

DEVELOPMENT OF A MODERATELY SIZED FINITE ELEMENT
PROGRAM FOR NONLINEAR STRUCTURAL ANALYSIS

(NASA-CR-150882) DEVELOPMENT OF A
MODERATELY SIZED FINITE ELEMENT PROGRAM FOR
NONLINEAR STRUCTURAL ANALYSIS Final Report
(Texas A&M Research Foundation) 182 p
HC A09/MF A01

N79-15324

Unclas
42218

CSCI 20K G3/39



**aerospace
engineering
department**

TEXAS A&M UNIVERSITY

WALTER E. HAISLER

TEXAS A&M RESEARCH FOUNDATION
COLLEGE STATION, TEXAS 77843

FINAL REPORT
CONTRACT NAS8-31332

FOR

NATIONAL AERONAUTICS AND SPACE ADMINISTRATION
GEORGE C. MARSHALL SPACE FLIGHT CENTER
GEORGE C. MARSHALL SPACE FLIGHT CENTER, ALABAMA 35812

TECHNICAL REPORT No. 3207-77-1

JUNE 1977

REPRODUCED BY
NATIONAL TECHNICAL
INFORMATION SERVICE
U.S. DEPARTMENT OF COMMERCE
SPRINGFIELD, VA. 22161

TEXAS ENGINEERING EXPERIMENT STATION

NOTICE

THIS DOCUMENT HAS BEEN REPRODUCED FROM THE BEST COPY FURNISHED US BY THE SPONSORING AGENCY. ALTHOUGH IT IS RECOGNIZED THAT CERTAIN PORTIONS ARE ILLEGIBLE, IT IS BEING RELEASED IN THE INTEREST OF MAKING AVAILABLE AS MUCH INFORMATION AS POSSIBLE.

Unclassified

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

REPORT DOCUMENTATION PAGE		READ INSTRUCTIONS BEFORE COMPLETING FORM
1 REPORT NUMBER 3207-77-1	2 GOVT ACCESSION NO	3 RECIPIENT'S CATALOG NUMBER
4 TITLE (and Subtitle) Development of a Moderately Sized Finite Element Program for Nonlinear Structural Analysis		5 TYPE OF REPORT & PERIOD COVERED Final
		6 PERFORMING ORG. REPORT NUMBER
7 AUTHOR(s) Walter E. Haisler		8 CONTRACT OR GRANT NUMBER(s) NAS8-31332
9 PERFORMING ORGANIZATION NAME AND ADDRESS Texas A&M Research Foundation College Station, Texas 77843		10 PROGRAM ELEMENT, PROJECT, TASK AREA & WORK UNIT NUMBERS
11 CONTROLLING OFFICE NAME AND ADDRESS National Aeronautics and Space Administration George C. Marshall Space Flight Center, Alabama 35812		12 REPORT DATE June 1977
		13 NUMBER OF PAGES 180
14 MONITORING AGENCY NAME & ADDRESS (if different from Controlling Office)		15 SECURITY CLASS (of this report)
		15a DECLASSIFICATION/DOWNGRADING SCHEDULE
16 DISTRIBUTION STATEMENT (of this Report) Unlimited		
17 DISTRIBUTION STATEMENT (of the abstract entered in Block 20, if different from Report)		
18 SUPPLEMENTARY NOTES		
19 KEY WORDS (Continue on reverse side if necessary and identify by block number) Finite Element Nonlinear Large Strain Plasticity		
20 ABSTRACT (Continue on reverse side if necessary and identify by block number) This report describes the AGGIE I computer program for predicting the linear and nonlinear, static and dynamic structural response of two- and three-dimensional continuum solids. The program is based on isoparametric finite elements and allows for 2-D plane stress, plane strain, and axisymmetric analyses and general 3-D analyses. Large strain kinematics is based on the total Lagrangian formulation. Materially nonlinear models include several elastic-plastic work-hardening models as well as an incompressible (continued)		

DD FORM 1473
1 JAN 73

EDITION OF 1 NOV 65 IS OBSOLETE
S/N 0102- LF-014-6601

Unclassified

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

Unclassified

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

20. Mooney-Rivlin model. Included in this report is a brief description of the theoretical bases of the program, the material models used, the element library and the overall program organization. Instructions for data input preparation are given in detail. Several sample problems are given along with the required program input and program generated solutions.

S/N 0102- LF- 014- 6601

Unclassified

SECURITY CLASSIFICATION OF THIS PAGE(When Data Entered)

DEVELOPMENT OF A MODERATELY SIZED FINITE ELEMENT
PROGRAM FOR NONLINEAR STRUCTURAL ANALYSIS

Final Report

Contract NAS8-31332

June 1977

Prepared by

Texas A&M Research Foundation
College Station, Texas 77843

Walter E. Haisler
Aerospace Engineering Department
Texas A&M University

Prepared for

National Aeronautics and Space Administration
George C. Marshall Space Flight Center
George C. Marshall Space Flight Center, Alabama 35812

NOTICE

This report was prepared as an account of work sponsored by the following agencies: Office of Naval Research and NASA. Neither the United States Government, nor these agencies, nor their employees, nor the authors of this report:

a) Makes any warranty, expressed or implied, with respect to the accuracy, completeness or usefulness of any information, apparatus, product or process disclosed in this report; or

b) Represents that the use of any information, apparatus, product or process disclosed in this report would not infringe privately owned rights; or

c) Assumes any liability with respect to the use of, or for damages resulting from the use of any information, apparatus, product or method disclosed in this report.

ABSTRACT

AGGIE I is a computer program for predicting the linear and nonlinear, static and dynamic structural response of two- and three-dimensional continuum solids. The program is based on isoparametric finite elements and allows for 2-D plane stress, plane strain, and axisymmetric analyses and general 3-D analyses. Large strain kinematics is based on the total Lagrangian formulation. Materially nonlinear models include several elastic-plastic work-hardening models as well as an incompressible Mooney-Rivlin model.

Included in this report is a brief description of the theoretical bases of the program, the material models used, the element library and the overall program organization. Instructions for data input preparation are given in detail. Several sample problems are given along with the required program input and program generated solutions.

ACKNOWLEDGMENT

This research was supported under NASA Contract NAS8-31332, George C. Marshall Space Flight Center, Alabama and ONR Contract N00014-76-C-0150, Office of Naval Research, Washington, D.C. Appreciation is expressed to Mr. John Key of NASA and Dr. Kenneth Saczalski of ONR who served as technical monitors for this research effort.

CONTENTS

ABSTRACT	i
ACKNOWLEDGEMENT	ii
1. INTRODUCTION	1
2. SUMMARY OF AGGIE I COMPUTER PROGRAM	3
3. FUTURE IMPROVEMENTS	16
4. PROGRAM INPUT	17
5. GUIDELINES FOR PROGRAM USE	130
6. PROGRAM RESOURCE REQUIREMENTS, CAPACITY AND LIMITATIONS	133
7. PROGRAM RESTART	134
8. EXAMPLE PROBLEMS	135
9. REFERENCES	174

1. INTRODUCTION

The development of computer programs for nonlinear structural analyses has progressed at a steady pace for the past ten years beginning with special application programs and progressing to larger, more general purpose programs like NONSAP, ANSYS, NASTRAN, and MARC.¹ Many of the larger general purpose programs which are currently available were originally developed for linear structures and later extended to include a nonlinear capability. The research reported herein was aimed at developing a moderately sized finite element program specifically for nonlinear structural analysis.

In beginning the development of such a program, a review¹ of existing nonlinear finite element programs was undertaken with the purpose being to determine what analysis capability currently existed, what techniques were being used, and to determine if any existing computer program might be suitable as a beginning base program. From this review, it was felt that the NONSAP² program developed by Bathe, Wilson and co-workers would form a suitable base from which to begin the work. The NONSAP program was modularized making it easy to modify, the program organization and logic was good, it was fairly well documented and it had a good library of material models and elements to conduct geometrically (large strain) and materially nonlinear analyses of two-dimensional structures (the 3-D capability was limited to linear elastic problems only).

Using the NONSAP program as a base, a moderately sized finite element program for nonlinear structural analysis has been developed. The program is based on two- and three-dimensional isoparametric solid elements. Nonlinearities accounted for in the program may be due to large displacements, large strains and nonlinear material behavior (linear be-

havior is also allowed). A number of material models have been incorporated; these include linear elastic, orthotropic, nonlinear elastic, nonlinear elastic incompressible and elastic-plastic models. Both static and dynamic response predictions are possible. The program also allows for the possibility of displacement incrementation for use in predicting buckling and post-buckling response. Dynamic dimensioning of most FORTRAN arrays allows efficient utilization of available high-speed memory. The program uses either an out-of-core or an in-core equation solver depending upon problem size and available main memory. The mesh description (node and element information) is input on a node by node and element by element basis or may be generated by using a rudimentary substructure option. Provision is made for both conservative and nonconservative (deformation dependent) loading.

A brief description of the theoretical basis of the program, the elements used, the available material models, and program organization is given in the next section. More detailed information regarding these items may be found in Refs. 2-5. A description of the input data required by the AGGIE I program is then presented. Sections are then presented which give suggestions and guidelines for use of the program; program resource requirements, capacity, limitations, and the program restart option. A last section contains selected example problems along with listings of the required input data and program results.

2. SUMMARY OF AGGIE I COMPUTER PROGRAM

The purpose of this section is to provide a summary of the theoretical basis of the computer program, the elements used, the available material models, and the program organization. Details of these may be found in Refs. 2-5.

Incremental Equilibrium Equations

The linearized incremental equilibrium equations which are solved at each time or load step are given by

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

where

$[M]$ = mass matrix

$[C]$ = damping matrix

$[K^t]$ = tangent stiffness matrix at time t

$\{R^{t+\Delta t}\}$ = external nodal force vector at time $t + \Delta t$

$\{F^t\}$ = nodal force vector due to internal element stresses at time t

$\{\ddot{u}^{t+\Delta t}\}$ = nodal acceleration vector at time $t + \Delta t$

$\{\Delta u\}$ = nodal displacement increment vector defined by $u^{t+\Delta t} = u^t + \Delta u$

For a static problem, the mass and damping matrices are omitted and the time increment Δt may be interpreted as an equivalent load increment.

The incremental equations given above yield only an approximate solution (because of the linearization used in their derivation) and hence will yield solutions which have accumulated error after several steps. To ensure that equilibrium is being satisfied at each step, one may use a modified Newton-Raphson method to perform equilibrium iteration. The iterative equations take the form

$$[M]\{\ddot{u}^{t+\Delta t}\}^{(i)} + [C]\{\dot{u}^{t+\Delta t}\}^{(i)} + [K^t]\{\Delta \Delta u\}^{(i)} = \{R^{t+\Delta t}\}^{(i-1)} - \{F^{t+\Delta t}\}^{(i-1)} \quad (2)$$

where (i) denotes the i^{th} equilibrium iteration and $\{\Delta u\}^{(i)}$ denotes the correction to the displacement increment at iteration i , that is,

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

$$\{u^{t+\Delta t}\}^{i+1} = \{u^t\}^{i+1} + \{\Delta u\}^{i+1}$$

It should be pointed out that in a general Newton type iteration, the tangent stiffness matrix $[K^t]$ would be updated at each iteration (i.e., use $[K^{t+\Delta t}]^{(i)}$); however, in the modified Newton method, this term is held constant during the iteration (or updated only periodically when convergence deteriorates). Input parameters allow the program user to specify the updating frequency.

The time integration procedure used in dynamic analyses is either the Wilson θ or the Newmark β methods.

The original NONSAP program utilized both the total Lagrangian and updated Lagrangian formulations in developing the incremental equilibrium equations. All element and material models developed during the current research program have been restricted to the total Lagrangian formulation; however, those material models in the original NONSAP program which permitted the use of the updated Lagrangian solution have been retained and are still available in AGGIE I.

Element Library

Several types of elements are currently available in the AGGIE I program. These include a three-dimensional truss element, two-dimensional isoparametric solid element with 4 to 8 nodes, and a three-dimensional isoparametric solid element with 8 to 21 nodes. The two-dimensional element is reducible to a constant strain triangular element and the three-dimensional element is reducible to a constant strain tetrahedron.

The truss element is shown in Fig. 1 and has in general three degrees-of-freedom per node. Material properties and cross-sectional area are assumed to be constant over the length of any element. The element may be used in linear elastic or nonlinear elastic analyses in combination with small or large displacements (for a large displacement analysis, only the updated Lagrangian formulation is available).

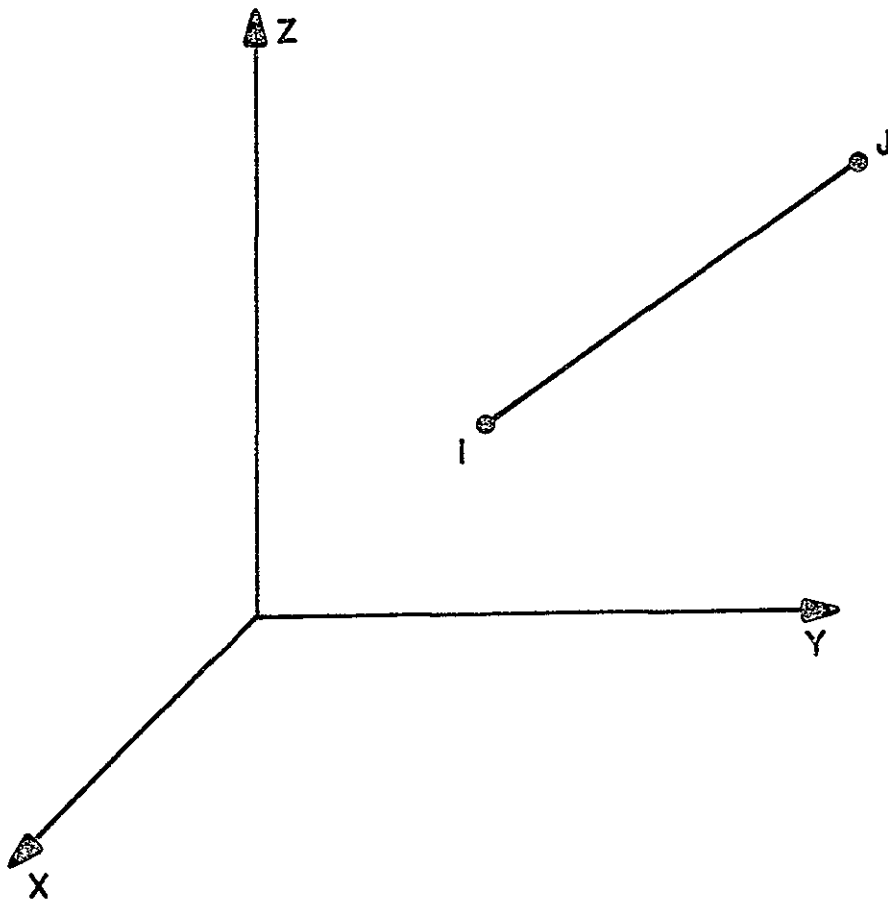
The general two-dimensional isoparametric solid element is shown in Fig. 2. The element may be used for plane stress, plane strain or axisymmetric analyses. The material models which are available for the two-dimensional element are shown in Fig. 2 and are described briefly in the next section. This element may be used to model axisymmetric shells; however, because of the continuum formulation used, the element does not represent Kirchhoff-type, thin shell behavior very well.

The general three-dimensional isoparametric solid element is shown in Fig. 3. The element may have from 8 to 21 nodes with 3 degrees-of-freedom per node. The material models which are currently available for the three-dimensional element are also listed in Fig. 3.

Material Models

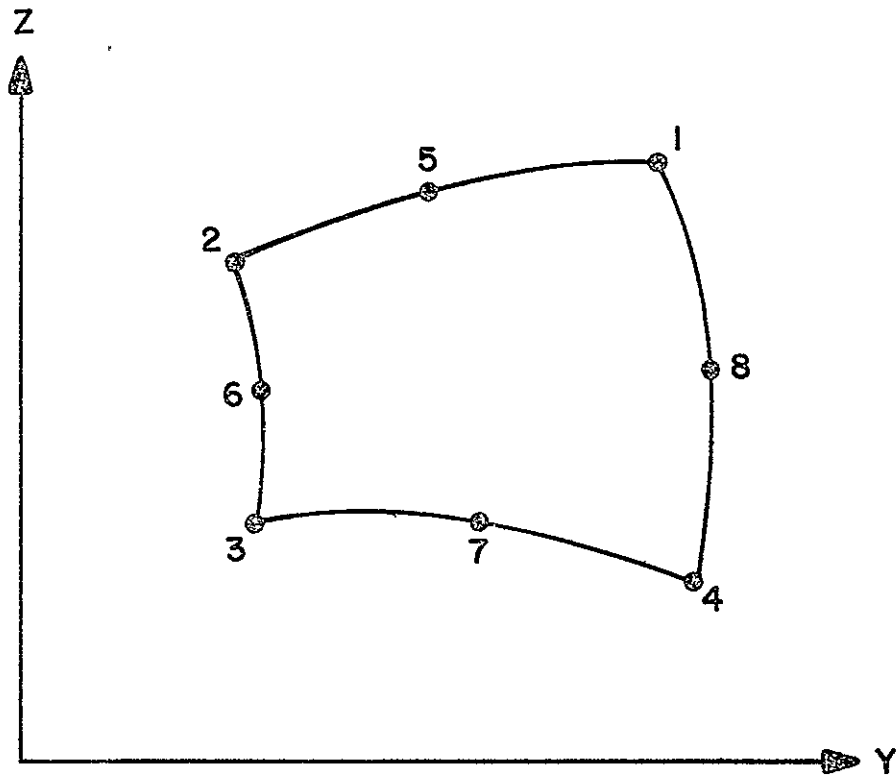
The original NONSAP program contained a number of material models for the two-dimensional elements but contained only a linear and nonlinear elastic material model for the three-dimensional elements. In the AGGIE program, new material models have been added for the two-dimensional elements and four material models have been added for the three-dimensional elements. Table 1 gives a complete listing of the available material models.

For the truss elements, two models are available: 1) linear elastic material and 2) nonlinear elastic material. The nonlinear elastic-material model makes use of a user-specified, piece-wise linear stress-strain curve.



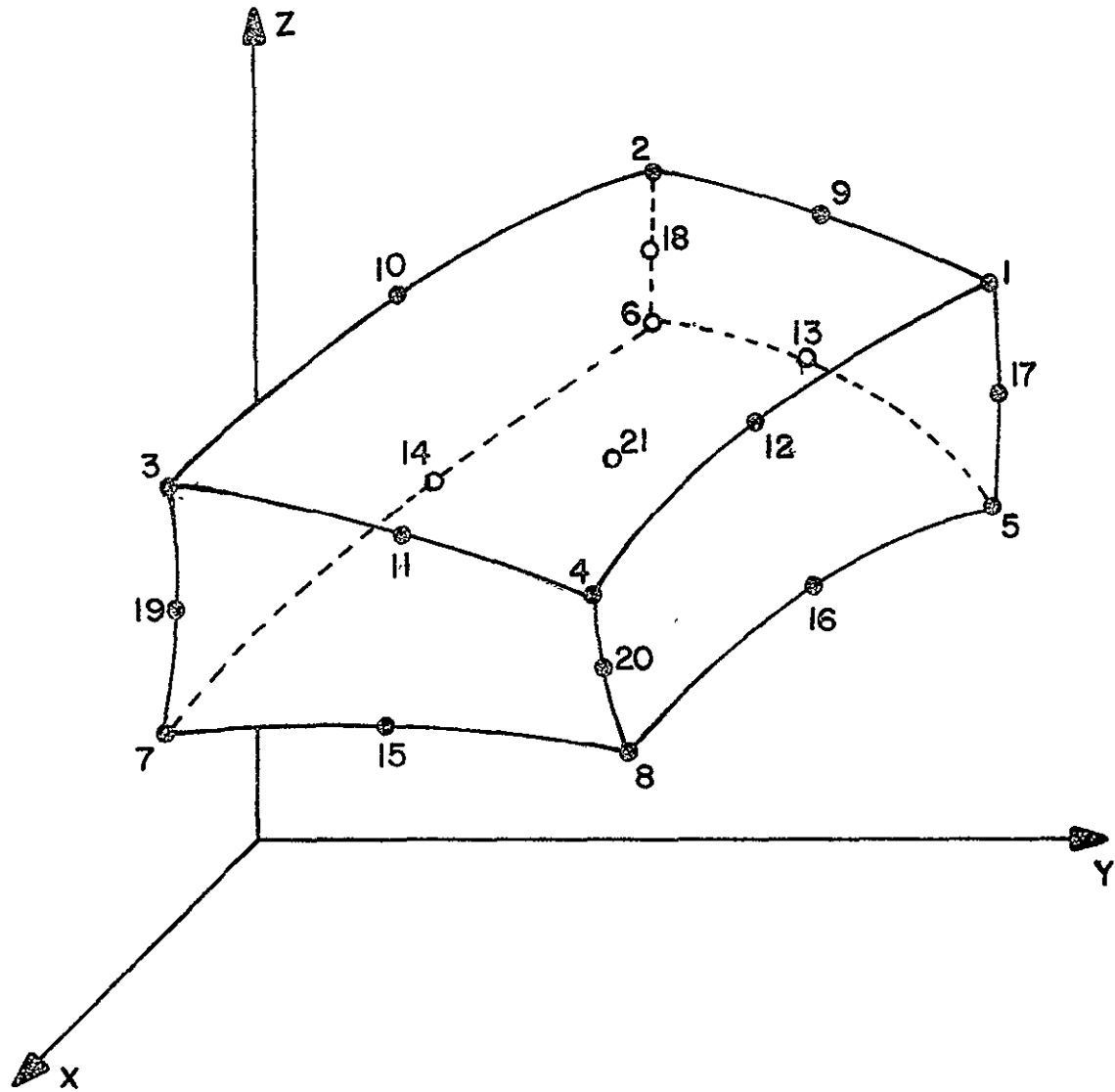
Available Material Models	Geometric Nonlinear Formulations		
	Linear	Total Lagrangian	Updated Lagrangian
1. linear elastic	X		X
2. nonlinear elastic	X		X

Fig. 1 Three-Dimensional Truss Element



Available Material Models	Geometric Nonlinear Formulation		
	Linear	Total Lagrangian	Updated Lagrangian
1. linear elastic, isotropic	X	X	X
2. linear elastic, orthotropic	X	X	
3. variable tangent moduli model	X	X	
4. curve description model (plane strain, axisymmetric)	X		
5. curve description model with tension cutoff	X		
6. elastic plastic, combined kinematic-isotropic hardening, von-Mises yield condition	X	X	
7. elastic-plastic, Drucker-Prager yield condition	X		
8. incompressible, nonlinear elastic, Mooney-Rivlin material (plane stress)	X	X	
9. incompressible, nonlinear elastic, Mooney-Rivlin material (plane strain or axisymmetric)	X	X	
10. elastic-plastic, mechanical sublayer model, von-Mises yield condition	X	X	

Fig. 2 Two-Dimensional Isoparametric Continuum Element for Plane Stress, Plane Strain or Axisymmetric Analysis



Available Material Models	Geometric Nonlinear Formulation		
	Linear	Total Lagrangian	Updated Lagrangian
1. linear elastic, isotropic	X	X	
2. linear elastic, orthotropic	X	X	
3. curve description model	X	X	
4. elastic-plastic, combined kinematic-isotropic hardening von-Mises yield condition	X	X	
5. elastic-plastic, mechanical sublayer model, von-Mises yield condition	X	X	
6. incompressible nonlinear elastic, Mooney-Rivlin material	X	X	

Fig. 3 Three-Dimensional Isoparametric Continuum Element

Table 1. Material Models

Model	1-D	2-D	3-D
1	linear elastic	linear elastic, isotropic	linear elastic, isotropic
2	nonlinear elastic	linear elastic, orthotropic	linear elastic, orthotropic
3		variable tangent moduli model	curve description model
4		curve description model (plane strain, axisymmetric)	elastic-plastic, combined kinematic-isotropic hardening, VonMises yield condition
5		curve description model with tension cutoff	elastic-plastic, mechanical sublayer model, Von-Mises yield condition
6		elastic-plastic, combined kinematic-isotropic hardening, Von Mises yield condition	incompressible nonlinear elastic, Mooney-Rivlin material
7		elastic-plastic, Drucker-Prager yield condition	
8		incompressible, nonlinear elastic, Mooney-Rivlin material (plane stress)	
9		incompressible, nonlinear elastic Mooney-Rivlin material (plane strain or axisymmetric)	
10		elastic-plastic, mechanical sublayer model, Von Mises yield condition	

For the two-dimensional solid elements, a number of models are available as shown in Table 1. Most of the material models may be used with either a geometrically linear (small strain) or nonlinear (large strain) formulation.

Two linear material models are available, i.e., either isotropic or orthotropic.

A newly developed nonlinear elastic (hyperelastic) incompressible model which makes use of Mooney-Rivlin material constants to describe the nonlinear stress-strain relation may be used for the analysis of some rubber-like materials. The plane-strain and axisymmetric models make use of Lagrange multiplier constraints in order to impose the incompressibility condition.⁵

Two new elastic-plastic material models provide a wide range of hardening rules which may be used in elastic-plastic analyses. Both models are based on the von Mises yield condition and the associated flow rule. For the hardening rule, one model makes use of a combined kinematic-isotropic hardening model while the second is based on a mechanical sublayer (Besseling) model. The combined kinematic-isotropic hardening model allows for the input of a yield surface size vs. equivalent uniaxial plastic strain curve if such data is available; otherwise this information is generated by the program for a user-specified value of the ratio of kinematic-to-isotropic hardening. Both models make use of the von Mises yield condition and associated flow rule. Details of both models may be found in Ref. 4.

Three geological material models originally in NONSAP have been retained in the present program. These are the variable tangent moduli model, the curve description model and the curve description model with tension cutoff.³ The variable tangent moduli model describes an isotropic

material in which the bulk and shear moduli are functions of the stress and strain invariants and the functional relationship used replaces an explicit yield conditions. In the curve description model, the instantaneous bulk and shear moduli are defined by piecewise linear functions of the current volume strain. With the tension cutoff model, tensile stresses due to applied loading cannot exceed the gravity in-situ pressure; the model assumes reduced stiffness in the direction of the tensile stress which exceed in magnitude the gravity pressure (i.e., when a "crack" develops).

The material models available for the three-dimensional solid elements are also shown in Table 1. Several new models were developed and implemented in the base program. These include the Mooney-Rivlin incompressible model, the linear orthotropic model, the elastic-plastic model with combined kinematic-isotropic hardening, and the elastic-plastic model with mechanical sublayer models. Each of these models may be used in either a geometrically linear analysis or a large strain analysis based on the total Lagrangian formulation.

Program Organization

The general program organization which was utilized in the base program NONSAP² has been retained in the present program. The significant differences are in the way the element matrices are assembled into the structural matrices and the allocation of the high-speed storage.

As in NONSAP, the solution process is divided into three phases:

1. Input and Mesh Generation Phase. Various control information is first read from data. Depending upon which input option is selected, the node and element information is either read from data or generated internally and then stored on tape or disk. Additionally, equation numbers

for all active degrees of freedom are generated and element-node connectivity arrays are determined and stored on disk. Externally applied pressure loads are converted to nodal forces and stored on disk.

2. Assemblage of Constant Structural Matrices. The stiffness, mass and damping matrices for all linear elements are computed, assembled and stored on disk.

3. Step-by-Step Solution. The incremental equations of equilibrium are solved for each step. At each step, equilibrium iteration is performed (if desired) until a converged solution is obtained. Displacements, velocities, accelerations, stresses and strains are computed and printed.

The program is designed to process elements by groups; all elements within an element group have like characteristics, i.e., they are all the same element type (e.g., 2-D axisymmetric), they all have the same type of geometric nonlinearity and the same material model. Each element group generally calls for a specific overlay segment of the program. Program efficiency is increased by grouping elements to avoid unnecessary or repeated calling of overlay segments. Element groups may be either linear (geometrically and materially linear) or nonlinear. During all phases, linear element groups are processed first. Stiffness properties for all linear element groups are computed during phase 2 and stored on disk. During phase 3, the structural stiffness for all linear element groups is read from disk; then stiffness matrices for all nonlinear element groups are computed (based on the current stress strain-state) and added to the linear contribution.

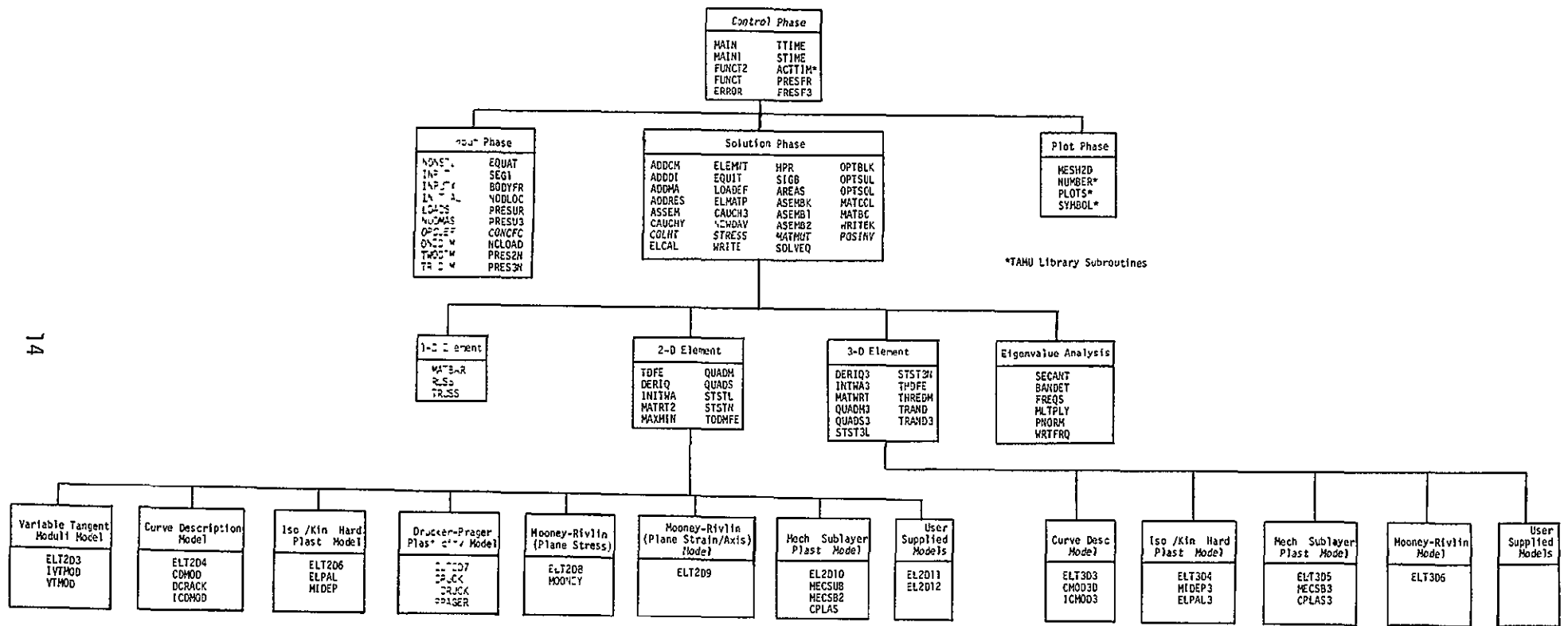
The program is designed with the idea of making maximum use of available high speed memory. A common block /AMAT/ dimensioned for MTOT locations is used to store the arrays that are required during the different

phases of program execution. During the input phase, space (NUMEST locations) is allocated in the common block for storage of element property data (element coordinates, element-node connectivity, material properties and working arrays for element stresses, strains and various variables associated with nonlinear material models). For each element group, the input is processed using the same NUMEST locations and necessary information is written on disk. During the solution phase, the element property data is read from disk and the same space is used repeatedly as each element group is processed. The program automatically allocates the NUMEST locations within the common block based on the storage requirements of the largest element group. During the input phase, the program determines the storage that will be required for the structural coefficient (e.g., stiffness in static analyses) matrix and the space (LBLOCK) that will be available with the common block. If sufficient space is available in the common block to store the coefficient matrix in core, then this is done. If sufficient space is not available, then the coefficient matrix is stored on disk in blocks of length LBLOCK and then processed block-by-block within the LBLOCK locations of common block AMAT. More detailed information on storage allocation may be found in the Programmer's Guide for the program.

As mentioned before, the AGGIE program contains over 125 subroutines and is overlaid. The overlay structure currently being used is shown in Table 2 (although additional overlaying may be possible and desirable).

Equation Solvers

The solution of the incremental equilibrium equations is obtained using either the in-core linear equation solver OPTSOL⁷ or the out-of-core linear equation solver OPTBLK.⁸ The coefficient matrix (left side



14

Table 2

Overlay Structure of AGGIE I Program

ORIGINAL PAGE IS
OF POOR QUALITY

of the equations) is stored as a one-dimensional array using a skyline storage scheme (by columns) i.e., only the terms in each column which fall below the skyline are stored. If the coefficient matrix will fit within the high-speed memory space which has been allocated to it, then the solution of the equations is done in-core by OPTSOL. If the coefficient matrix will not fit entirely in-core, then it is stored on disk, tape or other low-speed storage device by blocks and solved by OPTBLK in a block-by-block fashion. The blocksize is selected as large as possible depending upon available high-speed memory.

3. FUTURE IMPROVEMENTS

The AGGIE program is currently being extended to include the following improvements:

1. General 3-D thin shell element
2. General 3-D beam element
3. Thermal strains
4. Non-isothermal, combined kinematic-isotropic hardening model with temperature dependent material properties
5. Creep models
6. Nonlinear viscoelasticity

Additional improvements to be added in the future include: improved mesh generation, non-zero boundary conditions, and additional element geometries.

Users interested in obtaining further information on the status of current program improvements should contact: Dr. Walter E. Haisler, Aerospace Engineering Department, Texas A&M University, College Station, Texas 77843, Phone (713) 845-7541.

4. PROGRAM INPUT

This section describes the required input data for the AGGIE program. The input data is separated into 16 sections; some of these are generally not required for a particular problem. Multiple cases may be run by inputting the required input sequentially for each case. The data set for the last problem case should be followed by two blank cards; however, omission of these will only cause an end-of-file read error when the program tries to process the non-existent data set. An error in one input data set will cause the program to terminate and not process any data sets which might follow.

The input instructions here should be used in conjunction with the example problems and deck setups shown in a later section. Additionally, a section follows which outlines some of the known limitations of the program and gives some suggestions which may be useful when difficulties are encountered in using the program.

The input data consists of the following sections:

- I. Heading Card
- II. Program Control Cards
- III. Nodal Point Data
- IV. Truss Elements
- V. Two-Dimensional Solid Elements
- VI. Three-Dimensional Solid Elements
- VII. Node and Element Input by Substructures
- VIII. Load Curves
- IX. Nonconservative Pressure Loads
- X. Conservative Pressure Loads
- XI. Concentrated Nodal Forces
- XII. Rayleigh Damping
- XIII. Concentrated Nodal Masses
- XIV. Concentrated Nodal Dampers
- XV. Initial Conditions
- XVI. Frequency Solution Data Card

As noted previously, the AGGIE program is an extension of the NONSAP program. Hence, some sections of the input described here are necessarily the same as described in the NONSAP user's manual² (with only minor modifications). This is the case for input instructions which refer to program segments and material models which were originally in NONSAP and were left intact and whose input instructions were unchanged, e.g., nodal point input, all linear and geological material models, initial conditions, and concentrated mass and damping input. These input sections are marked with an asterisk (*).

I. HEADING CARD (20A4)

column	variable	description	notes
1-80	HEAD (20)	Heading information which will appear on output.	(1)

NOTES

- (1) Any information placed on this card will appear on the title page of the output.

II. PROGRAM CONTROL CARDS

II.1 Solution Control Variables (6I1, I4, 4I5, 2F10.0, 3I5, 6I2)

columns	variable	description	notes
1	IDOF(1)	Master X-translation code EQ.0 - admissible EQ.1 - deleted	(1)
2	IDOF(2)	Master Y-translation code	
3	IDOF(3)	Master Z-translation code	
4	IDOF(4)	Master X-rotation code	
5	IDOF(5)	Master Y-rotation code	
6	IDOF(6)	Master Z-rotation code	
7-10	NUMEG	Number of substructures or element groups	(2)
11-15	MGO	Mesh generation option EQ.0 - Individual node and element data input EQ.1 - Mesh described by substructures	(3)
16-20	MODEX	Solution mode EQ.0 - Data check only EQ.1 - Execution EQ.2 - Restart	(4)
21-25	IDINC	Displacement incrementation Flag EQ.0 - Load incrementation EQ.1 - Displacement incrementation	(5)
26-30	NSTE	Number of solution steps	(6)
31-40	DT	Load (time) step increment	(6)
41-50	TSTART	Load (time) value at which solution starts	(7)
51-55	IMASS	Flag indicating static or dynamic analysis EQ.0 - static analysis EQ.1 - dynamic analysis with lumped, diagonal mass EQ.2 - dynamic analysis with consistent mass matrix	(8)
56-60	IEIG	Frequency solution flag EQ.0 - no frequency solution EQ.1 - perform frequency analysis	(9)
61-65	IPL0T	Plot tape flag EQ.0 - no plot tape EQ.1 - plot tape generated	(10)

II. PROGRAM CONTROL CARDS (continued)

columns	variable	description	notes
66-67	ICOMB(1)	Substructure combining flags. Used with substructure mesh generation only (MGO=1).	
68-69	ICOMB(2)		
69-70	ICOMB(3)		
70-71	ICOMB(4)		
71-72	ICOMB(5)		
76-77	ICOMB(6)	EQ.0 - Program generates one element group for every substructure. GT.0 - Program combines substructures ICOMB[I(odd)] to ICOMB[I+1(even)] into one element group. Three sets of combinations are allowed. Leave blank if no substructure combining is desired.	

NOTES

- (1) The master translation codes are used to delete nodal degrees-of-freedom (dof) which are not required for this problem. The program allows for six degrees of freedom per node; however, since many problems do not require the complete set, these master translation codes are used to delete these dof and thereby conserving storage and execution time. Any later referenc to a deleted degree-of-freedom is ignored by the program.

All elements currently available in this program allow for translational degrees-of-freedom only and; hence, IDOF (I) for I=4,5 and 6 must equal 1. The 2-D continuum element is assumed to lie in the (Y-Z) plane and; hence, IDOF(1) must equal 1 for this case.

- (2) The input of element information is handled by element groups each of which has like characteristics as discussed later. If the nodal mesh is described by substructures (MGO=1), then NUMEG represents the number of substructures by which the complete structure geometry is described and which are read in Section VII. If the nodal mesh is input by individual nodes, then NUMEG represents the number of element groups to be input in Sections IV - VI. In either case, the elements comprising an element group or substructure must have the same characteristics, i.e., element type, type of analysis, material model and numerical integration order (these are described by the NPAR card in Section V.1 or VI.1).
- (3) The nodal coordinates and element-node connectivity may be

II. PROGRAM CONTROL CARDS (continued)

described by two approaches. The node numbers and coordinates may be input individually as described in Section III. Alternately, the geometry and element-node connectivity may be generated by describing "substructures" and letting the program automatically generate internal node numbers, coordinates, element numbers and element-node connectivity information. This approach is described in Section VII.

- (4) In the data check mode, all input is read, partially checked for validity, and printed. The restart mode is used to restart a problem for which a prior solution was obtained and a restart tape written. Detailed instruction on restart jobs is contained in Section PROGRAM RESTART.
- (5) For the usual case, load (time) is considered to be the independent variable and, hence, the load is incremented from zero to the final value. However, in static buckling problems it may be more convenient to let some characteristic displacement be the independent variable and increment this displacement (and solve for the corresponding load).
- (6) For dynamic problems, the time step DT must be selected to maintain accuracy and stability. The total time for which a solution is obtained is given by $NSTE*DT$. The loads must be specified over this time range and are described by force vs. time tables input in Section VIII.

For static problems the load increment DT is used to increment a load parameter. The loads are similarly described in terms of force vs. load parameter tables.

For load incrementation, the load (time) increment DT is not used since the incrementation is controlled by the displacement increment $DINC$ input in Section II.7.
- (7) Normally, $TSTART$ is set to zero. However, when restarting a problem, $TSTART$ is set to the value of the load parameter (or time) at which the solution is to restart and for which restart information is available on tape from a previous run.
- (8) For dynamic analyses, inertia effects are accounted for and, in addition, damping may be included.
- (9) Mode shapes and frequencies are computed prior to the dynamic solution if $IEIG = 1$ (Note - this part of the program is not operational in this version of AGGIE).
- (10) The program prepares a plot tape of initial mesh geometry. Subroutine $MESH2D$ utilizes the plot tape in combination with several TAMU Calcomp library programs to plot two-dimensional meshes. Although a plot tape is created for three-dimensional meshes, no plotting routines are included. Users utilizing the plot tape will undoubtedly have to interface the program with their installations plotting and graphics packages.

II. PROGRAM CONTROL CARDS (continued)

II.2 Nonlinear Solution Variables (4I5, F10.0)*

columns	variable	description	notes
1-5	ISREF	Number of load steps between reformation of stiffness matrix. If EQ.0 - default reset to 1.	(1)
6-10	NUMREF	Number of allowable stiffness reformations in each step. EQ.0 = default reset to 1	(2)
11-15	IEQUIT	Number of steps between equilibrium iterations. EQ.0 - default reset to 1	(3)
16-20	ITEMAX	Maximum number of equilibrium iterations permitted before reforming stiffness EQ.0 - default reset to 15.	(2)
21-30	RTOL	Tolerance used to measure equilibrium convergence. EQ.0 - default reset to 0.001	(4)

NOTES

- (1) For nonlinear element groups (materially or geometrically nonlinear), the stiffness matrix is reformed every ISREF load (time) steps based on information at the end of the previous step. In addition, history dependent variables are updated for use in any history dependent material law.
- (2) After ITEMAX equilibrium iterations in any load step, the stiffness matrix is reformed if NUMREF. GE.2 and iteration continued. The stiffness is always reformed at the beginning of the step prior to the first iteration (for those steps where reformation is to be performed) and then reformed NUMREF times with ITEMAX iterations between each reformation.
- (3) For nonlinear problems, equilibrium iteration is performed every IEQUIT steps until convergence of the displacements is obtained. For example, if IEQUIT is set to 3, then iteration is performed during load steps 3, 6, 9, etc. only until the solution converges to a tolerance of RTOL. If convergence does not occur within ITEMAX iterations, then the stiffness is reformed (if NUMREF.GT.1) and iteration continued; however, if stiffness reformation is not allowed then the solution is stopped.

*An asterisk is used to mark those sections of the input which are identical to the NONSAP Program

II. PROGRAM CONTROL CARDS (continued)

- (4) During equilibrium iteration, convergence of the solution is determined by the change in the Euclidean norm of the displacement vector, i.e., when

$$\frac{|| u^{(n-1)} || - || u^{(n)} ||}{|| u^{(n)} ||} \leq \text{RTOL}$$

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

II. PROGRAM CONTROL CARDS (continued)

II.3 Load Control Variables (5I5)

columns	variable	description	notes
1-5	NLCUR	Number of load curves to be input EQ.0 - Linear load curve in internally generated from load value TSTART to TSTART + DT*NSTE. GT.0 - Number of load curves to be read from data	(1)
6-10	NPRES	Number of conservative pressure load cards to be input	(2)
11-15	NPRESN	Number of nonconservative pressure load cards to be input	(2)
16-20	NCONCF	Number of concentrated nodal force cards to be input	(2)
21-25	NBODY	Body force flag EQ.0 - no body forces EQ.1 - body forces to be included (2-D only)	(3)

NOTES

- (1) The load curves describe the variation of the load functions with load parameter (time) and are input in Section VIII.

For displacement incrementation (IDINC.EQ.1), no load curves are input (set NLCUR.EQ.0). In displacement incrementation, the load curve multipliers are computed as part of the solution.

- (2) Conservative and nonconservative (deformation dependent) pressure loads applied to the face of an element or substructure and concentrated nodal forces are input in Sections IX - XI.

Nonconservative loads are not allowed for displacement incrementation (IDNIC.EQ.1).

- (3) Body forces may be applied to 2-D elements and are assumed to act in the global Z direction.

II. PROGRAM CONTROL CARDS (continued)

II.4 Print Control Variables (815)

columns	variable	description	notes
1-5	IPRI	Output printing interval EQ.0 - default set to 1	(1)
6-10	NPB	Number of displacement print blocks EQ.0 - print all nodal components GT.0 - print according to table (NPB.LE.8)	(2)
11-15	IDC	Displacement printout code EQ.0 - no displacement printout EQ.1 - print displacements	(3)
16-20	IVC	Velocity print out code EQ.0 - no velocity printout EQ.1 - print velocities	(3)
21-25	IAC	Acceleration print out code EQ.0 - no accelerations printed EQ.1 - print accelerations	(3)
26-30	IPGN	Controls printing of generated nodal data EQ.0 - print all generated nodes EQ.1 - no print out	(4)
31-35	IPGL	Controls printing of generated nodal force components EQ.0 - print all generated nodal force components EQ.1 - no print out	(5)
36-40	IRITRS	Restart tape flag EQ.0 - no restart tape written EQ.1 - Restart tape written	(6)
41-45	IPSM	Controls printing of storage information messages EQ.0 - print all messages EQ.1 - no print out	(7)

NOTES

- (1) Program results will be printed every IPRI steps. Additional print options controlling output of displacements, velocities, accelerations, strains and stresses are specified by the variables NPB, IDC, IVC, and

II. PROGRAM CONTROL CARDS (continued)

IAC on this card and on the element cards (Sections V.1 or VI.2).

- (2) Print blocks may be used to selectively print blocks or groups of nodal displacements (and velocities and accelerations). Each block of nodes is defined by the node number of the first and last node in the block (see card II.5)
- (3) If NPB.EQ.0, then all displacements are printed regardless of the values of IDC, IVC and IAC. If NPB.NE.0, then IDC, IVC, and IAC must be input as 1 if displacements, velocities and accelerations, respectively, are to be printed.
- (4) When using the substructure input of nodal and element data (MGO = 1), this flag may be used to suppress the output of nodes automatically generated by the program. It is strongly suggested that the generated nodes be output and checked (for at least the first run of a problem).
- (5) When pressure loads are input, the program transforms these to generalized nodal forces. This flag may be used to suppress the output of these nodal forces.
- (6) A restart tape(s) which contains solution information for the final solution step may be written for future use in a restart of a problem. Information regarding these restart tapes and the setup of a restart is contained in Section PROGRAM RESTART.
- (7) The program uses "dynamic storage allocation" to allocate space for several key variables in common block AMAT. During the course of a solution, messages are printed which reflect how much storage is being used for each phase of the solution.

II. PROGRAM CONTROL CARDS (continued)

II.5 Print Block Table (16I5)*-OMIT THIS CARD IF NPB.EQ.0

columns	variable	description	notes
1-5	IPNODE(1,1)	First node of print block no. 1	(1)
6-10	IPNODE(2,1)	Last node of print block no.1	
11-15	IPNODE(1,2)	First node of print block no. 2	
	etc.		

NOTES

- (1) The entries for each print block are input in pairs, i.e., the first and last node of the print block. Nodal information is printed at all nodes having node numbers between each pair of node numbers forming the print block. IF NPB.EQ.0, OMIT THIS CARD.

II. PROGRAM CONTROL CARDS (continued)

II.6 Mass And Damping Control Parameters*(4I5, 2F10.0)-OMIT THIS CARD IF IMASS.EQ.0

columns	variables	description	notes
1-5	IDAMP [†]	Flag indicating damping type EQ.0 - no Rayleigh damping EQ.1 - Rayleigh damping	(1)
6-10	IMASSN [†]	Number of concentrated nodal masses	(2)
11-15	IDAMPN [†]	Number of concentrated nodal dampers	(2)
16-20	IOPE	Time integration method used EQ.0 - default reset to 1 EQ.1 - Wilson's theta method EQ.2 - Newmark's method	(3)
21-30	OPVAR(1)	First integration parameter 1) If IOPE.EQ.1, parameter is θ EQ.0 - default reset to 1.4 2) If IOPE.EQ.2, parameter is γ EQ.0 - default reset to 0.5	(3)
31-40	OPVAR(2)	Second integration parameter 1) if IOPE.EQ.1, parameter is not applicable 2) if IOPE.EQ.2, parameter is β EQ.0 - default reset is specified in note (3)	(3)

NOTES

- (1) In a dynamic analysis (IMASS.GT.0), Rayleigh damping may be included. The damping coefficients are defined in Section XII.
- (2) Concentrated nodal masses and dampers may be added if desired. These are input in Section XIV.
- (3) For a dynamic analysis, the time integration method must be selected. The variables OPVAR(1) and OPVAR(2) contain the values of the parameters required in the time integration method specified by IOPE.

The Newmark θ -method requires one parameter (θ) which is normally set to 1.4.

The Wilson β -method uses the parameters γ and β . For linear analyses $\gamma = 1/2$ gives rise to no numerical damping and $\beta = 1/4$ corresponds to constant acceleration over a time step. For linear, unconditional stability, then $\gamma \geq 1/2$ and $\beta \geq (\gamma + 1/2)^2/4$. If β is not specified (input as zero), then β is set to the default value given by the linear, stability criterion.

[†] Not currently operational; must be input as zero.

II. PROGRAM CONTROL CARDS (continued)

II.7 Displacement Incrementation Variables (2I5, F10.0)-OMIT THIS CARD IF IDINC.EQ.0

columns	variable	description	notes
1-5	ININC	Node number where degree-of-freedom is to be incremented	(1)
6-10	IDIR	Displacement direction EQ.1 - X direction EQ.2 - Y direction EQ.3 - Z direction	(2)
11-20	DINC	Displacement increment	(3)

NOTES

- (1) The global node number where a displacement increment is to specified is required. If the substructure option for mesh generation (MGO=1) is being used, the required node number will not be known until a data check run is made and the automatically generated node numbers printed by the program.
- (2) IDIR refers to the global direction of the degree-of-freedom at a node.
- (3) The magnitude of the displacement increment is required; it may be positive or negative.

NODE AND ELEMENT INPUT

The input data required to specify the node numbers, nodal coordinates, element numbers, element-node connectivity and element material properties is different depending upon which input option (MGO) is selected.

If the individual nodal coordinates and element-node connectivity (MGO=0) are to be input, then the instructions in Sections III-VI should be followed.

If the nodal and element geometry are being automatically generated using the substructure option (MGO=1), then the input instructions in Section VII should be followed.

III. NODAL POINT DATA (MGO=0)

The input instructions in Section III-VI are used only if MGO=0.

III.1 Node and Element Group Data (3I5)

column	variable	description	notes
1-5	NUMNP	Total number of nodal points	(1)
6-10	NEGL	Number of linear element groups EQ.0 - all elements are nonlinear	(2)
11-15	NEGNL	Number of nonlinear element groups EQ.0 - all elements are linear	(2)

NOTES

- (1) NUMNP is the total number of nodes to be defined in Section III.2.
- (2) Elements are input by groups each of which have like characteristics (same element type, material model, type of geometric nonlinearity, and integration order). Linear elements are geometrically and materially linear and hence their stiffness matrices are constant and formed only once. Nonlinear elements have their stiffness properties continually updated during the solution.

The elements may be divided into groups for three reasons: (1) for input convenience - A particular group of elements may lend itself to being called a group simply because of the geometry of the structure, because of the node numbering, etc. Each element is assigned ascending element numbers from 1 to the total number of elements in that group. (2) There may be several regions each of which have different material properties, different element types, etc. and each has to be treated as a different group. (3) Conservation of dynamic storage. Each element group uses repeatedly the same storage space in common block AMAT and hence less space is required if the the total number of elements is divided into several groups.

For computational economy, all linear element groups are input together (first) and all nonlinear element groups are input together (after the linear element groups).

III. NODAL POINT DATA (continued)

III.2 Nodal Point Data (A1, I4, A1, I4, 5I5, 3F10.0, I5)

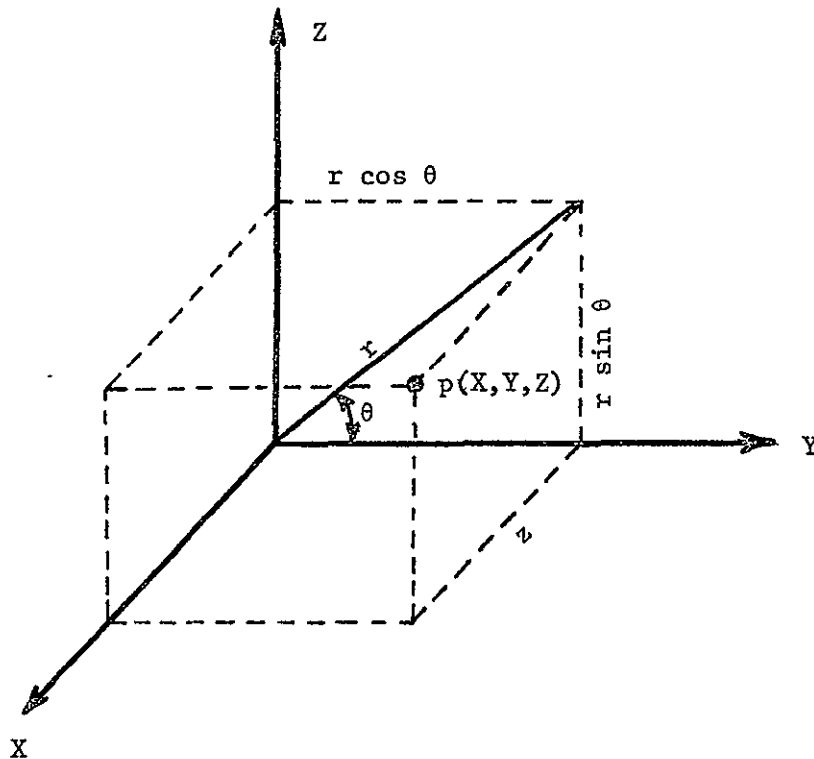
column	variable	description	notes
1	CT	Symbol describing the coordinate system for this node EQ. - (blank) cartesian (X,Y,Z) EQ.X - X-cylindrical	(1)
2-5	N	Node (joint) number GE.1 and LE.NUMNP	(2)
6	PSF	Print suppression flag (ignored unless N.EQ.1) EQ. - (blank) no suppression EQ.A - Suppress ordered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

NOTES

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

III. NODAL POINT DATA (continued)

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



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

- (2) Nodal data must be defined for all (NUMNP) nodes. Node data may be input directly (i.e., each node on its own individual card) or the generation option may be used if applicable (see note 6, below). Admissible node numbers range from "1" to the total number of nodes (NUMNP). Node numbers may not be repeated or omitted. The last node that is input must be "NUMNP".
- (3) The print suppression flag (PSF) is used to eliminate the second printing of ordered node coordinates or to suppress printing of equation number assignments (or both). The PSF character is entered on the card for node one (1) only.
- (4) Boundary condition codes can only be assigned the following values ($M=1,2,\dots,6$) --

III. NODAL POINT DATA (continued)

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

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

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

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

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

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

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

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

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

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

III. NODAL POINT DATA (continued)

coordinate data (if input on the card) are printed as read in the first list of node coordinates. The second listing of coordinates (if not suppressed with a PSF or "A" or "C") is an ordered printing of all nodes with coordinates converted to the (X,Y,Z) system. Note that boundary condition codes always reference the (X,Y,Z) system even if the node happens to be located using the cylindrical coordinate option.

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

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

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

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

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

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

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

As an example of the use of the node generation parameter, consider the sketch below with nodes labeled as shown:

1	5	9	13	17	21	25
2	6	10	14	18	22	26
3	7	11	15	19	23	27
4	8	12	16	20	24	28

III. NODAL POINT DATA (continued)

The nodal data may be generated using the following input:

Card 1: N=1, KN=4	}	Generates nodes 5, 9, 13, 17, 21
Card 2: N = 25, KN = 0		
Card 3: N = 2, KN = 4	}	Generates nodes 6, 10, 14, 18, 22
Card 4: N = 26, KN = 0		
Card 5: N = 3, KN = 4	}	Generates nodes 7, 11, 15, 19, 23
Card 6: N = 27, KN = 0		
Card 7: N = 4, KN = 4	}	Generates nodes 8, 12, 16, 20, 24
Card 8: N = 28, KN = 0		

In using the above, one must be careful so that the generated boundary codes are the ones that are desired. In the above example, the boundary conditions specified on card 1 will be applied to the generated nodes, 5, 9, 13, 17, and 21.

IV. TRUSS ELEMENTS *

TRUSS elements are two-node members allowed arbitrary orientation in the X,Y,Z system. The TRUSS transmits axial force only, and in general is a six (6) degree of freedom element (i.e., three global translation components at each end of the member, see Fig. 4).

IV.1. Element Control Card (20I4)

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

NOTES

- (1) TRUSS element numbers begin with one (1) and end with the total number of elements in this group, NPAR(2). Element data are input in Section IV.4, below.
- (2) The parameter NPAR(3) is applicable only if the element group is nonlinear. If NPAR(3).EQ.1, no geometric nonlinearities are taken into account, i.e. the geometric stiffness matrix is not included. If NPAR(3).EQ.2 large displacement effects are included in the analysis, but small strains are assumed in the calculation of element forces.
- (3) In any one element group only one material model can be used, and

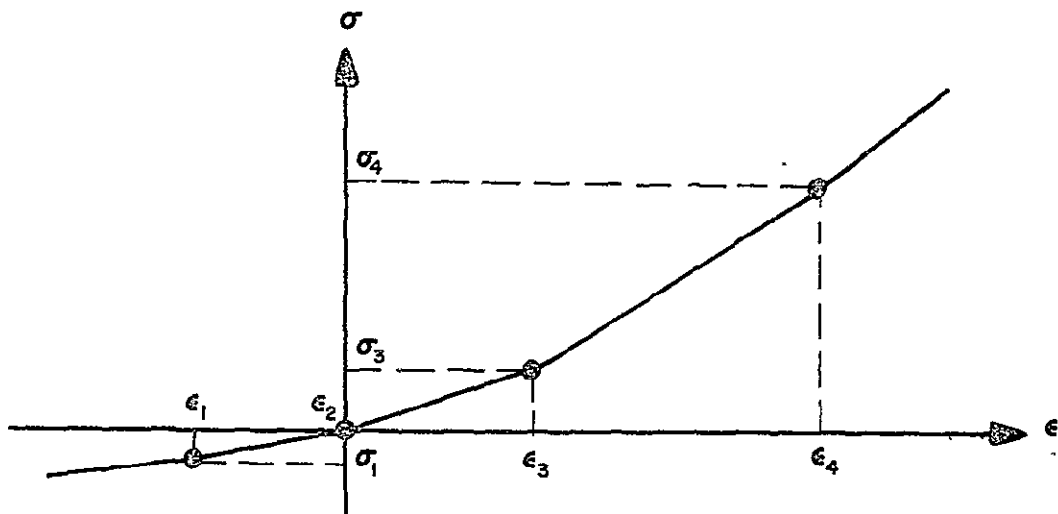
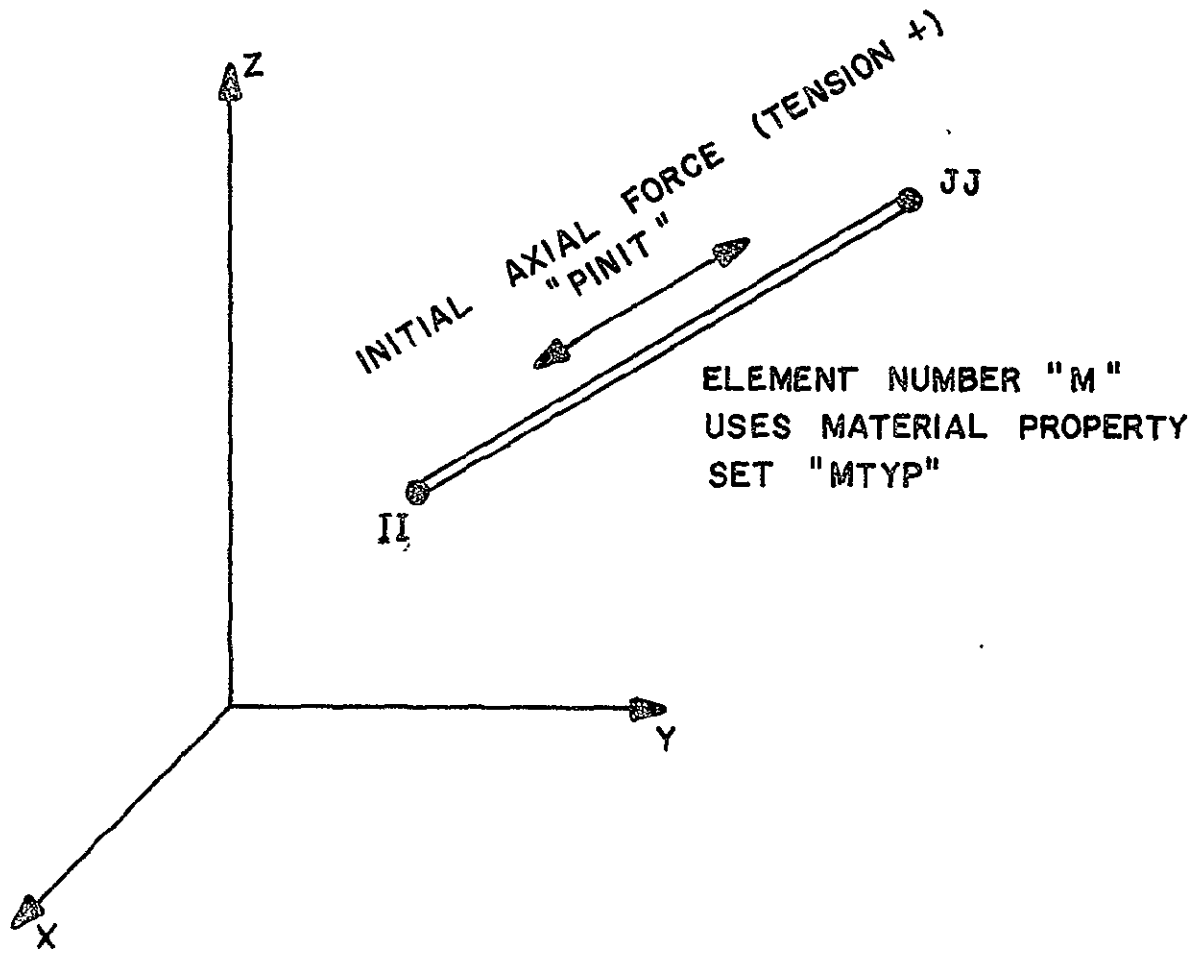


Fig. 4 Truss Model Parameters

IV. TRUSS ELEMENTS (continued)

this model type is defined by the entry NPAR(15). If NPAR(15).EQ.1 the model is defined by Young's modulus only and NPAR(17).EQ.0. If NPAR(15).EQ.2 the stress-strain curve is input in Section IV.3 below.

The model defined for the element group must be consistent with the nonlinear formulation used (defined by NPAR(3)) and the requirement of equilibrium iteration as defined on the Program Control Cards (Section II). As stated previously, equilibrium iterations can only be performed if the model allows for iteration, and if at least one nonlinear element group is used in the analysis.

- (4) The variable NPAR(16) defines the number of sets of material/section properties to be read in Sections IV.2 and IV.3 below.
- (5) NPAR(17) is the variable NCON used in Section IV.3(c).

IV. TRUSS ELEMENTS (continued)

IV.2. Linear Elastic Material/Section Property Cards

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

IV.2. a. Material Number Card (I5)

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

IV.2. b. Property card (4F10.0)

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

NOTES

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

IV. TRUSS ELEMENTS (continued)

IV.3. Nonlinear Elastic Material/Section Property Cards

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

IV.3.a. Material number card (15)

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

IV.3.b. Section property card (3F10.0)

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

IV.3.c. Stress-strain curve card (8F10.0)

column	variable	description	notes
1-10	PROP(1,N)	Strain at point 1, e^1	(2)
11-20	PROP(2,N)	Strain at point 2, e^2	
.	.	.	
.	.	.	
.	.	.	
	PROP(NCON/2,N)	Strain at point NCON/2	
	PROP(NCON/2+1,N)	Stress at point 1, σ^1	
	PROP(NCON/2+2,N)	Stress at point 2, σ^2	
	.	.	
	.	.	
	PROP(NCON,N)	Stress at point NCON/2	

IV. TRUSS ELEMENTS (continued)

NOTES

- (1) One section property card is defined to have the same area, density and initial strain.
- (2) The stress-strain curve is defined by straight lines between the input points (e^i, σ^i). From the stress-strain curve total stresses and the tangent modulus are evaluated for a given strain (see Figure 4).

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

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

IV. TRUSS ELEMENTS (continued)

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

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

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

NOTES

- (1) Refer to Figure 4.
- (2) PINIT is the axial force in the TRUSS at zero node displacements, and zero initial strain in the truss. The initial strain defined for the material property set number gives rise to an additional force in the truss.
- (3) Elements must be input in increasing element number order. If cards for elements [M+1, M+2, ..., M+J] are omitted, these "J" missing elements are generated using MTYP and PINIT of element "M" and by incrementing the node numbers of successive elements with the value of "KG"; KG is taken from the first card of the element generation sequence (i.e., from the "M-th" element card).

V. 2/D CONTINUUM ELEMENTS

2/D CONTINUUM elements are 4- to 8-node isoparametric quadrilaterals which must be input in the global Y-Z plane. Figure 5 shows some typical 2/D CONTINUUM elements. The element can represent either planar (plane stress or plane strain condition) or axisymmetric solids, as illustrated in Fig. 6. In both cases, each element node has two (2) translational degrees of freedom.

When the element is used to represent an axisymmetric solid or shell, the global Z-axis is the axis of revolution. All elements must be located in the +Y half-plane. These conventions are illustrated in Fig. 6.

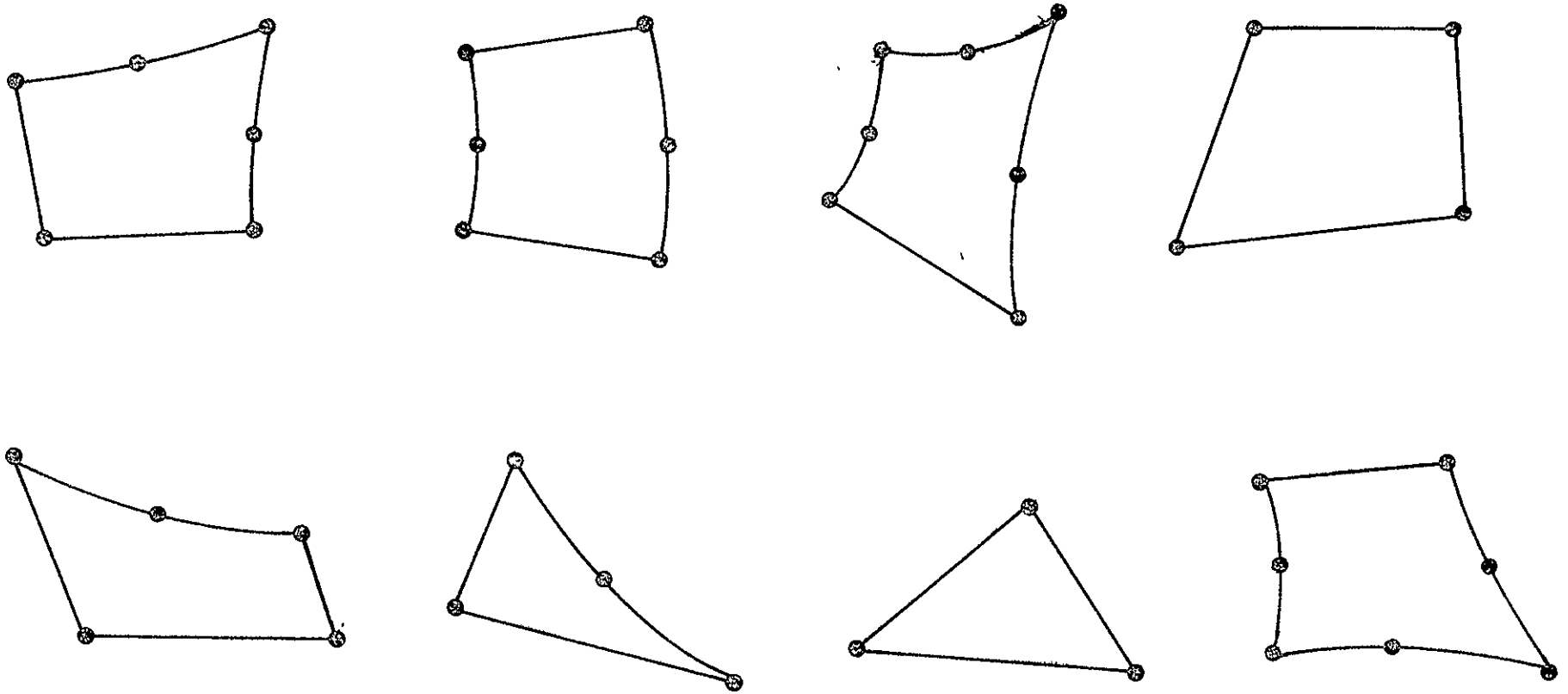
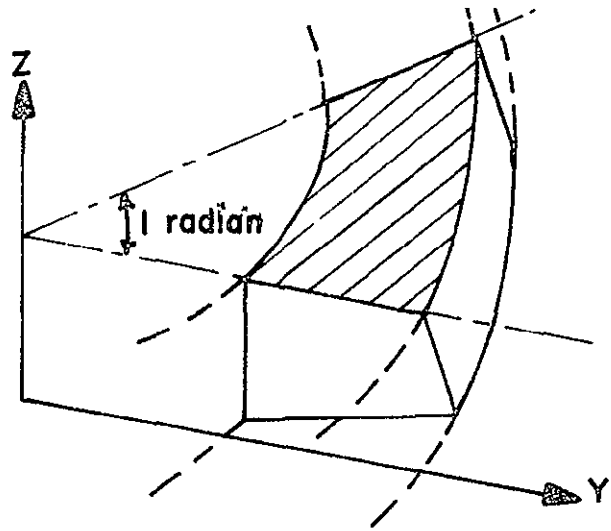
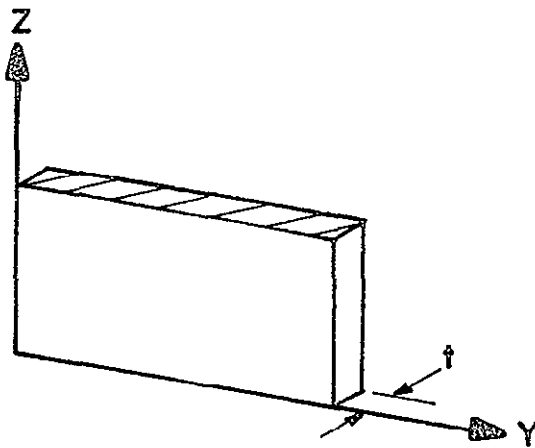


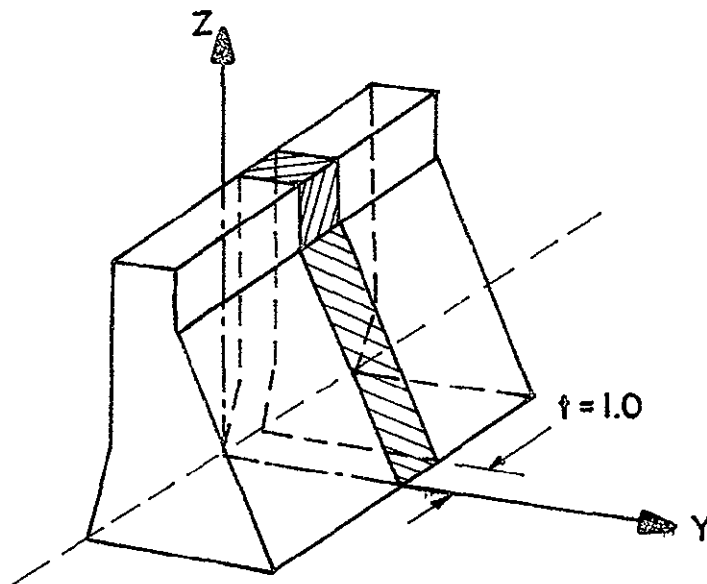
Fig. 5 Typical Two-Dimensional Isoparametric Elements*



AXISYMMETRIC FINITE ELEMENT MODEL OF A RING



PLANE STRESS FINITE ELEMENT MODEL OF A CANTILEVER



PLANE STRAIN FINITE ELEMENT MODEL OF A DAM

Fig. 6 Typical Two-Dimensional Applications*

47

V. 2/D CONTINUUM ELEMENTS (continued)

V.1. Element Group Control Card (20I4)*

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

V. 2/D CONTINUUM ELEMENTS (continued)

column	variables	description	notes
		EQ.8 - incompressible nonlinear elastic (Mooney-Rivlin/plane stress only)	
		EQ.9 - incompressible nonlinear elastic (Mooney-Rivlin/plane strain or axisymmetric)	
		EQ.10 - elastic-plastic (von Mises with mechanical sublayer hardening)	
		GE.11 - user supplied model	
61-64	NP(AR)(16)	Number of different sets of material properties GE.1	(6)
65-68	NP(AR)(17)	Number of constants per property set EQ.0 - if NP(AR)(15).LE.10 GE.0 - if NP(AR)(15).GE.11	(6)
69-72	NP(AR)(18)	Dimension of storage array required for element history EQ.0 - If NP(AR)(15).LE.10 GE.0 - If NP(AR)(15).GE.11	(6)

NOTES

- (1) 2D/CONTINUUM element numbers begin with one (1) and end with the total number of elements in this group, NP(AR)(2). Element data are input in Section V.4 below.
- (2) NP(AR)(3) is applicable for nonlinear element groups only and determines if geometrical nonlinearities are to be included in the analysis. If NP(AR)(3).EQ.1 displacements and strains are assumed to be infinitesimal. In the total Lagrangian and updated Lagrangian formulations all geometric effects (large displacements and large strains) are included in the analysis.
- (3) NP(AR)(7) limits the number of nodes that can be used to describe any of the elements in this group. A minimum of 4 and a maximum of 8 nodes are used to describe the 2/D CONTINUUM elements. For example, for the set of elements shown in Fig.V.1, NP(AR)(7) would be set to "7". Constant strain triangles are obtained by having nodes 3 and 4 coincide.
- (4) For rectangular elements an integration order of "2" is sufficient. If the element is distorted, a higher integration order need be used. Notice that apart from the larger computational effort in the calculation of the element matrices, more working storage may be required if a nonlinear material model is used (see note (6) below). The consistent mass matrix is always calculated with an integration order of 3.

V. 2/D CONTINUUM ELEMENTS (continued)

- (5) Element stresses are calculated at the points defined in the stress output location table assigned to the element (see Section V.3.). NPAR(13) defines the total number of stress output location tables input in Section V.3. NPAR(13).EQ.0 for all nonlinear element models, i.e. MODEL.GT.2 in Section V.2.
- (6) Only one material model (defined by the value of NPAR(15)) is allowed in an element group. If NPAR(15) is 1,2,...,10, the model exists in the current library for 2/D CONTINUUM elements, and the entries for NPAR(17) and NPAR(18) are ignored by the program. If, however, NPAR(15) is "11" or larger, material constants must be read into a storage array, "PROP", which has dimensions NPAR(17), NPAR(16), i.e., property constants are stored as,

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

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

$$IDWA = NPAR(18)*(NPAR(10)**2)$$
$$NPAR(2) = \text{number of elements}$$

i.e. for each integration point of each element, NPAR(18) storage locations are preserved during the solution.

The model defined for the element group must be consistent with the nonlinear formulation used (defined by NPAR(3)), the element type (defined by NPAR(5)), and the requirement of equilibrium iteration as defined in Section II. As stated previously, equilibrium iterations can only be performed if the model allows for iteration, and if at least one nonlinear element group is used in the analysis.

V. 2/D CONTINUUM ELEMENTS (continued)

V.2. Material Property Data

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

V.2. a. Material number card (I5,F10.0)*

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

NOTES

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

V.2.b. Material property card(s) (8F10.0)

MODEL 1 - linear isotropic*

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

NOTES

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

X. 2/D CONTINUUM ELEMENTS(continued)

MODEL 2 - linear orthotropic *

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

NOTES

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

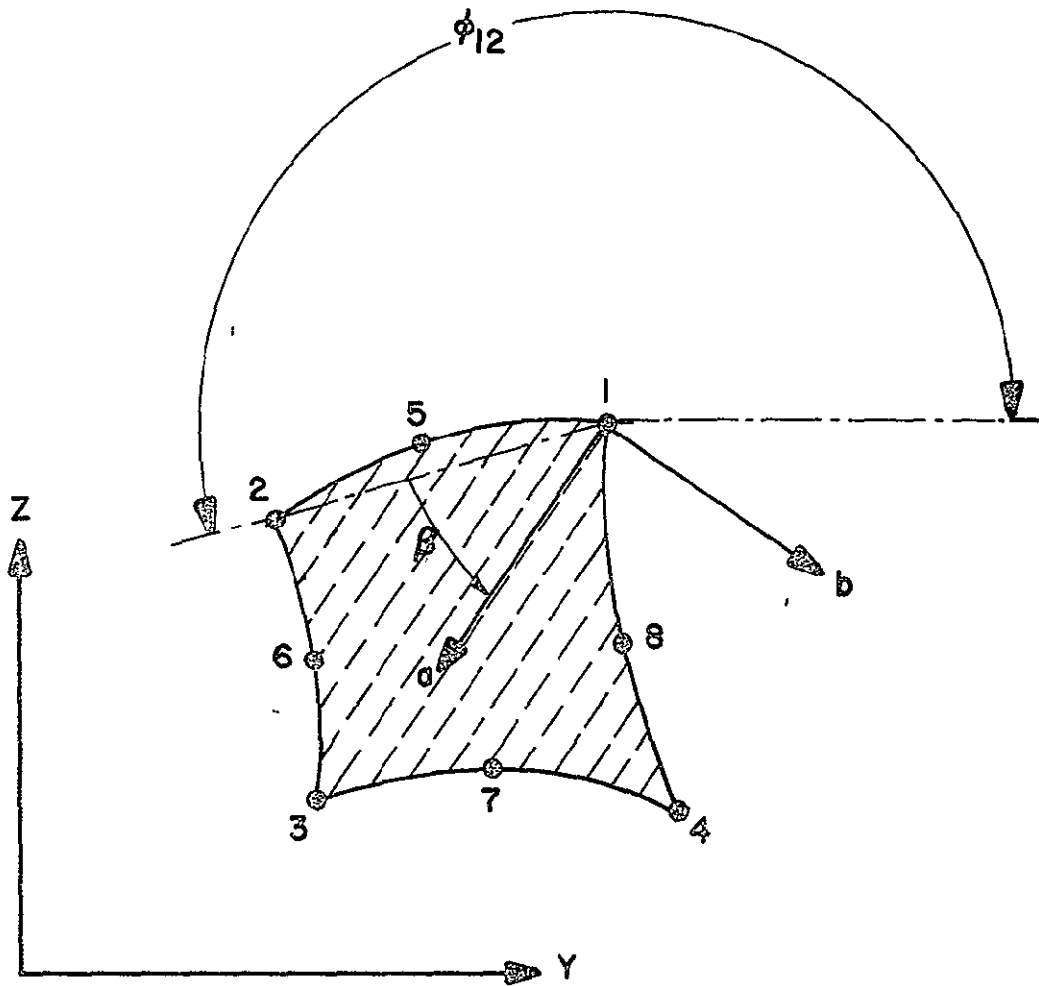


Fig. 7 Principal In-Plane Material Axes Orientation for Linear Orthotropic Material Model 2*

X. 2/D CONTINUUM ELEMENTS (continued)

MODEL 3 - variable tangent moduli*

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

NOTES

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

The material model cannot be used in plane stress analysis.

NOTATION

For the notation used in the description of the nonlinear material models refer to the report "Static and Dynamic Geometric and Material Nonlinear Analysis," SESM Report No. 74-4, February 1974 (Ref. 3).

V. 2/D CONTINUUM ELEMENTS (continued)

MODEL 4 or 5 - curve description model*

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

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

V. 2/D CONTINUUM ELEMENTS (continued)

column	variable	description	notes
31-40	PROP(20,N)	G_{LD}^2 ...	
41-50	PROP(21,N)	G_{LD}^3	
51-60	PROP(22,N)	G_{LD}^4	
61-70	PROP(23,N)	G_{LD}^5	
71-80	PROP(24,N)	G_{LD}^6	

NOTES

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

Note that

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

where "J" is the J-th table input point. All six (6) input points must be defined, as shown in Fig. 8. Linear interpolation is used to define the loading and unloading bulk moduli, ${}^tK_{LD}$, ${}^tK_{UN}$, and the loading shear modulus, ${}^tG_{LD}$, at time t. Note that these three cards must be supplied NPAR(16) times.

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

b. Tension cut-off definition card

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

column	variable	description	notes
1-10	GAMMA	material density	(1)
11-20	STIFAC	normal stiffness reduction factor	(2)
21-30	SHEFAC	shear stiffness reduction factor	

V. 2/D CONTINUUM ELEMENTS (continued)

NOTES

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

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

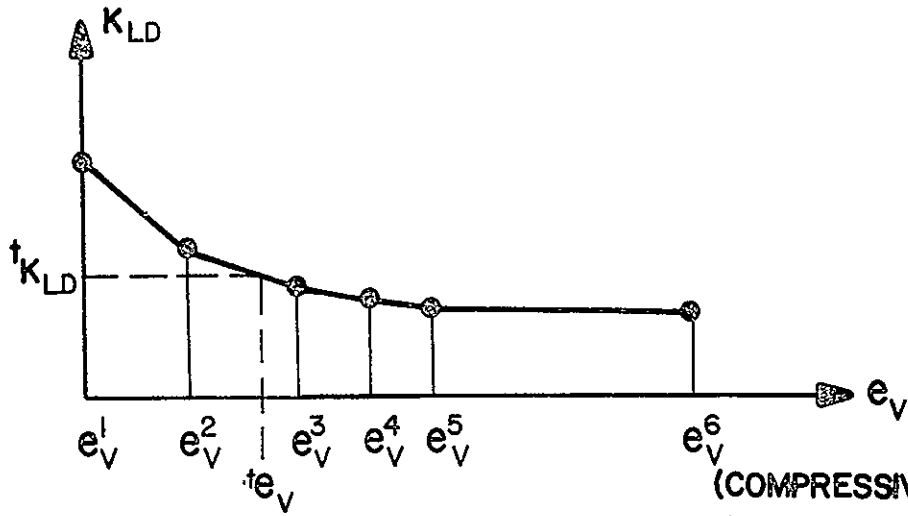
where h_i are the element interpolation functions, p_i is the pressure at the element nodes, and N is the total number of element nodes. The pressure p_i is calculated using $p_i = \text{GAMMA} * Z_i$, where Z_i is the coordinate of node i . This convention defines the assumed orientation of the Y-Z coordinate system. (All two-dimensional analyses must be carried out in the Y-Z plane). From the hydrostatic pressures and the bulk moduli input above, the in-situ volume strain at every integration point is calculated. When finding bulk and shear moduli from the curve description shown in Fig. 8, in-situ volume strain is included. That is, for MODEL 5

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

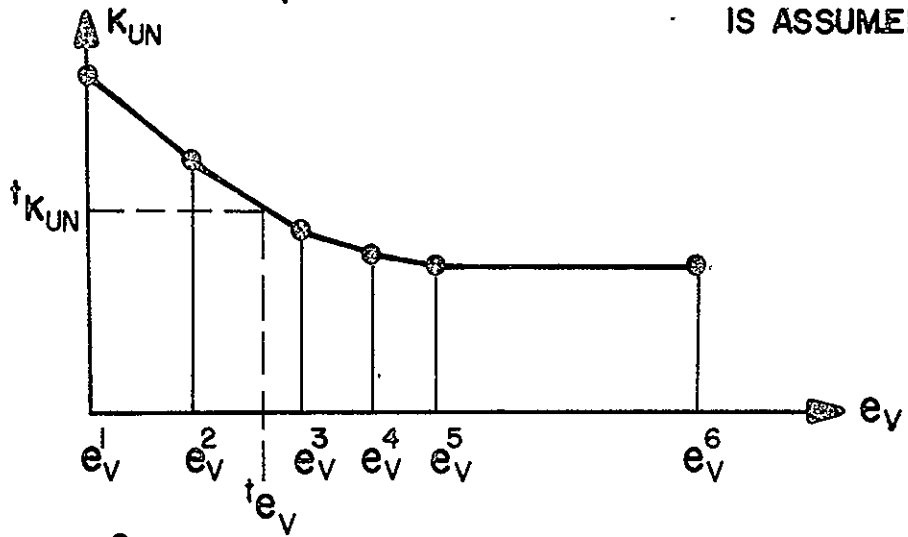
where t_{e_V} is total volume strain (abscissa of curves in Fig. 8), $t_{e_V}^P$ is the in-situ volume strain, and $t_{e_V}^L$ is the volume strain due to applied loads. Note that for MODEL 4 t_{e_V} is assumed to equal $t_{e_V}^L$.

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

LOADING BULK MODULUS



UNLOADING BULK MODULUS



LOADING SHEAR MODULUS

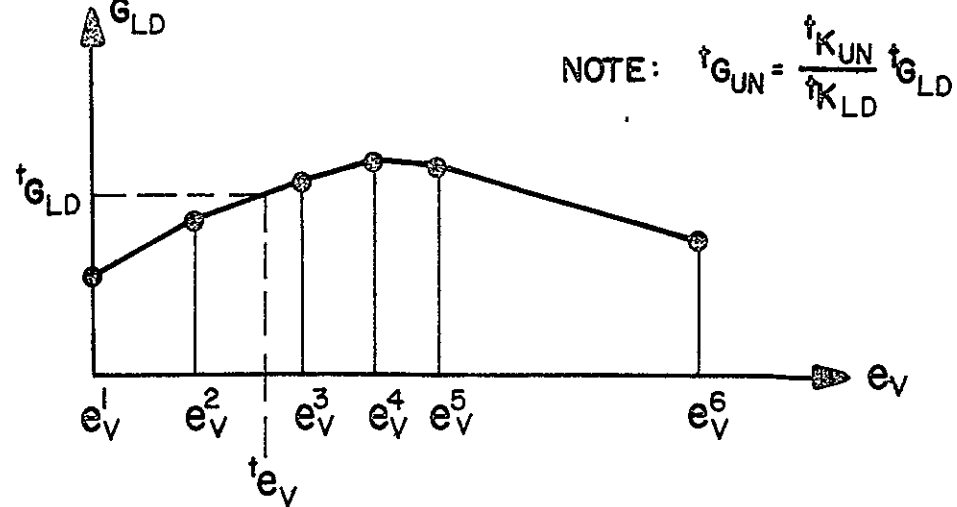


Fig. 8 Moduli vs. Volume Strain for Curve Description Model*

V. 2/D CONTINUUM ELEMENTS (continued)

MODEL 6 - Combined Isotropic-Kinematic Hardening, von Mises yield condition

	column	variable	description	notes
CARD 1	1-10	PROP(1,N)	Young's modulus, E	(1)
	11-20	PROP(2,N)	Poisson's ratio, ν	(1)
	21-30	PROP(3,N)	Number of points in piecewise linear Stress Strain Curve, RNSSC	(2)
	31-40	PROP(4,N)	Number of points in piecewise yield surface SIZE vs equivalent plastic curve, RNSIZE	(3)
	41-50	PROP(5,N)	Type of hardening rule being used, TPLAS	(4)
	51-60	PROP(6,N)	Ratio of isotropic hardening to kinematic hardening, RITK	(5)
	61-70	PROP(7,N)	Value of stress at first point in stress strain curve, $\bar{\sigma}_1$	(6)
	71-80	PROP(8,N)	Value of total strain at first point in stress strain curve, $\bar{\epsilon}_1$	(6)
CARDS 2,3,4	1-10	PROP(9,N)	Value of stress at second point in stress strain curve, $\bar{\sigma}_2$	
	11-20	PROP(10,N)	Value of strain at second point in stress strain curve, $\bar{\epsilon}_2$	
	:	:		
	:	:		
	:	:		
		PROP(5+2* RNSSC,N)	Value of stress at last point in stress strain curve, $\bar{\sigma}_{RNSSC}$	
		PROP(6+2* RNSSC,N)	Value of strain at last point in stress strain curve, $\bar{\epsilon}_{RNSSC}$	
		PROP(7+2* RNSSC,N)	Value of yield surface size at first point in yield surface size vs equivalent plastic strain curve, k_1	(7)
		PROP(8+2* RNSSC,N)	Value of equivalent plastic strain at first point in yield surface size vs equivalent plastic strain curve, $\bar{\epsilon}_1^p$	

V. 2/D CONTINUUM ELEMENTS (continued)

For MODEL "6" (continued)

column	variable	description	notes
:	:	:	
:	:	:	
:	:	:	
	PROP(5+2* RNSSC+2*RNSIZE ,N)	Value of yield surface size at last point in yield surface size vs equivalent plastic strain curve, k_{RNSIZE}	
	PROP(6+2* RNSSC+2*RNSIZE ,N)	Value of equivalent plastic strain at last point of yield surface size vs equivalent plastic strain curve, $\bar{\epsilon}^p_{RNSIZE}$	

NOTES

(1) MODEL 6 is nonlinear library material model for which NPAR(17).EQ.26 by default. NPAR(18).EQ.15, since 15 history dependent variables must be stored for each integration point.

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

(2) RNSSC is the number of points in a piecewise linear stress strain curve, and includes the end point of the last linear segment, but not the point STRESS=0, STRAIN=0. Maximum allowable value is RNSSC=5, the minimum value is RNSSC=2.

(3) RNSIZE is the number of points in a piecewise linear yield surface size vs equivalent plastic strain curve, and includes both the end point of the last linear segment as well as the point STRESS=yield stress, EQPLAS=0. RNSIZE=0 for TPLAS.NE.3.

(4) TPLAS.EQ.0Isotropic hardening
 EQ.1Kinematic hardening (Prager-Ziegler)
 EQ.2Combined isotropic, kinematic
 hardening, constant ratio
 EQ.3Combined isotropic, kinematic
 hardening, variable ratio

(5) RITK is the ratio of isotropic to kinematic hardening for TPLAS.EQ.2, i.e., with RITK.EQ.1 being pure isotropic, RITK.EQ.0 pure kinematic. If TPLAS.NE.2, RITK is ignored.

V. 2/D CONTINUUM ELEMENTS (continued)

For MODEL "6" (continued)

NOTES

- (6) Values of STRESS and STRAIN are input in pairs for the piecewise linear stress strain curve using the format 8F10.0. A maximum of five (5) pairs of data is allowed. Input RNSSC pairs of data. The input stress-strain curve must be monotonically increasing such that $PROP(5+2*I).LE. PROP(5+2*(I+1))$ and $PROP(6+2*I).LT.PROP(6+2*(I+1))$ for any I. See also Fig. 9.
- (7) Values of YSSIZE (Yield Surface SIZE) and EQPLAS(Equivalent PLASTic strain) are input in pairs for the piecewise linear yield surface size vs equivalent plastic strain curve. Data for this curve need not be input unless TPLAS.EQ.3. For TPLAS.LT.3, a compatible curve is generated internally in the program, appropriate for the hardening rule indicated by TPLAS. See also Fig. 9.

NOTE: Regardless of the value of RNSSC and RNSIZE, four cards will be read on 8F10.0 format to generate the PROP array. Blank entries for a property value will be read and ignored after the last non-zero entry. FOUR CARDS MUST BE SUPPLIED.

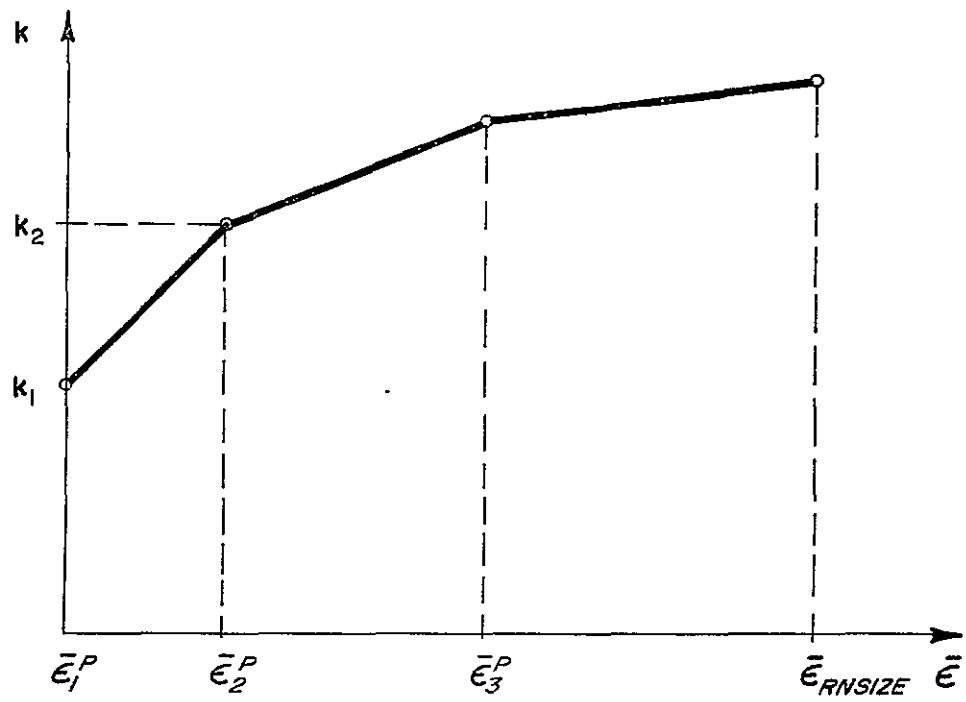
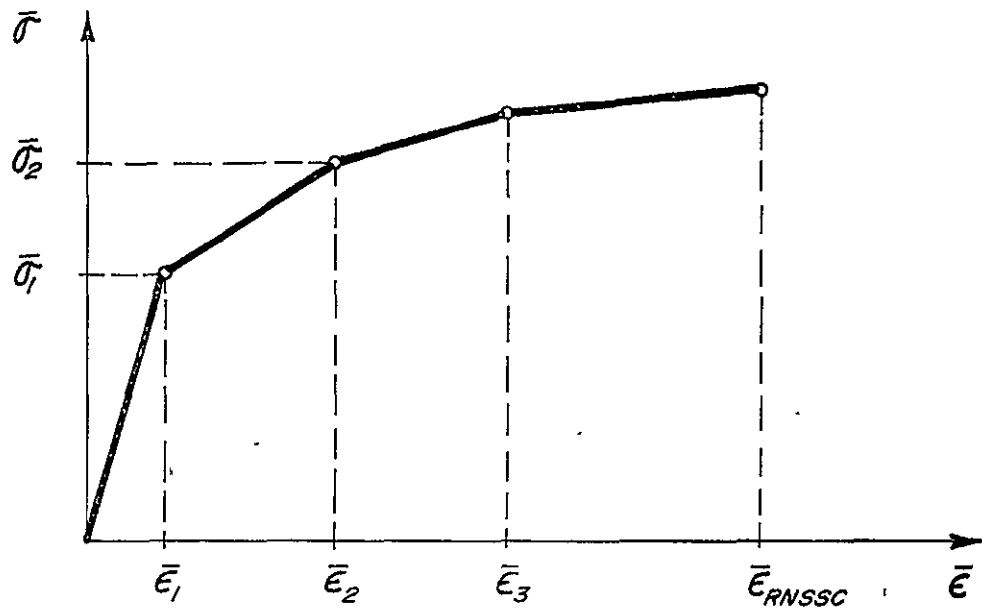


Fig. 9 Stress and Yield Surface Size vs. Strain Curves for Combined Isotropic-Kinematic Hardening Model

V. 2/D CONTINUUM ELEMENTS (continued)

MODEL 7 - elastic-plastic material, Drucker-Prager yield condition *

column	variable	description	notes
1-10	PROP(1,N)	Young's modulus, E	(1)
11-20	PROP(2,N)	Poisson's Ratio, ν	
21-30	PROP(3,N)	Angle of friction, θ	
31-40	PROP(4,N)	Cohesion, C	

NOTES

- (1) MODEL 7 is a nonlinear library material model requiring four (4) constants (i.e., NPAR(17).EQ.4 by default) per material property set. Eleven (11) parameters need to be stored per integration point, i.e. NPAR(18).EQ.11 by default. The model assumes elastic perfectly plastic conditions. For a further description of the material model parameters, refer to the report "Static and Dynamic Geometric and Material Nonlinear Analysis," SESM Report No. 74-4, Civil Engineering Department, University of California, Berkely, California, February 1974 (Ref. 3).

V. 2/D CONTINUUM ELEMENTS (continued)

MODEL 8 or 9 - incompressible nonlinear elastic material, Mooney-Rivlin model

column	variable		description	notes
1-10	PROP(1,N)	C_1	C_1 and C_2 are material constants in the Mooney-Rivlin expression for strain energy.	(1)
11-20	PROP(1,N)	C_2		

NOTES

- (1) MODEL 8 and 9 are nonlinear library material models for which NPAR(17) is set to "2". Since the material is elastic, no history-dependent variables need be stored at integration points and NPAR(18) is set to "0". MODEL 8 is for plane stress only and MODEL 9 is for plane strain or axisymmetric only. C_1 and C_2 are material constants in the expression for strain energy:

$$v = C_1(I_1-3) + C_2(I_2-3)$$

where I_1 and I_2 are the first and second strain invariants, respectively. A more complete description of the model is given in Ref. 5.

V. 2/D CONTINUUM ELEMENTS (continued)

MODEL 10 - Mechanical Sublayer Model, von Mises yield condition

	column	variable	description	notes
CARD 1	1-10	PROP(1,N)	Young's modulus, E	(1)
	11-20	PROP(2,N)	Poisson's ratio, ν	(1)
	21-30	PROP(3,N)	Number of points in piecewise linear <u>Stress Strain Curve</u> , RNSSC	(2)
	31-40	PROP(4,N)	Value of stress at first point in stress strain curve, $\bar{\sigma}_1$	(2)
	41-50	PROP(5,N)	Value of strain at first point in stress strain curve, $\bar{\epsilon}_1$	(2)
		.	.	
		PROP(8,N)	Value of stress at third point in stress strain curve, $\bar{\sigma}_2$	
CARD 2		PROP(9,N)	Value of strain at third point in stress strain curve, $\bar{\epsilon}_2$	
	.	.	.	
	.	.	.	
	.	.	.	
		PROP(2+2*RNSSC ,N)	Value of stress at last point in stress strain curve, $\bar{\sigma}_{RNSSC}$	
		PROP(3+2*RNSSC ,N)	Value of Strain at last point in stress strain curve, $\bar{\epsilon}_{RNSSC}$	

NOTES

- (1) MODEL 10 is a nonlinear library material model for which NPAR(17) EQ.11 by default. NPAR(18).EQ.20, since 20 history dependent variables must be stored for each integration point.

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

- (2) RNSSC is the number of points in a piecewise linear stress strain curve, and includes the end point of the last linear segment (unless that segment is perfectly plastic), but not the point STRESS=0, STRAIN=0. The maximum allowable value is RNSSC=4.0. If RNSSC.LE.2.0,

V. 2/D CONTINUUM ELEMENTS (continued)

a blank card B must still be supplied. It is also required that the input stress strain curve be monotonically increasing so that $\text{PROP}(2+2*I).\text{LT}.\text{PROP}(2+2*(I+1))$ and $\text{PROP}(3+2*I).\text{LT}.\text{PROP}(3+2*(I+1))$. Note that the inequality reads .LT., not .LE. Figure 10 depicts a typical piecewise-linear stress-strain curve.

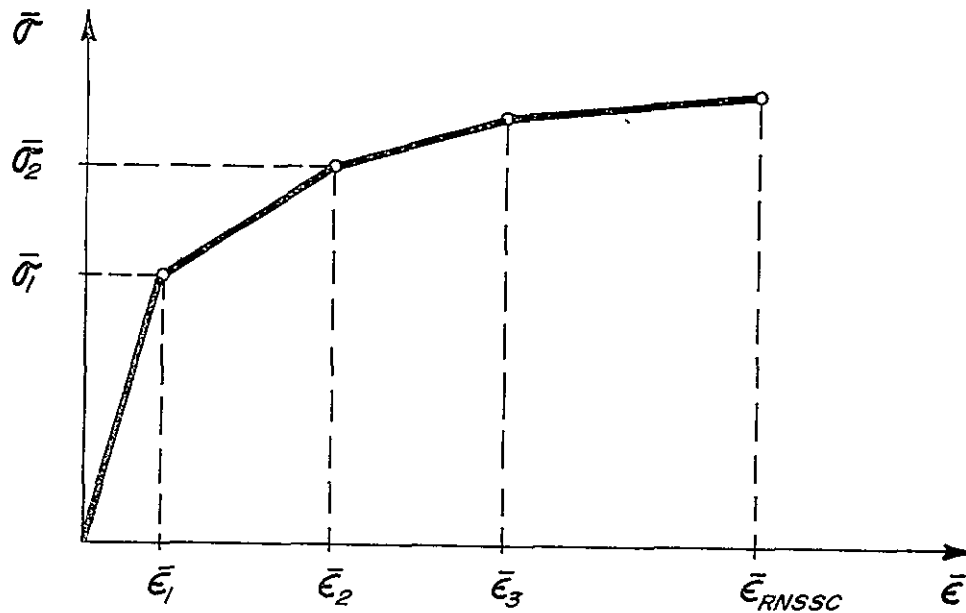


Fig. 10 Equivalent Uniaxial Stress-Strain Curve for Mechanical Sublayer Model

V. 2/D CONTINUUM ELEMENTS (continued)

MODEL 11 - user-supplied material model*

column	variable	description	notes
1-10	PROP(1,N)		(1)
11-20	PROP(2,N)		
21-30	PROP(3,N)		
.			
.	etc.		
.			

NOTES

- (1) MODEL 11 is a non-standard model (i.e., is not in the material model library) and requires that property data be read

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

where NPAR(17) is user-supplied (i.e., no default value exists). Subroutines which allocate storage locations (in "WA") for element history (if the model is nonlinear) and which calculate the incremental stress-strain law and current stresses must be supplied. For a discussion on how to incorporate a user-supplied material model, see Ref. 2.

V. 2/D CONTINUUM ELEMENTS (continued)

V.3. Stress Output Table Cards (9I5) *

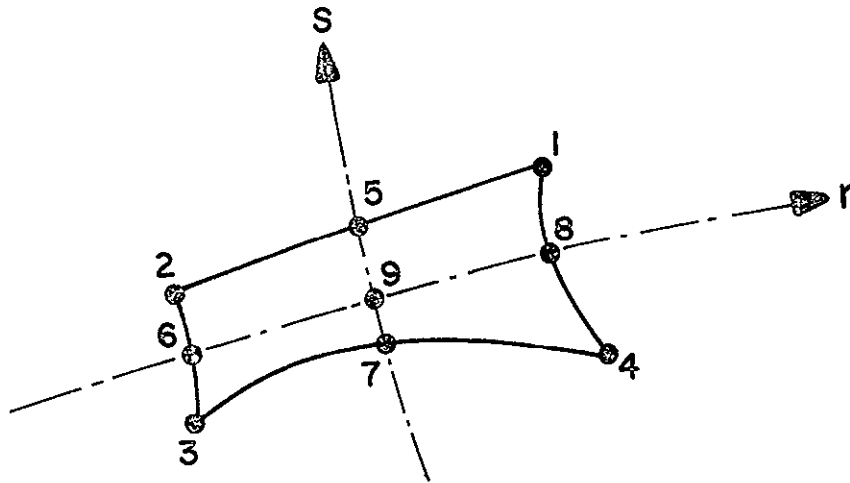
Skip this section of stresses at integration points are to be printed, i.e. NPAR(13).EQ.0; otherwise supply NPAR(13) cards.

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

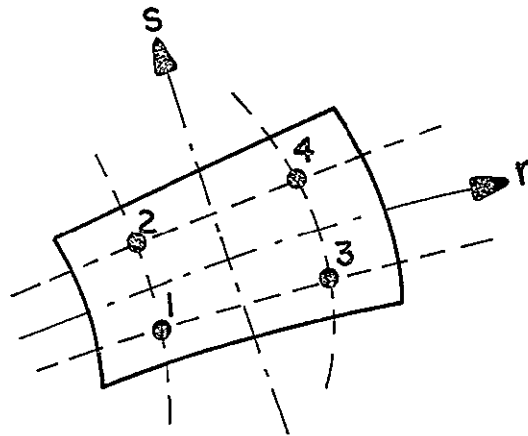
NOTES

- (1) Stress tables are defined to provide flexibility in element stress output requests. Each element can refer to one of the tables defined, and the element stresses are then calculated at the points specified in the table. Refer to Fig. 11 for selection of the stress calculation points. The first "0" entry in a table will terminate that table. For example, if ITABLE(N,1).EQ.7 and ITABLE(N,2).EQ.0, then stresses will be printed at point 7 only whenever this stress table is referred to.

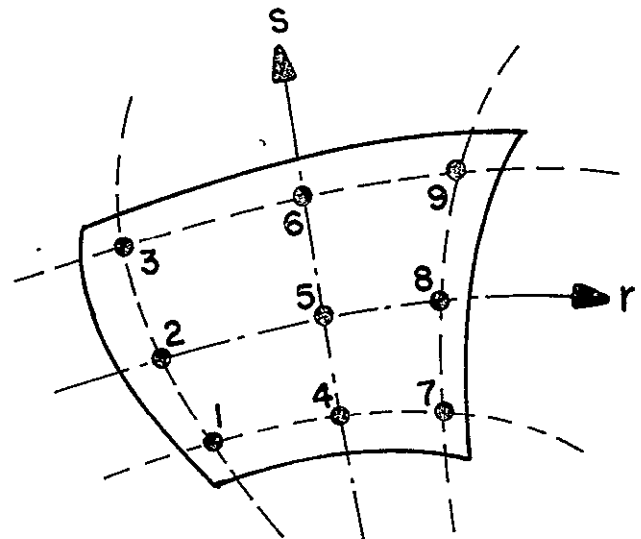
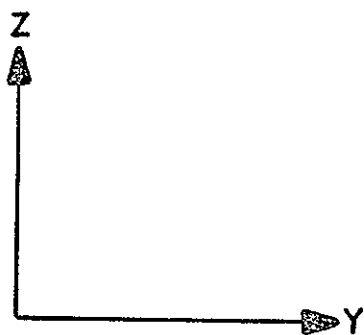
The stress tables are only used when MODEL "1" or "2" define the material behavior.



STRESS PRINTOUT TABLE POINTS FOR
2/D CONTINUUM ELEMENTS



2-POINT INTEGRATION



3-POINT INTEGRATION

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

Fig. 11 Stress Print-Out Locations for Two-Dimensional Elements*

V. 2/D CONTINUUM ELEMENTS (continued)

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

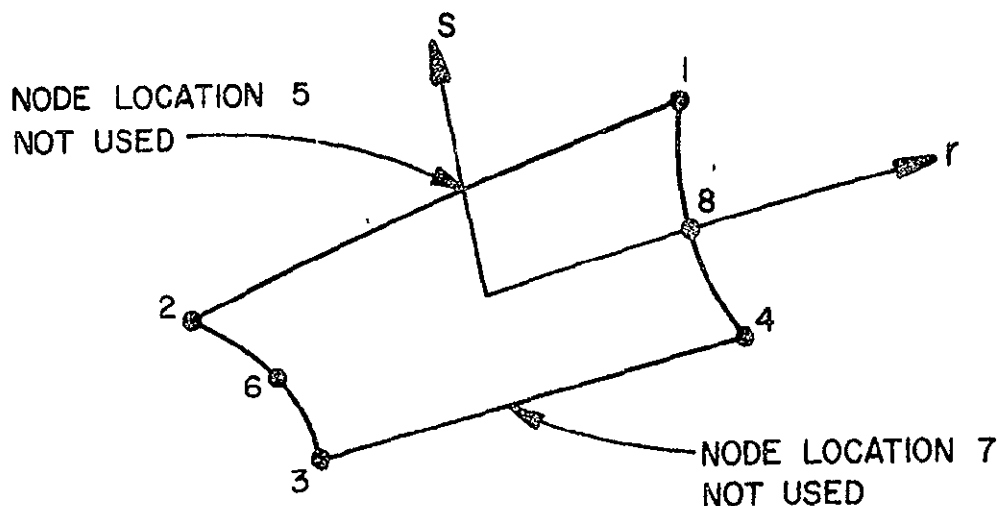
column	variable	description	notes
1-5	M	2/D CONTINUUM element number; GE.1 and LE.NPAR(2)	(1)
6-8	IEL	Number of nodes used to describe this element; EQ.0 - default set to "NPAR(7)" LE.NPAR(7)	(2)
9-10	IPS	Number of the stress output table to be used for stress calculations for this element (if a linear material model is used) or flag for printing stresses at element integration points (nonlinear material models; EQ.0 - no stress output for this element	(3)
11-20	BET	Material angle, β , in degrees to be used in connection with material model "2" (linear orthotropic)	
21-30	THIC	Element thickness; GT.0 - plane stress only	(4)
31-35	MTYP	Material property set number assigned to this element; GE.1 and LE.NPAR(16)	
36-40	KG	Node generation parameter used to compute node numbers for missing elements (given on first card of a sequence); EQ.0 - default set to "1"	(1)
41-45	NOD(1)	Global node number of element nodal point 1	(2)
46-50	NOD(2)	Global node number of element nodal point 2	
51-55	NOD(3)	Global node number of element nodal point 3	
56-60	NOD(4)	Global node number of element nodal point 4	
61-65	NOD(5)	Global node number of element nodal point 5	
66-70	NOD(6)	Global node number of element nodal point 6	
71-75	NOD(7)	Global node number of element nodal point 7	
76-80	NOD(8)	Global node number of element nodal point 8	

V. 2/D CONTINUUM ELEMENTS (continued)

NOTES

- (1) Elements must be input in ascending element number order. If data cards for elements $[M+1, M+2, \dots, M+J]$ are omitted, these "J" missing elements are generated using IEL, IPS, BET, THIC, and MTYP given on the card for element "M" and by incrementing node numbers of successive elements with the value "KG"; the value of KG used for incrementation is taken from the M-th element card, and only the non-zero nodes appearing on the M-th element card are incremented when generating missing element data. The first and last element must always be input.
- (2) The number of nodes in element "M" is defined by "IEL". However, all 8 entries for NOD(I) are read from the element data card; if $IEL.LT.8$ the particular node locations not used in this element need be input as "0" in NOD(I). Fig. 12 defines the input sequence that must be observed for element node input.

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



is defined by

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

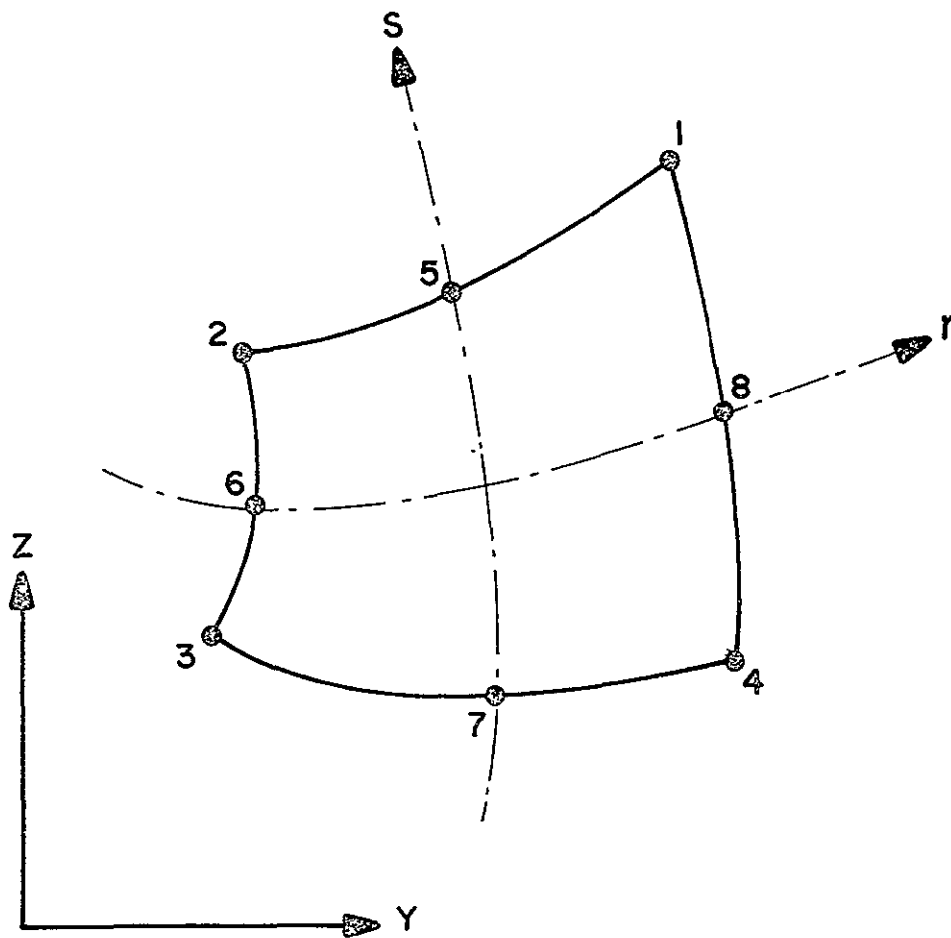


Fig. 12 Element Node Number Sequence for Two-Dimensional Elements

V. 2/D CONTINUUM ELEMENTS (continued)

NOTES

(2 continued)

where the nonzero entries (X) are the global mesh node numbers (Section III.2) of the 6 nodes.

- (3) If IPS.EQ.0 no stress output will be provided for this element. A nonzero entry for the element stress printout flag "IPS" has a different meaning when used with nonlinear material models than it does when used with linear material models:
 - a) For linear material models (MODELS "1" and "2") IPS specifies the stress output table (Section V.3) to be used for this element. If no stress output tables were input (i.e., if NPAR(13).EQ.0) then IPS.EQ.1 specifies that stresses are to be printed at all integration points for this element, as shown in Fig. 11.
 - b) For nonlinear material models (MODELS.GT."2") IPS.EQ.1 specifies that stresses are to be printed at all integration points for this element, as shown in Fig. 11.
- (4) The thickness of the element need only be defined in plane stress analysis. In plane strain analysis thickness is assumed to be "1". In axisymmetric analysis thickness is assumed to be one (1) radian.

VI. 3/D CONTINUUM ELEMENTS

3/D CONTINUUM elements are 8- to 21-node isoparametric or sub-parametric curvilinear hexahedra. Typical elements are illustrated in Figure 13.

These elements may be used to model three-dimensional solids or thick shells, two examples of which are shown in Figure 14.

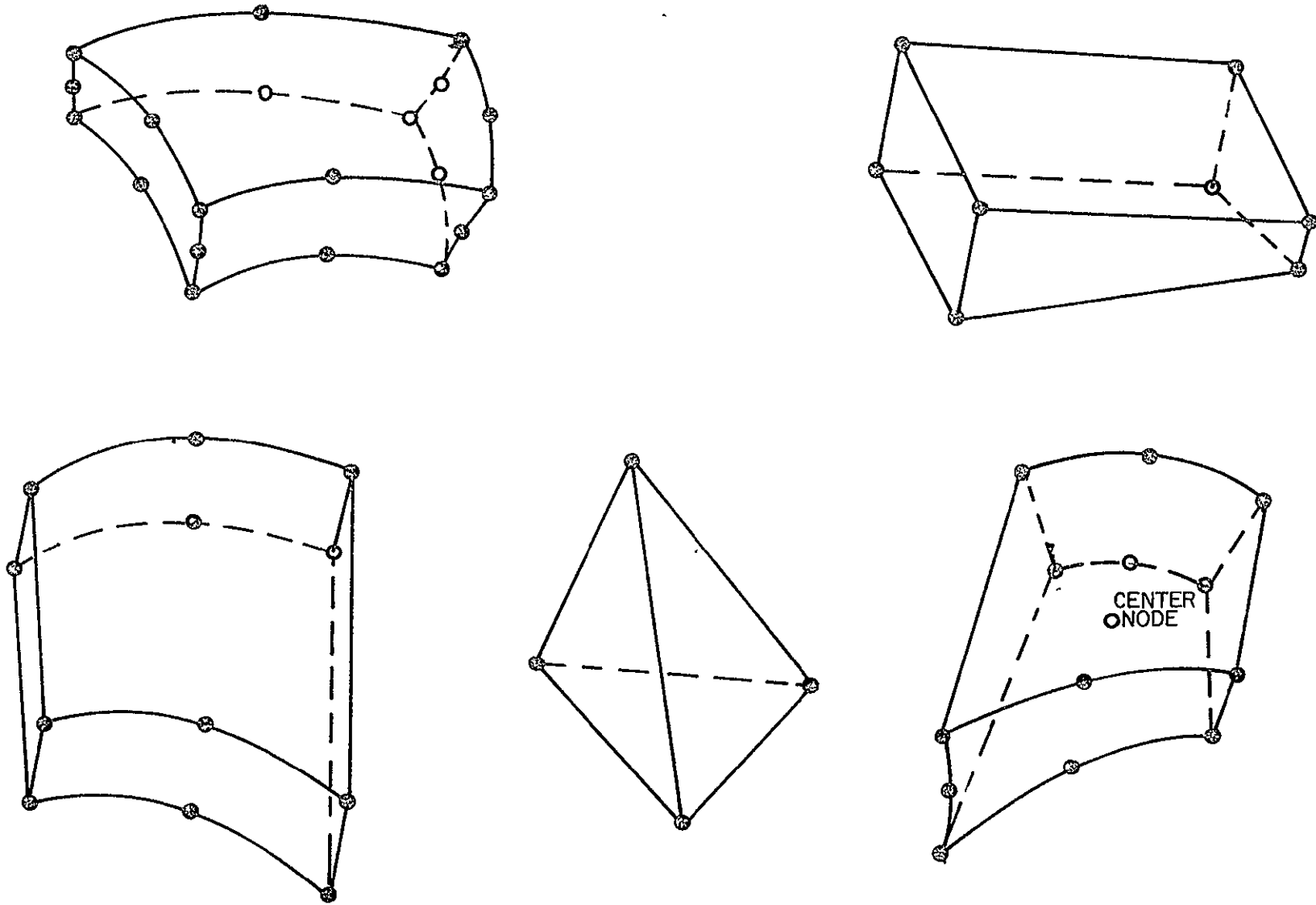
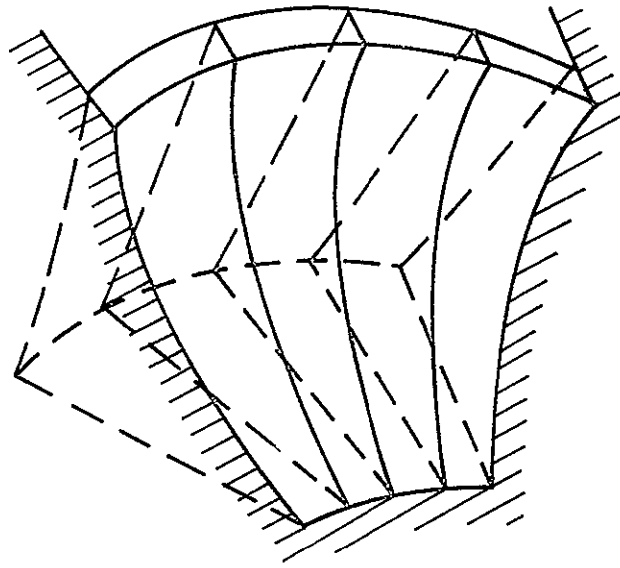
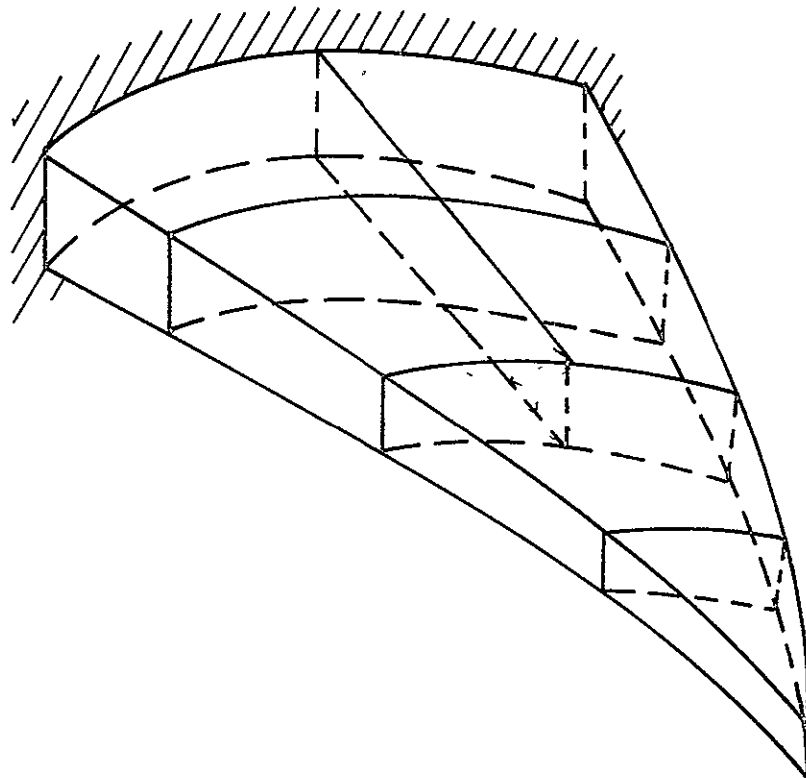


Fig. 13 Typical Three-Dimensional Isoparametric Elements*



3/D CONTINUUM ELEMENT MODEL OF ARCH DAM



3/D CONTINUUM ELEMENT MODEL OF THICK SHELL

Fig. 14 Possible Three-Dimensional Applications*

VI. 3/D CONTINUUM ELEMENTS (continued)

VI.1. Element Group Control Card (2014)*

column	variable	description	notes
1-4	NPAR(1)	Enter the number "3"	
5-8	NPAR(2)	Number of 3/D CONTINUUM elements in this group GE.1	(1)
9-10	NPAR(3)	Flag indicating type of geometric nonlinear analysis EQ.0 - default set to "1" EQ.1 - linear (small strain) EQ.2 - nonlinear (large strain), total Lagrangian EQ.3 - nonlinear (large strain), updated Lagrangian	(2)
25-28	NPAR(7)	Maximum number of nodes used to describe any one element EQ.0 - default set to "21"	(3)
37-40	NPAR(10)	Numerical integration order to be used in Gauss quadrature formula for r-s plane EQ.0 - default set to "2" GE.2 and LE.4	(4)
41-44	NPAR(11)	Numerical integration order to be used in Gauss quadrature formula for t-direction EQ.0 - default set to "2" GE.2 and LE.4	(4)
49-52	NPAR(13)	Number of stress output location tables EQ.0 - print stresses at integration points	(5)
57-60	NPAR(15)	Material model number GE.1 and LE.8 EQ.1 - linear elastic isotropic EQ.2 - linear elastic orthotropic EQ.3 - curve description model EQ.4 - elastic-plastic, combined kinematic isotropic hardening, von-Mises yield condition EQ.5 - elastic-plastic, mechanical sublayer hardening, von-Mises yield condition EQ.6 - incompressible nonlinear elastic, Mooney-Rivlin material GE.7 - user supplied model	(6)

VI. 3/D CONTINUUM ELEMENTS (continued)

column	variable	description	notes
61-64	NP(AR)(16)	Number of different sets of material properties GE.1	(6)
65-68	NP(AR)(17)	Number of constants per material property set EQ.0 - if NP(AR)(15).LE.6 GE.1 - if NP(AR)(15).GE.7	(6)
69-72	NP(AR)(18)	Dimension of storage array required for element history EQ.0 - if NP(AR)(15).LE.6 GE.0 - if NP(AR)(15).GE.7	(6)

NOTES

- (1) 3/D CONTINUUM element numbers begin with one (1) and end with the total number of elements in this group, NP(AR)(2). Data for the individual elements are input in Section VI.4 below.
- (2) NP(AR)(3) is applicable for nonlinear element groups only and determines the type of geometrical nonlinearities to be considered in the analysis.
- (3) NP(AR)(7) is the maximum number of nodes that can be used to describe any one of the elements in this group, i.e. elements in this group must have less than or equal to NP(AR)(7) nodes. A minimum of 8 and a maximum of 21 nodes are used to describe 3/D CONTINUUM elements as indicated in Fig. 16. Constant strain tetrahedra can be formed from 8-node elements by having nodes 1,2,3 and 4 coincide and nodes 7 and 8 coincide.
- (4) The selection of appropriate integration orders depends on the element shape and strain state being considered. When the quantities being integrated vary irregularly a higher order is needed. An integration order of "2" is sufficient for most problems (especially those modeled with 16 node elements); however, an integration order of 3 should be used for all 20 node elements. Shell or plate problems often use thin elements in which strain varies more or less linearly through the thickness, but more irregularly in the plane of the surface of the shell. In these cases it is advantageous to specify a lower order for integration through the thickness, i.e., NP(AR)(11) < NP(AR)(10).

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

$$n = \text{NP(AR)(11)} * \text{NP(AR)(10)} * \text{NP(AR)(10)}$$

VI. 3/D CONTINUUM ELEMENTS (continued)

NOTES

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

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

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

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

- (6) Only one material model (defined by the value of NPAR(15)) is allowed in an element group. If NPAR(15) is 1, 3, 4, 5, or 6 the model exists in the current library for 3/D CONTINUUM elements, and the entries for NPAR(17) and NPAR(18) are ignored by the program. If, however, NPAR(15) is "7" or larger, material constants must be read into a storage array, "PROP", which has dimensions NPAR(17) by NPAR(16); i.e., property constants are stored as,

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

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

$$IDWA = NPAR(18) * (NPAR(16)**2) * NPAR(11)$$

$$NPAR(2) = \text{number of elements}$$

i.e., for each integration point of each element NPAR(18) storage locations are preserved during the solution.

The model defined for the element group must be consistent with the nonlinear formulation used (NAPR(3)) and the requirement of equilibrium iteration. Equilibrium iteration can be performed only if the material model allows for it and if there is at least one nonlinear element group.

VI. 3/D CONTINUUM ELEMENTS (continued)

VI.2. Material Property Data

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

VI.2.a. Material number card (I5,F10.0)*

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

NOTES

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

VI.2.b. Material property card(s) (8F10.0)

MODEL 1 - linear isotropic elastic

column	variable	description	notes
1-10	PROP(1,N)	Young's modulus, E	(1)
11-20	PROP(2,N)	Poisson's ratio, ν	

NOTES

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

VI. 3/D CONTINUUM ELEMENTS (continued)

MODEL 2 - linear elastic orthotropic
first card

column	variable	description	notes
1-10	PROP(1,N)	E_{11} Young's modulus in the "1" principle direction	(1)
11-20	PROP(2,N)	E_{22} Young's modulus in the "2" principle direction	
21-30	PROP(3,N)	E_{33} Young's modulus in the "3" principle direction	
31-40	PROP(4,N)	G_{12} shear modulus	
41-50	PROP(5,N)	G_{13} shear modulus	
51-60	PROP(6,N)	G_{23} shear modulus	
61-70	PROP(6,N)	ν_{12} , Poisson's ratio, contraction in "1" direction due to stress in "2" direction	
71-80	PROP(7,N)	ν_{13} Poisson's ratio	

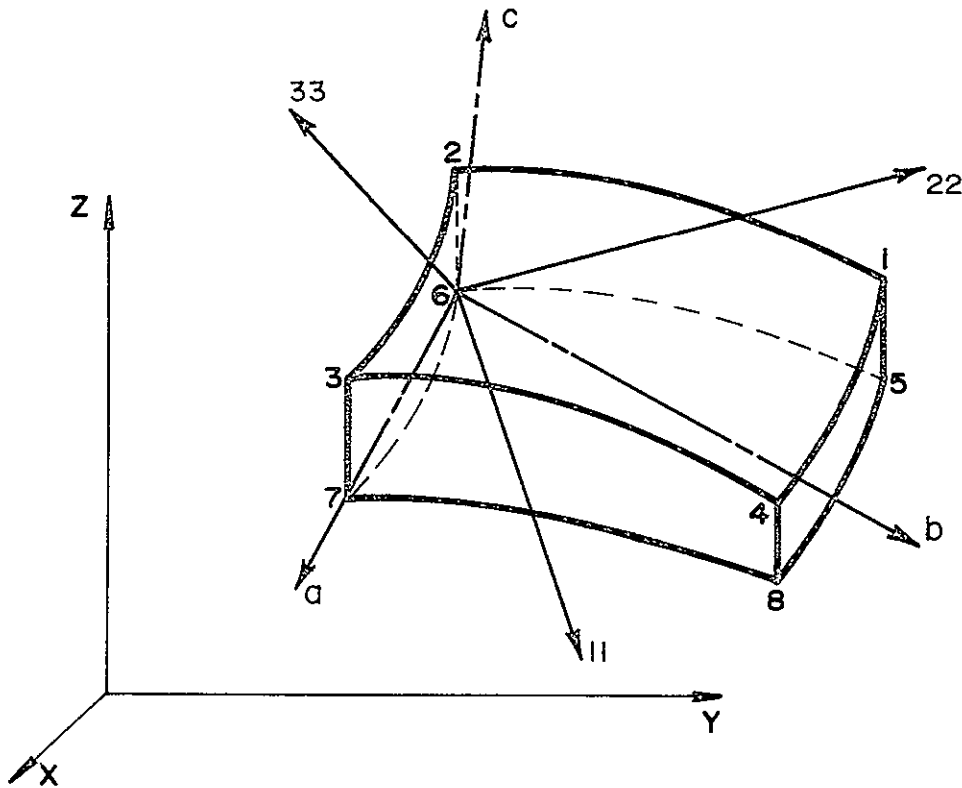
second card

1-10	PROP(8,N)	ν_{23} Poisson's ratio
------	-----------	----------------------------

NOTES

- (1) Model 2 is linear (i.e., NPAR(18).EQ.0 by default). The model requires nine (9) property constants, i.e., NPAR (17).EQ.9 by default. The constants (E_{11} , E_{22} , ..., ν_{23}) are defined in an orthogonal principle material coordinate system (11, 22, 33). These principle material axes are in turn defined relative to an orthogonal coordinate system (a,b,c) associated with each element as shown below.

VI. 3/D CONTINUUM ELEMENTS (continued)



The "a" direction corresponds to the direction of a vector connecting element nodes 6 and 7, "b" is normal to "a" and in the plane described by nodes 5-6-7, and "c" is normal to "a" and "b" such that $\vec{c} = \vec{a} \times \vec{b}$.

The angles α (ALPHA), β (BETA) and γ (GAMMA) are read individually for each element (Section VI.4) or substructure (Section VII.2).

The angles α , β , and γ defining the (1, 2, 3) axes relative to the (a, b, c) axes may be defined through a sequence of three axes rotations. Assume first that the two coordinate systems coincide. Now rotate the (a, b, c) system about the c axis with α degrees to obtain an (a'-b'-c') system (i.e., a'-b' is still in the a-b plane and α is the angle between a and a'). Next rotate the (a'-b'-c') system about the b' axis with β degrees to obtain an (a''-b''-c'') system (i.e., a''-b'' is now in the a'-c' plane and β is the angle between c' and c''). Finally rotate the (a''-b''-c'') system about the a'' axis with γ degrees to obtain an (a'''-b'''-c''') system and, equivalently, the (1-2-3) principal material axes. The angles α and γ are positive as defined by the right-hand rule while β is positive using the left-hand rule.

VI. 3/D CONTINUUM ELEMENTS (continued)

MODEL 3 - curve description model *

first card column	variable		description	notes
1-10	PROP(1,N)	e_V^1	volume strain at point "1"	(1)
11-20	PROP(2,N)	$e_V^2 \dots$		
21-30	PROP(3,N)	e_V^3		
31-40	PROP(4,N)	e_V^4		
41-50	PROP(5,N)	e_V^5		
51-60	PROP(6,N)	e_V^6		
61-70	PROP(7,N)	K_{LD}^1	loading bulk modulus at point "1"	
71-80	PROP(8,N)	$K_{LD}^2 \dots$		

second card

1-10	PROP(9,N)	K_{LD}^3		
11-20	PROP(10,N)	K_{LD}^4		
21-30	PROP(11,N)	K_{LD}^5		
31-40	PROP(12,N)	K_{LD}^6		
41-50	PROP(13,N)	K_{UN}^1	unloading bulk modulus at point "1"	
51-60	PROP(14,N)	$K_{UN}^2 \dots$		
61-70	PROP(15,N)	K_{UN}^3		
71-80	PROP(16,N)	K_{UN}^4		

third card

1-10	PROP(17,N)	K_{UN}^5		(2)
11-20	PROP(18,N)	K_{UN}^6		
21-30	PROP(19,N)	G_{LD}^1	loading shear modulus at point "1"	
31-40	PROP(20,N)	$G_{LD}^2 \dots$		
41-50	PROP(21,N)	G_{LD}^3		

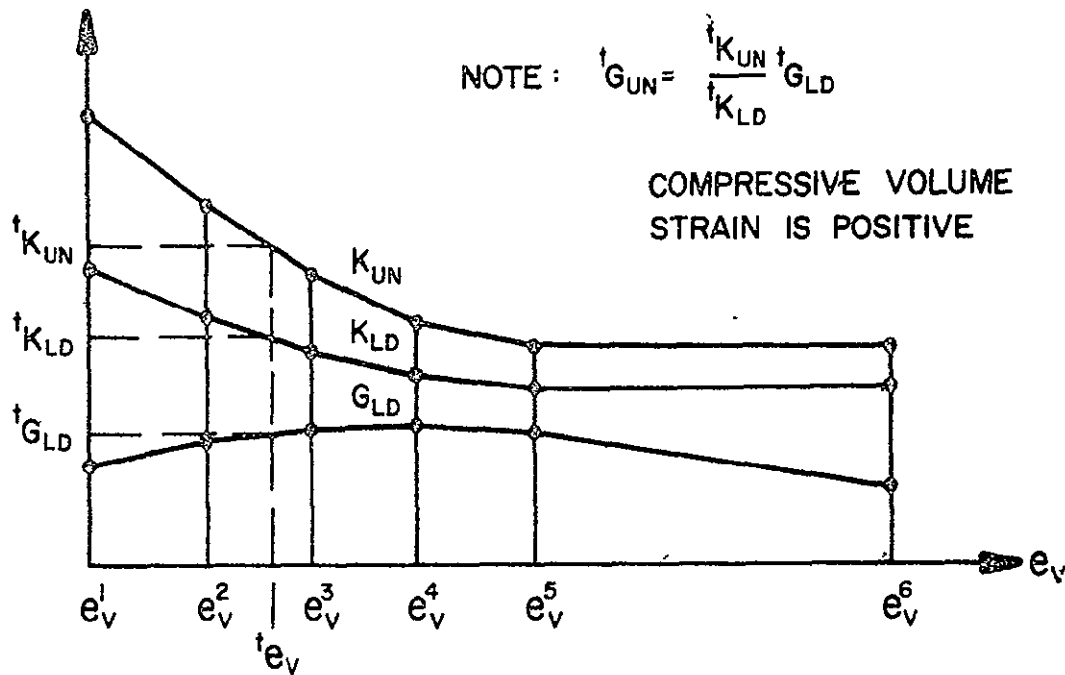
VI. 3/D CONTINUUM ELEMENTS (continued)

column	variable	description	notes
51-60	PROP(22,N)	G_{LD}^4	
61-70	PROP(23,N)	G_{LD}^5	
71-80	PROP(24,N)	G_{LD}^6	

NOTES

(1) MODEL 3 is a nonlinear library material model for which NPAR(17).EQ.24, be default. To characterize history, thirteen (13) variables--namely, the previously calculated stress and strains, and the maximum volumetric strain ever reached in loading--need be saved for each integration point. Therefore, NPAR(18) is set to "13" by default.

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



Note that

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

where "J" is the J-th table input point. Linear interpolation is used to define the loading and unloading bulk moduli, $t_{K_{LD}}$, $t_{K_{UN}}$, and the

VI. 3/D CONTINUUM ELEMENTS (continued)

loading shear modulus, $t_{G_{LD}}$, at time t between input points. If the volumetric strain during any time or load step exceeds e_V^6 , a diagnostic message results.

- (2) The unloading shear modulus at time t is calculated as

$$t_{G_{UN}} = (t_{K_{UN}}/t_{K_{LD}})t_{G_{LD}}.$$

NOTATION

For the notation used in the description of the nonlinear element models refer to the report, "Static and Dynamic Geometric and Material Non-linear Analysis," SESM Report No. 74-4, Feb. 1974 (Ref. 3).

VI. 3/D CONTINUUM ELEMENTS (continued)

MODEL 4 - Combined isotropic-kinematic hardening, von Mises yield condition.

	column	variable	description	notes
CARD 1	1-10	PROP(1,N)	Young's modulus, E	(1)
	11-20	PROP(2,N)	Poisson's ratio, ν	(1)
	21-30	PROP(3,N)	Number of points in piecewise linear <u>Stress Strain Curve</u> , RNSSC	(2)
	31-40	PROP(4,N)	Number of points in piecewise yield surface size vs equivalent plastic curve, RNSIZE	(3)
	41-50	PROP(5,N)	Type of hardening rule being used, TPLAS	(4)
	51-60	PROP(6,N)	Ratio of isotropic hardening to kinematic hardening, RITK	(5)
	61-70	PROP(7,N)	Value of stress at first point in stress strain curve, $\bar{\sigma}_1$	(6)
	71-80	PROP(8,N)	Value of total strain at first point in stress strain curve, $\bar{\epsilon}_1$	(6)
CARDS 2,3,4	1-10	PROP(9,N)	Value of stress at second point in stress strain curve, $\bar{\sigma}_2$	
	11-20	PROP(10,N)	Value of strain at second point in stress strain curve, $\bar{\epsilon}_2$	
	:	:		
	:	:		
	:	:		
		PROP(5+2* RNSSC,N)	Value of stress at last point in stress strain curve, $\bar{\sigma}_{RNSSC}$	
		PROP(6+2* RNSSC,N)	Value of strain at last point in stress strain curve, $\bar{\epsilon}_{RNSSC}$	
		PROP(7+2* RNSSC,N)	Value of yield surface size at first point (7) in yield surface size vs equivalent plastic strain curve, k_1 ,	(7)
		PROP(8+2* RNSSC,N)	Value of equivalent plastic strain at first point in yield surface size vs equivalent plastic strain curve, $\bar{\epsilon}_1^p$	

VI. 3/D CONTINUUM ELEMENTS (continued)

MODEL 4 (continued)

column	variable	description	notes
:	:	:	
:	:	:	
:	:	:	
	PROP(5+2* RNSSC+2*RNSIZE ,N)	Value of yield surface size at last point in yield surface size vs equivalent plastic strain curve, k	RNSIZE
	PROP(6+2* RNSSC+2*RNSIZE ,N)	Value of equivalent plastic strain at last point of yield surface size vs equivalent plastic strain curve, $\frac{y_p}{\epsilon_p}$	RNSIZE

NOTES

- (1) MODEL 4 is nonlinear library material model for which NPAR(17).EQ.26 by default. NPAR(18).EQ.21, since 21 history dependent variables must be stored for each integration point.

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

- (2) RNSSC is the number of points in a piecewise linear stress strain curve, and includes the end point of the last linear segment, but not the point STRESS=0, STRAIN=0. Maximum allowable value is RNSSC=5, the minimum value is RNSSC=2.
- (3) RNSIZE is the number of points in a piecewise linear yield surface size vs equivalent plastic strain curve, and includes both the end point of the last linear segment as well as the point STRESS=yield stress, EQPLAS=0. RNSIZE=0 for TPLAS.NE.3.
- (4) TPLAS EQ.0 Isotropic hardening
 EQ.1 Kinematic hardening (Prager-Ziegler)
 EQ.2 Combined isotropic, kinematic hardening, constant ratio
 EQ.3 Combined isotropic, kinematic hardening, variable ratio

VI. 3/D CONTINUUM ELEMENTS (continued)

- (5) RITK is the ratio of isotropic to kinematic hardening for TPLAS.EQ.2, with RITK.EQ.1 being pure isotropic, RITK.EQ.0 pure kinematic. If TPLAS.NE.2, RITK is ignored.
- (6) Values of STRESS and STRAIN are input in pairs for the piecewise linear stress strain curve using the format 8F10.0. A maximum of five (5) pairs of data is allowed. Input RNSSC pairs of data. The input stress-strain curve must be monotonically increasing such that $PROP(5+2*I) \leq PROP(5+2*(I+1))$ and $PROP(6+2*I) \leq PROP(6+2*(I+1))$ for any I. See Fig. 9 also.
- (7) Values of YSSIZE (Yield Surface SIZE) and EQPLAS (Equivalent PLASTic strain) are input in pairs for the piecewise linear yield surface size vs equivalent plastic strain curve. Data for this curve need not be input unless TPLAS.EQ.3. For TPLAS.LT.3, a compatible curve is generated internally in the program, appropriate for the hardening rule indicated by TPLAS. See Fig. 9 also.

NOTE: Regardless of the value of RNSSC and RNSIZE, four cards will be read on 8F10.0 format to generate the PROP array. Blank entries for a property value will be read and ignored after the last non-zero entry. FOUR CARDS MUST BE SUPPLIED.

VI. 3/D CONTINUUM ELEMENTS (continued)

MODEL 5 - mechanical sublayer model, von Mises yield condition

column	variable	description	notes	
RD 1	1-10	PROP(1,N) Young's modulus, E	(1)	
	11-20	PROP(2,N) Poisson's ratio, ν	(1)	
	21-30	PROP(3,N) Number of points in piecewise linear <u>Stress Strain Curve</u> , RNSSC	(2)	
	31-40	PROP(4,N) Value of stress at first point in stress strain curve, $\bar{\sigma}_1$	(2)	
	41-50	PROP(5,N) Value of strain at first point in stress strain curve, $\bar{\epsilon}_1$	(2)	
		.		
		.		
		.		
		PROP(8,N)	Value of stress at third point in stress strain curve	
	RD 2	PROP(9,N)	Value of strain at third point in stress strain curve	
.	.			
.	.			
.	.			
.	.			
	PROP(2+2*RNSSC, N)	Value of stress at last point in stress strain curve, $\bar{\sigma}_{RNSSC}$		
	PROP(3+2*RNSSC, N)	Value of Strain at last point in stress strain curve, $\bar{\epsilon}_{RNSSC}$		

NOTES

(1) MODEL 5 is a nonlinear library material model for which NPAR(17).EQ.11 by default. NPAR(18).EQ.30, since 30 history dependent variables must be stored for each integration point.

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

VI. 3/D CONTINUUM ELEMENTS (continued)

MODEL 5 (continued)

NOTES

- (2) RNSSC is the number of points in a piecewise linear stress strain curve, and includes the end point of the last linear segment (unless that segment is perfectly plastic), but not the point STRESS=0, STRAIN=0. The maximum allowable value is RNSSC=4.0. If RNSSC.LE.2.0, a blank card B must still be supplied. It is also required that the input stress strain curve be monotonically increasing so that $PROP(2+2*I).LT.PROP(2+2*(I+1))$ and $PROP(3+2*I).LT.PROP(3+2*(I+1))$. Note that the inequality reads .LT., not .LE. See Fig. 10 also.

VI. 3/D CONTINUUM ELEMENTS (continued)

MODEL 6 - Incompressible Nonlinear elastic, Mooney-Rivlin material

column	variable		description	notes
1-10	PROP(1,N)	C_1	C_1 and C_2 are material constants in the Mooney-Rivlin expression for strain energy.	(1)
11-20	PROP(2,N)	C_2		

NOTES

- (1) MODEL 6 is a nonlinear model for which NPAR (17) is set to "2". Since the material is hyperelastic, no history-dependent variables need be stored at integration points and NPAR(18) is set to "0". C_1 and C_2 are material constants in the expression for strain energy,

$$U = C_1(I_1-3) + C_2(I_2-3)$$

where I_1 and I_2 are the first and second strain invariants, respectively. A more complete theoretical description of the model is given in Ref. 5.

VI. 3/D CONTINUUM ELEMENTS (continued)

MODEL 7 - user-supplied material model

column	variable	description	notes
1-10	PROP(1,N)		(1)
11-20	PROP(2,N)		
21-30	PROP(3,N)		
.			
.	etc.		

NOTES

- (1) MODEL 7 is a non-standard model (i.e., is not in the material model library) and requires that property data be read

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

where NPAR(17) is user-supplied (i.e., no default value exists). Subroutines which allocate storage locations (in "WA") for element history (if the model is nonlinear) and which calculate the incremental stress-strain law and current stresses must be supplied.

VI. 3/D CONTINUUM ELEMENTS (continued)

VI.3. Stress Output Table Cards (16I5)*

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

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

NOTES

- (1) Stress tables are defined to provide flexibility in element stress output requests. Each element can refer to one of the tables defined, and the element stresses are then calculated at the points specified in that table. Figure 15 defines the locations of the 27 possible points within an element where stresses may be printed.

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

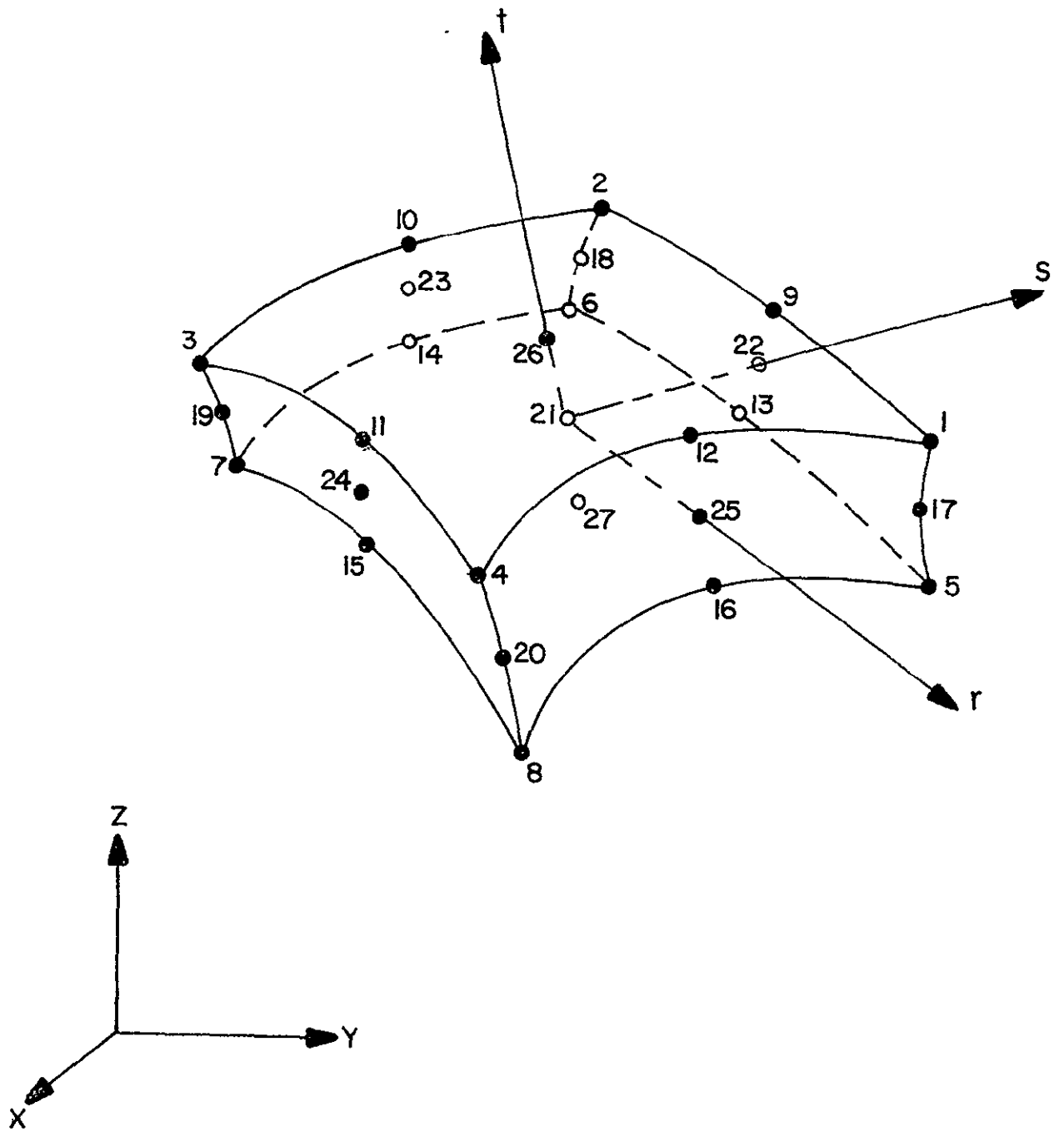


Fig. 15 Stress Output Points for Three-Dimensional Elements*

VI. 3/D CONTINUUM ELEMENTS (continued)

NOTES

stress table is referred to.

Stress output tables are used only with linear material models (NPAR(15).EQ.1 or 2). For elements using nonlinear material models (NPAR(15).GE.3), stresses can be printed at integration points only.

VI. 3/D CONTINUUM ELEMENTS (continued)

VI.4. Element Data Cards*

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

CARD 1 (7I5, 3F10.0)

column	variable	description	notes
1-5	M	3/D CONTINUUM element number GE.1 and LE.NPAR(2)	(1)
6-10	IELD	Number of nodes to be used in describing the element's displacement field EQ.0 - default set to NPAR(7) LE.NPAR(7)	(2)
11-15	IELX	Number of nodes to be used in the description of the element's geometry EQ.0 - default set to IELD GE.8 and LE.IELD EQ.IELD → isoparametric element LT.IELD → subparametric element	(3)
16-20	IPS	Number of the stress output location table to be used for this element (if a linear material model is used) or flag for printing stresses at element <u>integration</u> points EQ.0 - no stress output for this element	(4)
21-25	MTYP	Material property set number assigned to this element GE.1 and LE. NPAR(16)	
26-30	IST	Flag indicating that the stiffness and mass matrices for this element are the same as those for the preceding element EQ.0 - no EQ.1 - yes	(5)
31-35	KG	Node number increment for element data generation EQ.0 - default set to "1"	(6)
36-45	ALPHA	Angle α (degrees) used to define principal material axes for Model 2 (orthotropic)	(7)
46-55	BETA	Angle β (degree) used to define principal material axes for Model 2 (orthotropic)	
56-65	GAMMA	Angle γ (degrees) used to define principal material axes for Model 2 (orthotropic)	

VI. 3/D CONTINUUM ELEMENTS (continued)

CARD 2 (8I5)

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

CARD 3 (13I5)

column	variable	description	notes
1-5	NOD(9)	Global node number of element nodal point 9	(2)
6-10	NOD(10)	Global node number of element nodal point 10	
11-15	NOD(11)	Global node number of element nodal point 11	
16-20	NOD(12)	Global node number of element nodal point 12	
21-25	NOD(13)	Global node number of element nodal point 13	
26-30	NOD(14)	Global node number of element nodal point 14	
31-35	NOD(15)	Global node number of element nodal point 15	

VI. 3/D CONTINUUM ELEMENTS (continued)

column	variable	description	notes
36-40	NOD(16)	Global node number of element nodal point 16	
41-45	NOD(17)	Global node number of element nodal point 17	
46-50	NOD(18)	Global node number of element nodal point 18	
51-55	NOD(19)	Global node number of element nodal point 19	
56-60	NOD(20)	Global node number of element nodal point 20	
61-65	NOD(21)	Global node number of element nodal point 21	

NOTES

- (1) Element cards must be input in ascending element number order beginning with one (1) and ending with NPAR(2). Repetition of element numbers is illegal, but element cards may be omitted, and missing element data are generated according to the procedure described in note (6).
- (2) IELD is a count of the node numbers actually posted on Cards 2 and 3 which must immediately follow Card 1. IELD must be at least eight (8), but must be less than or equal to the limit NPAR(7) which was given on the element group control card, Section VI.1. Element displacements are assigned at the IELD non-zero nodes, and thus the order of the element matrices is three (i.e., translations X,Y,Z) times IELD. The eight corner nodes of the hexahedron must be input, but nodes 9 to 21 are optional, and any or all of these optional nodes may be used to describe the element's displacement field. However, all 21 entries for NOD(i) are read from element data cards 2 and 3; if IELD.LT.21 the particular node locations not used in this element must be input as zero (0) in NOD(i). Figure 16 defines the input sequence that must be observed for element input.

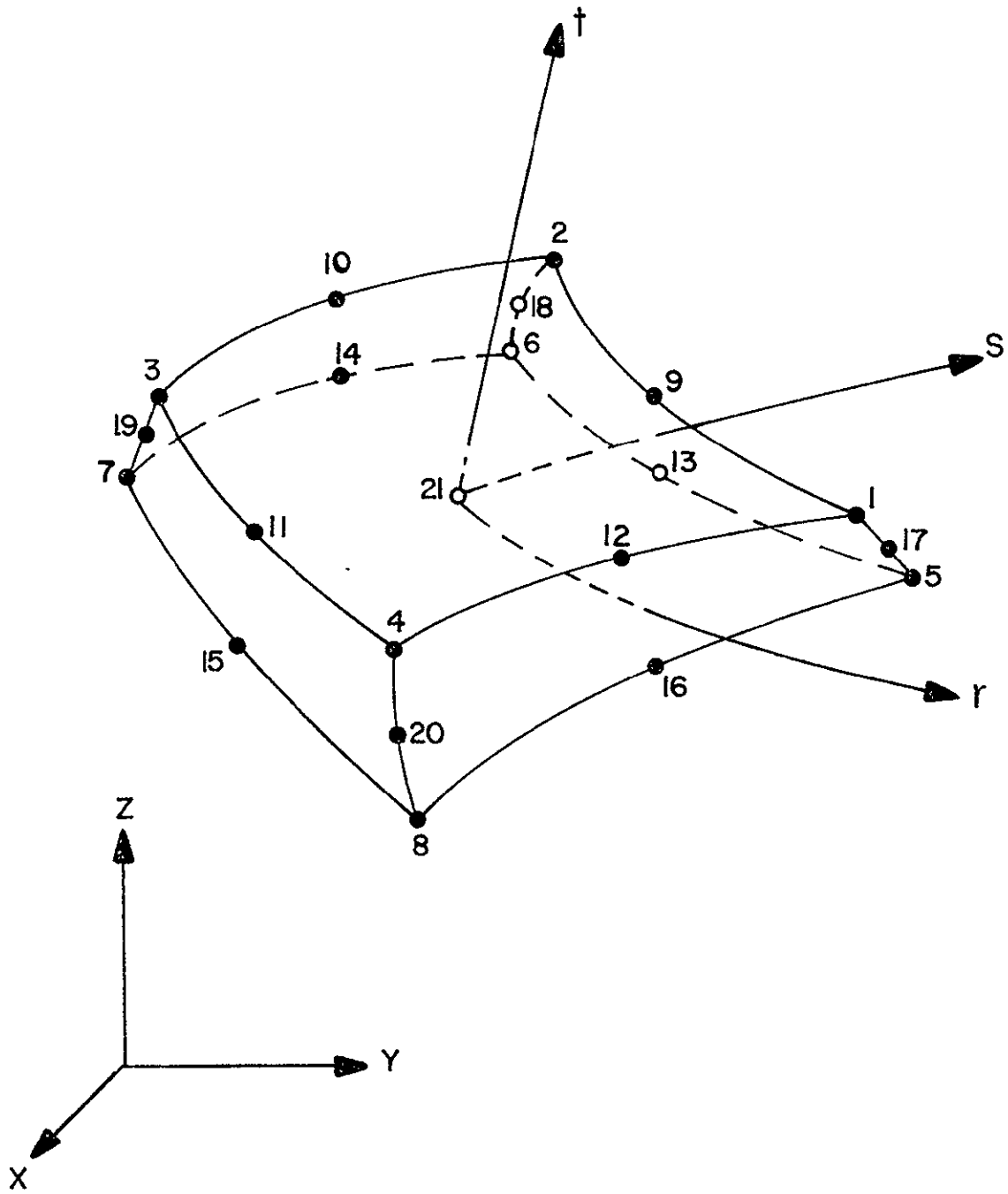
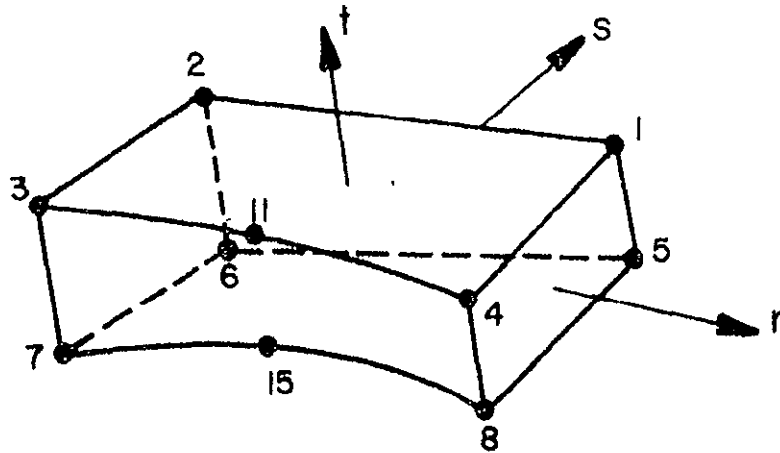


Fig. 16 Element Node Number Sequence for Three-Dimensional Elements

VI. 3/D CONTINUUM ELEMENTS (continued)

NOTES

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



is defined by

$$\text{NOD}(1) = [X \ X \ X \ X \ X \ X \ X \ X \ 0 \ 0 \ X \ 0 \ 0 \ 0 \ X \ 0 \ 0 \ 0 \ 0 \ 0 \ 0],$$

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

- (3) When element edges are straight it is unnecessary computationally to include side nodes in the numerical evaluation of coordinate derivatives, the Jacobian matrix, etc., and since regular element shapes are common, an option has been included to use fewer nodes in these geometric calculations than are used to describe element displacements. The first IELX nonzero nodes posted on Cards 2 and 3 are used to evaluate those parameters which pertain to element geometry only. IELX must be at least eight (8), and if omitted is re-set to IELD. A common application might be a 20 node element (i.e., IELD.EQ.20) with straight edges in which case IELX would be entered as "8".
- (4) If IPS.EQ.0 no stress output will be provided for this element. A nonzero entry for the element stress printout flag "IPS" has a different meaning when used with nonlinear material models than it does when used with linear material models:
 - a) For a linear material model (MODEL.EQ.1 or 2) "IPS" specifies the stress output table (Section VI.3) to be used for this element. If no stress output tables were input (i.e., NPAR(13).EQ.0) then IPS.EQ.1 specifies that stresses are to be printed

VI. 3/D CONTINUUM ELEMENTS (continued)

NOTES

at all integration points for this element.

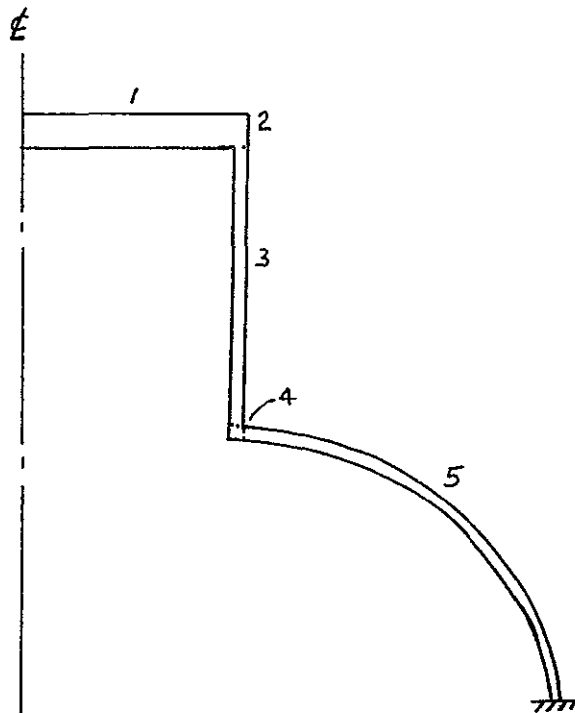
- b) For a nonlinear material model (MODEL.GE.3) IPS.EQ.1 specifies that stresses are to be printed at all integration points for this element.
- (5) The flag IST allows the program to bypass stiffness and mass matrix calculations providing the current element is identical to the preceding element; i.e., the preceding and current elements are identical except for a rigid body translation. If IST.EQ.0, new matrices are computed for the current element.
- (6) When element cards are omitted, element data are generated automatically as follows,
- a) All data on Card 1 for generated elements is taken to be the same as that given on the first element card in the sequence
 - b) Nonzero node numbers (given on Cards 2 and 3 for the first element) are incremented by the value "KG" (on the first element's Card 1) as element generation progresses; zero (or blank) node number entries are generated as zeroes.
- The last element cannot be generated and must be included.
- (7) The angles α , β and γ are required only for Model 2. They are defined in the notes for Model 2.

VII. NODE AND ELEMENT INPUT BY SUBSTRUCTURES

If the nodal coordinates, material property cards and element-node connectivity are to be input by using the substructure option (MGO=1), then this section (VII) is input in place of Section III-VI.

In brief, a complete structure is described as a collection of substructures. Each substructure is described by specifying its corner nodes and up to two midside nodes for each side, the number of mesh points to be generated in each coordinate direction, and various gradients to control the mesh spacing. Each substructure may be thought of as a single isoparametric element which is to be broken up into a number of smaller elements. The program then uses isoparametric shape functions to interpolate nodal coordinates and to generate element-node connectivity for all elements within the substructure. It is assumed that each element within the substructure has identical nonlinear material parameters, the same number of nodes, etc.

As an example, consider the pressure vessel shown below:



The above structure may be conveniently described by five substructures as shown. The required input would consist of the corner and midside nodes of each substructure plus information describing the number of mesh points in each of the coordinate directions (local element coordinates, say r and s) and the gradients controlling mesh size.

Enter one set of the cards in Section VII for each substructure. Linear substructures (small strain, elastic) must be input first followed by nonlinear substructures. Substructures must be input in sequence so that they attach to the right, top, or back (3-D) of a previously input substructure. The side of a substructure which attaches to a previously input substructure must have identical corner nodal coordinates and number of nodes along the matching boundary, i.e., the entire side must be matched.

VII.1 Control Card (10I5, 6F5.2)

column	variable	description	notes
1-5	NPAR(1)	Element Type EQ.1 - truss EQ.2 - 2-D continuum EQ.3 - 3-D continuum	(1)
6-10	NPAR(3)	Geometric nonlinear analysis type EQ.0 - small strain (linear) EQ.1 - large strain (total Lagrangian) EQ.2 - large strain (updated Lagrangian)	(2)
11-15	NPAR(5)	Element type code (2-D only) EQ.0 - axisymmetric EQ.1 - plane strain EQ.2 - plane stress	
16-20	NPAR(10)	Integration order (for r-s plane) EQ.0 - default reset to 2 GE.2 and LE.4	(3)
21-25	NPAR(11)	Integration order for t direction (3-D only) EQ.0 - default reset to 2 GE.2 and LE.4	(4)
26-30	NPAR(15)	Material model number (see table)	(5)
31-35	ITYPE	Number of boundary nodes used to describe this substructure: For 2-D; ITYPE=4 (linear), 8 (quadratic) or 12 (cubic) For 3-D, ITYPE=8 (linear), 20 (quadratic) or 32 (cubic)	(6)
36-40	IMESH	Number of mesh points to be generated in X direction (3-D only)	(7)
41-45	JMESH	Number of mesh points to be generated in Y direction	
46-50	KMESH	Number of mesh points to be generated in Z direction	
51-55	G1	Gradient used to determine mesh spacing from node 1 to node 2 of substructure EQ.0.0 - default reset to 1.0	(8)
56-60	G2	Gradient used to determine mesh spacing along top, front edge from node 1 to node 4 EQ.0.0 - default reset to 1.0	

VII.1 CONTROL CARD (10I5, 6F5.2) (continued)

column	variable	description	notes
61-65	G3	Gradient used to determine mesh spacing along top, left edge from node 4 to node 3 EQ.0.0 - default reset to 1.0	
66-70	G4	Gradient used to determine mesh spacing along top, rear edge from node 2 to node 3 EQ.0.0 - default reset to 1.0	
71-75	G5	Gradient used to determine mesh spacing along edge from node 1 to node 5 EQ.0.0 - default reset to 1.0	
76-80	G6	Gradient used to determine mesh spacing along edge from node 4 to node 8 EQ.0.0 - default reset to 1.0	

NOTES

- (1) All elements that are generated within this substructure will be of the same type, i.e., will have the same parameters for NPAR(1), NPAR(3), NPAR(5), NPAR(10), NPAR(11), NPAR(15), ITYPE, the same material model properties, etc.
- (2) NPAR(3) determines whether geometric nonlinearities (large displacements and large strains) are to be included in the analysis.
- (3) For rectangular elements, a Gaussian numerical order of 2 is generally sufficient. For distorted elements (curved boundary or skewed sides), a higher integration order is needed to accurately represent the volume integration. It should be noted that the computation time for stiffness and force vectors is strongly dependent on the integration order. Thus, 3 by 3 integration requires more than twice the computer time that 2 by 2 integration requires.
- (4) This entry is required only if this substructure represents 3-D elements (NPAR(1).EQ.3). For some applications such as plates or shells where the strain varies approximately linearly through the "thickness", the integration order through the thickness may be less than over the surface (r-s plane) of the shell.
- (5) The available material models are tabulated in Table 2.
- (6) The substructure can be thought of as a large isoparametric element whose substructure geometry is described by boundary nodes in exactly the same manner as a single element is described. The elements generated within a substructure are given the same number of nodes as the parent substructure. Hence, a linear substructure (4 nodes for the

VII.1 CONTROL CARD (10I5, 6F5.2) (continued)

NOTES

the 2-D case and 8 nodes for the 3-D case) should be used only if it is desired to generate linear elements within the substructure.

- (7) IMESH, JMESH, and KMESH specify the number of nodal points to be generated in the r,s and t directions, respectively. If EQ.0, no mesh is generated and the substructure is treated as a single element. When a substructure boundary is to match or mate together with a previously input substructure, the number of generated nodes must be identical on the adjacent boundaries. Subsequently input substructures must mate on the top, right or back boundaries of previously input substructures. See Figs. 17 and 18.

- (8) The gradient parameters G1 through G6 are used to determine size variation of the generated mesh (node points). If the gradient is greater than unity, the element size will decrease away from the origin. If the gradient is less than unity, the element size will increase away from the origin. Examples of the generated mesh for various gradients are shown in Figure 18.

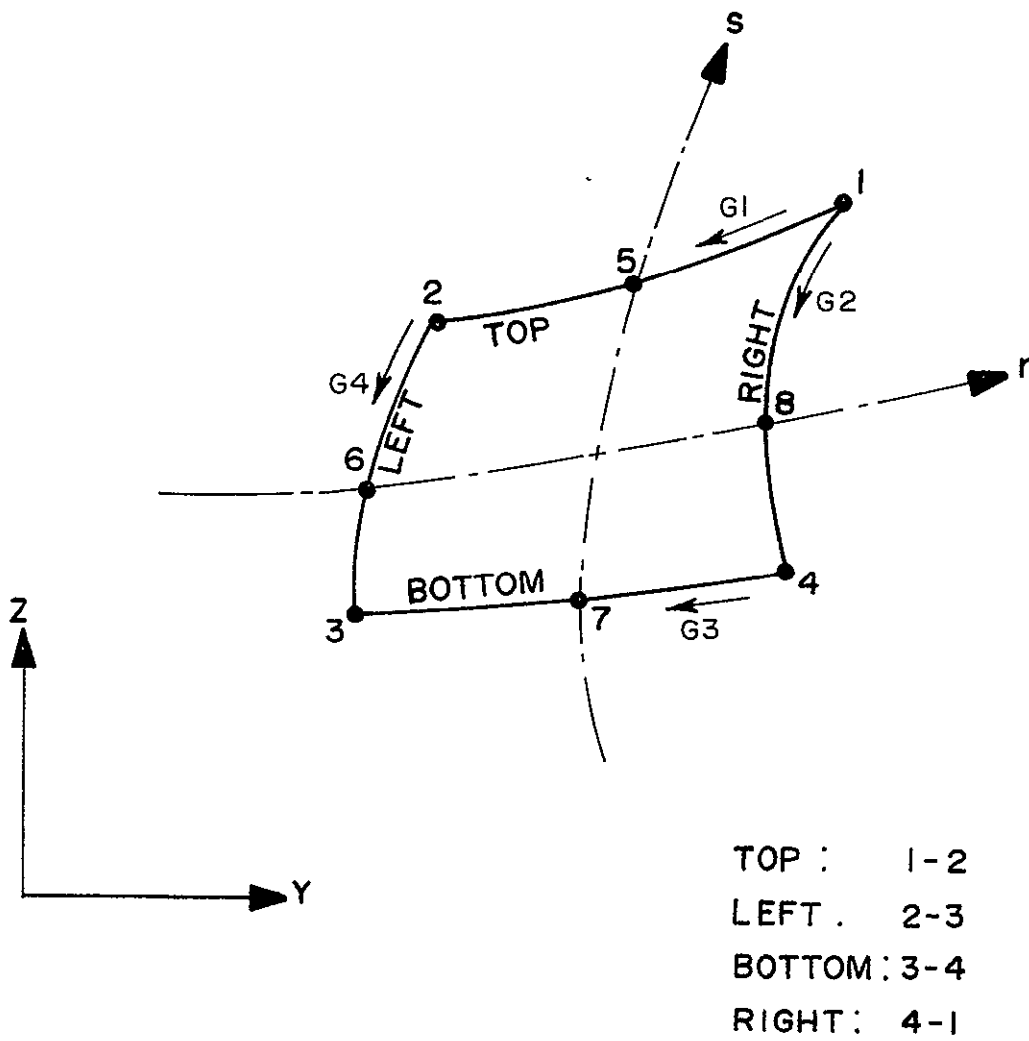


Fig. 17' Gradient Parameters for Two-Dimensional Substructure

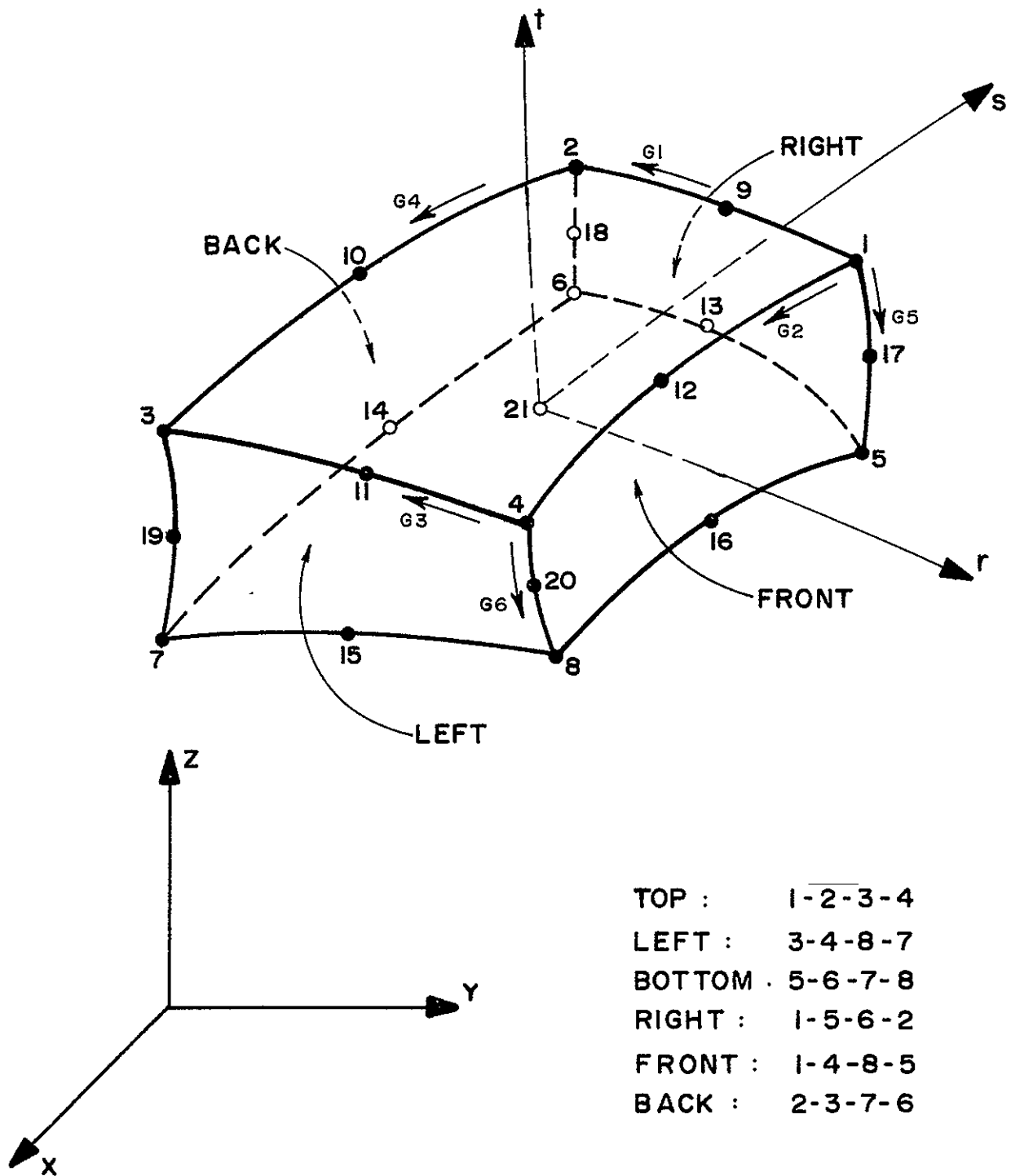


Fig. 18 Gradient Parameters for Three-Dimensional Substructure

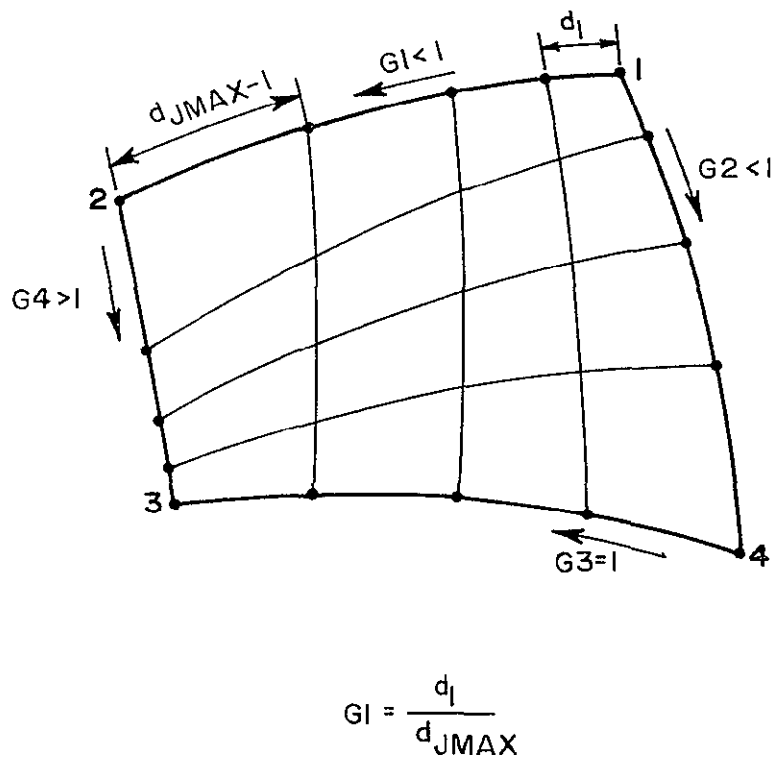


Fig. 19 Example of Generated Two-Dimensional Mesh

VII.2 Material Property Control Card (5F10.0, 4I5)-OMIT THIS CARD FOR A TRUSS ELEMENT GROUP (NPAR(1)=1).

column	variable	description	notes
1-10	THICK	Thickness of element (plane stress only)	(1)
11-20	DENSE	Mass density (required only in dynamic analyses or if body forces are included)	(2)
21-30	ALPHA or BETA	Dual meaning entry depending upon material model: 3-D, Model 2 - ALPHA - angle α (degrees) used to define principal material axes 2-D, Model 2 - BETA - orthotropic material angle β	(3) (4)
31-40	BETA or STIFAC	Dual meaning entry depending upon material model: 3-D, Model 2 - BETA - angle β (degrees) used to define principal material axes 2-D, Model 4,5 - STIFAC - Stiffness reduction factor after fracture	(3) (5)
41-50	GAMMA or SHEFAC	Dual meaning entry depending upon material model: 3-D, Model 2 - GAMMA - Angle γ (degrees) used to define principal material axes 2-D, Model 4,5 - SHEFAC - Shear reduction factor after fracture	(3) (5)
51-55	IPS	Stress print out flag. For linear material models, refers to number of stress output table to be used for all elements in this group. For non-linear material models, flag for printing stresses at integration points. EQ.0 - no stress output for this substructure	(6)
56-60	NPAR(16)	Number of material property sets (must be 1 for MGO=1 option)	(7)
61-65	NPAR(17)	Number of constants per property set (required only for used supplied material models)	(8)

VII.2 Material Property Control Card (5F10.0, 4I5) (continued)

column	variable	description	notes
66-70	NPAR(18)	Dimension of storage array required for element history-dependent variables (required only for user supplied material models).	(8)

NOTES

- (1) The element thickness is required for plane stress elements. Plane strain elements are assumed to have a thickness of one unit and axisymmetric elements have a thickness of one radian.
- (2) The mass density is used in dynamic analyses to compute the mass matrix. It is also used in 2-D analyses when body forces are included. The mass density must be input in proper units; no conversion factors are applied by the program.
- (3) The angles α , β and γ are required only for 3-D, Model 2 only. They are defined in the notes for this material model (Section VI.2.b, Model 2).
- (4) The orthotropic material angle β is required for 2-D, Model 2 only. It is defined further in the notes for this material model (Section V.2.b, Model 2).
- (5) STIFAC and SHEFAC are assumed material parameters used in conjunction with 2-D analyses with MODEL=4 or 5. See the notes for these parameters in Section V.2.b (MODEL 4).
- (6) If IPS.EQ.0, no stress output will be provided for the elements in this substructure. A nonzero entry has a different meaning when used with linear or nonlinear material models as follows:

For a linear material model (MODEL.EQ.1 or 2), IPS specifies the stress output table to be used for elements in this substructure. If no stress output tables were input (i.e., NPAR(13).EQ.0), then IPS.EQ.1 means that stressea are to be printed at all integration points of each element.

For a nonlinear material model (MODEL.GE.3), IPS,EQ.1 specifies that stressea are to be printed at all integration points of each element.
- (7) It is assumed that each element in the substructure has identical material properties and therefore NPAR(16) = 1.
- (8) For standard material models, in the program, these entries may be omitted. For user supplied models, the notes regarding these entries in Section V.1 and VI.1 should be consulted.

VII.3 Material Property Cards (and Stress Output Tables)

Material property cards are required for the particular material model (NPAR(15)) being utilized in this substructure. The required input is identical to that described in Sections IV.2, V.2, and VI.2. Depending upon the type of element, the material property card and stress output table input described in the following sections should be input:

- 1-D: Sec. IV.2.b, Models 1-2
- 2-D: Sec. V.2.b and V.3, Models 1-10
- 3-D: Sec. VI.2.b and VI.3, Models 1-6

VII.4. Nodal Coordinates of Substructure (I1, I3, 3I1, 3F20.0)

The following card is repeated for as many nodes as are being used to describe this substructure (ITYPE on Card VII.1).

column	variable	description	notes
1	IT	Coordinate system used to describe this node EQ.0 - cartesian (X,Y,Z) EQ.1 - X-cylindrical	(1)
2-4	N	Node number	(2)
5	IXTRAN	X-translation boundary condition code EQ.0 - free EQ.1 - fixed	(3)
6	IYTRAN	Y-translation boundary condition code	
7	IZTRAN	Z-translation boundary condition code	
11-30	X(N)	X coordinate; or Z coordinate if IT.EQ.1 (used only in 3-D)	(4)
31-50	Y(N)	Y coordinate; or R coordinate if IT.EQ.1	
51-70	Z(N)	Z coordinate; or θ coordinate if IT.EQ.1	

NOTES

- (1) Nodal coordinates may be input in either the global, cartesian or X-cylindrical coordinate systems. A complete description of the X-cylindrical coordinate system is given in Note 1 following Section III.1.
- (2) Node numbers must range from 1 to ITYPE (Sec. VII.1) but need not be input in ascending order. Figures 17 and 18 define the node numbering conventions that must be followed.
- (3) Boundary condition codes are used to delete (fix) certain degrees-of-freedom. If the same boundary conditions are specified for all the nodes on one edge of an element, then these boundary conditions are applied to all nodes generated by the program on that edge. If, for example, boundary conditions are applied only to one node on a particular edge, then these boundary conditions are retained for that node but all other generated nodes on that edge are given free boundary conditions.

VII.4. Nodal Coordinates of Substructure (continued)

NOTES

- (4) The nodal coordinates should be input in the proper coordinate system as prescribed by the coordinate system flag IT in CC 1. The origin of the global, cartesian and cylindrical coordinate systems are the same. It should be noted that the boundary condition codes always refer to degrees-of-freedom in the global X-Y-Z system even if the nodal coordinates were given in the cylindrical coordinate system.

VIII. LOAD CURVES (continued)

Hence, the definition of the applied pressure (a constant) and the magnitude of the load curves points is arbitrary. The only requirement is that the product of the constant pressure and the load curve multiplier evaluated at a particular time yield the correct magnitude that is to be applied at that time.

IX. NONCONSERVATIVE PRESSURE LOADS

Omit these cards if NPRESN.EQ.0; otherwise, input NPRESN cards.

Nonconservative pressure loads are assumed to be functions of the deformation, i.e., normal pressure loads remain normal to the deformed surface, etc. Nonconservative pressure loads can be applied only to elements for which large deformations (strains) are being considered. The same nonconservative pressure load, i.e., one card, may be used to describe loads being applied to one element, to a set of elements within a group or to an entire group (substructure).

column	variable	description	notes
1-5	IGROUP	Substructure or element group to which load is applied	(1)
6-10	IELM1	First element to which load is applied (ignored if MGO=1)	(2)
11-25	IELM2	Last element to which load is applied (ignored if MGO=1)	(2)
18-25	BONDRY	Boundary of substructure of element to which load is applied. Entry must be right justified: TOP LEFT BOTTOM RIGHT FRONT } only in 3-D elements BACK }	(3)
26-30	ICUR	Load curve number according to which the load is applied	(4)
31-40	PRESN	Normal pressure (positive if tensile)	(5)
41-50	PRESS	Shear pressure (positive according to Timoshenko notation) - shear pressure not allowed in 3-D elements	(5)

NOTES

- (1) IGROUP must be a valid group number, i.e., $1 \leq \text{IGROUP} \leq \text{NUMEG}$. The same group number may be repeated if necessary
- (2) IELM1 and IELM2 define an inclusive set of elements to which the pressure is to be applied. When describing the mesh by substructures, the program assumed that all elements in substructures IGROUP are loaded, i.e., these parameters are ignored if MGO=1.

VIII. LOAD CURVES*

If no load curves are to be input, omit the cards in Sec. VII. Otherwise, input one card VIII.1 followed by one or more cards VIII.2; then repeat this sequence until data for NLCUR load curves has been input.

VIII.1. Control card (I5)

column	variable	-description	notes
1-5	NPTS	Number of points used to input this load curve GE.2 and LE.100	

VIII.2. Load Curve Multiplier Data (8F10.0)

column	variable	description	notes
1-10	TIMV(1)	Load (time) at point 1	(1)
11-20	RV(1)	Load multiplier at point 1	
21-30	TIMV(2)	Load (time) at point 2	
31-40	RV(2)	Load multiplier at point 2	
.	.	.	.
.	.	.	.
71-80	RV(4)		
		Next card (if required)	
1-10	TIMV(5)		
11-20	RV(5)		
.	.	.	.
.	.	.	.
.	.	.	.

NOTES

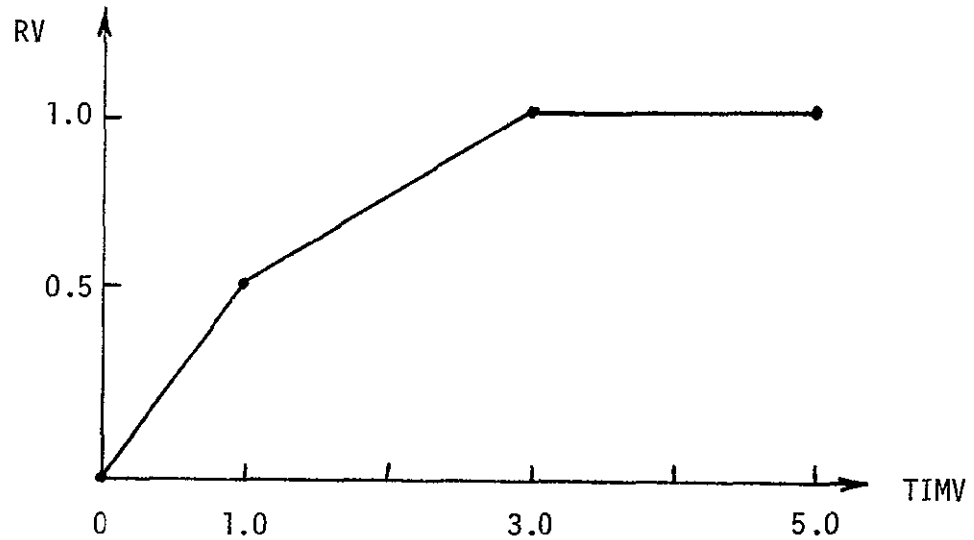
- (1) The data pairs [TIMV(i), RV(i)] define load multiplier functions which describe the variation of applied loads with time (or a load parameter in static problems). The magnitude of the load at any time (load parameter) is obtained by linearly interpolating the function and multiplying this times a constant input pressure assigned to an

VIII. LOAD CURVES (continued)

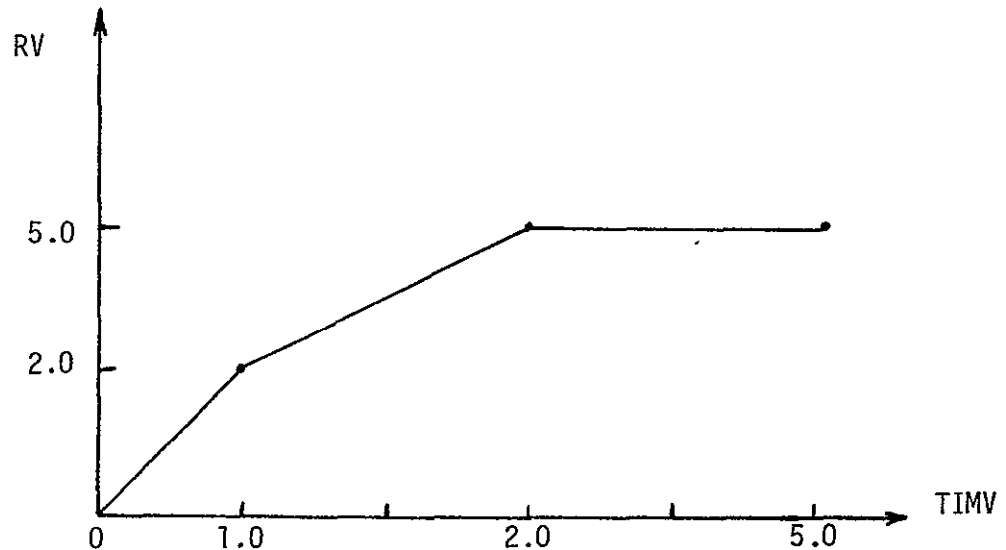
NOTES

element or a force assigned to a node (as input on cards IX, X, or XI). The first time, i.e., TIMV(1) must be equal to zero. The last time value for any curve, i.e., TIMV(NPTS) must be greater than or equal to the time at the end of the solution, i.e., $TIMV(NPTS) \geq TSTART + NSTE * DT$.

A typical load curve is shown below. The number



of points to be input is four. Suppose a certain element has a pressure of 500 psi which is applied according to the above load curve. At a time (load) of 1.0, the magnitude of the pressure will be $0.5 \times 500 = 250$ psi. At a time (load) of 2.0, the magnitude will be $0.75 \times 500 = 375$ psi, etc. Note that the same magnitudes would be obtained if the pressure specified for the element were 100 psi and the load curve was input as



IX. NONCONSERVATIVE PRESSURE LOADS (continued)

NOTES

- (3) The pressure load on this card is applied to the face of an element (elements or substructure) as shown in Figs. 20 & 21. Note that the faces are defined with respect to the element axes (r,s,t) which are in turn defined by the element node numbers
- (4) The pressure on this card is applied to the element face with a time variation described by load curve ICUR. Note that $1 \leq ICUR \leq NLCUR$.
- (5) The normal and shear pressure loads are positive as shown in Figs. 20 and 21.

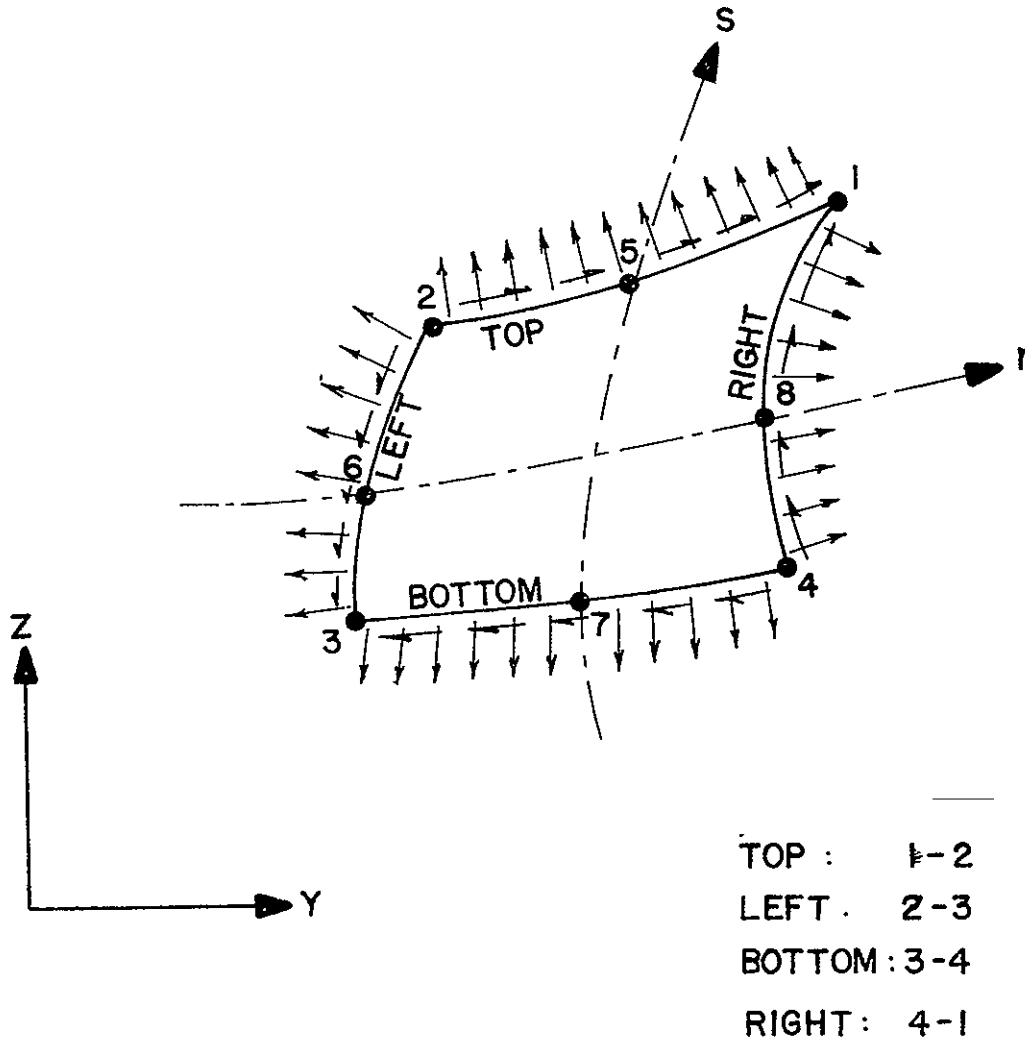


Fig. 20 Nomenclature for Inputting Pressure Distribution on Two-Dimensional Element

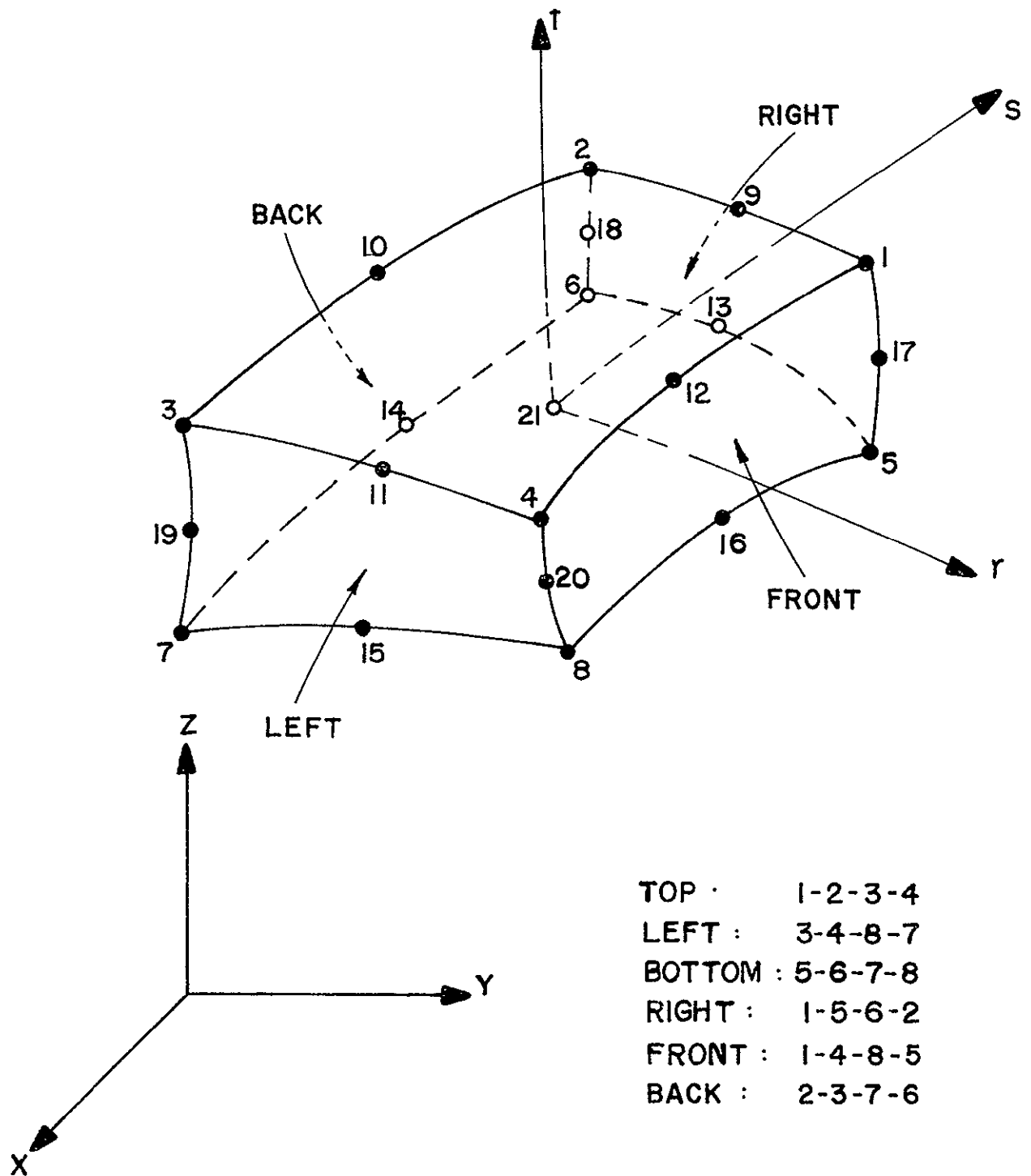


Fig. 21 Nomenclature for Inputting Pressure Distribution on Three-Dimensional Element

X. CONSERVATIVE PRESSURE LOADS

Omit these cards if NPRESQ.EQ.0; otherwise, input NPRESQ cards.

Conservative pressure loads are assumed to be independent of deformation.

The same conservative pressure load, i.e., one card, may be used to describe loads being applied to one element, to a set of elements within a group, or to an entire group (substructure).

column	variable	description	notes
1-5	JGROUP	Substructure or element group to which load is applied	(1)
6-10	IELM1	First element to which load is applied (ignored if MGO=1)	(2)
11-15	IELM2	Last element to which load is applied (ignored if MGO=1)	(2)
18-25	BONDRY	Boundary of substructure or element to which load is applied. Entry must be right justified: TOP LEFT BOTTOM RIGHT FRONT } only in 3-D elements BACK }	(3)
26-30	ICUR	Load curve number according to which the load is applied	(4)
31-40	PRESN	Normal pressure (positive if tensile)	(5)
41-50	PRESS	Shear pressure (positive according to Timoshenko notation) - shear pressure not allowed in 3-D elements	(5)

NOTES

- (1) JGROUP must be a valid group number, i.e., $1 \leq \text{JGROUP} \leq \text{NUMEG}$. The same group number may be repeated if necessary.
- (2) IELM1 and IELM2 define an inclusive set of elements to which the pressure is to be applied. When describing the mesh by substructures, the program assumed that all elements in substructures IGROUP are loaded, i.e., these parameters are ignored if MGO=1.
- (3) The pressure load on this card is applied to the face of an element

X. CONSERVATIVE PRESSURE LOADS (continued)

NOTES

(elements or substructure) as shown in Fig. IX.1. Note that the faces are defined with respect to the element axes (r,s,t) which are in turn defined by the element node numbers as shown in Fig. IX.1.

- (4) The pressure on this card is applied to the element face with a time variation described by load curve ICUR. Note that $1 \leq ICUR \leq NCLUR$.
- (5) The normal and shear pressure loads are positive as shown in Figs. 20 and 21.

XI. CONCENTRATED NODAL FORCES

Omit these cards if NCONFC.EQ.0; otherwise, input NCONFC cards.

The concentrated nodal forces are assumed to be conservative and independent of deformation.

column	variable	description	notes
1-5	JGROUP	Substructure to which load is applied (ignored if MGO=0)	(1)
6-10	NOD	Node to which load is applied. For MGO=0, NOD corresponds to the global node numbers as input in Section III. For MGO=1, NOD corresponds to the local node number of the substructure.	(2)
11-15	IDIR	Direction of load EQ.1 - X direction EQ.2 - Y direction EQ.3 - Z direction	(3)
16-20	JLCUR	Load curve according to which load is applied	(3)
21-30	FORCE	Nodal force	(4)

NOTES

- (1) JGROUP must lie in the range $1 \leq \text{JGROUP} \leq \text{NUMEG}$. The same group number may be repeated as necessary.
- (2) The location and direction of the concentrated nodal force is given by NOD and IDIR, respectively. For the node input option, MGO=0, NOD is a global node number as input in section and $1 \leq \text{NOD} \leq \text{NUMNP}$. For the mesh generation option using substructures, MGO=1, NOD corresponds to a local element node number as input in Section VII; valid range is $1 \leq \text{NOD} \leq 8$ for 2-D elements and $1 \leq \text{NOD} \leq 20$ for 3-D elements. The direction of the load is specified by IDIR and is always with respect to the global axes direction. Note that for 2-D problems, the geometry is defined with respect to the (Y-Z) plane and loads may only be applied in the Y and Z direction.
- (3) The concentrated force is to vary with time (or load parameter) according to load curve JLCUR. Note that $1 \leq \text{JLCUR} \leq \text{NLCUR}$.
- (4) The nodal force is positive in the global (X,Y,Z) directions.

XII. RAYLEIGH DAMPING (2F10.0)[†]

Omit this card if no Rayleigh damping is being included, IDAMP.EQ.0.

column	variable	description	notes
1-10	ADAMP	Rayleigh damping coefficient α	(1)
11-20	BDAMP	Rayleigh damping coefficient β	(1)

NOTES

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

It should be noted that α and β are applied to the linear mass and stiffness matrices.

[†]Not allowed in current program version.

XIII. CONCENTRATED NODAL MASSES (110, 6F10.0)[†]

Omit this section if IMASSN.EQ.0; otherwise, input IMASSN cards as follows.

column	variable	description	notes
1-10	N	Node number	(1)
11-20	XMASS(1)	X-direction mass	(2)
21-30	XMASS(2)	Y-direction mass	
31-40	XMASS(3)	Z-direction mass	
41-50	XMASS(4)	X-rotational mass	(3)
51-60	XMASS(5)	Y-rotational mass	
61-70	XMASS(6)	Z-rotational mass	

NOTES

- (1) Input IMASSN cards; node order is not important. Repeating nodes accumulates mass at that node. The node number corresponds to the global node number; hence $1 \leq N \leq \text{NUMNP}$. If the substructure option is used to generate node numbers, a data check run must be made first to determine the global node number assigned by the mesh generator. Once the node numbers are known, the node numbers required on this card may be input.
- (2) Mass components input for deleted (or non-existent) degrees of freedom are ignored by the program without a diagnostic message.
- (3) Rotational degrees of freedom are currently not used.

[†]Not allowed in current program version.

XIV. CONCENTRATED NODAL DAMPERS (I10, 6F10.0)[†]

Omit this section if IDAMPN.EQ.0; otherwise, input IDAMPN cards as follows.

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

NOTES

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

If the substructure option is used to generate node numbers, a data check run must be made first to determine the global node numbers assigned by the mesh generator. Once the node numbers are known, the node numbers required on this card may be input.

- (2) Damper components input for deleted (or non-existent) degrees of freedom are ignored by the program without a diagnostic message.
- (3) Rotational degrees of freedom are currently not used.

[†]Not allowed in current program version.

XV. INITIAL CONDITIONS

Initial conditions for all active degrees-of-freedom are defined in this section. If this is a restart run (MODEX.EQ.2), the cards in this section must be input but are ignored by the program. The variable ICON controls whether zero initial conditions are automatically generated or non-zero initial conditions are read from data.

XV.1. Control Flag (I5)

column	variable	description	notes
1-5	ICON	Flag indicating the type of conditions to be input initial EQ.0 - zero initial conditions are generated automatically EQ.1 - non-zero initial conditions are read from data cards.	

XV.2. Card Input of Initial Conditions (6E12.6)

If ICON.EQ.1, the program performs the following read operations:

```
READ(5,1000) (DIS(K), K = 1,NEQ)
```

```
READ(5,1000) (VEL(K),K = 1,NEQ)
```

```
READ(5,1000) (ACC(K),K = 1,NEQ)
```

```
1000 FORMAT (6E12.6)
```

where DIS, VEL, and ACC are the global displacement, velocity, and acceleration vectors, respectively. The variable NEQ is the total number of degrees-of-freedom, i.e., six times the total number of nodes minus all deleted degrees-of-freedom provided by fixed boundary conditions.

The list of equation numbers assigned to each degree-of-freedom may be obtained from the print-out of a previous run. It may also be determined by careful inspection of the node numbers and boundary conditions input by the program user.

For a static solution (IMASS.EQ.0), the VEL and ACC initial vectors are not read from card input.

XVI. FREQUENCY SOLUTION DATA CARD (5I, F10.0, I5)[†]

Input this card only if a frequency and mode shape solution is to be performed, i.e., if IEIG.EQ.1

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

NOTES

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

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

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

[†]Not operational in current program version.

5. GUIDELINES FOR PROGRAM USE

Developers of nonlinear programs have long cautioned prospective program users that the successful use of a large, nonlinear program is considerably different than the use of a linear program; in fact, it is often a tedious, frustrating and expensive exercise often requiring repeated attempts at solving a particular problem. This is due in part, to the required nonlinear solution algorithms which are often slowly convergent, and sometimes divergent. Various nonlinear kinematic and material models often introduce instabilities which are sensitive to numerical roundoff, grid modeling, numerical integration orders, and the like. This, coupled with the unfamiliarity of most program users with the inner workings and idiosyncrasies of a nonlinear program, leads to many nonlinear computer programs being shelved after a short time. This section presents some guidelines which hopefully will prove beneficial to less-experienced users of nonlinear programs. The following paragraphs are not intended to be a recipe for obtaining a good solution, rather they are a collection of assorted guidelines which should always be considered when setting up a problem or when difficulties arise.

For geometrically nonlinear (large strain) problems, it often is the case that at small strain levels where the gross deformations appear to be relatively linear with load, the program may experience considerable difficulty in converging to a solution. This is generally due to the initial coupling between bending, shear and membrane strains which are extremely nonlinear at low strain levels even though they may contribute insignificantly to the total gross deformation. This often requires relatively small load steps at the onset during which these nonlinearities develop, after which larger steps may be taken.

Difficulties in obtaining convergence may also occur at the onset of plastic deformation if a large portion of the structure undergoes simultaneous

plastic deformation. A similar situation when the structure has significant plastic zones and it experiences sudden load reversals causing the structure to unload elastically. This latter case causes discontinuities in the response of the structure and consequent difficulties in obtaining convergence. In cyclic load cases, it is mandatory that two or three relatively small load steps be taken after the load reversal, otherwise solution convergence will most likely not be achieved.

The solution scheme in the program allows the user to specify how often (load step frequency) the nonlinear stiffness is updated and how often equilibrium iteration is performed. If the nonlinear incremental stiffness is updated (reformed) infrequently, this will provide a more economical solution but will tend to introduce mathematical instability. If equilibrium iteration is not performed at each step, this will tend to stabilize the solution but will make it less accurate due to accumulated error from step to step. If in doubt, equilibrium iteration should be performed at each step. However, on a step where load reversal begins it is often advisable not to perform equilibrium iteration.

For 4 to 8 node, two-dimensional elements, a 2 x 2 Gaussian integration order for the stiffness is generally sufficient. For 16 node, three-dimensional elements, 2 x 2 x 2 integration is generally adequate; however, for 20 node elements a 3 x 3 x 3 order is necessary (a lower order integration will often provide inconsistent results).

Users should be discouraged from using the three-dimensional isoparametric element to model thin plate or shell behavior. In general, it has been demonstrated that the element gives a rather poor representation of thin shell behavior. In particular, as the thickness is reduced, the three-dimensional solution becomes excessively stiff when compared to a Kirchhoff-type thin shell or plate solution.

The substructure mesh generator option (MGO = 1) which is included in the program should be used with caution. The mesh generator may generate incorrect nodal coordinates, element connectivity and/or boundary conditions under certain conditions. Consequently, the user is cautioned to check generated grids carefully when using the MGO = 1 option and, preferably is urged to use the MGO = 0 option of mesh input.

6. PROGRAM RESOURCE REQUIREMENTS, CAPACITY, AND LIMITATIONS

The AGGIE program has been developed on a Amdahl 470 V/6 computer using the IBM 370 Fortran IV compiler and associated software. The minimum memory requirement is approximately 400k bytes when overlaid as shown in Table 2 and compiled in double precision. The memory requirement can be reduced to about 300k if certain modules are not required for a particular problem or if additional overlaying is done. The program requires 12 scratch disk files for the usual nonlinear problem (plus 1 other if displacement incrementation is used); see comment cards in the MAIN program for description of the required files.

As noted previously, the majority of all problem size - dependent arrays are incorporated into a single common block called AMAT. The size of this common block is defined in the MAIN program. If problem size dictates it, this common block should be expanded for more efficient program execution. The variable MTOT (in MAIN) should be set equal to the size of common block. In addition, the size of stiffness matrix block size (LBLOCK) is defined in MAIN. This portion of the common block used for 1 block of the stiffness matrix should be set as large as practical for efficient execution and decreased I/O time (however, it should not exceed approximately 75% of MTOT).

As explained earlier in the summary of the AGGIE program, the capacity of the program is, on a practical basis, "unlimited". If the complete model data will not fit into the array space allocated in common AMAT, the structure may be separated into element groups. In addition, the program uses an out-of-core equation solver (if necessary) to solve problems whose equilibrium equation will not fit entirely in the main core memory (with common AMAT).

7. PROGRAM RESTART

A problem involving a large number of degrees-of-freedom, a large number of load steps, or exhibiting complex nonlinear behavior can often be analyzed most efficiently by using the restart capability of the program. This generally requires running the program for several load steps and having the program write and save restart information which can then be used later to restart the program at that point and continue the analysis. This allows the user to analyze the output and determine the adequacy and accuracy of the solution at an intermediate point and then continue it if necessary without recomputing the prior solution history.

To perform a restart run, a first job must have been run which executed a successfully for NSTE load (or time steps) and which wrote and saved the necessary restart files. The files which must be saved are 1, 4, 7, 8, 9, and 10. The restart job is then executed by supplying the same input as used for the original run except that on the Solution Control Card II.1, the value of MODEX is set equal to 2, the Value of TSTART is set equal to the new starting value (which must be the same as the load (time) value at which the restart information was written), and the value of NSTE is set equal to the number of additional steps desired. If deemed necessary, the Nonlinear Solution Variables on Card II.2 may be changed for the restart run. All loading information on Cards VIII-XI may be modified, except that it should be noted that only loading information subsequent to TSTART is pertinent. No changes may be made to the element mesh, boundary conditions, material models, type of element nonlinearity, etc.

8. EXAMPLE PROBLEMS

The example problems which follow were chosen to demonstrate the various facets of the input required to run a particular problem, to acquaint the user with the type of problems that can be solved, and to provide documented results which may be used to check out the program upon its initial installation at a different computer facility. In an attempt to establish some confidence level in the program, the results are compared to either closed-form solutions, other finite element solutions or experimental results.

The particular problems presented here do not by any means represent the "typical" problem that can be solved nor the upper capacity or capability of the program. For example, the capacity of the program in terms of number of elements allowed is practically unlimited. Additional problems which have been solved with the program are presented in Ref. 8.

All solutions presented here were obtained with the computer program compiled in IBM Fortran G in double precision and executed on the Amdahl 470 V/6 located at the Data Processing Center of Texas A&M University. Using the overlay structure discussed previously, the program requires about 450K bytes of storage (excluding disk buffer space). All quoted computer execution times refer to CPU time only and do not include items such as I/O time, spooling time, virtual memory paging time, etc.

Example Problem 1

The first example problem involves the large, plastic deformation of a simply supported circular plate composed of high strainhardening 2024-0 aluminum. The plate under consideration is centrally loaded by a circular punch of radius 0.1875 inches to a maximum load of 4,000 pounds. The dimensions of the plate are given in Fig. 22 . The uniaxial material prop-

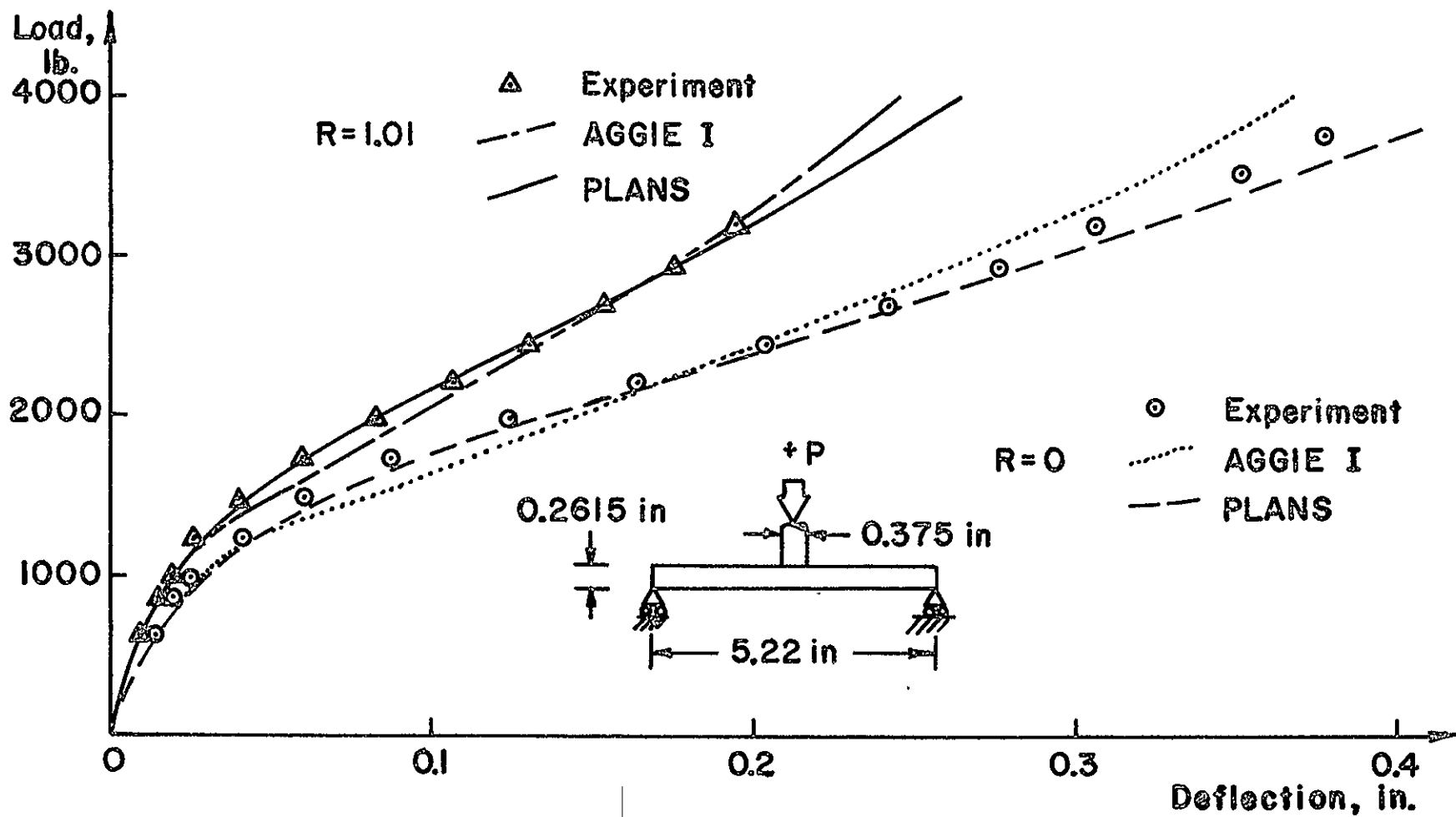


Fig. 22 Deflection history, circular plate with punch.

erties for this plate are given in Ref. 9 along with the plate test data and theoretical results obtained with the PLANS computer program. The material uniaxial stress-strain curve was approximated by five linear segments with nodal engineering stress, strain values as follows: 10,000 psi, 0.000995; 20,000 psi, 0.0131; 28,000 psi, 0.04502; 36,000 psi, 0.118; and 48,000 psi, 0.36311. The value of Poisson's ratio was not specified in Ref. 9; hence, a value of 0.325 was chosen for use in the present analysis.

Using a 100 pound load increment (40 steps), the plate was modeled using eleven and twenty-two 8-node isoparametric continuum elements with the results of both idealizations being the same indicating a converged solution. The predicted deflections of the lower surface of the plate at radii of 0.0 inch and 1.01 inches are presented in Fig. 22 along with theoretical and experimental data from Ref. 9.

The PLANS results shown in Fig. 22 were obtained by a finite element idealization of fourteen annular shell elements and using a 5.5 pound load increment. The fact that the annular elements of PLANS include bending effects but not the effects of transverse shear and normal stresses, whereas the elements of AGGIE are pure continuum element may account for the majority of the discrepancy between the results of the two programs.

The theoretical results of both PLANS and AGGIE were obtained using pure kinematic hardening; however, since the loading is monotonically increasing and each point in the structure will experience a near radial stress loading path, the results are the same as if either isotropic hardening or the mechanical sublayer models had been used.

The necessary input data for the eleven element idealization with kinematic hardening is shown in Fig. 23.

The CPU execution time for the eleven element idealization was approximately 46 seconds.

```

10      20      30      40      50      60      70      80
1234567890123456789012345678901234567890123456789012345678901234567890
**** SIMPLE SUPPORTED CIRCULAR PLATE WITH MONOTONIC PRESSURE LOAD, KH ****
100111  2  0  1  40  1.0  0.0
  0  0  2  25  0.001
  1  1  0  0  0  0  0  0
  2  2  1  0  0  0  0  0
  3  3  36  38
  58  1  1  1
  1  1  1
  3  1
  4  1
  5  1
  6  1
  8  1
  9  1
 10  1
 11  1
 36  1
 56  1
 14  1
 34  1
 39  1
 54  1
 12  1
 37  1
 57  1
 15  1
 35  1
 40  1
 55  1
 13  1
 38  1
 58  1  1  1
  2  2  2  0  8  3  1  1
  1  0.0  0.325
10050000.0
  1  8  1  1  6  1  3  8  4  2  5  7
  2  8  1  1  11  6  8  13  9  7  10  12
  2  9  2  0  8  3  6  1
  1  0.0
10050000.0
 20000.  0.0131  28000.  .04502  36000.  0.118  10000.0  .000995
  0.0  0.0  40.0  1.0
  1  1  2  TOP  1 -36216.59
  0

```

138

ORIGINAL PAGE IS
OF POOR QUALITY

Fig. 23 Input Data for Example Problem 1

Example Problem 2

The second problem involves the same simply supported circular plate considered in Example Problem 1, but subjected to a cyclic loading history of 1325 pounds, then -773 pounds, and finally 1325 pounds. Since the loading is not monotonic, the predicted structural response will depend on the particular hardening rule used in the analysis. Based on the results of the previous sample problem, the plate was modeled using nine 8-node isoparametric (axisymmetric) elements. In modeling the plate, it was assumed that the plug of material under the punch acts rigidly; hence, the material under the punch was removed and roller boundary conditions inserted at the punch boundary. The punch load was then approximated as a shear load distributed through the thickness at the punch boundary. Since the loading was not severe enough to introduce appreciable geometric nonlinearities, the analyses were run as materially nonlinear only.

The structural response predicted by AGGIE using the mechanical sublayer model is shown in Fig. 24 along with experimental data from Ref. 10. The material constants included a Poisson's ratio 0.325, Young's modulus of 10.05×10^6 psi, and a piecewise linear uniaxial stress-strain curve with the following nodal stress, strain values: 8,000 psi, 0.000796; 16250 psi, 0.00575; 25,000 psi, 0.02925; and 35,000 psi, 0.10. Compared to experiment, the results are qualitatively good. The required program input is shown in Fig. 25.

Results for combined kinematic-isotropic hardening with a constant ratio $\gamma = 0.5$ are shown in Fig. 26. The stress-strain curve was idealized slightly differently as: 8,000 psi, 0.000796; 12,000 psi, 0.0025; 18,000 psi, 0.00925; and 35,000 psi, 0.10. The results in this case are somewhat better than obtained with the mechanical sublayer model. The

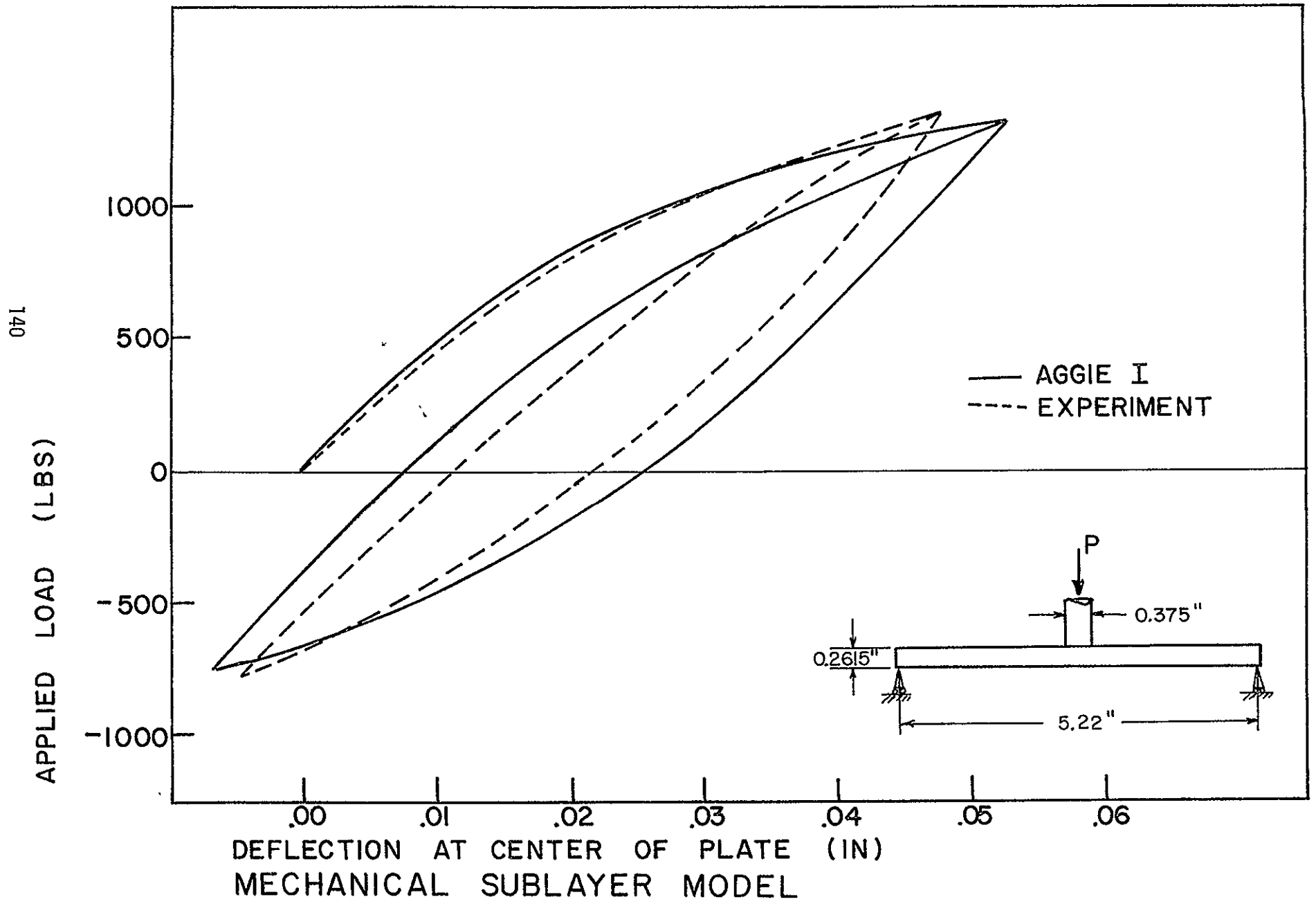


Fig. 24 Center Deflection of Curcular Plate Subjected to Cyclic Punch Load - Mechanical Sublayer Model

```

      10      20      30      40      50      60      70      80
123456789012345678901234567890123456789012345678901234567890
**** PROB2D7, CYCLIC PLATE 2024 AL. LEVINE AND SVALBONAS,MSL,SHEAR LOAD ****
100111 1 0 1 56 1.0 0.0
 1 1 2 25
 1 1 0 0 0 0 0
 2 1 1 0 0 0 0
 1 8
48 0 1
 1 1
 2 1
 3 1 1
 4
24
 5
25 0.92775 0.0 0.1875 0.2615
 6 0.352 0.2615 0.1875 0.13075
26 1.01 0.2615 0.1875 0.0
46 2.61 0.2615 0.26975 0.2615 5
 7 0.352 0.13075 0.92775 0.2615 5
27 1.01 0.13075 0.26975 0.0 5
47 2.61 0.13075 0.92775 0.0 5
29 1.21 0.2615 2.41 0.2615
44 2.41 1.21 5
30 1.21 5
45 2.41 0.352 5
 8 1.01 5
28 2.21 0.0
43 2.61
48 1 0 1
 2 9 1 0 8 3 10 1
10050000.0 0.0 0.325 4.0 8000. 0.00079602 16250. 0.00575 25000.
 1 0.02925 35000. 0.1 1 6 1 3 8 4 2 5 7
 2 8 1 1 5 11 6 8 13 9 7 10 12
 9 8 1 46 41 43 48 44 42 45 47
 8
 0.0 0.0 12.0 0.12 13.0 0.1198 14.0 0.119
34.0 35000. -0.07 35.0 -0.06975 36.0 -0.069 56.0 0.120
 1 1 1 LEFT 1 0.0 35851.0
 0

```

141

ORIGINAL PAGE IS
OF POOR QUALITY

Fig. 25 Input Data for Example Problem 2 - Mechanical Sublayer Model

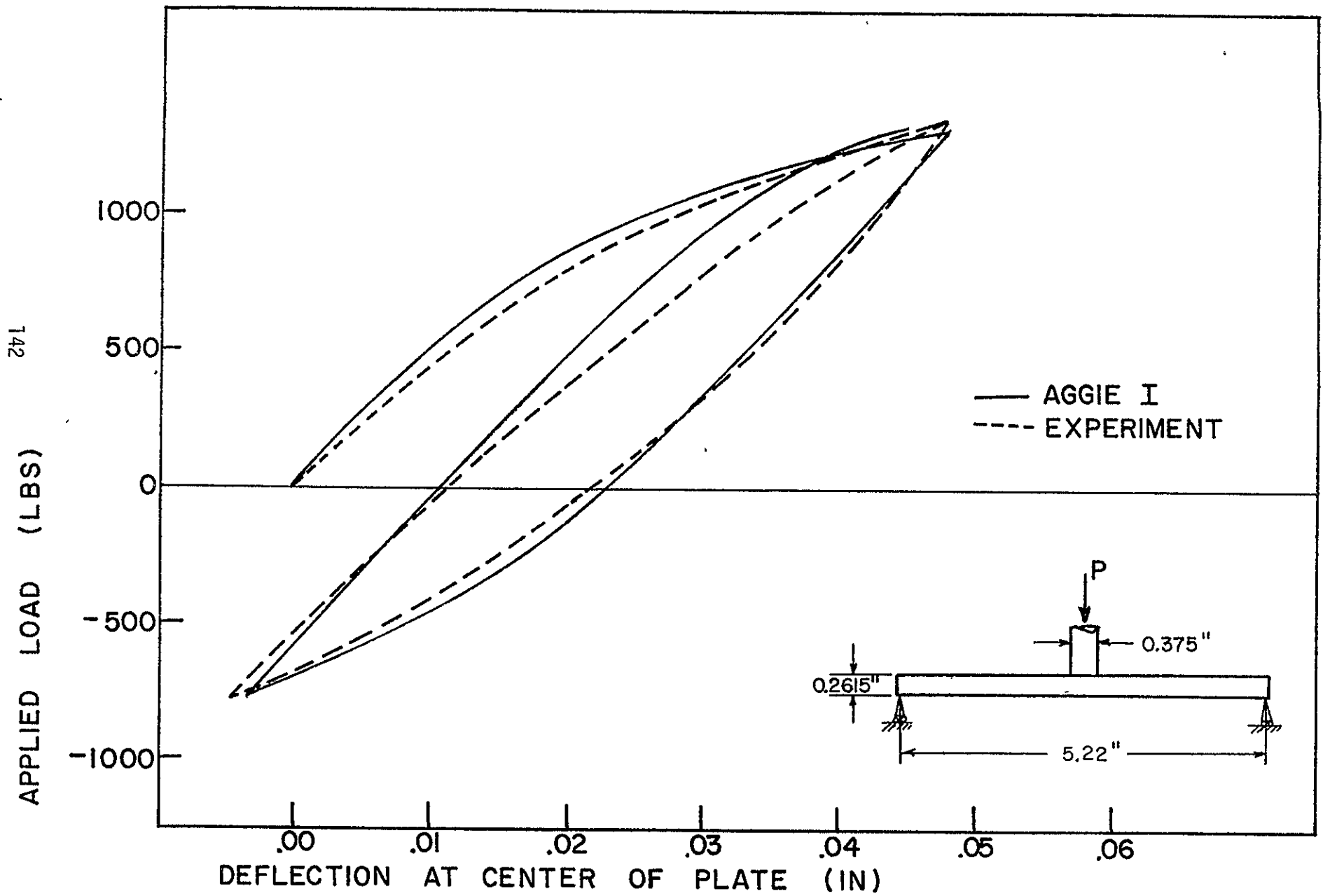


Fig. 26 Center Deflection of Circular Plate Subjected to Cyclic Punch Load - Combined Kinematic - Isotropic Hardening Model with $\gamma = 0.5$.

```

      10      20      30      40      50      60      70      80
123456789012345678901234567890123456789012345678901234567890
****-PROB2D7. CYCLIC PLATE 2024 AL, LEVINE AND SVALBONAS, IKH(R=.5), SHEAR LOAD **
100111 1 0 1 56 1.0 0.0
  0 0 2 25 0.001
  1 1 0 0 0 0 0 0
  2 1 1 0 0 0 0 0
  1 8
 48 0 1
  1 1 1
  2 1 1
  3 1 1
  4
 24 0.1875 0.2615
  5 0.1875 0.13075
  6 0.1875
  7 0.26975 0.2615 5
  8 0.92775 0.2615
  9 0.26975 5
 10 0.92775
 11 0.352 0.2615 5
 12 1.01 0.2615 5
 13 2.61 0.2615
 14 0.3520 0.13075 5
 15 1.01 0.13075 5
 16 2.61 0.13075
 17 1.21 0.2615 5
 18 2.41 0.2615
 19 1.21 5
 20 2.41 5
 21 0.352 5
 22 1.01 5
 23 2.61
 24 2 9 1 1 0 8 3 6 1
 25 1 0.0 0.325 5.0 0.00925 2.0 0.50 8000.0 0.00079602
 26 12000.0 0.0025 18000.0 25500.0 0.02925 35000.0 0.1
 27 1 8 1 1 5 6 1 3 8 4 2 5 7
 28 2 8 1 1 5 11 6 8 13 9 7 10 12
 29 9 8 1 46 41 43 48 44 42 45 47
 30 8
 31 0.0 0.0 12.0 0.12 13.0 0.11980 14.0 0.119
 32 34.0 -0.07 35.0 -0.06975 36.0 -0.0690 56.0 0.12
 33 1 1 1 LEFT 1 0.0 35851.0
 34 0

```

143

ORIGINAL PAPER IS
OF POOR QUALITY

Fig. 27 Input Data for Example Problem 2 - Combined Isotropic-Kinematic Hardening Model with $\gamma=0.5$

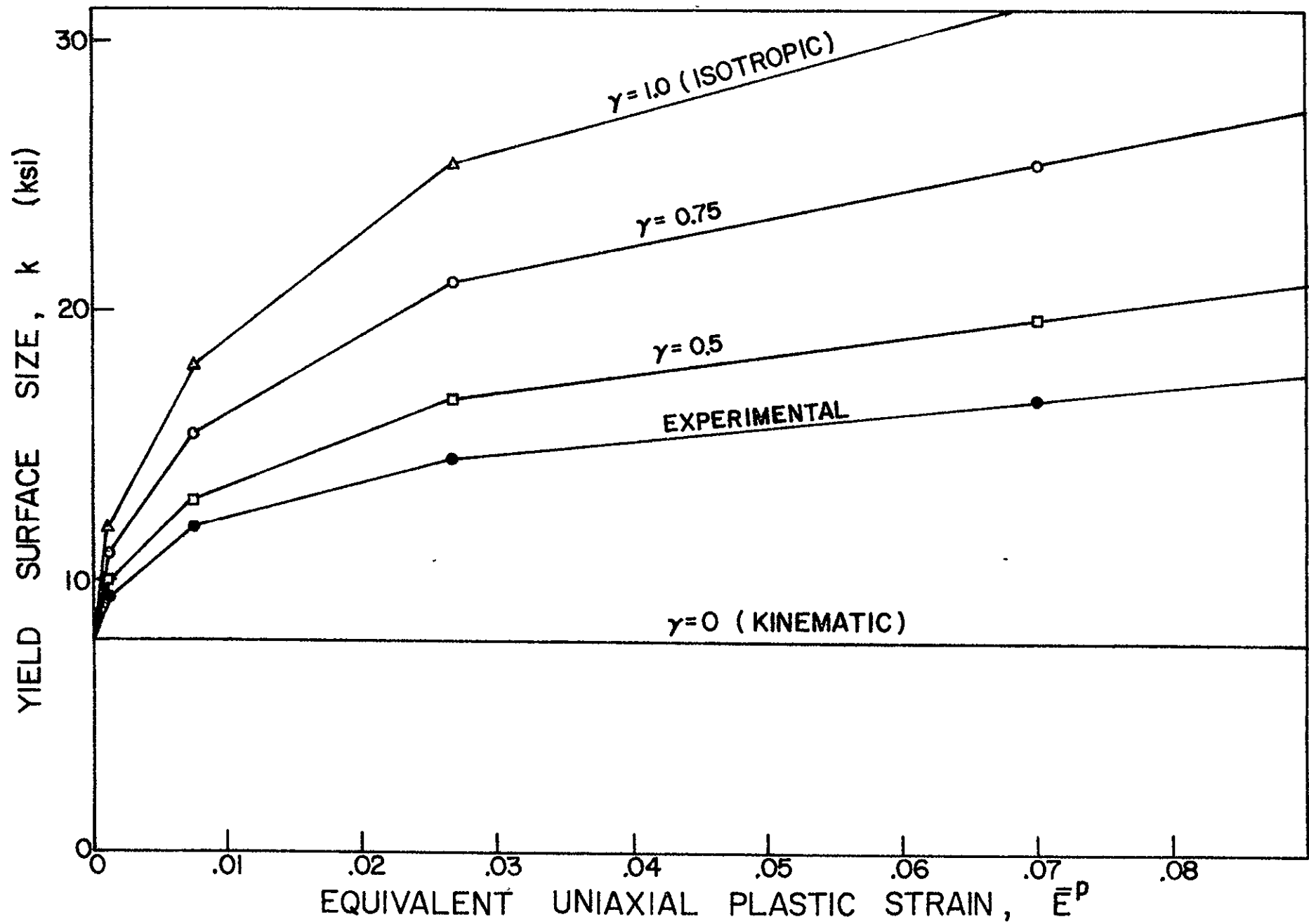


Fig. 28 Yield Surface Size vs. Equivalent Uniaxial Plastic Strain for 2024-O Aluminum

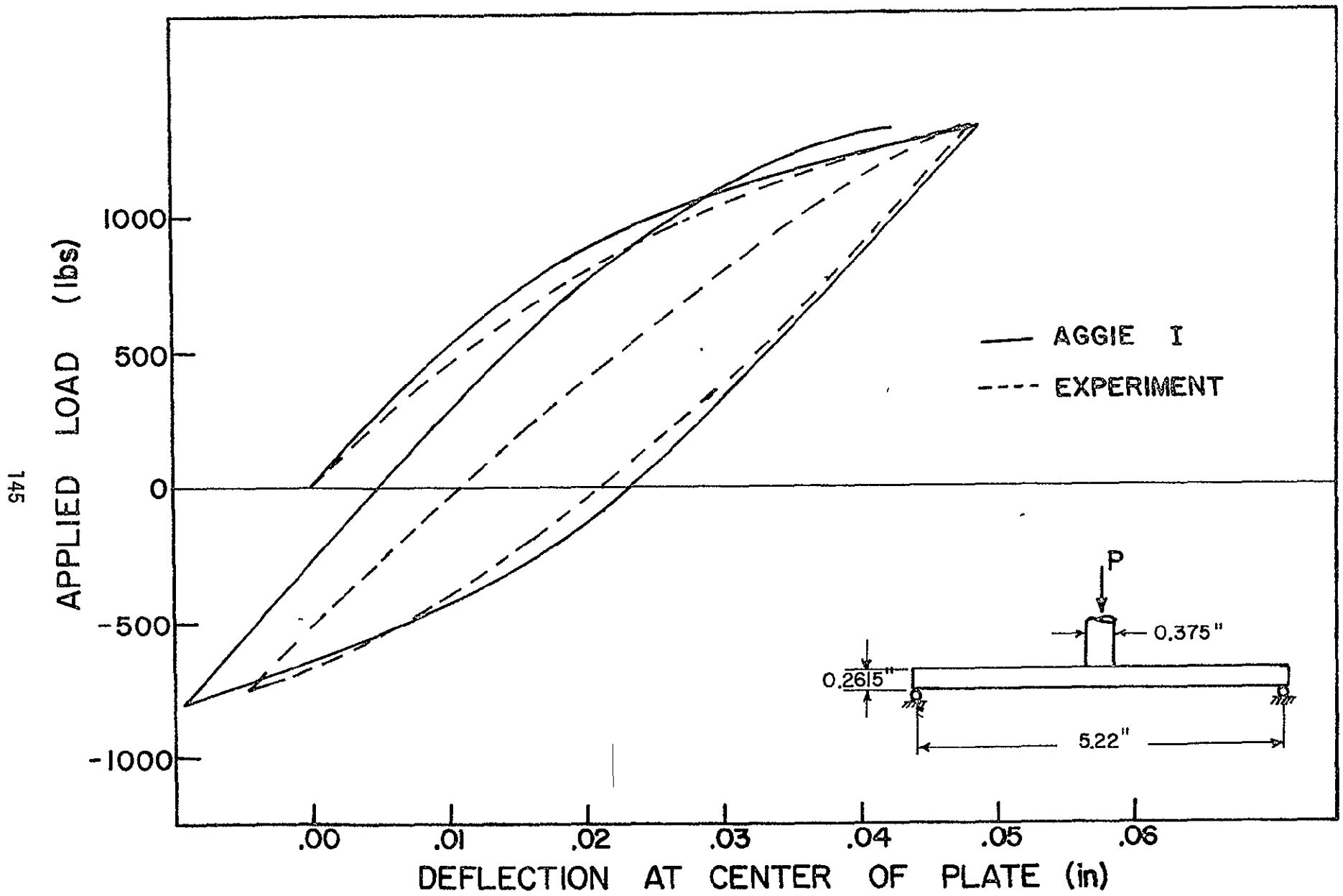


Fig. 29 Center Deflection of Circular Plate Subjected to Cyclic Punch Load - Combined Kinematic-Isotropic Hardening Model with Experimental Variable Ratio γ

```

      10      20      30      40      50      60      70      80
123456789012345678901234567890123456789012345678901234567890
**** PROB2D7, CYCLIC PLATE 2024 AL, LEVINE AND SVALBONAS, IKH(VAR.R), SHEAR LOAD *
100111 1 0 1 56 1.0 0.0
  0 0 2 25 0.001
  1 1 0 0 0 0 0 0
  2 1 1 0 0 0 0 0
  1 8
 48 0 1
  1 1 1 0.1875 0.2615
  2 1 1 0.1875 0.13075
  3 1 1 0 0.1875 0.0 0
  4 0.26975 0.2615 5
 24 0.92775 0.2615 0
  5 0.26975 0.0 5
 25 0.92775
  6 0.352 0.2615 5
 26 1.01 0.2615 5
 46 2.61 0.2615 0
  7 0.3520 0.13075 5
 27 1.01 0.13075 5
 47 2.61 0.13075
 29 1.21 0.2615 5
 44 2.41 0.2615
 30 1.21 0.0 5
 45 2.41
  8 0.352 5
 28 1.01 5
 48 2.61
  2 9 1 1 0 8 3 6 1
  1 0.0
10050000.0 0.325 5.0 5.0 3.0 8000.00.00079602
12000.0 0.0025 18000.0 0.00925 25500.0 0.02925 35000.0 0.1
 8000.0 0.0 11086.5 0.00457 13580.9 0.0142857 16075.4 0.038095
17322.6 0.061905
  1 8 1 1 6 1 3 8 4 2 5 7
  2 8 1 5 11 6 8 13 9 7 10 12
  9 8 1 46 41 43 48 44 42 45 47
  8
  0.0 0.0 12.0 0.12 13.0 0.11980 14.0 0.119
 34.0 -0.07 35. -0.06975 36.0 -0.0690 56.0 0.12
  1 1 1 LEFT 1 0.0 35851.0
  0

```

146

ORIGINAL PAGE IS
OF POOR QUALITY

Fig. 30 Input Data for Example Problem 2 - Combined Kinematic-Isotropic Hardening Model with Experimental Variable Ratio Data

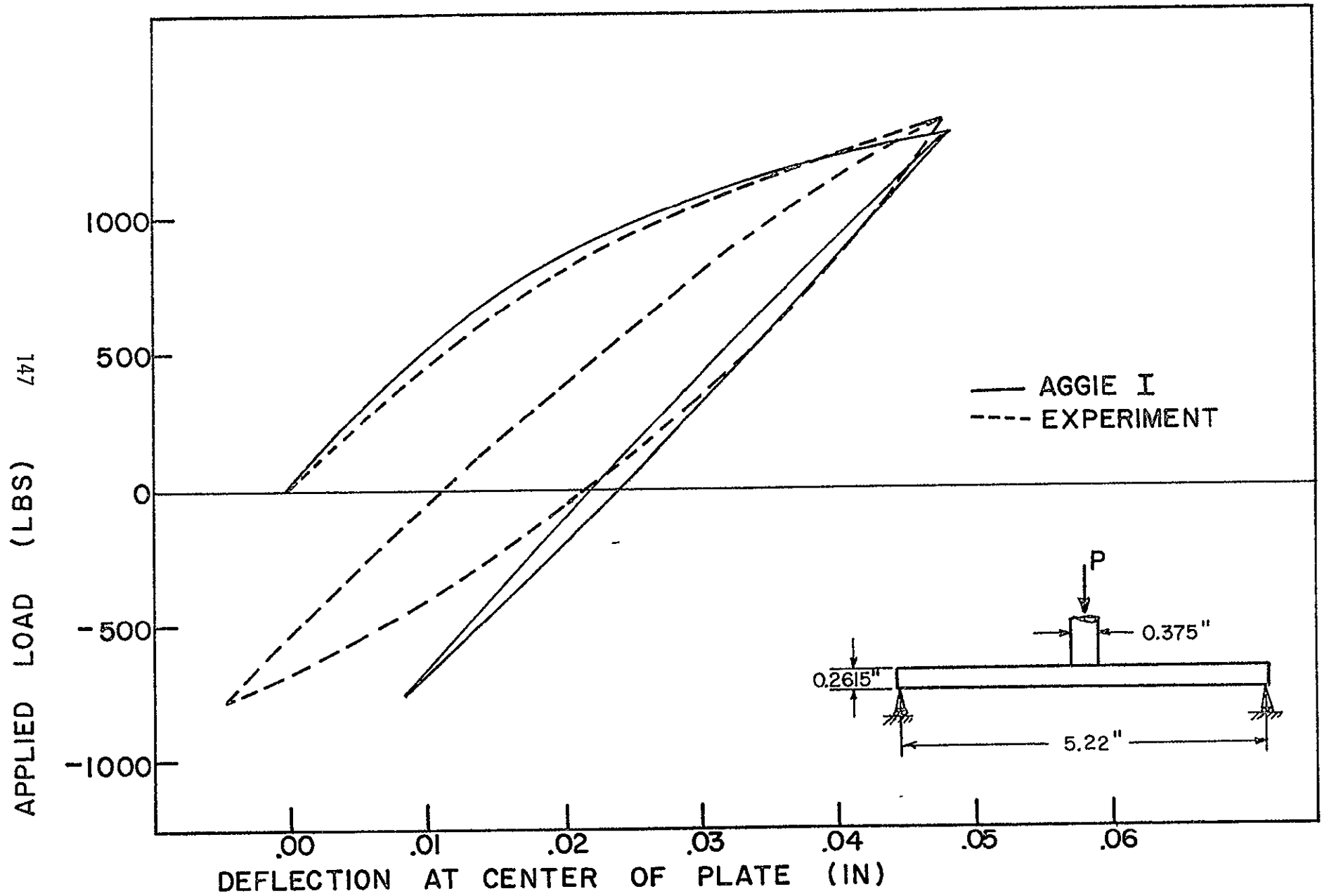


Fig. 31 Center Deflection of Circular Plate Subjected to Cyclic Punch Load - Isotropic Hardening

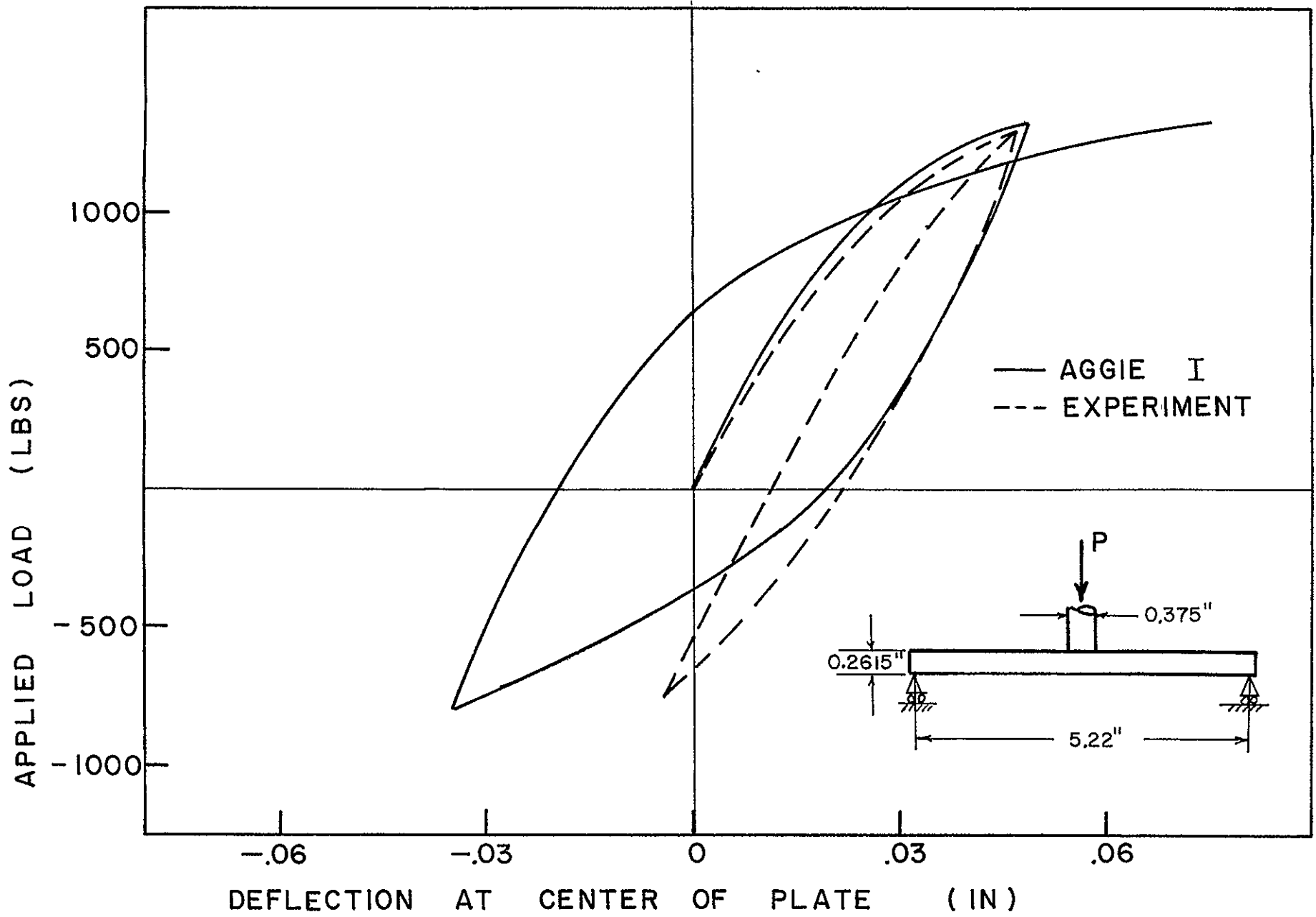


Fig. 32 Center Deflection of Circular Plate Subjected to Cyclic Punch Load - Kinematic Hardening

required input data is shown in Fig. 27 .

For the 2024-0 aluminum used in this test, a yield surface size vs. plastic strain curve has been obtained from experimental cyclic test data provided by Winter.¹¹ This experimental yield surface curve along with several others are shown in Fig. 28. This data has been used in the variable ratio, combined kinematic-isotropic hardening model (TPLAS.EQ.3) to obtain the results shown in Fig. 29. The results are qualitatively good. Input data for this solution is shown in Fig. 30.

For comparison purposes, solutions obtained with the pure isotropic and pure kinematic hardening models are shown in Figs. 31 and 32. The stress strain curve used was the same as used in the kinematic hardening model. The results are poor as would be expected. Since isotropic hardening predicts a reverse ideal Bauschinger effect, the structural response after the initial loading of 1325 pounds is predominately elastic, resulting in poor agreement with the experimental data. For kinematic hardening, the slight strainhardening rate predicted after a reversal of loading results in a very soft tangent stiffness and a corresponding overestimate of the displacement increments. Additional results for this problem may be found in Ref. 12.

The CPU execution time for each of these problems was approximately 44 seconds.

Example Problem 3

An infinitely long, thick-walled cylindrical tube made of incompressible material, 4 inches internal radius and 8 inches external radius, and subjected to an internal pressure of at most 40 psi is the next example problem. The material is assumed to be incompressible with Mooney-Rivlin material constants of $C_1=25$ psi and $C_2=7$ psi.² The finite element model consists of four 8-node axisymmetric elements with roller boundary conditions

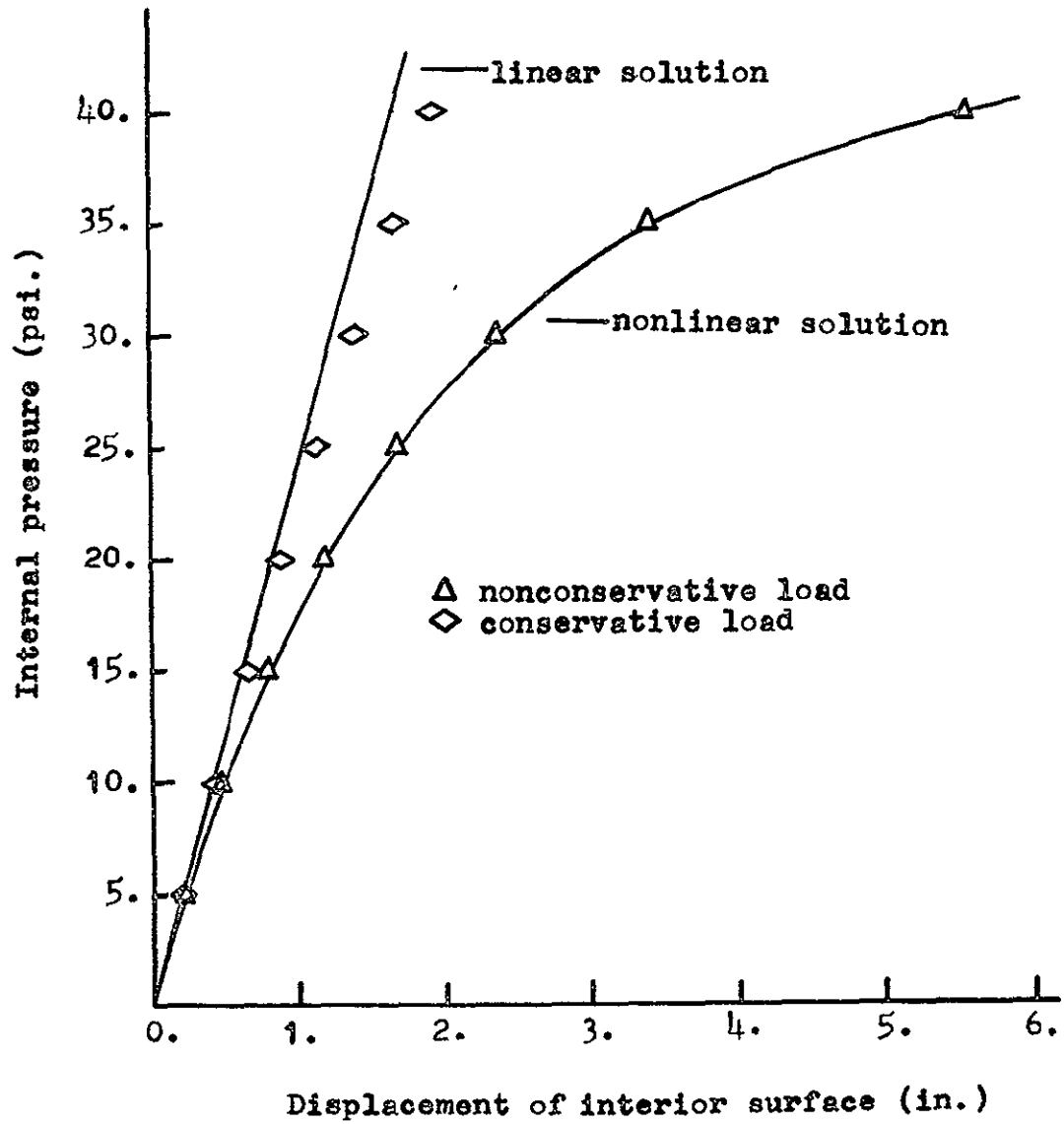
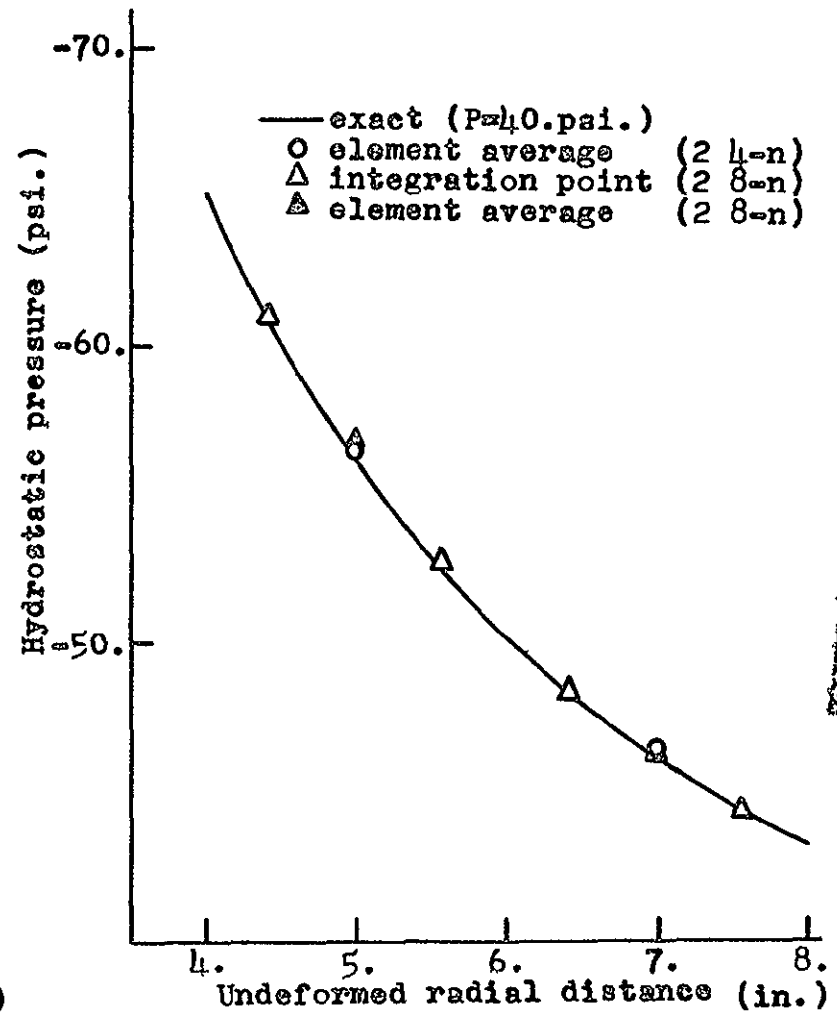
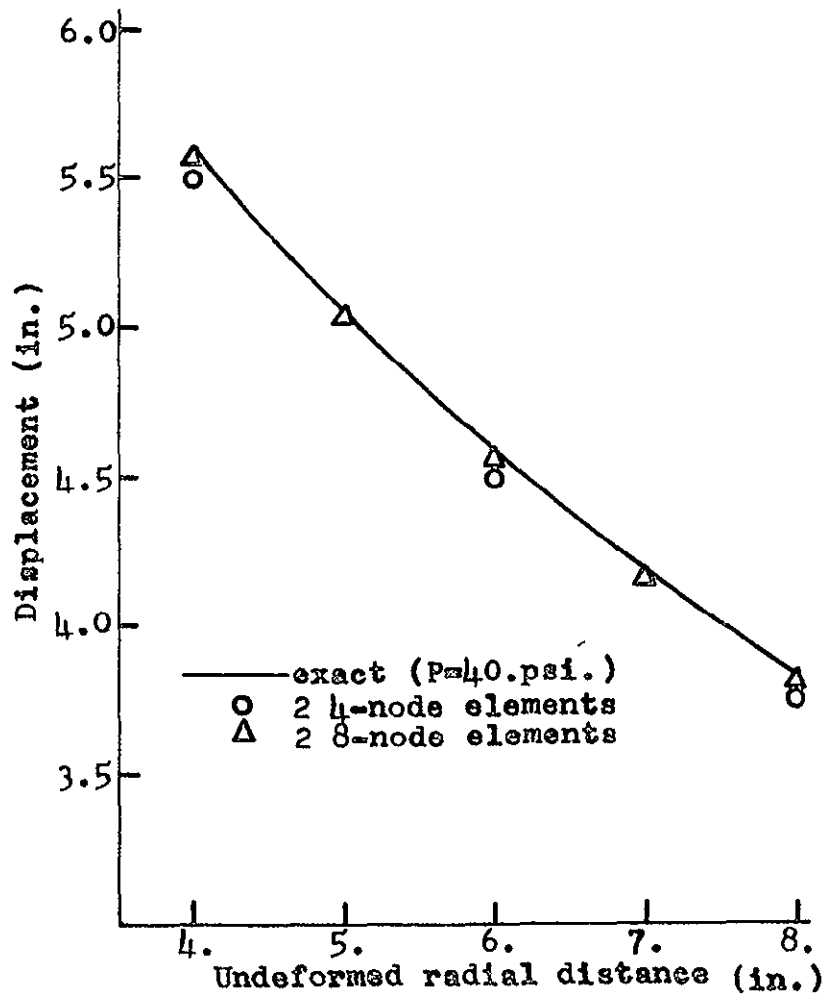


Fig. 33 Cylinder under internal pressure, Axisymmetric analysis, Displacement of interior surface vs. internal pressure



ORIGINAL PAGE IS
 OF POOR QUALITY

Fig. 34 Cylinder under internal pressure, Axisymmetric analysis,
 Displacement and hydrostatic pressure profiles

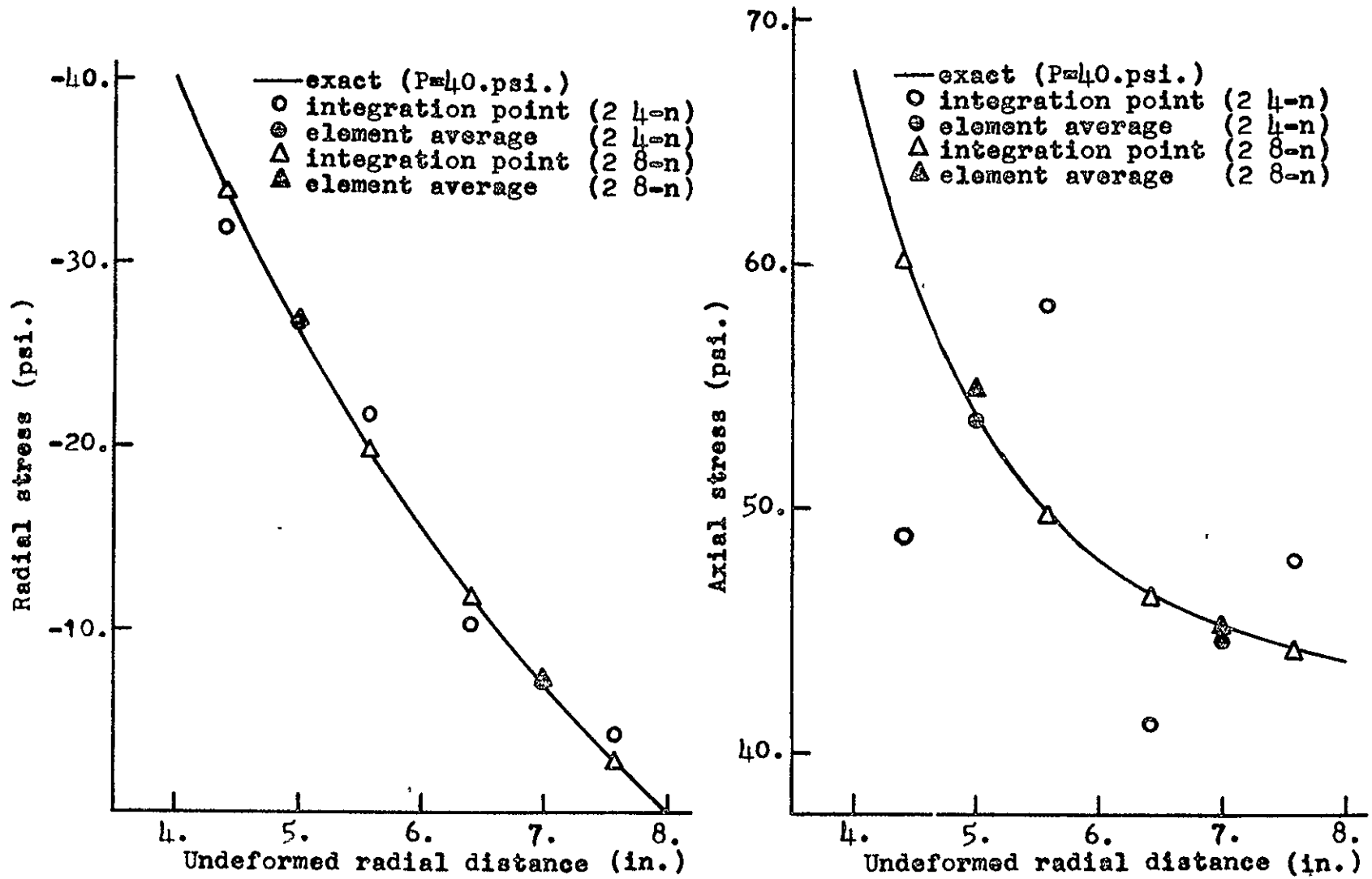


Fig. 35 Cylinder under internal pressure, Axisymmetric analysis, Stress profiles (radial and axial)

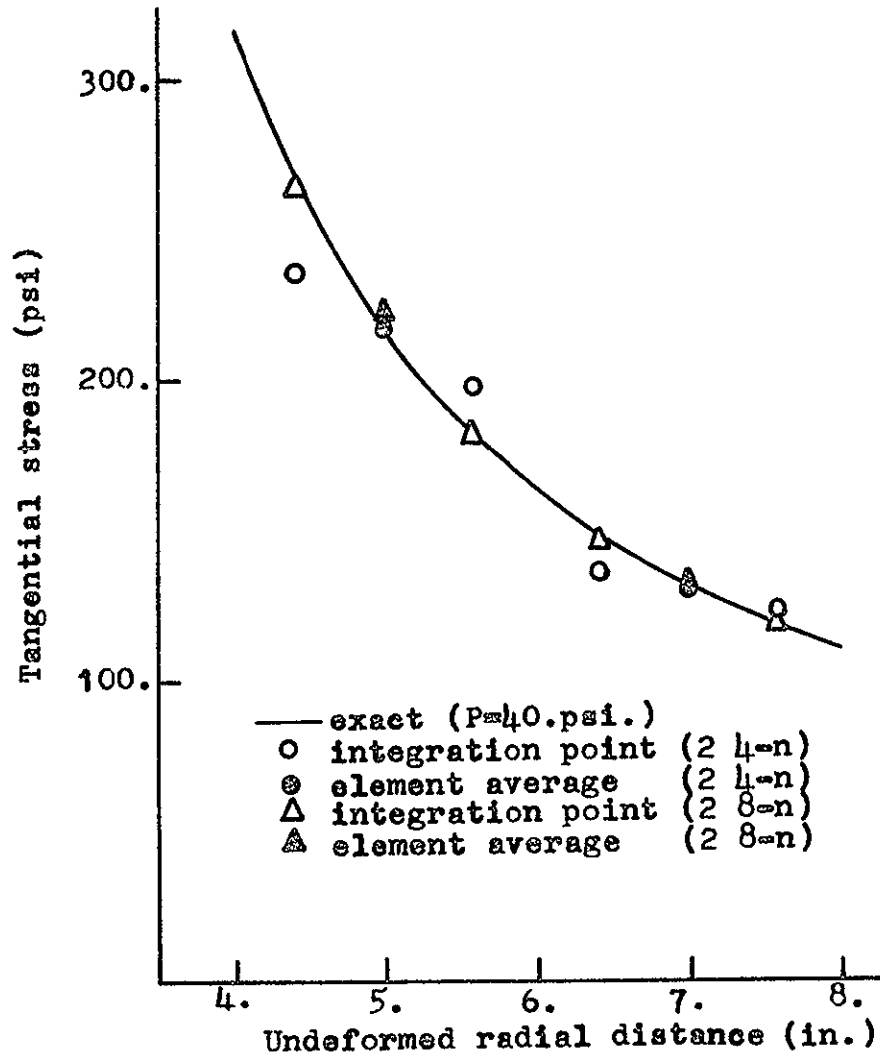


Fig. 36 Cylinder under internal pressure, Axisymmetric analysis, Stress profile (tangential)

```

10      20      30      40      50      60      70      80
123456789012345678901234567890123456789012345678901234567890
***THICK. INFINITE CYLINDER WITH INTERNAL PRESSURE, MOONEY-RIVLIN MATERIAL ***
100111 1 0 1 0 8 1.0 0.0 0 0 0
1 1 1 15 0.001
1 0 1 0 0 0 0 0
23 0 1 0 1 1 1 4.0 1.0 5
1 1 0 1 1 1 1 8.0 1.0 0
21 1 0 1 1 1 1 4.5 1.0 5
4 1 0 1 1 1 1 7.5 1.0 0
19 1 0 0 1 1 1 4.0 0.5 5
2 1 0 0 1 1 1 8.0 0.5 0
22 1 0 0 1 1 1 4.5 0.0 5
5 1 0 0 1 1 1 7.5 0.0 5
20 1 0 0 1 1 1 4.0 0.0 5
3 1 0 0 1 1 1 8.0 0.0 0
23 1 0 0 1 1 1
2 4 2 0 8 2 0 9 1 0 0
1 25.0 7.0
1 8 1 1 5 6 1 3 8 4 2 5 7
4 8 1 1 0 21 16 18 23 19 17 20 22
2 0.0 0.0 8.0 1.0
1 1 1 LEFT 1 -40.0 0.0
0

```

Fig. 37 Input Data for Example Problem 3

at the top and bottom surface to simulate the infinite length conditions (i.e., no displacement allowed in the direction of the cylinder axis).

Fig. 33 shows the displacement response obtained for a linear solution and nonlinear (total Lagrangian) solutions with conservative and nonconservative loading. The nonlinear solution with nonconservative loading is in excellent agreement with three closed-form exact solution;⁵ however, the conservative loading solution is considerably in error. This would be expected since the inner radius of the cylinder expands from 4 inches to about 10 inches and, hence, the surface area over which the pressure is applied increases by a factor of almost 6. A two element (8-node) solution yields virtually the same solution.

Displacement, hydrostatic pressure, and stress profiles through the thickness for a two element (4 and 8 node) model are shown in Figs. 34 through 36. Excellent agreement with exact solutions are noted in all cases except for some of the two, 4 node element solutions.

The input data used for the 4 element (8-node) idealization is shown in Fig. 37. The CPU execution time for this data set was approximately 7 seconds.

Additional details on the solution of various incompressible material problems may be found in Ref. 5.

Example Problem 4

This problem is identical to the previous problem except that a thin steel jacket has been added around the thick rubber cylinder. The steel jacket is 1/4 inch thick with material properties $E=30 \times 10^6$ psi, $\nu=0.3$, and an assumed uniaxial stress-strain curve approximated by stress, strain points: 50,000 psi, .00167; 60,000 psi, 0.01; and 65,000 psi, 0.10.

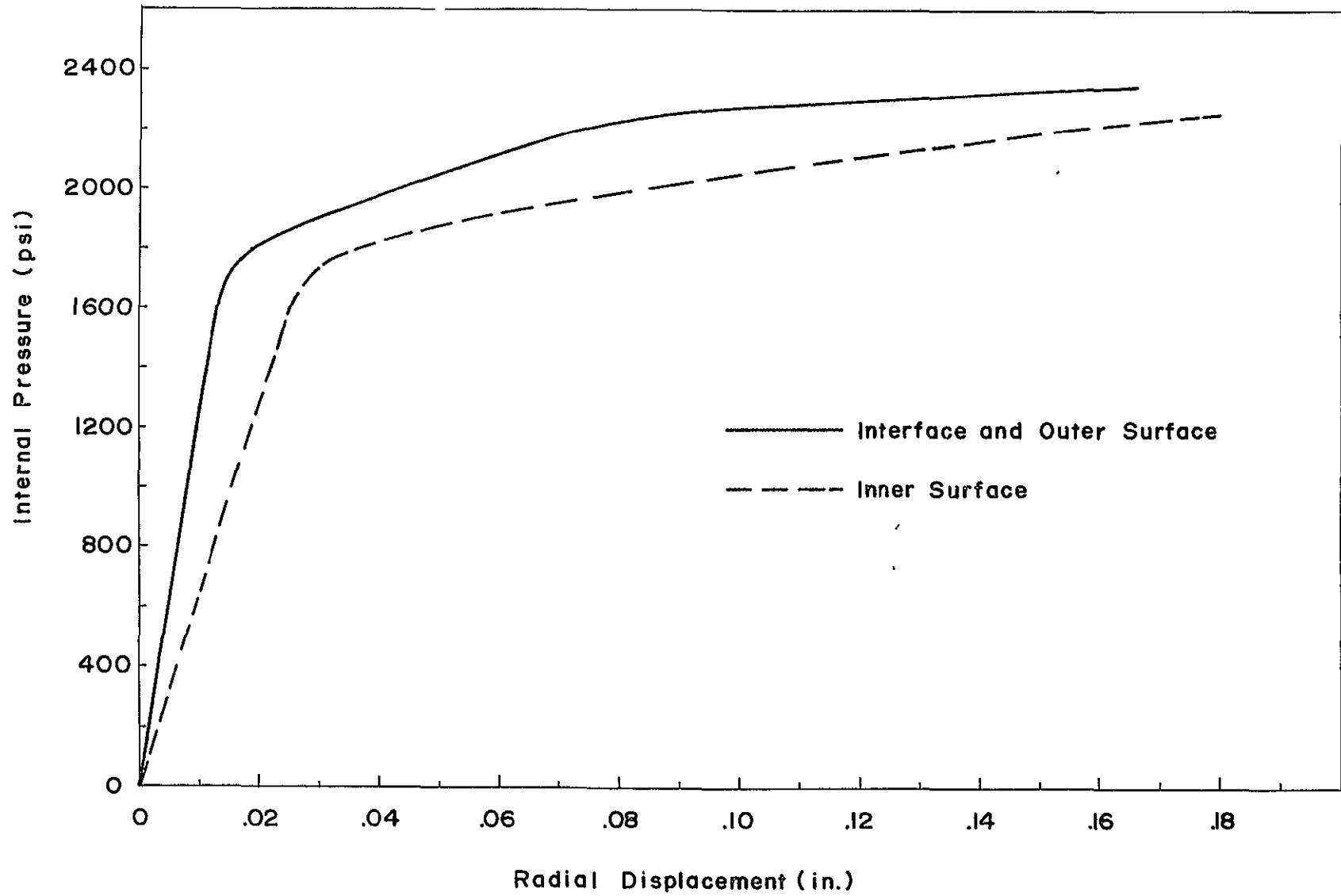


Fig. 38 Radial Displacement History for Thick Rubber-Steel Cylinder with Internal Pressure

```

10      20      30      40      50      60      70      80
1234567890123456789012345678901234567890123456789012345678901234567890
** THICK INFINITE CYLINDER WITH INTERNAL PRESSURE, MOONEY-RIVLIN AND STEEL ***
100111 2 0 1 0 25 1.0 0.0 0 0 0
1 1 1 15 0.001
1 1 0 0 0 0 0 0
1 0 1 0 0 0 0 0
33 0 2 1 1 1 1 4.0 1.0 1
1 1 0 1 1 1 1 8.0 1.0 1
9 1 0 1 1 1 1 8.25 1.0 0
13 1 0 0 1 1 1 4.0 0.5 1
14 1 0 0 1 1 1 8.0 0.5 1
18 1 0 0 1 1 1 8.25 0.5 0
20 1 0 0 1 1 1 4.0 0.0 1
21 1 0 0 1 1 1 8.0 0.0 1
29 1 0 0 1 1 1 8.25 0.0 0
33 1 0 1 1 1 1 0 9 1 0 0
2 4 2 0 8 2
1 25.0 0.0 7.0
1 8 1 1 1 3 1 21 23 2 14 22 15
2 8 1 1 1 5 3 23 25 4 15 24 16
3 8 1 1 1 7 5 25 27 6 16 26 17
4 8 1 1 1 9 7 27 29 8 17 28 18
2 2 2 0 8 3 0 6 1 0 0
1 30000000.0 0.0 0.3 3.0 0.0 50000.0 .00166667
60000.0 0.01 65000.0 0.1
1 8 1 1 1 11 9 29 31 10 18 30 19
2 8 1 1 1 13 11 31 33 12 19 32 20
4 0.0 1 0.0 LEFT 8.0 1600. 11.0 1900. 25.0 2600.
1 1 1 1 1 1 0.0
0

```

157

Fig. 39 Input Data for Example Problem 4

ORIGINAL PAGE IS
OF POOR QUALITY

The steel jacket is modeled with two 8-node elements and the isotropic hardening plasticity model (TPLAS.EQ.0). The modeling of the rubber remains the same as in Example Problem 3.

Figure 38 shows the radial displacement of the inner surface, rubber-steel interface, and outer surface as a function of pressure. The displacements for the interface and outer surface are indistinguishable on this scale plot. It should be noted that the steel jacket undergoes plastic deformation at a pressure of 1,725 psi. Linear thin shell membrane theory applied to the thin steel jacket by itself predicts the onset of plastic deformation at approximately 1,540 psi. The deformations in this case are considerably smaller than in the previous example; hence, this solution was obtained using conservative pressure loading.

Example Problem 5

The next example considers the dynamic response of a spherical cap subjected to a uniform normal pressure loading applied as a step function. The geometry and material properties are shown in Fig. 40. The material is aluminum and is assumed to have a yield stress of 24,000 psi with a secondary modulus E_t of 0.21×10^6 psi. The finite element model consists of ten 8-node axisymmetric elements. The solution is assumed to be geometrically and materially nonlinear with plasticity accounted for with the mechanical sublayer model.

Figure 41 shows a comparison of the apex deflection solutions obtained with AGGIE (incremental solution), DYNAPLAS¹³ (based on thin shell elements) and a pseudo force solution.¹⁴ Considering the different approximations in each solution, the agreement is quite good. Figure 42 presents displacement vs. time solutions for time steps of 2.5 and 5 μ seconds indicating a relatively converged solution with respect to time step.

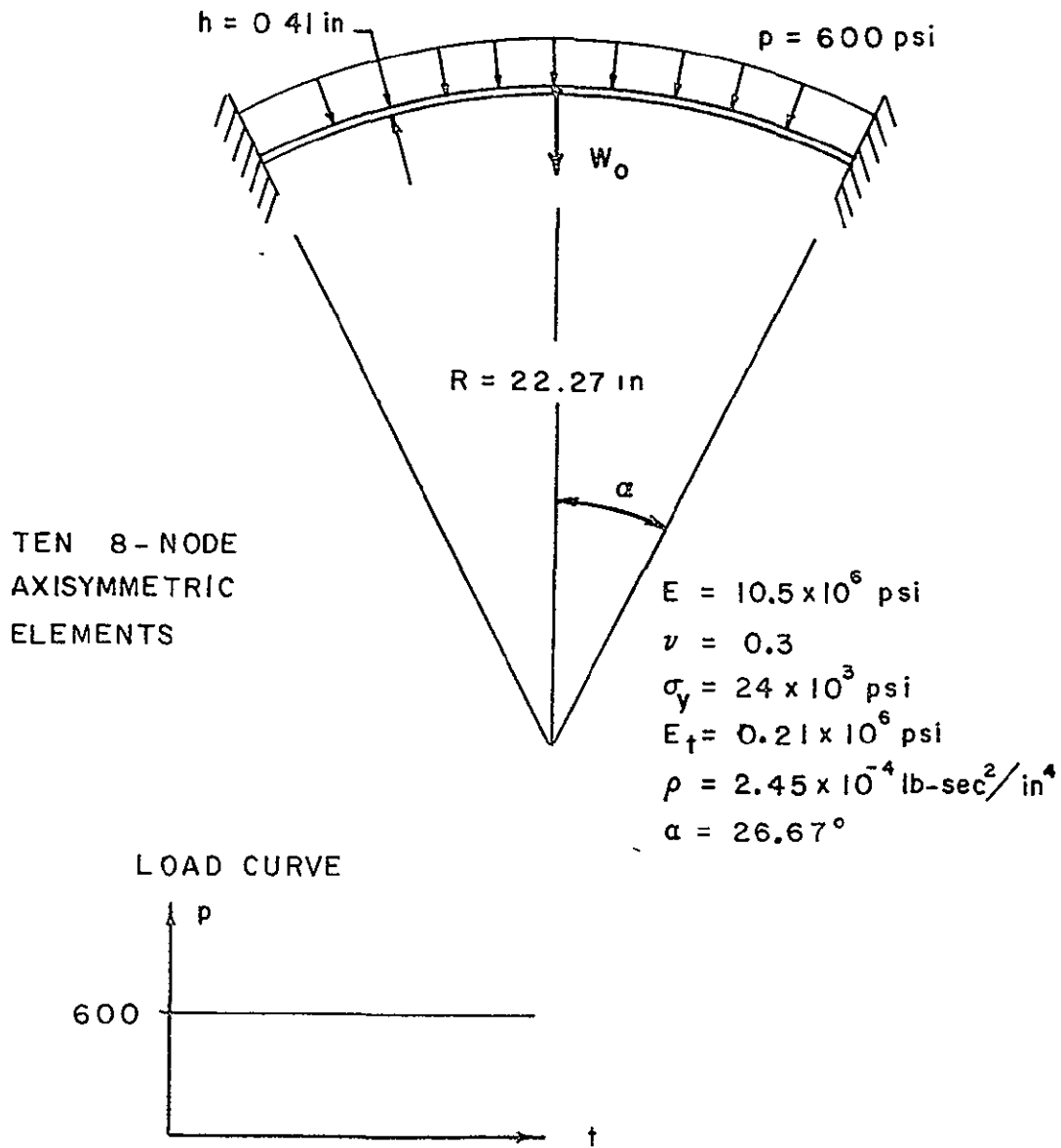


Fig. 40 Spherical cap with a uniform pressure loading.

ORIGINAL PAGE IS
OF POOR QUALITY

ALL SOLUTIONS:
 T.L. WITH MECHANICAL SUBLAYER
 $\Delta t = 5 \mu \text{ sec.}$

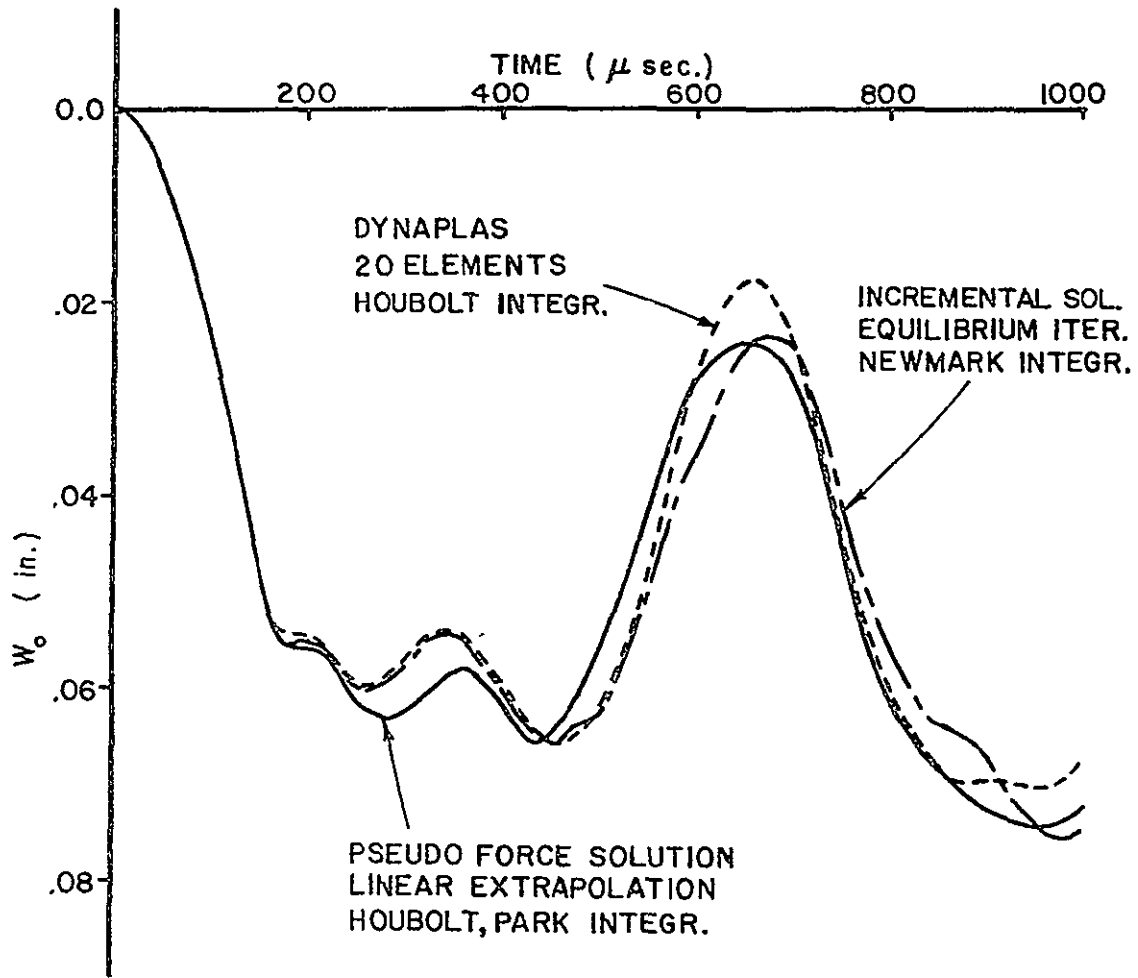


Fig. 41 Comparison of different nonlinear program solutions for a spherical cap.

ORIGINAL PAGE IS
OF POOR QUALITY

T.L. with Mechanical Sublayer
Pseudo Force Solutions - Houbolt Integration, Linear Extrap.
Incremental Solutions - Newmark Integration, Equilibrium Iter.

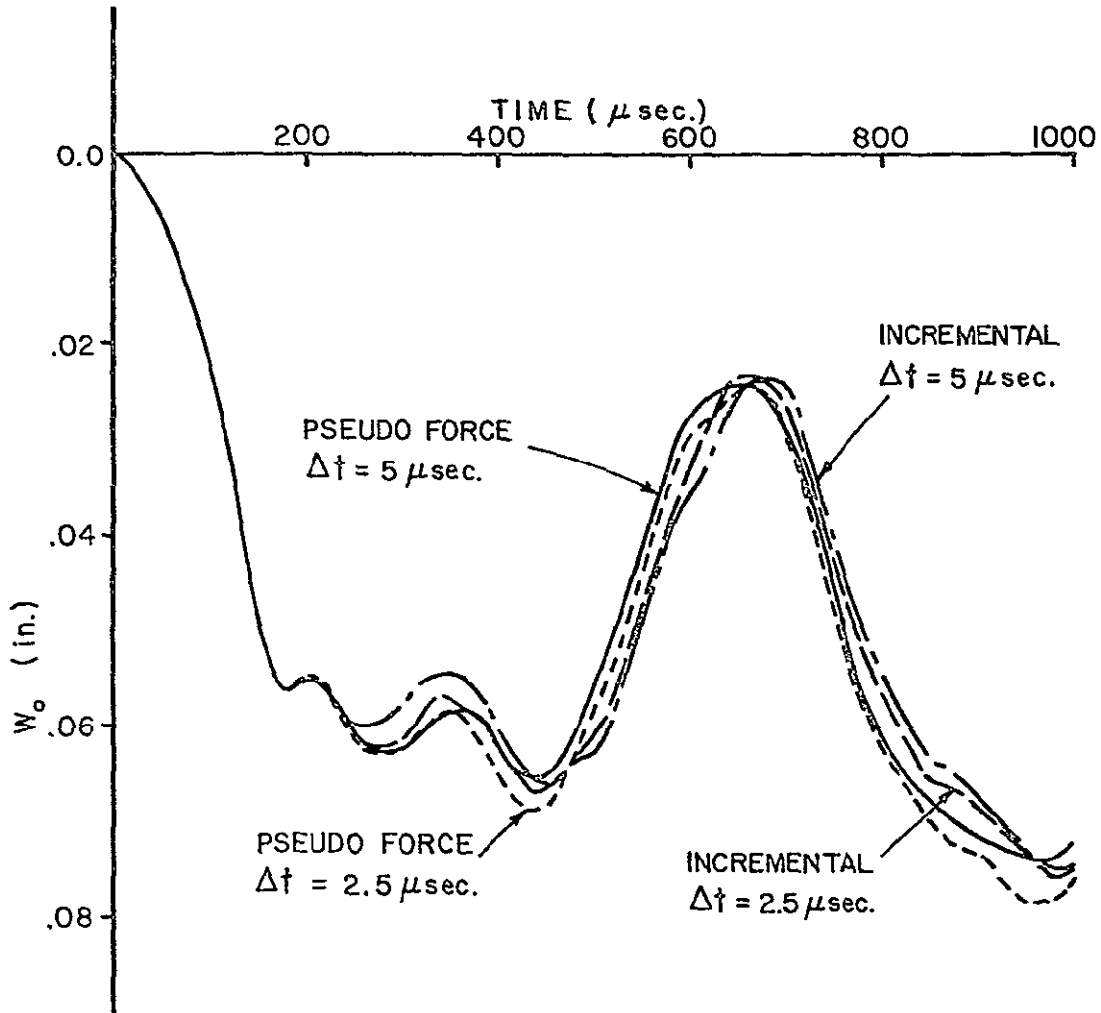


Fig. 42 Effects of time step size on nonlinear solutions for a spherical cap.

Example Problem 6

The next problem to be presented is the elastic-perfectly plastic analysis of a steel pressure vessel, as shown in Fig. 44 along with the finite element idealization used by AGGIE I. The results for the vertical deflection of point "A" predicted by the program using 45 8-node axisymmetric elements are shown in Fig. 45 along with experimental and theoretical results reported in Ref. 15. The material constants used were a Young's modulus of 29.12×10^6 psi, a Poisson's ratio of 0.3, and a yield stress of 40,540 psi. The finite element results presented by Nayak and Zienkiewicz¹⁵ were obtained using fifty-three axisymmetric isoparametric elements. Since the deflections are small, the analysis considered only material nonlinearity which was modeled by the isotropic hardening rule. Sixteen load steps were used to obtain the solution with a CPU execution time of 103 seconds. Similar results were obtained using only five load steps. The input for the sixteen step solution is shown in Fig. 46.

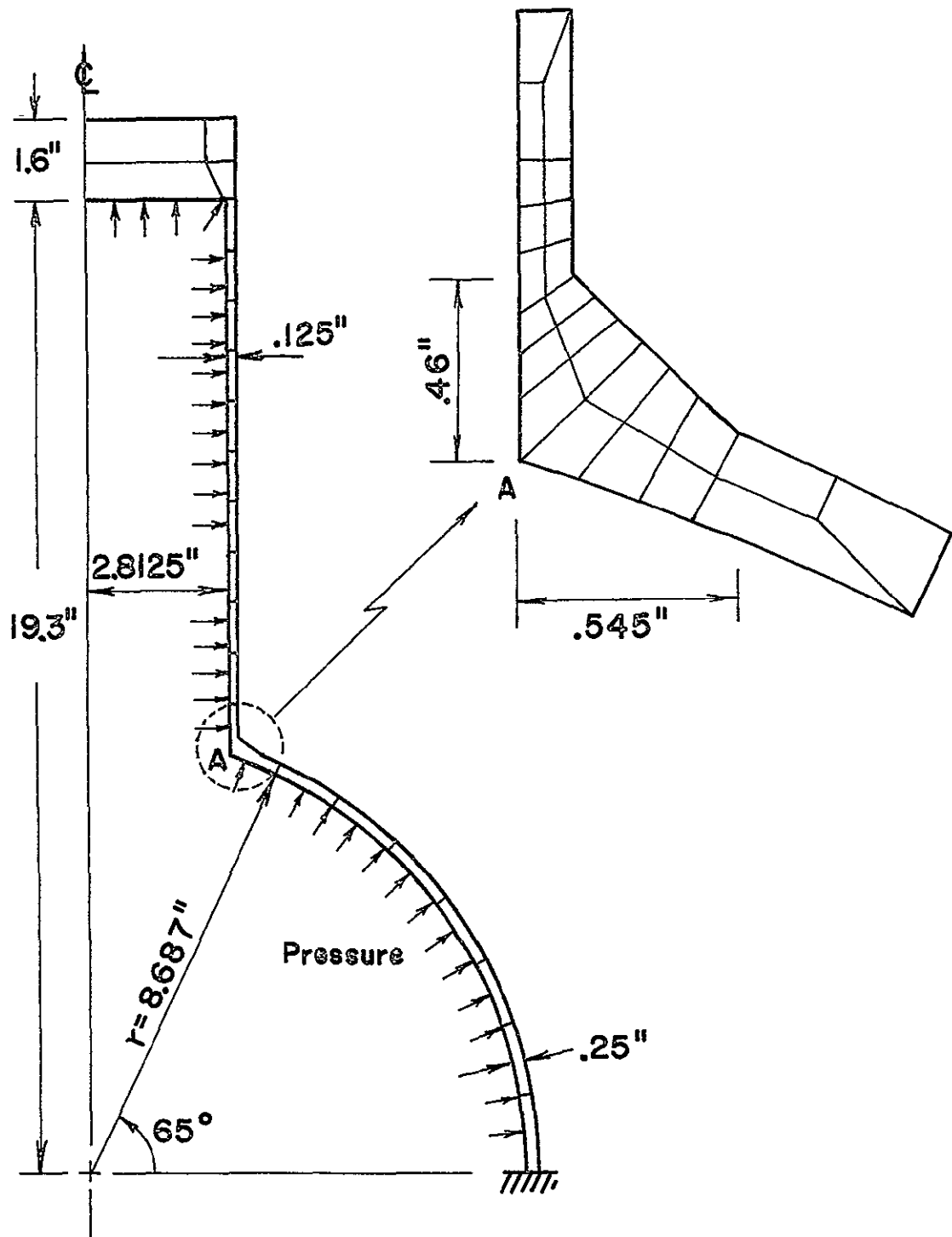


Fig. 44 Axisymmetric Steel Pressure Vessel Subjected to Uniform Internal Pressure

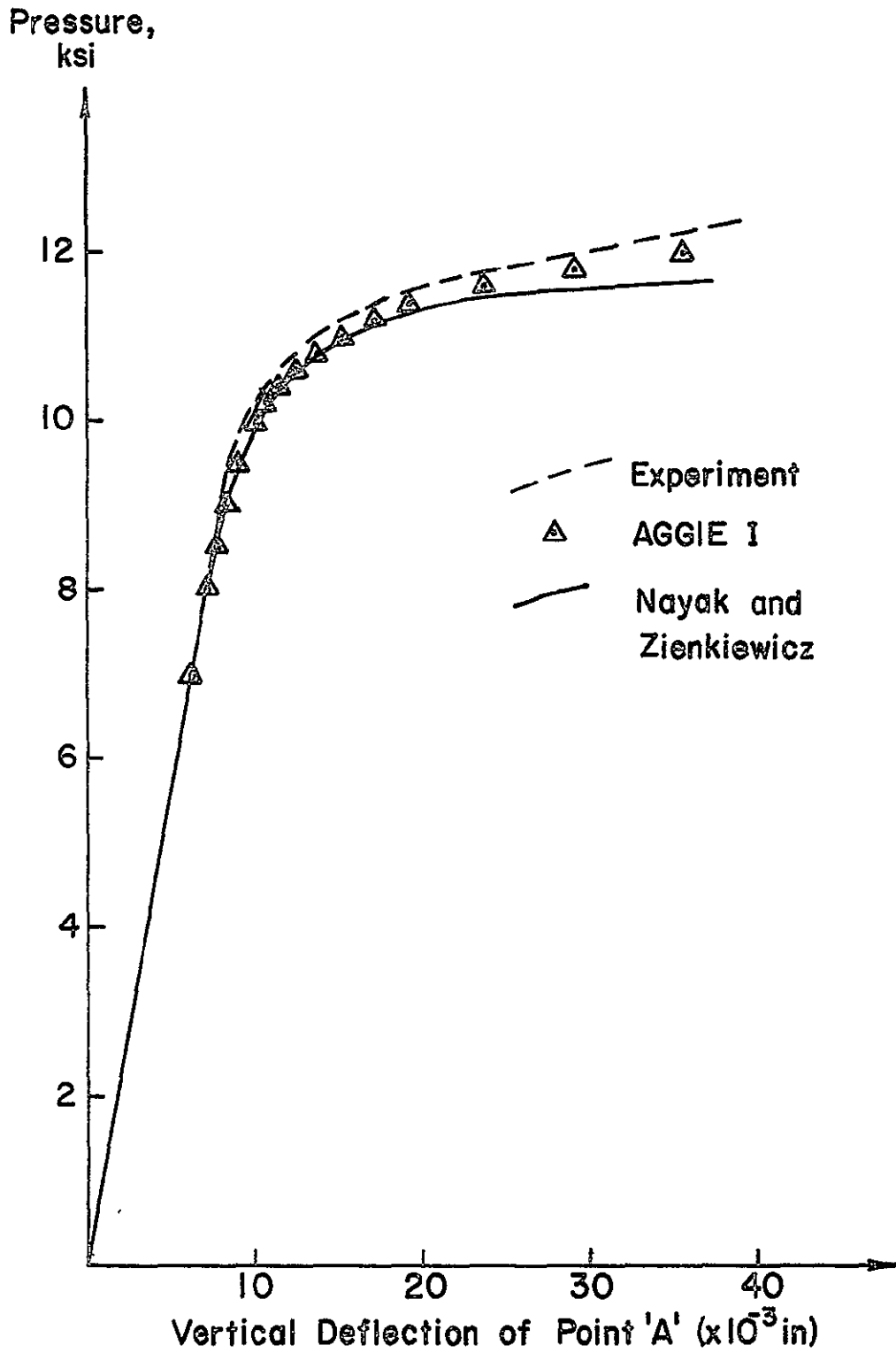


Fig. 45 Vertical Deflection of Point A for Axisymmetric Steel Pressure Vessel

10 20 30 40 50 60 70 80
 1234567890123456789012345678901234567890123456789012345678901234567890

***** PROBD06, STEEL PRESSURE VESSEL, NAYAK AND ZIENKIEWICZ *****

100111 18 1 1 0 16 1.0 0.0 0 0 0 1 6 7131418

1 1 15 0.001
 1 9 0 0 0 0 0 0
 1 0 1 0 0 0 0 0
 2 0 0 3 10 8 15 3
 0.0
 29120000. .3 1.0 40540. .00139217

1 3.671284839 7.873095745
 2 11 8.687 0.0
 3 11 8.93700 0.
 4 3.776939405 8.09967269
 5 7.326541489 4.667521697
 6 11 8.812 0.0
 7 7.537389351 4.801846599
 8 3.724112122 7.986384219
 2 0 0 3 10 8 3 3
 0.0 0 1

29120000. .3 1.0 40540. .00139217

1 3.518478424 8.079087422
 2 3.671284839 7.873095745
 3 3.776939405 8.099672692
 4 3.568388751 8.193690909
 5 3.594881631 7.976091583
 6 3.724112122 7.986384219
 7 3.672964917 8.147349121
 8 3.543433587 8.136389165
 2 0 0 3 10 8 3 3
 0.0 0 1

29120000. .3 1.0 40540. .00139217

1 3.31053933 8.166497
 2 3.263578661 8.050653595
 3 3.671284839 7.873095745
 4 3.518478424 8.079087422
 5 3.287058995 8.108575297
 6 3.468568097 7.964483935
 7 3.594881631 7.976091583
 8 3.414788571 8.123457577
 2 0 0 3 10 8 3 3
 0.0 0 1

29120000. .3 1.0 40540. .00139217

1 3.3575 8.282340415
 2 3.31053933 8.166497
 3 3.518478424 8.079087422
 4 3.568388751 8.193690909
 5 3.334019665 8.224418707

Fig. 46 Input Data for Example Problem 6

9. REFERENCES

1. Von Riesenmann, W.A., Stricklin, J.A., and Haisler, W.E., "Nonlinear Continua Software," Structural Mechanics Computer Programs, editors W. Pilkey, K. Saczalski and H. Schaefer, University Press of Virginia, Charlottesville, 1974, pp. 3-36.
2. Bathe, K.J., Wilson, E.L., and Iding, R.H., "NONSAP: A Structural Analysis Program for Static and Dynamic Response of Nonlinear for Static and Dynamic Response of Nonlinear Systems," Report No. UC SESM 74-3, Structural Engineering Laboratory, University of California, Berkeley, CA, Feb. 1974.
3. Bathe, K.J., Ozdemir, H., and Wilson, E.L., "Static and Dynamic Geometric and Material Nonlinear Analysis," Report No. UC SESM 74-4, Structural Engineering Laboratory, University of California, Berkeley, CA, Feb 1974.
4. Hunsaker, B., "The Application of Combined Kinematic-Isotropic and the Mechanical Sublayer Model to Small Strain Inelastic Structural Analysis by the Finite Element Method," Ph.D. Dissertation, Texas A&M University, Aug. 1976.
5. Takamatsu, T., "Nonlinear Finite Element Analysis of Incompressible Hyperelastic Materials Using Symmetric Stiffness Matrix," Ph.D. Dissertation, Texas A&M University, Dec. 1976.
6. Fung, Y.C., Foundations of Solid Mechanics, Prentice-Hall, Englewood Cliffs, New Jersey, 1965.
7. Mondkar, D.P. and Powell, G.H., "Towards Optimal In-Core Equation Solving," Computers & Structures, Vol. 4, 1974, pp. 531-548.
8. Mondkar, D.P. and Powell, G.H., "Large Capacity Equation Solver for Structural Analysis," Computers & Structures, Vol. 4, 1974, pp. 699-728.
9. Winter, R. and Levine, H.S., "Experiments on Large Plastic Deformations of Circular Plates with Work Hardening," Grumman Research Dept. Report RE-502, Grumman Aerospace Corp., Bethpage, N.Y., July 1975.
10. Levine, H.S. and Svalbonas, V., "Inelastic Nonlinear Analysis of Stiffened Shell of Revolution by Numerical Integration," Journal of Pressure Vessel Technology, May 1974, pp. 121-130.
11. Winter, R., Private Communications, Grumman Aerospace Corp., January 1977.
12. Haisler, W.E., "Numerical and Experimental Comparison of Plastic Work Hardening Rules," Proceedings of the 4th International Conference on Structural Mechanics in Reactor Technology, San Francisco, August 15-19, 1977.

13. Haisler, W.E., and Vaughan, D.K., "DYNAPLAS II - A Finite Element Program for the Dynamic Analysis of Stiffened Shells of Revolution," TEES Rept. 2926-73-2, Aerospace Engr. Dept., Texas A&M University, October 1973.
14. Richardson, E.Q., M.S. Thesis, Texas A&M University, August 1977.
15. Nayak, G.C. and Zienkiewicz, O.C., "Elasto-Plastic Stress Analysis - A Generalization for Various Constitutive Relations Including Strain Softening," International Journal for Numerical Methods in Engineering, Vol. 5, 1972, pp. 113-135.
16. Mescall, J.F., "Large Deflection of Spherical Shells Under Concentrated Loads," Journal of Applied Mechanics, December 1965, pp. 936-938.