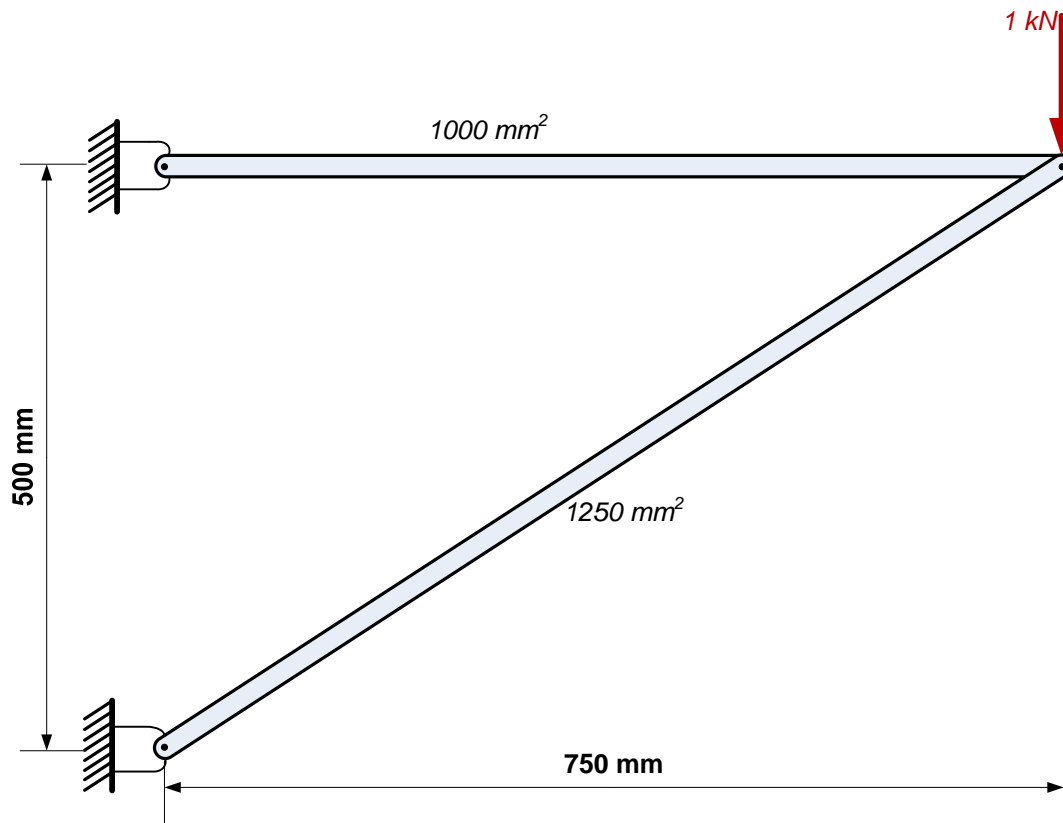


## Abaqus/CAE Truss Tutorial

### Problem Description:

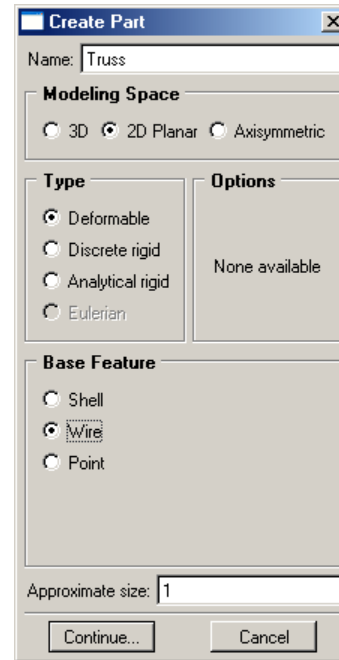
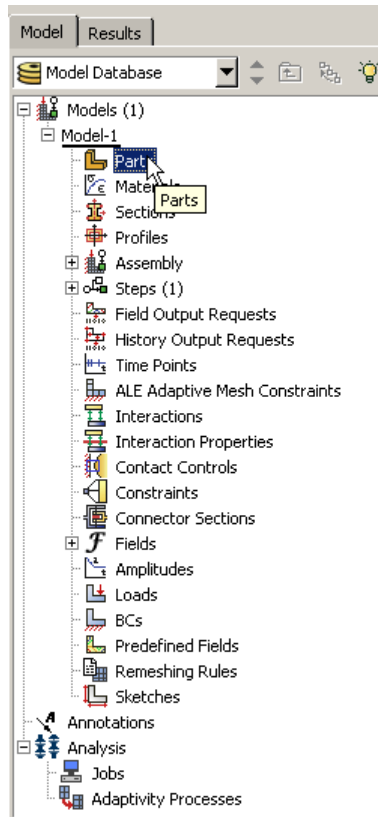
Solve for displacements of the free node and the reaction forces of the truss structure shown in the figure. This is the sample problem from the lecture note example.

Material is Steel with  $E = 210 \text{ GPa}$  and  $\nu = 0.25$ .

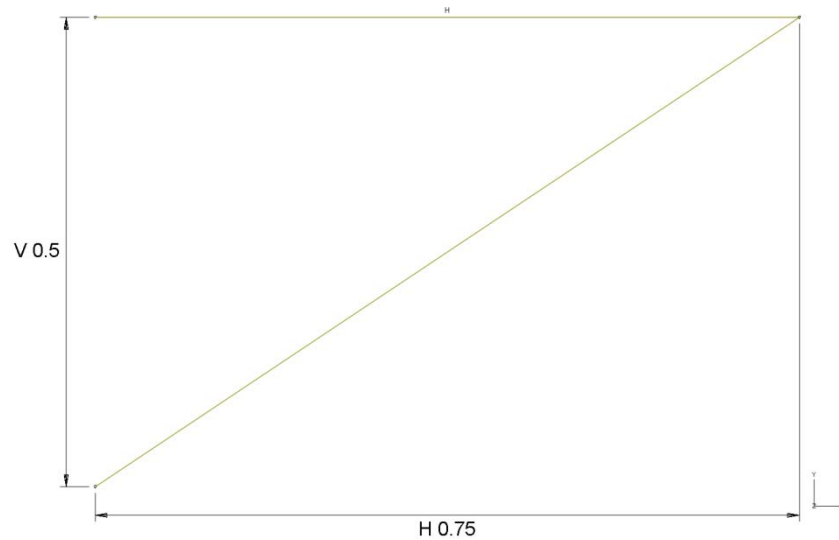


## Analysis Steps

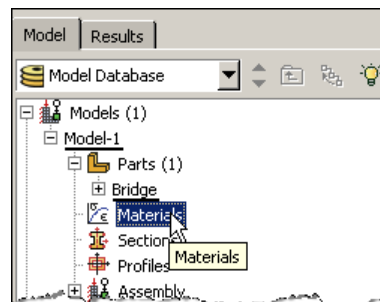
1. Start Abaqus and choose to create a new model database
2. In the model tree double click on the "Parts" node (or right click on "parts" and select Create)



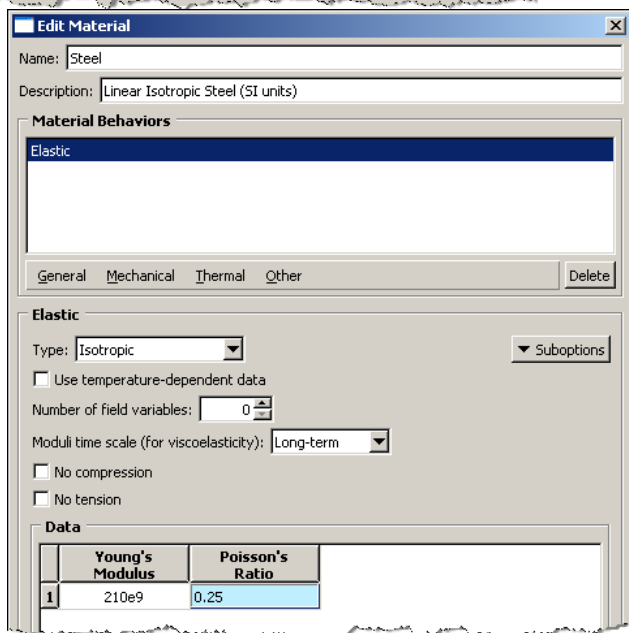
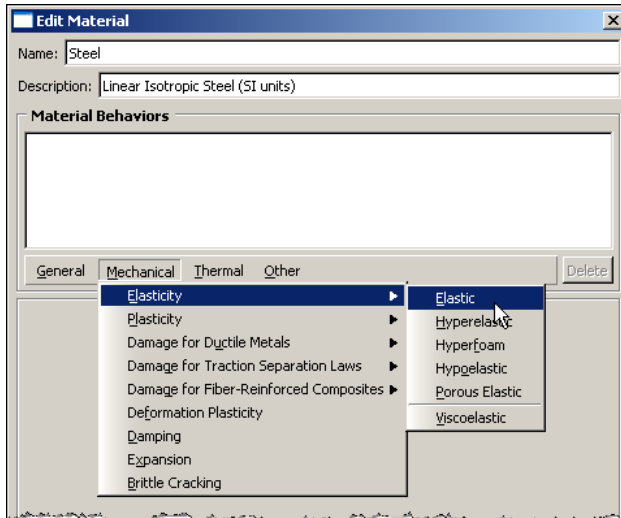
3. In the Create Part dialog box (shown above) name the part and
  - a. Select "2D Planar"
  - b. Select "Deformable"
  - c. Select "Wire"
  - d. Set approximate size = 1
  - e. Click "Continue..."
4. Create the geometry shown below (not discussed here)



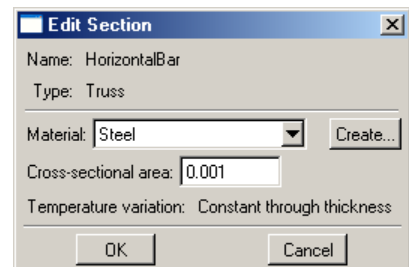
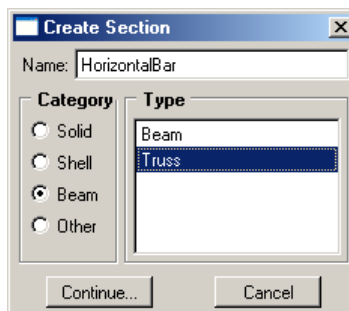
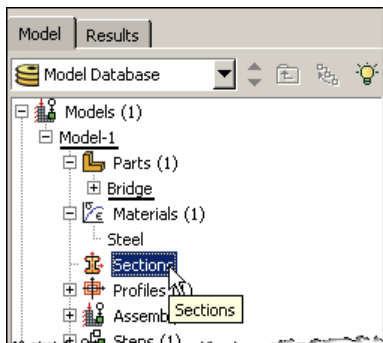
5. Double click on the “Materials” node in the model tree



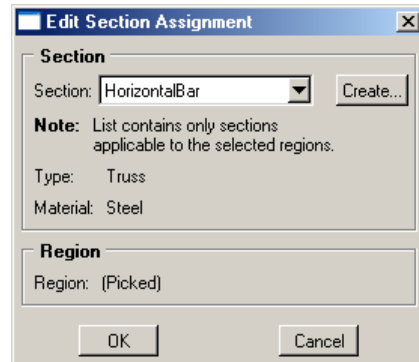
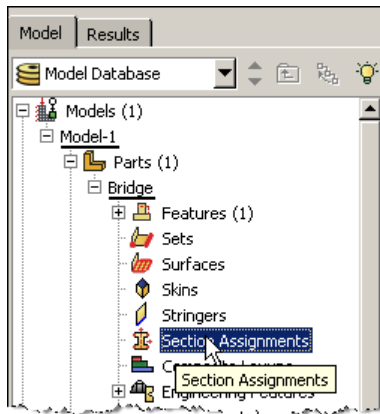
- a. Name the new material and give it a description
- b. Click on the “Mechanical” tab → Elasticity → Elastic
- c. Define Young’s Modulus and Poisson’s Ratio (use base SI units)
  - i. **WARNING: There are no predefined system of units within Abaqus, so the user is responsible for ensuring that the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



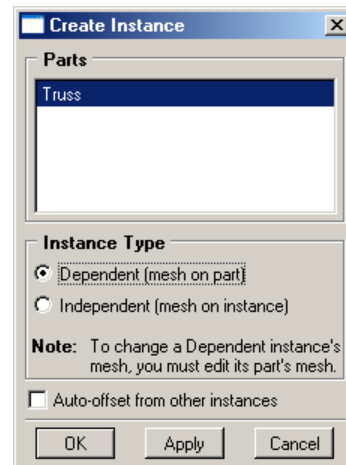
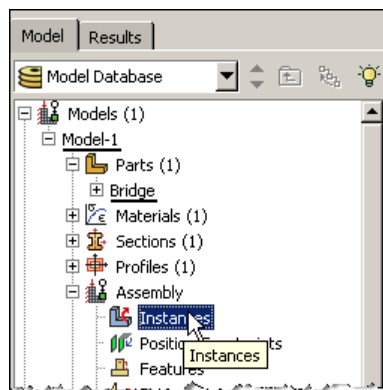
6. Double click on the "Sections" node in the model tree
  - a. Name the section "HorizontalBar" and select "Beam" for both the category and "Truss" for the type
  - b. Click "Continue..."
  - c. Select the material created above (Steel)
  - d. Set cross-sectional area = 0.001 (base SI units, m<sup>2</sup>)
  - e. Click "OK"



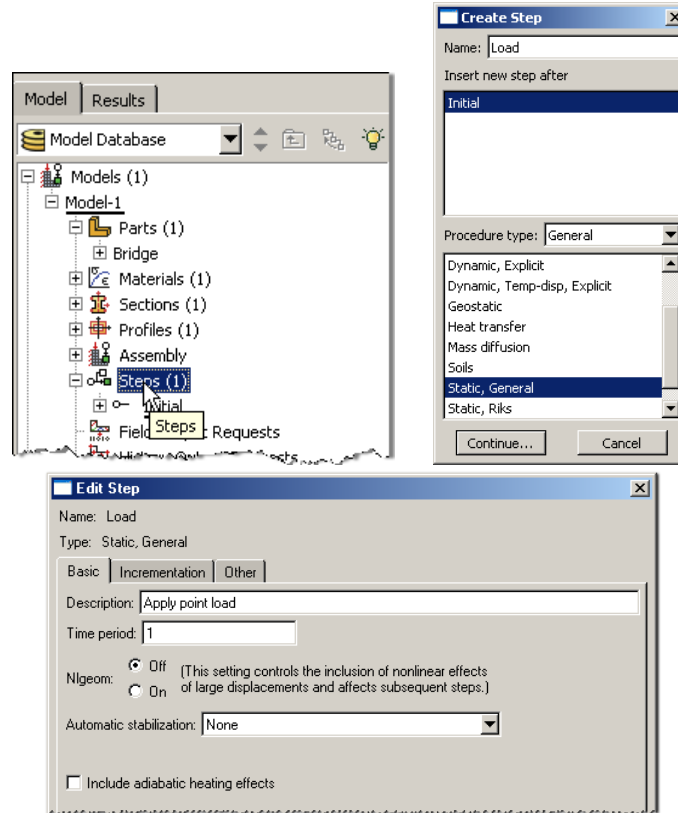
- f. Repeat for the "AngledBar"
  - i. Cross-sectional area=0.00125
- 7. Expand the "Parts" node in the model tree, expand the node of the part just created, and double click on "Section Assignments"
  - a. Select the horizontal portion of the geometry in the viewport
  - b. Click "Done"
  - c. Select the "HorizontalBar" section created above
  - d. Click "OK"



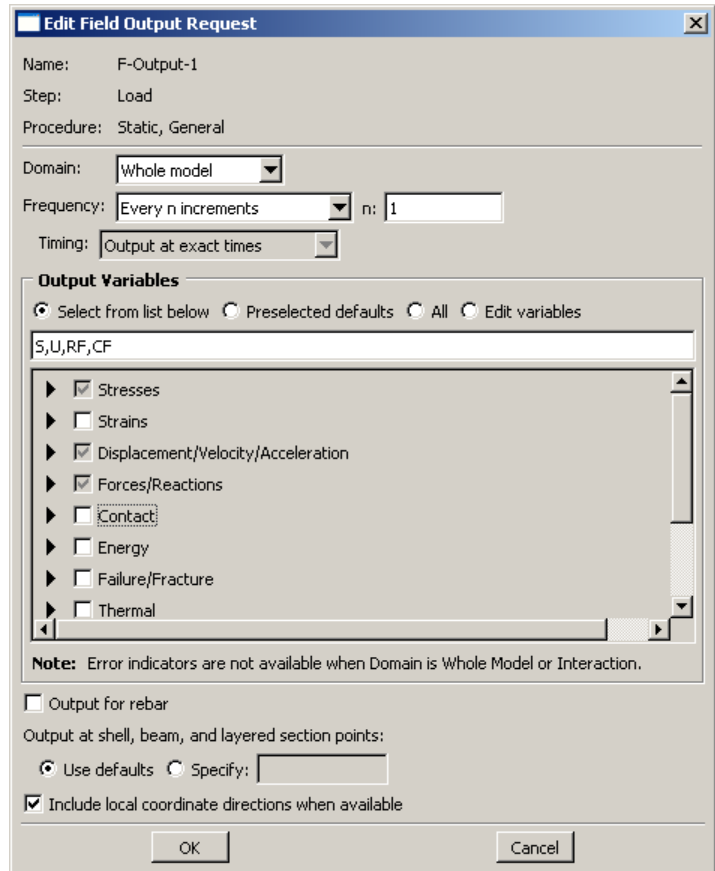
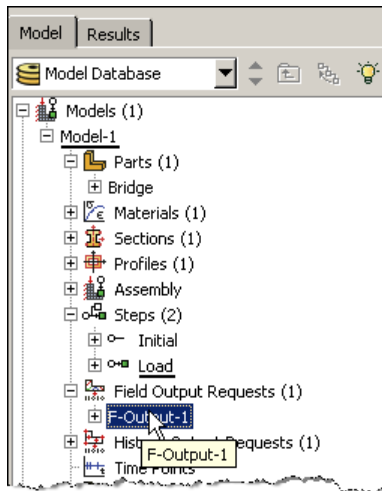
- e. Repeat for the angled portion of the geometry
- 8. Expand the "Assembly" node in the model tree and then double click on "Instances"
  - a. Select "Dependent" for the instance type
  - b. Click "OK"



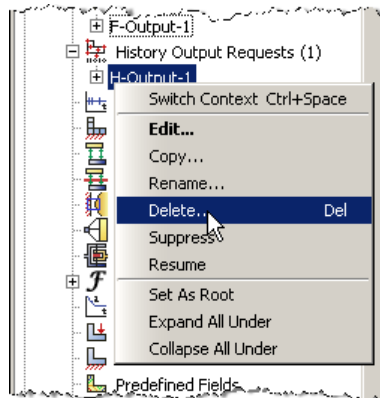
9. Double click on the “Steps” node in the model tree
  - a. Name the step, set the procedure to “General”, and select “Static, General”
  - b. Click “Continue...”
  - c. Give the step a description
  - d. Click “OK”



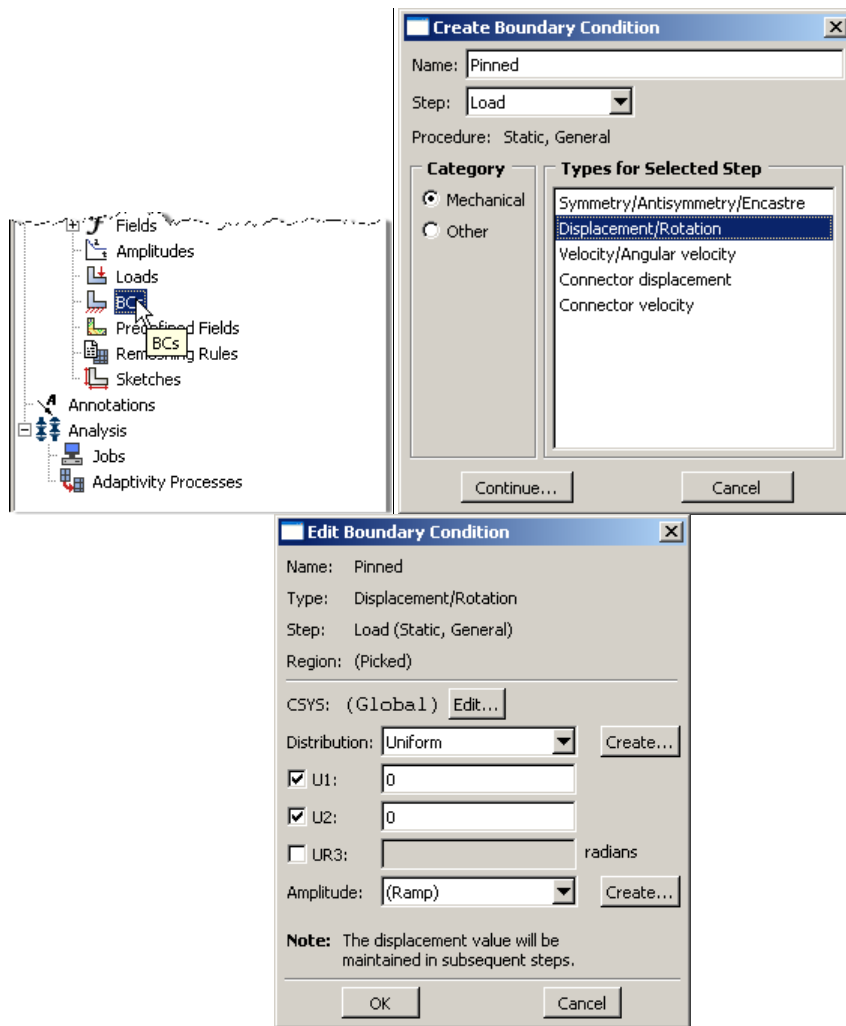
- 10. Expand the Field Output Requests node in the model tree, and then double click on F-Output-1 (F-Output-1 was automatically generated when creating the step)
  - a. Uncheck the variables "Strains" and "Contact"
  - b. Click "OK"



- 11. 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

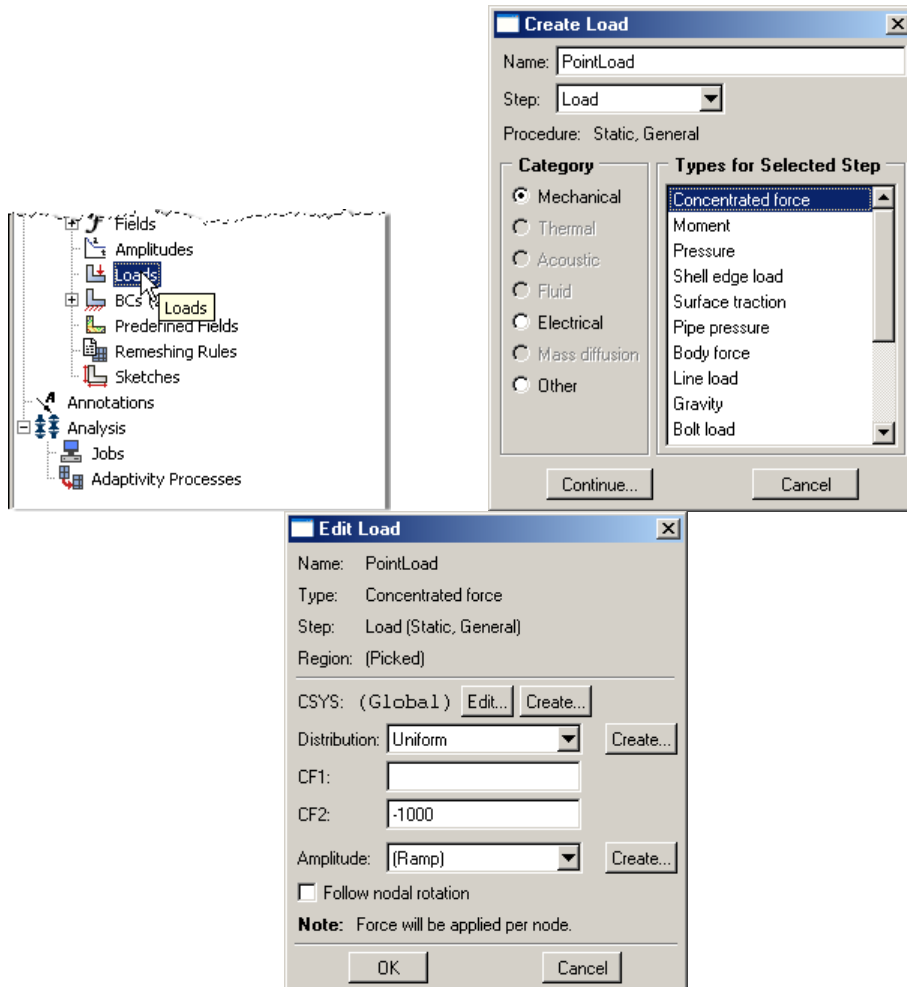


12. Double click on the “BCs” node in the model tree
  - a. Name the boundary conditioned “Pinned” and select “Displacement/Rotation” for the type
  - b. Click “Continue...”
  - c. Select the endpoints on the left (“shift” select ) and press “Done” in the prompt area
  - d. Check the U1 and U2 displacements and set them to 0
  - e. Click “OK”

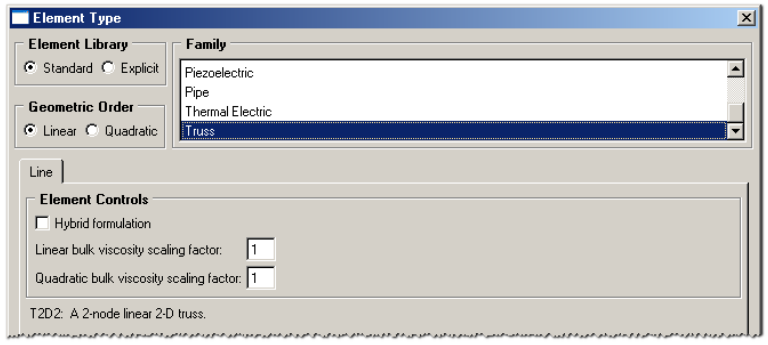
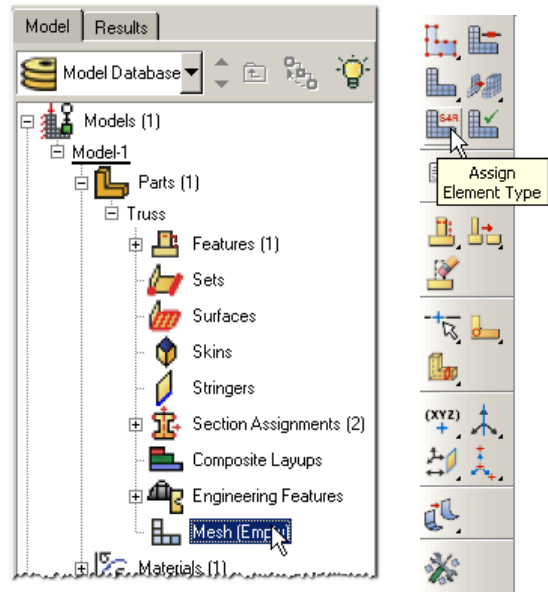




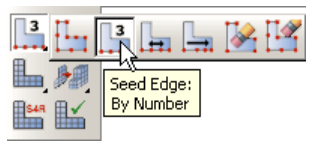
13. Double click on the "Loads" node in the model tree
  - a. Name the load "PointLoad" and select "Concentrated force" as the type
  - b. Click "Continue..."
  - c. Select the vertex on the right and press "Done" in the prompt area
  - d. Specify CF2 = -1000
  - e. Click "OK"



- 14. In the model tree double click on “Mesh” for the Truss part, and in the toolbox area click on the “Assign Element Type” icon
  - a. Select “Standard” for element type
  - b. Select “Linear” for geometric order
  - c. Select “Truss” for family
  - d. Note that the name of the element (B21) and its description are given below the element controls
  - e. Click “OK”



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



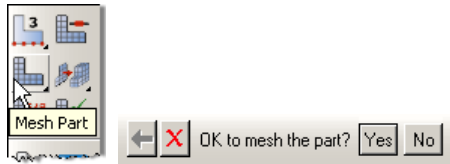
- a. Select the entire geometry and click “Done” in the prompt area



- b. Define the number of elements along the edges as 1 and click “Enter” in the prompt region, then “Done” in response to the next prompt.
- c.

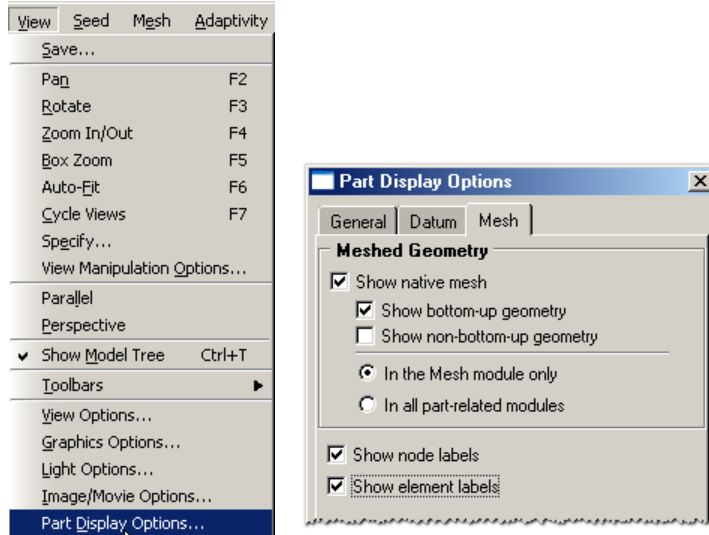
- 16. In the toolbox area click on the “Mesh Part” icon

- a. Click “Yes” in the prompt area



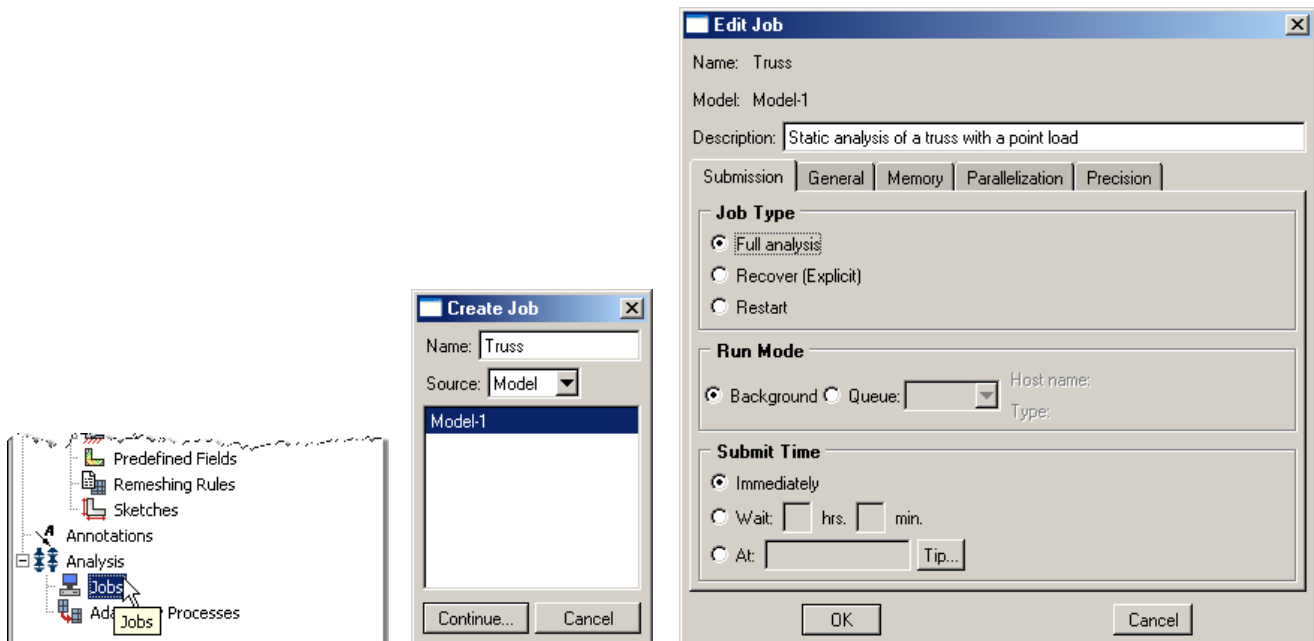
17. In the menu bar select View → Part Display Options

- a. On the Mesh tab check “Show node labels” and “Show element labels”
- b. Click “OK”

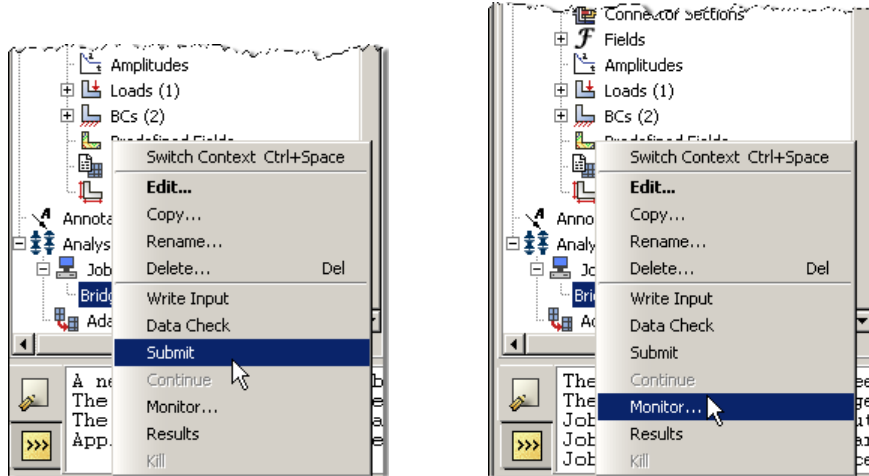


18. In the model tree double click on the “Job” node

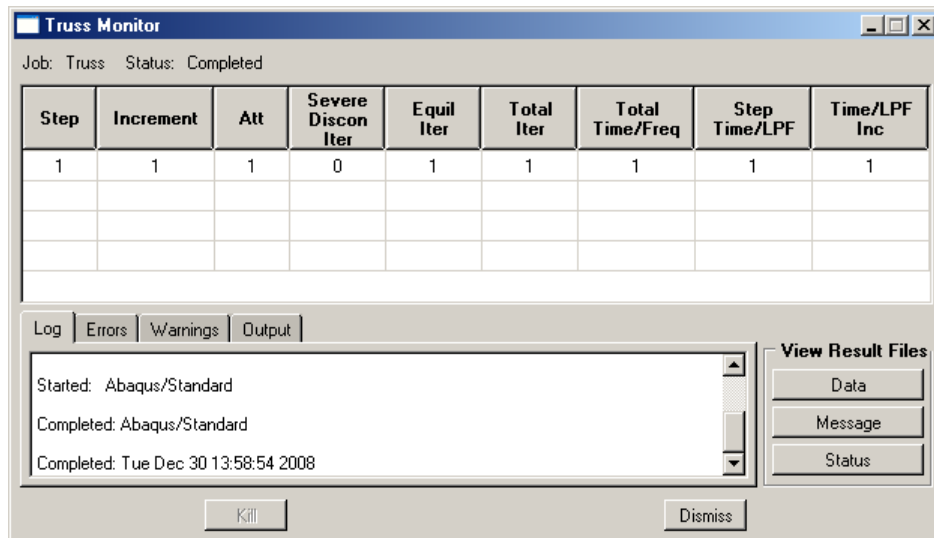
- a. Name the job “Truss”
- b. Click “Continue...”
- c. Give the job a description
- d. Click “OK”



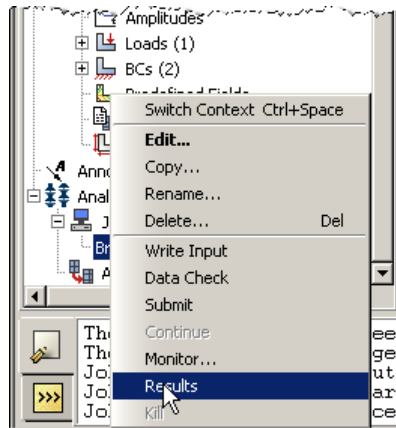
19. In the model tree right click on the job just created (Truss) and select "Submit"
  - a. While Abaqus is solving the problem right click on the job submitted (Truss), and select "Monitor"



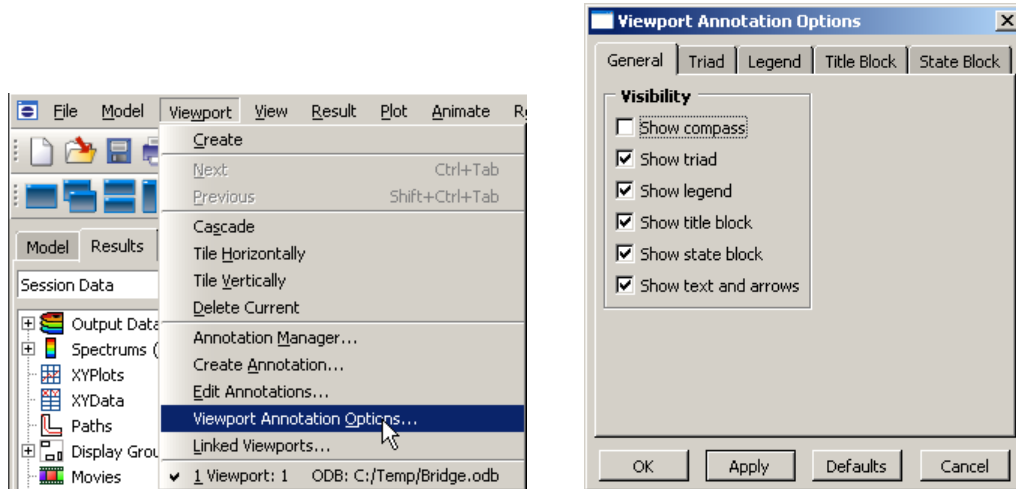
- b. In the Monitor window check that there are no errors or warnings
  - i. If there are errors, investigate the cause(s) before resolving
  - ii. If there are warnings, determine if the warnings are relevant, some warnings can be safely ignored



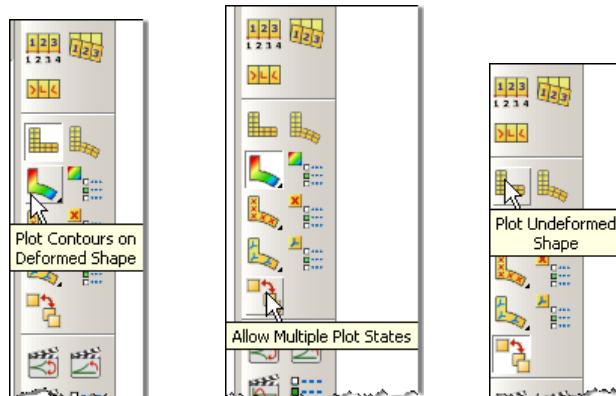
20. In the model tree right click on the submitted and successfully completed job (Truss), and select "Results"



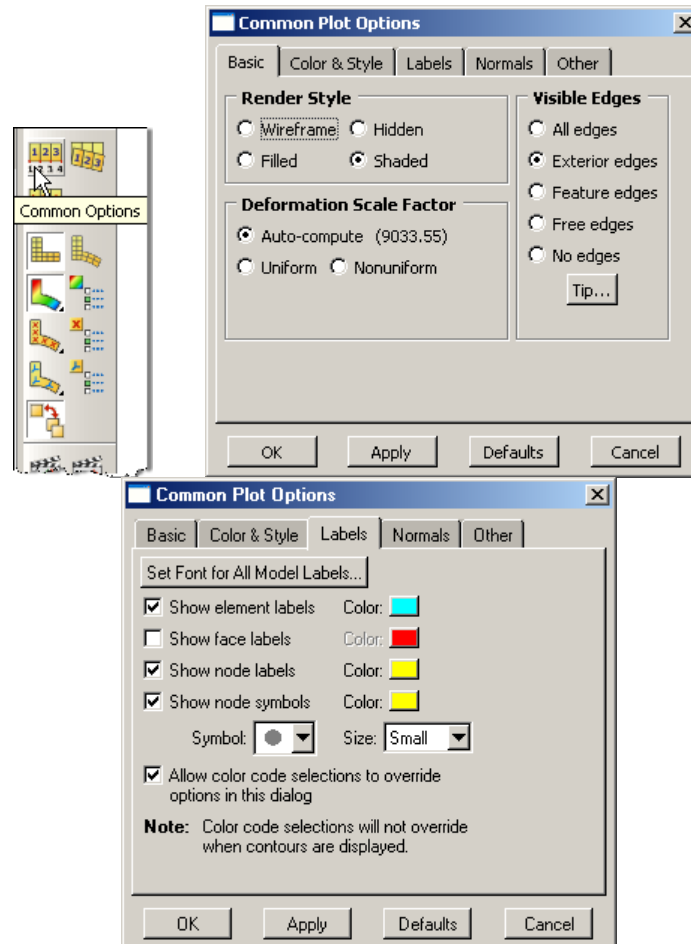
21. In the menu bar click on Viewport → Viewport Annotations Options
  - a. Uncheck the “Show compass option”
  - b. The locations of viewport items can be specified on the corresponding tab in the Viewport Annotations Options
  - c. Click “OK”

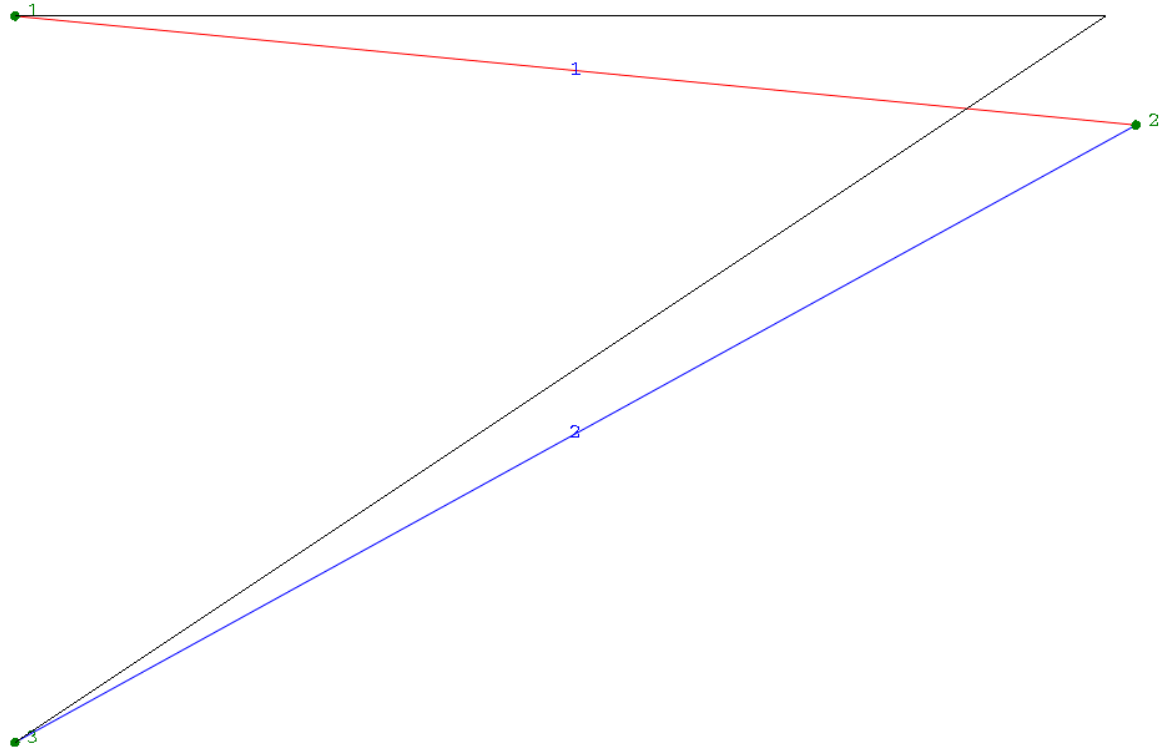
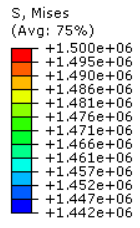


22. Display the deformed contour of the (Von) Mises stress overlaid with the undeformed geometry
  - a. In the toolbox area click on the following icons
    - i. “Plot Contours on Deformed Shape”
    - ii. “Allow Multiple Plot States”
    - iii. “Plot Undeformed Shape”



23. In the toolbox area click on the “Common Plot Options” icon
- Note that the Deformation Scale Factor can be set on the “Basic” tab
  - On the “Labels” tab check “Show element labels”, “Show node labels”, and “Show node symbols”
  - Click “OK”

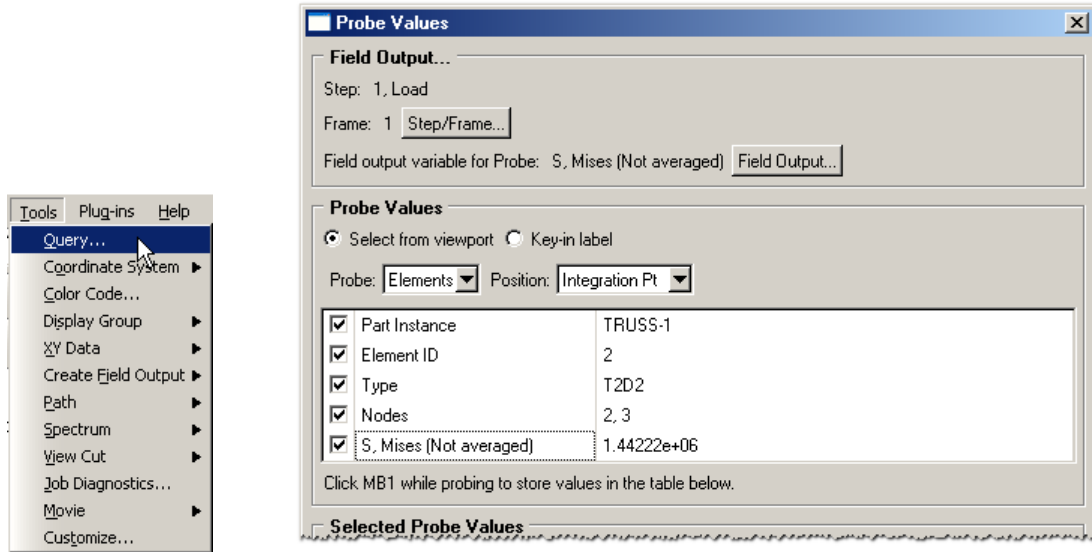




Static analysis of a truss with a point load  
ODB: Truss.odb Abaqus/Standard Version 6.8-1 Tue Dec 30 13:58:49 Pacific Standard Time 2008  
Step: Load, Apply point load  
Increment 1: Step Time = 1.000  
Primary Var: S, Mises  
Deformed Var: U Deformation Scale Factor: +3.907e+03

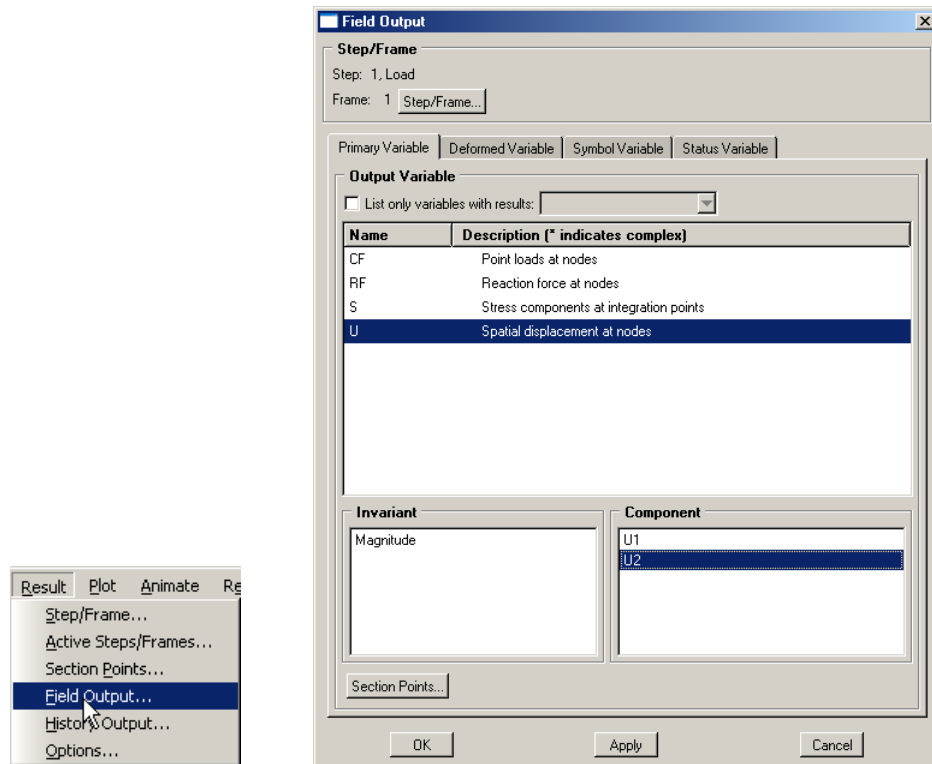


24. To determine the stress values, from the menu bar click Tools → Query → Probe Values, and click OK.
  - a. Check the boxes labeled “Nodes” and “S, Mises”
  - b. In the viewport mouse over the element of interest
  - c. Note that Abaqus reports stress values from the integration points, which may 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”

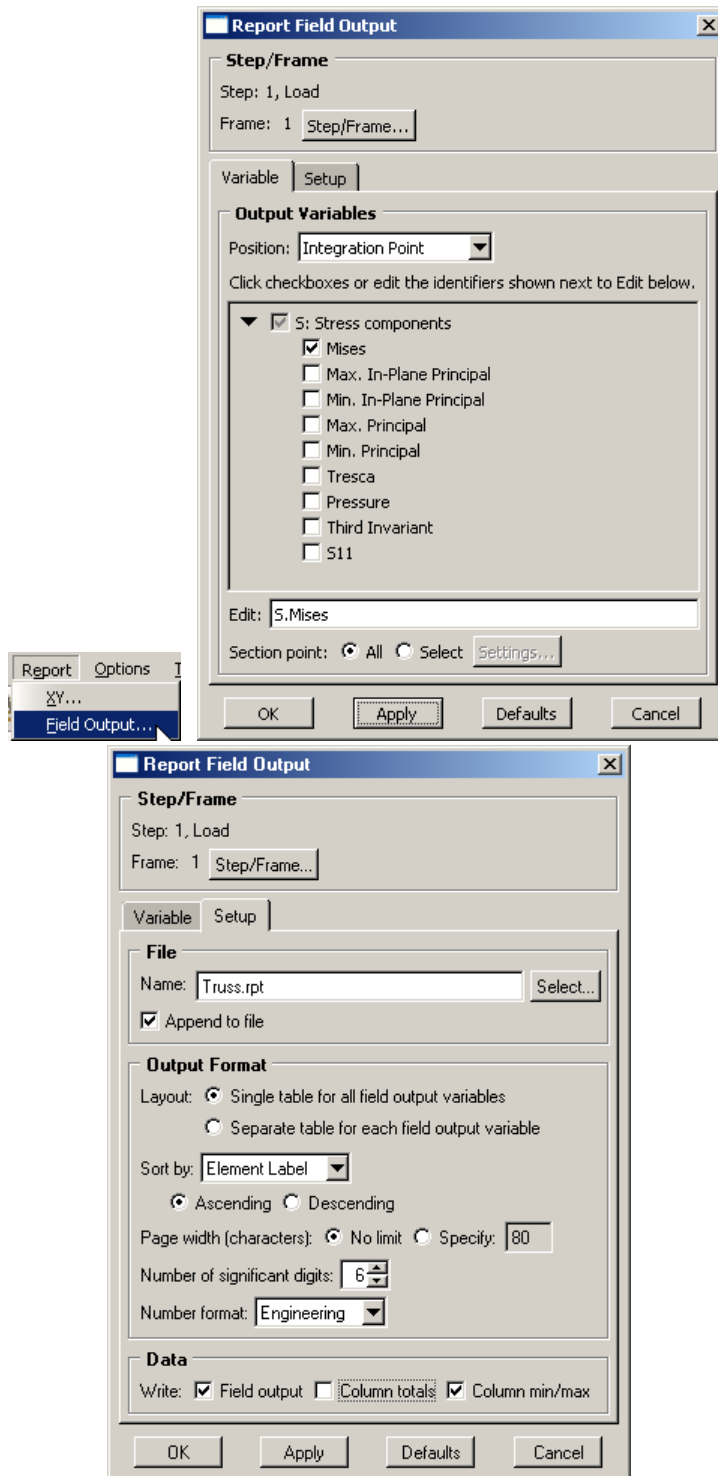


25. To change the output being displayed, in the menu bar click on Results → Field Output
  - a. Select “Spatial displacement at nodes”
    - i. Component = U2
  - b. Click “OK”

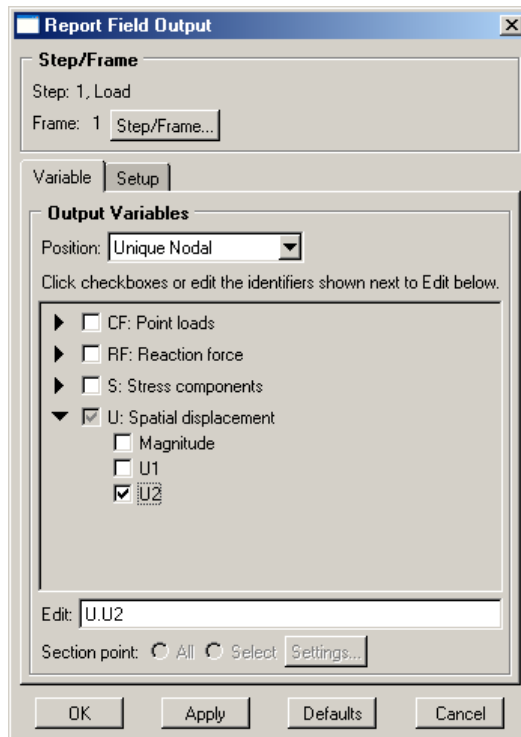




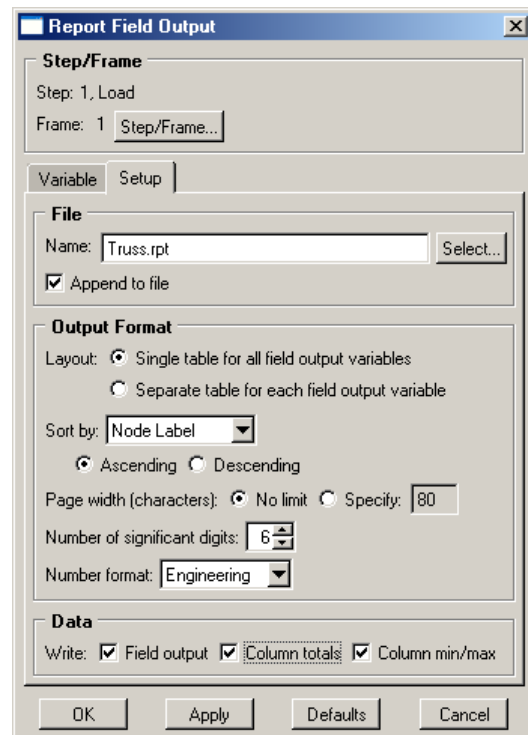
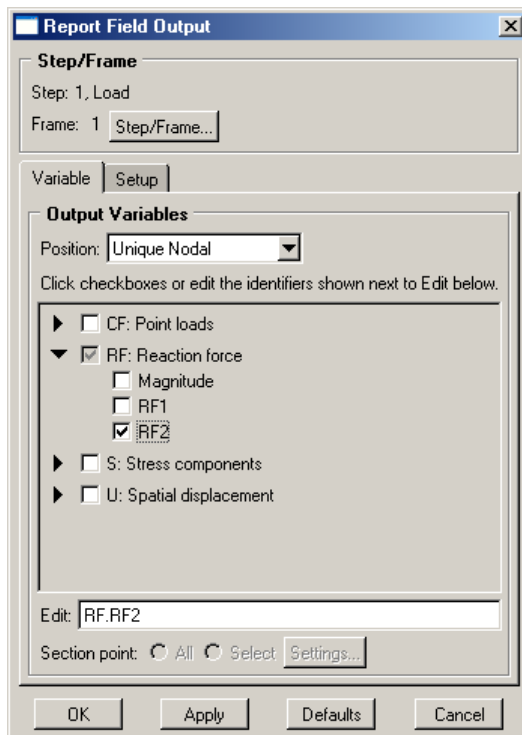
26. To create a text file containing the stresses, vertical displacements, and reaction forces (including the total), in the menu bar click on Report → Field Output
- For the output variable select (Von) Mises
  - On the Setup tab specify the name and the location for the text file
  - Uncheck the “Column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



- e. Back on the Variable tab change the position to “Unique Nodal”
- f. Uncheck the stress variable, and select the U2 spatial displacement
- g. 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”



27. Open the .rpt file with any text editor
- One thing to check is that the total downward reaction force is equal to the applied load (1,000 N)

```

ODB: C:/Temp/Truss.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: TRUSS-1

```

Node Label	RF.RF2 @Loc 1
1	-0.
2	0.
3	1.E+03
Minimum	0.
At Node	2
Maximum	1.E+03
At Node	3
Total	1.E+03