REPORT NO
UCSESM 70-20

STRUCTURES AND MATERIALS RESEARCH
DEPARTMENT OF CIVIL ENGINEERING

SAP

A General

Structural Analysis Program

By
EDWARD L. WILSON

REPORT TO
WALLA WALLA DISTRICT
U.S. ENGINEERS OFFICE

SEPTEMBER
1970

STRUCTURAL ENGINEERING LABORATORY
UNIVERSITY OF CALIFORNIA
BERKELEY CALIFORNIA

SAP

A General Structural Analysis Program

by

Edward L. Wilson

Report To

Walla Walla District
U.S. Engineers Office

Contract DACW 68-67-C-004

Structural Engineering Laboratory
University of California
Berkeley, California

September 1970

FOREWORD

The development of an effective computer program for structural analysis requires a knowledge from three scientific disciplines -- structural mechanics, numerical analyses and computer programming. The selection of accurate and efficient elements requires a modern background in structural mechanics. Because new elements are continuously being developed the computer program should have a simple mechanism for the insertion of new elements. The efficiency of the program depends on the numerical programming techniques employed. For example, incorrect techniques for the solution of equations may increase the execution time for a program by a factor of 1000. The most important aspects of programming are machine independence and optimum allocation of storage. If one is careful very efficient programs can be developed utilizing standard FORTRAN techniques.

In my opinion, all computer programs for structural analysis are 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. Therefore, it is very important that a large investment is not required for the development of a single program. However, if a program is small, machine independent, and has the ability to be easily modified it may have a longer life.

Many computer programs for the analysis of large complex structures have been developed during the past several years. Each of these programs represents the result of large expenditures of energy and money. One of these programs required approximately 75 man-years and two million dollars before completion; however, the resulting program

does not have modern structural elements and is not particularly efficient in operation. Also, it is partially machine dependent.

The computer program presented in this report is the result of over 10 years of research and development experience. However, the basic computer code was developed in a three-month period by a few engineers. In the past several months additional structural elements and a dynamic option have been added. The total development effort of the computer code to date has been approximately two man-years. In my opinion, however, the resulting program is one of the most powerful and efficient programs for the linear elastic analysis of complex structural systems that has been developed to date. Nevertheless, I am sure it will be obsolete within five years.

The slang name SAP was selected to remind the user that this program, like all computer programs, lacks intelligence. It is the responsibility of the engineer to idealize the structure correctly and assume responsibility for the results.

Berkeley, California
September 1970

E. L. Wilson

ACKNOWLEDGEMENT

In addition to the author, the following people participated in the computer program development: Lindsay R. Jones programmed parts of the main program and the beam element subroutines. Peter G. Smith participated in the initial organization of the program and in the development of the two-dimensional plane element. Te-ming Hsueh incorporated Carlos A. Felippa's quadrilateral shell element into the program. H. H. Dovey developed the dynamic options and modified the three-dimensional solid element initially programmed by Kenneth T. Kavanagh. William P. Doherty programmed parts of the equation solver, beam element, plane stress element and axisymmetric solid element.

During the past several years many organizations have sponsored various phases of the development of this computer program. However, the majority of the fundamental work on the equation solver and the solid elements was sponsored by the Walla Walla District of the Corps of Engineers. Melvin L. Scott and Marvin G. Brammer of the Walla Walla District defined the scope of this phase of the investigation and served as technical monitors for the project. All work conducted at the University of California was under the direction of Professors R. W. Clough and E. L. Wilson.

TABLE OF CONTENTS

		Page
FOREWORD		i
ACKNOWLEDGEMENTS		iii
I.	INTRODUCTION	1
II.	EQUILIBRIUM EQUATIONS FOR STRUCTURAL SYSTEMS	3
	2.1 The Direct Stiffness Method	3
	2.2 Boundary Conditions	5
III.	THREE DIMENSIONAL TRUSS ELEMENT	8
IV.	THREE DIMENSIONAL BEAM ELEMENT	12
	4.1 Definition of Principal Axes	12
	4.2 Master and Slave Degrees of Freedom	14
V.	PLANE STRESS ELEMENT	17
VI.	AXISYMMETRIC SOLID ELEMENT	18
	6.1 Coordinate Systems	19
	6.2 Displacement Assumption	21
	6.3 Element Strains	21
	6.4 Stress-Strain Relationship	25
	6.5 Numerical Integration	25
	6.6 Static Condensation	26
	6.7 Stress Evaluation	27
VII.	THREE DIMENSIONAL SOLID ELEMENT	29
	7.1 Coordinate Systems	29
	7.2 Displacement Functions	31
	7.3 Element Strains	32

Page

7.4 Stress-Strain Transformation 33

7.5 Stiffness Matrix 33

7.6 Matrix Condensation and Stress Evaluation 34

VIII. PLATE AND THIN SHELL ELEMENT 35

IX. BOUNDARY ELEMENT 37

X. DYNAMIC ANALYSIS 38

10.1 Reduction of the Number of Degrees of Freedom 38

10.2 Mode Shapes and Frequencies 41

10.3 Uncoupled Dynamic Equilibrium Equations 42

XI. EARTHQUAKE SPECTRAL ANALYSIS OF THREE DIMENSIONAL STRUCTURES 45

XII. COMPUTER PROGRAM ORGANIZATION 49

12.1 Solution of Equations 50

12.2 Formation of Equilibrium Equations 50

12.3 Joint Input Data and Degrees of Freedom 51

12.4 Calculation of Element Stiffness Matrices 51

12.5 Evaluation of Element Stresses 52

APPENDIX - DESCRIPTION OF INPUT DATA 53

I. INTRODUCTION

The purpose of this computer program is to perform linear, elastic analyses of three dimensional structural systems. The structural systems to be analysed may be composed of combinations of a number of structural element types. The present version contains the following element types:

1) three dimensional truss
2) three dimensional beam
3) plane stress and plane strain
4) two dimensional axisymmetric solid
5) three dimensional solid
6) plate and shell
7) boundary

Since several of these elements have not been published it will be necessary to present their development in this report. Only an outline and the unique characteristics of the program will be given; no attempt is made to document the program completely.

Systems composed of large numbers of joints and members may be analysed. The capacity of the program depends mainly on the total number of joints in the system. There is practically no restriction on the number of elements, number of load cases, or the "bandwidth" of the equations to be solved. Note, that while the program has the capacity to analyse very large systems, there is no loss of efficiency in the solution of smaller problems as compared to several special purpose programs presently available. The program is machine independent and is coded in standard FORTRAN IV.

New elements can be added to the program with a minimum of new programming; therefore, the nature of the program may change significantly within the next few years. These new elements will automatically have a dynamic load option since this part of SAP is element independent.

II. EQUILIBRIUM EQUATIONS FOR COMPLEX STRUCTURAL SYSTEMS

2.1 The Direct Stiffness Method

The governing joint equilibrium equations for a structural system can be derived by several different approaches. All methods yield a set of linear equations of the following form:

$$\underline{K}\ \underline{u} = \underline{R} \tag{2.1}$$

These equations set the sum of the internal element forces, $\underline{K}\ \underline{u}$, expressed in terms of joint displacements, $\underline{u}$, to the generalized loads, $\underline{R}$, acting at the joints. The matrix $\underline{u}$ contains all the joint displacements (degrees of freedom) of the system. The stiffness matrix $\underline{K}$ can be formed by the direct addition of element stiffness matrices; or

$$\underline{K} = \Sigma\ \underline{K}_m \tag{2.2}$$

For a typical element $\underline{m}$ the element stiffness matrix is given by

$$\underline{K}_m = \int_{Vol} \underline{a}_m^T\ \underline{c}_m\ \underline{a}_m\ d\ V_m \tag{2.3}$$

The stress-strain relationship for the element is of the form

$$\underline{\sigma}_m = \underline{c}_m\ \underline{\varepsilon}_m + \underline{\tau}_m \tag{2.4}$$

where $\underline{\varepsilon}_m$ are the element strains produced by the displacements $\underline{u}$ and $\underline{\tau}_m$ are the initial stresses in the element before deformation.

Within each element the strains are expressed (approximately) by the following equation:

$$\underline{\varepsilon}_m = \underline{a}_m \underline{u} \tag{2.5}$$

Note that $\underline{a}_m$ appear to be a very large matrix since $\underline{u}$ contains all degrees of freedom of the system. However, within the computer program only the non-zero columns of $\underline{a}_m$ are stored and their column numbers are stored as a separate identification array. The advantage of this notation is that the "direct" addition of element stiffness matrices as implied by equation (2.2) is correct.

The generalized loads, $\underline{R}$, are given by

$$\underline{R} = \underline{P} + \underline{T} - \underline{F} \tag{2.6}$$

where $\underline{P}$ is a matrix of concentrate joint loads and $\underline{T}$ is a matrix of generalized loads due to distributed surface stresses and is given by a summation of boundary element forces, or

$$\underline{T} = \sum \underline{T}_m \tag{2.7}$$

in which

$$\underline{T}_m = \int_{\text{Area}} \underline{b}_m^T \underline{t}_m \, ds_m \tag{2.8}$$

The surface stresses are $\underline{t}_m$ and the relationship between surface displacements and joint displacements is

$$\underline{u}_m (s) = \underline{b}_m \underline{u} \tag{2.9}$$

$\underline{F}$ is a matrix of generalized loads due to the initial stresses $\underline{\tau}_m$ and is given by a summation of element forces, or

$$\underline{F} = \Sigma \underline{F}_m \tag{2.10}$$

in which

$$\underline{F}_m = \int_{Vol} \underline{d}_m^T \underline{\tau}_m \, dV_m \tag{2.11}$$

The matrix $\underline{d}_m$ is the basic displacement field approximation within the element:

$$\underline{u}_m (x, y, z) = \underline{d}_m \underline{u} \tag{2.12}$$

2.2 Boundary Conditions

Equation (2.1) represents the relationship between all joint forces and all joint displacements and can be rewritten in partitioned form as:

$$\underline{K}_{aa} \underline{u}_a + \underline{K}_{ab} \underline{u}_b = \underline{R}_a \tag{2.13}$$

$$\underline{K}_{ba} \underline{u}_a + \underline{K}_{bb} \underline{u}_b = \underline{R}_b \tag{2.14}$$

where $\underline{R}_a$ = the specified joint loads

$\underline{R}_b$ = the unknown joint reactions

$\underline{u}_a$ = the unknown joint displacements

$\underline{u}_b$ = the specified joint displacements

The normal approach to the solution to this problem is to rewrite Eq. (2.13) in the following form:

$$\underline{K}_{aa}\ \underline{u}_a = \underline{R}_a - \underline{K}_{ab}\ \underline{u}_b = \underline{R}_a^* \tag{2.15}$$

Since $\underline{R}_a^*$ can be calculated directly, Eq. (2.15) can be solved for the unknown displacement.

In this report another approach is used which has certain programming advantages. If a displacement component, $\underline{u}_b$, is zero, the stiffness coefficients $\underline{K}_{ab}$, $\underline{K}_{ba}$ and K_{bb} are not added to the total stiffness matrix and that particular degree of freedom is disregarded in the equilibrium equations. If, however, a non-zero displacement is to be specified, $u_b = x$, Eq. (2.14) is modified by the addition of an equation of the following form:

$$k\ u_b = kx \tag{2.16}$$

where k is an arbitrary number. The resulting equation is:

$$\underline{K}_{ba}\ \underline{u}_a + (K_{bb} + k)\ u_b = R_b + kx \tag{2.17}$$

If k is selected to be several orders of magnitude greater than the stiffness coefficient K_{bb} the solution of this equation will be $u_b = x$. This may also be interpreted physically as adding a spring of large stiffness k to the structure; a large load kx is then applied; therefore, the relatively flexible structure will move along with the spring in order to produce the displacement x.

This technique of adding a stiff spring to the structure may also be used to specify skew boundary conditions. For zero specified displacement the resulting force in the spring is the support reaction.

III. THREE DIMENSIONAL TRUSS ELEMENT

The three-dimensional truss element will be explained in detail in order to illustrate the calculation of the stiffness matrix for a typical element.

A typical truss element connected to joints i and j is shown below. All dimensions are assumed positive; however, the development is correct for elements of different orientation.

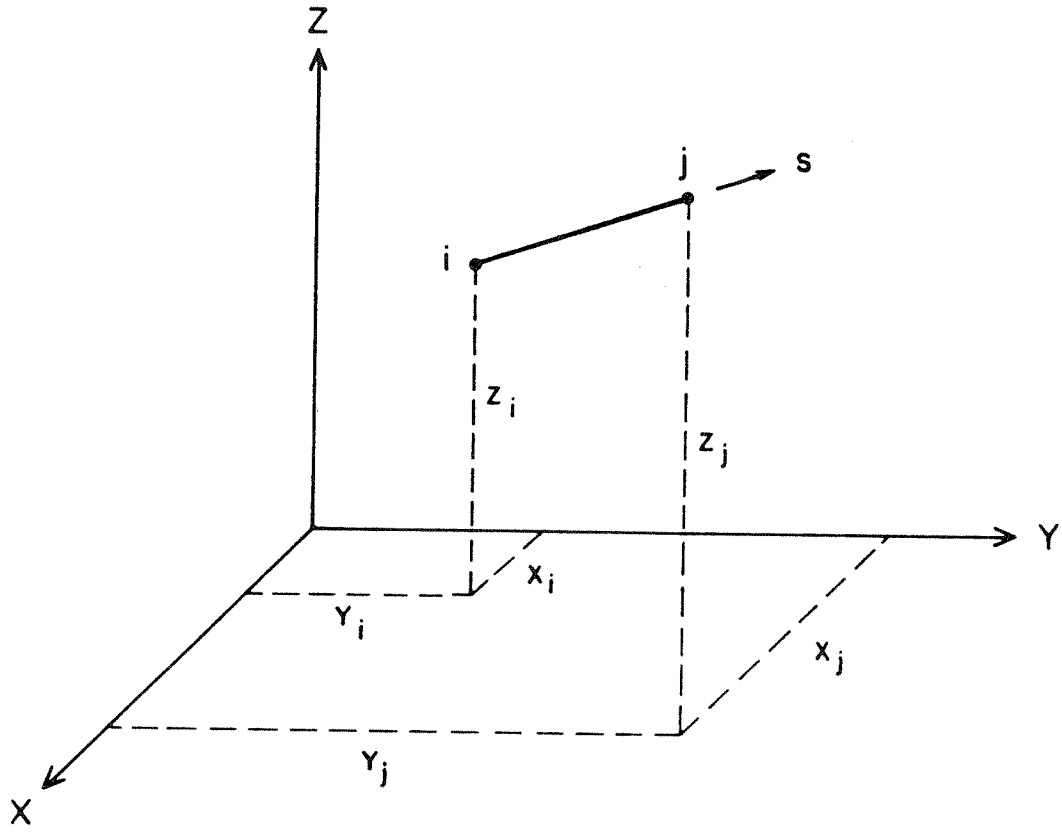


FIGURE 1 TYPICAL TRUSS ELEMENT

The length of the element is given by

$$L = \sqrt{L_x^2 + L_y^2 + L_z^2} \tag{3.1}$$

where

$$\begin{aligned} L_x &= x_j - x_i \\ L_y &= y_j - y_i \\ L_z &= z_j - z_i \end{aligned} \tag{3.2}$$

The axial displacement in the s-direction is assumed to be linear (constant strain).

$$u_s = u_{si} + \frac{s}{L}(u_{sj} - u_{si}) \tag{3.3}$$

where s equals zero at joint i. Therefore, the axial strain is

$$\epsilon_s = \frac{\partial u_s}{\partial s} = \frac{1}{L}(u_{sj} - u_{si}) \tag{3.4}$$

The axial displacement u_s is given in terms of the global displacements u_x, u_y, and u_z by

$$u_s = \frac{L_x}{L} u_x + \frac{L_y}{L} u_y + \frac{L_z}{L} u_z \tag{3.5}$$

The evaluation of equation (3.5) at joints i and j and the substitution into equation (3.4) yields the following expression for element strain:

$$\varepsilon_s = \frac{1}{L^2} \begin{bmatrix} - L_x & - L_y & - L_z & L_x & L_y & L_z \end{bmatrix} \begin{bmatrix} u_{xi} \\ u_{yi} \\ u_{zi} \\ u_{xj} \\ u_{yj} \\ u_{zy} \end{bmatrix} \tag{3.6}$$

Therefore, the strain-displacement matrix for the truss element is a 1 x 6 matrix and is given by

$$\underline{a} = \frac{1}{L^2} \begin{bmatrix} - L_x & - L_y & - L_z & L_x & L_y & L_z \end{bmatrix} \tag{3.7}$$

The axial stress is expressed in terms of axial strain by

$$\sigma_s = E \varepsilon_s \tag{3.8}$$

Therefore, the stress-strain relationship is a 1 x 1 matrix, or

$$\underline{c} = [E] \tag{3.9}$$

where E is the modulus of elasticity of the truss material.

From equation (2.3) the element stiffness can be calculated directly. Since the volume of the element is equal to the cross-sectional area of the element, A, times the length of the element, L, the element stiffness is of the form:

$$K = \frac{AE}{L^3} \begin{bmatrix} -L_x \\ -L_y \\ -L_x \\ L_x \\ L_y \\ L_z \end{bmatrix} \begin{bmatrix} -L_x & -L_y & -L_z & L_x & L_y & L_z \end{bmatrix} \tag{3.10}$$

From equations (3.8) and (3.6) the axial stress in terms of global displacements is

$$\sigma_s = \frac{E}{L^2} \begin{bmatrix} -L_x & -L_y & -L_z & L_x & L_y & L_z \end{bmatrix} \begin{bmatrix} u_{xi} \\ u_{yi} \\ u_{zi} \\ u_{xj} \\ u_{yj} \\ u_{zj} \end{bmatrix} \tag{3.11}$$

Within the computer program the stress-displacement relationship is always calculated at the same time as the element stiffness is evaluated; it is then placed on tape storage and is used later in the determination of element stresses after the joint displacements are determined.

IV. THREE DIMENSIONAL BEAM ELEMENT

The beam element included in this program considers torsion, bending about two axes, axial and shearing deformations. The element is prismatic and the development of its stiffness properties is standard and is given in many modern texts on structural analysis. Only the unique characteristics will be discussed in this section.

4.1 Definition of Principal Axes

The geometric location of a typical element is defined by joint numbers i and j. The place which locates the principal axis of the beam is defined by a third joint number k as shown in Figure 2. The relationship between the local coordinate system, s_1, s_2 and s_3 is most conveniently developed by the use of vector notation. The unit vectors in the $\hat{s}_1$ and $\hat{g}$ directions are given by

$$\hat{s}_1 = S_{1x}\,\hat{x} + S_{1y}\,\hat{y} + S_{1z}\,\hat{z}$$

$$\hat{g} = G_x\,\hat{x} + G_y\,\hat{y} + G_z\,\hat{z}$$

where $\hat{x}$, $\hat{y}$, and $\hat{z}$ are unit vectors in the x, y and z directions, respectively. The direction cosines are

$$S_{1x} = \frac{L_x}{L}\ ;\ S_{1y} = \frac{L_y}{L}\ ;\ S_{1z} = \frac{L_z}{L}$$

$$G_x = \frac{G_x}{G}\ ;\ G_y = \frac{G_y}{G}\ ;\ G_z = \frac{G_z}{G}$$

in which

$$L_x = X_j - X_i \; ; \; L_y = y_j - y_i \; ; \; L_z = z_j - z_i$$

$$G_x = X_k - X_i \; ; \; G_y = y_k - y_i \; ; \; G_z = z_k - z_i$$

$$L = \sqrt{L_x^2 + L_y^2 + L_z^2} \quad \text{and} \quad G = \sqrt{G_x^2 + G_y^2 + G_z^2}$$

The unit vector in the $\hat{s}_3$ direction is given by the vector product of $\hat{s}_1$ and $\hat{g}$ divided by the length of the vector in that direction

$$\hat{s}_3 = \frac{\hat{s}_1 \times \hat{g}}{|\hat{s}_3|} = S_{3x}\,\hat{x} + S_{3y}\,\hat{y} + S_{3z}\,\hat{z}$$

where the vector product is by definition the evaluation of

$$\underline{a} \times \underline{b} = \begin{bmatrix} \hat{x} & \hat{y} & \hat{z} \\ a_x & a_y & a_z \\ b_x & b_y & b_z \end{bmatrix}$$

The unit vector in the $\hat{s}_2$ direction is given by the vector product

$$\hat{s}_2 = \hat{s}_3 \times \hat{s}_1 = S_{2x}\,\hat{x} + S_{2y}\,\hat{y} + S_{2z}\,\hat{z}$$

The three unit vectors may be summarized by the following matrix equations:

$$\begin{bmatrix} \hat{s}_1 \\ \hat{s}_2 \\ \hat{s}_3 \end{bmatrix} \begin{bmatrix} S_{1x} & S_{1y} & S_{1z} \\ S_{2x} & S_{2y} & S_{2z} \\ S_{3x} & S_{3y} & S_{3z} \end{bmatrix} \begin{bmatrix} \hat{x} \\ \hat{y} \\ \hat{z} \end{bmatrix}$$

Within the computer program local displacements, forces and moments are transformed to the global system by this 3 x 3 matrix.

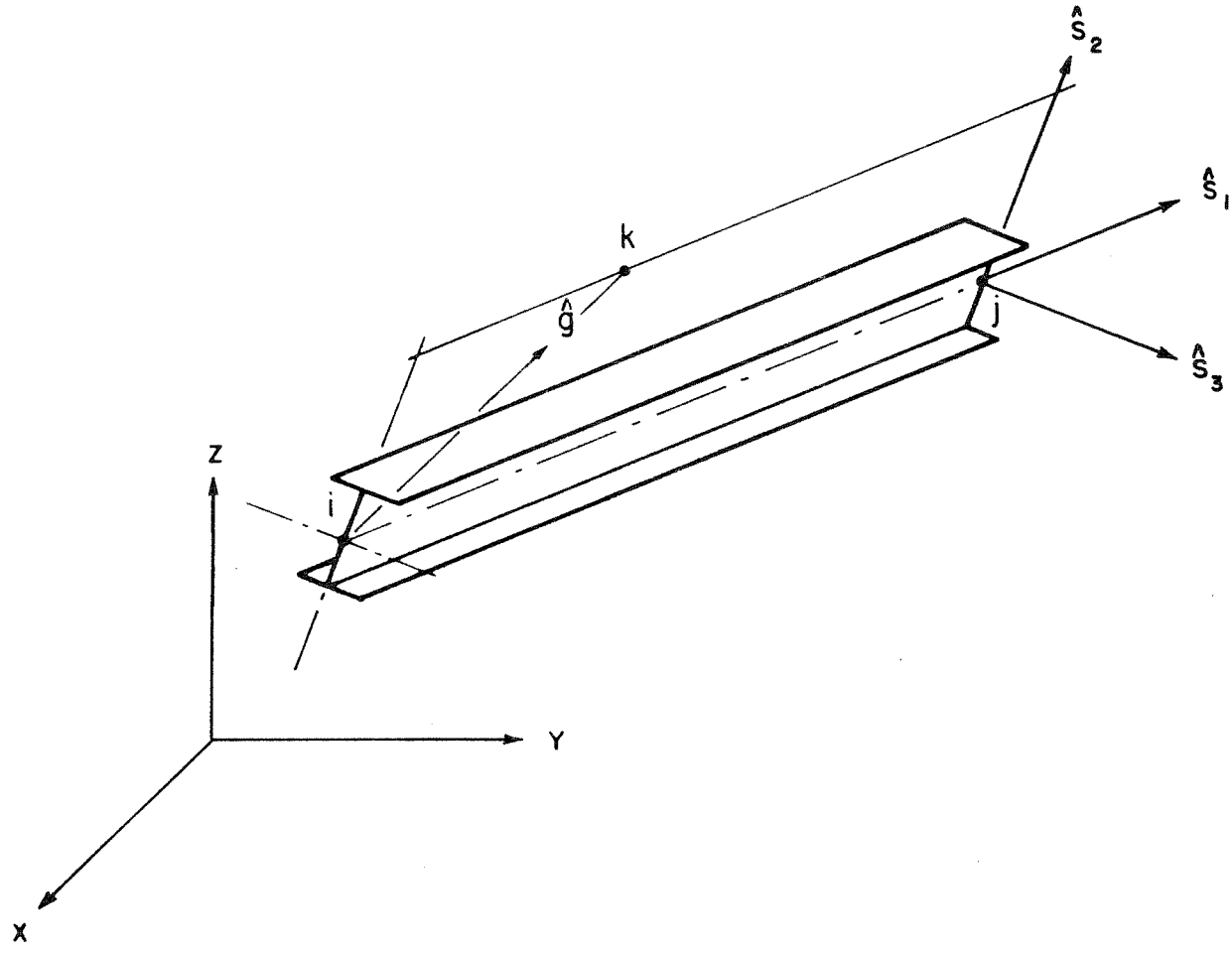


FIGURE 2 THREE DIMENSIONAL BEAM ELEMENT

4.2 Master and Slave Degrees of Freedom

The three dimensional beam elements can be connected to slave degrees of freedom. Slave degrees of freedom are eliminated from the formulation and replaced by the degrees of freedom of the master joint. This technique reduces the total number of joint equilibrium equations, in the system and greatly reduces the possibility of numerical sensitivities in many types of structures.

The geometry of the master and slave joints is shown in Figure 3. Beam elements may be connected directly to either a master or a slave joint. Any one of the six degrees of freedom of the slave joint may be eliminated. If all six degrees of freedom are made into slaves then the physical effect is that the two joints are connected with a rigid link.

If the x displacement of the slave joint is defined as a slave of the master joint the displacements will be transformed as follows:

$$u_{xs} = u_{xm} + (z_s - z_m)\,\theta_{ym} - (y_s - y_m)\,\theta_{zm}$$

For the y-displacement

$$u_{ys} = u_{ym} - (z_s - z_m)\,\theta_{xm} + (x_s - x_m)\,\theta_{zm}$$

For the z-displacement

$$u_{zs} = u_{zm} + (y_s - y_m)\,\theta_{xm} - (x_s - x_m)\,\theta_{ym}$$

and for the rotations

$$\theta_{xs} = \theta_{xm}$$

$$\theta_{ys} \quad \theta_{sm}$$

$$\theta_{zs} \quad \theta_{zm}$$

For the beam elements those transformations automatically take place for all elements connected to slave degrees of freedom. The computer program allows a joint to be a slave to more than one master; however, it is difficult to create a physical system with this property.

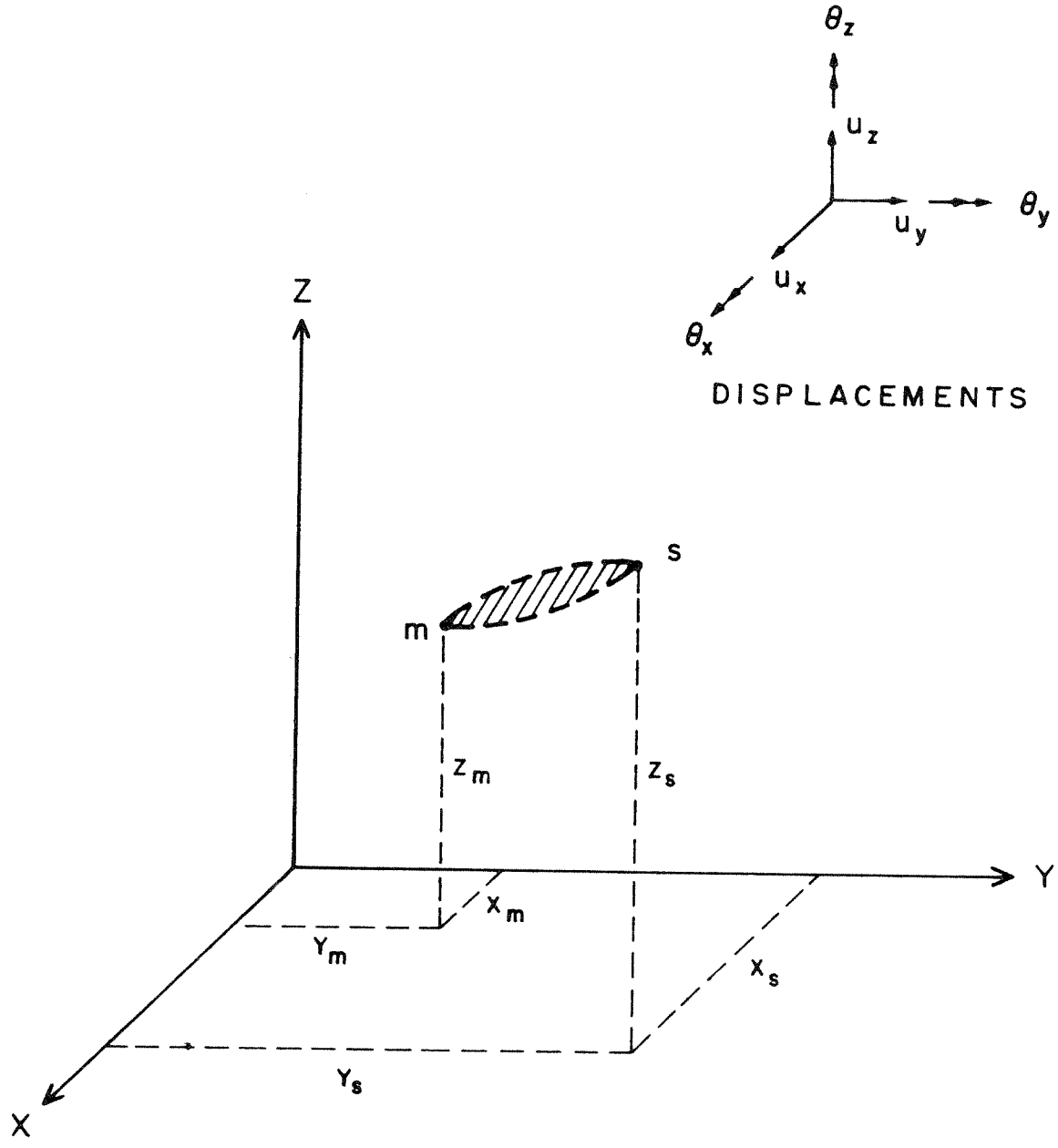


FIGURE 3 MASTER AND SLAVE JOINTS

V. PLANE STRESS ELEMENT

The plane stress element currently included in the program is a general quadrilateral with a one-point integration of the strain energy. Recent developments indicate that an improved quadrilateral element for plane stress can be developed; therefore, the existing plane stress element will not be described. It is hoped that in the near future it will be replaced by a new element which would be similar to the axisymmetric element described in the next section of this report.

The geometry of the quadrilateral element is defined by four points. The element must lie on a plane; therefore, only joints I, J, and K are used to define the plane. Joint L is assumed to be in the plane. The basic geometric transformations are of the same form as used for the three dimensional beam element.

For many plane structures in space there may be zero stiffness normal to the plane and the stiffness matrix may be singular in that direction. Therefore, this direction must be restrained by a boundary condition or a normal "boundary element".

VI. AXISYMMETRIC SOLID QUADRILATERAL ELEMENT

Many different elements for the two dimensional analysis of plane and axisymmetric solids have been developed during the past several years. Most of these elements have very poor bending characteristics. Six point triangles and eight point quadrilaterals are types of higher order elements which have excellent bending properties; however, elements of this nature require a large increase in computational effort in both the element stiffness formation and in the solution of the resulting set of equations.

The element presented in this section is a four point quadrilateral which has been modified in order to improve its bending behavior. This modification involves the addition of extra degrees of freedom within the elements which causes a violation of interelement displacement compatibility. The resulting element has good bending characteristics and for many problems is comparable in accuracy with the higher order elements.

The general form of the element stiffness matrix is

$$\underline{K} = \int_{vol} \underline{a}^T \underline{c} \underline{a} \, dv \tag{6.1}$$

where $\underline{c}$ is the relationship between stresses and strains and $\underline{a}$ is the strain-displacement matrix.

$$\underline{\varepsilon} = \underline{a} \underline{u} \tag{6.2}$$

For the axisymmetric element discussed here there are four components of strain and 12 element degrees of freedom. The displacement components are the eight nodal point displacements at the four corners of the elements and four internal degrees of freedom which will be eliminated before the element stiffness is added to the total stiffness matrix.

Since this element involves the application of new concepts and the uses of the "natural" coordinate system the details of its development will be given.

6.1 Coordinate Systems

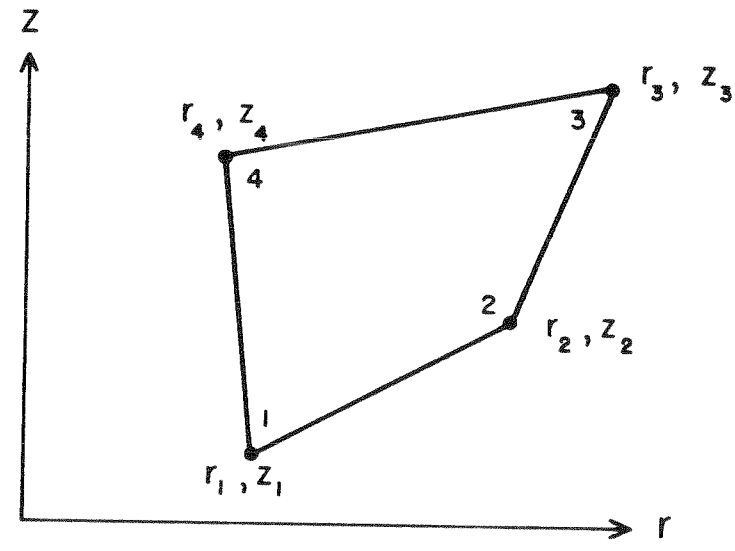
The geometry of the element is defined in global $r - z$ system by the coordinate of the four nodes as indicated in Figure 4a. Positions within the element may also be defined in the natural, or local, $s - t$ system as shown in Figure 4b. The relationship between these two systems is given by definition as

$$r(s, t) = \sum_{i=1}^{4} h_i r_i \qquad z(s, t) = \sum_{i=1}^{4} h_i z_i \qquad (6.3)$$

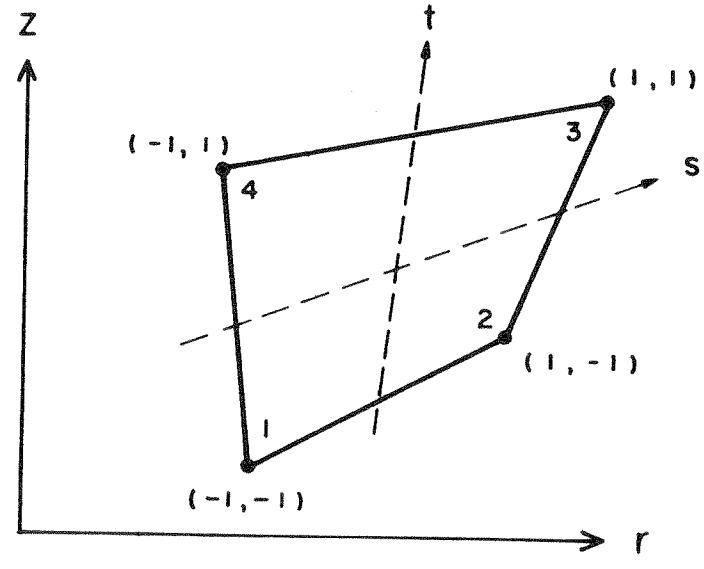
where

$$h_1 = (1 - s)(1 - t)/4 \qquad h_3 = (1 + s)(1 + t)/4$$

$$h_2 = (1 + s)(1 - t)/4 \qquad h_4 = (1 - s)(1 + t)/4$$



a. GLOBAL SYSTEM



b. LOCAL SYSTEM

FIGURE 4 COORDINATE SYSTEMS

6.2 Displacement Assumptions

The r and z displacement field approximation within the element is assumed to be of the following form

$$\begin{aligned} u_r(s, t) &= \sum_{i=1}^{4} h_i u_{ri} + h_5 \alpha_{r1} + h_6 \alpha_{r2} \\ u_z(s, t) &= \sum_{i=1}^{4} h_i u_{zi} + h_5 \alpha_{z1} + h_6 \alpha_{z2} \end{aligned} \tag{6.4}$$

where

$$h_5 = (1 - s^2) \quad \text{and} \quad h_6 = (1 - t^2)$$

The interpolation functions h_5 and h_6 are zero at the four nodes; however, they produce a parabolic incompatibility along the sides of the quadrilateral. (The amplitude factors α_{r1}, α_{r2}, α_{z1} and α_{z2} will be selected to better satisfy microscopic equilibrium by the application of the principle of minimum potential energy after the element stiffness is formed.)

6.3 Element Strains

For an axisymmetric solid subjected to axisymmetric loads the strain displacement equations are

$$\begin{aligned} \varepsilon_r &= \frac{\partial u_r}{\partial r} \\ \varepsilon_z &= \frac{\partial u_z}{\partial z} \\ \varepsilon_\theta &= \frac{u_r}{r} \\ \gamma_{rz} &= \frac{\partial u_r}{\partial z} + \frac{\partial u_z}{\partial r} \end{aligned} \tag{6.5}$$

Since the global displacements u_r and u_z are in terms of s and t, the appropriate derivatives cannot be calculated directly. Therefore, the application of the chain rule for differentiation is required. The chain rule may be written in matrix form as

$$\begin{bmatrix} \frac{\partial u}{\partial s} \\ \\ \frac{\partial u}{\partial t} \end{bmatrix} = \begin{bmatrix} \frac{\partial r}{\partial s} & \frac{\partial z}{\partial s} \\ \\ \frac{\partial r}{\partial t} & \frac{\partial z}{\partial t} \end{bmatrix} \begin{bmatrix} \frac{\partial u}{\partial r} \\ \\ \frac{\partial u}{\partial z} \end{bmatrix} \tag{6.6}$$

or inverted

$$\begin{bmatrix} \frac{\partial u}{\partial r} \\ \\ \frac{\partial u}{\partial z} \end{bmatrix} = \underline{J} \begin{bmatrix} \frac{\partial u}{\partial s} \\ \\ \frac{\partial u}{\partial t} \end{bmatrix} \tag{6.7}$$

where

$$\underline{J} = \frac{1}{J^*} \begin{bmatrix} z_{,t} & -z_{,s} \\ \\ -r_{,t} & r_{,s} \end{bmatrix}$$

in which the Jacobian Determinant of the transformation is

$$J^* = r_{,s}\, z_{,t} - r_{,t}\, z_{,s} \tag{6.8}$$

For both r and z displacements their derivatives with respect to the natural system are

$$\frac{\partial u}{\partial s} = \sum_{i=1}^{6} \frac{\partial h_i}{\partial s} u_i$$

$$\frac{\partial u}{\partial t} = \sum_{i=1}^{6} \frac{\partial h_i}{\partial t} u_i \qquad (6.9)$$

or in matrix form

$$\begin{bmatrix} \frac{\partial u}{\partial s} \\ \frac{\partial u}{\partial t} \end{bmatrix} = \underline{P} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ \alpha_1 \\ \alpha_2 \end{bmatrix} \qquad (6.10)$$

If the above equation is substituted into the inverted chain rule the global derivatives are expressed in terms of the discrete displacements.

$$\begin{bmatrix} \frac{\partial u}{\partial r} \\ \frac{\partial u}{\partial z} \end{bmatrix} = \underline{J}\,\underline{P}\,\underline{u} = \begin{bmatrix} T_{11} & T_{12} & T_{13} & T_{14} & T_{15} & T_{16} \\ T_{21} & T_{22} & T_{23} & T_{24} & T_{25} & T_{26} \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ \alpha_1 \\ \alpha_2 \end{bmatrix} \qquad (6.11)$$

These equations and the strain-displacement equations are now used to express the elements strains in terms of the 12 discrete displacements

$$\begin{bmatrix} \varepsilon_r \\ \varepsilon_z \\ \varepsilon_0 \\ \gamma_{rv} \end{bmatrix} = \begin{bmatrix} T_{11} & T_{12} & T_{13} & T_{14} & 0 & 0 & 0 & 0 & T_{15} & T_{16} & 0 & 0 \\ 0 & 0 & 0 & 0 & T_{21} & T_{22} & T_{23} & T_{24} & 0 & 0 & T_{25} & T_{26} \\ \frac{h_1}{r} & \frac{h_2}{r} & \frac{h_3}{r} & \frac{h_4}{r} & 0 & 0 & 0 & 0 & \frac{h_5}{r} & \frac{h_6}{r} & 0 & 0 \\ T_{21} & T_{22} & T_{23} & T_{24} & T_{11} & T_{12} & T_{13} & T_{14} & T_{25} & T_{26} & T_{15} & T_{16} \end{bmatrix} \begin{bmatrix} u_{r1} \\ u_{r2} \\ u_{r3} \\ u_{r4} \\ u_{z1} \\ u_{z2} \\ u_{z3} \\ u_{z4} \\ \alpha_{r1} \\ \alpha_{r2} \\ \alpha_{z1} \\ \alpha_{z2} \end{bmatrix} \quad (6.12)$$

or

$$\underline{\varepsilon} = \underline{a}\,(s, t)\,\underline{u}$$

in which

$$r = r\,(s, t) = \sum_{i=1}^{4} h_i\, r_i$$

This strain-displacement matrix $\underline{a}$ is expressed completely in terms of the natural coordinates s and t.

6.4 Stress-Strain Relationship

The stress-strain relationship can be of a general anisotropic form. In case of isotropic materials $\underline{\sigma} = \underline{c}\,\underline{\varepsilon}$ or

$$\begin{bmatrix} \sigma_r \\ \sigma_z \\ \sigma_\theta \\ \tau_{rz} \end{bmatrix} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & \nu & 0 \\ \nu & 1-\nu & \nu & 0 \\ \nu & \nu & 1-\nu & 0 \\ 0 & 0 & 0 & \frac{1-2\nu}{2} \end{bmatrix} \begin{bmatrix} \varepsilon_r \\ \varepsilon_t \\ \varepsilon_\theta \\ \gamma_{rz} \end{bmatrix} \tag{6.13}$$

6.5 Numerical Integration

For an axisymmetric solid the volume integral for the element stiffness can be converted to an area integral since $dV = r\ d\theta dA$. For a one radian segment of the solid

$$\underline{K} = \int_{Vol} \underline{a}^T\,\underline{c}\,\underline{a}\ dV = \int_{Area} r\,\underline{a}^T\,\underline{c}\,\underline{a}\ dA \tag{6.14}$$

In the natural system $dA = J^*\ ds\ dt$; therefore

$$\underline{K} = \int_{-1}^{1}\int_{-1}^{1} r\,\underline{a}^T\,\underline{c}\,\underline{a}\,J^*\ ds\ dt \tag{6.15}$$

This is of the form

$$\underline{K} = \int_{-1}^{1}\int_{-1}^{1} f(s, t)\ ds\ dt \tag{6.16}$$

A four point integration formula for a general quadrilateral is

$$\underline{K} = \sum_{j=1}^{4} \alpha_j\ f(s_j, t_j) \tag{6.17}$$

where the weighting factors and integration points are

j	α_j	s_j	t_j
1	1	$1/\sqrt{3}$	$1/\sqrt{3}$
2	1	$-1/\sqrt{3}$	$1/\sqrt{3}$
3	1	$-1/\sqrt{3}$	$-1/\sqrt{3}$
4	1	$1/\sqrt{3}$	$-1/\sqrt{3}$

Within the computer program the matrix product $r\ \underline{a}^T\ \underline{c}\ \underline{a}\ J^*$ is formed for each of the integration points. Then they are appropriately combined to form the complete 12 x 12 element stiffness matrix.

6.6 Static Condensation

The displacement amplitudes α_{r1}, α_{r2}, α_{z1} and α_{z2} can be eliminated at the element level since displacement compatibility is only inforced at the nodes for this element. Another interpretation of these displacement amplitudes is that they are Lagrangian Multipliers and are selected to improve equilibrium within the element. Also, if they are assumed to produce additional stress patterns this element may be thought of as a mixed model.

The forces and displacements associated with the element are

$$\underline{F} = \underline{K}\,\underline{u} \quad \text{or} \quad \begin{bmatrix} \underline{F}_{1\text{-}8} \\ 0 \end{bmatrix} = \begin{bmatrix} K_{aa} & K_{ab} \\ K_{ba} & K_{bb} \end{bmatrix} \begin{bmatrix} \underline{u}_{1\text{-}8} \\ \underline{\alpha} \end{bmatrix} \tag{6.18}$$

Since there are no external forces associated with the α's they may be expressed in terms of the corner displacements or

$$\underline{\alpha} = - \underline{K}_{bb}^{-1} \underline{K}_{ba} \underline{u}_{1-8} \tag{6.19}$$

Therefore, the corner forces in terms of corner deformation will be

$$\underline{F}_{1-8} = \left[\underline{K}_{aa} - \underline{K}_{ab} \underline{K}_{bb}^{-1} \underline{K}_{ba}\right] \underline{u}_{1-8} \tag{6.20}$$

The resulting 8 x 8 stiffness matrix has excellent bending properties and has been found to be an effective element for many axisymmetric and plane problems.

6.7 Stress Evaluation

After the displacements of the system are evaluated the stresses within the element can be calculated directly from the stress-strain relationship, $\underline{c}$, and the strain-displacement matrix, $\underline{a}$. In order to eliminate the need to recalculate the generalized displacements $\underline{\alpha}$, it is desirable to develop a stress-displacement transformation in terms of the eight nodal displacements. The stresses at an arbitrary point, s and t, are given by

$$\underline{\sigma} = \underline{c} \; \underline{a} \; \underline{u} \tag{6.21}$$

or if the a matrix is partitioned

$$\underline{\sigma} = \underline{c} \; \underline{a}_a \; \underline{u}_{1-8} + \underline{c} \; \underline{a}_b \; \underline{\alpha} \tag{6.22}$$

Since $\underline{\alpha} = - \underline{K}_{bb}^{-1} \underline{K}_{ba} \underline{u}_{1-8}$ the stresses may be written in terms of corner displacements

$$\underline{\sigma} = \underline{c} \; \underline{a}^{*} \; \underline{u}_{1-8} \tag{6.23}$$

where the effective strain-displacement matrix is

$$\underline{a}^* = \underline{a}_a - \underline{a}_b \; \underline{K}_{bb}^{-1} \; \underline{K}_{ba} \tag{6.24}$$

It is convenient to calculate this matrix within the computer program at the same time as the element stiffness is evaluated; then, it can be stored on tape for use after the displacements are determined.

VII. THREE-DIMENSIONAL SOLID ELEMENT

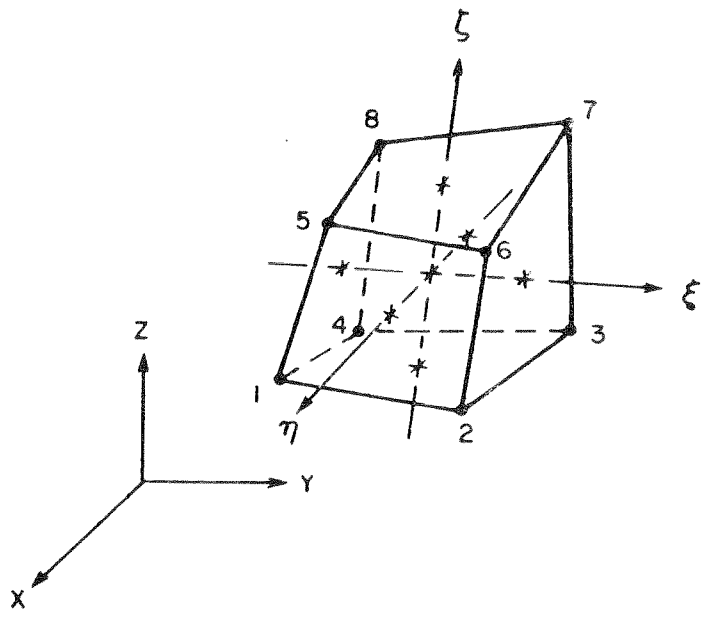
The three-dimensional solid element presented in this section is a eight node hexahedron. It is based on the element developed by Irons and Zienkiewicz; however, it has been modified by the addition of incompatible deformation modes which greatly improve the elements bending behavior.

7.1 Coordinate Systems

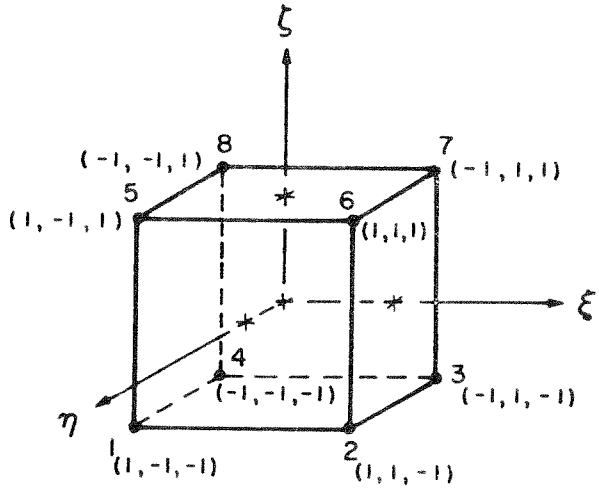
A typical hexahedron is shown in Figure 5. The coordinate system (x, y, z) is refered to as the global system; the x, y, z axes form an orthogonal, right handed system, and are used to define the locations of the nodes.

A system of local coordinates (η, ξ, ζ) for the element (sometime called the natural coordinates) is chosen such that η, ξ and ζ vary from -1 to 1; $(0, 0, 0)$ is located at the centroid of the element. In general, the relationship between the local and the global Cartesian coordinates is provided by a set of linear interpolation functions:

$$\begin{aligned} x &= \sum_{i=1}^{8} h_i \, x_i \\ y &= \sum_{i=1}^{8} h_i \, y_i \\ z &= \sum_{i=1}^{8} h_i \, z_i \end{aligned} \tag{7.1}$$



a. GLOBAL COORDINATES



b. LOCAL COORDINATES

FIGURE 5 THREE DIMENSIONAL COORDINATE SYSTEMS

where

$$h_1 = \frac{1}{8}(1 + \eta)(1 - \xi)(1 - \zeta)$$

$$h_2 = \frac{1}{8}(1 + \eta)(1 + \xi)(1 - \zeta)$$

$$h_3 = \frac{1}{8}(1 - \eta)(1 + \xi)(1 - \zeta)$$

$$h_4 = \frac{1}{8}(1 - \eta)(1 - \xi)(1 - \zeta)$$

$$h_5 = \frac{1}{8}(1 + \eta)(1 - \xi)(1 + \zeta)$$

$$h_6 = \frac{1}{8}(1 + \eta)(1 + \xi)(1 + \zeta)$$

$$h_7 = \frac{1}{8}(1 - \eta)(1 + \xi)(1 + \zeta)$$

$$h_8 = \frac{1}{8}(1 - \eta)(1 - \xi)(1 + \zeta)$$

7.2 Displacement Functions

The displacements within the element are assumed to be of the following form:

$$\begin{aligned} u_x &= \sum_{i=1}^{8} h_i u_{xi} + h_9 \alpha_{x1} + h_{10} \alpha_{x2} + h_{11} \alpha_{x3} \\ u_y &= \sum_{i=1}^{8} h_i u_{yi} + h_9 \alpha_{y1} + h_{10} \alpha_{y2} + h_{11} \alpha_{y3} \\ u_z &= \sum_{i=1}^{8} h_i u_{yi} + h_9 \alpha_{z1} + h_{10} \alpha_{z2} + h_{11} \alpha_{z3} \end{aligned} \qquad (7.2)$$

where

$$h_9 = (1 - \eta^2)$$

$$h_{10} = (1 - \xi^2)$$

$$h_{11} = (1 - \zeta^2)$$

7.3 Element Strains

For a three dimensional solid the six components of strain are given in terms of the displacements by:

$$\varepsilon = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{Bmatrix} = \begin{bmatrix} \dfrac{\partial u_x}{\partial x} \\ \dfrac{\partial u_y}{\partial y} \\ \dfrac{\partial u_z}{\partial z} \\ \dfrac{\partial u_x}{\partial y} + \dfrac{\partial u_y}{\partial x} \\ \dfrac{\partial u_x}{\partial z} + \dfrac{\partial u_z}{\partial x} \\ \dfrac{\partial u_y}{\partial z} + \dfrac{\partial u_z}{\partial y} \end{bmatrix} \tag{7.3}$$

Since u_x, u_y and u_z are not expressed directly in terms of the coordinates x, y and z the chain rule must be employed in order to evaluate the derivatives of the displacements. The chain rule written in matrix form is

$$\begin{Bmatrix} \dfrac{\partial}{\partial \xi} \\ \dfrac{\partial}{\partial \eta} \\ \dfrac{\partial}{\partial \zeta} \end{Bmatrix} = \begin{bmatrix} \dfrac{\partial x}{\partial \xi} & \dfrac{\partial y}{\partial \xi} & \dfrac{\partial z}{\partial \xi} \\ \dfrac{\partial x}{\partial \eta} & \dfrac{\partial y}{\partial \eta} & \dfrac{\partial z}{\partial \eta} \\ \dfrac{\partial x}{\partial \zeta} & \dfrac{\partial y}{\partial \zeta} & \dfrac{\partial z}{\partial \zeta} \end{bmatrix} \begin{Bmatrix} \dfrac{\partial}{\partial x} \\ \dfrac{\partial}{\partial y} \\ \dfrac{\partial}{\partial z} \end{Bmatrix} = [J] \begin{Bmatrix} \dfrac{\partial}{\partial x} \\ \dfrac{\partial}{\partial y} \\ \dfrac{\partial}{\partial z} \end{Bmatrix} \tag{7.4}$$

where the square matrix of derivatives is called the Jacobian matrix [J].

If this matrix is inverted it is possible to express the global derivatives of u_x, u_y and u_z in terms of the derivatives of the displacements with respect to the natural coordinates ξ, η and ζ. Therefore, it is possible to express the element strains in terms of the node displacements.

$$\underline{\varepsilon}\ (\xi, \eta, \zeta) = \underline{a}\ (\xi, \eta, \zeta)\ \underline{u} \tag{7.5}$$

A closed form expression for the matrix $\underline{a}$ is not required since within the computer program all operations are carried out in a numerical form for a specific values of the natural coordinates.

7.4 Stress-Strain Transformation

The stress-strain relationship for an elastic solid may be written as for an isotropic material as

$$\underline{c} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & \nu & & & \\ \nu & 1-\nu & \nu & & & \\ \nu & \nu & 1-\nu & & & \\ & & & \frac{1-2\nu}{2} & & \\ & & & & \frac{1-2\nu}{2} & \\ & & & & & \frac{1-2\nu}{2} \end{bmatrix} \tag{7.6}$$

7.5 Stiffness Matrix

When the strain matrix [a] and the stress matrix [c] have been evaluate the element stiffness matrix may be formulated from the

principle of minimum potential energy, with the result that the stiffness matrix be expressed in the form $K = \int_V \underline{a}^T \underline{c}\, \underline{a}\, dV$. However, because the strain matrix is expressed in natural coordinates, it is necessary to carry out the integration in natural coordinates, using the relationship

$$dV = dx\, dy\, dz = |J|\, d\xi\, d\eta\, d\zeta \tag{7.7}$$

where $|J|$ the determinant of the Jacobian matrix

$$K = \int_{-1}^{1} \int_{-1}^{1} \int_{-1}^{1} |J|\, a^T\, c\, a\, d\xi\, d\eta\, d\zeta \tag{7.8}$$

This is integrated numerically by Gaussian quadrature.

7.6 Matrix Condensation and Stress Evaluation

The elimination of the generalized displacements, $\underline{\alpha}$, and the calculation of stresses is identical to the proceeding described in the section on the Axisymmetric Solid Element.

VIII. PLATE AND THIN SHELL ELEMENT

The thin shell element included in this report is a quadrilateral of arbitrary geometry formed from four compatible triangles. The bending properties of this element are completely described in the following paper:

> "A Refined Quadrilateral Element For Analysis of Plate Bending," Proc. (Second) Conf. on Matrix Methods in Structural Mechanics, Wright-Patterson AFB, Ohio, 1968.

The element employs a partially restrained linear strain triangle to represent the membrane behavior. As shown in Figure 6, the central node is located at the average of the coordinates of the four corner nodes. The element has 17 interior degrees of freedom which are eliminated at the element level prior to assembling; therefore, the resulting quadrilateral element has 20 degrees of freedom, five per node, in the local element coordinate system.

For flat plates the stiffness associated with the rotation normal to the shell surface is not defined; therefore, the appropriate boundary condition must be enforced. For curved shells, the normal rotation can be included as an extra degree of freedom; or, it can be restrained by the addition of a "Boundary Element" which would add normal rotational stiffness to the node.

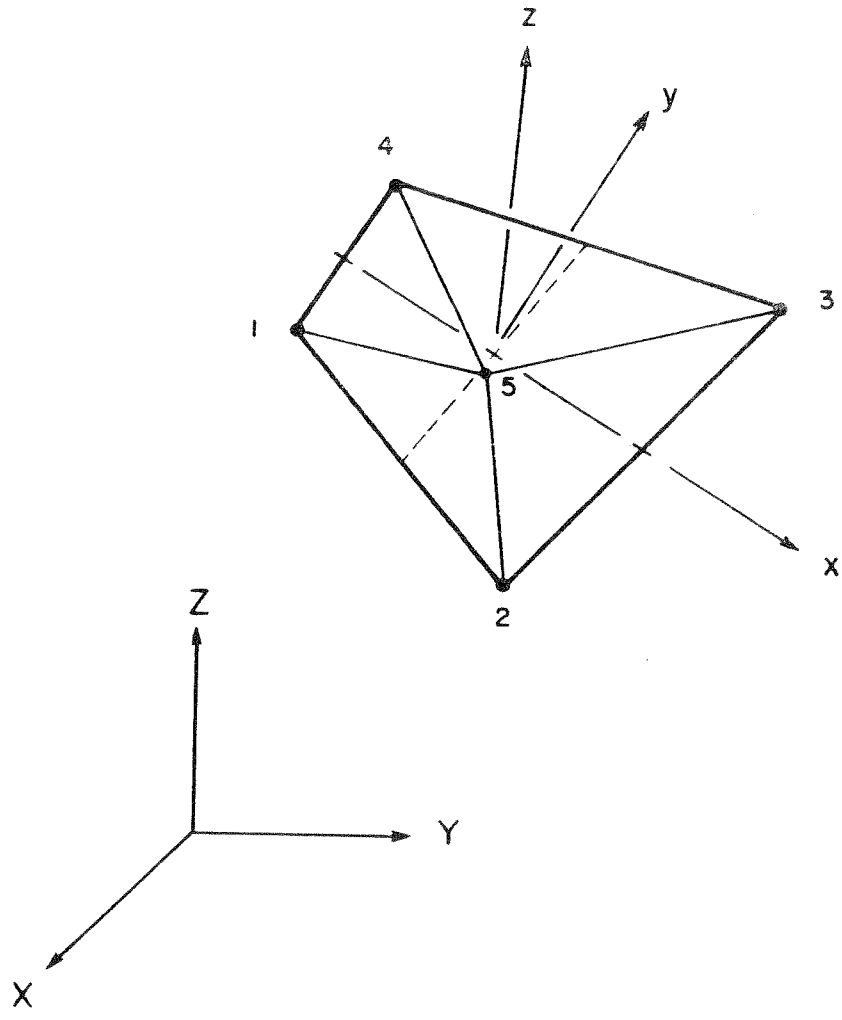


FIGURE 6 THIN SHELL ELEMENT

IX. BOUNDARY ELEMENT

The boundary element can be used for the following:

1. In the idealization of an external elastic support at a joint.
2. In the idealization of an incline roller support.
3. To specify a joint displacement
4. To eliminate the numerical difficulty associated with the "sixth" degree of freedom in the analysis of shells.

The element is a one dimensional element with an axial and torsional stiffness. These element stiffness coefficients are added direct to the total stiffness matrix. If a displacement is to be specified a load must be applied in the direction of the stiffness. If the boundary element stiffness is large compared to the stiffness of the structures it is possible to apply a load to produce the desired displacement.

X. DYNAMIC ANALYSIS

The computer program presented in this report has the following dynamic options:

1. Mode Shapes and Frequencies
2. Spectrum Analysis
3. Dynamic Response Analysis.

For all three of these options an approximate method for determining the mode shapes and frequencies is employed. For most structures the technique is adequate for the evaluation of the principal dynamic behavior without the expenditure of a large amount of computational effort.

10.1 Reduction of The Number of Degrees of Freedom

The exact formulation of the dynamic response of a structure involves an infinite number of degrees of freedom. However, for most structures the mass and stiffness properties of the structure may be "lumped" at a limited number of points (or joints) within the system. The advantage of this lumped parametered idealization is that the equilibrium of the structure is given by a set of second order, ordinary differential equations. These equations represent the force equilibrium at the joints of the structural system and may be expressed by the following matrix equation:

$$\underline{M}\,\ddot{\underline{u}} + \underline{C}\,\dot{\underline{u}} + \underline{K}\,\underline{u} = \underline{F} \qquad (10.1)$$

where

$\underline{M}$ = mass matrix

$\underline{C}$ = damping matrix

$\underline{K}$ = stiffness matrix

$\underline{F}$ = applied force vector

$\ddot{\underline{u}}$, $\dot{\underline{u}}$, $\underline{u}$ = joint acceleration, velocity, displacement vectors.

There are three unknown translation displacements and three rotational displacements for each joint in a three dimensional structure. Therefore, for large structural systems Equation (10.1) may involve several hundred degrees of freedom. Even on a large digital computer the exact solution of such a large system subjected to dynamic loads is not possible. However, Equation (10.1) may be reduced in size by certain approximations which are based on a physical insight into the behavior of the structure. It is possible to select a set of force patterns which are associated with the lower frequencies of the structure. By static analysis, a solution of the following equation is then performed for these load patterns.

$$\underline{K}\ \underline{A} = \underline{P} \tag{10.2}$$

where each column in the $\underline{P}$ matrix represents a static load pattern and the corresponding column in the $\underline{A}$ matrix represents the resulting static displacement pattern. As a result of the static analysis, a transformation between all displacements $\underline{u}$ and a limited number of generalized displacements is developed. Mathematically,

$$\underline{u} = \underline{A}\ \underline{V} \tag{10.3a}$$

Also, the velocities and acceleration are given by

$$\dot{\underline{u}} = \underline{A}\,\dot{\underline{V}} \tag{10.3b}$$

and

$$\ddot{\underline{u}} = \underline{A}\,\ddot{\underline{V}} \tag{10.3c}$$

Physically, the generalized displacements $\underline{V}$ represent multiplication factors for the static load displacement patterns $\underline{A}$. Or, the true displacement of the structure is expressed in terms of a linear combination of the static displacement patterns. The substitution of Equations (10.3) into Equation (10.1) and the premultiplication by the transpose of Matrix $\underline{A}$ yields:

$$\overline{\underline{M}}\,\ddot{\underline{V}} + \overline{\underline{C}}\,\dot{\underline{V}} + \overline{\underline{R}}\,\underline{V} = \overline{\underline{F}} \tag{10.4}$$

where

$$\overline{\underline{M}} = \underline{A}^T\,\underline{M}\,\underline{A} \tag{10.5}$$

$$\overline{\underline{C}} = \underline{A}^T\,\underline{C}\,\underline{A} \tag{10.6}$$

$$\overline{\underline{K}} = \underline{A}^T\,\underline{K}\,\underline{A} \tag{10.7}$$

$$\overline{\underline{F}} = \underline{A}^T\,\underline{F} \tag{10.8}$$

The premultiplication of the equation by $\underline{A}^T$ represents a transformation of the real forces to the generalized force system. An alternate form for the generalized stiffness matrix may be obtained by the combination of Equations (10.2) and (10.7). Or,

$$\overline{\underline{K}} = \underline{A}^T\,\underline{P} \tag{10.9}$$

This reduction procedure can be physically interpreted as an approximation on the mass and damping distribution in the structure. Since the displacement patterns satisfy static equilibrium there is not an approximation on the stiffness of the structure which would impose "locked in" constraints on the system.

10.2 Mode Shapes and Frequencies

The first step in solving Equation (10.4) is to determine the mode shapes and frequencies of the structure for "undamped free vibrations." This involves the solution of the following equation for its characteristic values:

$$\underline{\bar{K}}\ \underline{\phi}_n = \omega_n^2\ \underline{\bar{M}}\ \underline{\phi}_n \tag{10.10}$$

where

ω_n = frequency of n^{th} mode

$\underline{\phi}_n$ = mode shape of n^{th} mode

It should be noted that the mode shapes satisfy the orthogonality conditions

$$\left.\begin{aligned} \underline{\phi}_n^T\ \underline{\bar{M}}\ \underline{\phi}_m &= 0 \\ \underline{\phi}_n^T\ \underline{\bar{K}}\ \underline{\phi}_m &= 0 \end{aligned}\right\} \quad m \neq n$$

In addition the mode shapes are normalized so that

$$\underline{\phi}_n^T \; \overline{\underline{M}} \; \underline{\phi}_n = 1$$

Therefore, the following relationships apply:

$$\underline{\Phi}^T \; \overline{\underline{M}} \; \underline{\Phi} = \underline{I} \tag{10.11}$$

$$\underline{\Phi}^T \; \overline{\underline{K}} \; \underline{\Phi} = \underline{\omega}^2 \tag{10.12}$$

Where I is the unit matrix and $\underline{\omega}^2$ is a diagonal matrix of the frequency squared.

10.3 Uncoupled Dynamic Equilibrium Equations

The displacements $\underline{V}$, in Equation (10.4), are now expressed in terms of the mode shapes $\underline{\Phi}$ and the modal amplitudes $\underline{X}$ by introducing the following coordinate transformation:

$$\underline{V} = \underline{\Phi} \; \underline{X} \tag{10.13}$$

Therefore,

$$\dot{\underline{V}} = \underline{\Phi} \; \dot{\underline{X}} \tag{10.14}$$

$$\ddot{\underline{V}} = \underline{\Phi} \; \ddot{\underline{X}} \tag{10.15}$$

The substitution of Equations (10.13), (10.14) and (10.15) into Equation (10.4) yields

$$\overline{\underline{M}} \; \underline{\Phi} \; \ddot{\underline{X}} + \overline{\underline{C}} \; \underline{\Phi} \; \dot{\underline{X}} + \overline{\underline{K}} \; \underline{\Phi} \; \underline{X} = \overline{\underline{F}} \tag{10.16}$$

This equation is now premultiplied by $\underline{\Phi}^T$.

$$\underline{\Phi}^T \underline{\overline{M}} \underline{\Phi} \underline{\ddot{X}} + \underline{\Phi}^T \underline{\overline{C}} \underline{\Phi} \underline{\dot{X}} + \underline{\Phi}^T \underline{\overline{K}} \underline{\Phi} \underline{X} = \underline{\Phi}^T \underline{\overline{F}} \qquad (10.17)$$

For most structures the exact form of the damping matrix is unknown. Also, in most cases its effect on the vibration mode shapes of the structure is small; therefore, an assumption as to the form of this matrix is justifiable. Specifically, it is assumed that the damping matrix is selected so that orthogonality of the damping forces is maintained:

$$\underline{\phi}_n^T \underline{\overline{C}} \underline{\phi}_m = 0 \quad m \neq n$$

And so that

$$\underline{\phi}_n^T \underline{\overline{C}} \underline{\phi}_m = \alpha_n \quad m = n$$

in which $\alpha_n = 2 \lambda_n \omega_n$, and λ_n is the ratio of modal damping to critical damping in the n^{th} mode. Therefore, the following equation is satisfied

$$\underline{\Phi}^T \underline{\overline{C}} \underline{\Phi} = \underline{\alpha} \qquad (10.18)$$

where $\underline{\alpha}$ is a diagonal matrix. This implies that the damping is "uncoupled" with respect to the mode shapes.

The generalized force vector is now defined as

$$\underline{\overline{F}} = \underline{\Phi}^T \underline{\overline{F}} \qquad (10.19)$$

The substitution of Equations (10.11), (10.12), (10.18) and (10.19) into Equation (10.16) yields

$$\underline{I}\,\ddot{\underline{X}} + \underline{\alpha}\,\dot{\underline{X}} + \underline{\omega}^2\,\underline{X} = \overline{F} \tag{10.20}$$

Equation (10.20) is a set of "uncoupled", second order, ordinary differential equations which may be evaluated by standard step-by-step or Duhamel integration techniques. After these generalized coordinates are found the generalized displacements are determined by the direct application of Equation (10.13). The actual displacements of the structure are then calculated from Equation (10.3a).

XI. EARTHQUAKE SPECTRAL ANALYSIS OF THREE DIMENSIONAL STRUCTURES

In terms of absolute accelerations the equilibrium equations for a three dimensional structure subjected to ground motions in two directions are

$$\underline{m}\ \ddot{\underline{u}}_a + \underline{c}\ \dot{\underline{u}}_r + \underline{k}\ \underline{u}_r = 0 \tag{11.1}$$

where

$$\ddot{\underline{u}}_a = \begin{bmatrix} \ddot{\underline{u}}_{xa} \\ \ddot{\underline{u}}_{ya} \\ \ddot{\underline{u}}_{\theta a} \end{bmatrix} = \begin{bmatrix} \ddot{\underline{u}}_{xg} \\ \ddot{\underline{u}}_{yg} \\ 0 \end{bmatrix} + \begin{bmatrix} \ddot{\underline{u}}_{xr} \\ \ddot{\underline{u}}_{yr} \\ \ddot{\underline{u}}_{\theta r} \end{bmatrix} = \ddot{\underline{u}}_g + \ddot{\underline{u}}_r \tag{11.2}$$

Therefore,

$$\underline{m}\ \ddot{\underline{u}}_r + \underline{c}\ \dot{\underline{u}}_r + \underline{k}\ \underline{u}_r = \underline{p} \tag{11.3}$$

where

$$\underline{p} = -\begin{bmatrix} \underline{m}_x & 0 & 0 \\ 0 & \underline{m}_y & 0 \\ 0 & 0 & \underline{m}_\theta \end{bmatrix} \begin{bmatrix} \underline{u}_{xg} \\ \underline{u}_{yg} \\ 0 \end{bmatrix} = -\begin{bmatrix} \underline{m}_x & \underline{u}_{xg} \\ \underline{m}_y & \underline{u}_{yg} \\ 0 \end{bmatrix} \tag{11.4}$$

The solution of the free vibration problem

$$\underline{m}\ \ddot{\underline{u}}_r + \underline{k}\ \underline{u}_r = 0 \tag{11.5}$$

yields the three dimensional mode shapes $\underline{\phi}$ and frequencies $\underline{\omega}$. These are normalized so

$$\underline{\phi}^T \underline{m} \underline{\phi} = \underline{I} \tag{11.6}$$

$$\underline{\phi}^T \underline{k} \underline{\phi} = \underline{\omega}^2 \tag{11.7}$$

The true displacements are now expressed in terms of these mode shapes

$$\underline{u}_r = \underline{\phi} Z; \quad \dot{\underline{u}}_r = \underline{\phi} \dot{Z}; \quad \text{and} \quad \ddot{\underline{u}}_r = \underline{\phi} \ddot{Z} \tag{11.8}$$

where Z represents the response of each mode. Substitution of Eq. (11.8) into Eq. (11.3) yields

$$\underline{m} \underline{\phi} \ddot{\underline{Z}} + \underline{c} \underline{\phi} \dot{\underline{Z}} + \underline{k} \underline{\phi} \underline{Z} = \underline{p} \tag{11.9}$$

Premultiplication of Eq. (11.8) by $\underline{\phi}^T$ produced a set of second order differential equations of the form

$$\underline{M} \ddot{\underline{Z}} + \underline{C} \dot{\underline{Z}} + \underline{K} \underline{Z} = \underline{P} \tag{11.10}$$

where

$$\underline{M} = \underline{\phi}^T \underline{m} \underline{\phi} = \underline{I} \tag{11.11}$$

$$\underline{C} = \underline{\phi}^T \underline{c} \underline{\phi} = \left[2 \lambda \omega\right] \tag{11.12}$$

(This involves an assumption on the type of damping)

$$\underline{K} = \underline{\phi}^T \underline{k} \underline{\phi} = [\omega^2] \tag{11.13}$$

$$\underline{P} = \underline{\phi}^T \underline{p} = -[\phi_x^T \quad \phi_y^T \quad \phi_\theta^T] \begin{bmatrix} \underline{m}_x & \ddot{\underline{u}}_{xg} \\ \underline{m}_y & \ddot{\underline{u}}_{yg} \\ 0 & \end{bmatrix}$$

or

$$\underline{P} = \underline{P}_x + \underline{P}_y = - \underline{\phi}_x^T \underline{m}_x \ddot{\underline{u}}_{xg} - \underline{\phi}_y^T \underline{m}_y \ddot{\underline{u}}_{yg} \qquad (11.14)$$

Therefore, a typical equation for the n^{th} mode is of the form

$$\ddot{Z}_n + 2 \lambda \omega_n \dot{Z}_n + \omega_n^2 Z_n = P_{nx} + P_{ny} \qquad (11.15)$$

By definition the spectral displacement is the maximum displacement of a unit mass system governed by the equation

$$\ddot{v} + 2 \lambda \omega \dot{v} + \omega^2 v = \ddot{u}_g(t) \qquad (11.16)$$

For a given frequency and earthquake this spectral displacement has been calculated as $s(\omega)$

Therefore, for the three dimensional structure subjected to an acceleration in the x-direction the maximum response of the n^{th} mode will be

$$Z_{nx}^{(max)} = \underline{\phi}_{nx}^T \underline{m}_x s_x(\omega_n) \qquad (11.17)$$

Or the true three-dimensional maximum displacement of the structure subjected to an x ground acceleration for the n^{th} mode is

$$\underline{u}_{nx}^{(max)} = \Phi_n \; Z_{nx}^{(max)} \tag{11.18}$$

Note that all components of displacement may exist in a given mode for this type of loading.

Similarly, for ground acceleration in the y-direction

$$\underline{u}_{ny}^{(max)} = \Phi_n \; Z_{ny}^{(max)} \tag{11.19}$$

where

$$Z_{ny}^{(max)} = \Phi_{ny}^{T} \; \underline{m}_y \; s_y \; (\omega_n) \tag{11.20}$$

A good estimation of the maximum displacements and stresses is determined by calculating the root-mean-square of the maximum modal values.

XII. COMPUTER PROGRAM ORGANIZATION

The computer program is coded in standard FORTRAN IV and is practically machine independent. All storage is allocated at the time of execution; therefore, the minimum storage required will depend on the size of the structure. To increase the capacity of the program it is necessary to change one number in a dimension statement.

For static analysis the program is divided into four phases. A machine dependent overlay system is not used; instead, a COMMON storage area is used in each phase. These four are executed in the following sequence:

1. Data Input - Joint coordinates and loads are read or generated. As element properties are read or generated the element stiffness matrices are formed and placed on tape.
2. Formation of total stiffness is accomplished by reading the element stiffness tape and forming the joint equilibrium equations in blocks.
3. Equilibrium equations are solved for joint displacements, all load conditions are treated at the same time.
4. From the joint displacements, element stresses are calculated for all load conditions.

In the following sections these are explained in greater detail.

The dynamic analysis is very similar to the static analysis. After the displacements for all the static load patterns are determined, the mode shapes and frequencies are evaluated by the technique described in the previous section. The response of each mode is evaluated. The specified joint displacement and member stresses are then calculated. An unique printer plot subroutine allows time-dependent results to be presented in a compact form.

12.1 Solution of Equations

The computer program is built around a large capacity linear equation solver, USOL. The procedure used to solve the equations is not significantly different from the method developed by Gauss in 1827. The banded characteristics of the equations are recognized. Operations with zero coefficients are skipped. Data is transferred in and out of high speed storage in large blocks; therefore, a small amount of time is lost in the transfer of data.

The equilibrium equations (the stiffness matrix and loads) are stored in blocks on tape (or other low speed storage units). During the solution phase two blocks must be in high speed storage at any time. Therefore, the physical storage restriction is that there must be high speed storage available for at least two equations. For example, if the stiffness matrix has a band width of 250 and if there are 20,000 high speed storage locations available, the number of equations in a block will be ~~50~~ 40. Hence, for this example all data transfer will be in blocks of 10,000. The block size is automatically determined at the time of solution. Therefore, storage is utilized in the most efficient manner for a particular structure.

12.2 Formation of Equilibrium Equations

Before the total stiffness matrix is formed the element stiffness matrices are calculated and stored in sequence on low speed storage. The total stiffness matrix is formed two blocks at a time by making a pass through the element stiffness matrices and adding in the appropriate coefficients. In order to minimize the effort in searching through all the element stiffnesses the element stiffness matrices for several blocks are transferred to another storage unit; therefore, in the formation of

the next several blocks the time to search for the contributions to these blocks is reduced significantly.

12.3 Joint Input Data and Degrees of Freedom

The capacity of the program is controlled by the number of joint (nodal points) of the structural system. All joint data is retained in high speed storage during the formation of the element stiffness matrices. For each joint three coordinates and six boundary condition codes are required; therefore, the minimum required storage for a given problem is nine times the number of joints in the system.

Immediately after the joint data is supplied to the program a relationship between each joint degree of freedom and the corresponding equation number is established. Each of the six boundary condition codes for a given joint is replaced by the equation number for that degree of freedom. Restrained boundary conditions are identified by a zero equation number. Slave degrees of freedom (for beam elements) are identified by a negative joint number of the master node.

12.4 Calculation of Element Stiffness Matrices

After the coordinate of the joints are supplied and the equation numbers of the degrees of freedom established the stiffness and stress-displacement transformation matrices are calculated for each structural element in the system. Very little additional high speed storage is required for this phase since these matrices can be formed and placed on tape storage as the element properties are read. In addition to the element matrices the corresponding equation numbers are written on tape. After all element matrices are formed the joint coordinates and boundary condition information is not required; hence,

this storage area can be used subsequently for storage for the two blocks of the equilibrium equations. It is now possible to form and solve these equations as previously described.

12.5 Evaluation of Element Stresses

After the joint displacements are evaluated a pass is made through the element stress-displacement matrix tape and the element stresses are calculated and printed.

APPENDIX - DESCRIPTION OF INPUT DATA

The purpose of this computer program is to perform linear, elastic analyses of three dimensional structural systems. The structural systems to be analysed may be composed of combinations of a number of structural element types. The present version contains the following element types:

1) three dimensional truss
2) three dimensional beam
3) plane stress and plane strain
4) two dimensional axisymmetric solid
5) three dimensional solid
6) plate and shell
7) boundary

Systems composed of large numbers of joints and members may be analysed. The capacity of the program depends mainly on the total number of joints in the system. There is practically no restriction on the number of elements, number of load cases, or the "bandwidth" of the equations to be solved. Note, that while the program has the capacity to analyse very large systems, there is no loss of efficiency in the solution of smaller problems as compared to several special purpose programs presently available.

A general outline of the main features of the program is given here, followed by a more detailed user-manual.

JOINTS

Each joint in the system may have from 0 to 6 degrees of freedom as required. The user must ensure that the degrees of freedom specified for a given joint are compatible with the element-types which are adjacent to it.

Optimum solution efficiency is obtained by minimizing the number of degrees of freedom of the system. Also, joints connected only to Beam elements may use a special Slave-Master geometric constraint option to eliminate unnecessary degrees of freedom. (see Beam section)

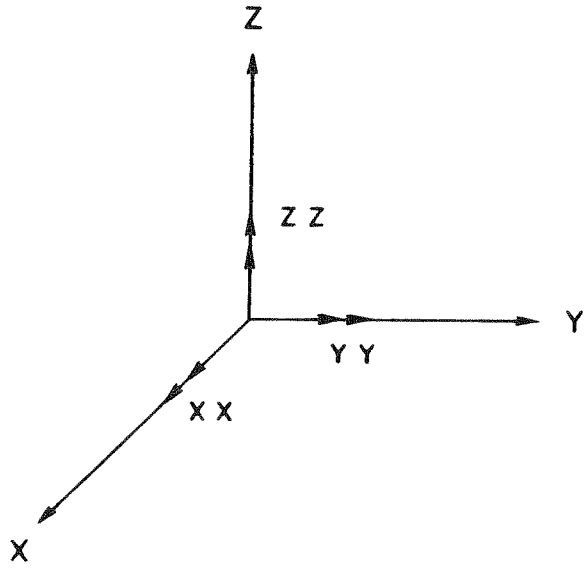
A right-handed orthogonal co-ordinate system, shown below, is used to describe the geometry of the structure. All joint loads and displacements are defined with reference to this system. A local co-ordinate system is used for each element type.

LOADING

Loads may be applied by means of both point loads acting at the joints and by element loading (e.g. gravity, temperature). Each element may have up to four different load cases called A, B, C, D. Loading for one solution consists of joint loads plus a linear combination of element load cases A, B, C and D. The types of loading which make up the element load cases are determined by each individual element type.

(N.B. The current version of the plane stress element is not standard in this regard (see description of the element) and care must be exercised when using this element in combination with other types.)

Imposed displacement loading is possible by means of a special boundary element.



GLOBAL COORDINATE SYSTEM

PROGRAM CAPACITY

The capacity of the program can be changed depending on the size of the problem to be solved. This is done by changing the two Fortran statements at the start of SAP, i.e.

```
COMMON  A(n)
MTOT = n
```

The minimum value of n needed is computed as follows:

n = 9* (number of joints) + M

where

M = the maximum value of each of the following:

(1) Truss elements

M = 5* NMAT NMAT = number of material types

(2) Beam elements

M = 3*NMAT+12*NFIX+6*NPROP

NFIX = number of fixed end force sets

NPROP = number of different beam properties

(3) Plane stress and plane strain elements.

M = 4*NMAT+11*NMAT*NTC

NTC = number of material temperatures

(4) Axisymmetric quadrilateral

M = 4*NMAT

(5) Three-dimensional solid elements

M = 4*NMAT+4*NLD+2475 NLD = number of element load sets

(6) Plate and shell elements

M = 12*NMAT

(7) Boundary elements

M = 0

Note: (1) A convenient general rule for computing a minimum value of n (except for 3-D solid) is:

n = 10* (number of joints)

(2) For optimum efficiency, however, a value of n, considerably greater than the minimum, should be used.

(3) If the value of n is set too small an error message is printed and program execution is terminated.

ELEMENT TYPES

The following is a brief outline of the elements currently included:

(1) Three-dimensional Truss Elements.

A uniform temperature change and inertia loads in three directions can be considered as the basic element loads. Axial forces and stresses are computed.

(2) Three-dimensional Beam Elements.

Beam elements are straight, prismatic beam members. Inertia loading (e.g. gravity) 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.

(3) Plane Stress and Plane Strain Elements.

An arbitrary quadrilateral (or triangular) element is used. The plane of the element may lie in any direction. Gravity, inertia, and temperature loadings may be considered. Stresses are computed at the center of the element.

(4) Axisymmetric Quadrilateral Elements.

An arbitrary quadrilateral (or triangular) element is used. The element is axisymmetric about the global Z-axis and the Y direction is considered radial. Temperature, surface pressure and inertia (Z direction) loading are included. Stresses are computed at the center of the element.

(5) Three-dimensional Solid Element.

A general 8 nodal point "brick" element, with 3 translational degrees of freedom per nodal point, is used. Isotropic material properties are assumed, and element loading consists of

temperature, surface pressure and inertia loads in three directions. Stresses (6 components) are computed at the center of the element and at the center of each face.

(6) Plate and Shell Elements.

An arbitrary quadrilateral element is used. Gravity, inertia, pressure and temperature loadings may be considered. Stresses are computed at the center of the element.

(7) Boundary Element.

This element is used to impose displacement boundary conditions and to compute support reactions.

INPUT DATA ASSOCIATED WITH JOINTS

The geometry of the joints, the boundary conditions, and the joint concentrated loads are numerically defined by a sequence of punched cards. The properties of the different structural members are described separately.

I. Heading Card (12A6)

Columns 1 - 72 Contain information to be printed with output

II. Control Card (4I5)

Columns	1 - 5	Number of joints in system
	6 - 10	Number of element groups
	11 - 15	Number of load conditions
	16 - 20	Number of frequencies (= 0 for static analysis)

III. Joint Data (7I5,3F10.0,I5)

The following information must be given for each joint in the system:

Columns	1 - 5	Joint Number
	6 - 10	X-direction
	11 - 15	Y-direction
	16 - 20	Z-direction
	21 - 25	Rotation about X-axis
	26 - 30	Rotation about Y-axis
	31 - 35	Rotation about Z-axis
	36 - 45	X-ordinate
	46 - 55	Y-ordinate
	56 - 65	Z-ordinate
	66 - 70	KN -- Joint cards need not be in joint-order sequence. If cards are omitted, the joint data for a series of joints is generated. KN is a mesh generation parameter on the last card of a mesh generation sequence. KN is the increment to be added to the previous nodal point number. The intermediate joints are located at equal intervals along the straight line. The boundary condition codes for the generated joint data are set equal to the boundary condition codes on the first joint card in the series.

Boundary Condition Codes:

Zero or blank indicates that the joint is free to move in that direction and loads may be applied.

One indicates that the joint is fixed in that direction.

If a particular degree of freedom is fixed for a series of cards, this may be indicated by a boundary condition code of -1 on the first joint card in the series and + 1 on the last joint card in the series.

IV. Element Data

A sequence of cards is required for each type of element in the structure. The form of this data for each type is described later.

V. Concentrated Load Data (2I5,6F10.0)

One card per load case for each joint which has non-zero concentrated loads or moments applied. The cards must be in joint-number sequence.

Columns	1 - 5	Joint number
	6 - 10	Load condition number
	11 - 20	Load X-direction
	21 - 30	Load Y-direction
	31 - 40	Load Z-direction
	41 - 50	Moment X-axis
	51 - 60	Moment Y-axis
	61 - 70	Moment Z-axis

This sequence of cards must be terminated with one blank card.

VI. Element Load Multipliers (4F10.0)

Four different types of loads associated with the element are possible. These element loads are referred to as load cases A, B, C, and D. By the use of "Element Load Multipliers," it is possible to add fractions of the basic element loads to any of the concentrated load conditions.

One card must be supplied for each load condition which contains the following information:

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

These cards must be in load-order sequence. The definitions of the element loads associated with a particular element type are discussed in detail under the section "Element Data".

1. THREE-DIMENSIONAL TRUSS MEMBERS

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 member load conditions. The truss members are described by the following sequence of cards:

A. Control Card (3I5)

Columns	1 - 5	The number 1
	6 - 10	Number of truss members
	11 - 15	Number of members with different properties

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

One card is required for each member which has a different cross-section or different material properties.

Columns	1 - 5	Material identification number
	6 - 15	Modulus of elasticity
	16 - 25	Coefficient of thermal expansion
	26 - 35	Mass density (mass/volume)
	36 - 45	Cross-sectional area
	46 - 55	Weight per unit length

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. Member Data Cards (4I5,F10,0,I5)

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

Columns		
	1 - 5	Member number (n)
	6 - 10	Joint number I
	11 - 15	Joint number J
	16 - 20	Member identification number
	21 - 30	Temperature change
	31 - 35	Optional parameter K causing automatic generation of number data.

If a series of elements exist such that the member number, N_i, is one greater than the previous member number (i.e. $N_i = N_{i-1} + 1$) and the joint 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 member identification number and the temperature for the generated elements are set equal to the values on the first card. If K is input as zero it is set to 1 by the program.

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. Inertia loadings in each coordinate direction form the basic member load conditions.

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

A. Control Card (5I5)

Columns	1 - 5	The number 2
	6 - 10	Number of beam elements
	11 - 15	Number of geometric property cards
	16 - 20	Number of fixed end moment sets
	21 - 25	Number of different materials

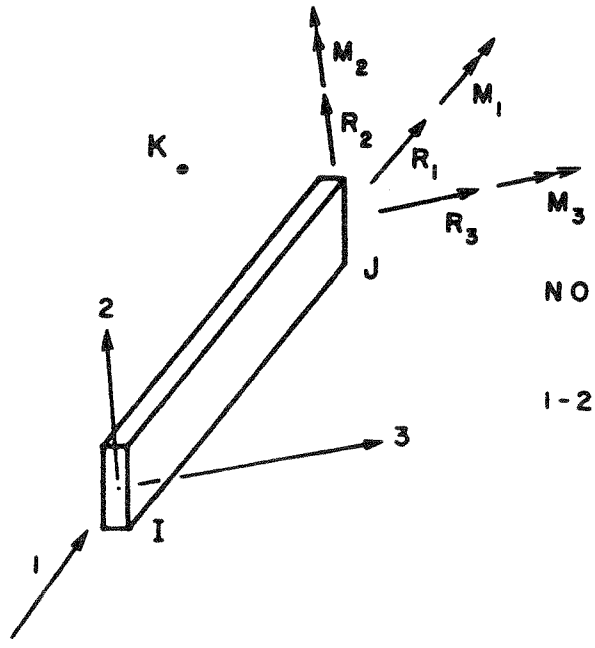
B. Material Property Cards (I5,3F10.0)

Columns	1 - 5	Material identification - any number from 1 to 10
	6 - 15	Young's modulus
	16 - 25	Poisson's ratio
	26 - 35	Mass density (mass/unit volume)

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

Columns	1 - 5	Identification - any integer 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 area is included 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 I-AXIS)

LOCAL COORDINATE SYSTEM FOR BEAM ELEMENT

D. Element Load Factors (4F10.0)

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

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.

Card 1:

Columns	1 - 5	Identification - any number from 1 to 99
	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

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	Identification - beam number	
6 - 10	Node I number	
11 - 15	Node J number	
16 - 20	Node K number - see below	
21 - 25	Material number	
26 - 30	Geometric property number	
31 - 35	A	Fixed-end force identification for element load cases A, B, C, and D respectively
36 - 40	B	
41 - 45	C	
46 - 50	D	
51 - 56	End release code - Node I	
57 - 62	End release code - 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 member end forces is known to be zero (hinge or roller), the digit corresponding to that component is a one.

Automatic Element Data Generation

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.

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

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

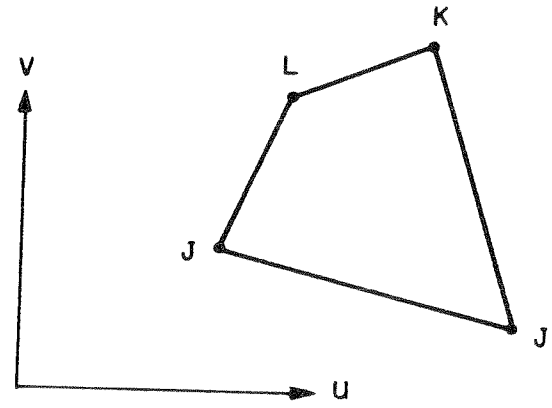
AND THE
a) material identification number
b) geometric property identification number
c) fixed-end force identification numbers for each element load case
d) element code
e) orientation of local 2-axis

are the same for each element in the series.

The generator option is the value of k and if left blank is taken to be one. The element data card for the last element in the structure must always be given.

*Where 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.

3. PLANE STRESS AND PLANE STRAIN ELEMENTS



LOCAL ELEMENT COORDINATE SYSTEM

A. Control Card (5I5)

Columns	1 - 5	The number 3
	6 - 10	Total number of plane stress and plane strain elements.
	11 - 15	Number of different materials.
	16 - 20	Maximum number of temperatures for any material.
	21 - 25	Identification for u-v plane:
		Blank or 0 — X-Y plane
		1 — Otherwise

If some elements do not lie in the X-Y plane, then a 1 is placed in column 25 and the u axis is assumed to coincide with side IJ of the element. v is normal to u and lies in the plane of the element.

B. Material Property Information

Orthotropic, temperature dependent material properties are possible.

For each different material the following group of cards must be supplied.

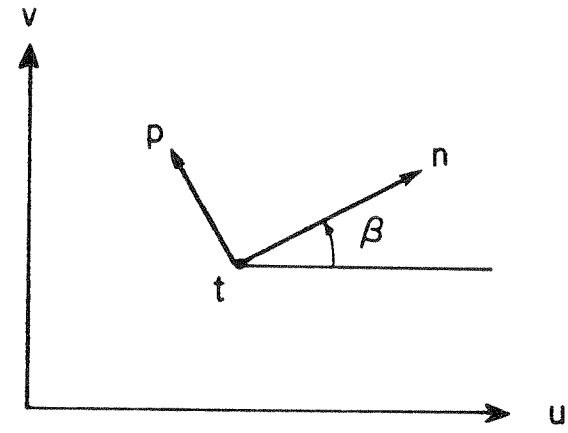
1. Material Identification Card (2I5,3F10.0)

Columns	1 - 5	Material identification -- any number from 1 to 6.
	6 - 10	Number of different temperatures for which properties are given. If this field is left blank the number is taken as one.

Columns 11 - 20 Weight density of material

21 - 30 Mass density of material.

31 - 40 Angle B in Degrees measured counter-clockwise from the u-axis to the n-axis.



PRINCIPAL MATERIAL AXES

The u-v axes are defined by the identification in column 25 of the CONTROL CARD for Plane Stress and Plane Strain elements.

The n-s axes are the principal axes for the orthotropic material.

Weight and mass densities are listed only if gravity and inertia loads are to be considered.

2. Material Property Cards - Two cards for each temperature.

Card 1: (8F10.0)

Columns 1 - 10 Temperature

11 - 20 Modulus of Elasticity - En

21 - 30 Modulus of Elasticity - Es

31 - 40 Modulus of Elasticity - Et*

41 - 50 Strain Ratio - νns

51 - 60 Strain Ratio - νnt*

61 - 70 Strain Ratio - νst*

71 - 80 Shear Modulus - Gns

Columns	1 - 10	Coefficient of Thermal expansion - αn
	11 - 20	Coefficient of Thermal expansion - αs
	21 - 30	Coefficient of Thermal expansion - αt*

* Plane stress is characterized by Et = νnt = νst = αt = 0
Plane strain is assumed for non-zero Et, νnt, νst.

Listing of the coefficients of thermal expansion is necessary only for thermal stress analysis. A blank second card would be used if thermal loads are not to be included.

C. Element Load Card (7F10.0)

Element load case A: Gravity Load

1 - 10	X portion of gravity load
11 - 20	Y portion of gravity load
21 - 30	Z portion of gravity load

Element load case B: Inertia load

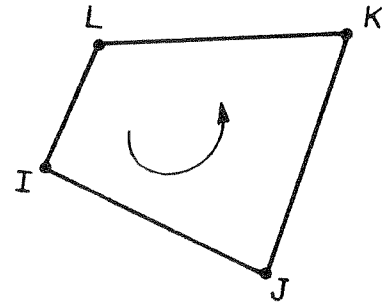
31 - 40	X component of acceleration
41 - 50	Y component of acceleration
51 - 60	Z component of acceleration

Element load case C: Thermal load

61 - 70	Reference Temperature (Stress free temperature)

No element loads are acting on plane stress or plane strain elements during element load case D.

D. Element Cards (6I5,2F10.0,3F5.0,I5)

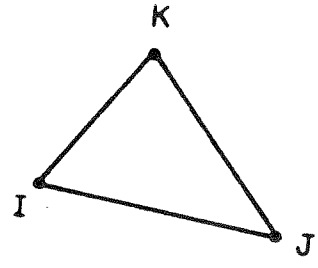


One Card per element.

Nodal points should be ordered counter-clockwise around the element.

Columns		
	1 - 5	Element Number
	6 - 10	Node I
	11 - 15	Node J
	16 - 20	Node K
	21 - 25	Node L*
	26 - 30	Material Identification (If left blank taken as one)
	31 - 40	Temperature
	41 - 50	Thickness (Blank for plane strain)
	51 - 55	Load factor for element load case A: gravity load
	56 - 60	Load factor for element load case B: inertia load
	61 - 65	Load factor for element load case C: thermal load
	66 - 70	Element data generator K described below under the heading ELEMENT DATA GENERATION. If this option is not used, then the field should be blank.

* Triangular elements are assumed when columns 21 - 25 are blank.



NOTE: Material properties versus temperature are input in tabular form for each different material. The properties for a particular element are evaluated by interpolation.

ELEMENT DATA GENERATION FOR PLANE STRESS OR PLANE STRAIN ELEMENTS

If a series of elements occurs such that each element number NE_i is one greater than the previous number NE_{i-1}

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

and the nodal point numbers belong to the following sequence:

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

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

$$NK_i = NK_{i-1} + k$$

$$NL_i = NL_{i-1} + k$$

then only the element card for the first element in the series need by input provided the following restrictions are satisfied.

(i) material identification number

(ii) temperature

(iii) thickness

(iv) element load factors must be the same for all elements in the series.

4. AXISYMMETRIC QUADRILATERAL ELEMENTS

Quadrilateral solid elements symmetrical about the Z-axis are identified by the number 4. The Y nodal point coordinate is interpreted as the radius R. Stresses are calculated at the center of each element. A uniform element temperature change, surface pressures, and inertia loads in the Z-direction are considered as the basic element load conditions. The axisymmetric elements are described by the following sequence of cards:

A. Control Card (3I5)

Columns	1 - 5	The number 4
	6 - 10	Number of axisymmetric elements
	11 - 15	Number of different materials

B. Material Property Cards (I5,4F10.0)

One card for each different material

Columns	1 - 5	Material identification number
	6 - 15	Modulus of elasticity
	16 - 25	Poisson's ratio
	26 - 35	Mass density
	36 - 45	Coefficient of thermal expansion

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

Card 1: Multipliers for element thermal load

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: Multipliers for element pressure loads

Card 3: Acceleration multipliers for the calculation of loads in the Z-direction.

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

One card per element in increasing numerical order

Columns	1 - 5	Element number	
	6 - 10	Nodal point I	The maximum difference "b" between these numbers is an indication of the band width. The execution time for the program will be proportional
	11 - 15	Nodal point J	
	16 - 20	Nodal point K	
	21 - 25	Nodal point L	
	26 - 30	Material identification number	this number squared
	31 - 40	Temperature change within element	
	41 - 50	Normal pressure acting on I-J surface	

For a right hand coordinate system, the nodal point numbers I, J, K, and L must be in sequence in a counter-clockwise direction around the element. Element cards must be in element number sequence. If element cards are omitted, the program automatically generates the omitted information by incrementing by one the preceding I, J, K and L. The material identification for the generated cards is set equal to the corresponding value on the last card. The last element card must always be supplied Triangular elements are also permissible; they are identified by repeating the last nodal point number (i.e., I, J, K, L).

5. THREE-DIMENSIONAL SOLID ELEMENT: 8 NODAL BRICK

A. Control Card (4I5)--

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

B. Material Property Cards (I5,4F10)--

Columns	1 - 5	Material identification--any integer from 1 to NMAT
	6 - 15	Modulus of elasticity, E (only elastic, isotropic materials are considered)
	16 - 25	Poisson's Ratio, ν
	26 - 35	Weight density of material
	36 - 45	Coefficient of thermal expansion, α_T

C. Element Distributed Load Set Cards (2I5,2F10,I5)--

Columns	1 - 5	Load set identification--any integer from 1 to NLD
	6 - 10	Load Type (KTYPE)
		KTYPE = 1 for constant surface pressure (normal)
		KTYPE = 2 for hydrostatic pressure
	11 - 20	PR = pressure for KTYPE = 1
		= specific gravity for KTYPE = 2
	21 - 30	YREF = Reference water level (hydrostatic pressure varies in Global Y direction); is ignored if KTYPE = 1
	31 - 35	Element face which pressure acts upon

See Note B for element face details and pressure sign convention.

D. Reference Temperature (2F10)--

Columns	1 - 10	Stress free temperature
	11 - 21	Acceleration due to gravity

E. Element Load Case Factors (5 cards of 4F10)--Pressure and thermal load factors on the element load cases are scaling factors in order to provide flexibility in modifying applied loads.

Card 1: Columns 1 - 10 | 11 - 20 | 21 - 30 | 31 - 40 — Pressure Load Factors for Element Load Case — A | B | C | D

Card 2: As card 1 but for thermal effects; see note E

Card 3: Gravity Factors for + X direction

Columns 1 - 10 | 11 - 20 | 21 - 30 | 31 - 40 — Percentage of Gravity Acting in + X Direction in Element Load Case — A | B | C | D

Card 4:

As card 3 but for + Y and + Z directions

Card 5:

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

Columns 1 - 5 Element number

6 - 10
11 - 15
16 - 20
21 - 25
26 - 30
31 - 35
36 - 40
41 - 45

Global Node Point Numbers Corresponding to Element Nodes (See Note A) — 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8

46 - 50 Integration Order (NINT)

51 - 55 Material Number

56 - 60 Generation Parameter (INC)

61 - 62
63 - 64
65 - 66
67 - 68

Distributed Load Set Number for Element Load Cases (Zero implies no load) — A | B | C | D

69 - 70 Face Numbers for Stress Output

71 - 80 Element Temperature

N. B. 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 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 NINT 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 |NINT| 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 <u>not</u> to succeeding elements in the series.

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

G. <u>Integration Order</u>

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

2. for rectangular element

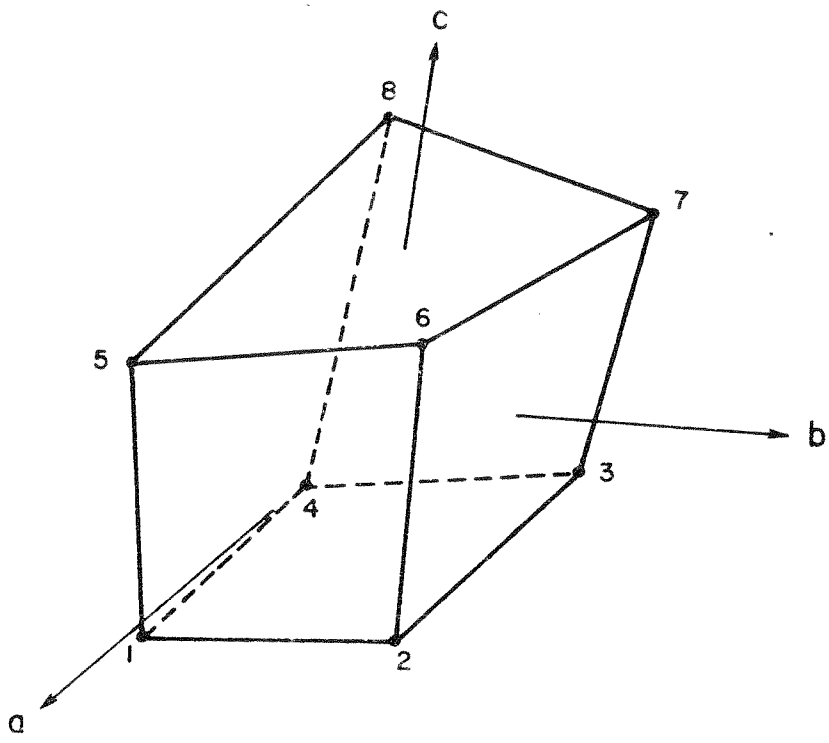
3. for skewed element

4. may be used if element is extremely distorted in shape, but not recommended.

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

H. Element Coordinate System

Local element coordinate system is a natural system for this element in which the element maps as a unit cube. Local element numbering is shown in Figure 1.



I. 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

Sign Convention: A positive distributed load acts in the positive (local) axis direction associated with each face.

J. Element Stresses are Output as Follows

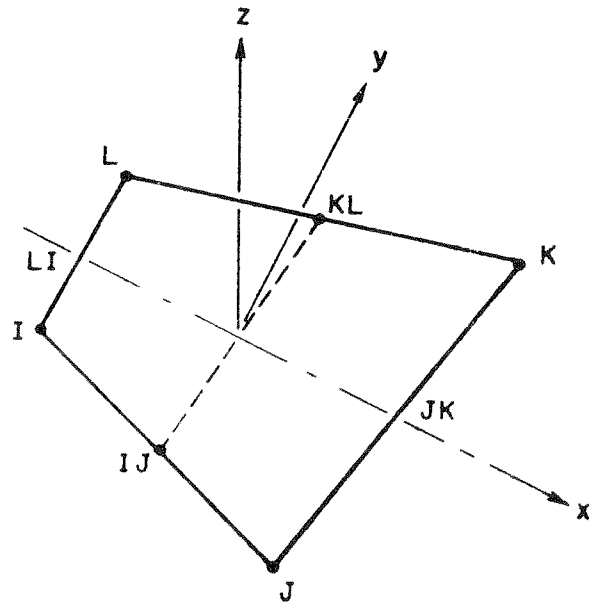
1. At the centroid of the element stresses are referred to the global axes. Three principal stresses are also presented.

2. At the centroid of an element face stresses are refereed to a set of local axes (x, y, z). These local axes are individually defined for each face in such a way: Let nodal points I, J, K and L are the four corners of the element face. Then

   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.



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 of the direction of the bigger stress to the local x axis are presented.

It is optional to choose whether one or two locations of an element where stresses are to be computed. In the output face zero designates the centroid of the element.

6. PLATE AND SHELL ELEMENTS (QUADRILATERAL)

A. Control Card (3I5)

Columns 1 - 5 The number 6
6 - 10 Total 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
31 - 40 Mass density
41 - 50 Thermal expansion coefficient α_x
51 - 60 Thermal expansion coefficient α_y
61 - 70 Thermal expansion coefficient α_z

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]

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

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_n**
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 $\partial T/\partial z$ across the shell thickness (a positive temperature gradient produces a negative curvature).

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

** 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

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

The corresponding increment for nodal number is K_n

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

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

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

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

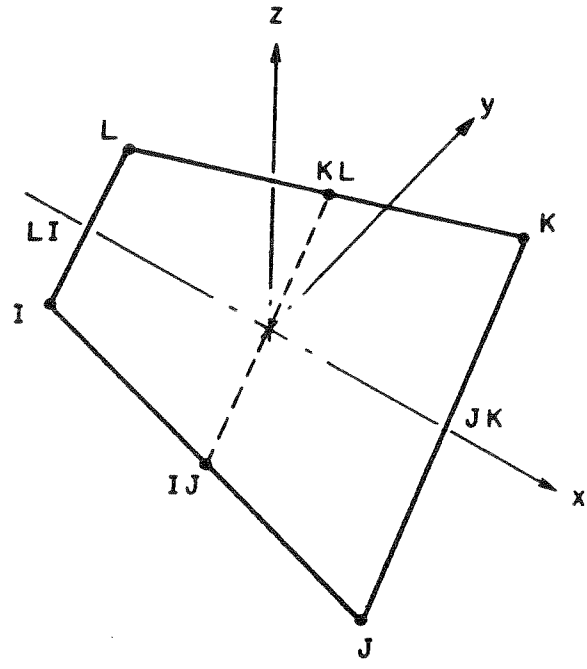
Material identification, element thickness, distributed lateral load, temperature and temperature gradient for generated elements are the same as the first element in the series. The last element card is always needed.

NOTE

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

7. BOUNDARY ELEMENT

This element can be used to constrain displacements to specified values and to compute support reactions. A large stiffness spring is used as the element.

The direction of the constraint is defined in either of two ways:

(1) A second node point (I) defines the direction

(2) The direction is defined as normal to two lines specified by nodes IJ and KL.

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 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
	16 - 20	Node K
	21 - 25	Node L

Leave columns 11 - 25 blank if only node I is needed.

26 - 30	Code for displacement	0 - free
31 - 35	Code for rotation	1 - constrained
36 - 40	Data generator Kn.	When a series of nodes are subject to the same kind of constraint, only the first and last cards in the series are needed. Kn on the first card is the increment added to the previous node number. the same constraint direction is used for each element generated.

Columns	41 - 50	Specified normal displacement
	51 - 60	Specified rotation about normal
	61 - 70	Spring stiffness (set to 10^{10} if left blank)

DYNAMIC ANALYSIS

Three types of dynamic analysis can be performed by SAP. The type of analysis is indicated by a number, NDYN, in columns 21 to 25 of the control card for the joint input data. If

NDYN = 0 Static Analysis
= 1 Mode Shapes and Frequencies
= 2 Dynamic Response Analysis for Arbitrary Time Dependent Loads
= 3 Response Spectrum Analysis

Concentrated joint masses are supplied with the Concentrated Load Data and are identified by a zero load condition number. They must be in joint number sequence and are punched in the same format as the joint loads.

I. Mode Shapes and Frequencies NDYN = 1

The number of frequencies desired is specified in columns 16 - 20 of the joint data control card. Of course, the number of load conditions must be greater than the number of frequencies since a Rayleigh-Ritz method is used.

II. Dynamic Response Analysis NDYN = 2

This option uses the mode shapes and frequencies calculated by the Rayleigh-Ritz technique. Dynamic loads may be applied as

1) ground motion in any of 3 directions

and/or

2) time varying loads applied to any point on the structure (except "slave" degrees of freedom).

A set of time functions are specified by a set of discrete points. Linear interpolation is used within the program to evaluate intermediate points. A particular force at any point on the structure is then described by a scalar multiplier and one of the time functions. A delay time may also be specified for the force. Ground motion is specified by specifying that a particular function describes ground acceleration in a given (X, Y, or Z) direction.

The following input data is required (to be supplied after element load multipliers).

1. Control Card (5I5,2F10)

Columns		
1 - 5	Number of different time functions	
6 - 10	Ground motion indicator:	0 - no ground motion 1 - some ground motion
11 - 15	Number of different delay times (NAT)	
16 - 20	Total Number of time steps of interval DT	
21 - 25	Output interval for displacement and stresses	
26 - 35	Time step DT	
36 - 45	Damping factor fraction of critical	

* If any period is less than 5 x DT it is not used in the solution.

2. Arbitrary Time Varying Loads. (4I5,F10)

One card for each degree of freedom of structure that loads are applied to. (In ascending node point number order.)

Columns		
1 - 5	Node point number where load is applied	
6 - 10	Displacement component (= 1 to 6 for X, Y, Z, XX, YY, ZZ)	
11 - 15	Time function number	
16 - 20	Delay time number	
21 - 30	Scalar multiplier for time function	

This sequence of cards must terminate with a blank card; this card must be supplied even if arbitrary time varying loads are not applied.

3. Ground Motion Control Card. (6I5)

Required if and only if ground motions are to be included.

Columns		
1 - 5 6 - 10 11 - 15	Time function number corresponding to ground acceleration in	{X, Y, Z} direction
16 - 20 21 - 25 26 - 30	Delay time number for functions in if blank - delay time is zero	{X, Y, Z} direction

Zero time function number indicates no ground motion for the particular direction.

4. Delay Time Cards (8F10.0)

As many cards as required at 8 times per card - in order. If no delays supply 1 blank card.

5. Time Function Definition Cards

One set for each time function.

Card I (I5,F10,12A5)

Columns 1 - 5 Number of definition points
6 - 15 Scale factor: set to 1.0 if blank (for changing units, etc.
16 - 75 Heading

Cards 2: As many as required at 6 points per card (12F6.0)

Columns 1 - 6 Time
7 - 12 Function value
13 - 18 Time
19 - 24 Function value
etc.

6. Output Definition Cards

To minimize output - values for which histories are required must be specified.

A. Displacement Output.

Control Card (2I5)

Columns 1 - 5 Output type indicator
= 1 - Printed histories and maximums
2 - Printer plotted histories and maximums
3 - Maximums only
6 - 10 Plot spacing indicator* (ISP)

* Horizontal width of printer plot is constant (10"). Vertical spacing may be varied -- Output values are printed on every (ISP + 1)th line.

Node Cards (7I5)

As many cards as required (in node point order)

Columns	1 - 5	Node point required
	6 - 10	As many as required the numbers 1 to 6 for displacement component X, Y, Z, XX, YY, ZZ.
	11 - 15	
	31 - 35	
	etc.	

The displacement components for a joint may be specified in any order. The first blank or zero number will terminate the information on the card. The sequence must be terminate with a blank card.

B. Element Stress Output

One set for each element type -- in same order as element cards.

Each set is the same as the displacement output except node cards become element cards (format 13I5) and last 12 integers specify the desired stress components as appropriate for individual elements. See static output for the given stress components for each element. Stress component number corresponds to the number in order of printing.

III. Response Spectrum NDYN = 3

After Element Load Multipliers the following information must be supplied:

1. Control Card (I5)

   1, 2, or 3 for Analysis in X, Y, or Z direction

2. Acceleration Spectrum Cards (12A6/I5,F10/(2F10))

   Card 1: Heading (12A6)

   Card 2: (I5,F10.0)

Columns	1 - 5	Number of Definition points
	6 - 15	Scale factor

   Card 3: (2F10.0) One card per point

Columns	1 - 10	Period
	11 - 20	Acceleration value