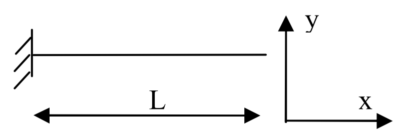
ES 128: Computer Assignment #5

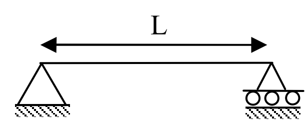
Due in class on Monday, 26 April 2010

When you use ABAQUS CAE to solve a natural frequency problem, the way to do this kind of analysis is: When you create a step: Select procedure type Linear perturbation→Select Frequency.

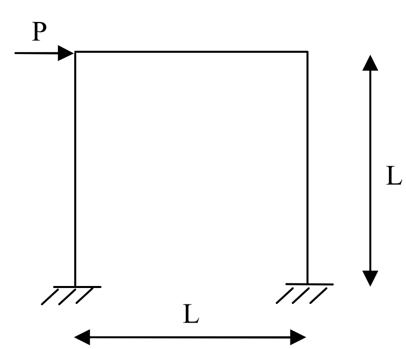
Task 1. Use ABAQUS to get the natural frequency for a bar under axial loading. The analytical solution is given as $f_1 = \frac{1}{4L}\sqrt{\frac{E}{\rho}}$. Use 1 element, 2 elements and so on till 10 elements to get the first natural frequency. You can model this bar as a truss. Plot your dimensionless frequency $\frac{f_1 L\sqrt{\rho}}{\sqrt{E}}$ as a function of the element number.



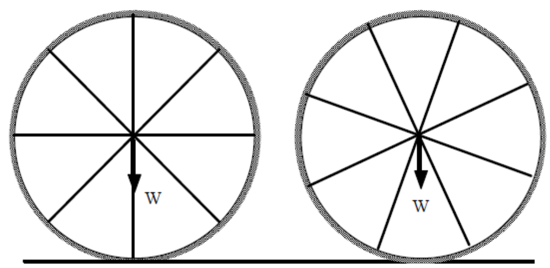
Task 2. Consider a beam with one end fixed and the other end simply supported. The analytical solution of the first natural frequency is $f_1 = \frac{\pi}{2L^2}\sqrt{\frac{EI}{\rho A}}$. Similar to the problem 1, plot your dimensionless frequency $\frac{f_1 L^2\sqrt{\rho A}}{\sqrt{EI}}$ as a function of the element number from 1 to 10.



Task 3. Use CAE to determine the first 3 normal mode of the frame. Plot the deformed shape of these modes. Print the natural frequencies associated with these modes.



Task 4. An eight-spoke wheel of 30 cm radius, intended to support weight W=10000 N, is shown in two positions. For an approximate design analysis, the spokes are modeled as truss elements and the rim as a uniform, initially circular, beam. (In practice one would also worry about local stress concentrations where the spokes are attached to the rim) The beam (rim) has a square cross section (2x2 cm^2) and is made of the same elastic material as the spokes (The Young's modulus is 200 GPa). The cross-section area of the spokes is 1 cm^2. The contact interaction between the wheel and ground is ignored in this problem. We fix the displacements and rotations of the node that is in contact with the ground.



We start by using only 8 beam elements (1 per 45° sector) and we calculate the downward displacement of the center of the wheel towards the floor and the maximum spoke tensile and compressive stresses (unit?).

Since we are using only 8 beam elements the solution is inaccurate. The issue here is that the beam elements are rather long. Introduce more beam elements – e.g., 2, 5, and 10 elements per 45° sector – until you are sure that the solution for stress and displacement is essentially independent (within 5% accuracy) of the degree of refinement. Your final mesh density is the *optimized* mesh density that yields both accuracy and efficiency. Support your conclusions by preparing tables showing the dependence of

· downward displacement of the center of the wheel towards the floor,
· maximum spoke tensile and compressive stresses (unit?), and
· maximum stress (unit?) in the rim,

on the number of elements in a sector. Also, be sure to indicate the locations of the stresses shown in your table.

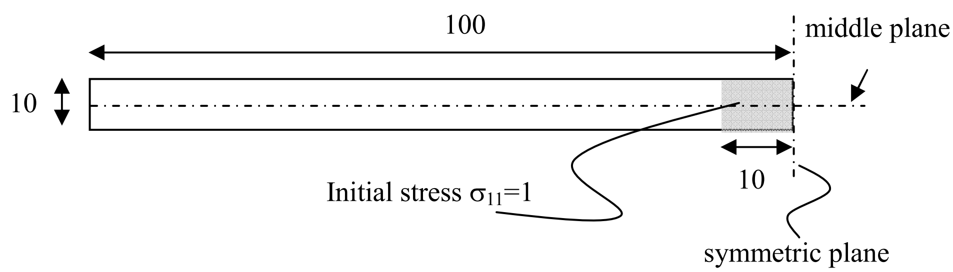
Use the optimized mesh density found above to rewrite your input file to solve the configuration shown on the right when the wheel rotates through 22.5°. Find out the maximum stresses in the rim and spokes and how much does the wheel center displace.

Let us determine the free vibration frequencies and mode shapes of the wheel in the first (0°) and second (22.5°) configuration.

Task 5. Now you will solve an initial value problem. Here you are asked to solve a plane stress problem for a long plate of dimension 200×10. The middle part of length 20 is under initial uniform stress $\sigma_{11} = 0.01$. The density is 1, Young's modulus $E = 1$. In the step module, select a step time period to be 500. The initial velocity in ABAQUS by default is zero.

A). For Poisson's ratio ν=0 and ν=0.3, watch the movie of stress σ_{11}, describe in words what you see in the whole process.

B). Take snapshots of the σ_{11} distribution along the middle plane of for three different time t=0, 50, 150. Plot them on a single figure. To do that, you need to create a path along the middle plane.



Some tips:

1) You just need to model half of the structure since the structure is symmetric in x direction.

2) To add an initial stress to the middle part.

   2.1). You need to partition the surface in the Part module: B72 (Partition face: Sketch)→draw a vertical line through the plate.

   2.2). In the Assembly module→go to Menu tools→Set→Create Set→give the set name as: middle→continue→select the middle part→click done.

   2.3). In Assembly module, go to menu→Model→Edit Keyswords→Right after
   *Elastic
   1.0,0.
   →add the following two lines
   *initial conditions, type=stress
   middle, 0.01
   (For detailed information, please refer to Keywords manual.)
   →click ok.

3) About mesh. In the mesh module, B12→Select element type to be Quad, Technique to be Sweep, so that you have rectangular meshes.