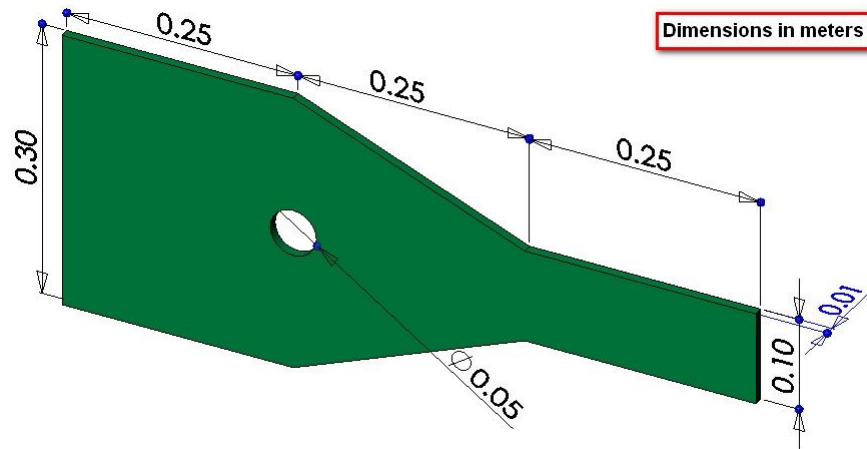# Abaqus Plane Stress Tutorial (ver. 6.12) using CST element

## Problem Description



The steel bracket is fixed to a wall along its left side. A tensile pressure force with a total magnitude of 5000 N is applied to the right side of the bracket. The bracket contains one line of symmetry, so only half of the geometry is to be modeled. Determine the stresses in the bracket.

## 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)
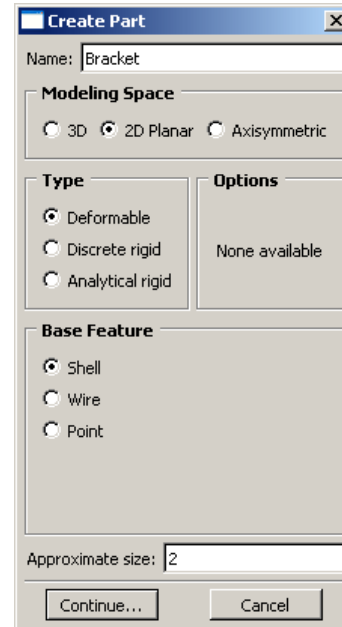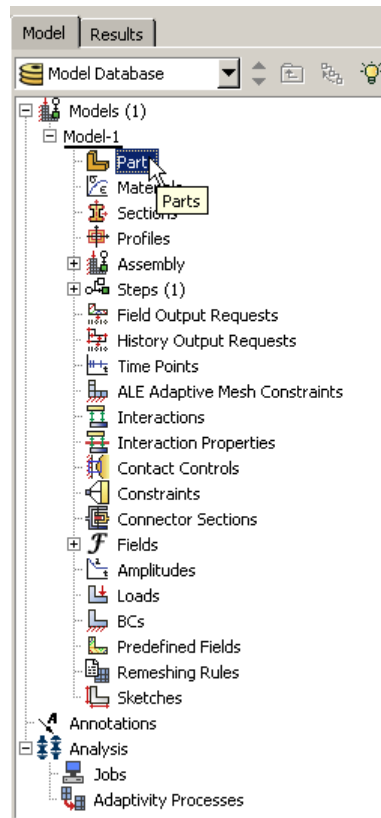
3.  In the Create Part dialog box (shown above) name the part and select
    a.  2D Planar
    b.  Deformable
    c.  Shell
    d.  Approximate size = 2
4.  Create the geometry shown below (not discussed here)
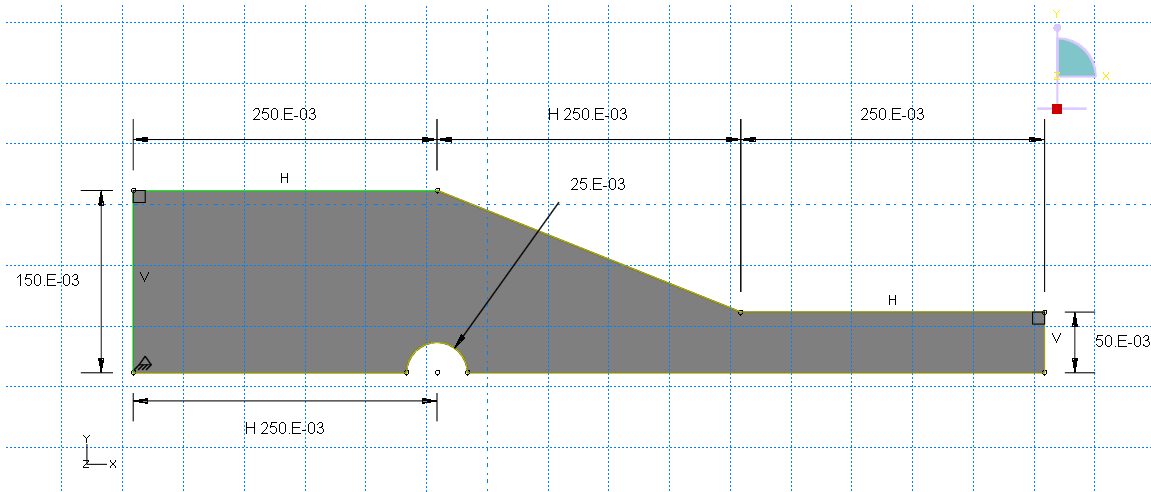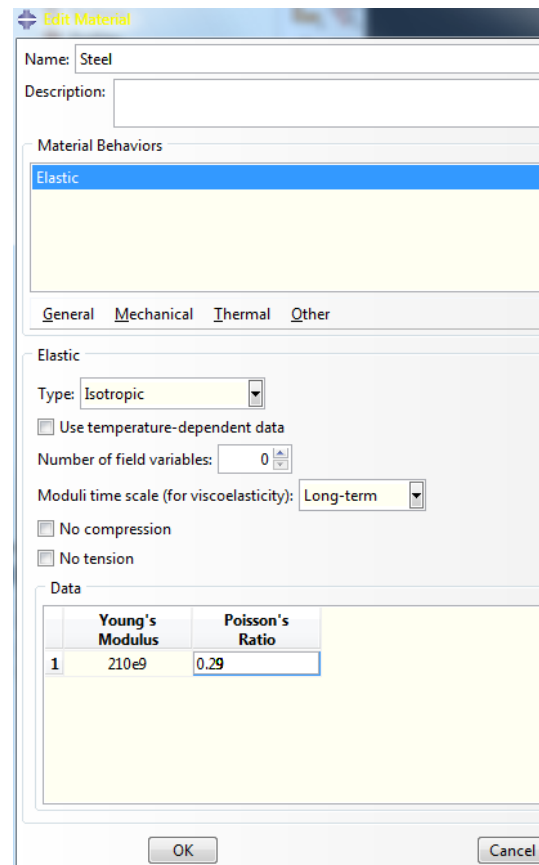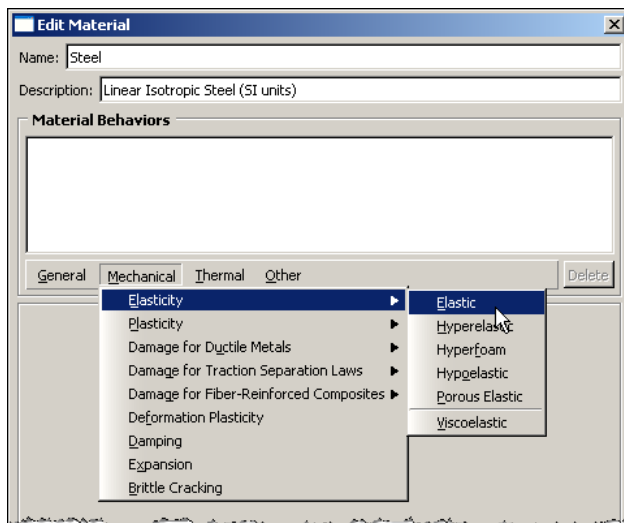
5. Double click on the "Materials" node in the model tree
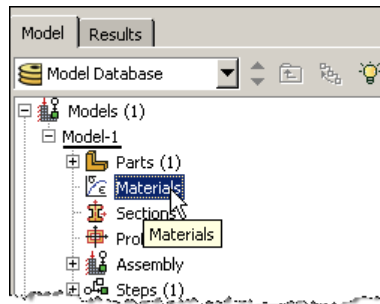
a. Name the new material and give it a description
b. Click on the "Mechanical" tab➜Elasticity➜Elastic
c. Define Young's Modulus and the Poisson's Ratio (use SI units)
   i. **WARNING: There are no predefined system of units within Abaqus, so the user is responsible for ensuring that the correct values are specified**
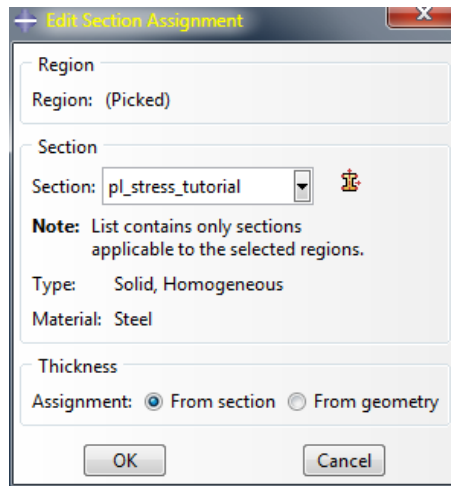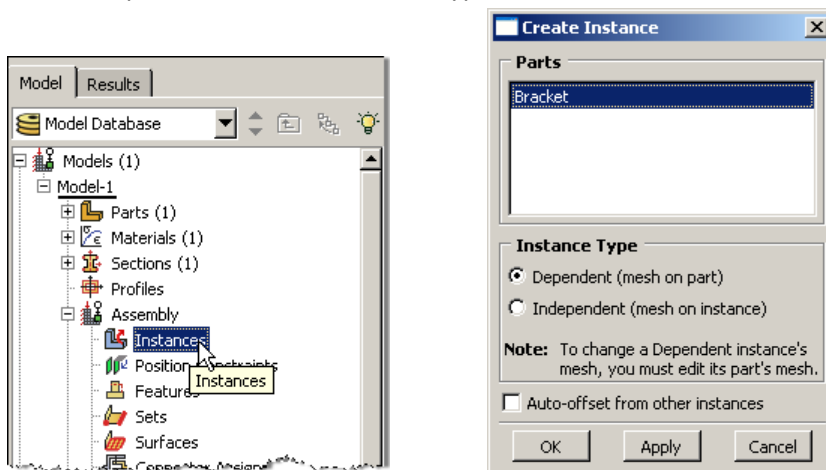
6. Double click on the "Sections" node in the model tree
a. Name the section "PlaneStressProperties" and select "Solid" for the category and "Homogeneous" for the type
b. Select the material created above (Steel) and set the thickness to 0.01
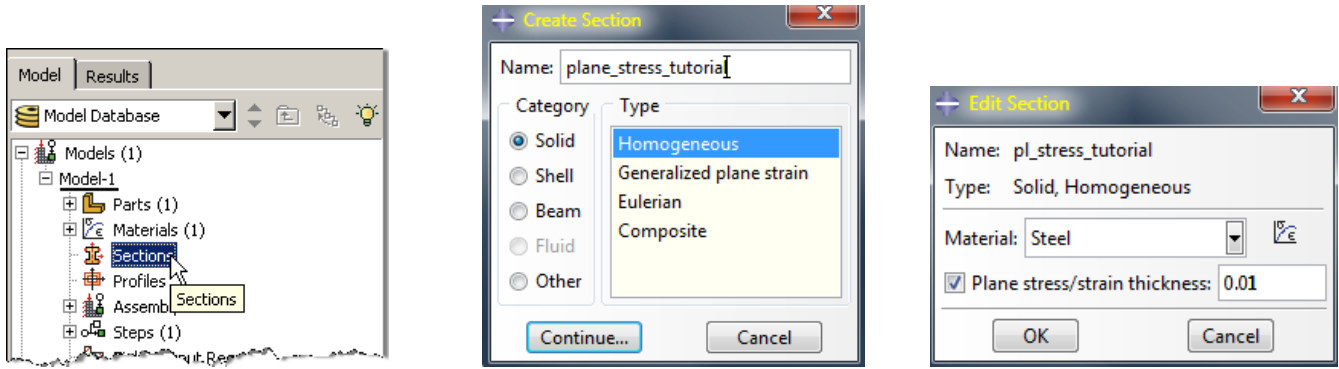
7. Click on the "Assign Section" icon
   a. Select the surface geometry in the viewport
   b. Be sure the section created above (plane_stress_tutorial) is selected.

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 left edge of the surface in the viewport
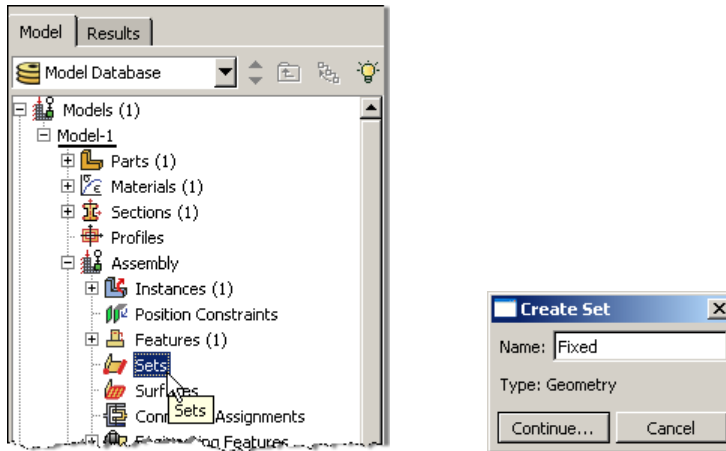
c.  Create another set named "Symmetry"
d.  Select the two lower horizontal edges of the surface in the viewport

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

a.  Name the surface "PressureLoad"
b.  Select the right edge of the surface in the viewport
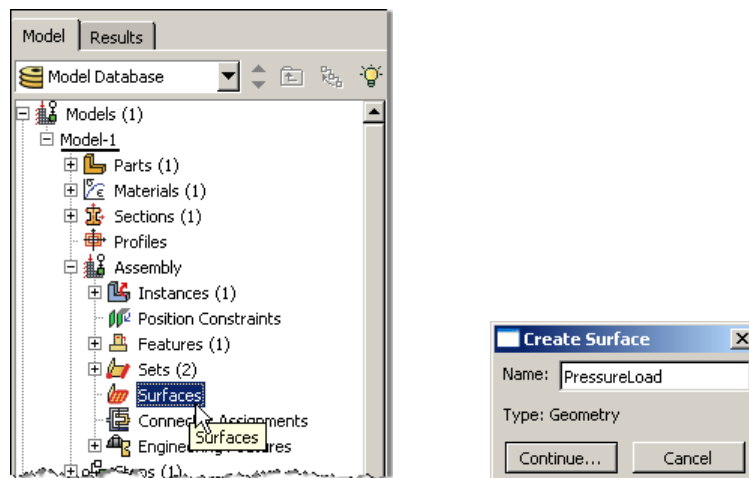


11. Double click on the "Steps" node in the model tree

a.  Name the step, set the procedure to "General", and select "Static, General"
b.  Give the step a description
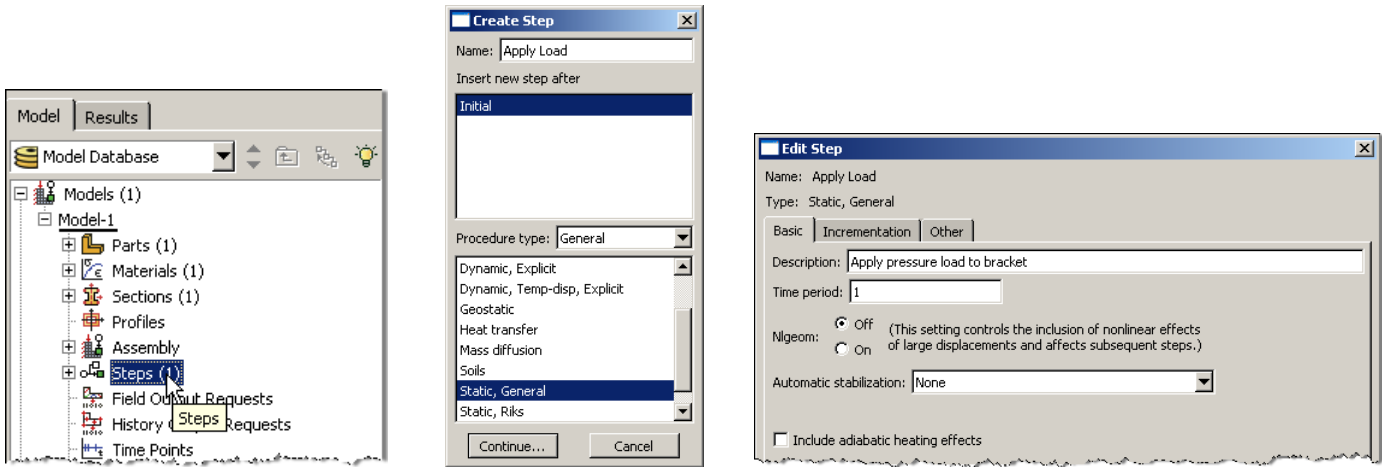
12. Expand the Field Output Requests node in the model tree, and then double click on F-Output-1 (F-Output-1 was automatically generated when creating the step)
    a. Uncheck the variables "Strains" and "Contact"
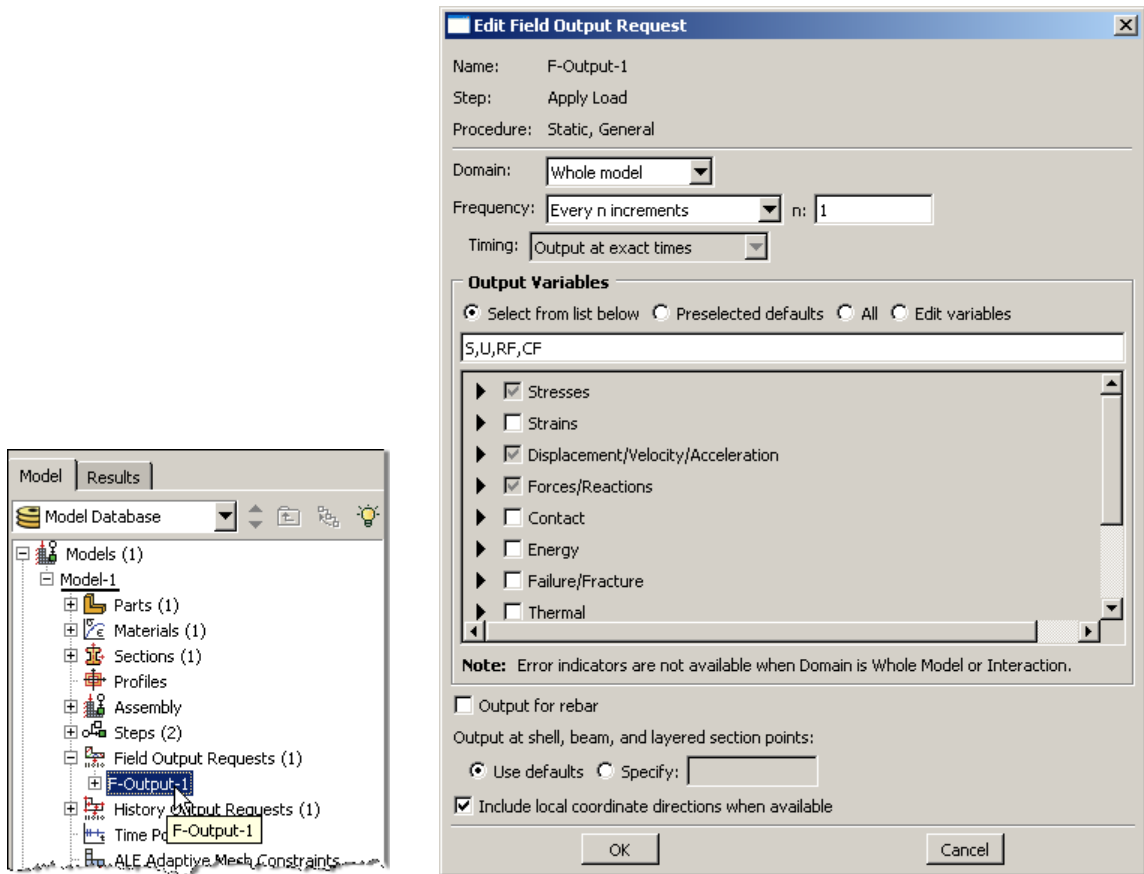


13. Expand the History Output Requests node in the model tree, and then right click on H-Output-1 (H-Output-1 was automatically generated when creating the step) and select Delete
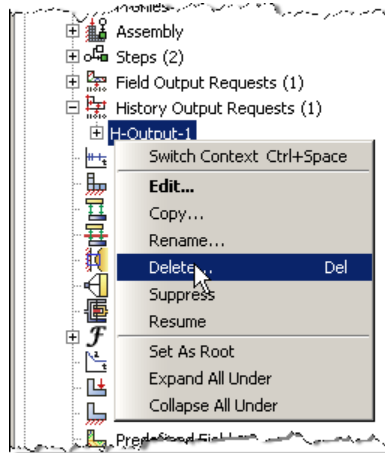
14. Double click on the "BCs" node in the model tree
    a.  Name the boundary conditioned "Fixed" and select "Displacement/Rotation" for the type
    b.  In the prompt area click on the Sets button
    c.  Select the set named "Fixed.
    d.  Check U1 and U2 to fully restrain the left edge..
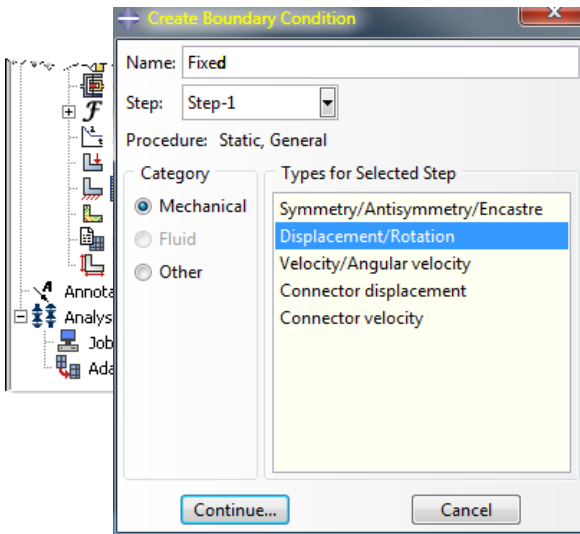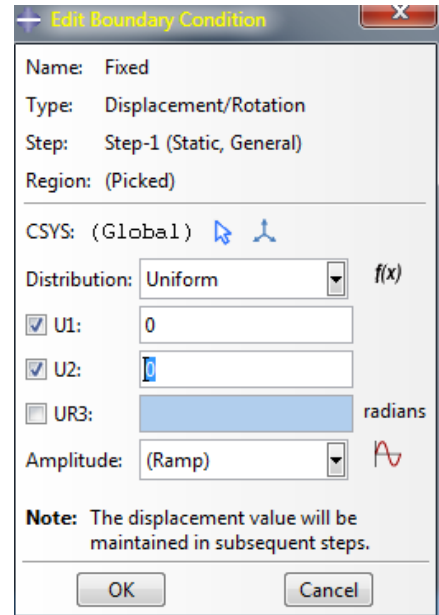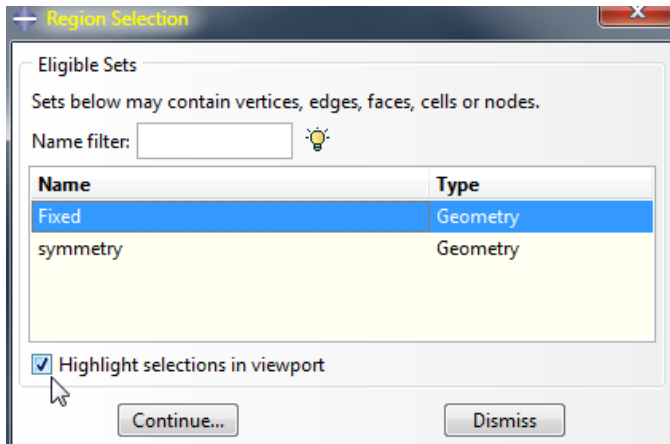
e.   Repeat the procedure for the symmetry restraint using the set named "Symmetry", check U2 for the boundary condition.
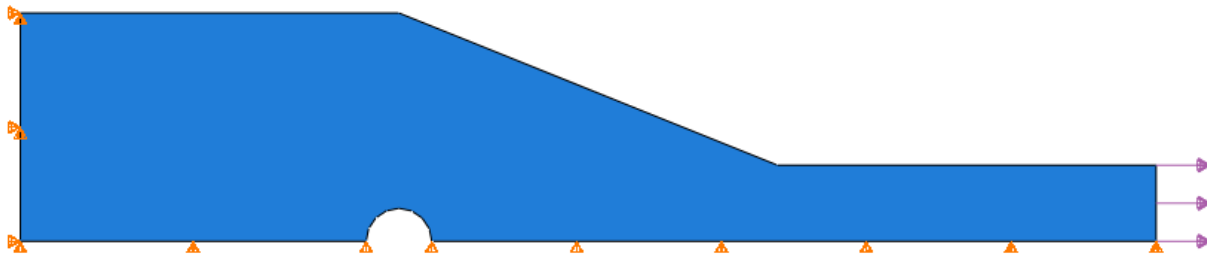
15. Double click on the "Loads" node in the model tree
    a.   Name the load "Pressure" and select "Pressure" as the type

    b.   Select surface named "Pressure"
    c.   For the magnitude enter -5e6
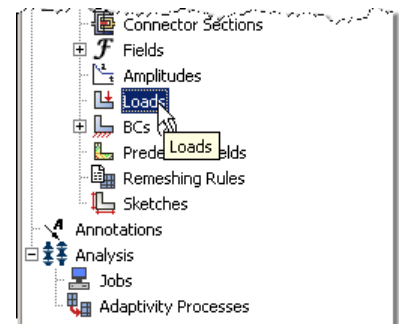
i.  Note that because we have been using standard SI units the load applied is -5x10$^6$ N/m$^2$ which is

a total of -2500 N distributed across the right edge of the surface $\left(\frac{-2500N}{(0.05m)(0.01m)}\right)$
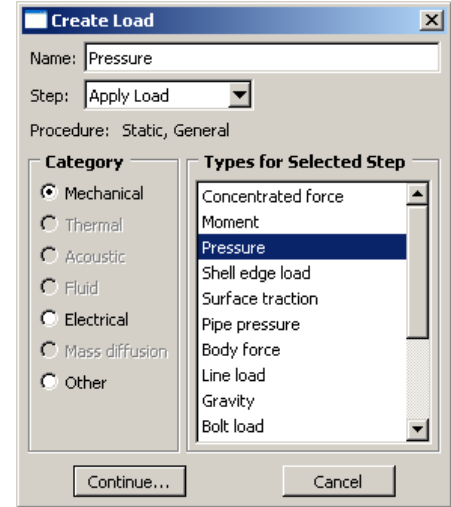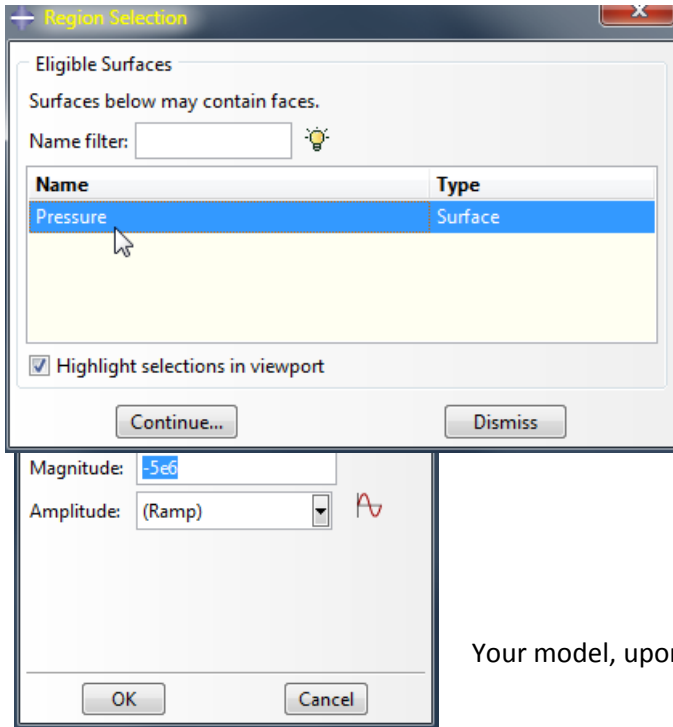
Your model, upon application of BC and load should look similar to the figure below.
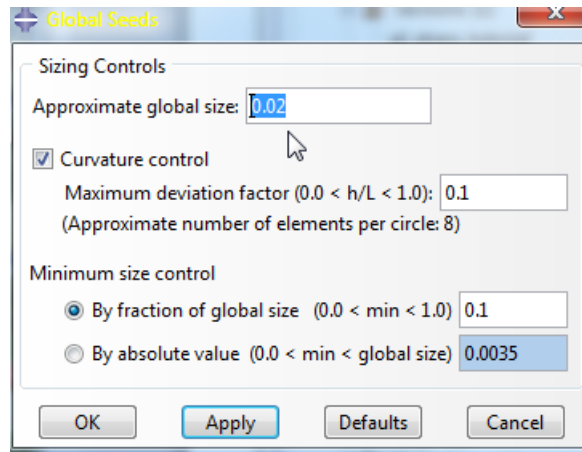
16. In the model tree double click on "Mesh" for the Bracket part, and in the toolbox area click on the "Assign Element Type" icon
    a. Select "Standard" for element type
    b. Select "Linear" for geometric order
    c. Select "Plane Stress" for family
    d. Pick the tab "Tri" and note the name of the CST element "CPS3" in the description window!
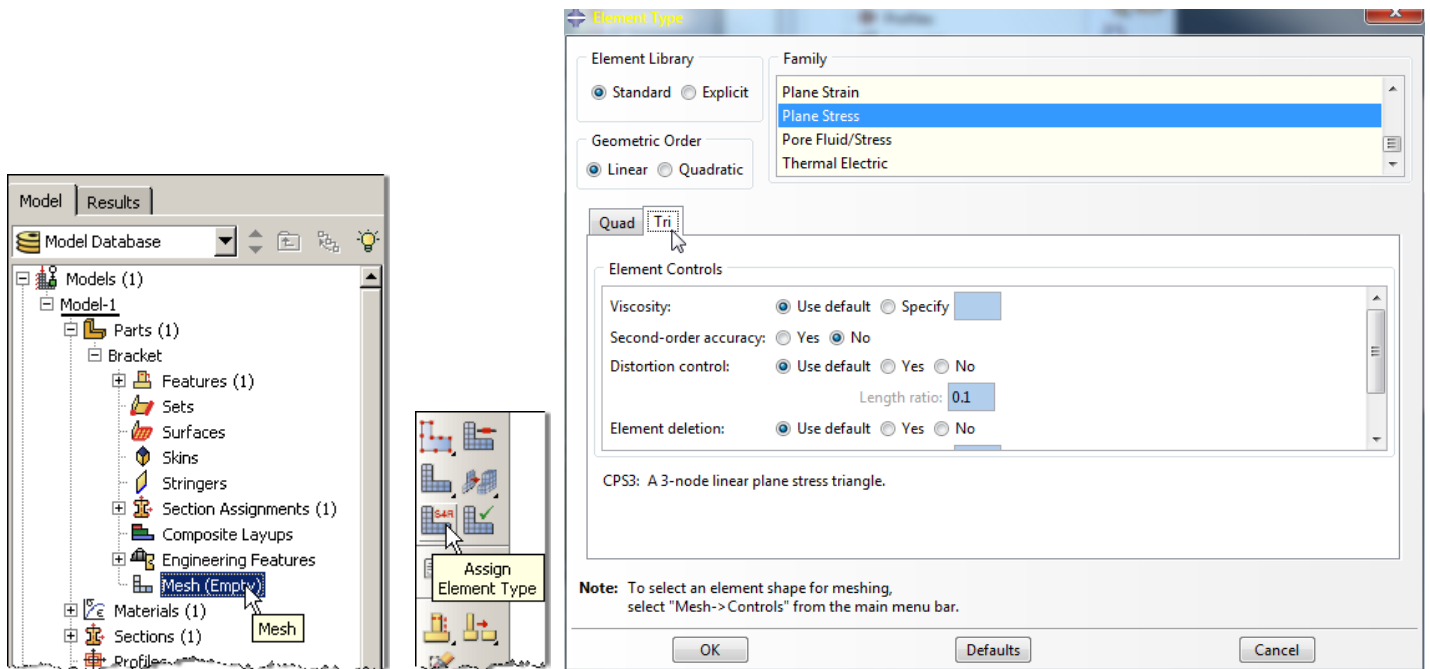
17. In the toolbox area click on the "Assign Mesh Controls" icon
    a. Modify the element shape "Tri" and click OK.
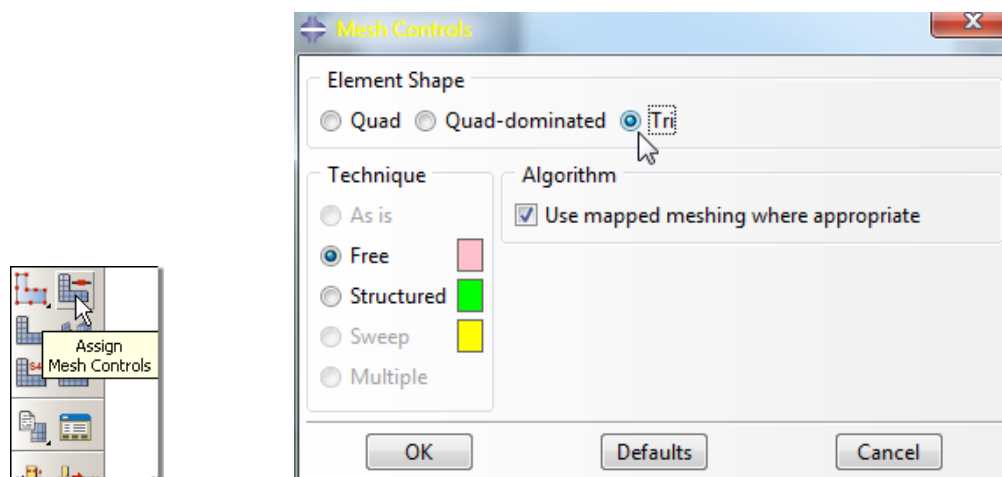
18. In the toolbox area click on the "Seed Part" icon
    a. Set the approximate global size to 0.02

**Global Seeds**

Sizing Controls

Approximate global size: 0.02

☑ Curvature control

Maximum deviation factor (0.0 < h/L < 1.0): 0.1
(Approximate number of elements per circle: 8)

Minimum size control

◉ By fraction of global size   (0.0 < min < 1.0) 0.1

◯ By absolute value  (0.0 < min < global size) 0.0035

OK    Apply    Defaults    Cancel
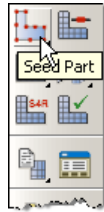
19. In the toolbox area click on the "Mesh Part" icon

20. In the model tree double click on the "Job" node
    a. Name the job "Bracket"
    b. Give the job a description
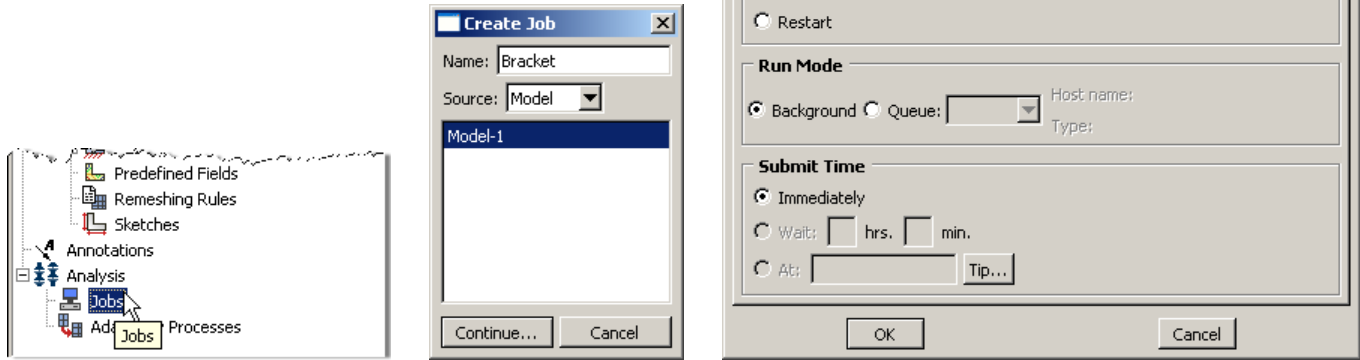
21. In the model tree right click on the job just created (Bracket) and select "Submit"
    a. While Abaqus is solving the problem right click on the job submitted (Bracket), and select "Monitor"
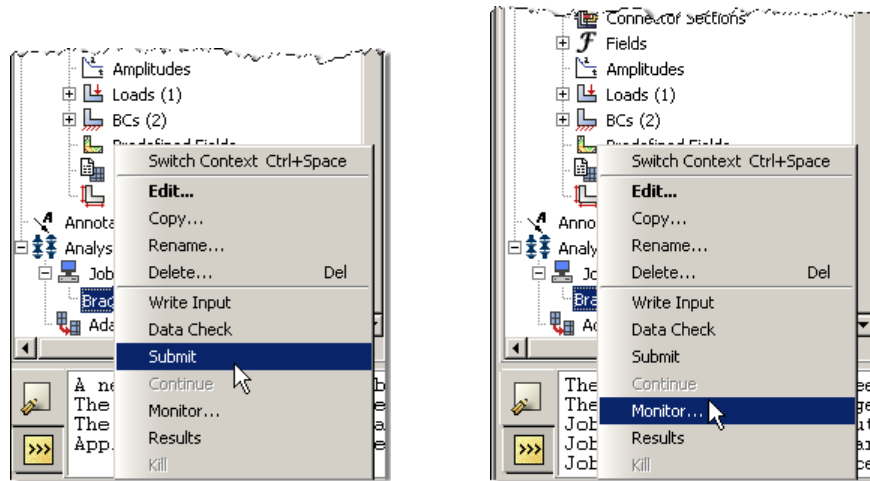


    b. In the Monitor window check that there are no errors or warnings
        i. If there are errors, investigate the cause(s) before resolving
        ii. If there are warnings, determine if the warnings are relevant, some warnings can be safely ignored
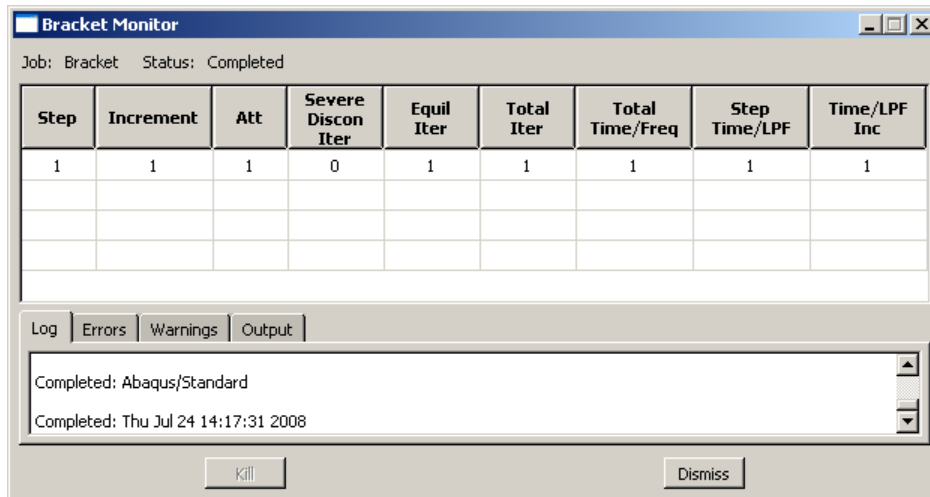
22. In the model tree right click on the submitted and successfully completed job (Bracket), and select "Results"
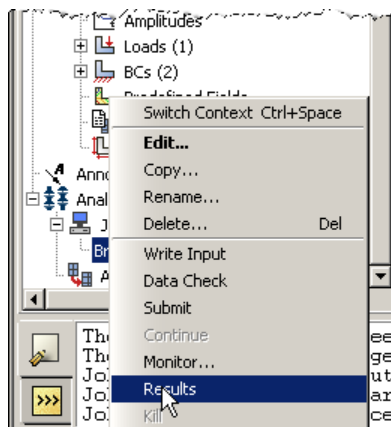


23. In the menu bar click on Viewport➔Viewport Annotations Options
    a. Uncheck the "Show compass option"
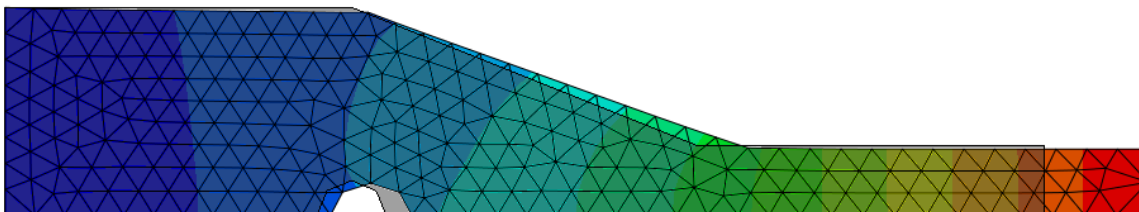    b. The locations of viewport items can be specified on the corresponding tab in the Viewport Annotations Options
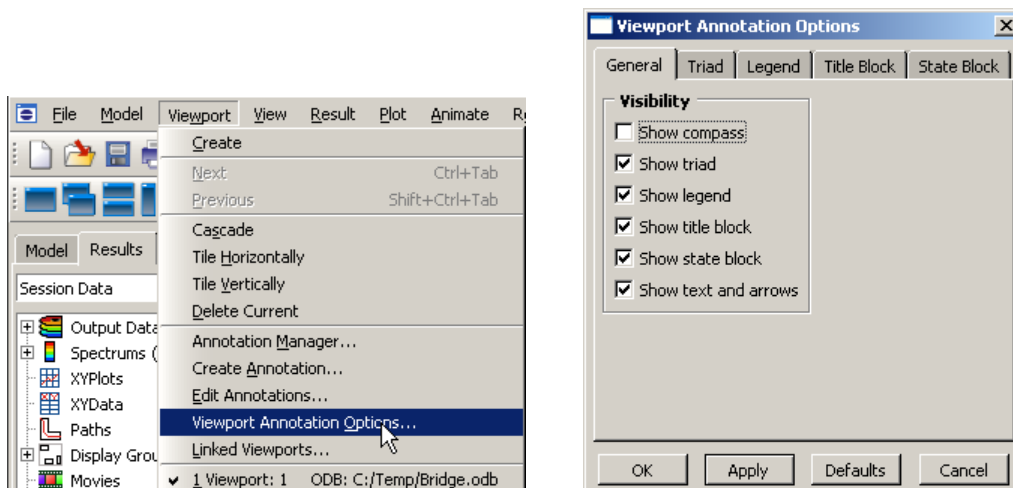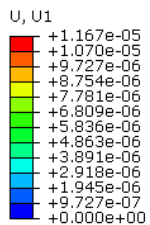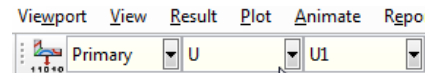
24. Display the deformed contour of the (Von) Mises stress
    a.  In the toolbox area click on the "Plot Contours on Deformed Shape" icon
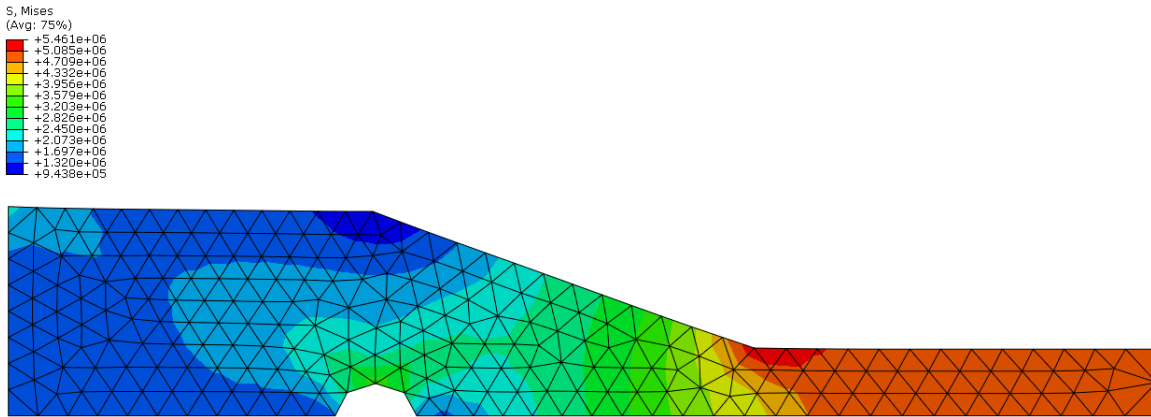


25. Request displacement plot along x-dir. With the undeformed shape superimposed.





The image was obtained using the "superimpose options" icon and setting the parameters as shown. Note the clear stretching of the right side of the bracket and the imposition of symmetry BC preventing the lower horizontal segement from moving in the vertical direction.
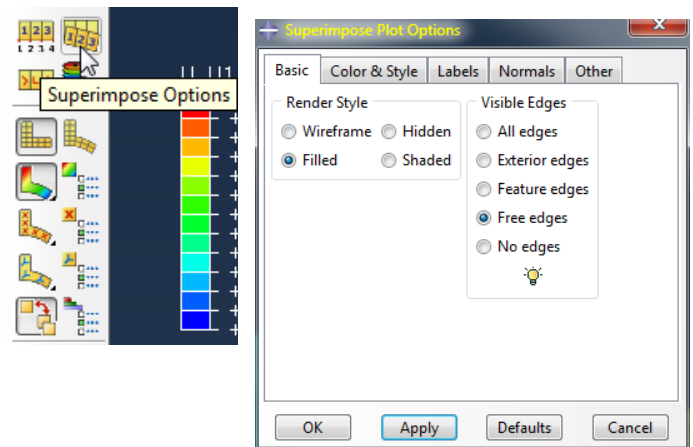
26. Activate the icon labeled "Results Options" shown on the right.
    a. Uncheck the box "Average element output at nodes."
    b. Change the stress output to S11.
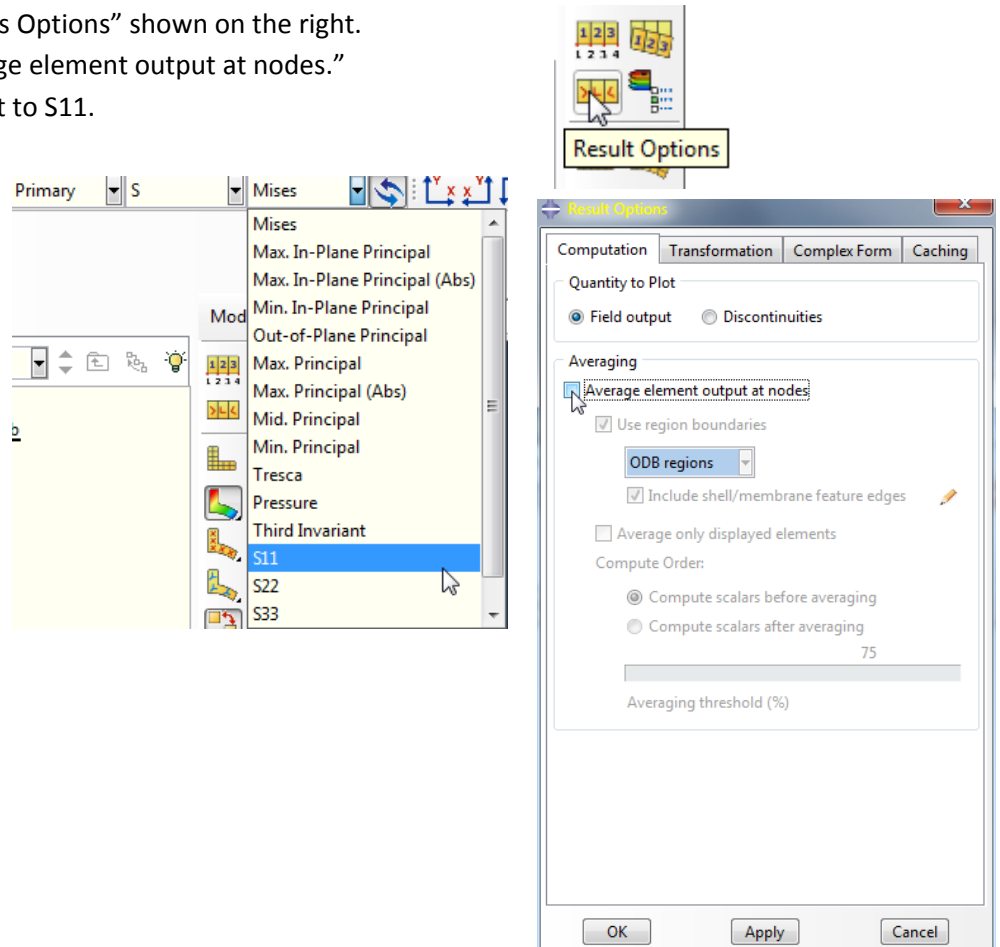
    c. Note the stress plot now shows each element acquiring a single color, denoting the Constant Stress nature of CST element!
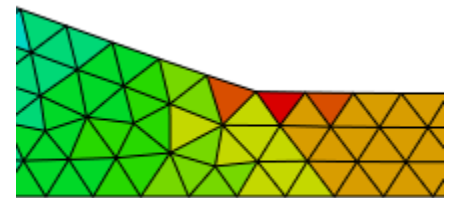
"Averaging" creates average stress values at the nodes based on the number of elements sharing the node (and stresses corresponding to those elements). It then creates a color gradient between values of adjacent nodes, thus the reason for "smooth" representation of colors. Averaged results can be useful for investigating mesh convergence at a point of interest where a node can be placed.