

2.31 Assignment 2

OPTIONAL

Due Monday 9/17 at 9:30 am

- 1) Follow the step by step instructions in sections 2.3.3 → 2.7 of Handout 4 to run your first FE analysis. Following the instructions in section 2.3.6 (page 2-22) edit the settings in the **Field Output Request Manager**. Add the axial strain in the elements (E11) to the list of variables that will be output to the **.odb** file.
- 2) Use the results in the printed output file (**.dat**) to calculate the axial forces in each of the members of the truss structure. Sketch the structure indicating magnitude and direction of the axial forces in the trusses. In the **.dat** file read the reaction forces at the constrained nodes. Add these forces to your sketch. Draw force diagrams at each of the nodes of the structure and check if the nodes are in equilibrium by summing the forces along the 1-axis and the 2-axis of the global coordinate system. Check if the entire structure is in equilibrium (remember to check global equilibrium with respect to the 1-direction, the 2-direction and rotation around the 3-axis).
- 3) Using the ABAQUS/Viewer postprocessor access the results in the **.odb** file. Display the deformed shape superposed to the undeformed shape. Using the **Query** tool obtain the axial strains (E11) in each element. Check if the values are consistent with the stresses printed in the **.dat** file and with the value you assigned for the Young's modulus.
- 4) If the yield stress for the steel is 600 MPa, up to what load can you continue to analyze this structure using a linear perturbation procedure? (neglect geometric sources of nonlinearity)