1974

Development and applications of a quadratic isoparametric finite element for axisymmetric stress and deflection analysis

F. X. Janucik

DEVELOPMENT AND APPLICATIONS OF A QUADRATIC ISOPARAMETRIC FINITE ELEMENT FOR AXISYMMETRIC STRESS AND DEFLECTION ANALYSIS

by

F. X. Janucik

A Thesis Submitted

in

Partial Fulfillment

of the

Requirements for the Degree of

MASTER OF SCIENCE

in

Mechanical Engineering

Approved by:

Prof. Neville Rieger
Thesis Advisor

Dr. Robert McCalley Jr.
(External Reviewer)

Prof. B. Karlekar

Prof. Richard B. Hetnarski

Prof. Robert M Desmond
(Department Head)

DEPARTMENT OF MECHANICAL ENGINEERING

ROCHESTER INSTITUTE OF TECHNOLOGY

ROCHESTER, NEW YORK

June, 1974

ACKNOWLEDGEMENTS

The author would like to express his appreciation to those who have assisted him during the course of this investigation and in particular:

To Professor N. F. Rieger, his thesis advisor, for his interest and guidance of the author's thesis research and for the profound effect that association with him has had on the author's professional career.

To Dr. R. B. McCalley, Jr., external thesis examiner, for his continued interest, insight, and encouragement in the author's thesis research.

To Dr. J. F. Loeber and Mr. T. Glasser, Knolls Atomic Power Laboratory, for their insight, suggestions, and assistance concerning this investigation.

To Professors B. Karlekar, R. Hetnarski, W. Halbleib, M. Sherman, and W. Walters, Rochester Institute of Technology, Mechanical Engineering Department, for their interest, suggestions, and encouragement concerning this thesis research.

To the author's wife Barbara for her encouragement and patience.

To Mrs. B. Engle for her superior job in typing the final manuscript of this investigation.

ABSTRACT

The theory and computer program for an axisymmetric finite element for static stress and deflection analysis is presented. The element is an eight noded isoparametric quadrilateral based on the displacement method which is capable of representing quadratic variation of element boundaries and displacements. Element stiffness properties are developed for linear elastic small displacement theory using homogeneous isotropic material. Test cases are compared with theoretical solutions from the theory of elasticity to identify program capabilities and limitations.

Ability to analyse axisymmetric problems and to represent curved element boundaries has been demonstrated. Example problems including a cylindrical pressure vessel, a disk of uniform thickness subjected to centrifugal body force, and stress concentrations in a cylindrical rod due to a spherical inclusion are presented. In each of these cases program predicted deflection and stress values were within 2% of theoretical values.

Limitations which have been identified include the prediction of discontinuous stresses at adjacent element boundaries, failure to match original element boundary stress conditions in substructure analyses, and the necessity of double precision calculations to correctly analyse problems whose theoretical solutions obey small displacement plate theory. Analysis of a spherical pressure vessel resulted in predicted displacements within 4% of theoretical values while stresses on element boundaries varied by 60% from theoretical values. Substructure analysis for the spherical inclusion problem resulted in prediction of boundary stresses which were incompatible with those originally obtained. Techniques to overcome this difficulty are proposed but are not tested. The inability to obtain reasonable results for flexural problems was found to be due to round off error in the single precision technique used for solving the structure equilibrium relations. Use of double precision calculations resulted in displacements and stresses within .25% and 4.% respectively of theory for the case of a clamped circular plate loaded by a uniform pressure normal to its surface.

TABLE OF CONTENTS

PAGE

LIST OF FIGURES

LIST OF TABLES

NOMENCLATURE

Scalars

r, z, θ	Cylindrical coordinates (radial, axial, circumferential)
P, Q	Local normalized curvilinear coordinates
A	Area
V	Volume
u, v	Displacement components in the radial and axial directions respectively
F_{ir}, F_{iz}	Components of force acting in the radial and axial directions respectively at nodal point i
e_r, e_θ, e_z	Normal components of strain in the r, θ, z directions
γ_{rz}	Shearing strain in cylindrical coordinates
$\sigma_r, \sigma_\theta, \sigma_z$	Normal stress components in the r, θ, z directions
τ_{rz}	Shearing stress in the rz plane
U	Strain energy
E	Young's modulus of elasticity
ν	Poisson's ratio
ϕ	An arbitrary parameter varying within an element (e.g. displacement, geometry)
ϕ_i	Value of unknown at element nodal point i
N_i	Element shape function associated with nodal point i
α	Unknown polynomial coefficient

Vectors and Matrices

$\{\ \}$	Row or column vector
$[\ ]$	Matrix
$[\ ]^T, \{\ \}^T$	Matrix or vector transposed
$[\ ]^{-1}$	Inverse of a matrix
$\det[\ ]$	Determinant of a matrix
$\{r_n\}$	Column vector of radial coordinates for element nodal points
$\{z_n\}$	Column vector of element nodal point axial coordinates
$\{u_n\}$	Column vector of element node radial displacement components
$\{v_n\}$	Column vector of element node axial displacement components
$\{W_o\}$	Column vector containing both radial and axial displacement components of the element nodes
$\{N\}$	Row vector of element shape functions
$\{e\}$	Column vector of strain components
$\{\sigma\}$	Column vector of stress components
$[B]$	Matrix relating displacement to strain
$[J]$	Jacobian matrix
$[G]$	Matrix relating element nodal point locations to the Jacobian matrix
$[X_o]$	Matrix of element nodes spatial coordinates
$[D]$	Matrix relating stress to strain
$[K]$	Element stiffness matrix

Vectors and Matrices

$\{F_o\}$	Column vector listing element nodal point forces
$\{\delta W\}$	Column vector of virtual displacements in radial and axial direction of an element's nodal points
$\{\bar{B}\}$	Column vector of element body force components
$\{\bar{P}\}$	Column vector of element surface force components
$[N']$	Matrix of shape functions
$\{\Delta\}$	Column vector of structure nodal point displacement components
$[K]^s$	Structural stiffness matrix
$\{R\}$	Column vector of structure nodal point force components
$[S]$	Matrix relating stress to displacement

1.0 INTRODUCTION

All linear elastic static stress and deflection problems of axially symmetric continua are, in theory, capable of being solved using the finite element method. (e.g. pressure vessels, cooling towers, rocket nozzles). Limitations to the finite element method occur when numerous elements are required to achieve a desired degree of accuracy thus resulting in large computer core requirements and/or excessive cost.

Prior to 1968, finite elements having only linear variation of boundaries were available. Thus, when a curved geometric boundary was to be modelled, one was forced to introduce large numbers of elements to achieve acceptable results. This required the solution of a greatly increased number of equilibrium equations and was recognized as a limiting factor in the application of the finite element method to this type of problem.

Introduction of the isoparametric concept by Ergatoudis [8] enabled development of elements with polynomic variation of boundaries and led to a reduction in the number of elements necessary to idealize curved boundaries.

The objective of this thesis is to present details of an isoparametric finite element for axisymmetric stress analysis which is capable of representing quadratic variation of element boundaries exactly. The development of the element, a computer program, and demonstrative applications are presented.

The element developed is an eight-noded quadrilateral based on the isoparametric element concept. Its material properties are isotropic and linear. Element force-displacement relations are obtained using the displacement method of minimum potential energy.

2.0 LITERATURE SURVEY

For the case of axisymmetric bodies subjected to axially symmetric boundary conditions, Timoshenko shows that the three dimensional equations of elasticity in cylindrical coordinates (r,θ,z), reduce to equations in two dimensions (r,z) [1]*, [2].

Two papers exist which are considered the classic presentations of finite element development based on this theory.

Clough and Rashid[3] present a straight sided plane triangular element whose displacements are assumed linear functions of element spatial coordinates r and z. Element stresses are constant and are assumed to be average values acting at the element's centroid. Element property expressions (e.g. stiffness matrix, load vectors), are developed in integral form based on the principal of virtual work and are recognized as being complicated and lengthy. Three example problems are presented: two dealing with pressure vessel analyses, and a third with the response of an elastic half space to a point load. Highly refined finite element models involving large numbers of elements are used in all examples which appear to agree quite well with theory. Results are presented in graphic form. No specific comparisons of predicted to theoretical values are given.

Wilson[4] presents additional development and modifications for the Clough and Rashid element which increases its

*Numbers in square brackets refer to the references listed in Section 12.0.

ability to analyse a broader class of structural problems. Presented is the development for determining steady state thermal effects and a procedure for analysing axisymmetric bodies experiencing asymetric loads. The technique for the latter consists of introducing harmonic displacement functions and summing a series of two dimensional analyses.[1] Wilson notes the advantage of quadrilateral elements for automated mesh generation and presents development for a quadrilateral element which is actually degenerated into four linear displacement triangles. Factors which prohibit direct formulation of quadrilateral elements are not considered.

Superiority of the linear displacement trapezoidal element over its triangular counterpart has been demosntrated based on strain energy considerations by Parsons and Wilson [32]. The internal work done by one trapezoid is shown to be lower than that of two corresponding triangular elements experiencing similar boundary conditions and the implication is made that more and smaller triangular elements are necessary to achieve results which are as accurate as those obtained with quadrilaterals. Among the disadvantages discussed is the difficulty to integrate for the stiffness matrix for shapes other than trapezoidal and introduction of inter-element displacement incompatability when adjacent elements are not rectangular.

---

1. For additional information, see Crose [5] or Ergatoudis [8]

The concept of an isoparametric element capable of overcoming the above disadvantages is credited to Taig by Irons[7] and Ergatoudis[8]. The technique of introducing a local curvilinear coordinate system is due to Taig[8] but was also developed independently, including consideration of curved element edge formulation and numerical integration convergence criteria, by Irons[7].

Ergatoudis, working in collaboration with Irons and Zienkiewicz, was the first to present plane quadrilateral elements based on the isoparametric concept[31]. Elements for two dimensional stress analysis were developed assuming linear, quadratic, and cubic boundary and displacement variations. Numerous example problems were presented and compared with solutions from the theory of elasticity. The necessity of numerical integration is noted but not discussed in depth. Conclusions are drawn favoring isoparametric quadrilateral elements having assumed variation functions of higher than first order. Subsequent work by Ergatoudis[8] includes the formulation of isoparametric, axisymmetric quadrilaterals having quadratic, cubic, and quintic displacement and boundary variations. Example problems of pressure vessels, circular plates, and rotating shafts in which excellent results were obtained are presented. Justification for the choice of particular elements in some examples is not provided.

The basic theory for deriving isoparametric elements is available in numerous texts. Theory is presented by Desai and Abel[17] and Martin and Carey[34] but the most comprehensive treatment of the concept is presented by Zienkiewicz[9] - [12].

Irons establishes the efficiency of numerical integration[7] and presents efficient integration techniques for the experienced analyst[13] - [15]. A recent paper by Gupta and Mohraz[16] presents an efficient technique for the numerical integration of element stiffness matrices which may readily be placed in a programmable form. Also included is a second technique which minimizes the number of mathematical computations necessary and hence computer time. A comparison of computer times between the two shows the proposed technique to be more efficient.

Example problems which demonstrate the increased efficiency of higher ordered isoparametric elements are presented by Dario and Bradley[21] for triangular elements and Ergatoudis[8], [31] for quadrilaterals.

3.0 BASIC STEPS OF THE FINITE ELEMENT DISPLACEMENT METHOD

Finite element development for stress and deflection analysis may be based on either of two variational principles; i) principle of minimum potential energy or ii) complementary energy theorem. The principle of minimum potential energy states that the true deformations of a body are those which make its potential energy a minimum. Application of this principle results in algebraic equations of equilibrium. The complementary potential energy theorem may be used to obtain algebraic equations of compatibility The more commonly used principle is that of minimizing potential energy since it facilitates assemblage of structural equilibrium relations. This technique is referred to as the displacement method of finite element analysis.

Models comprised of finite elements based on the displacement method tend to be stiffer than actual structures. This fact is due to the restraint introduced in prescribing intra-element displacement variation. Refinement of idealizations or the use of higher order elements minimizes this effect and provides convergence to true displacement shapes.

The six basic steps of the finite element technique based on the displacement method are:

1. Discretization of a continuum into an equivalent system of finite elements which are interconnected at nodal points.

2. Selection of a interpolation formula to approximate the variation of displacement on and within element boundaries.
3. Derivation of element stiffness matrices giving equilibrium relations between the forces and displacements at each element nodal point.
4. Assembly of the element stiffness matrices based on nodal point force equilibrium and displacement compatability to obtain structural equilibrium relations.
5. Solution of the structural equilibrium relations for unknown displacements.
6. Solution of element stresses based on element nodal point displacements.

These steps are applicable for development of all finite element types (e.g. plane stress/strain, axisymmetric, three dimensional solid). Development of a specific element type requires further consideration of the governing elasticity equations. The foregoing steps will now be applied to the development of an isoparametric finite element for axisymmetric static stress analysis.

4.0 DEVELOPMENT OF THE QUADRATIC-AXISYMMETRIC FINITE ELEMENT

4.1 Interpolation Formula and Isoparametric Concept

The selection of an interpolation formula describing the variation of some unknown ϕ (e.g. radial or axial displacement) within an element is of foremost importance in developing a finite element based on the displacement method. This formula is generally expressed as:

$$\phi = \sum_{i=1}^{n} N_i \phi_i \qquad (1)$$

where N_i is a normalized "shape function" of polynomial form in spatial coordinates

ϕ_i is the value of the unknown function ϕ at element node i

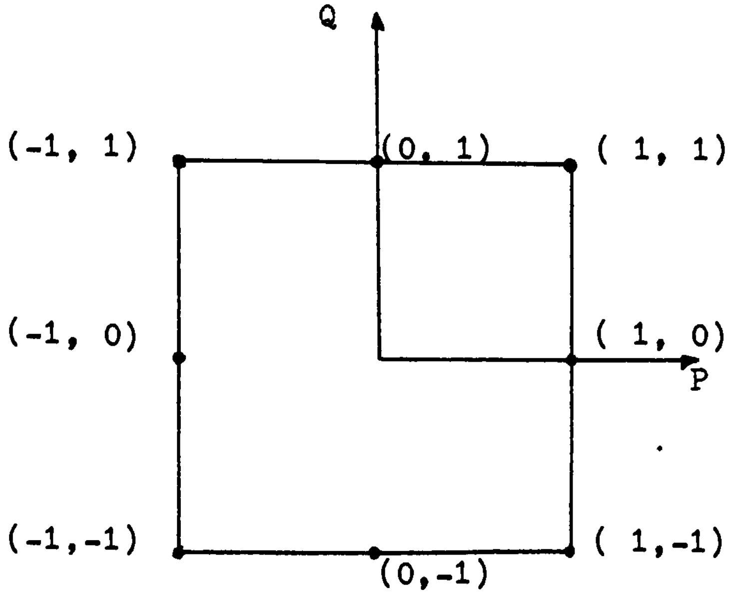
n is the number of nodes used to define the element

The shape functions in Eq. 1 may not be chosen arbitrarily if monotonic convergence is to be expected[10]. In order that finite element solutions converge to true solutions, shape functions must be chosen which:

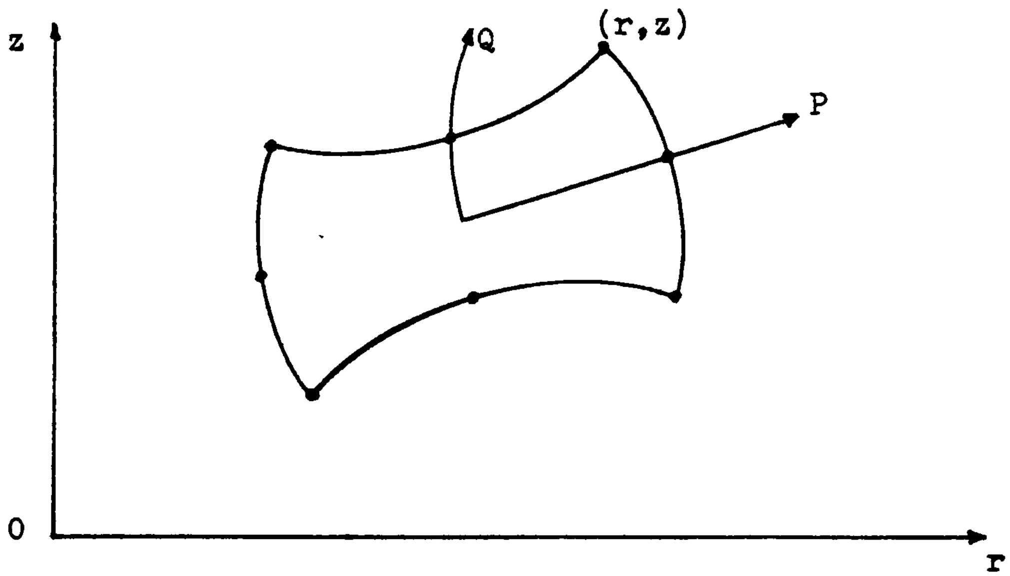
1. Are of such order and form that continuity of unknown ϕ occurs between elements.
2. Allow any arbitrary linear form of ϕ to be taken to represent constant derivatives.

With respect to element displacement, these requirements imply that no gaps or overlapping of adjacent element boundaries occur and that states of constant strain may be represented.

Although the quadrilateral element has been shown by Wilson and Parsons[32] to be superior to its triangular counterpart, the use of cartesian polynomials to define element shape functions is not suitable since convergence criteria can only be satisfied for the limited cases of elements being rectangles or parallelograms. The isoparametric concept enables specification of shape functions which will satisfy convergence criteria and also allow arbitrary element shapes which are consistent with assumed spatial variation. In the isoparametric concept, element shape functions are obtained for a square normalized element in a local coordinate system (P,Q). This coordinate system has its origin at the centroid of the element. Element boundaries have limits of -1 and 1 as shown in Fig. 1a. This normalized element and its shape functions are then associated with the curved element in spatial coordinates (r,z) shown in Fig. 1b. Therefore, coordinate system (P,Q) becomes curvilinear and both curved element displacement and geometry is expressed in terms of P and Q through Eq. 1.



a) local coordinate system



b) global coordinate system

Element Mapping

FIGURE 1

Zienkiewicz[12] suggested that shape functions which obey convergence properties may be obtained by inspection providing:

1. They have value of unity at the nodal point they refer to and zero at all other element nodes.
2. They have such an order of variation on element interfaces that the parameters specified on such interfaces uniquely define the function there.

Shape functions for a quadratic element which satisfy these criteria are presented in Table 1. The order in which these functions appear corresponds to the counterclockwise sequencing of nodal points shown in Figure 2.

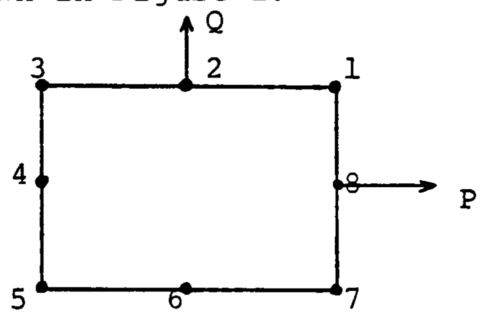


Figure 2. Location of Element Nodal Points and Associated Shape Functions

As stated previously, the order of the element interpolation formula (Eq.1) can be related to the number of nodes used to describe the element. Only six nodes would be required to specify a complete quadratic function in two variables. To maintain symmetry of the element, eight nodes are used. Expansion of Eq. 1 in terms of P and Q, using the above shape functions, the interpolation formula will be found to contain two terms of cubic order, PQ^2 and P^2Q. Therefore, although the element is referred to as quadratic, actual element variations are assumed which are higher order.

The axisymmetric problem in cylindrical coordinates may be completely specified in two dimensions. When axisymmetric boundary conditions exist, strain relations are completely specified in radial and axial coordinates (r,z), independant of θ. Thus, only two-dimensional finite elements in the r-z plane need be considered.

From Eq. 1, the variation of displacement within an element may be expressed as:

$$u = [N] \{u_n\} \tag{2}$$

$$v = [N] \{v_n\} \tag{3}$$

where u and v are radial and axial displacement components respectively at any point within the element.

[N]is a matrix of element shape functions:

$$[N] = [N_1, N_2, N_3, \ldots N_8]$$

$\{u_n\}, \{v_n\}$ are column vectors of element nodal point displacement components.

$$\{u_n\}^T = \{u_1, u_2, u_3, \ldots u_8\}$$

$$\{v_n\}^T = \{v_1, v_2, v_3, \ldots v_8\}.$$

By definition, element geometry is also defined by Eq. 1 and may be expressed as:

$$r = [N]\{r_n\} \tag{4}$$

$$z = [N]\{z_n\} \tag{5}$$

where r and z are element spatial coordinates in the radial and axial directions. $\{r_n\}$, $\{z_n\}$ are column vectors of element nodal point coordinates.

$$\{r_n\}^T = \{r_1, r_2, r_3, \ldots r_8\}$$

$$\{z_n\}^T = \{z_1, z_2, z_3, \ldots z_8\}$$

To demonstrate the element's ability to represent quadratic varying boundaries, consider Eqs. 4 and 5 for the case P = 1 which corresponds to the element edge defined by nodes 1, 7, and 8 in Fig. 2. From Table 1, shape functions N_2

through N_6 are zero and Eqs. 4 and 5 simplify to:

$$r = r_8 + \tfrac{1}{2}(r_1 - r_7)Q + \tfrac{1}{2}(r_1 + r_7 - 2r_8)Q^2$$

$$z = z_8 + \tfrac{1}{2}(z_1 - z_7)Q + \tfrac{1}{2}(z_1 + z_7 - 2z_8)Q^2$$

which represents a quadratic variation of element boundary.

These element displacement and geometry relations will now be used to establish element strain and stiffness properties.

TABLE I

ELEMENT SHAPE FUNCTIONS

i	N_i	$\frac{\partial N_i}{\partial P}$	$\frac{\partial N_i}{\partial Q}$
1	$\frac{1}{4}(1+P)(1+Q)(-1+P+Q)$	$\frac{1}{4}(1+Q)(2P+Q)$	$\frac{1}{4}(1+P)(2Q+P)$
2	$\frac{1}{2}(1-P^2)(1+Q)$	$-P(1+Q)$	$\frac{1}{2}(1-P^2)$
3	$\frac{1}{4}(1-P)(1+Q)(-1-P+Q)$	$\frac{1}{4}(1+Q)(2P-Q)$	$\frac{1}{4}(1-P)(2Q-P)$
4	$\frac{1}{2}(1-P)(1-Q^2)$	$-\frac{1}{2}(1-Q^2)$	$-Q(1-P)$
5	$\frac{1}{4}(1-P)(1-Q)(-1-P-Q)$	$\frac{1}{4}(1-Q()2P+Q)$	$\frac{1}{4}(1-Q)(2Q+P)$
6	$\frac{1}{2}(1-P^2)(1-Q)$	$-P(1-Q)$	$-\frac{1}{2}(1-P^2)$
7	$\frac{1}{4}(1+P)(1-Q)(-1+P-Q)$	$\frac{1}{4}(1-Q)(2P-Q)$	$\frac{1}{4}(1+P)(2Q-P)$
8	$\frac{1}{2}(1+P)(1-Q^2)$	$\frac{1}{2}(1-Q^2)$	$-Q(1+P)$

4.2 Strain-Displacement Relationships

As developed by Timoshenko [1], the linear strain-displacement relations for an axisymmetric body experiencing axisymmetric boundary conditions reduce to the following in cylindrical coordinates:

$$
\begin{aligned}
e_r &= \frac{\partial u}{\partial r} \\
e_\theta &= \frac{u}{r} \\
e_z &= \frac{\partial v}{\partial z} \\
\gamma_{rz} &= \frac{\partial u}{\partial z} + \frac{\partial v}{\partial r}
\end{aligned}
\qquad (6)
$$

where u and v are displacement components in the radial and axial directions respectively. Substituting Eqs. 2 and 3 into Eq. 6, the element strain may be expressed in matrix form as:

$$\{e\} = [B]\{W_o\} \qquad (7)$$

where $\{e\}^T = \{e_r \quad e_\theta \quad e_z \quad \gamma_{rz}\}$

$$[B] = [B_1 \quad B_2 \quad B_3 \; . \; . \; . \; B_8]$$

$$[B_i] = \begin{bmatrix} \frac{\partial N_i}{\partial r} & 0 \\ \frac{N_i}{r} & 0 \\ 0 & \frac{\partial N_i}{\partial z} \\ \frac{\partial N_i}{\partial z} & \frac{\partial N_i}{\partial r} \end{bmatrix}$$

and $\{W_o\}^T = \{u_1 \; v_1 \; u_2 \; v_2 \; . \; . \; . \; u_8 \; v_8\}$

The coefficients of matrix [B] contain derivatives of the element shape functions with respect to cylindrical coordinates. The shape functions are defined in terms of normalized coordinates (P,Q).

A relationship may be established between derivatives of two coordinate systems by the introduction of the Jacobian matrix of transformation from (r,z) to (P,Q) [23].

Applying the chain rule and differentiating shape function N_i with respect to P or Q, one obtains:

$$\frac{\partial N_i}{\partial P} = \frac{\partial N_i}{\partial r}\frac{\partial r}{\partial P} + \frac{\partial N_i}{\partial z}\frac{\partial z}{\partial P}$$

$$\frac{\partial N_i}{\partial Q} = \frac{\partial N_i}{\partial r}\frac{\partial r}{\partial Q} + \frac{\partial N_i}{\partial z}\frac{\partial z}{\partial Q}$$

or in matrix form:

$$\begin{Bmatrix} \dfrac{\partial N_i}{\partial P} \\ \\ \dfrac{\partial N_i}{\partial Q} \end{Bmatrix} = \begin{bmatrix} \dfrac{\partial r}{\partial P} & \dfrac{\partial z}{\partial P} \\ \\ \dfrac{\partial r}{\partial Q} & \dfrac{\partial z}{\partial Q} \end{bmatrix} \begin{Bmatrix} \dfrac{\partial N_i}{\partial r} \\ \\ \dfrac{\partial N_i}{\partial z} \end{Bmatrix}$$

where the matrix

$$\begin{bmatrix} \dfrac{\partial r}{\partial P} & \dfrac{\partial z}{\partial P} \\ \\ \dfrac{\partial r}{\partial Q} & \dfrac{\partial Z}{\partial Q} \end{bmatrix} = [J]$$

is called the Jacobian matrix.

Premultiplying both sides of the above equation by the inverse of the Jacobian, derivatives of the shape functions with respect to cylindrical coordinates may be expressed as:

$$\begin{Bmatrix} \dfrac{\partial N_i}{\partial r} \\ \\ \dfrac{\partial N_i}{\partial z} \end{Bmatrix} = [J]^{-1} \begin{Bmatrix} \dfrac{\partial N_i}{\partial P} \\ \\ \dfrac{\partial N_i}{\partial Q} \end{Bmatrix}$$

Determination of the Jacobian matrix is accomplished by differentiation of Eqs. 4 and 5 with respect to P and Q.

Applying the chain rule, the four coefficients of the Jacobian matrix become:

$$\frac{\partial r}{\partial P} = \sum_{i=1}^{8} \frac{\partial N_i}{\partial P} r_i ; \quad \frac{\partial z}{\partial P} = \sum_{i=1}^{8} \frac{\partial N_i}{\partial P} Z_i$$

$$\frac{\partial r}{\partial Q} = \sum_{i=1}^{8} \frac{\partial N_i}{\partial Q} r_i ; \quad \frac{\partial z}{\partial Q} = \sum_{i=1}^{8} \frac{\partial N_i}{\partial Q} Z_i$$

since the spatial coordinates of element nodes are constant.

These relations may be written in matrix form as:

$$[J] = [G] \; [X_O]$$

where

$$[G] = \begin{bmatrix} \frac{\partial N_1}{\partial P}, & \frac{\partial N_2}{\partial P}, & \frac{\partial N_3}{\partial P}, & \ldots, & \frac{\partial N_8}{\partial P} \\ \frac{\partial N_1}{\partial Q}, & \frac{\partial N_2}{\partial Q}, & \frac{\partial N_3}{\partial Q}, & \ldots, & \frac{\partial N_8}{\partial Q} \end{bmatrix}$$

$$[X_O]^T = \begin{bmatrix} r_1 & r_2 & r_3 & \ldots & r_8 \\ z_1 & z_2 & z_3 & \ldots & z_8 \end{bmatrix}$$

4.3 Stress-Strain Relations

Element stress-strain relations are presented for homogeneous, isotropic material.

For axially symmetric bodies, four components of stress exist. Normal stress components are in the axial, radial, and circumferential directions and shearing stress exists in the r z plane.

In the absence of initial strain, the relations between these element stresses and the element strains are:

$$\sigma r = \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} \quad e_r + \left(\frac{\nu}{1-\nu}\right) \left(e_\theta + e_z\right)$$

$$\sigma_\theta = \frac{E\ (1-\nu)}{(1+\nu)(1-2\nu)} \quad e_\theta + \left(\frac{\nu}{1-\nu}\right) \left(e_z + e_r\right)$$

$$\sigma_z = \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} \quad e_z + \left(\frac{\nu}{1-\nu}\right) \left(e_r + e_\theta\right)$$

$$\tau_{rz} = \frac{E}{2(1+\nu)} \quad \gamma_{rz}$$

where E represents Young's modulus

ν represents Poisson's ratio

These relations may then be expressed in matrix form as:

$$\left\{\sigma\right\} = [D] \left\{e\right\} \tag{9}$$

where

$$\left\{ \sigma^{T} \right\} = \left\{ \sigma_r \quad \sigma_\theta \quad \sigma_z \quad \tau_{rz} \right\}$$

$$[D] = \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1 & (\frac{\nu}{1-\nu}) & (\frac{\nu}{1-\nu}) & 0 \\ (\frac{\nu}{1-\nu}) & 1 & (\frac{\nu}{1-\nu}) & 0 \\ (\frac{\nu}{1-\nu}) & (\frac{\nu}{1-\nu}) & 1 & 0 \\ 0 & 0 & 0 & (\frac{1-2\nu}{2(1-\nu)}) \end{bmatrix}$$

4.4 Force - Displacement Relations

Relations between element nodal point forces and displacements may be obtained by the use of Castigliano's theorem. The strain energy of an axisymmetric element in a general state of stress may be expressed in matrix form as:

$$U = 1/2 \int_V \{\sigma\}^T \{e\} \, dV \qquad (10)$$

From Eqs. 7 and 9,

$$\{e\} = [B] \{W_o\}$$

$$\{\sigma\} = [D] \{e\} = [D] \; [B] \{W_o\}$$

The volume integral in Eq. 10 is expressed in cylindrical coordinates as:

$$\int_v dV = \int_z \int_r \int_o^{2\pi} r d\theta drdz = \int_z \int_r 2\pi rdrdz$$

From Eq. 4.

$$r = [N] \{r_o\}$$

Thus

$$\int_V dV = \int_z \int_r 2\pi \; [N] \{r_o\} \, drdz$$

Substituting the above relations into Eq. 10, the strain energy may be written as:

$$U = \int_z \int_r \pi \{W_o\}^T [B]^T [D] [B] \{W_o\} [N] \{r_o\} \, drdz \qquad (11)$$

Using the above relation and Castigliano's Theorem, the equilibrium relations between nodal point forces and displacements may be found.

Castigliano's Theorem states, "If the strain energy U of an elastic element is represented as a function of statically independant displacements, the partial derivative of this function with respect to displacements will give the actual forces at the displaced points in the directions of the displacements".[1]

Or

$$\frac{\partial U}{\partial \{W_o\}} = \{F_o\}$$

where $\{F_o\}$ refers to nodal point force components of an element.

$$\{F_o\}^T = \{F_{1r}, F_{1z}, F_{2r}, F_{2z}, \ldots F_{8r}, F_{8z}\}$$

Applying the above to Eq. 11 we obtain,

$$\{F_o\} = < \int_z \int_r 2\pi [B]^T [D] [B] [N] \{r_o\} \, drdz > \{W_o\}$$

or

$$\{F_o\} = [K] \{W_o\}$$

where

$$[K] = \int_z \int_r 2\pi [B]^T [D] [B] [N] \{r_o\} \, drdz \qquad (12)$$

and is the element stiffness matrix.

Evaluation of element stiffness by direct integration of Eq. 12 is not practical. Matrices [B] and [N]are expressed in curvilinear coordinates and would require transformation to cylindrical coordinates. Also, limits of integration are complicated by the curved boundaries shown in Fig. 1b.

These difficulties are overcome by transforming Eq. 12 to an integral in the local normalized coordinate system shown in Fig. 1a. This transformation is accomplished by recognizing that the determinant of the Jacobian matrix is equal to the ratio of differential areas in global (drdz) and local (dPdQ) coordinates [23].

$$drdz = \det[J]dPdQ$$

Applying this relation to Eq. 12 and changing limits of integration, the element stiffness matrix may be expressed as:

$$[K] = \int_{-1}^{1}\int_{-1}^{1} 2\pi[B]^T[D][B][N]\{r_o\} \det[J]\ dPdQ \qquad (13)$$

where all quantities within the integral are either constants or functions of P and Q.

Although limits of integration have been simplified, the quadratic form of the shape functions result in an expression to be integrated which is complex in form and not practical to integrate analytically. For this reason, evaluation of Eq. 13 is most readily accomplished by numerical integration using the Gauss quadrature technique. Details of the procedure used herein are presented in Appendix A.

4.5 Distribution of Element Loads to Nodal Points

In the finite element method, structural loading conditions are represented as point loads applied at the nodes of the idealized structure. In cases where distributed surface and body forces are present, these forces may be "intuitively" distributed to the nodal points, or a specific routine may be used.

In the case of higher order elements there is a departure from an easily conceived idealization and the allocation of distributed loads to nodal points by intuition may no longer be correct,[12]. However, nodal point loads, consistent with the assumed displacement functions, may be formulated for distributed loads by considering the Principal of Virtual Work, viz:

"If an element which is in equilibrium under a set of body forces ($\{\bar{B}\}$) and surface forces ($\{\bar{P}\}$), is given an arbitrary virtual displacement $\{\delta W\}$, which does not violate kinematic and geometrical boundary constraints, then the work done by the internal forces equals the work done by the applied loads during these displacements,"[19].

This statement leads to the matrix equation:

$$\int_A \{\delta W\}^T \{\bar{P}\}\, dA + \int_V \{\delta W\}^T \{\bar{B}\}\, dV = \{\delta W_o\}^T \{F\} \qquad (17)$$

where

dA = differential surface area of an element boundary

dV = differential volume within the element

$\{\delta W_o\}$ = virtual displacements of the element's nodes

$\{\delta W\}$ = virtual displacements within the element

Note also that $\{\delta W\} = [N']\{\delta W_o\}$ from assumed variation of intraelement displacement.

$$[N'] = \begin{bmatrix} N_1 & o & N_2 & o & N_3 & o & . & . & . & N_8 & o \\ o & N_1 & o & N_2 & o & N_3 & . & . & . & o & N_8 \end{bmatrix}$$

F = force components at element nodal points

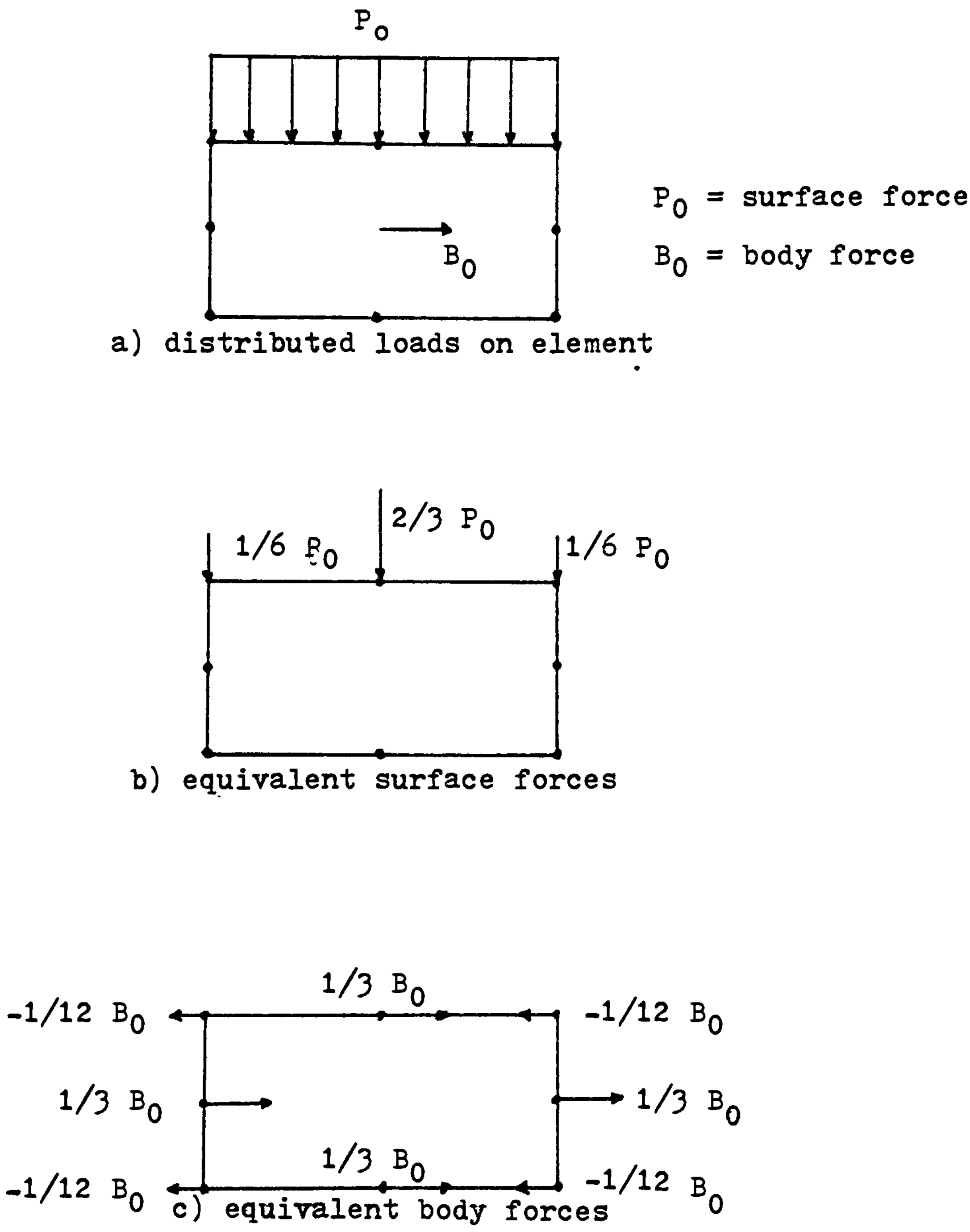
As $\{\delta W\}^T = \{\delta W_o\}^T [N']^T$

Eq. 17 becomes:

$$\int_A [N']^T\{\bar{P}\} dA + \int_V [N']^T\{\bar{B}\} dV = \{F\} \qquad (18)$$

The increased flexibility introduced in defining element shapes in cylindrical coordinates complicated the limits of integration in Eq. 18. It is found convenient to transform these integrals to the local coordinate system and integrate numerically as was done with the element stiffness matrix.

The option to internally generate these consistent loads has not been developed in the program presented, but, wherever required, allocation of distributed loads has been made as shown in Fig. 3.



Allocation of Surface and Body Forces
For a Regular Quadrilateral

Figure 3

5.0 STRUCTURAL EQUILIBRIUM RELATIONS-THE STRUCTURAL STIFFNESS MATRIX

For the previously developed element stiffness matrix [K], equilibrium equations relating element nodal point forces to displacements were obtained.
The next step of the displacement method is the determination of equilibrium relations between nodal point forces and displacements for the entire structure or, the structural stiffness matrix $[K]^s$.
The almost universally employed technique for obtaining this matrix is the direct stiffness method [17] which involves assembling the individual element stiffness matrices such that both displacement compatability and force equilibrium are satisfied at the nodal points, as follows:

1. All elements adjacent to a particular node must have the same displacement components at that node.
2. The external forces acting at a nodal point must equal the sum of the internal forces contributed by the elements meeting at the node.

Using these criteria, the structural stiffness matrix $[K]^s$ may be obtained by direct addition of the individual elements' stiffness coefficients to their appropriate locations in $[K]^s$.

These appropriate locations are determined by the nodal points defining each element.

Two important properties which the structural stiffness matrix possesses are

1. For linear elastic systems the element stiffness matrix is symmetric (i.e. $[K] = [K]^T$) and the assembled structural stiffness matrix is also symmetric.
2. Sequencing of elements and nodal points such that the maximum difference between nodal point numbers defining an element is a minimum, the resulting structural stiffness matrix will be banded as shown in Fig. 4.

Proof of these properties may be found in either reference [12] or [17].

Although these properties may not appear significant, they play an important role in an efficient scheme for solution of the structural equilibrium equations which requires a minimum amount of computer core capacity.

$$[K]^S = \begin{bmatrix} X & X & X & O & O & O & O & O & O & O \\ X & X & X & X & O & O & O & O & O & O \\ X & X & X & X & X & O & O & O & O & O \\ O & X & X & X & X & X & O & O & O & O \\ O & O & X & X & X & X & X & O & O & O \\ O & O & O & X & X & X & X & X & O & O \\ O & O & O & O & X & X & X & X & X & O \\ O & O & O & O & O & X & X & X & X & X \\ O & O & O & O & O & O & X & X & X & X \end{bmatrix}$$

Figure 4

A Banded Structural Stiffness Matrix (x = non-zero terms)

6.0 SOLUTION FOR STRUCTURAL NODAL POINT DISPLACEMENTS

Having found the structural stiffness matrix $[K]^s$, the structural equilibrium relations may be written as:

$$\{R\} = [K]^s \{\Delta\} \tag{19}$$

where $\{R\}$ = a vector of external force components acting at the nodes of the structure

$\{\Delta\}$ = a vector of displacement componets of the nodal points of the structure

The external forces applied at nodal points may be added directly to their appropriate locations in vector $\{R\}$. Also required is a sufficient number of prescribed displacement components in the vector $\{\Delta\}$ to prevent rigid body motion of the structure. Failure to constrain rigid body motions will result in matrix $[K]^s$ being singular and not possessing an inverse.

Introduction of prescribed displacements to Eq. 19 is accomplished by modification of $\{R\}$ and $[K]^s$ such that vector $\{\Delta\}$ will remain a vector of unknowns but yield the correct prescribed displacements when solved.

Having defined vector $\{R\}$ and introduced prescribed displacements, Eq. 19 may be solved.

Computer subroutines for the assemblage of the structural stiffness matrix and solving structure equilibrium equations were taken from an existing finite element computer program developed at the General Electric Research and Development Center by Levy[20] and used with only minor modifications for accomodation of the element developed.

Although no documentation of the above techniques is available in this report, the procedures used are similar to those presented by Cheung and King.[12].

The specific numerical technique used in finding displacements is a direct solution method using Gauss elimination for a tridiagonal matrix whose coefficients are themselves matrices.

Advantage of this technique is that a minimal amount of computer core required as all zero coefficients outside the bandwidth need not be retained. However, frequent accesses to peripheral storage devices during the Gauss elimination tends to increase total computer time.

As a result of the minimizing of core requirements possible using this technique, the computer program given in this thesis is capable of handling 600 nodes or 1200 displacement degrees of freedom. Such a problem corresponds to $[K]^s$ being of the order 1200x1200 and would require 1.44×10^6 words of computer storage with full retention of the structural stiffness matrix. The computer core required for the solution of this problem using the tridiagonal method is 10,100 words.

7.0 DETERMINATION OF ELEMENT STRESSES

Having determined nodal point displacements, it is then desirable to find stress components within the structure. Structural stress components are determined on a per-element basis and may be determined by a number of different techniques.

Three techniques currently used for obtaining element stress components, are:

1. Calculating stress components at element centroids and assuming these to be the average values of stress within each element. [17]
2. Assuming a polynomial variation of stress components and extrapolating these components to element boundaries. [20]
3. Calculation of consistent stress distributions based on the theory of conjugate approximations,[24] ,[25].

Of these three techniques, the second has been employed. The first technique was found to be too limited in stress information available while the third required sophistication beyond the scope of this thesis.

An advantage of the second technique is its ability to determine stresses on element boundaries, (where magnitudes are often a maximum) with a minimum of effort. Its disadvantage is that values of stress components calculated at a point similar to adjacent elements may exhibit finite discontinuities between the elements. This is demonstrated in section 8.4.

From Eq. 7, the matrix expression:

$$\{e\} = [B]\{W_o\}$$

was obtained which related element strain to its nodes' displacements.

From Eq. 9, the element stress vector was expressed as:

$$\{\sigma\} = [D]\{e\}$$

The relationship between stress and displacement is then:

$$\{\sigma\} = [D][B]\{W_o\}$$

where the matrix product [D] [B] is often referred to as the stress matrix [S] .

Stress components may be found at the midside nodes of each element by considering the element in its local normalizing coordinates.

As shown in Appendix A, the product [D] [B] is found at nine sampling points within an element when determining element stiffness. The locations of these sampling points are shown in Figure 5.

Since the locations and stress matrices of the sampling points are known, it is possible to extrapolate these matrices to the element's midside nodes.

Consider Fig. 5 for the case of P = 0. By definition, element nodes 2 and 6 lie on this line, and also sampling points 4, 5, and 6.

Assuming quadratic variation of the stress matrix as a function of Q, the stress matric [S(Q)] may be written as:

$$[S(Q)] = \alpha_1 + \alpha_2 Q + \alpha_3 Q^2$$

where α_1, α_2, and α_3 are unknown coefficients to be determined.

The stress matrices at nodes 2 and 6 become:

$$[S] \text{ node } 2 = \alpha_1 + \alpha_2 + \alpha_3$$

$$[S] \text{ node } 6 = \alpha_1 + \alpha_2 + \alpha_3$$

Denoting $[S]_i$ and a_i as the stress matrix and coordinate Q of sampling point i respectively, the following three equations are obtained.

$$[S]_4 = \alpha_1 + a_4\alpha_2 + a^2_4\,\alpha_3$$

$$[S]_5 = \alpha_1 + a_5\alpha_2 + a^2_5\,\alpha_3$$

$$[S]_6 + \alpha_1 + a_5\alpha_2 + a^2_6\,\alpha_3$$

The above represents 3 equations having 3 unknowns and may be solved forα_1, α_2, and α_3.

Using the same procedure for the case Q = 0, stress matrices at nodes 4 and 8 may be obtained in terms of the stress matrices at sampling points 2, 5, and 8.

Relationships between element midside node stress matrices and the stress matrices at the sampling points are presented in Table II.

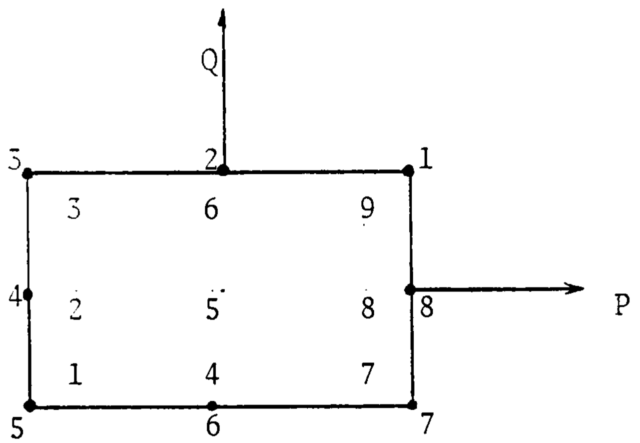


Figure 5. Quadrature Sampling Points• and Element Nodal Point Locations

TABLE II

Midside Node Stress Matrices

$$[S]\text{ node }2 = 0.1878\,[S]_4 - .6666\,[S]_5 + 1.4788\,[S]_6$$

$$[S]\text{ node }6 = 1.4788\,[S]_4 - .6666\,[S]_5 + 0.1878\,[S]_6$$

$$[S]\text{ node }4 = 1.4788\,[S]_2 - .6666\,[S]_5 + 0.1878\,[S]_8$$

$$[S]\text{ node }8 = 0.1878\,[S]_2 - .6666\,[S]_5 + 1.4788\,[S]_8$$

8.0 EXAMPLE PROBLEMS

A computer program has been written based on the foregoing development.

Numerous test cases have been examined to verify the computer program developed. Five test cases are presented to demonstrate program capabilities. Although limited in geometric and loading complexities, they are sufficiently representative to provide insight into the capabilities and limitations of the program.

The five test cases in order of presentation are:

TC 1. Cylindrical pressure vessel subjected to internal and external pressures.

TC 2. Stresses in a circular disk of uniform thickness due to centrifugal loading.

TC 3. Stress concentration in a cylindrical rod in tension due to a spherical inclusion.

TC 4. Spherical pressure vessel subjected to internal pressure.

TC 5. Bending of circular plates.

Results for cases similar to TC 1 and TC 3 have been published by Dario and Bradley [21] using quadratic triangular elements and results for cases similar to TC 2 and TC 5 using cubic and quartic quadrilateral elements have been presented by Ergatoudis [8].

All numerical results presented are in either tabular or graphical form as the actual computer output is too voluminous.

8.1 STRESSES AND DEFLECTIONS IN A CYLINDRICAL PRESSURE VESSEL TC 1

This case is presented to verify the ability of the program to solve axisymmetric problems and involves the classical thick cylinder problem from the theory of elasticity. The theoretical solution of this problem is due to Lame and is presented by Timoshenko[1]. Cylinder geometry and loading is presented in Fig. 6. Of primary interest is radial stress, hoop stress, and radial displacement. The theoretical displacement solution contains $1/r$ terms and stresses terms involving $1/r^2$.
Refinement of finite element models is necessary to approximate true stresses and displacements since actual variations are of higher order than those assumed within an element. Three finite element idealizations are presented having 1, 5, and 30 elements respectively. These models are shown in Fig. 7 and their results summarized in Table III. Graphs of radial stress, hoop stress, and radial displacement are presented in Fig. 8 for the 5 and 30 element models comparing their results with theory. For the 30 element model, stresses and deflections have converged to within a maximum difference of 1.0% of theoretical values at all locations.
Comparison information for the problem shown in Fig. 6 is available in a paper by Dario and Bradley[21] A

comparison between predicted stresses for the quadratic quadrilateral and linear and quadratic triangles is presented in Table VI, displacement information is not available. Superiority of the quadratic quadrilateral over the linear triangle is apparent. Advantage over its triangular counterpart is not as evident.

An unexpected result of this analysis was the prediction of displacements converging to the true solution from an upper bound. This contradicts the fact that elements based on the displacement method always prove too stiff. Two exceptions to this rule occur when either interelement displacement compatibility is not maintained or when element volume integration is approximate. Neither of these exceptions are believed to apply in this development. Also, similar displacement results were not obtained in other example problems. Explanation of this result is not available.

Results demonstrate functioning of the thesis program and also that the accuracy is a function of model refinement.

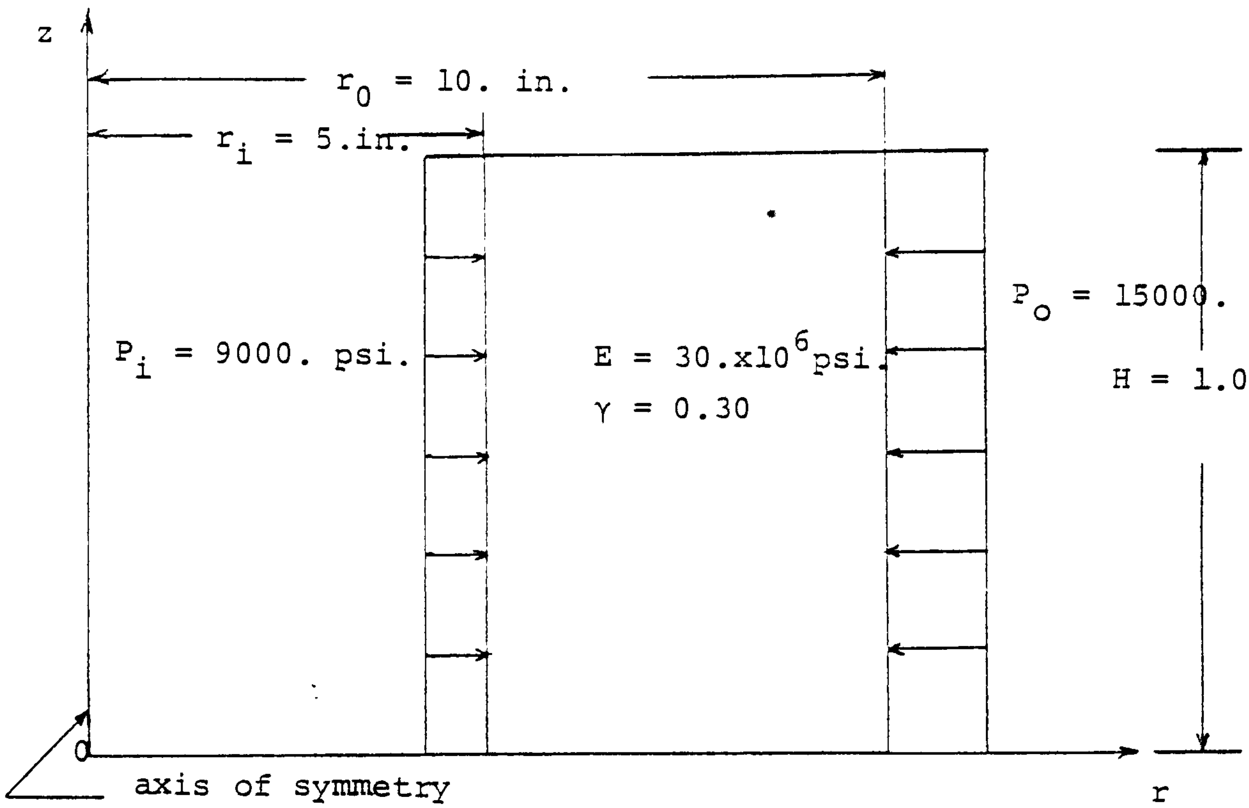
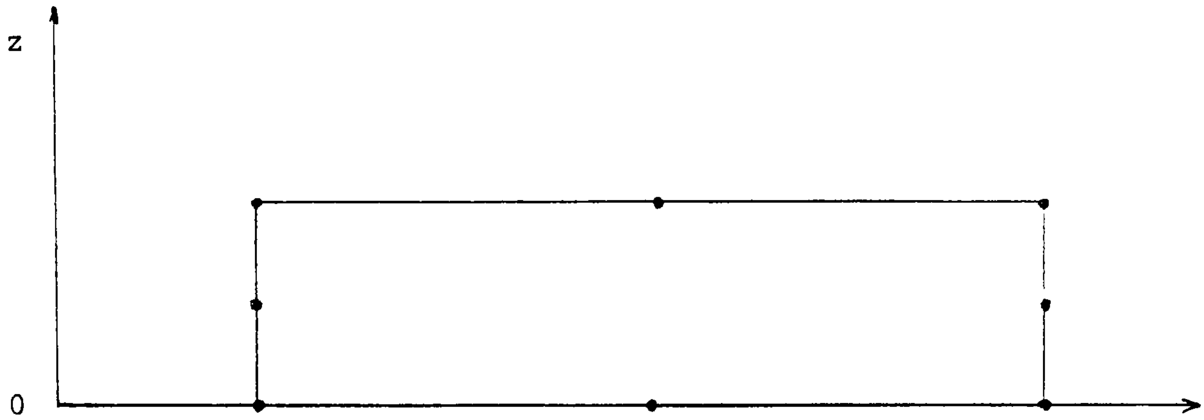
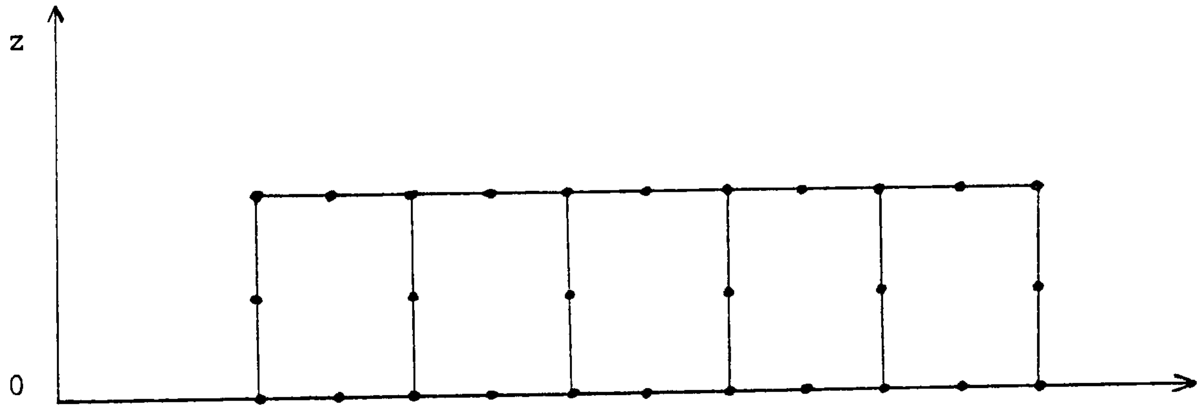


FIGURE 6

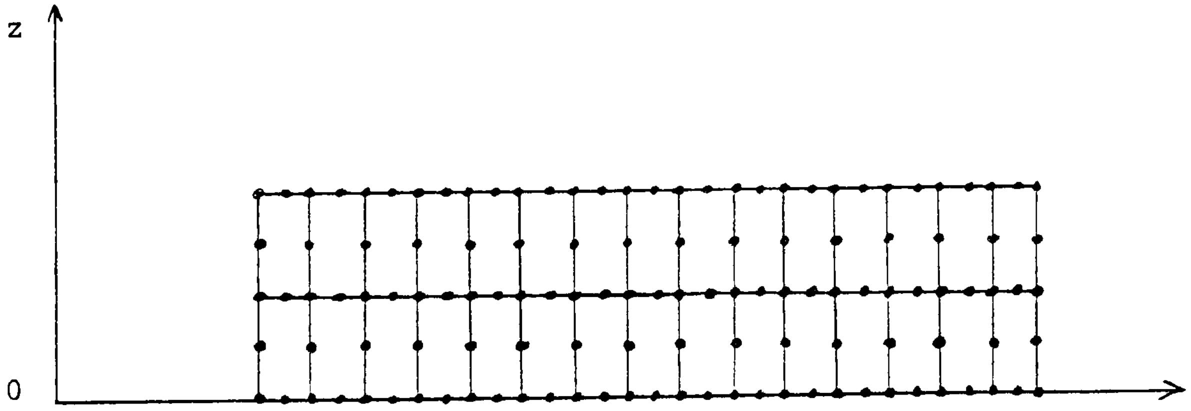
Thick Cylindrical Pressure Vessel



a) 14 unconstrained degrees of freedom



b) 54 unconstrained degrees of freedom



c) 248 unconstrained degrees of freedom

FIGURE 7

Finite Element Idealizations of TC 1

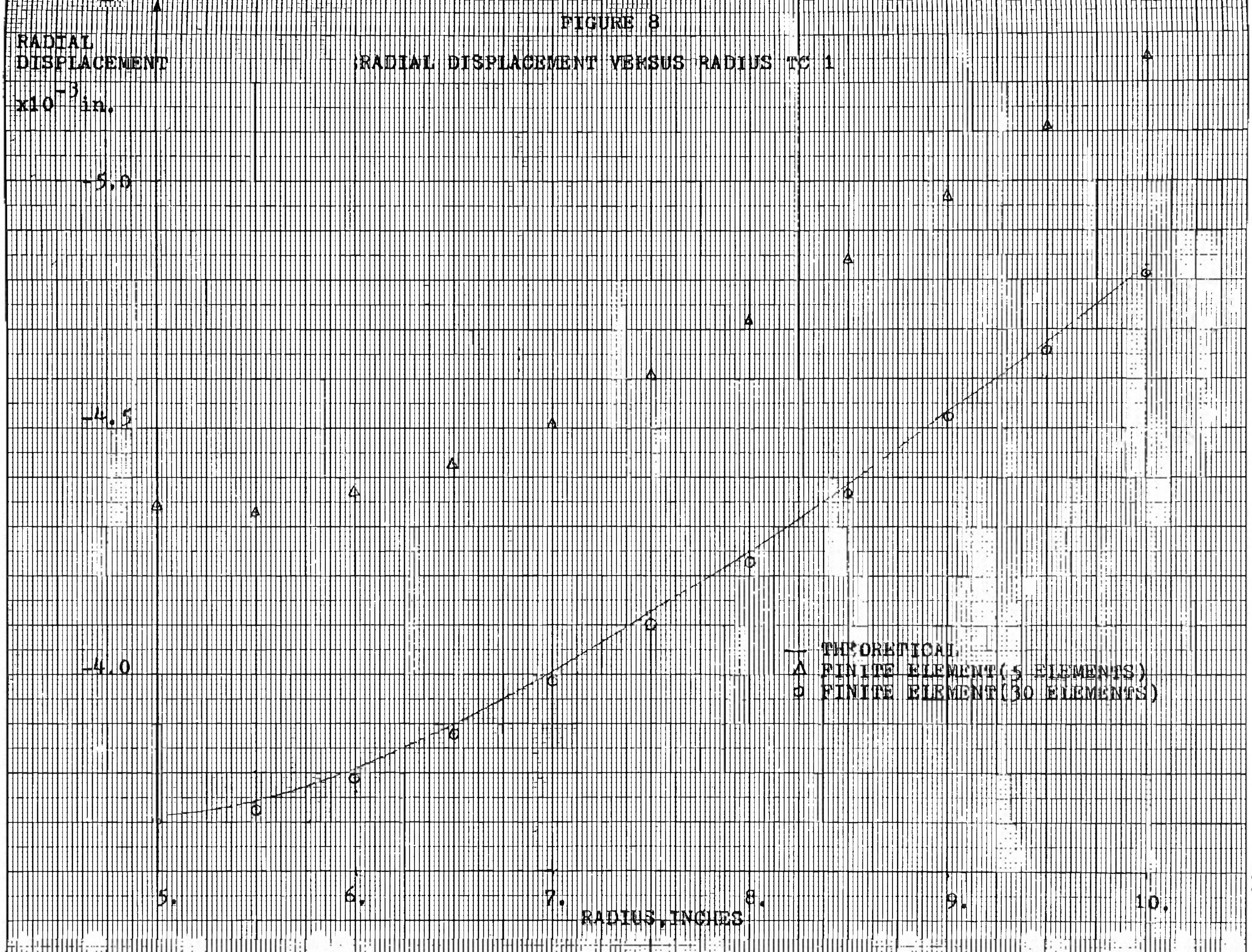


FIGURE 8

RADIAL DISPLACEMENT VERSUS RADIUS TC 1

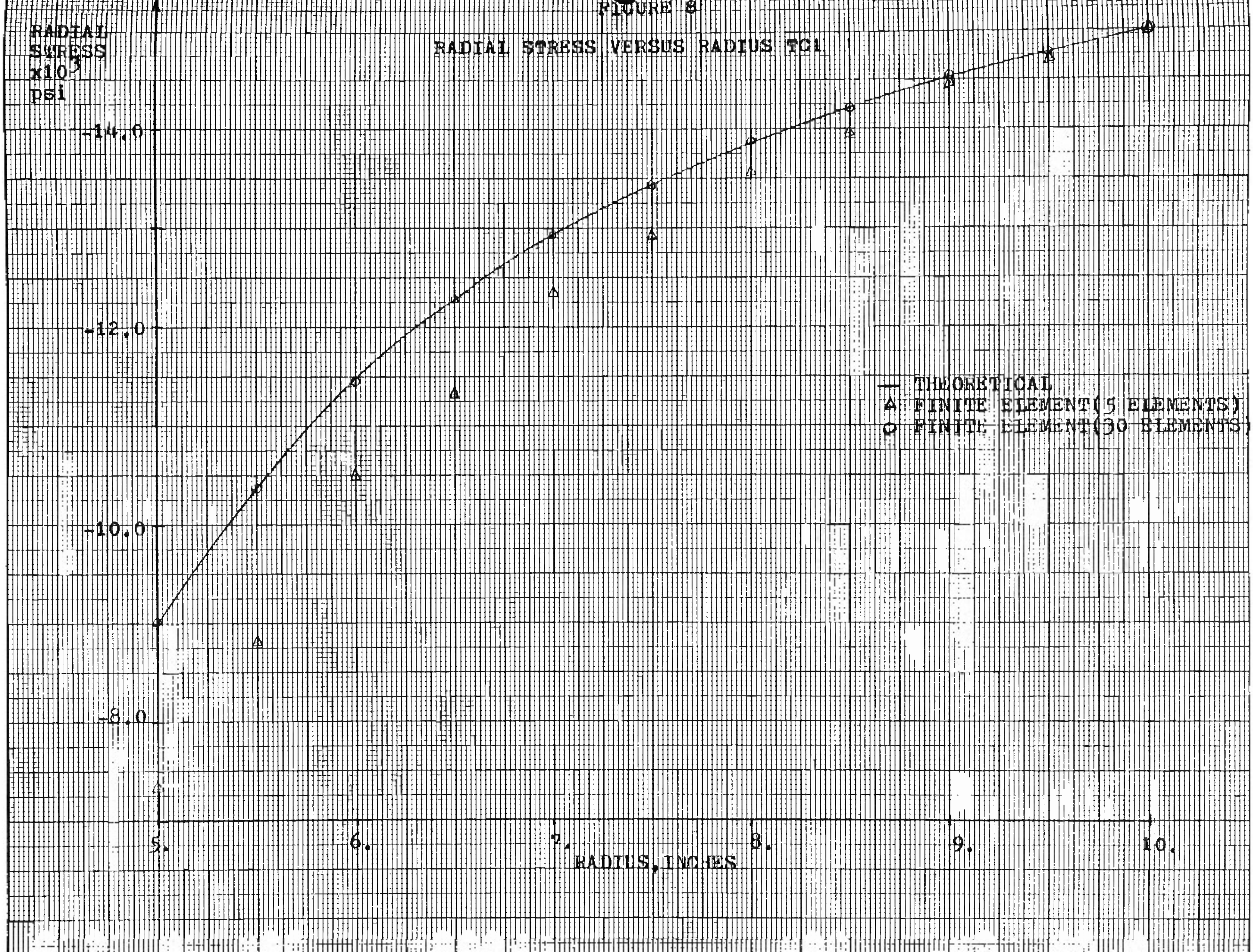


FIGURE 8

RADIAL STRESS VERSUS RADIUS TC1

FIGURE 8

HOOP STRESS VERSUS RADIUS TC1 (30 ELEMENTS)

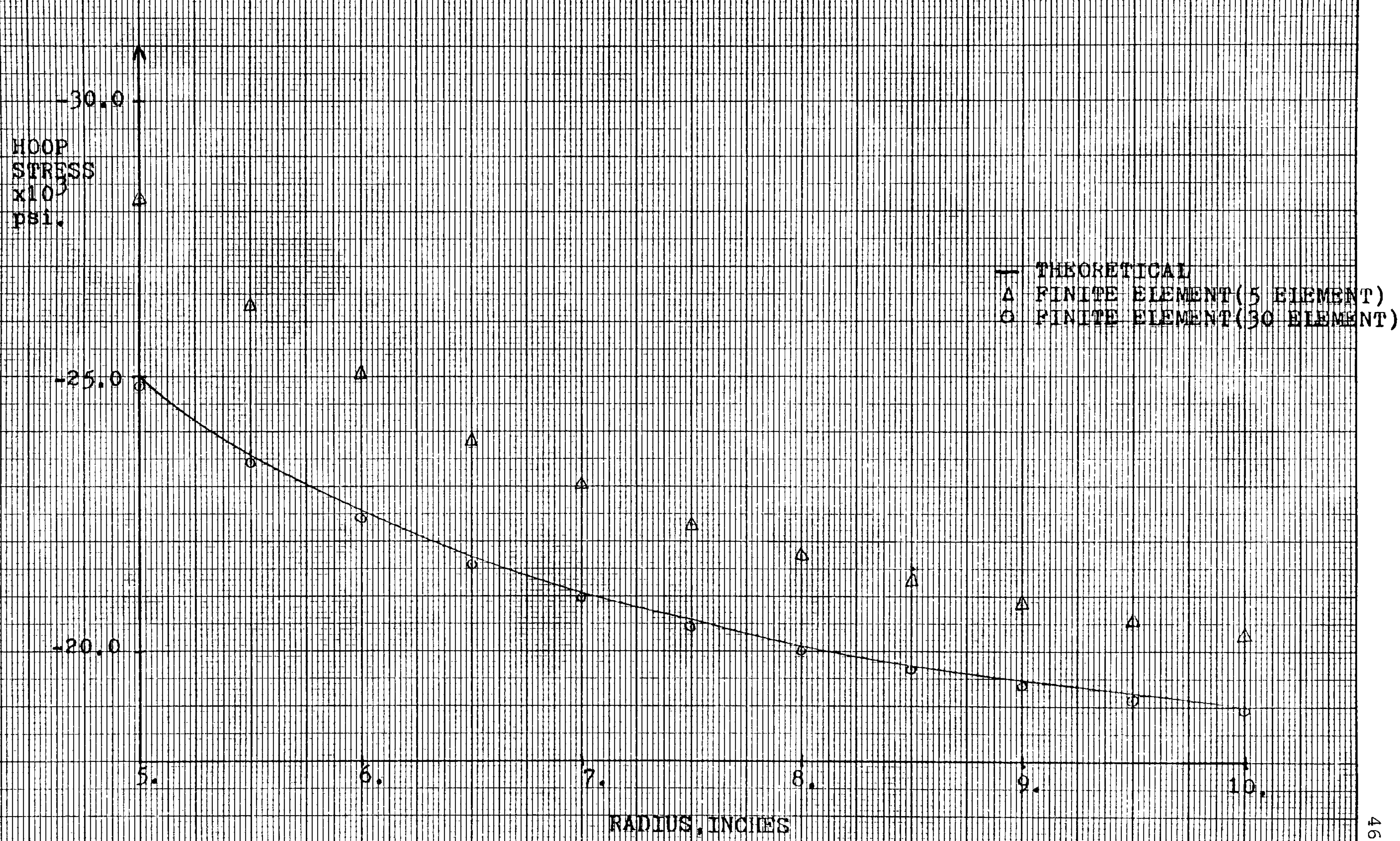


TABLE III

TC 1 SUMMARY OF RESULTS

DEGREES OF FREEDOM	σ_r MAX. psi.	% DIFF.	σ_θ MAX. psi.	% DIFF.	U_r MAX. inx10^{-3}	% DIFF.
Theoretical	-15000.	---	-25000.	---	-4.83	---
14*	-16223.	8.15	-28376.	13.50	-5.25	8.49
54**	-15023.	0.16	-28249.	13.00	-5.26	8.90
248***	-15004.	0.03	-24820.	0.72	-4.81	0.41

* 1 element

** 5 elements

*** 30 elements

TABLE VI

COMPARISON OF RESULTS OF THESIS VERSUS THOSE OF
LINEAR AND QUADRATIC TRIANGULAR ELEMENTS

	DARIO & BRADLEY	DARIO & BRADLEY	THESIS
Element shape:	triangle	triangle	quadrilateral
Displacement function:	linear	quadratic	quadratic
Model information			
Number of nodes:	105	105	110
Number of elements:	160	40	30
Maximum stress differences			
Radial stress:	6.43%	0.46%	0.03%
Hoop stress:	0.7%	0.1%	0.72%

8.2 Stresses in a Uniformly Thick Disk Due to Centrifugal Load TC 2

The second test case is a classic problem in the theory of elasticity and involves the determination of radial and hoop stresses in a circular disk of uniform thickness subjected to centrifugal loading. Problem geometry and loading conditions are shown in Fig. 9a. The finite element model used contained 30 elements and 125 nodes and is shown in Fig. 9b. Theoretical solutions for stresses are presented by Timoshenko [1] and are quadratic in nature.

Results from the finite element idealization are compared with their theoretical values in Fig. 10 and for all practical purposes may be considered exact.

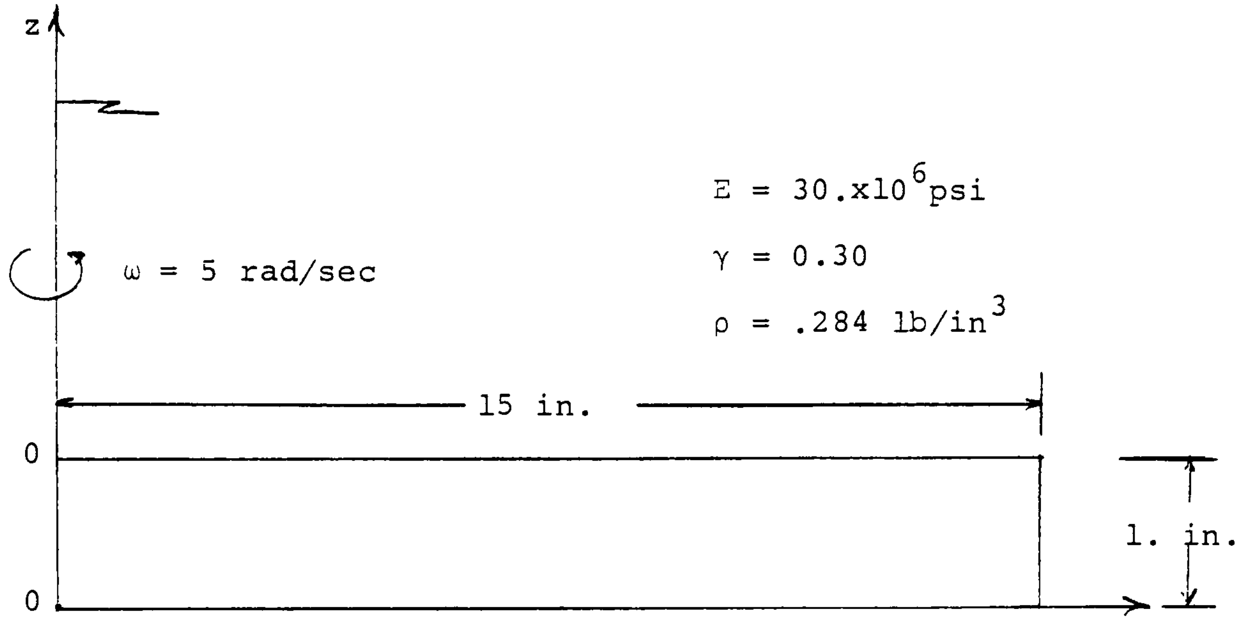
Consideration in this analysis was not only determination of accurate stress values but also the work necessary in specifying the body force loading condition.

Body forces were calculated for each element and specified as external forces acting at the model nodal points, consistent with the allocation scheme shown in Fig. 3.

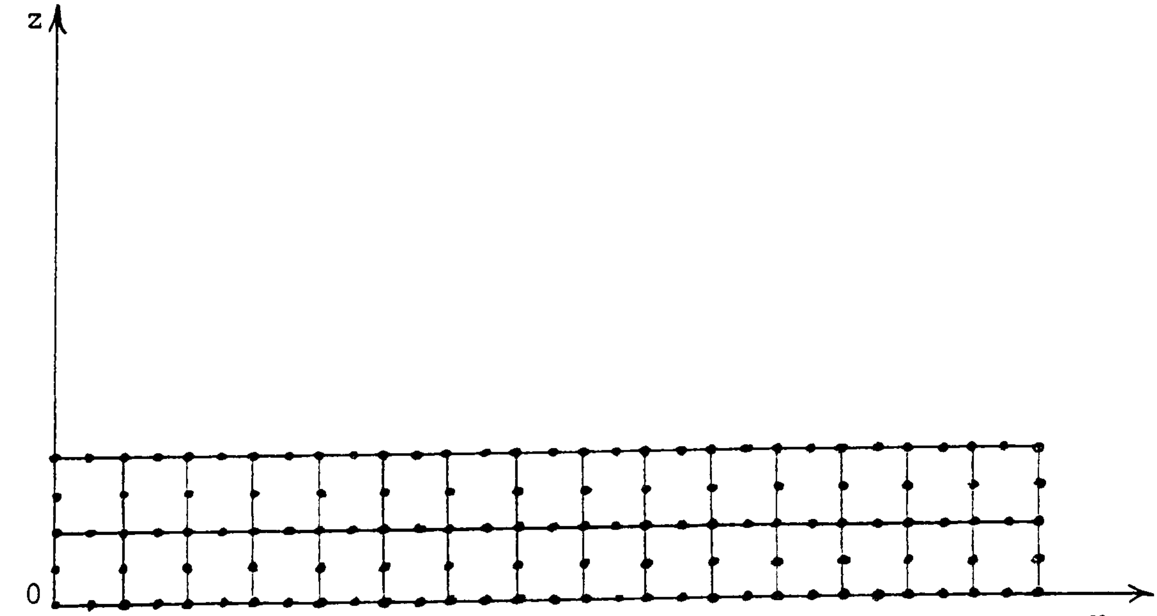
Using the above technique presents severe limitations in representing this type of problem which include:

1. An excessive amount of time to calculate element body forces and distribute them to the nodal points.
2. A necessarily large amount of input data for specification for the external nodal point forces calculated.
3. In the case of elements with curved boundaries, allocation of element body force to its nodes is no longer obvious as in the case presented and requires additional consideration.

All of the above limitations may be alleviated by the introduction of a subroutine in the program to internally calculate and distribute body forces to nodal points on a per element basis. Also, the third limitation cited is greatly reduced by using quadrature techniques. The computer program developed does not contain this option which is left for future development.



a) disk parameters



b) TC 2 finite element idealization

FIGURE 9

Uniformly Thick Disk Subjected

to Centrifugal Loading

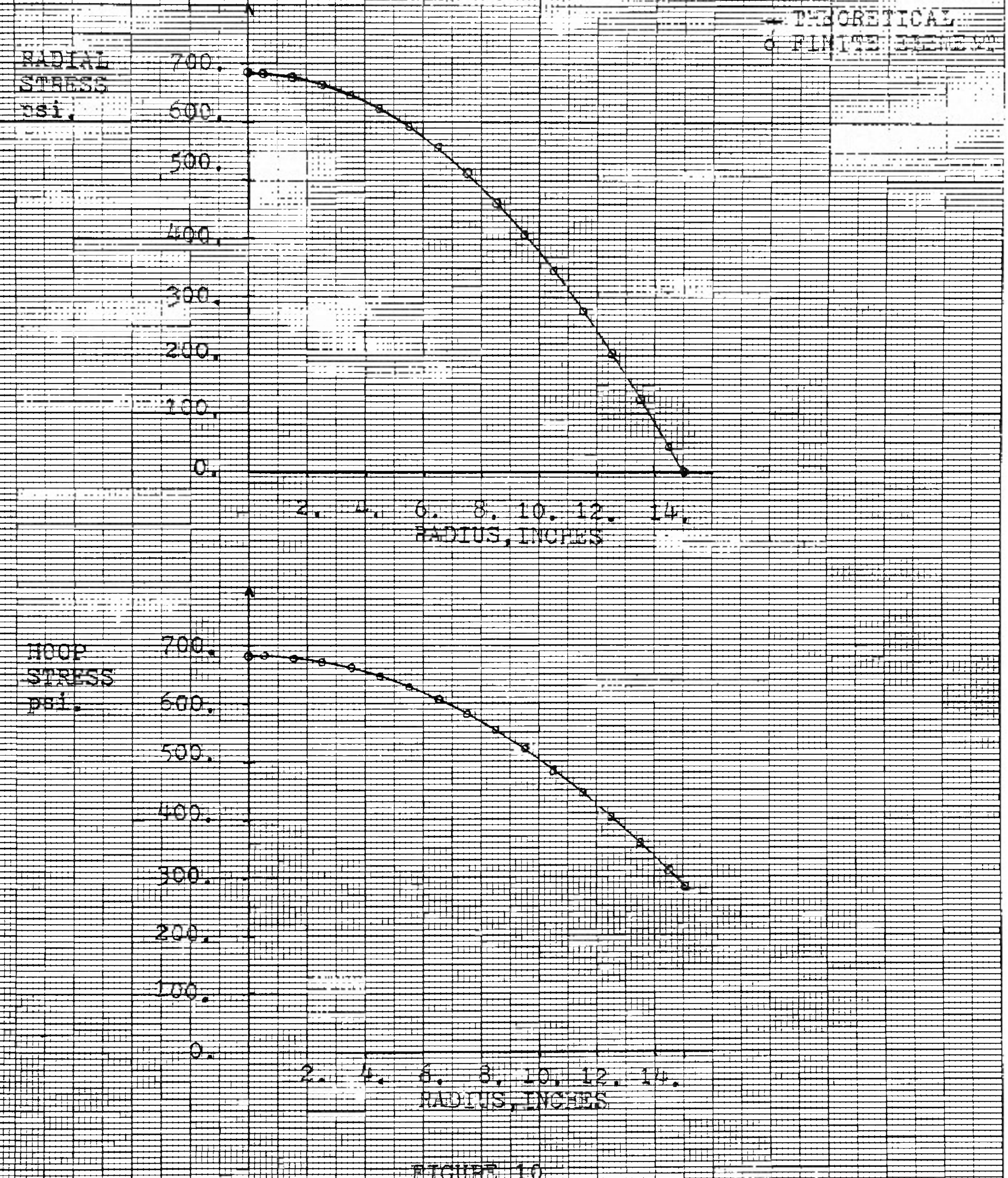


FIGURE 10
RADIAL AND HOOP STRESS RESULTS FOR TC 2

8.3 Stress Concentrations in a Cylindrical Rod Due to a Spherical Inclusion TC 3

The problem of axial stress concentration in a cylindrical rod containing a spherical inclusion was analysed as a test case to demonstrate the program's ability to represent curved boundaries and predict stress concentration values. The rod is subjected to a uniform tensile stress distribution as shown in Fig. 11.

The actual problem follows the notation of Dario and Bradley [21]. A closed form solution is presented by Timoshenko[1].

The finite element model developed, taking into account the symmetry of loading, is presented in Fig. 12. Only three elements are used to represent the inclusion boundary.

A graph comparing finite element to theoretical axial stress in the plane perpendicular to the z axis at z = 0 is presented in Fig. 13. The maximum difference between predicted and theoretical stress values was found to be 1.06%.

In an attempt to obtain further stress information in the localized area of concern, a second model was developed simulating a region consisting of the four elements noted in Fig. 13.

These four elements were divided into the eight elements shown in Fig. 14. Nodal points corresponding to nodes of the original model are circled. New model boundary conditions were specified as enforced displacements at the circled nodes obtained in the initial idealization.

The results for axial stress (σ_z). in the plane z = 0 for this model produced no correlation with that previously obtained. However, stresses at element midside nodes just away from the boundary (z = .166in.) did exhibit convergence and are shown in Fig. 13. The reason for boundary discrepancies is believed to be due to the introduction of additional nodes on the refinement's boundaries. It is felt that these additional nodes whose displacements are not prescribed result in deformation of the idealization's boundaries which are incompatable with the deformations of the original model. Possible techniques to overcome these discrepancies are:

1. Use element displacement functions (Eqs. 2 and 3) to determine prescribed displacements for all nodes of the refined model (Fig. 14.). This would assure displacement compatability between both models.

2. Determine stress element boundary values directly for each element of the original model using the relation:

$$\{\sigma\} = [D][B]\{W_o\}$$

Both of these techniques would require the development of an auxiliary program. The second technique appears to be more efficient since it would not require the formulation of additional structural models. Development of a program using the second technique cited has been initiated but is as yet unfinished. At present, discrepancies in boundary stresses of refined models are unresolved.

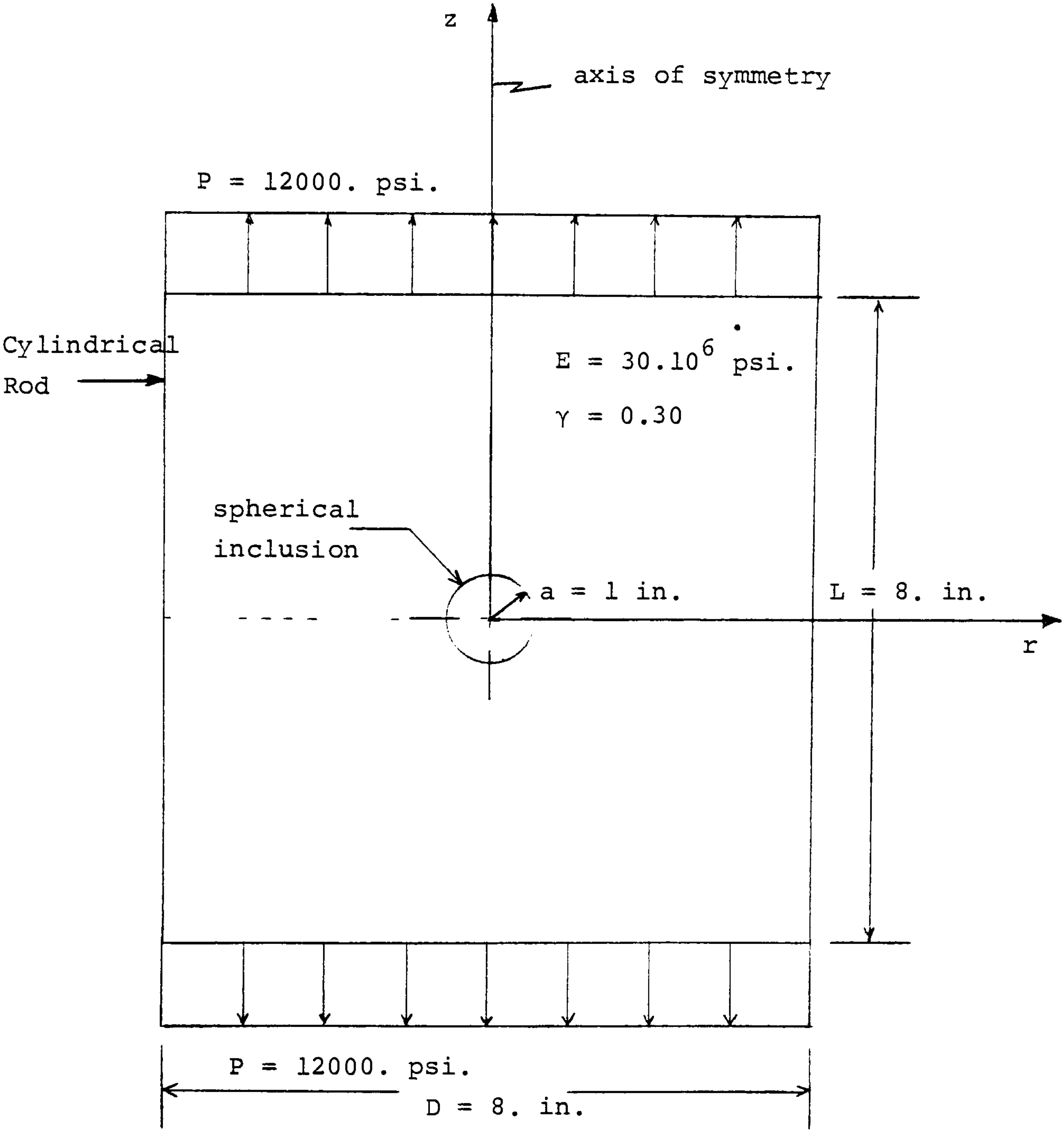


FIGURE 11

Cylindrical Rod Having a Spherical Inclusion

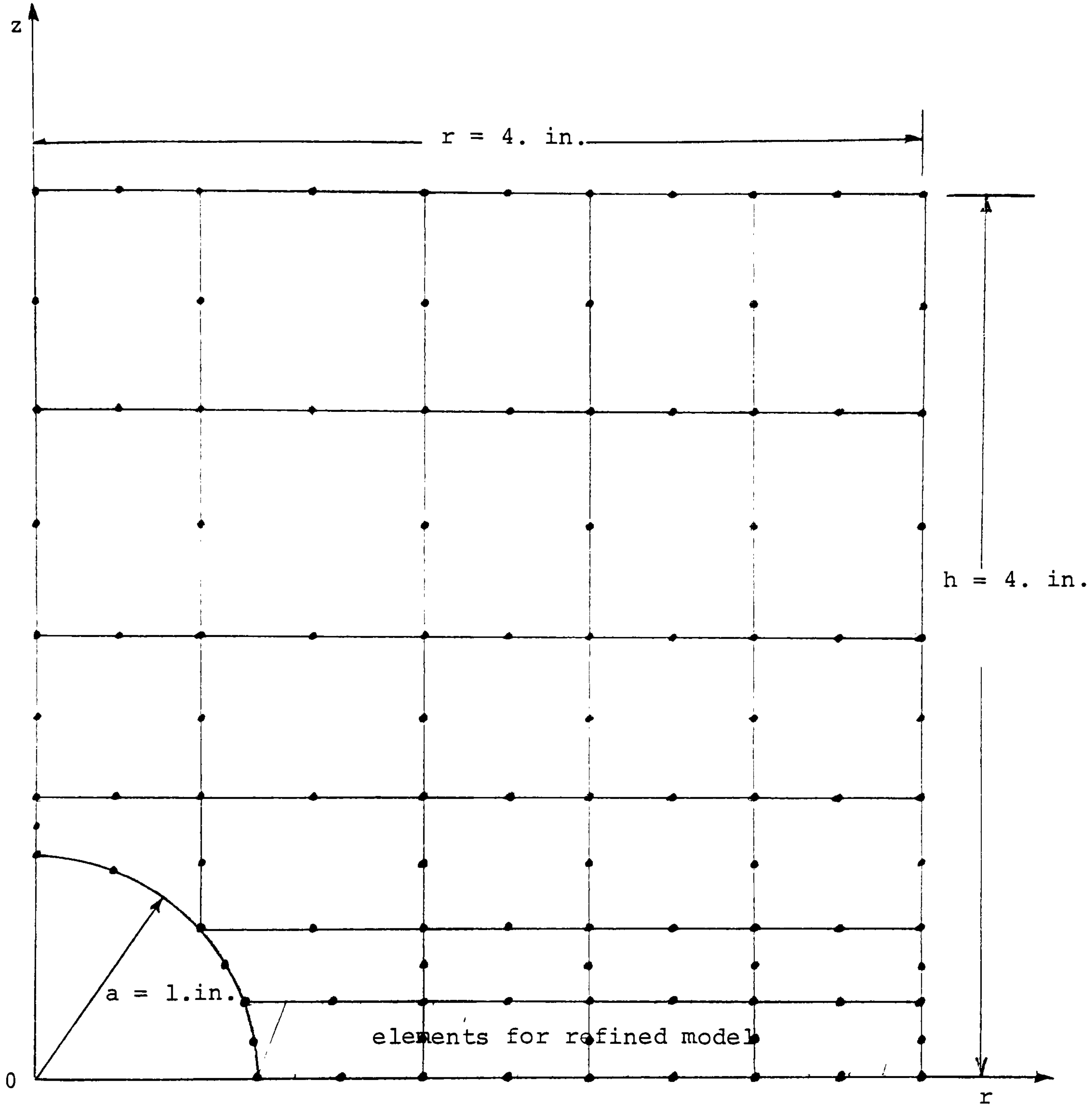
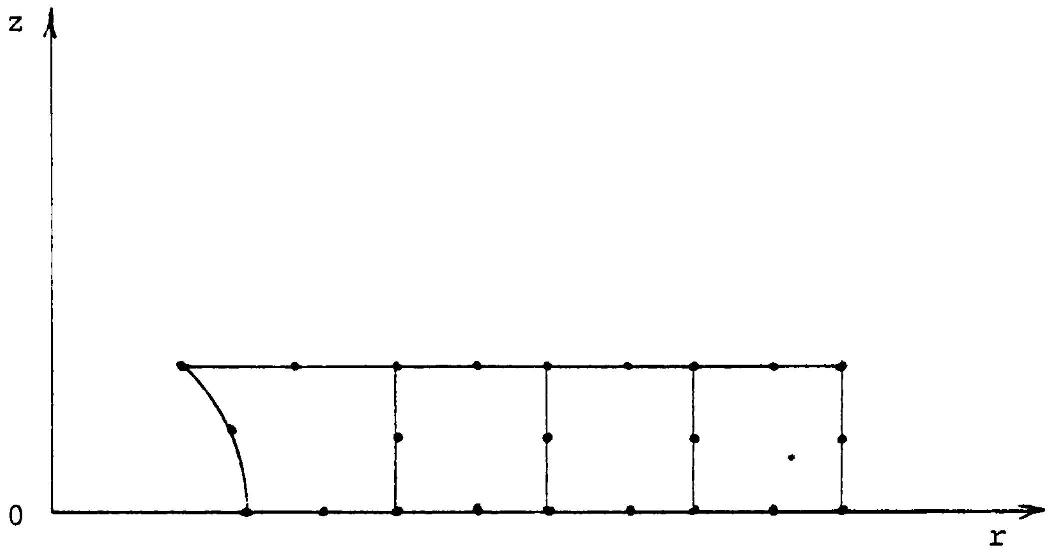


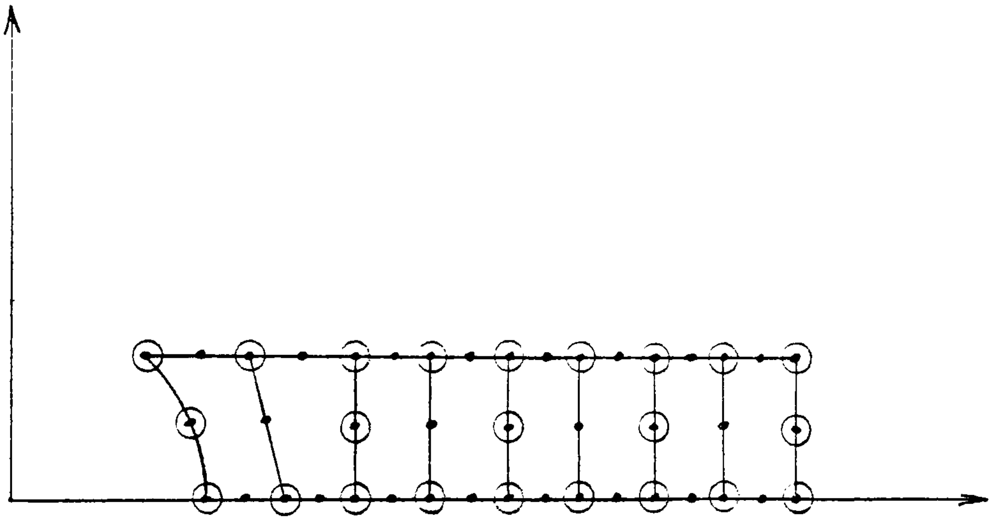
FIGURE 12

TC 3 Finite Element Idealization of

Spherical Inclusion in Cylindrical Rod



a) four elements from original model



b) refined idealization of elements in a)

FIGURE 14

TC 3 Refined Idealization

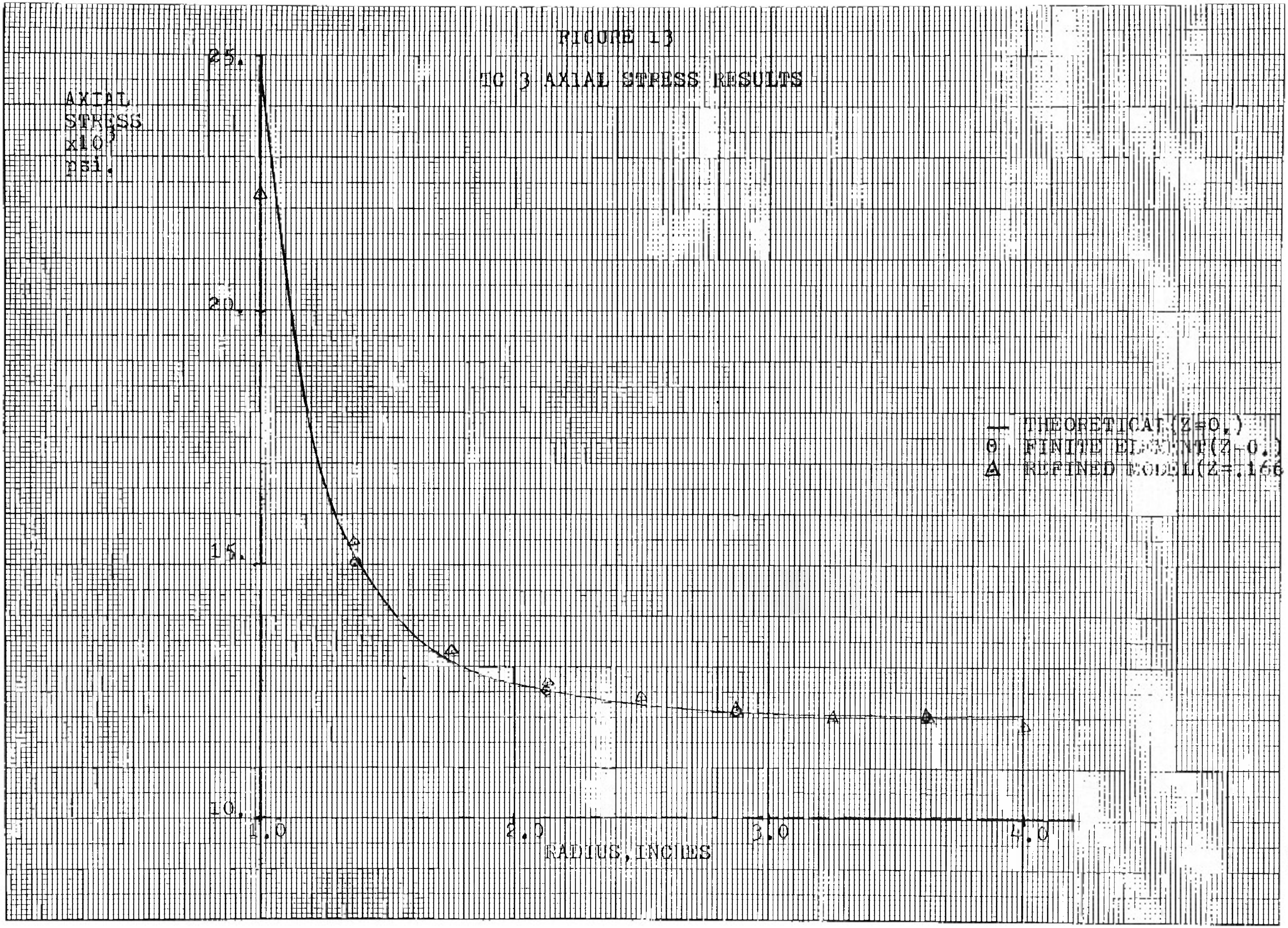


FIGURE 13

TC 3 AXIAL STRESS RESULTS

8.4 Stresses and Deflections in a Spherical Pressure Vessel TC 4

The fourth test case presented involves the determination of principal stresses and volumetric expansion of a thick spherical pressure vessel subjected to an internal pressure as shown in Fig. 15. Theoretical solutions for stress and displacement contain cubic and quartic functions of radius respectively. Of particular interest in this test case is the element's ability to represent the curved spherical surface.

Due to symmetry only half of the sphere was necessary in describing a finite element model. Difficulties with principal stress predictions resulted in the formulation of the four finite element models shown in Fig. 16. In all four cases the volumetric expansions obtained showed good correlation with theoretical results. Comparisons of the theoretical maximum displacement with the results from the four test cases is presented in Table IV. A graph showing theoretical, TC 4A, and TC 4D radial displacement as a function of radius is presented in Fig. 17.

The order of the displacement function for the quadratic element results in linear intraelement stress variation. In the case where actual stress is of higher than linear order, stresses computed for course finite element models will exhibit finite discontinuities at midside nodes of adjacent elements. This as pointed out by Desai and Abel[17], is due to the absence of force equilibrium in individual elements. Involving structural force equilibrium relations, the overall equilibrium of the body is approximated but not that of individual elements. Increased finite element refinement minimizes this effect.

TABLE IV

TC 4 SUMMARY OF MAXIMUM DISPLACEMENT RESULTS

MODEL	DEGREES OF FREEDOM	MAXIMUM DISPLACEMENT in. x 10^{-6}	PERCENT DIFFERENCE
Theoretical	closed form solution	22.22	--
TC 4A	64	18.82	15.3
TC 4B	88	20.20	9.09
TC 4C	112	20.64	7.11
TC 4D	224	21.36	3.87

It was found that only the finest mesh (TC 4D) predicted stress values that were at all close to theoretical values. Graphs comparing the theoretical principal hoop and radial stresses and the interelement linear variations of stress for TC 4D are shown in Figs. 18 and 19.

As can be seen from these graphs, large discontinuities in stress between the first two adjacent elements through the thickness of the sphere are predicted. These stress values are quite unreliable. Both the large discontinuities and the gradient of the theoretical curves suggest that a more refined finite element simulation is required in this region to improve stress results. Also, the extrapolation technique for stresses proposed in section 8.3 might improve these values.

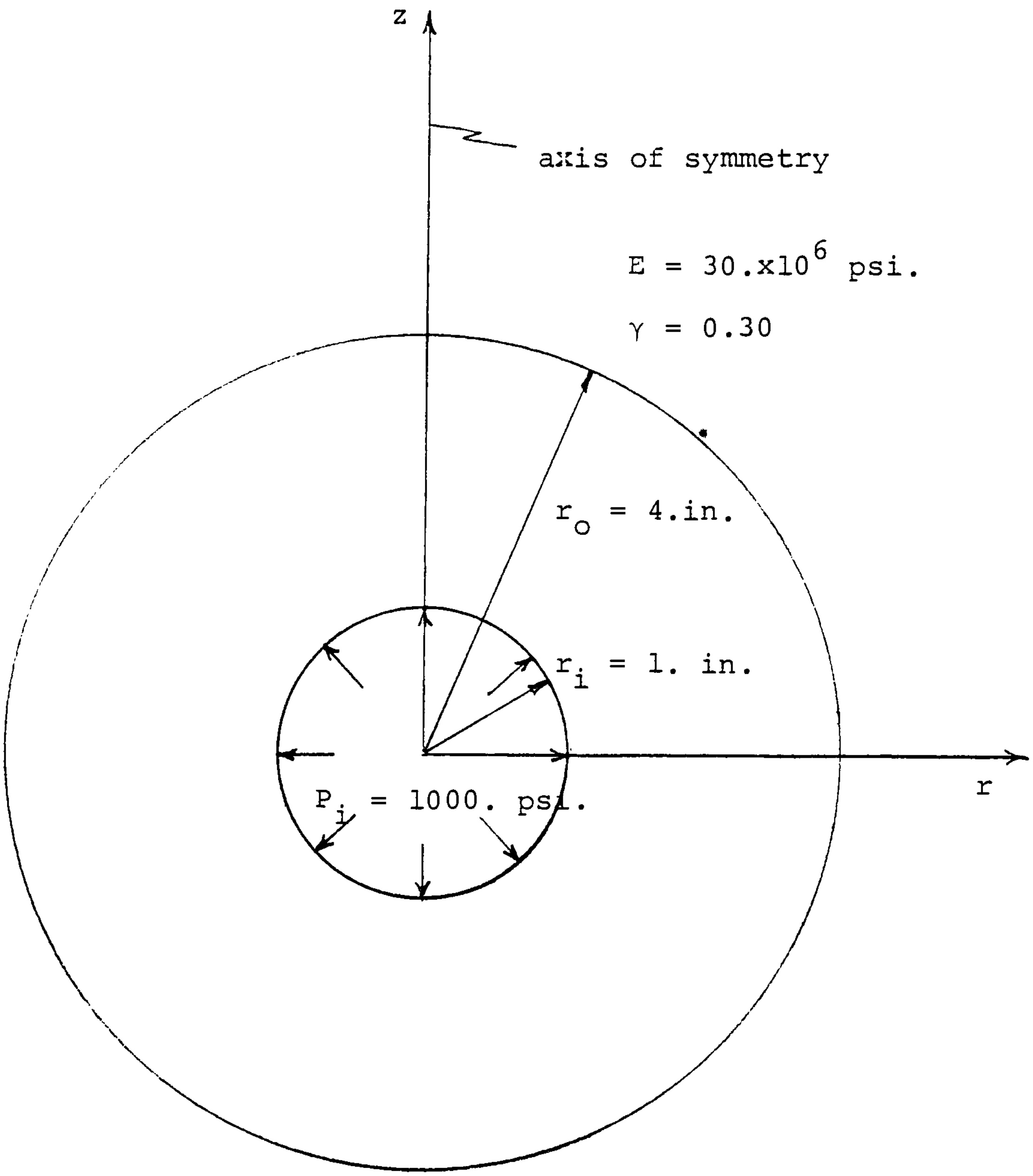
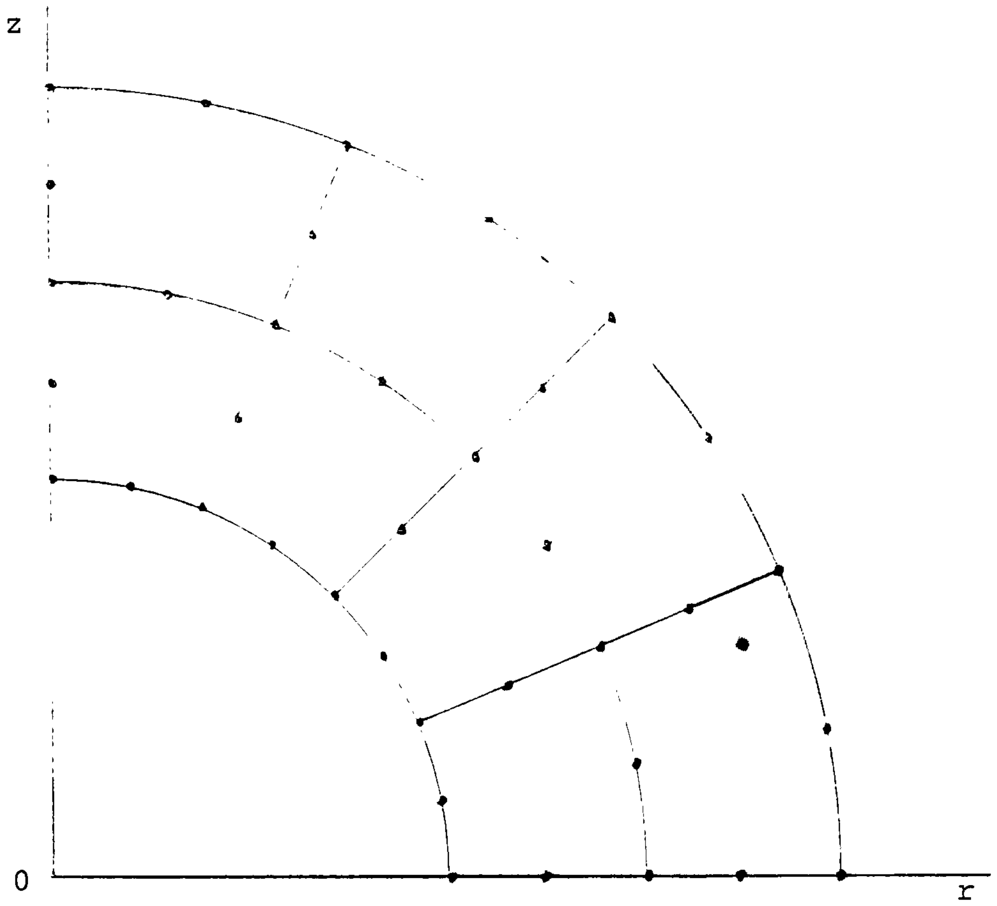


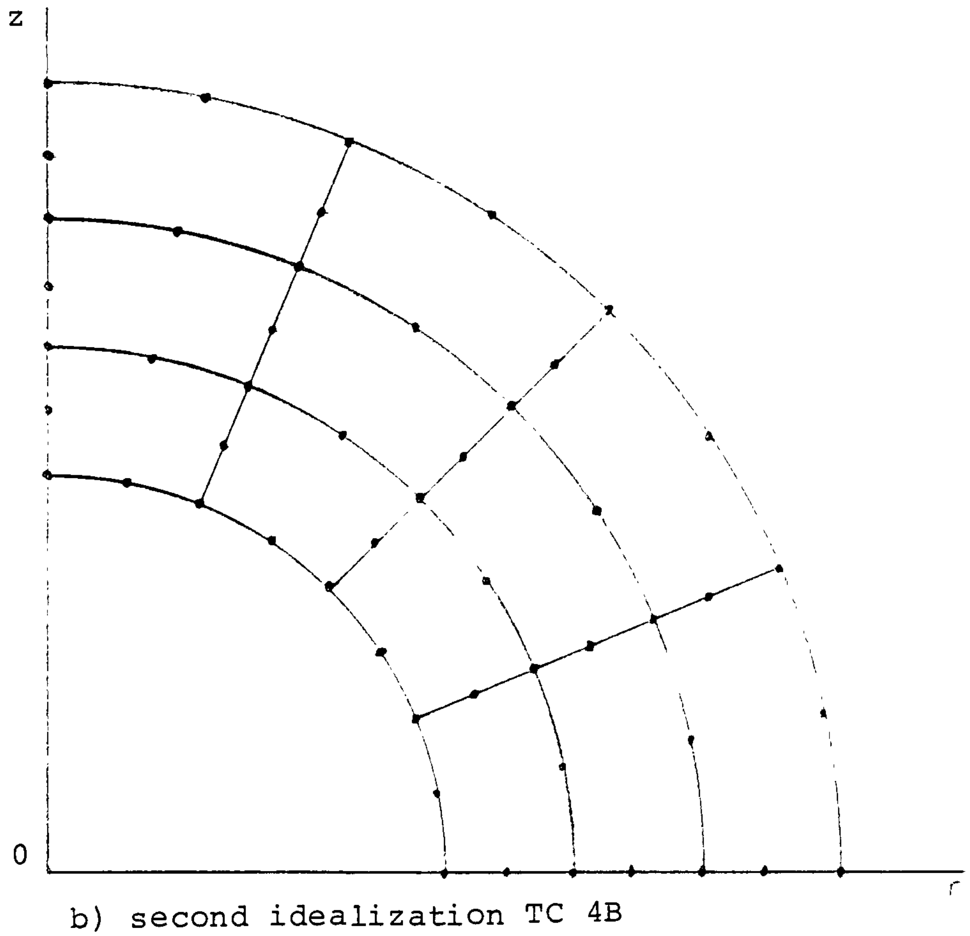
FIGURE 15

Spherical Pressure Vessel

Subjected to Internal Pressure

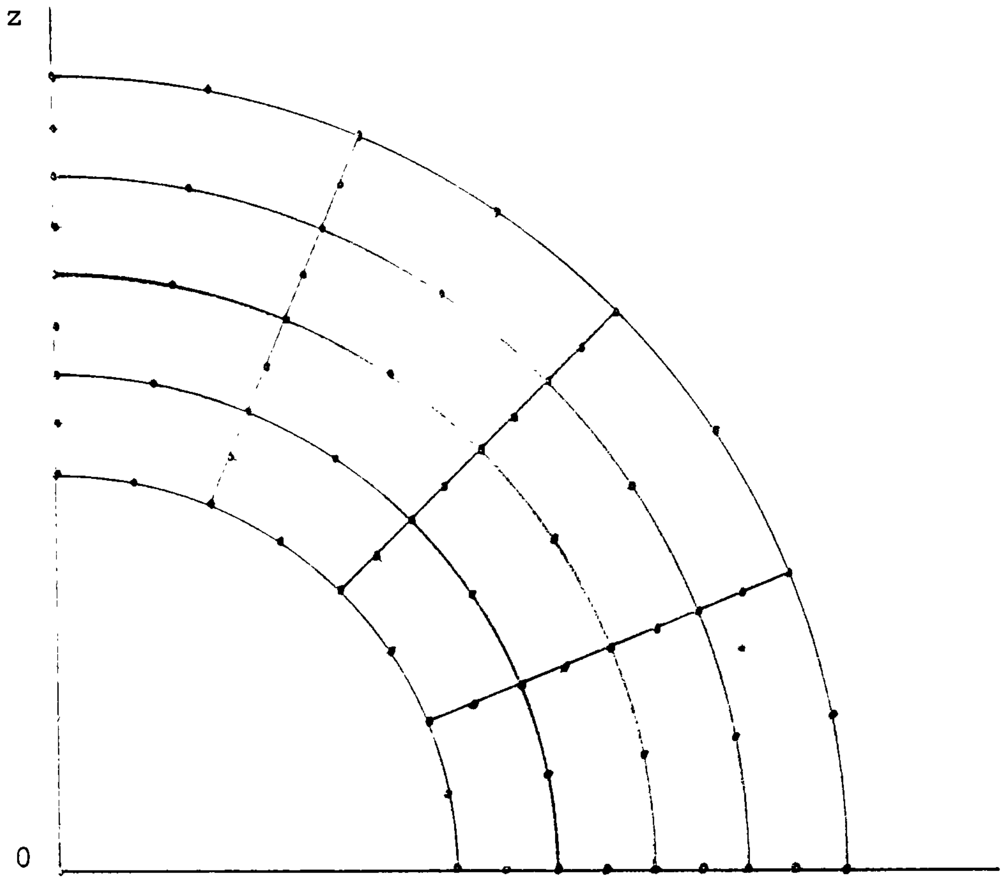


a) first finite element idealization TC 4_A

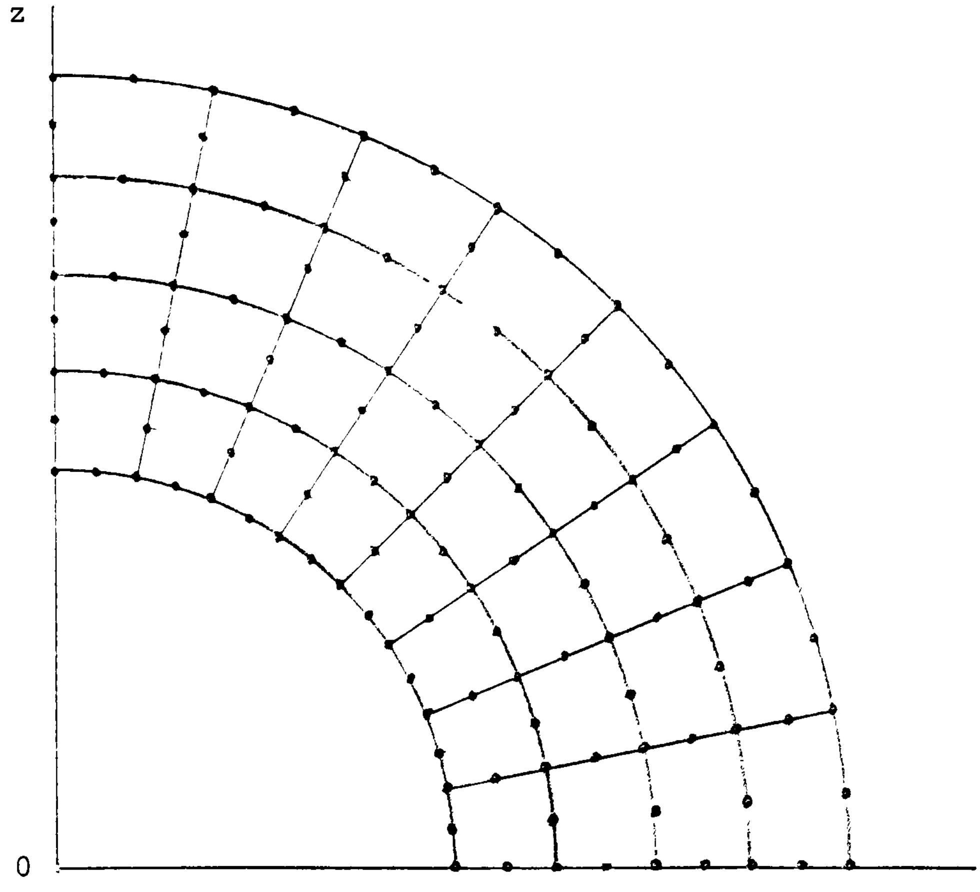


b) second idealization TC 4B

FIGURE 16
Finite Element Idealizations of TC4



c) third idealization TC 4C



d) fourth idealization TC 4D

FIGURE 16 (Continued)

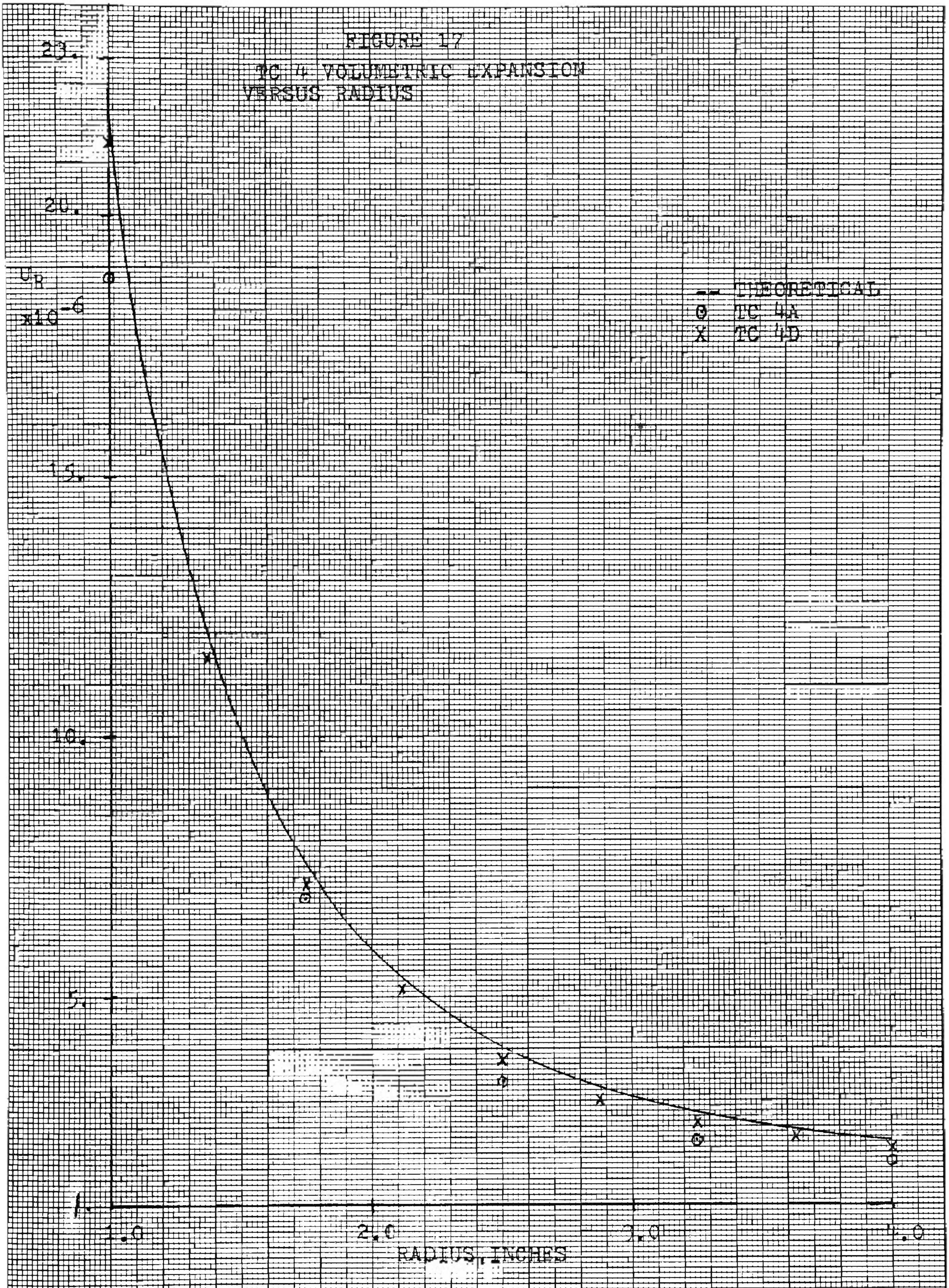


FIGURE 17
TC 4 VOLUMETRIC EXPANSION
VERSUS RADIUS
-- THEORETICAL
O TC 4A
X TC 4D
U_R
$x10^{-6}$
23.
20.
15.
10.
5.
1.
1.0
2.0
3.0
4.0
RADIUS, INCHES

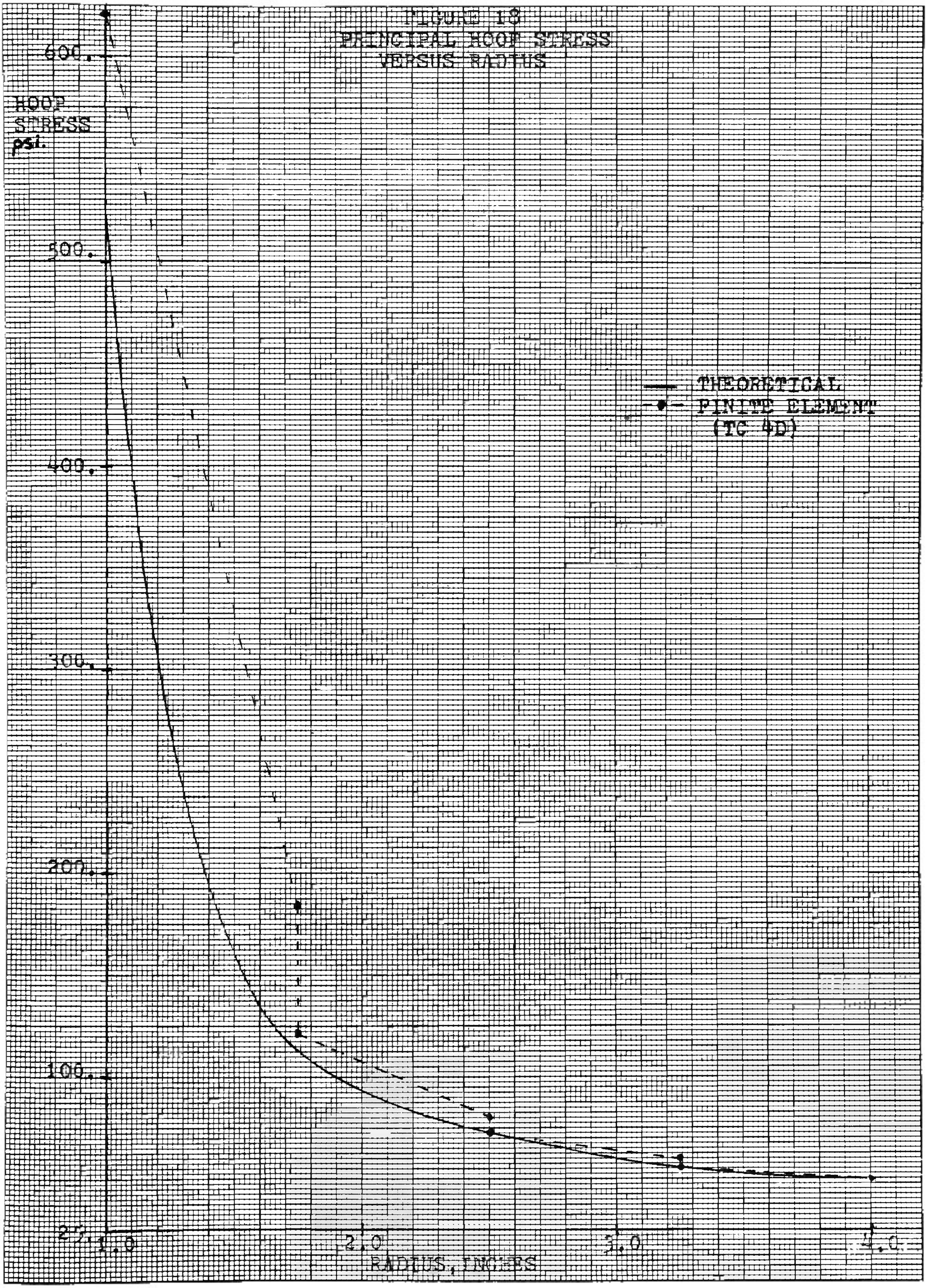


FIGURE 18
PRINCIPAL HOOP STRESS
VERSUS RADIUS
HOOP
STRESS
psi.
600.
500.
400.
300.
200.
100.
25.
1.0
2.0
3.0
4.0
RADIUS, INCHES
THEORETICAL
FINITE ELEMENT
(TC 4D)

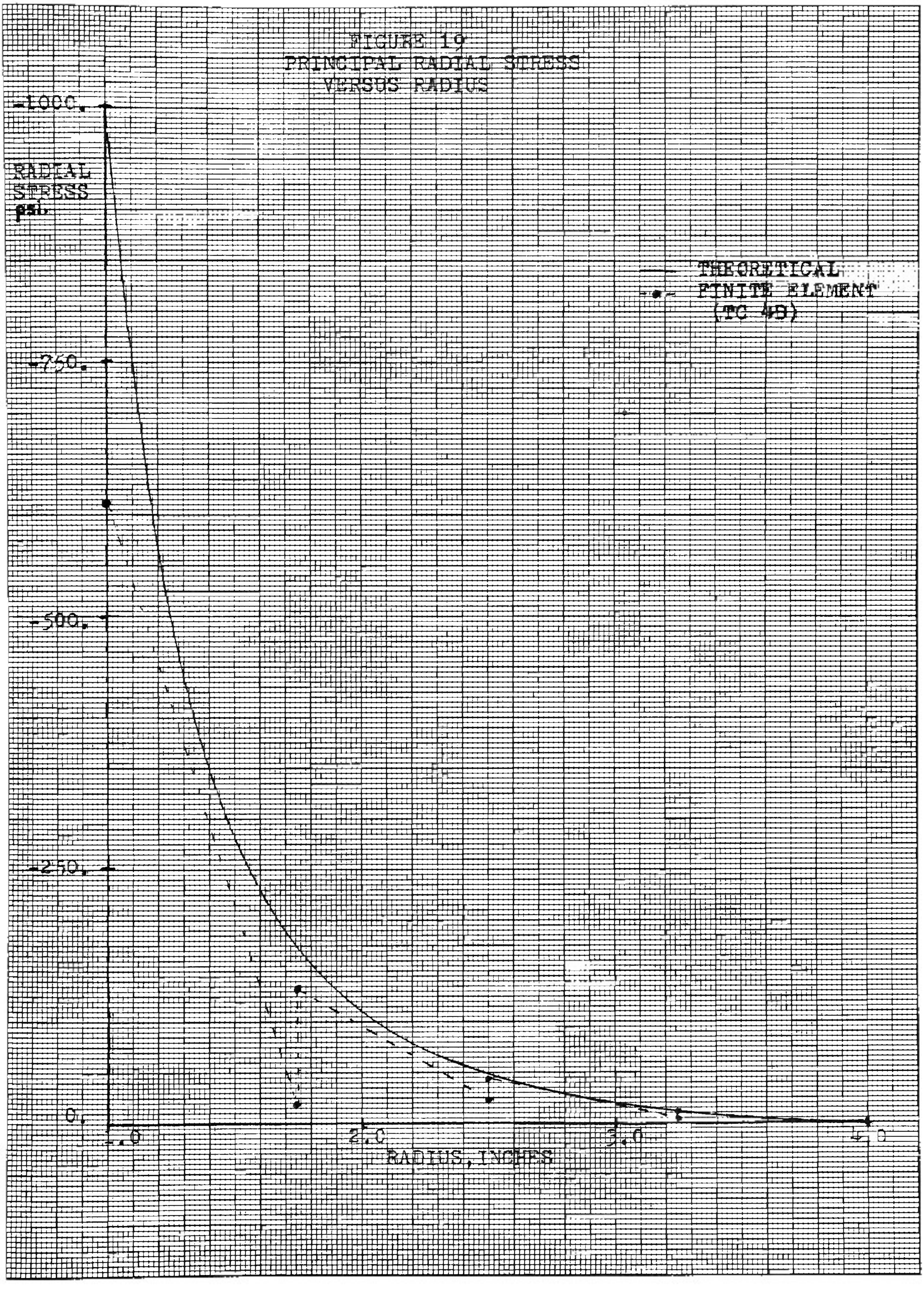


FIGURE 19
PRINCIPAL RADIAL STRESS
VERSUS RADIUS
-1000.
RADIAL
STRESS
psi.
THEORETICAL
FINITE ELEMENT
(TC 4B)
-750.
-500.
-250.
0.
1.0
2.0
3.0
4.0
RADIUS, INCHES

8.5 Circular Plate Bending-Investigation TC 5

The objective of this investigation was to determine this quadratic element's ability to predict displacements and stresses in structures obeying small displacement plate theory. This theory involves approximations in order that a linear differential equation of equilibrium is obtained. The criteria which a structure must meet to qualify as a plate obeying small displacement theory are stated by Timoshenko and Woinowski-Krieger [28] as:

1. There is no in-plane deformation of the middle plane of the plate which remains neutral during bending.
2. Lines initially normal to the middle plane of the plate experience linear variation of stress and strain.
3. Normal stresses in the direction transverse to the plate may be disregarded.

These criteria are satisfied provided transverse displacements are small in comparison with plate thickness and plate thickness is much smaller than radius.

The particular problem chosen to analyse was that of a circular plate clamped along its outer radius and loaded with a uniform pressure normal to its surface. Plate geometry and boundary conditions are shown in Fig. 20a.

A finite element idealization of this problem was developed for the case of a load intensity P_O = 10 psi. (Fig. 20b.) Structural displacement results were compared with the theoretical solution presented by Timoshenko and Woinowski-Krieger [28] and were found to be of unreasonable form and magnitude. This lack of correlation was discussed in detail with several knowledgable individuals in the field of finite element analysis [25], [26], [27], [33]. These discussions and a survey of available literature resulted in identification of several areas as the potential sources of discrepancy. These areas and comments on their subsequent investigations are:

Potential Sources of Discrepancy

1. Errors in element development or computer programming.
2. Errors in stiffness calculations due to the singularity in hoop strain (e_θ) for elements lying on the axis of symmetry.
3. Inappropriate structural idealization.
4. Incorrect specification of structure boundary conditions.
5. Violation of plate theory assumptions.

Comments

1.a) Investigations of element development and computer program by McCalley [26], Rieger[33], and the author did not identify any errors.

b) At the suggestion of McCalley, the eigenvalues and eigenvectors of a single element's stiffness matrix were calculated to verify element stiffness formulation. All principal stiffness values were found to be positive and the fundamental eigenvector was found to correspond to a rigid body axial translation. Both of these findings were consistent with a correctly formulated stiffness matrix.

c) It was established for a one element problem that structural force equilibrium was maintained.

2. The singularity in the hoop strain expression ($e_\theta = \frac{u}{r}$) will not provide error in stiffness formulation.

As noted by Ergatoudis [8], these expressions are evaluated at Gauss sampling points when stiffness matrices are evaluated numerically and these sampling points will not generally lie on element boundaries where r = 0. Also, results obtained in TC 2, TC 3, and TC 4 where elements were defined having an edge on the axis of symmetry did not exhibit similar difficulties.

3.a) The use of one element through plate thickness is justified by the second assumption of plate theory that lines initially normal to the middle plane of the plate experience linear variation of stress and strain. Since element displacement is quadratic, transverse stress and strain may vary linearly in the element. This fact is discussed by Griffin [30] for the case of beams in bending and also that a large number of elements are necessary along the length of a beam to account for curvature of axial fibers. Similar reasoning applies to the case of circular plates. However, increasing element refinement to 60 elements through the radius produced no appreciable difference in displacements.

b) At the suggestion of Glasser [27], solutions were obtained for models having four elements through the plate thickness. Due to limitations of computer core, a maximum of 16 elements along the radial direction could be specified. Resulting elements had aspect ratios of radial length/thickness of 10 and predicted unreasonable displacements. These results were inconclusive.

4. A total of 30 computer runs were made having minor modifications in specified boundary conditions. Alterations of plate geometry, force distribution, and displacement constraints did not produce appreciable changes in predicted results.
5. The possibility of violating the plate theory assumption that the middle plane of a plate remains neutral in bending was suggested by Rieger [33]. By reducing the load intensity P_O in Fig. 20a, a significant improvement was obtained in deflection results.

Based on these observations, it was concluded that one discrepancy which existed was due to violation of the assumptions of small displacement plate theory. It was also decided that the structural idealization shown in Fig. 20b was appropriate. The load intensity was changed to 1 psi (Fig. 20a) to reduce deflection magnitudes.

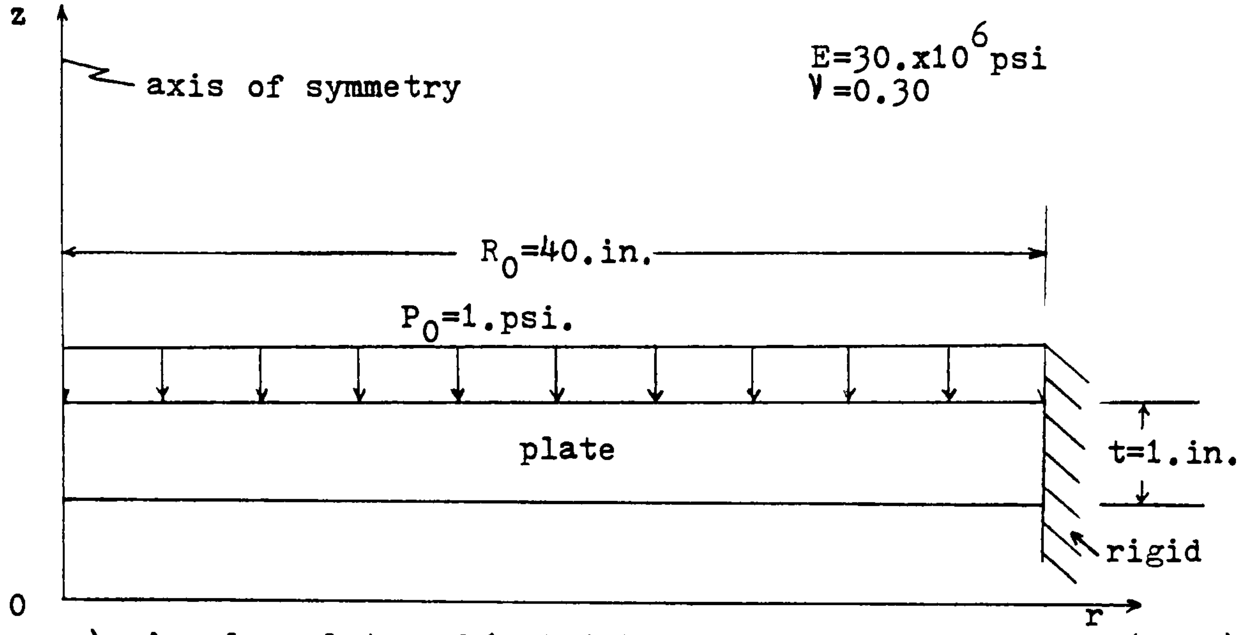
Displacement results were obtained for three finite element models having 20, 30, and 40 elements through plate radius and 1 element through its thickness. Computer calculations were performed in single precision arithmetic. A comparison of predicted and theoretical displacement results is presented in Fig. 21. Predicted displacement shapes were reasonable but their magnitudes did not exhibit lower bound convergence to theoretical values with model refinement.

These observations indicated additional error in either computer program or finite element idealization. In depth discussions with Halbleib [35] vindicated the finite element idealizations representing plate theory. Verification of a quadratic element's ability to represent flexural problems and the eventual determination of the source of error in the thesis program was made possible with the help of Loeber [25].

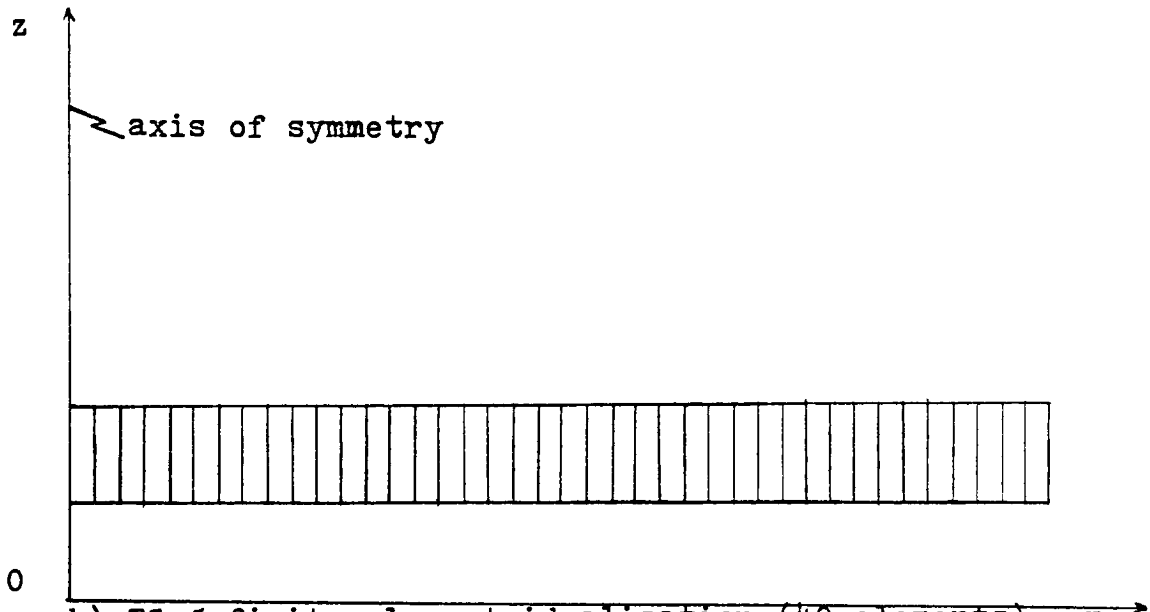
It was learned that a quadratic element similar to that developed was in use at the Knolls Atomic Power Laboratory (KAPL). In collaboration with Loeber, 20, 30, and 40 element idealizations similar to those run by the author were executed at KAPL. In all cases, displacement results were found to agree within 1% of theoretical values. Subsequent discussion with Loeber identified the major discrepancy between the thesis and KAPL programs as being the arithmetic precision of the computers involved. The Xerox Sigma 6 computer available to the author uses a 32 bit word in single precision arithmetic calculations while the CDC 7600 computer at KAPL uses a 60 bit word in single precison. It was learned that this leads to retention of 5 - 6 significant figures on the Sigma 6 as opposed to 14 - 15 on the CDC 7600. The reason that this lack of significant figures should have such a pronounced effect on a plate or shell type problem as opposed to the other problems presented is suggested by Zienkiewicz [10]. Zienkiewicz states that if a plate or shell's thickness becomes small, strains normal to its middle surface are associated with very large stiffness coefficients and roundoff problems will be encountered. In the previous example problems, structure geometry did not lead to this fact.

Based on these facts it was decided that the thesis program should be run using double precision calculations which would provide 13 - 14 significant figures. However, limitations of computer core available to the author did not make this possible. Arrangements were made to make 1 computer run of the 40 element model on a Univac 1108 computer using double precison (72 bit word). Maximum displacement results for this model agreed with those predicted by the KAPL program and varied .25% from theory. Predicted displacements for this run are presented in Fig. 21. Comparisons of radial and hoop stresses on the plate surface with theory are shown in Figs.

22 and 23 respectively and are within 4%. The ability of this quadratic element to analyse flexural problems has been demonstrated. Furthermore, the necessity of using double precision numerical calculations and obeying all assumptions of plate theory has been identified.



a) circular plate subjected to a uniform pressure load(TC 5)

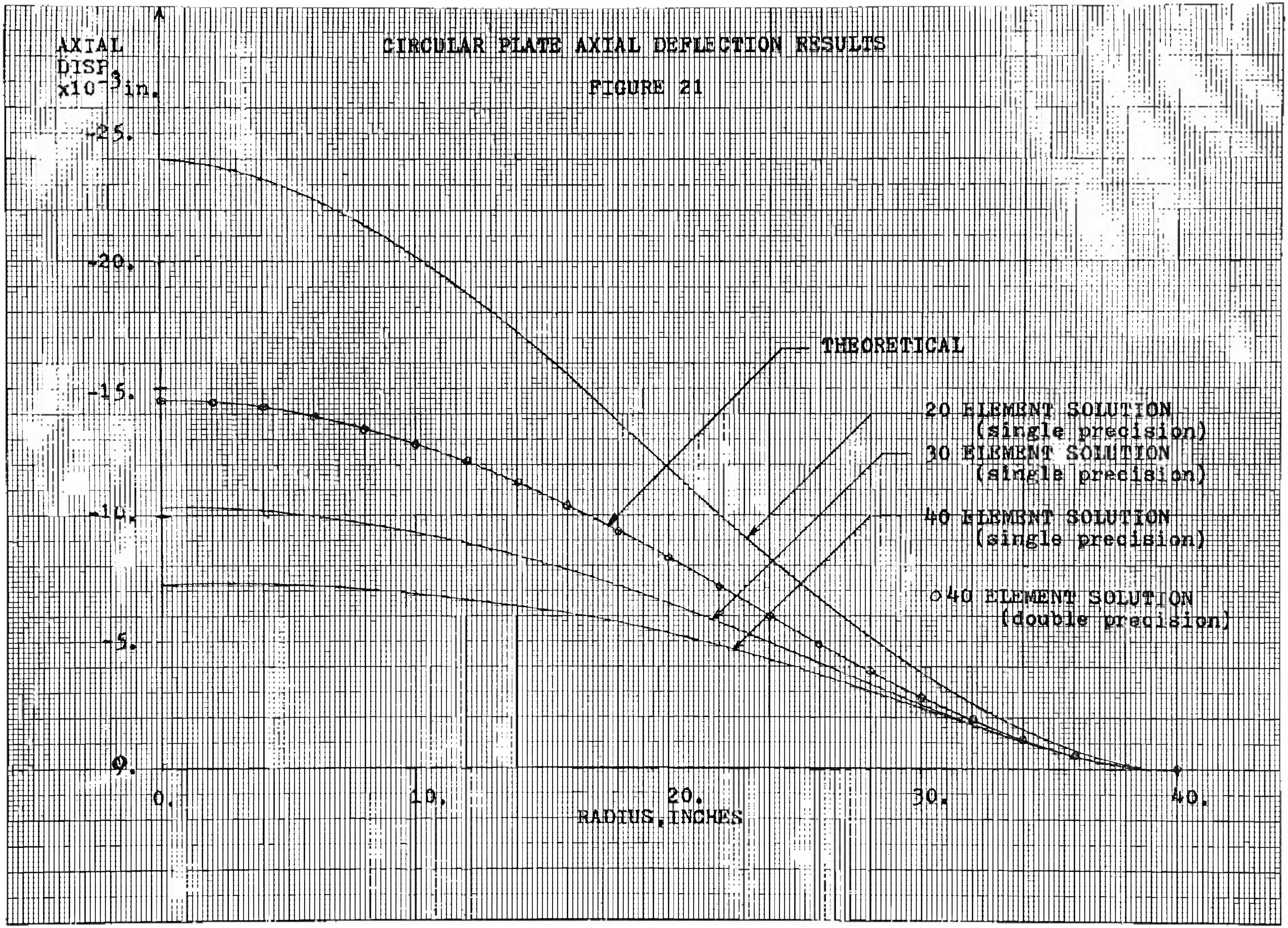


b) TC 5 finite element idealization (40 elements)

Circular Plate Subjected to Uniform Pressure

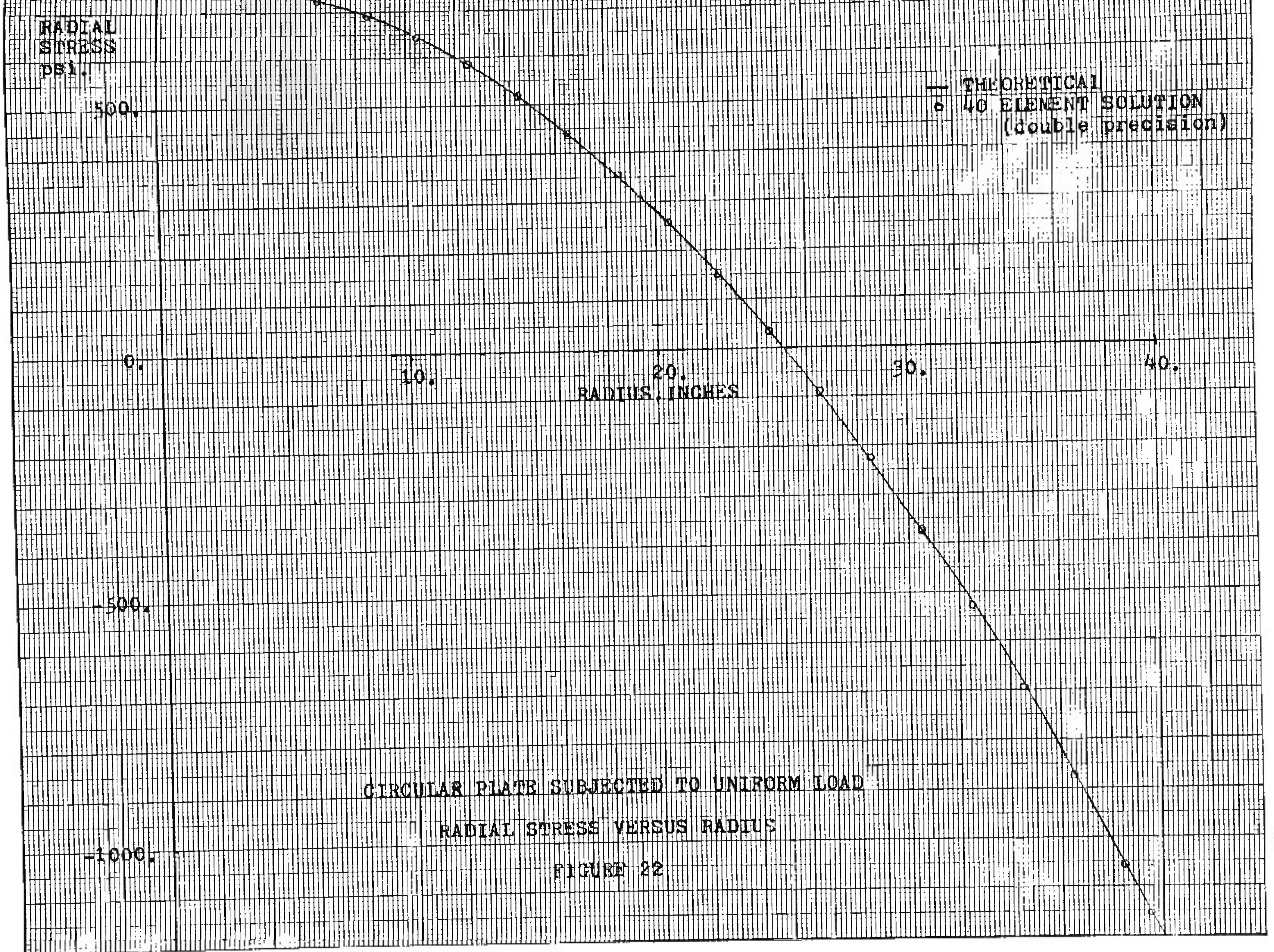
Problem and Idealization

FIGURE 20



CIRCULAR PLATE AXIAL DEFLECTION RESULTS

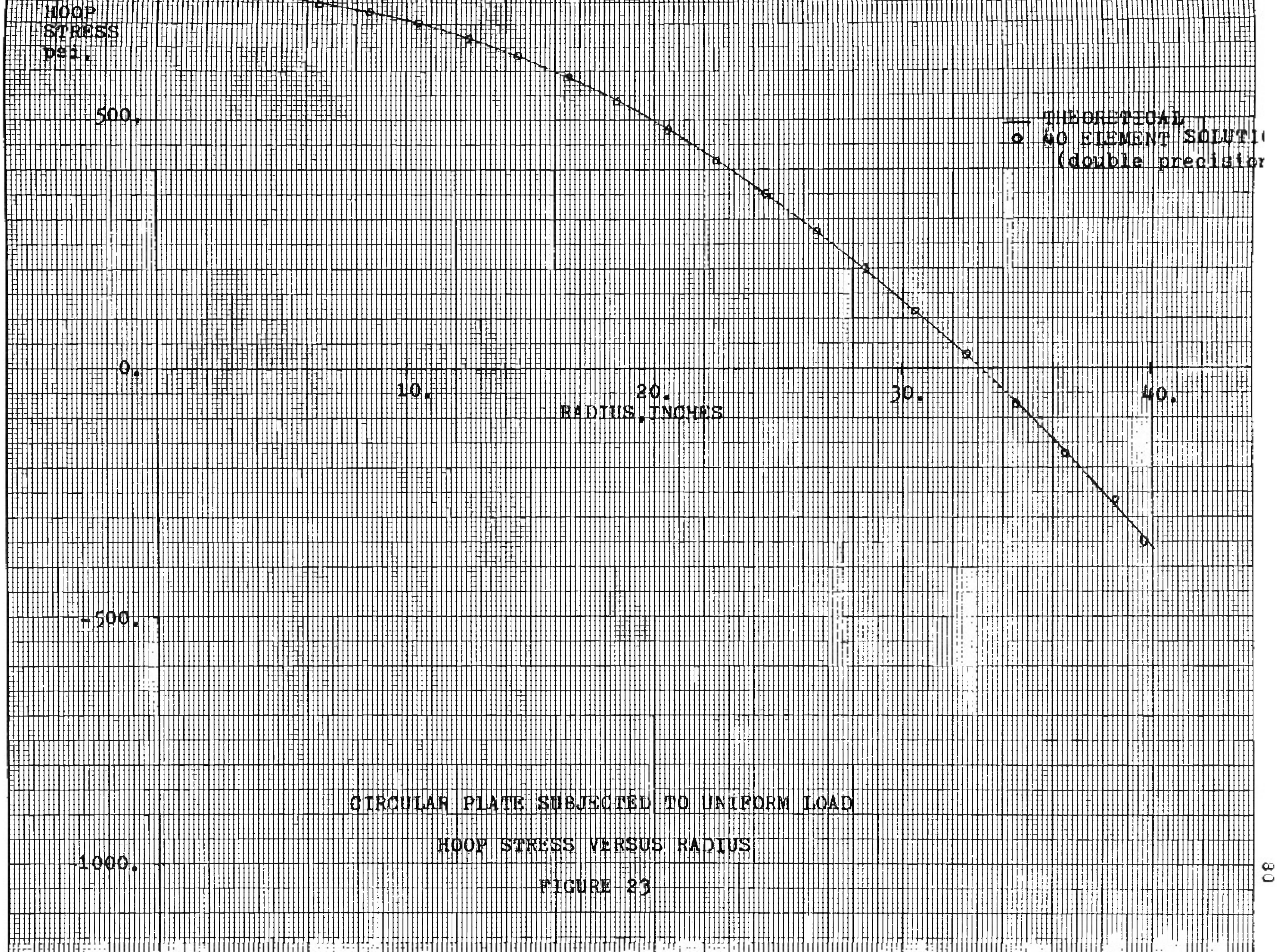
FIGURE 21



CIRCULAR PLATE SUBJECTED TO UNIFORM LOAD

RADIAL STRESS VERSUS RADIUS

FIGURE 22



CIRCULAR PLATE SUBJECTED TO UNIFORM LOAD

HOOP STRESS VERSUS RADIUS

FIGURE 23

9.0 DISCUSSION OF RESULTS

The theory for an axisymmetric finite element, using the isoparametric concept has been presented.

The isoparametric element requires the introduction of more sophisticated mathematical techniques than conventional straight sided elements. These mathematical sophistications lead to additional steps in element development, and to an increase in program computational time. However, it has been recognized that the curved isoparametric element will generally require fewer total elements to attain a specific degree of accuracy than will models using straight sided elements. Thus, superiority of either element type over the other is dependent on the particular area of concern. (e.g. development time, accuracy, computer time). The author believes that the isoparametric quadratic quadrilateral is an efficient element for axisymmetric analysis. Further tests of element convergence characteristics and comparisons with other elements is recommended for formal verification.

The necessity of numerical quadrature for evaluation of element stiffness matrices based on the isoparametric concept has been identified. Also, it was noted that the use of Gauss guadrature techniques as opposed to Newton-Cotes methods results in approximately a halving of the number of sampling points required for integral evaluations, and is therefore more efficient.

The tridiagonal method developed by Levy [20] for solution of structural nodal point displacements has been found to be an efficient technique when the amount of computer core required by the program is important. By restricting the maximum allowable difference in nodal point numbers defining an element, a banded structural stiffness matrix is obtained. Considering only the stiffness coefficients within the band, the total computer core required is greatly reduced. For the program developed, utilitization of its full capabilities would require computer core of 1.44×10^6 words for a sparse stiffness matrix. Using the tridiagonal method, this problem is capable of being solved using 10.1×10^3 words of computer core. Although this saving is impressive, several limitations of the technique have been identified which must be considered. Numerous accesses to peripheral storage devices tend to increase total computation time. Restriction of the maximum difference between element nodal points limits the number sequencing of structure nodes. This reduces flexibility in structural idealizations when large numbers of elements are necessary. When single precision computation is used (32 bit word), the round-off error or accuracy of this technique is sensitive to the form of the structural stiffness matrix.

However, no checks are provided for assessment of the error introduced.

The method presented for calculating structural element stress components by extrapolating stress matrices to element boundaries has in certain instances been found to result in stress components exhibiting finite discontinuities between adjacent elements. These discontinuities are inherent to the finite element displacement method, and are a result of interelement force equilibrium not having been satisfied. Although these discontinuities are frequently subjected to some sort of averaging, the author has chosen to identify them for use in the evaluation of the relative merit between different finite element idealizations.

The quadratic element has been found to be quite suitable for the analysis of thick pressure vessels. Both cylindrical and spherical pressure vessels have been analysed. By increasing the number of elements in the idealization, the cylindrical pressure vessel model was able to predict displacement and stress components which were within .7% of theoretical values. The spherical pressure vessel analysis resulted in a predicted maximum displacement value within 3.87% of theory but discontinuites in predicted stress values resulted in 60.% errors in stress values at boundary surfaces. This indicated that a more refined idealization was necessary using the technique of extrapolating stress matrices to element boundaries. It is suggested that alternate methods of determining element stresses may prove more efficient.

The analysis of stresses in a circular disk due to a centrifugal force loading condition has served to demonstrate both the quadratic element's ability to represent body force loading and the need for computer program capabilities to internally generate nodal point loads due to distributed surface and body forces. Predicted stress values obtained were within .1% of theoretical values. User's specification of nodal point loads was found to be possible only for regular shaped elements (defeating the purpose of elements having curved boundaries), time consuming, and susceptible to input errors.

Determination of axial stress concentrations in a cylindrical rod in tension due to a spherical inclusion resulted in accurate results being obtained with a minimum number of elements. Initial idealization resulted in predicted axial stress values in agreement with theoretical values to within 1.06%. Also demonstrated was an inability to match boundary conditions in substructure analysis of local patches of elements from the idealization. In this substructure analysis it was found that original boundary stresses were not reproduced on the boundaries of refined models. It is felt that this discrepancy is due to violation of displacement compatibility between original and substructure models. Resolution of this problem is suggested for future study.

Investigation of the quadratic element's ability to analyse flexural problems has shown that finite element idealizations for plate bending must comply with all restrictions imposed by plate theory if reasonable results are to be expected. Also, the inadequacy of single precision calculation using Levy's tridiagonal solution technique for this type of problem has been identified. When double precision calculations were used, deflection and stress results were obtained within .25% and 4% respectively of theory.

10.0 CONCLUSIONS

1. A finite element for axisymmetric problems having quadratically varing boundaries has been successfully developed based on the isoparametric concept.
2. The most efficient numerical integration technique to employ for element stiffness matrix evaluation is Gauss quadrature.
3. The tridiagonal method of solving structure force displacement equations is an efficient technique to employ when computer core must be minimized and computer time is secondary. However, this technique will provide erroneous results due to round off error for plate flexural problems unless double precision calculations are used.
4. The stress discontinuities which arise at adjacent elements boundaries may be used to assess the merit of finite element idealizations.
5. The quadratic element is an efficient tool for the analysis of thick pressure vessels.
6. Axisymmetric problems involving distributed surface and body forces may be successfully analysed with the quadratic element. Computer program calculation of their equivalent nodal point forces is recommended.
7. Substructure analysis may predict erroneous boundary stress and displacement results.

8. The quadratic element will predict reliable stress and displacement results where bending deformation predominates (e.g. thin plates) providing finite element idealizations meet the assumptions of plate theory and double precision calculations are used.

11.0 RECOMMENDATIONS

1. Development of a routine within the program presented to calculate nodal point loads due to distributed surface and body forces.
2. Investigation of necessary conditions in sub-structure analysis to insure reproduction of original boundary stresses.
3. Investigation of alternate methods for predicting element stress components.
4. Investigation of alternate techniques for the solution of the structural equilibrium equations for one which is less sensitive to round off error or modification of the existing program to a double precision version for the Sigma 6 computer.
5. Extension of the program's options by including thermal stress calculation and two dimensional plane stress/strain analysis capabilities.
6. Provide in depth comparisons with other computer programs and convergence studies to verify program efficiency.

12.0 REFERENCES

1. Timoshenko, S. and Goodier, J., Theory of Elasticity (McGraw Hill Book Company, Inc., New York, 1970)

2. Timoshenko, S., Strength of Materials, Part II (Van Nostrand Reinhold Company, New York, 1958), third edition.

3. Clough, R. and Rashid, Y., "Finite Element Analysis of Axisymmetric Solids", Journal of Engineering Mechanics, Amer. Soc. Civil Engrs., 71-85, (Feb. 1965).

4. Wilson, E., "Structural Analysis of Axisymmetric Solids", AIAA Journal, Vol. 3, No. 12, 2269-2274, (Dec. 1965)

5. Crose, J., "Stress Analysis of Axisymmetric Solids with Asymmetric Properties", AIAA Journal, Vol. 10, No. 7, 866-871, (July 1972).

6. Ergatoudis, J., Irons, B., Zienkiewicz, O., "Curved Isoparametric 'Quadrilateral' Elements for Finite Element Analysis", Int. Journal Solids Structures, Vol. 4, 31-42, (1968).

7. Irons, B., "Numerical Integration Applied to Finite Element Methods", Conference on the Use of Digital Computers in Structural Engineering, University of Newcastle, England, July 1966, (revision from author Nov. 1973).

8. Ergatoudis, J., "Isoparametric Finite Elements in Two and Three Dimensional Stress Analysis", PhD. Thesis, University College of Swansea, Wales, (Oct. 1968).

9. Zienkiewicz, O., Irons, B., "The Isoparametric Finite Element System-A New Concept in Finite Element Analysis", Conference on Recent Advances in Stress Analysis: New Concepts and Techniques and Their Practical Application, Royal Aeronautical Society, 35-40, (March 26-29, 1968).

10. Zienkiewicz, O., Irons, B., Ergatoudis, J., Ahmad, S., Scott, F., "Isoparametric and Associated Element Families for Two and Three Dimensional Analysis", Finite Element Methods in Stress Analysis, Ivar Holand and Kolbein Bell, eds., Trondheim, Norway: Tapir Publishing, The Technical University of Norway, 383-432, (1972).

11. Zienkiewicz, O., Irons, B., "Isoparametric Elements", Finite Element Techniques in Structural Mechanics, H. Tottenham and C. Brebbia, eds., Southhampton England Stress Analysis Publishers, Southhampton University, 283-301, (1970).

12. Zienkiewicz, O., The Finite Element Method in Engineering Science, (Mc Graw Hill Book Company, Inc., London, 1971), second edition.

13. Irons, B., "Engineering Applications of Numerical Integration in Stiffness Methods, AIAA Journal Vol. 4 No. 11, 2035-2037, (Nov. 1966).

14. Irons, B., "Economical Computer Techniques for Numerically Integrated Finite Elements" Inter. Journal for Num. Meth. in Eng., Vol. 1, 201-203, (1969).

15. Irons, B., "Quadrature Rules for Brick Based Finite Elements", Inter. Journal for Num. Meth. in Eng., 293-294, (1971).

16. Gupta, A., Mohraz, B., "A Method of Computing Numerically Integrated Stiffness Matrices", Inter. Journal for Num. Meth. in Eng., Vol. 5, 83-89, (1972).

17 Desai, C., Abel, J., Introduction to the Finite Element Method, (Van Nostrand Reinhold Company, New York, 1972), first edition.

18. Archer, J., "Consistent Matrix Formulations for Structural Analysis Using Finite Element Techniques", AIAA Journal, Vol. 3, No. 10, 1911-1918, (Oct. 1965)

19. Tottenham, H., "Basic Principles", Finite Element Techniques in Structural Mechanics, H. Trottenham and C. Brebbia, eds., Southhampton England: Stress Analysis Publishers, Southhampton Univeristy, 1-30, (1970).

20. Levy, S., "ISO2D", Computer Workshop in Finite Element Methods for Stress and Other Field Problems, Schenectady, New York: Union College Graduate and Special Programs, Union College, (Aug. 1973).

21. Dario, N., Bradley, W., "A Comparison of First and Second Order Axially Symmetric Finite Elements", Inter. Journal for Num. Meth. In Eng., Vol. 5, 573-583, (1973).

22. Hildebrand, F., Introduction to Numerical Analysis, (McGraw Hill Book Company, Inc., New York, 1974), second edition.

23. Spiegel, M., Advanced Calculus, Schaum's Outline Series, (McGraw Hill Book Company, Inc., New York, 1963).

24. Oden, J., Finite Elements of Nonlinear Continua, (McGraw Hill Book Company, Inc., New York, 1972) first edition.

25. Loeber, J., Knolls Atomic Power Laboratory, Schenectady, N. Y., Private Communication.

26. McCalley, R., General Electric Company, Schenectady, N.Y., Private Communication.

27. Glasser, T., Knolls Atomic Power Laboratory, Schenectady, N.Y., Private Communication.

28. Timoshenko, S., Woinowski-Krieger, Theory of Plates and Shells (McGraw Hill Book Company, Inc., New York, 1959), second edition.

29. Cook, R., "A Note on Certain Incompatable Elements", Inter. Journal for Num. Meth. in Eng., 146-147, (1973)

30. Griffin, D., "Use of the Computer for the Elastic Analysis of Pressure Vessels", The Computer in Pressure Vessel Analysis, Papers and Discussions from ASME Computer Seminar, Dallas, Texas, September 26, 1968.

31. Ergatoudis, J., "Quadrilateral Elements in Plane Analysis and Introduction to Solid Analysis", MS Thesis, University College of Swansea, Wales, (June, 1966).

32. Wilson, E., Parsons, B., "Trapezoidal Finite Elements - Their Derivation and Use for Axisymmetric Rotating Bodies", ASME Presentation at the Design Engineering Technical Conference, Cincinnati, Ohio, September 9-12, 1973. Paper 73 - DET - 47.

33. Rieger, N., Rochester Institute of Technology, Rochester, N.Y. Private Communication.

34. Martin, H., Carey, G., Introduction to Finite Element Analysis, (McGraw Hill Book Company, Inc., New York, 1973), first edition.

35. Halbleib, W., Rochester Institute of Technology, Rochester, N.Y. Private communication.

13.0 APPENDIX A

It is noted that the function to be integrated in Eq. 13 was of a sufficiently complex form to necessitate the use of numerical quadrature.

Numerical quadrature is a numerical procedure for the evaluation of definite integrals. Geometrically it requires the numerical determination of the area or volume under the integrand's curve[23]. To use this procedure, sampling points are chosen within the region of interest and the integrand is evaluated at them. Based on these integrand values and the number of sampling points chosen, an approximate value of the integral may then be obtained.

Numerical quadrature techniques may be divided into two basic categories [22], those whose sampling points are equally spaced over the region of interest (Newton-Cotes), and those whose sampling point are chosen at optimal locations and have weighting functions associated with them (Gauss).

Using the Newton-Cotes formulae requires n sampling points for the exact integration of a function of order n-1 whereas the Gauss technique requires n/2 sampling points.

Using the finite element method, structural idealization usually involves introduction of large numbers of elements for which stiffness matrices must be found. Efficient computing techniques rely on a minimizing of the number of mathematical operations necessary. For this reason Gaussian quadrature, which requires the fewest sampling points to integrate a function of specific order, is most frequently employed [7]. In Gaussian quadrature, the integral of a function f(x) is replaced by the summation:

$$\int_{-1}^{1} f(x)\,dx = \sum_{i=1}^{n} Hi\ f(a_i) \tag{14}$$

n is the number of sampling points

H_i is the weight coefficient associated with sampling point i

a_i is the abscissa of sampling point i

The theory for determination of optimal sampling points and weight coefficients may be found in Hildebrand [22]. Specific values for n = 2 through 24 are presented in Table V.

The procedure for evaluating the element stiffness matrix, [K] is as follows:

Eq. 13 may be rewritten as:

$$[K] = \int_{-1}^{1}\int_{-1}^{1} f\ (P,Q)\ dPdQ \tag{15}$$

where f(P,Q) is a matrix in P and Q equal to

$$2\pi[B]^T[D][B]\ [N]\ \{r_o\}\det\ [J]$$

Integration may be performed in a manner similar to the standard technique of evaluating double integrals. Substituting Eq. 14 intoEq. 15 while holding Q constant, one obtains:

$$[K] = \int_{-1}^{1} \sum_{i=1}^{n} H_i f(a_i, Q)\, dQ$$

Applying Eq. 14 again but with respect to Q, the expression for [K] becomes:

$$[K] = \sum_{j=1}^{n} \sum_{i=1}^{n} H_j H_i f(a_i, a_j) \qquad (16)$$

The number of sampling points in each direction used in Eq. 16 should be such that the volume of each element is exactly determined[12]. The minimum number of sampling points which are required is determined by the order of the determinant of the Jacobian matrix [J].

For the quadratic element developed, the minimum value of n is 2. However the element developed is for n = 3 for convenience in element stress calculations (Section 4.8).

Considering the case of n = 3, the element stiffness matrix is determined by the summation of the function f(P,Q) multiplied by its weight function at the nine sampling points shown in Figure 20.

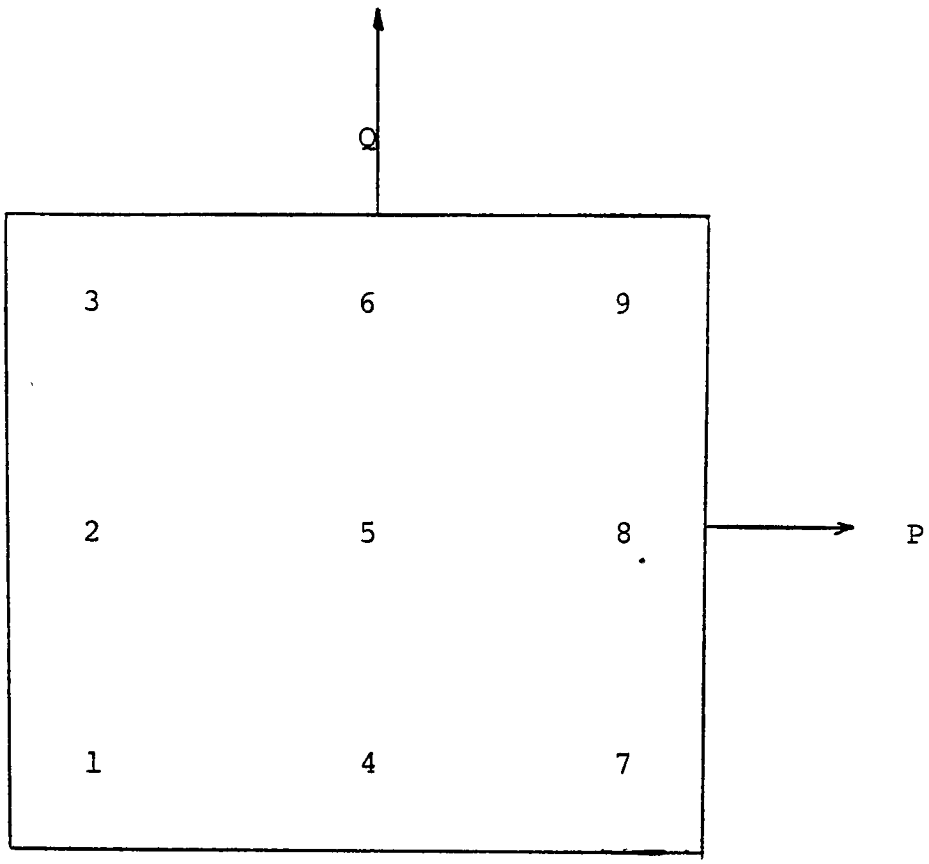


Figure 20

Sampling Point Locations
for Gaussian Quadrature

TABLE V

ABSCISSAS AND WEIGHT FACTORS FOR GAUSSIAN INTEGRATION

$$\int_{-1}^{+1} f(x)\,dx \approx \sum_{i=1}^{n} w_i f(x_i)$$

Abscissas $= \pm x_i$ (Zeros of Legendre Polynomials) Weight Factors $= w_i$

n = 2

$\pm x_i$	w_i
0.57735 02691 89626	1.00000 00000 00000

n = 3

$\pm x_i$	w_i
0.00000 00000 00000	0.88888 88888 88889
0.77459 66692 41483	0.55555 55555 55556

n = 4

$\pm x_i$	w_i
0.33998 10435 84856	0.65214 51548 62546
0.86113 63115 94053	0.34785 48451 37454

n = 5

$\pm x_i$	w_i
0.00000 00000 00000	0.56888 88888 88889
0.53846 93101 05683	0.47862 86704 99366
0.90617 98459 38664	0.23692 68850 56189

n = 6

$\pm x_i$	w_i
0.23861 91860 83197	0.46791 39345 72691
0.66120 93864 66265	0.36076 15730 48139
0.93246 95142 03152	0.17132 44923 79170

n = 7

$\pm x_i$	w_i
0.00000 00000 00000	0.41795 91836 73469
0.40584 51513 77397	0.38183 00505 05119
0.74153 11855 99394	0.27970 53914 89277
0.94910 79123 42759	0.12948 49661 68870

n = 8

$\pm x_i$	w_i
0.18343 46424 95650	0.36268 37833 78362
0.52553 24099 16329	0.31370 66458 77887
0.79666 64774 13627	0.22238 10344 53374
0.96028 98564 97536	0.10122 85362 90376

n = 9

$\pm x_i$	w_i
0.00000 00000 00000	0.33023 93550 01260
0.32425 34234 03809	0.31234 70770 40003
0.61337 14327 00590	0.26061 06964 02935
0.83603 11073 26636	0.18064 81606 94857
0.96816 02395 07626	0.08127 43883 61574

n = 10

$\pm x_i$	w_i
0.14887 43389 81631	0.29552 42247 14753
0.43339 53941 29247	0.26926 67193 09996
0.67940 95682 99024	0.21908 63625 15982
0.86506 33666 88985	0.14945 13491 50581
0.97390 65285 17172	0.06667 13443 08688

n = 12

$\pm x_i$	w_i
0.12523 34085 11469	0.24914 70458 13403
0.36783 14989 98180	0.23349 25365 38355
0.58731 79542 86617	0.20316 74267 23066
0.76990 26741 94305	0.16007 83285 43346
0.90411 72563 70475	0.10693 93259 95318
0.98156 06342 46719	0.04717 53363 86512

n = 16

$\pm x_i$	w_i
0.09501 25098 37637 440185	0.18945 06104 55068 496285
0.28160 35507 79258 913230	0.18260 34150 44923 588867
0.45801 67776 57227 386342	0.16915 65193 95002 538189
0.61787 62444 02643 748447	0.14959 59888 16576 732081
0.75540 44083 55003 033895	0.12462 89712 55533 872052
0.86563 12023 87831 743880	0.09515 85116 82492 784610
0.94457 50230 73232 576078	0.06225 35239 38647 892863
0.98940 09349 91649 932596	0.02715 24594 11754 094852

n = 20

$\pm x_i$	w_i
0.07652 65211 33497 333755	0.15275 33871 30725 850698
0.22778 58511 41645 078080	0.14917 29864 72603 746788
0.37370 60887 15419 560673	0.14209 61093 18382 051329
0.51086 70019 50827 098004	0.13168 86384 49176 626898
0.63605 36807 26515 025453	0.11819 45319 61518 417312
0.74633 19064 60150 792614	0.10193 01198 17240 435037
0.83911 69718 22218 823395	0.08327 67415 76704 748725
0.91223 44282 51325 905868	0.06267 20483 34109 063570
0.96397 19272 77913 791268	0.04060 14298 00386 941331
0.99312 85991 85094 924786	0.01761 40071 39152 118312

n = 24

$\pm x_i$	w_i
0.06405 68928 62605 626085	0.12793 81953 46752 156974
0.19111 88674 73616 309159	0.12583 74563 46828 296121
0.31504 26796 96163 374387	0.12167 04729 27803 391204
0.43379 35076 26045 138487	0.11550 56680 53725 601353
0.54542 14713 88839 535658	0.10744 42701 15965 634783
0.64809 36519 36975 569252	0.09761 86521 04113 888270
0.74012 41915 78554 364244	0.08619 01615 31953 275917
0.82000 19859 73902 921954	0.07334 64814 11080 305734
0.88641 55270 04401 034213	0.05929 85849 15436 780746
0.93827 45520 02732 758524	0.04427 74388 17419 806169
0.97472 85559 71309 498198	0.02853 13886 28933 663181
0.99518 72199 97021 360180	0.01234 12297 99987 199547

Compiled from P. Davis and P. Rabinowitz, Abscissas and weights for Gaussian quadratures of high order, J. Research NBS 56, 35–37, 1956, RP2645; P. Davis and P. Rabinowitz, Additional abscissas and weights for Gaussian quadratures of high order. Values for n=64, 80, and 96, J. Research NBS 60, 613-614, 1958, RP2875; and A. N. Lowan, N. Davids, and A. Levenson, Table of the zeros of the Legendre polynomials of order 1-16 and the weight coefficients for Gauss' mechanical quadrature formula, Bull. Amer. Math. Soc. 48, 739-743, 1942 (with permission).

14.0 APPENDIX B COMPUTER PROGRAM ISOAXI

B1. General Analyst's Information

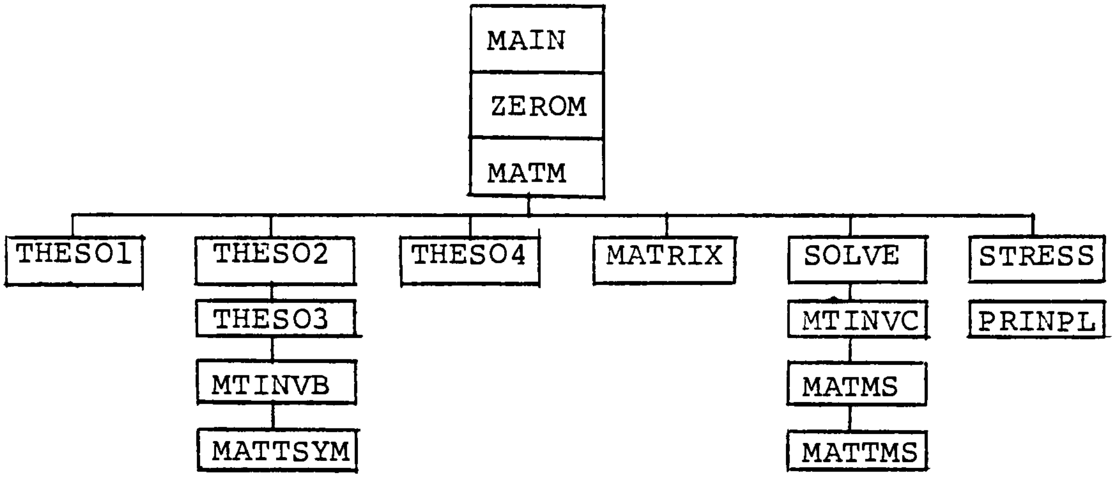
ISOAXI is a finite element computer program for the static stress analysis of axisymmetric structures having axisymmetric boundary conditions. The element used is a quadratic axisymmetric quadrilateral capable of representing geometric boundaries of quadratic variation. The assumed element displacement function is also quadratic. Element stress-strain relations are for a homogeneous isotropic material. Up to ten different materials may be specified per problem. Options not included at present are internal calculations of body forces and surface forces, thermal stress capabilities, and pre and post processors for mesh generation and computer plotting.

B2. Programmer's Information

Program development was accomplished using a Xerox Sigma 6 computer. ISOAXI is written in Fortran IV and makes use of three temporary files (2,3,4) for data storage and retrieval during execution. Data input is from a card reader (F:105) and output is to a line printer

(F:108). The program is overlayed as shown in <u>B3</u>. and required 10.1K words of main computer core for execution.

B3. COMPUTER PROGRAM STRUCTURE



B4. SUBROUTINE DESCRIPTIONS

The functions of the subroutines shown in B3. are as follows:

1. MAIN executive routine for calling subroutines
2. ZEROM* initializes a matrix to zero
3. MATM* multiplies two matrices
4. THESO1 calculates the shape functions and their partial derivatives at the nine sampling points.
5. THESO2 reads the majority of input data and writes it out for checking.
6. THESO3 calculates the element stiffness matrices and stress matrices at the sampling points of each element
7. MTINVB* finds the inverse of a matrix and also the value of its determinant
8. MTTSYM* multiplies two matrices, first transposed times the other and insures resulting matrix is symmetric
9. THESO4 extrapolates the stress matrices at the sampling points of each element to its midside nodes.
10. MATRIX* reads nodal point prescribed displacements and external forces, assembles the structural stiffness matrix and modifies it to accomodate prescribed dis placements.

11. SOLVE* solves the structural equilibrium relations for displacement using a modified Gauss elimination technique

12. MTINVC* inverts a matrix using the Gauss technique with pivoting

13. MATMS* multiplies two matrices

14. MATTMS* multiplies the transpose of a matrix with a second matrix

15. STRESS writes nodal point displacements, calculates and writes element stresses.

16. PRINPL* calculates principal stresses and direction cosines

*Subroutines obtained from Levy [20].

B5 INPUT DATA FORMAT

Data input to ISOAXI is in the form of punched cards. Data required may be generalized as consisting of seven sets, (A-G) with the number of cards required in each set depending on the particular problem being solved. The order in which input data should appear, its format, and definition is presented below.

DATA SET A PROBLEM PARAMETERS (FORMAT 7I4)

KPNT - number of nodal points in problem (max. 600)

KELM - number of elements in problem

NGEO - number of nodes having geometric constraints

NMAT - number of different materials in problem (max. 10)

NFREE- degrees of freedom per node (always 2)

NFOR - number of nodes subjected to external force

NPART - number of partitions in problem (max. 45)

DATA SET B NODAL POINT LOCATIONS (FORMAT 2F16.8)

X(1,J) - radial distance from origin of nodal point J

X(2,J) - axial distance from origin of nodal point J

(This set contains KPNT cards in sequential order from 1 through KPNT.)

DATA SET C MATERIAL PROPERTIES (FORMAT 2F16.4)

E(I) - Young's modulus of material I

P(I) - Poisson's ratio of material I

(This set contains NMAT cards in sequential order from 1 through NMAT)

DATA SET D ELEMENT DEFINITION (FORMAT 9I4)

N1,N2,...N8 the eight nodal points defining an element specified counter-clockwise with respect to coordinate axes and started at a corner node.

NM - the number I in data set C which corresponds to the material properties of the element

(This set contains KELM cards in sequential order from 1 through KELM)

DATA SET E PARTITION INFORMATION (FORMAT 4I4)

NSTART(I) - first element in partition I

NEND(I) - last element in partition I

NFIRST(I) - first node in partition I

NLAST(I) - last node in partition I

(This set contains NPART cards in sequential order from 1 through NPART)

DATA SET F PRESCRIBED NODAL DISPLACEMENTS (FORMAT 3I4, 2F16.8)

NO - the node having prescribed displacements

NA - 0 if radial displacement is specified, 1 if not

NB - 0 if axial displacement is specified, 1, if not

U - magnitude of specified radial displacement

V - magnitude of specified axial displacement

(This set contains NGEO cards)

<u>DATA SET G</u> EXTERNAL NODAL POINT FORCES (FORMAT I4,2F16.4)

NODE - the node at which external force acts

FORR - the radial component of force acting at the node*

FORZ - the axial component of force acting at the node*

* the total force through 2π radians

(This set contains NFOR cards)

B6 EXAMPLE OF COMPUTER INPUT-OUTPUT DATA

An example of computer program output is presented in Fig.24 and corresponds to the cylindrical pressure vessel problem (1 element solution) presented in section 8.1, Fig. 7. A sample listing of input data is not presented since computer output includes this information.

First output by the program is all input data information. This is done to facilitate data checking and also provide model documentation Printout of this information is in the same order as presented in section B5 and is noted in Fig. 24. Following this information, displacement components of all structural nodal points are output. Displacement output in Fig.24 for nodal point 1 indicates that radial and axial displacement components are:

$$u = -.43032356 \times 10^{-2} \text{in.}$$

$$v = .37970068 \times 10^{-3} \text{in.}$$

Following displacement output, element stress information is printed. Four sets of stress information are provided for each element's midside nodes. Each set contains the following information:

EL - Element number

NODES - The 8 nodes defining element EL

STRPT - Midside node to which stress components correspond.

SIGR - Normal component of stress in the radial direction

SIGTHETA - Normal component of stress in the theta direction

SIGZ - Normal stress component in the axial direction

TAURZ - Shear stress in the rz plane

PS - Principal stress value

L - Direction cosine between PS and r axis

M - Direction cosine between PS and z axis

N - Direction cosine between Ps and the θ axis

An example of interpretation of this information for element 1, defined by nodes 1 2 3 5 8 7 6 4, the stress components acting at node 4 (STRP 4) are:

σ_r = -12188.5 psi

σ_θ = -21995.3 psi

σ_z = -15.5 psi

τ_{rz} = -3.2 psi

These stress components correspond to principal stresses of:

15.5 psi in the -z direction (L=0.,M=-1.,N=0.)

-12188.4 psi in the $\underline{-}$rdirection (L=-1.,M=0.,N=0.)

-21995.3 psi in the $+\underline{\theta}$ direction (L=0.,M=0.,N=1)

FIGURE 24
COMPUTER OUTPUT - ISOAXI

```
RUN  (LMN,THESIS)
 TOTAL NUMBER OF NODAL POINTS..........        8
 TOTAL NUMBER OF ELEMENTS..............        1
 NUMBER OF GEOMETRIC CONSTRAINTS.......        0
 NUMBER OF DIFFERENT MATERIALS.........        1
 DEGREES OF FREEDOM PER NODE...........        2
 NUMBER OF NODES SUBJECTED TO FORCE....        6
 NUMBER OF PARTITIONS..................        1


         NODE             R CO-ORDINATE            Z CO-ORDINATE

           1                    5.0000                    1.0000
           2                    5.0000                     .5000
           3                    5.0000                     .0000
           4                    7.5000                    1.0000
           5                    7.5000                     .0000
           6                   10.0000                    1.0000
           7                   10.0000                     .5000
           8                   10.0000                     .0000
 MATERIAL  MODULUS OF ELASTICITY  POISSONS RATIO

       1             30000000.0000               .3000
 ELEMENT              ELEMENT  NODES            MATERIAL

       1    1    2    3    5    8    7    6    4    1
                 1                  1                1                   2
    3    1    0         .00000000         .00000000
    7    1    0         .00000000         .00000000
    1       37123.8906             .0000
    2      148495.5625             .0000
    3       37123.8906             .0000
    6     -157079.6250             .0000
    7     -628318.5000             .0000
    8     -157079.6250             .0000
 * * * *DISPLACEMENTS* * * * * * *

 NODE       RADIAL(R)          AXIAL(Z)
 ----       ---------          --------


    1   -.43032356E-02    .37970068E-03
    2   -.43273158E-02    .19016737E-03
    3   -.43445453E-02    .00000000E 00
    4   -.45335413E-02    .26179291E-03
    5   -.46213232E-02   -.90375456E-04
    6   -.52341036E-02    .19534706E-03
    7   -.52484572E-02    .00000000E 00
    8   -.52683308E-02   -.18555872E-03
 EL    1   NODES    1    2    3    5    8    7    6    4 STRESS    0

 SIGR    -9594.0SIGTHETA   -28376.7SIGZ          -.0TAURZ           -.0

 PS          -.2L  -.000064M-1.0-0500N    .000000
```

Figure 24 (Continued)

```
PS    -9594.?L -.99992?M -.?12642N  .? ? ? ?
PS  -22376.7L  .??? ??M  .? ??? ??N 1.? ? ? ?
EL    1  NODES   1   2   3   5   8   7   6   4 STRPT  -
SIGR  -12193.?SIGTHETA  -22146.7SIGZ      -1?. STAURZ      -? .?
PS      -11.5L -.???13?M-1.??? ???N  .? ? ???
PS  -12193.2L -.999997M -.?0243?N  .? ? ???
PS  -22146.7L  .??? ???M  .??? ???N 1.? ? ???
EL    1  NODES   1   2   3   5   8   7   6   4 STRPT  7
SIGR  -16223.5SIGTHETA  -?1843.?SIGZ       7. STAURZ      .?
PS        7.3L  .0??519M-1.??? ???N  .? ? ???
PS  -16223.5L  .999998M -.?136?4N  .? ? ???
PS  -2?843.?L  .??? ???M  .??? ??0N 1.? ? ???
EL    1  NODES   1   2   3   5   8   7   6   4 STRPT  4
SIGR  -12188.5SIGTHETA  -21995.3SIGZ      -1?. STAURZ      -? .?
PS      -15.5L -.000255M-1.??? ??0N  .? ? ???
PS  -12188.4L -.999999M -.?01209N  .? ? ???
PS  -21995.3L  .00? ???M  .??? ??0N 1.? ? ???
*STOP* 0
```

```
C* * * * * * * * * * * * * * * * * * * * * * * *
C      FINITE ELEMENT PROGRAM ISOAXI
C      FOR AXISYMMETRIC STATIC STRESS ANALYSIS
C      USING AN 8 NODED QUADRATIC ISOPARAMETRIC
C      ELEMENT SUBJECTED TO AXISYMMETRIC LOADING
C      AND GEOMETRIC CONSTRAINTS
C
C* * * * * * * * * * * * * * * * * * * * * * * *
C
C      MASTER OF SCIENCE THESIS
C      F.X.JANUCIK  JUNE,1974
C      ROCHESTER INSTITUTE OF TECHNOLOGY
C
C* * * * * * * * * * * * * * * * * * * * * * * *
C
C* * *PARAMETERS:
C
C      KPNT=MAXIMUM NUMBER OF NODES=600
C      NGEO=MAXIMUM NUMBER OF GEOMETRIC CONSTRAINTS=600
C      NMAT=MAXIMUM NUMBER OF DIFFERENT MATERIAL PROPERTIES=10
C      NFREE=NUMBER OF DEGREES OF FREEDOM PER NODE(ALWAYS 2)
C      NFOR=MAXIMUM NUMBER OF EXTERNAL NODAL FORCES=600
C      NPART=MAXIMUM NUMBER OF PARTITIONS=45
C
C* * * * * * * * * * * * * * * * * * * * * * * *
C
C* * *VARIABLES:
C
C      X(1,J)=RADIAL DISTANCE FROM ORIGIN OF NODE J
C      X(2,J)=AXIAL DISTANCE FROM ORIGIN OF NODE J
C      E(I)=YOUNGS MODULUS OF MATERIAL I
C      P(I)=POISSONS RATIO OF MATERIAL I
C      NSTART(I)=FIRST ELEMENT IN PARTITION I
C      NEND(I)=LAST ELEMENT IN PARTITION I
C      NFIRST(I)=FIRST NODE IN PARTITION I
C      NLAST(I)-LAST NODE IN PARTITION I
C      NF(I)=NODAL POINT NUMBER OF CONSTRAINT
C      NB(I,1)=0 IF CONSTRAINED, 1 IF NOT (RADIAL DIRECTION)
C      NB(I,2)=0 IF CONSTRAINED, 1 IF NOT (AXIAL DIRECTION)
C      BV(I,1)=MAGNITUDE OF RADIAL CONSTRAINT
C      BV(I,2)=MAGNITUDE OF AXIAL CONSTRAINT
C      U(1,K)=RADIAL FORCE THRU 2 PI RADIANS AT NODE K
C      U(2,K)=AXIAL FORCE THRU 2 PI RADIANS AT NODE K
C      STRPT=NODE AT WHICH STRESS COMPONENTS ARE CALCULATED
C      PS=PRINCIPAL STRESS
C      L,M,N=DIRECTION COSINES
C      SIGR,SIGTHETA,SIGZ,TAURZ=RADIAL,HOOP,AXIAL,
C                   AND SHEAR STRESSES RESPECTIVELY
C
C* * * * * * * * * * * * * * * * * * * * * * * * *
C
C* * *SUBROUTINES:
C
C      THES01
C      THES02
C      THES03
C      MATM
C      ZEROM
C      MTTSYM
C      MATMS
C      MATTMS
C      MTINVB
C      THES04
C      MATRIX
C      SOLVE
C      MTINVC
C      STRESS
C      PRINPL
C
C* * * * * * * * * * * * * * * * * * * * * * * * * *
C
C* * *MAIN EXECUTIVE PROGRAM:
      COMMON NPART,KPNT,KELM,NGEO,NMAT,NFREE,NFOR
     1,NSTART(45),NEND(45),NFIRST(45),NLAST(45)
      CALL THES01
      REWIND 2
      REWIND 3
      REWIND 4
      CALL THES02
      REWIND 2
```

```
      REWIND 4
      CALL THESO4
      REWIND 3
      REWIND 4
      CALL MATRIX
      REWIND 3
      REWIND 4
      CALL SOLVE
      REWIND 2
      REWIND 3
      CALL STRESS
      STOP
      END
```

```
C   MATRIX MULTIPLICATION   (DB)(LXN)=D(LXM)B(MXN)
      SUBROUTINE MATM(D,B,DB,L,M,N)
      DIMENSION D(L,M),B(M,N),DB(L,N)
      DO 110 J=1,N
      DO 110 I=1,L
      DB(I,J)=0.
      DO 110 K=1,M
  110 DB(I,J)=DB(I,J)+D(I,K)*B(K,J)
      RETURN
      END
```

```
C                 SUBROUTINE ZEROM
      SUBROUTINE  ZEROM(A,I,K)
      DIMENSION  A(1)
      II=I*K
      DO 10  J=1,II
   10 A(J)=0.0
      RETURN
      END
```

```
C****SUBROUTINE THES01
      SUBROUTINE THES01
C* * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *
C THIS SUBROUTINE CALCULATES THE SHAPE FUNCTIONS AND
C THEIR DERIVATIVES AT THE NINE GAUSS POINTS
C FUNCTIONS STORED IN CFUN(I,J),I=G.P.,J=SHAPE FUNCTION
C PARTIALS STORED IN AJP(K,L,M),K=WITH RESPECT TO KAI(1)
C ,WITH RESPECT TO ETA(2),L=SHAPE FUNCTION,M=GAUSS PT.
C* * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *
      COMMON NPART,KPNT,KELM,NGEO,NMAT,NFREE,NFOR
     1,NSTART(45),NEND(45),NFIRST(45),NLAST(45)
C
      DIMENSION  EKI(9),ETA(9),AJP(2,8,9),CFUN(9,8)
C
C**** PARTIAL DERIVATIVES OF SHAPE FUNCTIONS WITH RESPECT TO P
C
      FUC01(X,Y)=.25*((1.+Y)*(X+Y-1.)+(1.+X)*(1.+Y))
      FUC02(X,Y)=-X*(1.+Y)
      FUC03(X,Y)=.25*((-(1.+Y)*(-X+Y-1.))-(1.-X)*(1.+Y))
      FUC04(X,Y)=-.5*(1.-Y**2)
      FUC05(X,Y)=.25*((-(1.-Y)*(-X-Y-1.))-(1.-X)*(1.-Y))
      FUC06(X,Y)=-X*(1.-Y)
      FUC07(X,Y)=.25*((1.-Y)*(X-Y-1.)+(1.+X)*(1.-Y))
      FUC08(X,Y)=.5*(1.-Y**2)
C
C**** PARTIAL DERIVATIVES OF SHAPE FUNCTIONS WITH RESPECT TO Q
C
      FUC09(X,Y)=.25*((1.+X)*(X+Y-1.)+(1.+X)*(1.+Y))
      FUC10(X,Y)=.5*(1.-X**2)
      FUC11(X,Y)=.25*((1.-X)*(-X+Y-1.)+(1.-X)*(1.+Y))
      FUC12(X,Y)=-Y*(1.-X)
      FUC13(X,Y)=.25*((-(1.-X)*(-X-Y-1.))-(1.-X)*(1.-Y))
      FUC14(X,Y)=-.5*(1.-X**2)
      FUC15(X,Y)=.25*((-(1.+X)*(X-Y-1.))-(1.+X)*(1.-Y))
      FUC16(X,Y)=-Y*(1.+X)
C
C**** SHAPE FUNCTIONS
C
      FN1(X,Y)=-.25*(1.+X)*(1.+Y)*(1.-X-Y)
      FN2(X,Y)=.5*(1.-(X**2))*(1.+Y)
      FN3(X,Y)=.25*(1.-X)*(1.+Y)*(-X+Y-1.)
      FN4(X,Y)=.5*(1.-X)*(1.-(Y**2))
      FN5(X,Y)=.25*(1.-X)*(1.-Y)*(-X-Y-1.)
      FN6(X,Y)=.5*(1.-X**2)*(1.-Y)
      FN7(X,Y)=.25*(1.+X)*(1.-Y)*(X-Y-1.)
      FN8(X,Y)=.5*(1.+X)*(1.-Y**2)
C
C**** DEFINE GAUSS POINTS (EKI(I),ETA(I)),I=1,9
C
      X1=0.77459667
      X2=0.00000000
C
      EKI(1)=-X1
      EKI(2)=-X1
      EKI(3)=-X1
C
      EKI(4)=X2
      EKI(5)=X2
      EKI(6)=X2
C
      EKI(7)=X1
      EKI(8)=X1
      EKI(9)=X1
C
      ETA(1)=-X1
      ETA(2)=X2
      ETA(3)=X1
C
      ETA(4)=-X1
      ETA(5)=X2
      ETA(6)=X1
C
      ETA(7)=-X1
      ETA(8)=X2
      ETA(9)=X1
C
      DO 50 I=1,2
      DO 50 J=1,8
      DO 60 K=1,9
      AJP(I,J,K)=0.00000000
```

```
   60 CONTINUE
   50 CONTINUE
C*** EVALUATE PARTIAL DERIVATIVES AND SHAPE FUNCTIONS AT 9 SAMPLING POIN
      DO 1999 K=1,9
C**** PARTIAL OF N'S WITH RESPECT TO P
      AJP(1,1,K)=AJP(1,1,K)+ FUC01(EKI(K),ETA(K))
      AJP(1,2,K)=AJP(1,2,K)+ FUC02(EKI(K),ETA(K))
      AJP(1,3,K)=AJP(1,3,K)+ FUC03(EKI(K),ETA(K))
      AJP(1,4,K)=AJP(1,4,K)+ FUC04(EKI(K),ETA(K))
      AJP(1,5,K)=AJP(1,5,K)+ FUC05(EKI(K),ETA(K))
      AJP(1,6,K)=AJP(1,6,K)+ FUC06(EKI(K),ETA(K))
      AJP(1,7,K)=AJP(1,7,K)+ FUC07(EKI(K),ETA(K))
      AJP(1,8,K)=AJP(1,8,K)+ FUC08(EKI(K),ETA(K))
C**** PARTIAL OF N'S WITH RESPECT TO Q
      AJP(2,1,K)=AJP(2,1,K)+ FUC09(EKI(K),ETA(K))
      AJP(2,2,K)=AJP(2,2,K)+ FUC10(EKI(K),ETA(K))
      AJP(2,3,K)=AJP(2,3,K)+ FUC11(EKI(K),ETA(K))
      AJP(2,4,K)=AJP(2,4,K)+ FUC12(EKI(K),ETA(K))
      AJP(2,5,K)=AJP(2,5,K)+ FUC13(EKI(K),ETA(K))
      AJP(2,6,K)=AJP(2,6,K)+ FUC14(EKI(K),ETA(K))
      AJP(2,7,K)=AJP(2,7,K)+ FUC15(EKI(K),ETA(K))
      AJP(2,8,K)=AJP(2,8,K)+ FUC16(EKI(K),ETA(K))
 1999 CONTINUE
C**** AJP(2,8,9) IS FORMED
C
C**** SHAPE FUNCTIONS AT NINE SAMPLING POINTS
C
      DO 2001 I=1,9
      DO 2002 J=1,8
      CFUN(I,J)=0.00000000
 2002 CONTINUE
 2001 CONTINUE
      DO 2000 I=1,9
      CFUN(I,1)=FN1(EKI(I),ETA(I))
      CFUN(I,2)=FN2(EKI(I),ETA(I))
      CFUN(I,3)=FN3(EKI(I),ETA(I))
      CFUN(I,4)=FN4(EKI(I),ETA(I))
      CFUN(I,5)=FN5(EKI(I),ETA(I))
      CFUN(I,6)=FN6(EKI(I),ETA(I))
      CFUN(I,7)=FN7(EKI(I),ETA(I))
      CFUN(I,8)=FN8(EKI(I),ETA(I))
 2000 CONTINUE
C**** WRITE AJP AND CFUN TO TEMPORARY FILE 2
      END FILE 2
      REWIND 2
      WRITE(2)(((AJP(J,K,L),J=1,2),K=1,8),L=1,9),
     1((CFUN(I,M),I=1,9),M=1,8)
      RETURN
      END
```

```
C**** SUBROUTINE THES02
      SUBROUTINE THES02
C
      COMMON NPART,KPNT,KELM,NGEO,NMAT,NFREE,NFOR
     1,NSTART(45),NEND(45),NFIRST(45),NLAST(45)
C
      DIMENSION NOD(8),EMOD(10),P(10),XE(8,2)
     1,APJ(2,8,9),RFUN(9,8),X(600,2)
C
      REWIND 2
C
C**** READ SHAPE FUNCTIONS RFUN AND THEIR PARTIAL DERIVATIVES FROM FILE
C
      READ(2)(((APJ(J,K,L),J=1,2),K=1,8),L=1,9),
     1((RFUN(I,M),I=1,9),M=1,8)
      REWIND 2
      REWIND 4
C
C**** READ DATA SET A
      READ(105,10)KPNT,KELM,NGEO,NMAT,NFREE,NFOR,NPART
C
      WRITE(108,20)KPNT,KELM,NGEO,NMAT,NFREE,NFOR,NPART
      WRITE(108,41)
C
C**** READ DATA SET B
      DO 30 I=1,KPNT
      READ(105,40)(X(I,J),J=1,2)
      WRITE(108,50)I,(X(I,J),J=1,2)
   30 CONTINUE
C
      WRITE(108,79)
C
C**** READ DATA SET C
      DO 60 J=1,NMAT
      READ(105,70)EMOD(J),P(J)
      WRITE(108,80)J,EMOD(J),P(J)
   60 CONTINUE
      REWIND 3
      WRITE(108,109)
C
C**** READ DATA SET D
      DO 90 NX=1,KELM
      READ(105,100)(NOD(J),J=1,8),NEP
      WRITE(108,110)NX,(NOD(J),J=1,8),NEP
      DO 120 I=1,8
      JJ=NOD(I)
      DO 120 IX=1,NFREE
  120 XE(I,IX)=X(JJ,IX)
C
C**** COMPUTE ELEMENT STIFFNESS MATRIX
C
      CALL THES03(APJ,RFUN,NX,EMOD(NEP),P(NEP),XE,NOD)
   90 CONTINUE
C
   10 FORMAT(7I4)
   20 FORMAT(1X,39HTOTAL NUMBER OF NODAL POINTS...........,I4/
     11X,39HTOTAL NUMBER OF ELEMENTS...............,I4/
     21X,39HNUMBER OF GEOMETRIC CONSTRAINTS........,I4/
     31X,39HNUMBER OF DIFFERENT MATERIALS..........,I4/
     41X,39HDEGREES OF FREEDOM PER NODE............,I4/
     51X,39HNUMBER OF NODES SUBJECTED TO FORCE.....,I4/
     61X,39HNUMBER OF PARTITIONS...................,I4//)
C
   40 FORMAT(2F16.8)
   41 FORMAT(10X,4HNODE,10X,13HR CO-ORDINATE,10X,13HZ CO-ORDINATE/)
   50 FORMAT(10X,I4,7X,F16.4,7X,F16.4)
   70 FORMAT(2F16.4)
   79 FORMAT(1X,8HMATERIAL,2X,21HMODULUS OF ELASTICITY,2X,14HPOISSONS R
     1TIO/)
   80 FORMAT(1X,4X,I4,7X,F16.4,2X,F14.4)
  100 FORMAT(9I4)
  109 FORMAT(1X,7HELEMENT,9X,14HELEMENT  NODES,9X,8HMATERIAL/)
  110 FORMAT(1X,3X,I4,8I4,2X,I2)
C
C
C**** READ DATA SET E
C
      DO 2000 J=1,NPART
        READ(105,2001)NSTART(J),NEND(J),NFIRST(J),NLAST(J)
 2001 FORMAT(4I4)
```

```
      WRITE(108,2002)NSTART(J),NEND(J),NFIRST(J),NLAST(J)
 2002 FORMAT(10X,4(I4,10X))
 2000 CONTINUE
C
      RETURN
      END
```

```
C**** SUBROUTINE THES03 SEPTEMBER 18,1973 F. X. JANUCIK
      SUBROUTINE THES03(APJ,RFUN,NKO,E,P1,X,NODE)
C
      COMMON NPART,KPNT,KELM,NGEO,NMAT,NFREE,NFOR
     1,NSTART(45),NEND(45),NFIRST(45),NLAST(45)
C
      DIMENSION RFUN(9,8),APJ(2,8,9),CE(16,16),B(4,16),SCR(2,8),NODE(8
     1,X(8,2),AJO(2,2),C(9),D(4,4),AINT(4,16,9),CIN(16,16)
     1,RR(1,1),CFUN(1,8)
C
C**** DEFINE GAUSS QUADRATURE WEIGHT COEFFICIENTS
C
      W1=0.55555556
      W2=0.88888889
C
      C(1)=W1**2
      C(2)=W1*W2
      C(3)=W1**2
      C(4)=W1*W2
      C(5)=W2**2
      C(6)=W1*W2
      C(7)=W1**2
      C(8)=W1*W2
      C(9)=W1**2
C
C**** INITIALIZE ELEMENT STIFFNESS MATRIX CE TO ZERO
C
      CALL ZEROM(CE,16,16)
C
C**** CALCULATE ELASTICITY MATRIX D
C
C
C**** SET ELEMENT ELASTICITY MATRIX TO ZERO
C
      CALL ZEROM(D,4,4)
      EC1=P1/(1.-P1)
      EC2=(1.-2.*P1)/(2.*(1.-P1))
      EC3=E*(1.-P1)/((1.+P1)*(1.-2.*P1))
      EC4=EC1*EC3
C
      D(1,1)=EC3
      D(1,2)=EC4
      D(1,3)=EC4
C
      D(2,1)=D(1,2)
      D(2,2)=EC3
      D(2,3)=EC4
C
      D(3,1)=D(1,3)
      D(3,2)=D(2,3)
      D(3,3)=EC3
C
      D(4,4)=EC2*EC3
C
C**** CALCULATE ELEMENT STIFFNESS MATRIX
C
      DO 100 NGP=1,9
C**** CALCULATE JACOBIAN
      CALL MATM(APJ(1,1,NGP),X,AJO,2,8,2)
      DO 30 J=1,8
      CFUN(1,J)=RFUN(NGP,J)
   30 CONTINUE
C**** CALCULATE RADIAL DISTANCE TO GAUSS SAMPLING POINT
      CALL MATM(CFUN,X,RR,1,8,1)
C**** CALCULATE JACOBIAN INVERSE AND THE VALUE OF ITS DETERMINANT
      CALL MTINVB(AJO,2,AREA)
C**** CALCULATE VOLUME ASSOCIATED WITH SAMPLING POINT NGP
      VOL=6.283185*C(NGP)*RR(1,1)*AREA
      CALL MATM(AJO,APJ(1,1,NGP),SCR,2,2,8)
C**** INITIALIZE THE B MATRIX TO ZERO
      CALL ZEROM(B,4,16)
C**** CALCULATE THE B MATRIX AT SAMPLING POINT NGP
      DO 200 I=1,8
      K=2*I
      J=2*I-1
      B(1,J)=SCR(1,I)
      B(2,J)=RFUN(NGP,I)/RR(1,1)
      B(3,K)=SCR(2,I)
      B(4,J)=SCR(2,I)
      B(4,K)=SCR(1,I)
```

```
 200 CONTINUE
C**** INITIALIZE  MATRIX CIN TO ZERO
      CALL ZEROM(CIN,16,16)
C
C**** CALCULATE STIFFNESS CONTRIBUTION OF SAMPLING POINT NGP  MATRIX CIN
C
C**** MULTIPLY MATRIX B TIMES D TO GET STRESS MATRIX AINT AT POINT NGP
      CALL MATM(D,B,AINT(1,1,NGP),4,4,16)
C**** MULTIPLY MATRIX AINT TIMES THE B MATRIX TRANSPOSED TO GET CIN AT POINT NGP
      CALL MTTSYM(B,AINT(1,1,NGP),CIN,16,4,16)
      DO 400 J=1,16
      DO 400 K=1,16
      CE(J,K)=CE(J,K)+VOL*CIN(J,K)
 400  CONTINUE
 100  CONTINUE
C
C**** WRITE THE ELEMENT STIFFNESS MATRIX, NODES, AND NUMBER TO TEMPORARY FILE 3
      WRITE(3)((CE(J,I),J=1,16),I=1,16),(NODE(I),I=1,8),NKO
C
C**** WRITE THE STRESS MATRICES AT THE NINE SAMPLING POINTS OF EACH ELEMENT TO 4
      WRITE(4)(((AINT(I,J,K),I=1,4),J=1,16),K=1,9)
     1,(NODE(L),L=1,8)
      RETURN
      END
```

```
      SUBROUTINE MTTSYM(D,B,DB,L,M,N)
      DIMENSION D(M,L),B(M,N),DB(L,N)
      DO 110 J=1,N
      DO 110 I=1,L
      IF(I.LT.J) DB(I,J)=DB(J,I)
      IF(I.LT.J) GO TO 110
      DB(I,J)=0.00000000
      DO 120 K=1,M
 120  DB(I,J)=DB(I,J)+D(K,I)*B(K,J)
 110  CONTINUE
      RETURN
      END
```

```
C              MATRIX INVERSION WITH VALUE OF DETERMINANT 6/9/71
           SUBROUTINE MTINVB(A,N,DETERM)
C    A IS MATRIX BEING INVERTED
C    N IS MATRIX SIZE
           DIMENSION IPIVOT(9),A(N,N),INDEX(9,2),PIVOT(9)
C  * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *
C          INITIALIZATION
C  * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *
       10  DETERM=1.0
       15  DO 20 J=1,N
       20  IPIVOT(J)=0
       30  DO 550 I=1,N
C  * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *
C          SEARCH FOR PIVOT ELEMENT
C  * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *
       40  AMAX=0.0
       45  DO 105 J=1,N
       50  IF  (IPIVOT(J)-1)60,105,60
       60  DO  100 K=1,N
       70  IF  (IPIVOT(K)-1)80,100,740
       80  IF  (ABS(AMAX)-ABS(A(J,K)))85,100,100
       85  IROW=J
       90  ICOLUM=K
       95  AMAX=A(J,K)
      100  CONTINUE
      105  CONTINUE
      110  IPIVOT(ICOLUM)=IPIVOT(ICOLUM)+1
C  * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *
C          INTERCHANGE ROWS TO PUT PIVOT ELEMENT ON DIAGONAL
C  * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *
      130  IF  (IROW-ICOLUM)140,260,140
      140  DETERM=-DETERM
      150  DO 200 L=1,N
      160  SWAP=A(IROW,L)
      170  A(IROW,L)=A(ICOLUM,L)
      200  A(ICOLUM,L)=SWAP
      260  INDEX(I,1)=IROW
      270  INDEX(I,2)=ICOLUM
      310  PIVOT(I)=A(ICOLUM,ICOLUM)
      320  IF(N.LT.4)   DETERM=DETERM*PIVOT(I)
C  * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *
C          DIVIDE PIVOT ROW BY PIVOT ELEMENT
C  * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *
      330  A(ICOLUM,ICOLUM)=1.0
      340  DO 350 L=1,N
      350  A(ICOLUM,L)=A(ICOLUM,L)/PIVOT(I)
C  * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *
C          REDUCE NON-PIVOT ROWS
C  * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *
      380  DO 550 L1=1,N
      390  IF(L1-ICOLUM)400,550,400
      400  T=A(L1,ICOLUM)
           IF(T.EQ.0.0) GO TO 451
      420  A(L1,ICOLUM)=0.0
      430  DO 450 L=1,N
           A(L1,L)=A(L1,L)-A(ICOLUM,L)*T
     450   CONTINUE
     451   CONTINUE
      550  CONTINUE
C  * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *
C          INTERCHANGE COLUMNS
C  * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *
      600  DO 710 I=1,N
      610  L=N+1-I
      620  IF  (INDEX(L,1)-INDEX(L,2))630,710,630
      630  JROW=INDEX(L,1)
      640  JCOLUM=INDEX(L,2)
      650  DO 705 K=1,N
      660  SWAP=A(K,JROW)
      670  A(K,JROW)=A(K,JCOLUM)
      700  A(K,JCOLUM)=SWAP
      705  CONTINUE
      710  CONTINUE
      740  RETURN
           END
```

```
      SUBROUTINE THES04
C
C**** EXTRAPOLATES STRESS MATRICES AT SAMPLING POINTS TO ELEMENT MIDSIDE NOD
C
      COMMON NPART,KPNT,KELM,NGEO,NMAT,NFREE,NFOR
     1,NSTART(45),NEND(45),NFIRST(45),NLAST(45)
C
      DIMENSION STR(4,16,4),NA(4),NODE(8),DB(4,16,9)
C
      REWIND 4
C
C**** DEFINE WEIGHT COEFFICIENTS
      AA=1.47883056
      AB=-0.66666667
      AC=0.18783611
      DO 1 JOB=1,KELM
C**** READ  ELEMENT STRESS MATRICES AT GAUSS SAMPLING POINTS
C**** AND ALSO ELEMENT NODAL POINT NUMBERS
      READ(4)(((DB(I,J,K),I=1,4),J=1,16),K=1,9)
     1,(NODE(L),L=1,8)
C
C**** INITIALIZE MIDSIDE NODE STRESS MATRICES TO ZERO
      DO 2 I1=1,16
      DO 2 J1=1,4
      DO 2 K1=1,4
    2 STR(J1,I1,K1)=0.00000000
C
C**** CALCULATE STRESS MATRICES AT MIDSIDE NODES
C
      DO 3 I2=1,4
      DO 3 J2=1,16
      STR(I2,J2,1)=AC*DB(I2,J2,4)+AB*DB(I2,J2,5)+AA*DB(I2,J2,6)
      STR(I2,J2,2)=AA*DB(I2,J2,2)+AB*DB(I2,J2,5)+AC*DB(I2,J2,8)
      STR(I2,J2,3)=AA*DB(I2,J2,4)+AB*DB(I2,J2,5)+AC*DB(I2,J2,6)
      STR(I2,J2,4)=AC*DB(I2,J2,2)+AB*DB(I2,J2,5)+AA*DB(I2,J2,8)
    3 CONTINUE
C**** DEFINE MIDSIDE NODES ASSOCIATED WITH CALCULATED STRESS MATRICES
      DO 4 J6=1,4
      J7=2*J6
      NA(J6)=NODE(J7)
    4 CONTINUE
C
C**** WRITE STRESS MATRICES AND THE NODE ASSOCIATED WITH EACH AND ALSO ELEME
C**** NODES TO TEMPORARY FILE 2
C
      WRITE(2)(((STR(I,J,K),I=1,4),J=1,16),K=1,4)
     1,(NA(L),L=1,4),(NODE(L),L=1,8)
    1 CONTINUE
      RETURN
      END
```

```
      SUBROUTINE MATRIX
C**** ASSEMBLES THE STRUCTURAL STIFFNESS MATRIX IN BLOCK FORM
C**** READS IN THE STRUCTURAL GEOMETRIC CONSTRAINTS AND LOADING CONDITIO
      COMMON NPART,KPNT,KELM,NGEO,NMAT,NFREE,NFOR
     1,NSTART(45),NEND(45),NFIRST(45),NLAST(45)
C
      DIMENSION NF(600),NB(600,2),BV(600,2),UU(1200),U(2,600)
     1,NODE(8),C(16,16),ST(40,80)
C
      REWIND 4
      NFF=1
   52 CONTINUE
C**** READ GEOMETRIC CONSTRAINTS
C**** READ DATA SET F
      DO 50 I=NFF,NGEO
      READ(105,46) NF(I),(NB(I,J),J=1,2),(BV(I,J),J=1,2)
   46 FORMAT(3I4,2F16.8)
   50 WRITE(108,46) NF(I),(NB(I,J),J=1,2),(BV(I,J),J=1,2)
      CALL ZEROM(U,2,600)
      IF  (NFOR.EQ.0)  GO TO 6900
C**** READ NODAL POINT FORCES
C**** READ DATA SET G
      DO 69 I=1,NFOR
      READ(105,37)K,U(1,K),U(2,K)
   69 WRITE(108,37)K,U(1,K),U(2,K)
 6900 CONTINUE
   37 FORMAT(I4,3F16.4)
      INTER = 0
C**** CALCULATE STRUCTURAL STIFFNESS MATRIX ON BLOCK FORM
      DO 70 II=1,NPART
      REWIND 3
      CALL ZEROM (ST,40,80)
  975 CONTINUE
      NST=NSTART(II)
      NEN=NEND(II)
      K=NFIRST(II)
      L=NLAST(II)
      IF(II.NE.NPART)  KEND=NLAST(II+1)
      IF(II.EQ.NPART)KEND=NLAST(II)
      MINUS = K-1
      LMINUS=2*(L-MINUS)
      JNJ=0
      DO 981 J=K,L
      DO 981 I=1,NFREE
      JNJ=JNJ+1
  981 UU(JNJ)=U(I,J)
      DO 80 LK=1,KELM
      MM=LK-INTER
  162 READ(3) ((C(J,I),J=1,16),I=1,16),(NODE(I),I=1,8),NL
      DO 8210 I=1,8
      JNJ=NODE(I)
      IF (JNJ.LT.K) GO TO 8210
      IF (JNJ.GT.L) GO TO 8210
      JNJ=2*(JNJ-K)
      UU(JNJ+1)=UU(JNJ+1)
      DO 821 J=1,NFREE
      JNJ=JNJ+1
  821  UU(JNJ)=UU(JNJ)
 8210 CONTINUE
      IF(NL.LT.NST) GO TO 80
      IF(NL.GT.NEN) GO TO 80
  884 CONTINUE
      DO 800 LL=1,8
      DO 800 KK=1,8
      IF (NODE(KK)-K) 800,131,131
  131 IF (NODE(KK)-L) 132,132,800
  132 M=NFREE*(NODE(KK)-K)
      N=NFREE*(NODE(LL)-K)
      I=NFREE*(KK-1)
      J=NFREE*(LL-1)
      IF (N) 800,900,900
  900 DO 5 NJ=1,NFREE
      DO 5 MI=1,NFREE
      MMI=M+MI
      NNJ=N+NJ
      IMI=I+MI
      JNJ=J+NJ
    5 ST(MMI,NNJ) = ST(MMI,NNJ) + C(IMI,JNJ)
  800 CONTINUE
   80 CONTINUE
```

```
C * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *
C      INTRODUCTION OF PRESCRIBED DISPLACEMENTS
C * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *
      DO 290 I=1,NGEO
      M=NF(I)-K
      MM=NF(I)-1
      KKEND=KEND-NF(I)
      IF (M) 290,242,242
  242 IF(KKEND) 290,243,243
  243 DO 230 J=1,NFREE
      IF (NB(I,J)) 230,345,230
  345 NMI = NFREE*M+J
      LLEAR=NFREE*(L-K+1)
      DO 1345 KLEAR=1,LLEAR
      JNJ=KLEAR
      UU(JNJ)=UU(JNJ)-ST(KLEAR,NMI)*BV(I,J)
  233 IF(KLEAR.EQ.NMI)UU(JNJ)=BV(I,J)
      ST(KLEAR,NMI)=0.0
      IF(KLEAR.NE.NMI) GO TO 2345
      IF (II.EQ.NPART)GO TO 3344
      KKEND=KEND-1
      DO 3343 KB=L,KKEND
      NNI=KB+1
      DO 3343 JNJ=1,NFREE
      NMJ=NFREE*(NNI-K)+JNJ
      U(JNJ,NNI)=U(JNJ,NNI)-ST(NMI,NMJ)*BV(I,J)
 3343 CONTINUE
 3344 CONTINUE
      LLR=(KEND-K+1)*NFREE
      DO 3345 KKL=1,LLR
      ST(NMI,KKL)=0.0
 3345 CONTINUE
      ST(NMI,NMI)=1.0
 2345 CONTINUE
 1345 CONTINUE
  230 CONTINUE
  290 CONTINUE
      INTER=NEN
      MI=NFREE*MINUS+1
      NJ=NFREE*L
      M=NJ-MI+1
      IF (II-NPART)115,116,115
  115 NA=NFREE*(NLAST(II+1)-MINUS)
      GO TO 117
  116 NA=M+1
  117 N=NA-M
      MM=M+1
   70 WRITE(4)M,N,((ST(I,J),I=1,M),J=1,M),((ST(I,J),I=1,M),J=MM,NA),
     1(UU(I),I=1,M)
      RETURN
      END
```

```
      SUBROUTINE SOLVE
C
C**** SOLVES THE STRUCTURAL EQUILIBRUM RELATIONS FOR OISPLACEMENTS USING
C**** TRIOIAGONAL METHOO OF FORWARD E-IMINATION AND BACK SUBSTITUTION
C
      COMMON NPART,KPNT,KELM,NGEO,NMAT,NFREE,NFOR
     1,NSTART(45),NENO(45),NFIRST(45),NLAST(45)
C
      DIMENSION AM(40,40),BM(40,40),YM(40),TF(40),F(40),DIS(40),
     1  DBA(4,16,4),NA(4),NODE(8)
C
      DO 144 LL=1,NPART
      READ(4) M,N,((AM(I,J),I=1,M),J=1,M),((BM(I,J),I=1,M),J=1,N),
     1(F(I),I=1,M)
  150 DO 426 I=1,M
      IF (LL.EQ.1) GO TO 426
      F(I)=F(I)-TF(I)
  425 CONTINUE
      READ (3) (YM(J),J=1,M)
      DO 424 J=1,M
  424 AM(I,J)=AM(I,J)-YM(J)
 426  CONTINUE
      CALL MTINVC(AM,M,40)
C * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *
C     MATRIX INVERSION PROGRAM
C * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *
      WRITE(2) M,N,((AM(I,J),I=1,M),J=1,M),((BM(I,J),I=1,M),J=1,N),
     1(F(I),I=1,M)
      CALL MATMS(AM,F,DIS,M,M,40)
      IF (NPART-LL) 437,437,432
 432  CALL MATTMS(BM,DIS,TF,N,M,40)
      REWIND 3
      DO 20 J=1,N
      DO 25 I=1,M
 25   YM(I)=0.0
      DO 30 I=1,M
      BB=BM(I,J)
      IF (BB.EQ.0.0)  GO TO 30
      DO 40 K=1,M
 40   YM(K)=YM(K)+AM(K,I)*BB
 30   CONTINUE
 20   WRITE (3) (YM(I),I=1,M)
      REWIND 3
      DO 50 J=1,N
 50   READ (3) (AM(I,J),I=1,M)
      REWIND 3
      DO 60 J=1,N
      DO 65 I=1,N
 65   YM(I)=0.0
      DO 70 I=1,M
      BB=BM(I,J)
      IF (BB.EQ.0.0)  GO TO 70
      DO 80 K=1,N
 80   YM(K)=YM(K)+BB*AM(I,K)
 70   CONTINUE
 60   WRITE (3) (YM(I),I=1,N)
      REWIND 3
 144  CONTINUE
 437  REWIND 3
      WRITE (3) (DIS(I),I=1,M)
      JZ4=4*KELM
      IF (NPART-1) 600,600,601
 601  DO 441 LL=2,NPART
      REWINO 2
      DO 200 K=1,KELM
  200 READ(2) (((DBA(I,J,L),I=1,4),J=1,16),L=1,4),(NA(MO),MO=1,4)
     1,(NODE(MXA),MXA=1,8)
      DO 442 K=LL,NPART
 442  READ (2) M,N,((AM(I,J),I=1,M),J=1,M),((BM(I,J),I=1,M),J=1,N),
     1(F(I),I=1,M)
      CALL MATMS(BM,DIS,TF,M,N,40)
      DO 444 I=1,M
  444 F(I)=F(I)-TF(I)
      CALL MATMS(AM,F,DIS,M,M,40)
  441 WRITE (3) (OIS(I),I=1,M)
  600 CONTINUE
      RETURN
      END
```

```
C   MATRIX INVERSION, MODIFIED 2/4/72 BY S. LEVY
      SUBROUTINE MTINVC(A,N,NSIZE)
C   A IS MATRIX BEING INVERTED
C   N IS MATRIX SIZE
C   NSIZE IS MEMORY SIZE
      DIMENSION A(40     ,40    ),IPIVOT( 40),INDEX( 40,2),PIVOT( 40)
C * * * * * * * * * *
C     INITIALIZATION
C * * * * * * * * *
   15 DO 20 J=1,N
   20 IPIVOT(J)=0
   30 DO 550 I=1,N
C * * * * * * *
C     SEARCH FOR PIVOT ELEMENT
C * * * * * * * * *
   40 AMAX=0.0
   45 DO 105 J=1,N
   50 IF (IPIVOT(J)-1)60,105,60
   60 DO 100 K=1,N
   70 IF (IPIVOT(K)-1)80,100,740
   80 IF (ABS(AMAX)-ABS(A(J,K)))85,100,100
   85 IROW=J
   90 ICOLUM=K
   95 AMAX=A(J,K)
  100 CONTINUE
  105 CONTINUE
  110 IPIVOT(ICOLUM)=IPIVOT(ICOLUM)+1
C * * * * * * *
C     INTERCHANGE ROWS TO PUT PIVOT ELEMENT ON DIAGONAL
C * * * * * * *
  130 IF (IROW-ICOLUM)140,260,140
  140 CONTINUE
  150 DO 200 L=1,N
  160 SWAP=A(IROW,L)
  170 A(IROW,L)=A(ICOLUM,L)
  200 A(ICOLUM,L)=SWAP
  260 INDEX(I,1)=IROW
  270 INDEX(I,2)=ICOLUM
  310 PIVOT(I)=A(ICOLUM,ICOLUM)
  320 CONTINUE
C * * * * * * * *
C     DIVIDE PIVOT ROW BY PIVOT ELEMENT
C * * * * * * *
  330 A(ICOLUM,ICOLUM)=1.0
  340 DO 350 L=1,N
  350 A(ICOLUM,L)=A(ICOLUM,L)/PIVOT(I)
C * * * * * * * * * *
C     REDUCE NON-PIVOT ROWS
C * * * * * * *
  380 DO 550 L1=1,N
  390 IF(L1-ICOLUM)400,550,400
  400 T=A(L1,ICOLUM)
  420 A(L1,ICOLUM)=0.0
  430 DO 450 L=1,N
  450 A(L1,L)=A(L1,L)-A(ICOLUM,L)*T
  550 CONTINUE
C * * * * * * *
C     INTERCHANGE COLUMNS
C * * * * * * *
  600 DO 710 I=1,N
  610 L=N+1-I
  620 IF (INDEX(L,1)-INDEX(L,2))630,710,630
  630 JROW=INDEX(L,1)
  640 JCOLUM=INDEX(L,2)
  650 DO 705 K=1,N
  660 SWAP=A(K,JROW)
  670 A(K,JROW)=A(K,JCOLUM)
  700 A(K,JCOLUM)=SWAP
  705 CONTINUE
  710 CONTINUE
  740 RETURN
  750 END
```

```
C   MATRIX MULTIPLICATION TRANSPOSED DB(L)=D(MXL)*B(M)
         SUBROUTINE MATTMS(D,B,DB,L,M,NSIZE)
         DIMENSION D(40    ,40    ),B(NSIZE),DB(NSIZE)
C   NSIZE IS MEMORY SIZE
         DO 110 I=1,L
         DB(I)=0.
         DO 110 K=1,M
  110    DB(I)=DB(I)+D(K,I)*B(K)
         RETURN
         END
```

```
C   MATRIX MULTIPLICATION    DB(L)=D(LXM)*B(M)
         SUBROUTINE MATMS(D,B,DB,L,M,NSIZE)
         DIMENSION D(40    ,40    ),B(NSIZE),DB(NSIZE)
C   NSIZE IS MEMORY SIZE
         DO 110 I=1,L
         DB(I)=0.
         DO 110 K=1,M
  110    DB(I)=DB(I)+D(I,K)*B(K)
         RETURN
         END
```

```
C**** SUBROUTINE STRESS
C**** WRITES NODAL POINT DISPLACEMENTS AND ELEMENT STRESSES
      SUBROUTINE STRESS
      COMMON NPART,KPNT,KELM,NGEO,NMAT,NFREE,NFOR
     1,NSTART(45),NEND(45),NFIRST(45),NLAST(45)
C
      DIMENSION U(1200),STR(4,16,4),NODE(8),NA(4),
     1DEF(16),SIG(4),DIRCOS(3,3),PRIN(3)
C
      DO 600 II=1,NPART
      JJ=NPART+1-II
      M=NFREE*(NFIRST(JJ)-1)+1
      N=NFREE*NLAST(JJ)
  600 READ(3)(U(I),I=M,N)
C**** WRITE NODAL POINT DISPLACEMENTS
      WRITE(108,1)
    1 FORMAT(1X,33H* * * *DISPLACEMENTS* * * * * * *,//
     2        1X,33HNODE      RADIAL(R)         AXIAL(Z),/
     3        1X,33H----      ---------         --------,/)
       WRITE(108,2)((I,U(2*I-1),U(2*I)),I=1,KPNT)
    2 FORMAT(1X,I4,2E16.8)
C**** MUST PUT IN HEADING FOR STRESSES
      REWIND 2
      LL=1
   21 CONTINUE
C**** READ ELEMENT STRESS MARRICES AT MIDSIDE NODES
      READ(2)(((STR(I,J,K),I=1,4),J=1,16),K=1,4)
     2,(NA(L),L=1,4),(NODE(L),L=1,8)
C**** DEFINE VECTOR DEF OF ELEMENT DISPLACEMENTS
      DO 620 I=1,8
      JJ=NODE(I)
      DO 620 IJ=1,NFREE
      I3=NFREE*(I-1)+IJ
      J3=NFREE*(JJ-1)+IJ
  620 DEF(I3)=U(J3)
      DO 30 NX=1,4
C**** MULTIPLY STRESS MATRIX STR TIMES DEFLECTIONS TO OBTAIN STRSS MATRIX SI
      CALL MATM(STR(1,1,NX),DEF,SIG,4,16,1)
C**** CALCULATE PRINCIPAL STRESSES AND DIRECTION COSINES
      CALL PRINPL(DIRCOS,PRIN,SIG)
C**** WRITE ELEMENT NUMBER,ELEMENT NODES, AND MIDSIDE NODE NUMBER AT WHICH
C**** STRESSES WERE CALCULATED
      WRITE(108,5)LL,(NODE(I),I=1,8),NA(NX)
C**** WRITE THE FOUR COMPOMENTS OF STRESS AT NODE NA
      WRITE(108,6)(SIG(J),J=1,4)
C**** WRITE PRINCIPAL STRESSES AND THEIR DIRECTION COSINES
      WRITE(108,7)(PRIN(I),(DIRCOS(I,IJ),IJ=1,3),I=1,3)
   30 CONTINUE
      IF(LL-KELM)22,100,100
   22 LL=LL+1
      GO TO 21
  100 CONTINUE
C
    5 FORMAT(1X,2HEL,I4,2X,5HNODES,8I4,5HSTRPT,I4)
    6 FORMAT(1X,4HSIGR,F10.1,8HSIGTHETA,F10.1,4HSIGZ,F10.1,
     15HTAURZ,F10.1)
    7 FORMAT(1X,2HPS,F10.1,1HL,F9.6,1HM,F9.6,1HN,F9.6)
      RETURN
      END
```

```
C**** SUBROUTINE PRINPL
      SUBROUTINE PRINPL(DIRCOS,PRIN,SIG)
C
      DIMENSION SIG(4),DIRCOS(3,3),PRIN(3)
C
      CALL ZEROM(DIRCOS,3,3)
      PRIN(3)=SIG(2)
      DIRCOS(3,3)=1.0000
      SS=0.5*(SIG(1)+SIG(3))
      SQ=SQRT((SIG(1)-SIG(3))**2/4.00+SIG(4)**2)
      PRIN(1)=SS+SQ
      PRIN(2)=SS-SQ
      IF(SIG(4).NE.0.0)GO TO 10
      DIRCOS(1,1)=1.0000
      DIRCOS(2,2)=1.0000
      GO TO 20
   10 DO 15 J=1,2
      SQ=SQRT((PRIN(J)-SIG(1))**2+SIG(4)**2)
      DIRCOS(J,1)=SIG(4)/SQ
   15 DIRCOS(J,2)=-(PRIN(J)-SIG(1))/SQ
   20 RETURN
      END
```