The University of Iowa
College of Engineering
53:236 Optimization of Structural Systems
Fall Semester 2004

Assignment #6: Due: 12/01/2004

Upon completing Assignment #5, you have a program that given micro-structural design parameters (a,b,c), computes the set $\left(\mathbf{E}^Y; \partial\mathbf{E}^Y / \partial a; \partial\mathbf{E}^Y / \partial b; \partial\mathbf{E}^Y / \partial c\right)$. You should recall that each node of the structural finite element model will typically have three design variables (a,b,c). The variable c is related to the orientation of the microstructure θ as follows: $c = 2\theta / \pi$. Accordingly, all of the design variables are drawn from the same closed interval [0,1].

The objective of this assignment is to use this program in implementing the homogenization-based continuum structural topology optimization formulation of Bendsøe and Kikuchi.

Overview of Assignment

When this multi-part assignment is completed, you will have a functioning software package of three integrated pieces:

1. SLP is the optimization driver which will call BESTOP with designs $\mathbf{b}$, expecting to obtain function values and their design derivatives in return. **You will not need to make any changes to this program**.
2. BESTOP will receive the design vector $\mathbf{b}$ from the optimization driver, and it will then convert these into elastic moduli at integration points of individual finite elements in the analysis model. These elastic moduli will need to be written into a file called ``MATERIAL.data". BESTOP will then execute FENDAC which will read this file to obtain stiffness properties for each integration point. After execution is complete, BESTOP will read the resulting displacement field $\mathbf{u}(\mathbf{b})$ from the file "des.results". BESTOP will then compute the necessary function values (strain energy and global solid volume fraction) and their design derivatives. This information will be transferred/conveyed back to the optimization program SLP. You will need to make a few changes to BESTOP and these will be discussed below.
3. FENDAC will serve as the analysis program. When it receives elastic moduli from BESTOP it will then solve the linear elastic structural analysis problem and write the displacement solution $\mathbf{u}(\mathbf{b})$ in the ASCII results file "des.results" which will in turn be read by BESTOP. **You will not need to make any changes to FENDAC . In addition, FENDAC should be run in standard execution mode without doing any sensitivity analysis for evaluation of topology functionals.**

Details on BESTOP modifications:

BESTOP already does many of the tasks mentioned in the list above, so you will only need to make a few changes. In fact, all of your modifications to BESTOP will be contained within a single file (or subroutine) called plane.f

During each design iteration, BESTOP calls this routine a number of times to do different tasks. You need to be concerned with the following three:

ie = 1: During this stage, design BESTOP reconstructs the element connectivity data, and given the design variables for each element, it prints the elasticity tensor $\mathbf{E}^{Y}$ to the FENDAC input file "MATERIAL.data". You will need to provide the code for computing elasticity tensors and writing the results into "MATERIAL.data".

ie = 14: During this task, the displacement vector $\mathbf{u}$ and the adjoint displacement vector $\mathbf{u}^{a} = -\frac{1}{2}\mathbf{u}$ are passed to this subroutine. You will need to add the coding to compute both $F_E(\mathbf{b})$ and

$$\frac{dF_E(\mathbf{b})}{d\mathbf{b}} = \int_{\Omega_b} \boldsymbol{\varepsilon}^{a} \cdot \frac{\partial \boldsymbol{\sigma}}{\partial \mathbf{b}}\, d\Omega_b$$

For each finite element "i" you will compute the gradient and then assemble the result into the global design gradient vector which is a real array of dimensions "grad(3,numnp)"

ie = 15: During this task, the objective is to compute both:

$$\left\langle \phi_{\text{solid}} \right\rangle = \tfrac{1}{V} \int_{\Omega_b} \phi_{\text{solid}}\, \mathrm{d}\Omega_{\mathrm{b}} \text{ and } \frac{d\left\langle \phi_{\text{solid}} \right\rangle}{d\mathbf{b}}$$

Again, your sensitivity results will go into the same global design gradient array grad(3,numnp).

<u>More Details:</u>

1. It is recommended that you start out solving a very small problem involving only four elements. The starting FENDAC data file "des.data" and the starting BESTOP data file "bestop.data" for this problem are provided in the directory /usr/ui/class/examples/cee5330/53_236/hw6.

2. To make your own version of bestop you can copy the files "plane.f" and "makefile" from the directory /usr/ui/class/examples/cee5330/53_236/hw6. After modifying the file "plane.f" you only need to type "make" to compile it and create the new executable BESTOP.

3. BESTOP already does the following things which you will not need to concerned with:

- it reads the current design $\mathbf{b}$ from the file DESIGN created by SLP;
- it executes FENDAC and then reads $\mathbf{u}(\mathbf{b})$ from des.results;
- it writes the objective function to file "OBJECTIVE", the constraint function value to "CONSTRAIN" and their respective gradients to files "OBJGRAD" and "CONGRAD";

4. In your final report, you will want to provide documentation that you've thoroughly tested and debugged your implementation. In particular, you should have some examples in which you've tested your analytical design gradient expressions against converged finite difference gradients (c.f. Assignment #4). For this purpose, you will find a special version of IDESIGN in the /usr/ui/class/examples/cee5330/53_236/hw6 directory that calls BESTOP.

5. You should also apply your software to solve some compliance minimization problems. Discuss your results and the extent to which they do or do not make sense.