  - Select “Beam” for family
  - Note that the name of the element (B21) and its description are given below the element controls
  - Click “OK”



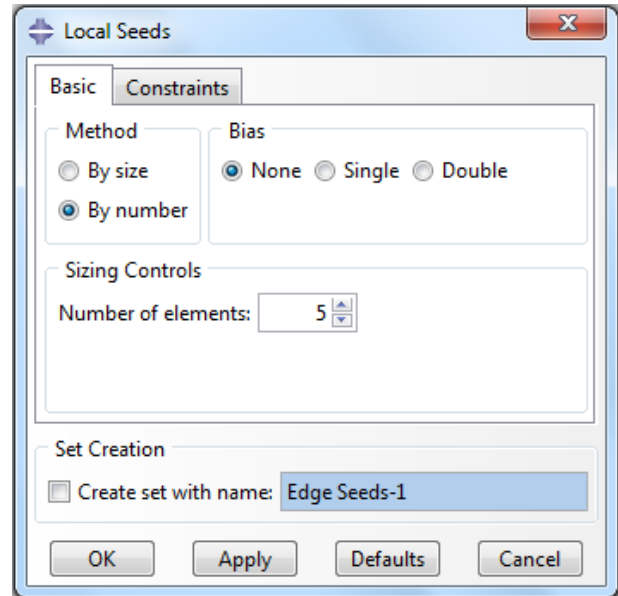


19. In the toolbox area click on the “Seed Edges” icon
  - a. Select the entire geometry, except the lower horizontal lines
  - b. Click “Done” in the prompt area

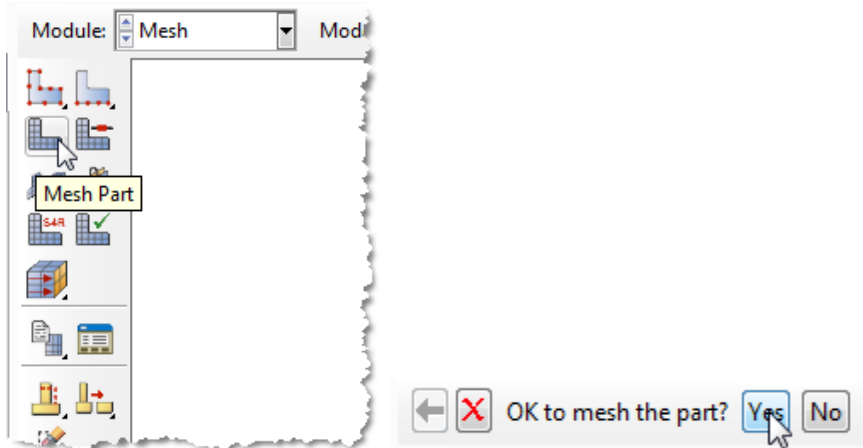




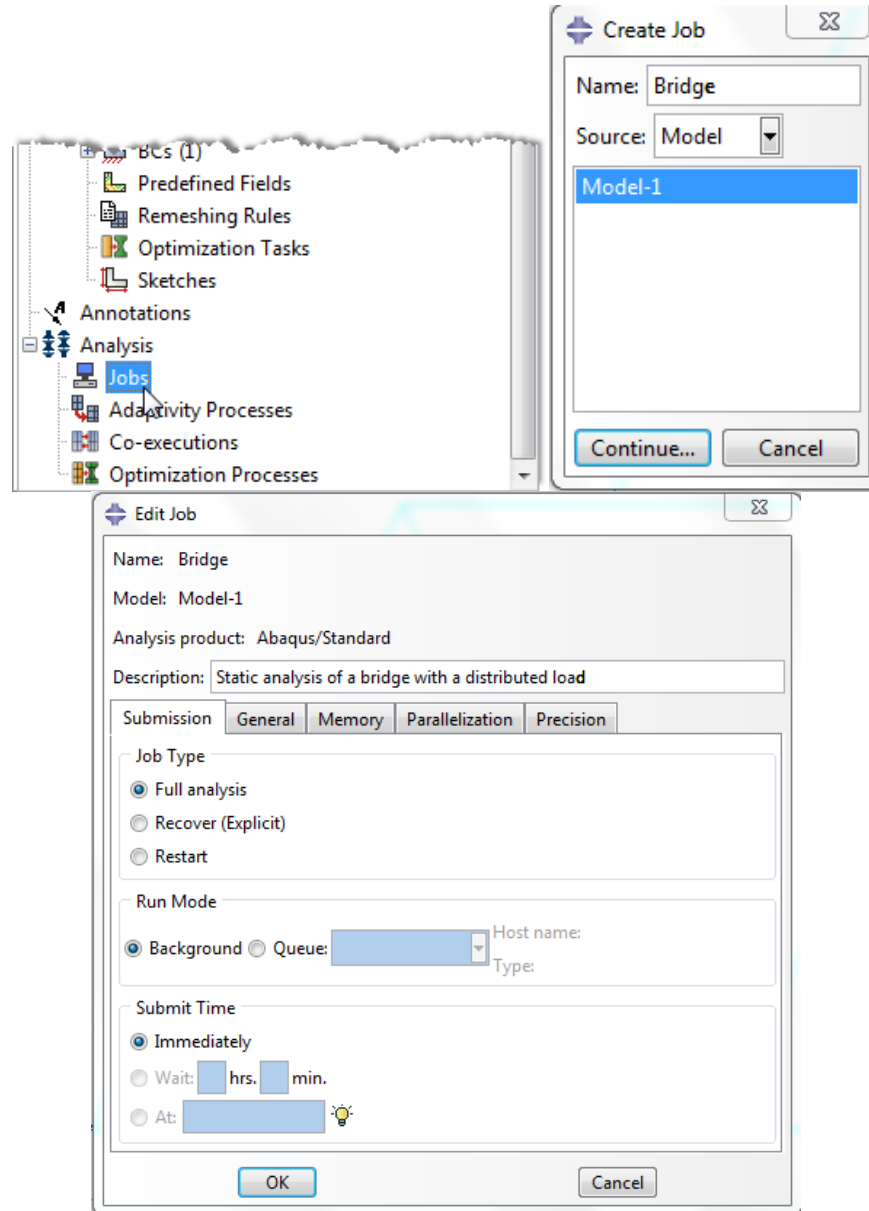
- c. Choose "By Number" Method and set the number of elements along the edges as 5 (under Sizing Controls)
- d. Repeat for the lower horizontal lines, except specify 10 elements along the edges



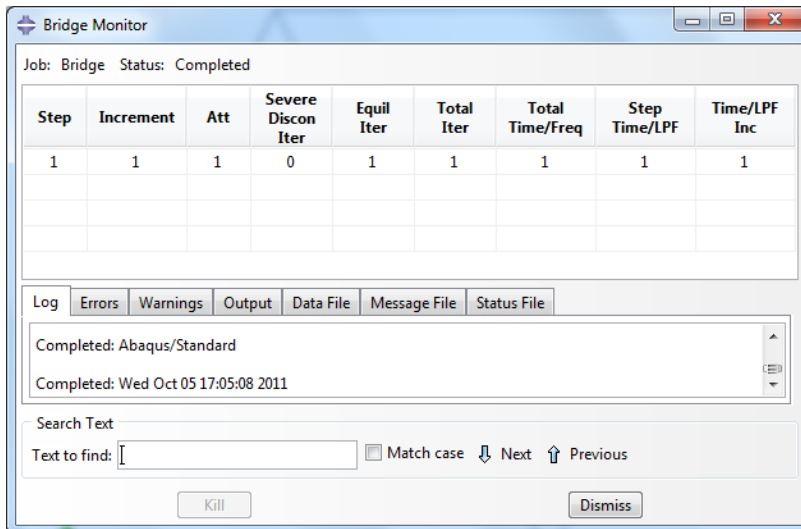
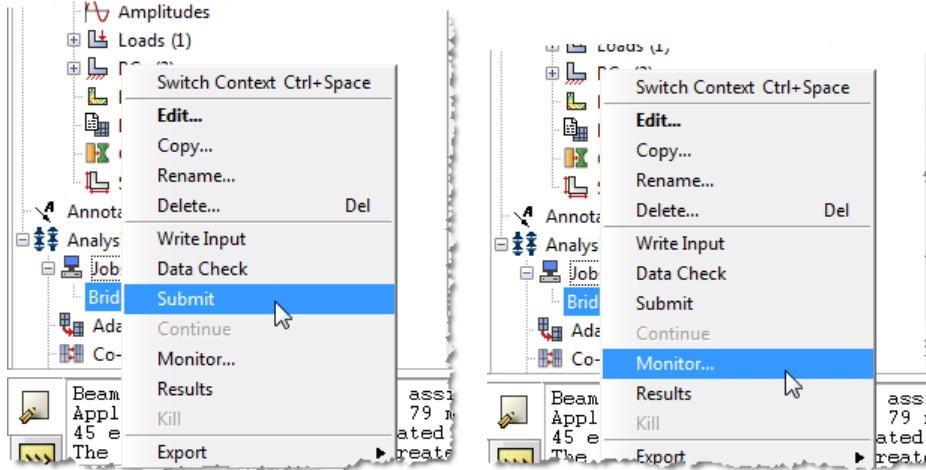
- 20. In the toolbox area click on the "Mesh Part" icon  
Click "Yes" in the prompt area



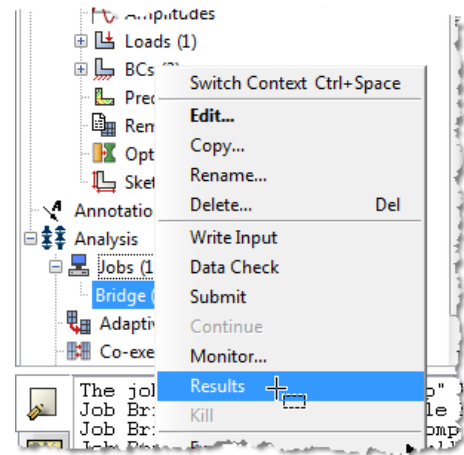
21. In the model tree double click on the “Job” node
- Name the job “Bridge”
  - Click “Continue...”
  - Give the job a description
  - Click “OK”



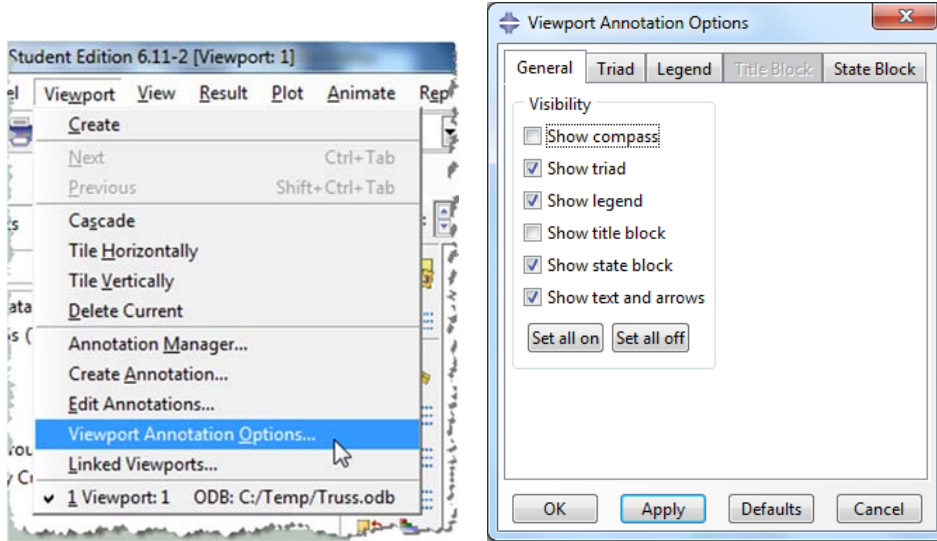
22. In the model tree right click on the job just created (Bridge) and select “Submit”
- While Abaqus is solving the problem right click on the job submitted (Bridge), and select “Monitor”
- In the Monitor window check that there are no errors or warnings
  - If there are errors, investigate the cause(s) before resolving
  - If there are warnings, determine if the warnings are relevant, some warnings can be safely ignored



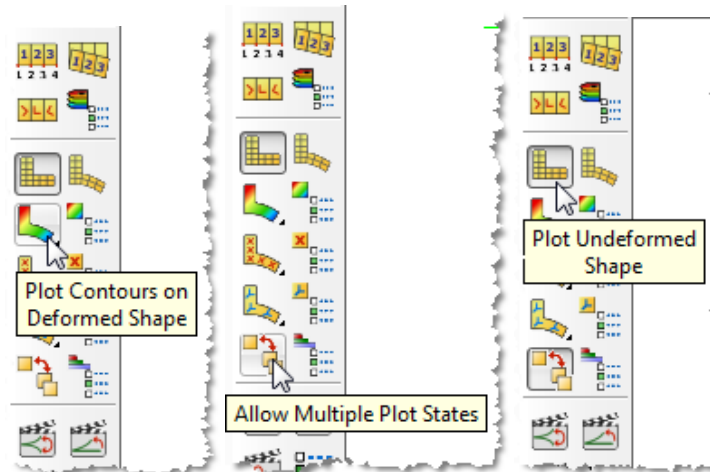
23. In the model tree right click on the submitted and successfully completed job (Bridge), and select “Results”



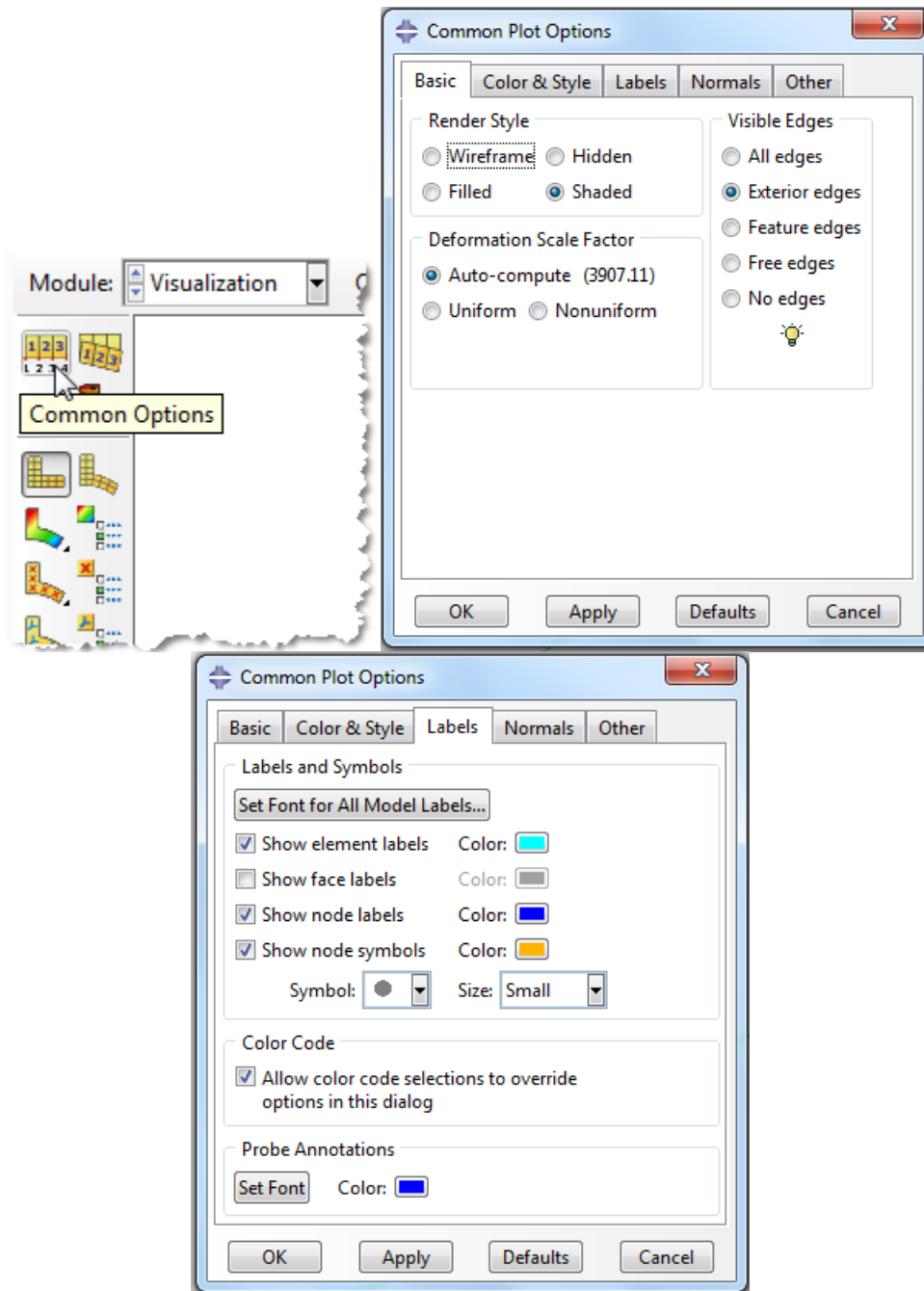
24. In the menu bar click on Viewport → Viewport Annotations Options
  - a. Uncheck the “Show compass option”
  - b. The locations of viewport items can be specified on the corresponding tab in the Viewport Annotations Options
  - c. Click “OK”

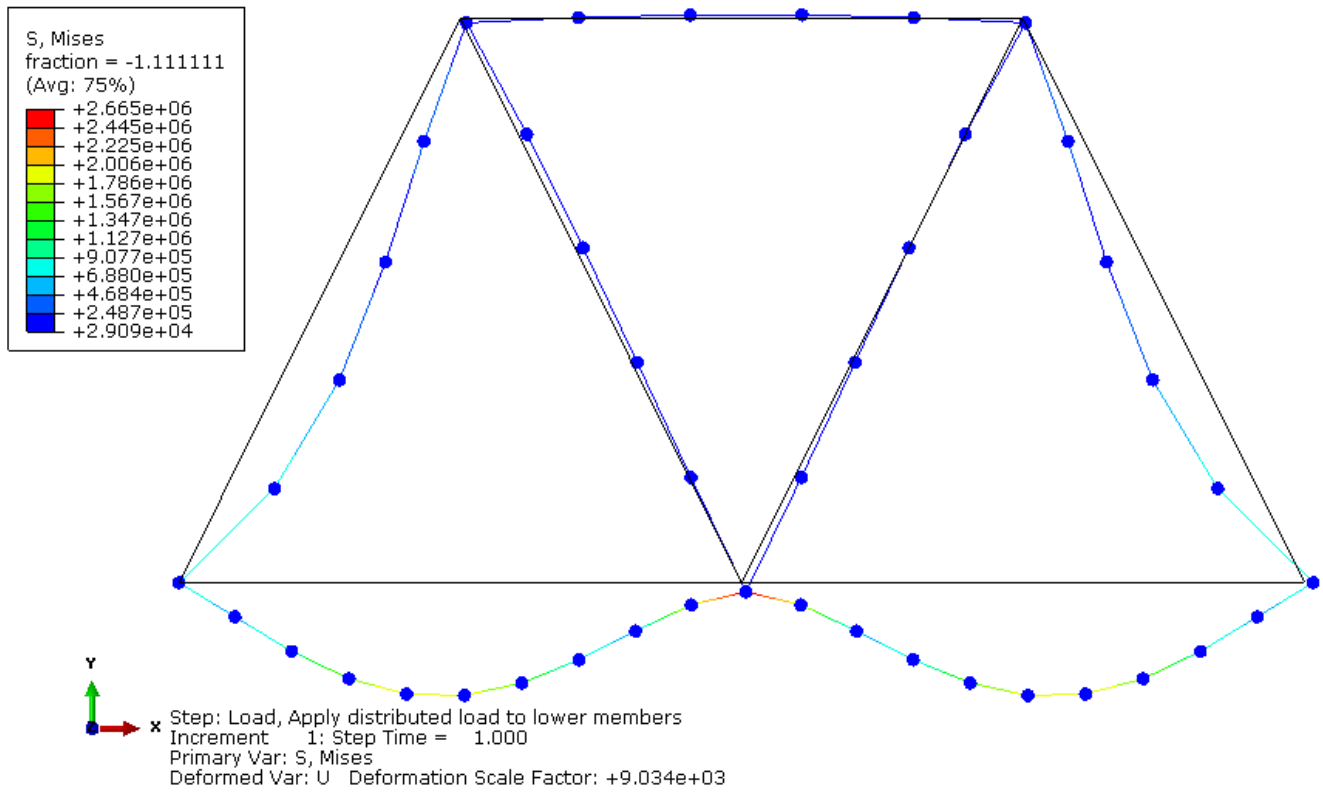


25. Display the deformed contour of the (Von) Mises stress overlaid with the undeformed geometry
  - a. Click on the icon for “Plot Contours on Deformed Shape”
  - b. Click on the icon for “Allow Multiple Plot States”
  - c. Click on the icon for “Plot Undeformed Shape”



26. In the toolbox area click on the “Common Plot Options” icon
  - a. Note that the Deformation Scale Factor can be set on the “Basic” tab
  - b. On the “Labels” tab check the show node symbols icon
  - c. Click “OK”





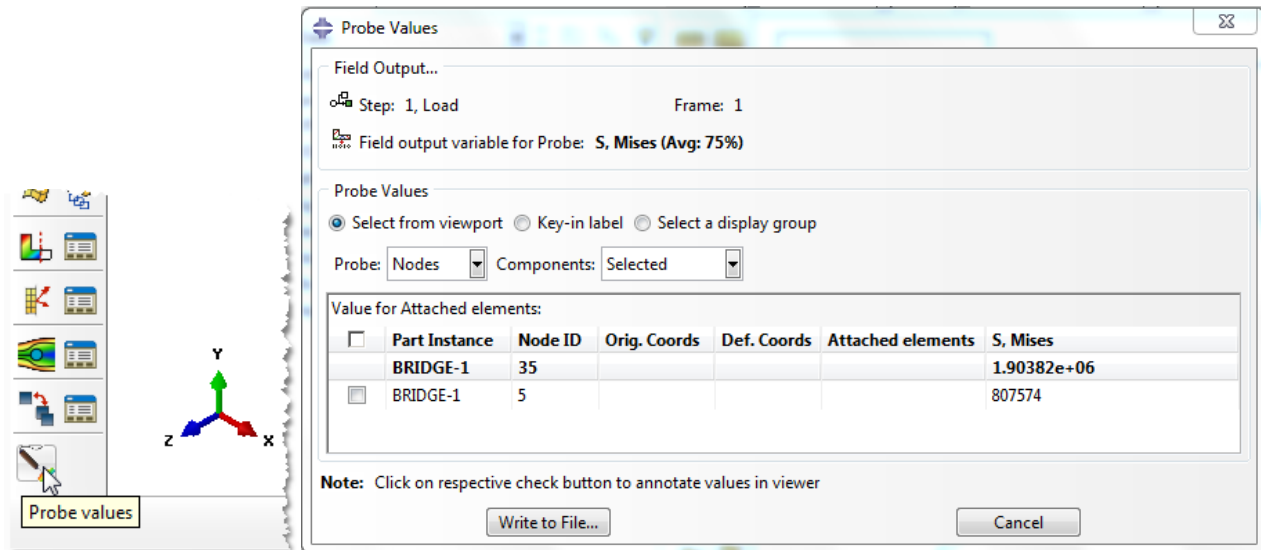
27. To determine the stress values, Click the “Probe Values” icon

- a. Check the boxes labeled “Nodes” and “S, Mises”
- b. In the viewport mouse over the element of interest

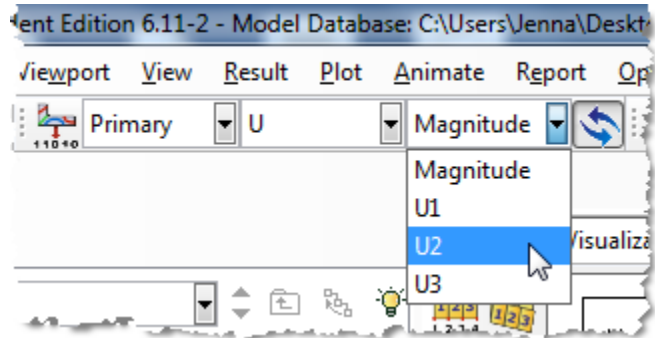
Note that Abaqus reports stress values from the integration points, which may differ slightly from the values determined by projecting values from the surrounding integration points to the nodes

The minimum and maximum stress values contained in the legend are from the stresses projected to the nodes

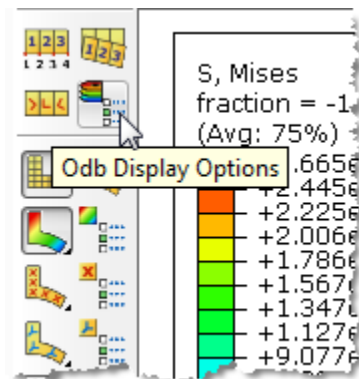
- c. Click on an element to store it in the “Selected Probe Values” portion of the dialogue box
- d. Click “Cancel”



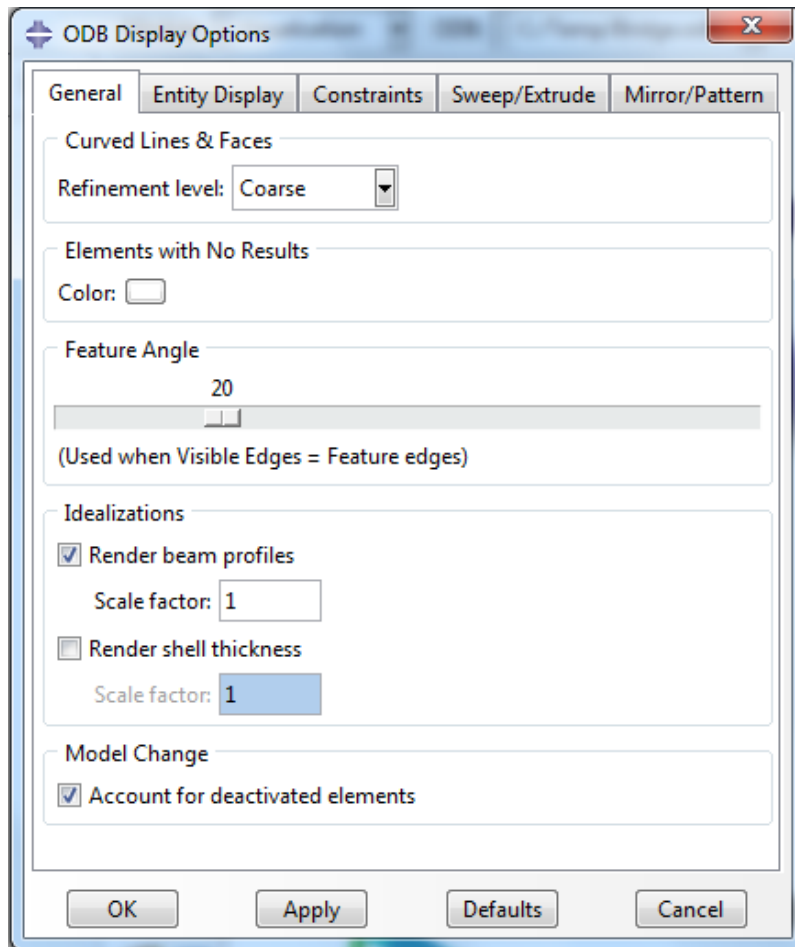
28. Change the output being displayed
  - a. Change the display option in the tool bar to "U"
  - b. Select component "U2"
  - c. Again nodal displacements can be found using the "Probe Values" tool



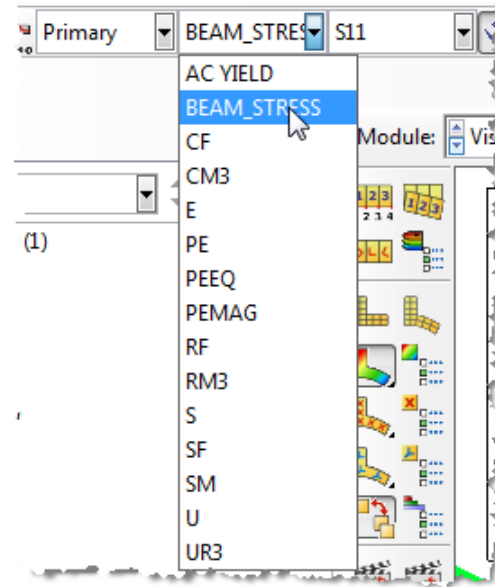
29. To investigate stresses through the beam section
  - a. Click on the "ODB display options" icon
  - b. Select "Display Beam Profiles"
  - c. Click "OK"



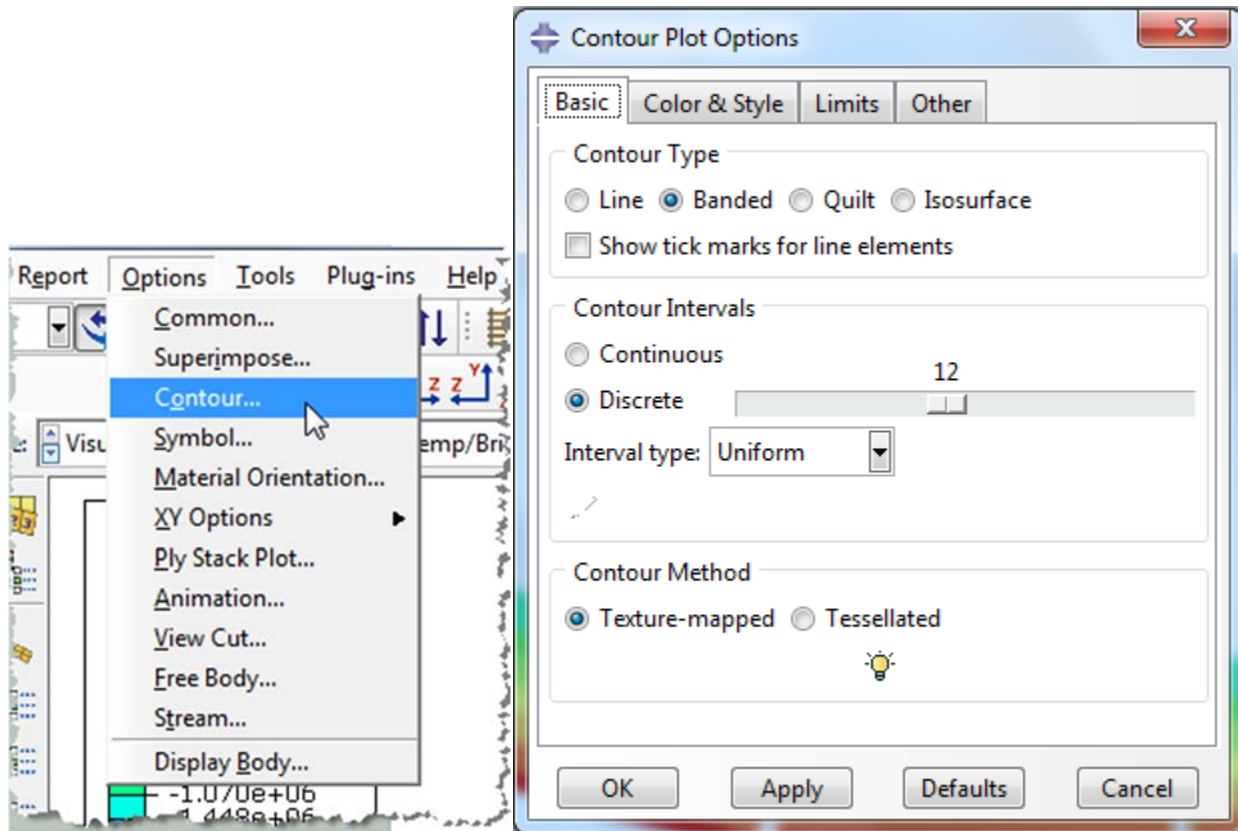




30. Change the output displayed to “Beam Stress”
  - a. Select the appropriate component, for example S11
  - b. The display will show the stress distribution across the beam section. \*Note: these values cannot be queried, but the resolution of the color display can be increased to more clearly highlight the location of the neutral axis.

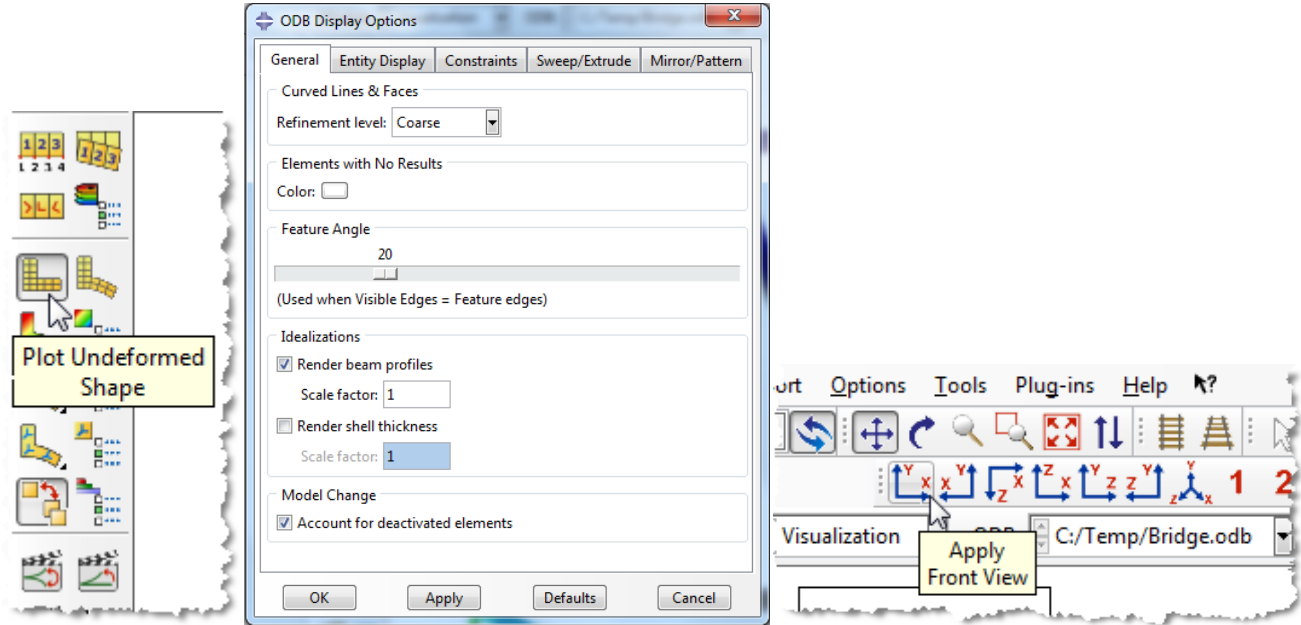


31. To adjust the display plot contours
  - a. In the menu bar, select Options -> Contour
  - b. Adjust the resolution
  - c. Click “OK”

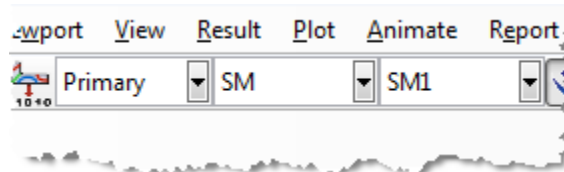


32. To see beam shear and moment diagrams

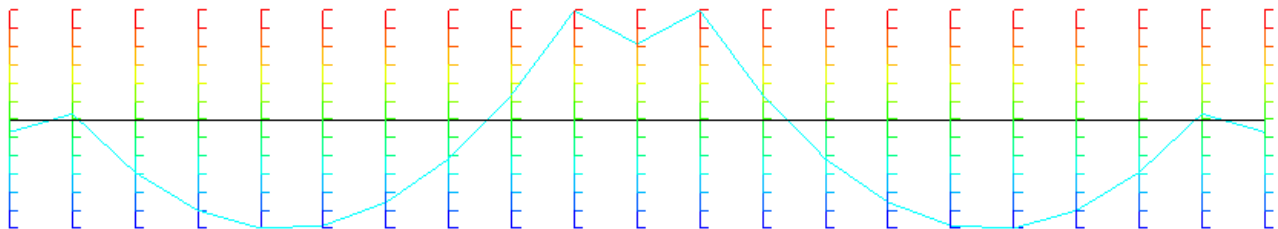
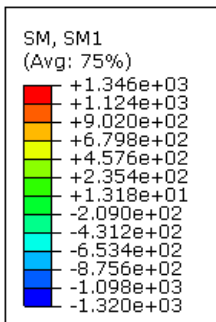
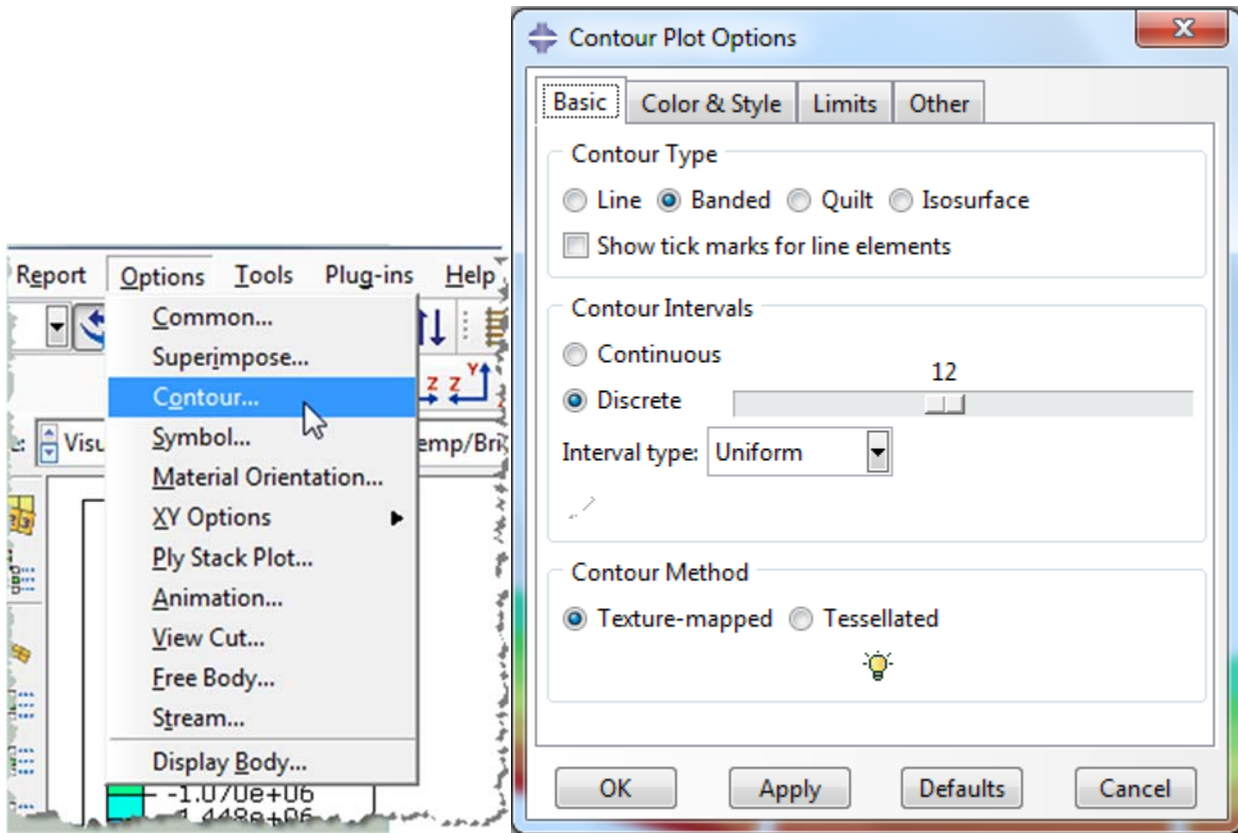
- a. Change to plot “Plot Contours on Undeformed Shape”
- b. Turn off “Render Beam Profiles” under ODB display options
- c. Adjust the view to be square to the part plane using the “Apply Front View” icon




- d. Choose the appropriate display output
  - i. For moment values choose “SM, component SM1”
  - ii. For shear force values choose “SF, component SF1”



- e. As the default display is not very useful for line type element, change to a “Bending-Moment” type plot under contour options
  - i. Select Options -> Contour
  - ii. Click “Show tick marks for line elements”
  - iii. \*Note- the display group has been reduced for clarity




 Step: Load, Apply distributed load to lower members  
 Increment 1: Step Time = 1.000  
 Primary Var: SM, SM1

**Beam stress:**

Select the "Beam stress" from the main toolbar.

If interested in the bending stress, select S11 as the stress component.

The example below shows the bending stress distribution across a wide-flange beam.

