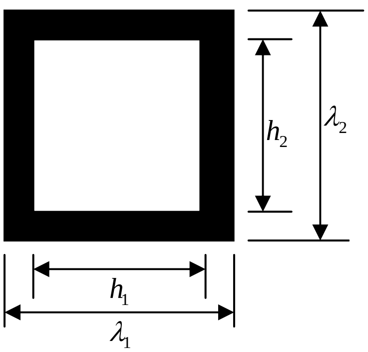
The University of Iowa
College of Engineering
53:236 Optimization of Structural Systems
Fall Semester 2007

Assignment #1: Due: 11/9/2007

In this assignment, you will have the opportunity to use strain-controlled computational homogenization to calculate all nine components (actually, six due to symmetry) of a solid-void composite's effective elasticity tensor $\mathbf{E}^Y$ under plane stress conditions:

$$\begin{bmatrix} \sigma_{11}^Y \\ \sigma_{22}^Y \\ \sigma_{12}^Y \end{bmatrix} = \begin{bmatrix} E_{11}^Y & E_{12}^Y & E_{13}^Y \\ E_{21}^Y & E_{22}^Y & E_{23}^Y \\ E_{31}^Y & E_{32}^Y & E_{33}^Y \end{bmatrix} \cdot \begin{bmatrix} \varepsilon_{11}^Y \\ \varepsilon_{22}^Y \\ \gamma_{12}^Y \end{bmatrix} \tag{1}$$



$$a_1 = \frac{h_1}{\lambda_1}; \quad a_2 = \frac{h_2}{\lambda_2}$$

Figure 1: Unit cell of a solid-void composite in which the black region is solid (E=100 GPa; ν=0.30) and the white region is void (E=.01 GPa; ν= 0.40)

Each student will be expected to compute the effective elasticity tensor $\mathbf{E}^Y$ for a range of six different unit cell parameters listed in Table 1.

Table 1. Microstructure parameters for which $\mathbf{E}^Y$ is to be computed.

Student No.	a_1	a_2
1	0.20	0.0, 0.20, 0.40, 0.60, 0.80, 1.0
2	0.40	0.0, 0.20, 0.40, 0.60, 0.80, 1.0
3	0.60	0.0, 0.20, 0.40, 0.60, 0.80, 1.0
4	0.80	0.0, 0.20, 0.40, 0.60, 0.80, 1.0
5	1.00	0.0, 0.20, 0.40, 0.60, 0.80, 1.0

This assignment will be carried out using the finite element code FENDAC in which various tasks needed for strain-controlled unit-cell homogenization have already been implemented. You can find the manual for this program on the class website at:

www.engineering.uiowa.edu/~swan/courses/53236/53236.html

To get started, a sample data file is provided in which a macroscopic strain of $\varepsilon_{11}^{Y} = 0.01$ is applied to the unit cell for (a_1=a_2=0.90).

To execute the finite element program FENDAC and to interpret its results, you will need to carefully step through the following process:

1. Log on to any ECSS LINUX work stations (l-lnx000 through l-lnx019).
2. Obtain a copy of the FEM data file typing:
   `cp -p /user/class/cee5330/examples/53_236/hw1/e11.data .`
3. Obtain a copy of a C-shell script by typing:
   `cp -p /user/class/cee5330/examples/53_236/hw1/setup53236 .`
4. To make sure you're running in a C-shell, type: `csh [Enter]`.
5. Execute the setup C-shell script by typing: `source setup53236` (Please note that you will need to re-execute steps 4 and 5 each time you log in to work on this assignment.)
6. Execute the finite element program FENDAC by typing: `f-m <Enter>` The program will then prompt you for the name of the data file. In response, type: `e11 <Enter>` The finite element program (FENDAC) will execute in the background. Wait for the program to finish executing (This should take a maximum of about 60 seconds). After executing the program, you will have a file called `e11.results` in your directory in addition to a number of `TAPE**.e11` files which contain information for the post-processing software. The information of most importance to you will be found in `TAPE88.e11`, since the values of $\left(\varepsilon_{11}^{Y}, \varepsilon_{22}^{Y}, \varepsilon_{33}, \gamma_{23}^{Y}, \gamma_{31}^{Y}, \gamma_{12}^{Y}\right)$ followed by $\left(\sigma_{11}^{Y}, \sigma_{22}^{Y}, \sigma_{33}, \sigma_{23}^{Y}, \sigma_{31}^{Y}, \sigma_{12}^{Y}\right)$ are printed in this file.
7. You can create similar data files that perform strain-controlled unit cell analysis ($\varepsilon_{22}^{Y} = 0.01$ and $\varepsilon_{12}^{Y} = 0.01$) as well. For each different combination of void dimensions, you will need to perform three such tests.
8. To view the undeformed mesh and deformed mesh associated with each unit cell computation, you can step through the following procedure:
   a. Execute the mesh-viewing program by typing `rpostj name <Enter>` where `name` is the name of a data set such as `e11`
   b. The viewing post-processor will then ask you if you want to show: (1) stress/strain data; (2) volume fraction distributions; and (3) unscaled/scaled plot deformation of the model. In response: enter "1" Yes for (1); "0" No for (2); and "1" Yes for (3).
   c. Once `rpostj` is finished, it will have created a file *.plt that can be viewed interactively with TecPlot.
9. Once you've finished all of your computations and post-processing, you can get out of the C-shell invoked in Step 4 by typing: `Ctrl D`

One you are finished with this assignment, you might want to delete or compress many of the large files that FENDAC has created to avoid going over your diskspace quota. It is recommended that you save the FENDAC input data files.