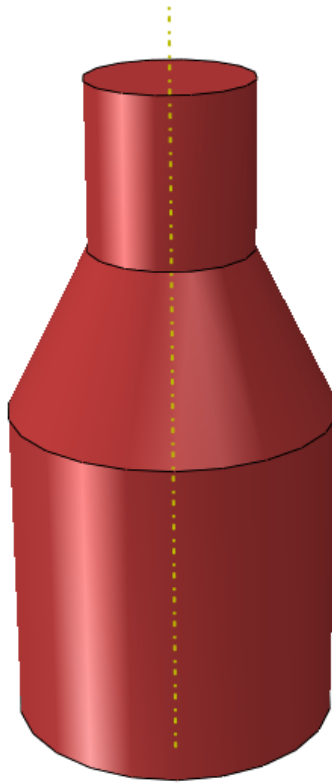# Abaqus/CAE Axisymmetric Tutorial

Problem Description

A round bar with varying diameter has a total load of 1000 N applied to its top face. The bottom of the bar is completely fixed. Determine stress and displacement values in the bar resulting from the load.

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 select
   a. Axisymmetric
   b. Deformable
   c. Shell
   d. Approximate size = 0.2
4. Create the geometry shown below (not discussed here)
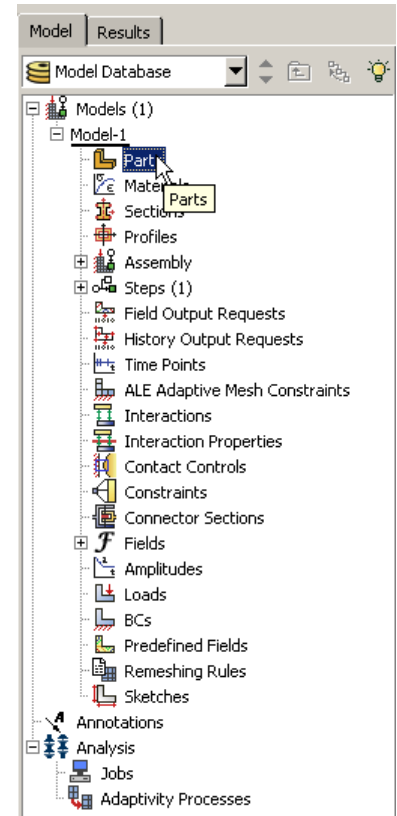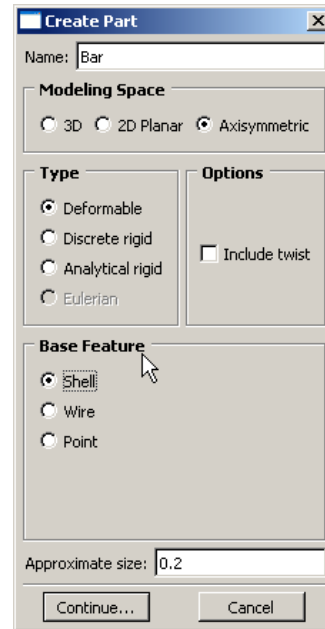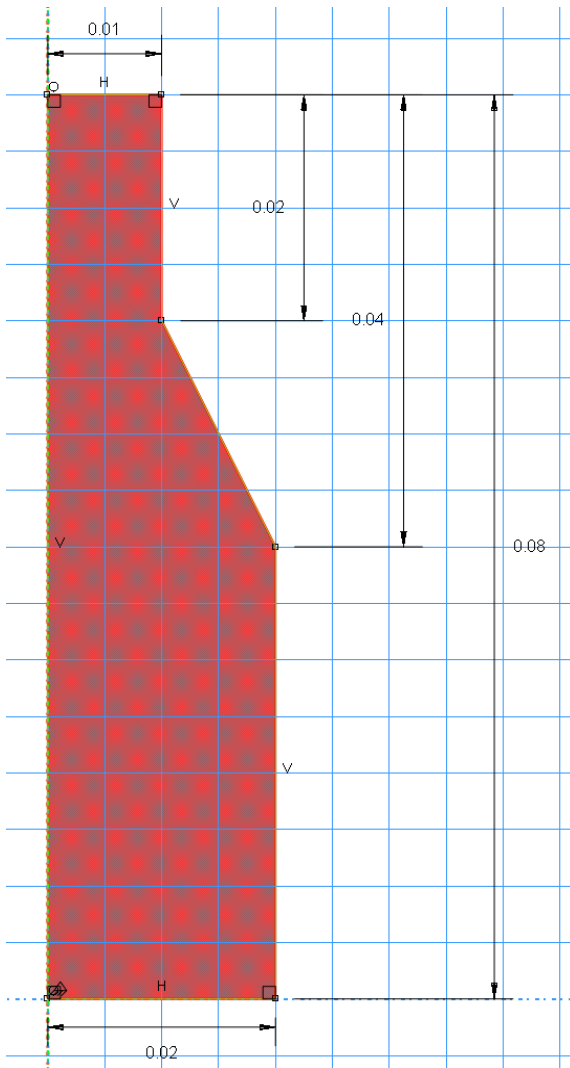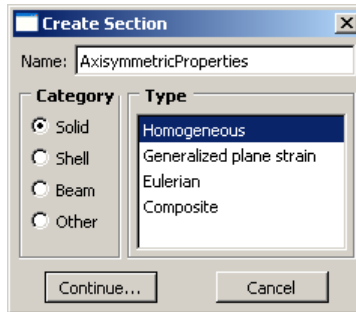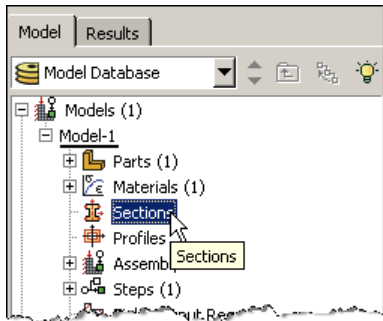
5. Double click on the "Materials" node in the model tree
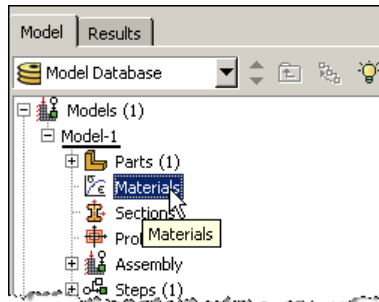
   a. Name the new material and give it a description
   b. Click on the "Mechanical" tab➜Elasticity➜Elastic
   c. Define Young's Modulus and the Poisson's Ratio (use SI units)
       i. **WARNING: There are no predefined system of units within Abaqus, so the user is responsible for ensuring that the correct values are specified**
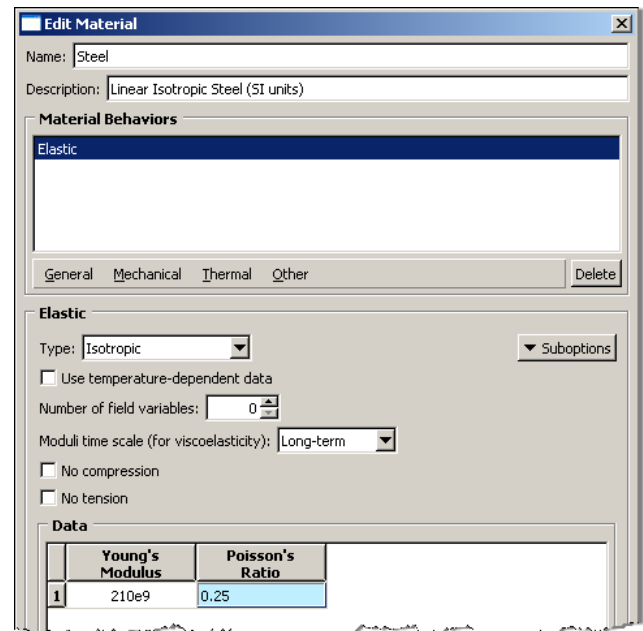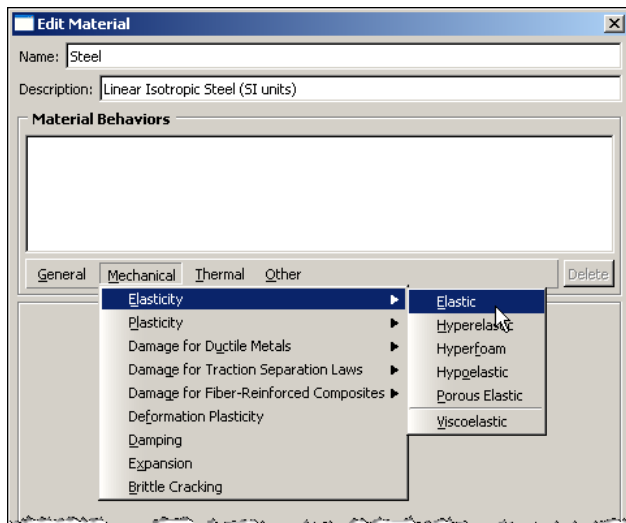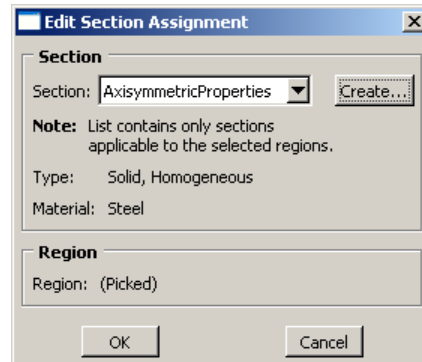

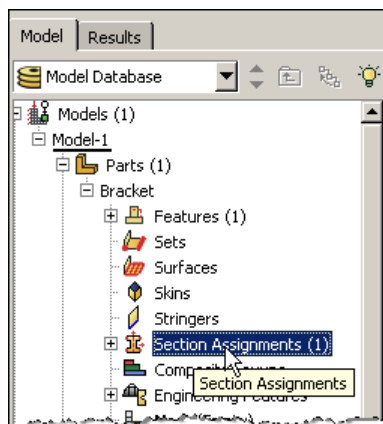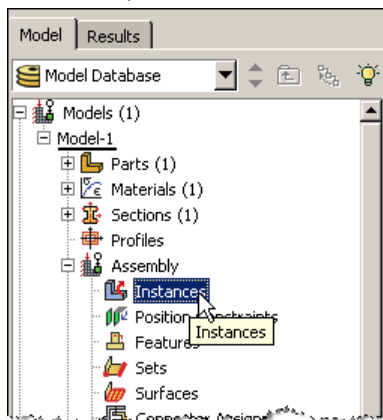6. Double click on the "Sections" node in the model tree
   a. Name the section "AxisymmetricProperties" and select "Solid" for the category and "Homogeneous" for the type
   b. Select the material created above (Steel)

7. Expand the "Parts" node in the model tree and double click on "Section Assignments"
    a. Select the surface geometry in the viewport
   b. Select the section created above (AxisymmetricProperties)

8. Expand the "Assembly" node in the model tree and then double click on "Instances"
   a. Select "Dependent" for the instance type

9. In the model tree, under the expanded "Assembly" node, double click on "Sets"
    a. Name the set "Fixed"
   b. Select the lower edge of the surface in the viewport
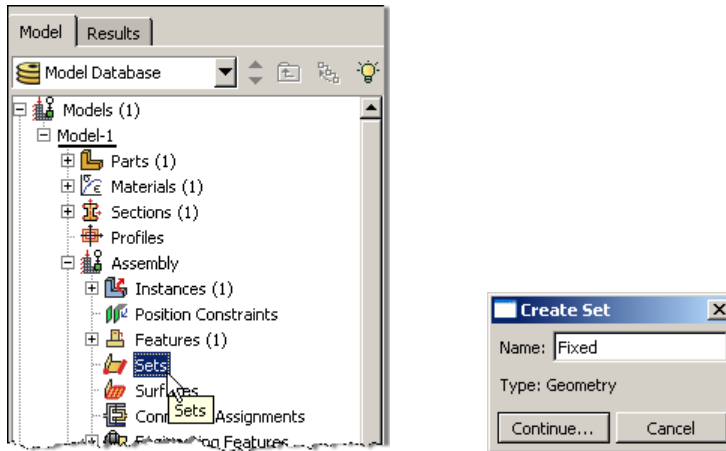
c. Create another set named "Symmetry"

d. Select the left edge of the surface in the viewport

10. In the model tree, under the expanded "Assembly" node, double click on "Surfaces"

a. Name the surface "PressureLoad"

b. Select the top edge of the surface in the viewport
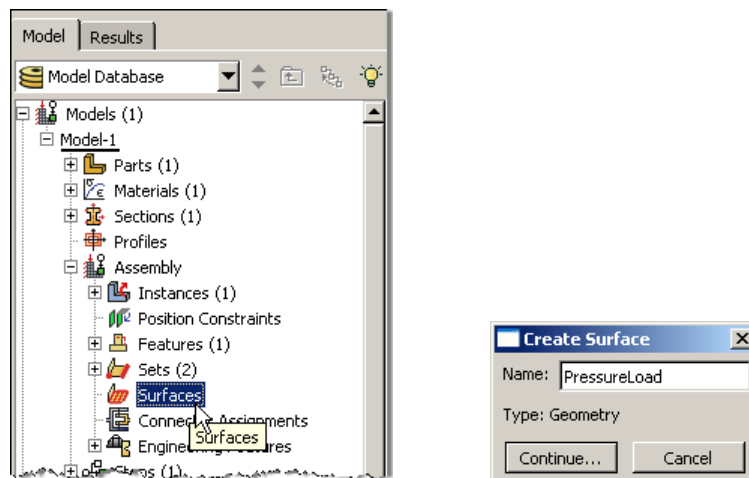
11. Double click on the "Steps" node in the model tree

a. Name the step, set the procedure to "General", and select "Static, General"

b. Give the step a description
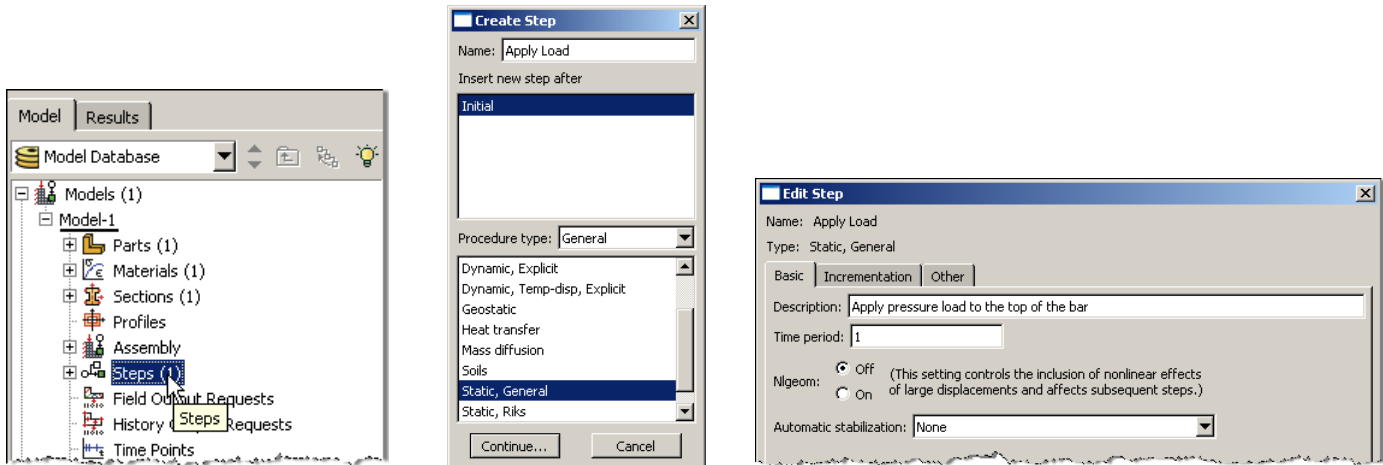
12. Expand the Field Output Requests node in the model tree, and then double click on F-Output-1 (F-Output-1 was automatically generated when creating the step)
   a. Uncheck the variables "Strains" and "Contact"
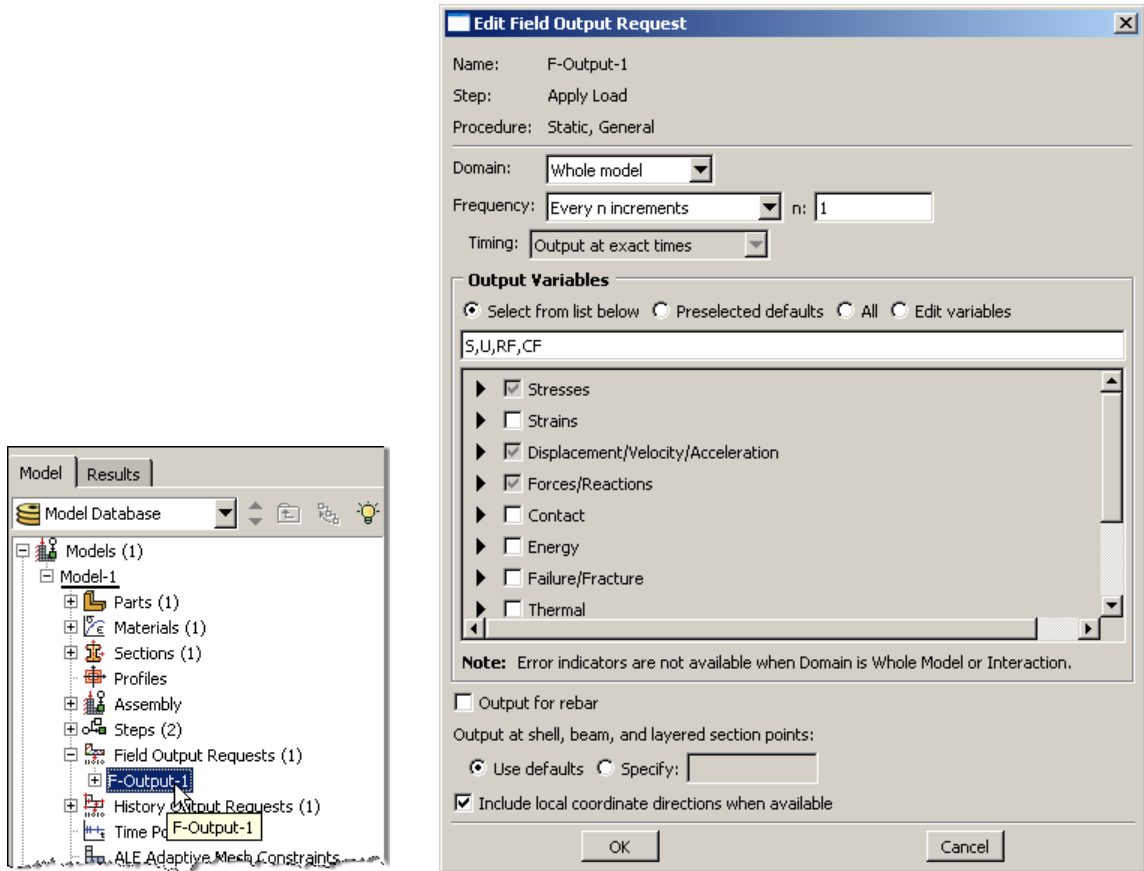
13. Expand the History Output Requests node in the model tree, and then right click on H-Output-1 (H-Output-1 was automatically generated when creating the step) and select Delete
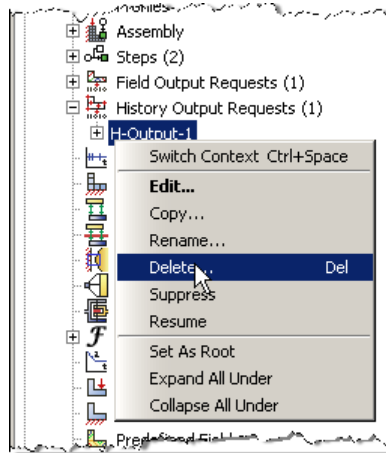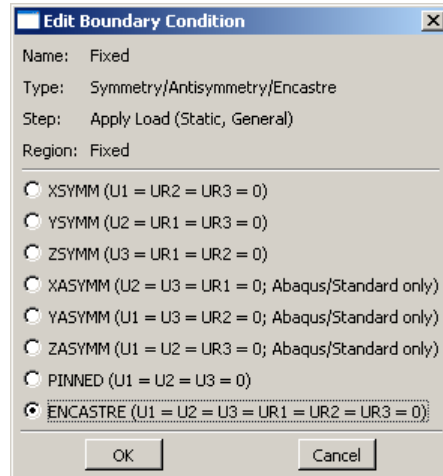
14. Double click on the "BCs" node in the model tree
    a. Name the boundary conditioned "Fixed" and select "Symmetry/Antisymmetry/Encastre" for the type
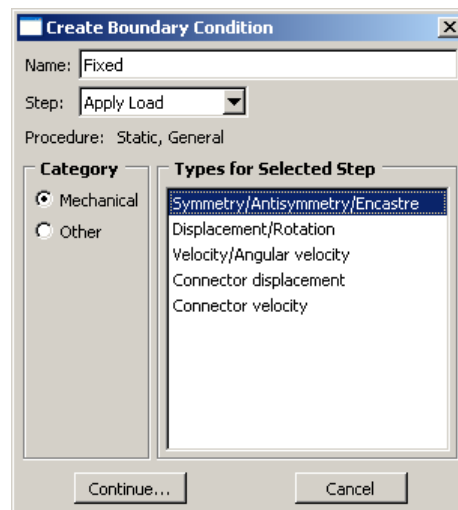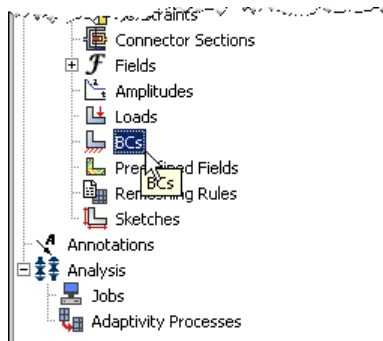
    b. In the prompt area click on the Sets button
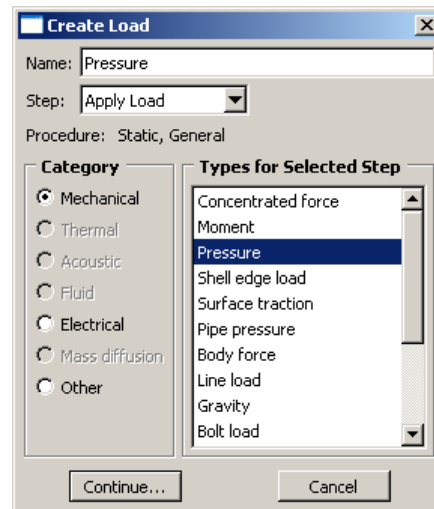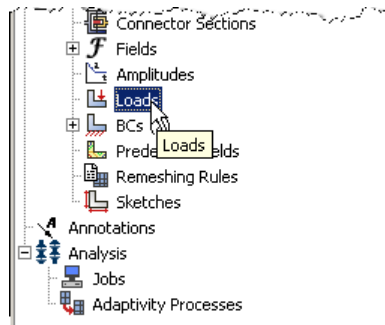   c. Select the set named "Fixed"
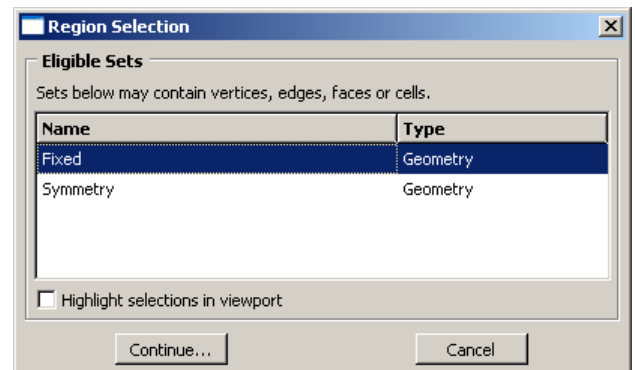
    d. Select "ENCASTRE" for the boundary condition

e. Repeat the procedure for the symmetry restraint using the set named "Symmetry", select "XSYMM" for the boundary condition
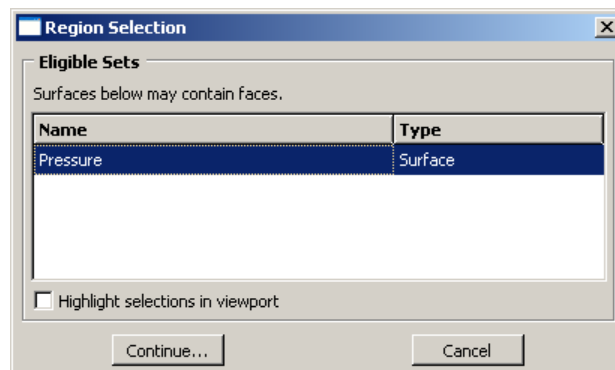
15. Double click on the "Loads" node in the model tree

a. Name the load "Pressure" and select "Pressure" as the type

b. Select surface named "Pressure"

c. For the magnitude enter

16. In the model tree double click on "Mesh" for the Bar 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 "Axisymmetric Stress" for family

    d. Note that the name of the element (CAX4R) and its description are given below the element controls
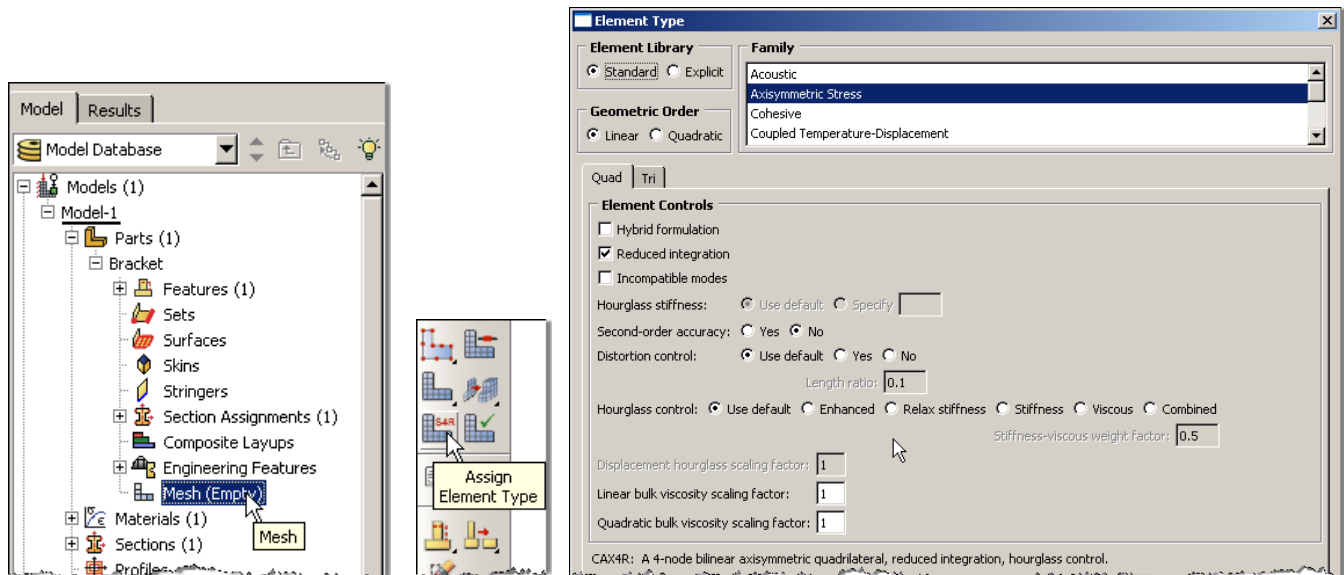
17. In the toolbox area click on the "Assign Mesh Controls" icon

    a. Change the element shape to "Quad"

    b. Change the Algorithm to "Medial axis" for a more structured mesh
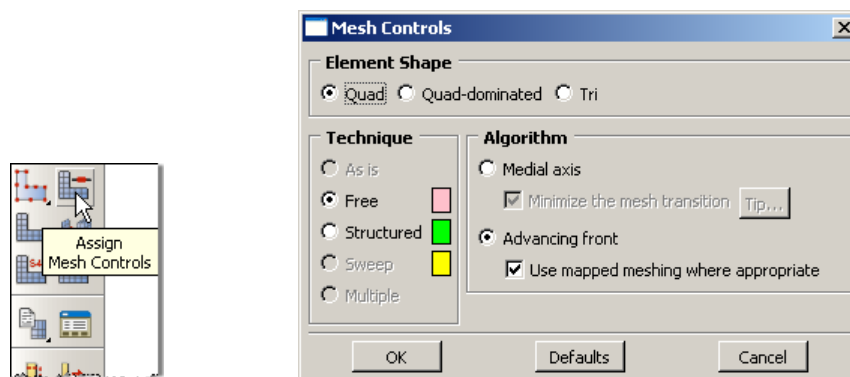
18. In the toolbox area click on the "Seed Part" icon
    a. Set the approximate global size to 0.005
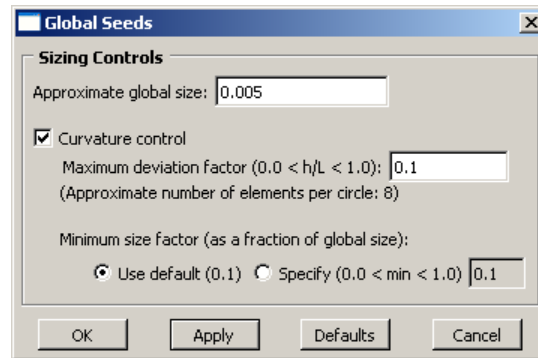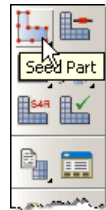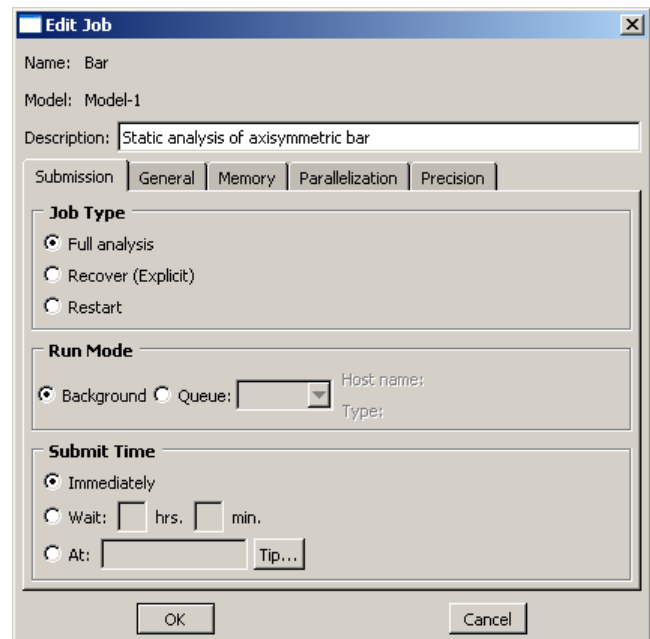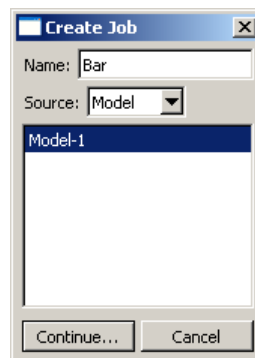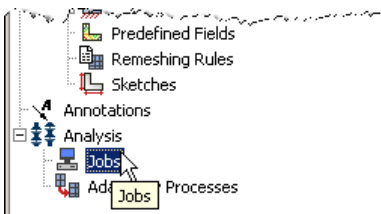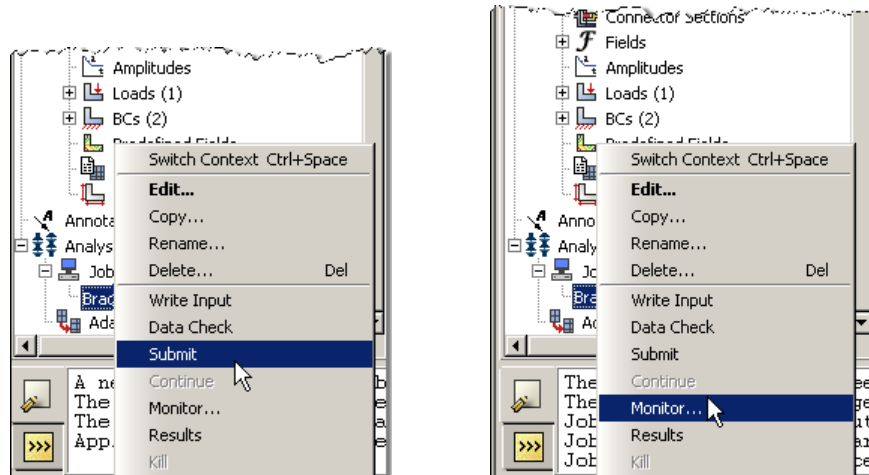
19. In the toolbox area click on the "Mesh Part" icon

20. In the model tree double click on the "Job" node
    a. Name the job "Bar"
   b. Give the job a description

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

22. In the model tree right click on the submitted and successfully completed job (Bar), and select "Results"
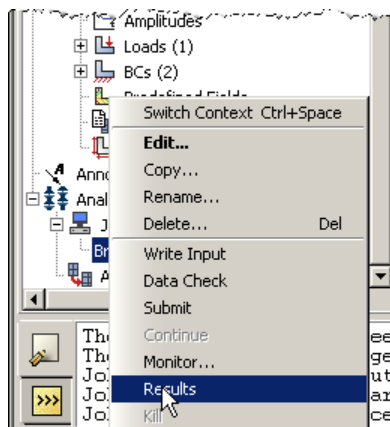
23. In the menu bar click on Viewport➔Viewport Annotations Options
    a. Uncheck the "Show compass option"
    b. The locations of viewport items can be specified on the corresponding tab in the Viewport Annotations Options
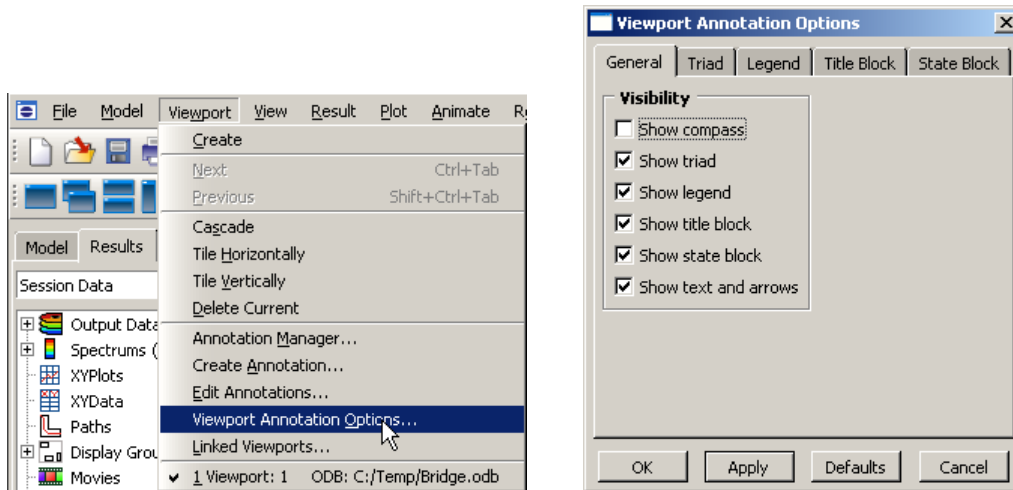
24. Display the deformed contour of the (Von) Mises stress
    a. In the toolbox area click on the "Plot Contours on Deformed Shape" icon
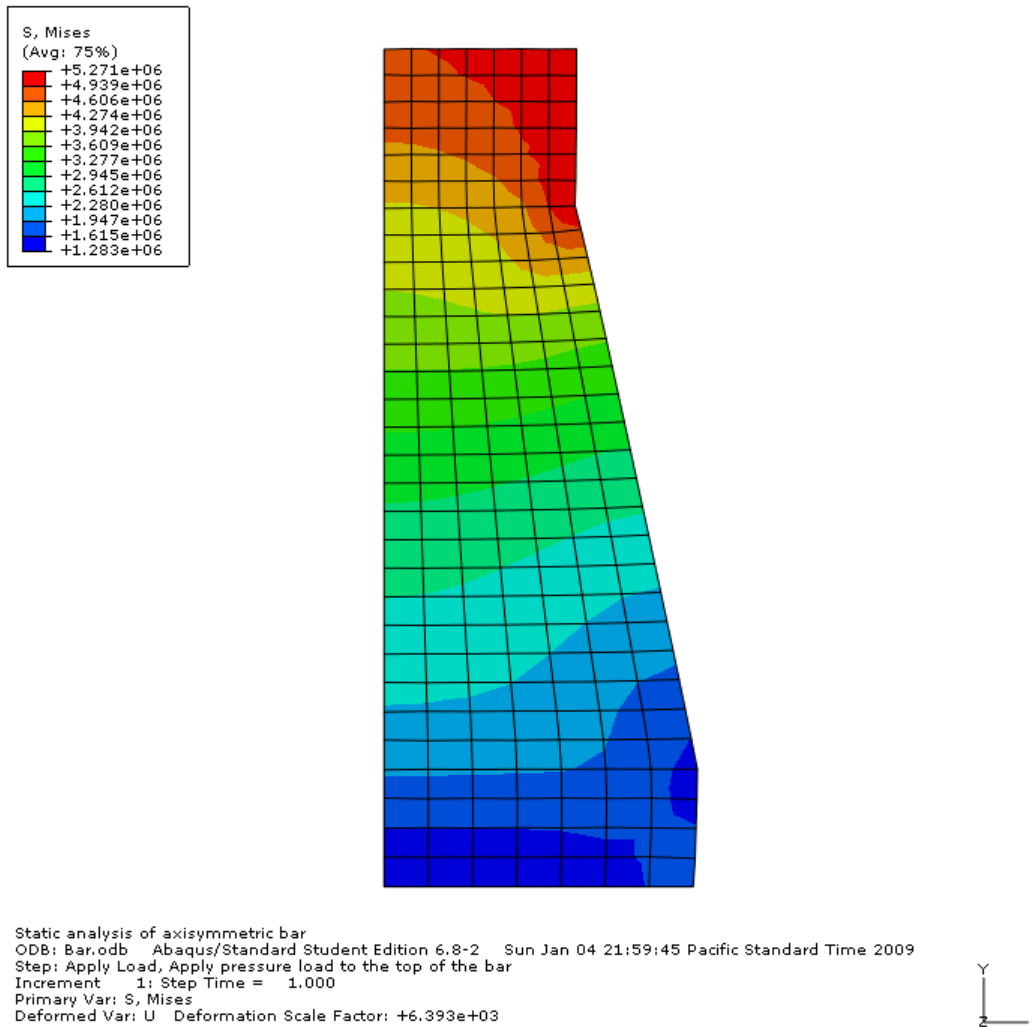
25. To determine the stress values, from the menu bar click Tools➔Query

   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 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
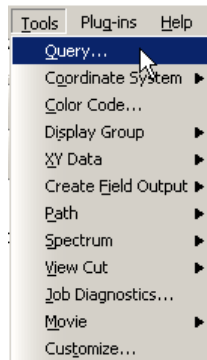
26. To change the output being displayed, in the menu bar click on Results➜Field Output
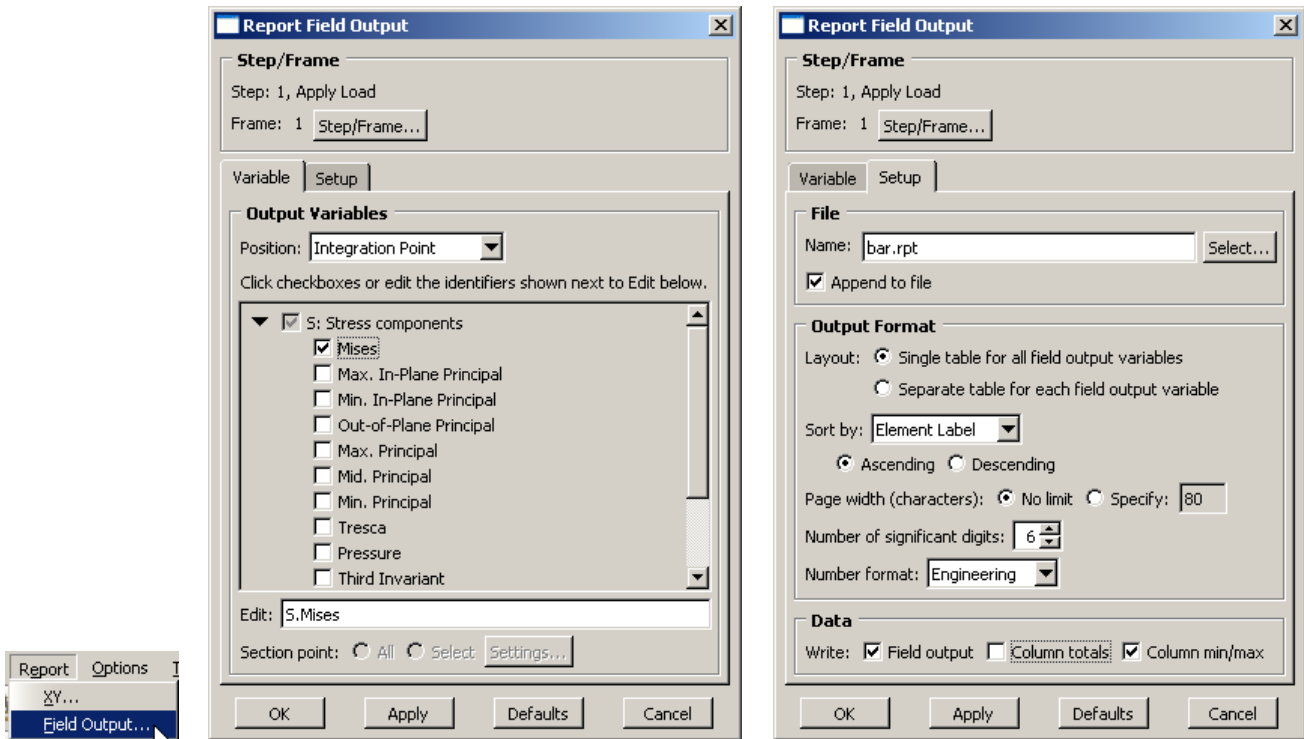
   a.   Select "Spatial displacement at nodes"

       i.   Invariant = Magnitude
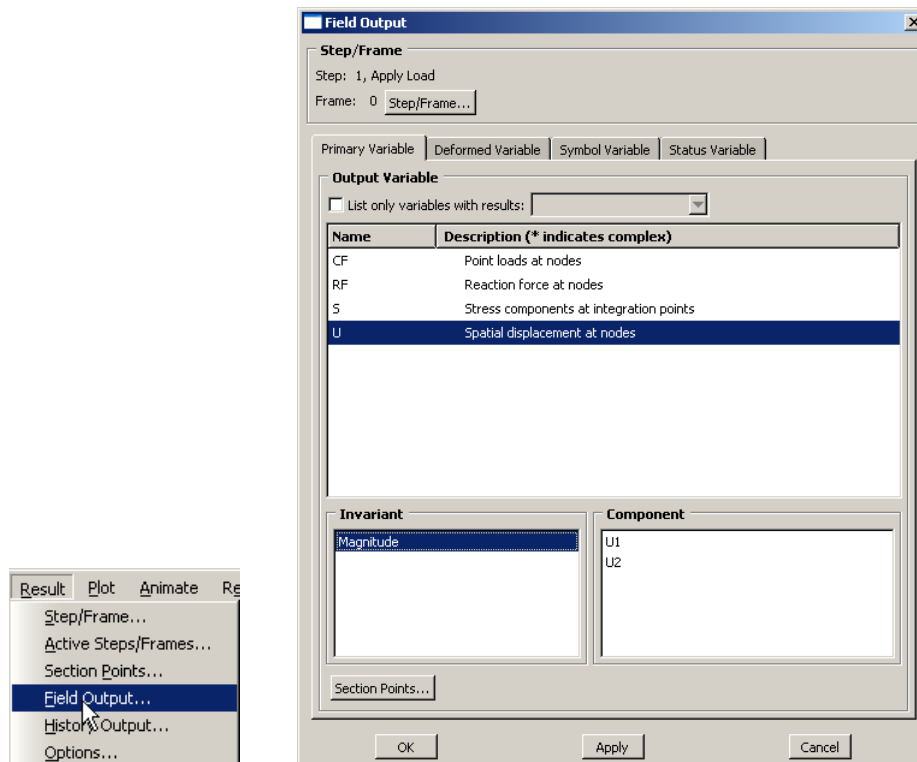


27. To create a text file containing the stresses and reaction forces (including total), in the menu bar click on Report➜Field Output

   a.   For the output variable select (Von) Mises

b.   On the Setup tab specify the name and the location for the text file
c.   Uncheck the "Column totals" option
d.   Click Apply

a.   Back on the Variable tab change the position to "Unique Nodal"
b.   Uncheck the stress variable, and select the RF1 reaction force
c.   On the Setup tab, check the "Column totals" option
d.   Click OK
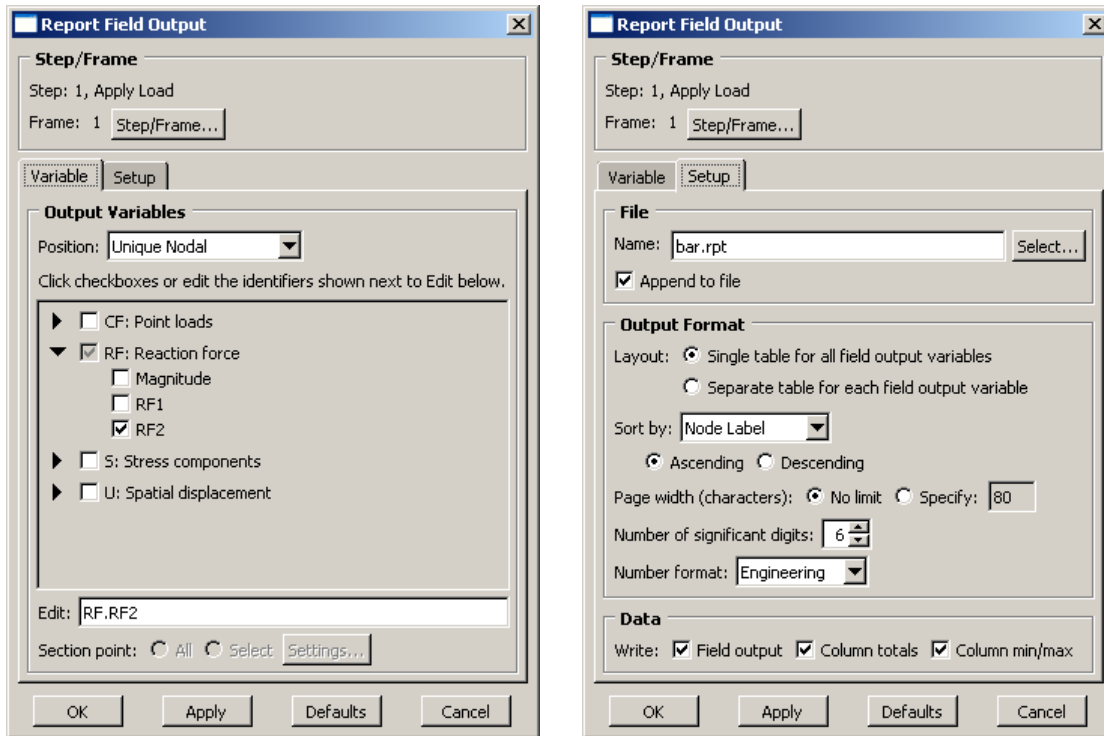
28. Open the .rpt file with any text editor
    a. One thing to check is that the total reaction force is equal to the applied load.