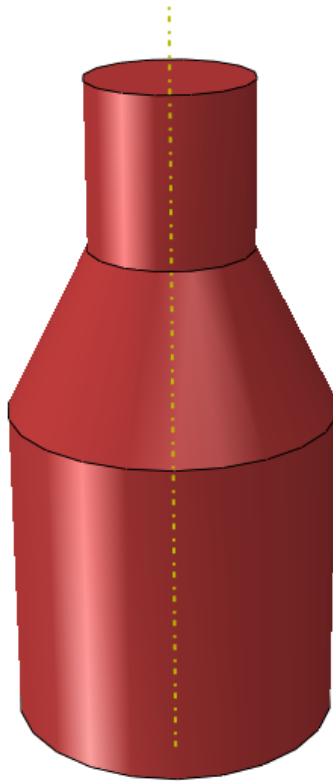# Abaqus/CAE Axisymmetric Tutorial (Version 2016)

Problem Description

A round bar with tapered diameter has a total load of 1000 N applied to its top face. The bottom of the bar is completely fixed. Determine maximum von Mises stress developed in the bar resulting from the load.
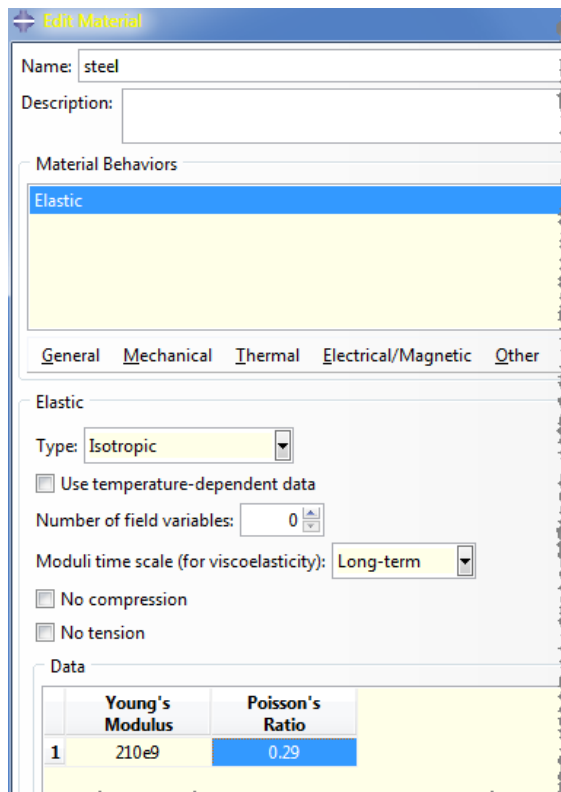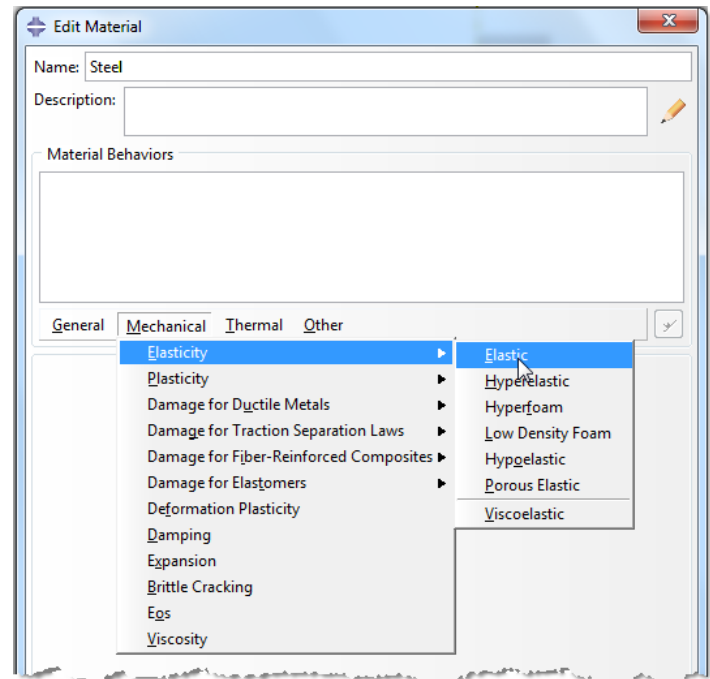
Analysis Steps

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 (part dimensions are in meters)
4. Create the geometry shown below (not discussed here).
5. Note axisymmetric parts must be drawn about the marked axis of rotation, and in the positive quadrant. Abaqus may require a "construction line" to be drawn on the vertical axis.
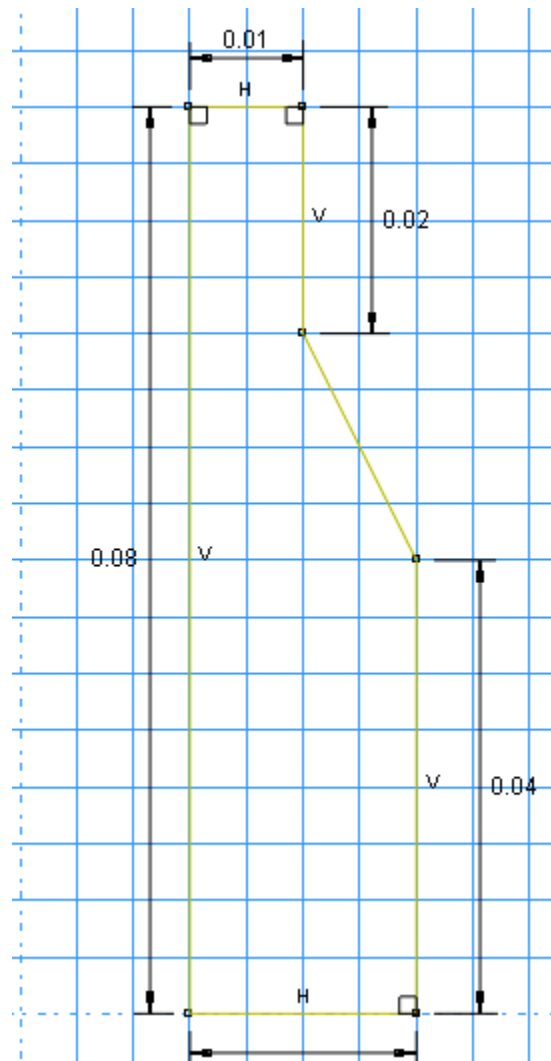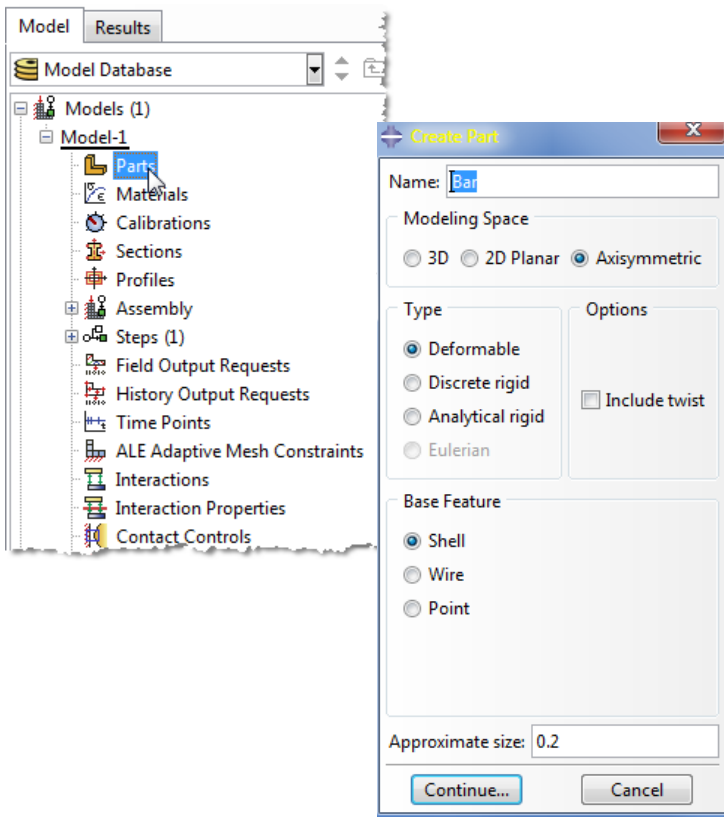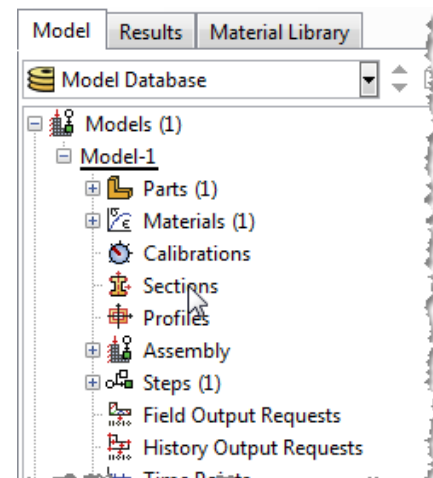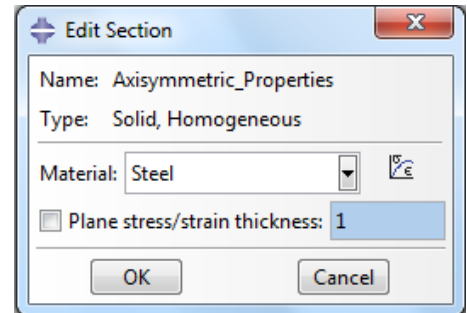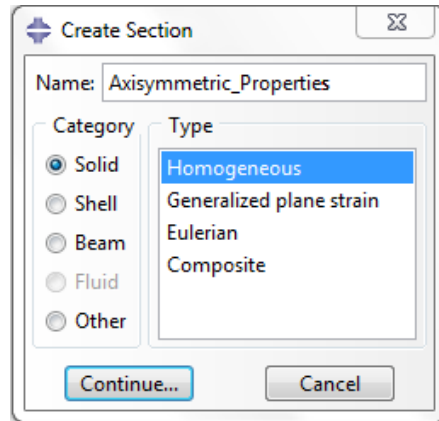
6. Double click on the "Materials" node in the model tree

   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**

Edit Material

Name: Steel

Description:

Material Behaviors

General | Mechanical | Thermal | Other

Elasticity ▶ | Elastic
Plasticity ▶ | Hyperelastic
Damage for Ductile Metals ▶ | Hyperfoam
Damage for Traction Separation Laws ▶ | Low Density Foam
Damage for Fiber-Reinforced Composites ▶ | Hypoelastic
Damage for Elastomers ▶ | Porous Elastic
Deformation Plasticity | Viscoelastic
Damping
Expansion
Brittle Cracking
Eos
Viscosity

Edit Material

Name: steel

Description:

Material Behaviors

Elastic

General | Mechanical | Thermal | Electrical/Magnetic | Other

Elastic

Type: Isotropic

☐ Use temperature-dependent data

Number of field variables: 0

Moduli time scale (for viscoelasticity): Long-term

☐ No compression
☐ No tension

Data

	Young's Modulus	Poisson's Ratio
1	210e9	0.29

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

Model | Results | Material Library

Model Database

Models (1)
  Model-1
    ⊞ Parts (1)
    ⊞ Materials (1)
    Calibrations
    Sections
    Profiles
    ⊞ Assembly
    ⊞ Steps (1)
    Field Output Requests
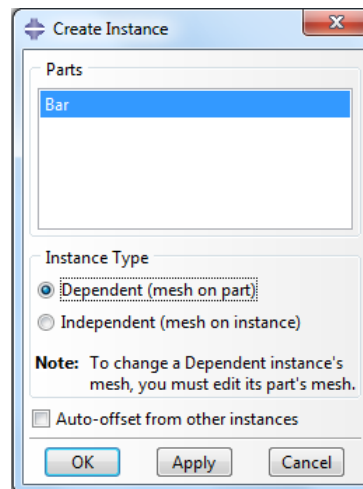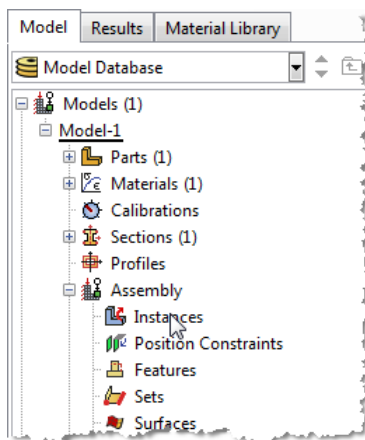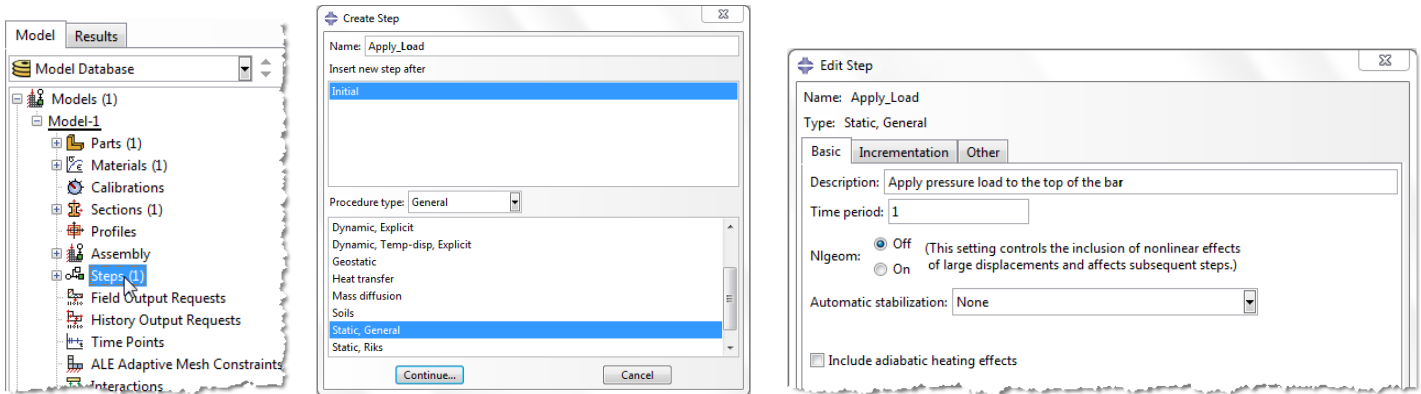   History Output Requests
    Time Points

8. 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 (Axisymmetric_Properties)
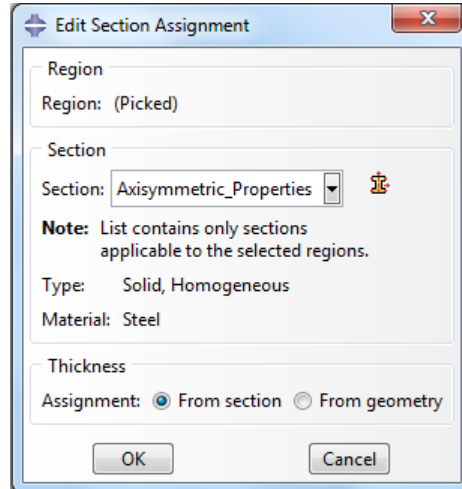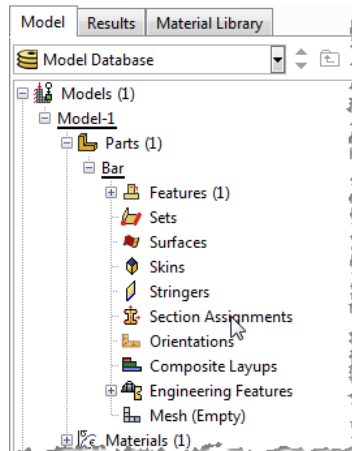
9. Expand the "Assembly" node in the model tree and then double click on "Instances"
   a. Select "Dependent" for the instance type
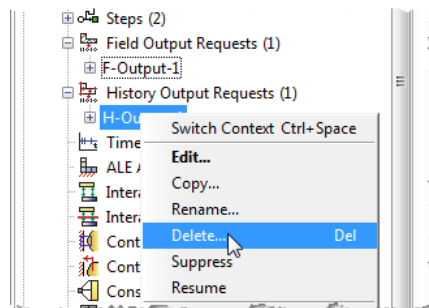
10. Double click on the "Steps" node in the model tree
    a. Name the step, set the procedure to "General", and select "Static, General"
   b. Give the step a description

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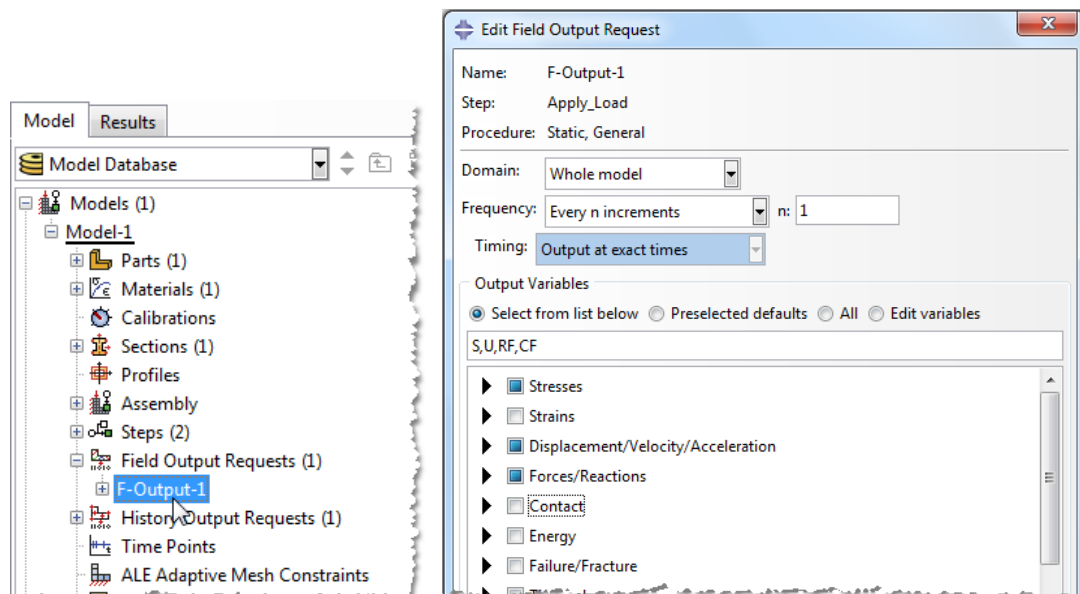
12. Expand the History Output Requests node in the model tree, and then right click on H-Output-1 (H-Output-1 was automatically generated when creating the step) and select Delete

13. Double click on the "BCs" node in the model tree
   a. Name the boundary conditioned "Fixed" and select "Displacement/Rotation" for the type
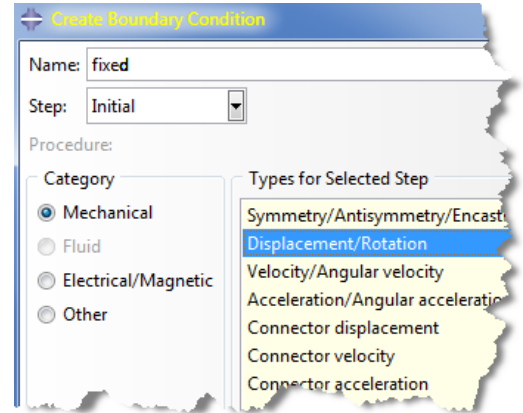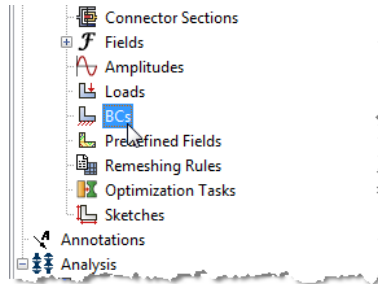
    b. Select the bottom edge of the geometry
    c. Set U1 and U2 to zero (full restraint)

    d. Select the vertical left edge for symmetry BC.
    e. Set U1 to zero to restrain the radial displacement.
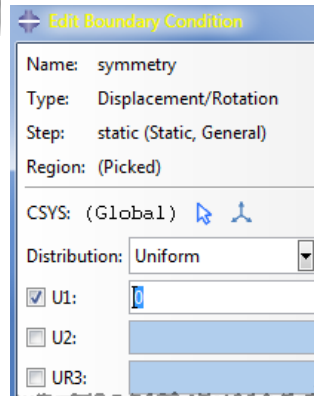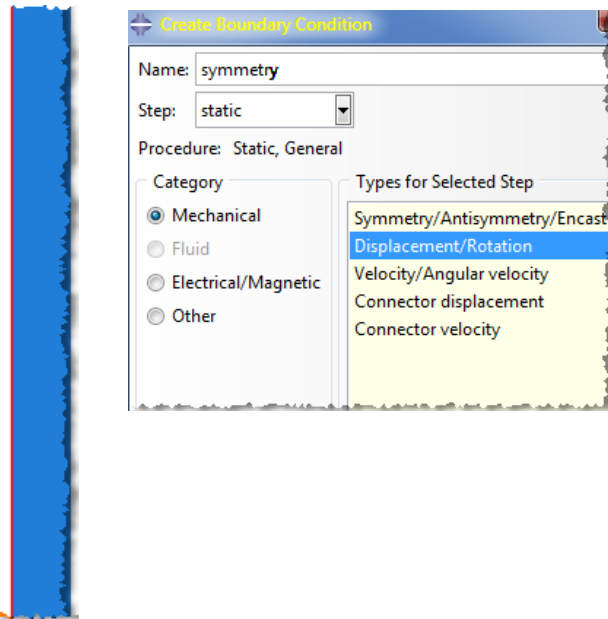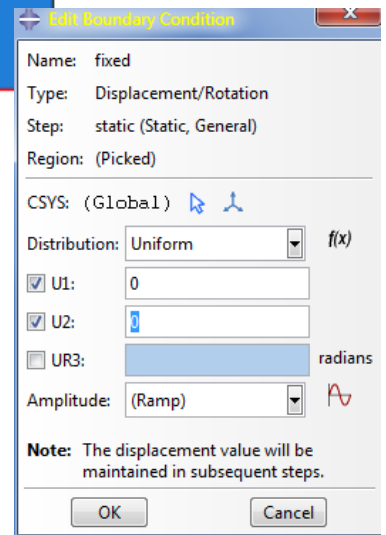
14. Double click on the "Loads" node in the model tree

   a. Name the load "Pressure" and select "Pressure" as the type
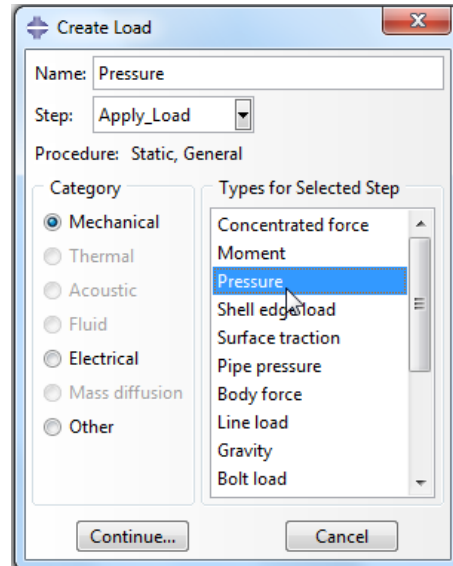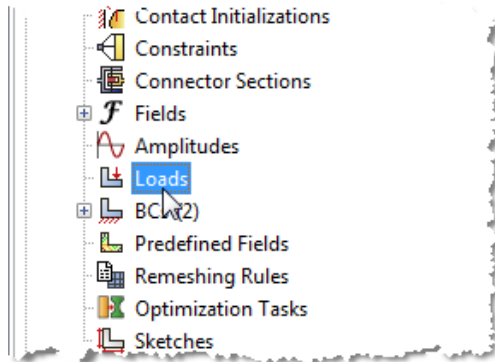
    b. Select surface named "Pressure"

   c. For the magnitude enter the applied pressure in $F/(\pi * r^2)$ or $\left[\dfrac{1000N}{\pi*(0.01m)^2}\right]$
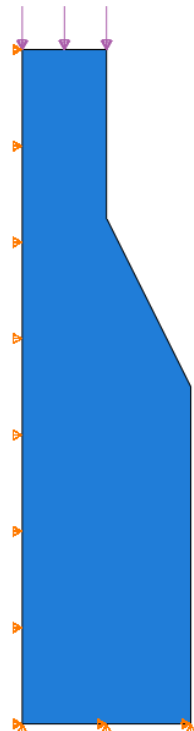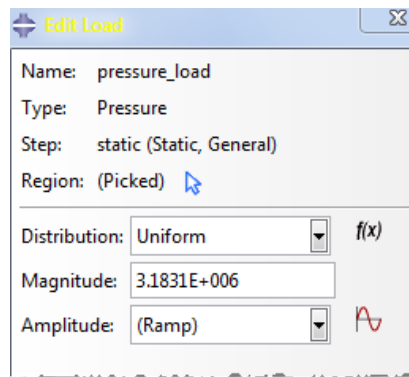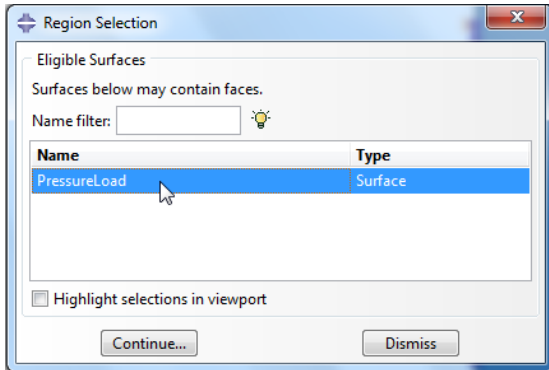
    Assembly with load and all boundary conditions is displayed to the right.

15. In the icon panel switch the Module to "Mesh" and switch the" Object:" from Assembly to 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. Uncheck "Reduced integration" and note that the name of the element (CAX4) and its description are given below the element controls
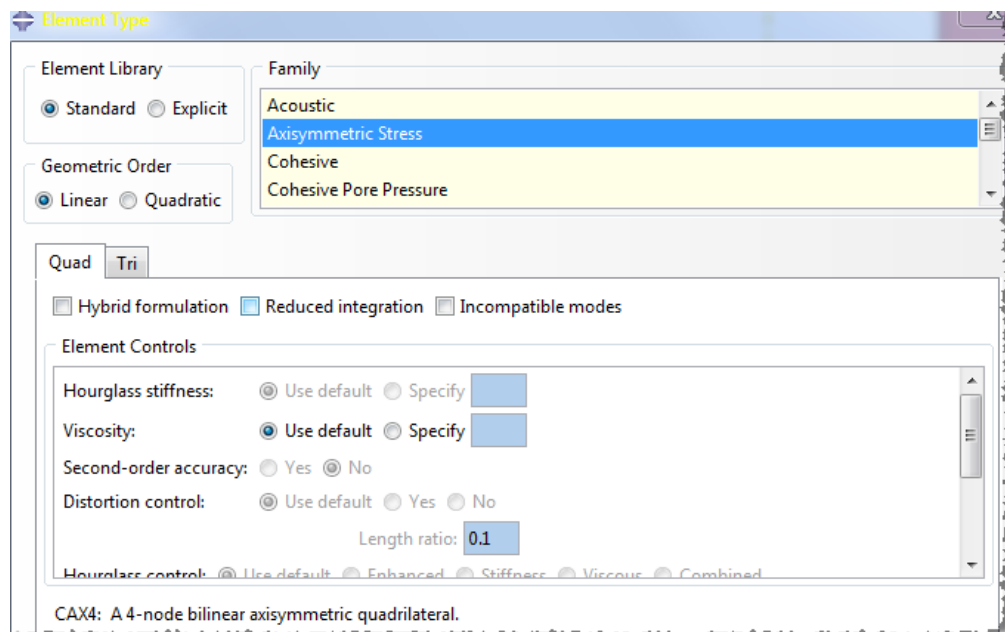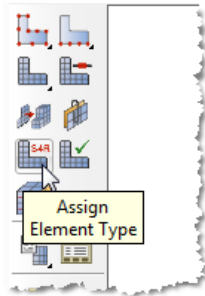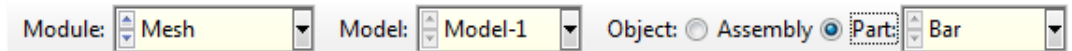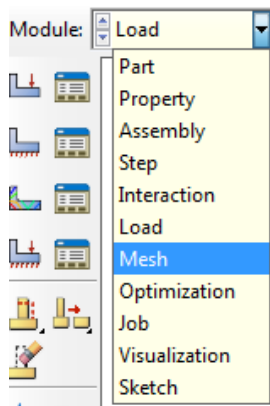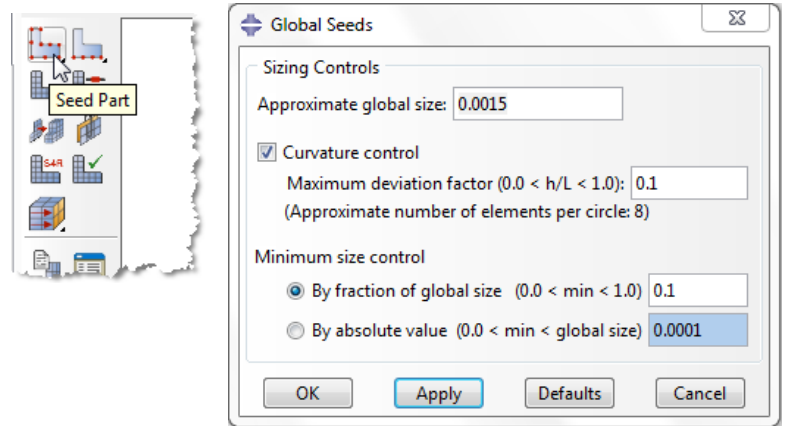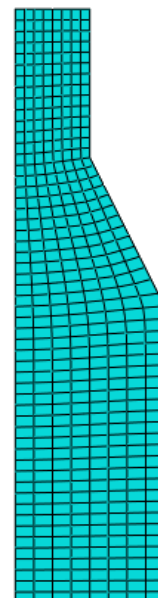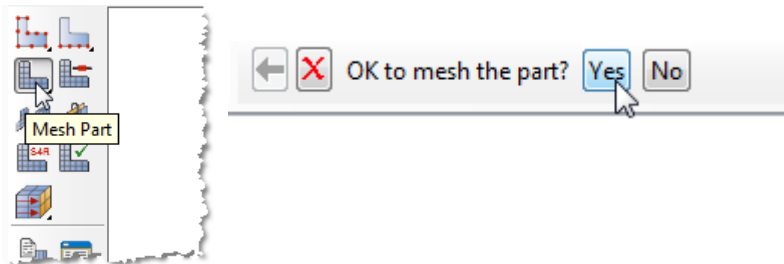
16. In the toolbox area click on the "Assign Mesh Controls" icon
    a. Change the element shape to "Quad"
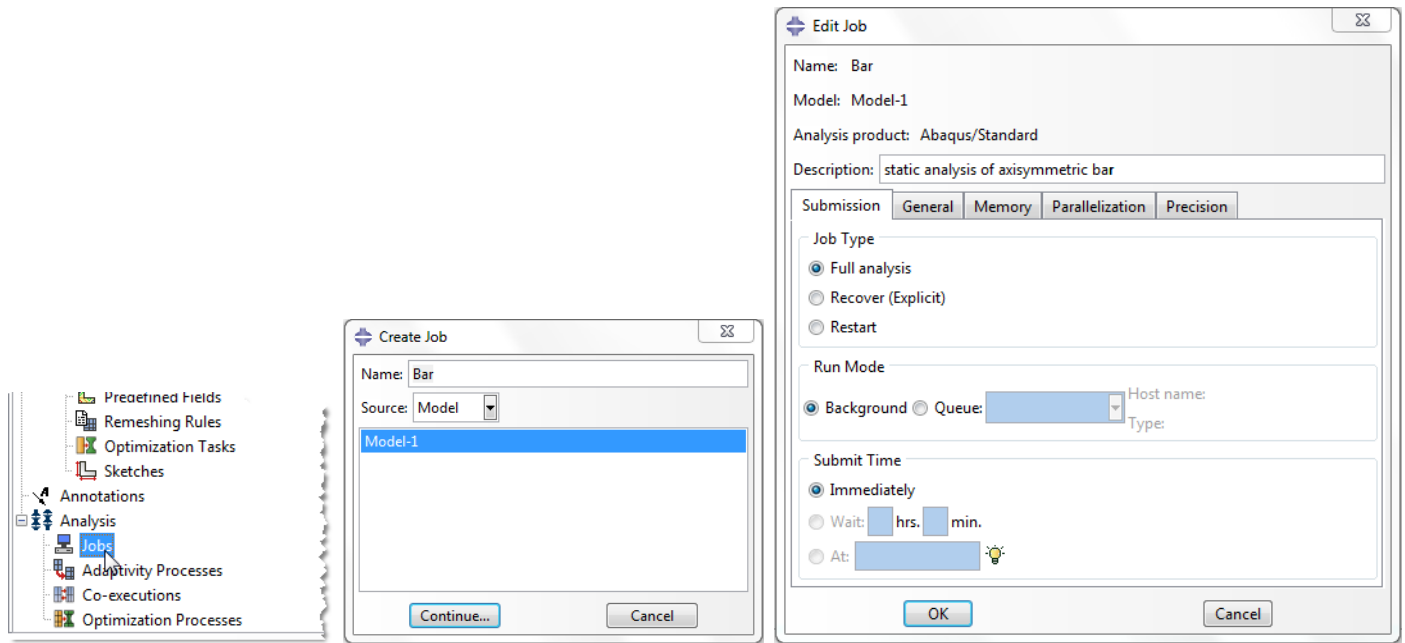   b. Change the Algorithm to "Medial axis" for a more structured mesh
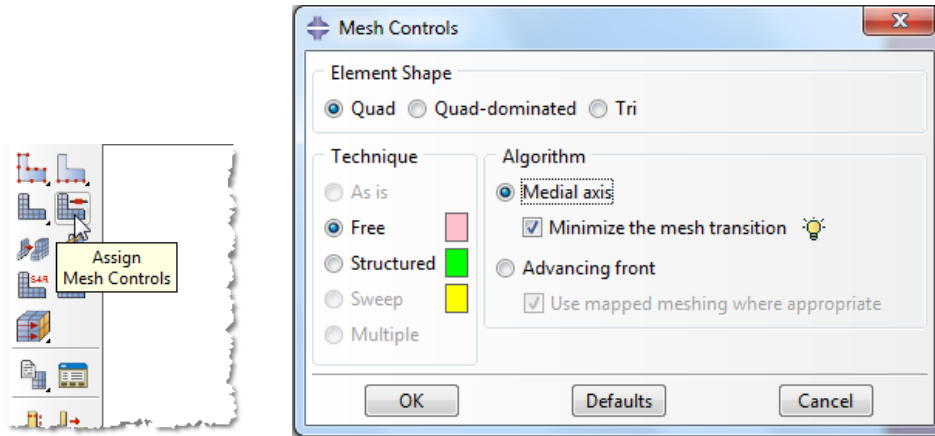
17. In the toolbox area click on the "Seed Part" icon
   a. Set the approximate global size to 0.0015

18. In the toolbox area click on the "Mesh Part" icon. A mesh of about 432 elements should be generated.

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

20. 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"
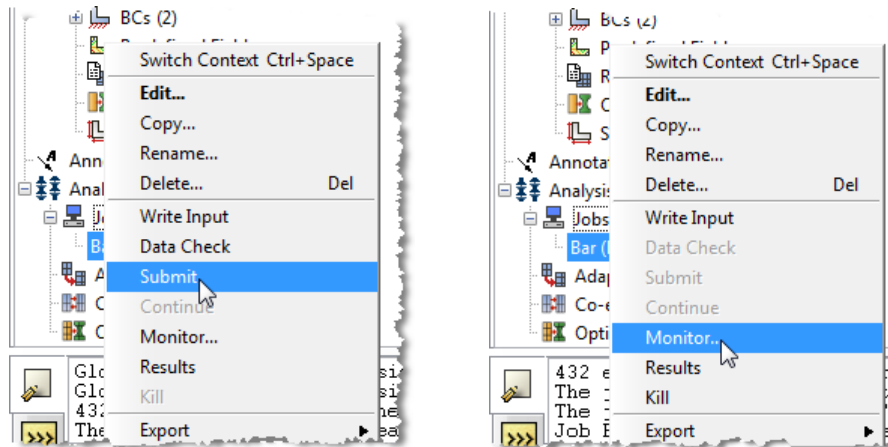

    b. In the Monitor window check that there are no errors or warnings
        i. If there are errors, investigate the cause(s) before resolving
        ii. If there are warnings, determine if the warnings are relevant, some warnings can be safely ignored

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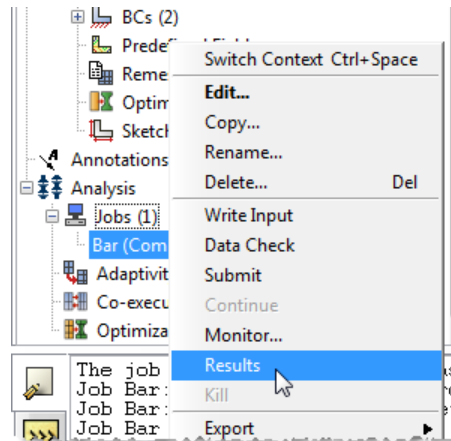
22. In the menu bar click on Viewport➜Viewport Annotations Options
    a. Uncheck the "Show compass option"
    b. The locations of viewport items can be specified on the corresponding tab in the Viewport Annotations Options
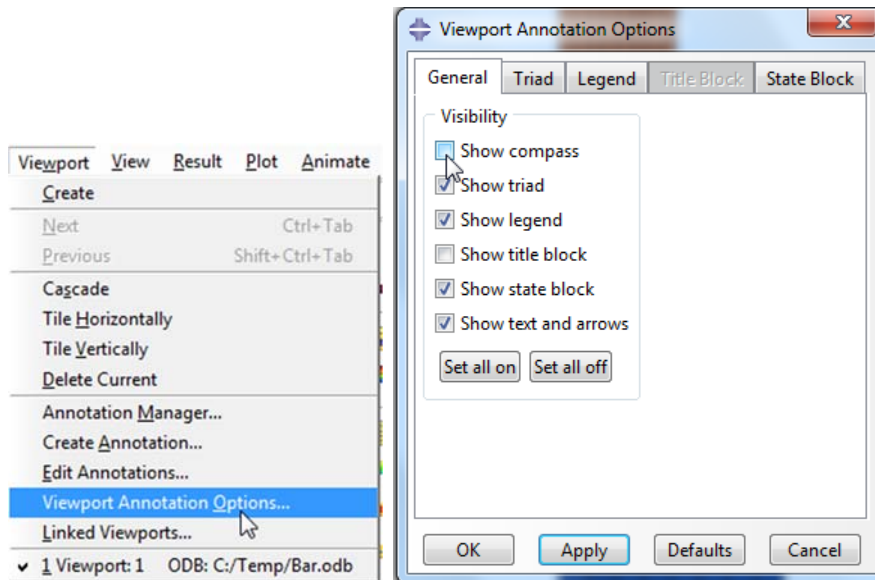
23. Display the deformed contour of the (Von) Mises stress
    a. In the toolbox area click on the "Plot Contours on Undeformed Shape" icon

24. To determine the stress values, click on the "Probe values" icon" from the icon panel.
    a. Check the boxes labeled "Nodes" and "S, Mises"
    b. In the viewport mouse over the node of interest
    c. Note that Abaqus reports stresses based on the "averaging" of the stress from the surrounding elements. You can also request stress values for the "element" which reports, by default, multiple stress values corresponding to the integration points.
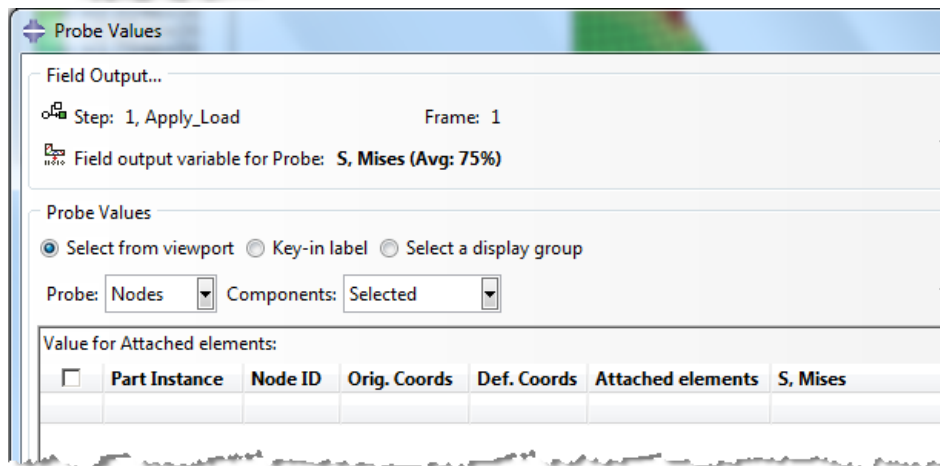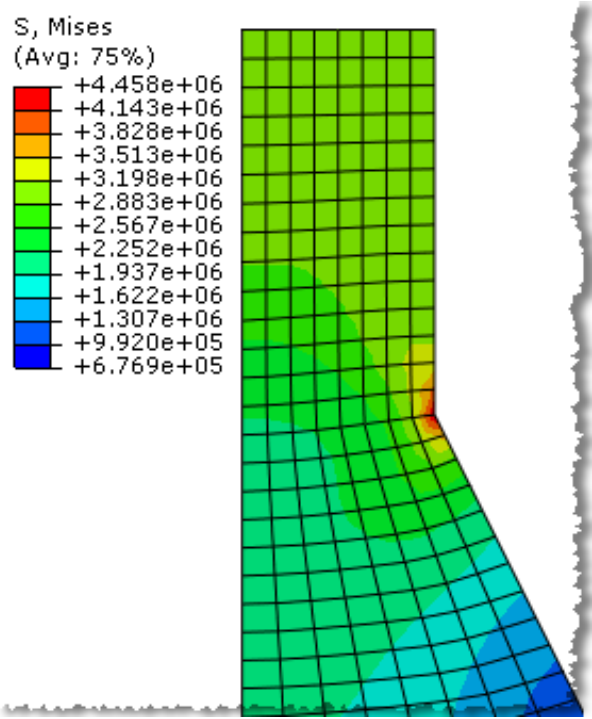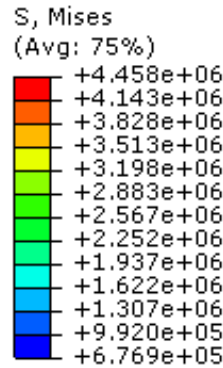
One final note: If you were to select "Reduced integration" element, stress values should be considerably lower, indicating the overstiff nature of the model. This situation can be remedied by a much more refined mesh!