REPORT NO.
UC SESM 74-3

STRUCTURAL ENGINEERING AND STRUCTURAL MECHANICS
DEPARTMENT OF CIVIL ENGINEERING

NONSAP

A STRUCTURAL ANALYSIS PROGRAM FOR STATIC AND DYNAMIC RESPONSE OF NONLINEAR SYSTEMS

by

KLAUS-JÜRGEN BATHE
EDWARD L. WILSON
ROBERT H. IDING

FEBRUARY 1974

COLLEGE OF ENGINEERING
UNIVERSITY OF CALIFORNIA
BERKELEY CALIFORNIA

Structures and Materials Research
Department of Civil Engineering
Division of Structural Engineering
and Structural Mechanics

Report No. UCSESM 74-3

NONSAP

A STRUCTURAL ANALYSIS PROGRAM FOR STATIC AND DYNAMIC RESPONSE OF NONLINEAR SYSTEMS

by

Klaus-Jürgen Bathe

Edward L. Wilson

Robert H. Iding

Structural Engineering Laboratory
University of California
Berkeley, California

February 1974

ABSTRACT

The current version of the computer program NONSAP for linear and nonlinear, static and dynamic finite element analysis is described. The solution capabilities, the numerical techniques used, the finite element library, the logical construction of the program and storage allocations are discussed. The user's manual of the program is given.

Some sample solutions are included, which are standard data cases available with the program.

TABLE OF CONTENTS

TABLE OF CONTENTS (Continued)

TABLE OF CONTENTS (Continued)

Page

- PART C -

APPENDICES

- PART A -

DESCRIPTION OF NONSAP

1. INTRODUCTION

The endeavor to perform nonlinear analyses has steadily increased in recent years [1], [13], [17], [23]. The safety of a structure may be increased and the cost reduced if a nonlinear analysis can be carried out. Primarily, nonlinear analyses of complex structures have become possible through the use of electronic digital computers operating on discrete representations of the actual structure. A very effective discretization procedure has proven to be the finite element method [25]. Based on this method, various large-scale general purpose computer programs with nonlinear capabilities are now in use [11].

The development of a nonlinear finite element analysis program is a formidable challenge. The proper formulation of the nonlinear problem and its idealization to a representative finite element system demands a modern background in structural mechanics. For the solution of the equilibrium equations in space and time, stable and efficient numerical techniques need be employed. The efficiency of a nonlinear program depends largely on optimum usage of computer hardware and software where, specifically, the appropriate allocation of high- and low-speed storage is important.

The earliest attempts to obtain nonlinear analysis programs essentially involved simple modifications of estabished programs for linear analysis, much in the same way as the linear structural theory was modified to account for some nonlinearities. However, to analyze systems with large geometrical and material nonlinearities, the program should be designed specifically for the required iteration process and not be merely an extension of a linear

analysis program. Naturally, a linear analysis program should be flexible and easy to modify or extend; however, this applies even more to a nonlinear analysis program. In particular, it should be realized that a great deal of research is still required and currently pursued in the nonlinear static and dynamic analysis of complex structures. Therefore, unless the general nonlinear analysis code is easy to modify, it may be obsolete within a few years after completion.

The nonlinear analysis program NONSAP presented in this report is not an extension of the linear analysis program SAP [6], but rather a completely new development [2]. Program NONSAP is designed with two primary objectives. The first aim is the efficient solution of a variety of practical nonlinear problems with the current capabilities of nonlinear analysis procedures and computer equipment. The second objective is to have a program which can be used effectively in the various research areas pertaining to nonlinear analysis. Because of continuous improvements in nonlinear analysis procedures, both objectives are attained simultaneously by the development of an efficient, modular, and easily modifiable general analysis code. The program is designed for a general incremental solution of nonlinear problems, but naturally can also be used for linear analysis.

The structural systems to be analyzed may be composed of combinations of a number of different finite elements. The program presently contains the following element types:

(a) three-dimensional truss element

(b) two-dimensional plane stress and plane strain element

(c) two-dimensional axisymmetric shell or solid element

(d) three-dimensional solid element

(e) three-dimensional thick shell element

The nonlinearities may be due to large displacements, large strains, and nonlinear material behavior. The material descriptions presently available are:

for the truss elements

(a) linear elastic

(b) nonlinear elastic

for the two-dimensional elements

(a) isotropic linear elastic

(b) orthotropic linear elastic

(c) Mooney-Rivlin material

(d) elastic-plastic materials, von Mises or Drucker-Prager yield conditions

(e) variable tangent moduli model

(f) curve description model (with tension cut-off)

for the three-dimensional elements

(a) isotropic linear elastic

(b) curve description model

Program NONSAP is an in-core solver. The capacity of the program is essentially determined by the total number of degrees

of freedom in the system. However, all structure matrices are stored in compacted form, i.e. only nonzero elements are processed, resulting in maximum system capacity and solution efficiency.

The system response is calculated using an incremental solution of the equations of equilibrium with the Wilson θ or Newmark time integration scheme. Before the time integration is carried out, the constant structure matrices, namely the linear effective stiffness matrix, the linear stiffness, mass and damping matrices, whichever applicable, and the load vectors are assembled and stored on low-speed storage. During the step-by-step solution the linear effective stiffness matrix is updated for the nonlinearities in the system. Therefore, only the nonlinearities are dealt with in the time integration and no efficiency is lost in linear analysis.

The incremental solution scheme used corresponds to a modified Newton iteration. To increase the solution efficiency, the user can specify an interval of time steps in which a new effective stiffness matrix is to be formed and an interval in which equilibrium iterations are to be carried out.

There is practically no high-speed storage limit on the total number of finite elements used. To obtain maximum program capacity, the finite elements are processed in blocks according to their type and whether they are linear or nonlinear elements. In the solution low-speed storage is used to store all information pertaining to each block of finite elements, which, in the case of nonlinear elements, is updated during the time integration.

The purpose in this part of the report is to present briefly the general program organization, the current element library

and the numerical techniques used. The different options available for static and dynamic analyses are described. In the presentation emphasis is directed to the practical aspects of the program. For detailed information on the formulation of the continuum mechanics equations of motion, the finite element discretization, and the material models used, reference is made to [5].

2. THE INCREMENTAL EQUILIBRIUM EQUATIONS OF STRUCTURAL SYSTEMS

The incremental nodal point equilibrium equations for an assemblage of nonlinear finite elements has been derived in [5]. At time t we have

$$M\ {}^{t+\Delta t}\ddot{u} + C\ {}^{t+\Delta t}\dot{u} + {}^{t}K\ u = {}^{t+\Delta t}R - {}^{t}F \qquad (1)$$

where

M = constant mass matrix

C = constant damping matrix

${}^{t}K$ = tangent stiffness matrix at time t

${}^{t+\Delta t}R$ = external load vector applied at time $t+\Delta t$

${}^{t}F$ = nodal point force vector equivalent to the element stresses at time t

${}^{t+\Delta t}\dot{u}$, ${}^{t+\Delta t}\ddot{u}$ = vectors of nodal point velocities and accelerations at time $t+\Delta t$

u = vector of nodal point displacement increments from time t to time $t+\Delta t$, ie. $u = {}^{t+\Delta t}u - {}^{t}u$

As was discussed in [5], the solution of Eq. (1) yields, in general, approximate displacement increments u. To improve the solution accuracy, and, in some cases, to prevent the development of instabilities, it may be necessary to use equilibrium iteration in each or preselected time steps. In this case we consider the equilibrium equations

$$M\ {}^{t+\Delta t}\ddot{u}^{(i)} + C\ {}^{t+\Delta t}\dot{u}^{(i)} + {}^{t}K\ \Delta u^{(i)} = {}^{t+\Delta t}R - {}^{t+\Delta t}F^{(i-1)} \qquad i=1,2,3... \qquad (2)$$

where M, C, ${}^{t}K$, and ${}^{t+\Delta t}R$ are as defined above, and ${}^{t+\Delta t}\ddot{u}^{(i)}$, ${}^{t+\Delta t}\dot{u}^{(i)}$, ${}^{t+\Delta t}u^{(i)} = {}^{t+\Delta t}u^{(i-1)} + \Delta u^{(i)}$ are the approximations to the accelerations, velocities, and displacements obtained in iteration i. The first iteration, i.e. i=1 in Eq. (2), corresponds to the solution of Eq. (1), where $\Delta u^{(1)} = u$, ${}^{t+\Delta t}u^{(0)} = {}^{t}u$, ${}^{t+\Delta t}\ddot{u}^{(1)} = {}^{t+\Delta t}\ddot{u}$, ${}^{t+\Delta t}\dot{u}^{(1)} = {}^{t+\Delta t}\dot{u}$, ${}^{t+\Delta t}F^{(0)} = {}^{t}F$. The vector of nodal point forces ${}^{t+\Delta t}F^{(i-1)}$ is equivalent to the element stresses in the configuration corresponding to the displacements ${}^{t+\Delta t}u^{(i-1)}$. The approximations to the velocities and accelerations, ${}^{t+\Delta t}\dot{u}^{(i)}$ and ${}^{t+\Delta t}\ddot{u}^{(i)}$, respectively, depend on the time integration scheme used [3].

It should be noted that the solution scheme used in Eq. (2) corresponds to a modified Newton iteration [16] [25].

In program NONSAP, the Wilson θ-method or the Newmark method is used for the step-by-step solution [3] [15]. Table 1 summarizes the algorithm in linear or nonlinear, static or dynamic analysis [5]. The specific operations performed during the step-by-step solution are discussed in Chapter 7.

2.1 Element to Structure Matrices and Force Vectors

The structure matrices in Table 1 are formed by direct addition of the element matrices and vectors [25]; for example

$$K = \sum K_m \tag{3}$$

where K_m is the stiffness matrix of the m'th element. Although K_m is formally of the same order as K, only those terms in K_m

TABLE 1 SUMMARY OF STEP-BY-STEP INTEGRATION

- - INITIAL CALCULATIONS - - -

1. Form linear stiffness matrix K, mass matrix M and damping matrix C; initialize ${}^{0}u$, ${}^{0}\dot{u}$, ${}^{0}\ddot{u}$

2. Calculate the following constants:

   tol ≤ 0.01 ; nitem ≥ 3 ; in static analysis $\theta = 1$ and go to 3.

   Wilson θ-method: $\theta \geq 1.37$, usually $\theta = 1.4$, $\tau = \theta\Delta t$

   $a_0 = 6/\tau^2$ $\quad a_1 = 3/\tau$ $\quad a_2 = 2a_1$ $\quad a_3 = 2$

   $a_4 = 2$ $\quad a_5 = \tau/2$ $\quad a_6 = a_0/\theta$ $\quad a_7 = -a_2/\theta$

   $a_8 = 1 - 3/\theta$ $\quad a_9 = \Delta t/2$ $\quad a_{10} = \Delta t^2/6$

   Newmark method: $\theta = 1.0$, $\delta \geq 0.50$, $\alpha \geq 0.25(0.5 + \gamma)^2$, $\tau = \Delta t$

   $a_0 = 1/(\alpha\Delta t^2)$ $\quad a_1 = \delta/(\alpha\Delta t)$ $\quad a_2 = 1/(\alpha\Delta t)$ $\quad a_3 = 1/(2\alpha) - 1$

   $a_4 = \delta/\alpha - 1$ $\quad a_5 = \Delta t(\delta/\alpha - 2)/2$ $\quad a_6 = a_0$ $\quad a_7 = -a_2$

   $a_8 = -a_3$ $\quad a_9 = \Delta t(1 - \delta)$ $\quad a_{10} = \delta\Delta t$

3. Form effective linear stiffness matrix: $\hat{K} = K + a_0 M + a_1 C$

4. In linear analysis triangularize $\hat{K}$

TABLE 1 (cont'd.)

- - FOR EACH TIME STEP - - -

A. IN LINEAR ANALYSIS

(i) Form effective load vector

$$^{t+\tau}\hat{R} = {}^{t}R + \theta({}^{t+\Delta t}R - {}^{t}R) + M(a_0\,{}^{t}u + a_2\,{}^{t}\dot{u} + a_3\,{}^{t}\ddot{u}) + C(a_1\,{}^{t}u + a_4\,{}^{t}\dot{u} + a_5\,{}^{t}\ddot{u})$$

(ii) Solve for displacement increments:

$$\hat{K}\,{}^{t+\tau}u = {}^{t+\tau}\hat{R}; \qquad u = {}^{t+\tau}u - {}^{t}u$$

(iii) Go to C.

B. IN NONLINEAR ANALYSIS

(i) If a new stiffness matrix is to be formed, update $\hat{K}$ for nonlinear stiffness effects to obtain $^{t}\hat{K}$; triangularize $^{t}\hat{K}$:

$$^{t}\hat{K} = LDL^{T}$$

(ii) Form effective load vector:

$$^{t+\tau}\hat{R} = {}^{t}R + \theta({}^{t+\Delta t}R - {}^{t}R) + M(a_2\,{}^{t}\dot{u} + a_3\,{}^{t}\ddot{u}) + C(a_4\,{}^{t}\dot{u} + a_5\,{}^{t}\ddot{u}) - {}^{t}F$$

(iii) Solve for displacement increments using latest D,L factors:

$$LDL^{T}\,u = {}^{t+\tau}\hat{R}$$

TABLE 1 (continued)

B. (iv) If required, iterate for dynamic equilibrium; then initialize $u^{(0)} = u$, $i = 0$

(a) $i = i + 1$

(b) Calculate (i-1)st approximation to accelerations, velocities, and displacements:

$$^{t+\tau}\ddot{u}^{(i-1)} = a_0\, u^{(i-1)} - a_2\, {}^{t}\dot{u} - a_3\, {}^{t}\ddot{u} \; ; \quad {}^{t+\tau}\dot{u}^{(i-1)} = a_1\, u^{(i-1)} - a_4\, {}^{t}\dot{u} - a_5\, {}^{t}\ddot{u} \; ;$$

$$^{t+\tau}u^{(i-1)} = u^{(i-1)} + {}^{t}u$$

(c) Calculate (i-1)st effective out-of-balance loads:

$$^{t+\tau}\hat{R}^{(i-1)} = {}^{t}R + \theta({}^{t+\Delta t}R - {}^{t}R) - M\, {}^{t+\tau}\ddot{u}^{(i-1)} - C\, {}^{t+\tau}\dot{u}^{(i-1)} - {}^{t+\tau}F^{(i-1)}$$

(d) Solve for i'th correction to displacement increments:

$$LDL^{T}\, \Delta u^{(i)} = {}^{t+\tau}\hat{R}^{(i-1)}$$

(e) Calculate new displacement increments:

$$u^{(i)} = u^{(i-1)} + \Delta u^{(i)}$$

(f) Iteration convergence if $||\Delta u^{(i)}||_2 \,/\, ||u^{(i)} + {}^{t}u||_2 < \text{tol}$

If convergence: $u = u^{(i)}$ and go to C;

If no convergence and $i <$ nitem: go to (a); otherwise restart using new stiffness matrix and/or a smaller time step size.

TABLE 1 (cont'd)

C. CALCULATE NEW ACCELERATIONS, VELOCITIES, AND DISPLACEMENTS

Wilson θ-method:

$$^{t+\Delta t}\ddot{u} = a_6\, u + a_7\, {}^{t}\dot{u} + a_8\, {}^{t}\ddot{u}$$

$$^{t+\Delta t}\dot{u} = {}^{t}\dot{u} + a_9\, (^{t+\Delta t}\ddot{u} + {}^{t}\ddot{u})$$

$$^{t+\Delta t}u = {}^{t}u + \Delta t\, {}^{t}\dot{u} + a_{10}\, (^{t+\Delta t}\ddot{u} + 2\, {}^{t}\ddot{u})$$

Newmark method:

$$^{t+\Delta t}\ddot{u} = a_6\, u + a_7\, {}^{t}\dot{u} + a_8\, {}^{t}\ddot{u}$$

$$^{t+\Delta t}\dot{u} = {}^{t}\dot{u} + a_9\, {}^{t}\ddot{u} + a_{10}\, {}^{t+\Delta t}\ddot{u}$$

$$^{t+\Delta t}u = {}^{t}u + u$$

which pertain to the element degrees of freedom are nonzero. The addition of the element matrices and vectors can, therefore, be performed by using the element matrices in compact form together with identification arrays which relate element to structure degrees of freedom.

In program NONSAP, either a diagonal or consistent mass matrix may be used. In addition, concentrated masses corresponding to selected degrees of freedom can be specified. Rayleigh damping is assumed with the addition of concentrated nodal point dampers. The assumptions used in lumped mass analysis and Raleigh damping have been discussed at various occasions [7] [25].

2.2 Boundary Conditions

If a displacement component is zero, the corresponding equation is not retained in the structure equilibrium equations, Eq. (2), and the corresponding element stiffness and mass terms are disregarded. If a non-zero displacement is to be specified at a degree of freedom i, say $u_i = x$, the equation

$$k\, u_i = k\, x \tag{4}$$

need be added into Eq. (2), where $k >> k_{ii}$. Therefore, the solution of Eq. (2) must give $u_i = x$. Physically, this can be interpreted as adding at the degree of freedom "i" a spring of large stiffness k and specifying a load, which, because of the relatively flexible structure at this degree of freedom, produces the required displacement x. This approach simplifies programming problems which are normally associated with specifying displacements.

A special boundary element could have been incorporated into NONSAP [6]. However, in the current version of NONSAP only translational displacements are considered (since only isoparametric elements are available, see Chapter 4). Therefore, nonzero displacement boundary conditions can be specified by using the truss element to provide the stiffness k in Eq. (4) and applying the load kx.

3. PROGRAM ORGANIZATION

The complete solution process in program NONSAP is divided into three distinct phases:

1) Input phase

The input phase consists of three steps:

a) The control information and the nodal point input data are read and generated by the program. In this phase the equation numbers for the active degrees of freedom at each nodal point are established.

b) The externally applied load vectors for each time (load) step are calculated and stored on tape (or other low-speed storage).

c) The element data are read and generated, the element connection arrays are calculated and all element information is stored on tape.

2) Assemblage of Constant Structure Matrices

Before the solution of Eq. (2) is carried out, the linear structure stiffness, mass, and damping matrices are assembled and stored on tape (or other low-speed storage). In addition, the effective linear structure stiffness matrix is calculated and stored (see Table 1).

3) Step-by-Step Solution

During this phase the solution of Eq. (2) is obtained at all time points. In addition to the displacement, velocity, and acceleration vectors (whichever applicable), the element stresses are calculated and printed. Before the

time integration is performed, the lowest frequencies and corresponding mode shapes may be calculated.

The details of the step-by-step solution are presented in Chapter 7.

It need be noted that these basic steps are independent of the element type used and are the same for either a static or dynamic analysis. However, only those matrices actually required in the analysis are assembled. For example, no mass and damping matrices are calculated in a static analysis.

Program NONSAP is an in-core solver and the high-speed storage capacity of the program is determined by the maximum storage that is required during the three phases. Figures 1 to 3 show the dynamic storage allocations used in each phase. We note that, in general, maximum high speed storage is required during the step-by-step solution. However, in some cases the storage required during the input phase may govern the system size that can be solved.

Figures 1 to 3 show that the lowest high speed storage locations are reserved throughout the solution for element group information. For the analysis, the finite elements of the complete assemblage need be divided into element groups according to their type, the nonlinear formulation (see Chapter 4), and the material models used (see Chapter 5). One element group must consist of the same type of elements, must use one nonlinear formulation and only one specific material model. The data pertaining to each individual element group need fit into the NUMEST storage locations, Fig. 1. Therefore, the minimum that

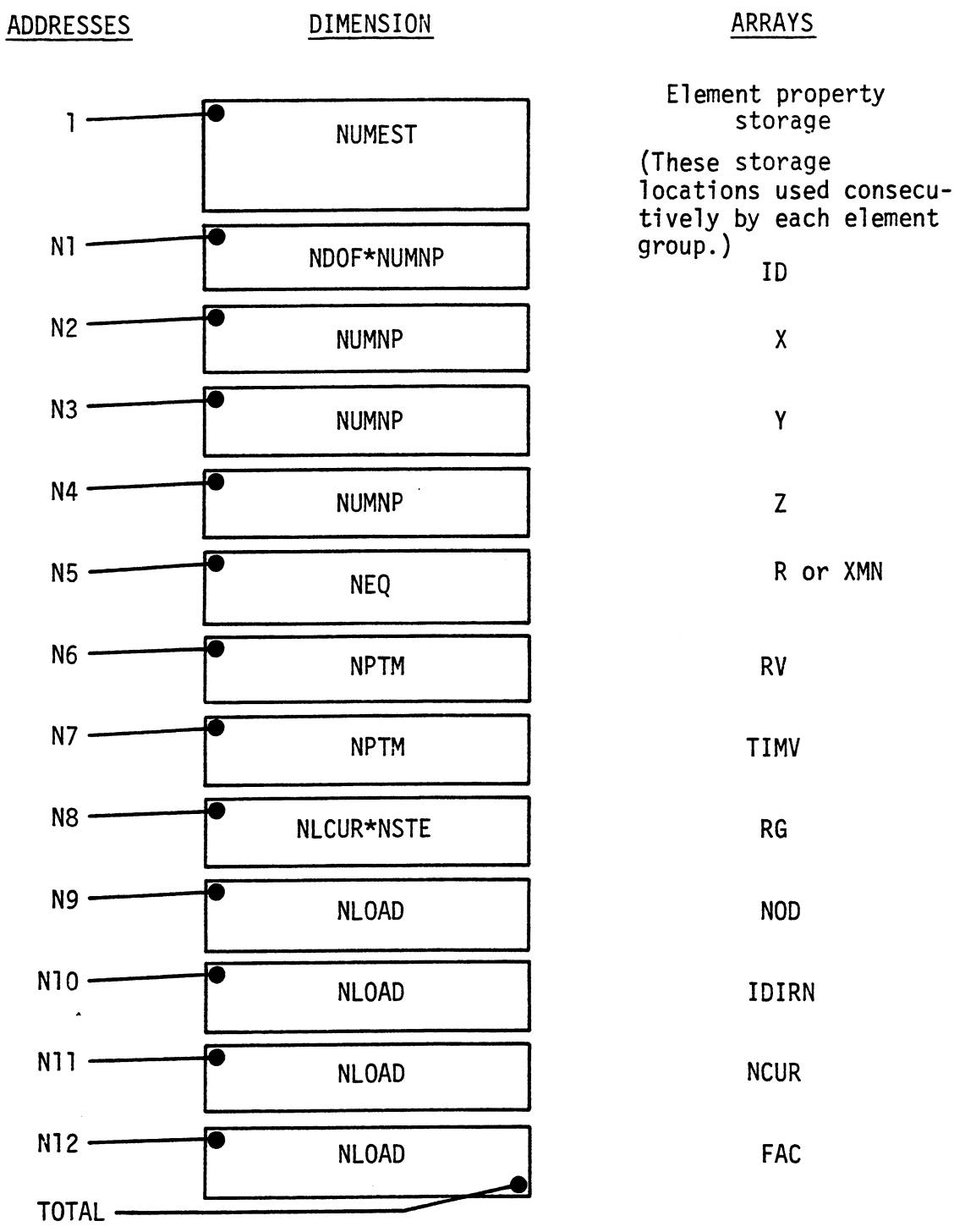


FIGURE I STORAGE ALLOCATION DURING INPUT PHASE

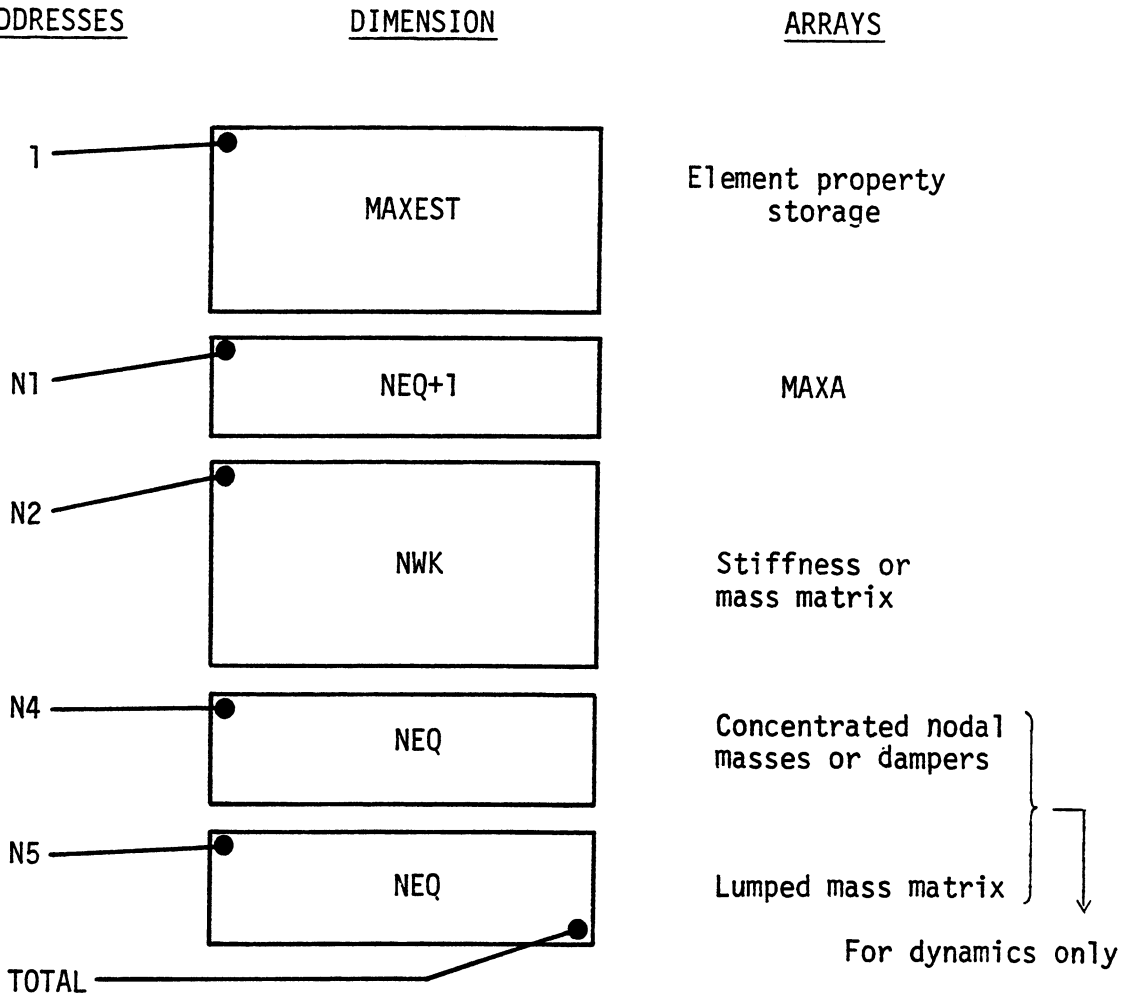
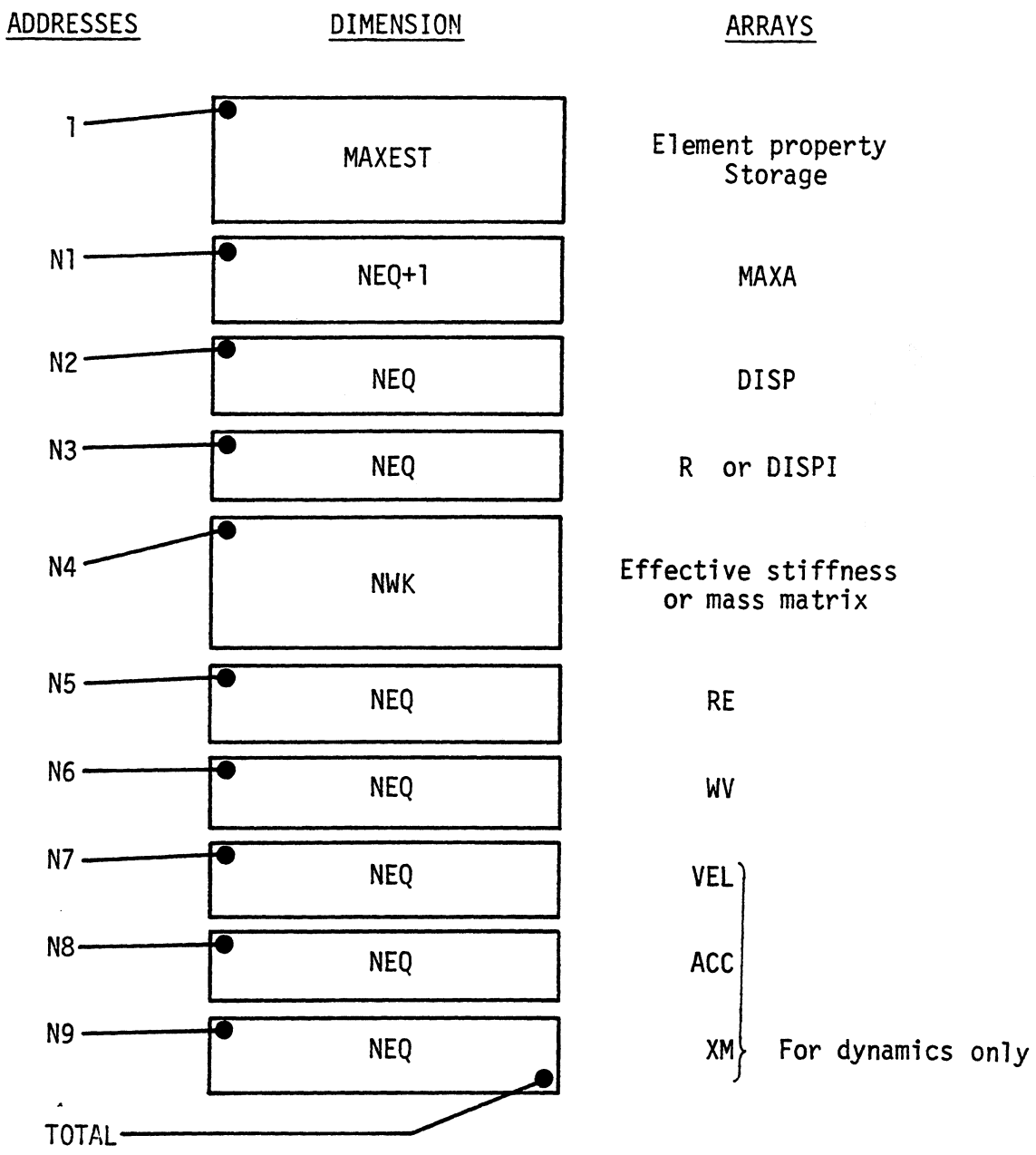


FIGURE 2 STORAGE ALLOCATION DURING MATRIX ASSEMBLAGE PHASE



Note: N7 = N5 + NDOF*NUMNP if NDOF*NUMNP > 2*NEQ

FIGURE 3 STORAGE ALLOCATION DURING TIME INTEGRATION

NUMEST should be specified is equal to the locations required to store the data pertaining to any one of the elements.

The use of element groups reduces input-output transfers during the solution process, since the data of the elements is retrieved in blocks during the solution of Eq. (2) and element stress calculations (see Chapter 7). Usually, NUMEST is some reasonable fraction of the total number of high speed storage locations available, and is not reset for each problem. During the input phase the program calculates the exact number of high speed storage locations required for each element group, and NUMEST is reset to MAXEST, which is the actual maximum of locations needed, see Figs. 2 and 3. Therefore, an optimum of high speed storage allocation is obtained during the step-by-step solution. Figure 4 shows the tape storage used for the element group information.

To further improve high speed storage capacity, NONSAP is an overlaid program. The overlay structure has been chosen to correspond to the three phases of execution listed above, the element library, the material models available, and the frequency calculation option. Figure 5 shows the overlay structure of NONSAP.

3.1 Nodal Point Input Data and Degrees of Freedom

The nodal point data read during the first step of the input phase consists of the boundary condition codes (stored in the ID array) and the global X, Y, Z coordinates of each nodal point. The same input is also required for program SAP [6]. A maximum of three boundary condition codes need currently be defined, since

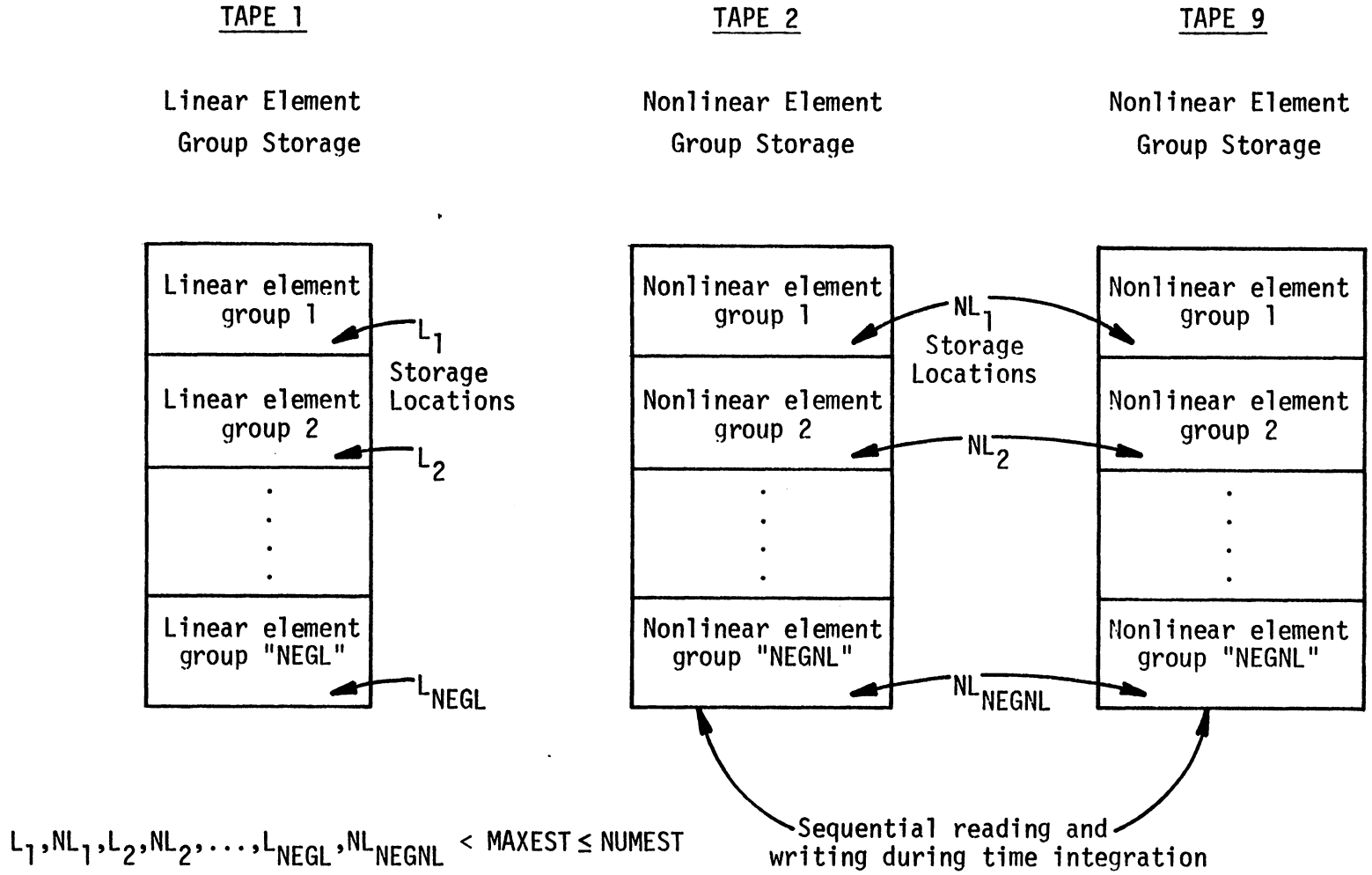


FIGURE 4 AUXILIARY STORAGE ORGANIZATION FOR ELEMENT GROUP INFORMATION

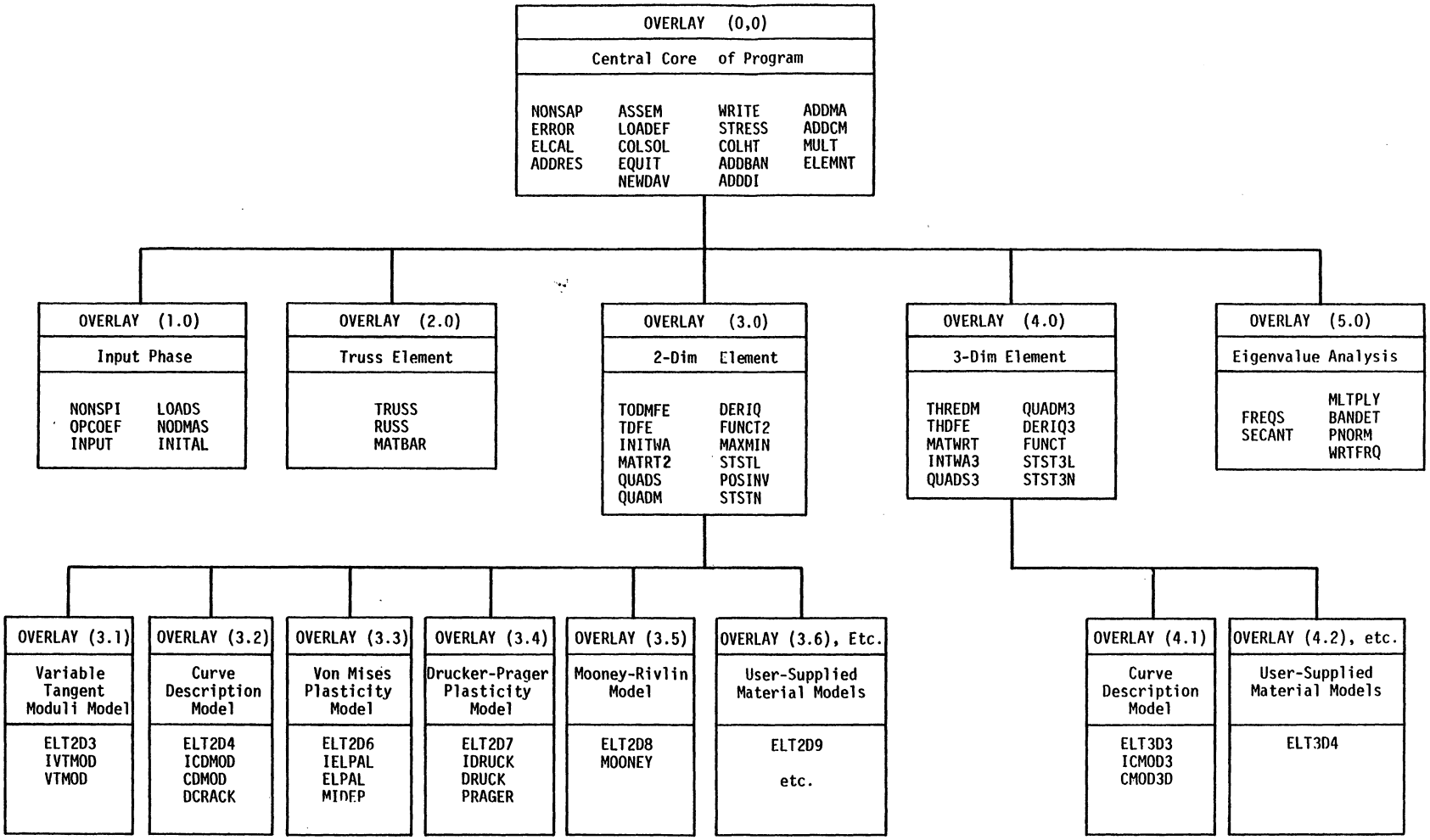


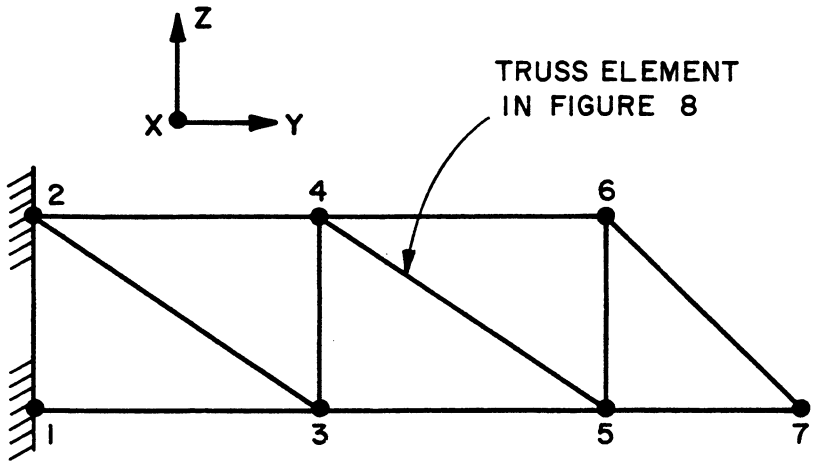
FIGURE 5 OVERLAY STRUCTURE OF NONSAP

a finite element node can have at most three (translational) degrees of freedom (see Chapter 4). As shown in Fig. 1, all nodal point data is retained in high speed storage during the complete input phase, i.e. during the calculation of the externally applied load vectors and the reading and generating of the element group information.

It need be noted that the user should allow only those degrees of freedom which are compatible with the elements connected to a nodal point. The program can deal with a maximum of six possible degrees of freedom (3 translations and 3 rotations) at each nodal point, and all non-active degrees of freedom need be deleted. Specifically, a "1" in the ID array denotes that no equation shall be associated with the degree of freedom, whereas a "0" indicates that this is an active degree of freedom [6]. Figure 6 shows for the simple truss structure the ID array as it was read and/or generated by the program. Once the complete ID and X, Y,Z arrays have been obtained, equation numbers are associated with all active degrees of freedom, i.e., the zeroes in the ID array are replaced by corresponding equation numbers, and each one is replaced by a zero, as shown in Fig. 7 for the simple truss example.

3.2 Calculation of External Load Vectors

The loading in the analysis can consist only of concentrated nodal point loading, i.e. all distributed body or surface loading must be transformed to nodal point loading prior to using NONSAP. The load corresponding to a degree of freedom is assumed to vary



NODAL POINT LAYOUT OF TRUSS

ID =

	1	2	3	4	5	6
1	1	1	1	1	1	1
2	1	1	1	1	1	1
3	1	0	0	1	1	1
4	1	0	0	1	1	1
5	1	0	0	1	1	1
6	1	0	0	1	1	1
7	1	0	0	1	1	1

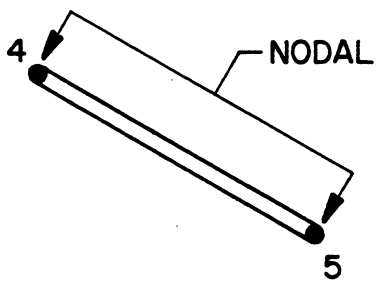
DEGREES OF FREEDOM

NODAL POINT NUMBERS

FIGURE 6 NODAL POINT LAYOUT OF TRUSS EXAMPLE AND ID-ARRAY AS READ AND/OR GENERATED

$$ID = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 2 & 0 & 0 & 0 \\ 0 & 3 & 4 & 0 & 0 & 0 \\ 0 & 5 & 6 & 0 & 0 & 0 \\ 0 & 7 & 8 & 0 & 0 & 0 \\ 0 & 9 & 10 & 0 & 0 & 0 \end{bmatrix}$$

FIGURE 7 ID ARRAY OF TRUSS EXAMPLE AFTER ALLOCATION OF EQUATION NUMBERS TO ACTIVE DEGREES OF FREEDOM



$$LM = \begin{bmatrix} 0 \\ 3 \\ 4 \\ 0 \\ 5 \\ 6 \end{bmatrix}$$

FIGURE 8 CONNECTION ARRAY (VECTOR LM) FOR A TYPICAL ELEMENT OF THE TRUSS EXAMPLE

with time as expressed by a time function and a load multiplier, both defined in the input.

3.3 Read-in of Element Data

In the last step of the input phase, element information for each element group is read and generated. Specifically, the element coordinates, the material properties, and the element connection arrays are established. Also, working vectors which store required element strains, stresses and other variables are initialized. For each element group this information is processed in the first NUMEST high speed storage locations and then written together in one block on secondary storage. During the next phases of the solution, therefore, the required element data can be read in blocks, sequentially one block at a time, into the same high speed storage locations.

The element connection array, i.e. vector LM of an element, is established from the ID matrix and the specified nodal points of the assemblage pertaining to the element. The connection array for a typical element of the truss example is shown in Figure 8.

It should be noted that the reading and generation of the element data of one group requires only one call of the specific element overlay needed since all elements in one group are of the same kind. After all element information has been established, the ID and X,Y,Z arrays are no longer required, and the corresponding storage area is used for the formation of the constant structure matrices and later for the solution of the equations of equilibrium.

3.4 Formation of Constant Structure Matrices

All structure matrices which are not time dependent are calculated before the time integration is carried out. At this stage it is necessary to distinguish between the different kinds of analyses possible, namely whether a linear or nonlinear, static or dynamic analysis is required. The storage allocation during this phase was given in Fig. 2, where it is shown that all required linear structure matrices are assembled using the same high-speed storage locations.

Figure 9 lists the sequence of assemblage and the tape storage used for the constant structure matrices corresponding to the different analyses. Note that only those matrices to be used later in the step-by-step solution are stored on tape. The assemblage of a structure matrix is effected by reading the data of all required element groups in succession, and by calculating and adding the element matrices to the structure matrix, as was discussed in Section 2.1.

It should be noted that in linear analysis the structure stiffness or effective stiffness matrix is triangularized before storage on tape. In the step-by-step solution only forward reductions and back-substitutions of the (effective) load vectors are then required (see Chapter 7).

3.5 The Compacted Storage Scheme

An important aspect is the efficient storage of the structure matrices and an effective solution of the equilibrium equations. The storage scheme need be optimized in order to obtain maximum

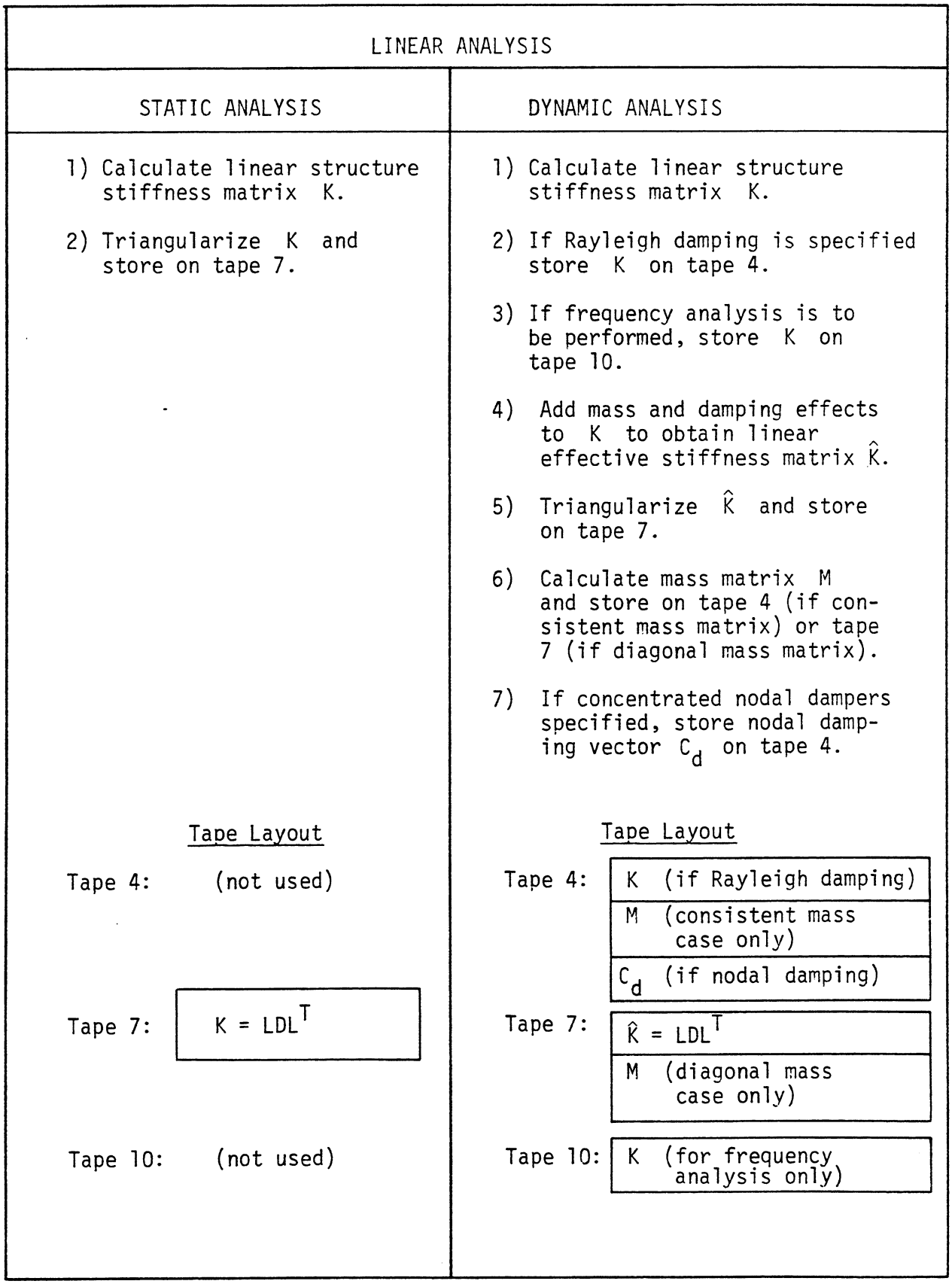


FIGURE 9 ASSEMBLAGE OF CONSTANT STRUCTURE MATRICES

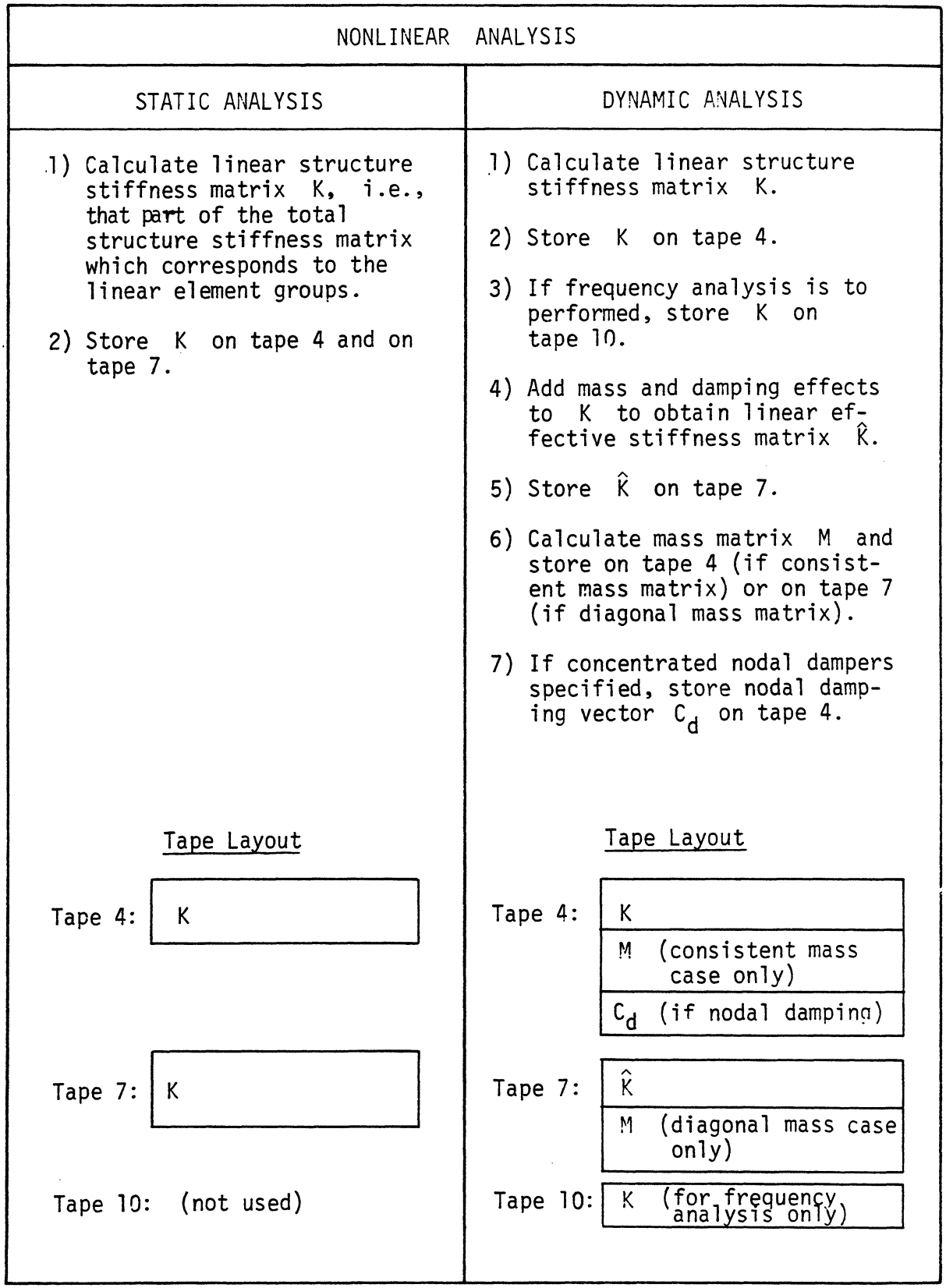


FIGURE 9 (CONT) ASSEMBLAGE OF CONSTANT STRUCTURE MATRICES

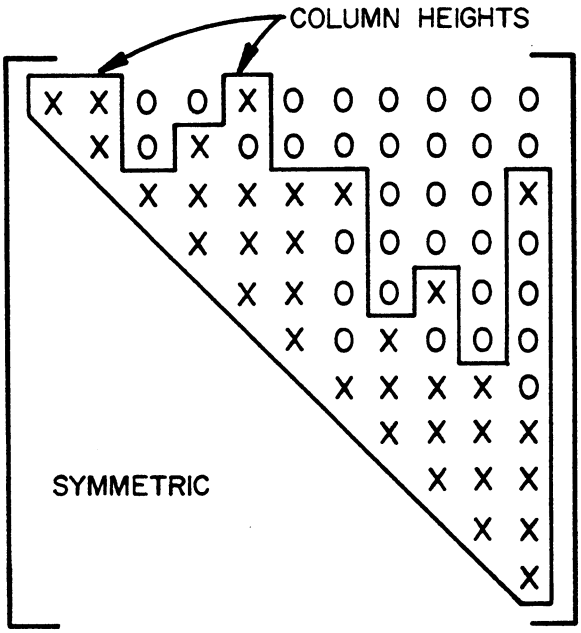
capacity. The effective solution of the equations is necessary to reduce total solution cost.

In program NONSAP a compacted storage scheme is used in which all structure matrices are stored as one-dimensional arrays, and only the elements below the skyline of a matrix are processed [24]. Figure 10 shows, as an example, the element pattern in a typical stiffness matrix before and after triangularization. It should be noted that, in general, zero elements within the skyline do not remain zero during the equation solution and must be stored, whereas all elements outside the skyline do not need to be considered. Therefore, by storing and processing in the equation solution only the elements within the skyline, a minimum number of high speed storage locations is used.

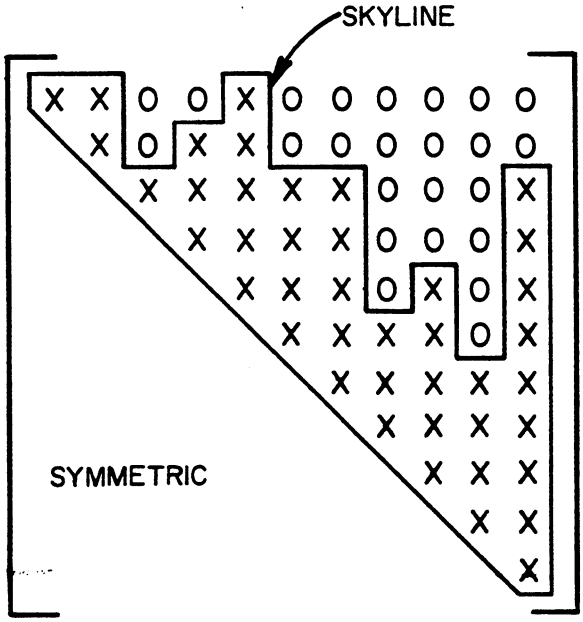
3.6 Equation Solution

The solution of equations is obtained using the linear equation solver COLSOL. This subroutine uses Gauss elimination on the positive definite symmetrical system of equations [24]. The algorithm performs a minimum number of arithmetic operations, since only the elements within the skyline of the matrix are processed, i.e. there are no operations with elements that remain zero during the solution. The algorithm is used in all analysis types, i.e. in linear, nonlinear, static or dynamic analysis, and consists of the LDL^T decomposition of the stiffness matrix (or effective stiffness matrix), and the reduction and back-substitution of the (effective) load vector. For example, in linear static analysis, the equations are $Ku = R$ and the program

X = NONZERO ELEMENT
O = ZERO ELEMENT



ELEMENTS IN ORIGINAL STIFFNESS MATRIX



ELEMENTS IN DECOMPOSED STIFFNESS MATRIX

FIGURE 10 TYPICAL ELEMENT PATTERN IN A STIFFNESS MATRIX

calculates

$$K = LDL^T \tag{5}$$

$$L\,V = R \tag{6}$$

$$D\,L^T\,U = V \tag{7}$$

where L and D are a lower triangular and a diagonal matrix, respectively.

4. THE ELEMENT LIBRARY

In the current version of program NONSAP all finite elements are isoparametric (or subparametric) elements [25].

Corresponding to the nonlinearities in the system, four different analysis procedures may be considered for a finite element:

Linear Elastic Analysis

The displacements of the element are assumed to be negligibly small and the strains infinitesimal. The material is isotropic or orthotropic linear elastic.

Materially Nonlinear Only Analysis

The displacements of the element are negligibly small, and the strains are infinitesimal. The material stress-strain description is nonlinear.

Total Lagrangian Formulation

The element may experience large displacements and large strains. The material stress-strain relationship is linear or nonlinear.

Updated Lagrangian Formulation

The element may experience large displacements and large strains. The material stress-strain description is linear or nonlinear.

The linear elastic analysis does not allow for any nonlinearities, whereas the materially nonlinear only analysis includes material nonlinearities, but no geometric nonlinearities [5]. The different linear and nonlinear material models currently

available in NONSAP are described in Chapter 5. The total Lagrangian and updated Lagrangian formulations may include all nonlinearities, and which formulation should be employed depends essentially on the definition of the material model used, as described in the next chapter.

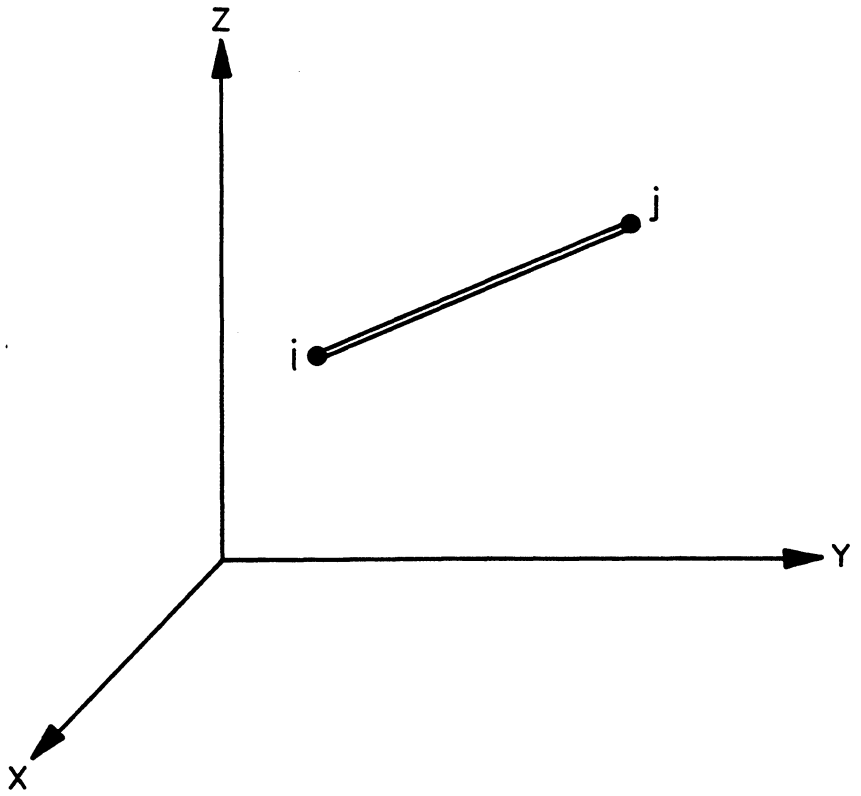
In the following, the finite elements currently available in NONSAP are briefly described. It should be noted that a particular element group must consist of finite elements of the same type, described by one of the four element formulations above, and must use one material model only. Since all four formulations and all material models have not been implemented for all element types, it is important to identify the nonlinear formulations and material models currently available in NONSAP for a specific element type, as illustrated in Figs. 11 to 13.

4.1 Truss Element

A three-dimensional truss element is available in NONSAP. The element is assumed to have constant area, and may be used in linear elastic analysis, materially nonlinear and/or large displacement geometrically nonlinear analysis. In the large displacement analysis, the updated Lagrangian formulation is used, but small strains are assumed in the calculation of element stresses.

The nonlinear elastic model is described in Chapter 5.

As noted earlier, the truss element can be used to specify nonzero boundary displacements [6].



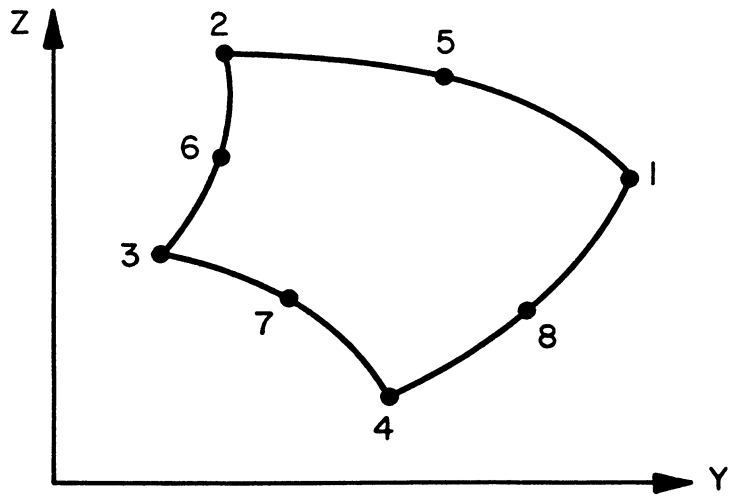
AVAILABLE
NONLINEAR FORMULATIONS

1. LINEAR ANALYSIS
2. MATERIALLY NONLINEAR ONLY
3. UPDATED LAGRANGIAN WITH LARGE DISPLACEMENTS BUT SMALL STRAINS

AVAILABLE
MATERIAL MODELS

1. LINEAR ELASTIC
2. NONLINEAR ELASTIC

FIGURE 11 TRUSS ELEMENT



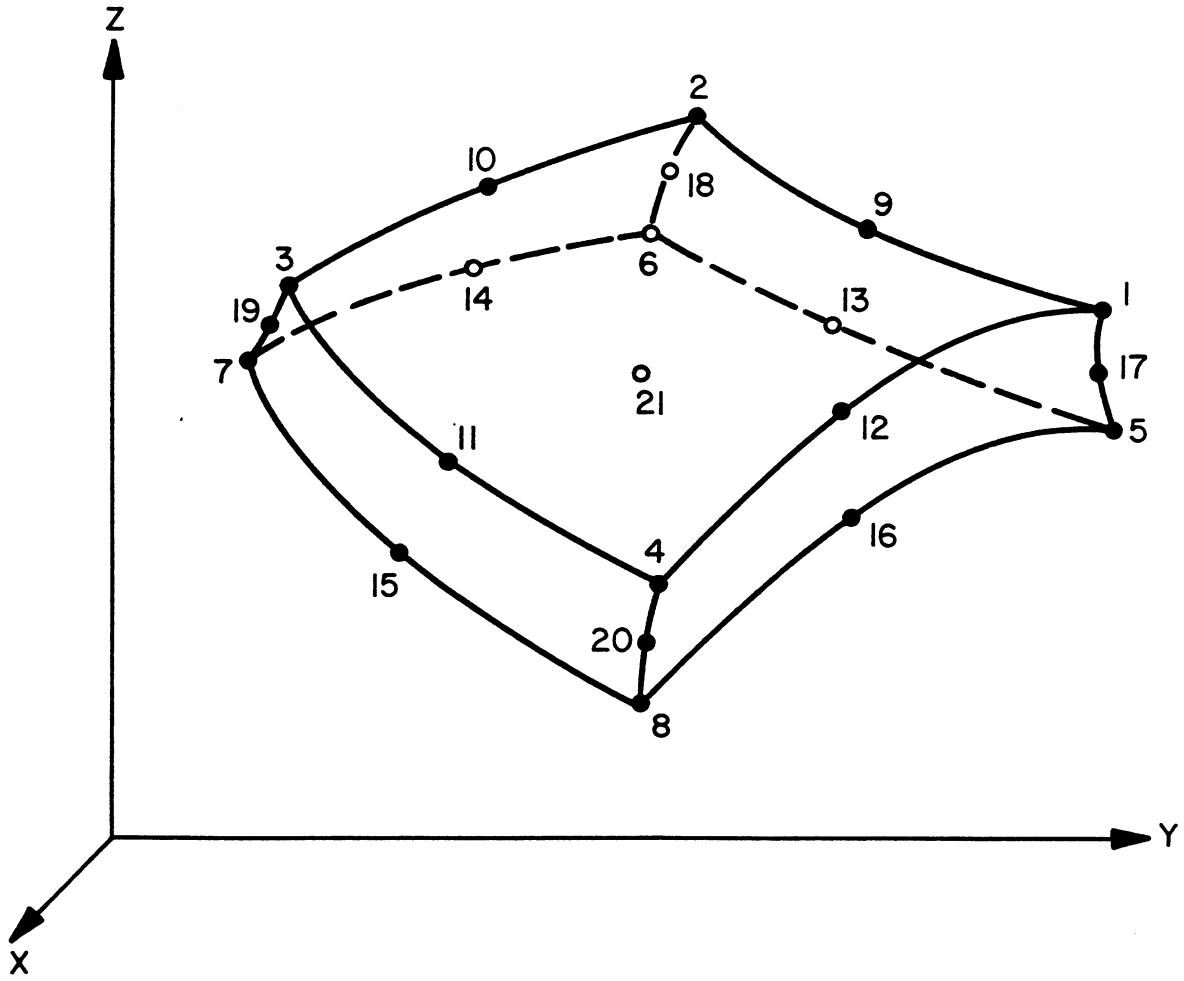
AVAILABLE
NONLINEAR FORMULATIONS

1. LINEAR ANALYSIS
2. MATERIALLY NONLINEAR ONLY
3. UPDATED LAGRANGIAN
4. TOTAL LAGRANGIAN

AVAILABLE
MATERIAL MODELS

1. LINEAR ISOTROPIC ELASTIC
2. LINEAR ORTHOTROPIC ELASTIC
3. VARIABLE TANGENT MODULI MODEL
4. CURVE DESCRIPTION NONLINEAR MODEL (WITH OR WITHOUT TENSION CUT-OFF ASSUMPTION)
5. PLASTICITY MODELS (VON MISES OR DRUCKER-PRAGER YIELD CONDITION)
6. NONLINEAR, ISOTROPIC INCOMPRESSIBLE ELASTIC (MOONEY-RIVLIN MATERIAL)

FIGURE 12 TWO-DIMENSIONAL PLANE STRESS, PLANE STRAIN AND AXISYMMETRIC ELEMENTS



AVAILABLE
NONLINEAR FORMULATIONS

1. LINEAR ANALYSIS
2. MATERIALLY NONLINEAR ONLY

AVAILABLE
MATERIAL MODELS

1. LINEAR ISOTROPIC ELASTIC
2. CURVE DESCRIPTION NONLINEAR MODEL

FIGURE 13 THREE-DIMENSIONAL SOLID AND THICK SHELL ELEMENT

4.2 Plane Stress and Plane Strain Element

A variable-number-node isoparametric finite element is available for two-dimensional plane stress and plane strain analysis. The element may have from 3 to 8 nodes, where any one of the nodes 5 to 8 can be omitted. The variable-number-node option allows effective modelling from coarse to finer finite element meshes.

The plane stress and plane strain element can be used in all four formulations. The material models available are described in the next chapter.

4.3 Axisymmetric Shell or Solid Element

The variable-number-node element described above is also available for axisymmetric two-dimensional analysis of shells or solids (with axisymmetric loading).

4.4 Three-Dimensional Solid or Thick Shell Element

A general three-dimensional isoparametric element with a variable number of nodes from 8 to 21 can be used. The first 8 nodes are the corner nodes of the element, nodes 9 to 20 correspond to midside nodes and node 21 is a center node. The element can be used for three-dimensional analysis of solids and thick shells. As for the two-dimensional elements, the possibility of choosing different element node configurations allows effective finite element modelling.

The three-dimensional element can currently only be used in linear isotropic analysis and in nonlinear analysis with material nonlinearities only.

5. THE MATERIAL MODELS

The largest number of material models is available for two-dimensional analysis, since it is anticipated that the two-dimensional elements will be used in most analyses. For the same reason, also all three nonlinear formulations can be used for the two-dimensional elements.

All material models available in NONSAP are discussed in [5].

5.1 Truss-Element Material Models

The truss element material behavior can be described by means of two models.

Linear Elastic Material

The material can be linear elastic defined by Young's modulus only.

Nonlinear Elastic Material

The nonlinear elastic material behavior is defined by specifying the stress as a piece-wise linear function of the current (infinitesimal) strain. Thus, the total stress and the tangent modulus are directly defined in terms of the total strain.

5.2 Two-Dimensional Element Material Models

The stress-strain relationship of the two dimensional elements can be described by various linear and nonlinear material models. In the definition of a material model, it may have been assumed that a specific nonlinear formulation is used. The application of the different material models is discussed in [5], where the assumptions

used are pointed out. (Table X.1 in the Appendix summarizes the formulations with which the material models can be used.)

Isotropic and Orthotropic Linear Elastic Material

The stress-strain relationships are defined by means of the constant Young's moduli and Poisson's ratios [25].

Mooney-Rivlin Material Model

A hyperelastic incompressible material model is available for the analysis of rubber-like materials [10] [16]. The stress-strain relationship is defined using the Mooney-Rivlin material constants. In NONSAP the model can only be used in plane stress analysis.

Elastic-Plastic Material Models

Elastic-plastic analysis using a plastic potential function can be carried out. The plasticity relations available are those based on the use of the von Mises yield condition and the Drucker-Prager yield condition. Both forms of describing material behavior have been employed extensively in practice [13] [19] [20]. Using the von Mises criterion, linear isotropic hardening can be assumed. In analysis using the Drucker-Prager yield condition, the material is assumed to be elastic perfectly plastic.

The Variable Tangent Moduli Model

The variable tangent moduli model is available for the analysis of geological materials [14]. The model describes an isotropic material, in which the bulk and shear moduli are

functions of the stress and strain invariants. The functional relationship used replaces an explicit yield condition.

The Curve Description Model

The curve description model is used in essentially the same way as the variable tangent moduli model. The model also describes the response of geological materials. In the model, the instantaneous bulk and shear moduli are defined by piecewise linear functions of the current volume strain. An explicit yield condition is not used, and whether the material is loading or unloading is defined by the history of the volume strain only.

In the analysis of some problems, tensile stresses due to applied loading cannot exceed the gravity in-situ pressure. In such conditions the model can be used to simulate tension cut-off, i.e., the material model assumes reduced stiffness in the direction of the tensile stresses which exceed in magnitude the gravity pressure.

5.3 Three-Dimensional Element Material Models

In principle, most two-dimensional models would also be applicable in three-dimensional analysis. However, in the current version of NONSAP, only the isotropic linear elastic model and the curve description model (without tension cut-off capability) are available.

6. EIGENSYSTEM SOLUTION

In dynamic analysis it is necessary to select a suitable time step Δt. The time increment must be small enough for solution accuracy, but for a cost effective solution, it should not be unnecessarily small. In order to estimate an appropriate time step, it may be necessary to solve for the fundamental frequencies of the system [3]. For this purpose an eigenvalue solution routine has been incorporated into NONSAP.

The algorithm considers the solution of the generalized eigenvalue problem

$$ {}^{0}K\ \phi = \omega^2\ M\ \phi \tag{8} $$

where ${}^{0}K$ is the tangent stiffness matrix at time 0, M is the mass matrix of the system and ω and ϕ are free vibration frequency and mode shape, respectively. The mass matrix can be diagonal (lumped mass assumption) or banded (consistent mass assumption), and the stiffness matrix ${}^{0}K$ is assumed to be positive definite. The solution to Eq. (8) can be written as

$$ {}^{0}K\ \Phi = M\ \Phi\ \Omega^2 \tag{9} $$

where Φ is a matrix with its columns equal to the mass-orthonormalized eigenvectors and Ω^2 is a diagonal matrix of the corresponding eigenvalues, i.e.

$$ \Phi = [\phi_1\ \ \phi_2\ \ \cdots\ \ \phi_n]\ ;\ \ \Omega^2 = \text{diag}\ (\omega_i{}^2) \tag{10} $$

The solution algorithm used in NONSAP is the determinant search method presented in [4]. Basically, the algorithm combines triangular factorization and vector inverse iteration in an optimum manner to calculate the required eigenvalues and eigen-

vectors; these are obtained in sequence starting from the least dominant eigenpair (ω_1^2 , ϕ_1). An efficient accelerated secant iteration procedure which operates on the characteristic polynomial

$$p(\omega^2) = \det (K - \omega^2 M) \qquad (11)$$

is used to obtain a shift near the next unknown eigenvalue. The eigenvalue separation theorem (Sturm sequence property) is used in this iteration. Each determinant evaluation requires a triangular factorization of the matrix $K - \omega^2 M$. Once a shift near the unknown eigenvalue has been obtained, inverse iteration is used to calculate the eigenvector and the eigenvalue is obtained accurately by adding the Rayleigh quotient correction to the shift value.

7. STEP-BY-STEP SOLUTION

The main phase in the analysis is the step-by-step solution of the equilibrium equations, Eq. (2). The algorithm used was presented in Table 1. The aim in this chapter is to describe in more detail the actual computer solution. Since the program can perform static and dynamic, linear and nonlinear analysis, it is convenient to consider in the following the different analysis types separately.

7.1 Linear Static Analysis

In a linear static analysis, all element groups are linear and only the linear stiffness matrix is calculated in the matrix assemblage phase. The stiffness matrix is triangularized before entering the step-by-step solution phase. It should be noted that this solution corresponds to a linear dynamic analysis, in which mass and damping effects are neglected. Therefore, by specifying time varying loads, the solution can be obtained for multiple load conditions.

Figure 14 shows the tape operations used in the analysis.

7.2 Linear Dynamic Analysis

In a linear dynamic analysis all elements are linear, with mass and possibly damping effects included. The mass matrix may be diagonal (lumped mass analysis) or banded (consistent mass analysis) and additional concentrated masses may be specified at selected degrees of freedom. The damping matrix C is assumed to be of the form

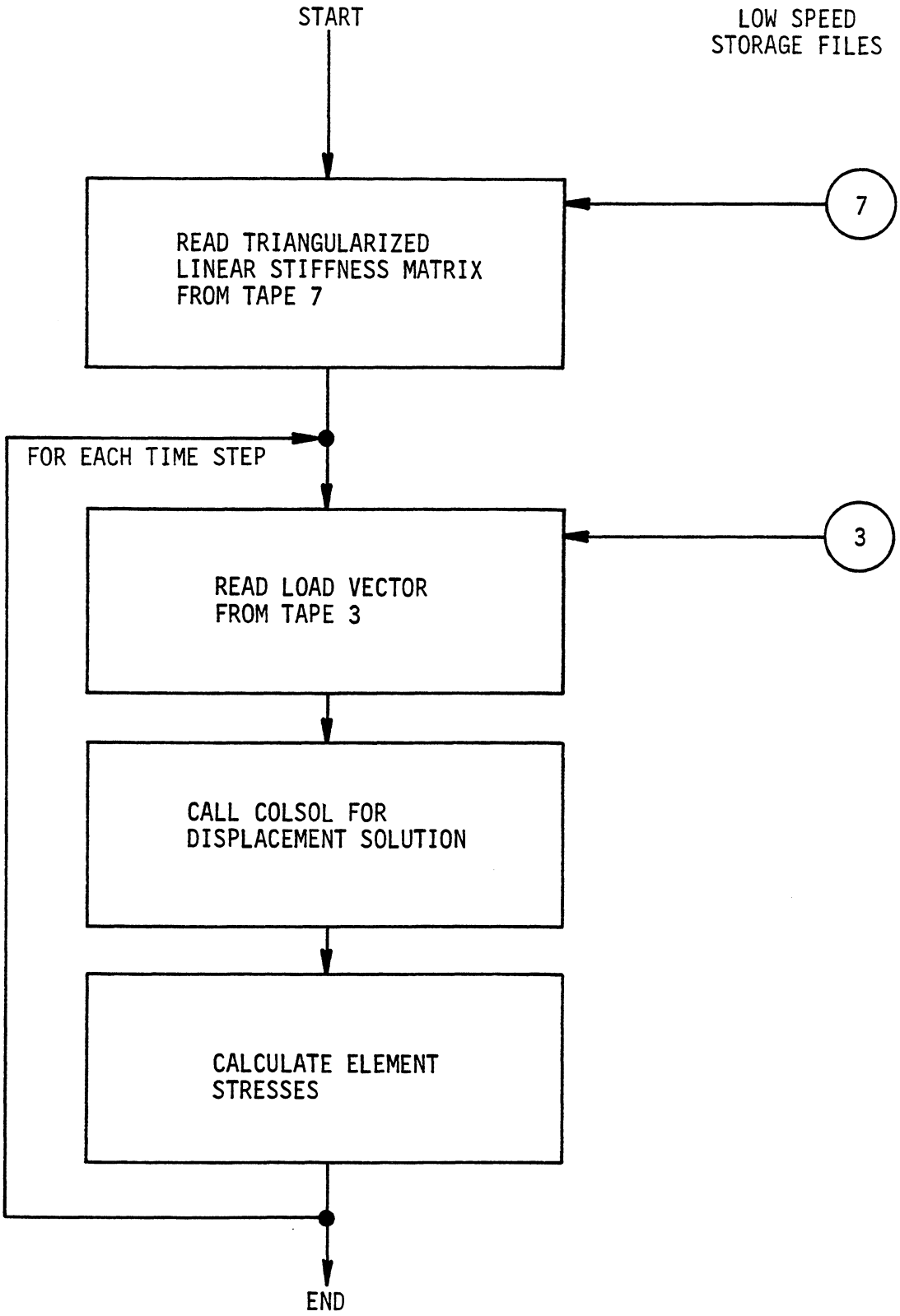


FIGURE 14 FLOW CHART FOR STEP-BY-STEP SOLUTION IN LINEAR STATIC ANALYSIS

$$C = \alpha M + \beta K + C_d \qquad (12)$$

where α and β are the Rayleigh damping coefficients and C_d is a diagonal damping matrix, assembled from concentrated dampers which are specified at selected degrees of freedom [7]. In Eq. (12), K and M are the linear stiffness and mass matrices of the complete element assemblage.

The tape operations performed during a linear dynamic analysis depend on the characteristics of the mass and damping matrices employed. Figure 15 illustrates the various possibilities for storage and retrieval of the matrices used.

7.3 Nonlinear Static Analysis

In nonlinear static analysis linear and nonlinear element groups are defined. Damping and mass effects are neglected.

Before the step-by-step solution, the linear stiffness matrix corresponding to the linear elements of the complete element assemblage was calculated (see Fig. 9). This matrix is now updated in preselected load steps by the stiffness matrices of the nonlinear elements to form the current tangent stiffness matrix. The interval of load steps in which a new tangent stiffness matrix is to be formed is input to the program.

Depending on the nonlinear formulations and the nonlinear material models used, and also depending on the magnitude of the load steps, the accuracy of the solution may be significantly increased using equilibrium iteration. In the program the interval of load steps, in which equilibrium iterations are to be performed, can be defined in the input control data.

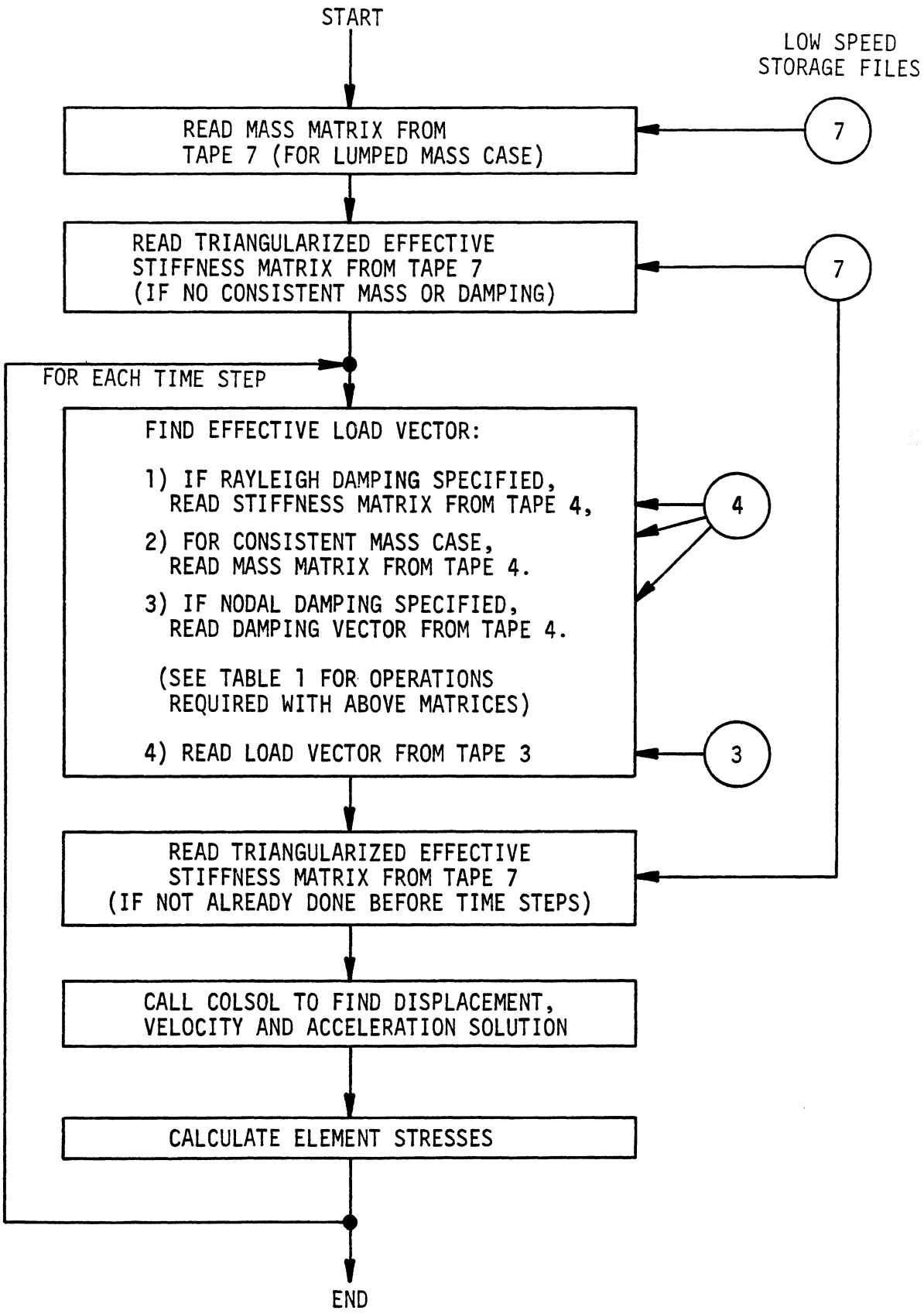


FIGURE 15 FLOW CHART FOR STEP-BY-STEP SOLUTION IN LINEAR DYNAMIC ANALYSIS

The storage of the matrices and tape operations carried out in the analysis are shown in Fig. 16.

7.4 Nonlinear Dynamic Analysis

A nonlinear dynamic analysis is carried out essentially in the same way as a nonlinear static analysis, but mass and possibly damping effects are included. The mass and damping matrices are defined as in linear dynamic analysis, where the Rayleigh damping coefficient β is now applied to the linear stiffness matrix of the element assemblage. It should be noted that the structure mass and damping matrices are calculated before the step-by-step solution, see Fig. 9.

The tape storage scheme and program flow in a nonlinear dynamic analysis are given in Fig. 17.

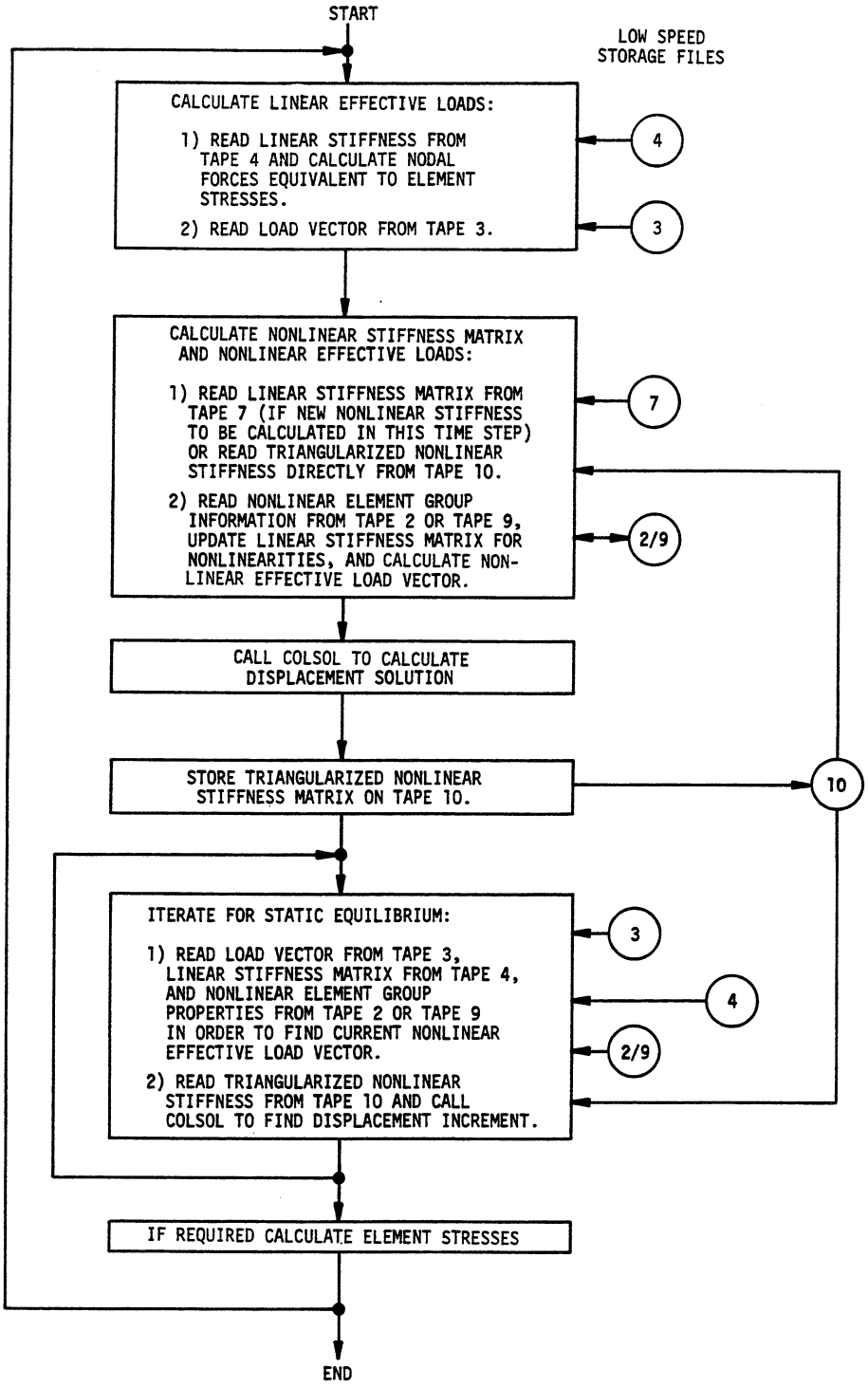


FIGURE 16 FLOW CHART FOR STEP-BY-STEP SOLUTION IN NONLINEAR STATIC ANALYSIS

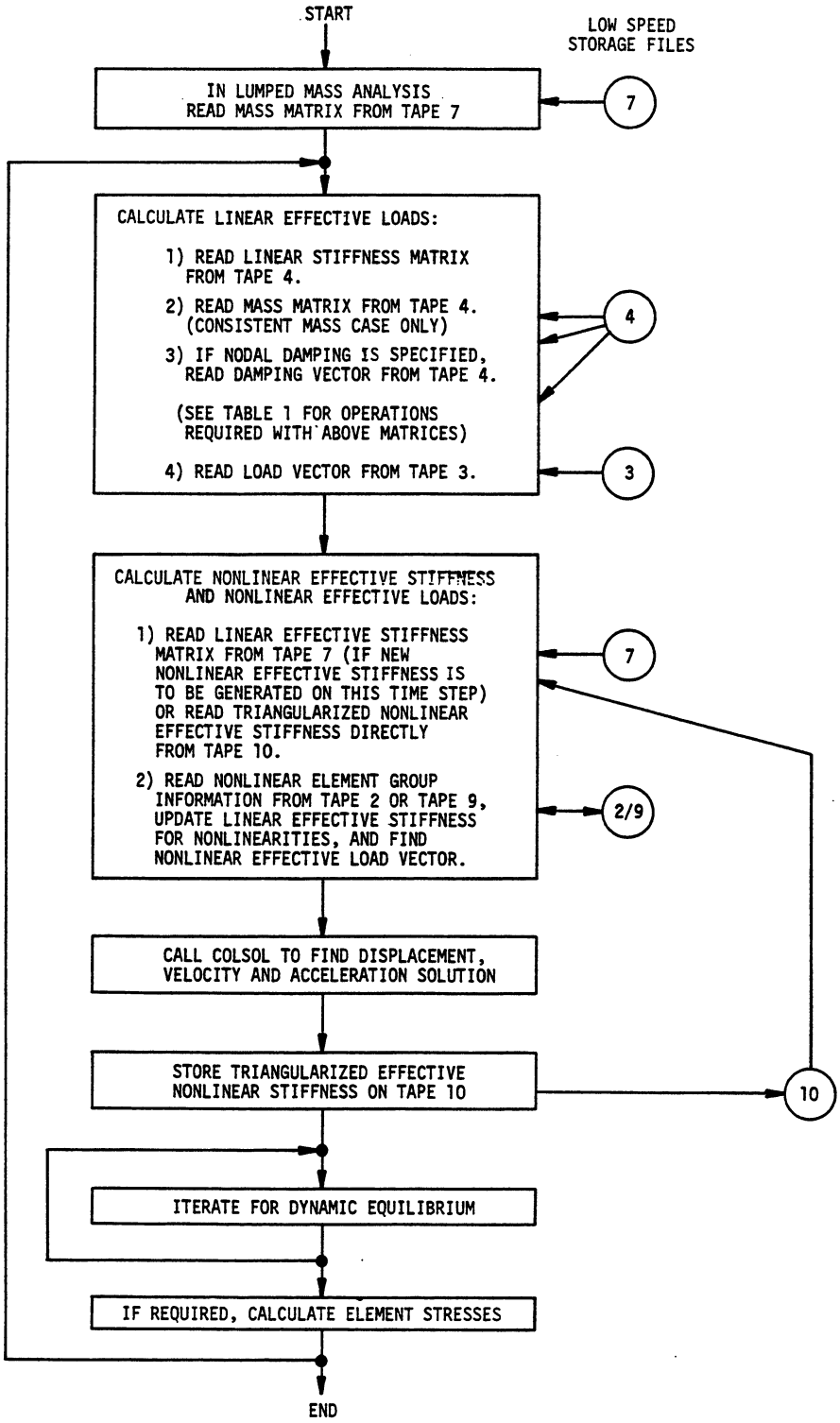


FIGURE 17 FLOW CHART FOR STEP-BY-STEP SOLUTION IN NONLINEAR DYNAMIC ANALYSIS

8. ANALYSIS RESTART

In nonlinear analysis it is often the case that the response of a structure has been calculated for some time (load) steps and that on interpretation of the results, it is decided to analyze the structure for more time (load) steps. If this is anticipated, the program can be used to restart at the end of the successfully completed analysis. The required control cards to store all relevant information at the end of a completed analysis and for restarting are given in Appendix A.

9. DATA CHECK RUN

In the analysis of large structures it is important to be able to check the data read and generated by the program. For this purpose an option is given in which the program simply reads, generates, and prints all data. This printout should be used to carefully verify the input data, since the program itself does not perform extensive data checking.

10. INSTALLATION OF NONSAP

NONSAP is written using FORTRAN IV and has been developed on a CDC/6400 computer.

On installation of NONSAP on machines other than the CDC series, it need be observed that arithmetic calculations must be performed using about 14 digit words. This means that, for example, on IBM and UNIVAC machines double precision need be used. The calculations to be performed in double precision are the formation of element stiffness matrices, the formation of the structure stiffness matrix, and the main steps in the solution of the incremental equations of motion (see Table 1). These calculations need primarily be carried out in double precision because of the truncation errors occurring when too few digits are used, which can cause large errors in the solution and numerical instabilities [21]. If NONSAP is used in single precision on IBM or UNIVAC machines, the results may be doubtful and in some cases the analysis will not be possible.

It may be noted here that the requirement of double precision is equally applicable in linear, nonlinear, static and dynamic analysis [6].

One important option which NONSAP does not have available is efficient pre- and postprocessing. Preprocessing is important for generation and checking of data, whereas postprocessing resulting in efficient display of the calculated response can be essential for obtaining a good understanding of the structural behavior.

With regard to the use of back-up storage, to keep the program system independent sequential accessing is used throughout, and no advantage is taken of system dependent efficient buffering techniques. In this context it should be noted that, considering the back-up storage reading and writing operations in NONSAP (see Chapter 7), the analysis of small order systems that could actually be carried out without use of back-up storage, may in some cases require virtually the same amount of back-up storage requests as systems of larger order. On installation of NONSAP on a specific machine, it should therefore be considered to increase the efficiency of the program use of back-up storage.

11. CONCLUDING REMARKS

The objective in this report was to present a brief description of the current version of the computer program NONSAP. The program is a general analysis tool for the linear and nonlinear, static and dynamic analysis of complex structures. A few applications of the program are presented in the next section.

Although NONSAP can be a very powerful analysis tool, it should be realized that depending on the problem considered, the program may not be easy to use and, for example, much more difficult to handle than the linear analysis program SAP IV [6]. The use of NONSAP requires a thorough understanding of the theoretical basis of the program, of the numerical techniques employed and their computer implementation. This is particularly the case because not many nonlinear solutions are yet possible on a routine basis [5] [23]. Therefore, it is necessary to apply the program only under the conditions and assumptions for which it was developed.

With regard to future developments of NONSAP, the program has primarily been developed to be used as a tool for further research in analysis techniques, nonlinear formulations and the characterization of nonlinear materials [5]. Therefore, additional developments are expected to be oriented towards applied research in those areas.

- PART B -

SAMPLE ANALYSES

SAMPLE ANALYSES

This section presents brief problem descriptions for a set of standard data cases available with the program. Naturally, the few sample analyses can only demonstrate to some degree the capabilities of the program. Additional problems solved with the program can be found in [5].

1. Static and Frequency Analysis of a Tower Cable

The cable stretched between a ground anchor point and a tower attach point shown in Fig. 18 was analyzed for static displacements and frequencies of vibration. The cable was modelled using 12 truss elements of linear elastic material, as shown in Fig. 18 . The cable had an initial tension of 7520.00 lbs. Insulators weighing 510. lbs. each were located at nodes 2, 4, and 6, and a cluster of 6 insulators totaling 3060. lbs. was located at node 8. Nodes 3, 5, 7, and 9 through 12 are intermediate nodes located along the cable without insulators. The total vertical load acting on the cable nodes was 5677.83 lbs. which includes the insulator weights and the cable self-weight.

Figure 18 shows the cable in the static equilibrium configuration with the total load applied. The nonlinear displacement response of node 8 is shown in Fig. 19 . Twenty equal load steps with a new stiffness matrix being calculated in every second step were used to reach the final cable configuration, and an average of four equilibrium iterations were performed in each load step.

For the frequency analysis a lumped mass matrix of the cable has been assumed to which the masses of insulators have been added. The periods of vibration of the cable about the static equilibrium configuration are given in Table 2.

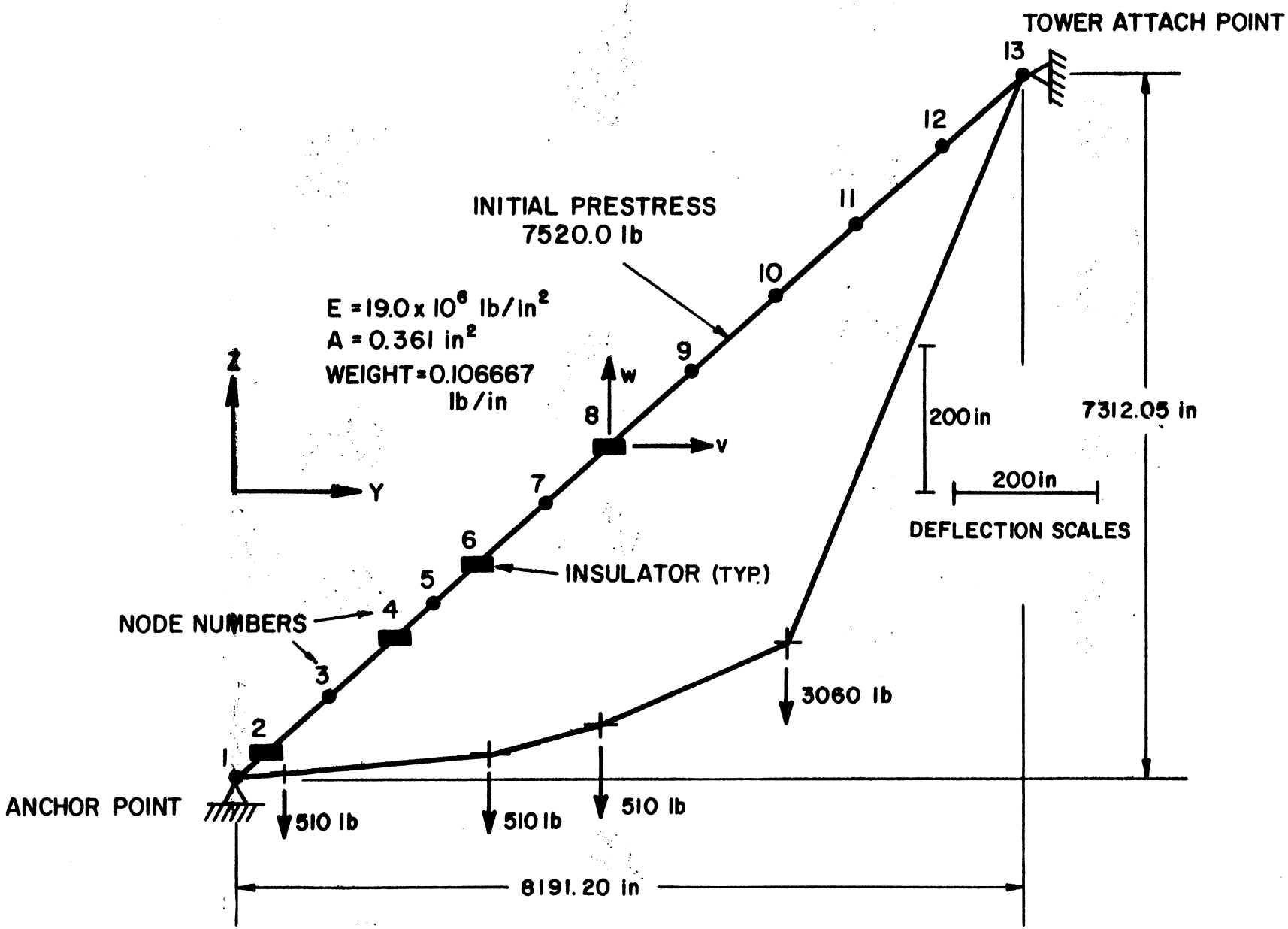


FIGURE 18 STATIC CONFIGURATION OF TOWER CABLE

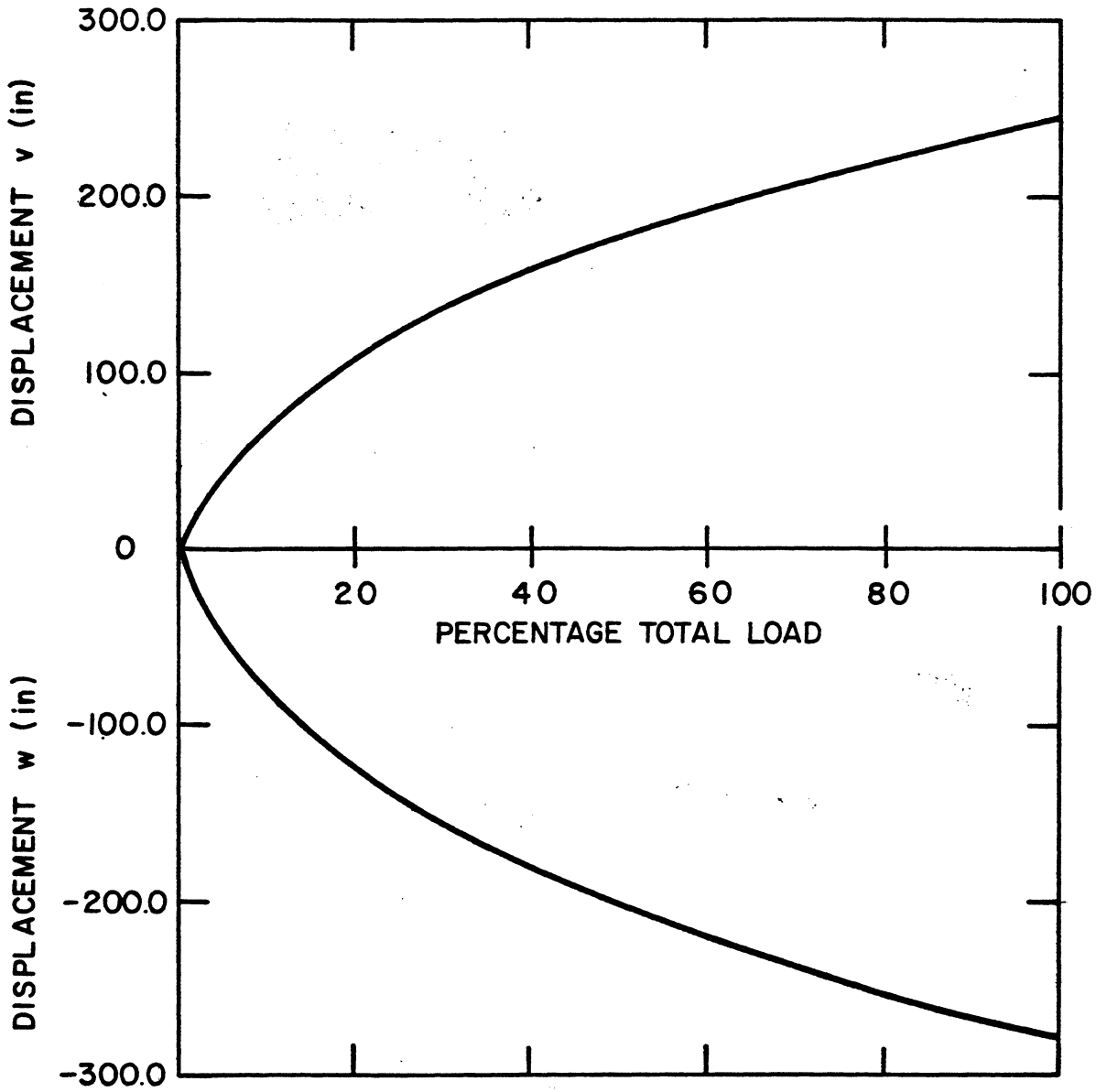


FIGURE 19 LOAD DEFLECTION CURVE OF TOWER CABLE

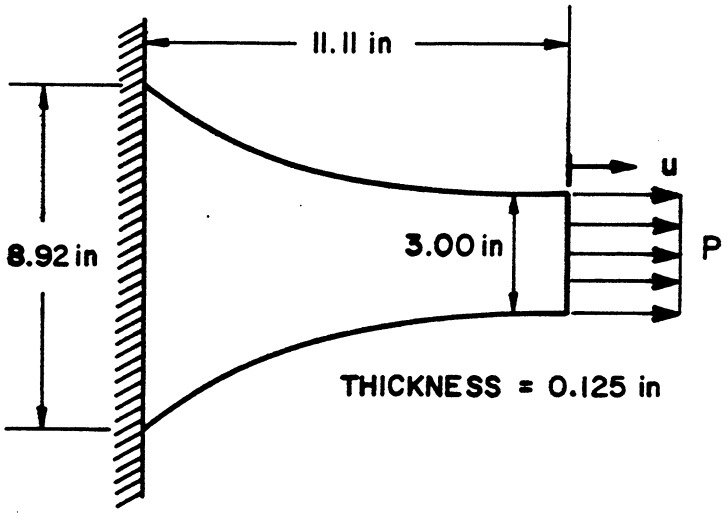
TABLE 2 VIBRATION PERIODS OF CABLE
IN STATIC EQUILIBRIUM CONFIGURATION

MODE NUMBER	PERIOD (SEC.)
1	4.420
2	2.309
3	1.211
4	1.164
5	0.9294

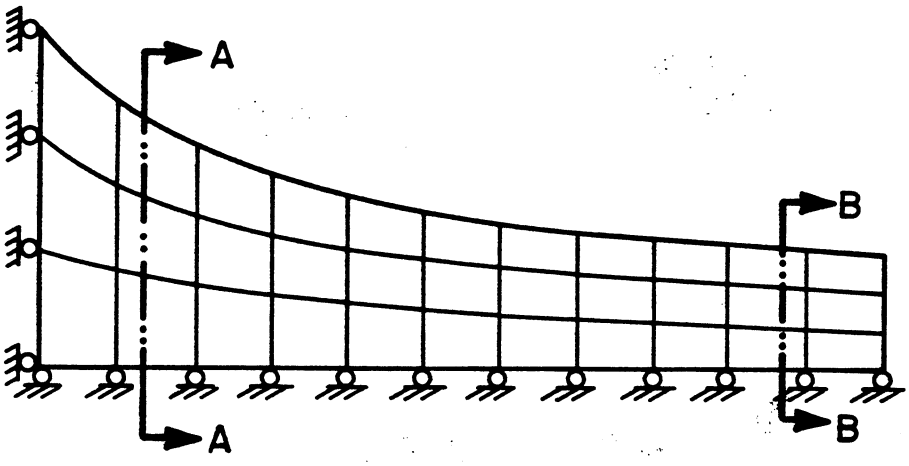
2. Large Displacement and Large Strain Static Analysis of a Rubber Sheet

The rubber sheet shown in Fig. 20 was analyzed for the uniform end loading indicated. The material was assumed to be of the Mooney-Rivlin type, for which experiments by Iding et al. gave C_1 = 21.605 lb./in.^{2}, C_2 = 15.743 lb./in.^{2} [10].

Figure 21 shows the static displacement response of the sheet. It is noted that the final displacement at the loaded end is of the order of the original length of the sheet, at which stage Green-Lagrange strains of 1.81 are measured. The final configuration of the sheet was reached in 4 equal load steps with an average of 5 equilibrium iterations in each step. Excellent agreement between the experimental results by Iding et al. [10] and those predicted by NONSAP has been obtained.

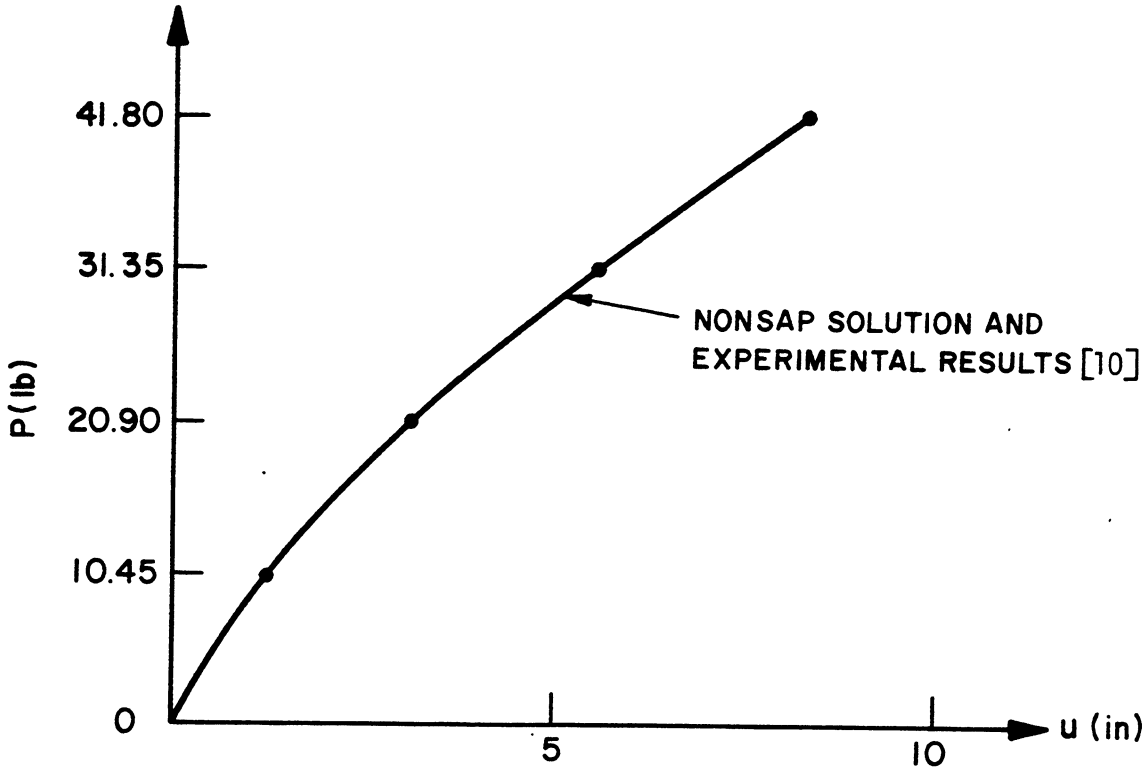


RUBBER SHEET

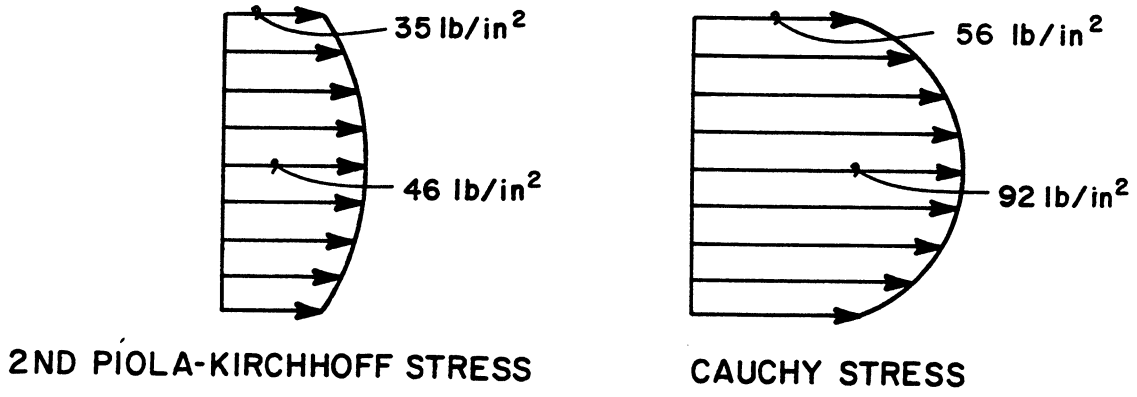


FINITE ELEMENT MESH (4 NODE ELEMENTS)

FIGURE 20 LARGE DISPLACEMENT AND LARGE STRAIN STATIC ANALYSIS OF A RUBBER SHEET



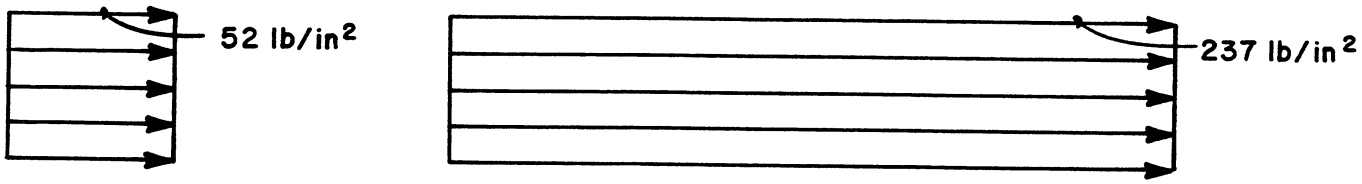
DEFLECTION AT LOADED END



2ND PIOLA-KIRCHHOFF STRESS

CAUCHY STRESS

STRESS DISTRIBUTION ACROSS SECTION A-A AT P=41.80lb



2ND PIOLA-KIRCHHOFF STRESS

CAUCHY STRESS

STRESS DISTRIBUTION ACROSS SECTION B-B AT P=41.80lb

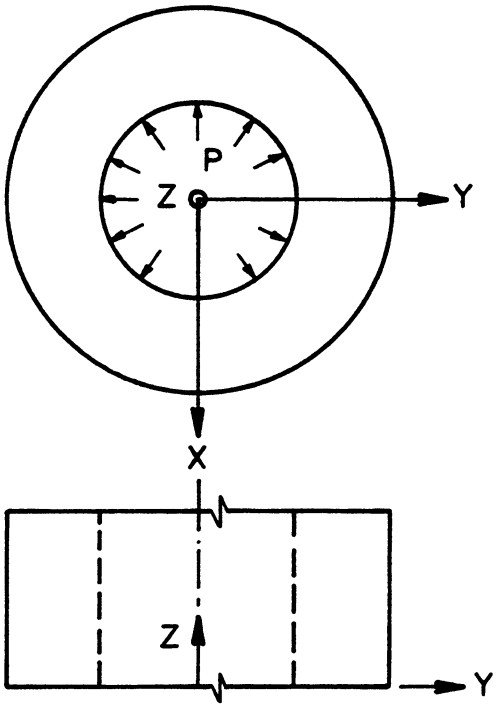
FIGURE 21 DISPLACEMENT AND STRESS RESPONSE OF A RUBBER SHEET

3. Elastic-Plastic Static Analysis of a Thick-Walled Cylinder

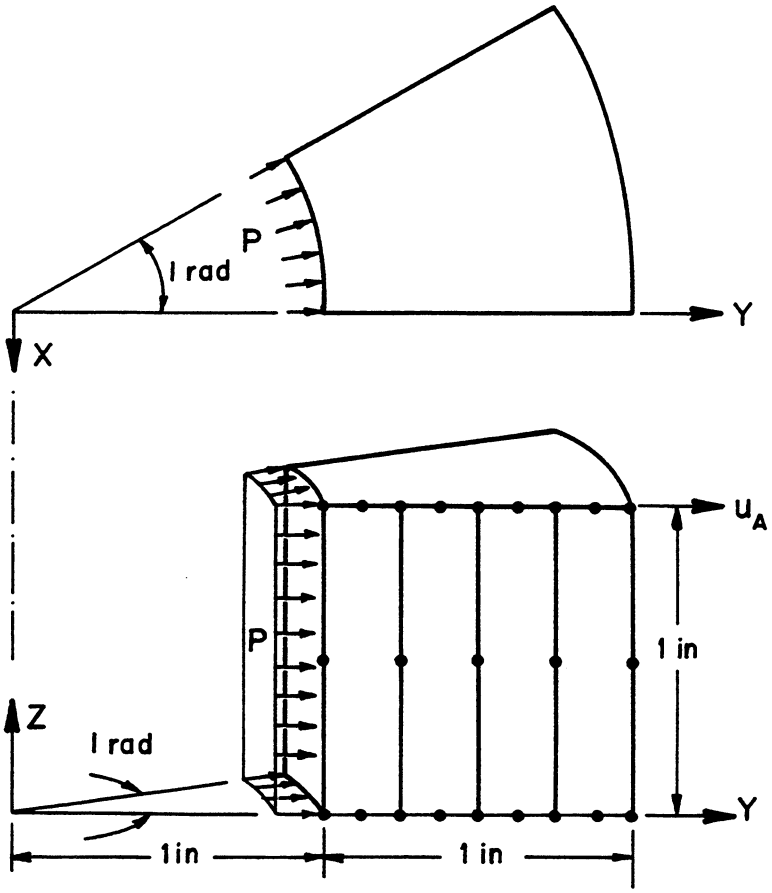
The thick walled cylinder in Fig. 22 subjected to internal pressure was analyzed using four 8-node axisymmetric elements.

The material of the cylinder was assumed to obey the von Mises yield condition with elastic perfectly plastic response. The same analysis was also carried out using the Drucker-Prager yield condition with material variables corresponding to those used in the von Mises condition, and identical results have been obtained.

Since displacements and strains are small, the analysis of the cylinder was carried out using the materially nonlinear only formulation. Figure 23 shows the radial displacement response of the cylinder as a function of the applied load, and Fig. 24 gives the stress distribution through the wall of the cyliner at a given level of internal pressure. Excellent agreement with the solution given by Hodge and White has been obtained [8].



TOP AND SIDE VIEWS



ENLARGED TOP VIEW AND
AXISYMMETRIC MESH

ELASTIC-PERFECTLY PLASTIC MATERIAL

VON MISES YIELD CONDITION

$G = 10^5/3$ lb/in^2

$\nu = 0.3$

$\sigma_y = 17.32$ lb/in^2

DRUCKER-PRAGER YIELD CONDITION

$G = 10^5/3$ lb/in^2

$\nu = 0.3$

ANGLE OF FRICTION = 0.0^0

COHESION = 8.66 lb/in^2

FIGURE 22 FINITE ELEMENT MESH OF THICK-WALLED CYLINDER

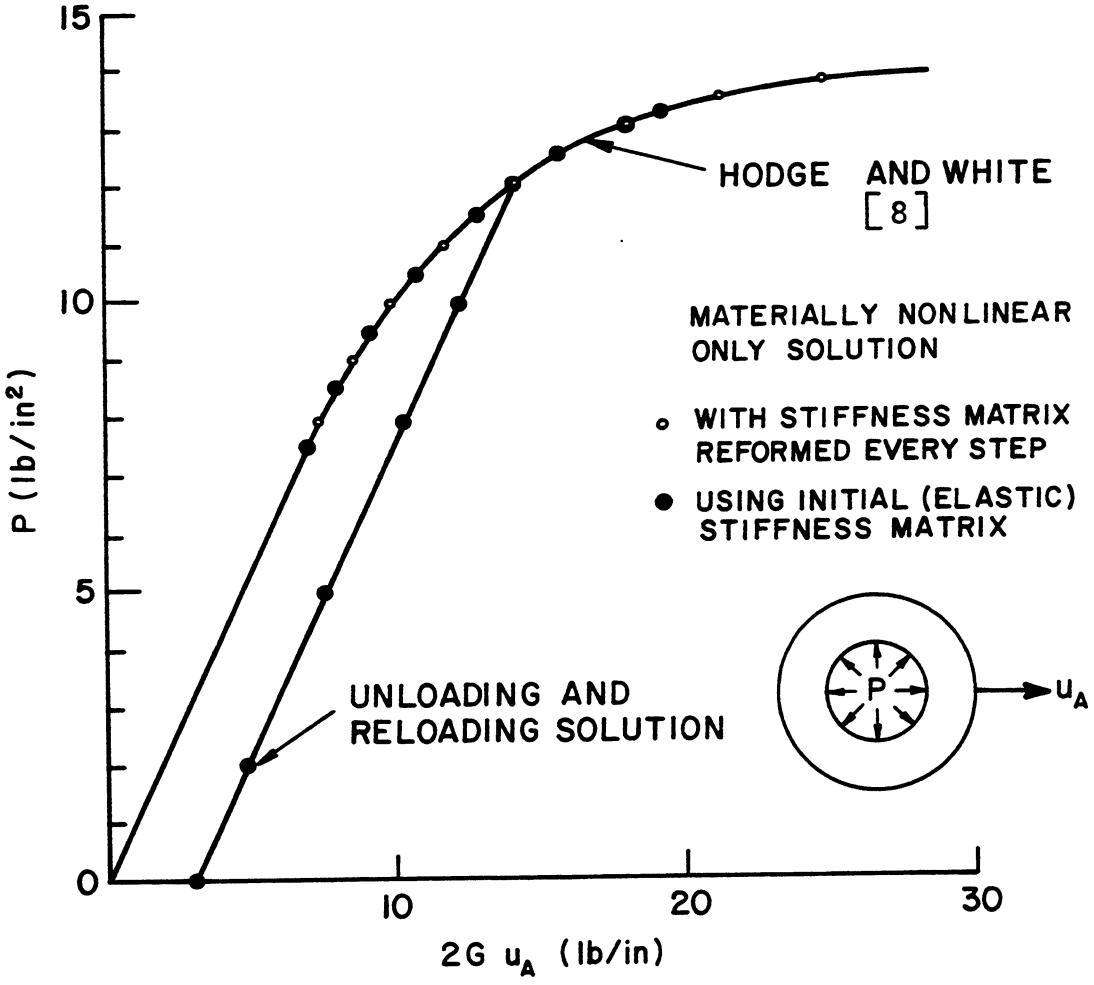


FIGURE 23 ELASTIC-PLASTIC DISPLACEMENT RESPONSE OF THICK-WALLED CYLINDER

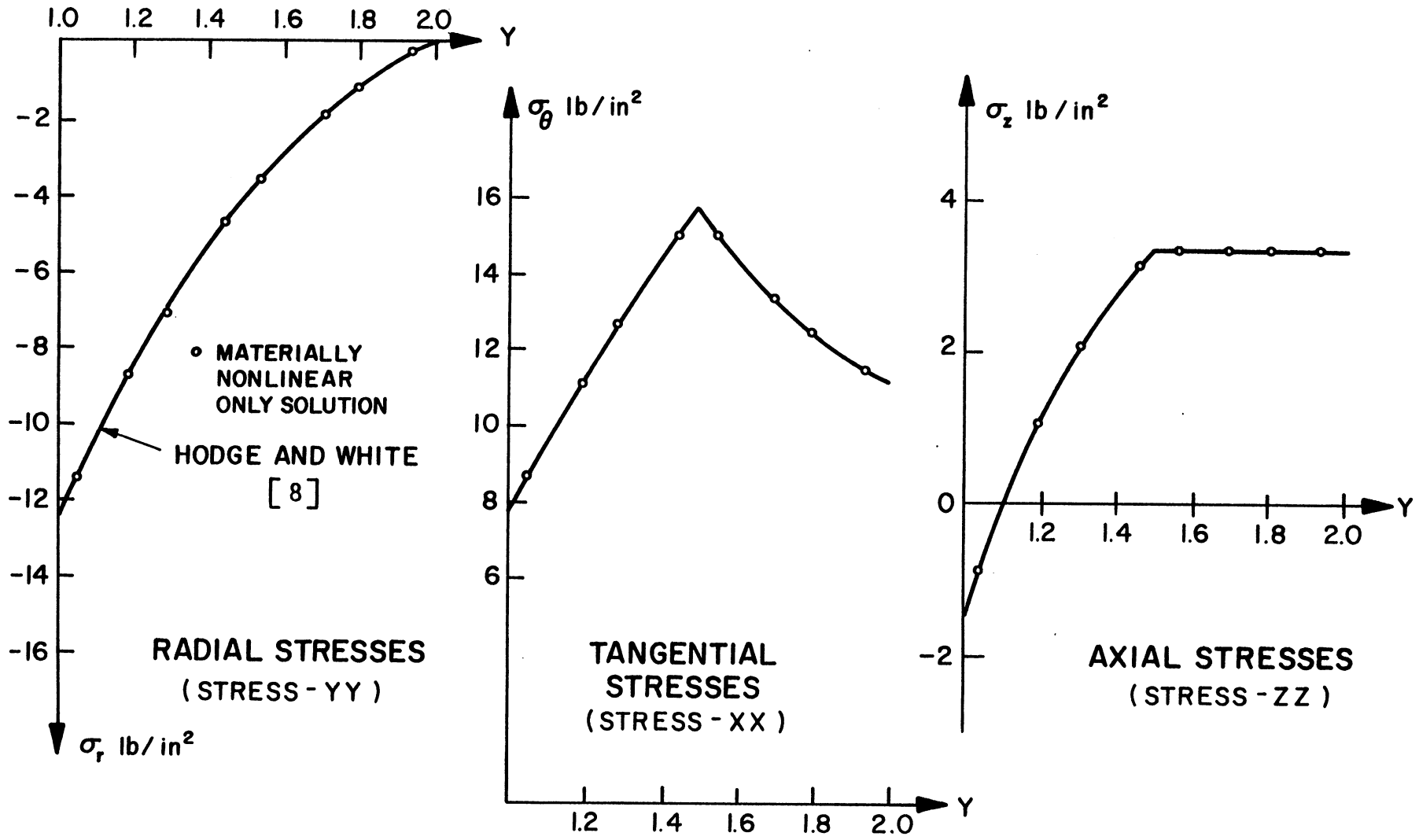
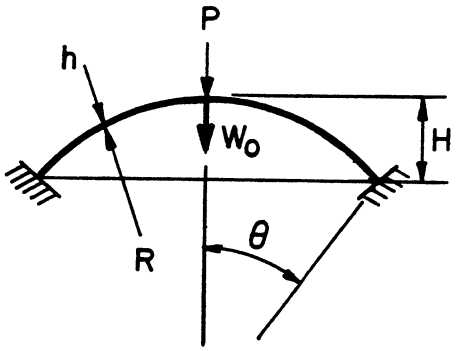


FIGURE 24 ELASTIC-PLASTIC STRESS DISTRIBUTION THROUGH THICKNESS OF THICK-WALLED CYLINDER AT P = 12.5 lb/in²

4. Static Large Displacement Analysis of a Spherical Shell

The spherical shell subjected to a concentrated apex load shown in Fig. 25 was analyzed for static response. The NONSAP solution could be compared with the response predicted by Stricklin [22] and Mescall [12].

Figure 25 shows the static load-deflection response calculated by NONSAP using the total Lagrangian formulation. Good correspondence with the solutions obtained by Stricklin and Mescall is observed.



TEN 8 NODE ELEMENTS
FOR AXISYMMETRIC
ANALYSIS

R = 4.76 in
h = 0.01576 in
H = 0.0859 in
θ = 10.9°
λ = 6
E = 10×10^6 lb/in^2
ν = 0.3
ρ = 0.000245 lb-sec^2/in^4

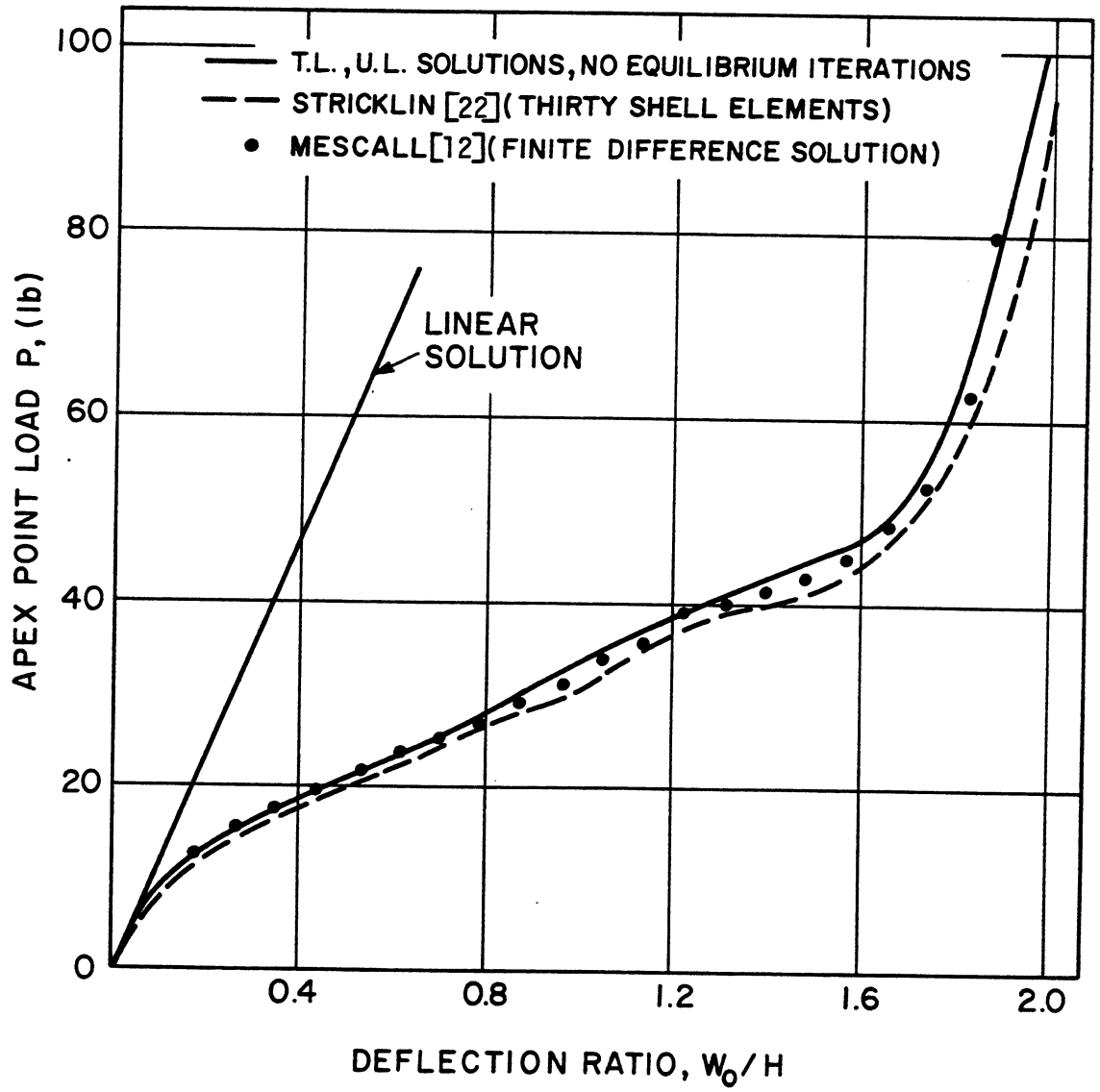


FIGURE 25 LOAD-DEFLECTION CURVES FOR SPHERICAL SHELL

5. Static and Dynamic Analysis of a Simply Supported Plate

A simply supported plate subjected to a concentrated mid-point load was analyzed for static and dynamic response. Figure 26 shows the finite element idealization used for one quarter of the plate. In the analysis, small displacements were assumed and the material was considered to be isotropic linear elastic (see Section 4.4). The Gauss integration order used in the derivation of element stiffness matrices was 2 in the r, s and t directions.

It should be noted that the finite element idealization is rather coarse and no high accuracy in the solution can be expected. Figure 27 shows the static and dynamic displacement response as predicted by NONSAP. The analysis was also performed using the same element idealization with SAP IV [6].

Table 3 below summarizes the frequency solution of the plate assuming symmetric response about the X and Y axes.

TABLE 3 VIBRATION PERIODS OF A SIMPLY SUPPORTED PLATE

MODE	PERIODS (SEC.)		
	Kirchhoff Plate Theory†	NONSAP Consistent Mass	NONSAP Lumped Mass
1	0.2366	0.2260	0.2264
2	0.0684	0.0609	0.0668
3	0.0362	0.0328	0.0359

† $T = 2\sqrt{\mu}/\pi\sqrt{D}\left(\frac{m^2}{40^2} + \frac{n^2}{60^2}\right)$; $D = \frac{E\,h^3}{12(1-\nu^2)}$; $\mu = \frac{\rho}{h}$

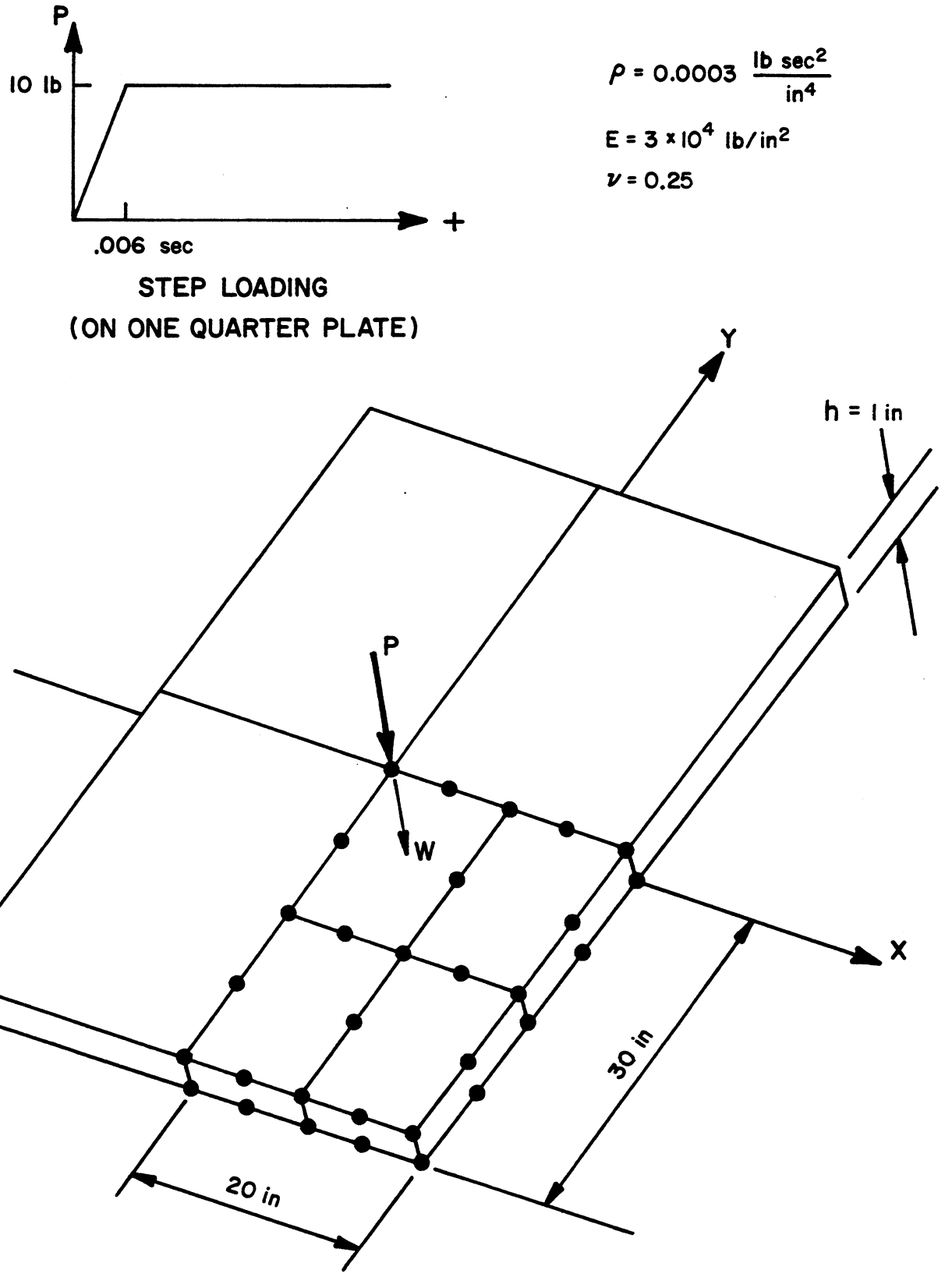


FIGURE 26 STATIC AND DYNAMIC ANALYSIS OF SIMPLY SUPPORTED PLATE

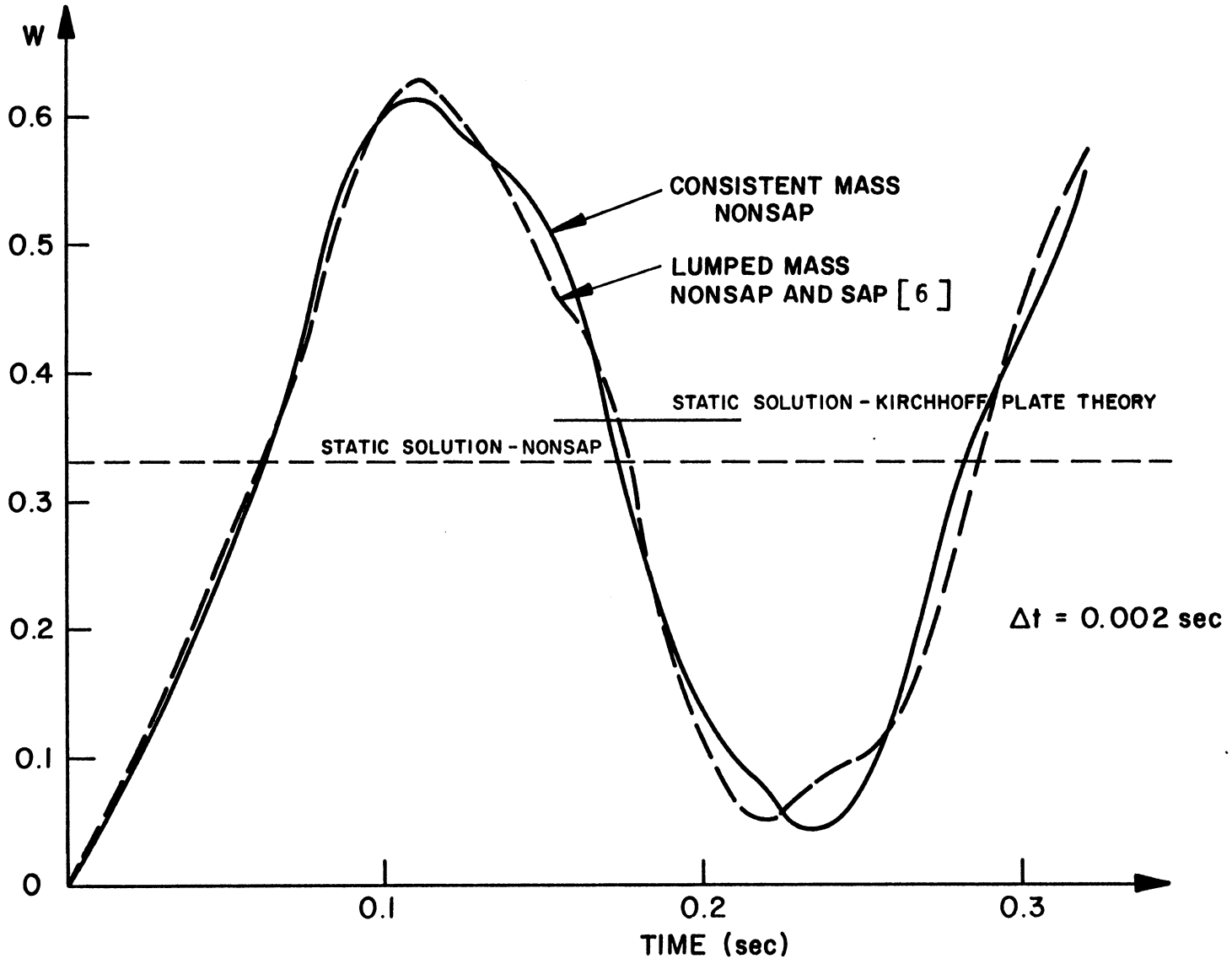


FIGURE 27 DISPLACEMENT RESPONSE OF SIMPLY SUPPORTED PLATE

REFERENCES

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

2. Bathe, K.J and Wilson, E.L., "NONSAP--A General Finite Element Program for Nonlinear Dynamic Analysis of Complex Structures," Paper M3/1, Proceedings 2nd Int. Conference on Structural Mechanics in Reactor Technology, Berlin, September, 1973.

3. Bathe, K.J., and Wilson, E.L., "Stability and Accuracy Analysis of Direct Integration Methods," International Journal of Earthquake Engineering and Structural Dynamics, Vol. 1, pp. 283-291, 1973.

4. Bathe, K.J., and Wilson, E.L., "Eigensolution of Large Structural Systems with Small Bandwidth," ASCE, J. Engg. Mech. Div., Vol. 99, pp. 467-479, June, 1973.

5. Bathe, K.J., Ozdemir, H., and Wilson, E.L., "Static and Dynamic Geometric and Material Nonlinear Analysis," SESM Report No. 74-4, Dept. of Civil Engineering, University of California, Berkeley, Feb. 1974.

6. Bathe, K.J., Wilson, E.L., and Peterson, F.E., "SAP IV--A Structural Analysis Program for Static and Dynamic Analysis of Linear Structural Systems," EERC Report No. 73-11, College of Engineering, University of California, Berkeley, June 1973.

7. Clough, R.W., and Bathe, K.J., "Finite Element Analysis of Dynamic Response," Proceedings, 2nd U. S.-Japan Symposium on Recent Advances in Computational Methods of Structural Analysis and Design, Berkeley, California, 1972.

8. Hodge, P.G., and White, G.H., "A Quantitative Comparison of Flow and Deformation Theories of Plasticity," J. Appl. Mech., Vol. 17, pp. 180-184, 1950.

9. Hurty, W. and Rubinstein, M.F., Dynamics of Structures, Prentice Hall, 1964.

10. Iding, R.H., "Identification of Nonlinear Materials by Finite Element Methods," SESM Report No. 73-4, Department of Civil Engineering, University of California, Berkeley, January 1973; and Private Communication to be published.

11. Marcal, P.V., "Survey of General Purpose Programs," Proceedings, 2nd U. S.-Japan Symposium on Recent Advances in Computational Methods of Structural Analysis and Design, Berkeley, California, 1972.

12. Mescall, J.F., "Large Deflections of Spherical Shells Under Concentrated Loads," J. App. Mech., Vol. 32, pp. 936-938, 1965.

13. Nayak, G.C., and Zienkiewicz, O.C., "Elasto-Plastic Stress Analysis. A Generalization for Various Constitutive Relations Including Strain Softening," Int. J. Num. Methods in Engg., Vol. 5, pp. 113-135, 1972.

14. Nelson, I., Baron, M.L., and Sandler, I., "Mathematical Models for Geological Materials for Wave Propagation Studies," Headquarters Defense Nuclear Agency, Report No. 2672, Washington, D.C.

15. Nickell, R.E., "Direct Integration Methods in Structural Dynamics," ASCE, J. Engg. Mech. Div., Vol. 99, pp. 303-317, 1973.

16. Oden, J.T., Finite Elements of Nonlinear Continua, McGraw-Hill, 1972.

17. Peterson, F.E., and Bathe, K.J., "Nonlinear Dynamic Analysis of Reactor Core Components," Report No. S-104.3, Engineering/Analysis Corporation, Berkeley, March, 1972.

18. Przemieniecki, J.S., Theory of Matrix Structural Analysis, McGraw-Hill, New York, 1968.

19. Pugh, C.E., Corum, J.M., Liu, K.C., and Greenstreet, W.L., "Currently Recommended Constitutive Equations for Inelastic Design Analysis of FFTF Components," Report No. TM-3602, Oak Ridge National Laboratory, Oak Ridge, Tennessee, 1972.

20. Reyes, S.F., and Deere, D.U., "Elastic-Plastic Analysis of Underground Openings," Proceedings, First Congress, International Society of Rock Mechanics, Lisbon, 1966.

21. Roy, J.R., "Numerical Errors in Structural Solutions," ASCE, J. Engg. Mech. Div., April 1971.

22. Stricklin, J.A., "Geometrically Nonlinear Static and Dynamic Analysis of Shells of Revolution," High Speed Computing of Elastic Structures, Proceedings of the Symposium of IUTAM, Univ. of Liege, August 1970.

23. Stricklin, J.A., Von Riesemann, W.A., Tillerson, J.R., and Haisler, W.E., "Static Geometric and Material Nonlinear Analysis," Proceedings, 2nd U.S.-Japan Symposium on Recent Advances in Computational Methods of Struct. Analysis and Design, Berkeley, Calif., 1972.

24. Wilson, E.L., Bathe, K.J., and Doherty, W.P., "Direct Solution of Large Systems of Linear Equations," Computers and Structures, to appear.

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

- PART C -

APPENDICES

APPENDIX - DATA INPUT TO NONSAP

Contents

APPENDIX (continued)

I. HEADING CARD (12A6)

note	columns	variable	entry
(1)	1 - 72	HED(12)	Enter the master heading information for use in labeling the output

NOTES/

(1) Begin each new data case with a new heading card.

II. MASTER CONTROL CARDS

Card 1 (I5, 6I1, I4, 3I5, 2F10.0, I5)

notes	columns	variable	entry
(1)	1 - 5	NUMNP	Total number of nodal points; EQ.0; program stop
(2)	6	IDOF(1)	Master X-translation code; EQ.0; admissible EQ.1; deleted
	7	IDOF(2)	Master Y-translation code
	8	IDOF(3)	Master Z-translation code
	9	IDOF(4)	Master X-rotation code
	10	IDOF(5)	Master Y-rotation code
	11	IDOF(6)	Master Z-rotation code
(3)	12 - 15	NEGL	Number of linear element groups; EQ.0; all elements are nonlinear
(4)	16 - 20	NEGNL	Number of nonlinear element groups; EQ.0; all elements are linear
(5)	21 - 25	MODEX	Flag indicating solution mode; EQ.0; data check only EQ.1; execution EQ.2; restart
(6)	26 - 30	NSTE	Number of solution time steps
(6)	31 - 40	DT	Time step increment
(7)	41 - 50	TSTART	Time at solution start
(8)	51 - 55	IPRI	Output printing interval; EQ.0; default set to "1"

NOTES/

(1) The total number of nodes (NUMNP) controls the amount of data to be read in Section III. If NUMNP.EQ.0, the program terminates execution.

(2) The codes IDOF(1), IDOF(2), ..., IDOF(6) given in cc 6-11 can have values of "0" or "1" and are used to conserve storage and reduce input for special problems not requiring a full six degrees of freedom at each node. If IDOF(I).EQ.1, the I-th (I=1,2,...,6) nodal displacement does not exist in this problem, and any reference in the data to the I-th node degree of freedom is ignored by the program. If, for example, this problem concerns the analysis of a 2/D continuum, the convention is that 2/D elements must be described in the global Y,Z plane and that only the global Y and Z translations are defined for the 2/D continuum elements; hence, IDOF(I) for I of 1,4,5 and 6 must be "1".

Table II.1 contains the IDOF sets for the current element types in the program. This table should be used together with the nodal point data card options (see Section III) to define all required degrees of freedom.

(3) The program distinguishes between linear and nonlinear elements. Linear elements have their stiffness matrices formed only once, and the formulation excludes consideration of either geometric or material nonlinearities; linear elements are used to represent those regions of a model which can be assumed to behave linearly and should be used (where possible) to improve solution efficiency.

An element group is a series of elements of a particular type (e.g., TRUSS, 2/D CONTINUUM, etc.) in which element numbers are assigned in ascending sequence beginning with "1" and ending with the total number of elements in that particular group. Elements forming a group must have the same values for the following parameters (see Sections IX, X, XI)

(a) Element type (NPAR(1), NPAR(5))

(b) Type of analysis (NPAR(3))

(c) Material model (NPAR(15), NPAR(17), NPAR(18))

(d) Integration order (NPAR(10), NPAR(11))

Elements defined by the same values of above mentioned parameters can be broken down into more than one group, although this may not be desirable for maximum solution efficiency. Consider the following example.

ELEMENT TYPE	MASTER BOUNDARY CONDITION CODES					
	IDOF(1)	IDOF(2)	IDOF(3)	IDOF(4)	IDOF(5)	IDOF(6)
1. TRUSS	0	0	0	1	1	1
2. 2/D CONTINUUM	1	0	0	1	1	1
3. 3/D CONTINUUM	0	0	0	1	1	1

TABLE II.1 - Possible Finite Element Degrees of Freedom for the Various Element Types Currently Operational in NONSAP

IV. APPLIED LOADS DATA

1. Control card (3I5)

note	columns	variable	entry
(1)	1 - 5	NLOAD	Number of cards used to prescribe loads acting at the nodes
(2)	6 - 10	NLCUR	Number of load curves (time functions)
(2)	11 - 15	NPTM	Maximum number of points used to describe any one of the load curves

NOTES/

(1) NLOAD determines the number of cards to be read in Section IV.3, below. The loads defined in Section IV.3 are concentrated node forces/moments that do not change direction as the structure deforms; i.e., the applied node forces are conservative loads.

(2) Time dependent loads are applied to the structure by means of load (or time) function [i.e., f(t)] references and function multipliers assigned with the loads. At time t the value of f(t) is found by linear interpolation in the table of f(t) vs. t; f(t) times the multiplier is the magnitude of the applied load at t. NPTM is the maximum number of [f(t),t] pairs used to describe any one of the NLCUR functions; an individual function may have fewer than NPTM [f(t),t] points as input, but no function can be input with more than NPTM points. At least two points are required per function; otherwise interpolation in time is not possible.

Suppose that a model contains 20 linear TRUSS elements and 33 linear 2/D CONTINUUM elements, 11 of which have isotropic material properties. Assuming that the 2/D isotropic properties are not given as equivalent orthotropic properties, a minimum of three (3) linear element groups (i.e., NEGL.EQ.3) is required in this example--

LINEAR GROUP NUMBER	ELEMENT TYPE	RANGE OF ELEMENT NUMBERS	TYPE OF MATERIAL MODEL
1	TRUSS	1 - 20	AE, constant
2	2/D CONTINUUM	1 - 11	Isotropic
3	2/D CONTINUUM	1 - 22	Orthotropic

Further, suppose that it is desirable to separate the 20 TRUSS elements into two groups of 5 and 15 elements and the 22 2/D CONTINUUM orthotropic elements into two groups, one with 8 and the other with 14 elements; then, NEGL.EQ.5 and

1	TRUSS	1 - 5	AE, constant
2	TRUSS	1 - 15	AE, constant
3	2/D CONTINUUM	1 - 11	Isotropic
4	2/D CONTINUUM	1 - 14	Orthotropic
5	2/D CONTINUUM	1 - 8	Orthotropic

Since NONSAP is an overlaid program, the order of the element groups (linear and nonlinear) is most efficient when all TRUSS element groups, 2/D CONTINUUM element groups,... are grouped together. In addition, all element groups using the same nonlinear material model are best input together.

It need be noted that the program must read data for all of the "NEGL" groups before reading any data for the nonlinear groups.

A restriction that must be noted when organizing elements into groups (either linear or nonlinear) is that only one type of material model is allowed for the elements in that group. Also, it is permissible to model a structure with nonlinear elements only, in which case NEGL.EQ.0 and NEGNL.GE.1.

(4) Nonlinear elements include the effects of material and/or geometric nonlinearities in the formation of stiffness and stress recovery matrices. The type of nonlinearities to be associated with an individual group of elements is defined by means of data given on the element group control card. In separating elements into nonlinear groups, note that only one type of material model is allowed for the group. The order in which groups are input was discussed in note (3) above.

The total number of element groups processed by the program is the sum NEGL+NEGNL . Also, NEGNL can be zero, but then the number of linear element groups must be at least one.

(5) The MODEX parameter determines whether the program is to check the data without executing an analysis (i.e., MODEX.EQ.0) or if the program is to solve the problem. In the data check only mode, the program only reads and prints all data. Therefore, the MODEX.EQ.0 option should be used for data checking. If MODEX.EQ.2, the problem is a restart job. Refer to Appendix A for setting up a restart job.

(6) DT is the solution time step, Δt, and is used similarly for static, quasi-static or dynamic problems. DT for static problems is not used to perform step-by-step time integration for system response, but rather is used as an equivalent "load step" or "loading increment". All forcing functions are input as tables of f(t) versus t, and the loads at solution step "n" (where "n" is between "1" and "NSTE") are found in the f(t) tables by linear interpolation at $t=n\Delta t$. NSTE must be at least one, except in the special case when frequencies and mode shapes are to be calculated but no time integration solution is desired.

(7) The time at solution start (TSTART) is an input convenience for restart (i.e., MODEX.EQ.2) jobs. TSTART in a restart job would be the final time to which a previous solution was run and saved for use in supplying initial conditions for this job. Since the time counter is incremented from TSTART, none of the forcing function tables need be revised (i.e., shifted in time) for the restart job.

(8) The print interval determines at which solution step interval program results are to be printed. If IPRI.EQ.4, output is produced at the end of solution steps 4, 8, 12, etc.

If IPRI is larger than the total number of solution steps (NSTE), then no output will be printed during the course of solution. If IPRI.LE.NSTE, then print directives must be given for displacements, velocities, and acceleration on Card 6 in this section and for the stress components on the element cards (Section IX.4, X.4, or XI.4).

Card 2 (4I5)

notes	columns	variable	entry
(1)	1 - 5	IMASS	Control flag indicating static or dynamic analysis; EQ.0; static analysis GT.0; dynamic analysis Also flag indicating mass matrix type; EQ.0; no mass effects EQ.1; lumped (diagonal) mass EQ.2; consistent mass matrix
(2)	6 - 10	IDAMP	Flag indicating damping type; EQ.0; no Rayleigh damping EQ.1; Rayleigh damping
(3)	11 - 15	IMASSN	Number of concentrated nodal masses
(3)	16 - 20	IDAMPN	Number of concentrated nodal dampers

NOTES/

(1) The control flag on static or dynamic analysis (IMASS) determines whether or not the program is to solve a problem including the contribution of inertia forces to system equilibrium, i.e. whether to solve a static or dynamic problem.

If IMASS.EQ.0, the program will solve a static (or quasi-static) problem, and no storage will be allocated for either the system mass matrix or the system velocity and acceleration vectors.

If IMASS.EQ.1 or 2, the analysis will be a dynamic analysis. For IMASS.EQ.1 a lumped (diagonal) mass matrix is allowed, in which case the diagonal mass coefficients are stored as a vector, and for IMASS.EQ.2 a consistent mass matrix is generated (from element data). It should be noted that the computation of the effective load vector at each solution time step is considerably more expensive when the consistent mass option is requested, and the extra effort involved in a consistent (as opposed to the diagonal) mass analysis may not be justified in many (if not most) problems.

The mass matrix is constant and is only formed once before the time integration is started.

NOTES/

(2) The damping matrix flag (IDAMP) determines whether or not the program is to include in a dynamic analysis Rayleigh damping. The Rayleigh damping coefficients are defined in Section V. Rayleigh damping can only be included when a dynamic analysis is specified (IMASS.GT.0).

(3) In a dynamic analysis, i.e. IMASS.GT.0, additional concentrated masses and/or additional concentrated dampers can be specified at selected degrees of freedom. The concentrated masses and concentrated dampers are input in Sections VI and VII.

A dynamic analysis with concentrated masses only (i.e., no lumped mass or consistent mass effects) can be accomplished by specifying IMASSN.GE.1, IMASS.EQ.1, and setting the mass density on all element cards to 0.0.

Card 3 (I5)

note	columns	variable	entry
(1)	1 - 5	IEIG	Flag indicating frequency solution mode; EQ.0; no frequency solution EQ.1; find lowest frequencies and associated mode shapes

NOTES/

(1) The control parameter IEIG determines if the program is to solve for vibration mode shapes and frequencies before solving for dynamic response of the model. If IEIG.EQ.1, then additional information which controls the eigensolution must be supplied in Section XII.

If frequencies and mode shapes only are desired (i.e. no step-by-step solution) set NSTE.EQ.0 on card 1 of this section.

II. MASTER CONTROL CARDS (continued)

Card 4 (4I5,E10.4)

note	columns	variable	entry
(1)	1 - 5	ISREF	Number of time steps between reforming effective stiffness matrix EQ.0; default set to "1"
(2)	6 - 10	NUMREF	Number of allowable stiffness reformations in each time step
(3)	11 - 15	IEQUIT	Number of time steps between equilibrium iterations EQ.0; default set to "1"
(3)	16 - 20	ITEMAX	Maximum number of equilibrium iterations permitted EQ.0; default set to "15"
(3)	21 - 30	RTOL	Relative tolerance used to measure equilibrium convergence EQ.0; default set to "1.E-3"

NOTES/

(1) The stiffness matrix reformation interval (ISREF) is ignored if the model is composed of linear element groups only; i.e., NEGNL.EQ.0. For linear problems the matrix is formed and decomposed only once.

For models containing nonlinear element groups, the system stiffness matrix is reformed every ISREF solution steps. For example, if ISREF.EQ.3, the stiffness matrix is formed in solution steps 1 (always), 3, 6, etc. based on conditions known at the end of steps 0, 2, 5, etc., respectively. Only the nonlinear portion of the complete system stiffness matrix is reformed; the linear portion is saved and reinstated when the complete matrix is calculated.

It should be noted that the history dependent variables defining a material law are only permanently updated, i.e. written on back-up storage, whenever a new stiffness matrix is formed. Therefore, all path dependent material models should be used (in a general analysis) with ISREF.EQ.1.

(2) NUMREF is currently ignored.

NOTES/

(3) If a structure is represented by nonlinear element groups with material models which allow for "equilibrium iteration", then the parameter IEQUIT determines at what solution step interval the program is to iterate for system equilibrium, ITEMAX is the maximum number of cycles of iteration allowed in the solution step and RTOL is used to measure convergence of the iteration in terms of change in system displacements. For example, if IEQUIT.EQ.5, ITEMAX.EQ.12 and RTOL.EQ.0.002, then providing the material model(s) allow for iteration, up to 12 cycles of iteration will be performed at solution steps 5,10,15, etc. with convergence declared if

$$\left| \frac{||u^{(n-1)}|| - ||u^{(n)}||}{||u^{(n)}||} \right| \leq 0.002$$

where $||u^{(n)}||$ is the Euclidean norm of the system displacement vector at cycle "n" of the iteration.

Whether or not a material model specifically allows iteration for equilibrium depends on how the model was incorporated into the program; the sections on the element material models distinguish between models that do and do not allow iteration.

Equilibrium iteration can only be performed if

(a) the structure contains at least one nonlinear element group, and

(b) the material models used to represent all nonlinear element groups allow for the possibility of equilibrium iteration.

The parameters IEQUIT, ITEMAX and RTOL will not be used if

(a) the structure is represented with linear elements only, or

(b) any one of the material models associated with a nonlinear element group does not allow for equilibrium iteration.

II. MASTER CONTROL CARDS (continued)

Card 5 (I10, 2F10.0)

note	columns	variable	entry
(1)	1 - 10	IOPE	Time integration method used EQ.0; default set to "1" EQ.1; Wilson's theta method EQ.2; Newmark's method
(2)	11 - 20	OPVAR(1)	First Integration Parameter 1) If IOPE.EQ.1 parameter is THETA EQ.0; default set to 1.4 2) If IOPE.EQ.2 parameter is DELTA EQ.0; default set to 0.5
(2)	21 - 30	OPVAR(2)	Second Integration Parameter 1) If IOPE.EQ.1 parameter is not applicable 2) If IOPE.EQ.2 parameter is ALPHA EQ.0; default as specified in note below

NOTES/

(1) For static problems (IMASS.EQ.0) this card is read, but the information is not used. For dynamic analysis IOPE determines which time integration algorithm will be used in the step-by-step solution.

(2) OPVAR(1) and OPVAR(2) store the integration parameters associated with the time integration method specified by IOPE.

a) Wilson's θ-method uses one parameter (THETA), which is usually specified as 1.4.

b) Newmark's method uses two parameters. DELTA is specified by the user (or the default value of 0.5 is used) and ALPHA is either set to a default value by the formula

ALPHA = 0.25*(DELTA + 0.50)**2,

or specified directly by the user.

II. MASTER CONTROL CARDS (continued)

Card 6 (4I5)

note	columns	variable	entry
(1)	1 - 5	NPB	Number of blocks of displacement/ velocity/acceleration printout EQ.0, print all nodal point components
(2)	6 - 10	IDC	Displacement printout code EQ.0, no displacement printout EQ.1, print displacements
(2)	11 - 15	IVC	Velocity printout code EQ.0, no velocity printout EQ.1, print velocities
(2)	16 - 20	IAC	Acceleration printout code EQ.0, no acceleration printout EQ.1, print accelerations

NOTES/

(1) For large meshes it is usually not necessary to print displacements, velocities, and accelerations at every node. Hence, nodes for which printout is desired are grouped into NPB printout blocks. Each block of nodes is defined by the node numbers of the first and last node in the block (see next card).

If NPB.EQ.0 all nodal quantities are printed regardless of the values of IDC, IVC, and IAC.

(2) The displacement solution at the nodes within the blocks is printed if IDC.EQ.1. In dynamic problems the velocity and/or acceleration solutions also are printed if IVC.EQ.1 and/or IAC.EQ.1.

II. MASTER CONTROL CARDS (continued)

Card 7 (16I5)

note	columns	variable	entry
(1)	1 - 5	IPNODE(1,1)	First node of printout block no. 1.
	6 - 10	IPNODE(2,1)	Last node of printout block no. 1.
	11 - 15	IPNODE(1,2)	First node of printout block no. 2.
		etc.	

NOTES/

(1) Two entries are expected for each printout block, namely, the first node of the block and the last node of the block. All nodal points between these two nodes will be included in the printout block.

If NPB.EQ.0 leave this card blank.

III. NODAL POINT DATA (A1,I4,A1,I4,5I5,3F10.0,I5)

note	columns	variable	entry
(1)	1	CT	Symbol describing the coordinate system for this node; EQ. ; (blank) cartesian (X,Y,Z) EQ.X; X-cylindrical
(2)	2 - 5	N	Node (joint) number; GE.1 and LE.NUMNP
(3)	6	PSF	Print suppression flag (ignored unless N.EQ.1); EQ. ; (blank) no suppression EQ.A; suppress ordered list of node coordinates EQ.B; suppress list of equation numbers EQ.C; both A and B, above
(4)	7 - 10	ID(1,N)	X-translation boundary code
	11 - 15	ID(2,N)	Y-translation boundary code
	16 - 20	ID(3,N)	Z-translation boundary code
	21 - 25	ID(4,N)	X-rotation boundary code
	26 - 30	ID(5,N)	Y-rotation boundary code
	31 - 35	ID(6,N)	Z-rotation boundary code
(5)	36 - 45	X(N)	X (or z)-coordinate
	46 - 55	Y(N)	Y (or r)-coordinate
	56 - 65	Z(N)	Z (or θ)-coordinate (degrees)
(6)	66 - 70	KN	Node number increment for node data generation; EQ.0; no generation

NOTES/

(1) Special cylindrical coordinate systems are allowed for the specification of node coordinates. If an "X" is entered in card column one (1), then the entries in cc 36-65 are with respect to a cylindrical (r,θ,z) system rather than the standard cartesian (X,Y,Z) system. The generator (i.e., the z-axis) of the "X-cylindrical" system coincides with the global X-axis, and θ is zero (0) in the global X-Y plane--

z-axis	θ-zero plane	TRANSFORMATIONS
X	X-Y	$X = z$ $Y = r\cos\theta$ $Z = r\sin\theta$

Cylindrical input is merely a user convenience for locating nodes in the (X,Y,Z) system, and no other references to the cylindrical system are implied; i.e., boundary conditions, output displacements, etc. are still referenced to the (X,Y,Z) system.

(2) Nodal data must be defined for all (NUMNP) nodes. Node data may be input directly (i.e., each node on its own individual card) or the generation option may be used if applicable (see note 6, below). Admissible node numbers range from "1" to the total number of nodes (NUMNP). Node numbers may not be repeated or omitted. The last node that is input must be "NUMNP".

(3) The print suppression flag (PSF) is used to eliminate the second printing of ordered node coordinates or to suppress printing of equation number assignments (or both). The PSF character is entered on the card for node one (1) only.

(4) Boundary condition codes can only be assigned the following values (M=1,2,...,6)--

ID(M,N) = 0; unspecified (free) displacement

ID(M,N) = 1; deleted (fixed) displacement

ID(M,N) = -1; deleted (fixed) displacement, used for generation

NOTES/(4) (continued)

An unspecified [ID(M,N)=0] degree of freedom is free to translate or rotate as the solution dictates. Concentrated forces (or moments) may be applied in this degree of freedom.

One system equilibrium equation is required for each unspecified degree of freedom in the model. The maximum number of equilibrium equations is always less than six (6) times the total number of nodes in the system.

Deleted [ID(M,N)=1] degrees of freedom are removed from the final set of equilibrium equations. Deleted degrees of freedom are used to define fixities (points of external reaction), and any loads applied in these degrees of freedom are ignored by the program. Nodes that are used for geometric reference only (i.e., nodes not assigned to elements) must have all six (6) degrees of freedom deleted. Nodal degrees of freedom having undefined stiffness (such as rotations in all TRUSS model, out-of-plane components in a two-dimensional model, etc.) must be deleted.

Independent of the actual entries posted for the ID(M,N) in cc 7-35 of the Nodal Data, any master boundary condition deletions (i.e., IDOF(M).EQ.1, M=1,2,...,6) which are given in cc 6-11 of Card 1 in Section II will be used for all nodes. Suppose that all rotations X,Y,Z have been deleted by means of the master codes (i.e., IDOF(M).EQ.1, M=4,5,6), then ID(M,N) (M=4,5,6) will be set to "1", and data in cc 21-35 is ignored by the program.

Two dimensional elements must use the Y-Z plane, i.e. plane strain, plane stress, and axisymmetric elements can only be defined in the Y-Z plane, and nodal degrees of freedom having undefined stiffness must be deleted.

For generation purposes the value ID(M,N)=-1 can also be used. In this case, if the corresponding value on the next input card is zero (0), it is set equal to "-1". Considering the deletion of degrees of freedom a minus one (-1) has the same meaning as a plus one (+1).

(5) For the case CT (cc 1) equal to the character "X", the data input in cc 36-65 are interpreted as the cylindrical (r,θ,z) coordinates of node "N". The table in note (1) contains the formulae used by the program to compute the cartesian (X,Y,Z) coordinates of node N from the cylindrical coordinate values given in cc 36-65. The origin of cartesian and cylindrical systems is the same point. Cylindrical coordinate data (if input on the card) are printed as read in the first list of node coordinates. The second listing of coordinates

NOTES/(5) (continued)

(if not suppressed with a PSF of "A" or "C") is an ordered printing of all nodes with coordinates converted to the (X,Y,Z) system. Note that boundary condition codes always reference the (X,Y,Z) system even if the node happens to be located using the cylindrical coordinate option.

(6) Node cards need not be input in node order sequence; eventually, however, all nodes in the set [1,NUMNP] must be defined. Node data for a series of nodes

$$[N_1,\ N_1+1*KN_1,\ N_1+2*KN_1,\ \ldots,\ N_2]$$

may be generated from information given on two (2) cards in sequence--

CARD 1 -- $CT_1,N_1,ID(N_1,1),\ldots,ID(N_1,6),X(N_1),\ldots,KN_1$

CARD 2 -- $CT_2,N_2,ID(N_2,1),\ldots,ID(N_2,6),X(N_2),\ldots,KN_2$

KN_1 is the node generation parameter given on the first card in the sequence. The first generated node is N_1+1*KN_1; the second generated node is N_1+2*KN_1, etc. Generation continues until node number N_2-KN_1 is established. Note that the node difference N_2-N_1 must be evenly divisible by KN_1.

In the generation the boundary condition codes (ID(L,J) values) of the generated nodes are set equal to those of node N_1. The coordinate values (cartesian (X,Y,Z) or cylindrical (r,θ,z)) are interpolated linearly. Note that coordinate generation is only possible if both nodes N_1 and N_2 are given in the same coordinate system.

IV. APPLIED LOADS DATA (continued)

2. Load function data

Input NLCUR sets of the following data cards in order of increasing load function number.

a. Control data (2I5)

note	columns	variable	entry
	1 5	NTF	Time function number; GE.1 and LE.NLCUR
	6 - 10	NPTS	Number of points (i.e., f(t), t pairs) used to input this time function; GE.2 and LE.NPTM

b. [f(t),t] data (8F10.0)

note	columns	variable	entry
(1)	1 - 10	TIMV(1)	Time at point 1, t_1
	11 - 20	RV (1)	Function value at point 1, $f(t_1)$
	21 - 30	TIMV(2)	Time at point 2, t_2
	31 - 40	RV (2)	Function value at point 2, $f(t_2)$
	. . .	. . .	. . .
	71 - 80	RV (4)	Function value at point 4, $f(t_4)$
	Next card (if required)		
(2)	1 - 10	TIMV(5)	Time at point 5, t_5
	11 - 20	RV (5)	Function value at point 5, $f(t_5)$
	. . .	. . .	. . .

NOTES/

(1) Time values at successive points must increase in magnitude (i.e., TIMV(1)<TIMV(2)<TIMV(3), etc.), and TIMV(1) must be equal to zero (i.e., TIMV(1).EQ.0.0). The last time value for the function [i.e., TIMV(NPTS)] must be greater than or equal to the time at the end of solution; i.e., TIMV(NPTS) ≥ TSTART + NSTE*DT otherwise an error condition is declared.

(2) Input as many cards in this section as are required to define NPTS points, four points per card.

IV. APPLIED LOADS DATA (continued)

3. Nodal Loads Data (3I5,F10.0)

Skip this section if NLOAD.EQ.0; otherwise input NLOAD cards in this section.

note	columns	variable	entry
(1)	1 - 5	NOD	Node number to which this load is applied; GE.1 and LE.NUMNP
	6 - 10	IDIRN	Degree of freedom number for this load component; EQ.0; solution terminated EQ.1; X-translation EQ.2; Y-translation EQ.3; Z-translation EQ.4; X-rotation EQ.5; Y-rotation EQ.6; Z-rotation
	11 - 15	NCUR	Load curve number that describes the time dependence of the load; GE.1 and LE.NLCUR
	16 - 25	FAC	Function multiplier used to scale f(t) for the load at "t"; EQ.0.0; default set to "1.0"

NOTES/

(1) If the same degree of freedom (IDIRN) at the same node (NOD) is given a multiple number of times, the program combines the loads algebraically with no error diagnostic.

V. RAYLEIGH DAMPING SPECIFICATION (2F10.0)

Omit this card if IDAMP.EQ.0

note	columns	variable	entry
(1)	1 - 10	ADAMP	Rayleigh damping coefficient α
(1)	11 - 20	BDAMP	Rayleigh damping coefficient β

NOTES/

(1) Rayleigh damping is defined as $C = \alpha M + \beta K$, where α and β are input as above.

It need be noted that β is applied to the linear stiffness matrix of the element assemblage.

VI. CONCENTRATED NODAL MASSES (I10,6F10.0)

Skip this section if IMASSN.EQ.0, on card 2 of Section II. Otherwise, input IMASSN cards as follows.

note	columns	variable	entry
(1)	1 - 10	N	Node number; GE.1 and LE.NUMNP
(2)	11 - 20	XMASS(1)	X-direction mass
	21 - 30	XMASS(2)	Y-direction mass
	31 - 40	XMASS(3)	Z-direction mass
(3)	41 - 50	XMASS(4)	X-rotational mass
	51 - 60	XMASS(5)	Y-rotational mass
	61 - 70	XMASS(6)	Z-rotational mass

NOTES/

(1) Input IMASSN cards; node order is not important. Repeating nodes accumulates mass at the node.

(2) Mass components input for deleted (or non-existant) degrees of freedom are ignored by the program without a diagnostic message.

(3) Rotational degrees of freedom are currently not used (see Table II.1).

VII. CONCENTRATED NODAL DAMPERS (I10,6F10.0)

Skip this section if IDAMPN.EQ.0, on card 2 of Section II, above. Otherwise, input IDAMPN cards as follows.

note	columns	variable	entry
(1)	1 - 10	N	Node number; GE.1 and LE.NUMNP
(2)	11 - 20	XDAMP(1)	X-direction damper
	21 - 30	XDAMP(2)	Y-direction damper
	31 - 40	XDAMP(3)	Z-direction damper
(3)	41 - 50	XDAMP(4)	X-rotation damper
	51 - 60	XDAMP(5)	Y-rotation damper
	61 - 70	XDAMP(6)	Z-rotation damper

NOTES/

(1) Input IDAMPN cards; node order is not important. Repeating nodes accumulates dampers at the node.

(2) Damper components input for deleted (or non-existent) degrees of freedom are ignored by the program without a diagnostic message.

(3) Rotational degrees of freedom are currently not used (See Table II.1).

VIII. INITIAL CONDITIONS

Initial conditions for the element are defined in this section. Initial conditions may be established using one (1) of three (3) methods--

METHOD 1 - For MODEX.EQ.2, this is a restart job. Refer to Appendix A for setting up a restart job. The variable "ICON" appearing on the card below is read by the program, but ignored; i.e., the control card (Section VIII.a) must still be input.

METHOD 2 - For MODEX.NE.2, and initial conditions of all zero, input ICON.EQ.0 with no additional data; all vector components are then automatically initialized to zero at time of solution start, TSTART.

METHOD 3 - For MODEX.NE.2, and known non-zero initial conditions, input ICON.EQ.1 and read the system vectors in compacted form from cards as described in Section VIII.b, below.

a. Control Card (I5)

note	columns	variable	entry
(1)	1 - 5	ICON	Flag indicating the type of initial conditions; EQ.0 and MODEX.NE.2, zero initial conditions are generated automatically EQ.1 and MODEX.NE.2, non-zero initial conditions are read from data cards immediately following

b. Card Input of System Vectors (6E12.6)

For the case MODEX.NE.2 and ICON.EQ.1 , the program performs the following read operations:

```
      READ (5,1000)  (DIS(K),K=1,NEQ)
      READ (5,1000)  (VEL(K),K=1,NEQ)
      READ (5,1000)  (ACC(K),K=1,NEQ)
 1000 FORMAT (6E12.6)
```

where DIS/VEL/ACC are the system initial displacement/

velocity/acceleration vectors, respectively. The variable NEQ is the total number of freedoms retained for evaluation; i.e., six (6) times the total nodes minus (-) all deletions provided by fixed boundary condition specifications.

The list of equation numbers can be obtained in Section III (variable PSF) and can be identified conveniently from the displacement (velocity and acceleration) print-out of a previous solution.

For the case of a static solution, the VEL/ACC system initial vectors are not read from card input. A static solution is performed if IMASS.EQ.0 (Section II, card 2).

ELEMENT INPUT

Input as many blocks of data in these sections as there are total element groups. Linear element groups (NEGL,total) are input first, and nonlinear elements (NEGNL groups, total) follow the linear element group data. Therefore, whether the elements in a group are linear or nonlinear, depends on whether the element group belongs to the first NEGL groups or the last NEGNL groups.

In any one group all elements input must be the same type; e.g., if nonlinear TRUSS elements are given as input, then all elements in the group must be nonlinear. Furthermore, in any one group, only one material model can be used, e.g., if the group consists of TRUSS elements, then either the material of all elements in the group is linear elastic, or it is nonlinear elastic. However, a number of different sets of material constants for a specific model can be used.

Since the program is an overlay system, in order to avoid unnecessary manipulation of overlays, it is most efficient to group all element groups of one kind and using one material model together.

IX. TRUSS ELEMENTS

TRUSS elements are two-node members allowed arbitrary orientation in the X,Y,Z system. The TRUSS transmits axial force only, and in general is a six (6) degree of freedom element (i.e., three global translation components at each end of the member), see Fig. IX.1.

1. Element control card (20I4)

note	columns	variable	entry
	1 - 4	NPAR(1)	Enter the number "1"
(1)	5 - 8	NPAR(2)	Number of TRUSS elements in this group; GE.1
(2)	9 - 12	NPAR(3)	Type of nonlinear analysis EQ.0 default set to "1" EQ.1 materially nonlinear only EQ.2 updated Lagrangian formulation
(3)	56 - 60	NPAR(15)	Material model number EQ.1 linear elastic EQ.2 nonlinear elastic
(4)	61 - 64	NPAR(16)	Number of different sets of section/material properties GE.1 EQ.0, default set to "1"
(5)	65 - 68	NPAR(17)	Number of material model constants per set EQ.0 if NPAR(15).EQ.1 GE.4 if NPAR(15).EQ.2

NOTES/

(1) TRUSS element numbers begin with one (1) and end with the total number of elements in this group, NPAR(2). Element data are input in Section IX.4, below.

(2) The parameter NPAR(3) is applicable only if the element group is nonlinear. If NPAR(3).EQ.1, no geometric nonlinearities are taken into account, i.e. the geometric stiffness matrix is not included. If NPAR(3).EQ.2 large displacement effects are included in the analysis, but small strains are assumed in the calculation of element forces.

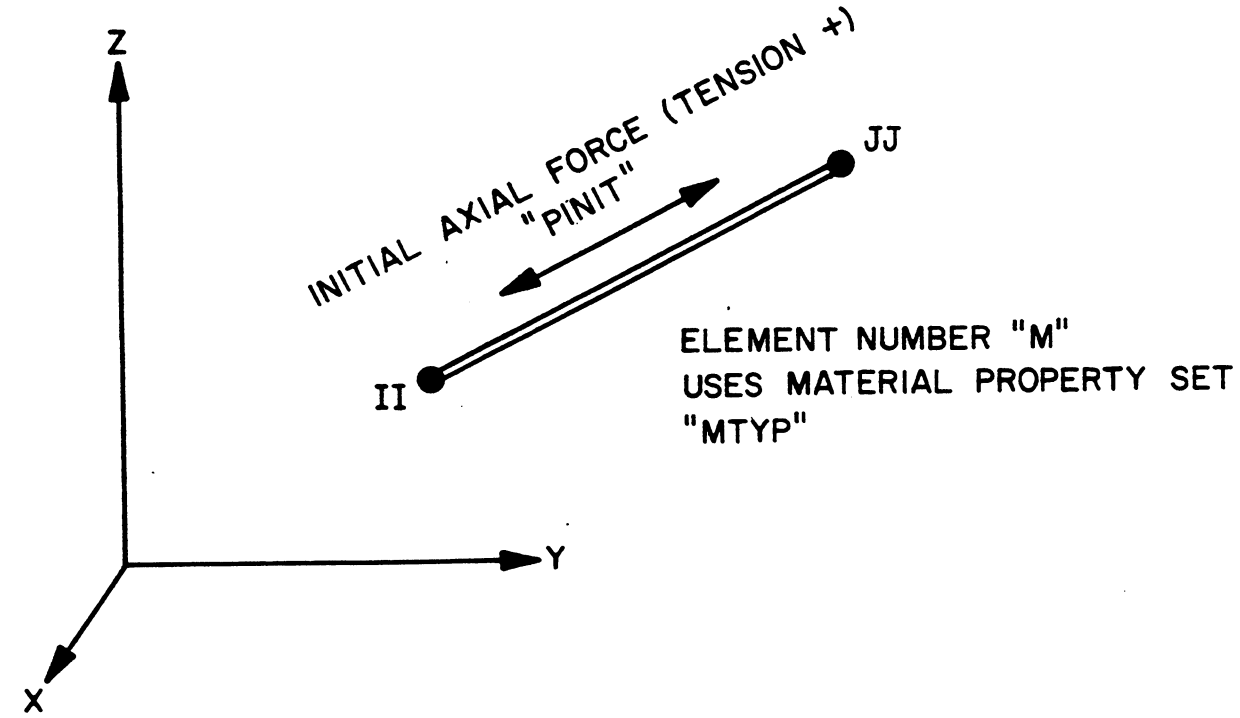


FIGURE IX.1 TRUSS ELEMENT

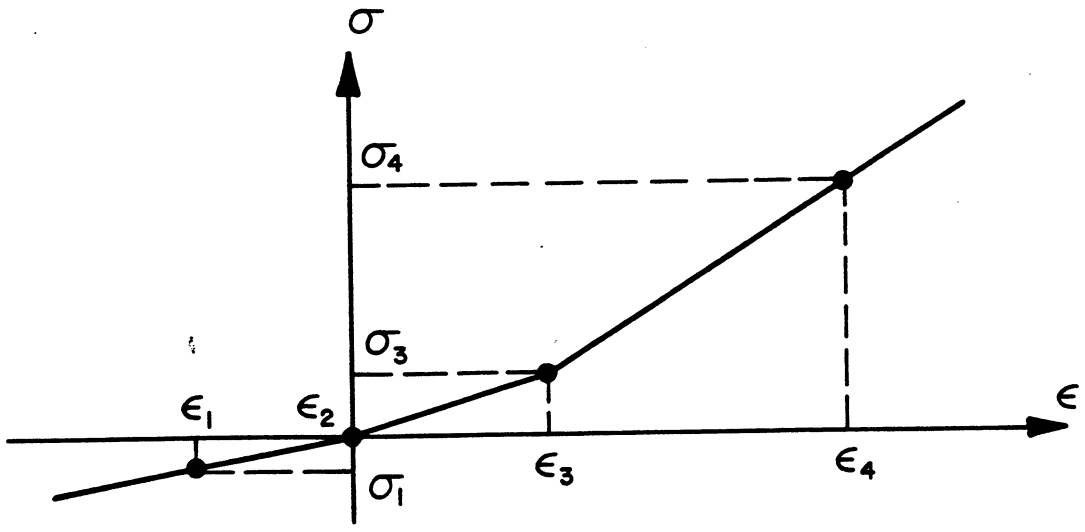


FIGURE IX.2 NONLINEAR MATERIAL MODEL FOR TRUSS

NOTES

(3) In any one element group only one material model can be used, and this model type is defined by the entry NPAR(15). If NPAR(15).EQ.1 the model is defined by Young's modulus only and NPAR(17).EQ.0. If NPAR(15).EQ.2 the stress-strain curve is input in Section IX.3 below.

The model defined for the element group must be consistent with the nonlinear formulation used (defined by NPAR(3)) and the requirement of equilibrium iteration as defined on card 4 of the Master Control Cards (Section II). As stated in note (3) of card 4, Section II, equilibrium iterations can only be performed if the model allows for iteration, and if at least one nonlinear element group is used in the analysis. Table IX.1 summarizes the formulations, material models and possibilities of equilibrium iteration that may be used.

(4) The variable NPAR(16) defines the number of sets of material/section properties to be read in Sections IX.2 and IX.3 below.

(5) NPAR(17) is the variable NCON used in Section IX.3(c).

FORMULATION	MATERIAL MODEL	POSSIBILITY OF EQUILIBIRUM ITERATION
LINEAR ANALYSIS	LINEAR ELASTIC	NO
MATERIALLY NONLINEAR ONLY	NONLINEAR ELASTIC	YES
UPDATED LAGRANGIAN FORMULATION	LINEAR ELASTIC	YES
	NONLINEAR ELASTIC	YES

TABLE IX.1 POSSIBLE ANALYSES USING TRUSS ELEMENTS

2. Linear Elastic Material/Section Property Cards

Skip this set of cards if NPAR(15).NE.1 .
Otherwise read NPAR(16) sets of cards.

a. material number card (I5)

columns	variable	entry
1 - 5	N	Material/section number GE.1; and LE.NPAR(16)

b. property card (4F10.0)

note	columns	variable	entry
(1)	1 - 10	E(N)	Young's modulus
	11 - 20	AREA(N)	Cross-sectional area
	21 - 30	DEN(N)	Mass density (for dynamic analysis)
	31 - 40	STRAI(N)	Initial strain (axial)

NOTES/

(1) NPAR(16) different linear elastic materials are input in this section, provided NPAR(15).EQ.1. Note that one material/section is defined to have the same Young's modulus, area, mass density and initial strain.

3. Nonlinear Elastic Material/Section Property Cards

Skip this set of cards if NPAR(15).NE.2, otherwise read NPAR(16) sets of cards.

a. material number card (I5)

columns	variable	entry
1 - 5	N	Material/section number GE.1 and LE.NPAR(16)

b. section property card (3F10.0)

note	columns	variable	entry
(1)	1 - 10	AREA(N)	Cross-sectional area
	11 - 20	DEN(N)	Mass density for dynamic analysis
	21 - 30	STRAI(N)	Initial axial strain

c. stress-strain curve card (8F10.0)

note	columns	variable	entry
(2)	1 - 10	PROP(1,N)	Strain at point 1, e^1
	11 - 20	PROP(2,N)	Strain at point 2, e^2
	.	.	.
	.	.	.
	.	.	.
		PROP(NCON/2,N)	Strain at point NCON/2
		PROP(NCON/2+1,N)	Stress at point 1, σ^1
		PROP(NCON/2+2,N)	Stress at point 2, σ^2
		.	.
		.	.
		.	.
		PROP(NCON,N)	Stress at point NCON/2

NOTES/

(1) One section property card is defined to have the same area, density and initial strain.

NOTES/(continued)

(2) The stress-strain curve is defined by straight lines between the input points (e^i,σ^i). From the stress-strain curve total stresses and the tangent modulus are evaluated for a given strain (see Fig. IX.2).

The variable NCON was defined in Section IX.1 by the variable NPAR(17).

This model can only be used in a nonlinear element group.

4. Element Data Cards (4I5,F10.0,3I5)

NPAR(2) elements must be input and/or generated in this section in ascending sequence beginning with "1".

note	columns	variable	entry
	1 - 5	M	TRUSS element number; GE.1 and LE.NPAR(2)
(1)	6 - 10	II	Node number at one end
	11 - 15	JJ	Node number at other end GE.1 and LE.NUMNP
	16 - 20	MTYP	Material property set number; GE.1 and LE.NPAR(16)
(2)	21 - 30	PINIT	Initial axial force in the TRUSS; GT.0, tension
	31 - 35	IPS	Flag for printing axial stress in TRUSS element; EQ.0, no printing EQ.1, print element stress
(3)	36 - 40	KG	Node generation increment used to compute node numbers for missing elements; EQ.0, default set to "1"

NOTES/

(1) Refer to Figure IX.1.

(2) PINIT is the axial force in the TRUSS at zero node displacements, and zero initial strain in the truss. The initial strain defined for the material property set number gives rise to an additional force in the truss.

(3) Elements must be input in increasing element number order. If cards for elements [M+1,M+2,...,M+J] are omitted, these "J" missing elements are generated using MTYP and PINIT of element "M" and by incrementing the node numbers of successive elements with the value "KG"; KG is taken from the first card of the element generation sequence (i.e., from the "M-th" element card).

X. 2/D CONTINUUM ELEMENTS

2/D CONTINUUM elements are 4- to 8-node isoparametric quadrilaterals which must be input in the global Y-Z plane. Figure X.1 shows some typical 2/D CONTINUUM elements. The element can represent either planar (plane stress or plane strain condition) or axisymmetric solids, as illustrated in Fig. X.2. In both cases, each element node has two (2) translational degrees of freedom.

When the element is used to represent an axisymmetric solid or shell, the global Z-axis is the axis of revolution. All elements must be located in the +Y half-plane. These conventions are illustrated in Fig. X.2.

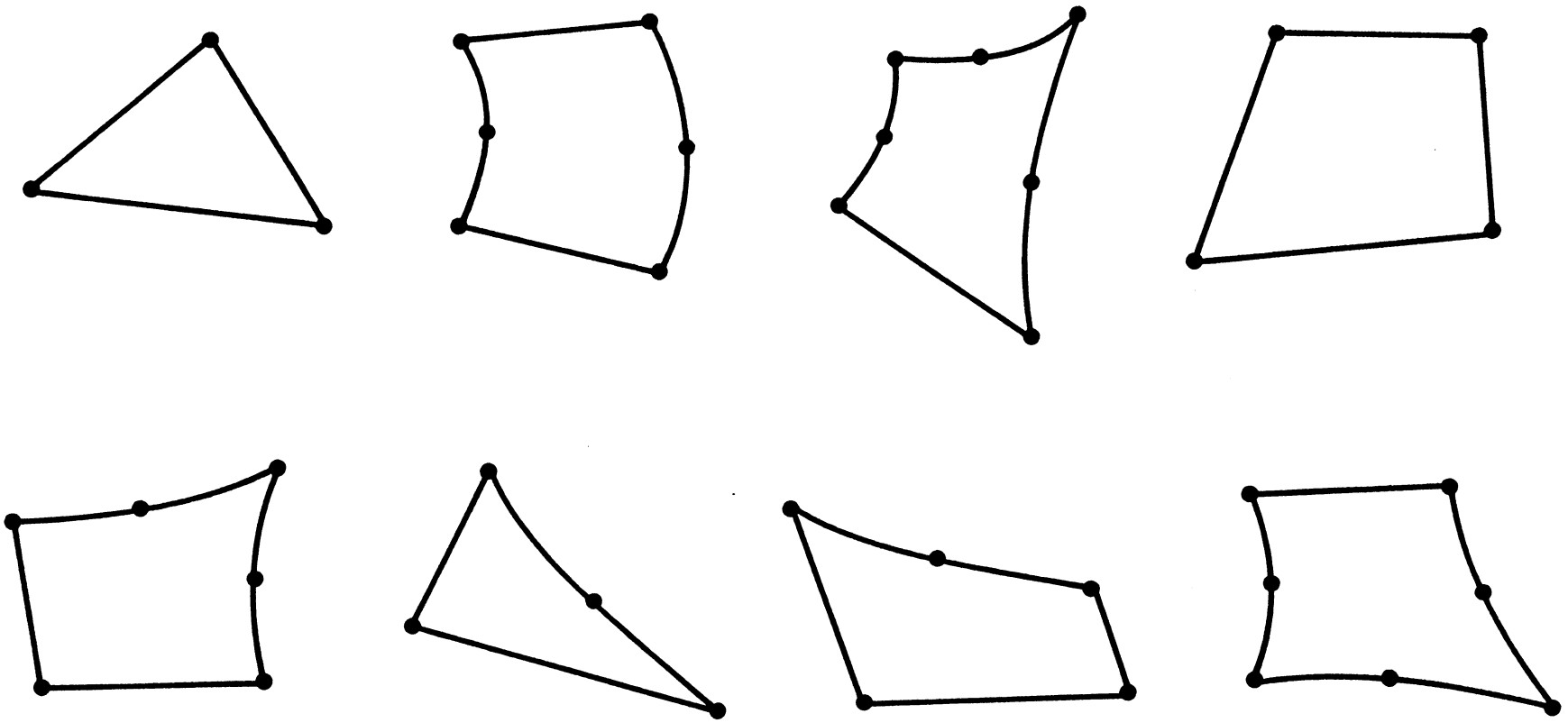
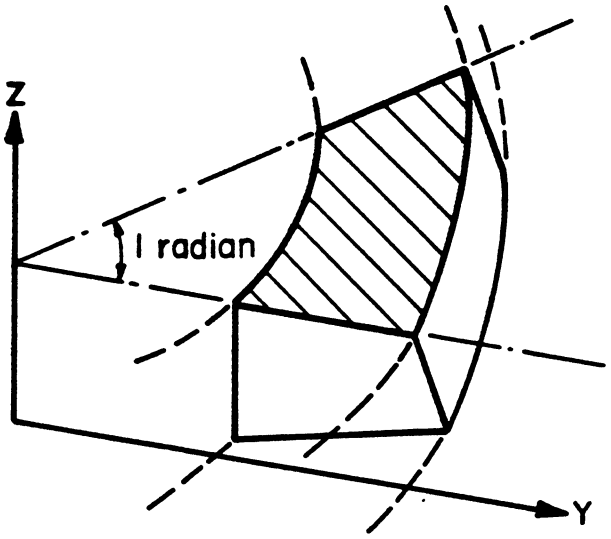
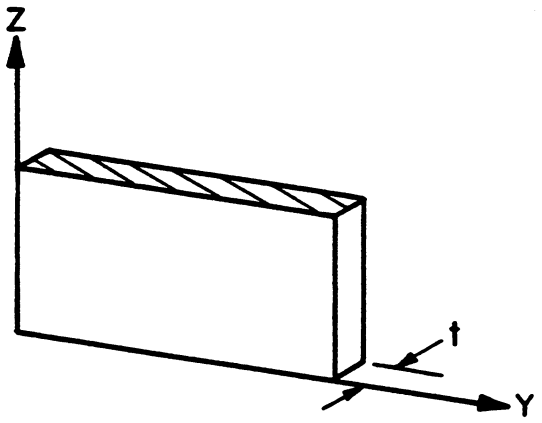


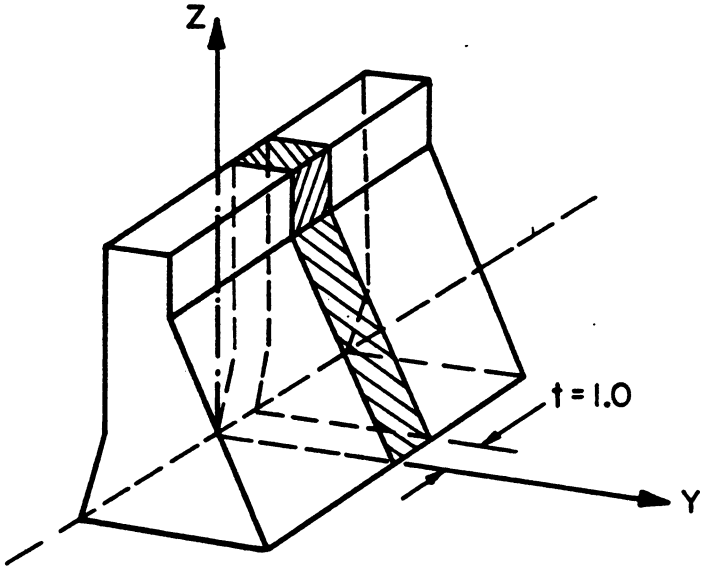
FIGURE X.1 TYPICAL NODAL POINT CONFIGURATIONS FOR 2/D CONTINUUM ELEMENTS DERIVED FROM THE GENERAL 8-NODE ELEMENT IN FIGURE X.6



AXISYMMETRIC FINITE ELEMENT MODEL OF A RING



PLANE STRESS FINITE ELEMENT MODEL OF A CANTILEVER



PLANE STRAIN FINITE ELEMENT MODEL OF A DAM

FIGURE X.2 POSSIBLE TWO DIMENSIONAL FINITE ELEMENT ANALYSES

X. 2/D CONTINUUM ELEMENTS (continued)

1. Element Group Control Card (20I4)

note	columns	variable	entry
	1 - 4	NPAR(1)	Enter the number "2"
(1)	5 - 8	NPAR(2)	Number of 2/D CONTINUUM elements in this group; GE.1
(2)	9 - 12	NPAR(3)	Code indicating type of nonlinear analysis; EQ.0, default set to "1" EQ.1, material nonlinear analysis only EQ.2, total Lagrangian EQ.3, updated Lagrangian
	17 - 20	NPAR(5)	Element type code; EQ.0, axisymmetric EQ.1, plane strain EQ.2, plane stress
(3)	25 - 28	NPAR(7)	Maximum number of nodes used to describe any one element; GE.4 and LE.8 EQ.0, default set to "8"
(4)	37 - 40	NPAR(10)	Numerical integration order to be used in Gauss quadrature formulae; EQ.0, default set to "2" GE.2 and LE.4
(5)	49 - 52	NPAR(13)	Number of stress output location tables; EQ.0, print stresses at integration points
(6)	57 - 60	NPAR(15)	Material model number; GE.1 and LE.12 EQ.1, linear isotropic EQ.2, linear orthotropic EQ.3, variable tangent moduli EQ.4, curve description model EQ.5, curve description model with tension cut-off (plane strain only) EQ.6, elastic-plastic (von Mises)

X. 2/D CONTINUUM ELEMENTS (continued)

note	columns	variable	entry
			EQ.7, elastic-plastic (Drucker-Prager) EQ.8, incompressible nonlinear elastic (Mooney-Rivlin/ plane stress only) GE.9, user-supplied
(6)	61 - 64	NPAR(16)	Number of different sets of material properties; GE.1
(6)	65 - 68	NPAR(17)	Number of constants per property set; EQ.0, if NPAR(15).LT.9 GE.1, if NPAR(15).GE.9
(6)	69 - 72	NPAR(18)	Dimension of storage array required for element history; EQ.0, if NPAR(15).LT.9 GE.0, if NPAR(15).GE.9

NOTES/

(1) 2D/CONTINUUM element numbers begin with one (1) and end with the total number of elements in this group, NPAR(2). Element data are input in Section X.4 below.

(2) NPAR(3) is applicable for nonlinear element groups only and determines if geometrical nonlinearities are to be included in the analysis. If NPAR(3).EQ.1 displacements and strains are assumed to be infinitesimal. In the total Lagrangian and updated Lagrangian formulations all geometric effects (large displacements and large strains) are included in the analysis.

(3) NPAR(7) limits the number of nodes that can be used to describe any of the elements in this group. A minimum of 4 and a maximum of 8 nodes are used to describe the 2/D CONTINUUM elements. For example, for the set of elements shown in Fig. X.1, NPAR(7) would be set to "7". Constant strain triangles are obtained by having nodes 3 and 4 coincide.

(4) For rectangular elements an integration order of "2" is sufficient. If the element is distorted, a higher integration order need be used. Notice that apart from the larger computational effort in the calculation of the element matrices, more working storage may be required if a nonlinear material model is used (see note (6) below). The consistent mass matrix is always calculated with an integration order of 3.

NOTES/ (5) Element stresses are calculated at the points defined in the stress output location table assigned to the element (see Section X.3). NPAR(13) defines the total number of stress output location tables input in Section X.3. NPAR(13).EQ.0 for all nonlinear element models, i.e. MODEL.GT.2 in Section X.2.

(6) Only one material model (defined by the value of NPAR(15)) is allowed in an element group. If NPAR(15) is 1,2,...,8, the model exists in the current library for 2/D CONTINUUM elements, and the entries for NPAR(17) and NPAR(18) are ignored by the program. If, however, NPAR(15) is "9" or larger, material constants must be read into a storage array, "PROP", which has dimensions NPAR(17), NPAR(16); i.e., property constants are stored as,

```
((PROP(I,J),I=1,NPAR(17)),J=1,NPAR(16))
```

Non-library models require that NPAR(17) be given (input) as a positive integer. In addition, NPAR(18) must be specified for non-standard nonlinear models for use in allocating storage in a working array, "WA", which contains element history. The NPAR(18) parameters which characterize element history must be retained for materially nonlinear elements so that properties can be chosen in the current solution step. The working storage array (WA) is dimensioned IDWA by NPAR(2); where

```
IDWA   =  NPAR(18)*(NPAR(10)**2)
NPAR(2)  =  number of elements
```

i.e. for each integration point of each element, NPAR(18) storage locations are preserved during the solution. See Appendix B for a discussion on how to incorporate a new material model.

The model defined for the element group must be consistent with the nonlinear formulation used (defined by NPAR(3)), the element type (defined by NPAR(5)), and the requirement of equilibrium iteration as defined on card 4 of the Master Control Cards (Section II). As stated in note (3) of card 4, Section II, equilibrium iterations can only be performed if the model allows for iterations, and if at least one nonlinear element group is used in the analysis. Table X.1 summarizes the formulations, material models, and iteration possibilities for 2/D CONTINUUM elements.

TABLE X.1 POSSIBLE ANALYSES USING 2D/CONTINUUM ELEMENTS

FORMULATION	MATERIAL MODEL	POSSIBILITY OF EQUILIBRIUM ITERATION
LINEAR ANALYSIS	linear isotropic	NO
	linear orthotropic	
MATERIALLY NONLINEAR ONLY	variable tangent moduli model	NO
	curve description model (plane strain or axisymmetric only)	NO
	curve description model with tension weakening (plane strain only)	NO
	elastic-plastic, von Mises yield condition	YES
	elastic-plastic, Drucker-Prager yield condition	YES
TOTAL LAGRANGIAN FORMULATION	linear isotropic	YES
	linear orthotropic	YES
	variable tangent moduli model	NO
	elastic-plastic, von Mises yield condition	YES
	incompressible non-linear elastic, Mooney-Rivlin material, (plane stress only)	YES
UPDATED LAGRANGIAN FORMULATION	linear isotopic	YES

X. 2/D CONTINUUM ELEMENTS (continued)

2. Material Property Data

NPAR(16) sets of cards must be input in this section. Card "a" (material number card") is the same for all material models, but card(s) "b" ("material property card(s)") depend on the material model number (NPAR(15).

a. material number card (I5,F10.0)

note	columns	variable	entry
	1 - 5	N	Material property set number; GE.1 and LE.NPAR(16)
(1)	6 - 15	DEN(N)	Mass density of the material used in calculation of the mass matrix, ρ; GE.0.0

NOTES/

(1) The mass density defined is used directly in the calculation of the element matrix, i.e., no acceleration constants are applied to the variable DEN(N).

b. material property card(s) (8F10.0)

For MODEL "1" (NPAR(15).EQ.1) → [linear isotropic]

note	columns	variable	entry
(1)	1 - 10	PROP(1,N)	Young's modulus, E
	11 - 20	PROP(2,N)	Poisson's ratio, ν

NOTES/

(1) MODEL 1 is a linear library model defined by two (2) positive constants (E,ν); i.e., if NPAR(15).EQ.1, NPAR(17) is set to "2" by default. MODEL 1 can be used with linear or nonlinear element groups. Since the material constants are independent of history, NPAR(18) is set to "0" by default.

For MODEL "2" (NPAR(15).EQ.2) → [linear orthotropic]

note	columns	variable	entry
(1)	1 - 10	PROP(1,N)	a-direction modulus, E_a
	11 - 20	PROP(2,N)	b-direction modulus, E_b
	21 - 30	PROP(3,N)	c-direction modulus, E_c
	31 - 40	PROP(4,N)	strain ratio, ν_{ab}
	41 - 50	PROP(5,N)	strain ratio, ν_{ac}
	51 - 60	PROP(6,N)	strain ratio, ν_{bc}
	61 - 70	PROP(7,N)	shear modulus, G_{ab}

NOTES/

(1) MODEL 2 is linear (i.e., NPAR(18).EQ.0 by default). The model requires seven(7) property constants; i.e., NPAR(17) .EQ.7 by default if NPAR(15).EQ.2. The constants (E_a,E_b, ...,G_{ab}) are defined in a material coordinate system (a,b,c) in which "c" corresponds to the out-of-plane direction (or global X), and the a-b plane coincides with the plane of the elements (Y-Z). a,b are principal material axes, and β is the angle (in degrees) between element edge 1-2 and the material +a-axis (Figure X.3). The material angle β is read individually for each element in Section X.4, below. MODEL 2 can be used with linear or geometrically nonlinear elements.

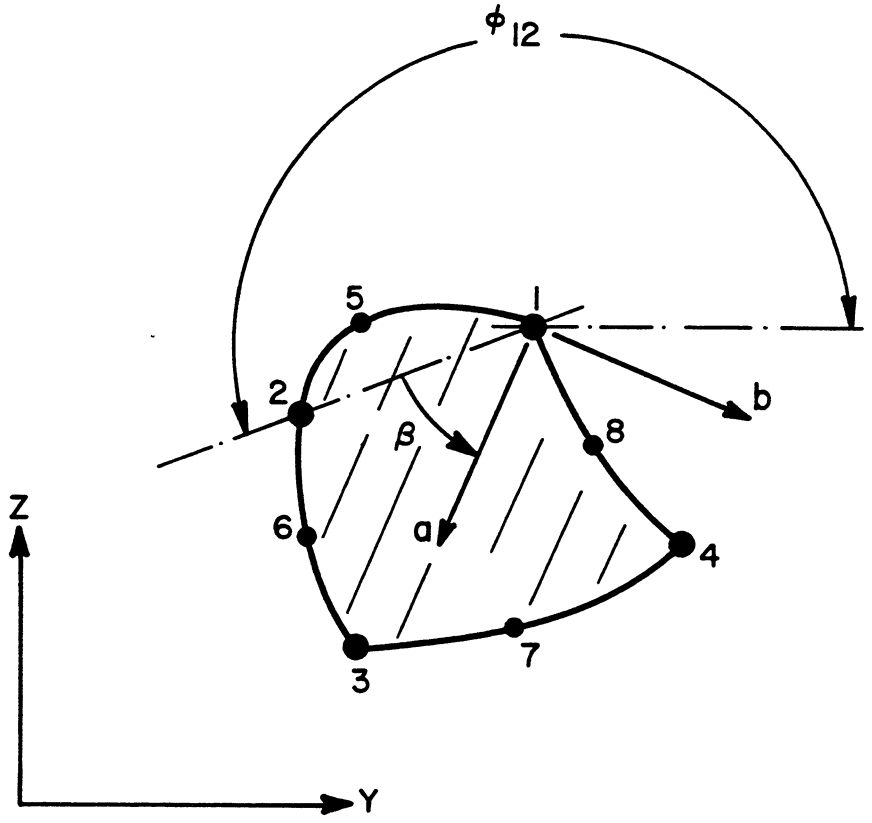


FIGURE X.3 PRINCIPAL IN-PLANE MATERIAL AXES ORIENTATION FOR THE LINEAR ORTHOTROPIC MATERIAL MODEL

For MODEL "3" (NPAR(15).EQ.3) → [variable tangent moduli]

note	columns	variable	entry	
(1)	1 - 10	PROP(1,N)	Unloading bulk modulus, K_{UN}	
	11 - 20	PROP(2,N)	K_0	Constants to define the loading bulk modulus
	21 - 30	PROP(3,N)	K_1	
	31 - 40	PROP(4,N)	K_2	
	41 - 50	PROP(5,N)	G_0	Constants to define the loading and unloading shear moduli
	51 - 60	PROP(6,N)	$\gamma_1 (<0)$	
	61 - 70	PROP(7,N)	$\bar{\gamma}_1 (<0)$	

NOTES/

(1) MODEL 3 is a nonlinear library material model requiring seven (7) constants (i.e., NPAR(17).EQ.7 by default) per property set. Ten variables, namely the previously calculated stresses, strains, the maximum pressure ever reached and a loading parameter, need be stored per integration point, i.e., NPAR(18) = 10 for this model.

The material model cannot be used in plane stress analysis.

NOTATION

For the notation used in the description of the nonlinear material models refer to the report "Static and Dynamic Geometric and Material Nonlinear Analysis," SESM Report No. 74-4 , February 1974.

For MODEL "4" or "5" (NPAR(15).EQ.4) or (NPAR(15).EQ.5) → [curve description model]

a. Material moduli cards (supply NPAR(16) sets of cards)

note	columns	variable	entry	
first card				
(1)	1 - 10	PROP(1,N)	e_V^1	volume strain at point "1"
	11 - 20	PROP(2,N)	e_V^2	....
	21 - 30	PROP(3,N)	e_V^3	
	31 - 40	PROP(4,N)	e_V^4	
	41 - 50	PROP(5,N)	e_V^5	
	51 - 60	PROP(6,N)	e_V^6	
	61 - 70	PROP(7,N)	K_{LD}^1	loading bulk modulus at point "1"
	71 - 80	PROP(8,N)	K_{LD}^2	....
second card				
	1 - 10	PROP(9,N)	K_{LD}^3	
	11 - 20	PROP(10,N)	K_{LD}^4	
	21 - 30	PROP(11,N)	K_{LD}^5	
	31 - 40	PROP(12,N)	K_{LD}^6	
	41 - 50	PROP(13,N)	K_{UN}^1	unloading bulk modulus at point "1"
	51 - 60	PROP(14,N)	K_{UN}^2	....
	61 - 70	PROP(15,N)	K_{UN}^3	
	71 - 80	PROP(16,N)	K_{UN}^4	
third card				
(2)	1 - 10	PROP(17,N)	K_{UN}^5	
	11 - 20	PROP(18,N)	K_{UN}^6	
	21 - 30	PROP(19,N)	G_{LD}^1	loading shear modulus at point "1"
	31 - 40	PROP(20,N)	G_{LD}^2	....
	41 - 50	PROP(21,N)	G_{LD}^3	
	51 - 60	PROP(22,N)	G_{LD}^4	
	61 - 70	PROP(23,N)	G_{LD}^5	
	71 - 80	PROP(24,N)	G_{LD}^6	

NOTES/

(1) MODELS 4 and 5 are nonlinear material models for which NPAR(17).EQ.24, by default. To characterize history, twelve (12) variables, namely, the previously calculated stresses and strains, the maximum volumetric strain ever reached in loading, ground pressure, the volume strain due to ground pressure, and the angle of "cracking" need be saved for each integration point, i.e. NPAR(18).EQ.12.

Note that

$$e_V^J > e_V^{J-1}, \quad K_{UN}^J \geq K_{LD}^J \quad \text{and} \quad G_{LD}^J < 1.5\, K_{LD}^J$$

where "J" is the J-th table input point. All six (6) input points must be defined, as shown in Fig. X.4. Linear interpolation is used to define the loading and unloading bulk moduli, ${}^tK_{LD}$, ${}^tK_{UN}$, and the loading shear modulus, ${}^tG_{LD}$, at time t. Note that these three cards must be supplied NPAR(16) times.

(2) The unloading shear modulus is calculated as $({}^tK_{UN}/{}^tK_{LD})\ {}^tG_{LD}$.

b. Tension cut-off definition card

This card defines the parameters to be used in the tension cut-off model, i.e., MODEL 5. Omit this card if MODEL.EQ.4.

note	columns	variable	entry
(1)	1 - 10	GAMMA	material density
(2)	11 - 20	STIFAC	normal stiffness reduction factor
	21 - 30	SHEFAC	shear stiffness reduction factor

NOTES/

(1) GAMMA is used to calculate the in-situ gravity pressure at the element integration points. It is assumed that the material provides a hydrostatic pressure distribution, calculated at an element integration point using

$$p = \sum_{i=1}^{N} h_i\, p_i$$

where h_i are the element interpolation functions, p_i is the pressure at the element nodes, and N is the total number of

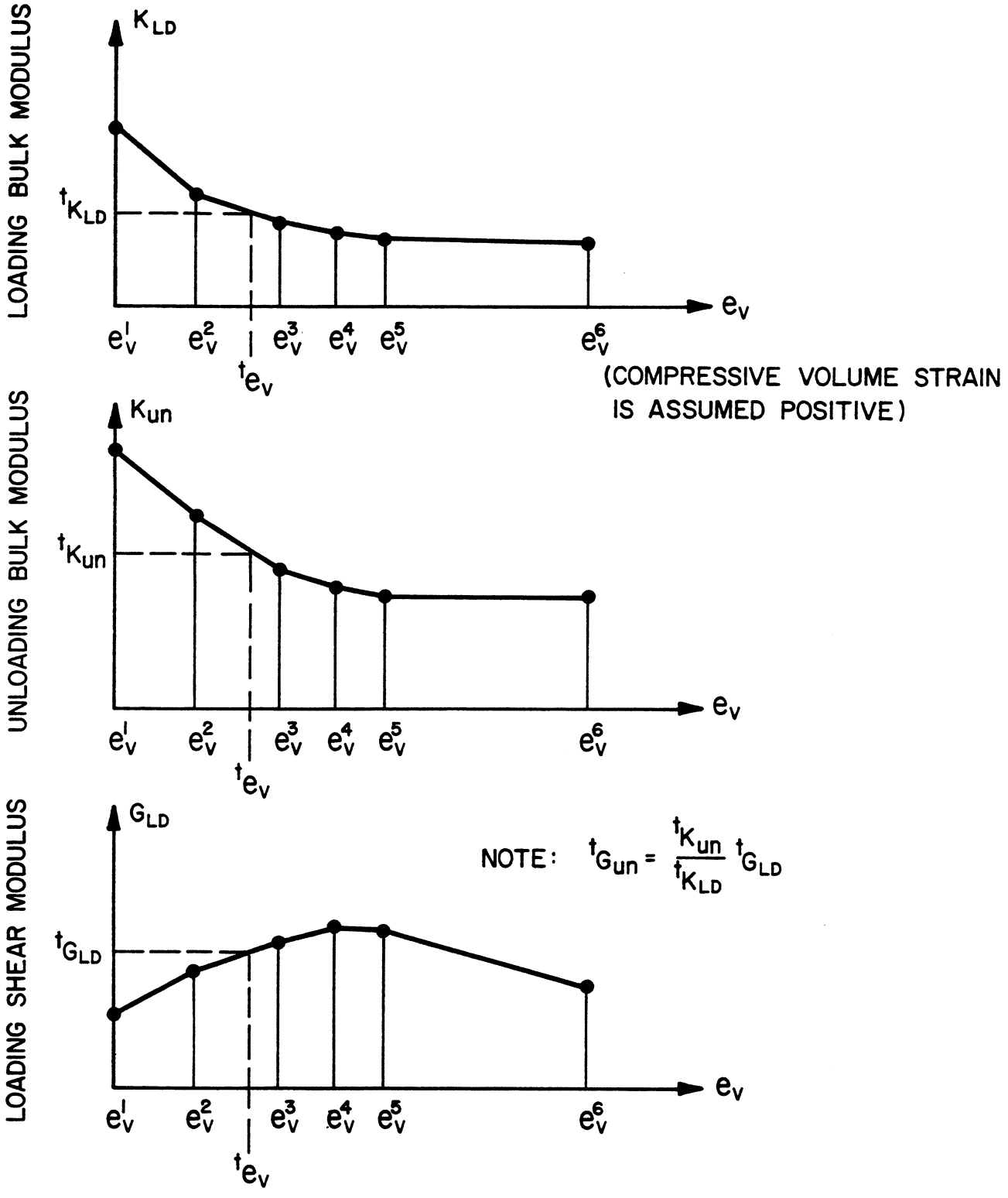


FIGURE X.4 MODULI VERSUS VOLUME STRAIN FOR THE CURVE DESCRIPTION MODEL

NOTES/(continued)

(1) element nodes. The pressure p_i is calculated using P_i = GAMMA * Z_i, where Z_i is the Z-coordinate of node i. This convention defines the assumed orientation of the Y-Z coordinate system. (All two-dimensional analyses must be carried out in the Y-Z plane.) From these hydrostatic pressures and the bulk moduli input above the in-situ volume strain at every integration point is calculated. When finding bulk and shear moduli from the curve description shown in Fig. X.4, in-situ volume strain is included. That is, for MODEL 5

$$^{t}e_V = {}^{t}e_V^P + {}^{t}e_V^L$$

where $^{t}e_V$ is total volume strain (abscissa of curves in Fig. X.4), $^{t}e_V^P$ is the in-situ volume strain, and $^{t}e_V^L$ is the volume strain due to applied loads. Note that for MODEL 4 $^{t}e_V$ is assumed to equal $^{t}e_V^L$.

(2) MODEL 5 contains the assumption that the material cannot withstand tensile stresses. When the tensile principal stress at an integration point exceeds the in-situ gravity pressure, a crack forms perpendicular to this principal stress and the stiffness across the crack is reduced by the factor STIFAC. Also, the shear stiffness at that integration point is reduced by the factor SHEFAC, which is meant to model the loss in shear strength due to cracking. Model 5 is to be used with plane strain elements only and unloading is not permitted.

For MODEL "6" (NPAR(15).EQ.6) → [elastic-plastic material, von Mises yield condition]

note	columns	variable	entry
(1)	1 - 10	PROP(1,N)	Young's modulus, E
(1)	11 - 20	PROP(2,N)	Poisson's ratio, ν
(2)	21 - 30	PROP(3,N)	Yield stress in simple tension, σ_y
(3)	31 - 40	PROP(4,N)	Strain hardening modulus, E_t

NOTES/

(1) MODEL 6 is a nonlinear library material model for which NPAR(17).EQ.4 by default. NPAR(18).EQ.10, since 10 history-dependent variables must be stored for each integration point.

The Young's modulus and Poisson's ratio define the initial elastic behavior of the material.

(2) The yield stress in simple tension defines the initial yield conditions.

(3) Linear isotropic strain hardening is assumed and E_t is the tangential modulus (after yield) obtained from a uniaxial tension test.

For MODEL "7" (NPAR(15).EQ.7) → [elastic-plastic material, Drucker-Prager yield condition]

note	columns	variable	entry
(1)	1 - 10	PROP(1,N)	Young's modulus, E
	11 - 20	PROP(2,N)	Poisson's ratio, ν
	21 - 30	PROP(3,N)	Angle of friction, θ
	31 - 40	PROP(4,N)	Cohesion, C

NOTES/

(1) MODEL 7 is a nonlinear library material model requiring four (4) constants (i.e., NPAR(17).EQ.4 by default) per material property set. Eleven (11) parameters need to be stored per integration point, i.e. NPAR(18).EQ.11 by default. The model assumes elastic perfectly plastic conditions.

For MODEL "8" (NPAR(15).EQ.8) → [incompressible nonlinear elastic material, Mooney-Rivlin model]

note	columns	variable	entry
(1)	1 - 10	PROP(1,N)	C_1
	11 - 20	PROP(2,N)	C_2

NOTES/

(1) MODEL 8 is a nonlinear library material model for which NPAR(17) is set to "2". Since the material is elastic, no history-dependent variables need be stored at integration points and NPAR(18) is set to "0". This rubber model can only be used in plane stress analysis.

For MODEL "≥9" (NPAR(15).GE.9) → (user-supplied material model)

note	columns	variable	entry
(1)	1 - 10	PROP(1,N)	
	11 - 20	PROP(2,N)	
	21 - 30	PROP(3,N)	
	⋮	etc.	

NOTES/

(1) MODEL "≥9" is a non-standard model (i.e., is not in the material model library) and requires that property data be read

(PROP(J,N),J=1,NPAR(17),N=1,NPAR(16))

where NPAR(17) is user-supplied (i.e., no default value exists). Subroutines which allocate storage locations (in "WA") for element history (if the model is nonlinear) and which calculate the incremental stress-strain law and current stresses must be supplied. For a discussion on how to incorporate a user-supplied material model, see Appendix B.

X. 2/D CONTINUUM ELEMENTS (continued)

3. Stress Output Table Cards (9I5)

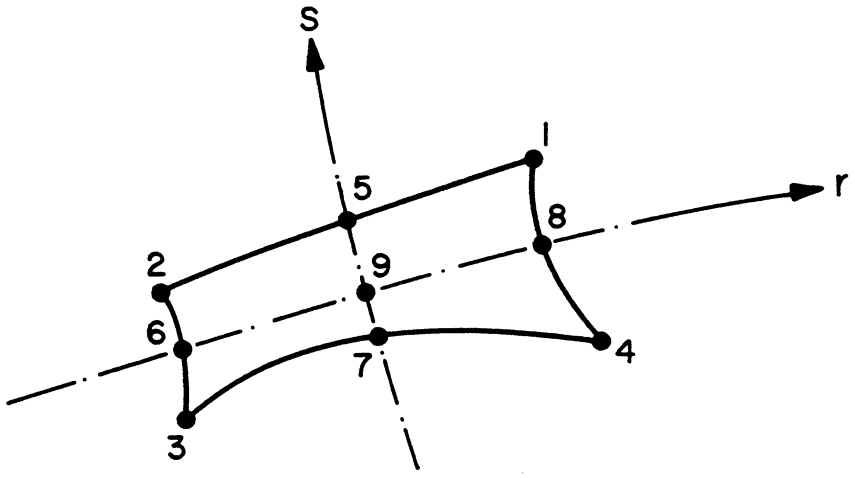
Skip this section if stresses at integration points are to be printed, i.e. NPAR(13).EQ.0; otherwise supply NPAR(13) cards.

note	columns	variable	entry
(1)	1 - 5	ITABLE(N,1)	Stress output location 1
	6 - 10	ITABLE(N,2)	Stress output location 2
	11 - 15	ITABLE(N,3)	Stress output location 3
	16 - 20	ITABLE(N,4)	Stress output location 4
	21 - 25	ITABLE(N,5)	Stress output location 5
	26 - 30	ITABLE(N,6)	Stress output location 6
	31 - 35	ITABLE(N,7)	Stress output location 7
	36 - 40	ITABLE(N,8)	Stress output location 8
	41 - 45	ITABLE(N,9)	Stress output location 9

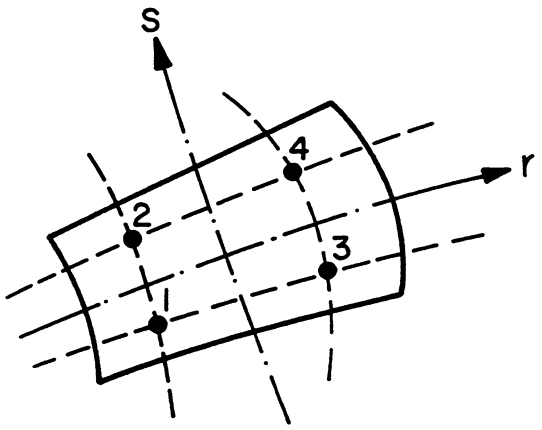
NOTES/

(1) Stress tables are defined to provide flexibility in element stress output requests. Each element can refer to one of the tables defined, and the element stresses are then calculated at the points specified in the table. Refer to Fig. X.5 for selection of the stress calculation points. The first "0" entry in a table will terminate that table. For example, if ITABLE(N,1).EQ.7 and ITABLE(N,2).EQ.0, then stresses will be printed at point 7 only whenever this stress table is referred to.

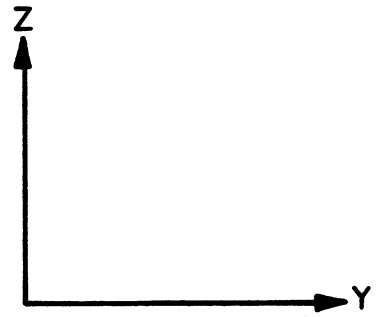
The stress tables are only used when MODEL "1" or "2" define the material behavior.

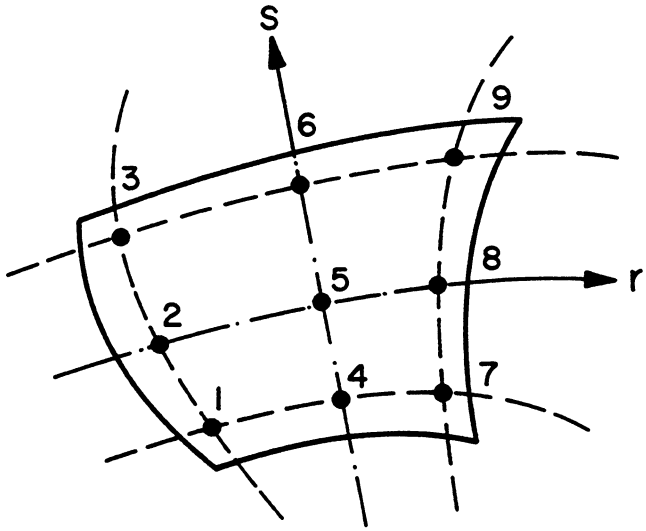


STRESS PRINTOUT TABLE POINTS FOR
2/D CONTINUUM ELEMENTS



2-POINT INTEGRATION





3-POINT INTEGRATION

PRINTING SEQUENCE FOR INTEGRATION
POINT STRESSES FOR 2/D CONTINUUM ELEMENTS

FIGURE X.5 STRESS PRINT-OUT CONVENTION
FOR 2/D CONTINUUM ELEMENTS

X. 2/D CONTINUUM ELEMENTS (continued)

4. Element Data Cards (I5,I3,I2,2F10,10I5)

note	columns	variable	entry
(1)	1 - 5	M	2/D CONTINUUM element number; GE.1 and LE.NPAR(2)
(2)	6 - 8	IEL	Number of nodes used to describe this element; EQ.0, default set to "NPAR(7)" LE.NPAR(7)
(3)	9 - 10	IPS	Number of the stress table to be used for stress calculations; EQ.0, no stress output for this element
	11 - 20	BET	Material angle, β, in degrees to be used in connection with material model "2" (linear orthotropic)
(4)	21 - 30	THIC	Element thickness; GT.0, plane stress only
	31 - 35	MTYP	Material property set number assigned to this element; GE.1 and LE.NPAR(16)
(1)	36 - 40	KG	Node generation parameter used to compute node numbers for missing elements (given on first card of a sequence); EQ.0, default set to "1"
(2)	41 - 45	NOD(1)	Global node number of element nodal point 1
	46 - 50	NOD(2)	Global node number of element nodal point 2
	51 - 55	NOD(3)	Global node number of element nodal point 3
	56 - 60	NOD(4)	Global node number of element nodal point 4
	61 - 65	NOD(5)	Global node number of element nodal point 5
	66 - 70	NOD(6)	Global node number of element nodal point 6
	71 - 75	NOD(7)	Global node number of element nodal point 7

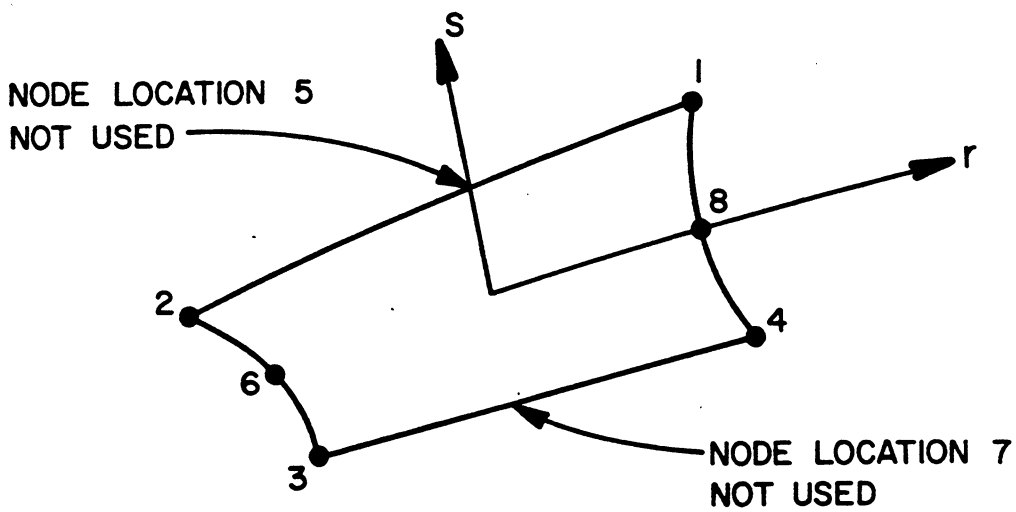
note	columns	variable	entry
	76 - 80	NOD(8)	Global node number of element nodal point 8

NOTES/

(1) Elements must be input in ascending element number order. If data cards for elements [M+1,M+2,...,M+J] are omitted, these "J" missing elements are generated using IEL,IPS,BET, THIC, and MTYP given on the card for element "M" and by incrementing node numbers of successive elements with the value "KG"; the value of KG used for incrementation is taken from the M-th element card, and only the non-zero nodes appearing on the M-th element card are incremented when generating missing element data.

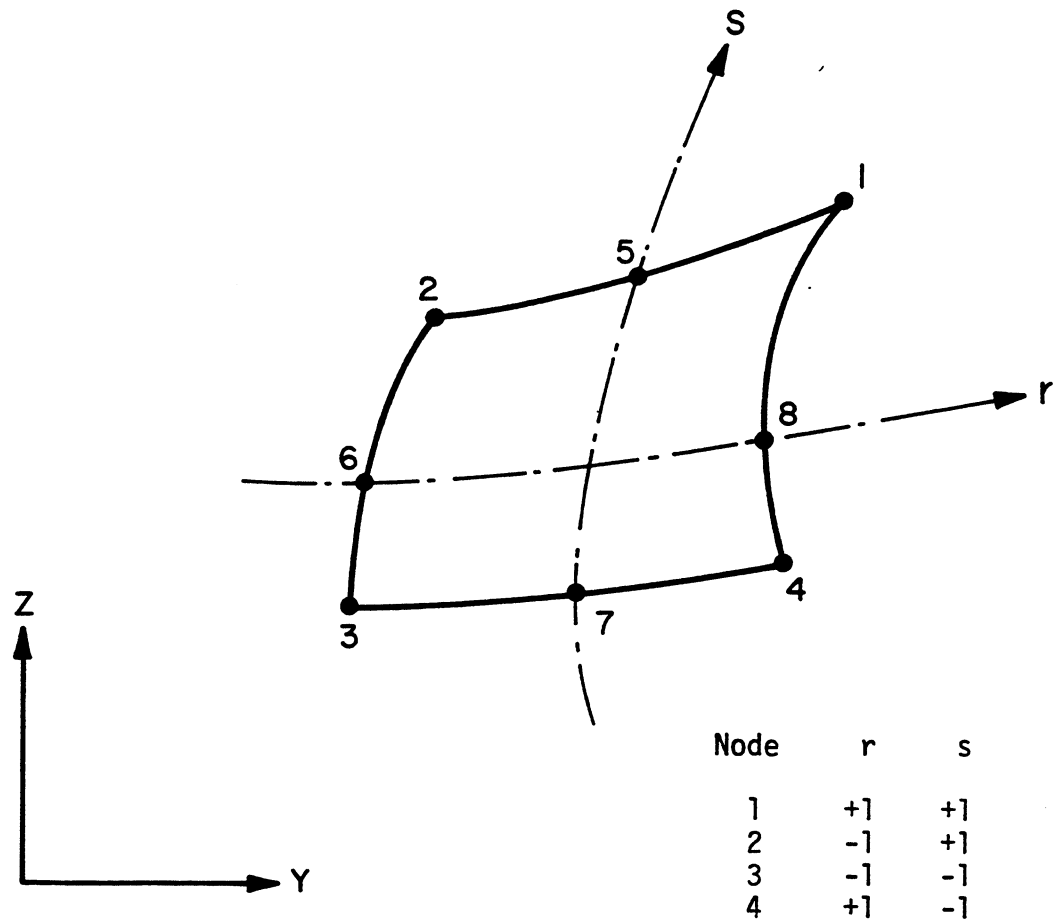
(2) The number of nodes in element "M" is defined by "IEL". However, all 8 entries for NOD(I) are read from the element data card; if IEL.LT.8 the particular node locations not used in this element need be input as "0" in NOD(I). Fig. X.6 defines the input sequence that must be observed for element node input.

For example, the 6-node element (IEL.EQ.6) shown below



is defined by

NOD(I) = [X X X X 0 X 0 X]



Node	r	s
1	+1	+1
2	-1	+1
3	-1	-1
4	+1	-1
5	0	+1
6	-1	0
7	0	-1
8	+1	0

FIGURE X.6 ELEMENT NODE NUMBER INPUT SEQUENCE FOR 2/D CONTINUUM ELEMENTS

NOTES/(2)

where the nonzero entries (X) are the global mesh node numbers (Section III.1) of the 6 nodes.

(3) If IPS.EQ.0 no stress output will be provided for this element. A nonzero entry for the element stress printout flag "IPS" has a different meaning when used with nonlinear material models than it does when used with linear material models:

a) For linear material models (MODELS "1" and "2") IPS specifies the stress output table (Section X.3) to be used for this element. If no stress output tables were input (i.e., if NPAR(13).EQ.0) then IPS.EQ.1 specifies that stresses are to be printed at all integration points for this element, as shown in Fig. X.5.

b) For nonlinear material models (MODELS.GT."2") IPS.EQ.1 specifies that stresses are to be printed at all integration points for this element, as shown in Fig. X.5.

(4) The thickness of the element need only be defined in plane stress analysis. In plane strain analysis thickness is assumed to be "1". In axisymmetric analysis thickness is assumed to be one (1) radian.

XI. 3/D CONTINUUM ELEMENTS

3/D CONTINUUM elements are 8- to 21-node isoparametric or subparametric curvilinear hexahedra. Typical elements are illustrated in Figure XI.1.

These elements may be used to model three-dimensional solids or thick shells, two examples of which are shown in Figure XI.2.

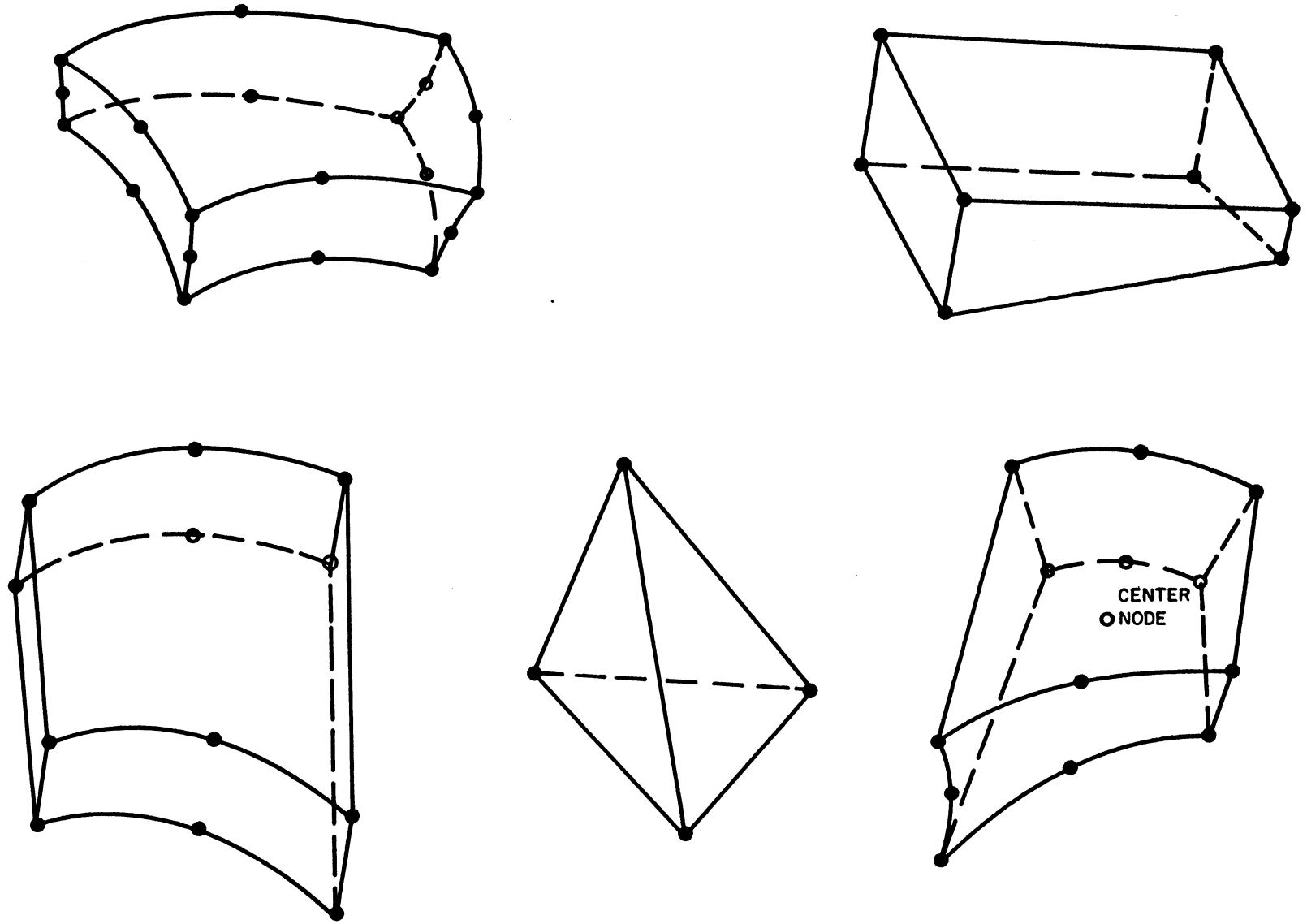
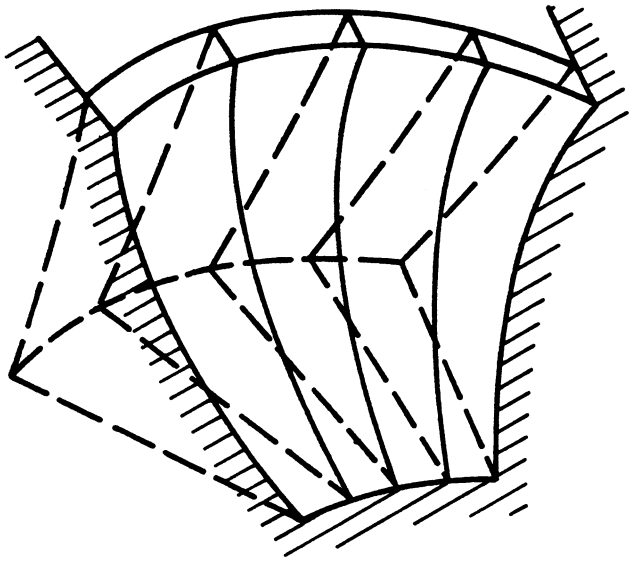
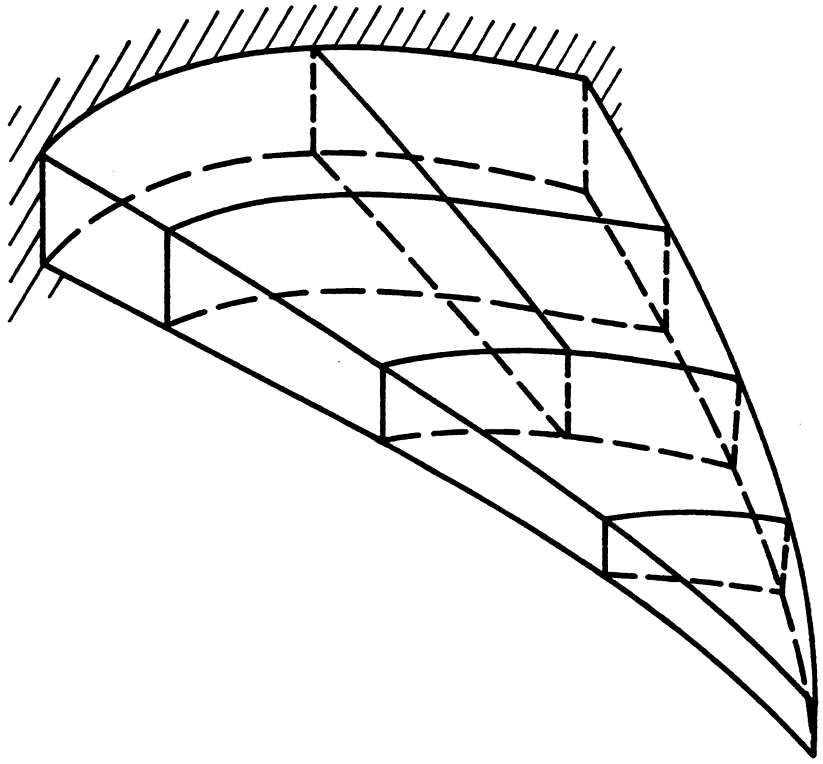


FIGURE XI.1 TYPICAL 3/D CONTINUUM ELEMENTS DERIVED FROM THE GENERAL ELEMENT IN FIGURE XI.3



3/D CONTINUUM ELEMENT MODEL OF ARCH DAM



3/D CONTINUUM ELEMENT MODEL OF THICK SHELL

FIGURE XI.2 POSSIBLE THREE-DIMENSIONAL FINITE ELEMENT ANALYSIS

XI. 3/D CONTINUUM ELEMENTS (continued)

1. Element Group Control Card (2014)

note	columns	variable	entry
	1 - 4	NPAR(1)	Enter the number "3"
(1)	5 - 8	NPAR(2)	Number of 3/D CONTINUUM elements in this group; GE.1
(2)	9 - 12	NPAR(3)	Flag indicating type of nonlinear analysis; EQ.0, default set to "1" EQ.1, material nonlinear analysis only EQ.2, total Lagrangian EQ.3, updated Lagrangian
(3)	25 - 28	NPAR(7)	Maximum number of nodes used to describe any one element; EQ.0, default set to "21"
(4)	37 - 40	NPAR(10)	Numerical integration order to be used in Gauss quadrature formula for r-s plane; EQ.0, default set to "2" GE.2 and LE.4
(4)	41 - 44	NPAR(11)	Numerical integration order to be used in Gauss quadrature formula for t-direction; EQ.0, default set to "2" GE.2 and LE.4
(5)	49 - 52	NPAR(13)	Number of stress output location tables; EQ.0, print stresses at integration points
(6)	57 - 60	NPAR(15)	Material model number; GE.1 and LE.8 EQ.1, linear elastic isotropic EQ.3, curve description model
(6)	61 - 64	NPAR(16)	Number of different sets of material properties; GE.1

note	columns	variable	entry
(6)	65 - 68	NPAR(17)	Number of constants per material property set; EQ.0, if NPAR(15).LE.3 GE.1, if NPAR(15).GT.3
(6)	69 - 72	NPAR(18)	Dimension of storage array required for element history; EQ.0, if NPAR(15).LE.3 GE.0, if NPAR(15).GT.3

NOTES/

(1) 3/D CONTINUUM element numbers begin with one (1) and end with the total number of elements in this group, NPAR(2). Data for the individual elements are input in Section XI.4 below.

(2) NPAR(3) is applicable for nonlinear element groups only and determines the type of geometrical nonlinearities to be considered in the analysis. In the present version of NONSAP only material nonlinearities are included in 3/D CONTINUUM elements; hence, NPAR(3) must be specified as one (1).

(3) NPAR(7) is the maximum number of nodes that can be used to describe any one of the elements in this group, i.e. elements in this group must have less than or equal to NPAR(7) nodes. A minimum of 8 and a maximum of 21 nodes are used to describe 3/D CONTINUUM elements as indicated in Fig. XI.3. Constant strain tetrahedra can be formed from 8-node elements by having nodes 1,2,3, and 4 coincide and nodes 7 and 8 coincide.

(4) The selection of appropriate integration orders depends on the element shape and strain state being considered. When the quantities being integrated vary irregularly a higher order is needed. An integration order of "2" is sufficient for most problems. Shell or plate problems often use thin elements in which strain varies more or less linearly through the thickness, but more irregularly in the plane of the surface of the shell. In these cases it is advantageous to specify a lower order for integration through the thickness, i.e., NPAR(11) < NPAR(10).

The expense of stiffness formation is dependent on the integration order, i.e.,

$$n = NPAR(11) * NPAR(10) * NPAR(10)$$

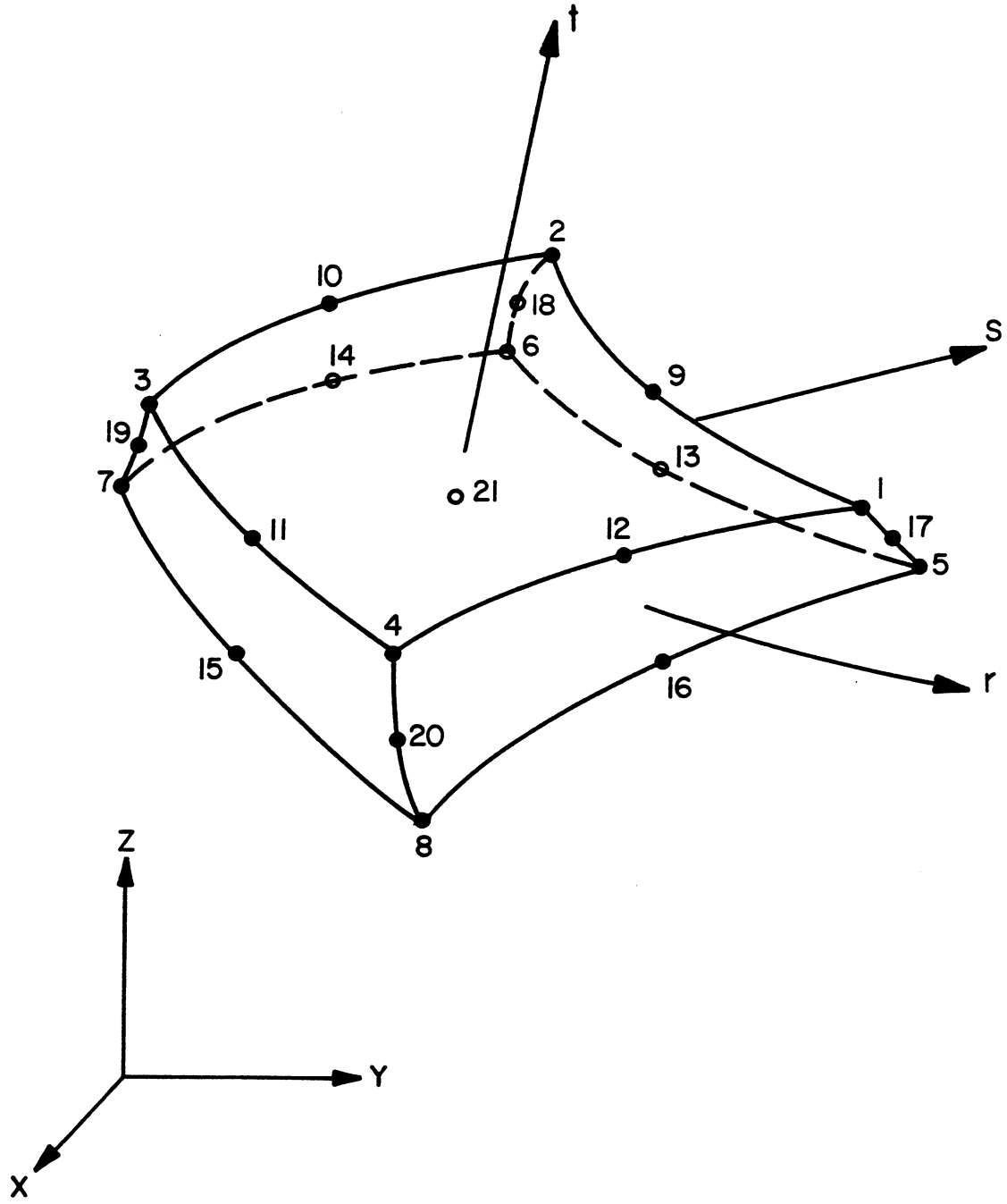


FIGURE XI.3 3/D CONTINUUM ELEMENT NODE NUMBERING CONVENTION

NOTES/(4)

where n is the number of points at which $B^T DB$ must be calculated in order to find the element stiffness by Gauss integration.

The consistent mass matrix is always calculated using an integration order of three in each direction.

(5) Element stresses are calculated at the points defined in the stress output location table (see Section XI.3) assigned to an element. NPAR(13) defines the total number of stress output location tables input in Section XI.3.

For nonlinear material models (NPAR(15).GE.3) stress output tables cannot be used, and stresses are printed at integration points; NPAR(13) is set to "0" in these cases.

(6) Only one material model (defined by the value of NPAR(15)) is allowed in an element group. If NPAR(15) is 1 or 3, the model exists in the current library for 3/D CONTINUUM elements, and the entries for NPAR(17) and NPAR(18) are ignored by the program. If, however, NPAR(15) is "4" or larger, material constants must be read into a storage array, "PROP", which has dimensions NPAR(17), NPAR(16); i.e., property constants are stored as,

(PROP(I,J),I=1,NPAR(17),J=1,NPAR(16)

Non-library models require that NPAR(17) be given (input) as a positive integer. In addition, NPAR(18) must be specified for non-standard nonlinear models for use in allocating storage in a working array, "WA", which contains element history. The NPAR(18) parameters which characterize element history must be retained for materially nonlinear elements so that properties can be chosen in the current solution step. The working storage array (WA) is dimensioned IDWA by NPAR(2), where

IDWA = NPAR(18) * (NPAR(10)**2) * NPAR(11)

NPAR(2) = number of elements

i.e., for each integration point of each element NPAR(18) storage locations are preserved during the solution. See Appendix B for the procedure to incorporate a new material model.

The model defined for the element group must be consistent with the nonlinear formulation used (NPAR(3)) and the requirement of equilibrium iteration as defined on Card 4 of the Master Control Cards (Section II). Equilibrium

NOTES/(6)

iteration can be performed only if the material model allows for it and if there is at least one nonlinear element group. Table XI.1 summarizes the possible analyses with the 3/D CONTINUUM material models implemented in the current version of NONSAP.

FORMULATION	MATERIAL MODEL	POSSIBILITY OF EQUILIBRIUM ITERATION
LINEAR ANALYSIS	Linear isotropic elastic	NO
MATERIALLY NONLINEAR ONLY ANALYSIS	Curve description model	NO

TABLE XI.1 POSSIBLE ANALYSIS TYPES WITH 3/D CONTINUUM ELEMENTS

XI. 3/D CONTINUUM ELEMENTS (continued)

2. Material Property Data

NPAR(16) sets of cards must be input in this section. Card "a" ("material number card") is the same for all material models, but card(s) "b" ("material property card(s)") depend on the material model number NPAR(15).

a. material number card (I5,F10.0)

note	columns	variable	entry
	1 - 5	N	Material property set number; GE.1 and LE. NPAR(16)
(1)	6 - 15	DEN(N)	Mass density of the material used in the calculations of the mass matrix; GE. 0.0

NOTES/

(1) The mass density defined is used directly in the calculation of the element mass matrix, i.e. no acceleration constants are applied to the variable DEN(N).

b. material property card(s) (8F10.0)

For MODEL "1" (NPAR(15).EQ.1) → [linear isotropic elastic]

note	columns	variable	entry
(1)	1 - 10	PROP(1,N)	Young's modulus, E
	11 - 20	PROP(2,N)	Poisson's ratio, ν

NOTES/

(1) MODEL 1 is a linear library model defined by two (2) positive constants (E,ν); i.e., if NPAR(15).EQ.1, NPAR(17) is set to "2" by default. MODEL 1 can be used with linear or nonlinear element groups. Since the material constants are independent of history NPAR(18) is "0" by default.

FOR MODEL "3" (NPAR(15).EQ.3) → [curve description model]

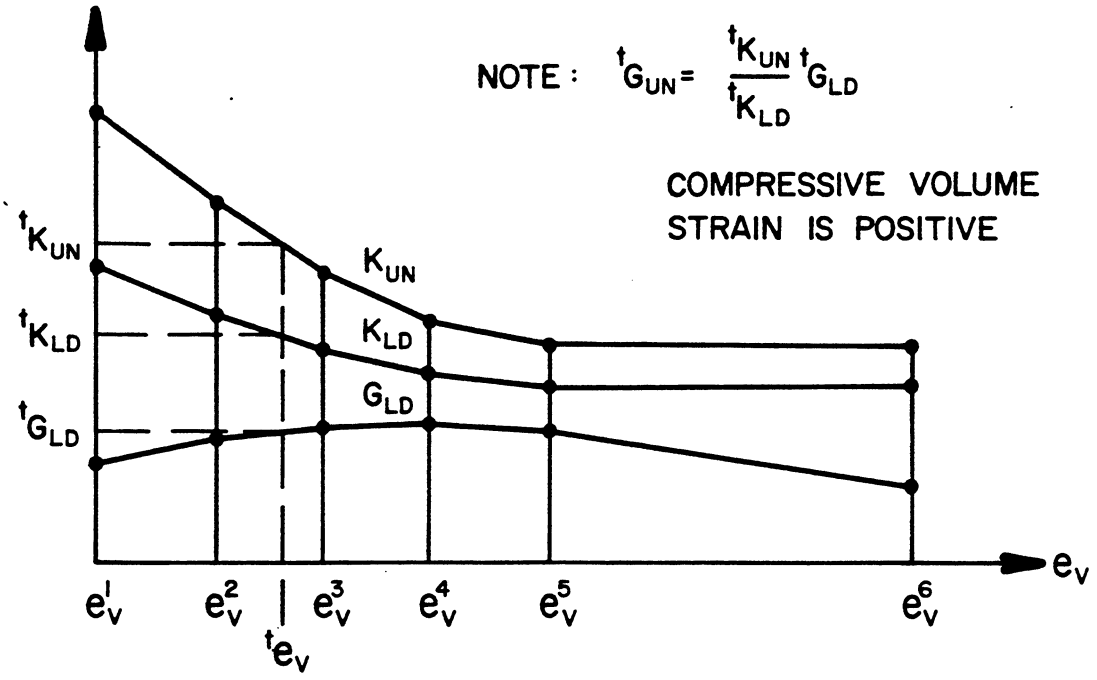
note	columns	variable	entry
first card			
(1)	1 - 10	PROP(1,N)	e_V^1 volume strain at point "1"
	11 - 20	PROP(2,N)	e_V^2....
	21 - 30	PROP(3,N)	e_V^3
	31 - 40	PROP(4,N)	e_V^4
	41 - 50	PROP(5,N)	e_V^5
	51 - 60	PROP(6,N)	e_V^6
	61 - 70	PROP(7,N)	K_{LD}^1 loading bulk modulus at point "1"
	71 - 80	PROP(8,N)	K_{LD}^2....
second card			
	1 - 10	PROP(9,N)	K_{LD}^3
	11 - 20	PROP(10,N)	K_{LD}^4
	21 - 30	PROP(11,N)	K_{LD}^5
	31 - 40	PROP(12,N)	K_{LD}^6
	41 - 50	PROP(13,N)	K_{UN}^1 unloading bulk modulus at point "1"
	51 - 60	PROP(14,N)	K_{UN}^2....
	61 - 70	PROP(15,N)	K_{UN}^3
	71 - 80	PROP(16,N)	K_{UN}^4
third card			
(2)	1 - 10	PROP(17,N)	K_{UN}^5
	11 - 20	PROP(18,N)	K_{UN}^6
	21 - 30	PROP(19,N)	G_{LD}^1 loading shear modulus at point "1"
	31 - 40	PROP(20,N)	G_{LD}^2....
	41 - 50	PROP(21,N)	G_{LD}^3
	51 - 60	PROP(22,N)	G_{LD}^4
	61 - 70	PROP(23,N)	G_{LD}^5
	71 - 80	PROP(24,N)	G_{LD}^6

NOTES/(1) MODEL 3 is a nonlinear library material model for which NPAR(17).EQ.24, by default. To characterize history, thirteen (13) variables--namely, the previously calculated

NOTES/(1) (continued)

stresses and strains, and the maximum volumetric strain ever reached in loading--need be saved for each integration point. Therefore, NPAR(18) is set to "13" by default.

All six input points must be defined, as indicated below:



Note that

$$e_V^J > e_V^{J-1}, \quad K_{UN}^J \geq K_{LD} \quad \text{and} \quad G_{LD}^J < 1.5\, K_{LD}^J$$

where "J" is the J-th table input point. Linear interpolation is used to define the loading and unloading bulk moduli, ${}^tK_{LD}$, ${}^tK_{UN}$, and the loading shear modulus, ${}^tG_{LD}$, at time t between input points. If the volumetric strain during any time or load step exceeds e_V^6, a diagnostic message results.

(2) The unloading shear modulus at time t is calculated as ${}^tG_{UN} = ({}^tK_{UN}/{}^tK_{LD})\, {}^tG_{LD}$.

NOTATION

For the notation used in the description of the nonlinear element models refer to the report, "Static and Dynamic Geometric and Material Nonlinear Analysis," SESM Report No. 74-4 , Feb. 1974.

For MODEL "≥4" (NPAR(15).GE.4) → (user-supplied material model)

note	columns	variable	entry
(1)	1 - 10	PROP(1,N)	
	11 - 20	PROP(2,N)	
	21 - 30	PROP(3,N)	
	⋮	etc.	

NOTES/

(1) MODEL "≥4" is a non-standard model (i.e., is not in the material model library) and requires that property data be read

(PROP(J,N),J=1,NPAR(17),N=1,NPAR(16))

where NPAR(17) is user-supplied (i.e., no default value exists). Subroutines which allocate storage locations (in "WA") for element history (if the model is nonlinear) and which calculate the incremental stress-strain law and current stresses must be supplied. For a discussion on how to incorporate a user-supplied material model, see Appendix B.

XI. 3/D CONTINUUM ELEMENTS (continued)

3. Stress Output Table Cards (16I5)

Skip this section if stresses are to be printed at integration points, i.e., if NPAR(13).EQ.0; otherwise supply NPAR(13) cards.

note	columns	variable	entry
(1)	1 - 5	ITABLE(N,1)	Stress output location 1
	6 - 10	ITABLE(N,2)	Stress output location 2
	11 - 15	ITABLE(N,3)	Stress output location 3
	16 - 20	ITABLE(N,4)	Stress output location 4
	21 - 25	ITABLE(N,5)	Stress output location 5
	26 - 30	ITABLE(N,6)	Stress output location 6
	31 - 35	ITABLE(N,7)	Stress output location 7
	36 - 40	ITABLE(N,8)	Stress output location 8
	41 - 45	ITABLE(N,9)	Stress output location 9
	46 - 50	ITABLE(N,10)	Stress output location 10
	51 - 55	ITABLE(N,11)	Stress output location 11
	56 - 60	ITABLE(N,12)	Stress output location 12
	61 - 65	ITABLE(N,13)	Stress output location 13
	66 - 70	ITABLE(N,14)	Stress output location 14
	71 - 75	ITABLE(N,15)	Stress output location 15
	76 - 80	ITABLE(N,16)	Stress output location 16

NOTES/(1) Stress tables are defined to provide flexibility in element stress output requests. Each element can refer to one of the tables defined, and the element stresses are then calculated at the points specified in that table. Figure XI.4 defines the locations of the 27 possible points within an element where stresses may be printed.

Any one table may contain a maximum of sixteen (16) stress output points. The first "0" entry in a table will terminate that table. For example, if ITABLE(N,1).EQ.21 and ITABLE(N,2) .EQ.0, then stresses will be printed only at point 21 (the centroid of the element) whenever this stress table is referred to.

Stress output tables are used only with linear material models (NPAR(15).EQ.1). For elements using nonlinear material models (NPAR(15).GE.3), stresses can be printed at integration points only.

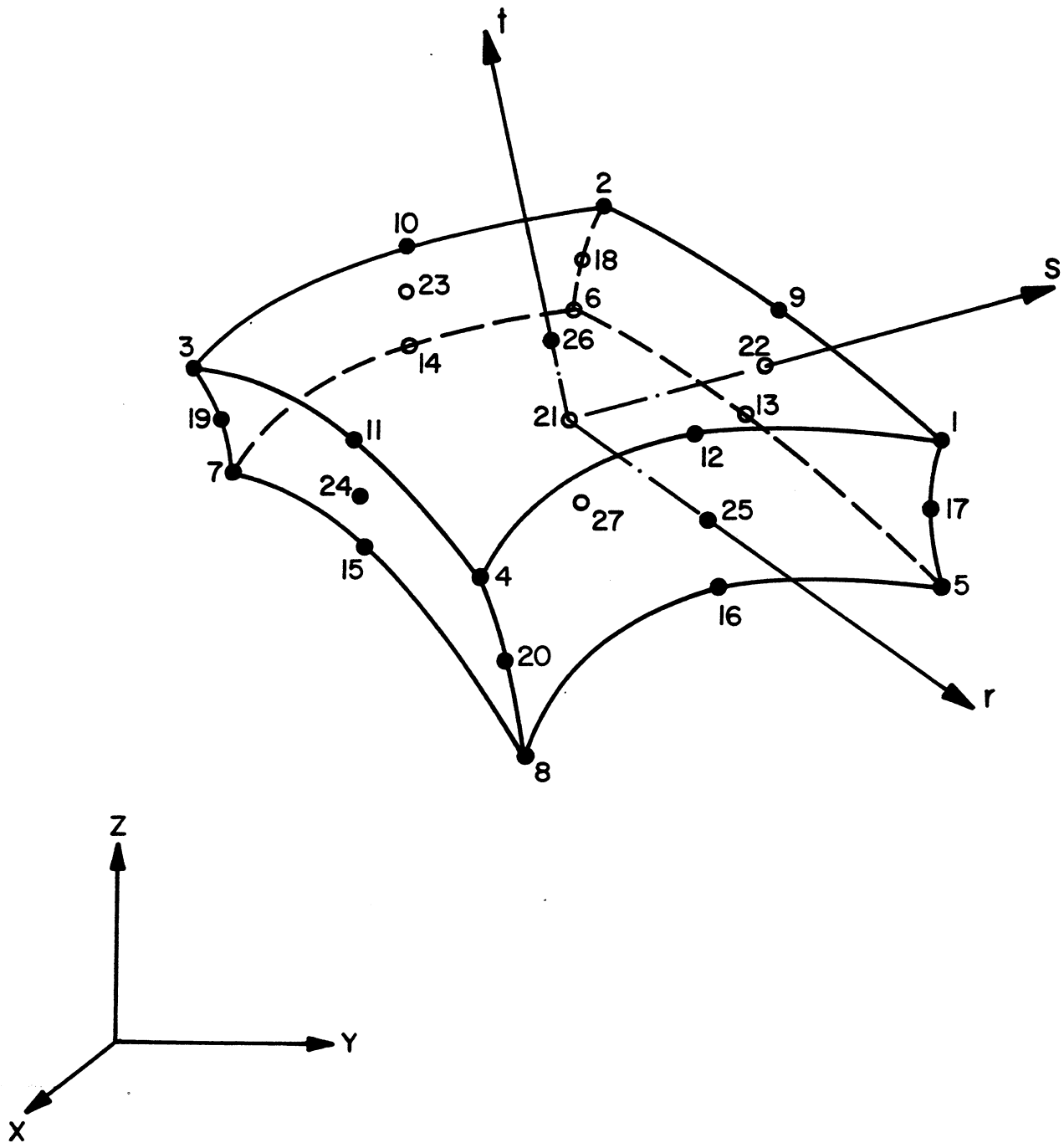


FIGURE XI.4 3/D CONTINUUM ELEMENT STRESS OUTPUT POINTS

XI. 3/D CONTINUUM ELEMENTS (continued)

4. Element Data Cards

Three cards must be prepared for each element that appears in the input:

CARD 1 (7I5)

note	columns	variable	entry
(1)	1 - 5	M	3/D CONTINUUM element number; GE.1 and LE. NPAR(2)
(2)	6 - 10	IELD	Number of nodes to be used in describing the element's displacement field; EQ.0, default set to NPAR(7) LE. NPAR(7)
(3)	11 - 15	IELX	Number of nodes to be used in the description of the element's geometry; EQ.0, default set to IELD GE.8 and LE.IELD EQ.IELD → isoparametric element LT.IELD → subparametric element
(4)	16 - 20	IPS	Number of the stress output location table to be used for this element (if a linear material model is used) or flag for printing stresses at element integration points; EQ.0, no stress output for this element
	21 - 25	MTYP	Material property set number assigned to this element; GE.1 and LE. NPAR(16)
(5)	26 - 30	IST	Flag indicating that the stiffness and mass matrices for this element are the same as those for the preceding element; EQ.0, no EQ.1, yes
(6)	31 - 35	KG	Node number increment for element data generation; EQ.0, default set to "1"

CARD 2 (8I5)

note	columns	variable	entry
(2)	1 - 5	NOD(1)	Global node number of element nodal point 1
	6 - 10	NOD(2)	Global node number of element nodal point 2
	11 - 15	NOD(3)	Global node number of element nodal point 3
	16 - 20	NOD(4)	Global node number of element nodal point 4
	21 - 25	NOD(5)	Global node number of element nodal point 5
	26 - 30	NOD(6)	Global node number of element nodal point 6
	31 - 35	NOD(7)	Global node number of element nodal point 7
	36 - 40	NOD(8)	Global node number of element nodal point 8

CARD 3 (13I5)

note	columns	variable	entry
(2)	1 - 5	NOD(9)	Global node number of element nodal point 9
	6 - 10	NOD(10)	Global node number of element nodal point 10
	11 - 15	NOD(11)	Global node number of element nodal point 11
	16 - 20	NOD(12)	Global node number of element nodal point 12
	21 - 25	NOD(13)	Global node number of element nodal point 13
	26 - 30	NOD(14)	Global node number of element nodal point 14
	31 - 35	NOD(15)	Global node number of element nodal point 15
	36 - 40	NOD(16)	Global node number of element nodal point 16
	41 - 45	NOD(17)	Global node number of element nodal point 17

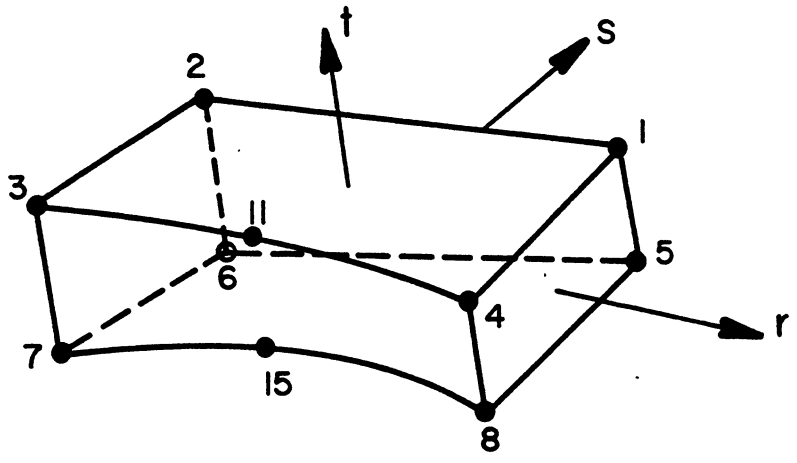
note	columns	variable	entry
	46 - 50	NOD(18)	Global node number of element nodal point 18
	51 - 55	NOD(19)	Global node number of element nodal point 19
	56 - 60	NOD(20)	Global node number of element nodal point 20
	61 - 65	NOD(21)	Global node number of element nodal point 21

NOTES/

(1) Element cards must be input in ascending element number order beginning with one (1) and ending with NPAR(2). Repetition of element numbers is illegal, but element cards may be omitted, and missing element data are generated according to the procedure described in note (6).

(2) IELD is a count of the node numbers actually posted on Cards 2 and 3 which must immediately follow Card 1. IELD must be at least eight (8), but must be less than or equal to the limit NPAR(7) which was given on the element group control card, Section XI.1. Element displacements are assigned at the IELD non-zero nodes, and thus the order of the element matrices is three (i.e., translations X,Y,Z) times IELD. The eight corner nodes of the hexahedron must be input, but nodes 9 to 21 are optional, and any or all of these optional nodes may be used to describe the element's displacement field. However, all 21 entries for NOD(I) are read from element data cards 2 and 3; if IELD .LT.21 the particular node locations not used in this element must be input as zero (0) in NOD(I). Figure XI.3 defines the input sequence that must be observed for element input.

For example, the 10-node element (IELD.EQ.10) shown below



NOTES/(2)

is defined by

NOD(I) = [X X X X X X X X 0 0 X 0 0 0 X 0 0 0 0 0 0],

where the nonzero entries (X) are the global mesh node numbers (Section III) of the 10 nodes.

(3) When element edges are straight it is unnecessary computationally to include side nodes in the numerical evaluation of coordinate derivatives, the Jacobian matrix, etc., and since regular element shapes are common, an option has been included to use fewer nodes in these geometric calculations than are used to describe element displacements. The first IELX nonzero nodes posted on Cards 2 and 3 are used to evaluate those parameters which pertain to element geometry only. IELX must be at least eight (8), and if omitted is re-set to IELD. A common application might be a 20 node element (i.e., IELD.EQ.20) with straight edges in which case IELX would be entered as "8".

(4) If IPS.EQ.0 no stress output will be provided for this element. A nonzero entry for the element stress printout flag "IPS" has a different meaning when used with nonlinear material models than it does when used with linear material models:

a) For a linear material model (MODEL.EQ.1) "IPS" specifies the stress output table (Section XI.3) to be used for this element. If no stress output tables were input (i.e., if NPAR(13).EQ.0) then IPS.EQ.1 specifies that stresses are to be printed at all integration points for this element.

b) For a nonlinear material model (MODEL.GE.3) IPS.EQ.1 specifies that stresses are to be printed at all integration points for this element.

(5) The flag IST allows the program to bypass stiffness and mass matrix calculations providing the current element is identical to the preceding element; i.e., the preceding and current elements are identical except for a rigid body translation. If IST.EQ.0, new matrices are computed for the current element.

(6) When element cards are omitted, element data are generated automatically as follows,

a) All data on Card 1 for generated elements is taken to be the same as that given on the first element card in the sequence;

NOTES/(6)

b) Nonzero node numbers (given on Cards 2 and 3 for the first element) are incremented by the value "KG" (on the first element's Card 1) as element generation progresses; zero (or blank) node number entries are generated as zeroes.

The last element cannot be generated.

XII. FREQUENCIES AND MODE SHAPES CALCULATION DATA

If the flag defining the eigensolution mode ("IEIG") given in Section II (Master Control Parameters, Card 3) is omitted, skip this block of data; i.e., if IEIG.EQ.0, no data is read in this section.

Frequency Solution Data Card (I5,F10.0,I5)

note	columns	variable	entry
(1)	1 - 5	NFREQ	Number of frequencies to be calculated; EQ.0, default set to "1"
(1)	6 - 15	COFQ	Cut-off circular frequency (units of radians per unit time); EQ.0, default set to "1.0E+08"
(2)	16 - 20	IFPR	Flag to suppress intermediate print-out of solution variables, interim informative messages, etc.; EQ.0, print-out suppressed EQ.1, print-out produced

NOTES/

(1) The lowest "NFREQ" vibration mode shapes and frequencies are evaluated for the structure linearized at time "TSTART". Frequencies are extracted in numerically ascending sequence, and unless the "N-th" frequency is > COFQ, the program will continue the eigensolution until all "NFREQ" modes have been determined. If the N-th frequency is > COFQ, where $1 \leq N \leq NFREQ$, the eigensolution is discontinued, and only the lowest "N" frequencies and mode shapes are printed.

At completion of solution, "physical" error bounds for each eigenvalue and eigenvector are calculated using $||(K-\omega^2 M)\phi||/||K\phi||$, where (ω,ϕ) is a calculated eigenpair.

(2) During the course of eigenvalue extraction various solution variables, informative diagnostics, etc. are printed if IFPR.EQ.1. This print-out is useful in tracing solution behavior.

APPENDIX A - CONTROL CARDS AND DECK SET-UP FOR ANALYSIS RESTART

The purpose of this appendix is to describe the procedure (including control cards and deck set-up) required for program restart following a successful step-by-step solution. The restart option has been included in the program in order to make it possible to continue a step-by-step solution at the point it was terminated in an earlier analysis. It may be noted that the restart option can conveniently be used to continue a nonlinear analysis of a structure for different loading conditions, each to be specified individually in a run of job type (2) below.

An analysis utilizing the restart feature requires that the job be run in two (2) steps:

JOB(1): Step-by-step solution for NSTE time or load steps, after which program files TAPE4, TAPE7, TAPE8, and TAPE9 are saved on the restart tape.

JOB(2): Reinstatement of program files TAPE4, TAPE7, TAPE8, and TAPE9 from the restart tape followed by a continuation of the step-by-step solution using the MODEX.EQ.2 option (see Section II, card 1).

For a given model, in the first job an analysis for "NSTE" time or load steps was carried out. The final material properties of the nonlinear elements, displacements, velocities, etc., and all linear structure matrices are saved on the restart tape. The restart tape together with the input defined in JOB(2) then contains all the information that is needed for the continuation of the analysis. More than one second job [JOB(2)] may be run using the restart tape as initial input, i.e., the restart tape is not destroyed.

Control cards and deck set-up for execution on the CDC 6400 computer at the University of California, Berkeley are given below:

JOB(1) - SUCCESSFUL SOLUTION OF STRUCTURE FOR "NSTE" STEPS

Notes	Card Deck
(1)	Job number, 1, 200, 120000, 300. User Name
(2)	REQUEST,TP1,HI,I. Reel No., Tape User Name
(3)	CØPYBF,TP1,NSAP UNLØAD,TP1
(4)	FLGØ, NSAP REWIND,TAPE4,TAPE7,TAPE8,TAPE9
(5)	REQUEST,RESTART,HI,I. Reel No., Tape User Name, ØUTPUT
(6)	CØPYBF,TAPE4,RESTART CØPYBF,TAPE7,RESTART CØPYBF,TAPE8,RESTART CØPYBF,TAPE9,RESTART
(7)	7-8-9

JOB(1) (continued)

Notes	Card Deck
(8)	Problem data deck blank card blank card
(9)	6-7-8-9

NOTES/

(1) The job control card parameters are defined as follows:
"1" = Number of tape drives required for the job.
"200" = CPU time limit (in octal seconds).
"120000" = Central memory field length (in octal).
"300" = Page limit for printing.

(2) Tape containing absolute version of program (TP1) is requested.

(3) Absolute version of the program is copied onto a disk file (NSAP).

(4) Program is loaded and execution is initiated.

(5) A blank tape (RESTART) is requested.

(6) The contents of disk files TAPE4, TAPE7, etc. are copied onto tape RESTART.

(7) End-of-record card: 7,8,9 punched in column 1.

(8) The data of the problem for "NSTE" solution steps is provided.

(9) End-of-file card: 6,7,8,9 punched in column 1.

JOB(2) - RESTART CONTINUATION OF ANALYSIS FOR "NSTE" MORE STEPS

Notes	Card Deck
	Job number, 1,200,120000,300. User Name
(1)	REQUEST,RESTART,HI,I. Reel No., User Name
	CØPYBF,RESTART,TAPE4
	CØPYBF,RESTART,TAPE7
	CØPYBF,RESTART,TAPE8
	CØPYBF,RESTART,TAPE9
	REWIND,TAPE4,TAPE7,TAPE8,TAPE9
	UNLØAD,RESTART
(2)	REQUEST,TP1,HI,I. Reel No., User Name
	CØPYBF,TP1,NSAP
	FLGØ,NSAP
	7-8-9
(3)	Problem data deck
	blank card
	blank card
	6-7-8-9

NOTES/

(1) The disk files TAPE4,TAPE7, etc. are re-created using the information saved on tape RESTART.

(2) The absolute version of the program is again obtained form tape TP1.

(3) The complete problem data deck is provided, i.e. all data cards provided in JOB(1) are also here. However, the input variable "TSTART" must be reset to the time at which the solution of JOB(1) was terminated. Also, the variable "NSTE" (number of additional time/load steps to be considered) and the loading data (Section IV of the Appendix) may be reset for JOB(2). Note that the loading data is interpolated starting at time TSTART. It is not possible to change the mesh, boundary conditions, element nonlinearity codes, material models, etc., i.e. the restart job [JOB(2)] is a mere continuation of the original job [JOB(1)]. The program reads all data, but bypasses the calculation of expensive information, which, instead, has been read from the tape RESTART. Note that in this data deck MODEX is set to "2".

It is not possible to calculate frequencies in a restart job [JOB(2)]. The flag IEIG.EQ.1 and the frequencies solution control card are read by the program but are not used.

APPENDIX B - IMPLEMENTATION OF USER-SUPPLIED NONLINEAR MATERIAL MODELS

NONSAP has been written to accept nonlinear material models not currently included in the material model library. The user must supply the overlay subroutines for the new material model. No changes to program NONSAP are required, except the insertion of the material model subroutines. The appropriate calls to the subroutines which need be supplied are already made from the program.

The purpose of this appendix is to discuss briefly how a new material model would be incorporated.

Additional material models may be specified for the two- and three-dimensional elements. The capability has not been programmed for the truss elements. User-supplied material models must have any number between 9 and 12 for the two-dimensional elements, and between 4 and 8 for the three-dimensional elements. The main subroutines called by NONSAP corresponding to these material model numbers are shown in Fig. B.1. Currently these subroutines do not perform any calculations, and to insert a new material model the user has to write the corresponding subroutine together with any others that are called by it.

For purpose of illustration we assume in the following that material model 9 shall be supplied for the two-dimensional elements. Figure B.1 shows that the main subroutine corresponding to this material model is ELT2D9.

It should be recalled that in materially nonlinear analysis stresses are only calculated at the Gauss integration points. Figure B.2 shows that during execution subroutine ELT2D9 must perform the following functions:

Call from INITWA

The working storage array WA is initialized during the element information input phase.

On the control card of an element group, NPAR(18) was specified as the number of working storage locations required per integration point. This working storage is available for any listing of stress, strain, loading conditions and other quantities. It should be noted that the user has to allocate the working storage, which will be the same throughout the solution.

Call from STSTN

The subroutine STSTN is called whenever it is necessary to calculate element stresses and/or the element tangent material law. This condition arises during three different solution phases:

(1) Formation of a new tangent stiffness matrix (element stresses and the tangent material matrix are calculated)

Two-dimensional element user-supplied material models

MODEL 9	MODEL 10	MODEL 11	MODEL 12
call to	call to	call to	call to
ELT2D9	EL2D10	EL2D11	EL2D12

Three-dimensional element user-supplied material models

MODEL 4	MODEL 5	MODEL 6	MODEL 7	MODEL 8
call to	call to	call to	call to	call to
ELT3D4	ELT3D5	ELT3D6	ELT3D7	ELT3D8

FIGURE B.1 SUBROUTINES USED IN THE DEFINITION OF NONLINEAR MATERIAL MODELS

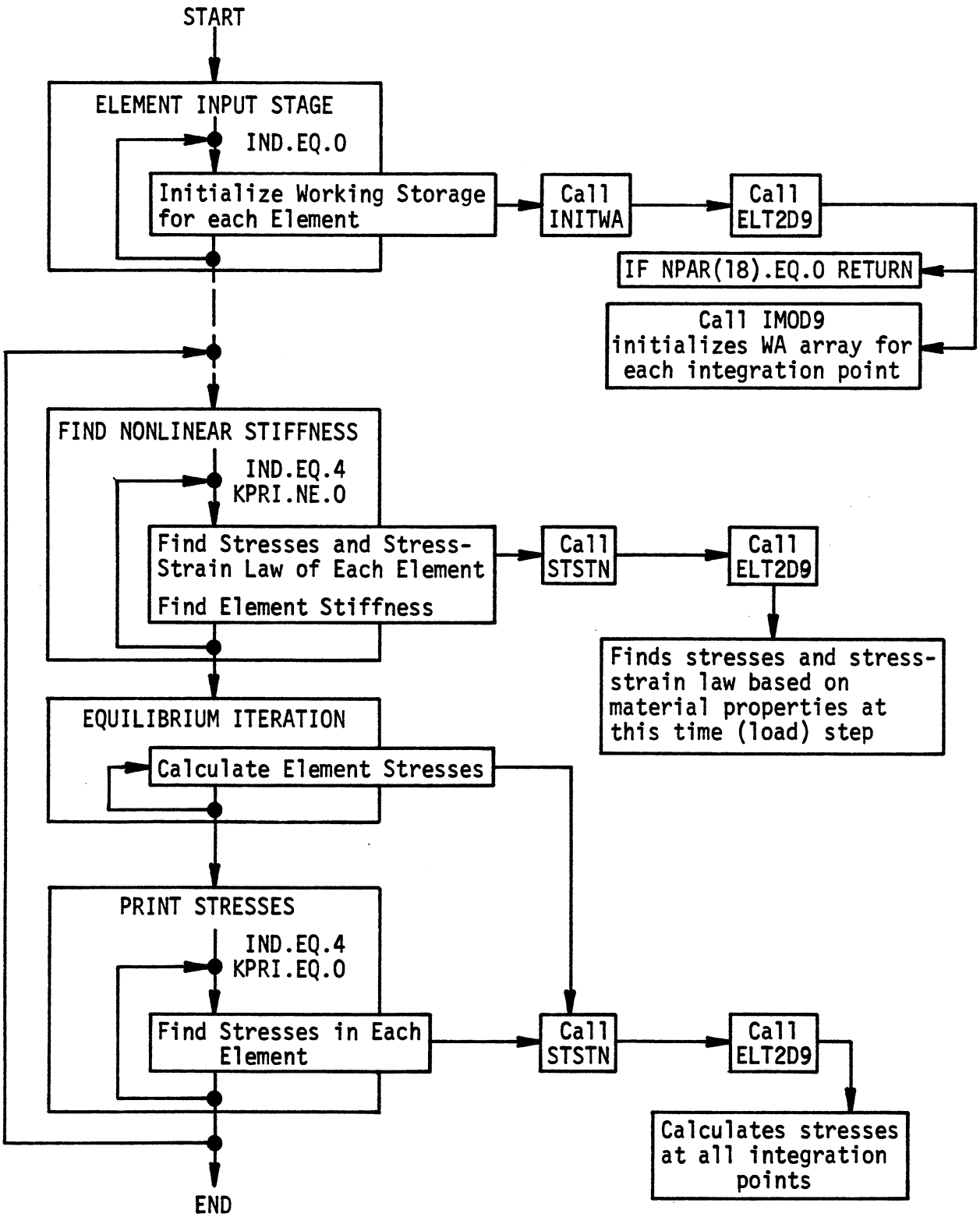

START
ELEMENT INPUT STAGE
IND.EQ.0
Initialize Working Storage for each Element
Call INITWA
Call ELT2D9
IF NPAR(18).EQ.0 RETURN
Call IMOD9 initializes WA array for each integration point
FIND NONLINEAR STIFFNESS
IND.EQ.4
KPRI.NE.0
Find Stresses and Stress-Strain Law of Each Element
Find Element Stiffness
Call STSTN
Call ELT2D9
Finds stresses and stress-strain law based on material properties at this time (load) step
EQUILIBRIUM ITERATION
Calculate Element Stresses
PRINT STRESSES
IND.EQ.4
KPRI.EQ.0
Find Stresses in Each Element
Call STSTN
Call ELT2D9
Calculates stresses at all integration points
END

FIGURE B.2

(2) Equilibrium iteration or incremental analysis without calculation of a new stiffness matrix (stresses only are calculated)

(3) Printing of stresses (stresses only are calculated)

In all cases the current strains are found in STSTN, which then calls ELT2D9 (through the overlay structure) for calculation of the corresponding stresses and material law. Note that for the calculation of the current stresses only, it may be necessary to evaluate the material law.

The main point in the programming of a new material law is that the working storage WA can be used to store any required history at each element integration point, such as previously calculated stresses, strains, loading conditions, etc. The array WA is dimensioned as

```
WA(IDWA,NE)
```

where

```
IDWA = NPAR(18) * (NPAR(10)**2)
  NE = NPAR(2)
```

Hence, the J'th column in WA stores the history of the J'th element, with the first NPAR(18) locations referring to the first integration point, etc.

In addition to the contents of WA, the array PROP is used, which stores the constant material data. This array was initialized during the read-in of the element information.