Homework #3: Sensitivity Analysis — ME 575

due 2/18/2015 before class via Learning Suite — 50 possible points

3.1 [5 pts] Read Chapter 4 (Sensitivity Analysis).

3.2 [30 pts] **Truss Sensitivities**

We will optimize a truss that is analyzed using finite elements. A truss element is the simplest type of finite element and only has an axial degree of freedom. If you took ME 372 you learned how to use truss elements in ANSYS and manually "optimized" a truss in one of your assignments.

The theory and derivation for truss elements is very simple, but for our purposes we will jump right to the result. Given a 2D element oriented arbitrarily in space (Fig. 1) we can relate the displacements at the nodes to the forces at the nodes through a stiffness relationship.

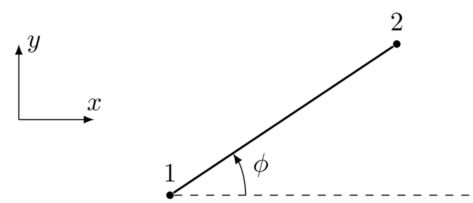

Figure 1: A truss element oriented at some angle ϕ, where ϕ is measured from a horizontal line emanating from the first node, oriented in the +x direction.

In matrix form the equation is $F = Kx$. In detail the equation is

$$\begin{bmatrix} X_1 \\ Y_1 \\ X_2 \\ Y_2 \end{bmatrix} = \frac{EA}{L} \begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix} \begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{bmatrix} \tag{1}$$

where the displacement vector is $x = [u_1, v_1, u_2, v_2]^T$. The meanings for the variables in the equation are described in Table 1.

Table 1: The variables used in the stiffness equation.

symbol	description
X_i	force in the x-direction at node i
Y_i	force in the y-direction at node i
E	modulus of elasticity of truss element material
A	area of truss element cross-section
L	length of truss element
c	$\cos\phi$
s	$\sin\phi$
u_i	displacement in the x-direction at node i
v_i	displacement in the y-direction at node i

The stress in the truss element can be computed from the equation $\sigma = Sx$ where σ is a scalar, x is the same vector as before, and the S matrix (really a row vector because stress is one-dimensional for truss elements) is

$$S = \frac{E}{L} \begin{bmatrix} -c & -s & c & s \end{bmatrix} \tag{2}$$

The global structure (an assembly of multiple finite elements) has the same equations: $F = Kx$ and $\sigma = Sx$, but now x contains displacements for all of the nodes in the structure $x = [x_1, x_2, \ldots, x_n]^T$. If we have n nodes and m elements then $F \in \mathcal{R}^{2n}$, $x \in \mathcal{R}^{2n}$, $K \in \mathcal{R}^{2n\times 2n}$, $S \in \mathcal{R}^{m\times 2n}$, and $\sigma \in \mathcal{R}^m$. The elemental stiffness and stress matrices are first computed and then put assembled into the global matrices. This is straightforward because the displacements and forces of the individual elements add linearly.

After we assemble the global matrices we must remove any degrees of freedom that are structurally rigid (already known to have zero displacement). Otherwise, the problem is ill-defined and the stiffness matrix will be ill-conditioned.

Given the geometry, materials, and external loading we can populate the stiffness matrix and force vector. We can then solve for the unknown displacements from

$$F = Kx \tag{3}$$

With the solved displacements we can compute the stress in each element using

$$\sigma = Sx \tag{4}$$

The specific problem we will solve is a 10-bar truss shown in Fig. 2. All horizontal and vertical segments are of the same length L, and meet at right angles. The two applied loads, P, are of equal magnitude. The parameters of the problem are specified in Table 2.

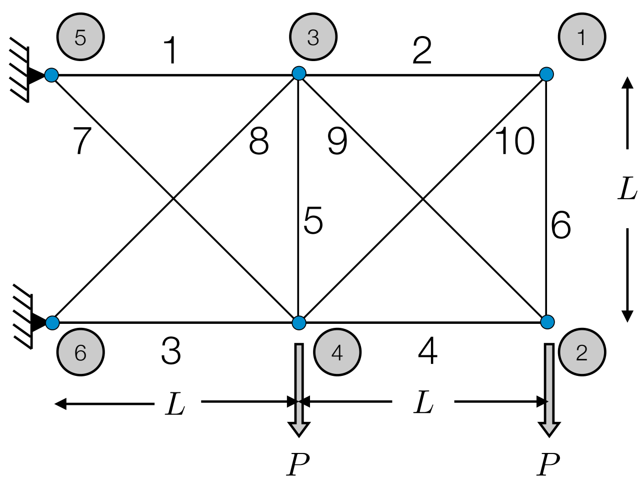

Figure 2: 10 bar truss geometry with node and element numberings.

Table 2: Parameters for the 10-bar truss problem.

parameter	value
modulus of elasticity (E)	1×10^7 psi
material density (ρ)	0.1 lb/in^3
applied load (P)	$100,000$ lb
length of square sides (L)	360 in

You will first need to create a subroutine that can compute the mass of the entire structure and the stress of each element (in an array) as a function of the cross-sectional areas of the members. Starter code that provides much of the implementation is available for you on Learning Suite (both Matlab and Python versions). To check your implementation, if every member has a cross section of 1 in^2, then the mass of the structure is 419.6 lb and the stress of element 1 is 195,365 psi and the stress of element 2 is 40,125 psi.

To optimize this structure we would need the following sensitivities:

- The sensitivities of mass with respect to changing the cross-sectional areas, dm/dA_i, for $i = 1, ..., m$.
- The sensitivities of stress with respect to changing the cross-sectional areas, $d\sigma_i/dA_j$, for $i = 1, ..., m$ and $j = 1, ..., m$.

Compute the sensitivities of the objective (mass) and the constraints (stress) using the following methods:

(a) A finite-difference formula of your choice.

(b) The complex-step derivative method.

(c) The analytic adjoint method.

Discuss your results and relative merits of the approaches.

Extra Credit:

Automatic differentiation is a bit tricker to setup with the current problem because of all the indexing shortcuts used in create submatrices (but it can be done). For extra credit apply an automatic differentiation tool to a simpler subroutine of your choice. Your subroutine should represent an engineering analysis and should have at least 10 lines of code. Briefly describe the analysis and compare the results from automatic differentiation with finite differencing.

3.3 [15 pts] **Truss Optimization**

With the provided gradients you can now optimize your truss. We will be discussing the theories and methods of constrained optimization in the next module, but for the purposes of this assignment you will just need to know how to use existing optimizers. If you are using Matlab then you will use fmincon. If you are using Python you have several options, but probably the easiest to get setup is SLSQP available in scipy.optimize (I would suggest using the wrapper "minimize" and select SLSQP as the algorithm, rather than directly using "fmin_slsqp"). I will provide some simple examples of using these, but you should consult the documentation for the solvers.

The objective is to minimize the mass, and the constraints are that every segment must not yield in compression or tension. The yield stress of all elements is 25×10^3 psi, except for member 9 which uses a stronger alloy with a yield stress of 75×10^3 psi. Mathematically the constraint is

$$|\sigma_i| < \sigma_{y_i} \quad \text{for } i = 1 \ldots 10 \tag{5}$$

Absolute values are not differentiable and so should be avoided in gradient-based optimization. We can easily remove the absolute value in a mathematically equivalent formulation by doubling the number of constraints:

$$\sigma_i < \sigma_{y_i} \quad \text{for } i = 1 \ldots 10 \tag{6}$$

$$\sigma_i > -\sigma_{y_i} \quad \text{for } i = 1 \ldots 10 \tag{7}$$

Each element should have a cross-sectional area of at least 0.1 in^2 for manufacturing reasons. This types of constraint is called a bound constraint. In solving this optimization problem you should think about using appropriate scaling.

Solve this optimization problem using both finite differencing and the analytic gradients from the adjoint method. For both methods you should show a convergence plot, report the optimal mass and corresponding cross-sectional areas, and compare the number of function calls required to converge. Discuss your findings.