

Abaqus/CAE (ver. 6.10) Material Nonlinearity Tutorial

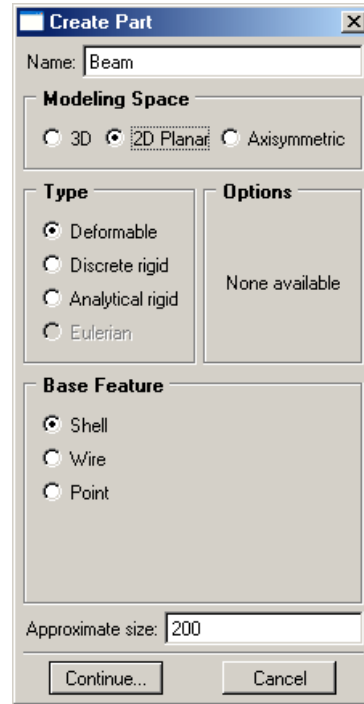
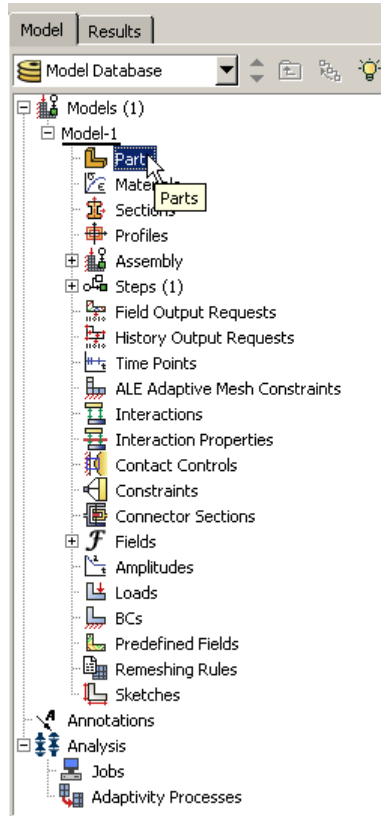
Problem Description

A rectangular steel cantilevered beam has a downward load applied to the one end. The load is expected to produce plastic deformation. An experimentally determined stress strain curve was supplied for the steel material. We will investigate the magnitude and depth of plastic strain.

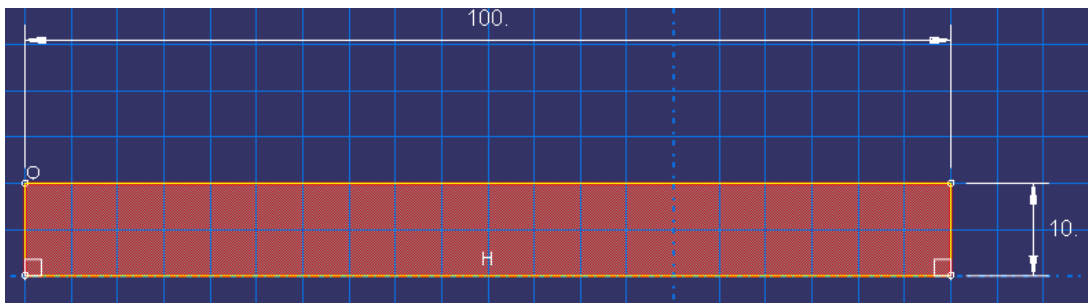


Analysis Steps

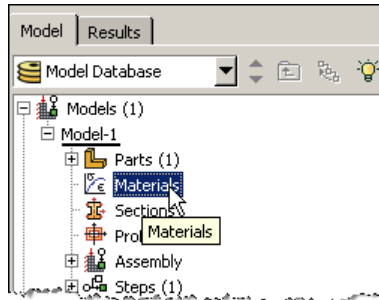
1. Start Abaqus and choose to create a new model database
2. In the model tree double click on the “Parts” node (or right click on “parts” and select Create)



3. In the Create Part dialog box (shown above) name the part and
 - a. Select “2D Planar”
 - b. Select “Deformable”
 - c. Select “Shell”
 - d. Set approximate size = 200
 - e. Click “Continue...”
4. Create the geometry shown below (not discussed here)

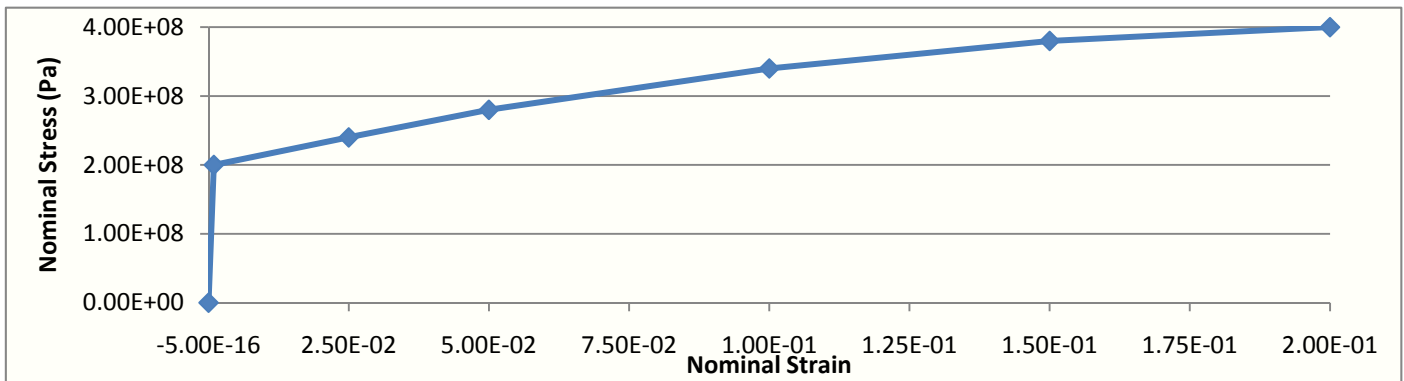


5. Double click on the “Materials” node in the model tree



- a. Name the new material and give it a description
- b. The stress strain data, shown below, was measured for the material used
 - i. This data is based on the nominal (engineering) stress and strain

Nominal Stress (Pa)	Nominal Strain
0.00E+00	0.00E+00
2.00E+08	9.50E-04
2.40E+08	2.50E-02
2.80E+08	5.00E-02
3.40E+08	1.00E-01
3.80E+08	1.50E-01
4.00E+08	2.00E-01

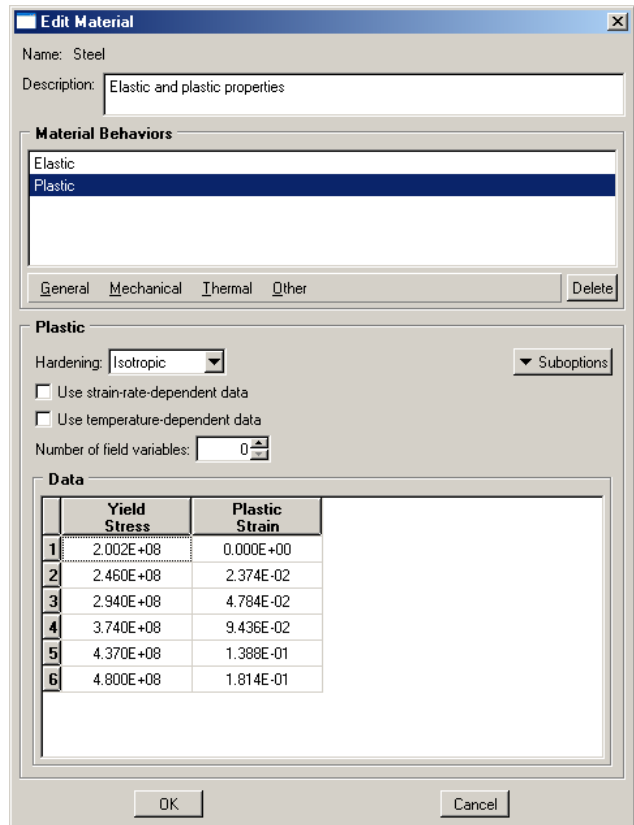
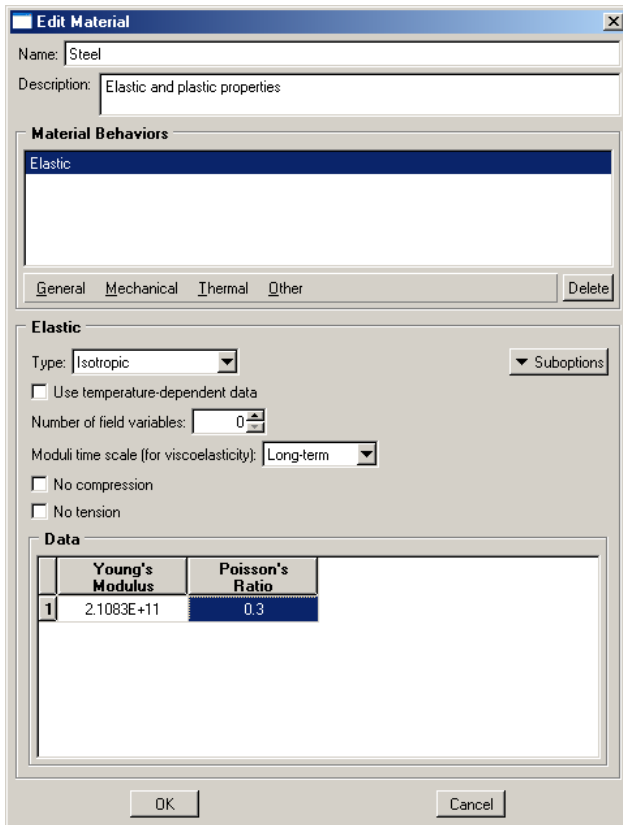


- ii. Abaqus expects the stress strain data to be entered as true stress and true plastic strain
 1. In addition the modulus of elasticity must correspond to the slope defined by the first point (the yield point)
- iii. To convert the nominal stress to true stress, use the following equation
 1. $\sigma_{tru} = \sigma_{nom}(1 + \epsilon_{nom})$
- iv. To convert the nominal strain to true strain, use the following equation
 1. $\epsilon_{tru} = \ln(1 + \epsilon_{nom})$
- v. To calculate the modulus of elasticity, divide the first nonzero true stress by the first nonzero true strain
- vi. To convert the true strain to true plastic strain, use the following equation
 1. $\epsilon_{pl} = \epsilon_{tru} - \frac{\sigma_{tru}}{E}$

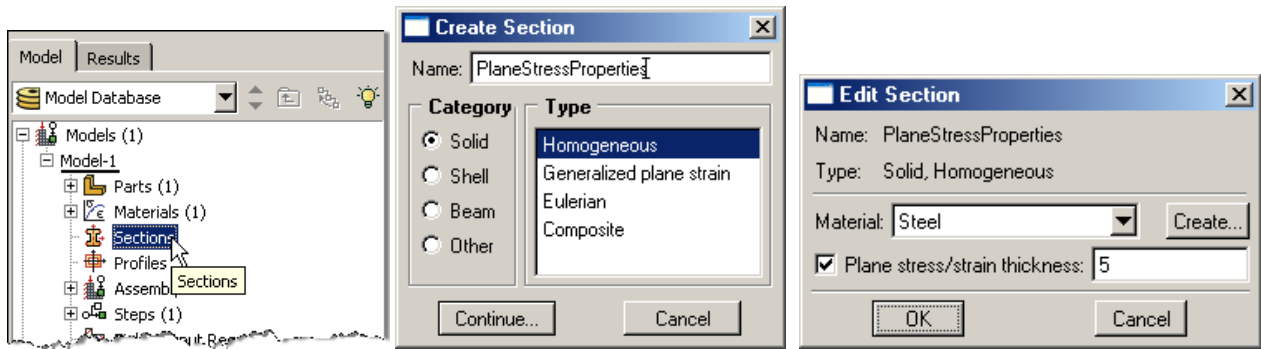
vii. The results should be

True Stress (Pa)	Plastic Strain	Elastic Modulus (Pa)
2.002E+08	0.000E+00	2.1083E+11
2.460E+08	2.374E-02	
2.940E+08	4.784E-02	
3.740E+08	9.436E-02	
4.370E+08	1.388E-01	
4.800E+08	1.814E-01	

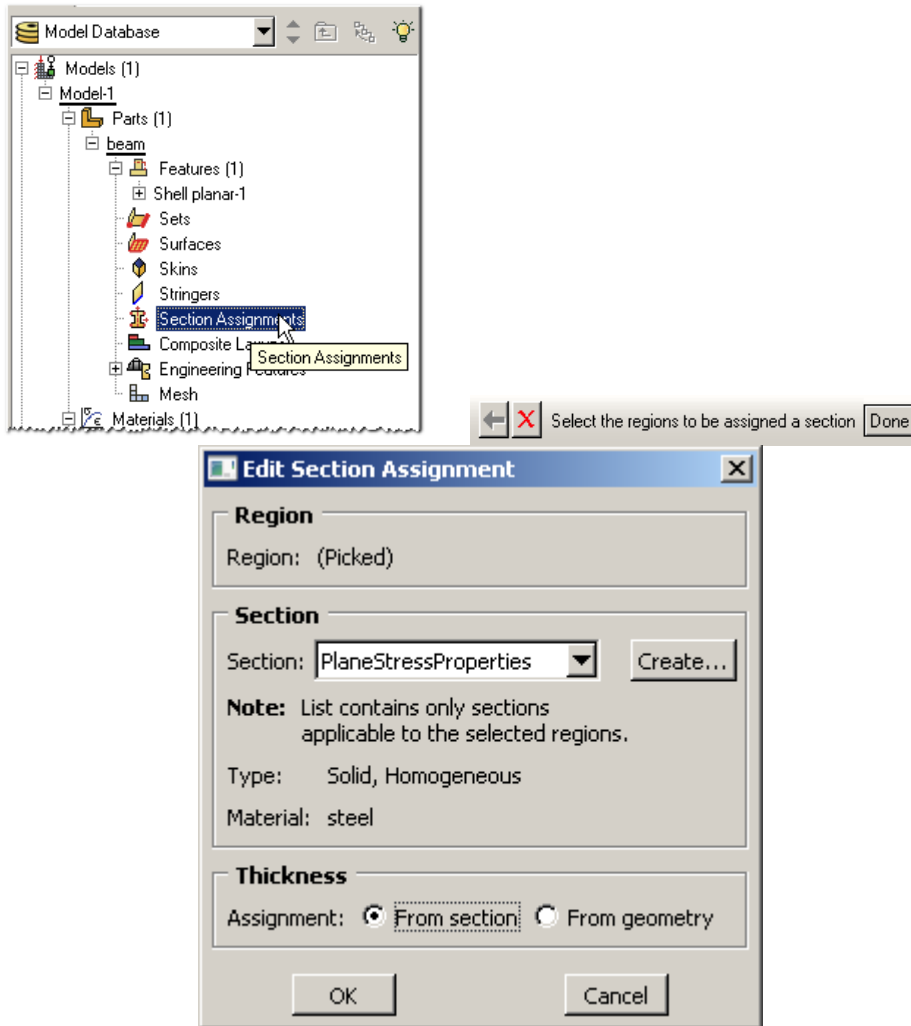
- c. Click on the “Mechanical” tab → Elasticity → Elastic
 - i. Enter the calculated modulus of elasticity, and Poisson’s ratio of 0.3
- d. Click on the “Mechanical” tab → Plasticity → Plastic
 - i. Enter the calculated true stress and plastic strain
 1. Note that you can simply copy your calculated values from Excel (or similar) and paste them into Abaqus
- e. Click “OK”



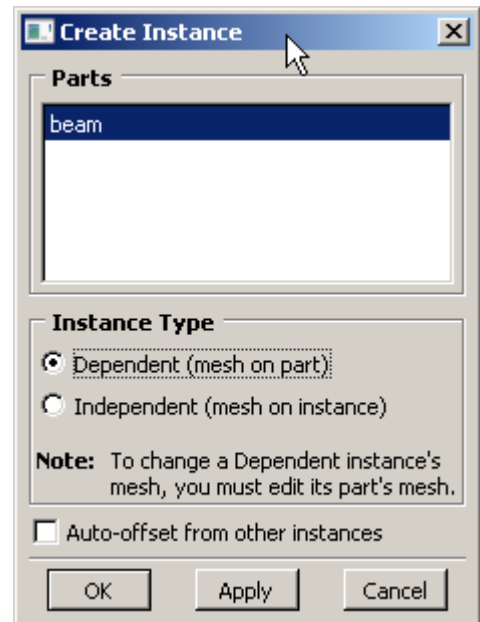
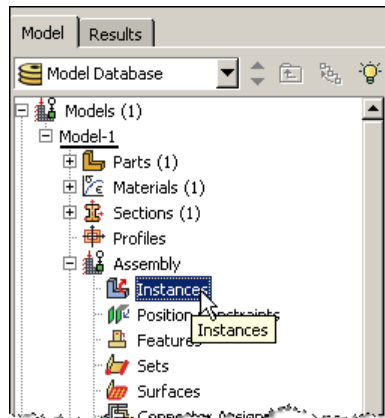
6. Double click on the “Sections” node in the model tree
 - a. Name the section “PlaneStressProperties” and select “Solid” for the category and “Homogeneous” for the type
 - b. Click “Continue...”
 - c. Select the material created above (Steel) and **set the thickness to 5.**
 - d. Click “OK”



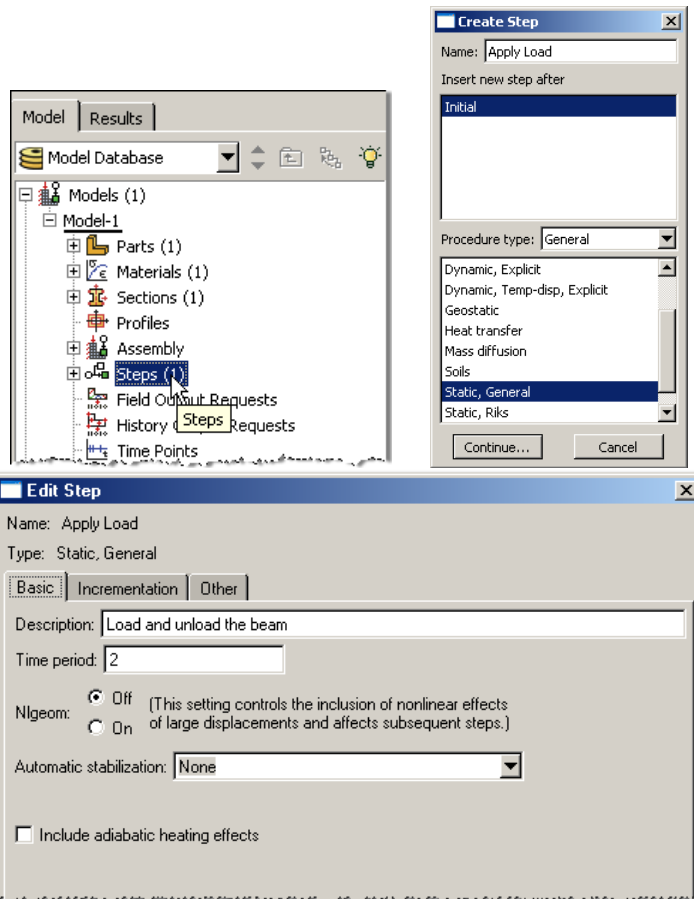
7. Expand the “Parts” node in the model tree, expand the node of the part just created, and double click on “Section Assignments”
 - a. Select the entire geometry in the viewport and press “Done” in the prompt area
 - b. Select the section created above (PlaneStressProperties)
 - c. Verify “From section” is selected under “Thickness”
 - d. Click “OK”



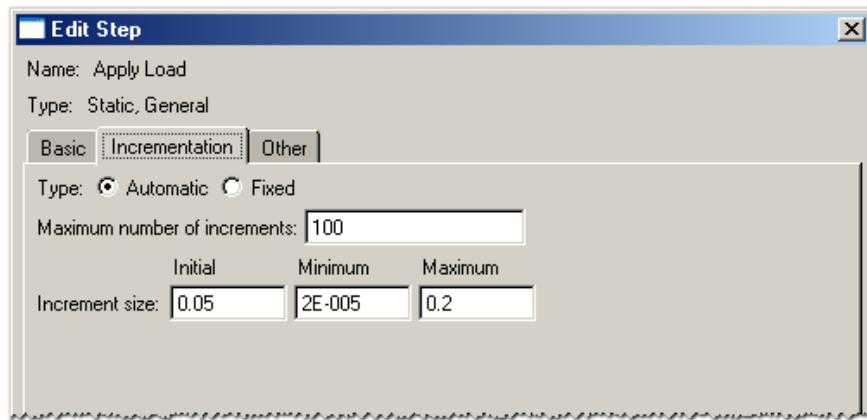
8. Expand the “Assembly” node in the model tree and then double click on “Instances”
 - a. Select “Dependent” for the instance type
 - b. Click “OK”



9. Double click on the “Steps” node in the model tree
 - a. Name the step, set the procedure to “General”, and select “Static, General”
 - b. On the Basic tab, give the step a description and **change the time period to 2**
 - i. For this analysis neglect the effects of geometric nonlinearities (Nlgeom = Off)



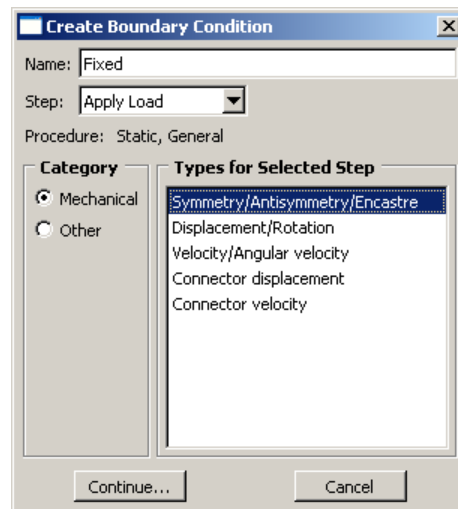
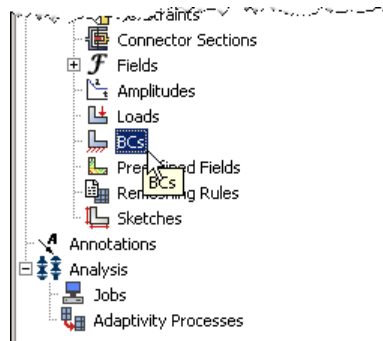
- c. On the Incrementation tab,
 - i. Set the **initial increment size to 0.05**
 - ii. Set the **maximum increment size to 0.2**



d. Click "OK"

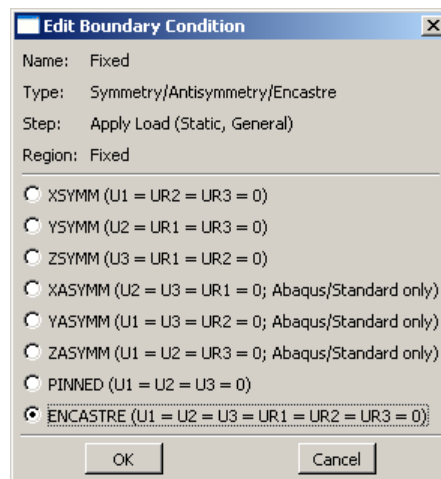
10. Double click on the "BCs" node in the model tree

a. Name the boundary conditioned "Fixed" and select "Symmetry/Antisymmetry/Encastre" for the type



b. Select the left edge and click "Done"

c. Select "ENCASTRE" for the boundary condition and click "OK"

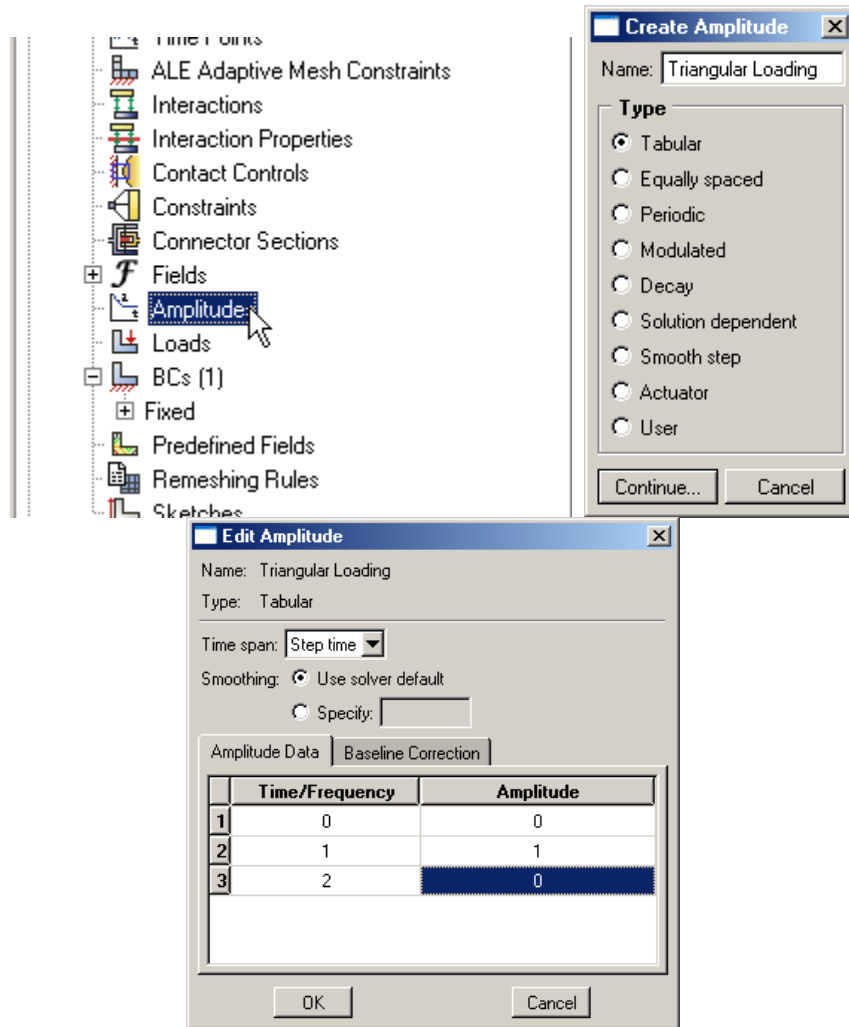


11. Double click on the "Amplitudes" node in the model tree

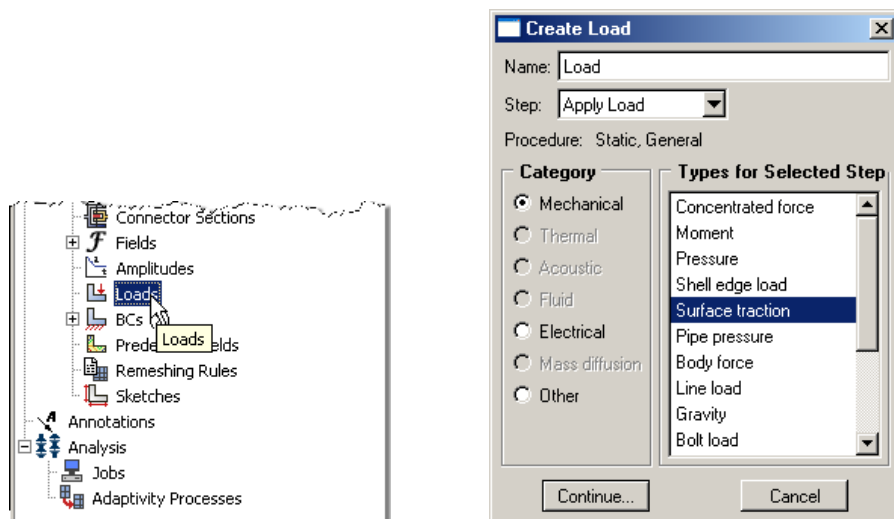
a. Name the amplitude "Triangular Loading" and select "Tabular"

b. Enter the data points shown below

- i. Abaqus multiplies the load by the amplitude definition, therefore 0 is no load and 1 is the full load

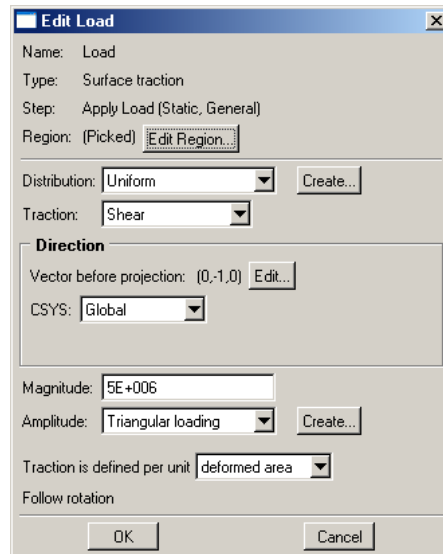


12. Double click on the “Loads” node in the model tree
 a. Name the load and select “Surface traction” as the type

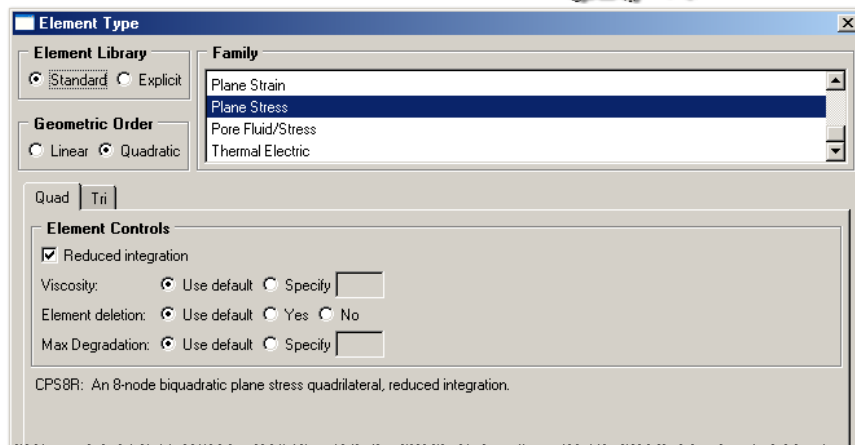
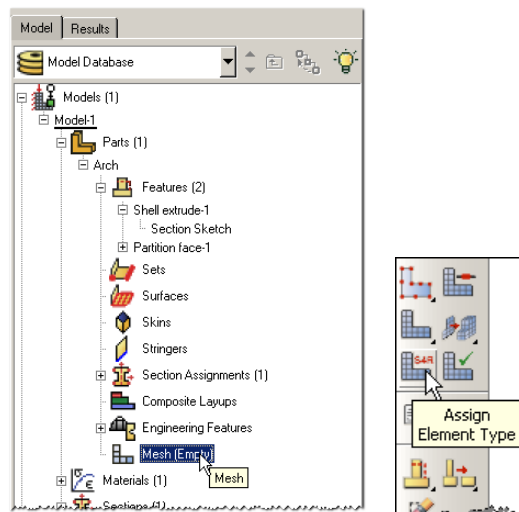


- b. Select the right edge

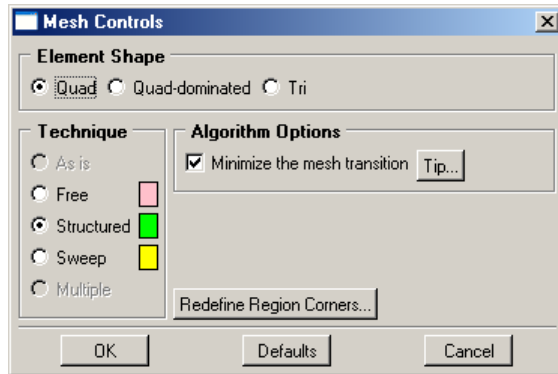
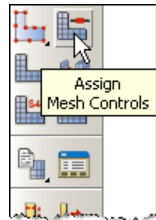
- c. Under Direction, click edit and select the upper-right corner as the first point, and the lower-right corner as the second point
- d. For the magnitude, enter 5e6
- e. For the amplitude, select the amplitude created above (Triangular loading)



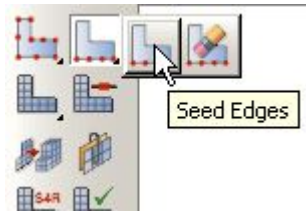
13. In the model tree double click on “Mesh” for the beam part, and in the toolbox area click on the “Assign Element Type” icon
 - a. Select the entire geometry
 - b. Select “Standard” for element type
 - c. Select “Quadratic” for geometric order
 - d. Select “Plane stress” for family
 - e. Note that the name of the element (S4R) and its description are given below the element controls
 - f. Select “OK”



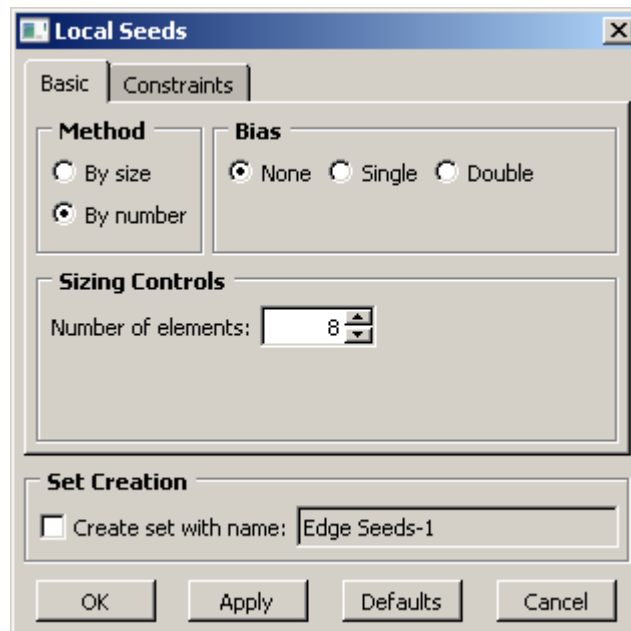
14. In the toolbox area click on the “Assign Mesh Controls” icon
 - a. Select the portion of the geometry associated with the boundary conditions and load
 - b. Change the element shape to “Quad”
 - c. Set the technique to “Structured”



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

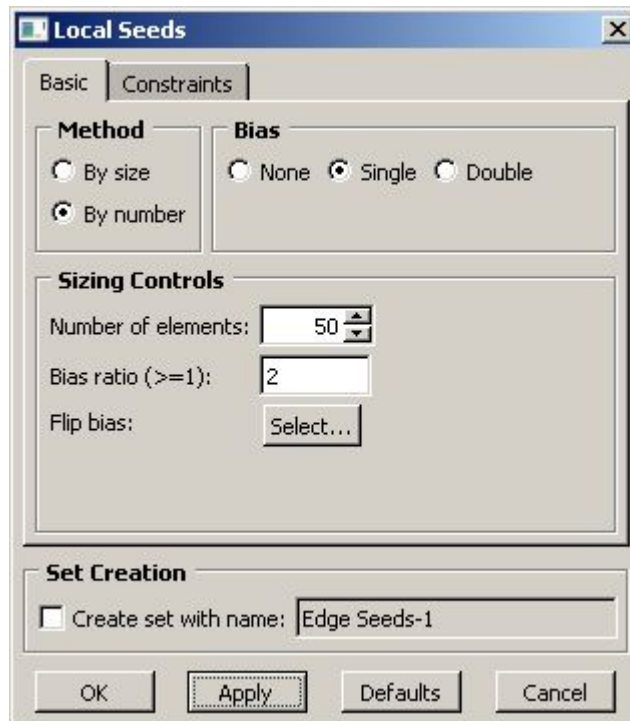


- a. Select the left and right edges, click “Done”
- b. Select “By number”
- c. Set “Bias” to “None”
- d. Under “Sizing Controls” enter 8 elements, Click “OK”

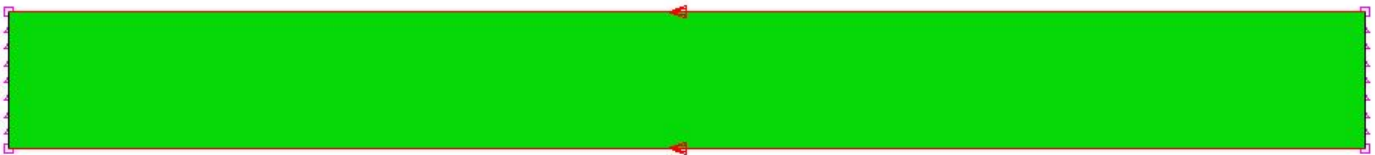


16. In the toolbox area ensure the “Seed Edges” icon is still selected

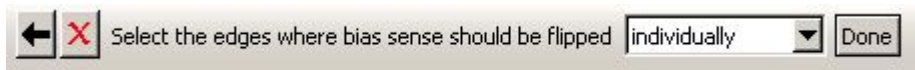
- a. Select the top and bottom edges
- b. Set “Method” to “By number” and “Bias” to “Single”
- c. Set the number of elements to 50
- d. Set the bias ratio to 2



- e. The bias arrows point towards the direction of the smaller elements, so in this case they should point to the left. If they don't, click the "Select" button located to the right of "Flip Bias"



- f. Select the top and bottom edges and select "Done"



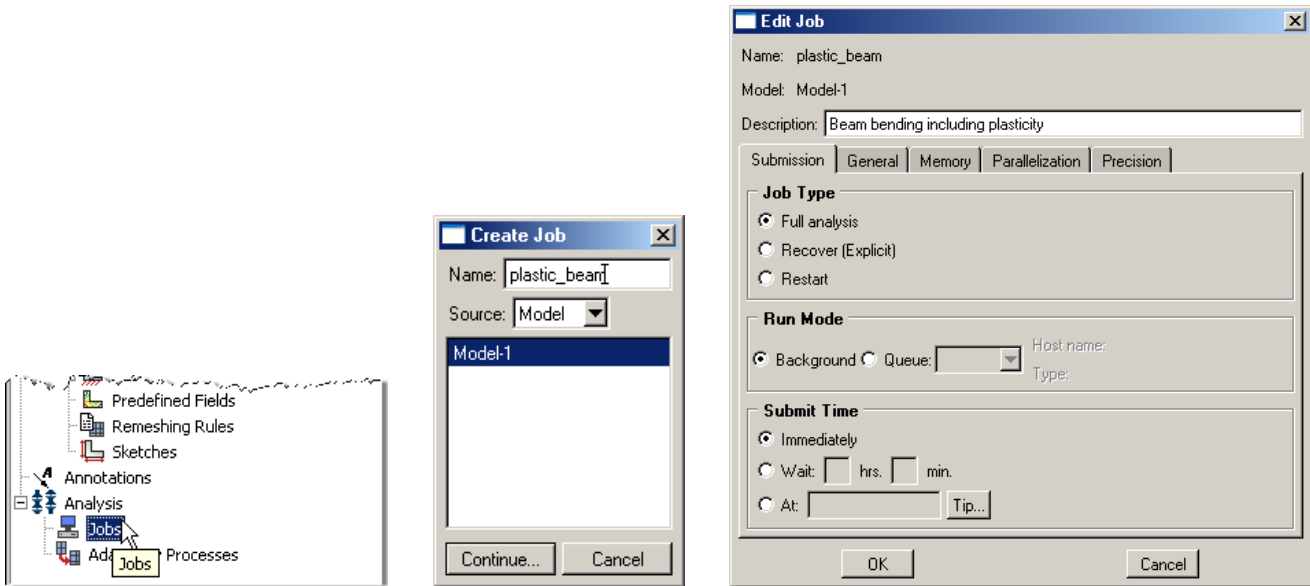
- g. The arrows should now point to the left
 h. Click the "OK" button

17. In the toolbox area click on the "Mesh Part" icon

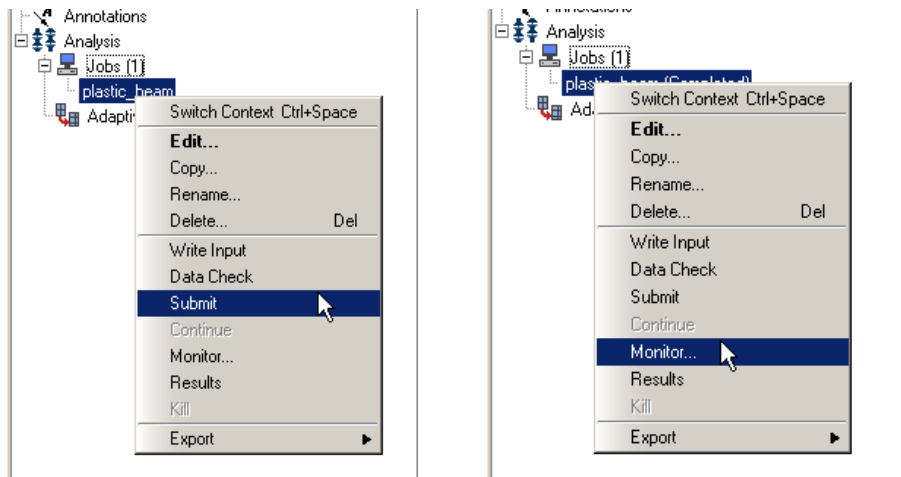


18. In the model tree double click on the "Job" node

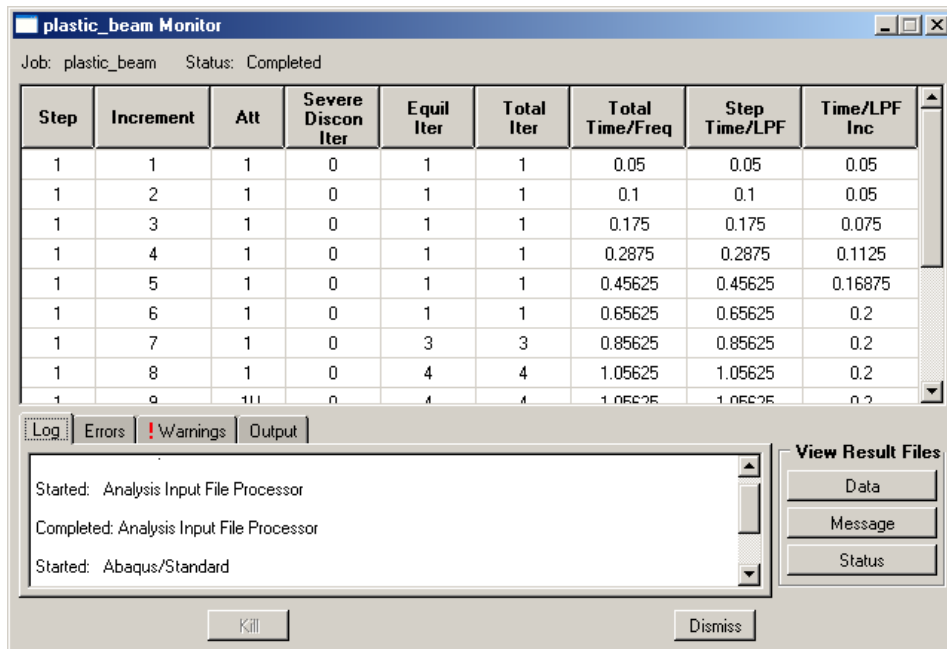
- a. Name the job “plastic_beam”
- b. Give the job a description



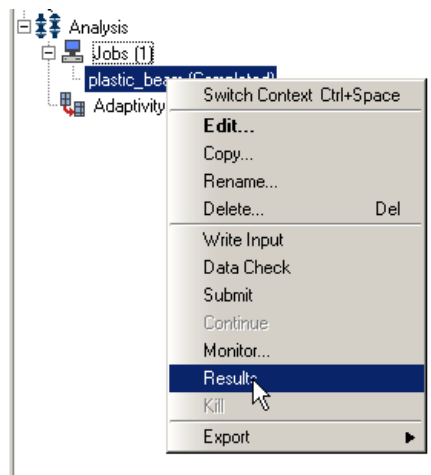
19. In the model tree right click on the job just created and select “Submit”
 - a. Ignore the message about unmeshed portions of the geometry
 - b. While Abaqus is solving the problem right click on the job submitted, and select “Monitor”
 - c.



- d. 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
 - iii. In the far right column, note how Abaqus adjusted the increment



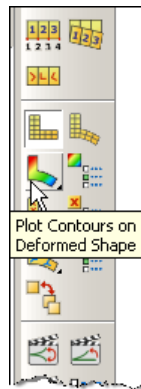
20. In the model tree right click on the submitted and successfully completed job, and select “Results”



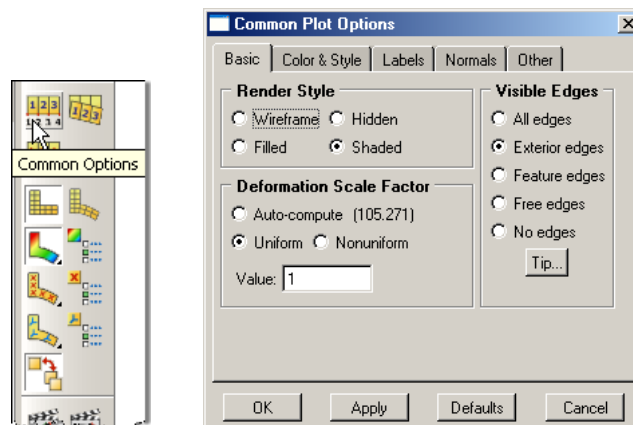
21. In the menu bar click on Viewport → Viewport Annotations Options
- Uncheck the “Show compass option”
 - The locations of viewport items can be specified on the corresponding tab in the Viewport Annotations Options



22. Display the deformed contour of the (Von) Mises stress
 - a. In the toolbox area click on the following icons
 - i. “Plot Contours on Deformed Shape”



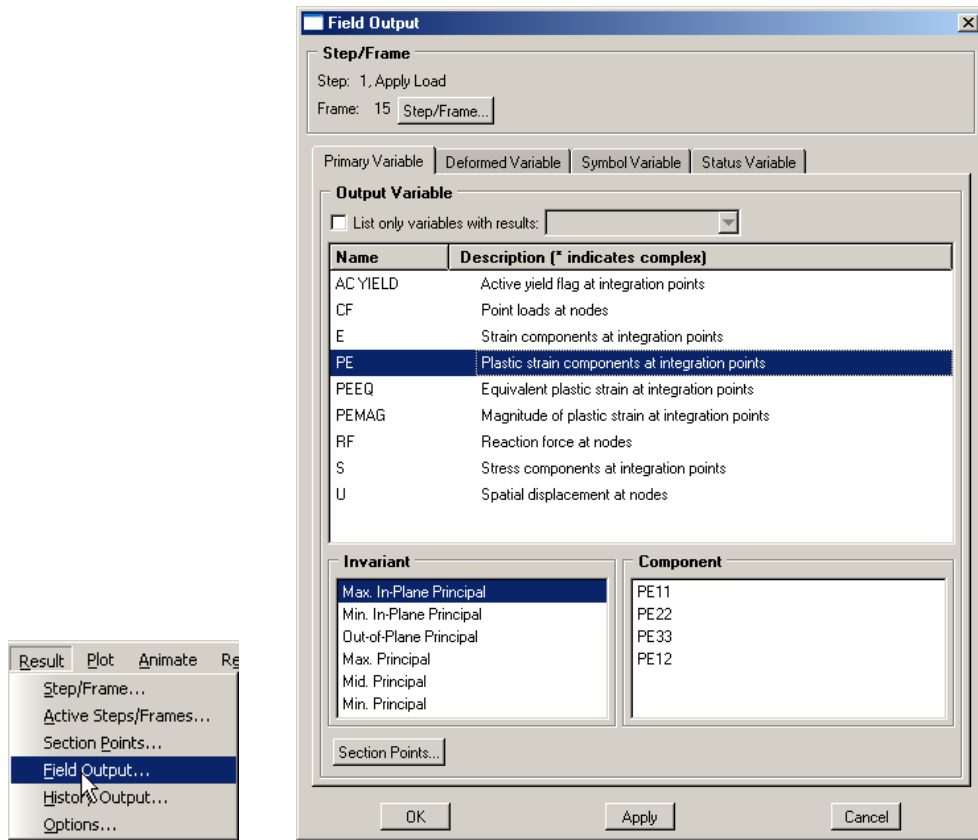
23. In the toolbox area click on the “Common Plot Options” icon
 - a. Set the Deformation Scale Factor to 1
 - b. Click “OK”



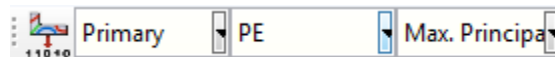
24. Click on the arrows on the context bar to change the time step being displayed
 - a. Click on the three squares to bring up the frame selector slider bar



25. To change the output being displayed, in the menu bar click on Results → Field Output
- Select one of the plastic strain related outputs (PE or PEEQ)
 - Click “OK”



Alternatively, you can select the output variable from the corresponding toolbar (shown below).



Hint: If you don't see the toolbar, go to view → Toolbars and activate the “Field output” to display the toolbar (a checkmark will appear next to it).

Note that PE displays individual plastic strain (similar to principal strain) components, while PEEQ variable provides the equivalent plastic strain value (similar to vonMises equivalent stress).