"Sensitivity and Optimization of Composite Structures using MSC/NASTRAN"

Gopal K. Nagendra* and Claude Fleury**

Introduction

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.

*Senior Engineer, The MacNeal-Schwendler Corporation, Los Angeles, CA
**Associate Professor, Dept. of Mechanical, Aerospace and Nuclear Engineering, UCLA, Los Angeles, CA
Consultant to MSC

Brief Overview of Theory

The theory of Design Sensitivity and composites is covered in detail in References 1 and 2. The brief overview of theory presented here is for the sake of completeness and to serve as a backdrop for the marriage of sensitivity and composites capability.

Composites

The user input PCOMP and MAT8 cards are converted to equivalent PSHELL and MAT2 cards in the Preface. From this point onwards, MSC/NASTRAN treats each element as a homogeneous shell with these generated equivalent properties for stiffness matrix, mass matrix, damping matrix, load vector generation. After the system equation is solved and displacements calculated, the individual lamina properties are recovered in stress-data-recovery module for calculation of lamina stresses and failure indices. PCOMP and MAT8 cards are coverted to equivalent PSHELL and MAT2 cards as follows:

$$T\ (\text{thickness}) = T_1 + T_2 + \ldots + T_{NPLY}$$

The middle surface of the element and moment of inertia of the cross-section are

$$\bar{z} = \frac{T_1 \times \frac{T_1}{2} + T_2(T_1 + \frac{T_2}{2}) + \ldots + T_{NPLY}(T_1 + T_2 + \ldots + T_{NPLY-1} + \frac{T_{NPLY}}{2})}{T}$$

$$I = \frac{T_1^3}{12} + T_1\left(\bar{Z} - \frac{T_1}{2}\right)^2 + \frac{T_2^3}{12} + T_2\{\bar{Z} - (T_1 + \frac{T}{2} 2)\}^2 + \ldots$$

$$+ \frac{T_{NPLY}^3}{12} + T_{NPLY}\{\bar{Z} - (T_1 + T_2 + \ldots + T_{NPLY-1} + \frac{T_{NPLY}}{2})\}^2$$

The $[G_1]$, $[G_2]$ and $[G_4]$ matrices for the equivalent MAT2 cards are calculated as follows

$$[G_1] = \frac{1}{T} \times \{[G_e]_1\, T_1 + [G_e]_2\, T_2 + \ldots + [G_e]_{NPLY}\, T_{NPLY}\}$$

$$[G_2] = \frac{1}{3I} \times \{[G_e]_1(Z_1^3 - Z_0^3) + [G_e]_2(Z_2^3 - Z_1^3) + \ldots + [G_e]_{NPLY}(Z_{NPLY}^3 - Z_{NPLY-1}^3)\}$$

$$[G_4] = \frac{1}{2T^2} \times \{[G_e]_1(Z_1^2 - Z_0^2) + [G_e]_2(Z_2^2 - Z_1^2) + \ldots + [G_e]_{NPLY}(Z_{NPLY}^2 - Z_{NPLY-1}^2)\}$$

The treatment of transverse shear stiffness and calculation of $[G_3]$ is more involved and is covered in References 3 and 4.

Three types of failure theories have been implemented in MSC/NASTRAN for the calculation of failure indices in the lamina. The lamina is assumed safe if the value of the failure index is less than 1 and to have failed if the failure index value exceeds 1. The theories are

1. Hill's Theory

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

2. Hoffman's Theory

$$F.I. = \left(\frac{1}{X_T} - \frac{1}{X_C}\right)\sigma_1 + \left(\frac{1}{Y_T} - \frac{1}{Y_C}\right)\sigma_2 + \frac{\sigma_1^2}{X_T X_C} + \frac{\sigma_2^2}{Y_T Y_C} + \frac{\sigma_1 \sigma_2}{Y_T Y_C} + \frac{\sigma_{12}^2}{S^2}$$

3. Tsai-Wu (Tensor Polynomial) Theory

$$F.I. = \left(\frac{1}{X_T} - \frac{1}{X_C}\right)\sigma_1 + \left(\frac{1}{Y_T} - \frac{1}{Y_C}\right)\sigma_2 + \frac{\sigma_1^2}{X_T X_C} + \frac{\sigma_2^2}{Y_T Y_C}$$

$$+ 2F_{12}\,\sigma_1\sigma_2 + \frac{\sigma_{12}^{\,2}}{S^2}$$

Where σ_1, σ_2 and σ_{12} are the lamina direct and shear stresses along the principal directions and X_T, X_C, Y_T, Y_C are the allowable tension and compression stresses along principal directions and S the allowable shear stress.

Design Sensitivity

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 varibles 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

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

Consider u_g as a function of b_j, then

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

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

or

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

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

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

or

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

The design sensitivity coefficient matrices may thus be expressed as

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

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

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

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.

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 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).

Any idealized structural system can be described by a finite set of quantities. In particular a finite element model is characterized by the node coordinates of the mesh, the types of elements, their thicknesses and material properties, the applied loads and boundary conditions, etc... Some of these quantities are fixed in advance and they will not be changed by the redesign process (prescribed parameters). The others are the design variables; they will be modified during each redesign process in order to gradually optimize the structural system. A function of the design variables must be defined, whose value permits selecting different feasible designs; this is the objective function (e.g. weight of an aerospace structure, deflection at the tip of a beam). A design is said to be feasible if it satisfies all the requirements that are imposed to the structural system when performing its task. Usually requiring that a design be feasible amounts to assigning upper or lower limits to quantities describing the structural response. Most often these behavior constraints are placed on stresses, displacements, frequencies, etc... Additional design constraints may be introduced for fabrication or analysis validity considerations (e.g. minimum thickess).

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, 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:

$$\text{minimize} \qquad f(x)$$

$$\text{such that} \quad h_j(x) > 0 \qquad j=1,2,\ldots,m$$

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

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 powerful main frame computers.

An alternative to this primal formulation is the so called "dual" approach [5], 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 constraints at each design iteration. The dual approach is especially powerful when used in conjunction with approximation concepts [6]. In particular, the convex linearization scheme (CONLIN) [7], 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 [5]. CONLIN can be viewed as a generalization of well established approaches to pure sizing structural optimization problems, namely "approximation concepts" and "optimality criteria" techniques [8], 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. These informations are 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 within 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, well organized to avoid high core requirement.

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

User Interface

The DSCONS and DVSET bulk data cards will contain information to specify the design contraints and element property parameters for composites. The remaining cards will require no modification. Both stresses and forces must be requested in the case control deck for elements for which lamina stresses or failure theories are design constraints. MSC/NASTRAN will not generate lamina stresses or failure theories, unless both forces and stresses are requested for the pertinent elements in case control. If PARAM, NØCOMPS is set to -1, stress output for individual plies will be suppressed. In addition, fields 5 and 6 on the PCØMP card - S_b (allowable shear stress of the bonding material or allowable interlaminar shear stress) and failure theory (Hill, Hoffman or Tsai-wu) must be specified if failure index information is desired or when failure indices are design constraints.

The DSCONS bulk data card specifies output quantities such as element forces and stresses, displacements, natural frequencies and buckling load factors as design constraints. For composites, two additional design constraints - the lamina stress and failure index have been added. The modified DSCONS card is shown in Figure 1. Specification of CSTR (Composite Lamina Stress) or CFOR (Composite Failure index) will be specified in field 4 of the DSCONS card. The limit value for failure index is 1.0. For specification of component in field 6 of DSCONS card, Tables 1 and 2 give lamina stress and failure index item codes. For example, if Normal-1 stress in lamina 5 of the element is a design constraint, the component in field 6 of DSCONS card would be equal to 11 *(5-1) + 3 = 47, with CSTR being specified in field 4 of the DSCONS card. Similarly if failure index for direct stresses in lamina 4 of the element is a design constraint, the component in field 6 of the DSCONS card would be equal to 8 *(4-1) + 5 = 29, with CFOR being specified in field 4 of the DSCONS card. In general for the n^{th} lamina, the components for stresses and failure indices respectively are given by

$$COMP_{stress} = 11 *(N-1) + \text{ITEM number}$$
$$COMP_{F.I.} = 8 *(N-1) + \text{ITEM number}$$

For composites, TYPE in field 3 of the DVSET card shown in Figure 2 will be PCOMP. The element properties may be thickness, orientation angle or material ID for any lamina in the element. The design variable is 'FIELD' specified on the 4th field of the DVSET card. Whenever a material card is used as a design variable, the design sensitivity bulk data deck must contain all of the original material cards from the base line run plus all of the varied material cards. In addition, for composites, if the material ID specified on a PCOMP card is used as a design variable, the design sensitivity bulk data deck must contain all of the original PCOMP cards from the base line run in addition to all of the original material cards from the base line run plus all of the varied material cards.

A footnote is in order here. It is quite possible that a user may wish to use the PCOMP as a design variable (say the thickness in a particular lamina), and may wish to use the element stress or force (as opposed to lamina stress or failure index) as a design constraint. He can do so by specifying STRESS or FORCE as a design constraint on the DSCONS card and using PCOMP in field 3 of the DVSET card.

For optimization in addition to specifying the design constraints and design variables, it is necessary to supply minimum and maximum side constraints. The normalized values of the side constraints can be input conveniently by means of the DTI cards - wherein the first record contains the normalized values of the minimum sizes and record 2 contains the normalized values of the maximum sizes. The calculation of structural mass, derivative of the objective function (structural mass) with respect to the design variables, the constraints and the sensitivity coefficients are calculated internally in MSC/NASTRAN. All the arrays are normalized with respect to the design variables.

An initial analysis is carried out to identify critical constraints and a data base created. In the succeeding run, information about constraints, design variables, maximum and minimum side constraints is supplied. 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 3 gives a schematic diagram of the program flow.

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 4. The plate is modelled using QUAD4 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 laminas 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 QUAD4 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 Figures 8 and 9 and Table 4. 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 onwards. The algorithm converged in 9 loops.

Figure 8 is a plot of the objective function (structural mass) versus the number of iterations. Figure 9 shows how a typical constraint converges as a function of the number of iterations. 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 Figures 5 and 7. The problem has been previously studied for frequency constraint in Reference 6. The structure is symmetric with respect to its middle surface which corresponds to the x-y plane in Figure 5. 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 5, that is material oriented at 0° has fibers running spanwise while material at 90° has fibers running chordwise. The skins are represented by QUAD4 and TRIA3 membrane elements and the webs are represented by shear panels. According to the linking scheme depicted in Figure 6, 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 QUAD4 elments, 12 TRIA3 elements adn 142 shear panels. The total number of nodes is 132.

The design constraint was the maximum deflection at the tip of wing equal to 10.0 in. The results are shown in Figure 10 and 11 for the objective function and the tip deflection for the number of iterations.

CONCLUSIONS

The design sensitivity capability for composites to be available in Version 66 of MSC/NASTRAN was designed for generality, whereby the design variables can be lamina thicknesses, orientation angles, material properties or a combination of all three. It is envisaged that this capability would constitute a powerful first step towards optimizing composite structures.

Futhermore as part of a research effort MSC/NASTRAN was linked to a general purpose optimizer CONLIN for fully automated structural design synthesis. The coupling of a large scale Finite element package like MSC/NASTRAN with a powerful optimizer like CONLIN would 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.

Acknowledgement

The authors express their appreciation to Ruthesther Willis for her typing of the manuscript.

REFERENCES:

[1] RSL, 'Design Sensitivity - Design Document', contract number - EC1038, Revised 1/20/83.

[2] RNS-11, 'Design Document for Layered Composite Capability in MSC/NASTRAN'.

[3] DNH-40, 'Improved transverse shear stiffness coefficients for the PCOMP

composite elements'

[4] Gopal Nagendra, 'Lecture Notes on MSC/NASTRAN composites structural analysis to Lockheed California Company', June 1985.

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

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

[7] C. Fleury and V. Braibant, "Structural Optimization - A New Dual Method Using Mixed Variables", Int. J. Num. Meth. Eng., to appear, 1986.

[8] 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.

Tables 1 and 2 are the composites equivalent of the element dependent data, Section 4.3 of User's Manual for tables of element Stress and Force item codes.

Component	Item	
1	Element ID	Repeat for each ply
2	Ply Number	
3	Normal - 1	
4	Normal - 2	
5	Normal - 12	
6	Shear - 1Z	
7	Shear - 2Z	
8	Θ - Shear angle	
9	Major Principal	
10	Minor Principal	
11	Max-Shear	

Table 1

Composite element stresses for QUAD4, TRIA3, QUAD8 and TRIA6.

Component	Item	
1	Element ID or -1	Repeat for each ply
2-3	Theory or blank	
4	Ply Number	
5	FP (failure index for direct stresses)	
6	FB or -1 (failure index for interlaminar shear-stress)	
7	MAX of FP, FB or -1	
8	Failure Flag	

Table 2

Composite element failure indices for QUAD4, TRIA3, QUAD8 and TRIA6

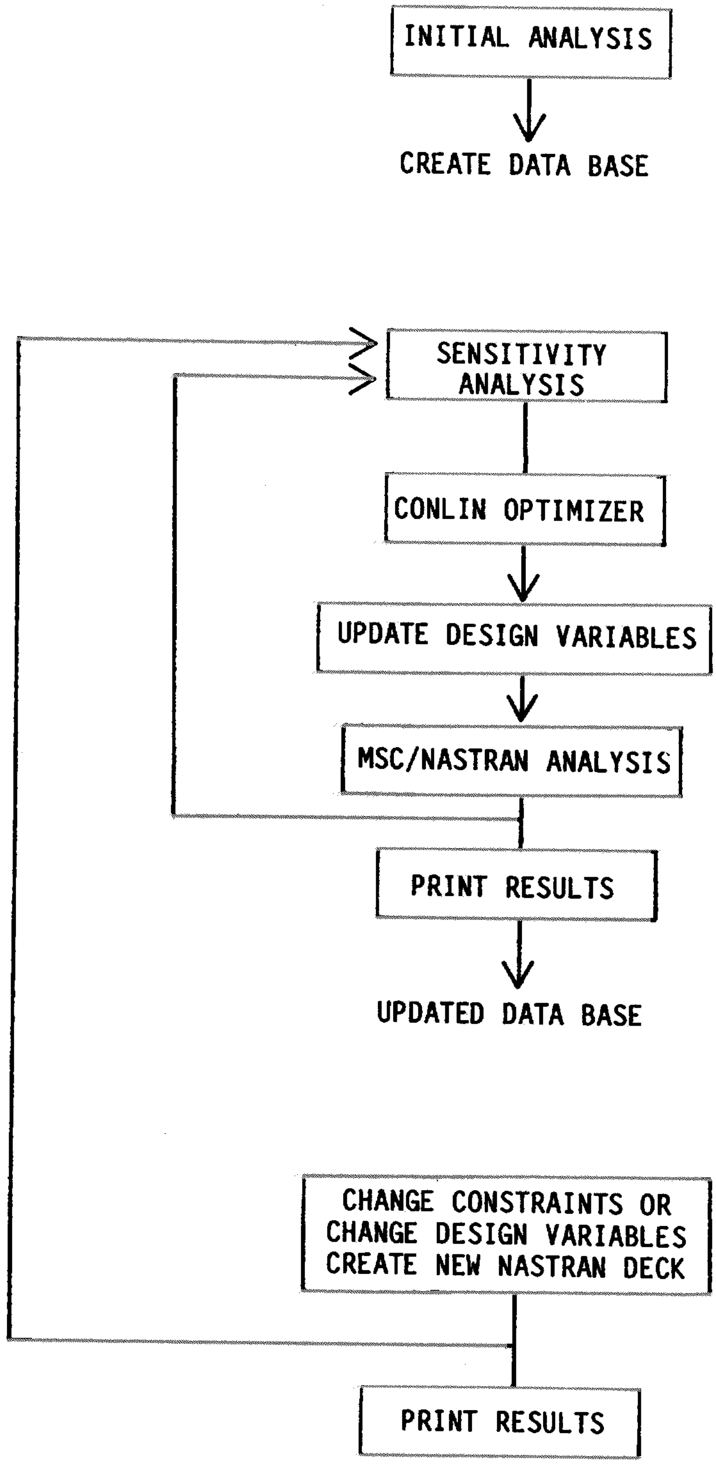

INITIAL ANALYSIS
CREATE DATA BASE
SENSITIVITY ANALYSIS
CONLIN OPTIMIZER
UPDATE DESIGN VARIABLES
MSC/NASTRAN ANALYSIS
PRINT RESULTS
UPDATED DATA BASE
CHANGE CONSTRAINTS OR
CHANGE DESIGN VARIABLES
CREATE NEW NASTRAN DECK
PRINT RESULTS

TABLE 3

ITERATION NUMBER	WEIGHT	σ_1 1(00)	σ_2 1(45)	σ_3 2(00)	σ_4 2(-45)	σ_5 3(00)
1	.3562	1.1632	1.1421	-	-	-
2	.3545	.9446	.9076	-	-	-
3	.3541	.9886	.9238	-	-	-
4	.3540	.9948	.9160	-	-	-
5	.3539	.99817	.9164	-	-	-
6*	.3612	1.2517	-	1.2079	1.2299	1.1118
7	.3600	.94754	-	.9454	.9601	.9092
8	.3599	.98164	-	.9777	.9982	.9382
9	.3599	.98289	-	.9793	.9997	.9400
10	.3599	.98244	-	.9795	.9998	.9405

*User Intervention

TABLE 4

DATA DECK

Input Data Card DSCØNS Design Constraint

Description: Defines a Design Constraint

Format and Example:

1	2	3	4	5	6	7	8	9	10
DSCØNS	DSCID	LABEL	TYPE	ID	CØMP	LIMIT	ØPT		
DSCØNS	21	COMPOS	CSTR	10	3	25.0+E3	MAX		

Field	Contents
DSCID	Design constraint identification number (Integer > 0). Must be unique for all DSCØNS.
LABEL	Label used to describe constraint in output (BCD)
TYPE	Type of constraint:

DISP	DISPLACEMENT
STRESS	ELEMENT STRESS
FØRCE	ELEMENT FØRCE
CSTR	LAMINA STRESS IN ELEMENTS FOR COMPOSITES
CFØR	FAILURE INDEX FOR A LAMINA IN ELEMENT FOR COMPOSITES
LAMA	EIGENVALUE or BUCKLING LØAD FACTØR
FREQ	FREQUENCY

ID — Identification number of constraint, i.e. GRID ID, ELEMENT ID, or MØDE NUMBER (Integer > 0)

CØMP — Component/Item to be constrained (Integer > 0)

For grid point components, refers to TX, TY, TZ, RX, RY and RZ using 1, 2, 3, 4, 5 and 6, respectively.

For scalar points, a value of 0 is used.

For element dependent data, refer to Section 4.3 for tables of Element Stress and Force Item codes.

For Buckling and Normal Modes CØMP is not used.

LIMIT — Value of limit (Real) (Default = 0.0)

ØPT — Constraint equation option (BCD - MAX or MIN). (Default is MAX)

if ØPT is MAX then:

$$\Psi = \left(\frac{\text{Constraint}}{|\text{Limit}|} - 1. * \text{sign(LIMIT)}\right) \quad \text{(If LIMIT} \neq 0)$$

$$\Psi = \text{Constraint} \quad \text{(If LIMIT} = 0)$$

if ØPT is MIN then:

$$\Psi = \left(1. * \text{sign(LIMIT)} - \frac{\text{Constraint}}{|\text{Limit}|}\right) \quad \text{(If LIMIT} \neq 0)$$

$$\Psi = \text{-Constaint} \quad \text{(If LIMIT} = 0)$$

Remark: DSCØNS cards must be selected in case control (SET2 includes DSCID).

FIGURE 1

NASTRAN DATA DECK

Input Data Card DVSET Design Variable Set Property

Description: Defines a set of element properties which vary in a fixed relation to a design variable

Format and Example:

1	2	3	4	5	6	7	8	9	10
DVSET	VID	TYPE	FIELD	PREF	ALPHA	PID1	PID2	PID3	
DVSET	21	PCØMP	13	.20	1.	99	101	110	ABC1
	PID4	PID5	etc.						
+BC1	111	122							

ALTERNATE FORM (1)

DVSET	VID	TYPE	FIELD	PREF	ALPHA	PID1	"THRU"	PID2	
DVSET	21	PCØMP	13	.20	1.	101	THRU	105	

ALTERNATE FORM (2)

DVSET	VID	TYPE	FIELD	MIDV		PID1	PID2	PID3	
DVSET	21	PCØMP	12	134		99	101		

Field	Contents
VID	Identification number (Integer > 0)
TYPE	Type of element property card, e.g., PSHELL. (See Remark 7.)
PID	Property card identification number (Integer > 0)
FIELD	Word number on the element property card to be varied. (Integer > 2). Field number for the Nth continuation property cards is 10 * N + FL where FL is the local field number of the continuation card. (See Remark 6.)
PREF	Reference value for element property (Real ≠ 0.0 or blank).
MIDV	Material identification of material property after a design change of DELTAB (See DVAR card). Note FIELD must specify a material ID field on the property card. (Integer > 0).
ALPHA	Exponent, alpha, of the actual element property versus the design variable (Real ≠ 0.0) (Default = 1.0)

FIGURE 2

(Continued)

DVSET (Cont.)

Remarks: 1. There is no restriction on the number of DVSET cards which may reference a given VID.

2. If PREF is blank, the corresponding value on the property card will be used. Non-blank PREF values are required when the basic property value is 0.0.

3. The form of PREF is

$$P = P_0 + P_{ref} * (B^{alpha} - 1.0)$$

4. The form of MIDV is

$$M = M_0 + (MIDV - M_0)/DELTAB * (B^{alpha} - 1.0)$$

5. MIDV material states correspond to a design vari[illegible] at $B = (1. + \Delta B)$.

6. For the BEAM and BEND elements, FIELD is a negative integer and corresponds to the WØRD number in the EPT section of the EST table as described in Section 1.15, preceded by a minus sign.

7. DVSET cards are selected by DVAR Bulk Data cards.

8. Since this card references only property cards ("Pxxx"), this implies that only elements with property cards may be used as design variables. This excludes elements such as CØNRØDs and CØNM2s. However, these elements may be designated as design constraints if they have force or stress output.

Input Data Card DVAR Design Variable

Description: Defines a Design Variable for a Design Sensitivity Analysis

Format and Example:

1	2	3	4	5	6	7	8	9	10
DVAR	BID	LABEL	DELTAB	VID	VID	VID	VID	VID	
DVAR	10	LFDØØR	.01	2	4	5	6	9	ABC1
	VID	etc.							
+BC1	10								

Field	Contents
BID	Design variable identification number (Integer > 0). Must be unique for all DVAR.
LABEL	Label used to describe variable in output (BCD)
DELTAB	The change ΔB in the dimensionless design variable, B, to be used in the calculation of the design sensitivity coefficients. (Real) (Default = .02)
VID	Identification number of DVSET card(s).

Remarks: 1. DVAR cards must be selected in case control (SET2 includes BID).

FIGURE 3

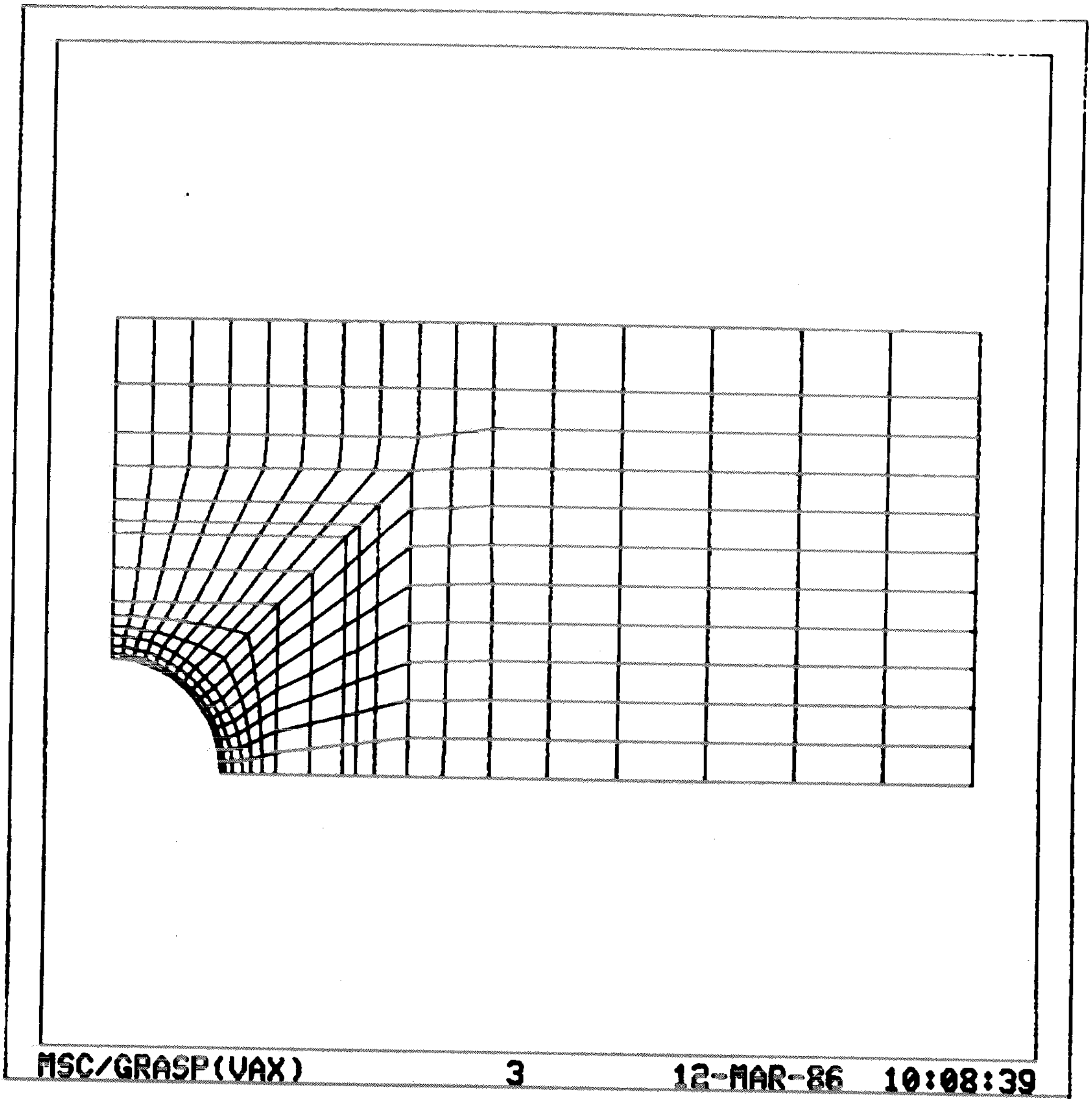


FIGURE 4

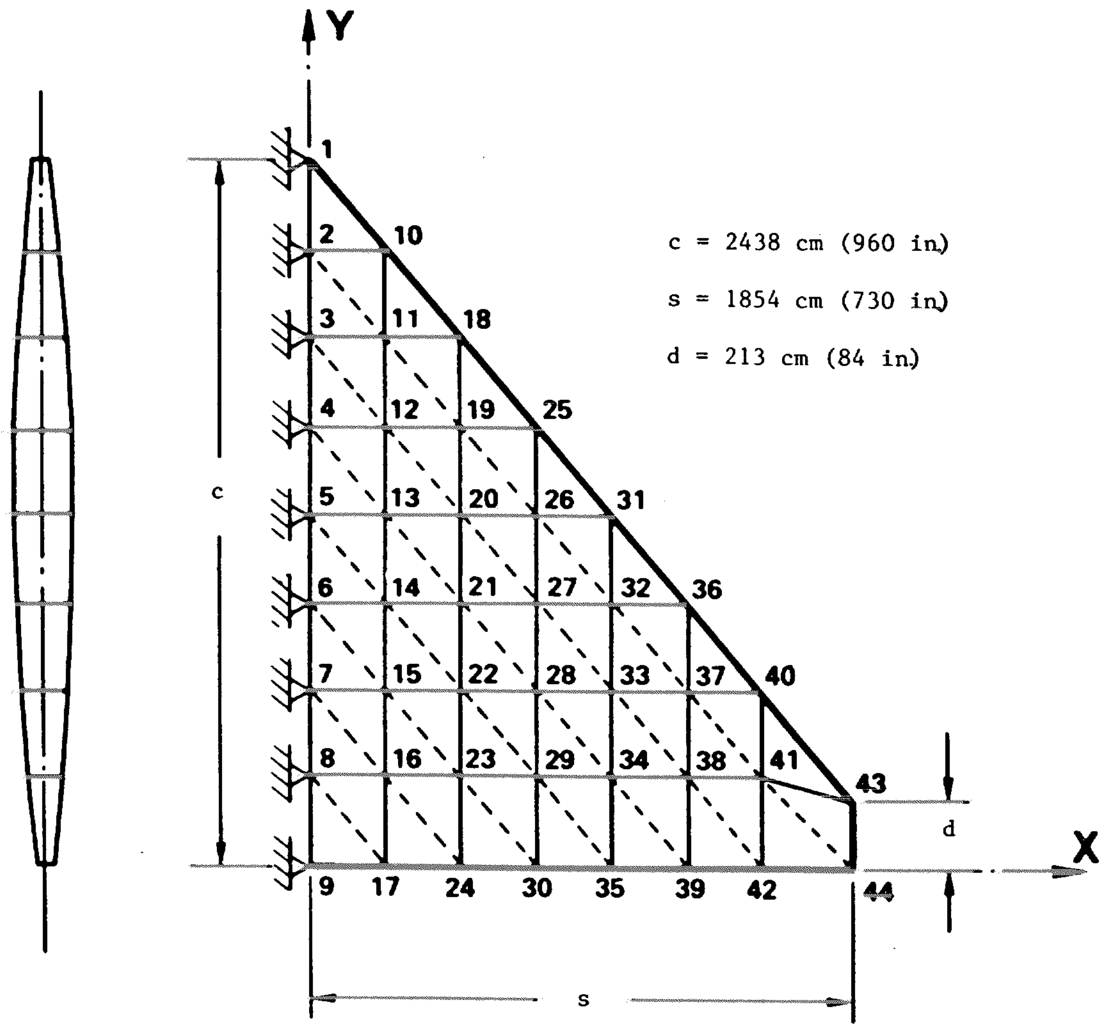


Figure 5. Delta Wing Analysis Model (Problem 2).

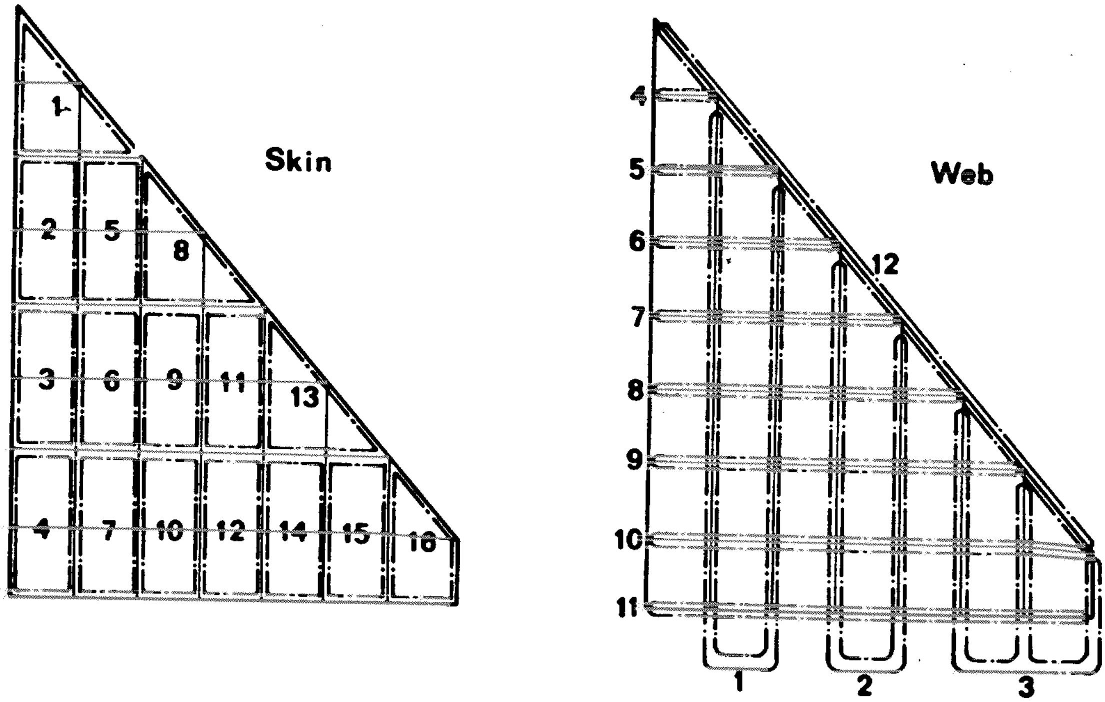


Figure 6. Delta Wing Design Model (Problem 2).

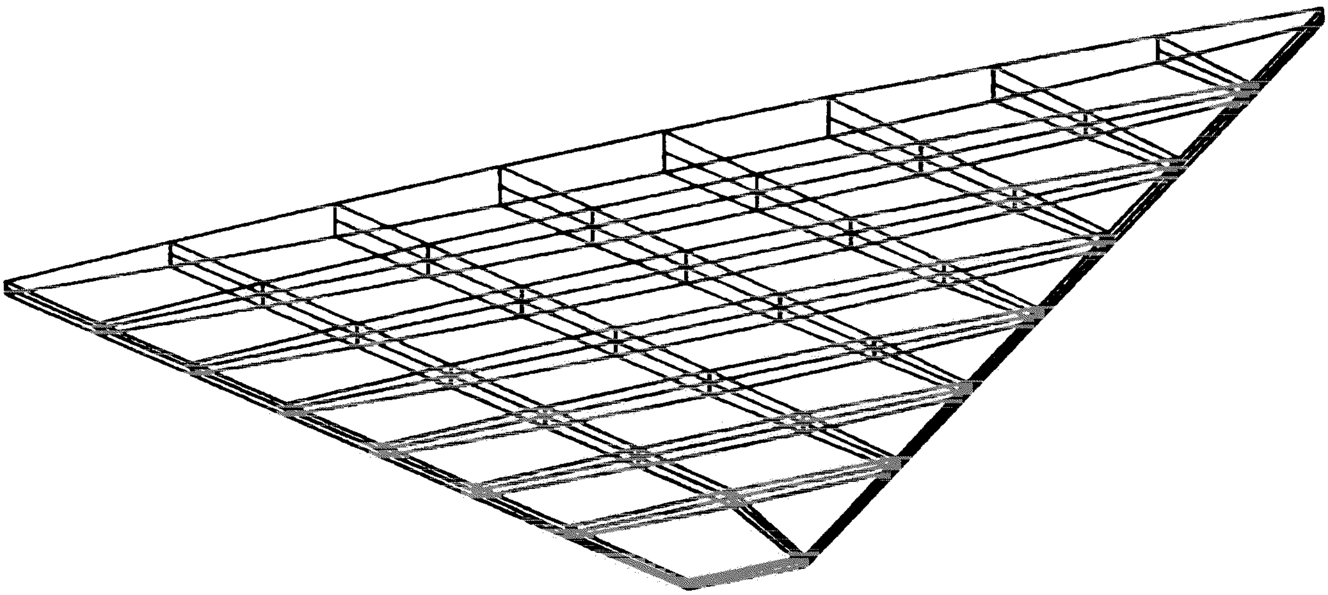


FIGURE 7

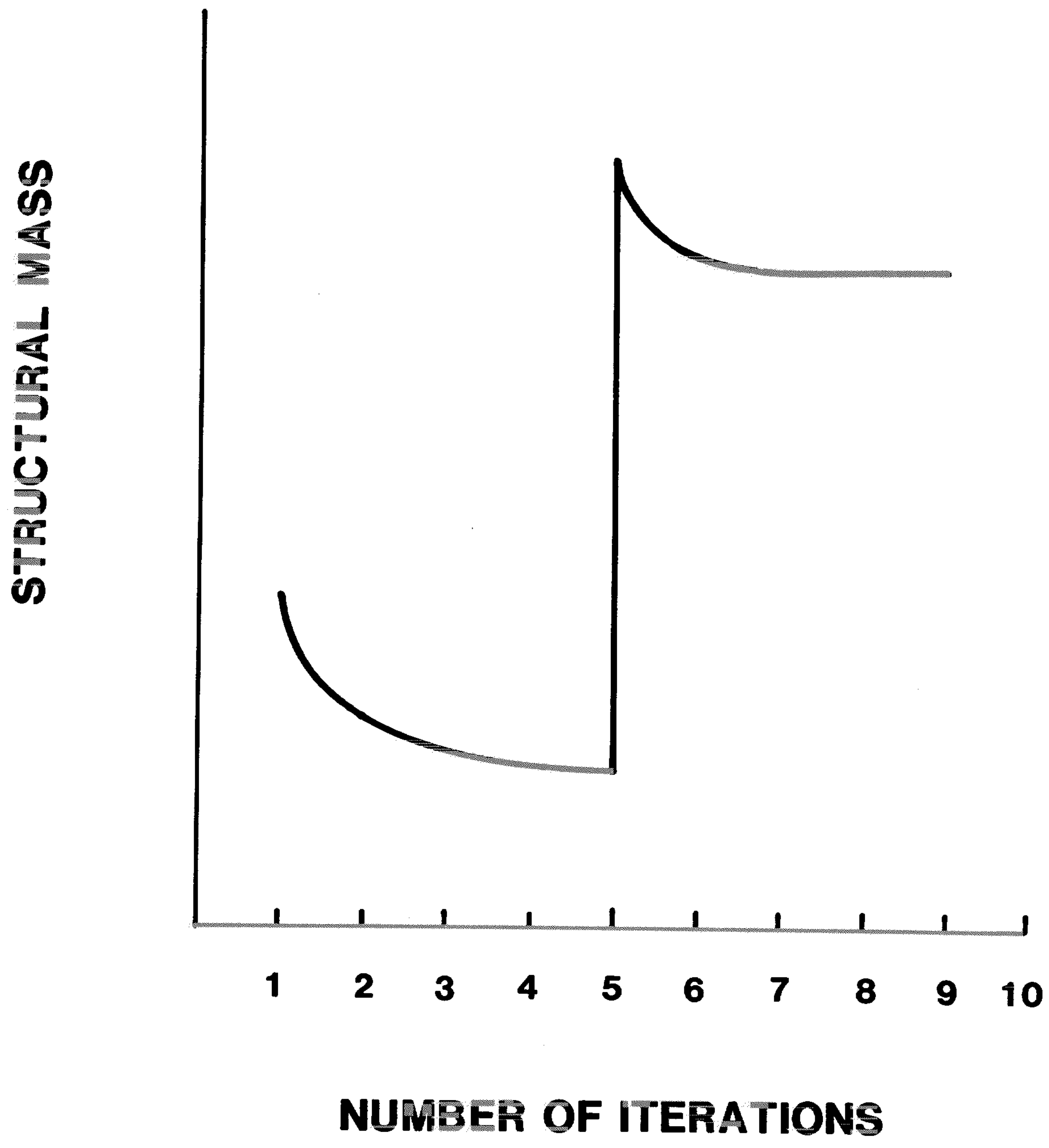
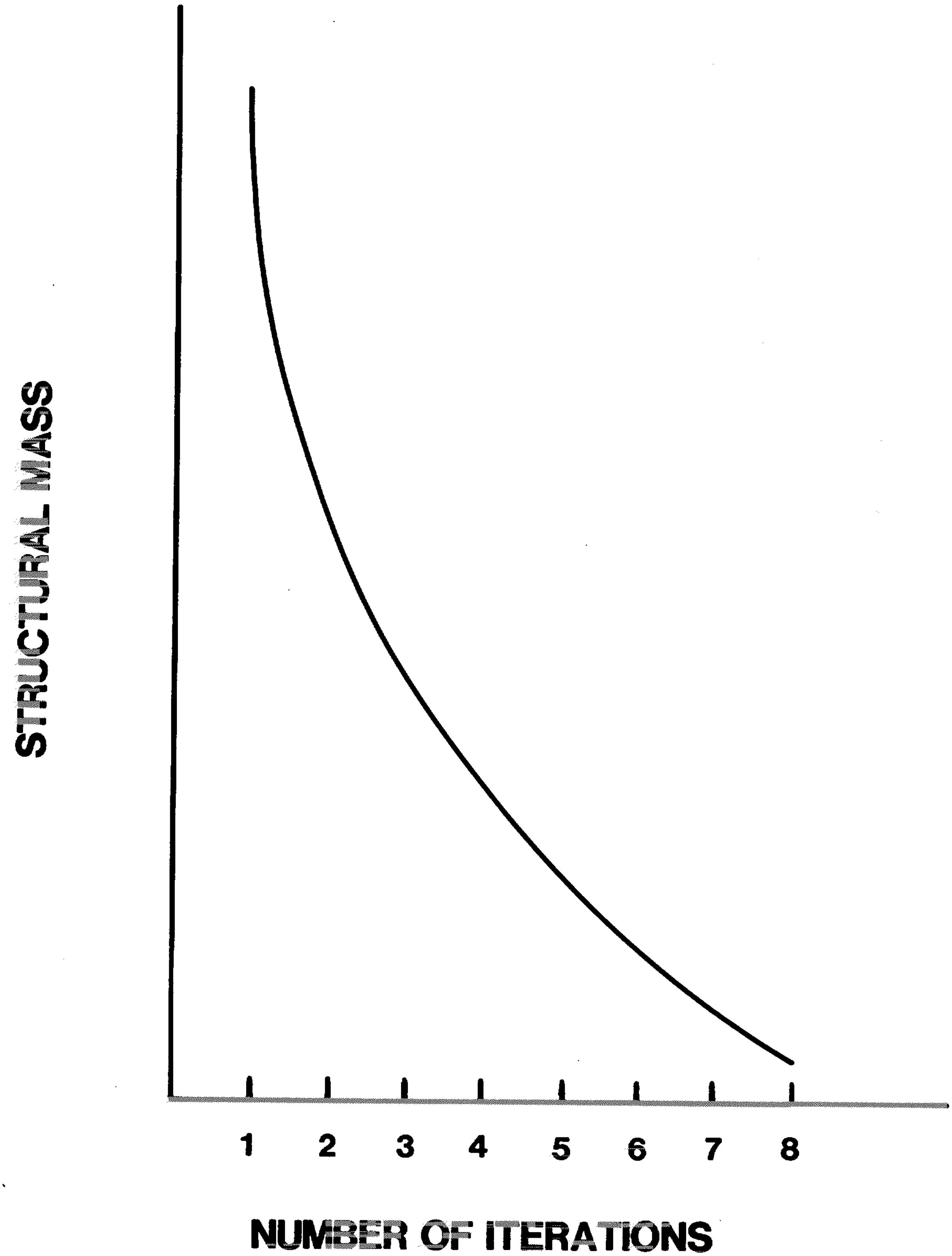


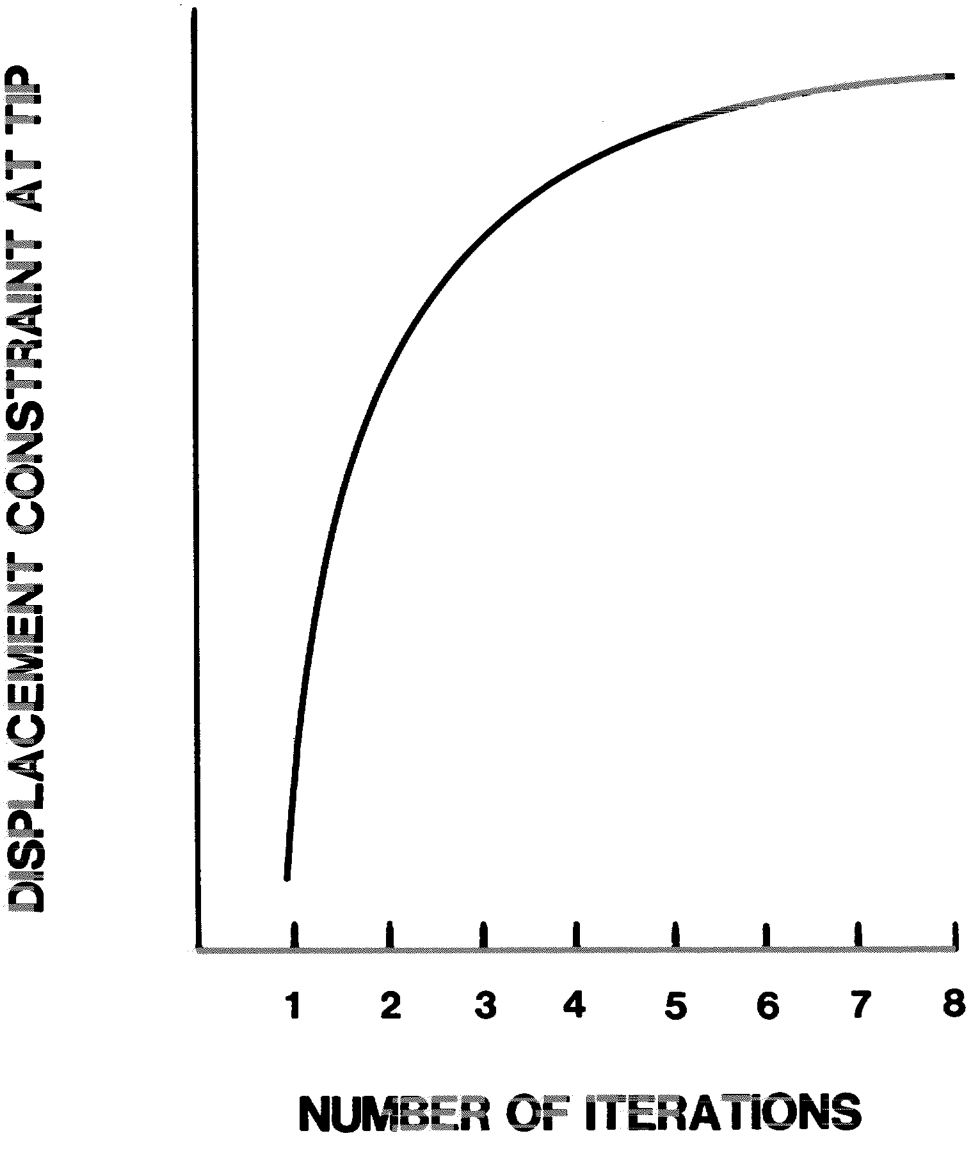
FIGURE 8


FAILURE INDEX CONSTRAINT 1
1
2
3
4
5
6
7
8
9
10
NUMBER OF ITERATIONS

FIGURE 9



STRUCTURAL MASS
1
2
3
4
5
6
7
8
NUMBER OF ITERATIONS


DISPLACEMENT CONSTRAINT AT TIP
1
2
3
4
5
6
7
8
NUMBER OF ITERATIONS