

12



structures library

UG-0026

AD-A266 907



SAP IV USER'S GUIDE

VERSION 1.0

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

S DTIC
ELECTE
JUL 19 1993
A D

by

F. R. Johnson and T. J. Holland

Naval Civil Engineering Laboratory
Port Hueneme, California 93043-4328

248150

93-16182



11808

January 1993

Approved for public release; distribution is unlimited

93 6 10 072

REPORT DOCUMENTATION PAGE			Form Approved OMB No. 0704-018	
Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection information, including suggestions for reducing this burden, to Washington Headquarters Service, Directorate for Information and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.				
1. AGENCY USE ONLY (Leave blank)		2. REPORT DATE December 1992		3. REPORT TYPE AND DATES COVERED Final; October 1986 - September 1992
4. TITLE AND SUBTITLE SAP IV USER'S GUIDE - VERSION 1.0 - A STRUCTURAL ANALYSIS PROGRAM FOR STATIC AND DYNAMIC RESPONSE OF LINEAR SYSTEMS			5. FUNDING NUMBERS PR - 51-142A	
6. AUTHOR(S) F. R. Johnson and T. J. Holland				
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) Naval Civil Engineering Laboratory 560 Laboratory Drive Port Hueneme, CA 93043-4328			8. PERFORMING ORGANIZATION REPORT NUMBER UG-0026	
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) Naval Facilities Engineering Command 200 Stovall Street Alexandria, VA 22332-2300			10. SPONSORING/MONITORING AGENCY REPORT NUMBER	
11. SUPPLEMENTARY NOTES				
12a. DISTRIBUTION/AVAILABILITY STATEMENT Approved for public release; distribution is unlimited.			12b. DISTRIBUTION CODE	
13. ABSTRACT (Maximum 200 words) SAP IV is a general purpose finite element analysis program for linear structural analysis. The program can analyze structures that are subject to either static or dynamic loads. The structures can be described in terms of truss, frame, plate, shell, or brick elements or combinations of elements. The static loads include concentrated loading applied to discrete points on the structural models; and loads, concentrated loading applied to discrete points on the structural models; and loads, concentrated or uniform, applied to individual elements in the model. The loads can be combined into load cases. The dynamic analysis features include modal analysis to obtain mode shapes and frequencies, response history analysis using either modal superposition or direct integration techniques, and response spectrum analysis.				
14. SUBJECT TERMS Structural analysis, finite element analysis, modal analysis, response spectra analysis, time history analysis, static structural analysis, dynamic structural analysis			15. NUMBER OF PAGES 119	
			16. PRICE CODE	
17. SECURITY CLASSIFICATION OF REPORT Unclassified		18. SECURITY CLASSIFICATION OF THIS PAGE Unclassified	19. SECURITY CLASSIFICATION OF ABSTRACT Unclassified	20. LIMITATION OF ABSTRACT UL

PREFACE

The information contained in this report is technical in nature. It does not contain policy of the Naval Civil Engineering Laboratory, the Naval Facilities Engineering Command, the Assistant Commander of Engineering and Design, or the Director of Engineering Systems Management Division. This report is one of a series which is designed to provide technical information and documentation for studies conducted in support of the Naval Facilities Engineering Command Graphics Engineering and Mapping Systems, Computer-Aided Design, Cost Engineering System, and Guide Specifications System as they relate to the Naval Facilities System and life cycle facilities management.

DTIC QUALITY INSPECTED 8

Accession For	
NTIS CRA&I	<input checked="" type="checkbox"/>
DTIC TAB	<input type="checkbox"/>
Unannounced	<input type="checkbox"/>
Justification	
By	
Distribution /	
Availability Codes	
Dist	Avail and/or Special
A-1	

ACKNOWLEDGEMENT

The authors acknowledge the contributions of the original developers of SAP IV and authors of the original user manual (Ref 1). Most of the information in this current version of the program and associated user guide originated with their efforts. The Naval Civil Engineering Laboratory (NCEL) was one of the contributors towards the development of the original work. Therefore, NCEL is interested in supporting continued utilization of this technology as a foundation for additional work.

DISCLAIMER

This program is furnished by the Government and is accepted and used by the recipient with the express understanding that the United States Government makes no warranties, expressed or implied, concerning the accuracy, completeness, reliability, usability, or suitability for any particular purpose of the information and data contained in this program or furnished in connection therewith, and the United States shall be under no liability whatsoever to any person by reason of any use made thereof. The program belongs to the Government. Therefore, the recipient further agrees not to assert any proprietary rights therein or to represent this program to any one as other than a Government program.

CONTENTS

	Page
INTRODUCTION	1
CAPABILITIES	1
GEOMETRIC MODEL	1
ELEMENT LIBRARY	1
Truss Element	1
Beam Element	1
Plane Stress Membrane Elements	2
Two-Dimensional Finite Elements	2
Three-Dimensional Solid Elements	2
Plate and Shell Elements	2
Boundary Element	2
General Three-Dimensional Isoparametric Elements	3
Three-Dimensional Pipe Elements	3
STATIC ANALYSIS	3
DYNAMIC ANALYSIS	3
SOLUTION METHOD	4
PROBLEM DESCRIPTION DATA PREPARATION INSTRUCTIONS	5
HEADING LINE	5
MASTER CONTROL LINE	5
NODAL POINT DATA	8
THREE-DIMENSIONAL TRUSS ELEMENTS, ELEMENT TYPE 1	12
THREE-DIMENSIONAL BEAM ELEMENTS, ELEMENT TYPE 2	15
PLANE STRESS MEMBRANE ELEMENTS, ELEMENT TYPE 3	21
TWO-DIMENSIONAL FINITE ELEMENTS, ELEMENT TYPE 4	26
THREE-DIMENSIONAL SOLID ELEMENTS, ELEMENT TYPE 5	35
PLATE AND SHELL QUADRILATERAL ELEMENT, ELEMENT TYPE 6	44
BOUNDARY ELEMENTS, ELEMENT TYPE 7	49
THICK SHELL AND THREE-DIMENSIONAL ELEMENTS, ELEMENT TYPE 8	53
THREE-DIMENSIONAL STRAIGHT OR CURVED PIPE ELEMENTS, ELEMENT TYPE 12	71
CONCENTRATED LOAD/MASS DATA	82
ELEMENT LOAD MULTIPLIERS	84
DYNAMIC ANALYSES	85
MODE SHAPES AND FREQUENCIES	86
RESPONSE HISTORY ANALYSIS	90
RESPONSE SPECTRUM ANALYSIS	98

	Page
EXECUTION INSTRUCTIONS	100
MS DOS COMPUTERS	100
OTHER COMPUTERS	100
DATA FILES CREATED DURING EXECUTION	101
DYNAMIC ANALYSIS RESTART INSTRUCTIONS	102
RESTART INSTRUCTIONS FOR OBTAINING ADDITIONAL EIGENVALUES AND EIGENVECTORS	104
PRE- AND POST-PROCESSING	105
PRE-PROCESSING	105
Manual Preparation	105
Automatic Generators	105
POST-PROCESSING	106
REFERENCES	107
SAP IV - VERSION 1.0 FEEDBACK REPORT	108
APPENDIX A - OUTPUT STRESS COMPONENTS	A-1

INTRODUCTION

SAP IV is a general purpose finite element analysis program for linear structural analysis. The program can analyze structures that are subject to either static or dynamic loads. The structures can be described in terms of truss, frame, plate, shell, or brick elements or combinations of elements. The static loads include concentrated loading applied to discrete points on the structural models; and loads, concentrated or uniform, applied to individual elements in the model. The loads can be combined into load cases. The dynamic analysis features include modal analysis to obtain mode shapes and frequencies, response history analysis using either modal superposition or direct integration techniques, and response spectrum analysis.

CAPABILITIES

GEOMETRIC MODEL

The structure to be analyzed is described as a mathematical model of three dimensional coordinates. These coordinates are numerically identified as nodes. Each node holds a discrete location in three dimensional space. The node can be free to move with translational or rotational displacement, or it can be restrained from movement. The coordinate system can be either a cartesian or cylindrical systems. A limited node generation feature is included.

ELEMENT LIBRARY

There are nine element types that can be used separately or combined to complete the mathematical description of the structure to be analyzed. Each element type is used to define the stiffness properties of the structure. The element types include:

Truss Element

The truss element is a three dimensional element. Axial forces and stress are calculated for each member. A uniform temperature change and inertia loads in three directions can be considered as the basic element load conditions.

Beam Element

The beam element is a three dimensional element. Forces (axial and shear) and moments (bending and torsion) are calculated with respect to the local beam coordinate system for each beam element. The beams are assumed

to be symmetrical about and loaded in the plans of their major and minor axes. Provision for unsymmetrical beams are not currently in the program, however, this feature has been developed. The effect of shear deformation and member end releases can be included in the analysis. The element loads include concentrated loads (no moments) due to gravity, and fixed-end forces. Member uniform and concentrated loads and thermal loads are specified using fixed-end forces (including moments).

Plane Stress Membrane Elements

Quadrilateral and triangular elements can be used for plane stress membrane elements of specified thickness which are oriented in an arbitrary plane. These elements have temperature dependent orthotropic material properties. Incompatible displacement modes can be included to improve the bending properties of the element. Gravity, inertia, pressure, and temperature loads are included.

Two-Dimensional Finite Elements

Quadrilateral and triangular elements can be used as: 1) axisymmetric solid elements symmetrical about the z-axis, 2) plane strain elements of unit thickness in the Y-Z plane, and 3) plane stress elements of specified thickness in the Y-Z plane. All of these elements have temperature dependent orthotropic material properties. Incompatible displacement modes can be included to improve the bending properties of the element. Gravity inertia, pressure, and temperature loads are included.

Three-Dimensional Solid Elements

General three-dimensional, eight-node, isoparametric elements with three translational degrees of freedom per node can be used. The elements use isotropic material properties. The element load cases include surface pressure, hydrostatic loads, inertia loads, and thermal loads. The six components of stress and the three principal stresses are computed at the center of each element. Surface stresses are also evaluated. No incompatible displacement modes are assumed in the formation of the element stiffness.

Plate and Shell Elements

A quadrilateral element is used to describe the plate or shell element. Anisotropic as well as isotropic material properties are possible. The element load cases include surface pressure, inertia loads, and thermal loads. For the analysis of shallow shell, rotational constraints normal to the surface may be imposed by using boundary elements at the nodes. The results include moments per unit length and membrane stresses.

Boundary Element

This element is used to constrain nodal displacements to specified values, to compute support reactions, and to provide linear elastic supports to nodes. The boundary element is essentially a spring which has a translational and rotational stiffnesses.

General Three-Dimensional Isoparametric Elements

A minimum of 8 and a maximum of 21 nodes are used to describe a general three dimensional isoparametric element. The element is used to represent orthotropic elastic media. The use of nodes 9 through 21 is optional. Element loads include: surface pressure, hydrostatic loads, inertia loads, and thermal loads. Six global stresses are computed at up to 7 user selected locations within the element.

Three-Dimensional Pipe Elements

Straight or curved cylindrical pipe elements may be used to describe pipe networks. Axial and shear forces, torque and bending moments are calculated for each member. Gravity loadings in the global (X, Y, Z) directions, uniform temperature changes, and extensional effects due to internal pressure form the basic member loading conditions.

STATIC ANALYSIS

The static analysis features include analysis for nodal, element, gravity, and temperature loads. The element loads have been described with the element library where each element type has particular element loads that can be considered. The nodal loads can be defined for each of the nodes. These loads include concentrated forces and moments for each of the three degrees of freedom. The nodal loads can be assigned to a load case. The load cases are completed by assigning the appropriate element load sets, of which there are four. The element loads are assigned to the load case using a multiplier factor. In this manner any number of loading conditions can be studied separately or combined.

DYNAMIC ANALYSIS

There are four types of dynamic analysis options available. The modal analysis option computes the mode shapes and associated modal frequencies. Response history analysis or time history analysis can be accomplished using two methods. The first uses the modal superposition technique which includes computing the mode shapes and frequencies. The second uses the direct integration method. The last dynamic analysis option is response spectrum analysis which also uses the mode shapes and frequencies to compute the R.M.S (root-mean-square) stresses and deflections due to an input displacement of acceleration spectrum.

SOLUTION METHOD

The numerical methods used in the SAP IV program are well documented in many text books dealing with the finite element method. The primary reference for this program is a report that is available through the Earthquake Engineering Research Center, College of Engineering, University of California, Berkeley, California (Reference 1). The original version of this computer program was also obtained from the Earthquake Engineering Research Center. Additional information on the techniques used in the program can be obtained from Reference 2. Discussion on the formulation of the element stiffness equations similar to those used in the program can be obtained in Reference 3.

PROBLEM DESCRIPTION DATA PREPARATION INSTRUCTIONS

HEADING LINE (A72)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(1)	1 - 72	HED	Enter the heading information to be printed with the output

NOTES:

(1) Begin each new data set with a new heading line. Several problems can be run at the same time by adding each problem data set one after another. To stop the program without an error requires two blank lines at the end of the last data set.

MASTER CONTROL LINE (815)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(1)	1 - 5	NUMNP	Total number of nodal points (joints) in the model
(2)	6 - 10	NELTYP	Number of element groups
(3)	11 - 15	LL	Number of structure load cases; GE. 1; static analysis EQ. 0; dynamic analysis
(4)	16 - 20	NF	Number of frequencies to be found in the eigenvalue solution; EQ. 0; Static analysis GE. 1; dynamic analysis
(5)	21 - 25	NDYN	Analysis type code: EQ. 0; static analysis EQ. 1; Eigenvalue/vector solution EQ. 2; forced dynamic response by mode superposition EQ. 3; response spectrum analysis EQ. 4; direct step-by-step integration
(6)	26 - 30	MODEX	Program execution mode: EQ. 0; problem solution EQ. 1; data check only
(7)	31 - 35	NAD	Total number of vectors to be used in a SUBSPACE ITERATION solution for eigenvalues/vectors: EQ. 0; default set to: MIN [2* NF, NF+41]

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(8)	36 - 40	KEQB	Number of degrees of freedom (equations) per block of storage: EQ. 0; calculated automatically by the program
(9)	40 - 45	N1OSV	Element stress porthole flag EQ. 0; no element stresses saved GT. 0; Save element stresses on TAPE10

NOTES:

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

(2) For each different element type (TRUSS, BEAM, etc) a new element group needs to be defined. Elements within groups are assigned integer labels ranging from "1" to the total number of elements in the group. Element groups are defined in the section dealing with elements, below. Element numbering must begin with one in each different group. It is possible to use more than one group for an element type. For example, all columns (vertical beams) of a building may be considered one group and the girders (horizontal beams) may be considered another group.

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

(4) For a static analysis, NF equals 0. If NDYN equals 1, 2, or 3, the lowest NF eigenvalues are determined by the program. Note that a dynamic solution may be restarted after eigenvalue extraction (providing a previous eigenvalue solution for the model was saved as described in Dynamic Analysis Restart Instructions section). NF for the original and restart runs must be the same.

(5) If NDYN equals 2 or NDYN equals 3 the program first solves for NF eigenvalues/vectors and then performs the forced response solution or the response spectrum analysis). Thus, the program expects to read the control line governing the eigensolution (mode shapes and frequencies) before reading data in either the Response History or Response Spectrum Analysis section. For the case NDYN equals 1, the program solves for NF eigenvalues/vectors, prints the results and proceeds to the next data case. The results for the eigenvalue solution phase (NDYN equals 1) may be saved for later use in automatic restart (refer to the two restart sections for the procedure required to affect this save operation for a subsequent restart), i.e., a

dynamic solution may be restarted without repairing the solution for modes and frequencies. If this data case is a restart job, set NDYN equals -2 for forced response solution, or set NDYN equals -3 for a response spectrum analysis. Note that the solution may be restarted a multiple of times (to run different ground spectra or different time-dependent forcing functions) because the program does not destroy the contents of the restart file.

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

(6) In the data-check-only mode (MODEX equals 1), the program writes only one file, "TAPE8", and this file may be saved for use as input to special purpose programs such as mesh plotters, etc. TAPE8 contains all data input in its completely generated form. If MODEX equals 1, most of the expensive calculations required during normal (MODEX equals 0) execution are passed. TAPE8, however, is also written during normal problem solution.

Note that a negative value for NDYN ("-2" or "-3"), when executing in the data-check-only mode, does not cause the program to read the restart file which contains the eigensolution information; instead, the program jumps directly from this line to the Response History or Response Spectrum section and continues reading and checking data lines without performing the solution.

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

(8) KEQB is a program testing parameter that allows the user to test multiple equation block solutions using small data cases that would otherwise be a one block problem. KEQB is also used to reduce the I/O files in the solution procedure. During the solution complete equation blocks (i.e., bandwidth times the number of equation in a block) are written and read from slow speed storage. If there are limitations on the size that can be transferred then KEQB can be used to reduce the block size. KEQB is normally left blank.

(9) NIOSV is a flag to determine whether element stresses are to be written in binary form onto TAPE10. This porthole file can then be used with a post processor.

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

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

NOTES:

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

$$\begin{aligned} X &= R \sin \theta \\ Y &= Y \\ Z &= R \cos \theta \end{aligned}$$

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

(2) Nodal point data must be defined for all (NUMNP) nodes. Node data may be input directly (i.e., each node on its own individual line), may be entered, or the generation option may be used if applicable (see note 5, below).

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

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

- IX(N,M) = 0; unspecified (free) displacement (or rotation) component
- IX(N,M) = 1; deleted (fixed) displacement (or rotation) component
- IX(N,M) = K; node number "K" ($1 < K \leq \text{NUMNP}$ and $K \neq N$) is the "master" node to which the Mth degree of freedom at node "N" is a "slave"

An unspecified (IX(N,M) = 0) degree of freedom is free to translate or rotate as the solution dictates. Concentrated forces (or moments) may be applied (see instructions pertaining to concentrated load/mass data) in this degree of freedom. One system equilibrium equation is required for each unspecified degree of freedom in the model. The maximum number of equilibrium equations is always less than six times the total number of nodes in the model.

Deleted (IX(N,M) = 1) degrees of freedom are removed from the final set of equilibrium equations. Deleted degrees of freedom are fixed (points of reaction), and loads applied to these degrees of freedom are ignored by the program. Nodes that are used for geometric reference only (i.e., nodes not assigned to any element) must have all six degrees of freedom deleted. Nodal degrees of freedom having undefined stiffness (such as rotations in an all TRUSS model, out-of-plane components in a two-dimensional planar model, etc.) should be deleted. Deletions have the beneficial effect of reducing the size of the set of equations that must be solved. The table below lists the types of degrees of freedom that are defined by each different element type. The table was prepared assuming that the element has general orientation in (X,Y,Z) space.

DEGREES OF FREEDOM WITH DEFINED STIFFNESS

ELEMENT TYPE	δX	δY	δZ	$\delta \theta_X$	$\delta \theta_Y$	$\delta \theta_Z$
1. TRUSS	x	x	x			
2. BEAM	x	x	x	x	x	x
3. MEMBRANE	x	x	x			
4. 2D/QUADRILATERAL		x	x			
5. 3D/BRICK	x	x	x			
6. PLATE/SHELL	x	x	x	x	x	x
7. BOUNDARY	x	x	x	x	x	x
8. THICK SHELL	x	x	x			
9. 3D/PIPE	x	x	x	x	x	x

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

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

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

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

(4) When CT (Column 1) is equal to the character "C", the values input in Columns 36-65 are interpreted as the cylindrical (R,Y, θ) coordinates of node "N". Y is the axis of symmetry. R is the distance of a point from the Y-axis. The angle θ is measured clockwise from the positive Z-axis when looking in the positive Y direction. The cylindrical coordinate values are printed as entered on the line, but immediately after printing the global cartesian values are computed from the input entries. Note that boundary condition codes always refer to the (X,Y,Z) system even if the node happens to be located with cylindrical coordinates.

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

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

may be generated from information given on two data lines in sequence:

LINE 1 $\{N_1, IX(N_1,1), \dots, IX(N_1,6), X(N_1), \dots, KN_1, T(N_1)\}$

LINE 2 $\{N_2, IX(N_2,1), \dots, IX(N_2,6), X(N_2), \dots, KN_2, T(N_2)\}$

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

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

THREE-DIMENSIONAL TRUSS ELEMENTS, ELEMENT TYPE 1

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

A. Control Line (3I5)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 - 5		The number 1
	6 - 10		Total number of truss elements
	11 - 15		Number of material property lines

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

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

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

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

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

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

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 - 10		Element load case A
	11 - 20		Element load case B
	21 - 30		Element load case C
	31 - 40		Element load case D

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

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

Line 4: As above to indicate the fraction of the thermal load to be added to each of the element load cases.

D. Truss Element Data Lines (4I5,F10.0,I5)

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

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 - 5		Element number
	6 - 10		Node number I
	11 - 15		Node number J
	16 - 20		Material property number
(2)	21 - 30		Reference temperature for zero stress
(1)	31 - 35		Optional parameter k used for automatic generation of element data.

NOTES:

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

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

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

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

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

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

Where $(T_i + T_j)/2.0$ - is the average of the nodal temperatures specified on the nodal point data lines for nodes i and j; and T_r is the zero stress reference temperature specified on the element line. For truss elements it is generally more convenient to set $T_i = T_j = 0.0$ such that $\Delta T = -T_r$ (note the minus sign).

Other types of member loadings can be specified using an equivalent ΔT :

(a) If a truss member has an initial lack of fit by an amount d (positive if too long) then $\Delta T = d/(\alpha L)$.

(b) If an initial prestress force P (positive if tensile) is applied to the member ends that is released after the member is connected to the rest of the structure then $\Delta T = -P/(\alpha AE)$.

In the above formulas A = cross section area, L = member length and α = coefficient of thermal expansion.

THREE-DIMENSIONAL BEAM ELEMENTS, ELEMENT TYPE 2

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

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

A. Control Line (515)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 - 5		The number 2
	6 - 10		Total number of beam elements
	11 - 15		Number of element property lines
	16 - 20		Number of fixed end force sets
	21 - 25		Number of material property lines

B. Material Property Lines (15,3F10.0)

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

C. Element Property Lines (15,6F10.0)

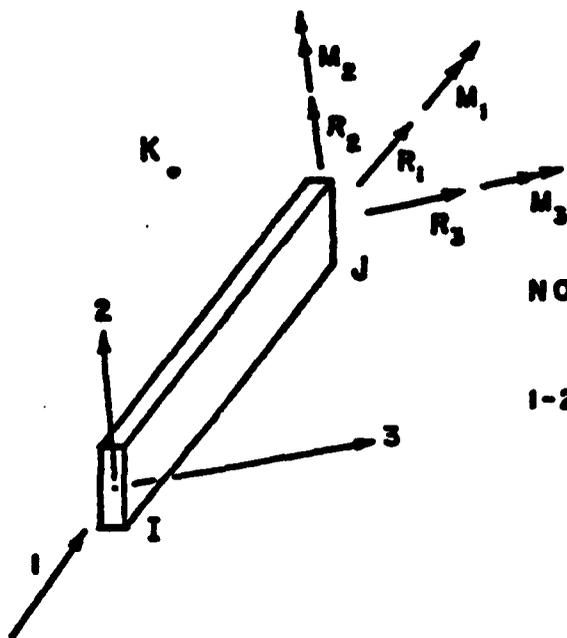
<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 - 5		Geometric property number
	6 - 15		Axial area
	16 - 25		Shear area associated with shear forces in local 2-direction

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	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

(1) One line is required for each unique set of properties. Shear areas need be specified only on short beams where shear deformations are on the same order of magnitude as the bending deformations and are to be included in the analysis.

(2) Section property computations should be done carefully.

- (a) The axial area is the total cross section area.
- (b) The shear area represent the beam cross sectional area effective in shear. This is used to account for shear deformations due to shear strains. These deformations are usually neglected, however, they may be important in deep beams. The shear area is defined as $A_s = K_s * A$ where K_s is the shear coefficient, a factor depending upon the shape of the cross section. The factor is always less than one. Reference 4 provides more information on this subject and it includes formulas for the shear coefficient for various common cross sections.
- (c) The torsional inertia is dependent upon the form and shape of cross section. For a circular cross section the torsional inertia is the polar moment of inertia; for other sections it is less than the polar moment of inertia, often much smaller. Common values for the torsional inertia have been tabulated in Reference 5.
- (d) The flexural inertia is the appropriate moment of inertia for the major or minor axes depending upon the orientation of the beam.



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

LOCAL COORDINATE SYSTEM FOR BEAM ELEMENT

D. Element Load Factors (4F10.0)

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

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

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 - 10		Element load case A
	11 - 20		Element load case B
	21 - 30		Element load case C
	31 - 40		Element load case D

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

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

E. Fixed-end Forces (15,6F10.0/15,6F10.0)

Two lines are required for each unique set of fixed-end forces occurring in the analysis. Distributed load, concentrated loads located between member ends, and thermal loads can be specified using the fixed-end forces.

Line 1:

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

Line 2:

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	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

(1) 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.

(2) The element loads for beams, usually referred to as member loads are input as fixed end forces and moments. These are true fixed end quantities without end condition corrections. Formulas for these fixed end quantities can be found in most structural engineering handbooks and textbooks.

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

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 - 5		Element number
	6 - 10		Node number I
	11 - 15		Node number J
	16 - 20		Node number K - see accompanying figure
	21 - 25		Material property number
	26 - 30		Section property number
	31 - 35	A } B } C } D }	Fixed-end force identification for element load cases A, B, C, and D respectively
	36 - 40		
	41 - 45		
	46 - 50		
(1)	51 - 56		End release code at node I
(1)	57 - 62		End release code at node J
(2)	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.

NOTES:

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

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

(2) 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 line for the first element in the series need be given as input, provided.

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

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

and the

- (b) material property number
- (c) element property number
- (d) fixed-end force identification numbers for each element load case
- (e) element release code
- (f) orientation of local 2-axis are the same for each element in the series.

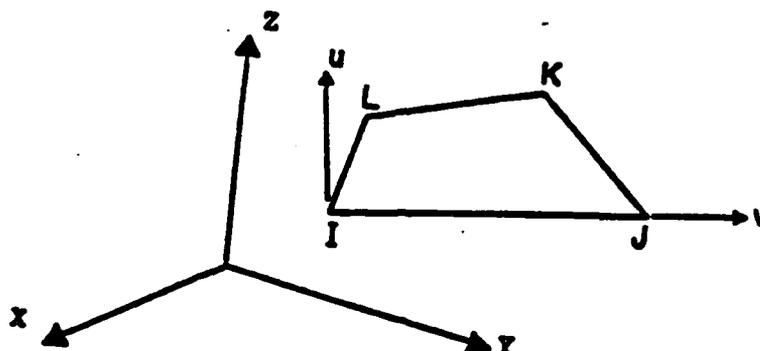
The value of k , if left blank, is taken to be one. The element data line for the last beam element must always be given.

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

PLANE STRESS MEMBRANE ELEMENTS, ELEMENT TYPE 3

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

A general quadrilateral element is shown below:



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

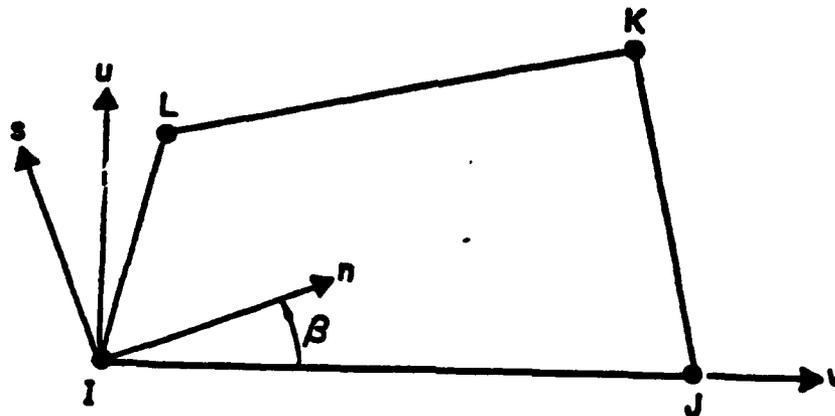
A. Control Line (6I5)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 - 5		The number 3
	6 - 10		Total number of plane stress elements
	11 - 15		Number of material property lines
	16 - 20		Maximum number of temperature points for any one material; see Section B below.
(5)	30		Nonzero numerical value will suppress the introduction of incompatible displacement modes.

Material Property Information: Orthotropic, temperature-dependent material properties are possible. For each different material, the following group of lines must be supplied.

B. Material Property Identification Line (2I5,3F10.0)

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



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

C. Two Temperature Data Lines

Line 1: (8F10.0)

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

Line 2: (3F10.0)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 - 10		Coefficient of thermal expansion - α_n
	11 - 20		Coefficient of thermal expansion - α_s
	21 - 30		Coefficient of thermal expansion - α_t

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

D. Element Load Factors (5F10.0)

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

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

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 - 10		Fraction of thermal load
	1 - 20		Fraction of pressure load
	21 - 30		Fraction of gravity in X-direction
	31 - 40		Fraction of gravity in Y-direction
	41 - 50		Fraction of gravity in Z-direction

E. Element Data Lines (6I5,2F10.0,2I5,F10.0)

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

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 - 5		Element number
	6 - 10		Node I
	11 - 15		Node J
	16 - 20		Node K
(4)	21 - 25		Node L (Node L must equal Node K for triangular element)
	26 - 30		Material identification number
(3)	31 - 40		Reference temperature for zero stresses within element
	41 - 50		Normal pressure on I-J side of element
(2)	51 - 55		Stress evaluation option "n"
(1)	56 - 60		Element data generator "k"
	61 - 70		Element thickness

NOTES:

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

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

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

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

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

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

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

(3) Thermal Data - Nodal temperatures as specified nodal point data lines are used by element type 3 and 4 in the following two ways (see element type 4 for more information):

(a) Temperature-dependent material properties are approximated by interpolating (or extrapolating) the input material properties at the temperature T_0 corresponding to the origin of the local s-t coordinate system. The material properties throughout the element are assumed constant corresponding to this temperature.

(b) For computation of nodal loads due to thermal strains in the element a bilinear interpolation expansion for the temperature change $\Delta T(s,t)$ is used.

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

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

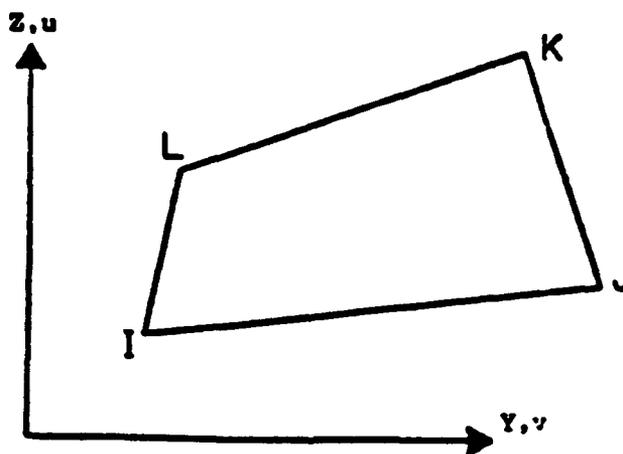
TWO-DIMENSIONAL FINITE ELEMENTS, ELEMENT TYPE 4

Quadrilateral (and triangular) elements can be used as:

- Axisymmetric solid elements symmetrical about the Z-axis. The radial direction is specified as the Y-axis. Care must be exercised in combining this element with other types of elements
- Plane strain elements of unit thickness in the Y-Z plane.
- Plane stress elements of specified thickness in the Y-Z plane.

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

A general quadrilateral element is shown below:



A. Control Line (6I5)

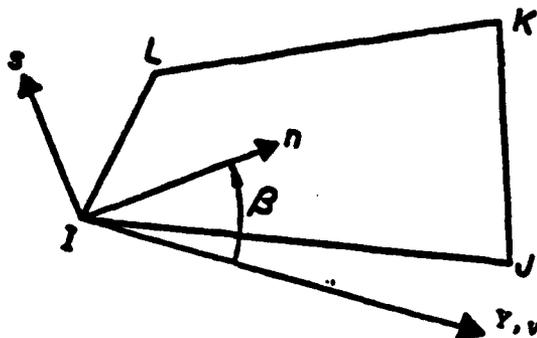
<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 - 5		The number 4
	6 - 10		Total number of elements
	11 - 15		Number of different materials
	16 - 20		Maximum number of temperature lines for any one material - see Section C below.

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	25		<ul style="list-style-type: none"> 0 for axisymmetric analysis 1 for plane strain analysis 2 for plane stress analysis
(5)	30		Nonzero numerical value will suppress the introduction of incompatible displacement modes. Incompatible modes cannot be used for triangular elements and are automatically suppressed.

Material Property Information: Orthotropic, temperature-dependent material properties are possible. For each different material the following group of lines must be supplied.

B. Material Property Identification Line (2I5,3F10.0)

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



PRINCIPAL MATERIAL AXES

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

C. Two Temperature Data Lines:

Line 1: (8F10.0)

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

Line 2: (3F10.0)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 - 10		Coefficient of thermal expansion - α_n
	11 - 20		Coefficient of thermal expansion - α_s
	21 - 30		Coefficient of thermal expansion - α_t

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

D. Element Load Factors (5F10.0)

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

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

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 - 10		Fraction of thermal load
	11 - 20		Fraction of pressure load
	21 - 30		Fraction of gravity in X-direction
	31 - 40		Fraction of gravity in Y-direction
	41 - 50		Fraction of gravity in Z-direction

E. Element Data Lines (6I5,2F10.0,2I5,F10.0)

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

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 - 5		Element number
	6 - 10		Node I
	11 - 15		Node J
	16 - 20		Node K
(4)	21 - 25		Node L (Node L must equal Node K for triangular elements)
	26 - 30		Material identification number
(3)	31 - 40		Reference temperature for zero stresses within element
	41 - 50		Normal pressure on I-J side of element
(2)	51 - 55		Stress evaluation option "n"
(1)	56 - 60		Element data generator "k"
	61 - 70		Element thickness (For plane strain set equal to 1.0 by program)

NOTES:

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

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

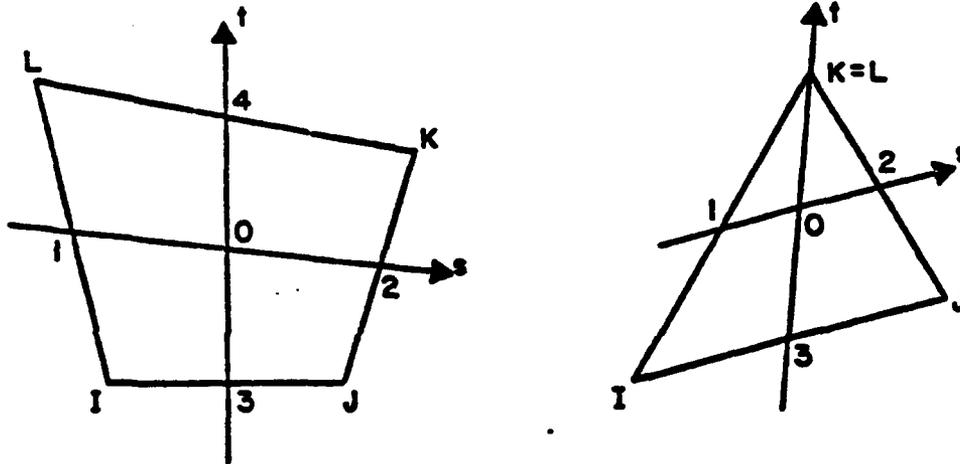
$$J_n = J_{n-1} + k$$

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

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

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

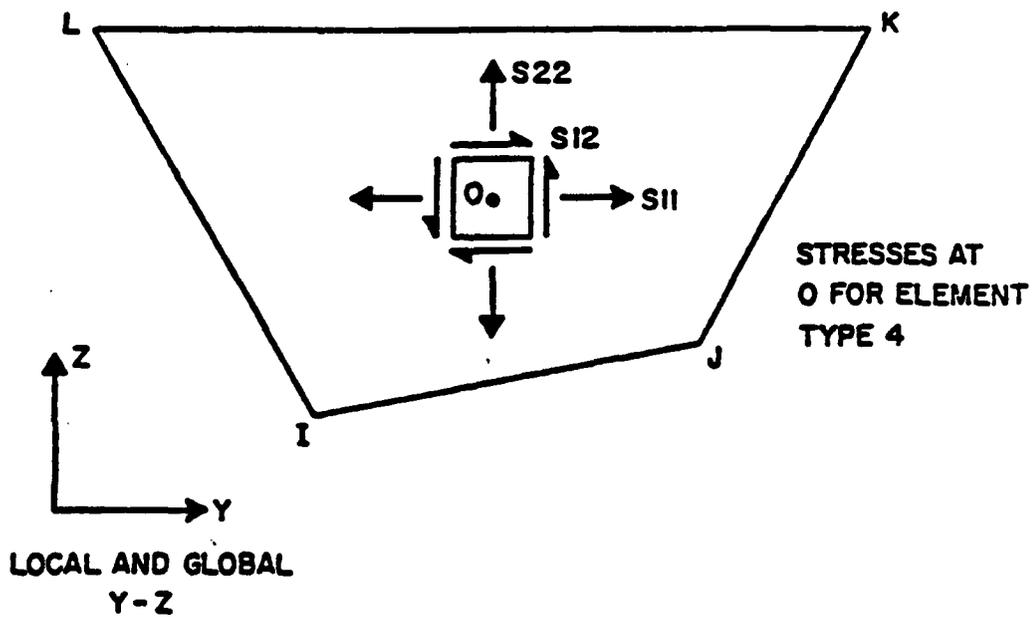
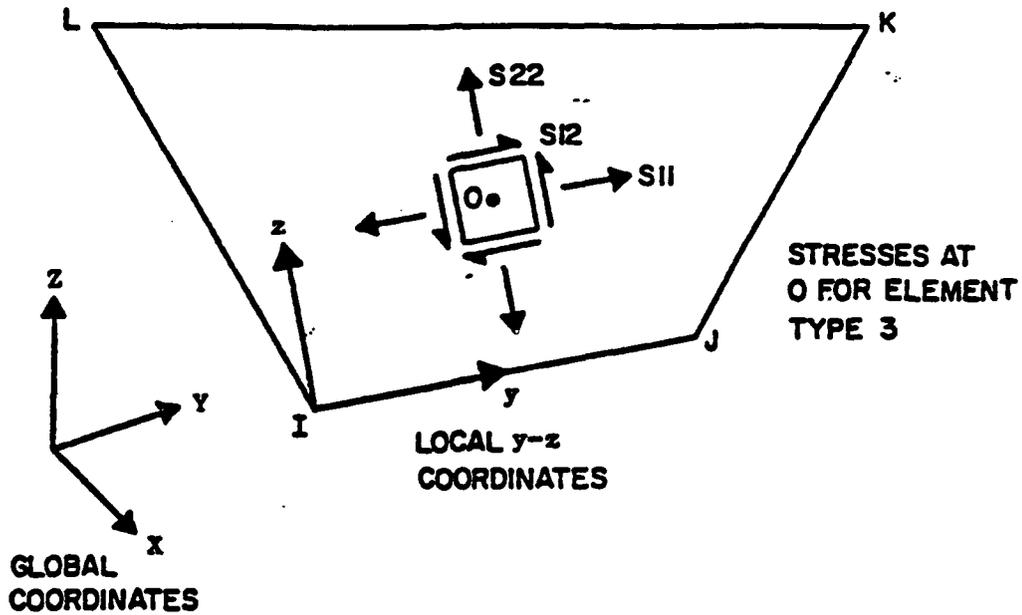
(2) Stress Print Option - The following description of the stress print option applies to both element types 3 and 4. The value of the stress print option "n" can be given as 1, 0, 8, 16, or 20.



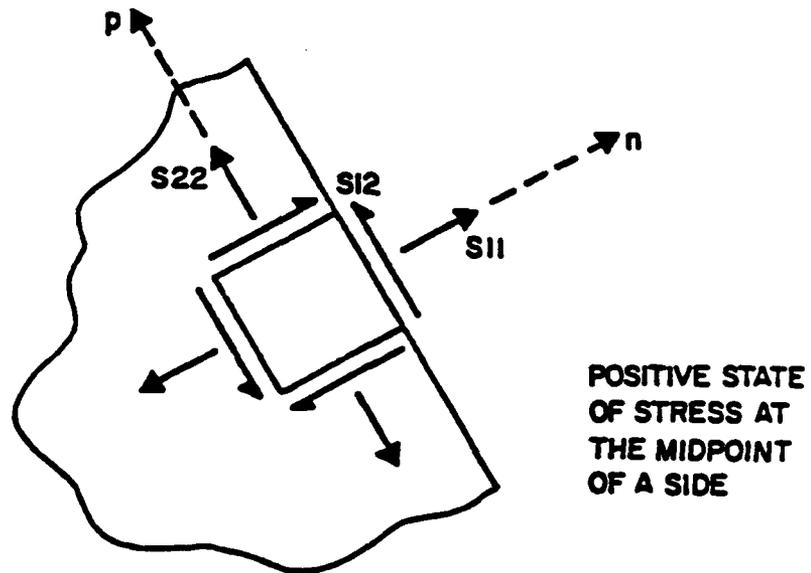
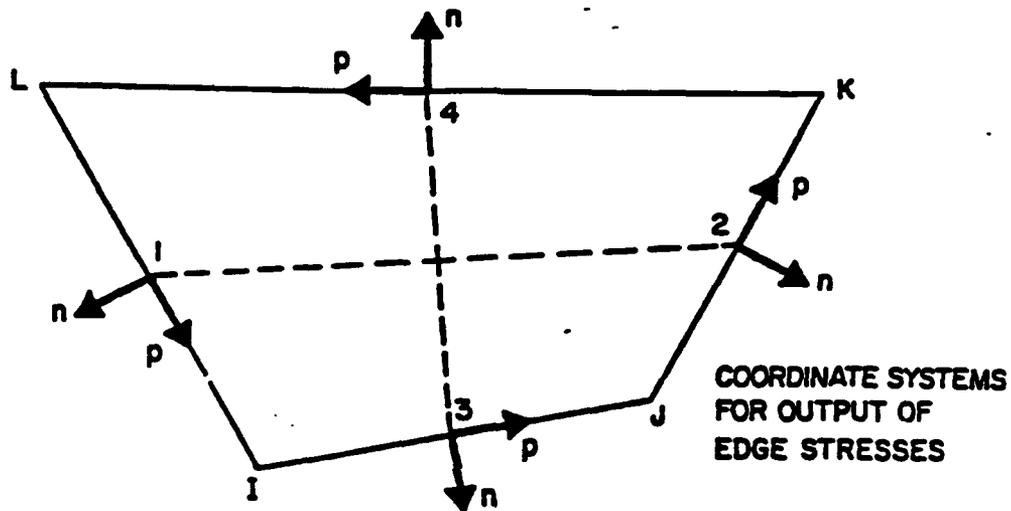
0 = origin of natural s - t coordinates (Fig. 5-2). Points 1, 2, 3, and 4 are midpoints of sides. The points at which stresses are output depend on the value of n as described in the following table.

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

The stresses at 0 are printed in a local y-z coordinate system. For element type 3, side I-J defines the local y-z axes in the plane of the element. For element type 4 the local y-z axes are parallel to the global Y-Z axes.



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

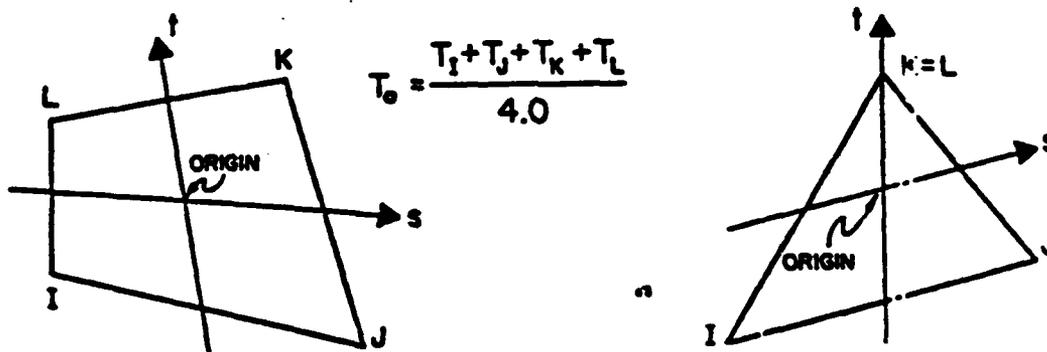


The stresses for an element are output under the following headings: S11, S22, S12, S33, S-MAX, S-MIN, ANGLE. The normal stresses S11 and S22 and the shear stress S12 are as described above. S-MAX and S-MIN are the principle stresses in the plane of the element and S33 is the third principal stress acting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

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

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

- (a) Temperature-dependent material properties are approximated by interpolating (or extrapolating) the input material properties at the temperature T_0 corresponding to the origin of the local s-t coordinate system (see Figure below for description of local element coordinates). The material properties throughout the element are assumed constant corresponding to this temperature.



- (b) For computation of nodal loads due to thermal strains in the element a bilinear interpolation expansion for the temperature change $\Delta T (s, t)$ is used.

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

where T_i are the nodal temperatures specified on the joint data lines, T_r is the reference stress free temperature and $h_i (s, t)$ are the interpolation functions.

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

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

THREE-DIMENSIONAL SOLID ELEMENTS, ELEMENT TYPE 5

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

A. Control Line (4I5)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 - 5		The number 5
	6 - 10		Number of 8-node solid elements
	11 - 15		Number of different materials
	16 - 20		Number of element distributed load sets

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

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

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

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 - 5		Load set identification number
(5)	6 - 10		LT (load type) LT = 1 if this line specifies a uniformly distributed load. LT = 2 if this line specifies a hydrostatically varying pressure.
(5)	11 - 20		P If LT = 1, P is the magnitude of the uniformly distributed load If LT = 2, P is the weight density of the fluid causing the hydrostatic pressure
(5)	21 - 30		Y If LT = 1, leave blank If LT = 2, Y is the global Y coordinate of the surface of fluid causing hydrostatic pressure loading
(4) (5)	31 - 35		Element face number on which surface load acts. Face numbers are from 1 to 6 as described in note (5) for uniformly distributed loads and can be only faces 2, 4 or 6 for hydrostatically varying pressures.

D. Acceleration Due to Gravity (F10.2)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 - 10		Acceleration due to gravity (for calculation of mass matrix)

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

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

Line 1:

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(7)	1 - 10		} Pressure load multipliers
	11 - 20		
	21 - 30		
	31 - 40		

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

Line 2:

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(6)	1 - 10		TA
	11 - 20		TB
	21 - 30		TC
	31 - 40		TD
			} Thermal load multipliers

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

Line 3:

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 - 10		GXA
	11 - 20		GXB
	21 - 30		GXC
	31 - 40		GXD
			} Gravity load multipliers for + X global direction

Line 4:

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 - 10		GYA
	11 - 20		GYB
	21 - 30		GYC
	31 - 40		GYD
			} Gravity load Multipliers for + Y global direction

Line 5:

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 - 10		GZA
	11 - 20		GZB
	21 - 30		GZC
	31 - 40		GZD
			} Gravity load multipliers for + Z global direction

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

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

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 - 5		Element number
	6 - 10		Global node point numbers corresponding to element nodes
(3)	11 - 15		
	16 - 20		
	21 - 25		
	26 - 30		
	31 - 35		
	36 - 40		
	41 - 45		
(2)	46 - 50		Integration Order
	51 - 55		Material Number
(1)	56 - 60		Generation Parameter (INC)
	61 - 62	LSA	LSA is the distributed surface load set identification number of the distributed load acting on this element to be assigned to element load case A. LSB, LSC and LSD refer to element load cases B, C, and D, respectively
	63 - 64	LSB	
	65 - 66	LSC	
	67 - 68	LSD	
(8)	69 - 70		Face number for stress output
	71 - 80		Stress-free element temperature

NOTES:

(1) Element Generation

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

- If on first line for the series the integration order is:
 - > 0 Same value is used for succeeding elements.
 - = 0 A new element stiffness is not formed. Element stiffness is assumed to be identical to that of the preceding element.
 - < 0 Absolute value is used for the first element of the series, and the same element stiffness is used for succeeding element.
- If on first line for the series, the distributed load number (for any load case) is:
 - > 0 Same load is applied to succeeding elements.
 - < 0 The load case is applied to this element but not to succeeding elements in the series.

(c) Element line for the last element must be supplied.

(2) Integration Order

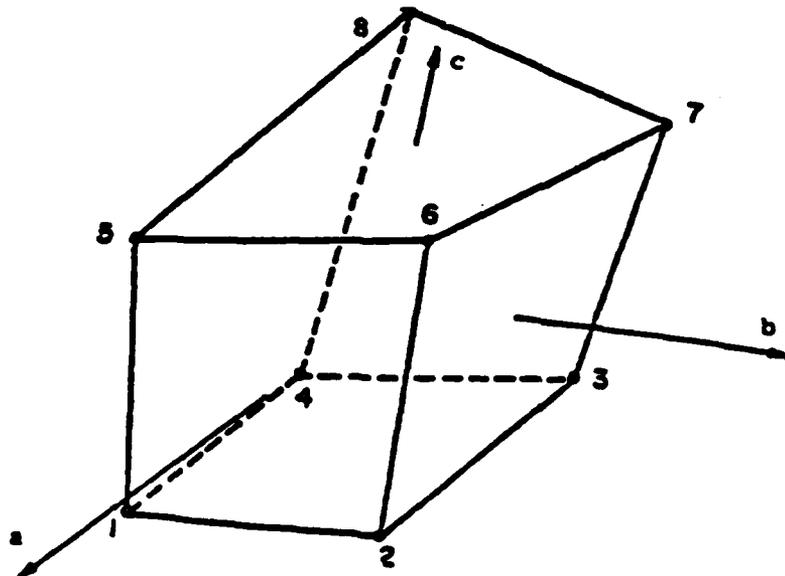
Computation time (for element stiffness) increases with the third power of the integration order. Therefore, the smallest satisfactory order should be used. This is found to be:

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

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

(3) Element coordinate System

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



(4) Identification of Element Faces

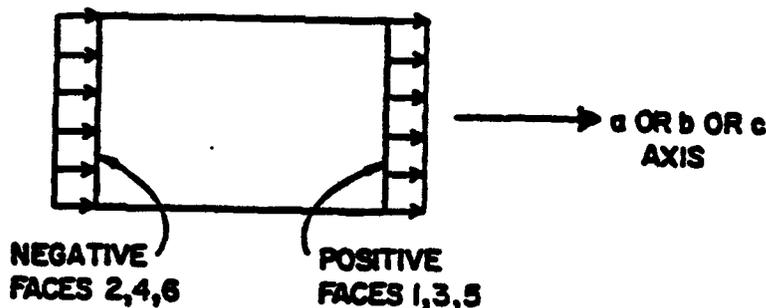
Element faces are numbered as follows:

Face 1 corresponds to + a direction	}	Faces 1,3,5 are positive faces
2 corresponds to - a direction		Faces 2,4,6 are negative faces
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		

(5) Distributed Surface Loads

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

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



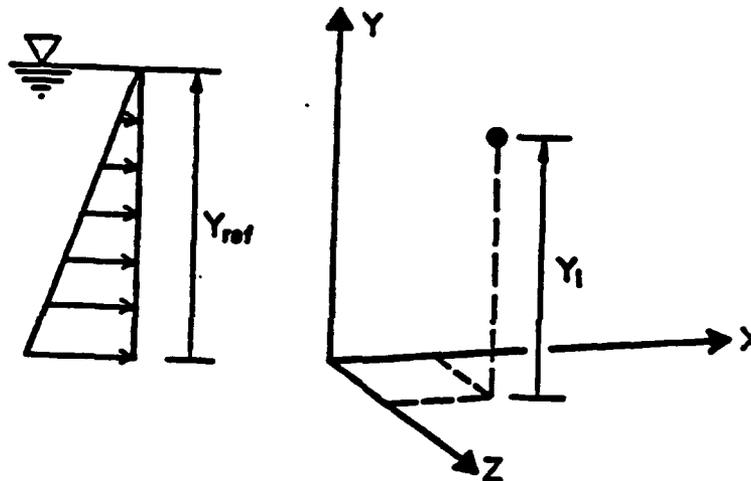
POSITIVE SURFACE LOADING P

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

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

$$P_i = Y (Y_i - Y_{ref})$$

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



If $P_i > 0$, corresponding to surface tension, the contribution is ignored. If an element face is such that $Y_i > Y_{ref}$ for all i (16 integration points are used by program) then no nodal loads will be applied to the element. If some $P_i > 0$ and some $P_i < 0$ for a particular face, then approximate nodal loads are obtained for the partially loaded surface.

(6) Thermal Loads

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

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

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

T_o = stress free element temperature specified on the element line

(7) Element Load Cases

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

$$\begin{aligned} \text{Load case A} = \Sigma & (\text{PA} \times \text{pressure loading} \\ & + \text{TA} \times \text{thermal loading} \\ & + \text{GXA} \times \text{gravity X loading} \\ & + \text{GYA} \times \text{gravity Y loading} \\ & + \text{GZA} \times \text{gravity Z loading}) \end{aligned}$$

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

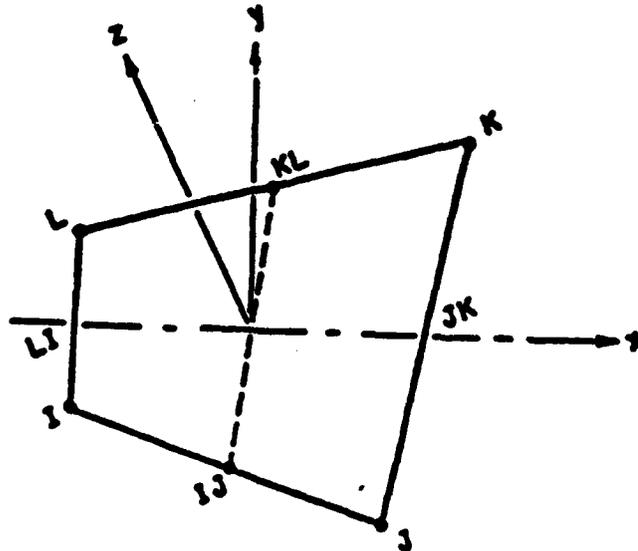
(8) Output of Element Stresses

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

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

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

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



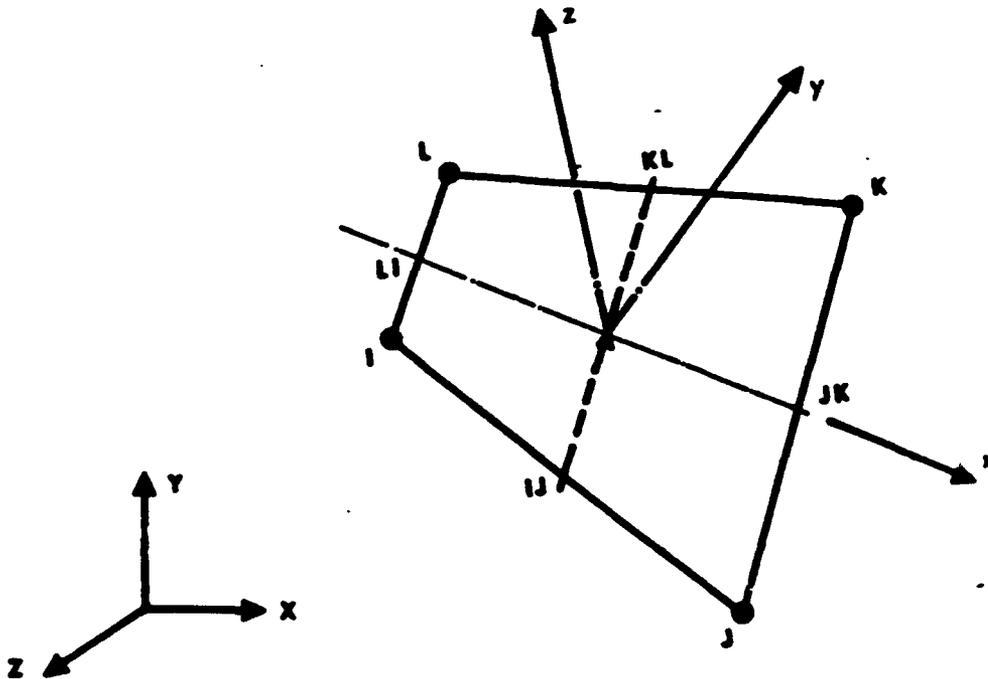
The corresponding nodal points I, J, K and L in each face are given in the table.

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

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

PLATE AND SHELL QUADRILATERAL ELEMENT, ELEMENT TYPE 6

Quadrilateral or triangular shaped thin plate/shell elements are identified as element type "6". For the quadrilateral, four nodes I, J, K, and L may be located arbitrarily in the global (X,Y,Z) system. A local element coordinate system (x,y,z) is defined as follows:



LOCAL COORDINATE SYSTEM FOR THE 3-DIMENSIONAL THIN PLATE/SHELL ELEMENT

1. The x-axis is specified by the line LI-JK, where LI and JK are the midpoints of sides L-I and J-K, respectively.
2. The z-axis is normal to the x-direction and to the line joining midpoints IJ and KL.
3. The y-axis is normal to both the z and x-directions to complete the remainder of the right-handed system.

This local system is used to express all physical and kinematic shell properties (stresses, strains, material, law, etc.), except that gravity forces act in the global (X,Y,Z) coordinate system.

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

Plate/shell elements are described by the following sequence of data lines.

A. Control Line (3I5)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 - 5		The number 6
	6 - 10		Number of shell elements
	11 - 15		Number of different materials

B. Material Property Information

Anisotropic material properties are permitted. For each different material, two lines must be supplied.

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

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 - 10		Material identification number
	31 - 40		Mass density
	41 - 50		Thermal expansion coefficient α_x
	51 - 60		Thermal expansion coefficient α_y
	61 - 70		Thermal expansion coefficient α_{xy}

Line 2: (6F10.0)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 - 10		Elasticity element C
	11 - 20		Elasticity element C^{xx}
	21 - 30		Elasticity element C^{xy}
	31 - 40		Elasticity element C^{xs}
	41 - 50		Elasticity element C^{yy}
	51 - 60		Elasticity element G^{ys} xy

Elements in plane stress
material matrix [C]

$$\begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \tau_{xs} \end{bmatrix} = \begin{bmatrix} C & C & C \\ C^{xx} & C^{xy} & C^{xs} \\ C^{xy} & C^{yy} & G^{ys} \\ C^{xs} & G^{ys} & xy \end{bmatrix} \begin{bmatrix} \epsilon_{xx} \\ \epsilon_{yy} \\ \gamma_{xy} \end{bmatrix}$$

C. Element Load Multipliers (5 lines)

Line 1: (4F10.0)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	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

Line 2: (4F10.0)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	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

Line 3: (4F10.0)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	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

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

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

D. Element Data Lines (8I5,F10.0)

One line for each element

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(1)	1 - 5		Element number
(2)	6 - 10		Node I
	11 - 15		Node J
	16 - 20		Node K
	21 - 25		Node L
(3)	26 - 30		Node O

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	31 - 35		Material identification (if left blank, taken as one)
(1)	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)

NOTES:

(1) Element Data Generation

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

The increment for element number is one

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

The corresponding increment for nodal number is K_n

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

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

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

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

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

(2) Nodal Points and Coordinate Systems

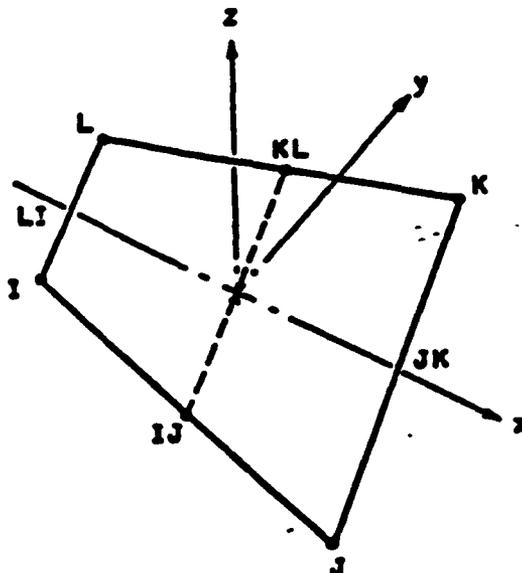
The nodal point numbers I, J, K and L are in sequence in a counter-clockwise direction around the element. The local element coordinate system (x, y, z) is defined as follows:

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

z Normal to x and to the line joining midpoints IJ and KL.

y Normal to x and z to complete the right-handed system.

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



For analysis of flat plates, the stiffness associated with the rotation normal to the plate surface is not defined, therefore, the rotation normal degree of freedom must not be included in the analysis.

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

(3) Node 0

When columns 26 - 30 are left blank, mid-node properties are computed by averaging the four nodes. The thin plate and shell element is a quadrilateral of arbitrary geometry formed from four compatible triangles. The central node is located at the average of the coordinate of the four corner nodes if a node k is not specified.

(4) Element Stress Calculations

Output are moments per unit length and membrane stresses.

ELEMENT TYPE	MAXIMUM NUMBER OF COMPONENTS	STRESS COMPONENT NUMBER	OUTPUT SYMBOL	DESCRIPTION
6	6	(1)	(XX-S/R)	XX-Stress Resultant
		(2)	(YY-S/R)	YY-Stress Resultant
		(3)	(XY-S/R)	XY-Stress Resultant
		(4)	(XX-M/R)	XX-Moment Resultant
		(5)	(YY-M/R)	YY-Moment Resultant
		(6)	(XY-M/R)	XY-Moment Resultant

BOUNDARY ELEMENTS, ELEMENT TYPE 7

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

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

A. Control Line (2I5)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(4)	1 - 5		The number 7.
	6 - 10		Total number of boundary elements.

B. Element Load Multipliers (4F10.0)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	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 Lines (8I5,3F10.0)

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

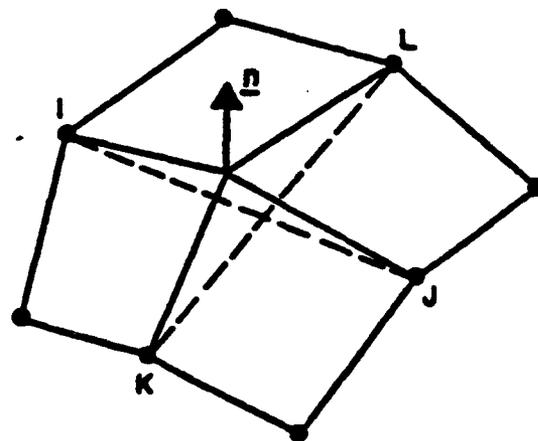
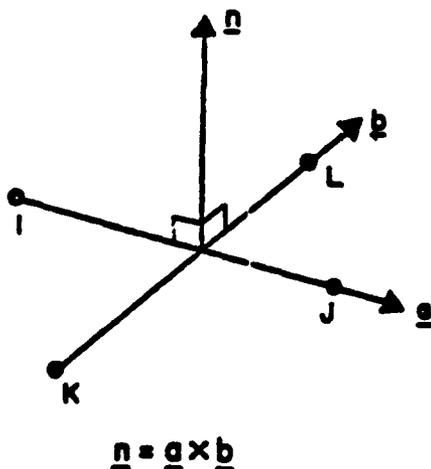
Notes	Columns	Variables	Entry
	1 - 5		Node N, at which the element is placed
(1)	6 - 10		Node I
	11 - 15		Node J
	16 - 20		Node K
	21 - 25		Node L
(2)	26 - 30		Code for displacement
	31 - 35		Code for rotation
(3)	36 - 40		Data generator K
	41 - 50		Specified displacement along element axis
	51 - 60		Specified rotation about element axis
	61 - 70		Spring stiffness (set to 10^{10} if left blank) for both extension and rotation.

NOTES:

- (1) Direction of boundary element

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

- (a) A second nodal point I defines the direction of the element from node N to node I.
- (b) Four nodal points I, J, K and L specify the direction of the element as the normal to the plane defined by two intersecting straight lines (vectors \underline{a} and \underline{b} , see Fig. below). The direction vector \underline{n} is given by the vector cross product $\underline{n} = \underline{a} \times \underline{b}$ and defines the direction of the boundary element.



**ROTATIONAL CONSTRAINT
IN THIN SHELL ANALYSIS**

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

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

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

(2) Displacement and rotation codes

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

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

(3) Data generator K_n

When a series of nodes are such that:

- (a) All have identical boundary elements attached
- (b) All boundary elements have same direction
- (c) All specified displacements and rotations are identical
- (d) The nodal sequence form an arithmetic sequence, i.e., $N, N + K_n, N + 2K_n$ etc.,

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

(4) Element load multipliers

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

(5) Recommendations for use of boundary elements

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

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

In dynamic analysis "artificially stiff" boundary elements should not be used (see note (8) in the Mode Shapes and Frequencies section).

THICK SHELL AND THREE-DIMENSIONAL ELEMENTS, ELEMENT TYPE 8

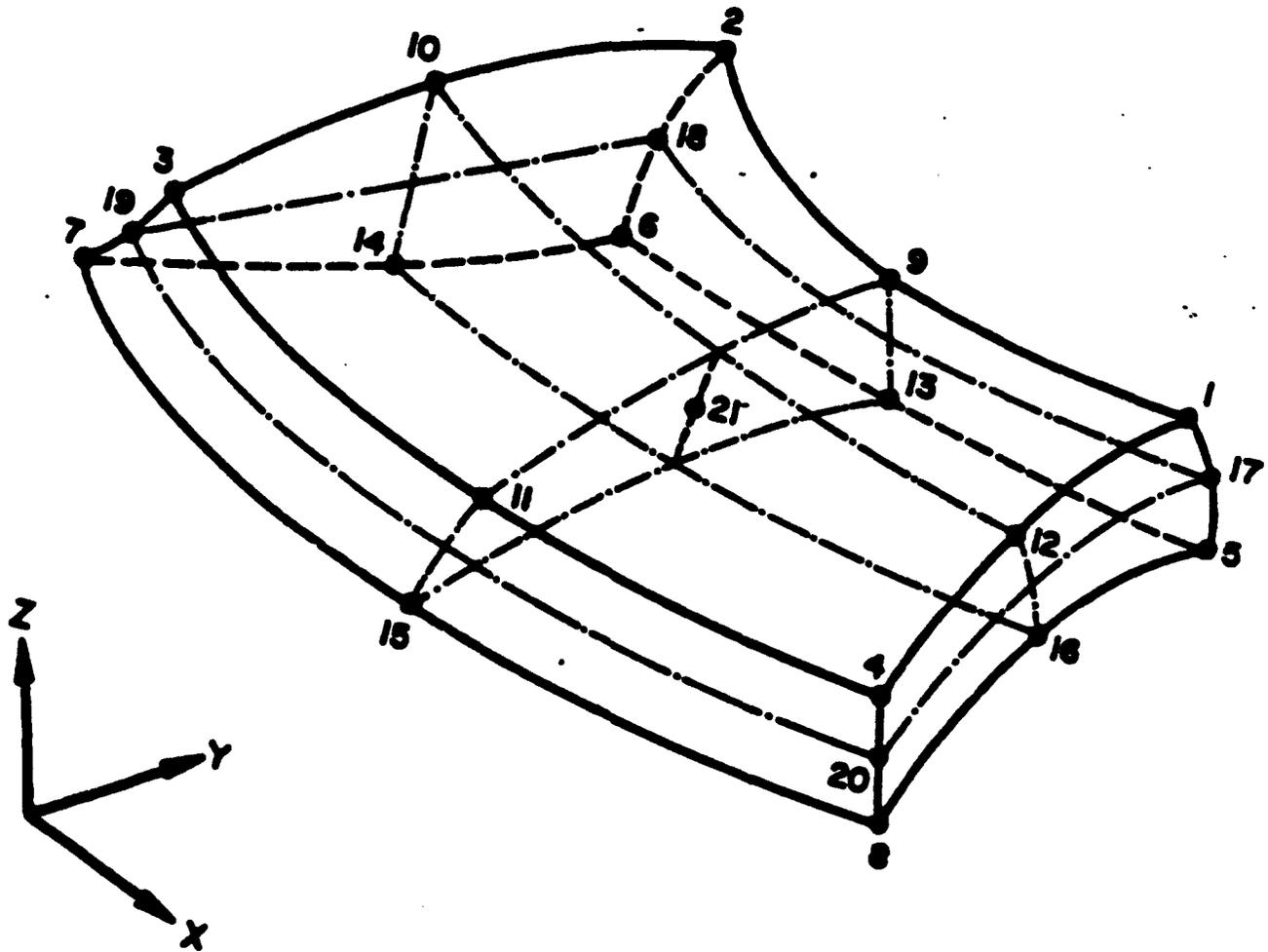
A minimum of 8 and a maximum of 21 nodes are used to describe a general three dimensional isoparametric element; the element is used to represent orthotropic, elastic media. The element type is identified by the number eight (8). Three translational degrees of freedom are assigned to each node, and at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

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

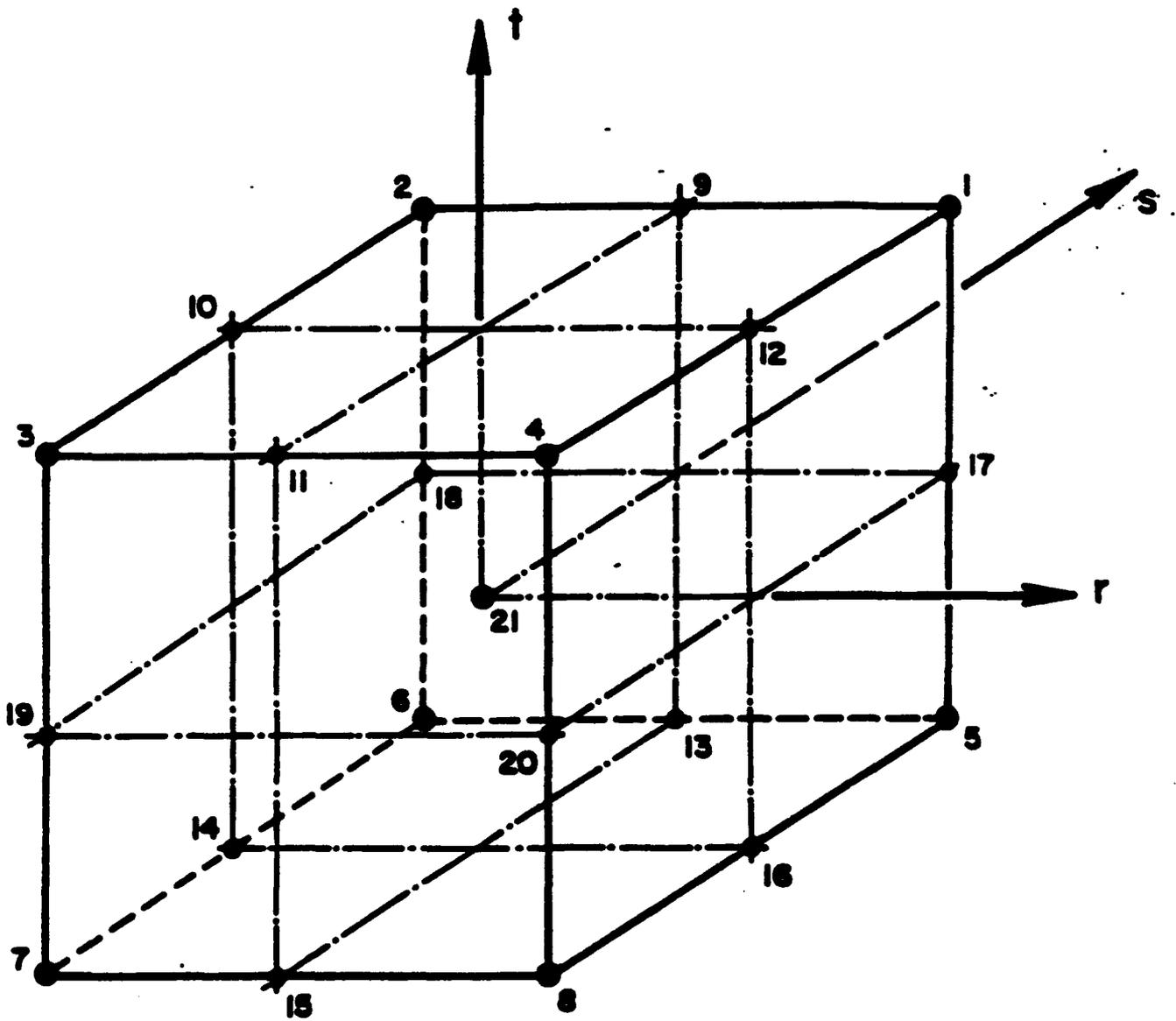
Node temperatures input with the nodal point data are used to form an average element temperature, which is the basis of material property selection for the element. If thermal loads are applied, node temperatures are used to establish the temperature field within the element, and the temperature interpolation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

A. Control Line (10I5)

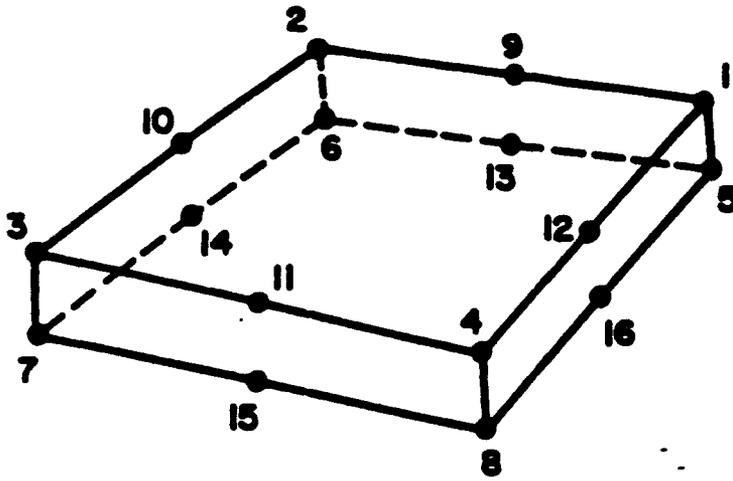
<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 - 5		Enter the number "8"
	6 - 10	NSOL21	Number of solid elements
	11 - 15	NUMMAT	Number of different materials
(1)	16 - 20	MAXTP	Maximum number of temperature points used in the table for any material; EQ. 0 the default is set to "1"
(2)	21 - 25	NORTHO	Number of different sets of martial axis orientation data; EQ. 0; all properties are defined in the X,Y,Z, system
(3)	26 - 30	NDLS	Number of different distributed load (i.e., pressure) sets
(4)	31 - 35	MAXNOD	Maximum number of nodes used to describe any one element; $8 \leq \text{MAXNOD} \leq 21$ EQ. 0 the default is set to "21"
(5)	36 - 40	NOPSET	Number of sets of data requesting stress output at various element locations; EQ. 0; centroid output only



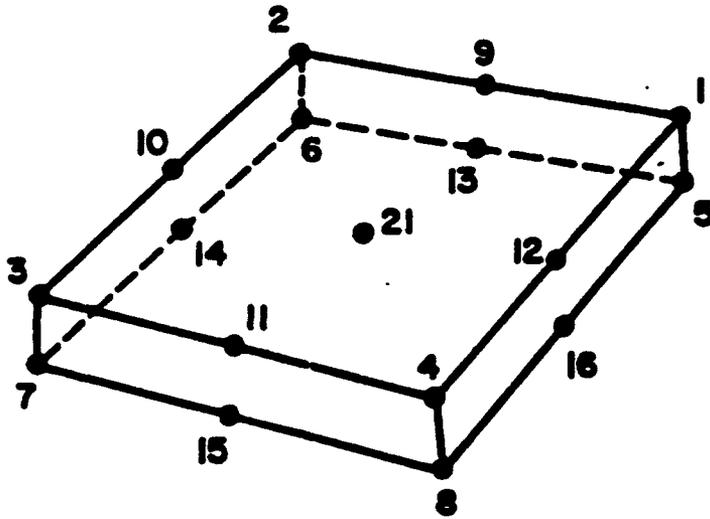
THREE DIMENSIONAL ISOPARAMETRIC ELEMENT



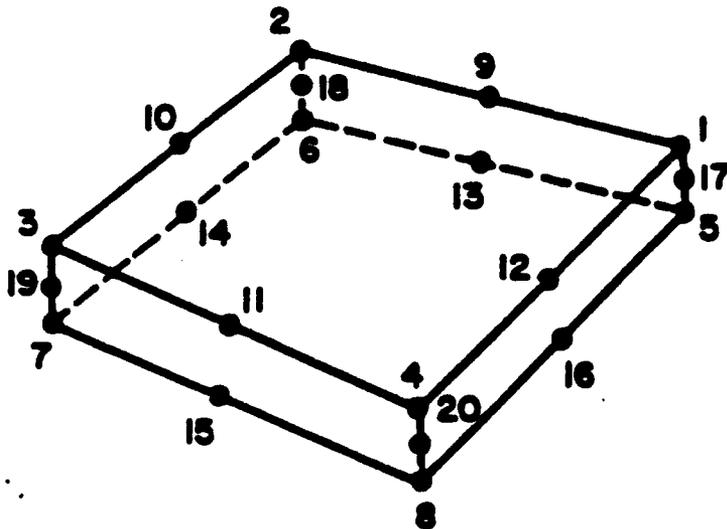
HEXAHEDRAL ELEMENT IN NATURAL COORDINATES



a. 16 - NODE ELEMENT



b. 17 - NODE ELEMENT



c. 20 - NODE ELEMENT

COMMONLY USED ELEMENT GEOMETRIES

A. Control Line (10I5) (continued)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(6)	41 - 45	INTRS	Standard integration order for the natural (r,s) directions; $2 \leq \text{INTRS} \leq 4$ EQ. 0 the default is set to "2"
(6)	46 - 50	INTT	Standard integration order for the natural (t)- direction; $2 \leq \text{INTT} \leq 4$ EQ. 0 the default is set to "2"

NOTES:

(1) The variable MAXTP limits the number of temperature points that can be input for any one of the NUMMAT material sets; i.e., the variable NTP on the material property line (section B below) cannot exceed the value of MAXTP.

(2) NORTH0 specifies the number of material axes orientation lines to be read and if omitted, all orthotropic material axes are assumed to coincide with the global cartesian axes X,Y,Z.

(3) NDLS specifies the number of line pairs of distributed surface load data to be read. NDLS must be a positive integer if any pressure loads are to be applied to solid element faces.

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

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

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

B. Material Property Lines

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

Material Identification Line (2I5,2F10.0,6A6)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(1)	1 - 5	M	Material identification number; $1 \leq M \leq \text{NUMMAT}$
	6 - 10	NTP	Number of different temperatures at which properties are given; LE.MAXTP EQ. 0 the default is set to "1"
(2)	11 - 20	WTDEN	Weight density of the material used to computed static gravity loads
	21 - 30	MASSDN	Mass density of the material used to compute the mass matrix in a dynamic analysis; EQ. 0; default set to "WTDEN/386.4"
	31 - 66		Material description used to label the output.

NOTES:

(1) Material numbers (M) must be input in ascending sequence beginning with "1" and ending with "NUMMAT"; omissions or repetitions are not permitted.

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

Material Lines (7F10.0,6F10.0)

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

Material Line 1

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(1)	1 - 10		Temperature, T_n
(2)	11 - 20		E_{11} at T_n
	21 - 30		E_{22} at T_n
	31 - 40		E_{33} at T_n
	41 - 50		ν_{12} at T_n
	51 - 60		ν_{13} at T_n
	61 - 70		ν_{23} at T_n

Material Line 2

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 - 10		G_{12} at T_n
	11 - 20		G_{13} at T_n
	21 - 30		G_{23} at T_n
	31 - 40		α_1 at T_n
	41 - 50		α_2 at T_n
	51 - 60		α_3 at T_n

NOTES:

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

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

$$\begin{bmatrix} \epsilon_{11} \\ \epsilon_{22} \\ \epsilon_{33} \\ \gamma_{12} \\ \gamma_{23} \\ \gamma_{31} \end{bmatrix} = \begin{bmatrix} 1/E_{11} & -\nu_{12}/E_{22} & -\nu_{13}/E_{33} & 0 & 0 & 0 \\ -\nu_{21}/E_{11} & -1/E_{22} & -\nu_{23}/E_{33} & 0 & 0 & 0 \\ -\nu_{31}/E_{11} & -\nu_{32}/E_{22} & -1/E_{33} & 0 & 0 & 0 \\ 0 & 0 & 0 & 1/G_{12} & 0 & 0 \\ 0 & 0 & 0 & 0 & 1/G_{23} & 0 \\ 0 & 0 & 0 & 0 & 0 & 1/G_{13} \end{bmatrix} \begin{bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \tau_{12} \\ \tau_{23} \\ \tau_{31} \end{bmatrix} - [\Delta\alpha_1 \quad \Delta\alpha_2 \quad \Delta\alpha_3 \quad 0 \quad 0 \quad 0]^T$$

where ϵ_{ii} and σ_{ii} are normal strains and stresses in the X_i directions; γ_{ij} and τ_{ij} are shear strains and stresses on the principal material planes; α_{ij} are the coefficients of thermal expansion, and ΔT is the increase in temperature from stress free (reference temperature (T_0)) distributed over the element volume specified on the element data line (i.e., $\Delta T = T_n - T_0$).

Material Axes Orientation Sets (4I5)

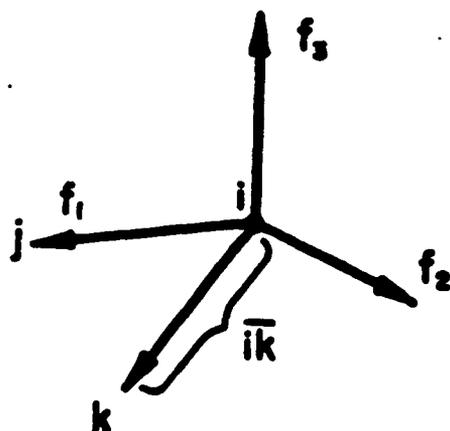
If NORTH0 is zero on the Control Line, skip this data section, and all material axes (X_1, X_2, X_3) will be assumed to coincide with the global cartesian system (X, Y, Z). Otherwise, NORTH0 lines must be input as follows:

Notes	Columns	Variables	Entry
(1)	1 - 5	M	Identification number; GE. 1 and LE.NORTH0
(2)	6 - 10	NI	Node number for point "i"
	11 - 15	NJ	Node number for point "j"
	16 - 20	NK	Node number for point "k"

NOTES:

(1) Identification number (M) must be input in increasing sequence beginning with "1" and ending with NORTH0".

(2) Orthotropic material axes orientations are specified by means of the three node numbers NI,NJ,NK. For the special case where orthotropic material axes coincide with the global axes (X, Y, Z), it is not necessary to input data in this section; see Section F, note (4). Let f_1, f_2, f_3 be the three orthogonal vectors which define the axes of material orthotropy, then their directions are as shown below:



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

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

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

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

C. Distributed Surface Load Data

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

Surface Load Control Line (315)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(1)	1 - 5	N	Load set identification number; $1 \leq N \leq \text{NDLS}$
(2)	6 - 10	NFACE	Element face number on which this distributed load is acting; $1 \leq \text{NFACE} \leq 6$
(3)	11 - 15	LT	Load type code; EQ. 1; prescribed normal pressure intensities EQ. 2; hydrostatically varying pressure field EQ. 0; default set to "1"

NOTES:

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

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

Table 8.1. Corner Node Numbers for the Solid Element Faces

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

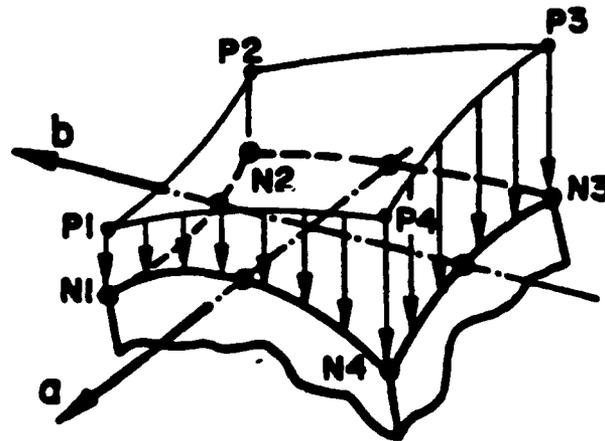
(3) Two types of surface pressure loads may be applied to faces of the elements. If LT equals 0 (or 1), a normal pressure distribution is prescribed directly by means of pressure intensities at the face corner nodes. If LT equals 2, the face is exposed to hydrostatic pressure due to

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

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(1)	1 - 10	P1	Pressure at face node N ₁
(2)	11 - 20	P2	Pressure at face node N ₂
			EQ. 0; default set to "P1"
	21 - 30	P3	Pressure at face node N ₃
			EQ. 0; default set to "P1"
	31 - 40	P4	Pressure at face node N ₄
			EQ. 0; default set to "P1"

NOTES:

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



The face corner node numbers are given on the element data line (Section F) and positive pressure tends to compress the volume of the element.

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

$$p(a,b) = P_1 x h_1 + P_2 x h_2 + P_3 x h_3 + P_4 x h_4$$

where

$$\begin{array}{lll} h_1 = (1/4) & (1+a) & (1+b) \\ h_2 = (1/4) & (1-a) & (1+b) \\ h_3 = (1/4) & (1-a) & (1-b) \\ h_4 = (1/4) & (1+a) & (1-b) \end{array}$$

in quadrilateral natural face coordinates (a,b).

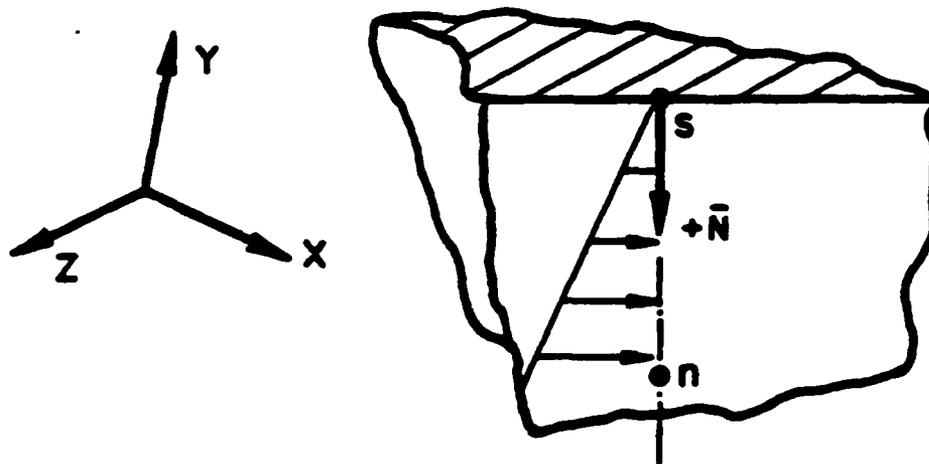
(2) If any of the entries P2,P3,P4 are omitted, these values are reset to the value of P1; i.e., for a uniformly distributed pressure (p), we have P1 equals p and columns 11-40 blank. If P2 is zero specify a small number.

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

NOTES:

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

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



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

D. Stress Output Request Location Sets (7I5)

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

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(1)	1 - 5	LOC1	Location number of output point 1
	6 - 10	LOC2	Location number of output point 2
	11 - 15	LOC3	Location number of output point 3
	16 - 20	LOC4	Location number of output point 4
	21 - 25	LOC5	Location number of output point 5
	26 - 30	LOC6	Location number of output point 6
	31 - 35	LOC7	Location number of output point 7
	:	}	Up to 27 points
	:		
	:		

NOTES:

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

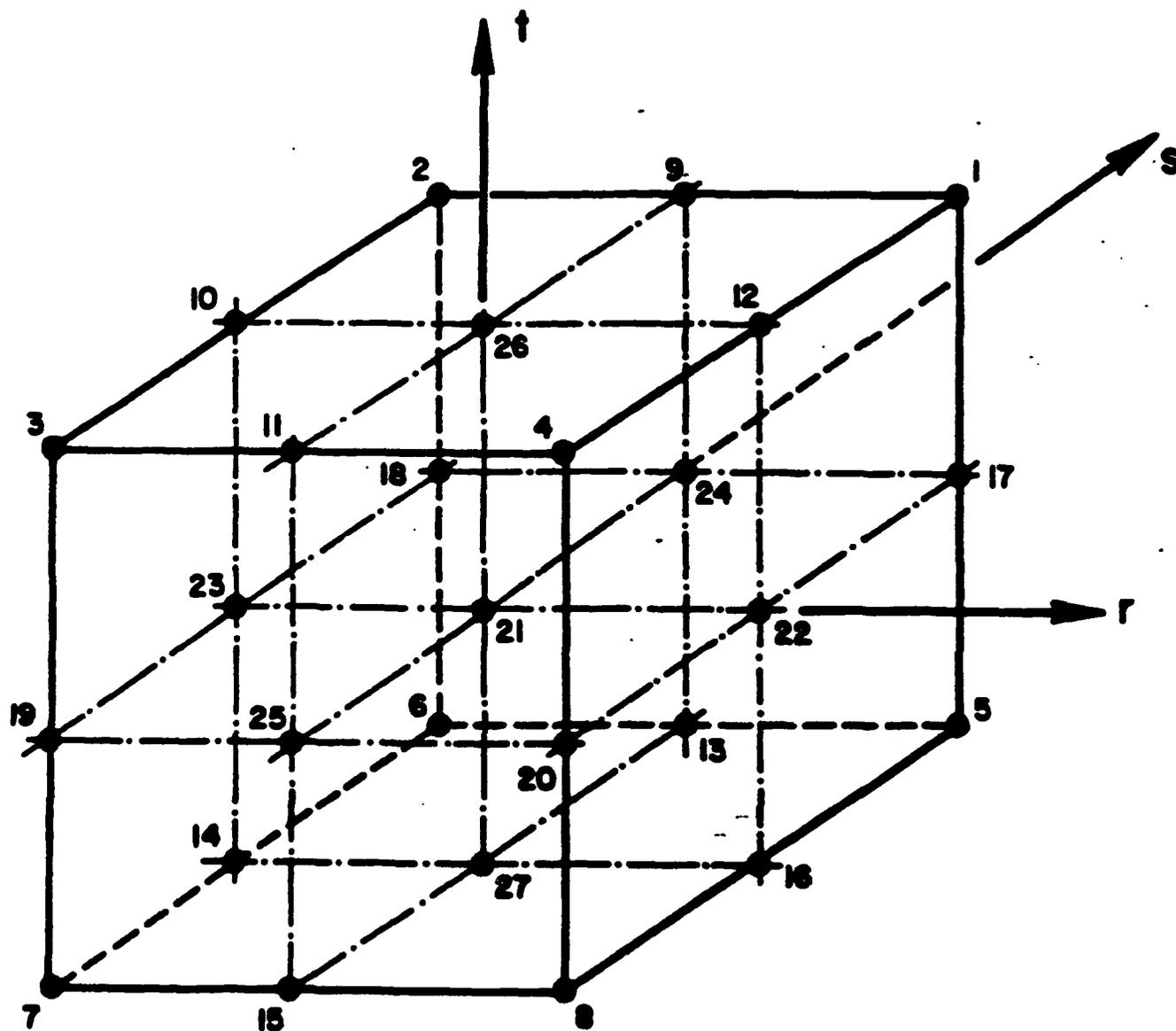
FACE 2, ..., FACE 6 correspond to output locations 22, 23, ..., 27 respectively. (See Table 8.1).

E. Element Load Case Multipliers

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

X-Direction Gravity, Line 1 (4F10.0)

<u>Notes</u>	<u>Columns</u>	<u>Variable</u>	<u>Entry</u>
(1)	1 - 10	GXA	Fraction of X-direction gravity to be applied in element load case A
	31 - 40	GXD	Fraction of X-direction gravity to be applied in element load case D



ELEMENT STRESS OUTPUT LOCATION NUMBERS

Y-Direction Gravity, Line 2 (4F10.0)

Z-Direction Gravity, Line 3 (4F10.0)

Thermal loads, Line 4 (4F10.0)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(2)	1 - 10	TA	Fraction of thermal loads to be applied in element load case A
	11 - 20	TB	Fraction of thermal loads to be applied in element load case B
	21 - 20	TC	Fraction of thermal loads to be applied in element load case C
	31 - 40	TD	Fraction of thermal loads to be applied in element load case D

Pressure loads, Line 5 (4F10.0)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(3)	1 - 10	PA	Fraction of pressure loads to be applied in element load case A
	11 - 20	PB	Fraction of pressure loads to be applied in element load case B
	21 - 20	PC	Fraction of pressure loads to be applied in element load case C
	31 - 40	PD	Fraction of pressure loads to be applied in element load case D

NOTES:

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

(2) Thermal loads are computed knowing the node temperatures input in the Nodal Point Data Section, the stress free reference temperature (T_0) input in Section F and the element's material properties and node coordinates. The temperature distribution within the element is described using the same interpolation functions which describe the variation of displacements within the element.

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

F. Element Lines

Two line (if MAXNOD equals 8) or three lines (if MAXNOD.GT. 8) must be prepared for each element that appears in the input, and the format for these lines is as follows:

Element Line 1 (6I5,F10.,4I5,4I2)

<u>Notes</u>	<u>Columns</u>	<u>Variable</u>	<u>Entry</u>
(1)	1 - 5	M	Element number; $1 \leq M \leq \text{NSOL21}$
(2)	6 - 10	NDIS	Number of nodes to be used in describing the element's displacement field; EQ. 0; default set to "MAXNOD"
(3)	11 - 15	NXYZ	Number of nodes to be used in the description of element geometry; EQ. 0; default set to "NDIS" EQ. NDIS isoparametric element LT.NDIS subparametric element
	16 - 20	NMAT	Material identification number GE. 1 AND LE.NUMMAT
(4)	21 - 35	MAXES	Identification number of the material axis orientation set; $1 \leq \text{MAXES} \leq \text{NORTH0}$ EQ. 0; material axes default to the global X, Y, Z system
(5)	26 - 30	IOP	Identification number of the stress output location set; $1 \leq \text{IOP} \leq \text{NOPSET}$ EQ. 0; centroid output only
	31 - 40	TZ	Stress free reference temperature, T
(6)	41 - 45	KG	Node number increment for element data generation; EQ. 0; default set to "1"
	46 - 50	NRSINT	Integration order for natural coordinate (r,s) direction; EQ. 0; default set to "INTRS"
	51 - 55	NTINT	Integration order for natural coordinate (t) direction; EQ. 0; default set to "INTT"
(7)	56 - 60	IREUSE	Flag indicating that the stiffness and mass matrices for this element are the same as those for the preceding element; EQ. 0; no EQ. 1; yes
(8)	61 - 62	LSA	Pressure set for element load case A
	63 - 64	LSB	Pressure set for element load case B
	65 - 66	LSC	Pressure set for element load case C
	67 - 68	LSD	Pressure set for element load case D $\text{LSD} \leq \text{NDLS}$ defined in Section A

Element Line 2 (16I5)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(9)	1 - 5		Node 1 number
	6 - 10		Node 2 number
	11 - 15		Node 3 number
	16 - 20		Node 4 number
	21 - 25		Node 5 number
	26 - 30		Node 6 number
	31 - 35		Node 7 number
	36 - 40		Node 8 number
(10)	41 - 45		Node 9 number
	46 - 50		Node 10 number
	51 - 55		Node 11 number
	56 - 60		Node 12 number
	61 - 65		Node 13 number
	66 - 70		Node 14 number
	71 - 75		Node 15 number
	76 - 80		node 16 number

Element Line 3 (5I5) (required if MAXNOD.GT. 8)

<u>Note</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 - 5		Node 17 number
	6 - 10		Node 18 number
	11 - 15		Node 19 number
	16 - 20		Node 20 number
	21 - 25		Node 21 number

NOTES:

(1) Element lines must be input in ascending element number order beginning with "1" and ending with "NSOL21". Repetition of element numbers is not permitted, but element lines may be omitted, and missing element data are generated according to the procedure described in note (7).

(2) NDIS is a count of the node numbers actually posted on Lines 2 and 3 which must immediately follow Line 1. NDIS must be at least eight (8), but must be less than or equal to the limit (MAXNOD) which was given on the Control Line, Section A. Element displacements are assigned at the NDIS nonzero nodes, and thus, the order of the element matrices is three (i.e., translations X,Y,Z) times NDIS. The eight corner nodes of the hexahedron must be input, but nodes 9 to 21 are optional, and any or all of these optional nodes may be used to describe the element's displacement field.

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

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

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

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

- (a) all data on Line 1 for generated elements is taken to be the same as that given on the first element line in the sequence;
- (b) nonzero node numbers (given on Lines 2 and 3 for the first element) are incremented by the value "KG" (which is given on Line 1 of the first element) as element generation progresses; zero (or blank) node number entries are generated as zeroes.
- (c) The last element cannot be generated.

(7) The flag IREUSE allows the program to bypass stiffness and mass matrix calculations providing the current element is identical to the preceding element; i.e., the preceding and current elements are identical except for the rigid body translation. If IREUSE equals 0, new matrices are computed for the current element. If IREUSE equals 1 it is also assumed that the node temperatures of the element (for calculation of thermal loads) are the same as those of the preceding element.

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

(9) The first eight node numbers establish the corners or vertices of a general hexahedron and must be all nonzero, (see the Figures associated with Section A). Node numbers must be input in the sequence indicated otherwise volume and surface area integrations will be indefinite.

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

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

THREE-DIMENSIONAL STRAIGHT OR CURVED PIPE ELEMENTS, ELEMENT TYPE 12

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

A. Control Line (1415)

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

NOTES:

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

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

B. Material Property Lines

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

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

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

Material Identification Line (2I5,6A6)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(1)	1 - 5	M	Material identification number; $1 \leq M \leq \text{NUMMAT}$
	6 - 10	NT	Number of different temperatures at which properties are given; EQ. 0; one temperature point is assumed to be input
	11 - 46		Material description used to label the output for this material

NOTES:

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

Material Lines (4F10.0)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(1)	1 - 10	T(N)	Temperature, T_n
	11 - 20	E(N)	Young's modulus, E_n
	21 - 30	XNU(N)	Poisson's ratio, ν_n
	31 - 40	ALP(N)	Thermal expansion coefficient, α_n

NOTES:

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

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

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(1)	1 - 5	N	Section property identification number; $1 \leq N \leq \text{NSECT}$
(2)	6 - 15		Outside diameter of the pipe, d_o
	16 - 25		Pipe wall thickness, t
	26 - 35		Shape factor for shear distortion, α_v
(3)	36 - 45		Weight per unit length of section, Y_1^v
(4)	46 - 55		Mass per unit length of section, ρ_1
	56 - 73		Section description (used to label the output)

NOTES:

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

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

(a) inner and outer pipe radii;

$$r_o = d_o/2$$

$$r_i = r_o - t$$

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

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

(c) principal moments of inertia (bending);

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

$$I_z = I_y$$

(d) polar moment of inertia (torsion);

$$J_x = 2I_y$$

(e) effective shear areas (shear distortions);

$$A_y = A_x/\alpha_v$$

$$A_z = A_y$$

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

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

$$\approx 2.0$$

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

(3) The weight per unit length of section (Y_1) is used to compute gravity loadings on the elements. Fixed end shears, moments, torques, etc. are computed automatically and applied as equivalent nodal loads. These forces will not act on the structure unless first assigned to one of the element load cases (A,B,C,D) in Section D below.

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

$$\rho_1 = Y_1 / 386.4$$

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

C. Branch Point Node Numbers

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

First Line (10I5)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(1)	1 - 5		Node number at branch point 1
	6 - 10		Node number at branch point 2
	.		.
	.		.
	45 - 50		Node number at branch point 10

Second Line (10I5) -- if required

<u>Notes</u>	<u>Columns</u>	<u>Variable</u>	<u>Entry</u>
	1 - 5		Node number at branch point 11

NOTES:

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

D. Element Load Case Multipliers

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

X-Direction Gravity, Line 1 (4F10.0)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(1)	1 - 10		Fraction of X-direction gravity to be applied in element load case A
	11 - 20		Fraction of X-direction gravity to be applied in element load case B
	21 - 30		Fraction of X-direction gravity to be applied in element load case C
	31 - 40		Fraction of X-direction gravity to be applied in element load case D

Y-Direction Gravity, Line 2 (4F10.0)

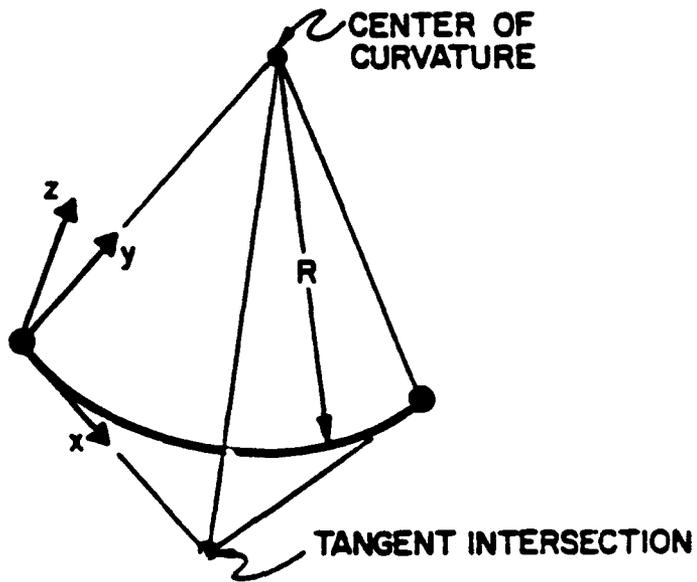
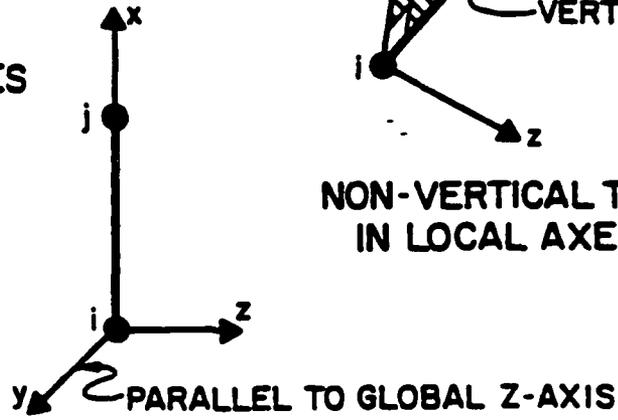
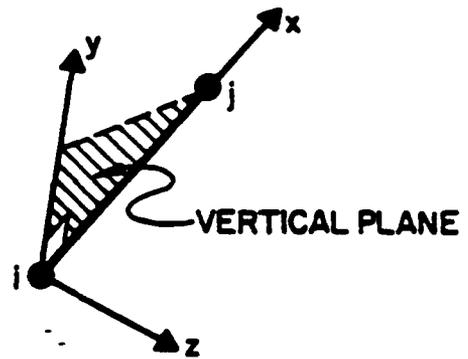
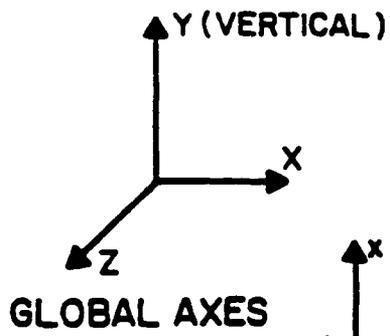
Z-Direction Gravity, Line 3 (4F10.0)

Thermal loads, Line 4 (4F10.0)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(2)	1 - 10		Fraction of thermal loading to be applied in element load case A
	11 - 20		Fraction of thermal loading to be applied in element load case B
	21 - 30		Fraction of thermal loading to be applied in element load case C
	31 - 40		Fraction of thermal loading to be applied in element load case D

Internal Pressure, Line 5 (4F10.0)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(3)	1 - 10		Fraction of pressure-induced loading applied in element load case A
	11 - 20		Fraction of pressure-induced loading applied in element load case B
	21 - 30		Fraction of pressure-induced loading applied in element load case C
	31 - 40		Fraction of pressure-induced loading applied in element load case D



LOCAL COORDINATE SYSTEMS FOR PIPE ELEMENTS

NOTES:

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

(2) No thermal loading will result if the coefficient of thermal expansion has been omitted from all the material lines. Otherwise, thermal loads are computed for each element using the ΔT between the average element temperature (T) and the stress-free temperature (T_0) given with each pipe element line (Section E, below).

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

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

E. Pipe Element Lines

Line Type 1

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(1)	1 - 4 5	N	Pipe element number; GE. 1 and LE.NPIPE Geometric type code: "T" (or blank); tangent section "B" ; bend (circular) section
	6 - 10	I	Node I number
	11 - 15	J	Node J number
	16 - 20	MAT	Material identification number; GE. 1 and LE.NUMMAT
	21 - 25	ISECT	Section property identification number GE. 1 and LE.NSECT
(2)	26 - 35		Stress-free temperature, T_0
(3)	36 - 45		Internal pressure, p
(4)	46 - 55		Positive projection of a local y- vector on the global X-axis; A(yX)
	56 - 65		Positive projection of a local y- vector on the global Y-axis; A(yY)
	66 - 75		Positive projection of a local y- vector on the global Z-axis A (yZ)
(5)	76 - 80	KG	Node number increment for tangent element generation; EQ. 0; default set to "1"

NOTES:

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

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

$$\Delta T = T_a - T_o$$

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

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

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

in which

$$h = tR/r^2$$

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

and

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

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

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

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

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

- (a) the local y-axis is directed positively toward and intersects the center of curvature of the bend (i.e., radius vector);
- (b) the local x-axis is tangent to the arc of the bend and is directed positively from node I to node J.

Note that for all elements, the local x, y, z system is a right-handed set (see Figure on page 76).

(5) If a tangent element sequence exists such that 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 line for the first tangent in the series need be input. The node numbers for the missing tangents are computed using the formulae:

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

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

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

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

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

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

Line Type 2 (F10.0, .3X, A2, 4F10.0)

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

NOTES:

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

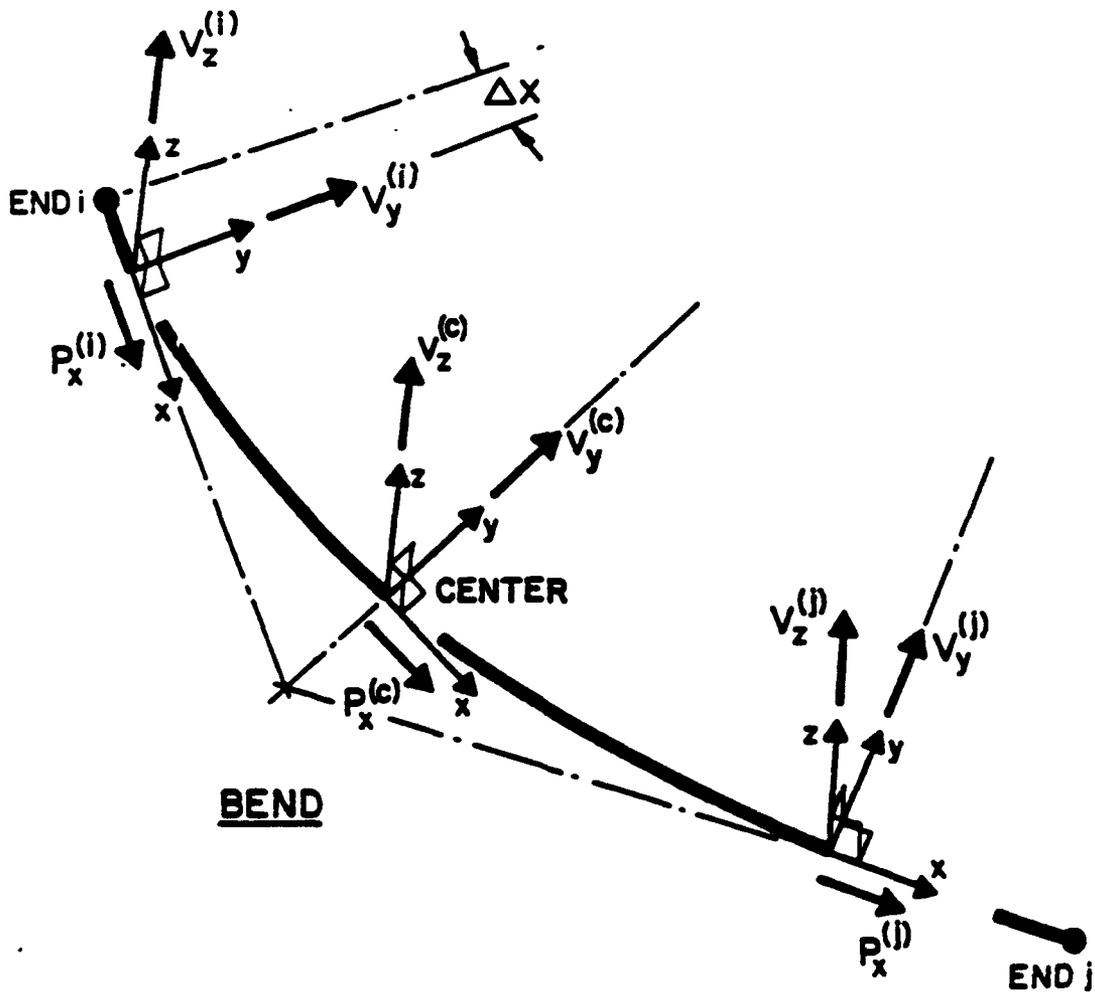
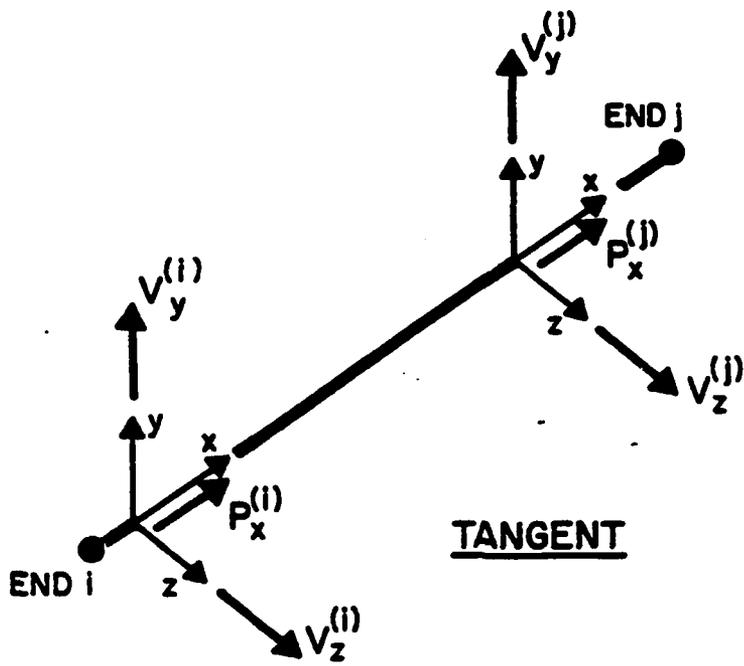
(2) If the tangent intersection point is used, the program computes a radius for the bend and compares the computed value with the input radius. An error is declared if the two radii are different by more than the specified fraction (or multiple) of the section wall thickness. The lengths of the two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

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

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

F. Element Stress Output

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



FORCE SIGN CONVENTION FOR PIPE ELEMENT OUTPUT

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

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

NOTES:

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

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

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

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

ELEMENT LOAD MULTIPLIERS (4F10.0)

<u>Notes</u>	<u>Columns</u>	<u>Variable</u>	<u>Entry</u>
(1,2)	1 - 10	EM (1)	Multipliers for element load case A
	11 - 20	EM (2)	Multipliers for element load case B
	21 - 30	EM (3)	Multipliers for element load case C
	31 - 40	EM (4)	Multipliers for element load case D

END OF DATA SET FOR STATIC ANALYSIS (NDYN = 0)

NOTES:

(1) One line must be included for each static (NDYN equals 0) structure load case requested on the Master Control Line ("LL"). Each line references the respective load case number in ascending order. The four element load sets (A,B,C,D), if created in any of the element data sets are combined with any concentrated loads specified in the Concentrated Load/Mass section for the structure loads cases. For example, suppose an analysis case calls for seven static structure loading conditions (i.e., LL = 7), then the program expects to read seven lines in this section. Further, suppose line number three (i.e., structure load case three) in this section contains the entries:

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

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

Static analysis input is complete with this section. If a new problem is not desired, provide two additional lines: one representing the heading line with "END" entered beginning in Column 1, the second representing the master control card with the number of nodes (NUMNP) equal to zero. Begin a new problem data set with a new heading line.

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

DYNAMIC ANALYSES

Four types of dynamic analysis can be performed by the program. The type of analysis is indicated by the number "NDYN" specified in columns 21-25 of the Master Control Line. If:

- NDYN .EQ. 1; Determination of system mode shapes and frequencies only (complete Mode Shape and Frequency section only)
- NDYN .EQ. 2; Dynamic Response Analysis for arbitrary time dependent loads using mode superposition (complete both Mode Shape and Frequency, and Response History sections)
- NDYN .EQ. 3; Response Spectrum Analysis (complete both Mode Shape and Frequency, and Response Spectrum sections)
- NDYN .EQ. 4; Dynamic Response Analysis for arbitrary time dependent loads using step-by-step direct integration (complete Response History section only)

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

In case of a direct step-by-step integration analysis (NDYN equals 4) do not provide the eigenvalue solution control line.

For the special case of dynamic analysis restart (NDYN equals -2 or -3), data input consists of the Heading Line, the Master Control Line, and either the Response History section (NDYN = -2) or the Response Spectrum section (NDYN = -3), below. Restarting is possible only if a previous eigenvalue solution using the same model was performed with NDYN equals 1, and the results from this eigenvalue solution were saved on the restart file (refer to dynamic analysis restart instructions).

Up to this section the program processes (i.e., expects to read) essentially the same blocks of data for either the static or dynamic analysis cases; certain of these preceding data lines, however, are read by the program but are not used in the dynamic analysis phase. In general, the purpose of the preceding data sections is to provide information leading to the formation of the systems stiffness and mass matrices (appropriately modified for displacement boundary conditions). For example, element load sets (A, B, C, D) may be constructed as through a static case were to be considered, but these data are not used in a dynamic analysis; i.e., the same data file, through the Element Library section, can be used for either type of analysis. The concept of structure loading conditions is not applicable for dynamic analysis, and input for Concentrated Load/Mass Data and Element Load Multipliers must be prepared specially.

MODE SHAPES AND FREQUENCIES (3I5,2F10.0)

This section is required for NDYN equals 1, 2 or 3.

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

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

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

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

END OF DATA SET FOR NDYN = 1

NOTES:

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

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

- (a) the DETERMINANT SEARCH algorithm requires that the upper triangular band of the system stiffness matrix fit into memory (in-core); i.e., one equation "block".
- (b) the SUBSPACE ITERATION algorithm is used if only portions (fractions) of the system matrix can be retained in memory; i.e., the matrix (even though in band form) must be manipulated in blocks.

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

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

Define:

MBAND = maximum equation bandwidth (coefficients)
= (maximum element node number difference)
X (average number of degree of freedom per node)
NEQ = total number of degrees of freedom in the model
= (6) x (total number of nodes) - [number of fixed (deleted) degrees of freedom]
NEQB = number of equations per block of storage
= MTOT/ MBAND/ 2 (for large systems)

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

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

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

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

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

$$[|\lambda(k+1) - \lambda(k)|/\lambda(k)] < RTOL$$

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

$$\varepsilon_i = ||r_i||_2 / ||K\phi_i||_2$$

where

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

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

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

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

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

Only those modes whose frequencies are less than COFQ will be used in the RESPONSE HISTORY or RESPONSE SPECTRUM analyses.

(6) The starting iteration vectors, together with control information, must be written onto TAPE10 before the program execution is started. The section Restart Instruction for Obtaining Additional Eigenvalues and Eigenvectors describes the creation of TAPE10 and gives the required control lines.

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

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

RESPONSE HISTORY ANALYSIS (NDYN equals 2 or NDYN equals 4)

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

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

and/or

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

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

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

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

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

A. Response History Control Line (5I5,2F10.0)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(1)	1 - 5	NFN	Number of different time functions; GE. 1
(2)	6 - 10	NGM	Ground motion indicator; EQ. 0; no ground motion is input EQ. 1; read ground motion control line (Section C below)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(3)	11 - 15	NAT	Number of different arrival times for the forcing functions; EQ. 0; all arrival times are zero
(4)	16 - 20	NT	Total number of solution time steps; .GE. 1
(5)	21 - 25	NOT	Output print interval for stresses, displacements, etc. 1 ≤ NOT ≤ NT
(4)	26 - 35	DT	Solution time step, Δt; GT. 0
(6)	36 - 45	DAMP	Damping factor to be applied to all NF modes (fraction of critical) GE. 0

In case of direct integration (NDYN equals 4) use

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

NOTES:

- (1) At least one time function must be input.
- (2) If no ground acceleration acts on the structure, set "NGH" to zero and skip the ground motion control line, (Section C below). Both ground acceleration and nodal force input are allowed.
- (3) If no arrival time values are input, all forcing functions begin acting on the structure at time zero. The same arrival time value may be reference by different forcing function. "NAT" determines the number of nonzero entries that the program expects to read in the arrival time line, (Section D below).
- (4) The program performs a step-by-step integration of the equations of motion using a scheme which is unconditionally stable with respect to time step size, Δt. In case NDYN equals 2 the modal uncoupled equations of motion are integrated. In case NDYN equals 4 the coupled system equations are integrated. If "T" is the period of the highest numbered mode (normally the NFth mode) that is to be included in the response calculation, Δt should be chosen such that Δt/T < 0.1. A larger time step (i.e., Δt > 0.1T) will not cause failure (instability), but participation of the higher modes is "filtered" from the predicted response. In general, with increasing time step size the solution is capable of capturing less of the higher frequency participation.
- (5) The program computes system displacement at every solution time step, but printing of displacements and recovery of element stresses is only performed at solution step intervals of "NOT". NOT must be at least "1" and is normally selected in the range of 10 to 100.

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

(7) For direct integration (NDYN equals 4) the damping matrix used is $C = \alpha M + \beta K$, where α and β are defined in columns 36 - 55.

B. Time-Varying Load Lines (4I5,F10.0)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(1)	1 - 5	NP	Nodal point number where the load component (force or moment) is applied; $1 \leq NP \leq \text{NUMNP}$
(2)	10	IC	EQ. 0 last line only Degree of freedom number; $1 \leq IC \leq 6$ ($\delta X=1, \delta Y=2, \delta Z=3, \phi X=4, \phi Y=5, \phi Z=6$)
(3)	11 - 15	IFN	Time function number; $1 \leq \text{IFN} \leq \text{NIFN}$
(4)	16 - 20	IAT	Arrival time number; EQ. 0; load applied at solution start GE. 1; nonzero arrival time
(5)	21 - 30	P	Scalar multiplier for the time function; EQ. 0; no load applied

NOTES:

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

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

(3) A nonzero time function number ("IFN") must be given for each forcing function. IFN must be between 1 and NFN. The time functions are input tabularly in the time function definition section, below. Function values at times between input time point are computed with linear interpolation.

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

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

C. Ground Motion Control Line (615)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(1)	1 - 5	NFNX	Time function number describing the ground acceleration in the X-direction
	6 - 10	NFNY	Time function number describing the ground acceleration in the Y-direction
	11 - 15	NFNZ	Time function number describing the ground acceleration in the Z-direction
(2)	16 - 20	NATX	Arrival time number, X-direction
	21 - 25	NATY	Arrival time number, Y-direction
	26 - 30	NATZ	Arrival number, Z-direction

NOTES:

(1) This line must be input only in the ground motion indicator ("NGM") was set equal to one (1) on the Response History Control Line, above. A zero time function number indicates that no ground motion is applied for that particular direction.

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

D. Arrival Time Lines

a. Line One (8F10.0)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(1)	1 - 10	AT(1)	Arrival time number 1
	11 - 20	AT(2)	Arrival time number 2
		.	.
		.	.
	71 - 80	AT(8)	Arrival time number 8

b. Line Two (8F10.0) - (required if NAT.GT. 8)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 -10	AT(9) etc.	Arrival time number 9 etc.

NOTES:

(1) The entry ("NAT") given in column 11-15 on the Response History Control Line above specifies the total number of arrival time entries to be read in this section. Input as many lines as are required to define "NAT" different arrival times, eight entries per line. If no arrival times were requested (NAT equals 0) supply one blank line in this section.

E. Time Function Definition Lines

Supply one set (line 1 and card(s) 2) of input for each of the "NFN" time functions requested in column 1-5 of the Response History Control Line above. At least one set of time function lines is expected in this section. The line sets are input in ascending function number order.

Line 1 (15,F10.0,12A5)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(1)	1 - 5	NLP	Number of function definition points; GE. 2
(2)	6 - 15	SFTR	Scale factor to be applied to f(t) values; EQ. 0; default set to "1.0"
	16 - 75	HED(12)	Label information (to be printed with output) describing this function table

NOTES:

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

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

Line 2 (12F6.0)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(1)	1 - 6	T(1)	Time values at point 1, t_1
	7 - 12	F(1)	Function value at point 1, $f(t_1)$
	13 - 18	T(2)	Time value at point 2, t_2
	19 - 24	F(2) etc.	Function value at point 2, $f(t_2)$ etc.

NOTES:

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

F. Output Definition Lines

To minimize the amount of the output which would be produced by the program if all displacement, stresses, etc. were printed, output requests for specific components must be given in this section. Time histories for selected components appear in tables; the solution step output printing interval is specified as "NOT" which is given in column 21-25 of the Response History Control Line above.

Displacement Output Requests. Control Line (215)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(1)	1 - 5	KKK	Output type indicator; EQ. 1; print histories and maxima EQ. 2; printer plot histories and recovery of maxima EQ. 3; recovery maxima only
(2)	6 - 10	ISP	Printer plot spacing indicator

NOTES:

(1) The type of output to be produced by the program applies to all displacement requests. KKK equals 0 is not permitted.

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

Node Displacement Request Lines (715)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(1)	1 - 5	NP	Node number GE. 1 and LE.NUMNP EQ. 0 last line only
(2)	6 - 10	IC(1)	Displacement component, request 1
	11 - 15	IC(2)	Displacement component, request 2
	16 - 20	IC(3)	Displacement component, request 3
	21 - 25	IC(4)	Displacement component, request 4
	26 - 30	IC(5)	Displacement component, request 5
	31 - 36	IC(6)	Displacement component, request 6 GE. 1 and LE.6 EQ. 0 terminates requests for the node

NOTES:

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

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

Element Stress Component Output Requests. Control Line (2I5)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(1)	1 - 5	KKK	Output type indicator; EQ. 1; print histories and maxima EQ. 2; printer plot of histories and recovery of maxima EQ. 3; recover maxima only
(2)	6 - 10	ISP	Plot spacing indicator

NOTES:

(1) The type of output to be produced by the program applies to all displacement requests. KKK equals 0 is not permitted.

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

Element Stress Component Request Lines (13I5)

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

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(1)	1 - 5	NEL	Element number GE. 1
(2)	6 - 10	IS(1)	EQ. 0; last line in the group only Stress component number for output, request 1
	11 - 15	IS(2)	Stress component number for output, request 2
	.	.	.
	61 - 65	IS(12)	Stress component number for output, request 12

END OF DATA SET FOR NDYN = 2 OR NDYN = 4

NOTES:

(1) Terminate each different element output group (type) with a blank line. Elements within a group must be in element number order (ascending); element number repetitions are not permitted.

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

RESPONSE SPECTRUM ANALYSIS (NDYN equals 3)

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

A. Control Line (3F10.0,I5)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(1)	1 - 10	FX	Factor for X-direction input
	11 - 20	FY	Factor for Y-direction input
	21 - 30	FZ	Factor for Z-direction input
(2)	31 - 35	IST	EQ. 0; not acting
			Input spectrum type;
			EQ. 0; displacement vs. period EQ. 1; acceleration vs. period

NOTES:

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

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

B. Spectrum Lines

a. Heading Line (12A6)

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
	1 - 72	HED(12)	Heading information used to label the spectrum table

b. Control Line (I5,F10.0)

<u>Notes</u>	<u>Columns</u>	<u>Variable</u>	<u>Entry</u>
	1 - 5	NPTS	Number of definition points in the spectrum table; GE. 2
	6 - 15	SFTR	Scale factor used to adjust the displacement (or acceleration) ordinates in the spectrum table EQ. 1.0; no adjustment

c. **Spectrum Data (2F10.0)**

<u>Notes</u>	<u>Columns</u>	<u>Variables</u>	<u>Entry</u>
(1)	1 - 10	T	Period (reciprocal of frequency)
(2)	11 - 20	S	Value of displacement (or acceleration if IST equals 1)

END OF DATA SET FOR NDYN = 3

NOTES:

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

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

EXECUTION INSTRUCTIONS

MS-DOS COMPUTERS

Execution of SAP IV on a microcomputer can only be done on a fixed disk system with a math coprocessor chip. The primary memory requirement is 640K, and the fixed disk must have at least 10 Megabytes available during a large run. Some smaller runs can have smaller disk space requirements. The program reads data from the "SAP4.DAT" file and if the run is a restart, the appropriate restart files are used.

The master diskette contains several files. The three SAP IV diskettes are 360K double sided, double density disks so most machines can read them. There are three parts of the SAP4 executable program: SAP4.EXE, SAP4COM.OVL, and SAP4DYN.OVL. The RUNSAP4.BAT file is set up so the user can name the data file as desired and the output file will be named with the same name, however the input data file must use the extension .DAT and the output file will be assigned the extension .OUT. The RUNSAP4.BAT file also erases all of the TAPE files that are required for restarts and post-processing. This convention can be changed to suit user restart needs.

The current version of SAP IV, supplied on the distribution diskette, has been set up so it will solve medium sized problems. The problem size is determined by the number of nodes, resulting in the following size limitations:

- Maximum number of nodes is 1,500
- Maximum storage allocation (MTOT) is 15,000
- Executable program size is 400K bytes
- Number of disk files simultaneously open is 13
- Recommended number of file transfer buffers is 13 (7K)

There are a number of options available to modify the program so it can address larger problems. NCEL can assist users who have requirements that exceed the current capacity of the program. Options, such as extended memory and random-access memory (RAM) disk are available.

OTHER COMPUTERS

This program can be used on other computers. Contact NCEL Code L51 for instructions.

DATA FILES CREATED DURING EXECUTION

SAP IV creates eleven data files during execution. These files are used for various reasons depending upon the type of solution being conducted. Their size is also problem dependent. Most problems will run if 10 megabytes of disk space are available. The files are unformatted, meaning they can not be examined unless a special program is used. However, the TAPE11, TAPE12, SAP4.DAT, and the SAP4.OUT files are formatted so they can be examined. The table below provides some information about each file.

<u>File</u>	<u>Contents</u>
TAPE1	Restart: Element description data
TAPE2	Restart: Stiffness matrix, displacement
TAPE3	Scratch: Mass matrix
TAPE4	Scratch: Nodal displacement
SAP4.DAT	Initial input data
SAP4.OUT	Printer output data
TAPE7	Restart: Subspace iteration
TAPE8	Restart: Dynamic analysis Postprocessing: Geometry and element data
TAPE9	Restart: Mass matrix
TAPE10	Postprocessing: Element stress results 4
TAPE11	Postprocessing: Time history results
TAPE12	Postprocessing: Problem size data
TAPE13	Scratch: Stress-displacement transformation data

DYNAMIC ANALYSIS RESTART INSTRUCTIONS

The purpose of this section is to describe the procedure required for program restart following an eigenvalue/eigenvector analysis. The restart option has been included in the program in order to make a repeated forced response or spectrum analysis possible without solving each time for the required eigensystem. For medium-to-large size models, eigenvalue solution is computation intense when compared to the forced response calculations; hence, excessive computation time may be incurred if the entire job has to be rerun due to improper specification of forcing functions or input spectra, inadequate requests, etc. For small models (less than 100 nodes) the extra effort required for restart is normally not justified.

A complete dynamic analysis utilizing the restart feature requires that the analysis be conducted in two steps:

- Step 1: Eigenvalue extraction solution only, after which program files TAPE1, TAPE2, TAPE7, TAPE8, and TAPE9 are saved.
- Step 2: Response Analysis (NDYN equals -2) or a Response Spectrum Analysis (NDYN equals -3) using previously saved files TAPE1, TAPE2, TAPE7, TAPE8, and TAPE9.

For a given model, the first step creating the restart tapes is executed once. The restart tapes then contain all the initial information required by the program at the beginning of a forced response analysis, Step 2. More than one step 2 may be executed using the restart tapes.

Step 1 PSEUDO Problem Data File:

HEADING LINE
MASTER CONTROL LINE with
 LL = 0
 NF ≥ 1
 NDYN = 0
 MODEX = 0
JOINT DATA
ELEMENT DATA
CONCENTRATED MASS DATA
ELEMENT LOAD MULTIPLIERS
DYNAMIC ANALYSIS: Mode shapes and Frequencies
 Blank line
 Blank line

Step 2 PSEUDO Problem Data File:

HEADING LINE
MASTER CONTROL LINE with
LL = 0
NF \geq 1
NDYN = -2 or -3
MODEX = 0
DYNAMIC ANALYSIS

Dynamic Response Analysis (NDYN equals -2)
or
Response Spectrum Analysis (NDYN equals -3)
Blank line
Blank line

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

**RESTART INSTRUCTIONS FOR OBTAINING ADDITIONAL
EIGENVALUES AND EIGENVECTORS**

In the dynamic analysis of large-order systems, the solution of the required eigensystem is normally the most computation intensive phase. The option described in this section demonstrates how it is possible to use NFO previously calculated eigenvalues and vectors when the solution for $NF \geq NFO$ eigenvalues and eigenvectors is required.

Assume that in Step 1, the solution for NFO eigenvalues and eigenvectors was performed. At the end of this step, TAPE2 and TAPE7 must have been saved. Assuming that in Step 2 the solution of NF additional eigenvalues and eigenvectors is required, then prior to the execution of this job, the previously saved TAPE2 and TAPE7 files are combined into one file, TAPE10, and that file is rewound.

PRE- AND POST-PROCESSING

The key to using the finite element method is computer applications. A program like SAP IV requires a large amount of coordinate, connectivity, element property, and load information to solve common structural problems. The results of the analysis are often very voluminous especially for dynamic analysis problems. This section describes various ways the user can simplify the task of preparing the input data and of examining the output data.

PRE-PROCESSING

Pre-processing involves computer applications to generate the modal coordinates, modal connectivity, element properties, and load information. A number of options are available to simplify data preparation:

Manual Preparation

The SAP IV program uses fixed format line (line) image method of input. The lines of data are collected into an input data file. This method of input is very laborious for large problems and therefore is not recommended. However, small problems can often be quickly posed using this method with the aid of a screen editor program. The screen editor program tab features for right and decimal justification will simplify preparing the input file. The screen editor program must be capable of preparing an alpha-numeric file for SAP IV with the 10 numbers and 26 capital characters and no special control characters. Experience has shown that this method of input can be very expedient and easy to modify for the smaller class problems. Additionally, the SAP IV coordinate and element generation features, while somewhat limited, can be used to simplify preparing the data.

Automatic Generators

There are a few automatic input data generators that have been successfully used to generate SAP IV data.

GEN3D is Naval Civil Engineering Laboratory program that generates the node and element connectivity data. This program is technically one of the most advanced mech generators that is available. The primary limitation is the user interface is less not desirable, since it does not support immediate user interaction.

Finite Element Modeling (FEM) software is the best way available to prepare input data for SAP IV. Such a package encourages the user to develop a three-dimensional model of the structure and then provides comprehensive commands to superimpose a finite element mesh on top of the model. The mesh contains the coordinate and element connectivity information. The package can also be used to generate the element properties and load cases. The features include ways to check the quality of the mesh, element, and load information before the analysis program is executed.

The Naval Civil Engineering Laboratory has developed some special purpose generation software to prepare data information for specific types of problems. A generator for overhead electric traveling bridge cranes has been prepared. A two-dimensional drydock model generator has also been prepared. The primary strength of the SAP IV program is having the flexibility of developing problem specific generation software.

Automatic load generation has been accomplished. Special software has been developed to generate hydrostatic pressures on complex curved surfaced. Time series analysis software has been used to prepare dynamic test data so it can be used as input data. The dynamic loads include ocean waves, ship motion, wind, and blast pressures.

POST-PROCESSING

Post-processing involves reducing the large quantities of analysis results into meaningful presentations that can be quickly assimilated and evaluated. SAP IV prepares a number files (TAPE1, TAPE8, TAPE10, TAPE11, TAPE12) that contain the model undeformed geometry, the displacement, and the various stress, force, moment, velocity, and accelerations quantities. Programs have been prepared that read these files so that information can be further processed to meet specific analysis objectives.

The undeformed geometry can be plotted using the SAPLOT program. This is helpful to verify the correctness of the coordinates and the element connectivity.

The deformed geometry can be plotted using the SAPPLOTD program. This program will plot the deformed structure with or without the undeformed structure in the background.

Various programs have been written by the Naval Civil Engineering Laboratory to process the force, moment, and stress results. A beam stress program has been used to evaluate the beam element results with respect to bending and shear stress. Moment and shear diagrams have also been produced. The two and three dimensional element results have been studied using MOVIE.BYU which prepares contour and continuous tone displays of requested information. An interface has been prepared to aid studying time history data using time series analysis techniques.

REFERENCES

1. Earthquake Engineering Research Center, Report EERC 73-11, "SAP IV A Structural Analysis Program for Static and Dynamic Response of Linear Systems," by Klaus-Jurgen Bathe, Edward L. Wilson, and Fred E. Peterson, Berkeley, CA, April 1974.
2. Bathe, Klaus-Jurgen, "Finite Element Procedures in Engineering Analysis," Prentice-Hall, Englewood Cliffs, NJ, 1982.
3. Gallagher, Richard H., "Finite Element Analysis: Fundamentals," Prentice-Hall, Englewood Cliffs, NJ, 1975.
4. Cowper, G.R., "The Shear Coefficient in Timoshenko's Beam Theory," Journal of Applied Mechanics, Volume 33, Transactions of the ASME, 1966, pp 335-340.
5. Roark, Raymond J., "Formulas for Stress and Strain," McGraw-Hill, New York, NY, 1954.

SAP IV - VERSION 1.0

FEEDBACK REPORT

The Naval Civil Engineering Laboratory is fully dedicated to supporting GEMS users. A primary requirement for this task is to establish a priority listing of user requirement. It would be of great value to the development of new software if you, the user, would complete the feedback questions below. Since each individual user may have specific requirement, please reproduce this page as many times as necessary.

Please circle the number that best applies in questions 1 through 4, complete the other questions, fold at tic marks, and mail to NCEL with franked label on reverse side or to address at bottom of page.

1. Was the software beneficial (productive)?

No benefit 0 1 2 3 4 5 6 7 8 9 10 Very beneficial

2. Was it easy to use (user friendly)?

Difficult 0 1 2 3 4 5 6 7 8 9 10 Very easy

3. Does this software make decisions more reliable?

No 0 1 2 3 4 5 6 7 8 9 10 Yes

4. Does it better document the design?

No 0 1 2 3 4 5 6 7 8 9 10 Yes

5. Did it save time?

Yes___ No___ Estimated percent saved_____

6. What would make future software more user friendly?

7. What further support would you like to have on the GEMS systems?

8. What other comments or remarks would you like to add?

Activity_____

Telephone_____

Mail address is:

NAVFAC GEMS Support Group
Naval Civil Engineering Laboratory
Code L54
Port Hueneme, CA 93043-4328

APPENDIX A
OUTPUT STRESS COMPONENTS

ELEMENT TYPE	MAXIMUM NUMBER OF COMPONENTS	STRESS COMPONENT NUMBER	OUTPUT SYMBOL	DESCRIPTION
1. TRUSS	(2)	(1)	(P/A)	AXIAL STRESS
		(2)	(P)	AXIAL FORCE
<hr/>				
2. BEAM	(12)	(1)	(PI(I))	1-Force at end I
		(2)	(V2(I))	2-Shear at end I
		(3)	(V3(I))	3-Shear at end I
		(4)	(TI(I))	1-Torque at end I
		(5)	(M2(I))	2-Moment at end I
		(6)	(M3(I))	3-Moment at end I
		(7)	(P1(J))	1-Force at end J
		(8)	(V2(J))	2-Shear at end J
		(9)	(V3(J))	3-Shear at end J
		(10)	(TI(J))	1-Torque at end J
		(11)	(M2(J))	2-Moment at end J
		(12)	(M3(J))	3-Moment at end J
<hr/>				
3. PLANE- STRESS/ PLANE- STRAIN	(20)	(1)	(11-S0)	V- Stress at point 0
		(2)	(22-S0)	U- Stress at point 0
		(3)	(33-S0)	T- Stress at point 0
		(4)	(12-S0)	UV-Stress at point 0
		(5)	(11-S1)	V- Stress at point 1
		(6)	(22-S1)	U- Stress at point 1
		(7)	(33-S1)	T- Stress at point 1
		(8)	(12-S1)	UV-Stress at point 1
		(9)	(11-S2)	V- Stress at point 2
		(10)	(22-S2)	U- Stress at point 2
		(11)	(33-S2)	T- Stress at point 2
		(12)	(12-S2)	UV-Stress at point 2
		(13)	(11-S3)	V- Stress at point 3
		(14)	(22-S3)	U- Stress at point 3
		(15)	(33-S3)	T- Stress at point 3
		(16)	(12-S3)	UV-Stress at point 3
		(17)	(V -S4)	V- Stress at point 4
		(18)	(U -S4)	U- Stress at point 4
		(19)	(T -S4)	T- Stress at point 4
		(20)	(UV-S4)	UV-Stress at point 4

4. TWO-DIMENSIONAL ELEMENTS	(20)	(1)	(11-S0)	V- Stress at point 0
		(2)	(22-S0)	U- Stress at point 0
		(3)	(33-S0)	T- Stress at point 0
		(4)	(12-S0)	UV-Stress at point 0
		(5)	(11-S1)	V- Stress at point 1
		(6)	(22-S1)	U- Stress at point 1
		(7)	(33-S1)	T- Stress at point 1
		(8)	(12-S1)	UV-Stress at point 1
		(9)	(11-S2)	V- Stress at point 2
		(10)	(22-S2)	U- Stress at point 2
		(11)	(33-S2)	T- Stress at point 2
		(12)	(12-S2)	UV-Stress at point 2
		(13)	(11-S3)	V- Stress at point 3
		(14)	(22-S3)	U- Stress at point 3
		(15)	(33-S3)	T- Stress at point 3
		(16)	(12-S3)	UV-Stress at point 3
		(17)	(V -S4)	V- Stress at point 4
		(18)	(U -S4)	U- Stress at point 4
		(19)	(T -S4)	T- Stress at point 4
		(20)	(UV-S4)	UV-Stress at point 4

5. EIGHT NODE BRICK	(12)	(1)	(XX-SL1)	XX-Stress at location 1
		(2)	(YY-SL1)	YY-Stress at location 1
		(3)	(ZZ-SL1)	ZZ-Stress at location 1
		(4)	(XX-SL1)	XY-Stress at location 1
		(5)	(YY-SL1)	YZ-Stress at location 1
		(6)	(ZZ-SL1)	ZX-Stress at location 1
		(7)	(XX-SL2)	XX-Stress at location 2
		(8)	(YY-SL2)	YY-Stress at location 2
		(9)	(ZZ-SL2)	ZZ-Stress at location 2
		(10)	(XX-SL2)	XY-Stress at location 2
		(11)	(YY-SL2)	YZ-Stress at location 2
		(12)	(ZX-SL2)	ZX-Stress at location 2

6. PLATE/SHELL	(6)	(1)	(XX-S/R)	XX-Stress Resultant
		(2)	(YY-S/R)	YY-Stress Resultant
		(3)	(XY-S/R)	XY-Stress Resultant
		(4)	(XX-M/R)	XX-Moment Resultant
		(5)	(YY-M/R)	YY-Moment Resultant
		(6)	(XY-M/R)	XY-Moment Resultant

7.	BOUNDARY	(2)	(1)	(BDRY-F)	BOUNDARY Force
			(2)	(BDRY-M)	BOUNDARY Moment
8.	THICK SHELL AND 3-DIAM	(42)	(1)	(SXX(0))	XX-Stress at centroid (0)
			(2)	(SYY(0))	YY-Stress at centroid (0)
			(3)	(SZZ(0))	ZZ-Stress at centroid (0)
			(4)	(SKY(0))	KY-Stress at centroid (0)
			(5)	(SKZ(0))	KZ-Stress at centroid (0)
			(6)	(SKX(0))	KX-Stress at centroid (0)
			(8)	(SXX(1))	XX-Stress at center face 1
			(9)	(SYY(1))	YY-Stress at center face 1
			(10)	(SZZ(1))	ZZ-Stress at center face 1
			(11)	(SKY(1))	KY-Stress at center face 1
			(12)	(SKZ(1))	KZ-Stress at center face 1
			(13)	(SXX(2))	XX-Stress at center face 2
			(14)	(SYY(2))	YY-Stress at center face 2
			(15)	(SZZ(2))	ZZ-Stress at center face 2
			(16)	(SKY(2))	KY-Stress at center face 2
			(17)	(SKZ(2))	KZ-Stress at center face 2
			(18)	(SKX(2))	KX-Stress at center face 2
			(19)	(SXX(3))	XX-Stress at center face 3
			(20)	(SYY(3))	YY-Stress at center face 3
			(21)	(SZZ(3))	ZZ-Stress at center face 3
			(22)	(SKY(3))	KY-Stress at center face 3
			(23)	(SKZ(3))	KZ-Stress at center face 3
			(24)	(SKX(3))	KX-Stress at center face 3
			(25)	(SXX(4))	XX-Stress at center face 4
			(26)	(SYY(4))	YY-Stress at center face 4
			(27)	(SZZ(4))	ZZ-Stress at center face 4
			(28)	(SKY(4))	KY-Stress at center face 4
			(29)	(SKZ(4))	KZ-Stress at center face 4
			(30)	(SKX(4))	KX-Stress at center face 4
			(31)	(SXX(5))	XX-Stress at center face 5
			(32)	(SYY(5))	YY-Stress at center face 5
			(33)	(SZZ(5))	ZZ-Stress at center face 5
			(34)	(SKY(5))	KY-Stress at center face 5
			(35)	(SKZ(5))	KZ-Stress at center face 5
			(36)	(SKX(5))	KX-Stress at center face 5
			(37)	(SXX(6))	XX-Stress at center face 6
			(38)	(SYY(6))	YY-Stress at center face 6
			(39)	(SZZ(6))	ZZ-Stress at center face 6
			(40)	(SKY(6))	KY-Stress at center face 6
			(41)	(SKZ(6))	KZ-Stress at center face 6
			(42)	(SKX(6))	KX-Stress at center face 6

9. PIPE

A. TANGENT (12)

(1)	(PX(I))	X-Force	at end I
(2)	(VY(I))	Y-Shear	at end I
(3)	(VZ(I))	Z-Shear	at end I
(4)	(TX(I))	X-Torque	at end I
(5)	(MY(I))	Y-Moment	at end I
(6)	(MZ(I))	Z-Moment	at end I
(7)	(PX(J))	X-Force	at end J
(8)	(VY(J))	Y-Shear	at end J
(9)	(VZ(J))	Z-Shear	at end J
(10)	(TX(J))	X-Torque	at end J
(11)	(MY(J))	Y-Moment	at end J
(12)	(MZ(J))	Z-Moment	at end J

B. BEND (18)

(1)	(PX(I))	X-Force	at end I
(2)	(VY(I))	Y-Shear	at end I
(3)	(VZ(I))	Z-Shear	at end I
(4)	(TX(I))	X-Torque	at end I
(5)	(MY(I))	Y-Moment	at end I
(6)	(MZ(I))	Z-Moment	at end I
(7)	(PX(C))	X-Force	at center of arc
(8)	(VY(C))	Y-Shear	at center of arc
(9)	(VZ(C))	Z-Shear	at center of arc
(10)	(TX(C))	X-Torque	at center of arc
(11)	(MY(C))	Y-Moment	at center of arc
(12)	(MZ(C))	Z-Moment	at center of arc
(13)	(PX(J))	X-Force	at end J
(14)	(VY(J))	Y-Shear	at end J
(15)	(VZ(J))	Z-Shear	at end J
(16)	(TX(J))	X-Torque	at end J
(17)	(MY(J))	Y-Moment	at end J
(18)	(MZ(J))	Z-Moment	at end J
