University of California, Berkeley
CEE C133/ME C180, Engineering Analysis Using the Finite Element Method
Spring 2009
Instructor: S. Govindjee
GSI: N. Hodge

Lab 4

This week, you will be implementing another model in COMSOL. The model is located in the Help facility, under

Structural Mechanics Module/Model Library/Nonlinear Material Models/Hyperelastic Seal.

This model contains some additional geometric modeling features, contact, and a nonlinear material model. Your assignment is as follows:

- Implement and run the model as described in the help.
- *Do not* implement the second stage of the solution, which is described after **Adding Internal Pressure**.
- Be sure to save your work in an *.mph* file.
- Send me your *.mph* file via email, and hand in a single plot of the deformed geometry, a surface plot of the von Mises stress, and an arrow plot on the subdomains of the displacements, by the end of lecture on Thursday, February 19.

Note that I will be in the lab during the regular time on Tuesday, Feburary 17. Persons enrolled in the Monday lab section are free to come to the Tuesday lab section to ask questions about this weeks assignment.