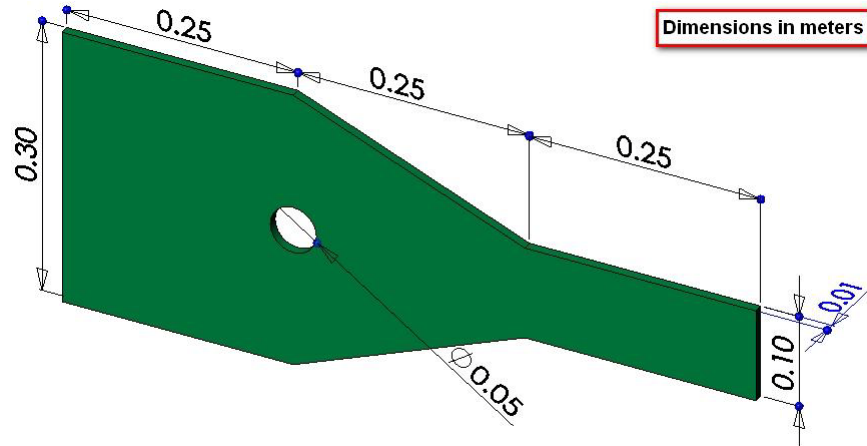


Abaqus Plane Stress Tutorial

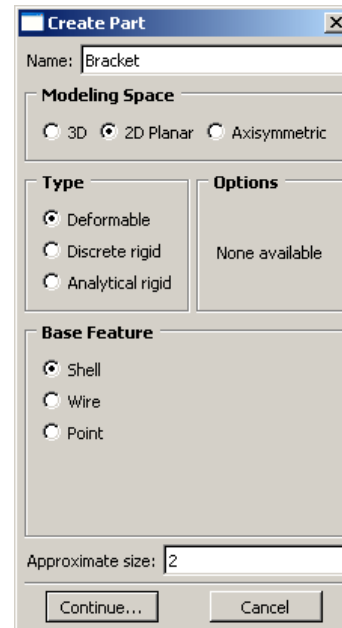
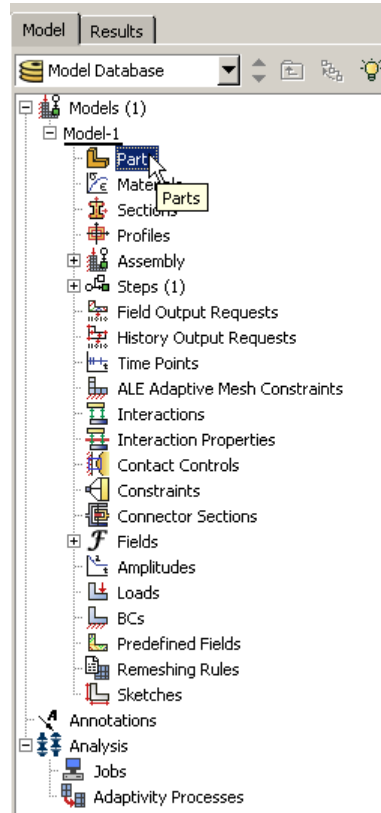
Problem Description



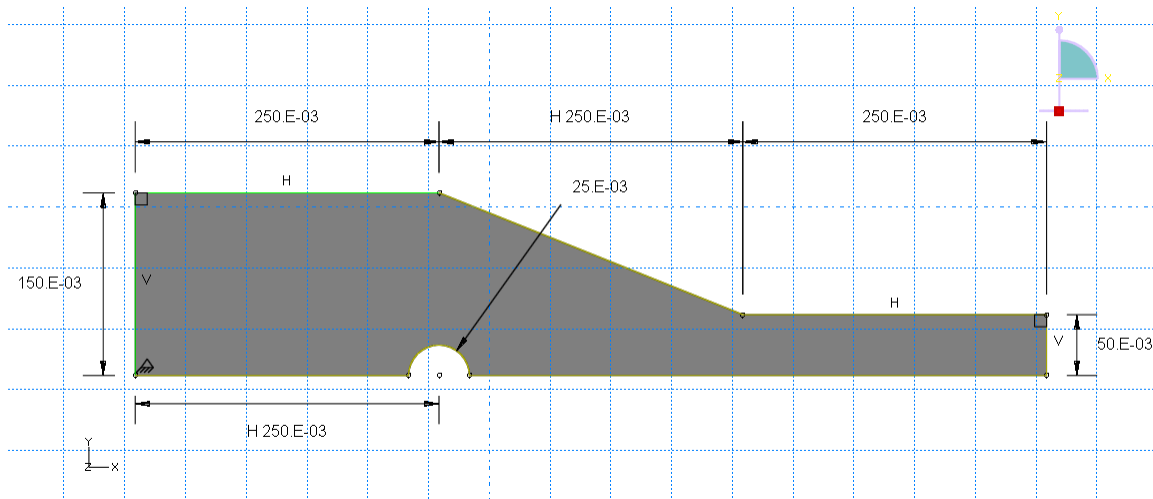
The steel bracket is fixed to a wall along its left side. A tensile pressure force with a total magnitude of 5000 N is applied to the right side of the bracket. The bracket contains one line of symmetry, so only half of the geometry is to be modeled. Determine the stresses in the bracket.

Analysis Steps

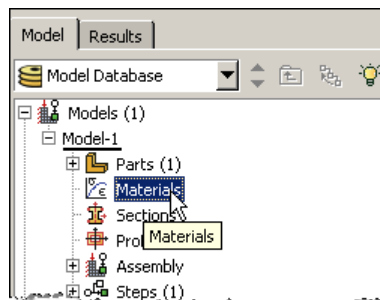
1. Start Abaqus and choose to create a new model database
2. In the model tree double click on the “Parts” node (or right click on “parts” and select Create)



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)

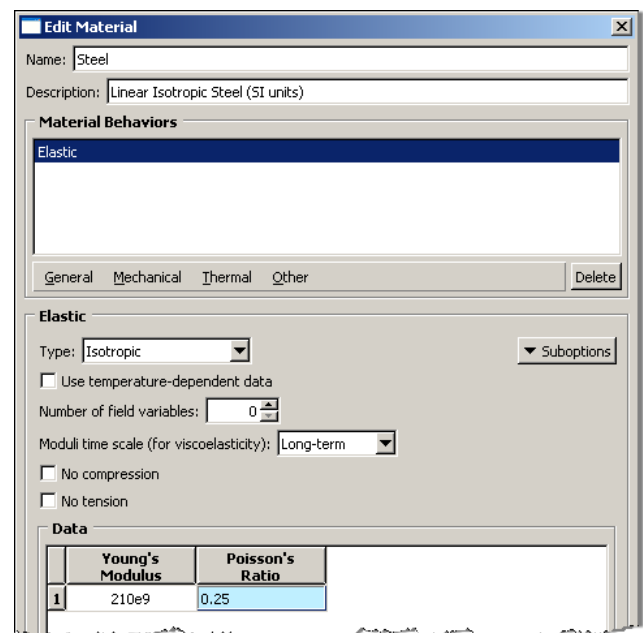
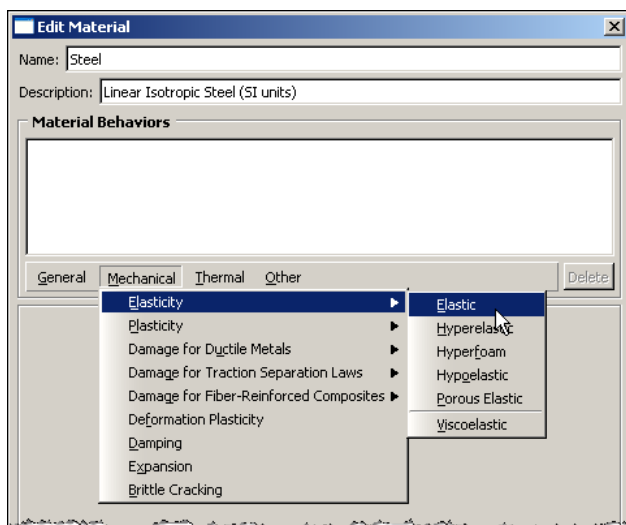


5. Double click on the “Materials” node in the model tree

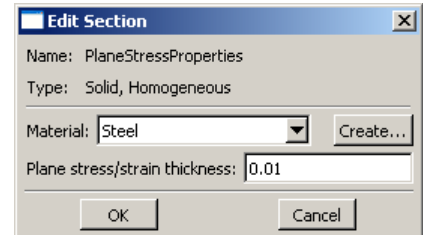
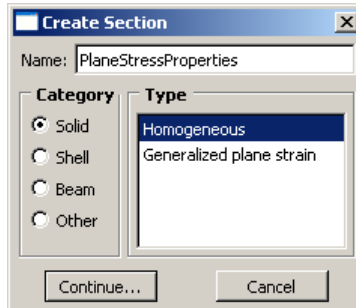
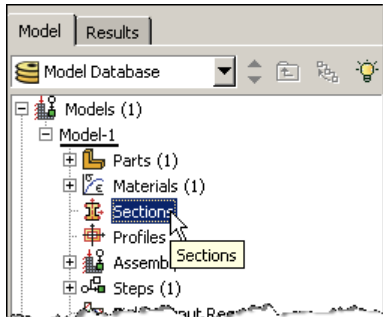


- Name the new material and give it a description
- Click on the “Mechanical” tab → Elasticity → Elastic
- Define Young’s Modulus and the Poisson’s Ratio (use SI units)

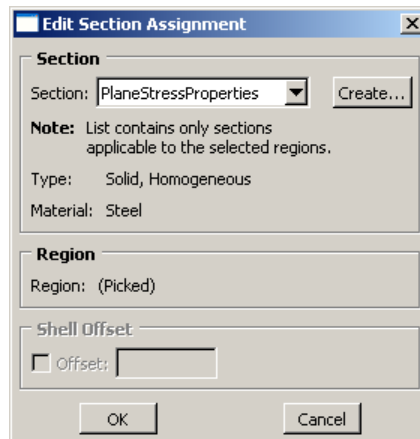
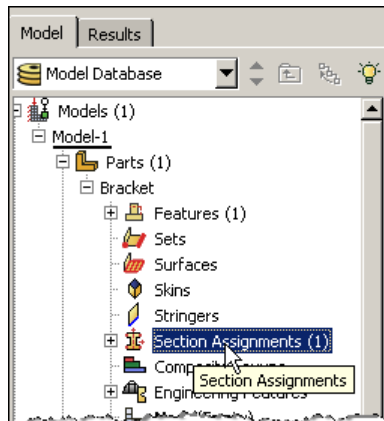
i. WARNING: There are no predefined system of units within Abaqus, so the user is responsible for ensuring that the correct values are specified



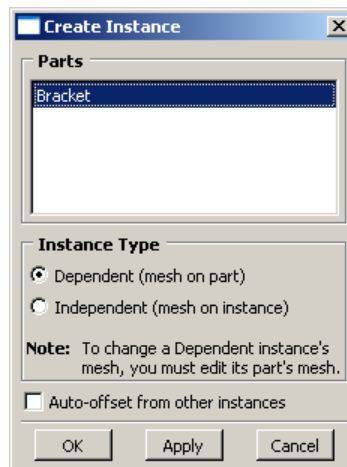
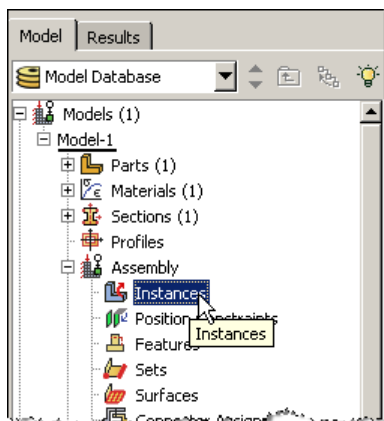
6. Double click on the "Sections" node in the model tree
 - a. Name the section "PlaneStressProperties" and select "Solid" for the category and "Homogeneous" for the type
 - b. Select the material created above (Steel) and set the thickness to 0.01



7. Expand the "Parts" node in the model tree and double click on "Section Assignments"
 - a. Select the surface geometry in the viewport
 - b. Select the section created above (PlaneStressProperties)

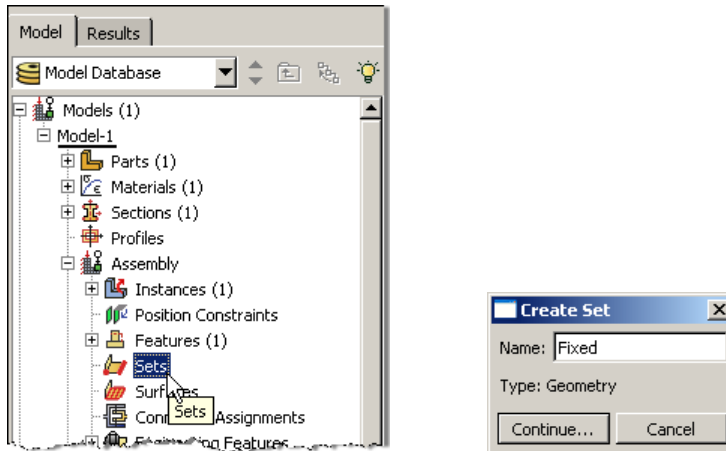


8. Expand the "Assembly" node in the model tree and then double click on "Instances"
 - a. Select "Dependent" for the instance type

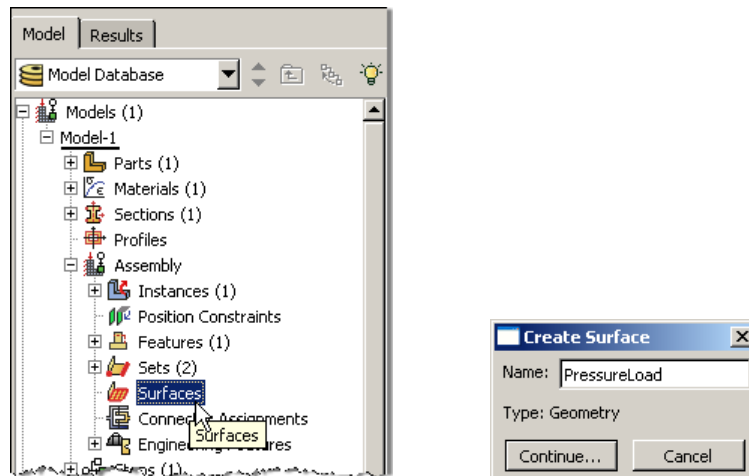


9. In the model tree, under the expanded "Assembly" node, double click on "Sets"

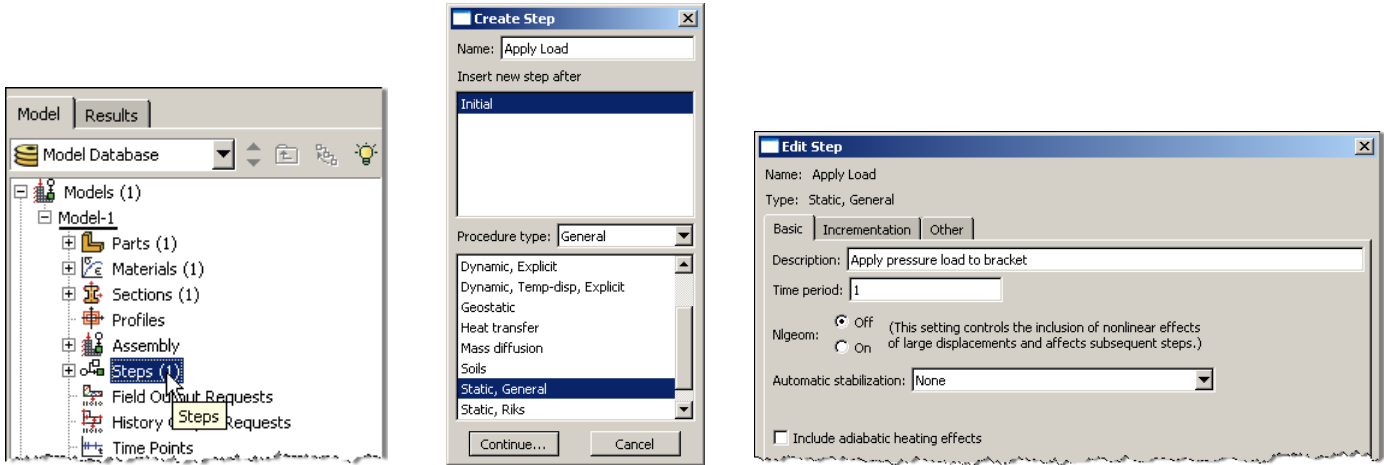
- a. Name the set "Fixed"
- b. Select the left edge of the surface in the viewport



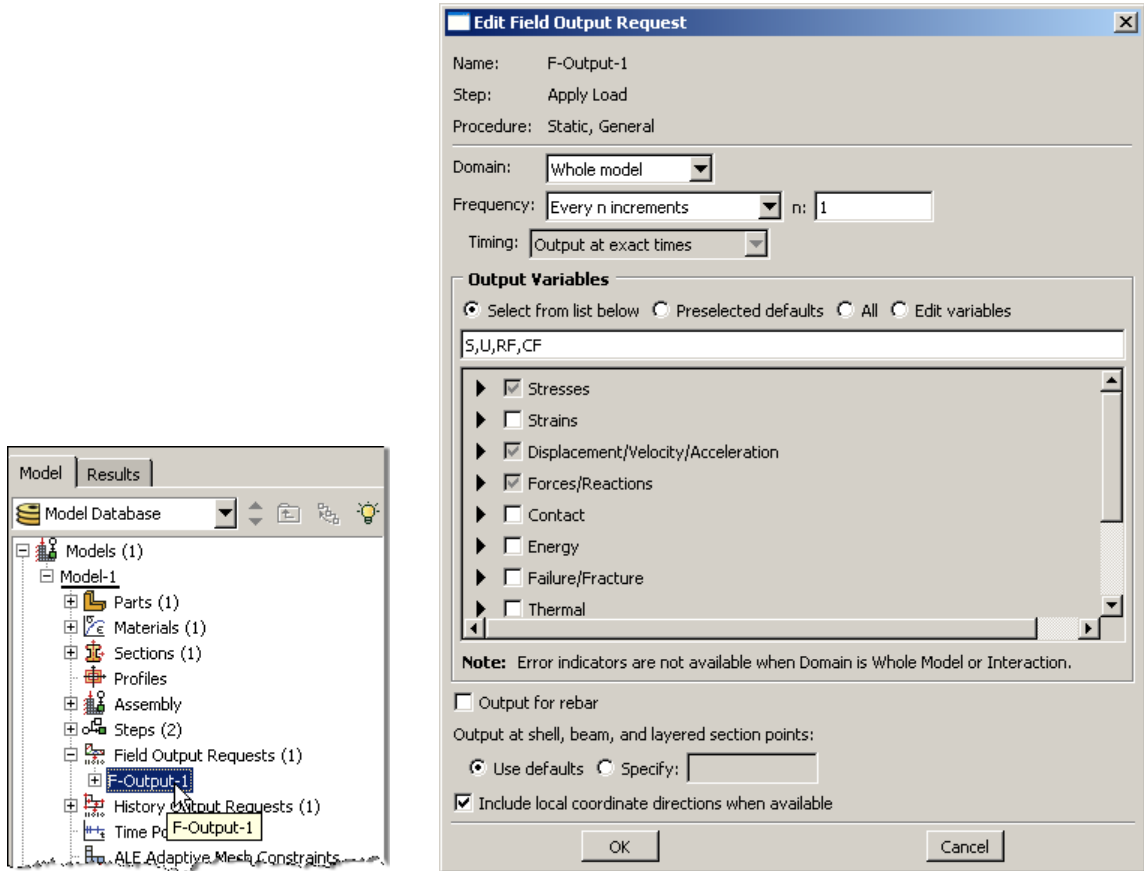
- c. Create another set named "Symmetry"
 - d. Select the two lower horizontal edges of the surface in the viewport
10. In the model tree, under the expanded "Assembly" node, double click on "Surfaces"
- a. Name the surface "PressureLoad"
 - b. Select the right edge of the surface in the viewport



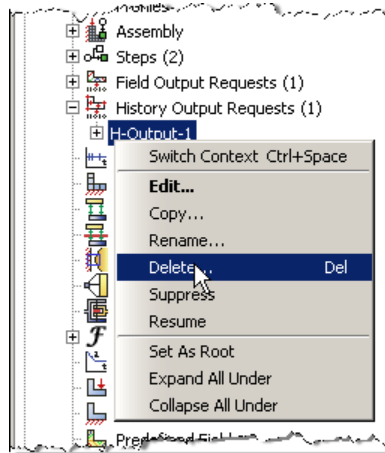
11. Double click on the "Steps" node in the model tree
- a. Name the step, set the procedure to "General", and select "Static, General"
 - b. Give the step a description



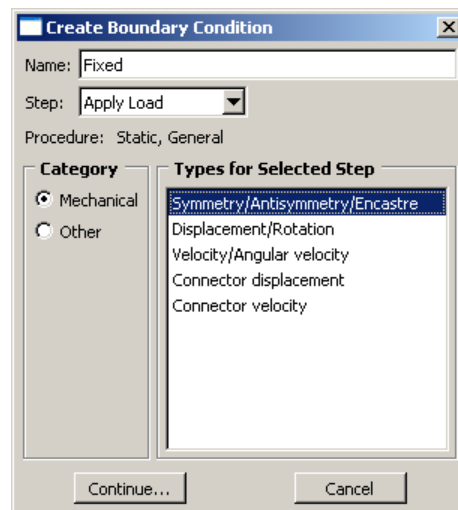
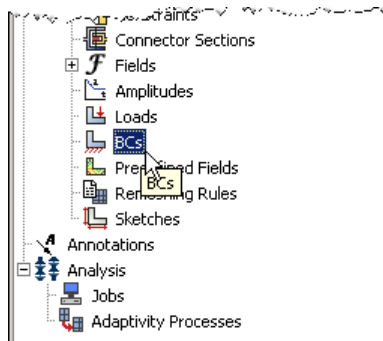
12. Expand the Field Output Requests node in the model tree, and then double click on F-Output-1 (F-Output-1 was automatically generated when creating the step)
 - a. Uncheck the variables "Strains" and "Contact"



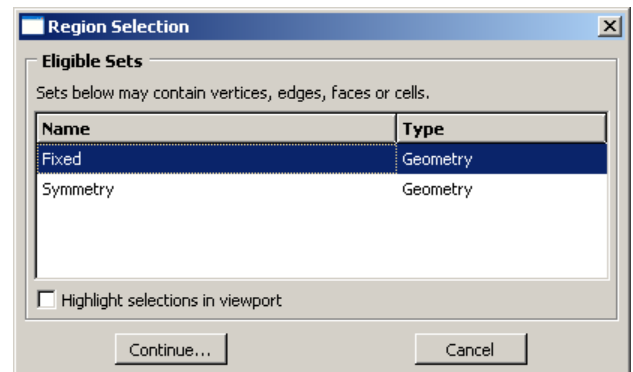
13. Expand the History Output Requests node in the model tree, and then right click on H-Output-1 (H-Output-1 was automatically generated when creating the step) and select Delete



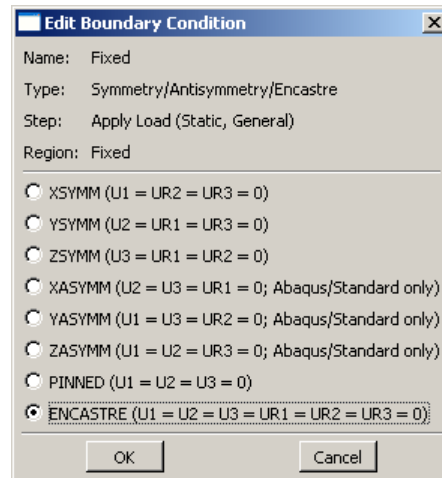
- 14. Double click on the “BCs” node in the model tree
 - a. Name the boundary conditioned “Fixed” and select “Symmetry/Antisymmetry/Encastre” for the type



- b. In the prompt area click on the Sets button
 - c. Select the set named “Fixed”



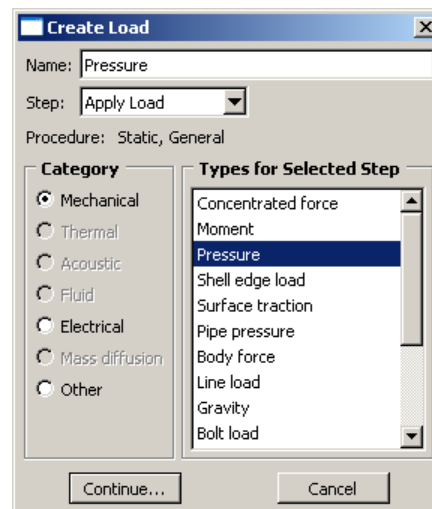
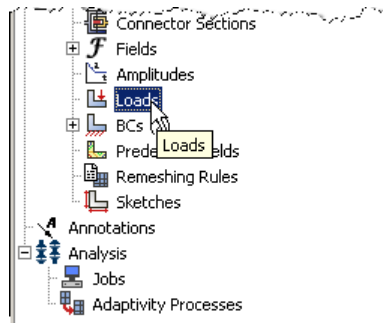
- d. Select “ENCASTE” for the boundary condition (means “clamped” BC).



- e. Repeat the procedure for the symmetry restraint using the set named "Symmetry", select "YSYMM" 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

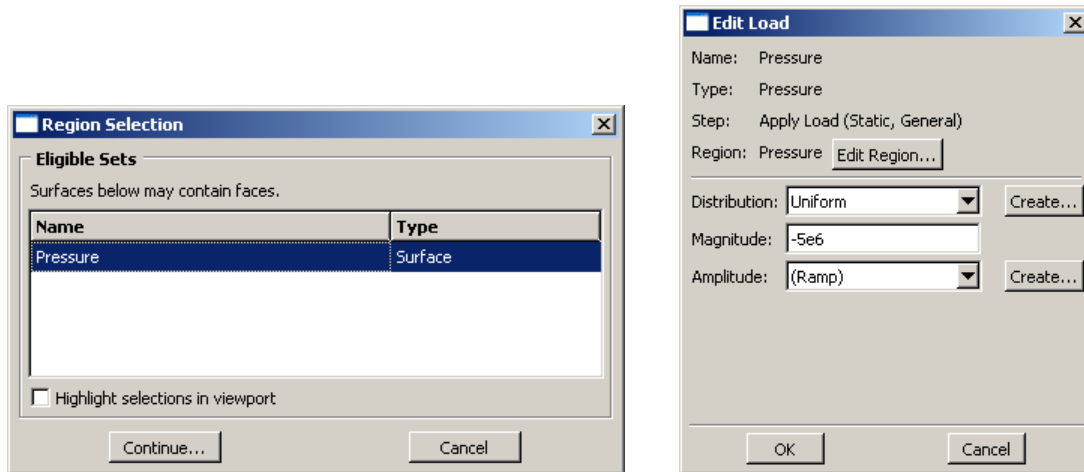


- b. Select surface named "Pressure"

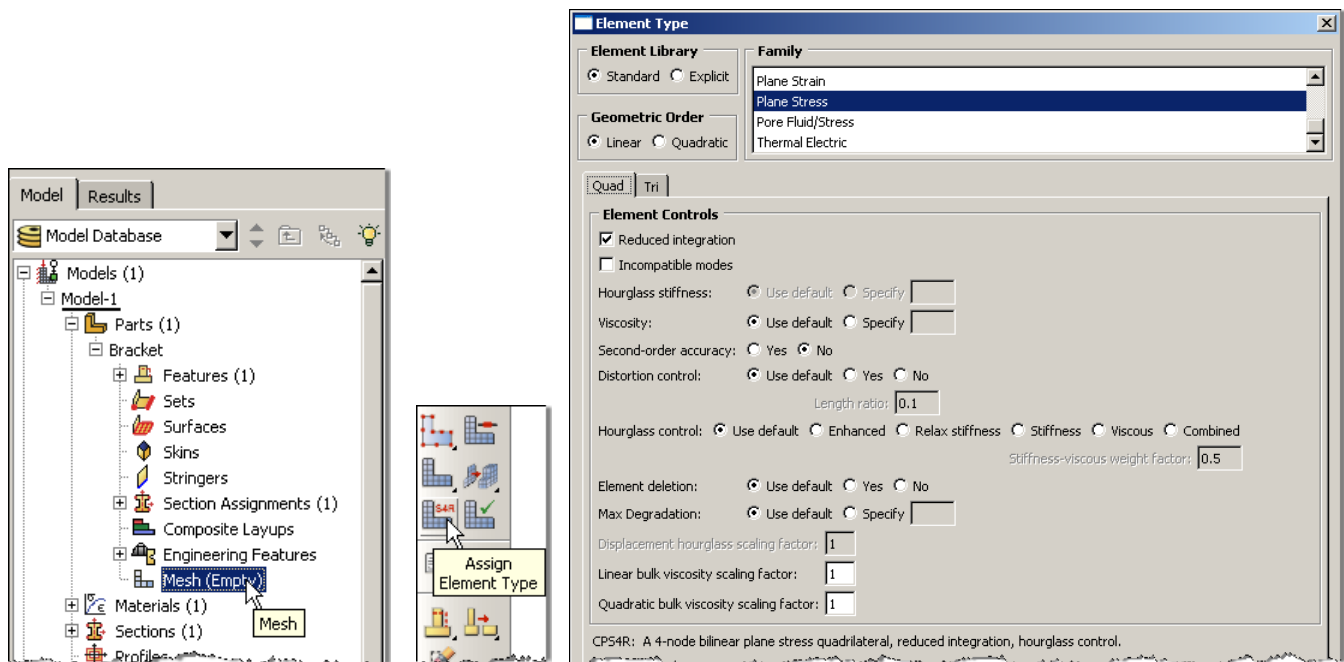
- c. For the magnitude enter -5e6

- i. Note that because we have been using standard SI units the load applied is $-5 \times 10^6 \text{ N/m}^2$ which is

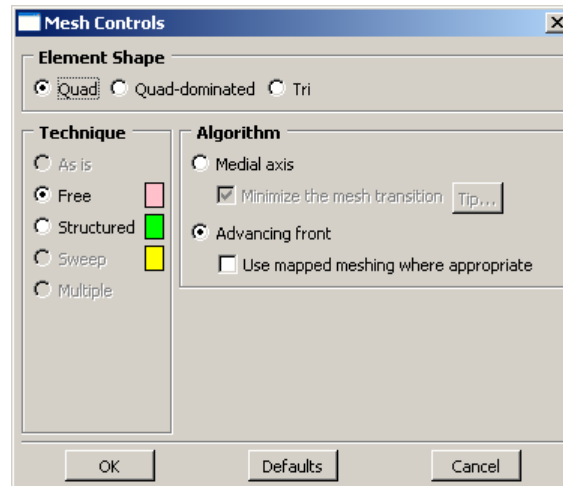
a total of -2500 N distributed across the right edge of the surface $\left(\frac{-2500 \text{ N}}{(0.05\text{m})(0.01\text{m})} \right)$



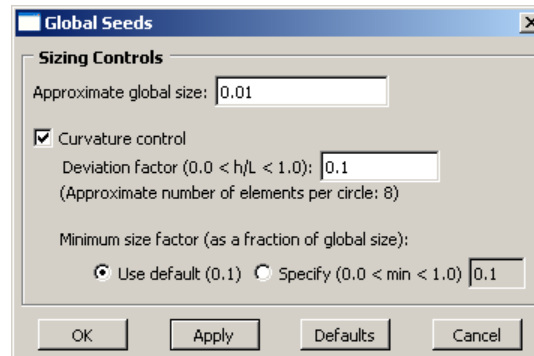
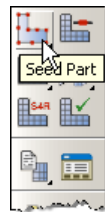
16. In the model tree double click on “Mesh” for the Bracket part, and in the toolbox area click on the “Assign Element Type” icon
- Select “Standard” for element type
 - Select “Linear” for geometric order
 - Select “Plane Stress” for family
 - Note that the name of the element (CPS4R) and its description are given below the element controls



17. In the toolbox area click on the “Assign Mesh Controls” icon
- Change the element shape to “Quad”



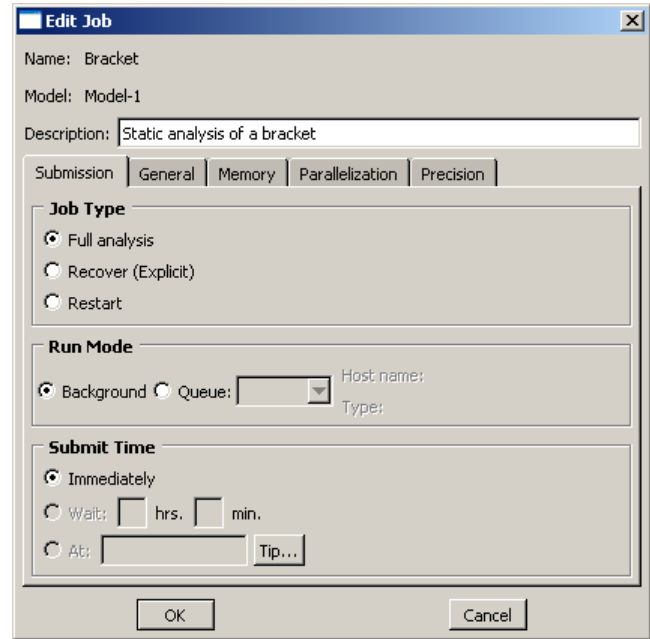
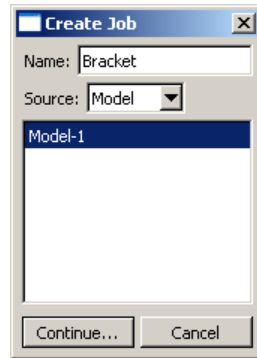
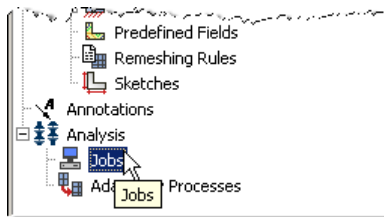
18. In the toolbox area click on the “Seed Part” icon
 - a. Set the approximate global size to 0.01



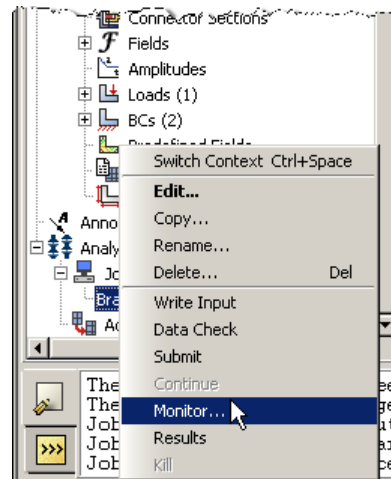
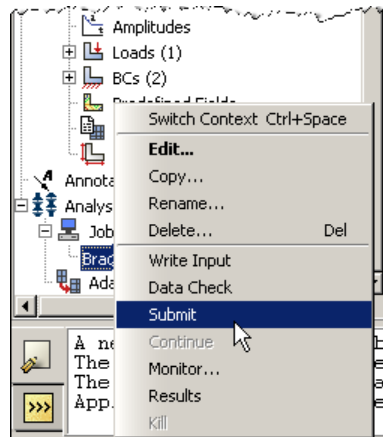
19. In the toolbox area click on the “Mesh Part” icon



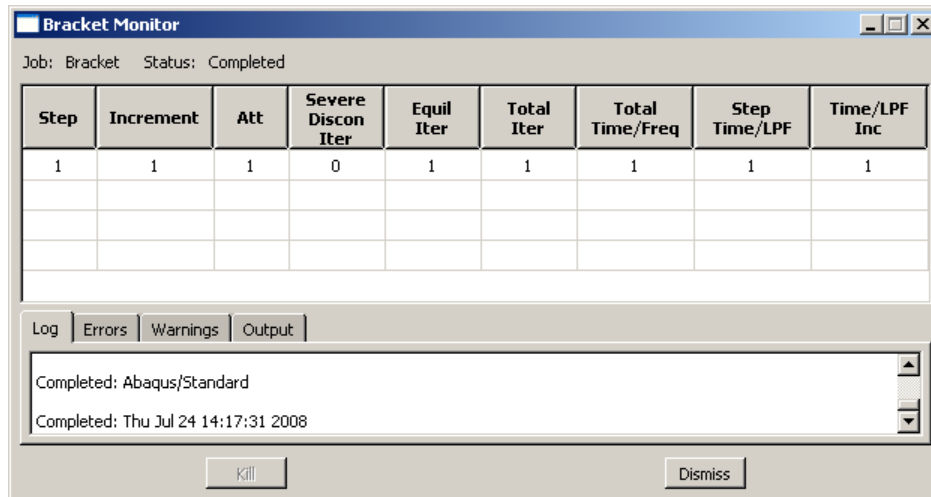
20. In the model tree double click on the “Job” node
 - a. Name the job “Bracket”
 - b. Give the job a description



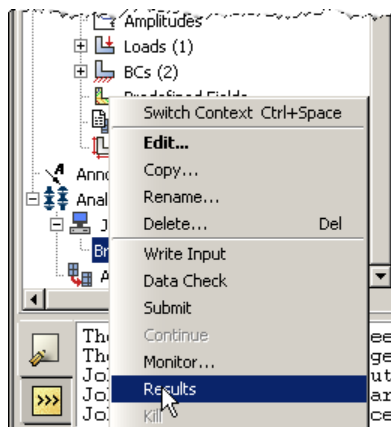
21. In the model tree right click on the job just created (Bracket) and select "Submit"
 - a. While Abaqus is solving the problem right click on the job submitted (Bracket), and select "Monitor"



- 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

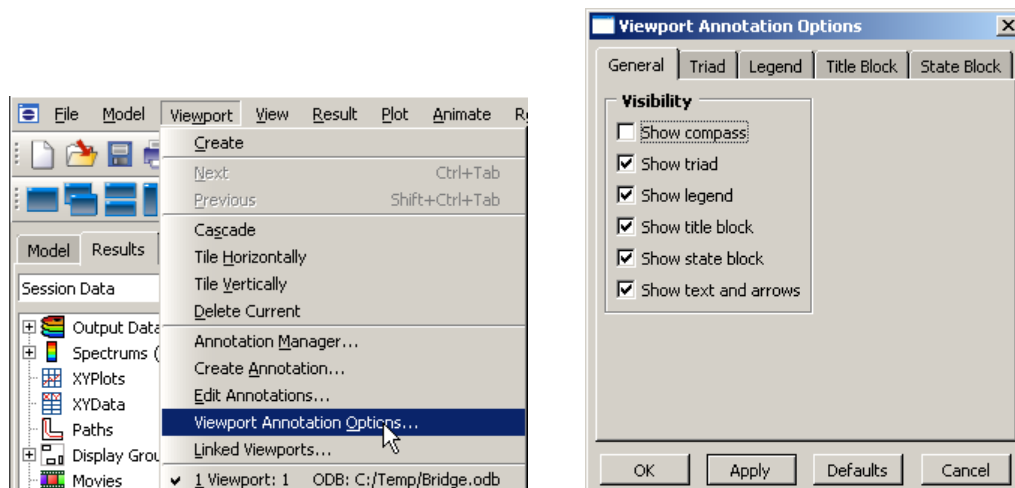


22. In the model tree right click on the submitted and successfully completed job (Bracket), and select “Results”



23. In the menu bar click on Viewport → Viewport Annotations Options

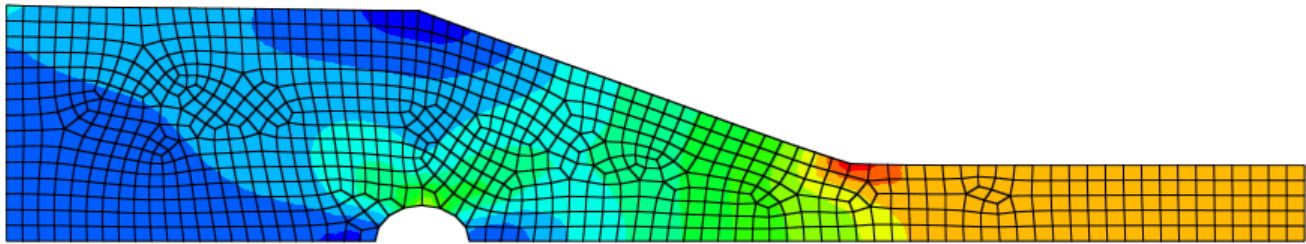
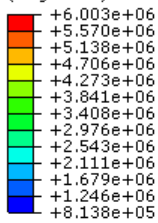
- a. Uncheck the “Show compass option”
- b. The locations of viewport items can be specified on the corresponding tab in the Viewport Annotations Options



24. Display the deformed contour of the (Von) Mises stress

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

S, Mises
(Avg: 75%)

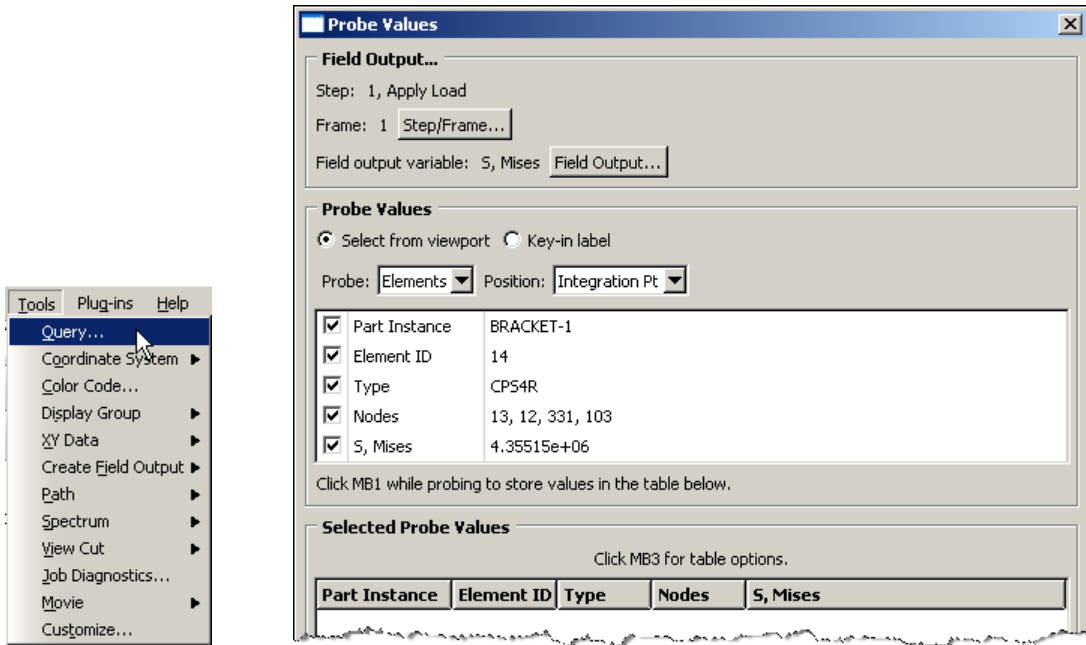


Static analysis of a bracket
 ODB: Bracket.odb Abaqus/Standard Student Edition 6.7-2 Thu Jul 24 15:07:45 Pacific Daylight Time 2008
 Step: Apply Load, Apply pressure load to bracket
 Increment 1: Step Time = 1.000
 Primary Var: S, Mises
 Deformed Var: U Deformation Scale Factor: +6.376e+03

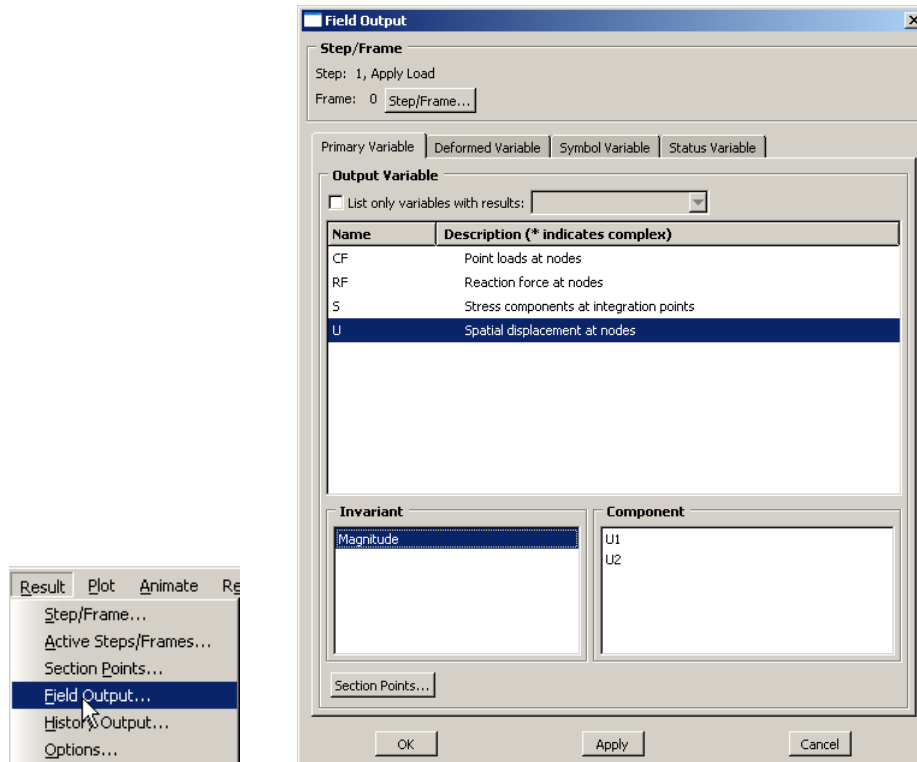


25. To determine the stress values, from the menu bar click Tools → Query

- a. Check the boxes labeled “Nodes” and “S, Mises”
- 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

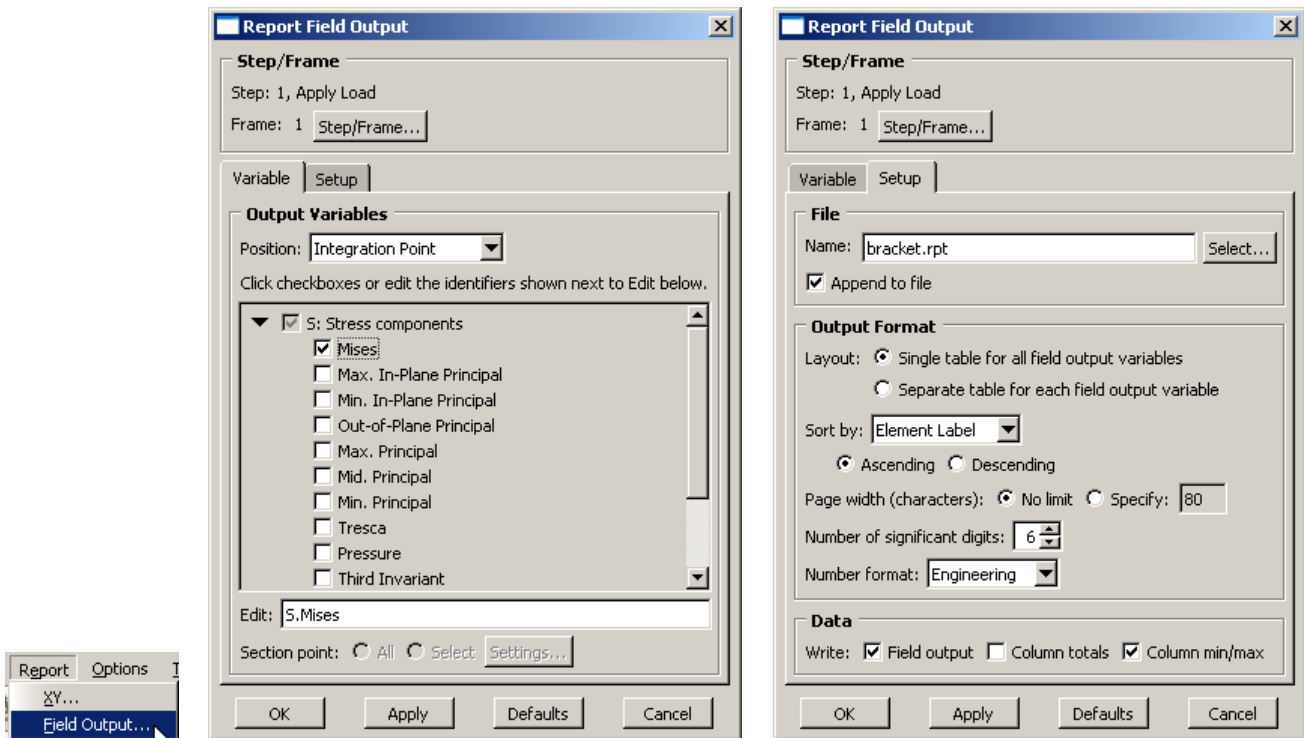


26. To change the output being displayed, in the menu bar click on Results → Field Output
 - a. Select “Spatial displacement at nodes”
 - i. Invariant = Magnitude

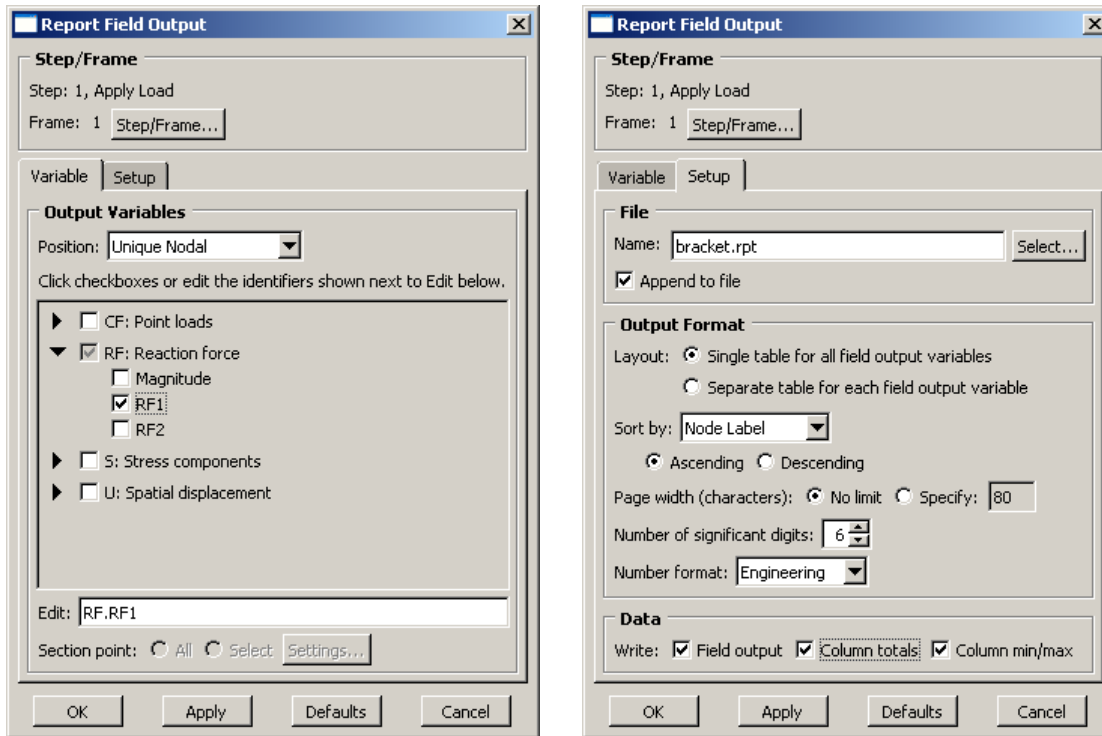


27. To create a text file containing the stresses and reaction forces (including total), in the menu bar click on Report → Field Output
 - a. For the output variable select (Von) Mises
 - b. On the Setup tab specify the name and the location for the text file

- c. Uncheck the “Column totals” option
- d. Click Apply



- a. Back on the Variable tab change the position to “Unique Nodal”
- b. Uncheck the stress variable, and select the RF1 reaction force
- c. On the Setup tab, check the “Column totals” option
- d. Click OK



28. Open the .rpt file with any text editor
 - a. One thing to check is that the total reaction force is equal to the applied load (-2,500 N)

Output sorted by column "Node Label".

Field Output reported at nodes for part: BRACKET-1

Node Label	RF.RF1 @Loc 1
:	:
8	-76.1136
:	:
62	-130.944
63	-152.996
64	-156.267
65	-160.712
66	-173.951
:	:
318	-152.139
319	-152.673
320	-153.997
321	-154.675
322	-157.14
323	-159.558
324	-164.544
325	-166.167
326	-176.78
327	-211.343
Minimum	-211.343
At Node	327
Maximum	0.
At Node	949
Total	-2.50000E+03