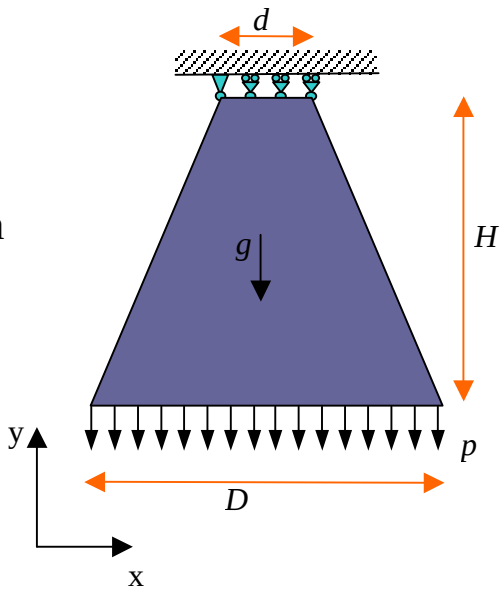
2.31 Assignment 4

Due Wed, Oct 3 at 9:30 am



The trapezoidal plate shown hangs under its own weight, plus an applied distributed load, p = 0.1 MPa .
The in-plane dimensions of the plate are: D = 10 m, H=10 m, d=2 m, and the plate thickness is t= 1m. The material is linear elastic with a Young's modulus E=200GPa, and Poisson ratio ν= 0.3. The density of the plate is ρ=7140 Kg/m^3.

1) **Pen and paper work (no FE)**
{PLEASE obtain results in symbolic form first and then substitute in the numerical values!}

Consider a simplified 1D model of the plate where you consider only one component of stress $\sigma = \sigma_{yy}$, and idealize the plate as a 'bar' of variable cross section A(y)

a. Calculate the total vertical reaction at the support.
b. Calculate and plot the axial stress profile as a function of position y (y=0→H) along the bar, σ(y).
c. What is the max stress level and where is it located?

2) **FE model**

a. Create a coarse FE model of the plate as a plane stress structure in 2D space using ABAQUS/CAE.

Things to keep in mind as you set up the model:

PROPERTY : In section property you want to create a *solid homogeneous* section of thickness 1m.
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 weight load, you must choose gravity, 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 the distributed load, *p*, choose a pressure load with

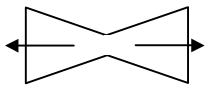
negative magnitude (you are pulling, not pushing on the surface).

MESH : Seed the assembly with a global element size of 1.0. In the mesh dialog choose standard linear quads, and make sure you click off the reduced integration option. (you should have CPS4 as element type)

JOB : Create a job called *linearcoarse* (or something of this sort). Click on the preprocessor printout options so you get your model in the text file *linearcoarse.dat*
Do not write anything to the *.dat* file. Submit the analysis

Look at the results in the VISUALIZATION module of ABAQUS/CAE.

a) Plot and print the contours of s22 (=σ_{yy}) over the entire plate → attach the plot to your assignment. Comment on the differences between the FE stress field and your pen and paper calculations. Where do you think the main discrepancies are coming from?
[hint: our plate is essentially ½ of this structure →



b) Plot and print the profile of the vertical reaction forces along the top of the plate. Check if the sum of the reaction forces is equal to your estimate of the total vertical reaction at the support. Here is how to do it:

- Start by displaying the undeformed mesh and toggling on the *show node symbol* button under **Option→Undeformed Shape→Labels**
- Now you need to define a path along the top of the plate using the **path** tool in the **Tool** menu. The easiest approach is to create the path using a **node list.** You can generate the list by clicking on the *select* button and then picking sequential nodes along the path on the top of the plate.
- To create the X-Y plot of y-reaction forces along the path use the **path** option under **Tools→X-Y data→Create**. In the dialog box that pops up, choose your path, and click on the *Field Output* button to select the y-reaction force (RF2) as the plot variable. Click on *Plot* if you want to see the plot and print it. Attach this plot to your assignment.
  Save your XY data using the *Save As* button, say that you call the curve tops22. Now the x-y data are available to print.
- Using **Report→XY** gives you a dialog box where you can select the curve, tops22, for which you want to print the x-y values to file, as well as the name of the print file (under the *Setup* tab). This will be an ASCII file. Attach a printout of this file to your assignment. Sum up the forces to obtain the total.

How does the FE prediction compare w/ your 1-D estimate? Why?