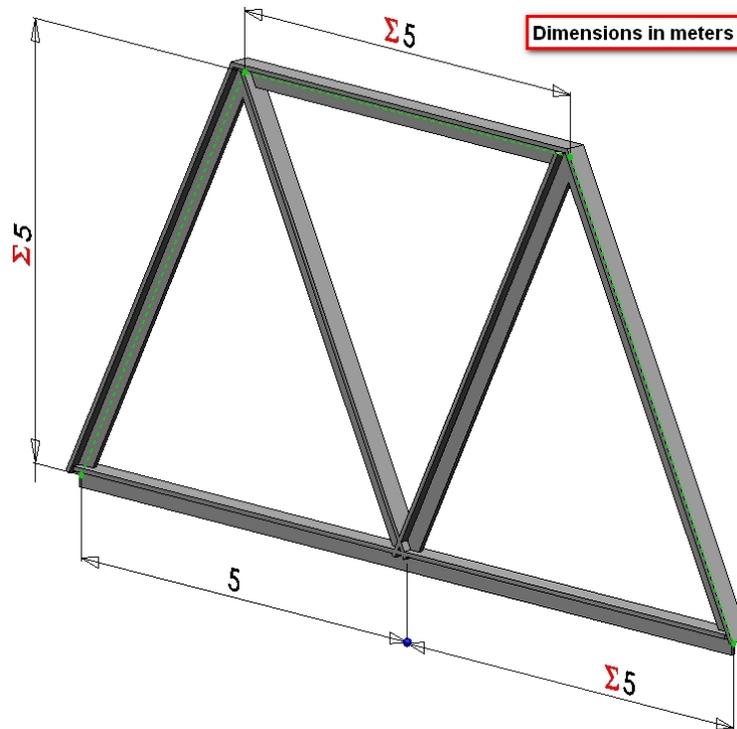


Abaqus Beam Tutorial

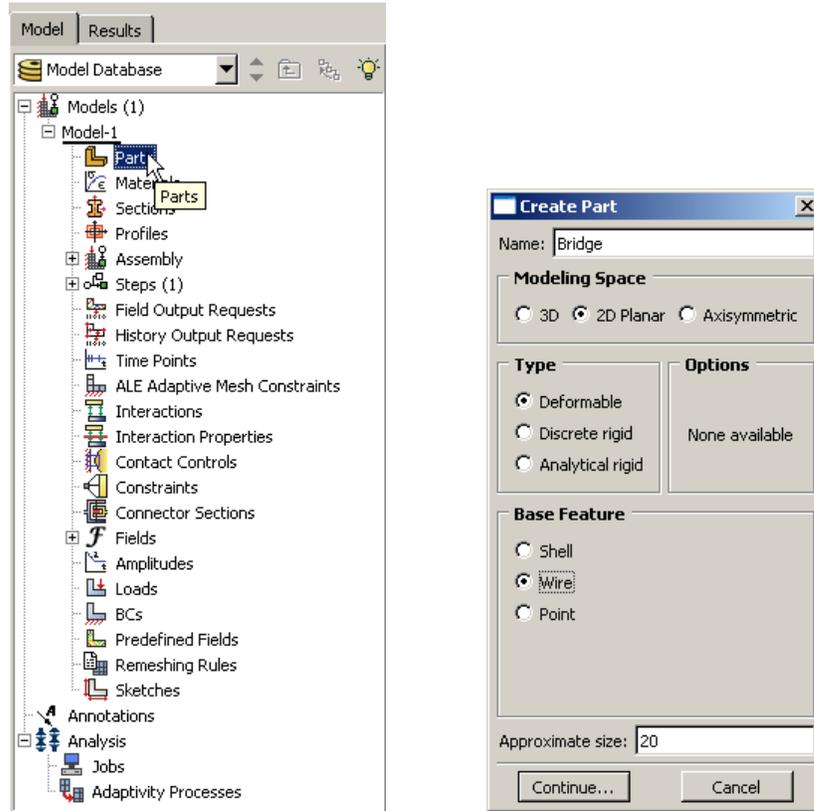
Problem Description



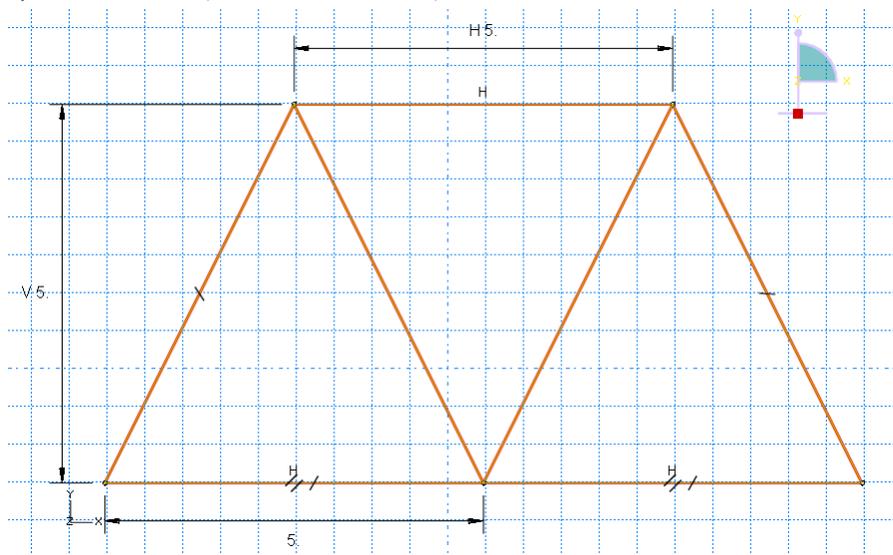
The two dimensional bridge structure, which consists of steel T-sections, is simply supported at its lower corners. A uniform distributed load of 1000 N/m is applied to the lower horizontal members in the vertical downward direction. Determine the stresses and the vertical displacements.

Analysis Steps

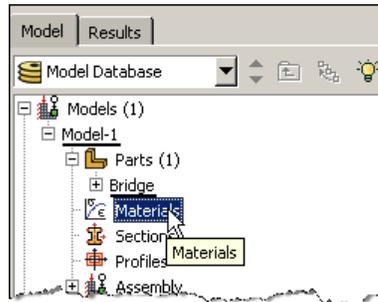
1. Start Abaqus and choose to create a new model database
2. In the model tree double click on the “Parts” node (or right click on “parts” and select Create)



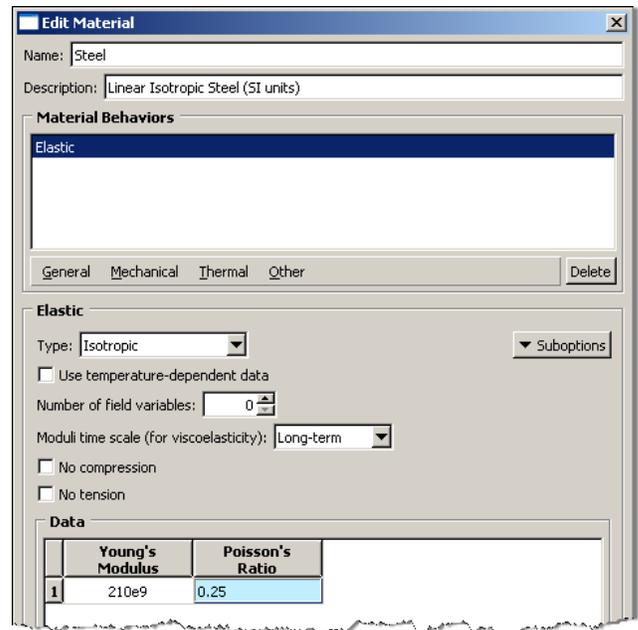
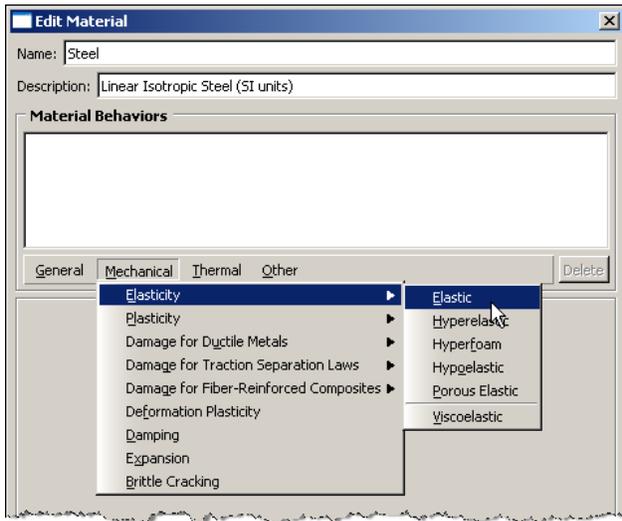
3. In the Create Part dialog box (shown above) name the part and
 - a. Select “2D Planar”
 - b. Select “Deformable”
 - c. Select “Wire”
 - d. Set approximate size = 20
 - e. Click “Continue...”
4. Create the geometry shown below (not discussed here)



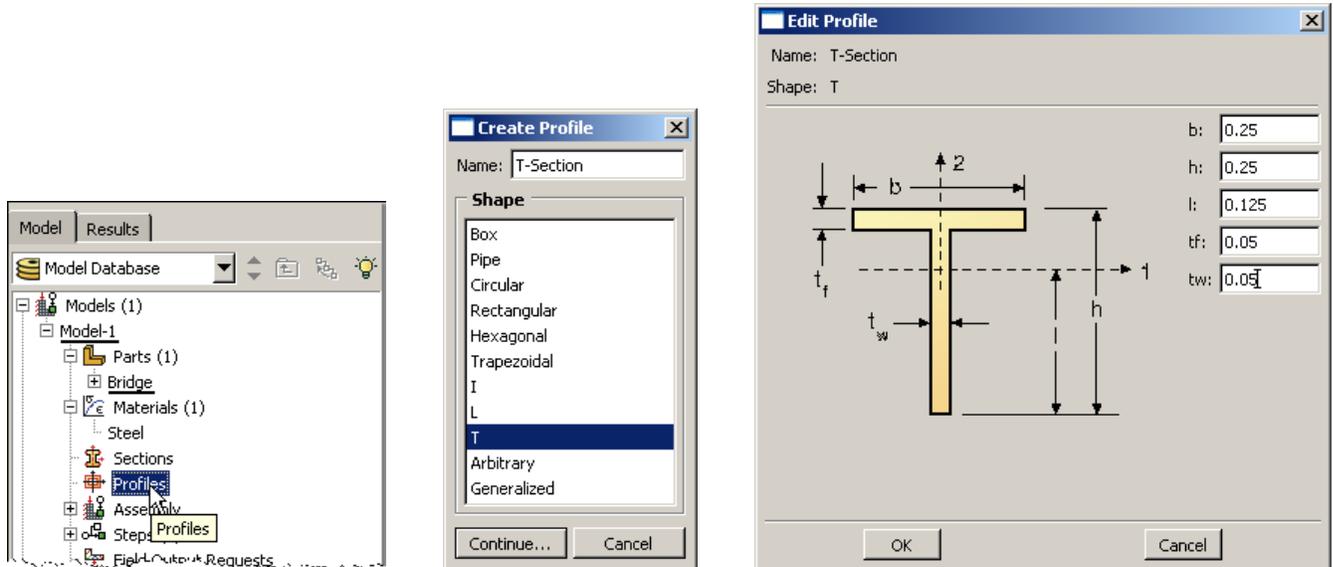
5. Double click on the “Materials” node in the model tree



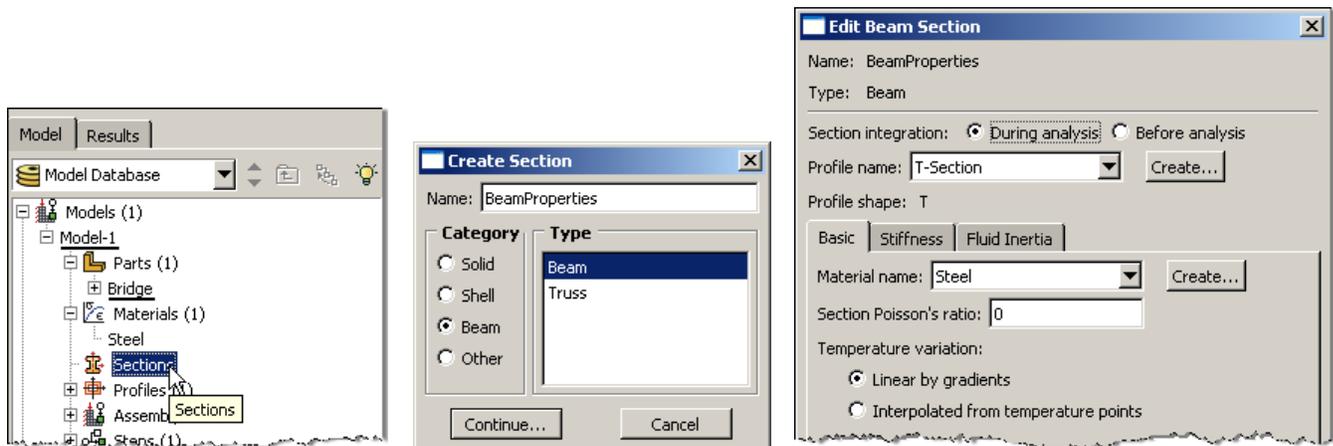
- a. Name the new material and give it a description
- b. Click on the “Mechanical” tab → Elasticity → Elastic
- c. Define Young’s Modulus and Poisson’s Ratio (use SI units)
 - i. **WARNING: There are no predefined system of units within Abaqus, so the user is responsible for ensuring that the correct values are specified**
- d. Click “OK”



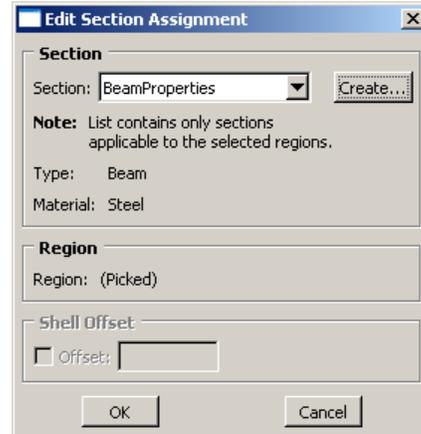
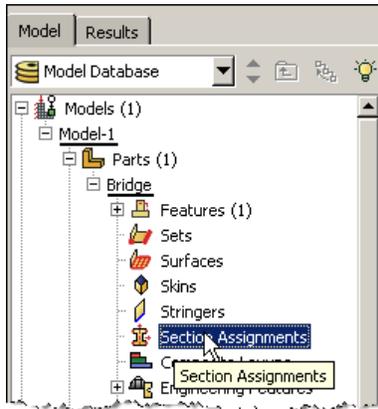
6. Double click on the "Profiles" node in the model tree
 - a. Name the profile and select "T" for the shape
 - i. Note that the "T" shape is one of several predefined cross-sections
 - b. Click "Continue..."
 - c. Enter the values for the profile shown below
 - d. Click "OK"



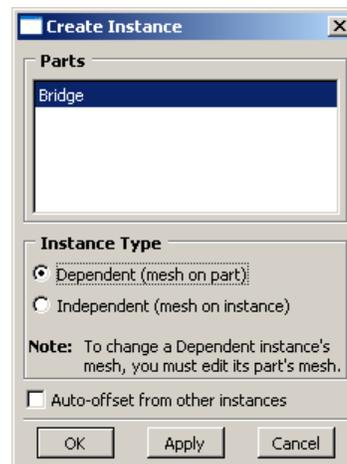
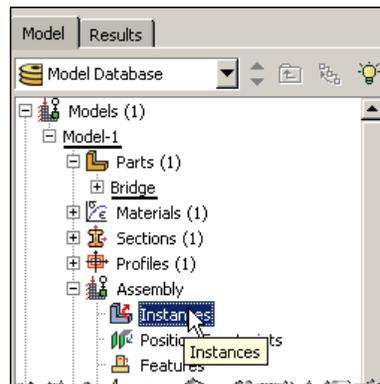
7. Double click on the "Sections" node in the model tree
 - a. Name the section "BeamProperties" and select "Beam" for both the category and the type
 - b. Click "Continue..."
 - c. Leave the section integration set to "During Analysis"
 - d. Select the profile created above (T-Section)
 - e. Select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



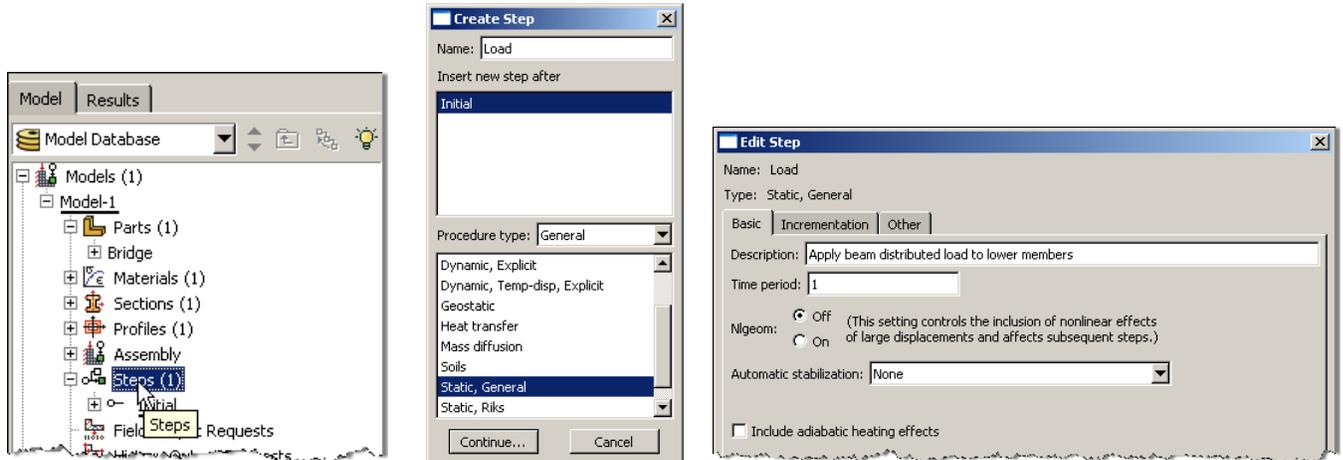
8. Expand the “Parts” node in the model tree, expand the node of the part just created, and double click on “Section Assignments”
 - a. Select the entire geometry in the viewport and press Enter
 - b. Select the section created above (BeamProperties)
 - c. Click “OK”



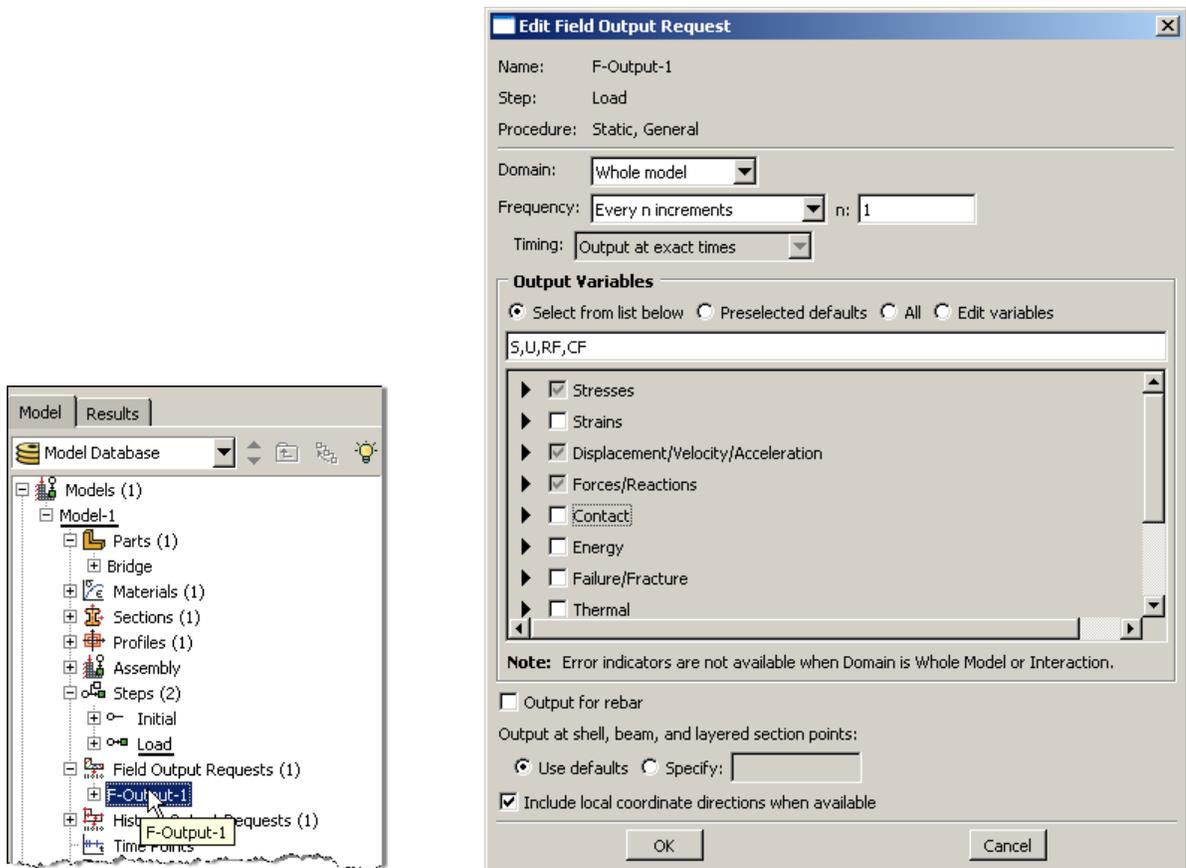
9. Expand the “Assembly” node in the model tree and then double click on “Instances”
 - a. Select “Dependent” for the instance type
 - b. Click “OK”



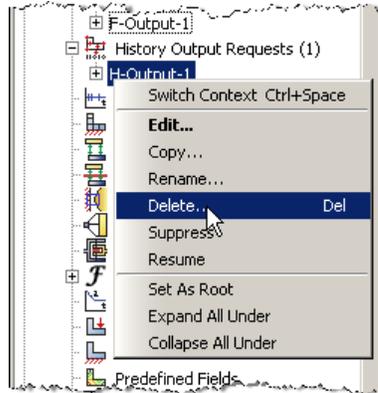
10. Double click on the “Steps” node in the model tree
 - a. Name the step, set the procedure to “General”, and select “Static, General”
 - b. Click “Continue...”
 - c. Give the step a description
 - d. Click “OK”



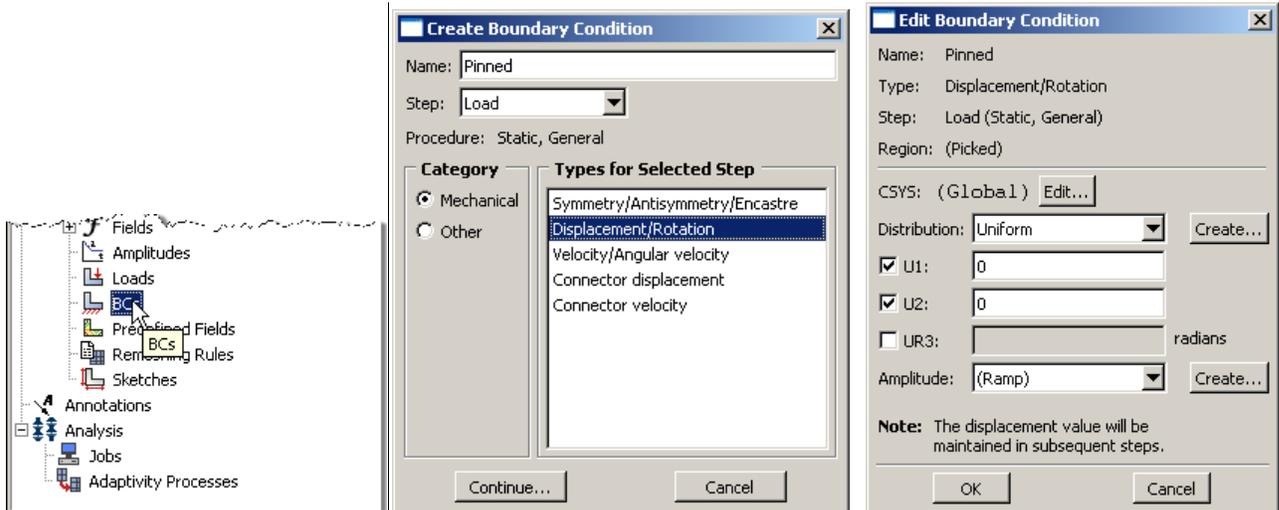
11. Expand the Field Output Requests node in the model tree, and then double click on F-Output-1 (F-Output-1 was automatically generated when creating the step)
 - a. Uncheck the variables “Strains” and “Contact”
 - b. Click “OK”



12. Expand the History Output Requests node in the model tree, and then right click on H-Output-1 (H-Output-1 was automatically generated when creating the step) and select Delete



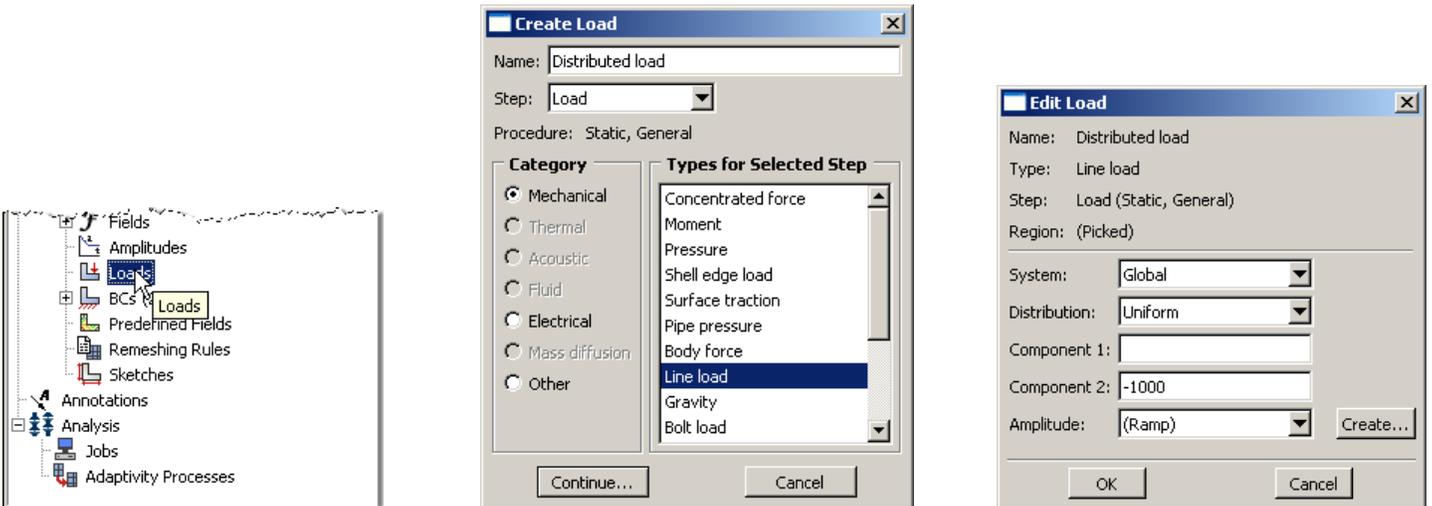
13. Double click on the “BCs” node in the model tree
- Name the boundary conditioned “Pinned” and select “Displacement/Rotation” for the type
 - Click “Continue...”
 - Select the lower-left vertex of the geometry and press “Done” in the prompt area
 - Check the U1 and U2 displacements and set them to 0
 - Click “OK”



- f. Repeat for the lower-right vertex, but model a roller restraint (only U2 fixed) instead

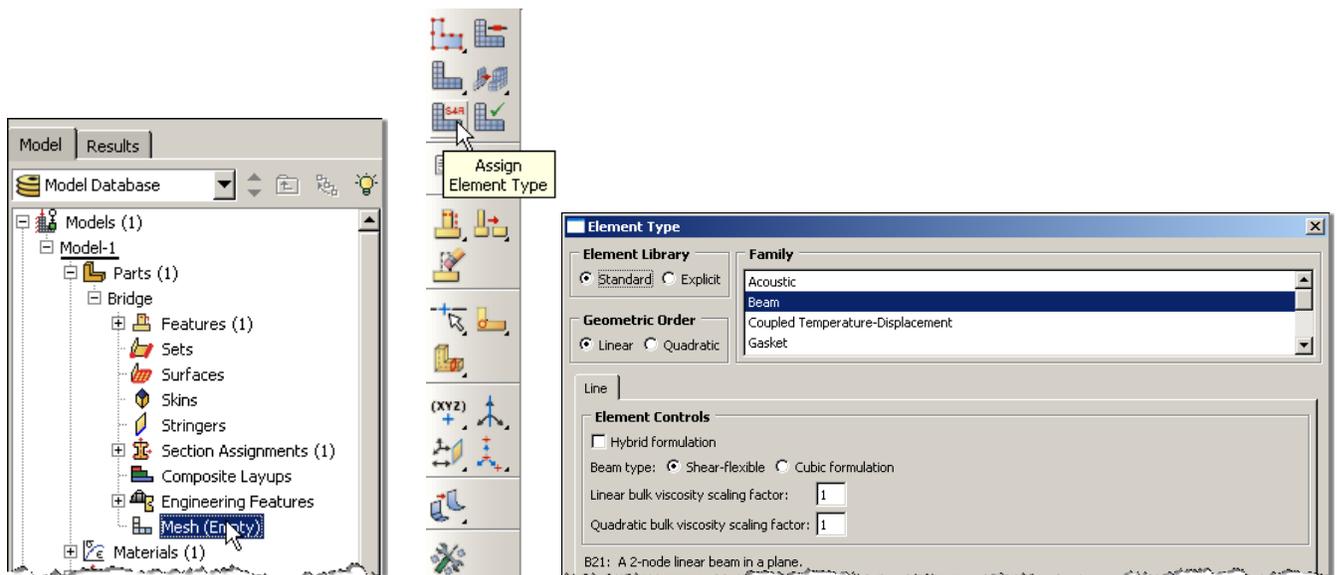
14. Double click on the “Loads” node in the model tree

- 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
 - i. 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”

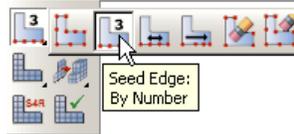


15. In the model tree double click on “Mesh” for the Bridge part, and in the toolbox area click on the “Assign Element Type” icon

- a. Highlight all members in the viewport and select Done
- b. Select “Standard” for element type
- c. Select “Linear” for geometric order
- d. Select “Beam” for family
- e. Note that the name of the element (B21) and its description are given below the element controls
- f. Click “OK”



16. In the toolbox area click on the “Seed Edge: By Number” icon (hold down icon to bring up the other options)



a. Select the entire geometry, except the lower horizontal lines, and click “Done” in the prompt area



b. Define the number of elements along the edges as 5

c. Repeat for the lower horizontal lines, except specify 10 elements along the edges

17. In the toolbox area click on the “Mesh Part” icon

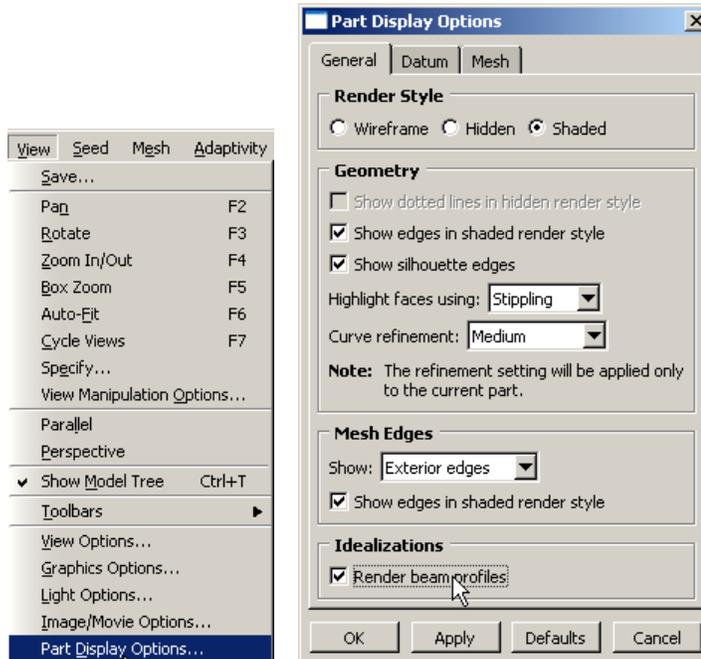
a. Click “Yes” in the prompt area



18. In the menu bar select View → Part Display Options

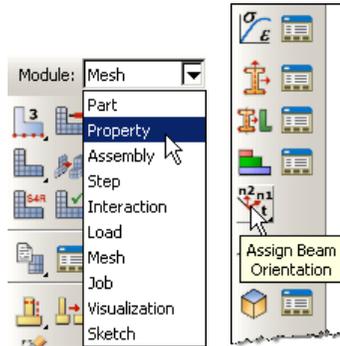
a. Check the Render beam profiles option on the General tab

b. Click “OK”

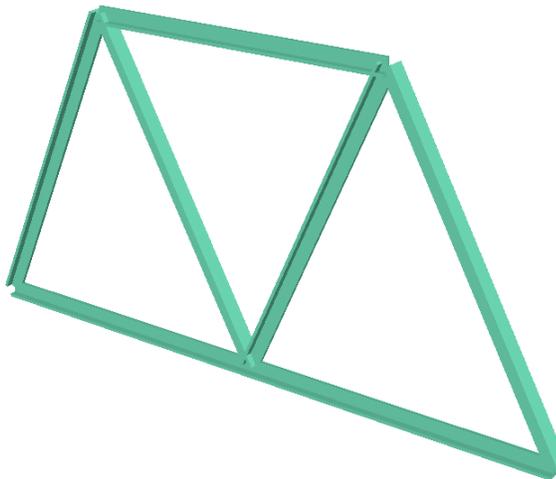


19. Change the Module to "Property"

- a. Click on the "Assign Beam Orientation" icon
- b. Select the entire geometry from the viewport
- c. Click "Done" in the prompt area
- d. Accept the default value of the approximate n1 direction

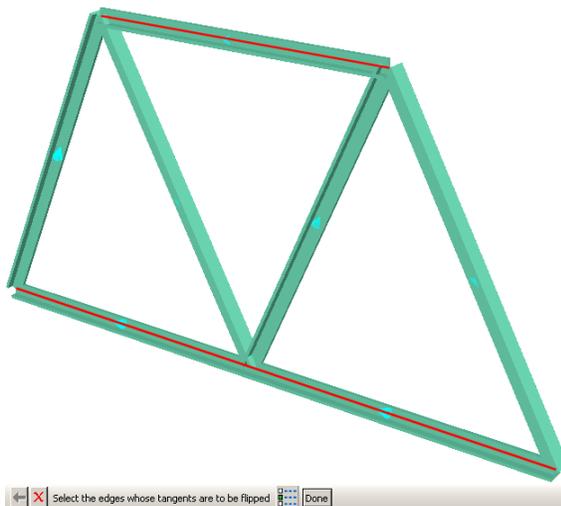
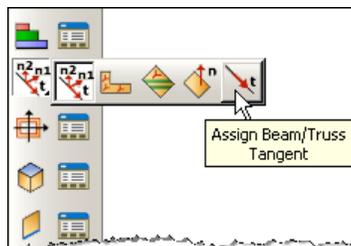


20. Note that the preview shows that the beam cross sections are not all orientated as desired (see Problem Description)



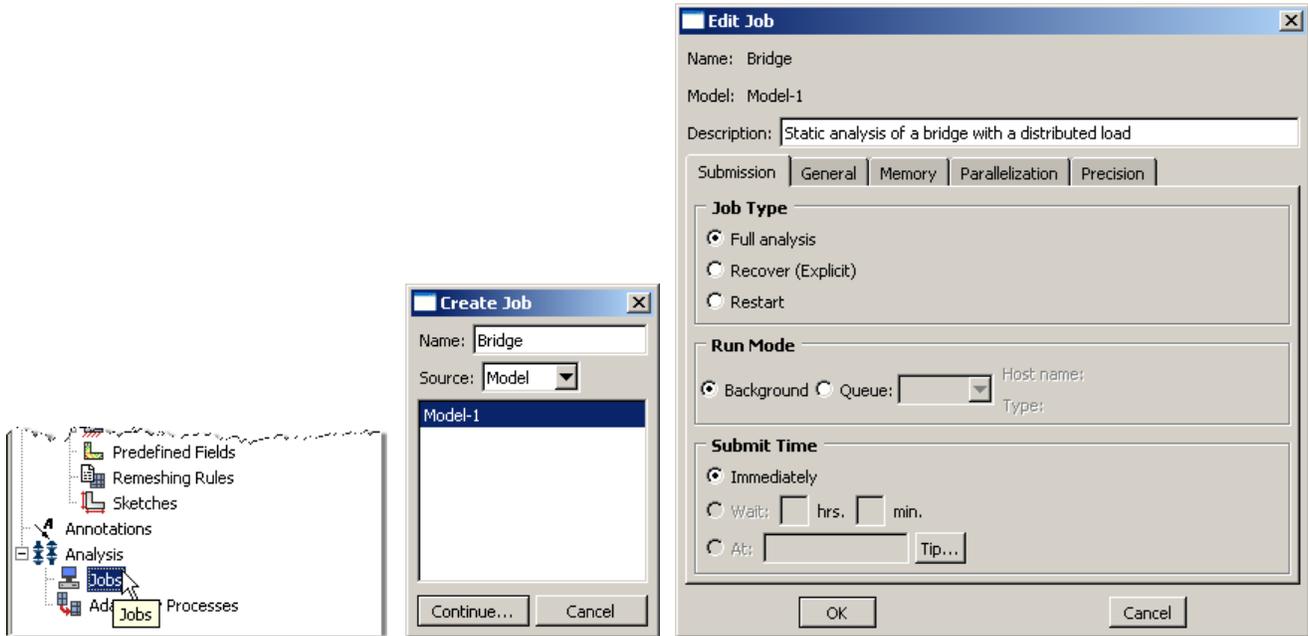
21. In the toolbox area click on the "Assign Beam/Truss Tangent" icon

- a. Click on the sections of the geometry that are off by 180 degrees



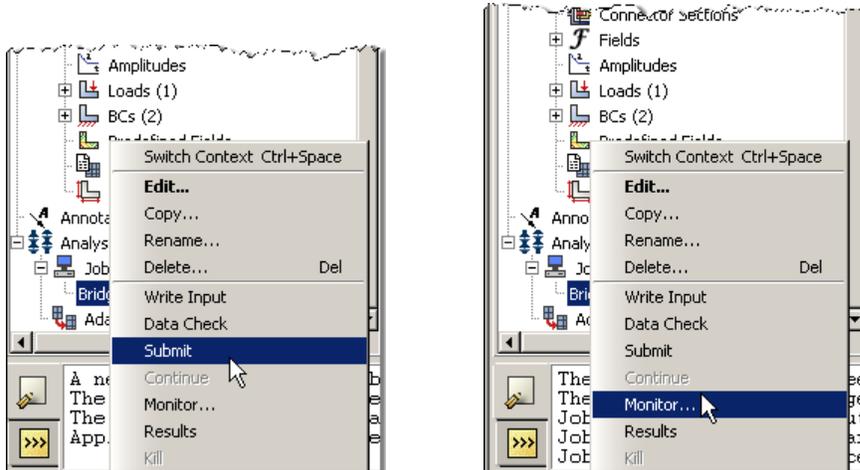
22. In the model tree double click on the “Job” node

- a. Name the job “Bridge”
- b. Click “Continue...”
- c. Give the job a description
- d. Click “OK”

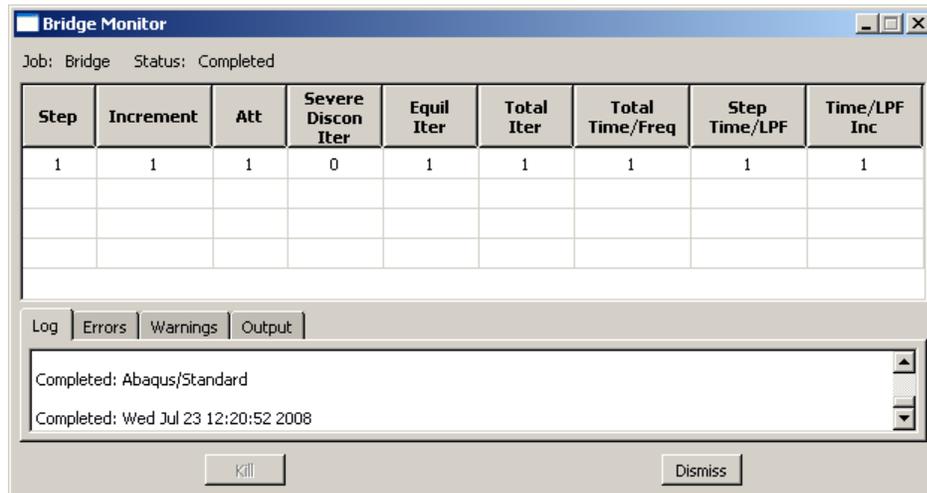


23. In the model tree right click on the job just created (Bridge) and select “Submit”

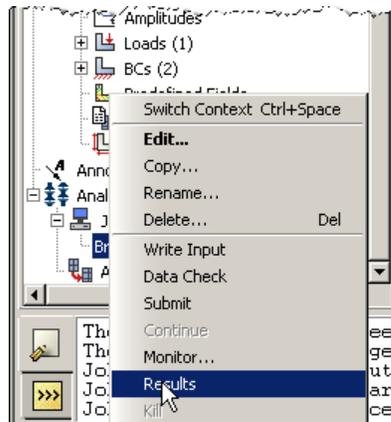
- a. While Abaqus is solving the problem right click on the job submitted (Bridge), and select “Monitor”



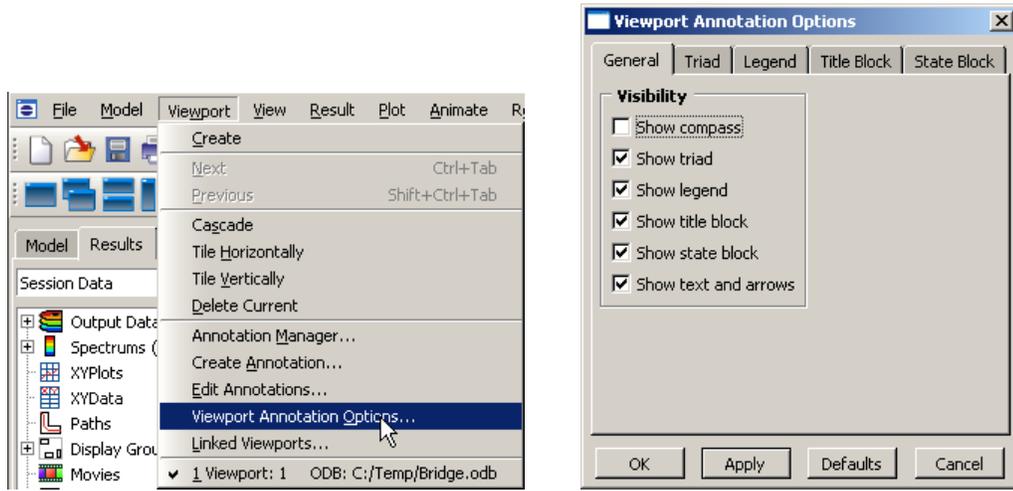
- b. In the Monitor window check that there are no errors or warnings
 - i. If there are errors, investigate the cause(s) before resolving
 - ii. If there are warnings, determine if the warnings are relevant, some warnings can be safely ignored



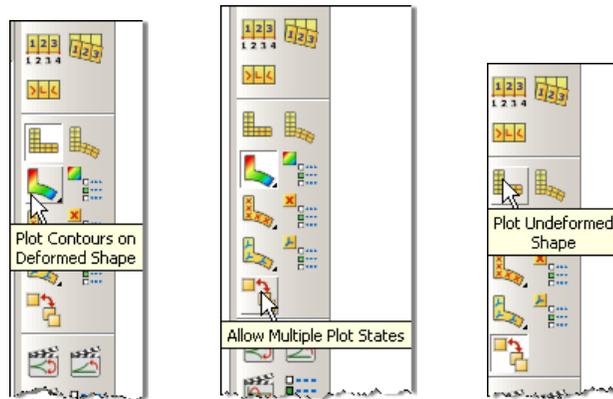
24. In the model tree right click on the submitted and successfully completed job (Bridge), and select "Results"



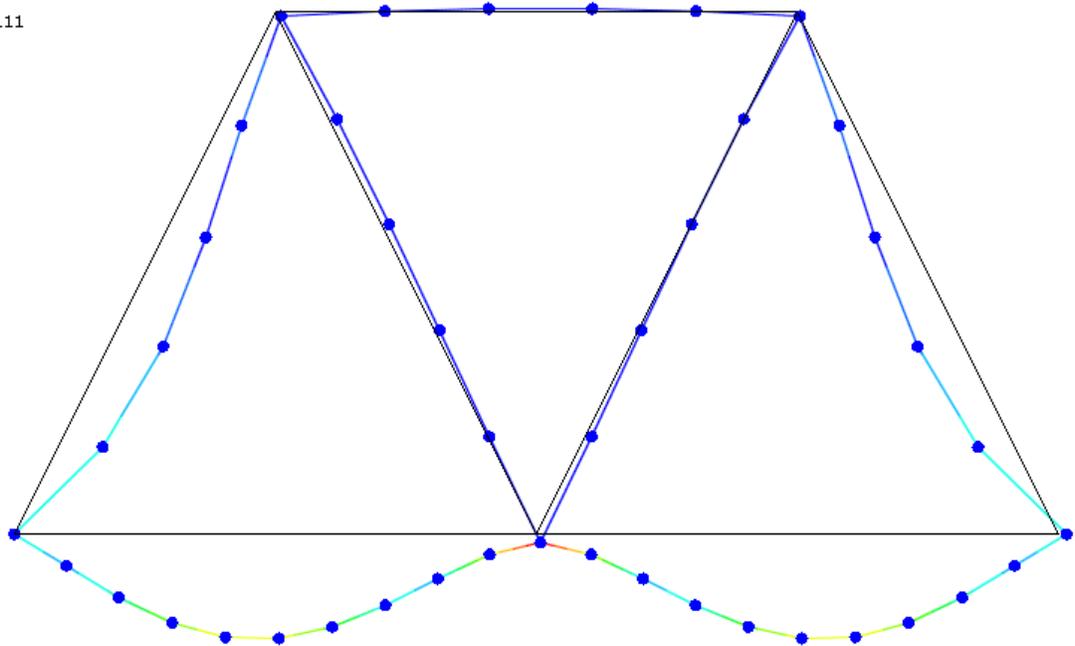
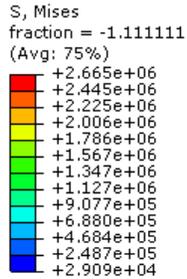
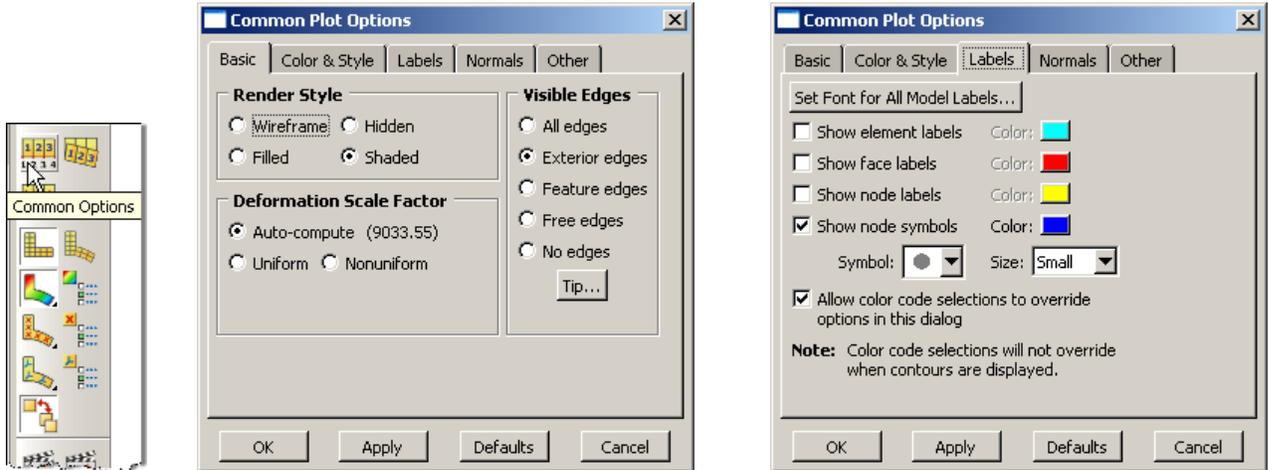
25. In the menu bar click on Viewport → Viewport Annotations Options
 - a. Uncheck the “Show compass option”
 - b. The locations of viewport items can be specified on the corresponding tab in the Viewport Annotations Options
 - c. Click “OK”



26. Display the deformed contour of the (Von) Mises stress overlaid with the undeformed geometry
 - a. In the toolbox area click on the following icons
 - i. “Plot Contours on Deformed Shape”
 - ii. “Allow Multiple Plot States”
 - iii. “Plot Undeformed Shape”



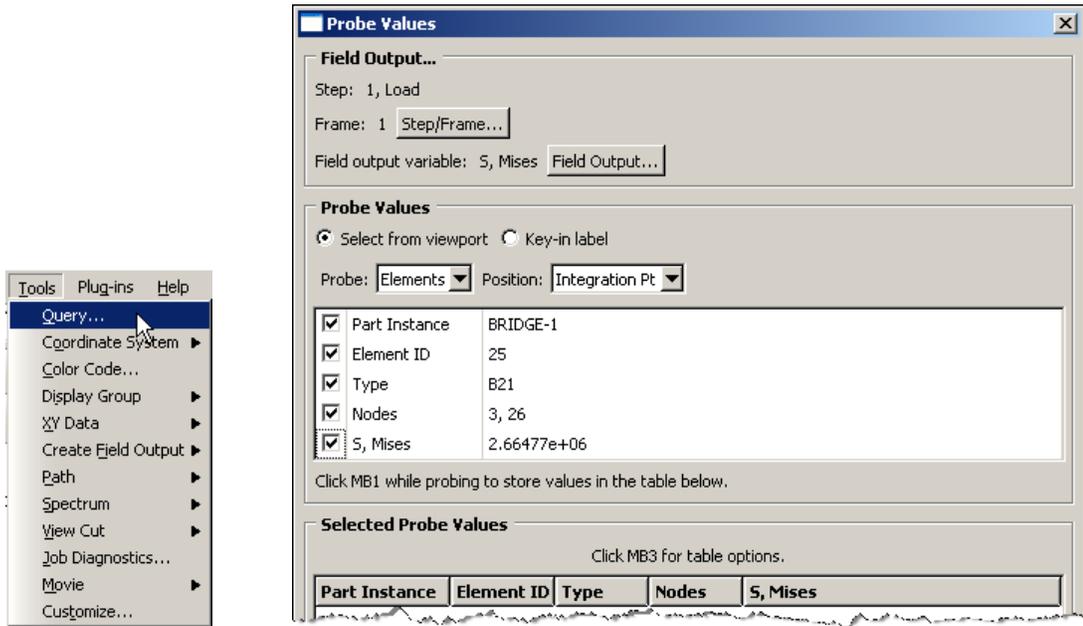
27. In the toolbox area click on the “Common Plot Options” icon
 - a. Note that the Deformation Scale Factor can be set on the “Basic” tab
 - b. On the “Labels” tab check the show node symbols icon
 - c. Click “OK”



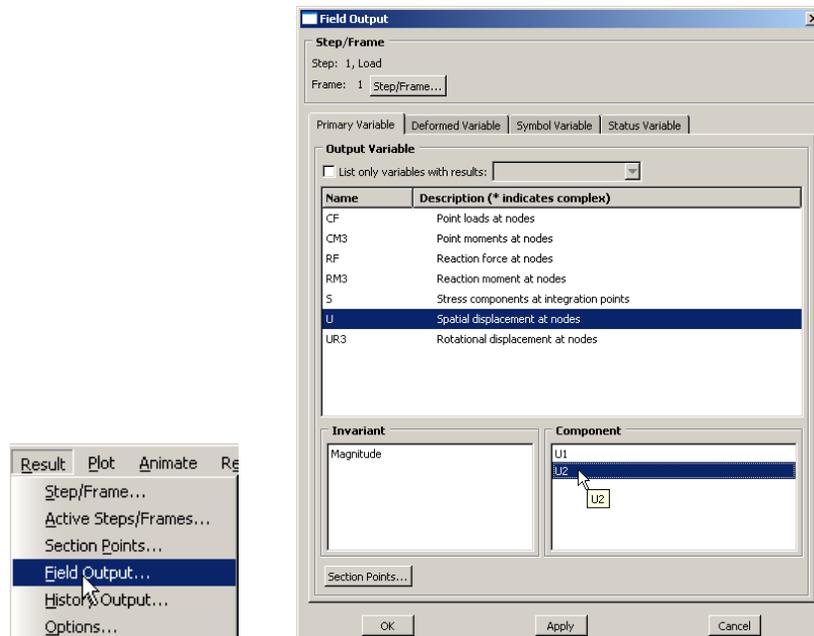
Static analysis of a bridge with a distributed load
 ODB: Bridge.odb Abaqus/Standard Student Edition 6.7-2 Wed Jul 23 12:20:47 Pacific Daylight Time 2008
 Step: Load, Apply beam distributed load to lower members
 Increment 1: Step Time = 1.000
 Primary Var: S, Mises
 Deformed Var: U Deformation Scale Factor: +9.034e+03



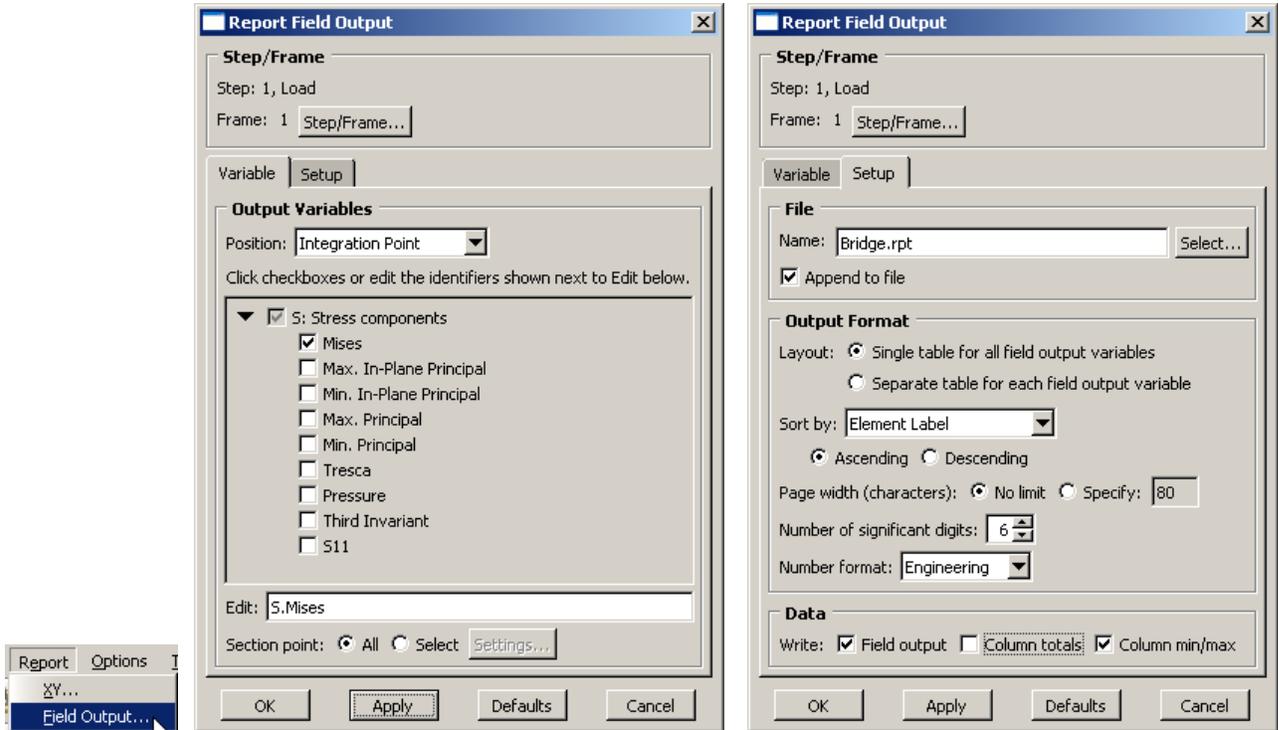
28. To determine the stress values, from the menu bar click Tools → Query
 - a. Check the boxes labeled “Nodes” and “S, Mises”
 - b. In the viewport mouse over the element of interest
 - c. Note that Abaqus reports stress values from the integration points, which may differ slightly from the values determined by projecting values from the surrounding integration points to the nodes
 - i. The minimum and maximum stress values contained in the legend are from the stresses projected to the nodes
 - d. Click on an element to store it in the “Selected Probe Values” portion of the dialogue box
 - e. Click “Cancel”



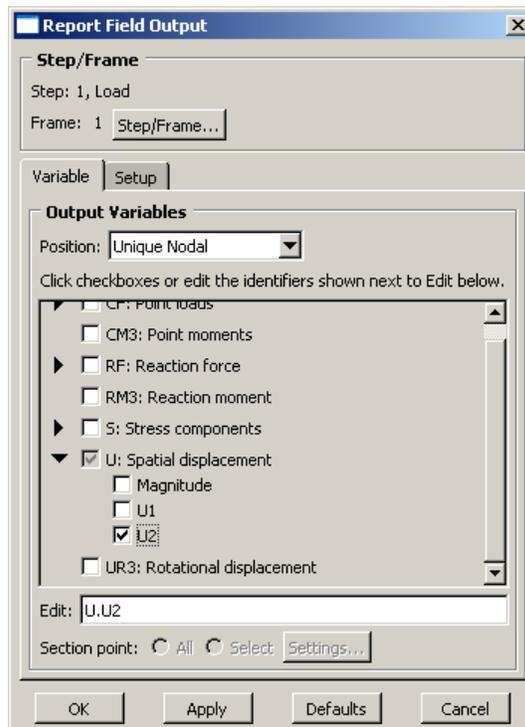
29. To change the output being displayed, in the menu bar click on Results → Field Output
 - a. Select “Spatial displacement at nodes”
 - i. Component = U2
 - b. Click “OK”



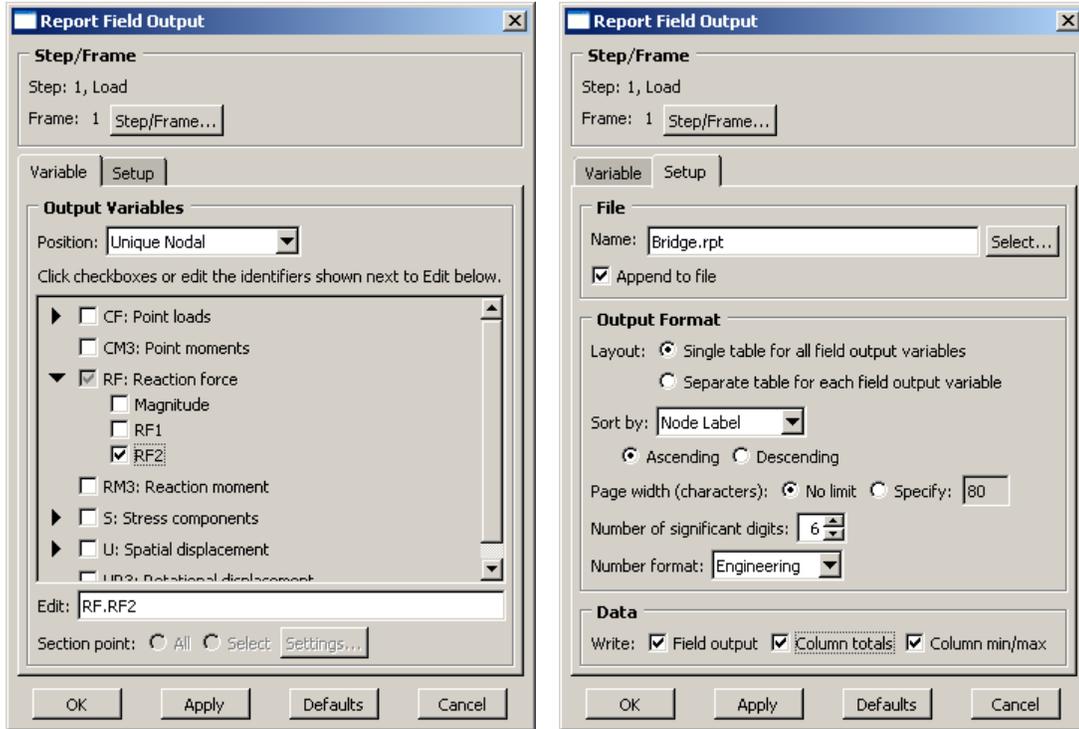
30. To create a text file containing the stresses, vertical displacements, and reaction forces (including the total), in the menu bar click on Report → Field Output
- For the output variable select (Von) Mises
 - On the Setup tab specify the name and the location for the text file
 - Uncheck the “Column totals” option
 - Click “Apply”



- Back on the Variable tab change the position to “Unique Nodal”
- Uncheck the stress variable, and select the U2 spatial displacement
- Click “Apply”



- h. On the Variable tab, uncheck Spatial displacement and select the RF2 reaction force
- i. On the Setup tab, check the "Column totals" option
- j. Click "OK"



- 31. Open the .rpt file with any text editor
 - a. One thing to check is that the total reaction force is equal to the applied load (-10,000 N)

```

ODB: C:/Temp/Bridge.odb
Step: Load
Frame: Increment      1: Step Time =      1.000

Loc 1 : Nodal values from source 1

Output sorted by column "Node Label".

Field Output reported at nodes for part: BRIDGE-1

      Node      RF.RF2
      Label      @Loc 1
-----
      1          0.
      2          0.
      3          0.
      4          5.E+03
      5          5.E+03
      :
      :
      :
      :
Minimum          0.
      At Node      43

Maximum          5.E+03
      At Node      5

      Total          10.E+03

```