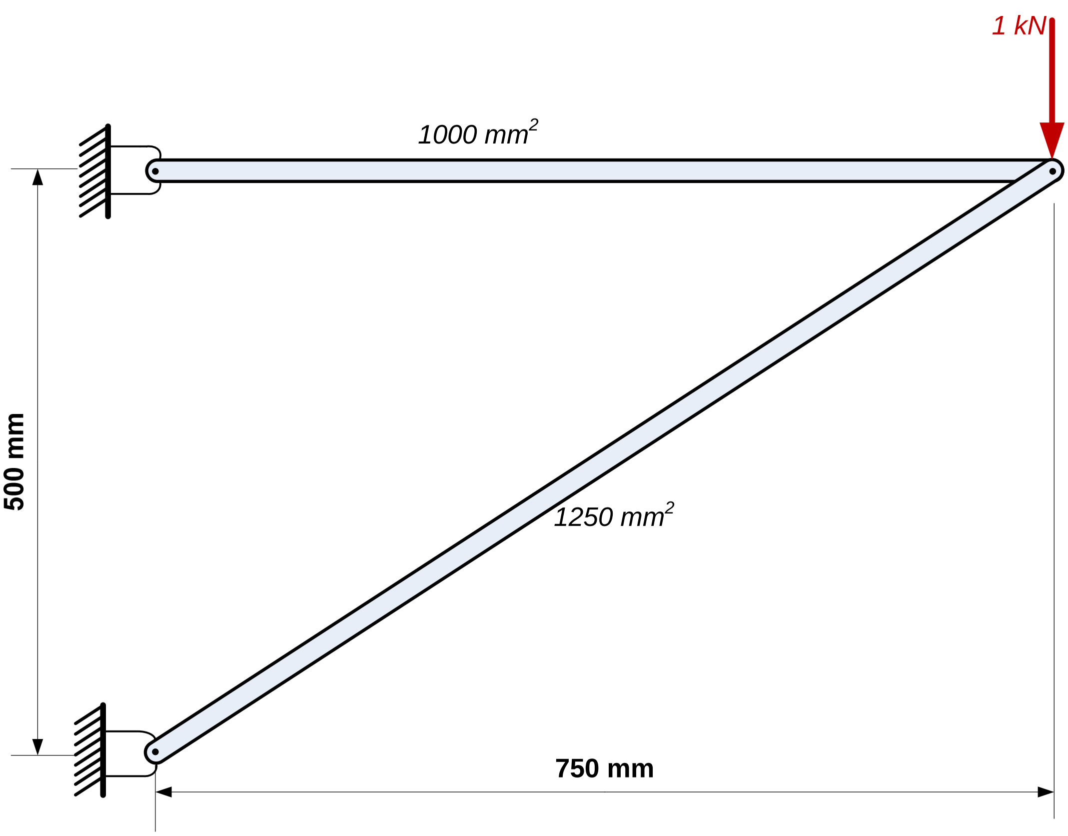
Abaqus/CAE Truss Tutorial *(Version 6.12)*

Problem Description:

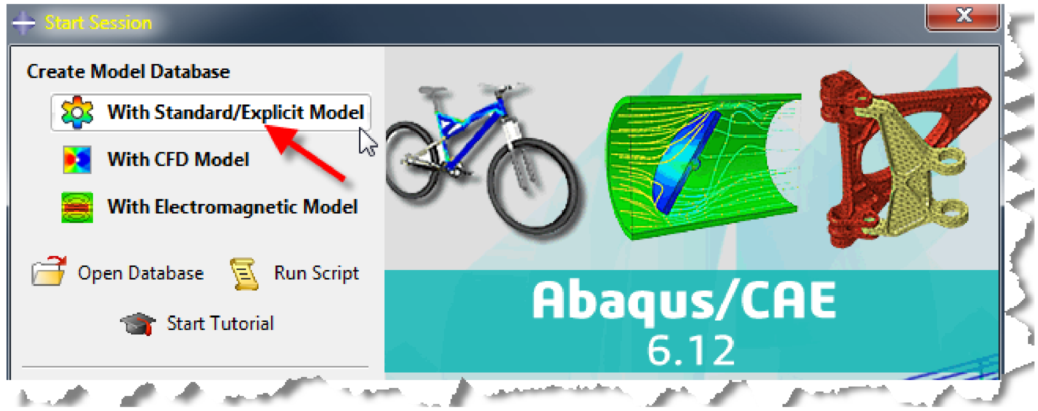
Given the truss structure shown below with pinned supports at the wall and 1kN applied load; solve for displacements of the free node and the reaction forces of the truss structure. The truss material is steel with E = 210 GPa and υ =0.25.

This sample problem is similar to the lecture note example.

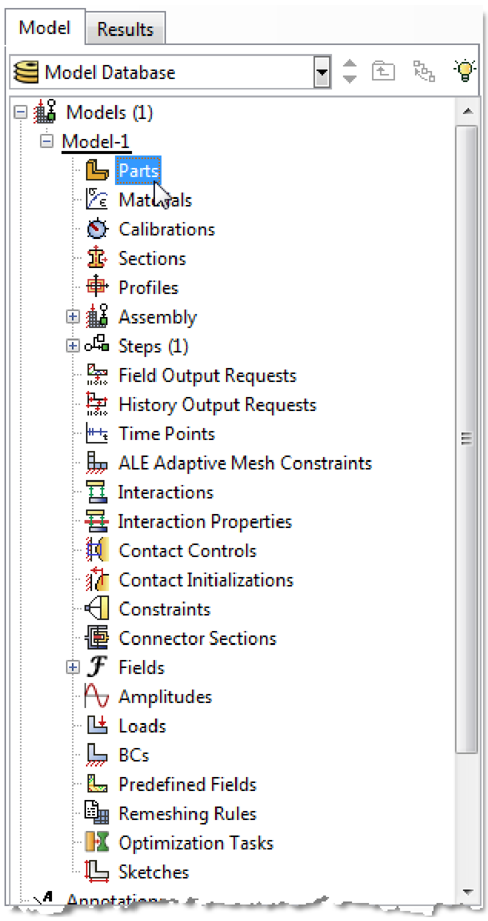


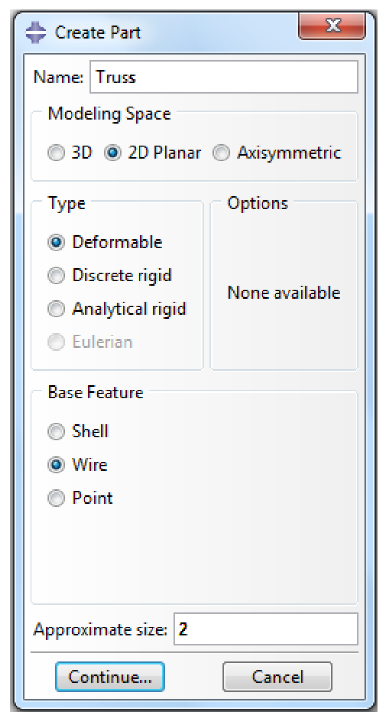
Analysis Steps

1. Start Abaqus and choose to create a new model database



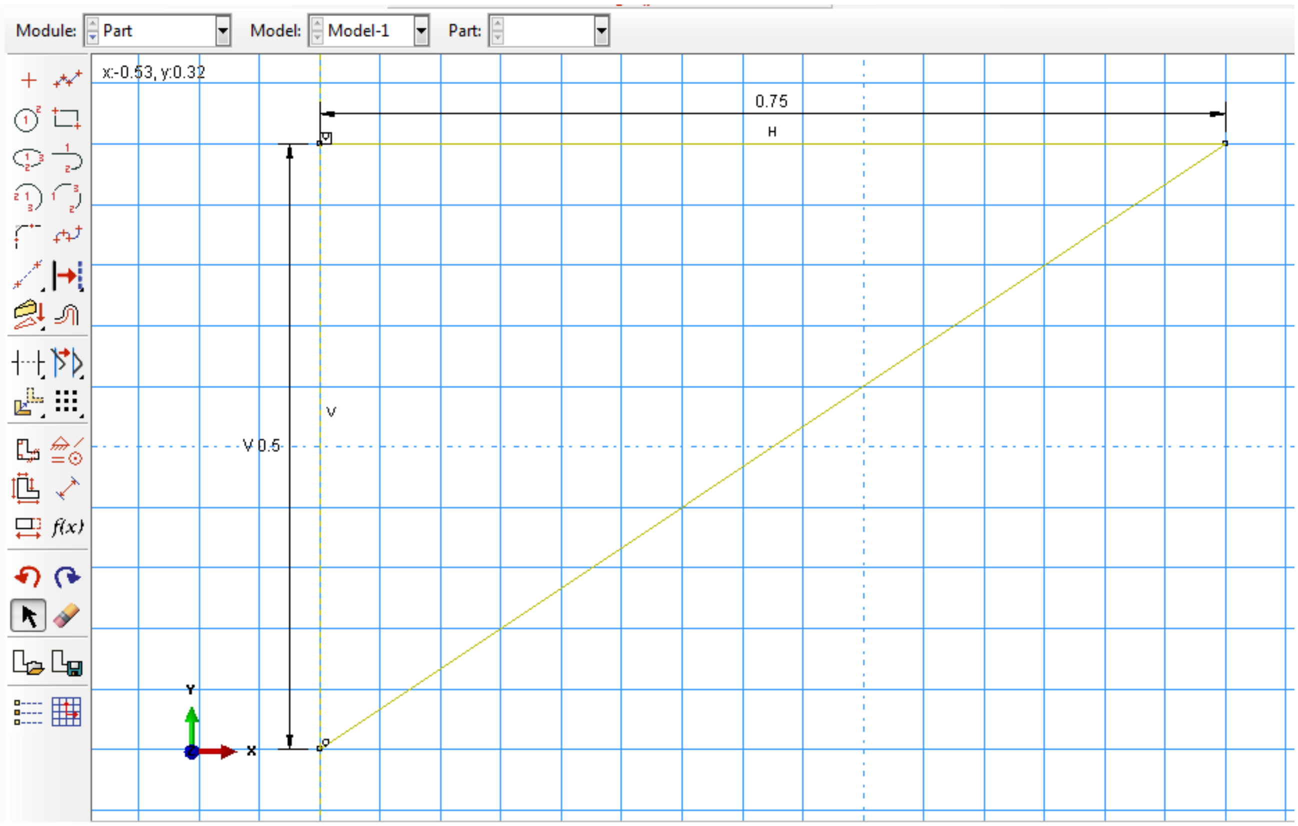
2. In the model tree double click on the “Parts” node (or right click on “parts” and select Create)



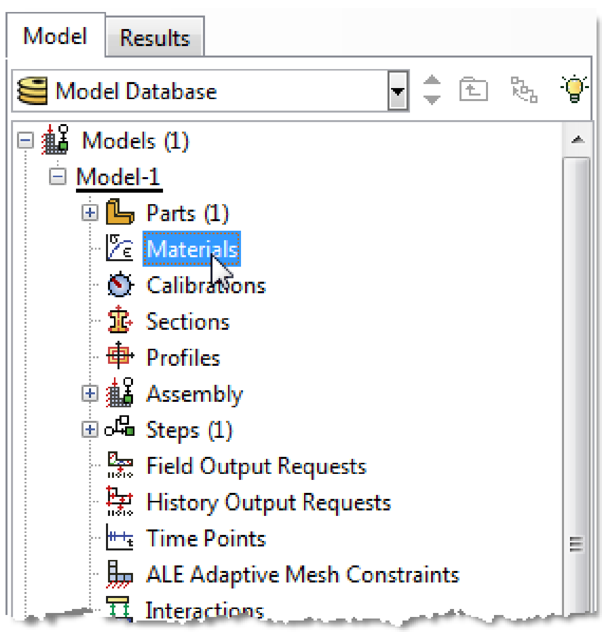


3. In the Create Part dialog box (shown above) name the part and
    a. Select “2D Planar”
    b. Select “Deformable”
    c. Select “Wire”
    d. Set approximate size = 1
    e. Click “Continue…”

4. Create the geometry shown below (details not discussed here)



5. Double click on the "Materials" node in the model tree



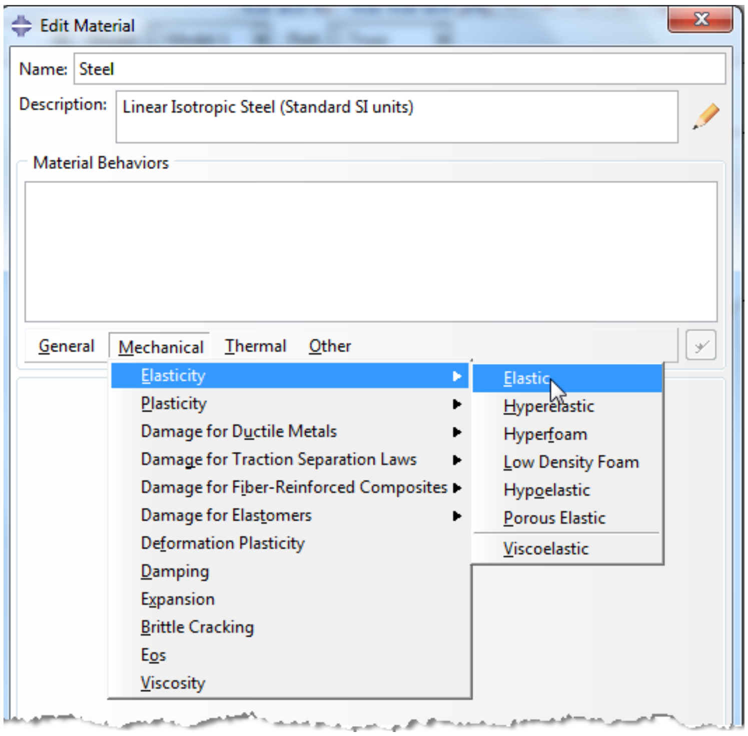
a. Name the new material and give it a description

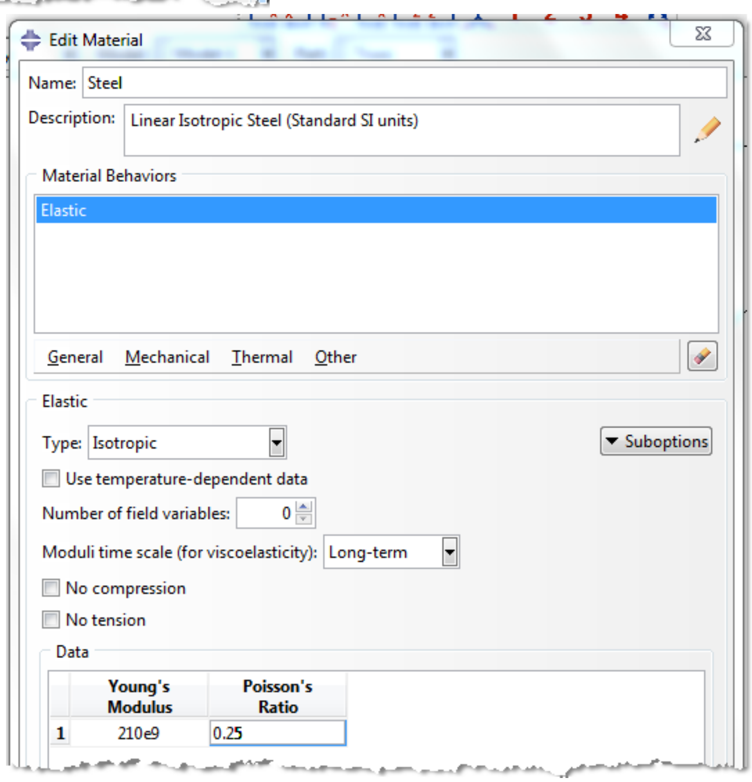
b. Click on the "Mechanical" tab→Elasticity→Elastic

c. Define Young's Modulus and Poisson's Ratio (use base SI units)

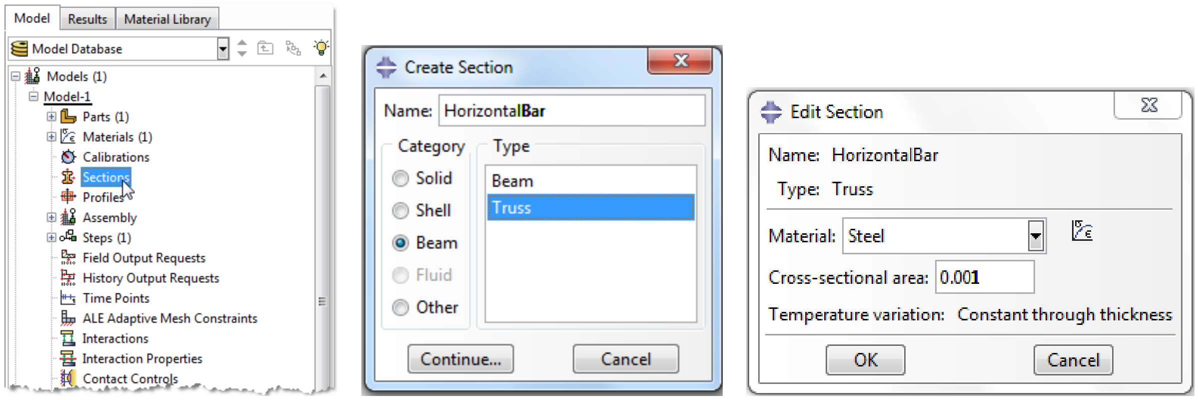
  i. **WARNING: There are no predefined system of units within Abaqus, so the user is responsible for ensuring that the correct values are specified**

d. Click "OK"



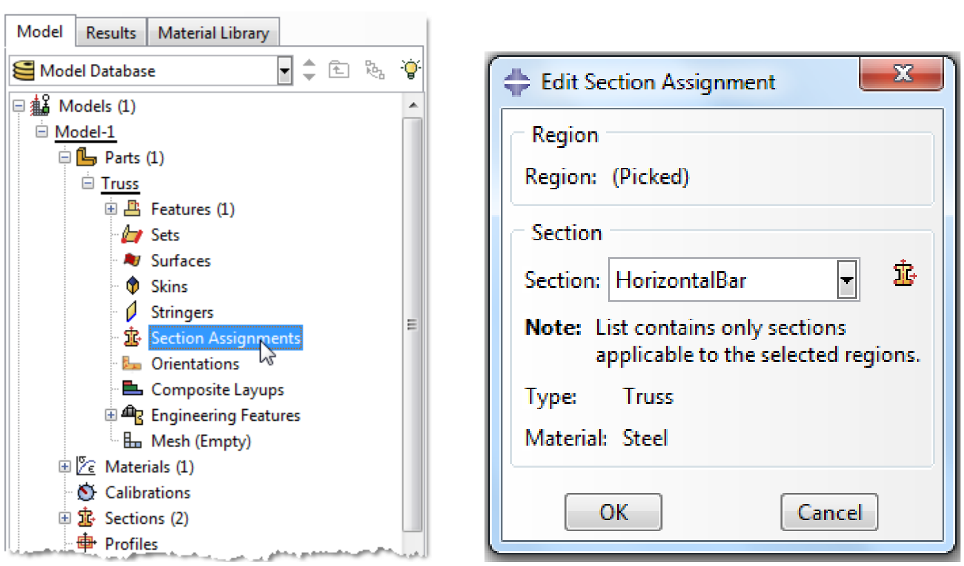


6. Double click on the “Sections” node in the model tree
    a. Name the section “HorizontalBar” and select “Beam” for both the category and “Truss” for the type
    b. Click “Continue…”
    c. Select the material created above (Steel)
    d. Set cross-sectional area = 0.001 (base SI units, m^2)
    e. Click “OK”



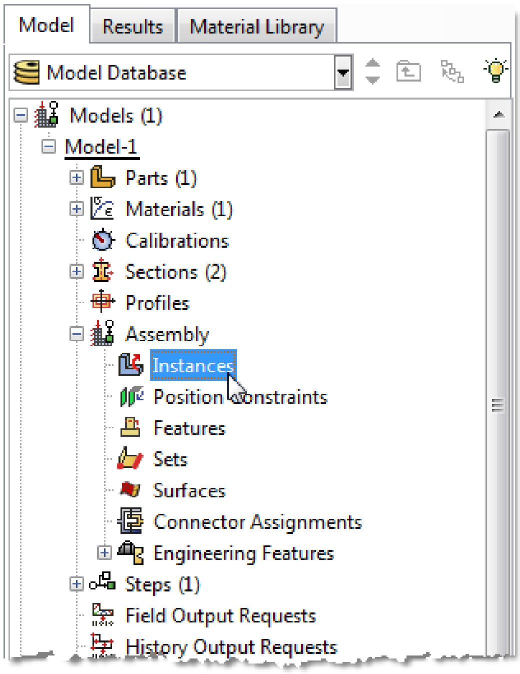
    f. Repeat for the “AngledBar”
        i. Cross-sectional area=0.00125

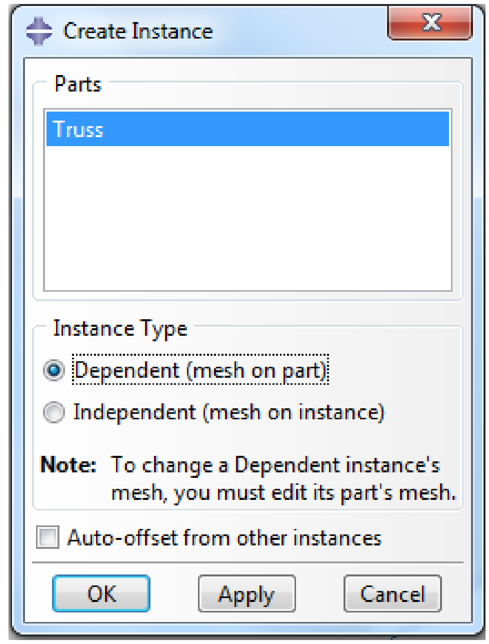
7. Expand the “Parts” node in the model tree, expand the node of the part just created, and double click on “Section Assignments”
    a. Select the horizontal portion of the geometry in the viewport
    b. Click “Done”
    c. Select the “HorizontalBar” section created above
    d. Click “OK”



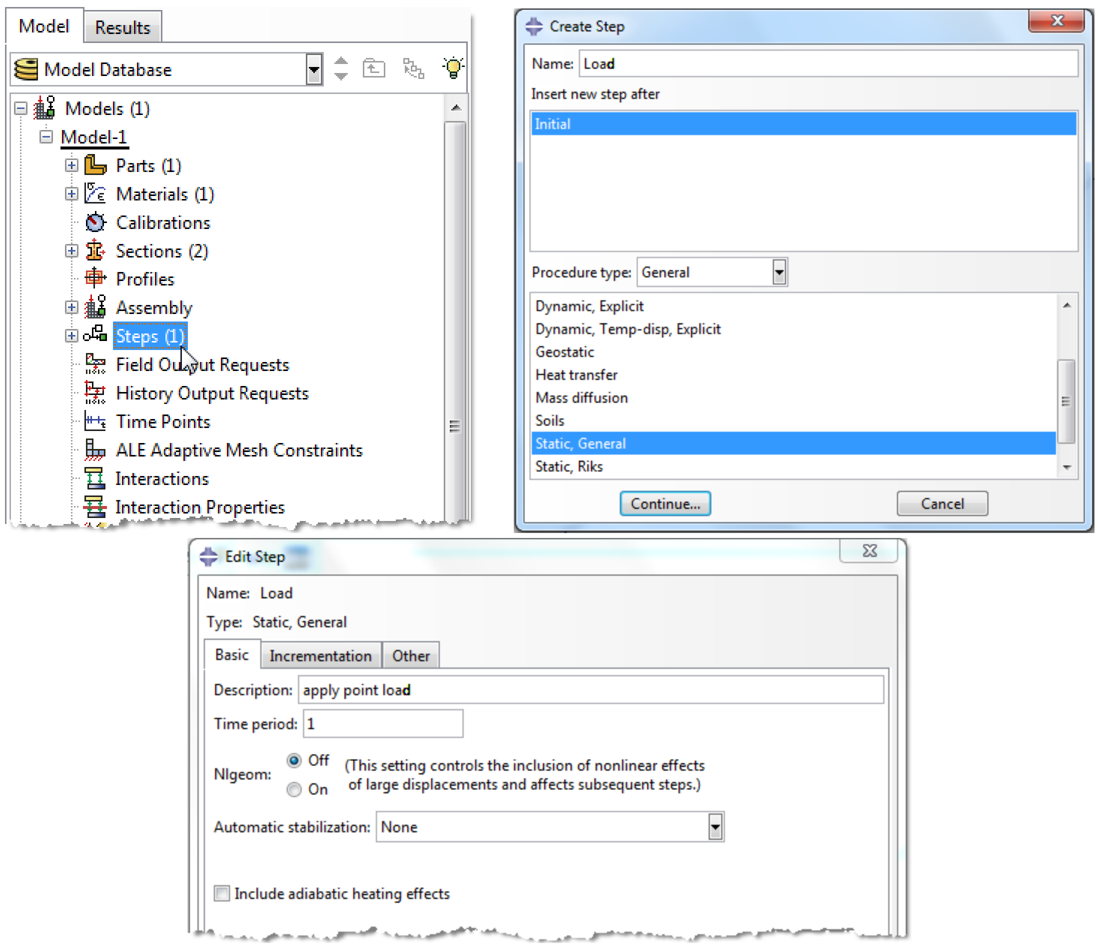
    e. Repeat for the angled portion of the geometry

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”

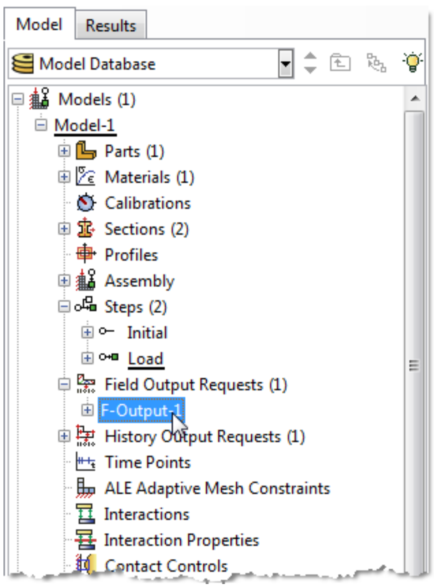


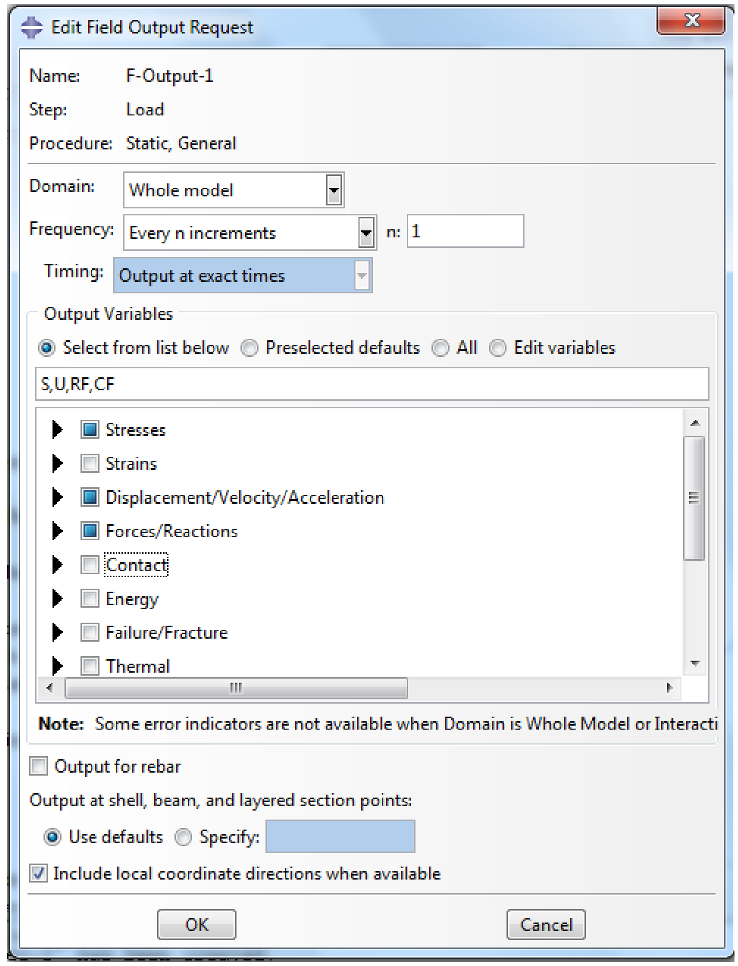


9. Double click on the "Steps" node in the model tree
    a. Name the step, set the procedure to "General", and select "Static, General"
    b. Click "Continue…"
    c. Give the step a description
    d. Click "OK"

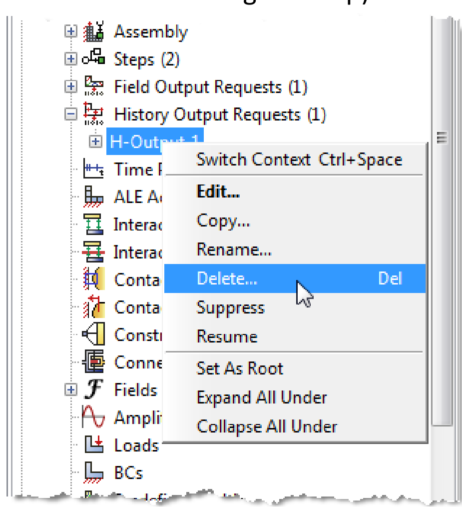


10. Expand the Field Output Requests node in the model tree, and then double click on F-Output-1 (F-Output-1 was automatically generated when creating the step)
    a. Uncheck the variables “Strains” and “Contact”
    b. Click “OK”

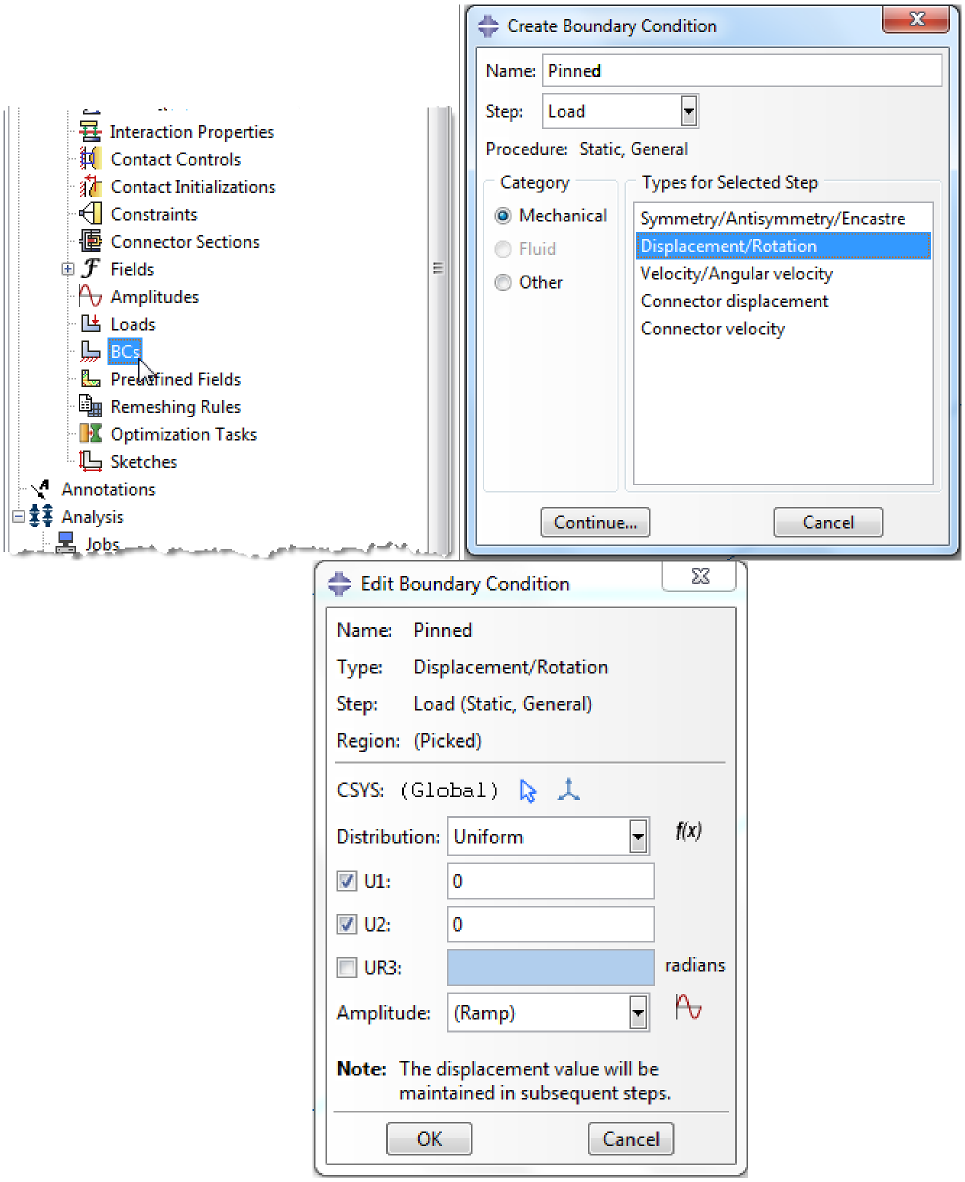




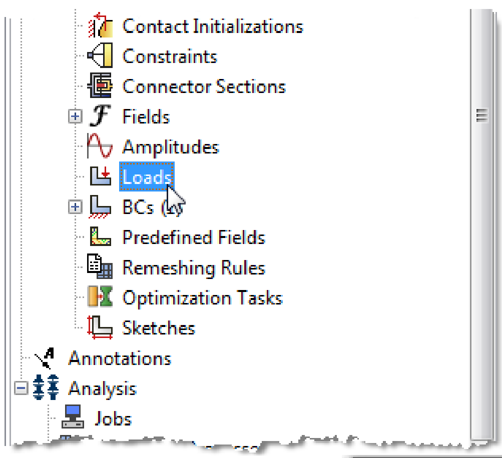
11. Expand the History Output Requests node in the model tree, and then right click on H-Output-1 (H-Output-1 was automatically generated when creating the step) and select Delete

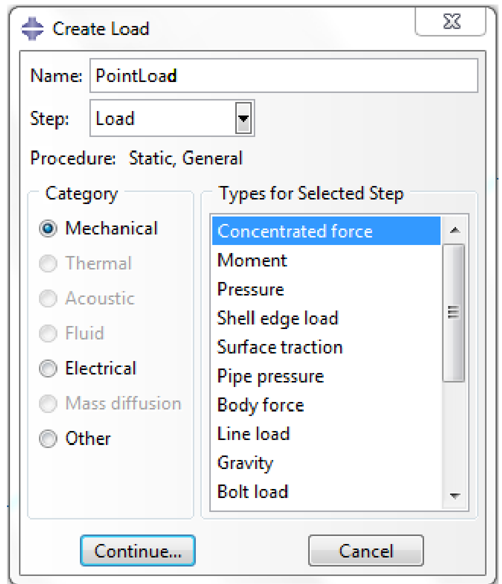


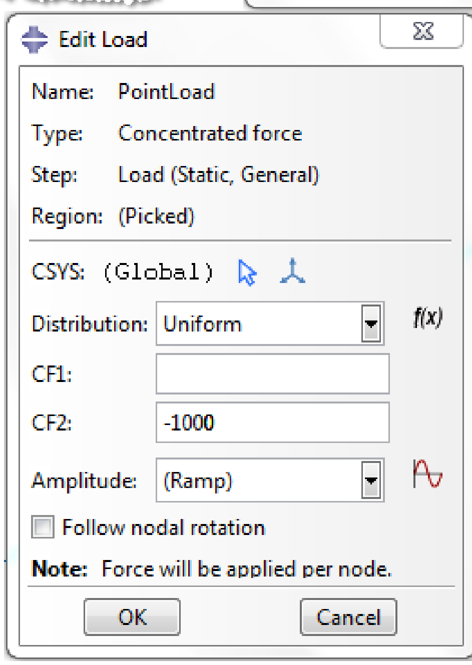
12. Double click on the “BCs” node in the model tree
    a. Name the boundary conditioned “Pinned” and select “Displacement/Rotation” for the type
    b. Click “Continue…”
    c. Select the endpoints on the left (“shift” select) and press “Done” in the prompt area
    d. Check the U1 and U2 displacements and set them to 0
    e. Click “OK”



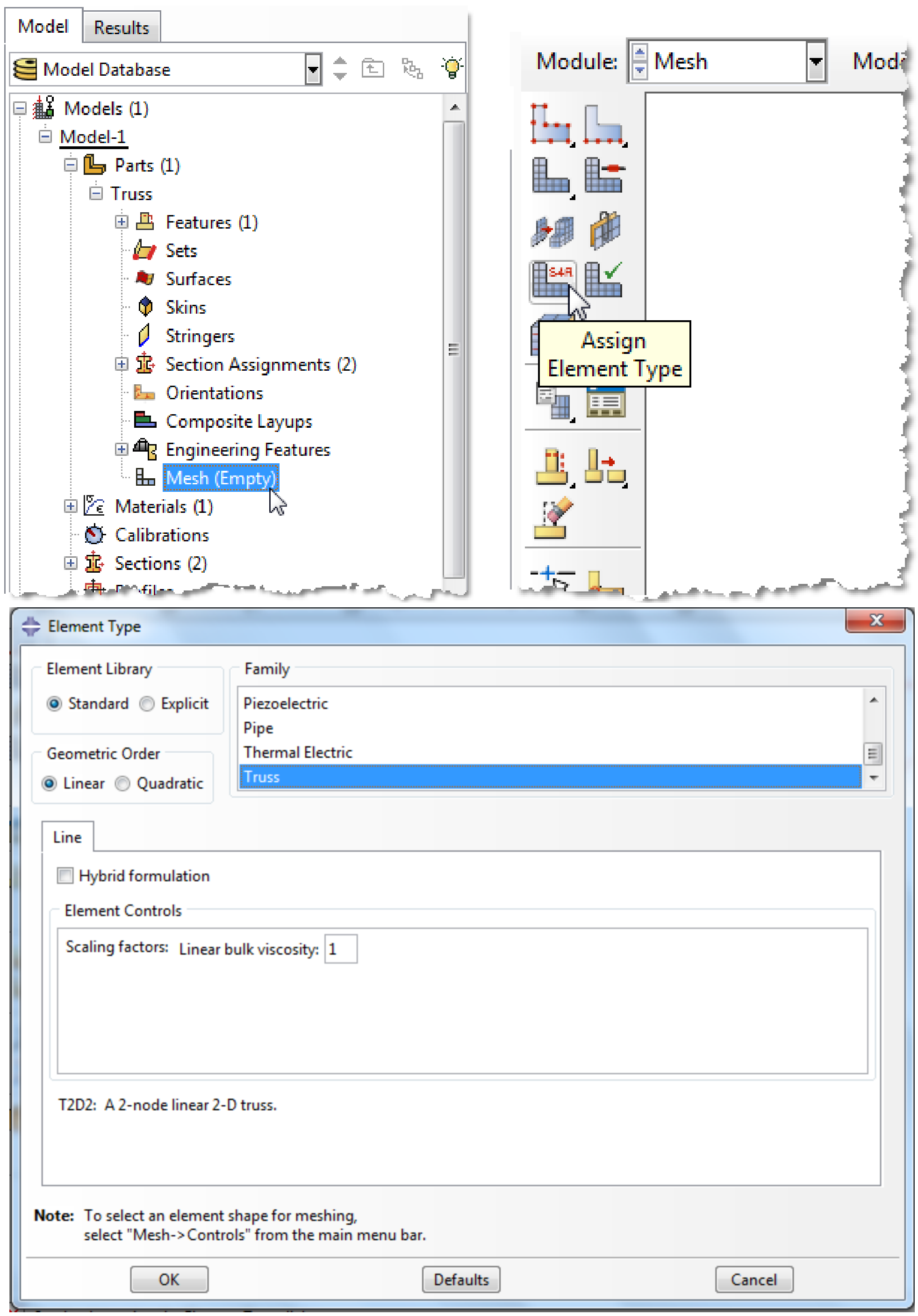
13. Double click on the "Loads" node in the model tree
    a. Name the load "PointLoad" and select "Concentrated force" as the type
    b. Click "Continue…"
    c. Select the vertex on the right and press "Done" in the prompt area
    d. Specify CF2 = -1000
    e. Click "OK"



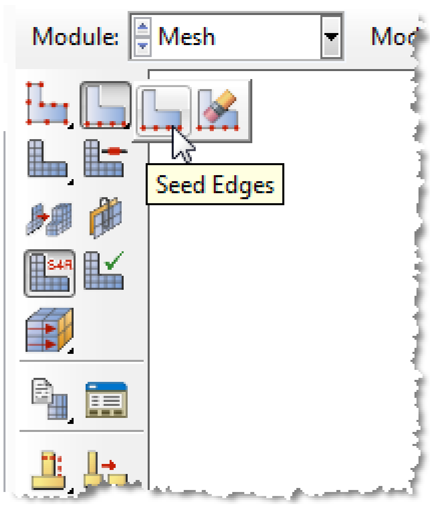


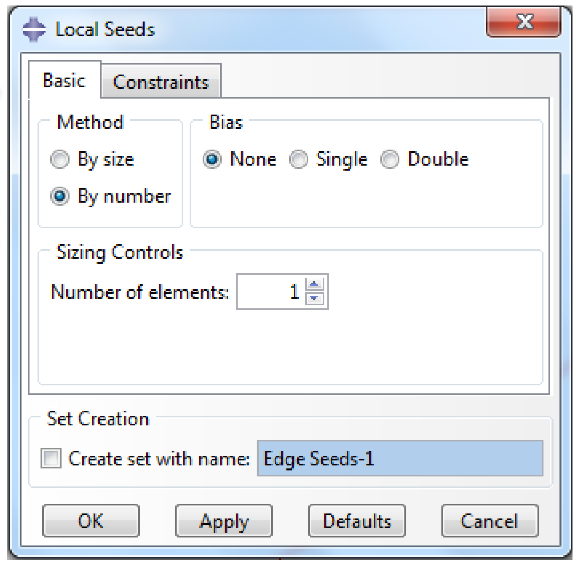


14. In the model tree double click on “Mesh” for the Truss part, and in the toolbox area click on the “Assign Element Type” icon
    a. Select “Standard” for element type
    b. Select “Linear” for geometric order
    c. Select “Truss” for family
    d. Note that the name of the element (B21) and its description are given below the element controls
    e. Click “OK”

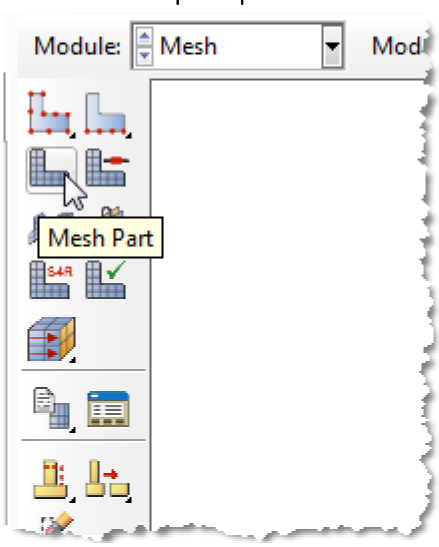


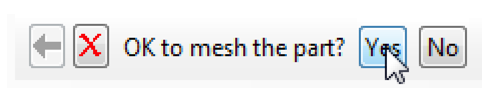
15. In the toolbox area click on the "Seed Edges" icon
    a. Select the entire geometry
    b. Choose method as "By number"
    c. Define the number of elements along the edges as "1"
    d. Click "OK"



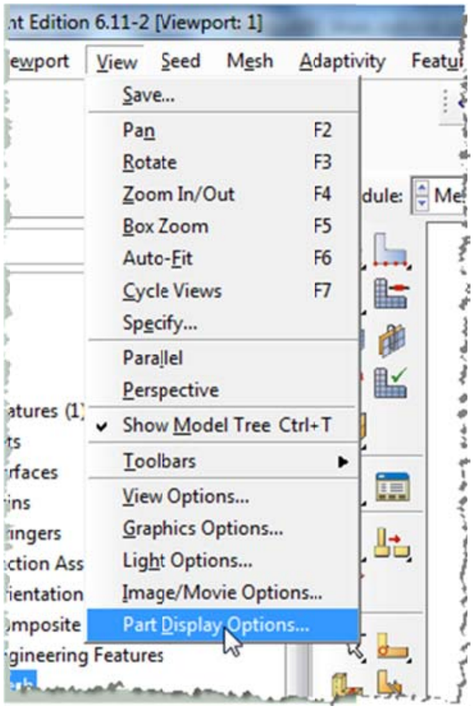


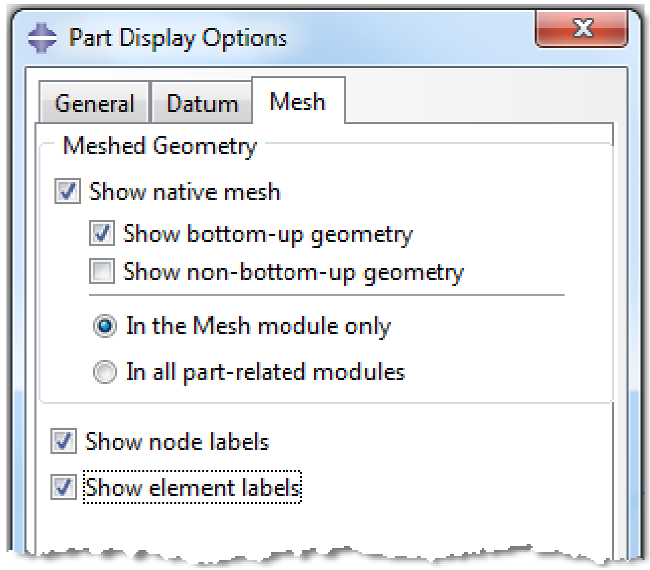
16. In the toolbox area click on the "Mesh Part" icon
    a. Click "Yes" in the prompt area



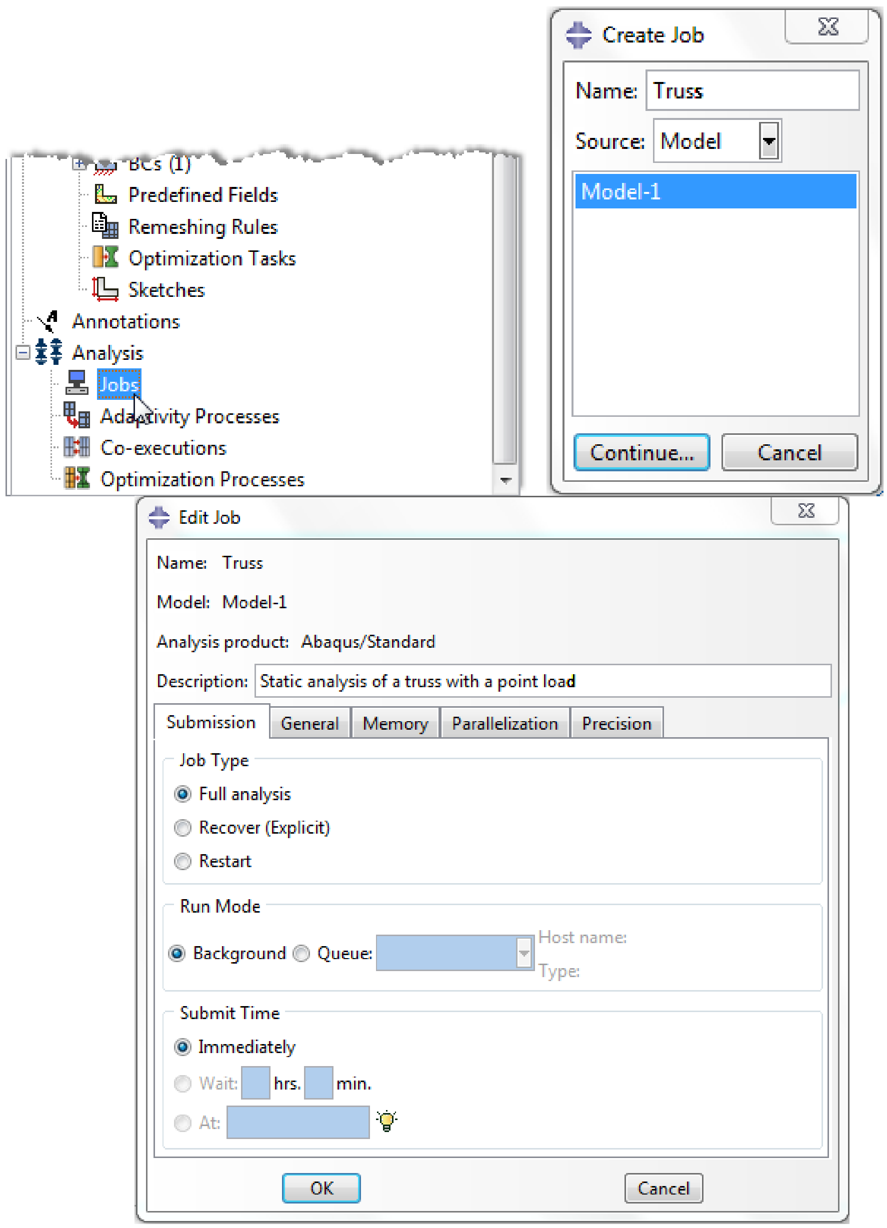


17. In the menu bar select View➔Part Display Options
    a. On the Mesh tab check “Show node labels” and “Show element labels”
    b. Click “OK”

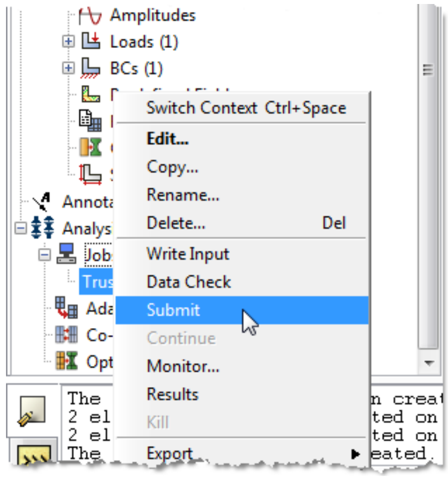


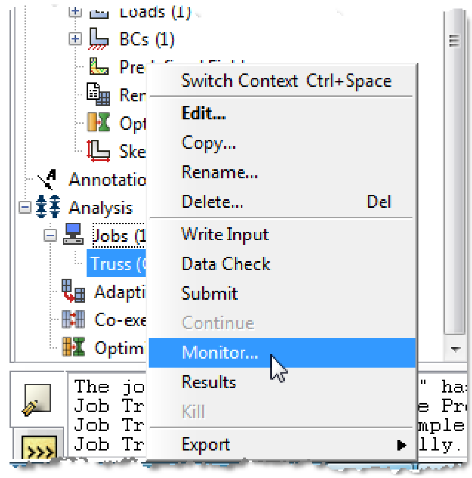


18. In the model tree double click on the “Job” node
    a. Name the job “Truss”
    b. Click “Continue…”
    c. Give the job a description
    d. Click “OK”

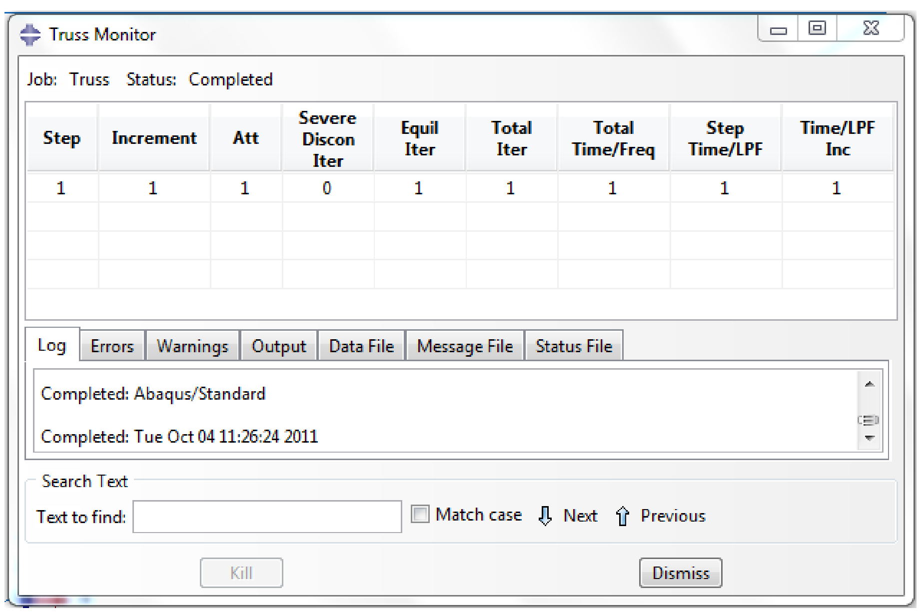


19. In the model tree right click on the job just created (Truss) and select “Submit
    a. While Abaqus is solving the problem right click on the job submitted (Truss), and select “Monitor”

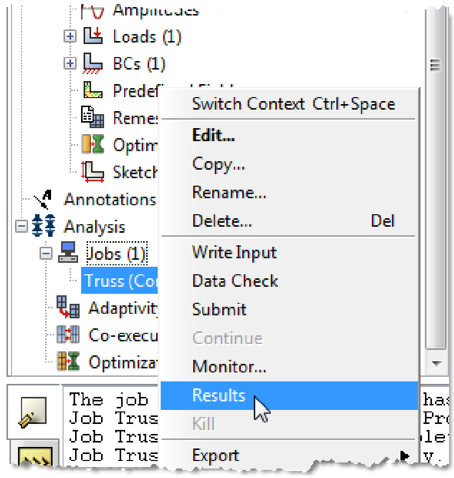




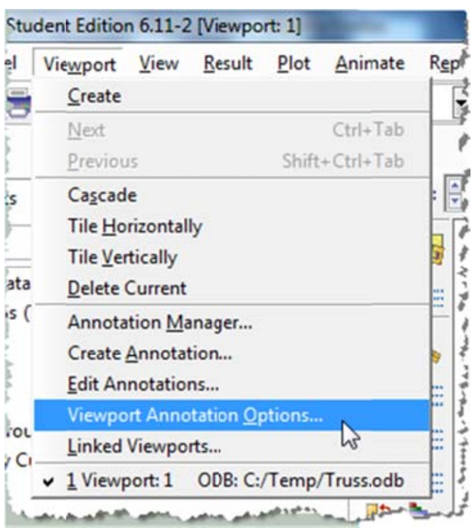
b. In the Monitor window check that there are no errors or warnings
   i. If there are errors, investigate the cause(s) before resolving
   ii. If there are warnings, determine if the warnings are relevant, some warnings can be safely ignored

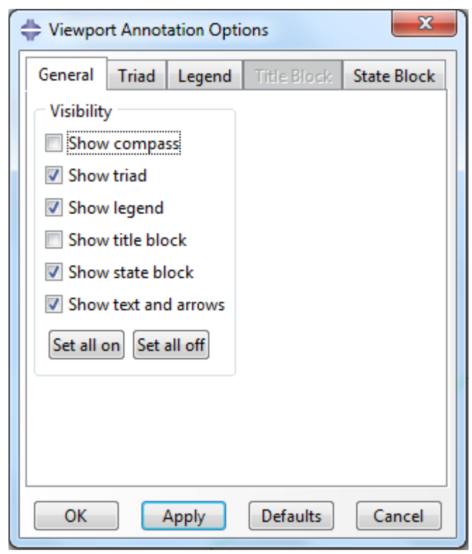


20. In the model tree right click on the submitted and successfully completed job (Truss), and select “Results”

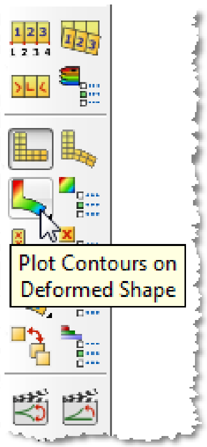


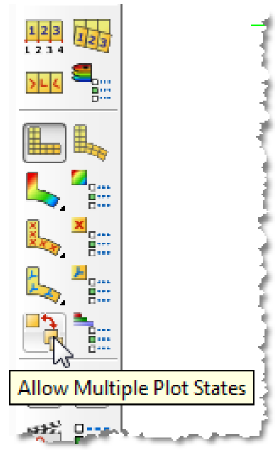
21. In the menu bar click on Viewport➔Viewport Annotations Options
    a. Uncheck the “Show compass option”
    b. The locations of viewport items can be specified on the corresponding tab in the Viewport Annotations Options
    c. Click “OK”

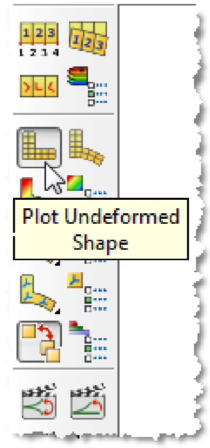




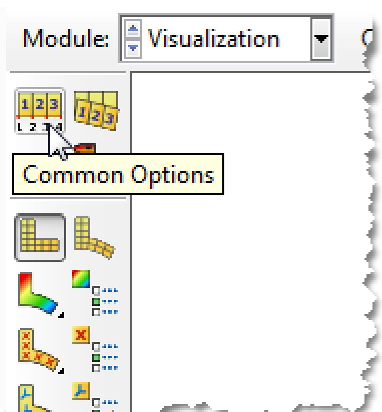
22. Display the deformed contour of the (Von) Mises stress overlaid with the undeformed geometry
    a. In Abaqus 6.11 the default display field output is (Von) Mises stress
    b. In the toolbox area click on the following icons
        i. “Plot Contours on Deformed Shape”
        ii. “Allow Multiple Plot States”
        iii. “Plot Undeformed Shape”

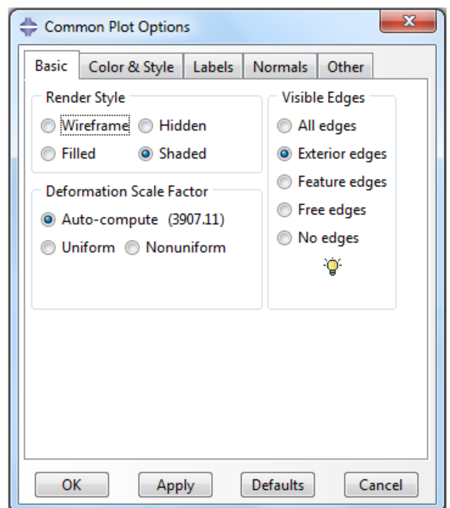


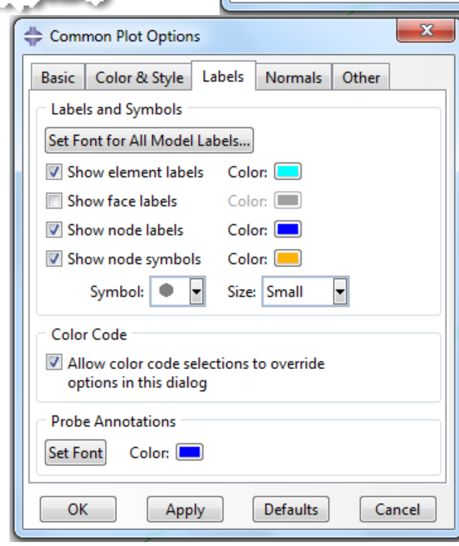


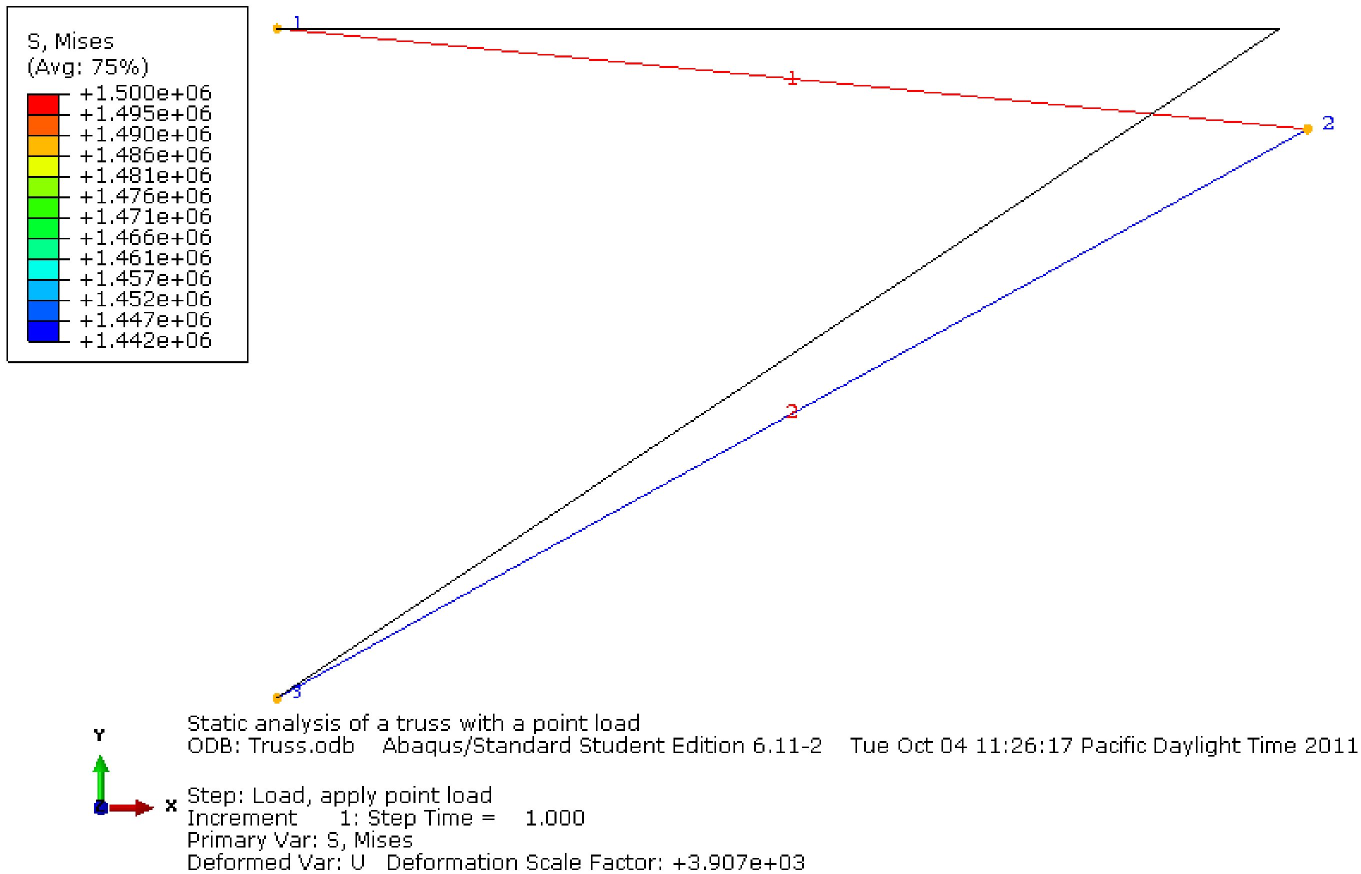


23. In the toolbox area click on the “Common Plot Options” icon
    a. Note that the Deformation Scale Factor can be set on the “Basic” tab
    b. On the “Labels” tab check “Show element labels”, “Show node labels”, and “Show node symbols”
        i. *Note* the default label colors may need to be adjusted for visibility. They can be changed in this option box by clicking on the color and picking a new one.
    c. Click “OK”

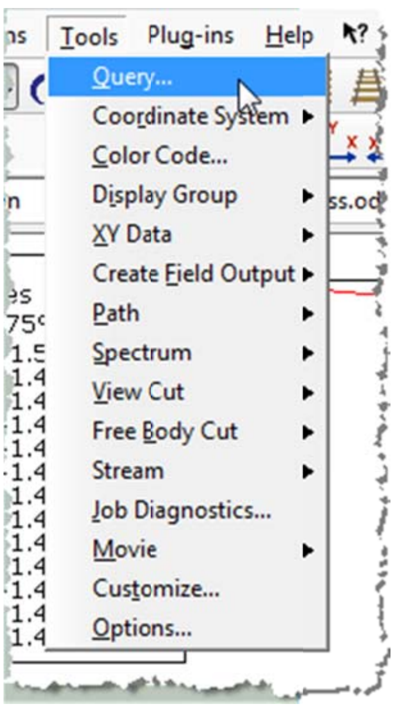


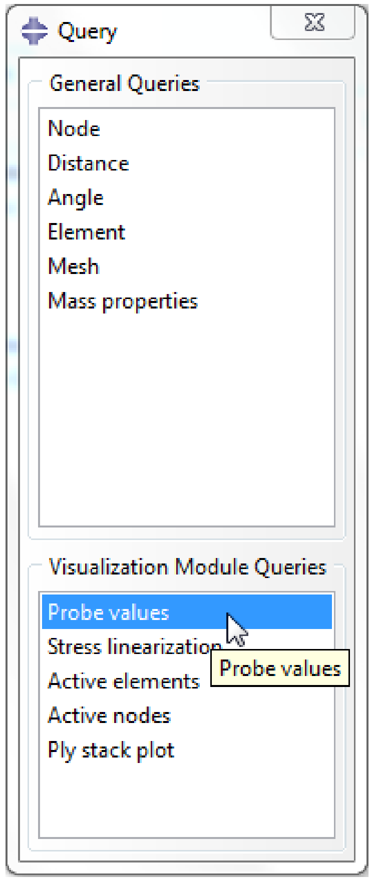


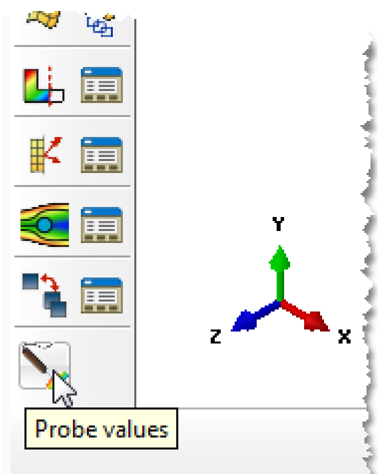



S, Mises
(Avg: 75%)
+1.500e+06
+1.495e+06
+1.490e+06
+1.486e+06
+1.481e+06
+1.476e+06
+1.471e+06
+1.466e+06
+1.461e+06
+1.457e+06
+1.452e+06
+1.447e+06
+1.442e+06
1
2
3
1
2
Y
X
Static analysis of a truss with a point load
ODB: Truss.odb Abaqus/Standard Student Edition 6.11-2 Tue Oct 04 11:26:17 Pacific Daylight Time 2011
Step: Load, apply point load
Increment 1: Step Time = 1.000
Primary Var: S, Mises
Deformed Var: U Deformation Scale Factor: +3.907e+03

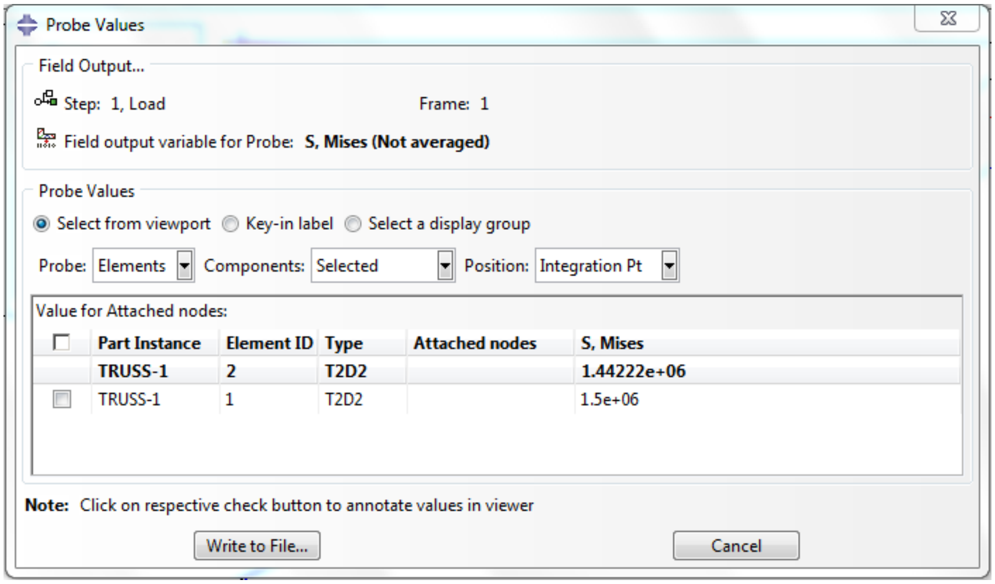
24. To determine the local stress values use the probe feature.
    a. This can be found in two locations
        i. In the menu bar click Tools→Query → Probe Values
        ii. In the vertical icon tool bar





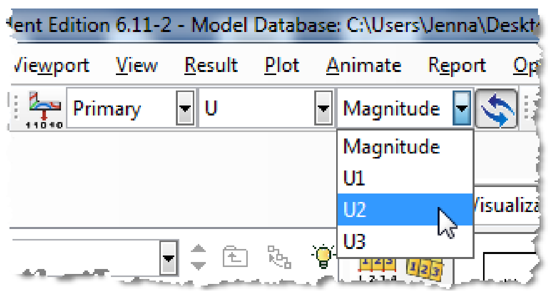


    b. In the viewport mouse over the element of interest
    c. Click on an element to store it in the “Selected Probe Values” portion of the dialogue box
    d. Note that Abaqus reports stress values from the integration points, which may differ slightly from the values determined by projecting values from the 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
        ii. The probe can be set to query either the elements or nodes.
    e. Click “Cancel” and there is no need to save the queried points at this time.



25. To change the displayed output, either:

   a. Change the field output using the tool bar option shortcut



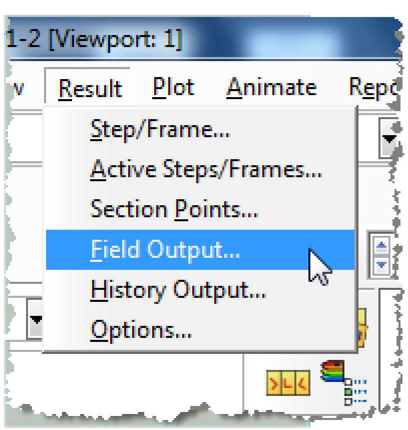
   b. Use the menu bar

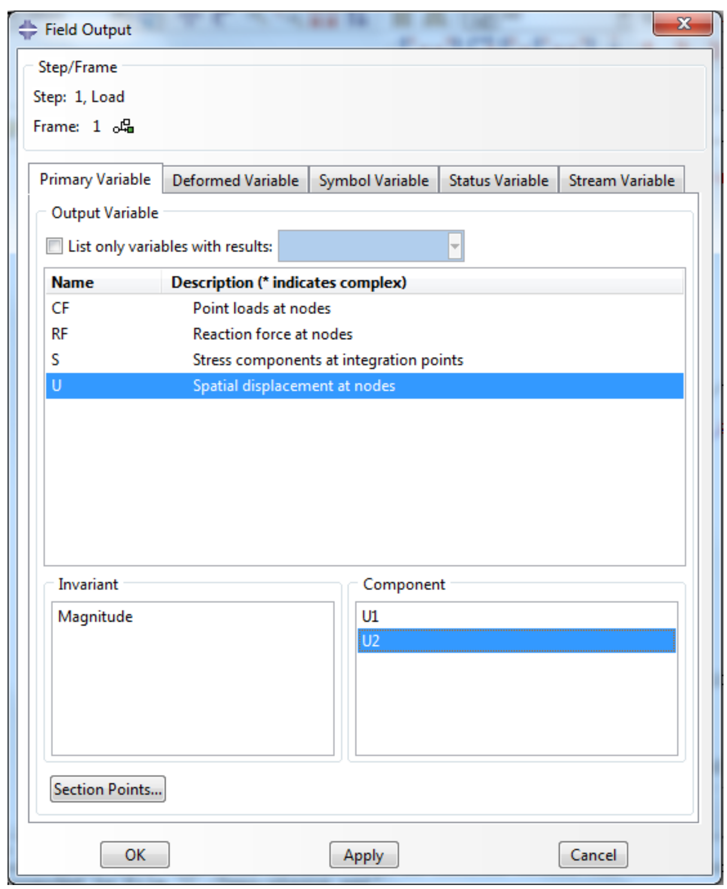
      i. Click on Results→Field Output

      ii. Select "U - Spatial displacement at nodes"

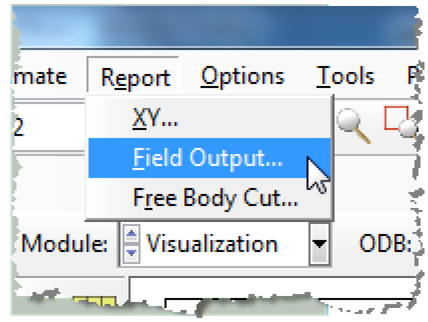
      iii. Component = U2

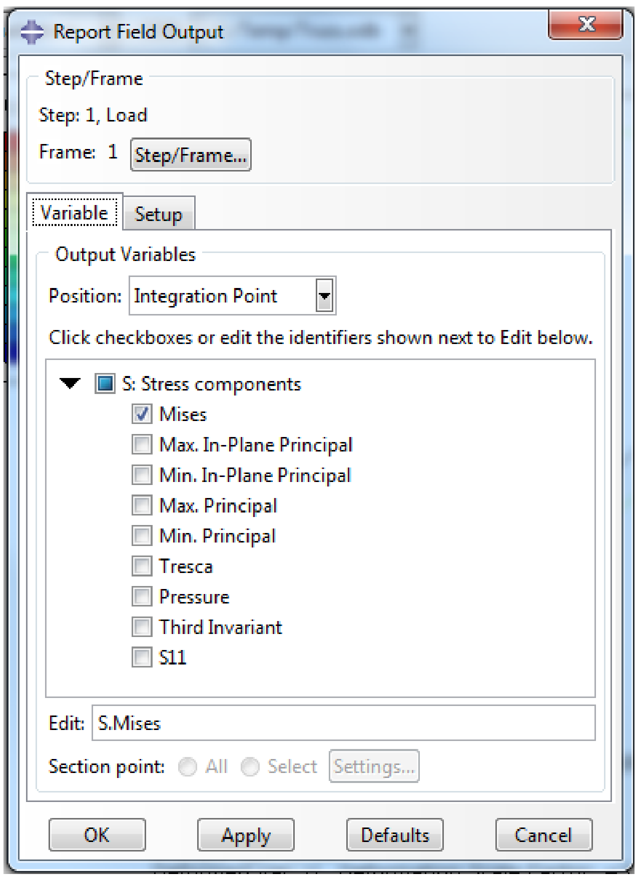
      iv. Click "OK"

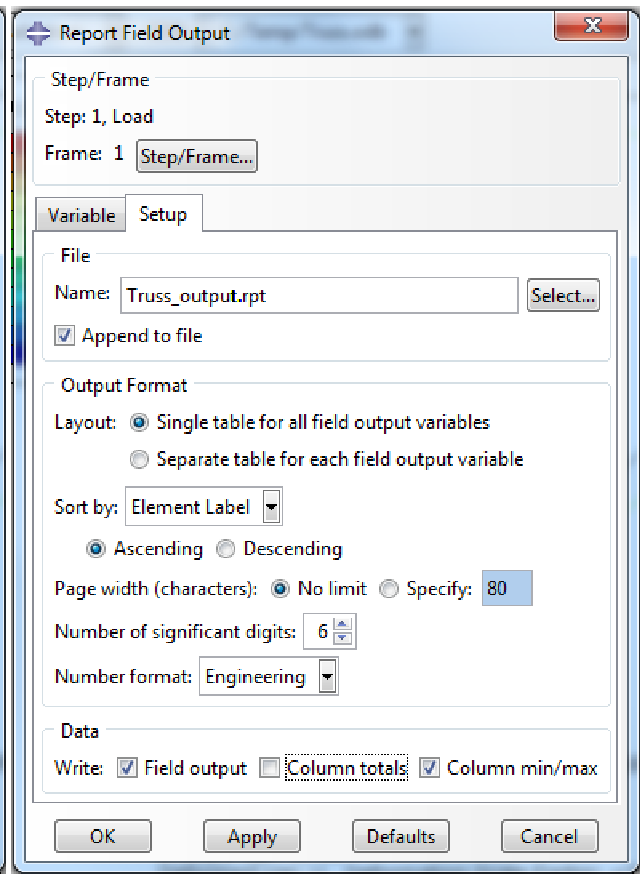




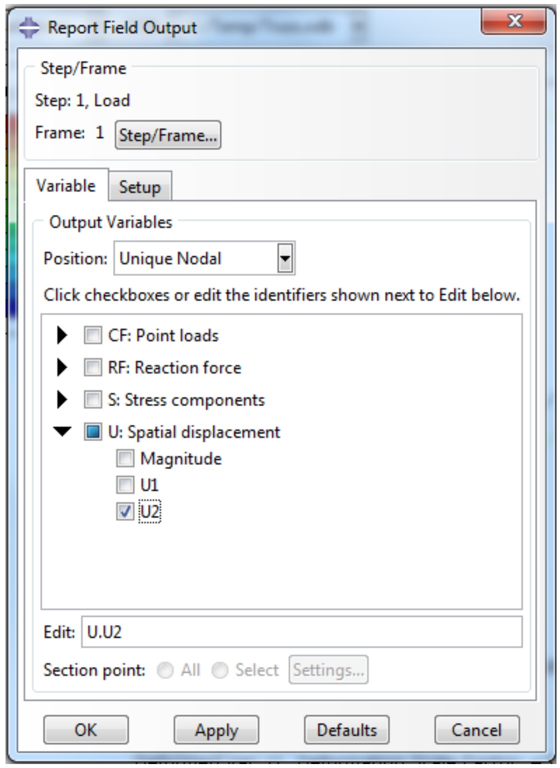
26. To create a text file containing the stresses, vertical displacements, and reaction forces (including the total), in the menu bar click on Report→Field Output
    a. For the output variable select (Von) Mises
    b. On the Setup tab specify the name and the location for the text file
    c. Uncheck the "Column totals" option
    d. Click "Apply"



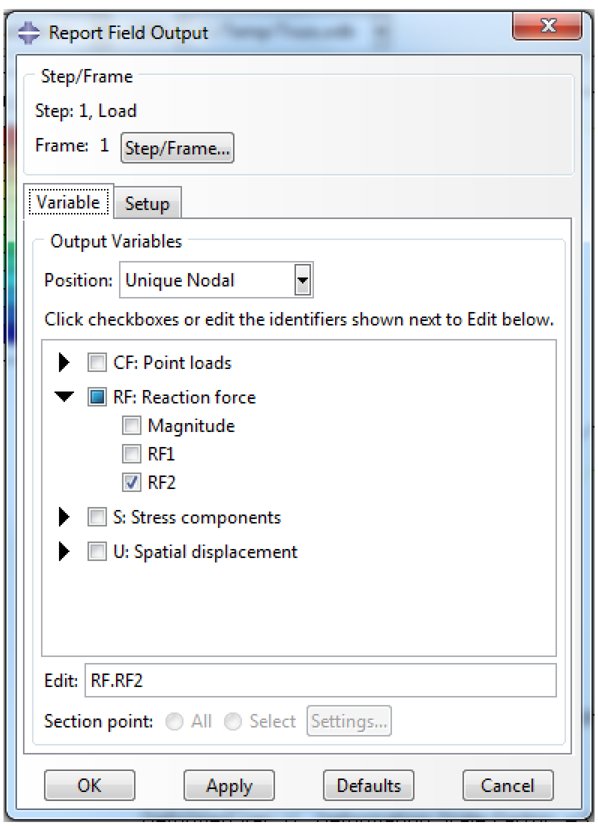


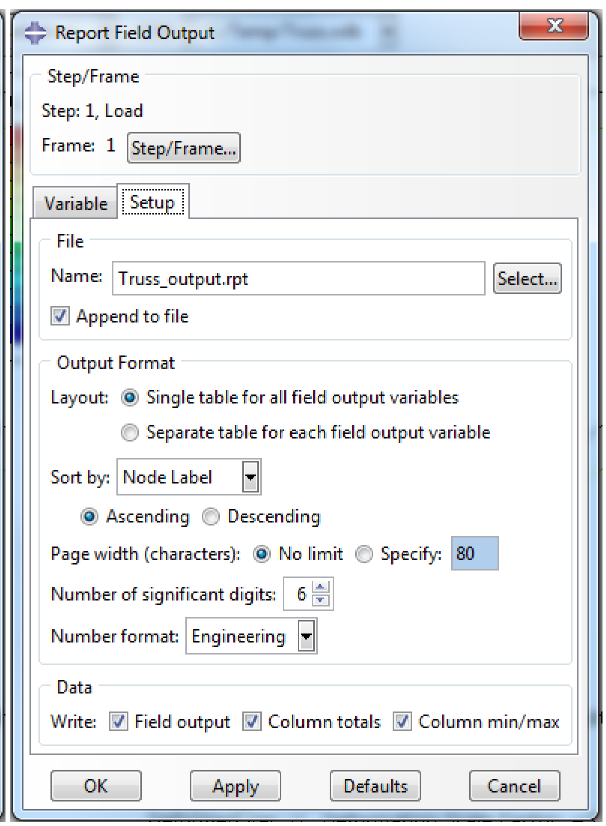


    e. Back on the Variable tab change the position to "Unique Nodal"
    f. Uncheck the stress variable, and select the U2 spatial displacement
    g. Click "Apply"

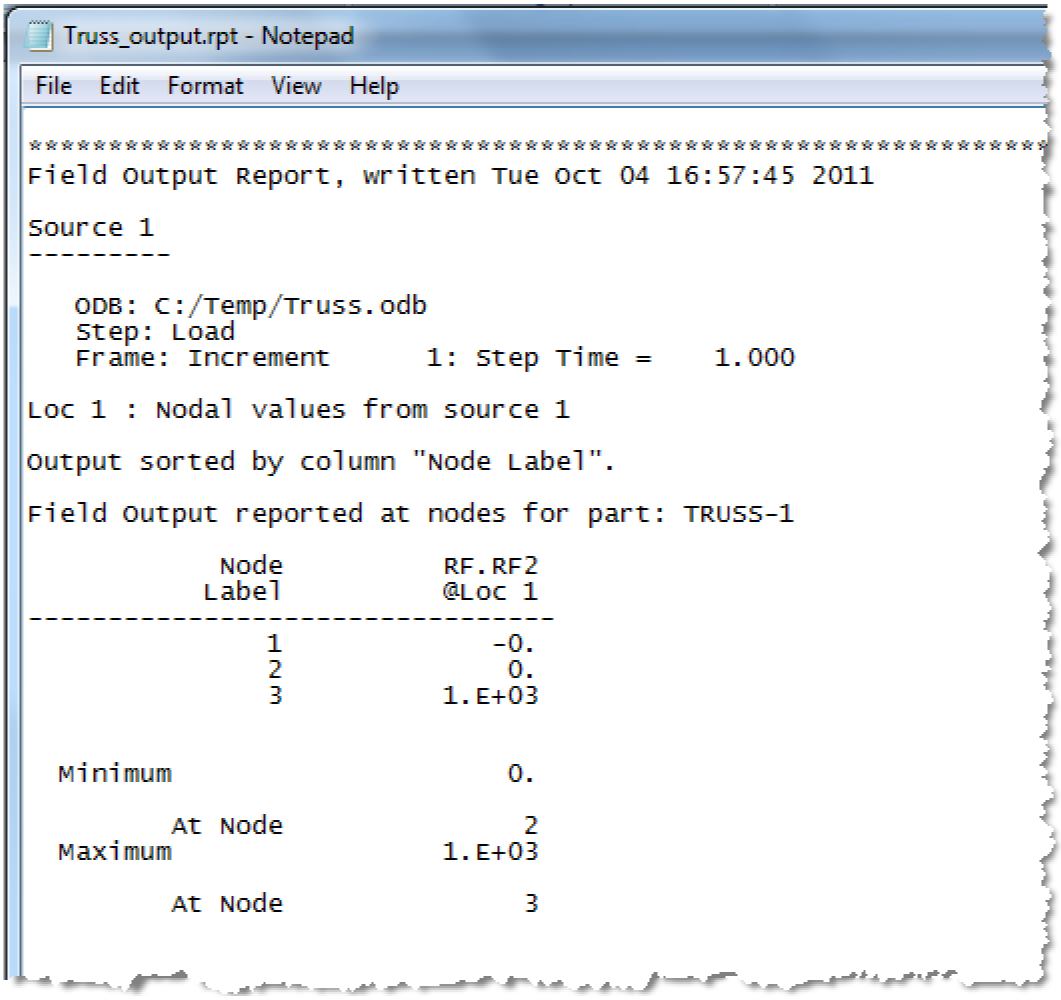


h. On the Variable tab, uncheck Spatial displacement and select the RF2 reaction force
i. On the Setup tab, check the "Column totals" option
j. Click "OK"





27. Open the .rpt file with any text editor (such as Notepad)
    a. One thing to check is that the total downward reaction force is equal to the applied load (1,000 N)
    b. Each time "Apply" is pressed the .rpt file is updated as long as the "Append to file" option is checked under "Setup"



Truss_output.rpt - Notepad

File Edit Format View Help

```
********************************************************************
Field Output Report, written Tue Oct 04 16:57:45 2011

Source 1
---------

   ODB: C:/Temp/Truss.odb
   Step: Load
   Frame: Increment      1: Step Time =    1.000

Loc 1 : Nodal values from source 1

Output sorted by column "Node Label".

Field Output reported at nodes for part: TRUSS-1

            Node          RF.RF2
           Label          @Loc 1
---------------------------------
               1             -0.
               2              0.
               3          1.E+03


  Minimum                     0.

         At Node               2
  Maximum                 1.E+03

         At Node               3
```