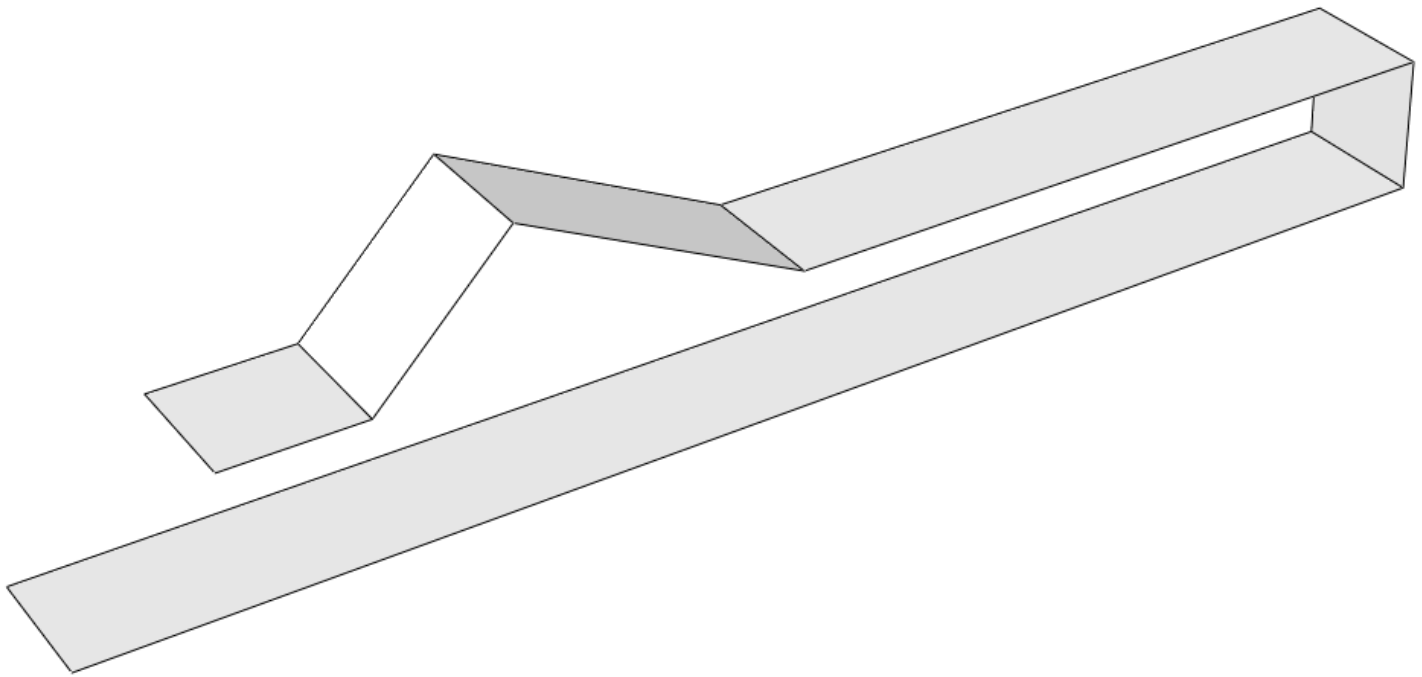


Abaqus CAE (ver. 6.9) Contact Tutorial

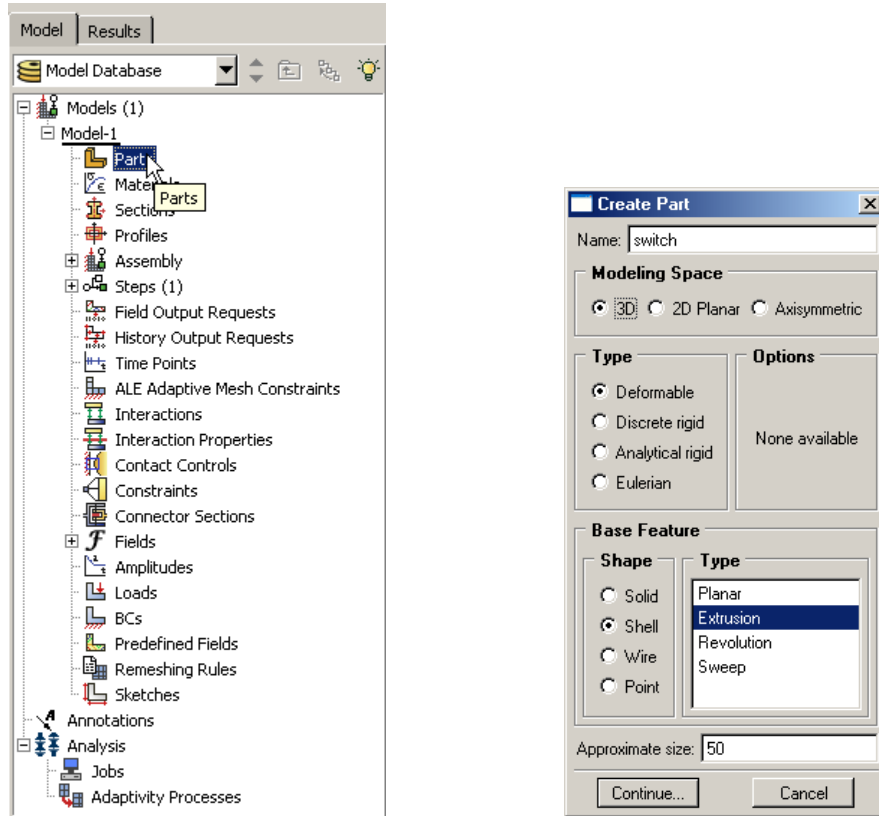
Problem Description



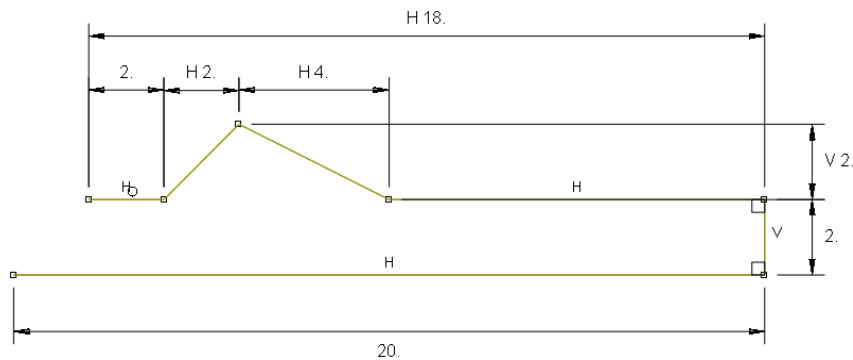
Note: You do not need to extrude the right vertical edge of the sensor.

Analysis Steps

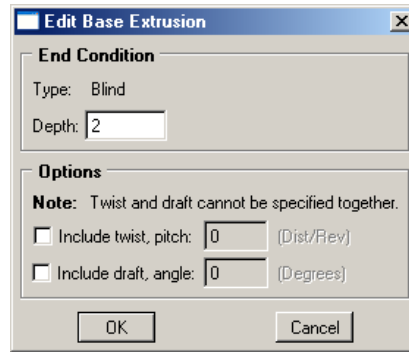
1. Start Abaqus and choose to create a new model database
2. In the model tree double click on the “Parts” node (or right click on “parts” and select Create)



3. In the Create Part dialog box (shown above) name the part and
 - a. Select “3D”
 - b. Select “Deformable”
 - c. Select “Shell”
 - d. Select “Extrusion”
 - e. Set approximate size = 50
 - f. Click “Continue...”
4. Create the geometry shown below (not discussed here)



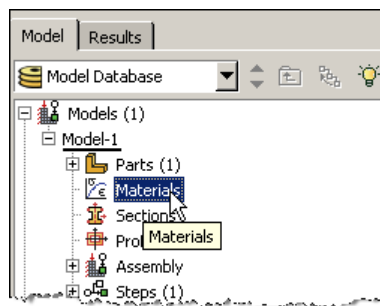
a. Click “Done”



b. Set Depth = 2

c. Click “OK”

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

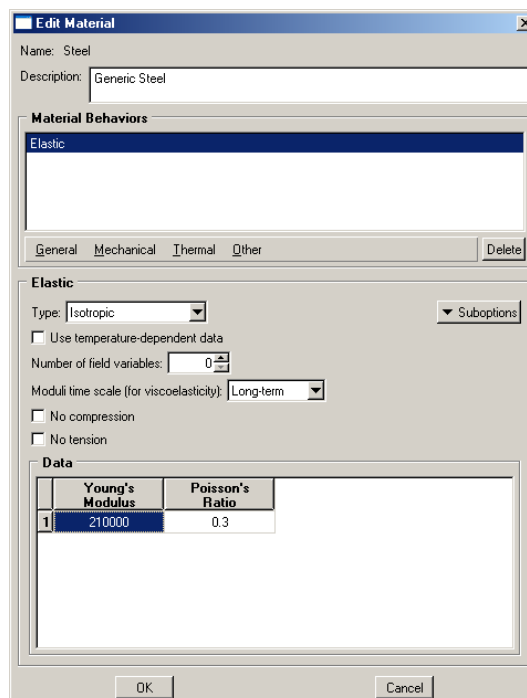


a. Name the new material and give it a description

b. Click on the “Mechanical” tab → Elasticity → Elastic

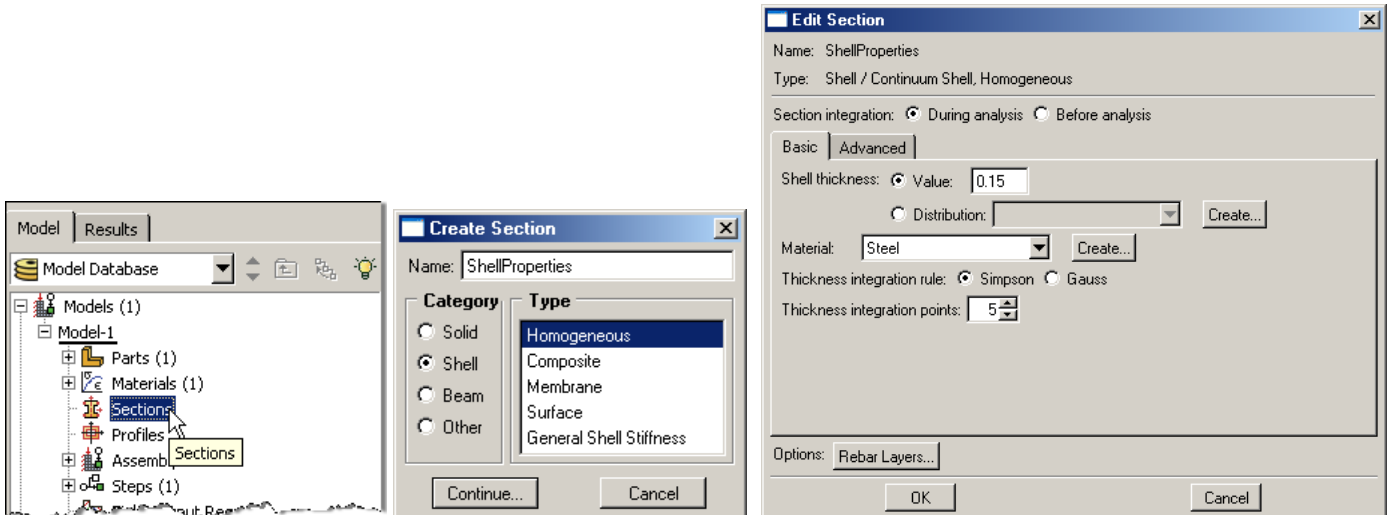
c. Define Young’s Modulus and the Poisson’s Ratio (use SI (mm) units)

d. Click “OK”



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

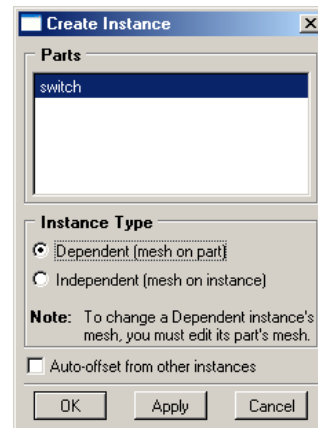
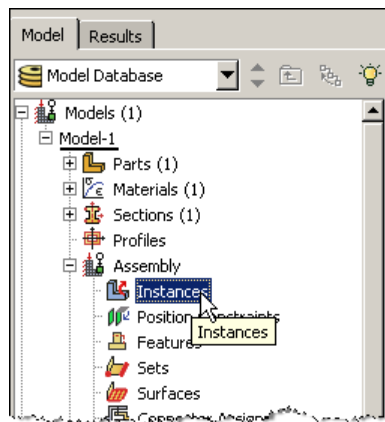
- Name the section "ShellProperties" and select "Shell" for the category and "Homogeneous" for the type
- Click "Continue..."
- Select the material created above (Steel) and set the thickness to 0.15
- Click "OK"



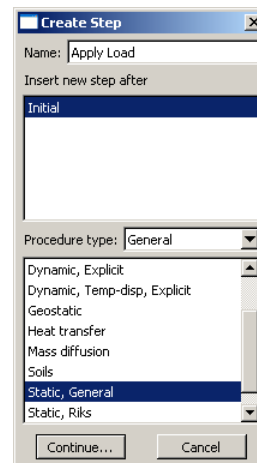
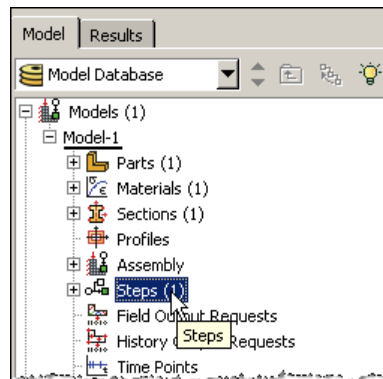
- Expand the "Parts" node in the model tree, expand the node of the part just created, and double click on "Section Assignments"
 - Select the entire geometry, except for the vertical face, in the viewport and press "Done" in the prompt area
 - Select the section created above (ShellProperties)
 - Specify shell offset if necessary
 - Click "OK"



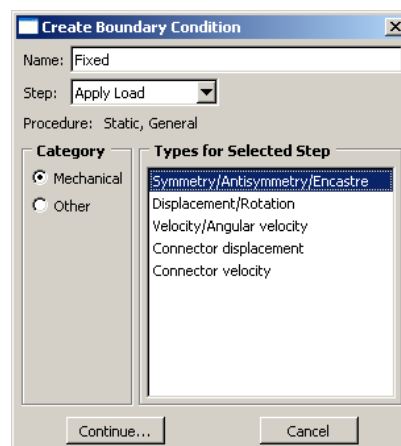
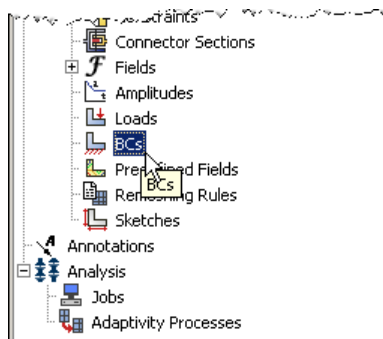
- Expand the "Assembly" node in the model tree and then double click on "Instances"
 - Select "Dependent" for the instance type
 - Click "OK"



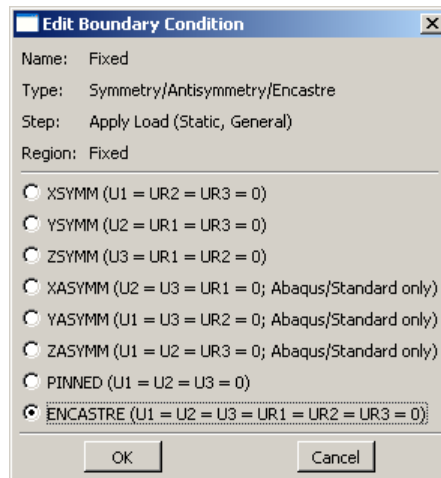
9. Double click on the “Steps” node in the model tree
 - a. Name the step, set the procedure to “General”, select “Static, General”, and click “Continue...”
 - b. Accept the default settings



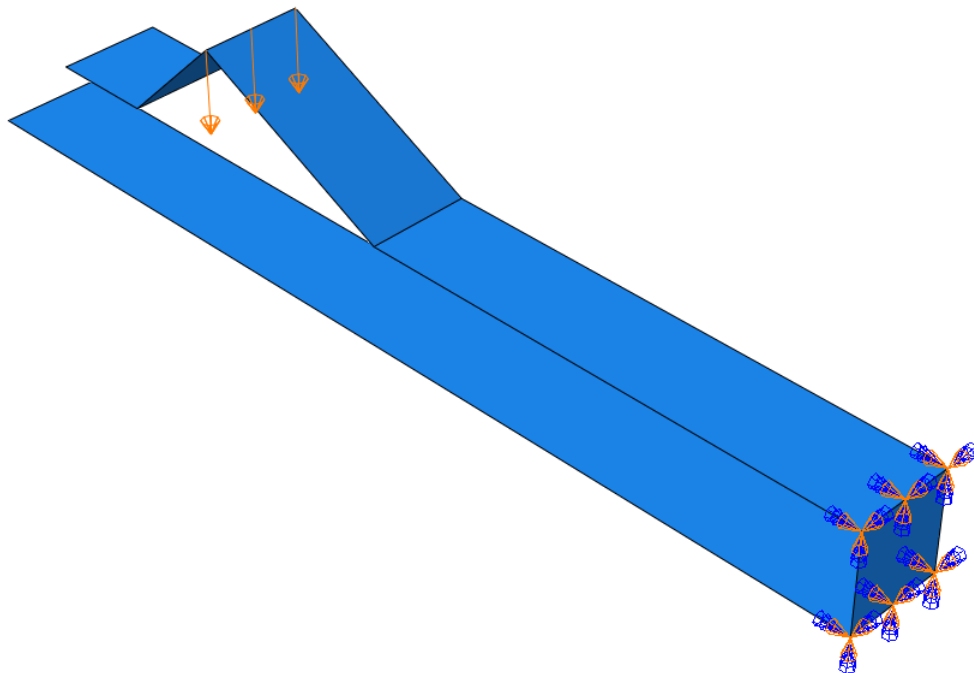
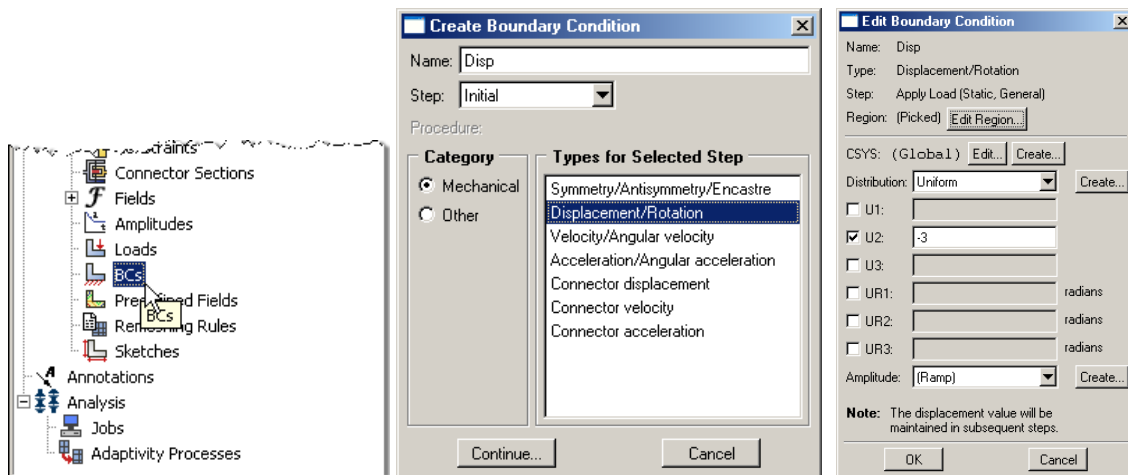
10. Double click on the “BCs” node in the model tree
 - a. Name the boundary conditioned “Fixed” and select “Symmetry/Antisymmetry/Encastre” for the type



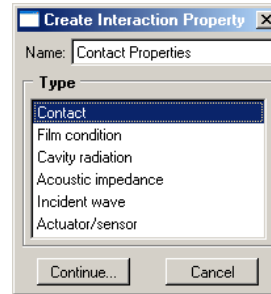
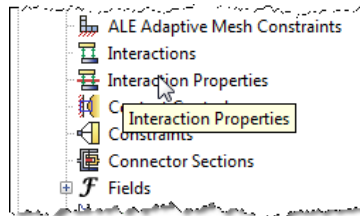
- b. Select the horizontal edges on the vertical surface and click “Done”
- c. Select “ENCASTRE” for the boundary condition and click “OK”



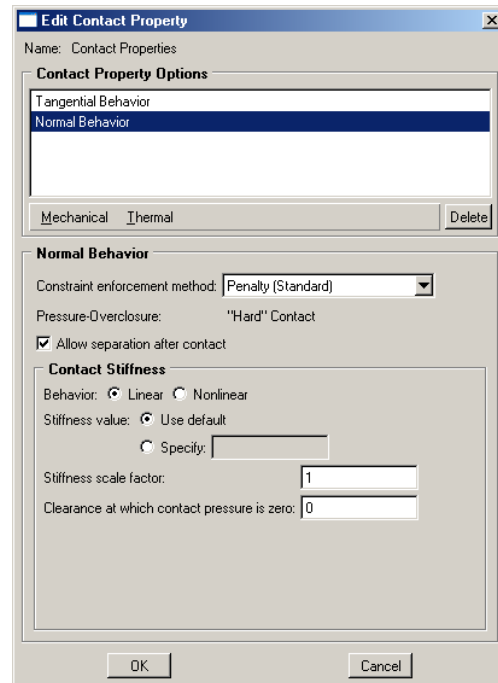
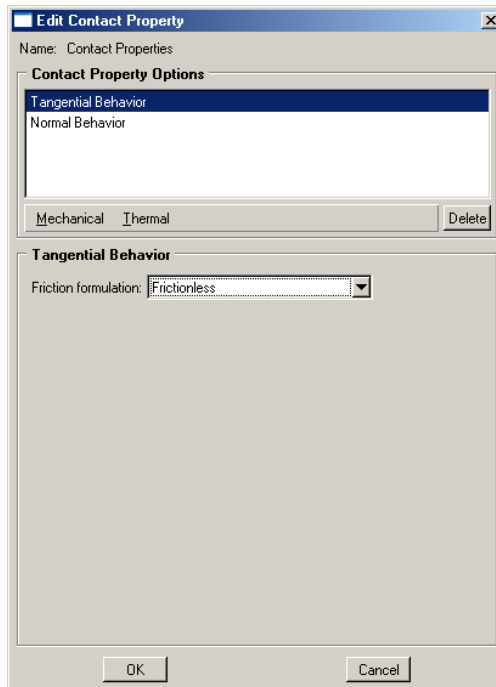
11. Double click on the “BCs” node in the model tree
 - a. Name the boundary conditioned “Disp” and select “Displacement/Rotation” for the type
 - b. Select the top edge of the triangular portion of the geometry
 - c. Set the y-displacement to -3



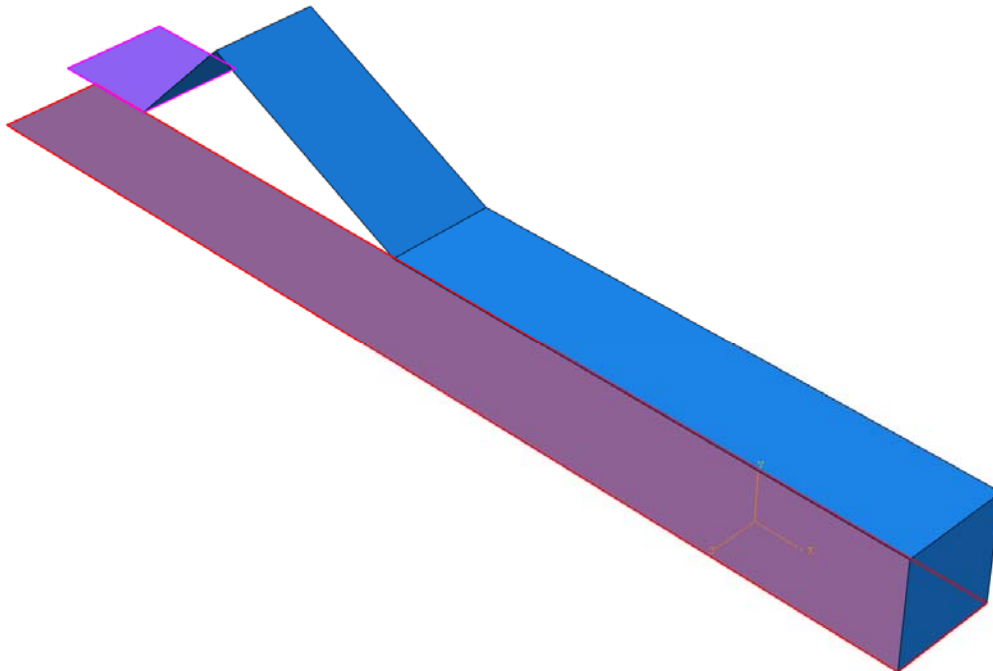
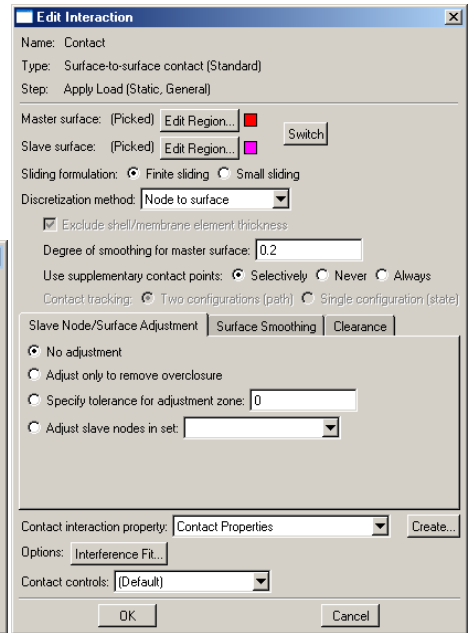
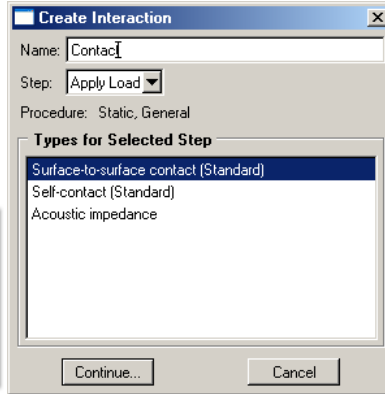
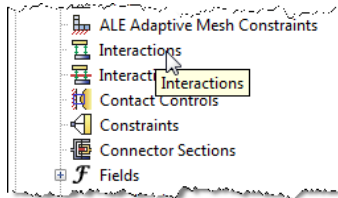
12. Double click on the “Interaction Properties” node in the model tree
 - a. Name the interaction properties and select “Contact” for the type



- b. On the Mechanical tab Select “Tangential Behavior”
 - i. Set the friction formulation to “Frictionless”
 - c. On the Mechanical tab Select “Normal Behavior”
 - i. Because the surfaces do not start in contact, change the constraint enforcement method to “Penalty”

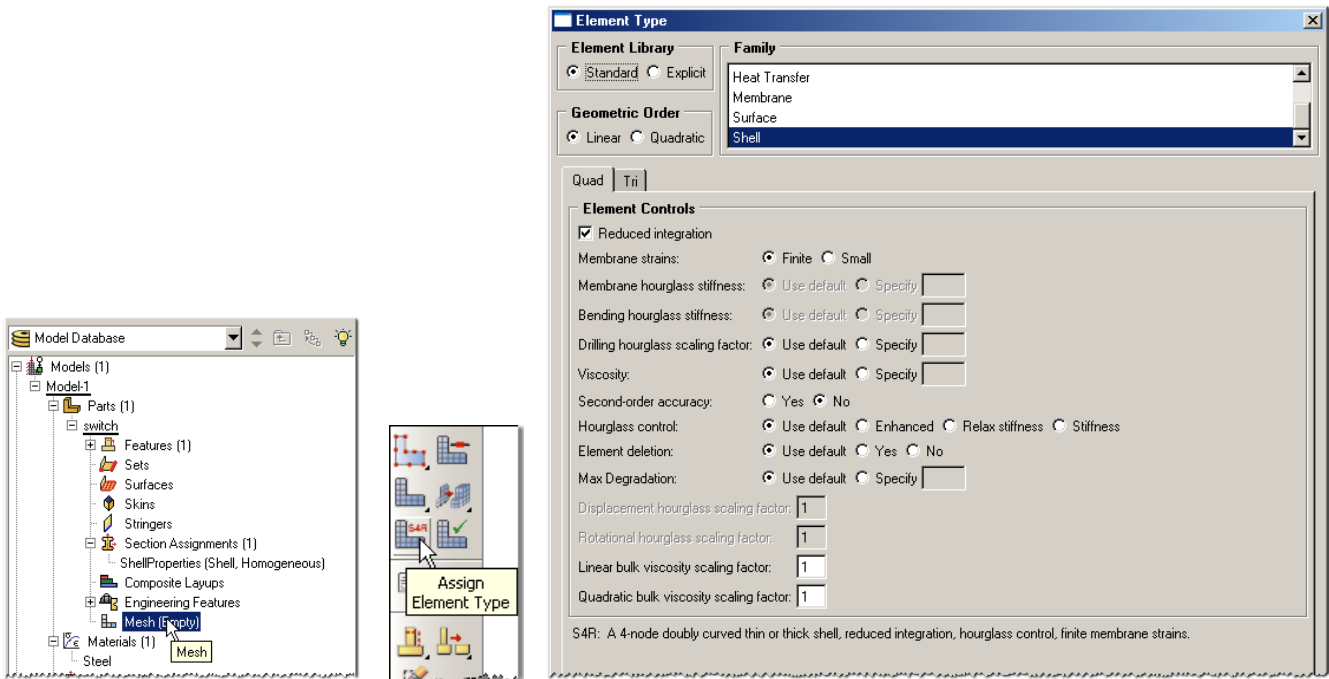


13. Double click on the “Interactions” node in the model tree
 - a. Name the interaction, select “Surface-to-surface contact”, and click continue
 - b. For the master surface select the lower portion of the geometry and click done
 - i. While applying the fixed displacement, the nodes at the tip of the upper portion of the geometry will make contact at an unknown location on the lower surface
 - ii. Nodes on the slave surface cannot penetrate the surface formed by the element faces on the master surface
 - c. Select the color of the surface corresponding to the top surface
 - d. For the slave surface, set the slave type to “Surface”
 - e. Select the upper portion of the geometry at the free end and click done
 - f. Select the color of the surface corresponding to the bottom surface
 - g. Change the contact interaction properties to the one created above (if not already done)



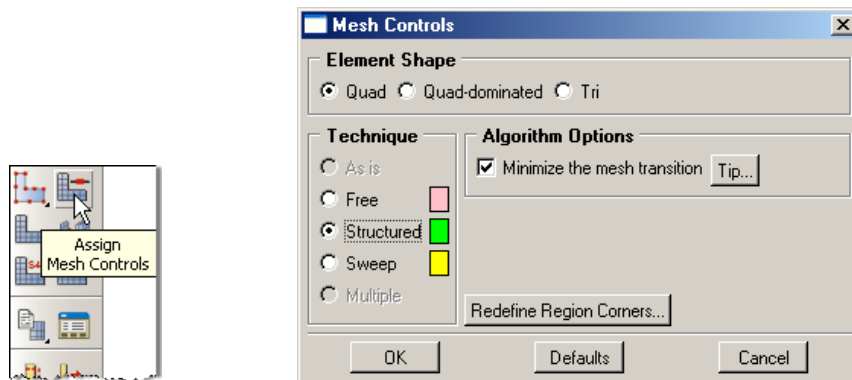
14. In the model tree double click on “Mesh” for the Arch part, and in the toolbox area click on the “Assign Element Type” icon

- Select the portion of the geometry associated with the boundary conditions and load
- Select “Standard” for element type
- Select “Linear” for geometric order
- Select “Shell” for family
- Note that the name of the element (S4R) and its description are given below the element controls
- Select “OK”



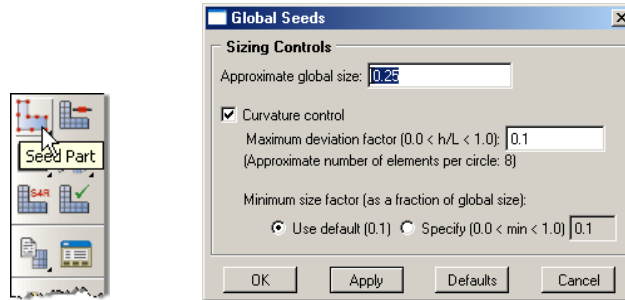
15. In the toolbox area click on the “Assign Mesh Controls” icon

- Select the portion of the geometry associated with the boundary conditions and load
- Change the element shape to “Quad”
- Change the technique to “Structured”

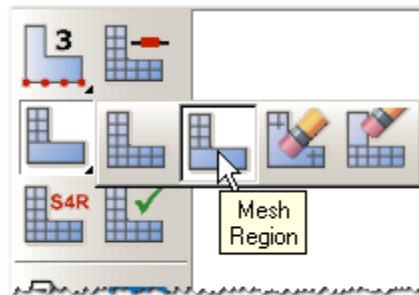


16. In the toolbox area click on the “Seed Part” icon

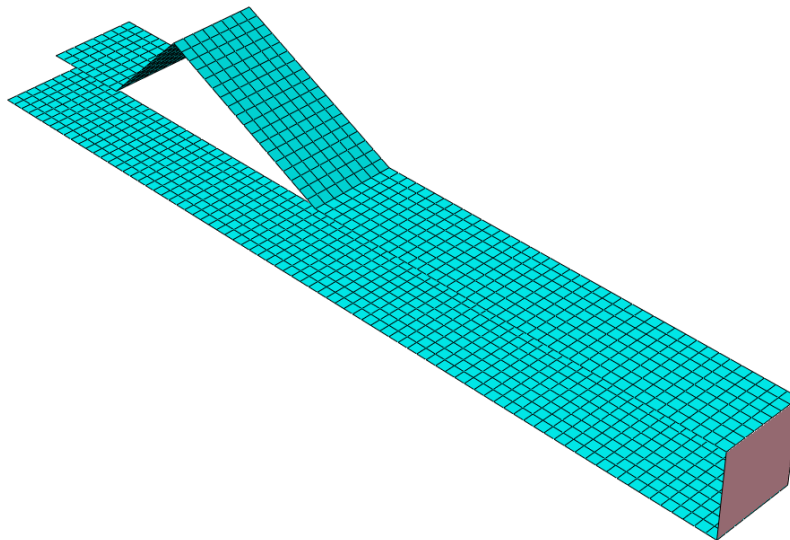
- a. Set the approximate global size to 0.25



17. In the toolbox area click on the “Mesh Region” icon

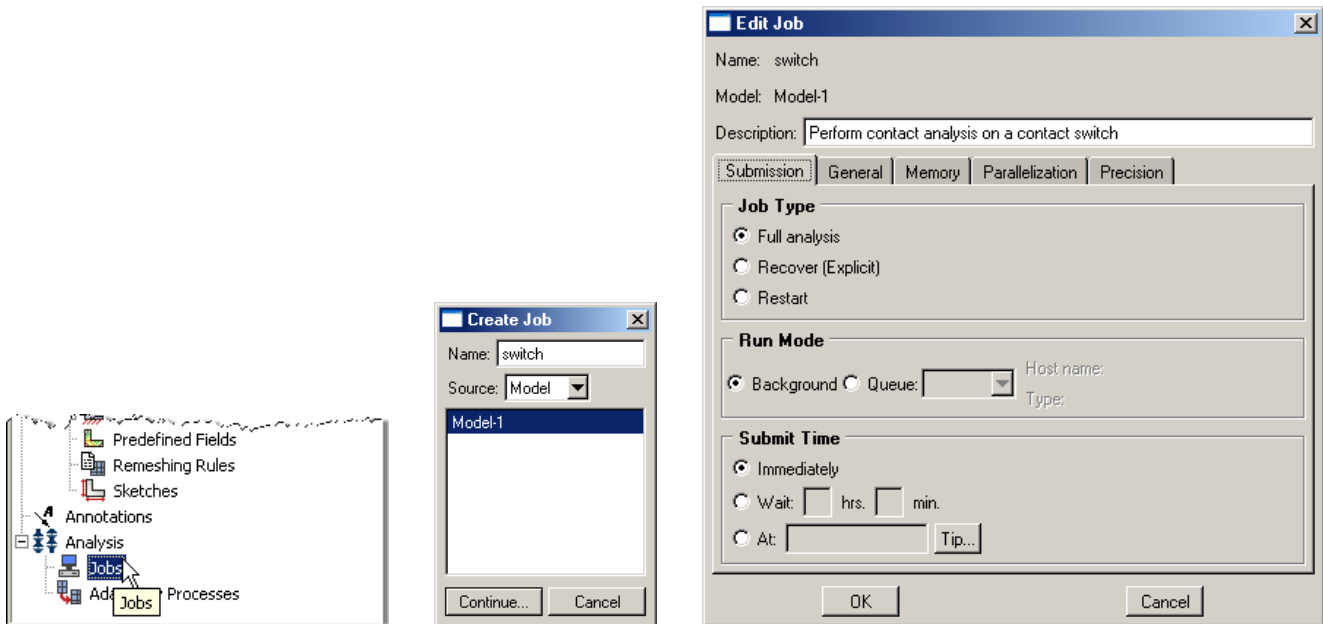


- b. Select the entire geometry, except for the vertical face
- c. Select “Done”

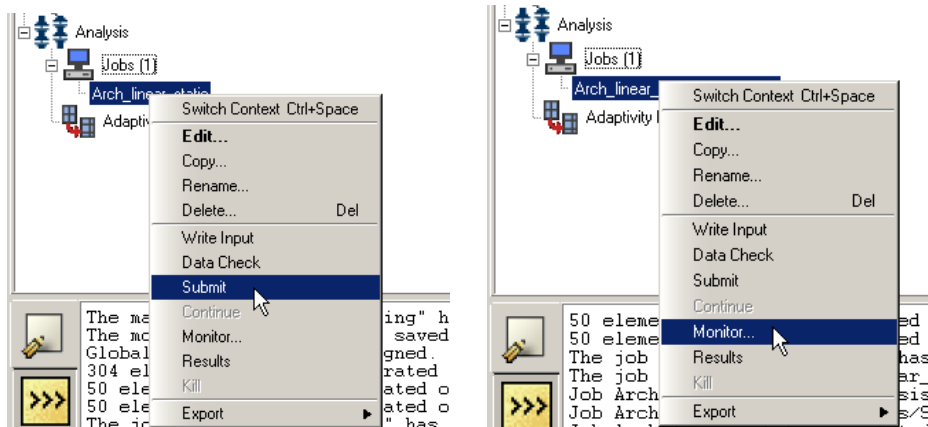


18. In the model tree double click on the “Job” node

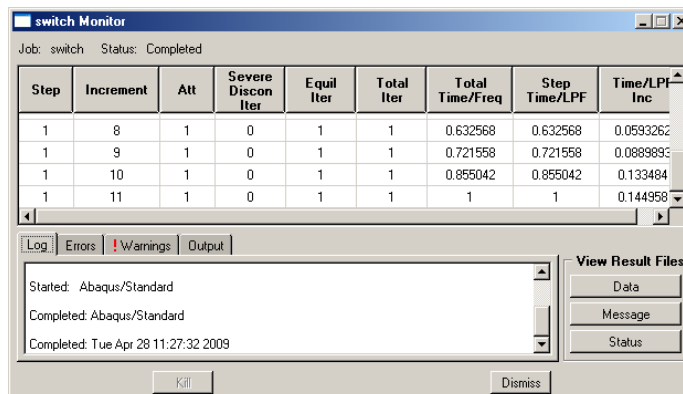
- a. Name the job “switch”
- b. Give the job a description



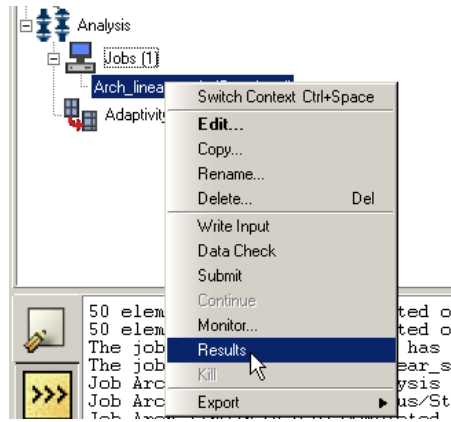
19. In the model tree right click on the job just created and select “Submit”
- d. Ignore the message about unmeshed portions of the geometry
 - e. While Abaqus is solving the problem right click on the job submitted, and select “Monitor”



- f. 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, and select “Results”



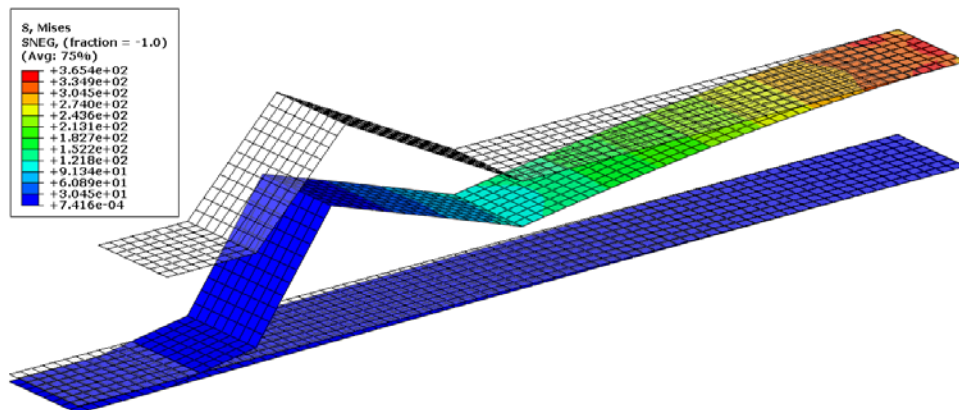
21. Display the deformed contour of the (Von) Mises stress overlaid with the undeformed geometry

- 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"



22. In the toolbox area click on the "Common Plot Options" icon

- a. Set the Deformation Scale Factor to 1
- b. Click "OK"



23. To change the output being displayed, in the menu bar click on Results → Field Output

- a. Select the contact pressure at surface nodes (CPRESS)
- b. Click "OK"

