Design Optimization Codes for Structures: DOCS Computer Program

D. T. Nguyen*
Northeastern University, Boston, Massachusetts
and
J. S. Arora† and A. D. Belegundu‡
The University of Iowa, Iowa City, Iowa

A finite-element-based computer program called Design Optimization Codes for Structures (DOCS) has been developed. The program can be used to optimize elastic structural systems for a variety of design environments. The main features of level 2.0 of the code are: 1) two- and three-dimensional structural systems can be modeled using rod, beam, constant strain triangular, shear panel, and fiber-reinforced composite finite elements (beam and rod elements of various cross-sectional shapes available); 2) large structural systems are handled by the use of substructural formulation; 3) damage-tolerant design capability is provided by including in the design process the probable future damage to the structure; 4) specifications of the American Institute of Steel Construction and Aluminum Association of America can be used in the design process; 5) acceleration loads can be specified for the structure; 6) free format input and several other flexibilities have been incorporated into the code. These include design-variable linking, fixed design status for parts of the structure, direct input of substructural stiffness and mass properties, and specification of various constraint combinations such as stress, deflection, member buckling, natural frequency, and member sizes. This paper describes these features of the DOCS program and presents some structural design applications.

Introduction

THIS paper describes a computer program that has been developed for design optimization of structural systems. The program uses the finite-element method for analysis of the structure, the well-developed design sensitivity analysis methods, and a very simple optimization procedure. The program is called DOCS, which stands for Design Optimization Codes for Structures. The current version of the program has a limited finite-element library. Different types of constraints and their combinations can be specified. A unique feature of the program is that the effect of probable future damage to the structure can be included in the design process. Definition of the design optimization problem for the program is kept quite general. A free format input for the program has been designed which closely matches the input form for the NASTRAN computer program.

During the past two decades considerable research has been done to develop optimization methods for the design of structural systems. Benefits of the use of optimization techniques in the design process have been demonstrated. It is clear that, with the use of optimization techniques, the practical design process can be accelerated and automated to some extent. However, research continues on the development of new algorithms and the testing of the existing algorithms to determine which of the methods is most suitable for structural optimization.

Whereas application of the optimization techniques for design of industrial products has appeared in the recent literature, the use of such techniques is not widespread. One reason for this lack of usage is that general-purpose, well-tested computer codes are not available for the design optimization of engineering systems. Some special-purpose applications-oriented computer programs have been developed. These include TRUSSOPT[1] and other programs presented in Ref. 2. These programs have been used successfully to optimize certain classes of structural design problems. What is needed is general-purpose design-oriented software that can treat a variety of design environments. To develop such a capability, one needs to integrate existing structural analysis programs into optimization algorithms to develop general and user-oriented software for structural optimization. Some progress has been made in this regard.[2] The current version of the computer program DOCS presents an intermediate step in achieving this objective of general-purpose design-optimization software.

Design Problem Definition and Solution Methods

Design Problem

The most practical way of analyzing the behavior of a large and complex structure is the finite-element approach. The governing equilibrium equation for a finite-element model of structures subjected to quasistatic loads is

$$K(b)z=S(b) \tag{1}$$

where

$K(b)$ = $n\times n$ symmetric nonsingular structural stiffness matrix
$S(b)$ = n vector representing equivalent nodal loads for the finite-element model
z = n vector of nodal displacements for the finite-element model of the system
b = k vector of design variables for the problem

Another equilibrium equation that governs the free-vibration response of a structure or its buckling behavior is the eigenvalue problem

$$K(b)y=\zeta M(b)y \tag{2}$$

where

$M(b)$ = $n\times n$ structural mass matrix for the vibration problem and the geometric stiffness matrix for the buckling problem

---

Received Aug. 2, 1982; revision received Feb. 9, 1983.

*Assistant Professor, Department of Civil Engineering.
†Professor, Division of Materials Engineering.
‡Assistant Professor, Division of Materials Engineering. Currently at GMI Engineering and Management Institute, Flint, Mi.

ζ = eigenvalue related to the natural frequency or the buckling load for the problem
y = eigenvector

When any future damage to the structure has to be included in the design process, Eqs. (1) and (2) must be solved for each damage condition. Only then can one impose stress displacement and eigenvalue-type constraints in the design process. When the structure is very large, it is sometimes advantageous to partition the structure into several smaller substructures and analyze Eqs. (1) and (2) using substructural matrices. It is understood that these equations can be solved using substructures if necessary.[3,4]

A mathematical model for optimal design of the linearly elastic structural systems is defined as follows:

Find a design variable vector b that minimizes a cost function

$$\psi_0(b,z,\zeta) \tag{3}$$

and satisfies Eqs. (1) and (2) and the constraints

$$\psi_i(b,z,\zeta) \leq 0, \qquad 1,2,\ldots,m \tag{4}$$

This design optimization model is quite general, since the stress displacement and eigenvalue-type constraints for the damaged as well as undamaged structure can be included in Eq. (4). Similarly, the cost function of Eq. (3) can represent a variety of design objectives. However, in DOCS, total weight of the structure is treated as the cost function. For more discussion on the design optimization model, see Ref. 3.

Design Sensitivity Analysis

A major step in any optimization method is the calculation of gradients of cost and constraint functions. Calculation of such gradients for the structural design problem is called the design sensitivity analysis. It is critically important to realize that ψ_0 and ψ_i are implicit functions of the design variable b because z and ζ are implicit functions of b. To calculate gradient of these functions, one must use implicit differentiation procedures. Such differentiation procedures have been developed in the literature. References 3, 5, and 6 can be consulted for more details on such procedures. In DOCS, a substructural formulation is used in the optimization process. Therefore, the design sensitivity analysis with substructures must be used. An efficient method for design sensitivity analysis with substructures has been presented in Refs. 7 and 8. That method is utilized in DOCS computer program to calculate gradient of stress and displacement constraint functions for any damaged or undamaged structure.

Optimization Algorithm

Once cost and constraint functions and their gradients have been evaluated, gradient-based algorithm can be used to optimize the structure. However, some algorithms are more suitable than others for structural optimization problems. The main consideration in selecting an optimization algorithm is that it should need a minimum number of function and gradient evaluations. Structural analysis and design sensitivity analysis are the two major calculations in optimal design of structural systems. To evaluate stresses, displacements, and eigenvalues, one must analyze large finite-element models. This can be quite expensive. Therefore, it is desirable to keep the number of function evaluations at a minimum.

Most existing optimization methods require a one-dimensional search in their algorithm. The one-dimensional search may require several function evaluations. It can be highly inefficient, since each call to function evaluation requires analysis of the structure. Therefore, it is desirable to use an optimization method that does not depend upon one-dimensional search in the design process. A very simple-minded optimization algorithm has been incorporated in the DOCS computer program. The algorithm is a generalization of the steepest descent method for unconstrained problems. In this algorithm, a constrained steepest descent search is used to compute changes in design.[3] A certain amount of user interaction and experience with the algorithm is desirable when using the program.

Recently, a more sophisticated optimization algorithm has been developed which does not rely on one-dimensional search in the design process.[9,10] This method will be incorporated into future updates of DOCS.

Capabilities and Features of DOCS

In this section, salient features and capabilities of the DOCS computer program are described.

Finite-Element Library

The basic finite elements such as axial force element (truss), constant strain triangle (CST), and symmetric shear panels (SSP) are available in DOCS. These elements are simple, since their stress is independent of design variables. (Thickness of plate and area of axial force element are normally chosen as design variables.) Also, for these elements the stiffness matrix is linear in design variables. Thus these calculations are quite straightforward. For many structures, however, it is necessary to have finite elements with bending stiffness. For this reason, beam elements with various cross-sectional shapes are included in DOCS. Two- and three-dimensional beam elements having various cross-sectional shapes can be specified. Also, the ends of a member may be specified to be pin-jointed member connections. The plate bending element is not available in the current version of DOCS.

Multilayered Fiber-Reinforced Composite Material

In structural optimization problems where the objective is to minimize the weight, composite materials with a high ratio of material strength to density offer a great advantage in minimizing the weight of the structure. The option of using multilayered composite material for the CST element is provided in the code. When this option is used, fiber direction cosines with respect to a global coordinate system are supplied by the user. The program internally calculates the direction cosines of the fiber with respect to the element local axis. The thickness of each layer is treated as a design variable in this case.

Damage-Tolerant Design Option

In DOCS, structures can be optimized with or without projected future damage conditions included in the design process. The damage to the structure may be defined by removing either some nodes or some selected members. Partial of full damage to a member can be specified as input data by assigning proper reduction factors for stiffness. A member is considered completely damaged if a reduction factor of one is assigned to it. In such a case, the member is effectively removed from the model. The basic assumption in the method is that the structure is indeterminate and remains geometrically stable after damage to its members and/or nodes. In other words, the structure does not fail in a mechanism-type motion after it is damaged.

Substructural Formulation

It has been shown in Refs. 8 and 11 that substructural formulation offers computational advantages in analysis as well as design sensitivity analysis for the class of problems formulated and solved in DOCS computer program. Large-scale design problems that cannot fit into the computer memory can be treated by partitioning the structure into many smaller substructures. In DOCS, the structure can be analyzed and designed with or without substructures. Also, a provision is made so that addition of substructures to the existing structure can be achieved by supplying the stiffness and mass properties of additional substructures directly to the specified nodes. A modified subspace iteration technique[4] is used to solve the eigenvalue problem of Eq. (2) with substructures.

Types of Constraints

In the DOCS computer program, several options for specifying constraints are available. These include fundamental frequency, member stress, nodal displacement, member buckling, and minimum and maximum sizes of members. In frequency analysis, the structural mass matrix can be obtained based on either the consistent or lumped mass approach. Direct and buckling stresses may be imposed simultaneously for axial force members. For CST and shear panels, an interaction equation of the following form is imposed:

$$\left(\frac{\sigma_1}{F_1}\right)^2+\left(\frac{\sigma_2}{F_2}\right)^2-\left(\frac{\sigma_1\sigma_2}{F_1F_2}\right)+\left(\frac{\sigma_{12}}{F_{12}}\right)^2\leq 1 \tag{5}$$

where σ_1, σ_2, and σ_{12} are the calculated stresses, and F_1, F_2, and F_{12} are the allowable stresses in directions 1 and 2, respectively. For isotropic material, $F_1=F_2$ and $F_{12}=F_1/\sqrt{3}$. For the composite CST element, direction 1 is along the fibers, and direction 2 is normal to it.

For beam elements, the familiar interaction equation is imposed:

$$\frac{f_a}{F_a}+C_1\frac{f_{bz}}{F_{bz}}+C_2\frac{f_{by}}{F_{by}}\leq 1 \tag{6}$$

In addition, the shear stress constraint is imposed as

$$f_s/F_s\leq 1$$

Here f_a is the axial stress in the element; f_{bz} and f_{by} the bending stresses due to bending about the y and z axes, respectively; f_s the shear stress; F_a, F_{bz}, F_{by}, and F_s the corresponding allowable stresses; and C_1 and C_2 the magnification factors.[12] The allowable stresses have been coded according to the AISC specifications[12] as well as the specifications of the Aluminum Association.[13] The user may choose between these two specifications as well as a "user-supplied constraint code" in which the values of C_1, C_2, F_a, F_{bz}, F_{by}, and F_s are read in as input data. In addition, for beams with hollow tubular cross sections, the von Mises yield condition may be imposed in lieu of the above interaction equations. The von Mises constraint is

$$\sigma^2+3\tau^2\leq\sigma_a^2$$

where σ is the axial stress in the tube, τ the shear stress, and σ_a the allowable yield stress.

Design Variable Linking

For ease of fabrication, it is often necessary to impose symmetry conditions in the structure. To impose such conditions, several members must be of the same size and shape. This is especially true with large-scale problems. With the design variable linking option available in DOCS, many members may be assigned the same design variables. Also, an option is provided in the program such that design of certain parts of the structure may be kept fixed. With this option, the value of design variables for this part of the structure will not change in the design process.

Various Types of Cross-Sectional Shapes

The axial force and beam elements may have different cross-sectional shapes. The cross-sectional area of the axial force element is normally chosen as design variables. However, the user may choose dimensions of various cross-sectional shapes as design variables. The same is true for the beam finite element. Several cross-sectional shapes are available in DOCS.[14]

Unit Design Variable Concept

In DOCS, calculations that do not change during the optimization process are performed outside the optimization loop to reduce computational effort. Element stress, stiffness, and mass matrices, for example, are calculated and stored on direct access files with unit design variable values outside the optimization loop. During the assembly process, element stiffness and mass matrices are recalled from these files. These matrices are then multiplied by the actual value of design variables prior to assembling the structural stiffness and mass matrices.

Body Forces

If the structure is given an acceleration in some direction, then the resulting body forces are computed and added to any other external loads that may have been applied to the structure. The resultant applied load vector S is calculated and used in the right-hand side of Eq. (1). The input acceleration vector may consist of translational as well as rotational components. Furthermore, the acceleration is allowed to vary in magnitude along some particular axis in the global coordinate system. This is handled by specifying in the input data the components of the acceleration vector at various points along the axis. Acceleration of any point in the structure is obtained through linear interpolation. A lumped mass matrix is used in computing the body forces due to accelerations. Rotational inertia of the elements is neglected. It should be noted that, since the mass of the structure is a function of the design variables, the body forces due to acceleration are computed within the optimization loop.

Fully Stressed Design

The computer program DOCS has an option in which one can perform fully stressed design for initial few iterations. This option is quite useful as it gives a very good starting design for the optimization process. Thus, substantial computing efforts can be saved with the use of the fully stressed design option.

Free-Format Input

To reduce the task of preparing input data specially for large-scale problems, all input data to DOCS are provided in free format. The input data deck is divided into three subdecks: executive control, case control, and bulk data. For each type of data deck, well-defined cards are developed to provide input to the program. These input data cards are quite similar to those for the well-known structural analysis program NASTRAN. Example of an input data card is given in Table 1. Thus, the user can augment the NASTRAN data deck for use with the DOCS computer program.

Table 1 Sample input data card for CROD element

Bulk data card:	CROD							
Description:	Specifies the properties of the truss element							
Format:								
1	10	11					76	80
CROD		l	m_1	n_1	n_2	m_2		
Examples:								
CROD		10	6	8	69	96		

Remarks:

1) l = element identification number ($l>0$)
   m_1 = materials property code number
   n_1,n_2 = grid points to which the element is connected ($n_1\neq n_2$, $n_i>0$)
   m_2 = design variable code number (group number for members)
2) l,n_i, and m_j are all integer constants.
3) m_2 is unique unless design-variable linking is implied.

Computational Aspects

Several direct and sequential access files are used in the computer program. These files contain data generated outside as well as within the optimization loop.

Decompositions of the boundary stiffness matrix and interior stiffness matrices for substructures are stored on the direct access files. These matrices can be retrieved any time for future calculations. Although the restart option is not available in the current version of the program, it can be easily incorporated.

To calculate gradients of the cost and constraint functions, the most efficient method of the design sensitivity analysis is used. The program automatically uses the substructural formulation for design sensitivity analysis if that option is specified. The design sensitivity coefficients are stored on direct access files. These design sensitivity coefficients are useful in their own right, since they can be used in the traditional design process. This is explained in Ref. 15. To calculate the gradients in the most efficient manner, the specific form of the various constraint functions is automatically utilized in the program. For further explanations of this, see Ref. 3. One of the main features of the computer program is that it requires function as well as gradient evaluations only once during an optimization cycle.

Reliability

Many small-scale examples have been used in the debugging phase of the computer program. These problems are such that closed-form solutions for the problems are possible. Thus, the sensitivity calculations in the program can be easily checked. The structural analysis part of the DOCS program has been verified by comparing solutions with the SAP4 solutions.

Machine Independence

The program is written in standard FORTRAN. Except for the file operations, the program is essentially machine independent. The program has been developed on the IBM 370/158 computer. It recently was converted to PRIME and VAX minicomputers. The program is in the process of being converted to the CDC 7600 computer and the HARRIS minicomputer.

Table 2 Design data for one-bay, two-story frame

Modulus of elasticity	10,900 ksi
Material weight density	0.1 lb/in.^{3}
Yield stress in tension/compression	58 ksi
Yield stress in shear	34 ksi
Lower limit on design variables (same for each group)	{1.0, 0.3, 0.1, 0.1} in.
Cross section used	I section with four design variables
Upper limit on design variables	None

Displacement limits for 18 degrees of freedom: limit on the ratio of the displacement of a joint to its height from the base is 0.002. Also, displacement limit within any element is $lm/360$, where l_m is the length of the element. Thus, the limits are as follows:

Node	x_1 limit, in.	x_2 limit, in.
1	0.360	0.360
2	0.670	0.670
3	0.720	0.720
4	0.360	0.360
5	0.670	0.670
6	0.720	0.720

Two loading cases (shown in Fig. 1) as follows:

Loading condition	Load description
I	45.0 kip along x_1 direction at nodes 1, 3
II	0.5 kip/in. uniform loads along negative x_2 on elements 2, 3, 6, 7

Applications

Several structural design examples have been solved using the computer program DOCS.[16] These include small-scale as well as large-scale problems. In this section, three such applications are presented.

Example 1: Design of Framed Structures

A one-bay, two-story frame (shown in Fig. 1) is optimized. Design data for the frame are given in Table 2. Design variable linking is used to obtain a symmetric structure. There are four design groups, each consisting of two finite elements. The cross section of each element is an I section with four design variables: depth, flange width, web thickness, and flange thickness. There are thus a total of 16 design variables. The aluminum stress code is imposed to limit the stresses within each finite element.

The initial design, iteration history, and final design, along with other relevant information on the optimization process, are given in Table 3. For this example, the method converged in just four iterations.

Example 2: Design of Composite Closed Helicopter Tail-Boom without Damage Consideration

In this example, a helicopter tail-boom is modeled as a closed structure that is obtained by using 48 CST elements as a skin to cover 108 truss members. The structure has 28 joints and 72 degrees of freedom.[3] The geometry of the idealized structure and the design loads are given in Fig. 2. The truss element numbering system for a typical panel is shown in Fig. 3. The CST elements are used over the truss member to cover the tail-boom. Connecting nodes for the CST elements are defined as 1-7-3, 5-7-1, etc. The top and bottom skin members are made of boron-epoxy layered composite, each having three layers. The material properties of boron-epoxy are given in Table 4. The skin elements on two sides are made of an aluminum alloy sheet (7075-T6 clad aluminum). CST elements 1-4 represent the top skin, 5-12 the two side skins, and 13-16 the bottom skin for the first two bays of the helicopter. The same pattern for the CST element numbering system is repeated for the remaining four bays. The material properties of clad aluminum (CST elements) and aluminum alloy (truss elements) are given in Table 5. Table 5 also contains other design data for the structure.

The problem is to minimize the total weight of the structure and at the same time to insure that member stress, nodal displacement, member buckling, and natural frequency

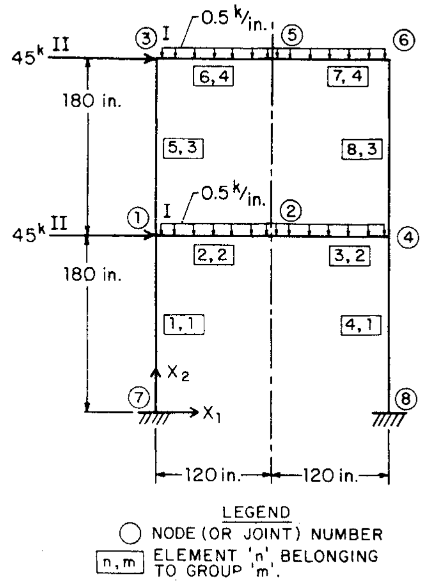


Fig. 1 One-bay, two-story frame.

Table 3 Results for one-bay, two-story frame

Initial design (depth, flange width, web thickness, and flange thickness):
$b^{(0)} = \{(30.0, 15.0, 2.0, 2.0), (30.0, 15.0, 2.0, 2.0), (30.0, 15.0, 2.0, 2.0), (30.0, 15.0, 2.0, 2.0)\}$ in.

Initial weight = 13.44 kip
Final design:
$b = \{(30.142, 14.945, 0.6954, 1.7743), (30.1370, 14.9780, 1.0789, 1.8639) (30.0210, 14.870, 0.3795, 1.0947), (30.0000, 14.892, 0.8436, 1.1752)\}$ in.

Iteration	Weight, kip	Maximum violation
1	13.440	0.0208
2	8.724	0.0720
3	8.088	0.0249
4	7.551	0.0026

Active constraints at optimum: Displacement limit along x_1 direction of node 5 in loading condition 1.

Total CPU time = 43.5 s on IBM 370/158 in double precision.

Table 4 Material properties and allowable strength of boron-epoxy

Elastic modulus, E_{11}	30,000 ksi
Elastic modulus, E_{22}	27,000 ksi
Poisson's ratio	0.21
Shear modulus	6500 ksi
Specific weight	0.0725 lb/in.^{3}
Allowable stress along fiber direction (tension)	176.0 ksi
Allowable stress along fiber direction (compression)	390.0 ksi
Allowable stress along transverse fiber direction (tension)	11.4 ksi
Allowable stress along transverse fiber direction (compression)	44.6 ksi
Allowable shear stress	2.1 ksi

constraints are satisfied under the projected loading condition. The design parameters to be calculated are the cross-sectional area of truss members and layer thickness of CST members. A lower bound constraint is also imposed on the cross-sectional area and thickness of an element. An upper bound constraint is imposed only on the CST elements.

The members of the truss are taken to be tubular sections. Assuming the sections to be thin, the moment of inertia and cross-sectional area are given as $I = \pi R^3 t$ and $A = 2\pi Rt$, where R is the mean radius and t the thickness of the tube. In calculating the Euler buckling load, the moment of inertia is assumed to be given as $I = \beta A^2$. Therefore, $\beta = I/A^2$ is given as $R/4\pi t$. If R/t is conservatively selected as 12-14, then $\beta \cong 1.0$. This value of β is used in the calculations. The working stress for a truss member is assumed to be approximately 60% (± 25 ksi) of the yield stress (42 ksi) for the material used. This working stress corresponds to a safety factor of roughly 1.68. Displacement limits of ± 0.5 in. at the nodal points are based on approximately ⅓-deg misalignment at the center of the tail-boom. The lower limit on the member cross-sectional area is taken as 0.0415 in.^{2}, which corresponds to a tube with 0.50-in. o.d. and 0.028-in. wall thickness. The single loading condition for the structure is given in Table 5. To maintain symmetry and to facilitate fabrication of the structure, 108 truss members of the structure are divided into a total of 42 groups, and each group is assigned a design variable. Therefore, each panel of the structure (shown in Fig. 3) has seven design variables for truss members.

Design variable linking is also used for skin members for ease of fabrication. The top and bottom skins each have nine design variables. The skin elements on two sides have nine design variables. Thus, this example has a total of 69 design variables. The lower and upper bounds on layer thickness are 0.0067 and 0.17 in., respectively.

In this example, the tail-boom structure is divided into three substructures by partitioning it at nodes 9-12 and 17-20. Nodes numbered 25-28 are also treated as boundary nodes. Substructure 1 has four boundary nodes (9-12) and eight interior nodes (1-8); substructure 2 has eight boundary nodes (9-12, 17-20); and substructure 3 has eight boundary nodes (17-20, 25-28).

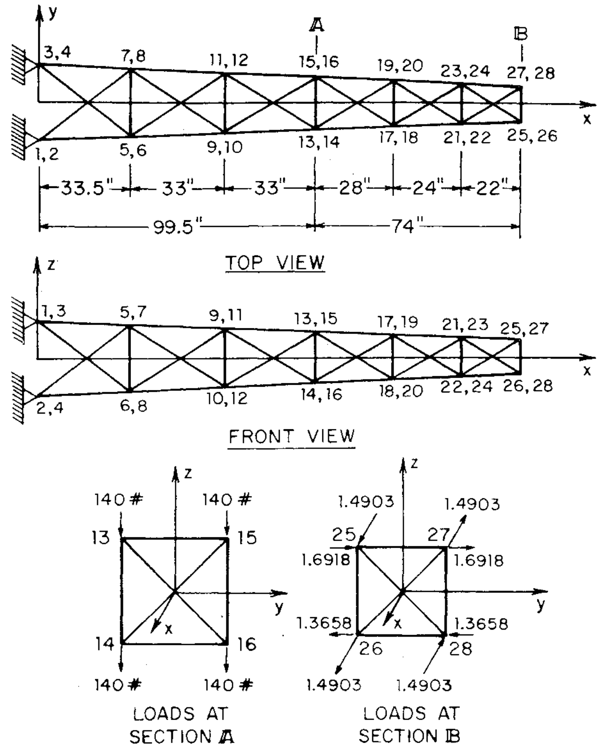
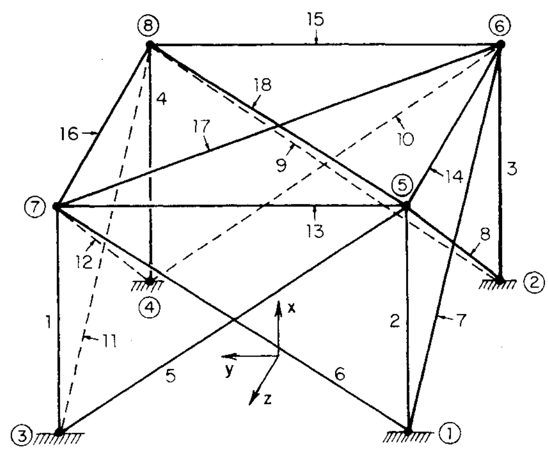


Fig. 2 Arrangement of members for the helicopter tail-boom.



Following grouping of members, with members of a group to have same cross-sectional areas is used:

Group No.	Member Numbers
1	2, 3
2	1, 4
3	5, 6, 9, 10
4	7, 8, 11, 12
5	13, 15
6	14, 16
7	17, 18

Fig. 3 Truss member numbering for the first panel.

Table 5 Design data for helicopter tail-boom

Loading for the structure

Node No.	Load component in direction, kip: x	y	z
13	0.0	0.0	−0.140
14	0.0	0.0	−0.140
15	0.0	0.0	−0.140
16	0.0	0.0	−0.140
25	1.4903	1.6918	0.0
25	1.4903	−1.3658	0.0
27	−1.4903	1.6918	0.0
28	−1.4903	−1.3658	0.0

Lower bound on natural frequency = 29 Hz

Data for truss elements

- Material: 2024-T3 aluminum alloy
- Modulus of elasticity = 10,500 ksi
- Stress limits = ±25.0 ksi
- Material density = 0.1 lb/in.^{3}
- Moment of inertia: $I=\beta A^2$; $\beta=1.0$
- Displacement limits = ±0.50 in.
- Lower limit on cross-sectional area = 0.0415 in.^{2}
- Upper limit on cross-sectional area = none

Data for skin elements (on two sides)

- Material: 7075-T6 clad aluminum
- Modulus of elasticity = 10,400 ksi
- Stress limit = 40.2 ksi
- Material density = 0.098 lb/in.^{3}
- Lower limit on thickness = 0.02 in.
- Upper limit on thickness = 0.50 in.

Table 6 Optimum designs for closed helicopter tail-boom with composite material

Design variable No.	Member No.	Example 3 (six damage conditions)	Example 2 (no damage)	Design variable No.	Member No.	Example 3 (six damage conditions)	Example 2 (no damage)
1	2,3	0.17959D 00	0.41500D−01	41	104,106	0.41500D−01	0.41500D−01
2	1,4	0.29341D 00	0.41500D−01	42	107,108	0.19239D 00	0.50965D−01
3	5,6,9,10	0.24897D 00	0.41500D−01	43	1,2,3,4	0.67000D−02	0.11076D−01
4	7,8,11,12	0.14791D 00	0.48855D−01	44	1,2,3,4	0.67000D−02	0.67000D−02
5	13,15	0.41500D−01	0.41500D−01	45	1,2,3,4	0.92961D−02	0.16028D−01
6	14,16	0.41500D−01	0.41500D−01	46	5,6,7,8	0.86139D−02	0.72838D−02
7	17,18	0.21617D 00	0.41500D−01	47	5,6,7,8	0.16479D−01	0.11069D−01
8	20,21	0.84621D−01	0.41500D−01	48	5,6,7,8	0.67000D−02	0.67000D−02
9	19,22	0.13649D 00	0.41500D−01	49	9,10,11,12,	0.67000D−02	0.82097D−02
10	23,24,27,28	0.11340D 00	0.41500D−01		13,14,15,16		
11	25,26,29,30	0.16358D 00	0.48511D−01	50	9,10,11,12,	0.67000D−02	0.82097D−02
12	31,33	0.41500D−01	0.41500D−01		13,14,15,16		
13	32,34	0.41500D−01	0.41500D−01	51	9,10,11,12	0.67000D−02	0.82097D−02
14	35,36	0.63878D−01	0.41500D−01		13,14,15,16		
15	38,39	0.46714D−01	0.41500D−01	52	17,18,19,20	0.94398D−02	0.67000D−02
16	37,40	0.55394D−01	0.41500D−01	53	17,18,19,20	0.67000D−02	0.67000D−02
17	41,42,45,46	0.41500D−01	0.41500D−01	54	17,18,19,20	0.29438D−01	0.92058D−02
18	43,44,47,48	0.48723D−01	0.50746D−01	55	21,22,23,24	0.67000D−02	0.68632D−02
19	49,51	0.41500D−01	0.41500D−01	56	21,22,23,24	0.12386D−01	0.15024D−01
20	50,52	0.41500D−01	0.41500D−01	57	21,22,23,24	0.67000D−02	0.67000D−02
21	53,54	0.61319D−01	0.41500D−01	58	25,26,27,28,	0.12349D−01	0.97540D−02
22	56,57	0.41673D−01	0.41500D−01		29,30,31,32		
23	55,58	0.44047D−01	0.41500D−01	59	25,26,27,28,	0.12349D−01	0.97540D−02
24	59,60,63,64	0.41500D−01	0.41500D−01		29,30,31,32		
25	61,62,65,66	0.63692D−01	0.41694D−01	60	25,26,27,28,	0.12349D−01	0.97540D−02
					29,30,31,32		
26	67,69	0.41500D−01	0.41500D−01				
27	68,70	0.41500D−01	0.41500D−01	61	33,34,35,36	0.67000D−02	0.67000D−02
28	71,72	0.54351D−01	0.41500D−01	62	33,34,35,36	0.85875D−02	0.67000D−02
29	74,75	0.53918D−01	0.41500D−01	63	33,34,35,36	0.16374D−01	0.90408D−02
30	73,76	0.72680D−01	0.41500D−01	64	37,38,39,40	0.12710D−01	0.67000D−02
31	77,78,81,82	0.18427D 00	0.41500D−01	65	37,38,39,40	0.14739D−01	0.69886D−02
32	79,80,83,84	0.17073D 00	0.44785D−01	66	37,38,39,40	0.67000D−02	0.67000D−02
33	85,87	0.41500D−01	0.41500D−01	67	41,42,43,44,	0.12710D−01	0.80420D−02
34	86,88	0.41500D−01	0.41500D−01		45,46,47,48		
35	89,90	0.87936D−01	0.41500D−01				
36	92,93	0.41500D−01	0.41500D−01	68	41,42,43,44,	0.12710D−01	0.80420D−02
37	91,94	0.41500D−01	0.41500D−01		45,46,47,48		
38	95,96,99,100	0.41500D−01	0.41500D−01	69	41,42,43,44,	0.12710D−01	0.80420D−02
39	97,98,101,102	0.70261D−01	0.41500D−01		45,46,47,48		
40	103,105	0.41594D−01	0.41500D−01				
				Structural weight, lb		61.50	40.34
				CPU/cycle, (IBM 370-158), s		42.11	9.77

Table 7 Critical constraints at optimum for closed tail-boom with composite material

Without any damage conditions:	
Stress constraint for members:	30, 12, 66, 48, 84
Lower bound on design variables:	8-10, 12-14, 1-3, 5-7
	44, 48
	22-24, 26-28, 15-17, 9-21
	52, 53, 57
	36-41, 29-31, 33-35
	61, 62, 64, 66
With six damage conditions:	
Stress constraint for members:	Under damage condition:
27,36	1
48	2
66	3
102	4
71	5
Displacement in the y direction of node:	Under damage condition:
25	2, 6
Lower bound on design variables:	12, 13, 5, 6
	43, 44, 48-51
	24, 26, 27, 17, 19, 20
	53, 55, 57
	36-38, 41, 33, 34
	61, 66

Table 8 Damage condition definitions for truss members and skin elements

Damage condition	Member(s) damaged	Node(s) damaged	Reduction in area, %
	Truss members		
1	21, 25, 28, 32, 33, 35, 39, 44, 45	10	100
2	1, 6, 12, 13, 16, 17, 19, 23, 29	7	100
3	58, 63, 65, 69, 70, 72, 76, 82, 84	20	100
4	73, 78, 84, 85, 88, 89, 91, 95, 101	23	100
5	56, 59, 62, 67, 68, 72, 74, 78, 79	17	100
6	3,7, 10, 14, 15, 17, 21, 26, 27	6	100
	Skin elements		
1	7, 8, 11, 12, 21, 25	10	100
2	1, 2, 3, 13, 14, 15	7	100
3	23, 32, 37, 38, 45, 56	20	100
4	33-35, 45-47	23	100
5	20, 28, 33, 34, 41, 42	17	100
6	5-7, 9-11	6	100

The initial design is selected as follows: area of truss members = 0.09 in.^{2}, and thickness for all CST elements = 0.0067 in. The initial weight is 47.76 lb, and the final weight obtained after 25 iterations is 40.34 lb, as given in Table 6. The critical constraints at the optimum for this example are given in Table 7.

Example 3: Damage-Tolerant Design of Composite Helicopter Tail-Boom

In this example, the helicopter tail-boom of example 2 is redesigned to withstand six possible damage conditions. Each damage condition consists of damage to a node of the structure. All members connected to the node are then ineffective in carrying loads. These damage conditions are defined in Table 8.

The initial design weighs 66.63 lb. The final design after 22 iterations weighs 61.5 lb, as given in Table 6. Critical constraints at the optimum are given in Table 7.

Several conclusions can be drawn from the results for examples 2 and 3. Comparing the optimum weights for examples 2 and 3, the same conclusion as drawn earlier in Ref. 11 can be stated: that is, it is necessary to increase the weight in most cases to achieve a damage-tolerant design objective. It has been concluded in Ref. 11 that the closed tail-boom is more efficient in carrying loads than the open tail-boom. By using composite material only for the top and bottom skin elements, the optimum weight obtained in this example (61.50 lb) is further reduced, as compared to the optimum weight (77.21 lb) obtained in Ref. 16, where the same closed tail-boom is optimized without composite material. It should be noted that even more significant weight savings can be achieved when two sides of skin elements are also made of layered composite material, and the fiber directions are oriented in an optimum manner. In such a case, the fiber orientations should also be used as design variables. However, if the fiber directions are also chosen as design variables, then the design sensitivity analysis becomes more involved. Also, because of the ease of fabrication, 0, ± 45, and 90 deg are normally selected as fiber directions. This eliminates the need of including fiber directions as design variables. In the computer program DOCS, the number of layers and their fiber directions must be specified by the designer.

Computational effort when multilayered composite material is used is expected to be more than in the case in which conventional material is used. This is because, with more design variables introduced, more calculations are needed for both the analysis and the design sensitivity.

Discussion and Conclusions

A general finite-element computer program DOCS has been developed for structural synthesis. The program has been used in this paper to solve the composite closed helicopter tail-boom structure, with and without damage considerations, and a framed structure. The code has been implemented on various computer systems such as IBM, CDC, PRIME, and VAX with minimum efforts. This is because all of the READ/WRITE statements for the direct access files are written in a separate subroutine that makes the program portable.

The reliability of the program DOCS has been checked by solving several small-scale design problems in addition to the examples presented in the paper. These additional examples are given in Ref. 17.

For some small-scale examples, the design sensitivity analysis has been checked using closed-form solutions. The analysis part of the program has been tested by comparing solutions with the well-known analysis code SAP4.

A variety of design environments can be handled with DOCS. For example, multilayered composite panels may be used to model the structure; acceleration loads may be specified; effect of projected future damage to the structure can be studied; fundamental frequency constraint can be imposed; various cross-sectional shapes for beam-type members can be used; stress constraints for beam-type elements can be user supplied; substructural formulation may be used; supporting structures can be modeled through the use of an "additional substructures" option; and design of certain parts of the structure can be kept fixed during the optimization process. These options make the program suitable for design of practical structural systems.

The program DOCS (current version) presents an intermediate step in achieving the goal of developing a general design capability. Several extensions of the program are possible. These include expansion of the finite-element library for the program and treatment of general boundary conditions and dynamic loads. These are topics of further research and development for the program DOCS.

References

[1]Arora, J.S. and Haug, E.J., "Efficient Optimal Design of Structures by Generalized Steepest Descent Programming," *International Journal for Numerical Methods in Engineering,* Vol. 10, April 1976, pp. 747-766; Vol. 10, June 1976, pp. 1420-1427.

[2]Gallagher, R.H., and Atrek, E. (eds), *Optimum Structural Design, Proceedings of the International Symposium,* University of Ariz., Tucson, Arizona, Oct. 1981.

[3]Haug, E.J. and Arora, J.S., *Applied Optimal Design,* Wiley-Interscience, New York, 1979.

[4]Arora, J.S. and Nguyen, D.T., "Eigensolution for Large Structural Systems with Substructures," *International Journal for Numerical Methods in Engineering,* Vol. 15, March 1980, pp. 333-341.

[5]Arora, J.S. and Haug, E.J., "Methods of Design Sensitivity Analysis in Structural Optimization," *AIAA Journal,* Vol. 17, Sept. 1979, pp. 970-974.

[6]Haug, E.J. and Arora, J.S., "Design Sensitivity Analysis of Elastic Mechanical Systems," *Computer Methods in Applied Mechanics and Engineering,* Vol. 15, 1978, pp. 35-62.

[7]Arora, J.S. and Govil, A.K., "Design Sensitivity Analysis with Substructuring," *Journal of Engineering Mechanics,* Vol. 103, No. EM4, Aug. 1977, pp. 537-548.

[8]Arora, J.S. and Govil, A.K., "An Efficient Method for Optimal Structural Design by Substructuring," *Computers and Structures,* Vol. 7, No. 4, 1977, pp. 507-515.

[9]Arora, J.S., "An Algorithm for Optimum Structural Design without Line Search," Optimum Structural Design, Vol. II, edited by R.H. Gallagher, K.M. Ragsdell, E. Atrek, and O.C. Zienkiewicz, Wiley, New York, 1983.

[10]Belsare, S.V. and Arora, J.S., "An Algorithm for Engineering Design Optimization," *International Journal for Numerical Methods in Engineering* (to be published), 1983.

[11]Nguyen, D.T. and Arora, J.S., "Fail-Safe Optimal Design of Complex Structures with Substructures," *Journal of Mechanical Design,* Vol. 104, Oct. 1982, pp. 861-868.

[12]"Manual of Steel Construction," 7th ed., American Institute of Steel Construction, Chicago, Ill., 1973.

[13]"Specification for Aluminum Structures," Sec. 1, 3rd ed., Aluminum Association, Inc., 1976.

[14]Arora, J.S. and Ryu, Y.S., "User's Manual for DOCS," Division of Materials Engineering, The University of Iowa, Iowa City, Iowa, Tech. Rept. CAD-SS-82.1, Jan. 1982.

[15]Arora, J.S. and Haririan, M., "Use of Response Gradients in Structural Design," Division of Materials Engineering, The University of Iowa, Iowa City, Iowa, Tech. Rept. CAD-SS-81.5, Nov. 1981.

[16]Nguyen, D.T. and Arora, J.S., "Fail-Safe Optimal Design of Complex Structures with Substructures and Composite Materials," Division of Materials Engineering, The University of Iowa, Iowa City, Iowa, Tech. Rept. CAD-SS-82.3, Feb. 1982.

[17]Nguyen, D.T., Belegundu, A.D., and Arora, J.S., "Sample Solutions with the Computer Program DOCS," Division of Materials Engineering, The University of Iowa, Iowa City, Iowa, Tech. Rept. CAD-SS-82.9, July 1982.