N SF-RA-E-75-265

PB 282 966

REPORT NO.
EERC 75-37
DECEMBER 1975

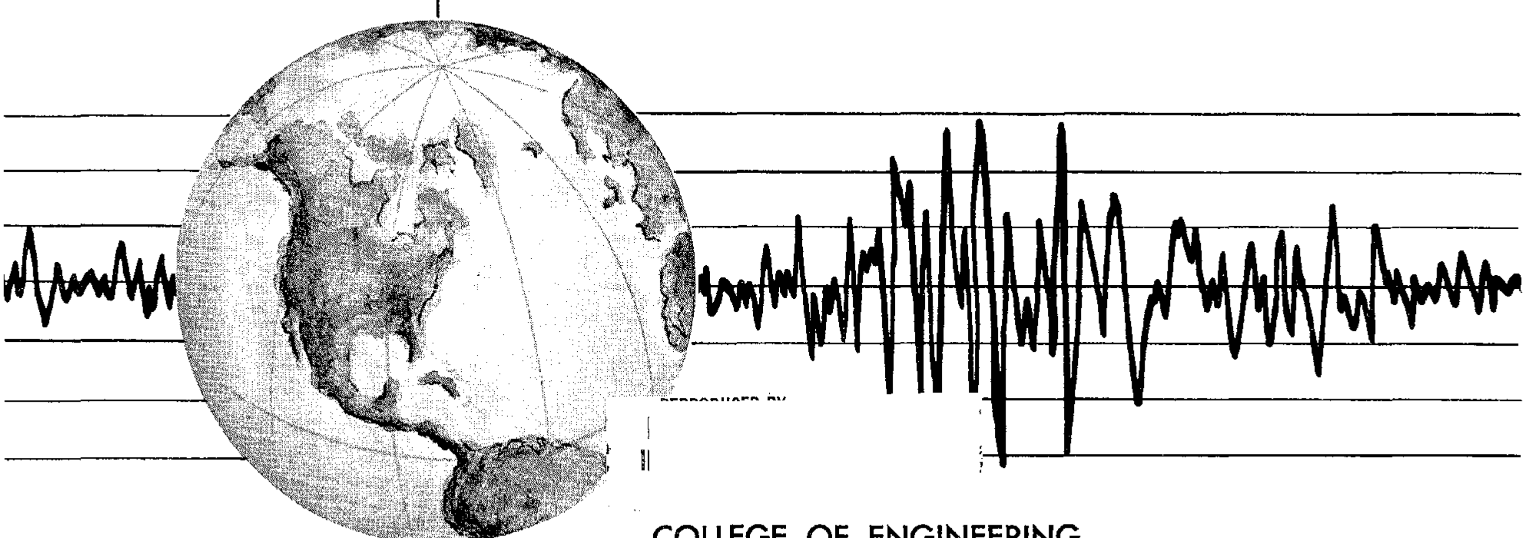
EARTHQUAKE ENGINEERING RESEARCH CENTER

ANSR - I

GENERAL PURPOSE PROGRAM FOR ANALYSIS OF NONLINEAR STRUCTURAL RESPONSE

by
DIGAMBAR P. MONDKAR
and
GRAHAM H. POWELL

Report to Sponsor:
National Science Foundation
Grant NSF-GI-36387



COLLEGE OF ENGINEERING
UNIVERSITY OF CALIFORNIA · Berkeley, California

ANSR-I
GENERAL PURPOSE PROGRAM FOR ANALYSIS OF NONLINEAR STRUCTURAL RESPONSE

by

Digambar P. Mondkar
Assistant Research Engineer

and

Graham H. Powell
Professor of Civil Engineering

Prepared under the sponsorship of
the National Science Foundation

Report No. 75-37
Earthquake Engineering Research Center
College of Engineering
University of California
Berkeley, California

December 1975

ABSTRACT

ANSR is a general purpose computer program for static and dynamic analysis of nonlinear structures. This report documents the features and organization of the current version of the program. The theoretical formulations and solution schemes used in the program are described, and details are given about the structure and organization of the auxiliary program for adding new finite elements to the program. Several examples are presented to illustrate the scope of ANSR. The user's manual for the program is described.

ACKNOWLEDGEMENTS

This work has been performed under the sponsorship of the National Science Foundation, under Grant NSF-GI-36387. Computer facilities were provided by the Computer Centers of the University of California, Berkeley and the Lawrence Berkeley Laboratory.

The assistance of Ms. Ellen Antonucci who typed the manuscript, and Mr. Al Klash, Ms. Lea Rambeau and Ms. Gail Feazell who drafted the drawings, is gratefully acknowledged.

TABLE OF CONTENTS

Page

1. INTRODUCTION

1.1 OBJECTIVE

Because of the need for rational investigations of nonlinear structures subjected to static and dynamic forces, a continuing study has been undertaken with the long term objective of developing a general purpose computer program for nonlinear structural analysis. This study has progressed through two phases, namely (1) review and development of theories, computational techniques and algorithms that can be applied in nonlinear structural analysis, and (2) development of a preliminary version of a general purpose computer code based on the studies in phase (1).

The studies conducted under phase (1) have included the following.

(a) Consistent formulation of the equations of motion, which are applicable to any type and/or degree of nonlinearity.

(b) Finite element formulation of the nonlinear equations of motion.

(c) Techniques for solution of the nonlinear equations of motion.

(d) Constitutive plasticity laws for material behavior, and algorithms for materal stress computations for path dependent plasticity models.

The results and findings of these studies have been reported in detail in [1]. Based on these investigations, a preliminary version of a general purpose computer program has been completed. The objectives of this report are to explain the concepts behind the program, to document its features and organization, and to illustrate its use.

1.2 PROGRAM CONCEPTS

During recent years a considerable amount of effort has been invested in developing computer codes for nonlinear structural analysis (see, for example, the survey paper by Marcal [2]). Most programs have been developed for special purpose applications, and are limited in scope. A few general purpose programs, notably MARC-CDC [3] and ANSYS [4] are commercially available and are being used quite extensively. The program NONSAP [5] is also being applied, but to a more limited extent.

Although such programs exist, they do not satisfy all practical needs, and do not necessarily apply the most effective and efficient techniques for the solution of nonlinear problems. In view of the practical importance of nonlinear analysis, and the large amounts of computer time which are commonly required for such analysis, it is believed that additional program development efforts are warranted. The program described in this report is intended to form the basis of a practical general purpose computer code which combines broad scope and large capacity with computional efficiency. The program structure has been designed, therefore, to satisfy the following requirements.

(a) Modularity: The program should be modular so that new program capabilities, such as new finite elements, new constitutive laws, etc., can be added by developing a few subroutines, without changes to the existing program. This has been achieved by structuring the program as a base program to which a number of auxiliary programs can be added. Storage allocation and computations common to all finite elements are performed within the base program, whereas computations associated with specific elements are carried out within the auxiliary programs.

(b) Computational efficiency: The program should incorporate efficient computational algorithms, including efficient equation solvers, stress computation algorithms, etc.

(c) Effective use of core and disc storage: The program should be organized to make optimum use of the available core (high-speed) storage, and to minimize the input/output cost for data transfer to disc (low-speed) storage. This has been achieved by using dynamic allocation of core storage and buffering of data into large blocks for transfer to disc storage.

(d) Solution Strategy: The program should include a flexible solution strategy so that a wide range of nonlinear structural systems can be analyzed. This is desirable because no single solution scheme can be identified as optimal for all types of nonlinear behavior. Flexibility has been achieved by implementing a strategy defined in terms of a number of solution parameters. By assigning different values to these parameters, a wide variety of solution schemes can be constructed.

1.3 PROGRAM FEATURES AND LIMITATIONS

1.3.1 Structural Idealization

(1) The structure to be analyzed is idealized as an assemblage of discrete finite elements connected at nodes. The theory and solution procedure are based on the finite element formulation of the displacement method, with the nodal displacements as the field variables.

(2) Each node may possess up to six displacement degrees of freedom, as in a typical three dimensional frame analysis.

(3) Provison is made for degrees of freedom to be deleted or combined. This feature provides the program user with ample flexibility in the idealization of the structure, and may permit the size of the problem to be substantially reduced.

(4) The structure mass is assumed to be lumped at the nodes, so that the mass matrix is diagonal. The program could be modified to consider a coupled (consistent) mass matrix.

(5) Viscous damping effects may be included, if desired. Damping effects proportional to mass, initial elastic stiffness and/or tangent stiffness can be specified. These effects may be specified to vary in magnitude from one group of elements to the next.

1.3.2 Static and Dynamic Loadings

(1) Loads are assumed to be applied only at the nodes. Static and/or dynamic loads may be specified; however, static loads, if any, must be applied prior to the dynamic loads.

(2) For static analysis, a number of static force patterns must be specified. Static loads are then applied in a series of load increments, each load increment being specified as a linear combination of the static force patterns. This feature permits nonproportional loads to be applied. Each load increment can be specified to be applied in a number of equal steps.

(3) The dynamic loading may consist of earthquake ground accelerations, time dependent nodal loads, and prescribed initial values of the nodal velocities and accelerations. These dynamic loadings can be specified to act singly or in combination.

(4) Earthquake excitations are defined by time histories of ground acceleration. Three different time histories may be specified,

one for each of the X, Y and Z axes of the structure. For any given axis, all support points are assumed to move identically and in phase. The accelerations for any time history may be specified at equal time intervals (as in an artificially generated earthquake record) or at unequal time intervals (as in a measured earthquake record).

(5) Any number of time histories of dynamic force may be specified. As with the earthquake records, these time histories may be specified to be at equal or unequal time intervals. Any dynamic force record may be prescribed to act at a node or a group of nodes, either as forces in the X, Y or Z directions, or as moments about the X, Y or Z axes.

(6) Values of initial translation and/or rotational velocity and acceleration may be specified at each node. Structures subjected to impulsive loads can be analyzed by prescribing appropriate initial velocities. For the case of static analysis followed by dynamic analysis, the displacements at the start of the dynamic analysis are assumed to be those at the end of the static analysis.

1.3.3 Finite Element Library

(1) The element library is very limited at the time of writing. However, the program is organized to permit the addition of new finite elements to the library with relative ease. The following elements are available.

(a) Three dimensional truss element, which may yield in tension and yield or buckle elastically in compression. Large displacement effects may be included.

(b) Two dimensional 4-to-8 node finite element for plane stress, plane strain and axisymmetric analysis. Large displacement effects may be included. The material may be specified to be isotropic linearly elastic, orthotropic linearly elastic or isotropic elastic-perfectly plastic with the von Mises yield function.

(2) Nonlinearities are introduced at the element level only, and may be due to large displacements, large strains and/or nonlinear materials. The programmer adding a new element may include any type or degree of nonlinearity in the behavior of the element.

1.3.4 Solution Procedure

(1) The program incorporates a solution strategy defined in terms of a number of control parameters. By assigning appropriate values to these parameters, a wide variety of solution schemes, including step-by-step, iterative and mixed schemes, may be constructed. This permits the program user considerable flexibility in selecting optimal schemes for particular types of nonlinear behavior.

(2) For static analysis, a different solution scheme may be employed for each load increment. The use of this feature can reduce the solution time for structures in which the response must be computed more precisely for certain ranges of loading than for others. In such cases, a sophisticated solution scheme with equilibrium iteration might be used for the critical ranges of loading, whereas a simpler step-by-step scheme without iteration might suffice for other loading ranges.

(3) The dynamic response is computed by step-wise time integration of the incremental equations of motion using Newmark's β-γ-δ operator. A variety of integration operators may be obtained by assigning appropriate values to the parameters β and γ. However,

the most commonly used scheme will be the "constant average acceleration" scheme, with $\beta = 1/4$, $\gamma = 1/2$ and $\delta = 0$. Viscous damping effects may be introduced by specifying a positive value to the parameter δ. In most cases, however, damping effects will be introduced more explicitly, in mass dependent or stiffness dependent form.

1.3.5 Other Features

(1) Data checking runs may be made prior to execution runs. During data checking, the program reads and prints all input data, and also prints any generated data, but performs no substantial analysis.

(2) In its current version the program requires that the stiffness matrix be stored in core. This matrix is stored column-wise in a compacted form omitting most zero elements. Because the stiffness matrix is modified, rather than completely reformed, as the tangent stiffness changes, it is also necessary to save a duplicate stiffness matrix. The modification requires least numerical operations if the duplicate stiffness matrix can be held in core. However, this may not always be possible, and hence provision is made for the user to specify whether the duplicate matrix should be stored in core or on disc. Modifications for very large systems, using blocking and out-of-core storage of the stiffness matrix, are planned for a future version of the program.

(3) The element information is stored either in core or on disc. When stored on disc, the information is blocked to minimize input/output cost for data transfer between core and disc storages. Thus, the number of finite elements are not limited by the availability

of the core storage, except that the available core storage must be sufficient to fit information for at least one of the elements requiring the largest number of data locations.

(4) During solution, the decomposition (triangularization) of the structure stiffness matrix is carried out on only that part of the updated stiffness matrix which follows the first modified coefficient. Significant savings in solution time can sometimes be obtained by numbering those nodes connecting nonlinear elements to be last, so that the operations on the structure stiffness matrix are limited to the end of the matrix.

(5) Because the structure stiffness matrix is stored in compacted form, rather than in banded form, there will be relatively small penalties in storage requirements and equation solving time if there are local increases in the matrix bandwidth. Hence, if a few nodes are to be added to a structure for which an input data deck has already been prepared, the additional computational cost incurred by numbering these nodes last may be less than the man-hour cost involved in renumbering all of the nodes and preparing a new input data deck.

(6) At present no restart capability is included in the program. Such a capability will be added in a future version.

1.4 REPORT LAYOUT

Chapter 2 of this report reviews the theory and computational algorithms used in the program. In particular, the nonlinear incremental equations of motion using the Lagrangian description of deformation and their finite element formulation are discussed, and computational techniques for the solution of the nonlinear equations are presented. The solution strategy included in the program is outlined in detail in Chapter 3. The structure and the organization of the auxiliary program for adding new elements to the program are explained in Chapter 4. The results for some example structures which have been used for program verification are discussed in Chapter 5, and concluding remarks are presented in Chapter 6.

Appendix A constitutes a detailed user's guide for the program. Appendices B1 and B2 contain a brief description of the truss element and two dimensional element, respectively.

2. THEORETICAL FORMULATIONS

2.1 INTRODUCTION

In this chapter, the theoretical and computational techniques used in the program are briefly reviewed. Firstly, the variational form of the incremental equations of motion is stated and linearization of these equations for computational purposes is discussed. Using isoparametric finite elements, a finite element formulation of the equations of motions is obtained. Secondly, techniques including the step-by-step and iterative procedures for the solution of the nonlinear equations are discussed, and an acceleration scheme to improve convergence in constant stiffness iteration is reviewed. Finally, an algorithm for the numerical integration of the equations of motion using Newmark's β-γ-δ operator and optional iteration is described.

For a more detailed treatment of the subjects covered in this chapter, reference may be made to [1].

2.2 EQUATIONS OF MOTION

2.2.1 Kinematics

The kinematics of deformation of a body can be described by the three configurations indicated in Fig. 1, namely (a) configuration C_o at time $t = 0$, (b) configuration C_1 at time t, and (c) neighboring configuration C_2 at time $\tau = t + \Delta t$. In the Lagrangian description, the initial undeformed configuration C_o is taken as the reference configuration, and the conjugate pair consisting of the second (symmetric) Piola-Kirchhoff stress and the Lagrangian strain are used to derive all relationships [6].

Systems of orthogonal curvilinear coordinates and corresponding base vectors can be associated with each of the three configurations. However, for the derivations herein, a cartesian system X_I (I = 1, 2, 3) in the reference configuration will be used. If this system is chosen as the global coordinate system to describe the motion, the kinematics of deformation is governed by the following relationships*.

$$^2u_I = {}^1u_I + u_I \tag{1}$$

$$2\,{}^2E_{IJ} = {}^2u_{I|J} + {}^2u_{J|I} + {}^2u_{K|I}\,{}^2u_{K|J} \tag{2}$$

$$2\,{}^1E_{IJ} = {}^1u_{I|J} + {}^1u_{J|I} + {}^1u_{K|I}\,{}^1u_{K|J} \tag{3}$$

Here $^1\underset{\sim}{u}$ and $^2\underset{\sim}{u}$ are the displacement vectors of a generic material point $\underset{\sim}{X}$, in configurations C_1 and C_2 respectively; $\underset{\sim}{u}$ is the displacement vector from C_1 to C_2; $^1\underset{\sim}{E}$ and $^2\underset{\sim}{E}$ are the Lagrangian strain tensors in C_1 and C_2 respectively; and $(\cdot)_{I|J}$ denotes differentiation of the I^{th} component of the undesignated variable with respect to X_J, viz. $u_{I|J} = \partial u_I / \partial X_J$.

The strain increment E_{IJ} between configuration C_1 and C_2 can be obtained as

$$E_{IJ} = {}^2E_{IJ} - {}^1E_{IJ} \tag{4}$$

and can be expressed in terms of the linear component, e_{IJ} and the non-linear component, η_{IJ} as follows.

$$2E_{IJ} = 2\,e_{IJ} + 2\,\eta_{IJ} \tag{5}$$

*In all relationships stated in this chapter, each index has the range 1 to 3 unless stated otherwise. The usual tensor summation convention on a repeated index is implied.

where

$$2\, e_{IJ} = u_{I|J} + u_{J|I} + u_{K|I}\ {}^1u_{K|J} + {}^1u_{K|I}\ u_{K|J} \tag{6}$$

and

$$2\, \eta_{IJ} = u_{K|I}\ u_{K|J} \tag{7}$$

2.2.2 Incremental Equations of Motion

The incremental equations of motion are obtained by applying the Principle of Virtual Displacements to the two neighboring configurations C_1 and C_2. The virtual work relationships in these deformed configurations are expressed with reference to the initial configuration C_o. The variational form of the nonlinear incremental equations is then obtained by taking the difference between the two virtual work relationships. A detailed derivation of the incremental equations may be found in references [1,7,8].

The incremental equations written in cartesian component form are as follows.

$$\int_{V_o} [S_{IJ}(\delta e_{IJ} + \delta\eta_{IJ}) + {}^1S_{IJ}\ \delta\eta_{IJ} + {}^o\rho\ \delta u_K\ \ddot{u}_K] dV = [\int_{A_2} \delta u_I\ {}^2t_I\ d\bar{a} + \int_{V_2} \delta u_I\ {}^2f_I\ d\bar{v}] - [\int_{A_1} \delta u_I\ {}^1t_I\ da + \int_{V_1} \delta u_I\ {}^1f_I\ dv] \tag{8}$$

in which V_o is the volume of the body in C_o; dV is the differential volume in C_o; A_1 and A_2 are the surface areas over which tractions are prescribed, and V_1, V_2 are volumes of the body in C_1 and C_2, respectively; corresponding differential quantities are denoted by da, $d\bar{a}$, dv, $d\bar{v}$; ${}^1\underset{\sim}{t}$, ${}^1\underset{\sim}{f}$, ${}^2\underset{\sim}{t}$, ${}^2\underset{\sim}{f}$ are the prescribed surface traction per unit area and body force per unit volume in C_1 and C_2, respectively; ${}^{o}\rho$ is the mass density in C_o; $\underset{\sim}{u}$ and $\ddot{\underset{\sim}{u}}$ are the increments in displacement and acceleration respectively; ${}^{1}\underset{\sim}{S}$ is the second (symmetric) Piola-Kirchhoff (P-K) stress tensor in C_1; the increment in P-K stress between C_1 and C_2 is defined as $\underset{\sim}{S} = {}^{2}\underset{\sim}{S} - {}^{1}\underset{\sim}{S}$; and $\delta(\cdot)$ denotes variation on the undesignated variable.

Equations (8) are applicable to both large displacement, small strain response and large displacement, large strain response. If changes in the geometry of the body are neglected, the equations reduce to those for small displacement response. Material nonlinearity can be included by specifying an appropriate relationship between stress and strain. Therefore, these equations can be used to study structural systems with only geometric nonlinearity, only material nonlinearity, or combined geometric and material nonlinearities.

2.2.3 Linearization of Equations of Motion

Equations (8) are nonlinear in the displacements, and must be linearized for computational purposes. The linearization process must account for three considerations, as follows.

(a) It must be assumed that the stress increment, S_{IJ}, is linearly related to the strain increment, E_{IJ}. That is,

$$S_{IJ} = C_{IJKL} E_{KL} \tag{9}$$

in which C_{IJKL} is the constitutive tensor. For finite strain increments this will be true only for linearly elastic materials. For other types of material, a linear relationship can be established only in terms of stress rate and strain rate, so that this assumption is true only for infinitesimal increments of strain.

(b) Substitution of equation (9) into equation (8) results in terms such as $C_{IJKL}\ (e_{KL}\ \delta\eta_{IJ} + \eta_{KL}\ \delta e_{IJ})$ and $C_{IJKL}\ \eta_{KL}\ \delta\eta_{IJ}$. These terms are functions of the unknown displacement increment, and hence these terms must either be omitted or their effects must be approximated.

(c) If the prescribed surface tractions and body forces are deformation dependent (for example, hydrostatic pressure loads), then the integrals over the area A_2 and volume V_2 in configuration C_2 can be evaluated only approximately.

The effect of using a linear approximation is that the external virtual work of the surface tractions and body forces in configuration C_1 will not generally be equal to the internal virtual work of the state of stress in C_1. Corrections to compensate for this error must be made to reduce departure of the computed response from the true response.

The form of the linearized equations, including the correction for inequality of the virtual work, can be written as follows.

$$\int_{V_o} [C_{IJKL}\ e_{KL}\ \delta e_{IJ} + {}^1S_{IJ}\ \delta\eta_{IJ} + {}^o\rho\ \delta u_K\ \ddot{u}_K] dV$$

$$= \left[\int_{A_o} \delta u_I\ {}^2t_I\ dA + \int_{V_o} \delta u_I\ {}^2f_I\ dV \right] - \int_{V_o} ({}^1S_{IJ}\ \delta e_{IJ} + {}^o\rho\ \delta u_K\ {}^1\ddot{u}_K) dV \quad (10)$$

in which A_o and dA are the surface area with prescribed tractions and the differential area, respectively, in C_o; and ${}^1\ddot{\underset{\sim}{u}}$ is the acceleration vector in C_1. All other terms have been defined earlier. It is assumed that the surface tractions and body forces are not deformation-dependent, their components being defined per unit area and volume in the undeformed configuration C_o.

2.3 FINITE ELEMENT FORMULATION

To obtain a finite element formulation, the integrals in equation (10) are evaluated over the volume and area of the element. Once the behavior of the element has been defined in terms of the nodal values of the field variables (in this case nodal displacements), the complete finite element model can be assembled by applying well-documented procedures [9].

The characteristics of any finite element can be derived by specifying appropriate approximations for the displacement field within the element. These approximations can then be substituted into the variational equation (10) and the strain-displacement relationships (6,7) to obtain the discrete formulation.

2.3.1 Displacement Approximations

The displacement field within the element is approximated by

$$ {}^1u_I(\underset{\sim}{X},t) = N^m(\underset{\sim}{X})\ {}^1q^e_{mI}(t) \qquad (11) $$

where ${}^1u_I(\underset{\sim}{X},t)$ are the components of displacement of the material point $\underset{\sim}{X}$ at time t in configuration C_1; $N^m(\underset{\sim}{X})$ is the interpolation function corresponding to node m of element e; and ${}^1q^e_{mI}(t)$ is the I^{th} component of displacement at node m. The repeated index m is summed over all nodes of the element.

The same interpolation functions are used to approximate increments of displacement between configurations C_1 and C_2. That is,

$$ u_I(\underset{\sim}{X},t) = N^m(\underset{\sim}{X})\ q^e_{mI}(t) \qquad (12) $$

where u_I and q^e_{mI} are components of the displacement increments of the material point $\underset{\sim}{X}$ and the node m, respectively.

Equations (11) and (12) can both be rewritten in matrix form, as

$$ {}^{1}\underset{\sim}{u} = \underline{N} \cdot {}^{1}\underset{\sim}{q}^{e} \tag{13a} $$

and

$$ \underset{\sim}{u} = \underline{N} \cdot \underset{\sim}{q}^{e} \tag{13b} $$

2.3.2 Geometric Approximations

For the isoparametric family of finite elements [10], the geometry within the element is approximated by the same interpolation functions that are used to specify the displacement field within the element. Thus, the coordinates of the material point $\underset{\sim}{X}$ are expressed in terms of the nodal coordinates as

$$ X_I = N^m(\underset{\sim}{X}) \; X^e_{mI} \tag{14} $$

where X_I is the I^{th} coordinate of the material point and X^e_{mI} is the I^{th} coordinate of node m of element e.

2.3.3 Strain-Displacement Relationships

From equation (6), the linear part of the Lagrangian strain increment can be written in matrix form as

$$ \underset{\sim}{e} = {}^{1}\underline{F} \cdot \underset{\sim}{u}_{\partial} \tag{15} $$

where ${}^{1}\underline{F}$ is the matrix of deformation gradients evaluated in the current deformed configuration C_1 and $\underset{\sim}{u}_{\partial}$ is the vector of displacement gradients.

Substituting the displacement approximation, equation (12), we obtain the linear strain-displacement transformation in terms of the nodal displacements. That is,

$$\underset{\sim}{u}_\partial = \underline{N}_\partial \cdot \underset{\sim}{q}^e \tag{16}$$

Hence,

$$\underset{\sim}{e} = {}^1\underline{F} \cdot \underline{N}_\partial \cdot \underset{\sim}{q}^e \tag{17a}$$

or

$$\underset{\sim}{e} = {}^1\underline{B} \cdot \underset{\sim}{q}^e, \; {}^1\underline{B} = {}^1\underline{F} \cdot \underline{N}_\partial \tag{17b}$$

where $\underline{N}_\partial$ is the matrix of derivatives of the interpolation functions with respect to the coordinates X_I $(I = 1, 2, 3)$.

The deformation gradient matrix ${}^1\underline{F}$ contains displacement gradients in configuration C_1 (i.e. terms such as ${}^1u_{I|J}$) which can be obtained as follows:

$${}^1\underset{\sim}{u}_\partial = \underline{N}_\partial \cdot {}^1\underset{\sim}{q}^e \tag{18}$$

because the same functions approximate both $\underset{\sim}{u}$ and ${}^1\underset{\sim}{u}$.

An explicit relationship such as that derived for linear strains is not defined between nonlinear strains η_{IJ} and nodal displacements. However, the second integral on the left hand side of equation (10) can be evaluated as follows. We can write

$$2\,{}^1S_{IJ}\,\eta_{IJ} = \underset{\sim}{u}_\partial^T \cdot {}^1\hat{\underline{S}} \cdot \underset{\sim}{u}_\partial \tag{19}$$

where ${}^1\hat{\underline{S}}$ is the symmetric stress matrix containing Piola-Kirchhoff stress components in the current configuration C_1, and the superscript T denotes transposition. This equation follows from the fact that the nonlinear strains are quadratic in the displacement gradients, as stated in equation (7).

Substituting equation (16) into equation (19) we have

$$2\ {}^{1}S_{IJ}\ \eta_{IJ} = (\underline{N}_{\partial} \cdot \underset{\sim}{q}^{e})^{T} \cdot {}^{1}\hat{\underline{S}} \cdot (\underline{N}_{\partial} \cdot \underset{\sim}{q}^{e}) \tag{20}$$

Hence, taking a variation on η_{IJ} we have

$${}^{1}S_{IJ}\ \delta\eta_{IJ} = (\delta \underset{\sim}{q}^{e})^{T} \cdot (\underline{N}_{\partial}^{T} \cdot {}^{1}\hat{\underline{S}} \cdot \underline{N}_{\partial}) \cdot \underset{\sim}{q}^{e} \tag{21}$$

Equation (21) can be used to evaluate the integral in equation (10).

2.3.4 Element Matrices

Following typical finite element methodology, the element matrices are obtained by evaluating each term in the variational equation (10) over the surface area A_e and volume V_e of the element in its undeformed configuration C_o. The following relationships result.

(a) Consistent Mass Matrix.

$$\underline{M}^{e} = \int_{V_e} {}^{o}\rho\ \underline{N}^{T} \cdot \underline{N}\ dV \tag{22}$$

(b) Linear Stiffness Matrix.

$${}^{1}\underline{K}_{E}^{e} = \int_{V_e} {}^{1}\underline{B}^{T} \cdot \underline{C} \cdot {}^{1}\underline{B}\ dV \tag{23}$$

(c) Geometric Stiffness Matrix.

$${}^{1}\underline{K}_{G}^{e} = \int_{V_e} \underline{N}_{\partial}^{T} \cdot {}^{1}\hat{\underline{S}} \cdot \underline{N}_{\partial}\ dV \tag{24}$$

(d) Nodal Loads due to the State of Stress

$${}^{1}\underset{\sim}{R}^{e} = \int_{V_e} {}^{1}\underline{B}^{T} \cdot {}^{1}\underset{\sim}{S}\ dV \tag{25}$$

(e) Nodal Loads due to Applied Forces.

$$ {}^2\underset{\sim}{p}^e = \int_{A_e} \underline{N}^T \cdot {}^2\underset{\sim}{t}\, dA + \int_{V_e} \underline{N}^T \cdot {}^2\underset{\sim}{f}\, dV \qquad (26) $$

Here $\hat{\underline{N}}$ is the matrix of interpolation functions relating displacements of the surface area over which tractions are prescribed to the nodal displacements.

In most cases, the integrals in equations (22-26) must be evaluated numerically. For nonlinear materials, the constitutive matrix $\underline{C}$ is evaluated in the current configuration.

2.3.5 Discrete Incremental Equations

The discrete incremental equations of motion for an undamped system are

$$ \underline{M} \cdot \ddot{\underset{\sim}{q}} + [{}^1\underline{K}_E + {}^1\underline{K}_G] \cdot \underset{\sim}{q} = {}^2\underset{\sim}{P} - (\underline{M} \cdot {}^1\ddot{\underset{\sim}{q}} + {}^1\underset{\sim}{R}) \qquad (27) $$

where $\underset{\sim}{q}$ and $\ddot{\underset{\sim}{q}}$ are the vectors of increments of nodal displacement and acceleration, respectively; and ${}^1\ddot{\underset{\sim}{q}}$ is the vector of nodal accelerations in the current configuration C_1.

The structure matrices $\underline{M}$, ${}^1\underline{K}_E$ and ${}^1\underline{K}_G$ and the vectors ${}^2\underset{\sim}{P}$, ${}^1\underset{\sim}{R}$ are obtained from the element matrices using well known assembly procedures [9].

The equations of equilibrium for static analysis can be obtained from equation (27) by omitting the terms containing accelerations. Viscous effects may be included by modifying equation (27) as follows.

$$ \underline{M} \cdot \ddot{\underset{\sim}{q}} + {}^1\underline{C} \cdot \dot{\underset{\sim}{q}} + [{}^1\underline{K}_E + {}^1\underline{K}_G] \cdot \underset{\sim}{q} $$

$$ = {}^2\underset{\sim}{P} - (\underline{M} \cdot {}^1\ddot{\underset{\sim}{q}} + {}^1\underline{C} \cdot {}^1\dot{\underset{\sim}{q}} + {}^1\underset{\sim}{R}) \qquad (28) $$

in which $\dot{\underset{\sim}{q}}$ and ${}^{1}\dot{\underset{\sim}{q}}$ are the vectors of velocity increment and velocity in configuration C_1, respectively; and ${}^{1}\underline{C}$ is a damping matrix.

Because of the linearization, equation (27) or (28) will yield only an approximate solution for the displacement increment between configurations C_1 and C_2. In general, the structure response will be computed by applying the load in small steps, and in some cases equilibrium iterations may have to be carried out to obtain results with a sufficient degree of accuracy. The selection of a scheme for the solution of these equations constitutes an important part of the design of a general nonlinear analysis computer program. Because no single solution scheme can be identified as optimal for all types of nonlinear behavior, a general strategy incorporating a variety of solutions schemes has been implemented in the computer program.

2.4 SOLUTION TECHNIQUES

2.4.1 Classification

Most solution procedures for nonlinear analysis can be classified as either step-by-step or iterative. Both procedures have been widely used in static nonlinear analysis, and both are applicable to dynamic nonlinear analysis in which the response is computed by step-wise marching in time.

2.4.2 Step-by-Step and Iterative Procedures

In the step-by-step solution procedure the load is applied in several small steps and the structure is assumed to respond linearly within each step, the response being obtained without iteration. This procedure is simple to apply and has been widely used, particularly for elasto-plastic problems. However, unless the load steps are very small the computed response may deviate appreciably from the true response, because equilibrium is not satisfied exactly at any step. The accuracy of the computed response can be improved by applying equilibrium correction terms, as was noted earlier.

Two types of iterative procedure are commonly used, namely Newton-Raphson iteration and Constant Stiffness iteration. In Newton-Raphson iteration the structure tangent stiffness matrix is reformed at every iteration, and a disadvantage of this procedure is that a large amount of computational effort may be required to form and decompose the stiffness matrix. In Constant Stiffness iteration the stiffness matrix is formed only once, usually in configuration C_1. Constant stiffness iteration will typically converge more slowly than Newton-Raphson iteration, and schemes to accelerate convergence may be desirable. It may also be advantageous to use mixed strategies incorporating a combination of Newton-Raphson and Constant Stiffness iteration. Mixed iteration schemes are considered later in this paper. Step-by-step and iterative procedures are shown diagrammatically in Figs. 2 and 3.

If we define a vector function $\underset{\sim}{f}$ such that

$$\underset{\sim}{f}(\underset{\sim}{v},\underset{\sim}{P}) = \underset{\sim}{P} - [\underline{M} \cdot \ddot{\underset{\sim}{v}} + \underline{C} \cdot \dot{\underset{\sim}{v}} + \underset{\sim}{R}(v)] \tag{29}$$

then the displacements $^2\underset{\sim}{q}$ are the solution of the nonlinear equations $\underset{\sim}{f}(^2\underset{\sim}{q},\, ^2\underset{\sim}{P}) = 0$. Computational formulas for step-by-step and iterative procedures can be obtained by considering a first order Taylor series expansion of $\underset{\sim}{f}(^2\underset{\sim}{q},\, ^2\underset{\sim}{P})$ about the known displacements $^1\underset{\sim}{q}$, and assuming $^2\underset{\sim}{q} = {}^1\underset{\sim}{q} + \underset{\sim}{q}$ [1 , 7]. Convergence in iterative procedures can be checked by comparing the norm of the residual load vector $\underset{\sim}{f}(^2\underset{\sim}{q},\, ^2\underset{\sim}{P})$ in any iteration with the norm of the vector $^2\underset{\sim}{P}$.

2.4.3 Acceleration Scheme

The principal advantage of Constant Stiffness iteration is that the tangent stiffness matrix is not formed and inverted (decomposed) at every iteration. Hence, this procedure is computationally attractive for structures with a large number of degrees of freedom. However, the procedure can be expected to converge more slowly than Newton-Raphson iteration, and the use of a scheme to accelerate convergence may be desirable.

The computer program includes the "alpha-constant" acceleration scheme [11]. In this scheme the displacement increments during any iteration are scaled in an attempt to obtain the same result as if Newton-Raphson iteration were employed. The computational steps of the scheme are given in Table 1. For each iteration the scheme requires two steps of displacement computation (steps c, g), and two steps of residual load computation (steps a, e). Computation of the residual load involves state determination (i.e. determination of the state of stress given the state of strain), which requires the material characterization and assumes particular significance for nonlinear materials with memory.

The scheme as implemented in the program differs from the one indicated in Table 1, in that steps (a) through (d) and steps (e) through (i) are performed every second iteration. Further, the scaling matrix is reinitialized to a unit matrix if the iteration fails to converge within a specified number of iterations or if an increase in residual load is detected from one iteration to the next. These restrictions are intended to prevent possible divergence.

The scheme can be expected to improve the rate of convergence for moderately nonlinear structures in which the load-displacement curve is of a softening type. However, for structures with stiffening load-displacement response, or for loadings which produce stress reversals, the acceleration scheme has been found to be unreliable and should be used with **caution**. The scheme can be used in both static and dynamic analysis.

2.5 INTEGRATION OF EQUATIONS OF MOTION

2.5.1 Newmark's β-γ-δ Operator

For integration of equation (28), the time domain is divided into a number of time steps, and it is required to compute the displacements, velocities and accelerations in configuration C_2 (time $\tau = t + \Delta t$) given the previous deformation history from time 0 to time t. An implicit, single step, two-parameter (β,γ) family of integration operators has been described by Newmark [12], in which it is assumed that the increments in velocity and acceleration are related to the increment in displacement and the state of motion at time t. With Newmark's operator, the equations of motion reduce to the following form.

$$\underset{\sim}{K}^*_t \cdot \underset{\sim}{q} = \underset{\sim}{f}^*_t \qquad (30)$$

Here, the effective stiffness matrix $\underline{K}_t^*$ is given by

$$\underline{K}_t^* = \frac{1}{\beta(\Delta t)^2} \underline{M} + \frac{\gamma}{\beta \Delta t} \underline{C}_t + \underline{K}_t \tag{31}$$

and the effective load fector $\underset{\sim}{f}_t^*$ is given by

$$\begin{aligned} \underset{\sim}{f}_t^* &= \underset{\sim}{f}(\underset{\sim}{q}_t, \underset{\sim}{P}_\tau) + \underline{M} \cdot [\frac{1}{\beta \Delta t} \dot{\underset{\sim}{q}}_t + \frac{1}{2\beta} \ddot{\underset{\sim}{q}}_t] \\ &+ \underline{C}_t \cdot [\frac{\gamma}{\beta} \dot{\underset{\sim}{q}}_t + \Delta t (\frac{\gamma}{2\beta} - 1) \ddot{\underset{\sim}{q}}_t] \end{aligned} \tag{32}$$

In equations (30-32) quantities associated with configuration C_1 are denoted with the subscript **t**, and those associated with configuration C_2 with the subscript τ. The tangent stiffness matrix $\underline{K}_t$ is equal to ${}^1\underline{K}_E + {}^1\underline{K}_G$. From equation (29), the vector $\underset{\sim}{f}(\underset{\sim}{q}_t, \underset{\sim}{P}_\tau)$ is given by

$$\underset{\sim}{f}(\underset{\sim}{q}_t, \underset{\sim}{P}_\tau) = \underset{\sim}{P}_\tau - (\underline{M} \cdot \ddot{\underset{\sim}{q}}_t + C_t \cdot \dot{\underset{\sim}{q}}_t + \underset{\sim}{R}_t) \tag{33}$$

A detailed derivation of equations (30-32) may be found elsewhere [1].

Newmark's operator has been used extensively, and its stability and accuracy characteristics in linear analysis have been documented [13]. However, the stability limits may no longer be valid in nonlinear analysis. Moreover, the accuracy of the response in nonlinear analysis will depend on the type of nonlinearity, the solution scheme, and the iteration process, so that the accuracy and stability of the operator for nonlinear analysis can be studied only by numerical experimentation.

A number of operators can be obtained by specifying various values of the parameters β and γ . The "constant average acceleration" operator with $\beta = 1/4$, $\gamma = 1/2$ has been shown to be unconditionally stable for linear analysis. It is possible to introduce artificial viscous effects by specifying a damping parameter δ [13]. In most cases, however, viscous effects in structures will be introduced explicitly through a damping matrix, $\underset{\sim}{C}_t$.

2.5.2 Integration Algorithm with Iteration

With Newmark's operator, an integration algorithm can be designed in which iterations are performed within a time step to satisfy equilibrium subject to a specified tolerance. The solution algorithm is shown in Table 2. If the time step is sufficiently small, it may not be necessary to reformulate the effective stiffness matrix $\underset{\sim}{K}_t^*$ of equation (31) at every time step, and it can be retained unchanged over several steps.

2.5.3 Damping Matrix

The damping matrix for a typical finite element can be derived by procedures analogous to those used for the element stiffness, provided internal damping characteristics are specified. A possible form for the damping matrix is

$$\underset{\sim}{C}_t^e = \int_{V_e} [\beta_m \, {}^o\rho \; \underset{\sim}{N}^T \cdot \underset{\sim}{N} + \beta_o \, {}^1\underset{\sim}{B}^T \cdot \underset{\sim}{D} \cdot {}^1\underset{\sim}{B} + \beta_T \, {}^1\underset{\sim}{B}^T \cdot \underset{\sim}{C} \cdot {}^1\underset{\sim}{B}] dV \quad (34)$$

in which β_m, β_o and β_T are proportionality factors; $\underset{\sim}{D}$ is the elastic constitutive tensor; and $\underset{\sim}{C}$ is the tangent constitutive tensor in configuration C_1. The first integral term in this equation defines a mass proportional damping matrix, whereas the second and third terms give damping matrices proportional to the initial elastic stiffness and the current tangent stiffness matrices, respectively.

3. COMPUTATIONAL STEPS

3.1 INTRODUCTION

In this chapter, the computational tasks involved in the nonlinear analysis problem are discussed. An overall solution strategy defined in terms of a number of control parameters is then outlined, and significance of these parameters is discussed in detail. Several solution schemes obtained by specifying different values to the control parameters are also presented.

3.2 COMPUTATIONAL TASKS

The nonlinear analysis problem involves three major tasks embedded in an overall solution strategy. These tasks are (a) linearization, (b) equation solution, and (c) state determination.

The evaluation of the tangent stiffness matrix in static analysis, or of the effective stiffness matrix in dynamic analysis, constitutes the linearization phase. The finite element formulation of the stiffness matrices has been presented in Chapter 2, and the form of the effective stiffness matrix using Newmark's operator for dynamic analysis has been considered.

The computation of an effective load vector, and the solution of a set of linear equations to determine a displacement increment constitute the equation solution phase. Formulas for the load vectors in static and dynamic analysis have been presented. Techniques for solving linear equations are well known and need not be discussed here. A particularly efficient algorithm for a symmetric matrix decomposition has been presented in [14] for in-core operation, and in [15] for out-of-core operation. This algorithm has been incorporated in the computer program.

When the displacement increment is computed, it is necessary to calculate the corresponding increments in stress and strain, and thus obtain a new state. This is the state determination phase. The relationships for computing strain increments from displacement increments have been stated earlier in Chapter 2. The problem of computing a stress increment from a given strain increment involves the material constitutive relationship, and for materials with dependence on strain history, the question of path dependence needs careful consideration. For such materials, a "true" loading path can be followed by using the step-by-step procedure with very small load steps. However, if an iterative procedure is used with large or moderately large load steps, it may be important to distinguish between "path dependent" and "path independent" state determinations.

3.2.1 Path Dependent and Path Independent State Determination

With path dependent state determination, the stresses are updated at the end of each iteration, based on the strain increments computed for that iteration. In contrast, with path independent state determination the stress increment is computed for all strain increments accumulated up to any iteration, and the stresses are updated only after the iteration process has converged. For cases in which the strains increase monotonically during iteration, the results with the two types of state determination can be expected to be in close agreement. However, if the strains do not increase monotonically, the two methods may give significantly different results. Path dependent state determination appears to be more consistent for Newton-Raphson iteration, whereas for Constant Stiffness iteration it is more logical to use path independent state determination.

3.3 SOLUTION STRATEGY

In the general case, the nonlinear response will be computed by a combination of the step-by-step and iterative procedures. Depending on the degree of nonlinearity, equilibrium iterations may or may not be needed, and the tangent or effective dynamic stiffness matrix may or may not be reformulated in every step. Because of the large computational effort typically required for decomposition of the tangent stiffness matrix, it will usually be desirable to seek a strategy in which the number of stiffness matrix reformulations is minimized. Nevertheless, in some cases it may also be necessary to consider the computational effort for the state determination calculations.

Basically, there are only two solution procedures included in the computer program, namely (a) Newton-Raphson iteration and (b) Constant Stiffness iteration; the step-by-step procedure is treated as a special case of Newton-Raphson iteration. However, it is possible to obtain a number of solution schemes by specifying different values to the control parameters, as described in the following section.

3.3.1 Parameters of the Solution Strategy

The control parameters are as follows:

(a) Number of steps (NSTEP). For static analysis, this parameter is the number of equal load steps. For dynamic analysis, it equals the number of time steps.

(b) Type of iteration procedure (ITYP). This parameter takes a value of zero for Newton-Raphson iteration and a value of one for constant stiffness iteration. If a value greater than one is assigned, constant stiffness iteration with the alpha-constant acceleration scheme will be used. The value of the parameter is then used to control reinitialization of the scaling matrix (α matrix). Reinitialization to a unit matrix is carried out if the number of iterations before convergence exceeds the value of the parameter.

(c) Type of state determination (KPATH). This parameter controls the state determination process for elasto-plastic materials. For Newton-Raphson iteration, only path dependent state determination can be used, and the parameter is ignored. For constant stiffness iteration the parameter takes a value of zero for path independent and one for path dependent state determination.

(d) Stiffness reformulation code (KRUSE). As noted in the preceding section, if the load step or time step is sufficiently small, then the tangent stiffness matrix may be kept constant over several steps. The value of this parameter can be used to control the frequency of stiffness reformulation. If the parameter is assigned a value of zero, then the tangent stiffness matrix or the effective matrix from the preceding step will be retained. If a value equal to n ($n \geq 1$) is specified, the stiffness matrix will be undated every n steps.

(e) Maximum number of iteration cycles within a step (MAXCYC). This parameter specifies the number of stiffness updates (cycles of iteration) permitted during constant stiffness iteration within a load step or time step. Typically, for Newton-Raphson iteration, this parameter will be assigned a value of one. However, with constant stiffness iteration it may be expected that convergence will be slow and that the stiffness may need to be updated occasionally. If convergence is not attained within a specified number of iterations, the stiffness will be updated, and a new "cycle" of iteration, with a new constant stiffness, will begin. If the solution does not converge after a specified number of iteration cycles, the solution will typically be terminated, although continuation with the next load step may be specified through use of the parameter IQUIT.

(f) Maximum number of iterations per cycle (MAXIT). This parameter controls the maximum number of iterations permitted in each iteration cycle.

(g) Fine convergence tolerance (TOLF). This parameter defines the convergence tolerance in the last load step of any series for static analysis. For dynamic analysis it specifies the convergence tolerance to be used in certain time steps, the frequency of which is controlled by the parameter NITF. The convergence criterion is based on residual load in any iteration.

(h) Coarse convergence tolerance (TOL). This convergence tolerance is used in all steps except the last in static analysis, and in all time steps except those at the specified frequency in dynamic analysis. This tolerance can be specified to be equal to the finer tolerance, TOLF, if desired, but will commonly be larger.

(i) Stiffness reformulation tolerance (TOLK). In certain cases it may be desirable to perform, in any load step, Newton-Raphson iterations initially, but as the changes in stiffness become progressively smaller, to retain the same stiffness and convert the solution procedure to constant stiffness iteration. With Newton-Raphson iteration, if the residual load vector at the end of any iteration satisfies this tolerance criterion, the previous stiffness will be retained until convergence.

(j) Frequency of time steps for fine convergence tolerance (NITF). This parameter controls the time step interval in dynamic analysis, at which iteration will be performed to satisfy the fine convergence tolerance, TOLF.

(k) Limit on displacement increment (DISLIM). It may be desirable to limit increments of displacement during any step of iteration. In most structures, it is possible to identify a principal component of displacement, which may be controlled using this parameter.

(l) Execution termination code (IQUIT). If convergence is not obtained within the specified number of iteration "cycles" and iterations per cycle, then it may be desirable to terminate execution to prevent waste of computer time. This parameter is assigned a value of one to terminate execution, or a value of zero to proceed to the next load step or time step.

3.3.2 Solution Schemes

By assigning appropriate values to the parameters described in the preceding section, it is possible to construct a variety of solution schemes. The basic procedures and some of the mixed solution procedures are identified in this section, and values of the parameters for these schemes are suggested in Table 3. The tolerances are rather loosely identified as "small", "moderate" or "large". Specific values of these tolerances will be influenced by the particular structure being analyzed. Some solution schemes are as follows:

1. Step-by-step procedure (without iteration) with stiffness reformulation every step.

2. Newton-Raphson iteration.

3. Constant stiffness iteration with stiffness reformulation every step.

4. Constant stiffness iteration using initial elastic stiffness throughout.

5. Step-by-step procedure with stiffness reformulation every step, but with constant stiffness iteration in the last load step for static analysis, or at specified time step intervals for dynamic analysis.

6. Automatic stiffness reformulation procedure, in which the stiffness is reformulated only if convergence is not obtained in a specified number of constant stiffness iterations.

7. Mixed iteration procedure in which Newton-Raphson iterations are followed by constant stiffness iterations.

All of these schemes can be used for either static or dynamic analysis. A limit on the displacement increment in any iteration may be specified, and for schemes with constant stiffness iteration the alpha-constant acceleration scheme may be used. Different convergence tolerances may be specified to obtain results more accurately in some steps than in others.

4. ADDITION OF ELEMENTS TO PROGRAM

4.1 INTRODUCTION

The computer program is organized so as to facilitate addition of new elements to the existing element library of the program. For this purpose, the program is divided into two parts, namely, (1) the base program consisting of a series of subroutines performing specific tasks required for static and dynamic analysis, and (2) a number of auxiliary programs, each program consisting of a package of subroutines required for a specific type of finite element in the element library. The user wishing to add a new element to the library is mainly concerned with the structure and organization of the auxiliary program, which will be described in the subsequent sections. The organization of the base program will not be described in this report; however, sufficient details will be given to provide an understanding of the linkage and information transmittal between the base program and the auxiliary program.

4.2 TRANSMITTAL OF INFORMATION

During input, the elements are arranged into groups, such that all elements in any group are of the same type. Depending on the type of element, the base program refers to the package of subroutines of the auxiliary program, at various phases of the computation. Information is transmitted to or returned from the subroutines of the auxiliary program through the argument lists and through labelled COMMON blocks.

For each element, a block of information is created, and is continually updated. All information to be retained for any element must be contained within this block. Because the core storage will usually be inadequate to store the information blocks for all elements, this information must be retained on secondary storage, usually a disc file, and retrieved from time to time. If each element information block were to be individually transmitted to or retrieved from the disc file each time it is required during computation, the number of input-output operations would be excessive. Therefore, to reduce the I/O cost, the base program automatically assembles "super" blocks of element information, each super block consisting of as many element information blocks as can be fitted into the available core storage. If the information for all elements can be held in core, there is no need to use a disc file, and hence there is no I/O cost associated with transmittal of element data to a disc.

The base program transfers the element information to a subroutine in the auxiliary program through the array COMS in the argument list. The address assigned to the array COMS in the base program corresponds to the first word of information for the corresponding element. To transfer the data from the COMS array to the element information block, the following FORTRAN statements must appear at the beginning of each auxiliary subroutine.

```
      COMMON/INFEL/IMEM, ..........
      DIMENSION COMS (1), COM (1)
      EQUIVALENCE (IMEM, COM (1))
      DO 100 J = 1, NINFC
  100 COM (J) = COMS (J)
```

in which NINFC = number of words in the COMMON block INFEL. The contents of the block INFEL will be described subsequently.

The data within the block INFEL will usually be updated during computations in the subroutine, so that it is necessary to transmit the updated data back to the array COMS at the end of the subroutine. This is achieved through the following FORTRAN statements.

```
      DO 200 J = 1, NINFC
  200 COMS (J) = COM (J)
```

It may be noted that in most cases only a part of the data is updated. Hence, it may be more efficient to transfer the modified data selectively. However, it can be expected that the computer time required to transfer data from array COMS to the block INFEL, and vice versa, will be a small proportion of the total execution time.

4.3 LABELLED COMMON BLOCKS

4.3.1 COMMON Blocks

The labelled COMMON blocks used in subroutines of the auxiliary program are as follows.

(a) COMMON/TAPES/NIU, NOU, NT1, NT2, NT3, NT4, NT5, NTEMP

(b) COMMON/INFEL/IMEM, KST, LM(...),...

(c) COMMON/WORK/WORK (2000)

4.3.2 Input/Output Unit Block (/TAPES/)

This block contains disc file units assigned by the base program. These should not be changed in any of the subroutines of the auxiliary program. NIU is the input unit to read data and NOU is the output unit to print data; whereas disc units NT1 through NT5 are used by the base program as scratch files for manipulation of data. Unit NTEMP is a temporary storage/retrieval disc available to the programmer during execution of the auxiliary program. If this unit is used in any auxiliary subroutine, it must be positioned at its starting point by the statement REWIND NTEMP before control is returned to the base program.

4.3.3 Element Information Block (/INFEL/)

This block contains all data to be retained for any element. The data can be arranged by the programmer in any desired order except for the following restrictions.

(1) The first word of the block must be the element number. The variable name IMEM is suggested.

(2) The second word must be the stiffness update code, as explained subsequently. Variable name KST is suggested.

(3) The third word must be the first word of the element location matrix. The suggested variable name is LM. The length of the vector LM equals the number of degrees of freedom of the element.

The remaining data of the block INFEL can be arranged in any desired order. This data will typically consist of element material properties, nodal coordinates, strain-displacement transformation matrices, current stiffness matrix, strains and stresses at integration points, envelope values of stresses and strains, plastic strains, etc.

4.3.4 Work Block (/WORK/)

This block provides a core area for use by the programmer. The work area provided by this block can be used for storage and manipulation of data during execution of any subroutine in the auxiliary program. Because this area is also used for temporary data storage by subroutines of the base program, it must not be used to transfer data between auxiliary subroutines.

4.4 AUXILIARY PROGRAM

4.4.1 General

Each auxiliary program consists of a package of subroutines required for a specific type of finite element. Each program consists of four main subroutines, as follows.

(a) INEL: Input and initialization of element information.

(b) STIF: Formation of element tangent stiffness in static analysis, or of element effective stiffness in dynamic analysis.

(c) RESP: Computation of element deformations (strains) and actions (stresses); determination of yield status; updating of element information; computation of equivalent nodal loads in equilibrium with the current state of stress; computation of equivalent damping loads; and printing of strain and stress results. As will be explained subsequently, control is exercised by the base program to perform selectively any one or a combination of the above operations.

(d) OUT: Output of envelope values of element deformations (strains) and actions (stresses) at specified load increments in static analysis or at specified time intervals in dynamic analysis.

Each of these four routines must be identified by a number designating the element type, suffixed to the subroutine name. For example, the names of subroutines for the element type 1 must be INEL1, STIF1, RESP1 and OUT1. The programmer can also write, if needed, additional secondary subroutines which are referenced by any one of the four main subroutines. At the end of such a subroutine control will be returned to a main subroutine, whereas at the end of a main subroutine control will be returned to the base program. Information may be transferred to and from secondary subroutines through argument lists, through the WORK common block, or through other labelled COMMON blocks created specifically for such information transfer.

Explanations of the tasks performed by each of the main subroutines, and the meanings of the variables of the argument lists, are given in the following sections.

4.4.2 Subroutine INEL

This subroutine is referenced by the base program once for each group of elements of the corresponding element type. For example, subroutine INEL1 will be called once for each group of elements containing elements of type 1.

The purpose of the subroutine is to read the input data for all elements in the group, and to initialize the variables in the element information block INFEL.

The subroutine requires labelled COMMON blocks TAPES and INFEL. The labelled COMMON block WORK may be used if desired. The argument list is as follows.

LPAR: A vector of dimension 10, which upon entry contains up to 10 control parameters for each element group.

FLPAR: A vector of dimension 6 which upon entry contains up to 6 control parameters for each element group.

NDOF: Number of element degrees of freedom.

NINFC: Number of words of information stored for each element in the element group. This number equals the length of the labelled COMMON block INFEL for elements of the type being considered.

NJT: Total number of nodes in the structure. This value is assigned by the base program.

NDKOD: An array of dimension (NJT x 6), which upon entry contains the numbers of the structure degrees of freedom. That is, NDKOD (I,1) thru NDKOD (I,6) contain the numbers of the structure degrees of freedom corresponding to the X displacement, Y displacement, Z displacement, X rotation, Y rotation and Z rotation, respectively, at node I. These values are generated by the base program, and must not be changed in the auxiliary program.

X,Y,Z: Vectors of dimension NJT each, which upon entry contain nodal coordinates. That is X(I), Y(I) and Z(I) contain the X, Y and Z coordinates, respectively of node I. These values are generated by the base program, and must not be changed in the auxiliary program.

The title card of the subroutine, for example for element type 1, must be as follows.

SUBROUTINE INEL 1 (LPAR, FLPAR, NDOF, NINFC, NDKOD, X, Y,Z, NJT)

The values of the control parameters in vectors LPAR and FLPAR are established within the base program by reading the first data card of each element group using a (10I5, 6F5.0) format. The first three control parameters in LPAR and the first two control parameters in FLPAR are stored by the base program as control parameters for the element group, and are used subsequently. These parameters must be as follows.

LPAR(1): A number identifying the type of element in the group. For example, if 4 is entered, the subroutines called for this group will be INEL4, STIF4, RESP4 and OUT4. Presently, this parameter can be assigned values 1 through 10.

LPAR(2): Number of elements in the group.

LPAR(3): Element number of the first element in the group.

FLPAR(1): Initial stiffness damping factor β_o.

FLPAR(2): Current tangent stiffness damping factor β_T.

All other words in LPAR and FLPAR can be assigned values, as needed, by the programmer. For example, in the currently developed two dimensional variable node finite element, the remaining words in LPAR are the number of material property sets, the number of nodes for each element in the group, the integration order in local r-direction, the integration order in local s-direction, the indicator for type of behavior (i.e. plane stress, plane strain or axisymmetric), the number of plastic strain increments for numerical integration during plastic loading, and the order of Runge-Kutta integration for numerical integration during plastic loading. These remaining control parameters are not retained by the base program for subsequent use, so that they must be stored as part of element information block INFEL if they are required later in the program.

All subsequent data for the elements are read within the subroutine INEL, with the sequence and input formats to be decided by the programmer.

The following steps must be performed within the subroutine.

(a) Set the values of the variables NDOF and NINFC.

(b) If desired, establish reference tables of material properties, fixed end forces, initial stresses etc. for later use in specifying properties for each element. The WORK block may be used to store these tables temporarily.

(c) Specify properties of each element in the group. This data will typically consist of node numbers, material properties, the initial state of stress, an indicator for inclusion of large displacement effects, etc. Any reference tables established in (b) may be used. Generation options may be incorporated, provided the elements are generated in element number sequence and information for only one element at a time is stored in the COMMON block INFEL.

(d) For each element, the following initialization operations must be performed.
(1) Set up the element location matrix, LM, within the COMMON block INFEL. This can be done with reference to the numbers of the structure degrees of freedom contained in the array NDKOD, and the element node numbers.
(2) Set IMEM to the element number within the group. Set the stiffness update code KST to one (KST = 1).
(3) Set any status indicators established within the COMMON block INFEL to appropriate values. Such indicators will typically be used to indicate whether or not large displacement effects are to be considered; to monitor yield status; to control printing of stress-strain history results; etc.
(4) Compute and save, within the block INFEL, strain-displacement transformation matrices for formation of element stiffness terms and for state determination calculations to be carried out in the auxiliary routines STIF and RESP, respectively. It should be noted that the nodal coordinates X, Y, Z are not transferred by the base program to the auxiliary routines STIF and RESP. However, the programmer may retain the nodal coordinates for the nodes to which the elements connects, as part of the INFEL block, if desired.
(5) Call subroutine BAND with the statement

```
CALL BAND (LM, NDOF)
```

This permits the base program to establish information on the profile of the structure stiffness matrix. This call must be made subsequent to the setting up of the element location matrix LM.

(6) Call subroutine COMPACT with the statement

CALL COMPACT

This transfers data from the INFEL block to a disc file assigned by the base program. This call must be made after the element information in the block INFEL has been fully initialized. It transfers the first NINFC words of the block to the scratch file.

4.4.3 Subroutine STIF

This subroutine is referenced by the base program each time a change in element stiffness is to be calculated, unless the solution control parameters are such that the structure stiffness from a previous step is to be retained. The subroutine is referenced in the following situations.

(a) For the first step in either a static analysis or a dynamic analysis, the subroutine is referenced by the base program once for each element. For static analysis, the load steps are numbered sequentially in decreasing order by the base program (ISTEP = 0, -1, -2, ..., etc.) whereas for dynamic analysis the time steps are numbered sequentially in increasing order (ISTEP = 1, 2, 3, ..., etc.). Thus, when ISTEP = 0, the subroutine is called once for each element to form the initial elastic stiffness; whereas when ISTEP = 1, it is called once for each element to form the effective stiffness matrix, which includes contributions due to the inertial and/or damping matrix terms.

(b) The static solution control parameters or the dynamic solution control parameters determine the frequency with which the subroutine will be referenced. Situations will arise when the solution control parameters specify no reference to the subroutine even when a stiffness change is indicated for one or more elements. However, these situations are dealt with in the base program.

As with the subroutine INEL1, the subroutine STIF1 will be called for elements of type 1. The purpose of the routine is to compute a change in element stiffness, and transfer this change to the base program for subsequent assembly into the structure stiffness matrix. Because the structure stiffness matrix is not necessarily updated at every load step, time step, or iteration, the change in the element stiffness must reflect the change since the last update.

The subroutine requires the labelled COMMON block INFEL. The labelled COMMON block WORK may be used if desired. The argument list is as follows.

ISTEP: Load step number, or time step number. This value is assigned by the base program.

NDOF: See Section 4.4.2. This value is now assigned by the base program.

NINFC: See Section 4.4.2. This value is now assigned by the base program.

CDKO: Value of constant $a_4\beta_0$ to be used in computing the contribution of the damping terms to the effective stiffness matrix in dynamic analysis (See Table 2). This value is assigned by the base program.

CDKT: Value of constant $a_4\beta_T$ to be used in computing the contribution of the damping terms to the effective stiffness matrix in dynamic analysis (See Table 2). This value is assigned by the base program.

COMS: A vector of dimension NINFC, which upon entry contains the element information. The address assigned to COMS in the base program corresponds to the first word of information for the element.

FK: An array of dimension of at most (NDOF x NDOF), into which is to be placed the change in the element stiffness matrix since the last update. See explanation below.

INDFK: Indicator to specify the storage arrangement of the element stiffness matrix in the array FK. The programmer is required to assign a value of zero or one to INDFK in this subroutine, as explained in the following.

The element stiffness matrix can be stored in the array FK either (1) as a square symmetric matrix of dimension (NDOF x NDOF) or (2) as a vector in which the columns of the lower part of the symmetric stiffness matrix are stacked together compactly. The number of words in the vector of form (2) will be NDOF x (NDOF + 1)/2. The programmer is required to assign, to INDFK, a value of zero if the element stiffness is stored as in (1), or a value of one if the element stiffness is stored as in (2). The base program uses INDFK in the assembly of the element stiffness matrix into the structure stiffness matrix.

The title card of the subroutine, for example for element type 1, must be as follows.

SUBROUTINE STIF 1 (ISTEP, NDOF, NINFC, CDKO, CDKT, FK, INDFK)

The following steps must be performed within the subroutine.

(a) Transfer the data from the array COMS to the element information block INFEL. The procedure explained in Section 4.2 must be used.

(b) Set INDFK to zero or one, as appropriate.

(c) For static analysis ($ISTEP \leq 0$), compute the change in the element tangent stiffness matrix. When ISTEP = 0, this change equals the initial elastic stiffness matrix. For dynamic analysis ($ISTEP \geq 1$), compute the change in the element effective stiffness matrix. Store the change in array FK, the storage scheme depending on the value assigned to INDFK.

(d) Set the stiffness update code (KST) to zero. Update any other data in the COMMON block INFEL.

(e) Transfer the information in the block INFEL to the array COMS. The procedure explained in Section 4.2 must be used.

4.4.4 Subroutine RESP

This subroutine is referenced by the base program for each element at each interation within a load step in static analysis, and at each interation within a time step in dynamic analysis.

As with the subroutine INELI, the subroutine RESP1 will be called for elements of type 1.

The tasks to be performed in this subroutine are: (T1) compute the element deformations (strains) and actions (stresses); (T2) determine the change of status if any; (T3) compute equivalent nodal loads in equilibrium with the current state of stress; (T4) compute equivalent damping loads; (T5) accumulate envelope values of element deformations (strains) and actions (stresses); (T6) update the element information; and (T7) print the strain and stress results. As explained subsequently, the base program specifies, through the indicator KUPD, which of the above tasks should be performed at any iteration in a load step or time step.

The subroutine requires the labelled COMMON blocks TAPES and INFEL. The labelled COMMON block WORK may be used if desired. The argument list for this routine is as follows.

NDOF: See Section 4.4.2. This value is assigned by the base program.

NINFC: See Section 4.4.2. This value is assigned by the base program.

MFST: Element number of the first element in the group. This value is assigned by the base program, and equals the control parameter LPAR(3). See Section 4.4.2.

KPR: Print indicator for element stress and strain results. This value is assigned by the base program. KPR is set equal to zero if the results are not to be printed, otherwise, it is set equal to the element group number.

COMS: A vector of dimension NINFC, which upon entry contains the element information. The address assigned to COMS in the base program corresponds to the first word of information for the element.

Q: A vector of dimension NDOF, which upon entry contains the increments in the element nodal displacements.

VEL: A vector of dimension NDOF, which upon entry contains the element nodal velocities.

ACC: A vector of dimension NDOF, which upon entry contains the element nodal accelerations.

FE: A vector of dimension NDOF, in which the nodal loads in equilibrium with the current state of stress must be returned.

FD: A vector of dimension NDOF, in which the damping loads at the element nodes must be returned.

TIME: Time, in seconds, at the current time step. This value is assigned by the base program. In static analysis, TIME = 0.0.

DKO: Initial stiffness damping factor, B_0. This value is assigned by the base program.

DKT: Tangent stiffness damping factor, β_T. This value is assigned by the base program.

C7: Value of a constant to be used in computing the contribution of damping to the effective load vector in dynamic analysis. This equals (a_5-1) as shown in Table 2. This value is assigned by the base program.

C8: Value of constant a_6 to be used in computing the contribution of damping to the effective load vector in dynamic analysis, as shown in Table 2. This value is assigned by the base program.

KUPD: An indicator controlling which task or combination of tasks is to be performed in this routine, as explained subsequently. The base program sets KUPD to a value of 1 through 4.

KITRN: An indicator specifying the form of the effective load vector in dynamic analysis. This value is assigned by the base program. If KITRN = 0, the effective load vector is required to be computed as in step 4 of Table 2. If KITRN = 1, the effective load vector is required to be computed as in step 7 of Table 2.

The values of MFST and KPR should be used by the programmer to print the element group number and an appropriate heading when the element stress and strain results are printed. Additionally, the programer can print selectively the results for certain elements within the group, with the aid of appropriate indicator stored as part of the element information.

The indicator KUPD is required to be used as follows, in performing the tasks (T1) through (T7) specified earlier.

(1) KUPD = 1: Perform tasks (T1) through (T7)

(2) KUPD = 2: Perform tasks (T1) through (T4) and (T7)

(3) KUPD = 3: Perform task (T7) only

(4) KUPD = 4: Perform tasks (T3), (T4) and (T7)

The computation of damping stresses and equivalent nodal loads due to damping is to be performed in dynamic analysis only (i.e. when TIME > 0.0).

The title card of the subroutine, for example for element type 1, must be as follows.

```
SUBROUTINE RESP1 (NDOF, NINFC, MFST, KPR, COMS, Q, VEL, ACC,
                  FE, FD, TIME, DKO, DKT, C7, C8, KUPD,
                  KITRN)
```

The following steps must be performed within the subroutine

(a) Transfer the data from the array COMS to the element information block INFEL. The procedure explained in Section 4.2 must be used.

(b) Perform the task (T1) through (T7), depending on the value of the indicator KUPD. If the element changes its status because of material yielding or unloading, set the stiffness update code (KST) to one. If large displacement effects are included for the element, KST must always be set to 1, because there will be a continuous change in the element geometry and hence in its stiffness. KST must be set prior to updating the element information in the block INFEL (i.e. prior to performing task (T6)).

(c) Transfer the information in the block INFEL to the array COMS. The procedure explained in Section 4.2 must be used. The transfer of this information must be carried out only if KUPD = 1.

4.4.5 Subroutine OUT

This subroutine is referenced by the base program for each element at selected static load increments and at specified time step intervals.

As with the subroutine INEL1, the subroutine OUT1 will be called for elements of type 1.

The purpose of this routine is to print the envelope values of stresses, strains and the corresponding times at which these maxima have occurred. The sequence and formats for printing these results are to be decided by the programmer. If the programmer decides to omit storing envelope values and corresponding times in the block INFEL, a dummy OUT subroutine must be supplied.

The subroutine requires the labelled COMMON blocks TAPES and INFEL. The labelled COMMON block WORK may be used if desired. The argument list is as follows.

NINFC: See Section 4.4.2. This value is assigned by the base program.

MFST: See Section 4.4.4. This value is assigned by the base program.

COMS: A vector of dimension NINFC, which upon entry contains the element information. The address assigned to COMS in the base program corresponds to the first word of information for the element.

The title card of the subroutine, for example for element type 1, must be as follows.

```
SUBROUTINE OUT1 (COMS, NINFC, MFST)
```

The following steps must be performed within the subroutine.

(a) Transfer the data from the array COMS to the element information block INFEL. The procedure explained in Section 4.2 must be used.

(b) Print an appropriate heading for the results if IMEM equals MFST.

(c) Print the envelope results.

4.5 CONCLUDING REMARKS

The preceding sections describe the organization of the auxiliary program, and the required transfers of information between the base program and the auxiliary program. The auxiliary program is structured so as to provide considerable flexibility to the programmer, except for certain rules regarding arrangement of data in the element information block and its transfer within the auxiliary subroutines.

Listings of two auxiliary programs, one for a three dimensional truss element and the other for a two dimensional 4-to-8 node finite element can be studied to aid in understanding of the organization of the auxiliary program.

5. SAMPLE APPLICATIONS

The results of the analyses of a number nonlinear structures are presented in this chapter. The main objective of these analyses has been verification of the program features and the various solution schemes implemented in the program. Wherever possible, the results have been compared with experimental, analytical and/or numerical results obtained by other investigators.

The structures analyzed include simple truss-bar systems and more complex plane and axisymmetric finite element systems. Structures with only geometric nonlinearity, with only material nonlinearity, and with combined material and geometric nonlinearity are included. Load-displacement relationships of both softening and stiffening types have been considered, and responses under both static and dynamic loadings have been computed. In most example analyses the results have been obtained using more than one solution scheme.

5.1 TRUSS-SPRING PROBLEM - LARGE DISPLACEMENT ELASTIC STATIC RESPONSE

Figure 4 shows the load-displacement response of a simple, geometrically nonlinear two-bar truss. The response was first obtained under an apex load of up to 48 lbs. using a load step of 1 lb. and Newton-Raphson iteration in each step. The solutions were repeated with loads steps of 3 lbs. and 6 lbs., and identical responses were obtained. In all cases convergence was fast, requiring an average of two to three iterations per load step with a specified tolerance of 0.01 lb. for the norm of the residual load vector. The Newton-Raphson iteration procedure required totals of 99, 41 and 26 stiffness formulations, and equal numbers of state determinations, for load steps of 1 lb., 3 lbs. and 6 lbs., respectively.

Excellent agreement between the present results and those of Noor [16] and Stricklin et al [17] was obtained, as shown in Fig. 4. For the present study, the response was also computed with constant stiffness iteration using the initial elastic stiffness, a load step of 6 lbs. and the alpha-constant acceleration scheme. Convergence was obtained in this case with an average of nine iterations per step, and the response was indistinguishable from that obtained with Newton-Raphson iteration. However, the process failed to converge beyond a load of 42 lbs. within a limit of 20 iterations.

The results of the Newton-Raphson iteration and the constant stiffness iteration with over-relaxation, using a load step of 6 lbs., are given in Table 4 , and are compared with those of Stricklin and Noor. As can be seen, the agreement is very close.

5.2 TWO-BAY PLANE TRUSS - LARGE DISPLACEMENT ELASTO-PLASTIC STATIC AND DYNAMIC RESPONSE

A two-bay plane truss, as shown in Fig. 5 , has been analyzed considering both large and small displacements. In each case the truss members were assigned an elasto-plastic constitutive relationship of Ramberg-Osgood type, with the parameters indicated in Fig. 5 . This relationship was modelled by decomposing it into a number of elastic-perfectly plastic components acting in parallel. The response of the truss under both static and dynamic loadings has been computed. The results are as follows.

(a) Static Response: Figure 6 shows the vertical displacement response obtained using a load step of 1 kip, and using (1) for large displacements, constant stiffness iteration with stiffness reformulation every step; (2) for large displacements, step-by-step with equilibrium correction and stiffness reformulation every step but no iteration; (3) for small displacements, constant stiffness iteration with stiffness reformulation whenever the structure yields. For iteration the convergence tolerance was specified to be 0.05 kip. Convergence was rapid, requiring an average of 4 iterations per step beyond a load of 6 kips, with almost linear behavior below 6 kips. The number of stiffness reformulations in each of the three cases were 11, 11 and 5, respectively, for 11 load steps.

With large displacement effects, the response shows very close agreement with results obtained by Noor [16]. As can be seen from Fig. 6 , the effect of large displacements on the elasto-plastic response of the truss is small for the range of loading considered. The solution at a load of 10 kips is compared with those of Goldberg et al [18] and Noor [16] in Table 5 . Again, the agreement is close.

(b) Dynamic Response: The dynamic response of the truss to a step load of 10 kips is shown in Fig. 7 . A lumped mass idealization was used, with the mass of each element lumped at the element ends. Newmark's average constant acceleration method (β = 1/4, γ = 1/2, δ = 0) was used, and the response was computed considering both large displacements and material nonlinearity.

The response was computed using three different time steps, namely, Δt_1 = 62 μsecs, Δt_2 = 124 μsecs and Δt_3 = 248 μsecs, corresponding to ratios of $T_o/100$, $T_o/50$ and $T_o/25$, respectively, where T_o is the fundamental period of the elastic truss. The linear response shown in Fig. 7 was computed using a time step of 124 μsecs only. For the nonlinear analyses, step-by-step procedure with equilibrium corrections was used, with the effective stiffness being reformed every step. With the time step of $\Delta t_1 = 62$ μsecs, the residual loads at each step were small, so that the response can be considered to be "exact" for the purposes of comparison. The agreement between the three response analyses can be seen to be close.

With the time step $\Delta t_3 = 248$ μsecs, the analysis was repeated with equilibrium iterations in each step. An average of 1.6 constant stiffness iterations per step were required. An indicated in Fig. 7 , this response closely matches that obtained with a time step of 124 μsecs and no equilibrium iterations. Computationally, the procedure with iteration and $\Delta t_3 = 248$ μsecs required a total of 25 stiffness formulations and 40 state determinations, whereas corresponding numbers for the procedure without iteration and $\Delta t_2 = 124$ μsecs were 50 and 50, respectively. Hence, the solution procedure with iteration proved to be computationally more efficient in this case.

5.3 THICK CYLINDER UNDER INTERNAL PRESSURE - SMALL DISPLACEMENT ELASTO-PLASTIC STATIC RESPONSE

An axially-restrained thick cylinder under internal pressure is a classical example which provides a convenient means of checking elasto-plastic computations. Only small displacement effects have been considered, and the material has been assumed to be elastic perfectly plastic, with the von Mises yield criterion. The geometry of the cylinder and the material parameters are shown in Fig. 8 (a). The finite element model of the cylinder is shown in Fig. 8 (b). Four 8-node axisymmetric finite elements were used through the thickness, with 2 x 2 Gauss quadrature integration.

The response was computed for two different loading sequences as follows:

(a) Internal pressure increased monotonically in steps of 0.5 lb/in^2 until the plastic region extended through 3/4 of the thickness of the cylinder.

(b) Cyclic increments of pressure, namely, $P = 0.0 \rightarrow 10.0 \rightarrow 12.5 \rightarrow 0.0 \rightarrow -10.0 \rightarrow -12.5 \rightarrow 0.0$ lbs/in^2.

The outer surface radial displacements for monotonically increasing pressure are shown in Fig. 9 . The results obtained with (a) Newton-Raphson iteration and (b) constant stiffness iteration using the initial elastic stiffness are plotted. These results show close agreement with an analytical solution given by Hodge and White [19]. The Newton-Raphson iteration required an average of 1.6 iterations per step and a total of 17 stiffness formulations. The constant stiffness iteration required an average of 7 iterations per step with only one stiffness formulation. In both cases the iteration was carried out to a convergence tolerance of 0.01 lb/in^2. Equilibrium iterations were required beyond a pressure of about 8 lbs/in^2.

The distribution of stresses within the cylinder at a pressure of 12.5 lbs/in^2, when the plastic zone extended through half of the thickness of the cylinder, is shown in Fig. 10 . Again the agreement between the results of the numerical solution and the results of Hodge and White [19] is very close.

To obtain the response for cyclic pressure loading, the constant stiffness iteration procedure using the initial elastic stiffness was used. The results for the outer surface radial displacements and stresses are shown in Figs. 11 and 12 . Only the stress distribution for the sequence of positive loading-unloading (i.e. $P = 0.0 \rightarrow 10.0 \rightarrow 12.5 \rightarrow 0.0$) are plotted, because the stresses for negative loading-unloading were symmetrical with respect to the initial state.

5.4 SHALLOW SPHERICAL CAP WITH APEX LOAD - LARGE DISPLACEMENT ELASTIC STATIC AND DYNAMIC RESPONSE

The behavior of a spherical cap under concentrated apex load has been studied for both static and dynamic loads. The geometry of the shell and the material properties are shown in Fig. 13 . Only geometric nonlinearity has been considered, and the material has been assumed to be linearly elastic. The finite element model of the cap consisted of ten 8-node axisymmetric elements, with 2 x 2 Gauss integration. For the dynamic analysis a lumped mass idealization was used and the integration was carried out using Newmark's average constant acceleration method ($\beta = 1/4$, $\gamma = 1/2$, $\delta = 0$).

For the static analysis the effects of load step size and iteration were studied. The results were compared with experimental results and numerical solutions obtained by other investigators. For the dynamic analysis, results were computed with different time steps and the need for equilibrium iterations was studied.

(a) Static analysis: The relationship between apex load and apex displacement is shown in Fig. 13. This response was computed using two load step sizes, namely 1 lb. and 5 lbs., up to a load of 100 lbs. For the load steps size of 1 lb., the simple step-by-step procedure with equilibrium correction but without iteration was used, with the stiffness reformed at each step. With the larger load step of 5 lbs., the displacement was calculated using this same step-by-step procedure and also using Newton-Raphson iteration.

The behavior (shell parameter $\lambda = 6$, $\lambda^2 = \sqrt{12(1-\nu)^2}\, a^2/(Rt)$) as shown in Fig. 13 is highly nonlinear, with initial softening and subsequent stiffening. The step-by-step procedure with a 1 lb. load step yields results which are in close agreement with those of the Newton-Raphson iteration using a 5 lb. load step. However, with the 5 lb. load step the step-by-step procedure showed considerable drift from the true solution. The computational economy of the procedures is indicated in Table 6 .

Figure 13 also shows a comparison with the numerical results of Haisler et al [20] and the experimental studies of Evan-Iwanovski et al [21]. It should be noted, that the experimental cap model had a shell parameter $\lambda = 6.23$. The present analysis also agrees fairly well with the finite difference solution of Mescall [22].

(b) Dynamic analysis: The displacement response of the cap subjected to a dynamic step load of 100 lbs. is shown in Fig. 14 .

The response was obtained using two different time steps, namely, $\Delta t_1 = 2$ μsecs (approximately $T_o/60$) and $\Delta t_2 = 4$ μsecs (approximately $T_o/30$), where T_o is the fundamental period of the cap. The following analyses were performed.

(1) Linear response (i.e. small displacement elastic analysis) with time step Δt_1.

(2) Linear response with time step Δt_2.

(3) Nonlinear response using step-by-step procedure with equilibrium correction but no iteration. The stiffness was reformed every step, and the time step Δt_1 was used.

(4) Same as analysis (3), but with the time step Δt_2.

(5) Same as analysis (3), but with constant stiffness iteration in each time step to a convergence tolerance of 10 lbs. The results of this analysis are assumed to be "exact" for the purposes of comparison.

The results are shown in Fig. 14 for a time duration of 500 μsecs.

The linear responses obtained with the two time steps are practically the same, except that some differences can be observed after the third cycle of vibration. For the nonlinear response, with a time step of 2 μsecs the difference between the response without iteration (analysis (3)) and that with iteration (analysis (5)) is small up to a duration of about 400 μsec; after which some amplitude decay and period elongation for analysis (3) is observed. Analysis (4) yields a response which differs considerably from the "exact" response. This indicates a need to perform equilibrium iterations when larger time steps are used.

Analysis (5) required an average of 4.1 iterations per step to achieve convergence to the tolerance of 10% of the applied load. Analysis (3) and (5) required equal numbers of stiffness formulations, namely 250. Analysis (4) required 125 stiffness formulations but the results were inaccurate.

5.5 SHALLOW ARCH WITH APEX LOAD - LARGE DISPLACEMENT ELASTIC STATIC RESPONSE

The large displacement response of a shallow elastic arch subjected to a concentrated apex load has been studied. The geometry of the arch and the material properties are shown in Fig. 15 . The material was assumed to be elastic and isotropic. The finite element model of the arch consisted of eight 8-node plane stress elements between the fixed end and the apex (i.e. symmetry was taken into account), and 2 x 2 Gauss integration was used.

The purpose of this analysis was to estimate the buckling load of the arch, and to compare the analysis with available experimental results and analyses carried out by other investigators. The effects of load step size and iteration were studied.

Figure 15 shows the displacement response with increasing load. Two different load steps, namely 1 lb. and 5 lbs., were used. Two different solution procedures were considered, namely (1) step-by-step procedure with equilibrium correction but no iteration and (2) constant stiffness iteration with stiffness reformed in each load step. The results obtained are shown in Fig. 15 and compared with the experimental results of Gjelsvik et al [23] and the numerical results of Mallet et al [24]. The iteration procedure with a load step of 5 lbs. failed to converge within 15 iterations and a tolerance of 0.5 lbs. beyond a load of 30 lbs. With a load step of 1 lb., the results without iteration and with iteration (tolerance = 0.1 lb.) were almost identical. With a load step of 2.5 lbs. and iterations (tolerance = 0.25 lb.) the results were again very close. However, with a 5 lb. load step the results diverged from those of the other procedures.

The computed results compare reasonably well with those of Mallet et al [24] and with experiment [23]. Dupuis et al [25] have also analyzed the arch using 32 "degenerate" shell elements of the type developed by Khojasteh-Bakht [26], with results showing considerable deviation from experiment.

5.6 CANTILEVER BEAM WITH UNIFORM LOAD - LARGE DISPLACEMENT ELASTIC STATIC AND DYNAMIC RESPONSE

The response of a cantilever beam subjected to uniform pressure has been computed, considering large displacement effects but assuming the material to be linearly elastic. The cantilever dimensions and material properties are given in Fig. 16 (a). The finite element model of the cantilever is shown in Fig. 16 (b). Five 8-node plane stress elements were used along the length, with 2 x 2 Gauss integration. A lumped mass idealization was used for the dynamic analysis, with one fourth of the mass of each element lumped at each of the four corner nodes. Newmark's average constant acceleration method (β = 1/4, γ = 1/2, δ = 0) was used for the dynamic response.

The aim of the analysis was to study the effect of equilibrium iterations, and in the case of static analysis to compare the results with an analytical solution. In the dynamic analysis, various ratios of time step to the initial elastic period were investigated, and iterative schemes were explored.

(a) Static analysis: The computed tip displacement for static loading is shown in Fig. 17 . This response was calculated using two solution schemes, namely (a) Newton-Raphson iteration, and (b) the step-by-step procedure with equilibrium correction but no iteration. In the step-by-step procedure the stiffness was reformed every step. The total pressure of 10 lbs/in^2 was applied in 20 equal steps for the Newton-Raphson iteration, and in 100 steps for the step-by-step procedure. The Newton-Raphson procedure required an average of 3.5 iterations per step, and a total of 71 stiffness formulations. Both results compare well with the analytical results of Holden [27].

(b) Dynamic analysis: The computed dynamic response of the cantilever subjected to a step load is shown in Fig. 18.

The computations were carried out using two different time steps, namely $\Delta t_1 = 45$ μsecs ($\approx T_o/120$) and $\Delta t_2 = 135$ μsecs ($\approx T_o/40$), where T_o is the fundamental period of the cantilever. The following analyses were considered:

(1) Small displacement analysis with $\Delta t = \Delta t_2$ (i.e. linear response).

(2) Large displacement analysis using the step-by-step procedure with equilibrium correction but no iteration. Time steps of both Δt_1 and Δt_2 were used, with the stiffness being reformed every step.

(3) Same as analysis (2), but with constant stiffness iteration within each time step to attain convergence to a tolerance of 0.1 lb.

(4) Large displacement analysis using the iterative procedure with "automatic" stiffness reformulation, and a time step Δt_2. In this procedure, constant stiffness iterations were performed in any step using the previously formed stiffness, and the stiffness was reformed only if convergence was not obtained within five iterations. The execution would have been terminated if the procedure failed to converge within a maximum of 3 cycles of 5 iterations each in any step. The convergence tolerance was 0.1 lb.

Analysis (3) with the short time step Δt_1 will be assumed to be "exact" for the purposes of comparison.

The results of the analyses are shown in Fig. 18 for a duration of 0.0135 secs. As is to be expected, the nonlinear response is stiffer than the linear response. The results obtained with analysis (4) are very close to those obtained with analysis (3) with a time step of 135 μsecs, and are not plotted. With a time step of 45 μsecs, the results obtained with and without iteration are very close, indicating that with such a small step equilibrium iterations can be omitted. With a time step of 135 μsecs, the results with iteration are very close to the "exact" results, whereas the results without iteration are grossly in error. This again emphasizes the need for equilibrium iterations when larger time steps are used. The results indicate that a time step of approximately $T_o/40$ is needed to obtain accurate response.

The computational efficiency of the various procedures can be assessed by comparing the numbers of stiffness reformulations, state determinations and iterations per time step. These are presented in Table 7 .

Among the analyses giving accurate results, the "automatic" stiffness formulation procedure,analysis (4), appears to be computationally most efficient. This efficiency can be attributed to the fact that the stiffness is reformed only when necessary, rather than arbitrarily at every time step.

5.7 CLAMPED BEAM - LARGE DISPLACEMENT ELASTIC STATIC AND DYNAMIC RESPONSE

The behavior of a fixed beam subjected to a central concentrated load has been studied. The geometry and material properties are shown in Fig. 19. Only large displacement effects have been considered, and the material was assumed to be linearly elastic.

The dynamic response of this beam has been studied by McNamara [28], and a similar problem has been solved by Weeks [29].

The finite element model of the beam consisted of five 8-node plane stress elements in the half span, with 2 x 2 Gauss integration. For the dynamic response a lumped mass idealization was used and the integration was carried out using Newmark's operator ($\beta = 1/4$, $\gamma = 1/2$, $\delta = 0$). In the studies by McNamara [28] the beam half-span was modeled using five beam bending elements, whereas Weeks [29] used a simpler model with approximate nonlinear effects.

The purpose of study has been to investigate the choice of load and time steps to obtain accurate results for a structure having a strongly stiffening response. The effects of equilibrium iteration on the response were also studied.

(a) <u>Static analysis</u>: The variation of central displacement with load is shown in Fig. 19 , for both linear and nonlinear response. The highly stiffening behavior of the beam can be seen, the linear displacement being several times larger than that obtained with a nonlinear solution. The load was applied in steps of 10 lbs. up to 100 lbs., and then in steps of 100 lbs. up to a total load of 700 lbs. Newton-Raphson iteration was used to obtain the response shown. An average of 3 iterations per load step and a total of 47 stiffness formulations were required to obtain convergence to a tolerance of 1 lb. For the same loading steps, using the step-by-step procedure with equilibrium correction but no iteration, the results obtained were very close to those obtained by Newton-Raphson iteration. The computations were repeated with Newton-Raphson iteration using a uniform load step of 100 lbs. and a limitation of 0.5 in. on the displacement increment in any iteration. The results are shown in

Fig. 19 , and are identical to those obtained with the variable load steps. An average of 3.9 iterations were required per step, with a total of 27 stiffness formulations. Newton-Raphson iteration with 100 lb. load steps and without a displacement limit failed to converge within 5 iterations in the first load step to the specified tolerance of 1 lb. The displacement limit has the effect of scaling down the load on any iteration, thereby preventing the solution from departing too far from the true solution.

(b) Dynamic analysis: The response of the clamped beam to a step load of 640 lbs. was also studied. As can be seen from the static analysis, the linear displacement at a load of 640 lbs. is several times larger than that with nonlinear effects. Therefore, it would be expected that the beam subjected to a dynamic load will vibrate with a period considerably shorter than the linear period of vibration. This influences the choice of time step.

Figure 20 shows a comparison of the linear and nonlinear responses of the beam. The linear responses were obtained with time steps of $\Delta t = 50$ μsecs and $\Delta t = 100$ μsecs. The nonlinear response was obtained with a time step $\Delta t = 50$ μsecs, using the step-by-step procedure without equilibrium iteration, the stiffness being reformed every step. The linear responses obtained with the two time steps are almost identical, and the period of vibration compares well with the value $T_o \approx 9056$ μsecs obtained using a beam formula [30]. On the other hand, the period of the first cycle of nonlinear vibration is seen to be approximately 2300 μsecs. The considerable difference in the maximum displacements of the linear and nonlinear solutions can also be noted.

Figure 21 shows the nonlinear responses obtained using various time steps and solution strategies for a duration of 5000 μsecs. Three different time steps, namely Δt_1 = 100 μsecs, Δt_2 = 50 μsecs, and Δt_3 = 25 μsecs were used, corresponding to ratios of approximately $T_o/90$, $T_o/180$ and $T_o/360$, respectively, in which T_o is the fundamental period of the elastic beam. The following analyses were performed.

(1) Nonlinear analysis using the step-by-step procedure with equilibrium correction but no iteration, with a time step Δt_1, and stiffness reformulation every step.

(2) Same as analysis (1), but with a time step Δt_2.

(3) Same as analysis (1), but with a time step Δt_3.

(4) Nonlinear analysis using iteration with "automatic" stiffness reformulation and a time step of Δt_2. The solution algorithm in this analysis was the same as that described in Section 5.6. Equilibrium iterations were performed in each step to a convergence tolerance of 64 lbs.

Analysis (3) is assumed to be "exact" for the purposes of comparison. The response with Δt_2 = 50 μsecs and without iteration (analysis (2)) is reasonably close to the "exact" response, except in some regions, whereas analysis (1) yields a response with considerable differences, again indicating a need for iteration to obtain accurate response. The response of analysis (4) is close to the exact response.

Computationally, analyses (1), (2) and (3) required 50, 100 and 200 stiffness formulations, respectively, and the same numbers of state determinations. Analysis (4) with the "automatic" stiffness reformulation required only 43 stiffness formulations, but 489 state determinations.

McNamara [28] analyzed the beam using a central difference operator with a time step of 5 μsecs, and obtained a maximum displacement of 0.90 inch compared to 0.77 inch in the present analysis. The nonlinear period of the first cycle as reported by McNamara was 2889 μsecs, compared with a period of 2300 μsecs of the present analysis. The reasons for the discrepancies between these two investigations are not clear and further study is necessary.

5.8 EIGHT-STORY SHEAR BUILDING - SMALL DISPLACEMENT ELASTO-PLASTIC DYNAMIC RESPONSE

An eight-story shear building modelled as a spring-mass system has been analyzed for dynamic response when subjected to the first six seconds of an artificially generated ground motion. The system parameters are shown in Fig. 22. The structure has alternating large and small masses, so that high frequency oscillations occur in the higher modes. The fundamental period, T_o of the elastic structure is 0.6 secs. Damping proportional to the initial stiffness was specified. The dynamic response was computed using Newmark's integration operator (β = 1/4, γ = 1/2, δ = 0).

For this structure large inelastic excursions were expected. In addition, because of the presence of the high frequency oscillations, it is not computationally economical to select a time step which is less than the shortest period of the structure.

The response was computed using four different time steps, namely Δt_1 = 0.0025 (T_o/240), Δt_2 = 0.005 (T_o/120), Δt_3 = 0.01 (T_o/60), and Δt_4 = 0.02 (T_o/30). Three solution strategies were used, as follows:

(a) Step-by-step procedure with equilibrium correction, and no iteration within any time step. The stiffness was reformulated whenever the structure yielded or unloaded.

(b) Step-by-step procedure with equilibrium iteration within each time step, and using the initial elastic stiffness throughout. A convergence tolerance of 0.1 kip was used.

(c) Same as strategy (b), but with a convergence tolerance of 0.5 kip.

Note that the convergence tolerances are both small fractions of the story shears at yield. The results of the studies are presented in Table 8, in terms of story ductility demands. The ductility demand for a story is defined as the maximum story drift divided by the drift at yield. Measures of the computational effort, particularly the numbers of stiffness formulations are also tabulated.

Results using a strategy identical to strategy (a), with time steps of Δt_1, Δt_2 and Δt_3, have previously been obtained by Ghose [31]. Those with the time step Δt_1 will be assumed to be "exact" for comparison.

As can be seen from Table 8, the ductilities computed with strategy (a) and a time step Δt_3 are reasonably accurate compared to the "exact" response. Using a time step Δt_2, the results with strategy (b) are as accurate as those with strategy (a). However, strategy (b) is computationally more efficient, as it requires only one stiffness formulation and triangularization, compared to 125 for strategy (a). Only an average of 1.7 iterations per step were required to achieve convergence to the tolerance of 0.1 kip. With a time step of $\Delta t_4 = 0.02$ secs, the results were inaccurate for all the three strategies, indicating that this time step is too large for accurate numerical integration. It may be noted that iteration to the fine convergence tolerance does not appreciably improve the results over iteration to the coarse tolerance.

Considering both accuracy and computational effort, strategy (b) with a time step Δt_2 = 0.005 secs was the best solution strategy among those considered.

5.9 PRESTRESSED CABLE NET-STATIC RESPONSE

The static response of a plane prestressed cable net is shown in Fig. 23. The overall geometry and elastic properties of the network are indicated in Fig. 23(a). More detailed information for the network is given by Argyris et al [32]. The interior cables are prestressed to a load of about 2000 kgf., whereas the exterior cables carry a prestressing force of nearly 18000 kgf. Because of the symmetry about the in-plane axes only one quarter of the net was analyzed. The cables were modelled using weightless three-dimensional truss elements.

The vertical displacement response of the net subjected to a lateral load of 10000 kgf. at each node was computed using Newton-Raphson iteration. The load was applied in 4 equal increments of 250 kgf. each followed by 9 equal increments of 1000 kgf. each. An average of 3.0 iterations was required per step. The results of the present analysis compare favorably with those of Argyris et al [32]. The final vertical displacement at node 1 was 155.63 as compared with 162.95 cm reported by Argyris.

5.10 PLANE STRAIN PUNCH PROBLEM - SMALL DISPLACEMENT ELASTO-PLASTIC STATIC RESPONSE

The behavior of a solid plane strain specimen subjected to indentation by a rigid punch has been studied. The material of the specimen was assumed to be elastic-perfectly plastic and only small displacement effects were included. The specimen dimensions and material properties are given in Fig. 24. For the finite element model, higher order (8-node) plane strain elements with 2x2 Gauss point integration were used in the region of the specimen directly under the punch, where yielding of the material is expected, whereas lower order (7-node and 4-node) elements were used in regions of the specimen away from the punch. Two different types of analysis were carried out, namely an imposed load analysis and an imposed displacement analysis. For the imposed load analysis, the punch was modelled as a number of finite elements with a very large value of Young's modulus. For the imposed displacement analysis, displacements were imposed through very stiff vertical springs. In both cases friction at the interface was assumed to be zero.

Figure 24 shows a plot of mean pressure versus punch indentation (both quantities are nondimensionalized). For the imposed displacements case, the total punch indentation of 0.008 was applied in three ways, namely (1) 16 equal steps of 0.0005 each, (2) 4 equal steps of 0.002 each, and (3) a single step of 0.008. In each case the response was computed using constant stiffness iterations with stiffness reformulation whenever the structure yielded or unloaded and path independent state determination. The results of the analysis with 16 equal steps are shown in Fig. 24. Nearly identical results were obtained with 4 equal steps (these results are not plotted). With the displacement applied in a single step very similar results were also obtained. The analysis with 16 steps required 13 stiffness formulations, 24 state determinations and an average of 1.6 iterations per step with a convergence tolerance of 1% of the yield stress. The corresponding numbers were 4, 26 and 6.5 for 4 equal steps, and 1, 25 and 25 for one step of imposed indentation. A larger convergence tolerance, equal to 5% of the yield stress, was specified for the single step case.

The analysis was also carried out using Newton-Raphson iteration and 4 equal steps. In this analysis path dependent state determination was used. However, the results obtained were totally incorrect, as shown in Fig. 24. Also, at the center of the most severely strained element a horizontal strain of 0.00915 and an effective stress of 8.47 were computed at the imposed displacement of 0.008, compared with a strain of 0.00979 and an effective stress of 13.0 (indicating yield) obtained using constant stiffness iteration with path independent state determination. This illustrates the need to use path independent state determination for imposed displacement cases.

For the imposed load computations, the total pressure of 1.2 was applied in 4 equal steps of 0.20 each followed by 10 equal steps of 0.04 each. The response was computed using Newton-Raphson iteration with path dependent state determination. The results show very close agreement with 16 steps of imposed displacement. This analysis required 18 stiffness formulations, 25 state determinations, and an average of 2.6 iterations per step beyond a pressure of about 0.92, up to which the response is seen to be linear. Because the strains increase progressively for the imposed loads, Newton-Raphson iteration with a path dependent state determination yielded correct results, in contrast to the totally incorrect results obtained with this solution scheme for the imposed displacements case. In both analyses, collapse was obtained at a pressure of 1.2. Essentially identical results were obtained by Nayak and Zienkiewicz [33], and the collapse load closely agrees with that predicted by a slip-line solution of the problem [34].

The yielded integration points for the imposed load case at a pressure of 1.2 are shown in the left half of Fig. 25. Yielded zones are shown for the imposed displacement case in the right half of this figure, at displacements of 0.002, 0.005 and 0.008, respectively.

6. CONCLUDING REMARKS

This report is intended to serve as a preliminary documentation report for the computer program. It is necessary to note the theoretical and computational formulations on which this program is based, and to interpret the results of the computer analysis in accordance with those formulations.

A number of new capabilities for the program are being developed. These consist of addition of new finite elements, including a three dimensional beam element, a three dimensional isoparametric solid element, and a degenerate thick and thin shell element. Additional material constitutive relationships, including elasto-plastic isotropic and kinematic hardening models, a parallel component model, and soil material models will be developed. It is also planned to add a restart capability to the program, and to include additional solution control parameters to improve the range of available solution schemes.

REFERENCES

1. Mondkar, D.P. and Powell, G.H., "Static and Dynamic Analysis of Nonlinear Structures", Report No. EERC 75-10, Earthquake Engineering Research Center, University of California, Berkeley, 1975.

2. Marcal, P.V., "Survey of General Purpose Programs", Proceedings, Second U.S.-Japan Seminar on Matrix Methods of Structural Analysis and Design, Berkeley, Aug. 1972.

3. MARC-CDC User's Manual, Control Data Corporation, Minneapolis, Minnesota, U.S.A.

4. ANSYS, Engineering Analysis Systems User's Manual, Swanson Analysis Systems, Inc., Pennsylvania, U.S.A.

5. Bathe, K-J and Wilson, E.L., "NONSAP-A Nonlinear Analysis Program", Nuclear Engineering and Design, Vol. 29, pp. 266-293, 1974.

6. Malvern, L.E., "Introduction to the Mechanics of A Continuous Medium", Prentice Hall, Inc., 1969.

7. Oden, J.T., "Finite Elements of Nonlinear Continua", McGraw-Hill, New York, 1972.

8. Larsen, P.K., "Large Displacement Analysis of Shells of Revolution Including Creep, Plasticity and Viscoelasticity", Report No. SESM 71-22, University of California, Berkeley, 1971.

9. Zienkiewicz, O.C., "The Finite Element Method in Engineering Science", McGraw-Hill, London, 1971.

10. Ergatoudis, I., Irons, B.M. and Zienkiewicz, O.C., "Curved, Isoparametric Quadrilateral Elements for Finite Element Analysis", International Journal of Solids and Structures, Vol. 4, pp. 31-42, 1968.

11. Nayak, G.C. and Zienkiewicz, O.C., "Note on the Alpha-Constant Stiffness Method For the Analysis of Nonlinear Problems", International Journal Numerical Methods in Engineering, Vol. 4, pp. 579-582, 1972.

12. Newmark, N.M., "A Method of Computation For Structural Dynamics", Journal of the Engineering Mechanics Division, ASCE, Vol. 85, No. EM3, pp. 67-94, 1959.

13. Goudreau, G.L. and Taylor, R.L., "Evaluation of Numerical Integration Methods in Elastodynamics", Computer Methods in Applied Mechanics and Engineering, Vol. 2, pp. 65-68, 1972.

14. Mondkar, D.P. and Powell, G.H., "Towards Optimal In-Core Equation Solving", International Journal Computers and Structures, Vol. 4, pp. 531-548, 1974.

15. Mondkar, D.P. and Powell, G.H., "Large Capacity Equation Solver for Structural Analysis", International Journal Computers and Structures, Vol. 4, pp. 699-728, 1974.

16. Noor, A.K., "Nonlinear Analysis of Space Trusses", Journal of the Structural Division, ASCE, Vol. 100, No. ST3, pp. 533-546, 1974.

17. Stricklin, J.A., Haisler, W.E. and Von Riesemann, W.A., "Geometrically Nonlinear Structural Analysis by Direct Stiffness Method", Journal of the Structural Division, ASCE, Vol. 97, No. ST9, pp. 2299-2314, 1971.

18. Goldberg, J.E. and Richard, R.M., "Analysis of Nonlinear Structures", Journal of the Structural Division, ASCE, Vol. 89 ,No. ST4, pp. 333-351, 1963.

19. Hodge, P.G. and White, G.H., "A Quantitative Comparison of Flow and Deformation Theories of Plasticity", Journal of Applied Mechanics, Vol. 17, pp. 180-184, 1950.

20. Haisler, W.E., Stricklin, J.A. and Stebbins, F.J., "Development and Evaluation of Solution Procedures For Geometrically Nonlinear Structural Analysis", AIAA Journal, Vol. 10, No. 3, pp. 264-272, 1972.

21. Evan-Iwanovski, R.M., Cheng, H.S. and Loo, T.C., "Experimental Investigation of Deformations and Stability of Spherical Shells Subjected to Concentrated Load at the Apex", Proceedings, Fourth U.S. National Congress of Applied Mechanics, ASME, pp. 563-575, 1962.

22. Mescall, J.F., "Large Deflections of Spherical Shells Under Concentrated Loads", Journal of Applied Mechanics, Vol. 32, pp. 936-938, Dec. 1965.

23. Gjelsvik, A. and Bodner, S.R., "The Energy Criterion and Snap Buckling of Arches", Journal of the Engineering Mechanics Division, ASCE, Vol. 88, No. EM5, pp. 87-134, 1962.

24. Mallet, R.H. and Berke, L., "Automated Method for the Large Deflection and Instability Analysis of Three Dimensional Truss and Frame Assemblies", AFFDL-TR-66-102, Dec. 1966.

25. Dupuis, G.A., Hibbitt, H.D., McNamara, S.F. and Marcal, P.V., "Nonlinear Material and Geometric Behavior of Shell Structures", International Journal Computers and Structures, Vol. 1, pp. 223-239, 1971.

26. Khojasteh-Bakht, M., "Analysis of Elastic-Plastic Shells of Revolution Under Axisymmetric Loading", SESM Report 67-8, Department of Civil Engineering, University of California, Berkeley, 1967.

27. Holden, J.T., "On the Finite Deflections of Thin Beams", International Journal of Solids and Structures, Vol. 8, pp. 1051-1055, 1972.

28. McNamara, J.F., "Solution Schemes for Problems of Nonlinear Structural Dynamics", Journal of Pressure Vessel Technology, ASME, Vol. 96, pp. 96-102, 1974.

29. Weeks, G., "Temporal Operators For Nonlinear Structural Dynamics Problems", Journal of the Engineering Mechanics Division, ASCE, Vol. 98, No. EM5, pp. 1087-1104, 1972.

30. Hurty, W.C. and Rubinstein, M.F., "Dynamics of Structures", Prentice Hall, Inc., 1967.

31. Ghose, A., "Computational Procedures for Inelastic Structural Dynamics", University of California, Berkeley, 1974.

32. Argyris, J.H., Dunne, P.C. and Angelopoules, T., "Nonlinear Oscillations Using the Finite Element Technique", Computer Methods in Applied Mechanics and Engineering, Vol. 2, pp. 203-250, 1973.

33. Nayak, G.C. and Zienkiewicz, O.C., "Elasto-Plastic Stress Analysis. A Generalization for Various Constitutive Relations Including Strain Softening", International Journal Numerical Methods in Engineering, Vol. 5, pp. 113-135, 1972.

34. Hill, R., "The Mathematical Theory of Plasticity", Clarendon Press, Oxford, 1950.

TABLE 1

Acceleration Scheme For Constant Stiffness Iteration

1. Given an initial guess ${}^{1}\tilde{q}$ and the tangent stiffness matrix $\underline{K}$, initialize

$${}^{2}\tilde{q}^{o} = {}^{1}\tilde{q}$$

$$\underline{\alpha}_{o} = \underline{I}_{n}$$

in which $\underline{\alpha}_o$ is a diagonal scaling matrix and $\underline{I}_n$ is a unit matrix of size n, n being equal to the number of degrees of freedom.

2. For the i^{th} iteration (i = 0,1,2,...., number of iterations) perform the following steps:

(a) Compute $\tilde{f}^{i} = \tilde{f}\ ({}^{2}\tilde{q}^{i}\ ,\ {}^{2}\tilde{P})$ = residual load.

(b) Check convergence: if $||\tilde{f}^{i}||_2/||\ {}^{2}\tilde{P}||_2 >$ tolerance, proceed to next step; otherwise terminate iteration. $||\cdot||_2$ denotes Euclidian vector norm.

(c) Compute $\Delta\tilde{q}^{i} = \underline{K}^{-1}\ \tilde{f}^{i}$

(d) Update ${}^{2}\tilde{q}^{i} = {}^{2}\tilde{q}^{i} + \underline{\alpha}_{i}\ \Delta\tilde{q}^{i}$

(e) Compute $\tilde{f}^{i} = \tilde{f}\ ({}^{2}\tilde{q}^{i}\ ,\ {}^{2}\tilde{P})$

(f) Check convergence: same as step (b).

(g) Compute $\delta\tilde{q}^{i} = \underline{K}^{-1}\ \tilde{f}^{i}$

(h) Update $\alpha_{i+1,j} = \alpha_{i,j} + \Delta q^{i,j}/\delta q^{i,j}$

where $j = 1,2,\ldots\ldots,n$ (number of degrees of freedom).

(i) Update ${}^{2}\tilde{q}^{i+1} = {}^{2}\tilde{q}^{i} + \delta\tilde{q}^{i}$

TABLE 2

Step-by-Step Integration Algorithm With Iteration

I. Basic Computation

1. Specify the parameters β, γ, time step Δt and convergence tolerance TOL.

2. Compute the following constants

$$a_1 = \frac{1}{\beta(\Delta t)^2} \qquad a_2 = \frac{1}{\beta\Delta t} \qquad a_3 = \frac{1}{2\beta}$$

$$a_4 = \frac{\gamma}{\beta\Delta t} \qquad a_5 = \frac{\gamma}{\beta} \qquad a_6 = \Delta t\left(\frac{\gamma}{2\beta} - 1\right)$$

3. Specify initial conditions $\underset{\sim}{q}_o$, $\dot{\underset{\sim}{q}}_o$ and $\ddot{\underset{\sim}{q}}_o$

II. For Each Time Step

4. Compute, if required, the effective stiffness matrix $\underline{K}^*_t$ and form the effective vector $\underset{\sim}{f}^*_t$. That is

$$\underline{K}^*_t = a_1 \underline{M} + a_4 \underline{C}_t + \underline{K}_t$$

$$\underset{\sim}{f}^*_t = \underset{\sim}{f}(\underset{\sim}{q}_t, \underset{\sim}{P}_\tau) + \underline{M} \cdot [a_2 \dot{\underset{\sim}{q}}_t + a_3 \ddot{\underset{\sim}{q}}_t]$$

$$+ \underline{C}_t \cdot [a_5 \dot{\underset{\sim}{q}}_t + a_6 \ddot{\underset{\sim}{q}}_t]$$

where $\tau = t + \Delta t$, $\underline{K}_t$ is the tangent stiffness matrix, and

$$\underset{\sim}{f}(\underset{\sim}{q}_t, \underset{\sim}{P}_\tau) = \underset{\sim}{P}_\tau - (\underline{M} \cdot \ddot{\underset{\sim}{q}}_t + \underline{C}_t \cdot \dot{\underset{\sim}{q}}_t + \underset{\sim}{R}_t)$$

5. Solve for $\underset{\sim}{q}$. That is $\underset{\sim}{q} = [\underline{K}^*_t]^{-1} \cdot \underset{\sim}{f}^*_t$

6. Update state of motion. At time $\tau = t + \Delta t$

$$\ddot{\underset{\sim}{q}}_\tau = \ddot{\underset{\sim}{q}}_t + a_1 \underset{\sim}{q} - a_2 \dot{\underset{\sim}{q}}_t - a_3 \ddot{\underset{\sim}{q}}_t$$

$$\dot{\underset{\sim}{q}}_\tau = \dot{\underset{\sim}{q}}_t + a_4 \underset{\sim}{q} - a_5 \dot{\underset{\sim}{q}}_t - a_6 \ddot{\underset{\sim}{q}}_t$$

$$\underset{\sim}{q}_\tau = \underset{\sim}{q}_t + \underset{\sim}{q}$$

7. Compute residual load vector

$$\hat{\underset{\sim}{f}} = \underset{\sim}{f}(\underset{\sim}{q}_\tau, \underset{\sim}{P}_\tau) = \underset{\sim}{P}_\tau - (\underline{M} \cdot \ddot{\underset{\sim}{q}}_\tau + \underline{C}_\tau \cdot \dot{\underset{\sim}{q}}_\tau + \underset{\sim}{R}_\tau)$$

in which $\underset{\sim}{R}_\tau$ is the load vector in equilibrium with the current state of stress.

8. Convergence check: if $||\hat{\underset{\sim}{f}}||_2/||\underset{\sim}{P}_\tau||_2 \leq$ TOL, no equilibrium iteration is needed and repeat steps 4 through 8 for next time step; otherwise proceed as follows. Here $||\cdot||_2$ denotes Euclidean norm of the undesignated vector.

9. Initialize the <u>diagonal</u> scaling matrix $\underline{\alpha}$ to a unit matrix $\underline{I}_n$ (or any other value from previous process) of size n, where n equals the number of structure degrees of freedom.

III. <u>For Each Iteration Within the Time Step</u>

10. Compute $\Delta \underset{\sim}{q} = [\underline{K}_t^*]^{-1} \cdot \hat{\underset{\sim}{f}}$

11. Update the state of motion. That is

$$\ddot{\underset{\sim}{q}}_\tau = \ddot{\underset{\sim}{q}}_\tau + a_1 \underline{\alpha} \cdot \Delta \underset{\sim}{q}$$

$$\dot{\underset{\sim}{q}}_\tau = \dot{\underset{\sim}{q}}_\tau + a_4 \, \underline{\alpha} \cdot \Delta \underset{\sim}{q}$$

$$\underset{\sim}{q}_\tau = \underset{\sim}{q}_\tau + \underline{\alpha} \cdot \Delta \underset{\sim}{q}$$

12. Compute residual load vector as in step 7.

13. Convergence check: if $||\hat{\underset{\sim}{f}}||_2/||\underset{\sim}{P}_\tau||_2 \leq$ TOL, go to the next time step; otherwise proceed as follows.

14. Compute $\delta \underset{\sim}{q} = [\underline{K}_t^*]^{-1} \cdot \hat{\underset{\sim}{f}}$

15. Update the scaling matrix $\underline{\alpha}$. That is

$$\alpha_j = \alpha_j + \Delta q_j/\delta q_j \quad (j = 1, 2, \ldots, n)$$

16. Update the state of motion. That is

$$\ddot{\underset{\sim}{q}}_\tau = \ddot{\underset{\sim}{q}}_\tau + a_1 \, \delta \underset{\sim}{q}$$

$$\dot{\underset{\sim}{q}}_\tau = \dot{\underset{\sim}{q}}_\tau + a_4 \, \delta \underset{\sim}{q}$$

$$\underset{\sim}{q}_\tau = \underset{\sim}{q}_\tau + \delta \underset{\sim}{q}$$

TABLE 3. CONTROL PARAMETERS FOR SOME SOLUTION SCHEMES

Control Parameter	SOLUTION SCHEME						
	1	2	3	4	5	6	7
NSTEP	As appropriate	As appropriate	As appropriate	As appropriate	As appropriate	As appropriate	As appropriate
ITYP	0	0	≥ 1	≥ 1	1	1	0
KPATH	Not needed	Not needed	0 or 1	0 or 1	0 or 1	0 or 1	Not needed
KRUSE	1	1	1	0	1	0	1
MAXCYC	1	1	1	1	1	say, 3	1
MAXIT	1	As appropriate	As appropriate	large	large	say, 5	As appropriate
TOLF	large	small	small	small	small	small	small
TOL	large	small or moderate	small or moderate	small or moderate	large	small or moderate	small or moderate
TOLK	large	= TOLF	large	large	large	large	moderate
NITF	1	As appropriate	As appropriate	As appropriate	= NSTEP	As appropriate	As appropriate
DISLIM	As appropriate	As appropriate	As appropriate	As appropriate	As appropriate	As appropriate	As appropriate
IQUIT	0	1	1	1	0	1	1

TABLE 4. LARGE DISPLACEMENT ELASTIC STATIC RESPONSE OF TRUSS-SPRING PROBLEM - COMPARISON WITH OTHER RESULTS

LOAD	VERTICAL DISPLACEMENT, INCHES				MEMBER FORCES, POUNDS		
P	PRESENT STUDY		STRICKLIN	NOOR	PRESENT STUDY		NOOR
POUNDS	N-R ITER.	CON. STIFF.	[17]	[16]	N-R ITER.	CON. STIFF.	[16]
6	0.2354	0.2352	0.2354	0.2354	-207.6	-207.5	-207.2
12	0.9970	0.9956	1.0000	1.0000	-499.9	-499.9	-495.0
18	1.7646	1.7647	1.765	1.767	-207.6	-207.6	-204.0
24	2.0000	2.0000	2.0000	2.000	0.0	0.0	0.0
30	2.1617	2.1615	2.162	2.162	+174.7	+174.5	+171.0
36	2.2893	2.2893	2.289	2.289	+331.1	+331.1	+323.3
42	2.3961	2.3959	-	-	+474.5	+474.3	-
48	2.4892	no converg.	-	-	+608.8	no converg.	-

TABLE 5. LARGE DISPLACEMENT INELASTIC STATIC RESPONSE OF TWO BAY TRUSS--COMPARISON WITH OTHER RESULTS, AT P = 10.0 KIPS

(a) MEMBER FORCES, KIPS

MEMBER	PRESENT STUDY	GOLDBERG [18]*	NOOR [16]**
1	+11.27	+11.26	+11.28
2	+ 6.19	+ 6.18	+ 6.19
3	- 6.30	- 6.31	- 6.31
4	-11.20	-11.19	-11.21
5	- 0.50	- 0.58	- 0.51
6	+ 3.34	+ 3.27	+ 3.33
7	+ 6.92	+ 7.04	+ 6.94
8	- 5.57	- 5.46	- 5.56
9	- 4.16	- 4.22	- 4.17
10	+ 4.46	+ 4.37	+ 4.45

(b) DISPLACEMENT W, INCHES

-	1.0511	0.923	1.0574

* DISPLACEMENT METHOD

** MIXED (FORCE AND DISPLACEMENT) METHOD

TABLE 6. SPHERICAL CAP - COMPARISON OF COMPUTATIONAL EFFICIENCY

Procedure	No. of Stiffness Formulations	No. of State Determinations	Average No. of Iterations/Step
(1) Step-by-step with equilibrium correction, Load step 1 lb.	100	100	No iteration
(2) Step-by-step with equilibrium correction, Load step 5 lbs.	20	20	No iteration
(3) Newton-Raphson iteration, Load step 5 lbs.	83	83	4.2

TABLE 7. CANTILEVER BEAM - COMPARISON OF COMPUTATIONAL EFFICIENCY

Procedure	No. of Stiffness Formulations	No. of State Determinations	Average No. of Iterations/Step
(2) with $\Delta t = \Delta t_1$	300	300	No iteration
*(2) with $\Delta t = \Delta t_2$	100	100	No iteration
(3) with $\Delta t = \Delta t_1$ classified "exact"	300	753	2.5
(3) with $\Delta t = \Delta t_2$	100	776	7.8
(4) with $\Delta t = \Delta t_2$ "Automatic" Stiffness formulation	71	529	5.3

* Procedure gives inaccurate response

TABLE 8 : DYNAMIC RESPONSE OF 8-STORY SHEAR BUILDING--COMPARISON OF VARIOUS SOLUTION STRATEGIES

STORY	RESULTS FROM REFERENCE [31]					RESULTS FROM PRESENT STUDY							
	Δt = .0025	Δt = .005		Δt = .01		Δt = .005		Δt = .02		Δt = .02		Δt = .02	
	DUCTILITY	DUCT.	ERROR	DUCT.	ERROR	DUCT.	ERROR	DUCT.	ERROR	DUCT.	ERROR	DUCT.	ERROR
8	2.87	2.72	-5.23	2.38	-17.07	2.95	+ 2.79	2.68	- 6.62	2.56	-10.80	2.95	+ 2.09
7	4.46	4.56	+2.24	4.78	+ 7.17	4.27	- 4.26	3.84	-13.90	3.43	-23.09	3.08	-30.94
6	2.86	2.95	+3.15	2.79	- 2.45	3.21	+12.24	4.47	+56.29	2.08	-27.27	2.24	-21.68
5	3.50	3.46	-1.14	3.26	- 6.86	3.64	+ 4.00	2.00	-42.86	2.26	-35.43	2.26	-35.43
4	2.75	2.88	+4.73	2.78	+ 1.09	2.81	+ 2.18	1.84	-33.09	2.28	-17.09	2.44	-11.27
3	3.13	3.17	+1.28	3.02	- 3.51	3.08	- 1.60	2.49	-20.45	2.54	-18.85	2.51	-19.81
2	3.86	3.83	-0.78	3.77	- 2.33	3.96	+ 2.59	4.43	+14.77	3.73	- 3.37	3.79	- 1.81
1	4.20	4.15	-1.19	4.09	- 2.62	4.19	- 0.24	4.95	+17.86	3.91	- 6.90	3.84	- 8.57
No. of Stiffness Formulations	131	125		112		1		85		1		1	
No. of State Determinations	2400	1200		600		2043		300		1161		455	
Average No. of Iterations/step	-	-		-		1.7		-		3.9		1.5	
Solution Strategy	(a)	(a)		(a)		(b)		(a)		(b)		(c)	

Error = [(Ductility) $_{\Delta t}$ - (Ductility) $_{.0025}$] x 100 / (Ductility) $_{.0025}$

(a) Step-by-step, with equilibrium correction, no iteration.

(b) Step-by-step, with equilibrium iteration using initial elastic stiffness. Convergence tolerance 0.1 kip.

(c) Same as (b), but Convergence Tolerance 0.5 kip.

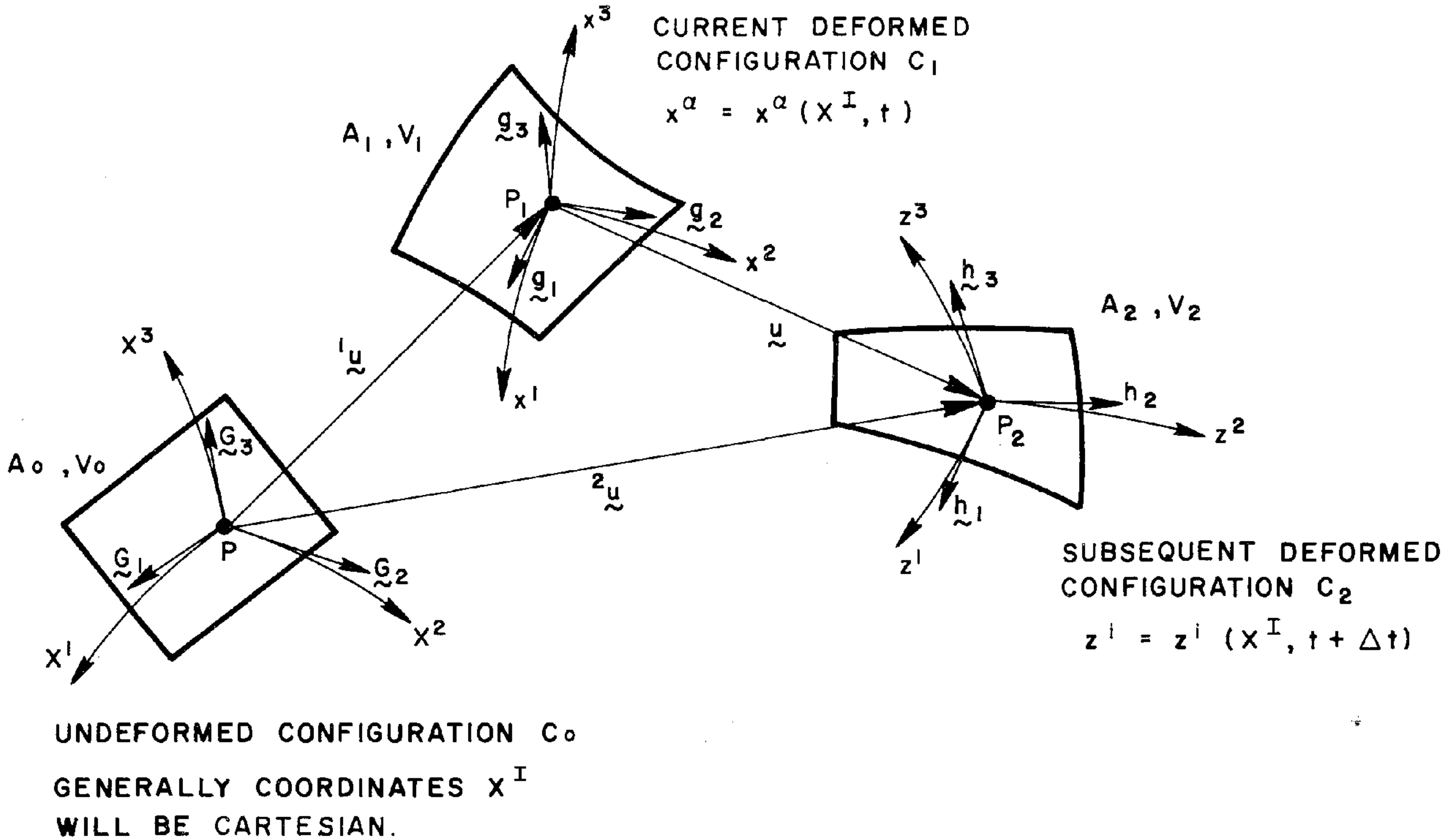


FIG. 1 DEFORMATION MAPPINGS OF THE BODY

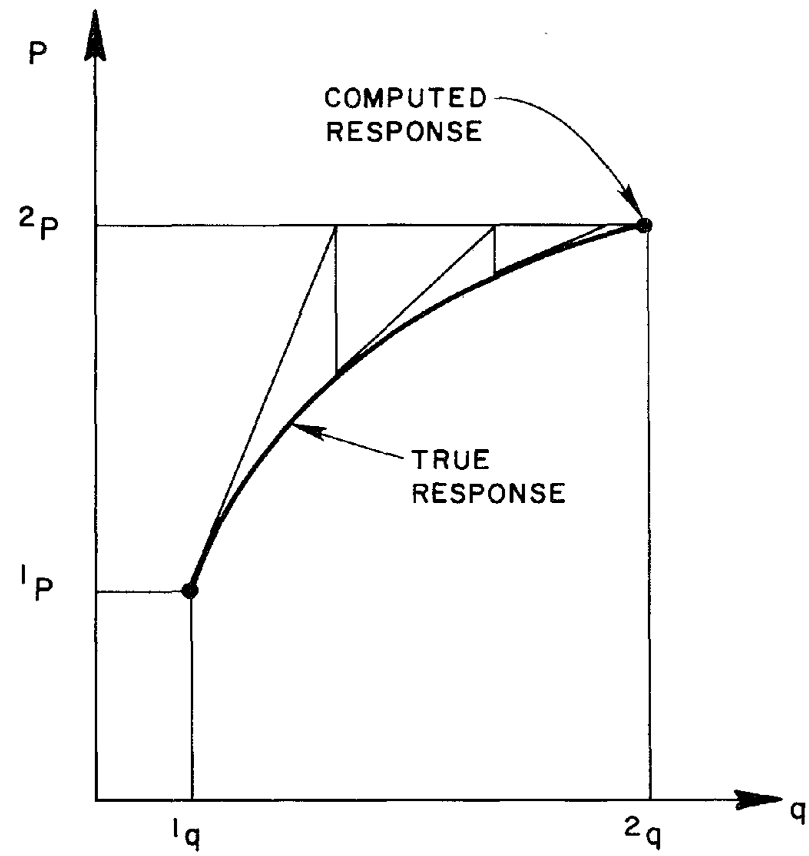
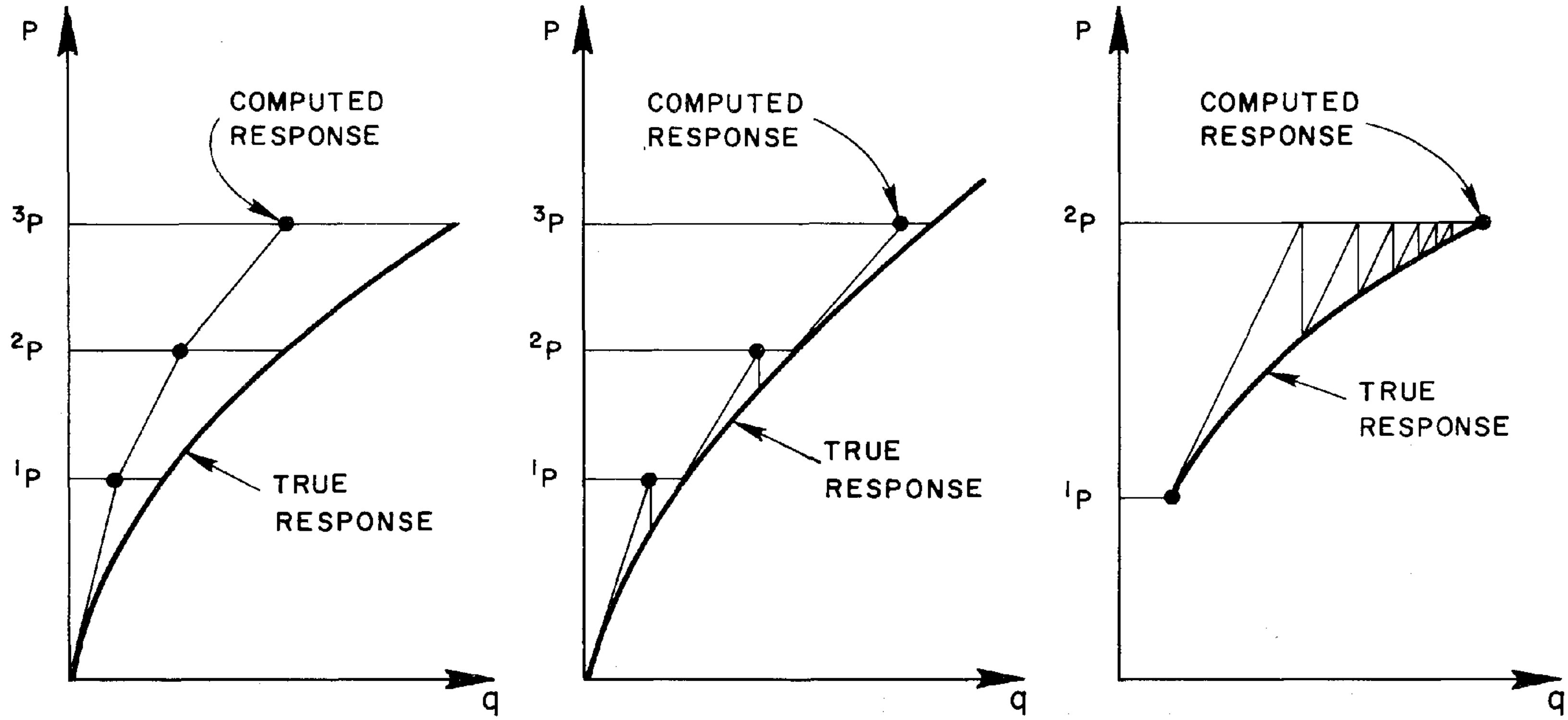


FIG. 2 NEWTON-RAPHSON ITERATION PROCEDURE



(a) STEP-BY-STEP WITHOUT EQUILIBRIUM CORRECTION

(b) STEP-BY-STEP WITH EQUILIBRIUM CORRECTION

(c) CONSTANT STIFFNESS ITERATION

FIG. 3 STEP-BY-STEP AND CONSTANT STIFFNESS ITERATION PROCEDURES

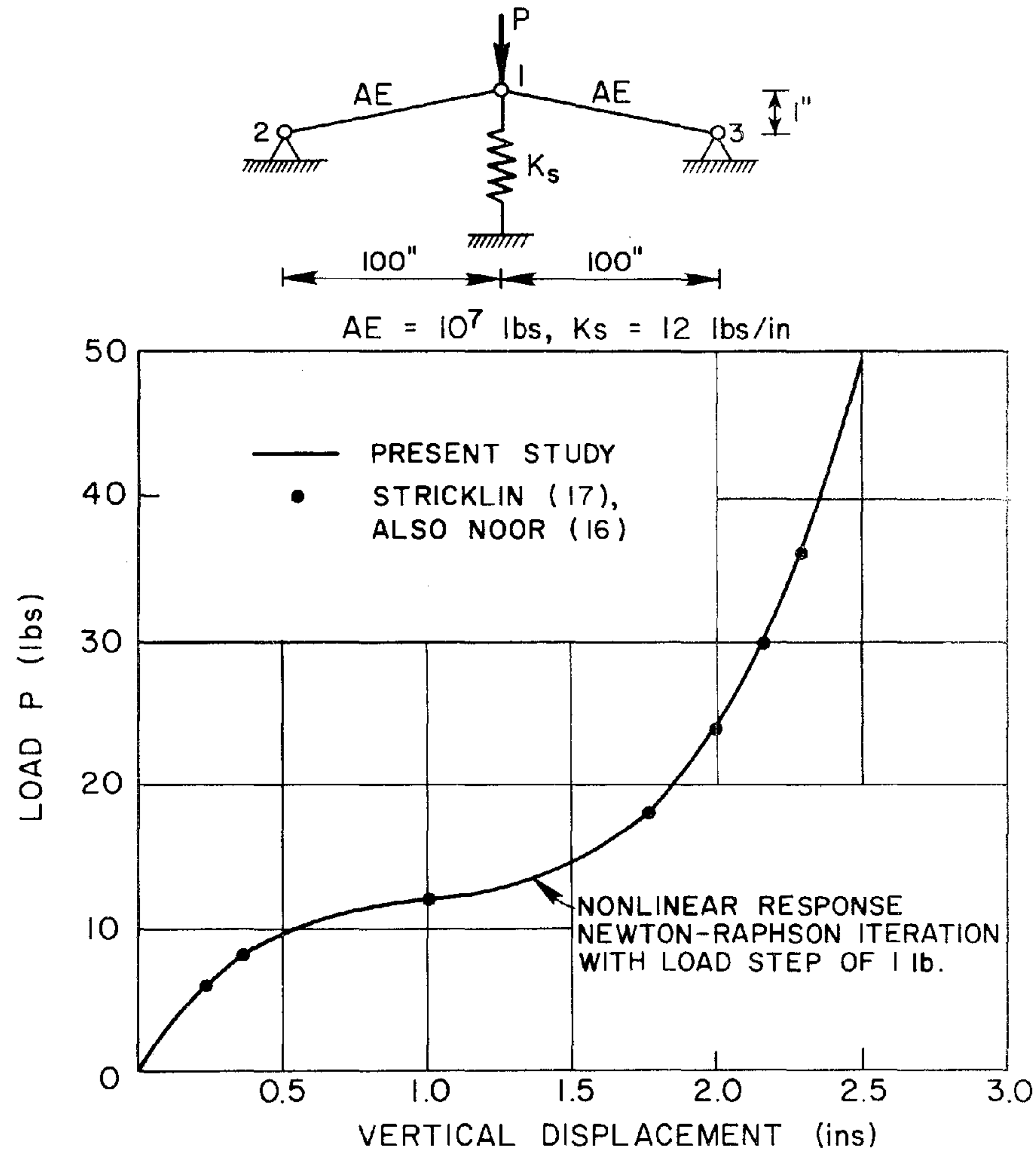
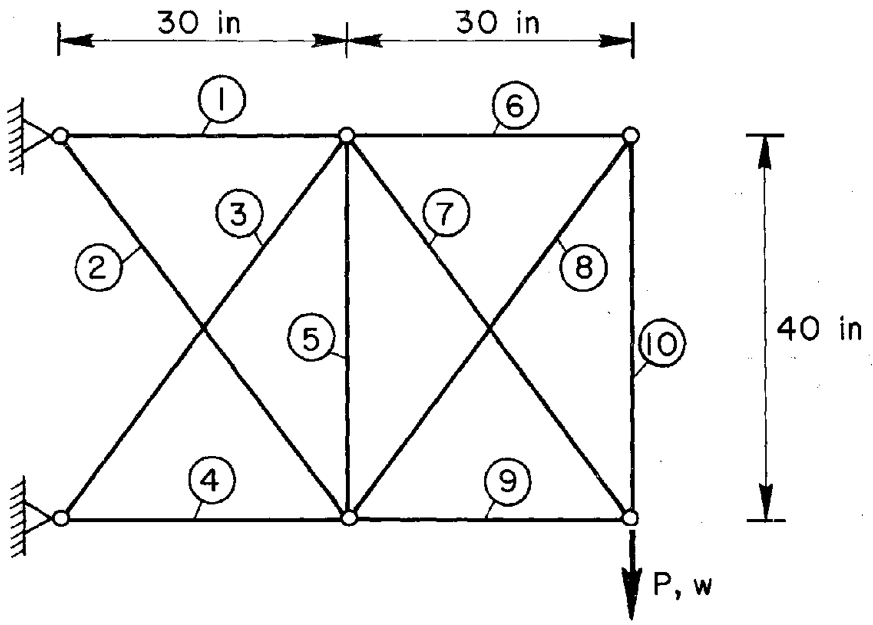


FIG. 4 LOAD-DISPLACEMENT RESPONSE FOR TRUSS-SPRING PROBLEM



Member	Area
Horizontals	0.25 in^2
Verticals and diagonals	0.20 in^2

Stress-Strain Law (Ramberg-Osgood Curve)

$$\frac{\varepsilon}{\varepsilon_o} = \frac{\sigma}{\sigma_o} \left[1 + k \left(\frac{\sigma}{\sigma_o}\right)^{n-1}\right]$$

σ_o = 40.52 ksi $\quad$ k = 3/7

ε_o = (0.4052) 10^{-2} in/in $\quad$ n = 7

E = 10000 ksi

ρ = (2.5) 10^{-4} lb. sec^2/in^4

To = Fundamental Period ≈ 6200 μsecs.

FIG. 5 PLANE TRUSS

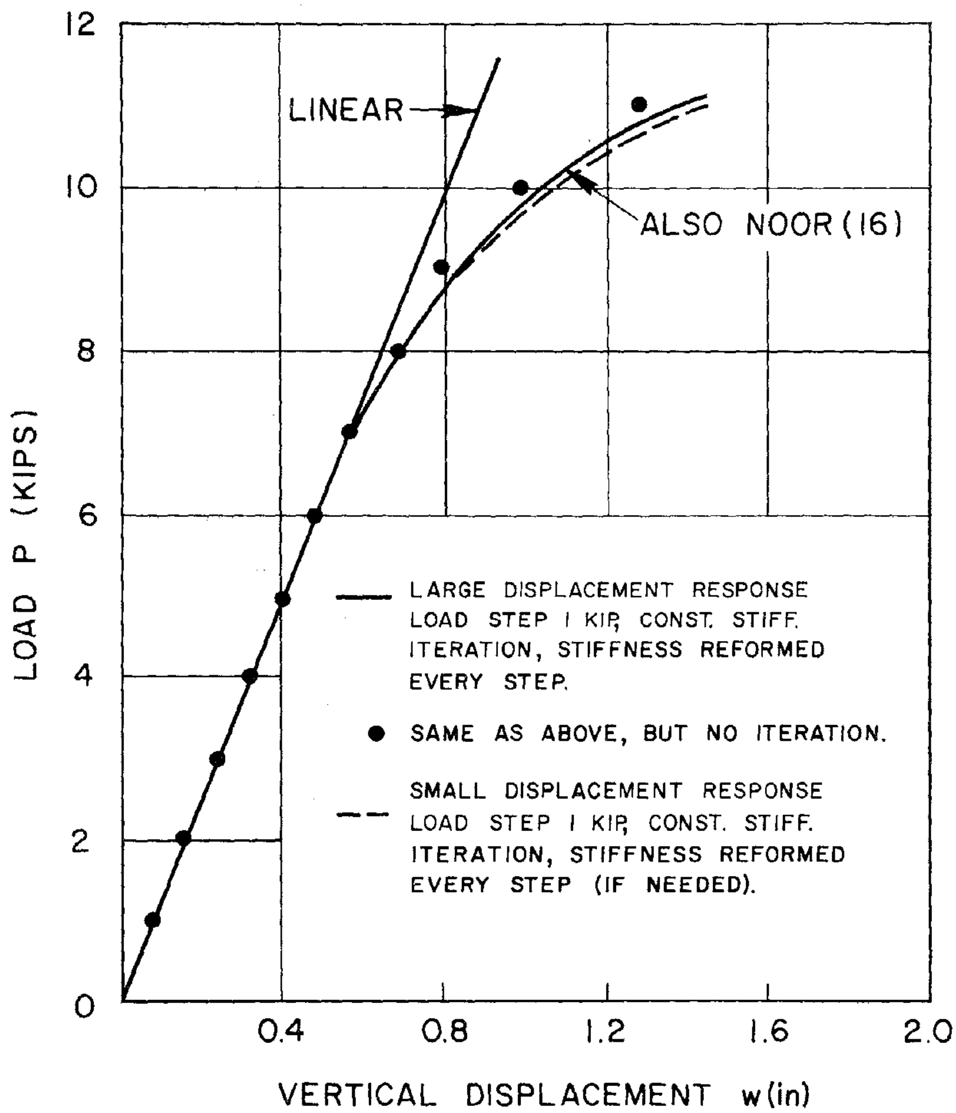


FIG. 6 STATIC RESPONSE OF PLANE TRUSS

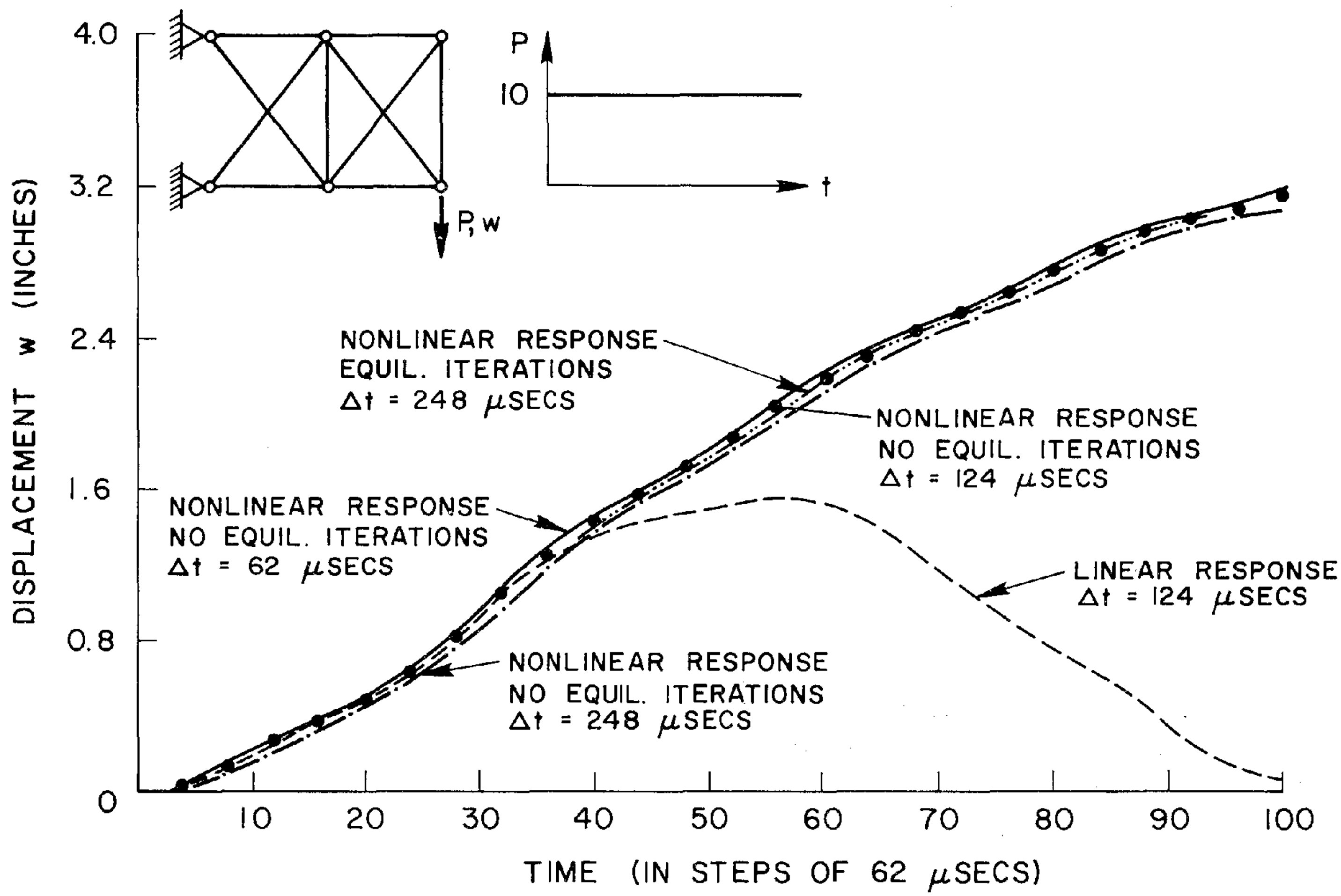
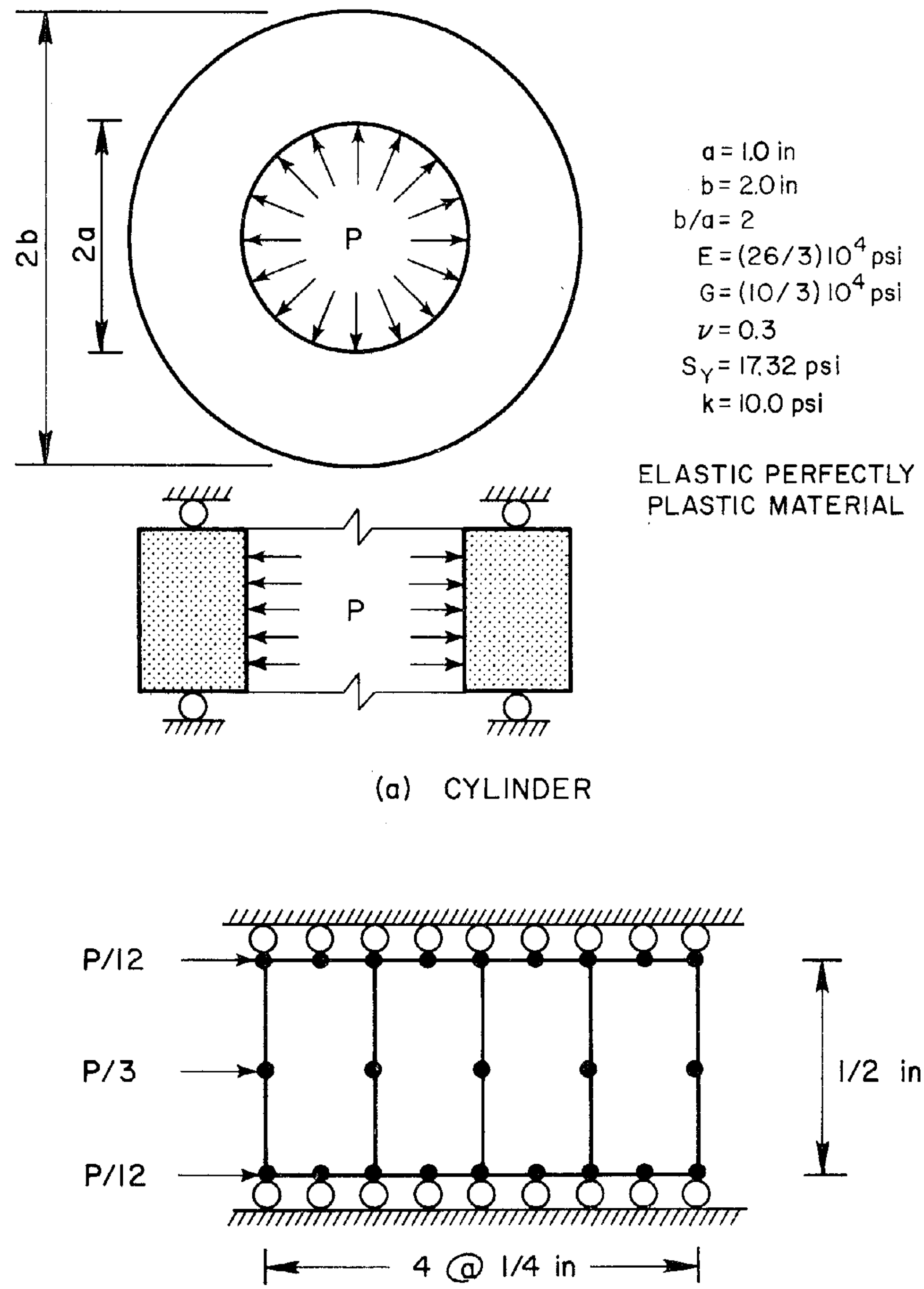


FIG. 7 DYNAMIC RESPONSE OF INELASTIC PLANE TRUSS



(a) CYLINDER

(b) FINITE ELEMENT MODEL

FIG. 8 AXIALLY RESTRAINED THICK CYLINDER

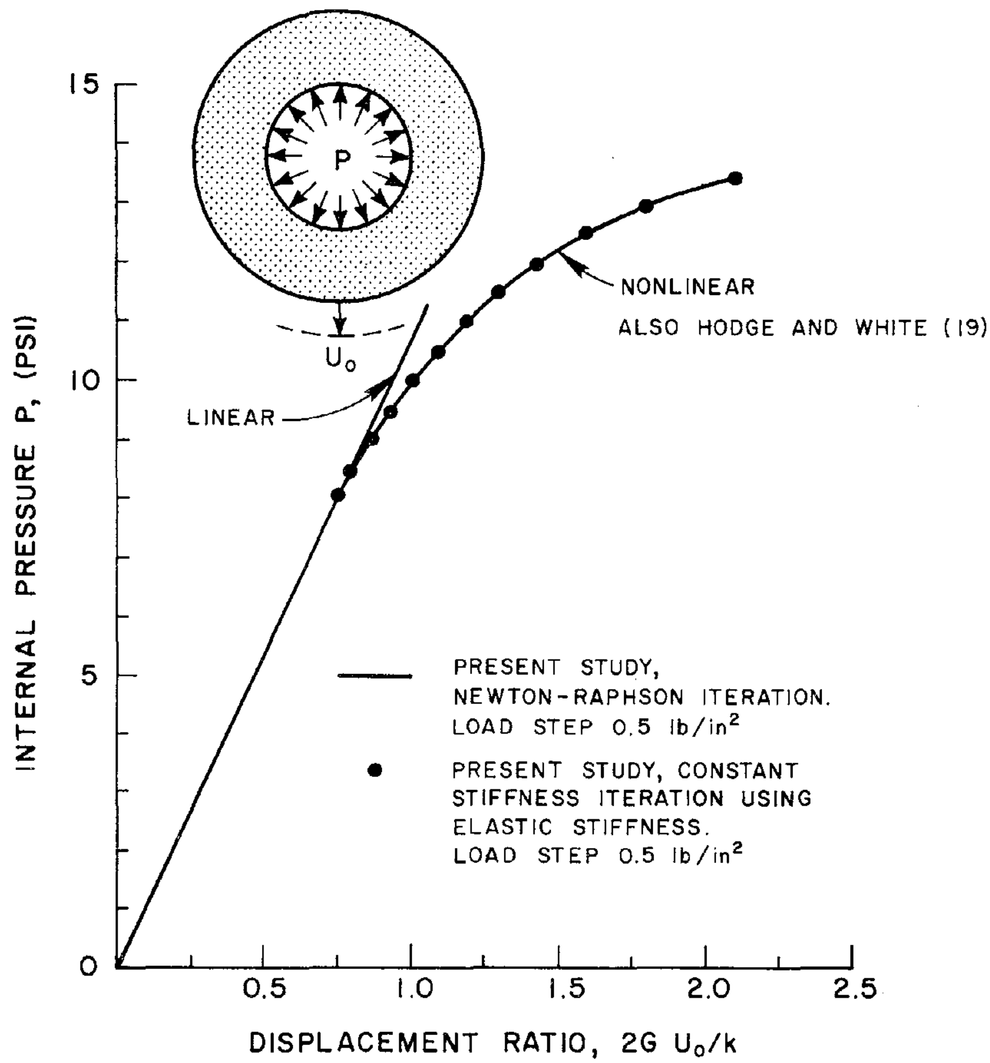


FIG. 9 SMALL DISPLACEMENT ELASTO-PLASTIC RESPONSE OF THICK CYLINDER, OUTER RADIAL DISPLACEMENT.

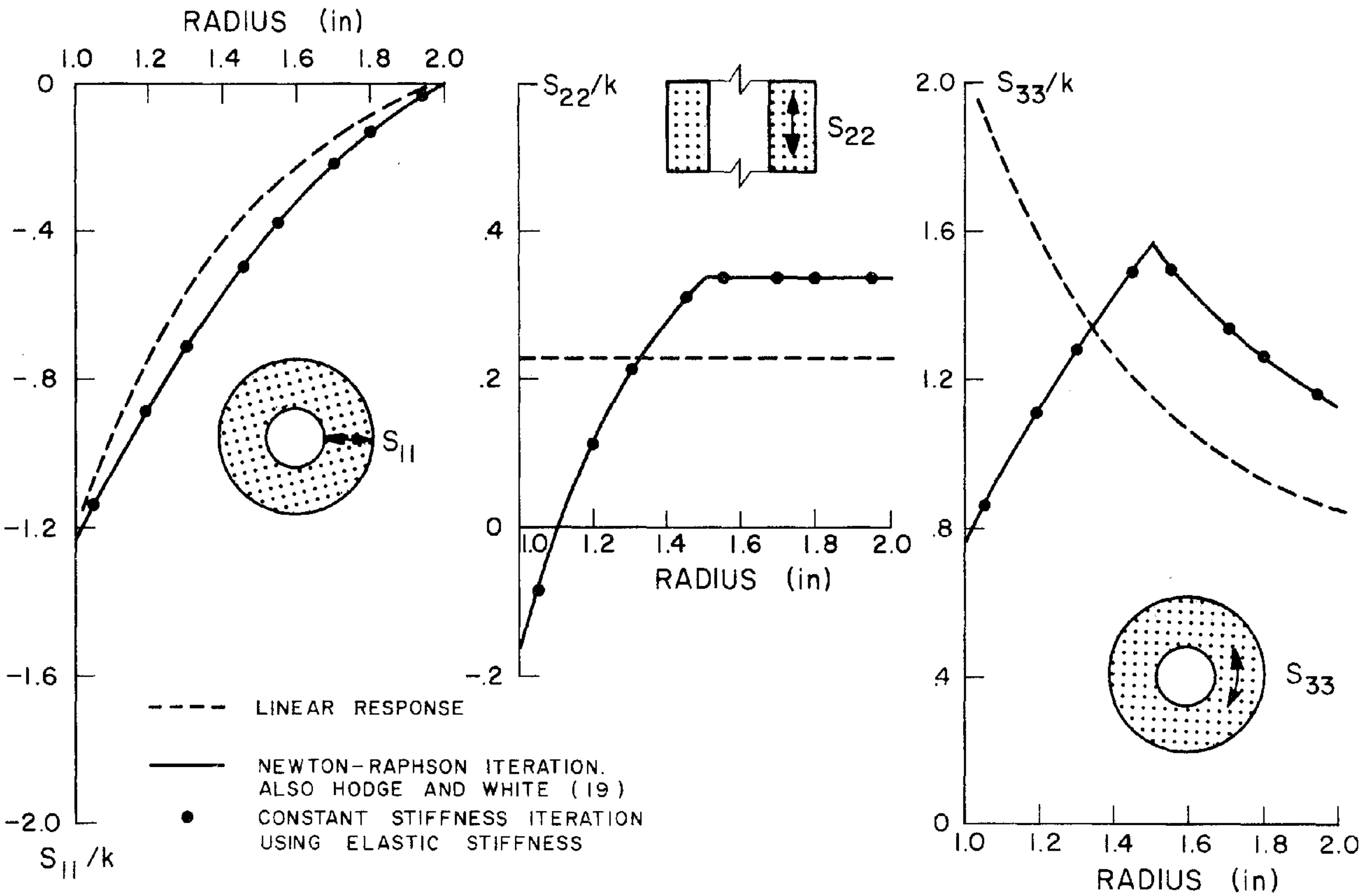


FIG. 10 SMALL DISPLACEMENT ELASTO-PLASTIC RESPONSE OF THICK CYLINDER, STRESSES AT PRESSURE 12.5 PSI

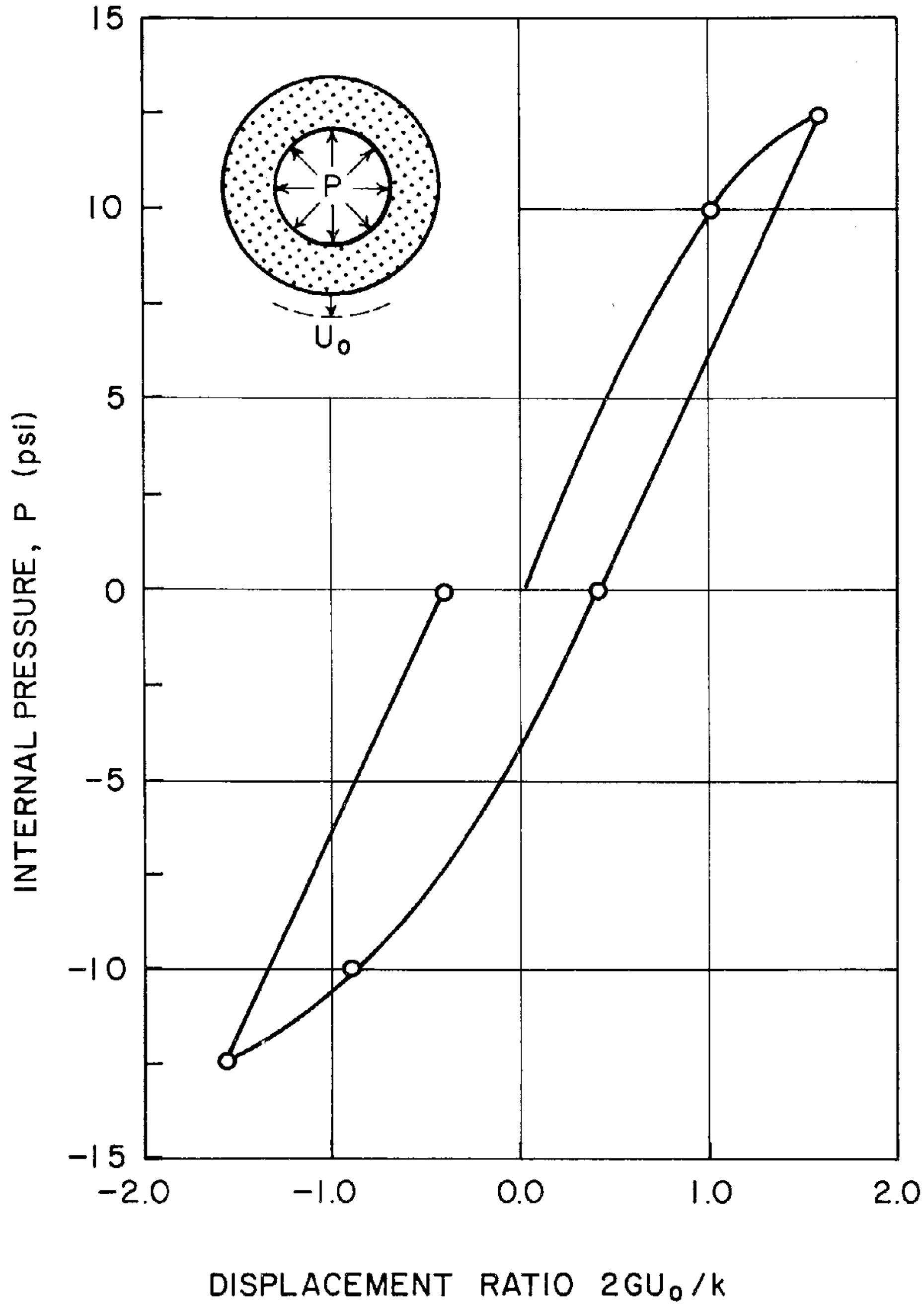


FIG. 11 SMALL DISPLACEMENT ELASTO-PLASTIC RESPONSE OF THICK CYLINDER WITH LOAD REVERSAL

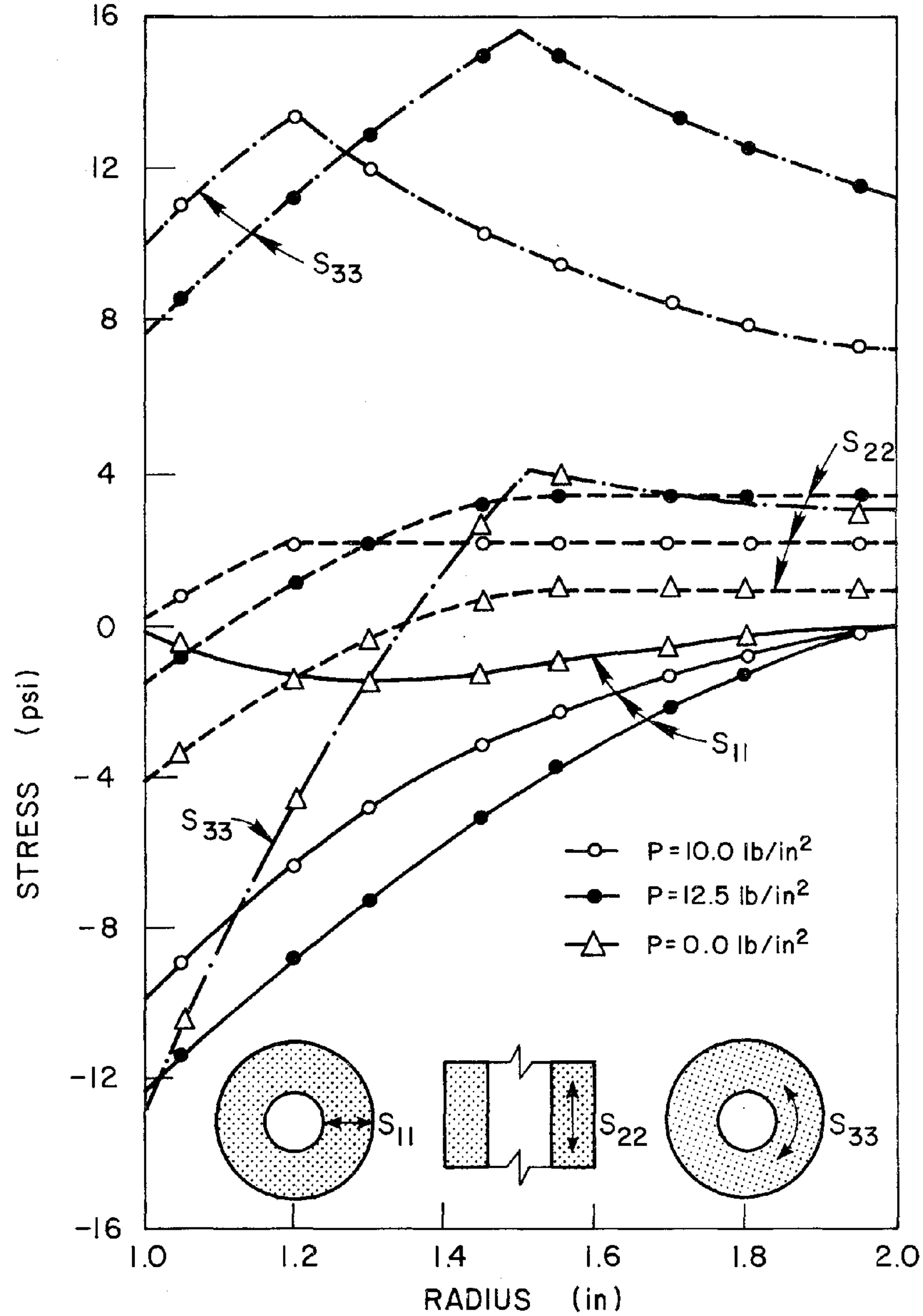


FIG. 12 SMALL DISPLACEMENT ELASTO-PLASTIC RESPONSE OF THICK CYLINDER, STRESSES WITH LOAD REVERSAL

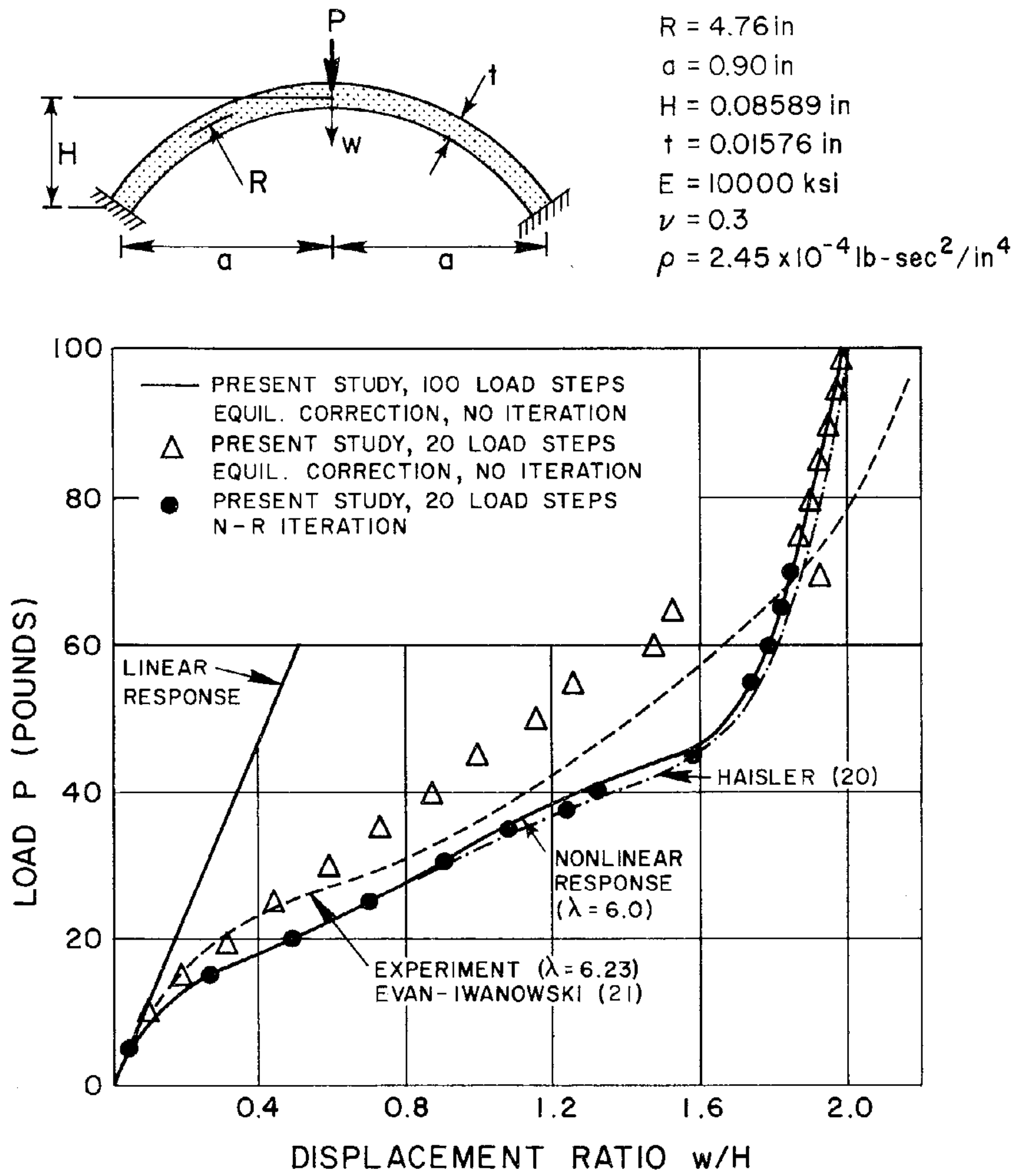


FIG. 13 LARGE DISPLACEMENT ELASTIC STATIC RESPONSE OF SHALLOW CAP

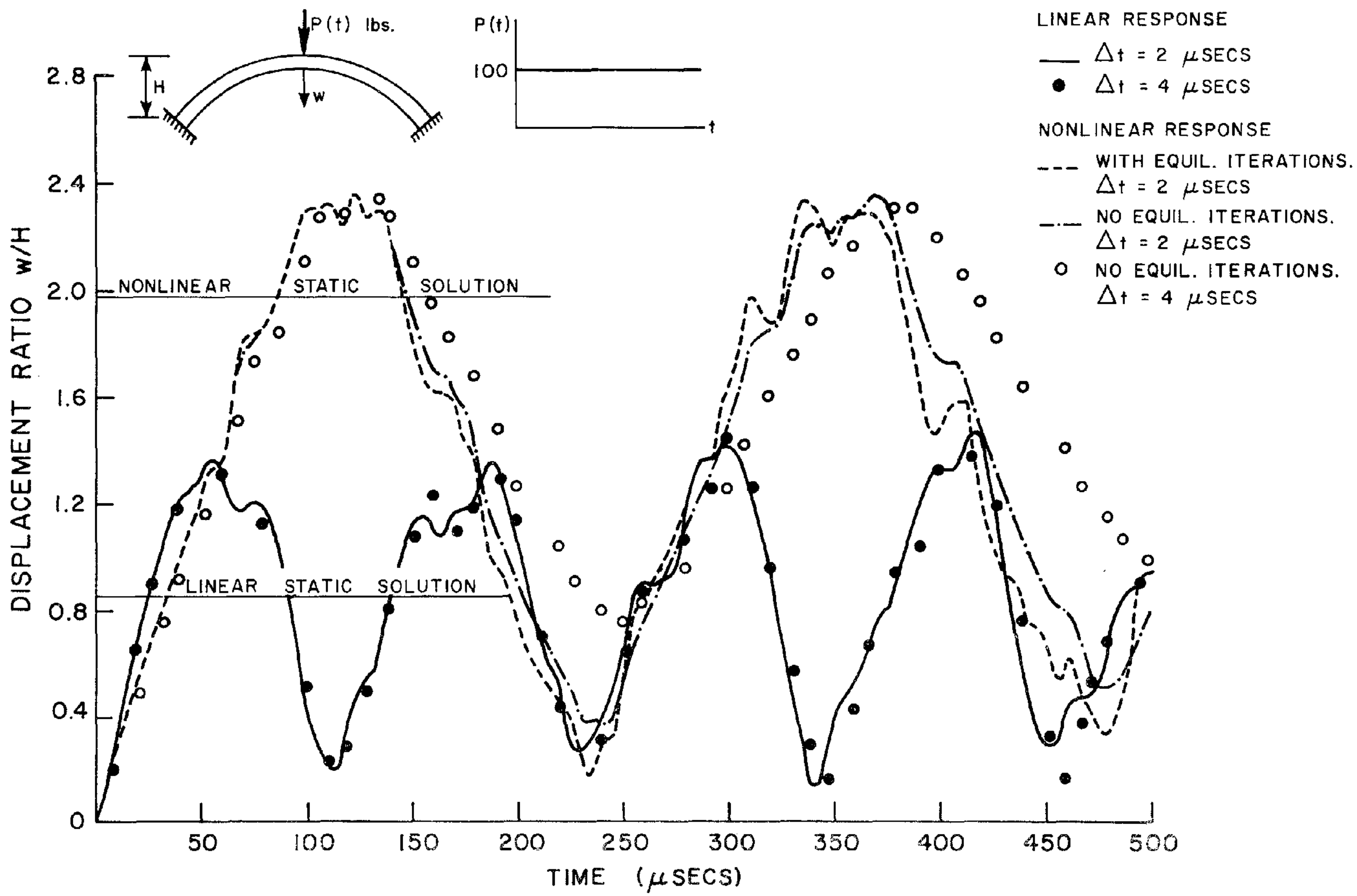


FIG. 14 LARGE DISPLACEMENT ELASTIC DYNAMIC RESPONSE OF SHALLOW CAP

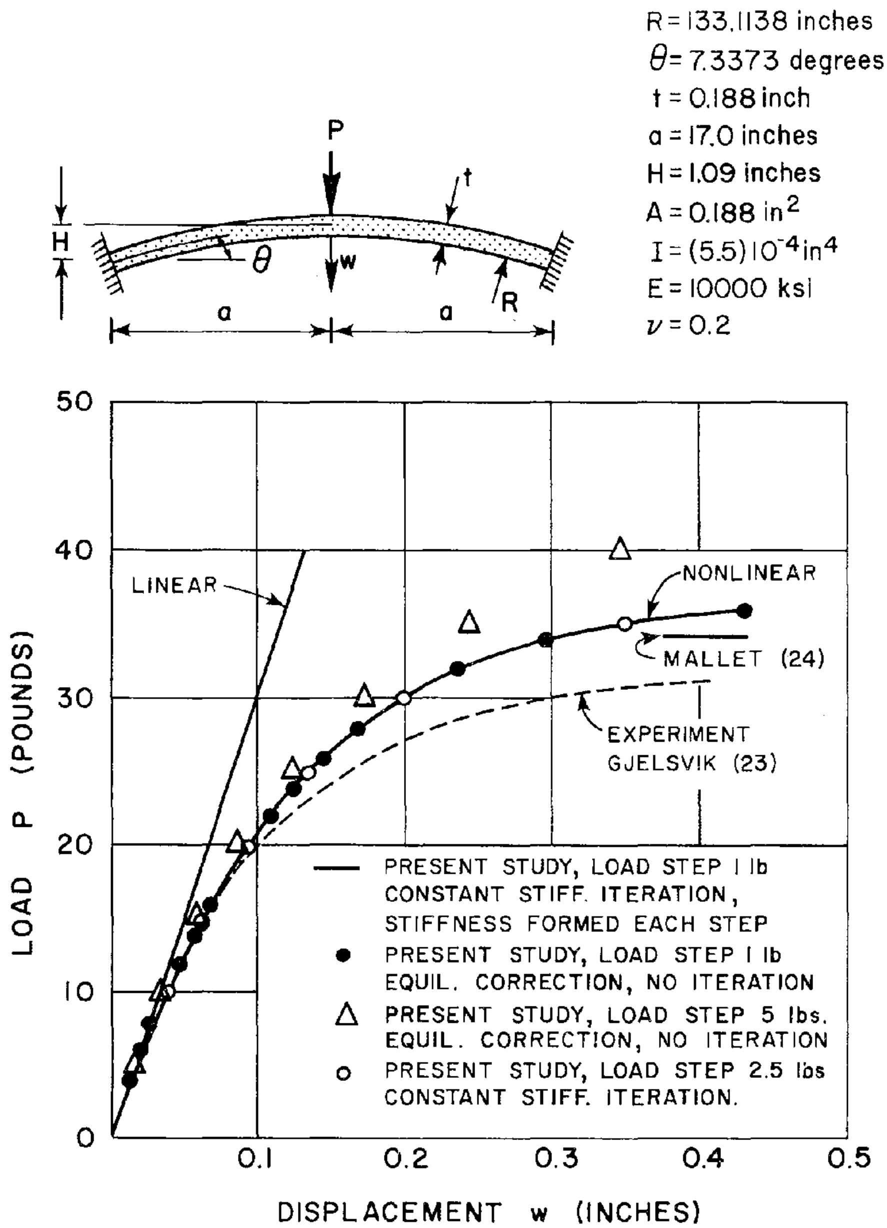
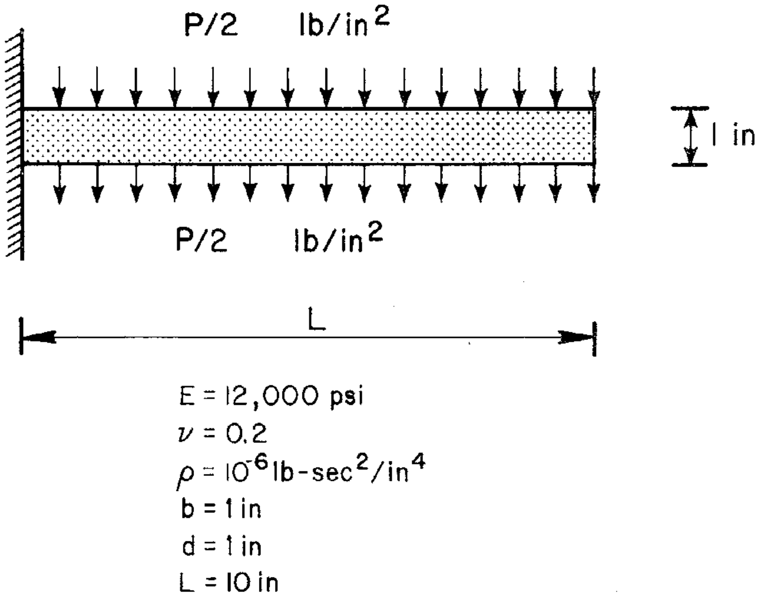
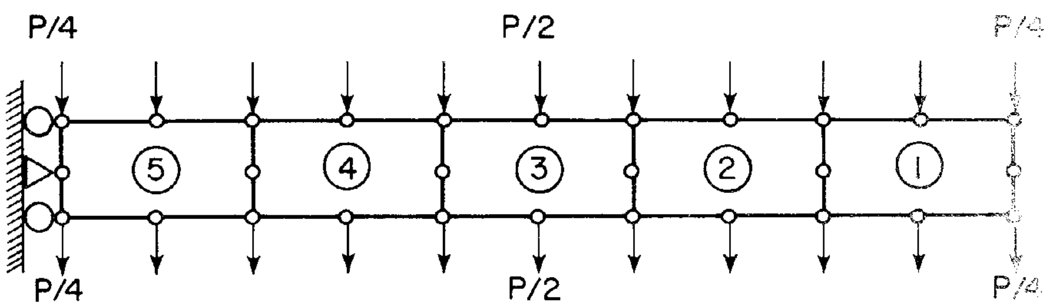


FIG. 15 LARGE DISPLACEMENT ELASTIC STATIC RESPONSE OF SHALLOW ARCH.



(a) BEAM PROPERTIES



(b) FINITE ELEMENT MODEL

FIG. 16 CANTILEVER BEAM

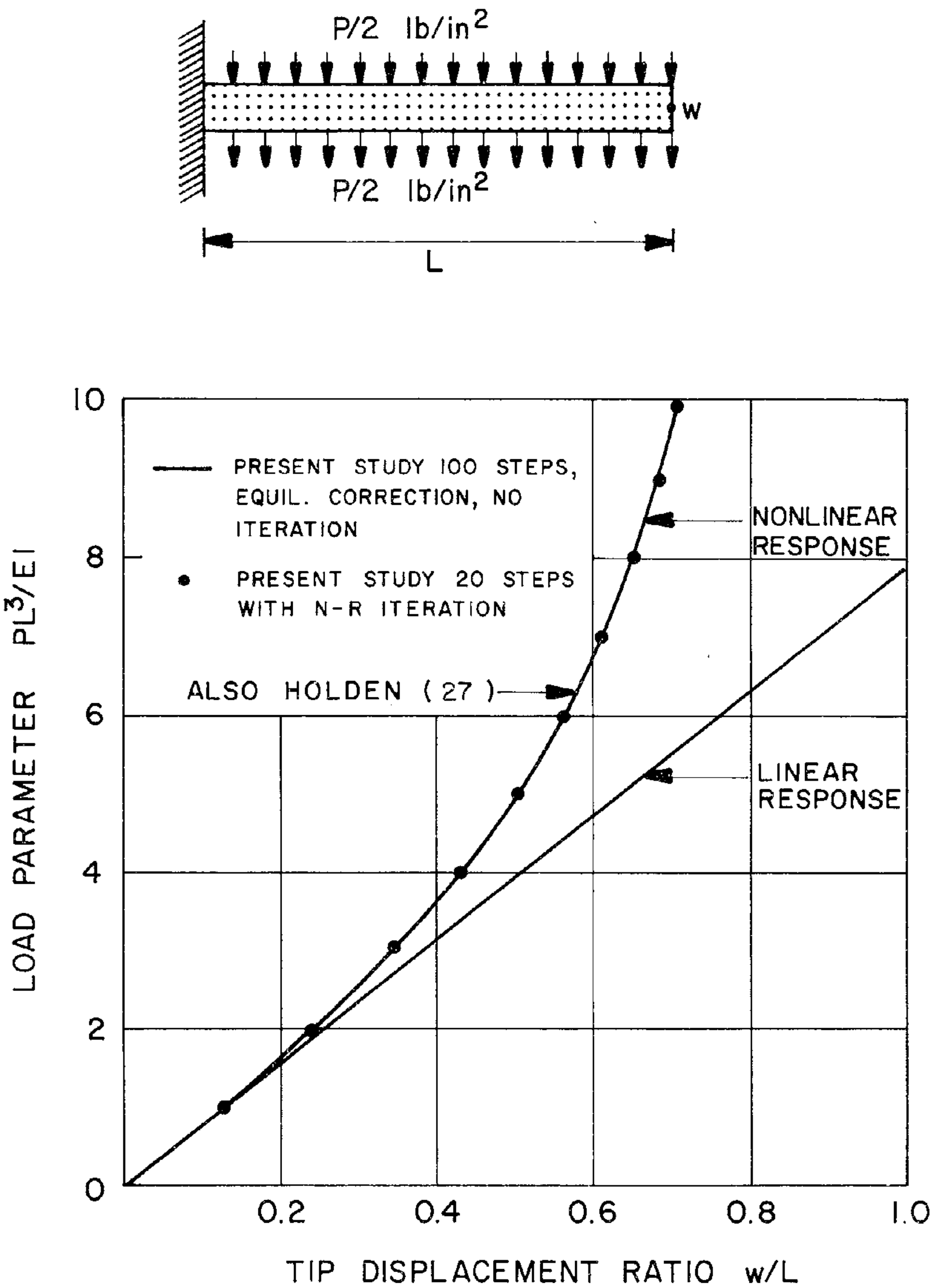


FIG. 17 LARGE DISPLACEMENT ELASTIC STATIC RESPONSE OF CANTILEVER BEAM

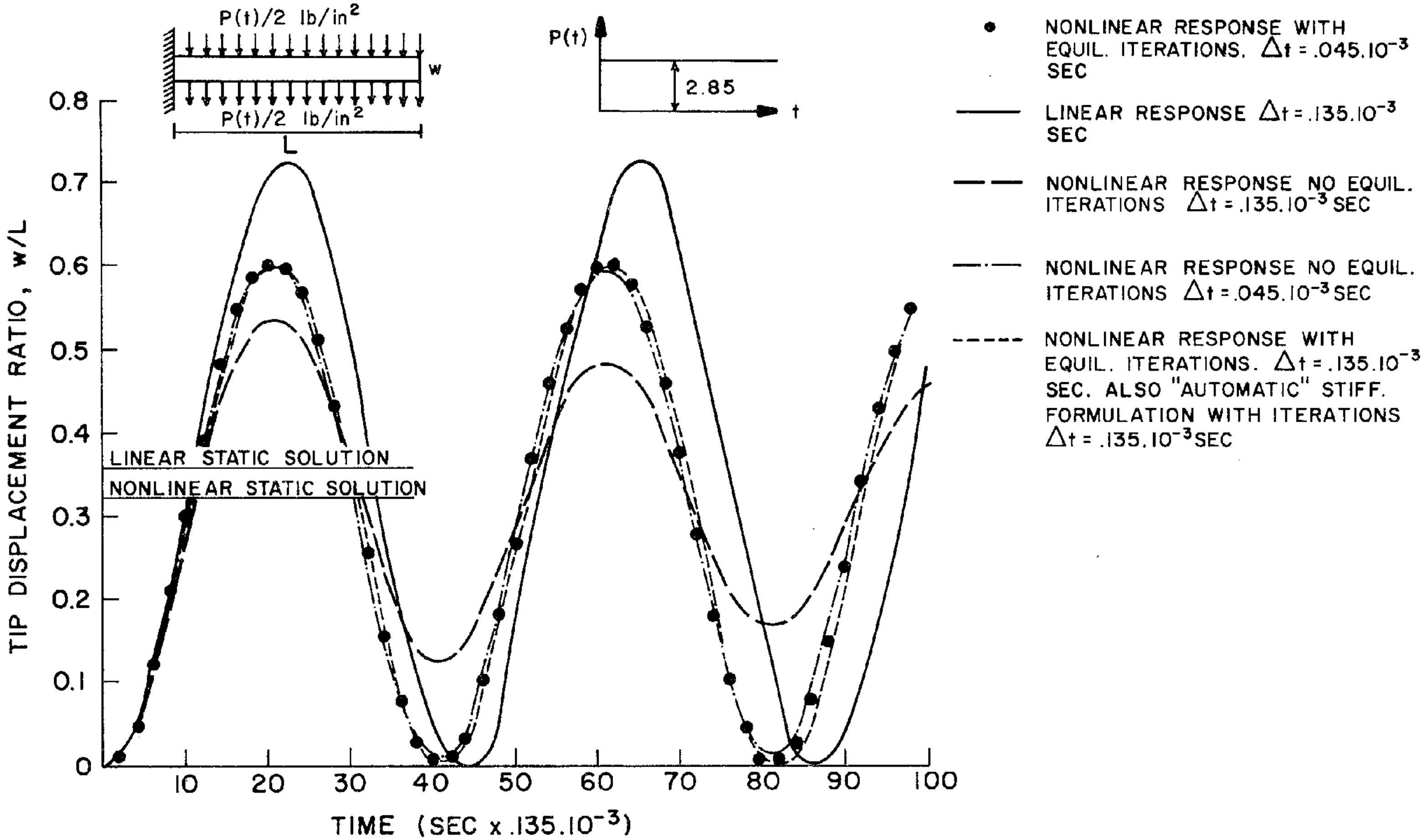


FIG. 18 LARGE DISPLACEMENT ELASTIC DYNAMIC RESPONSE OF CANTILEVER BEAM

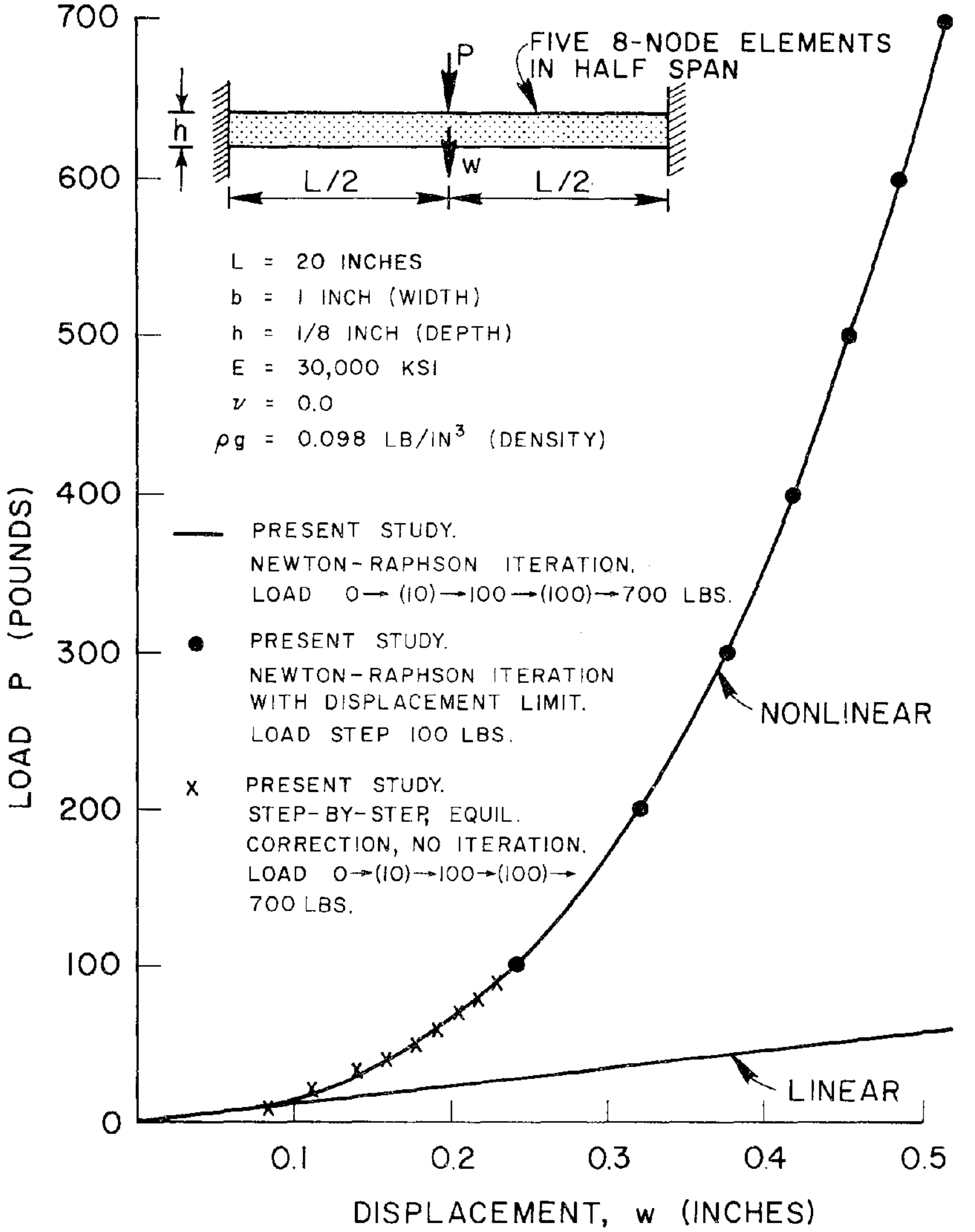


FIG. 19 LARGE DISPLACEMENT ELASTIC STATIC RESPONSE OF CLAMPED BEAM

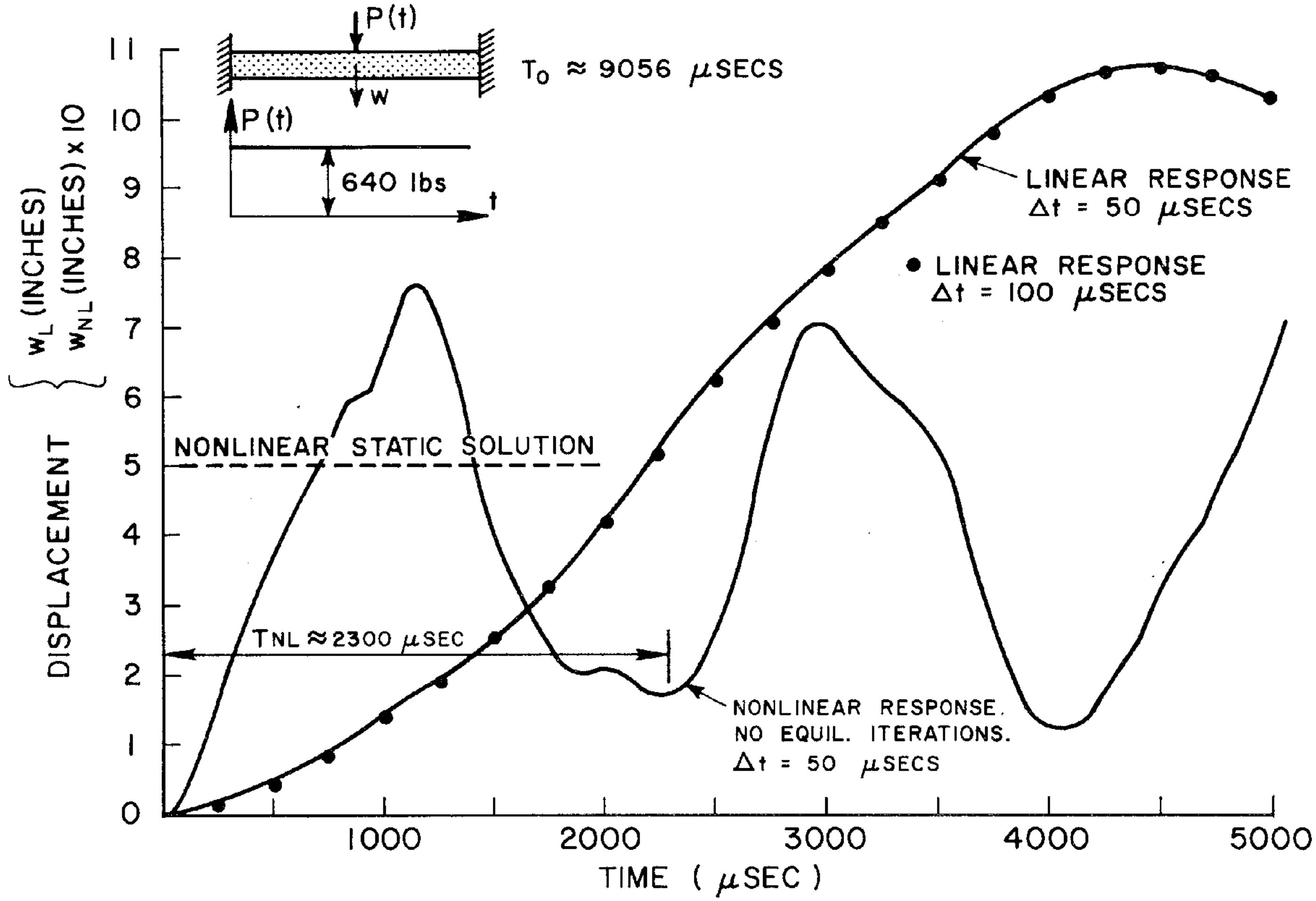


FIG. 20 COMPARISON OF LINEAR AND NONLINEAR DYNAMIC RESPONSES FOR CLAMPED BEAM

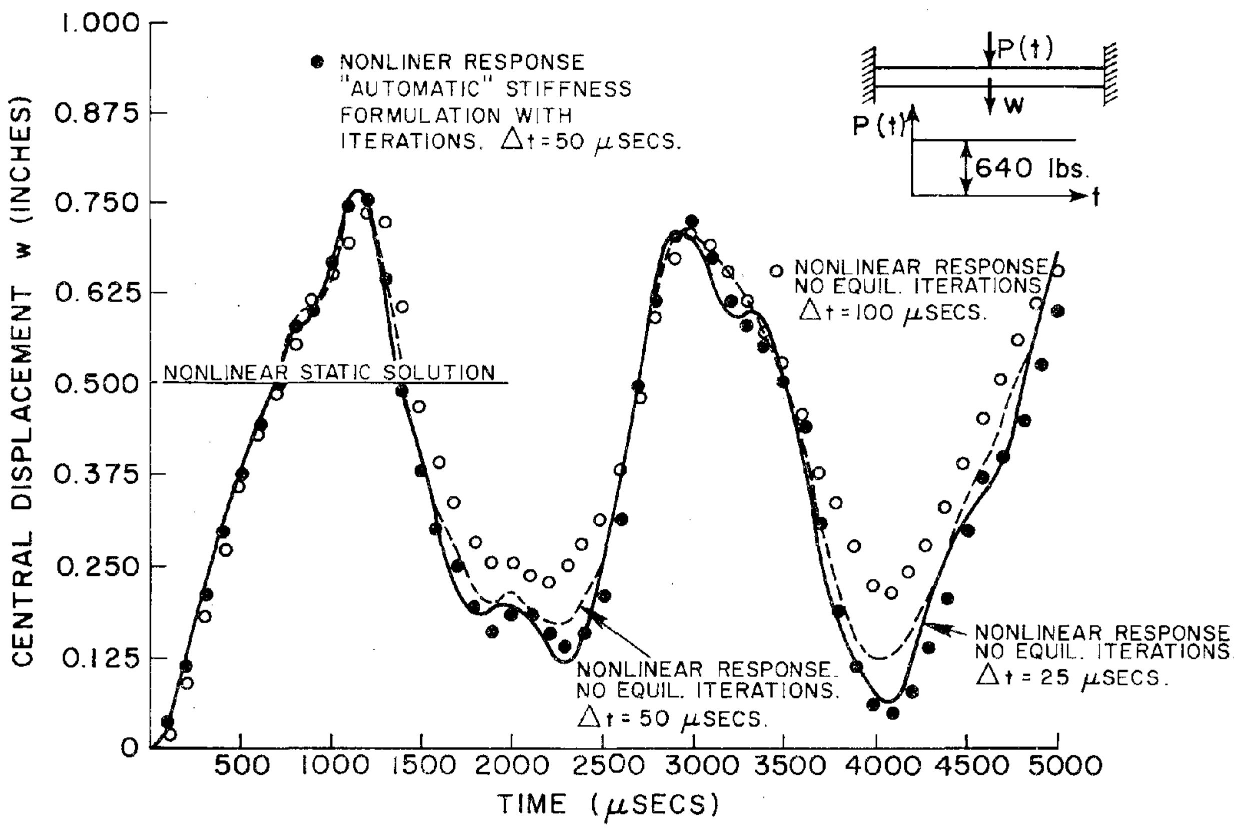
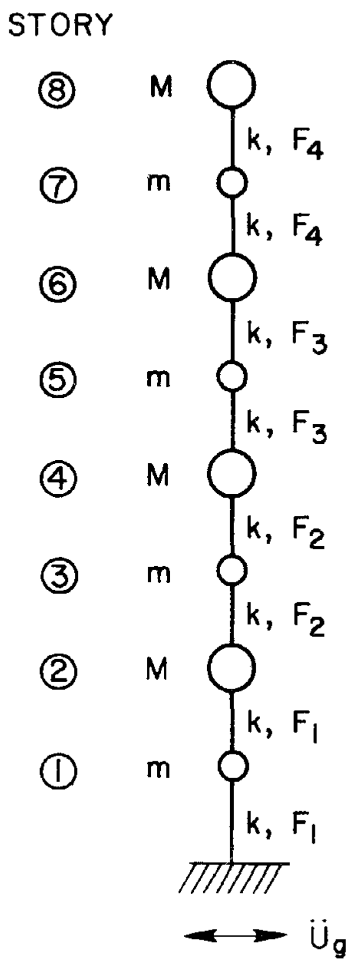
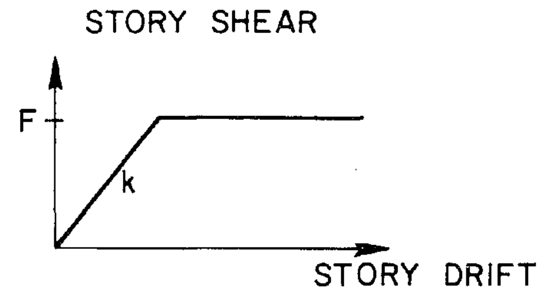


FIG. 21 LARGE DISPLACEMENT ELASTIC DYNAMIC RESPONSE OF CLAMPED BEAM





$M = 0.1196$ kip-sec^2/inch

$m = M/100$

$k = 219.34$ kips/inch

$F_1 = 69.09$ kips

$F_2 = 60.32$ kips

$F_3 = 47.38$ kips

$F_4 = 27.20$ kips

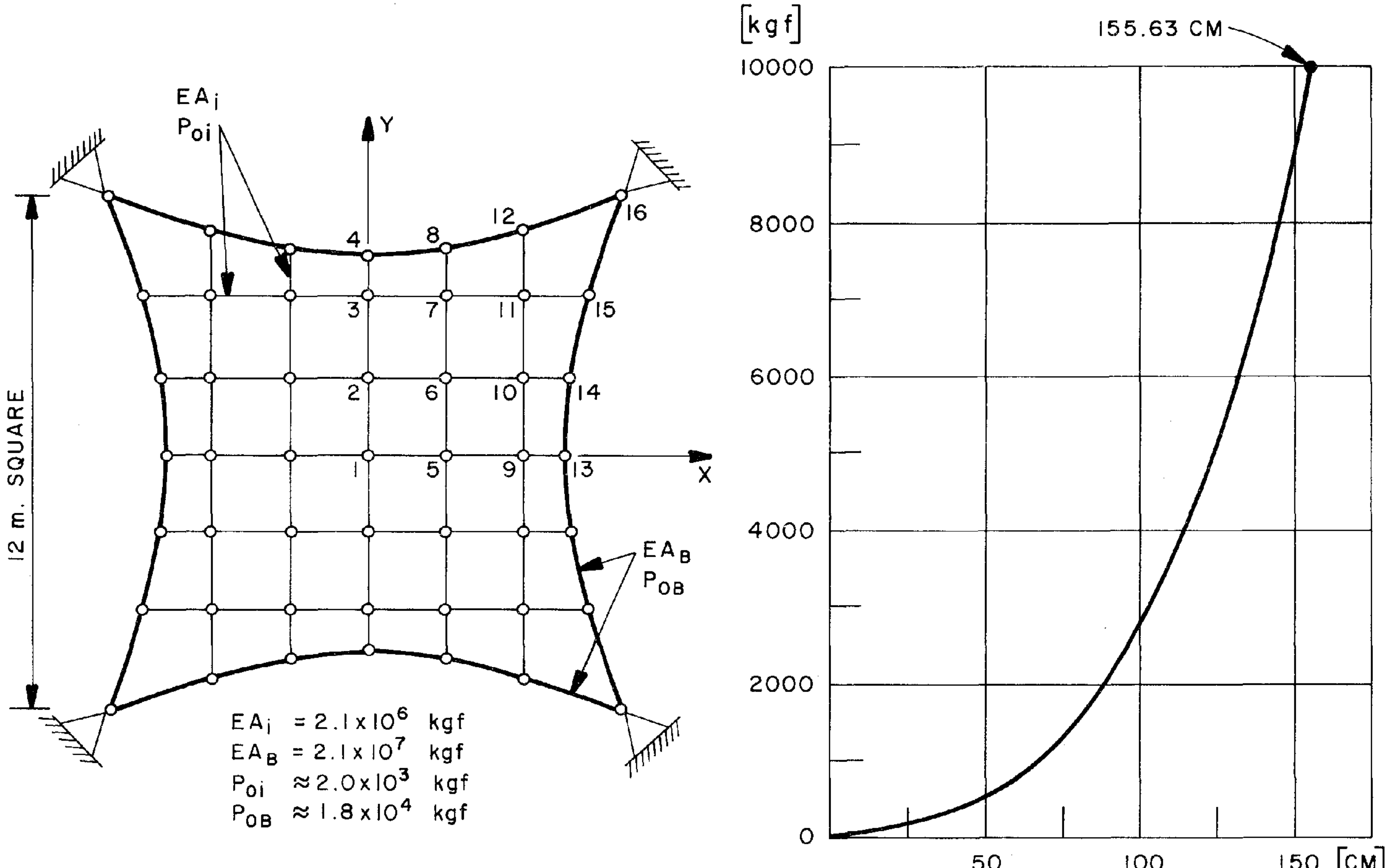
Fundamental Period: $T_o = 0.6$ sec

Damping: $c = \beta k$

(i.e. proportional to elastic stiffness)

β corresponds to 0.5% at T_o

FIG. 22 EIGHT-STORY SHEAR BUILDING



(a) GEOMETRY OF CABLE NET

(b) VERTICAL DISPLACEMENT AT NODE 1

FIG. 23 STATIC RESPONSE OF A PLANE PRESTRESSED CABLE NET

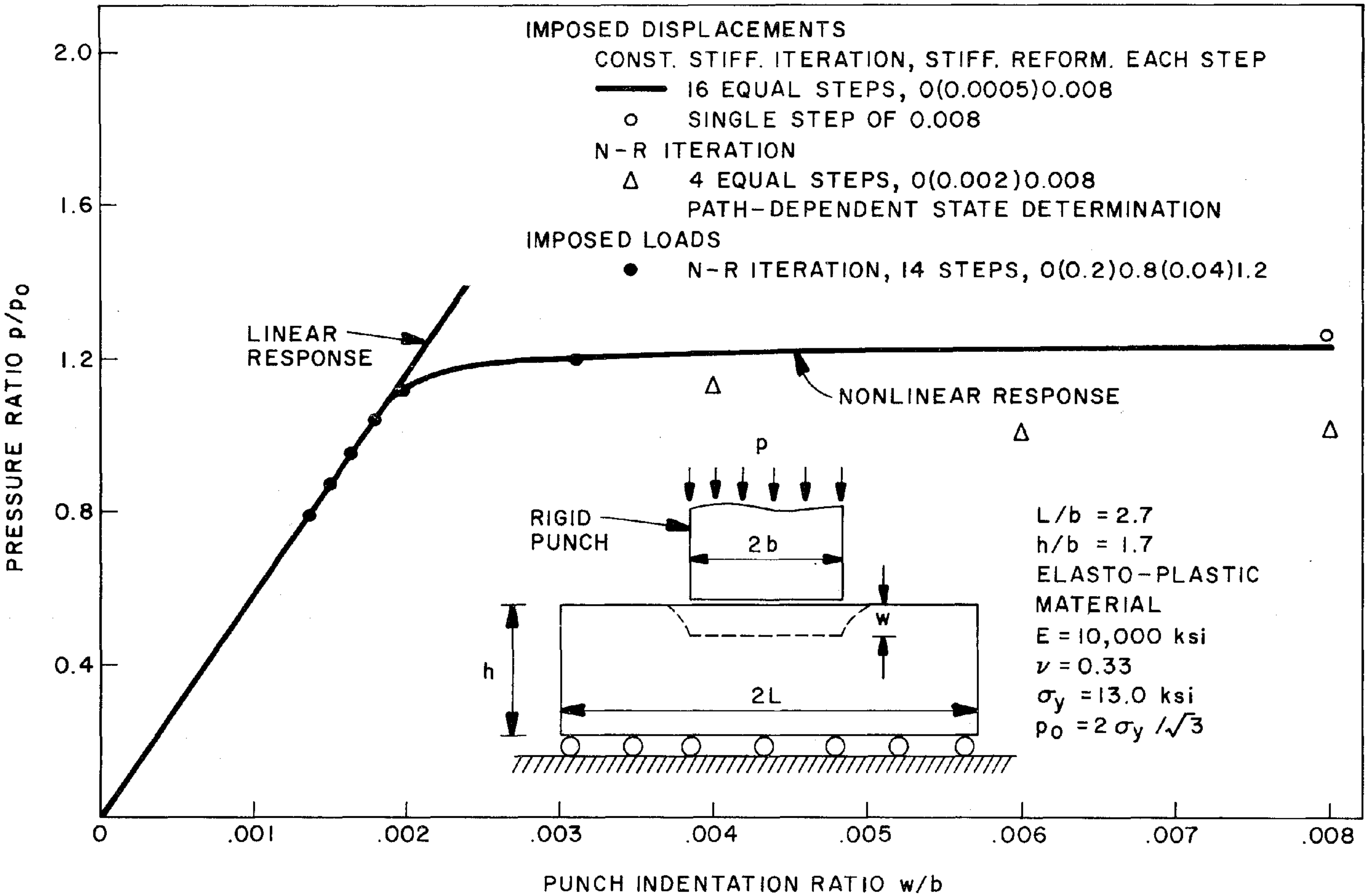


FIG. 24 STATIC RESPONSE OF PLANE STRAIN PUNCH PROBLEM

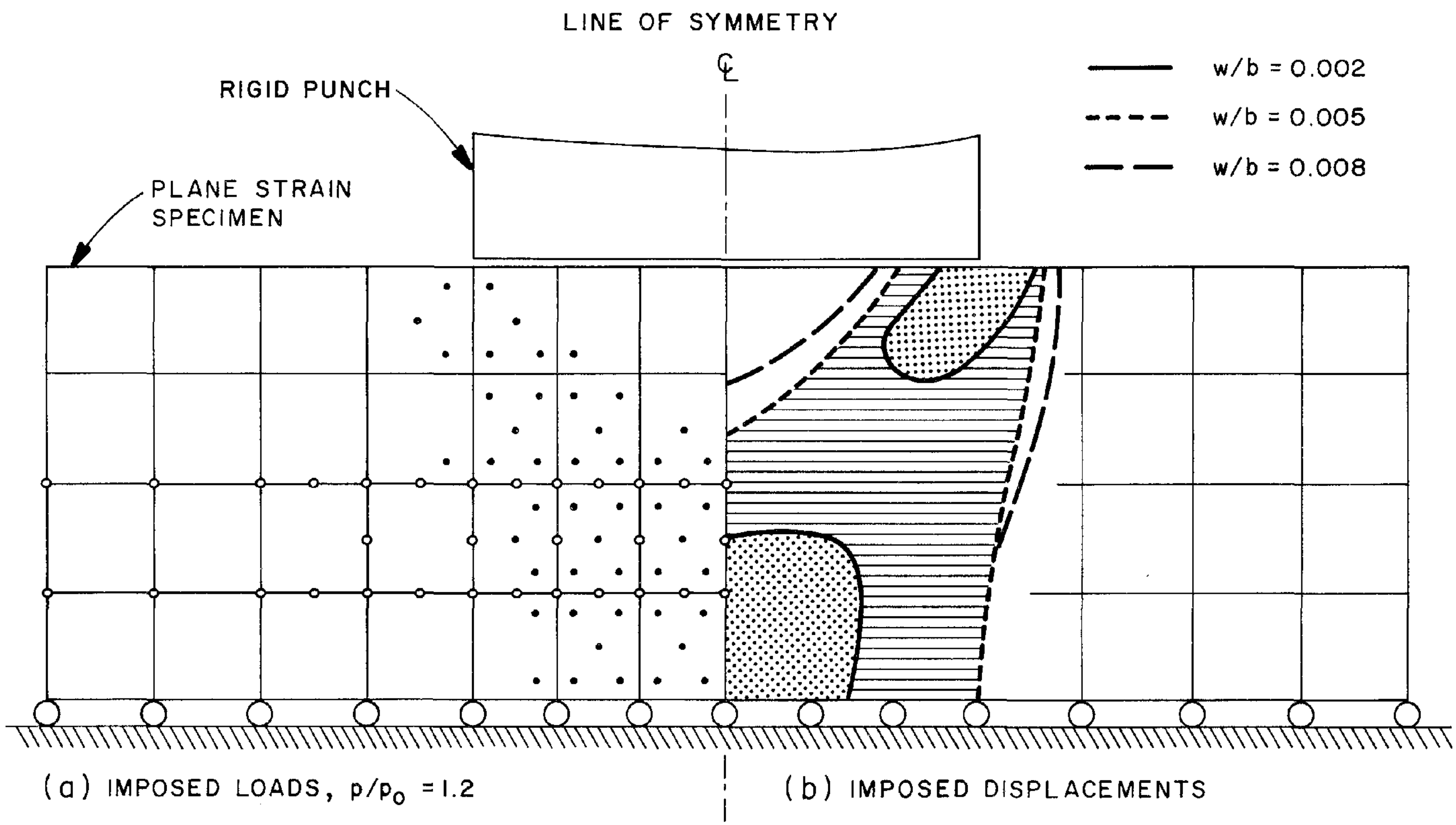


FIG. 25 PLASTIC ZONE PROFILES FOR PLANE STRAIN PUNCH PROBLEM

APPENDIX A

PROGRAM USER'S GUIDE

University of California
Berkeley

Division of Structural Engineering
and Structural Mechanics

Computer Programming Series

PROGRAM IDENTIFICATION

ANSR-I: General Purpose Computer Program for Analysis of Nonlinear Structures. Version I, December 1975.

Developed by: D. P. Mondkar and G. H. Powell,
University of California, Berkeley

CONTENTS

A. PROBLEM INITIATION AND TITLE (A5, 3X, 18A4) - ONE CARD

Columns	1 - 5:	Punch the word START
	6 - 8:	Blank
	9 - 80:	Problem title, to be printed with output.

B. NODE INFORMATION

B1. CONTROL INFORMATION (9I5) - One card

Columns	1 - 5:	Total number of nodes.
	6 - 10:	Number of "control" nodes, for which coordinates are specified directly (NCNOD). See section B2.
	11 - 15:	Number of coordinate generation commands (NODGC). See section B3.
	16 - 20:	Number of commands specifying nodes with zero displacements (NDCON). See section B4.
	21 - 25:	Number of commands specifying nodes with equal displacements (NIDDOF). See section B5.
	26 - 30:	Number of commands specifying nodal masses (NMSGC). See section B6.
	31 - 35:	Number of element groups (NELGR, max. 20). See section E.
	40:	Execution code (KEXEC) as follows. (a) zero or blank: full execution. (b) 1: data checking only (c)-1: full execution, but only if the structure stiffness and element data can be held in core.
	45:	Stiffness storage code (KSCHM), as follows. (a) zero or blank: duplicate stiffness matrix held in core. (b) 1: duplicate stiffness matrix stored on scratch file.

B2. CONTROL NODE COORDINATES (I5, 3F10.0) - NCNOD cards

Columns	1 - 5:	Node number, in any sequence.
	6 - 15:	X coordinate.
	16 - 25:	Y coordinate.
	26 - 35:	Z coordinate.

B3. COORDINATE GENERATION (4I5, F10.0, 10I5) - NODGC cards

Columns 1 - 5: Node number at beginning of generation line. This must either be a control node, or must have been generated by a previous generation command.

6 - 10: Node number at end of generation line. This node must also have been specified previously.

11 - 15: Number of nodes to be generated along line. If the nodes to be generated are listed in Columns 31 - 80, this number may not exceed 10.

16 - 20: Node number difference between successive generated nodes, and between first generated node and node at beginning of generation line. May be negative. Leave blank if generated nodes are listed in Columns 31 - 80.

21 - 30: Spacing between nodes, as follows.

(a) zero or blank: generated nodes are spaced uniformly along the generation line.

(b) less than 1.0: spacing between nodes is this proportion of the length of the generation line.

(c) 1.0 or larger: spacing between nodes is equal to this distance.

31 - 80: Up to 10 fields, each I5. List nodes to be generated, in sequence along generation line. Required only if Columns 16 - 20 are blank.

Note: It is not necessary to provide coordinate generation commands for nodes which are sequentially numbered between the beginning and end nodes of any straight line, and which are equally spaced along that line. After all generation commands have been executed, the coordinates for each group of unspecified nodes are automatically generated assuming sequential numbering and equal spacing along a line joining the specified nodes immediately preceding and following the group. That is, any generation command with a node number difference of one and equal spacing is superfluous.

B4. NODES WITH ZERO DISPLACEMENTS (I5, 4X, 6I1, 13I5) - NDCON cards

Columns	1 - 5:	Node number, or number of first node in a series of nodes covered by this command. See Note following for repetition of nodes.
	10:	Constraint code for X displacement, as follows. (a) zero or blank: displacement, not contrained to be zero. (b) 1: displacement constrained to be zero.
	11:	Code for Y displacement.
	12:	Code for Z displacement.
	13:	Code for XX rotation.
	14:	Code for YY rotation.
	15:	Code for ZZ rotation.
	16 - 20:	Number of last node in series of nodes covered by this command. Leave blank or punch zero for a single code, or if the nodes in the series are listed in Columns 31 - 80.
	21 - 25:	Node number difference between successive nodes in series. Leave blank for a single node, or if the nodes in the series are listed in Columns 31 - 80.
	29 - 30:	Number of nodes listed in Columns 31 - 80, following. This list is considered only if Columns 16 - 20 are blank or zero. Leave blank for a single node.
	31 - 80:	Up to 10 fields, each I5. List second, etc. nodes of series.

Note: If constraint codes are specified more than once for any node, the last specified value is assumed. For plane or axisymmetric problems, the first command should cover all nodes and should constrain all except the relevant displacements. Additional cards to modify the constraint codes at particular nodes should then be added.

B5. NODES WITH EQUAL DISPLACEMENTS (6I1, 4X, 14I5) - NIDDOF cards

Columns	1:	Equal displacement code for X displacement, as follows. (a) zero or blank: displacement not contrained to be identical. (b) 1: displacement constrained to be identical for all nodes in group.
	2:	Code for Y displacement.
	3:	Code for Z displacement.
	4:	Code for XX rotation.
	5:	Code for YY rotation.
	6:	Code for ZZ rotation.
	7 - 10:	Blank
	11 - 15:	Number of nodes in group.
	16 - 80:	Up to 13 fields, each I5. List nodes in group. The first node must be the smallest numbered node in the group. See Note following.

Note: If the group has more than thirteen nodes, specify the remaining nodes on additional equal displacement commands. The smallest numbered node in the group must be the first node in the list for all commands defining the group. Greater computational efficiency may be obtained by constraining nodes to have equal displacements. However, the effect of specifying equal displacements may be to increase the band width of the structure stiffness matrix. This may result in an increase in the required stiffness matrix storage and/or the computational effort required to solve the equations of motion. Equal displacements specifications should therefore be used with caution. It should be noted that the equation solver used in the program is less sensitive to local increases in the stiffness matrix band width than a conventional equation solver based on a banded storage scheme.

B6. NODAL MASSES (I5, 6F10.0, 2I5) - NMSGC cards

Columns	1 - 5:	Node number, or number of first node in a series of nodes covered by this command.
	6 - 15:	Mass associated with X-displacement degree of freedom.
	16 - 25:	Mass associated with Y-displacement degree of freedom.
	26 - 35:	Mass associated with Z-displacement degree of freedom.
	36 - 45:	Mass associated with X-rotation degree of freedom.
	46 - 55:	Mass associated with Y-rotation degree of freedom.
	56 - 65:	Mass associated with Z-rotation degree of freedom.
	66 - 70:	Number of last node in series of nodes covered by this command. Leave blank for a single node.
	71 - 75:	Node number difference between successive nodes in series. Leave blank for a single node.

Note: The specification commands for lumped masses will generally permit the user to input the nodal masses with only a few data cards. Any node may, if desired, appear in more than one specification command. In such cases the mass associated with any degree of freedom will be the sum of the masses specified in separate commands. If certain nodes are constrained to have an equal displacement, the mass associated with this displacement will be the sum of the masses specified for the individual nodes. If a mass is specified for any degree of freedom that is constrained to be zero, it is ignored.

C. LOAD SPECIFICATION

C1. CONTROL CARD (8I5, 3F10.0) - One card

Columns

1 - 5: Code for static and/or dynamic analysis, (KSTAT).
- (a) zero or blank: dynamic analysis only.
- (b) 1: static analysis followed by dynamic analysis.
- (c) -1: static analysis only.

6 - 10: Number of static force patterns to be specified (NSPAT). See section C2. If blank or zero, no static loads will be applied.

11 - 15: Number of static force application commands (NSLGC). See Section F.

20: Code for ground motion records (IGM), as follows.
- (a) zero or blank: no ground motion records.
- (b) 1: ground motion records will be specified. See Section C3.

21 - 25: Number of dynamic force records to be specified (NDLR). See Section C4.

26 - 30: Largest number of points on any dynamic force record. This number is used for storage allocation.

31 - 35: Number of commands defining points of application of dynamic force records (NDLGC). See Section C5.

36 - 40: Number of integration time steps to be considered in dynamic analysis.

41 - 50: Integration time step, Δt.

51 - 60: Integration method parameter, δ, in Newmark's $\beta - \gamma - \delta$ method.

61 - 70: Integration method parameter, β, in Newmark's $\beta - \gamma - \delta$ method. If zero or blank, β is assumed to be equal to $0.25\,(1 + \delta)^2$.

C2. STATIC LOAD PATTERNS - NSPAT sets of cards as follows.

Each set consists of a control card followed by as many cards as needed to define the nodal loads. Load patterns are assumed to be input in numerical sequence.

C2(a) CONTROL CARD (I5, 3X, 18A4)

Columns	1 - 5:	Number of nodal load commands for this pattern (NSLC).
	9 - 80:	Load pattern title, to be printed with output.

C2(b) NODAL LOADS (I5, 6F10.0, 2I5) - NSLC cards

Columns	1 - 5:	Node number, or number of first node in a series of nodes covered by this command.
	6 - 15:	Load in X-direction, positive in positive direction of X-axis.
	16 - 25:	Load in Y-direction, positive in positive direction of Y-axis.
	26 - 35:	Load in Z-direction, positive in positive direction of Z-axis.
	36 - 45:	Moment about X-axis, positive by right hand screw rule.
	46 - 55:	Moment about Y-axis, positive by right hand screw rule.
	56 - 65:	Moment about Z-axis, positive by right hand screw rule.
	66 - 70:	Number of last node in series. Leave blank for a single node.
	71 - 75:	Node number difference between successive nodes in series. Leave blank for a single node, or if node number difference equals one.

C3. GROUND MOTION (ACCELERATION) RECORDS.

Omit if IGM, Section C1, is zero or blank. Accelerations are assumed to be in acceleration units, not as multiples of the acceleration due to gravity.

C3(a) CONTROL CARD (4I5, 6F10.0) - One card

Columns	1 - 5:	Number of time points defining ground motion record in X-direction (NIPX). Leave blank or punch zero for no ground motion in this direction.
	6 - 10:	Number of time points defining ground motion record in Y-direction (NIPY). Leave blank or punch zero for no ground motion in this direction.
	11 - 15:	Number of time points defining ground motion record in Z-direction (NIPZ). Leave blank or punch zero for no ground motion in this direction.
	16 - 20:	Print code, as follows (a) zero or blank: records are not printed. (b) 1: records are printed as input and scaled. (c) -1: records are printed as input, scaled and interpolated at time step intervals.
	21 - 30:	Input time interval for X-ground motion. If blank or zero, both time and acceleration values must be input; otherwise only acceleration values must be input, the times being automatically determined. See Section C3(b).
	31 - 40:	Input time interval for Y-ground motion. If blank or zero, both time and acceleration values must be input; otherwise only acceleration values must be input. See Section C3(c).
	41 - 50:	Input time interval for Z-ground motion. If blank or zero, both time and acceleration values must be input; otherwise only acceleration values must be input. See Section C3(d).
	51 - 60:	Scale factor by which X-ground accelerations are to be multiplied.
	61 - 70:	Scale factor by which Y-ground accelerations are to be multiplied.
	71 - 80:	Scale factor by which Z-ground accelerations are to be multiplied.

C3(b) X RECORD - One card followed by as many cards as needed.

Omit if NIPX is blank or zero.

(i) FIRST CARD (15A4, 5A4)

Columns 1 - 60: Record title, to be printed with output.

61 - 80: Input format to read NIPX points defining the record. For example, if the format is 12F6.0, punch (12F6.0).

(ii) FOLLOWING CARDS

As many cards as needed to specify NIPX input points, with the format defined in columns 61 - 80 of the first card. If both time and acceleration values are input, the time must immediately precede the corresponding accleration.

C3(c) Y RECORD - One card followed by as many cards as needed.

Omit if NIPY is blank or zero.

(i) FIRST CARD (15A4, 5A4)

Columns 1 - 60: Record title, to be printed with output.

61 - 80: Input format to read NIPY points defining the record.

(ii) FOLLOWING CARDS

As many cards as needed to specify NIPY input points, with the format defined in columns 61 - 80 of the first card.

C3(d) Z RECORD - One card followed by as many cards as needed.

Omit if NIPZ is blank or zero.

(i) FIRST CARD (15A4, 5A4)

Columns 1 - 60: Record title, to be printed with output.

61 - 80: Input format to read NIPZ points defining the record.

(ii) FOLLOWING CARDS

As many cards as needed to specify NIPZ input points, with the format defined in columns 61 - 80 of the first card.

Note: The acceleration scale factor may be used to increase or decrease the accelerations, or to convert from multiples of the acceleration due to gravity to acceleration units.

C4. DYNAMIC FORCE RECORDS - NDLR sets of cards, as follows.

Each set consists of one card followed by as many cards as needed to define the record. Records are assumed to be numbered in sequence as input.

C4(a) FIRST CARD (2I5, 2F10.0, 8A4, 2X, 4A4)

Columns 1 - 5: Number of time points defining record (NIPT).

6 - 10: Print code, as follows.
(a) zero or blank: record is not printed.
(b) 1: record is printed as input and scaled.
(c) -1: record is printed as input and scaled and as interpolated at time step intervals.

11 - 20: Input time interval. If blank or zero, both time and force values must be input; otherwise only force values.

21 - 30: Scale factor by which force values are to be multiplied.

31 - 62: Record title, to be printed with output.

65 - 80: Input format to read points defining the record.

C4(b) FOLLOWING CARDS

As many cards as needed to specify NIPT input points, with the format defined in columns 65 - 80 of the first card. If both time and force values are input, the time must immediately precede the corresponding force.

C5. DYNAMIC FORCE APPLICATION (16I5) - NDLGC Cards (See Section C1)

Acceleration records, if specified, are applied automatically, assuming all support points to move in phase. Force records are applied as defined by the cards of this section.

Columns 1 - 5: Dynamic force record number.

10: Direction code, as follows.
(a) 1: X translation.
(b) 2: Y translation.
(c) 3: Z translation.
(d) 4: X rotation.
(e) 5: Y rotation.
(f) 6: Z rotation.

11 - 80: Up to 14 fields, each I5. List the nodes at which the record is to be applied. Each node in the list is subjected to the scaled force record.

Note: The dynamic forces as specified by the dynamic force record number are applied in the positive direction defined by the direction code. To apply forces in the negative direction, the scale factor by which the force values are multiplied (Section C4) should be negative.

C6. DAMPING SPECIFICATION (3F10.0) - One Card

Omit if code for static and/or dynamic analysis, KSTAT (Section C1) equals -1.

Columns 1 - 10: Mass proportional damping factor, β_M.

11 - 20: Tangent stiffness proportional damping factor, β_T See Note following.

21 - 30: Initial stiffness proportional damping factor, β_o See Note following.

Note: If desired, it is possible to specify different values of the factors β_T and β_O for each element group. See Section E for explanation of this option.

D. OUTPUT SPECIFICATION

This set of cards consists of a control card followed by as many cards as needed to specify node numbers for output. See Note following.

D(a) CONTROL CARD (10I5, 7A4) - One card

Columns

1 - 5: Time interval for printout of nodal displacement, velocity and acceleration time histories, expressed as a multiple of the integration time step. Leave blank or punch zero for no time history output or if there is no dynamic analysis.

6 - 10: Time interval for printout of element action time histories (stresses, forces, etc.) expressed as a multiple of the integration time step. Leave blank or punch zero for no time history output or if there is no dynamic analysis.

11 - 15: Time interval for printout of intermediate envelopes of nodal displacements and element actions, expressed as a multiple of the integration time step. Leave blank or punch zero for no intermediate envelope output or if there is no dynamic analysis. Envelopes are automatically output at the end of the dynamic analysis.

16 - 20: Number of nodes for X-displacement, velocity and acceleration output (NODSX). For output at all nodes, punch -1.

21 - 25: Number of nodes for Y-displacement, velocity and acceleration output (NODSY). For output at all nodes, punch -1.

26 - 30: Number of nodes for Z-displacement, velocity and acceleration output (NODSZ). For output at all nodes, punch -1.

31 - 35: Time interval for punched output of nodal displacement, velocity and acceleration time histories, expressed as a multiple of the integration time step. Leave blank or punch zero for no punched output or if there is no dynamic analysis.

36 - 40: Number of nodes for punched output of X-displacement, velocity and acceleration response (NODXP). For output at all nodes, punch -1.

41 - 45: Number of nodes for punched output of Y-displacement, velocity and acceleration response (NODYP). For output at all nodes, punch -1.

Column 46 - 50: Number of nodes for punched output of Z-displacement, velocity and acceleration response (NODZP). For output at all nodes, punch -1.

51 - 78: Format for punched output. See Note following.

Note: Results for the same nodes and elements are printed for both static and dynamic analyses, except that velocities and accelerations are not printed for static analyses. Punched output is provided only for dynamic analyses.

Envelope values are printed for the dynamic analysis, and may be printed at the end of each static load increment if so specified on Card F(a). For punched output, the quantities output are node number, direction (i.e. X, Y or Z), displacement, velocity, acceleration and time. The node number and direction must be output in I5 and A5 format respectively; whereas other quantities may be output in any desired format, specified between parentheses in column 51 - 78, For example

(I5, A5, 3E15.5, 15X, E10.4)

D(b) FOLLOWING CARDS - SIX SETS OF CARDS, AS FOLLOWS.

(1) List of nodes for X response printout (16I5) - As many cards as needed to specify NODSX number of nodes, sixteen to a card. Omit if NODSX equals zero or -1.

(2) List of nodes for Y response printout (16I5) - As many cards as needed to specify NODSY number of nodes, sixteen to a card. Omit if NODSY equals zero, or -1.

(3) List of nodes for Z response printout (16I5) - As many cards as needed to specify NODSZ number of nodes, sixteen to a card. Omit if NODSZ equals zero, or -1.

(4) List of nodes for X response punched output (16I5) - As many cards as needed to specify NODXP number of nodes, sixteen to a card. Omit if NODXP equals zero, or -1.

(5) List of nodes for Y response punched output (16I5) - As many cards as needed to specify NODYP number of nodes, sixteen to a card. Omit if NODYP equals zero, or -1.

(6) List of nodes for Z response punched output (16I5) - As many cards as needed to specify NODZP number of nodes, sixteen to a card. Omit if NODZP equals zero, or -1.

E. ELEMENT SPECIFICATION

Elements must be divided into "groups". All elements in any group must be of the same type. However, elements of the same type may be divided into separate groups if desired.

Element groups may be input in any sequence. The total number of element groups may not exceed 20. The elements in any group must be numbered sequentially, the number of the first element in the group being any convenient number.

E1. THREE DIMENSIONAL TRUSS ELEMENTS

See Appendix B1 for description of element. Number of words of information per element = 96.

E1(a) CONTROL INFORMATION (10I5, 6F5.0) - One card

Columns		
	5:	Element group indicator. Punch 1 (to indicate that the group consists of three dimensional truss elements)
	6 - 10:	Number of elements in this group.
	11 - 15:	Element number of the first element in this group. If blank or zero, assumed to be equal to 1.
	16 - 20:	Number of material types. If blank or zero, assumed to be equal to 1.
	21 - 50:	Blank (not used for this element type).
	51 - 55:	Initial stiffness damping factor βo. If blank or zero, βo is assumed to be equal to the system βo value input in card C6.
	56 - 60:	Current tangent stiffness damping factor, β_T If blank or zero, β_T is assumed to be equal to the system β_T value input in card C6.

E1(b) MATERIAL PROPERTY INFORMATION (I5,4F10.0) - One card for each different material type.

Columns		
	1 - 5:	Material number, in sequence starting with 1.
	6 - 15:	Young's modulus of elasticity, E.
	16 - 25:	Strain hardening modulus as a proportion of Young's modulus (i.e. the ratio E_h/E).

Column 26 - 35: Yield stress in tension

36 - 45: Yield stress in compression, or elastic buckling stress in compression (Input as a positive value)

E1(c) ELEMENT GENERATION COMMANDS (4I5, 2F10.0, 4I5) - As many cards as needed to generate all elements in this group.

Cards must be entered in order of increasing element number. Cards for the first and last element must be included. See Note for explanation of generation procedure.

Columns 1 - 5: Element number, or number of first element in a sequentially numbered series of elements to be generated by this card.

6 - 10: Node number at element end i.

11 - 15: Node number at element end j.

16 - 20: Material number. If blank or zero, assumed to be equal to 1.

21 - 30: Cross sectional area.

31 - 40: Initial axial force on the element.

41 - 45: Node number increment for element generation. If blank or zero assumed to be equal to 1.

50: Code for large displacement effects. Leave blank or punch zero, for small displacement effects. Punch 1 for large displacement effects.

55: Time history output code. Leave blank or punch zero for no time history output. Punch 1 if time history output is required.

60: Buckling code. Leave blank or punch zero if element yields in compression without buckling. Punch 1 if element buckles elastically in compression.

E2. TWO DIMENSIONAL FINITE ELEMENTS

See Appendix B2 for description of element. The number of words of information per element varies with the number of nodes and the integration order.

The element must lie in the XY plane.

E2(a) CONTROL INFORMATION (10I5, 6F5.0) - One card

Columns 5: Element group indicator. Punch 3 (to indicate that group consists of two dimensional finite elements).

6 - 10: Number of elements in this group.

11 - 15: Element number of the first element in this group. If blank or zero, assumed to be equal to 1.

16 - 20: Number of material types. If blank or zero, assume to be equal to 1.

21 - 25: Number of nodes for each element in group. If blank or zero, assumed to be equal to 4. Must not exceed 8.

26 - 30: Integration order in r-direction. If blank or zero, assumed to be equal to 2. Should not exceed 4

31 - 35: Integration order in s-direction. If blank or zero, assumed to be equal to 2. Should not exceed 4.

36 - 40: Indicator for type of behavior: Punch 0 for Plane stress.
Punch 1 for Plane strain.
Punch 2 for Axisymmetric Solid.

41 - 45: (For elasto-plastic material models only). Number of plastic strain increments for numerical integration during plastic loading. If blank or zero, assumed to be equal to 20.

46 - 50: (For elasto-plastic material models only). Order of Runge-Kutta integration for numerical integration during plastic loading. If blank or zero, assumed to be equal to 1 (i.e. simple Euler integration). Must not exceed 4.

51 - 55: Initial stiffness damping factor β_O. If blank or zero, β_O is assumed to be equal to the system β_O value input in card C6.

56 - 60: Current tangent stiffness damping factor, β_T. If blank or zero, β_T is assumed to be equal to the system β_T value input in card C6.

E2(b) MATERIAL PROPERTY INFORMATION - Two cards for each material type

(i) FIRST CARD (I5, 4F10.0)

Columns 1 - 5: Material identification number

6 - 15: Young's Modulus of Elasticity, E_A, along material axis -A.

16 - 25: Poisson's Raio, μ_{AB}

26 - 35: Yield stress, S_y

(ii) SECOND CARD (5F10.0)

Leave blank for isotropic elastic or elasto-plastic materials. The data on this card will be ignored for any element if the material behavior code for the element specifies an isotropic material.

Columns 1 - 10: Young's Modulus of Elasticity, E_B, along material axis -B.

11 - 20: Young's Modulus of Elasticity, E_C, along material axis -C.

21 - 30: Poisson's ratio, μ_{AC}.

31 - 40: Poisson's ratio, μ_{BC}.

41 - 50: Shear modulus, G_{AB}.

E2(c) ELEMENT GENERATION COMMANDS - (10I5, F5.0, I5, I2, 1X, 2I1, 3F5.0)

As many cards as needed to generate all elements in this group.

Cards must be entered in order of increasing element number. Cards for the first and the last element must be included. See Note for explanation of generation procedure.

Columns 1 - 5: Element number, or number of first element in a sequentially numbered series of elements to be generated by the card.

Columns 6 - 10:	Node number at node 1.	
11 - 15:	Node number at node 2.	
16 - 20:	Node number at node 3.	
21 - 25:	Node number at node 4.	
26 - 30:	Node number at node 5.	Leave blank if any node is absent.
31 - 35:	Node number at node 6.	
36 - 40:	Node number at node 7.	
41 - 45:	Node number at node 8.	
46 - 50:	Material number. If blank or zero, assumed to be equal to 1.	
51 - 55:	Element thickness for plane stress behavior. Leave blank for plane strain or axisymmetric behavior.	
56 - 60:	Node number increment for element generation: if blank or zero assumed to be equal to 1.	
62:	Code for material behavior: Punch 0 for elastic isotropic material. Punch 1 for elastic orthotropic material. Punch 2 for elasto-plastic isotropic material.	
64:	Code for large displacement effects. Leave blank or punch zero, for small displacement effects. Punch 1 for large displacement effects.	
65:	Time history output code. Leave blank or punch zero for no time history output. Punch 1 for time history output at the "output" point, natural coordinates of which are specified in columns 66 thru 75. Punch 2 for time history output at Gauss integration points (in addition to the "output" point). It specified, the time history will be output at intervals specified on card D(a).	
66 - 70:	r-coordinate of "output" point. If blank, r = 0.0	
71 - 75:	s-coordinate of "output" point. If blank, s = 0.0	
76 - 80:	Angle ψ (degrees) which the material A-axis makes with side connecting node 4 to node 1. Leave blank for isotropic material.	

Note: In the element generation commands, the elements must be specified in increasing numerical order. Cards may be provided for sequentially numbered elements, in which case each card specifies one element and the generation option is not used. Alternatively, the cards for a group of elements may be omitted, in which case the data for the missing group is generated as follows:

(1) All elements are assigned the same material number, cross sectional area, code for large displacement effects, etc. as for the element preceding the missing group of elements.

(2) The node numbers for each missing element are obtained by adding the specified node number increment to the node numbers of each preceding element. The node number increment is that specified for the element preceding the missing set of elements.

In the printout of the element data, generated data is prefixed by an asterisk.

F. STATIC ANALYSIS SPECIFICATION - NSLGC sets of cards (See Section C1).

Each set consists of a solution procedure card followed by one or more cards defining a linear combination of static force patterns. Each set defines an increment of static load.

F(a). SOLUTION PROCEDURE CARD (8I5, 4F10.0) - One card

Columns 1 - 5: Number of equal steps in which load increment is to be applied, positive if results envelopes are not to be printed at the end of the increment, otherwise negative.

6 - 10: Iteration type, as follows.
- (a) zero or blank: Newton-Raphson iteration
- (b) n: Constant stiffness iteration with alpha-constant over-relaxation, the alpha matrix being reinitialized every n iterations.

15: Type of state determination calculation to be used for constant stiffness iteration as follows:
- (a) zero or blank: path independent.
- (b) 1: path dependent.

Path dependent state determination is always used for Newton-Raphson iteration.

16 - 20: Stiffness reformation code, as follows.
- (a) zero or blank: stiffness used in preceding step is retained.
- (b) n: stiffness is reformed evey n load steps.

25: Termination code, as follows.
- (a) zero or blank: If the solution does not converge within the maximum number of iterations for any load step, the next load step will be applied.
- (b) 1: If the solution does not converge, the execution will terminate.

26 - 30: Print code, as follows.
- (a) -1: results are not printed for this increment.
- (b) zero or blank: results are printed at the end of the increment only.
- (c) 1: results are printed after each load step.
- (d) 2: results are printed every iteration. This option should be used for debugging purposes only.

Columns 31 - 35: Maximum number of cyles of iteration within any load step.

36 - 40: Maximum number of iterations within any cycle.

41 - 50: Nodal force convergence tolerance to be used in last step of load increment.

51 - 60: Nodal force convergence tolerance to be used in all except last step of load increment.

61 - 70: Nodal force tolerance for change of stiffness in Newton-Raphson iteration. If the unbalanced force reduces below this tolerance, the stiffness will not be reformed for the next iteration.

71 - 80: Maximum nodal displacement (translation or rotation) increment permitted in any iteration step. Leave blank for unlimited displacement. Displacement limits should be specified only with Newton-Raphson iteration.

F(b). FOLLOWING CARDS (8F10.0) - As many cards as needed

Columns 1 - 80: Up to eight fields, each F10.0. For each static force pattern in turn, specify a scale factor by which the pattern is to be multiplied. The scaled patterns are added together to produce the load increment.

Scale factors may be positive or negative. Leave the corresponding field blank or punch zero to ignore any force pattern.

G. DYNAMIC ANALYSIS SPECIFICATION

G1. DYNAMIC SOLUTION PROCEDURE CARD (7I5, 4F10.0, I5) - One card

Omit if KSTAT (Section C1) equals -1.

Columns 1 - 5: Iteration type, as follows.
(a) zero or blank: Newton-Raphson iteration.
(b) n > 0: Constant stiffness iteration with alpha-constant over-relaxation, the alpha matrix being reinitialized every n iterations.

10: Type of state determination calculation to be used for constant stiffness iteration, as follows.
(a) zero or blank: path independent.
(b) 1: path dependent
Path dependent state determination is always used for Newton-Raphson iteration.

15: Stiffness reformation code, as follows.
(a) zero or blank: stiffness used in preceding time step is retained.
(b) n: stiffness is reformed every n time steps.

20: Termination code, as follows.
(a) zero or blank: If the solution does not converge within the maximum number of iterations for any time step, the next time step will be applied.
(b) 1: If the solution does not converge, the execution will terminate.

21 - 25: Maximum number of cycles of iteration within any time step.

26 - 30: Maximum number of iterations within any cycle.

31 - 35: Number of time steps between application of "fine" convergence tolerance. The "coarse" tolerance is used at intermediate steps.

36 - 45: "Fine" nodal force convergence tolerance.

46 - 55: "Coarse" nodal force convergence tolerance.

56 - 65: Nodal force tolerance for change of stiffness in Newton-Raphson iteration. If the unbalanced force reduces below this tolerance, the stiffness will not be reformed for the next iteration.

Columns 66 - 75: Maximum nodal displacement (translation or rotation) increment permitted in any iteration step. Leave blank for unlimited displacement. Displacement limits should be specified only with Newton-Raphson iteration.

76 - 80: Number of initial condition generation commands (NICGC). See Section G2.

G2. INITIAL CONDITION SPECIFICATION (I5, 2F10.0, 11I5) - NICGC cards (See Section G1).

Columns 1 - 5: Direction code, as follows.
(a) 1: X translation
(b) 2: Y translation
(c) 3: Z translation
(d) 4: X rotation
(e) 5: Y rotation
(f) 6: Z rotation

6 - 15: Initial velocity.

16 - 25: Initial acceleration.

26 - 80: Up to 11 fields, each I5. List up to 11 nodes having the same initial conditions.

H. NEW PROBLEM

Data for a new problem may follow immediately starting with Section A. Any number of structures may be analyzed in a single computer run.

I. TERMINATION CARD (A4) - One card to terminate the complete data deck.

Columns 1 - 4: Punch the word STOP.

APPENDIX B1

THREE DIMENSIONAL TRUSS ELEMENT

(a) General Characteristics

Truss elements may be arbitrarily oriented in space, but can transmit axial load only (Fig. B1.1). Large displacement effects may or may not be included. When this effect is specified, it is included in both static and dynamic analyses.

Two alternative modes of inelastic behavior may be specified, namely (1) yielding in both tension and compression (Fig. B1.2a) and (2) yielding in tension with elastic buckling in compression (Fig. B1.2b). Strain hardening effects may be considered. It should be noted that the inelastic behavior is specified in terms of stress and strain, rather than axial force and axial deformation. For computations, a strain hardening stress-strain relationship is decomposed into two components, one linearly elastic and the other elastic-perfectly plastic. A truss bar for linearly elastic behavior may be obtained by specifying a very high value of the yield stress.

Initial axial forces in the truss elements can be specified. These initial forces will typically be the forces in the elements under static loading, as calculated by a separate analysis. For consistency, these forces should be in equilibrium with the static load producing them, but this is not essential as the computer program makes corrections for any equilibrium unbalance resulting from the initial forces. Thus it is possible to compute the displacements of a truss-bar structure with specified initial forces.

(b) Output Results

For static analyses, the results may be output at each iteration, at each load step or at the end of each load increment. Results envelopes (i.e. maximum positive and negative values) may be printed at the end of each load increment. For dynamic analysis the time history results and results envelopes may be printed at specified time step intervals. For both static and dynamic analyses, the results are printed only for those elements for which the results are requested, except that results envelopes are printed for all elements.

The results output consists of the following:

(1) Element number.

(2) Node numbers at end i and end j.

(3) Yield code: zero indicates that the element is elastic, and one indicates that it is yielding or buckling.

(4) Axial force, tension positive.

(5) Total axial deformation, elongation positive.

(6) Accumulated positive and negative plastic deformations (elongation positive) up to the current load or time. These deformations are computed by accumulating the plastic extensions during all positive and negative plastic excursions. For an element which buckles in compression (Fig. B1.2b), the accumulated negative plastic deformations are printed as zero.

The results envelopes consist of the following output:

(1) Element number.

(2) Node numbers at end i and end j.

(3) Maximum positive and negative values of axial force, and the corresponding times at which these peak values occur (printed as zero in static analysis).

(4) Maximum positive and negative values of total deformation, and the corresponding times.

(5) Accumulated plastic deformations.

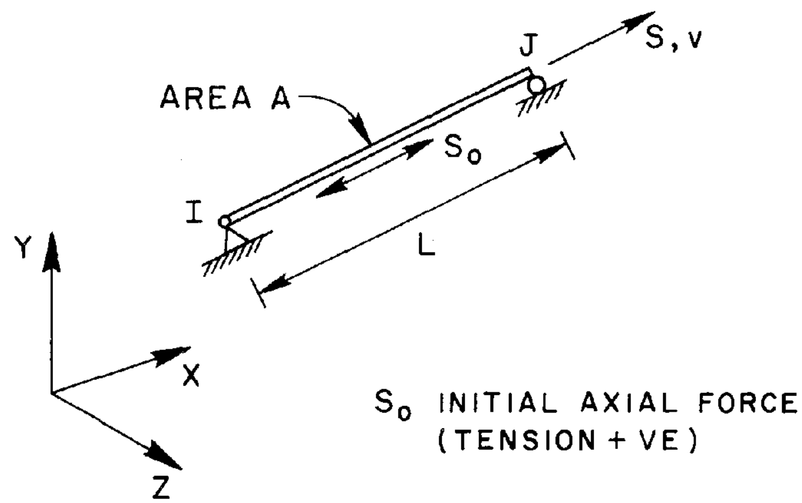
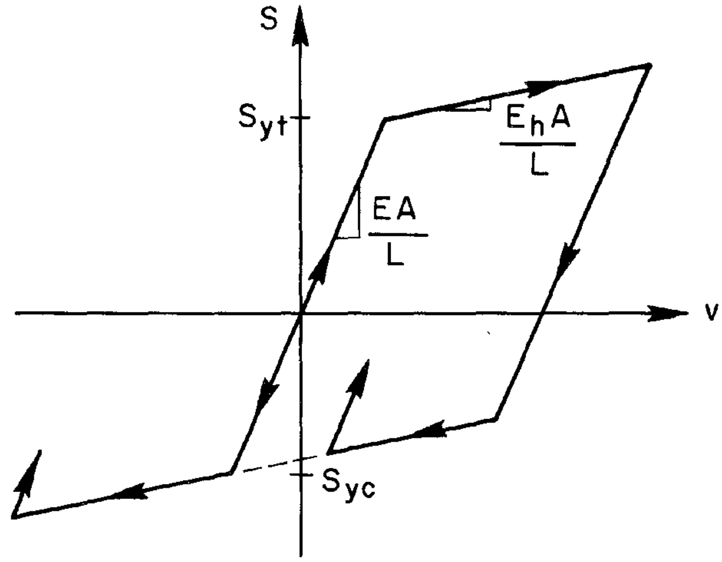
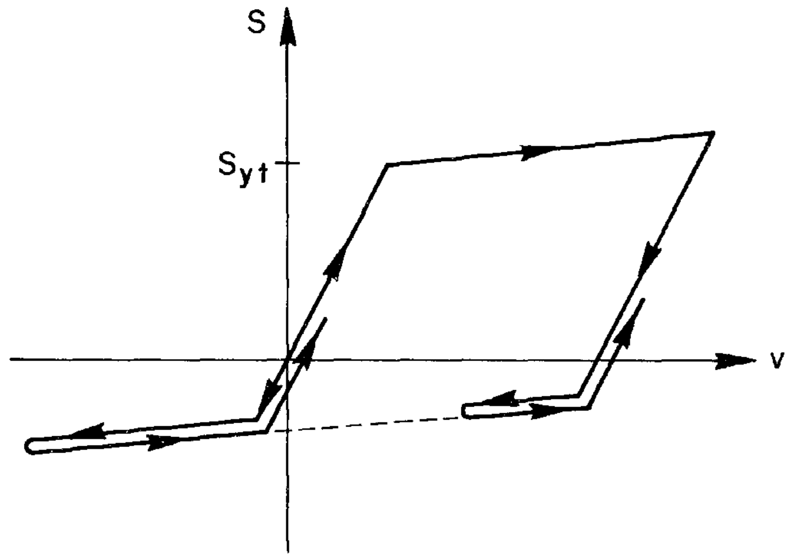


FIG. B1.1 TRUSS ELEMENT



(a) YIELD IN TENSION AND COMPRESSION



(b) YIELD IN TENSION, BUCKLING IN COMPRESSION

FIG. B1.2 INELASTIC BEHAVIOR FOR TRUSS ELEMENT

APPENDIX B2

TWO DIMENSIONAL 4-TO-8 NODE ELEMENT

(a) General Characteristics

Two dimensional elements must lie in the X Y plane, and can have from four to eight nodes. The local node numbering is shown in Fig. B2.1. The element maps into a rectangular element in a local r-s coordinate system, such that nodes 1 through 4 are located at the four corners and nodes 5 through 8 are located at the midsides of the rectangle. The four corner nodes must always be specified and any one or more of the midside nodes may be specified. Thus, the basic 4-node and higher order 5-to-8 node elements can be combined to produce a variety of finite element discretizations.

Three different formulations, namely (1) plane stress, (2) plane strain, and (3) axisymmetric solid elements are included. In the plane strain formulation it is assumed that the element has a unit thickness, whereas in the axiysmmetric formulation a unit radian segment ($\theta = 1$, Fig. B2.1) is considered. The nodal loads for plane strain and axisymmetric structures must be computed accordingly. The element matrices are computed using Gauss quadrature integration. The integration order in the r-direction and s-direction may be specified separately. Presently any integration order up to 4 may be input in either direction; however, a 2x2 Gauss quadrature integration is recommended for most cases.

Large displacement effects may be included, if desired. These effects are included in both the static and dynamic analyses. The material may be either (1) isotropic linearly elastic, (2) orthotropic linearly elastic or (3) isotropic elastic-perfectly plastic with von Mises yield function. For orthotropic material behavior, material properties are defined with respect to right handed rectangular coordinate axes A-B-C, with the axes A, B lying in the plane of the element (Fig. B2.1).

(b) Results Output

For static analysis, the results may be output at each iteration, at each load step or at the end of each load increment. Results envelopes may be printed at the end of each load increment. For dynamic analysis, time history results and results envelopes may be printed at specified time step intervals. The results are output only for those elements for which results are requested, except that results envelopes are printed for all elements.

The results output consists of the following.

(1) Element number.

(2) Yield code at "output" point and each Gauss point (if so specified by the time history output code). Zero indicates that the material is elastic, and one indicates that it is yielding.

(3) Stress components (SIG11, SIG22, SIG12, and SIG33) at "output" point, and at each Gauss point (if so specified by the time history output code).

(4) Strain components (STR11, STR22, STR12 and STR33) at "output" point, and at each Gauss point (if so specified by the time history output code).

(5) Effective stress at "output" point, and at each Gauss point (if so specified by the time history output code). Here the effective stress is defined as

$$\text{EFFSIG} = \sqrt{1.5 * \text{YSS}}$$

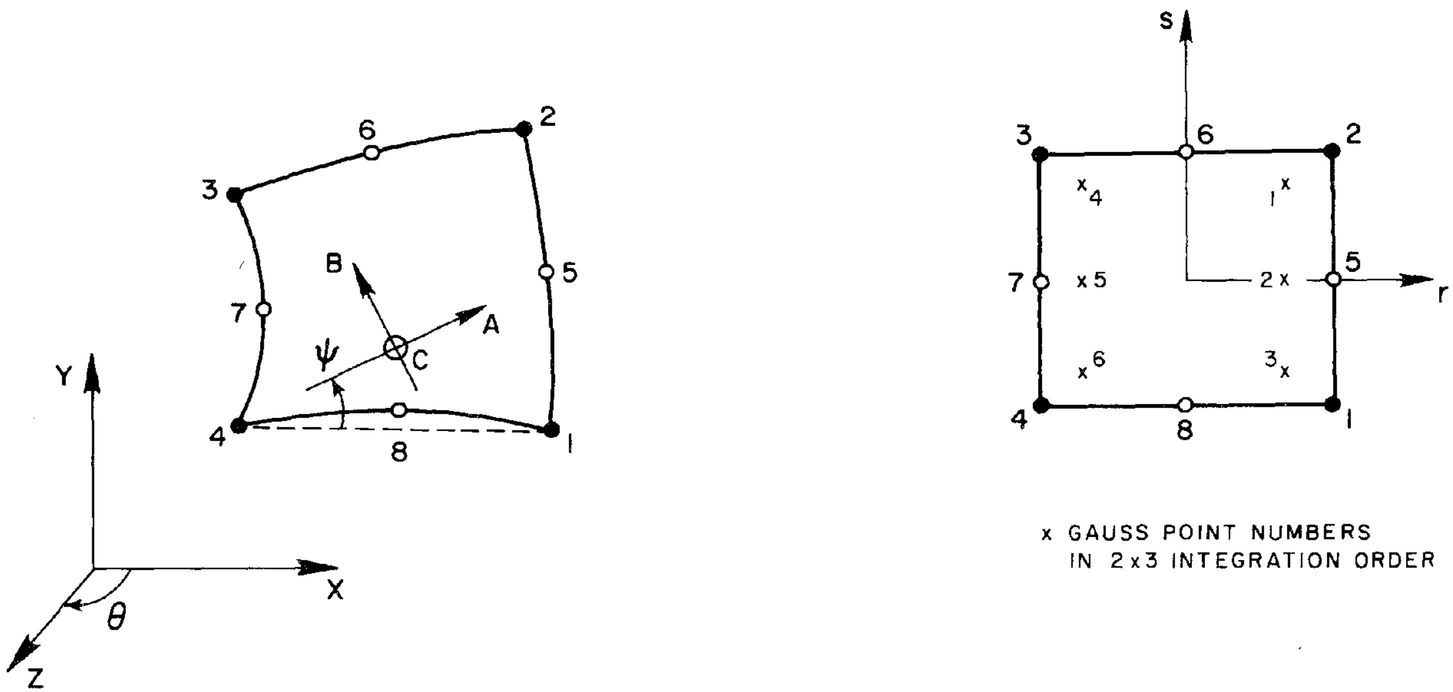
where $\text{YSS} = (\text{SIG11})^2 + (\text{SIG22})^2 + (\text{SIG33})^2 + 2(\text{SIG12})^2$

The results envelopes consist of the following output.

(1) Element number.

(2) Maximum positive and negative values of the stress components (SIG11, SIG22, SIG12 and SIG33) at "output" point and at each Gauss point, and the corresponding times at which these peak values occur. Times are printed as zero for static analysis.

(3) Same as in (2) for the strain components.



(a) TWO DIMENSIONAL ELEMENT IN GLOBAL X-Y-Z SYSTEM

(b) TWO DIMENSIONAL ELEMENT IN LOCAL r-s SYSTEM

FIG. B2.1 TWO DIMENSIONAL FINITE ELEMENT

EERC 67-1 "Feasibility Study Large-Scale Earthquake Simulator Facility," by J. Penzien, J. G. Bouwkamp, R. W. Clough and D. Rea - 1967 (PB 187 905)

EERC 68-1 Unassigned

EERC 68-2 "Inelastic Behavior of Beam-to-Column Subassemblages Under Repeated Loading," by V. V. Bertero - 1968 (PB 184 888)

EERC 68-3 "A Graphical Method for Solving the Wave Reflection-Refraction Problem," by H. D. McNiven and Y. Mengi 1968 (PB 187 943)

EERC 68-4 "Dynamic Properties of McKinley School Buildings," by D. Rea, J. G. Bouwkamp and R. W. Clough - 1968 (PB 187 902)

EERC 68-5 "Characteristics of Rock Motions During Earthquakes," by H. B. Seed, I. M. Idriss and F. W. Kiefer - 1968 (PB 188 338)

EERC 69-1 "Earthquake Engineering Research at Berkeley," - 1969 (PB 187 906)

EERC 69-2 "Nonlinear Seismic Response of Earth Structures," by M. Dibaj and J. Penzien - 1969 (PB 187 904)

EERC 69-3 "Probabilistic Study of the Behavior of Structures During Earthquakes," by P. Ruiz and J. Penzien - 1969 (PB 187 886)

EERC 69-4 "Numerical Solution of Boundary Value Problems in Structural Mechanics by Reduction to an Initial Value Formulation," by N. Distefano and J. Schujman - 1969 (PB 187 942)

EERC 69-5 "Dynamic Programming and the Solution of the Biharmonic Equation," by N. Distefano - 1969 (PB 187 941)

Note: Numbers in parenthesis are Accession Numbers assigned by the National Technical Information Service. Copies of these reports may be ordered from the National Technical Information Service, 5285 Port Royal Road, Springfield, Virginia, 22161. Accession Numbers should be quoted on orders for the reports (PB --- ---) and remittance must accompany each order. (Foreign orders, add $2.50 extra for mailing charges.) Those reports without this information listed are not yet available from NTIS. Upon request, EERC will mail inquirers this information when it becomes available to us.

EERC 69-6 "Stochastic Analysis of Offshore Tower Structures," by A. K. Malhotra and J. Penzien - 1969 (PB 187 903)

EERC 69-7 "Rock Motion Accelerograms for High Magnitude Earthquakes," by H. B. Seed and I. M. Idriss - 1969 (PB 187 940)

EERC 69-8 "Structural Dynamics Testing Facilities at the University of California, Berkeley," by R. M. Stephen, J. G. Bouwkamp, R. W. Clough and J. Penzien - 1969 (PB 189 111)

EERC 69-9 "Seismic Response of Soil Deposits Underlain by Sloping Rock Boundaries," by H. Dezfulian and H. B. Seed - 1969 (PB 189 114)

EERC 69-10 "Dynamic Stress Analysis of Axisymmetric Structures under Arbitrary Loading," by S. Ghosh and E. L. Wilson - 1969 (PB 189 026)

EERC 69-11 "Seismic Behavior of Multistory Frames Designed by Different Philosophies," by J. C. Anderson and V. V. Bertero - 1969 (PB 190 662)

EERC 69-12 "Stiffness Degradation of Reinforcing Concrete Structures Subjected to Reversed Actions," by V. V. Bertero, B. Bresler and H. Ming Liao - 1969 (PB 202 942)

EERC 69-13 "Response of Non-Uniform Soil Deposits to Travel Seismic Waves," by H. Dezfulian and H. B. Seed - 1969 (PB 191 023)

EERC 69-14 "Damping Capacity of a Model Steel Structure," by D. Rea, R. W. Clough and J. G. Bouwkamp - 1969 (PB 190 663)

EERC 69-15 "Influence of Local Soil Conditions on Building Damage Potential during Earthquakes," by H. B. Seed and I. M. Idriss - 1969 (PB 191 036)

EERC 69-16 "The Behavior of Sands under Seismic Loading Conditions," by M. L. Silver and H. B. Seed - 1969 (AD 714 982)

EERC 70-1 "Earthquake Response of Concrete Gravity Dams," by A. K. Chopra - 1970 (AD 709 640)

EERC 70-2 "Relationships between Soil Conditions and Building Damage in the Caracas Earthquake of July 29, 1967," by H. B. Seed, I. M. Idriss and H. Dezfulian - 1970 (PB 195 762)

EERC 70-3 "Cyclic Loading of Full Size Steel Connections," by E. P. Popov and R. M. Stephen - 1970 (PB 213 545)

EERC 70-4 "Seismic Analysis of the Charaima Building, Caraballeda, Venezuela," by Subcommittee of the SEAONC Research Committee: V. V. Bertero, P. F. Fratessa, S. A. Mahin, J. H. Sexton, A. C. Scordelis, E. L. Wilson, L. A. Wyllie, H. B. Seed and J. Penzien, Chairman - 1970 (PB 201 455)

EERC 70-5 "A Computer Program for Earthquake Analysis of Dams," by A. K. Chopra and P. Chakrabarti - 1970 (AD 723 994)

EERC 70-6 "The Propagation of Love Waves across Non-Horizontally Layered Structures," by J. Lysmer and L. A. Drake - 1970 (PB 197 896)

EERC 70-7 "Influence of Base Rock Characteristics on Ground Response," by J. Lysmer, H. B. Seed and P. B. Schnabel - 1970 (PB 197 897)

EERC 70-8 "Applicability of Laboratory Test Procedures for Measuring Soil Liquefaction Characteristics under Cyclic Loading," by H. B. Seed and W. H. Peacock - 1970 (PB 198 016)

EERC 70-9 "A Simplified Procedure for Evaluating Soil Liquefaction Potential," by H. B. Seed and I. M. Idriss - 1970 (PB 198 009)

EERC 70-10 "Soil Moduli and Damping Factors for Dynamic Response Analysis," by H. B. Seed and I. M. Idriss - 1970 (PB 197 869)

EERC 71-1 "Koyna Earthquake and the Performance of Koyna Dam," by A. K. Chopra and P. Chakrabarti - 1971 (AD 731 496)

EERC 71-2 "Preliminary In-Situ Measurements of Anelastic Absorption in Soils Using a Prototype Earthquake Simulator," by R. D. Borcherdt and P. W. Rodgers - 1971 (PB 201 454)

EERC 71-3 "Static and Dynamic Analysis of Inelastic Frame Structures," by F. L. Porter and G. H. Powell - 1971 (PB 210 135)

EERC 71-4 "Research Needs in Limit Design of Reinforced Concrete Structures," by V. V. Bertero - 1971 (PB 202 943)

EERC 71-5 "Dynamic Behavior of a High-Rise Diagonally Braced Steel Building," by D. Rea, A. A. Shah and J. G. Bouwkamp - 1971 (PB 203 584)

EERC 71-6 "Dynamic Stress Analysis of Porous Elastic Solids Saturated with Compressible Fluids," by J. Ghaboussi and E. L. Wilson - 1971 (PB 211 396)

EERC 71-7 "Inelastic Behavior of Steel Beam-to-Column Subassemblages," by H. Krawinkler, V. V. Bertero and E. P. Popov - 1971 (PB 211 335)

EERC 71-8 "Modification of Seismograph Records for Effects of Local Soil Conditions," by P. Schnabel, H. B. Seed and J. Lysmer - 1971 (PB 214 450)

EERC 72-1 "Static and Earthquake Analysis of Three Dimensional Frame and Shear Wall Buildings," by E. L. Wilson and H. H. Dovey - 1972 (PB 212 904)

EERC 72-2 "Accelerations in Rock for Earthquakes in the Western United States," by P. B. Schnabel and H. B. Seed - 1972 (PB 213 100)

EERC 72-3 "Elastic-Plastic Earthquake Response of Soil-Building Systems," by T. Minami - 1972 (PB 214 868)

EERC 72-4 "Stochastic Inelastic Response of Offshore Towers to Strong Motion Earthquakes," by M. K. Kaul - 1972 (PB 215 713)

EERC 72-5 "Cyclic Behavior of Three Reinforced Concrete Flexural Members with High Shear," by E. P. Popov, V. V. Bertero and H. Krawinkler - 1972 (PB 214 555)

EERC 72-6 "Earthquake Response of Gravity Dams Including Reservoir Interaction Effects," by P. Chakrabarti and A. K. Chopra - 1972 (AD 762 330)

EERC 72-7 "Dynamic Properties on Pine Flat Dam," by D. Rea, C. Y. Liaw and A. K. Chopra - 1972 (AD 763 928)

EERC 72-8 "Three Dimensional Analysis of Building Systems," by E. L. Wilson and H. H. Dovey - 1972 (PB 222 438)

EERC 72-9 "Rate of Loading Effects on Uncracked and Repaired Reinforced Concrete Members," by S. Mahin, V. V. Bertero, D. Rea and M. Atalay - 1972 (PB 224 520)

EERC 72-10 "Computer Program for Static and Dynamic Analysis of Linear Structural Systems," by E. L. Wilson, K.-J. Bathe, J. E. Peterson and H. H. Dovey - 1972 (PB 220 437)

EERC 72-11 "Literature Survey - Seismic Effects on Highway Bridges," by T. Iwasaki, J. Penzien and R. W. Clough - 1972 (PB 215 613)

EERC 72-12 "SHAKE-A Computer Program for Earthquake Response Analysis of Horizontally Layered Sites," by P. B. Schnabel and J. Lysmer - 1972 (PB 220 207)

EERC 73-1 "Optimal Seismic Design of Multistory Frames," by V. V. Bertero and H. Kamil - 1973

EERC 73-2 "Analysis of the Slides in the San Fernando Dams during the Earthquake of February 9, 1971," by H. B. Seed, K. L. Lee, I. M. Idriss and F. Makdisi - 1973 (PB 223 402)

EERC 73-3 "Computer Aided Ultimate Load Design of Unbraced Multistory Steel Frames," by M. B. El-Hafez and G. H. Powell - 1973

EERC 73-4 "Experimental Investigation into the Seismic Behavior of Critical Regions of Reinforced Concrete Components as Influenced by Moment and Shear," by M. Celebi and J. Penzien - 1973 (PB 215 884)

EERC 73-5 "Hysteretic Behavior of Epoxy-Repaired Reinforced Concrete Beams," by M. Celebi and J. Penzien - 1973

EERC 73-6 "General Purpose Computer Program for Inelastic Dynamic Response of Plane Structures," by A. Kanaan and G. H. Powell - 1973 (PB 221 260)

EERC 73-7 "A Computer Program for Earthquake Analysis of Gravity Dams Including Reservoir Interaction," by P. Chakrabarti and A. K. Chopra - 1973 (AD 766 271)

EERC 73-8 "Behavior of Reinforced Concrete Deep Beam-Column Subassemblages under Cyclic Loads," by O. Kustu and J. G. Bouwkamp - 1973

EERC 73-9 "Earthquake Analysis of Structure-Foundation Systems," by A. K. Vaish and A. K. Chopra - 1973 (AD 766 272)

EERC 73-10 "Deconvolution of Seismic Response for Linear Systems," by R. B. Reimer - 1973 (PB 227 179)

EERC 73-11 "SAP IV: A Structural Analysis Program for Static and Dynamic Response of Linear Systems," by K.-J. Bathe, E. L. Wilson and F. E. Peterson - 1973 (PB 221 967)

EERC 73-12 "Analytical Investigations of the Seismic Response of Long, Multiple Span Highway Bridges," by W. S. Tseng and J. Penzien - 1973 (PB 227 816)

EERC 73-13 "Earthquake Analysis of Multi-Story Buildings Including Foundation Interaction," by A. K. Chopra and J. A. Gutierrez - 1973 (PB 222 970)

EERC 73-14 "ADAP: A Computer Program for Static and Dynamic Analysis of Arch Dams," by R. W. Clough, J. M. Raphael and S. Majtahedi - 1973 (PB 223 763)

EERC 73-15 "Cyclic Plastic Analysis of Structural Steel Joints," by R. B. Pinkney and R. W. Clough - 1973 (PB 226 843)

EERC 73-16 "QUAD-4: A Computer Program for Evaluating the Seismic Response of Soil Structures by Variable Damping Finite Element Procedures," by I. M. Idriss, J. Lysmer, R. Hwang and H. B. Seed - 1973 (PB 229 424)

EERC 73-17 "Dynamic Behavior of a Multi-Story Pyramid Shaped Building," by R. M. Stephen and J. G. Bouwkamp - 1973

EERC 73-18 "Effect of Different Types of Reinforcing on Seismic Behavior of Short Concrete Columns," by V. V. Bertero, J. Hollings, O. Kustu, R. M. Stephen and J. G. Bouwkamp - 1973

EERC 73-19 "Olive View Medical Center Material Studies, Phase I," by B. Bresler and V. V. Bertero - 1973 (PB 235 986)

EERC 73-20 "Linear and Nonlinear Seismic Analysis Computer Programs for Long Multiple-Span Highway Bridges," by W. S. Tseng and J. Penzien - 1973

EERC 73-21 "Constitutive Models for Cyclic Plastic Deformation of Engineering Materials," by J. M. Kelly and P. P. Gillis - 1973 (PB 226 024)

EERC 73-22 "DRAIN - 2D User's Guide," by G. H. Powell - 1973 (PB 227 016)

EERC 73-23 "Earthquake Engineering at Berkeley - 1973" - 1973 (PB 226 033)

EERC 73-24 Unassigned

EERC 73-25 "Earthquake Response of Axisymmetric Tower Structures Surrounded by Water," by C. Y. Liaw and A. K. Chopra - 1973 (AD 773 052)

EERC 73-26 "Investigation of the Failures of the Olive View Stairtowers during the San Fernando Earthquake and Their Implications in Seismic Design," by V. V. Bertero and R. G. Collins - 1973 (PB 235 106)

EERC 73-27 "Further Studies on Seismic Behavior of Steel Beam-Column Subassemblages," by V. V. Bertero, H. Krawinkler and E. P. Popov - 1973 (PB 234 172)

EERC 74-1 "Seismic Risk Analysis," by C. S. Oliveira - 1974 (PB 235 920)

EERC 74-2 "Settlement and Liquefaction of Sands under Multi-Directional Shaking," by R. Pyke, C. K. Chan and H. B. Seed - 1974

EERC 74-3 "Optimum Design of Earthquake Resistant Shear Buildings," by D. Ray, K. S. Pister and A. K. Chopra - 1974 (PB 231 172)

EERC 74-4 "LUSH - A Computer Program for Complex Response Analysis of Soil-Structure Systems," by J. Lysmer, T. Udaka, H. B. Seed and R. Hwang - 1974 (PB 236 796)

EERC 74-5 "Sensitivity Analysis for Hysteretic Dynamic Systems: Applications to Earthquake Engineering," by D. Ray - 1974 (PB 233 213)

EERC 74-6 "Soil-Structure Interaction Analyses for Evaluating Seismic Response," by H. B. Seed, J. Lysmer and R. Hwang - 1974 (PB 236 519)

EERC 74-7 Unassigned

EERC 74-8 "Shaking Table Tests of a Steel Frame - A Progress Report," by R. W. Clough and D. Tang - 1974

EERC 74-9 "Hysteretic Behavior of Reinforced Concrete Flexural Members with Special Web Reinforcement," by V. V. Bertero, E. P. Popov and T. Y. Wang - 1974 (PB 236 797)

EERC 74-10 "Applications of Reliability-Based, Global Cost Optimization to Design of Earthquake Resistant Structures," by E. Vitiello and K. S. Pister - 1974 (PB 237 231)

EERC 74-11 "Liquefaction of Gravelly Soils under Cyclic Loading Conditions," by R. T. Wong, H. B. Seed and C. K. Chan - 1974

EERC 74-12 "Site-Dependent Spectra for Earthquake-Resistant Design," by H. B. Seed, C. Ugas and J. Lysmer - 1974

EERC 74-13 "Earthquake Simulator Study of a Reinforced Concrete Frame," by P. Hidalgo and R. W. Clough - 1974 (PB 241 944)

EERC 74-14 "Nonlinear Earthquake Response of Concrete Gravity Dams," by N. Pal - 1974 (AD/A006583)

EERC 74-15 "Modeling and Identification in Nonlinear Structural Dynamics, I - One Degree of Freedom Models," by N. Distefano and A. Rath - 1974 (PB 241 548)

EERC 75-1 "Determination of Seismic Design Criteria for the Dumbarton Bridge Replacement Structure, Vol. I: Description, Theory and Analytical Modeling of Bridge and Parameters," by F. Baron and S.-H. Pang - 1975

EERC 75-2 "Determination of Seismic Design Criteria for the Dumbarton Bridge Replacement Structure, Vol. 2: Numerical Studies and Establishment of Seismic Design Criteria," by F. Baron and S.-H. Pang - 1975

EERC 75-3 "Seismic Risk Analysis for a Site and a Metropolitan Area," by C. S. Oliveira - 1975

EERC 75-4 "Analytical Investigations of Seismic Response of Short, Single or Multiple-Span Highway Bridges," by Ma-chi Chen and J. Penzien - 1975 (PB 241 454)

EERC 75-5 "An Evaluation of Some Methods for Predicting Seismic Behavior of Reinforced Concrete Buildings," by Stephen A. Mahin and V. V. Bertero - 1975

EERC 75-6 "Earthquake Simulator Study of a Steel Frame Structure, Vol. I: Experimental Results," by R. W. Clough and David T. Tang - 1975 (PB 243 981)

EERC 75-7 "Dynamic Properties of San Bernardino Intake Tower," by Dixon Rea, C.-Y. Liaw, and Anil K. Chopra - 1975 (AD/A008406)

EERC 75-8 "Seismic Studies of the Articulation for the Dumbarton Bridge Replacement Structure, Vol. I: Description, Theory and Analytical Modeling of Bridge Components," by F. Baron and R. E. Hamati - 1975

EERC 75-9 "Seismic Studies of the Articulation for the Dumbarton Bridge Replacement Structure, Vol. 2: Numerical Studies of Steel and Concrete Girder Alternates," by F. Baron and R. E. Hamati - 1975

EERC 75-10 "Static and Dynamic Analysis of Nonlinear Structures," by Digambar P. Mondkar and Graham H. Powell - 1975 (PB 242 434)

EERC 75-11 "Hysteretic Behavior of Steel Columns," by E. P. Popov, V. V. Bertero and S. Chandramouli - 1975

EERC 75-12 "Earthquake Engineering Research Center Library Printed Catalog" - 1975 (PB 243 711)

EERC 75-13 "Three Dimensional Analysis of Building Systems," Extended Version, by E. L. Wilson, J. P. Hollings and H. H. Dovey - 1975 (PB 243 989)

EERC 75-14 "Determination of Soil Liquefaction Characteristics by Large-Scale Laboratory Tests," by Pedro De Alba, Clarence K. Chan and H. Bolton Seed - 1975

EERC 75-15 "A Literature Survey - Compressive, Tensile, Bond and Shear Strength of Masonry," by Ronald L. Mayes and Ray W. Clough - 1975

EERC 75-16 "Hysteretic Behavior of Ductile Moment Resisting Reinforced Concrete Frame Components," by V. V. Bertero and E. P. Popov - 1975

EERC 75-17 "Relationships Between Maximum Acceleration, Maximum Velocity, Distance from Source, Local Site Conditions for Moderately Strong Earthquakes," by H. Bolton Seed, Ramesh Murarka, John Lysmer and I. M. Idriss - 1975

EERC 75-18 "The Effects of Method of Sample Preparation on the Cyclic Stress-Strain Behavior of Sands," by J. Paul Mulilis, Clarence K. Chan and H. Bolton Seed - 1975

EERC 75-19 "The Seismic Behavior of Critical Regions of Reinforced Concrete Components as Influenced by Moment, Shear and Axial Force," by B. Atalay and J. Penzien - 1975

EERC 75-20 "Dynamic Properties of an Eleven Story Masonry Building," by R. M. Stephen, J. P. Hollings, J. G. Bouwkamp and D. Jurukovski - 1975

EERC 75-21 "State-of-the-Art in Seismic Shear Strength of Masonry - An Evaluation and Review," by Ronald L. Mayes and Ray W. Clough - 1975

EERC 75-22 "Frequency Dependencies Stiffness Matrices for Viscoelastic Half-Plane Foundations," by Anil K. Chopra, P. Chakrabarti and Gautam Dasgupta - 1975

EERC 75-23 "Hysteretic Behavior of Reinforced Concrete Framed Walls," by T. Y. Wong, V. V. Bertero and E. P. Popov - 1975

EERC 75-24 "Testing Facility for Subassemblages of Frame-Wall Structural Systems," by V. V. Bertero, E. P. Popov and T. Endo - 1975

EERC 75-25 "Influence of Seismic History on the Liquefaction Characteristics of Sands," by H. Bolton Seed, Kenji Mori and Clarence K. Chan - 1975

EERC 75-26 "The Generation and Dissipation of Pore Water Pressures During Soil Liquefaction," by H. Bolton Seed, Phillippe P. Martin and John Lysmer - 1975

EERC 75-27 "Identification of Research Needs for Improving a Seismic Design of Building Structures," by V. V. Bertero - 1975

EERC 75-28 "Evaluation of Soil Liquefaction Potential during Earthquakes," by H. Bolten Seed, I. Arango and Clarence K. Chan 1975

EERC 75-29 "Representation of Irregular Stress Time Histories by Equivalent Uniform Stress Series in Liquefaction Analyses," by H. Bolton Seed, I. M. Idriss, F. Makdisi and N. Banerjee 1975

EERC 75-30 "FLUSH - A Computer Program for Approximate 3-D Analysis of Soil-Structure Interaction Problems," by J. Lysmer, T. Udaka, C.-F. Tsai and H. B. Seed - 1975

EERC 75-31 "ALUSH - A Computer Program for Seismic Response Analysis of Axisymmetric Soil-Structure Systems," by E. Berger, J. Lysmer and H. B. Seed - 1975

EERC 75-32 "TRIP and TRAVEL - Computer Programs for Soil-Structure Interaction Analysis with Horizontally Travelling Waves," by T. Udaka, J. Lysmer and H. B. Seed - 1975

EERC 75-33 "Predicting the Performance of Structures in Regions of High Seismicity," by J. Penzien - 1975

EERC 75-34 "Efficient Finite Element Analysis of Seismic Structure - Soil - Direction," by J. Lysmer, H. Bolton Seed, T. Udaka, R. N. Hwang and C.-F. Tsai - 1975

EERC 75-35 "The Dynamic Behavior of a First Story Girder of a Three-Story Steel Frame Subjected to Earthquake Loading," by Ray W. Clough and Lap-Yan Li - 1975

EERC 75-36 "Earthquake Simulator Study of a Steel Frame Structure, Volume II - Analytical Results," by David T. Tang - 1975

EERC 75-37 "ANSR-I General Purpose Computer Program for Analysis of Non-Linear Structural Response," by Digambar P. Mondkar and Graham H. Powell - 1975

EERC 75-38 "Nonlinear Response Spectra for Probabilistic Seismic Design and Damage Assessment of Reinforced Concrete Structures," by Masaya Murakami and Joseph Penzien - 1975