REPORT NO.
63-2

STRUCTURES AND MATERIALS RESEARCH
DEPARTMENT OF CIVIL ENGINEERING

FINITE ELEMENT ANALYSIS OF TWO-DIMENSIONAL STRUCTURES

BY

EDWARD L. WILSON

NATIONAL SCIENCE FOUNDATION
RESEARCH GRANT G18986

JUNE, 1963

STRUCTURAL ENGINEERING LABORATORY
UNIVERSITY OF CALIFORNIA
BERKELEY CALIFORNIA

5733

Structures and Materials Research
Department of Civil Engineering
Report No. 63-2

FINITE ELEMENT ANALYSIS OF TWO-DIMENSIONAL STRUCTURES

by

Edward L. Wilson

Faculty Investigator: Ray W. Clough

Prepared under the sponsorship of
National Science Foundation
Research Grant GI8986

University of California
Berkeley, California

June, 1963

TABLE OF CONTENTS

TABLE OF CONTENTS (Cont'd.)

LIST OF FIGURES

LIST OF SYMBOLS

ε_x	Strain in x Direction
ε_y	Strain in y Direction
γ	Shearing Strain
σ_x	Stress in x Direction
σ_y	Stress in y Direction
τ_{xy}	Shearing Stress
u	Displacement in x Direction
v	Displacement in y Direction
C_1, C_2, C_3, C_4	Constants which Define the Strain Distribution Within an Element
a_j, b_j, a_k, b_k	Element Dimensions
E	Modulus of Elasticity
ν	Poisson's Ratio
$[A]$	Displacement Transformation Matrix
$[B]$	Force Transformation Matrix
$[C]$	Stress-Strain Relationship
$[S]$	Element Corner Forces
$[k]$	Element Stiffness Matrix
$[r]$	Nodal Point Displacements
$[R]$	Nodal Point Loads
$[K]$	Stiffness Matrix for Complete Structure
$[\;]^T$	Matrix Transpose
β	Over-Relaxation Factor
δ	Group Relaxation Factor
σ_1, σ_2	Major and Minor Principal Stresses
θ	Direction of Major Principal Stress from Horizontal
ϕ	Slope of Boundary Point

σ_t	Constrained Thermal Stress
α_t	Thermal Coefficient of Expansion
ΔT	Temperature Increase
x,y	Coordinates of Nodal Points
$[K]_i$	Stiffness for Load Increment i
$[\Delta R]_i$	Load Increment i
$[C]_i$	Incremental Stress-Strain Relationship
$[\Delta\sigma]_i$	Incremental Stresses for Increment i
$[\sigma]_i$	Total Stresses After Increment i
$[\Delta r]_i$	Incremental Displacements for Increment i
$[r]_i$	Total Displacements After Increment i
$[K]_n$	Effective Stiffness for Approximation n
$[r]_n$	Displacements Resulting from Approximation n

INTRODUCTION

Nature of Problem

Many of the recent technological advancements have required the refined analysis and design of complex structural systems. An important class of structure, which the analyst must be prepared to cope with, is the arbitrary two-dimensional structure subjected to in-plane loads. Although the governing differential equations have been known for over a century, closed form solutions have been obtained only for a limited number of practical structures. Thus, the engineer must often rely on experimental or numerical procedures to solve this problem.

Throughout the past fifty years experimental methods, such as photoelasticity, have proved to be versatile tools in the analysis of arbitrary two-dimensional structures. However, for structures with nonlinear material properties or with body loading this approach becomes extremely difficult.

Within the past ten years the development of the digital computer has motivated renewed and more extensive development of numerical methods. The ultimate purpose of most numerical procedures is to reduce the continuous problem into a model with a finite number of degrees of freedom. In general, this reduction involves either a physical or mathematical approximation.

Collocation, Least Squares, Galerkin and Ritz methods are examples of mathematical approaches which involve the selection of the best solution from some assumed family of trial solutions. Perhaps the finite difference method, which involves the replacement of derivatives in the differential equation and boundary conditions with difference equations, is the most powerful of the mathematical approximations.

In the physical approach,* the structure is approximated by a finite number of discrete elements interconnected at a finite number of nodal points. Approximations are made on the behavior of the elements in an attempt to approximate the behavior of the continuous structure. Based on these approximations, equilibrium equations, in terms of unknown nodal point displacements, are developed for each nodal point. A solution of this set of equations constitutes a solution of the finite element system. This finite element approximation is the basis for the numerical work presented in this thesis.

Method of Analysis

From a historical standpoint, a one-dimensional finite element was first introduced by Saint-Venant in his work on torsion and flexure of beams. This work was of primary importance in the basic development of the field of structural analysis. Methods of structural analysis have been previously applied to two-dimensional structures. McHenry (2), Grinter (3) and Hrennikoff (4) have approximated two-dimensional elements by systems of one-dimensional elements. Turner, Clough, Martin and Topp (5) first introduced the two-dimensional plate-element in the analysis of aircraft structures. Clough (6) has presented both rectangular and triangular plate-element

---

*This method has been considered by some (1) to be a type of finite difference approach. Since it differs considerably from normal finite-difference methods, the physical terminology "finite element method" seems more appropriate.

models for plane stress structures. The triangular plate-element is the specific type of element used throughout this investigation.

The finite element idealization of a two-dimensional problem produces a highly indeterminate structural system. Because of the tremendous number of numerical operations which are inherent in such an analysis, the use of modern digital computers is mandatory. Perhaps the matrix formulation of structural analysis, as presented by Argyris (7), provides the most general approach to the computer analysis of structural systems. However, due to computer storage limitations, this formal matrix procedure is modified in this study -- only the non-zero coefficients of the matrices are considered by the computer program.

Purpose and Scope

The purpose of this dissertation is to generalize and extend the finite element method as applied to two-dimensional structures.

First, the stiffness matrix for the general triangular element is rederived and procedures for treating structures with anisotropic material properties, body forces and mixed boundary conditions are discussed. In addition, an iterative method of solution is introduced which enables the finite element procedure to be applied to large practical problems (over 500 elements in large computers). Also, in order to eliminate previous difficulties in evaluating element stresses a procedure to determine nodal point stresses is presented.

Second, the overall validity of the finite element procedure is examined and several examples are presented to indicate the practical application of the method to linear systems.

Finally, the finite element method is extended to include structures with nonlinear material properties. Two solution techniques, a step-by-step procedure and a successive approximation approach, are given.

All work has been programmed for the IBM 7090 digital computer. To enable others to use this procedure, a Fortran listing of the computer program for linear systems is presented in the Appendix.

METHOD OF ANALYSIS - LINEAR STRUCTURES

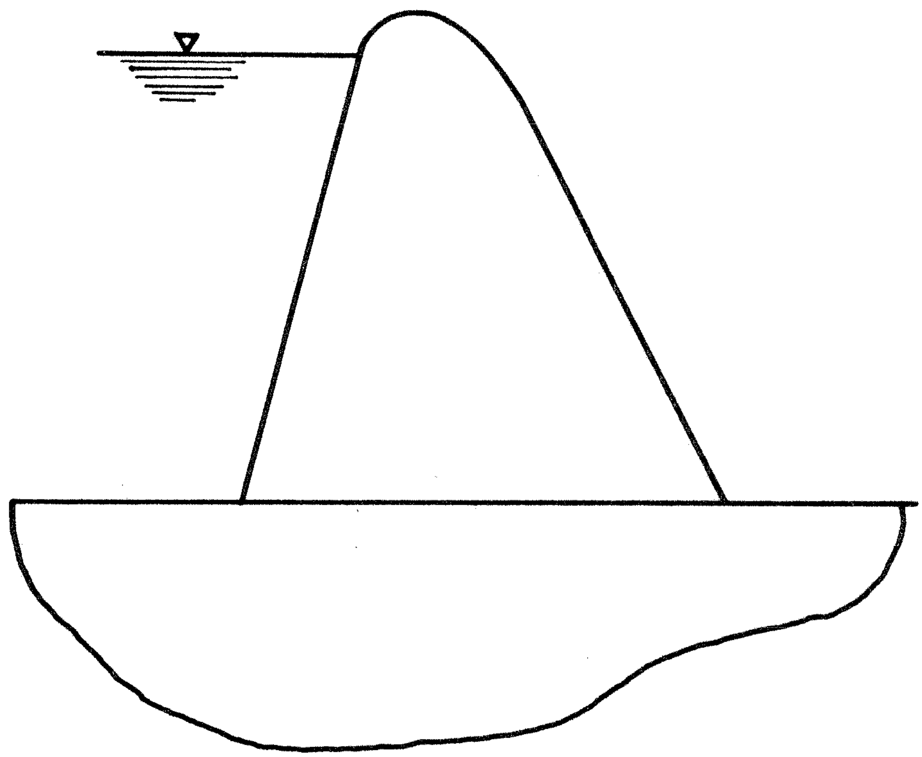
The "finite element method" is a general method of structural analysis in which a continuous structure is replaced by a finite number of elements interconnected at a finite number of nodal points. (Such an idealization is inherent in the conventional analysis of frames and trusses). In this investigation the finite element method is used to determine the stresses and displacements developed in two-dimensional elastic structures of arbitrary geometry and material properties. An assemblage of triangular plate elements is used to represent the continuous structure. Forces acting on the actual structure are replaced by statically equivalent concentrated forces acting at the nodal points of the finite element system. Figure 1 illustrates a very coarse-mesh idealization of the cross-section of a gravity dam.

Basic Assumptions

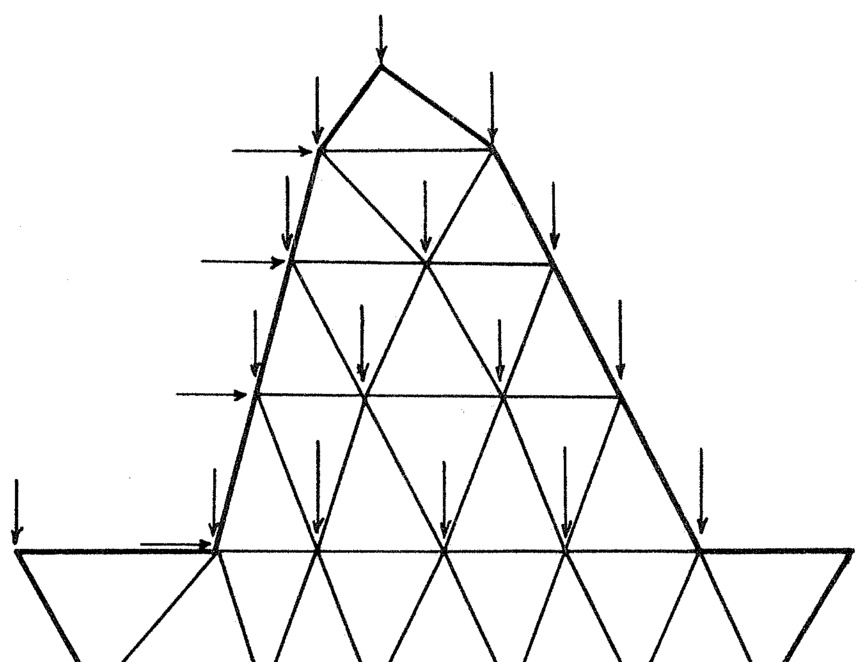
Continuity between elements of the system is maintained by requiring that within each element "lines initially straight remain straight in their displaced position."* This requirement is satisfied if the strains ε_x, ε_y and γ are assumed to be constant within each element. Therefore, the stresses σ_x, σ_y and τ_{xy} which act on the edges of each element are also constant. These stresses are replaced by stress resultants which act at the corners of the element. Based on these assumptions, it is possible to determine the stiffness of a

---

*This is similar to the approximation made in classical beam theory that "transverse sections, originally plane, remain plane and normal to the longitudinal fibers of the beam after bending."



a. ACTUAL DAM SECTION



b. ELEMENT AND LOAD APPROXIMATION

FIG. 1 - THE TRIANGULAR FINITE ELEMENT IDEALIZATION

typical element, which is an expression for the corner forces resulting from unit corner displacements. After this relationship is developed standard methods of structural analysis are employed to solve the complete system of elements.

Stiffness of a Typical Element

Strain-Displacement Relationship - The first step in the development of the stiffness of a typical element is to express the three components of strain within each element in terms of the six corner displacements. The geometry of a typical element is defined in Fig. 2. The assumed displacement pattern is illustrated in Fig. 3. This linear displacement field is defined in terms of $u_{(x,y)}$ and $v_{(x,y)}$ by equations of the following form:

$$u_{(x,y)} = u_i + C_1 x + C_2 y \qquad (1a)$$

$$v_{(x,y)} = v_i + C_3 x + C_4 y \qquad (1b)$$

The constants C_1, C_2, C_3, and C_4 can be expressed readily in terms of the corner displacements and the geometry of the element:

$$\begin{bmatrix} C_1 \\ C_2 \\ C_3 \\ C_4 \end{bmatrix} = \frac{1}{a_j b_k - a_k b_j} \begin{bmatrix} b_j - b_k & 0 & b_k & 0 & -b_j & 0 \\ a_k - a_j & 0 & -a_k & 0 & a_j & 0 \\ 0 & b_j - b_k & 0 & b_k & 0 & -b_j \\ 0 & a_k - a_j & 0 & -a_k & 0 & a_j \end{bmatrix} \begin{bmatrix} u_i \\ v_i \\ u_j \\ v_j \\ u_k \\ v_k \end{bmatrix} \qquad (2)$$

Ao; $u(x,y) = u_i + x[(b_j - b_k)u_i + b_k u_j - b_j u_k] + y[(a_k - a_j)u_i - a_k u_j + a_j u_k]$

$v(x,y)$ = etc Note how displ comp. within element are expressed in terms of a) corner displ. b) geometry of element

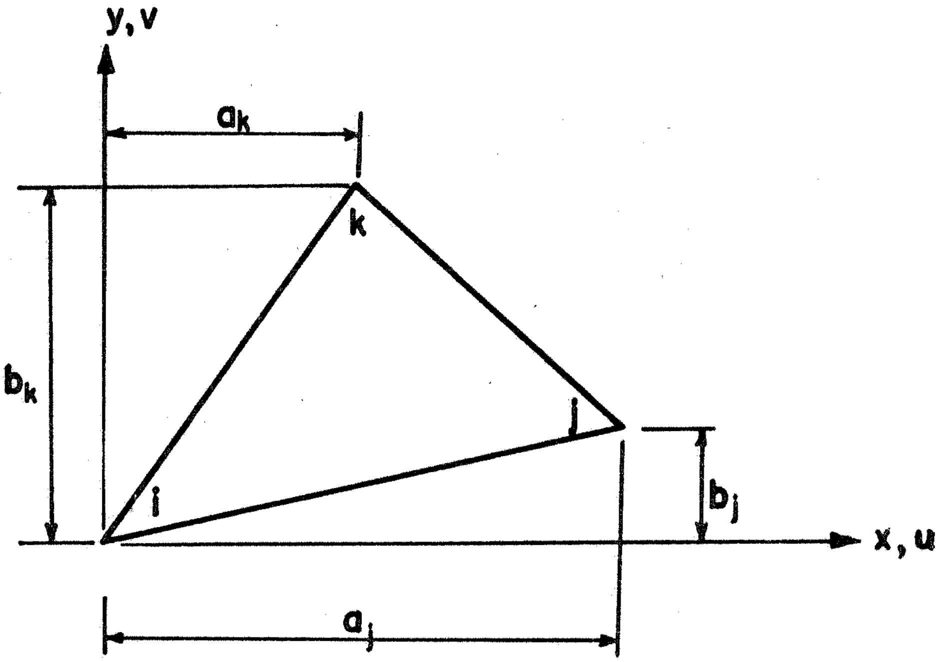


FIG. 2 - ELEMENT DIMENSIONS

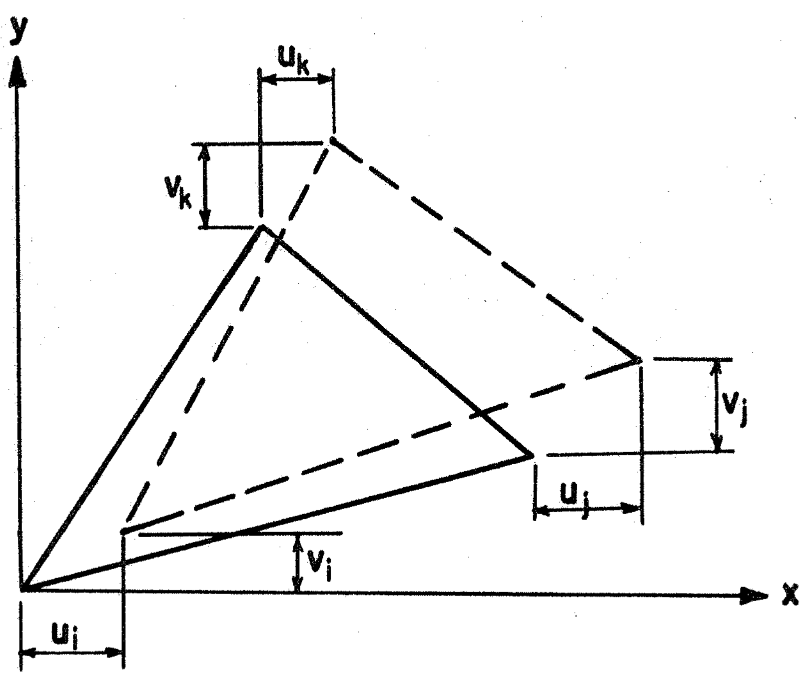


FIG. 3 - ASSUMED DISPLACEMENT PATTERN

The strains within the element can be obtained from the assumed displacement field, Eq. 1, by considering the basic definitions of strain:

$$\varepsilon_x \equiv \frac{\partial u}{\partial x} = c_1 \qquad (3a)$$

$$\varepsilon_y \equiv \frac{\partial v}{\partial y} = c_4 \qquad (3b)$$

$$\gamma \equiv \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} = c_2 + c_3 \qquad (3c)$$

If Eqs. 2 and 3 are combined, element strains in terms of corner displacements are expressed by the following matrix equation:

$$\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma \end{bmatrix} = \frac{1}{a_j b_k - a_k b_j} \begin{bmatrix} b_j - b_k & 0 & b_k & 0 & -b_j & 0 \\ 0 & a_k - a_j & 0 & -a_k & 0 & a_j \\ a_k - a_j & b_j - b_k & -a_k & b_k & a_j & -b_j \end{bmatrix} \begin{bmatrix} u_i \\ v_i \\ u_j \\ v_j \\ u_k \\ v_k \end{bmatrix} \qquad (4a)$$

or in symbolic form

$$[\varepsilon] = [A]\,[r] \qquad (4b)$$

<u>Stress-Strain Relationship</u> - One important advantage of the finite element method in two-dimensional elasticity is that structures with anisotropic material properties can be considered. In general, the stress-strain relationship is of the form

$$\begin{bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{bmatrix} = \begin{bmatrix} c_{11} & c_{12} & c_{13} \\ c_{21} & c_{22} & c_{23} \\ c_{31} & c_{32} & c_{33} \end{bmatrix} \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma \end{bmatrix} \qquad (5a)$$

or in symbolic form

$$[\sigma] = [C][\varepsilon] \tag{5b}$$

For example, the stress-strain relationship for an isotropic material in the state of plane strain is of the form

$$\begin{bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{bmatrix} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & 0 \\ \nu & 1-\nu & 0 \\ 0 & 0 & \frac{1-2\nu}{2} \end{bmatrix} \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma \end{bmatrix} \tag{6}$$

<u>Stress-Resultants</u> - The next step in the development of the stiffness of a typical element is to replace the uniform stresses acting on the edges of the element with stress-resultants acting at the corners of the element. For an element of unit thickness, Fig. 4 illustrates a set of statically equivalent corner forces for each component of stress. The corner forces expressed in terms of the three components of stress are

$$\begin{bmatrix} S_x^i \\ S_y^i \\ S_x^j \\ S_y^j \\ S_x^k \\ S_y^k \end{bmatrix} = \frac{1}{2} \begin{bmatrix} b_j-b_k & 0 & a_k-a_j \\ 0 & a_k-a_j & b_j-b_k \\ b_k & 0 & -a_k \\ 0 & -a_k & b_k \\ -b_j & 0 & a_j \\ 0 & a_j & -b_j \end{bmatrix} \begin{bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{bmatrix} \tag{7a}$$

or in symbolic form

$$[S] = [B][\sigma] \tag{7b}$$

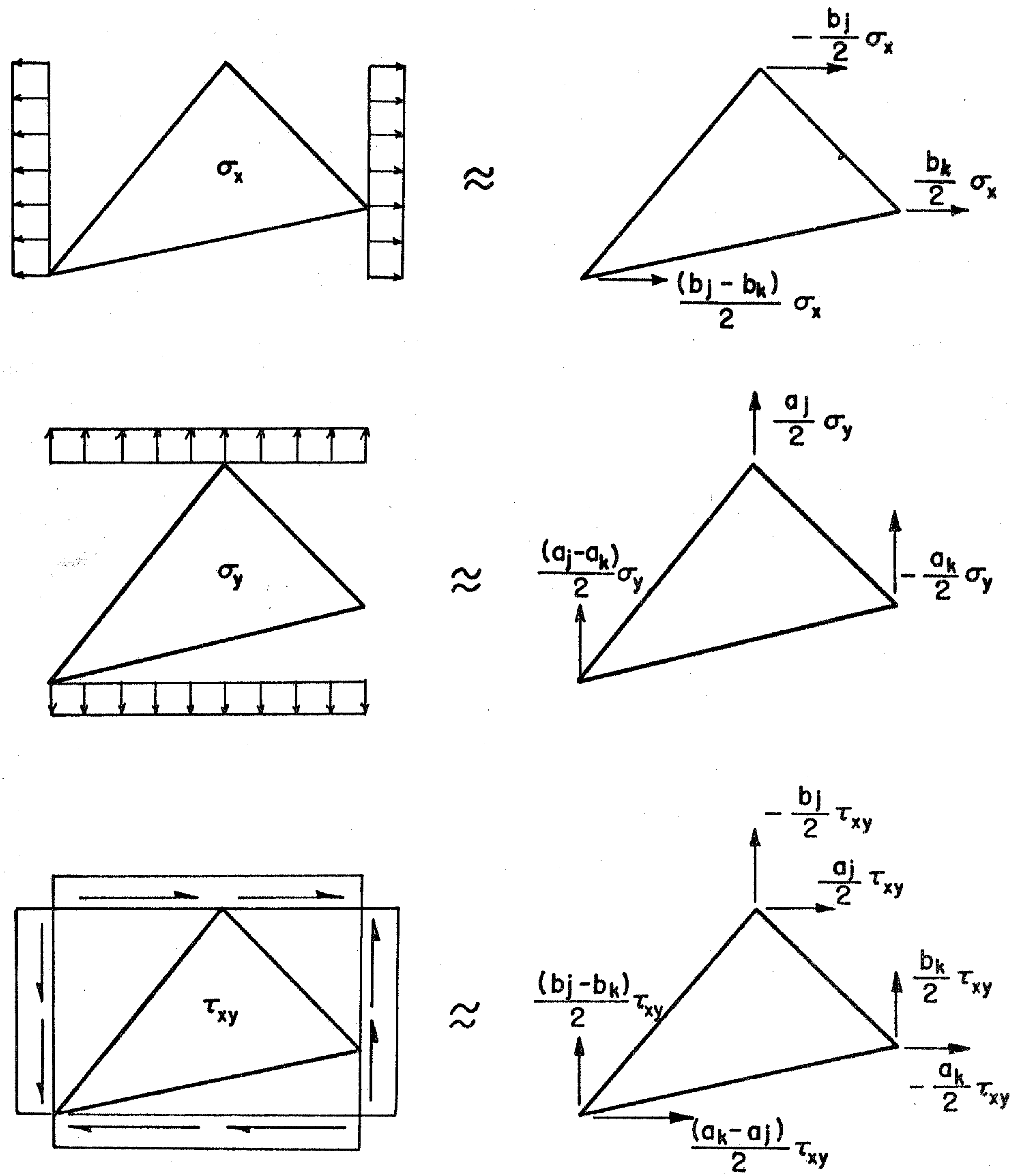


FIG. 4 - STRESS RESULTANTS

<u>Element Stiffness</u> - Element stresses can be expressed in terms of corner displacements by substituting Eq. 4b into Eq. 5b. Or

$$\left[\sigma\right] = \left[C\right] \left[A\right] \left[r\right] \tag{8}$$

The substitution of Eq. 8 into Eq. 7b yields

$$\left[S\right] = \left[B\right] \left[C\right] \left[A\right] \left[r\right] \tag{9}$$

Eq. 9, which is an expression for corner forces in terms of corner displacements, can be rewritten in the following form:

$$\left[S\right] = \left[k\right] \left[r\right] \tag{10}$$

where $\left[k\right]$ is the 6x6 stiffness matrix* for the element and is given by

$$\left[k\right] = \left[B\right] \left[C\right] \left[A\right] \tag{11a}$$

It should be pointed out that the algebraic manipulation necessary to establish the individual terms of this stiffness matrix for a typical element is not required, since within the computer program these stiffness matrices are numerically formed by standard matrix subroutines.

---

*This stiffness matrix could have been derived without the development of a force transformation matrix $\left[B\right]$. The more direct approach involves energy considerations and leads to the following equation:

$$\left[k\right] = \int \left[A\right]^T \left[C\right] \left[A\right] dv \tag{11b}$$

Where the integral is evaluated over the volume of the triangle. The only purpose of selecting the stress resultant approach is to give a physical interpretation of the method.

Equilibrium Equations For Complete Structure

The equilibrium of the complete system of elements, which is an expression for nodal point loads in terms of nodal point displacements, can be expressed by the following matrix equation:

$$\left[R\right] = \left[K\right]\left[r\right] \qquad (12)$$

where the stiffness of the complete structure $\left[K\right]$ can be found by a systematic addition of the stiffnesses of all elements in the system. This addition can best be illustrated if Eq. 10 is rewritten in terms of a typical element q

$$\begin{bmatrix} S_i^{(q)} \\ S_j^{(q)} \\ S_k^{(q)} \end{bmatrix} = \begin{bmatrix} k_{ii}^{(q)} & k_{ij}^{(q)} & k_{ik}^{(q)} \\ k_{ji}^{(q)} & k_{jj}^{(q)} & k_{jk}^{(q)} \\ k_{ki}^{(q)} & k_{kj}^{(q)} & k_{kk}^{(q)} \end{bmatrix} \begin{bmatrix} r_i \\ r_j \\ r_k \end{bmatrix} \qquad (13)$$

where, in terms of arbitrary nodal points l and m, $S_l^{(q)}$ and r_m are vectors of the form

$$S_l^{(q)} = \begin{bmatrix} S_x \\ S_y \end{bmatrix}_l^{(q)} \qquad (14a)$$

$$r_m^{(q)} = \begin{bmatrix} r_x \\ r_y \end{bmatrix}_m^{(q)} \qquad (14b)$$

and the stiffness coefficient $k_{lm}^{(q)}$ is a 2x2 submatrix of the form

$$k_{lm}^{(q)} = \begin{bmatrix} k_{xx} & k_{xy} \\ k_{xy} & k_{yy} \end{bmatrix}_{lm}^{(q)} \qquad (14c)$$

The term $k_{lm}^{(q)}$ represents the forces developed on element q at nodal point l due to unit displacements at nodal point m. Therefore, the general stiffness term, K_{lm}, for the complete structure, which is the sum of the forces acting on all elements at nodal point l due to unit displacements at nodal point m, is given by

$$K_{lm} = \sum_{q} k_{lm}^{(q)} \tag{15}$$

It should be pointed out that K_{lm} exists only if l equals m, or if l and m are adjacent nodal points in the physical system.

Solution of Equilibrium Equations

For most practical problems, Eq. 12 represents a system of several hundred equations. Three major difficulties are encountered in a direct solution of such a large set of equations. First, the storage required by the complete stiffness matrix is equal to N^2, where N is the number of equations. Second, the time required for solution is proportional to N^3. Third, accuracy of solution can be a serious problem. Even on a large computer, such as the IBM 7090, these problems still exist; however, these difficulties can be minimized by the use of iterative methods.

Iterative Procedure - The specific iterative method used is a modification of the well-known Gauss-Seidel iteration procedure which, when applied to Eq. 12, involves the repeated calculation of new displacements from the equation,*

$$r_n^{(s+1)} = K_{nn}^{-1}\left[R_n - \sum_{i=1,n-1} K_{ni}\, r_i^{(s+1)} - \sum_{i=n+1,N} K_{ni}\, r_i^{(s)}\right] \tag{16}$$

---

*The stiffness matrix [K] is positive definite; therefore, the method will always converge (1).

where n is the number of the unknown and s is the cycle of iteration.

The only modification of the procedure introduced in this analysis is the application of Eq. 16 simultaneously to both components of displacement at each nodal point. Therefore, r_n and R_n become vectors with x and y components, and the stiffness coefficients may be expressed in the 2x2 submatrix form of Eq. 14c.

<u>Over-Relaxation Factor</u> - The rate of convergence of the Gauss-Seidel procedure can be greatly increased by the use of an over-relaxation factor (8). However, in order to apply this factor it is first necessary to calculate the change in the displacement of nodal point n between cycles of iteration:

$$\Delta r_n^{(s)} = r_n^{(s+1)} - r_n^{(s)} \tag{17}$$

The substitution of Eq. 16 into Eq. 17 yields for the change in displacement

$$\Delta r_n^{(s)} = K_{nn}^{-1} \left[R_n - \sum_{i=1,n-1} K_{ni}\, r_i^{(s+1)} - \sum_{i=n,N} K_{ni}\, r_i^{(s)} \right] \tag{18}$$

The new displacement of nodal point n is then determined from the following equation:

$$r_n^{(s+1)} = r_n^{(s)} + \beta\, \Delta r_n^{(s)} \tag{19}$$

where β is the over-relaxation factor.

The selection of an over-relaxation factor, which gives the best convergence, depends on the characteristics of the particular problem. However, experience has indicated that for most two-dimensional structures the optimum over-relaxation factor is between 1.8 and 1.95.

<u>Group Relaxation</u> - Southwell (9) has illustrated the advantage of block and group relaxation techniques. However, these methods are based on a physical intuition of the specific structure being analyzed. The incorporation of such an approach into a general computer program does not seem practical. Therefore, a group relaxation method, which does not depend on a specific property of the structure, is developed. This procedure is similar to Rayleigh's energy method (10) which is used in the calculation of critical loads and frequencies.

After s cycles of iteration, it is assumed that $\delta\left[r^{(s)}\right]$ represents a good approximation of the final displacements of the system. In order to solve for δ it is necessary to consider the energy of the system when subjected to this deformation pattern. The energy which is supplied externally to the system is given by

$$U_e = \delta\left[r^{(s)}\right]\left[R\right] \tag{20}$$

The energy stored elastically within the elements of the system will be

$$U_i = \delta^2 \left[r^{(s)}\right]^T\left[K\right]\left[r^{(s)}\right] \tag{21}$$

Now, if the internal and external energy is equated, δ is found to be

$$\delta = \frac{\left[r^{(s)}\right]\left[R\right]}{\left[r^{(s)}\right]^T\left[K\right]\left[r^{(s)}\right]} \tag{22}$$

Therefore, before the start of the next cycle of iteration, the displacements may be modified as follows:

$$\left[r^{(s)}\right]^* = \delta \left[r^{(s)}\right] \tag{23}$$

The determination of δ involves approximately the same number of numerical operations as one cycle of Gauss-Seidel iteration. However, this "group relaxation" need only be applied once every several cycles.

Physical Interpretation of Method - There is important physical significance in the terms of Eq. 18. The term $(k_{nn})^{-1}$ is the flexibility of nodal point n. This represents the nodal point displacements resulting from unit nodal point forces, and can be written in the form of a submatrix

$$K_{nn}^{-1} = \begin{bmatrix} f_{xx} & f_{xy} \\ f_{yx} & f_{yy} \end{bmatrix} \tag{24}$$

The summation terms in Eq. 18 represent the elastic forces acting at nodal point n due to the deformations of the plate elements:

$$Q_n^{(s+1)} = \sum_{i=1,n} K_{ni}\, r_i^{(s+1)} + \sum_{i=n+1,N} K_{ni}\, r_i^{(s)} \tag{25}$$

The difference between these elastic forces and the applied loads is the total unbalanced force which in submatrix form may be written:

$$\begin{bmatrix} X \\ Y \end{bmatrix}_n^{(s+1)} = \begin{bmatrix} R_x \\ R_y \end{bmatrix}_n + \begin{bmatrix} Q_x \\ Q_y \end{bmatrix}_n^{(s+1)} \tag{26}$$

Eq. 19, which gives the new displacement of nodal point n, may now be rewritten in the following sub-matrix form:

$$\begin{bmatrix} r_x \\ r_y \end{bmatrix}_n^{(s+1)} = \begin{bmatrix} r_x \\ r_y \end{bmatrix}_n^{(s)} + \beta \begin{bmatrix} f_{xx} & f_{xy} \\ f_{yx} & f_{yy} \end{bmatrix}_n \begin{bmatrix} X \\ Y \end{bmatrix}^{(s+1)} \tag{27}$$

With β equal to one, the application of this equation is physically equivalent to releasing nodal point n and permitting it to move freely to a new equilibrium position. With β greater than one, the nodal point is moved beyond its equilibrium position before proceeding to the next point.

It is important to note that any desired nodal point displacement $r_n^{(o)}$ may be assumed for the first cycle of iteration. A good choice of these displacements will greatly speed the convergence of the solution. In fact, if all displacements were assumed correctly, the unbalanced forces given by Eq. 26 would be zero and no iteration would be necessary. However, in a practical case there always will be unbalanced forces in the system at first, and the iteration process continually reduces them toward zero.

Boundary Conditions

Equation 27 is valid for all nodal points which are free to move in both the x and y directions; however, in order for it to be applied to boundary nodal points the flexibility coefficients must be modified to account for the specific types of restraint which may exist. Since these flexibility coefficients are independent of the cycle of iteration, this modification is performed before the start of iteration.

For a general boundary nodal point that is free to move along a line which makes an angle ϕ with the x-axis, Fig. 5 illustrates the forces and displacements associated with the application of Eq. 23. The unbalanced forces X and Y are determined from Eq. 26. The unknown reaction is represented by R.

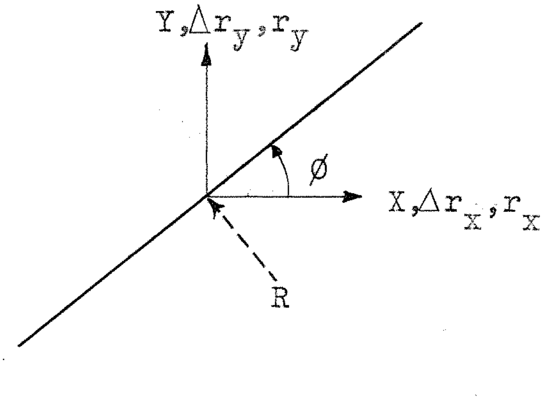


Fig. 5 GENERAL BOUNDARY POINT

Applying Eq. 18 to this nodal point, the displacements Δr_x and Δr_y are expressed in the form

$$\begin{vmatrix} \Delta r_x \\ \\ \Delta r_y \end{vmatrix} = \begin{vmatrix} f_{xx} & f_{xy} \\ \\ f_{yx} & f_{yy} \end{vmatrix} \begin{vmatrix} X - R \sin \phi \\ \\ Y + R \cos \phi \end{vmatrix}$$

Eliminating the unknown reaction R yields

$$\Delta r_x = \frac{f_{xx} - \alpha\, f_{yx}}{1 - \alpha \tan \phi}\, X + \frac{f_{xy} - \alpha\, f_{yy}}{1 - \alpha \tan \phi}\, Y \qquad (29a)$$

where

$$\alpha = \frac{f_{xx} \tan \phi - f_{xy}}{f_{xy} \tan \phi - f_{yy}}$$

Also, by definition,

$$\Delta r_y = \Delta r_x \tan \phi \qquad (29b)$$

Therefore, the effective flexibility coefficients are given by

$$\begin{aligned} f^*_{xx} &= \frac{f_{xx} - \alpha\, f_{xy}}{1 - \alpha \tan \phi} \\ f^*_{xy} &= \frac{f_{xy} - \alpha\, f_{yy}}{1 - \alpha \tan \phi} \\ f^*_{yx} &= \tan \phi\, f^*_{xx} \\ f^*_{yy} &= \tan \phi\, f^*_{xy} \end{aligned} \qquad (30a)$$

For points fixed in both x and y directions, $\Delta r_x = \Delta r_y = 0$, the modified flexibility coefficients will be

$$f^*_{xx} = f^*_{xy} = f^*_{yx} = f^*_{yy} = 0 \qquad (30b)$$

Nodal Point Loads

Three types of loadings will be discussed here--dead loads, live loads and thermal loads. In all cases loads must be reduced to concentrated nodal point forces.

Dead Loads - The dead weight of each plate element is given by the product of its area, its thickness (which is taken as unity) and its unit weight. It is assumed that the element's weight is distributed equally to each of the three nodal points to which the element is attached. Therefore, the total dead load at any nodal point is taken as one-third of the weights of all elements attached at that point, applied in a downward, or negative "y", direction.

Live Loads - Forces acting on the boundary of the structure are replaced by statically equivalent concentrated forces acting at the nodal points of the finite element system.

Thermal Loads - The thermal stress analysis of a finite element system is divided into two parts. First, assuming all nodal points are restrained, the stresses developed within all elements due to temperature changes are determined. In a plane strain system these stresses for a typical element are given by

$$\sigma_x = \sigma_y = \sigma_t = \frac{E\,\alpha_t}{(1+\nu)(1-2\nu)}\,\Delta T \qquad (31)$$

where α_t = thermal coefficient of expansion

ΔT = change of temperature

For a typical element the corner forces which are necessary to maintain these stresses are determined from Eq. 7b.

$$\begin{bmatrix} R_x^i \\ R_y^i \\ R_x^j \\ R_y^j \\ R_x^k \\ R_y^k \end{bmatrix} = -\frac{1}{2} \begin{bmatrix} b_k - b_j \\ a_j - a_k \\ -b_k \\ a_k \\ b_j \\ -a_j \end{bmatrix} \sigma_t \qquad (32)$$

Second, in order to eliminate these forces the system is analyzed for nodal point loads which are equal in magnitude but opposite in sign to these restraining forces. The final thermal stress distribution is the sum of stresses due to these thermal loads and the initial stresses in the restrained system.

Element Stresses

Equation 8 expresses the three components of stress σ_x, σ_y, and τ_{xy} within a typical element in terms of the six corner displacements of the element. Therefore, after Eq. 12 is solved for the nodal point displacements of the finite element system, the stresses within each element are determined by the direct application of Eq. 8 to all elements of the system.

Although the normal and shear stresses (σ_x, σ_y, & τ_{xy}) completely define the state of stress in the elements, it frequently is of interest to know the principal stresses σ_1, and σ_2 and their directions θ. These principal stress values are determined from the stress values related to the x, y coordinate axes by the standard transformation formulas of elementary mechanics.

Nodal Point Stresses

Practical application of the method indicates that the computed nodal point displacements are realistic but, unless a very fine mesh is used, considerable difficulty may be encountered in plotting and evaluating element stresses. Furthermore, it is often desirable to obtain nodal point stresses, since maximum stresses normally are developed on the boundaries of a structure. Therefore, the purpose of this section is to introduce a method of determining nodal point stresses.

It has been shown that nodal point stresses, obtained by averaging the element stresses of all elements connected to the nodal point, produce good results for interior nodal points; however, this approach breaks down when applied to boundary nodal points (11).

Experience has indicated that the three components of element stresses do not represent the state of stress at any one point within the element. For example, consider the element shown in Fig. 6. Since stresses must be consistent

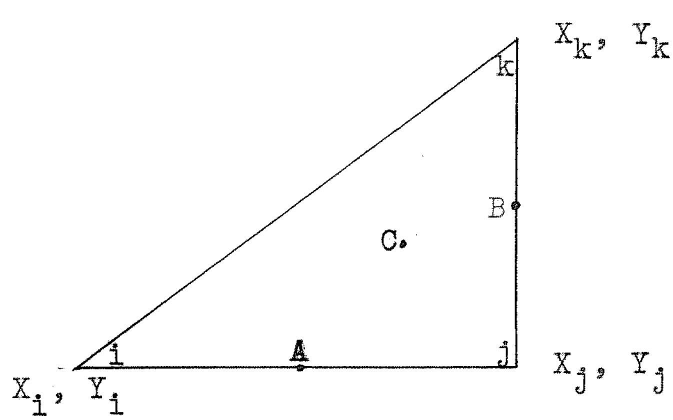


Fig. 6 TYPICAL ELEMENT n ATTACHED TO NODAL POINT i

with the nodal point displacements, σ_x approximates the horizontal stress of point A, σ_y approximates the vertical stress at point B, and τ_{xy} approximates the shearing stress at some interior point C. For this case, in determining stresses at nodal point i, it is apparent that the horizontal stress at A must be weighted more heavily than the vertical stress at B. Therefore, a "weighted average" method, which reflects this behavior, is used to determine nodal point stresses.

The method involves the following calculations to determine the three components of stress at point i.

$$\sigma_x = \frac{1}{S_x} \sum_n \frac{a^{(n)}}{a^{(n)} + b^{(n)}} \sigma_x^{(n)} \qquad (33a)$$

$$\sigma_y = \frac{1}{S_y} \sum_n \frac{b^{(n)}}{a^{(n)} + b^{(n)}} \sigma_y^{(n)} \qquad (33b)$$

$$\tau_{xy} = \frac{1}{N} \sum_n \tau_{xy}^{(n)} \qquad (33c)$$

where

$$a^{(n)} = \left| X_k^{(n)} + X_j^{(n)} - 2X_i \right|$$

$$b^{(n)} = \left| Y_k^{(n)} + Y_j^{(n)} - 2Y_i \right|$$

$$S_x = \sum \frac{a^{(n)}}{a^{(n)} + b^{(n)}}$$

$$S_y = \sum \frac{b^{(n)}}{a^{(n)} + b^{(n)}}$$

The summation is performed on all N elements connected at nodal point i.

The method is illustrated by several examples in the next section of this dissertation. In general, the procedure yields results which agree very closely with the direct averaging method for interior nodal points but differ considerably for boundary nodal points.

Solution of Complete Problem

The complete analysis of a two-dimensional structure by the finite element method involves three separate phases. First, the structure must be idealized by a system of triangular elements. Second, this system of elements must be solved for displacements and stresses. Third, the displacements and stresses must be illustrated in a graphical form in order to be evaluated conveniently.

The selection of the finite element system for a particular problem is completely arbitrary; therefore, structures with practically any shape boundaries may be considered. If all elements and nodal points are numbered, in any convenient manner, it is possible to define the system in the form of three numerical arrays--nodal point array, element array and boundary point array. The nodal point array contains the coordinates and the loads or displacements that are associated with each nodal point of the system. The element array contains, for each element in the system, the location of the element (the three nodal point numbers where the element is attached) and other possible parameters which are associated with the element (i.e. elastic constants, density and temperature changes). The boundary array indicates the type of restraint that exists at boundary nodal points.

These three arrays, along with some basic control information, constitute the numerical input for the digital computer program, which determines the displacements and stresses of the finite element system. The most important characteristic of this program is that only the non-zero coefficients of the stiffness matrix are developed and retained; therefore, it is possible to treat large systems without exceeding the storage capacity of the computer. In addition to nodal point displacements, the three coordinate stresses (σ_x, σ_y, τ_{xy}) and the principal stresses and directions (σ_1, σ_2, θ) for each element and nodal point of the system are generated by the program. The details involved in the use of this program and a Fortran listing of the program are included in the Appendix.

The computer output may be plotted in two forms--stress vectors at various points in the structure, which illustrate the magnitudes and directions of the principal stresses; or stress contours, for any component of stress, which are lines connecting points of equal stress. In general, it is this phase, the plotting and evaluation of results, which is the most time consuming part of the complete analysis of a structure.

VALIDITY OF METHOD AND APPLICATION TO LINEAR SYSTEMS

Before the finite element method is extended to nonlinear problems, several basic questions regarding its application to linear systems must be answered. First, in approximating a continuous structure by a system of triangular elements, does the behavior of the finite element system converge to the behavior of the continuous structure as the dimensions of the elements become infinitesimal? Second, what is the order of magnitude of the errors which are inherent in the finite element approximation? Third, for a given structure, how does the finite element solution compare with an exact solution? In this section these questions are discussed and the overall validity of the method is illustrated by several examples.

Convergence

In order to compare the behavior of an element system of infinitesimal dimensions to the behavior of a continuous structure, it is necessary to examine separately the equilibrium conditions, stress-strain relationships and compatibility requirements which are imposed on a system of finite elements.

Based on stress-resultants, equilibrium is satisfied only at the nodal points of the finite element system. However, it is apparent that the errors introduced by approximating stresses by stress-resultants are reduced to zero as the dimensions of the elements become infinitesimal. It should be pointed out the overall statics (summation of forces across any section) are satisfied, regardless of the mesh size.

The stress-strain relationship, Eq. ~~52~~ 5a, is independent of the element size. Since this is not a source of error in the finite element approximation, the method can be readily extended to structures with nonlinear material properties.

Since the sides of the finite elements remain straight after deformation, continuity of u and v is always maintained, regardless of the size of the finite elements. Compatibility of the continuous problem is normally obtained by requiring that the following equation be satisfied:

$$\frac{\partial^2 \varepsilon_x}{\partial y^2} + \frac{\partial^2 \varepsilon_y}{\partial x^2} = \frac{\partial^2 \gamma}{\partial x \, \partial y} \qquad (34)$$

This is a restriction on the strains, or the derivatives of u and v; therefore, for certain types of problems (multiply-connected bodies) this requirement has been found to be insufficient to establish a unique solution, due to the possibility of discontinuities in the displacements. It is apparent that the finite element solution does not have such a limitation.

Error Term

Equation 4a, which can be thought of as a finite difference operator, expresses element strains (derivatives of displacements) in terms of nodal point displacements. The derivation of this equation is based on physical compatibility requirements; however, the same equation could have been developed from a mathematical approach.

A Taylor series expansion, in the case of a function $f_{(x,y)}$, is written in the following symbolic form:

$$f_{(x+h,y+k)} = f_{(x,y)} + \left(h \frac{\partial}{\partial x} + k \frac{\partial}{\partial y}\right) f'_{(x,y)} + \frac{1}{2!}\left(h \frac{\partial}{\partial x} + k \frac{\partial}{\partial y}\right)^2 f''_{(x,y)} + --- \qquad (35)$$

Now, if second and higher order derivatives are neglected, the application of this equation at nodal point i of a typical triangle, Fig. 2, yields the following equations:

$$\begin{aligned} u_j &= u_i + a_j \frac{\partial u}{\partial x} + b_j \frac{\partial u}{\partial y} \\ u_k &= u_i + a_k \frac{\partial u}{\partial x} + b_k \frac{\partial u}{\partial y} \\ v_j &= v_i + a_j \frac{\partial v}{\partial x} + b_j \frac{\partial v}{\partial y} \\ v_k &+ v_k + a_k \frac{\partial v}{\partial x} + b_k \frac{\partial v}{\partial y} \end{aligned} \qquad (36)$$

Eqs. 3 and 36 are now combined to yield Eq. 4a.

The importance of this approach is that the errors which are inherent in the approximation are apparent. Typically, the error term for the displacement u is of the form

$$E_{(u)} = \frac{1}{2!}\left(a_j^2 \frac{\partial^2 u}{\partial x^2} + 2\, a_j\, b_j \frac{\partial^2 u}{\partial x \partial y} + b_j^2 \frac{\partial^2 u}{\partial^2 y^2}\right) \qquad (37)$$

It is evident that near stress concentrations, where the rate of change of strain (second derivative of displacement) is large compared with the strain, errors may be large, unless the mesh size is small. Also, from a numerical analysis standpoint it can be said that the triangular finite element approximation is a "first order method with an error term of order h^2."

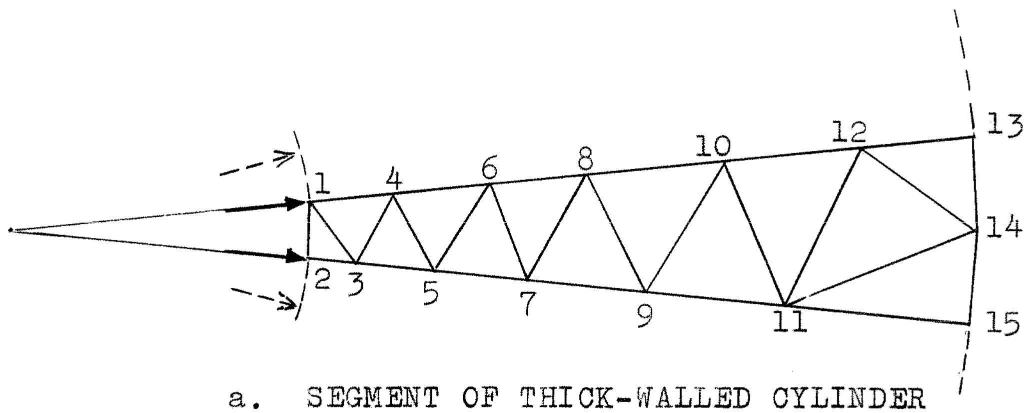
It is of practical importance to note that errors in stresses for structural problems do not appear to accumulate. The overall stress distribution is not affected by local errors at stress concentrations.

Examples - Linear Structures

Thick-Walled Cylinder - A thick-walled cylinder subjected to uniform internal pressure is selected to illustrate the general application of the method. A segment of the cylinder, as shown in Fig. 7a, was idealized as a system of 13 elements and 15 nodal points. The internal pressure was approximated by concentrated forces acting at nodal points 1 and 2. By utilizing the procedure suggested by Eqs. 30 the boundary points were allowed to move only in a radial direction.

The tangential and radial stresses are given in Figs. 7b and 7c. Considering the very coarse mesh idealization, there is good agreement with the exact solution. As expected, the errors are largest near the application of the loads. The radial displacement at the point of load application was in error by approximately .5%.

This example, when treated by classical methods, is best formulated in polar coordinates. However, the finite element procedure, which is basically formulated in rectangular coordinates, is readily applied to this class of problems. This ability of the finite element method to handle arbitrary boundary conditions is an important advantage over other numerical techniques.



a. SEGMENT OF THICK-WALLED CYLINDER

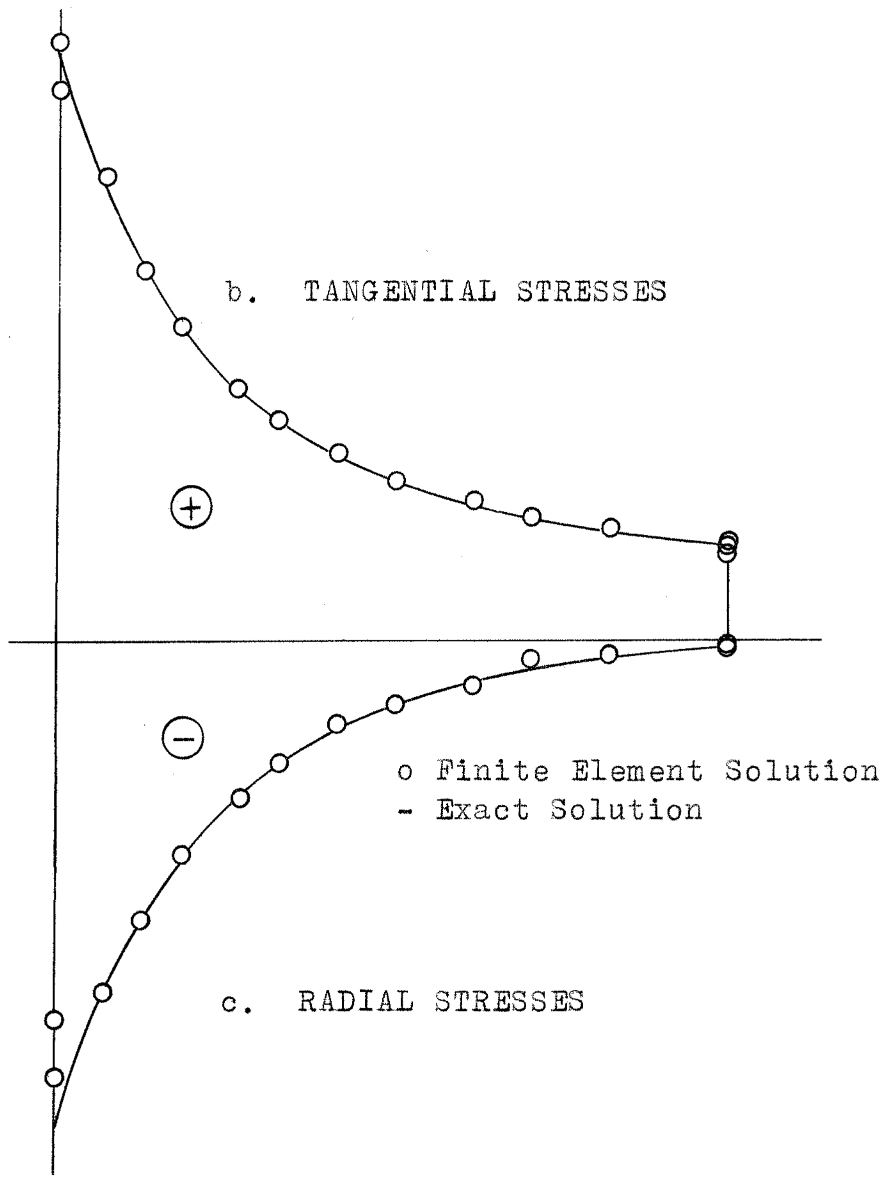
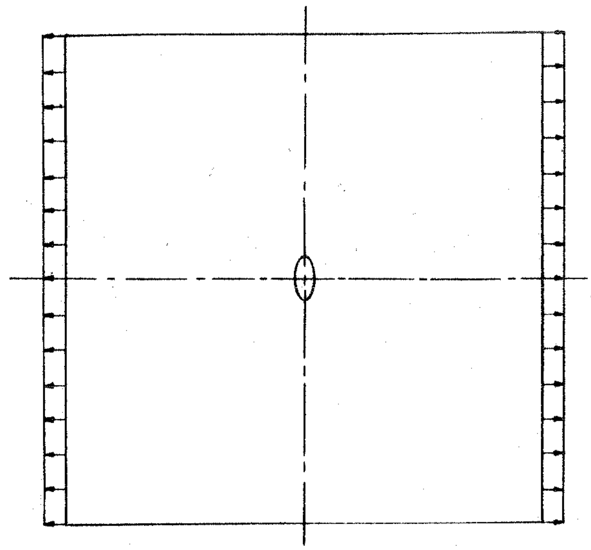


FIG. 7 THICK-WALLED CYLINDER

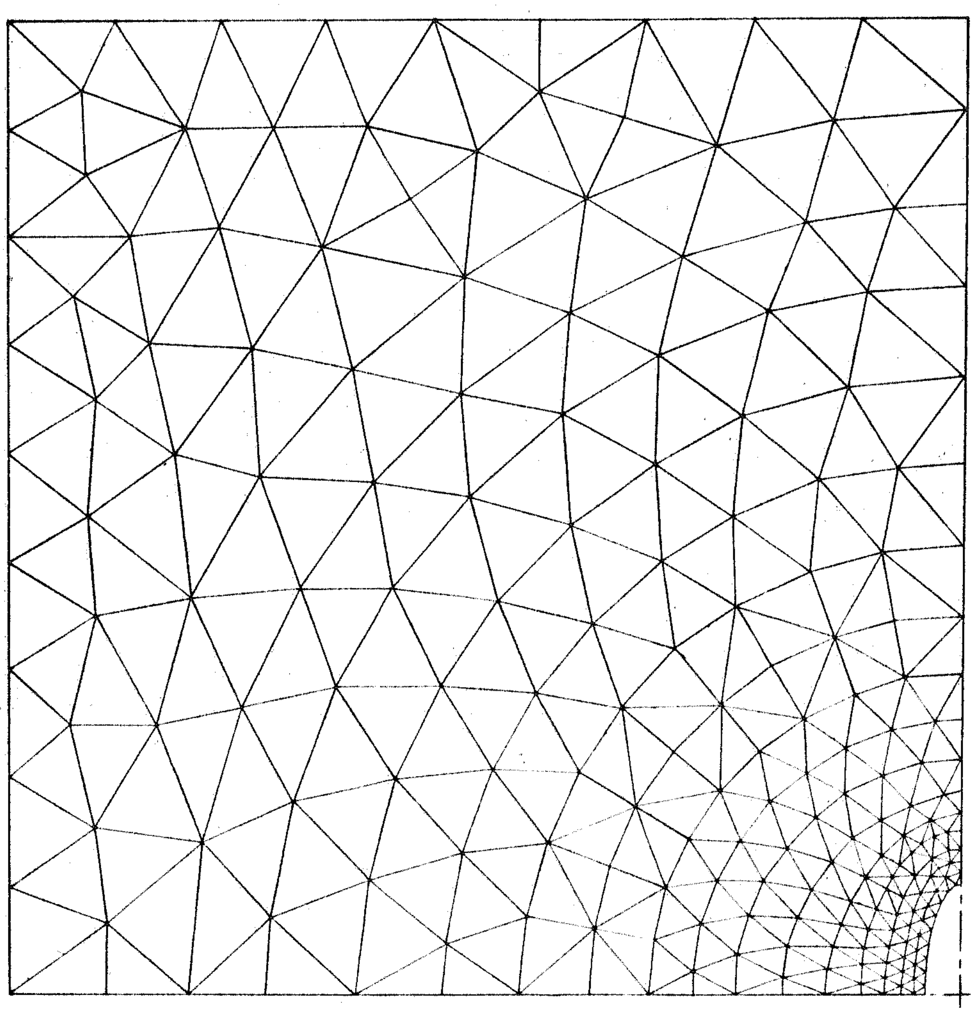
Infinite Plate With Elliptic Hole - A square plate, with a central elliptic hole, subjected to uniform tensile stresses as shown in Fig. 8a was analyzed by the finite element method. The ratio of major principal axis to minor principal axis is three. The ratio of length of major axis to the length of the plate is 12. The finite element layout for one-fourth the plate is illustrated in Fig. 8b. This idealization involves 384 elements and 224 nodal points. Therefore, a solution of over 400 equations is required. A solution, accurate to three significant figures, was obtained in 65 cycles of Gauss-Seidel iteration with an over-relaxation factor of 1.87. The total computer time required to form the stiffness matrix and solve for the unknown displacements and stresses was 50 seconds for the IBM 7090.

Since the dimensions of the plate are large, compared to the size of the elliptic hole, the solution is comparable with the exact solution for the infinite plate. Figure 9 shows stress contours for major and minor principal stresses and normal stress on the x and y axes. Again, agreement with the infinite plate solution is good, except at the stress concentration near the hole where the finite element method yields 564, as compared with the exact value of 700.

This example demonstrates another advantage of the finite element method--flexibility in mesh layout. In this case, in order to reduce errors, the elements are small near points of expected stress concentrations and large where the rate of change of stresses is expected to be small. Therefore, "engineering judgment" in selecting the mesh layout for a particular problem may be utilized effectively.



a. SQUARE PLATE WITH ELLIPTIC HOLE



b. ELEMENT LAYOUT

FIG. 8 - IDEALIZATION OF INFINITE PLATE WITH ELLIPTIC HOLE

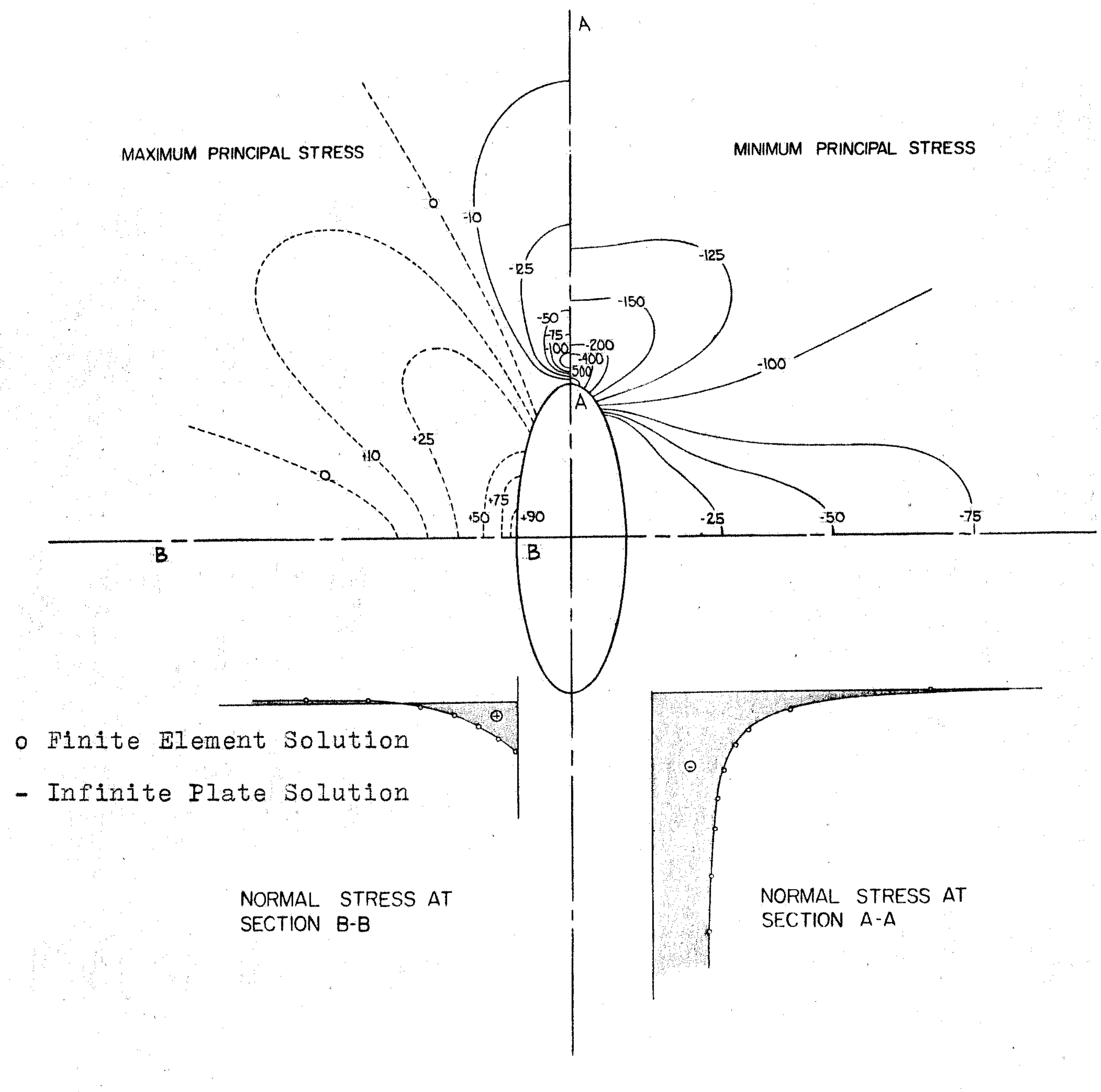


FIG. 9 - STRESS DISTRIBUTION - INFINITE PLATE WITH ELLIPTIC HOLE

Gravity Dam - An extensive investigation, utilizing the finite element method of analysis, has been made to determine the effect of a vertical crack on the displacements and stresses of a gravity dam. A complete report of this study, which involves 27 separate analyses, is given in Reference (12). Analyses were performed for various combinations of live loads, dead loads and thermal loads, for several crack heights, and for two types of foundation material (isotropic and orthotropic). For the purpose of demonstrating the application of the finite element method to a practical problem only one of these two analyses will be included here.

The dam, a 230 foot high concrete structure, was idealized by two systems of triangular finite elements, as shown in Fig. 10. Each analysis was divided into two parts--a coarse mesh analysis to determine foundation displacements (along the dashed line in the base) and a fine mesh analysis with these foundation displacements applied as boundary conditions. In this way the effects of foundation deformations were included in the fine mesh analysis without the need of additional elements.

The specific analysis considered here is for a section cracked to 7/9 of its height and subjected to live load and dead load. The principal stresses within each element, plotted in vector form, are given in Fig. 11. Perhaps a more readable form of the results is given by Fig. 12 in the form of stress contours which are lines connecting points of equal stress. A statics check, which is the only form of engineering check that can readily be performed on a problem of this nature, yields errors in base forces of less than two per cent.

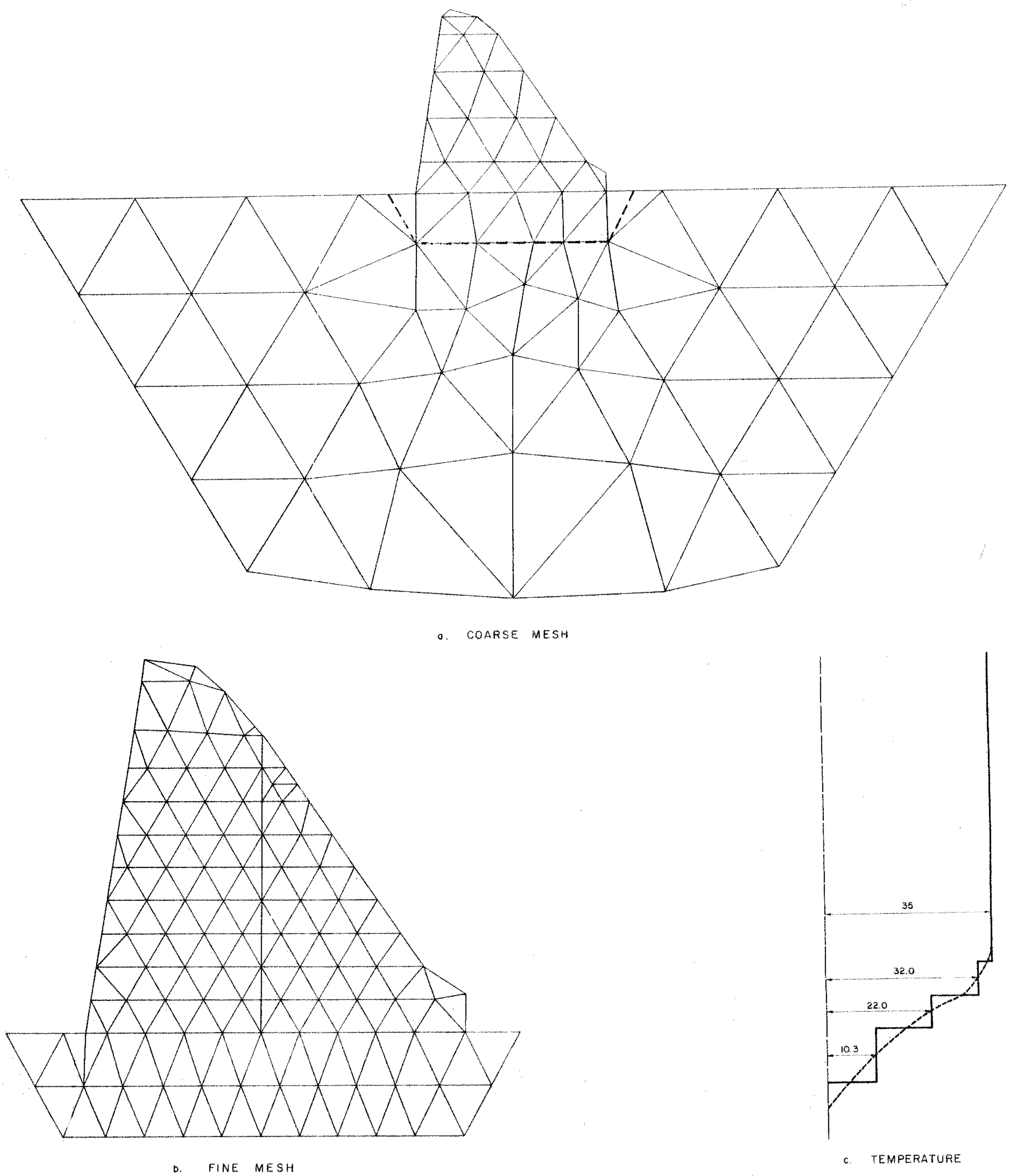


FIG. 10 FINITE ELEMENT IDEALIZATION — GRAVITY DAM

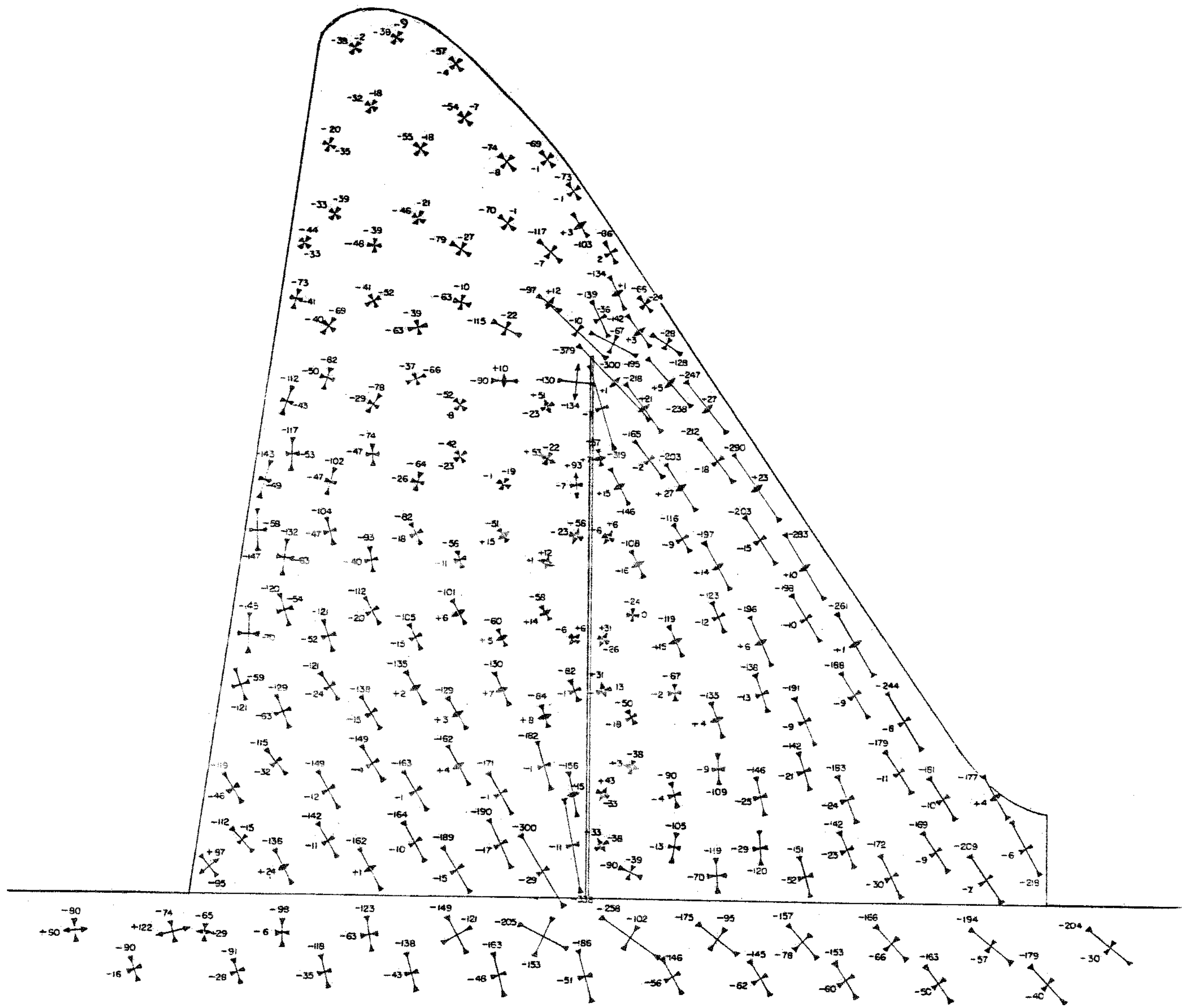


FIG. II STRESS VECTORS—GRAVITY DAM

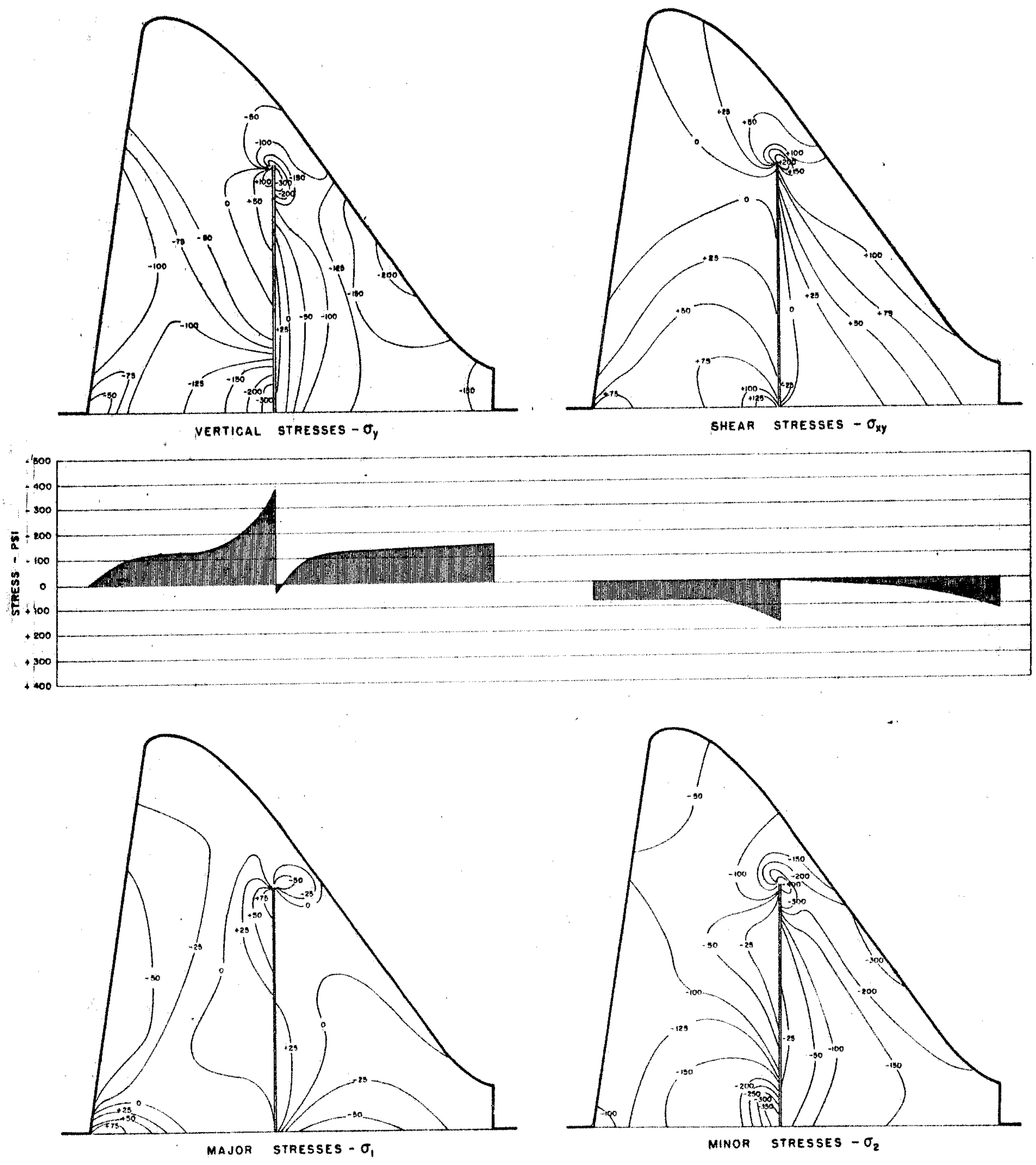
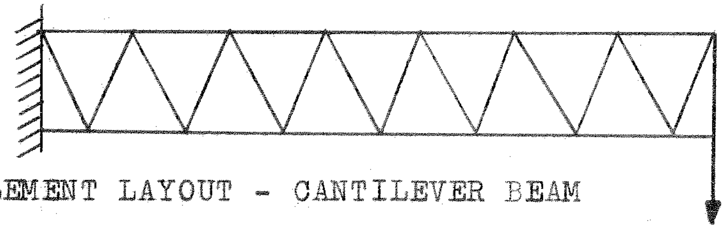


FIG. 12 STRESS CONTOURS—GRAVITY DAM

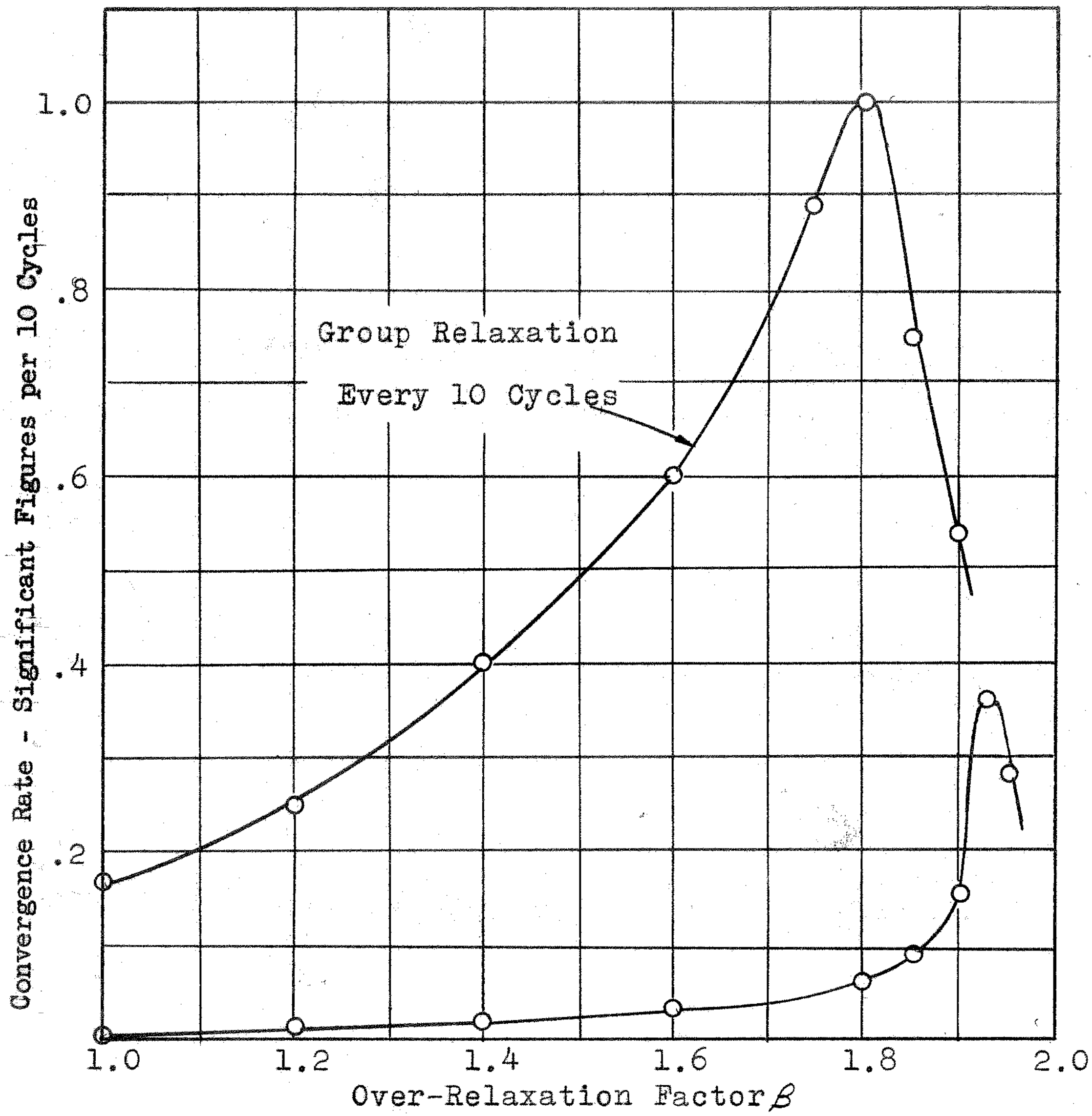
In addition to the practical significance, this example illustrates another favorable characteristic of the finite element approach--the combining of analyses of different mesh size. Therefore, in the region of stress concentrations, points where the errors from one analysis may be large, separate fine mesh studies may be made utilizing the results from a previous coarse mesh analysis.

<u>Cantilever</u> <u>Beam</u> - A very coarse finite element idealization of a cantilever beam is shown in Fig. 13a. This example was selected to illustrate the effect of over-relaxation and group relaxation on the convergences of the iterative procedure. Since the overall displacements of the beam are large, compared to the individual element deformations, the system is "poorly conditioned" from an iteration standpoint.

In general, the Gauss-Seidel procedure tends to converge at a fixed number of significant figures per cycle. This rate of convergence vs. over-relaxation factor is plotted in Fig. 13b. The same over-relaxation factor is used throughout any one iterative solution. In the case of straight over-relaxation the rate of convergence is extremely dependent on the over-relaxation factor and for this problem the optimum value of β is approximately 1.93. If a group relaxation, as previously described, is performed every 10 cycles of iteration the rate of convergence is improved considerably and the optimum value of the over-relaxation factor β is reduced to approximately 1.80. Another important effect of group relaxation is that the rate of convergence is not as dependent on the over-relaxation factor. For practical purposes, any value of β between 1.6 and 1.9 produces adequate convergence. For



a. ELEMENT LAYOUT - CANTILEVER BEAM



b. CONVERGENCE RATE VS. OVER-RELAXATION

FIG. 13 CANTILEVER BEAM

this example group relaxation was also applied every 5 cycles and every 15 cycles of iteration; this produced only minor changes in the convergence rates obtained by applying group relaxation every 10 cycles.

Since the group relaxation approach has been used only in the latter part of this investigation, it is impossible to draw any general conclusion as to its effectiveness. However, it should be pointed out that the basic formulation of the group relaxation procedure is independent of the type of structure. Therefore, it may be applied to structures such as plate and shells where difficulty has been traditionally encountered with relaxation procedures. Experience to date indicates that it may prove to be a very powerful adjunct of the iteration solution.

EXTENSION TO NONLINEAR STRUCTURES

The analysis of practical two-dimensional structures with nonlinear behavior is virtually impossible by classical methods. However, the finite element method may be extended to include structures with both nonlinear geometry and nonlinear material properties. Two possible solution techniques by which these problems may be treated are discussed here. First, the analysis of a nonlinear structure may be accomplished by a step-by-step procedure. This involves replacing the nonlinear analysis by the sum of linear analyses of separate structures, each subjected to a small increment of load. Second, the equilibrium position of a structure subjected to a given set of loads may be determined by a trial and error procedure. In this case the structure is solved repeatedly until the final geometry and stresses agree with the assumed geometry and stresses.

In Reference (11) a step-by-step matrix procedure, which includes the effect of large displacements, is presented for finite element systems. A disadvantage of this is that it is necessary to perform a complete linear analysis for each increment of load. Another form of geometry nonlinearity was encountered in the analysis of a cracked gravity dam. The dam, idealized as a system of triangular plate elements, was analyzed by a step-by-step procedure in which the water load was applied incrementally. Since the crack closed during the application of load, the behavior of the structure was nonlinear. The computer program for linear analysis, which is given in the Appendix, was modified to perform this analysis. A complete report of this study is given in Reference (12).

In the step-by-step approach, if a direct solution technique is employed for each increment of load the procedure may be numerically inefficient. It is possible to reduce this difficulty if an iterative solution is used, since the solution from the previous load increment serves as a good initial approximation for the next load increment. In fact, if the nonlinear effects are small, compared to the previous increment of load, a solution may be found in only a few additional cycles of iteration.

Similarly, in the successive approximation approach the iterative solution technique offers a distinct advantage. The solution from the previous approximation serves as a good initial guess for the next approximation. Therefore, as the procedure converges fewer and fewer cycles of iteration are required per solution.

The discussion in this section will be restricted to structures with nonlinear material properties. Examples of both the step-by-step and the successive approximation techniques are presented.

Step-By-Step Analysis

A step-by-step procedure has been successfully applied to the analysis of frames and trusses with nonlinear material properties (13). It is possible to use a similar approach in the analysis of systems composed of triangular plate elements. In this approach the effect of nonlinear material properties on the behavior of a structure subjected to load $\left[R\right]$ is approximated by the sum of a series of linear structures, each subjected to a load increment $\left[\Delta R\right]$. These structures have different stiffness characteristics due to the previously applied loads. For triangular finite element systems the stiffness properties are contained in the stress-strain relationship, Eq. 5. Therefore, the first step in the procedure is to determine the incremental stiffness for the element. The stiffness, which represents the changes in corner forces due to small changes in corner deformations, is a function of the magnitude of stresses within the element. For one-dimensional elements these incremental stiffnesses, in terms of a few basic parameters, are easily developed. However, for two-dimensional elements, because of the lack of experimental data, the procedure is more difficult. For this reason only a very idealized material will be considered here. However, it should be emphasized that this is not a limitation of the step-by-step approach, for any material can be considered if its biaxial stress-strain relationship is known.

After the incremental element stiffnesses are determined for step i, the analysis involves the following sequence of operations:

Formation of Complete Stiffness - The incremental stiffness $\left[K\right]_i$ for the complete structure is developed by the direct application of Eq. 15.

Determination of Incremental Displacements - The following equation is solved, by the previously described iterative procedure, for the incremental nodal point displacements:

$$\left[\Delta R\right]_i = \left[K\right]_i \left[\Delta r\right] \tag{38}$$

Determination of Incremental Element Stresses - The change in element stresses is determined from the incremental nodal point displacements by applying Eq. 8,

$$[\Delta\sigma]_i = [C]_i[A][\Delta r]_i \tag{39}$$

where $[C]_i$ is the incremental stress-strain relationship for step i.

Determination of Total Displacements and Stresses - The total nodal point displacements and element stresses are added to the displacements and stresses that were determined in the previous step.

$$[r]_i = [r]_{i-1} + [\Delta r]_i \tag{40a}$$

$$[\sigma]_i = [\sigma]_{i-1} + [\Delta\sigma]_i \tag{40b}$$

Based on these element stresses, a new incremental stress-strain relationship may be approximated for the next increment of load. This procedure may be repeated until any desired magnitude of load is applied to the structure.

It is apparent that any desired degree of accuracy may be obtained simply by decreasing the magnitude of the load increments. Also, the problem of uniqueness of solution is avoided since the incremental load approach automatically forces the structure into its most reasonable position.

The computer program for linear analysis, which is given in the Appendix, was modified to perform step-by-step analyses of finite element systems with nonlinear material properties. Since this is a simple modification that depends on the specific class of material that is being considered, the details of this nonlinear program will not be given.

Successive Approximations

Perhaps the most direct solution to structures with nonlinear material properties is one of successive approximations. Basically, this involves the repeated solution of the following equation for the displacements of the system:

$$\left[R\right] = \left[K\right]_{n-1} \left[r\right]_n \qquad (41)$$

where $\left[R\right]$ = the external loads acting on the system

$\left[K\right]_{n-1}$ = the effective stiffness of the system resulting from the previous approximation

$\left[r\right]_n$ = the displacements of the system for approximation n

The effective stiffness must be based on the results that were obtained in the previous approximation (n-1). Since deformations are assumed to be small, the development of the effective stiffness depends only on the estimation of an effective stress-strain relationship for each element in the structure.

It is apparent that this approach has several disadvantages. First, the procedure has no guarantee of converging. However, experience has indicated that for structures where the nonlinear effects are small, compared to the initial linear analysis, the procedure does converge. Second, in order to obtain a unique solution the method is restricted to <u>elastic</u> materials; or, in other words, materials with single-valued stress-strain relationships. This will insure a strain energy function which will be positive for all values of strain. Therefore, for small deformations, the classical proof (14) for uniqueness of solution can be applied. Third, stresses and displacements are determined for only one load magnitude, whereas the step-by-step analysis yields the complete behavior of the structures during the application of loads.

In spite of these limitations, the method of successive approximations can be applied to certain types of problems for which the step-by-step approach fails. An example of such a structure is one made of a material with bilinear properties--a material with different properties in tension and compression. In this case the stress-strain relationship is a function of the signs and directions of the principal stresses, rather than their magnitudes. The method of successive approximation was applied only to this type of material in this study.

Again, since each approximation involves a solution to a linear problem, the computer program for linear analysis is readily modified to include the effects of bilinear material properties. It is only necessary to reform the stiffness matrix after each approximation and to resolve the structure. In order to develop this stiffness matrix for the next approximation the sign and direction of the principal stresses must be examined and a new bilinear stress-strain relationship approximated.

Bilinear Stress-Strain Relationship - For a bilinear material there are three possible stress-strain relationships which depend on the state of stress.

Type I - Both σ_1 and σ_2 are compression

Type II - Both σ_1 and σ_2 are tension

Type III - σ_1 is tension and σ_2 is compression

For Type I and II conditions the stress-strain relationships, in the reference coordinate system x-y, are of the normal form. For Type III condition the stress-strain relationship is a function of the angle of the principal stresses θ. Therefore, this must be considered as a special case.

In terms of the principal coordinate system the stress-strain relationship is written in the form

$$\left[\bar{\sigma}\right] = \left[\bar{C}\right]\left[\bar{\varepsilon}\right] \tag{42}$$

In the case of Type III material this must be of the orthotropic form.

The principal strains are expressed in terms of strains in the x-y coordinate system by the following standard transformation:

$$\begin{bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \bar{\gamma} \end{bmatrix} = \begin{bmatrix} c^2 & s^2 & s\cdot c \\ s^2 & c^2 & -s\cdot c \\ 2\cdot s\cdot c & -2\cdot s\cdot c & s^2+c^2 \end{bmatrix} \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma \end{bmatrix} \tag{43a}$$

where $c = \cos\theta$

$s = \sin\theta$

or symbolically

$$\left[\bar{\varepsilon}\right] = \left[\bar{A}\right]\left[\varepsilon\right] \tag{43b}$$

It is of interest to point out that $\bar{\gamma}$ must be zero for all elements in the final solution of the structure; however, in order to maintain three degrees of freedom within each element it is necessary to retain it as a possible strain component. The final solution will be independent of the shear modulus which is associated with this shear strain.

Stresses in the x-y coordinate system are expressed in terms of principal stresses by the transformation

$$[\sigma] = [\bar{A}]^T [\bar{\sigma}] \tag{44}$$

After combining Eqs. 42, 43 and 44 the stress-strain relationship in the x-y coordinate system for a bilinear material is found to be

$$[\sigma] = [\bar{A}]^T [\bar{C}] [\bar{A}] [\varepsilon] \tag{45}$$

<u>Examples of Nonlinear Structures</u>

<u>Step-by-Step Analysis of Square Plate</u> - A square plate, with a central circular hole, subjected to a uniform tensile stress is shown in Fig. 14a. It is assumed that the plate is made of a material which has a stress-strain relationship of the following form: the material behaves linearly with a modulus E_1 until some yield condition is reached; beyond this condition the material behaves linearly with a reduced modulus E_2. For any one element this yield condition is defined as the point when the strain energy density reaches a predetermined value. When the ratio E_1/E_2 is large this reduces to the normal assumption which is made in ideal plasticity. In this example E_1/E_2 is set equal to 500.

The finite element idealization of one-fourth the plate is given by Fig. 14b. It is important to point out that it was possible to reduce the mesh size in the region of the plate where the nonlinear behavior was expected.

Eighteen equal load increments were applied to the structure. The displacement of point B vs. load increment is shown in Fig. 14c. The stress distributions on section A-A for various load conditions are shown in Fig. 14d. Since solutions

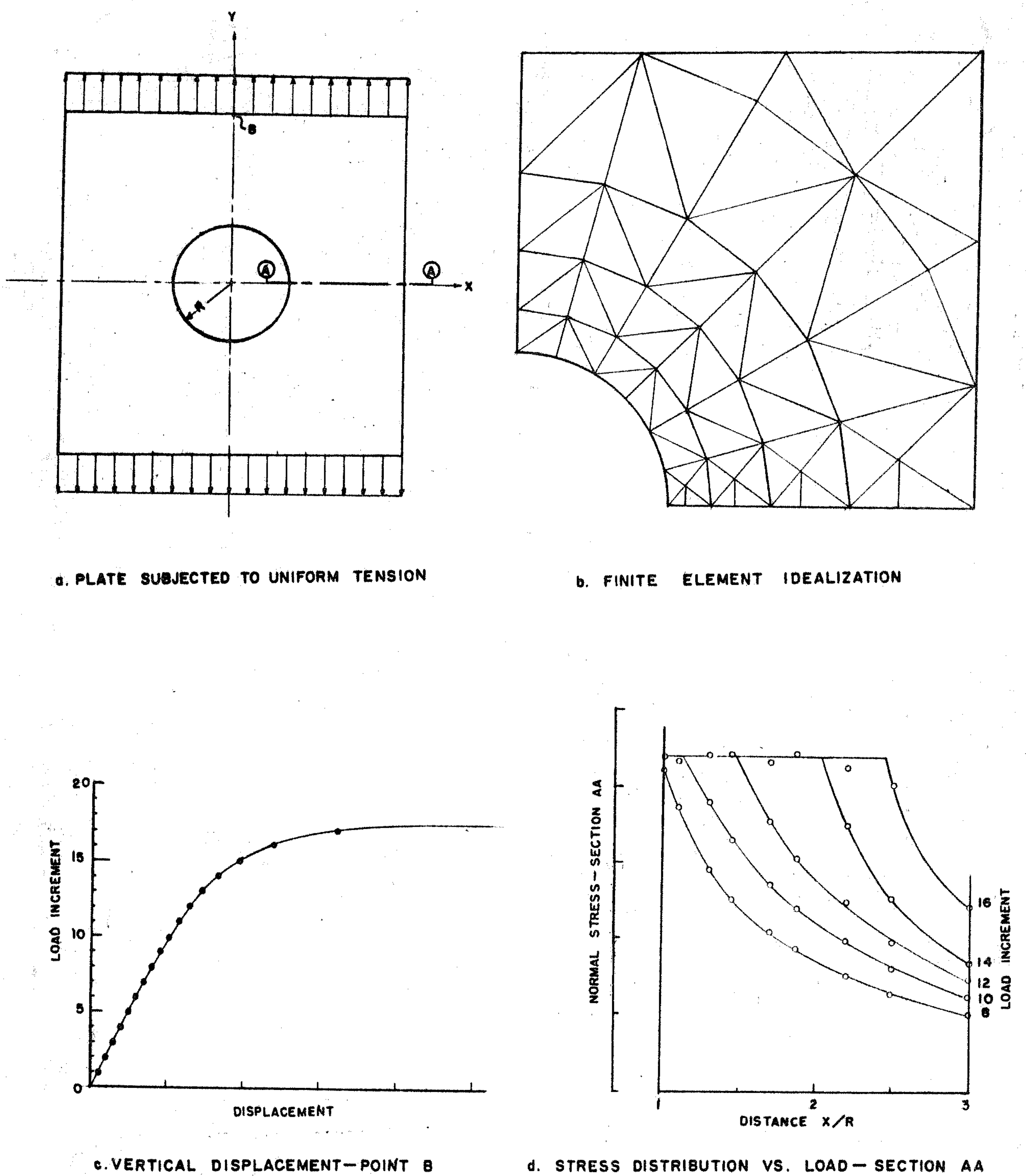


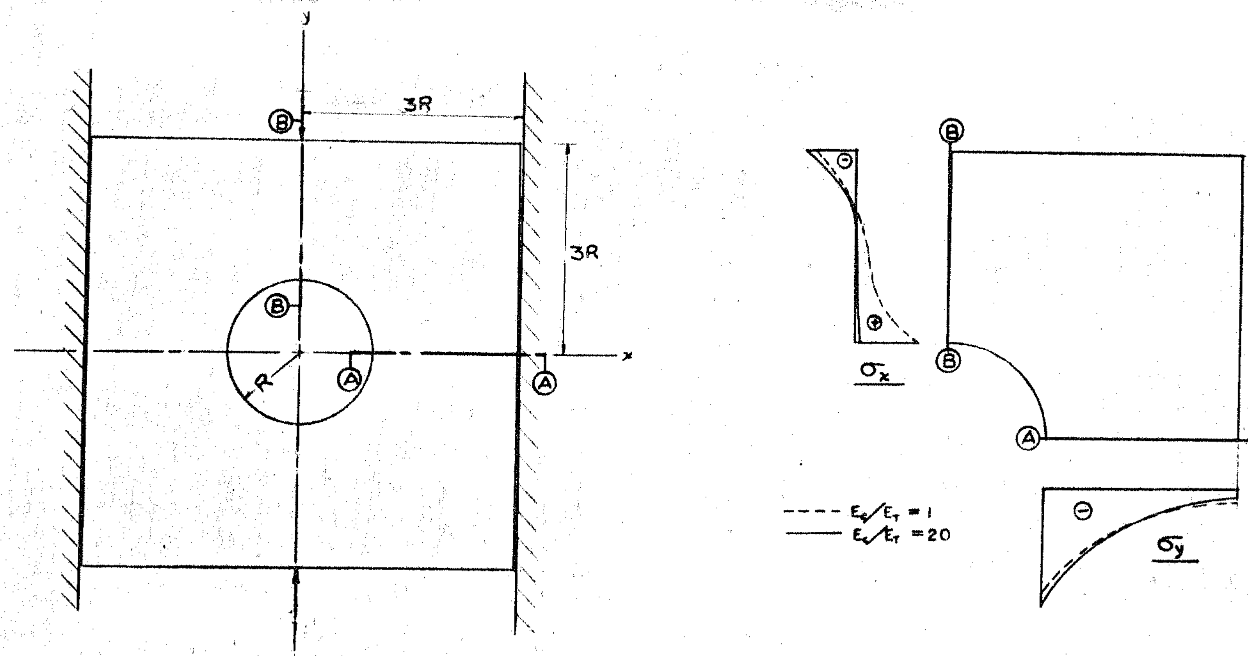
FIG. 14 STEP-BY-STEP ANALYSIS OF PLATE SUBJECTED TO UNIFORM TENSION

to plane stress problems with nonlinear material properties do not exist, it was impossible to verify the method by comparing results with previous solutions. However, for this example the results do seem reasonable. The minor irregularity in yield stress across section A-A is due to the discrete nature of the loading. These inconsistencies can be reduced by decreasing the magnitudes of the load increments.

It should be pointed out that the finite element method does not lose any of its flexibility when extended to structures with nonlinear material properties. It is still possible to treat structures of arbitrary geometry and loading. In addition, each element in the structure may have different nonlinear characteristics.

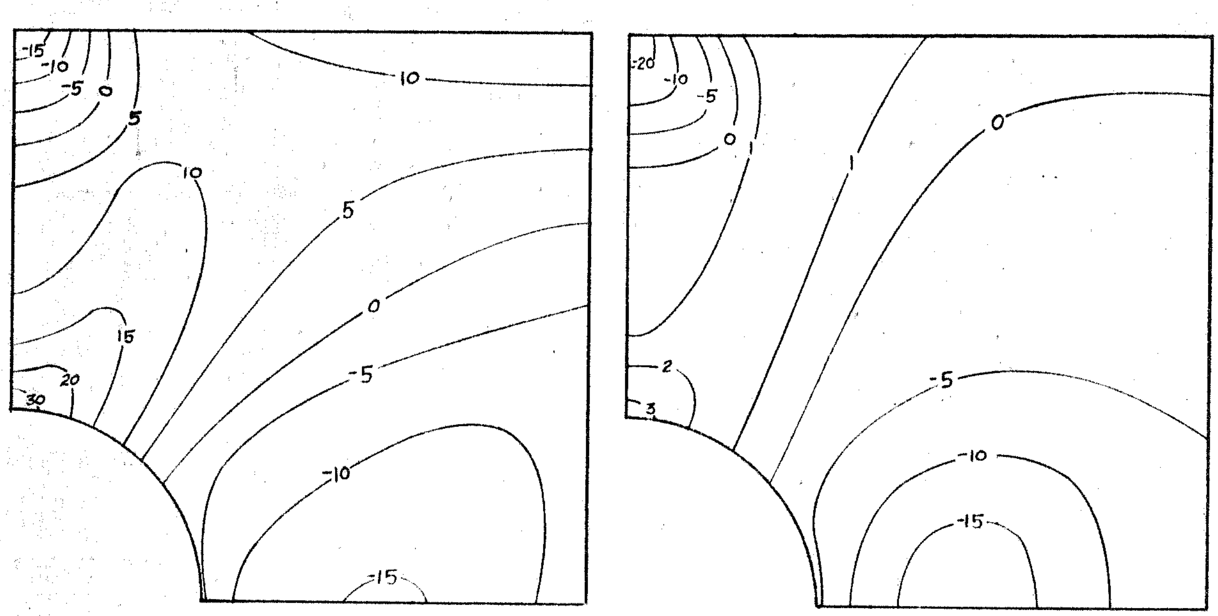
Analysis of Plate With Bilinear Properties - A square plate, with a central circular hole, which is made of a material with different properties in tension and compression, was selected to illustrate the method of successive approximations. This plate, as shown in Fig. 15a, is subjected to equal and opposite compressive forces and its edges are confined laterally (at $x = \pm 3R$, $u = 0$). It was assumed that the ratio of elastic modulus in compression to elastic modulus in tension was 20.

The finite element layout for one-fourth the plate is shown in Fig. 14b. For the first approximation the plate was assumed to be completely in compression; this solution required 90 cycles of iteration. After 20 solutions and a total of 450 cycles of iteration, the difference in stresses and displacements obtained in successive approximations was negligible.



a. CONFINED PLATE

b. NORMAL STRESSES — SECTIONS AA & BB



c. MAJOR PRINCIPAL STRESSES — $E_c/E_T = 1$

d MAJOR PRINCIPAL STRESSES — $E_c/E_T = 20$

FIG. 15 ANALYSIS OF PLATE WITH BILINEAR MATERIAL PROPERTIES

For purposes of comparison, a solution of the isotropic problem, $E_C/E_T = 1$, was also obtained. Figure 15b illustrates the comparison of normal stresses on sections AA and BB. The effect of the low tensile modulus only slightly increases the compressive stresses on section AA but alters radically the tension stresses on section BB. The overall effect of the bilinear analysis on the major principal stresses (maximum tensile stresses) is found by comparing the stress contours which are given in Figs. 15c and 15d; in general, the tensile stresses are reduced by approximately a factor of 10. Again, because of the lack of previous work, no comparison with other solutions is possible.

Since concrete, rocks and many other materials can be approximated by a bilinear model, this analysis represents an important initial step in predicting the behavior of a large class of practical structures. In addition, it may be possible to utilize the bilinear model in the analysis of tapes and belts--due to buckling, these structures have a low effective modulus in compression compared to their modulus in tension.

FINAL REMARKS

This dissertation has demonstrated the general applicability and validity of the finite element analysis of two-dimensional structures. Since the method encompasses several classes of practical engineering structures its importance as a tool in analysis is evident. Due to the general availability of large computers the method may be immediately used by members of the engineering profession. This approach reduces the analysis of a two-dimensional structure to a simple procedure that may be carried out without a detailed knowledge of the method or of the computer program. In order to use the program it is only necessary for the engineer to select an element idealization of the structure and to supply the computer program with data that numerically defines the system of elements. This information, which is in the form of three arrays of numbers, may be assembled by an engineer for large practical structures in only a few hours. An example of such an application is a recent investigation of the stress distribution of underground structures (15).

In addition, this investigation has stimulated supplementary development of the finite element approach.

Presently, it is necessary for the individual to select the finite element idealization of a structure. It is apparent that this involves a certain amount of routine work and a large amount of input data. It may be possible to eliminate some of this work by building into the computer program this "mesh generation" from more fundamental geometric data. However, it should be remembered that the finite element method derives much of its flexibility from the ability of the user arbitrarily to select the element layout. Similarly, it is now necessary for the user to evaluate and plot the computer output, which is always a time consuming task. This may eventually be eliminated if automatic plotting equipment becomes more widely available.

The analysis of certain problems in plate bending is another possible application of the present computer program. The differential equations for both plate bending and plane stress (in terms of the stress function) problems are of the same form. Therefore, it may be possible to determine the solution of the homogeneous part of the differential equation for the plate bending problem by solving a plane stress problem with equivalent boundary conditions. The final solution to the plate bending problem would then be the sum of this homogeneous solution and the particular solution which is readily obtainable for most problems.

Perhaps the next logical step in the extension of the finite element method to two-dimensional problems is its application to time dependent effects (viscous or inertia forces). In this case a step-by-step approach, with respect to time, may be a feasible solution technique (16). Since in this approach a complete linear analysis is required for each increment of time, the necessity of utilizing group and over-relaxation factors is apparent.

Another extension of finite element procedures and the numerical techniques present here is the axial-symmetric problems in three-dimensional elasticity. Only a segment of the symmetric problem needs to be considered. The thickness of each element depends on its distance from the axial of symmetry. As in the plane problem only two degrees of freedom are possible at each nodal point. The only modification in the procedure is in the determination of the 6x6 element stiffness matrix, Eq. 11. This involves including the tangential component of stress in the stress-strain relationship, which expands $[C]$ in Eq. 5 to a 4x4 matrix. Of course, the tangential strain, which varies uniformly over the area of the element, must be included in the displacement transformation matrix; this expands $[A]$ to a 4x6 matrix. Each term in the element stiffness matrix is then obtained by integrating over the volume of the element as suggested by Eq. 11b. To include the effects of nonlinear material properties for axial symmetric problems, it is only necessary to know the nonlinear triaxial stress-strain behavior of the material; then it is possible to apply the same solution techniques which have been used in two-dimensional problems.

ACKNOWLEDGMENTS

The work described in this report was conducted during the course of study for the degree of Doctor of Engineering in Civil Engineering at the University of California.

All work was carried out under the direction of Professor R. W. Clough and the author wishes to express his deepest gratitude for this guidance.

In addition, grateful appreciation is due to Professors F. Baron and D. H. Lehmer for their critical review of this manuscript.

Finally, the National Science Foundation, whose grant made this work possible, and the University of California Computer Center, which provided the computer facilities, are to be thanked for their generous support.

REFERENCES

1. Crandall, Stephen H., "Engineering Analysis," McGraw-Hill Book Company, Inc., New York, 1956.

2. McHenry, D., "A Lattice Analogy for the Solution of Stress Problems," Journal, Institution of Civil Engineers, December 1943, pp. 59-82.

3. Grinter, L. E., "Statistical State of Stress Studied by Grid Analysis," Numerical Methods of Analysis in Engineering, The Macmillan Company, New York, 1949.

4. Hrennikoff, A., "Solution of Problems of Elasticity by the Framework Method," Journal of Applied Mechanics, December 1941.

5. Turner, M. J., Clough, R. W., Martin, H. C., and Topp, L. J., "Stiffness and Deflection Analyses of Complex Structures," Journal of Aeronautical Sciences, Vol. 23, No. 9, September 1956, p. 805.

6. Clough, R. W., "The Finite Element Method in Plane Stress Analysis," Proceedings, ASCE, 2nd Conference on Electronic Computation, Pittsburgh, Pa., September 1960.

7. Argyris, J. H., "Energy Theorems and Structural Analysis," Aircraft Engineering, Vols. 26-27, October 1954-May 1955.

8. Lehman, F. G., "Simultaneous Equations Solved by Over-Relaxation," Proceedings, ASCE, 2nd Conference on Electronic Computation, Pittsburgh, Pa., September 1960.

9. Southwell, R. V., "Relaxation Methods in Engineering Science," Oxford University Press, 1940.

10. Temple, G., and Beckley, W. G., "Rayleigh's Principle and Its Application to Engineering," Oxford University Press, 1933.

11. Turner, M. J., Martin, H. C., and Weikel, R. C., "Further Development and Applications of the Stiffness Method," AGARD, Structures and Materials Panel, Paris, France, July, 1962.

12. Clough, R. W., "The Stress Distribution of Norfork Dam," Series 100, Issue 19, Institute of Engineering Research, University of California, Berkeley, California, August 1962.

13. Wilson, E. L., "Matrix Analysis of Nonlinear Structures," Proceedings, ASCE, 2nd Conference on Electronic Computation, Pittsburgh, Pa., September 1960.

14. Love, A. E. H., "A Treatise on the Mathematical Theory of Elasticity," Dover Publications, New York.

15. Clough, R. W., and Raphael, J. M., "Stress Analysis in Non-Uniform Media by the Finite Element Method," Final Report to Department of Water Resources, State of California, Report No. SESM-63-2, Institute of Engineering Research, University of California, Berkeley, California, March 1963.

16. Wilson, E. L., and Clough, R. W., "Dynamic Response by Step-by-Step Matrix Analysis," Symposium on Use of Computers in Civil Engineering, Laboratorio Nacional de Engenharia Civil, Lisbon, Portugal, October 1962.

APPENDIX

COMPUTER PROGRAM

COMPUTER PROGRAM

The digital computer program performs three major tasks in the complete analysis of a system of two-dimensional triangular plate elements. First, the equilibrium equations for the system are formed from a basic numerical description of the system. Second, this set of equations is solved for the displacements of the nodal points by an iteration procedure. Third, the internal stresses are determined from these displacements.

The program which is given here is coded in the FORTRAN language and has been used with both the IBM 704 and IBM 7090 computers. This program is restricted to plane stress structures.* Gravity loads, temperature loads, non-homogeneous material properties and arbitrary boundary conditions are possible. The maximum size system that can be analyzed is governed by the availability of computer storage. For computers with 32K (32768) storage the maximum number of elements is 550 and the maximum number of nodal points is 340. Only the main sequence of operations of the computer program will be described; the details of coding will be omitted.

---

*The program may also be used for plane strain analysis if the elastic constants, which are used by the program, are modified as follows:

$$E^* = \frac{E}{1-\nu^2}$$

$$\nu^* = \frac{\nu}{1-\nu}$$

Sequence of Operations

Input Data - For the purpose of numerically defining a structure, all nodal points and elements are numbered as illustrated in Fig. 16. The following sequence of punched cards numerically defines the structure.

A. Title Card (72H)
Columns 2 to 72 of this card contain information to be printed with result.

B. Control Card (6I4, 2E12.5, 1I1)

Cols.	1-4	Number of elements
	5-8	Number of nodal points
	9-12	Number of restrained boundary points
	13-16	Cycle interval for the print of the force unbalance
	17-20	Cycle interval for the print of displacements and stresses
	21-24	Maximum number of cycles problem may run
	25-36	Convergence limit for unbalanced forces
	37-48	Over-relaxation factor
	49	Non-zero punch to suppress printing of input data

C. Element array - 1 card per element (4I4, 4E12.4, F8.4)

Cols.	1-4	Element number
	5-8	Nodal point number i
	9-12	Nodal point number j
	13-16	Nodal point number k
	17-28	Modulus of elasticity
	29-40	Density of element
	41-52	Poisson's Ratio
	53-64	Coefficient of thermal expansion
	65-72	Temperature change within element

D. Nodal point array - 1 card per nodal point (1I4, 4F8.1, 2F12.8)

Cols.	1-4	Nodal point number	
	5-12	X-ordinate	
	13-20	Y-ordinate	
	21-28	X-load	
	29-36	Y-load	
	37-48	X-displacement	on free nodal points, these are initial guesses, on restrained nodal points, these are specified displacements.
	49-60	Y-displacement	

E. Boundary array - 1 card per point (2I4,IF6)

Cols.	1-4	Nodal point number
	5-8	0 if Nodal point is fixed in both directions 1 if Nodal point is fixed in X-direction. 2 if Nodal point is free to move along a line of slope S.
	9-16	Slope S (type 2 boundary point only)

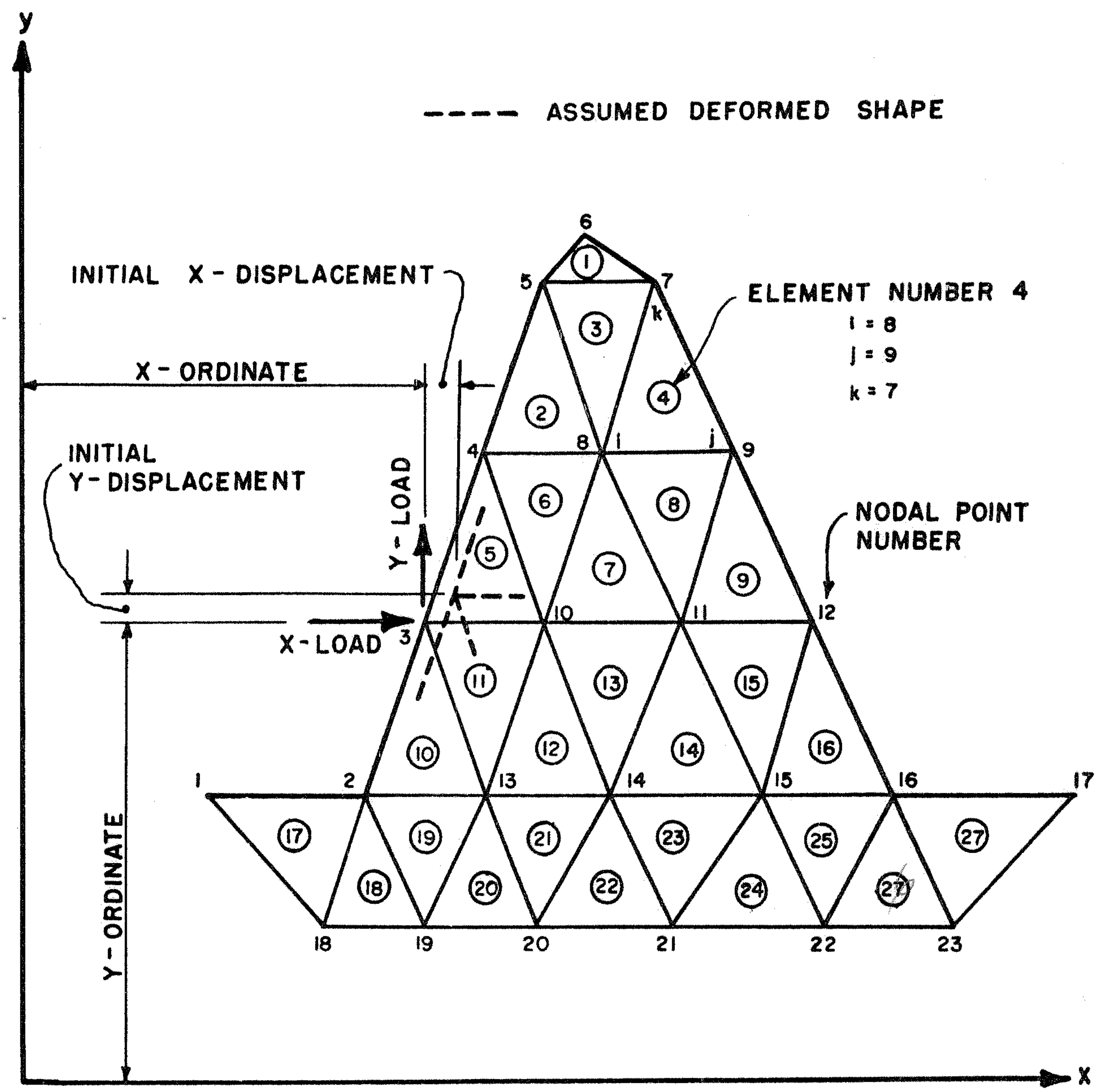


FIG. 16 NUMBERING SYSTEM FOR ELEMENTS AND NODAL POINTS

It should be noted that for a fixed boundary point, the initial displacement is the final displacement of the point, since it is not altered by the iteration procedure.

Formation of Element Stiffness Matrices - The stiffness matrix for each element is determined from Eq. 11. The basic element dimensions are calculated from the coordinates of the connecting nodal points:

$$a_j = X_j - X_i$$
$$b_j = Y_j - Y_i$$
$$a_k = X_k - X_i$$
$$b_k = Y_k - Y_i$$

where i, j and k are the nodal point numbers of the three connecting points and are given in the element array.

Formation of Complete Stiffness of System - Because of the large matrices that are developed in the solution of practical problems, the stiffness matrix used in Eq. 12 is not formed. Since the complete stiffness matrix contains many zero elements, only the non-zero elements are developed and retained by the program; thus, it is possible to treat large plane stress systems without exceeding the storage capacity of the computer.

Formation of Nodal Point Loads - The loads acting on the nodal points are composed of live loads, dead loads, and temperature loads. The equations which are used to determine these loads have been presented in the preceding sections of this report.

Formation of Nodal Flexibilities - The nodal point flexibilities are determined from the previously developed stiffness coefficients. The flexibilities associated with the boundary nodal points are modified by the application of Eqs. 30, as required.

Iterative Solution - The repeated application of Eq. 27 at all nodal points constitutes the iterative procedure. The sum of the absolute magnitude of the unbalanced forces at all nodal points (given by Eq. 26) is also computed for each cycle; this sum, when compared to the covergence limit, serves as a check on the convergence of the procedure. In all analyses presented in this report, this sum was reduced to less than 1/10000 of the value obtained in the first cycle of iteration.

Calculation of Stresses - From the nodal point displacements, with the aid of Eq. 8 and Eq. 33, the stresses σ_x, σ_y, and τ_{xy} are calculated for all element and nodal points. As added information, the principal stresses σ_1 and σ_2 and directions θ are also calculated.

Output Information - At desired points in the iteration procedure, nodal displacements, element stresses and nodal point stresses are printed. Fig. 17 illustrates the form of the computer output, in a typical case.

Timing

For the IBM 7090 the computational time required by the program is approximately equal to 0.006 n.m seconds, where n equals the number of nodal points and m equals the number of cycles of iteration. The number of cycles required depends on the accuracy of the initially assumed displacements and on the desired degree of convergence.

NODAL POINT	X-DISPLACEMENT	Y-DISPLACEMENT
1	0.889282E-02	C.8892E2E-03
2	0.9C1966E-02	-C.901966E-03
3	0.780701E-02	-C.78C7C1E-03
4	0.710852E-02	C.710852E-03
5	0.653899E-02	-C.653899E-03
6	0.590251E-02	C.59C251E-03
7	0.557266E-02	-C.557266E-03
8	0.516712E-02	C.516712E-03
9	0.486274E-02	-C.486274E-03
10	0.455815E-02	C.455815E-03
11	0.439421E-02	-C.439421E-03
12	0.421725E-02	C.421725E-03
13	0.4C7061E-02	0.407061E-03
14	0.4C6246E-02	C.262212E-04
15	0.4C7286E-02	-C.407286E-03

N-POINT	X-STRESS	Y-STRESS	XY-STRESS	MAX-STRESS	MIN-STRESS	DIRECTION
1	-1.9910	2.8814	0.C177	2.88	-1.99	-89.79
2	-2.2800	3.1106	0.1642	3.12	-2.29	-88.26
3	-1.8382	2.4179	0.C536	2.42	-1.84	-89.28
4	-1.4487	1.9338	-0.C289	1.93	-1.45	89.51
5	-1.1172	1.6183	0.C399	1.62	-1.12	-89.17
6	-0.8312	1.3193	-0.C255	1.32	-0.83	89.32
7	-0.6460	1.1353	0.C191	1.14	-0.65	-89.39
8	-0.45C2	0.9763	-0.C204	0.98	-0.45	89.18
9	-0.3436	0.8389	0.C145	0.84	-0.34	-89.30
10	-0.2248	0.7219	-0.C134	0.72	-0.23	89.19
11	-0.1031	0.6400	0.C064	0.64	-0.10	-89.51
12	-0.C709	0.5920	-0.C177	0.59	-0.07	88.47
13	-0.C331	0.4581	-0.C205	0.46	-0.03	87.61
14	-0.C421	0.4853	0.C013	0.49	-0.04	-89.86
15	-0.C655	0.5145	0.C239	0.52	-0.07	-87.65

ELEMENT	X-STRESS	Y-STRESS	XY-STRESS	MAX-STRESS	MIN-STRESS	DIRECTION
1	-2.28C0	3.11C6	0.1642	3.12	-2.29	-88.26
2	-1.8765	2.3062	-0.1289	2.31	-1.88	88.24
3	-1.4384	1.8954	0.1256	1.90	-1.44	-87.85
4	-1.0554	1.6169	-0.C833	1.62	-1.06	88.22
5	-0.8491	1.2704	0.C773	1.27	-0.85	-87.91
6	-0.6017	1.1397	-0.C704	1.14	-0.60	87.69
7	-0.4764	0.9552	0.C504	0.96	-0.48	-87.98
8	-0.3169	0.8366	-0.C412	0.84	-0.32	87.95
9	-0.2282	0.7057	0.C343	0.71	-0.23	-87.90
10	-0.1325	0.6443	-0.C333	0.65	-0.13	87.55
11	-0.0331	0.6016	0.C006	0.60	-0.03	-89.94
12	-0.C655	0.5145	0.C239	0.52	-0.07	-87.65
13	-0.0331	0.4581	-0.C205	0.46	-0.03	87.61

FIG 17 COMPUTER OUTPUT

__Program Listing__ - For the sake of completeness, a Fortran listing of the basic computer program for linear analysis is included. This will enable others who may wish to utilize this approach to avoid some of the tedious details of programming. It also should be pointed out that the portions of the program which are associated with the formation and solution of the stiffness matrix are independent of the type of structure and therefore may be used for other problems in structural analysis.

PROGRAM LISTING

```
C      PLANE STRESS ITERATION--JUNE 1962
C
C      DIMENSION AND COMMON STATEMENTS
C
       DIMENSION NPNUM(340),XORD(340),YORD(340),
      1DSX(340),DSY(340),XLOAD(340),YLOAD(340),NP(340,10),SXX(340,9),
      2SXY(340,9),SYX(340,9),SYY(340,9),NAP(340)
       DIMENSION NUME(550),NPI(550),NPJ(550),NPK(550),ET(550),XU(550),
      1RO(550),COED(550),DT(550),THERM(550),AJ(550),BJ(550),AK(550),
      2BK(550),SIGXX(550),SIGYY(550),SIGXY(550),SLOPE(340)
       DIMENSION NPB(340),NFIX(340),LM(3),A(6,6),B(6,6),S(6,6)
       COMMON SXX,SXY,SYX,SYY
       EQUIVALENCE (SIGXX,RO,NPB), (SIGYY,COED,NFIX), (SIGXY,DT,SLOPE)
C
C      READ AND PRINT OF DATA
C
   150 READ 100
       PRINT 99
       PRINT 100
       READ 1, NUMEL,NUMNP,NUMBC,NCPIN,NOPIN,NCYCM,TOLER,XFAC,T1
       PRINT 101,NUMEL
       PRINT 102,NUMNP
       PRINT 103,NUMBC
       PRINT 104,NCPIN
       PRINT 105,NOPIN
       PRINT 106,NCYCM
       PRINT 107,TOLER
       PRINT 108,XFAC
       READ 2,(NUME(N),NPI(N),NPJ(N),NPK(N),ET(N),RO(N), XU(N),COED(N),
      1DT(N),N=1,NUMEL)
       READ 3,(NPNUM(M),XORD(M),YORD(M),XLOAD(M),YLOAD(M),
      1DSX(M),DSY(M),M=1,NUMNP)
       IF (T1) 160,155,160
   155 PRINT 110
       PRINT 2,(NUME(N),NPI(N),NPJ(N),NPK(N),ET(N),RO(N),XU(N),COED(N),
      1DT(N),N=1,NUMEL)
       PRINT 111
       PRINT 109,(NPNUM(M),XORD(M),YORD(M),XLOAD(M),YLOAD(M),
      1DSX(M),DSY(M),M=1,NUMNP)
C
C      INITIALIZATION
C
   160 NCYCLE=0
       NUMPT=NCPIN
       NUMOPT=NOPIN
       DO 175 L=1,NUMNP
       DO 170 M=1,9
       SXX(L,M)=0.0
       SXY(L,M)=0.0
       SYX(L,M)=0.0
       SYY(L,M)=0.0
   170 NP(L,M)=0
       NP(L,10)=0
   175 NP(L,1)=L
```

```
C
C     MODIFICATION OF LOADS AND ELEMENT DIMENSIONS
C
      DO 180 N=1,NUMEL
      ET(N)=ABSF(ET(N))
      I=NPI(N)
      J=NPJ(N)
      K=NPK(N)
      AJ(N)=XORD(J)-XORD(I)
      AK(N)=XORD(K)-XORD(I)
      BJ(N)=YORD(J)-YORD(I)
      BK(N)=YORD(K)-YORD(I)
  176 AREA=(AJ(N)*BK(N)-BJ(N)*AK(N))/2.
      IF (AREA) 701,701,177
  177 THERM(N)=ET(N)*COED(N)*DT(N)/(XU(N)-1.)
      DL=AREA*RO(N)/3.
      XLOAD(I)=THERM(N)*(BK(N)-BJ(N))/2.+XLOAD(I)
      XLOAD(J)=-THERM(N)*BK(N)/2.+XLOAD(J)
      XLOAD(K)=THERM(N)*BJ(N)/2.+XLOAD(K)
      YLOAD(I)=THERM(N)*(AJ(N)-AK(N))/2.+YLOAD(I)-DL
      YLOAD(J)=THERM(N)*AK(N)/2.+YLOAD(J)-DL
  180 YLOAD(K)=-THERM(N)*AJ(N)/2.+YLOAD(K)-DL
C
C     FORMATION OF STIFFNESS ARRAY
C
      DO 200 N=1,NUMEL
      AREA=(AJ(N)*BK(N)-AK(N)*BJ(N))*.5
      COMM=.25*ET(N)/((1.-XU(N)**2)*AREA)
      A(1,1)=BJ(N)-BK(N)
      A(1,2)=0.0
      A(1,3)=BK(N)
      A(1,4)=0.0
      A(1,5)=-BJ(N)
      A(1,6)=0.0
      A(2,1)=0.0
      A(2,2)=AK(N)-AJ(N)
      A(2,3)=0.0
      A(2,4)=-AK(N)
      A(2,5)=0.0
      A(2,6)=AJ(N)
      A(3,1)=AK(N)-AJ(N)
      A(3,2)=BJ(N)-BK(N)
      A(3,3)=-AK(N)
      A(3,4)=BK(N)
      A(3,5)=AJ(N)
      A(3,6)=-BJ(N)
      B(1,1)=COMM
      B(1,2)=COMM*XU(N)
      B(1,3)=0.0
      B(2,1)=COMM*XU(N)
      B(2,2)=COMM
      B(2,3)=0.0
      B(3,1)=0.0
      B(3,2)=0.0
      B(3,3)=COMM*(1.-XU(N))*.5
C
      DO 182 J=1,6
      DO 182 I=1,3
      S(I,J)=0.0
```

```
      DO 182 K=1,3
  182 S(I,J)=S(I,J)+B(I,K)*A(K,J)
      DO 183 J=1,6
      DO 183 I=1,3
  183 B(J,I)=S(I,J)
      DO 184 J=1,6
      DO 184 I=1,6
      S(I,J)=0.0
      DO 184 K=1,3
  184 S(I,J)=S(I,J)+B(I,K)*A(K,J)
C
      LM(1)=NPI(N)
      LM(2)=NPJ(N)
      LM(3)=NPK(N)
      DO 200 L=1,3
      DO 200 M=1,3
      LX=LM(L)
      MX=0
  185 MX=MX+1
      IF(NP(LX,MX)-LM(M)) 190,195,190
  190 IF(NP(LX,MX)) 185,195,185
  195 NP(LX,MX)=LM(M)
      IF (MX-10) 196,702,702
  196 SXX(LX,MX)=SXX(LX,MX)+S(2*L-1,2*M-1)
      SXY(LX,MX)=SXY(LX,MX)+S(2*L-1,2*M)
      SYX(LX,MX)=SYX(LX,MX)+S(2*L,2*M-1)
  200 SYY(LX,MX)=SYY(LX,MX)+S(2*L,2*M)
C
C     COUNT OF ADJACENT NODAL POINTS
C
      DO 206 M=1,NUMNP
      MX =1
  205 MX=MX+1
      IF (NP(M,MX)) 206,206,205
  206 NAP(M)=MX-1
C
C     INVERSION OF NODAL POINT STIFFNESS
C
      DO 210 M=1,NUMNP
      COMM=SXX(M,1)*SYY(M,1)-SXY(M,1)*SYX(M,1)
      TEMP=SYY(M,1)/COMM
      SYY(M,1)=SXX(M,1)/COMM
      SXX(M,1)=TEMP
      SXY(M,1)=-SXY(M,1)/COMM
  210 SYX(M,1)=-SYX(M,1)/COMM
C
C     MODIFICATION OF BOUNDARY FLEXIBILITIES
C
      PRINT 112
      READ  4, (NPB(L),NFIX(L),SLOPE(L),L=1,NUMBC)
      PRINT 4, (NPB(L),NFIX(L),SLOPE(L),L=1,NUMBC)
      DO 240 L=1,NUMBC
      M=NPB(L)
      NP(M,1)=0
      IF(NFIX(L)-1) 225,220,215
  215 C=(SXX(M,1)*SLOPE(L)-SXY(M,1))/(SYX(M,1)*SLOPE(L)-SYY(M,1))
      R=1.-C*SLOPE(L)
      SXX(M,1)=(SXX(M,1)-C*SYX(M,1))/R
      SXY(M,1)=(SXY(M,1)-C*SYY(M,1))/R
```

```
      SYX(M,1)=SXX(M,1)*SLOPE(L)
       SYY(M,1)=SXY(M,1)*SLOPE(L)
      GO TO 240
  220 SYY(M,1)=SYY(M,1)-SYX(M,1)*SXY(M,1)/SXX(M,1)
      GO TO 230
  225 SYY(M,1)=0.0
  230 SXX(M,1)=0.0
  235 SXY(M,1)=0.0
      SYX(M,1)=0.0
  240 CONTINUE
C
C      ITERATION ON NODAL POINT DISPLACEMENTS
C
  243 PRINT 119
  244 SUM=0.0
      DO 290 M=1,NUMNP
      NUM=NAP(M)
      IF (SXX(M,1)+SYY(M,1)) 275,290,275
  275 FRX=XLOAD(M)
      FRY=YLOAD(M)
      DO 280 L=2,NUM
      N=NP(M,L)
      FRX=FRX-SXX(M,L)*DSX(N)-SXY(M,L)*DSY(N)
  280 FRY=FRY-SYX(M,L)*DSX(N)-SYY(M,L)*DSY(N)
      DX=SXX(M,1)*FRX+SXY(M,1)*FRY-DSX(M)
      DY=SYX(M,1)*FRX+SYY(M,1)*FRY-DSY(M)
      DSX(M)=DSX(M)+XFAC*DX
      DSY(M)=DSY(M)+XFAC*DY
      IF (NP(M,1)) 285,290,285
  285 SUM=SUM+ABSF(DX/SXX(M,1))+ABSF(DY/SYY(M,1))
  290 CONTINUE
C
C     CYCLE COUNT AND PRINT CHECK
C
      NCYCLE=NCYCLE +1
      IF (NCYCLE-NUMPT) 305,300,300
  300 NUMPT=NUMPT+NCPIN
      PRINT 120,NCYCLE,SUM
  305 IF (SUM-TOLER) 400,400,310
  310 IF(NCYCM-NCYCLE) 400,400,315
  315 IF (NCYCLE-NUMOPT) 244,320,320
  320 NUMOPT=NUMOPT+NOPIN
C
C     PRINT OF DISPLACEMENTS AND STRESSES
C
  400 PRINT 99
      PRINT 100
      PRINT 121
      PRINT 122,(NPNUM(M),DSX(M),DSY(M),M=1,NUMNP)
      PRINT 123
      DO 420 N=1,NUMEL
      I=NPI(N)
      J=NPJ(N)
      K=NPK(N)
      EPX=(BJ(N)-BK(N))*DSX(I)+BK(N)*DSX(J)-BJ(N)*DSX(K)
      EPY=(AK(N)-AJ(N))*DSY(I)-AK(N)*DSY(J)+AJ(N)*DSY(K)
      GAM= (AK(N)-AJ(N))*DSX(I)-AK(N)*DSX(J)+AJ(N)*DSX(K)
     1   +(BJ(N)-BK(N))*DSY(I)+BK(N)*DSY(J)-BJ(N)*DSY(K)
      COMM=ET(N)/((1.-XU(N)**2)*(AJ(N)*BK(N)-AK(N)*BJ(N)))
```

```
       X=COMM*(EPX+XU(N)*EPY)+THERM(N)
       Y=COMM*(EPY+XU(N)*EPX)+THERM(N)
      XY=COMM*GAM*(1.-XU(N))*.5
      SIGXX(N)=X
      SIGYY(N)=Y
      SIGXY(N)=XY
      C=(X+Y)/2.0
      R=SQRTF(((Y-X)/2.0)**2+XY**2)
      XMAX=C+R
      XMIN=C-R
      PA=0.5*57.29578*ATANF ( 2.* XY/(Y-X))
      IF (2.*X-XMAX-XMIN) 405,420,420
  405 IF (PA) 410,420,415
  410 PA=PA+90.0
      GO TO 420
  415 PA=PA-90.0
  420 PRINT 124, (NUME(N),X,Y,XY,XMAX,XMIN,PA)
C
      PRINT 823
      DO 900  M=1,NUMNP
       X=0.0
       Y=0.0
      XY=0.0
      SRX=0.0
      SRY=0.0
      R=0.0
      DO 860 N=1,NUMEL
      I=NPI(N)
      J=NPJ(N)
      K=NPK(N)
      IF (M-I) 830,850,830
  830 IF (M-J) 835,845,835
  835 IF (M-K) 860,840,860
  840 I=NPK(N)
      K=NPI(N)
      GO TO 850
  845 I=NPJ(N)
      J=NPI(N)
  850 A=ABSF(XORD(J)+XORD(K)-2.*XORD(I))
      B=ABSF(YORD(J)+YORD(K)-2.*YORD(I))
      RY=B/(A+B)
      SRY=SRY+RY
      Y=Y+SIGYY(N)*RY
      RX=A/(A+B)
      SRX=SRX+RX
      X=X+SIGXX(N)*RX
      R=R+1.0
      XY=XY+SIGXY(N)
  860 CONTINUE
      X=X/SRX
      Y=Y/SRY
      XY=XY/R
      C=(X+Y)/2.0
      R=SQRTF(((Y-X)/2.0)**2+XY**2)
      XMAX=C+R
      XMIN=C-R
      PA=0.5*57.29578*ATANF ( 2.* XY/(Y-X))
      IF (2.*X-XMAX-XMIN) 805,820,820
  805 IF (PA) 810,820,815
```

```
  810 PA=PA+90.0
      GO TO 820
  815 PA=PA-90.0
  820 PRINT 124,(M,X,Y,XY,XMAX,XMIN,PA)
  900 CONTINUE
C
      IF (SUM-TOLER) 440,440,430
  430 IF (NCYCM-NCYCLE) 440,440,243
C
  440 GO TO 150
C
C     PRINT OF ERRORS IN INPUT DATA
C
  701 PRINT 711,(N)
      GO TO 440
  702 PRINT 712,(LX)
      GO TO 440
C
C        FORMAT STATEMENTS
C
    1 FORMAT (6I4,2E12.5,4I1)
    2 FORMAT (4I4,4E12.4,F8.4)
    3 FORMAT (1I4,4F8.1,2F12.8)
    4 FORMAT (2I4,1F8.3)
    5 FORMAT (3E15.8)
   99 FORMAT (1H1)
  100 FORMAT (72H  BCD INFORMATION
     1                          )
  101 FORMAT(29H0NUMBER OF ELEMENTS          =1I4/)
  102 FORMAT(29H NUMBER OF NODAL POINTS      =1I4/)
  103 FORMAT(29H NUMBER OF BOUNDARY POINTS   =1I4/)
  104 FORMAT(29H CYCLE PRINT INTERVAL        =1I4/)
  105 FORMAT(29H OUTPUT INTERVAL OF RESULTS  =1I4/)
  106 FORMAT(29H CYCLE LIMIT                 =1I4/)
  107 FORMAT(29H TOLERANCE LIMIT             =1E12.4/)
  108 FORMAT(29H OVER RELAXATION FACTOR      =1F6.3)
  109 FORMAT (1I8,4F12.1,2F12.8)
  110 FORMAT (74H1EL.    I    J    K        E          DENSITY     POISSON
     1  ALPHA      DELTA T)
  111 FORMAT (80H1       NP         X-ORD         Y-ORD         X-LOAD        Y-LOA
     1D       X-DISP        Y-DISP)
  112 FORMAT (20H BOUNDARY CONDITIONS)
  119 FORMAT(34H0          CYCLE        FORCE UNBALANCE)
  120 FORMAT (1I12,1E20.6)
  121 FORMAT (42H0NODAL POINT X-DISPLACEMENT Y-DISPLACEMENT)
  122 FORMAT (1I12,2E15.6)
  123 FORMAT(120H1  ELEMENT                X-STRESS                 Y-STRESS
     1        XY-STRESS            MAX-STRESS     MIN-STRESS        DIRECTION)
  124 FORMAT (1I10,3F20.4,5X,3F15.2)
  711 FORMAT (32H0ZERO OR NEGATIVE AREA, EL. NO.=1I4)
  712 FORMAT (33H0OVER 8 N.P. ADJACENT TO N.P. NO.1I4)
  823 FORMAT(120H1  N-POINT                X-STRESS                 Y-STRESS
     1        XY-STRESS            MAX-STRESS     MIN-STRESS        DIRECTION)
C
      END
```