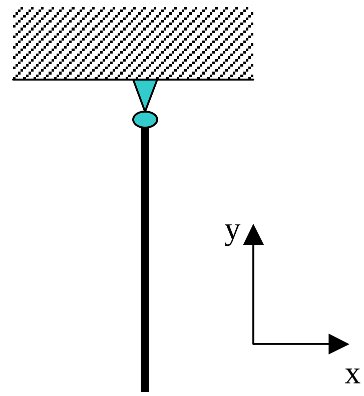
2.31 Assignment 3

Due Monday 9/25 at 9:30 am



The uniform bar shown hangs from a hinge under its own weight. The length of the bar is L =1m; its cross section has an area $A=1mm^2$; the material is linear elastic with a Young's modulus E=200GPa, and Poisson ratio ν=0.3. The density of the bar is $\rho=7000$ Kg/m^3.

1) **<u>Pen and paper work (no FE)</u>** {PLEASE obtain results in symbolic form (i.e in term of A, E, L, ρ..) first and <u>then</u> substitute in the numerical values!}

   a. Calculate the reaction force at the hinge F_{hinge}.
   b. Calculate and plot the axial stress profile as a function of position y (y=0→L) along the bar, $\sigma(y)$.
   c. Calculate and plot the corresponding strain profile, $\varepsilon(y)$.
   d. Integrate the strain profile to obtain the displacement profile along the bar, v(y): remember that $\varepsilon=dv/dy$ in 1D. Plot the profile and obtain the displacement at the lower end of the bar, δ_{max}, and the deflection at the center of the bar, δ_{mid}.

2) **<u>FE model</u>**

   a. Create a coarse FE model of the bar as a truss structure in 2D space using ABAQUS/CAE.
   Things to keep in mind as you set up the model:

PROPERTY :	In section property beware of units. In material properties you have to input Mechanical props (E, ν) as well as the density (under the General tab).
STEP :	Choose static, linear perturbation.
LOAD :	When you create the Load, you must choose gravity (not concentrated load), select the entire part, and give as magnitude of the load the gravity acceleration (which has components {0, -9.8 m/s} in the coordinate system of the sketch above). . When you create BCs be careful. Think of what dof you must constrain.
MESH :	Seed the assembly so as to have a single element for the whole bar. (Use standard linear truss elements).

JOB : Create a job called *onelem* (or something of this sort). Click on the preprocessor printout options so you get your model in the text file *onelem.dat*
Use the keyword editor (bottom of page 2-28 of handout) to request that ABAQUS prints to the *.dat* file the stresses, strains, displacements and reaction forces:

```
*EL PRINT, FREQ=1
S,
E,
*NODE PRINT, FREQ=1
U,
RF,
```

Submit the analysis

Look at the results in the *onelem.dat* file and compare them with the analytical results from part 1:

<u>Forces</u>: is F_{hinge} equal to the reaction force at the highest node?
<u>Displacements</u>: is δ_{max} equal to the displacement at the lowest node? Do we have any FE displacement information at the bar midpoint ?
<u>Stresses and strains</u>: superpose a plot of the FE stress profile to the analytical stress profile: how do they compare? Why is the FE stress not equal to the total weight divided by the bar cross sectional area?

Comments on what you think are the reasons for which some results match while others do not.

b. Repeat the whole thing you did in 2.a but now change mesh refinement by seeding the part so as to have 10 truss elements along the bar. Note that you do not need to redo everything. Just back up to the mesh module and change only the seeding of the part and the mesh. Resubmit the job **<u>with a different name</u>** otherwise you overwrite the other results (*tenelem* seems like a great choice to me). Look at the results in the new *tenelem.dat* file and compare them with the analytical results from part 1 and the FE results in part 2.a

   Comments on the differences and where they come from.

c. Say that you have to analyze this structure and your main design concern is max deflection. What mesh would you use? Why? Alternatively, say that you are concerned with possible failure and you are worried about the stress levels. What mesh should you choose? Why?