NUCLEAR ENGINEERING AND DESIGN 29 (1974) 266 293 © NORTH-HOLLAND PUBLISHING COMPANY

NONSAP – A NONLINEAR STRUCTURAL ANALYSIS PROGRAM*

Klaus Jürgen BATHE and Edward L. WILSON

Division of Structural Engineering and Structural Mechanisc, Department of Civil Engineering, University of California, Berkeley, California 94720, USA

Received 24 June 1974

The current version of the computer program NONSAP for linear and nonlinear, static and dynamic finite element analysis is presented. The solution capabilities, the numerical techniques used, the finite element library, the logical construction of the program and storage allocations are discussed. The solutions of some sample problems considered during the development of the program are presented.

1. Introduction

The endeavor to perform nonlinear analyses has steadily increased in recent years [1–4]. 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 [5]. Based on this method, various large-scale general purpose computer programs with nonlinear capabilities are now in use [6].

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 established 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 of completion.

The nonlinear analysis program NONSAP presented in this paper is not an extension of the linear analysis program SAP [7], but rather a completely new development [8]. 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.

---

* Invited paper M3/1* presented at the Second International Conference on Structural Mechanics in Reactor Technology, Berlin, Germany, 10–14 September, 1973.

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, and

(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:

(1) For the truss elements: linear elastic and nonlinear elastic.

(2) For the two-dimensional elements: isotropic linear elastic; orthotropic linear elastic; Mooney–Rivlin material; elastic–plastic materials, von Mises or Drucker–Prager yield conditions; variable tangent moduli model; and curve description model (with tension cut-off).

(3) For the three-dimensional elements: isotropic linear elastic and 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 is 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 paper is to present the general program organization, the current element library, the numerical techniques used and some sample solutions. 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, see refs [9] and [10].

2. Incremental equilibrium equations of structural systems

The incremental nodal point equilibrium equations for an assemblage of nonlinear finite elements have been derived in refs [9] and [10]. At time t we have

$$\boldsymbol{M}\,{}^{t+\Delta t}\ddot{\boldsymbol{u}} + \boldsymbol{C}\,{}^{t+\Delta t}\dot{\boldsymbol{u}} + {}^{t}\boldsymbol{K}\,\boldsymbol{u} = {}^{t+\Delta t}\boldsymbol{R} - {}^{t}\boldsymbol{F}, \tag{1}$$

where $\boldsymbol{M}$ is the constant mass matrix; $\boldsymbol{C}$ is the constant damping matrix; ${}^{t}\boldsymbol{K}$ is the tangent stiffness matrix at time t; ${}^{t+\Delta t}\boldsymbol{R}$ is the external load vector applied at time $t + \Delta t$; ${}^{t}\boldsymbol{F}$ is the nodal point force vector equivalent to the element stresses at time t; ${}^{t+\Delta t}\dot{\boldsymbol{u}}$, ${}^{t+\Delta t}\ddot{\boldsymbol{u}}$ are vectors of nodal point velocities and accelerations at time $t + \Delta t$; and $\boldsymbol{u}$ is the vector of nodal point displacement increments from time t to $t + \Delta t$, i.e. $u = {}^{t+\Delta t}u - {}^{t}u$.

As discussed in refs [9] and [10], 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

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

where M, C, ${}^{t}K$, and ${}^{t+\Delta t}R$ are as defined above, and

$${}^{t+\Delta t}\ddot{u}^{(i)}, \qquad {}^{t+\Delta t}\dot{u}^{(i)}, \qquad {}^{t+\Delta t}u^{(i)} = {}^{t+\Delta t}u^{(i-1)} + \Delta u^{(i)}$$

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

$$\Delta u^{(1)} = u, \qquad {}^{t+\Delta t}u^{(0)} = {}^{t}u, \qquad {}^{t+\Delta t}\ddot{u}^{(1)} = {}^{t+\Delta t}\ddot{u},$$
$${}^{t+\Delta t}\dot{u}^{(1)} = {}^{t+\Delta t}\dot{u}, \qquad {}^{t+\Delta t}F^{(0)} = {}^{t}F.$$

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 [11]. It should be noted that the solution scheme used in eq. (2) corresponds to a modified Newton iteration [3, 5].

In program NONSAP, the Wilson θ-method or the Newmark method is used for the step-by-step solution [11, 12]. Table 1 summarizes the algorithm in linear or nonlinear, static or dynamic analysis [10]. The specific operations performed during the step-by-step solution are discussed in section 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 [5, 13]; for example

$$K = \Sigma K_m, \qquad (3)$$

where K_m is the stiffness matrix of the mth element. Although K_m is formally of the same order as K, only those terms in K_m 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 on various occasions [5, 14, 15].

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 nonzero displacement is to be specified at a degree of freedom i, say $u_i = x$, the equation

$$ku_i = kx \qquad (4)$$

need be added into eq. (2), where $k \gg k_{ii}$. 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 [7]. However, in the current version of NONSAP only translational displacements are considered (since only isoparametric elements are available, see section 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. This phase consists of three steps:

Table 1.
Summary of step-by-step integration.

Initial calculations

---

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

(2) Calculate the following constants:

tol $\leqslant 0.01$; *nitem* $\geqslant 3$; in static analysis $\theta = 1$ and go to (3).

Wilson θ-method: $\theta \geqslant 1.37$, 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: $\theta = 1.0$, $\delta \geqslant 0.50$, $\alpha \geqslant 0.25(0.5 + \gamma)^2$, $\tau = \Delta t$

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

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

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

(3) Form effective linear stiffness matrix: $\hat{K} = K + a_0M + a_1C$.

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

For each time step

(A) *In linear analysis*

(i) Form effective load vector:

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

(ii) Solve for displacement increments:

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

(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 ${}^t\hat{K}$:

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

(ii) Form effective load vector:

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

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

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

(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:

$${}^{t+\tau}\ddot{u}^{(i-1)} = a_0u^{(i-1)} - a_2{}^t\dot{u} - a_3{}^t\ddot{u}; \qquad {}^{t+\tau}\dot{u}^{(i-1)} = a_1u^{(i-1)} - a_4{}^t\dot{u} - a_5{}^t\ddot{u};$$

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

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

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

Table 1 (continued).

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

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

(e) Calculate new displacement increments:

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

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

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.

(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:

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

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

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

(a) The control information and 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 is applicable), the element stresses are calculated and printed. Before the time integration is performed, the lowest frequencies and corresponding mode shapes may be calculated. Details of the step-by-step solution are presented in section 7.

It should 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. Figs 1–3 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 1–3 show that the lowest high-speed storage locations are reserved throughout the solution for element group information. For the analysis, the finite

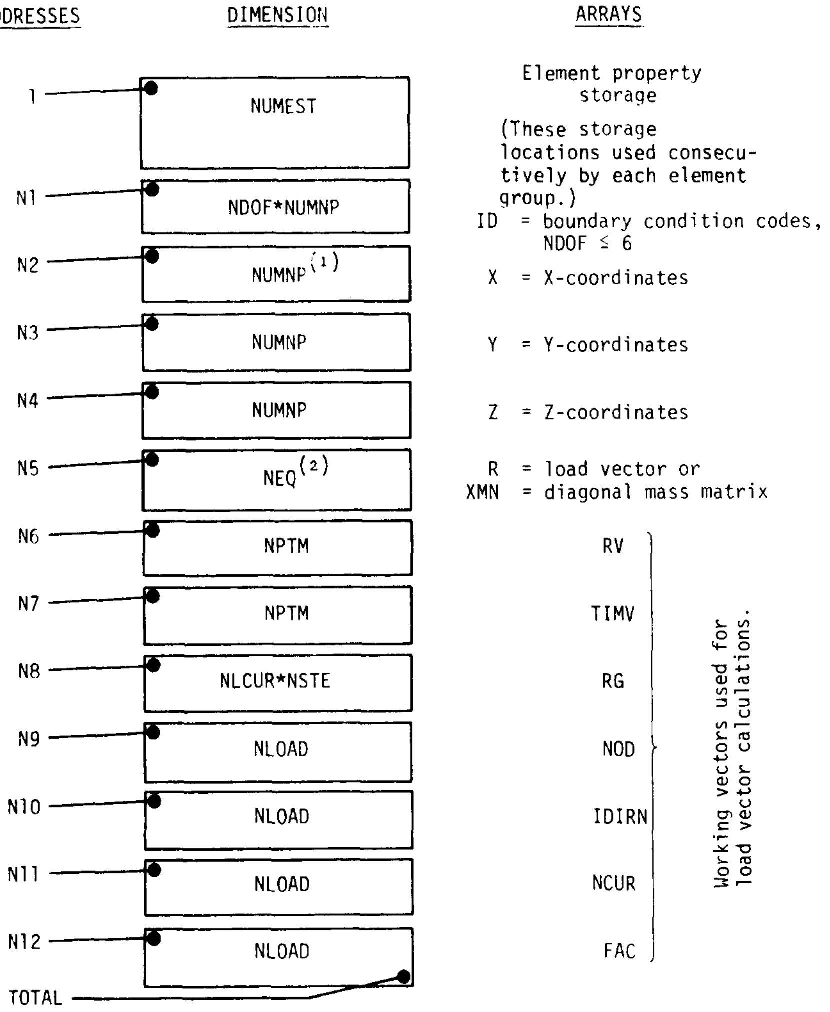


Fig. 1. Storage allocation during input phase. (1)NUMNP = number of nodal points, (2)NEQ = number of equations.

elements of the complete assemblage need be divided into element groups according to their type, the nonlinear formulation (see section 4), and the material models used (see section 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 to fit into the NUMEST storage locations, fig. 1. Therefore, the minimum that 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 section 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 number of locations needed, see figs 2 and 3. Therefore, an optimum of high-speed storage allocation is obtained during the step-by-step solution. Fig. 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

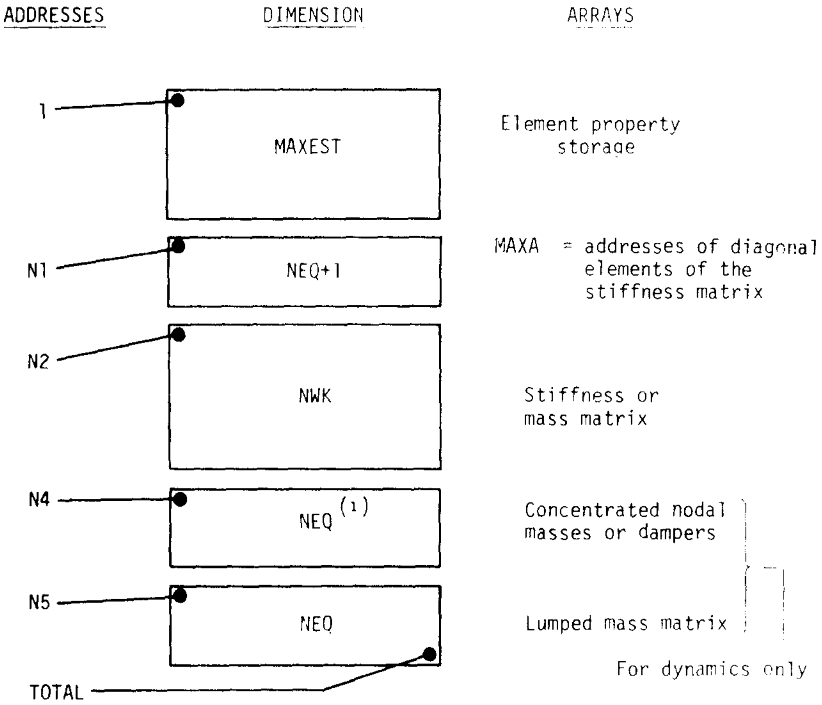


Fig. 2. Storage allocation during matrix assemblage phase. (1)NEQ = number of equations.

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. Fig. 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 [7]. A maximum of three boundary condition codes need currently be defined, since a finite element node can have at most three (translational) degrees of freedom (see section 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 (three translations and three 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 [7]. Fig. 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 zeros 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 with

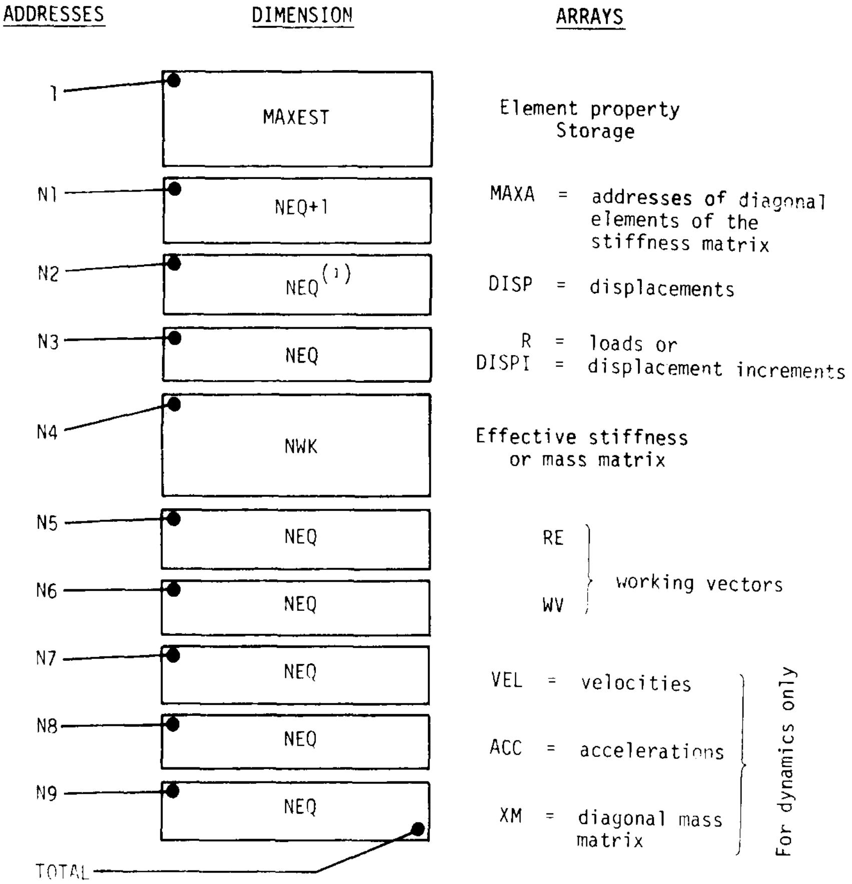


Fig. 3. Storage allocation during time integration. (1)NEQ = number of equations.

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 fig. 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.

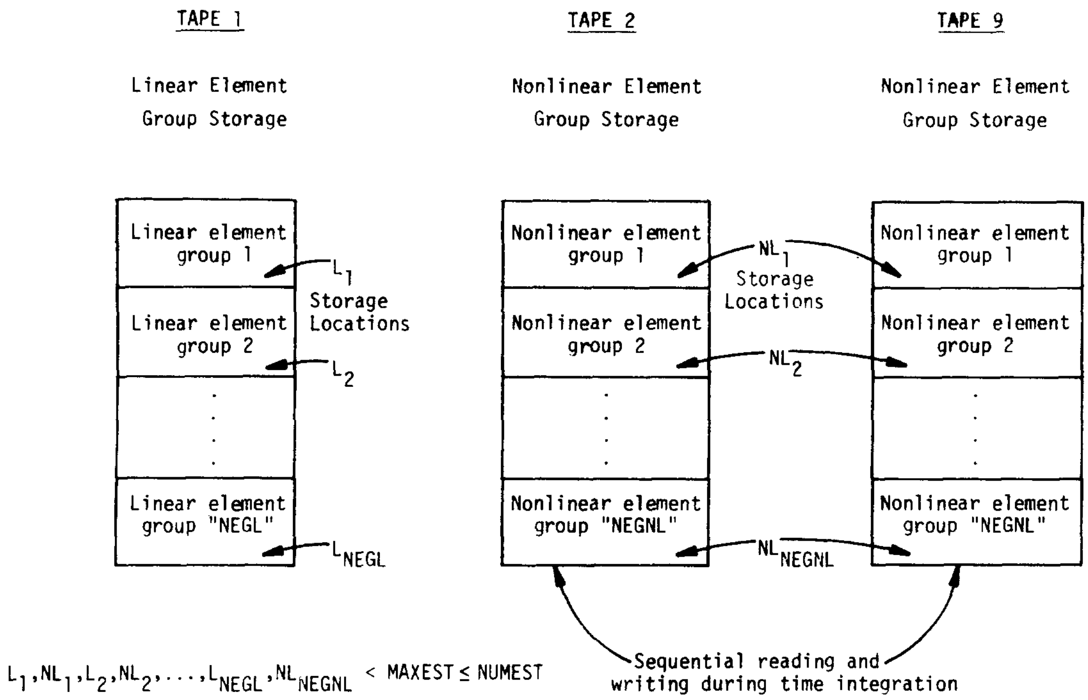


Fig. 4. Auxiliary storage organization for element group information.

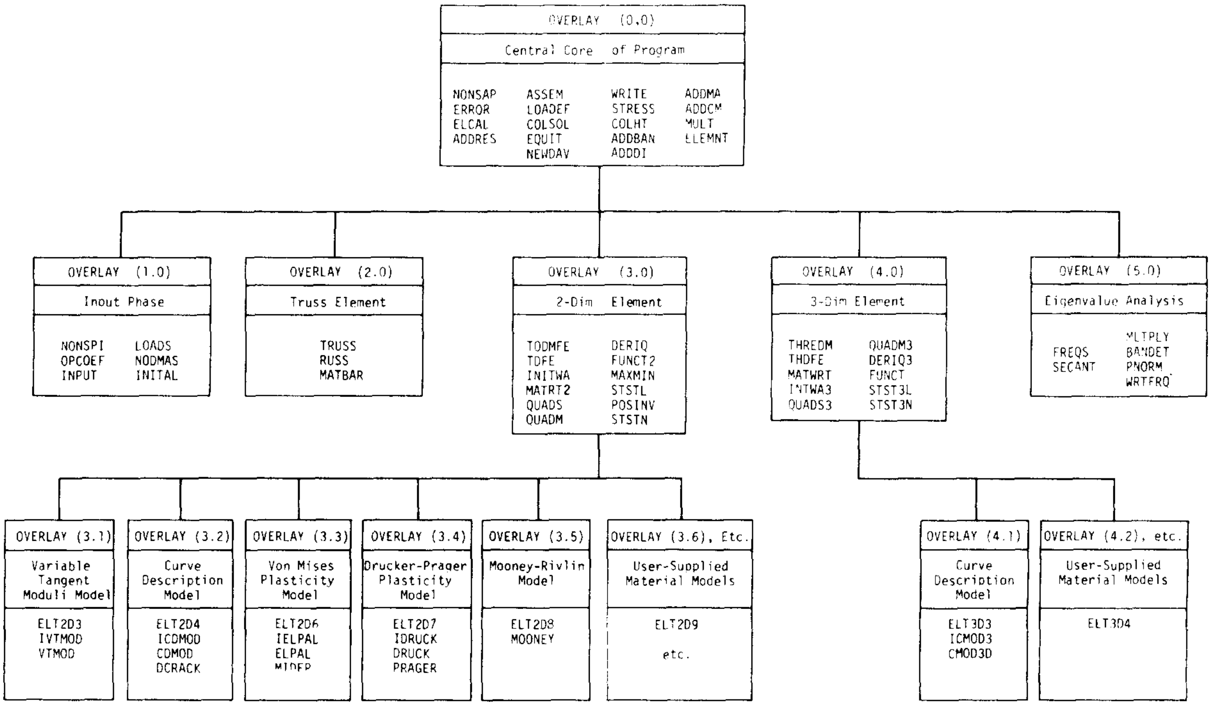
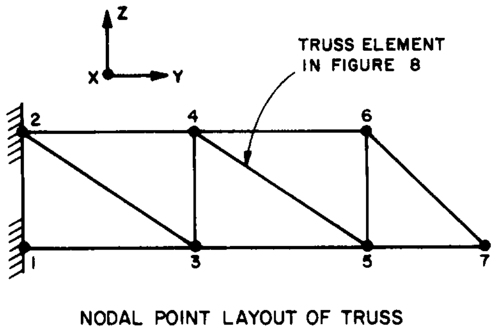


Fig. 5. Overlay structure of NONSAP.



$$
ID = \begin{array}{c|cccccc}
 & 1 & 2 & 3 & 4 & 5 & 6 \\
\hline
1 & 1 & 1 & 1 & 1 & 1 & 1 \\
2 & 1 & 1 & 1 & 1 & 1 & 1 \\
3 & 1 & 0 & 0 & 1 & 1 & 1 \\
4 & 1 & 0 & 0 & 1 & 1 & 1 \\
5 & 1 & 0 & 0 & 1 & 1 & 1 \\
6 & 1 & 0 & 0 & 1 & 1 & 1 \\
7 & 1 & 0 & 0 & 1 & 1 & 1
\end{array}
$$

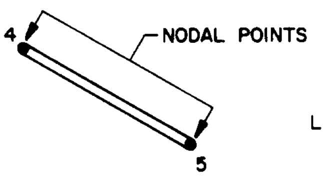
DEGREES OF FREEDOM

NODAL POINT NUMBERS

Fig. 6. Nodal point layout of truss example and *ID* array as read and/or generated.

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

Fig. 7. *ID* array of truss example after allocation of equation numbers to active degrees of freedom.



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

Fig. 8. Connection array (vector *LM*) for a typical element of the truss example.

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 section 7).

3.5. 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 to obtain maximum 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 [16]. Fig. 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 [16]. The algorithm performs – for practical purposes – a minimum number of arithmetic operations, since only the elements within the skyline of the matrix are processed. The algorithm is used in all analysis types, i.e. in linear, nonlinear, static or dynamic analysis, and consists of the $\boldsymbol{LDL}^{\mathrm{T}}$ 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 $\boldsymbol{Ku} = \boldsymbol{R}$ and the program calculates

$$\boldsymbol{K} = \boldsymbol{LDL}^{\mathrm{T}}, \tag{5}$$

$$\boldsymbol{Lv} = \boldsymbol{R}, \tag{6}$$

$$\boldsymbol{DL}^{\mathrm{T}}\boldsymbol{u} = \boldsymbol{v}, \tag{7}$$

where $\boldsymbol{L}$ and $\boldsymbol{D}$ are a lower triangular and a diagonal matrix, respectively.

4. Element library

In the current version of program NONSAP all finite elements are isoparametric (or subparametric) elements [5]. Corresponding to the nonlinearities in the system, four different analysis procedures may be considered for a finite element:

(1) 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.

(2) Material nonlinear only analysis. The displacements of the element are negligibly small, and the strains are infinitesimal. The material stress–strain description is nonlinear.

(3) Total Lagrangian formulation. The element may experience large displacements and large strains. The material stress–strain relationship is linear or nonlinear.

(4) 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 [10]. The different linear and nonlinear material models currently available in NONSAP are described in section 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 [10].

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 11–13.

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 section 5. As noted earlier, the truss element can be used to specify nonzero boundary displacements [7].

4.2. Plane stress and plane strain element

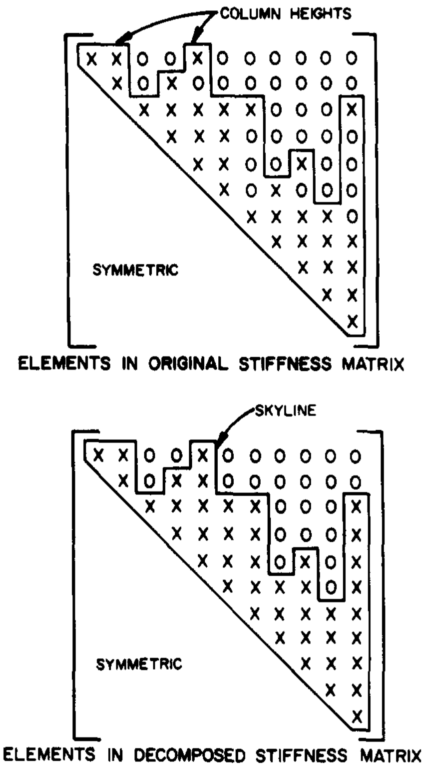
A variable-number-nodes isoparametric finite element is available for two-dimensional plane stress and plane strain analysis. The element may have from 4 to 8 nodes, where any one of the nodes 5–8 can be omitted. The variable-number-nodes option allows effective modelling from coarse to finer finite element meshes. The plane stress and plane strain element can be used in all four formulations. The material models available are described in section 5.

LINEAR ANALYSIS	
STATIC ANALYSIS	DYNAMIC ANALYSIS
1) Calculate linear structure stiffness matrix K. 2) Triangularize K and store on tape 7.	1) Calculate linear structure stiffness matrix K. 2) If Rayleigh damping is specified store K on tape 4. 3) If frequency analysis is to be performed, store K on tape 10. 4) Add mass and damping effects to K to obtain linear effective stiffness matrix $\hat{K}$. 5) Triangularize $\hat{K}$ and store on tape 7. 6) Calculate mass matrix M and store on tape 4 (if consistent mass matrix) or tape 7 (if diagonal mass matrix). 7) If concentrated nodal dampers specified, store nodal damping vector C_d on tape 4.
Tape Layout	Tape Layout
Tape 4: (not used)	Tape 4: K (if Rayleigh damping) M (consistent mass case only) C_d (if nodal damping)
Tape 7: $K = LDL^T$	Tape 7: $\hat{K} = LDL^T$ M (diagonal mass case only)
Tape 10: (not used)	Tape 10: K (for frequency analysis only)

Fig. 9. Assemblage of constant structure matrices.

<table>
<tr><th colspan="2">NONLINEAR ANALYSIS</th></tr>
<tr><th>STATIC ANALYSIS</th><th>DYNAMIC ANALYSIS</th></tr>
<tr><td>1) Calculate linear structure stiffness matrix K, i.e., that part of the total structure stiffness matrix which corresponds to the linear element groups.
2) Store K on tape 4 and on tape 7.</td><td>1) Calculate linear structure stiffness matrix K.
2) Store K on tape 4.
3) If frequency analysis is to performed, store K on tape 10.
4) Add mass and damping effects to K to obtain linear effective stiffness matrix $\hat{K}$.
5) Store $\hat{K}$ on tape 7.
6) Calculate mass matrix M and store on tape 4 (if consistent mass matrix) or on tape 7 (if diagonal mass matrix).
7) If concentrated nodal dampers specified, store nodal damping vector C_d on tape 4.</td></tr>
<tr><td>Tape Layout
Tape 4: K
Tape 7: K
Tape 10: (not used)</td><td>Tape Layout
Tape 4: K
M (consistent mass case only)
C_d (if nodal damping)
Tape 7: $\hat{K}$
M (diagonal mass case only)
Tape 10: K (for frequency analysis only)</td></tr>
</table>

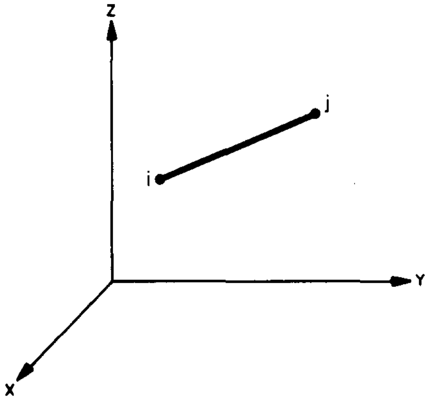
Fig. 9 (continued).



X	X	0	0	X	0	0	0	0	0	0
	X	0	X	0	0	0	0	0	0	0
		X	X	X	X	X	0	0	0	X
			X	X	X	0	0	0	0	0
				X	X	0	0	X	0	0
					X	0	X	0	0	0
						X	X	X	X	0
							X	X	X	X
								X	X	X
									X	X
										X

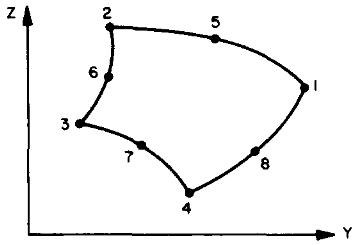
X	X	0	0	X	0	0	0	0	0	0
	X	0	X	X	0	0	0	0	0	0
		X	X	X	X	X	0	0	0	X
			X	X	X	X	0	0	0	X
				X	X	X	0	X	0	X
					X	X	X	X	0	X
						X	X	X	X	X
							X	X	X	X
								X	X	X
									X	X
										X

Fig. 10. Typical element pattern in a stiffness matrix. X = nonzero element, and 0 = zero element.



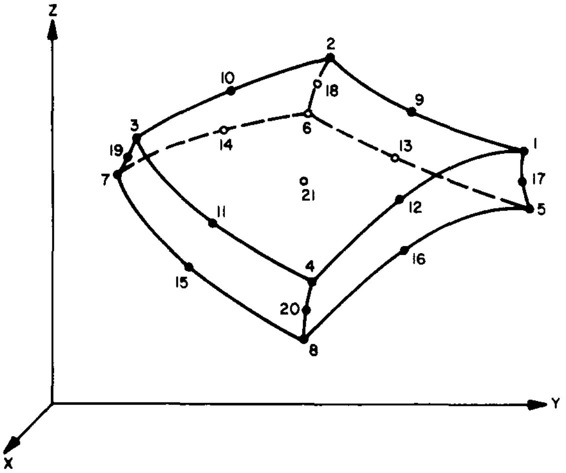
AVAILABLE NONLINEAR FORMULATIONS	AVAILABLE MATERIAL MODELS
1. LINEAR ANALYSIS	1. LINEAR ELASTIC
2. MATERIALLY NONLINEAR ONLY	2. NONLINEAR ELASTIC
3. UPDATED LAGRANGIAN WITH LARGE DISPLACEMENTS BUT SMALL STRAINS	

Fig. 11. Truss element.



AVAILABLE NONLINEAR FORMULATIONS	AVAILABLE MATERIAL MODELS
1. LINEAR ANALYSIS	1. LINEAR ISOTROPIC ELASTIC
2. MATERIALLY NONLINEAR ONLY	2. LINEAR ORTHOTROPIC ELASTIC
3. UPDATED LAGRANGIAN	3. VARIABLE TANGENT MODULI MODEL
4. TOTAL LAGRANGIAN	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)

Fig. 12. Two-dimensional plane stress, plane strain and axisymmetric elements.



AVAILABLE NONLINEAR FORMULATIONS	AVAILABLE MATERIAL MODELS
1. LINEAR ANALYSIS	1. LINEAR ISOTROPIC ELASTIC
2. MATERIALLY NONLINEAR ONLY	2. CURVE DESCRIPTION NONLINEAR MODEL

Fig. 13. Three-dimensional solid and thick shell element.

4.3. Axisymmetric shell or solid element

The variable-number-nodes 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 eight nodes are the corner nodes of the element, nodes 9–20 correspond to midside nodes and nodes 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.

5. 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 ref. [10].

5.1. Truss element material models

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

(1) Linear elastic material. The material can be linear elastic defined by Young's modulus only.

(2) Nonlinear elastic material. The nonlinear elastic material behavior is defined by specifying the stress as a piecewise 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 ref. [10], where the assumptions used are pointed out.

5.2.1. Isotropic and orthotropic linear elastic material
The stress–strain relationships are defined by means of the constant Young's moduli and Poisson's ratios [28].

5.2.2. Mooney–Rivlin material model
A hyperelastic incompressible material model is available for the analysis of rubber-like materials [3, 17]. The stress–strain relationship is defined using the Mooney–Rivlin material constants. In NONSAP the model can only be used in plane stress analysis.

5.2.3. 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 [2, 18–20]. 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.

5.2.4. Variable tangent moduli model
The variable tangent moduli model is available for the analysis of geological materials [21]. 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.

5.2.5. 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. To estimate an appropriate time step, it may be necessary to solve for the fundamental frequencies of the system [11]. For this purpose an eigenvalue solution routine has been incorporated into NONSAP.

The algorithm considers the solution of the generalized eigenvalue problem

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

where ${}^0\boldsymbol{K}$ is the tangent stiffness matrix at time 0, $\boldsymbol{M}$ is the mass matrix of the system and ω and $\boldsymbol{\phi}$ 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 ${}^0\boldsymbol{K}$ is assumed to be positive definite. The solution to eq. (8) can be written as

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

where $\boldsymbol{\Phi}$ is a matrix with its columns equal to the mass-orthonormalized eigenvectors and $\boldsymbol{\Omega}^2$ is a diagonal matrix of the corresponding eigenvalues, i.e.

$$ \boldsymbol{\Phi} = [\boldsymbol{\phi}_1 \boldsymbol{\phi}_2 \ldots \boldsymbol{\phi}_n];\ \boldsymbol{\Omega}^2 = \mathrm{diag}\,(\omega_i^2). \tag{10}$$

The solution algorithm used in NONSAP is the determinant search method presented in ref. [22]. Basically, the algorithm combines triangular factorization and vector inverse iteration in an optimum manner to calculate the required eigenvalues and eigenvectors; these are obtained in sequence starting from the least dominant eigenpair $(\omega_1^2, \boldsymbol{\phi}_1)$. An efficient accelerated secant iteration procedure which operates on the characteristic polynomial

$$ p(\omega^2) = \det\,(\boldsymbol{K} - \omega^2\boldsymbol{M}) \tag{11}$$

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 $\boldsymbol{K} - \omega^2\boldsymbol{M}$. 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 section 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. Fig. 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)

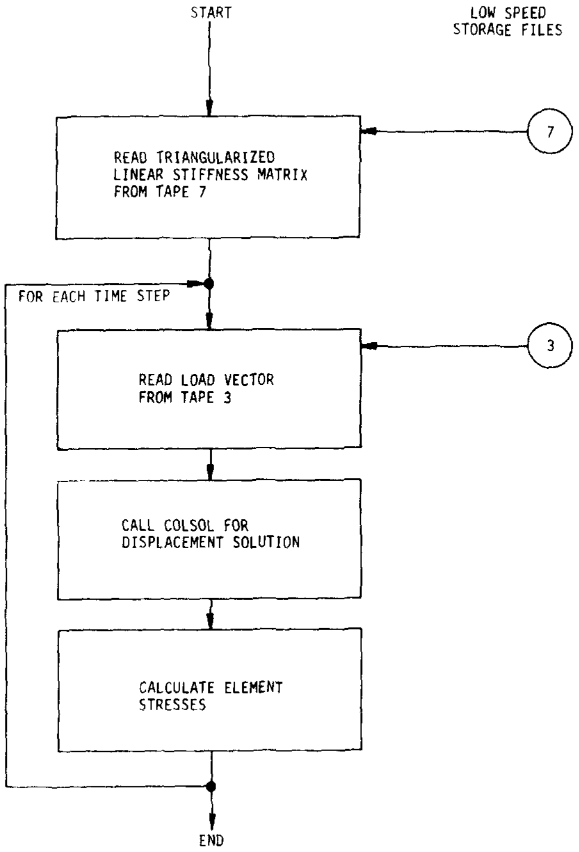


Fig. 14. Flow chart for step-by-step solution in linear static analysis.

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.

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

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 [14]. 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. Fig. 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 per-

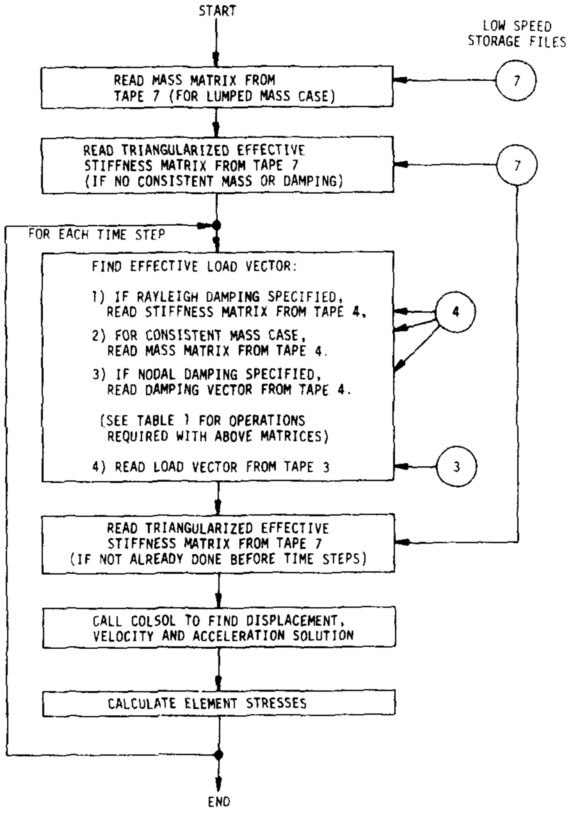


Fig. 15. Flow chart for step-by-step solution in linear dynamic analysis.

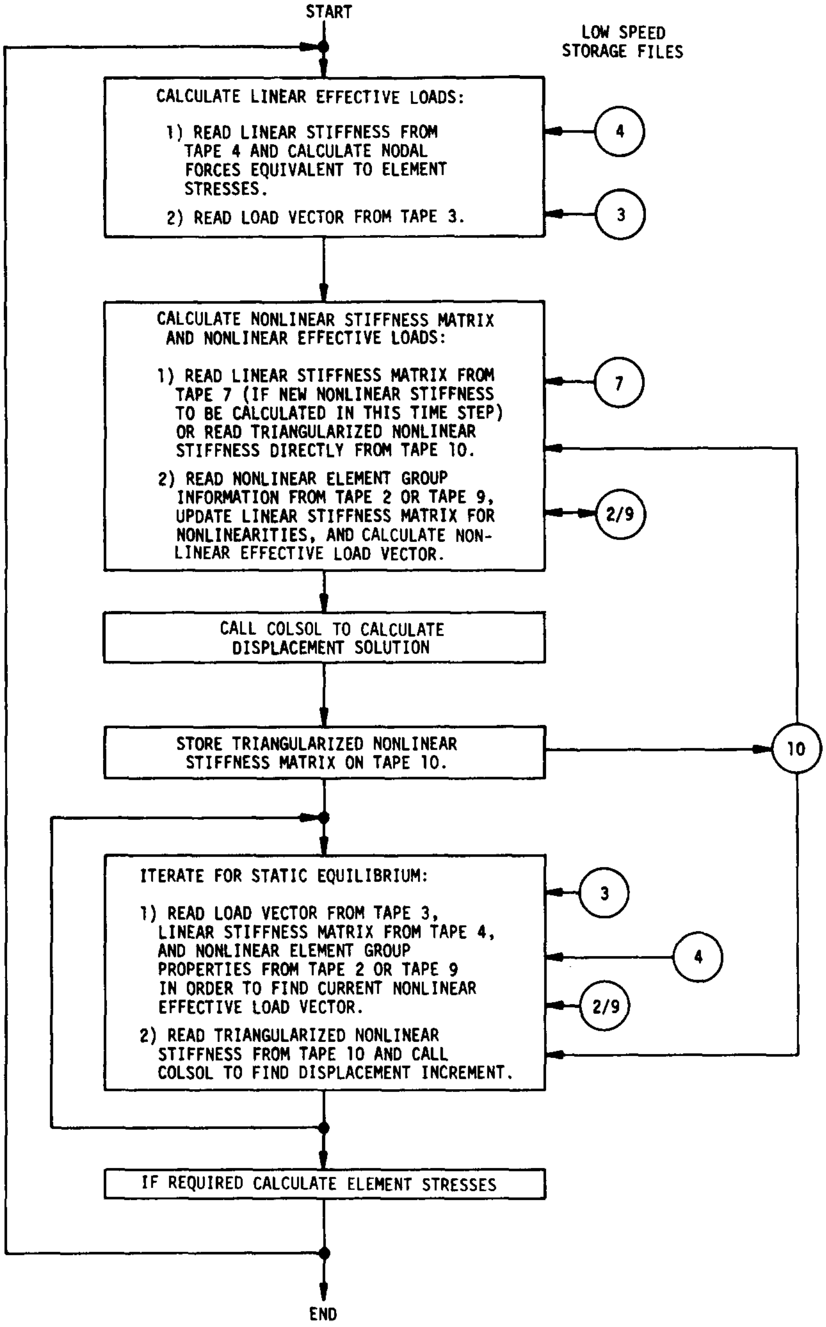


Fig. 16. Flow chart for step-by-step solution in nonlinear static analysis.

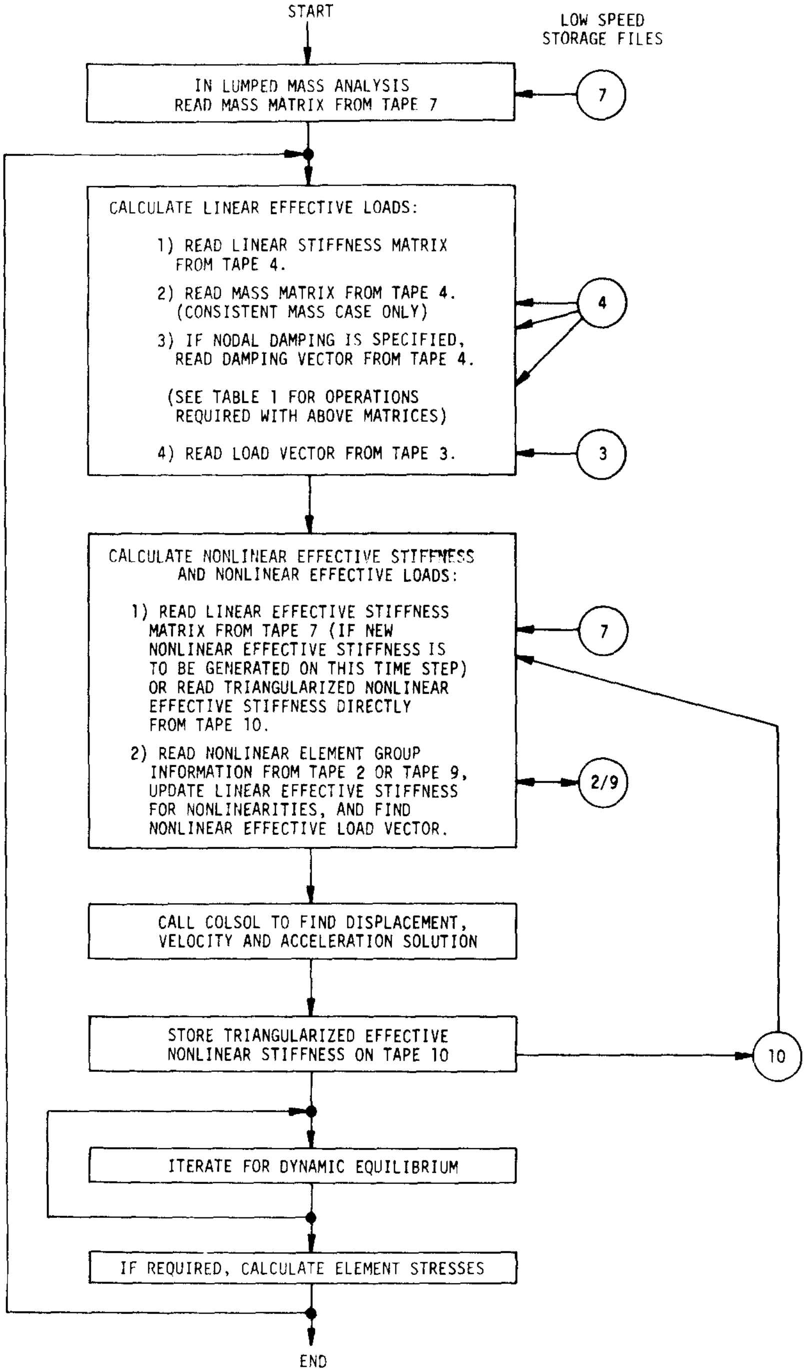


Fig. 17. Flow chart for step-by-step solution in nonlinear dynamic analysis.

formed, can be defined in the input control data. 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.

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.

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. Sample analyses

In the following the solutions of some problems are presented that have been considered during the development of NONSAP. Additional problem solutions are given in refs [9] and [10]. All solutions have been obtained using the algorithm presented in table 1, in which the selected parameters were $tol = 0.001$, $nitem = 15$, $\theta = 1.4$, $\delta = 0.50$ and $\alpha = 0.25$. Since only relatively small order systems have been considered the solution times have always been small [10].

10.1. 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. The total vertical load acting on the cable nodes was 5677.83 lb which includes the insulator weights and the cable selfweight.

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.

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

10.2. Static and dynamic displacement analysis of a cantilever

The cantilever in fig. 20 under uniformly distributed load was analyzed using a finite element idealization of five eight-node plane stress elements. The material of the cantilever was assumed to be isotropic linear elastic.

The static response of the cantilever using 100 load steps to reach the final equilibrium configuration is shown in fig. 21, in which the NONSAP solution is compared with an analytical solution by Holden [23]. Excellent agreement between the solutions has been obtained. The dynamic response of the cantilever is shown in fig. 22, where also the importance of equilibrium iterations in an analysis using a relatively large time step Δt is demonstrated [10].

10.3. Static large displacement analysis of a spherical shell

The spherical shell subjected to a concentrated apex load shown in fig. 23 was analyzed for static response. The NONSAP solution could be compared with the response predicted by Stricklin [24] and Mescall [25].

Table 2.

Vibration periods of cable in static equilibrium configuration.

Mode number	Period (sec)
1	4.42
2	2.31
3	1.21
4	1.16
5	0.929

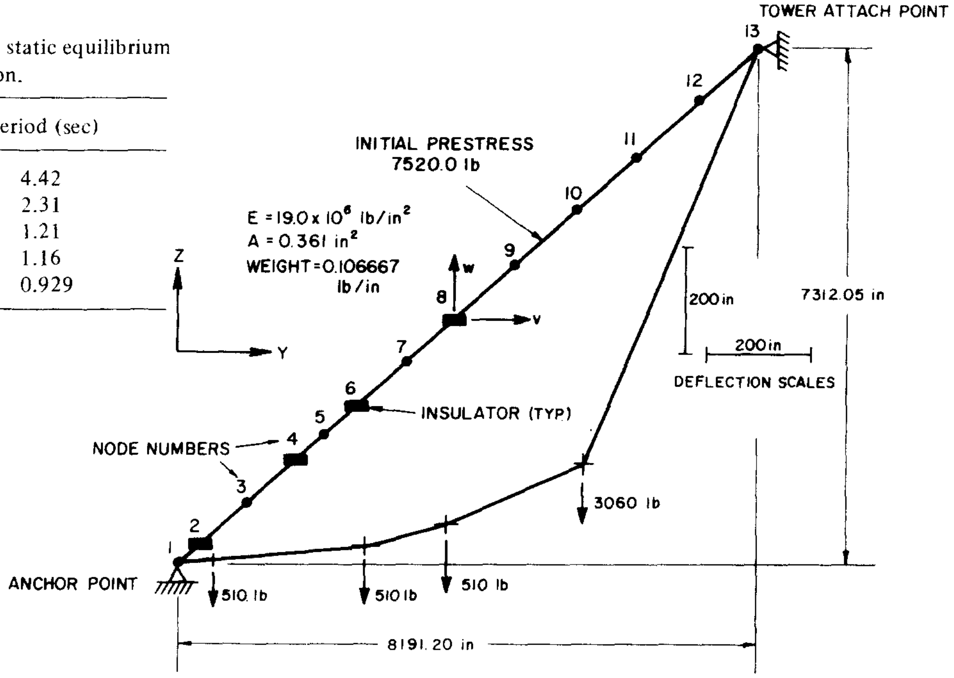


Fig. 18. Static configuration of tower cable.

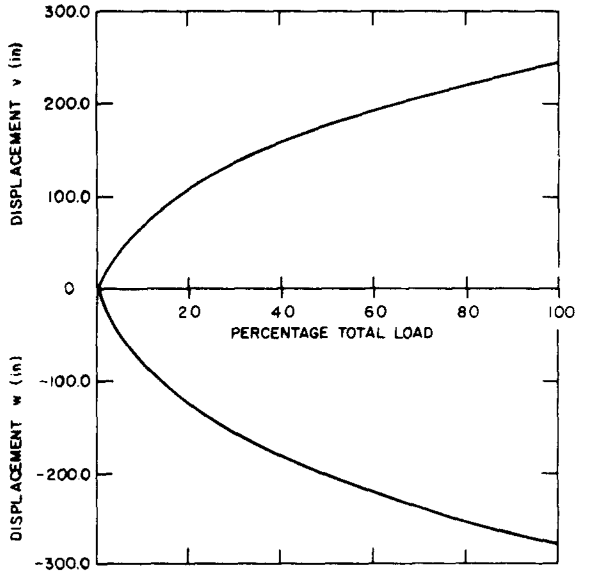


Fig. 19. Load deflection curve of tower cable.

Figure 23 shows the static load-deflection response calculated by NONSAP. Good correspondence with the solutions obtained by Stricklin and Mescall is observed.

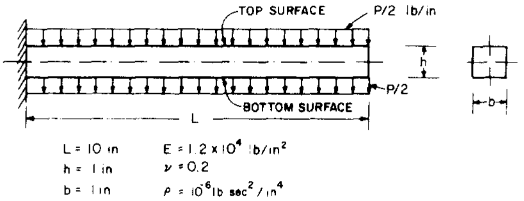


Fig. 20. Cantilever under uniformly distributed load.

10.4. Large displacement and large strain static analysis of a rubber sheet

The rubber sheet shown in fig. 24 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} [17].

Figure 25 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 four equal load steps with an average of five equilibrium iterations in each step. Excellent

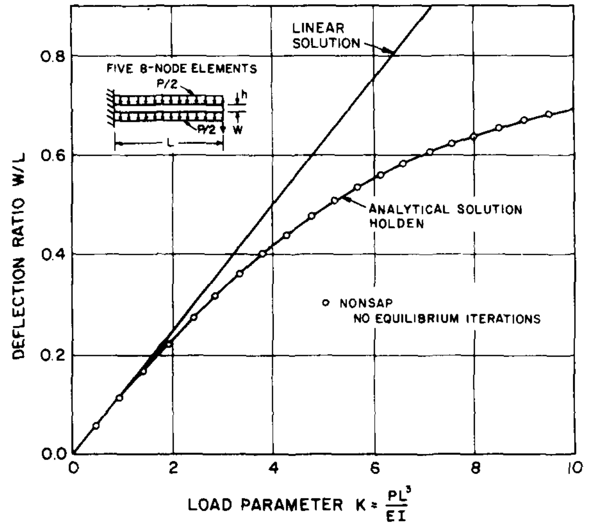


Fig. 21. Large deflection analysis of cantilever under uniformly distributed load.

agreement between the experimental results by Iding et al. [17] and those predicted by NONSAP has been obtained.

10.5. Elastic–plastic static analysis of a thick-walled cylinder

The thick-walled cylinder in fig. 26 subjected to internal pressure was analyzed using four eight-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. Since displacements and strains are small, the analysis of the cylinder was carried out using the materially nonlinear only formulation. Fig. 27 shows the radial displacement response of the cylinder as a function of the applied load, and fig. 28 gives the stress distribution

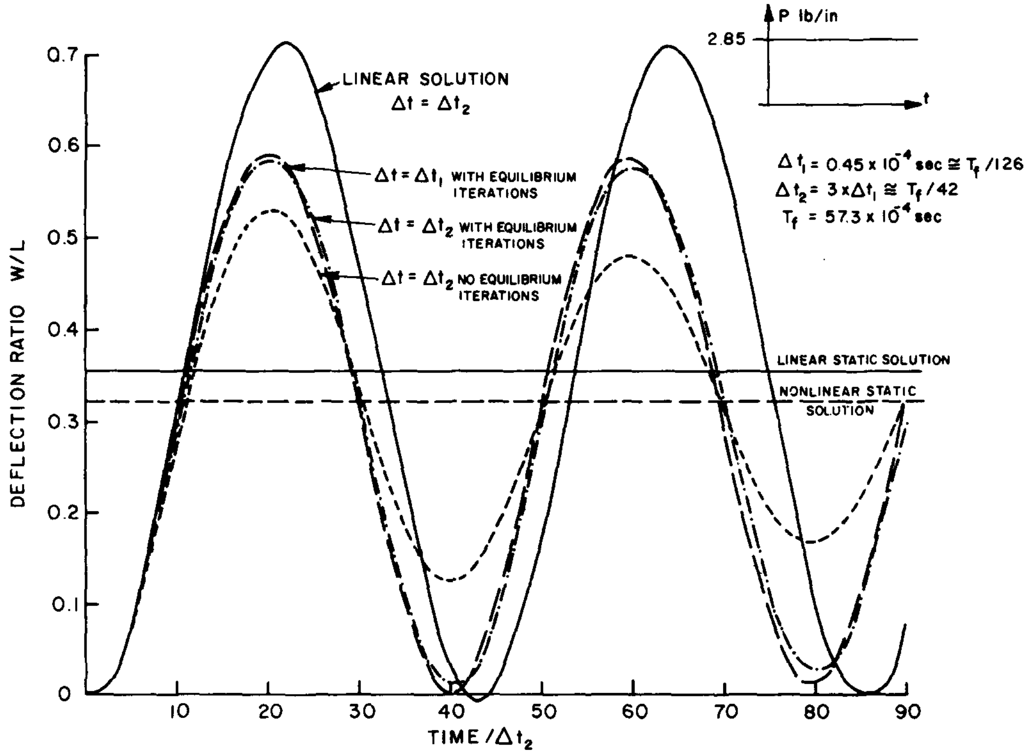


Fig. 22. Large displacement dynamic response of cantilever under uniformly distributed load, Newmark method $\delta = 0.50$, $\alpha = 0.25$.

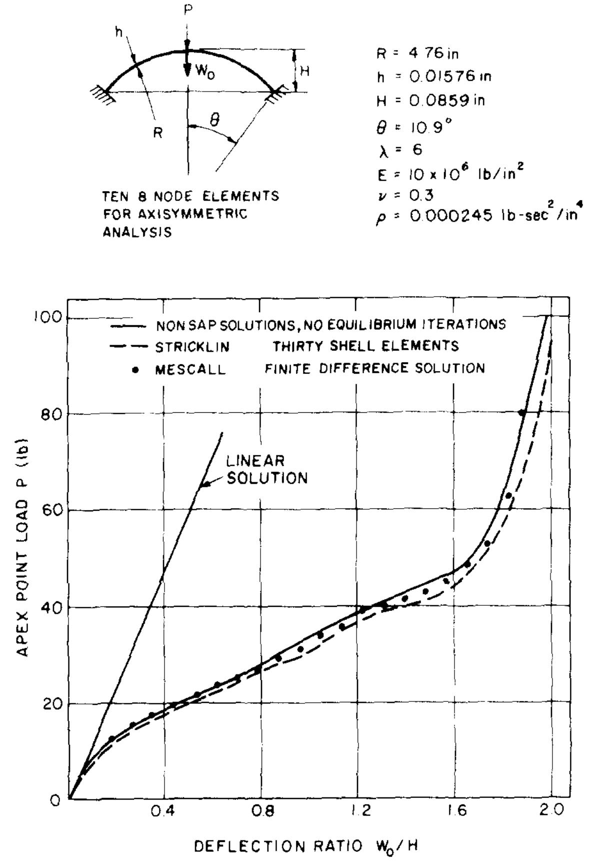


Fig. 23. Load–deflection curves for spherical shell.

through the wall of the cylinder at a given level of internal pressure. Excellent agreement with the solution given by Hodge and White has been obtained [26].

10.6. Elastic–plastic small displacement dynamic analysis of a simply-supported beam

The beam shown in fig. 29 was analyzed for the step loading indicated. The material of the beam was taken to be elastic perfectly plastic using the von Mises yield condition. In the analysis small displacements were assumed, i.e. materially nonlinear only solutions were calculated.

The dynamic response of the beam is shown in fig. 30, in which the NONSAP solutions are compared with solutions provided by Baron et al. [27] and Nagarajan and Popov [28].

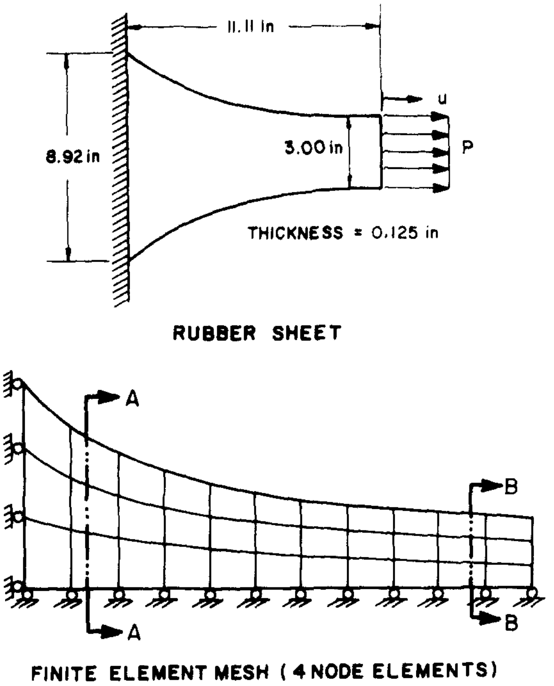


Fig. 24. Large displacement and large strain static analysis of a rubber sheet.

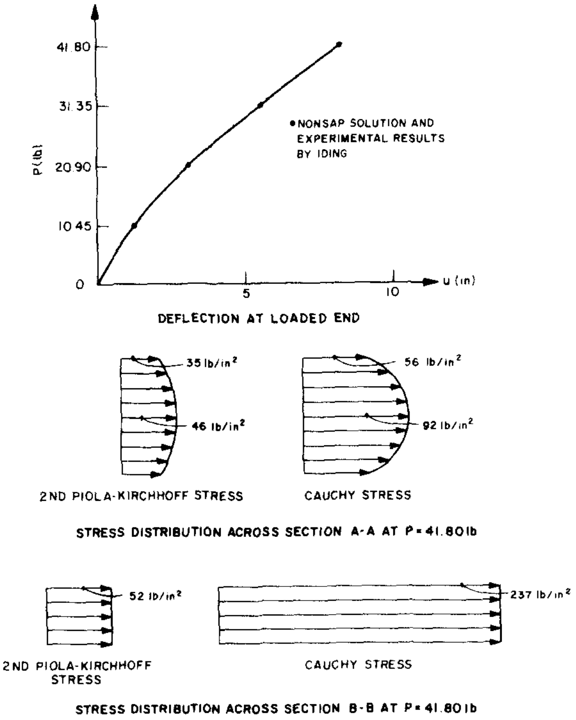


Fig. 25. Displacement and stress response of a rubber sheet.

10.7. Static analysis of an underground opening

A simplified analysis of an undergorund opening under static overburden pressure was carried out. Fig. 31 shows the underground opening, the finite element mesh and the material data used. The analysis was performed using the materially nonlinear only formulation, i.e. large displacement effects were neglected. The rock material was assumed to be a no-tension material with constant Young's modulus and Poisson's ratio.

Figure 32 gives the load–deflection relations for two points of the opening. The influence of the no-tension material assumption on the displacements can be observed. Fig. 33 shows the crack regions around the opening at two load levels.

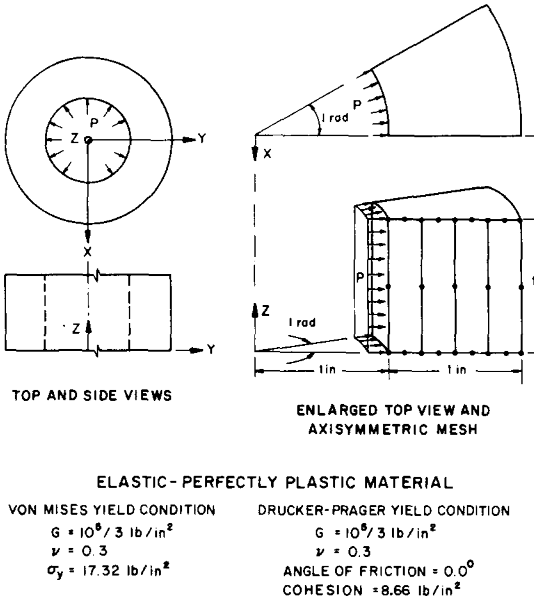


Fig. 26. Finite element mesh of thick-walled cylinder.

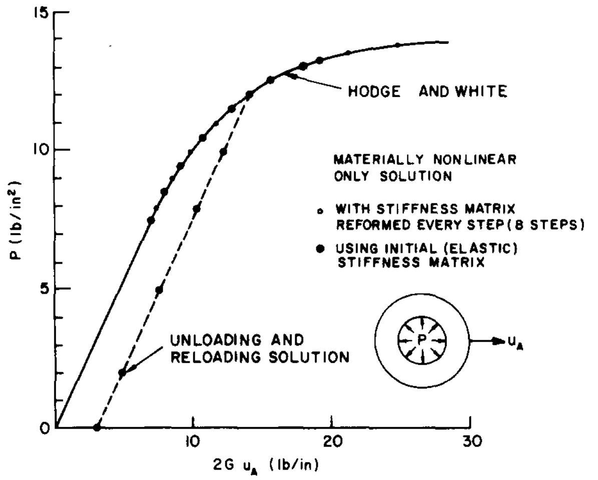


Fig. 27. Elastic–plastic displacement response of thick-walled cylinder.

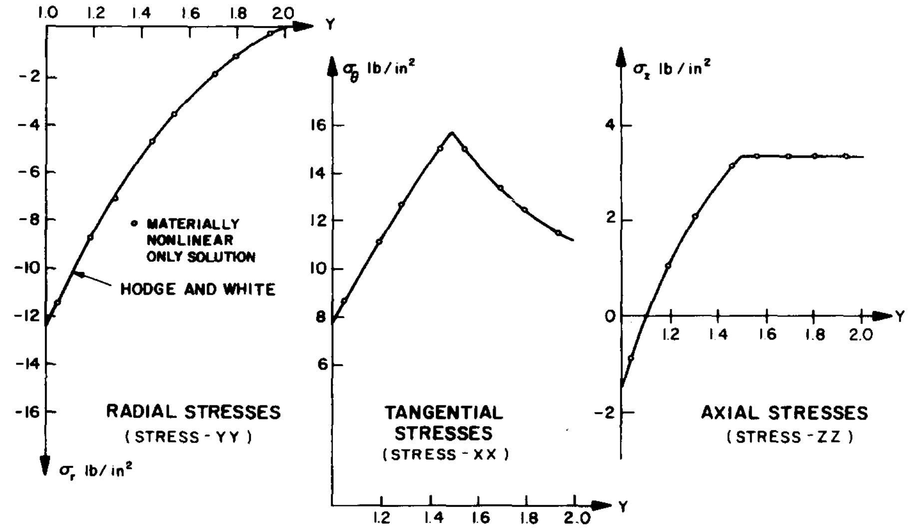


Fig. 28. Elastic–plastic stress distribution through thickness of thick-walled cylinder at $P = 12.5$ lb/in^2.

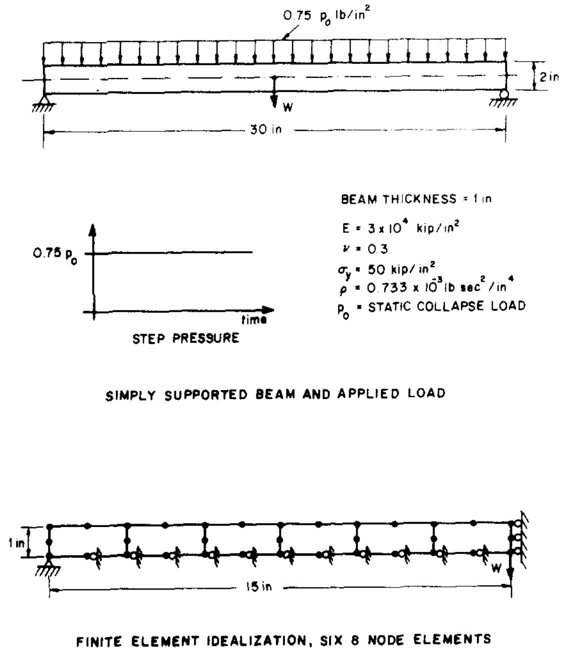


Fig. 29. Elastic–plastic dynamic analysis of simply-supported beam.

11. Concluding remarks

The objective in this paper 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 have been presented.

Although NONSAP can be a 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 [7]. 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 [4, 10]. Therefore, it is necessary to apply the program only under the conditions and assumptions for which it was developed.

One important option which NONSAP does not have available is efficient pre- and postprocessing.

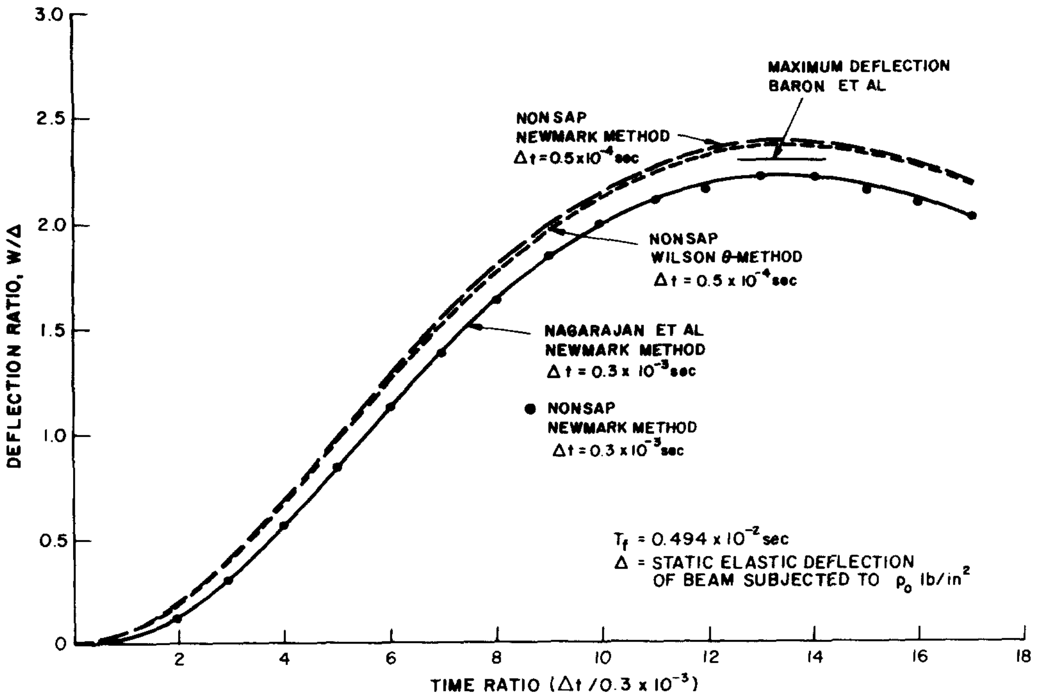


Fig. 30. Initial elastic–plastic displacement response of simply-supported beam.

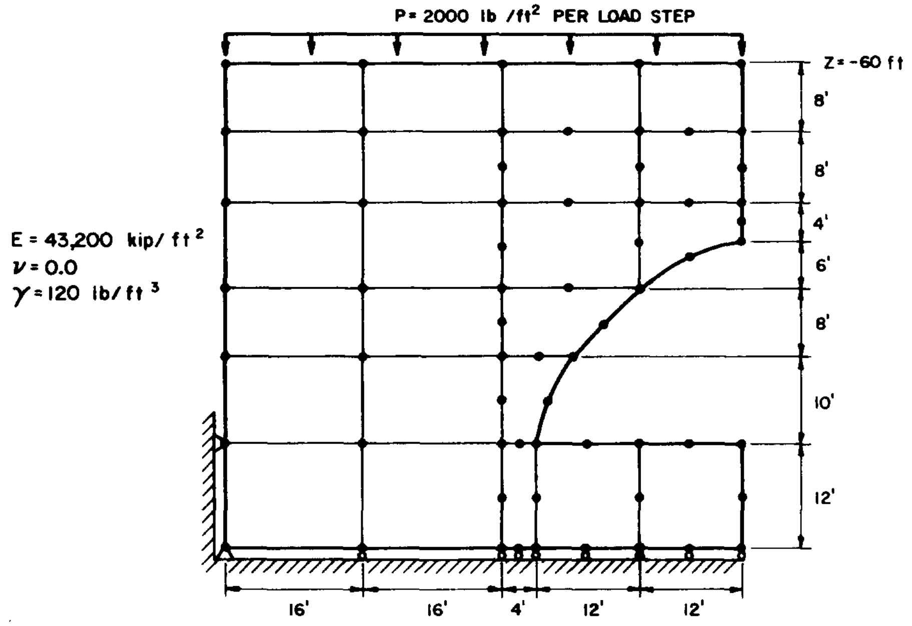


Fig. 31. Finite element mesh for analysis of underground opening.

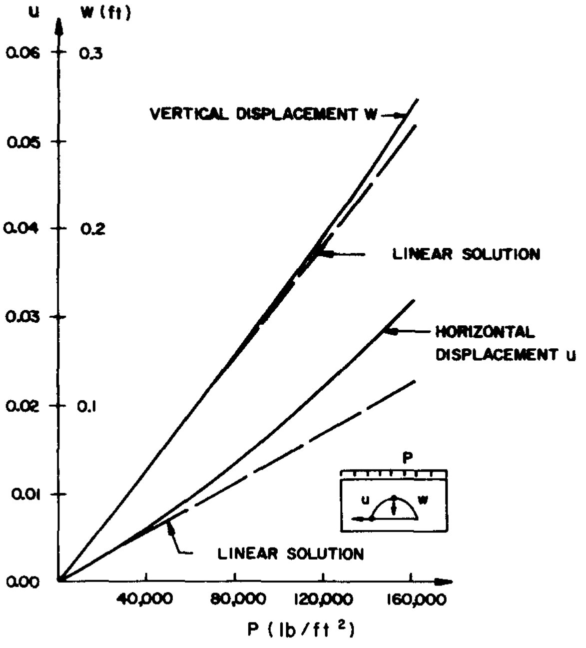


Fig. 32. Load–deflection response of underground opening.

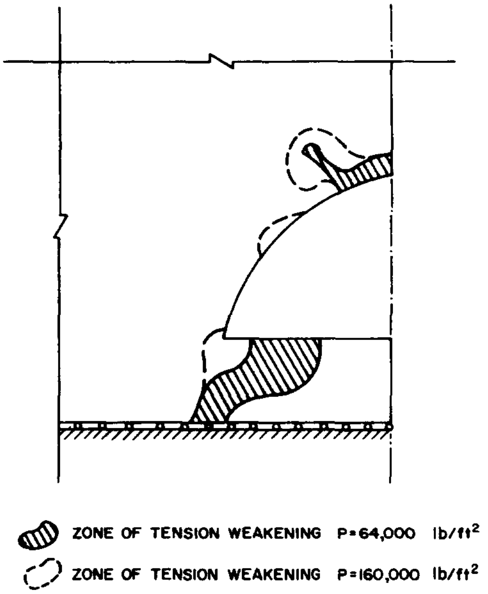


Fig. 33. Cracked regions around underground opening.

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 future work on NONSAP, it is hoped that the program can be further developed in various areas. It can be important to have out-of-core solution capability, the element library need be increased and additional material models are required. Altogether, the program provides a basis for further work in a variety of problem areas, such as in thermal elastic–plastic and creep analysis, buckling analysis, and soil response calculations.

Acknowledgements

The authors would like to express their appreciation to R.H. Iding, H. Ozdemir and E. Ramm for their assistance during the development of NONSAP.

References

[1] J.H. Argyris, P.C. Dunne and T. Angelopoulos, Nonlinear oscillations using the finite element technique, Computer Methods in Applied Mechanics and Engineering, 2 (1973) 203–250.

[2] G.C. Nayak and O.C. Zienkiewicz, Elasto-plastic stress analysis. A generalization for various constitutive relations including strain softening, Int. J. Numer. Method Eng. 5 (1972) 113–135.

[3] J.T. Oden, Finite Elements of Nonlinear Continua, McGraw-Hill, New York (1972).

[4] J.A. Stricklin, W.A. Von Riesemann, J.R. Tillerson and W.E. Haisler, Static geometric and material nonlinear analysis, Proceedings, 2nd US–Japan Symposium on Recent Advances in Computational Methods of Structural Analysis and Design, Berkeley, California (1972).

[5] O.C. Zienkiewicz, The Finite Element Method in Engineering Science, McGraw-Hill, London (1971).

[6] P.V. Marcal, Survey of general purpose programs, Proceedings, 2nd US–Japan Symposium on Recent Advances in Computational Methods of Structural Analysis and Design, Berkeley, California (1972).

[7] K.J. Bathe, E.L. Wilson and F.E. Peterson, 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; also: E.L. Wilson, K.J. Bathe, F.E. Peterson, H.H. Dovey, SAP – A structural analysis program for linear systems, Nucl. Eng. Des. 25 (1973) 257–274.

[8] K.J. Bathe and E.L. Wilson, NONSAP – a general finite element program for nonlinear dynamic analysis of complex structures, Paper M3/1*, Preprints 2nd Int. Conference on Structural Mechanics in Reactor Technology, Berlin, Sept. 1973.

[9] K.J. Bathe, E. Ramm and E.L. Wilson, Finite element formulations for large deformation dynamic analysis, Int. J. Numer. Meth. Eng. (to appear).

[10] K.J. Bathe, H. Ozdemir and E.L. Wilson, Static and dynamic geometric and material nonlinear analysis, SESM Report, No. 74–4, Dept. of Civil Engineering, University of California, Berkeley, Feb. (1974).

[11] K.J. Bathe and E.L. Wilson, Stability and accuracy analysis of direct integration methods, Int. J. Earthquake Eng. Struct. Dyn. 1 (1973) 283–291.

[12] R.E. Nickell, Direct integration methods in structural dynamics, ASCE, J. Eng. Mech. Div. 99 (1973) 303–317.

[13] J.S. Przemieniecki, Theory of Matrix Structural Analysis, McGraw-Hill, New York (1968).

[14] R.W. Clough and K.J. Bathe, Finite element analysis of dynamic response, Proceedings, 2nd US–Japan Symposium on Recent Advances in Computational Methods of Structural Analysis and Design, Berkeley, California (1972).

[15] W. Hurty and M.F. Rubinstein, Dynamics of Structures, Prentice-Hall (1964).

[16] E.L. Wilson, K.J. Bathe and W.P. Doherty, Direct solution of large systems of linear equations. Comput. Struct. 4 (1974) 363–372.

[17] R.H. Iding, Identification of nonlinear materials by finite element methods, SESM Report, No. 73–4, Department of Civil Engineering, University of California, Berkeley, Jan. (1973); and Private communication to be published.

[18] G.C. Nayak and O.C. Zienkiewicz, Convenient form of stress invariants for plasticity, ASCE, J. Struct. Div. 98 (1972) 949–954.

[19] C.E. Pugh, J.M. Corum, K.C. Liu and W.L. Greenstreet, Currently recommended constitutive equations for inelastic design analysis of FFTF components, Report, No. TM-3802, Oak Ridge National Laboratory, Oak Ridge, Tennessee (1972).

[20] S.F. Reyes and D.U. Deere, Elastic–plastic analysis of underground openings, Proceedings, First Congress, International Society of Rock Mechanics, Lisbon (1966).

[21] I. Nelson, M.L. Baron and I. Sandler, Mathematical models for geological materials for wave propagation studies, Headquarters Defense Nuclear Agency Report, No. 2672, Washington, D.C.

[22] K.J. Bathe and E.L. Wilson, Eigensolution of large structural systems with small bandwidth, ASCE, J. Eng. Mech. Div. 99 (1973) 467–479.

[23] J.T. Holden, On the finite deflections of thin beams, Int. J. Solids Struct. 8 (1972) 1051–1055.

[24] J.A. Stricklin, Geometrically nonlinear static and dynamic analysis of shells of revolution, High Speed

Computing of Elastic Structures, Proceedings of the Symposium of IUTAM, University of Liege, Aug. (1970).

[25] J.F. Mescall, Large deflections of spherical shells under concentrated loads, J. Appl. Mech. 32 (1965) 936–938.

[26] P.G. Hodge and G.H. White, A quantitative comparison of flow and deformation theories of plasticity, J. Appl. Mech. 17 (1950) 180–184.

[27] M.L. Baron, H.H. Bleich and P. Weidlinger, Dynamic elastic–plastic analysis of structures, ASCE, J. Eng. Mech. Div. 87 (1961) 23–42.

[28] S. Nagarajan and E.P. Popov, Elastic–plastic dynamic analysis of axisymmetric solids, SESM Report, No. 73–9, Department of Civil Engineering, University of California, Berkeley (1973).