ANSYS

Capability: Static and dynamic linear and nonlinear structural analysis and heat transfer analysis. Program has plasticity, creep, and large displacement and rotation capability.

Method: Finite element displacement method. Program uses the incremental method of solution accounting for plasticity with isotropic and kinematic hardening. Program uses the wave-front method coupled with an explicit time integration scheme for the solution of the nonlinear equations of motion. Eigenvalues are extracted via Jacobi iteration with Guyan reduction.

Language: FORTRAN

Hardware: Program runs on CDC, IBM, and UNIVAC machines.

Usage: Program has been extensively used in the nuclear industry and indications of its reliability are available.

Developer: John A. Swanson
Swanson Analysis Systems, Inc.
870 Pine View Drive
Elizabeth, PA 15037

Table 15-1. General Purpose Finite Element Programs.

Features	Program[a]				
	ANSYS	MARC	NASTRAN	SAP	STARDYNE
Straight beam, straight pipe, solid and flat plate elements	X	X	X	X	X
Axisymmetric elements	X	X	X	X	O
Curved beam/curved pipe elements	O/X	X/X	O/O	O/X	O/X
Curved shell elements	O	X	O	O	O
Inviscid fluid element	O	O	X	O	O
Buckling analysis	O	X	X	O	O
Shock spectra	X	O	O	X	X
Mesh generation	Yes	Yes	Yes	Some	Some
Nonlinear analysis	Extensive	Extensive	Limited	Limited[b]	None
Pages in manual describing elements, input and output (approximate)	830	820	980	130	560
Proprietary/public	Prop.	Prop.	Public	Public	Prop.
Availability[c]	CDC, W, D	CDC, D	CDC, W	CDC, D	CDC

[a] X = program has this capability; O = program lacks this capability.
[b] Nonlinear capability in MODSAP version.
[c] CDC = Control Data Corporation (Cyber); W = Westinghouse Telecomputer Center, Pittsburgh, PA; D = developer (see text).

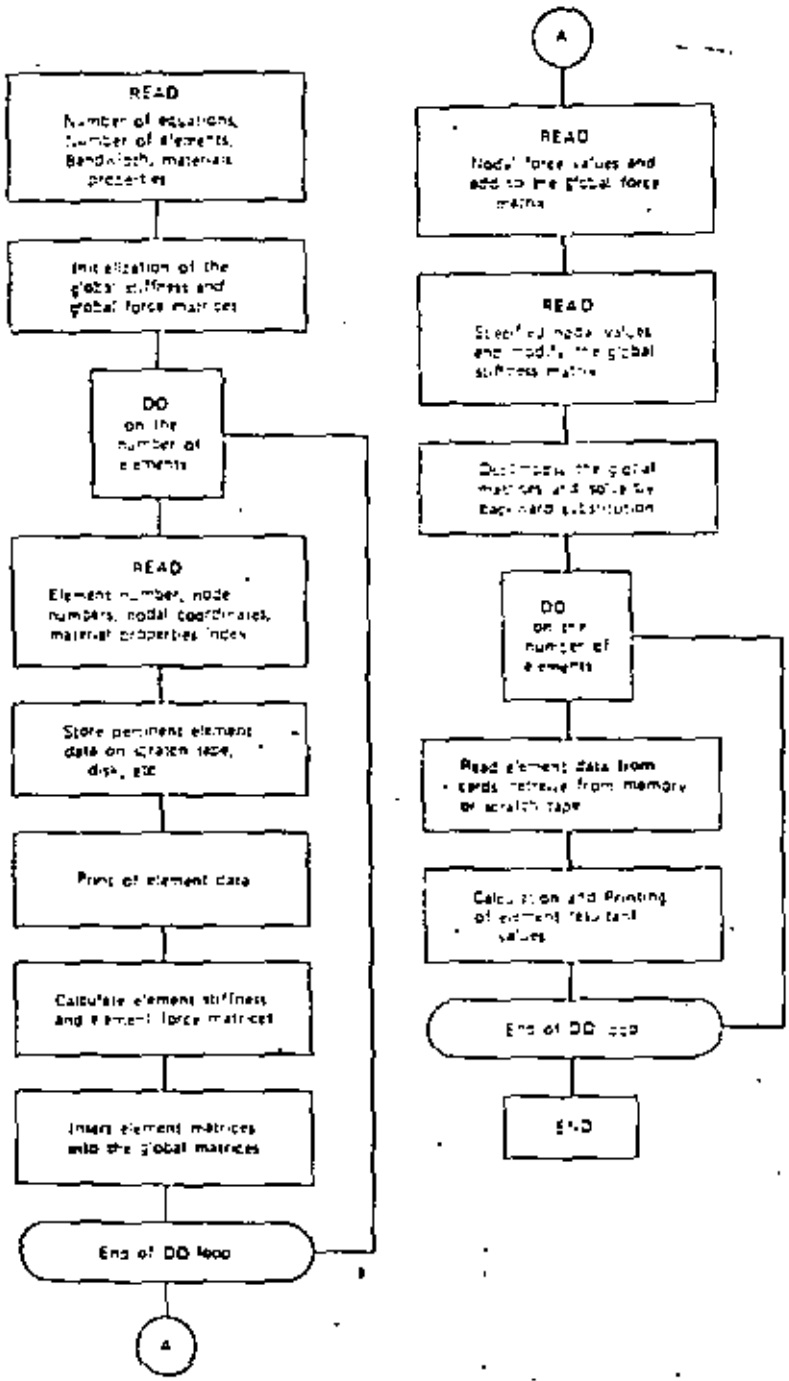
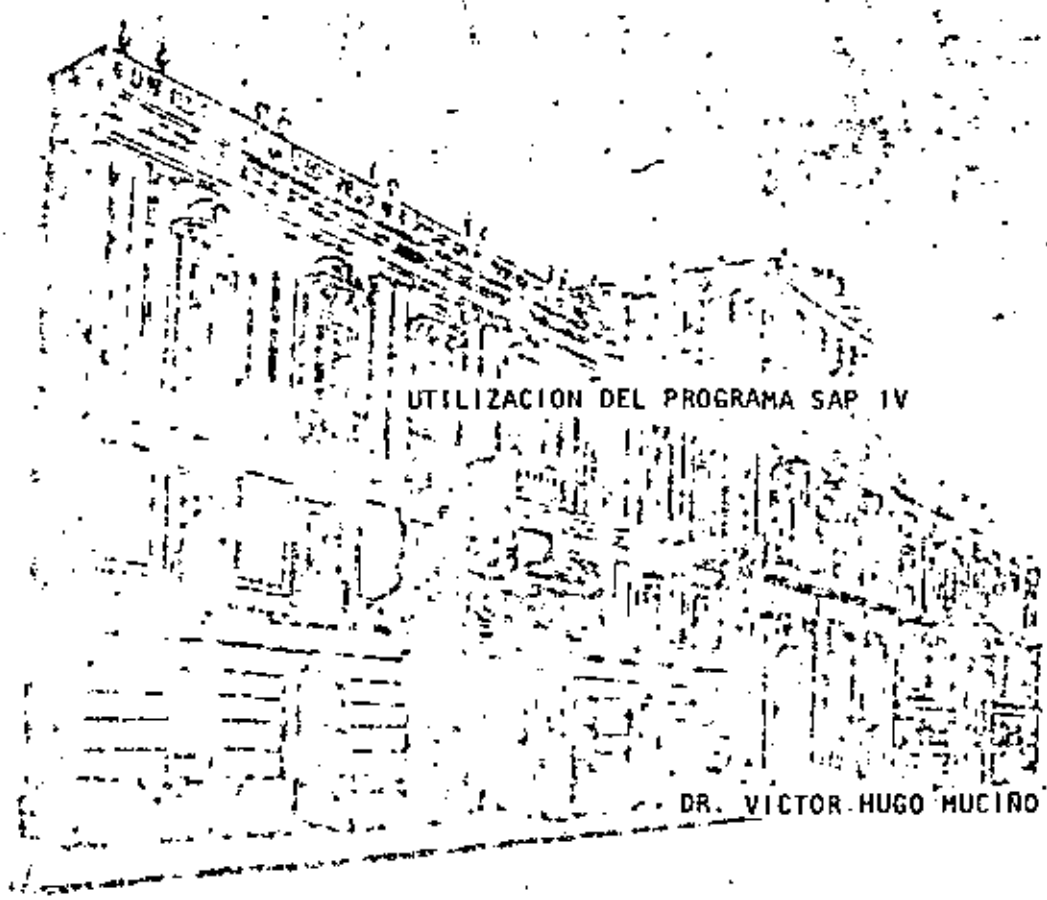


Fig. 7.3 General computer flow diagram for a finite element program.

EL METODO DEL ELEMENTO FINITO EN LA INGENIERIA MECANICA



UTILIZACION DEL PROGRAMA SAP IV

DR. VICTOR HUGO HUCIN QUINTERO

ABRIL, 1982

REPORT NO.
EERC 73-11
JUNE 1973
REVISED APRIL 1974

EARTHQUAKE ENGINEERING RESEARCH CENTER

SAP IV

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

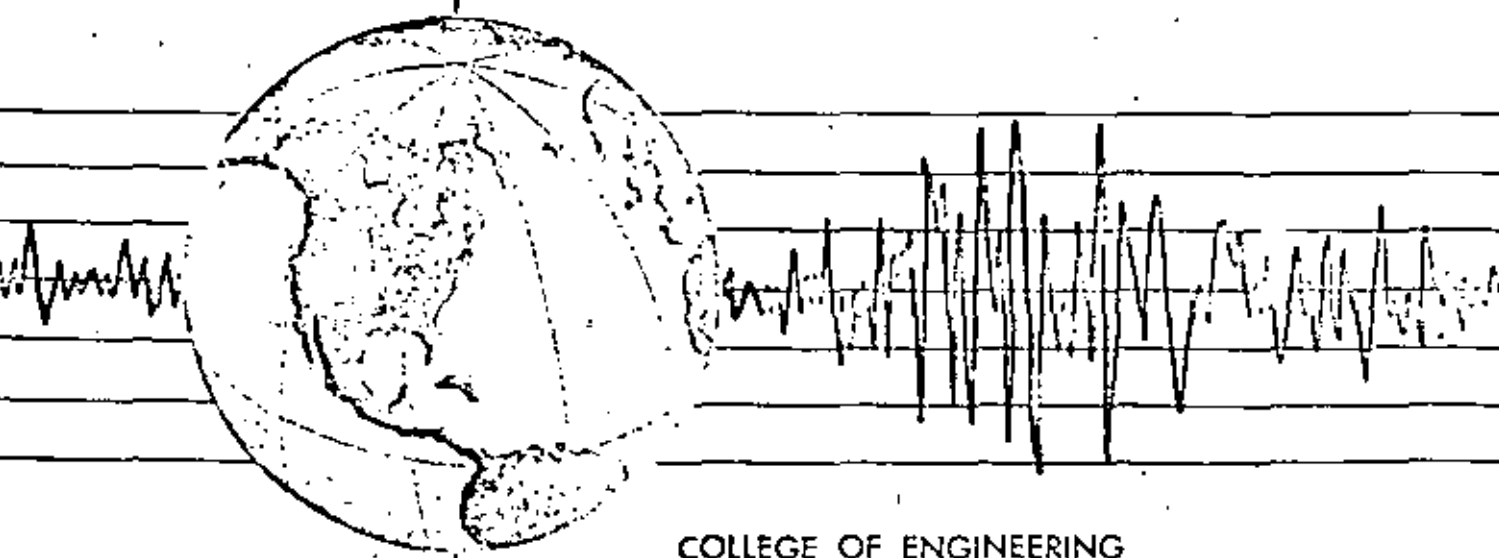
by

KLAUS-JÜRGEN BATHE

EDWARD L. WILSON

FRED E. PETERSON

A Report to the
National Science Foundation



COLLEGE OF ENGINEERING

UNIVERSITY OF CALIFORNIA · Berkeley, California

SAP IV

A STRUCTURAL ANALYSIS PROGRAM

FOR STATIC AND DYNAMIC RESPONSE OF LINEAR SYSTEMS

by

Klaus-Jürgen Bathe

Edward L. Wilson

Fred E. Peterson

June 1973

Revised April 1974

ABSTRACT

The computer program SAP IV for the static and dynamic analysis of linear structural systems is presented.

The report is divided into three parts. In the first part the reader is introduced to the logical construction of the program, the dynamic high speed storage allocation, the analysis capabilities, the finite element library and the numerical techniques used. Typical running times are given. In the second part of the report several sample analyses are described. These problems have been selected as standard problems whose solutions are provided with the program. In the last part of the report the user's manual of the program is given.

ACKNOWLEDGEMENTS

The development of the computer programs SAP including SAP IV has been supported by many organizations during the past years. The final phase of development and documentation of SAP IV was sponsored by Grants GI 36387 and GK 31586 from the National Science Foundation.

The release of the previous version of the program, SAP III, was restricted to agencies which sponsored our research. We are pleased that many institutions in Europe and the United States responded positively and that today we can make the latest version of the program available for duplication and mailing costs only. By making our work freely available, we hope that all those interested may profit from the developments that have taken place.

We would like to thank the following agencies, and in particular Engineering/Analysis Corporation, Berkeley, for their contributions towards the development of this program:

France

Informatique Internationale, Rungis

West-Germany

Germanischer Lloyd, Hamburg; Hochtief, Essen; Interatom, Bensberg/Köln; Kraftwerk Union, Erlangen; MAN, München

United States

Agbabian and Associates, Los Angeles, Calif.; Bechtel Corporation, San Francisco, Calif.; Beloit Corporation, Beloit, Wisconsin; Byron Jackson Pump Division of Borg Warner, Los Angeles, Calif.; Dames and Moore, San Francisco, Calif.; Engineering Mechanics Research Corporation, Troy, Michigan; Fluor Corporation, Los Angeles, Calif.; General Electric Company, San Jose, Calif.; Horza Engineering, Chicago, Illinois; International Harvester Company, Chicago, Illinois;

United States (continued)

Lockheed Missile and Space Company, Sunnyvale, Calif.; Martin and Associates, Los Angeles, Calif.; Philadelphia Gear Corporation, King of Prussia, Pennsylvania; Pregnoff/Matheu/Beebe, San Francisco, Calif.; Sargent and Lundy Engineers, Chicago, Illinois; Stone and Webster Engineering Corporation, Boston, Massachusetts; United Engineering, Philadelphia, Pennsylvania; U.S. Army Corps of Engineers - Waterways Experiment Station, Vicksburg, Mississippi; U.S. Army Corps of Engineers - Walla Walla District, Washington, D.C.; U.S. Department of the Interior, Bureau of Mines, Denver, Colorado; U.S. Naval Civil Engineering Laboratory, Port Hueneme, Calif.; Westinghouse Electric Corporation, Pittsburgh, Pennsylvania; Woodward-McNeill and Associates, Orange, Calif.; Yee and Associates, Honolulu, Hawaii.

TABLE OF CONTENTS

TABLE OF CONTENTS (Cont.)

- PART B -

SAMPLE ANALYSES

TABLE OF CONTENTS (Cont.)

- PART C -

APPENDICES

TABLE OF CONTENTS (Cont.)

- PART A -

DESCRIPTION OF SAP IV

1. INTRODUCTION

The development of an effective computer program for structural analysis requires a knowledge of three scientific disciplines -- structural mechanics, numerical analysis and computer application. The development of accurate and efficient structural elements requires a modern background in structural mechanics. The efficiency of a program depends largely on the numerical techniques employed and on their effective computer implementation. With regard to programming techniques, an optimum allocation of high and low speed storage is necessary.

A most important aspect of a general purpose computer program is, however, the ease with which it can be modified, extended and updated; otherwise, it may very well be that the program is obsolete within a few years after completion. This is because new structural elements are developed, better numerical procedures are available, or new computer equipment which requires new coding techniques is produced.

The structural analysis program SAP was designed to be modified and extended by the user. Additional options and new elements may easily be added. The program has the capacity to analyze very large three-dimensional systems; however, there is no loss in efficiency in the solution of smaller problems. Also, from the complete program, smaller special purpose programs can easily be assembled by simply using only those subroutines which are actually needed in the execution. This makes the program particularly usable on small size computers.

The current program version SAP IV for the static and dynamic analysis of linear structural systems is the result of several years' research and development experience. The program has proven to be a very flexible and efficient analysis tool. The program is coded in FORTRAN IV and operates without modifications on the CDC 6400, 6600 and 7600 computers. The first version of program SAP was published in September 1970 [28]. An improved static analysis program, namely SOLID SAP, or SAP II, was presented in 1971 [29]. Work was then started on a new static and dynamic analysis program. The program SAP III for static and dynamic analysis was released towards the end of 1972, but only to those agencies which supported our research. In relation to SAP III, the current version SAP IV has improvements throughout, and in particular has available a new variable-number-nodes thick shell and three-dimensional element, and out-of-core direct integration for time history analysis.

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

(a) three-dimensional truss element,

(b) three-dimensional beam element,

(c) plane stress and plane strain element,

(d) two-dimensional axisymmetric solid,

(e) three-dimensional solid,

(f) variable-number-nodes thick shell and three-dimensional element,

(g) thin plate or thin shell element,

(h) boundary element,

(i) pipe element (tangent and bend).

These structural elements can be used in a static or dynamic analysis. The capacity of the program depends mainly on the total number of nodal points in the system, the number of eigenvalues needed in the dynamic analysis and the computer used. There is practically no restriction on the number of elements used, the number of load cases or the order and bandwidth of the stiffness matrix. Each nodal point in the system can have from zero to six displacement degrees of freedom. The element stiffness and mass matrices are assembled in condensed form; therefore, the program is equally efficient in the analysis of one-, two- or three-dimensional systems.

The formation of the structure matrices is carried out in the same way in a static or dynamic analysis. The static analysis is continued by solving the equations of equilibrium followed by the computation of element stresses. In a dynamic analysis the choice is between

1. frequency calculations only,
2. frequency calculations followed by response history analysis,
3. frequency calculations followed by response spectrum analysis,
4. response history analysis by direct integration.

To obtain the frequencies and vibration mode shapes solution routines are used which calculate the required eigenvalues and eigenvectors directly without a transformation of the structure stiffness matrix and mass matrix to a reduced form. In the direct integration an unconditionally stable integration scheme is used, which also operates on the original structure stiffness matrix and mass matrix. This way the program operation and necessary input data for a dynamic analysis is a simple addition to what is needed for a static analysis.

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 and typical running times are given. In the presentation, emphasis is directed to the practical aspects of the program. For information on the development of the structural elements and the numerical techniques used the reader is referred to appropriate references.

2. THE EQUILIBRIUM EQUATIONS FOR COMPLEX STRUCTURAL SYSTEMS

2.1 Element to Structure Matrices

The nodal point equilibrium equations for a linear system of structural elements can be derived by several different approaches [1] [2] [9] [15] [23] [34]. All methods yield a set of linear equations of the following form

$$M\ddot{u} + C\dot{u} + Ku = R \tag{1}$$

where M is the mass matrix, C is the damping matrix and K is the stiffness matrix of the element assemblage; the vectors u, $\dot{u}$, $\ddot{u}$ and R are the nodal displacements, velocities, accelerations and generalized loads, respectively. The structure matrices are formed by direct addition of the element matrices; for example

$$K = \sum K_m \tag{2}$$

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 which pertain to the element degrees of freedom are nonzero. The addition of the element matrices can therefore be performed by using the element matrices in compact form together with identification arrays which relate element to structure degrees of freedom. The algorithm used in the program is described in Section 3.3.

In the program the structure stiffness matrix and a diagonal mass matrix are assembled. Therefore, a lumped mass analysis is assumed, where the structure mass is the sum of the individual element mass matrices plus additional concentrated masses which are specified at

selected degrees of freedom. The damping is assumed to be proportional and is specified in form of a modal damping factor. The assumptions used in lumped mass analyses and in the use of proportional damping have been discussed at various occasions [9] [11] [17] [33].

2.2 Boundary Conditions

If a displacement component is zero, the corresponding equation is not retained in the structure equilibrium equations, Eq. (1), 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

$$k u_i = k x \tag{3}$$

is added into Eq. (1), where $k \gg k_{ii}$. Therefore, the solution of Eq. (1) 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.

3. PROGRAM ORGANIZATION FOR CALCULATION OF THE STRUCTURE STIFFNESS MATRIX AND MASS MATRIX

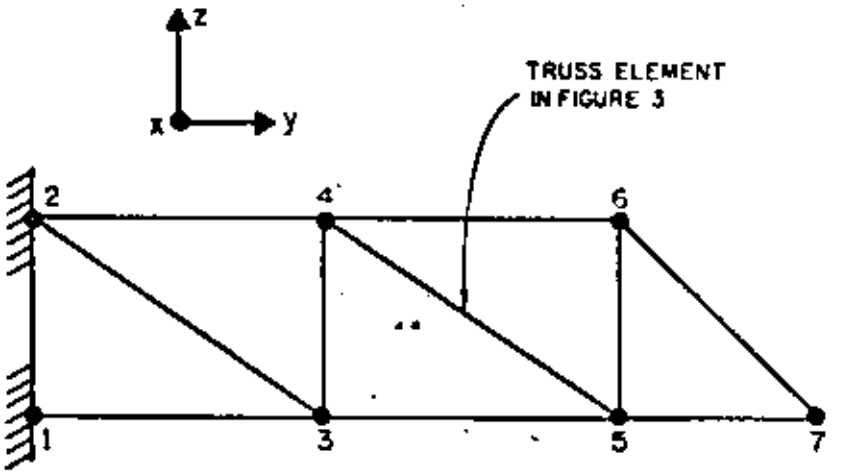
The calculation of the structure stiffness matrix and mass matrix is accomplished in three distinct phases:

1. The nodal point input data is read and generated by the program. In this phase the equation numbers for the active degrees of freedom at each nodal point are established.
2. The element stiffness and mass matrices are calculated together with their connection arrays; the arrays are stored in sequence on tape (or other low-speed storage).
3. The structure stiffness matrix and mass matrix are formed by addition of the element matrices and stored in block form on tape.

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.

3.1 Nodal Point Input Data and Degrees of Freedom

The capacity of the program is controlled by the number of nodal points of the structural system. For each nodal point six boundary condition codes (stored in the array ID), three coordinates (stored in the arrays X,Y,Z) and the nodal point temperatures (stored in the array T) are required (generation capability is provided). All nodal point data is retained in high speed storage during the formation of the element stiffness and mass matrices. Since the required high speed storage for the element subroutines is relatively small, the minimum required storage for a given problem is a little larger than ten times the



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

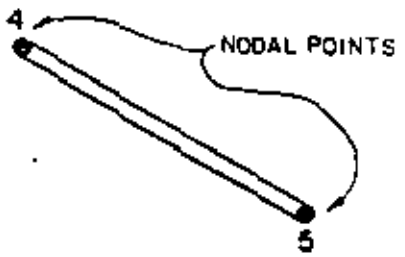
DEGREES OF FREEDOM

NODAL POINT NUMBERS

FIGURE 1: 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 2: 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 3: CONNECTION ARRAY (VECTOR LM) FOR A TYPICAL ELEMENT OF THE TRUSS-EXAMPLE

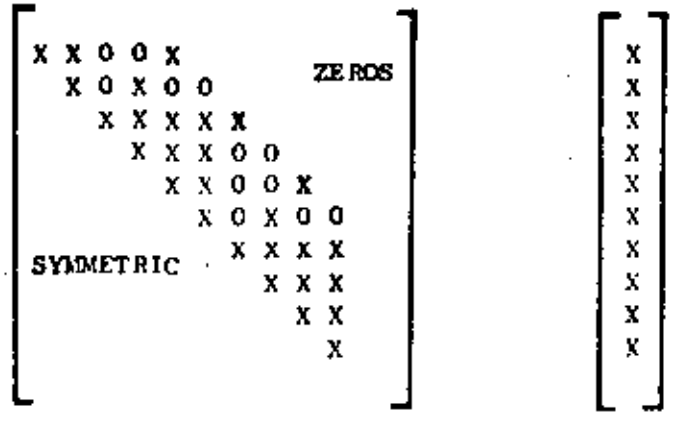
from the ID matrix and the specified structure nodal points pertaining to the element. The connection array for a typical element of the truss element is shown in Fig. 3.

The element matrices are calculated in groups, i.e., always all elements in one group together, thus calling the corresponding element subroutine only once for each element group. After all element matrices have been established, the ID and X,Y,Z arrays are not needed any more, and the corresponding storage area is used for the formation of the structure matrices and later for the solution of the equations of equilibrium.

3.3 Formation of Structure Stiffness and Mass

The stiffness matrix and mass matrix of the structure are formed in blocks, as shown in Fig. 4 for the truss-example. The number of equations per block depends on the available high speed storage and is calculated in the program as indicated in Fig. 5. It is noted that on reasonable size computers very large systems can be analyzed for static and dynamic response. With the number of equations per block known, the stiffness and mass matrix are assembled two blocks at a time by direct addition of the element matrices. In this process it is necessary to pass through the element matrices which are stored on tape. In order to minimize tape reading, in each pass element matrices which pertain to the next several blocks are written on another tape. This way the tape reading necessary for the formation of these blocks is reduced significantly.

A flow diagram of the program organization for the calculation of the structure stiffness matrix and mass matrix is shown in Fig. 6.



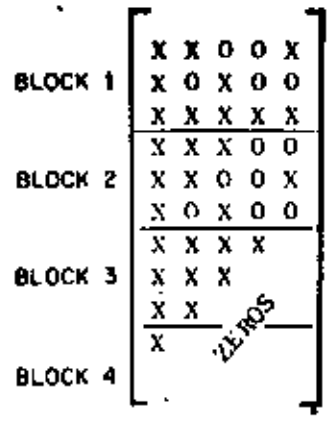
STIFFNESS MATRIX

MASS MATRIX

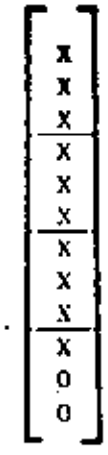
ACTUAL STRUCTURE MATRICES

0 = ZERO ELEMENT

X = NONZERO ELEMENT



STIFFNESS MATRIX



MASS MATRIX

BLOCK STORAGE OF STRUCTURE MATRICES

FIGURE 4: STORAGE OF STIFFNESS MATRIX AND MASS MATRIX ON TAPE

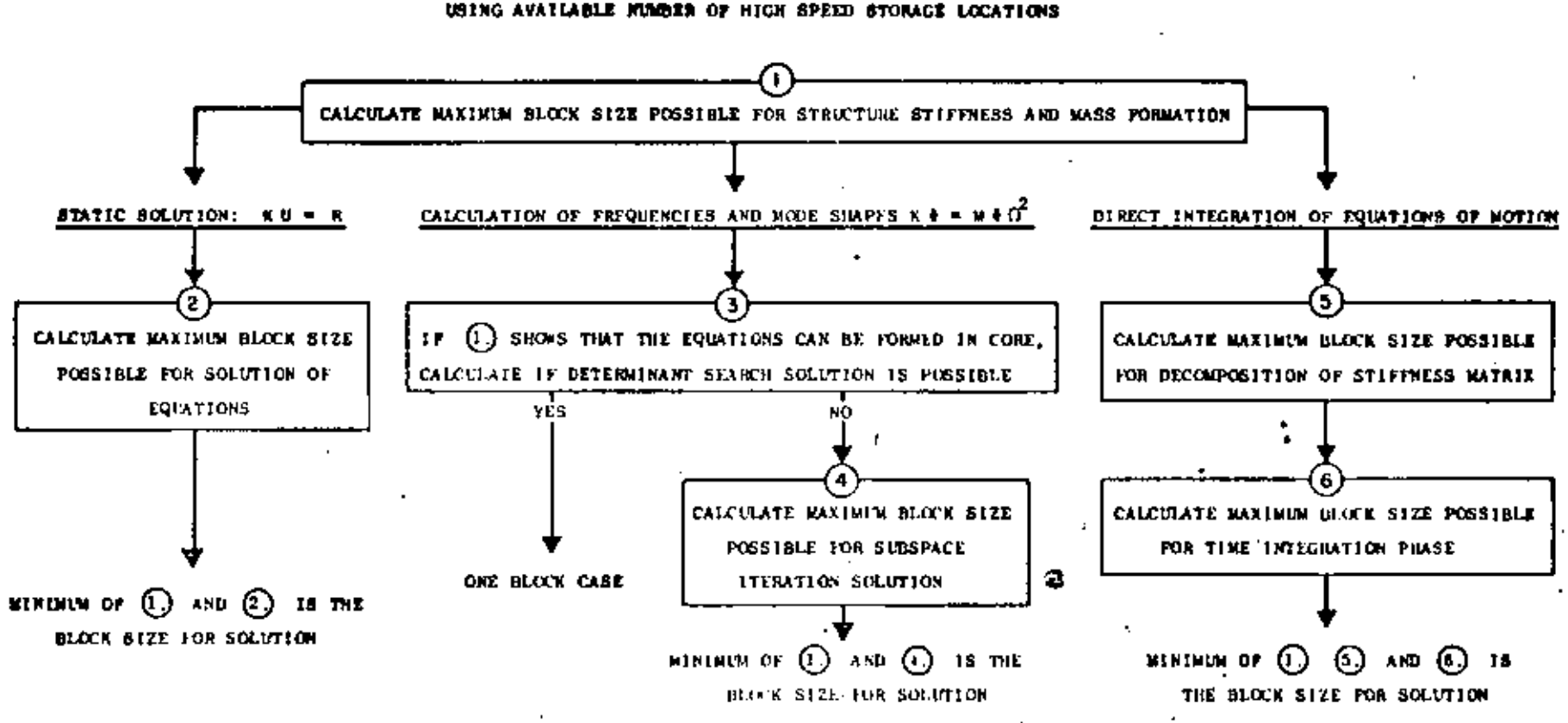


FIGURE 5: FLOWCHART SHOWING CALCULATION OF NUMBER OF EQUATIONS IN A BLOCK

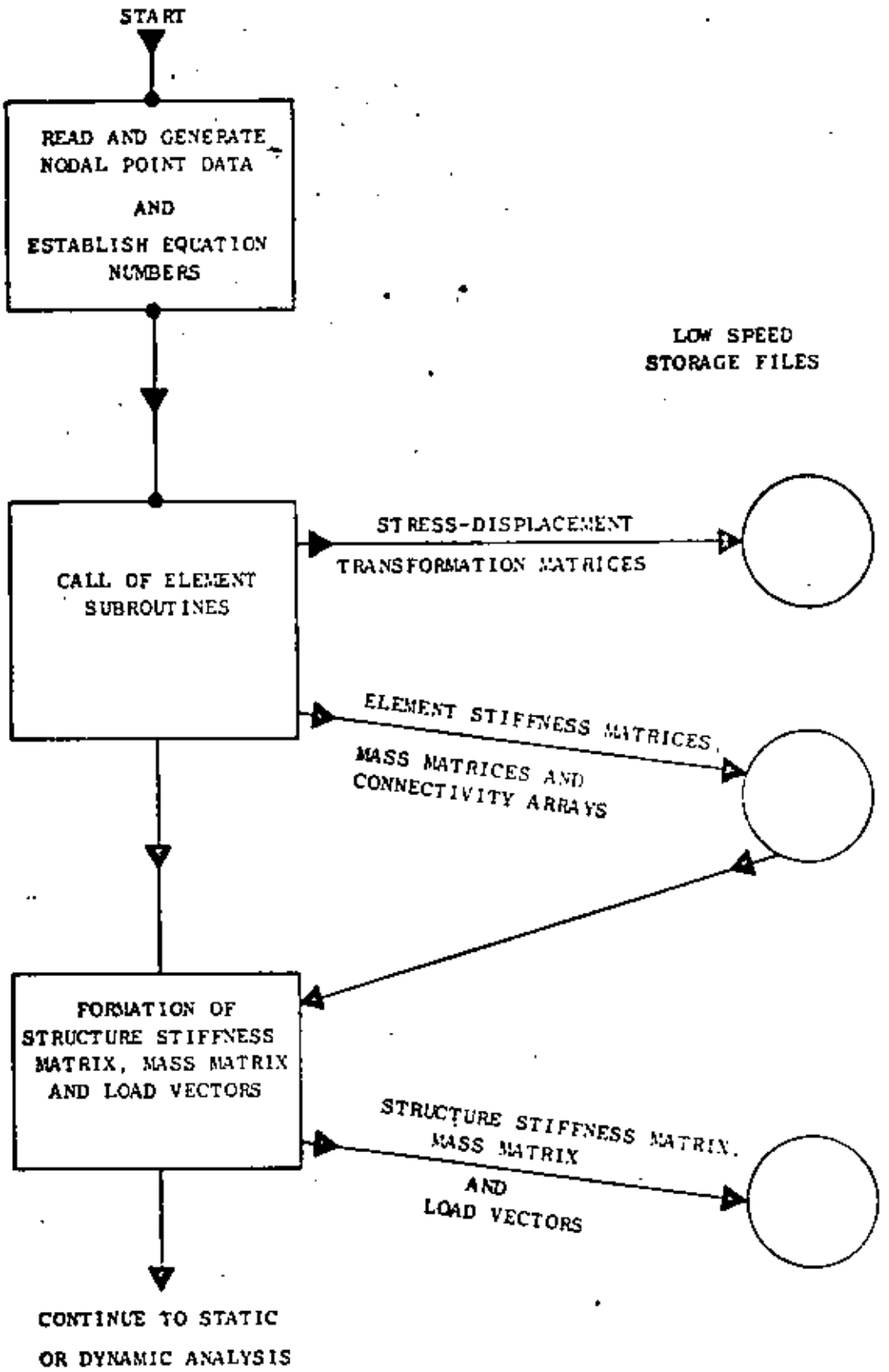


FIGURE 6: FLOWCHART FOR CALCULATION OF STRUCTURE STIFFNESS MATRIX AND MASS MATRIX

With the matrices stored in block form on tape either a static or a dynamic analysis can now be carried out. 15

4. THE ELEMENT LIBRARY

16

The element library of SAP IV consists of eight different element types. These elements can be used in either a static or dynamic analysis. They are shown in Fig. 7 and are briefly described below.

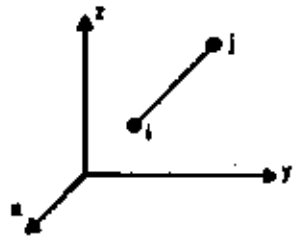
4.1 Three Dimensional Truss Element

The derivation of the truss element stiffness is given in Refs. [23] [29]. The element can be subjected to a uniform temperature change.

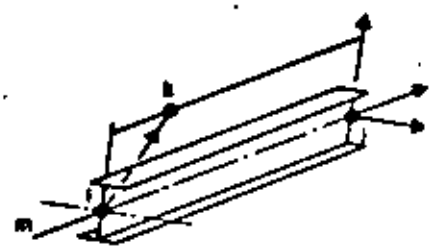
4.2 Three-Dimensional Beam Element

The beam element included in the program considers torsion, bending about two axes, axial and shearing deformations. The element is prismatic. The development of its stiffness properties is standard and is given in Ref. [23]. Inertia loading in three directions and specified fixed-end-forces form the element load cases. Forces (axial and shear) and moments (bending and torsion) are calculated in the beam local co-ordinate system.

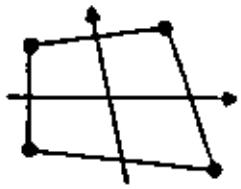
A typical beam element is shown in Fig. 7b. A plane which defines the principal bending axis of the beam is specified by the plane i, j, k. Only the geometry of nodal point k is needed; therefore, no additional degrees of freedom for nodal point k are used in the computer program. A unique option of the beam member is that the ends of the beam can be geometrically constrained to a master node. Slave degrees of freedom at the end of the beam are eliminated from the formulation and replaced by the transformed degrees of freedom of the master node [18] [29]. This technique reduces the total number of joint equilibrium

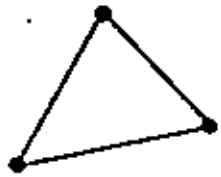


a. TRUSS ELEMENT

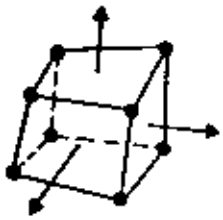


b. THREE-DIMENSIONAL BEAM ELEMENT

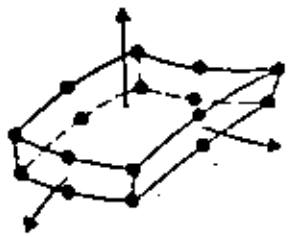




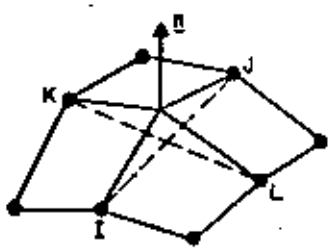
c. PLANE STRESS, PLANE STRAIN AND AXISYMMETRIC ELEMENTS

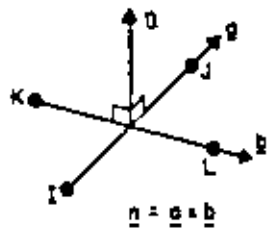


d. THREE-DIMENSIONAL SOLID

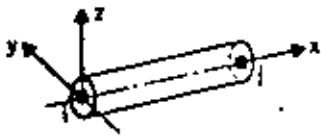


e. VARIABLE-NUMBER-NODES THICK SHELL AND THREE-DIMENSIONAL ELEMENT

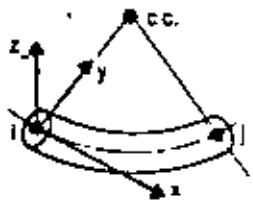




f. THIN SHELL AND BOUNDARY ELEMENT



TANGENT



BEND

g. PIPE ELEMENT

FIGURE 7: ELEMENT LIBRARY OF SAP IV

equations in the system (while possibly increasing the bandwidth) and greatly reduces the possibility of numerical sensitivities in many types of structures. Also, the method can be used to specify rigid floor diaphragms in building analysis.

4.3 Plane Stress, Plane Strain and Axisymmetric Elements

A plane stress quadrilateral (or triangular) element with orthotropic material properties is available. Each plane stress element may be of different thickness and may be located in an arbitrary plane with respect to the three-dimensional coordinate system. The plane strain and axisymmetric elements are restricted to the y-z plane. Gravity, inertia and temperature loadings may be considered. Stresses may be computed at the center of the element and at the center of each side. The element is based on an isoparametric formulation [19] [34]. Incompatible displacement modes can be included in order to improve the bending properties of the element [26] [29] [32].

4.4 Three-Dimensional Solid Element

A general eight nodal point "brick" element, with three translational degrees of freedom per nodal point can be used, Fig. 7d. Isotropic material properties are assumed and element loading consists of temperature, surface pressure and inertia loads in three directions. Stresses (six components) may be computed at the center of the element and at the center of each face. The element employs incompatible modes, which can be very effective if rectangular elements are used [26].

4.5 Variable-Number-Nodes Thick Shell and Three-Dimensional Element

A general three-dimensional isoparametric or subparametric element which may have from 8 to 21 nodes can be used for three-dimensional

or thick shell analysis, Fig. 7e [7] [8]. General orthotropic material properties can be assigned to the element. The loading may consist of applied surface pressure, hydrostatic loads, inertia loads in three directions, and thermal loads. Six global stresses are output at up to seven locations within an element.

4.6 Thin Plate and Shell Element

The thin shell element available in the program is a quadrilateral of arbitrary geometry formed from four compatible triangles. The bending and plane stress properties of the element are described in references [12] [14]. The shell element uses the constant strain triangle and the LCCT9 element to represent the membrane and bending behavior, respectively. The central node is located at the average of the coordinates of the four corner nodes. The element has six interior degrees of freedom which are eliminated at the element level prior to assembly; therefore, the resulting quadrilateral element has twenty-four degrees of freedom, i.e., six degrees of freedom per node in the global coordinate system.

In the analysis of flat plates the stiffness associated with the rotation normal to the shell surface is not defined; therefore, the rotation normal degree of freedom must not be included in the analysis. For curved shells, the normal rotation need be included as an extra degree of freedom. In case the curvature is very small, the degree

of freedom should be restrained by the addition of a "Boundary Element" with a small normal rotational stiffness, say of less or about 10% of the element bending stiffness [13] [34].

4.7 Boundary Element

The boundary element, shown in Fig. 7f, can be used for the following:

1. in the idealization of an external elastic support at a node;
2. in the idealization of an inclined roller support;
3. to specify a displacement, or
4. to eliminate the numerical difficulty associated with the 'sixth' degree of freedom in the analysis of nearly flat shells.

The element is one-dimensional with an axial or torsional stiffness. The element stiffness coefficients are added directly to the total stiffness matrix (see Section 2.2).

4.8 Pipe Element

The pipe element (Fig. 7g) can represent a straight segment (tangent) or a circularly curved segment (bend); both elements require a uniform section and uniform material properties. Elements can be directed arbitrarily in space. The member stiffness matrices account for bending, torsional, axial and shearing deformations. In addition, the effect of internal pressure on the stiffness of curved pipe elements is considered.

The types of structure loads contributed by the pipe elements include gravity loading in the global directions and loads due to thermal distortions and deformations induced by internal pressure. Forces and moments

acting at the member ends (i,j) and at the center of each bend are calculated in coordinate systems aligned with the member's cross section.

The pipe element stiffness matrix is formed by first evaluating the flexibility matrix corresponding to the six degrees of freedom at end j as given by Poley [22]. With the corresponding stiffness matrix, the equilibrium transformations outlined by Hall et al [16] are used to form the complete element stiffness matrix. Distortions due to element loads are premultiplied by the stiffness matrix to compute restrained nodal forces due to thermal, pressure or gravity loads.

5. STATIC ANALYSIS

A static analysis involves the solution of the equilibrium equations

$$\mathbf{K}\,\mathbf{u} = \mathbf{R} \tag{4}$$

followed by the calculation of element stresses.

5.1 Solution of Equilibrium Equations

The load vectors R have been assembled at the same time as the structure stiffness matrix and mass matrix were formed. The solution of the equations is obtained using the large capacity linear equation solver SESOL [31]. This subroutine uses Gauss elimination on the positive-definite symmetrical system of equations. The algorithm performs a minimum number of operations; i.e. there are no operations with zero elements. In the program, the $\mathbf{L}^T\mathbf{D}\mathbf{L}$ decomposition of K is used, hence Eq. (4) can be written as

$$\mathbf{L}^T\mathbf{v} = \mathbf{R} \tag{5}$$

and

$$\mathbf{v} = \mathbf{D}\mathbf{L}\mathbf{u} \tag{6}$$

where the solution for v in Eq. (5) is obtained by a reduction of the load vectors; the displacement vectors u are then calculated by a back-substitution.

In the solution, the load vectors are reduced at the same time as K is decomposed. In all operations it is necessary to have at any one time the required matrix elements in high-speed storage. In the

reduction, two blocks are in high speed storage (as was also the case in the formation of the stiffness matrix and mass matrix), i.e., the "leading" block, which finally stores the elements of L and D, and in succession those blocks which are affected by the decomposition of the "leading" block. Table 1 gives some typical solution times.

5.2 Evaluation of Element Stresses

After the nodal point displacements have been evaluated, sequentually the element stress-displacement matrices are read from low speed storage and the element stresses are calculated.

TABLE 1 SOLUTION OF EQUATIONS USING SESOL

NUMBER OF EQUATIONS	HALF BANDWIDTH	CENTRAL PROCESSOR SEC	COMPUTER USED
8036	344	1786†	CDC 6600
2696	488	1200	CDC 6600
4214	205	31	CDC 7600

† The inner DO - loop in the factorization of the stiffness matrix has been coded in machine language for this solution.

6. CALCULATION OF FREQUENCIES AND MODE SHAPES

The dynamic analysis of a structural system using mode superposition requires as the first step the solution of the generalized eigenvalue problem

$$K\phi = \omega^2 M\phi \tag{7}$$

where ω and ϕ are free vibration frequency and mode shape, respectively. As was described in Section 3.3 the program stores the stiffness and mass matrix in blocks on tape, Fig. 4. The mass matrix is diagonal with partly zero diagonal elements. The program assumes that only the lowest p eigenvalues and corresponding eigenvectors are needed. The solution of Eq. (7) can therefore be written as

$$K\Phi = M\Phi\Omega^2 \tag{8}$$

where Ω^2 is a diagonal matrix with the p smallest eigenvalues, i.e. $\Omega^2 = \text{diag}(\omega_i^2)$, and Φ stores the corresponding M-orthonormalized eigenvectors $\phi_1, \phi_2, \ldots \phi_p$. Two different solution procedures are used in the program, a determinant search technique or a subspace iteration solution. The determinant search solution is carried out when the stiffness matrix can be contained in high-speed storage in one block. Therefore, for systems of large order and bandwidth the subspace iteration method is used. Both solution techniques solve the generalized eigenvalue problem directly without a transformation to the standard form [3].

6.1 The Determinant Search Solution

The determinant search technique is best suited for the analysis of large systems in which K and M have small bandwidths [4]. Basically, the solution 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 ω_1^2, ϕ_1. An efficient accelerated secant iteration procedure which operates on the characteristic polynomial

$$p(\omega^2) = \det(K - \omega^2 M) \tag{9}$$

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; the eigenvalue is obtained by adding the Rayleigh quotient correction to the shift value. Table 2 shows typical solution times.

6.2 The Subspace Iteration Solution

When the system is too large to be completely contained in high speed storage, i.e. more blocks than one are used, the subspace iteration solution is carried out. The iteration can be interpreted as a repeated application of the Ritz method [5] [9], in which the computed eigenvectors from one step are used as the trial basis vectors for the next iteration until convergence to the required p eigenvalues and

TABLE 2 CALCULATION OF FREQUENCIES AND MODE SHAPES USING DETERMINANT SEARCH METHOD

SYSTEM	SYSTEM ORDER n	MAXIMUM HALF BAND WIDTH	NUMBER OF REQ'D. FREQN. AND MODE SHAPES p	COMPUTER USED	CENTRAL PROCESSOR SEC
PLANE FRAME	297	30	3	CDC 6400	40
PIPING SYSTEM	566	12	7	CDC 6600	11
BUILDING	340	32	7	CDC 6600	20
CONTAINER	265	65	40	CDC 7600	58

eigenvectors is obtained.

The solution is carried out by iterating simultaneously with q linearly independent vectors, where $q > p$. In the k'th iteration the vectors span the q-dimensional subspace $\mathcal{E}_k$ and 'best' eigenvalue and eigenvector approximations are calculated; i.e. when the vectors span the p-dimensional least dominant subspace, the required eigenvalues and eigenvectors are obtained.

Let V_o store the starting vectors, then the k'th iteration is described as follows:

Solve for vectors $\bar{V}_k$ which span $\mathcal{E}_k$

$$K \bar{V}_k = M V_{k-1} \tag{10}$$

Calculate the projections of K and M onto $\mathcal{E}_k$ (i.e. the generalized stiffness matrix and mass matrix corresponding to $\mathcal{E}_k$)

$$K_k = \bar{V}_k^T K \bar{V}_k \tag{11}$$

$$M_k = \bar{V}_k^T M \bar{V}_k \tag{12}$$

Solve for the eigensystem of K_k and M_k

$$K_k Q_k = M_k Q_k \Omega_k^2 \tag{13}$$

and calculate the k'th improved approximation to the eigenvectors

$$V_k = \bar{V}_k Q_k \tag{14}$$

Provided that the starting subspace is not orthogonal to any of the required eigenvectors, the iteration converges to the desired result, i.e. $\Omega_k^2 \to \Lambda^2$ and $V_k \to \Phi$ as $k \to \infty$.

The number of vectors q used in the iteration is taken greater than the desired number of eigenvectors in order to accelerate the convergence of the process. The number of iterations required to achieve satisfactory convergence depends, of course, on the quality of the starting vectors V_o. Unless requested otherwise (see Section 6.3), the program generates q starting vectors where $q = \min(2p, p+8)$, which has proven to be effective in general applications. At convergence a Sturm sequence check can be requested to verify that the lowest p eigenvalues have been found.

Table 3 lists a few typical solution times using the program generated starting vectors.

6.3 Dynamic Optimization

The solution of the eigenvalue problem may be required when a good estimate of the required eigensystem is already known, such as in dynamic optimization. In this case the subspace iteration method is ideally suited for solution. The number of iteration vectors q and the vectors V_o together with the maximum number of iterations can in this case be specified by the user. Also, in case the number of eigenvalues and vectors required is increased, the already calculated eigenvectors can be specified as part of the starting iteration vectors in order to accelerate convergence.

TABLE 3 CALCULATION OF FREQUENCIES AND MODE SHAPES USING SUBSPACE ITERATION METHOD

SYSTEM	SYSTEM ORDER n	MAXIMUM HALF BAND WIDTH	NUMBER OF REQ'D. FREQN. AND MODE SHAPES p	COMPUTER USED	CENTRAL PROCESSOR SEC
PLANE FRAME	297	30	3	CDC 6400	25
PIPING SYSTEM	566	12	28	CDC 6600	142
BLDG. WITH FOUNDATION	1174	138	45	CDC 6600	890
3-DIM BLDG. FRAME	468	156	4	CDC 6400	160

7. DYNAMIC ANALYSES

In dynamic response analysis the solution of the equations

$$M\ddot{u} + C\dot{u} + Ku = R(t) \tag{15}$$

is required, where R(t) can be a vector of arbitrary time varying loads or of effective loads which result from ground motion. Specifically, in the case of ground motion, if it is assumed that the structure is uniformly subjected to the ground acceleration $\ddot{u}_g$ [9], the equilibrium equations considered are

$$M\ddot{u}_r + C\dot{u}_r + Ku_r = -M\ddot{u}_g \tag{16}$$

where u_r is the relative displacement of the structure with respect to the ground, i.e. $u_r = u - u_g$.

The program can carry out a history analysis for solution of Eqs. (15) or (16), or a response spectrum analysis for solution of Eq. (16). The history analysis can be carried out using mode superposition or direct integration. The response spectrum analysis necessitates, of course, first the solution of the required eigensystem.

7.1 Response History Analysis by Mode Superposition

In the mode superposition analysis, it is assumed that the structural response can be described adequately by the p lowest vibration modes, where $p \ll n$. Using the transformation $u = \phi X$, where the columns in ϕ are the p M-orthonormalized eigenvectors, Eq. (15) can be written as

$$\ddot{X} + \Delta\dot{X} + \Omega^2 X = \phi^T R \tag{17}$$

where

$$\Delta = \text{diag}(2\omega_i \xi_i); \qquad \Omega^2 = \text{diag}(\omega_i^2) \tag{18}$$

In Eq. (18) it is assumed that the damping matrix C satisfies the modal orthogonality condition

$$\phi_i^T C \phi_j = 0 \qquad (i \neq j) \tag{19}$$

Equation (17) therefore represents p uncoupled second order differential equations. These are solved in the program using the Wilson θ-method, which is an unconditionally stable step-by-step integration scheme [6]. The same time step is used in the integration of all equations to simplify the calculation of stress components at preselected times.

In the case of prescribed ground motion $u_r = \phi X$ and in Eq. (17) the right hand side is given by $-\phi^T M \ddot{u}_g$, where the ground acceleration is considered as the sum of the components in the x, y and z directions as described in Section 7.3.

7.2 Response History Analysis by Direct Integration

The solution of the equations of motion, Eqs. (15) and (16), can be obtained by direct integration [6]. In the program the Wilson θ-method is used, which is unconditionally stable. The algorithm employed is summarized in Table 4. It need be noted that Rayleigh damping is assumed, i.e. $C = \alpha M + \beta K$ [11]. This form of damping is easily taken account of in the analysis, because no storage and no multiplications for a damping matrix are required.

TABLE 4: STEP-BY-STEP DIRECT INTEGRATION ALGORITHM

Initial Calculations

1. Calculate the following constants (Assume $C = \alpha M + \beta K$).

$\theta = 1.4, \quad \tau = \theta\Delta t$

$a_0 = (6 + 3\alpha\tau)/(\tau^2 + 3\beta\tau)$

$b_0 = \alpha - \beta a_0$

$a_1 = 6/\tau^2 + 3b_0/\tau$

$a_2 = 6/\tau + 2b_0$

$a_3 = 2 + \tau b_0/2$

$a_4 = 6/[\theta(3\beta\tau + \tau^2)]$

$b_1 = \beta a_4$

$a_5 = 3b_1/\tau - 6/(\tau^2\theta)$

$a_6 = 2b_1 - 6/(\tau\theta)$

$a_7 = b_1\tau/2 + 1 - 3/\theta$

$a_8 = \Delta t/2$

$a_9 = \Delta t^2/3$

$a_{10} = \frac{1}{2} a_9$

2. Form effective stiffness matrix $K^* = K + a_0 M$.

3. Triangularize K^*

For Each Time Increment

1. Form effective load vector R_t^*

$$R_t^* = R_t + \theta(R_{t+\Delta t} - R_t) + M[a_1 u_t + a_2 \dot{u}_t + a_3 \ddot{u}_t]$$

2. Solve for effective displacement vector u_t^*

$$K^* u_t^* = R_t^*$$

3. Calculate new acceleration, velocity and displacement vectors,

$$\ddot{u}_{t+\Delta t} = a_4 u_t^* + a_5 u_t + a_6 \dot{u}_t + a_7 \ddot{u}_t$$

$$\dot{u}_{t+\Delta t} = \dot{u}_t + a_8(\ddot{u}_{t+\Delta t} + \ddot{u}_t)$$

$$u_{t+\Delta t} = u_t + \Delta t\, \dot{u}_t + a_9 \ddot{u}_t + a_{10} \ddot{u}_{t+\Delta t}$$

4. Calculate element stresses if desired.

7.3 Response Spectrum Analysis

In this analysis the ground acceleration vector in Eq. (16) is written as

$$\ddot{u}_g = \ddot{u}_{gx} + \ddot{u}_{gy} + \ddot{u}_{gz} \tag{20}$$

where $\ddot{u}_{gx}$, $\ddot{u}_{gy}$ and $\ddot{u}_{gz}$ are the ground accelerations in the x, y and z directions, respectively. The equation for the response in the r'th mode is therefore

$$\ddot{x}_r + 2\xi_r \omega_r \dot{x}_r + \omega_r^2 x_r = r_{rx} + r_{ry} + r_{rz} \tag{21}$$

where x_r is the r'th element in X and

$$r_{rx} = -\phi_r^T M \ddot{u}_{gx} \; ; \; r_{ry} = -\phi_r^T M \ddot{u}_{gy} \; ; \; r_{rz} = -\phi_r^T M \ddot{u}_{gz} \tag{22}$$

Using the definition of the spectral displacement [10], the maximum absolute modal displacements of the structure subjected to an acceleration into the x direction are

$$u_{rx}^{(max)} = \phi_r |\phi_r^T M I_x| S_x(\omega_r) \tag{23}$$

where $S_x(\omega_r)$ is the spectral displacement into the x direction corresponding to the frequency ω_r and I_x is a null vector except that those elements are equal to one which correspond to the x-translational degrees of freedom. Similarly, for the responses due to a ground acceleration into the y and z-directions

$$u_{ry}^{(max)} = \phi_r |\phi_r^T M I_y| S_y(\omega_r) \quad ; \quad u_{rz}^{(max)} = \phi_r |\phi_r^T M I_z| S_z(\omega_r) \tag{24}$$

and the total maximum response in the r'th mode is assumed to be

$$u_r^{(max)} = u_{rx}^{(max)} + u_{ry}^{(max)} + u_{rz}^{(max)} \quad (25)$$

Program SAP IV calculates the maximum responses in each of the p lowest modes, where the spectra (displacements or accelerations) into the x, y and z-directions are assumed to be proportional to each other. The total response for displacements and stress resultants is calculated as the square root of the sum of the squares of the modal maximum responses [10], [36].

7.4 Restart Capability in Mode Superposition Analysis

The most expensive phase in mode-superposition analysis is usually the calculation of frequencies and mode shapes. However, once the required eigensystem has been solved for, it can be used to analyze the structure for different loading conditions. Also, in a design process the history or spectrum analysis for the same loading can be carried out economically a few times, for example, to study the stress history in different parts of the structure.

In the program, at completion of the eigensystem solution, all variables required for a response history or response spectrum analysis together with the frequencies and mode shapes are written on low speed storage. The program execution may be stopped at this stage and the information on low speed storage be copied to a physical tape. Later, this tape would be copied back to low speed storage before starting a response analysis. If, after a number of response analyses using the eigensystem on the tape, it is decided that more frequencies and mode shapes need be calculated, the information on the tape can be used to

reduce the cost of the new eigensystem solution as described in Section 6.3.

7.5 Mode Superposition Versus Direct Integration

For an effective response history analysis the user must decide appropriately whether to use mode superposition or direct integration. It should be realized that the direct integration is equivalent to a mode superposition analysis in which all the eigenvalues and vectors have been calculated and the uncoupled equations in Eq. (17) with $p = n$ are integrated with a common time step Δt. Naturally, the integration can only be accurate for those modes for which Δt is smaller than a certain fraction of the period T. Using the Wilson θ-algorithm the integration errors effectively "filter" the high mode response, for which $\Delta t/T$ is large, out of the solution. This filtering is due to the amplitude decay observed in the numerical solution when $\Delta t/T$ is large. As an example, Fig. 8 shows the amplitude decay for the initial value problem indicated [6].

The effective filtering of the high frequency response from the solution may be beneficial. Integration accuracy cannot be obtained in the response of the modes for which $\Delta t/T$ is large and the filtering process allows one to obtain a total system solution in which the low mode response is accurately observed.

It is therefore noted that the direct integration is quite equivalent to a mode superposition analysis, in which only the lowest modes of the system, but a sufficient number to take proper account of the applied loading, are considered. The exact number of modes effectively included in the analysis depends on the time step size Δt and the distribution of the periods.

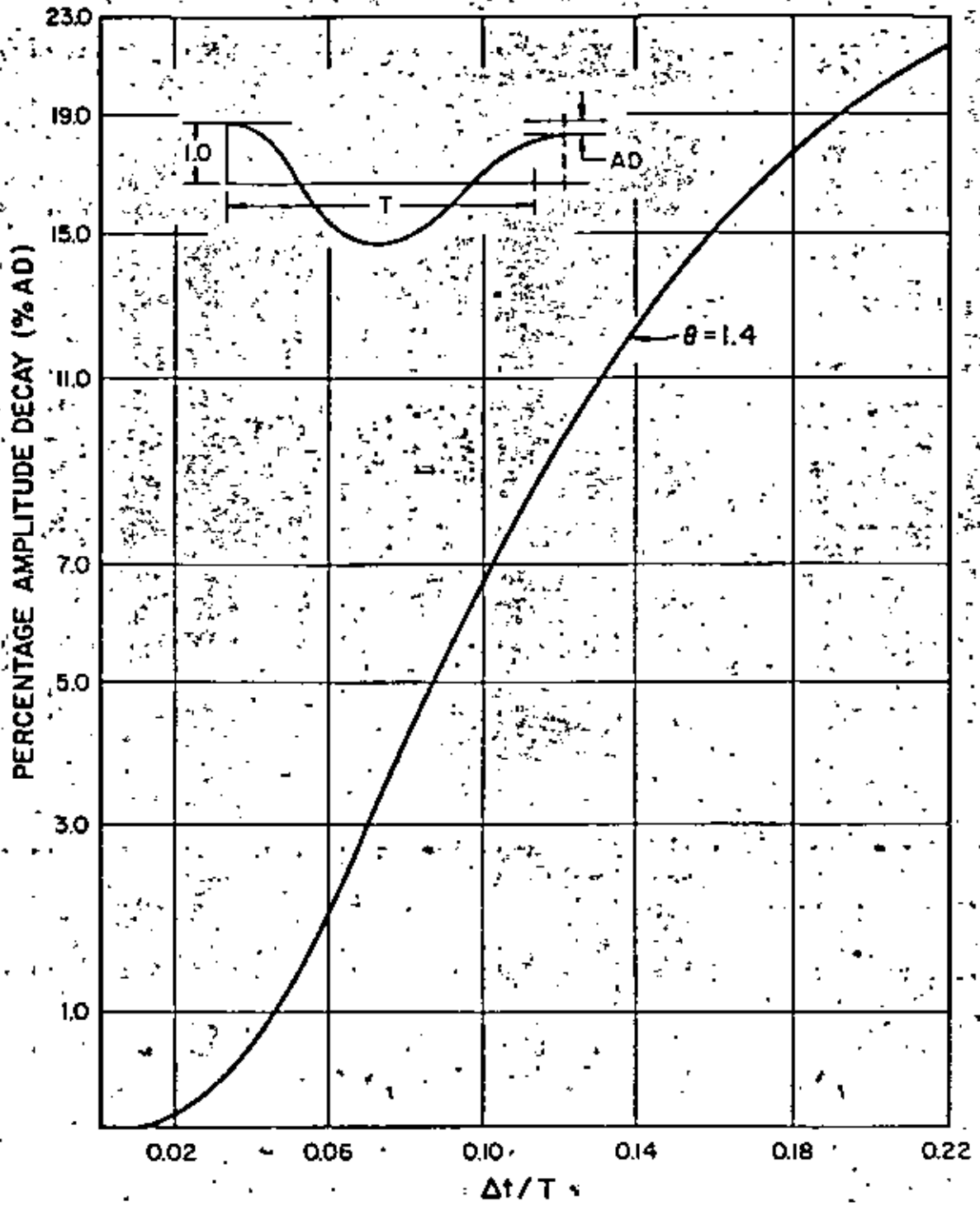


FIGURE 8 : AMPLITUDE DECAY WILSON θ - METHOD

The advantages of mode superposition are essentially that frequencies and mode shapes are obtained and that a variety of response history and response spectrum analyses can be carried out with relatively small additional cost. Also, if the structure is slightly changed or more eigenvalues and vectors are required, i.e., the frequency domain to be considered shall be extended, the eigensystem solved for already can be used to reduce the cost of the new eigensystem solution (see Section 7.4).

The direct step-by-step integration, however, is more effective, when many modes need be included in the analysis and the response is required over relatively few time steps, such as in shock problems. It should be noted that the tape reading required in the direct integration analysis of large out-of-core systems can be costly because in the solution for the response in each time step the triangularized effective stiffness matrix must be taken into high speed storage.

8. 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 and generates all data, prints it and also writes the full data on low speed storage. At completion of data read and generation the information on low speed storage can be copied to a physical tape. This tape may then be used to plot the finite element mesh.

9. INSTALLATION OF SAP IV ON A SYSTEM OTHER THAN A CDC COMPUTER

SAP IV is written using FORTRAN IV and has been developed on a CDC computer. The program has also been installed with relatively little effort on IBM and UNIVAC machines.

The program or parts of it can essentially be used on any reasonably sized computer. SAP IV consists of about 14000 cards, and is organized in a standard Fortran overlay structure to reduce the required high speed storage for program execution. The main overlay essentially consists of the main program. The secondary overlays are, respectively, the element routines, the equation solver, the eigenvalue routines, the mode superposition history analysis program, the spectrum analysis program and the direct integration routine. Using only specific overlays efficient special purpose programs are obtained. For example, using the main overlay plus the secondary overlays of the pipe element, the eigenvalue routines and the response history analysis a special purpose pipe response history analysis program by mode superposition is obtained. On the CDC 6400 of the University of California, Berkeley, the complete program with 12000_{10} high speed storage locations allocated for solution processing, i.e. the blank common block A has a length of 12000, requires a field length of about 114000_8 for execution.

On installation of SAP IV on other machines than the CDC series, it must be observed that arithmetic calculations should 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 in static and dynamic analysis the formation of element stiffness matrices, the formation of the structure stiffness

matrix and main steps in the solution of the equations of motion, namely, the solution of $Ku = R$, the solution of the generalized eigenvalue problem $K\phi = \omega^2 M\phi$ and in the direct integration the solution of the effective displacements u_t^* (see Table 4). These calculations need primarily be performed in double precision because of truncation errors occurring when too few digits are used, which can cause large errors in the solution and numerical instabilities [20] [25].

With regard to the use of back-up storage, to keep the program system independent sequential accessing is used throughout. Therefore, since no advantage is taken of efficient buffering and direct access techniques, it need be noted that the use of secondary storage can be much improved when tailored to a specific system.

- PART B -

SAMPLE ANALYSES

SAMPLE ANALYSES

In this part of the report brief problem descriptions for a set of standard data cases available with program SAP IV are given. Naturally, the few sample analyses can only demonstrate to a small degree the capabilities of the program. In general, detailed problem descriptions can be found in the references from which the sample analyses have been taken.

1. Static Analysis of Pipe Network

The pipe network shown in Fig. 9 corresponds to a sample problem solution presented in the User's Manual for the "ADLPIPE" piping analysis computer code [35]. The purpose of this analysis is to predict the static response of the system under the combined effects of:

(1) concentrated loads

(2) vertical (y-direction) gravity loads

(3) uniform temperature increase

(4) non-zero displacements imposed at one support point

Table 5 compares the reactions printed in the SAP and ADLPIPE solutions. The two solutions are in fair agreement; the SAP results satisfy equilibrium to all six digits appearing in the printed output. In the table of applied loads, a total weight of 6284.03 lbs results from 950.686 inches of pipe weighing 6.61 lbs per inch.

2. Static Shell Analysis

The clamped spherical shell shown in Fig. 10 is analyzed for stresses produced by a uniform pressure applied on its outside surface. The SAP model represents a five degree wedge of the shell with eighteen

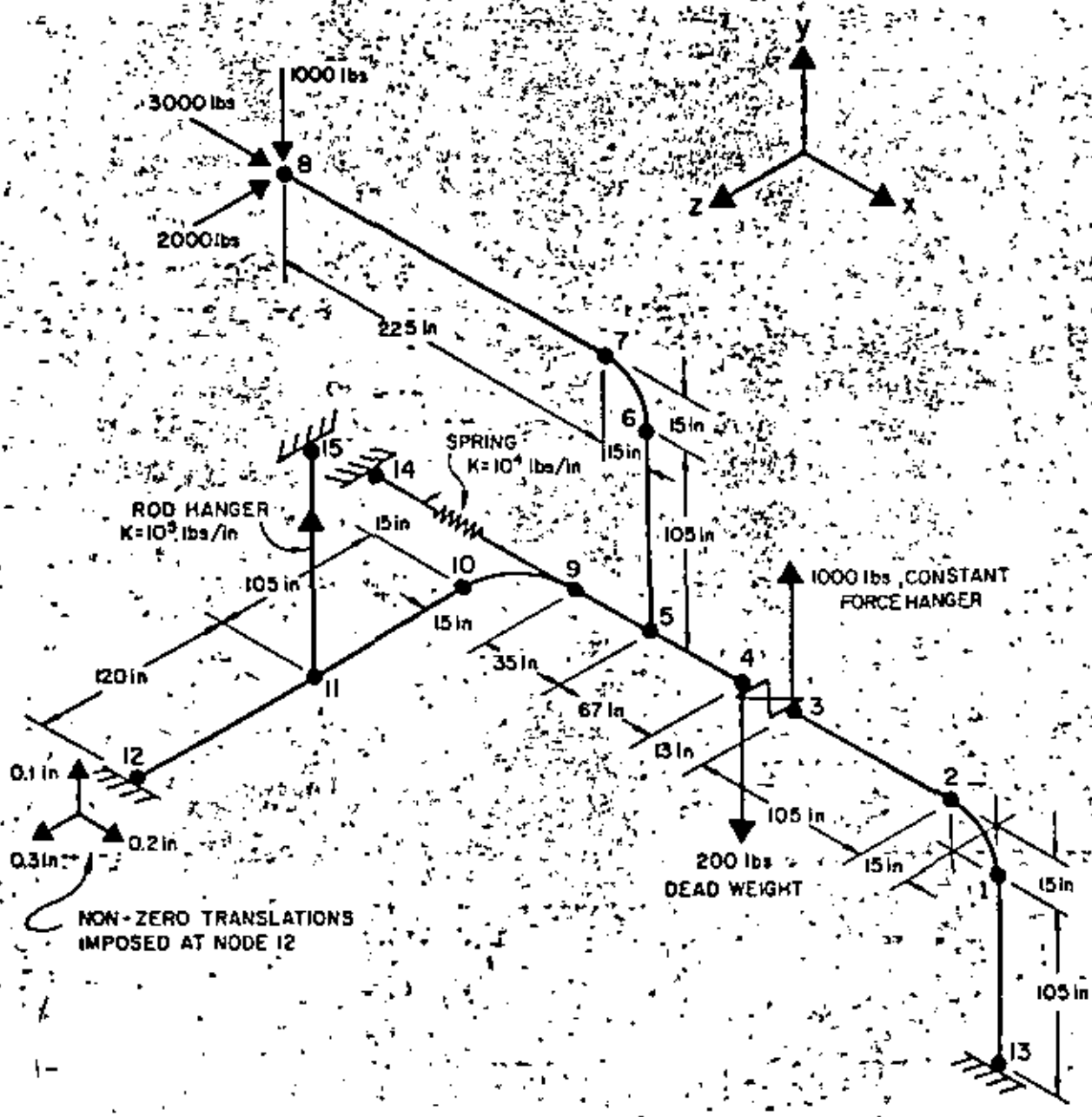


FIGURE 9: SAP MODEL OF PIPE NETWORK GIVEN IN ADLPIPE MANUAL

TABLE 5 FORCE EQUILIBRIUM SUMMARY
(SAP ANALYSIS OF ADLPIPE EXAMPLE 1)

A. REACTIONS

NODE	SAP			ADLPIPE		
	FX	FY	FZ	FX	FY	FZ
9	5643.51	.	.	5659.	.	.
11	.	-4044.59	.	.	-4052.	.
12	2350.08	4023.01	-4960.70	2361.	4026.	-4966.
13	-10993.59	4505.61	2960.70	-11021.	4509.	2966.
TOTAL	-3000.00	4484.03	-2000.00	-3001.	4483.	-2000.

B. APPLIED LOADS

LOADING TYPE	DIRECTION		
	X	Y	Z
CONCENTRATED:			
at node 3	.	1000.00	.
at node 4	.	-200.00	.
at node 8	3000.	1000.00	2000.
DISTRIBUTED WEIGHT:		-6284.03	
TOTAL	3000.	-4484.03	2000.

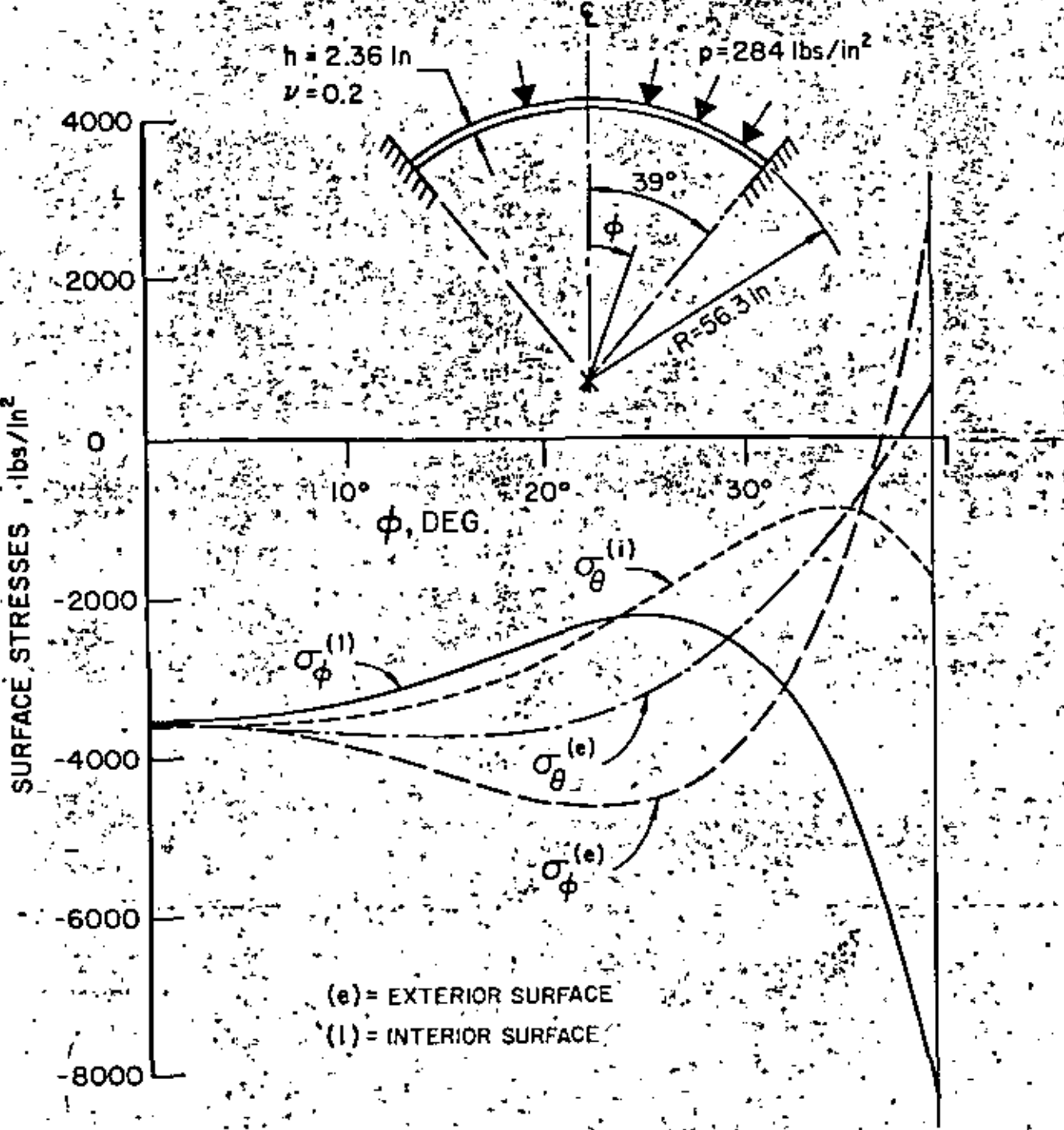


FIGURE 10: DISTRIBUTION OF SURFACE STRESSES IN A CLAMPED SPHERICAL SHELL UNDER EXTERNAL PRESSURE

thin shell elements along the thirty-nine degree meridian. The curves drawn in Fig. 10 are plots of meridian (ϕ) and circumferential (θ) direction surface stresses predicted by the SAP program at the element centroids.

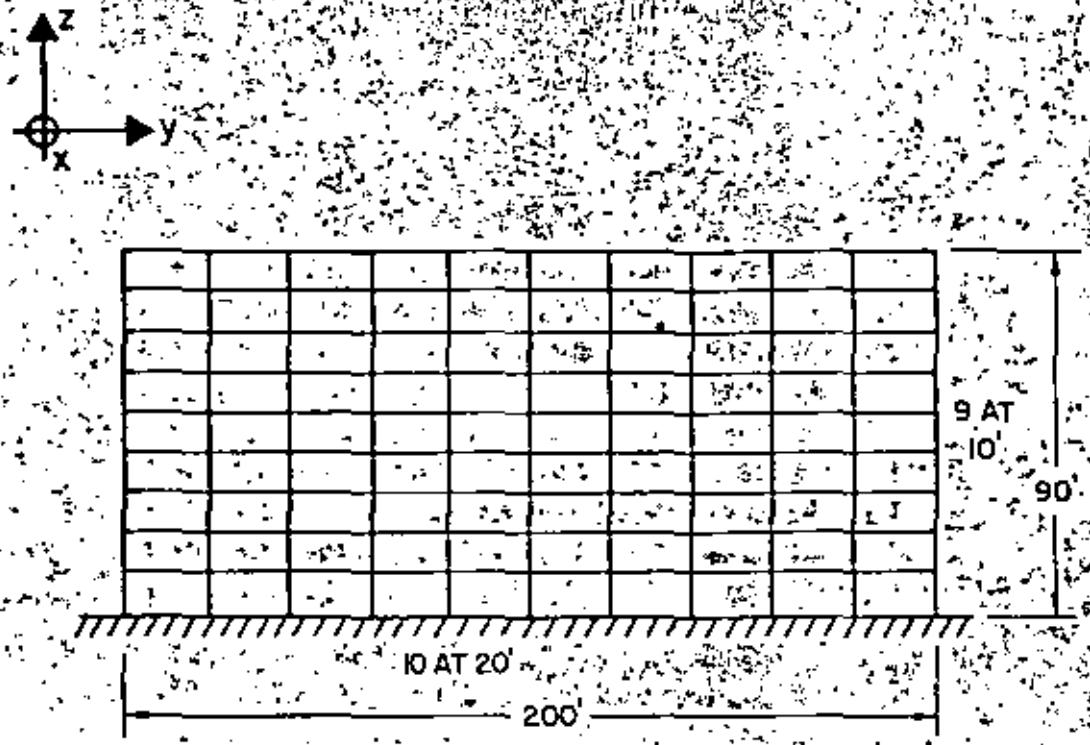
The solution of this problem is given in the text by Timoshenko [27], where the stress distribution of Fig. 10 may be found for comparison. It should be noted that program SAP calculates membrane stresses (force per unit area) and bending resultants (moment per unit length) from which the surface stresses in the figure have been evaluated.

3. Frequency and Mode Shape Analysis of Plane Frame

The lowest three frequencies and corresponding mode shapes of the plane frame shown in Fig. 11 are calculated. The results can be compared with the solutions published in references [4] [5]. Note that depending on the high speed storage available either a determinant search or a subspace iteration solution may be performed. The three lowest vibration periods of the frame are given in Table 6.

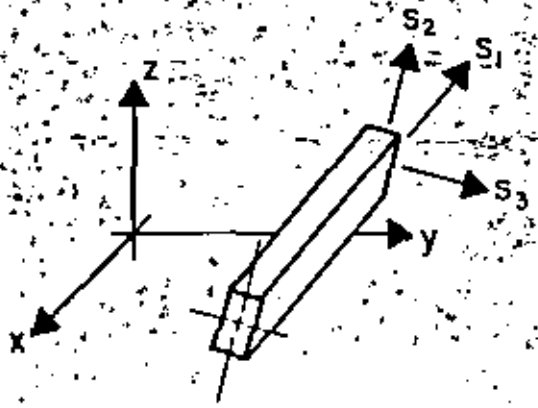
4. Response Spectrum Analysis of Pipe Network

A response spectrum analysis of the pipe assemblage shown in Fig. 12 is carried out. This is example 1 in the User's Manual for the "PIPDYN" computer program [36]. Good correspondence between the SAP and PIPDYN solutions is obtained. Table 7 compares local z-direction member end moments calculated by the two programs. In the analysis the lowest five modes are considered. Both, horizontal and vertical (proportional) spectra are simultaneously specified.



(a) ELEVATION OF FRAME

DATA: YOUNG'S MODULUS = 432000, MASS DENSITY = 1.0
FOR ALL BEAMS AND COLUMNS $A_1 = 3.0$, $I_1 = I_2 = I_3 = 1.0$
UNITS: FT, KIPS



(b) BEAM ELEMENT DEFINITION

S_1, S_2 AND S_3 = BEAM LOCAL AXES

I_1, I_2 AND I_3 = FLEXURAL INERTIA ABOUT S_1, S_2, AND S_3

A_1 = AREA ASSOCIATED WITH S_1

FIGURE 11: SAP MODEL OF PLANE FRAME

TABLE 6 PERIODS OF PLANE FRAME

MODE NUMBER	PERIOD (SEC)
1	8.183
2	2.673
3	1.543

TABLE 7 COMPARISON OF MOMENT PREDICTIONS
(SAP ANALYSIS OF PIPDYN EXAMPLE 1)

ELEMENT NUMBER	MOMENT MZ (Kip in) IN ELEMENT LOCAL COORDINATES (at element ends 1, see Ref. 29 pp. 54)	
	SAP	PIPDYN
1	376.9	377.0
2	30.67	30.68
3	152.9	152.9
4	100.6	100.6
5	83.27	83.27
6	46.17	46.19
7	1.081	1.082
8	21.59	21.81
9	7.052	7.038
10	7.537	7.571
11	160.3	160.4
12	78.07	78.09
13	26.08	25.80

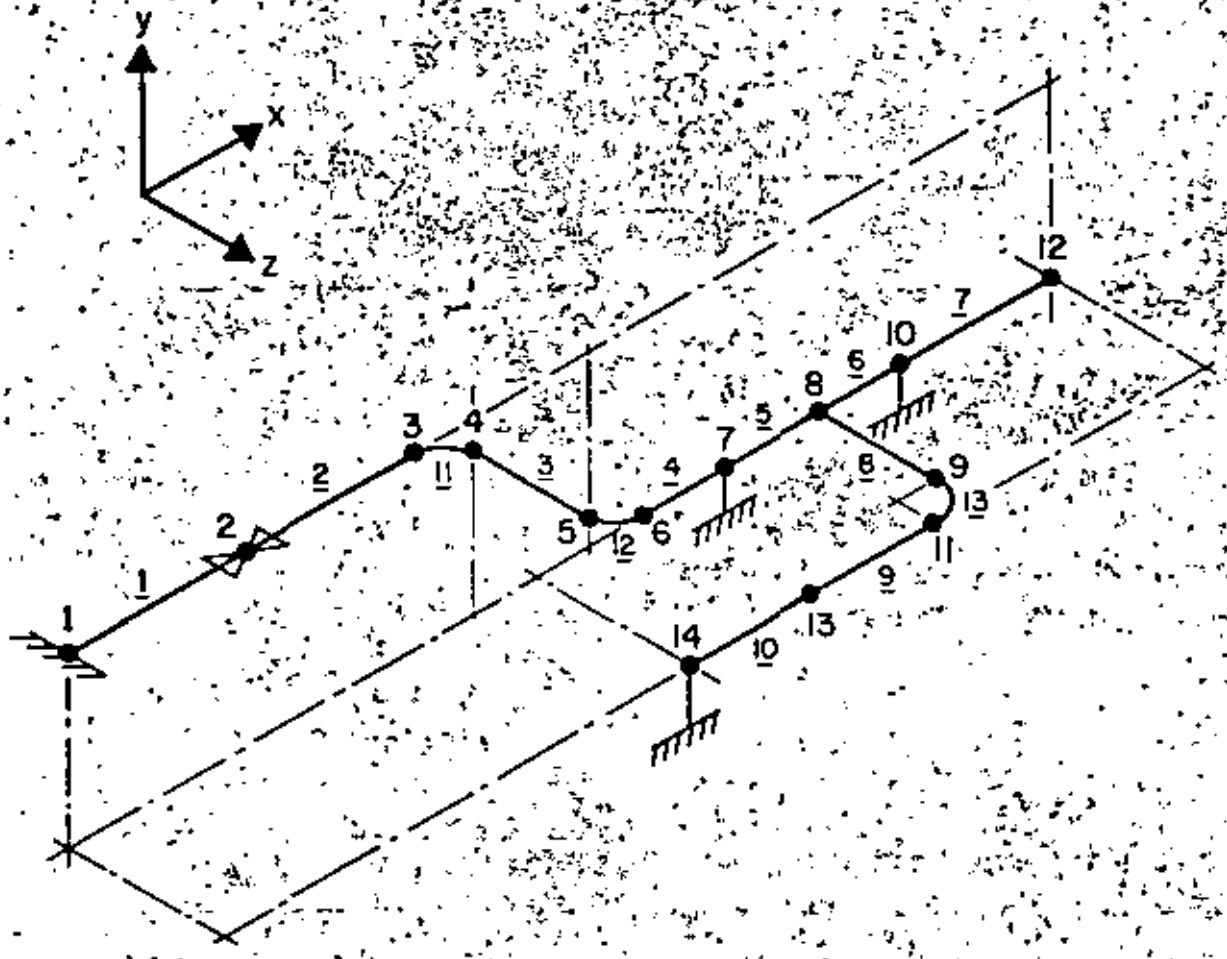


FIGURE 12: SAP MODEL OF PIPDYN EXAMPLE 1, RESPONSE SPECTRUM ANALYSIS

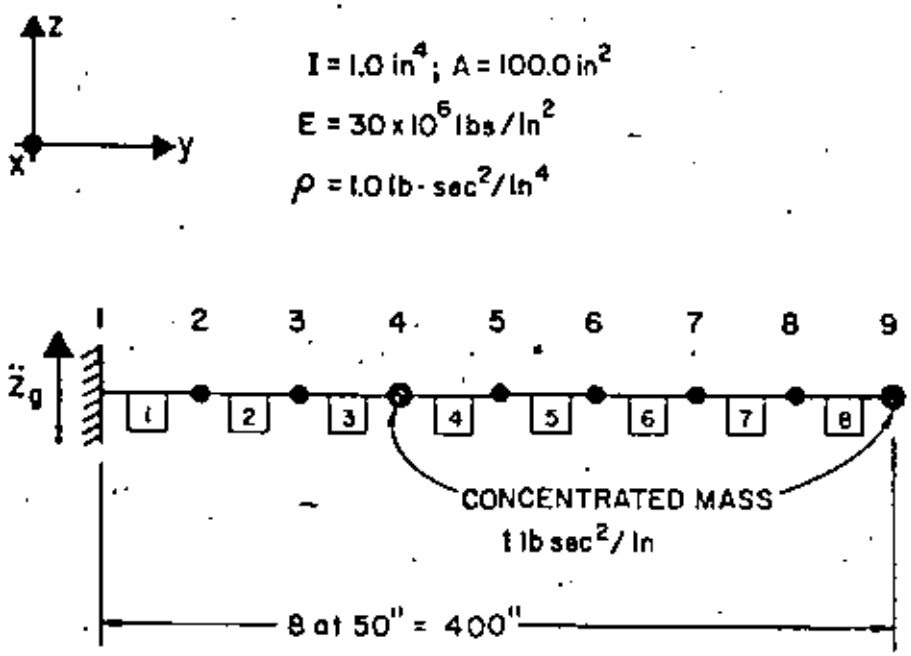
5. Mode Superposition Time History Response Analysis of Cantilever

The cantilever beam shown in Fig. 13 is analyzed for the ground acceleration shown in the same figure. The solution to this problem is obtained independently using the "DRA2" computer code [21]. This program calculates the dynamic response by direct integration of the (coupled) equations of motion using the Wilson θ-algorithm [6].

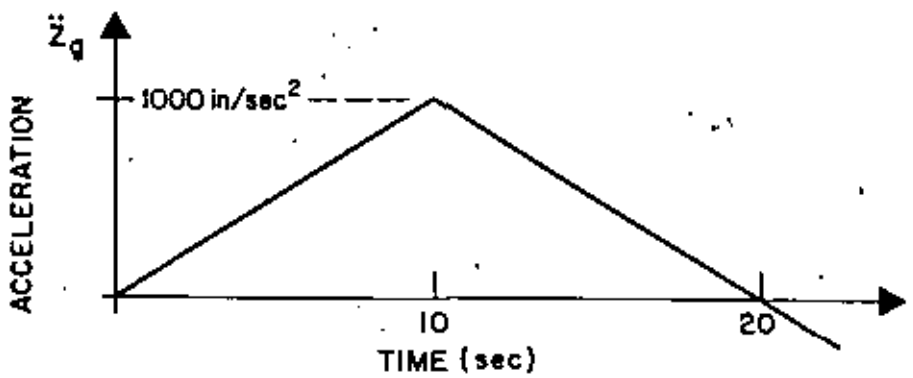
The response history of the beam model is evaluated in SAP using mode superposition including all eight flexural modes developed in the cantilever; Table 8 lists the periods of these eight modes computed by SAP. Figure 14 shows the variation of the transverse displacements and of the fixed-end moment calculated by SAP. The DRA2 predictions agree with the SAP results to 5 or more digits and, consequently, are not shown for comparison.

6. Mode Superposition Time History Response Analysis of Cylindrical Tube

The response of the simply supported cylindrical tube shown in Fig. 15 for a suddenly applied load is calculated by mode superposition. Using symmetry one half of the tube is idealized as an assemblage of axisymmetric elements with a total of 61 degrees of freedom. In the mode superposition analysis only the lowest twenty modes are considered; some of the vibration periods are listed in Table 9. Figure 15 shows a comparison of the radial displacements calculated by the program with a Timoshenko-Love solution [24].



(a) NODE AND BEAM NUMBER ASSIGNMENTS FOR THE CANTILEVER MODEL



(b) GROUND ACCELERATION APPLIED AT NODE 1

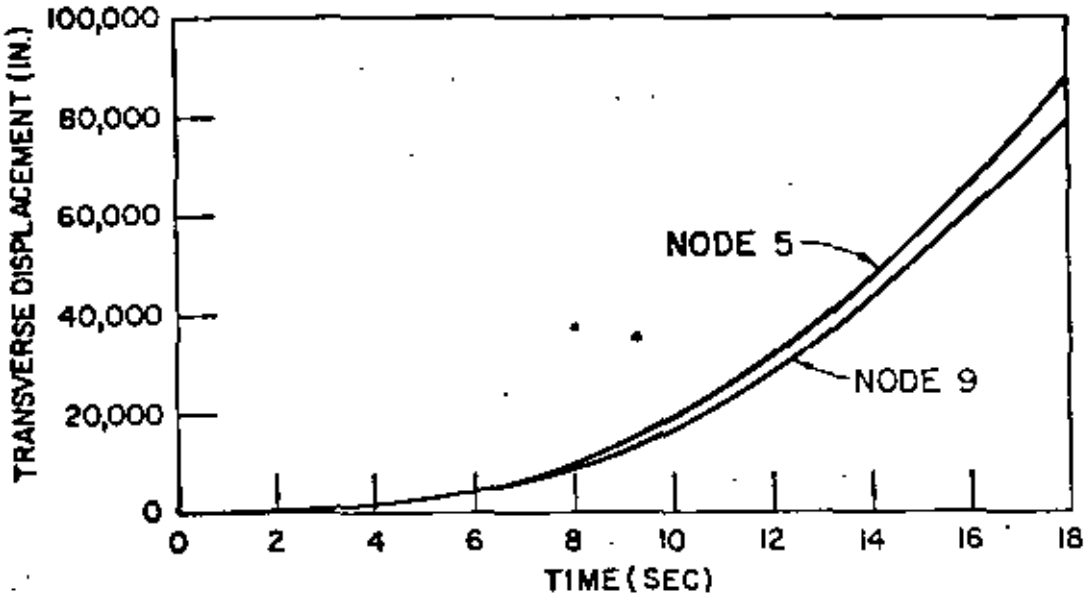
FIGURE 13: RESPONSE HISTORY ANALYSIS OF CANTILEVER BEAM

TABLE 8 CANTILEVER BEAM ANALYSIS - NATURAL PERIODS FOR THE EIGHT (LOWEST) FLEXURAL MODES

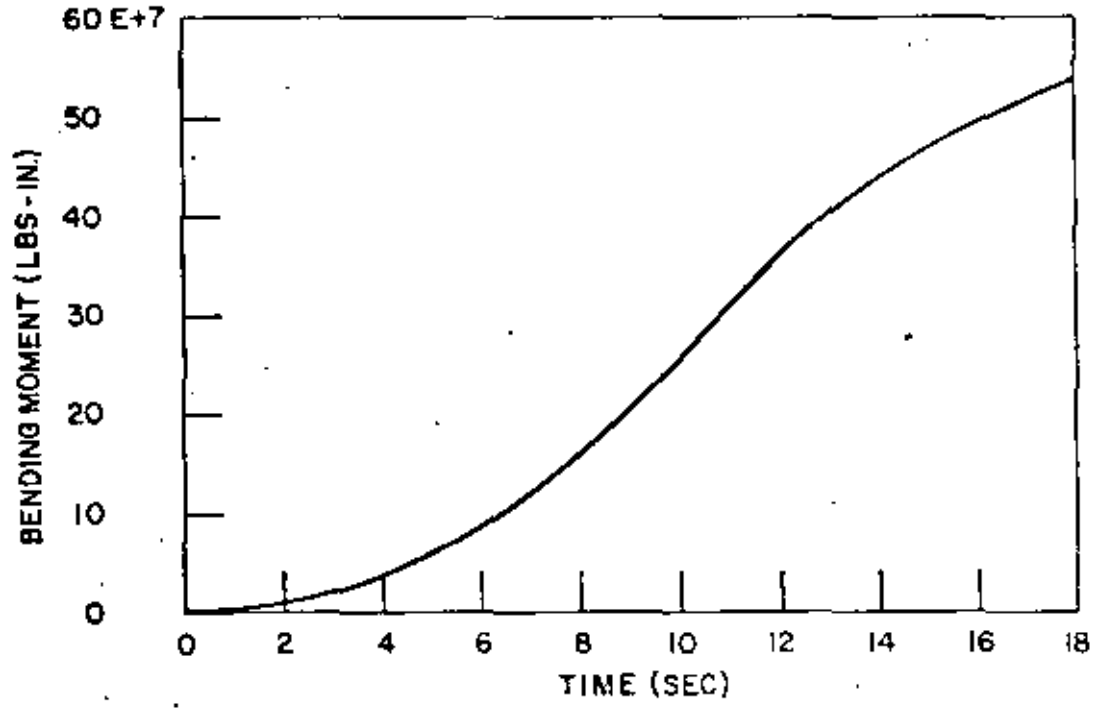
MODE NUMBER	PERIOD (SEC)
1	525.79
2	85.368
3	30.965
4	16.059
5	9.9006
6	6.8276
7	5.1865
8	4.3777

TABLE 9 CYLINDRICAL TUBE ANALYSIS - SOME NATURAL PERIODS

MODE NUMBER	PERIOD (SEC x 10^{-3})
1	1.2788
5	0.62140
10	0.32983
15	0.17463
20	0.11497

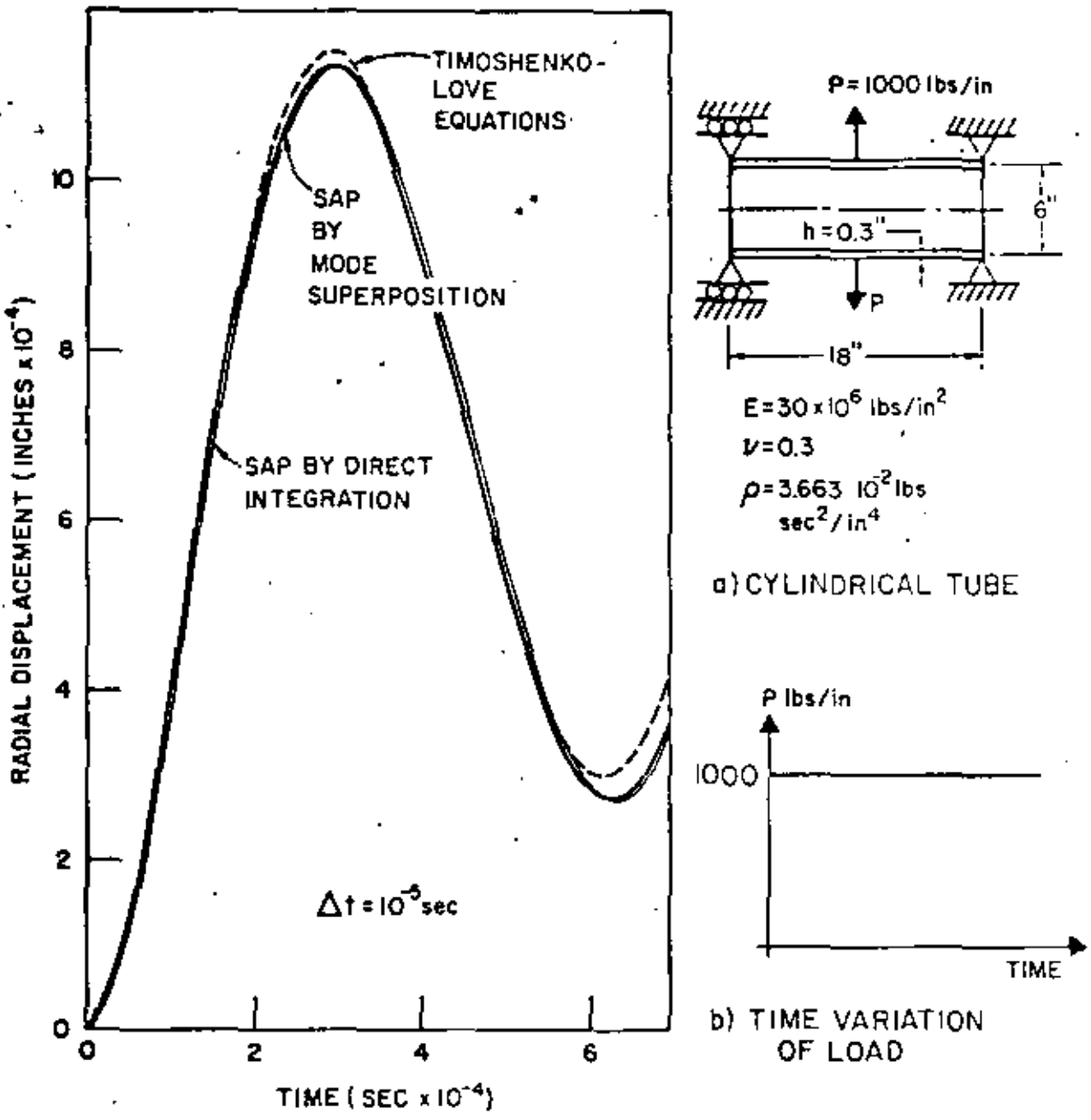


(a) TRANSVERSE DEFLECTIONS



(b) MOMENT AT NODE 1
(FIXED END OF CANTILEVER)

FIGURE 14: CANTILEVER RESPONSE



c) RADIAL DISPLACEMENT VERSUS TIME

FIGURE 15: RESPONSE HISTORY ANALYSIS OF CYLINDRICAL TUBE

7. Direct Integration Time History Response Analysis of Cylindrical Tube

The response of the simply supported tube shown in Fig. 15 for the applied load is calculated by direct integration. The same finite element idealization and time step Δt as in the mode superposition is used. Figure 15 shows the radial displacements as calculated by the program.

REFERENCES

1. Argyris, J. H., and Kelsey, A., "Energy Theorems and Structural Analysis," Aircraft Engineering, Vol. 31, Oct. and Nov. 1954, Feb. to May 1955. Also published by Butterworth's Scientific Publications, London, 1960.

2. Argyris, J. H., "Continua and Discontinua," Proceedings Conference on Matrix Methods in Structural Mechanics, Wright Patterson AFB, Ohio, 1965.

3. Bathe, K. J., and Wilson, E. L., "Solution Methods for Eigenvalue Problems in Structural Mechanics," Int. J. Num. Methods in Engg., Vol. 6, No. 2, 1973.

4. Bathe, K. J., and Wilson, E. L., "Eigensolution of Large Structural Systems with Small Bandwidth," ASCE Journal of Eng. Mech. Div., June, 1973.

5. Bathe, K. J., and Wilson, E. L., "Large Eigenvalue Problems in Dynamic Analysis," ASCE Journal of Eng. Mech. Div., Dec. 1972.

6. Bathe, K. J., and Wilson, E. L., "Stability and Accuracy Analysis of Direct Integration Methods," Int. J. of Earthquake Engg. and Struct. Dynamics, Vol. 1, No. 2, 1973.

7. Bathe, K.J., and Wilson, E.L., "Thick Shell Structures", Proceedings International Symposium on Structural Mechanics Software, University of Maryland, College Park, Maryland, June 1974.

8. Bathe, K.J., Wilson, E.L., and Iding, R.H., "NONSAP - A Structural Analysis Program for Static and Dynamic Response of Nonlinear Systems", SESM Report 74-3, Department of Civil Engineering, University of California, Berkeley, 1974.

9. Clough, R. W., "Analysis of Structural Vibrations and Dynamic Response", Proceedings 1st U.S.-Japan Symposium on Recent Advances in Matrix Methods of Structural Analysis and Design, Tokyo, Japan, 1968.

10. Clough, R. W., "Earthquake Analysis by Response Spectrum Superposition," Bulletin of the Seismological Society of America, Vol. 52, July 1962.

11. Clough, R. W., and Bathe, K. J., "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.

12. Clough, R. W., and Felippa, C. A., "A Refined Quadrilateral Element for Analysis of Plate Bending," Proceedings 2nd Conference on Matrix Methods in Structural Mechanics, Wright Patterson AFB, Ohio, 1968.

13. Clough, R. W., and Wilson, E. L., "Dynamic Finite Element Analysis of Arbitrary Thin Shells," Computers and Structures, Vol. 1, No.1, 1971.

14. Felippa, C. A., "Refined Finite Element Analysis of Linear and Nonlinear Two-dimensional Structures," SESM Report 66-2, Dept. of Civil Engineering, University of California, Berkeley, 1966.

15. Felippa, C. A., and Clough, R. W., "The Finite Element Method in Solid Mechanics," Proceedings Symposium on Numerical Solutions of Field Problems in Continuum Mechanics, Durham, North Carolina, 1968.

16. Hall, A. S., Tezcan, S. S., and Bulent, D., Discussion of paper "Curved Beam Stiffness Coefficients," ASCE Journal of Struct. Div., Feb., 1969.

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

18. Irons, B. M., "Structural Eigenvalue Problems: Elimination of Unwanted Variables," Journal A.I.A.A., Vol. 3, 1965.

19. Irons, B. M., "Numerical Integration Applied to Finite Element Methods," Conf. on Use of Digital Computers in Structural Engineering, University of New Castle, England, July 1966.

20. MacNeal, R.H., "The NASTRAN Theoretical Manual", NASA Report No. NASA SP-221, September 1970.

21. Peterson, F. E., and Bathe, K. J., "Nonlinear Dynamic Analysis of Reactor Core Components," Report S-104.3, Engineering Analysis Corporation, Berkeley, California, March 1972.

22. Poley, S., "Mesh Analysis of Piping Systems," IBM New York Scientific Center Technical Report No. 320-2939, March 1968.

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

24. Reismann, H., and Padlog, J., "Forced, Axisymmetric Motions of Cylindrical Shells," Journal of the Franklin Institute, Vol. 284, No. 5, Nov. 1967.

25. Roy, J. R., "Numerical Errors in Structural Solutions," ASCE Journal of the Structural Division, April 1971.

26. Strang, G., and Fix, G.J., "An Analysis of the Finite Element Method", Prentice Hall, Inc., Englewood Cliffs, New Jersey, 1973.

27. Timoshenko, S., Theory of Plates and Shells, 2nd Edition, McGraw-Hill, 1959, pp. 544.

28. Wilson, E. L., "SAP - A General Structural Analysis Program," SESM Report 70-20, Dept. of Civil Engineering, University of California, Berkeley, 1970.

29. Wilson, E. L., "SOLID SAP - A Static Analysis Program for Three-Dimensional Solid Structures," SESM Report 71-19, Dept. of Civil Engineering, University of California, Berkeley, 1971.

30. Wilson, E. L., "Earthquake Analysis of Reactor Structures," Proceedings Symposium on Seismic Analysis of Pressure Vessels and Piping Components, The American Society of Mechanical Engineers, 1971.

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

32. Wilson, E. L., Taylor, R. L., Doherty, W. P., and Ghaboussi, J., "Incompatible Displacement Models," ONR Symposium on Matrix Methods in Structural Mechanics, University of Illinois, Urbana, Illinois, Sept. 1971.

33. Wilson, E. L., and Penzien, J., "Evaluation of Orthogonal Damping Matrices," Int. J. for Num. Methods in Engg., Vol. 4, No. 1, 1972.

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

Computer Program Manuals:

35. "ADL Pipe Static-Thermal-Dynamic Pipe Stress Analysis," Arthur D. Little, Inc., Cambridge, Massachusetts, January 1971.

36. "Construction Industry Programs, PIPDYN: Dynamic Analysis of Piping Systems," Computer Sciences Corporation, Los Angeles, California.

EL METODO DEL ELEMENTO FINITO EN LA INGENIERIA MECANICA

UTILIZACION DEL PROGRAMA SAP IV

DR. VICTOR HUGO MUCIÑO QUINTERO

ABRIL, 1982

PRINCIPALES INSTRUCCIONES PARA CREAR UN ARCHIVO
POR MEDIO DE TERMINAL Y PARA CORRER EL PROGRAMA
SAP IV

1.- Para crear un archivo de datos teclear
NAME FILENAME DATA
2.- Para secuenciarlo teclear
SEQ 100+100
3.- Para salirse de secuencia oprimir dos veces la tecla RETURN.
4.- Para volver a secuenciarlo teclear
SEQ (n. de secuencia -100
5.- Para corregir una línea teclear
FIX No. de línea /lo que se quiere quitar/lo que se quiere poner
6.- Para revisar lo que se corrigió teclear
L=
7.- Para listar el programa teclear
LIST
8.- Para listar una línea teclear
LIST No. de línea
9.- Para guardar el archivo teclear
SAVE
10.- Para volver a llamar al archivo teclear
GET FILE NAME
11.- Para correr el programa SAP IV y que los resultados aparezcan en la pantalla teclear
RUN:SAPIV/CORR;FILE FILE5(TITLE="FILENAME")
12.- Para correr el programa SAP IV y que los resultados aparezcan en la impresora teclear
RUN:SAPIV/CORR;FILE FILE5(TITLE="FILENAME"),
FILE6(PRINTER)
Nota FILENAME puede ser cualquier nombre, pero tiene que ser el mismo en el archivo de datos y al correr el programa

B U E N A S U E R T E

APPENDIX - DATA INPUT TO SAP IV

I. HEADING CARD (12A6)

notes	columns	variable	entry
(1)	1 - 72	HED(12)	Enter the heading information to be printed with the output

NOTES/

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

II. MASTER CONTROL CARD (8I5)

notes	columns	variable	entry
(1)	1 - 5	NUMNP	Total number of nodal points (joints) in the model
(2)	6 - 10	NELTYP	Number of element groups
(3)	11 - 15	LL	Number of structure load cases: GE.1; static analysis EQ.0; dynamic analysis
(4)	16 - 20	NF	Number of frequencies to be found in the eigenvalue solution; EQ.0; static analysis GE.1; dynamic analysis
(5)	21 - 25	NDYN	Analysis type code: EQ.0; static analysis EQ.1; eigenvalue/vector solution EQ.2; forced dynamic response by mode superposition EQ.3; response spectrum analysis EQ.4; direct step-by-step integration
(6)	26 - 30	MODEX	Program execution mode: EQ.0; problem solution EQ.1; data check only
(7)	31 - 35	NAD	Total number of vectors to be used in a SUBSPACE ITERATION solution for eigenvalues/vectors: EQ.0; default set to: MIN{2*NF, NF+8}
(8)	36 - 40	KEQB	Number of degrees of freedom (equations) per block of storage: EQ.0; calculated automatically by the program

NOTES.

(1) Nodes are labeled with integers ranging from "1" to the total number of nodes in the system, "NUMNP". The program exits with no diagnostic message if NUMNP is zero (0). Thus, two blank cards are used to end the last data case in a run; i.e., one blank heading card (Section I) and one blank card for this section.

(2) For each different element type (TRUSS, BEAM, etc.) a new element group need be defined. Elements within groups are assigned integer labels ranging from "1" to the total number of elements in the group. Element groups are input in Section IV, below.

Element numbering must begin with one (1) in each different group. It is possible to use more than one group for an element type. For example, all columns (vertical beams) of a building may be considered one group and the girders (horizontal beams) may be considered another group.

(3) At least one (1) load condition must be specified for a static (NDYN.EQ.0) analysis. If the data case calls for one of the dynamic analysis options (NDYN.EQ.1, 2, 3, or 4), no load cases can be requested (i.e., LL is input as "0"). The program always processes Sections V (Concentrated Load/Mass Data) and VI (Element Load Multipliers) and expects to read some data. For the case of a dynamic analysis (NDYN.GE.1) only mass coefficients can be input in Section V, and one (1) blank element load multiplier card is expected in Section VI.

(4) For a static analysis, NF.EQ.0. If NDYN.EQ.1, 2 or 3, the lowest NF eigenvalues are determined by the program. Note that a dynamic solution may be re-started after eigenvalue extraction (providing a previous eigenvalue solution for the model was saved on tape as described in Appendix A). NF for the original and re-start runs must be the same.

(5) If NDYN.EQ.2 or NDYN.EQ.3 the program first solves for NF eigenvalues/vectors and then performs the forced response solution (or the response spectrum analysis). Thus, the program expects to read the control card governing the eigensolution (Section VII.A) before reading data in either Sections VII.B or VII.C. For the case NDYN.EQ.1, the program solves for NF eigenvalues/vectors, prints the results and proceeds to the next data case. The results for the eigenvalue solution phase (NDYN.EQ.1) may be saved for later use in automatic re-start (Appendix A lists the control cards that are required to affect this save operation), i.e. a dynamic solution may be restarted without repeating the solution for modes and frequencies. If this data case is a re-start job, set NDYN.EQ.-2 for a forced response solution, or set NDYN.EQ.-3 for a response spectrum analysis. Note that the solution may be re-started a multiple of times (to run different ground spectra or different time-dependent forcing functions) because the program does not destroy the contents of the re-start tape.

If NDYN.EQ.4 the program performs the response solution by direct step-by-step integration and no eigenvalue solution control card should be provided.

II. MASTER CONTROL CARD (continued)

(6) In the data-check-only mode (MODEX.EQ.1), the program writes only one file, "TAPE8", and this file may be saved for use as input to special purpose programs such as mesh plotters, etc. TAPE8 contains all data input in its completely generated form. If MODEX.EQ.1, most of the expensive calculations required during normal (MODEX.EQ.0) execution are passed. TAPE8, however, is not written during normal problem solution.

Note that a negative value for NDYN ("-2" or "-3"), when executing in the data-check-only mode, does not cause the program to read the re-start tape which contains the eigensolution information; instead, the program jumps directly from this card to Section VII.B (or Section VII.C) and continues reading and checking data cards without performing the solution.

(7) If the program is to solve for eigenvalues using the SUBSPACE ITERATION algorithm, the entry in cc 31-35 can be used to change the total number of iteration vectors to be used from the default minimum of 2*NF or NF+8 (whichever is smaller) to the value "NAD". The effect of increasing NAD over the default value is to accelerate convergence in the calculations for the lowest NF eigenvalues. NAD is principally a program testing parameter and should normally be left blank.

(8) KEQB is a program testing parameter which allows the user to test multiple equation block solutions using small data cases which would otherwise be one block problems. KEQB is normally left blank.

III. NODAL POINT DATA (A1,I4,6I5,3F10.0,I5,F10.0)

notes	columns	variable	entry
(1)	1	CT	Symbol describing coordinate system for this node; EQ. ; (blank) cartesian (X,Y,Z) EQ.C; cylindrical (R,Y,θ)
(2)	2 - 5	N	Node number
(3)	6 - 10	IX(N,1)	X-translation boundary condition code
	11 - 15	IX(N,2)	Y-translation boundary condition code
	16 - 20	IX(N,3)	Z-translation boundary condition code
	21 - 25	IX(N,4)	X-rotation boundary condition code
	26 - 30	IX(N,5)	Y-rotation boundary condition code
	31 - 35	IX(N,6)	Z-rotation boundary condition code EQ.0; free (loads allowed) EQ.1; fixed (no load allowed) GT.1; master node number (beam nodes only)
(4)	36 - 45	X(N)	X (or R) -ordinate
	46 - 55	Y(N)	Y -ordinate
	56 - 65	Z(N)	Z (or θ) -ordinate (degrees)
(5)	66 - 70	KN	Node number increment
(6)	71 - 80	T(N)	Nodal temperature

NOTES/

(1) A special cylindrical coordinate system is allowed for the global description of nodal point locations. If a "C" is entered in card column one (1), then the entries given in cc 36-65 are taken to be references to a global (R,Y,θ) system rather than to the standard (X,Y,Z) system. The program converts cylindrical coordinate references to cartesian coordinates using the formulae:

$$X = R \sin\theta$$
$$Y = Y$$
$$Z = R \cos\theta$$

Cylindrical coordinate input is merely a user convenience for locating nodes in the standard (X,Y,Z) system, and no other references to the cylindrical system are implied; i.e., boundary condition specifications, output displacement components, etc. are referenced to the (X,Y,Z) system.

(2) Nodal point 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 5, below).

Admissible nodal point numbers range from "1" to the total number of nodes "NUMNP". Illegal references are: N.LE.0 or N.GT.NUMNP.

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

IX(N,M) = 0; unspecified (free) displacement (or rotation) component

IX(N,M) = 1; deleted (fixed) displacement (or rotation) component

IX(N,M) = K; node number "K" ($1 \leq K \leq$ NUMNP and $K \neq N$) is the "master" node to which the Mth degree of freedom at node "N" is a "slave"

An unspecified (IX(N,M) = 0) degree of freedom is free to translate or rotate as the solution dictates. Concentrated forces (or moments) may be applied (Section V, below) in this degree of freedom. One (1) 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 model.

Deleted (IX(N,M) = 1) degrees of freedom are removed from the final set of equilibrium equations. Deleted degrees of freedom are fixed (points of 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 any element) must have all six (6) degrees of freedom deleted. Nodal degrees of freedom having undefined stiffness (such as rotations in an all TRUSS model, out-of-plane components in a two-dimensional planar model, etc.) should be deleted. Deletions have the beneficial effect of reducing the size of the set of equations that must be solved. The table below lists the types of degrees of freedom that are defined by each different element type. The table was prepared assuming that the element has general orientation in (X,Y,Z) space.

DEGREES OF FREEDOM WITH DEFINED STIFFNESS

ELEMENT TYPE	δX	δY	δZ	$\delta\theta_X$	$\delta\theta_Y$	$\delta\theta_Z$
1. TRUSS	x	x	x			
2. BEAM	x	x	x	x	x	x
3. MEMBRANE	x	x	x			
4. 2D/QUADRILATERAL		x	x			
5. 3D/BRICK	x	x	x			
6. PLATE/SHELL	x	x	x	x	x	x
7. BOUNDARY	x	x	x	x	x	x

	DEGREES OF FREEDOM WITH DEFINED STIFFNESS					
ELEMENT TYPE	δX	δY	δZ	$\delta\theta_X$	$\delta\theta_Y$	$\delta\theta_Z$
8. THICK SHELL	x	x	x			
9. 3D/PIPE	x	x	x	x	x	x

Hence, for an all 3D/BRICK model, only the X,Y,Z translations are defined at the node, and the number of equations can be cut in half by deleting the three (3) rotational components at every node. If a node is common to two or more different element types, then the non-trivial degrees of freedom are found by combination. For example, all six (6) components are possible at a node common to both BEAM and TRUSS elements; i.e., the BEAM governs.

A "master/slave" option is allowed to model rigid links in the system. For this case, IX(N,M) = K means that the Mth degree of freedom at node "N" is "slave" to (dependent on) the same (Mth) degree of freedom at node "K"; node "K" is said to be the master node to which node N is slave. Note that no actual beam need to run from node K to node N, however the following restrictions hold:

(a) Node one (1) cannot be a master node; i.e., $K \neq 1$.
(b) Nodes "N" and "K" must be beam-only nodes; i.e., no other element type may be connected to either node N or K.
(c) A node "N" can be slave to only one master node, "K"; multiple nodes, however, can be slave to the same master.
(d) If the beam from "N" to "K" is to be a rigid link arbitrarily oriented in the X,Y,Z space, then all six (6) degrees of freedom at node "N" must be made slaves to node "K"

Displacement/rotation components for slave degrees of freedom at node "N" are not recovered for printing; i.e., zeroes appear as output for slave degrees of freedom.

(4) When CT (Col. 1) is equal to the character "C", the values input in CC 36-65 are interpreted as the cylindrical (R,Y,θ) coordinates of node "N". Y is the axis of symmetry. R is the distance of a point from the Y-axis. The angle θ is measured clockwise from the positive Z-axis when looking in the positive Y direction. The cylindrical coordinate values are printed as entered on the card, but immediately after printing the

global cartesian values are computed from the input entries. Note that boundary condition codes always refer to the the (X,Y,Z) system even if the node happens to be located with cylindrical coordinates.

(5) Nodal point cards need not be input in node-order sequence; eventually, however, all nodes in the integer set {1, NUMNP} must be defined. Joint data for a series of nodes

$$\{N_1,\ N_1+1 \times KN_2,\ N_1+2 \times KN_2, \ldots, N_2\}$$

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

CARD 1 / N_1, IX(N_1,1),...,IX(N_1,6),X(N_1),...,KN_1,T(N_1)/

CARD 2 / N_2, IX(N_2,1),...,IX(N_2,6),X(N_2),...,KN_2,T(N_2)/

KN_2 is the mesh generation parameter given on the second card of a sequence. The first generated node is $N_1+1 \times KN_2$; the second generated node is $N_1+2 \times KN_2$, etc. Generation continues until node number $N_2 - KN_2$ is established. Note that the node difference $N_2 - N_1$ must be evenly divisible by KN_2. Intermediate nodes between N_1 and N_2 are located at equal intervals along the straight line between the two points. Boundary condition codes for the generated data are set equal to the values given on the first card. Node temperatures are found by linear interpolation between T(N_1) and T(N_2). Coordinate generation is <u>always</u> performed in the (X,Y,Z) system, and no generation is performed if KN_2 is zero (blank).

(6) Nodal temperatures describe the actual (physical) temperature distribution in the structure. Average element temperatures established from the nodal values are used to select material properties and to compute thermal strains in the model (static analysis only).

IV. ELEMENT DATA

TYPE 1 - THREE-DIMENSIONAL TRUSS ELEMENTS

Truss elements are identified by the number 1. Axial forces and stresses are calculated for each member. A uniform temperature change and inertia loads in three directions can be considered as the basic element load conditions. The truss elements are described by the following sequence of cards:

A. Control Card (3I5)

Columns	1 - 5	The number 1
	6 - 10	Total number of truss elements
	11 - 15	Number of material property cards

B. Material Property Cards (I5,5F10.0)

There need be as many of the following cards as are necessary to define the properties listed below for each element in the structure.

Columns	1 - 5	Material identification number
	6 - 15	Modulus of elasticity
	16 - 25	Coefficient of thermal expansion
	26 - 35	Mass density (used to calculate mass matrix)
	36 - 45	Cross-sectional area
	46 - 55	Weight density (used to calculate gravity loads)

C. Element Load Factors (4F10.0) Four cards

Three cards specifying the fraction of gravity (in each of the three global coordinate directions) to be added to each element load case.

Card 1: Multiplier of gravity load in the +X direction

Columns	1 - 10	Element load case A
	11 - 20	Element load case B
	21 - 30	Element load case C
	31 - 40	Element load case D

Card 2: As above for gravity in the +Y direction

Card 3: As above for gravity in the +Z direction

Card 4: This indicates the fraction of the thermal load to be added to each of the element load cases.

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

One card per element in increasing numerical order starting with one.

Columns	1 - 5	Element number

Columns	6 - 10	Node number I
	11 - 15	Node number J
	16 - 20	Material property number
	21 - 30	Reference temperature for zero stress
	31 - 35	Optional parameter k used for automatic generation of element data.

NOTES/

(1) If a series of elements exist such that the element number, N_i, is one greater than the previous element number (i.e. $N_i = N_{i-1} + 1$) and the nodal point number can be given by

$$I_i = I_{i-1} + k$$

$$J_i = J_{i-1} + k$$

then only the first element in the series need be provided. The element identification number and the temperature for the generated elements are set equal to the values on the first card. If k (given on the first card) is input as zero it is set to 1 by the program.

(2) The element temperature increase ΔT used to calculate thermal loads is given by

$$\Delta T = (T_i + T_j)/2.0 - T_r$$

where $(T_i + T_j)/2.0$ is the average of the nodal temperatures specified on the nodal point data cards for nodes i and j; and T_r is the zero stress reference temperature specified on the element card. For truss elements it is generally more convenient to set $T_i = T_j = 0.0$ such that $\Delta T = -T_r$ (note the minus sign). Other types of member loadings can be specified using an equivalent ΔT. If a truss member has an initial lack of fit by an amount d (positive if too long) then $\Delta T = d/(\alpha L)$. If an initial prestress force P (positive if tensile) is applied to the member ends that is released after the member is connected to the rest of the structure then $\Delta T = - P/(\alpha A E)$. In the above formulas A = cross section area, L = member length and α = coefficient of thermal expansion.

IV. ELEMENT DATA (continued)

TYPE 2 - THREE-DIMENSIONAL BEAM ELEMENTS

Beam elements are identified by the number 2. Forces (axial and shear) and moments (bending and torsion) are calculated (in the beam local coordinate system) for each beam. Gravity loadings in each coordinate direction and specified fixed end forces form the basic element load conditions.

The beam elements are described by the following sequence of cards:

A. Control Card (5I5)

Columns	1 - 5	The number 2
	6 - 10	Total number of beam elements
	11 - 15	Number of element property cards
	16 - 20	Number of fixed end force sets
	21 - 25	Number of material property cards

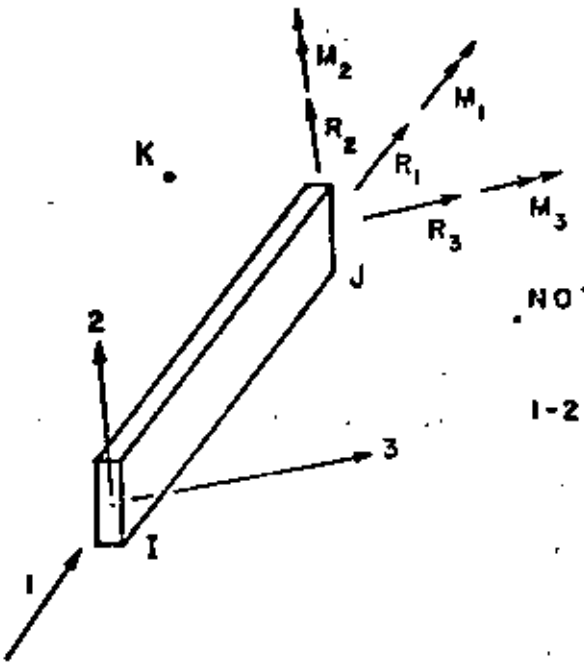
B. Material Property Cards (I5,3F10.0)

Columns	1 - 5	Material identification number
	6 - 15	Young's modulus
	16 - 25	Poisson's ratio
	26 - 35	Mass density (used to calculate mass matrix)
	36 - 45	Weight density (used to calculate gravity loads)

C. Element Property Cards (I5,6F10.0)

Columns	1 - 5	Geometric property number
	6 - 15	Axial area
	16 - 25	Shear area associated with shear forces in local 2-direction
	26 - 35	Shear area associated with shear forces in local 3-direction
	36 - 45	Torsional inertia
	46 - 55	Flexural inertia about local 2-axis
	56 - 65	Flexural inertia about local 3-axis

One card is required for each unique set of properties. Shear areas need be specified only if shear deformations are to be included in the analysis.



NOTE:
K IS ANY NODAL POINT WHICH LIES IN THE LOCAL 1-2 PLANE (NOT ON THE 1-AXIS)

LOCAL COORDINATE SYSTEM FOR BEAM ELEMENT

D. Element Load Factors (4F10.0)

Nodal point loads (no moments) due to gravity are computed. Three cards need be supplied which specify the fraction of these loads (in each of the three global coordinate directions) to be added to each element load case.

Card 1: Multiplier of gravity load in the +X direction

Columns		
1 - 10	Element load case A
11 - 20	Element load case B
21 - 30	Element load case C
31 - 40	Element load case D

Card 2: As above for gravity in the +Y direction

Card 3: As above for gravity in the +Z direction

E. Fixed-End Forces (I5,6F10.0/I5,6F10.0)

Two cards are required for each unique set of fixed-end forces occurring in the analysis. Distributed loads and thermal loads can be specified using the fixed-end forces.

Card 1:

Columns	
1 - 5	Fixed-end force number
6 - 15	Fixed-end force in local 1-direction at Node I
16 - 25	Fixed-end force in local 2-direction at Node I
26 - 35	Fixed-end force in local 3-direction at Node I
36 - 45	Fixed-end moment about local 1-direction at Node I
46 - 55	Fixed-end moment about local 2-direction at Node I
56 - 65	Fixed-end moment about local 3-direction at Node I

IV. ELEMENT DATA (continued)

Card 2:

Columns		
	1 - 5	Blank
	6 - 15	Fixed-end force in local 1-direction at Node J
	16 - 25	Fixed-end force in local 2-direction at Node J
	26 - 35	Fixed-end force in local 3-direction at Node J
	36 - 45	Fixed-end moment about local 1-direction at Node J
	46 - 55	Fixed-end moment about local 2-direction at Node J
	56 - 65	Fixed-end moment about local 3-direction at Node J

Note that values input are literally fixed-end values. Corrections due to hinges and rollers are performed within the program. Directions 1, 2 and 3 indicate principal directions in the local beam coordinates

F. Beam Data Cards (10I5,2I6,I8)

Columns		
	1 - 5	Element number
	6 - 10	Node number I
	11 - 15	Node number J
	16 - 20	Node number K - see accompanying figure
	21 - 25	Material property number
	26 - 30	Element property number
	31 - 35	A
	36 - 40	B
	41 - 45	C
	46 - 50	D
		Fixed-end force identification for element load cases A, B, C, and D respectively
	51 - 56	End release code at node I
	57 - 62	End release code at node J
	63 - 70	Optional parameter k used for automatic generation of element data. This option is described below under a separate heading. If the option is not used, the field is left blank.

The end release code at each node is a six digit number of ones and/or zeros. The 1st, 2nd, . . . , 6th digits respectively correspond to the force components R1, R2, R3, M1, M2, M3 at each node.

If any one of the above element end forces is known to be zero (hinge or roller), the digit corresponding to that component is a one.

NOTES/

(1) If a series of elements occurs in which each element number NE_i is one greater than the previous number NE_{i-1}

i.e., $$NE_i = NE_{i-1} + 1$$

only the element data card for the first element in the series need be given as input, provided

IV. ELEMENT DATA (continued)

(1) The end nodal point numbers are $NI_i = NI_{i-1} + k$

$NJ_i = NJ_{i-1} + k$

and the

(2) material property number
(3) element property number
(4) fixed-end force identification numbers for each element load case
(5) element release code
(6) orientation of local 2-axis

are the same for each element in the series.

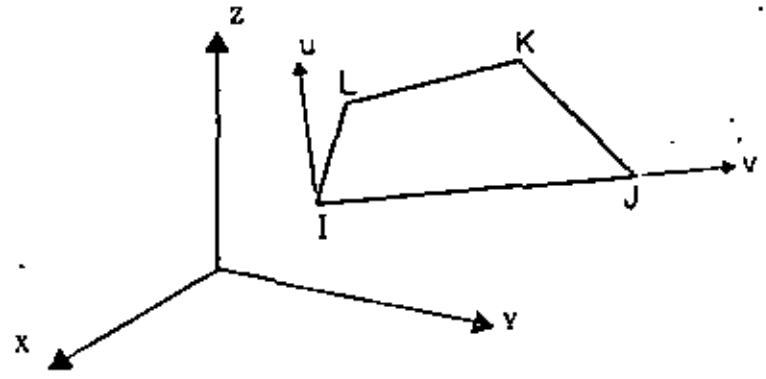
The value of k, if left blank, is taken to be one. The element data card for the last beam element must always be given.

(2) When successive beam elements have the same stiffness, orientation and element loading, the program automatically skips recomputation of the stiffness. Note this when numbering the beams to obtain maximum efficiency.

TYPE 3 – PLANE STRESS MEMBRANE ELEMENTS

Quadrilateral (and triangular) elements can be used for plane stress membrane elements of specified thickness which are oriented in an arbitrary plane. All elements have temperature-dependent orthotropic material properties. Incompatible displacement modes can be included at the element level in order to improve the bending properties of the elements.

A general quadrilateral element is shown below:



A local element coordinate system is defined by a u-v system. The v-axis coincides with the I-J side of the element. The u axis is normal to the v-axis and is in the plane defined by nodal points I, J and L. Node K must be in the same plane if the element stiffness calculations are to be correct. The following sequence of cards define the input data for a set of TYPE 3 elements.

A. Control Card (6I5)

Columns		
	1 – 5	The number 3
	6 – 10	Total number of plane stress elements
	11 – 15	Number of material property cards
	16 – 20	Maximum number of temperature points for any one material; see Section B below.
	30	Non-zero numerical punch will suppress the introduction of incompatible displacement modes.

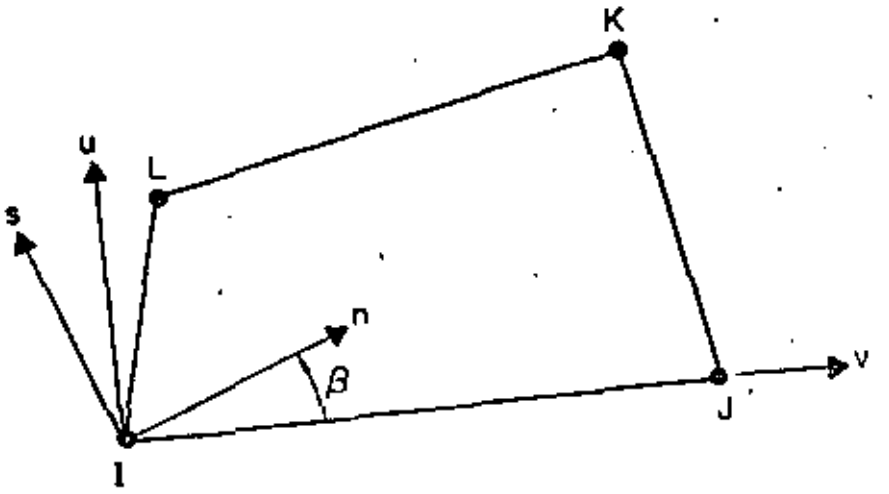
B. Material Property Information

Orthotropic, temperature-dependent material properties are possible. For each different material, the following group of cards must be supplied.

IV. ELEMENT DATA (continued)

1. Material Property Card (2I5,3F10.0)

Columns		
	1 - 5	Material identification number
	6 - 10	Number of different temperatures for which properties are given. If this field is left blank, the number is taken as one.
	11 - 20	Weight density of material (used to calculate gravity loads)
	21 - 30	Mass density (used to calculate mass matrix)
	31 - 40	Angle β in degrees, measured counter-clockwise from the v-axis to the n-axis.



The n-s axes are the principal axes for the orthotropic material. Weight and mass densities need be listed only if gravity and inertia loads are to be considered.

2. Two cards for each temperature:

Card 1: (8F10.0)

Columns		
	1 - 10	Temperature
	11 - 20	Modulus of Elasticity - E_n
	21 - 30	Modulus of Elasticity - E_s
	31 - 40	Modulus of Elasticity E_t
	41 - 50	Strain Ratio - ν_{ns}
	51 - 60	Strain Ratio - ν_{nt}
	61 - 70	Strain Ratio - ν_{st}
	71 - 80	Shear Modulus - G_{ns}

Card 2: (3F10.0)

Columns		
1 - 10	Coefficient of thermal expansion - α_n
11 - 20	Coefficient of thermal expansion - α_s
21 - 30	Coefficient of thermal expansion - α_t

All material constants must always be specified. For plane stress, the program modifies the constitutive relations to satisfy the condition that the normal stress σ_t equals zero.

C. Element Load Factors (5F10.0)

Four cards are used to define the element load cases A, B, C and D as fraction of the basic thermal, pressure and acceleration loads.

First card, load case A: Second card, load case B, etc.

Columns	
1 - 10	Fraction of thermal load
11 - 20	Fraction of pressure load
21 - 30	Fraction of gravity in X-direction
31 - 40	Fraction of gravity in Y-direction
41 - 50	Fraction of gravity in Z-direction

D. Element Cards (6I5,2F10.0,2I5,F10.0)

One card per element must be supplied (or generated) with the following information:

Columns	
1 - 5	Element number
6 - 10	Node I
11 - 15	Node J
16 - 20	Node K
21 - 25	Node L (Node L must equal Node K for triangular elements)
26 - 30	Material identification number
31 - 40	Reference temperature for zero stresses within element
41 - 50	Normal pressure on I-J side of element
51 - 55	Stress evaluation option "n"
56 - 60	Element data generator "k"
61 - 70	Element thickness

NOTES/

(1) Element Data Generation - Element cards must be in element number sequence. If cards are omitted, data for the omitted elements will be generated. The nodal numbers will be generated with respect to the first card in the series as follows:

$$I_n = I_{n-1} + k$$

$$J_n = J_{n-1} + k$$

$$K_n = K_{n-1} + k$$

$$L_n = L_{n-1} + k$$

All other element information will be set equal to the information on the last card read. The data generation parameter "k" is specified on that card.

(2) Stress Print Option - See element type 4

(3) Thermal Data - See element type 4

(4) Use of Triangles - See element type 4

(5) Use of Incompatible Modes - See element type 4

IV. ELEMENT DATA (continued)

TYPE 4 - TWO-DIMENSIONAL FINITE ELEMENTS

Quadrilateral (and triangular) elements can be used as:

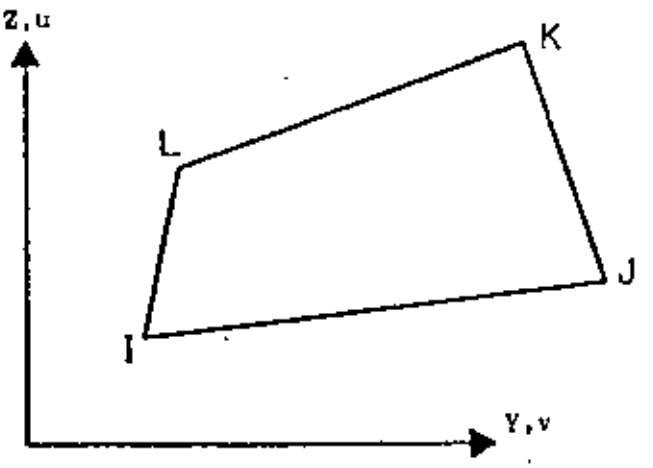
(i) Axisymmetric solid elements symmetrical about the Z-axis. The radial direction is specified as the Y-axis. Care must be exercised in combining this element with other types of elements.

(ii) Plane strain elements of unit thickness in the Y-Z plane.

(iii) Plane stress elements of specified thickness in the Y-Z plane.

All elements have temperature-dependent orthotropic material properties. Incompatible displacement modes can be included at the element level in order to improve the bending properties of the element.

A general quadrilateral element is shown below:



A. Control Card (6I5)

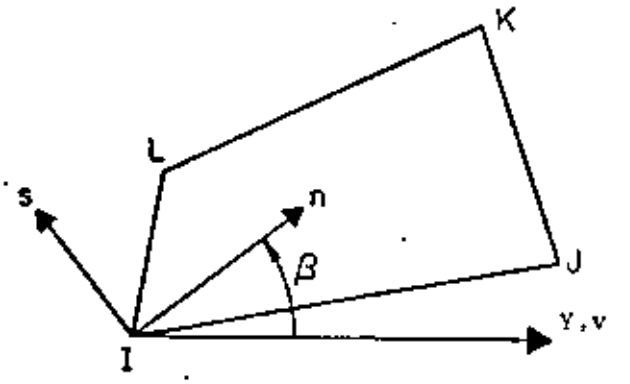
Columns		
1 - 5		The number 4
6 - 10		Total number of elements
11 - 15		Number of different materials
16 - 20		Maximum number of temperature cards for any one material - see Section B below.
25	0	for axisymmetric analysis
	1	for plane strain analysis
	2	for plane stress analysis
30		Non-zero numerical punch will suppress the introduction of incompatible displacement modes. Incompatible modes cannot be used for triangular elements and are automatically suppressed.

B. Material Property Information

Orthotropic, temperature-dependent material properties are possible. For each different material the following group of cards must be supplied.

1. Material Property Card (2I5,3F10.0)

Columns	1 - 5	Material identification number
	6 - 10	Number of different temperature for which properties are given. If this field is left blank, the number is taken as one.
	11 - 20	Weight density of material (used to calculate gravity loads)
	21 - 30	Mass density (used to calculate mass matrix)
	31 - 40	Angle β in degrees, measured counter-clockwise from the v-axis to the n-axis.



PRINCIPAL MATERIAL AXES

The n-s axes are the principal axes for the orthotropic material. Weight density is needed only if gravity and inertia loads are to be considered.

2. Two cards for each temperature:

Card 1: (8F10.0)

Columns	1 - 10	Temperature
	11 - 20	Modulus of elasticity - E_n
	21 - 30	Modulus of elasticity - E_s
	31 - 40	Modulus of elasticity - E_t
	41 - 50	Strain ratio - ν_{ns}
	51 - 60	Strain ratio - ν_{nt}
	61 - 70	Strain ratio - ν_{st}
	71 - 80	Shear modulus - G_{ns}

Card 2: (3F10.0)

Columns		
1 - 10	Coefficient of thermal expansion -	α_n
11 - 20	Coefficient of thermal expansion -	α_s
21 - 30	Coefficient of thermal expansion -	α_t

All material constants must always be specified. In plane stress, the program modifies the constitutive relations to satisfy the condition that the normal stress σ_t equals zero.

C. Element Load Factors

Four cards are used to define the element load cases A, B, C and D as fraction of the basic thermal, pressure and acceleration loads.

First card, load case A; Second card, load case B; etc.

Columns	
1 - 10	Fraction of thermal load
11 - 20	Fraction of pressure load
21 - 30	Fraction of gravity in X-direction
31 - 40	Fraction of gravity in Y-direction
41 - 50	Fraction of gravity in Z-direction

D. Element Cards (6I5,2F10.0,2I5,F10.0)

One card per element must be supplied (or generated) with the following information:

Columns	
1 - 5	Element number
6 - 10	Node I
11 - 15	Node J
16 - 20	Node K
21 - 25	Node L (Node L must equal Node K for triangular elements)
26 - 30	Material identification number
31 - 40	Reference temperature for zero stresses within element
41 - 50	Normal pressure on I-J side of element
51 - 55	Stress evaluation option "n"
56 - 60	Element data generator "k"
61 - 70	Element thickness (For plane strain set equal to 1.0 by program)

NOTES/

(1) Element Data Generation - Element cards must be in element number sequence. If cards are omitted the omitted element data will be generated. The nodal numbers will be generated with respect to the first card in the series as follows:

$$I_n = I_{n-1} + k$$

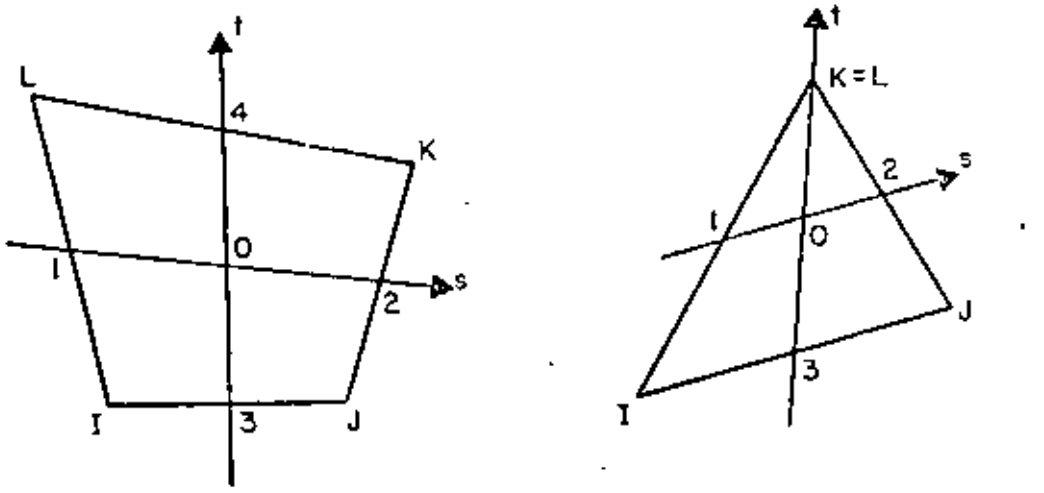
$$J_n = J_{n-1} + k$$

$$K_n = K_{n-1} + k$$

$$L_n = L_{n-1} + k$$

All other element information will be set equal to the information on the last card read. The data generation parameter k is given on that card.

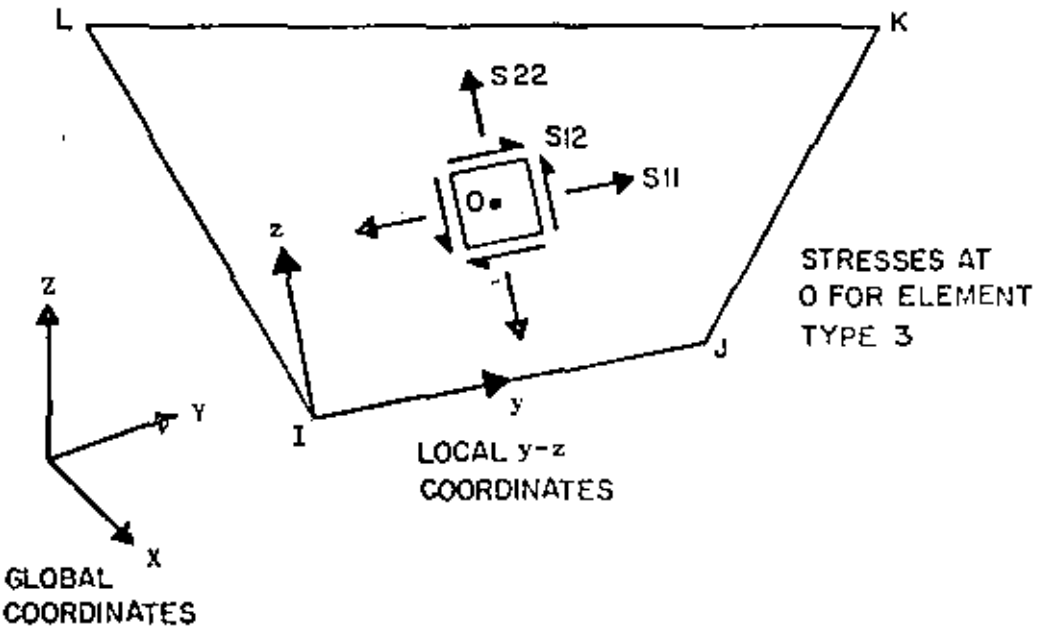
(2) Stress Print Option - The following description of the stress print option applies to both element types 3 and 4. The value of the stress print option "n" can be given as 1, 0, 8, 16 or 20.

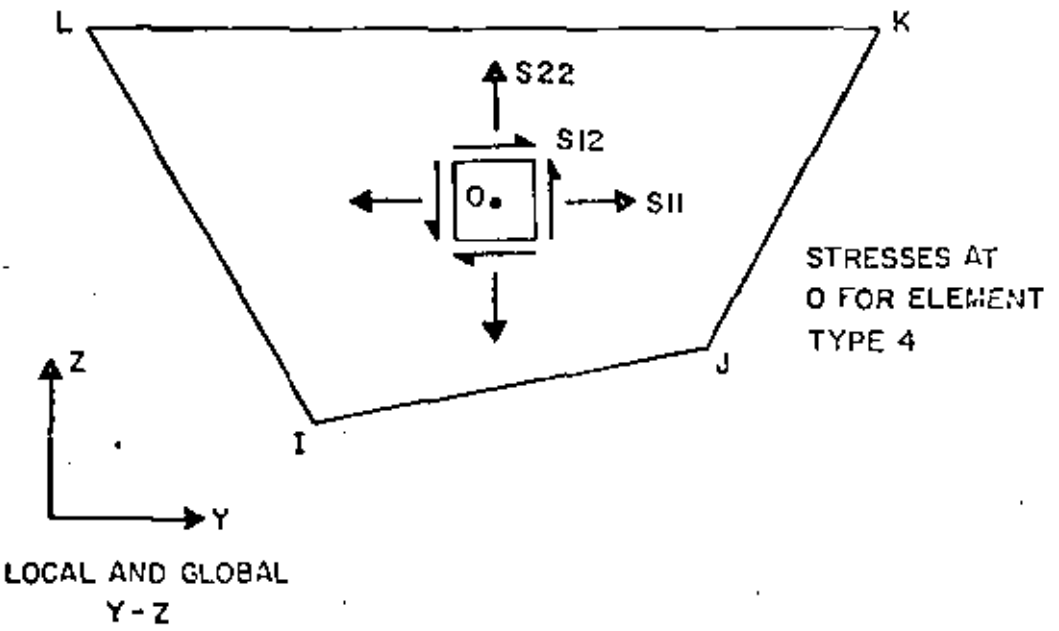


0 = origin of natural s-t coordinates (Fig. 5-2). Points 1, 2, 3 and 4 are midpoints of sides. The points at which stresses are output depend on the value of n as described in the following table.

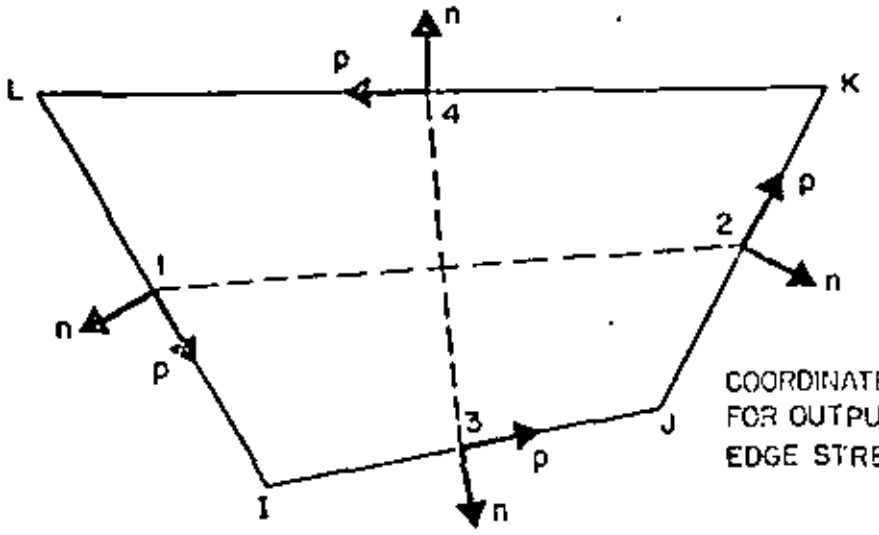
n	Stresses output at
1	None
0	0
8	0, 1
16	0, 1, 2, 3
20	0, 1, 2, 3, 4

The stresses at O are printed in a local y-z coordinate system. For element type 3, side I-J defines the local y-z axes in the plane of the element. For element type 4 the local y-z axes are parallel to the global Y-Z axes.

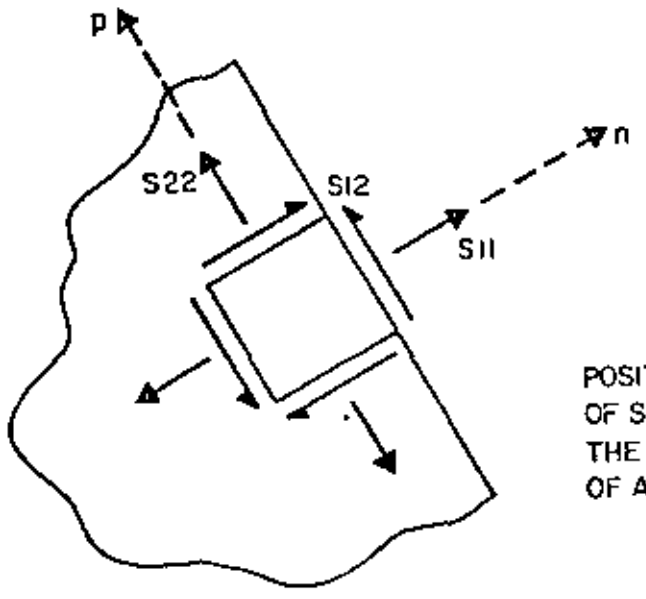




For both element types 3 and 4 the stresses at each edge midpoint are output in a rectangular n-p coordinate system defined by the outward normal to the edge (n axis) and the edge (p axis). The positive p axis for points 1, 2, 3 and 4 is from L to I, J to K, I to J and K to L respectively (positive direction is counterclockwise about element).



COORDINATE SYSTEMS FOR OUTPUT OF EDGE STRESSES



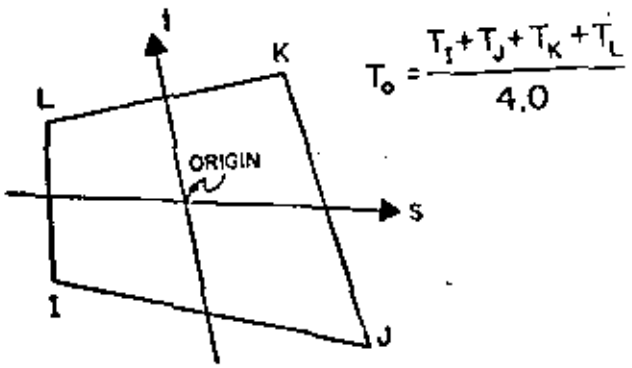
POSITIVE STATE OF STRESS AT THE MIDPOINT OF A SIDE

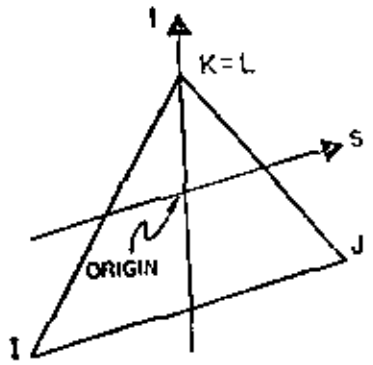
The stresses for an element are output under the following headings: S11, S22, S12, S33, S-MAX, S-MIN, ANGLE. The normal stresses S11 and S22 and the shear stress S12 are as described above. S-MAX and S-MIN are the principal stresses in the plane of the element and S33 is the third principal stress acting on the plane of the element. ANGLE is the angle in degrees from (1) the local y axis at point 0, or (2) the n axis at the midpoints, to the axis of the algebraically largest principal stress.

For triangular elements the stress print option is as described above except that n = 20 is not valid. If n = 20 is input, n will be set to 16 by the program.

(3) Thermal Data - Nodal temperatures as specified on the nodal point data cards are used by element types 3 and 4 in the following two ways:

(1) Temperature-dependent material properties are approximated by interpolating (or extrapolating) the input material properties at the temperature T_o corresponding to the origin of the local s-t coordinate system (see Fig. 5.2 for description of local element coordinates). The material properties throughout the element are assumed constant corresponding to this temperature.





(2) For computation of nodal loads due to thermal strains in the element a bilinear interpolation expansion for the temperature change ΔT (s,t) is used.

$$\Delta T\ (s,t) = \sum_{i=1}^{4} h_i(s,t)\ T_i - T_r$$

where T_i are the nodal temperatures specified on the joint data cards, T_r is the reference stress free temperature and h_i (s,t) are the interpolation functions given by Eq. 5.7.

(4) Use of Triangles - In general, the elements are most effective when they are rectangular, i.e. the elements are not distorted. Therefore, regular and rectangular element mesh layouts should be used as much as possible. In particular, the triangle used is the constant strain triangle; and it should be avoided, since its accuracy is not satisfactory.

(5) Use of Incompatible Modes - Incompatible displacement modes have been found to be effective only when used in rectangular elements. They should always be employed with care. Since incompatible modes are used for all elements of a group it is recommended to use separate element groups for elements with incompatible modes and elements without incompatible modes, respectively. (See Section II, note (2)).

TYPE 5 - THREE-DIMENSIONAL SOLID ELEMENTS (EIGHT NODE BRICK)

General three-dimensional, eight-node, isoparametric elements with three translational degrees of freedom per node are identified by the number 5. Isotropic material properties are assumed. The element load cases (A, B, C and D) are defined as a combination of surface pressure, hydrostatic loads, inertia loads in three directions and thermal loads. The six components of stress and three principal stresses are computed at the center of each element. Also, surface stresses are evaluated. Nine incompatible displacement modes are assumed in the formation of element stiffness matrices. For 8-node elements without incompatible modes use element type 8.

A. Control Card (4I5)

Columns	1 - 5	The number 5
	6 - 10	Number of 8-node solid elements
	11 - 15	Number of different materials
	16 - 20	Number of element distributed load sets

B. Material Property Cards (I5,4F10.0) One card for each different material

Columns	1 - 5	Material identification number
	6 - 15	Modulus of elasticity (only elastic, isotropic materials are considered)
	16 - 25	Poisson's ratio
	26 - 35	Weight density of material (for calculation of gravity loads or mass matrix)
	36 - 45	Coefficient of thermal expansion

C. Distributed Surface Loads (2I5,2F10.2,I5) One card is required for each unique set of uniformly distributed surface loads and for each reference fluid level for hydrostatically varying pressure loads. See notes (4) and (5) for sign convention.

Columns	1 - 5	Load set identification number
	6 - 10	LT (load type) LT = 1 if this card specifies a uniformly distributed load. LT = 2 if this card specifies a hydrostatically varying pressure.
	11 - 20	P If LT = 1, P is the magnitude of the uniformly distributed load If LT = 2, P is the weight density of the fluid causing the hydrostatic pressure
	21 - 30	Y If LT = 1, leave blank If LT = 2, Y is the global Y coordinate of the surface of fluid causing hydrostatic pressure loading
	31 - 35	Element face number on which surface load acts. Face numbers are from 1 to 6 as

described in note (5) for uniformly distributed loads and can be only faces 2, 4 or 6 for hydrostatically varying pressures.

D. Acceleration due to gravity (F10.2)

Columns 1 - 10 Acceleration due to gravity (for calculation of mass matrix)

E. Element Load Case Multipliers (5 cards of 4F10.2)

Multipliers on the element load cases are scaling factors in order to provide flexibility in modifying applied loads.

Card 1:	Columns	1 - 10	PA	Pressure load multipliers
		11 - 20	PB	
		21 - 30	PC	
		31 - 40	PD	

PA is a factor used to scale the complete set of distributed surface loads. This scaled set of loads is assigned to element load case A. Note that zero is a valid multiplier. PB, PC and PD are similar to PA except that scaled loads are assigned to element load cases B, C and D respectively. For the majority of applications these factors should be 1.0

Card 2:	Columns	1 - 10	TA	Thermal load multipliers
		11 - 20	TB	
		21 - 30	TC	
		31 - 40	TD	

TA is a factor used to scale the complete set of thermal loads. The scaled set of loads are then assigned to element load case A. TB, TC and TD are similar and refer to element load cases B, C and D respectively.

Card 3:	Columns	1 - 10	GXA	Gravity load multipliers for + X global direction
		11 - 20	GXB	
		21 - 30	GXC	
		31 - 40	GXD	
Card 4:	Columns	1 - 10	GYA	Gravity load multipliers for + Y global direction
		11 - 20	GYB	
		21 - 30	GYC	
		31 - 40	GYD	
Card 5:	Columns	1 - 10	GZA	Gravity load multipliers for + Z global direction
		11 - 20	GZB	
		21 - 30	GZC	
		31 - 40	GZD	

Gravity loads are computed from the weight density of the material and from the geometry of the element. GXA is a multiplier which reflects the location of the gravity axis and any load factors used. The program computes the weight of the element, multiplies it by GXA and assigns the resulting loads to the + X direction of element load case A. Consequently GXA is the product of the component of gravity along the + X global axis (from - 1.0 to 1.0) and any desired load factor. GXB, GXC and GXD are similar to GXA and refer to element load cases B, C and D respectively. GYA and GZA refer to the global Y and Z directions respectively.

F. Element Cards (12I5,4I2,2I1,F10.2)

Columns		
1 - 5	Element number	
6 - 10	Global node point numbers corresponding to element nodes (See note (3))	1
11 - 15		2
16 - 20		3
21 - 25		4
26 - 30		5
31 - 25		6
36 - 40		7
41 - 45		8
46 - 50	Integration Order	
51 - 55	Material Number	
56 - 60	Generation Parameter (INC)	
61 - 62	LSA	LSA is the distributed surface load set identification number of the distributed load acting on this element to be assigned to element load case A. LSB, LSC and LSD refer to element load cases B, C and D respectively
63 - 64	LSB	
65 - 66	LSC	
67 - 68	LSD	
69 - 70	Face numbers for stress output	
71 - 80	Stress-free element temperature	

NOTES/

(1) Element Generation

1. Element cards must be in ascending order
2. Generation is possible as follows:
   If a series of element cards are omitted,
   a. Nodal point numbers are generated by adding INC to those of the preceding element. (If omitted, INC is set equal to 1.)
   b. Same material properties are used as for the preceding element.
   c. Same temperature is used for succeeding elements.

d. If on first card for the series the integration order is:
   - >0 Same value is used for succeeding elements.
   - = 0 A new element stiffness is not formed. Element stiffness is assumed to be identical to that of the preceding element.
   - <0 Absolute value is used for the first element of the series, and the same element stiffness is used for succeeding elements.

e. If on first card for the series, the distributed load number (for any load case) is:
   - >0 Same load is applied to succeeding elements.
   - <0 The load case is applied to this element but not to succeeding elements in the series.

3. Element card for the last element must be supplied.

(2) Integration Order

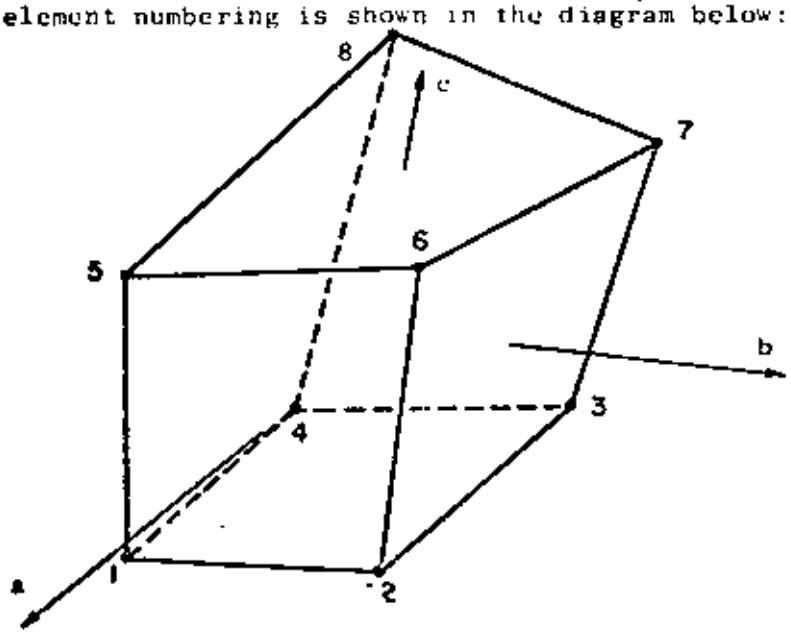
Computation time (for element stiffness) increases with the third power of the integration order. Therefore, the smallest satisfactory order should be used. This is found to be:

2 for rectangular element
3 for skewed element
4 may be used if element is extremely distorted in shape, but not recommended.

Mesh should be selected to give "rectangular" elements as far as possible.

(3) Element Coordinate System

Local element coordinate system is a natural system for this element in which the element maps onto a cube. Local element numbering is shown in the diagram below:



(4) Identification of Element Faces

Element faces are numbered as follows:

Face 1 corresponds to + a direction
2 corresponds to - a direction
3 corresponds to + b direction
4 corresponds to - b direction
5 corresponds to + c direction
6 corresponds to - c direction
0 corresponds to the center of the element

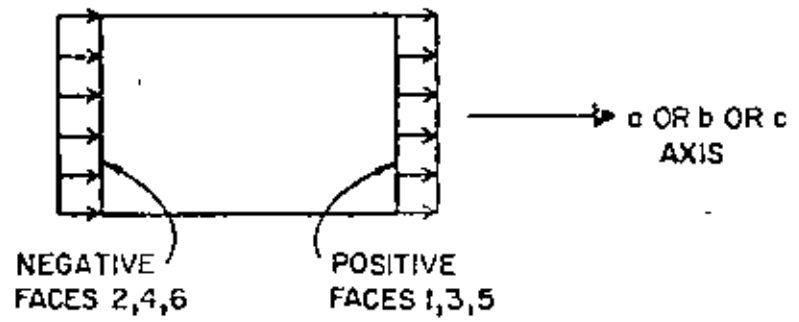
} Faces 1,3,5 are positive faces

} Faces 2,4,6 are negative faces

(5) Distributed Surface Loads

Two types of surface loadings may be specified; load type 1 (LT = 1), uniformly distributed surface load and load type 2 (LT = 2), hydrostatically varying surface pressure (but not surface tension). Both loading types are for loads normal to the surface and do not include surface shears. Surface loadings that do not fall into these categories must be input as nodal loads on the concentrated load data cards (see Section V).

(1) LT = 1: A positive surface load acts in the direction of the outward normal of a positive element face and along the inward normal of a negative element face as shown in the following diagram.



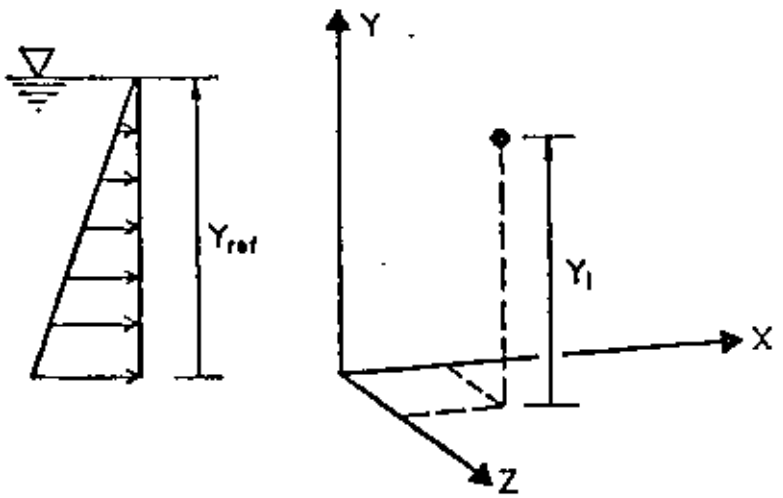
POSITIVE SURFACE LOADING P

If the uniformly distributed surface loading P is input as a positive quantity then it describes pressure loading on faces 2, 4 or 6 and tensile loading on faces 1, 3 or 5. If P is input as a negative quantity then it describes tensile loading on faces 2, 4 or 6 and pressure on faces 1, 3 or 5.

(2) LT = 2: A hydrostatically varying surface pressure on element faces 2, 4 or 6 can be specified by a reference fluid surface and a fluid weight density γ as input. Only one hydrostatic surface pressure card need be input in order to specify a hydrostatic loading on the complete structure. The consistent nodal loads are calculated by the program as follows. At each numerical integration point "i" on an element surface the pressure P_i is calculated from

$$P_i = \gamma (Y_i - Y_{ref}) \quad .$$

where Y_i is the global Y coordinate of the point in question and Y_{ref} specifies the fluid surface assuming gravity acts along the -Y axis



If $P_i > 0$, corresponding to surface tension, the contribution is ignored. If an element face is such that $Y_i > Y_{ref}$ for all i (16 integration points are used by program) then nonodal loads will be applied to the element. If some $P_i > 0$ and some $P_i < 0$ for a particular face, then approximate nodal loads are obtained for the partially loaded surface.

(6). Thermal Loads

Thermal loads are computed assuming a constant temperature increase ΔT throughout the element.

$\Delta T = T_{avg} - T_o$

T_{avg} = the average of the 8 nodal point temperatures specified on nodal point data cards

T_o = stress free element temperature specified on the element card.

(7). Element Load Cases

Element load case A consists of all the contributions from distributed loadings, thermal loadings and gravity loading for all the elements taken collectively.

Load case A = Σ (PA × pressure loading
+ TA × thermal loading
+ GXA × gravity X loading
+ GYA × gravity Y loading
+ GZA × gravity Z loading)

Element load case A for the set of three dimensional solid elements is added to element load case A for the other element types in the analysis. The treatment of element load cases B, C and D is analogous to that of element load case A. The loading cases for the structure are obtained by adding linear combinations of element load cases A, B, C and D to the nodal loads specified on the joint data cards.

(8) Output of Element Stresses

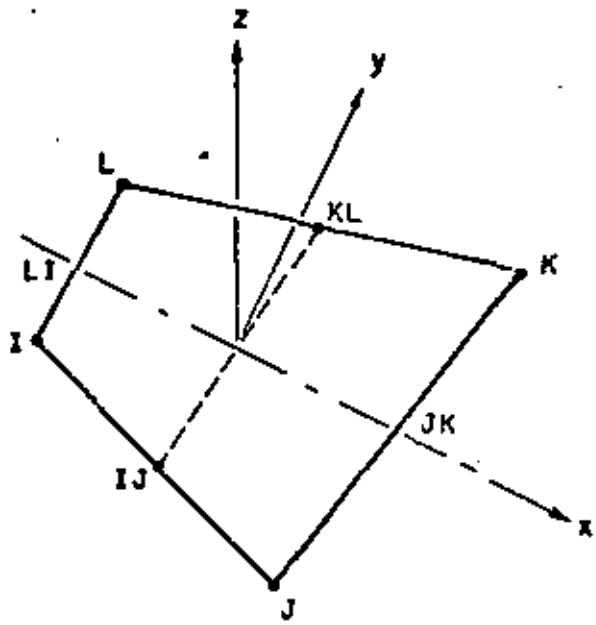
1. At the centroid of the element, stresses are referred to the global axes. Three principal stresses are also presented.

2. At the center of an element face, stresses are referred to a set of local axes (x,y,z). These local axes are individually defined for each face as follows: Let nodal points I, J, K and L be the four corners of the element face. Then

   x is specified by LI - JK, where LI and JK are midpoints of sides L-I and J-K,

   z is normal to x and to the line joining midpoints IJ and KL.

   y is normal to x and z, to complete the right-handed system.



The corresponding nodal points I, J, K and L in each face are given in the table.

FACE	NODAL POINTS			
	I	J	K	L
1	1	2	6	5
2	4	3	7	8
3	3	7	6	2
4	4	8	5	1
5	8	5	6	7
6	4	1	2	3

Two surface principal stresses and the angle between the algebraically largest principal stress and the local x axis are printed with the output. It is optional to choose one or two locations of an element where stresses are to be computed. In the output, "face zero" designates the centroid of the element.

IV. ELEMENT DATA (continued)

TYPE 6 - PLATE AND SHELL ELEMENTS (QUADRILATERAL)

A. Control Card (3I5)

Columns		
1 - 5	The number 6
6 - 10	Number of shell elements
11 - 15	Number of different materials

B. Material Property Information

Anisotropic material properties are possible. For each different material, two cards must be supplied.

Card 1: (I10,20X,4F10.0)

Columns	
1 - 10	Material identification number
31 - 40	Mass density
41 - 50	Thermal expansion coefficient α_x
51 - 60	Thermal expansion coefficient α_y
61 - 70	Thermal expansion coefficient α_{xy}

Card 2: (6F10.0)

Columns	
1 - 10	Elasticity element C_{xx}
11 - 20	Elasticity element C_{xy}
21 - 30	Elasticity element C_{xs}
31 - 40	Elasticity element C_{yy}
41 - 50	Elasticity element C_{ys}
51 - 60	Elasticity element G_{xy}

Elements in plane stress material matrix [C]

$$\begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \tau_{xs} \end{Bmatrix} = \begin{bmatrix} C_{xx} & C_{xy} & C_{xs} \\ C_{xy} & C_{yy} & C_{ys} \\ C_{xs} & C_{ys} & G_{xy} \end{bmatrix} \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \gamma_{xy} \end{Bmatrix}$$

C. Element Load Multipliers (5 cards)

Card 1: (4F10.0)

Columns	
1 - 10	Distributed lateral load multiplier for load case A
11 - 20	Distributed lateral load multiplier for load case B
21 - 30	Distributed lateral load multiplier for load case C
31 - 40	Distributed lateral load multiplier for load case D

Card 2: (4F10.0)

Columns	
1 - 10	Temperature multiplier for load case A
11 - 20	Temperature multiplier for load case B
21 - 30	Temperature multiplier for load case C
31 - 40	Temperature multiplier for load case D

Card 3: (4F10.0)

Columns	
1 - 10	X-direction acceleration for load case A
11 - 20	X-direction acceleration for load case B
21 - 30	X-direction acceleration for load case C
31 - 40	X-direction acceleration for load case D

Card 4: (4F10.0) Same as Card 3 for Y-direction

Card 5: (4F10.0) Same as Card 3 for Z-direction

D. Element Cards (8I5,F10.0)

One card for each element

Columns		
1 - 5	Element number
6 - 10	Node I
11 - 15	Node J
16 - 20	Node K
21 - 25	Node L
26 - 30	Node O
31 - 35	Material identification (if left blank, taken as one)
36 - 40	Element data generator K_D
41 - 50	Element thickness
51 - 60	Distributed lateral load (pressure)
61 - 70	Mean temperature variation T from the reference level in undeformed position
71 - 80	Mean temperature gradient dT/z across the shell thickness (a positive temperature gradient produces a negative curvature).

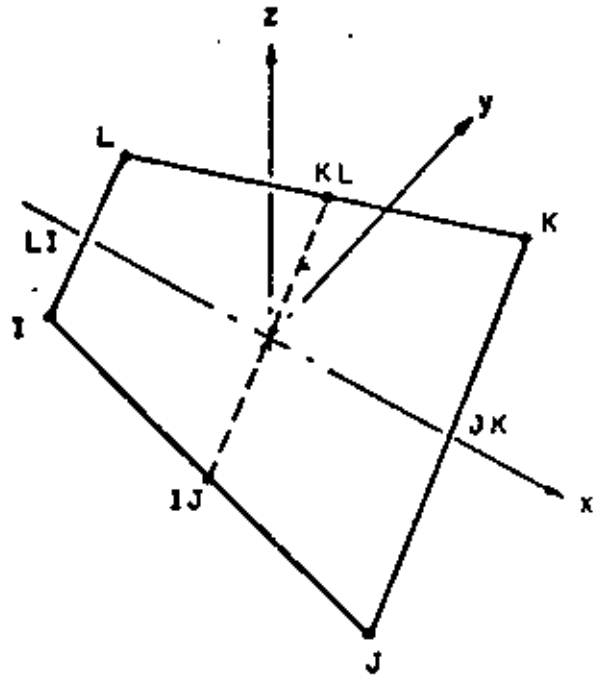
NOTES/

(1) Nodal Points and Coordinate Systems

The nodal point numbers I, J, K and L are in sequence in a counter-clockwise direction around the element. The local element coordinate system (x, y, z) is defined as follows:

- x Specified by LI - JK, where LI and JK are midpoints of sides L-I and J-K.
- z Normal to x and to the line joining midpoints IJ and KL.
- y Normal to x and z to complete the right-handed system.

This system is used to express all physical and kinematic shell properties (stresses, strains, material law, etc.), except that the body force density is referred to the global coordinate system (X, Y, Z).



For the analyses of shallow shells, rotational constraints normal to the surface may be imposed by the addition of boundary elements at the nodes (element type #7).

(2) Node 0

When columns 26 - 30 are left blank, mid-node properties are computed by averaging the four nodes.

(3) Element Data Generation

Element cards must be in element number sequence. If element cards are omitted, the program automatically generates the omitted information as follows:

The increment for element number is one

$$\text{i.e.} \quad NE_{i+1} = NE_i + 1$$

The corresponding increment for nodal number is K_n

$$\text{i.e.} \quad NI_{i+1} = NI_i + K_n$$

$$NJ_{i+1} = NJ_i + K_n$$

$$NK_{i+1} = NK_i + K_n$$

$$NL_{i+1} = NL_i + K_n$$

Material identification, element thickness, distributed lateral load, temperature and temperature gradient for generated elements are the same. Always include the complete last element card.

IV. ELEMENT DATA (continued)

(4) Element Stress Calculations

Output are moments per unit length and membrane stresses.

IV. ELEMENT DATA (continued)

TYPE 7 - BOUNDARY ELEMENTS

This element is used to constrain nodal displacements to specified values, to compute support reactions and to provide linear elastic supports to nodes. If the boundary condition code for a particular degree of freedom is specified as 1 on the structure nodal point data cards, the displacement corresponding to that degree of freedom is zero and no support reactions are obtained with the printout. Alternatively, a boundary element can be used to accomplish the same effect except that support reactions are obtained since they are equal to the member end forces of the boundary elements which are printed. In addition the boundary element can be used to specify non-zero nodal displacements in any direction which is not possible using the nodal point data cards.

The boundary element is defined by a single directed axis through a specified nodal point, by a linear extensional stiffness along the axis or by a linear rotational stiffness about the axis. The boundary element is essentially a spring which can have axial displacement stiffness and axial rotational stiffness. There is no limit to the number of boundary elements which can be applied to any joint to produce the desired effects. Boundary elements have no effect on the size of the stiffness matrix.

INPUT DATA

A. Control Card (2I5)

Columns	1 - 5	The number 7.
	6 - 10	Total number of boundary elements.

B. Element Load Multipliers (4F10.0)

Columns	1 - 10	Multiplier for load case A
	11 - 20	Multiplier for load case B
	21 - 30	Multiplier for load case C
	31 - 40	Multiplier for load case D

C. Element Cards (8I5,3F10.0)

One card per element (in ascending nodal point order) except where automatic element generation is used.

Columns	1 - 5	Node N, at which the element is placed
	6 - 10	Node I
	11 - 15	Node J — Leave columns 11 - 25 blank if only node I is needed.
	16 - 20	Node K — Leave columns 11 - 25 blank if only node I is needed.
	21 - 25	Node L — Leave columns 11 - 25 blank if only node I is needed.
	26 - 30	Code for displacement
	31 - 35	Code for rotation
	36 - 40	Data generator K_n
	41 - 50	Specified displacement along element axis
	51 - 60	Specified rotation about element axis
	61 - 70	Spring stiffness (set to 10^{10} if left blank) for both extension and rotation.

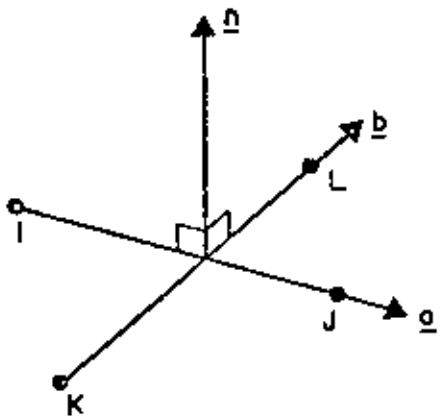
NOTES/

(1) Direction of boundary element

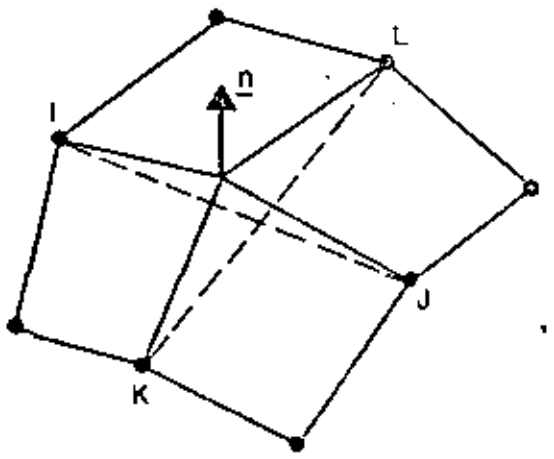
The direction of the boundary element at node N is specified in one of two ways.

(i) A second nodal point I defines the direction of the element from node N to node I.

(ii) Four nodal points I, J, K and L specify the direction of the element as the normal to the plane defined by two intersecting straight lines (vectors $\underline{a}$ and $\underline{b}$, see Fig. below).



$\underline{n} = \underline{a} \times \underline{b}$



ROTATIONAL CONSTRAINT IN THIN SHELL ANALYSIS

The four points I, J, K and L need not be unique. A useful application for the analysis of shallow thin shells employs the boundary element to approximate rotational constraint about the surface normal as shown above.

$\underline{n}$ is given by the vector cross product $\underline{n} = \underline{a} \times \underline{b}$ and defines the direction of the boundary element.

Note that node I in case (i) and nodes I, J, K and L in case (ii) are used only to define the direction of the element and if convenient may be any nodes used to define other elements. However 'artificial nodes' may be created to define directions of boundary elements. These 'artificial nodes' are input on the nodal point data cards with their coordinates and with all the boundary condition codes specified as 1 (one).

IV. ELEMENT DATA (continued)

It should be noted that node N is the structure node to which the boundary element is attached. In case (i), a positive displacement moves node N towards node I. Correspondingly, a positive force in the element means compression in the element. In case (ii), a positive displacement moves node N into the direction n (see Fig.).

(2) Displacement and rotation codes

Displacement code = 1: When this code is used, the displacement δ, specified in columns 41-50, and the spring stiffness k, specified in columns 61-70, are used by the program in the following way. The load P, evaluated from $P = k\delta$, is applied to node N in the direction node N to node I in case (i) and into direction n in case (ii), if δ is positive. If k is much greater than the stiffness of the structure at node N without the boundary element, then the net effect is to produce a displacement very nearly equal to δ at node N. If $\delta = 0$, then $P = 0$ and the stiff spring approximates a rigid support. Note that the load P will contribute to the support reaction for nonzero δ. The boundary condition codes specified on the structure nodal point data cards must be consistent with the fact that a load P is being applied to node N to effect the desired displacement (even when this displacement is zero).

Rotation code = 1: This case is analogous to the situation described above. A torque T, evaluated from $T = k\theta$, is applied to node N about the axis (direction) of the element. The rotation θ is specified in columns 51-60.

(3) Data generator K_n

When a series of nodes are such that:

(i) All have identical boundary elements attached
(ii) All boundary elements have same direction
(iii) All specified displacements and rotations are identical
(iv) The nodal sequence forms an arithmetic sequence, i.e., N, $N + K_n$, $N + 2K_n$ etc.,

then only the first and last node in the sequence need be input. The increment K_n is input in columns 36-40 of the first card.

(4) Element load multipliers

Each of the four possible element load cases A, B, C and D associated with the boundary elements consists of the complete set of displacements as specified on the boundary element cards multiplied by the element load multiplier for the corresponding load case. As an example, suppose that displacement of node N is specified as 1.0, spring stiffness as 10^{10} and no other boundary element displacements are specified. Let case A multiplier be 0.0 and case B multiplier be 2.0. For element load case A the specified displacement is 0.0 x 1.0 = 0.0 while that for B is 2.0 x 1.0 = 2.0. Linear combinations of element load cases A, B, C and D for all types of elements collectively for a particular problem are specified on the structure element load multiplier cards. As far as the boundary element is concerned, this device is useful when a particular node has a support displacement in one load case but is fixed in others.

(5) Recommendations for use of boundary elements

If a boundary element is aligned with a global displacement direction, only the corresponding diagonal element in the stiffness matrix is modified. Therefore, no stiffness matrix ill-conditioning results. However, when the boundary element couples degrees of freedom, large off-diagonal elements introduce ill-conditioning into the stiffness matrix which can cause solution difficulties.

In the analysis of shallow shells boundary elements with stiffness a fraction of the element bending stiffness should be used (say less than or about 10%).

In dynamic analysis "artificially stiff" boundary elements should not be used. (See note (8) in Section VII.A).

TYPE 8 - VARIABLE-NUMBER-NODES THICK SHELL AND THREE-DIMENSIONAL ELEMENTS

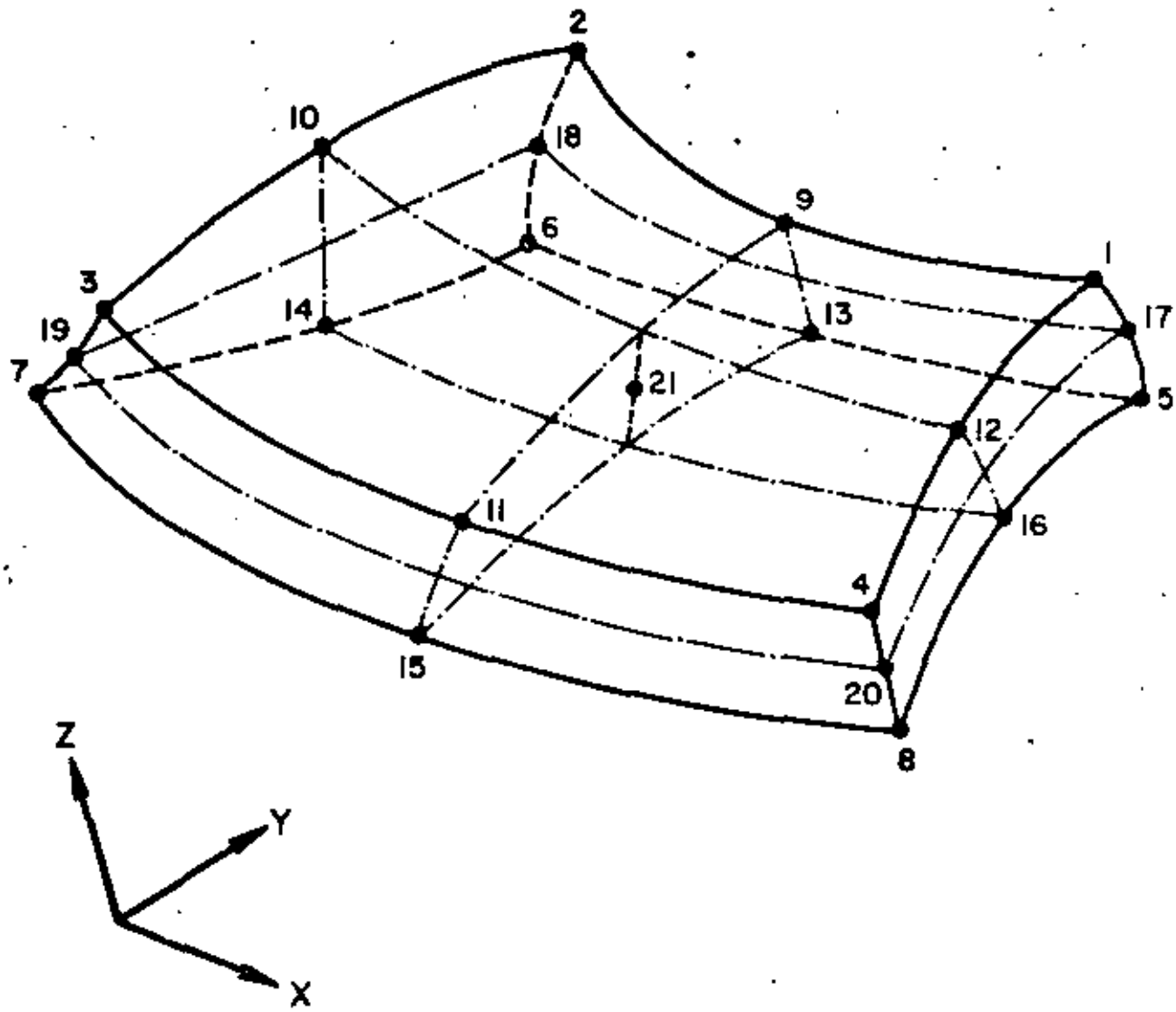
A minimum of 8 and a maximum of 21 nodes are used to describe a general three dimensional isoparametric element; the element is used to represent orthotropic, elastic media. The element type is identified by the number eight (8). Three translational degrees of freedom are assigned to each node, and at least the eight corner nodes must be input to define a hexahedron. Input of nodes 9 to 21 is optional; the figures below illustrate some of the most commonly used node combinations.

Element load cases (A,B,C,...) are formed from combinations of applied surface pressure, hydrostatic loads, inertia loads in the three directions X,Y,Z and thermal loads. Six global stresses are output at up to seven (7) locations within the element; these output locations are selected by means of appropriate data entries.

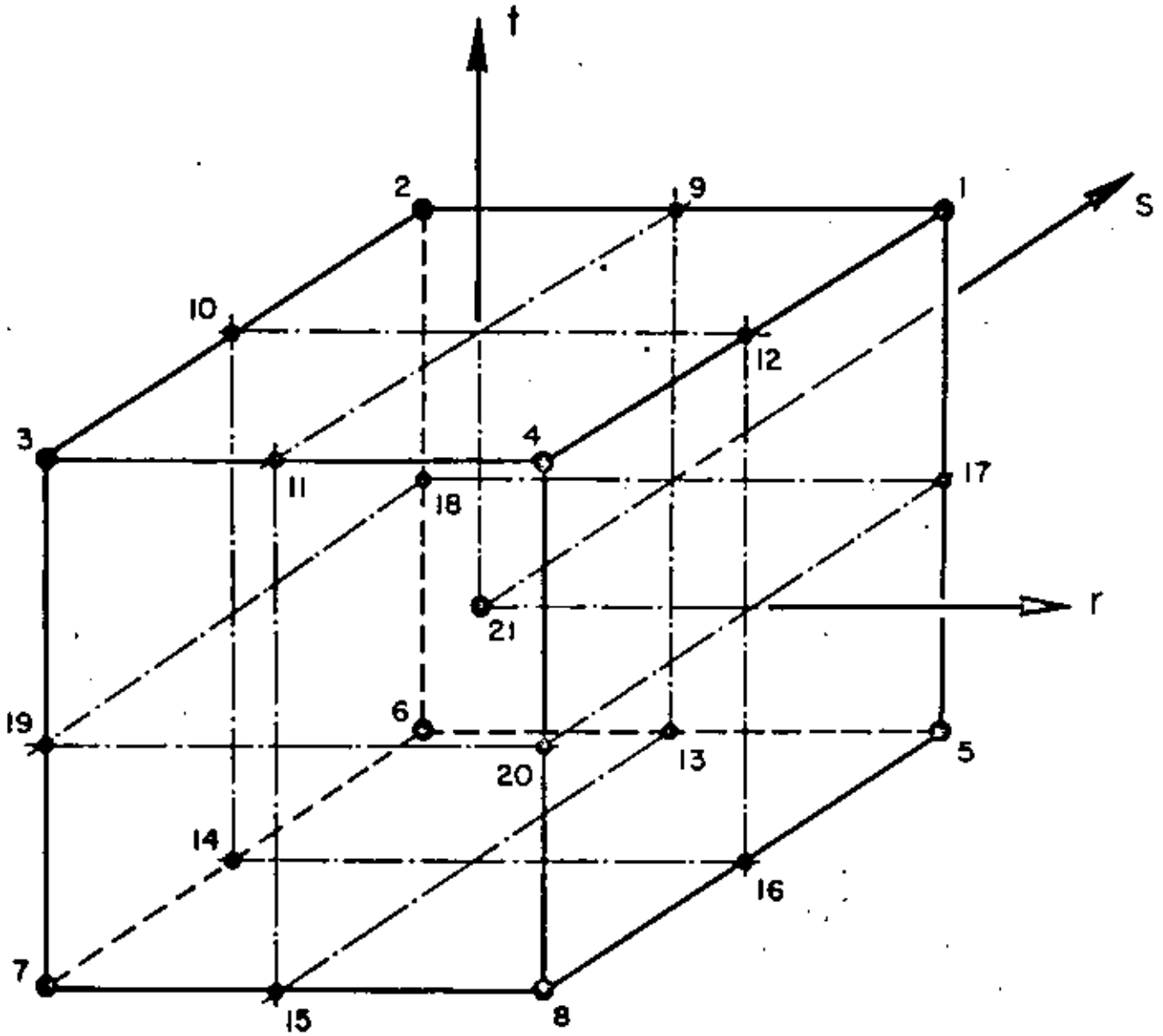
Node temperatures input in Section III are used to form an average element temperature, which is the basis of material property selection for the element. If thermal loads are applied, node temperatures are used to establish the temperature field within the element, and the temperature interpolation functions are the same as those assumed to represent element displacements.

1. Control Card (1015)

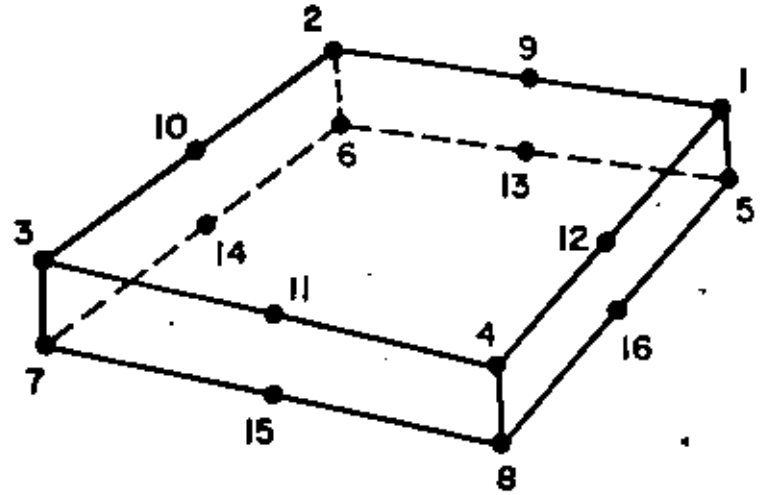
notes	columns	variable	entry
	5		Enter the number "8"
	6 - 10	NSOL21	Number of solid elements; GE.1
	11 - 15	NUMMAT	Number of different materials; GE.1
(1)	16 - 20	MAXTP	Maximum number of temperature points used in the table for any material; EQ.0; default set to "1"
(2)	21 - 25	NORTHO	Number of different sets of material axis orientation data; EQ.0; all properties are defined in the X,Y,Z, system
(3)	26 - 30	NDLS	Number of different distributed load (i.e., pressure) sets
(4)	31 - 35	MAXNOD	Maximum number of nodes used to describe any one element; GE.8 and LE.21 EQ.0; default set to "21"
(5)	36 - 40	NOPSET	Number of sets of data requesting stress output at various element locations: EQ.0; centroid output only



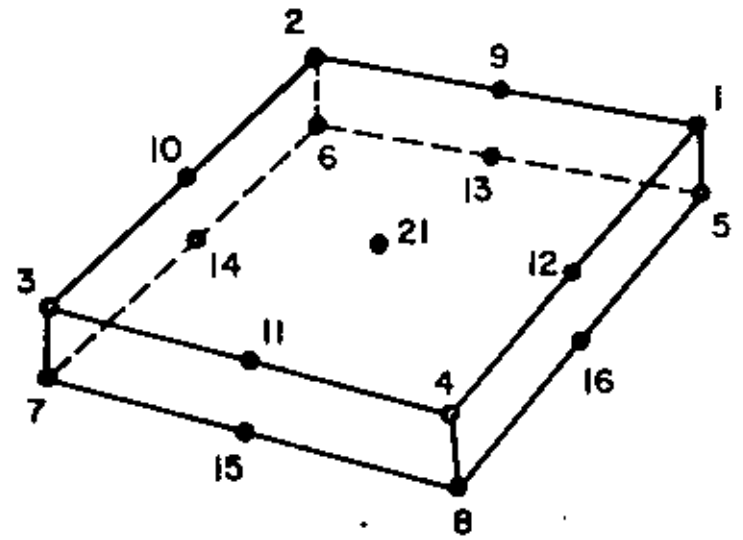
THREE DIMENSIONAL ISOPARAMETRIC ELEMENT



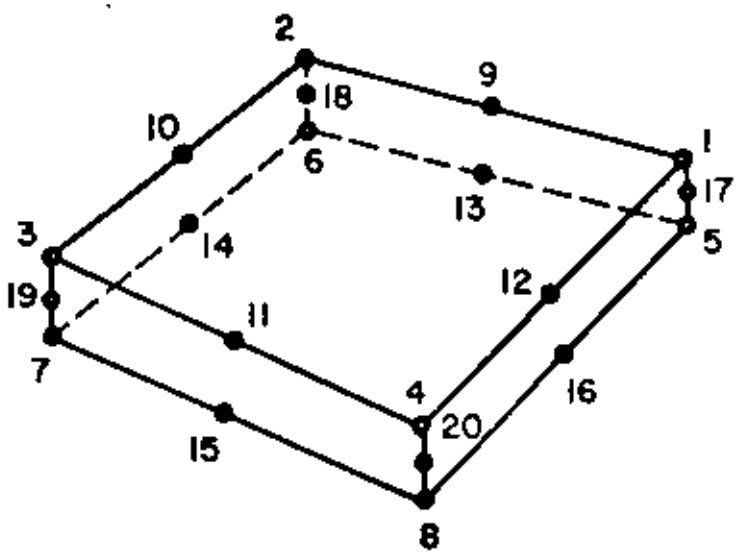
HEXAHEDRAL ELEMENT IN NATURAL COORDINATES



a. 16 – NODE ELEMENT



b. 17 – NODE ELEMENT



c. 20 – NODE ELEMENT

COMMONLY USED ELEMENT GEOMETRIES

IV. ELEMENT DATA (continued)

1. Control Card (1015) (continued)

notes	columns	variable	entry
(6)	41 - 45	INTRS	Standard integration order for the natural (r,s) directions; GE.2 and LE.4 EQ.0; default set to "2"
	46 - 50	INTT	Standard integration order for the natural (t)-direction; GE.2 and LE.4 EQ.0; default set to "2"

NOTES/

(1) The variable MAXTP limits the number of temperature points that can be input for any one of the NUMMAT material sets; i.e., the variable NTP in Section 2 cannot exceed the value of MAXTP.

(2) NORTHO specifies the number of cards to be read in Section 3, and if omitted, all orthotropic material axes are assumed to coincide with the global cartesian axes X,Y,Z.

(3) NDLS specifies the number of card pairs to be read in Section 4. NDLS must be a positive integer if any pressure loads are to be applied to solid element faces.

(4) MAXNOD specifies the maximum number of non-zero node numbers assigned to any one of the NSOL21 elements input in Section 7. Locations of the element's 21 possible nodes are shown in the figure below in which the element is shown mapped into its natural r,s,t coordinate system. The eight corner nodes must be input for every element, and nodes 9 to 21 are input optionally. If MAXNOD is 9 or greater, all 21 node entries are read for each element (Cards 2 and 3, Section 7), but only the first MAXNOD non-zero entries encountered when reading in sequence from 1 to 21 will be used for element description. As an example, for the 16-17- and 20-node elements MAXNOD has values of 16, 17, 20, respectively.

(5) As a means of controlling the amount of solution output, stress output location sets are defined in Section 5, and the total number of these output requests is specified by the variable NOPSET. For the case of NOPSET.EQ.0, no data is input in Section 5, and the only stress output produced by the program is at the element centroid. Otherwise, stress output can be requested at up to seven (7) locations (selected from a table of 27 possible locations) by means of the data entries given in Section 5.

NOTES (continued)

(6) The entries INTRS and INTT control the number of integration points to be used in numerical evaluation of integrals over volumes in the (r,s) and (t)-coordinate directions, respectively. When solid elements are used to represent shell structures, the through-the-thickness integrations (i.e., in the natural t-axis direction) can be evaluated less accurately than those in-plane (i.e., in the r,s plane). For this case INTRS might be 3 and INTT would be chosen typically as 2. The entries INTRS and INTT are standard or reference values and are used if the integration order entries on the element cards (Card 1, Section 7) are omitted. Non-zero entries for integration order(s) given on the element cards over-ride the standard values posted on this card.

2. Material Property Cards

Orthotropic, temperature dependent material properties are allowed. For each different material that is requested on the Control Card, the following set of data must be supplied (i.e., NUMMAT sets total):

a. Material identification card (2I5,2F10.0,6A6)

notes	columns	variable	entry
(1)	1 - 5	M	Material identification number; GE.1 and LE.NUMMAT
	6 - 10	NTP	Number of different temperatures at which properties are given; LE.MAXTP EQ.0; default set to "1"
(2)	11 - 20	WTDEN	Weight density of the material used to computed static gravity loads
	21 - 30	MASSDN	Mass density of the material used to compute the mass matrix in a dynamic analysis; EQ.0; default set to "WTDEN/386.4"
	31 - 66		Material description used to label the output.

NOTES/

(1) Material numbers (M) must be input in ascending sequence beginning with "1" and ending with "NUMMAT"; omissions or repetitions are illegal.

(2) Weight density is used to compute static node forces due to applied gravity loads; mass density is used to calculate element mass matrices for use in connection with a dynamic analysis.

IV. ELEMENT DATA (continued)

b. Material cards (7F10.0,6F10.0)

NTP pairs of cards are input in order of algebraically increasing value of temperature.

First Card

notes	columns	variable	entry
(1)	1 - 10		Temperature, T_n
(2)	11 - 20		E_{11} at T_n
	21 - 30		E_{22} at T_n
	31 - 40		E_{33} at T_n
	41 - 50		ν_{12} at T_n
	51 - 60		ν_{13} at T_n
	61 - 70		ν_{23} at T_n

Second Card

notes	columns	variable	entry
	1 - 10		G_{12} at T_n
	11 - 20		G_{13} at T_n
	21 - 30		G_{23} at T_n
	31 - 40		α_1 at T_n
	41 - 50		α_2 at T_n
	51 - 60		α_3 at T_n

NOTES/

(1) The 12 entries following the temperature value T_n are physical properties known at T_n. When two or more temperature points describe a material, interpolation based on average element temperature is performed to establish a property set for the element. Hence, the range of temperature points for a material table must span the expected range of average element temperatures for all elements associated with the material.

(2) The 12 constants $(E_{11}, E_{22}, \ldots, \alpha_3)$ are defined with respect to a set of axes (X_1, X_2, X_3) which are the principal material directions for an orthotropic, elastic medium. The stress-strain relations with respect to the (X_1, X_2, X_3) system is written as follows :

$$\begin{bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ \gamma_{12} \\ \gamma_{23} \\ \gamma_{31} \end{bmatrix} = \begin{bmatrix} 1/E_{11} & -\nu_{12}/E_{22} & -\nu_{13}/E_{33} & 0 & 0 & 0 \\ -\nu_{21}/E_{11} & 1/E_{22} & -\nu_{23}/E_{33} & 0 & 0 & 0 \\ -\nu_{31}/E_{11} & -\nu_{32}/E_{22} & 1/E_{33} & 0 & 0 & 0 \\ 0 & 0 & 0 & 1/G_{12} & 0 & 0 \\ 0 & 0 & 0 & 0 & 1/G_{23} & 0 \\ 0 & 0 & 0 & 0 & 0 & 1/G_{13} \end{bmatrix} \begin{bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \tau_{12} \\ \tau_{23} \\ \tau_{31} \end{bmatrix}$$

$$- \begin{bmatrix} \Delta T\alpha_1 & \Delta T\alpha_2 & \Delta T\alpha_3 & 0 & 0 & 0 \end{bmatrix}^T$$

where ε_{ii} and σ_{ii} are normal strains and stresses in the X_i directions; γ_{ij} and τ_{ij} are shear strains and stresses on the principal material planes; α_i are the coefficients of thermal expansion, and ΔT is the increase in temperature from stress free distributed over the element volume.

3. Material Axes Orientation Sets (615)

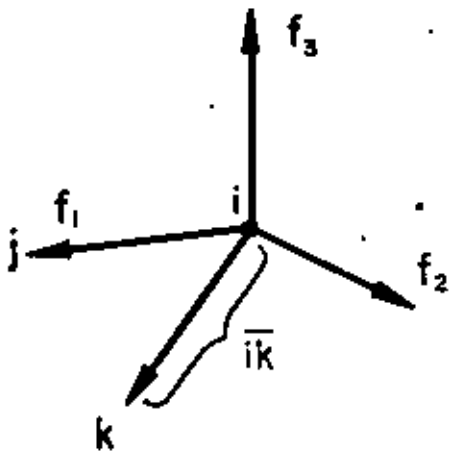
If NORTHO is zero on the Control Card, skip this data section, and all material axes (X_1,X_2,X_3) will be assumed to coincide with the global cartesian system X,Y,Z. Otherwise, NORTHO cards must be input as follows:

notes	columns	variable	entry
(1)	1 - 5	M	Identification number; GE.1 and LE.NORTHO
(2)	6 - 10	NI	Node number for point "i"
	11 - 15	NJ	Node number for point "j"
	16 - 20	NK	Node number for point "k"

NOTES/

(1) Identification numbers (M) must be input in increasing sequence beginning with "1" and ending with "NORTHO".

(2) Orthotropic material axes orientations are specified by means of the three node numbers NI,NJ,NK. For the special case where orthotropic material axes coincide with the global axes (X,Y,Z), it is not necessary to input data in this section; see Section 7, note (4). Let $\underline{f}_1,\underline{f}_2,\underline{f}_3$ be the three orthogonal vectors which define the axes of material orthotropy, then their directions are as shown below:



$$\underline{f}_1 = \vec{ij}$$

$$\underline{f}_3 = \vec{ij} \times \vec{ik}$$

$$\underline{f}_2 = \underline{f}_3 \times \underline{f}_1$$

Node numbers NI,NJ,NK are only used to locate points i,j,k, respectively, and any convenient nodes may be used.

4. Distributed Surface Load Data

NDLS pairs of cards are to be input in this section in order of increasing set number (N). These data describe surface loads acting on element faces and may be prescribed directly in terms of face corner node pressures or indirectly by means of a hydrostatic pressure field.

a. Control Card (315)

notes	columns	variable	entry
(1)	1 - 5	N	Load set identification number; GE.1 and LE.NDLS
(2)	6 - 10	NFACE	Element face number on which this distributed load is acting; GE.1 and LE.6
(3)	11 - 15	LT	Load type code; EQ.1; prescribed normal pressure intensities EQ.2; hydrostatically varying pressure field EQ.0; default set to "1"

NOTES/

(1) The surface load data sets established in this section are assigned to the elements in Section 7.

(2) Hexahedra have six quadrilateral faces each uniquely described by four node numbers at the corners of the face. The face number convention established for elements is given in the Table below.

(3) Two types of surface pressure loads may be applied to faces of the elements. If LT.EQ.0 (or 1), a normal pressure distribution is prescribed directly by means of pressure intensities at the face corner nodes. If LT.EQ.2, the face is exposed to hydrostatic pressure due to fluid head.

FACE NUMBER	NATURAL COORDINATES	CORNER NODE NUMBERS N_1	N_2	N_3	N_4
1	(+1, s, t)	1	4	8	5
2	(-1, s, t)	2	3	7	6
3	(r,+1, t)	1	5	6	2
4	(r,-1, t)	4	8	7	3
5	(r, s,+1)	1	2	3	4
6	(r, s,-1)	5	6	7	8

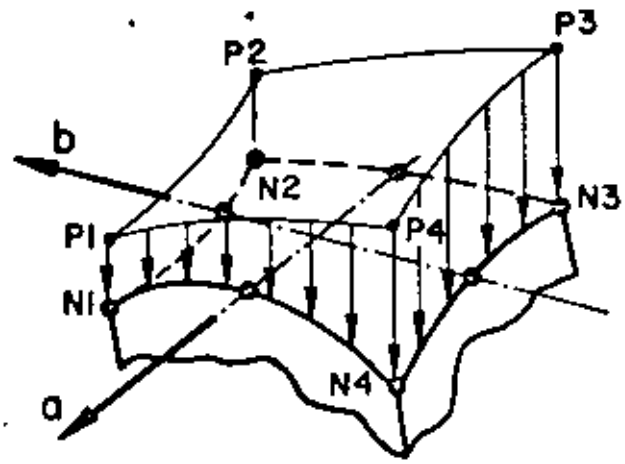
TABLE Corner Node Numbers for the Solid Element Faces

b. Normal Pressure Data (4F10.0) (LT.EQ.1, only)

notes	columns	variable	entry
(1)	1 - 10	P1	Pressure at face node N_1
(2)	11 - 20	P2	Pressure at face node N_2; EQ.0; default set to "P1"
	21 - 30	P3	Pressure at face node N_3; EQ.0; default set to "P1"
	31 - 40	P4	Pressure at face node N_4; EQ.0; default set to "P1"

NOTES/

(1) The pressure distribution acting on an element face is defined by specifying intensities P1,P2,P3,P4 at the face corner nodes as shown below:



The face corner node numbers are given in the Table and positive pressure tends to compress the volume of the element.

The variation of pressure over the element face, p(a,b), is given as:

$$p(a,b) = P1 \times h_1 + P2 \times h_2 + P3 \times h_3 + P4 \times h_4$$

where

$$h_1 = (1/4)\ (1+a)\ (1+b)$$
$$h_2 = (1/4)\ (1-a)\ (1+b)$$
$$h_3 = (1/4)\ (1-a)\ (1-b)$$
$$h_4 = (1/4)\ (1+a)\ (1-b)$$

in quadrilateral natural face coordinates (a,b).

(2) If any of the entries P2,P3,P4 are omitted, these values are re-set to the value of P1; i.e., for a uniformly distributed pressure (p), we have P1.EQ.p and cc 11-40 blank. If P2 is zero specify a small number.

IV. ELEMENT DATA (continued)

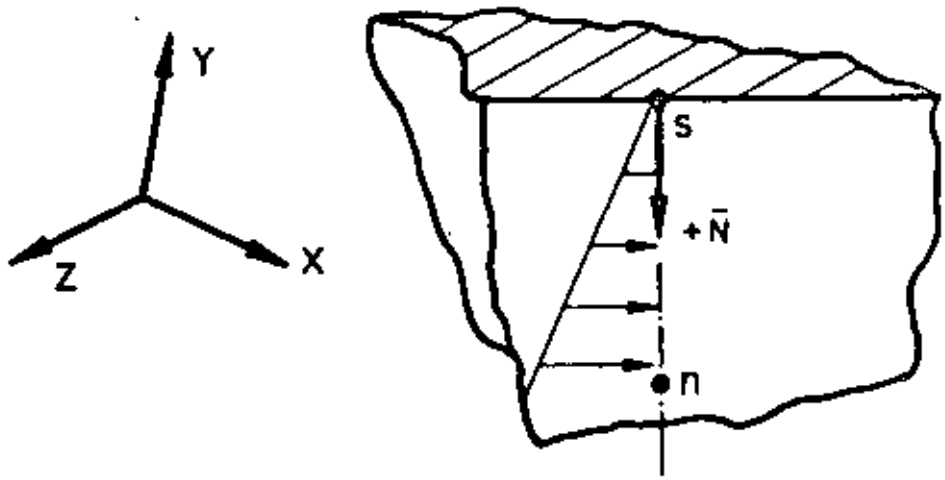
c. Hydrostatic Pressure Data (7F10.0) (LT.EQ.2, only)

notes	columns	variable	entry
(1)	1 - 10	GAMMA	Weight density of the fluid, γ; GT.0
(2)	11 - 20	XS	X-ordinate of point s in the free surface of the fluid
	21 - 30	YS	Y-ordinate of point s in the free surface of the fluid
	31 - 40	ZS	Z-ordinate of point s in the free surface of the fluid
	41 - 50	XN	X-ordinate of a point n on the normal to the fluid surface
	51 - 60	YN	Y-ordinate of a point n on the normal to the fluid surface
	61 - 70	ZN	Z-ordinate of a point n on the normal to the fluid surface

NOTES/

(1) GAMMA is the weight density (i.e., units of force per unit of fluid volume) of the fluid in contact with element face number NFACE.

(2) Point "s" is any point in the free surface of the fluid, and point "n" is located such that the direction from s to n is normal to the free surface and is positive with increasing depth.



Hydrostatic pressure in contact with an element face causes element compression; i.e., pressure resultant acts toward the element centroid. Nodes located above the fluid surface are automatically assigned zero pressure intensities if an element face is not (or only partially) submerged in the fluid.

5. Stress Output Request Location Sets (7I5)

If NOPSET is zero on the Control Card, skip this section, and global stresses will be computed and output at the element centroid only. Otherwise, NOPSET cards must be input as follows:

notes	column	variable	entry
(1)	1 - 5	LOC1	Location number of output point 1
	6 - 10	LOC2	Location number of output point 2
	11 - 15	LOC3	Location number of output point 3
	16 - 20	LOC4	Location number of output point 4
	21 - 25	LOC5	Location number of output point 5
	26 - 30	LOC6	Location number of output point 6
	31 - 35	LOC7	Location number of output point 7
			LE. 27

NOTES/

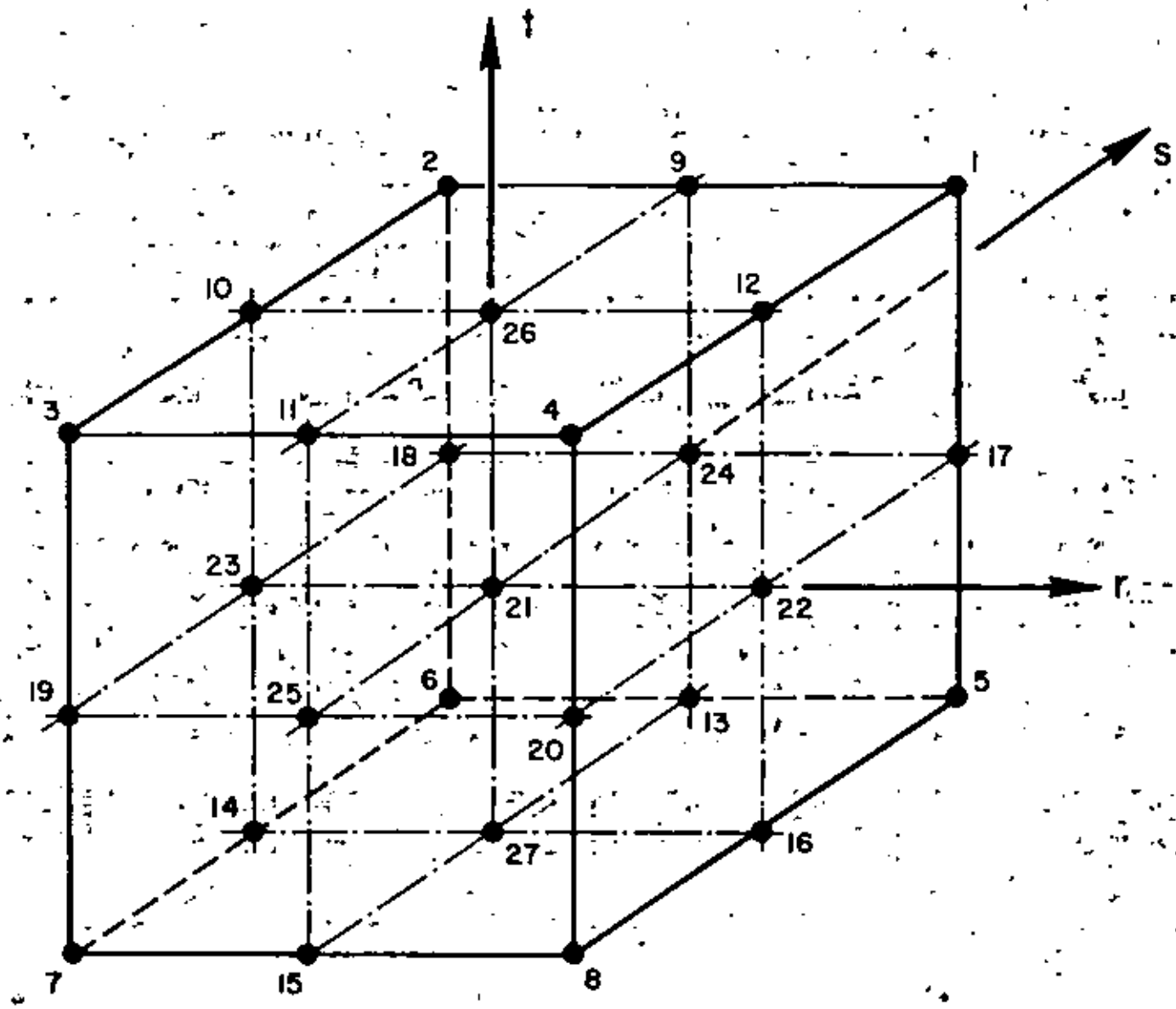
(1) 27 element locations are assigned numbers as shown in the Figure below. Locations 1 to 21 correspond to node numbers 1 to 21, respectively. Locations 22 to 27 are element face centroids. The first zero (or blank) entry on a location card terminates reading of location numbers for the output set; hence, fewer than seven locations can be requested in an output set. Location numbers must be input in order of increasing magnitude; i.e., LOC2 is greater than LOC1, LOC3 is greater than LOC2, etc. In dynamic analysis, FACE 1, FACE 2,..., FACE 6 correspond to output locations 22,23,...,27 respectively. (See Table VII.1).

6. Element Load Case Multipliers

Five (5) cards must be input in this section specifying the fraction of gravity (X,Y,Z), the fraction of thermal loads and the fraction of pressure loads to be added to each of the element loading combinations (A,B,...). Load case multiplier data affect static analysis calculations only.

Card 1. X-direction gravity (4F10.0)

notes	columns	variable	entry
(1)	1 - 10	GXA	Fraction of X-direction gravity to be applied in element load case A
	31 - 40	GXD	Fraction of X-direction gravity to be applied in element load case D



ELEMENT STRESS OUTPUT LOCATION NUMBERS

IV. ELEMENT DATA (continued)

Card 2 Y-direction gravity (4F10.0)

Card 3 Z-direction gravity (4F10.0)

Card 4 Thermal loads (4F10.0)

notes	columns	variable	entry
(2)	1 - 10	TA	Fraction of thermal loads to be applied in element load case A
		. . .	
	31 - 40	TD	Fraction of thermal loads to be applied in element load case D

Card 5 Pressure loads (4F10.0)

notes	columns	variable	entry
(3)	1 - 10	PA	Fraction of pressure loads to be applied in element load case A
		. . .	
	31 - 40	PD	Fraction of pressure loads to be applied in element load case D

NOTES/

(1) Gravity loads on the structure due to static body forces are computed from the weight density of element materials and the element geometry. These loads are assigned to the element load combinations by means of the entries on Cards 1,2 and 3 for forces in the X,Y,Z directions, respectively.

(2) Thermal loads are computed knowing the node temperatures input in Section III, the stress free reference temperature (T_o) input in Section 7 and the element's material properties and node coordinates. The temperature distribution within the element is described using the same interpolation functions which describe the variation of displacements within the element.

(3) Pressure loads are first assigned to element load cases (A,B,...) by means of the entries (scale factors) on Card 5, and the distributed load sets which were input in Section 4 are then applied to the elements individually for cases (A,B,...) by means of load set references given in Section 7.

7. Element Cards

Two cards (if MAXNOD.EQ.8) or three cards (if MAXNOD.GT.8) must be prepared for each element that appears in the input, and the

format for these cards is as follows:

Card 1 (6I5,F10.,4I5,4I2)

notes	columns	variable	entry
(1)	1 - 5	M	Element number; GE.1 and LE.NSOL2I
(2)	6 - 10	NDIS	Number of nodes to be used in describing the element's displacement field; EQ.0; default set to "MAXNOD"
(3)	11 - 15	NXYZ	Number of nodes to be used in the description of element geometry; EQ.0; default set to "NDIS" EQ.NDIS → isoparametric element LT.NDIS → subparametric element
	16 - 20	NMAT	Material identification number; GE.1 and LE.NUMMAT
(4)	21 - 35	MAXES	Identification number of the material axis orientation set; GE.1 and LE.NORTHO EQ.0; material axes default to the global X,Y,Z system
(5)	26 - 30	IOP	Identification number of the stress output location set; GE.1 and LE.NOPSET EQ.0; centroid output only
	31 - 40	TZ	Stress free reference temperature, T_o
(6)	41 - 45	KG	Node number increment for element data generation; EQ.0; default set to "1"
	46 - 50	NRSINT	Integration order for natural coordinate (r,s) directions; EQ.0; default set to "INTRS"
	51 - 55	NTINT	Integration order for natural coordinate (t) direction; EQ.0; default set to "INTT"
(7)	56 - 60	IREUSE	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
(8)	61 - 62	LSA	Pressure set for element load case A
	63 - 64	LSB	Pressure set for element load case B
	65 - 66	LSC	Pressure set for element load case C
	67 - 68	LSD	Pressure set for element load case D; LE.NDLS

IV. ELEMENT DATA (continued)

Card 2 (1615)

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

Card 3 (515) (required if MAXNOD.GT.8)

note	columns	variable	entry
	1 - 5		Node 17 number
	6 - 10		Node 18 number
	11 - 15		Node 19 number
	16 - 20		Node 20 number
	21 - 25		Node 21 number

NOTES/

(1) Element cards must be input in ascending element number order beginning with "1" and ending with "NSOL21". 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 (7).

(2) NDIS is a count of the node numbers actually posted on Cards 2 and 3 which must immediately follow Card 1. NDIS must be at least eight (8), but must be less than or equal to the limit (MAXNOD) which was given on the Control Card, Section 1. Element displacements are assigned at the NDIS non-zero nodes, and thus, the order of the element matrices is three (i.e., translations X,Y,Z) times NDIS. 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.

(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 NXYZ non-zero nodes posted on Cards 2 and 3 are used to evaluate those parameters which pertain to element geometry only. NXYZ must be at least eight (8), and if omitted is re-set to NDIS. A common application might be a 20 node element (i.e., NDIS.EQ.20) with straight edges, in which case NXYZ would be entered as "8".

(4) MAXES (unless omitted) refers to one of the material axes set defined in Section 3. If omitted, the material (NMAT) orientation is such that the (X_1,X_2,X_3) axes coincide with the (X,Y,Z) axes, respectively.

(5) IOP (unless omitted) refers to one of the output location sets given in Section 5. If IOP.EQ.0, stress output is quoted at the element centroid only. Stress output at a point consists of three normal and three shear components referenced to the global (X,Y,Z) axes.

(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;

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

The last element cannot be generated.

(7) The flag IREUSE 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 IREUSE.EQ.0, new matrices are computed for the current element. If IREUSE.EQ.1 it is also assumed that the node temperatures of the element (for calculation of thermal loads) are the same as those of the preceding element.

(8) Pressure loads are assigned (i.e., applied) to the element by means of load set references in cc 61-62 for combination A, cc 63-64 for B, etc. A zero entry means that no pressure acts on the element for that particular element load combination.

(9) The first eight node numbers establish the corners or vertices of a general hexahedron and must be all non-zero, (see Figure in Section 1 on control cards). Node numbers must be input in the sequence indicated otherwise volume and surface area integrations will be indefinite.

(10) The number of cards required as input for each element depends on the variable MAXNOD. For the case of MAXNOD.EQ.8, only Card 2 is required. If MAXNOD.GT.8, Cards 2 and 3 are required for all elements.

Nodes 9 to 21 are optional, and only those nodes actually used to describe the element are input. The program will read all 21 entries if MAXNOD was given as 9 or greater, but only NDIS non-zero values are expected to be read on Cards 2 and 3. If for example one element is described by 10 nodes, then cc 1-40 on Card 2 would be the eight corner node numbers, and the remaining two node numbers would be posted somewhere on Cards 2 and 3.

TYPE 9 - THREE-DIMENSIONAL STRAIGHT OR CURVED PIPE ELEMENTS

Pipe elements are identified by the number twelve (12). Axial and shear forces, torque and bending moments are calculated for each member. Gravity loadings in the global (X,Y,Z) directions, uniform temperature changes (computed from input nodal temperatures), and extensional effects due to internal pressure form the basic member loading conditions. Pipe element input is described by the following sequence of cards:

1. Control Card (14I5)

notes	columns	variable	entry
	4 - 5		Enter the number "12"
(1)	6 - 10	NPIPE	Number of pipe elements
	11 - 15	NUMMAT	Number of material sets
	16 - 20	MAXTP	Maximum number of temperature points used in the table for any material GE.1; at least one point
	21 - 25	NSECT	Number of section property sets; GE.1
(2)	26 - 30	NBRP	Number of branch point nodes at which output is required; EQ.0; no branch point output is produced
	31 - 35	MAXTAN	Maximum number of tangent elements common to any one branch point node; EQ.0; default set to "4"
	36 - 40	NPAR(8)	Blank
	41 - 45	NPAR(9)	Tangent stiffness load matrix dump flag EQ.1; Print EQ.0; Suppress printing
	46 - 50	NPAR(10)	Bend stiffness load matrix dump flag EQ.1; Print EQ.0; Suppress printing
	51 - 55	NPAR(11)	Element parameters dump flag EQ.1; Print EQ.0; Suppress printing

NOTES/

(1) The number of pipe elements ("NPIPE") counts both tangent and bend geometries, and both the material and section property tables can reference either the bend or tangent element types.

(2) A branch point is defined as a nodal location where at least three (3) tangent pipe elements connect. The two input parameters "NBRP" and "MAXTAN" reserve storage for an index array created during the processing of pipe element data; posting a larger number of maximum common tangents than actually exist is not considered a fatal error condition. Branch point data is read if requested, but not currently used; i.e. to be used in future program versions.

IV. ELEMENT DATA (continued)

2. Material Property Cards

Temperature-dependent Young's modulus (E), Poisson's ratio (ν) and thermal expansion coefficient (α) are allowed. If more than one (1) temperature point is input for a material table, then the program selects properties using linear interpolation between input temperature values. The temperature used for property selection is the average element temperature which is denoted as T_a:

$$T_a = (T_i + T_j)/2.$$

where T_i and T_j are the input nodal temperatures for ends "i" and "j" of the pipe. For each different material, the following set of cards must be input:

a. material identification card (2I5,6A6)

notes	columns	variable	entry
(1)	1 - 5	M	Material identification number; GE.1 and LE.NUMMAT
	6 - 10	NT	Number of different temperatures at which properties are given; EQ.0; one temperature point is assumed to be input
	11 - 46		Material description used to label the output for this material

NOTES/

(1) Material identification number must be input between one ("1") and the total number of materials specified ("NUMMAT").

b. material cards (4F10.0)

notes	columns	variable	entry
(1)	1 - 10	T(N)	Temperature, T_n
	11 - 20	E(N)	Young's modulus, E_n
	21 - 30	XNU(N)	Poisson's ratio, ν_n
	31 - 40	ALP(N)	Thermal expansion coefficient, α_n

NOTES/

(1) Supply one card for each temperature point in the material table; at least one card is required. Temperatures must be input in increasing (algebraic) order. If two or more points are used, care must be taken to insure that the table covers the expected range of average temperatures existing in the elements to which the material table is assigned.

3. Section Property Cards (I5,5F10.0,3A6)

notes	columns	variable	entry
(1)	1 - 5	N	Section property identification number; GE.1 and LE.NSECT
(2)	6 - 15		Outside diameter of the pipe, d_o
	16 - 25		Pipe wall thickness, t
	26 - 35		Shape factor for shear distortion, α_v
(3)	36 - 45		Weight per unit length of section, γ_1
(4)	46 - 55		Mass per unit length of section, ρ_1
	56 - 73		Section description (used to label the output)

NOTES/

(1) Section property identification numbers must be input in an ascending sequence beginning with one ("1") and ending with the total number of section specified ("NSECT").

(2) Assuming that (y,z) are the section axes and that the x-axis is normal to the section, the properties for the section are computed from the input parameters [d_o, t and α_v] as follows:

(a) inner and outer pipe radii;

$$r_o = d_o/2$$

$$r_i = r_o - t$$

(b) cross-sectional area (axial deformations);

$$A_x = \pi(r_o^2 - r_i^2)$$

(c) principal moments of inertia (bending);

$$I_y = (\pi/4)\ (r_o^4 - r_i^4)$$

$$I_z = I_y$$

(d) polar moment of inertia (torsion);

$$J_x = 2I_y$$

(e) effective shear areas (shear distortions);

$$A_y = A_x/\alpha_v$$

$$A_z = A_y$$

Note that the shape factor for shear distortion (α_v) may be input directly. If the entry is omitted, the shape factor is computed using the equation:

$$\alpha_v = (4/3)\ (r_o^3 - r_i^3)/[(r_o^2 + r_i^2)\ (r_o - r_i)]$$

$$\simeq 2.0$$

An input value for α_v greater than one hundred (100.) causes the program to neglect shear distortions entirely. If used, the same shape factor is applied to both in and out-of-plane shear distortions.

(3) The weight per unit length of section (γ_1) is used to compute gravity loadings on the elements. Fixed end shears, moments, torques, etc. are computed automatically and applied as equivalent nodal loads. These forces will not act on the structure unless first assigned to one of the element load cases (A,B,C,D) in Section IV,L.5, below.

(4) The mass per unit length is only used to form the lumped mass matrix for a dynamic analysis case. If no entry is input, then the program will re-define the mass density from the weight density using:

$$\rho_1 = \gamma_1/386.4$$

Either a non-zero weight density or mass density will cause the program to assign masses to all pipe element nodes.

4. Branch Point Node Numbers

If the number of output branch point nodes has been omitted from the control card (i.e., cc 26-30 blank), skip this section of input, and no branch point data will be read. Otherwise, supply node numbers for a total number of branch points requested on the control card, ten (10) nodes per card:

first card (10I5)

notes	columns	variable	entry
(1)	1 - 5		Node number at branch point 1
	6 - 10		Node number at branch point 2
	. . .		. . .
	45 - 50		Node number at branch point 10

second card (10I5) -- if required

notes	columns	variable	entry
	1 - 5		Node number at branch point 11
	. . .		. . .

NOTES/

(1) A node does not define a branch point unless at least three (3) tangent elements are common to the node. Branch point output is only produced for static analysis cases.

5. Element Load Case Multipliers

Five (5) cards must be input in this section specifying the fraction of gravity (in each of the X,Y,Z coordinate directions), the fraction of thermal loading and the fraction of internal pipe pressure loading to be added to each of four (4) possible element loading combinations (A,B,C,D).

Card 1 X-direction gravity (4F10.0)

notes	columns	variable	entry
(1)	1 - 10		Fraction of X-direction gravity to be applied in element load case A
	11 - 20		Fraction of X-direction gravity to be applied in element load case B
	21 - 30		Fraction of X-direction gravity to be applied in element load case C
	31 - 40		Fraction of X-direction gravity to be applied in element load case D

Card 2 Y-direction gravity (4F10.0)

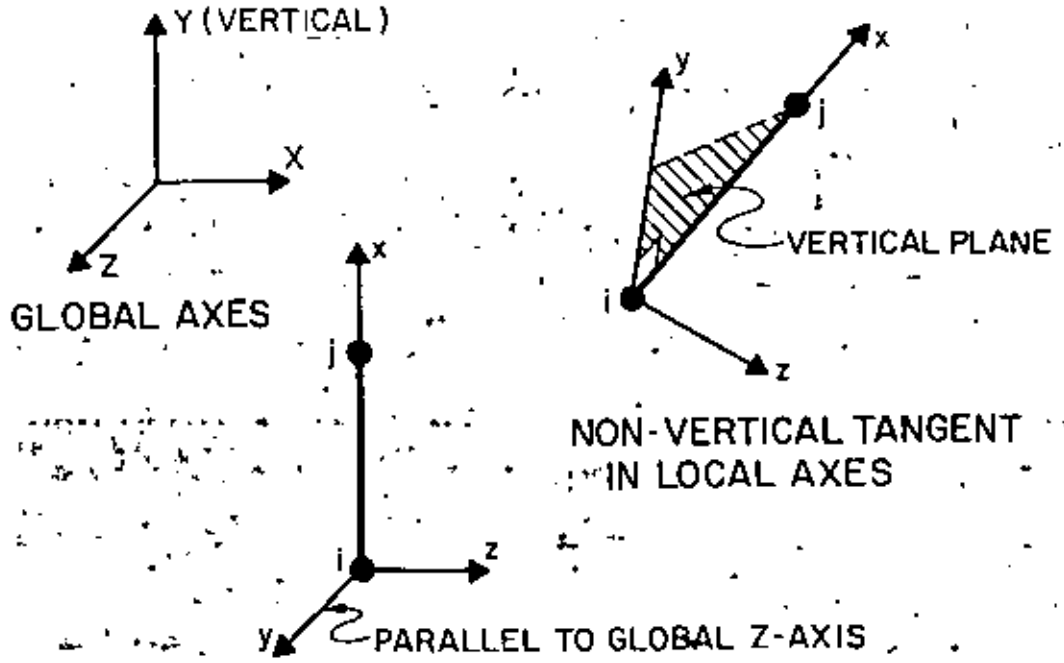
Card 3 Z-direction gravity (4F10.0)

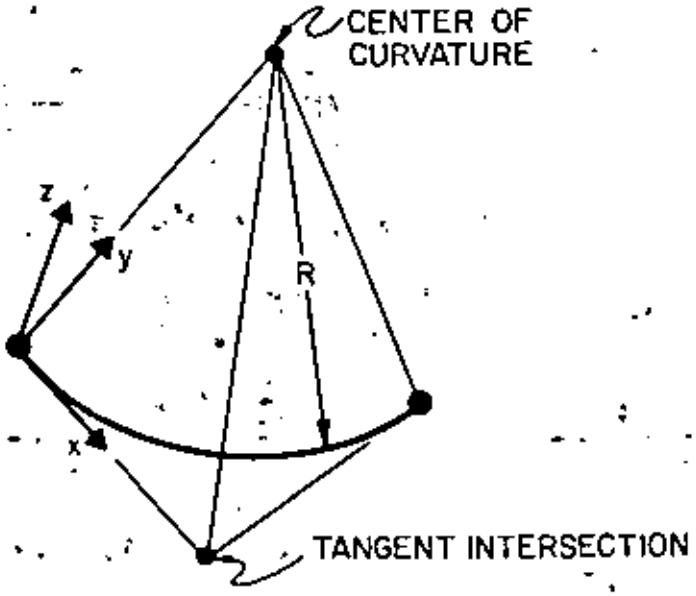
Card 4 Thermal loads (4F10.0)

notes	columns	variable	entry
(2)	1 - 10		Fraction of thermal loading to be applied in element load case A
	11 - 20		Fraction of thermal loading to be applied in element load case B
	21 - 30		Fraction of thermal loading to be applied in element load case C
	31 - 40		Fraction of thermal loading to be applied in element load case D

Card 5 Internal pressure (4F10.0)

notes	columns	variable	entry
(3)	1 - 10		Fraction of pressure-induced loadin. applied in element load case A
	11 - 20		Fraction of pressure-induced loading applied in element load case B
	21 - 30		Fraction of pressure-induced loading applied in element load case C
	31 - 40		Fraction of pressure-induced loading applied in element load case D





LOCAL COORDINATE SYSTEMS FOR PIPE-ELEMENTS

5. Element Load Case Multipliers (continued)

NOTES/

(1) No gravity loads will be produced if the weight per unit length was input as zero on all section property cards. Otherwise, a multiplier of 1.0 input for an element load case means that 100% of deadweight will be assigned to that load combination.

(2) No thermal loading will result if the coefficient of thermal expansion has been omitted from all the material cards. Otherwise, thermal loads are computed for each element using the ΔT between the average element temperature (T_a) and the stress-free temperature (T_o) given with each pipe element card (Section IV.L.6, below).

(3) Element distortions are computed for each element due to internal pressure, and these loads are combined into element load cases by means of appropriate non-zero entries in Card 5.

Gravity, thermal or pressure-induced loads cannot act on the structure unless first combined in one or more of the element load sets (A,B,C,D). Once defined, element load cases are assigned (via scale factors) to the structure load cases by means of Element Load Multipliers given in Section VI. An element load case combination may be used a multiple number of times when defining the various structure loading conditions.

6. Pipe Element Cards

a. card type 1

notes	columns	variable	entry
(1)	1 - 4	N	Pipe element number; GE.1 and LE.NPIPE
	5		Geometric type code: "T" (or blank); tangent section "B" ; bend (circular) section
	6 - 10	I	Node I number
	11 - 15	J	Node J number
	16 - 20	MAT	Material identification number; GE.1 and LE.NUMMAT
	21 - 25	ISECT	Section property identification number; GE.1 and LE.NSECT
(2)	26 - 35		Stress-free temperature, T_o
(3)	36 - 45		Internal pressure, p
(4)	46 - 55		Positive projection of a local y-vector on the global X-axis; A(yX)

IV. ELEMENT DATA (continued)

6. Pipe Element Cards (continued)

notes	columns	variable	entry
	56 - 65		Positive projection of a local y-vector on the global Y-axis; A(yY)
	66 - 75		Positive projection of a local y-vector on the global Z-axis A (yZ)
(5)	76 - 80	KG	Node number increment for tangent element generation; EQ.0; default set to "1"

NOTES/

(1) Card type 1 is used for both tangent and bend elements; a second card (card type 2, below) must be input immediately following card type 1 if the pipe element is a bend (i.e., "B" in cc 5). Note that element cards must be input in ascending sequence beginning with one ("1") and ending with the total number of pipe elements. If tangent elements are omitted, generation of the intermediate elements will occur; the generation algorithm is described below. An attempt to generate bend type elements is considered to be an error.

(2) The stress-free temperature, T_o, is subtracted from the average element temperature, T_a, to compute the uniform temperature difference acting on the element:

$$\Delta T = T_a - T_o$$

The entire element is assumed to be at this uniform value of temperature difference.

(3) The value of pressure is used to compute a set of self-equilibrating joint forces arising from member distortions due to pressurization; i.e., the mechanical equivalent of thermal loads. For bend elements, the pressure is also used to compute the bend flexibility factor, k_p. The curved pipe subjected to bending is more flexible than elementary beam theory would predict. The ratio of "actual" flexibility to that predicted by beam theory. is denoted by k_p, where

$$k_p = (1.65/h)/[1 + (6p/Eh)(R/t)^{4/3}] \geq 1$$

in which

$$h = tR/r^2$$

$$r = (d_o - t)/2$$

6. Pipe Element Cards (continued)

and

t = pipe wall thickness
R = radius of the circular bend
r = mean radius of the pipe cross section
d_o = outside diameter of the pipe
E = Young's modulus
p = internal pressure

The flexibility factor is computed and applied to all bend elements; pressure stiffening is neglected if the entry for internal pressure ("p") is omitted.

(4) The global projections of the local y-axis for a tangent member may be omitted (cc 46-75 blank); for this case, the following convention for the local system is assumed:

(a) tangents parallel to the global Y-axis (vertical axis) have their local y-axes directed parallel to and in the same direction as the global Z-axis;

(b) tangents not parallel to the global Y-axis have their local y-axes contained in a vertical (global) plane such that local y projects positively on the positive global Y-axis.

For bend elements, the global projections of the local y-axis are not used; instead, the local axis convention is defined as follows:

(a) the local y-axis is directed positively toward and intersects the center of curvature of the bend (i.e., radius vector);

(b) the local x-axis is tangent to the arc of the bend and is directed positively from node I to node J.

Note that for all elements, the local x, y, z system is a right-handed set (see figure).

(5) If a tangent element sequence exists such that each element number (NE_i) is one (1) greater than the previous number (NE_{i-1}); i.e.,

$$NE_i = NE_{j-1} + 1$$

only the element card for the first tangent in the

6. Pipe Element Cards (continued)

series need be input. The node numbers for the missing tangents are computed using the formulae:

$$NI_i = NI_{i-1} + KG$$

$$NJ_i = NJ_{i-1} + KG$$

where "KG" is the node number increment input in cc 76-80 for the first element in the series, and the

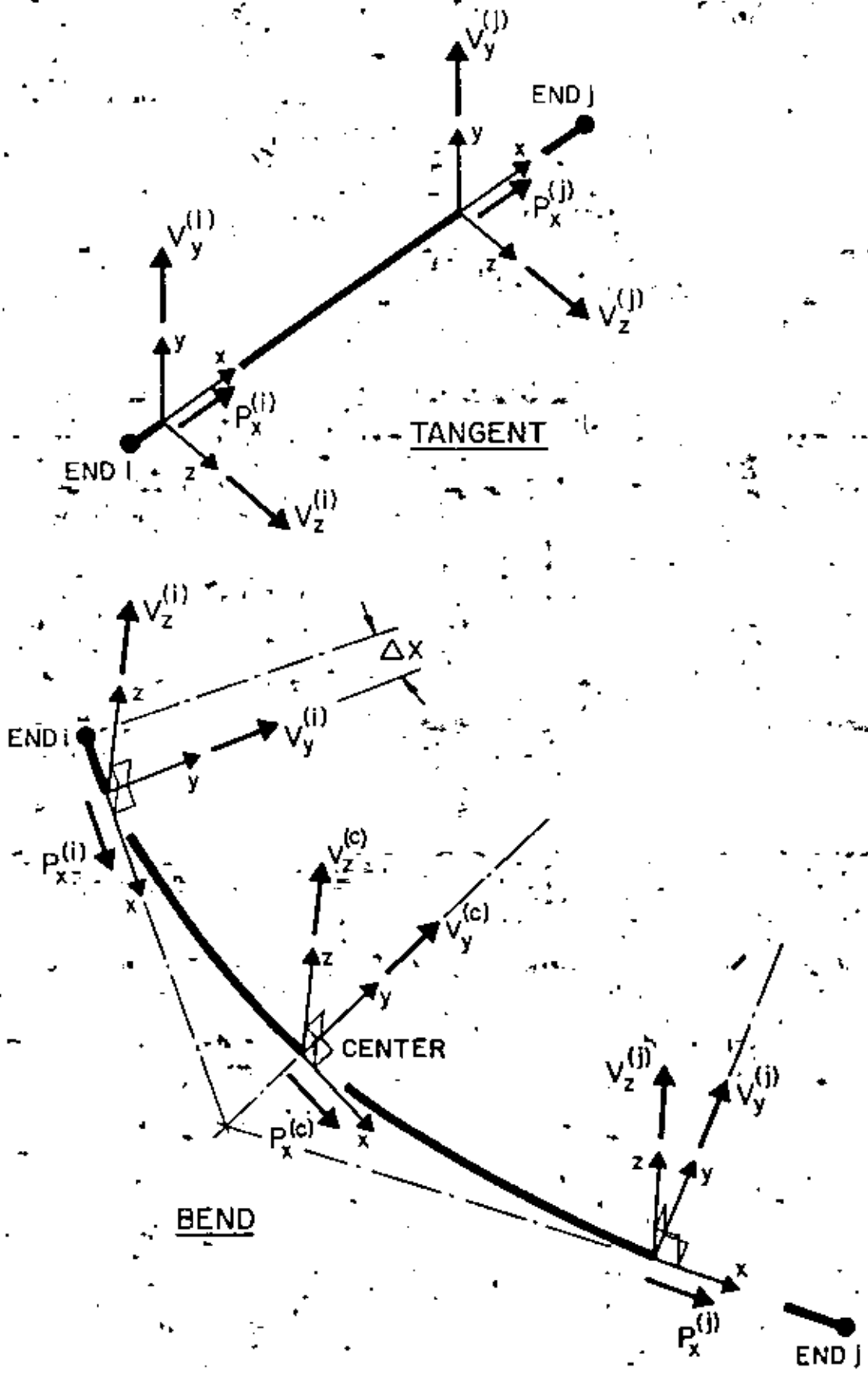
(a) material identification number
(b) section property identification number
(c) stress-free temperature
(d) internal pressure
(e) y-axis global projections

for each tangent in the generation sequence are taken to be the same as those input on the first card in the series. The node number increment ("KG") is reset to one (1) if left blank on the first card in the series. The last (highest) element cannot be generated; i.e., it must be input.

Bend element data cannot be generated because two input cards are required for each bend. Also, the element just prior to a bend-element must appear on an input card. Several bends may be input in a sequence, but each bend must appear (on two cards) in the input stream.

b) card type 2 (F10.0,3X,A2,4F10.0)

notes	columns	variable	entry
(1)	1 - 10	R	Radius of the bend element, R
(2)	14 - 15		Third point type code: "TI" (or blank); third point is the tangent intersection point "CC" ; third point is the center of curvature
	16 - 25		X-ordinate of the third point, X_3
	26 - 35		Y-ordinate of the third point, Y_3
	36 - 45		Z-ordinate of the third point, Z_3
	46 - 55		Fraction of wall thickness to be used for dimensional tolerance tests; EQ.0; default set to "0.1"



FORCE SIGN CONVENTION FOR PIPE ELEMENT OUTPUT

IV. ELEMENT DATA (continued)

6. Pipe Element Cards (continued)

NOTES/

(1) The radius of the bend ("R") must be input regardless of the method ("T1" or "CC") used to define the third point for the bend.

(2) If the tangent intersection point is used, the program computes a radius for the bend and compares the computed value with the input radius. An error condition is declared if the two radii are different by more than the specified fraction (or multiple) of the section wall thickness. The lengths of the two tangent lines (I to TI and J to TI) are compared for equality, and an error will be flagged if the two values are discrepant by more than the dimensional tolerance.

If the center of curvature is input, the distances from the third point to nodes I and J are compared to the input radius; discrepancies larger than the user defined tolerance are noted as errors.

This second element card is only to be input for the bend type element.

Element Stress Output

Stress output for pipe elements consists of forces and moments acting in the member cross sections at the ends of each member and at the midpoints of the arcs in bend elements. Output quantitites act on the element segment connecting the particular output station and end i; i.e., j to i, center to i, or ΔX to i (where $\Delta X \rightarrow 0$). Positive force/moment vectors are directed into the positive local (x,y,z) directions, as shown in the accompanying figure.

V. CONCENTRATED LOAD/MASS DATA (2I5,6F10.4)

notes	columns	variable	entry
(1)	1 - 5	N	Nodal point number
(2)	6 - 10	L	Structure load case number; GE.1; static analysis EQ.0; dynamic analysis
	11 - 20	FX(N,L)	X-direction force (or translational mass coefficient)
	21 - 30	FY(N,L)	Y-direction force (or translational mass coefficient)
	31 - 40	FZ(N,L)	Z-direction force (or translational mass coefficient)
	41 - 50	MX(N,L)	X-axis moment (or rotational inertia)
	51 - 60	MY(N,L)	Y-axis moment (or rotational inertia)
	61 - 70	MZ(N,L)	Z-axis moment (or rotational inertia)

NOTES/

(1) For a static analysis case (NDYN.EQ.0), one card is required for each nodal point ("N") having applied (non-zero) concentrated forces or moments. All structure load cases must be grouped together for the node ("N") before data is entered for the next (higher) node at which loads are applied.. Only the structure load cases for which node N is loaded need be given, but the structure load case numbers ("L") which are referenced must be supplied in ascending order. Node loadings must be defined (input) in increasing node number order, but again, only those nodes actually loaded are required as input. The static loads defined in this section act on the structure exactly as input and are not scaled, factored, etc. by the element load case (A,B,C,D) multipliers (Section VI, below). Nodal forces arising from element loadings are combined (additively) with any concentrated loads given in this section. Applied force/moment vectors act on the structure, positive in the positive global directions. Only one card is allowed per node per load case.

For a dynamic analysis case (NDYN.EQ.1,2, 3 or 4), structure load cases have no meaning, but the program expects to read data in this section nonetheless. In place of concentrated loads, lumped mass coefficients for the nodal degrees of freedom may be input for any (or all) nodes. The mass matrix is automatically constructed by the program from element geometry and associated material densities; the mass coefficients read in this section are combined (additively) with the existing element-based lumped mass matrix. For mass input, a node may only be specified once, and the load case number ("L") must be zero (or blank).

The program terminates reading loads (or mass) data when a zero (or blank) node number ("N") is encountered; i.e., terminate this section of input with a blank card. For the special case of a static analysis with no concentrated loads applied, input only one (1) blank card in this section. Similarly, a dynamic analysis in which the mass matrix is not to be augmented by any entries in this section requires only one (1) blank card as input.

(2) For a static analysis, structure load case numbers range from "1" to the total number of load cases requested on the Master Control Card ("LL"); thus, $1 \leq L \leq LL$, NDYN.EQ.0. For a dynamic analysis, only zero (0) references are allowed; thus, L = 0, NDYN.EQ.1,2 3, or 4.

VI. ELEMENT LOAD MULTIPLIERS (4F10.0)

notes	columns	variable	entry
(1,2)	1 - 10	EM(1)	Multiplier for element load case A
	11 - 20	EM(2)	Multiplier for element load case B
	21 - 30	EM(3)	Multiplier for element load case C
	31 - 40	EM(4)	Multiplier for element load case D

NOTES/

(1) One card must be given for each static (NDYN.EQ.0) structure load case requested on the Master Control Card ("LL"). The cards must reference load case numbers in ascending order. The four (4) element load sets (A,B,C,D), if created during the processing of element data (Section IV, above), are combined with any concentrated loads specified in Section V for the structure load cases. For example, suppose an analysis case calls for seven (7) static structure loading conditions (i.e., LL = 7), then the program expects to read seven (7) cards in this section. Further, suppose card number three (3) in this section contains the entries:

[EM(1),EM(2),EM(3),EM(4)] = [-3.0,0.0,2.0,0.0]

Structure load case three (3) will then be constructed using 100% of any concentrated loads specified in Section V minus (-) 300% of the loads in element set A plus (+) 200% of the loads in element set C. Load sets B and D will not be applied in structure load case 3. Element load sets may be referenced any number of times in order to construct different structure loading conditions. Element-based loads (gravity, thermal, etc.) can only be applied to the structure by means of the data entries in this section.

(2) If this case calls for one of the dynamic analysis options, supply only one blank card in this section. If the job is a dynamic re-start case (NDYN.EQ.-2 or -3), skip this section.

Static analysis input is complete with this section. Begin a new data case with a new Heading Card (see Section I).

VII. DYNAMIC ANALYSES

Four (4) types of dynamic analysis can be performed by the program. The type of analysis is indicated by the number "NDYN" specified in card columns 21-25 of the Master Control Card (Section II). If

NDYN.EQ.1; Determination of system mode shapes and frequencies only
(complete input Section VII.A, only)

NDYN.EQ.2; Dynamic Response Analysis for arbitrary time dependent loads using mode superposition
(complete both Sections VII.A and B below)

NDYN.EQ.3; Response Spectrum Analysis
(complete both Sections VII.A and C, below)

NDYN.EQ.4; Dynamic Response Analysis for arbitrary time dependent loads using step-by-step direct integration
(complete Section VII.B below)

In any given dynamic analysis case only one (1) value of NDYN will be considered. However, if NDYN.EQ.2 or 3, the program must first solve the eigenvalue problem for structure modes and frequencies. These eigenvalues/vectors are then used as input to either the Forced Response Analysis (NDYN.EQ.2) or to the Response Spectrum Analysis (NDYN.EQ.3). Hence, options 1, 2 or 3 all require that the control parameters for eigenvalue extraction be supplied in Section VII.A, below.

In case of a direct step-by-step integration analysis (NDYN.EQ.4) do not provide the eigenvalue solution control card of Section VII.A.

For the special case of dynamic analysis re-start (NDYN.EQ.-2 or -3), data input consists of the Heading Card (Section I), the Master Control Card (Section II), and either of Sections VII.B (-2) or VII.C (-3), below. Re-starting is possible only if a previous solution using the same model was performed with NDYN.EQ.1, and the results from this eigenvalue solution were saved on the re-start file. (See Appendix A.)

Up to this section the program processes (i.e., expects to read) essentially the same blocks of data for either the static or dynamic analysis cases; certain of these preceding data cards, however, are read by the program but are not used in the dynamic analysis phase. In general, the purpose of the preceding data sections is to provide information leading to the formation of the system stiffness and mass matrices (appropriately modified for displacement boundary conditions). For example, element load sets (A,B,C,D) may be constructed as though a static case were to be considered, but these data are not used in a dynamic analysis; i.e., the same data deck through Section IV can be used for either type of analysis. The concept of structure loading conditions is not defined for the dynamic case, and input for Sections V and VI must be prepared specially.

A diagonal (lumped) mass matrix is formed automatically using element geometry and assigned material density or densities. The mass matrix so defined contains only translational mass coefficients calculated from tributary element volumes common to each node. Known rotational inertias must be input for the individual nodal degrees of freedom in Section V, above.

Non-zero impressed displacements (or rotations) input by means of the BOUNDARY element (type "7") are ignored; instead the component is restrained against motion during dynamic motion of the structure.

The program does not change the order of the system by performing a condensation of those nodal degrees of freedom having no (zero) mass coefficients; i.e., a zero mass reduction is not performed. No distinction is made between static and dynamic degrees of freedom; i.e., they are identical in sequence, type and total number.

VII. DYNAMIC ANALYSES (continued)

A. MODE SHAPES AND FREQUENCIES (NDYN.EQ.1, 2 or 3) (3I5,2F10.0)

notes	columns	variable	entry
(1)	1 - 5	IFPR	Flag for printing intermediate matrices, norms, etc. calculated during the eigenvalue solution; EQ.0; do not print EQ.1; print
(2)	6 - 10	IFSS	Flag for performing the STURM SEQUENCE check; EQ.0; check to see if eigenvalues were missed EQ.1; pass on the check
(3)	11 - 15	NITEM	Maximum number of iterations allowed to reach the convergence tolerance; EQ.0; default set to "16"
(4)	16 - 25	RTOL	Convergence tolerance (accuracy) for the highest ("NF") requested eigenvalue; EQ.0; default set to "1.0E-5"
(5)	26 - 35	COFQ	Cut-off frequency (cycles/unit time) EQ.0; NF eigenvalues will be extracted GT.0; extract only those values below COFQ
(6)	36 - 40	NFO	Number of starting iteration vectors to be read from TAPE10

NOTES/

(1) Extra output produced by the eigenvalue solutions can be requested; output produced by this option can be quite voluminous. Normal output produced by the program consists of an ordered list of eigenvalues followed by the eigenvectors for each mode. The number of modes found and printed is specified by the variable "NF" given in card columns 16-20 of the Master Control Card.

(2) The program performs the solution for eigenvalues/vectors using either of two (2) distinct algorithms:

(a) the DETERMINANT SEARCH algorithm requires that the upper triangular band of the system stiffness matrix fit into high speed memory (core); i.e., one equation "block".

(b) the SUBSPACE ITERATION algorithm is used if only portions (fractions) of the system matrix can be retained in core; i.e., the matrix (even though in band form) must be manipulated in blocks.

A. MODE SHAPES AND FREQUENCIES (continued)

The program will automatically select the SUBSPACE ITERATION procedure for eigenvalue solution if the model is too large for the in-core algorithm.

The entries "IFSS", "NITEM" and "RTOL" are ignored if the program can use the DETERMINANT SEARCH to find eigenvalues. Whether or not a model is too large for the DETERMINANT SEARCH depends on the amount of core allocated (by the programmer and not the user) for array storage. The program variable "MTOT" equals the amount of working storage available.

Define:

MBAND = maximum equation bandwidth (coefficients)
= (maximum element node number difference) x (average number of degrees of freedom per node)

NEQ = total number of degrees of freedom in the model
= (6) x (total number of nodes) - [number of fixed (deleted) degrees of freedom]

NEQB = number of equations per block of storage
= MTOT/ MBAND/ 2 (for large systems)

If NEQB is less than NEQ, the model is too large for the DETERMINANT SEARCH algorithm, and the SUBSPACE ITERATION procedure will be used.

If the SUBSPACE ITERATION algorithm is used the user may request that the STURM SEQUENCE check be performed. By experience the algorithm has always produced the lowest NF eigenvalues, but there is no formal mathematical proof that the calculated NF eigenvalues will always be the lowest ones. The STURM SEQUENCE check can be used to verify that the lowest NF eigenvalues have been obtained. It should be noted that the computational effort expended in performing the STURM SEQUENCE check is not trivial. A factorization of the complete system matrix is performed at a shift just to the right of the NFth eigenvalue.

If during the SUBSPACE ITERATION the NFth eigenvalue fails to converge to a tolerance of "RTOL" (normally 1.0E-5, or 5 significant figures) within "NITEM" (normally "16") iterations, then the STURM SEQUENCE flag ("IFSS") is ignored.

A. MODE SHAPES AND FREQUENCIES (continued)

(3) The maximum number of iterations to reach convergence ("NITEM") applies only to the SUBSPACE ITERATION algorithm. If cc 11-15 are left blank, a default value of "16" for NITEM is assumed.

(4) The convergence tolerance ("RTOL") is applicable only if the SUBSPACE ITERATION algorithm is used. This tolerance test applies to the NFth eigenvalue, and all eigenvalues lower than the NFth one will be more accurate than RTOL. The lowest mode is found most accurately with precision decreasing with increasing mode number until the highest requested mode ("NF") is accurate to a tolerance of RTOL. Iteration is terminated after cycle number (k+1) if the NFth eigenvalue (λ, say) satisfies the inequality:

$$[\,|\lambda(k+1) - \lambda(k)|/\lambda(k)] < \text{RTOL}$$

If the determinant search algorithm is used, the eigenpairs are obtained to a high precision, which is indicated by the "physical error bounds"

$$\epsilon_i = \|r_i\|_2 \,/\, \|K\phi_i\|_2$$

where

$$r_i = (K - \omega_i^2 M)\,\phi_i ,$$

and $(\omega_i^2\ \phi_i)$ are the i'th eigenvalue and eigenvector obtained in the solution.

(5) The cut-off frequency ("COFQ") is used by both eigenvalue algorithms to terminate computations if all eigenvalues below the specified frequency have been found.

The DETERMINANT SEARCH algorithm computes eigenvalues in order from "1" to "NF". If the Nth eigenvalue ($1 \le N < NF$) has a frequency greater than "COFQ", the remaining (NF-N) eigenvalues are not computed.

A. MODE SHAPES AND FREQUENCIES (continued)

The SUBSPACE ITERATION algorithm terminates calculation when the Nth eigenvalue is accurate (i.e., does not change with iteration) to a tolerance of RTOL. As before, the Nth eigenvalue is the nearest eigenvalue higher than COFQ. If the SUBSPACE ITERATION solution determines N eigenvalues less than COFQ (where, N <NF), the STURM SEQUENCE check (if requested) is performed using the Nth (rather than the NFth) eigenvalue as a shift.

Only those modes whose frequencies are less than COFQ will be used in the TIME HISTORY or RESPONSE SPECTRUM analyses (Sections VII.B and C, below).

(6) The starting iteration vectors, together with control information, must be written onto TAPE10 before the program execution is started. Appendix B describes the creation of TAPE10 and gives the required control cards.

(7) The program does not calculate rigid body modes, i.e. the system must have been restraint so that no rigid body modes are present. In exact arithmetic the element d_{nn} of the matrix D in the triangular factorization of the stiffness matrix, i.e. $K = LDL^T$, is zero if a rigid body mode is present. In computer arithmetic the element d_{nn} is small when compared with the other elements of the matrix D. If this condition occurs the program stops with a message.

Note: If many "artificially" stiff boundary elements are used, the average of the elements of D will be artificially large. Consequently, d_{nn} may be small in comparison, and although no rigid body modes may be present, the program will stop. In a dynamic analysis it is recommended not to use very stiff boundary elements.

END OF DATA CASE INPUT (NDYN.EQ.1)

B. RESPONSE HISTORY ANALYSIS (NDYN.EQ.2 or NDYN.EQ.4)

The NDYN.EQ.2 option uses the ("NF") mode shapes and frequencies computed in the preceeding Section (VII.A) to perform a mode superposition solution for forced response. The NDYN.EQ.4 option initiates a direct step-by-step integration of the coupled system equations, i.e. no eigenvalue solution has been performed and no transformation to the eigenvector basis is now carried out. The data input is identical to the case NDYN.EQ.2 except for the definition of damping. Dynamic response can be produced by two (2) general types of forcing function:

(1) ground acceleration input in any (or all) of the three (3) global (X,Y,Z) directions;

and/or

(2) time varying loads (forces/moments) applied in any (or all) nodal degrees of freedom (except - "slave" - degrees of freedom)

Time dependent forcing functions (whether loads or ground acceleration components) are described in two steps. First, a number (1 or more are possible) of non-dimensional time functions are specified tabularly by a set of descrete points: $[f(t_i), t_i]$, where $i = 1,2,\ldots,k$. Each different time function may have a different number of definition points (k). A particular forcing function applied at some point on the structure is then defined by a scalar multiplier ("β", say) and reference to one of the input time functions ("f(t)", say). The actual force (or acceleration) at any time ("τ", say) equals $\beta \times f(\tau)$; $f(\tau)$ is found by linear interpolation between two of the input time points $\{t_i, t_{i+1}\}$, where $t_i \leq \tau \leq t_{i+1}$.

Assuming that the solution begins at time zero (0), an independent arrival time (t_a, where $t_a \geq 0$) may be assigned to each forcing function. The forcing function is not applied to the system until the solution time ("τ", say) equals the arrival time, t_a. Interpolation for function values is based on relative time within the function table; i.e., $g(\tau) = f(\tau - t_a)$.

The structure is assumed to be at rest at time zero; i.e., zero initial displacements and velocities are assumed at time of solution start.

The following data are required for a Forced Dynamic Response Analysis:

1. Control Card (5I5,2F10.0)

notes	columns	variable	entry
(1)	1 - 5	NFN	Number of different time functions; GE.1

VII. DYNAMIC ANALYSES (continued)

B. - RESPONSE HISTORY ANALYSIS (continued)

notes	columns	variable	entry
(2)	6 - 10	NGM	Ground motion indicator; EQ.0; no ground motion is input EQ.1; read ground motion control card (Section VII.B.3)
(3)	11 - 15	NAT	Number of different arrival times for the forcing functions; EQ.0; all arrival times are zero
(4)	16 - 20	NT	Total number of solution time steps; GE.1
(5)	21 - 25	NOT	Output print interval for stresses, displacements, etc.; GE.1 and LE.NT
(4)	26 - 35	DT	Solution time step, Δt; GT.0
(6)	36 - 45	DAMP	Damping factor to be applied to all NF modes (fraction of critical); GE.0

In case of NDYN.EQ.4 use

(6)	36 - 45	ALPHA	Damping factor α
(7)	46 - 55	BETA	Damping factor β

NOTES/

(1) At least one (1) time function must be input.

(2) If no ground acceleration acts on the structure, set "NGM" to zero and skip Section VII.B.3, below. Both ground acceleration and nodal force input are allowed.

(3) If no arrival time values are input, all forcing functions begin acting on the structure at time zero. The same arrival time value may be referenced by different forcing functions. "NAT" determines the number of non-zero entries that the program expects to read in Section VII.B.4, below.

(4) The program performs a step-by-step integration of the equations of motion using a scheme which is unconditionally stable with respect to time step size, Δt. In case NDYN.EQ.2 the modal uncoupled equations of motion are integrated. In case NDYN.EQ.4 the coupled system equations are integrated. If "T" is the period of the highest numbered mode (normally the NFth mode) that is to be included in the response calculation, Δt should be chosen such that $\Delta t/T < 0.1$. A

B. RESPONSE HISTORY ANALYSIS (continued)

larger time step (i.e., $\Delta t > 0.1T$) will not cause failure (instability), but participation of the higher modes is "filtered" from the predicted response. In general, with increasing time step size the solution is capable of capturing less of the higher frequency participation.

(5) The program computes system displacements at every solution time step, but printing of displacements and recovery of element stresses is only performed at solution step intervals of "NOT". NOT must be at least "1" and is normally selected in the range of 10 to 100.

(6) The damping factor ("DAMP") is applied to all NF modes. The admissible range for DAMP is between 0.0 (no damping) and 1.0 (100% of critical viscous damping).

(7) In case NDYN.EQ.4 the damping matrix used is $C = \alpha M + \beta K$, where α and β are defined in columns 30 to 55.

VII. DYNAMIC ANALYSES (continued)

B. RESPONSE HISTORY ANALYSIS (continued)

2. Time-Varying Load Cards (4I5,F10.0)

notes	columns	variable	entry
(1)	1 - 5	NP	Nodal point number where the load component (force or moment) is applied; GE.1 and LE.NUMNP EQ.0 last card only
(2)	10	IC	Degree of freedom number; GE.1 and LE.6 (δX=1, δY=2, δZ=3, ϕX=4, ϕY=5, ϕZ=6)
(3)	11 - 15	IFN	Time function number; GE.1 and LE.NTFN
(4)	16 - 20	IAT	Arrival time number; EQ.0; load applied at solution start GE.1; non-zero arrival time
(5)	21 - 30	P	Scalar multiplier for the time function; EQ.0; no load applied

NOTES/

(1) One card is required for each nodal degree of freedom having applied time varying loads. Cards must be input in ascending node point order. This sequence of cards must be terminated with a blank card. A blank card must be supplied even if no loads are applied to the system.

(2) The same node may have more than one degree of freedom loaded; arrange degrees of freedom references ("IC") in ascending sequence at any given node.

(3) A non-zero time function number ("IFN") must be given for each forcing function. IFN must be between 1 and NFN. The time functions are input tabularly in Section VII.B.5, below. Function values at times between input time points are computed with linear interpolation.

(4) If "IAT" is zero (or blank), the forcing function is assumed to act on the system beginning at time zero. If IAT is input as a positive integer between 1 and NAT, the IATth arrival time (defined in Section VII.B.4, below) is used to delay the application of the forcing function; i.e., the forcing function begins acting on the structure when the solution reaches the IATth arrival time value.

(5) The actual magnitude of force (or moment) acting on the model at time, t, equals the product: ("P") x (value of function number "IFN" at time, t).

VII. DYNAMIC ANALYSES (continued)

B. RESPONSE HISTORY ANALYSIS (continued)

3. Ground Motion Control Card (6I5)

notes	columns	variable	entry
(1)	1 - 5	NFNX	Time function number describing the ground acceleration in the X-direction
	6 - 10	NFNY	Time function number describing the ground acceleration in the Y-direction
	11 - 15	NFNZ	Time function number describing the ground acceleration in the Z-direction
(2)	16 - 20	NATX	Arrival time number, X-direction
	21 - 25	NATY	Arrival time number, Y-direction
	26 - 30	NATZ	Arrival time number, Z-direction

NOTES/

(1) This card must be input only if the ground motion indicator ("NGM") was set equal to one (1) on the Control Card (Section, VII.B.1, above). A zero time function number indicates that no ground motion is applied for that particular direction.

(2) Zero arrival time references mean that the ground acceleration (if applied) begins acting on the structure at time zero (0). Non-zero references must be integers in the range 1 to NAT.

VII. DYNAMIC ANALYSES (continued)

B. RESPONSE HISTORY ANALYSIS (continued)

4. Arrival Time Cards

a. card one (8F10.0)

notes	columns	variable	entry
(1)	1 - 10	AT(1)	Arrival time number 1
	11 - 20	AT(2)	Arrival time number 2
		. . .	. . .
	71 - 80	AT(8)	Arrival time number 8

b. card two (8F10.0) - (required if NAT.GT.8)

notes	columns	variable	entry
	1 - 10	AT(9)	Arrival time number 9
		etc.	etc.

NOTES/

(1) The entry ("NAT") given in cc 11-15 on the Control Card (Section VII.B.1, above) specifies the total number of arrival time entries to be read in this section. Input as many cards as are required to define "NAT" different arrival times, eight (8) entries per card. If no arrival times were requested (NAT.EQ.0), supply one (1) blank card in this section.

VII. DYNAMIC ANALYSES (continued)

B. RESPONSE HISTORY ANALYSIS (continued)

5. Time Function Definition Cards

Supply one set (card 1 and card(s) 2) of input for each of the "NFN" time functions requested in cc 1-5 of the Control Card (Section VII.B.1, above). At least one set of time function cards is expected in this section. The card sets are input in ascending function number order.

a. card 1 (I5,F10.0,12A5)

notes	columns	variable	entry
(1)	1 - 5	NLP	Number of function definition points; GE.2
(2)	6 - 15	SFTR	Scale factor to be applied to f(t) values; EQ.0; default set to "1.0"
	16 - 75	HED(12)	Label information (to be printed with output) describing this function table

NOTES/

(1) At least two points (i.e., 2 pairs: $f(t_i), t_i$) must be specified for each time function. Less than two points would preclude linear interpolation in the table for f(t).

(2) The scale factor "SFTR" is used to multiply function values only; i.e., input time values are not changed. If the scale factor is omitted, SFTR is re-set by the program to "1.0" thereby leaving input function values unchanged.

B. RESPONSE HISTORY ANALYSIS (continued)

5. Time Function Definition Cards (continued)

b. card(s) 2 (12F6.0)

notes	columns	variable	entry
(1)	1 - 6	T(1)	Time values at point 1, t_1
	7 - 12	F(1)	Function value at point 1, $f(t_1)$
	13 - 18	T(2)	Time value at point 2, t_2
	19 - 24	F(2)	Function value at point 2, $f(t_2)$
		etc.	etc.

NOTES:

(1) Input as many card(s) 2 as are required to define "NLP" pairs of $t_i, f(t_i)$, six (6) pairs per card. Pairs must be input in order of ascending time value. Time at point one must be zero, and care must be taken to ensure that the highest (last) input time value (t_{NLP}) is at least equal to the value of time at the end of solution; i.e., the time span for all functions must cover the solution time period otherwise the interpolation for function values will fail. For the case of non-zero arrival times associated with a particular function, the shortest arrival time reference ("t_A", say) plus (+) the last function time ("t_{NLP}") must at least equal the time at the end of the solution period (t_{END}, say); i.e., $t_A + t_{NLP} \geq t_{END}$.

B. RESPONSE HISTORY ANALYSIS (continued)

6. Output Definition Cards

To minimize the amount of output which would be produced by the program if all displacements, stresses, etc. were printed, output requests for specific components must be given in this section. Time histories for selected components appear in tables; the solution step output printing interval is specified as "NOT" which is given in cc 21-25 of the Control Card (Section VII.B.1, above).

a. displacement output requests

(1) control card (215)

notes	columns	variable	entry
(1)	1 - 5	KKK	Output type indicator; EQ.1; print histories and maxima EQ.2; printer plot histories and recovery of maxima EQ.3; recover maxima only
(2)	6 - 10	ISP	Printer plot spacing indicator

NOTES/

(1) The type of output to be produced by the program applies to all displacement requests. KKK.EQ.0 is illegal.

(2) "ISP" controls the vertical (down the page) spacing for printer plots. Output points are printed on every (ISP+1)th line. The horizontal (across the page) width of printer plots is a constant ten (10) inches (100 print positions). ISP is used only if KKK.EQ.2.

B. RESPONSE HISTORY ANALYSIS (continued)

6. Output Definition Cards

a. displacement output requests (continued)

(2) node displacement request cards (7I5)

notes	columns	variable	entry
(1)	1 - 5	NP	Node number GE.1 and LE.NUMNP EQ.0 last card only
(2)	6 - 10	IC(1)	Displacement component, request 1
	11 - 15	IC(2)	Displacement component, request 2
	16 - 20	IC(3)	Displacement component, request 3
	21 - 25	IC(4)	Displacement component, request 4
	26 - 30	IC(5)	Displacement component, request 5
	31 - 36	IC(6)	Displacement component, request 6 GE.1 and LE.6 EQ.0 terminates requests for the node

NOTES/

(1) Only those nodes at which output is to be produced (or at which maxima are to be determined) are entered in this section. Cards must be input in ascending node number order. Node numbers may not be repeated. This section must be terminated with a blank card.

(2) Displacement component requests ("IC") range from 1 to 6, where 1=δX,2=δY,3=δZ,4=ϕX;5=ϕY;6=ϕZ. The first zero (or blank) encountered while reading IC(1),IC(2),...,IC(6) terminates information for the card. Displacement components at a node may be requested in any order. As an example, suppose that δY, ϕX and ϕZ are to be output at node 34; the card could be written as /34,2,4,6,0/, or /34,6,4,2,0/, etc. but only four (4) fields would have non-zero entries./

B. RESPONSE HISTORY ANALYSIS (continued)

6. Output Definition Cards

b. element stress component output requests

(1) control card (2I5)

notes	columns	variable	entry
(1)	1 - 5	KKK	Output type indicator; EQ.1; print histories and maxima EQ.2; printer plot of histories and recovery of maxima EQ.3; recover maxima only
	6 - 10	ISP	Plot spacing indicator

NOTES/

(1) See Section VII.B.6.a.(1), above.

(2) element stress component request cards (13I5)

Requests are grouped by element type; "NELTYP" groups must be input. A group consists of a series of element stress component request cards terminated by a blank card. Element number references within an element type (TRUSS, say) grouping must be in ascending order. Element number references may be omitted but not repeated. The program processes element groups in the same order as originally input in the Element Data (Section IV, above). If no output is to be produced for an element type, then input one blank card for its group.

notes	columns	variable	entry
(1)	1 - 5	NEL	Element number GE.1 EQ.0; last card in the group only
(2)	6 - 10	IS(1)	Stress component number for output, request 1
	11 - 15	IS(2)	Stress component number for output, request 2
	. . . .	. . .	. . .
	61 - 65	IS(12)	Stress component number for output, request 12

NOTES/

(1) Terminate each different element output group (type) with a blank card. Elements within a group must be in element number order (ascending); element number repetitions are illegal.

(2) The first zero (or blank) request encountered while reading IS(1), IS(2),..., IS(12) terminates information for the card. No more than twelve (12) different components may be output for any one of the elements. Table VII.1 lists the stress component numbers and corresponding descriptions for the various element types. Some element types (TRUSS, for example) have fewer than 12 components defined; only the stress component numbers listed in Table VII.1 are legal references.

END OF DATA CASE INPUT (NDYN.EQ.2 or NDYN.EQ.4)

TABLE VII.1

ELEMENT TYPE	MAXIMUM NUMBER OF COMPONENTS	STRESS COMPONENT NUMBER	OUTPUT SYMBOL	D E S C R I P T I O N
1. TRUSS	(2)	(1)	(P/A)	AXIAL STRESS
		(2)	(P)	AXIAL FORCE
2. BEAM	(12)	(1)	(P1(I))	1-FORCE AT END I
		(2)	(V2(I))	2-SHEAR AT END I
		(3)	(V3(I))	3-SHEAR AT END I
		(4)	(T1(I))	1-TORQUE AT END I
		(5)	(M2(I))	2-MOMENT AT END I
		(6)	(M3(I))	3-MOMENT AT END I
		(7)	(P1(J))	1-FORCE AT END J
		(8)	(V2(J))	2-SHEAR AT END J
		(9)	(V3(J))	3-SHEAR AT END J
		(10)	(T1(J))	1-TORQUE AT END J
		(11)	(M2(J))	2-MOMENT AT END J
		(12)	(M3(J))	3-MOMENT AT END J
3. PLANE-STRESS/ PLANE-STRAIN				
4. AXISYM-METRIC	(20)	(1)	(11-S0)	V- STRESS AT POINT 0
		(2)	(22-S0)	U- STRESS AT POINT 0
		(3)	(33-S0)	T- STRESS AT POINT 0
		(4)	(12-S0)	UV-STRESS AT POINT 0
		(5)	(11-S1)	V- STRESS AT POINT 1
		(6)	(22-S1)	U- STRESS AT POINT 1
		(7)	(33-S1)	T- STRESS AT POINT 1
		(8)	(12-S1)	UV-STRESS AT POINT 1
		(9)	(11-S2)	V- STRESS AT POINT 2
		(10)	(22-S2)	U- STRESS AT POINT 2
		(11)	(33-S2)	T- STRESS AT POINT 2
		(12)	(12-S2)	UV-STRESS AT POINT 2
		(13)	(11-S3)	V- STRESS AT POINT 3
		(14)	(22-S3)	U- STRESS AT POINT 3
		(15)	(33-S3)	T- STRESS AT POINT 3
		(16)	(12-S3)	UV-STRESS AT POINT 3

ELEMENT TYPE	MAXIMUM NUMBER OF COMPONENTS	STRESS COMPONENT NUMBER	OUTPUT SYMBOL	D E S C R I P T I O N
		(17)	(V -S4)	V- STRESS AT POINT 4
		(18)	(U -S4)	U- STRESS AT POINT 4
		(19)	(T -S4)	T- STRESS AT POINT 4
		(20)	(UV-S4)	UV-STRESS AT POINT 4
5. EIGHT NODE BRICK	(12)	(1)	(XX-SL1)	XX-STRESS AT LOCATION 1
		(2)	(YY-SL1)	YY-STRESS AT LOCATION 1
		(3)	(ZZ-SL1)	ZZ-STRESS AT LOCATION 1
		(4)	(XY-SL1)	XY-STRESS AT LOCATION 1
		(5)	(YZ-SL1)	YZ-STRESS AT LOCATION 1
		(6)	(ZX-SL1)	ZX-STRESS AT LOCATION 1
		(7)	(XX-SL2)	XX-STRESS AT LOCATION 2
		(8)	(YY-SL2)	YY-STRESS AT LOCATION 2
		(9)	(ZZ-SL2)	ZZ-STRESS AT LOCATION 2
		(10)	(XY-SL2)	XY-STRESS AT LOCATION 2
		(11)	(YZ-SL2)	YZ-STRESS AT LOCATION 2
		(12)	(ZX-SL2)	ZX-STRESS AT LOCATION 2
6. PLATE/ SHELL	(6)	(1)	(XX-S/R)	XX-STRESS RESULTANT
		(2)	(YY-S/R)	YY-STRESS RESULTANT
		(3)	(XY-S/R)	XY-STRESS RESULTANT
		(4)	(XX-M/R)	XX-MOMENT RESULTANT
		(5)	(YY-M/R)	YY-MOMENT RESULTANT
		(6)	(XY-M/R)	XY-MOMENT RESULTANT
7. BOUN-DARY	(2)	(1)	(BDRY-F)	BOUNDARY FORCE
		(2)	(BDRY-M)	BOUNDARY MOMENT
8. THICK SHELL AND 3-DIM.	(42)	(1)	(SXX(0))	XX-STRESS AT CENTROID (0)
		(2)	(SYY(0))	YY-STRESS AT CENTROID (0)
		(3)	(SZZ(0))	ZZ-STRESS AT CENTROID (0)
		(4)	(SXY(0))	XY-STRESS AT CENTROID (0)
		(5)	(SYZ(0))	YZ-STRESS AT CENTROID (0)
		(6)	(SZX(0))	ZX-STRESS AT CENTROID (0)
		(7)	(SXX(1))	XX-STRESS AT CENTER OF FACE 1

ELEMENT TYPE	MAXIMUM NUMBER OF COMPONENTS	STRESS COMPONENT NUMBER	OUTPUT SYMBOL	DESCRIPTION
		(8)	(SYY(1))	YY-STRESS AT CENTER OF FACE 1
		(9)	(SZZ(1))	ZZ-STRESS AT CENTER OF FACE 1
		(10)	(SXY(1))	XY-STRESS AT CENTER OF FACE 1
		(11)	(SYZ(1))	YZ-STRESS AT CENTER OF FACE 1
		(12)	(SZX(1))	ZX-STRESS AT CENTER OF FACE 1
		(13)	(SXX(2))	XX-STRESS AT CENTER OF FACE 2
		(14)	(SYY(2))	YY-STRESS AT CENTER OF FACE 2
		(15)	(SZZ(2))	ZZ-STRESS AT CENTER OF FACE 2
		(16)	(SXY(2))	XY-STRESS AT CENTER OF FACE 2
		(17)	(SYZ(2))	YZ-STRESS AT CENTER OF FACE 2
		(18)	(SZX(2))	ZX-STRESS AT CENTER OF FACE 2
		(19)	(SXX(3))	XX-STRESS AT CENTER OF FACE 3
		(20)	(SYY(3))	YY-STRESS AT CENTER OF FACE 3
		(21)	(SZZ(3))	ZZ-STRESS AT CENTER OF FACE 3
		(22)	(SXY(3))	XY-STRESS AT CENTER OF FACE 3
		(23)	(SYZ(3))	YZ-STRESS AT CENTER OF FACE 3
		(24)	(SZX(3))	ZX-STRESS AT CENTER OF FACE 3
		(25)	(SXX(4))	XX-STRESS AT CENTER OF FACE 4
		(26)	(SYY(4))	YY-STRESS AT CENTER OF FACE 4
		(27)	(SZZ(4))	ZZ-STRESS AT CENTER OF FACE 4
		(28)	(SXY(4))	XY-STRESS AT CENTER OF FACE 4
		(29)	(SYZ(4))	YZ-STRESS AT CENTER OF FACE 4
		(30)	(SZX(4))	ZX-STRESS AT CENTER OF FACE 4
		(31)	(SXX(5))	XX-STRESS AT CENTER OF FACE 5
		(32)	(SYY(5))	YY-STRESS AT CENTER OF FACE 5
		(33)	(SZZ(5))	ZZ-STRESS AT CENTER OF FACE 5
		(34)	(SXY(5))	XY-STRESS AT CENTER OF FACE 5
		(35)	(SYZ(5))	YZ-STRESS AT CENTER OF FACE 5
		(36)	(SZX(5))	ZX-STRESS AT CENTER OF FACE 5
		(37)	(SXX(6))	XX-STRESS AT CENTER OF FACE 6
		(38)	(SYY(6))	YY-STRESS AT CENTER OF FACE 6
		(39)	(SZZ(6))	ZZ-STRESS AT CENTER OF FACE 6
		(40)	(SXY(6))	XY-STRESS AT CENTER OF FACE 6
		(41)	(SYZ(6))	YZ-STRESS AT CENTER OF FACE 6
		(42)	(SZX(6))	ZX-STRESS AT CENTER OF FACE 6

* *

9. PIPE

A. TANGENT (12)

(1)	(PX(I))	X-FORCE AT END I
(2)	(VY(I))	Y-SHEAR AT END I
(3)	(VZ(I))	Z-SHEAR AT END I
(4)	(TX(I))	X-TORQUE AT END I
(5)	(MY(I))	Y-MOMENT AT END I
(6)	(MZ(I))	Z-MOMENT AT END I
(7)	(PX(J))	X-FORCE AT END J
(8)	(VY(J))	Y-SHEAR AT END J
(9)	(VZ(J))	Z-SHEAR AT END J
(10)	(TX(J))	X-TORQUE AT END J
(11)	(MY(J))	Y-MOMENT AT END J
(12)	(MZ(J))	Z-MOMENT AT END J

B. BEND (18)

(1)	(PX(I))	X-FORCE AT END I
(2)	(VY(I))	Y-SHEAR AT END I
(3)	(VZ(I))	Z-SHEAR AT END I
(4)	(TX(I))	X-TORQUE AT END I
(5)	(MY(I))	Y-MOMENT AT END I
(6)	(MZ(I))	Z-MOMENT AT END I
(7)	(PX(C))	X-FORCE AT CENTER OF ARC
(8)	(VY(C))	Y-SHEAR AT CENTER OF ARC
(9)	(VZ(C))	Z-SHEAR AT CENTER OF ARC
(10)	(TX(C))	X-TORQUE AT CENTER OF ARC
(11)	(MY(C))	Y-MOMENT AT CENTER OF ARC
(12)	(MZ(C))	Z-MOMENT AT CENTER OF ARC
(13)	(PX(J))	X-FORCE AT END J
(14)	(VY(J))	Y-SHEAR AT END J
(15)	(VZ(J))	Z-SHEAR AT END J
(16)	(TX(J))	X-TORQUE AT END J
(17)	(MY(J))	Y-MOMENT AT END J
(18)	(MZ(J))	Z-MOMENT AT END J

* *
* *

C. RESPONSE SPECTRUM ANALYSIS (NDYN.EQ.3)

This option combines all (NF) mode shapes and frequencies computed during the eigenvalue solution (Section VII.A) to calculate R.M.S. stresses/deflections due to an input displacement (or acceleration) spectrum. The input spectrum is applied in varying proportions in the global X,Y,Z directions. For the case of a non-zero cut-off frequency "COFQ" (Section VII.A), only those modes whose frequencies are less than COFQ will be combined in the R.M.S. analysis.

1. Control Card (3F10.0,I5)

notes	columns	variable	entry
(1)	1 - 10	FX	Factor for X-direction input
	11 - 20	FY	Factor for Y-direction input
	21 - 30	FZ	Factor for Z-direction input
			EQ.0; not acting
(2)	31 - 35	IST	Input spectrum type;
			EQ.0; displacement vs. period
			EQ.1; acceleration vs. period

NOTES/

(1) All three (3) direction factors may be non-zero in which case the entries represent the X,Y,Z components of the input direction vector.

(2) "IST" defines the type of spectrum table to be input immediately following. The spectral displacements ("S_d") and accelerations ("S_a") are assumed to be related as follows: $S_a = (4\pi^2 f^2)(S_d)$.

VII. DYNAMIC ANALYSES (continued)

C. RESPONSE SPECTRUM ANALYSIS (continued)

2. Spectrum Cards

a. heading card (12A6)

notes	columns	variable	entry
	1 - 72	HED(12)	Heading information used to label the spectrum table

b. control card (I5,F10.0)

notes	columns	variable	entry
	1 - 5	NPTS	Number of definition points in the spectrum table; GE.2
	6 - 15	SFTR	Scale factor used to adjust the displacement (or acceleration) ordinates in the spectrum table EQ.1.0; no adjustment

c. spectrum data (2F10.0)

notes	columns	variable	entry
(1)	1 - 10	T	Period (reciprocal of frequency)
(2)	11 - 20	S	Value of displacement (or acceleration if IST.EQ.1)

NOTES/

(1) Input one definition point per card; "NPTS" cards are required in this section. Cards must be arranged in ascending value of period.

(2) "S" is interpreted to be a displacement quantity if "IST" was input as zero. For IST.EQ.1, "S" is an acceleration value.

END OF DATA CASE INPUT (NDYN.EQ.3)

APPENDIX A - CONTROL CARDS AND DECK SET-UP FOR DYNAMIC ANALYSIS RE-START

The purpose of this appendix is to describe the procedure (including control cards and deck set-up) required for program re-start following an eigenvalue/eigenvector extraction analysis. The re-start option has been included in the program in order to make a repeated forced response or spectrum analysis possible without solving each time for the required eigensystem. For medium-to-large size models, eigenvalue solution is quite costly when compared to the forced response calculations; hence, excessive costs may be incurred if the entire job has to be re-run due to improper specification of forcing functions or input spectra, inadequate requests, etc. For small models (less than 100 nodes, say) the extra effort required for re-start is normally not justified.

A complete dynamic analysis utilizing the re-start feature requires that the job be run in two (2) steps:

JOB(1): Eigenvalue extraction solution only, after which program files TAPE1,TAPE2,TAPE7,TAPE8, and TAPE9 are saved on the re-start tape.

JOBS(2): Re-instatement of program files TAPE1,TAPE2,TAPE7,TAPE8, and TAPE9 from the re-start tape followed by a Dynamic Response Analysis (NDYN.EQ.-2) or a Response Spectrum Analysis (NDYN.EQ.-3).

For a given model, the first job [JOB(1)] creating the re-start tape is run only once. The re-start tape then contains all the initial information required by the program at the beginning of a forced response analysis. More than one second job [JOBS(2)] may be run using the re-start tape as initial input; i.e., the re-start 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) - EIGENVALUE SOLUTION/RE-START TAPE CREATION

Notes Card Deck

(1) Job number, 1, 200, 120000,300. User Name
(2) REQUEST, TP1,I. Reel No., Tape User Name
(3) COPYBF, TP1,SAP4
UNLOAD,TP1
(4) LGO,SAP4
REWIND,TAPE1,TAPE2,TAPE7,TAPE8,TAPE9
(5) REQUEST,RESTART,I. Reel No., Tape User Name; OUTPUT
(6) COPYBF,TAPE1,RESTART
COPYBF,TAPE2,RESTART
COPYBF,TAPE7,RESTART
COPYBF,TAPE8,RESTART
COPYBF,TAPE9,RESTART
(7) 7-8-9

PROBLEM DATA DECK:
I. HEADING CARD
II. MASTER CONTROL CARD with
(LL.EQ.0)
(NF.GE.1)
(NDYN.EQ.1)
(MODEX.EQ.0)
III. JOINT DATA
IV. ELEMENT DATA
V. CONCENTRATED MASS DATA
VI. ELEMENT LOAD MULTIPLIERS
VII. DYNAMIC ANALYSIS
A. Mode Shapes and Frequencies
blank card
blank card

(8) 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 binary version of program (TP1) is requested.
(3) Binary version of the program is copied onto a disk file (SAP4).
(4) Program is loaded and execution is initiated.
(5) A blank tape (RESTART) is requested.
(6) The contents of disk files TAPE1,TAPE2, etc. are copied onto tape RESTART.
(7) End-of-record card: 7,8,9 punched in column 1.
(8) End-of-file card: 6,7,8,9 punched in column 1.

JOB (2) - RE-START FOR RESPONSE HISTORY ANALYSIS (NDYN.EQ.-2)
or RESPONSE SPECTRUM ANALYSIS (NDYN.EQ.-3)

Notes Card Deck

```
     Job number, 1,200,120000,300.   User Name
     REQUEST, RESTART, I.   Reel No., User Name
     COPYBF,RESTART,TAPE1
     COPYBF,RESTART,TAPE2
(1)  COPYBF,RESTART,TAPE7
     COPYBF,RESTART,TAPE8
     COPYBF,RESTART,TAPE9
     REWIND,TAPE1,TAPE2,TAPE7,TAPE8,TAPE9
     UNLOAD,RESTART
     REQUEST,TP1,I.  Reel No., User Name
(2)  COPYBF,TP1,SAP4
     LGO,SAP4
     7-8-9

     PROBLEM DATA DECK
          I.    HEADING CARD
          II.   MASTER CONTROL CARD with
                (LL.EQ.0)
                (NF.GE.1)
                (NDYN.EQ.-2 or -3)
(3)             (MODEX.EQ.0)
          VII.  DYNAMIC ANALYSIS
             B.    Dynamic Response Analysis (NDYN.EQ.-2)
          or
                C.    Response Spectrum Analysis (NDYN.EQ.-3)
     blank card
     blank card

     6-7-8-9
```

NOTES/

(1) The disk files TAPE1,TAPE2, etc. are re-created using the information saved on tape RESTORE.

(2) The binary version of the program is again obtained from tape TP1.

(3) Normally, the number of frequencies ("NF") entered on the MASTER CONTROL CARD for a re-start case has the same value as was specified earlier when the eigenvalue problem was solved in JOB(1). If a value for the cut-off frequency ("COFQ") was entered on the "Mode Shapes and Frequencies" control card [in JOB(1)] and the program extracted fewer than "NF" frequencies (eigenvalues), then only the actual number of eigenvalues computed by the program in JOB(1) is specified for "NF" in this re-start run.

APPENDIX B: CONTROL CARDS AND DECK SET-UP FOR USE OF STARTING ITERATION VECTORS

In the dynamic analysis of large-order systems, the solution of the required eigensystem is normally the most expensive phase. The option described in this appendix demonstrates how it is possible to use NFØ previously calculated eigenvalues and vectors when the solution for NF ≥ NFØ eigenvalues and eigenvectors is required.

Assume that in Job(1), the solution for NFØ eigenvalues and eigenvectors was performed. At the end of this job, TAPE2 and TAPE7 must have been saved on a physical tape, say "RESTART". Assuming that in JOB(2) the solution of NF eigenvalues and eigenvectors is required, then prior to the execution of this job, tape RESTART needs to be copied onto TAPE10.

This procedure was performed with the following control cards on the CDC 6400 of the University of California at Berkeley:

JOB(1) - SOLUTION FOR NFØ EIGENVALUES/RESTART TAPE CREATION

Notes	Card Deck
(1)	Job No., 1,200,120000,500. User Name
	REQUEST,TP1,I. Reel No., Tape User Name
	CØPYBF,TP1,SAP4
	UNLOAD,TP1
(2)	REQUEST,TAPE2,NB
	REQUEST,TAPE7,NB
	LGØ,SAP4
	REWIND,TAPE2,TAPE7
(3)	REQUEST,RESTART,I. Reel No.,Tape User Name, OUTPUT
(4)	CØPYBR,TAPE2,RESTART,1
	CØPYBF,TAPE7,TP3
	7-8-9
	PROBLEM DATA DECK
	6-7-8-9

Notes/

(1) See Notes (1) - (4) in Appendix A.

(2) The computer is directed to write on disk files TAPE2 and TAPE7 in an unblocked format.

(3) A blank tape (RESTART) is requested onto which the contents of files TAPE2 and TAPE7 are to be written.

(4) The contents of files TAPE2 and TAPE7 are written as <u>one file</u> onto tape RESTART.

JOB(2) - SOLUTION FOR ADDITIONAL EIGENVALUES USING THE INFORMATION STORED ON TAPE "RESTART"

Notes	Card Deck
	Job No.,1,200,120000,500. User Name
(1)	REQUEST,RESTART,I. Reel No., Tape User Name
	REQUEST,TAPE10,NB
	REQUEST,TAPE2,NB
	REQUEST,TAPE7,NB
(2)	CØPYBF,RESTART,TAPE10
	UNLOAD,RESTART
(3)	REWIND,TAPE10
	REQUEST,TP1,I. Reel No., Tape User Name
	CØPYBF,TP1,SAP4
	LGØ,SAP4
	7-8-9
	PROGRAM DATA DECK
	6-7-8-9

Notes/

(1) TAPE10 (as TAPE2 and TAPE7 if they are to be used for further restarts,) is requested to be an unblocked file.

(2) The contents of tape RESTART are copied into TAPE10 as one file.

(3) Program execution.

EARTHQUAKE ENGINEERING RESEARCH CENTER REPORTS

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

EERC 68-1 Unassigned

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

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

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

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

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

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

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

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

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

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

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

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

Note: Numbers in parentheses are Accession Numbers assigned by the National Technical Information Service. Copies of these reports may be ordered from the National Technical Information Service, Springfield, Virginia, 22151. Either the accession number or a complete citation should be quoted on orders for the reports.

Revised 4/23/73

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

EERC 72-1 "Static and Earthquake Analysis of Three Dimensional Frame and Shear Wall Buildings" by E. L. Wilson and H. H. Dovey - 1972 (PB 212 589)

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

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

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

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

EERC 72-6 "Earthquake Response of Gravity Dams Including Reservoir Interaction Effects" by P. Chakrabarti and A. K. Chopra - 1972.

EERC 72-7 "Dynamic Properties of Pine Flat Dam", by D. Rea, C. Y. Liau and A. K. Chopra - 1972.

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

EERC 72-9 "Rate of Loading Effects on Uncracked and Repaired Reinforced Concrete Members", by V.V. Bertero, D. Rea, S. Mahin and M. Atalay - 1973

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

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

EERC 72-12 "SHAKE, a Computer Program for Earthquake Response Analysis of Horizontally Layered Sites", by P.B. Schnabel and J. Lysmer - 1972.

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

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

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

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

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

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

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

EERC 73-8 "Seismic Behavior of Spandrel Frames - A Review and Outline for Future Research", by R. Razani and J.G. Bouwkamp - 1973.

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

EERC 73-10 "Deconvolution of Seismic Response for Linear Systems", by R. B. Reimer - 1973.

EERC 73-11 "SAP IV Structure Analysis Program for Static and Dynamic Response of Linear Systems", by K. -J. Bathe, E. L. Wilson, and F. E. Peterson - 1973 (revised).

EERC 73-12 "Analytical Investigations of the Seismic Response of Tall Flexible Highway Bridges", by W. S. Tseng and J. Penzien - 1973.

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

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

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

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

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

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

EERC 73-19 "Olive View Medical Center Material Studies, Phase 1", by B. Bresler and V. Bertero - 1973.

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

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

EERC 73-22 "DRAIN-2D Users' Guide" by G. H. Powell - 1973.

EERC 73-23 "Earthquake Engineering at Berkeley - 1973" by D. Rea - 1973.

EERC 73-24 "Seismic Input and Structural Response During the 1971 San Fernando Earthquake" by R. B. Reimer, R. W. Clough, and J. M. Raphael - 1973.

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

EERC 73-26 "Investigation of the Failures of the Olive View Stairtowers During the San Fernando Earthquake and Their Implications on Seismic Design", by V. V. Bertero and Robert G. Collins - 1973.

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

EL METODO DEL ELEMENTO FINITO EN LA INGENIERIA MECANICA

A N E X O

DR. VICTOR HUGO MUCINO QUINTERO

ABRIL, 1982

AN ASME PUBLICATION
$3.00 per copy $1.50 to ASME Members

THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS
345 E 47 St., New York, N.Y. 10017

An Exact Condensation Procedure for Chain-Like Structures Using a Finite Element-Transfer Matrix Approach

V. H. Mucino
Research Asst.,
Department of Mechanical Engineering
Assoc. Mem. ASME

V. Pavelic
Professor of Mechanical Engineering,
Mem. ASME

The main objective of this study is to describe a new scheme to carry out the static or dynamic analysis of elastic systems using a combined Finite Element-Transfer Matrix Approach. The proposed scheme offers the advantage of automatic matrix size reduction without having to truncate degrees of freedom, and preserving the strain and kinetic energy throughout the condensation. Although limited to chain-like elastic systems, the method is generalized to non-repetitive configurations with substructures having intermediate active degrees of freedom.

Introduction

The analysis of large and complex systems often requires a discretization so refined that the resulting stiffness and mass matrices become too large for the computer to handle. To overcome this difficulty, several "reduction techniques" have been proposed, having as primary objective the size reduction of the system matrices, through a truncation of degrees of freedom (d.o.f.), which involves the selection of certain "master" and "slave" d.o.f., also known in literature as retained and truncated d.o.f., respectively.

Guyan [1] is credited with establishing the concepts involved in performing the reduction, which is based upon the assumption that for dynamic analysis, the kinetic energy of the lower frequency modes is less sensitive to the truncation than the kinetic energy of the higher frequency modes, while the strain energy is preserved through the truncation.

In this procedure, the problems involved are two-fold: first, the results are dependent on the ability and experience of the analyst, to arbitrarily select the master d.o.f. in such a way that the motion of the principal modes can be characterized adequately by the retained d.o.f., and second, that the truncation modifies to an extent the distribution of the inertial properties of the structure, which in turn introduces some error in the results obtained. Further, no criteria currently exists to relate the number and location of the retained d.o.f. and the error introduced by the truncation. Common sense, experience and technical intuition in some cases are about the only possible tools to come up with an efficient truncation, unless the problem in hand is fairly simple. However, for practical purposes, even though these techniques are used, they produce limited success results.

The idea of matrix condensation lends itself particularly well to the concept of substructuring, which involves the "Macrodiscretization" of a large system into a set of subsystems known as substructures, which in turn are discretized using a finite element method, having as its main purpose to extract the most significant modes and to assemble the system as a whole in terms of the principal modes of each substructure. This area received significant attention in the aerospace industry and is well documented under the subject of "Modal Synthesis Techniques." Hurty [2], Bamford [3] and Goldman [4], among others, have developed extensive studies in this area and the theory need not be repeated here.

These techniques have been well adapted to the present finite element practice, and several codes, such as NASTRAN [5], ANSYS [6] and SUPERB [7], among others, offer the features of "substructuring" and "dynamic condensation."

It is to be noted that the use of these techniques is primarily directed towards the dynamic analysis area, in which not only the stiffness matrix is stored, but also, the mass, and in some cases, the damping matrices are stored, thus reducing the problem size memory storage capacity requirements to enhance the computer analysis work.

While matrix methods of analysis have significantly contributed to the development of these techniques, particularly the "Direct Stiffness Method" [8], upon which the finite element method is based, other methods have not enjoyed the same degree of application, but may potentially be proved useful for the analysis of structures, such is the case for the "Transfer Matrix Method" [9], which can be viewed as a continuity function for an enclosed system with transferable boundaries. Its advantages and limitations are documented by Dimarogonas [10] and Fishelson [11], but it has had some successful applications for very particular types of problems, as have the studies published by Prohl [12], Leckie [13], and Lin and McDaniel [14].

Contributed by the Design Engineering Division of THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS for presentation at the Century 2 Design Technology Transfer Conference, San Francisco, Calif., Aug. 19-21, 1980. Manuscript received at ASME Headquarters March, 1980. Paper No. 80-C2/DET-129.

The generalization achieved by the finite element method and the correspondence or correlation between the "Direct Stiffness" and the "Transfer Matrix" methods prompted various researchers to investigate the possibility of combining the advantages of both methods. Pestel and Leckie [15], treated the field transfer matrix as a different way of expressing the stiffness matrix. Later Dukanish [16] presented a combined Finite Element-Transfer Matrix (FE-TM) Method for the dynamic analysis of tapered or rectangular plates. In his approach, a finite element formulation was used to obtain the stiffness and mass matrices for a strip of elements whose boundaries were successively connected and whose end boundaries were characterized by state vectors, as defined in the standard transfer matrix method. Then a transformation of matrices was performed as described by Pestel and Leckie [15] and an algorithm similar to that proposed by Holzer [17] was used to successively solve for the natural frequencies of the system. McDaniel and Eversole [18] followed a similar approach to treat a stiffened plate structure and gave some numerical values of merit in the computing time efficiency of the algorithm as compared with regular finite element formulation without condensation.

In this paper a further generalization for the FE-TM method is presented with special emphasis on the non-repetitive configuration, but still chain-like type of structures, without restricting the substructures to be of the same nature. A special feature, described herein, is the treatment given to the intermediate d.o.f. which are condensed into a more compact form rather than regarding them as slave or truncated d.o.f. Condensation in this sense implies that all the d.o.f. contribute to both kinetic and strain energy.

Theory

The Equations of Motion. The equations of motion of any elastic structure able to store energy in terms of elastic and inertial properties can be obtained from the applicable form of the Lagrange equation as follows:

$$\frac{\partial}{\partial t}\left[\frac{\partial L}{\partial \dot{x}_i}\right] - \frac{\partial L}{\partial x_i} = Q_i \qquad (1)$$

Where the Lagrangian function (L) is given by the following expression:

$$L = 1/2 \sum_{i=1}^{n} \sum_{j=1}^{n} M_{ij} \dot{X}_i \dot{X}_j - 1/2 \sum_{i=1}^{n} \sum_{j=1}^{n} K_{ij} X_i X_j \qquad (2)$$

In this expression, it is assumed that the characteristics of the system can be approximated by expressing the kinetic energy (first term), and the strain energy (second term) in terms of a finite number (n) of generalized coordinates of d.o.f.

The substitution of equation (2) in equation (1) yields the resulting equations of motion, which expressed in matrix notation have the following general form:

$$[M]\{\ddot{X}\} + [K]\{X\} = \{F(t)\} \qquad (3)$$

Systems Matrices and Substructures. In finite element practice, the mass matrix $[M]$ can be formulated using a lumped mass approach as described by Bisplinghoff et. al. [19]. This formulation results in a diagonal matrix.

Also, a consistent mass formulation can be used to describe the distributed mass properties of the system. Archer [20] introduced the concept of consistent mass matrix, and gave it a physical interpretation analogous to that of the stiffness matrix. The later approach results in a banded matrix and the natural frequencies obtained using this consistent mass formulation are upper bounds to the exact frequencies of the system.

The formulation of the equations of motion using either a lumped or consistent mass matrix, generally satisfy the requirements of minimum potential energy. The explicit form of the equations of motion is as follows:

$$\begin{bmatrix} m_{11} & m_{12} & \cdots & m_{1n} \\ m_{21} & m_{22} & \cdots & m_{2n} \\ m_{n1} & m_{n2} & \cdots & m_{nn} \end{bmatrix} \begin{Bmatrix} \ddot{x}_1 \\ \ddot{x}_2 \\ \ddot{x}_n \end{Bmatrix} + \begin{bmatrix} k_{11} & k_{12} & \cdots & k_{1n} \\ k_{21} & k_{22} & \cdots & k_{n2} \\ k_{n1} & k_{n2} & \cdots & k_{nn} \end{bmatrix} \begin{Bmatrix} x_1 \\ x_2 \\ x_n \end{Bmatrix} = \begin{Bmatrix} f_1 \\ f_2 \\ f_n \end{Bmatrix} \qquad (4)$$

This system of equations is applicable to any elastic structure if damping can be neglected. If finite elements are used to discretize the overall structure, and the system is composed of several substructures, the overall system matrices have the following form:

Nomenclature

- $\frac{\partial}{\partial t}$ = partial derivative with respect to time
- $\dot{x}_i$, $\dot{x}_j$ = generalized velocities
- x_i, x_j = generalized coordinates
- Q_i = generalized forces
- L = Lagrangian function
- $\{\ddot{X}\}$, $\{X\}$ = vector of generalized (accelerations, displacements)
- $\{\ddot{X}_m\}$, $\{X_m\}$ = vector of (accelerations, displacements) of master d.o.f.
- $\{\ddot{X}_s\}$, $\{X_s\}$ = vector of (accelerations, displacements) of slave d.o.f.
- $\{X_L\}$, $\{X_R\}$ = vector of d.o.f. of the (left, right) boundaries of a substructure
- $\{X_I\}$ = vector of intermediate d.o.f. of a substructure
- m_{ij} = mass coefficient associated with generalized coordinates "i" and "j"
- $[M]$ = global mass matrix
- $[M_{mm}]$
- $[M_{ms}]$ $[M_{sm}]$ = partitions of the global mass matrix corresponding to the master and slave d.o.f.
- $[M_{ij}]$
- K_{ij} = stiffness coefficient associated with generalized coordinates "i" and "j"
- $[K]$ = global stiffness matrix
- $[K_{mm}]$
- $[K_{ms}]$ $[K_{sm}]$ = partitions of the global stiffness matrix corresponding to the master and slave d.o.f.
- $[K_{ii}]$
- $[K_{LL}]$
- $[K_{LR}]$ $[K_{RL}]$ = partitions of the global stiffness matrix corresponding to the left and right boundaries d.o.f.
- $[K_{RR}]$
- R = order of the global stiffness matrix
- r_i = order of the substructure "i" stiffness matrix
- d.o.f. = number of degrees of freedom per node
- N = number of nodes at the interfaces

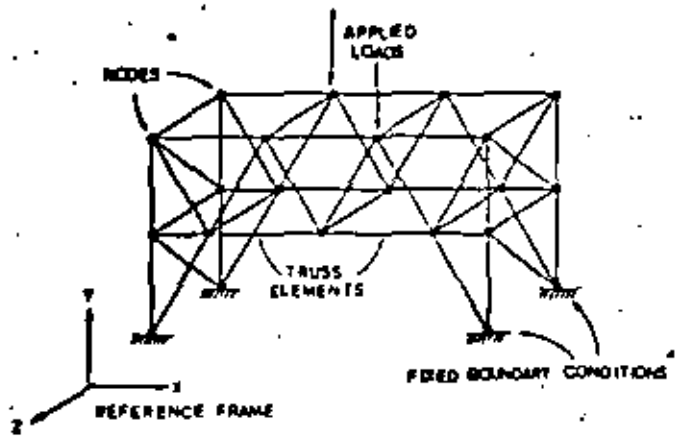


Fig. 1 Multidegree of freedom general structure with constrained boundary conditions and applied load vectors

$$\begin{bmatrix} \begin{bmatrix} M_1 \end{bmatrix} & & \\ & \begin{bmatrix} M_2 \end{bmatrix} & \\ & & \end{bmatrix}_{r\times r} \begin{Bmatrix} \ddot{X}_1 \\ \ddot{X}_I \\ \ddot{X}_2 \end{Bmatrix}_{r\times 1} + \begin{bmatrix} \begin{bmatrix} K_1 \end{bmatrix} & & \\ & \begin{bmatrix} K_2 \end{bmatrix} & \\ & & \end{bmatrix}_{r\times r} \begin{Bmatrix} X_1 \\ X_I \\ X_2 \end{Bmatrix}_{r\times 1} = \begin{Bmatrix} f_1 \\ f_I \\ f_2 \end{Bmatrix}_{r\times 1} \quad (7)$$

where:

$\{X_i\}$ are the degrees of freedom associated with subsystem "i" only $i = 1,2$ and $\{X_I\}$ are the degrees of freedom connecting the two substructures.

For the example used here, the order of the global matrices is given by the following relationship.

$$R = r_1 + r_2 - (\text{d.o.f.}) \times N \quad (8)$$

where:

r_i is the order of the ith substructure matrix, $i = 1,2$, N is

$$[M] = \begin{bmatrix} \begin{bmatrix} m_1 \end{bmatrix} & \begin{bmatrix} \end{bmatrix} & & \\ \begin{bmatrix} \end{bmatrix} & \begin{bmatrix} m_2 \end{bmatrix} & \begin{bmatrix} \end{bmatrix} & \\ & \begin{bmatrix} \end{bmatrix} & \begin{bmatrix} m_n \end{bmatrix} \end{bmatrix} \quad \text{and} \quad [K] = \begin{bmatrix} \begin{bmatrix} k_1 \end{bmatrix} & \begin{bmatrix} \end{bmatrix} & & \\ \begin{bmatrix} \end{bmatrix} & \begin{bmatrix} k_2 \end{bmatrix} & \begin{bmatrix} \end{bmatrix} & \\ & \begin{bmatrix} \end{bmatrix} & \begin{bmatrix} k_n \end{bmatrix} \end{bmatrix} \quad (5)$$

The overlap between the blocks represents the common boundaries between two adjacent substructures. Physically, the overlap between matrices represents the degrees of freedom connecting the two subsystems.

The order of these matrices is directly given by the total number of d.o.f. in the overall system. As an example, consider the structural system shown in Fig. 1.

If a lumped mass matrix is used, and no damping is assumed, the equations describing the motion of the structure under a harmonic driving force are as follows:

$$[M]_{r\times r} \{\ddot{X}\}_{r\times 1} + [K]_{r\times r} \{X\}_{r\times 1} = \{f\}_{r\times 1} \quad (6)$$

If the system as shown in Fig. 1 is assembled to another alike system, as shown in Fig. 2, such that some nodes are common to both systems, the resulting equations become:

the number of nodes at the interface and d.o.f. is the number of degrees of freedom per node.

In general, the substructures do not have to be of the same order, and several substructures can be assembled following the same procedure. The general expression for the order of the global matrices of the chain-like system shown in Fig. 3 is given by:

$$R = \sum_{i=1}^{n} r_i - \sum_{i=1}^{n-1} (\text{d.o.f.})_i \times N_i \quad (9)$$

It should be noted that the interfaces may or may not have the same number of nodes. The important fact to note here is that the more substructures there are in the system, the larger the order of the system matrices will be. This is not the case for the proposed method described in the following sections.

Nomenclature (cont.)

$\{F(t)\}$ = vector of applied time dependent forces
$\{F_m\}$ $\{F_s\}$ = vector of forces associated with (master, slave) d.o.f.
$\{F^*\}$ = reduced vector of applied forces after condensation
$\{F_L\}$ $\{F_R\}$ = vectors of forces for the (left, right) boundary d.o.f.
$\{F_I\}$ = vector of forces at the intermediate d.o.f.
$[D]$ = dynamic stiffness matrix
$[D_{mm}]$
$[D_{ms}]$ $[D_{sm}]$ = partitions of the global dynamic stiffness matrix corresponding to the master and slave d.o.f.
$[D_{ss}]$
$[D^*]$ = reduced dynamic stiffness matrix after condensation
$[T_i]$ = transfer matrix of substructure i
$[T_{11}]$
$[T_{12}]$ $[T_{21}]$ = partitions corresponding to the overall transfer matrix of a substructure with active intermediate d.o.f.
$[T_{22}]$
Z_R Z_L = state vectors of the (right, left) boundaries
$[A]$ $[B]$ $[C]$
$[D]$ $[E]$ $[F]$ = partitions of the global stiffness matrix corresponding to the (left, right and intermediate) d.o.f.
$[G]$ $[H]$ $[I]$
$[\psi_{11}]$
$[\psi_{21}]$ $[\psi_{12}]$ = partitions of the reduced set of equations after the intermediate d.o.f. have been eliminated in the global system
$[\psi_{22}]$
$\{R_1\}$ = vectors of remainder terms after the intermediate d.o.f. have been eliminated in the global system
$\{R_2\}$
$\{S_1\}$ = complementary vectors for the extended transfer matrix of equation (32)
$\{S_2\}$
w = frequency of vibration

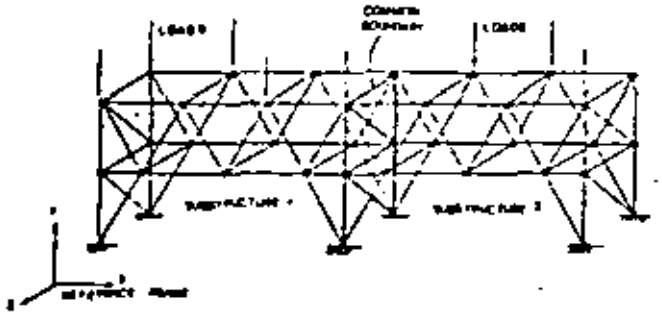


Fig. 2 Superstructure composed of two alike substructures having a common interface boundary

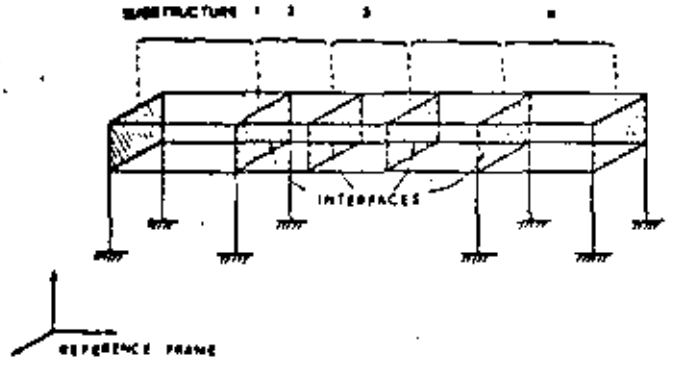


Fig. 3 Multisegmented superstructure with "n" substructures chain-like connected. The substructures are of a non-repetitive nature.

Condensation Techniques. As stated earlier, the condensation of d.o.f. has as its primary objective, the matrix size reduction and is conceptually done in four steps which are:

1 Selection of master set of d.o.f.
2 Partition of the system matrices.
3 Obtaining the solution for the master set of d.o.f.
4 Performing expansion or recovery for slave d.o.f.

The selection of the master set of d.o.f. is generally left to the analyst, who designates certain d.o.f. as being the most representative of the motion of the system. Once the master set has been specified, rearrangement of rows and columns is performed on the mass and stiffness matrices, in order to make the partitions given in the following equation:

$$\begin{bmatrix} Mmm & Mms \\ Msm & Mss \end{bmatrix} \begin{Bmatrix} \ddot{X}m \\ \ddot{X}s \end{Bmatrix} + \begin{bmatrix} Kmm & Kms \\ Ksm & Ksm \end{bmatrix} \begin{Bmatrix} Xm \\ Xs \end{Bmatrix} = \begin{Bmatrix} Fm \\ Fs \end{Bmatrix} \quad (10)$$

Where the subscript (m) indicates the terms associated with the "master set" of d.o.f., and subscript (s) indicates the terms associated with the "slave d.o.f." Assuming a harmonic solution, the following expression can be obtained:

$$\left(\begin{bmatrix} Kmm & Kms \\ Ksm & Kss \end{bmatrix} - w^2 \begin{bmatrix} Mmm & Mms \\ Msm & Mss \end{bmatrix} \right) \begin{Bmatrix} Xm \\ Xs \end{Bmatrix} = \begin{Bmatrix} Fm \\ Fs \end{Bmatrix} \quad (11)$$

this equation can be written as follows:

$$\begin{bmatrix} Dmm & Dms \\ Dsm & Dss \end{bmatrix} \begin{Bmatrix} Xm \\ Xs \end{Bmatrix} = \begin{Bmatrix} Fm \\ Fs \end{Bmatrix} \quad (12)$$

or

$$[D]\{X\} = \{F\} \quad (13)$$

Where the matrix $[D]$ is known as the "Dynamic Stiffness Expanding" equation (12), solving for $\{X_s\}$ and substituting, several times, the following system of equations is obtained:

$$[D^*]\{Xm\} = \{F^*\} \quad (14)$$

where

$$[D^*] = [Dmm] - [Dms]\ [Dss]^{-1}\ [Dsm] \quad (15)$$

and

$$\{F^*\} = \{Fm\} + [Dms]\ [Dss]^{-1}\ \{Fs\} \quad (16)$$

Equation (14) constitutes the "Reduced" set of equations, whose matrix order is dependent on the number of master d.o.f. The expanded solution can be obtained using the recovery equations; these equations are given by the following expression:

$$\{X_s\} = [Dss]^{-1}\{F_s\} - [Dsm]\{Xm\} \quad (17)$$

A special case in the condensation results when the master d.o.f. are chosen in such a way that there are no driving forces acting on the slave d.o.f.; in this case equations (16) and (17) become:

$$\{F^*\} = \{Fm\} \quad (18)$$

and

$$\{Xs\} = [Dss]^{-1}[Dsm]\{Xm\} \quad (19)$$

Aside from the inherent approximation in the discretization of the system, the solution expressed by equations (14) and (17) do not fully satisfy the Lagrange equation (1), since the kinetic energy is not minimized, considering the slave d.o.f. This argument is well documented by Guyan [21] and Clough [22], among others. Therefore, the truncation of d.o.f. introduces some error in the results obtained.

The Finite Element-Transfer Matrix Approach

Prior to the discussion and derivation of the proposed method, the fundamental concepts of combining the finite element and the transfer matrix method will be reviewed briefly. A more detailed description can be found in references [15, 16] and [18].

The application of the direct stiffness method to an elastic system subject to a static load vector results in the following equation:

$$[K]\{X\} = \{F\} \quad (20)$$

Now, let's consider the system described by equation (20) as a structure such that the degrees of freedom can be partitioned into "left" and "right" d.o.f. Then equation (20) becomes:

$$\begin{bmatrix} K_{ll} & K_{lR} \\ K_{Rl} & K_{RR} \end{bmatrix} \begin{Bmatrix} X_l \\ X_R \end{Bmatrix} = \begin{Bmatrix} F_l \\ F_R \end{Bmatrix} \quad (21)$$

By expanding this expression and solving for $\{X_R\}$ and $\{F_R\}$ in terms of $\{X_l\}$ and $\{F_l\}$, the following equations can be obtained:

$$\{X_R\} = [-(K_{lR})^{-1}\ [K_{ll}]]\ \{X_l\} + [K_{lR}]^{-1}\{F_l\} \quad (22)$$

and

$$\{F_R\} = [[K_{Rl}] - (K_{RR})\ [K_{lR}]^{-1}\ [K_{ll}]]\{X_l\}$$

$$+[K_{RR}]\,[K_{LR}]^{-1}\,\{F_L\} \tag{23}$$

which arranged in matrix form become:

$$\begin{Bmatrix} X_R \\ F_R \end{Bmatrix} = \left[\begin{array}{c|c} -[K_{LR}]^{-1}[K_{LL}] & [K_{LR}]^{-1} \\ \hline [K_{RL}]-[K_{RR}][K_{LR}]^{-1}[K_{LL}] & [K_{RR}][K_{LR}]^{-1} \end{array}\right] \begin{Bmatrix} X_L \\ F_L \end{Bmatrix} \tag{24}$$

or simplifying the notation, it can be written as follows:

$$\begin{Bmatrix} X_R \\ F_R \end{Bmatrix} = \begin{bmatrix} T_{11} & T_{12} \\ T_{21} & T_{22} \end{bmatrix} \begin{Bmatrix} X_L \\ F_L \end{Bmatrix} \tag{25}$$

or

$$\{Z_R\} = [T]\{Z_L\} \tag{26}$$

Equation (26) can be recognized as the transfer matrix relationship between the state vectors $\{Z_R\}$ and $\{Z_L\}$, which were derived directly from the stiffness relationship between the displacement vector $\{X\}$ and force vector $\{F\}$, given by equation (20).

In this example, only the filed transfer matrix was derived. In a similar manner, the point transfer matrix could be derived.

The Proposed Method of Analysis

Consider now, that the structure to be analyzed is such that it can be broken down into substructures which are chain-like connected as shown in Fig. 4. The substructures have certain number of d.o.f. which are at the interfaces and some which are intermediate between the two interfaces. Then taking the vector of d.o.f. for one substructure, and dividing it into three subsets:

$$X = \begin{Bmatrix} X_L \\ X_I \\ X_R \end{Bmatrix}$$

where

$\{X_L\}$ are the d.o.f. at the left interface
$\{X_I\}$ are the intermediate d.o.f., and
$\{X_R\}$ are the d.o.f. at the right interface

Using this partition in equation (13) applied to one substructure, the following expressions can be written:

$$\begin{bmatrix} A & B & C \\ D & E & F \\ G & H & I \end{bmatrix} \begin{Bmatrix} X_L \\ X_I \\ X_R \end{Bmatrix} = \begin{Bmatrix} F_L \\ F_I \\ F_R \end{Bmatrix} \tag{27}$$

solving for the X_I and substituting in the remaining equations, the following expressions are obtained:

$$\{[A]-[B][E]^{-1}[D]\}\{X_L\}$$

$$+\{[C]-[B][E]^{-1}[F]\}\{X_R\}+[B][E]^{-1}\{F_I\}=\{F_L\}$$

$$\{[G]-[H][E]^{-1}[D]\}\{X_L\}$$

$$+\{[I]-[H][E]^{-1}[F]\}\{X_R\}+[H][E]^{-1}\{F_I\}=\{F_R\} \tag{28}$$

which can also be written in matrix form as follows:

$$\begin{bmatrix} \psi_{11} & \psi_{12} \\ \psi_{21} & \psi_{22} \end{bmatrix} \begin{Bmatrix} X_L \\ S_R \end{Bmatrix} + \begin{Bmatrix} R_1 \\ R_2 \end{Bmatrix} = \begin{Bmatrix} F_L \\ F_R \end{Bmatrix} \tag{29}$$

where $\{\psi_{ij}\}$ and $\{R_i\}$ are the short hand notation of the matrices in the square brackets of equations (28).

By expanding and rearranging equation (29), it can be shown after various matrix manipulations that the left and right boundaries can be related by the following expression.

$$\begin{Bmatrix} X_R \\ F_R \end{Bmatrix} = \left[\begin{array}{c|c} -\psi_{12}^{-1}\psi_{11} & \psi_{12}^{-1} \\ \hline \psi_{21}-\psi_{22}\psi_{12}^{-1}\psi_{11} & \psi_{22}\psi_{12}^{-1} \end{array}\right] \begin{Bmatrix} X_L \\ F_L \end{Bmatrix} + \left[\begin{array}{c} -\psi_{12}^{-1}R_1 \\ \hline \psi_{22}\psi_{12}^{-1}R_1+R_2 \end{array}\right] \tag{30}$$

or simplifying the notation:

$$\begin{Bmatrix} X_R \\ F_R \end{Bmatrix} = \begin{bmatrix} T_{11} & T_{12} \\ T_{21} & T_{22} \end{bmatrix} \begin{Bmatrix} X_L \\ F_L \end{Bmatrix} + \begin{Bmatrix} S_1 \\ S_2 \end{Bmatrix} \tag{31}$$

where T_{ij} correspond to the terms included in the partitions of the matrix of equation (30).

Adding one dummy equation to the system, i.e., $(1 = 1)$ the following equation can be obtained:

$$\begin{Bmatrix} X_R \\ F_R \\ 1 \end{Bmatrix} = \begin{bmatrix} T_{11} & T_{12} & S_1 \\ T_{21} & T_{22} & S_2 \\ 0 & 0 & 1 \end{bmatrix} \begin{Bmatrix} X_L \\ F_L \\ 1 \end{Bmatrix} \tag{32}$$

which is the expanded transfer matrix relating the state of the left and right boundaries through the intermediate degrees of freedom.

For dynamic analysis, the stiffness matrix $[K]$ can be substituted by the dynamic stiffness matrix given in equations (11) and (13). The procedure then to obtain the transfer matrix is analogous to that just described.

Once the transfer matrix has been formulated for each substructure, the assembly of the system as a whole is made following standard transfer matrix method procedures.

The relation between the left and right interface state vectors, of a substructure in a chain-like connected system is given by equation (32), which in short hand notation has the form of equation (26) repeated here for convenience of the reader.

$$\{Z_R\}_n = [T_n]\{Z_L\}_n \tag{26}$$

When two substructures are linked together, the right interface of substructure (n), becomes also the left interface of substructure $(n+1)$, therefore:

$$\{Z_L\}_{n+1} = \{Z_R\}_n \tag{33}$$

The relationship between state vectors for substructure $(n+1)$ is then

$$\{Z_R\}_{n+1} = [T_{n+1}]\{Z_L\}_{n+1} \tag{34}$$

Combining equations (26), (33) and (34) the equation results:

$$\{Z_R\}_{n+1} = [T_{n+1}][T_n]\{Z_L\}_1 \tag{35}$$

In this case, the general expression for the total system with "n" substructures as shown in Fig. 4 is given by

$$\{Z\}_n = [T_n][T_{n-1}]\cdots[T_2][T_1]\{Z_0\} \tag{36}$$

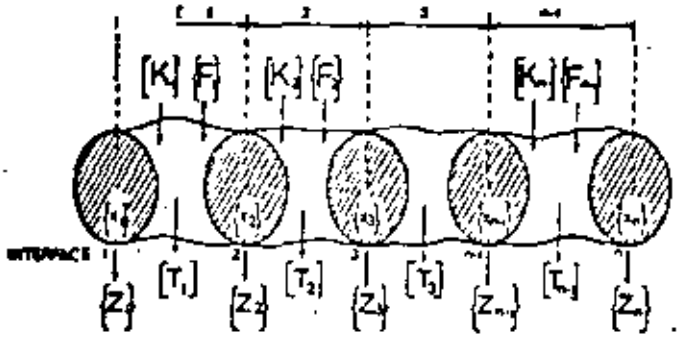


Fig. 4 A chain-like connected system, composed of "n" substructures or stations, with defined stiffnesses $[k]_i$ and applied force vectors $\{F\}_i$ and state vectors $\{Z\}$ defined at the connecting boundaries

or

$$\{Z\}_n = [U]\{Z_0\} \tag{37}$$

where

$$[U] = [T_n][T_{n-1}] \ldots [T_1] \tag{38}$$

It should be noted that by multiplying the transfer matrices $[T_i]$, the order of matrix $[U]$ does not increase but remains compatible with the matrices being multiplied. If the system is such that all substructures have the same transfer matrix the order of the system transfer matrix $[U]$ remains the same.

This feature results in a reduced size matrix which embodies the entire system. The end state vectors $\{Z\}_n$ and $\{Z\}_1$ contain the boundary conditions of the structure in terms of displacements in the direction of the d.o.f. and forces at the nodes located in the interfaces.

Once the system has been assembled, this is when all the transfer matrices have been multiplied as expressed by equation (38). Subsequently the boundary conditions have to be satisfied by solving for the unknown terms in the end state vectors. After the end state vectors are known the intermediate state vectors can be obtained by recursively applying equation (26) until all state vectors are known.

For dynamic analysis, the dynamic stiffness matrix contains the frequency terms. Those frequency values which satisfy the boundary conditions are the natural frequencies for the system. The procedure to obtain the natural frequencies and the modes is similar to that proposed by Holzer [17]. In this method a natural frequency value is assumed for which the system is "treated," where the test consists in multiplying the transfer matrices and observing whether or not the boundary conditions are satisfied. If the boundary conditions are not satisfied, a different "test" frequency must be chosen; and calculations must be repeated, until the boundary conditions are satisfied producing an actual natural frequency of the system. This iterative procedure is shown schematically in the computer flowchart in Fig. 5.

Operational Aspects of the Finite Element-Transfer Matrix Method

Due to the inherent complications of matrix operations, it is necessary to point out some important aspects to be considered in developing a suitable computer algorithm.

The proposed method is oriented towards the analysis of complex systems which can be modeled by means of substructures connected in a chain-like manner, for instance, beams with intermediate supports, bridges, multithrow crankshafts, etc. The complications involved in obtaining the stiffness and mass matrices are directly associated with the type of finite elements used to describe the structure. Several books [23, 24 among others] are available with detailed descriptions of the procedures required to obtain the system matrices of equations (3) and (4).

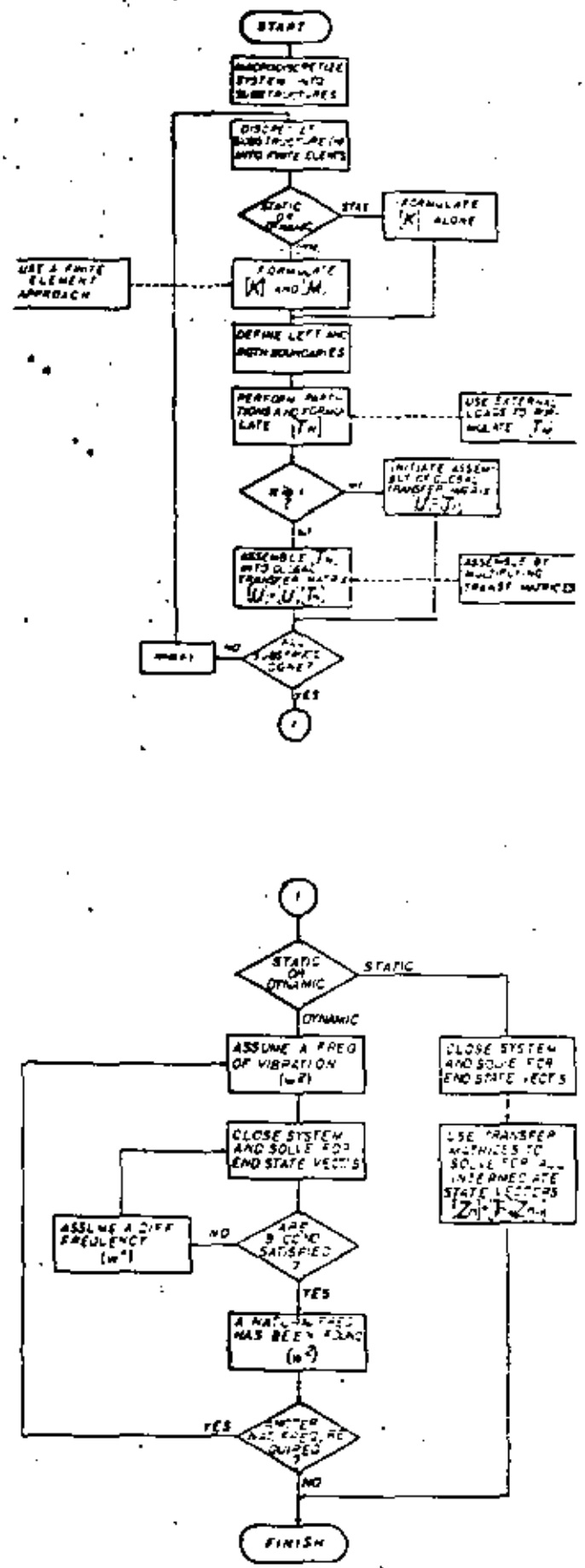


Fig. 5 Computer implementation algorithm for the generalized finite element transfer matrix method for the static or dynamic analysis of chain-like structures

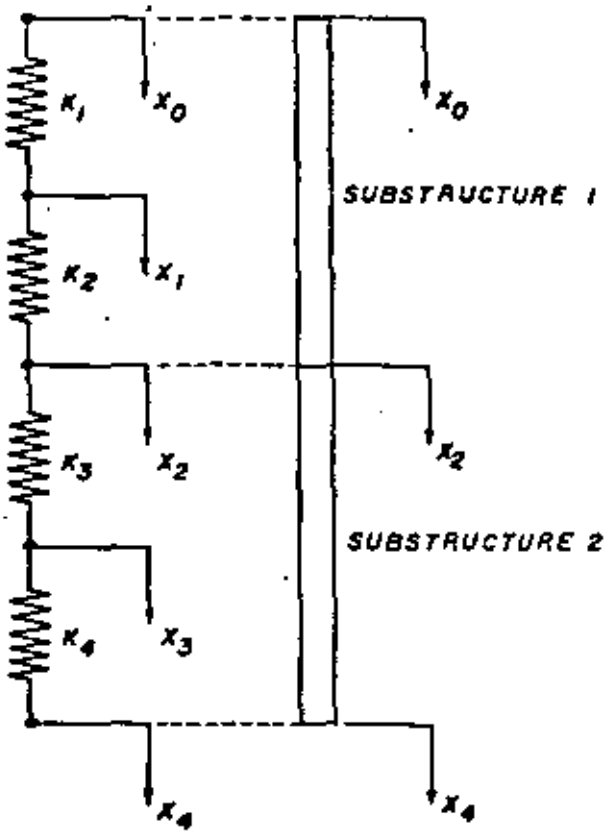


Fig. 6 Simple chain-like system and synthesis by substructuring

The derivation of the transfer matrix for a substructure, however, requires the inversion of submatrix $[E]$ in equation (27) and $[\psi_{12}]$ in equation (30). These inversions are sources of some numerical errors. However, these inversions are done only once for each substructure and are not affected by the load vector. This is an advantage, especially if all the substructures have the same configuration. This is the case in periodic structures such as those treated by Emgels and Mairovitch [25]. Note also that the order of these matrices is smaller than the order of the stiffness and mass matrices for a given substructure, since only the intermediate d.o.f. are considered in the matrix to be inverted.

Finally, it can be noted that the matrix $[A]$ is banded and it does not require full storage in the computer memory. It is the assembly of the various substructures that makes storage requirements increase, since the order of the global matrices increases too. In the *FE-TM* method the substructure matrix $[T_i]$ is fully populated and requires full storage in the computer memory, but the global transfer matrix $[U]$ does not increase in size since it results from consecutive matrix multiplications as indicated by equation (36).

Some other aspects in obtaining the solution of the system are parallel to those involved in standard transfer matrix applications and discussion may be found, for instance, in papers by Pestel and Leckie [9] or [15].

Although the proposed method is oriented towards more complex structures, a simple example is given in the appendix with the purpose of illustrating the treatment of two substructures which have a common boundary and are chain-like connected. In this example, the stiffness matrix $[A]$ is first derived for each element in the substructure and then assembled using the standard direct stiffness method. Subsequently, the transfer matrix $[T]$ is formulated for each substructure by applying the transformations of equations (28), (30) and (32) to the stiffness matrix found earlier. Finally, global transfer matrix $[U]$ is obtained by multiplying the transfer matrices of each substructure.

Treatment of a larger and more complex system is analogous to that described in this example and the use of the finite element method allows more complex elements to be used to discretize the substructures and to obtain the substructure stiffness and mass matrices. Such applications have been done by the authors using 3-D isoparametric solid elements and will be reported in our next papers which are now in preparation.

Summary and Conclusions

A brief description of the currently available condensation and substructuring techniques has been made, pointing out some of the main features of these techniques and how they apply to the actual type of systems addressed in this study. The correlation between the stiffness and transfer matrix for simple elements was discussed, and a generalization of the concept was developed for complex substructures having intermediate active d.o.f. A detailed derivation of the equations involved in the proposed method was made, and a general computer algorithm flowchart (Fig. 5) was presented showing the main steps required for computer implementation of this method for practical applications to an actual physical system.

It is important to note that special attention must be paid to the numerical aspects involved in the matrix operations, in order to reduce the possibility of numerical error.

From inspection of the equations derived, and from the example given in the appendix, the following conclusions can be drawn which apply for chain-like connected systems.

1 Matrix reduction can be achieved by applying the FE-TM approach to the substructures of a system.

2 No selection of Master and Slave degrees of freedom is required in the FE-TM method, thus reducing the possibility of misrepresentation of the system.

3 All the degrees of freedom are included in the formulation of the reduced equations, and no sacrifice is required in approximating the kinetic energy of the system.

4 Intermediate active d.o.f. can be properly condensed, along with any external loads acting on them as shown by equation (28).

5 The advantages of the finite element method apply to the proposed method in terms of discretizing the system using substructures.

6 The advantages of the Transfer Matrix method also apply to the proposed method, specifically the fact that by multiplying the transfer matrices, the order of the resulting matrix does not increase.

Future improvements in this area perhaps will include the formulation of transfer matrices for structures with complex finite elements and in addition, the inclusion of branches in the system may be considered.

Some of this work is already in progress at this institution, specifically, transfer matrix for structures modeled with 3D-solid finite elements.

References

1 Guyan, R. J., "Reduction of Stiffness and Mass Matrices," *A.I.A.A. Journal*, Vol. 3, No. 2, Feb. 1965, p. 380.

2 Hurty, W. C., "Introduction to Modal Synthesis Techniques," Paper No. 1 of XXII Special Publication 88, No. Hou-72, 1971, Synthesis of Vibrating Systems.

3 Hansford, R. M., "A Modal Combination Program for Dynamic Analysis of Structures," Technical Memorandum 33-290, Jet Propulsion Laboratory, Pasadena, Calif., July 1966.

4 Goldman, R. L., "Vibration Analysis by Dynamic Partitioning," *A.I.A.A. Journal*, Vol. 7, No. 6, June 1969, p. 1152.

5 MacNeal, R. H., "The NASTRAN Theoretical Manual," (Level 15.5), The MacNeal Schendler Corporation, Los Angeles, Ca., 1974.
6 DeSalvo, G. J., and Swanson, J. A., "The ANSYS User's Manual," Swanson Analysis Systems, Inc., Elizabeth, Pa., 1974.
7 SUPERB's User Manual, Structural Dynamics Research Corporation, Milford, Ohio, 1974.
8 Clough, R. W., and Penzien, J., *Dynamics of Structures*, McGraw-Hill, N.Y., 1975, p. 158.
9 Leckie, F. A., and Pestel, E., "Transfer Matrix Fundamentals," *Intern. J. Mech. Sci.*, Vol. 2, 1960, pp. 137-167.
10 Dimarogonas, A. D., *Vibration Engineering*, West Publishing Co., N.Y., 1976, p. 406.
11 Ehrleman, R. L., *Flexible Rotor-Bearing System Dynamics*, ASME Special Publication, Book No. H00082, 1972.
12 Prohl, M. A., "A General Method for Calculating Critical Speeds of Flexible Rotors," Transactions ASME, Vol. 67, 1945, pp. A142, A148.
13 Leckie, F. A., "The Application of Transfer Matrices to Plate Vibrations," *Ingenieur-Archiv*, Vol. XXXII, 1963, pp. 100-111.
14 Lin, Y. K., and McDaniel, T. J., "Dynamics of Beam-Type Periodic Structures," ASME, *Journal of Engineering for Industry*, Nov. 1969, p. 1133.
15 Pestel, E. C., and Leckie, F. A., *Matrix Methods in Elastodynamics*, McGraw-Hill, N.Y., 1963, p. 148.
16 Dokainish, M. A., "A New Approach for Plate Vibrations: Combination of Transfer Matrix and Finite Element Technique," ASME, *Journal of Engineering for Industry*, May 1972, pp. 526-530.
17 Holzer, H., "Die Berechnung der Drehschwingungen," Springer Verlag OHG, Berlin, 1921, Republished by J. W. Edwards, Pub., Inc., Ann Arbor, Mich.
18 McDaniel, T. J., and Eversole, K. B., "A Combined Finite Element-Transfer Matrix Structural Analysis Method," *Journal of Sound and Vibration*, Vol. 51, No. 2, 1977, pp. 157-169.
19 Bisplinghoff, R. L., Ashley, H., and Halfman, R., *Aeroelasticity*, Addison-Wesley Publishing Co., Inc., Cambridge, Mass., 1955.
20 Archer, J. S., "Consistent Mass Matrix for Distributed Mass Systems," Proc. ASCE, *Journal of the Structural Division*, Vol. 89, No. ST4, Aug. 1963.
21 Gavan, R. J., "Distributed Mass Matrix for Plate Element Bending," Technical Note, *A.I.A.A. Journal*, Sept. 1964, p. 567.
22 Clough, R. W., and Penzien, J., *Dynamics of Structures*, McGraw-Hill, N.Y., 1975, p. 235.
23 Zienkiewicz, O. C., *The Finite Element Method*, McGraw-Hill, N.Y., 1977.
24 Cook, R. D., *Concepts and Applications of Finite Element Analysis*, John Wiley & Sons, Inc., N.Y., 1974.
25 Engels, R. C., and Merovitch, L., "Response of Periodic Structures—by Modal Analysis."

APPENDIX

Transfer Matrix derivation for the two substructure system shown, Fig. 6.

Stiffness Matrix of Substructure 1:

$$\begin{bmatrix} K_1 & -K_1 & 0 \\ -K_1 & K_1+K_2 & -K_2 \\ 0 & -K_2 & K_2 \end{bmatrix} \begin{Bmatrix} X_0 \\ X_1 \\ S_2 \end{Bmatrix} = \begin{bmatrix} f_0 \\ f_1 \\ f_2 \end{bmatrix}$$

Stiffness Matrix of Substructure 2:

$$\begin{bmatrix} K_3 & -K_3 & 0 \\ -K_3 & K_3+K_4 & -K_4 \\ 0 & -K_4 & K_4 \end{bmatrix} \begin{Bmatrix} X_2 \\ X_3 \\ X_4 \end{Bmatrix} = \begin{bmatrix} f_2 \\ f_3 \\ f_4 \end{bmatrix}$$

Assembled Overall System Stiffness Matrix:

$$\begin{bmatrix} K_1 & -K_1 & 0 & 0 & 0 \\ -K_1 & K_1+K_2 & -K_2 & 0 & 0 \\ 0 & -K_2 & -K_2+K_3 & -K_3 & 0 \\ 0 & 0 & -K_3 & K_3+K_4 & -K_4 \\ 0 & 0 & 0 & -K_4 & K_4 \end{bmatrix} \begin{Bmatrix} X_0 \\ X_1 \\ X_2 \\ X_3 \\ X_4 \end{Bmatrix} = \begin{Bmatrix} f_0 \\ f_1 \\ f_2 \\ f_3 \\ f_4 \end{Bmatrix}$$

Partitions on Substructure 1 Stiffness Matrix for Transfer Matrix Formulation:

$$\left[\begin{array}{c|c|c} K_1 & -K_1 & 0 \\ \hline -K_1 & K_1+K_2 & -K_2 \\ \hline 0 & -K_2 & K_2 \end{array}\right] \left[\begin{array}{c} X_0 \\ \hline X_1 \\ \hline X_2 \end{array}\right] = \left[\begin{array}{c} f_0 \\ \hline f_1 \\ \hline f_2 \end{array}\right]$$

Therefore

$$A = K_1 \quad B = -K_1 \quad C = 0$$
$$D = -K_1 \quad E = K_1+K_2 \quad F = -K_2$$
$$G = 0 \quad H = -K_2 \quad I = K_2$$

Then, using equations (30) and (32)

$$\psi_{11} = \frac{K_1K_2}{K_1+K_2} \qquad \psi_{12} = \frac{K_1K_2}{K_1+K_2} \qquad R_1 = -\frac{K_1f_1}{K_1+K_2}$$

$$\psi_{12}' = -\frac{K_1+K_2}{K_1K_2}$$

$$\psi_{21} = -\frac{K_1K_2}{K_1+K_2} \qquad \psi_{22} = \frac{K_1K_2}{K_1+K_2} \qquad R_2 = -\frac{K_2f_1}{K_1+K_2}$$

and

$$T_{11} = -\left(-\frac{K_1+K_2}{K_1K_2}\right)\left(\frac{K_1K_2}{K_1+K_2}\right) = 1 \qquad T_{12} = -\frac{K_1+K_2}{K_1K_2}$$

$$T_{21} = -\left(\frac{K_1K_2}{K_1+K_2}\right) + \left(\frac{K_1K_2}{K_1+K_2}\right)\left(\frac{K_1+K_2}{K_1K_2}\right)\left(\frac{K_1K_2}{K_1+K_2}\right) = 0 \qquad T_{22} = \left(\frac{K_1K_2}{K_1+K_2}\right)\left(-\frac{K_1+K_2}{K_1K_2}\right) = -1$$

$$S_1 = -\left(-\frac{K_1+K_2}{K_1K_2}\right)\left(\frac{-K_1f_1}{K_1+K_2}\right) = -\frac{f_1}{K_2}$$

$$S_2 = \left(\frac{K_2K_1}{K_1+K_2}\right)\left(-\frac{K_1+K_2}{K_1K_2}\right)\left(\frac{-K_1f_1}{K_1+K_2}\right) + \left(\frac{-K_2f_1}{K_1+K_2}\right) = \frac{f_1(K_1-K_2)}{(K_1+K_2)}$$

The Transfer Matrix for Substructure 1 is

$$[T_1] = \begin{bmatrix} 1-\frac{K_1+K_2}{K_1K_2} & \frac{f_1}{K_2} \\ 0 & -1 & \frac{f_1(K_1-K_2)}{(K_1+K_2)} \\ 0 & 0 & 1 \end{bmatrix}$$

Therefore

$$\begin{Bmatrix} X_2 \\ f_2 \\ 1 \end{Bmatrix} = \begin{bmatrix} 1-\frac{K_1+K_2}{K_1K_2} & \frac{f_1}{K_2} \\ 0 & -1 & \frac{f_1(K_1-K_2)}{(K_1+K_2)} \\ 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} X_0 \\ f_0 \\ 1 \end{Bmatrix}$$

The Global Transfer Matrix is

$$\begin{Bmatrix} X_4 \\ f_4 \\ 1 \end{Bmatrix} = \begin{bmatrix} 1 & \left(\frac{K_3+K_4}{K_3K_4}-\frac{K_1+K_2}{K_1K_2}\right)\left(-\frac{f_1}{K_2}-\frac{K_1+K_4}{K_1K_4}\left(\frac{f_1(K_1-K_2)}{(K_1+K_2)}\right)-\frac{f_1}{K_4}\right) \\ 0 & 1 & \left(-\frac{f_1(K_1-K_2)}{(K_1+K_2)}+\frac{f_1(K_3-K_4)}{K_3+K_4}\right) \\ 0 & 0 & 1 \end{bmatrix} \begin{Bmatrix} X_0 \\ f_0 \\ 1 \end{Bmatrix}$$

RELACION DE ASISTENCIA EN EL CURSO "EL METODO DEL ELEMENTO FINITO EN LA INGENIERIA MECANICA" LLEVADO A CABO DEL 12 AL 16 DE ABRIL - DE 1982.

1.- Marco Antonio Alarcón Ramírez
Profesor asociado B M.T.
Facultad de Estudios Superiores
Cuautitlán
Carretera Cuautitlán Teoloyucan
Km. 3
872-03-45
872-03-55

Norte 92 # 6507
San Pedro El Chico
C.P. 07480
760-54-95

2.- Armando Andrade García
Becario Tesis Licenciatura
Instituto de Investigaciones - Electricas
Interior Internado Palmira
4-38-11

Caminito de Contreras No. 8
Sta. Maria Cuernavaca, Morelos
C.P. 1200
3-73-04

3.- Guillermo Canseco López
Encargado de la Rama de Ingenieria Mecanica
Av. Insurgentes Sur 1806, 4o. Piso
Florida
C.P. 03010
534-87-88

Rafael Oliva No. 15
Coyoacan
C.P. 04210
544-57-58

4.- Alejandro Castañeda N.
Instituto Mexicano del Petroleo
Ave. Eje Central Lazaro Cardenas
No. 152
C.P. 14805
567-66-00
567-91-00

5.- Jorge Castillo Rivera
Profesor de Asignatura B
ENEP Cuautitlán
Cuautitlán Edo. de Méx.

Dios Rio No. 29
Los parques Ixcalli
Edo. de México
277-83-40

6.- Elías Iza Calás
Jefe Ofna. Ingeniería Mecánica
Comisión Federal de Electricidad
Av. Apaseo s/n
Irapuato, Gto.
727-27

Paseo Mirador del Valle No. 5598 B
Irapuato, Gto.
6-65-82

7.- Dagoberto de la Serna V
Facultad de Ingeniería, UNAM

8.- Lucia Teresa Matus Santana
Profesor T.C.
Unidad Profesional de Ingeniería Química e Industrial ext. IPN
Unidad Profesional de Ingeniería del IPN Edificio 3
Lindavista
754-40-44

Piñón No. 284
Nueva Santa Maria
C.P. 02830
556-26-36

9.- Margarita Navarrete Montesinos
Ayudante de Investigador
Ciudad Universitaria
Instituto de Ingeniería UNAM

Sur 119 # 516 .
Col.-Esc. 201
Iztapalapa
C.P. 09060
582-64-36

10.- Arturo Noriega Salgado
Evaluación en Ingeniería mecánica
Comisión Nacional de Seguridad Nuclear y Salvaguardia
Insurgentes Sur No. 1806
Col. Florida
C.P. 01030
524-26-65

Calle Emil Zapate Edif. 25-301
Heroés de la Revolución (Huizachal) Edo. de Méx.

11.- Ignacio Pérez Pérez
Ingenieros Civiles Asociados, S.A.
Mineria No. 145
516-04-60

Independencia No. 55
Zacahitzco
559-53-54

12.- Joaquín Sánchez Becerra
Gerente de Ingeniería
Tasmi, S. A.
Carretera MTY-Monclova KM 11.5
El Carmen N. L.
76-52-18

Av. Central No. 1241
Central
71-70-58

13.- Fco. Gabriel Sánchez Aldana
Proyectista
Diseños Avanzados
Cantú 9-102
Anzures
545-27-22

Icazbalceta No. 27
San Rafael
Cuauhtemoc
535-23-89

14.- Jorge Silva Ballesteros
Jefe de la Rama de Estructuras
Comisión Nacional de Seguridad Nuclear
Insurgentes Sur No. 1806
Florida
524-26-65

Marmoleria No. 324
20 de Noviembre
789-89-75