N87-18865

SENSITIVITY AND OPTIMIZATION OF COMPOSITE STRUCTURES USING MSC/NASTRAN

Gopal K. Nagendra[*] and Claude Fleury[**]
* The MacNeal-Schewendler Corporation, Los Angeles, CA
**UCLA
Los Angeles, CA

ABSTRACT

Design sensitivity analysis for composites will soon be available in MSC/NASTRAN. The design variables for composites can be lamina thicknesses, orientation angles, material properties or a combination of all three. With the increasing use of composites in aerospace and automotive industries, this general capability can be used in its own right for carrying out sensitivity analysis of complicated real-life structures.

As part of a research effort, the sensitivity analysis has been coupled with a general-purpose optimizer. This preliminary version of the optimizer is capable of dealing with minimum weight structural design with a rather general design variable linking capability at the element level or system level. Only sizing type of design variables (i.e., lamina thicknesses) can be handled by the optimizer.

Test cases have been run and validated by comparison with independent finite element packages. The linking of design sensitivity capability for composites in MSC/NASTRAN with an optimizer would give designers a powerful automated tool to carry out practical optimization design of real-life complicated composite structures.

PRECEDING PAGE BLANK NOT FILMED

INTRODUCTION

The purpose of this paper is to present the considerations and the resultant approach used to implement design sensitivity capability for composites into MSC/NASTRAN. MSC/NASTRAN is a large-scale, general-purpose computer program which solves a wide variety of engineering problems by the finite-element method. In addition, as part of a research effort, MSC/NASTRAN has been coupled with a general-purpose optimizer CONLIN to optimize composite structures with sizing type variables.

The following sections will cover:

- The analysis of laminated composites in MSC/NASTRAN

- Theory for design sensitivity analysis for composites

- The program architecture and considerations that go into implementing such a capability into a large-scale general-purpose computer program

- Basic optimization concepts and a brief description of the optimizer CONLIN used for this study

- Numerical studies to validate the results

THE ANALYSIS OF LAMINATED COMPOSITE MATERIALS IN MSC/NASTRAN

Laminated composites may be conceptually viewed as a "stack" of laminae with different orientations of the principal material directions in the individual lamina. An exploded view of three cross-ply laminated plates is illustrated in Figure 1. The n-laminae (n = 2,3,4) of each of the three configurations are normal to the z-axis of the indicated coordinate system and the 1- and 2-axes appended to the individual lamina denote principal material axis directions. The directions of the principal material axes of each lamina alternate as implied by the use of the word "cross-ply" to describe the configuration. The xy-plane of the coordinate axes is defined in the geometric middle plane of the laminae.

An entire "stack" of laminae may be modeled with a single plate or shell element because the material properties of the "stack" are completely reflected in the matrices of elastic moduli for the element. These matrices are automatically calculated in MSC/NASTRAN from user-supplied definitions of the thickness, the material properties, and the relative orientation of these properties for the individual lamina. Once these matrices of elastic moduli are calculated, the analysis proceeds in a standard manner. This capability for the automatic representation of laminated composites is available in linear static analysis, real and complex eigenvalue analysis, buckling analysis, geometric nonlinear analysis, and a dynamic analysis [1].

In the analysis of isotropic materials, strength is independent of the orientation of the body under load and one may compare the largest computed principal stress with an allowable stress to establish the integrity of the structure. Laminated composites, on the other hand, are orthotropic materials and may exhibit unequal properties in tension and compression. Thus, the strengths of these orthotropic laminae are a function of body orientation relative to the imposed stresses.

As the evaluation of the matrices of material moduli for laminated composites provides sufficient information to determine the actual stress field sustained by the material, the determination of structural integrity will depend on the definition of an allowable stress field. The basic ingredient of this definition is the establishment of a set of allowable stresses or strengths in the principal material directions.

X_t = Allowable tensile stress in the principal x(or 1)-direction of the material

X_c = Allowable compressive stress in the principal x(or 1)-direction of the material

Y_t = Allowable tensile stress in the principal y(or 2)-direction of the material

Y_c = Allowable compressive stress in the principal y(or 2)-direction of the material

S = Allowable shear stress in the principal material system

Failure index of an element is a measure designed to test whether the state of stress in the worst-stressed lamina is within or outside the lamina's failure envelope.

In addition, the interlaminar shear stress will be checked against the allowable bonding stress (S_b) specified by the user.

The failure index for the laminate is the larger of the two values so obtained. Three failure criteria are available in MSC/NASTRAN. They are Hill, Hoffman and Tsai-wu. In this paper Hill's failure criterion will be used, i.e.,

$$\frac{\sigma_1^2}{X^2} + \frac{\sigma_2^2}{Y^2} - \frac{\sigma_1\sigma_2}{X^2} + \frac{\sigma_{12}^2}{S^2} = 1$$

$X = X_t$ if σ_1 is tensile
$\quad = X_c$ if σ_1 is compressive

$Y = Y_t$ if σ_2 is tensile
$\quad = Y_c$ if σ_2 is compressive

For the product term, $X = X_t$ if σ_1 and σ_2 are of the same sign; $X = X_c$

otherwise. Basically the equation represents a failure envelope in the stress space (figure 2).

If the state of stress in the orthotropic lamina $(\sigma_1, \sigma_2, \sigma_{12})$ is such that the stress point is within or on the envelope, the lamina is said to be "safe"; if the point is outside, the lamina is said to have "failed".

DESIGN SENSITIVITY CAPABILITY FOR COMPOSITES IN MSC/NASTRAN

Design sensitivity analysis for composites will soon be available in MSC/NASTRAN. The design variable for composites can be lamina thicknesses, orientation angles, material properties, or a combination of all three. With the increasing use of composites in aerospace and automotive industries, this general capability can be used in its own right for carrying out sensitivity analysis of complicated real-life structures.

Design sensitivity analysis estimates the effects of interrelated design variables such as element properties and materials on the structural response quantities, such as displacement, stress, natural frequency, buckling loads, and for composites lamina stresses and failure indices. Design sensitivity coefficients are defined as the gradients of the design constraints with respect to the design variables at the current design point. The method chosen for incorporation into MSC/NASTRAN is called the Pseudo load technique, based on a first variation (finite-difference scheme) of the systems equilibrium equations with respect to the design variables.

Let $\psi_i(b_j, u_g)$ be a set of design constraints which are functions of b_j design variables and displacements u_g. The design constraints are expressed as

$$\psi_i(b_j, u_g) < 0$$

The first variation in ψ_i is given as

$$\delta\psi_i = \underset{1 \times j}{[\frac{\partial\psi_i}{\partial b_j}]_{u\text{-fixed}}} \underset{j \times 1}{\{\delta b_j\}} + \underset{1 \times n}{[\frac{\partial\psi_i}{\partial u_g}]_{b\text{-fixed}}} \underset{n \times 1}{\{\delta u_g\}}$$

consider u_g as a function of b_j, then

$$\{\delta u_g\} = \underset{n \times j}{[\frac{\partial u_g}{\partial b_j}]} \underset{j \times 1}{\{\delta b_j\}}$$

150

Thus

$$\{\delta\psi_i\} = \left(\left[\frac{\partial\psi_i}{\partial b_j}\right] + \left[\frac{\partial\psi_i}{\partial u_g}\right]\left[\frac{\partial u_g}{\partial b_r}\right]\right)\{\delta b_j\}$$

or

$$\frac{\delta\psi_i}{\delta b_j} \simeq \frac{\Delta\psi_i}{\Delta b_j} = \left[\frac{\partial\psi_i}{\partial b_j}\right] + \left[\frac{\partial\psi_i}{\partial u_g}\right]\left[\frac{\partial u_g}{\partial b_j}\right]$$

The matrix $\frac{\partial u_g}{\partial b_j}$ can be evaluated by taking the first variation of the systems equilibrium equation

$$[K_g]\{u_g\} = \{P_g\}$$

or

$$[K_g]\{\Delta u_g\} + [\Delta k_g]\{u_g\} = \{\Delta P_g\}$$

or

$$\{\Delta u_g\} = [k_g]^{-1}(\{\Delta P_g\} - [\Delta k_g]\{u_g\})$$

or

$$[\Delta u_g] = [K_g]^{-1}(\{\Delta P_g(\Delta b_1)\}, \{\Delta P_g(\Delta b_2)\}, \ldots, \{\Delta P_g(\Delta b_j)\})$$

$$- [K_g]^{-1}([\Delta k_g(\Delta b_1)]\{u_g\}, [\Delta k_g(\Delta b_2)]\{u_g\}, \ldots, [\Delta k_g(\Delta b_j)]\{u_g\})$$

The elements of $\left[\frac{\partial\psi_i}{\partial u_g}\right]$ matrix for an element constraint, such as stress, force, or a failure index, can be expressed by the relationship

$$\{\psi_i\} = [S]_e\{u_g\}$$

or

$$\left[\frac{\partial\psi_i}{\partial u_g}\right] = [S]_{ig}$$

The design sensitivity coefficient matrices may thus be expressed as

$$[\Lambda_{ij}] \cong (\{\frac{\Delta\psi_i}{\Delta b_1}\}, \{\frac{\Delta\psi_i}{\Delta b_2}\}, \ldots, \{\frac{\Delta\psi_i}{\Delta b_j}\}) \, |_{u_{fixed}}$$

$$+ [S_{ig}](\{\frac{\Delta U_{g1}}{\Delta b_1}\}, \{\frac{\Delta U_{g2}}{\Delta b_2}\}, \ldots, \{\frac{\Delta U_{gj}}{\Delta b_j}\}) \, |_{b_{fixed}}$$

From this equation it is easy to see that the number of additional case control records (additional loading cases) required for design sensitivity analysis is equal to the number of design variables for each subcase (Pseudo-Load Technique).

A typical term of the coefficient matrix may thus be written as

$$\Lambda_{ij} = (\frac{S^{B+\Delta B} u^B}{\Delta B} - \frac{S^B u^B}{\Delta B}) + (\frac{S^B u^{B + \Delta B}}{\Delta B} - \frac{S^B u^B}{\Delta B})$$

where B represents the base line or original state and B + ΔB represents the perturbed state. The first expression in parentheses on the right-hand side is thus the change in response quantity due to a change in design variable for the original solution vector. The second term represents the change in response quantity due to a change in displacement for the unperturbed design variable. For displacement constraints, the first term in parentheses on the right-hand side is identically zero.

The design constraints for composites can be lamina stresses or failure indices, displacements, frequency, buckling loads, or forces. The design variables can be lamina thicknesses, orientation angles or material properties.

PROGRAM ARCHITECTURE

In order to understand the reasons behind how a development is introduced into a large finite-element program, a knowledge of the program architecture and technical purpose is necessary. A brief description of MSC/NASTRAN is presented as background.

The cornerstone of MSC/NASTRAN's architecture is its Executive System, whose essential functions are to establish and control and sequence of calculations, to allocate files, and to maintain a restart capability. Engineering calculations are performed by approximately 200 functional modules which communicate only with the Executive System and not with each other. Flexibility is maintained by a macro-instruction language called DMAP, which

is under user control, but which also serves to establish preformatted calculation sequences for the major types of analysis, including linear analysis, buckling, vibration mode analysis, and design sensitivity.

The calculation of finite element data is concentrated exclusively in a few modules. The element matrices for stiffness, structural damping, and differential stiffness for elements of the structural model are generated in the Element Matrix Generator (EMG) module. These element matrices are subsequently assembled to form the elastic stiffness matrix, the structural damping matrix, the mass matrix, or the differential stiffness matrix.

The element contribution to the load vector is generated in the SSG1 module and the element stress and force are generated in the SDR2 module. In all these modules, the finite element descriptions are defined in the Element Summary Table, the EST table. The EST table contains the element connection, material property and sectional property information.

Taking advantage of the table-driven concept used by the element modules, much of the element dependent development could be avoided in implementing design sensitivity if a procedure could be developed which would involve only building EST tables that would cause existing modules to form the necessary element data.

How a given capability is introduced into a commercial general-purpose finite-element program is as important an issue to the user as its theoretical sophistication. If the user views a capability as hard to use, as having an insufficient capacity to solve his problem, or taking an inordinate time to comprehend its output, the product is of little practical use. In addition, the program developer, while heeding the user's needs, has to keep sight of the program as a whole when adding new capabilities. This involves interfacing well with existing capabilities, maintaining program reliability and generality, and producing software that makes effective use of computer resources.

The user interface is a major consideration in the design of a new capability. The following issues were considered in DSA:

1. DSA input should be straightforward, but allow flexibility to model complex structural design concepts

2. DSA output should be concise and easily understood

3. Avoid arbitrary program limits which restrict the allowable element types, constraint quantities, and problem size

4. Provide an interface for external optimization postprocessors

A brief discussion of the processes involved in a typical DSA STATIC analysis in MSC/NASTRAN will help bring into perspective the work involved in the various parts of the DSA solution.

DSA in a STATICS analysis is based on solving for $\{\Delta u_g\}$ in the first-order variation of the nodal equilibrium equation:

$$[K^O_{gg}]\{\Delta u_g\} = \{\Delta P_g\} - [\Delta K_{gg}]\{u^O_g\}$$

The DSA problem in the paper is considered to be the additional task required after the solution of primary analysis. By restarting from the primary STATIC analysis, the solution of the DSA system equation only involves the calculation of the right-hand side and the backward pass operation in the solution of Δu.

The work involved in solving the system equations (backward pass operation) is a function of the product of the number of design variables and loading conditions. The following DSA tasks are required in addition to solving the system equations:

1. DSA Data Organization

2. DSA Data Assembly

3. DSA Data Recovery

These tasks are functions of the triple product of the number of design variables, design constraints and loading conditions. For large DSA problems, the data organization, assembly and recovery tasks are the dominant users of computer resources.

Another major consideration was to support all structural finite-element types in MSC/NASTRAN. Since a large number of the elements developed are semiempirical, the determination of consistent element derivative formulations cannot be practically accomplished. Therefore, a method was developed to calculate element derivatives by a differencing scheme about the current design point. This method involved the calculation of the element matrix at the design point plus or minus the user-specified design variable increment. These element data are differenced with the data at the design point to determine the corresponding element derivatives. For example, the following shows the change in element stiffness due to a change in the design variable.

$$[\Delta K_{gg}] = [K_{gg + \Delta B}] - [K^O_{gg}]$$

Another benefit of differencing about the design point is that it avoids most potential numerical problems. This is because the evaluation of the perturbated element data is computed near a design point which has already been determined to be numerically acceptable in the primary analysis.

To get an idea of the magnitude of the task involved, about 15 existing subroutines comprising approximately 6000 lines of Fortran had to be modified, in addition to coding approximately 10 new subroutines comprising 1000 lines of Fortran. There are approximately 15 tables created for data organization and manipulation.

An initial analysis is carried out to identify critical constrains and a data base created. In the succeeding run, information about constraints, design variables, and maximum and minimum side constraints is supplied. A special DMAP package was created which exploits the data base technology.

The user can control the number of iterations performed. He can restart from the previous step. This is especially convenient, as he can scan the output and intervene manually to either add or delete constraints or modify design variables. Table 1 gives a schematic diagram of the program flow.

OPTIMIZATION CONCEPTS AND CONVEX LINEARIZATION

In order to validate the new design sensitivity capability for composite structures, it was decided to introduce a numerical optimization module in a special version of MSC/NASTRAN. It has then become possible to solve some well-documented structural optimization problems, and to compare the results with those produced by other finite-element systems having similar sensitivity and optimization capabilities. In our opinion this pilot implementation represents the most complete and reliable way of verifying that the sensitivity analysis results are correct and accurate enough for a meaningful exploitation. It should however be mentioned that only sizing types of design variables (i.e., lamina thicknesses) are permitted in our optimization module. This is because no proper formulation is currently available to deal with optimization problems involving other types of design variables (e.g., orientation angles and material properties).

Structural optimization methods using finite-element models have now reached a high level of reliability and efficiency. These methods can currently address practical problems involving various types of design variables (e.g., component transverse sizes, shape variables) and design constraints (e.g., geometry requirements, maximum allowable stresses, bounds on deflections, or frequencies). In addition the types of finite-element models tractable by these methods have recently been largely extended so that virtually all finite-element models that can be analyzed can now be addressed by optimization techniques (e.g., bar, beam, membrane, plate, and shell).

A numerical optimization problem is characterized by a given objective function $f(x)$, which is to be minimized by determining the magnitudes of design variables x, such that certain constraints on the x_i's are achieved. This leads to a mathematical programming problem of the "primal" form:

minimize $\qquad f(x)$

such that $\qquad \underline{h}_j(x) \geqslant 0 \qquad j=1,2,\ldots,m$

$\qquad \overline{x}_i \geqslant x_i \geqslant \underline{x}_i \qquad i=1,2,\ldots,n$

where m is the number of behavior constraints and n, the number of design variables.

Such a problem can be solved iteratively by using numerical optimization techniques. Each iteration begins with a complete analysis of the system behavior in order to evaluate the objective function and constraint values along with their sensitivities to changes in the design variables (i.e., first derivatives). A design iteration is concluded by employing the results of these behavioral and sensitivity analyses in a primal minimization algorithm which searches the n-dimensional design space for a new primal point that decreases f(x) while remaining feasible (i.e., satisfying the constraints $h_j(x)$). Many such iterations are usually required before achieving the optimum design. Until recently, because of the high-computation cost of each iteration (full FEM analysis), structural optimization techniques based on primal algorithms have been only conceivable on power mainframe computers.

An alternative to this primal formulation is the so-called "dual" approach [2], in which the constrained primal minimization problem is replaced by maximizing a quasi-unconstrained dual function depending only on the Lagrangian multipliers associated with the behavior constraints. These multipliers are the dual variables subject to simple non-negativity constraints. The efficiency of this dual formulation is due to the fact that maximization is performed in the dual space whose dimensionality is relatively low and depends on the active constraint at each design iteration. The dual approach is especially powerful when used in conjunction with approximation concepts [3]. In particular, the convex linearization scheme (CONLIN) [4], recently introduced to solve general structural optimization problems, exhibits very good performance, even when dealing with the inherently difficult problems involving changes in geometry.

In CONLIN each function defining the optimum design problem is linearized with respect to appropriate intermediate variables (called "mixed" variables) yielding a convex, separable problem approximation. The initial problem is therefore transformed into a sequence of explicit subproblems having a simple algebraic structure. The convex linearization scheme exhibits remarkable properties that makes it attractive to replace the original primal subproblem by its dual [2]. CONLIN can be viewed as a generalization of well-established approaches to pure sizing structural optimization problems, namely "approximation concepts" and "optimality criteria" techniques [5], and as such it is capable of addressing a broader class of problems with considerable facility of use.

Because of its many attractive features the CONLIN algorithm has been selected to implement optimization capabilities in our pilot program. At each successive iteration point, the CONLIN method only requires evaluation of the objective and constraint functions and their first derivatives with respect to the design variables. This information is provided by the FEM analysis and sensitivity analysis results. The CONLIN optimizer will then select by itself an appropriate approximation scheme on the basis of the sign of the derivatives. CONLIN benefits from many interesting features.

- The CONLIN approach is very general, requiring only values and derivatives of the functions describing the optimization problem to be solved; it permits therefore straight interfacing to the FEM software;

- Because it is based on conservative approximation concepts, CONLIN does not demand a high level of accuracy for the sensitivity analysis results, which can therefore be obtained by finite differencing;

- CONLIN usually generates a nearly optimal design with less than 10 FEM analyses;

- CONLIN has an inherent tendency to produce a sequence of steadily improving feasible designs;

- The CONLIN method is simple enough to lead to a relatively small computer code, and well organized to avoid high core requirements.

These features have considerably facilitated the implementation of reliable and efficient optimization capabilities in our special version of MSC/NASTRAN.

NUMERICAL EXAMPLES

Two example problems were chosen to validate the capability and to highlight some of the salient features.

EXAMPLE 1 RECTANGULAR PLATE WITH A CIRCULAR HOLE.

A rectangular plate with a circular hole is subjected to a specified displacement along the x-direction. The quarter model of the plate is shown in Figure 3. The plate is modelled using QUADRILATERAL elements. Each element consists of 4 laminae stacked at 0°, 45°, 90° and -45°, respectively. The region near the hole is divided into 13 regions. The 0° lamina for each of the 13 regions is treated as a single design variable. The laminae at 45° and -45° are linked and are treated as a single design variable for each of the 13 regions. Similarly the 90° lamina is treated as single design variable for each of the 13 regions. Thus there are a total of 39 design variables for this problem. The model consists of 288 QUADRILATERAL elements and 317 grids.

The design constraints are the failure indices using the Hill criterion selected for different lamina in specified elements. The model was optimized for these selected constraints. The results are shown in Table 2. The results were examined after iteration 5 to examine if the failure index exceeded 1 for any of the elements which were not specified as constraints originally. The violated elements were input as constraints and the optimization loop started from this point onward. The algorithm converged in 9 loops. As can be seen, the user can intervene at specific points in the algorithm and monitor the progress. This capability is particularly important and convenient for realistic design of structures.

EXAMPLE 2

The second demonstration problem is a delta wing structure with graphite/ epoxy skins and titanium webs subjected to pressure loading and temperature loading. The wing is shown in Figure 4. The problem has been previously studied for frequency constraint in Reference 3. The structure is symmetric

with respect to its middle surface which corresponds to the x-y plane in Figure 4. The skins are assumed to be made up of 0°, ±45° and 90° high strength graphite/epoxy laminates. It is understood that orientation angles are given with respect to the x reference co-ordinate in Figure 4, that is, material oriented at 0° has fibers running spanwise while material at 90° has fibers running chordwise. The skins are represented by QUADRILATERAL and TRIANGULAR membrane elements and the webs are represented by shear panels. According to the linking scheme depicted in Figure 5, it can be seen that the total number of independent design variables is equal to 60 made up as follows: 16 for 0° material, 16 for ±45° material, 16 for 90° material and 12 for the web material. The model contains 56 QUADRILATERAL elements, 12 TRIANGULAR elements and 142 shear panels. The total number of nodes is 132.

The design constraint was the maximum deflection at the tip of the wing equal to 10.0 in. The results are shown in Figure 6 for the objective function and the tip deflection for the number of iterations.

After the Delta-wing was optimized for tip deflection of 10.00 in., parametric studies were carried out to study the effect of ΔB on the response quantity. The fundamental frequency was chosen as the constraint and ply-angle chosen as the design variable. All the 0° laminae in the wing were linked together, as were the 45°/-45° and 90° laminae. The value of ΔB was vaied from 10.0 to 10^{-7} and the results are shown in Table 3. As can be seen from Table 3, the sensitivity coefficients gradually converge till ΔB equal to 10^{-5} and then begin to diverge. Thus for ΔB less than 10^{-5}, round off errors become significant enough to degrade the solution. The robustness of the finite difference approach is, however, evident since even for ΔB as large as 0.1, the % error is only of the order of magnitude of 1%.

It was also decided to investigate the effect of linked design variables on fundamental frequency as a constraint. The results are shown in Table 4, where ply-thicknesses, orientation angles and Young's modulus along the principal direction were chosen as the design variables. The sensitivity analysis, by itself has little value unless used in an optimization context.

Table 5 gives the CPU times on the VAX-11/780 machine for the two example problems. As can be seen, the optimizer itself takes very little time. Since normally, 5 to 10 iterations are required for optimization, sensitivity analysis and the reanalysis after updating the design variables constitute the expensive portions of the design process. Efforts to enhance efficiency for sensitivity and reanalysis would go a long way toward making the design of realistic structures a viable proposition.

CONCLUSIONS

The design sensitivity capability for composites to be available in the next release of MSC/NASTRAN was designed for generality, whereby the design variables can be laminae thicknesses, orientation angles, material properties or a combination of all three. It is envisaged that this capability would constitute a powerful first step toward optimizing composite structures.

158

Furthermore as part of a research effort MSC/NASTRAN was linked to a general-purpose optimizer CONLIN for fully automated structural design synthesis. It has been demonstrated that the coupling of a large-scale finite element package like MSC/NASTRAN with a powerful optimizer will give designers a powerful tool to carry out practical optimization of real-life complicated structures. It should however be mentioned that only sizing type of design variables (i.e. lamina thicknesses) are permitted in our optimization module. This is because no proper formulation is currently available to deal with optimization problems involving other types of design variables (e.g., orientation angles and material properties).

A unique feature of the coupling is the capability for the user to intervene at any stage of the redesign process and to modify design constraints or design variables and to carry on from the previous stage. Man-machine interaction is an essential ingredient for realistic optimization of structural problems.

REFERENCES:

[1] G. K. Nagendra, "The Analysis of Laminated Composite Materials in MSC/NASTRAN", Chapter 6 of Computer-Aided Engineering of Composite Materials and Structures (J. Backlund), Elsevier, 1987.

[2] C. Fleury, "Structural Weight Optimization by Dual Methods of Convex Programming", International Journal for Numerical Methods in Engineering, Vol. 14, 1979, pp. 1761-1783.

[3] C. Fleury and L. A. Schmit, "Dual Methods and Approximation Concepts in Structural Synthesis", NASA Contractor Report, NASA-CR-3226, 1980, 196 pages.

[4] C. Fleury and V. Braibant, "Structural Optimization - A New Dual Method Using Mixed Variables", Int. J. Num. Meth. Eng., Vol. 23, No: 3 1986, pp. 409-428.

[5] C. Fleury, "Reconciliation of Mathematical Programming and Optimality Criteria Approaches to Structural Optimization", Chapter 10 of Foundations of Structural Optimization: a Unified Approach (A. J. Morris, ed.), John Wiley and Sons, 1982, pp. 363-404.
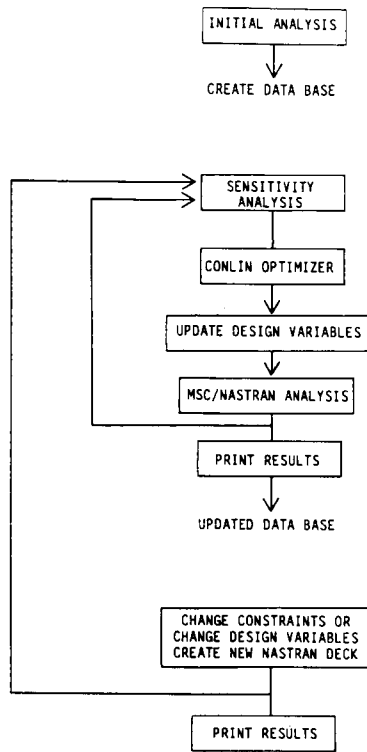
TABLE 1. FLOW CHART FOR OPTIMIZATION PROCEDURE

```
                        ┌─────────────────┐
                        │ INITIAL ANALYSIS │
                        └─────────────────┘
                                 │
                                 ▼
                          CREATE DATA BASE


    ┌──────────────────────►┌─────────────────┐
    │   ┌──────────────────►│   SENSITIVITY   │
    │   │                   │    ANALYSIS     │
    │   │                   └─────────────────┘
    │   │                            │
    │   │                            ▼
    │   │                   ┌─────────────────┐
    │   │                   │ CONLIN OPTIMIZER │
    │   │                   └─────────────────┘
    │   │                            │
    │   │                            ▼
    │   │                   ┌─────────────────────┐
    │   │                   │ UPDATE DESIGN VARIABLES │
    │   │                   └─────────────────────┘
    │   │                            │
    │   │                            ▼
    │   │                   ┌─────────────────────┐
    │   └───────────────────│ MSC/NASTRAN ANALYSIS │
    │                       └─────────────────────┘
    │                                │
    │                                ▼
    │                       ┌─────────────────┐
    │                       │  PRINT RESULTS  │
    │                       └─────────────────┘
    │                                │
    │                                ▼
    │                         UPDATED DATA BASE
    │
    │                       ┌──────────────────────────┐
    │                       │ CHANGE CONSTRAINTS OR     │
    │                       │ CHANGE DESIGN VARIABLES   │
    │                       │ CREATE NEW NASTRAN DECK   │
    │                       └──────────────────────────┘
    │                                │
    └────────────────────────────────┘
                                     ▼
                            ┌─────────────────┐
                            │  PRINT RESULTS  │
                            └─────────────────┘
```

TABLE 2.   RESULTS OF OPTIMIZATION

ANALYSIS NUMBER	WEIGHT	σ_1 1(00)	σ_2 1(45)	σ_3 2(00)	σ_4 (2-45)	σ_5 3(00)
1	.3575	1.1632	1.1421	-	-	-
2	.3562	.9446	.9076	-	-	-
3	.3545	.9886	.9238	-	-	-
4	.3541	.9948	.9160	-	-	-
5	.3540	.9982	.9164	-	-	-
6*	.3539	.9990	.9983	1.0999	1.1053	1.1634
7	.3554	.9388	.9469	.9799	.9921	.9871
8	.3552	.9552	.9651	.9855	.9994	.9805
9	.3552	.9585	.9690	.9854	.9999	.9769
10	.3552	.9594	.9700	.9853	1.0000	.9757

---

*User Intervention

TABLE 3. VARIATION OF SENSITIVITY COEFFICIENTS WITH RESPECT TO ΔB

ΔB	Λ
10.0	-.072539
1.0	-.082207
.1	-.082766
.01	-.082803
.001	-.082819
.0001	-.082836
.00001	-.082940
.000001	-.079559*
.0000001	-.098504

*Degrades

TABLE 4. SENSITIVITY COEFFICIENTS FOR FUNDAMENTAL FREQUENCY

DESIGN VARIABLE	SENSITIVITY COEFFICIENTS	
Ply-Thickness	0°	0.38200
	45°/-45°	-0.89380
	90°	-0.01774
Ply-Angles	0°	-0.08280
	45°/-45°	-0.10490
	90°	-0.00633
Material Properties	0°	1.48870
	45°/-45°	0.22640
	90°	0.00444

TABLE 5. ESTIMATE OF CPU TIME ON VAX 11-780

	Example 1	Example 2
	Rectangle with Cutout 650 DOF 39 Design Variables	Delta Wing 400 DOF 60 Design Variables
Initial Analysis	430 Secs.	300 Secs.
Sensitivity Analysis	257 Secs. (39 Constraints) 190 Secs. (10 Constraints)	180 Secs. (6 Constraints)
Optimization	3 Secs.	3 Secs.
Analysis	135 Secs.	115 Secs.
Per Iteration	395 Secs. (39 Constraints) 330 Secs. (10 Constraints)	300 Secs. (6 Constraints)
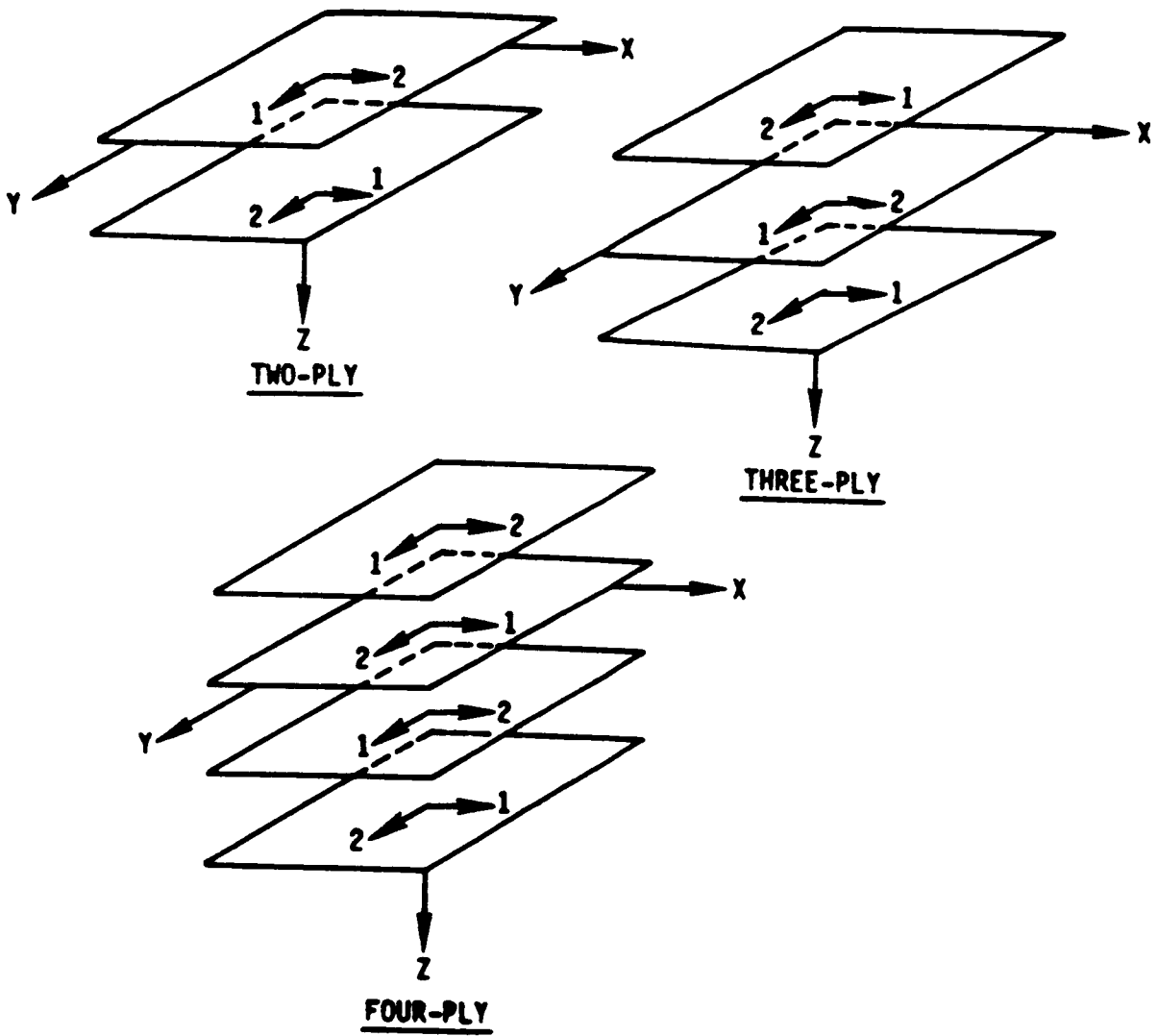
Figure 1. Exploded view of three cross-ply laminated plane structures.
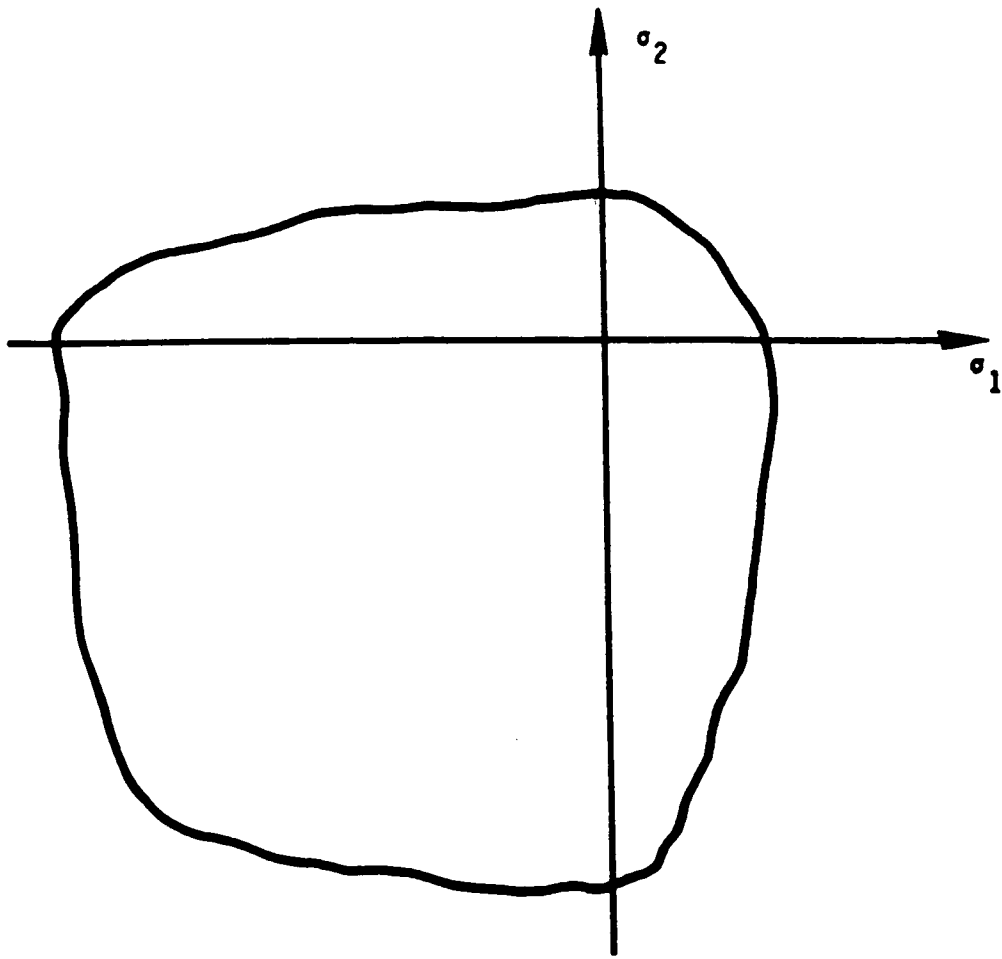
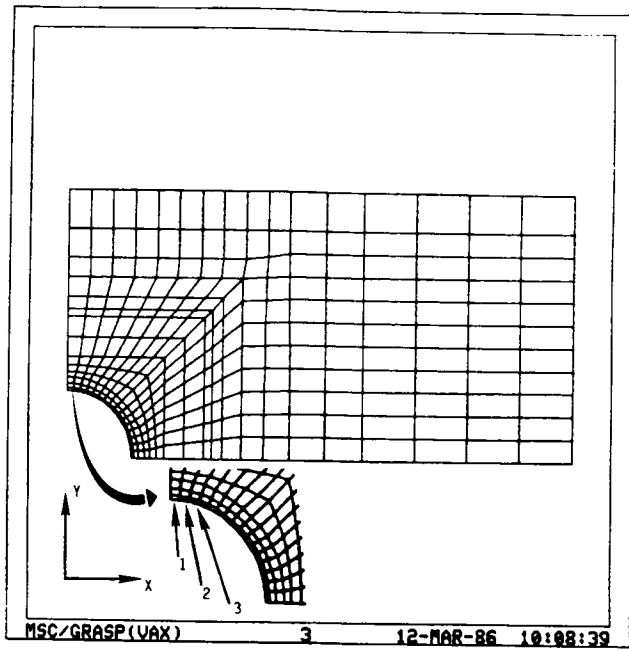Figure 2. Typical failure envelope for a material such as concrete.

Figure 3. Finite-element model of one-quarter plate.

c = 2438 cm (960 in.)
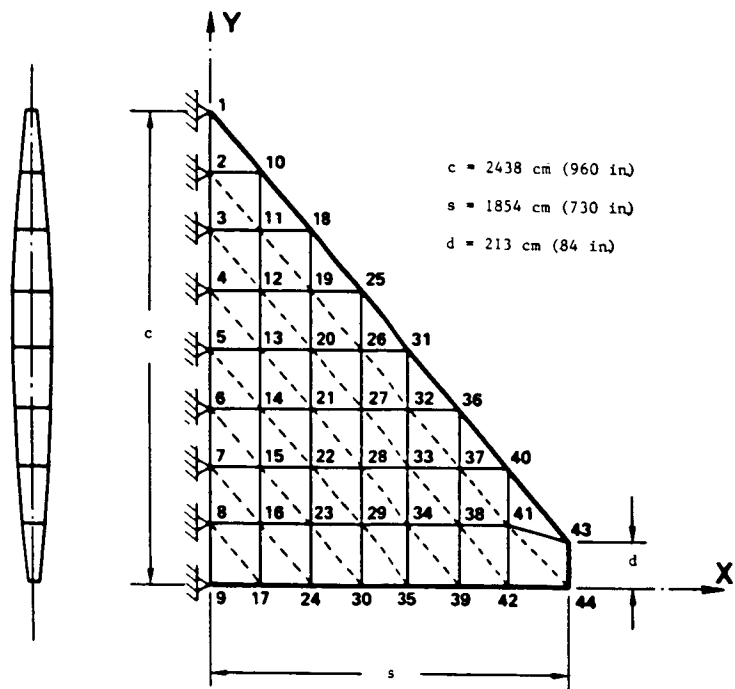
s = 1854 cm (730 in.)

d = 213 cm (84 in.)

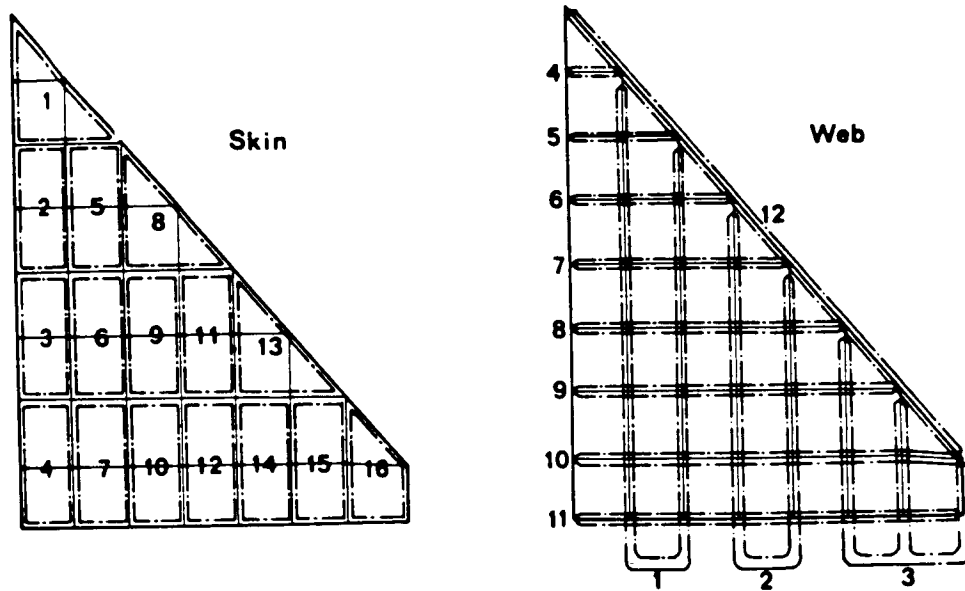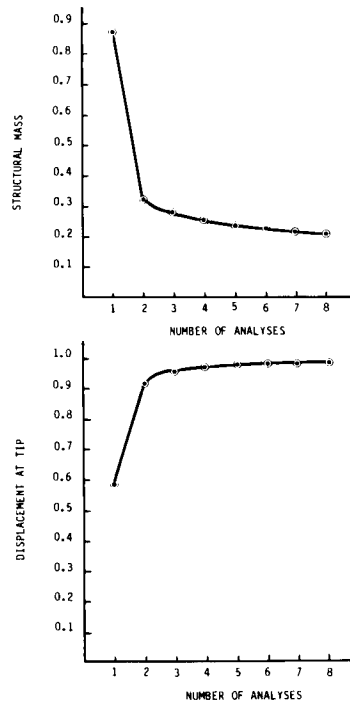Figure 4. Delta wing analysis model (problem 2).

Figure 5. Delta wing design model (problem 2).

Figure 6. Convergence of delta wing design.