A STUDY OF THE EFFECT OF ELEMENT SHAPE

ON

THE ACCURACY OF VARIOUS 2D FINITE ELEMENT FORMULATIONS

By

© KIRAN KUPPANDA SUBBAYYA

A Thesis

Submitted to the School of Graduate Studies

in Partial Fulfilment of the Requirements

for the Degree

Master of Engineering

McMaster University

September 1980

ACCURACY OF 2D FINITE ELEMENTS

MASTER OF ENGINEERING (1980) McMASTER UNIVERSITY

(Mechanical) Hamilton, Ontario.

TITLE: A Study of the Effect of Element Shape on the Accuracy of Various 2D Finite Element Formulations

AUTHOR: Kiran Kuppanda Subbayya, B.E.(Mysore University, India.)

SUPERVISOR: Professor D.S. Weaver

NUMBER OF PAGES: x, 185

ABSTRACT

Linear Isoparametric elements and elements based on the Constant Strain Triangle are developed and incorporated into a computer program, for two dimensional stress analysis on mini computers. These elements are compared at element level using the Weighted Eigenvalue method and in assemblage using sensitivity analysis. Further, a method of predicting results based on the data generated during the comparisons at the element level is also presented. As an application of the computer program, analysis of a porcelain insulator is also included.

ACKNOWLEDGEMENT

The Author wishes to acknowledge and express his sincere appreciation to his supervisor, Dr. D.S. Weaver, to Monserco Limited and to Dilworth, Secord, Meagher and Associates Ltd. for the cooperation extended during the preparation of this thesis.

The Author is especially grateful to Mr. Rodney P. Bell, Vice-president, Monserco Limited for the authorization granted.

Special thanks to Mrs. Helen Hill, for assisting in the preparation of the original manuscript.

TABLE OF CONTENTS

TABLE OF CONTENTS (CONTD.)

LIST OF TABLES AND FIGURES

TABLES

FIGURES

LIST OF TABLES AND FIGURES (CONTD.)

FIGURES (CONTD.)

LIST OF TABLES AND FIGURES (CONTD.)

FIGURES (CONTD.)

LIST OF TABLES AND FIGURES (CONTD.)

FIGURES (CONTD.)

1. INTRODUCTION

In the engineering industry, Finite Element (FE) Techniques are fast becoming everyday tools. Today, the stress analyst has at his disposal[22] numerous proprietary computer programs: NASTRAN, STARDYNE, ANSIS, SAPVI, to cite a few. In industry, these programs are used as 'Black Boxes' which tends to make the analyst accept the results without any question. Further, these proprietary programs require sizeable computer core (memory), dictating the use of a large expensive computing system. In practice, any one FE model ends up being analyzed more than once or probably twice. This is due to the incorrect assumptions made, changes in design and error in input data. With the sophisticated diagnostics available on these large systems, error in input data can be minimized. However, the first two events cannot be foreseen till the analysis is complete. Reanalysis increases the cost of analysis, not only due to the additional computing time required but also due to the multiple royalties paid.

To be competitive in the market, the analyst must be capable of carrying out analysis economically. He also must be willing to use the data as made available and provide results in a format suitable to the client. This requires extensive pre and post processing of data which is not easily and economically accomplished using the 'black box' programs on large computing systems.

From the above reasoning, it is clear that any analyst using the FE techniques is required to make two major decisions:

(a) with regards to the required accuracy of results

(b) with regards to the cost of analysis

In fact, each of these decisions affect the other. The accuracy of the analysis is dependent on the basic element formulation used, the mesh size and the aspect ratio of the elements. The type of element formulation and the mesh size also govern the cost of analysis.

Recently, with the advent of relatively inexpensive mini computers, it is now feasible to analyze sizeable FE models economically using specially tailored programs; however, using such a set-up requires the analyst to discard the 'black box' approach and have an insight to the FE techniques.

In this thesis, a general purpose two diminsional stress analysis program (KAXI) is developed for use on a mini computer and verified against classical solutions. For ease of understanding and programming, the commonly available element formulations in two dimensions are derived. Comparisons are then carried out using these various element formulations for accuracy as a function of aspect ratio. The comparisons are carried out using rectangular elements. These comparisons should give the reader an insight to the selection of element type for analysis. Finally, two representative applications of the program are described.

Implementation of such a computer program provides full control over program modifications, pre and post processing of data to suit the needs. Though analysis using mini computers looks very promising, the concept does have its drawbacks. Due to the limited core memory, the size of the analytical model is limited and due to extensive use of secondary storage, the turn around time is comparatively slower. However, the program KAXI was developed and installed on a PDP 11/34 mini computer with 29 k words accessible memory. The size limitations for this set-up is 500 nodal points, 500 elements and 10 load cases.

The contents of this work are grouped as follows:

Chapter 1: Introduction to the work

Chapter 2: Background; outlines the FE method as applied to stress analysis. The elements, two quadrilateral elements based on the constant strain triangle (CST) formulation and three quadrilateral elements based on the isoparametric (ISO) formulations, are developed in steps for ease of understanding and programming.

Chapter 3: Program KAXI, describes the program developed on the mini computer, provides a list of subroutines used, the capabilities and limitations of the program and its verifications. A computer listing of the program is included as an appendix.

Chapter 4: Comparison of element formulations, describes the comparisons carried out in this work. The comparisons are carried out at two levels:

(a) at element level, using single element with varying aspect ratios under various modes of loading.

(b) in an assemblage, using sensitivity analysis to measure accuracy as a function of mesh size.

Chapter 5: A proposed method of predicting results; describes an attempt made at predicting results in an assemblage of isoparametric elements, based on the results of comparison at the element level.

Chapter 6: Representative applications of the program KAXI, describes the analysis carried out on a porcelain composite insulator as an axisymmetric case and as a plane stress case and the attempt to correlate the two results. It also describes the results of a tube sheet analysis to study the effect of using the different elèments.

Chapter 7: Conclusions, as the name implies describes the conclusions arrived at during this work and directs the reader towards the additional work that needs to be done for better understanding of the effects of element shape.

Appendix 1: Describes the eigenvalue and eigenvector extraction using a generalized Jacobi method.

Appendix 2: Describes the development of general stress-strain relationship (matrix) for two dimensional stress analysis.

Appendix 3: Contains the user manual for the program KAXI, its computer listing and outputs from program verifications.

2. BACKGROUND

2.1 Outline of FE Technique

It is not possible to obtain a closed form solution for all mathematical models of engineering problems. Analytical solutions can be obtained only for certain simplified situations. For problems involving complex shapes, material properties, loading and boundary conditions, the analyst has to resort to numerical techniques that give approximate solutions which are acceptable. Approximate solutions often involve replacing the continuum with a finite number of well-defined components, the process being called discretization. This discretization has been approached differently by mathematicians and engineers. The former have developed general techniques applicable directly to the differential equations governing the problem such as finite differnce approximations[1,2], various weighted residual procedures[3,4], or functionals. On the other hand, engineers generally have approached the problem more intuitively by creating an analogy between real discrete elements and finite portions of a continuum. The work of Newmark[5], Argyris[6] and Turner et al[7] done in the late '40s and early '50s stand out in this context. But Clough[8] appears to be the first to use the term "Finite Element" in a direct use of standard methodology applicable to discrete systems.

In structural analysis, the principle of minimum potential energy, i.e. "The potential energy of a loaded elastic structure as represented by the sum of the internal energy stored as a result of the deformations and the potential of the applied loading is minimum if the body is in equilibrium" is usually employed to obtain a set of equilibrium equations for each finite element. The equilibrium equations for the entire body representing the continuum are then obtained by combining the equations for each element in such a way that the continuity of displacements is preserved at the interconnecting nodes. These equations are modified for the given boundary conditions and then solved to obtain the unknown displacements and stresses.[5]

The total potential or the potential energy of a conservative elastic system is defined as

$$\Pi = U + W_P \tag{2.1}$$

where U is the strain energy

and W_P is the potential of the applied forces

Assuming the forces are constant during deformation, the variation of the work done by the load W can be related to the potential of the load as

$$\delta W = -\delta W_P \tag{2.2}$$

i.e. for a conservative system, the principle of minimum potential energy is

$$\delta\pi = \delta U - \delta W = 0 \qquad (2.3)$$

$$\text{and} \quad U = \int_{vol} \tfrac{1}{2}\{\varepsilon\}^T \{\sigma\}dv \qquad (2.4)$$

where $\{\varepsilon\}$ is the strain vector
and $\{\sigma\}$ is the stress vector

Introducing the stress-strain relationship $\{\sigma\} = [D]\{\varepsilon\}$ into equation (2.4), at the element level

$$U_e = \int_{v} \tfrac{1}{2} \{\varepsilon\}^T [D] \{\varepsilon\}dv \qquad (2.5)$$

where $[D]$ is the stress-strain matrix

Introducing the strain-displacement relationship of $\{\varepsilon\} = [B]\{\delta\}$ into equation (2.5),

$$U_e = \int_{v} \tfrac{1}{2} \{\delta\}^T [B]^T [D][B] \{\delta\}dv \qquad (2.6)$$

where $\{\delta\}$ is the displacement vector and $[B]$ the strain displacement matrix unique to a type of element formulation

Applying Castigliano's theorem, the element stiffness $[k]$ is given by

$$[k] = \int_{v} [B]^T [D] [B]dv \qquad (2.7)$$

The total stiffness assemblage for the structure can be written as:

$$[K]\ \{d\} = \{F\} \tag{2.8}$$

where $[K]$ = the total stiffness matrix given by $\sum_{i=1}^{n} k_i$, n being the number of elements in the structure

$\{d\}$ = the global displacement vector

$\{F\}$ = the global load vector

Knowing $[K]$ and the applied loads $\{F\}$, the set of simultaneous equations in (2.8) is solved to evaluate the displacement vector $\{d\}$.

Assuming that the material properties within an element do not change, the matrix $[D]$ defining the stress-strain relationship can be considered independent of the element formulation. The method of formulating the $[D]$ matrix for an orthotropic material in two dimensions is described in Appendix 2.

The matrix $[B]$ defining the strain-displacement relationship is a function of the element formulation. In the next two sections, details of setting up the $[B]$ matrix and evaluating the element stiffness matrix $[k]$ are described.

Most comparisons to evaluate the relative behaviour of the different element types has been carried out in plane stress conditions by many authors[9-12]. These comparisons are carried out in assemblage for a cantilever beam under bending and shear loading. Some excellent work with comparisons at a single element level under various loading conditions has been carried out by Melosh[13], Khanna[14], Clough[15] and Rigby and McNeice[16]. Melosh suggests the trace concept in which the element whose stiffness matrix has the lower trace is likely to be better for general loading cases. Clough uses this method to study three-dimensional elements. Khanna proposes a stiffness difference matrix method derived from strain energy principles. Rigby and McNeice propose a general strain energy based method called the weighted eigenvalue method. It is a complete method which includes the eigenvectors of the stiffness matrix and the applied loading on the element. Further it is independent of the element size and order of the element formulation. This method is used for comparison at element level as described in Section 4.2.

In this study, two families of elements are compared: isoparametric and constant strain. The elements based on the isoparametric formulation are:

(1) Linear plane isoparametric quadrilateral element, designated as Q4.

(2) Linear plane isoparametric quadrilateral element with two internal degrees of freedom, designated as Q5.

(3) Linear plane isoparametric quadrilateral

element with four internal degrees of freedom, designated as Q6.

The elements based on the formulation of constant strain triangle (CST) are:

(1) Linear plane constant strain weight averaged quadrilateral element made up of four CST.

(2) Linear plane constant strain quadrilateral element made up of four CST's with a common central node.

2.2 Isoparametric Element Formulation

To ensure that a minimum number of elements can represent a complex form occuring in real problems, we need elements of arbitrary shape such as a quadrilateral element. Elements of basic one, two, or three dimensional types can be mapped into distorted form (quadrilateral) using a one to one correspondence between cartesian and curvilinear coordinate systems for the element.

The family of elements using the element shape function for mapping is the so-called isoparametric family. This concept appears to have been first applied in FE analysis by Irons.[18] These elements are formulated using a 'natural' or 'intrinsic' coordinate system defined by the element geometry.

These elements satisfy the criteria of invariance and continuity of displacements within the element.[9] When the element shape is defined by the same function as the displacement, the rigid body criteria and the constant strain energy criteria are also satisfied.[10] The shape function $[N]$ defining a family of elements relates a unit square in the natural coordinate system (ξ,η), whose shape and size are defined by the nodal cartesian coordinates.

$$[N] = f(\xi,\eta) \tag{2.9}$$

By definition of isoparametric formulation, the coordinates are related as:

$$x = \sum_{i=1}^{n} N_i x_i \quad \text{and} \quad y = \sum_{i=1}^{n} N_i y_i$$

and the displacements are related as:

$$u = \sum_{i=1}^{n} N_i u_i \quad \text{and} \quad v = \sum_{i=1}^{n} N_i v_i$$

where i represents the node number

and n represents the number of nodes

In matrix form, the mapping[9] (shape[11]) function relationships can be written as:

$$\begin{Bmatrix} x \\ y \end{Bmatrix} = [N]\{X\} \quad \text{and} \quad \begin{Bmatrix} u \\ v \end{Bmatrix} = [N]\{d\} \tag{2.10}$$

where $\{X\}$ and $\{d\}$ denote the nodal coordinate and displacement vectors.

i.e. $\{X\}^T = \{x_1, y_1, x_2, y_2 \ldots\ldots\}$

and $\{d\}^T = \{u_1, v_1, u_2, v_2 \ldots\}$

To evaluate the strains, $\partial u/\partial x$, $\partial u/\partial y$, $\partial v/\partial x$ and $\partial v/\partial y$ are to be calculated.

From equation (2.9) and (2.10)

$$\frac{\partial\{u\}}{\partial x} = \frac{\partial [N]\{d\}}{\partial x} \quad \text{and} \quad [N] = f(\xi,\eta)$$

Using the chain rule,

$$\frac{\partial N_i}{\partial \xi} = \frac{\partial N_i}{\partial x}\frac{\partial x}{\partial \xi} + \frac{\partial N_i}{\partial y}\frac{\partial y}{\partial \xi}$$

and $$\frac{\partial N_i}{\partial \eta} = \frac{\partial N_i}{\partial x}\frac{\partial x}{\partial \eta} + \frac{\partial N_i}{\partial y}\frac{\partial y}{\partial \eta}$$

In matrix form,

$$\begin{Bmatrix} \frac{\partial N_i}{\partial \xi} \\ \frac{\partial N_i}{\partial \eta} \end{Bmatrix} = \begin{bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} \end{bmatrix} \begin{Bmatrix} \frac{\partial N_i}{\partial x} \\ \frac{\partial N_i}{\partial y} \end{Bmatrix}$$

Inverting,

$$\begin{Bmatrix} \frac{\partial N_i}{\partial x} \\ \frac{\partial N_i}{\partial y} \end{Bmatrix} = [J]^{-1} \begin{Bmatrix} \frac{\partial N_i}{\partial \xi} \\ \frac{\partial N_i}{\partial \eta} \end{Bmatrix} \qquad (2.11)$$

Where $[J]$, the Jacobian is defined as,[11]

$$\underset{(2 \times 2)}{[J]} = \underset{(2 \times n)}{\begin{bmatrix} \frac{\partial}{\partial \xi} \{N\}^T \\ \frac{\partial}{\partial \eta} \{N\}^T \end{bmatrix}} \underset{(n \times 2)}{\{x_i\} \; \{y_i\}} \qquad (2.12)$$

For plane strain/plane stress analysis, the strain vector is given in matrix notation as,

$$\underset{(3 \times 1)}{\{\varepsilon\}} = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \underset{(3 \times 4)}{\begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 \\ 0 & 1 & 1 & 0 \end{bmatrix}} \underset{(4 \times 1)}{\begin{Bmatrix} \partial u/\partial x \\ \partial u/\partial y \\ \partial v/\partial x \\ \partial v/\partial y \end{Bmatrix}} \qquad (2.13)$$

Using equations (2.11) and (2.12), the equation (2.13) can be written as,

$$\underset{(3x1)}{\{\varepsilon\}} = \underset{(3 \times 4)}{\begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 \\ 0 & 1 & 1 & 0 \end{bmatrix}} \underset{(4 \times 4)}{\begin{bmatrix} [J]^{-1} & \begin{matrix} 0 & 0 \\ 0 & 0 \end{matrix} \\ \begin{matrix} 0 & 0 \\ 0 & 0 \end{matrix} & [J]^{-1} \end{bmatrix}} \underset{(4 \times 2n)}{\begin{bmatrix} \frac{\partial N_i}{\partial \xi} & 0 & \\ \frac{\partial N_i}{\partial \eta} & 0 & \\ 0 & \frac{\partial N_i}{\partial \xi} & \cdots \\ 0 & \frac{\partial N_i}{\partial \eta} & \cdots \end{bmatrix}_{i=1,n}} \underset{(2nx1)}{\{d\}} \qquad (2.14)$$

i.e. $\{\varepsilon\} = \underset{(3 \times 2n)}{[B]} \{d\}$

For axisymmetric analysis, the strain vector is given in matrix notation[10] as,

$$\underset{(4\times1)}{\{\varepsilon\}} = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_\theta \\ \gamma_{xy} \end{Bmatrix} = \underset{(4\times5)}{\begin{bmatrix} 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 1/r \\ 0 & 1 & 1 & 0 & 0 \end{bmatrix}} \underset{(5\times1)}{\begin{Bmatrix} \partial u/\partial x \\ \partial u/\partial y \\ \partial v/\partial x \\ \partial v/\partial y \\ u \end{Bmatrix}} \qquad (2.15)$$

where the additional term, ε_θ, represents the strain in the hoop direction.

Using equations (2.11) and (2.12), the equation (2.15) can be written as,

$$\underset{(4\times1)}{\{\varepsilon\}} = \underset{(4\times5)}{\begin{bmatrix} 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 1/r \\ 0 & 1 & 1 & 0 & 0 \end{bmatrix}} \underset{(5\times5)}{\begin{bmatrix} [J]^{-1} & & 0 & 0 & 0 \\ & & 0 & 0 & 0 \\ 0 & 0 & [J]^{-1} & & 0 \\ 0 & 0 & 0 & 0 & 1 \end{bmatrix}} \underset{\substack{i=1,n \\ (5\times2n)}}{\begin{bmatrix} \frac{\partial N_i}{\partial \xi} & 0 \\ \frac{\partial N_i}{\partial \eta} & 0 \\ 0 & \frac{\partial N_i}{\partial \xi} \\ 0 & \frac{\partial N_i}{\partial \eta} \\ 1 & 0 \end{bmatrix}} \underset{(2n\times1)}{\{d\}} \qquad (2.16)$$

$$\text{i.e. } \{\varepsilon\} = \underset{(4\times2n)}{[B]} \{d\}$$

The element stiffness is calculated using the value of [B] matrix calculated in either equation (2.14) or (2.16) and introducing into equation (2.7).

Further, the element volume is calculated as $dv = t\,dx\,dy$ for plane strain/stress analysis, and $dv = 2\pi R\,dx\,dy$ for axisymmetric analysis.

In the natural coordinate system[10]

$$dv = t \det [J] \, d\xi d\eta$$

or

$$= 2\pi R \det [J] \, d\xi d\eta$$

The element stiffness then can be written as,

$$[k] = t \int_{-1}^{1}\int_{-1}^{1} [B]^T [D] [B] \det [J] \, d\xi d\eta \qquad (2.17)$$

For plane strain/stress analysis with constant thickness.

$$[k] = 2\pi \int_{-1}^{1}\int_{-1}^{1} [B]^T [D] [B] \det [J] \, R d\xi d\eta \qquad (2.18)$$

For axisymmetric analysis.

This numerical integration is necessary since the integrand polynomial cannot be integrated exactly.

For the integration Gauss quadrature is chosen, since for any given number of sampling points, it gives the greatest accuracy.

i.e. $$I = \int_{-1}^{1}\int_{-1}^{1} f(\xi,\eta) d\xi d\eta = \sum_{i=1}^{m} \sum_{j=1}^{m} W_i W_j f(\xi_j, \eta_i) \qquad (2.19)$$

where m is the number of sampling points

W_i is the weight coefficient at that point

and f is the value of the function at the sampling point

INTEGRATION ORDER (n)	SAMPLING POINT (a_j)	WEIGHTING FACTOR (H_j)
1	0.0	2.0
2	±0.5773502691896	1.
3	±0.7745966692415	0.5555555555556
	0.0	0.8888888888889
4	±0.8611363115941	0.3478548451375
	±0.3399810435849	0.6521451548625

$$\int_{-1}^{1} f(x)\,dx = \sum_{j=1}^{n} H_j f(a_j)$$

SAMPLING POINTS AND WEIGHTING FACTORS

IN GAUSS-LEGENDRE NUMERICAL INTEGRATION[9,11,17]

TABLE 1

In particular, Gauss-Legendre numerical integration is used in this formulation. The sampling points and the weights used are tabulated in Table 1.

To make sure that the mapping is unique at any sampling point, the sign of the det $[J]$ is checked. If the sign is negative, the mapping is not unique and errors may arise.

The development of $[B]$ matrices for elements Q4, Q5 and Q6 are given in the following three subsections and have been computerized in subroutines ISOQUAD and ISOBMTX.

2.2.1 Q4 Element

Q4 element is a basic linear quadrilateral element characterized by 4 corner nodes. A typical general quadrilateral element (Q4), its parent element and the mapped element are shown in Figure 1.

The nodal coordinates for the element in the natural coordinate system (ξ,η) are:

Node 1 - (1,1)

Node 2 - (-1,1)

Node 3 - (-1,-1)

Node 4 - (1,-1)

The shape functions are defined as,[12]

$$N_i = \frac{1}{4}(1 + \xi\xi_i)(1 + \eta\eta_i) \qquad (2.20)$$

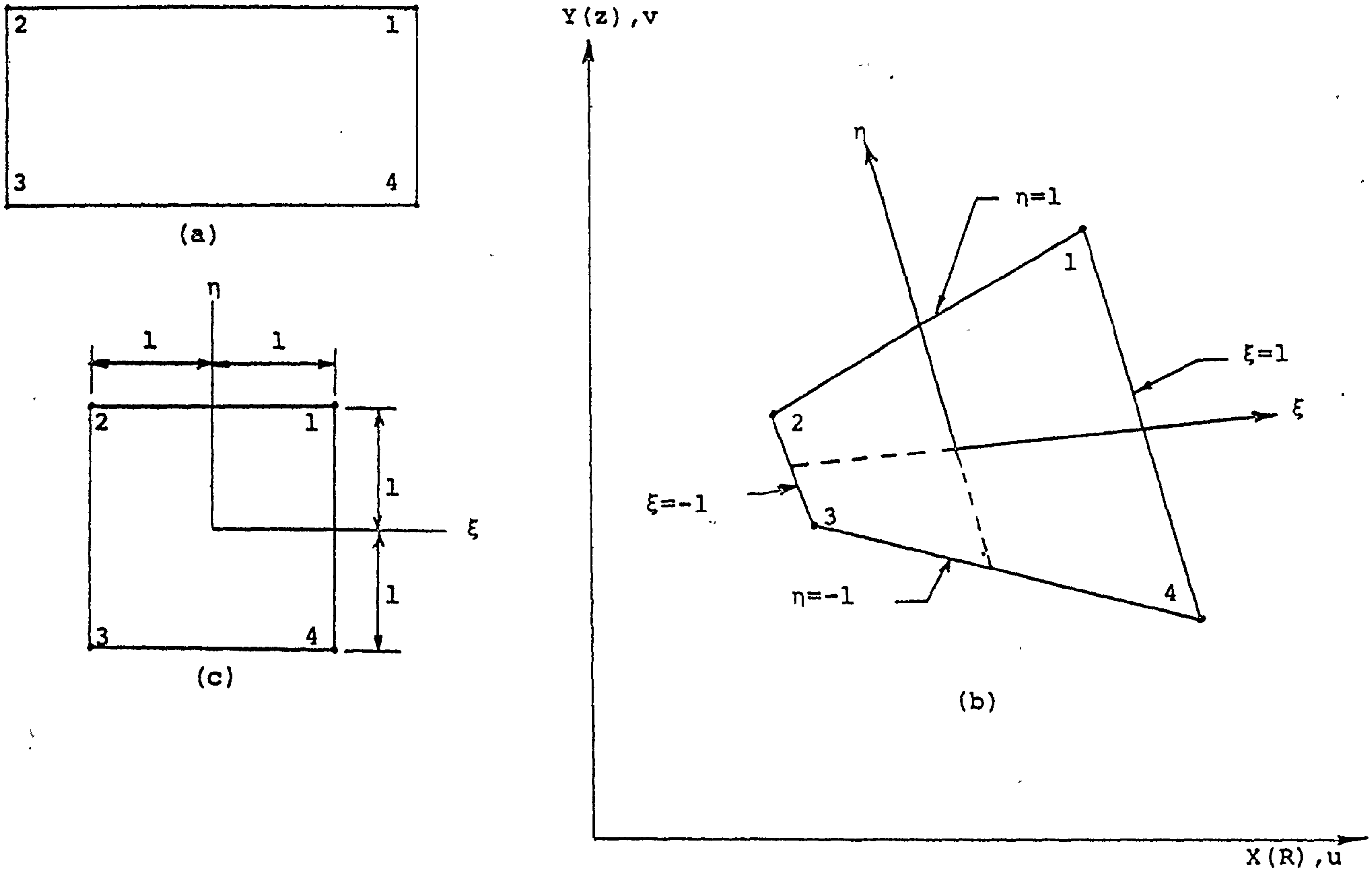


FIGURE 1 (a) Parent Element; (b) Quadrilateral Isoparametric Element; (c) Mapping into a Square resulting at the nodes

$$N_1 = \tfrac{1}{4}(1 + \xi)(1 + \eta)$$

$$N_2 = \tfrac{1}{4}(1 - \xi)(1 + \eta)$$

$$N_3 = \tfrac{1}{4}(1 - \xi)(1 - \eta)$$

and $$N_4 = \tfrac{1}{4}(1 + \xi)(1 - \eta)$$

and $$\begin{Bmatrix} x \\ y \end{Bmatrix} = \begin{bmatrix} N_1 & O & N_2 & O & N_3 & O & N_4 & O \\ O & N_1 & O & N_2 & O & N_3 & O & N_4 \end{bmatrix} \begin{Bmatrix} x_1 \\ y_1 \\ \cdot \\ \cdot \\ \cdot \\ x_4 \\ y_4 \end{Bmatrix}$$

and a linear displacement field.

At each sampling point (ξ,η), the derivatives of the shape functions with respect to the natural coordinates are given by,

$$\frac{\partial N_1}{\partial \xi} = \frac{1}{4}(1 + \eta) \; , \quad \frac{\partial N_1}{\partial \eta} = \frac{1}{4}(1 + \xi)$$

$$\frac{\partial N_2}{\partial \xi} = -\frac{1}{4}(1 + \eta) \; , \quad \frac{\partial N_2}{\partial \eta} = \frac{1}{4}(1 - \xi)$$

$$\frac{\partial N_3}{\partial \xi} = -\frac{1}{4}(1 - \eta) \; , \quad \frac{\partial N_3}{\partial \eta} = -\frac{1}{4}(1 - \xi)$$

$$\frac{\partial N_4}{\partial \xi} = \frac{1}{4}(1 - \eta) \quad , \quad \frac{\partial N_4}{\partial \eta} = -\frac{1}{4}(1 + \xi)$$

The Jacobian $[J]$ is given by:

$$J_{11} = \sum_{i=1}^{4} \frac{\partial N_i}{2\xi} X_i \quad , \quad J_{12} = \sum_{i=1}^{4} \frac{\partial N_i}{2\xi} Y_i$$

$$J_{21} = \sum_{i=1}^{4} \frac{\partial N_i}{2\eta} X_i \quad , \quad J_{22} = \sum_{i=1}^{4} \frac{\partial N_i}{2\eta} Y_i$$

Substituting these into equation (2.14) or (2.16), introducing the $[D]$ matrix and integrating as per equation (2.19) gives the element stiffness as defined by (2.17) or (2.18).

Further at each sampling point,

$$\det [J] = (J_{11} \times J_{22} - J_{21} \times J_{12})$$

is evaluated and checked for zero or negative value, which will indicate non-uniqueness in mapping.

The Jacobian is inverted and the integration is carried out using the Gauss Quadrature. The resulting $[B]$ matrix is a (3 x 8) for the plane stress/strain analysis and a (4 x 8) for the axisymmetric analysis. Using equations (2.17) or (2.18) the element stiffness $[K]$(8 x 8) is generated.

The major disadvantage of the Linear Q4 element is that it behaves badly under pure bending as the sides remain straight during deformation, i.e. parasitic shear exists at all sampling points for all but one point Gauss rule and the resulting element is too stiff in bending[10]. This is because deformations require strain energy storage by both shear strain and normal strain.

2.2.2 Q5 Element

By introducing additional degrees of freedom, a specific defect in an element may be corrected or its accuracy increased by virtue of the additional terms in the assumed displacement field.

Two internal degrees of freedom may be added to the Q4 element, resulting in the Q5 element such that[10]

$$u = \sum_{i=1}^{5} N_i u_i \quad \text{and} \quad v = \sum_{i=1}^{5} N_i v_i$$

where N_1 through N_4 are the same as for the Q4 element.

This introduces another shape function,

$$N_5 = (1 - \xi^2)\ (1 - \eta^2) \tag{2.21}$$

These degrees of freedom u_5 and v_5 vanish at all element edges and are strictly internal with no affect on the interelement compatibility. These can be viewed as generalized coordinates or displacements relative to the displacements at $\xi = \eta = 0$ dictated by the external degrees of freedom.

Introducing these generalized coordinates into the formulation results in,

$$\begin{Bmatrix} x \\ y \end{Bmatrix} = \begin{bmatrix} N_1 & O & N_2 & O & N_3 & O & N_4 & O & N_5 & O \\ O & N_1 & O & N_2 & O & N_3 & O & N_4 & O & N_5 \end{bmatrix} \begin{Bmatrix} x_1 \\ y_1 \\ \cdot \\ \cdot \\ \cdot \\ x_5 \\ y_5 \end{Bmatrix}$$

At each sampling point (ξ,η) the derivatives of the shape functions are the same as given in section 2.2.1. In addition we have,

$$\frac{\partial N_5}{\partial \xi} = -2(1-\eta^2)\xi \quad , \quad \frac{\partial N_5}{\partial \eta} = -2(1-\xi^2)\eta \qquad (2.22)$$

For the evaluation of the Jacobian, $[J]$, the summation must be over all the nodes, including the hypothetical internal node (0,0). This is mentioned here to maintain the completeness of evaluation, though these have no mathematical values at $\xi = 0$, $\eta = 0$, the centre.

The integration is carried out as described in section 2.2.1. The resulting $[B]$ matrix is a (3 x 10) matrix for plane strain/stress analysis and a (4 x 10) matrix for axisymmetric analysis. Hence the resulting stiffness matrix (10 x 10) must be reduced to the general (8 x 8) matrix to be compatible with the equations of equilibrium. We eliminate the extra degrees of freedom so that these extra unknowns do not increase the number of overall equations.

The method used for reduction of these additional degrees of freedom is called the "Static Condensation" method. The theory as outlined by Desai et al[11], is given below.

The equilibrium equations are rearranged and partitioned as:

$$\begin{bmatrix} k_{11} & | & k_{12} \\ --- & + & --- \\ K_{21} & | & k_{22} \end{bmatrix} \begin{Bmatrix} \{d_1\} \\ --- \\ \{d_2\} \end{Bmatrix} = \begin{Bmatrix} \{F_1\} \\ --- \\ \{F_2\} \end{Bmatrix}$$

where $\{d_2\}$ is the vector of displacements and $\{F_2\}$ is the vector of loads at the internal node (the additional degrees of freedom) that has to be eliminated. These equations can be rewritten as:

$$[k_{11}] \{d_1\} + [k_{12}] \{d_2\} = \{F_1\} \qquad (2.23)$$

$$[k_{21}] \{d_1\} + [k_{22}] \{d_2\} = \{F_2\} \qquad (2.24)$$

Solving equation (2.24) for $\{d_2\}$, we have

$$\{d_2\} = \frac{\{F_2\} - [k_{21}]\{d_1\}}{[k_{22}]}$$

$$= -[k_{22}]^{-1} [k_{21}] \{d_1\} + [k_{22}]^{-1} \{F_2\}$$

Substituting in equation (2.23) we have,

$$\left[k_{11}\right]\{d_1\} + \left[k_{12}\right]\left\{-\left[k_{22}\right]^{-1}\left[k_{21}\right]\{d_1\} + \left[k_{22}\right]^{-1}\{F_2\}\right\} = \{F_1\}$$

i.e. $$\left\{\left[k_{11}\right] - \left[k_{12}\right]\left[k_{22}\right]^{-1}\left[k_{21}\right]\right\}\{d_1\} = \{F_1\} - \left[k_{12}\right]\left[k_{22}\right]^{-1}\{F_2\}$$

which can be written as

$$\left[\bar{k}\right]\{d_1\} = \{\bar{F}\}$$

where the effective stiffness

$$\left[\bar{k}\right] = \left[k_{11}\right] - \left[k_{12}\right]\left[K_{22}\right]^{-1}\left[k_{21}\right] \qquad (2.25)$$

and the effective load vector

$$\{\bar{F}\} = \{F_1\} - \left[k_{12}\right]\left[k_{22}\right]^{-1}\{F_2\} \qquad (2.26)$$

The above technique is carried out using the symmetric backward Gaussian elimination which is very efficient. In our matrix, the number of external degrees of freedom r = 8 and the total number of degrees of freedom, s = 10. The following operation is carried out for the condensation.

Equation 2.25 can be written as

$$(k_{ij})_{new} = (k_{ij})_{old} - K_{ip} \frac{k_{pj}}{k_{pp}} \qquad (2.27)$$

for p = s,(s-1),(s-2).....(r+1) i.e. p=10,9

where i,j = 1,2,3........(p-1) for every p.

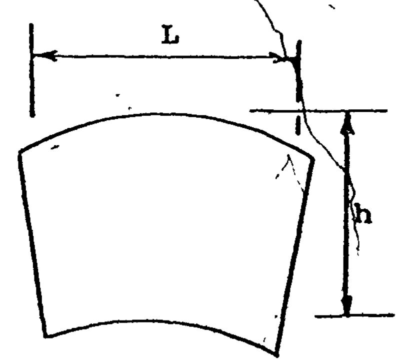
The equation (2.27) is incorporated in the subroutine ISOSTIF to arrive at a condensed (8 x 8) element stiffness matrix. Unfortunately the Q5 element behaves only slightly better than the basic Q4 element under pure bending.

2.2.3 Q6 Element

A better way to improve the behaviour in bending of linear elements was suggested by Wilson et al[19] using appropriate restraints of the rigid body motion resulting in:

$$u = \frac{M}{EI}\xi\eta$$

$$v = \frac{ML^2}{8EI}(1-\xi^2) + \nu\frac{Mh^2}{8EI}(1-\eta^2)$$



where EI is the bending stiffness

and ν is the poisson's ratio

But the linear element can only produce displacements of the form

$$u = C_1 \xi \eta \qquad \text{about the x axis}$$

and $v = 0$ where C_1 is a constant.

Similarly bending about the y axis may be considered.

This leads to the conclusion that, the Q4 element errs when bending is about both axes by omitting displacements of the form

$$u = C_1(1 - \xi^2) + C_2(1 - \eta^2)$$

$$v = C_3(1 - \xi^2) + C_4(1 - \eta^2)$$

The basic element can be modified to include this to give a better element, Q6, as

$$u = \sum_{i=1}^{4} N_i u_i + (1 - \xi^2)\, u_5 + (1 - \eta^2)\, u_6$$

$$v = \sum_{i=1}^{4} N_i v_i + (1 - \xi^2)\, v_5 + (1 - \eta^2)\, v_6 \qquad (2.28)$$

where N_1 through N_4 are the same as for the Q4 element, equation (2.20).

Though these additional degrees of freedom are internal, they affect the edge displacements and make the elements incompatible with their neighbours. This element when rectangular, is invariant under a four point Gauss quadrature but looses accuracy for arbitrary shape. Hence this element must be used with caution.

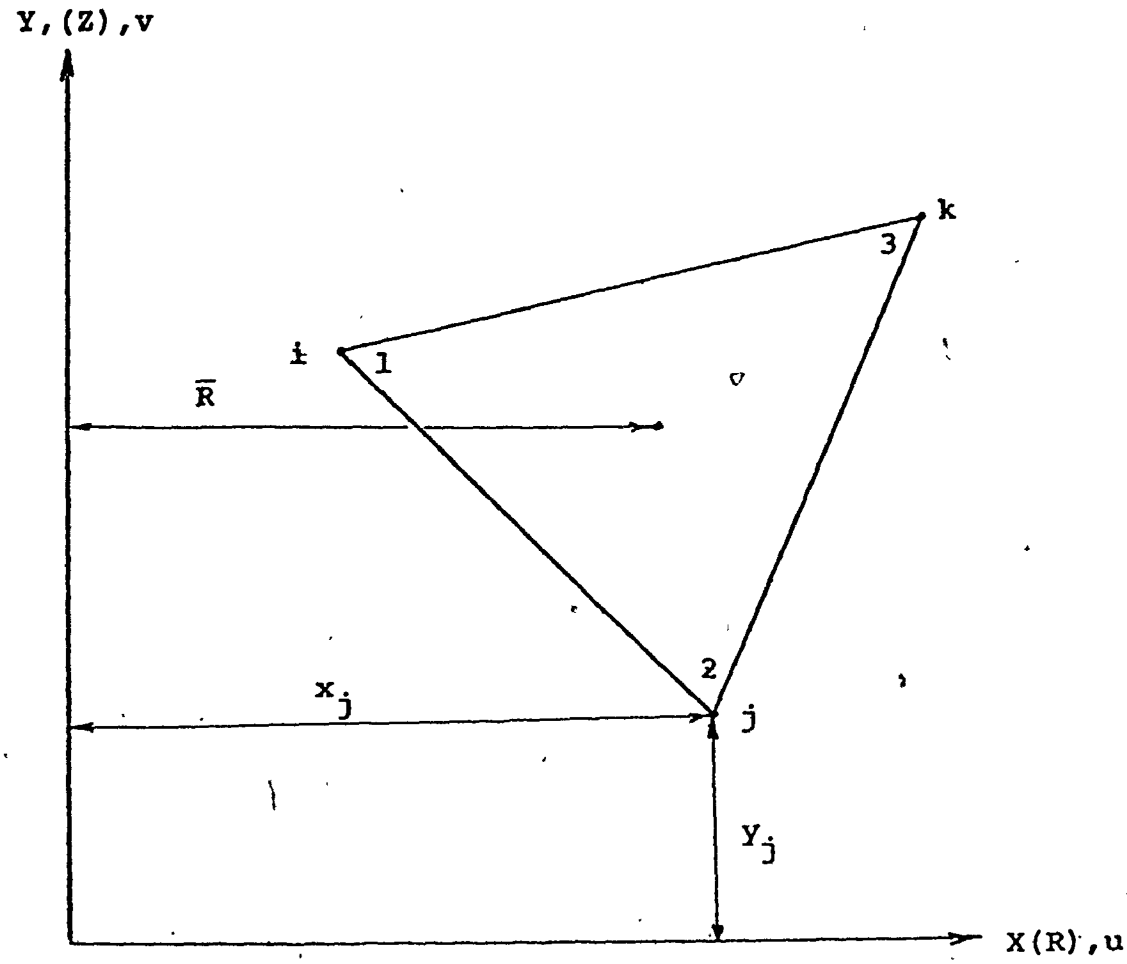
For axisymmetric analysis[10], when r, the radius, is very large compared to the element size i.e. $r \to a$, the element behaves as expected. But when r is not very large in comparison with a, corner node displacements produce hoop strain. Resistance to this activates a mode associated with non zero shear strain which should not be present al all. This makes the Q6 element not very useful in practical axisymmetric analysis.

2.3 Constant Strain Formulations

The element used for comparison in this work is a quadrilateral, which for this formulation, consists of an assemblage of constant strain triangles (CST) with primary external nodes. The CST element is one of the basic and widely used and accepted elements due to its simplicity and adaptability.

The element formulations considered here are different from each other only in the manner the quadrilateral basic element is broken down into the parent CST's. In this section, the details of setting up the element stiffness for the parent CST is described. In the next two subsections, the different methods of assembling are described.

For two dimensional problems, the linear displacement function for a parent CST shown in Figure 2 can be written as [9]



$\overline{R}$ applicable only for axisymmetric analysis

CONSTANT STRAIN TRIANGLE (CST) ELEMENT
FOR TWO-DIMENSIONAL ANALYSIS

FIGURE 2

$$u = \alpha_1 + \alpha_2 x + \alpha_3 y$$

and $$v = \alpha_4 + \alpha_5 x + \alpha_6 y$$

or in Matrix Form

$$\begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} 1 & x & y & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & x & y \end{bmatrix} \begin{Bmatrix} \alpha_1 \\ \cdot \\ \cdot \\ \cdot \\ \alpha_6 \end{Bmatrix} \qquad (2.29)$$

Thus $\varepsilon_x = \alpha_2$, $\varepsilon_y = \alpha_6$ and $\gamma_{xy} = \alpha_3 + \alpha_5$; hence the name constant strain triangle.[10]

Further, if $\alpha_1 \neq 0$, a rigid body motion exists in the x direction. Similarly, $\alpha_4 \neq 0$ produces a rigid body motion in the y direction. Also $\alpha_3 = -\alpha_5 \neq 0$ will produce a rigid body rotation. Hence the completeness, compatibility and isotropy requirements are met.

Expressing the nodal displacements in terms of the generalized coordinates α, using equation (2.29) we have

$$\{d\} = \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix} = \begin{bmatrix} 1 & x_1 & y_1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & x_1 & y_1 \\ 1 & x_2 & y_2 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & x_2 & y_2 \\ 1 & x_3 & y_3 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & x_3 & y_3 \end{bmatrix} \{\alpha\} \qquad (2.30)$$

Solving for $\alpha_1, \alpha_2, \ldots\ldots\alpha_6$ in terms of the nodal displacements {d}, we have [9]

$$u = \frac{1}{2\Delta}\Big[(a_i + b_i x + c_i y)\, u_i + (a_j + b_j x + c_j y) u_j + (a_k + b_k x + c_k y)\, u_k\Big] \qquad (2.31)$$

and

$$v = \frac{1}{2\Delta}\Big[(a_i + b_i x + c_i y)\, v_i + (a_j + b_j x + c_j y)\, v_j + (a_k + b_k x + c_k y)\, v_k\Big] \qquad (2.32)$$

where

$$2\Delta = \det \begin{vmatrix} 1 & x_i & y_i \\ 1 & x_j & y_j \\ 1 & x_k & y_k \end{vmatrix}$$

= 2 x (area of triangle with nodes i,j, and k)

and $a_i = (x_j y_k - x_k y_j)$

$b_i = (y_j - y_k)$

$c_i = (x_k - x_j)$

with other coefficients obtained by a cyclic permutation of the subscripts.

We can write equations (2.31) and (2.32) as

$$u = N_1 u_1 + N_2 u_2 + N_3 u_3$$

$$v = N_1 v_1 + N_2 v_2 + N_3 v_3$$

where $N_i = (a_i + b_i x + c_i y)/2\Delta$

with $i = 1, 2, 3$

In matrix form

$$\begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 \end{bmatrix} \{d\} \qquad (2.33)$$

To evaluate the strains, $\partial u/\partial x$, $\partial v/\partial y$, $\partial u/\partial y$ and $\partial v/\partial x$ are to be evaluated.

From equation (2.33)

$$\frac{\partial u}{\partial x} = \sum_{i=1}^{3} \frac{\partial N_i}{\partial x} u_i \quad ; \quad \frac{\partial u}{\partial y} = \sum_{i=1}^{3} \frac{\partial N_i}{\partial y} u_i$$

$$\frac{\partial v}{\partial x} = \sum_{i=1}^{3} \frac{\partial N_i}{\partial x} v_i \quad ; \quad \frac{\partial v}{\partial y} = \sum_{i=1}^{3} \frac{\partial N_i}{\partial y} v_i$$

i.e. $\frac{\partial u}{\partial x} = \frac{1}{2\Delta} \Sigma b_i u_i = \frac{1}{2\Delta} (b_i u_1 + b_j u_2 + b_k u_3)$

Similarly

$$\left.\begin{aligned}
\frac{\partial u}{\partial y} &= \frac{1}{2\Delta} (c_i u_1 + c_u u_2 + c_k u_3) \\
\frac{\partial v}{\partial x} &= \frac{1}{2\Delta} (b_i v_1 + b_j v_2 + b_k v_3) \\
\text{and} \quad \frac{\partial v}{\partial y} &= \frac{1}{2\Delta} (c_i v_1 + c_j v_2 + c_k v_3)
\end{aligned}\right\} \qquad (2.34)$$

For plane stress/plane strain analysis substituting equation (2.34) in (2.13) we have

$$\underset{(3\times1)}{\{\varepsilon\}} = \frac{1}{2\Delta} \underset{(3\times4)}{\begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 \\ 0 & 1 & 1 & 0 \end{bmatrix}} \underset{(4\times6)}{\begin{bmatrix} b_i & 0 & b_j & 0 & b_k & 0 \\ c_i & 0 & c_j & 0 & c_k & 0 \\ 0 & b_i & 0 & b_j & 0 & b_k \\ 0 & c_i & 0 & c_j & 0 & c_k \end{bmatrix}} \underset{(6\times1)}{\{d\}}$$

$$\text{i.e. } \{\varepsilon\} = \underset{(3\times6)}{[B]} \{d\} \qquad (2.35)$$

The element stiffness is then calculated using equation (2.7) as

$$[k_e] = t \iint_{v_e} [B]^T [D] [B] \; dxdy$$

For axisymmetric analysis, the element volume is given by

$$dv = 2\pi R \; dRdz \qquad \text{or} \qquad 2\pi R \; dxdy$$

Introducing into equation (3.9)

$$[k_e] = 2\pi \iint_{v_e} [B]^T [D] [B] \; R \; dxdy$$

Inspecting this equation, the integration cannot be performed as simply as for plane stress/strain analysis. This is due to the fact that $[B]$ matrix depends on the coordinate R. Three ways of overcoming this problem have been used in literature:

1. Numerical integration similar to equation (2.19)
2. Explicit multiplication and term by term integration.
3. Evaluation of $[B]$ for a centroidal point.

In this report, the method (3) of using the centroidal point is used.

$$\text{as} \quad [k]_e = 2\pi \bar{R} \Delta [B]^T [D][B]$$

where the centroid of the traingular element is given by

$$\bar{x} = (x_1 + x_2 + x_3)/3$$

$$\bar{y} = (y_1 + y_2 + y_3)/3$$

$$\bar{R} = \bar{x}$$

This is equivalent to a 'one-point' integration technique, since the volume of a swept body of revolution is given exactly by the product of the area and the path swept around by its centroid.

With a fine mesh, this method sometimes gives a result superior to the numerical integration[9]. The reason for this is the occurance of logarithmic terms in the exact formulation. These involve ratios of the type x_i/x_j and, when the element is at a large distance from the axis, such terms tend to unity and the evaluation of the logarithm is inaccurate.

Using the centroid method and substituting equations (2.34) in (2.15) we have

$$\underset{(4\times1)}{\{\varepsilon\}} = \frac{1}{2\Delta} \underset{(4\times5)}{\begin{bmatrix} 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 1/r \\ 0 & 1 & 1 & 0 & 0 \end{bmatrix}} \underset{(5\times6)}{\begin{bmatrix} b_i & 0 & b_j & 0 & b_k & 0 \\ c_i & 0 & c_j & 0 & c_k & 0 \\ 0 & b_i & 0 & b_j & 0 & b_k \\ 0 & b_i & 0 & b_j & 0 & b_k \\ N_1 & 0 & N_2 & 0 & N_3 & 0 \end{bmatrix}} \underset{(6\times1)}{\{d\}}$$

$$\text{i.e. } \{\varepsilon\} = \underset{(4\times6)}{[B]}\{d\} \qquad (2.36)$$

While Δ, the area of the triangle is calculated, its sign is checked to make sure the coordinates and the numbering are consistent.

The subroutine 'CSTBMTX' in appendix 3 lists the $[B]$ matrix generation described above, both for plane stress/strain and axisymmetric analyses.

Two types of assemblages are considered in this work: Weight averaged quadrilateral (AVEQ) using four CST's and a quadrilateral (CSTQ) made up of four CST's with a common central node.

2.3.1 Weight Averaged Quadrilateral Element (AVEQ)

A quadrilateral can be considered as an assemblage of two CST's. Experience has shown that this method gives biased results as the elements become skewed. To overcome this detrimental effect, a method of using four triangles to offset each others biased behaviour was developed. In this formulation, a quadrilateral element is made up by stacking four triangles on top of each other as shown in Figure 3.

It can be observed from the figure that, while evaluating the element matrices, the element area is

STRUCTURE OF THE WEIGHT AVERAGED QUADRILATERAL ELEMENT (AVEQ)

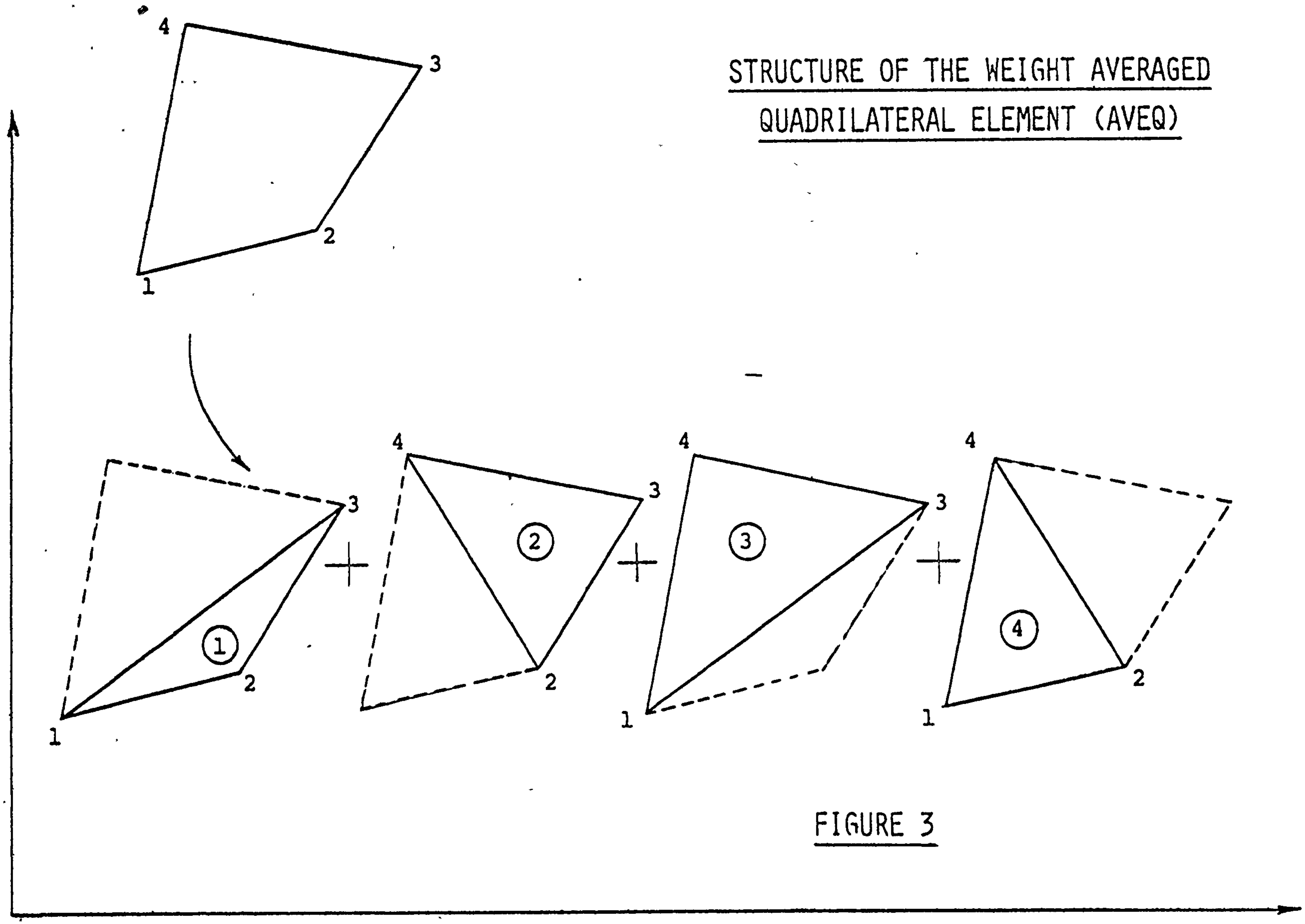


FIGURE 3

swept over twice. Further, in the final element stiffness matrix as shown later, the diagonal terms are dependent on three CST's, whereas the off-diagonal terms are dependent only on two CST's. In the final [B] matrix, the coefficients are averaged to counteract the twice swept area, as

$$B_{ij} = 0.5 * B_{ij} \qquad \text{for} \quad i = 1, 4 \quad j = 1, 4$$

This averaging is valid for the linear elements and the effect of cross coupling or the off-diagonal terms is reduced in comparison to the diagonal terms, off-setting the detrimental effect of the neighbouring triangular elements. The final quadrilateral element stiffness is assembled as:

AVEQ	=	CST1	+	CST2	+	CST3	+	CST4
k_{11}		k_{11}				k_{11}		k_{11}
k_{12}		k_{12}						k_{12}
k_{13}		k_{13}				k_{13}		
k_{14}						k_{14}		k_{14}
k_{22}		k_{22}		k_{22}				k_{22}
k_{23}		k_{23}		k_{23}				
k_{24}				k_{24}				k_{24}
k_{33}		k_{33}		k_{33}		k_{33}		
k_{34}				k_{34}		k_{34}		
k_{44}				k_{44}		k_{44}		k_{44}

Where each k is a 2 x 2 matrix for the 2 nodal degrees of freedom (x,y or r,z).

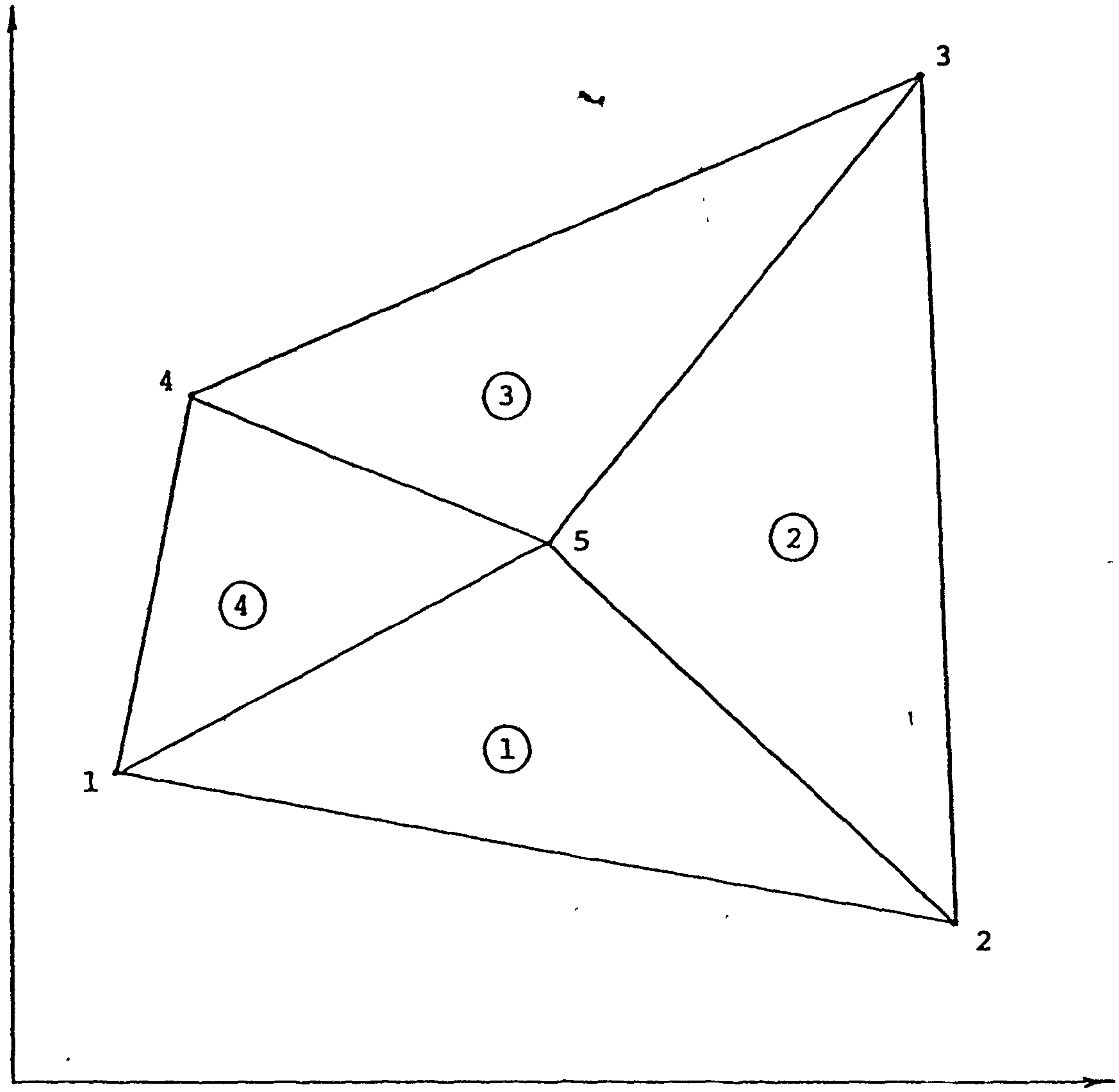
$$\text{i.e.}\quad \underset{(8 \times 8)}{\underset{\text{AVEQ}}{\left[k_e\right]}} = \begin{bmatrix} k_{11} & k_{12} & k_{13} & k_{14} \\ & k_{22} & k_{23} & k_{24} \\ \text{symmetric} & & k_{33} & k_{34} \\ & & & k_{44} \end{bmatrix}$$

These matrices can be generated quite effectively, using the cyclic property of the node numbers. Though this formulation is slower, it has the advantage that it needs no complicated programming. Subroutine 'AVEQUAD' in appendix 3 evaluates the element stiffness as outlined in this section.

2.3.2 Quadrilateral Element Made Up of Four Triangles With A Common Central Node (CSTQ)

This formulation is the most common method of grouping 4 CST's to form a quadrilateral element. This formulation also ensures that the results are independent of the skew of the subdivision triangles.

The stiffness of the quadrilateral element is assembled in three steps. First, the hypothetical inside node is located. For obvious reasons, the node is located at the centroid of the quadrilateral element. The stiffness of the resulting four triangular elements is assembled using direct stiffness method to give a



Node 5 is located at the centroid of the quadrilateral element

QUADRILATERAL ELEMENT (CSTQ) CONSTRUCTED FROM 4 CST's AND A COMMON CENTRAL NODE

FIGURE 4

(10x10) stiffness matrix. The degrees of freedom corresponding to the internal node are condensed out before assembling the total global stiffness matrix.

Referring to Figure 4, the coordinates for the central node (node 5) are calculated as

$$x_5 = \frac{1}{4} \sum_{i=1}^{4} x_i$$

and

$$y_5 = \frac{1}{4} \sum_{i=1}^{4} y_i$$

The element stiffness is assembled as:

CSTQ	=	CST1	+	CST2	+	CST3	+	CST4
k_{11}	=	k_{11}						k_{11}
k_{12}	=	k_{12}						
k_{13}								
k_{14}	=							k_{14}
k_{15}	=	k_{15}						k_{15}
k_{22}	=	k_{22}		k_{22}				
k_{23}	=			k_{23}				
k_{24}								
k_{25}	=	k_{25}		k_{25}				
k_{33}	=			k_{33}		k_{33}		
k_{34}	=					k_{34}		
k_{35}	=			k_{35}		k_{35}		
k_{44}	=					k_{44}		k_{44}
k_{45}	=					k_{45}		k_{45}
k_{55}	=	k_{55}		k_{55}		k_{55}		k_{55}

Where each k is a 2 x 2 matrix for the 2 nodal degrees of freedom (x,y or r,z)

$$\text{i.e.} \quad \underset{(10 \times 10)}{\left[k_e\right]_{CSTQ}} = \begin{bmatrix} k_{11} & k_{12} & k_{13} & k_{14} & k_{15} \\ & k_{22} & k_{23} & k_{24} & k_{25} \\ & & k_{33} & k_{34} & k_{35} \\ & \text{symmetrical} & & k_{44} & k_{45} \\ & & & & k_{55} \end{bmatrix}$$

The internal degrees of freedom are condensed out using the static condensation method as described in Section 2.2.2. This results in an element more flexible than the AVEQ element. Subroutine 'CSTQUAD' in appendix 3 evaluates the element stiffness as outlined in this section.

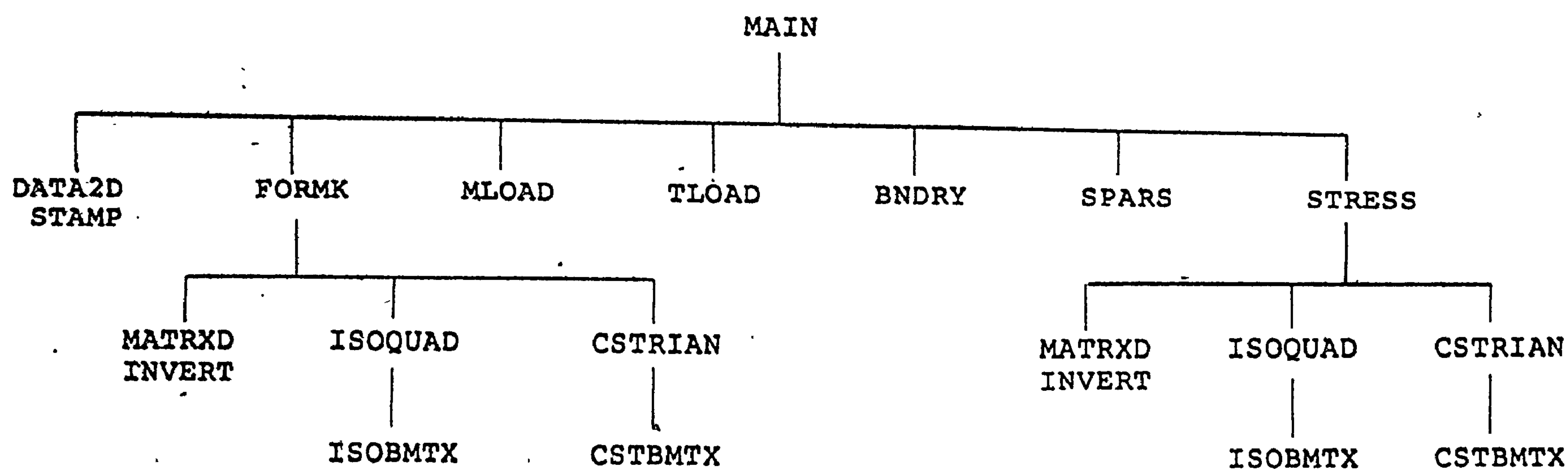
3. PROGRAM KAXI

3.1 Program Structure

Based on the element formulations described in Section 2, a general purpose Two Dimensional Stress Analysis program (KAXI) was developed and implemented on a PDP 11/34 mini computer. Due to the limited memory available (29k words) for programming on the computer, the program modules had to be overlaid and secondary storage (disc) had to be used extensively. The program is overlaid into a third order tree structure as shown in Figure 5.

The subroutines used for specific operations, like element stiffness generations, etc. were put together as modules making the program modular. This modular approach makes the program versatile in that a module can be replaced by another compatible module with minimum effort. In other words, the solution technique module, the element formulation module or the input-output modules can be easily modified or changed to suit a specific requirement. This results in providing the analyst with adequate control over the program and for easy pre and post processing of data.

In the present version of the program, the well known "Sparse Matrix Technique" modified for using secondary storage is used for the solution of the equilibrium equations. The full fixity boundary condition is imposed by inserting a large term ($1.x10^{20}$) on the diagonal of the total stiffness



STRUCTURE OF THE PROGRAM KAXI

FIGURE 5

matrix at the degree of freedom to be restricted. The list of the various subroutines used is given below:

MAIN:	Main line for the program
DATA2D:	Routine for input and output of geometry data
STAMP:	Routine to print out title block on output
FORMK:	Routine to generate the upper triangular total stiffness matrix
MATRXD:	Routine to generate a general stress-strain matrix for an anisotropic linear material in two dimensions
INVERT:	Routine for inverting a square matrix
ISOQUAD:	Routine to generate element stiffness matrix for a quadrilateral element, using isoparametric formulation
ISOBMTX:	Routine to generate the strain-displacement matrix for a quadrilateral element, using isoparametric formulation
CSTRAIN:	Routine to generate element stiffness matrix for a constant strain triangle
CSTBMTX:	Routine to generate the strain-displacement matrix for a constant strain triangle
MLOAD:	Routine to generate the load vector for mechanical loading
TLOAD:	Routine to generate the load vector for thermal loading
BNDRY:	Routine to read and insert boundary conditions
SPARS:	Solution routine using sparse matrix method
STRESS:	Routine for calculating the global stresses at element centroids and output the stresses

The computer listings of these subroutines are given in Appendix 3.

3.2 Capabilities and Limitations of the Program

The program is capable of handling a model made up of up to 500 nodes and up to 500 elements. The quadrilateral elements can either be Q4, Q5 or Q6 and the triangular elements are constant strain triangles. Due to the element behaviour, the triangular elements must only be used to accurately model the geometry. The program is capable of handling up to 10 different materials for the elements. The materials can either be linear isotropic or linear anisotropic. Though the routine MATRXD can accomodate materials whose major axes are shifted by an angle β with respect to the global (model) axes, this option is not activated at present.

The program is capable of handling both mechanical and thermal loadings as separate load cases totalling up to a maximum of 10 load cases. The mechanical load cases must be specified before the thermal load cases. A mechanical load case can comprise of either nodal loading, pressure loading or a combination of the two. The mechanical load formulation is not a consistent load formulation but a lumped load formulation . The element centroid temperature data for the thermal loading is read off a direct access file set up by a thermal analysis program. The program cannot handle initial stresses. However, this could be taken care of with minimum modifications. The program can accomodate three types of boundary conditions; full fixities, linear translatory springs and specified nodal displacements in global coordinate system.

The output of the program consists of the geometry data, loading data, boundary conditions data, global displacement at nodes and element centroidal stresses in global system. Further, the element geometry, i.e. the element definitions and the nodal coordinate data, the displacements and stresses are stored in direct access files for post processing.

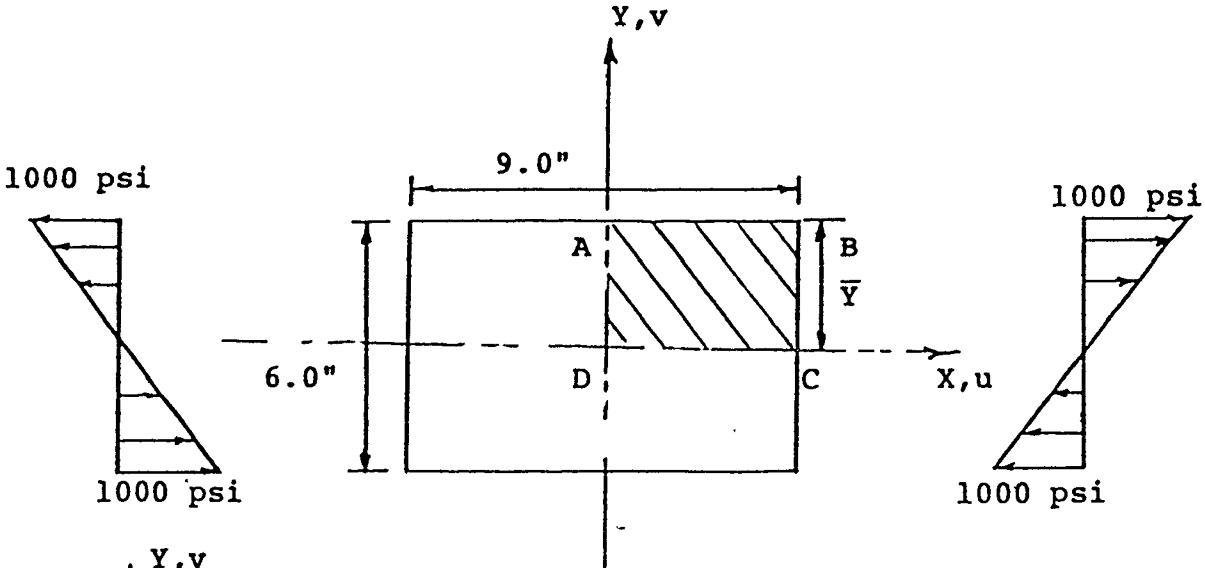
3.3 Program Verification

The program KAXI has been verified for all three types of analysis: plane stress, plane strain and axisymmetric. Mechanical loading was used for the verifications. For the axisymmetric analysis, in addition to the mechanical loading, thermal loading was also considered.

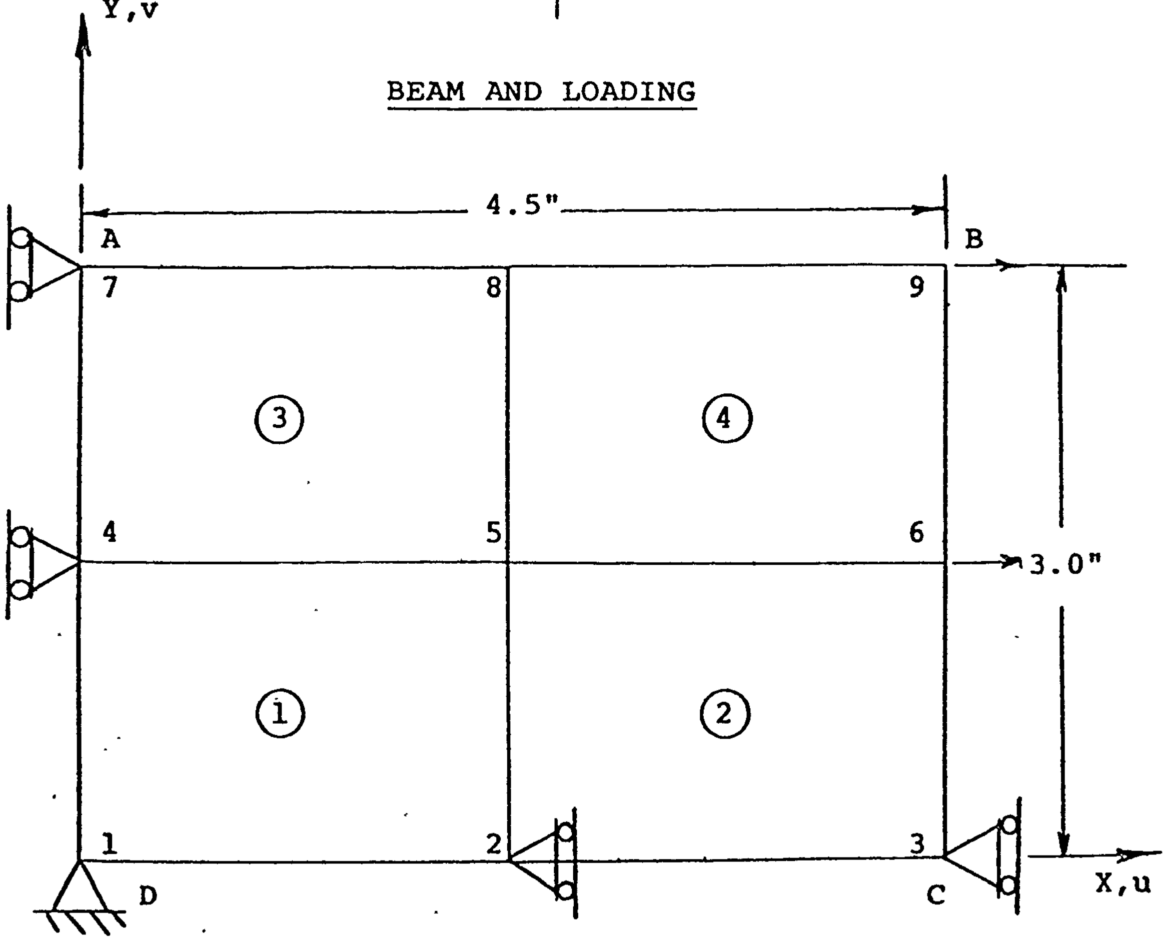
The results using elements AVEQ and CSTQ are also included in the verifications for the purposes of comparison. These were calculated using separate computer programs. Further, for the element formulation CSTQ, no stress recovery was carried out. Sample outputs for the verifications are given in Appendix 3.

3.3.1 Plane Stress Analysis

The problem of a beam under pure bending from Ref. 11 was used for the verification. The model analyzed is shown in Figure 6. The model is made up of 4 elements and 9 nodes.



BEAM AND LOADING



E = 30.0E06 psi

ν = 0.3

t = 1.0"

FIGURE 6

TEST CASE FOR PLANE STRESS ANALYSIS:[11] BEAM BENDING

The nodes 2,3,4 and 7 were restrained in the x direction whereas node 1 was restrained in both x and y directions. The loading was introduced at nodes 6 and 9 as nodal forces.

The displacements at nodes 9 (location B) and 3 (location C) and the stress σ_x at the centroids of the elements 1 and 3 are computed using the programs and compared with the theoretical results in Table 2.

The theoretical displacements were calculated using[11]

$$u = \frac{P}{E\bar{y}} xy$$

$$\text{and} \quad v = \frac{-P}{2E\bar{y}} (x^2 + \nu y^2)$$

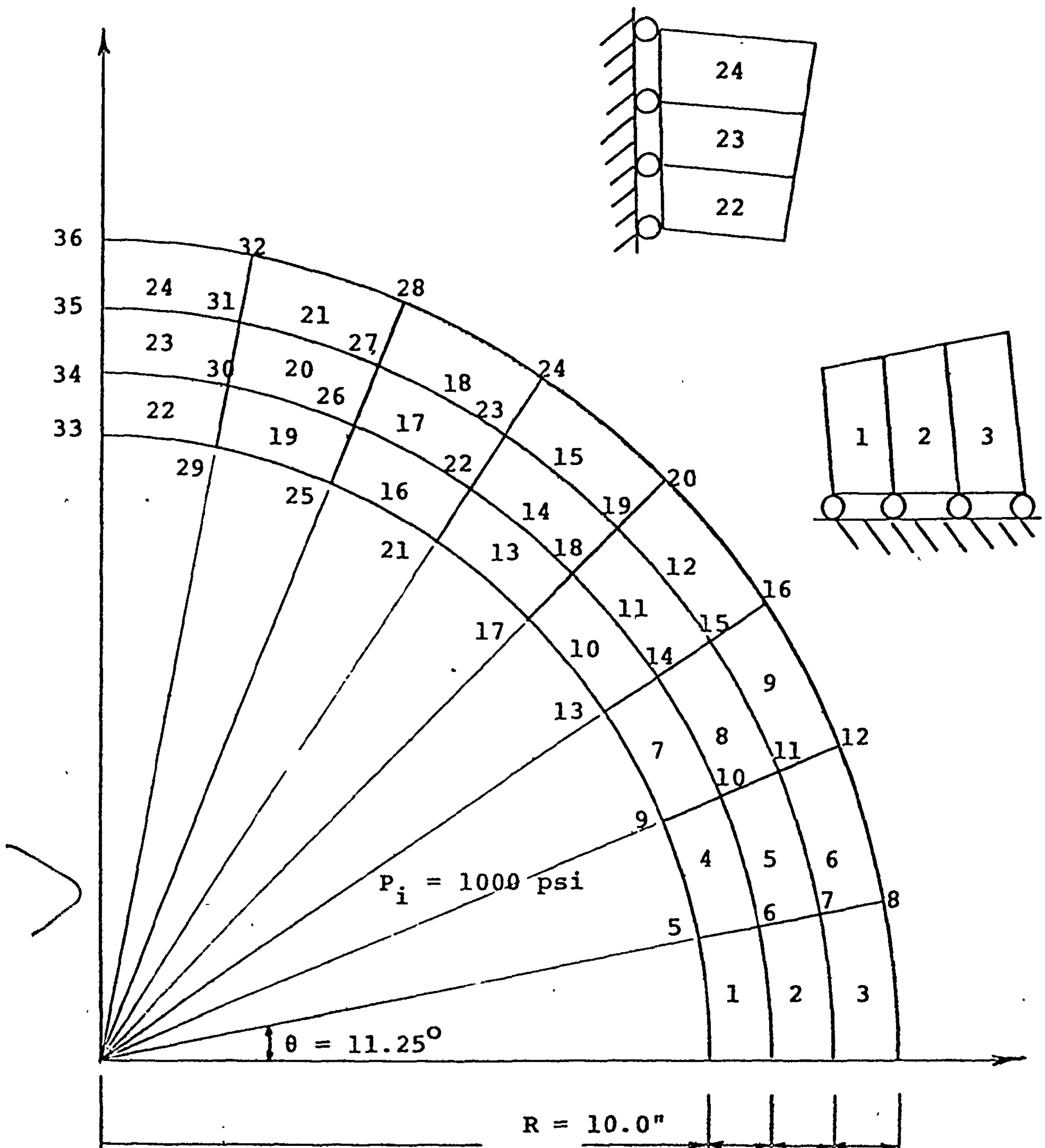
where P = the bending stress

$\bar{y}$ = the distance from the neutral axis to the top of the fibre

Referring to Table 2, it is evident that the Isoparametric element family results agree quite well with the theoretical results. It is also clear that, for a mesh of aspect ratio 1:1.5, the Q6 element gives exact results. The inadequacy of constant strain formulations and the Q4 formulation in a predominantly bending mode can be clearly seen. Though the AVEQ element results have the right trend, the displacements are only 75% of the theoretical displacements.

3.3.2 Plane Strain Analysis

A quarter model of a thick cylinder with R_i=10" and t = 3" under an internal pressure loading (P_i=1000 psi) was analyzed. The model analyzed is shown in Figure 7. The



TEST CASE FOR PLANE STRAIN ANALYSIS:

LONG PIPE UNDER MECHANICAL LOADING

E = 30.0 E06 psi

ν = 0.3

α = 7.E-06 in/in

FIGURE 7

model was set up with three elements through the pressure wall and a sector angle of 11.25° resulting in a maximum aspect ratio of 1:1.25. The model is made up of 24 elements and 36 nodes. The boundary nodes 1 through 4 are restrained in the x direction allowing only radial displacements at the axes. The pressure loading was applied on the inner surface.

The stress distribution through the pressure wall and the radial displacements at the inner and the outer surfaces are compared with the theoretical results obtained by using classical methods as below:[21]

The stresses at any radius r are given by

$$\sigma_R = \frac{a^2}{(b^2-a^2)} \left(1 - \frac{b^2}{r^2}\right) P_i$$

$$\sigma_\theta = \frac{a^2}{(b^2-a^2)} \left(1 + \frac{b^2}{r^2}\right) P_i$$

and the radial displacements are given by

$$\delta_a = \frac{P_i}{E} a\left(\frac{b^2+a^2}{b^2-a^2} + \nu\right)$$

$$\delta_b = \frac{P_i}{E} \left(\frac{2ba^2}{b^2-a^2}\right)$$

where $a = R_i$, the inside radius

$b = R_i + t$, the outside radius

$\nu = 0.3$, the Poisson's ratio

$E = 30.E06$ psi, the Young's modulus

$P_i = 1000$ psi, the internal pressure

D I S P L A C E M E N T S

LOCATION AND DIRECTION	THEORY	ELEMENT FORMULATION				
		Q4	Q5	Q6	AVEQ	CSTQ
B,u	1.5000	1.3965	1.4367	1.500	1.1127	1.3614
B,v	-1.2750	-1.1901	-1.2230	-1.275	-0.9538	-1.1161
C,u	0.000	0.000	0.000	0.000	0.000	0.000
C,v	-1.1250	-1.0571	-1.0842	-1.125	-0.8504	-1.0327

σ_x S T R E S S (PSI)

ELM 1	250.00	235.786	241.211	250.00	Not calculated	
ELM 3	750.00	707.398	724.584	750.00		

Ref: Figure 6 Displacements in 1.E-0.4 in.

RESULTS OF PROGRAM VERIFICATION FOR PLANE STRESS ANALYSIS

TABLE 2

LOAD CASE: MECHANICAL LOADING; P_i = 1000 psi

	RADIUS	THEORY	ELEMENT FORMULATION				
			Q4	Q5	Q6	AVEQ	CSTQ
RADIAL STRESS σ_R (PSI)	10.0	-1000.00	-980.10	-983.4	-973.70	-758.50	STRESSES NOT CALCULATED
	10.5	-772.29	-764.22	-767.04	-756.92	-563.92	
	11.5	-402.73	-396.09	-398.09	-387.27	-231.73	
	12.5	-118.26	-112.76	-114.19	-102.95	24.52	
	13.0	0.00	-2.89	-15.32	7.21	124.20	
HOOP STRESS σ_θ (PSI)	10.0	3898.55	3881.00	3883.00	3892.00	3693.00	
	10.5	3670.84	3664.19	3666.48	3675.75	3491.03	
	11.5	3301.28	3294.57	3296.61	3306.27	3147.33	
	12.5	3016.81	3010.28	3012.12	3022.09	2883.65	
	13.0	2898.55	2900.00	2902.00	2912.00	2782.00	
RADIAL DISP. (in)	10.0	13.995	13.067	13.079	13.093	12.952	13.055
	13.0	12.560	11.385	11.394	11.414	11.296	11.376

Ref: Figure 7 Displacements in 1.E-0.4 in.

RESULTS OF PROGRAM VERIFICATION FOR PLANE STRAIN ANALYSIS

TABLE 3

In Table 3, for comparison, the global centroidal stresses at elements 10, 11 and 12 transformed into local stresses are tabulated. The surface stresses shown were arrived at using least squares fit on the local stresses and then extrapolating to the surface. The radial displacements at the inside and the outside surface tabulated correspond to the displacements at nodes 17 and 20.

Referring to Table 3, the results from the programs agree quite well with the theoretical results. Once again, the isoparametric family of elements is superior to the constant strain family of elements. Though element AVEQ gives acceptable results for the displacements and the hoop stresses, it falls short for radial stresses. This degradation in the results is not very significant during overall analysis since the radial stresses are about an order of magnitude smaller compared to the hoop stresses and the stress field is governed by the predominant hoop stress.

3.3.3 Axisymmetric Analysis

A quarter model of a thick spherical shell with $R_i = 10''$ and $t = 3''$ was analyzed under an internal pressure loading ($P_i = 1000$ psi) and a linearized (thin shell) thermal gradient. The details of the model, the boundary conditions imposed and the pressure loading are identical to the plane strain test model described in section 3.3.2. The model used is given in Figure 8. The thermal loading was applied as element centroidal temperatures arrived at using:[20]

$$T_r = T_i \frac{a}{(b-a)} \left(\frac{b}{r} - 1 \right) \quad \text{at any radius } r$$

where $T_i = 500^\circ F$, the inside surface temperature

$a = R_i$, the inside radius

$b = R_i + t$, the outside radius

The temperature distribution used is,

@ $r = 10.5"$, $T_r = 396.825^\circ F$

@ $r = 11.5"$, $T_r = 217.391^\circ F$

@ $r = 12.5"$, $T_r = 66.667^\circ F$

The stress distribution through the wall and the radial displacements (for pressure load only) at the inner and the outer surfaces are compared with the theoretical results obtained by using classical methods as below:

For pressure loading, the stresses at any radius r are given by,[21]

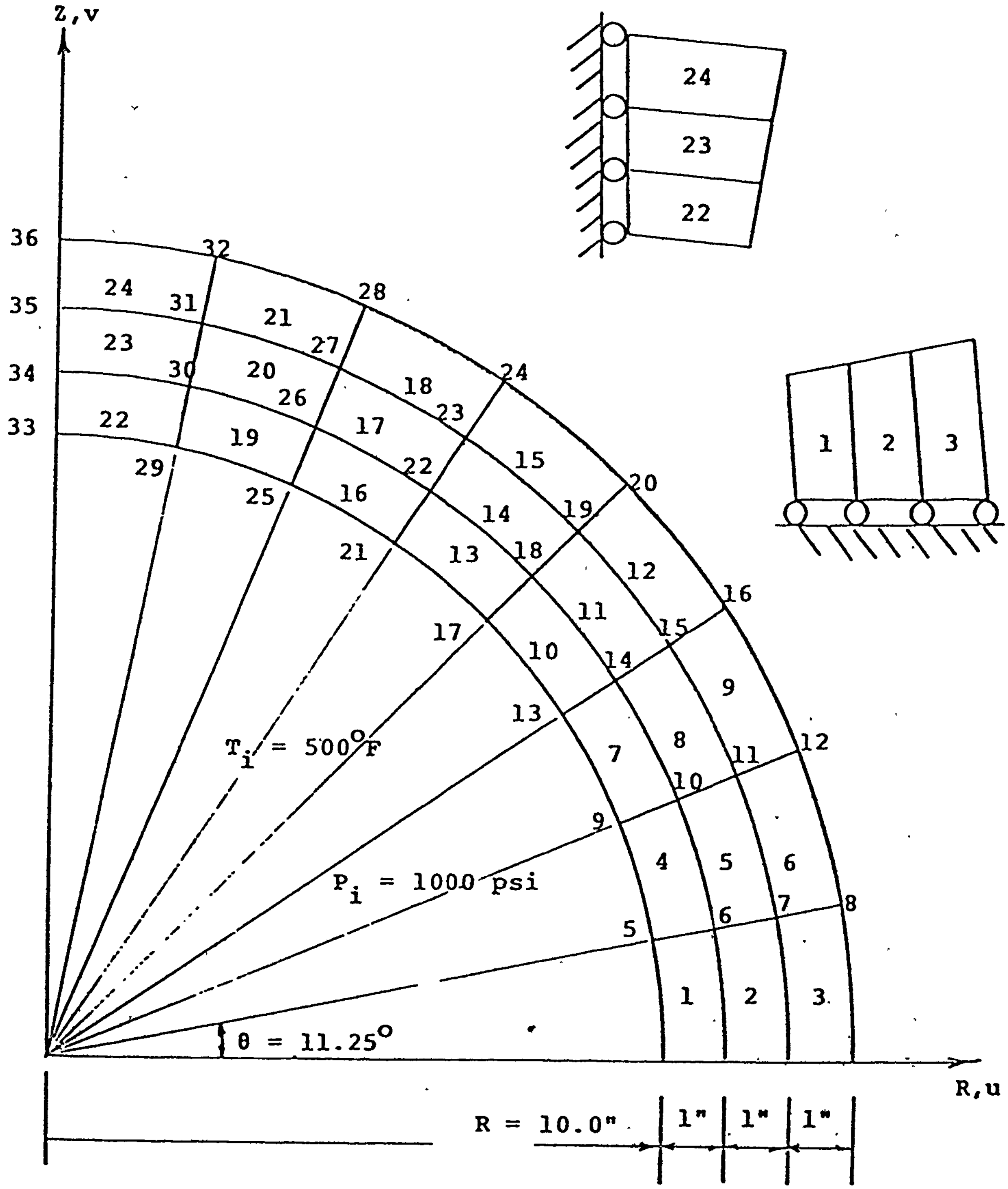
$$\sigma_R = \frac{a^3}{(b^3-a^3)} \left(1 - \frac{b^3}{r^3}\right) P_i$$

$$\sigma_\theta = \frac{a^3}{(b^3-a^3)} \left(1 + \frac{b^3}{2r^3}\right) P_i$$

and the radial displacements are given by,

$$\delta_a = \frac{P_i}{E} a \left(\frac{(1-\nu)\ (b^3+2a^3)}{2(b^3-a^3)} + \nu\right)$$

$$\delta_b = \frac{P_i}{E} b \left(\frac{3(1-\nu)a^3}{2(b^3-a^3)}\right)$$



TEST CASE FOR AXISYMMETRIC ANALYSIS

SPHERICAL VESSEL UNDER MECHANICAL AND THERMAL LOADING

FIGURE 8

$E = 30.0$ E06 psi

$\nu = 0.3$

$\alpha = 7.$E-06 in/in

Linearized temp. Gradient $T_i = 500^oF$

where the variable definitions are the same as in section 3.2.2.

For the thermal loading (linearized gradient) the stresses at any radius r are given by,[20]

$$\sigma_R = \frac{E\,\alpha\,T_i}{(1-\nu)} \; \frac{ab}{(b^3-a^3)} \left[a+b- \frac{1}{r}\,(b^2+ab+a^2) + \frac{a^2b^2}{r^3} \right]$$

$$\sigma_\theta = \frac{E\,\alpha\,T_i}{(1-\nu)} \; \frac{ab}{(b^3-a^3)} \left[a+b-\frac{1}{2r}\,(b^2+ab+a^2) - \frac{a^2b^2}{2r^3} \right]$$

In Tables 4 and 5, for comparison, the global centroidal stresses at elements 10, 11 and 12 transformed into local stresses are tabulated. The surface stresses shown were arrived at using least squares fit on the local stresses and then extrapolating to the surface. The radial displacements at the inside and the outside surface tabulated correspond to displacements at nodes 17 and 20. Further, for reasons given in section 2.2.3, element Q6 was not included in the verifications.

Referring to Table 4, the results from the programs agree quite well with the theoretical results. Though the isoparametric family gave better results for plane stress and plane strain analyses, it does not show any significant superiority for axisymmetric analysis. For the mechanical loading, though the AVEQ element gives lower stresses compared to the isoparametric family, the hoop stresses

LOAD CASE: MECHANICAL LOADING; P_i = 1000 psi

	RADIUS	THEORY	ELEMENT FORMULATION Q4	Q5	Q6	AVEQ	CSTQ
RADIAL STRESS σ_R (PSI)	10.0	-1000.00	-963.60	-980.70	ELEMENT TYPE NOT ANALYZED	-946.50	STRESSES NOT CALCULATED
	10.5	-750.08	-734.73	-748.37		-721.20	
	11.5	-371.40	-360.49	-369.04		-351.80	
	12.5	-104.31	-97.62	-103.48		-90.70	
	13.0	0.00	-7.95	-13.36		-0.76	
HOOP STRESS σ_θ (PSI)	10.0	1753.13	1776.00	1776.00		1739.00	
	10.5	1628.18	1651.42	1651.41		1623.65	
	11.5	1438.83	1445.95	1446.02		1434.94	
	12.5	1305.29	1299.53	1299.26		1302.13	
	13.0	1253.13	1248.00	1248.00		1257.00	
RADIAL DISP. (IN)	10.0	5.0906	5.0417	5.0585		5.0011	5.0506
	13.0	3.8012	3.7683	3.7773		3.7399	3.7726

Ref: Figure 8 Displacements in 1.E-04 in.

RESULTS OF PROGRAM VERIFICATION FOR AXISYMMETRIC ANALYSIS

TABLE 4

LOAD CASE: THERMAL LOADING; LINEAR GRADIENT T_i = 500°F, T_o = 0°F

	RADIUS	THEORY	Q4	Q5	Q6	AVEQ	CSTQ
RADIAL STRESS σ_R (PSI)	10.0	0.00	-843.60	-332.80	ELEMENT TYPE NOT ANALYZED	-1405.00	STRESSES NOT CALCULATED
	10.5	-6534.94	-5421.67	-5129.17		-5739.17	
	11.5	-9507.72	-8464.80	-8490.66		-8416.21	
	12.5	-4352.88	-3357.27	-3543.92		-3103.84	
	13.0	0.00	2253.00	2045.00		2548.00	
HOOP STRESS σ_θ (PSI)	10.0	-87964.93	-87100.00	-86780.00		-87610.00	
	10.5	-53750.08	-54367.61	-54188.09		-54707.95	
	11.5	1566.55	1434.76	1415.77		1402.24	
	12.5	44206.52	44363.53	44236.18		44590.85	
	13.0	62030.08	61000.00	60850.00		61340.00	

Ref: Figure 8

RESULTS OF PROGRAM VERIFICATION FOR AXISYMMETRIC ANALYSIS

TABLE 5

which are the significant components as calculated are closer to the theoretical results.

Referring to Table 5, all element formulations give similar results for the thermal loading. Only the radial stress distribution for the isoparametric family of elements is marginally better than the constant strain formulations. However, the hoop stress which is governing is not very much different for the various formulations.

The predominant mode of deformation in the above test case is not one of bending and hence all element formulations give similar results. To study the behaviour during a predominant bending mode, a thin circular ring fixed at the inner radius and loaded at the outer radius was analyzed. The model used in the analysis and the results are shown in Figure 9.

The model is made up of 30 elements and 44 nodes resulting in an aspect ratio of 1:3. The nodes at the inside edge (R_i=25") were restrained in both radial and axial directions. Node 44 at the outer edge (R_o=50") was loaded with 1000 lbs./in. of circumference. The maximum stress and the maximum displacements are compared with theoretical results obtained by using[17]

$$\sigma_{max} = \frac{kP}{t^2} \quad \text{and} \quad \delta_{max} = \frac{k_1 P b^2}{E t^3}$$

where $b = R_o$, the outside radius

$a = R_i$, the inside radius

$E = 30.E06$ psi, Young's Modulus

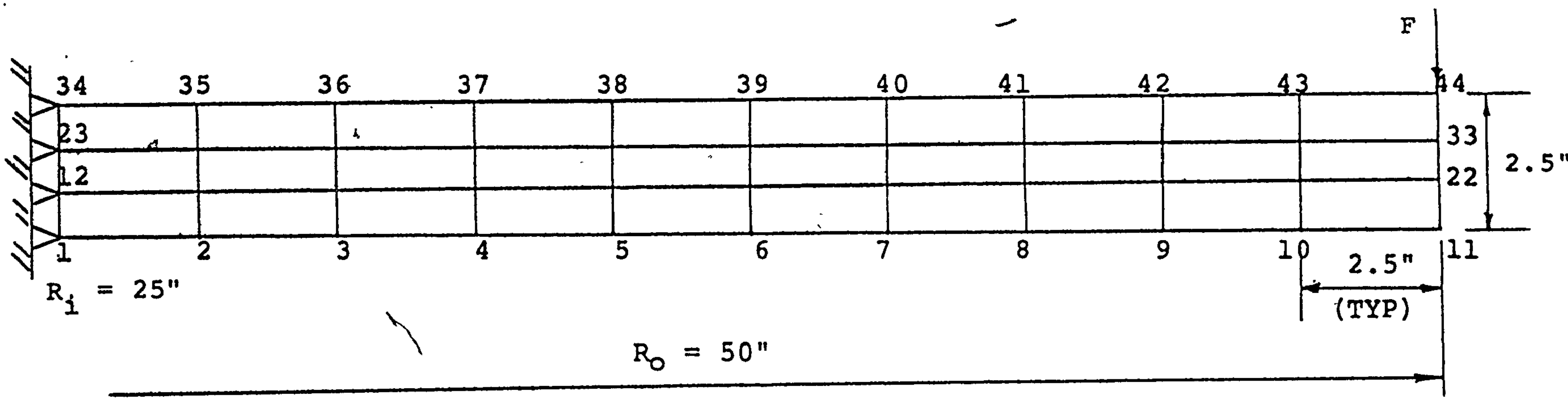
$t = 2.5"$, the thickness

$P = 2\pi R_o F$

$F = 1000$ lbs/in

$$\left.\begin{array}{l} k = 0.7530 \\ k_1 = 0.0877 \end{array}\right\} \text{constants for } b/a = 2\text{, Ref. 11}$$

Referring to Figure 9, we can see that the Isoparametric family of elements give better results compared to the constant strain elements.



E = 30.E06 psi
ν = 0.3
F = 1000 lbs

	THEORY	ELEMENT FORMULATION Q4	Q5	Q6	AVEQ	CSTQ
σ_{MAX} (psi)	-37849.9	-30500.	-31860.	-	-20330.	-
δ_{MAX} (in)	-0.14694	-0.11338	-0.11970	-	-0.07355	-0.10663

Stresses extrapolated to the surface

AXISYMMETRIC MODEL OF A RING HELD AT ITS INNER RADIUS AND LOADED AT THE OUTER RADIUS

FIGURE 9

4. COMPARISON OF ELEMENT FORMULATIONS

The element formulation most suitable for any particular application is the one which produces the maximum strain energy under the prescribed set of loading conditions. The comparison has been carried out at two levels:

(a) at the element level, using a single element,

(b) at the assemblage level, using sensitivity analysis.

4.1 Comparison at the Element Level

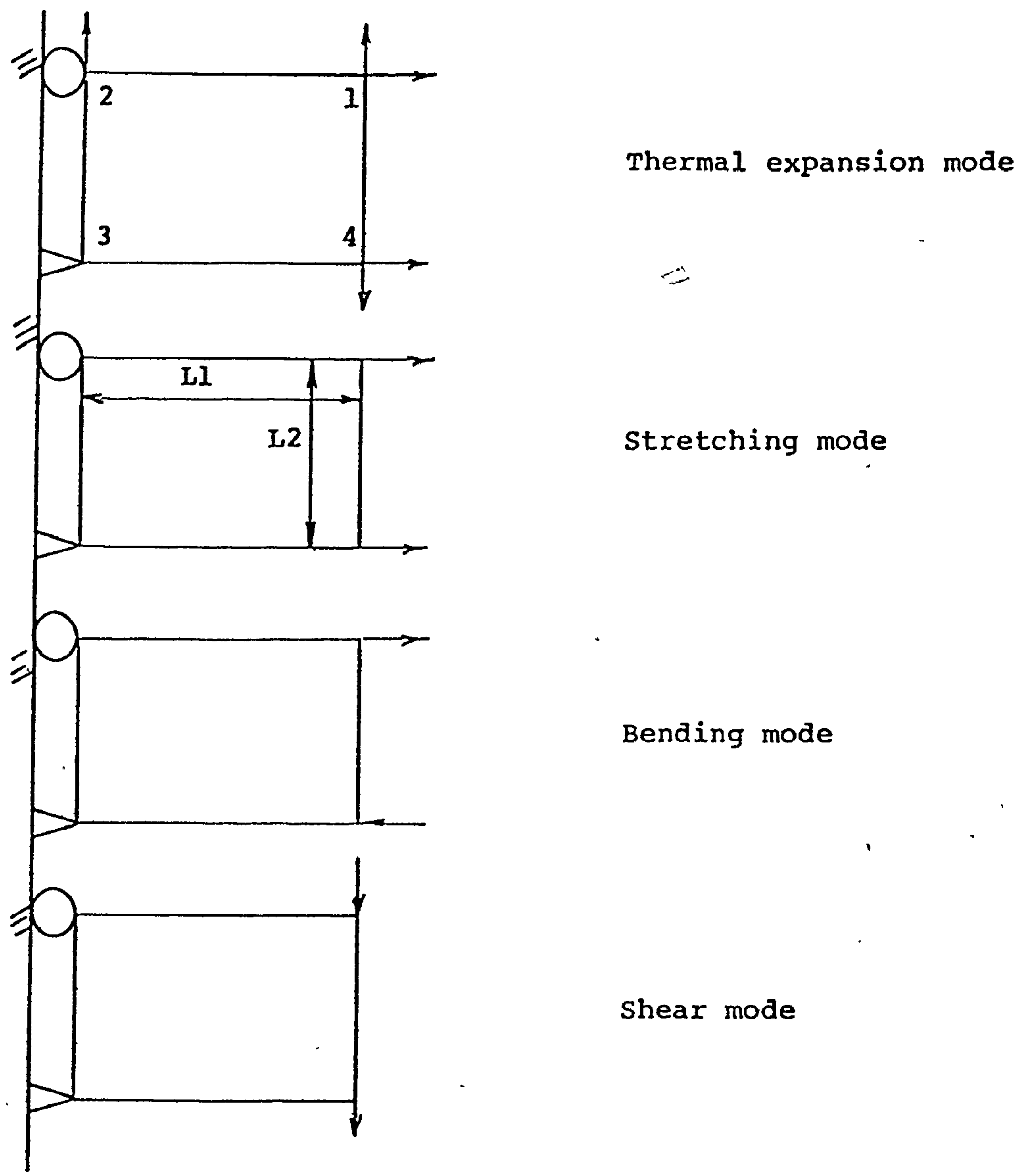
Since most of the modelling done in the engineering industry is using rectangular elements, these were chosen as the basic element for comparison, with four basic modes of loading as shown in Figure 10. The comparison was based on the strain energy criterion under the basic modes of loading. The strain energy was calculated using the "Weighted Eigenvalue Method" as described in section 4.2. Strain energy densities were calculated to facilitate comparisons irrespective of the element size. Two nodes of the element were restrained as shown in Figure 10 to remove all the rigid body modes of deformation. The other two nodes were loaded with unit loads to produce:

(a) a unit thermal expansion mode (biaxial stretch),

(b) a uniaxial stretching mode,

(c) a pure bending mode,

(d) shear mode.



E = 1., ν = .3, α = 1.
Aspect Ratio = L1/L2
All loads are unit loads
BoCo: node 3 fixed in x and y
node 2 fixed in x

ELEMENT AND LOAD CONFIGURATION FOR
STRAIN ENERGY COMPARISONS
FIGURE 10

Only the thermal load vector used is a consistent load vector.

The elements were checked for zero eigenvalues to make sure that there are only as many zero eigenvalues as there ought to be. Too few suggests a formulation that does not permit rigid body motion whereas, too many suggests zero energy deformation modes. Accordingly, there are three rigid body modes for the two dimensional element and one for the axisymmetric element. Further, the non-zero eigenvalues were checked to see whether or not they have the same value for similar modes. The non-zero eigenvalues and deformation modes of Q4 element in plane strain are shown in Figure 11.

The results from these comparisons were plotted as $\sqrt{U_0}$ vs.geometry variable, which is the aspect ratio for the rectangular elements. $\sqrt{U_0}$, the root of the strain energy density is in fact proportional to the deformation of linear elements. The plots are shown in Figures 12 through 23. This family of plots show that the elements behave similarly for all three types of analysis; plane stress, plane strain and axisymmetric.

The criterion for the element choice can be stated as; For any type of analysis, under a specific loading, the element having the largest strain energy density is the most flexible and hence better.

However, it is important to note that the Q6 formulation fails the 'patch test' i.e., when nodal degrees of freedon are given values corresponding to a state of constant strain, the displacement field does not produce a constant strain state through the element. Hence, for non-rectangular Q6 elements in plane stress and plane strain and rectangular Q6 elements in axisymmetric analysis, this comparison data should be used with caution. In the comparison, the radius is made large compared to the element size so that there is not much loss of element quality during axisymmetric analysis.

4.2 The Criterion for Comparison

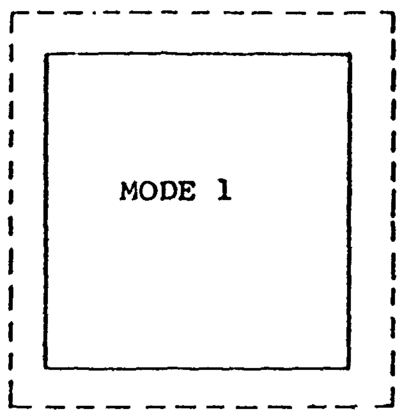
In the available literature only few researchers like Melosh[13], Clough[15], Khanna[14], Rigby and McNeice[16] have reported studies involving accuracy of element formulations. In this work, since elements with non-compatible deformation or zero strain energy modes are included for comparison, the simplified argument "the least stiff element is the best, i.e. the element with the smallest eigenvalues and smallest matrix trace" is not valid. The Weighted Eigenvalue method which is a complete strain energy based method presented by Rigby and McNeice is used for comparison.

The element strain energy as given by

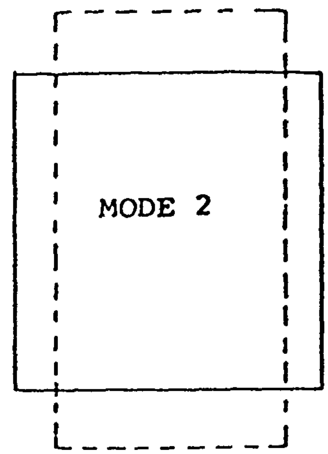
$$U_e = \tfrac{1}{2}\, d^T k d \tag{4.1}$$

Considering the basic equilibrium equation at the element level,

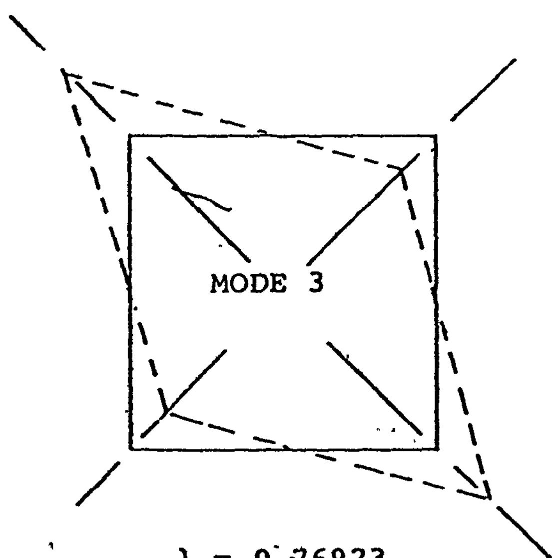
$$kd = F \tag{4.2}$$



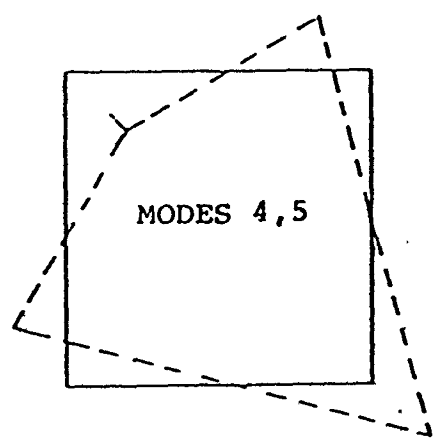
λ_1= 1.92308
Extension



λ_2= 0.76923
Pinching



λ_3= 0.76923
Shear



$\lambda_{4,5}$= 0.57692
Flexural

------ Deformed Shape
Modes 1,2 and 3 are Constant Strain modes, Modes 4 and 5 are Linear Strain modes.

$E = 1., \nu = .3$
Base element = unit square

DEFORMATION MODES AND NON ZERO EIGENVALUES FOR A LINEAR ISOPARAMETRIC ELEMENT(Q4) IN PLANE STRAIN CONDITION

FIGURE 11

Let the nodal loads F be made proportional to the nodal displacements d as $F = \lambda d$, this renders a standard eigenvalue problem described by the equation

$$kd = \lambda d \tag{4.3}$$

which can be written in general terms as

$$k\phi_i = \lambda_i \phi_i \qquad (i = 1,2 \ldots\ldots \ell) \tag{4.4}$$

where k is the stiffness matrix which is non-singular following the removal of the rigid body modes. λ_i are the eigenvalues and ϕ_i are the corresponding normalized eigenvectors. ℓ is the number of modes following the removal of the rigid body modes.

The eigenvectors are normalized such that their orthogonality relationships become

$$\phi_i^T \phi_j = 1, \text{ for } i = j \qquad = 0, \text{ for } i \neq j \tag{4.5}$$

The displacement vector d in equation (4.3) can be expanded as a linear combination of eigenvectors.

i.e. $$d = \sum_{i=1}^{\ell} a_i \phi_i \tag{4.6}$$

Using equations (4.1), (4.4), (4.5) and (4.6), the strain energy can be written as

$$U = \tfrac{1}{2} \sum_{i=1}^{\ell} a_i \lambda_i a_i \tag{4.7}$$

The undetermined coefficients a_i can be solved for by pre-multiplying equation (4.2) by each eigenvector and using the orthogonality requirements for the k^{th} eigenvector

$$\phi_k^T k \sum_{i=1}^{\ell} a_i \phi_i = \phi_k^T F$$

$$\phi_k^T k a_k \phi_k = \phi_k^T F$$

$$\phi_k^T \lambda_k a_k \phi_k = \phi_k^T F \qquad (4.8)$$

and therefore

$$a_k = (1/\lambda_k)\ \phi_k^T F \qquad (4.9)$$

Therefore, in general, the strain energy becomes

$$U_e = \tfrac{1}{2} \sum_{i=1}^{\ell} \frac{1}{\lambda_i}\ \phi_i^T F\ \phi_i^T F$$

or $$U_e = \tfrac{1}{2} \sum_{i=1}^{\ell} \frac{C_i}{\lambda_i} \qquad (4.10)$$

where $$C_i = (\phi_i^T F)^2 \qquad (4.11)$$

Using equations (4.10) and (4.11), the strain energy within an element can be determined once the eigenvalues, the eigenvectors and the applied loading are known. Since flexible elements tend to deform more for any given set of loading, the higher the strain energy better the element.

The eigenvalue and eigenvector extraction method used is given in Appendix 1.

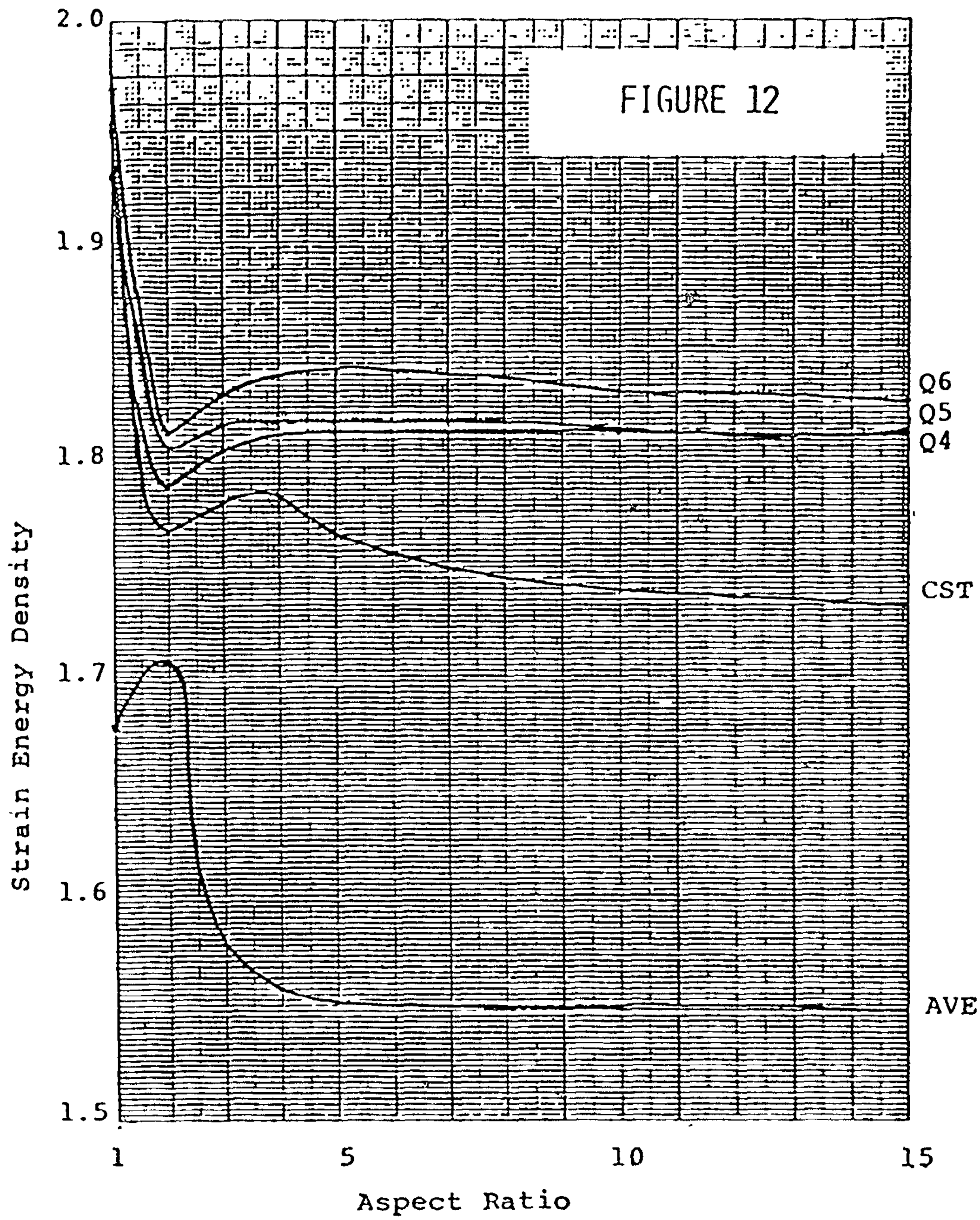
4.3 Results of Comparison at Element Level

The Figures 12 through 23 show plots of $\sqrt{U_0}$ against a range of aspect ratios. This range covers both the low and the high aspect ratios encountered in the industry. In a realistic model, the aspect ratio of the elements in regions of interest are usually kept below 1:3. For economy, in areas away from these regions, the aspect ratios may by as high as 1:10. Hence, in this work, as aspect ratio of 1:15 was chosen as the upper limit for comparisons.

Referring to these plots, it is observed that the behaviour of the various elements is similar for any given loading for all three types of analyses.

For the unit thermal loading, element AVEQ does not follow the general trend, under plane stress condition. Further at an aspect ratio of 1:2, the strain energy densities of the elements show a dip indicating that this aspect ratio is not better than a slightly higher one.

For the stretching mode, all elements follow the same trend resulting in almost identical behaviour. Since this is a constant strain deformation mode, the constant strain elements AVEQ and CSTQ exhibit a slightly higher strain energy. This is very noticable when the isoparametric elements exhibit a dip in their strain energy densities at an aspect ratio of 1:2.

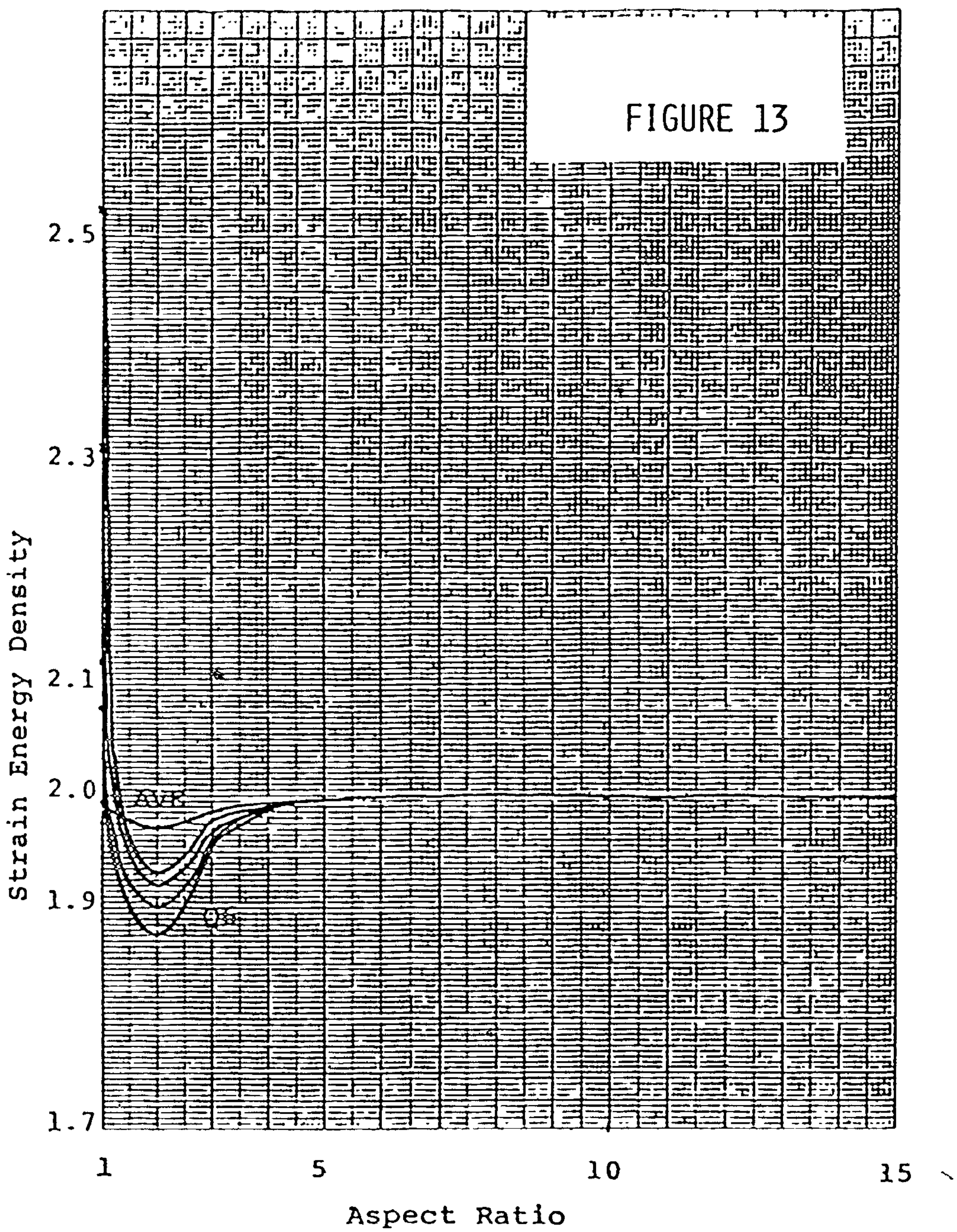


ELM. STRAIN ENERGY DENSITY COMPARISON (RECT. ELM.)

STRAIN ENERGY DENSITIES (SQRT(U/VOL))

TYPE OF ANALYSIS= PLANE STRESS   UNIT THERML MODE

ASP	CST QD	AVE QD	ISO Q4	ISO Q5	ISO Q6
1.00	0.19520E+01	0.16793E+01	0.19300E+01	0.19489E+01	0.19703E+01
2.00	0.17708E+01	0.17128E+01	0.17910E+01	0.18082E+01	0.18128E+01
3.00	0.17837E+01	0.15815E+01	0.18074E+01	0.18178E+01	0.18323E+01
4.00	0.17865E+01	0.15600E+01	0.18123E+01	0.18185E+01	0.18397E+01
5.00	0.17672E+01	0.15537E+01	0.18139E+01	0.18180E+01	0.18432E+01
10.00	0.17413E+01	0.15481E+01	0.18133E+01	0.19145E+01	0.18320E+01
15.00	0.17367E+01	0.15474E+01	0.18120E+01	0.18126E+01	0.18277E+01
20.00	0.17351E+01	0.15472E+01	0.18113E+01	0.18116E+01	0.18259E+01

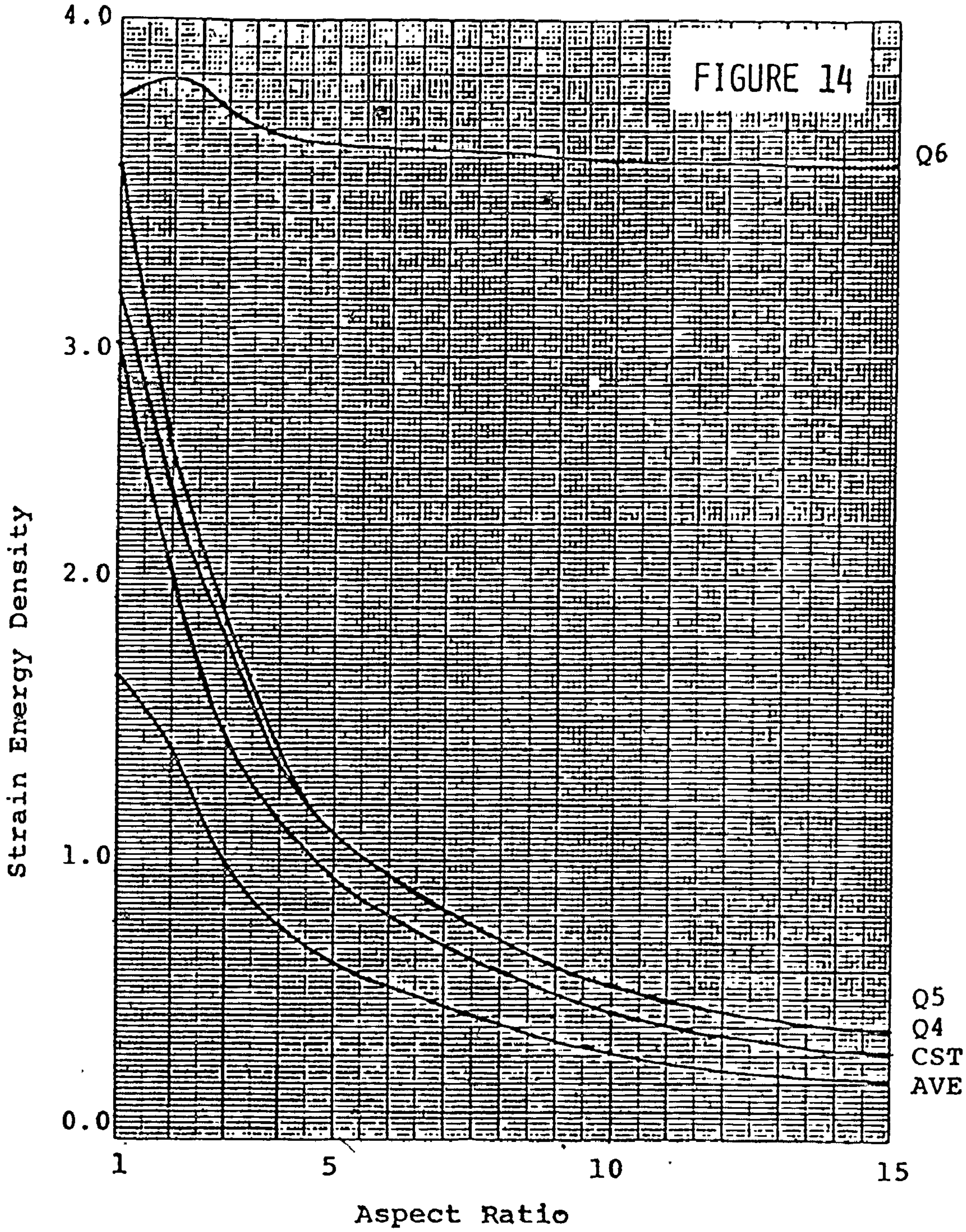


ELM. STRAIN ENERGY DENSITY COMPARISON (RECT. ELM.)

STRAIN ENERGY DENSITIES [SQRT(U/VOL)]

TYPE OF ANALYSIS= PLANE STRESS   STRETCHING MODE

ASP	CST QD	AVE QD	ISO Q4	ISO Q5	ISO Q6
1.00	0.25265E+01	0.19862E+01	0.23139E+01	0.21203E+01	0.20874E+01
2.00	0.19332E+01	0.19717E+01	0.19182E+01	0.19032E+01	0.18768E+01
3.00	0.19780E+01	0.19867E+01	0.19743E+01	0.19731E+01	0.19654E+01
4.00	0.19887E+01	0.19926E+01	0.19871E+01	0.19868E+01	0.19830E+01
5.00	0.19931E+01	0.19953E+01	0.19921E+01	0.19920E+01	0.19898E+01
10.00	0.19984E+01	0.19988E+01	0.19982E+01	0.19981E+01	0.19976E+01
15.00	0.19993E+01	0.19995E+01	0.19992E+01	0.19992E+01	0.19990E+01
20.00	0.19996E+01	0.19997E+01	0.19995E+01	0.19995E+01	0.19994E+01



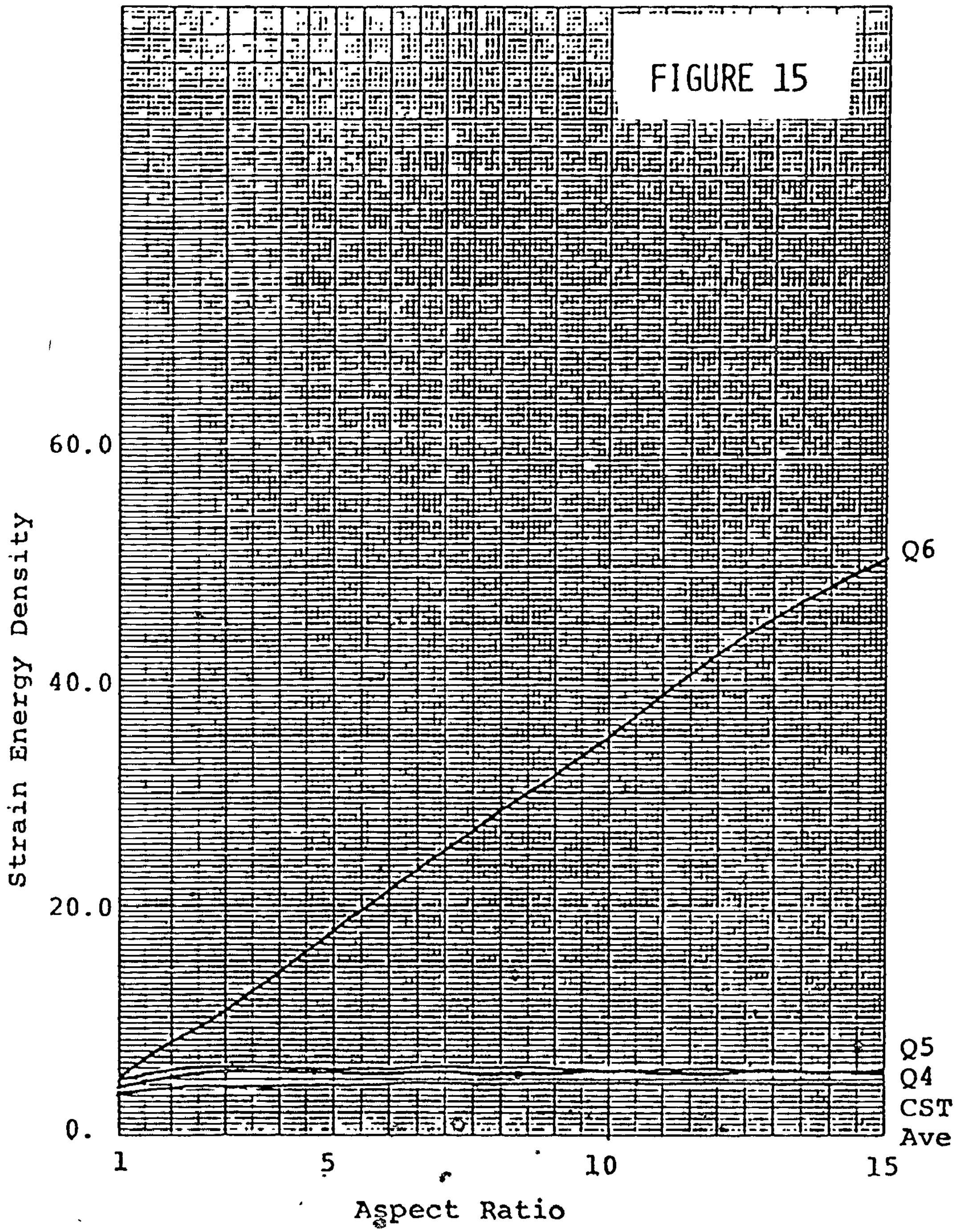
ELM. STRAIN ENERGY DENSITY COMPARISON (RECT. ELM.)

STRAIN ENERGY DENSITIES [SQRT(U/VOL)]

TYPE OF ANALYSIS= PLANE STRESS   BENDING MODE

ASP	CST QD	AVE QD	ISO Q4	ISO Q5	ISO Q6
1.00	0.28491E+01	0.16536E+01	0.30338E+01	0.34725E+01	0.37240E+01
2.00	0.20481E+01	0.14055E+01	0.22844E+01	0.24870E+01	0.37881E+01
3.00	0.14616E+01	0.98516E+00	0.16994E+01	0.17784E+01	0.36451E+01
4.00	0.11715E+01	0.76179E+00	0.13332E+01	0.13696E+01	0.35787E+01
5.00	0.92778E+00	0.62056E+00	0.10894E+01	0.11087E+01	0.35471E+01
10.00	0.45776E+00	0.31914E+00	0.55798E+00	0.56065E+00	0.34839E+01
15.00	0.30452E+00	0.21398E+00	0.37267E+00	0.37353E+00	0.34724E+01
20.00	0.22823E+00	0.16082E+00	0.27951E+00	0.27989E+00	0.34686E+01

FIGURE 15



ELM. STRAIN ENERGY DENSITY COMPARISON (RECT. ELM.)

STRAIN ENERGY DENSITIES [SQRT(U/VOL)]

TYPE OF ANALYSIS= PLANE STRESS SHEAR MODE.

ASP	CST QD	AVE QD	ISO Q4	ISO Q5	ISO Q6
1.00	0.47746E+01	0.38797E+01	0.49374E+01	0.53105E+01	0.55303E+01
2.00	0.52415E+01	0.41803E+01	0.56481E+01	0.60082E+01	0.82976E+01
3.00	0.54129E+01	0.43443E+01	0.60022E+01	0.62315E+01	0.11380E+02
4.00	0.54844E+01	0.44259E+01	0.61732E+01	0.63221E+01	0.14623E+02
5.00	0.55194E+01	0.44699E+01	0.62643E+01	0.63665E+01	0.17945E+02
10.00	0.55686E+01	0.45363E+01	0.64002E+01	0.64285E+01	0.34961E+02
15.00	0.55781E+01	0.45497E+01	0.64275E+01	0.64403E+01	0.52176E+02
20.00	0.55814E+01	0.45545E+01	0.64372E+01	0.64444E+01	0.69443E+02

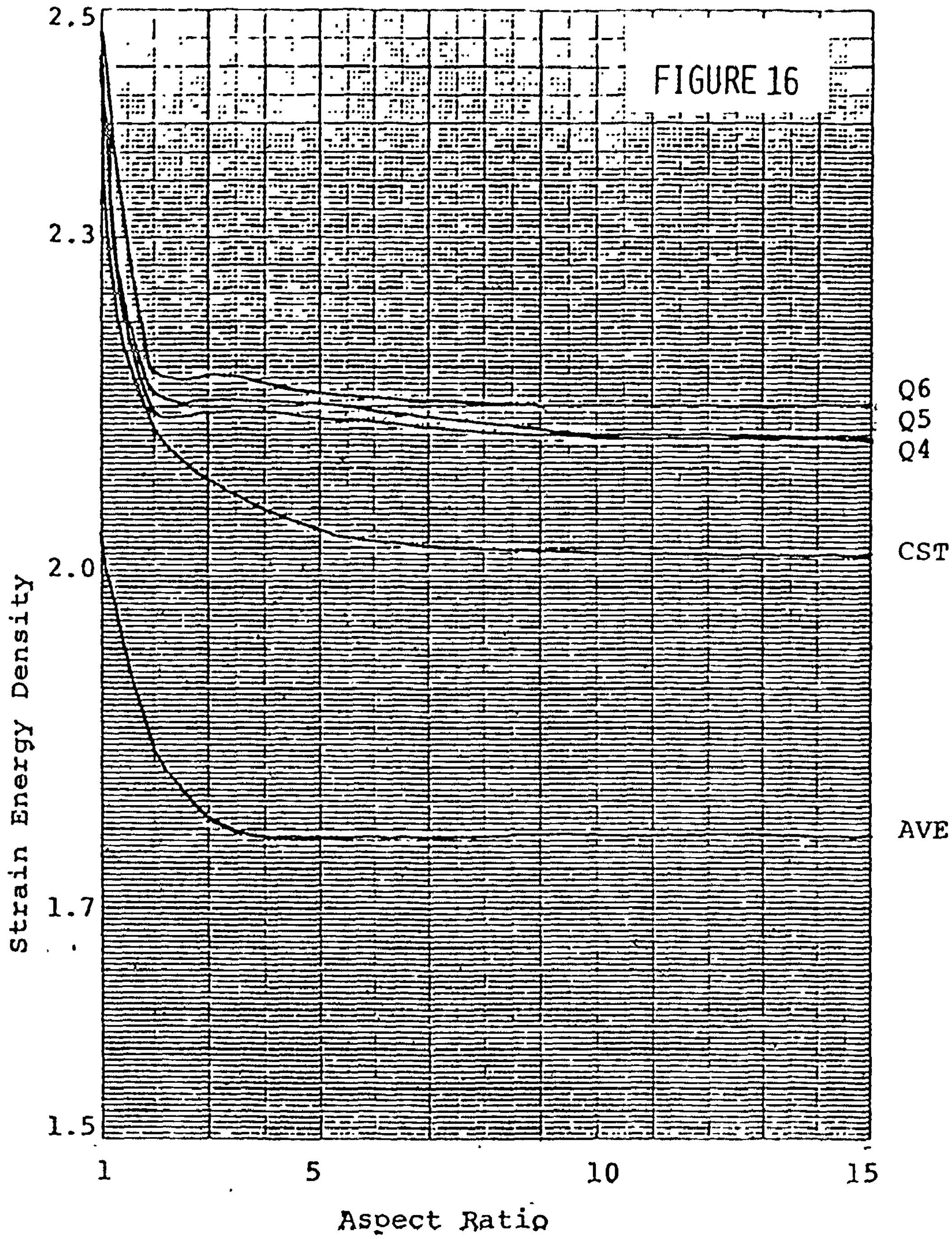


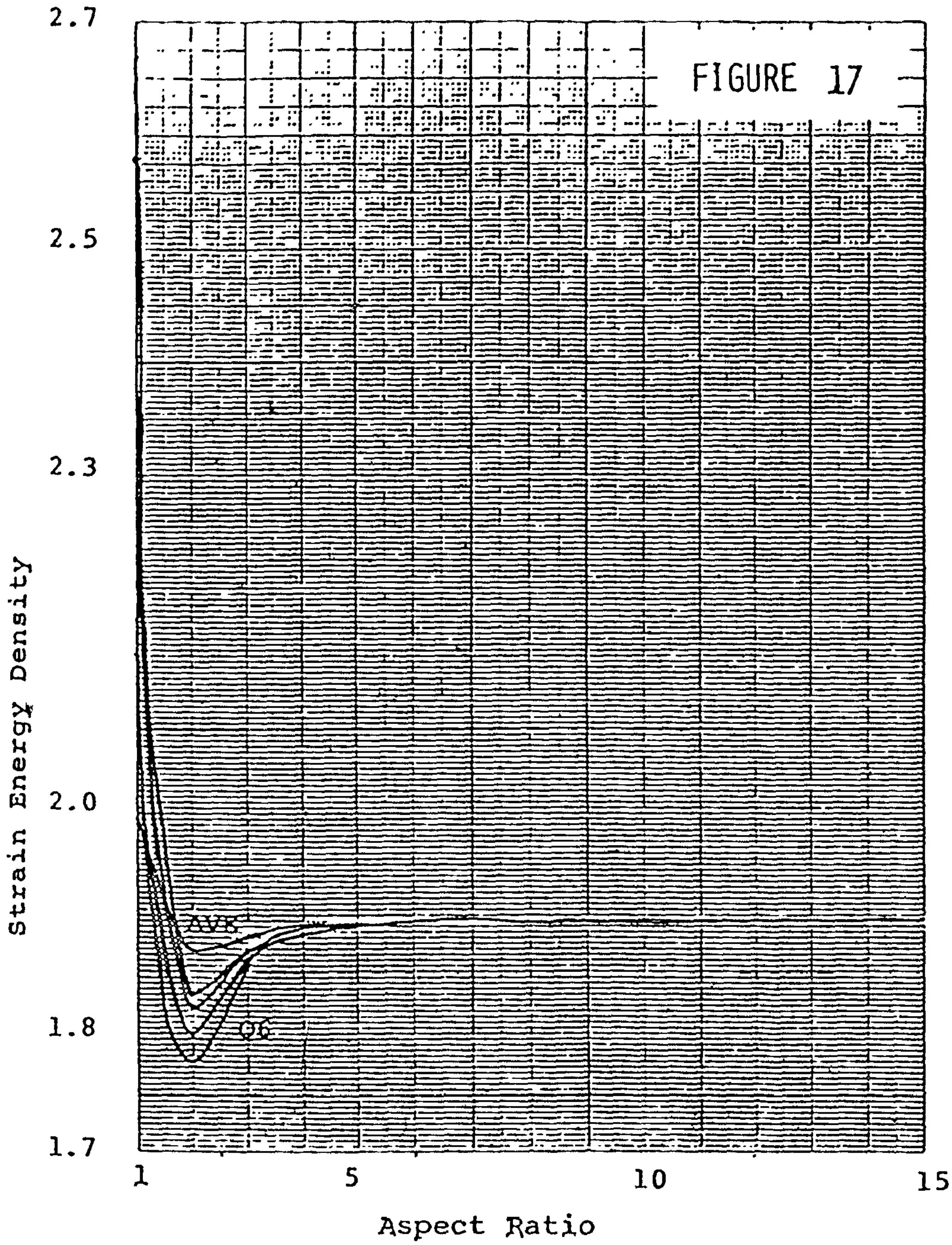
FIGURE 16

ELM. STRAIN ENERGY DENSITY COMPARISON (RECT. ELM.)

STRAIN ENERGY DENSITIES [SQRT(U/VOL)]

TYPE OF ANALYSIS= PLANE STRAIN   UNIT THERML MODE

ASP	CST QD	AVE QD	ISO Q4	ISO Q5	ISO Q6
1.00	0.24509E+01	0.20374E+01	0.24393E+01	0.24580E+01	0.24749E+01
2.00	0.21305E+01	0.18437E+01	0.21426E+01	0.21645E+01	0.21805E+01
3.00	0.20874E+01	0.17858E+01	0.21439E+01	0.21544E+01	0.21765E+01
4.00	0.20539E+01	0.17795E+01	0.21467E+01	0.21534E+01	0.21722E+01
5.00	0.20399E+01	0.17780E+01	0.21392E+01	0.21526E+01	0.21622E+01
10.00	0.20217E+01	0.17774E+01	0.21211E+01	0.21258E+01	0.21509E+01
15.00	0.20184E+01	0.17774E+01	0.21169E+01	0.21188E+01	0.21491E+01
20.00	0.20172E+01	0.17774E+01	0.21154E+01	0.21163E+01	0.21484E+01

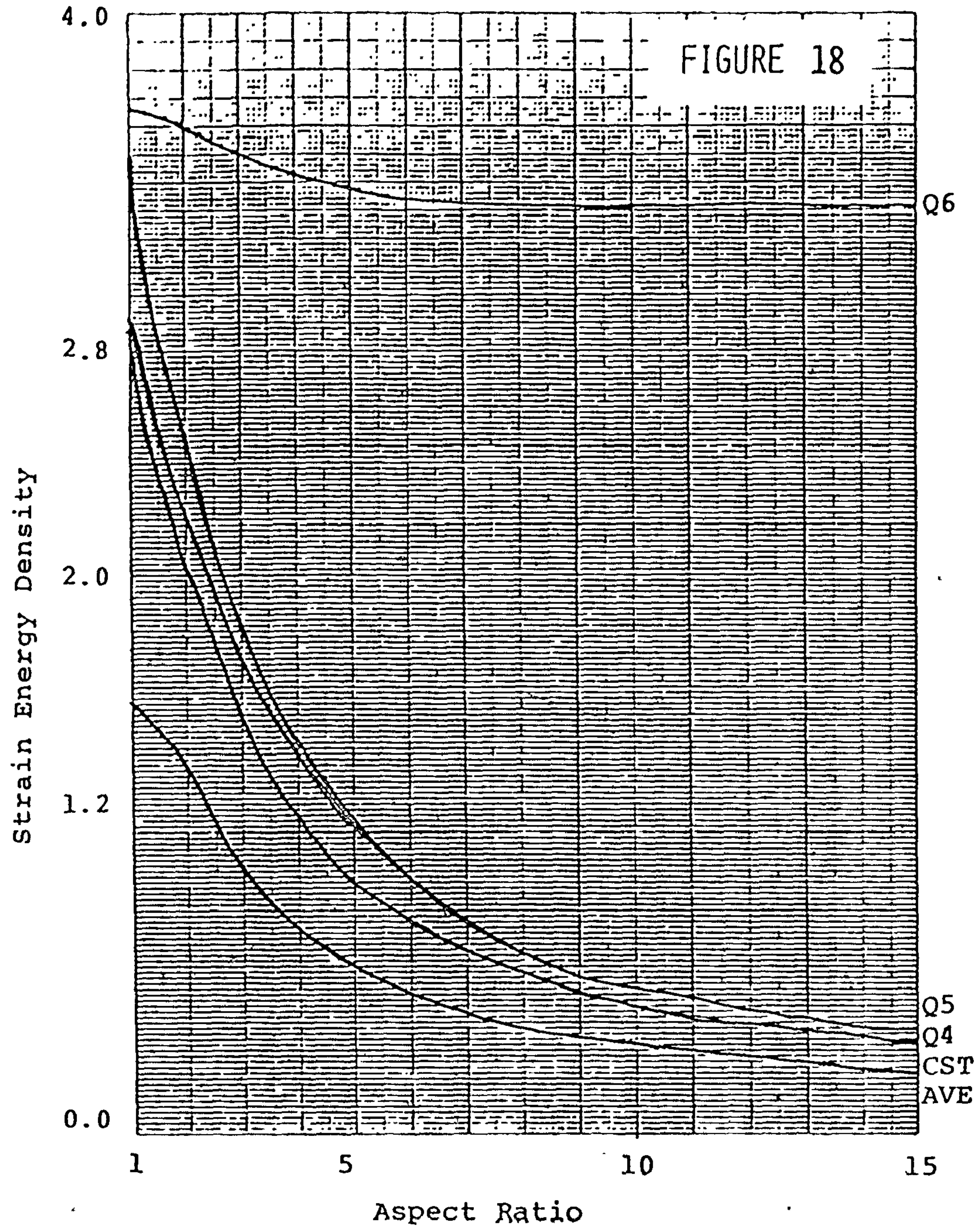


ELM. STRAIN ENERGY DENSITY COMPARISON (RECT. ELM.)

STRAIN ENERGY DENSITIES [SQRT(U/VOL)]

TYPE OF ANALYSIS= PLANE STRAIN   STRETCHING MODE

ASP	CST QD	AVE QD	ISO Q4	ISO Q5	ISO Q6
1.00	0.26776E+01	0.19954E+01	0.25835E+01	0.21425E+01	0.21216E+01
2.00	0.18384E+01	0.18850E+01	0.18263E+01	0.18041E+01	0.17818E+01
3.00	0.18634E+01	0.18958E+01	0.18798E+01	0.18772E+01	0.18670E+01
4.00	0.18953E+01	0.19009E+01	0.18935E+01	0.18929E+01	0.18876E+01
5.00	0.19002E+01	0.19034E+01	0.18990E+01	0.18989E+01	0.18956E+01
10.00	0.19061E+01	0.19067E+01	0.19058E+01	0.19058E+01	0.19050E+01
15.00	0.19071E+01	0.19074E+01	0.19070E+01	0.19070E+01	0.19066E+01
20.00	0.19074E+01	0.19076E+01	0.19074E+01	0.19074E+01	0.19072E+01



ELM. STRAIN ENERGY DENSITY COMPARISON (RECT. ELM.)

STRAIN ENERGY DENSITIES [SQRT(U/VOL)]

TYPE OF ANALYSIS= PLANE STRAIN   BENDING MODE

ASP	CST QD	AVE QD	ISO Q4	ISO Q5	ISO Q6
1.00	0.28726E+01	0.15504E+01	0.29106E+01	0.34984E+01	0.36627E+01
2.00	0.20401E+01	0.13077E+01	0.22081E+01	0.24925E+01	0.36190E+01
3.00	0.14976E+01	0.94557E+00	0.16738E+01	0.18100E+01	0.34899E+01
4.00	0.11223E+01	0.74424E+00	0.13539E+01	0.13894E+01	0.34314E+01
5.00	0.90101E+00	0.61125E+00	0.11083E+01	0.11361E+01	0.33807E+01
10.00	0.45436E+00	0.31790E+00	0.55893E+00	0.56535E+00	0.33217E+01
15.00	0.30353E+00	0.21361E+00	0.37266E+00	0.37437E+00	0.33120E+01
20.00	0.22781E+00	0.16066E+00	0.27944E+00	0.28012E+00	0.33087E+01

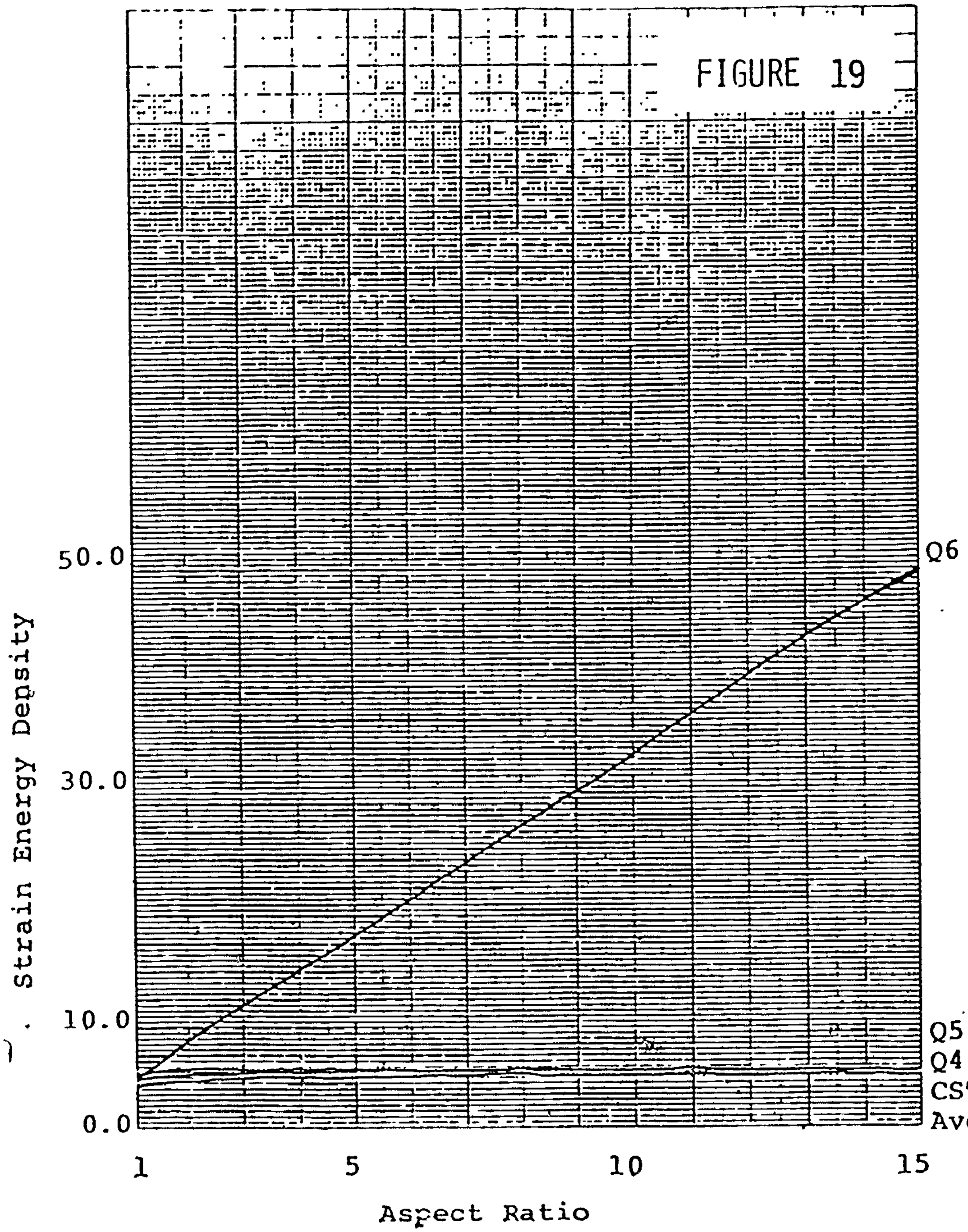


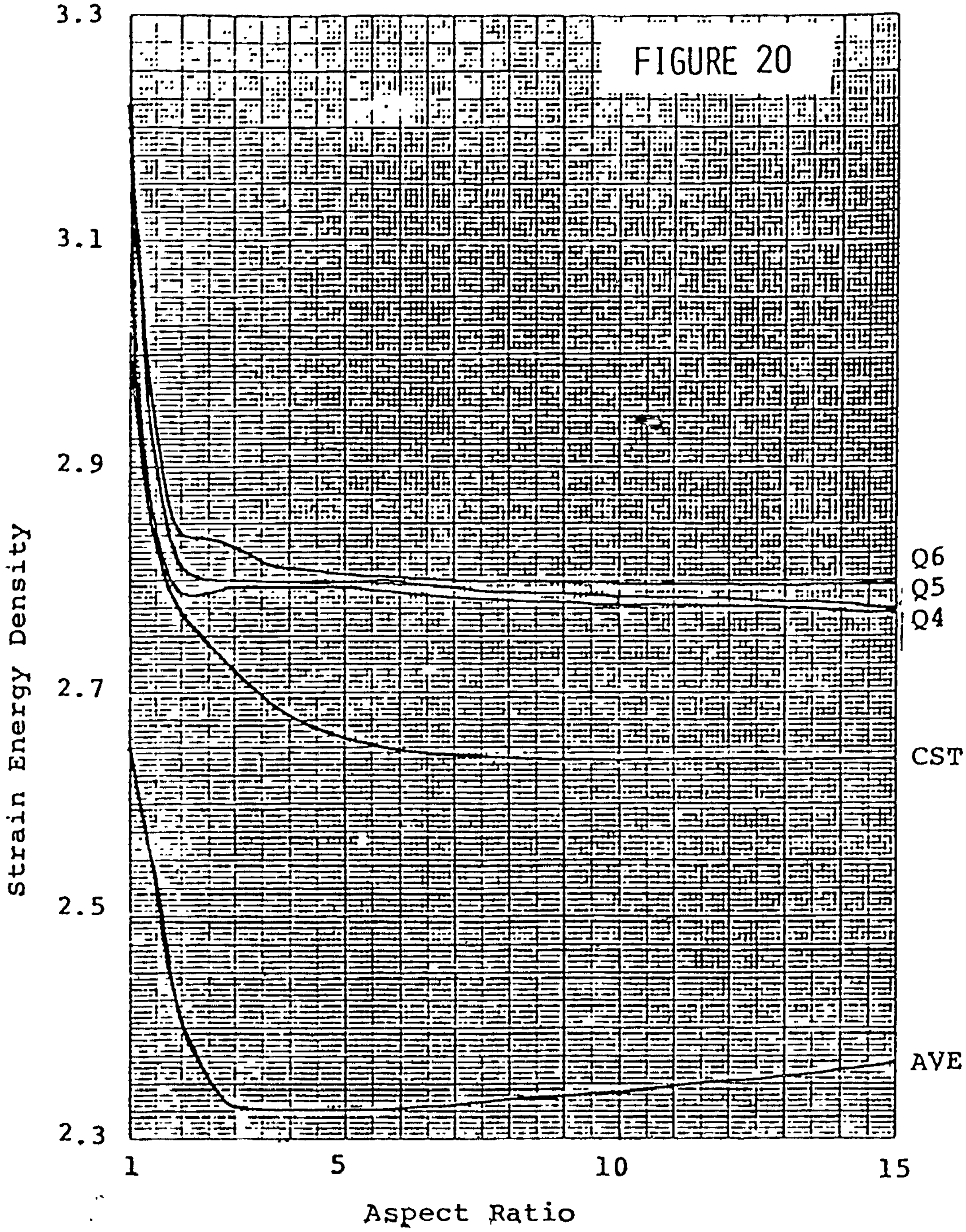
FIGURE 19

ELM. STRAIN ENERGY DENSITY COMPARISON (RECT. ELM.)

STRAIN ENERGY DENSITIES [SQRT(U/VOL)]

TYPE OF ANALYSIS= PLANE STRAIN   SHEAR MODE

ASP	CST QD	AVE QD	ISO Q4	ISO Q5	ISO Q6
1.00	0.47082E+01	0.37733E+01	0.47471E+01	0.52443E+01	0.53858E+01
2.00	0.52155E+01	0.40957E+01	0.54814E+01	0.59786E+01	0.79674E+01
3.00	0.54016E+01	0.42884E+01	0.58917E+01	0.62178E+01	0.10896E+02
4.00	0.54774E+01	0.43885E+01	0.60995E+01	0.63140E+01	0.13982E+02
5.00	0.55150E+01	0.44438E+01	0.62127E+01	0.63613E+01	0.17145E+02
10.00	0.55675E+01	0.45289E+01	0.63856E+01	0.64271E+01	0.33365E+02
15.00	0.55776E+01	0.45463E+01	0.64208E+01	0.64397E+01	0.49782E+02
20.00	0.55811E+01	0.45526E+01	0.64334E+01	0.64441E+01	0.66252E+02

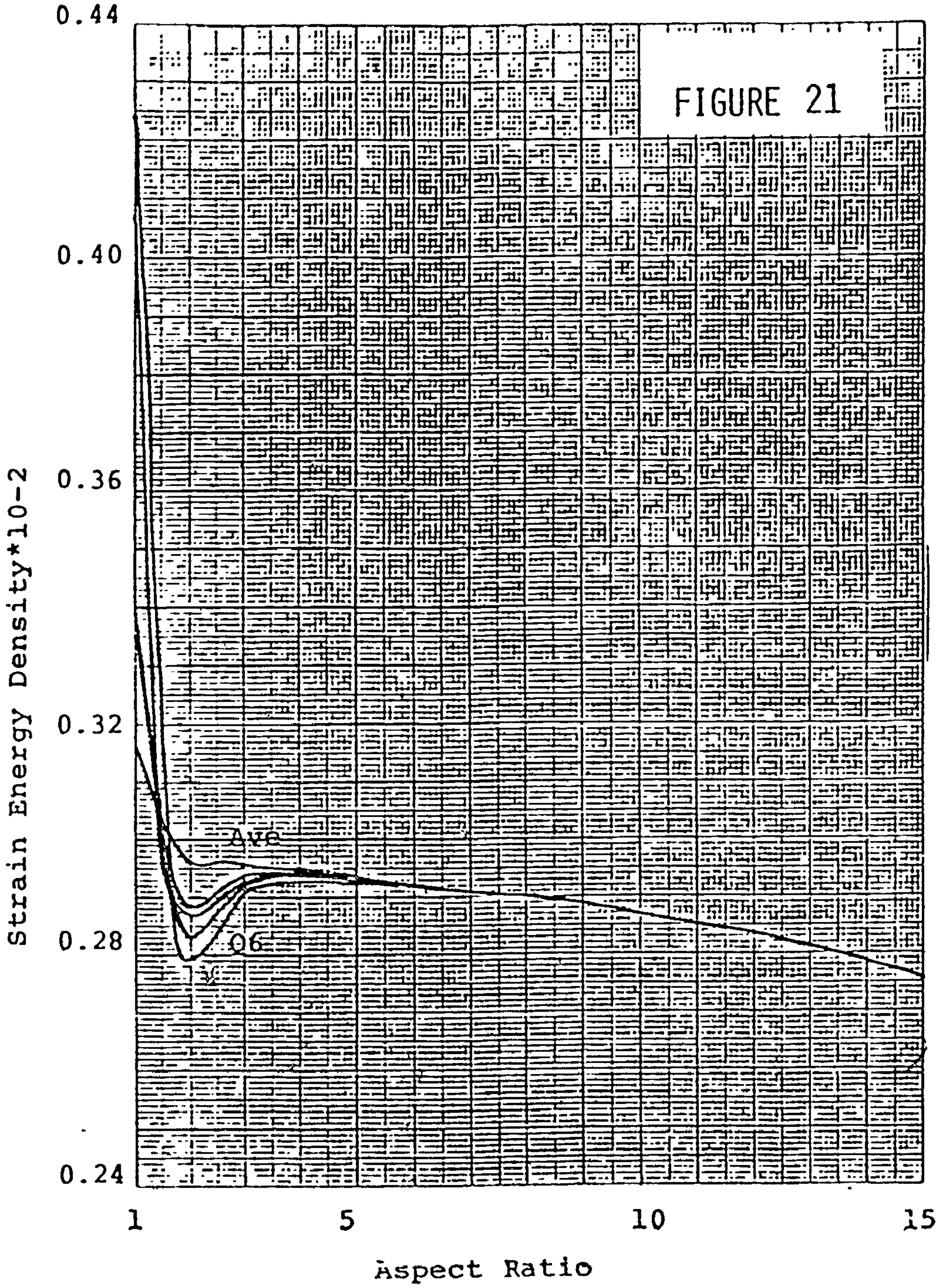


ELM. STRAIN ENERGY DENSITY COMPARISON (RECT. ELM.)

STRAIN ENERGY DENSITIES [SQRT(U/VOL)]

TYPE OF ANALYSIS= AXISYMMETRIC   UNIT THERML MODE

ASP	CST QD	AVE QD	ISO Q4	ISO Q5	ISO Q6
1.00	0.31913E+01	0.26517E+01	0.31747E+01	0.32005E+01	0.32211E+01
2.00	0.27704E+01	0.24078E+01	0.27855E+01	0.26144E+01	0.28352E+01
3.00	0.27196E+01	0.23323E+01	0.27869E+01	0.28002E+01	0.28294E+01
4.00	0.26755E+01	0.23265E+01	0.27902E+01	0.27992E+01	0.28263E+01
5.00	0.26575E+01	0.23274E+01	0.27914E+01	0.27979E+01	0.28129E+01
10.00	0.26378E+01	0.23405E+01	0.27706E+01	0.27790E+01	0.27982E+01
15.00	0.26376E+01	0.23535E+01	0.27688E+01	0.27730E+01	0.27962E+01
20.00	0.26401E+01	0.23655E+01	0.27735E+01	0.27767E+01	0.27960E+01

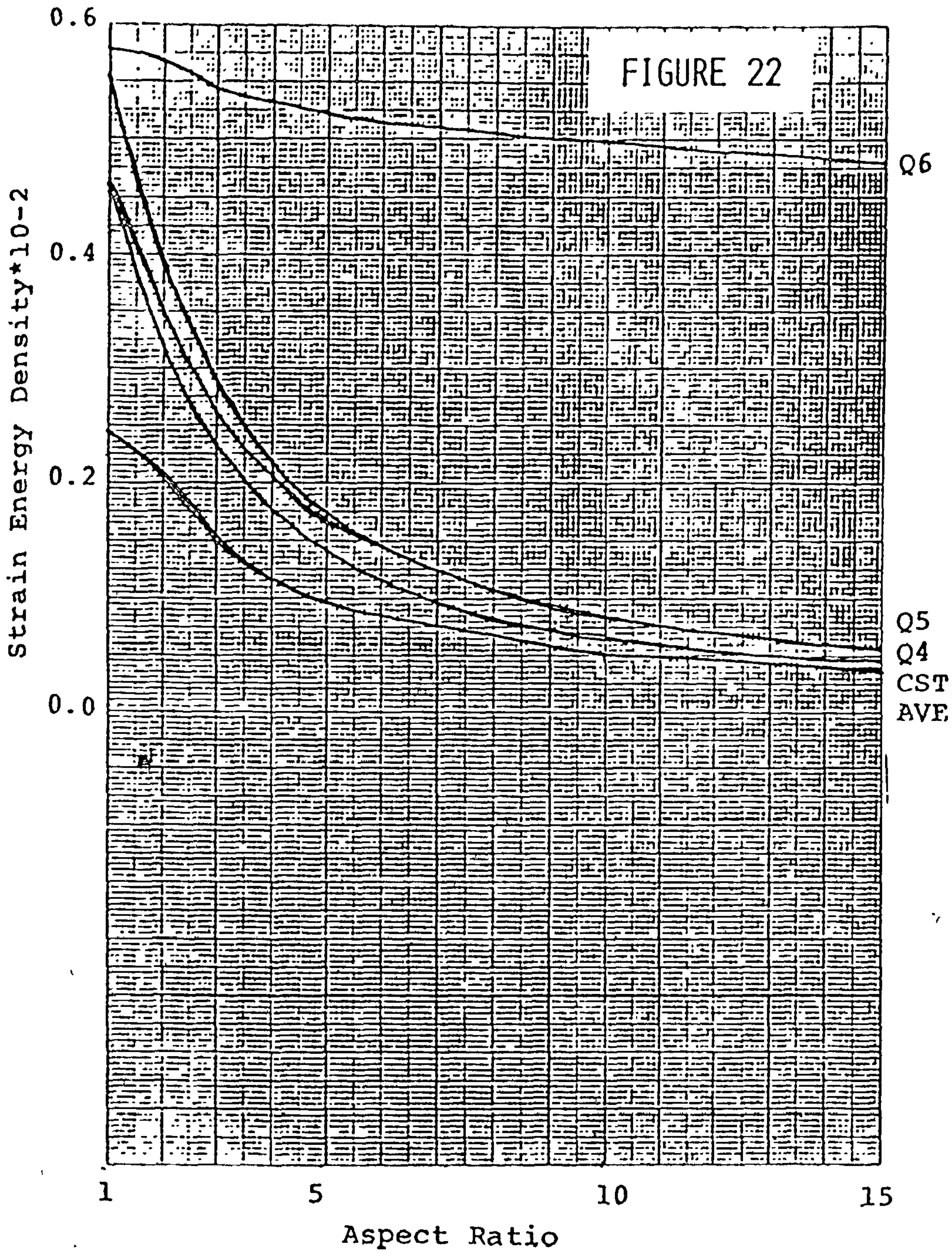


ELM. STRAIN ENERGY DENSITY COMPARISON (RECT. ELM.)

STRAIN ENERGY DENSITIES [SQRT(U/VOL)]

TYPE OF ANALYSIS= AXISYMMETRIC STRETCHING MODE

ASP	CST QD	AVE QD	ISO Q4	ISO Q5	ISO Q6
1.00	0.42360E-02	0.31586E-02	0.40787E-02	0.33922E-02	0.33576E-02
2.00	0.28879E-02	0.29621E-02	0.28696E-02	0.28337E-02	0.27982E-02
3.00	0.29402E-02	0.29596E-02	0.29347E-02	0.29304E-02	0.29135E-02
4.00	0.29397E-02	0.29484E-02	0.29370E-02	0.29360E-02	0.29271E-02
5.00	0.29283E-02	0.29333E-02	0.29267E-02	0.29263E-02	0.29209E-02
10.00	0.28451E-02	0.28464E-02	0.28444E-02	0.28444E-02	0.28439E-02
15.00	0.27586E-02	0.27596E-02	0.27578E-02	0.27578E-02	0.27590E-02
20.00	0.26752E-02	0.26763E-02	0.26741E-02	0.26741E-02	0.26766E-02

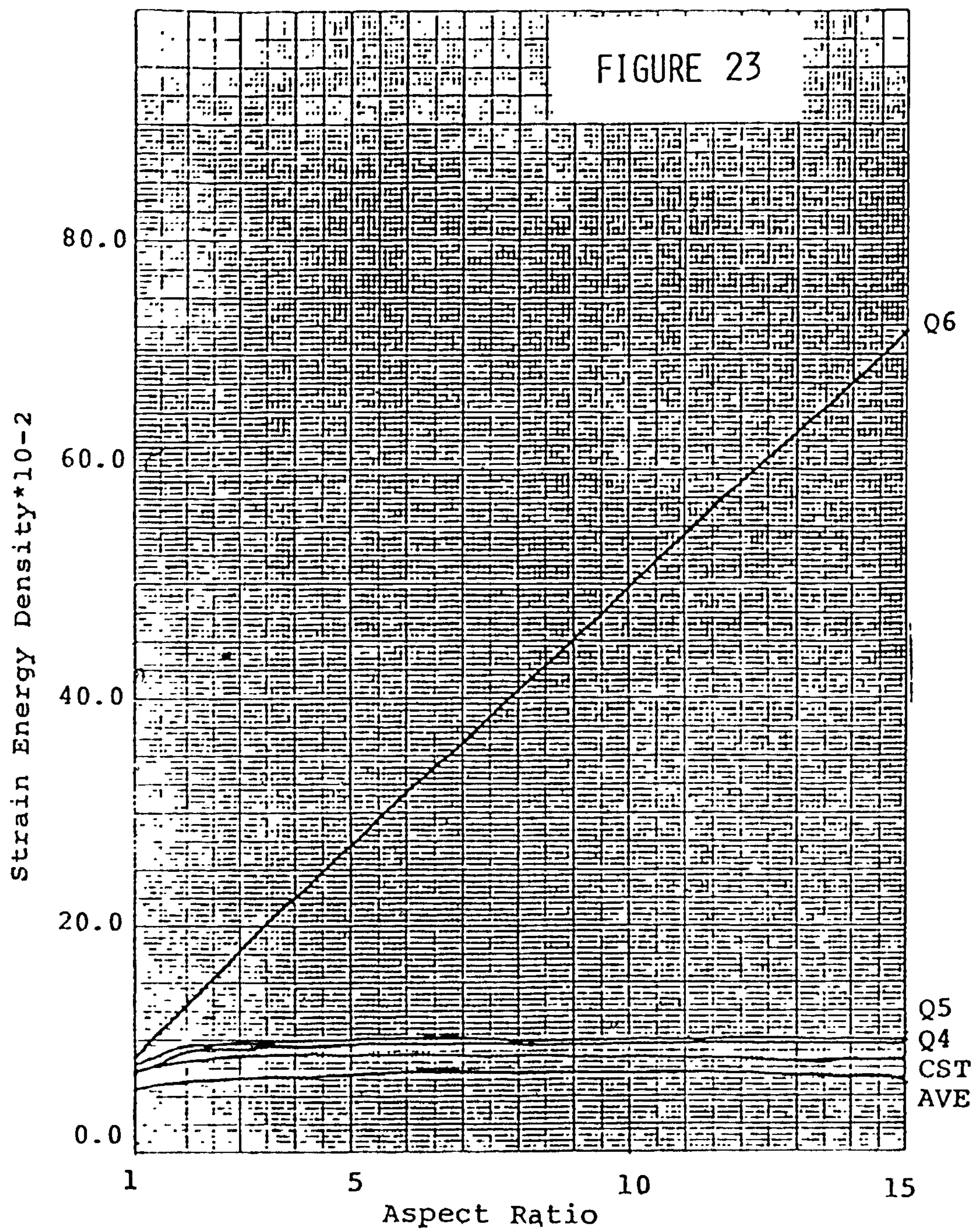


ELM. STRAIN ENERGY DENSITY COMPARISON (RECT. ELM.)

STRAIN ENERGY DENSITIES [SQRT(U/VOL)]

TYPE OF ANALYSIS= AXISYMMETRIC BENDING MODE

ASP	CST QD	AVE QD	ISO Q4	ISO Q5	ISO Q6
1.00	0.45489E-02	0.24525E-02	0.46074E-02	0.55377E-02	0.57960E-02
2.00	0.32061E-02	0.20639E-02	0.34745E-02	0.39207E-02	0.56851E-02
3.00	0.23537E-02	0.14830E-02	0.26113E-02	0.28278E-02	0.54437E-02
4.00	0.17535E-02	0.11612E-02	0.20846E-02	0.21680E-02	0.53245E-02
5.00	0.14003E-02	0.94903E-03	0.17333E-02	0.17477E-02	0.52109E-02
10.00	0.68907E-03	0.48187E-03	0.85670E-03	0.86959E-03	0.49573E-02
15.00	0.44969E-03	0.31632E-03	0.55991E-03	0.56396E-03	0.47889E-02
20.00	0.32996E-03	0.23254E-03	0.41310E-03	0.41520E-03	0.46376E-02



ELM. STRAIN ENERGY DENSITY COMPARISON (RECT. ELM.)

STRAIN ENERGY DENSITIES [SQRT(U/VOL)]

TYPE OF ANALYSIS= AXISYMMETRIC SHEAR MODE

ASP	CST QD	AVE QD	ISO Q4	ISO Q5	ISO Q6
1.00	0.74578E-02	0.59768E-02	0.75179E-02	0.83048E-02	0.85242E-02
2.00	0.82285E-02	0.64599E-02	0.86430E-02	0.94316E-02	0.12523E-01
3.00	0.84915E-02	0.67364E-02	0.92570E-02	0.97757E-02	0.17014E-01
4.00	0.85790E-02	0.68662E-02	0.95503E-02	0.96926E-02	0.21691E-01
5.00	0.86059E-02	0.69249E-02	0.96940E-02	0.99316E-02	0.26425E-01
10.00	0.85278E-02	0.69180E-02	0.97903E-02	0.98574E-02	0.49798E-01
15.00	0.63878E-02	0.68093E-02	0.96736E-02	0.97043E-02	0.71988E-01
20.00	0.82431E-02	0.66882E-02	0.95271E-02	0.95446E-02	0.92867E-01

For the bending mode, the strain energy density of the Q6 element remains almost constant, whereas it falls off with increasing aspect ratio for the other elements.

For the shear mode, the strain energy density of the Q6 element increases with aspect ratio whereas for the other elements the strain energy density remains almost constant.

From these strain energy density curves, it is observed that the strain energy density falls off with increasing aspect ratio, initially very steeply and then levelling off. This can be interpreted as, there will be a steep fall in accuracy initially and then the accuracy almost remains constant. Further, it is observed that, in general the isoparametric family of elements exhibit higher strain energy densities indicating their superiority. At low aspect ratios, the elements behave quite differently and discretion must be used while selecting.

Further, at an aspect ratio of 1:2, there seems to be a discontinuity dip in the curves for the isoparametric elements. However, the fall in strain energy density is quite small compared to the strain energy density at an aspect ratio of 1:1. This area needs to be further investigated.

4.4 Comparison in Small Assemblage

The effect of element shape on the results in small assemblage for the various formulations were studied in plane stress, plane strain and axisymmetric conditions.

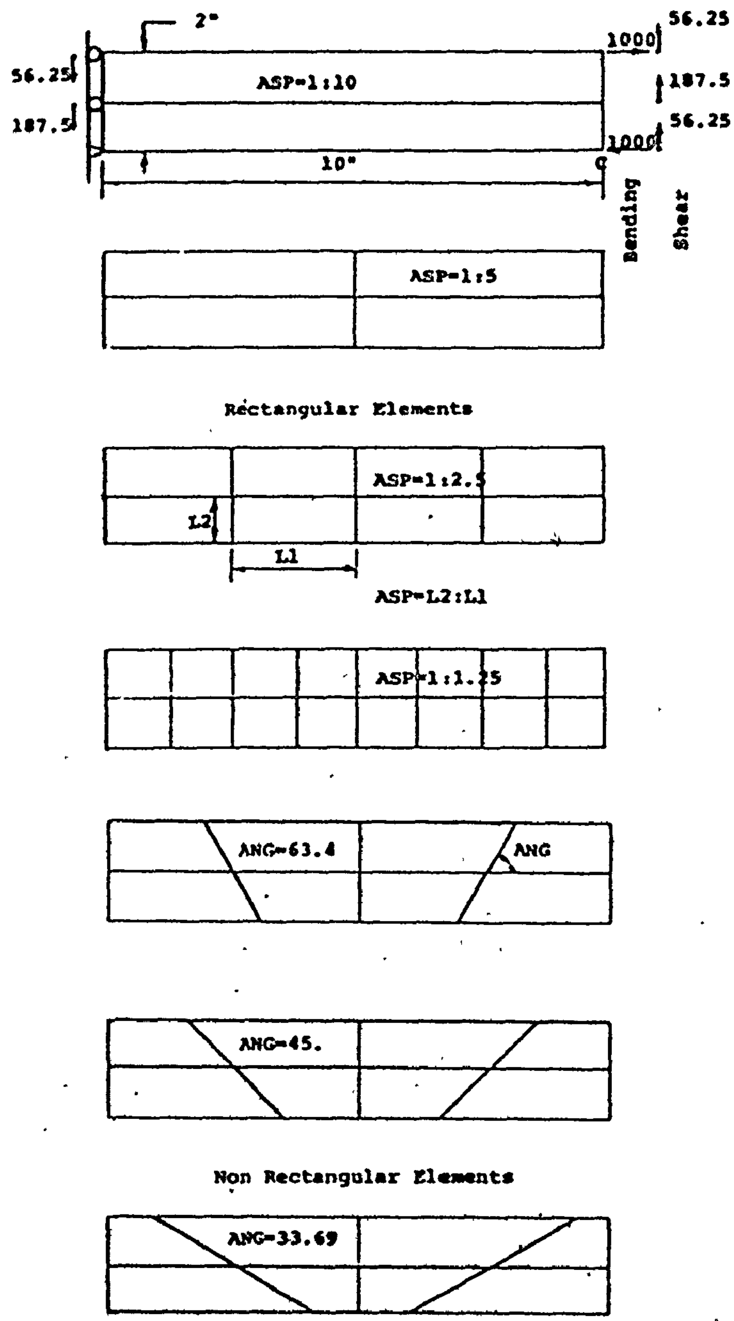
(a) Plane Stress Condition

A cantilever beam under bending and shear loads was considered for comparison. The comparison was carried out using both rectangular and non-rectangular elements. For the assemblage with rectangular elements, the comparison was carried out at aspect ratios 1:1.25, 1:5.0 and 1:10.0. For the assemblage with non-rectangular elements, the comparison was carried out with included angles of 90°, 63.4°, 45° and 33.69°. The models used are shown in Figure 24.

The tip deflection is plotted against the aspect ratio or the included angle for comparison in Figures 25 through 27.

All elements except the Q6 element follow similar trends. The accuracy of the Q6 element remains unchanged with change in aspect ratio for rectangular elements under the bending load. Its accuracy falls off only slightly under shear loading and for non-rectangular elements under both loadings.

For rectangular elements, Q4 and Q5 elements behave very similarly under both loadings, the Q5 element being only marginally better. The tip defelction for the Q5 element at the low aspect ratio of 1:1.25 is 90% of the theoretical under bending and 94% of the theoretical under shear. At the high aspect ratio of 1:10, the deflection is only 8.5% of the theoretical under bending and 9% of the theoretical under shear. At the low aspect ratio (1:1.25) the difference in results between the Q5 element and the AVEQ element is



E=1500.   v=0.25
Deflections compared at point C

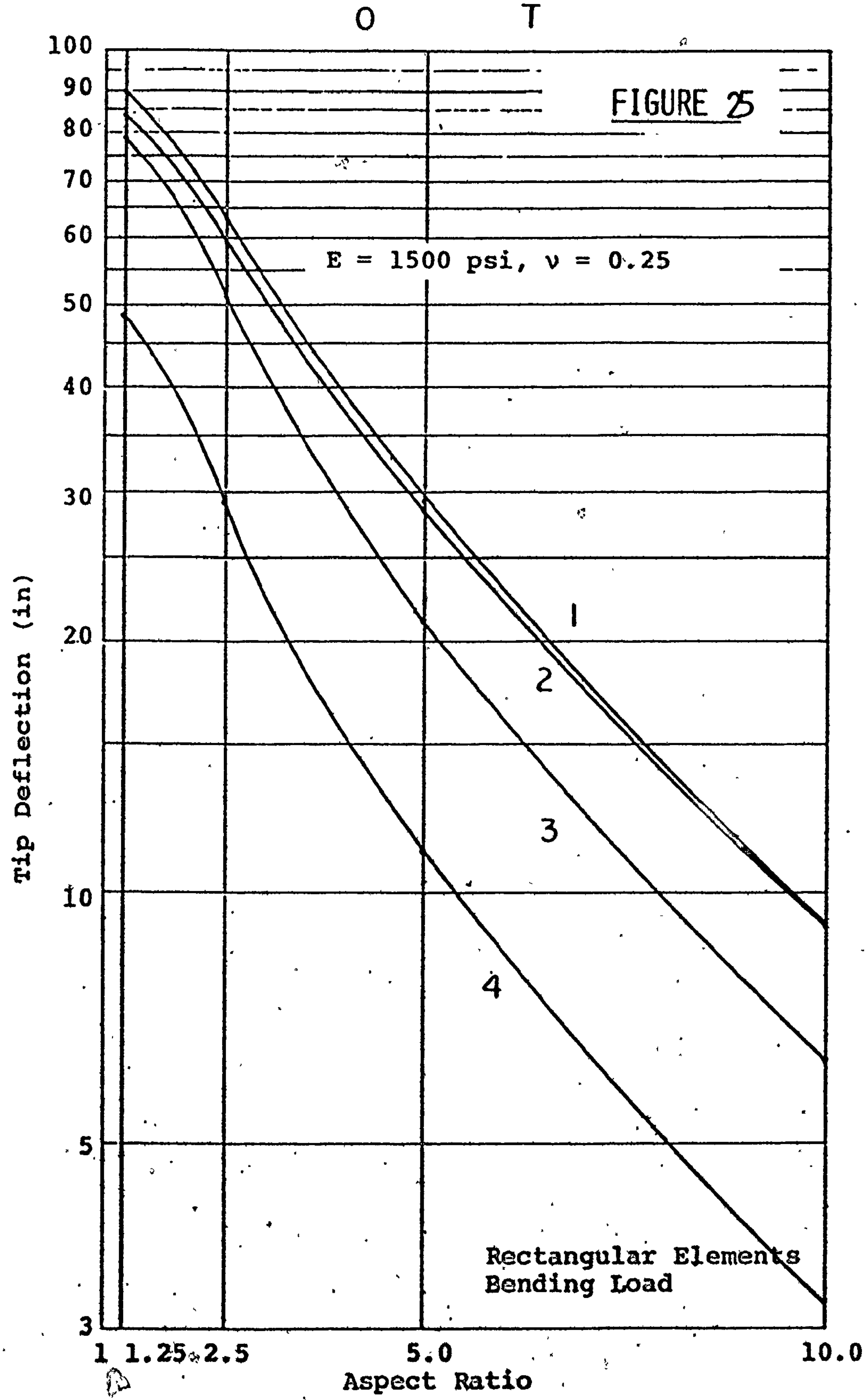
Legend for plots(figs. 28,29 and 30)

0 - Q6 Elm.
1 - Q5
2 - Q4
3 - CST
4 - AVE

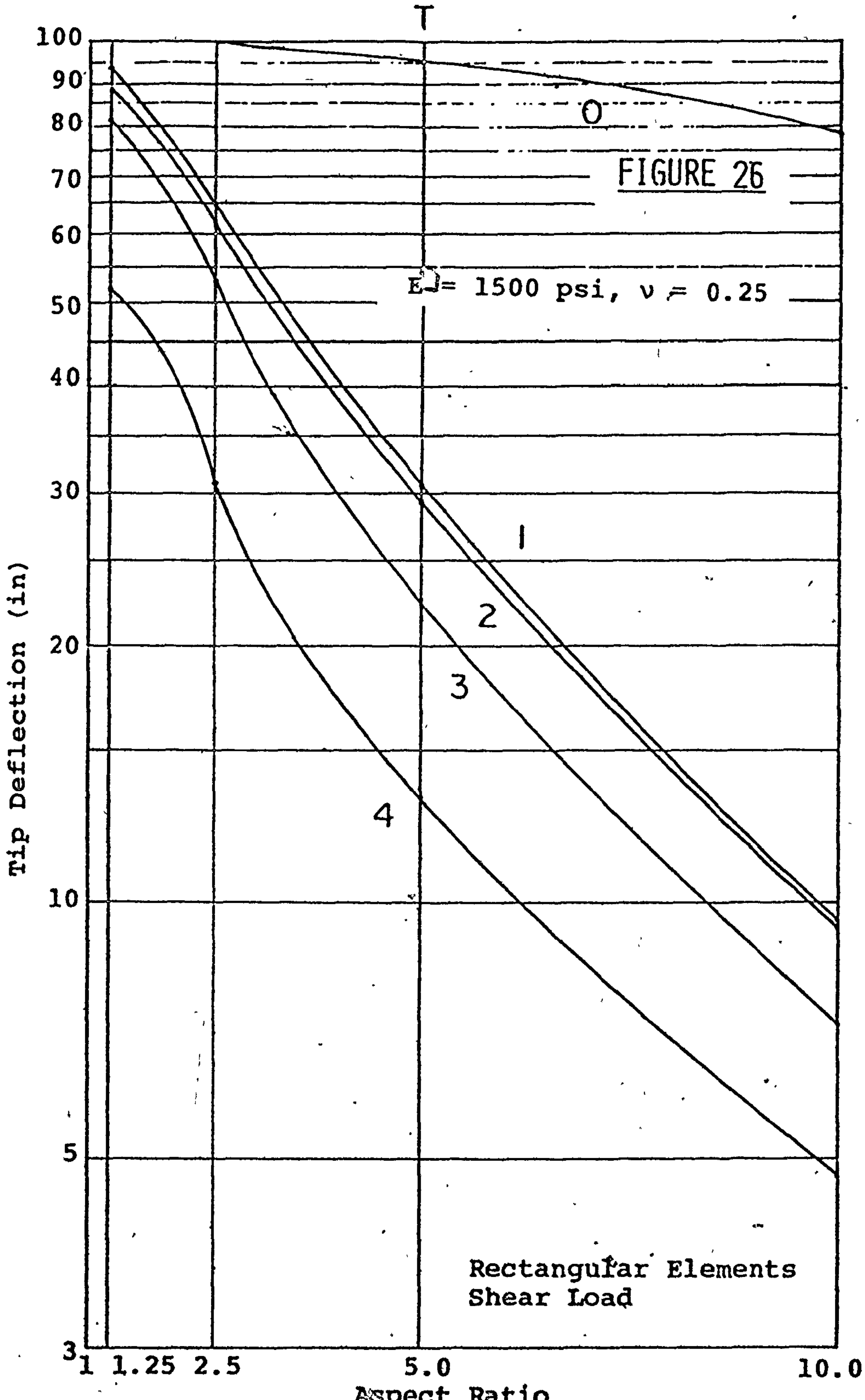
T - Theory
100" - Bending
103" - Shear

TEST CASE FOR COMPARISON OF ELEMENTS IN ASSEMBLAGE (PLANE STRESS)

FIGURE 2b



EFFECT OF ASPECT RATIO ON TIP DEFLECTION
PLANE STRESS ANALYSIS



EFFECT OF ASPECT RATIO ON TIP DEFLECTION

PLANE STRESS ANALYSIS

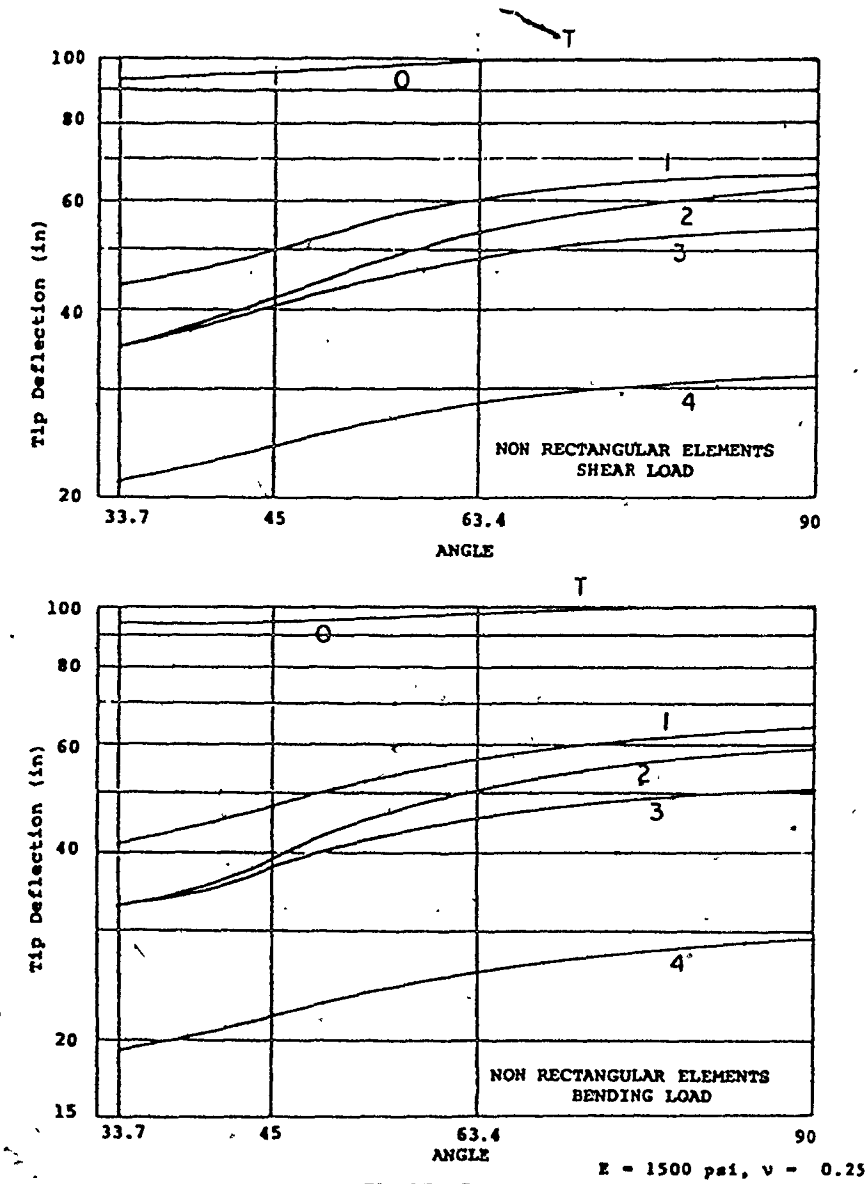


FIGURE 27

EFFECT OF DISTORTION ON TIP DEFLECTION
PLANE STRESS ANALYSIS

about 42% of the theoretical value whereas at the high aspect ratio (1:10) the difference is reduced to only about 5% of the theoretical value.

For non-rectangular elements, the CSTQ element approaches the behaviour of Q4 element at low included angle and at larger included angle, the element Q4 behaves very similarly to the element Q5, under both bending and shear. At an included angle of 90° (aspect ratio 1:2.5), the Q5 element results in about 65% of the theoretical value in both bending and shear, whereas at very acute included angle of 33.69°, it drops to about 45% of the theoretical value. Under the same conditions the AVEQ element results in about 30% of the theoretical value at an included angle of 90° and drops to about 19% of the theoretical value under bending and to about 22% of the theoretical value under shear.

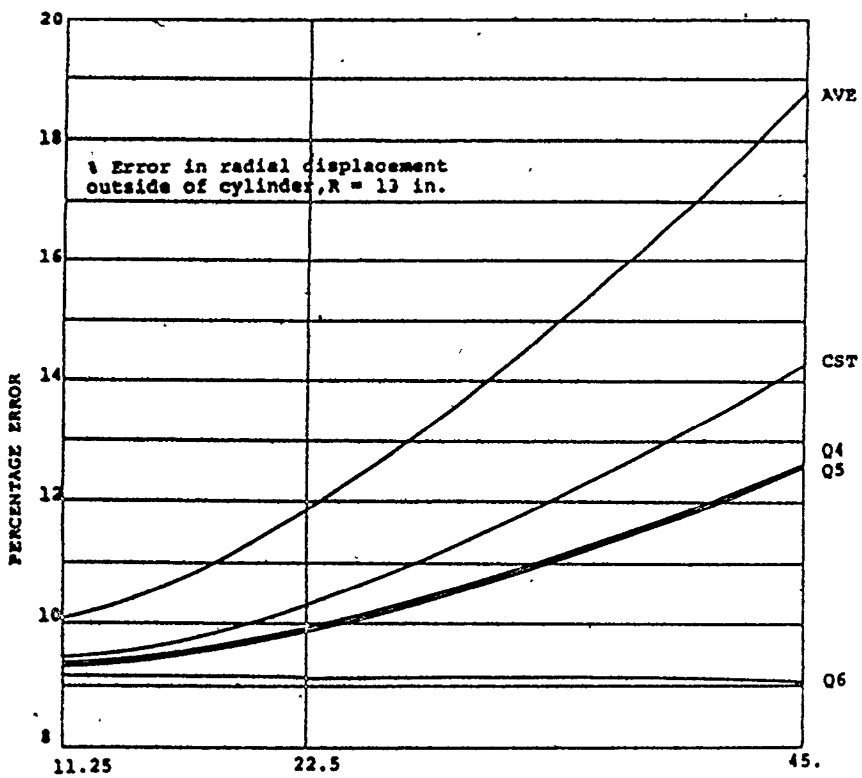
(b) Plane Strain Condition

A long cylinder under mechanical loading was considered for comparison. The radial displacements under internal pressure loading at both inside and outside radii were compared, for the various element formulations. The comparisons were made at sector angles 11.25°, 22.5° and 45°. A typical model made up of 11.25° sectors is shown in Figure 7. The percentage error in displacements is plotted for the different sector angles as shown in Figures 28 and 29.

From these figures it is seen that the elements Q4 and Q5 behave almost identically. For any sector angle the Q6 element gives the least error whereas the AVEQ element gives the largest error. Also it is observed that the error in radial displacement at the outside radius is larger compared to the error at the inside radius.

At the outer radius, referring to Figure 28, the error ranges between 9.25% for the Q6 element to 10% for the AVEQ element at a sector angle of 11.25^{o}. At a larger sector angle of 45^{o}, the error for the Q6 element remains almost the same whereas for the AVEQ elements it increases to 18.75%. The Q4 and Q5 elements exhibit errors of 9.25% at a sector angle of 11.25^{o} and 12.5% at a sector angle of 45^{o}. The CSTQ element though exhibits similar error as the Q4 element at the low sector angle , it exhibits an error of 14.4% at the larger sector angle.

At the inner radius, referring to Figure 29, the error ranges between 6.5% for the Q6 element to 7.5% for the AVEQ element at a sector angle of 11.25^{o}. At a larger sector angle of 45^{o}, the error for the Q6 element increases slightly to 8% whereas the error for the AVEQ element increases to 17.5%. The Q4 and Q5 elements exhibit errors of 6.75% at the low sector angle and 10.5% at the larger sector angle. The CSTQ element exhibits almost the same behaviour as the Q4 element at the low sector angle but results in about 12.5% error at the large sector angle.

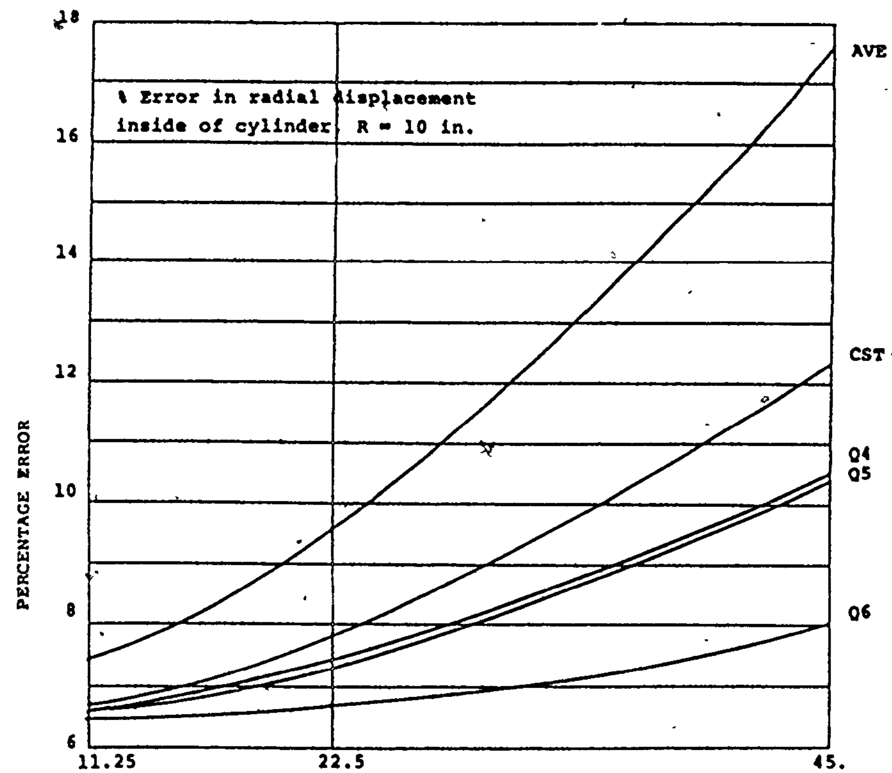


Refer fig. 7 Pressure loading P_i = 1000. psi.

$E = 30. * 10^6$ $\nu = 0.3$

FIGURE 28

EFFECT OF SECTOR ANGLE ON DISPLACEMENT PLANE STRAIN ANALYSIS



Refer fig. 7 Pressure loading P_i = 1000. psi.

$E = 30. * 10^6$ $\nu = 0.3$

FIGURE 29

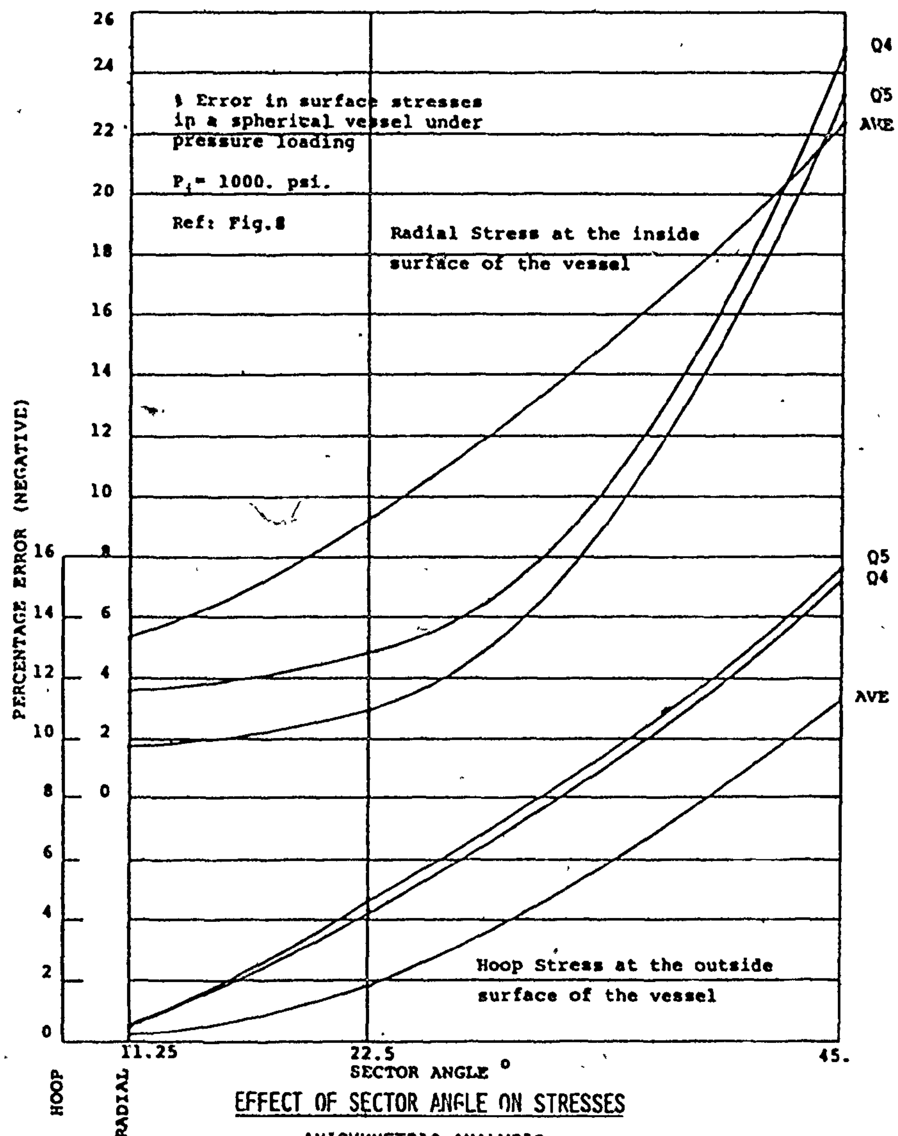
EFFECT OF SECTOR ANGLE ON DISPLACEMENT PLANE STRAIN ANALYSIS

From these observations, it can be inferred that with isoparametric family of elements larger sector angles can be used, still resulting in acceptable accuracies.

(c) Axisymmetric Condition

A spherical vessel under both mechanical and thermal loadings was considered for comparison. Deformations and stresses were compared for various element formulations. For reasons given in section 2.2.3, Q6 element formulation was not considered during comparison. The comparisons were made at sector angles 11.25^o, 22.5^o and 45^o. A typical model made up of 11.25^o sectors is shown in Figure 8. For comparison under mechanical loading (pressure loading), the radial stress at the inside surface and the hoop stress at the outside surface were compared. The percentage errors in these quantities for the three sector angles are plotted in Figures 30 and 31. For comparison under thermal loading (linear temperature gradient) the hoop stresses at the inside and the outside surfaces were compared. The percentage errors in these quantities are plotted for the three sector angles in Figure 32.

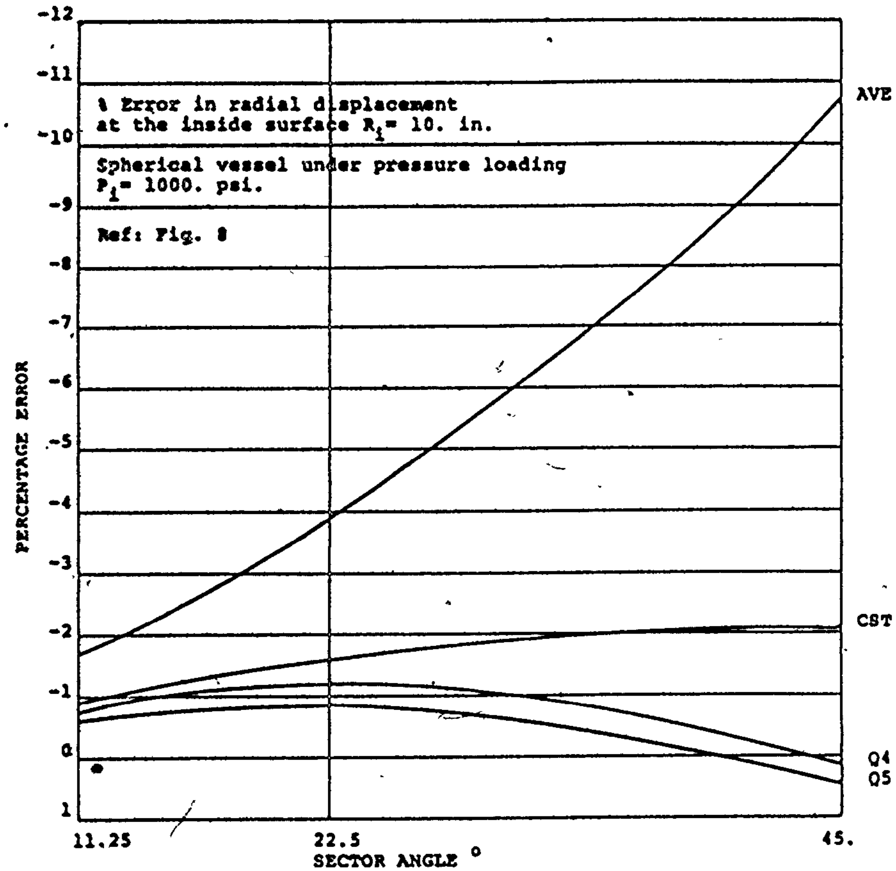
Under mechanical loading, the elements do not result in zero radial stresses at the outside surface but in a finite valve. This radial stress has a maximum value of about 7% of the pressure at 45^o sector angle and about 2% of the pressure at 11.25^o sector angle. The isoparametric elements give better results for radial stresses at the low



EFFECT OF SECTOR ANGLE ON STRESSES

AXISYMMETRIC ANALYSIS

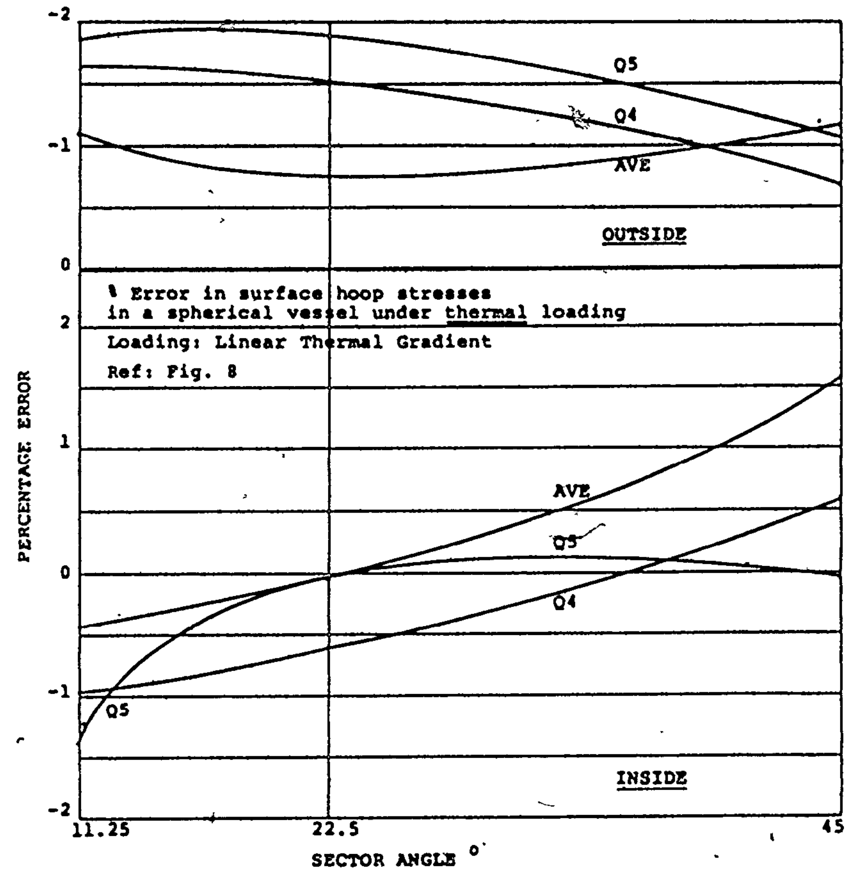
FIGURE 39



EFFECT OF SECTOR ANGLE ON DEFORMATION

AXISYMMETRIC ANALYSIS

FIGURE 31



EFFECT OF SECTOR ANGLE ON STRESSES

AXISYMMETRIC ANALYSIS

FIGURE 32

sector angles but the AVEQ element gives a better result at 45° sector angle. This holds good at the inside surface also. Referring to Figure 33, at 11.25° sector, the error in radial stress is -1.8% for the Q5 element, -3.5% for the Q4 element and -5.25% for the AVEQ element. With increasing sector angle, the errors increase exponentially for the isoparametric elements and almost linearly for the AVEQ element resulting in -25% error for Q4 element, -23% for the Q5 element and -22.5% for the AVEQ element. All element formulations have low errors (<1%) in hoop stress at low sector angle but increases to almost 16% for Q4 and Q5 elements and to 11.5% for AVEQ element at a sector angle of 45°. For deformations, referring to Figure 34, the Q4 and Q5 elements exhibit an error of only ±1.5% for the three sector angles. At 11.25° sector angle, the AVEQ element shows only -1.5% error but increases exponentailly to -10.5% at 45° sector angle.

Under thermal loading, the elements do not give zero radial stresses at the surfaces but result in a finite stress which is very small compared to the governing hoop stresses. The error in hoop stress at the surfaces for the various sector angles is plotted in Figure 32. Referring to the figure, the error is within ±2% of the theoretical value for all the element formulations.

5. A PROPOSED METHOD OF PREDICTING RESULTS

In high technology industry, such as Nuclear, it is common practice to have an analysis reviewed by a third party. One of the most common questions asked by the reviewer concerns the selection of one type of element over other types available and the resulting loss of accuracy if any. Hence it is of interest to the analyst to have a feel for what the results would be if a different element formulation was used in the analysis. In this chapter a method of predicting the results based on the family of curves generated during the comparison at element level is proposed.

The family of curves given in Figures 12 through 23 are plots of the square root of the strain energy density as a function of the variation in element geometry for various element formulations under any given type of loading. Since the formulations are based on a linear displacement field, the ordinate (Y) in the curves represent linear functions of displacements for any element type under a specific type of loading.

For any given aspect ratio, we can arrive at two values of Y, Y_1 and Y_2 for two different element formulations. This assumes that the material properties are identical, i.e.

$$Y_1 = C\delta_1$$

where C is a constant

$$Y_2 = C\delta_2$$

i.e. $$\delta_2 = \frac{Y_2}{Y_1}\delta_1 \qquad (5.1)$$

It is apparent from this equation that, the deformation behaviour of element type 2 can be predicted knowing the behaviour of element type 1 under similar conditions.

This equation holds good for assemblages since they behave in a fashion similar to their discrete elements.

The equation (5.1) is only valid for comparison of elements with identical material properties and identical value of loading for any given mode of deformation. The method of predicting can be extended to embrace different material properties and different values of loading as:

The element stiffness is given by:

$$k = \int_V [B]^T[D][B]\, dv \qquad (5.2)$$

For an isotropic material, the stress strain matrix $[D]$ can be written as

$$[D] = E\,[D'] \qquad (5.3)$$

where E is the Young's Modulus and $[D]$ is the stress-strain matrix independent of E.

Substituting (5-3) in (5-2) we have:

$$k = E\int_V [B]^T[D'][B]\, dv \qquad (5.4)$$

Using $k' = \int [B]^T [D'] [B]\, dv$ we have

$$k = Ek' \tag{5.5}$$

Consider an element with a modulus of elasticity E_1 under loading F_1 which deforms by δ_1. The equilibrium equations can be written as

$$k\delta_1 = F_1 \qquad \text{or} \qquad E_1 k' \delta_1 = F_1$$

If the same element with a different modulus of elasticity E_2 under loading F_2 deforms by δ_1', then δ_1' can be calculated as

$$\delta_1' = \frac{E_1}{E_2} \frac{F_2}{F_1} \delta_1 \tag{5.6}$$

Substituting δ_1' for δ_1 in equation (5.1), we have

$$\delta_2 = \frac{Y_2}{Y_1} \frac{E_1}{E_2} \frac{F_2}{F_1} \delta_1 \tag{5.7}$$

This relationship is valid as long as the bi-directional coupling in the two isotropic materials remains the same, i.e. the Poisson's ratio $\nu_1 = \nu_2$.

Using equation (5.7) with the family of curves in Figures 12 through 23, the deformation behaviour of any assemblage modelled in one type of element can be predicted knowing its deformation behaviour using another type of element. Though this method is general, it has the following constraints:

(a) the Poisson's ratio must be identical

and (b) the loading must represent approximately the deformation mode upon which the particular curve is based.

Even with these constraints, it is expected that this approach gives the relative accuracy of various element formulations in the deformation of realistic assemblage under similar modes of loading.

The use of this method of predicting results for assemblages under similar modes of loading has resulted in quite good correlations. A test model of a cantilever beam as shown in Figure 24 under pure bending and shear was used. The tests were carried out at two aspect ratios, 1:2 and 1:5. The results for bending mode are given in Tables 6 through 9.

For rectangular elements of the isoparametric family, the maximum variation (spread) between the predicted results and the calculated results is ±17%, which is quite reasonable. This spread reduces as the 'Aspect Ratio' increases as all the elements tend to give similar results. The spread in the prediction for the two types of test loading are:

	BENDING MODE		SHEARING MODE	
ASPECT RATIO	DISPLACEMENT	STRESS	DISPLACEMENT	STRESS
1:2	±17%	±17%	±7%	±6%
1:5	±4%	±6%	±4%	±3%

ELM	RATIO*	PREDICTED	CALCULATED	% ERROR
CST	REFERENCE ELEMENT			
AVE	0.6784	14.495	11.055	31.12
Q4	1.2301	26.284	28.054	-6.31
Q5	1.2609	26.942	29.457	-8.54
Q6	3.7521	80.171	91.0	-11.9
CST	1.474	16.295	21.367	-23.74
AVE	REFERENCE ELEMENT			
Q4	1.8123	20.035	28.054	-28.58
Q5	1.8587	20.548	29.457	-30.24
Q6	5.5308	61.143	91.0	-32.81
CST	0.8130	22.808	21.367	6.74
AVE	0.5515	15.472	11.055	39.95
Q4	REFERENCE ELEMENT			
Q5	1.0251	28.758	29.457	-2.37
Q6	3.0503	85.573	91.0	-5.96
CST	0.7931	23.362	21.367	9.34
AVE	0.5380	15.848	11.055	43.35
Q4	0.9755	28.735	28.054	2.43
Q5	REFERENCE ELEMENT			
Q6	2.9757	87.655	91.0	-3.68
CST	0.2665	24.252	21.367	13.5
AVE	0.1808	16.453	11.055	48.83
Q4	0.3278	29.830	28.054	6.33
Q5	0.3361	30.585	29.457	3.83
Q6	REFERENCE ELEMENT			

ASPECT RATIO = 1:5 MODE = Bending

PREDICTION OF RESULTS: TIP DEFLECTION *1.E-04

TABLE 6

$*\text{RATIO} = \frac{Y_{ELM}}{Y_{REF\ ELM}}$

ELM	RATIO*	PREDICTED	CALCULATED	% ERROR
CST	REFERENCE ELEMENT			0.00
AVE	0.6410	38.486	33.471	14.98
Q4	1.0823	64.982	64.711	0.42
Q5	1.2218	73.358	72.198	1.61
Q6	1.7739	106.507	91.0	17.04
CST	1.5601	52.218	60.041	-13.03
AVE	REFERENCE ELEMENT			
Q4	1.6885	56.516	64.711	-12.66
Q5	1.9060	63.796	72.198	-11.64
Q6	2.7675	92.631	91.0	1.79
CST	0.9239	59.787	60.041	-0.42
AVE	0.5922	38.322	33.471	14.49
Q4	REFERENCE ELEMENT			
Q5	1.1288	73.046	72.198	1.17
Q6	1.6390	106.06	91.0	16.55
CST	0.8185	59.094	60.041	-1.58
AVE	0.5247	37.882	33.471	13.18
Q4	0.8859	63.960	64.711	-1.16
Q5	REFERENCE ELEMENT			
Q6	1.4520	104.832	91.0	15.2
CST	0.5637	51.297	60.041	-14.56
AVE	0.3613	32.878	33.471	-1.77
Q4	0.6101	55.519	64.711	-14.20
Q5	0.6887	62.672	72.198	-13.19
Q6	REFERENCE ELEMENT			

ASPECT RATIO = 1:2 MODE = Bending

PREDICTION OF RESULTS: TIP DEFLECTION *1.E-04

TABLE 7

ELM	RATIO*	PREDICTED	CALCULATED	% ERROR
CST	REFERENCE ELEMENT			
AVE				
Q4				
Q5				
Q6				
CST				
AVE	REFERENCE ELEMENT			
Q4	1.6885	931.5	1066.7	-12.67
Q5	1.9060	1051.5	1190.1	-11.64
Q6	2.7675	1526.8	1500.0	1.79
CST				
AVE	0.5922	631.7	551.7	14.5
Q4	REFERENCE ELEMENT			
Q5	1.1288	1204.1	1190.1	1.18
Q6	1.6390	1748.3	1500.0	16.55
CST				
AVE	0.5247	624.4	551.7	13.19
Q4	0.8859	1054.3	1066.7	-1.16
Q5	REFERENCE ELEMENT			
Q6	1.4520	1728.0	1500.0	15.20
CST				
AVE	0.3613	541.95	551.7	-1.77
Q4	0.6101	915.2	1066.7	-14.20
Q5	0.6887	1033.1	1190.1	-13.20
Q6	REFERENCE ELEMENT			

ASPECT RATIO = 1:2 MODE = Bending

PREDICTION OF RESULTS: RADIAL STRESS

TABLE 8

$*\text{RATIO} = \frac{Y_{ELM}}{Y_{REF\ ELM}}$

ELM	RATIO*	PREDICTED	CALCULATED	% ERROR
CST	REFERENCE ELEMENT			
AVE				
Q4				
Q5				
Q6				
CST				
AVE	REFERENCE ELEMENT			
Q4	1.8123	330.2	462.4	-28.59
Q5	1.8587	338.7	485.5	-30.25
Q6	5.5308	1007.7	1500.0	-32.82
CST				
AVE	0.5515	255.0	182.2	39.96
Q4	REFERENCE ELEMENT			
Q5	1.0251	474.0	485.5	-2.37
Q6	3.0503	1410.5	1500.0	-5.97
CST				
AVE	0.5380	261.2	182.2	43.36
Q4	0.9755	473.6	462.4	2.42
Q5	REFERENCE ELEMENT			
Q6	2.9757	1444.7	1500.0	-3.69
CST				
AVE	0.1808	271.2	182.2	48.85
Q4	0.3278	491.7	462.4	6.34
Q5	0.3361	504.2	485.5	3.84
Q6	REFERENCE ELEMENT			

ASPECT RATIO = 1:5 MODE = Bending

PREDICTION OF RESULTS: RADIAL STRESSES

TABLE 9

The AVEQ element results could not be correlated within any reasonable accuracy.

This method will hold good for large assemblages <u>as long as the predominant loading remains the same.</u>

6. REPRESENTATIVE APPLICATIONS OF THE PROGRAM KAXI

6.1 Analysis of a Porcelain Composite Insulator

An analysis of a typical porcelain composite electrical insulator is carried out to study the stress patterns in the component under the weight of the cable it is supporting. The intent of the analysis was to determine whether the true stress picture could be simulated using a two dimensional photo-elastic model.

For this analysis, one basic assumption was made: The insulator was considered to be an axisymmetric body of revolution. This is true for the insulator except at the top, near the supported end where it is three dimensional. Two types of analyses were carried out: an axisymmetric analysis to derive the true stress pattern and a plane stress analysis to simulate the photo-elastic model.

From the element comparisons in Chapter 4, the Q5 element formulation was found to have overall superiority and was chosen for this analysis. Since the model size governs the cost of analysis, an acceptable mesh with aspect ratios less than 1:4 as shown in Figure 33 was developed. This resulted in 314 nodes and 272 elements.

The various material properties used are:

1. Steel: Modulus of Elasticity, $E = 30 \times 10^6$ psi
   Poissons' ratio, $\nu = 0.3$
   and Shear modulus, $G = 11.5 \times 10^6$ psi

2. Grouting: Modulus of elasticity, $E = 4 \times 10^6$ psi
Poissons' ratio, $\nu = 0.1$
and Shear modulus, $G = 1.82 \times 10^6$ psi

3. Porcelain: Modulus of elasticity, $E = 10 \times 10^6$ psi
Poissons' ratio, $\nu = 0.1$
and Shear modulus, $G = 4.55 \times 10^6$ psi

The loading used is 2500 lbs tension pull.

From the results, it is evident that a large share of the load is carried by the hoop strength which is not represented in the plane stress case. This makes it difficult to compare the two models using principal stresses and directions. However, one way of comparison is to use the 'effective stress' approach.[23]

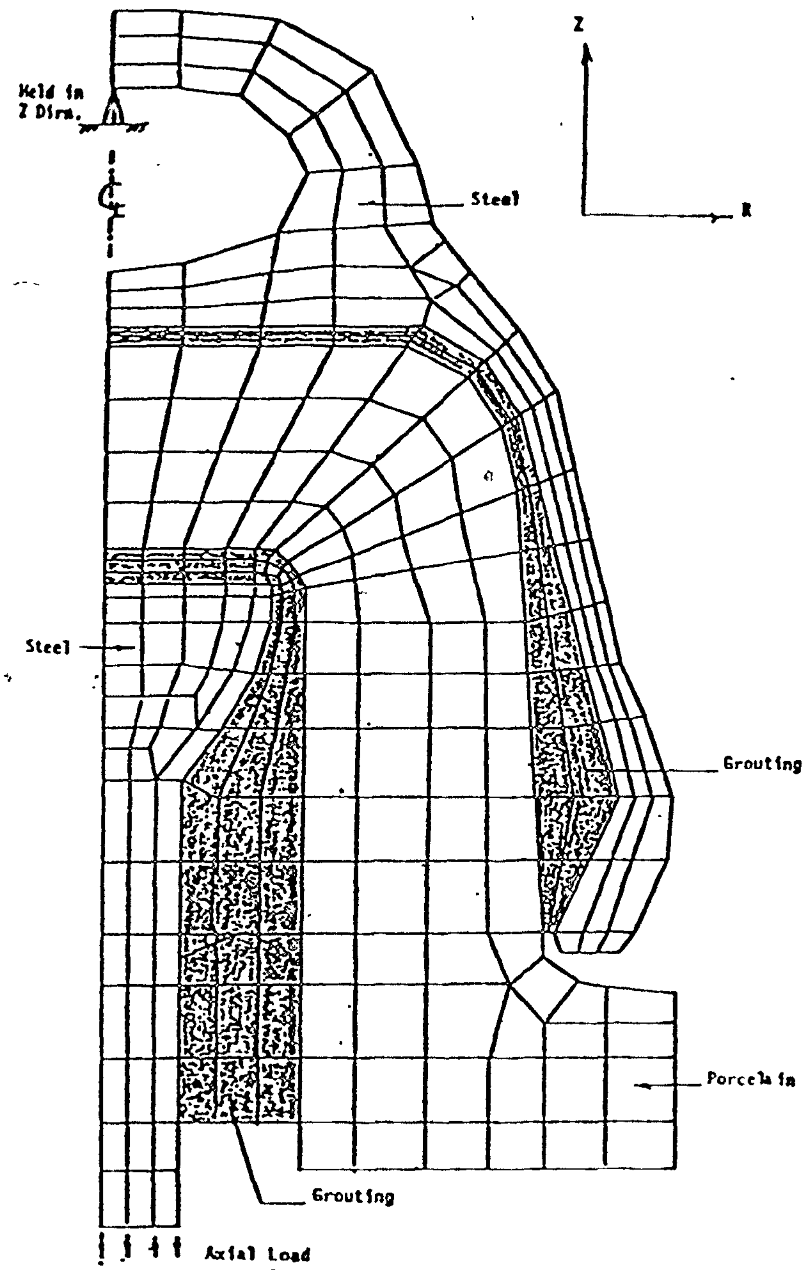
The effective stress is defined as

$$\sigma_e = \tfrac{1}{2} \{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2\}^{\frac{1}{2}}$$

where σ_1, σ_2 and σ_3 are principal stresses arrived at from the global stresses.

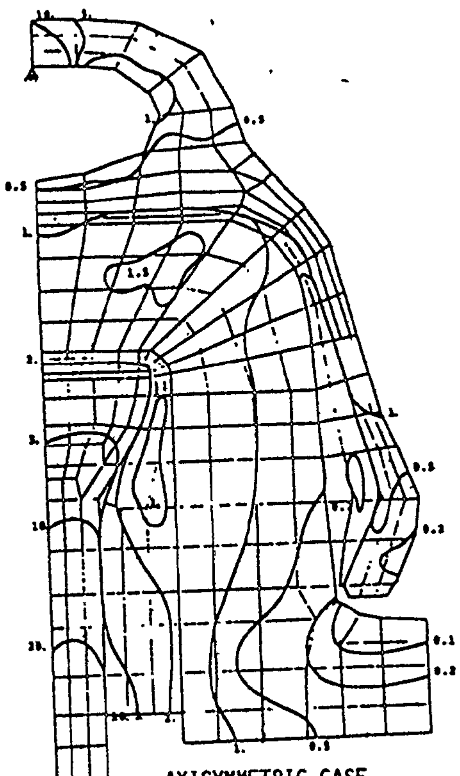
The effective stresses are plotted as iso-stress lines as shown in Figures 34 and 35. From the figures, it is seen that, though the basic iso-stress lines have similar shapes, their magnitudes do not correspond. In the highly stressed regions, the stresses are almost an order of magnitude different, the axisymmetric model giving higher values.

This concludes that an accurate prediction of the stress pattern in a real insulator is not practical using plane stress photo-elastic models.

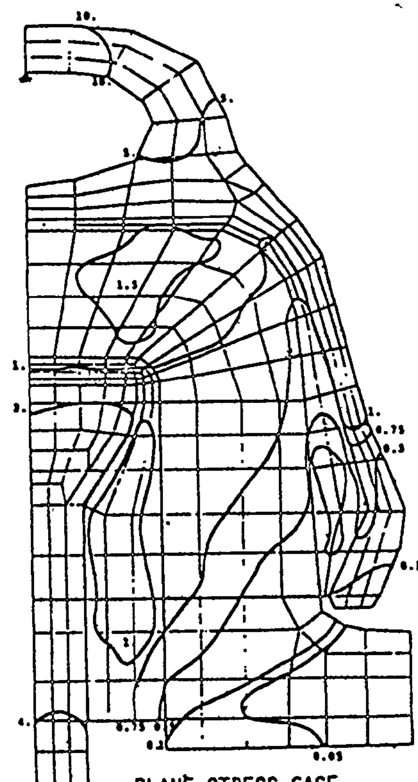


FINITE ELEMENT MODEL FOR THE PORCELAIN COMPOSITE INSULATOR

FIGURE 33



AXISYMMETRIC CASE



PLANE STRESS CASE

(STRESS IN PSI)

FIGURES 34 AND 35

PLOT OF ISO - STRESS LINES OF EFFECTIVE STRESS FOR THE INSULATOR

6.2 Analysis of a Tube Sheet in a Heat Exchanger

An axisymmetric analysis of a Tube Sheet assembly was carried out for comparing the element formulations: Q4, Q5 and AVEQ. The finite element model includes the tube sheet, the channel cover, the bolting, the primary inlet nozzle and the secondary shell flange assembly. This approach of using axisymmetric analysis to represent a three dimensional case is considered adequate to represent fairly the maximum stress levels.

The model used for analysis is shown in Figure 36. The primary inlet nozzle is located on the axis of the heat exchanger and the holes in the tube sheet are modelled as an annular array. The model is exposed to an axisymmetric temperature change, whereas in an actual heat exchanger, only a portion of the tube sheet/cover is exposed to the inlet water temperature while the remainder is exposed to somewhat lower temperatures. The axisymmetric model will therefore undergo larger thermal distortions and consequently experience larger thermal stresses than a three dimensional model in which only a portion of the tube sheet is heated.

The bolts were modelled as an annular ring occupying or overlaying the same space as the ligaments between the bolts. The bolt(ring) elements are connected to the flange, the tube sheet and the channel cover at its ends only.

The material properties of the bolt-ring and the bolt-ligament ring were adjusted to reflect the stiffness of the bolts and bolt-ligaments relative to the adjacent homogeneous material, i.e. since the bolts and bolt-ligaments cannot sustain any significant hoop stress, the modulus of elasticity in the hoop direction is given an arbitrary small value.

The material properties of the elements representing the holes or ligament region in the tube sheet were modified to account for the holes in accordance with the ASME Code. The elements in the gasket region were also given modified elasticity constants to account for the grooves and to account for possible slippage at the contact face.

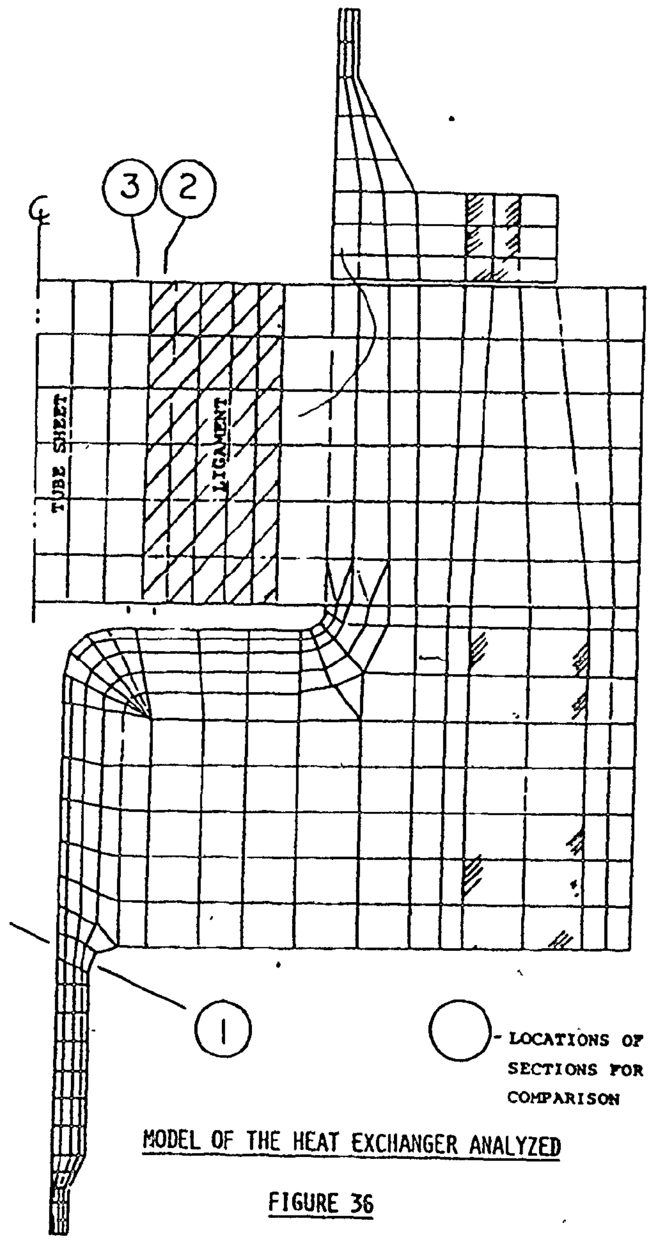
The temperature distribution in the model was determined using a finite element thermal analysis program. The temperature distribution at critical times during the specified thermal transients was used as input to the stress program and the thermal stresses at these times were subsequently used to determine the range of peak stresses for fatigue analysis.

Stresses due to a unit 1000 psi internal pressure and due to thermal heatup (100 deg. F to 492 deg. F in 80 min.) were calculated. The load cases were run individually to permit prorating.

The element centroidal stresses and extrapolated surface stresses in the hoop direction are plotted for the three element types, for comparison in Figure 37. From the figure it is observed that all the elements behave quite similarly. The hoop stress distribution due to pressure loading is almost identical for the Q4 and Q5 elements. The

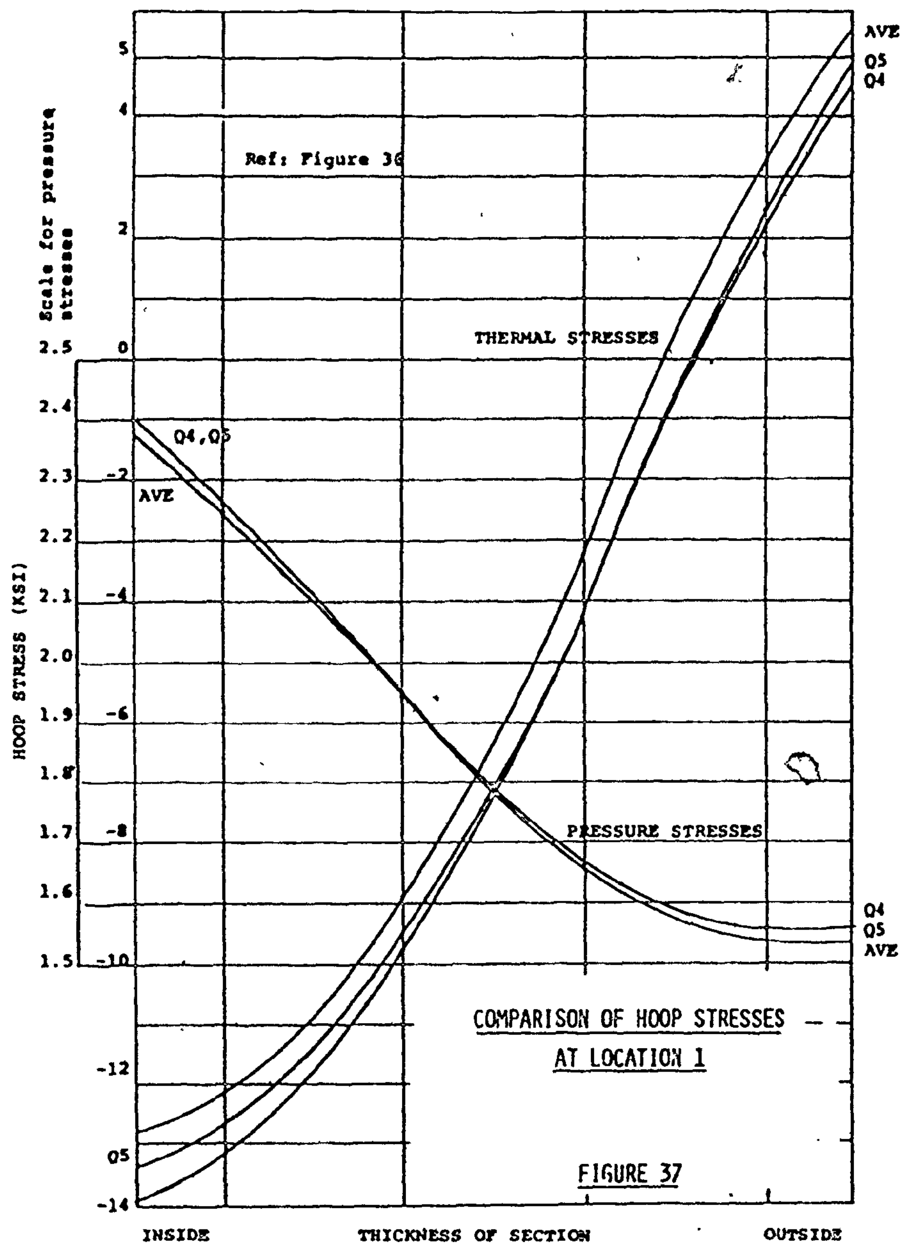
AVEQ element results are within ±1% of the results using Q4 elements. For the thermal loading, the AVEQ element results in a lower stress at the inside and a higher stress at the outside compared to the Q4 and Q5 elements. The spread is only about ±7%.

Further, a comparison has been carried out for the normal stress distribution at the tube sheet-ligament interface, locations 2 and 3 in Figure 36. The stresses are plotted in Figure 38. It is of interest to note that the stresses for element Q4 and AVEQ do not extrapolate to a small value at the surfaces. Further, in the middle of the section, the Q5 element provides with the maximum stress difference. Though these normal stresses do show significant variations, the hoop stresses which govern the stress state have very little variations, thus the overall stress state of the component is not significantly changed by using different elements. This is true when the thermal stresses swamp the other stresses.



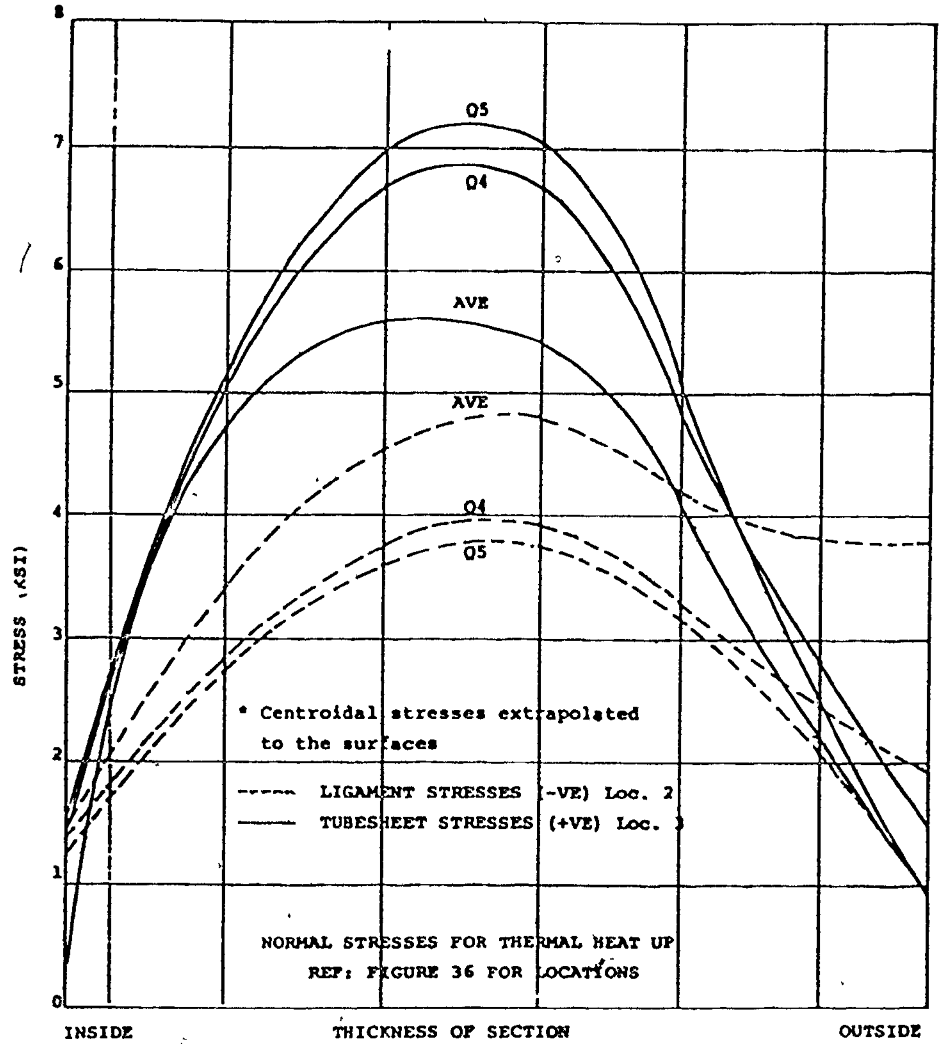
MODEL OF THE HEAT EXCHANGER ANALYZED

FIGURE 36



COMPARISON OF HOOP STRESSES AT LOCATION 1

FIGURE 37



COMPARISON OF THE NORMAL STRESS DISTRIBUTION
AT THE TUBESHEET - LIGAMENT INTERFACE

FIGURE 38

7. CONCLUSIONS

Two dimensional, including axisymmetric finite elements using both constant strain formulation and isoparametric formulations have been developed and incorporated into a computer program for use on a mini-computer. The verifications against classical problems in Chapter 3 show that the program KAXI produces results within ±10% of the theoretical results using isoparametric elements. The cost of using this program on a mini-computer is about less than half the cost of using proprietary programs on a large computer.

The various elements developed have been compared for their behaviour at element level based on the work of Rigby and McNeice and also in small assemblages using sensitivity analysis. (only rectangular elements were used for comparison). From the comparisons, it was concluded that each element behaves quite similarly in plane stress, plane strain and axisymmetric conditions. The accuracy of the elements in general fell quite steeply between aspect ratios of 1:1 and 1:2 and then levelled off. However, elements whose aspect ratios were greater than 1:5 gave similar results, regardless of the formulation. The different elements behave quite differently at an aspect ratio of 1:1, but tend to show similarities in behaviour at large aspect ratios. It can also be concluded from the study, that for a two dimensional problem, Q6 element is well suited due to its excellent bending behaviour, whereas for the axisymmetric case, Q5 element shows a suitable overall behaviour.

Further, an attempt was made to predict results based on the data gathered during element comparisons at element level. In small assemblages, where a single mode of deformation predominates, the predictions were quite good. However, when there was no identifiable single predominant mode, the accuracy of prediction suffered. It is still of use to an analyst to get a feel for the results, if a different kind of element could have been used in his analysis.

For a better understanding of the effects of element shapes, more work needs to be done in comparing non-rectangular elements under various loadings. Also additional work needs to be done to understand the coupling of the predominant modes of deformation and use this information for the prediction of results.

REFERENCES

1. Southwell, R.V. Relaxation Methods in Theoretical physics, Clarendon Press, Oxford, 1946.
2. Allen, D.N.De G. Relaxation Methods, McGraw-Hill, New York, 1955.
3. Crandall, S.H. Engineering Analysis, McGraw-Hill, New York, 1956.
4. Finlayson, B.A. The Method of Weighted Residuals and Variational Principles, Academic Press, New York, 1972.
5. Newmark, N.M. 'Numerical Methods of Analysis in Bars, Plates and Elastic Bodies', Numerical Methods of Analysis in Engineering, MacMillan, New York, 1949.
6. Argyris, J.H. Energy Theorems and Structural Analysis, Butterworth and Co., London, 1960.
7. Turner, M.J., Clough, R.W., Martin, H.C. and Topp, L.C. 'Stiffness and Deflection Analysis of Complex Structures', Journal of the Aeronautical Sciences, Vol 23, No. 9, 1956.
8. Clough, R.W. 'The Finite Element in Plane Stress Analysis', Proceedings, 2nd A.S.C.E. Conference on Electronic Computation, Pittsburg, Pa., Sept. 8 and 9, 1960.
9. Zeinkiewicz, O.C. The Finite Element Method in Engineering Science, McGraw-Hill, London, 1971.
10. Cook, R.D. Concepts and Applications of Finite Element Analysis, John Wiley and Sons Inc., New York, 1974.

11. Desai, Chandrakant S. and Abel, John F.Introduction to Finite Element Method, Van Nostrand Reinhold Co., New York, 1972.

12. Bathe, Klaus-Jurgen and Wilson, Edward L. Numerical Methods in Finite Element Analysis, Prentice Hall Inc., Engelwood Cliffs, N.J., 1976.

13. Melosh, R.J. 'Structural Analysis of Solids', Int. Journal Structural Div., Vol. 89, No. ST-4, Aug. 1963., pp. 205 - 223.

14. Khanna, J. and Hooley, R.F. 'Comparison and Evaluation of Stiffness Matrices', AIAA Journal, Vol. 4, No. 12, Dec. 1966.

15. Clough, R.W. 'Comparison of Three Dimensional Finite Elements', Proceedings of the Symposium on Application of Finite Element Methods in Civil Engineering, Vanderbilt University, Nashville, Tenn., Nov. 1969.

16. Rigby, G.L. and McNeice, G.M. 'A Strain Energy Basis for Studies of Element Stiffness Matrices', AIAA Journal ,Vol. 10, No. 11, Nov. 1972., pp. 1490 - 1493.

17. Timoshenko, S. Strength of Materials, Part II, 3rd Ed., Van Nostrand Co., Princeton, N.J., 1956.

18. Irons, B.M. 'Engineering Application of Numerical Integration in Stiffness Method', AIAA Journal, No. 14, 1966, pp. 2035 - 2037.

19. Bathe, K.J., Wilson, E.L. and Peterson, F.E., SAP IV : A Structural Analysis Program for Static and Dynamic

Response of Linear Systems, Report EERC-73-11, Earthquake Engineering Research Centre, University of California, Berkeley, Ca., June 1973.

20. Timoshenko, S. and Goodier, J.N. Theory of Elasticity, Second Ed., McGraw-Hill Book Co., New York, 1951.

21. Roark, Raymond J. and Young, Warren C. Formulas for Stress and Strain, Fifth Ed., McGraw-Hill Book Co., New York, 1975.

22. Pilkey, W., Saczalski, K. and Schaeffer, Eds. Structural Mechanics Computer Programs, Survey, Assessments and Availability, University Press of Virginia, Charlottesville, 1974.

23. Dally, J.W. and Riley, W.F. Experimental Stress Analysis, McGraw-Hill Book Co., New York, 1965.

APPENDIX 1

EIGENVALUE AND EIGENVECTOR EXTRACTION

USING GENERALIZED

JACOBI TRANSFORMATION METHOD

Eigenvalue and Eigenvector Extraction Using Generalized Jacobi Transformation Method

The equations of motion for an undamped system can be written as:

$$M\ddot{x} + Kx = 0 \tag{1}$$

where M = the mass matrix
K = the stiffness matrix
x = the displacement vector

Considering harmonic motion, $x = \phi \sin \omega t$, the generalized eigenvalue problem is obtained as:

$$-\omega^2 M\phi + K\phi = 0$$
$$\text{OR} \tag{2}$$
$$(-\lambda M + K)\,\phi = 0$$

where λ = the eigenvalues
ϕ = the corresponding eigenvectors

The solution for equation (2) can be obtained by making use of the property that the eigenvalues of the problem $K\phi = \lambda M\phi$ are the roots of the characteristic polynomial

$$P(\lambda) = \left| K - \lambda M \right| \tag{3}$$

which derives from the basic relationship of equation (2). This can be satisfied only for nontrivial ϕ provided the matrix $\left[K - \lambda M\right]$ is singular, i.e. $\left|K - \lambda M\right| = 0$. This determinant vanishes only for particular eigenvalues λ and for each eigenvalue there is a corresponding eigenvector ϕ.

There are two classes of eigenvalue extraction methods:

(1) Transformation methods: Jacobi, House-holder, Q-R, Givens, etc.

(2) Iterative and minimization schemes.

The generalized Jacobi iteration method used is briefly outlined below.

This method is a transformation method which makes use of the basic properties of the eigenvectors as follows:

$$\phi^T K \phi = \Lambda^2 \qquad (4)$$

$$\phi^T M \phi = I \qquad (5)$$

$$\text{where} \quad \Lambda^2 = \begin{bmatrix} \lambda_1^2 & 0 & 0 \; \ldots\ldots 0 \\ & \lambda_2^2 & 0 \; \ldots\ldots 0 \\ & & \quad\quad \lambda_n^2 \end{bmatrix}$$

The matrix ϕ, which is used to diagonalize K and M as shown in euation (4) and (5) is unique. Thus, the principle is to diagonalize K and M by successive pre- and post-multiplication by matrices P_n^T and P_n, where n = 1, 2, 3 ... Letting $K_1 = K$ and $M_1 = M$,

$$K_2 = P_1^T K_1 P_1$$

$$K_3 = P_2^T K_2 P_2 \qquad (6)$$

$$\downarrow$$

$$K_{j+1} = P_n^T K_n P_n$$

and,

$$M_2 = P_1^T M_1 P_1$$

$$M_3 = P_2^T M_2 P_2 \qquad (7)$$

$$\downarrow$$

$$M_{j+1} = P_n^T M_n P_n$$

where P_n are the matrices formed to let K_n and M_n approach a diagonal form. That is,

$$K_n \rightarrow \Lambda^2 \text{ and } M_{n+1} \rightarrow I$$

$$\text{and } n \rightarrow \infty$$

and with the last iteration ℓ

$$\hat{\phi} = P_1 P_2 \ldots P_\ell \qquad (8)$$

It should be noted that K_{n+1} and M_{n+1} need not converge to Λ^2 and I respectively and all that is required is that they converge to diagonal form since,

$$\Lambda^2 = \text{diag}\left[\frac{K^{(\ell+1)}}{M^{\ell+1}}\right] \qquad (9)$$

$$\text{and, } \phi = P_1 P_2 \ldots P_\ell = \text{diag}\frac{1}{\sqrt{M^{\ell+1}}} \qquad (10)$$

The matrices P_n must be selected such that M and K tend toward diagonal form. The matrix P_n is formed as follows:

$$P_n = \begin{bmatrix} 1 & & & & & & & \\ & 1 & & & & & & \\ & & \ddots & & & & & \\ & & & 1 & \alpha & & & \\ & & & \gamma & 1 & & & \\ & & & & & \ddots & & \\ & & & & & & 1 & \\ & & & & & & & 1 \end{bmatrix} \qquad \begin{matrix} \leftarrow i^{th} \text{ row}, \ \alpha \text{ in } j^{th} \text{ column} \\ \leftarrow j^{th} \text{ row}, \ \gamma \text{ in } i^{th} \text{ column} \end{matrix} \tag{11}$$

where α and γ are selected such that the elements(i, j) in K_n and M_n are simultaneously reduced to zero. Performing the multiplications $P_n^T K_n P_n$ and $P_n^T M_n P_n$ where n denotes the n^{th} iteration, and using the requirement that m_{ij}^{n+1} amd K_{ij}^{n+1} will be zero, the following equations are obtained for α and γ

$$\alpha k_{ii}^{n} + (1 + \alpha\gamma) k_{ij}^{n} + \gamma k_{jj}^{n} = 0 \tag{12}$$

$$\alpha m_{ii}^{n} + (1 + \alpha\gamma) m_{ij}^{n} + \gamma m_{jj}^{n} = 0 \tag{13}$$

Solving for α and γ,

$$\bar{k}_{ii}^{n} = k_{ii}^{n} m_{ij}^{n} - m_{ii}^{n} k_{ij}^{n} \tag{14}$$

$$\bar{k}_{jj}^{n} = k_{jj}^{n} m_{ij}^{n} - m_{jj}^{n} k_{ij}^{n} \tag{15}$$

$$\bar{k}^n = k_{ii}^n m_{jj}^n - k_{jj}^n m_{ii}^n \tag{16}$$

$$x = \frac{\bar{k}^n}{2} + \mathrm{sign}(\bar{k}^n)\sqrt{\left(\frac{\bar{k}^n}{2}\right)^2 + \bar{k}_{ii}^n \bar{k}_{jj}^n} \tag{17}$$

α and are given by

$$\gamma = \frac{\bar{k}_{jj}^n}{x} \tag{18}$$

$$\gamma = \frac{\bar{k}_{ii}^n}{x} \tag{19}$$

Once the P_n matrix is determined, the relations (6) to (10) are used and the process is iterated until convergence is reached. The convergence is measured by comparing successive eigenvalue approximations and by testing if all off-diagonal elements are small enough. In the computer program this is evaluated as,

$$\frac{\lambda_i^{(n+1)} - \lambda_i^{(n)}}{\lambda_i^{(n+1)}} \leqslant \text{RTOL} \tag{20}$$

where RTOL is the tolerance specified.

$$\lambda_i^{(n+1)} = \frac{k_{ii}^{(n+1)}}{m_{ii}^{(n+1)}}$$

$$\lambda_i^{n} = \frac{k_{ii}^{n}}{m_{ii}^{n}}$$

and

$$\left[\frac{(k_{ij}^{(n+1)})^2}{k_{ii}^{(n+1)} k_{jj}^{(n+1)}}\right]^{\frac{1}{2}} \leqslant (0.01^{NSWEEP})^2 \qquad (21)$$

$$\left[\frac{(m_{ij}^{(n+1)})^2}{m_{ii}^{(n+1)} m_{jj}^{(n+1)}}\right]^{\frac{1}{2}} \leqslant (0.01^{NSWEEP})^2 \qquad (22)$$

where NSWEEP = Sweep Number

The relations given in this section form the basis of the subroutine JACOBI used in the extraction of eigenvalues and eigenvectors. The complete listing is given below.

```
      SUBROUTINE JACOBI(A,B,X,EIGV,N)
C
C     TO SOLVE THE GENERALIZED EIGENPROBLEM USING THE
C     GENERALIZED JACOBI ITERATION
      IMPLICIT REAL*8(A-H,O-Z)
      COMMON /CNTRL/IR,IOUT,IC
      DIMENSION A(16,16),B(16,16),X(16,16),EIGV(16),D(16)
      NSMAX=6
      RTOL=1.E-06
      IFPR=1
C     INITIALIZE EIGENVALUE AND EIGENVECTOR MATRICES
      DO 10 I=1,N
      IF(A(I,I).GT.0..AND.B(I,I).GT.0.)GO TO 10
      WRITE(IOUT,2020)I,A(I,I),B(I,I)
      STOP
10    EIGV(I)=A(I,I)/B(I,I)
      DO 30 I=1,N
      DO 20 J=1,N
   20 X(I,J)=0.0
   30 X(I,I)=1.0
      IF(N.EQ.1) RETURN
C     INITIALIZE SWEEP COUNTER AND BEGIN ITERATION
      NSWEEP=0
      NR=N-1
   40 NSWEEP=NSWEEP+1
      IF(IFPR.EQ.1)WRITE(IC,2000)NSWEEP
C     CHECK IF PRESENT OFF-DIAGONAL ELEMENT IS LARGE ENOUGH TO REQU
C     ZEROING
      EPS=(.01**FLOAT(NSWEEP))**2
      DO 210 J=1,NR
      JJ=J+1
      DO 210 K=JJ,N
      EPTOLA=(A(J,K)*A(J,K))/(A(J,J)*A(K,K))
      EPTOLB=(B(J,K)*B(J,K))/(B(J,J)*B(K,K))
      IF((EPTOLA.LT.EPS).AND.(EPTOLB.LT.EPS)) GO TO 210
C     IF ZEROING IS REQUIRED,CALCULATE THE ROTATION MATRIX ELEMENTS
C     CA AND CG
      AKK=A(K,K)*B(J,K)-B(K,K)*A(J,K)
      AJJ=A(J,J)*B(J,K)-B(J,J)*A(J,K)
      AB=A(J,J)*B(K,K)-A(K,K)*B(J,J)
      CHECK=(AB*AB+4.*AKK*AJJ)/4.0
      IF(CHECK) 50,60,60
   50 WRITE(IOUT,2020)
      STOP
   60 SQCH=DSQRT(CHECK)
      D1=AB/2.0+SQCH
      D2=AB/2.0-SQCH
      DEN=D1
      IF(DABS(D2).GT.DABS(D1)) DEN=D2
      IF(DEN) 80,70,80
   70 CA=0.0
      CG=-A(J,K)/A(K,K)
      GO TO 90
   80 CA=AKK/DEN
      CG=-AJJ/DEN
C     PERFORM THE GENERALIZED ROTATION TO ZERO THE PRESENT OFF-
C     DIAGONAL ELEMENT
90    IF((N-2).EQ.0)GO TO 190
      JP1=J+1
      JM1=J-1
      KP1=K+1
      KM1=K-1
      IF((JM1-1).LT.0)GO TO 130
      DO 120 I=1,JM1
      AJ=A(I,J)
      BJ=B(I,J)
      AK=A(I,K)
      BK=B(I,K)
      A(I,J)=AJ+CG*AK
      B(I,J)=BJ+CG*BK
      A(I,K)=AK+CA*AJ
  120 B(I,K)=BK+CA*BJ
130   IF((KP1-N).GT.0)GO TO 160
      DO 150 I=KP1,N
      AJ=A(J,I)
      BJ=B(J,I)
      AK=A(K,I)
      BK=B(K,I)
      A(J,I)=AJ+CG*AK
      B(J,I)=BJ+CG*BK
      A(K,I)=AK+CA*AJ
  150 B(K,I)=BK+CA*BJ
160   IF((JP1-KM1).GT.0)GO TO 190
      DO 180 I=JP1,KM1
      AJ=A(J,I)
      BJ=B(J,I)
      AK=A(I,K)
      BK=B(I,K)
      A(J,I)=AJ+CG*AK
      B(J,I)=BJ+CG*BK
      A(I,K)=AK+CA*AJ
  180 B(I,K)=BK+CA*BJ
  190 AK=A(K,K)
      BK=B(K,K)
      A(K,K)=AK+2.0*CA*A(J,K)+CA*CA*A(J,J)
      B(K,K)=BK+2.0*CA*B(J,K)+CA*CA*B(J,J)
      A(J,J)=A(J,J)+2.0*CG*A(J,K)+CG*CG*AK
      B(J,J)=B(J,J)+2.0*CG*B(J,K)+CG*CG*BK
      A(J,K)=0.0
      B(J,K)=0.0
C     UPDATE THE EIGENVECTOR MATRIX EACH ROTATION
      DO 200 I=1,N
      XJ=X(I,J)
      XK=X(I,K)
      X(I,J)=XJ+CG*XK
  200 X(I,K)=XK+CA*XJ
  210 CONTINUE
C     UPDATE THE EIGENVALUES AFTER EACH SWEEP
```

```
      DO 220 I=1,N
      IF(A(I,I).GT.0.0.AND.B(I,I).GT.0.0) GO TO 220
      IF(A(I,I).LT.-1.E-10)GO TO 221
      A(I,I)=1.E-10
      GO TO 220
221   WRITE(IOUT,2020)I,A(I,I),B(I,I)
C     STOP
  220 EIGV(I)=A(I,I)/B(I,I)
      IF(IFPR.EQ.0) GO TO 230
      WRITE(IOUT,2030)
      WRITE(IOUT,2010) (EIGV(I),I=1,N)
C     CHECK FOR CONVERGENCE
  230 DO 240 I=1,N
      TOL=RTOL*D(I)
      DIF=DABS(EIGV(I)-D(I))
      IF(DIF.GT.TOL) GO TO 280
  240 CONTINUE
C     CHECK ALL OFF DIAGONAL ELEMENTS TO SEE IF ANOTHER
C     SWEEP IS REQUIRED
      EPS=RTOL**2
      DO 250 J=1,NR
      JJ=J+1
      DO 250 K=JJ,N
      EPSA=(A(J,K)*A(J,K))/(A(J,J)*A(K,K))
      EPSB=(B(J,K)*B(J,K))/(B(J,J)*B(K,K))
      IF((EPSA.LT.EPS).AND.(EPSB.LT.EPS))GO TO 250
      GO TO 280
  250 CONTINUE
C     FILL OUT BOTTOM TRIANGLE OF RESULANTANT MATRICES
C     AND SCALE EIGENVECTORS
255   CONTINUE
      DO 260 I=1,N
      DO 260 J=1,N
      B(J,I)=B(I,J)
260   A(J,I)=A(I,J)
      DO 270 I=1,N
      BB=DSQRT(B(I,I))
      DO 270 K=1,N
  270 X(K,I)=X(K,I)/BB
C
C     FIND LARGEST TERM AND NORMALIZE
C
      DO 300 I=1,N
      BB=0.
      DO 310 K=2,N
      IF(DABS(BB).GE.DABS(X(K,I)))GO TO 310
      BB=X(K,I)
310   CONTINUE
      DO 320 K=1,N
320   X(K,I)=X(K,I)/BB
300   CONTINUE
      RETURN
C     UPDATE D MATRIX AND START NEW SWEEP ,IF ALLOWED
  280 DO 290 I=1,N
  290 D(I)=EIGV(I)
      IF(NSWEEP.LT.NSMAX) GO TO 40
      GO TO 255
 2000 FORMAT(27H0SWEEP NUMBER IN *JACOBI* =,I4)
 2010 FORMAT(1H0,6E20.12)
 2020 FORMAT(25H0*** ERROR SOLUTION STOP /
     1       31H MATRICES NOT POSITIVE DEFINITE /
     2 4H  I=,I4,7HA(I,I)=,E20.12,7HB(I,I)=,E20.12)
 2030 FORMAT(36H0CURRENT EIGENVALUES IN *JACOBI* ARE,/)
      END
```

APPENDIX 2

DEVELOPMENT OF GENERAL STRESS - STRAIN MATRIX

FOR TWO-DIMENSIONAL ANALYSIS

1. <u>The Constitutive Equations</u>

The basic relationship between stress and strain is the Hooke's Law for uniaxial deformation states. As an extension, in a general three dimensional case, for an elastic and anisotropic continuum, the generalized Hooke's Law can be written as:[3]

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \begin{bmatrix} D_{11} & D_{12} & \cdot & \cdot & \cdot & D_{16} \\ D_{21} & & & & & \\ \cdot & & & & & \\ \cdot & & & & & \\ \cdot & & & & & \\ D_{61} & & & & & D_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} \qquad \text{(A2.1)}$$

i.e. $\{\sigma\} = D\{\varepsilon\}$

Where $[D]$ is the general stress-strain matrix or the matrix of linear constitutive equations.

To evaluate the above $[D]$ matrix, we need 36 coeffecients of elasticity and due to symmetry results in 21 elastic constants for an anisotropic material, and in 9 elastic constants for an orthotropic material. For the orthotropic material the equation (A2.1) can be written in terms of the 9 constants as:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \begin{vmatrix} D_{11} & D_{12} & D_{13} & 0 & 0 & 0 \\ & D_{22} & D_{23} & 0 & 0 & 0 \\ & & D_{33} & 0 & 0 & 0 \\ & & & D_{44} & 0 & 0 \\ & & & & D_{55} & 0 \\ & & & & & D_{66} \end{vmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} \quad \text{(A2.2)}$$

These constants are difficult to evaluate and the general practice is to write the constitutive equations as 'strain-stress' equations. For orthotropic material, the strain-stress equations in terms of Young's modulii and Poisson's ratios can be written as:[11]

$$\left.\begin{aligned} \varepsilon_x &= \frac{1}{E_x}\sigma_x - \frac{\nu_{yx}}{E_y}\sigma_y - \frac{\nu_{zx}}{E_z}\sigma_z \\ \varepsilon_y &= \frac{-\nu_{xy}}{E_x}\sigma_x + \frac{1}{E_y}\sigma_y - \frac{\nu_{zy}}{E_z}\sigma_z \\ \varepsilon_z &= \frac{-\nu_{xz}}{E_x}\sigma_x - \frac{\nu_{yz}}{E_y}\sigma_y + \frac{1}{E_z}\sigma_z \end{aligned}\right\} \quad \text{(A2.3)}$$

$$\gamma_{xy} = \frac{\tau_{xy}}{G_{xy}} , \quad \gamma_{yz} = \frac{\tau_{yz}}{G_{yz}} , \quad \gamma_{zx} = \frac{\tau_{zx}}{G_{zx}}$$

Where E = Young's modulus

ν = Poissons' ratio

and G = Shear modulus

However only nine of these twelve material properties are independent since

$$\frac{E_x}{\nu_{xy}} = \frac{E_y}{\nu_{yx}} , \quad \frac{E_y}{\nu_{yz}} = \frac{E_z}{\nu_{zy}} , \quad \frac{E_z}{\nu_{zx}} = \frac{E_x}{\nu_{xz}} \qquad (A2.4)$$

To arrive at equation (A2.2), the equations in (A2.3) have to be inverted. Using equations (A2.9), the equations (A2.3) can be expressed in matrix form as shown in the next page, equation (A2.5).

The (4x4) matrix outlined by the dashed line in (A2.5) corresponds to the special case of axisymmetric analysis.

For general 2D analysis, a routine MATRXD was written to evaluate the stress-strain matrices for either plane stress, plane strain or axisymmetric analysis. The matrix [C] is generated based on the material properties for axisymmetric case. It is then inverted to get the [D] matrix. This matrix is then modified for either plane stress or plane strain case as necessary.

For plane stress case, the matrix [D] is modified as

$$D_{ij} = D_{ij} - D_{3j} * D_{i3}/D_{33} \qquad (A2.6)$$

for $i = 1, 2, 4$ and $j = i \rightarrow 4$

$D_{ij} = 0,$ for $i = 1, 4$ and $j = 3, 4$

and $D_{33} = D_{44}$ (A2.7)

Thus reducing the (4 x 4) matrix into a (3 x 3) matrix.

For plane strain analysis only the modifications in equation (A2.7) are carried out.

$$
[C] = \left[\begin{array}{cccc|cc}
\frac{1}{E_x} & -\frac{\nu_{xy}}{E_x} & -\frac{\nu_{xz}}{E_z} & 0 & 0 & 0 \\
 & \frac{1}{E_y} & -\frac{\nu_{yz}}{E_y} & 0 & 0 & 0 \\
 & & \frac{1}{E_z} & 0 & 0 & 0 \\
 & \text{symmetric} & & \frac{1}{G_{xy}} & 0 & 0 \\
\hline
 & & & & \frac{1}{G_{yz}} & 0 \\
 & & & & & \frac{1}{G_{zx}}
\end{array}\right]
$$

where $\{\epsilon\} = [C]\{\sigma\}$ (A2.5)

2. Transformation of Material Properties[10]

For completely generalizing the matrix [D] , it is necessary to consider material properties not specified in the model or the system coordinate system. The transformation necessary is a coordinate transformation to permit the use of element properties known with reference to one coordinate system, to be used in another coordinate system.

To be general, the transformation matrix for two cartesian coordinate systems arbitrarily oriented with respect to one another is evaluated here. Consider the two systems (x,y,z) and (x',y',z') shown in Figure A2.1, any state of stress or strain may be expressed in either coordinate systems as:

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

In their respective axis system

$$\{\sigma'\} = [D']\{\varepsilon'\} \qquad \text{(A2.8)}$$

These stresses and strains are related to their counter parts in any other system through a transformation matrix [T] as:

$$\begin{aligned}\{\sigma'\} &= [T_\sigma]\{\sigma\} \\ \{\varepsilon'\} &= [T_\varepsilon]\{\varepsilon\}\end{aligned} \qquad \text{(A2.9)}$$

The coefficients of the transformation matrix contain sine and cosine functions (direction cosines) of the angles between the axes. Further the transformation matrices in equation (A2.9) are related by

$$\left[T_\sigma\right]^{-1} = \left[T_\varepsilon\right]^T \text{ or vice-versa} \qquad \text{(A2.10)}$$

A general coordinate system in three dimensions and a table of their direction cosines are shown in Figure A2.1[2,6].

Using the matrix property of equation (A2.10), $\left[T_\varepsilon\right]$ can be written in terms of $\left[T_\sigma\right]$ as

$$\left[T_\varepsilon\right] = \left[\begin{array}{c|c} T_{11} & \tfrac{1}{2}T_{12} \\ \hline 2T_{21} & T_{22} \end{array}\right] \qquad \text{(A2.11)}$$

Where each submatrix is a (3 x 3) matrix defined by

$$\left[T_\sigma\right] = \left[\begin{array}{c|c} T_{11} & T_{12} \\ \hline T_{21} & T_{22} \end{array}\right]$$

Knowing that regardless of the coordinate system, increment in the strain energy density due to a virtual displacement must remain the same, we can write

$$\delta U = \{\delta\varepsilon\}^T \{\sigma\} = \{\delta\varepsilon'\}^T\{\sigma'\} \qquad \text{(A2.12)}$$

Substituting equations (A2.8) and (A2.9) in equations

(A2.12) we have

$$\delta U = \{\delta\varepsilon\}^T \left[D\right] \{\varepsilon\}$$

$$= \{\delta\varepsilon\}^T \left[T_\varepsilon\right]^T \left[D'\right] \left[T_\varepsilon\right] \{\varepsilon\}$$

$$\text{or} \quad \left[D\right] = \left[T_\varepsilon\right]^T \left[D'\right] \left[T_\varepsilon\right] \qquad \text{(A2.13)}$$

For plane stress and plane strain cases, in two dimensions,
$\{\sigma\}^T = \{\sigma_x \quad \sigma_y \quad \tau_{xy}\}$

referring to Figure A2.2

$$\left[T_\varepsilon\right] = \begin{bmatrix} l_1^2 & m_1^2 & l_1 m_1 \\ l_2^2 & m_2^2 & l_2 m_2 \\ 2l_1 l_2 & 2m_1 m_2 & (l_1 m_2 + l_2 m_1) \end{bmatrix} \qquad \text{(A2.14)}$$

Similarly for the axisymmetric case,
$\{\sigma\}^T = \{\sigma_x \quad \sigma_y \quad \sigma_\theta \quad \tau_{xy}\}$

Since σ_θ is independent of the coordinate system,

$$\left[T_\varepsilon\right] = \begin{bmatrix} l_1^2 & m_1^2 & 0 & l_1 m_1 \\ l_2^2 & m_2^2 & 0 & l_2 m_2 \\ 0 & 0 & 1 & 0 \\ 2l_1 l_2 & 2m_1 m_2 & 0 & (l_1 m_1 + l_2 m_2) \end{bmatrix} \qquad \text{(A2.15)}$$

The subroutine MATRXD listed in Appendix 3 evaluates the [D] matrix for any orthotropic material with an angular shift of principal axes β^o, using the above transformations.

$$
\left[T_\sigma\right] = \left[\begin{array}{ccc|ccc}
l_1^2 & m_1^2 & n_1^2 & 2l_1m_1 & 2m_1n_1 & 2n_1l_1 \\
l_2^2 & m_2^2 & n_2^2 & 2l_2m_2 & 2m_2n_2 & 2n_2l_2 \\
l_3^2 & m_3^2 & n_3^2 & 2l_3m_3 & 2m_3n_3 & 2n_3l_3 \\
\hline
l_1l_2 & m_1m_2 & n_1n_2 & (l_1m_2+l_2m_1) & (m_1n_2+m_2n_1) & (n_1l_2+n_2l_1) \\
l_2l_3 & m_2m_3 & n_2n_3 & (l_2m_3+l_3m_2) & (m_2n_3+m_3n_2) & (n_2l_3+n_3l_2) \\
l_3l_1 & m_3m_1 & n_3n_1 & (l_3m_1+l_1m_3) & (m_3n_1+m_1n_3) & (n_3l_1+n_1l_3)
\end{array}\right]
$$

TABLE A2.1

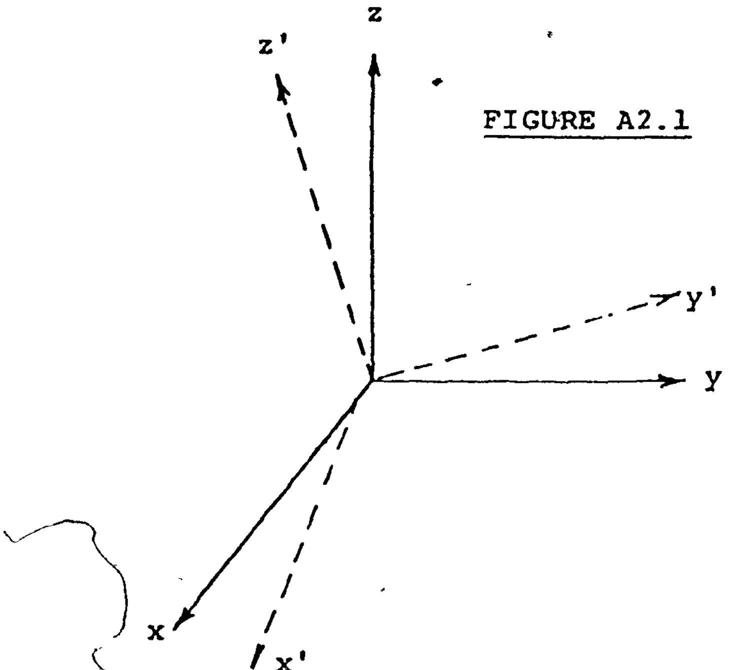
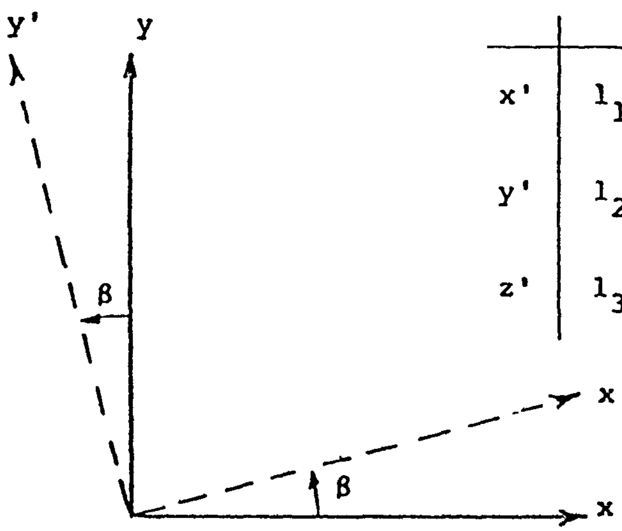


FIGURE A2.1

	x	y	z
x'	l_1	m_1	n_1
y'	l_2	m_2	n_2
z'	l_3	m_3	n_3

COORDINATES AND DIRECTION COSINES FOR A GENERAL THREE DIMENSIONAL CASE



	x	y	z
x'	l_1=Cos β	m_1=Sin β	n_1=o
y'	l_2=-Sin β	m_2=Cos β	n_2=o
z'	l_3=o	m_3=o	n_3=o

FIGURE A2.2

COORDINATES AND DIRECTION COSINES

FOR A TWO DIMENSIONAL CASE

APPENDIX 3

3.1 User Manual for Program KAXI

3.2 Computer Listing of Program KAXI

3.3 Typical Outputs from Program Verifications

USER MANUAL

FOR

PROGRAM: KAXI

TWO DIMENSIONAL STRESS ANALYSIS PROGRAM

(USING LINEAR ISOPARAMETRIC QUADRILATERAL

AND CONSTANT STRAIN TRIANGULAR ELEMENTS)

1. INPUT DATA FOR KAXI

The data should be stored in a data file

DL1: KAXDATA.DAT;1

NOTE: Triangular elements to be used only when absolutely necessary. The program is valid for a consistent set of units.

CARD 1: TITLE

FORMAT (20A4)

Where

Title = Job/Run identification

CARD 2: IGO, ITYP, NE, NP, NMAT, NLD, NTH, NMAX, I1, MODEX, AMBT, THIC

FORMAT (10I5, 2E10.2)

(1) IGO = switch for mode of reading model geometry
= 0, for reading from direct access files set up during thermal analysis
= 1, for reading from sequential data file

ITYP = Mode of analysis
= 1, for plane stress analysis
= 2, for plane strain analysis
= 3, for axisymmetric analysis defaults to 3

NE = number of internal elements, ≤500

NP = number of nodes, ≤500

NMAT = number of material types, ≤10

(2) NLD = number of mechanical load cases

NTH = number of thermal load cases >0, only if thermal analysis has been carried out

NMAX = max. connectivity, default = 100

I1 = 0, for printout of input data
= 1, for no printout

MODEX = mode of execution
= 0, for solution
= 1, for data check
AMBT = ambient temperature
THIC = thickness for plane stress, default = 1.

Notes: 1. The permanent files referred to are KAXELEMNT.DAT;1 and KAXCOORD.DAT;1 on DL1:
2. The program can handle up to a maximum of 10 load cases (NTH + NLD $\leq$ 10)

CARD 3: (PROP(I,J),J=1,10)
FORMAT (10E8.2)
where
PROP(I,1) = modulus of elasticity in R or X
PROP(I,2) = modulus of elasticity in the Z or Y direction
PROP(I,3) = modulus of elasticity in the tangential (θ) direction
PROP(I,4) = poissons ratio in the RZ or XY direction
PROP(I,5) = poissons ratio in the Rθ or Xθ direction
PROP(I,6) = poissons ratio in the Zθ or Yθ direction
PROP(I,7) = coefficient of thermal expansion in the R or X direction
PROP(I,8) = coefficient of thermal expansion in the Z or Y direction
PROP(I,9) = coefficient of thermal expansion in the tangential or θ direction

PROP(I,10) = shear modulus of elasticity in RZ or XY plane

Note: For isotropic materials, read in only E_R, ν_{RZ} and α_R values. The complete set of properties are generated. Further, the shear modulus if not defined, is calculated as $E_R/2(1 + \nu_{RZ})$

IF IGO = 0, BYPASS CARD SETS 4+ AND 5+

CARD 4+: (NODE(J),J=1,4), MAT, IEL

FORMAT (6I5)

where

(1) NODE (1), NODE (2), NODE (3), NODE (4) = Element definition

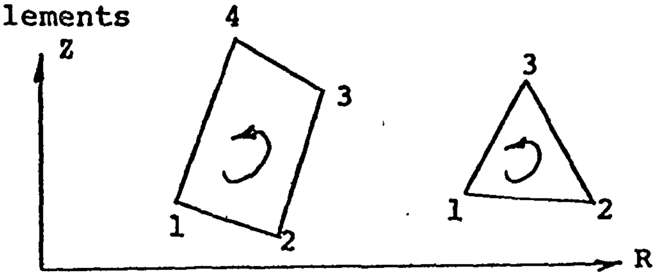
(2) MAT = Material type

IEL = Element type
= 4, for Q4 element
= 5, for Q5 element
= 6, for Q6 element
= 3, for CST element

Defaults to 5

Notes:
1. The node numbers must be defined as shown on the next sheet.
2. The material types can be up to 10. If the material type is left blank, it is assumed to be the same as that for the previous element.

For triangular elements
NODE (4) = 0



NODE NUMBER DEFINITION:

CARD 5+: N,X,Z
FORMAT (15,2F10.3)
where
N = node number
X = R or X coordinate
Z = Z or Y coordinate
This set must consist of NP cards.
Note: This node number data need not be in sequence.

IF NLD = 0, BYPASS CARD SET 6+

CARD 6+: ID,NAME
FORMAT (A2,24A2)
where
ID = FR, for nodal loading
= PR, for pressure loading
= blank for end of load case
NAME = load case identification
If the load case does not have nodal loads, i.e. ID = PR, bypass card 6A+.

CARD 6A+: NQ, R1, R2

FORMAT (I5,2E10.3)

where

NQ = node number at which nodal load is applied

R1 = load in the R or X direction/unit circ.

R2 = load in the Z or Y direction/unit circ.

Putting NQ = 0, ends this data block

If the load case does not have pressure loading, i.e. ID = FR, bypass Cards 6B+.

CARD 6B+: I1, I2, PR

FORMAT (2I5,F10.2)

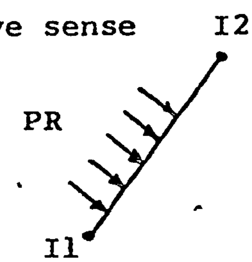
where

I1, I2 are node numbers defining the surface on which the pressure is acting

PR = pressure acting

Putting I1 = 0, ends this data block

Note: The pressure in the positive sense must be acting from left to right, looking from I1 towards I2.



There should be NLD sets of cards 6+, 6A+/6B+.

IF NTH = 0, BYPASS CARD SET 7+

CARD 7+: ID, NAME

FORMAT (I5,24A2)

where

ID = thermal load case number corresponding to thermal run, i.e. location of temperature data on file KAXTTEMP.DAT;1

NAME = load case identification

There should be NTH cards of 7+.

CARD 8+: NB, NFIX

FORMAT (3I5)

where

NB = node number at which the boundary is to be fixed

NFIX = fixity condition in R(X) and Z(Y) directions

= 1, for fix

= 0, for free

Putting NB = 0, ends this data set

CARD 9+: NB, BFIX

FORMAT (I5,2E10.3)

where

NB = node number at which linear springs are to be inserted

BFIX = spring rate in R(X) and Z(Y) directions

Putting NB = 0, ends this data set

CARD 10+: NB, BFIX

FORMAT (I5,2E10.3)

where

NB = node number at which displacements are to be specified

BFIX = displacements in R(X) and Z(Y) direction

Putting NB = 0, ends this data set

2. OUTPUT DATA FROM KAXI

The output file DLO: KAXIOUT.LST consists of the following:

1. Model Definition
2. Loading Data
3. Boundary Conditions Data
4. Global Displacements at the Nodes and Global Stresses at the Element Centroids for Each Load Case.

The stresses and displacements are stored in two direct access files:

DL1:KAXSTRES.DAT;1 for stresses
DL1:KAXDISP.DAT;1 for displacements

FORTRAN IV LISTING

OF

PROGRAM "KAXI"

```
C  ************************************************************
C  TWO DIMENSIONAL STRESS ANALYSIS , DEVELOPED BY K.K.SUBBAYYA
C  FOR PLANE STRESS,PLANE STRAIN AND AXISYMMETRIC ANALYSIS
C  VERIFIED ON PDP 11-34, FORTRAN IV COMPILER, MAR 1979.
C  MAIN LINE
C  ************************************************************
C
      IMPLICIT REAL*8(A-H,O-Z)
      COMMON/CNTRL/IR,IW,IC,ITYP,MODEX,THIC,NE,NP,NLD,NTH,NMAT,NMAX,
     1AMBT,TITLE(20),IPAGE
      COMMON /POP/PROP(10,10)
      COMMON CORD(500,2)
C-----------------------------------------------------------------
C  FILE DEFINITIONS:
C  1:ELEMENT DEFINITION: (NODE(I),I=1,4),MAT,IEL
C  2:COORDINATE DATA: X,Z
C  3:STIFFNESS/RHS FILE: (SI(J),J=1,NMAX),(ISP(J),J=1,NMAX),RHS(J),J=1,
C    LCASE)-NMAX.LE.100, LCASE.LE.10
C  4:DISPLACEMENTS FILE FOR 10 RHS.: (DIS(J),J=1,LCASE)
C  5:STRESS/TEMP. DATA: (ST(K,J),J=1,5),K=1,10)
C  6:INPUT DATA FILE
C  7:OUTPUT SPOOL FILE
C  8:TI:
C  9:TEMPERATURE OUTPUT FROM MAXIT: (TEMP(J),J=1,10)
C  10:UNIT THERMAL LOAD VECTOR: (UNIT(J),J=1,8)
C-----------------------------------------------------------------
      CALL ASSIGN (1,'DL1:KAXELEMNT.DAT;1')
      DEFINE FILE 1(500,6,U,NF1)
      CALL ASSIGN (2,'DL1:KAXCOORD.DAT;1')
      DEFINE FILE 2(500,8,U,NF2)
      CALL ASSIGN (3,'DL1:KAXSTIFF.DAT;1')
      DEFINE FILE 3(1000,540,U,NF3)
      CALL ASSIGN (4,'DL1:KAXSDISP.DAT;1')
      DEFINE FILE 4(500,80,U,NF4)
      CALL ASSIGN(5,'DL1:KAXSTRES.DAT;1')
      DEFINE FILE 5(500,200,U,NF5)
      CALL ASSIGN (6,'DL1:KAXDATA.DAT;1')
      OPEN (UNIT=7,NAME='DL0:KAXIOUT.LST',DISPOSE='SAVE')
      CALL ASSIGN (8,'TI:')
      CALL ASSIGN (9,'DL1:KAXTTEMP.DAT;1')
      DEFINE FILE 9(500,40,U,NF9)
      CALL ASSIGN (10,'DL1:KAXTUNIT.DAT;1')
      DEFINE FILE 10(500,32,U,NF10)
C-----------------------------------------------------------------
C
      IR=6
      IW=7
      IC=8
      CALL TIME(TIM)
      WRITE(IW,106)TIM
106   FORMAT('     START OF RUN , TIME= 'A8)
      WRITE(IC,102)TIM
102   FORMAT('   START OF DATA INPUT , TIME= ',A8)
      CALL DATA2D
      IF(MODEX.GT.0)GO TO 1
      CALL TIME(TIM)
      WRITE(IC,103)TIM
103   FORMAT('    START OF STIFFNESS ROUTINE , TIME= ',A8)
      CALL FORMK
      CALL TIME(TIM)
      WRITE(IC,104)TIM
104   FORMAT('    START OF LOADING CALC. , TIME = ',A8)
      IF(NLD.LE.0)GO TO 10
      CALL MLOAD
10    IF(NTH.LE.0)GO TO 20
      CALL TLOAD
20    CONTINUE
      CALL BNDRY
      IF(MODEX.GT.0)GO TO 1
      CALL TIME(TIM)
      WRITE(IC,105)TIM
105   FORMAT('    START OF SOLUTION , TIME= ',A8)
      CALL SPARS
      CALL TIME(TIM)
      WRITE(IC,107)TIM
107   FORMAT('    START STRESS CALC. , TIME = ',A8)
      CALL STRESS
      GO TO 2
1     WRITE(IW,101)
101   FORMAT(5X,'CHECK AND CORRECT DATA IF NECESSARY')
2     CONTINUE
      CALL TIME(TIM)
      WRITE(IC,108)TIM
108   FORMAT('    END OF RUN , TIME= ',A8)
      WRITE(IW,109)TIM
109   FORMAT(20(/),5X,'END OF RUN , TIME= ',A8)
      CLOSE(UNIT=7,DISPOSE='SAVE')
      STOP
      END
```

```
      SUBROUTINE DATA2D
C
C  INPUT DATA ROUTINE,HANDLES UPTO 500 QUAD ELMS,500 NODES AND 10 MATL.
C
      IMPLICIT REAL*8(A-H,O-Z)
      COMMON/CNTRL/IR,IW,IC,ITYP,MODEX,THIC,NE,NP,NLD,NTH,NMAT,NMAX,
     1AMBT,TITLE(20),IPAGE
      COMMON/POP/PROP(10,10)
      COMMON CORD(500,2)
      DIMENSION NOP(5),NODE(5),D1(3),D2(3),DUM(10,5)
      DIMENSION TYPE(3,3),TAM(3)
      DATA  TYPE/'PLAN','E ST','RESS','PLAN','E ST','RAIN','AXIS','YMME'
     1,'TRIC'/
      DATA TAM/'R-RZ','Z-RT','T-ZT'/
      DATA ISO/' Q'/
      DATA ICST/'CS'/
C
C  READ CONTROL PARAM.
C
C  IGO=0,FOR READING NODES AND COORDS. FROM DIRECT ACCESS FILES
C   ITYP=1,FOR PLANE STRESS AND =2,FOR PLANE STRAIN AND =3,FOR AXI-
C   -SYMMETRIC; DEFAULTS TO 3
C  NE=NO. OF ELMS.
C  NP=NO. OF NODES
C  NMAT=NO. OF MATL. TYPES
C  NLD=NO.OF MECH. LOAD CASES
C  NTH=NO. OF TH. LOAD CASES
C  NMAX=MAX. CONNECTIVITY (OPT. HALF BAND WIDTH)
C  I1=1,FOR NO PRINTOUT OF INPUT DATA
C  MODEX=1, FOR DATA CHECK
C  MODEX=0,FOR SOLN.
C  AMBT=AMBIENT TEMP.,
C  THIC=THICKNESS FOR PLANE STRESS , DEFAULT=1.
C
      IPAGE=1
      READ(IR,101)TITLE
101   FORMAT(20A4)
      READ(IR,102)IGO,ITYP,NE,NP,NMAT,NLD,NTH,NMAX,I1,MODEX,AMBT,THIC
102   FORMAT(10I5,2E10.2)
C
C  CHECK ON CONTROL PARAM.
C
      IF(THIC.LE.0.)THIC=1.
      IF(NMAX.LE.0)NMAX=100
      IF(NP.LE.500)GO TO 1
      MODEX=1
      WRITE(IW,207)NP
207   FORMAT(/,5X,'NO. OF NODES EXCEEDS LIMIT , SPECIFIED= ',I5)
1     IF(NE.LE.500)GO TO 2
      MODEX=1
      WRITE(IW,208)NE
208   FORMAT(/,5X,'NO. OF ELEMENTS EXCEEDS LIMIT, SPECIFIED= ',I5)
2     IF(NMAT.LE.10)GO TO 3
      MODEX=1
      WRITE(IW,209)NMAT
209   FORMAT(/,5X,'NO. OF MATL TYPES EXCEEDS LIMIT , SPECIFIED= ',I5)
3     NCASE=NLD+NTH
      IF(NCASE.LE.10)GO TO 4
      MODEX=1
      WRITE(IW,210)NCASE
210   FORMAT(/,5X,'TOTAL NO. OF LOAD CASES GT. 10,SPECIFIED= ',I5)
4     CONTINUE
      IF(ITYP.LE.0.OR.ITYP.GT.3)ITYP=3
C
C  READ MATL. DATA
C
      DO 10 I=1,NMAT
      READ(IR,103)(PROP(I,J),J=1,10)
103   FORMAT(10E8.3)
      DO 11 J=1,9,3
      K1=J+1
      K2=J+2
      IF(PROP(I,J).GT.0.)GO TO 41
      WRITE(IW,301)J,I
301   FORMAT(/,5X,'MATL. PROP. NO.',I5,' FOR MATL. TYPE ',I5,' NOT VAL'
     1'ID')
      MODEX=1
41    CONTINUE
      IF(PROP(I,K1).LE.0.)PROP(I,K1)=PROP(I,J)
      IF(PROP(I,K2).LE.0.)PROP(I,K2)=PROP(I,J)
11    CONTINUE
      IF(PROP(I,10).LE.0.)PROP(I,10)=PROP(I,1)/2./(1.+PROP(I,4))
10    CONTINUE
C
C  BYPASS IF READING OFF DIRECT ACCESS FILES
      IF(IGO.LE.0)GO TO 20
C
C  READ ELEMENT DATA
C  DEFAULT ELEMENT TYPE =Q5 FOR ISOPARAMETRIC ELEMENTS
C
      IELO=5
      MATO=1
      DO 9 L=1,NE
      READ(IR,102)(NODE(J),J=1,4),MAT,IEL
      IF(MAT.EQ.0)MAT=MATO
      MATO=MAT
      IF(IEL.EQ.0)IEL=IELO
      IELO=IEL
9     WRITE(1'L)(NODE(J),J=1,4),MAT,IEL
C
C  READ NODAL COORD. DATA
C
      DO 6 L=1,NP
      DO 6 K=1,2
6     CORD(L,K)=-0.000
C
      N1=0
      DO 5 L=1,NP
```

```
      READ(IR,104)N,X,Z
104   FORMAT(I5,2F10.3)
      N1=MAX0(N1,N)
      IF(CORD(N,1).EQ.-0.000.AND.CORD(N,2).EQ.-0.000)GO TO 7
      WRITE(IW,302)L,N
302   FORMAT(5X,'DUPLICATE NODE CORD. DEF. ,CARD! ',I5,' NODE! ',I5)
      MODEX=1
7     WRITE(2'N)X,Z
      CORD(N,1)=X
5     CORD(N,2)=Z
C
      IF(N1.EQ.NP)GO TO 23
      MODEX=1
      WRITE(IW,217)N1,NP
217   FORMAT(5X,'NODE POINT DATA DOES NOT AGREE WITH SPEC., N1=',I5,
     1' NP=',I5)
      GO TO 23
20    CONTINUE
      DO 25 L=1,NP
25    READ(2'L)CORD(L,1),CORD(L,2)
C
C  PRINT INPUT DATA
C
23    WRITE(IW,199)IPAGE
199   FORMAT('1',//,T95,'PAGE',I5)
      IPAGE=IPAGE+1
      WRITE(IW,200)TITLE,(TYPE(J,ITYP),J=1,3)
200   FORMAT(/,5X,90('*'),/,5X,'*',88(' '),'*',/,5X,'*',4X,20A4,
     14X,'*',2(/,5X,'*',88(' '),'*'),/5X,'*',4X,'TWO DIMENSIONAL STRESS
     2ANALYSIS      -      ',3A4,' ANALYSIS',T95,'*',/,5X,'*',88(' '),'*')
      CALL STAMP
      LINE=37
      IF(I1.NE.0)GO TO 500
      WRITE(IW,202)NP,NE,NMAT,NTH,NLD,AMBT
202   FORMAT(10X,'I N P U T   D A T A !',//,10X,'NO. OF NODAL POINTS',
     1T40,I5,/,10X,'NO. OF ELEMENTS',T40,I5,/,10X,'NO. OF MATL. TYPES',
     2T40,I5,/,10X,'NO. OF TH. LOAD CASES',T40,I5,/,10X,'NO. OF MECH. LO
     3AD CASES',T40,I5,/,10X,'AMBIENT TEMP.          ',T35,F10.3,/)
      IF(ITYP.EQ.1)WRITE(IW,216)THIC
216   FORMAT(10X,'MATL. THICKNESS',T35,F10.3,/)
      WRITE(IW,211)
211   FORMAT(//,10X,'MATERIAL PROPERTIES',//,10X,'MATL.',9X,'MODULUS
     1OF  POISSONS   COEFF. OF      SHEAR',/,10X,'TYPE E-MU      ELASTICI
     2TY  RATIO      TH. EXP.       MOD.')
      DO 32 I=1,NMAT
      WRITE(IW,213)
213   FORMAT(/)
      LINE=LINE+4
      IF(LINE.LE.55)GO TO 31
      WRITE(IW,214)TITLE,IPAGE
      LINE=21
      IPAGE=IPAGE+1
      WRITE(IW,211)
31    CONTINUE
      DO 36 JJ=1,3
36    WRITE(IW,212)I,TAM(JJ),(PROP(I,K),K=JJ,10,3)
212   FORMAT(I13,2X,A4,2X,E12.3,F8.2,2E15.3)
32    CONTINUE
      WRITE(IW,214)TITLE,IPAGE
214   FORMAT('1',///,10X,20A4,T95,'PAGE',I5)
      IPAGE=IPAGE+1
      WRITE(IW,203)
      LINE=12
203   FORMAT(//,10X,'E L E M E N T   D A T A',//,2(5X,'ELM',16X,'NODE',
     117X,'- TYPE OF -',10X),/,2(5X,'NO.',5X,9('-'),' NUMBERS ',9('-'),
     25X,'MAT',5X,'ELM',10X))
      DO 24 I=1,NE,2
      READ(1'I)(NODE(J),J=1,5),I1
      IJ=ICST
      IF(NODE(4).GT.0)IJ=ISO
      LINE=LINE+1
      IF(LINE.LE.55)GO TO 13
      LINE=12
      WRITE(IW,214)TITLE,IPAGE
      IPAGE=IPAGE+1
      WRITE(IW,203)
13    N=I+1
      IF(N.GT.NE)GO TO 15
      READ(1'N)(NOP(J),J=1,5),I2
      IK=ICST
      IF(NOP(4).GT.0)IK=ISO
      WRITE(IW,201)I,(NODE(J),J=1,5),IJ,I1,N,(NOP(J),J=1,5),IK,I2
201   FORMAT(2(6(5X,I3),5X,A2,I1,10X))
      GO TO 24
15    WRITE(IW,201)I,(NODE(J),J=1,5)
24    CONTINUE
C
      WRITE(IW,214)TITLE,IPAGE
      IPAGE=IPAGE+1
      WRITE(IW,204)
      LINE=10
204   FORMAT(//,10X,'N O D A L   C O O R D I N A T E   D A T A',//,
     13(10X,'NODE NO.',7X,'R( X )',3X,'Z( Y )'))
      DO 33 I=1,NP,3
      N=1
      DO 34 J=I,I+2
      IF(J.GT.NP)GO TO 35
      D1(N)=CORD(J,1)
      D2(N)=CORD(J,2)
      NOP(N)=J
      N=N+1
34    CONTINUE
35    WRITE(IW,206)(NOP(K),D1(K),D2(K),K=1,N-1)
      LINE=LINE+1
      IF(LINE.LE.55)GO TO 33
      LINE=10
      WRITE(IW,214)TITLE,IPAGE
      IPAGE=IPAGE+1
```

```
      WRITE(IW,204)
33    CONTINUE
206   FORMAT(3(I15,5X,2F10.3))
500   CONTINUE
C
C  WRITE ONTO FILE THE AMBIENT TEMP.
C
      DO 43 I=1,10
      DO 44 J=1,4
44    DUM(I,J)=0.
43    DUM(I,5)=AMBT
      DO 45 I=1,NE
45    WRITE(5'I)((DUM(J,K),K=1,5),J=1,NCASE)
      RETURN
      END
```

```
      SUBROUTINE STAMP
      COMMON/CNTRL/IR,IW
      DIMENSION IDATE(5)
      CALL DATE(IDATE)
      WRITE(IW,201)IDATE
201   FORMAT(5X,90('*'),/,2(5X,'*',88(' '),'*',/),5X,'*',2X,'PREPARED BY
     1!.......................,CHECKED BY!.........................,DATE RUN!',
     25A2,3X,'*',/,5X,'*',T95,'*',/,5X,'*',88(' '),'*',/,5X,90('*'),//)
      RETURN .
      END
```

```
      SUBROUTINE FORMK
C
C  ROUTINE TO ASSEMBLE THE TOTAL STIFFNESS MATRIX(GLOBAL),SET UP A
C  POINTER MATRIX AND STORE ON DISC/TAPE
C
C  STIFF=ELEMENT STIFFNESS MATRIX (12*12)
C  SI=COMPACT STIFFNESS MATRIX EACH ROW
C  ISP=POINTER MATRIX EACH ROW
C  FILE 3  STIFFNESS FILE
C
      IMPLICIT REAL*8(A-H,O-Z)
      COMMON/CNTRL/IR,IW,IC,ITYP,MODEX,THIC,NE,NP,NLD,NTH,NMAT,NMAX,
     1AMBT
      COMMON /POP/PROP(10,10)
      COMMON CORD(500,2)
      DIMENSION STIFF(12,12),SI(100),ISP(100),NODE(4),RHS(12)
      DIMENSION D(4,4),DE(4)
      NEQ=NP*2
      NCASE=NLD+NTH
C
C  ZERO STIFFNESS FILE
C
      DO 40 I=1,NMAX
      SI(I)=0.
40    ISP(I)=0
      DO.41 I=1,NEQ
      WRITE(3'I)(SI(J),J=1,NMAX),I,(ISP(J),J=2,NMAX),(SI(J),J=1,10)
41    CONTINUE
C
C  START SCAN FOR EACH ELEMENT
C
      VOLUME=0.
      MATO=0
      DO 1 N=1,NE
      READ(1'N)(NODE(L),L=1,4),MAT,IEL
      IF(MAT.EQ.MATO)GO TO 42
      BETA=0.
      CALL MATRXD(MAT,BETA,D,DE)
      MATO=MAT
42    CONTINUE
      WRITE(IC,201)N
201   FORMAT('+',5X,'CALCULATING STIFFNESS FOR ELEMENT ',I5,'    ')
101   CONTINUE
C
C  CALCULATE ELEMENT STIFFNESS
C
      NON=4
      IF(NODE(4).GT.0)GO TO 43
      NON=3
      CALL CST(1,N,NODE,D,DE,STIFF,RHS,VOL)
      GO TO 44
43    CALL ISOQ(IEL,1,2,N,NON,NODE,D,DE,STIFF,RHS,VOL)
44    VOLUME=VOLUME+VOL
      I=0
      DO 350 JJ=1,NON
C
C  COMPUTE ROW NO. AND READ CURRENT VALUES FROM FILE 3
C
      NROW=(NODE(JJ)-1)*2
      DO 350 J=1,2
      NROW=NROW+1
      I=I+1
      READ(3'NROW)(SI(II),II=1,NMAX),(ISP(II),II=1,NMAX)
C
C  COMPUTE COLUMN NO.
C
      II=0
      DO 330 KK=1,NON
      NCOL=(NODE(KK)-1)*2
      DO 330 K=1,2
      NCOL=NCOL+1
      II=II+1
C
C  TRIANGULAR MATRIX FOR SOLUTION
C
      IF(NROW.GT.NCOL)GO TO 330
C
C  SEARCH POINTER FOR COL. NO.
C
      DO 310 M=1,NMAX
      IF(ISP(M).EQ.NCOL)GO TO 320
      IF(ISP(M).LE.0)GO TO 315
310   CONTINUE
      WRITE(IW,420)NROW,NCOL,M,NMAX
420   FORMAT(10X,'ERROR  - POINTER OVERFLOW',' ROW= ',I5,' COL= ',I5,
     1'COUNTER= ',I5,' GREATER THAN ',I5)
      MODEX=1
      GO TO 1
315   ISP(M)=NCOL
320   SI(M)=STIFF(I,II)+SI(M)
C  END LOOP FOR COL.
330   CONTINUE
C
C  UPDATE FILE WITH NEW VALUES
C
      WRITE(3'NROW)(SI(II),II=1,NMAX),(ISP(II),II=1,NMAX)
C  END LOOP FOR ROWS
350   CONTINUE
C  STORE UNIT THERMAL LOAD VECTORS
C
      IF(NTH.LE.0)GO TO 1
      WRITE(10'N)(RHS(K),K=1,NON*2)
C  END LOOP FOR ELEMENTS
1     CONTINUE
      WRITE(IW,202)VOLUME
202   FORMAT(/,5X,'VOLUME OF THE MODEL= ',F10.1)
      RETURN
      END
```

```
      SUBROUTINE MATRXD(MAT,BETA,D,DE)
C
C  ROUTINE TO CALCULATE THE STRESS-STRAIN MATRIX FOR 2D ELEMENTS (D)
C  VALID FOR ELASTIC MATERIALS ONLY
C  ALSO CALCULATE MATRIX [DE] (4)
C  FOR PLANE STRESS,ITYP=1,(D)=(3*3)
C  FOR PLANE STRAIN,ITYP=2,(D)=(3*3)
C  FOR AXISYMMETRIC,ITYP=3,(D)=(4*4)
C  MATL=ELEMENT MATERIAL TYPE (MAX.=10)
C  PROP=MATERIAL PROPERTIES ARRAY
C  PROP(MAT,1)=ER OR EX
C  PROP(MAT,2)=EZ OR EY
C  PROP(MAT,3)=ET
C  PROP(MAT,4)=MU RZ  OR MU XY
C  PROP(MAT,5)=MU RT  OR MU XT
C  PROP(MAT,6)=MU ZT  OR MU YT
C  PROP(MAT,10)=G RZ
C
      IMPLICIT REAL*8(A-H,O-Z)
      COMMON/CNTRL/IR,IW,IC,ITYP,MODEX
      COMMON /POP/PROP(10,10)
      DIMENSION D(4,4),DE(4),T(4,4),TD(4,4)
C
C  INITIALIZE MATRIX
C
      DO 4 I=1,4
      DE(I)=0.
      DO 4 J=1,4
      TD(I,J)=0.
4     D(I,J)=0.
C
      DO 6 I=1,3
6     T(2,I)=PROP(MAT,I+6)
C
      D(1,1)=1./PROP(MAT,1)
      D(2,2)=1./PROP(MAT,2)
      D(3,3)=1./PROP(MAT,3)
      D(4,4)=1./PROP(MAT,10)
      D(1,2)=-PROP(MAT,4)*D(2,2)
      D(1,3)=-PROP(MAT,5)*D(3,3)
      D(2,3)=-PROP(MAT,6)*D(3,3)
      D(2,1)=D(1,2)
      D(3,1)=D(1,3)
      D(3,2)=D(2,3)
C
C  INVERT MATRIX TO OBTAIN STRESS-STRAIN MATRIX [D] (4*4)
C
      CALL INVERT(D)
C
      IF(BETA.EQ.0.)GO TO 500
C
      DO 508 I=1,4
      DO 508 J=1,4
508   T(I,J)=0.
      ANG=BETA/57.2957795
      SN=DSIN(ANG)
      CS=DCOS(ANG)
      SC=SN*CS
      S2=SN*SN
      C2=CS*CS
C
C  SET UP TRANSFORMATION MATRIX [T] (4,4)
C
      T(1,1)=C2
      T(1,2)=S2
      T(1,4)=SC
      T(2,1)=S2
      T(2,2)=C2
      T(2,4)=-T(1,4)
      T(3,3)=1.
      T(4,1)=-2.*SC
      T(4,2)=-T(4,1)
      T(4,4)=C2-S2
C
C  FORM [T]T [D]
C
      DO 501 I=1,4
      DO 501 J=1,4
      SUM=0.
      DO 504 K=1,4
504   SUM=SUM+T(K,I)*D(K,J)
501   TD(I,J)=SUM
C
C  FORM [T]T [D] [T]
C
      DO 505 I=1,4
      DO 505 J=1,4
      SUM=0.
      DO 506 K=1,4
506   SUM=SUM+TD(I,K)*T(K,J)
      D(I,J)=SUM
505   D(J,I)=SUM
C
500   CONTINUE
      JK=4
C
      GO TO(1,2,3),ITYP
C
C  MODIFY THE D MATRIX FOR PLANE STRESS ANALYSIS
1     CONTINUE
C
      D(1,1)=D(1,1)-D(3,1)*D(1,3)/D(3,3)
      D(1,2)=D(1,2)-D(3,2)*D(1,3)/D(3,3)
      D(1,4)=D(1,4)-D(3,4)*D(1,3)/D(3,3)
      D(2,2)=D(2,2)-D(3,2)*D(2,3)/D(3,3)
      D(2,4)=D(2,4)-D(3,4)*D(2,3)/D(3,3)
      D(4,4)=D(4,4)-D(3,4)*D(4,3)/D(3,3)
      DO 503 I=1,4
```

```
      DO 503 J=1,4
503   D(J,I)=D(I,J)
C
C  MODIFY THE D MATRIX FOR PLANE STRAIN ANALYSIS
2     CONTINUE
      JK=3
C
      SAVE=D(4,4)
      DO 502 I=1,4
      DO 502 J=3,4
      D(I,J)=0.
502   D(J,I)=0.
      D(3,3)=SAVE
3     CONTINUE
      T(2,JK)=0.
C
C  MODIFY COEFF. OF THERMAL EXP.
C
      IF(BETA.EQ.0.)GO TO 5
      DO 507 I=1,3
507   T(1,I)=PROP(MAT,I+6)
      T(2,1)=C2*T(1,1)+S2*T(1,2)
      T(2,2)=S2*T(1,1)+C2*T(1,2)
      T(2,3)=T(1,3)
      T(2,JK)=2.*SC*(T(1,1)-T(1,2))
5     CONTINUE
C  CALCULATE THE MATRIX [DE] (4)
C
      DO 509 I=1,JK
      DO 509 J=1,JK
509   DE(I)=DE(I)+D(I,J)*T(2,J)
C
      RETURN
      END
```

```
      SUBROUTINE INVERT(A)
C
C  CALLED BY MATRXD
C
C     INVERSION OF A POSITIVE DEFINITE MATRIX
C
      IMPLICIT REAL*8(A-H,O-Z)
      COMMON/CNTRL/IR,IW,ITYP,MODEX
      DIMENSION A(4,4)
C
      DO 200 N=1,4
C
      IF(DABS(A(N,N)).GT.1.E-20)GO TO 50
      WRITE(IW,2000)
2000  FORMAT(/,2X,'MATRIX SINGULAR')
      MODEX=1
      RETURN
C
   50 D = 1.0/ A(N,N)
      DO 100 J=1,4
  100 A(N,J) = -A(N,J) * D
C
      DO 150 I=1,4
      IF(N.EQ.I) GO TO 150
      DO 140 J=1,4
      IF(N.EQ.J) GO TO 140
      A(I,J) = A(I,J) + A(I,N)*A(N,J)
  140 CONTINUE
  150 A(I,N) = A(I,N)* D
C
      A(N,N)= D
C
 200  CONTINUE
      RETURN
      END
```

```
      SUBROUTINE ISOQ(IEL,IPASS,NINT,NEL,NON,NODE,D,DE,S,RHS,VOL)
C
C  ROUTINE TO CALCULATE THE STIFFNESS MATRIX FOR AN ISOPARMETRIC QUAD
C  ELEMENT FOR PLANE STRESS,PLANE STRAIN OR AXISYMMETRIC ANALYSIS
C  FOR ELEMENT Q4,Q5 OR Q6
C
C  IEL =ELEMENT TYPE , 4,5 OR 6   ,DEFAULT=4.
C  IPASS=1, FOR STIFFNESS GENERATION AND =2, FOR STRESS RECOVERY
C  NINT =GAUSS NUMERICAL INTEGRATION ORDER
C
      IMPLICIT REAL*8(A-H,O-Z)
      COMMON /CNTRL/IR,IW,IC,ITYP,MODEX,THIC,NELM,NC
      COMMON CORD(500,2)
      DIMENSION D(4,4),B(4,12),S(12,12),XG(4,4),WGT(4,4),DB(4),NODE(4),
     1DE(4),RHS(12)
C
C  MATRIX XG STORES GAUSS-LEGENDRE SAMPLING POINTS
C  MATRIX WGT STORES GAUSS-LEGENDRE WEIGHTING FACTORS
C
      DO 1 I=1,4
      DO 1 J=1,4
      WGT(I,J)=0.
1     XG(I,J)=0.
C
      XG(1,2)=-.5773502691896
      XG(2,2)=-XG(1,2)
      XG(1,3)=-.7745966692415
      XG(3,3)=-XG(1,3)
      XG(1,4)=-.8611363115941
      XG(4,4)=-XG(1,4)
      XG(2,4)=-.3399810435849
      XG(3,4)=-XG(2,4)
C
      WGT(1,1)=2.
      WGT(1,2)=1.
      WGT(2,2)=1.
      WGT(1,3)=.5555555555556
      WGT(3,3)=WGT(1,3)
      WGT(2,3)=.8888888888889
      WGT(1,4)=.3478548451375
      WGT(4,4)=WGT(1,4)
      WGT(2,4)=.6521451548625
      WGT(3,4)=WGT(2,4)
C
      NS=IEL*2
      PHI2=8.*DATAN(1.)
C
C  CHECK FOR TRIANGLE
      IF(NON.LE.3)NODE(4)=NODE(1)
C
      VOL=0.
C  INITIALIZE STIFFNESS MATRIX
C  AND THE UNIT THERMAL LOAD VECTOR
      DO 30 I=1,NS
      RHS(I)=0.
      DO 30 J=1,NS
30    S(I,J)=0.
C
      IST=3
      IF(ITYP.EQ.3)IST=4
C  LOOP OVER THE INTEGRATION ORDER
      DO 80 LX=1,NINT
      RI=XG(LX,NINT)
      DO 80 LY=1,NINT
      SI=XG(LY,NINT)
C
C  EVALUATE THE DERIVATIVE OPERATOR B AND THE JAC. DETERMINANT DET
      CALL ISTDM(IEL,NODE,B,DET,RI,SI,XBAR,NEL)
      IF(MODEX.GT.0)RETURN
C
C  ADD CONTRIBUTION TO ELEMENT STIFFNESS
C
      XBAR=XBAR*PHI2
      IF(ITYP.LT.3)XBAR=THIC
      WT=WGT(LX,NINT)*WGT(LY,NINT)*XBAR*DET
      VOL=VOL+WT
      DO 70 J=1,NS
      DO 40 K=1,IST
      RHS(J)=RHS(J)+B(K,J)*DE(K)*WT
      DB(K)=0.
      DO 40 L=1,IST
40    DB(K)=DB(K)+D(K,L)*B(L,J)
      DO 60 I=J,NS
      STIFF=0.
      DO 50 L=1,IST
50    STIFF=STIFF+B(L,I)*DB(L)
60    S(I,J)=S(I,J)+STIFF*WT
70    CONTINUE
80    CONTINUE
C
      DO 90 J=1,NS
      DO 90 I=J,NS
90    S(J,I)=S(I,J)
C
C  CALCULATE [B] MATRIX AT ELEMENT CG. FOR STRESS RECOVERY
C
      IF(IPASS.EQ.2)CALL ISTDM(IEL,NODE,B,DET,0.,0.,XBAR,NEL)
      IF(IEL.EQ.4)GO TO 20
C
C  CONDENSE THE INTERNAL DOFS. FOR Q5 AND Q6
C
      NS1=NS-8
      DO 4 K=1,NS1
      LL=NS-K
      KK=LL+1
      DO 3 L=1,LL
      PIVOT=S(L,KK)/S(KK,KK)
      DO 5 M=1,L
```

```
5     S(M,L)=S(M,L)-S(M,KK)*PIVOT
      GO TO(3,6),IPASS
6     CONTINUE
      DO 7 J=1,4
7     B(J,L)=B(J,L)-B(J,KK)*PIVOT
3     RHS(L)=RHS(L)-RHS(KK)*PIVOT
4     CONTINUE
      DO 8 I=1,8
      DO 8 J=1,8
8     S(J,I)=S(I,J)
C
20    IF(NON.EQ.3)NODE(4)=0
C
      IF(IPASS.EQ.1)RETURN
C
C  RETURN [B] MATRIX IN [S]
C
      DO 2 I=1,4
      DO 2 J=1,8
2     S(I,J)=B(I,J)
      RETURN
      END
```

```
      SUBROUTINE ISTDM(IEL,NODE,B,DET,R,S,XBAR,NEL)
C
C  ROUTINE TO EVALUATE THE STRAIN - DISPLACEMENT MATRIX B (4*12) AT A
C  POINT  (R,S) FOR A 2D QUAD ELEMENT  Q4 , Q5 OR Q6
C  [B] = (4*8) , FOR Q4
C      = (4,10) , FOR Q5
C      = (4,12) , FOR Q6
C
      IMPLICIT REAL *8(A-H,O-Z)
      COMMON /CNTRL/IR,IW,IC,ITYP,MODEX,THIC
      COMMON CORD(500,2)
      DIMENSION NODE(4),B(4,12),H(6),P(2,6),XJ(2,2),XJI(2,2)
C
C
      RP=1.+R
      SP=1.+S
      RM=1.-R
      SM=1.-S
C
C  INTERPOLATION FUNCTIONS / SHAPE FUNCTIONS
C
      H(1)=RP*SP/4.
      H(2)=RM*SP/4.
      H(3)=RM*SM/4.
      H(4)=RP*SM/4.
C
C  DERIVATIVES OF THE SHAPE FUNCTIONS WRT THE NATURAL COORD.
C
C  1. WITH RESPECT TO R
C
      P(1,1)=SP/4.
      P(1,2)=-P(1,1)
      P(1,3)=-SM/4.
      P(1,4)=-P(1,3)
C
C  2. WITH RESPECT TO S
C
      P(2,1)=RP/4.
      P(2,2)=RM/4.
      P(2,3)=-P(2,2)
      P(2,4)=-P(2,1)
C
C  ADD ADDITIONAL SHAPE FUNCTIONS AND THEIR DERIVATIVES
C
      GO TO(4,4,4,4,5,6),IEL
5     CONTINUE
C  FOR ELEMENT Q5
      H(5)=RP*RM*SP*SM
      P(1,5)=-2.*R*SP*SM
      P(2,5)=-2.*S*RP*RM
      GO TO 4
6     CONTINUE
C  FOR ELEMENT Q6
      H(5)=RP*RM
```

```
      H(6)=SP*SM
      P(1,5)=-2.*R
      P(2,5)=0.
      P(1,6)=0.
      P(2,6)=-2.*S
4     CONTINUE
C
C  COMPUTE THE JACOBIAN MATRIX AT POINT (R,S)
C
      DO 30 I=1,2
      DO 30 J=1,2
      DUM=0.
      DO 20 K=1,4
20    DUM=DUM+P(I,K)*CORD(NODE(K),J)
30    XJ(I,J)=DUM
C
C  COMPUTE THE DETERMINANT OF THE JACOBIAN AT THE ABOVE POINT
C
      DET=XJ(1,1)*XJ(2,2)-XJ(2,1)*XJ(1,2)
      IF(DET.GT.0.)GO TO 40
      WRITE(IW,201)NEL
201   FORMAT(//,5X,'ZERO OR NEGETIVE DET. FOR ELM. ',I5)
      MODEX=1
      RETURN
40    CONTINUE
C
C  COMPUTE THE INVERSE OF THE JAC.
C
      DUM=1./DET
      XJI(1,1)=XJ(2,2)*DUM
      XJI(1,2)=-XJ(1,2)*DUM
      XJI(2,1)=-XJ(2,1)*DUM
      XJI(2,2)=XJ(1,1)*DUM
C
C  INITIALIZE [B] MATRIX
C
      DO 110 I=1,4
      DO 110 J=1,IEL*2
110   B(I,J)=0.
C
C  EVALUATE THE GLOBAL DERIVATIVE MATRIX B (4*12)
C  PLACE THE TERM RELATED TO THE SHEAR STRAIN IN THE CORRECT ROW
C  ROW 3, FOR PLANE STRESS OR PLANE STRAIN AND ROW 4, FOR AXISYMMETRIC
C
      K2=0
      K3=3
      IF(ITYP.EQ.3)K3=4
C
      DO 60 K=1,IEL
      K2=K2+2
      DO 50 I=1,2
      B(1,K2-1)=B(1,K2-1)+XJI(1,I)*P(I,K)
50    B(2,K2)=B(2,K2)+XJI(2,I)*P(I,K)
      B(K3,K2)=B(1,K2-1)
60    B(K3,K2-1)=B(2,K2)
C
C  IN CASE OF PLANE STRAIN / PLANE STRESS ANALYSIS EXCLUDE THE NORMAL
C  STRAIN COMPONENT
C
      IF(ITYP.LT.3)RETURN
C
C  COMPUTE THE RADIUS AT POINT (R,S)
C
      XBAR=0.
      DO 70 K=1,4
70    XBAR=XBAR+H(K)*CORD(NODE(K),1)
C
C  EVALUATE THE HOOP STRAIN -DISPLACEMENT RELATION
C
      IF(XBAR.GT.0.00001)GO TO 90
C  FOR THE CASE OF ZERO RADIUS EQUATE THE RADIAL TO THE HOOP STRESS
C
      DO 80 K=1,IEL*2
80    B(3,K)=B(1,K)
      RETURN
C
C  FOR NON ZERO RAD.
C
90    DUM=1./XBAR
      K2=0
      DO 100 K=1,IEL
      K2=K2+2
100   B(3,K2-1)=H(K)*DUM
      RETURN
      END
```

```
      SUBROUTINE CST(IPASS,N,NODE,D,DE,STIFF,RHS,VL)
C
C  ROUTINE TO CALCULATE THE ELEMENT STIFFNESS BASED ON A CST
C  OUTPUT FROM THE ROUTINE CONSISTS OF STIFFNESS MATRIX(6,6)
C  ,UNIT THERMAL LOAD VECTOR(6) AND THE ELM. VOLUME.
C  IPASS=1,FOR STIFFNESS GENERATION ,ARRAY STIFF CONTAINS STIFFNESS
C       =2,FOR STRESS RECOVERY,ARRAY STIFF CONTAINS MATRIX [B]
C
      IMPLICIT REAL*8(A-H,O-Z)
      COMMON /CNTRL/IR,IW,IC,ITYP,MODEX,THIC,NELM,NC
      COMMON /POP/PROP(10,10)
      COMMON CORD(500,2)
      DIMENSION STIFF(12,12),RHS(12),NODE(4),DE(4),D(4,4),B(4,6),
     1     DB(4,6)
C
      PHI2=8.*DATAN(1.)
      NON=3
      MN=NON*2
      ICMP=4
      IF(ITYP.NE.3)ICMP=3
C
      CALL MATRXB(N,NODE,AR,RMN,B)
      VL=PHI2*RMN*AR
C
      IF(IPASS.EQ.2)GO TO 15
C
C  CALCULATE UNIT THERMAL LOAD VECTOR
C
      DO 7 K=1,6
      RHS(K)=0.
      DO 7 J=1,ICMP
7     RHS(K)=RHS(K)+B(J,K)*DE(J)*VL
C
C  CALCULATE MATRIX [D][B] (4,6)
C
      DO 5 I=1,ICMP
      DO 5 J=1,MN
      DB(I,J)=0.
      DO 5 K=1,ICMP
5     DB(I,J)=DB(I,J)+D(I,K)*B(K,J)
C
C  CALCULATE MATRIX [B]T[D][B] (6,6)
C
      DO 6 I=1,MN
      DO 6 J=1,MN
      STIFF(I,J)=0.
      DO 6 K=1,ICMP
6     STIFF(I,J)=STIFF(I,J)+B(K,J)*DB(K,I)*VL
C
      RETURN
C
C  RETURN [B] IN [STIFF]
C
15    CONTINUE
      DO 2 I=1,4
      DO 2 J=1,MN
2     STIFF(I,J)=B(I,J)
C
      RETURN
      END
```

```
      SUBROUTINE MATRXB(NO,NODE,AREA,RMN,B)
C
C  ROUTINE TO CALCULATE THE STRAIN-DISPLACEMENT MATRIX (B) (4*6)
C  ITYP=1,FOR PLANE STRESS ANALYSIS
C  ITYP=2,FOR PLANE STRAIN ANALYSIS
C  ITYP=3,FOR AXISYMMETRIC ANALYSIS
C  ROUTINE VALID FOR TRIANGULAR ELEMENTS ONLY
C
C  CORD=NODE COORD. ARRAY
C  NODE=NODE NOS. AND TYP ARRAY
C  AREA=ELEMENT AREA
C
      IMPLICIT REAL*8(A-H,O-Z)
      COMMON /CNTRL/IR,IW,IC,ITYP,MODEX,THIC
      COMMON CORD(500,2)
      DIMENSION B(4,6),NODE(4)
C
      PHI2=8.*DATAN(1.)
C
C  INITIALIZE B MATRIX
C
      DO 1 I=1,4
      DO 1 J=1,6
1     B(I,J)=0.
      I=NODE(1)
      J=NODE(2)
      K=NODE(3)
C
C  CALCULATE THE AREA OF THE ELEMENT
C
      AREA=(CORD(J,1)*CORD(K,2)-CORD(K,1)*CORD(J,2)-CORD(I,1)*CORD(K,2)+
     1CORD(I,1)*CORD(J,2)+CORD(K,1)*CORD(I,2)-CORD(J,1)*CORD(I,2))/2.
      IF(AREA.LE.0.)GO TO 2
C
C  SETUP LOCAL COORD. SYSTEM
C
      BI=CORD(J,2)-CORD(K,2)
      BJ=CORD(K,2)-CORD(I,2)
      BK=CORD(I,2)-CORD(J,2)
      CI=CORD(K,1)-CORD(J,1)
      CJ=CORD(I,1)-CORD(K,1)
      CK=CORD(J,1)-CORD(I,1)
C
C  BRANCHOUT AS PER TYPE OF ANALYSIS
C
      IF(ITYP.EQ.3)GO TO 4
C
C  FOR PLANE STRESS/STRAIN ELEMENT
C
      RMN=THIC/PHI2
      B(1,1)=BI
      B(1,3)=BJ
      B(1,5)=BK
      B(2,2)=CI
      B(2,4)=CJ
      B(2,6)=CK
      B(3,1)=CI
      B(3,2)=BI
      B(3,3)=CJ
      B(3,4)=BJ
      B(3,5)=CK
      B(3,6)=BK
      GO TO 5
4     CONTINUE
C
C  FOR AXISYMMETRIC ANALYSIS
C
C  ADDITIONAL LOCAL COORDINATES
      AI=CORD(J,1)*CORD(K,2)-CORD(K,1)*CORD(J,2)
      AJ=CORD(K,1)*CORD(I,2)-CORD(I,1)*CORD(K,2)
      AK=CORD(I,1)*CORD(J,2)-CORD(J,1)*CORD(I,2)
      RMN=(CORD(I,1)+CORD(J,1)+CORD(K,1))/3.
      ZMN=(CORD(I,2)+CORD(J,2)+CORD(K,2))/3.
      ZR=ZMN/RMN
      FI=AI/RMN+BI+CI*ZR
      FJ=AJ/RMN+BJ+CJ*ZR
      FK=AK/RMN+BK+CK*ZR
C
      B(2,2)=CI
      B(2,4)=CJ
      B(2,6)=CK
      B(1,1)=BI
      B(1,3)=BJ
      B(1,5)=BK
      B(3,1)=FI
      B(3,3)=FJ
      B(3,5)=FK
      B(4,1)=CI
      B(4,2)=BI
      B(4,3)=CJ
      B(4,4)=BJ
      B(4,5)=CK
      B(4,6)=BK
C
5     CONTINUE
      DO 3 I=1,4
      DO 3 J=1,6
3     B(I,J)=B(I,J)/AREA/2.
      RETURN
C
2     WRITE(IW,101)NO,(NODE(J),J=1,3)
101   FORMAT(/,5X,'ELM. NO.',I5,' WITH NODES ',3I5,' HAS NEGATIVE AREA')
      MODEX=1
      RETURN
      END
```

```
      SUBROUTINE MLOAD
C
C  ROUTINE TO READ MECH. LOADING AND GENERATE THE R.H.S. VECTOR
C  CAN HANDLE NODAL LOADS AND SURFACE PRESSURE
C
      IMPLICIT REAL*8(A-H,O-Z)
      COMMON/CNTRL/IR,IW,IC,ITYP,MODEX,THIC,NE,NP,NLD,NTH,NMAT,NMAX,
     1AMBT,TITLE(20),IPAGE
      COMMON CORD(500,2)
      DIMENSION RHS(1000),SI(100),ISP(100),DUMY2(10),R(2),NAME(24)
      DIMENSION SUM(2)
      DATA ID1,ID2,ID3/'  ','FR','PR'/
      NEQ=NP*2
      LI=0
      DO 1 III=1,NLD
C
      SUM(1)=0.
      SUM(2)=0.
C
      LI=LI+1
      WRITE(IC,203)LI
203   FORMAT('+',5X,'SETTING UP NODAL LOADS FOR CASE ',I3,'     ')
C
C  INITIALIZE LOAD
      DO 2 J=1,NEQ
2     RHS(J)=0.
C
      WRITE(IW,100)TITLE,IPAGE
100   FORMAT('1',///,10X,20A4,T95,'PAGE',I5,//)
      IPAGE=IPAGE+1
C
C  BRANCH OUT AS NECESSARY
C
      LINE=5
173   READ(IR,101)ID,NAME
101   FORMAT(A2,24A2)
      IF(ID.EQ.ID2)GO TO 171
      IF(ID.EQ.ID3)GO TO 172
      IF(ID.EQ.ID1)GO TO 161
      MODEX=1
      WRITE(IW,201)LI
201   FORMAT(/,5X,'HEADER CARD FOR LOAD CASE NO ',I5,' NOT VALID')
      GO TO 1
C
C  NODAL LOADS
C
171   WRITE(IW,109)LI,NAME
109   FORMAT(10X,'LOAD CASE= ',I5,5X,24A2,///,10X,'NODAL LOADS',/,10X,
     1'NODE NO.',8X,'RADIAL(X)',7X,'AXIAL(Y)')
      LINE=12
C     READ,PRINT AND STORE LOAD CARD
166   READ(IR,9)NQ,R(1),R(2)
      IF(NQ.LE.0)GO TO 173
      LINE=LINE+1
      IF(LINE.LE.55)GO TO 20
      LINE=10
      WRITE(IW,100)TITLE,IPAGE
      IPAGE=IPAGE+1
      WRITE(IW,109)LI,NAME
20    WRITE(IW,91)NQ,(R(K),K=1,2)
91    FORMAT(10X,I5,5X,2(5X,F10.3))
      FAC=THIC
      IF(ITYP.EQ.3)FAC=6.2832*CORD(NQ,1)
      DO 170 K=1,2
      ICC=(NQ-1)*2+K
      RHH=R(K)*FAC
      SUM(K)=SUM(K)+RHH
170   RHS(ICC)=RHS(ICC)+RHH
      GO TO 166
 9    FORMAT(I5,2E10.3)
C
C  PRESSURE LOADING
C  CONVERSION OF SURFACE STRESSES INTO EQ. NODAL FORCES
C  PRESS. MUST ACT FROM LEFT TO RIGHT , LOOKING FROM I1 TO I2
C
172   WRITE(IW,110)LI,NAME
110   FORMAT(10X,'LOAD CASE= ',I5,5X,24A2,///,10X,'PRESSURE LOADING',/,
     112X,'NODE NO.',4X,'PRESS.',/,11X,'FROM   TO')
      LINE=14
165   READ(IR,111)I1,I2,PR
111   FORMAT(2I5,F10.2)
      IF(I1.LE.0)GO TO 173
      LINE=LINE+1
      IF(LINE.LE.55)GO TO 21
      LINE=12
      WRITE(IW,100)TITLE,IPAGE
      IPAGE=IPAGE+1
      WRITE(IW,110)LI,NAME
21    WRITE(IW,112)I1,I2,PR
112   FORMAT(10X,2I5,F10.2)
      DIST=DSQRT((CORD(I1,1)-CORD(I2,1))**2+(CORD(I1,2)-CORD(I2,2))**2)
      RBAR=.5*(CORD(I1,1)+CORD(I2,1))
      DZ=CORD(I2,2)-CORD(I1,2)
      DR=CORD(I2,1)-CORD(I1,1)
      FAC=THIC
      IF(ITYP.EQ.3)FAC=6.2832*RBAR
      FORCE=FAC*DIST*PR
      FR=.5*FORCE*DZ/DIST
      FZ=-.5*FORCE*DR/DIST
      IG=1
      KR=2*I1-1
164   KZ=KR+1
      RHS(KR)=RHS(KR)+FR
      RHS(KZ)=RHS(KZ)+FZ
      SUM(1)=SUM(1)+FR
      SUM(2)=SUM(2)+FZ
      KR=2*I2-1
      GO TO(162,165),IG
```

```
162   IG=2
      GO TO 164
161   CONTINUE
      IJK=LI-1
      IF(IJK.LE.0)GO TO 19
      DO 167 J=1,NEQ
      READ(3'J)(SI(K),K=1,NMAX),(ISP(K),K=1,NMAX),(DUMY2(K),K=1,IJK)
167   WRITE(3'J)(SI(K),K=1,NMAX),(ISP(K),K=1,NMAX),(DUMY2(K),K=1,IJK)
     1,RHS(J)
      GO TO 4
19    CONTINUE
      DO 168 J=1,NEQ
      READ(3'J)(SI(K),K=1,NMAX),(ISP(K),K=1,NMAX)
168   WRITE(3'J)(SI(K),K=1,NMAX),(ISP(K),K=1,NMAX),RHS(J)
4     CONTINUE
      WRITE(IW,114)(SUM(J),J=1,2)
114   FORMAT(//,10X,'NODAL LOAD SUMMATION : SIGMAR(X)=',E10.3,
     1'  SIGMAZ(Y)=',E10.3)
1     CONTINUE
      RETURN
      END
```

```
      SUBROUTINE TLOAD
C
C     ROUTINE TO CALCULATE NODAL LOADS DUE TO THERMAL LOADING FOR TWO
C     DIMENSIONAL AND AXISYMMETRIC ELEMENTS (CONSISTENT LOAD VECTOR)
C     LOAD VECTOR IS BASED ON THE UNIT LOAD VECTOR CALULATED IN
C     ELEMENT STIFFNESS ROUTINE FOR DT=1. ABOVE AMBIENT, FROM FILE
C     SET UP FOR 10 LOAD CASES, 500 ELEMENTS AND 500 NODES
C
      IMPLICIT REAL*8(A-H,O-Z)
      COMMON/CNTRL/IR,IW,IC,ITYP,MODEX,THIC,NE,NP,NLD,NTH,NMAT,NMAX,
     1    AMBT,TITLE(20),IPAGE
      COMMON/POP/PROP(10,10)
      COMMON CORD(500,2)
      DIMENSION NODE(4),TOLD(10),TMP(10),DUMY(10,5),RHS(10),ID(10)
      DIMENSION SI(100),ISP(100),SUM(10,2),UNIT(8),NAME(20)
C
      NCASE=NLD+NTH
      DO 30 I=1,NTH
      DO 30 J=1,2
30    SUM(I,J)=0.
C
C     READ OFF STORAGE LOCATION FOR THERMAL DATA (FROM MAXIT)
C
      IMX=0
      DO 4 I=1,NTH
      READ(IR,101)ID(I)
4     IMX=MAX0(IMX,ID(I))
101   FORMAT(I5,20A2)
C
      DO 5 I=1,NTH
5     BACKSPACE IR
C
      WRITE(IC,208)
208   FORMAT(5X,'START SETTING UP THERMAL RHS.')
C
C     LOOP OVER ALL ELEMENTS
C
      DO 3 II=1,NE
C     READ ELEMENT DEFINITION
      READ(1'II)(NODE(J),J=1,4)
C     READ TEMP. DATA GENERATED IN MAXIT
      READ(9'II)(TMP(J),J=1,IMX)
C     READ STRESS FILE TO STORE TEMP FOR STRESS RECOVERY
      READ(5'II)((DUMY(I,J),J=1,5),I=1,NCASE)
C
      NON=4
      IF(NODE(4).LE.0)NON=3
      NDF=NON*2
C
C     READ UNIT LOAD VECTOR BASED ON DT=1.
      READ(10'II)(UNIT(J),J=1,NDF)
C     CALCULATE THERMAL RHS, AS A RATIO OF UNIT THERMAL LOAD VECTOR
C
      DO 7 JJ=1,NTH
```

```
      LI=NLD+JJ
      L=ID(JJ)
      TOLD(JJ)=TMP(L)-AMBT
7     DUMY(LI,5)=TMP(L)
C
      IJ=0
      DO 6 I=1,NON
      NROW=(NODE(I)-1)*2
      DO 6 J=1,2
      NROW=NROW+1
      IJ=IJ+1
C     READ RHS DATA FILE FOR STORING THERMAL RHS.
      READ(3'NROW)(SI(K),K=1,NMAX),(ISP(K),K=1,NMAX),(RHS(K),K=1,
     1    NCASE)
C
C     LOOP OVER ALL THERMAL LOADING CASES
C
      DO 8 JJ=1,NTH
      LI=NLD+JJ
      RHH=TOLD(JJ)*UNIT(IJ)
      RHS(LI)=RHS(LI)+RHH
8     SUM(JJ,J)=SUM(JJ,J)+RHH
C     STORE RHS.
6     WRITE(3'NROW)(SI(K),K=1,NMAX),(ISP(K),K=1,NMAX),(RHS(K),K=1,
     1    NCASE)
C     STORE TEMP.
3     WRITE(5'II)((DUMY(I,J),J=1,5),I=1,NCASE)
C
C     WRITE OUT TEMP DATA FOR EACH LOAD CASE
C
      DO 100 II=1,NTH
      LI=NLD+II
      READ(IR,101)ID(10),NAME
      WRITE(IW,303)TITLE,IPAGE
303   FORMAT('1',///,10X,20A4,T95,'PAGE',I5)
      WRITE(IW,201)LI,ID(10),NAME
      IPAGE=IPAGE+1
201   FORMAT(//,10X,'LOAD CASE NO.= ',I3,3X,'THERMAL CASE (MAXIT RUN)'
     1    ,' NO.= ',I3,' :',20A2,//,10X,'ELEMENT TEMPERATURES :',/)
      LINE=11
C
C     LOOP OVER ALL ELEMENTS , READ TEMP. OFF FILE AND WRITE OUT
C
      DO 12 I=1,NE,8
      DO 13 J=1,8
      IJ=I+J-1
      J1=J-1
      IF(IJ.GT.NE)GO TO 15
      J1=J
      READ(9'IJ)(TMP(K),K=1,IMX)
      TOLD(J)=TMP(ID(10))
13    ID(J)=IJ
15    WRITE(IW,202)(ID(K),TOLD(K),K=1,J1)
      LINE=LINE+1
      IF(LINE.LE.55)GO TO 12
      LINE=11
      WRITE(IW,303)TITLE,IPAGE
      IPAGE=IPAGE+1
      WRITE(IW,201)LI,ID(10),NAME
202   FORMAT(5X,8(I3,F8.2,3X))
12    CONTINUE
C
      WRITE(IW,206)(SUM(II,J),J=1,2)
206   FORMAT(//,10X,'NODAL LOAD SUMMATION : SIGMAR(X)=',E10.3,
     1    '  SIGMAZ(Y)=',E10.3)
100   CONTINUE
      RETURN
      END
```

```
      SUBROUTINE BNDRY
C
C  ROUTINE TO INSERT BOUNDARY CONDITIONS INTO THE STIFFNESS MATRIX
C  HANDLES FIXITIES,SPRINGS AND SPECIFIED DISPLACEMENTS
C
      IMPLICIT REAL*8(A-H,O-Z)
      COMMON/CNTRL/IR,IW,IC,ITYP,MODEX,THIC,NE,NP,NLD,NTH,NMAT,NMAX,
     1AMBT,TITLE(20),IPAGE
      COMMON CORD(500,2)
      DIMENSION SI(100),ISP(100),NFIX(2),BFIX(2),RH(10)
      TERM=1.E20
C
C  FIXED BOUNDARIES-PUT LARGE TERM ON THE DIAG.
C
      IBD=1
42    READ(IR,201)NB,NFIX
201   FORMAT(3I5)
      IF(NB.LE.0)GO TO 50
      IF(IBD.GT.1)GO TO 69
      IBD=2
      WRITE(IW,206)TITLE,IPAGE
206   FORMAT('1',///,10X,20A4,T95,'PAGE',I5,//)
      IPAGE=IPAGE+1
      WRITE(IW,200)
200   FORMAT(10X,'BOUNDARY CONDITIONS 0=FREE,1=FIX',//,10X,'NODE',
     13X,'R',4X,'Z',/)
69    WRITE(IW,203)NB,NFIX
203   FORMAT(8X,3I5)
      DO 41 I=1,2
      IF(NFIX(I).LE.0)GO TO 41
      NROW=(NB-1)*2+I
      READ(3'NROW)(SI(J),J=1,NMAX),(ISP(J),J=1,NMAX),(RH(J),J=1,LCASE)
      SI(1)=TERM
      WRITE(3'NROW)(SI(J),J=1,NMAX),(ISP(J),J=1,NMAX),(RH(J),J=1,LCASE)
41    CONTINUE
      GO TO 42
50    CONTINUE
C
C  INSERT SPRING CONSTANTS ON THE DIAGONAL
C
      IBD=1
54    READ(IR,104)NB,BFIX
104   FORMAT(I5,2E10.3)
      IF(NB.LE.0)GO TO 60
      IF(IBD.GT.1)GO TO 58
      IBD=2
      WRITE(IW,206)TITLE,IPAGE
      IPAGE=IPAGE+1
      WRITE(IW,204)
204   FORMAT(10X,'VALUES OF SPRING CONSTANTS',/,10X,'NODE',7X,
     1'SPRING CONSTANT  (FORCE/LENGTH)',/,21X,'R',9X,'Z',/)
58    WRITE(IW,202)NB,BFIX
202   FORMAT(7X,I5,3X,2E10.3)
      DO 51 I=1,2
      IF(BFIX(I).LT.1.)GO TO 51
      NROW=(NB-1)*2+I
      READ(3'NROW)(SI(J),J=1,NMAX),(ISP(J),J=1,NMAX),(RH(J),J=1,LCASE)
      SI(1)=SI(1)+BFIX(I)
      WRITE(3'NROW)(SI(J),J=1,NMAX),(ISP(J),J=1,NMAX),(RH(J),J=1,LCASE)
51    CONTINUE
      GO TO 54
60    CONTINUE
C
C  INSERT SPECIFIED DISPLACEMENTS
C
      IBD=1
62    READ(IR,104)NB,BFIX
      IF(NB.LE.0)GO TO 67
      IF(IBD.GT.1)GO TO 68
      IBD=2
      WRITE(IW,206)TITLE,IPAGE
      IPAGE=IPAGE+1
      WRITE(IW,205)
205   FORMAT(10X,'SPECIFIED DISPLACEMENTS AT NODES',//,
     110X,'NODE',9X,'R',14X,'Z',/)
68    WRITE(IW,66)NB,BFIX
66    FORMAT(8X,I5,2(5X,E10.4))
      DO 61 K=1,2
      IF(BFIX(K).LE.0.)GO TO 61
      NROW=(NB-1)*2+K
      READ(3'NROW)(SI(J),J=1,NMAX),(ISP(J),J=1,NMAX),(RH(J),J=1,LCASE)
      DO 56 J=1,LCASE
56    RH(J)=TERM*BFIX(K)
      SI(1)=TERM
      WRITE(3'NROW)(SI(J),J=1,NMAX),(ISP(J),J=1,NMAX),(RH(J),J=1,LCASE)
61    CONTINUE
      GO TO 62
67    CONTINUE
C
C  FIX NODES ON THE AXIS , IN R,(X) DIRECTION FOR AXISYMMETRIC MODELS
C
      IF(ITYP.NE.3)GO TO 70
      DO 71 I=1,NP
      X=CORD(I,1)
      IF(X.GT.0.0001.OR.X.LT.-0.0001)GO TO 71
      NROW=(I-1)*2+1
      READ(3'NROW)(SI(J),J=1,NMAX),(ISP(J),J=1,NMAX),(RH(J),J=1,LCASE)
      SI(1)=TERM
      WRITE(3'NROW)(SI(J),J=1,NMAX),(ISP(J),J=1,NMAX),(RH(J),J=1,LCASE)
71    CONTINUE
70    CONTINUE
      RETURN
      END
```

```
      SUBROUTINE SPARS
C
      IMPLICIT REAL*8 (A-H,O-Z)
      COMMON/CNTRL/IR,IW,IC,ITYP,MODEX,THIC,NE,NP,NLD,NTH,NMAT,NMAX,
     1AMBT,TITLE(20),IPAGE
      DIMENSION SI1(100),SI2(100),ISP1(100),ISP2(100),IMET(1000),
     1DIS(10,2),RHS1(10),RHS2(10)
C
      NEQ=NP*2
      NCASE=NTH+NLD
C
C  SOLUTION BY SPARSE MATRIX METHOD
C  IMET SHOLUD BE DIMENSIONED TO MAX. DOF/EQN.
C  CAN HANDLE 500 NODES , 1000 DOF.
C  ROUTINE CAN HANDLE UPTO 10 RHS.
C
      WRITE(IW,200)IPAGE
200   FORMAT('1',///,T95,'PAGE',I5)
      IPAGE=IPAGE+1
      WRITE(IW,201)TITLE
201   FORMAT(20(/),5X,90('*'),2(/,5X,'*',88(' '),'*'),/,5X,'*',
     14X,20A4,4X,'*',2(/,5X,'*',88(' '),'*'),/,5X,'*',4X,
     2'T W O   D I M.  S T R E S S   A N A L Y S I S - S O L U T I O N
     3',T95,'*',3(/,5X,'*',88(' '),'*'),/,5X,90('*'),//)
C      SET UP EQUATION TABLE
      READ  (3'  1) (SI1(J),J=1,NMAX),(IMET(J),J=1,NMAX)
      DO 340 N=2,NEQ
      WRITE(IC,301)N
301   FORMAT('+',5X,'SETTING UP EQN TABLE FOR EQN. ',I5,'  ')
      READ(3'N)(SI2(J),J=1,NMAX),(ISP2(J),J=1,NMAX),(RHS2(J),J=1,NCASE)
      DO 280 M=1,NMAX
      IF((IMET(M)-N+1).EQ.0) GO TO 280
      DO 240 L=1,NMAX
      IF(ISP2(L).EQ.0) GO TO 260
      IF(ISP2(L).EQ.IMET(M)) GO TO 280
  240 CONTINUE
      WRITE(IW,100) N
  100 FORMAT(//'    OVERFLOW ON EQUATION TABLE  EQU. NO.',I5//)
      STOP 'ABORTED IN SPARSE'
  260 ISP2(L)=IMET(M)
  280 CONTINUE
      WRITE(3'N)(SI2(J),J=1,NMAX),(ISP2(J),J=1,NMAX),(RHS2(J),J=1,NCASE)
      DO 320 M=1,NMAX
  320 IMET(M)=ISP2(M)
  340 CONTINUE
C
C      START ELIMINATION  -  LOOP ON EQUATIONS
C
      DO 520 I=1,(NEQ-1)
      WRITE(IC,302)I
302   FORMAT('+',5X,'START ELIMINATION FROM EQN ',I5,'   ')
      READ(3'I)(SI1(J),J=1,NMAX),(ISP1(J),J=1,NMAX),(RHS1(J),J=1,NCASE)
C
C      MODIFY R.H.S. VECTOR
C
      DO 521 J=1,NCASE
521   RHS1(J)=RHS1(J)/SI1(1)
C
C      LOOP ON ROW TO BE ELIMINATED
C
      DO 460 M=2,NMAX
      IN=ISP1(M)
      IF(IN.EQ.0) GO TO 480
C
C      READ ROW FROM FILE
C
      READ(3'IN)(SI2(J),J=1,NMAX),(ISP2(J),J=1,NMAX),(RHS2(J),J=1,NCASE)
       DO 380 N=1,NMAX
      IA=ISP2(N)
      IF(IA.EQ.0) GO TO 400
C
C      SEEK APPROPRIATE ROWS
C
  380 IMET(IA)=N
  400 CONTINUE
      TEMP=SI1(M)/SI1(1)
C
C      LOOP ON COLUMN TO BE ELIMINATED
C
      DO 420 N=1,NMAX
      IA=ISP1(N)
      IF(IA.EQ.0) GO TO 440
      IF(IA.LT.IN) GO TO 420
      IM=IMET(IA)
C
C      MODIFY TERM IN MATRIX
C
      SI2(IM)=SI2(IM)-TEMP*SI1(N)
  420 CONTINUE
C
C      MODIFY LOAD VECTOR
C
440   CONTINUE
      DO 441 J=1,NCASE
441   RHS2(J)=RHS2(J)-RHS1(J)*SI1(M)
460   WRITE(3'IN)(SI2(J),J=1,NMAX),(ISP2(J),J=1,NMAX),(RHS2(J),J=1,NCASE
     1)
  480 CONTINUE
C
C      RESET ROW FOR BACK SUBSTITUTION
C
      DO 500 M=2,NMAX
  500 SI1(M)=SI1(M)/SI1(1)
      WRITE(3'I)(SI1(J),J=1,NMAX),(ISP1(J),J=1,NMAX),(RHS1(J),J=1,NCASE)
  520 CONTINUE
      READ(3'NEQ)(SI1(J),J=1,NMAX),(ISP1(J),J=1,NMAX),(RHS1(J),J=1,NCASE
     1)
      DO 522 J=1,NCASE
```

```
522   RHS1(J)=RHS1(J)/SI1(1)
      WRITE(3'NEQ)(SI1(J),J=1,NMAX),(ISP1(J),J=1,NMAX),(RHS1(J),J=1,NCAS
     1E)
C
C      BACK SUBSTITUTION
C
      DO 560 N=1,(NEQ-1)
      I=NEQ-N
      WRITE(IC,303)I
303   FORMAT('+',5X,'BACKSUBSTITUTING EQN ',I5,'      ')
      READ(3'I)(SI1(J),J=1,NMAX),(ISP1(J),J=1,NMAX),(RHS1(J),J=1,NCASE)
      DO 540 M=2,NMAX
      L=ISP1(M)
      IF(L.EQ.0) GO TO 560
      READ(3'L)(SI2(J),J=1,NMAX),(ISP2(J),J=1,NMAX),(RHS2(J),J=1,NCASE)
      DO 541 J=1,NCASE
541   RHS1(J)=RHS1(J)-SI1(M)*RHS2(J)
  540 CONTINUE
560   WRITE(3'I)(SI1(J),J=1,NMAX),(ISP1(J),J=1,NMAX),(RHS1(J),J=1,NCASE)
C
C      PUT RESULT ON FILE 4
C
      JJ=1
      DO 602 M=1,NEQ,2
      K=1
      DO 603 N=M,M+1
      READ(3'N)(SI1(J),J=1,NMAX),(ISP1(J),J=1,NMAX),(DIS(J,K),J=1,NCASE)
      K=K+1
603   CONTINUE
      WRITE(4'JJ)((DIS(J,K),K=1,2),J=1,NCASE)
      JJ=JJ+1
602   CONTINUE
      RETURN
      END
```

```
      SUBROUTINE STRESS
C
C     ROUTINE TO CALCULATE THE CENTROIDAL STRESSES FOR TWO DIMENSIONAL
C     AND AXISYMMETRIC ELEMENTS
C     [SIGMA]=[D][B][DIS]-[D][EO]+[SIGMAO]
C     FOR AXISYMMETRIC (4*1)=(4*4)(4*8)(8*1)-(4*4)(4*1)+(4*1)
C     FOR PLANE STRESS/STRAIN (3*1)=(3*3)(3*8)(8*1)-(3*3)(3*1)+(3*1)
C     IN THIS SUBROUTINE INITIAL STRESSES [SIGMAO] ARE NEGLECTED
C
      IMPLICIT REAL*8(A-H,O-Z)
      COMMON/CNTRL/IR,IW,IC,ITYP,MODEX,THIC,NE,NP,NLD,NTH,NMAT,NMAX,
     1    AMBT,TITLE(20),IPAGE
      COMMON /POP/PROP(10,10)
      COMMON CORD(500,2)
      DIMENSION DIS(1000),DMAT(10,4,4),D(4,4),DE(10,4),DEO(4),DUM(4,2)
     1    ,DUMY(10,5),NODE(4),B(12,12),RHS(12)
C
      ICMP=4
      IF(ITYP.NE.3)ICMP=3
      JK=ICMP-1
      NCASE=NLD+NTH
      NEQ=NP*2
C
C     CALCULATE THE [D] MATRIX(4*4) AND THE [DEO] MATRIX(4*1) FOR DT=1
C
      DO 11 IJ=1,NMAT
      CALL MATRXD(IJ,0.,D,DEO)
      DO 11 I=1,4
      DE(IJ,I)=DEO(I)
      DO 11 J=1,4
11    DMAT(IJ,I,J)=D(I,J)
C
C     LOOP OVER ALL LOAD CASES
C
      DO 100 III=1,NCASE
      WRITE(IC,207)III
207   FORMAT('+',5X,'CALCULATING STRESSES FOR LOAD CASE ',I5,'  ')
      WRITE(IW,201)TITLE,IPAGE,III
      IPAGE=IPAGE+1
      LINE=15
201   FORMAT('1',///,10X,20A4,T95,'PAGE',I5,/,10X,'L O A D   C A S E'
     1    ,'   N U M B E R  ',I5,/)
      WRITE(IW,202)
202   FORMAT(/,5X,'N O D E   D I S P L A C E M E N T S   (G L O B A L)
     1    ',//,3(3X,'NODE',17X,'R-',12X,'Z-'),/,3(' NUMBER',5X,2(3X,
     2    'TRANSLATION')),/)
C
C     INITIALIZE DISPLACEMENTS ARRAY, READ DISP. OFF FILE INTO CORE
C     AND PRINT OUT
C
      DO 4 JK=1,NEQ
4     DIS(JK)=0.
C
      DO 1 JK=1,NP,3
```

```
      DO 6 I=1,3
      DUM(I,1)=0.
      DUM(I,2)=0.
6     NODE(I)=0
C
      DO 2 KJ=1,3
      J1=KJ-1
      J=JK+J1
      IF(J.GT.NP)GO TO 5
      J1=KJ
      READ(4'J)((DUMY(JJ,KK),KK=1,2),JJ=1,NCASE)
      N=(J-1)*2
      DO 3 K=1,2
      N=N+1
      DIS(N)=DUMY(III,K)
3     DUM(KJ,K)=DUMY(III,K)
2     NODE(KJ)=J
5     WRITE(IW,203)(NODE(K),(DUM(K,JJ),JJ=1,2),K=1,J1)
      LINE=LINE+1
      IF(LINE.LT.55)GO TO 1
      WRITE(IW,201)TITLE,IPAGE,III
      IPAGE=IPAGE+1
      WRITE(IW,202)
      LINE=15
203   FORMAT(3(I7,5X,2E14.5))
1     CONTINUE
C
C
      WRITE(IW,201)TITLE,IPAGE,III
      IPAGE=IPAGE+1
      WRITE(IW,204)
204   FORMAT(/,10X,'E L E M E N T   S T R E S S E S   (G L O B A L)',
     1    //,3X,'ELEMENT',11X,'SIGMAR',11X,'SIGMAZ',7X,'SIGMATHETA',12X,
     2    'TAU-RZ',5X,'TEMP.',/)
      LINE=12
C
C     LOOP OVER ELEMENTS FOR STRESS RECOVERY
C
      MATO=0
      DO 99 JJJ=1,NE
      READ(5'JJJ)((DUMY(I,J),J=1,5),I=1,NCASE)
      READ(1'JJJ)(NODE(J),J=1,4),MAT,IEL
C
      IF(MAT.EQ.MATO)GO TO 18
      MATO=MAT
      DO 19 I=1,ICMP
      DEO(I)=DE(MAT,I)
      DO 19 J=1,ICMP
19    D(I,J)=DMAT(MAT,I,J)
18    CONTINUE
C
C     CALCULATE MAXTRIX [B] (4,8) OR (4,6)
C
      NON=4
      IF(NODE(4).GT.0)GO TO 7
      NON=3
      CALL CST(2,JJJ,NODE,D,DEO,B,RHS,VOL)
      GO TO 14
7     CALL ISOQ(IEL,2,2,JJJ,NON,NODE,D,DEO,B,RHS,VOL)
14    MN=NON*2
C
C     CALCULATE MATRIX [D][B] (4*8)
      DO 9 I=1,ICMP
      I1=I+4
      DO 9 J=1,MN
      RHS(J)=0.
      B(I1,J)=0.
      DO 9 K=1,ICMP
9     B(I1,J)=B(I1,J)+D(I,K)*B(K,J)
C
C     CALCULATE MATRIX [D][EO] (4*1)
C
      DT=DUMY(III,5)-AMBT
      DO 20 I=1,ICMP
20    DUM(I,1)=DT*DE(MAT,I)
C
      DO 8 I=1,NON
      K=(I-1)*2
      DO 8 J=1,2
      IJ=J+K
      L=(NODE(I)-1)*2+J
8     RHS(IJ)=DIS(L)
      IA=K+2
      DO 12 I=1,ICMP
      I1=I+4
      B(I,1)=0.
      DO 13 J=1,IA
13    B(I,1)=B(I,1)+B(I1,J)*RHS(J)
12    B(I,1)=B(I,1)-DUM(I,1)
C
      DO 21 I=1,ICMP
21    DUMY(III,I)=B(I,1)
C
C     INTERCHANGE HOOP STRESS AND SHEAR STRESS IF NESASSARY
C     AND STORE STRESSES ON FILE DL1:AXISTRESS.DAT
      IF(ITYP.EQ.3)GO TO 22
      DUMY(III,4)=DUMY(III,3)
      DUMY(III,3)=0.
22    WRITE(5'JJJ)((DUMY(I,J),J=1,5),I=1,NCASE)
C
C     WRITE OUT STRESS COMPONENTS
C
      LINE=LINE+1
      IF(LINE.LE.55)GO TO 10
      WRITE(IW,201)TITLE,IPAGE,III
      WRITE(IW,204)
      LINE=12
      IPAGE=IPAGE+1
```

```
10    WRITE(IW,204)JJJ,(DUMY(III,J),J=1,5)
204   FORMAT(I10,4F17.3,F10.2)
99    CONTINUE
100   CONTINUE
      RETURN
      END
```

TYPICAL OUTPUTS FROM

PROGRAM VERIFICATION

For details of the models used for verification refer to section 3 of the main body.

PAGE 1

REDUCIBLE NET, PAGE 164 AND 453, DESAI AND ABEL

TWO DIMENSIONAL STRESS ANALYSIS - PLANE STRESS ANALYSIS

PREPARED BY:....................CHECKED BY:....................DATE RUN:11-JUN-79

I N P U T   D A T A :

NO. OF NODAL POINTS	9
NO. OF ELEMENTS	4
NO. OF MATL. TYPES	1
NO. OF TH. LOAD CASES	0
NO. OF MECH. LOAD CASES	1
AMBIENT TEMP.	0.000
MATL. THICKNESS	1.000

MATERIAL PROPERTIES

MATL. TYPE	E-MU	MODULUS OF ELASTICITY	POISSONS RATIO	COEFF. OF TH. EXP.	SHEAR MOD.
1	R-RZ	0.300E+08	0.30	0.700E-05	0.115E+08
1	Z-RT	0.300E+08	0.30	0.700E-05	
1	T-ZT	0.300E+08	0.30	0.700E-05	

REDUCIBLE NET, PAGE 164 AND 458, DESAI AND ABEL

PAGE 2

E L E M E N T   D A T A

ELM NO.	NODE NUMBERS				TYPE OF MAT	TYPE OF ELM	ELM NO.	NODE NUMBERS				TYPE OF MAT	TYPE OF ELM
1	1	2	5	4	1	06	2	2	3	6	5	1	06
3	4	5	8	7	1	06	4	5	6	9	8	1	06

REDUCIBLE NET, PAGE 166 AND 453, DESAI AND ABEL PAGE 3

NODAL COORDINATE DATA

NODE NO.	R(X)	Z(Y)	NODE NO.	R(X)	Z(Y)	NODE NO.	R(X)	Z(Y)
1	0.000	0.000	2	2.250	0.000	3	4.500	0.000
4	0.000	1.500	5	2.250	1.500	6	4.500	1.500
7	0.000	3.000	8	2.250	3.000	9	4.500	3.000

VOLUME OF THE MODEL= 13.5

REDUCIBLE NET, PAGE 166 AND 453, DESAI AND ABEL PAGE 4

LOAD CASE= 1 BENDING STRESS OF 1000. PSI.

NODAL LOADS

NODE NO.	RADIAL(X)	AXIAL(Y)
6	750.000	0.000
9	625.000	0.000

NODAL LOAD SUMMATION : SIGMAR(X)= 0.130E+04 SIGMAZ(Y)= 0.000E+00

REDUCIBLE NET, PAGE 166 AND 453, DESAI AND ABEL PAGE 5

BOUNDARY CONDITIONS 0=FREE,1=FIX

NODE	R	Z
1	1	1
2	1	0
3	1	0
4	1	0
7	1	0

```
****************************************************************
*                                                              *
*  REDUCIBLE NET, PAGE 166 AND 458, DESAI AND ABEL             *
*                                                              *
*  T W O   D I M.  S T R E S S   A N A L Y S I S - S O L U T I O N
*                                                              *
****************************************************************
```

REDUCIBLE NET, PAGE 166 AND 458, DESAI AND ABEL — PAGE 7

L O A D   C A S E   N U M B E R   1

N O D E   D I S P L A C E M E N T S   (G L O B A L)

NODE NUMBER	R- TRANSLATION	Z- TRANSLATION	NODE NUMBER	R- TRANSLATION	Z- TRANSLATION	NODE NUMBER	R- TRANSLATION	Z- TRANSLATION
1	0.12500E-17	-0.40926E-33	2	-0.81749E-26	-0.28125E-04	3	-0.12500E-17	-0.11250E-03
4	0.75000E-17	-0.37500E-05	5	0.37500E-04	-0.31875E-04	6	0.75000E-04	-0.11625E-03
7	0.62500E-17	-0.15000E-04	8	0.75000E-04	-0.43125E-04	9	0.15000E-03	-0.12750E-03

REDUCIBLE NET, PAGE 166 AND 458, DESAI AND ABEL — PAGE 8

L O A D   C A S E   N U M B E R   1

E L E M E N T   S T R E S S E S   (G L O B A L)

ELEMENT	SIGMAR	SIGMAZ	SIGMATHETA	TAU-RZ	TEMP.
1	250.000	0.000	0.000	-0.000	0.00
2	250.000	-0.000	0.000	-0.000	0.00
3	750.000	0.000	0.000	0.000	0.00
4	750.000	-0.000	0.000	0.000	0.00

PAGE 1

```
*****************************************************************************************
*                                                                                       *
*    PIPE QUARTER MODEL UNDER PRESSURE AND THERMAL LOADING (USING Q5)                   *
*                                                                                       *
*    TWO DIMENSIONAL STRESS ANALYSIS      -      PLANE STRAIN ANALYSIS                  *
*                                                                                       *
*****************************************************************************************
*                                                                                       *
*                                                                                       *
*  PREPARED BY:....................CHECKED BY:.....................DATE RUN:13-JUN-79   *
*                                                                                       *
*                                                                                       *
*****************************************************************************************
```

I N P U T   D A T A :

NO. OF NODAL POINTS	36
NO. OF ELEMENTS	24
NO. OF MATL. TYPES	1
NO. OF TH. LOAD CASES	1
NO. OF MECH. LOAD CASES	1
AMBIENT TEMP.	100.000

MATERIAL PROPERTIES

MATL. TYPE	E-MU	MODULUSOF ELASTICITY	POISSONS RATIO	COEFF. OF TH. EXP.	SHEAR MOD.
1	R-RZ	0.300E+08	0.30	0.700E-05	0.115E+08
1	Z-RT	0.300E+08	0.30	0.700E-05	
1	T-ZT	0.300E+08	0.30	0.700E-05	

PIPE QUARTER MODEL UNDER PRESSURE AND THERMAL LOADING (USING Q5) PAGE 2

E L E M E N T   D A T A

ELM NO.	NODE NUMBERS				TYPE OF MAT	TYPE OF ELM	ELM NO.	NODE NUMBERS				TYPE OF MAT	TYPE OF ELM
1	1	2	6	5	1	Q5	2	2	3	7	6	1	Q5
3	3	4	8	7	1	Q5	4	5	6	10	9	1	Q5
5	6	7	11	10	1	Q5	6	7	8	12	11	1	Q5
7	9	10	14	13	1	Q5	8	10	11	15	14	1	Q5
9	11	12	16	15	1	Q5	10	13	14	18	17	1	Q5
11	14	15	19	18	1	Q5	12	15	16	20	19	1	Q5
13	17	18	22	21	1	Q5	14	18	19	23	22	1	Q5
15	19	20	24	23	1	Q5	16	21	22	26	25	1	Q5
17	22	23	27	26	1	Q5	18	23	24	28	27	1	Q5
19	25	26	30	29	1	Q5	20	26	27	31	30	1	Q5
21	27	28	32	31	1	Q5	22	29	30	34	33	1	Q5
23	30	31	35	34	1	Q5	24	31	32	36	35	1	Q5

PIPE QUARTER MODEL UNDER PRESSURE AND THERMAL LOADING (USING Q5) PAGE 3

N O D A L   C O O R D I N A T E   D A T A

NODE NO.	R(X)	Z(Y)	NODE NO.	R(X)	Z(Y)	NODE NO.	R(X)	Z(Y)
1	10.000	0.000	2	11.000	0.000	3	12.000	0.000
4	13.000	0.000	5	9.808	1.950	6	10.789	2.146
7	11.769	2.341	8	12.750	2.536	9	9.239	3.827
10	10.163	4.210	11	11.087	4.592	12	12.010	4.975
13	8.315	5.556	14	9.146	6.111	15	9.978	6.667
16	10.809	7.222	17	7.071	7.071	18	7.778	7.778
19	8.485	8.485	20	9.192	9.192	21	5.556	8.315
22	6.111	9.146	23	6.667	9.978	24	7.222	10.809
25	3.827	9.239	26	4.210	10.163	27	4.592	11.087
28	4.975	12.010	29	1.950	9.808	30	2.146	10.789
31	2.341	11.769	32	2.536	12.750	33	0.000	10.000
34	0.000	11.000	35	0.000	12.000	36	0.000	13.000

VOLUME OF THE MODEL= 53.8

PIPE QUARTER MODEL UNDER PRESSURE AND THERMAL LOADING (USING Q5) PAGE 4

LOAD CASE= 1 1000 PSI. INTERNAL PRESS.

PRESSURE LOADING

NODE NO. FROM	TO	PRESS.
1	5	1000.00
5	9	1000.00
9	13	1000.00
13	17	1000.00
17	21	1000.00
21	25	1000.00
25	29	1000.00
29	33	1000.00

NODAL LOAD SUMMATION : SIGMAR(X)= 0.100E+05 SIGMAZ(Y)= 0.100E+05

PIPE QUARTER MODEL UNDER PRESSURE AND THERMAL LOADING (USING Q5) PAGE 5

LOAD CASE NO.= 2 THERMAL CASE (MAXIT RUN) NO.= 1 !LOG GRADIENT T(I)=500. DEG F

ELEMENT TEMPERATURES !

1	407.02	2	233.65	3	74.75	4	407.02	5	233.65	6	74.75	7	407.02	8	233.65
9	74.75	10	407.02	11	233.65	12	74.75	13	407.02	14	233.65	15	74.75	16	407.02
17	233.65	18	74.75	19	407.02	20	233.65	21	74.75	22	407.02	23	233.65	24	74.75

NODAL LOAD SUMMATION ! SIGMAR(X)= 0.152E-10 SIGMAZ(Y)= 0.455E-12

PIPE QUARTER MODEL UNDER PRESSURE AND THERMAL LOADING (USING Q5) PAGE 6

BOUNDARY CONDITIONS 0=FREE,1=FIX

NODE	R	Z
1	0	1
2	0	1
3	0	1
4	0	1
33	1	0
34	1	0
35	1	0
36	1	0

```
******************************************************************************************
*                                                                                        *
*   PIPE QUARTER MODEL UNDER PRESSURE AND THERMAL LOADING (USING Q5)                     *
*                                                                                        *
*   T W O   D I M.   S T R E S S   A N A L Y S I S - S O L U T I O N                     *
*                                                                                        *
******************************************************************************************
```

PIPE QUARTER MODEL UNDER PRESSURE AND THERMAL LOADING (USING 05) PAGE 8
L O A D   C A S E   N U M B E R   1

N O D E   D I S P L A C E M E N T S   (G L O B A L)

NODE NUMBER	R- TRANSLATION	Z- TRANSLATION	NODE NUMBER	R- TRANSLATION	Z- TRANSLATION	NODE NUMBER	R- TRANSLATION	Z- TRANSLATION
1	0.13079E-02	0.19457E-16	2	0.12370E-02	0.34763E-16	3	0.11821E-02	0.31508E-16
4	0.11394E-02	0.14272E-16	5	0.12828E-02	0.25506E-03	6	0.12132E-02	0.24131E-03
7	0.11594E-02	0.23062E-03	8	0.11176E-02	0.22229E-03	9	0.12084E-02	0.50049E-03
10	0.11428E-02	0.47336E-03	11	0.10921E-02	0.45235E-03	12	0.10527E-02	0.43607E-03
13	0.10875E-02	0.72660E-03	14	0.10285E-02	0.68720E-03	15	0.98281E-03	0.65669E-03
16	0.94736E-03	0.63302E-03	17	0.92479E-03	0.92479E-03	18	0.87464E-03	0.87464E-03
19	0.83580E-03	0.83580E-03	20	0.80566E-03	0.80566E-03	21	0.72660E-03	0.10875E-02
22	0.68720E-03	0.10285E-02	23	0.65669E-03	0.98281E-03	24	0.63302E-03	0.94736E-03
25	0.50049E-03	0.12084E-02	26	0.47336E-03	0.11428E-02	27	0.45235E-03	0.10921E-02
28	0.43607E-03	0.10527E-02	29	0.25506E-03	0.12828E-02	30	0.24131E-03	0.12132E-02
31	0.23062E-03	0.11594E-02	32	0.22229E-03	0.11176E-02	33	0.19457E-16	0.13079E-02
34	0.34763E-16	0.12370E-02	35	0.31508E-16	0.11821E-02	36	0.14272E-16	0.11394E-02

PIPE QUARTER MODEL UNDER PRESSURE AND THERMAL LOADING (USING 05) PAGE 9
L O A D   C A S E   N U M B E R   1

E L E M E N T   S T R E S S E S   (G L O B A L)

ELEMENT	SIGMAR	SIGMAZ	SIGMATHETA	TAU-RZ	TEMP.
1	-724.550	3623.747	0.000	-432.427	100.00
2	-362.606	3261.284	0.000	-360.347	100.00
3	-84.163	2982.441	0.000	-305.000	100.00
4	-393.414	3292.624	0.000	-1231.354	100.00
5	-86.741	2985.292	0.000	-1026.389	100.00
6	149.213	2749.130	0.000	-868.525	100.00
7	218.065	2681.159	0.000	-1843.133	100.00
8	422.938	2475.600	0.000	-1536.028	100.00
9	580.516	2317.468	0.000	-1299.736	100.00
10	1017.243	1882.204	0.000	-2174.162	100.00
11	1088.876	1809.647	0.000	-1811.856	100.00
12	1144.000	1753.922	0.000	-1533.119	100.00
13	1882.204	1017.243	0.000	-2174.162	100.00
14	1809.647	1088.876	0.000	-1811.856	100.00
15	1753.922	1144.000	0.000	-1533.119	100.00
16	2681.159	218.065	0.000	-1843.133	100.00
17	2475.600	422.938	0.000	-1536.028	100.00
18	2317.468	580.516	0.000	-1299.736	100.00
19	3292.624	-393.414	0.000	-1231.354	100.00
20	2985.292	-86.741	0.000	-1026.389	100.00
21	2749.130	149.213	0.000	-868.525	100.00
22	3623.747	-724.550	0.000	-432.427	100.00
23	3261.284	-362.606	0.000	-360.347	100.00
24	2982.441	-84.163	0.000	-305.000	100.00

PIPE QUARTER MODEL UNDER PRESSURE AND THERMAL LOADING (USING Q5) PAGE 10
L O A D C A S E N U M B E R 2

N O D E D I S P L A C E M E N T S (G L O B A L)

NODE NUMBER	R-TRANSLATION	Z-TRANSLATION	NODE NUMBER	R-TRANSLATION	Z-TRANSLATION	NODE NUMBER	R-TRANSLATION	Z-TRANSLATION
1	0.90207E-02	0.53305E-15	2	0.11621E-01	0.64057E-15	3	0.12427E-01	0.35036E-15
4	0.11726E-01	0.13743E-15	5	0.88472E-02	0.17590E-02	6	0.11398E-01	0.22672E-02
7	0.12189E-01	0.24245E-02	8	0.11501E-01	0.22874E-02	9	0.83338E-02	0.34521E-02
10	0.10736E-01	0.44472E-02	11	0.11481E-01	0.47557E-02	12	0.10833E-01	0.44874E-02
13	0.75003E-02	0.50117E-02	14	0.96625E-02	0.64564E-02	15	0.10333E-01	0.69043E-02
16	0.97497E-02	0.65146E-02	17	0.63786E-02	0.63786E-02	18	0.82173E-02	0.82173E-02
19	0.87875E-02	0.87875E-02	20	0.82915E-02	0.82915E-02	21	0.50117E-02	0.75003E-02
22	0.64564E-02	0.96625E-02	23	0.69043E-02	0.10333E-01	24	0.65146E-02	0.97497E-02
25	0.34521E-02	0.83338E-02	26	0.44472E-02	0.10736E-01	27	0.47557E-02	0.11481E-01
28	0.44874E-02	0.10833E-01	29	0.17590E-02	0.88472E-02	30	0.22672E-02	0.11398E-01
31	0.24245E-02	0.12189E-01	32	0.22874E-02	0.11501E-01	33	0.53305E-15	0.90207E-02
34	0.64057E-15	0.11621E-01	35	0.35036E-15	0.12427E-01	36	0.13743E-15	0.11726E-01

PIPE QUARTER MODEL UNDER PRESSURE AND THERMAL LOADING (USING Q5) PAGE 11
L O A D C A S E N U M B E R 2

E L E M E N T S T R E S S E S (G L O B A L)

ELEMENT	SIGMAR	SIGMAZ	SIGMATHETA	TAU-RZ	TEMP.
1	-2318.836	-38928.158	0.000	3639.668	407.02
2	-3263.306	2154.298	0.000	-536.578	233.65
3	-1035.580	36706.260	0.000	-3755.100	74.75
4	-5107.386	-36139.383	0.000	10367.454	407.02
5	-2848.015	1741.161	0.000	-1536.524	233.65
6	1835.799	33835.567	0.000	-10688.341	74.75
7	-10254.827	-30993.024	0.000	15517.697	407.02
8	-2088.574	979.329	0.000	-2295.844	233.65
9	7145.233	28525.143	0.000	-15997.796	74.75
10	-16983.228	-24265.785	0.000	18304.447	407.02
11	-1093.310	-16.315	0.000	-2707.644	233.65
12	14080.782	21588.830	0.000	-18870.762	74.75
13	-24265.785	-16983.228	0.000	18304.447	407.02
14	-16.315	-1093.310	0.000	-2707.644	233.65
15	21588.830	14080.782	0.000	-18870.762	74.75
16	-30993.024	-10254.827	0.000	15517.697	407.02
17	979.329	-2088.574	0.000	-2295.844	233.65
18	28525.143	7145.233	0.000	-15997.796	74.75
19	-36139.383	-5107.386	0.000	10367.454	407.02
20	1741.161	-2848.015	0.000	-1536.524	233.65
21	33835.567	1835.799	0.000	-10688.341	74.75
22	-38928.158	-2318.836	0.000	3639.668	407.02
23	2154.298	-3263.306	0.000	-536.578	233.65
24	36706.260	-1035.580	0.000	-3755.100	74.75

```
*********************************************************************************
*                                                                               *
*   SPHERICAL VESSEL UNDER PRESSURE AND THERMAL LOADING (USING Q4)              *
*                                                                               *
*   TWO DIMENSIONAL STRESS ANALYSIS     -     AXISYMMETRIC ANALYSIS             *
*                                                                               *
*********************************************************************************
*                                                                               *
*                                                                               *
* PREPARED BY:....................CHECKED BY:....................DATE RUN:15-JUN-79 *
*                                                                               *
*                                                                               *
*********************************************************************************
```

I N P U T   D A T A :

NO. OF NODAL POINTS	36
NO. OF ELEMENTS	24
NO. OF MATL. TYPES	1
NO. OF TH. LOAD CASES	1
NO. OF MECH. LOAD CASES	1
AMBIENT TEMP.	100.000

MATERIAL PROPERTIES

MATL. TYPE	E-MU	MODULUSOF ELASTICITY	POISSONS RATIO	COEFF. OF TH. EXP.	SHEAR MOD.
1	R-RZ	0.300E+08	0.30	0.700E-05	0.115E+08
1	Z-RT	0.300E+08	0.30	0.700E-05	
1	T-ZT	0.300E+03	0.30	0.700E-05	

SPHERICAL VESSEL UNDER PRESSURE AND THERMAL LOADING (USING Q4)

E L E M E N T   D A T A

ELM NO.	NODE NUMBERS				TYPE OF MAT	TYPE OF ELM
1	1	2	6	5	1	Q4
3	3	4	8	7	1	Q4
5	6	7	11	10	1	Q4
7	9	10	14	13	1	Q4
9	11	12	16	15	1	Q4
11	14	15	19	18	1	Q4
13	17	18	22	21	1	Q4
15	19	20	24	23	1	Q4
17	22	23	27	26	1	Q4
19	25	26	30	29	1	Q4
21	27	28	32	31	1	Q4
23	30	31	35	34	1	Q4

ELM NO.	NODE NUMBERS				TYPE OF MAT	TYPE OF ELM
2	2	3	7	6	1	Q4
4	5	6	10	9	1	Q4
6	7	8	12	11	1	Q4
8	10	11	15	14	1	Q4
10	13	14	18	17	1	Q4
12	15	16	20	19	1	Q4
14	18	19	23	22	1	Q4
16	21	22	26	25	1	Q4
18	23	24	28	27	1	Q4
20	26	27	31	30	1	Q4
22	29	30	34	33	1	Q4
24	31	32	36	35	1	Q4

SPHERICAL VESSEL UNDER PRESSURE AND THERMAL LOADING (USING Q4) PAGE 3

N O D A L   C O O R D I N A T E   D A T A

NODE NO.	R(X)	Z(Y)	NODE NO.	R(X)	Z(Y)	NODE NO.	R(X)	Z(Y)
1	10.000	0.000	2	11.000	0.000	3	12.000	0.000
4	13.000	0.000	5	9.808	1.950	6	10.789	2.146
7	11.769	2.341	8	12.750	2.536	9	9.239	3.827
10	10.163	4.210	11	11.087	4.592	12	12.010	4.975
13	8.315	5.556	14	9.146	6.111	15	9.978	6.667
16	10.809	7.222	17	7.071	7.071	18	7.778	7.778
19	8.485	8.485	20	9.192	9.192	21	5.556	8.315
22	6.111	9.146	23	6.667	9.978	24	7.222	10.809
25	3.827	9.239	26	4.210	10.163	27	4.592	11.087
28	4.975	12.010	29	1.950	9.808	30	2.146	10.789
31	2.341	11.769	32	2.536	12.750	33	0.000	10.000
34	0.000	11.000	35	0.000	12.000	36	0.000	13.000

VOLUME OF THE MODEL= 2482.8

SPHERICAL VESSEL UNDER PRESSURE AND THERMAL LOADING (USING Q4) PAGE 4

LOAD CASE= 1 1000 PSI. INTERNAL PRESS.

PRESSURE LOADING

NODE NO. FROM	TO	PRESS.
1	5	1000.00
5	9	1000.00
9	13	1000.00
13	17	1000.00
17	21	1000.00
21	25	1000.00
25	29	1000.00
29	33	1000.00

NODAL LOAD SUMMATION : SIGMAR(X)= 0.490E+06 SIGMAZ(Y)= 0.314E+06

```
SPHERICAL VESSEL UNDER PRESSURE AND THERMAL LOADING (USING Q4)                              PAGE    5

LOAD CASE NO.=   2   THERMAL CASE (MAXIT RUN) NO.=   1 ;LOG GRADIENT T(I)=500. DEG F
ELEMENT TEMPERATURES :

 1  407.02     2  233.65     3   74.75     4  407.02     5  233.65     6   74.75     7  407.02     8  233.65
 9   74.75    10  407.02    11  233.65    12   74.75    13  407.02    14  233.65    15   74.75    16  407.02
17  233.65    18   74.75    19  407.02    20  233.65    21   74.75    22  407.02    23  233.65    24   74.75

NODAL LOAD SUMMATION : SIGMAR(X)= 0.229E+08  SIGMAZ(Y)=-0.575E-09




SPHERICAL VESSEL UNDER PRESSURE AND THERMAL LOADING (USING Q4)                              PAGE    6

BOUNDARY CONDITIONS 0=FREE,1=FIX
NODE   R    Z

  1    0    1
  2    0    1
  3    0    1
  4    0    1
 33    1    0
 34    1    0
 35    1    0
 36    1    0


*************************************************************************************
*                                                                                   *
*   SPHERICAL VESSEL UNDER PRESSURE AND THERMAL LOADING (USING Q4)                  *
*                                                                                   *
*   T W O   D I M.  S T R E S S   A N A L Y S I S - S O L U T I O N                 *
*                                                                                   *
*************************************************************************************
```

SPHERICAL VESSEL UNDER PRESSURE AND THERMAL LOADING (USING Q4) PAGE 8
LOAD CASE NUMBER 1

NODE DISPLACEMENTS (GLOBAL)

NODE NUMBER	R-TRANSLATION	Z-TRANSLATION	NODE NUMBER	R-TRANSLATION	Z-TRANSLATION	NODE NUMBER	R-TRANSLATION	Z-TRANSLATION
1	0.49962E-03	0.58651E-15	2	0.44270E-03	0.10494E-14	3	0.40202E-03	0.10265E-14
4	0.37279E-03	0.47920E-15	5	0.48983E-03	0.98622E-04	6	0.43394E-03	0.87401E-04
7	0.39407E-03	0.79683E-04	8	0.36541E-03	0.74016E-04	9	0.46085E-03	0.19382E-03
10	0.40820E-03	0.17197E-03	11	0.37066E-03	0.15628E-03	12	0.34372E-03	0.14506E-03
13	0.41397E-03	0.28236E-03	14	0.36674E-03	0.25025E-03	15	0.33320E-03	0.22715E-03
16	0.30923E-03	0.21053E-03	17	0.35114E-03	0.36178E-03	18	0.31160E-03	0.32029E-03
19	0.28378E-03	0.29041E-03	20	0.26411E-03	0.26879E-03	21	0.27478E-03	0.43032E-03
22	0.24521E-03	0.38097E-03	23	0.22470E-03	0.34533E-03	24	0.21076E-03	0.31931E-03
25	0.18772E-03	0.48720E-03	26	0.16975E-03	0.43203E-03	27	0.15801E-03	0.39228E-03
28	0.15098E-03	0.36287E-03	29	0.92822E-04	0.53042E-03	30	0.89198E-04	0.47442E-03
31	0.84263E-04	0.43275E-03	32	0.83945E-04	0.40078E-03	33	-0.18184E-17	0.59265E-03
34	0.11490E-16	0.51639E-03	35	0.75610E-17	0.46541E-03	36	0.64435E-17	0.43043E-03

SPHERICAL VESSEL UNDER PRESSURE AND THERMAL LOADING (USING Q4) PAGE 9
LOAD CASE NUMBER 1

ELEMENT STRESSES (GLOBAL)

ELEMENT	SIGMAR	SIGMAZ	SIGMATHETA	TAU-RZ	TEMP.
1	-713.257	1604.519	1613.214	-230.731	100.00
2	-344.745	1419.788	1424.007	-175.985	100.00
3	-84.758	1289.868	1291.047	-136.994	100.00
4	-535.911	1431.207	1613.258	-657.946	100.00
5	-209.286	1285.819	1423.372	-501.320	100.00
6	20.336	1184.053	1289.903	-389.745	100.00
7	-206.897	1110.670	1613.764	-987.648	100.00
8	41.133	1038.716	1422.801	-750.556	100.00
9	214.047	988.958	1288.447	-582.186	100.00
10	226.172	690.516	1615.242	-1170.268	100.00
11	368.628	716.828	1423.765	-886.287	100.00
12	466.015	735.896	1288.959	-685.422	100.00
13	699.101	234.325	1618.138	-1177.529	100.00
14	722.718	369.779	1428.201	-889.915	100.00
15	738.585	464.532	1295.485	-685.666	100.00
16	1134.554	-192.407	1618.642	-1010.205	100.00
17	1053.330	53.333	1440.909	-763.589	100.00
18	993.446	216.997	1315.271	-587.028	100.00
19	1468.632	-493.182	1626.644	-679.214	100.00
20	1291.596	-188.568	1462.440	-547.138	100.00
21	1208.271	24.772	1361.039	-416.789	100.00
22	1437.970	-1041.903	1451.970	-436.931	100.00
23	1431.528	-478.356	1445.711	-314.725	100.00
24	1418.051	-113.519	1426.066	-224.718	100.00

SPHERICAL VESSEL UNDER PRESSURE AND THERMAL LOADING (USING Q4) PAGE 10
L O A D   C A S E   N U M B E R   2

N O D E   D I S P L A C E M E N T S   (G L O B A L)

NODE NUMBER	R- TRANSLATION	Z- TRANSLATION	NODE NUMBER	R- TRANSLATION	Z- TRANSLATION	NODE NUMBER	R- TRANSLATION	Z- TRANSLATION
1	0.83547E-02	0.43226E-13	2	0.11406E-01	0.55664E-13	3	0.12057E-01	0.32177E-13
4	0.10857E-01	0.12751E-13	5	0.81936E-02	0.16306E-02	6	0.11187E-01	0.22268E-02
7	0.11825E-01	0.23536E-02	8	0.10649E-01	0.21194E-02	9	0.77171E-02	0.32011E-02
10	0.10536E-01	0.43682E-02	11	0.11138E-01	0.46166E-02	12	0.10030E-01	0.41576E-02
13	0.69438E-02	0.46485E-02	14	0.94812E-02	0.63424E-02	15	0.10023E-01	0.67028E-02
16	0.90268E-02	0.60357E-02	17	0.59035E-02	0.59197E-02	18	0.80624E-02	0.80747E-02
19	0.85250E-02	0.85328E-02	20	0.76786E-02	0.76831E-02	21	0.46358E-02	0.69672E-02
22	0.63346E-02	0.95006E-02	23	0.67007E-02	0.10039E-01	24	0.60379E-02	0.90389E-02
25	0.31886E-02	0.77535E-02	26	0.43636E-02	0.10568E-01	27	0.46196E-02	0.11165E-01
28	0.41659E-02	0.10054E-01	29	0.16204E-02	0.82500E-02	30	0.22194E-02	0.11241E-01
31	0.23546E-02	0.11873E-01	32	0.21343E-02	0.10689E-01	33	0.62407E-15	0.83779E-02
34	-0.46073E-15	0.11466E-01	35	-0.48332E-15	0.12162E-01	36	0.10541E-15	0.10971E-01

SPHERICAL VESSEL UNDER PRESSURE AND THERMAL LOADING (USING Q4) PAGE 11
L O A D   C A S E   N U M B E R   2

E L E M E N T   S T R E S S E S   (G L O B A L)

ELEMENT	SIGMAR	SIGMAZ	SIGMATHETA	TAU-RZ	TEMP.
1	-5856.287	-53594.880	-54080.717	4744.387	407.02
2	-8462.775	-96.738	-30.983	-831.393	233.65
3	-2966.698	44934.862	45391.055	-4766.758	74.75
4	-9488.388	-49955.141	-54080.718	13513.233	407.02
5	-7818.119	-736.140	-31.077	-2378.596	233.65
6	679.286	41288.860	45389.868	-13567.600	74.75
7	-16192.622	-43239.583	-54081.106	20223.383	407.02
8	-6636.762	-1914.896	-33.294	-3554.140	233.65
9	7420.167	34545.295	45387.406	-20305.366	74.75
10	-24951.697	-34462.264	-54080.019	23849.866	407.02
11	-5089.829	-3453.979	-32.029	-4190.672	233.65
12	16223.876	25739.321	45388.501	-23949.223	74.75
13	-34425.954	-24964.723	-54078.606	23843.801	407.02
14	-3414.480	-5114.239	-24.717	-4189.226	233.65
15	25754.295	16213.087	45399.310	-23948.234	74.75
16	-43170.568	-16193.029	-54080.553	20208.385	407.02
17	-1870.529	-6655.918	-14.429	-3556.815	233.65
18	34564.851	7416.847	45426.671	-20312.466	74.75
19	-49918.672	-9492.346	-54127.440	13448.134	407.02
20	-643.861	-7863.291	-12.324	-2417.786	233.65
21	41358.835	640.348	45473.626	-13559.887	74.75
22	-53531.322	-5310.483	-54002.649	4862.337	407.02
23	205.918	-7662.770	278.222	-919.465	233.65
24	45221.648	-2686.150	45676.570	-4994.355	74.75

SUBROUTINE CSTQUAD

```
      SUBROUTINE CSTOD(N,NON,NODE,D,DE,STIFF,RHS,VOL)
C
C  ROUTINE TO CALCULATE THE ELEMENT STIFFNESS BASED ON 4 CST'S WITH A
C  COMMON CENTRAL NODE.
C
C  OUTPUT FROM THE ROUTINE CONSISTS OF CONDENSED STIFFNESS MATRIX(8,8)
C  ,UNIT THERMAL LOAD VECTOR(8) AND THE ELM. VOLUME.
C
C  CAN HANDLE SINGLE TRIANGULAR ELEMENTS ALSO.
C
      IMPLICIT REAL*8(A-H,O-Z)
      COMMON /CNTRL/IR,IW,IC,ITYP,MODEX,THIC,NELM,NC,NMAX,NCASE,TAMB,
     1     TITLE(20)
      COMMON /POP/PROP(8,10)
      COMMON CORD(501,2)
      DIMENSION STIFF(10,10),RHS(10),NODE(4),DE(4),D(4,4),NOD(4),ND(3),
     1     B(4,6),DB(4,6),ESTIFF(6,6),RH(6)
C
      PHI2=8.*ATAN(1.)
C
C  INITIALIZE ARRAYS
C
      DO 1 I=1,10
      RHS(I)=0.
      DO 1 J=1,10
1     STIFF(I,J)=0.
C
      DO 14 I=1,3
      NOD(I)=NODE(I)
14    ND(I)=I
      LOOP=1
      VOL=0.
C
C  SKIP FOR A SINGLE TRIANGLE
C
      IF(NON.EQ.3)GO TO 12
      LOOP=4
C
C  CALCULATE THE CENTRAL NODE COORD.
C
      XX=0.
      YY=0.
      DO 3 I=1,4
      II=NODE(I)
      XX=XX+CORD(II,1)
3     YY=YY+CORD(II,2)
      NC1=NC+1
      CORD(NC1,1)=XX/4.
      CORD(NC1,2)=YY/4.
12    CONTINUE
C
C  LOOP OVER ALL ELEMENTS
C
      DO 15 ID=1,LOOP
C
C  SPLIT QUAD INTO TRIANGLES BASED ON THE PASS NO.
C
      IF(NON.EQ.3)GO TO 13
      DO 4 J=1,2
      I1=ID-1+J
      IF(I1.GT.4)I1=I1-4
      NOD(J)=NODE(I1)
4     ND(J)=I1
      NOD(3)=NC1
      ND(3)=5
13    CONTINUE
C
C  EVALUATE THE STRAIN-DISPLACEMENT MATRIX [B] (4,6)
C
      CALL MATRXB(N,NOD,AR,RMN,B)
      VL=PHI2*RMN*AR
      VOL=VOL+VL
C
C  CALCULATE MATRIX [D][B] (4,6)
C
      DO 5 I=1,4
      DO 5 J=1,6
      DB(I,J)=0.
      DO 5 K=1,4
5     DB(I,J)=DB(I,J)+D(I,K)*B(K,J)
C
C  CALCULATE MATRIX [B]T[D][B] (6,6)
C
      DO 6 I=1,6
      DO 6 J=1,6
      ESTIFF(I,J)=0.
      DO 6 K=1,4
6     ESTIFF(I,J)=ESTIFF(I,J)+DB(K,I)*B(K,J)*VL
C
C  CALCULATE UNIT THERMAL LOAD VECTOR
C
      DO 7 K=1,6
      RH(K)=0.
      DO 7 J=1,4
7     RH(K)=RH(K)+B(J,K)*DE(J)*VL
C
C  ASSEMBLE TOT. STIFFNESS MATRIX [K] (10,10)
C
      I=0
      DO 8 JJ=1,3
C  COMPUTE ROW NO.
      NROW=(ND(JJ)-1)*2
      DO 8 J=1,2
      NROW=NROW+1
      I=I+1
      RHS(NROW)=RHS(NROW)+RH(I)
      II=0
      DO 8 KK=1,3
```

```
C  COMPUTE COL NO.
      NCOL=(ND(KK)-1)*2
      DO 8 K=1,2
      NCOL=NCOL+1.
      II=II+1
8     STIFF(NROW,NCOL)=STIFF(NROW,NCOL)+ESTIFF(I,II)
C
15    CONTINUE
      IF(NON.EQ.3)RETURN
C
C  CONDENSE OUT THE INTERNAL DOF.
C
      DO 9 J=1,2
      IJ=10-J
      IK=IJ+1
      PIVOT=STIFF(IK,IK)
      DO 10 K=1,IJ
      F=STIFF(IK,K)/PIVOT
      STIFF(IK,K)=F
      DO 11 I=K,IJ
      STIFF(I,K)=STIFF(I,K)-F*STIFF(I,IK)
11    STIFF(K,I)=STIFF(I,K)
10    RHS(K)=RHS(K)-STIFF(IK,K)*RHS(IK)
9     RHS(IK)=RHS(IK)/PIVOT
C
      RETURN
      END
```

SUBROUTINE AVEQUAD

```
      SUBROUTINE CSTAV(IG,NEL,NUN,NODE,D,DE,STIFF,RHS,VOL)
C
C  ROUTINE TO CALCULATE THE ELEMENT STIFFNESS MATRIX FOR A TWO DIMEN.
C  TRIANGULAR ELEMENT  OR QUAD ELM. SPLIT INTO TRI. ELMS.
C
C  D=DMATRIX(4*4)
C  B=BMATRIX(4*6)
C
      IMPLICIT REAL*8(A-H,O-Z)
      COMMON/CNTRL/IR,IW,IC,ITYP,MODEX,THIC,NELM,NC
      COMMON CORD(501,2)
      DIMENSION STIF(6,6),B(4,6),DB(4,6),D(4,4),NODE(4),NOD(4),DE(4),
     1STIFF(10,10),RHS(10),RH(6),ND(4)
C  FOR THE WEIGHTED AVERAGE ELEMENTS,
C  FAC=1., FOR TRIANGULAR ELEMENTS
C  FAC=2., FOR SPLIT QUAD. ELEMENTS
C
      PHI2=8.*ATAN(1.)
C
      DO 1 I=1,10
      RHS(I)=0.
      DO 1 J=1,10
1     STIFF(I,J)=0.
C
      VOL=0.
C
C  LOOP OVER
C
      IGO=ABS(IG)
      DO 10 IDO=1,IGO*2
C
      DO 3 I=1,4
      ND(I)=I
3     NOD(I)=NODE(I)
      IF(NON.EQ.3)GO TO 2
C
      DO 4 J=1,2
      I1=IDO
      IF(IDO.EQ.2.AND.IG.EQ.1)I1=IDO+1
      IF(IDO.EQ.1.AND.IG.EQ.-1)I1=IDO+3
      N=I1+J-1
      IF(N.GT.4)N=N-4
      ND(J)=N
4     NOD(J)=NODE(N)
      N=N+1
      IF(N.GT.4)N=N-4
      NOD(3)=NODE(N)
      ND(3)=N
C
2     FAC=IGO
C
C  FORM STRAIN-DISPLACEMENT MATRIX B(4*6)
C
      CALL MATRXB(NEL,NOD,AREA,RMN,B)
C
C  CALCULATE MATRIX (D)(B) (4*6)
C
      DO 5 I=1,4
      DO 5 J=1,6
      DB(I,J)=0.
      DO 5 K=1,4
5     DB(I,J)=DB(I,J)+D(I,K)*B(K,J)
C
      VL=PHI2*RMN*AREA/FAC
C
C  CALCULATE (B)T(D)(B) VOL (6*6)
C
      DO 6 I=1,6
      DO 6 J=1,6
      STIF(I,J)=0.
      DO 6 K=1,4
6     STIF(I,J)=STIF(I,J)+DB(K,I)*B(K,J)*VL
C
      DO 7 K=1,6
      RH(K)=0.
      DO 7 J=1,4
7     RH(K)=RH(K)+B(J,K)*DE(J)*VL
C
      VOL=VOL+VL
C
      I=0
      DO 350 JJ=1,3
C
C  COMPUTE ROW NO.
C
      NROW=(ND(JJ)-1)*2
      DO 350 J=1,2
      NROW=NROW+1
      I=I+1
      RHS(NROW)=RHS(NROW)+RH(I)
C
C  COMPUTE CULUMN. NO.
C
      II=0
      DO 350 KK=1,3
      NCOL=(ND(KK)-1)*2
      DO 350 K=1,2
      NCOL=NCOL+1
      II=II+1
C
      STIFF(NROW,NCOL)=STIFF(NROW,NCOL)+STIF(I,II)
350   CONTINUE
      IF(NON.EQ.3)RETURN
10    CONTINUE
C
      RETURN
      END
```