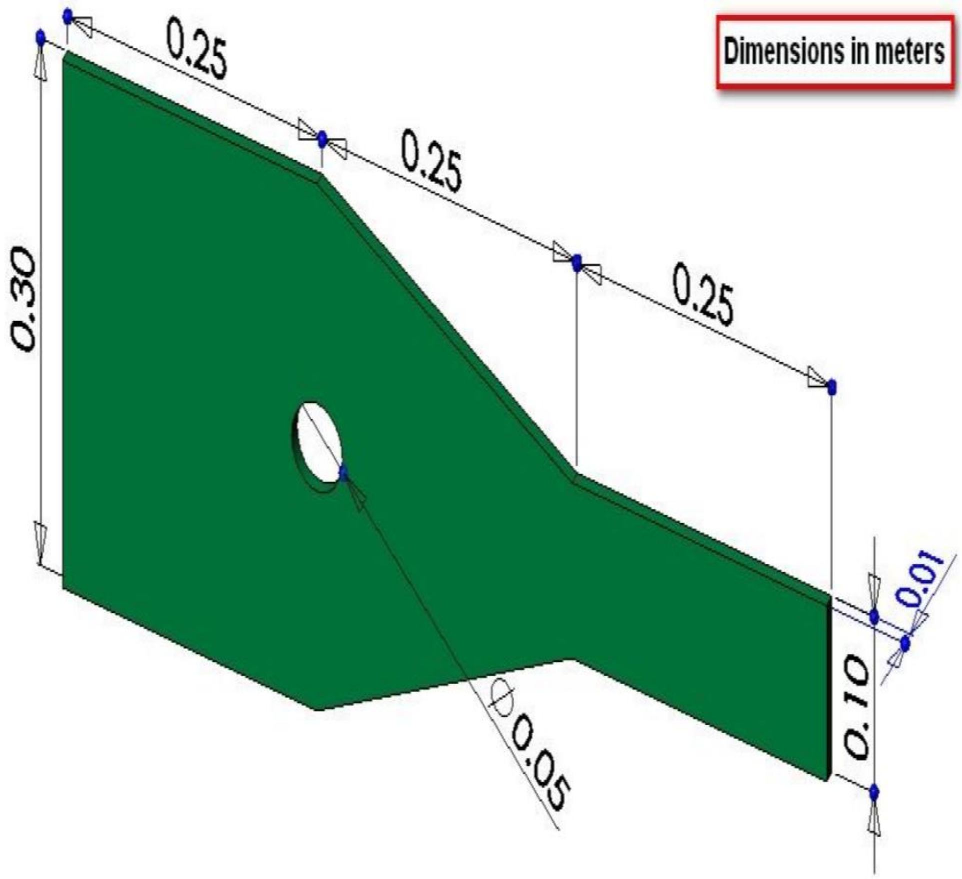
Abaqus/CAE tutorial – 2D plane stress

Prepared by Dr. Hamed

Problem description



The steel bracket shown is fixed to a wall along its left side. A tensile force with a magnitude of 5000N is applied to the right side of bracket. Determine the displacement and stress field in the bracket.

Note that:

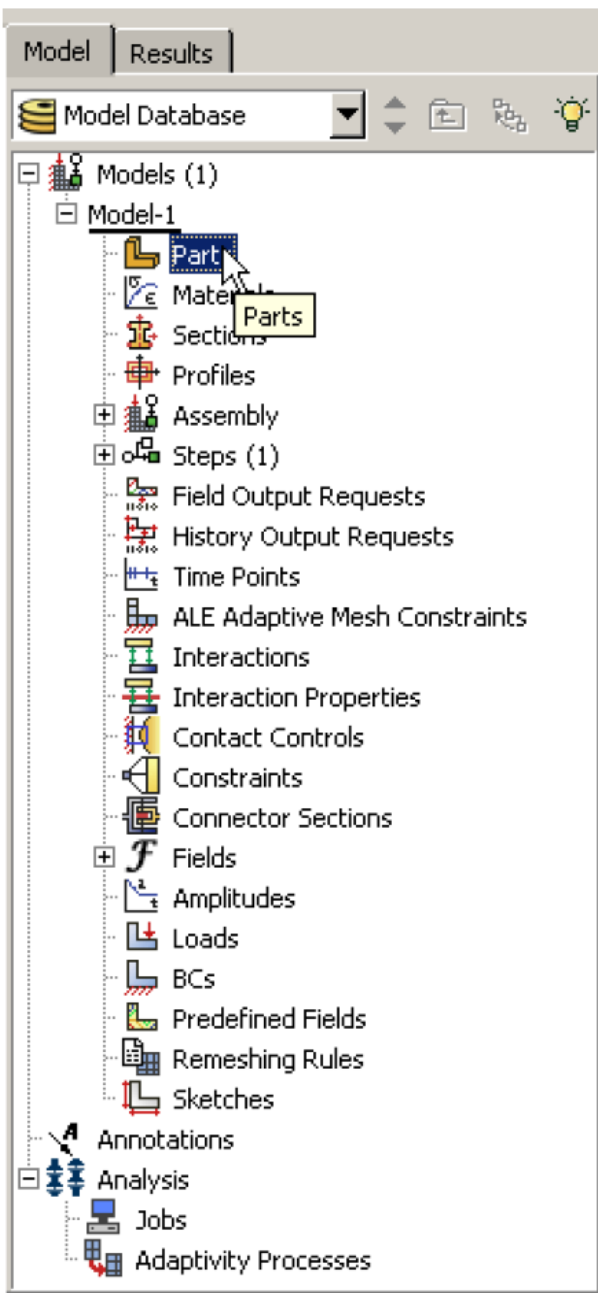
- Small thickness of the bracket and in-plane loading motivate the use of **2D plane stress** analysis.
- The structure contains a line of **symmetry**. So, only half of it will be modeled.

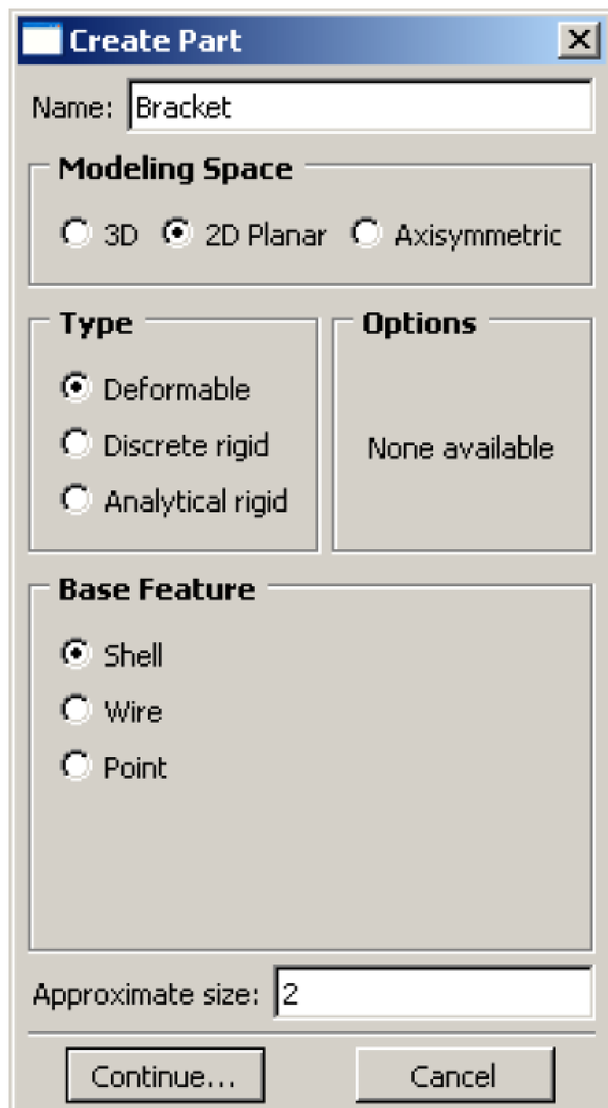
(1) Start Abaqus and choose to create a new model.

(2) In the model tree, double click on “parts” node

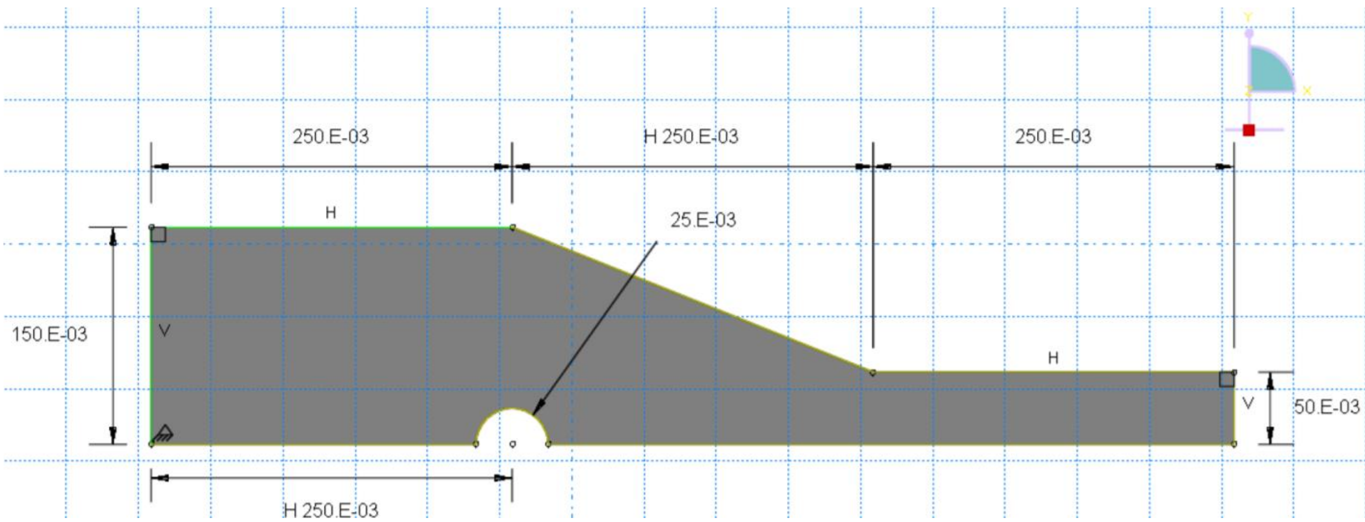
(3) In the Create Part dialog box, name the part and select:

- 2D planar
- Deformable
- Shell
- Approximate size=2



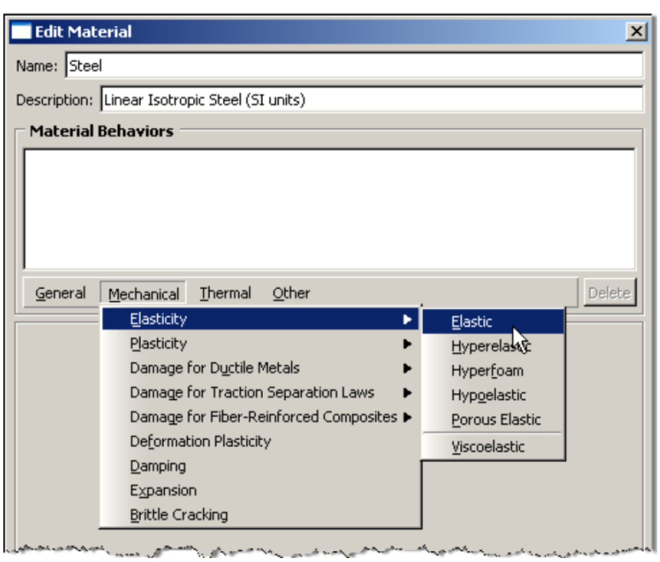


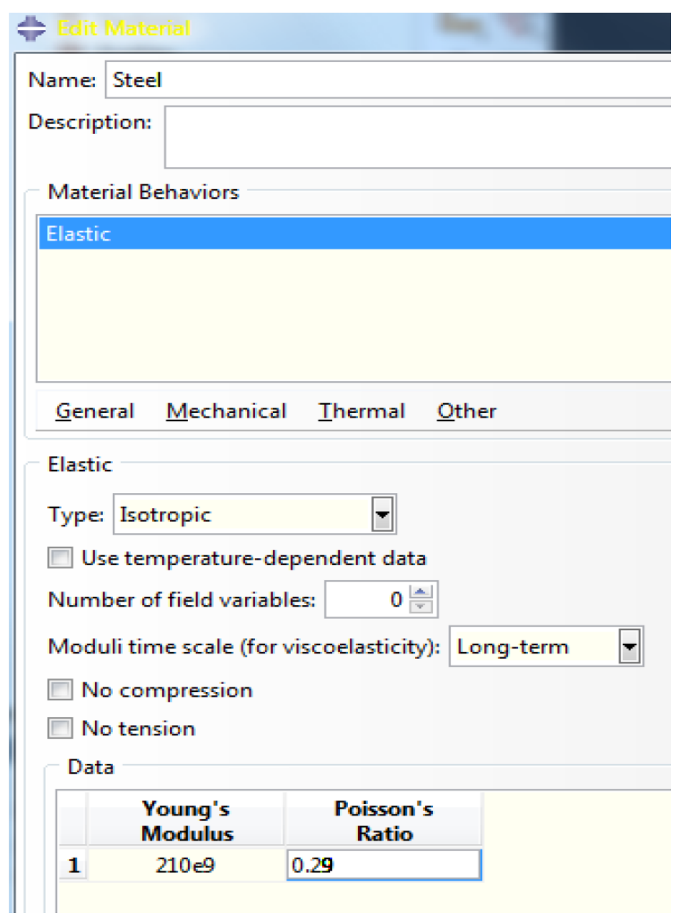
(4) Create the geometry shown below.



(4) Double click on the “Materials” node in the model tree

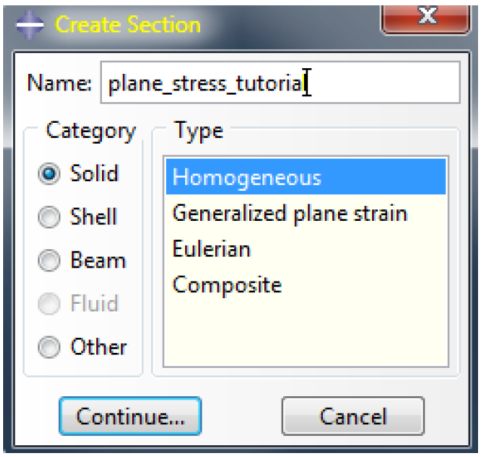
- Name the material and give it a description
- Click on the “Mechanical” tab → Elasticity → Elastic
- Define Young’s modulus and Poisson’s ratio in SI units

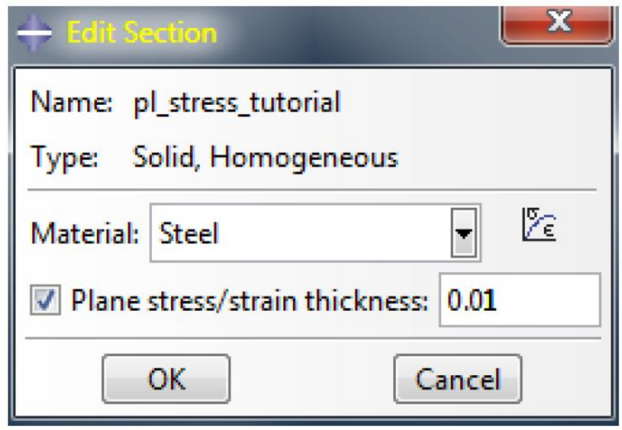




(6) Double click on the “Sections” node in the model tree

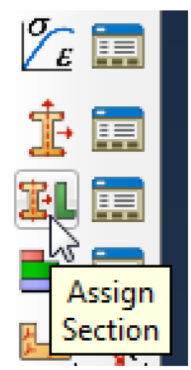
- Name the section “PlaneStressProperties” and select Solid for the category and “Homogeneous” for type
- Select the material created above (Steel) and set the thickness to be 0.01

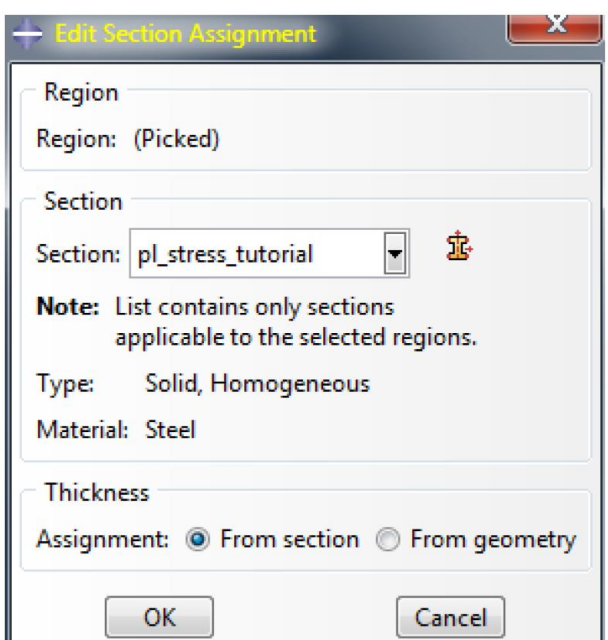




(7) Click on the “Assign Section” icon

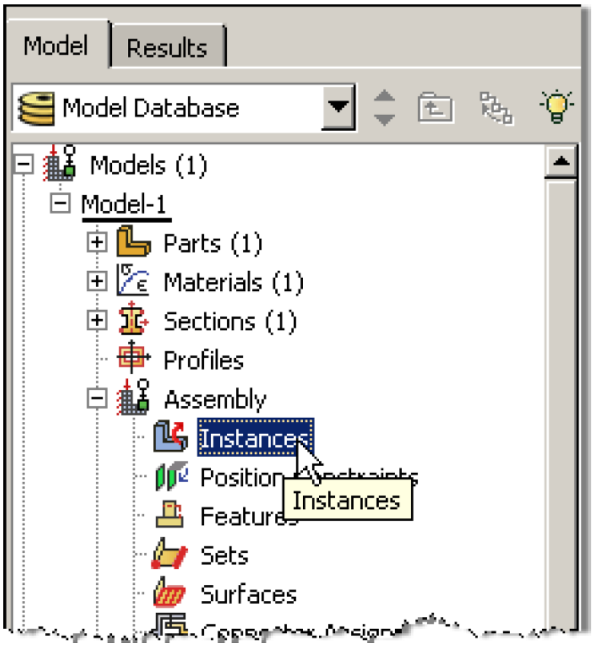
- Select the surface geometry in the viewport
- Make sure that the section created above (plane_stress_tutorial) is selected.

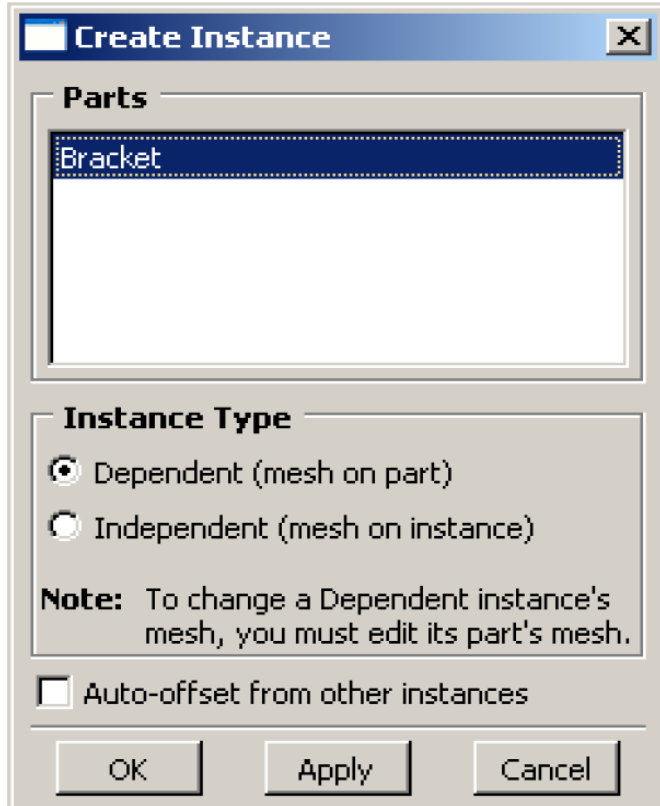




(8) Expand the “Assembly” node in the tree model and then double click on “Instances”

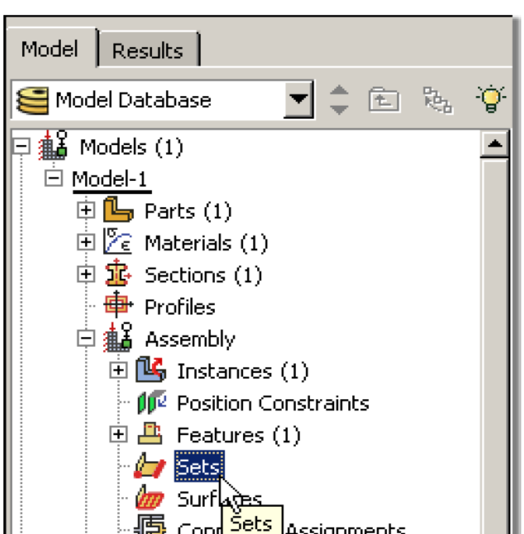
- Select “Dependent” for the instance type

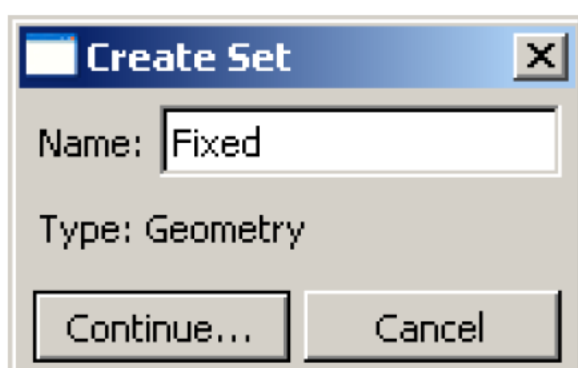




(9) In the model tree, under the expanded “Assembly” node, double click on “Sets”

- Name the set “Fixed”
- Select the left edge of the surface in the viewport



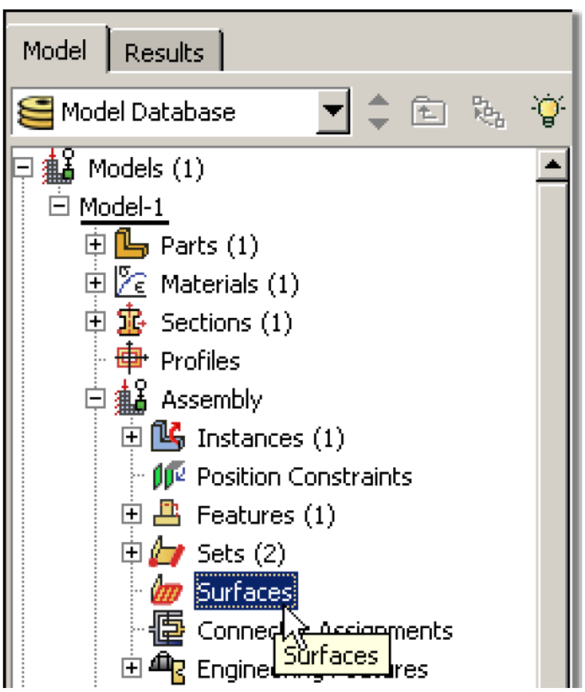


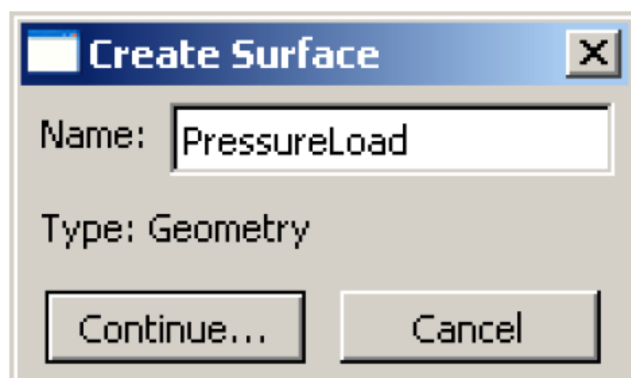
(9) (continue)

- Create another Set named “Symmetry”
- 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”

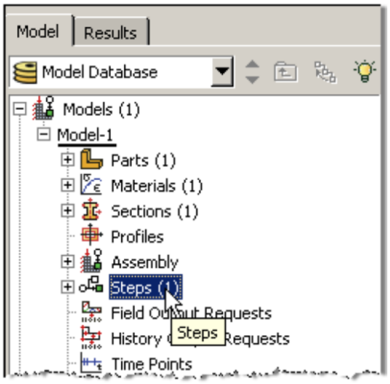
- Name the surface “PressureLoad”
- Select the right edge of the surface in the viewport

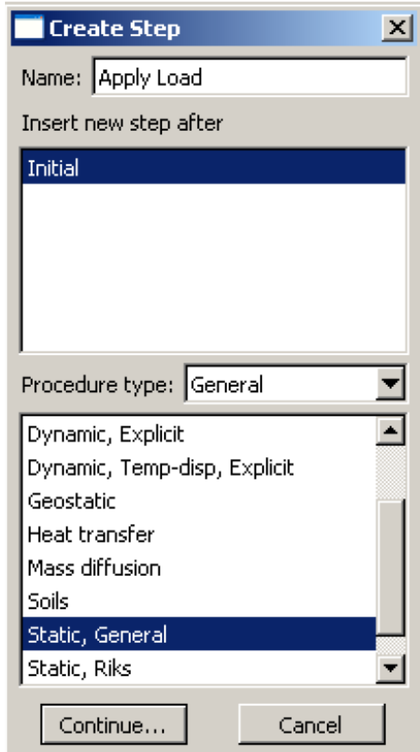


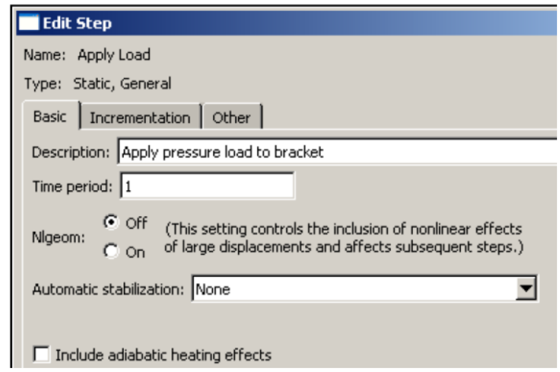


(11) Double click on the “Steps” node in the model tree

- Name the step, set the Procedure to “General” and select “Static, General”.
- Give the step a description

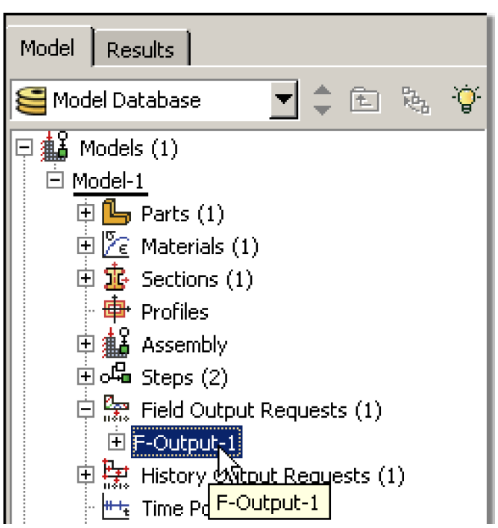


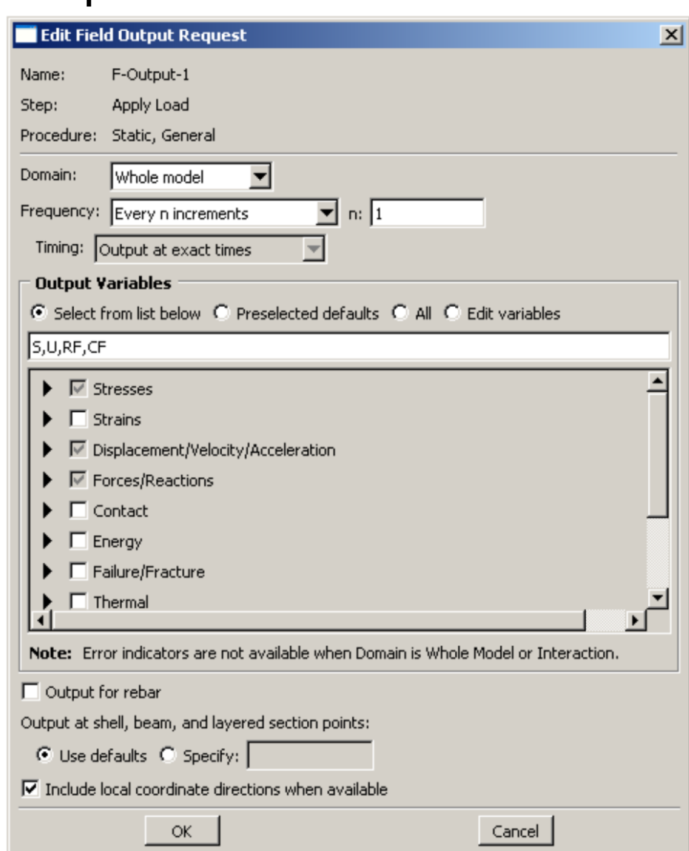




(12) Expand the Field Output Requests node in the model tree, and double click on F-Output-1

- Uncheck the variables “Strains” and “Contact”
- Give the step a description

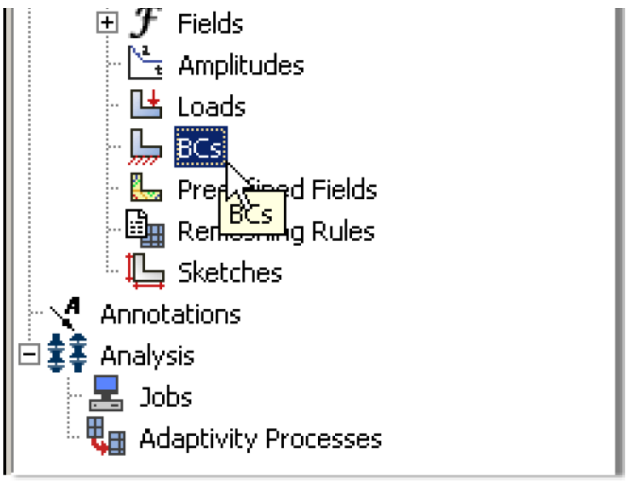


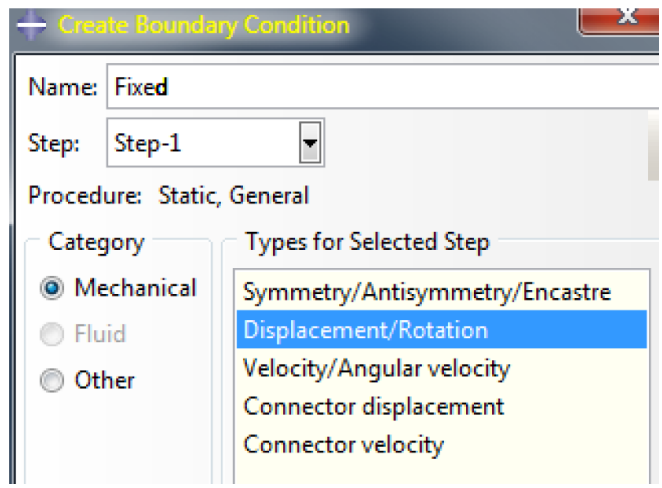


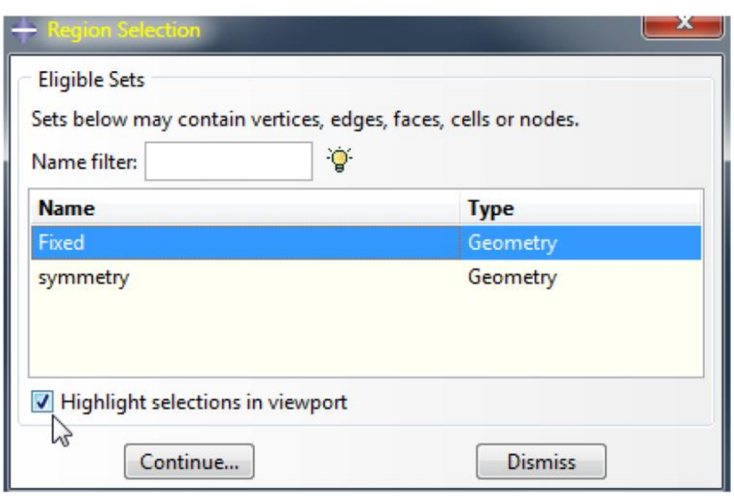
(13) Expand the History Output Requests node in the model tree, right click on H-Output-1 and select Delete

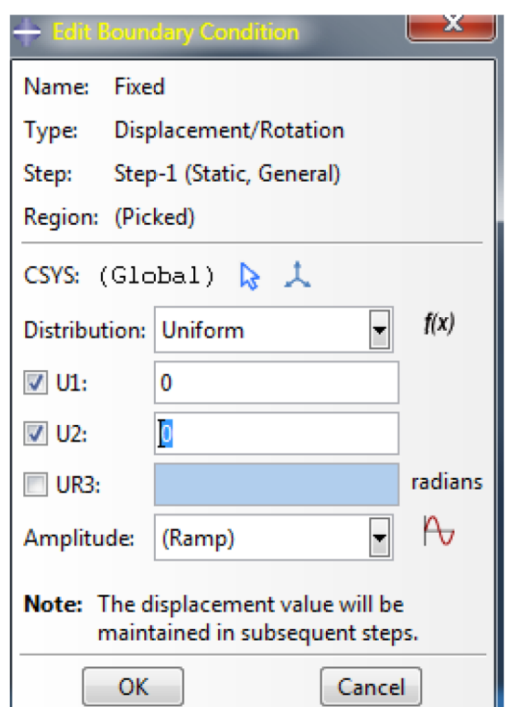
(14) Double click on the “BCs” node in the model tree

- Name the BC “Fixed” and select “Displacement/Rotation” for type
- In the prompt area click on the Set button
- Select the set named “Fixed”
- Check U1 and U2 to fully restrain the left edge







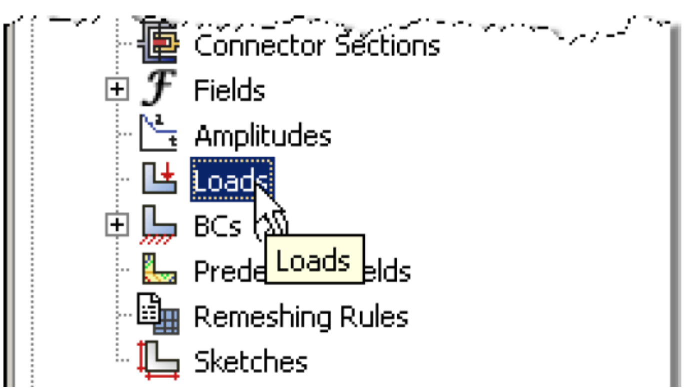


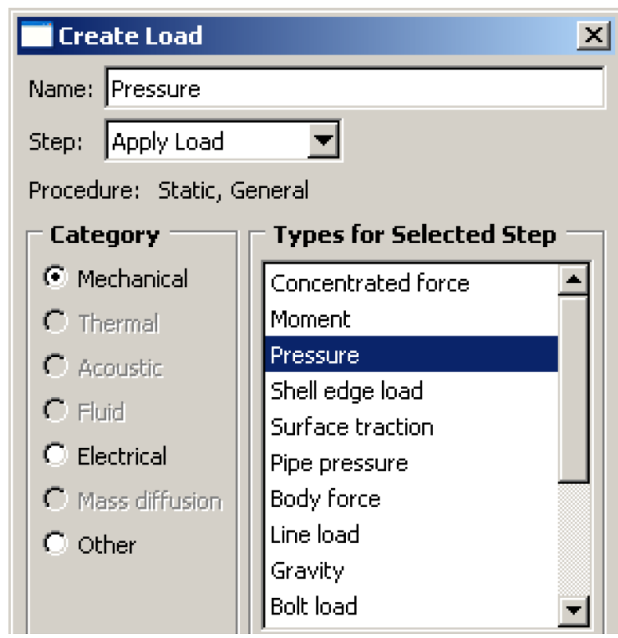
(14) (continue)

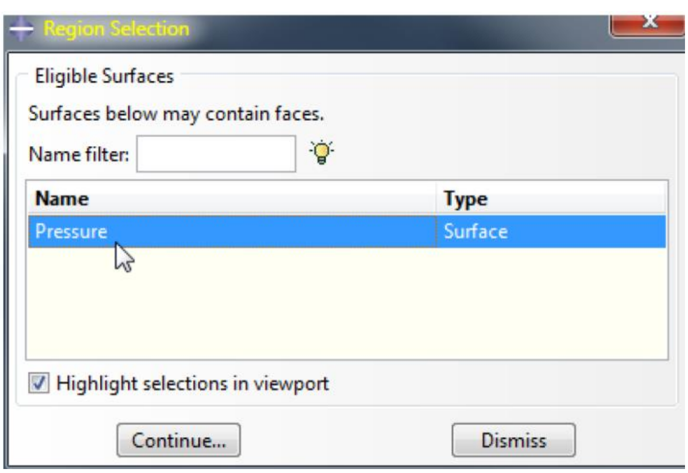
- Repeat the procedure for the symmetry BC using the set named “Symmetry”. Set U2 to be 0 for this BC.

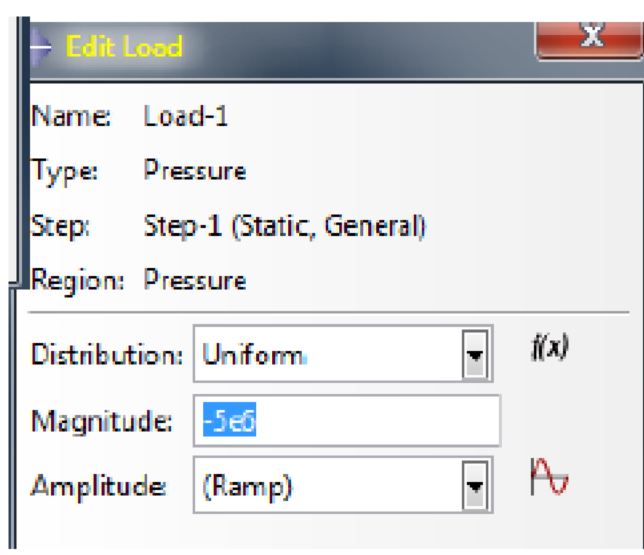
(15) Double click on the “Loads” node in the model tree

- Name the load “Pressure” and select “Pressure” as type
- Select the surface named “Pressure”
- The applied pressure is -5×10^6 N/m^2, which is a load of -5000/2 N distributed across the right cross sectional area of surface (-2500 N/(0.05 m x 0.01 m))

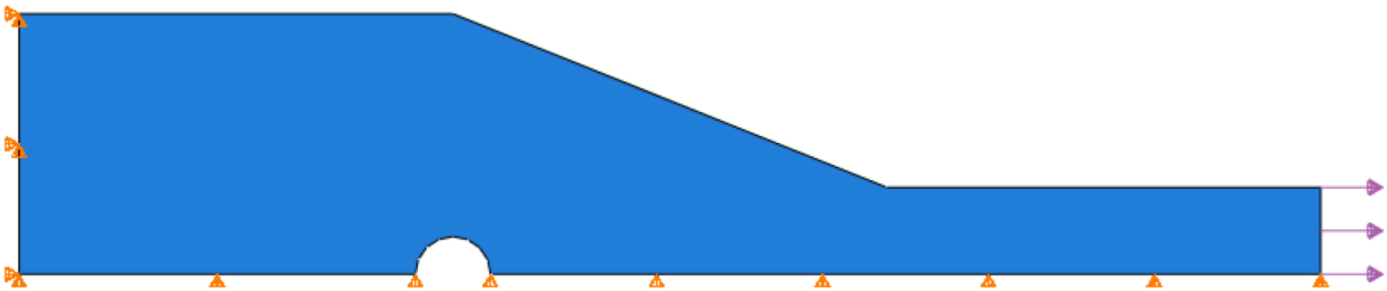






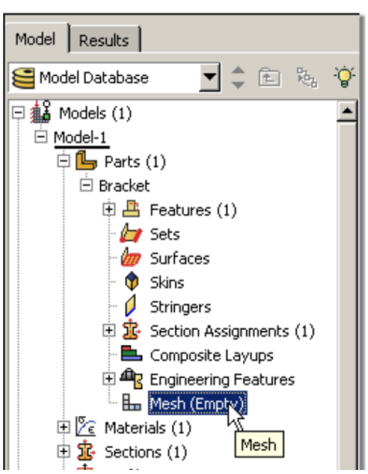


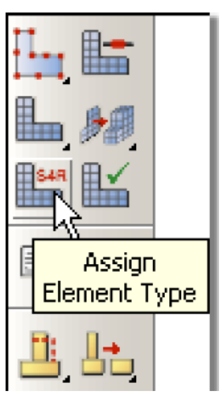
After application of BCs and load, your model should look like the figure below.

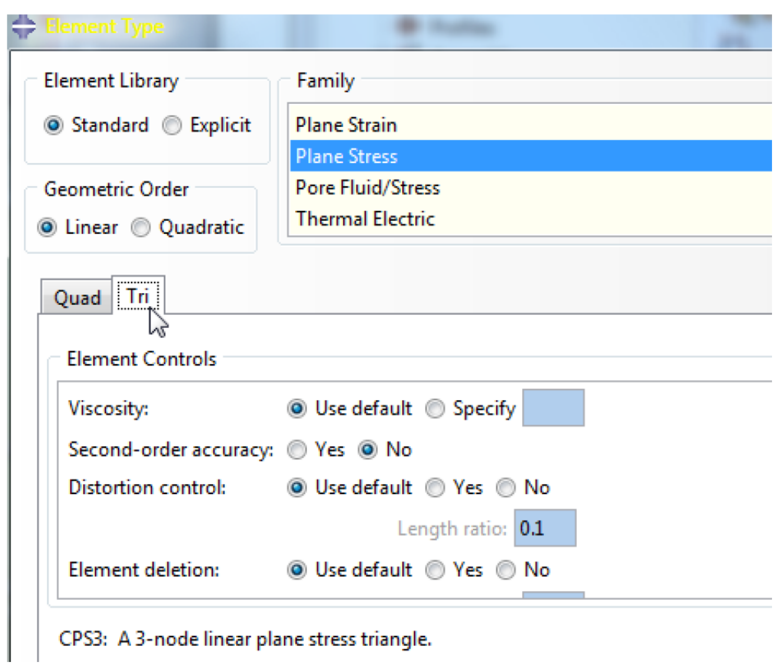


(16) In the model tree, double click on the “Mesh” for the Bracket part, and in the toolbox area click on the “Assign Element Type” icon

- Select “Standard” for element type
- Select linear for geometric order
- Select “Plane Stress” for family
- Pick the tab “Tri” and note the name and description of CST triangular element in the description window: **CPS3**

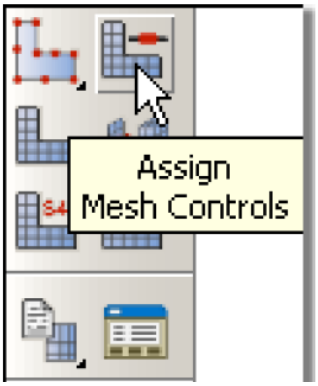


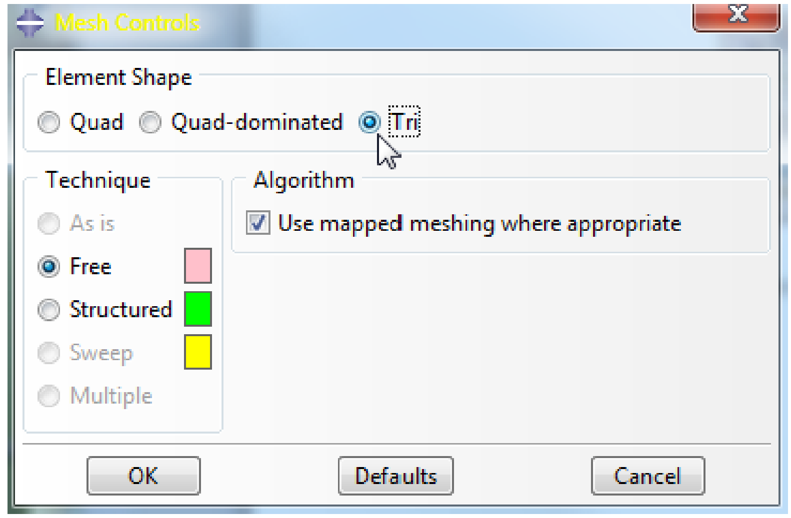




(17) In the toolbox area, click on “Assign Mesh Controls”

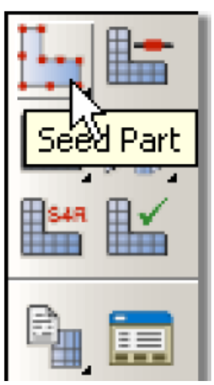
- Select the element shape as “Tri” and click OK

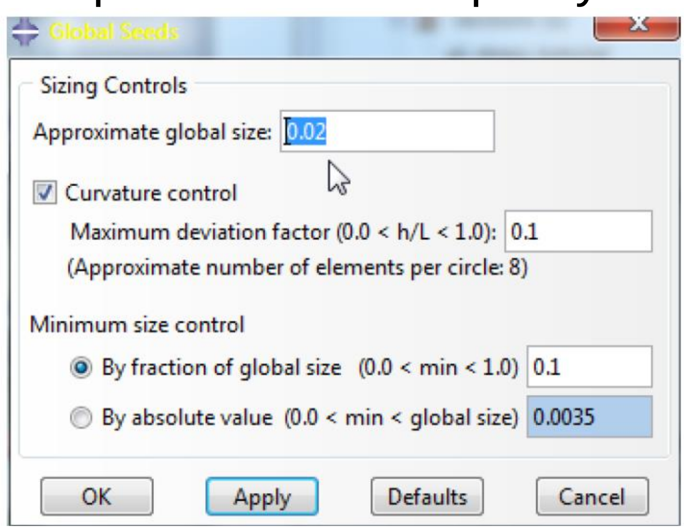




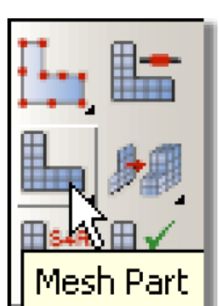
(18) In the toolbox area, click on “Seed Part” icon

- Select the approximate global size to 0.02
- Pay attention to options for mesh quality control



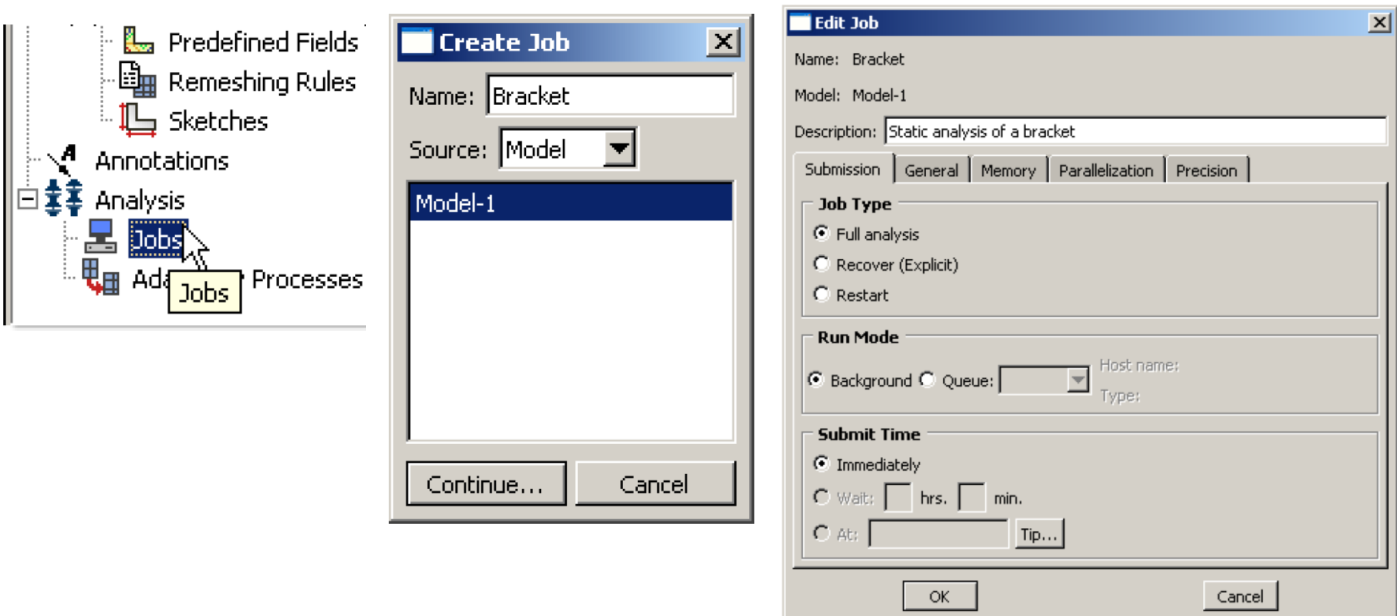


(19) In the toolbox area, click on “Mesh Part” icon



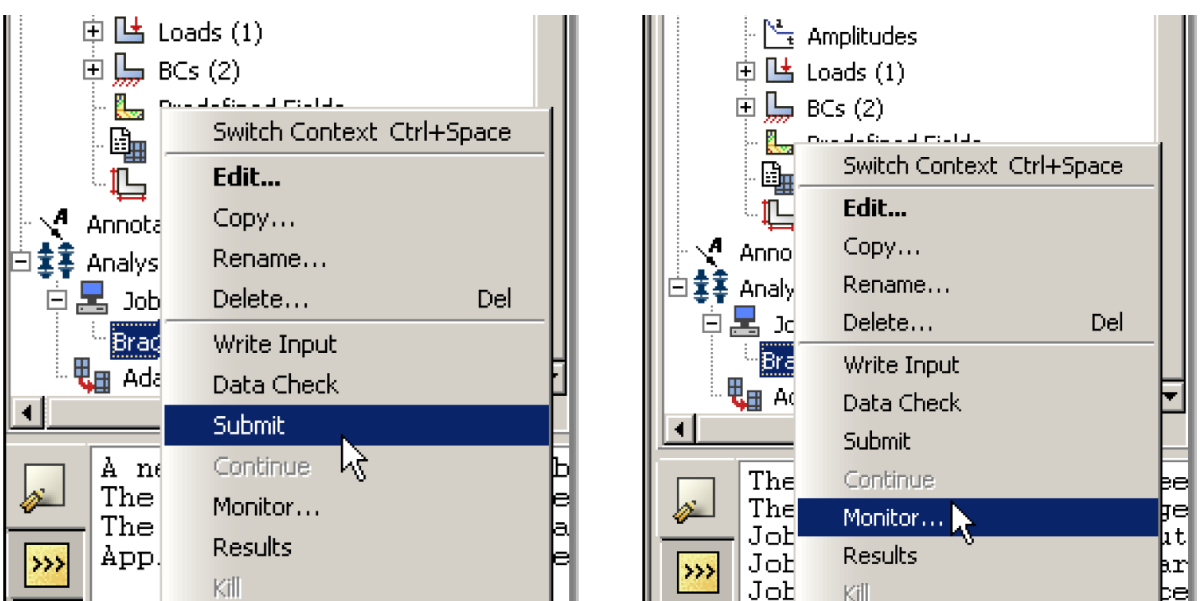
(20) In the model tree double click on the “Job” mode

- Name the job “Bracket”
- Give the job a description

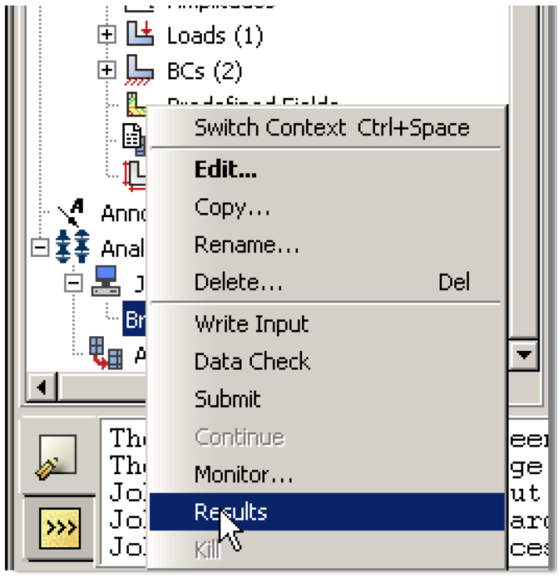


(21) In the model right click on the job created (Bracket) and click on “Submit”

- While Abaqus is solving the problem, right click on the submitted job (Bracket) and select “Monitor”
- In the monitor window, check that there are no errors or warnings

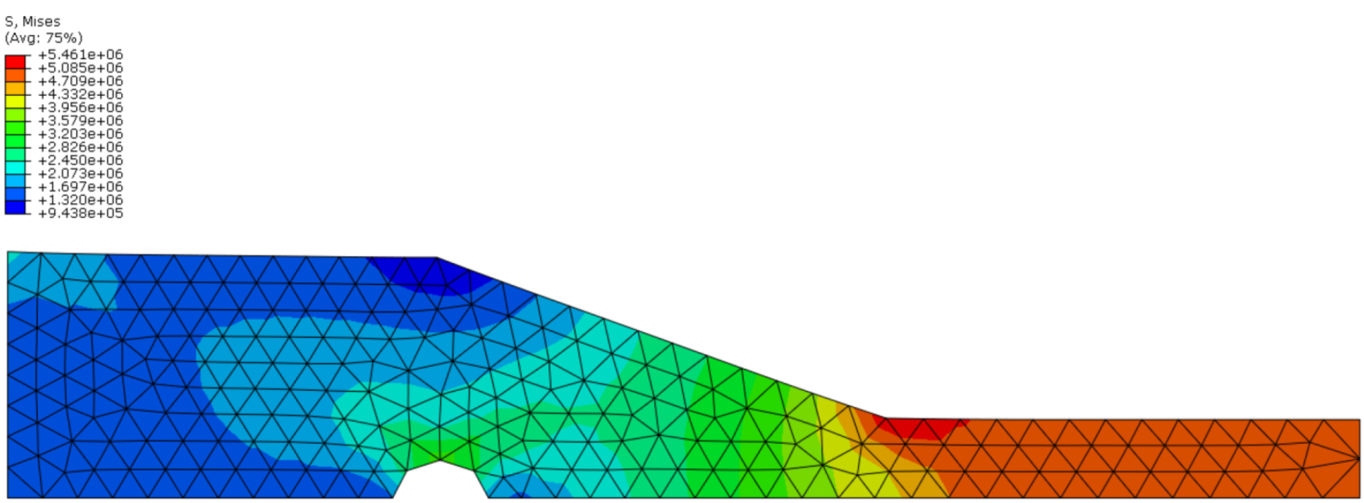


(22) In the model tree right click on submitted and successfully completed job (Bracket) and select “Results”

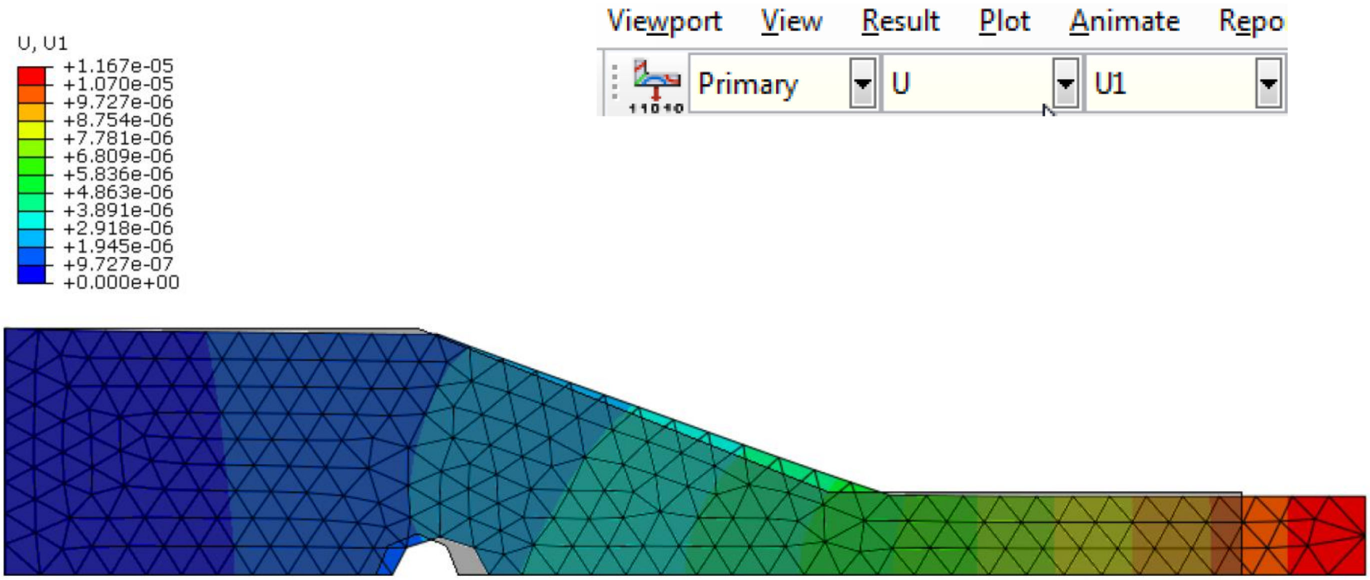


(23) Display the contour of Mises stress on the deformed shape

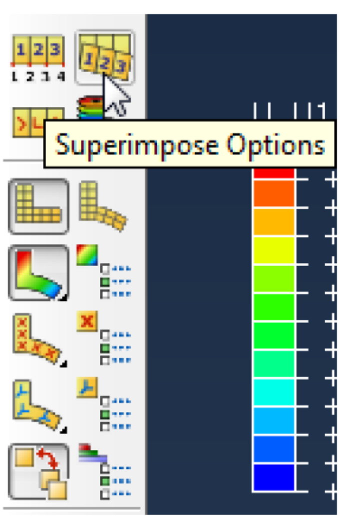
- In the toolbox area click on the “Plot Contours on Deformed Shape” icon

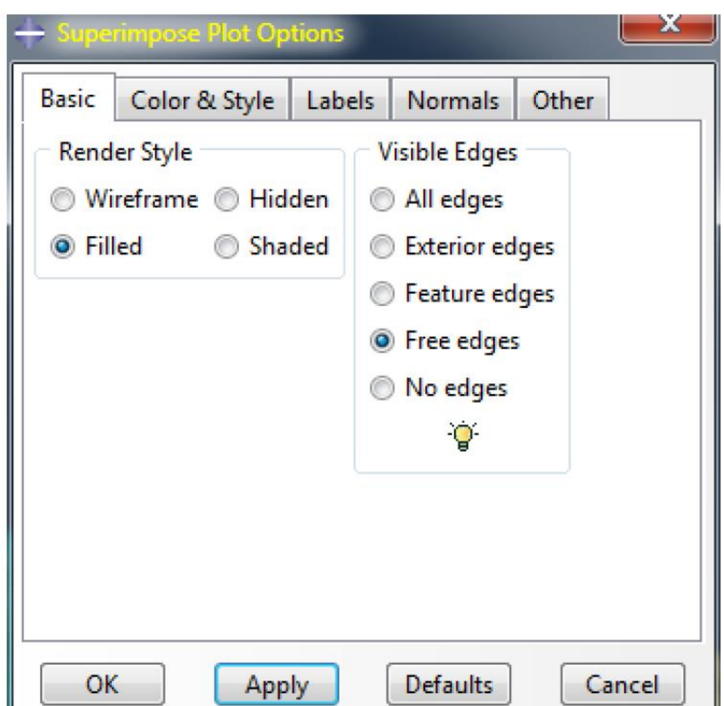


(24) Request displacement plot along x direction, with the undeformed shape superimposed.



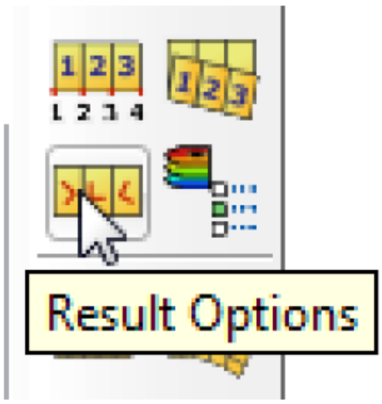
The image was obtained using the “Superimpose Options” icon and setting the parameters as shown. Note the clear stretching of the right side of the bracket and the imposition of symmetry BC that prevents lower horizontal surface from moving in the vertical direction.

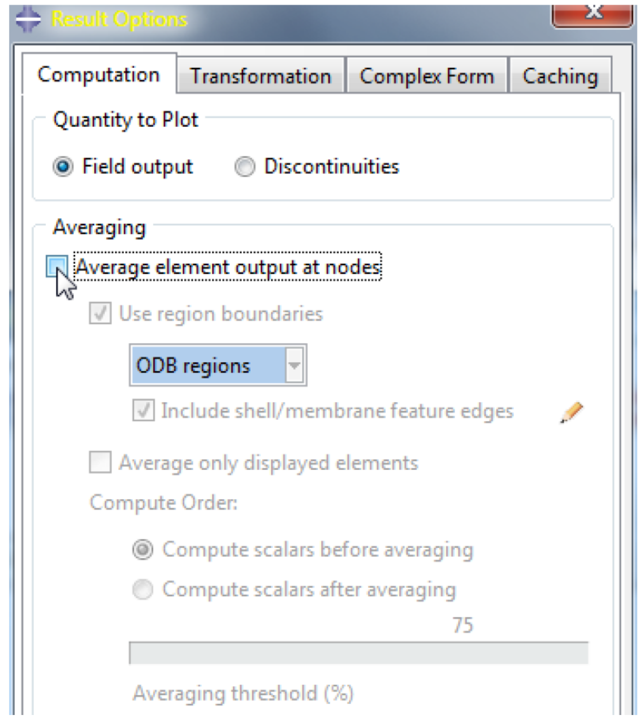


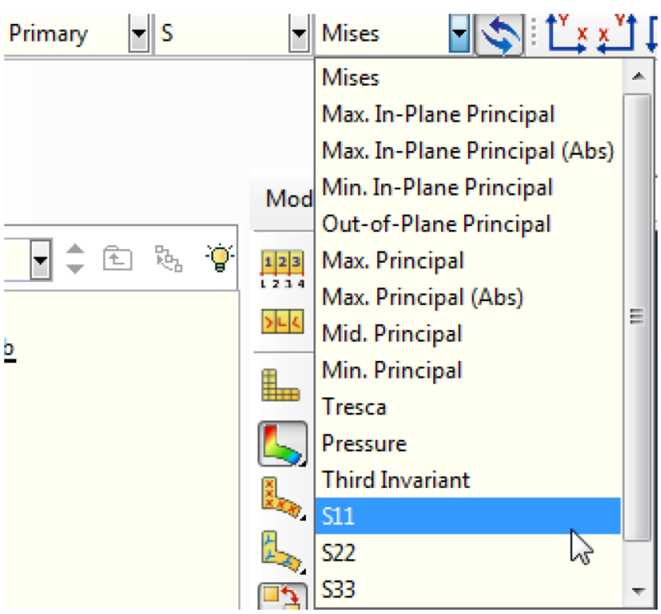


(25) Activate the icon “results Options” on the toolbox area.

- Uncheck the box “Average element output at nodes”
- Change the stress output to S11.







- Note the stress value in each element is depicted only by a single color, denoting the **constant stress** nature of 3-node triangular element (**CST**)