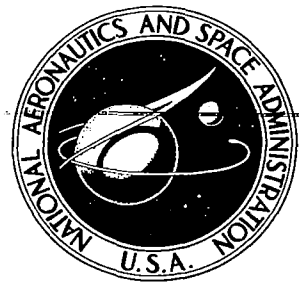
NASA CONTRACTOR REPORT

NASA CR-2797



NASA CR-27

DESAP 2 - A STRUCTURAL DESIGN PROGRAM WITH STRESS AND BUCKLING CONSTRAINTS

Volume I: Theoretical and User's Manual

J. Kiusalaas and G. B. Reddy

Prepared by
THE PENNSYLVANIA STATE UNIVERSITY
University Park, Pa. 16802
for George C. Marshall Space Flight Center

NATIONAL AERONAUTICS AND SPACE ADMINISTRATION • WASHINGTON, D. C. • MARCH 1977

1. REPORT NO.	2. GOVERNMENT ACCESSION NO.	3. RECIPIENT'S CATALOG NO.
NASA CR-2797		

4. TITLE AND SUBTITLE	5. REPORT DATE
DESAP 2 - A Structural Design Program with Stress and Buckling Constraints. Vol. I: Theoretical and User's Manual	March 1977
	6. PERFORMING ORGANIZATION CODE
7. AUTHOR(S) J. Kiusalaas and G. B. Reddy	8. PERFORMING ORGANIZATION REPORT # M-205
9. PERFORMING ORGANIZATION NAME AND ADDRESS Department of Engineering Science and Mechanics The Pennsylvania State University University Park, Pennsylvania 16802	10. WORK UNIT NO.
	11. CONTRACT OR GRANT NO. NCA-8-71/72
12. SPONSORING AGENCY NAME AND ADDRESS National Aeronautics and Space Administration Washington, D. C. 20546	13. TYPE OF REPORT & PERIOD COVERED Contractor Report Final
	14. SPONSORING AGENCY CODE

15. SUPPLEMENTARY NOTES

This work was done under the sponsorship of the Structures and Propulsion Laboratory of the Marshall Space Flight Center, Alabama.

16. ABSTRACT

DESAP 2 is a finite element program for computer-automated, minimum weight design of elastic structures with constraints on stresses (including local instability criteria) and buckling loads. No limits are placed on the number of load conditions for stress-constrained design, but only one of these load conditions can be chosen as the potential buckling load. A substantial portion of DESAP 2, particularly the analysis of the prebuckling state, is derived from the SOLID SAP finite element program developed at the University of California, Berkeley. The stress-constrained design is based on the classical stress ratio method, which drives the design towards a fully stressed state. The constraints on the buckling load are handled by solving the appropriate optimality criterion by successive iterations. During each iteration, the element sizes determined by the stress ratio method are used as the minimum size constraints. The element subroutines have been organized in a manner that permits the user to make additions and changes with a minimal programming effort. Consequently, DESAP 2 can readily be changed into a special-purpose program to handle the user's specific design requirements and failure criteria.

DESAP 2 is a companion program of DESAP 1: "A Structural Design Program with Stress and Displacement Constraints." With the exception of a few cards, the same input data deck can be used for both programs.

This is Volume I of three volumes.

17. KEY WORDS	18. DISTRIBUTION STATEMENT
	STAR Category 39

19. SECURITY CLASSIF. (of this report)	20. SECURITY CLASSIF. (of this page)	21. NO. OF PAGES	22. PRICE
Unclassified	Unclassified	133	$6.00

* For sale by the National Technical Information Service, Springfield, Virginia 22161

ACKNOWLEDGEMENTS

The development of the program was sponsored by the National Aeronautics and Space Administration, George C. Marshall Space Flight Center, Huntsville, Alabama, under cooperative Agreement NCA-8-71/72 with the Pennsylvania State University, University Park, Pennsylvania. The technical monitor of the work was J. E. Key.

The authors would also like to acknowledge the contributions of M. Isreb, who assisted in the development of the shear panel element and the stress-constrained redesign subroutines.

TABLE OF CONTENTS

VOLUME 1

A. INTRODUCTION

DESAP 2 is a finite element program for automated design (synthesis) of linear-elastic structures under static loads. The design objective is to find the element sizes (cross-sectional areas, plate thicknesses, etc.) that minimize the total structural weight. The layout of the structure is not changed during the design procedure.

The primary constraints used in the synthesis algorithm are upper limits on stresses and lower bounds on buckling loads. The stress limits may be prescribed in the form of yield criteria, local instability criteria, or both. There are no restrictions on the number of load conditions that may be imposed on the structure for the stress-constrained design. Considerations of economy, however, limit the number of load conditions for buckling-constrained design to one.

The program also allows the use of secondary constraints, which consist of minimum allowable element sizes and size proportion constraints (the requirement that the sizes of specified elements be equal, or have a prescribed ratio).

The method of design is very similar to that used for stress and displacement constraints in DESAP 1[1]; in fact, many parts of DESAP 1 and DESAP 2 are identical. The design procedure is an iterative process, each iteration consisting of four parts:

1) Prebuckling analysis of the current design.
2) Redesign with respect to stress constraints based on the results of prebuckling analysis.

3) Buckling analysis of the current design under the action of the internal forces obtained in part 1).

4) Redesign with respect to buckling; the sizes of elements determined in part 2) are used as the minimum size constraints.

The entire computer program is built around the SOLID SAP finite element program developed by E. L. Wilson[2]. It was necessary, of course, to carry out extensive modifications of the existing SOLID SAP subroutines in order to accomodate the special requirements of the redesign operations. Apart from the modifications, several major additions to the program were made. These included the entire buckling analysis package, all the redesign subroutines, and the subroutines for the shear panel element.

The classical stress ratio method is employed in the redesign with respect to stress constraints. This procedure will drive the final design to the fully stressed design, which does not necessarily coincide with the minimum weight distribution of the material. Unfortunately, at the present state-of-the-art, more rigorous design methods are entirely impractical for structures of reasonable size, since they require the inversion of the structural stiffness matrix, or the use of numerical search techniques.

For the constraints on the general buckling loads, the redesign procedure described in [3,4] is used. The redesign formulas are derived directly from the optimality criterion; consequently, the element sizes are driven towards the minimum weight design.

The design procedure for stress as well as the buckling constraints is based on the assumption that the loading is independent of the

element sizes (dead loading of prescribed magnitude). Allowance has been made for size-dependent loads, e.g., thermal stresses and gravity loading, but these loads must be used with special precautions.

As pointed out repeatedly in existing literature, neither the fully stressed design, nor the optimal weight design are necessarily unique, i.e., they have a local rather than a global character. It follows, therefore, that the choice of the initial design plays an important role in determining the design to which the synthesis algorithm converges. Numerous examples seem to indicate, however, that the weight differences between the various converged designs are slight, although there may be large differences in the distribution of the material.

A very important feature of the program is the organization of the element subroutines such that they can easily be adapted to the user's special requirements with the smallest possible programming changes. In making this provision, we recognized the fact that it is virtually impossible to create an all-purpose synthesis program. The reason is that structural design requires a much larger and more varied volume of input information than analysis. In the case of beam elements, for example, the cross-sectional area, the two moments of inertia and the torsional constant suffice for the purposes of analysis. In design we must add the section moduli, local buckling information and the yield criterion; in addition we must specify how all these properties vary with the design variable. Since each design situation may use elements of special construction and shape, and different design criteria, it is clearly impractical to make provisions for all the possible forms of design data in a single program.

Finally, it must be pointed out that DESAP 2, despite its flexibility, can still handle only a limited amount of design information. Consequently, the results of the program are to be taken as a preliminary design in the sense that it gives the desired proportions of the structure, but detailing must be still carried out by the designer through conventional design practices.

B. THEORETICAL BACKGROUND

B.1 Element Properties

The size of a typical (i^{th}) element is denoted by A_i. It represents the cross-sectional area or the panel thickness for one and two-dimensional elements, respectively. The weight of the element can be written as

$$W_i = \rho_i A_i , \qquad i = 1, 2, \ldots, I \tag{B.1.1}$$

where ρ_i is the unit weight of the element. If the minimum cost, rather than weight is the design objective, then ρ_i should be taken as the unit cost.

It is seldom desirable to have each A_i as an independent design variable. Equal size constraints can be imposed by introducing the design variables D_m, $m = 1, 2, \ldots, M$, where $M \leq I$, which are independently variable, and expressing each element size in the form

$$A_i = \eta_i D_m , \tag{B.1.2}$$

where η_i is called the design variable fraction of the element. Both η_i and m must be specified for each element. Equal sizing of two elements is obtained, for example, by prescribing the same m for each element together with $\eta_i = 1$. In addition, the scheme permits proportional sizing if the same m, but different values of η_i are used.

The stiffness matrix of each element is restricted to the form

$$[K_i] = [k_i^{(1)}]A_i + [k_i^{(2)}]A_i^{n_i} \tag{B.1.3}$$

where $[K_i]$ is the element stiffness matrix, $[k_i^{(1)}]$ and $[k_i^{(2)}]$ are the unit stiffness matrices of the element, and n_i is the inertia exponent. The first part of (B.1.3) represents the action of direct stresses, whereas the second part is due to bending or torsion. The value of n_i depends on the physical nature of the design variable. For example, $n_i = 1$ for thin-walled beams if the wall thickness only is being varied. If all dimensions of the cross section are scaled uniformly, then $n_i = 2$. For plates, where the thickness is subject to design, we have $n_i = 3$. The unit stiffness matrices of each element are stored separately on an external storage device, together with m, η_i and n_i.

It should be noted that n_i is determined by the relationship between the size and the moment of the inertia of the element:

$$I_i = j_i A_i^{n_i} \quad , \tag{B.1.4}$$

where j_i is the unit moment of inertia.

The vector of internal forces $\{N_i\}$ of an element is recovered from the element nodal displacements $\{u_i\}$ by the formula

$$\{N_i\} = [S_i]\{u_i\} + \{T_i\} \quad . \tag{B.1.5}$$

If the element stiffness matrix is given by (B.1.3), then the force recovery matrix $[S_i]$ has the same form:

$$[S_i] = [s_i^{(1)}]A_i + [s_i^{(2)}]A_i^{n_i} \quad , \tag{B.1.6}$$

where $[s_i^{(1)}]$ and $[s_i^{(2)}]$ are the unit force recovery matrices. The force vector $\{T_i\}$ also consists of two parts:

$$\{T_i\} = \{t_i^{(1)}\}A_i + \{t_i^{(0)}\} \quad . \tag{B.1.7}$$

$\{t_i^{(1)}\}$ being the unit thermal force vector and $\{t_i^{(0)}\}$ represents the contribution of size-independent loading; e.g., fixed-end forces of beam elements due to dead loading. Again, $[s_i^{(1)}]$, $[s_i^{(2)}]$, $\{t_i^{(1)}\}$ and $\{t_i^{(0)}\}$ are stored for each element.

The element load vector $\{Q_i\}$ is similarly separated into two components:

$$\{Q_i\} = \{q_i^{(1)}\}A_i + \{q_i^{(0)}\} , \qquad (B.1.8)$$

where $\{q_i^{(1)}\}$ is the unit load vector due to size-dependent loading (thermal and gravity loads), and $\{q_i^{(0)}\}$ represents the contribution of dead loads; both vectors are stored for each element.

The form of equations (B.1.7) and (B.1.8) allows only for a uniform temperature increase within an element, i.e., it assumes that thermal expansion causes extension without bending. This excludes, for example, the effects of thermal gradients through the thickness of plates, for which we would require additional terms of the type $\{t_i^{(2)}\}A_i^{n_i}$ and $\{q_i^{(2)}\}A_i^{n_i}$ for $\{T_i\}$ and $\{Q_i\}$, respectively.

Up to now the discussion has been confined to the properties of the elements associated with linear analysis, i.e. the analysis of the pre-buckling state. In the analysis of general buckling, the elastic stiffness matrix $[K_i]$ of each element must be supplemented by its geometric stiffness matrix $[G_i]$. The latter represents the contribution of the nonlinear terms in the strain-displacement relations to the strain energy of the element, the linear terms being accounted for by $[K_i]$.

The buckling analysis of DESAP 2 is limited to bifurcation instability (classical, Euler-type buckling), in which case only the quadratic terms in the components of the rotation vector $\{\omega\}$ are included in the formation of $[G_i]$. The contribution of these terms to the strain energy of an element is---see Ref. [5], p. 7.1-6.

$$U_G = \frac{1}{2}\int_V [(\sigma_x^0+\sigma_y^0)\omega_z^2 + (\sigma_y^0+\sigma_z^0)\omega_x^2 + (\sigma_z^0+\sigma_x^0)\omega_y^2 - 2\tau_{xy}^0\omega_x\omega_y - 2\tau_{yz}^0\omega_y\omega_z - 2\tau_{zx}^0\omega_z\omega_x]dV, \tag{B.1.9}$$

where the superscript "0" denotes that the stresses are evaluated in the prebuckling state, and V refers to the volume of the element. The rotations are given in terms of the buckling displacements u,v,w by

$$\omega_x = \frac{1}{2}(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}),\ \omega_y = \frac{1}{2}(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}),\ \omega_z = \frac{1}{2}(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}). \tag{B.1.10}$$

For a plate or a plane stress element lying in the x-y plane, $\sigma_z^0 = \tau_{yz}^0 = \tau_{zx}^0 = 0$. In addition, Kirchoff's hypothesis (normals remain normal to the middle surface) is equivalent to setting $\frac{\partial v}{\partial z} = -\frac{\partial w}{\partial y}$ and $\frac{\partial u}{\partial z} = -\frac{\partial w}{\partial x}$, so that (B.1.9) becomes

$$U_G = \frac{1}{2}\int_A \{N_x^0[\omega_z^2+(\frac{\partial w}{\partial x})^2] + N_y^0[\omega_z^2+(\frac{\partial w}{\partial y})^2] + 2N_{xy}^0\frac{\partial w}{\partial x}\frac{\partial w}{\partial y}\}dA, \tag{B.1.11}$$

where N_x^0, N_y^0 and N_{xy}^0 are the membrane stress resultants acting in the prebuckling state, and A is the middle surface area of the element.

In DESAP 2 the stress resultants are taken as constant within each element (the resultants acting at the center of the element are used); consequently they can be taken outside the integral sign in (B.1.11).

The geometric stiffness matrix $[G_i]$ of an element is defined as

$$U_G = \frac{1}{2}\{u_i\}^T[G_i]\{u_i\} \ , \tag{B.1.12}$$

where $\{u_i\}$ is the nodal displacement vector of the element associated with buckling deformation. Comparing (B.1.12) with (B.1.11), we conclude that the geometric stiffness matrix has the form

$$[G_i] = [g_i^{(1)}]N_x^0 + [g_i^{(2)}]N_y^0 + [g_i^{(3)}]N_{xy}^0 \ . \tag{B.1.13}$$

The <u>unit geometric stiffness matrices</u> $[g_i^{(k)}]$ of the element can be calculated from

$$\begin{aligned}
\{u_i\}^T[g_i^{(1)}]\{u_i\} &= \int_A [\omega_z^2 + (\frac{\partial w}{\partial x})^2]dA \ , \\
\{u_i\}^T[g_i^{(2)}]\{u_i\} &= \int_A [\omega_z^2 + (\frac{\partial w}{\partial y})^2]dA \ , \\
\{u_i\}^T[g_i^{(3)}]\{u_i\} &= 2\int_A \frac{\partial w}{\partial x}\frac{\partial w}{\partial x}\,dA \ .
\end{aligned} \tag{B.1.14}$$

It is conventional to neglect ω_z for plate elements, but all the terms in (B.1.14) are included for the plane stress element.

In the case of one-dimensional elements parallel to the x-axis, we use

$$U_G = \frac{1}{2}\int_L N^0[(\frac{\partial v}{\partial x})^2 + (\frac{\partial w}{\partial x})^2]dx \ , \tag{B.1.15}$$

where N^0 is the axial tensile force prior to buckling, and L denotes the length of the element. The axial force is again taken to be constant throughout L, which enables us to write

$$[G_i] = [g_i^{(1)}]N^0 \ , \tag{B.1.16}$$

where the unit stiffness matrix is obtained from

$$\{u_i\}^T[g_i^{(1)}]\{u_i\} = \int_L [(\frac{\partial v}{\partial x})^2 + (\frac{\partial w}{\partial x})^2]dx \ . \tag{B.1.17}$$

Equation (B.1.15) can be derived from (B.1.11) only by assuming that the element does not undergo torsion in the course of buckling. As a result, DESAP 2 is not capable of handling torsional or lateral-torsional buckling of beams. The geometric stiffness matrix for the latter is very complex---see Ref. [5], p. 7.2-8 -- and does not readily lend itself to the redesign process.

The unit geometric stiffness of each element is calculated in DESAP 2 as soon as the element data is read in, and is stored on an auxiliary storage device. Prior to each buckling analysis, the unit matrices are read back into the core, and the geometric stiffness matrices of the elements are reconstituted according to (B.1.13) or (B.1.6).

B.2 Stress Constraints

At the present time, the only practical means of handling stress constraints for large structures seems to be the concept of fully stressed design (FSD). In a fully stressed structure each element reaches its maximum permissible stress level under at least one load condition, unless it is governed by the minimum size constraint. FSD does not generally coincide with the minimum weight design, except for statically determinate structures, but the differences in weight are small in most cases.

A typical failure criterion of an element can be expressed in the general form

$$f(\{N_i\},\ \{N_i^*\},A_i) = 1\ , \tag{B.2.1}$$

where $\{N_i\}$ is the internal force vector of the element and $\{N_i^*\}$ contains the allowable forces. Equation (B.2.1) may represent a criterion for any kind of failure, such as yielding, fracture or local instability.

Let A_i be the element size for the current design, and A_i' the size predicted for next (improved) design. The corresponding internal forces are denoted by $\{N_i\}$ and $\{N_i'\}$, respectively. For the sake of clarity, we limit the discussion at this time to a single load condition, and assume that each element is subject to a single failure criterion.

If the improved design is to be fully stressed, it must satisfy

$$f(\{N_i'\},\ \{N_i^*\},A_i') = 1\ . \tag{B.2.2}$$

The main difficulty in using (B.2.2) is that it requires a knowledge of $\{N_i'\}$, i.e. the changes in the nodal forces of each element caused by the

redesign. This information can be acquired only by an inversion of the structural stiffness matrix, which is impractical for large structures, since the banded form of the structural stiffness matrix is lost upon its inversion.

The problem is greatly simplified if we assume that the nodal forces do not change upon redesign. Equation (B.2.2) then becomes

$$f(\{N_i\}, \{N_i^*\}, A_i') = 1 \ , \qquad \text{(B.2.3)}$$

which can immediately be solved for A_i', one element at a time. We have now arrived at the stress ratio method of redesign, which is adopted for DESAP 1.

The assumption of no change in the nodal forces is valid only for statically determinate structures under dead (size-independent) loading, in which case a single redesign will result in FSD. For all other problems, equation (B.2.3) is an approximation, and it must be applied iteratively, updating $\{N_i\}$ each time, before FSD is reached. If size-dependent loading is present (e.g., gravity or thermal loads), the approximation may be poor and cause difficulties of convergence.

When several load conditions and design criteria are used, the element size is calculated for each combination of loading and design formula, and the largest value is chosen as A_i'.

Once the new size of the element is known, the corresponding design variable is determined by (B.1.2): $D_m' = A_i'/\eta_i$, or the minimum size constraint: $D_m' = D_m^*$ (D_m^* is the prescribed lower limit), whichever is larger. If the design variable D_m is common to several elements,

i.e., if equal or proportional size constraints are used, the maximum value of D_m' is used; that is

$$D_m' = \max_{i \varepsilon m} A_i'/\eta_i \quad \text{or} \quad D_m' = D_m^* , \tag{B.2.4}$$

whichever is greater. The notation $i \varepsilon m$ indicates that maximum is obtained from all elements i that share the design variable number m.

The ratio

$$R_m = D_m'/D_m$$

is called the stress ratio of the design variable. In order to monitor the progress of the design sequence, the maximum and minimum stress ratios of the structure,

$$R_{max} = \max_m R_m \quad \text{and} \quad R_{min} = \min_m R_m , \tag{B.2.6}$$

are computed after each redesign cycle. The new design $\{D'\}$ is said to be a stress-critical design if

$$1 - \delta \leq R_{max} \leq 1 + \delta , \tag{B.2.7}$$

where δ is a small parameter prescribed by the user. Similarily the design is considered to be fully stressed if, in addition to being critical, it also satisfies

$$1 - \delta \leq R_{min} \leq 1 + \delta . \tag{B.2.8}$$

The values of R_{max}, R_{min} and the corresponding design variable and load numbers are printed out after each redesign cycle.

B.3 Buckling Constraints

Buckling analysis is a very expensive operation that consumes several times more computer time than analysis of the prebuckling state. Moreover, the cost of buckling-constrained design is proportional to the number of load conditions used, since each load condition results in a different geometric stiffness matrix, thus requiring separate analysis.

In order to keep the computer costs within reasonable limits, DESAP 2 allows only one load condition to be used for buckling constraints. This restriction does not apply to stress constraints, where the number of load conditions is limited only by the available core storage.

Constraints on general buckling loads are handled in DESAP 2 by the technique described in References [3,4]. The procedure is not only applicable to buckling loads, but can also be used for displacement constraints as was done in DESAP 1 [1].

In the analysis of buckling, the loads applied to the structure are taken to be proportional to a single load parameter p. Denoting the critical values of p by p_r, $p_1 \leq p_2 \leq \ldots$, the constraints on the buckling loads are

$$p_r \geq p^*, \qquad r = 1,2,\ldots, R, \tag{B.3.1}$$

where p* is the prescribed lower bound and R equals the number of degrees of freedom in the finite element model. The load vector of the

structure at buckling is thus given by

$$\{Q^*\} = p^*\{Q\} \ , \tag{B.3.2}$$

where $\{Q\}$ is the load vector used in prebuckling analysis. If $\{Q\}$ represents the design load with respect to stress constraints, then p^* can be interpreted as the additional factor of safety against general buckling.

The constraints (B.3.1) can be divided into two categories: if the equality sign governs the optimal design, the constraint is said to be active; if the inequality occurs, the constraint is passive. This division is not known, of course, until the final design is reached, but for notational convenience we presume that the active constraints are listed first, so that (B.3.1) can be replaced by

$$\begin{aligned} p_r &= p_r^* \ , \qquad r = 1,2,\ldots, R_{act} \\ p_r &> p_r^* \ , \qquad r = R_{act} + 1,\ldots, R \ , \end{aligned} \tag{B.3.3}$$

where R_{act} denotes the number of active buckling constraints. DESAP 2 allows for $R_{act} \leq 2$, i.e. it assumes that number of active buckling modes never exceeds two. Theoretically it is possible for the optimal design to be governed by three or more modes (all having the same buckling load), but such cases have not yet been encountered.

In addition to buckling constraints, we must also account for limits on element sizes:

$$A_i \geq A_i^*, \quad \text{i.e., } D_m \geq D_m^* \ . \tag{B.3.4}$$

A passive design variable is governed by the minimum size constraint in the final design, i.e., $D_m = D_m^*$, whereas an active design variable is determined by the displacement constraint, in which case $D_m' > D_m^*$.

In the mathematical treatment that follows, it is convenient to replace (B.3.1) and (B.3.4) by the equality constraints

$$p_r - a_r^2 = p_r^* , \qquad D_m - b_m^2 = D_m^* , \tag{B.3.5}$$

where a_r and b_m are to be viewed as variables free from constraints.

The design objective is to minimize the total weight of the structure $W = \sum_i W_i$. Substituting for W_i from (B.1.1) and (B.1.2), we have

$$W = \sum_i \rho_i \eta_i D_m . \tag{B.3.6}$$

Minimizing (B.3.6) subject to (B.3.5) is equivalent to making the following function stationary:

$$V = \sum_i \rho_i \eta_i D_m - \sum_r \lambda_r (p_r - a_r^2) - \sum_m \mu_m (D_m - b_m^2) , \tag{B.3.7}$$

where λ_r and μ_m are non-negative Lagrangian multipliers. The operations $\partial V/\partial D_m = 0$, $\partial V/\partial a_r = 0$ and $\partial V/\partial b_m = 0$ yield, respectively,

$$- \sum_{i \in m} \rho_i \eta_i - \sum_r \lambda_r p_{r,m} - \mu_m D_m = 0 , \tag{B.3.8}$$

$$\lambda_r a_r = 0 , \tag{B.3.9}$$

$$\mu_m b_m = 0 , \tag{B.3.10}$$

where we used the notation $(\ )_{,m} = \partial(\ )/\partial D_m$.

Inspection of (B.3.5), (B.3.9) and (B.3.10)reveals that

$$\lambda_r \begin{cases} \geq 0 & \text{if } p_r = p_r^* \quad \text{(active constraints)} \\ = 0 & \text{if } p_r > p_r^* \quad \text{(passive constraints)}, \end{cases}$$

$$\mu_m \begin{cases} \geq 0 & \text{if } D_m = D_m^* \quad \text{(passive design variables)} \\ = 0 & \text{if } D_m > D_m^* \quad \text{(active design variables)}, \end{cases}$$

which enables us to rewrite the optimality criterion (B.3.8) as

$$\sum_{i\epsilon m} \rho_i \eta_i - \sum_{r\ act} \lambda_r p_{r,m} \begin{cases} \geq 0 & \text{if } D_m = D_m^* \\ = 0 & \text{if } D_m > D_m^* \end{cases} . \qquad \text{(B.3.11)}$$

The notation r act shows that the sum is to be taken over the active displacement constraints. Introducing the <u>unit weight</u> of the m^{th} design variable

$$\rho_m = \sum_{i\epsilon m} \rho_i \eta_i , \qquad \text{(B.3.12)}$$

(B.3.11) becomes for the <u>active design variables</u>

$$\frac{1}{\rho_m} \sum_{r=1}^{R\ act} \lambda_r p_{r,m} = 1 \qquad \text{(B.3.13)}$$

The redesign formula for the active design variables is obtained directly from the optimality criterion. Multiplying both sides of (B.3.13) by $(1-\alpha)D_m$, where α is a constant to be determined later, and rearranging terms, we get

$$D_m = \alpha D_m + (1-\alpha)\frac{D_m}{\rho_m} \sum_{r=1}^{R\ act} \lambda_r p_{r,m} .$$

The function of the design algorithm is to solve the last equation by successive iterations. The redesign formula used in each iterative step is

$$D_m' = \alpha D_m + (1-\alpha) \frac{D_m}{\rho_m} \sum_{r=1}^{Ract} \lambda_r p_{r,m} , \qquad (B.3.14)$$

where D_m is the current design variable, and D_m' represents its improved value. The constant α can now be recognized as the relaxation factor, which is utilized to control the convergence characteristics of the iterative procedure.

Each iterative design, i.e. each application of (B.3.14) is followed by an analysis of resulting structure, so that λ_r and $p_{r,m}$ can be updated. The computation of these parameters will be discussed next.

Buckling is governed by the matrix characteristic value problem

$$[K]\{u\} = p[G]\{u\} , \qquad (B.3.15)$$

where [K] and [G] are the elastic and geometric stiffness matrices of the structure, respectively, and {u} is the nodal displacement vector due to buckling. The gradients of the buckling parameters can be shown to be [3,6]

$$p_{r,m} = \frac{\{u^{(r)}\}^T \left([K_{,m}] - p_r [G_{,m}] \right) \{u^{(r)}\}}{\{u^{(r)}\}^T [G] \{u^{(r)}\}} , \qquad (B.3.16)$$

where $\{u^{(r)}\}$ is the nodal displacement vector associated with p_r.

It is assumed, as was done in the stress-constrained design, that the changes in the internal forces between two successive redesign

cycles are negligible, in which case we can set $[G_{,m}] = [0]$. In addition, the buckling modes are normalized with respect to [G], so that (B.3.16) becomes

$$p_{r,m} = \{u^{(r)}\}^T [K_{,m}] \{u^{(r)}\} \quad . \tag{B.3.17}$$

Noting that only the elements that share the design variable D_m contribute to $[K_{,m}]$, (B.3.17) can be rewritten as

$$p_{r,m} = \sum_{i \in m} \{u_i^{(r)}\}^T [K_{i,m}] \{u_i^{(r)}\} \quad . \tag{B.3.18}$$

Utilizing (B.1.2) and (B.1.3), the derivatives of the element stiffness matrices in (B.3.18) are

$$[K_{i,m}] = \frac{\partial [K_i]}{\partial D_m} = \eta_i ([k_i^{(1)}] + n_i A_i^{n_i - 1} [k_i^{(2)}]) \quad . \tag{B.3.19}$$

Since the unit stiffness matrices of each element are computed and stored at the beginning of the program (they do not change upon redesign), the gradients of the buckling load parameters are readily calculated once the buckling modes of the current design are available.

It must be reiterated that (B.3.18) is valid only when the internal forces prior to buckling are independent of the design variables. This condition is satisfied only if the prebuckling state is statically determinate and size-dependent loads (gravity loading and thermal stresses) are absent.

If the prebuckling state is statically indeterminate, (B.3.18) is only an approximation, which would not necessarily lead to a true

optimal design. The use of the equation is however, compatible with the stress ratio method of redesign, where the changes of internal forces were also neglected. Note that the use of the exact gradient expressions (B.3.16) is precluded for the reason stated in Sec. B.2: it would require the knowledge of $\{N_{i,m}\}$, the gradients of the internal force vector of each element.

If the prebuckling state of the structure is statically determinate and if the loading is not size-dependent, the user of DESAP 2 should specify this in the input data. Since the geometric stiffness will be unchanged during redesign, the program will compute [G] only once, namely after the first prebuckling analysis. Otherwise [G] will be recomputed in each redesign cycle from the unit geometric stiffness matrices of the elements.

The Lagrangian multipliers are chosen such as to make the improved design buckling-critical, i.e., they are calculated from the condition $p'_r = p^*_r$, $r = 1, 2, \ldots, R_{act}$. Using the notation $\delta p_r = p'_r - p_r$ and $\delta D_m = D'_m - D_m$, the change in the nodal displacements can be estimated from the linear approximation

$$\delta p_r = \sum_{m=1}^{M} p_{r,m} \delta D_m \ . \qquad \text{(B.3.20)}$$

Passive design variables are governed by the minimum size constraint after redesign. Hence, $\delta D_m = D^*_m - D_m$, making their contribution to (B.3.20)

$$(\delta p_r)_{pass} = \sum_{m\ pass}^{M} p_{r,m}(D^*_m - D_m) \ . \qquad \text{(B.3.21)}$$

The improved values of active elements are given by (B.3.14), which results in

$$(\delta p_r)_{act} = -(1-\alpha) \sum_{m\ act}^{M} p_{r,m} D_m \left(1 - \frac{1}{\rho_m} \sum_{s=1}^{R_{act}} \lambda_s p_{s,m}\right) \quad . \quad (B.3.22)$$

The total change in p_r is therefore,

$$\delta p_r = (\delta p_r)_{pass} + (\delta p_r)_{act} \quad . \quad (B.3.23)$$

Setting $\delta p_r = p_r^* - p_r$, and substituting (B.3.21) and (B.3.22) in (B.3.23), we obtain the following simultaneous equations for the Lagrangian multipliers λ_s, $s = 1,2,\ldots, R_{act}$:

$$(1-\alpha) \sum_{s=1}^{R_{act}} \lambda_s \sum_{m\ act}^{M} \frac{p_{s,m} p_{r,m}}{\rho_m} D_m \quad (B.3.24)$$

$$= (1-\alpha) \sum_{m\ act}^{M} p_{r,m} D_m - \sum_{m\ pass}^{M} p_{r,m} (D_m^* - D_m) + p_r^* - p_r \quad ,$$

$$r = 1,2, \ldots, R_{act} \quad .$$

The solution of (B.3.24) requires the prior knowledge of the number of active buckling constraints R_{act}, and the active-passive identities of the design variables. As this information is generally not available, the redesign is carried out by an iterative procedure diagramed in Fig. B.3.1. As noted previously, a maximum of two buckling modes are assumed to be active at any stage of the design procedure, i.e., we take $R_{act} \leq 2$.

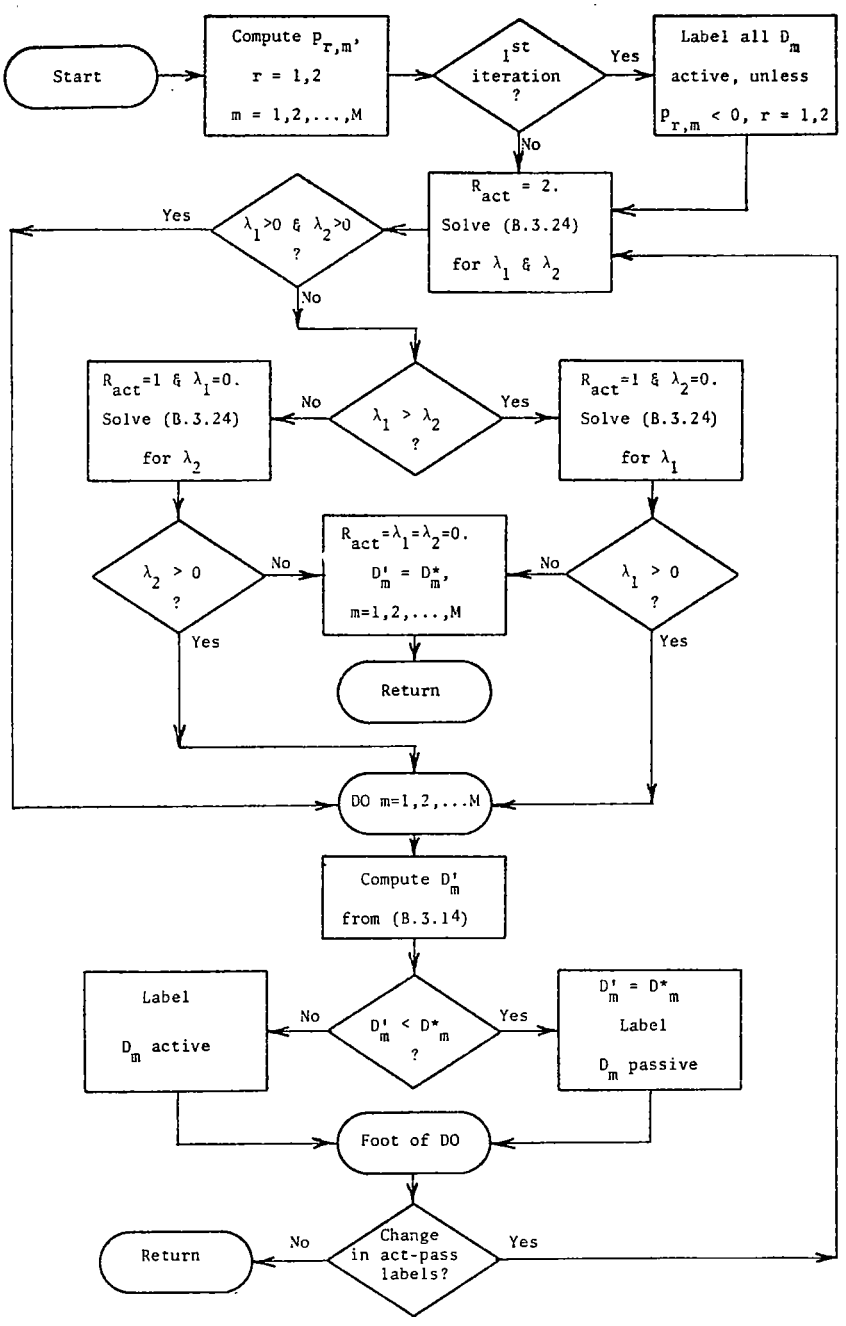


Figure B.3.1

Flow Diagram for One Redesign Cycle.
(Buckling Constraints)

The flow diagram in Fig. B.3.1 is somewhat simplified, as it omits the paths that are followed if the equations for λ_1 and λ_2 are singular or have a small determinant. In general, if $\lambda \to \pm \infty$ is indicated by the equations, the corresponding buckling constraint is taken as passive, i.e. λ_r is treated in the same manner as if it were negative.

After each buckling analysis, but before buckling-constrained redesign, the buckling ratios of the current design

$$Q_r = p^*/p_r \ , \tag{B.3.25}$$

are computed and printed. A buckling mode (r^{th} mode) is considered to be potentially active only if

$$Q_r \geq \omega R_{max} \ , \tag{B.3.26}$$

where $\omega < 1$ is a user-supplied constant and R_{max} is the maximum stress ratio defined in (B.2.6). If the above inequality is violated, the corresponding mode is ignored in the subsequent buckling-constrained redesign operation.

A design is said to be buckling-critical if

$$1 - \delta \leq Q_{max} = \max_r Q_r \leq 1 + \delta \ , \tag{B.3.27}$$

where δ is a small parameter that was also used in defining a stress-critical design --- see (B.2.7). The design is considered acceptable

if one of the following conditions are met:

1) All design variables are passive and $Q_{max} \leq 1 + \delta$.

2) The design is buckling-critical and the optimality criterion (B.3.13) is satisfied within a prescribed latitude:

$$1 - 5\delta \leq \frac{1}{\rho_m} \sum_{r=1}^{R_{act}} \lambda_r p_{r,m} \leq 1 + 5\delta \qquad \text{(B.3.28)}$$

for all active elements. Experience with the program has shown that the "latitude" in (B.3.28) should be considerably less stringent than in (B.3.27); hence the use of 5δ in the last inequality.

The quantity

$$\frac{1}{\rho_m} \sum_{r=1}^{R_{act}} \lambda_r p_{r,m} ,$$

which we call the optimality index of the design variable D_m, is printed for each design variable after every redesign cycle, together with its active-passive classification.

B.4 Stress and Buckling Constraints

If both stress and buckling constraints are present, each redesign cycle is divided into two parts. First, the structure is redesigned with respect to the stresses only; the buckling constraints are ignored in this phase of design. The next step is the buckling-constrained design, where the design variables just obtained from the stress-constrained phase are used as the minimum size constraints.

The buckling-constrained redesign phase will be skipped if the analysis of the current design shows that

$$Q_{max} \leq \omega R_{max} , \tag{B.4.1}$$

where R_{max} and Q_{max} are the maximum stress and buckling ratios defined in (B.2.6) and (B.3.27), respectively. The constant $\omega < 1$ is also used in conjunction with (B.3.26) in choosing the potentially active buckling constraints.

B.5 Uniform Scaling Operation

We recall that the redesign equations for both, stress and buckling constraints, were based on certain simplifying approximations. The stress ratio redesign formula (B.2.3) and the gradients of the buckling parameters (B.3.18) were both derived under the assumption that the internal forces are unchanged during a redesign cycle. In addition, the predicted changes in the buckling parameters (B.3.20) were linearized. As a consequence of these approximations, the redesign process becomes an iterative procedure consisting of repeated applications of the redesign formulas until convergence is achieved.

It is frequently advantageous to interrupt the design procedure by the so-called uniform scaling operation, where all the design variables are changed by the same scale factor μ:

$$D'_m = \mu D_m \ . \tag{B.5.1}$$

The scale factor is calculated from the condition that the scaled design $\{D'\}$ should be critical, i.e.

$$1 - \delta \leq \max(R_{max}, Q_{max}) \leq 1 + \delta. \tag{B.5.2}$$

DESAP 2 gives the user the option of using the uniform scaling operation whenever the current design is not critical. Once the design has been scaled to the critical state (more than one scaling operation may be required to achieve this), the redesign equations will be applied in the usual manner.

This method of design offers two advantages over the use of the redesign equations above. Firstly, the use of the scaling operation

results in a sequence of critical designs which are very useful in monitoring the design process. In particular, the weight comparison of the critical designs can be used to terminate the design whenever the weight reduction becomes small, or ceases altogether. This is especially important when approximate optimization techniques, such as the fully stressed design principle, are used.

The second advantage of using scaling is that it prevents intermediate designs from departing excessively from the critical state. This in turn has a stabilizing influence on the convergence of the design process, and in some problems it even makes the difference between a convergent process and no convergence at all.

The scale factor for the stress constraints is simply

$$\mu^{(S)} = R_{max} \ , \tag{B.5.3}$$

where R_{max} is the maximum stress ratio obtained by the stress ratio method -- see (B.2.5). The scale factor is <u>exact</u> --- that is, the resulting design will be precisely stress-critical --- if the internal forces remain unchanged upon scaling. This condition is satisfied, apart from statically determinate structures, if all element stiffness matrices have the form

$$[K_i] = [k_i]A_i^n \ , \tag{B.5.4}$$

where n is common to all elements, and if the loading is size-independent.

It can easily be shown that if (B.5.4) is satisfied, the buckling-constrained uniform scaling is also exact, the scale factor

being

$$\mu^{(B)} = (Q_{max})^{1/n} , \qquad (B.5.5)$$

where Q_{max} is the maximum buckling ratio defined in (B.3.27).

For the problems where the scaling operation is inexact, the scale factor for the buckling constraints must be computed from (B.3.20). To obtain the scale factor $\mu_r^{(B)}$ which would make the r^{th} buckling load critical, we substitute $\delta p_r = p_r^* - p_r$ and $\delta D_m = D_m' - D_m = (\mu_r^{(B)}-1)D_m$, obtaining

$$p_r^* - p_r = (\mu_r^{(B)}-1) \sum_{m=1}^{M} p_{r,m} D_m .$$

Solving for the scale factor, we get

$$\mu_r^{(B)} = (p_r^*-p_r)/(\sum_{m=1}^{M} p_{r,m} D_m) + 1 . \qquad (B.5.6)$$

A scale factor is computed for each potentially active buckling constraint from (B.5.6), and the maximum value is used for $\mu^{(B)}$, i.e.

$$\mu^{(B)} = \max_r \mu_r^{(B)} , \quad r = 1,\ldots,R_{act} . \qquad (B.5.7)$$

The factor used in the scaling operation (B.5.1) is the larger of $\mu^{(S)}$ and $\mu^{(B)}$, i.e.

$$\mu = \max(\mu^{(S)},\mu^{(B)}) . \qquad (B.5.8)$$

If the structure meets the conditions for exact scaling, the user should specify this in the input to DESAP 2, together with the inertia exponent n in (B.5.4). The buckling scale factor will then be

computed from (B.5.5) and the analysis of the scaled structure will be omitted, resulting in a considerable saving in computer time. Otherwise, (B.5.6) and (B.5.7) are used to calculate $\mu^{(B)}$, and the scaled design will be reanalyzed and if not critical, will be scaled again.

DESAP 2 gives the user the option of dispensing with scaling altogether. The no-scaling option could be used if the scaling operation is inexact and the intermediate critical designs are of no interest. This could save a substantial amount of computer time, but at the expense of losing some control over the convergence of the design procedure.

Scaling should not be used if the loading is dominated by thermal or gravity loads and the element stiffness matrices have the form $[K_i] = [k_i]A_i$ (linear size-stiffness relationship). It is readily seen that under these circumstances a uniform scaling operation is entirely ineffective, since it leaves the stresses and buckling loads unchanged.

If the uniform scaling option is chosen, the weight of each critical design is calculated and the smallest of these weights, W_{min}, is stored. The design procedure is terminated whenever

$$(W-W_{min})/W_{min} \geq \varepsilon \ , \qquad (B.5.9)$$

where W is the weight of the current critical design and ε is a small prescribed constant. This cut-off criterion prevents further redesign operations if these are going to result in a weight increase.

The scaling operation is always by-passed if $\mu > 2$, in order to avoid the corresponding large weight increase. Experience has shown that by preceding uniform scaling with a redesign operation, a lower weight is obtained than from the scaling-redesign sequence. It should be pointed out that a large scale factor would arise only during the first design cycle if the initial design is poorly chosen.

C. ORGANIZATION OF THE PROGRAM

C.1 Overlay Tree

The structure of the program is illustrated with the overlay tree in Fig. C.1.1 Details of the element subroutines are shown separately in Figs. C.1.2 to C.1.7.

Additional elements can be added to the program by replacing the call NOELEM in ELTYPE with the new element subroutines. The element code numbers (MTYPE) 5 and 8 have been reserved for this purpose.

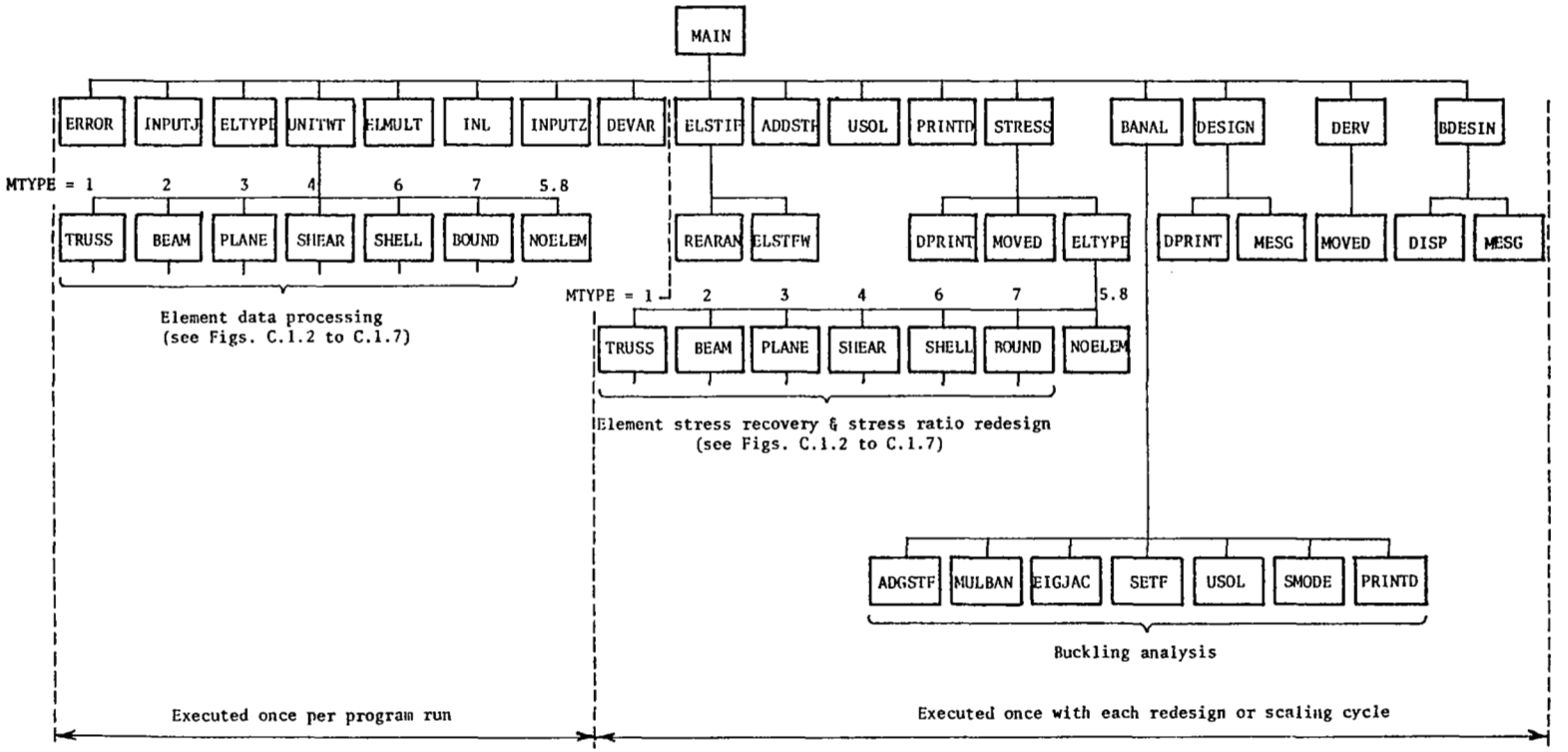


Figure C.1.1

Overlay Tree for DESAP 2.

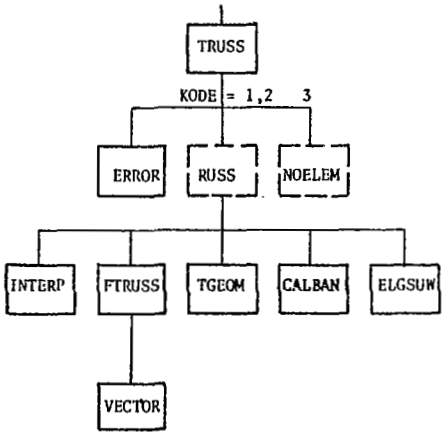


Figure C.1.2

Overlay Tree for Bar Element

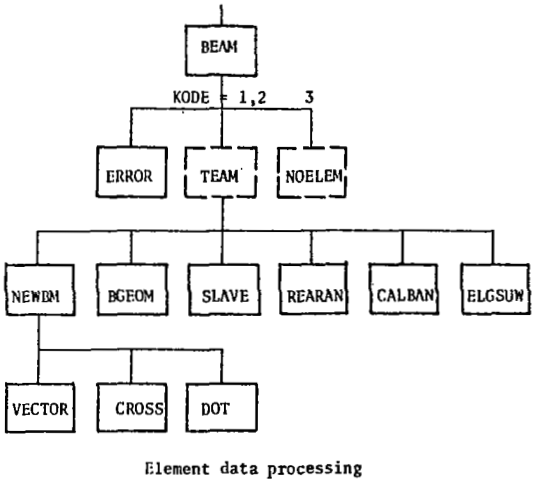
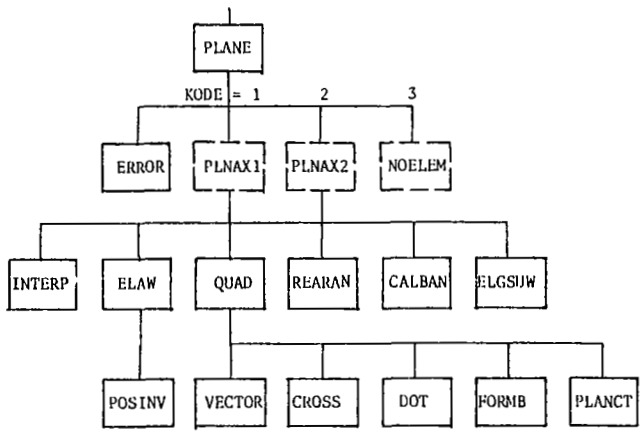
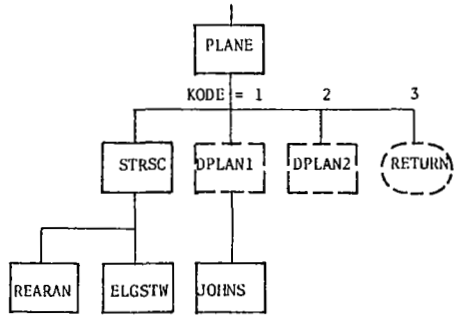


Figure C.1.3

Overlay Tree for Beam Element



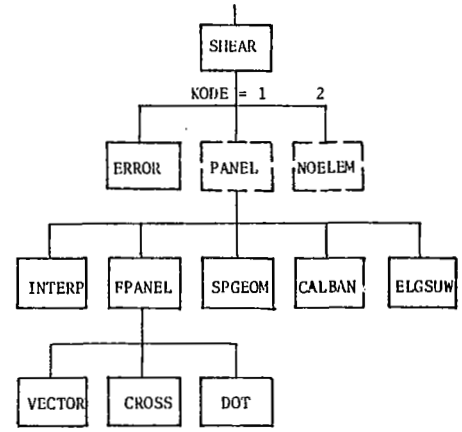
Element data processing



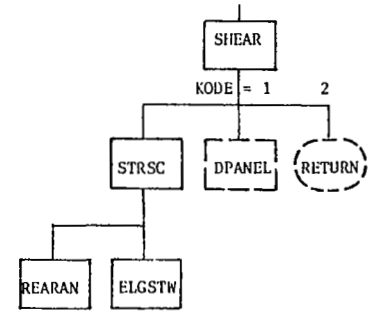
Element stress recovery & stress ratio redesign

Figure C.1.4

Overlay Tree for Plane Stress Element



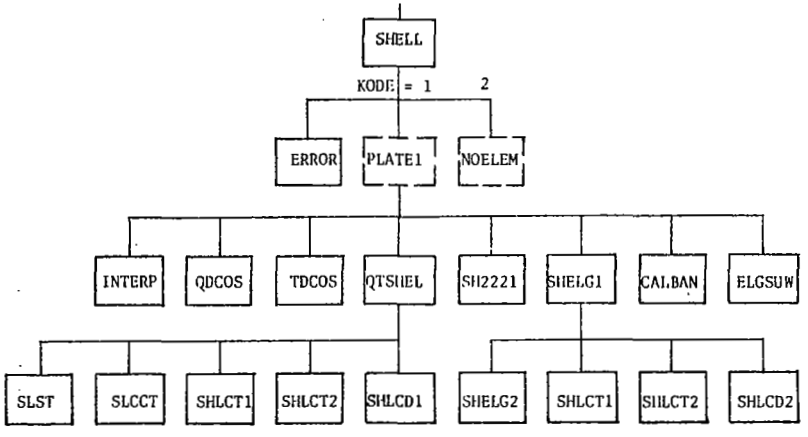
Element data processing



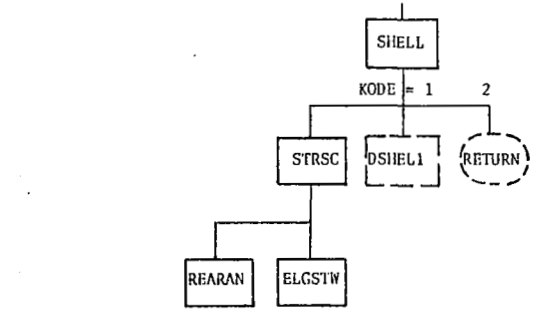
Element stress recovery & stress ratio redesign

Figure C.1.5

Overlay Tree for Shear Panel



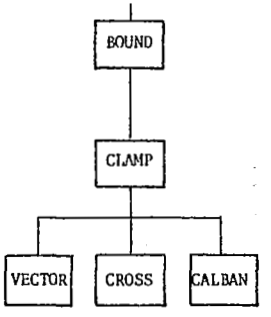
Element data processing



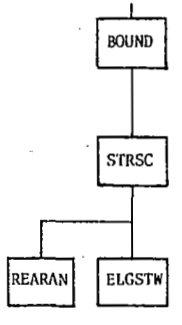
Element stress recovery & stress ratio redesign

Figure C.1.6

Overlay Tree for Plate-Shell Element.



Element data processing



Element stress recovery & stress ratio redesign

Figure C.1.7

Overlay Tree for Boundary Element

C.2 Element Subroutines

The element subroutines of DESAP [2] are constructed in such a manner as to facilitate modifications by the user. Each element sub-routine package consists of two parts: element data processing, and stress recovery combined with stress ratio redesign. The element data, including the design information, is read in by the processor sub-routines shown by the dashed lines in Figs. C.1.2 to C.1.6. Provision has been made for more than one such processor subroutine with each element package, the different processors being distinguished by their construction code numbers (KODE). This scheme allows the user to provide his own processor subroutines which will best satisfy his special design requirements.

After the data for an element is read in, it is immediately processed and the results placed on auxiliary storage devices. Standard computations, such as the formation of the unit elastic and geometric stiffness matrices, transformation of coordinates, etc , are handled by calling the appropriate element computational subroutines, identified by the solid lines. No modification by the user is required here, since these sub-routines are independent of the construction details and design criteria.

The processed element data consists of:

1) Element unit elastic stiffness matrices and load vectors.

2) Element unit geometric stiffness matrices.

3) Element unit force recovery matrices and force vectors.

4) Data associated with failure and local instability criteria.

The first three sets of data are stored in a standard form (common to all elements), whereas the format of the failure and local instability data

is completely flexible --- it may, for example, contain parameters of the failure equations (as in the subroutines presently provided), or purely numerical data in tabular form.

Following each prebuckling analysis of the structure, the element nodal forces are recovered by subroutine STRSC. The appropriate design subroutine is then used to compute the stresses and carry out stress ratio redesign. The design subroutines, also identified by the dashed lines in Figs. C.1.2 to C.1.6, must be compatible with the corresponding processor subroutines --- that is, if the user supplies his own processor, he must also have a matching design subroutine, both labelled with the same construction code.

Note that the boundary element, Fig. C.1.7, is not subject to redesign and, therefore, lacks a construction code and design subroutine.

The user-supplied processor and design subroutines are to be inserted in the place of NOELEM and RETURN, respectively, shown in Figs. C.1.2 to C.1.6.

C.3 Storage Requirements

The common in-core storage blocks used in DESAP 2 are:

1) COMMON/JUNK/---368 locations---is used for storage of miscellaneous data, as its name implies.
2) COMMON/ELPAR/---33 locations---serves for storage of parameters that control the execution of the program.
3) COMMON/UNITS/---12 locations---contains the assignments (numbers) of the input-output units.
4) COMMON/EM/---5548 locations---is used primarily for processing of element data.
5) COMMON/CONTR/---29 locations---contains control data for the redesign operations.
6) Unlabelled storage area A(n). This is the main working area of the program and its dimension largely controls the allowable size of the structure. The capacity of the program can be adjusted by changing "n" in the following statements at the beginning of the main program:

```
DIMENSION A(n)
REAL*8 AD (n/2)
EQUIVALENCE (A(1), AD(1))
MTOT = n
```

The availability of the required storage in A(n) is checked at various stages of the program, and if insufficient, an error message is printed (subroutine ERROR) and the execution of the program terminated. The minimum value of "n" required in each subroutine where A(n) is used, is listed in Table C.3.3. It is advisable, however, to use the largest possible value for "n", since this would reduce the time spent on the

transfer of data between the auxiliary storage devices and the core.

COMMON/COMPL---852 locations---is used only in the plate-shell element subroutines.

The usage of the common blocks in the various subroutines is shown in Table C.3.4.

Parameter	Description
LL	Number of load conditions subject to stress constraints
M	Max (LL,2)
MM	2*NEQB+(MBAND-1)/NEQB
MBAND	Band width of structural stiffness matrix
NELTYP	Number of element types in the structure
NEQ	Number of degrees of freedom for the structure, i.e., number of equations
NEQB	Number of equations in a block = min{(n-4*LL)/[4*(MBAND+LL)],n/[4*(MBAND+M)+1],NEQ}
ND	Number of degrees of freedom for an element (see Table C.3.2)
NG	Number of unit geometric stiffness matrices for an element (see Table C.3.2)
NI	Dimension of element design information array (see Table C.3.2)
NS	Number of stresses calculated for an element (see Table C.3.2)
NU	Number of unit stiffness matrices for an element (see Table C.3.2)
NUMDV	Number of design variables
NUMEL	Number of elements in structure
NUMNP	Number of nodal points in structure
NUMGEO	Number of different geometric properties used for an element type
NUMTC	Maximum number of temperatures for which material properties of an element type are specified
NUMFX	Number of fixed end-force sets (beam only)
NV	Number of unit load vectors for an element (see Table C.3.2)
NW	Number of initial stress vectors for an element (see Table C.3.2)

Table C.3.1

List of Parameters that Determine the Storage Requirements.

Element	Parameters						
	NU	NV	NW	ND	NS	NI	NG
Truss	1	1	1	6	1	5	1
Beam	2	2	2	24	12	11	1
Plane	1	2	1	12	15	10	3
Shear	1	1	1	12	4	3	1
Plate-Shell	2	2	1	24	6	5	3
Boundary	1	1	1	6	2	0	0

Table C.3.2

Values of Storage Parameters for Various Elements.

Subroutine	Minimum Required Value of "n"
INL	6*NUMNP+6*LL+2*NEQB*LL
ADDSTF	4*(MBAND+LL)*NEQB+4*LL
USOL	4*(MBAND+M)*NEQB+NEQB
PRINTD	6*NUMNP+6*M+2*NEQB*M
STRESS	3*NUMDV+4*LL+2*NEQB+NEQ
MULBAN	NEQB*(MBAND+LL)+2*MM*NVEC
DERV	2*NUMDV+NEQB
BDESIN	6*NUMDV+4*LL
ELTYPE	NUMDV+10*NUMNP+m, where "m" depends on the element type (see below)

Subroutine	Minimum Required Value of "m"
TRUSS	(2+5*NUMTC)*NUMMAT+2*NUMGEO
BEAM	10*NUMGEO+6*NUMMAT+12*NUMFX
PLANE	(2+8*NUMTC)*NUMMAT+5*NUMGEO
SHEAR	(2+4*NUMTC)*NUMMAT
SHELL	(2+7*NUMTC)*NUMMAT

Table C.3.3

Core Storage Requirements for A(n)

The storage parameters are defined in Table C.3.1.

	JUNK	ELPAR	EM	UNITS	CONTR	COMPL
MAIN	x	x	x	x	x	
INL	x					
CALBAN		x		x	x	
ELSTIF			x			
ADDSTF			x			
STRESS	x	x				
STRSC	x		x	x	x	
BANAL	x	x		x	x	
ADGSTF			x			
DERV			x			
DESIGN	x			x	x	
BDESIN	x			x	x	
ERROR				x		
TRUSS	x	x		x		
RUSS	x		x	x	x	
FTRUSS	x		x			
DTRUSS	x					
TGEOM			x			
BEAM	x	x		x		
TEAM	x		x	x	x	
NEWBM	x		x			
SLAVE			x			
DBEAM	x					
BGEOM	x		x			
PLANE	x	x	x	x		
ELAW	x					
QUAD	x	x	x		x	
FORMB	x					
PLNAX1	x		x	x		
DPLAN1	x					
PLNAX2	x		x	x		
DPLAN2	x					
SHEAR	x	x		x		
PANEL	x		x	x	x	
FPANEL	x		x			
DPANEL	x					
SPGEOM	x					
SHELL	x	x		x		
PLATE1	x		x	x	x	x
QTSHEL	x		x			x
SLST						x
SLCCT						x
DSHEL1	x					
SHELG1			x			x
SHELG2						x
BOUND	x	x		x		
CLAMP	x		x	x		

Table C.3.4

Usage of Common Blocks in Various Subroutines.

C.4 Requirements for Auxiliary Storage Devices

DESAP 2 requires nine sequentially accessible, auxiliary storage units. The units, together with the required storage capacities, are listed below. The storage parameters can be found in Table C.3.1.

I1---is used for permanent storage of element load multipliers and the minimum allowable design variables.

No. of locations = 4*(LL+NUMDV)

I2---is used for:

a) temporary storage of element stiffness matrices,

b) scratch file during solution of simultaneous equations,

c) storage of displacement vectors,

d) storage of the coordinate vectors during buckling analysis,

e) temporary storage of the derivatives of the buckling parameters.

No. of locations = the larger of the following:

a) $\sum_{NUMEL}$ [2+ND*(ND*2+9)],

b) 2*|MBAND/NEQB|*MBAND*NEQB,

c) 2*NEQ*M,

d) 2*NUMDV.

I3---is used for:

a) scratch file during solution of simultaneous equations,

b) temporary storage of element geometric stiffness matrices,

c) storage of structural stiffness matrix and the coordinate vectors in the solution of equations during buckling analysis,

d) storage of mode shapes during buckling analysis.

No. of locations = the larger of the following:

a) 2*NEQ*(MBAND+LL),

b) $\sum_{NUMEL}$ [2+ND*(1+2*ND),

c) 2*NEQ*(MBAND+2),

d) 4*NEQ.

I8---is used for permanent storage of the equation number matrix, control parameters for element subroutines (NPAR), element unit force recovery matrices, element stress-constrained design information, and unit weights of the design variables.

No. of locations = 6*NUMNP+14*NELTYP+NUMDV

$$+ \sum_{NUMEL} [7+ND+NI+(ND*NU+4*NW)*2*NS].$$

I9---is used for:

a) scratch file in the assembly of structural stiffness and geometric stiffness matrices,

b) scratch file in the solution of simultaneous equations (during prebuckling and buckling analysis),

c) scratch file during buckling analysis.

No. of locations = the larger of the following:

a) $|(NUMEL)^{1/2}|*[ND*(ND*2+9)+2]$

b) |MBAND/NEQB|*2*MBAND*NEQB,

c) 4*NEQ.

I10---is used for storage of structural stiffness matrix, load vectors and geometric stiffness matrix (statically indeterminate structures only).

No. of locations = 2*NEQ*(2*MBAND+LL+2).

I11---is used for:

a) permanent (indeterminate structures) or temporary (determinate structures) storage of element unit geometric stiffness matrices,

b) permanent storage of the structural geometric stiffness matrix (statically determinate structures only).

No. of locations = the larger of the following:

a) $\sum_{NUMEL}$ (4+ND*ND*NG),

b) 2*NEQ*MBAND.

I12---is used for permanent storage of element unit stiffness matrices and load vectors, and permanent storage of structural load vectors due to size-independent loads.

No. of locations = $\sum_{NUMEL}$ [7+ND+2*ND*(ND+NU+4*NV)]

+ 2*LL*NEQ.

I13---is used for:

a) storage of coordinate vectors used in buckling analysis,

b) scratch file during buckling analysis.

No. of locations = the larger of the following:

a) 4*NEQ,

b) 2*NEQ*(MBAND+2).

The assignments of the auxiliary storage devices, together with those of the input-output units (IR = card reader, IW = printer, and IP = card punch), can be changed by altering the assignment numbers at the beginning of the Main Program.

C.5 Assembly and Solution of Simultaneous Equations

We denote the load vector of the structure representing the ℓth load condition by $\{Q^{(\ell)}\}$ and the corresponding nodal displacement vector by $\{u^{(\ell)}\}$. The equilibrium equations governing the prebuckling state thus are

$$[K][U] = [Q] \ , \tag{C.5.1}$$

where [K] is the stiffness matrix of the structure,

$$\begin{aligned} [U] &= [\{u^{(1)}\}\ \{u^{(2)}\}\ \ldots\ \{u^{(L)}\}] \ , \\ [Q] &= [\{Q^{(1)}\}\ \{Q^{(2)}\}\ \ldots\ \{Q^{(L)}\}] \ , \end{aligned} \tag{C.5.2}$$

and L represents the number of load conditions.

The assembly of the equations and their solution, carried out by ADDSTF and USOL, respectively, are essentially the same as in the SOLID SAP program. The only significant difference is that DESAP 2 uses double precision arithmetic, whereas single precision was employed in SOLID SAP. Reconversion into single precision is not recommended, even if your machine carries above-average number of digits, because roundoff errors in the analysis tend to be magnified during the redesign cycle. The cumulative buildup of error may even lead to a non-convergence of the design process in large structures.

The equations, i.e., the matrices [K] and [Q], are formed in blocks in the unlabelled common area A(n), and stored on auxiliary storage devices. The banded structure of the stiffness matrix is

exploited throughout the formation of the equations and their solution. The size of a block is determined automatically by the computer once the bandwidth and the number of load vectors have been established. There must be sufficient space in A(n) for at least two equations.

The equations are solved by the Gaussian elimination procedure, one block at a time. The structural stiffness matrix is not destroyed during the solution procedure.

C.6 Solution of Buckling Equations

As pointed out in (B.5.15), buckling analysis leads to a matrix eigenvalue problem of order R (R = number of degrees of freedom):

$$[K]\{u\} = p[G]\{u\} \ . \tag{C.6.1}$$

DESAP 2 uses an iterative Rayleight-Ritz method[7] to extract only the potentially active eigenvalues (load parameters) $p^{(1)}, p^{(2)},\ldots, p^{(R_{act})}$ and the corresponding eigenvectors (buckling modes) $\{u^{(1)}\}, \{u^{(2)}\},\ldots, \{u^{(R_{act})}\}$.

The first step is to introduce the R-dimensional coordinate vectors $\{z^{(r)}\}$, $r = 1,2,\ldots,R_{act}$, and to express the nodal displacement vector in the form

$$\{u\} = \sum_{s=1}^{R_{act}} y_s\{z^{(s)}\} \ , \tag{C.6.2}$$

where y_s are called the Rayleigh-Ritz coordinates. Equation (C.6.2) can also be written as

$$\{u\} = [Z]\{y\} \ , \tag{C.6.3}$$

where $[Z] = [\{z^{(1)}\}\ \{z^{(2)}\}\ \ldots\ \{z^{(R_{act})}\}]$.

The coordinate vectors can be interpreted as the first approximation for the active buckling modes, whereas {u} represents an improved approximation.

Once the coordinate vectors have been chosen, the Rayleigh-Ritz coordinates can be obtained by minimizing the potential energy of

the buckling deformations:

$$E = \frac{1}{2}\left(\{u\}^T[K]\{u\} - p\{u\}^T[G]\{u\}\right) . \qquad (C.6.4)$$

Substituting (C.6.3) in (C.6.4) and applying the operations $\partial E/\partial y_s = 0$, results in the reduced (order R_{act}) eigenvalue problem

$$[K']\{y\} = p[G']\{y\} , \qquad (C.6.5)$$

where

$$[K'] = [Z]^T[K][Z], \quad [G'] = [Z]^T[G][Z] . \qquad (C.6.6)$$

Since $R_{act} \leq 2$ in DESAP 2, the solution of (C.6.5) can easily be written down once [K'] and [G'] have been formed.

The improved estimate for the buckling modes can now be obtained by substituting the eigenvectors of (C.6.5) $\{y^{(1)}\}$, $\{y^{(2)}\}$, ..., $\{y^{(R_{act})}\}$ in (C.6.3). Thus

$$\{u^{(r)}\} = [Z]\{y^{(r)}\}, \quad r = 1,2,\ldots,R_{act} ,$$

or

$$[U] = [Z][Y] , \qquad (C.6.7)$$

where

$$[U] = [\{u^{(1)}\}\ \{u^{(2)}\}\ \ldots\ \{u^{(R_{act})}\}]$$

and

$$[Y] = [\{y^{(1)}\}\ \{y^{(2)}\}\ \ldots\ \{y^{(R_{act})}\}] .$$

It is shown in [7] that the knowledge of [U] enables us to obtain a better approximation to the coordinate vectors:

$$[Z] = [K^{-1}][G][U] \ , \tag{C.6.8}$$

which can be used in turn to obtain closer estimates of the critical load parameters and the buckling modes. The process is repeated until the change in the eigenvalues becomes sufficiently small:

$$\max_r \ |\Delta p^{(r)}/p^{(r)}| \leq 0.005 \ , \tag{C.6.9}$$

where $\Delta p^{(r)}$ is the change in $p^{(r)}$ between the last two iterations.

Since the inversion of a banded matrix is a very inefficient operation, the improved coordinate vectors are not calculated directly from (C.6.8), but are obtained from the solution of the simultaneous equations

$$[K][Z] = [C] \ , \tag{C.6.10}$$

where [C] = [G][U].

The initial choice of the coordinate vectors [Z] can be made in two ways:

(1) the coordinates are chosen by the user and punched on data cards in the order of the equation numbers, or

(2) the coordinates are assigned by the computer by means of a random number generator in subroutine INPUTZ.

These coordinate vectors are used only to start the iterative analysis of the initial design. In the analysis of all subsequent designs, the initial coordinate vectors are taken as the buckling modes of the previous design. Since the buckling modes of two successive designs do not change much, the number of iterations required for each buckling analysis will be small, with the possible exception of the initial design.

The geometric stiffness matrix of the structure [G] is assembled by the subroutine ADGSTF. It differs from ADDSTF, which is used for the assembly of the elastic stiffness matrix [K], in minor details only. There is no need to recompute [K] for the buckling analysis, since it is already available from auxiliary storage where it was placed during analysis of the prebuckling state. The simultaneous equations (C.6.10) are solved by the same subroutine that was employed for the equilibrium equations of the prebuckling state, namely USOL. A flow diagram of the buckling analysis algorithm appears in Fig. C.6.1.

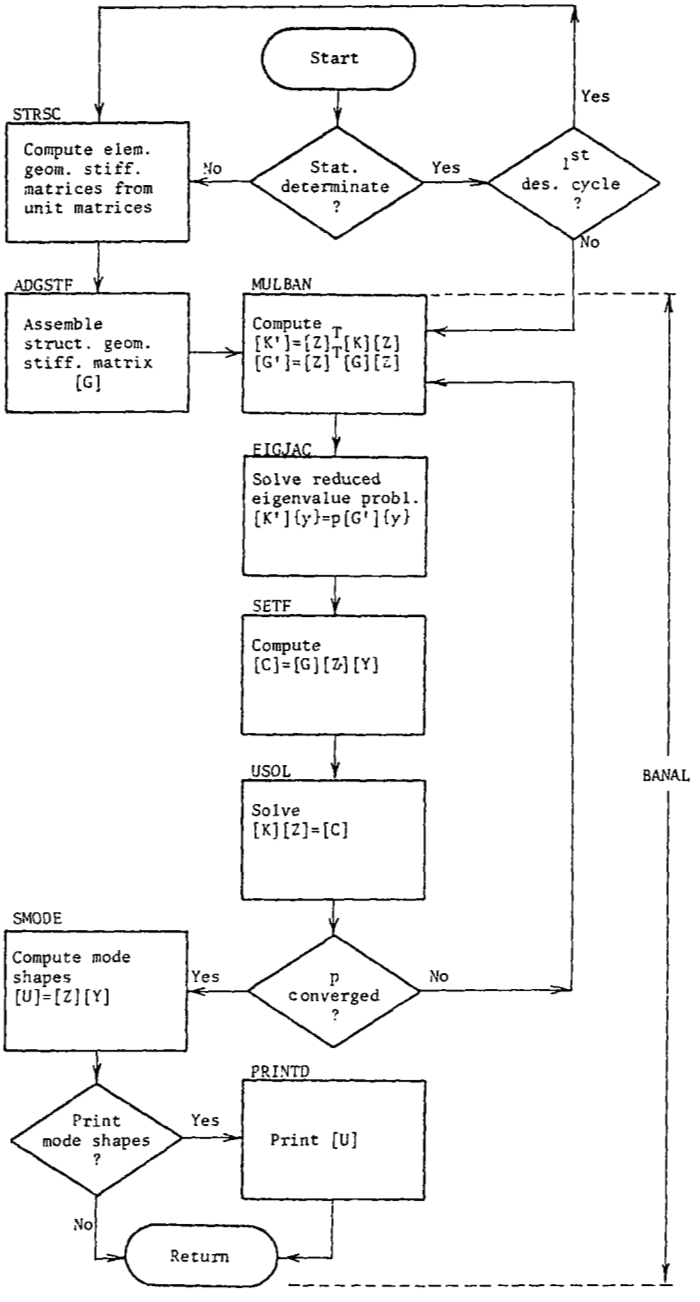


Figure C.6.1

Flow Diagram for Buckling Analysis.

C. 7 Cutoff Criteria and Restart Option

The design process is terminated whenever one of the following cutoff criteria are satisfied:

1) The number of design cycles equals a prescribed number NCYCL. A redesign cycle is defined as an application of the redesign equations; uniform scaling operation, if used, does not count as a redesign cycle.

This feature enables the user to evaluate the progress of the design procedure after a few design cycles and take corrective action if necessary (particularly, adjust the relaxation parameter α).

2) The number of successive uniform scaling operations equals a prescribed number NSCALE. Program termination here indicates that uniform scaling is an ineffective operation and should not be used.

3) The structural weight of critical design begins to increase, i.e.,

$$(W - W_{min})/W_{min} \geq \varepsilon \ , \tag{C.7.1}$$

where the terms are defined in (B.5.9).

4) The design is acceptable, i.e., convergence is complete. An acceptable design satisfies one of the following optimality conditions:

(i) The design is fully stressed and the buckling constraints are not violated, i.e.,

$$1 - \delta \leq R_{max} \leq 1 + \delta, \quad 1 - \delta \leq R_{min} \leq 1 + \delta$$
$$\text{and } Q_{max} \leq 1 + \delta \ , \tag{C.7.2}$$

where R_{max}, R_{min} and Q_{max} are the stress and buckling ratios defined in (B.2.6) and (B.3.27), respectively, and δ is a prescribed (small) constant.

(ii) The design is buckling-critical, the optimality criterion is satisfied for the buckling constraints, and stress constraints are not violated, i.e.,

$$1 - \delta \leq Q_{max} \leq 1 + \delta;$$

$$1 - 5\delta \leq \frac{1}{\rho_m} \sum_{r=1}^{R_{act}} \lambda_r p_{r,m} \leq 1 + 5\delta \quad \text{for all active } D_m, \tag{C.7.3}$$

and $R_{max} \leq 1 + \delta$.

At the user's option, DESAP 2 will produce a restart deck just prior to the termination of the program. The restart deck contains the last values of the design variables together with their minimum allowable values. It can be used to replace the original design variable data input deck if the user decides to resubmit the program and use the last design as the starting point.

D. DESCRIPTION OF INPUT DATA

D.1 General Input Data

I. Heading Card (20A4)

1	80
HED	

HED = Any alphameric statement that is to appear as the first line of output.

II. Program Control Card (4I5)

1	6	11	16	20
NUMNP	NELTYP	LL	NUMDV	

NUMNP = Number of nodal points in the structure. Include special nodes used for beam elements (slave nodes and points used to define principal axes) and boundary elements (points used to define directions of the elements).

NELTYP = Number of different element types used in the structure. Count each construction code of an element as a separate element type.

LL = Number of separate loading conditions for which the structure is to be designed. Note that only one of these loading conditions may be subjected to buckling constraints.

NUMDV = Number of independent design variables in the structure.

III. Design Control Card (3I5, 2F10.0, 3I5)

1	6	11	16	26	36	41	46	50
NCYCL	NSCALE	KSCALE	DELTA	EPSIL	KPUNCH	KPRINT	LBUCK	

NCYCL = Maximum allowable number of redesign cycles (see Sec. C.7). If NCYCL = 0, analysis of initial design only will be carried out.

NSCALE = Maximum allowable number of successive scaling operations (see Sec. C.7). If blank, computer sets NSCALE = 3.

KSCALE = Code for uniform scaling operation.
KSCALE < 0: omit uniform scaling operation.
KSCALE = 0: carry out uniform scaling whenever the design is not critical, and analyse the scaled structure. Used when scaling is not exact (see Sec. B.5).
KSCALE = n > 0, where n is the exponent of A_i in $[K_i] = [k_i]A_i^n$. Uniform scaling will be carried out whenever the design is not critical, but analysis of the scaled structure will be skipped. To be used only when scaling is exact (see Sec. B.5).

DELTA = The small parameter δ that specifies the latitude of the cut-off criteria listed in Sec. C.7. If blank, computer sets DELTA=0.05.

EPSIL = The small parameter ε that specifies the allowable weight increase of the structure in the weight-cutoff criterion in Sec. C.7. If blank, computer sets EPSIL = 0.1.

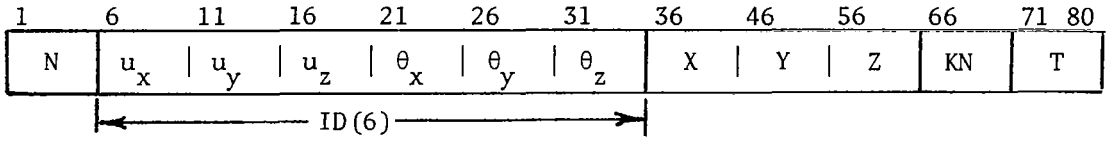
KPUNCH = Code for restart card deck.
KPUNCH $\neq$ 0: punch design variable data cards for the last design before program termination occurs (see Sec. C.7).
KPUNCH = 0: omit punching.

KPRINT = Code for printout of nodal displacements, incl. buckling modes.
KPRINT $\neq$ 0: print nodal displacements after each analysis of the structure.
KPRINT = 0: omit printout of nodal displacements.

LBUCK = Code for constraints on buckling loads.
LBUCK = 0: no buckling constraints exist.
LBUCK = n > 0: the nth load condition is subjected to constraints on the buckling loads.

IV. Nodal Point Data (7I5, 3F10.0, I5, F10.0)

One card for each node; cards do not have to be in node number sequence, but the last card in the deck must be for the last node.



N = Node number.

ID = Motion code for nodal displacements (u_x, u_y, u_z) and rotations θ_x, θ_y, θ_z) with respect to the global coordinate axes.
ID(i) = 0: motion is permitted.
ID(i) = 1: motion is not permitted (the corresponding degrees of freedom are eliminated).
ID(i) = m, m > 1: node N is connected to a master node m. (used for beam elements only---see Ch. E).

X,Y,Z = Global coordinates of the node.

KN = Node number increment used in automatic generation of nodal points (see below).
KN = 0: automatic generation is not used.
KN > 0: use automatic generation.

T = Nodal point temperature.
Note: temperature distribution in the structure is prescribed by nodal temperatures. The reference temperature (i.e. temperature of the stress-free state) is specified on the element cards.

Automatic node generation---if a series of nodes exists that satisfies the following requirements:
(a) the nodes are equally spaced along a straight line;
(b) the temperature distribution along the line of nodes is linear;
(c) the motion codes for all the nodes are identical;
(d) the node numbers form the sequence n, n+KN, n+2*KN,...,
then only cards for the first and last node of the series are required. The data for the intermediate nodes will be generated automatically. The motion code for the series will be taken from the first card, the node increment KN from the last card of the series.

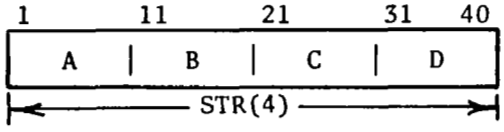
Automatic motion code generation---if a particular degree of freedom is to be eliminated from a series of nodes, this can be achieved by using -1 as the motion code on the first card, and 1 on the last card of the series. The computer will automatically use 1 as the motion code for the first and all intermediate node cards of the series.

V. Element Data

This data differs for various element types and construction codes; it is described from Ch. E onwards.

VI. Structural Load Multipliers (4F10.0)

One card for each separate load condition; the cards must be in ascending order of load condition numbers.



STR(1) = Fraction of element load A which is to be added to the load condition (structural load multiplier for element load A).
STR(2) = Structural load multiplier for element load B.
STR(3) = Structural load multiplier for element load C.
STR(4) = Structural load multiplier for element load D.

Element loads A, B, C and D are defined with description of element data (Ch. E onwards). The element loads represent thermal and gravity loading or distributed loads of prescribed magnitude. The structural load multipliers enable the user to choose any fraction of the element loads for any load condition.

VII. Buckling Control Card (F10.0, 4I5, 2F10.0)

This card is to be omitted if no buckling constraints are imposed (LBUCK = 0).

1	11	16	21	26	31	41 50
COEFFT	MODEIN	NMODE	INDET	NVEC	ALPA	OMEGA

COEFFT = The constraint p* on the critical load parameters, defined in eqs. (B.3.1) and (B.3.2). It represents the desired factor of safety against general buckling for the prescribed loading. If blank, computer sets COEFFT = 1.0.

MODEIN = Code for generation of the initial coordinate vectors (mode shapes) $\{z^{(r)}\}$---see Sec. C.6.1.
MODEIN = 0: coordinate vectors are to be generated by the computer as random numbers.
MODEIN = 1: coordinate vectors are read in with input data.

NMODE = Number of buckling modes on which the buckling constraint is to be imposed.
NMODE = 1: the optimal design is expected to be determined by a single (fundamental) buckling mode.
NMODE = 2: the optimal design is expected to be determined by two buckling modes simultaneously (i.e., the lowest two buckling loads are expected to coalesce). If blank, computer sets NMODE = 1; if NMODE $>$ 2, computer sets NMODE = 2.

Note: The use of NMODE = 1 is recommended for the first run, since it works for a vast majority of problems and requires somewhat less computer time than NMODE = 2.

INDET = Code for static determinacy of the prebuckling state.
INDET = 0: the internal forces are statically indeterminate prior to buckling.
INDET = 1: the internal forces are statically determinate prior to buckling.

Note: the code applies only to the load condition that is subjected to the buckling constraints. If INDET = 1, the geometric stiffness matrix of the structure is assembled only once; if INDET = 0, the matrix is reassembled prior to each buckling analysis---see Sec. B.3.9.

NVEC = Number of coordinate vectors $\{z^{(r)}\}$ used in the iterative buckling analysis. If NVEC < NMODE or NVEC > 2, computer sets NVEC = 2.

Note: the number of coordinate vectors used must equal at least the number of eigenvalues (NMODE) to be extracted. It is advisable, however, to use NVEC = 2 (the maximum allowable value) in all cases (even if NMODE = 1), since this results in faster convergence.

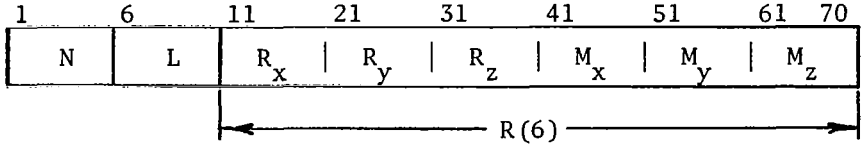
ALPA = The relaxation factor $\alpha < 1$ used in the buckling-constrained redesign formula (B.3.13).

Note: the chances that the iterative design procedure converges uniformly increase with increasing values of ALPA. Consequently, it is better to use an ALPA that is too large (resulting in under-relaxation), than one that is too small (over-relaxation). Large values of ALPA, however, may require a larger number of design iterations before the cutoff criteria are activated. In order to minimize computer time, it is advisable to start with a "normal" value, which can be estimated from [3] $\alpha = n/(n+1)$, where n is the inertia exponent that dominates the structural stiffness matrix. After a few redesign cycles ALPA may be increased or decreased, depending on the convergence characteristics of the problem.

OMEGA = The comparison parameter $\omega < 1$ used in selecting the potentially active buckling constraints (see Sec. B.4). If blank, computer sets OMEGA = 0.8.

VIII. Nodal Load Data (2I5, 6F10.4)

One card per load condition for each node that carries concentrated loads or moments. Cards must be arranged in an ascending order of node numbers. The data must end with a blank card. If no nodal loads are present, use the blank card only.



N = Node number.

L = Load condition number.

R = Concentrated forces (R_x,R_y,R_z) and moments (M_x,M_y,M_z) in the global coordinate directions. Positive values denote loads in the positive coordinate directions, negative values in the negative coordinate directions (see Fig. D.1.1).

Note: the right-hand screw rule is used to determine positive directions for the moments, as shown in Fig. D.1.1. The same sign convention is employed for nodal displacements and rotations in the printout of the analysis.

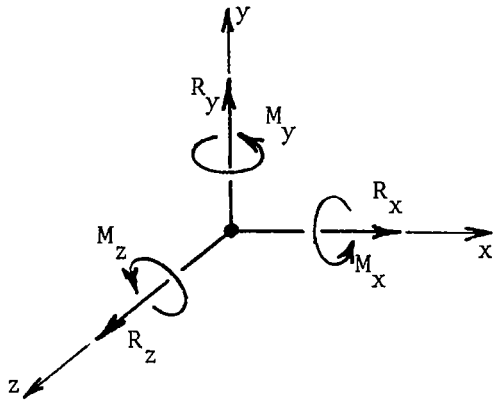


Fig. D.1.1

Positive Directions of
Nodal Forces and Moments.

IX. Initial Coordinate Vectors (8F10.5)

One set of cards for each coordinate vector $\{z^{(r)}\}$ used in buckling analysis. The number of vectors must equal NVEC specified on the Buckling Control Card. The components of each coordinate vector are to be read in by blocks in the order of their equation numbers* If the initial coordinate vectors are to be generated by the computer (specified by MODEIN = 0 on the Buckling Control Card), omit all cards.

X. Design Variable Data

One card for each design variable; cards must be in ascending order of design variable numbers.

1	6	16 25
N	AOLD	AMIN

N = Design variable number.

*Note that prior knowledge of the equation numbers and the number of equations in each block are required. This information is available only after the first run of the problem has been completed.

AOLD = Initial value of design variable. If AOLD < AMIN, computer sets AOLD = AMIN.

AMIN = Minimum allowable value of design variable.

Automatic data generation---if there exists a sequence of design variables N = n, n+1, n+2, ... that have identical values of AOLD and AMIN, it is sufficient to include only the last card of sequence in the deck. The omitted data will be generated automatically.

E. BAR ELEMENT

E.1 General Information

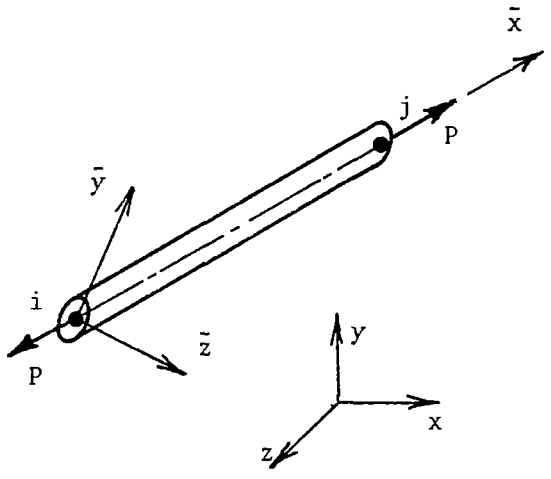


Fig. E.1.1

Typical Bar Element.

A typical bar element is shown in Fig. E.1.1. The element coordinate axes $\bar{y}$ and $\bar{z}$ coincide with the principal axes of the cross section, whereas $\bar{x}$ is the centroidal axis.

The displacements are assumed to be linear in $\bar{x}$; consequently, the axial force P will be constant throughout the length of the element.

Since bar elements do not carry bending or torsion, the nodal rotations must be suppressed on the Nodal Point Data cards, utilizing the motion code ID(6)---see Sec. D.1. For the same reason, only zero concentrated moments are permitted on the Nodal Load Data cards.

The basic element loads consist of gravity loading (in the three global coordinate directions), and temperature increase. The temperature increase of an element is assumed to be uniform, and is calculated from

$$\Delta T = 1/2(T_i+T_j) - T_{ref} , \qquad (E.1.1)$$

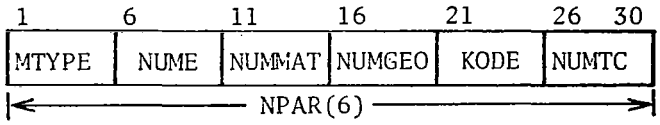
where T_i and T_j are the nodal temperatures (see Nodal Point Data), and T_{ref} is the reference temperature specified on the element cards. The element load vector due to gravity is obtained by assigning half the gravitational force acting on the element to each node.

The program can handle temperature-dependent material properties, in which case the properties should be listed for two or more temperatures. The lowest and highest of these temperatures must cover the range of temperatures listed on Nodal Point Data cards. The material properties of each element are then obtained from the listing by linear interpolation.

The output from the analysis will consist of the axial force P for each element and each load condition.

The element data deck for each construction code used must start with:

IV A. Element Control Card (6I5)



NPAR(1) = Code for element type (MTYPE). For bar elements use NPAR(1) = 1.

NPAR(2) = Number of bar elements with the specified construction code (NUME).

NPAR(3) = Number of different materials used for the specified construction code (NUMMAT).

NPAR(4) = Number of different geometric (cross-sectional) properties used for the specified construction code (NUMGEO).

NPAR(5) = Construction code (KODE).

NPAR(6) = Maximum number of temperatures for which material properties are given (NUMTC). If blank, computer sets NPAR(6) = 1.

The remaining element data depends on the construction code used, and is described separately below.

E.2 Construction Code No. 1

This construction code is designed mainly for thin-walled bars where the wall thickness only is to be changed during the optimization process. The size of the element, namely the cross-sectional area A, is then related to the moments of inertia by

$$I_y = j_y A, \quad I_z = j_z A \quad , \tag{E.2.1}$$

where j_y and j_z are the unit moments of inertia.

If P is positive (tensile), the element is redesigned with respect to the allowable tensile stress only. The stress ratio redesign formula (B.2.3) then becomes

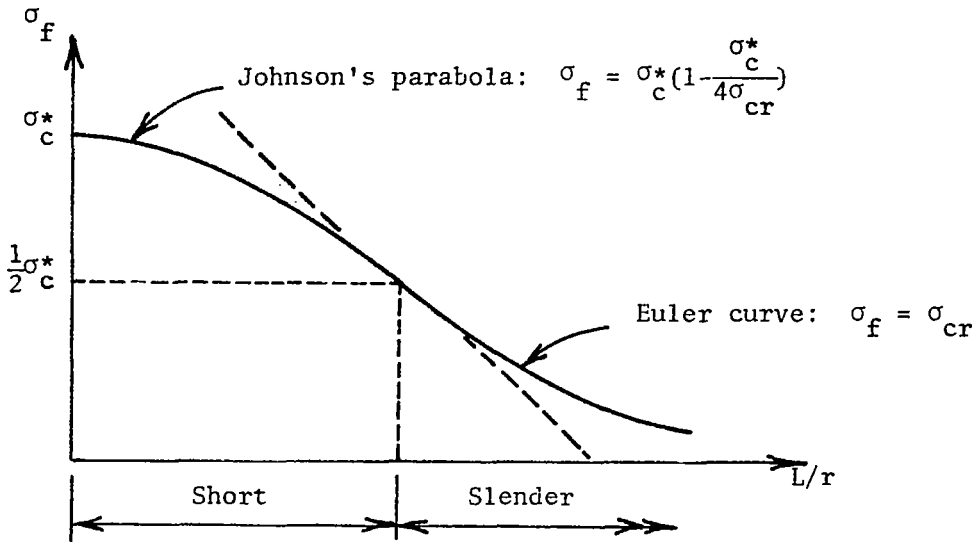
$$A'/A = P/P_t^* \quad , \tag{E.2.2}$$

where A' is the improved area and $P_t^* = \sigma_t^* A$, σ_t^* being the tensile strength.

For negative (compressive) values of P, Johnson's parabolic formula, [Sec. C1, Ref. 8] shown in Fig. E.2.1, is used as the failure criterion. For slender bars, failure is governed by Euler buckling, in which case the stress ratio redesign formula is

$$A'/A = -P/P_{cr} \quad , \tag{E.2.3}$$

where $P_{cr} = \sigma_{cr} A$ is the Euler buckling load of the current design. For short bars, the allowable stress is given by Johnson's parabola; the



σ_f = failure stress
L = length of element
r = radius of gyration of the cross section
σ^*_c = compressive strength (yield stress or crippling stress)
$\sigma_{cr} = C\pi^2E/(L/r)^2$ = Euler buckling stress
E = Young's modulus
C = end-fixity coefficient for buckling

Figure E.2.1

Johnson's Parabolic Formula

resulting stress ratio formula can be shown to be

$$A'/A = - \frac{PP_{cr}}{P_c(P^*_{cr} - P^*_c/4)} \qquad (E.2.4)$$

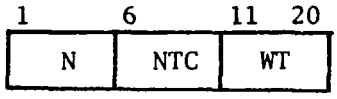
where $P^*_c = \sigma^*_c A$ and $P_{cr} = \sigma_{cr} A$. The ratio A'/A is computed about both principal axes of bending, and the larger of the values is chosen as the stress ratio of the element.

The elements may be guarded against torsional buckling (applicable to open sections), or local buckling of the wall by choosing appropriate minimum size constraints.

VI B. Material Properties Data

A separate data deck, described below, is required for each material. The decks do not have to be in a numerical sequence of material numbers.

Material Control Card (2I5, F10.0)



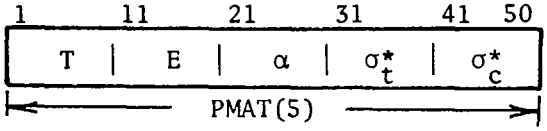
N = Material number.

NTC = Number of temperatures for which the properties of this material are given. If blank, computer sets NTC = 1.

WT = Weight-density of the material.

Material Properties Cards (5F10.0)

One card for each temperature for which the properties of this material are given. Cards must be in ascending order of temperatures.



PMAT(1) = Temperature for which the properties are given (T).

PMAT(2) = Young's modulus of elasticity (E).

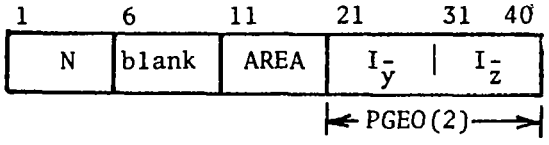
PMAT(3) = Coefficient of linear thermal expansion (α).

PMAT(4) = Tensile strength (σ_t^*).

PMAT(5) = Compressive strength (σ_c^*). If blank, computer sets PMAT(5)=PMAT(4).

VI C. Geometric Properties Data (I5, 5X, 3F10.0)

One card for each geometric property; cards do not have to be in numerical order.



N = Geometric property number.

AREA = Cross-sectional area or reference section (see below). If blank, computer sets AREA = 1.0.

PGEO(1) = Moment of inertia of the reference section about $\bar{y}$-axis ($I_{\bar{y}}$). If blank, computer sets PGEO(1) = 10.0**6, thus eliminating Euler buckling as a design consideration.

PGEO(2) = Moment of inertia of the reference section about $\bar{z}$-axis ($I_{\bar{z}}$). If blank, computer sets PGEO(2) = 10.0**6, eliminating Euler buckling.

The Geometric Properties Data is used solely for the computation of the unit moments of inertia $j_{\bar{y}}$ and $j_{\bar{z}}$. Therefore, the reference section does not have to coincide with the initial design, but can represent any acceptable intermediate design. The unit moments of inertia could, of course, be read in directly by setting AREA = 1.0 (or blank), in which case PGEO would be interpreted as the unit inertias.

VI D. Element Load Multipliers (4F10.0)

Four cards as shown below:

1	11	21	31 40	
A	B	C	D	Card 1: x-gravity
A	B	C	D	Card 2: y-gravity
A	B	C	D	Card 3: z-gravity
A	B	C	D	Card 4: thermal loading

EMUL(4,4)

EMUL = Fractions of the basic element loads (gravity loading in the three global coordinate directions and thermal loading) which are to be included in the element loads A, B, C and D.

Element Load Multipliers simply define the element loads A, B, C and D. Various multiples of these element loads can be added to each structural load condition by the use of the Structural Load Multipliers (see Sec. D.1).

VI E. Element Data (6I5, 4F10.0, I5)

One card for each element; cards must be arranged in an ascending order of element numbers.

1	6	11	16	21	26	31	41	51	61	71 75
IEL	II	JJ	IMAT	IGEO	IDV	FRC	REFT	ELPYY	ELPZZ	INC

IEL = Element number.

II = Number of node i (see Fig. E.1.1).

JJ = Number of node j.

IMAT = Material number of element.

IGEO = Geometric property number of element.

IDV = Design variable number of element.

FRC = The design variable fraction η_i in eqn. (B.1.2). If blank, computer sets FRC = 1.0.

REFT = Reference temperature---see Nodal Point Data in Sec. D.1.

ELPYY = The end-fixity coefficient C for buckling about $\bar{y}$-axis---see Fig. E.1.2.

ELPZZ = The end-fixity coefficient C for buckling about $\bar{z}$-axis.

INC = Node number increment in automatic generation of element data (see below). If blank, computer sets INC = 1.

Automatic element data generation---if there exists a series of elements IEL = m, m+1, m+2,..., which satisfies the following requirements:

(a) The node numbers form the sequences
II = i, i+INC, i+2*INC,...
JJ = j, j+INC, i+2*INC,...;

(b) The rest of the data is identical for all elements of the series,

then only the last card of the series will be required. The element data for the other elements in the series will be generated automatically.

E.3 Construction Code No. 2

This construction code assumes that all the cross-sectional dimensions are changed by the same proportion upon redesign. Consequently, (E.2.1) is replaced by

$$I_{\bar{y}} = j_{\bar{y}} A^2 \quad , \quad I_{\bar{z}} = j_{\bar{z}} A^2 \tag{E.3.1}$$

This scheme allows the user to choose the optimal proportions of the cross section (usually determined by local buckling considerations, such as crippling stress or torsional buckling), and assures him that these proportions are not changed upon redesign.

If P is tensile, (E.2.2) remains valid as the redesign formula. For compression, the quadratic size-inertia relationships result in the following modifications to (E.2.3) and (E.2.4), respectively:

$$A'/A = (-P/P_{cr})^{1/2} \quad , \tag{E.3.2}$$

$$A'/A = P_c^*/(4P_{cr}) - P/P_c^* \quad . \tag{E.3.3}$$

The input data and the remainder of the redesign procedure are identical to those used in construction code No. 1.

F. BEAM ELEMENT

F.1 General Information

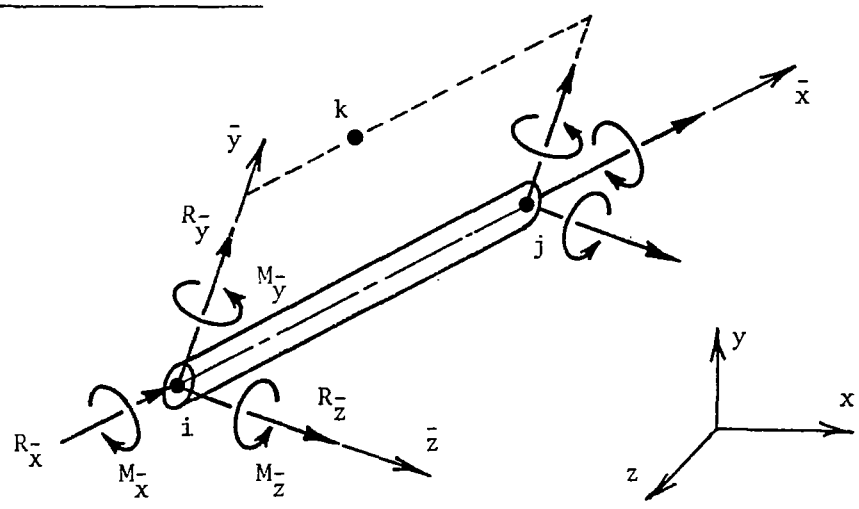


Figure F.1.1

Typical Beam Element Showing Directions of Positive End-Forces and Moments.

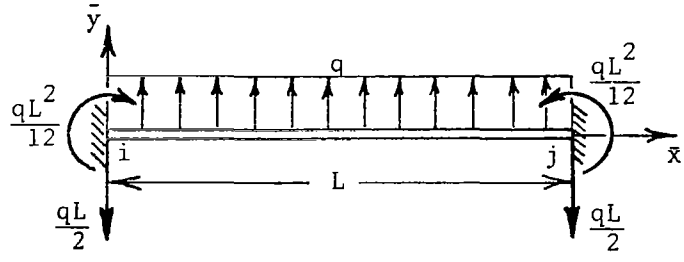
The element coordinate axes $\bar{y}$ and $\bar{z}$ shown in Fig. F.1.1, are the principal axes of the cross section. The direction of the $\bar{y}$-axis is defined by the plane of i-j-k, where k is a "third" nodal point that must be specified in addition to the nodes i and j. The node k may be a conventional nodal point of the structure, or a point used solely for defining the principal axes. In the latter case, the motion code on the node data card should be used to eliminate all degrees of freedom for node k.

Cubic polynomials in $\bar{x}$ are used for the lateral displacements, whereas the axial displacement is assumed to be linear as in the bar element. The bending moments are thus confined to linear functions

of $\bar{x}$. It follows that the finite element analysis of beam elements is exact (apart from numerical errors) for elements that carry concentrated nodal forces or moments only.

The basic element loads are gravity loads in the three global coordinate directions, and fixed-end forces referred to the element coordinate directions. Neither thermal stresses nor temperature-dependent material properties are included in the current version of the program.

Distributed lateral loads are specified in the form of fixed-end forces and moments. A fixed-end force (or moment) is simply the end force (or moment) acting in a clamped-clamped beam subjected to the specified lateral loading. An example on the use of fixed-end forces is shown in Fig. 1.2.



	$R_{\bar{x}}$	$R_{\bar{y}}$	$R_{\bar{z}}$	$M_{\bar{x}}$	$M_{\bar{y}}$	$M_{\bar{z}}$
Fixed-end forces for node i	0	$-\frac{qL}{2}$	0	0	0	$-\frac{qL^2}{12}$
Fixed-end forces for node j	0	$-\frac{qL}{2}$	0	0	0	$\frac{qL^2}{12}$

Figure F.1.2

Example on the Use of Fixed-End Forces.

The use of fixed-end forces results in the correct end-forces (and moments) in a beam element, but the distribution of the internal forces (and moments) is generally inaccurate between the ends.

Any degree of freedom of an end-node (such as i or j) can be connected to the degrees-of-freedom of a "master" node m. If such a connection is present, the number of the master node should be used as the motion code for the "slave" node (see Nodal Point Data in Sec. D.1). Note that this scheme precludes using node number 1 for the master node.

If the x-displacement of node i is specified as a slave of node m, the displacement is expressed in terms of the degrees-of-freedom of the master node as follows:

$$u_{xi} = u_{xm} + (z_i - z_m)\theta_{ym} - (y_i - y_m)\theta_{zm} \ . \qquad \text{(F.1.1)}$$

The corresponding formulas for the y and z-displacements can be obtained from (F.1.1) by cyclic permutation of x,y,z.

If the rotation about x-axis of node i is specified to be the slave degree-of-freedom, the computer sets

$$\theta_{xi} = \theta_{xm} \qquad \text{(F.1.2)}$$

Analagous equations are used for slave rotations about the y and z-axis.

Slave-master relationship is used to represent various rigid connections between two nodes. The edge-reinforced plate in Fig. F.1.3 can be offered as an example. By specifying all the degrees-of-freedom of node j to be the slaves of node m, a rigid link is created between the two nodes. The link enforces displacement continuity between the

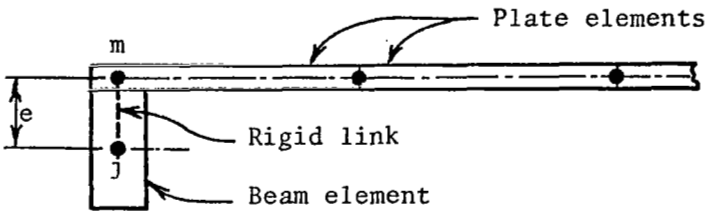


Figure F.1.3

Example of Master and Slave Nodes.

beam and plate element, but at the same time it accounts for the inter-element eccentricity e.

It is important to note that the eccentricity e is kept constant throughout the redesign procedure, i.e., no allowance is made for the changes in e caused by redesign.

The printed output following each analysis consists of the end-forces and moments shown in Fig. F.1.1.

The first card of the element data deck for each construction code used must be:

VI A. Element Control Card (6I5)

1 6	11	16	21	26 30
MTYPE | NUME	NUMMAT	NUMGEO	KODE	NUMFX
←——— NPAR(6) ———→				

NPAR(1) = Code for element type (MTYPE). For beam elements use NPAR(1) = 2.

NPAR(2) = Number of elements with the specified construction code (NUME).

NPAR(3) = Number of different materials used for the specified construction code (NUMMAT).

NPAR(4) = Number of different geometric (cross-sectional) properties used for the specified construction code (NUMGEO).

NPAR(5) = Construction code (KODE).

NPAR(6) = Number of different fixed-end force sets used for the specified construction code (NUMFX).

The rest of the element data is dependent on the specified construction code, and is listed separately below.

F.2 Construction Code No. 1

The size A of the element is taken as the cross-sectional area, and the size-stiffness relationships are assumed to be linear:

$$\bar{I}_x = \bar{j}_x A, \quad \bar{I}_y = \bar{j}_y A, \quad \bar{I}_z = \bar{j}_z A \quad , \tag{F.2.1}$$

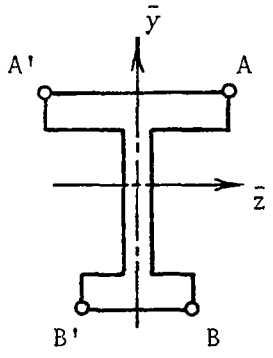
where $\bar{I}_x$ is the torsional constant of the cross section, $\bar{I}_y$ and $\bar{I}_z$ are the principal moments of inertia, and $\bar{j}_x$, $\bar{j}_y$, $\bar{j}_z$ are the size-independent unit values.

This construction code is intended primarily for the use of thin-walled sections where the wall thickness only is to be varied during design. Equations (F.2.1) represent a good approximation for closed sections, provided that the wall thickness is not too large. For open sections, the expressions for $\bar{I}_y$ and $\bar{I}_z$ are still valid, but the torsional constant has the form $\bar{I}_x = \bar{j}_x A^3$. It must be noted, however, that $\bar{I}_x$ is very small in comparison to $\bar{I}_y$ and $\bar{I}_z$ for thin-walled, open sections. Therefore, its contribution to the rigidity of the structure can be neglected in most cases, i.e. $\bar{j}_x = 0$ may be used.

The construction code is also valid for plane bending of beams with rectangular cross sections where the width of the cross section is subject to design.

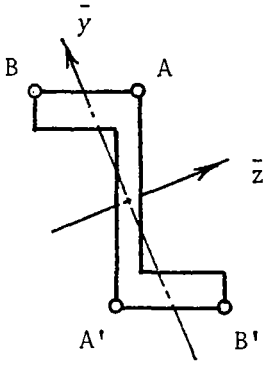
The stress ratio redesign formula is based on the allowable stresses only; instability criteria are not used. Local instabilities could be guarded against by choosing adequate minimum size constraints.

The stresses are calculated at four points at each end section of the element, as indicated in Fig.F.2.1 by points A, B, A' and B'. Locations of A and B on the cross section are chosen by the user; points A' and B' are located by the computer to be in "mirror image" position of A and B.



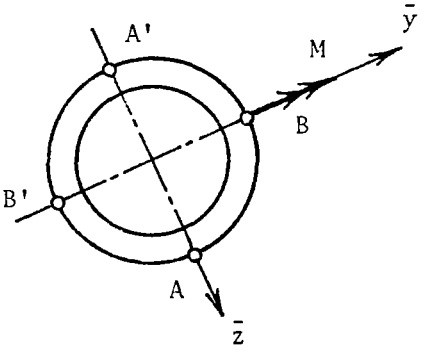
Symmetric

KSEC = 1



Zee

KSEC = 2



Circular

KSEC = 3

Figure F.2.1

Allowable Families of Cross Sections.

The input is limited to three families of cross sections:

(a) Cross sections with at least one axis of symmetry. It is also advisable to have the centroid and the shear center coincide, since the program takes the center of twist to be the centroid of the cross section. The axis of symmetry must be specified as the $\bar{y}$-axis.

(b) Zee section with identical flanges. The $\bar{y}$-axis must be specified as the principal axis that passes through the centroids of the flanges as shown in Fig. F.2.1.

(c) Circular cross section. The orientation of $\bar{y}$ and $\bar{z}$ axes is completely arbitrary; the computer will automatically rotate the axes such that the $\bar{y}$-axis will coincide with direction of the resultant moment M (see Fig. F.2.1).

The user must specify the section moduli at points A and B only; the computer will automatically generate the moduli for A' and B' as shown below.

(a) Symmetric sections:

$$(Z_{\bar{y}})_{A'} = -(Z_{\bar{y}})_A, \quad (Z_{\bar{z}})_{A'} = (Z_{\bar{z}})_A, \quad (Z_{\bar{x}})_{A'} = (Z_{\bar{x}})_A \ ,$$
$$(Z_{\bar{y}})_{B'} = -(Z_{\bar{y}})_B, \quad (Z_{\bar{z}})_{B'} = (Z_{\bar{z}})_B, \quad (Z_{\bar{x}})_{B'} = (Z_{\bar{x}})_B \ . \qquad \text{(F.2.1)}$$

(b) Zee sections:

$$(Z_{\bar{y}})_{A'} = -(Z_{\bar{y}})_A, \quad (Z_{\bar{z}})_{A'} = -(Z_{\bar{z}})_A, \quad (Z_{\bar{x}})_{A'} = (Z_{\bar{x}})_A \ ,$$
$$(Z_{\bar{y}})_{B'} = -(Z_{\bar{y}})_B, \quad (Z_{\bar{z}})_{B'} = -(Z_{\bar{z}})_B, \quad (Z_{\bar{x}})_{B'} = (Z_{\bar{x}})_B \ . \qquad \text{(F.2.2)}$$

The normal stresses due to bending and axial load are computed from the formulas

$$\sigma_B = \mp (M_{\bar{y}}/Z_{\bar{y}} - M_{\bar{z}}/Z_{\bar{z}}) \ , \tag{F.2.3}$$

$$\sigma_A = \mp R_{\bar{x}}/A \ , \tag{F.2.4}$$

respectively, where the minus sign applies to node i, and the plus sign to node j. The shear stress due to torsion is obtained from

$$\tau = M_{\bar{x}}/Z_{\bar{x}} \ . \tag{F.2.5}$$

The stress ratio redesign formula is based on the modified Von Mises yield criterion

$$(\sigma/\sigma^*)^2 + (\tau/\tau^*)^2 = 1 \ , \tag{F.2.6}$$

where $\sigma = \sigma_A + \sigma_B$, and σ^*, τ^* are the allowable stresses. If $\sigma > 0$, σ^* is taken as the allowable tensile stress σ_t^*; if $\sigma < 0$, σ^* represents the allowable compressive stress σ_c^*. It should be noted that (F.2.6) reduces to the original Von Mises criterion if $\sigma_t^* = \sigma_c^*$ and $\tau^* = \sigma_t^*/\sqrt{3}$.

Because only the wall thickness changes during design, the section moduli have the form

$$Z_{\bar{x}} = z_{\bar{x}}A \ , \quad Z_{\bar{y}} = z_{\bar{y}}A \ , \quad Z_{\bar{z}} = z_{\bar{z}}A \ , \tag{F.2.7}$$

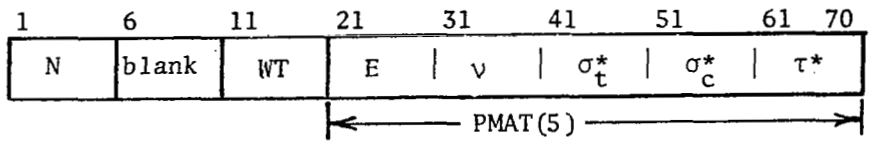
where $z_{\bar{x}}$, $z_{\bar{y}}$, and $z_{\bar{z}}$ are the size-independent unit section moduli. Utilizing (F.2.7) and (F.2.3) to (F.2.6) in the failure criterion (F.2.6), we obtain for the stress ratio redesign formula the following equation:

$$\frac{A'}{A} = \left[\left(\frac{\sigma_A+\sigma_B}{\sigma^*}\right)^2 + \left(\frac{\tau}{\tau^*}\right)^2\right]^{1/2} \ , \tag{F.2.8}$$

where σ_A, σ_B, and τ are the stresses of the current design A.

VI B. Material Properties Data (I5, 5X, 6F10.0)

One card for each material used; the cards do not have to be in numerical sequence.



N = Material number.

WT = Weight-density of the material.

PMAT(1) = Young's modulus of elasticity (E).

PMAT(2) = Poisson's ratio (ν).

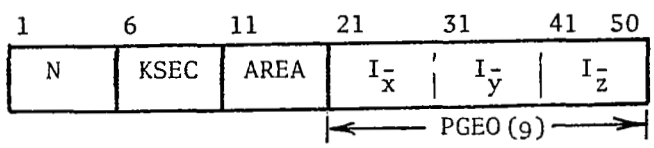
PMAT(3) = Allowable tensile stress (σ_t^*).

PMAT(4) = Allowable compressive stress (σ_c^*). If blank, computer sets PMAT(4) = PMAT(3).

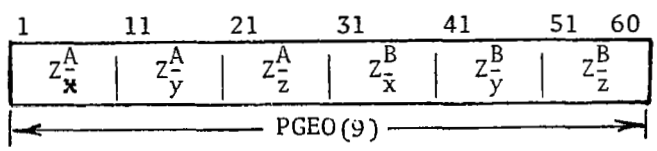
PMAT(5) = Allowable shear stress (τ*). If blank, computer sets PMAT(5) = 0.577*PMAT(3).

VI C. Geometric Properties Data (2I5, 4F10.0/6F10.0)

Two cards for each geometric property; cards do not have to be in numerical sequence.



Card 1



Card 2

N = Geometric property number.

KSEC = Code for cross section (see Fig. F.2.1).
KSEC = 1 or blank: symmetric section.
KSEC = 2: Zee section.
KSEC = 3: circular section.

AREA = Cross-sectional area of reference section.
If blank, computer sets AREA = 1.0.

PGEO(1) = Torsional constant of the reference section ($I_{\bar{x}}$).

PGEO(2) = Moment of inertia of the reference section about $\bar{y}$-axis ($I_{\bar{y}}$).

PGEO(3) = Moment of inertia of the reference section about $\bar{z}$-axis ($I_{\bar{z}}$).
If KSEC = 3 (circular section), computer sets
PGEO(3) = PGEO(2).

PGEO(4) = Section modulus of point A of the reference section for torsion ($Z_{\bar{x}}^{A}$).

PGEO(5) = Section modulus of point A of the reference section for bending about $\bar{y}$-axis ($Z_{\bar{y}}^{A}$).

PGEO(6) = Section modulus of point A of the reference section for bending about $\bar{z}$-axis ($Z_{\bar{z}}^{A}$).

PGEO(7) = Section modulus of point B of the reference section for torsion ($Z_{\bar{x}}^{B}$).

PGEO(8) = Section modulus of point B of the reference section for bending about $\bar{y}$-axis ($Z_{\bar{y}}^{B}$).

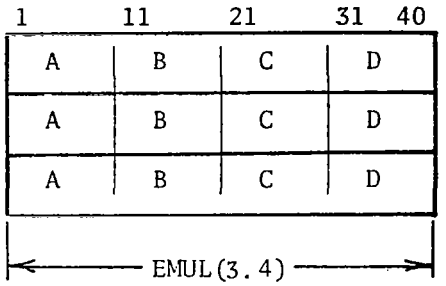
PGEO(9) = Section modulus of point B of the reference section for bending about $\bar{z}$-axis ($Z_{\bar{z}}^{B}$).

Note: If PGEO(i), i = 4,5,...,9 are left blank, the corresponding torsional or bending stress is set to zero in the analysis. For KSEC = 3 (circular section), stresses are calculated at points A and A' only; consequently PGEO(6) to PGEO(9) are set to zero by the computer.

The Geometric Properties Data is used only for the computation of unit inertias and section moduli. The reference section thus does not have to be the initial design.

VI D. Element Load Multipliers (4F10.0)

Three cards as shown below.



Card 1: x-gravity

Card 2: y-gravity

Card 3: z-gravity

EMUL = Fractions of basic element loads (gravity loads in the global coordinate directions) which are to be included in the element loads A, B, C and D.

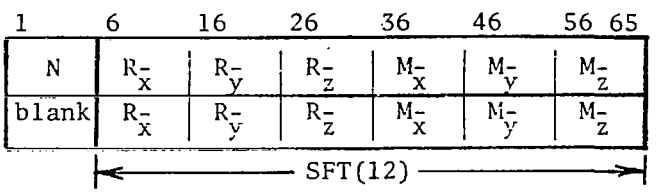
Lumped nodal forces are used to represent gravity loads, as was done for Bar Elements (see Sec. E.1). Fixed-end forces are not computed.

The element load multipliers define the contribution of gravity to element loads A, B, C and D. Additional contribution to the element loads is made by the fixed-end forces; this information is specified separately by Fixed-End Force Data and Element Data.

Specified multiples of element loads A, B, C and D can be added to each structural load condition by the use of Structural Load Multipliers (see Sec. D.1).

VI E. Fixed-End Force Data (I5, 6F10.0/5X, 6F10.0)

Two cards for each set of fixed-end forces used in the analysis; cards do not have to be in numerical order. If no fixed-end forces are present (NUMFX = 0), omit all cards.



Card 1: node i

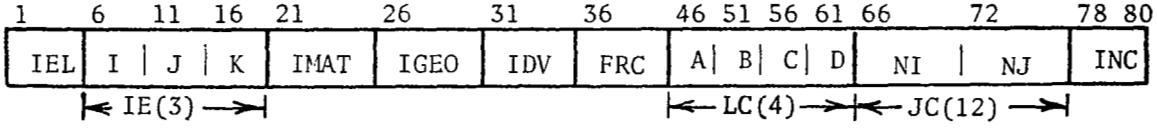
Card 2: node j

N = Fixed-end force set number.

SFT = Fixed-end forces and moments. Use the sign convention shown in Fig. F.1.1.

VI F Element Data (7I5, F10.0, 4I5, 2I6, I3)

One card for each element; cards must be in an ascending order of element numbers.



IEL = Element number.

I = Number of node i (see Fig. F.1.1).

J = Number of node j.

K = Number of node k. This node defines the direction of $\bar{y}$-axis, and should not lie on the line of i-j.

IMAT = Material number of element.

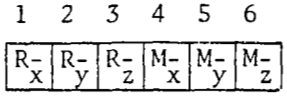
IGEO = Geometric property number of element.

IDV = Design variable number of element.

FRC = The design variable fraction η_i in eqn. (B.1.2). If blank, computer sets FRC = 1.0.

LC(4) = Numbers of fixed-end force sets that are to be included in element loads A, B, C and D, respectively (also see Element Load Multipliers).

NI = End release code for node i. The end release code consists of a six digit number, each digit corresponding to an end-force as shown below.



If any of the end-forces is known to be zero, due to the presence of a hinge or a roller, the digit one must be used; otherwise the digit should be left blank.

NJ = End release code for node j.

INC = Node number increment in automatic generation of element data (see below). If blank, computer sets INC = 1.

Automatic element data generation---if there exists a series of elements IEL = m, m+1, m+2,..., which satisfies the following requirements:

(a) The numbers of the end-nodes form the sequences
I=i, i+INC, i+2*INC,...,
J=j, j+INC, j+2*INC,...;
(b) The rest of the data is identical to all elements of the series, including the node number K,

then only the last card of the series will be required. The element data for the other elements of the series will be generated automatically.

F.3 Construction Code No. 2

In this construction code all the dimensions of the cross section are scaled by the same factor upon redesign. This scheme gives the user an opportunity to choose optimal proportions for the cross section, and maintain these proportions throughout the design process.

The expressions for the bending and torsional properties of the cross section now take form

$$I_{\bar{x}} = j_{\bar{x}} A^2, \quad I_{\bar{y}} = j_{\bar{y}} A^2, \quad I_{\bar{z}} = j_{\bar{z}} A^2 \quad , \tag{F.3.1}$$

$$Z_{\bar{x}} = z_{\bar{x}} A^{3/2}, \quad Z_{\bar{y}} = z_{\bar{y}} A^{3/2}, \quad Z_{\bar{z}} = z_{\bar{z}} A^{3/2} \quad , \tag{F.3.2}$$

and the formula for stress ratio redesign becomes

$$\left(\frac{A'}{A}\right)^3 = \left(\frac{\sigma_A}{\sigma^*}\sqrt{\frac{A'}{A}} + \frac{\sigma_B}{\sigma^*}\right)^2 + \left(\frac{\tau}{\tau^*}\right)^2 \quad . \tag{F.3.3}$$

This equation is solved iteratively for A'/A in the manner described below.

Let $r^{\nu} = (A'/A)^{\nu}$ be the ratio obtained from the previous iteration (iteration ν), and $r^{\nu+1} = (A'/A)^{\nu+1}$ the improved value predicted by the current iteration. If

$$\frac{2|\sigma_A \sigma_B|}{(\sigma^*)^2} \geq \left(\frac{\tau}{\tau^*}\right)^2 \quad , \tag{F.3.4}$$

the improved value is obtained from the solution of the cubic equation in $(r^{\nu+1})^{1/2}$:

$$\sqrt{1 - \left(\frac{\tau}{\tau^*}\right)^2 (r^{\nu})^{-3}}\ (r^{\nu+1})^{3/2} - \frac{\sigma_A}{\sigma^*}(r^{\nu+1})^{1/2} - \frac{\sigma_B}{\sigma^*} = 0 \ . \tag{F.3.5}$$

If (F.3.4) is not satisfied, the iteration formula is taken as cubic equation in $r^{\nu+1}$:

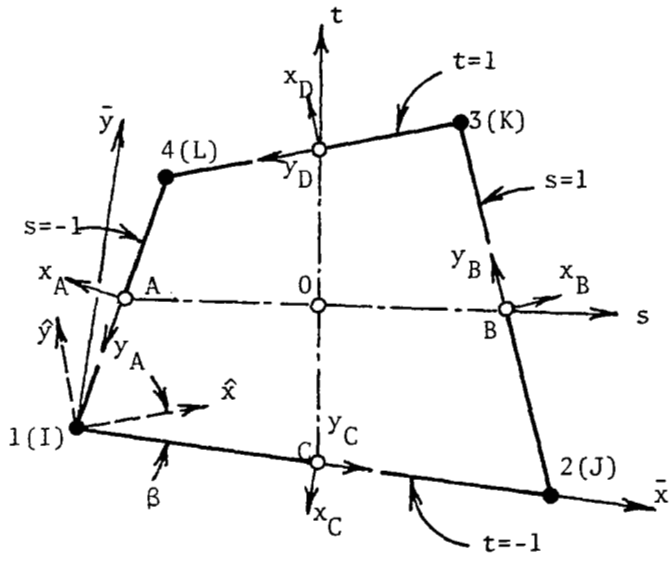
$$(r^{\nu+1})^3 - \frac{\sigma_A}{\sigma^*} r^{\nu+1} - \left[\left(\frac{\sigma_B}{\sigma^*}\right)^2 + \left(\frac{\tau}{\tau^*}\right)^2 + \frac{2\sigma_A\sigma_B}{(\sigma^*)^2}(r^{\nu})^{1/2}\right] = 0 \qquad \text{(F.3.6)}$$

If $\tau = 0$, (F.3.5) yields an exact solution of (F.3.3). Similarly, (F.3.6) is an exact redesign equation if $\sigma_A = 0$ or $\sigma_B = 0$. In both cases only one iterative cycle is needed. Otherwise the number of iterations is limited to six in each redesign cycle, i.e., the results of the sixth iteration are used for the new design even if the iterative solution failed to reach convergence.

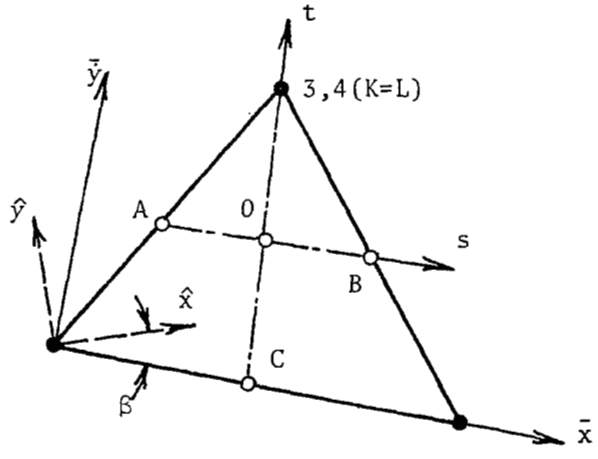
The input data and the remaining details of analysis and redesign procedures are identical to those used for Construction Code No. 1.

G. PLANE STRESS QUADRILATERAL OR TRIANGULAR ELEMENT

G.1 General Information



(a)



(b)

Figure G.1.1

Quadrilateral and Triangular Plane Stress Elements.

The basic plane stress element is the quadrilateral shown in Fig. G.1.1(a). The nodes 1 and 2 determine the local Cartesian coordinate axes $\bar{x}$ and $\bar{y}$, and the relative orientation of the natural element coordinates s and t in the manner indicated on the drawing. Note that the natural element coordinates take the values +1 or -1 at the sides of the element. All four nodes of the element should be on the same plane. If the nodes are not co-planar errors will occur in the computation of element properties. The magnitude of the error becomes more serious with increasing warp of the element.

The displacement field within the element is taken in the form

$$\begin{aligned}
\bar{u}(s,t) &= \sum_{i=1}^{4} h_i(s,t)\bar{u}_i + h_5(s,t)\alpha_1 + h_6(s,t)\alpha_2 \ , \\
\bar{v}(s,t) &= \sum_{i=1}^{4} h_i(s,t)\bar{v}_i + h_5(s,t)\alpha_3 + h_6(s,t)\alpha_4 \ , \\
\bar{w}(s,t) &= \sum_{i=1}^{4} h_i(s,t)\bar{w}_i \ ,
\end{aligned} \tag{G.1.1}$$

where $\bar{u}$, $\bar{v}$ and $\bar{w}$ are the components of the displacement field in the direction of the local Cartesian coordinate axes, $\bar{u}_i$, $\bar{v}_i$ and $\bar{w}_i$ denote the displacement components of node i, and α_j are generalized coordinates in addition to the nodal displacements. The shape functions in (G.1.1) are

$$\begin{aligned}
h_1 &= \tfrac{1}{4}(1-s)(1-t) \qquad & h_2 &= \tfrac{1}{4}(1+s)(1-t) \\
h_3 &= \tfrac{1}{4}(1+s)(1+t) \qquad & h_4 &= \tfrac{1}{4}(1-s)(1+t) \\
h_5 &= \tfrac{1}{4}(1-s)^2 \qquad & h_6 &= \tfrac{1}{4}(1-t)^2 \ .
\end{aligned} \tag{G.1.2}$$

The functions h_1 to h_4 are Lagrangian interpolation polynomials; they result in a linear displacement field along each side of the element, and thus maintain displacement continuity between the elements. The functions h_5 and h_6, on the other hand, represent incompatible displacement modes, because they cause the sides of the elements to be deformed into parabolas.

According to the examples in [2], the introduction of the incompatible modes can lead to a significant improvement of the prebuckling analysis for certain problems.

Note that h_5 and h_6 vanish at the nodes. Consequently, α_j's are internal degrees of freedom, which can be eliminated at the element level by the static condensation procedure. The user of the program, however, has the option of suppressing the incompatible modes altogether, in which case the computer would set $\alpha_j = 0$, $j = 1,2,3,4$.

All the area integrals in the computation of element properties are carried out numerically, using Legendre-Gauss quadrature with four integration points. It can be shown that the numerical integral is exact only for rectangular elements. Since the integration error increases with the skewness of the element, highly skewed quadrilaterals are to be avoided if possible.

The triangular element used in the program is simply a limiting case of the quadrilateral when two of the nodes are allowed to coincide, as seen in Fig. G.1.1(b). It requires no special provisions within the program.

The basic element loads are gravitational forces in the three coordinate directions, thermal loading, and in-plane pressure acting on

one side of the element (the pressure loading is not allowed for the undirectionally reinforced element described under Construction Code 1).

Temperature-dependent material properties may be used in the program, in which case the material properties should be specified for several temperatures. The material properties of each element are obtained from the specified properties by linear interpolation. The highest and lowest temperatures at which the material properties are prescribed should cover the entire range of element temperatures.

The output of each analysis cycle consists of the in-plane stress resultants at the origin of the natural coordinates---point 0 in Fig. G.1.1(a), and at points A, B, C, and D. The stress resultants at point 0 are printed out twice: once with respect to the local Cartesian coordinates $\bar{x}$ and $\bar{y}$, and once with respect to the coordinates $\hat{x}$ and $\hat{y}$ defined by the angle β in Fig. G.1.1. The mid-side forces are expressed in terms of the directions x_A, y_A, etc. shown in Fig. G.1.1(a). The convention for positive stress resultants is given in Fig. G.1.2.

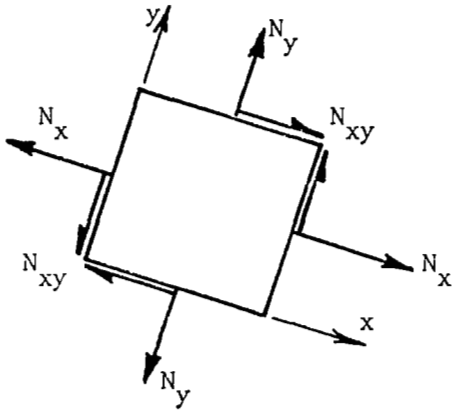


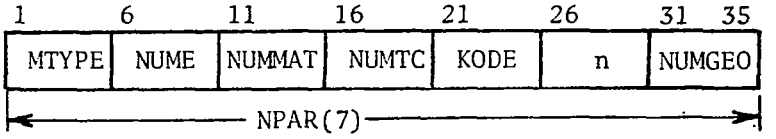
Figure G.1.2

Positive Stress Resultants

By using an appropriate value of the stress-printout option code NS, the user can suppress the computation of some or all of the stresses.

The element data deck for each construction code must start with:

VI A. Element Control Card



NPAR(1) = Code for element type (MTYPE). For plane stress elements use NPAR(1) = 3.

NPAR(2) = Number of elements with the specified construction code (NUME).

NPAR(3) = Number of different materials used for the given construction code (NUMMAT).

NPAR(4) = Maximum number of temperatures for which material properties are given (NUMTC). If blank, computer sets NPAR(4) = 1.

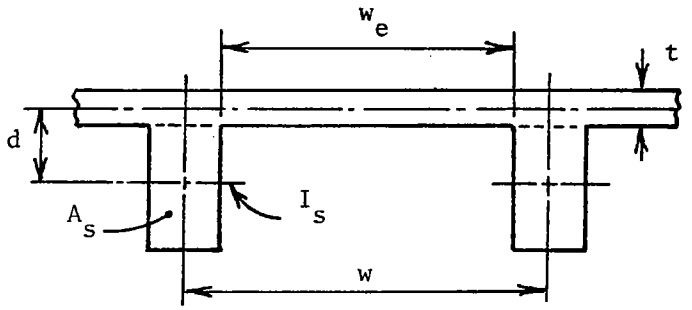
NPAR(5) = Construction code (KODE).

NPAR(6) = Code for use of incompatible displacement modes (n).
NPAR(6) = 0: use incompatible modes.
NPAR(6) > 0: suppress incompatible modes.

NPAR(7) = Number of genetic properties (not used for Construction Code No. 2).

G.2 Construction Code No. 1

This construction code deals with unidirectionally stiffened panels. All the cross-sectional dimensions shown in Fig. G.2.1, including the stiffener spacing w, are assumed to change by the same proportion during redesign.



Note: A_s and I_s are the cross-sectional area and moment of inertia of the stiffener, respectively.

Figure G.2.1

Cross Section of Unidirectionally Stiffened Panel.

The sheet thickness t is chosen as the size of the element, i.e., it is taken as the independent dimension of the cross section.

For the purposes of analysis, the panel is treated as a homogeneous, orthotropic slab with equivalent extensional and shear flexibilities:

$$\begin{Vmatrix} \varepsilon_{\hat{x}} \\ \varepsilon_{\hat{y}} \\ \gamma_{\hat{x}\hat{y}} \end{Vmatrix} = \frac{1}{E} \begin{Vmatrix} 1/\bar{t} & -\nu/\bar{t} & 0 \\ -\nu/\bar{t} & 1/t & 0 \\ 0 & 0 & 2(1+\nu)/t \end{Vmatrix} \begin{Vmatrix} N_{\hat{x}} \\ N_{\hat{y}} \\ N_{\hat{x}\hat{y}} \end{Vmatrix} \qquad \text{(G.2.1)}$$

In (G.2.1), $\hat{x}$ and $\hat{y}$ are the principal material coordinates shown in Fig. G.2.2; $N_{\hat{x}}$, $N_{\hat{y}}$ and $N_{\hat{x}\hat{y}}$ represent the average membrane stress resultants; and

$$\bar{t} = t + A_s/w$$

is the "average" thickness of the panel.

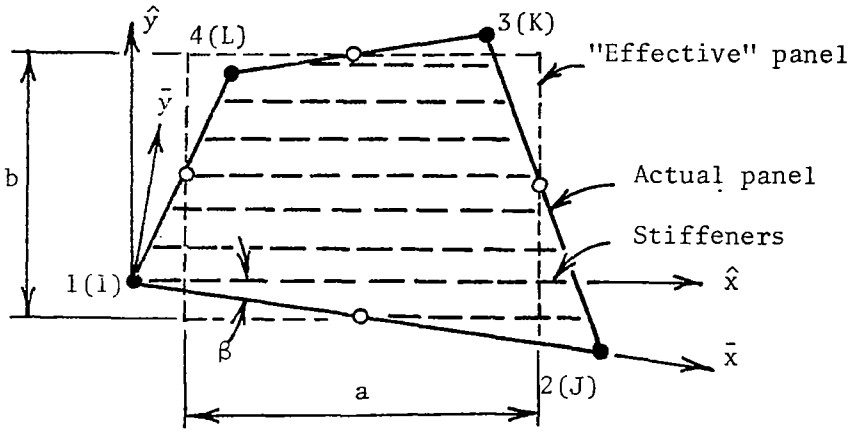


Figure G.2.2

Unidirectionally Stiffened Panel

The following possible modes of failure are considered in redesign operation:

(1) stresses exceeding their allowable values in the sheet;

(2) general buckling of the panel;

(3) buckling of the sheet between the stiffeners;

(4) failure of stiffeners.

The stress resultants acting at the middle of the panel (s = 0, t = 0) only are used in evaluating these failure modes.

Other types of local buckling failure can be handled simply by choosing suitable cross-sectional proportions of the panel, i.e., proportioning the reinforcement such that sheet buckling would always occur prior to (or simultaneously with) the local buckling. The user may even optimize the cross-sectional proportions by maximizing the configuration-efficiency coefficient of the panel in a manner similar to Ref. [9], p. 107.

(1) Stress limits

The modified Von Mises yield criterion

$$(\sigma_{\hat{x}}/\sigma^*)^2 + (\sigma_{\hat{y}}/\sigma^*)^2 - \sigma_{\hat{x}}\sigma_{\hat{y}}/(\sigma^*)^2 + (\tau_{\hat{x}\hat{y}}/\tau^*)^2 = 1 \qquad \text{(G.2.2)}$$

is used for the stress-constrained design of the sheet, where σ^* and τ^* are allowable normal and shear stresses, respectively. Different allowable stresses may be specified for tension (σ_t^*) and compression (σ_c^*). Equation(G.2.2) reduces to the original Von Mises yield criterion only if $\sigma_t^* = \sigma_c^*$ and $\tau^* = \sigma^*/\sqrt{3}$.

Using (G.2.2), the stress ratio redesign formula can be shown to be

$$t'/t = [(N_{\hat{x}}/N_{\hat{x}}^*)^2 + (N_{\hat{y}}/N_{\hat{y}}^*)^2 - N_{\hat{x}}N_{\hat{y}}/(N_{\hat{x}}^*N_{\hat{y}}^*) + (N_{\hat{x}\hat{y}}/N_{\hat{x}\hat{y}}^*)^2]^{1/2}, \qquad \text{(G.2.3)}$$

where $N_{\hat{x}}^* = \sigma^*\bar{t}$, $N_{\hat{y}}^* = \sigma^* t$ and $N_{\hat{x}\hat{y}}^* = \tau^* t$.

(2) General buckling of panel

The panel is treated as a homogeneous, orthotropic plate with simple supports. It is assumed that only the compressive stress $\sigma_{\hat{x}}$ in the direction of the stiffeners influences buckling.

Since the buckling formula for a general quadrilateral is not available, the panel is treated as a rectangle with the "effective" width b shown in Fig. G.2.2. The effective width is computed automatically;

however, the user has the option of substituting his own value.

According to Ref. [8], Sec. C.2, p. 26, the compressive buckling stress of a stiffened plate is

$$(\sigma_{\hat{x}})_{cr} = \frac{k\pi^2 E}{12(1-\nu^2)} \left(\frac{t}{b}\right)^2 \quad , \tag{G.2.4}$$

where the buckling coefficient is given by

$$k = 2 \frac{\left\{ 1 + \frac{N-1}{N} \frac{EI_s}{wD} \left[1 + \frac{A_s d^2 I_s}{1+0.88A_s/(wt)} \right] \right\}^{1/2} + 1}{N^2 \left(\frac{N-1}{N} \frac{A_s}{wt} + 1 \right)} \quad . \tag{G.2.5}$$

In (G.2.5), N is the number of bays between stiffeners, and

$$D = \frac{Et^3}{12(1-\nu^2)}$$

The remaining symbols are defined in Fig. G.2.1.

For large numbers of stiffeners, we can approximate (N-1)/N = 1 and N = b/w, and obtain for the critical stress resultant

$$(N_{\hat{x}})_{cr} = \frac{k_1 \pi^2 E}{12(1-\nu^2)b^4} t^5 \tag{G.2.6}$$

where

$$k_1 = 2\left(\frac{w}{t}\right)^2 \left\{ \left[1 + \frac{12(1-\nu^2) I_s}{wt^3} \left(1 + \frac{A_s d^2/I_s}{0.12+0.88\bar{t}/t} \right) \right]^{1/2} + 1 \right\} \quad . \tag{G.2.7}$$

Note that the coefficient k_1 does not change upon redesign since the cross-sectional proportions are kept constant. The stress ratio redesign formula hence becomes

$$t'/t = [-N_{\hat{x}}/(N_{\hat{x}})_{cr}]^{1/5} \tag{G.2.8}$$

(3) Buckling of sheet between stiffeners

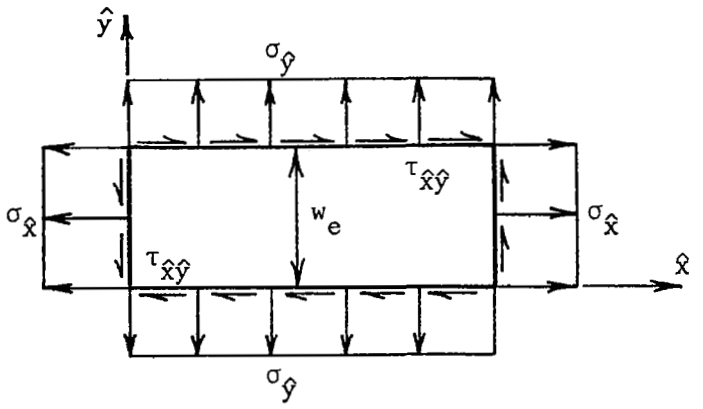


Figure G.2.3

Sheet Between Stiffeners.

The sheet between stiffeners, shown in Fig. G.2.3, is treated as a long, simply supported plate of effective width w_e. The general form of the compressive buckling stress for each of the three load conditions shown is

$$\sigma_{cr} = k \frac{\pi^2 E}{12(1-\nu^2)} \left(\frac{t}{w_e}\right)^2 \qquad \text{(G.2.9)}$$

For simple supports, the buckling coefficients for longitudinal compression, transverse compression and shear are, respectively

$$k_{\hat{x}} = 4.00, \quad k_{\hat{y}} = 1.00, \quad k_{\hat{x}\hat{y}} = 5.35. \qquad \text{(G.2.10)}$$

If all loads are applied simultaneously, the buckling criterion (interaction formula) is taken as

$$-\sigma_{\hat{x}}/(\sigma_{\hat{x}})_{cr} - \sigma_{\hat{y}}/(\sigma_{\hat{y}})_{cr} + \tau^2_{\hat{x}\hat{y}}/(\tau_{\hat{x}\hat{y}})^2_{cr} = 1 \quad . \qquad \text{(G.2.11)}$$

The interaction formula has an exact theoretical basis only when $\sigma_{\hat{y}} = 0$ or $\tau_{\hat{x}\hat{y}} = 0$; otherwise is an approximation.

The stress ratio redesign formula obtainable from (G.2.9) to (G.2.11) is

$$\frac{t'}{t} = \frac{-N_{\hat{x}} - 4(t/\bar{t})N_{\hat{y}} + \{[N_{\hat{x}} + 4(t/\bar{t})N_{\hat{y}}]^2 + [1.495(t/\bar{t})N_{\hat{x}\hat{y}}]^2\}^{1/2}}{2(N_{\hat{x}})_{cr}} , \quad (G.2.12)$$

where

$$(N_{\hat{x}})_{cr} = (\sigma_{\hat{x}})_{cr}\bar{t} .$$

Equation (G.2.12) remains valid if $N_{\hat{x}}$ and $N_{\hat{y}}$ are positive (tensile).

(4) Failure of stiffeners

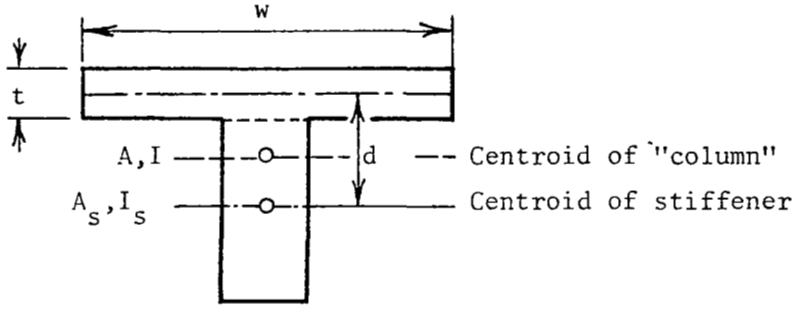


Figure G.2.4

Stiffener and Sheet Treated as a Column.

A stiffener and the attached sheet (see Fig. G.2.4) are treated as a column of length "a". The value of "a" is calculated by the computer in the manner indicated in Fig. G.2.2, or it may be user-specified.

If $N_{\hat{x}}$ is positive (tensile), the failure analysis is skipped. For compressive loading, the Euler-Johnson failure criterion shown in Fig. E.2.1 is used.

The cross-sectional area of the column in Fig. G.2.4 is

$$A = A_s + wt \tag{G.2.13}$$

and its moment of inertia can be shown to be

$$I = wt^3/12 + I_s + d^2[wt(A_s/A)^2 + A_s(1-A_s/A)^2] \ . \tag{G.2.14}$$

The unit moment of inertia, which is invariant of the value of t, can be calculated from

$$j = I/t^4 \ . \tag{G.2.15}$$

For long columns that fail by Euler buckling, the critical compressive stress resultant is

$$N_{cr} = \frac{C\pi^2 EI}{a^2 w} = \frac{C\pi^2 Ej}{a^2(w/t)}\, t^3 \ , \tag{G.2.16}$$

from which we obtain the redesign formula

$$t'/t = (-N_{\hat{x}}/N_{cr})^{1/3} \ . \tag{G.2.17}$$

Johnson's parabola, which governs the failure of short columns, leads to the following quadratic equation in t'/t:

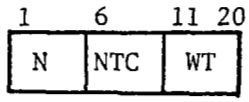
$$N_c^*(t'/t)^2 + N_{\hat{x}}(t/t') - (N_c^*)^2/(4N_{cr}) = 0 \ , \tag{G.2.18}$$

where $N_c^* = \sigma_s^* \bar{t}$, σ_s^* being the compressive strength of the stiffener (yield stress or crippling stress).

VI B Material Properties Data

A separate data deck is required for each material used. The decks do not have to be in numerical sequence of material numbers.

Temperature-Independent Data (2I5, F10.0)



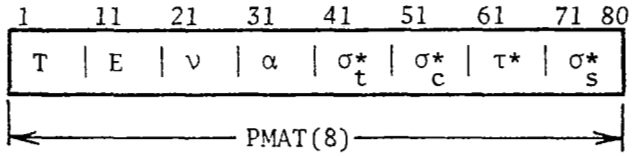
N = Material number

NTC = Number of temperatures for which properties of this material are given. If blank, computer sets NTC = 1.

WT = Weight-density of the material.

Temperature-Dependent Data (8F10.0)

One card for each temperature for which the properties of this material are given. Cards must be in ascending order of temperatures.



PMAT(1) = Temperature for which the properties are given (T).

PMAT(2) = Young's modulus of elasticity (E).

PMAT(3) = Poisson's ratio (ν).

PMAT(4) = Coefficient of linear thermal expansion (α).

PMAT(5) = Tensile strength of sheet (σ^*_t).

PMAT(6) = Compressive strength of sheet (σ^*_c).

PMAT(7) = Shear strength of sheet (τ^*).

} If blank, computer sets PMAT(6) = PMAT(5), PMAT(7) = 0.577 PMAT(5).

PMAT(8) = Compressive strength (yield or crippling stress) of stiffener (σ^*_s). If blank, computer sets PMAT(8) = PMAT(6).

VI C Geometric Properties Data (I5, 6F10.0)

A separate card is required for each element geometry. The cards do not have to be in numerical sequence of geometric property numbers.

1	6	16	26	36	46	56 65
N	TH	W	SA	SI	D	WE

N = Geometric property number.

TH = Sheet thickness t of the reference cross section (see note below).

W = Distance w between the stiffeners of the reference cross section.

SA = Cross-sectional area A_s of the stiffener in the reference cross section.

SI = Moment of inertia I_s of the stiffener about its own centroidal axis in the reference cross section.

D = Distance d between the centroid of the stiffener and the mid-plane of the sheet in the reference cross section.

WE = Effective width w_e of the sheet between the stiffeners in the reference cross section (used in buckling analysis of the sheet). If blank, computer sets WE = W.

The Geometric Properties Data is used only for the computation of unit cross-sectional properties, i.e., properties for unit sheet thickness. The reference cross section, therefore, can be chosen as any design that has the desired cross-sectional proportions; it does not have to be the initial design.

VI D Element Load Multipliers (4F10.0)

Four cards as indicated below:

1	11	21	31 40	
thermal	x-gravity	y-gravity	z-gravity	Card 1: Element load A
thermal	x-gravity	y-gravity	z-gravity	Card 2: Element load B
thermal	x-gravity	y-gravity	z-gravity	Card 3: Element load C
thermal	x-gravity	y-gravity	z-gravity	Card 4: Element load D

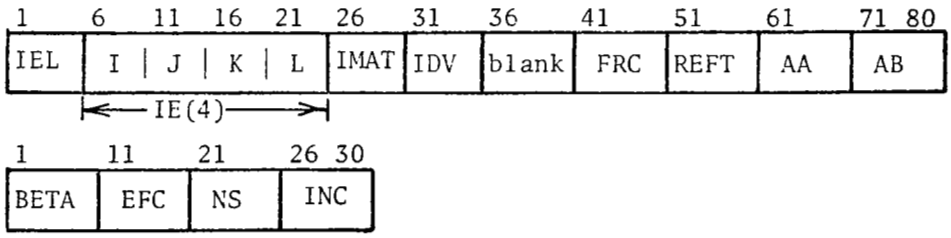
← EMUL(4,4) →

EMUL = Fractions of basic element loads (thermal loading and gravity loads in the three global coordinate directions) which are to be included in the element loads A,B,C and D.

Element Load Multipliers simply define the element loads A,B,C and D. Various multiples of these element loads can be added to each structural load condition by the use of Structural Load Multipliers (see Sec. D.1).

VI E Element Data (7I5, 5X, 4F10.0/2F10.0, 2I5)

Two cards for each element; cards must be arranged in an ascending order of element numbers.



IEL = Element number.

IE = Node numbers I,J,K and L of the four corner nodes (see Fig. G.1.1). Nodes 1 and 2 should define the side that is most parallel to the stiffeners as indicated in Fig. G.2.2. For a triangular element (their use is not recommended for stiffened panels), the same node number must be used for nodes 3 and 4.

IMAT = Material number of element.

IDV = Design variable number of element.

FRC = The design variable fraction η_i in eqn. (B.1.2): $A_i = \eta_i D_m$. Note that A_i, the size of the element, is taken as the sheet thickness t. If blank, computer sets FRC = 1.0.

REFT = Reference temperature (temperature of the stress-free state) of the element.

AA = The effective panel length "a" used in computing the Euler buckling load (G.2.16) of the stiffeners. If blank, computer calculates "a" in the manner shown in Fig. G.2.2.

AB = The effective panel width b used in computing the general buckling stress (G.2.6) of the panel. If blank, computer calculates b in the manner shown in Fig. G.2.2.

BETA = The angle β between the stiffeners and the local $\bar{x}$-axis (see Fig. G.2.2).

EFC = The end-fixity coefficient C used in Euler buckling load (G.2.16) of the stiffeners. If blank, computer sets EFC = 1.0 (simple supports).

NS = Stress printout code:

NS	Points for which stresses are printed (see Fig. G.1.1)
1	None
3	0
6	0,A
12	0,A,B,C
15	0,A,B,C,D

If blank, computer sets NS = 3. If NS = 15 and the element is triangular, computer sets NS = 12. Note that design of the element is always based on stresses at point 0, regardless of which printout code is used.

INC = Node number increment in automatic generation of element data (see below). If blank, computer sets INC = 1.

Automatic element data generation---if there exists a series of elements IEL = m, m+1, m+2,..., which satisfies the following requirements:

(a) the node numbers form the sequences
I=i, i+INC, i+2*INC,...,
J=j, j+INC, j+2*INC,...,
K=k, k+INC, k+2*INC,...,
L=ℓ, ℓ+INC, ℓ+2*INC,...,

(b) the remainder of the data is identical for all elements of the series,

then only the last card of the series will be required. The element data for the other elements of the series will be generated automatically.

G.3 Construction Code No. 2

Isotropic, homogeneous panels subjected to stress constraints only are treated under this construction code. The variable dimension of the element (element size) is the panel thickness t.

Two failure criteria are used in the stress ratio redesign:

a) Von Mises yield criterion:

$$(\sigma_1/\sigma^*)^2 + (\sigma_2/\sigma^*)^2 - (\sigma_1/\sigma^*)(\sigma_2/\sigma^*) = 1 \quad , \tag{G.3.1}$$

where σ_1 and σ_2 are the principal stresses, and σ^* is the allowable normal stress. Different allowable stresses may be specified for tension (σ_t^*) and compression (σ_c^*). The corresponding stress ratio redesign formula is

$$t'/t = [(N_1/N^*)^2 + (N_2/N^*)^2 - (N_1/N^*)(N_2/N^*)]^{1/2} \quad , \tag{G.3.2}$$

where $N^* = \sigma^* t$, and N_1, N_2 are the principal stress resultants.

b) Maximum shear stress theory of failure:

$$\tau_{max}/\tau^* = 1 \quad , \tag{G.3.3}$$

τ_{max} being the maximum shear stress and τ^* its allowable value. This leads to the stress ratio redesign formula

$$t'/t = (N_s)_{max}/N_s^* \quad , \tag{G.3.4}$$

where $(N_s)_{max}$ is the maximum shear stress resultant, and $N_s^* = \tau^* t$.

The value of t'/t is chosen as the larger of (G.3.2) and (G.3.4); however, the user can suppress the use of (G.3.4) by setting $\tau^* = 0$ on the material data cards.

The redesign formulas are applied to all points of the panel where the stress printout is requested (points O,A,B,C or D shown in Fig. G.1.1). This is in contrast to Construction Code No. 1, where point O only was used for redesign.

VI B Material Properties Data

A separate data deck is required for each material used. The decks do not have to be in numerical sequence of material numbers.

Material Control Card (2I5, F10.0)

1	6	11 21
N	NTC	WT

N = Material number.

NTC = Number of temperatures for which properties of this material are given. If blank, computer sets NTC = 1.

WT = Weight-density of the material.

Temperature-Dependent Data (7F10.0)

One card for each temperature for which the properties of this material are given. Cards must be in ascending order of temperatures.

1	11	21	31	41	51	61 70
T	E	ν	α	σ_t^*	σ_c^*	τ^*
←			PMAT(7)			→

PMAT(1)= Temperature for which the properties are given (T).

PMAT(2)= Young's modulus of elasticity (E).

PMAT(3)= Poisson's ratio (ν).

PMAT(4)= Coefficient of linear expansion (α).

PMAT(5)= Tensile strength (σ_t^*).

PMAT(6)= Compressive strength (σ_c^*). If blank, computer sets PMAT(6)=PMAT(5).

PMAT(7)= Shear strength (τ^*). If blank, maximum shear stress theory of failure will not be used in redesign.

VI C Element Load Multipliers (4F10.0)

Four cards as shown below:

1	11	21	31	41	50	
thermal	pressure	x-gravity	y-gravity	z-gravity		Card 1: A
thermal	pressure	x-gravity	y-gravity	z-gravity		Card 2: B
thermal	pressure	x-gravity	y-gravity	z-gravity		Card 3: C
thermal	pressure	x-gravity	y-gravity	z-gravity		Card 4: D

EMUL(4,5)

EMUL = Fractions of basic element loads (thermal loading, pressure acting on side of element, and gravity loads in the three global coordinate directions) which are to be included in the element loads A,B,C and D.

Element Load Multipliers define element loads A,B,C and D. Various multiples of these loads can be added to each structural load condition by the use of Structural Load Multipliers (see Sec. D.1).

VI D Element Data (7I5, F5.0, 3F10.0, 2I5)

One card for each element; cards must be arranged in ascending order of element numbers.

1	6	11	16	21	26	31	36	41	51	61	71	76 80
IEL	I	J	K	L	IMAT	IDV	FRC	REFT	PRESS	BETA	NS	INC

IE(4)

IEL = Element number.

IE = Node numbers I,J,K and L of the four corner nodes (see Fig. G.1.1). Nodes 1 and 2 should define the side on which pressure load is applied, if any. For a triangular element, the same node number must be used for nodes 3 and 4.

IMAT = Material number of element.

IDV = Design variable number of the element,

FRC = The design variable fraction η_i in eqn. (B.1.2): $A_i = \eta_i D_m$. Note that A_i, the size of the element, is taken as the panel thickness t. If blank, computer sets FRC = 1.0.

REFT = Reference temperature (temperature of the stress-free state) of the element.

PRESS = Compressive force per unit length (pressure resultant) applied to side 1-2 of the element.

BETA = The angle β which defines the $\hat{x}$ and $\hat{y}$ axes as shown in Fig. G.1.1. The angle is used only to control the stress printout at point 0 (see Sec. G.1).

NS = Stress printout code (see Sec. G.2). Note that the design of the element is based on the stresses at all the points for which stress printout is requested.

INC = Node increment in automatic generation of element data (see Sec. G.2).

H. QUADRILATERAL SHEAR PANEL

H.1 General Information

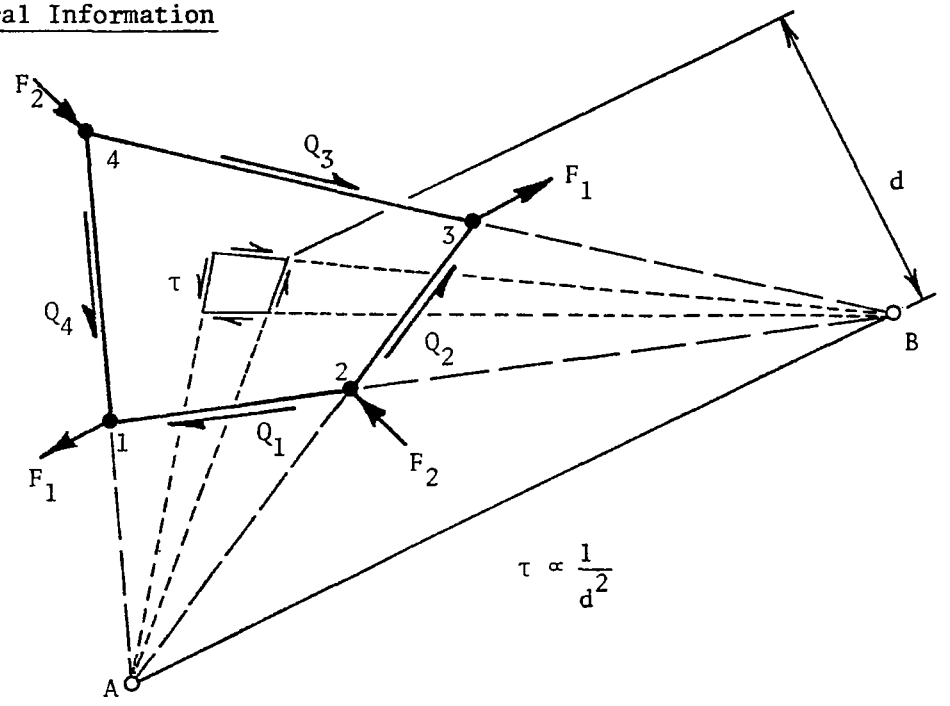


Figure H.1.1

Garvey's Shear Panel Idealization.

A shear panel, shown in Fig. H.1.1, is assumed to resist only shear tractions applied to its edges; it has no rigidity with respect to normal edge tractions. The resultant forces of the shear tractions Q_i, i = 1,2,3,4, must form a self-equilibrating system, so that only one of resultants is statically independent.

All four corner nodes should be on the same plane, otherwise errors will arise in the computation of element properties.

Following NASTRAN [5], the equivalent nodal forces F_1 and F_2 are obtained by "lumping" one half of each adjacent edge force into the corner. This method was chosen purely on the basis of convenience, because it

results in corner forces that are collinear with the diagonals of the element. Again, only one of the corner forces is statically independent, the remainder being determined by equilibrium requirements.

In order to calculate the strain energy (i.e., the stiffness matrix) of the element, an assumption must be made regarding the distribution of the shear stress τ in the panel. We adopted the distribution suggested by Garvey [10], which satisfies equilibrium equations, but not compatibility of the resulting displacement field (except for a parallelogram and a rectangle). The assumed shear planes are shown in Fig. H.1.1; the magnitude of τ is taken to be inversely proportional to the square of the distance d from the "baseline" AB. The details of formulating the element properties, including the geometric stiffness matrix, can be found in Ref. [5].

Two potential difficulties arise in the use of shear panels:

a) The state of stress assumed by Garvey can exist only in rectangular panel, i.e., only a rectangle can be in a state of pure shear when subjected to tangential boundary tractions. It is important, therefore, not to use severely skewed shear panels in order to avoid erroneous results.

b) The absence of extensional rigidity will, in general, lead to a singular stiffness matrix of the structure, unless each shear panel is surrounded by elements that are capable of absorbing the extensional corner forces (forces in direction other than F_1 and F_2). Bar elements are commonly used for this purpose, where the stiffness of each bar usually includes the extensional rigidities of the adjacent shear panels.

Shear panels have traditionally been used in aerospace industry to idealize stiffened panels: the shear resistance of the sheet is

accounted for by the shear panel, whereas the extentional rigidities of the sheet and the stiffeners are lumped into surrounding bar elements. This idealization was undoubtedly useful in the days of hand computation, but its utility has become dubious with the arrival of the finite element method. An orthotropic, plane stress quadrilateral element, such as described in Sec. G.2, is not only a more precise representation of a stiffened panel, but it also requires less input data and no more computer time.

Shear panel has not been excluded from DESAP 2, partly because it may still be a useful element in special applications, and partly in recognition of its popularity with aircraft designers.

Only gravity loads are permitted as the basic element loads. Thermal stesses are excluded, of course, since thermal expansion cannot be resisted by a state of pure shear. Temperature-dependent material properties, however, are allowed.

The printout that follows each analysis cycle consists of the shear flow $N_i = \tau_i t$ at each of the four nodes, and the average shear flow of the panel, given by

$$N_{av} = \frac{1}{4} \sum_{i=1}^{4} N_i \quad . \tag{H.1.1}$$

The first card of the element data deck for each construction code must be:

VI A Element Control Card (5I5)

1	6	11	16	21	25
MTYPE	NUME	NUMMAT	NUMTC	KODE	
← NPAR(5) →					

NPAR(1) = Code for the element type (MTYPE). For shear panel use MTYPE = 4.

NPAR(2) = Number of elements with the specified construction code (NUME).

NPAR(3) = Number of different materials used for this construction code (NUMMAT).

NPAR(4) = Maximum number of temperatures for which the material properties are given (NUMTC). If blank, computer sets NPAR(4) = 1.

NPAR(5) = Construction code (KODE).

H.2 Construction Code No. 1

Homogeneous, isotropic shear panels are treated under this construction code. The thickness t of the panel is the variable dimension (size of the element). Two design criteria are used in the redesign equations:

(1) The average shear stress τ_{av} should not exceed its prescribed limit τ^*. The corresponding stress ratio redesign formula is

$$t'/t = N_{av}/N^* \ , \tag{H.2.1}$$

where N_{av} is the average shear flow defined in (H.1.1), and $N^* = \tau^* t$.

(2) The average shear stress should not exceed the buckling stress τ_{cr}. For a rectangular panel, the critical shear stress is

$$\tau_{cr} = k \frac{\pi^2 E}{12(1-\nu^2)} \left(\frac{t}{b}\right)^2 \tag{H.2.2}$$

where b is the unsupported width (shorter dimension) of the panel, and k represents the buckling coefficient for shear. The stress ratio redesign equation obtainable from (H.2.2) is

$$t'/t = (N_{av}/N_{cr})^{1/3} \ , \tag{H.2.3}$$

where $N_{cr} = \tau_{cr} t$.

A good approximation for the buckling coefficient is

$$k = C_1 + C_2 (b/a)^2, \quad 0 < b/a \leq 1 \ , \tag{H.2.4}$$

where a is the longer dimension of the panel, and C_1 and C_2 are constants that depend on the edge support conditions only. The values of these

constants for various edge supports are given in Fig. H.2.1. The figure also shows the corresponding support codes ISU that are used to specify the boundary conditions on element cards.

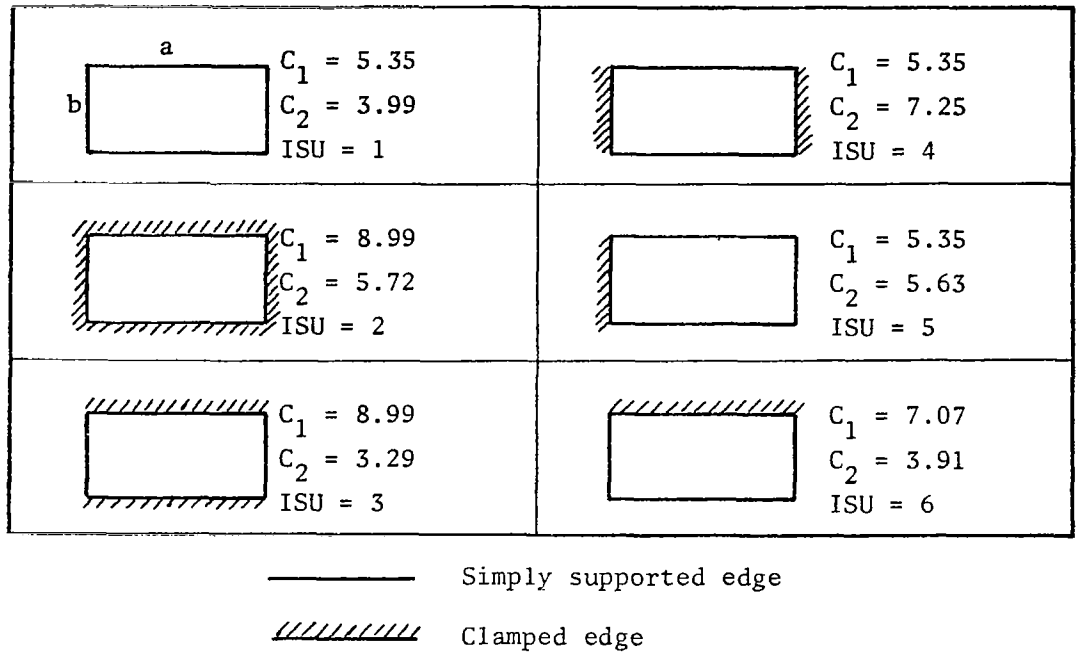


Figure H.2.1

Edge Supports for Buckling of Shear Panels.

Equation (H.2.4) agrees precisely with the theoretical buckling coefficients in the limiting cases b/a = 0 and b/a = 1; in between these values, the approximation is somewhat on the conservative side.

The unsupported dimensions of the panel, a and b, are user-specified. However, if the dimensions are left blank, the computer will use the "average" panel dimensions shown in Fig. H.2.2.

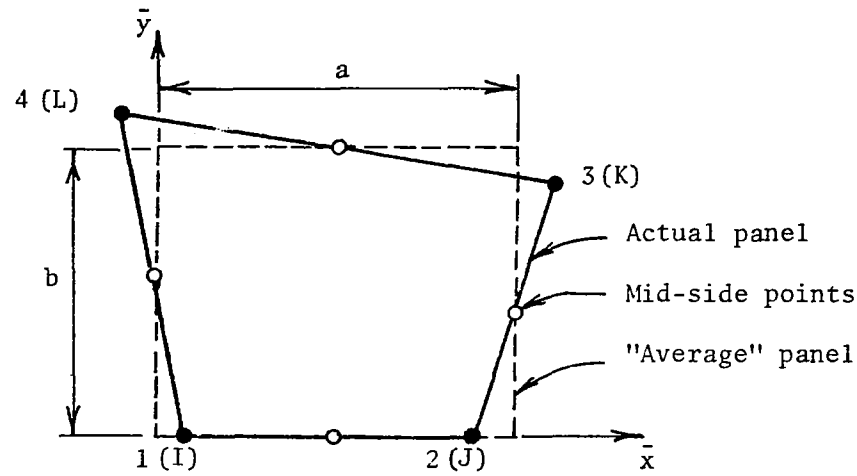


Figure H.2.2

Shear Panel and Its Average Dimensions Used in Buckling Analysis.

VI B Material Properties Data

A separate data deck is required for each material used. The decks do not have to be in numerical sequence of material numbers.

Material Control Card (2I5, F10.0)

1	6	11 20
N	NTC	WT

N = Material number.

NTC = Number of temperatures for which the properties of this material are given. If blank, computer sets NTC = 1.

WT = Weight-density of the material.

Temperature-Dependent Data (4F10.0)

One card for each temperature for which the material properties are given. Cards must be in ascending order of temperatures.

1	11	21	31 40
T	E	ν	τ^*

PMAT(4)

PMAT(1) = Temperature for which the material properties are given (T).

PMAT(2) = Young's modulus of elasticity (E).

PMAT(3) = Poisson's ratio (ν).

PMAT(4) = Shear strength (τ^*).

VI C Element Load Multipliers

Three cards as shown below.

1	11	21	31 40	
A	B	C	D	Card 1: x-gravity
A	B	C	D	Card 2: y-gravity
A	B	C	D	Card 3: z-gravity

EMUL(4,4)

EMUL = Fractions of basic element loads (gravity forces in the three global coordinate directions) which are to be included in the element load cases A,B,C and D.

Element Load Multipliers define element loads A,B,C and D. Various multiples of these loads can be added to each structural load condition by the use of Structural Load Multipliers (see Sec. D.1).

VI D Element Data (I5, 3F10.0, I5)

One card for each element; card must be in ascending order of element numbers.

1	6	11	16	21	26	31	36	41	51	61	71 75
IEL	I	J	K	L	IMAT	IDV	ISU	FRC	AL	BL	INC

IE(4)

IEL = Element number.

I = Node number of node 1 (see Fig. H.2.2).

J = Node number of node 2.

K = Node number of node 3.

L = Node number of node 4.

IMAT = Material number of element. If blank, computer sets IMAT = 1.

IDV = Design variable number of element.

ISU = Code for edge support conditions used in local buckling analysis (see Fig. H.2.1). If blank, local buckling analysis will be suppressed.

FRC = The design variable fraction η_i in eqn. (B.1.2): $A_i = \eta_i D_m$. Note that A_i, the size of the element, is taken as the panel thickness t. If blank, computer sets FRC = 1.0.

AL = The longer unsupported dimension "a" of the panel used in local buckling analysis. If blank, "a" is calculated as shown in Fig. H.2.2.

BL = The shorter unsupported dimension "b" of the panel used in local buckling analysis. If blank, "b" is calculated as shown in Fig. H.2.2.

INC = Node number increment in automatic generation of element data (see below). If blank, computer sets INC = 1.

Automatic element data generation---if there exists a series of elements IEL = m, m+1, m+2,...which satisfies the following requirements:

a) the node numbers form the sequences

I = i, i+INC, i+2*INC,...,
J = j, j+INC, j+2*INC,...,
K = k, k+INC, k+2*INC,...,
L = ℓ, ℓ+INC, ℓ+2*INC,...;

b) the remainder of the data is identical for all elements of the series,

then only the last card of the series will be required. The element data for the other elements of the series will be generated automatically.

I. PLATE QUADRILATERAL OR TRIANGULAR ELEMENT

I.1 General Information

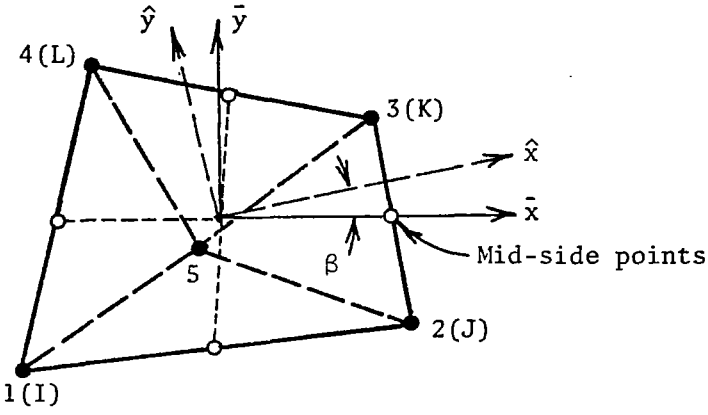


Figure I.1.1

Quadrilateral Plate Element.

The basic plate-shell element is the plane quadrilateral shown in Fig. I.1.1. The element is made up of four sub-triangles; the coordinates of the internal node (node 5) are obtained by averaging the coordinates of the four corner nodes. The local Cartesian coordinates $\bar{x}$ and $\bar{y}$ are determined by the mid-points of the sides in the manner shown on the drawing. If the element is orthotropic, $\hat{x}$ and $\hat{y}$ denote the principal material coordinates; their direction is specified by the angle β.

The membrane behavior of the element is obtained by treating each sub-triangle as a constant strain element, and then eliminating the degrees-of-freedom of node 5 by matrix condensation.

The procedure of Clough and Felippa[11] is employed to derive the bending properties of the quadrilateral. A so-called LCCT-9 element, with nine degrees of freedom, is used for each subtriangle. The lateral displacement field of this element is a cubic function, but the displacements are partially constrained such that the normal slope along

each side is linear. After the four triangles are assembled into the quadrilateral element, the degrees of freedom of node 5 are again eliminated by static condensation.

In calculating the contribution of the lateral displacements to the geometric stiffness matrix of the element, one modification is made to the above procedure: the shape functions used by Clough and Felippa for the subtriangles are replaced by the nonconforming functions suggested by Zienkiewicz---Ref. [12], equation (10.26). A quintic numerical integration formula (p. 151 of Ref. [12]) is then used to compute the unit geometric stiffness matrices for each of the subtriangles.

The membrane and bending characteristics of the quadrilateral are condensed and stored separately in the form of unit stiffness matrices and load vectors as described in Sec. B.1. The procedure of condensing the unit stiffness matrices before they are combined into the element stiffness matrix is valid only if all the subtriangles are coplanar. Therefore, if the four corner nodes do not lie on the same plane, i.e., if the quadrilateral is warped, errors will occur in the computation of element properties.

If a triangle, rather than a quadrilateral, is used in the program, a single LCCT-9 element will be used for bending, and a constant strain triangle for the membrane action. The triangular element with its local coordinate system is shown in Fig. I.1.2.

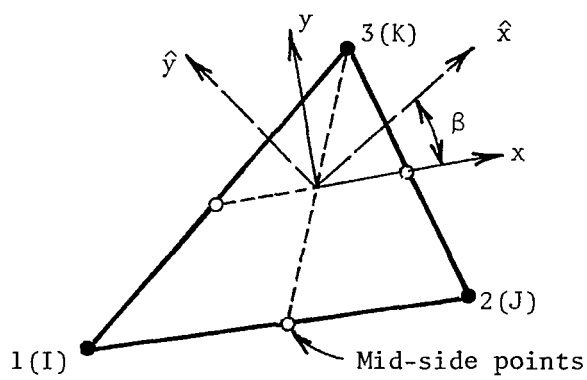


Figure I.1.2

Triangular Plate Element.

The basic element loads consist of lateral pressure, thermal loading, and gravity loads in the three global coordinate directions. The temperature is assumed to be uniform through the thickness of the plate, i.e., thermal stresses caused by temperature gradients in the direction normal to the element are neglected.

Provision has been made for temperature-dependent material properties. The properties of each material used should be listed for a range of temperatures covering che entire temperature spectrum of the structure. Linear interpolation is used by the computer to calculate the properties of each element for the specitied nodal temperatures.

The output from analysis contains the stress resultants referred to the coordinates $\hat{x}$ and $\hat{y}$. The positive directions of the stress resultants are shown in Fig. I.1.3. For a quadrilateral, the stress resultants are calculated at node 5 (Fig. I.1.1) by averaging the values

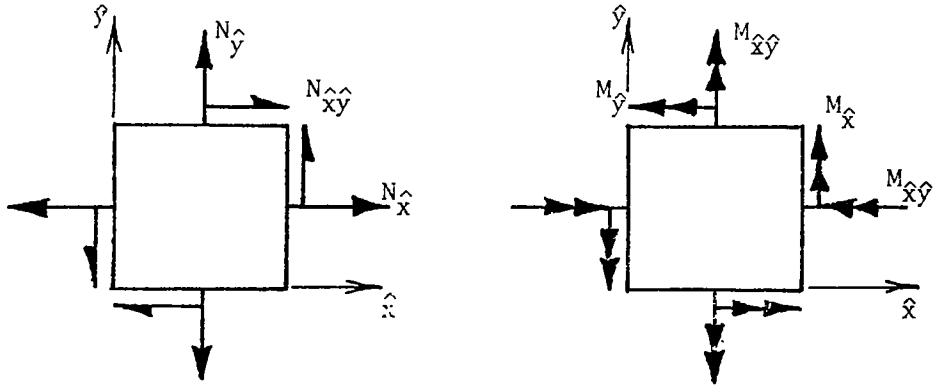


Figure I.1.3

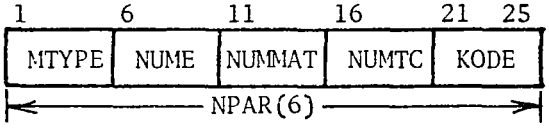
Positive Stress Resultants.

for the four sub-triangles. In the case of the triangular element, the stress resultants are computed at node 3 in Fig. I.1.2.

If the plate elements are used to model a curved surface, the nodal rotation about the normal to the surface may be suppressed by the use of boundary elements (see Ch. J).

The element data deck for each construction code used must begin with:

VI A Element Control Card (5I5)



NPAR(1) = Code for element type (MTYPE). For plate elements use NPAR(1) = 6.

NPAR(2) = Number of elements with the specified construction code (NUME).

NPAR(3) = Number of different materials used for the specified construction code (NUMMAT).

NPAR(4) = Maximum number of temperatures for which the properties of any one material are given (NUMTC).

NPAR(5) = Construction code (KODE).

The remainder of element data is listed separately for each construction code.

I.2 Construction Code No. 1

This construction code treats isotropic, homogeneous plate elements. The design variable (size of the element) is the plate thickness t.

The stress ratio redesign formula is obtained from the modified Von Mises yield criterion

$$(\sigma_{\hat{x}}/\sigma^*)^2 + (\sigma_{\hat{y}}/\sigma^*)^2 - \sigma_{\hat{x}}\sigma_{\hat{y}}/(\sigma^*)^2 + (\tau_{\hat{x}\hat{y}}/\tau^*)^2 = 1 \ , \qquad (I.2.1)$$

where σ^* and τ^* are the allowable normal and shear stresses referred to the $\hat{x}$ and $\hat{y}$ axis, respectively. Different values of σ^* may be used for tension (σ_t^*) and compression (σ_c^*).

Note that (I.2.1) represents the original, isotropic Von Mises yield criterion if $\sigma_t^* = \sigma_c^*$ and $\tau^* = \sigma^*/\sqrt{3}$. In that case, the design would be independent of the angle β in Figs. I.1.2 and I.1.3; consequently, β will only control the printout of the stress resultants.

The stress ratio formula obtainable from (I.2.1) is the fourth-order equation in t'/t:

$$(t'/t)^4 - a(t'/t)^2 \mp b(t'/t) - c = 0, \qquad (I.2.2)$$

where the minus sign preceding b is applicable to the upper surface of the plate and the plus sign to the lower surface, and

$$\begin{aligned} a &= (N_{\hat{x}}/N^*)^2 + (N_{\hat{y}}/N^*)^2 + (N_{\hat{x}\hat{y}}/N_s^*)^2 - N_{\hat{x}}N_{\hat{y}}/(N^*)^2 \\ b &= 2[(N_{\hat{x}}/N^*)(M_{\hat{x}}/M^*) + (N_{\hat{y}}/N^*)(M_{\hat{y}}/M^*) + (N_{\hat{x}\hat{y}}/N_s^*)(M_{\hat{x}\hat{y}}/M_s^*)] \\ &\quad - [(N_{\hat{x}}/N^*)(M_{\hat{y}}/M^*) + (N_{\hat{y}}/N^*)(M_{\hat{x}}/M^*)] \\ c &= (M_{\hat{x}}/M^*)^2 + (M_{\hat{y}}/M^*)^2 + (M_{\hat{x}\hat{y}}/M_s^*)^2 - (M_{\hat{x}}/M^*)(M_{\hat{y}}/M^*) \ . \end{aligned} \qquad (I.2.3)$$

In (I.2.3) we used the notation

$$N^* = t\sigma^* \qquad N_s^* = t\tau^*$$
$$M^* = t^2\sigma^*/6 \quad , \qquad M_s^* = t^2\tau^*/6 \quad . \tag{I.2.4}$$

Equation (I.2.2) is solved by the method of successive iterations, the iteration equation being

$$(r^{\nu+1})^4 - (a \pm b/r^{\nu})(r^{\nu+1})^2 - c = 0 \quad ,$$

i.e.,

$$r^{\nu+1} = \left[\frac{1}{2}(a \pm b/r^{\nu}) + \sqrt{\frac{1}{4}(a \pm b/r^{\nu})^2 + c}\right]^{1/2} \quad , \tag{I.2.5}$$

where r^{ν} is the current value of t'/t, and $r^{\nu+1}$ represents the improved value. The starting value of r^{ν} is taken as one, and the number of iterations is limited to ten. Note that for pure bending action or pure membrane state of stress b = 0, in which case, (I.2.5) becomes an exact expression for t'/t.

The redesign equation is applied only to the point where the stress resultants are calculated, namely node 5 for a quadrilateral, and node 3 for a triangle. No local instability criteria are used with this construction code.

VI B Material Properties Data

A separate data deck, described below, is required for each material used for this construction code. The decks do not have to be in the sequence of material numbers.

Material Control Card (2I5, F10.0)

1	6	11 20
N	NTC	WT

N = Material number.

NTC = Number of temperatures for which the properties of this material are given. If blank, computer sets NTC = 1.

WT = Weight-density of the material.

Material Properties Cards (7F10.0)

One card for each temperature for which the properties of this material are given. Cards must be in ascending order of temperatures.

1	11	21	31	41	51	61	70
T	E	ν	α	σ_t^*	σ_c^*	τ^*	

PMAT(7)

PMAT(1)= Temperature for which the material properties are given (T).

PMAT(2)= Young's modulus of elasticity (E).

PMAT(3)= Poisson's ratio (ν).

PMAT(4)= Coefficient of linear thermal expansion (α).

PMAT(5)= Tensile strength (σ_t^*).

PMAT(6)= Compressive strength (σ_c^*). If blank, computer sets PMAT(6)=PMAT(5).

PMAT(7)= Shear strength (τ^*). If blank, computer sets PMAT(7)=0.577*PMAT(5).

VI C Element Load Multipliers (4F10.0)

Five cards shown below:

1	11	21	31	40
A	B	C	D	
A	B	C	D	
A	B	C	D	
A	B	C	D	
A	B	C	D	

EMUL

Card 1: lateral pressure.

Card 2: thermal loading.

Card 3: x-gravity.

Card 4: y-gravity.

Card 5: z-gravity.

EMUL = Fractions of basic element loads (lateral pressure, thermal loading and gravity loads in the three global coordinate directions) which are to be included in element loads A,B,C and D.

Element Load Multipliers simply define the element loads A,B,C and D. Various multiples of these element loads can be added to each structural load condition by the use of Structural Load Multipliers (see Sec. D.1).

IV D Element Data (8I5, 3F10.0)

One card for each element; cards must be arranged in an ascending order of element numbers.

1	6	11	16	21	26	31	36	41	51	61	71	80
IEL	I	J	K	L	IMAT	INC	IDV	PRESS	REFT	FRC	BETA	
	←——IE(4)——→											

IEL = Element number.

IE = Node numbers I,J,K and L of the four corner nodes (see Fig. I.1.1). For a triangular element use L = 0.

IMAT = Material number of the element.

INC = Node number increment in automatic generation of element data (see note below). If blank, computer sets INC = 1.

IDV = Design variable number of element.

PRESS = Lateral pressure (in the positive $\bar{z}$-direction) acting on the element.

REFT = Reference temperature (temperature of the stress-free state) of the element.

FRC = The design variable fraction η_i in (B.1.2): $A_i = \eta_i D_m$. Note that the size of the element A_i is the thickness of the plate. If blank, computer sets FRC = 1.0.

BETA = The angle β (in degrees) that defines the $\hat{x}$ and $\hat{y}$ axes as shown in Figs. I.1.1 and I.1.2. Note that the stress resultants printed out are referred to the $\hat{x}$ and $\hat{y}$ axes.

Automatic element data generation---if there exists a series of elements IEL = m, m+1, m+2,..., which satisfy the following conditions:

(a) the node numbers form the sequences

I = i, i+INC, i+2*INC,...,
J = j, j+INC, j+2*INC,...,
K = k, k+INC, k+2*INC,...,
L = ℓ, ℓ+INC, ℓ+2*INC,...,

(b) the remainder of the data is identical for all elements of the series,

then only the last card of the series will be required. The element data for the other elements of the series will be generated automatically.

J. BOUNDARY ELEMENTS

A boundary element is essentially a line element with specified extensional and/or rotational spring constant. The element can be used for the following purposes:

(a) modelling of elastic supports,

(b) enforcement of specified nodal displacements or rotations in a given direction,

(c) computation of support reactions.

The direction of the boundary element can be specified in two ways, as shown in Fig. J.1.

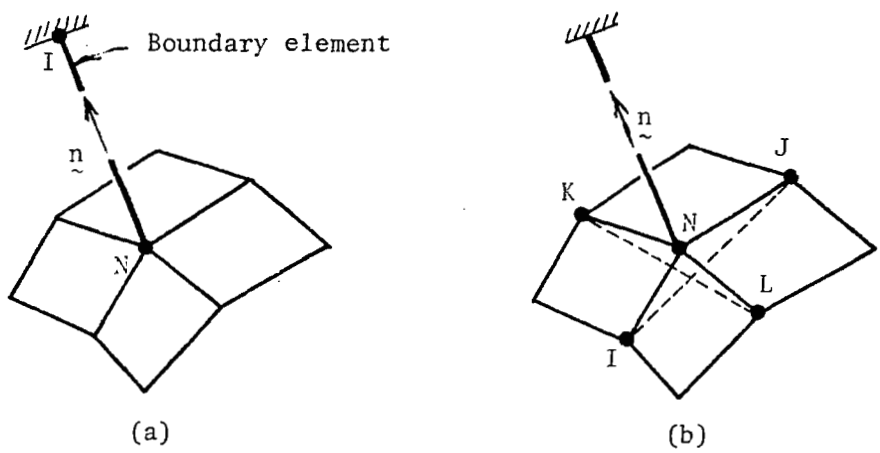


Figure J.1

Two Options for Specifying Direction n of a Boundary Element.

In method (a) the direction is determined by the structural node N and a second node I. The latter may also be a structural node, or a node specially created for the purpose. In the latter case, the degrees of

freedom of I should be suppressed on the node card. In method (b) the direction of the element is taken as perpendicular to the lines I-J and K-L. Again, the points I,J,K and L may be structural nodes, or special points listed on the node cards (with suppressed degrees of freedom). This option is particularly useful when boundary elements are used to suppress the normal rotations for thin shells.

The results of the analysis consist of the axial force and/or torque in the boundary element. If the boundary element is an extensional spring, the axial force is computed from the formula

$$P = k\delta \quad , \tag{J.1}$$

where k is the spring constant and δ the computed displacement of node N (see Fig. J.2).

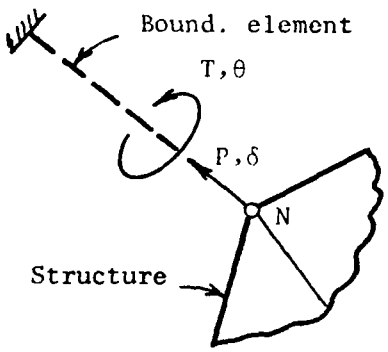


Figure J.2
Positive Displacements and Forces Acting on the Structure

Similarly, the torque in a rotational spring obtained from

$$T = k\theta \quad , \tag{J.2}$$

with T being the torque and θ the computed rotation.

Positive directions of the forces and displacements are defined in Fig. J.2.

If a nonzero displacement δ^* is prescribed for node N, the following force is applied to the node prior to analysis:

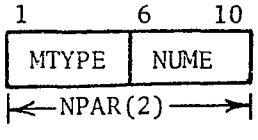
$$P^* = k\delta^* \ . \qquad (J.3)$$

This applied force will result in the specified displacement only if the spring constant k of the boundary element is made much larger than the corresponding stiffness of the structure. A specified rotation θ^* is handled in the analogous fashion.

A regular elastic support (without a prescribed displacement) is obtained by setting $\delta^* = 0$ and/or $\theta^* = 0$. If, in addition, k is made sufficiently large, the boundary element will approximate a rigid support. The latter option provides a means of enforcing "skewed" boundary conditions due to, for example, inclined roller supports.

Caution must be exercised in the use of boundary elements. Round-off errors in the input data and computation of element properties have the effect of introducing springs in directions other than the desired orientation. The spring constants of these undesired "elements" are proportional to the roundoff errors. Although some of the computational errors are minimized by the use of double precision arithmetic, it is still important not to make the spring constants of the boundary elements too large and to specify the directions of the elements with great precision.

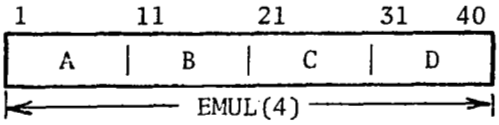
VI A Element Control Card (2I5)



NPAR(1) = Code for element type (MTYPE). For boundary elements use NPAR(1) = 7.

NPAR(2) = Number of boundary elements (NUME).

VI B Element Load Multipliers (4F10.0)

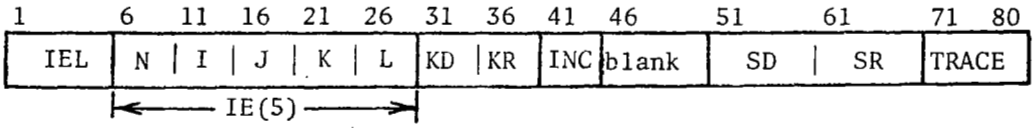


EMUL = Fractions of specified nodal displacements and rotations which are to be included in element load cases A,B,C and D.

Element Load Multipliers define the element loads A,B,C and D. Various multiples of these element loads can be added to each structural load condition by the use of Structural Load Multipliers as explained in Sec. D.1. For boundary elements, the load multipliers are useful if some structural load cases include specified nodal displacements, while others do not.

VI C Element Data (9I5, 5X, 3F10.0)

One card for each boundary element; cards must be in ascending order of element numbers.



IEL = Element number.

N = Node number of node N (structural node to which the boundary element is attached).

I, J, K, L = Node numbers, I,J,K,L that define the direction of the element. If node I only is used as shown in Fig. J.1(a), set J = K = L = 0.

KD = Code for extensional boundary element.
KD = 0: extensional spring is not used.
KD = 1: extensional spring is used.

KR = Code for rotational boundary element.
KR = 0: rotational spring is not used.
KR = 1: rotational spring is used.

INC = Node increment used in automatic generation of element data (see below).

SD = Specified displacement of node N.

SR = Specified rotation of node N (radians).

TRACE = Spring stiffness for extensional and/or rotation spring. If blank, computer sets TRACE = 10.0**10.

Automatic element generation---if there exists a series of elements which satisfy the following conditions:

(a) the element numbers form the sequence
MEMB = m, m+1, m+2,...;

(b) the node numbers of successive elements are
N = n, n+INC, n+2*INC,...,
I = i, i+INC, i+2*INC,...,
J = j, j+INC, j+2*INC,...,
K = k, k+INC, k+2*INC,...,
L = ℓ, ℓ+INC, ℓ+2*INC,...,
} or, J = K = L = 0 for all elements if node I only is used for direction;

(c) the remainder of the data is identical for all elements of the series;

then only the last card of the series will be required. The element data for other elements of the series will be generated automatically.

REFERENCES

[1] Kiusalaas, J. and Reddy, G. B., "DESAP 1 - A Structural Design Program with Stress and Displacement Constraints". Department of Engineering Science and Mechanics, The Pennsylvania State University, University Park, PA., March 1976.

[2] Wilson, E. L., "SOLID SAP - A Static Analysis Program for Three-Dimensional Solid Structures". Report UC SESM 71-19, Structural Engineering Laboratory, University of California, Berkeley, Sept. 1971 (revised Dec. 1972).

[3] Kiusalaas, J., "Minimum Weight Design of Structures Via Optimality Criteria". NASA TN D-7115, Dec. 1972.

[4] Kiusalaas, J., "Optimal Design of Structures with Buckling Constraints". Int. J. Solids Structures, Vol. 9, pp. 863-878, 1973.

[5] MacNeal, R. H. (editor), The Nastran Theoretical Manual. Office of Technology Utilization, National Aeronautics and Space Administration, Sept. 1970.

[6] Fox, R. L., Optimization Methods for Engineering Design. Addison-Wesley, 1971.

[7] Admire, J. R., "Modal Analysis of Structures by an Iterative Rayleigh-Ritz Technique". Research Achievements Review, Vol. 4, Report No. 1, NASA TM X-64528, 1970.

[8] NASA MSFC Astronautic Structures Manual. NASA, George C. Marshall Space Flight Center, Analytical Mechanics Division, MSFC-Form 454 (revised Oct. 1967).

[9] Gerard, G., Introduction to Structural Stability Theory. McGraw-Hill, 1962.

[10] Garvey, S. J., "The Quadrilateral Shear Panel". Aircraft Engineering, pp. 134-135, 144, May 1951.

[11] Clough, R. W. and Felippa, C. A., "A Refined Quadrilateral Element for Analysis of Plate Bending". Proc. Second Conference on Matrix Methods in Structural Mechanics, Wright-Patterson AFB, Ohio, pp. 399-440, 1968. AFFDL-TR-68-150.

[12] Zienkiewicz, O. C., The Finite Element Method in Engineering Science. McGraw-Hill, 1971.

[13] Timoshenko, S. P. and Gere, J. M., Theory of Elastic Stability (Second Edition). McGraw-Hill Book Co., 1961.

[14] Khot, N. S., Venkayya, V. B. and Berke, L., "Optimization of Structures for Strength and Stability Requirements". AFFDL-TR-73-98, Dec. 1973.

[15] Taylor, J. E. and Liu, C. Y., "Optimal Design of Columns". AIAA Journal, Vol. 6, pp. 1497-1502, 1968.

[16] Chen, C., "Optimal Design and Isoperimetric Eigenvalue Problems". Ph.D. Thesis in Engineering Mechanics, The Pennsylvania State University, University Park, Pennsylvania, June 1969.

NASA-Langley, 1977