University of California, Berkeley
CEE C133/ME C180, Engineering Analysis Using the Finite Element Method
Spring 2010
Instructor: S. Govindjee
GSI: N. Hodge

Lab 1: Introduction to FEA and COMSOL

- General comments
  - COMSOL has a nice, menu driven GUI, and is similar to other commercial FEA codes in this regard.
  - COMSOL has *a lot* of features and options; but, since it hasn't been around as long, it probably also has more bugs.
- COMSOL Model Navigator
  - Note that "plane stress" is under both "COMSOL Multiphysics" and "Structural Mechanics". The difference is that different application modes can have different menus, as well as different default values in the dialog boxes.
  - Even the sub-modes can be different: check out the two different plane stress entries. Note the description on the right hand side of the dialog box.
- main menus (correspond nicely to major tasks in FEA)
  - Draw (create raw geometry)
  - Physics (define data for the problem)
  - Mesh (discretize the geometrry)
  - Solve
  - Postprocessing
- geometric modeling (Draw mode)
  - "solid" entities: Not much to say . . .
  - modeling tools
    * Union
    * Intersection
    * Difference
    * Note that modeling tools do not allow for interactions between objects of different dimensionality (i.e., lines can only interact with lines, surfaces can only interact with surfaces, etc.).
  - line entities: right click to close, shift-right click to leave as curves

- Physics settings
  - A well posed problem in solid mechanics needs both *material properties* and *boundary conditions*
  - In COMSOL, problem data must be applied directly to the geometry
  - Material properties are found under "Subdomain Settings" (menu or double click on a domain)
  - Boundary conditions are found under "Boundary Settings" (menu or double click on a boundary)
- Mesh operations
  - For most of this class, we will be using auto-generated meshes, which are nearly always triangular; note that there is a "free, quad" meshing mode.
  - Check out "Mesh/Interactive Meshing" for lots of control.
  - Create mesh
  - Refine mesh
- Solution
  - Check out the "Solve/Solver Parameters" dialog box
- Post processing
  - Check the "Plot Parameters" dialog box
  - Type of plot: can turn each type of plot on and off individually.
  - Quantity to plot
  - Plot type must be appropriate for the quantity to be plotted, e.g., is a vector plot ok for stresses???
  - One significant uniqueness of solid mechanics: the domain is moving; check out the deformed shape plot.

Exercise

- One last note: COMSOL has a nice help facility
- Navigate to
  - Structural Mechanics Module
  - Model Library (HTML)
  - Benchmark Models
  - Large Deformation Beam

Set up and run this model, including the parametric solution. Stop at the section labeled "COMPUTING THE LINEARIZED BUCKLING LOAD".

Before you leave class, be sure to turn in the following to me: a deformed plot of the beam, with scale factor 0.1, and any two solution quantities that you like, as long as they can be easily discerned. Be sure to write your name on the plot you turn in.