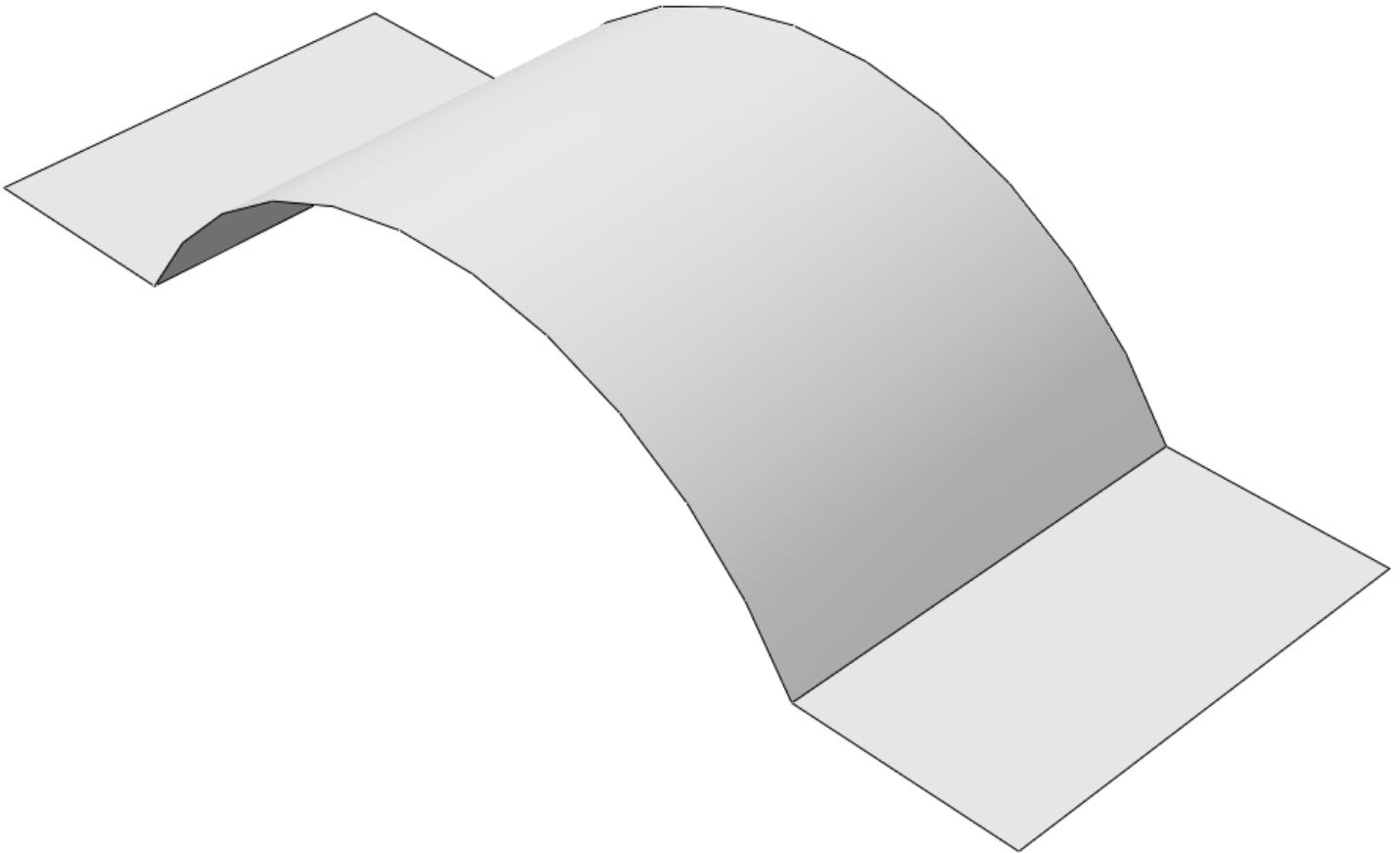


# ***Abaqus/CAE (ver. 6.11) Shell Tutorial***

## **Problem Description**

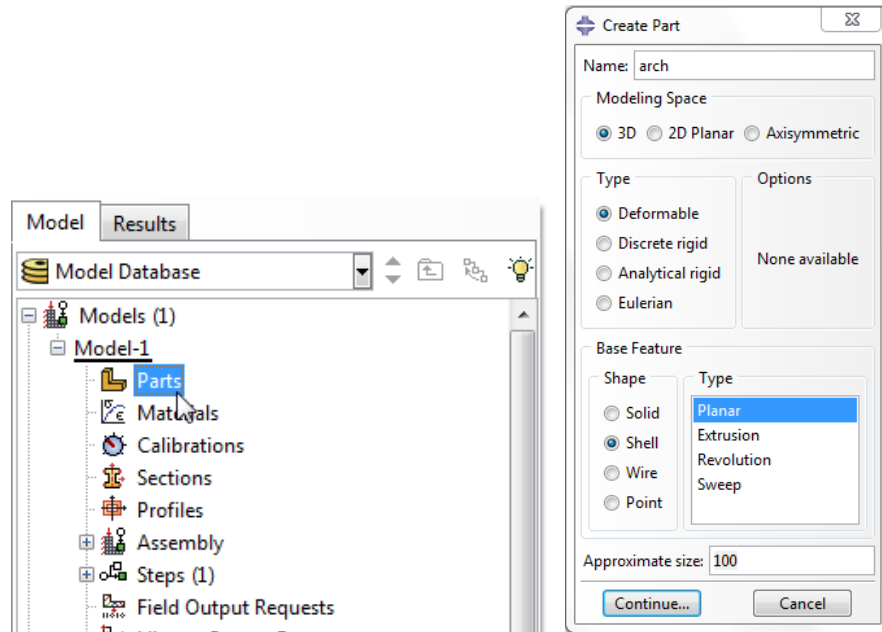
The aluminum arch ( $E = 70 \text{ GPa}$ ,  $\nu = 0.3$ ) shown below is completely clamped along the flat faces. The arch supports a pressure of 100 MPa.



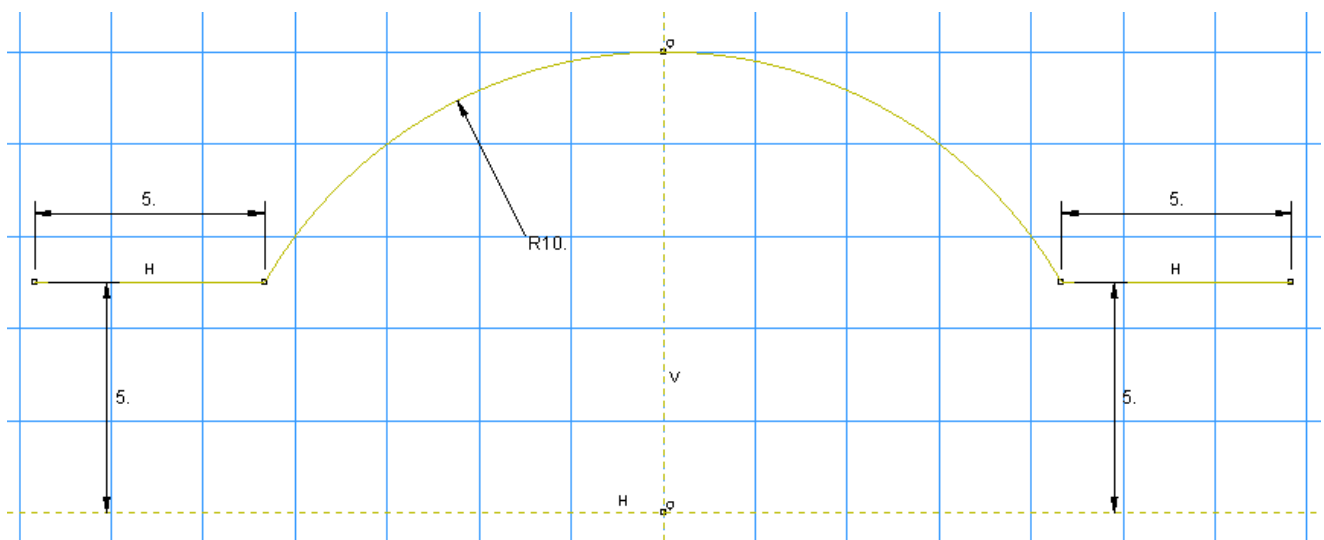
In this example, we also practice how to mesh a “portion” of geometry and how to avoid modeling unnecessary segments!

## Analysis Steps

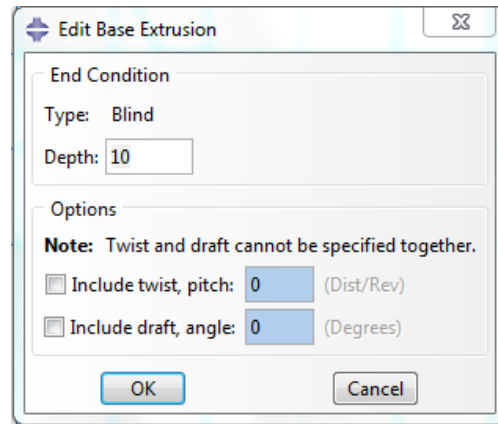
1. Start Abaqus and choose to create a new model database
2. In the model tree double click on the “Parts” node (or right click on “parts” and select Create)



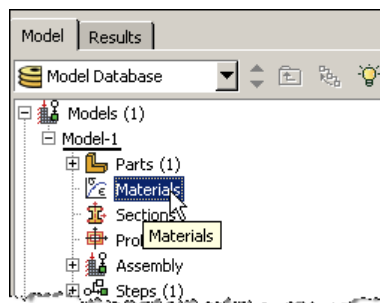
3. In the Create Part dialog box (shown above) name the part and
  - a. Select “3D”
  - b. Select “Deformable”
  - c. Select “Shell”
  - d. Select “Extrusion”
  - e. Set approximate size = 100
  - f. Click “Continue...”
4. Create the geometry shown below (not discussed here). Units shown are in mm.



- a. Click “Done”
- b. Set Depth = 10
- c. Click “OK”



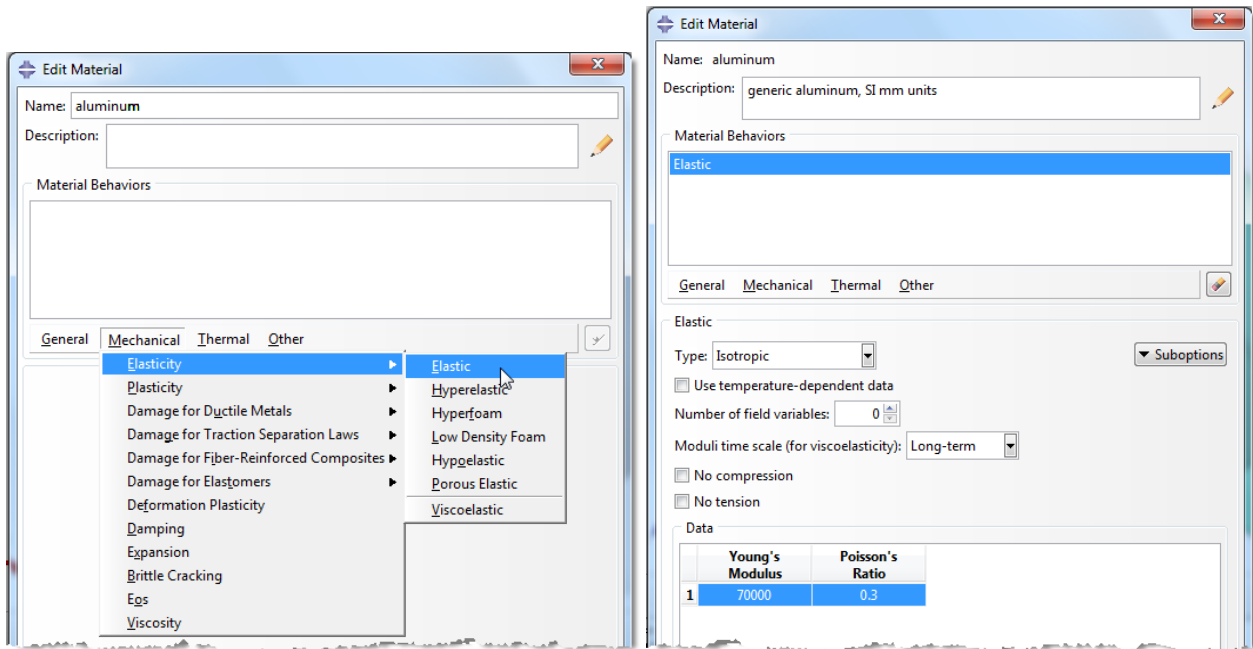
5. Double click on the “Materials” node in the model tree



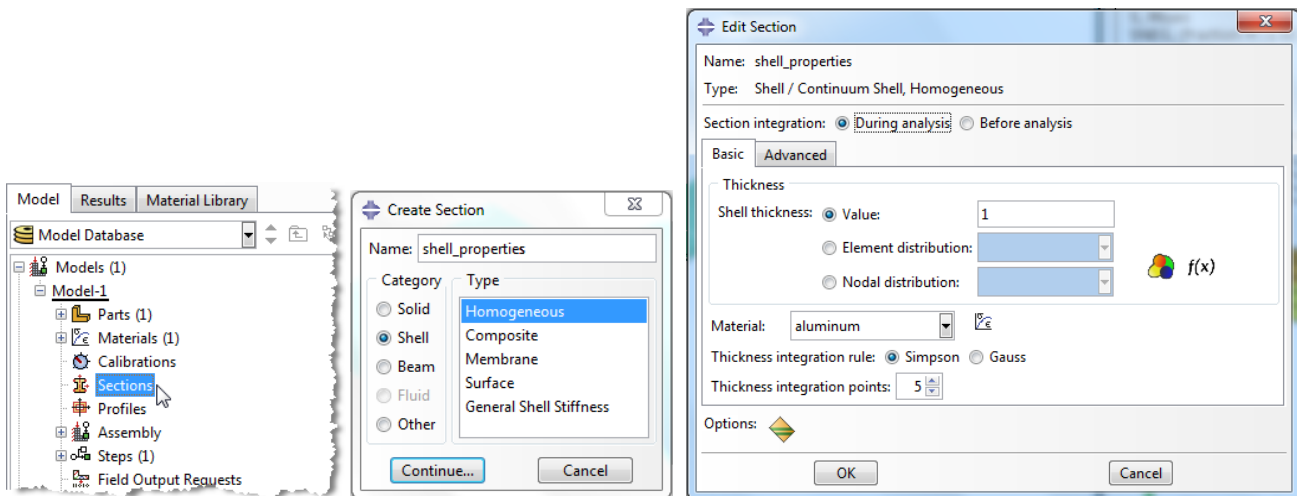
- a. Name the new material and give it a description
- b. Click on the “Mechanical” tab → Elasticity → Elastic
- c. Define Young’s Modulus and the Poisson’s Ratio (use SI (mm) units)
  - i. **WARNING: There are no predefined system of units within Abaqus, so the user is responsible for ensuring that the correct values are specified**
  - ii. **See the table of consistent units below**

Quantity	SI	SI (mm)	US Unit (ft)	US Unit (inch)
Length	m	mm	ft	in
Force	N	N	lbf	lbf
Mass	kg	tonne ( $10^3$ kg)	slug	$\text{lbf s}^2/\text{in}$
Time	s	s	s	s
Stress	$\text{Pa (N/m}^2\text{)}$	$\text{MPa (N/mm}^2\text{)}$	$\text{lbf/ft}^2$	$\text{psi (lbf/in}^2\text{)}$
Energy	J	$\text{mJ (}10^{-3}\text{ J)}$	ft lbf	in lbf
Density	$\text{kg/m}^3$	$\text{tonne/mm}^3$	$\text{slug/ft}^3$	$\text{lbf s}^2/\text{in}^4$

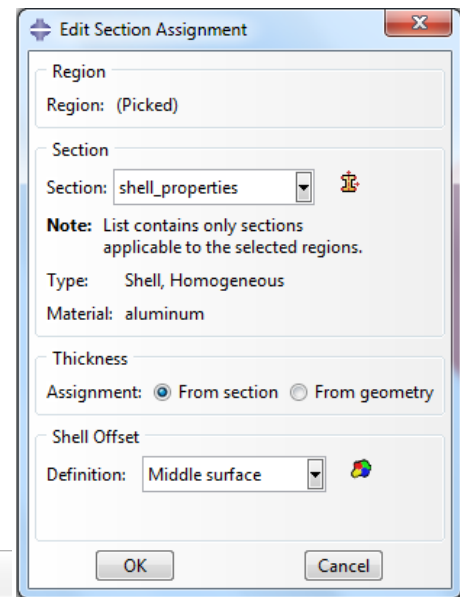
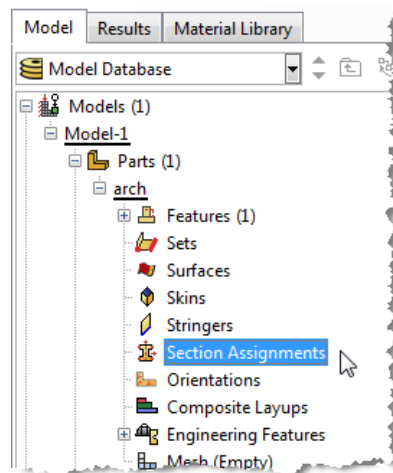
- d. Click “OK”



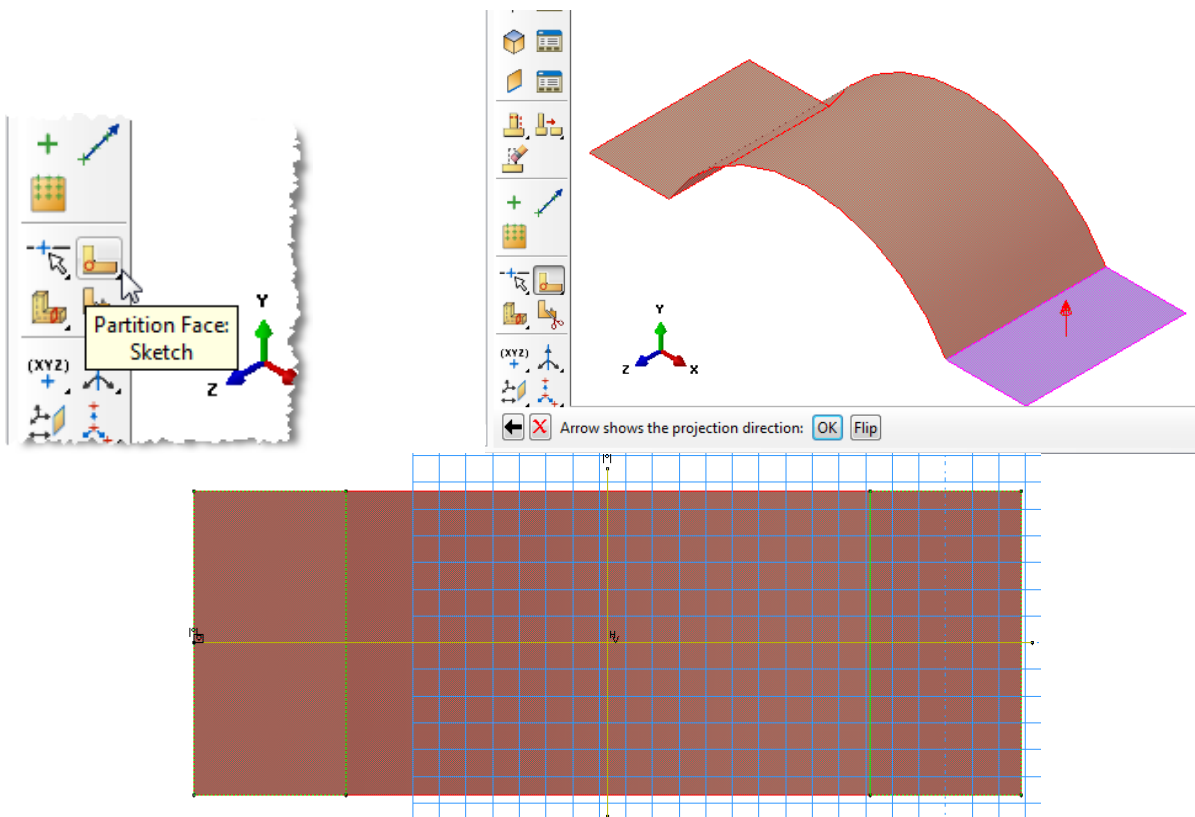
6. Double click on the "Sections" node in the model tree
  - a. Name the section "shell\_properties" and select "Shell" for the category and "Homogeneous" for the type
  - b. Click "Continue..."
  - c. Select the material created above (aluminum) and set the thickness to 1 (mm).
  - d. Adjust the thickness integration points if necessary
    - i. For Simpson integration the number of points must be odd and between 3 and 15
    - ii. For Gauss integration the number of points must be between 2 and 15
  - e. Click "OK"



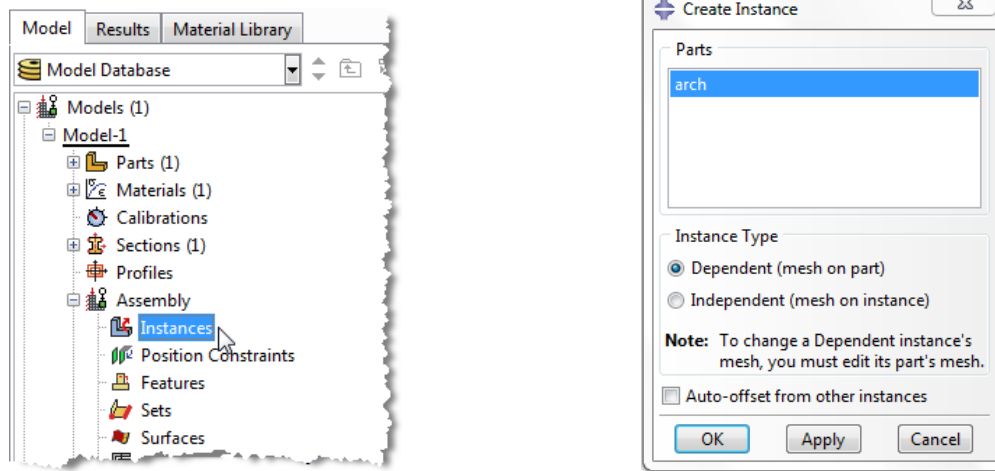
7. Expand the "Parts" node in the model tree, expand the node of the part just created, and double click on "Section Assignments"
  - a. Select the entire geometry in the viewport and press "Done" in the prompt area
  - b. Select the section created above (shell\_properties).
  - c. Specify shell offset if necessary. For this example use the default of "middle surface".
  - d. Click "OK"



8. In the toolbox area click on the “Partition Face: Sketch” icon
  - a. Select all faces and click “Done”
  - b. Select one of the flat faces as the sketch plane
  - c. Specify “Through All” for the projection distance. Note the arrow should encompass the entire part.
  - d. Select “Flip” if the arrow showing the project direction is incorrect, and/or press “OK”
  - e. Select one of the edges on the end of the part as the vertical sketch direction
  - f. Create a sketch that will divide the part into quarters. For example: draw a vertical line, select the equal distance constraint, pick the node at the upper right, pick the node at the upper left, then pick the drawn vertical line. The constraint will move the line to the midpoint.
  - g. Select “Done”

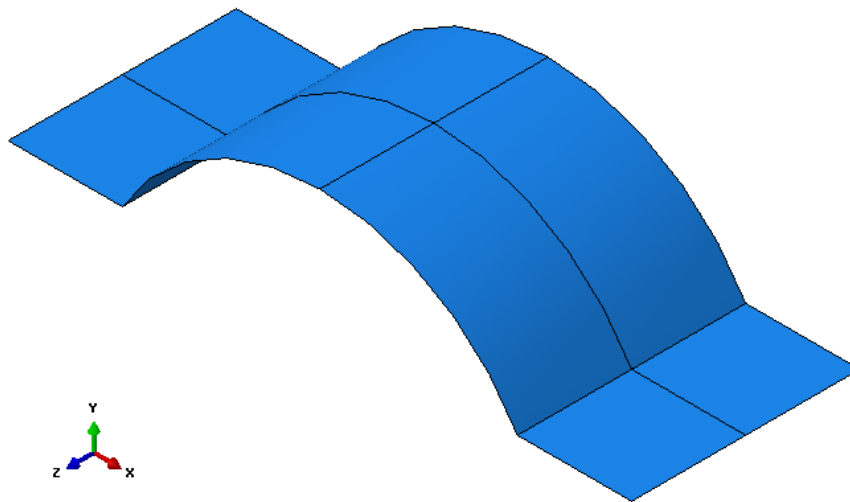


9. Expand the “Assembly” node in the model tree and then double click on “Instances”
  - a. Select “Dependent” for the instance type
  - b. Click “OK”

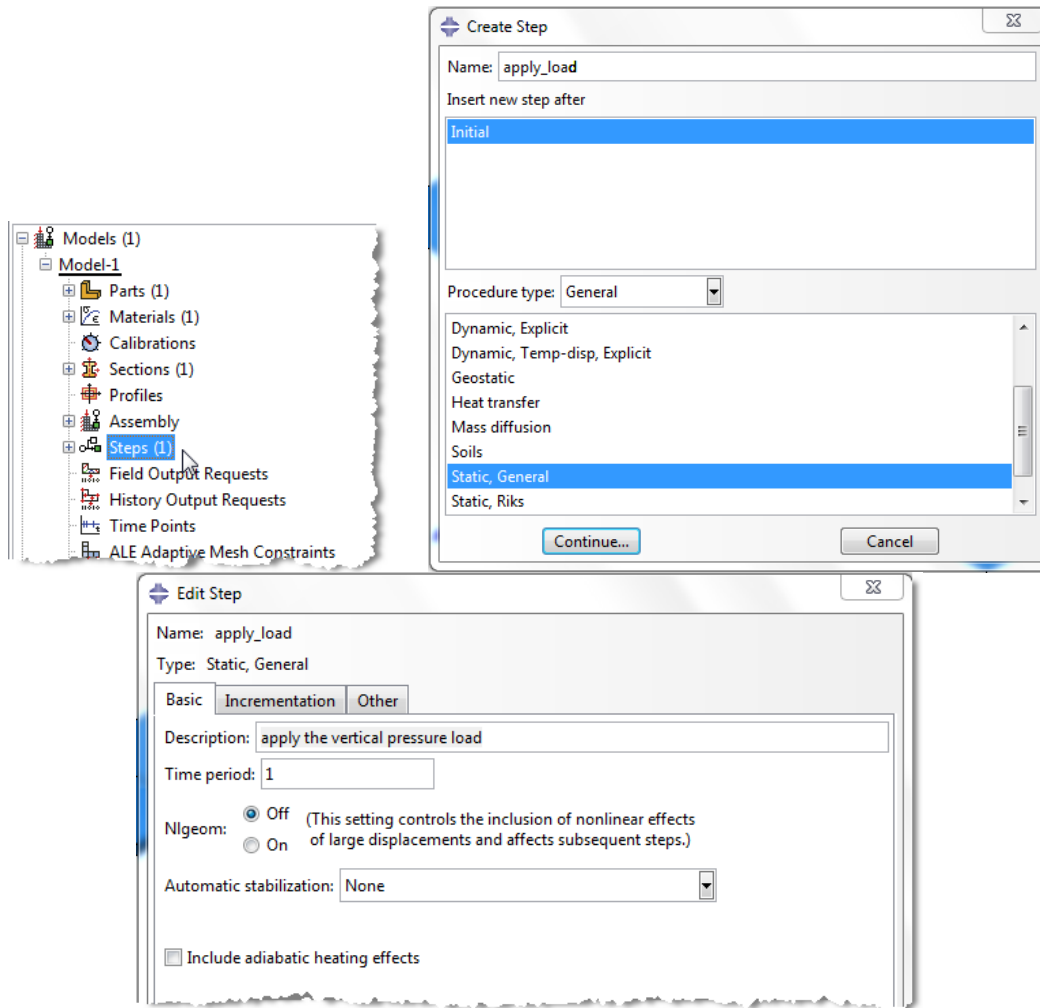


**10. Save the model**

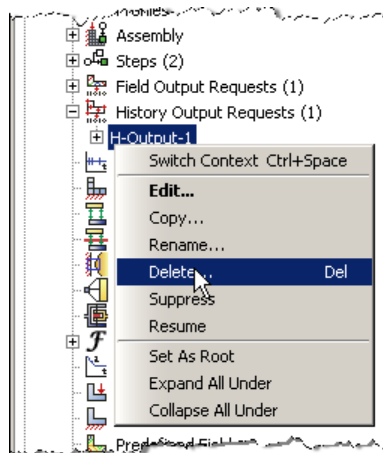
- a. This model will be used as a starting place for further tutorials



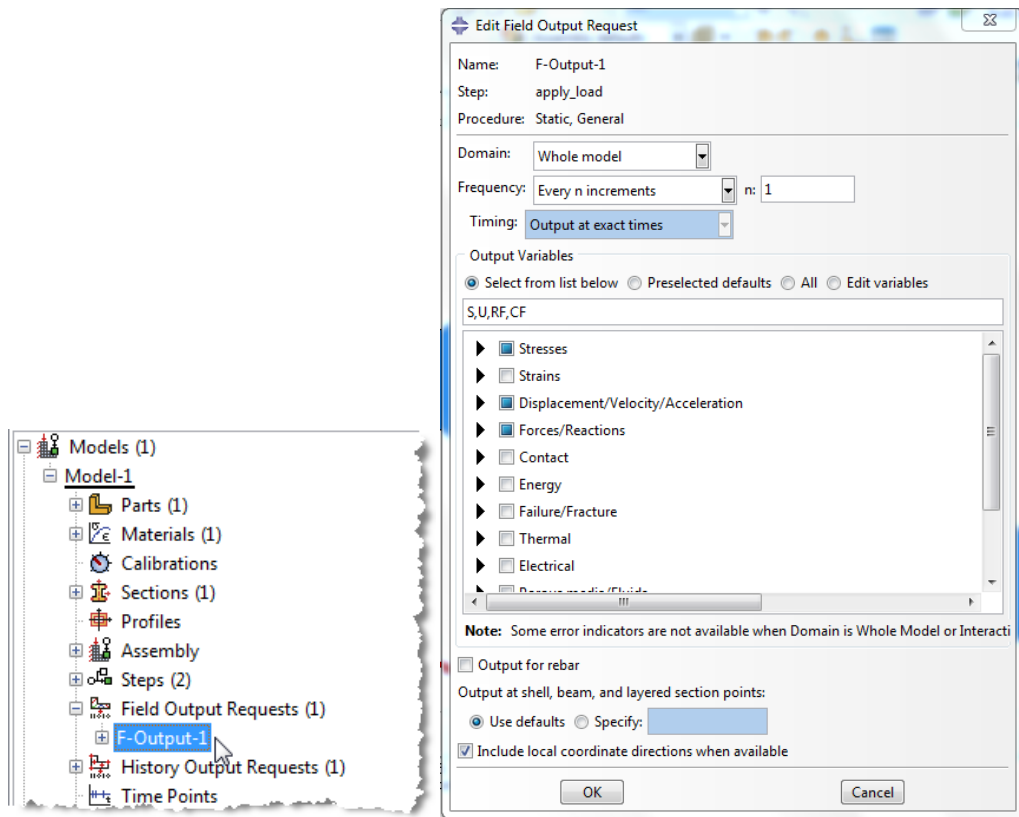
11. Double click on the “Steps” node in the model tree
  - a. Name the step, set the procedure to “General”, and select “Static, General”
  - b. Give the step a description



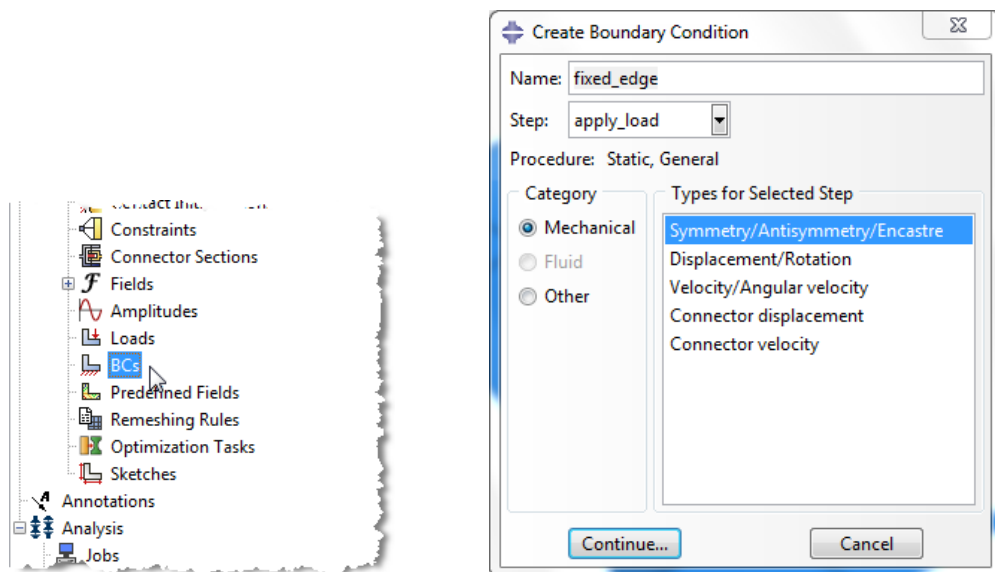
12. Expand the History Output Requests node in the model tree, and then right click on H-Output-1 (H-Output-1 was automatically generated when creating the step) and select Delete



13. 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”

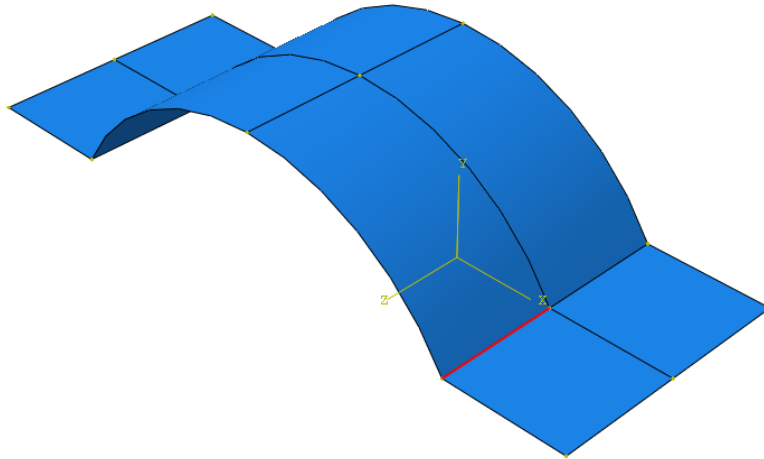


14. Because the part is symmetrical and the flat surfaces are fully restrained only a quarter of the arch needs to be modeled.
15. Because the flat surfaces are assumed to be fully restrained we do not need to include them, and can instead fix just the edge.
16. Double click on the “BCs” node in the model tree
  - a. Name the boundary conditioned “Fixed” and select “Symmetry/Antisymmetry/Encastre” for the type



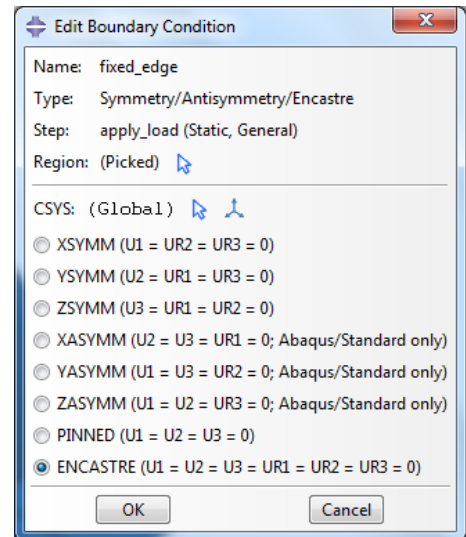


b. Select the edge shown below and click “Done”



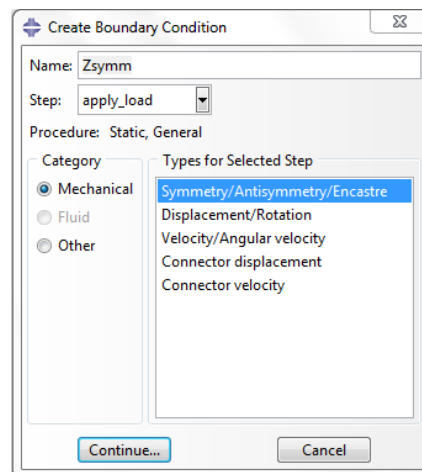
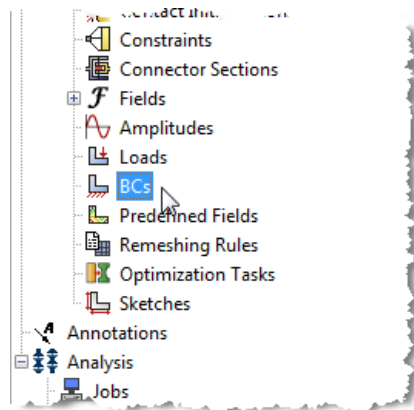
c. Select “ENCASTRE” for the boundary condition and click “OK”

**Note:** Restraining the entire surface will be inefficient, requiring unnecessary meshing of the portion of the geometry which will have no influence on the stiffness properties, and thus the result of simulation. Therefore, the restraint is applied to the shown edge to reduce the problem size. Noting this, the geometry creation could have been simplified right from the start!

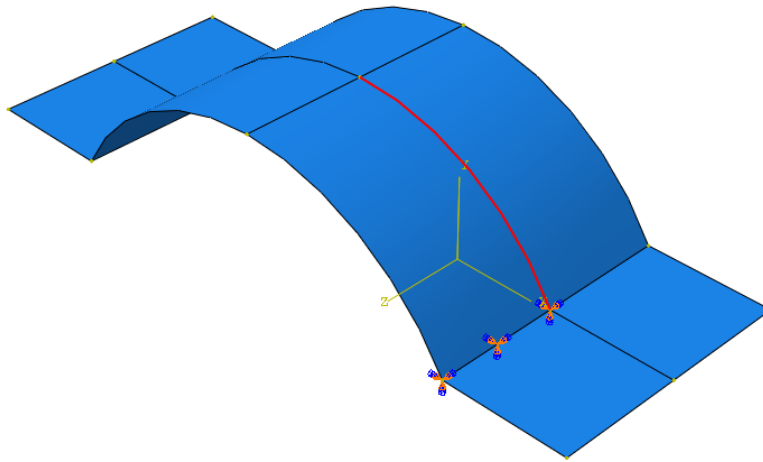


17. Double click on the “BCs” node in the model tree

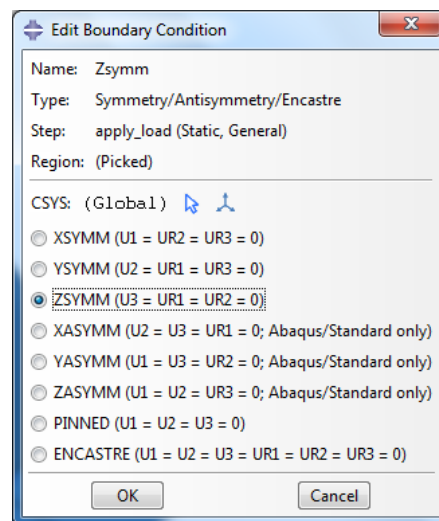
a. Name the boundary conditioned “Zsymm” and select “Symmetry/Antisymmetry/Encastre” for the type



b. Select the edge shown below and click “Done”



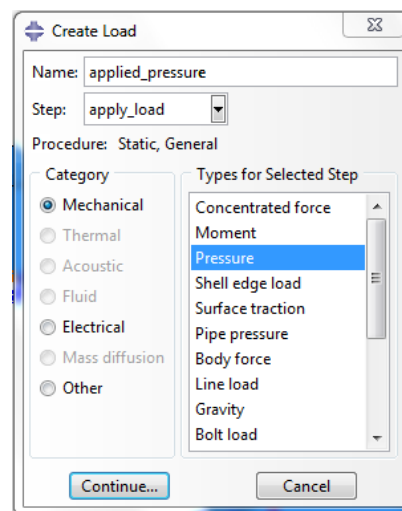
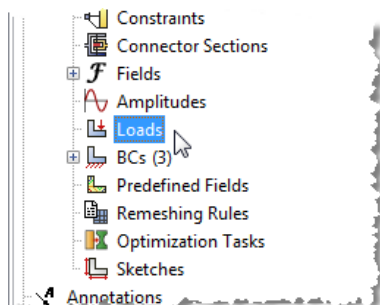
c. Select “ZSYMM” for the boundary condition



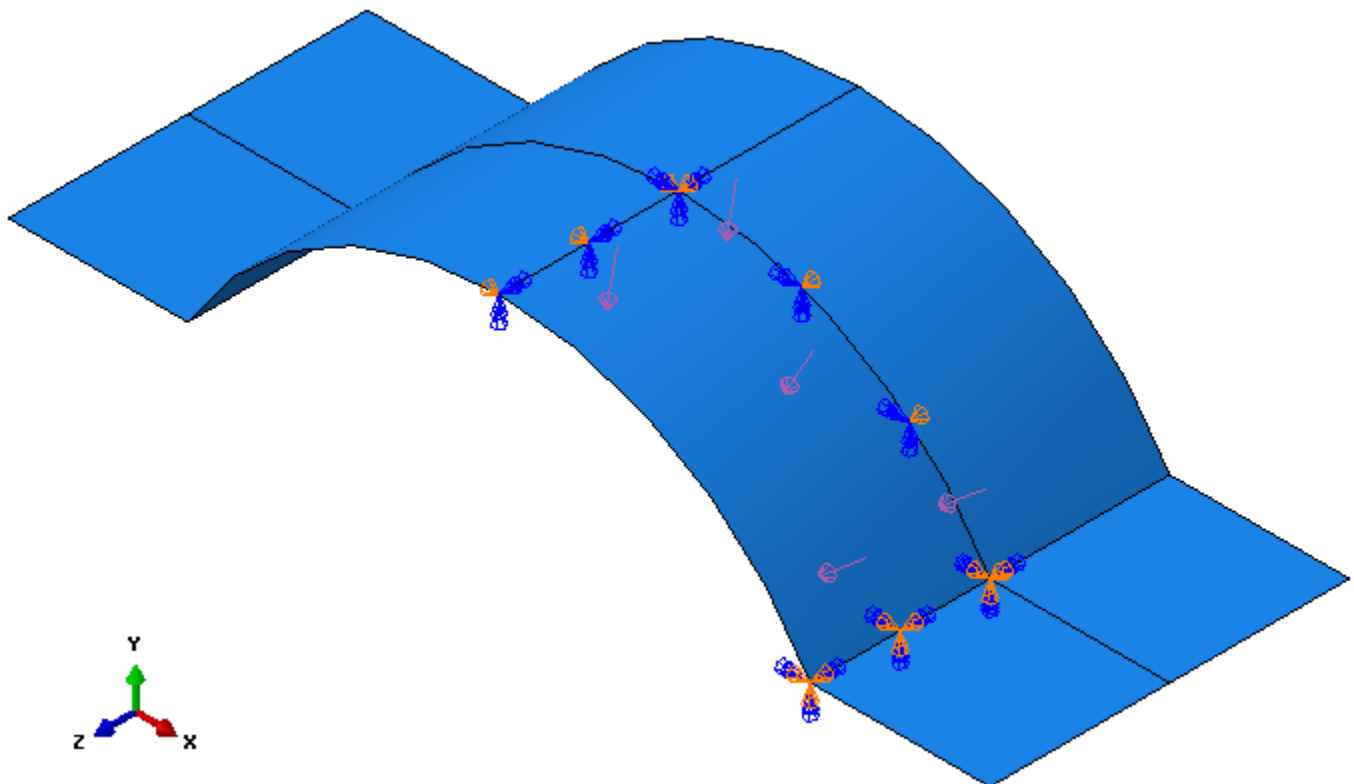
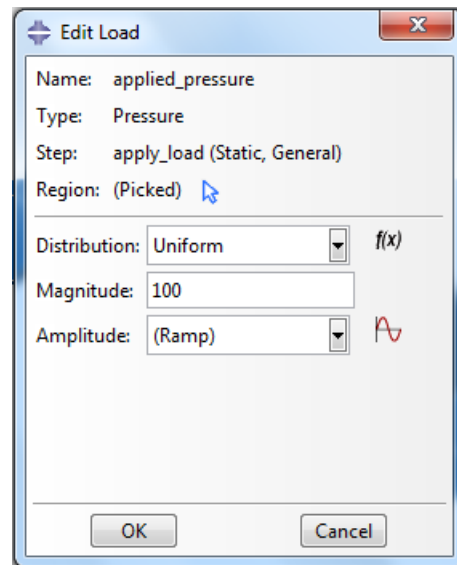
d. Repeat for the other edge and select “Xsymm” to apply x-dir symmetry condition.

18. Double click on the “Loads” node in the model tree

a. Name the load “Pressure” and select “Pressure” as the type

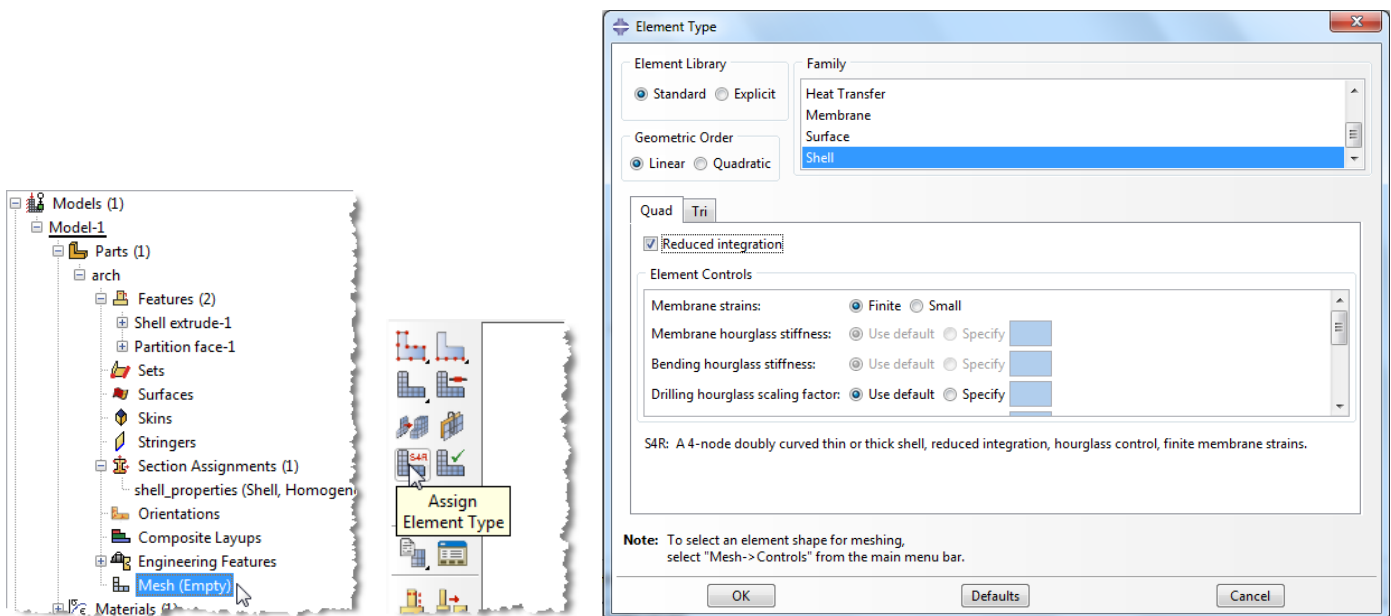


- b. Select the quarter of the arch surface with the boundary conditions applied to it
- c. Select the color corresponding to the top surface
- d. For the magnitude enter 600



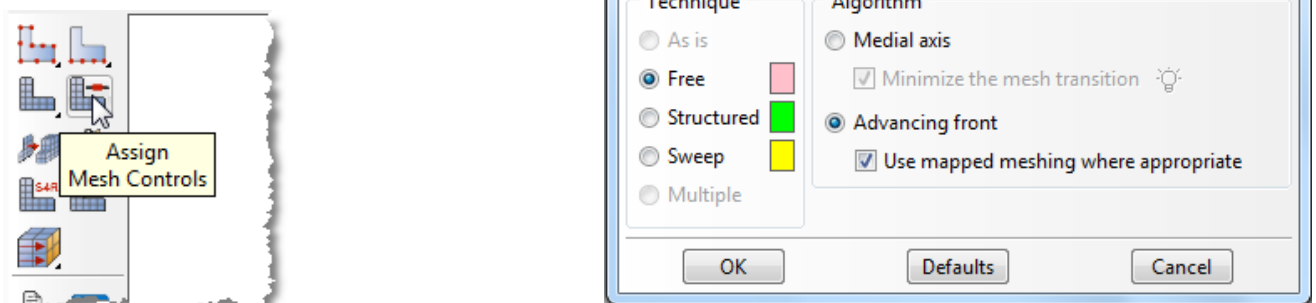
19. In the model tree double click on “Mesh” for the Arch part, and in the toolbox area click on the “Assign Element Type” icon

- 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”



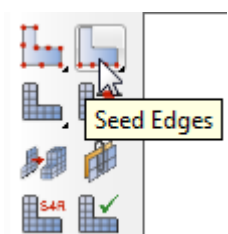
20. In the toolbox area click on the “Assign Mesh Controls” icon

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

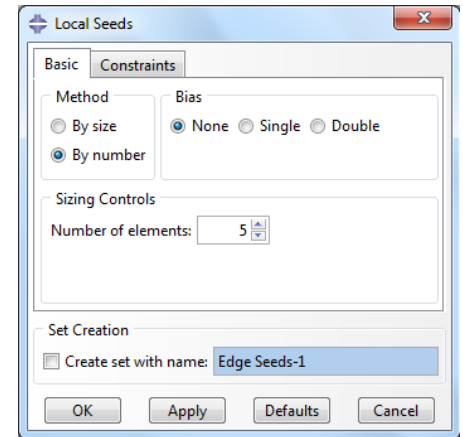
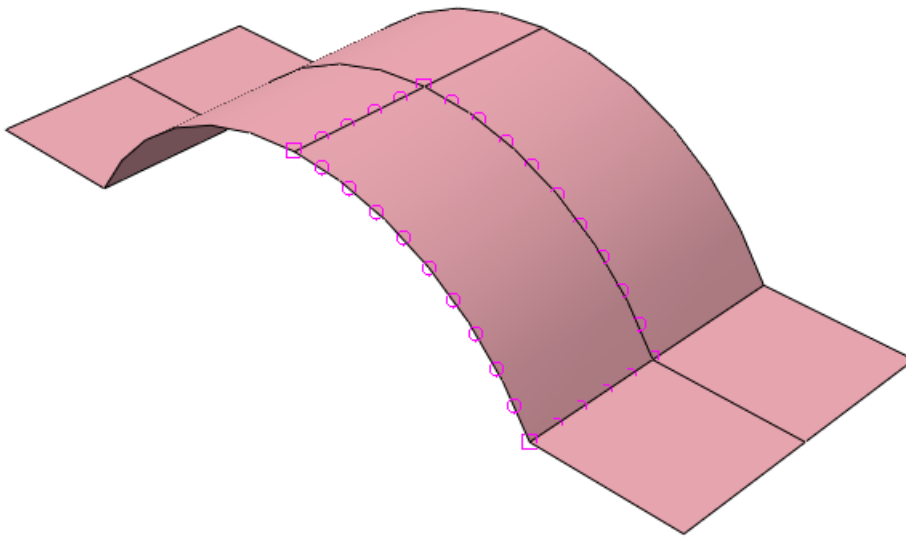


21. In the toolbox area click on the “Seed Edges” icon

- Select the shorter edges of the portion of the geometry associated with the boundary conditions and load
  - Select “By Number” method and Specify 5 elements

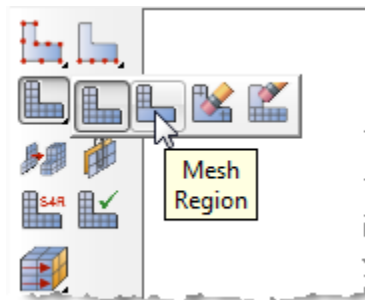


- b. Repeat step a. for the longer curved edges of the portion of the geometry associated with the boundary conditions and load
  - ii. Specify 10 elements

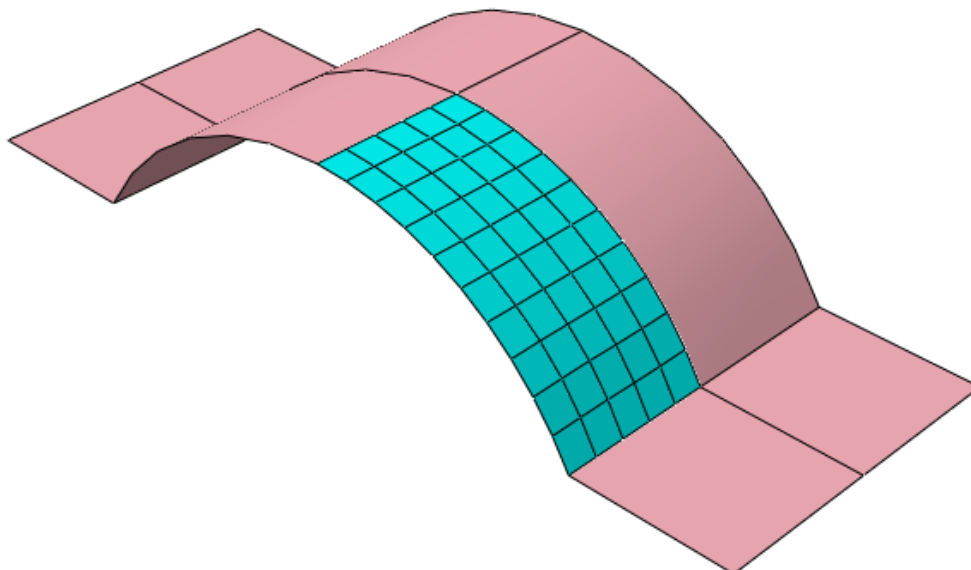


- c. Select "Done"

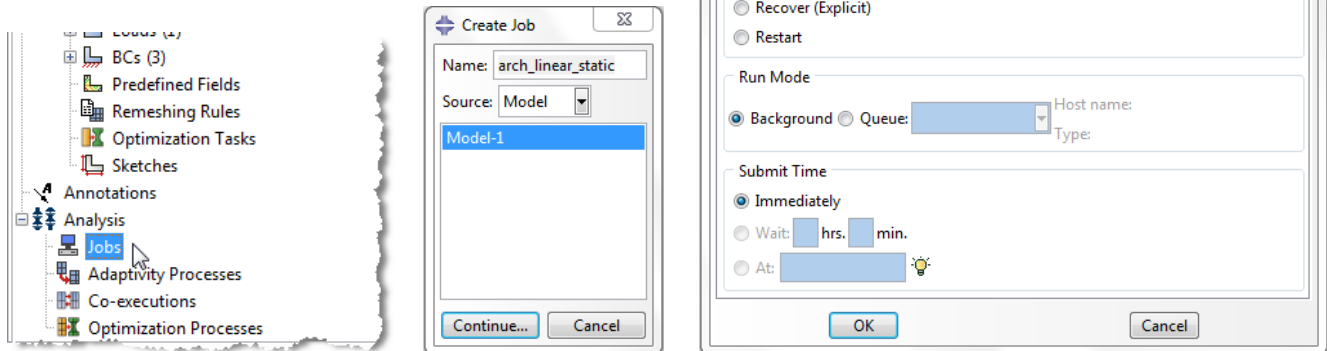
22. In the toolbox area click on the "Mesh Region" icon



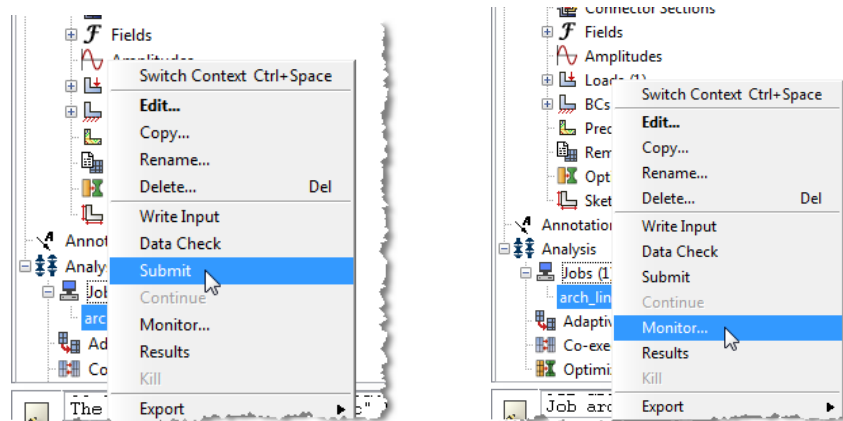
- d. Select the portion of the geometry associated with the boundary conditions and load
- e. Select "Done"



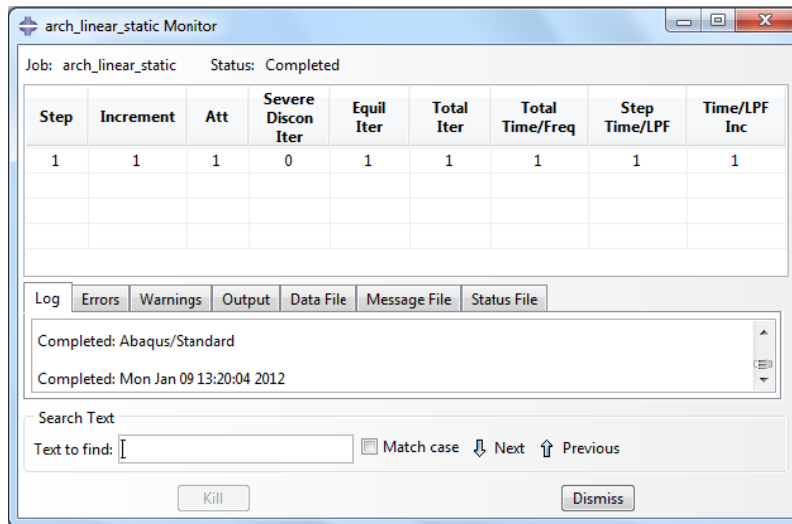
23. In the model tree double click on the “Job” node
  - a. Name the job “arch\_linear\_static”
  - b. Give the job a description



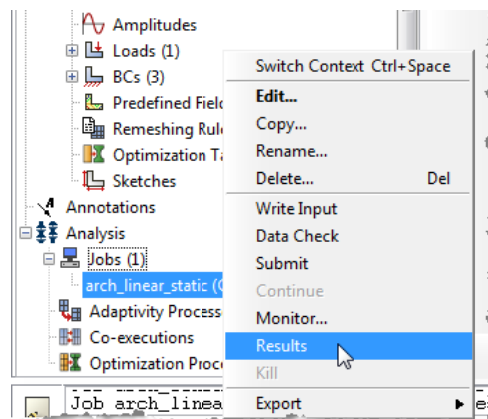
24. In the model tree right click on the job just created (arch\_linear\_static) and select “Submit”
  - f. Ignore the message about unmeshed portions of the geometry, click “yes” to continue.
  - g. While Abaqus is solving the problem right click on the job submitted (arch\_linear\_static), and select “Monitor”



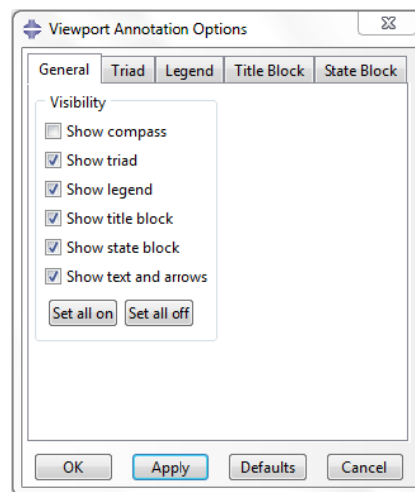
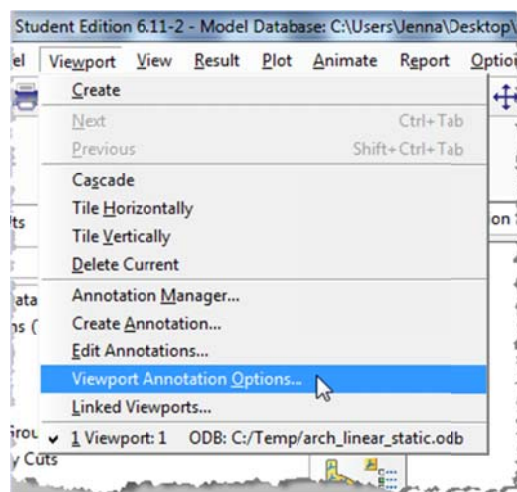
- h. In the Monitor window check that there are no errors or warnings
  - iii. If there are errors, investigate the cause(s) before resolving
  - iv. If there are warnings, determine if the warnings are relevant, some warnings can be safely ignored



25. In the model tree right click on the submitted and successfully completed job (arch\_linear\_static), and select “Results”



26. In the menu bar click on Viewport→Viewport Annotations Options
- Uncheck the “Show compass option”
  - The locations of viewport items can be specified on the corresponding tab in the Viewport Annotations Options



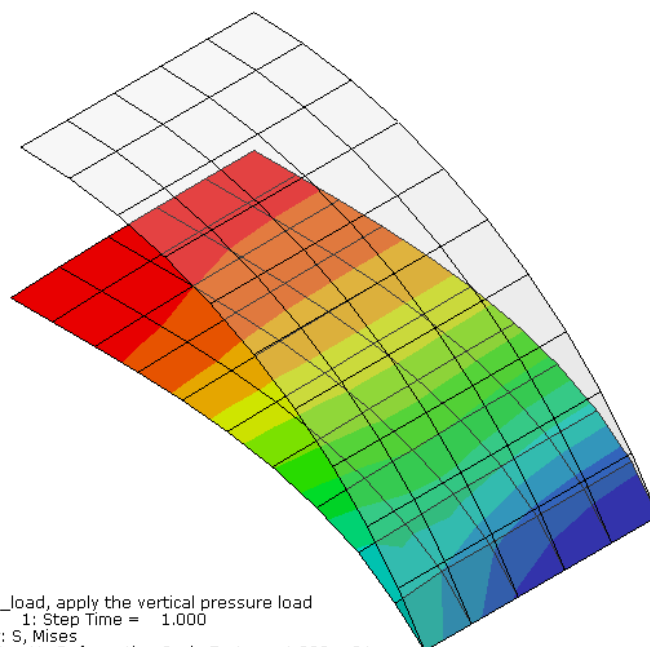
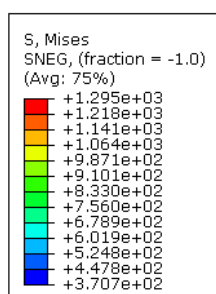
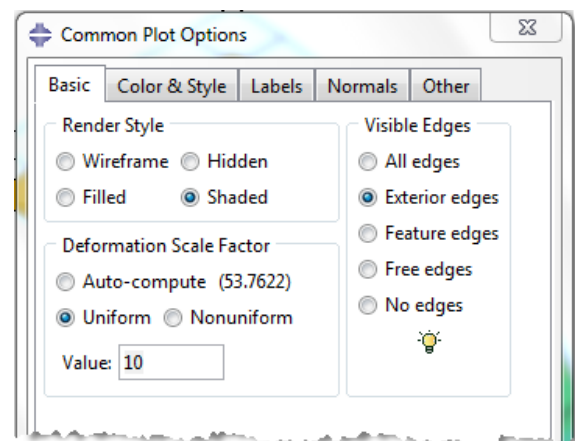
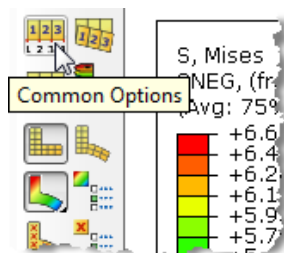
27. Display the deformed contour of the (Von) Mises stress overlaid with the undeformed geometry

- 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"



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

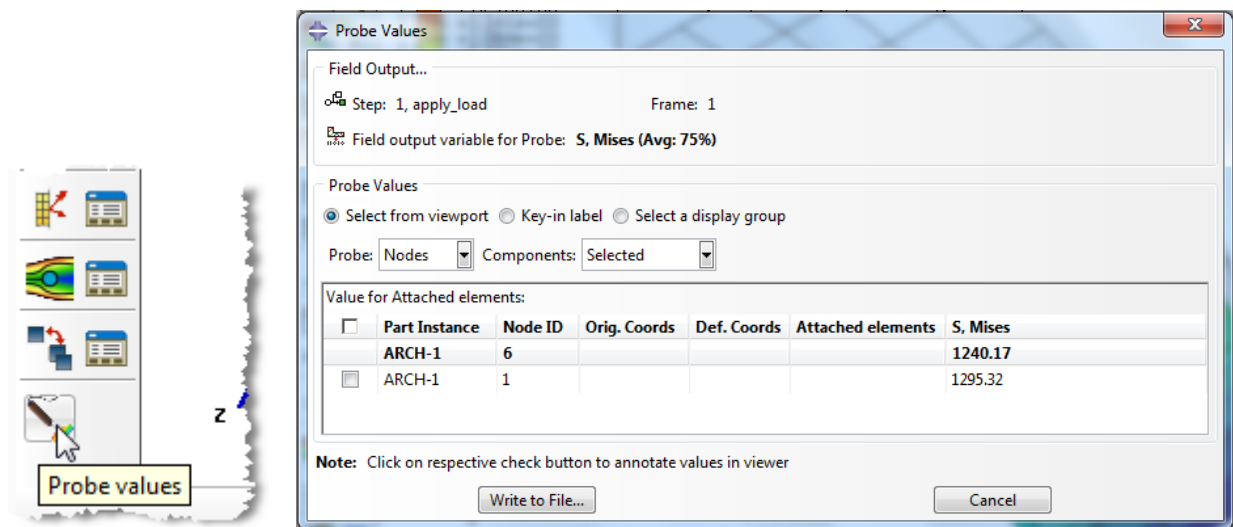
- a. Set the Deformation Scale Factor to 10
- b. Click "OK"



Step: apply\_load, apply the vertical pressure load  
 Increment 1: Step Time = 1.000  
 Primary Var: S, Mises  
 Deformed Var: U Deformation Scale Factor: +1.000e+01



29. To determine the stress values, click on the “probe values” icon
  - a. Set the probe to “Nodes”
  - 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



30. The field output tool bar can be used to change the output displayed
  - a. The middle drop down tab selects the field output of interest.
  - b. The right drop down is used to select the variant or component.

