THE FINITE ELEMENT ANALYSIS OF ELASTO-STATIC PROBLEM INCLUDING AUTOMATIC MESH GENERATION

by
Jaw-Ping Pan

Submitted in Partial Fulfillment of the Requirements
for the Degree of
Master of Science
in the
Civil Engineering
Program

Paul X Bellini   21 MAY 81
Advisor   Date

Sally M. Hotchkiss   July 28, 1981
Acting Dean of the Graduate School   Date

YOUNGSTOWN STATE UNIVERSITY
April, 1981

ABSTRACT

THE FINITE ELEMENT ANALYSIS OF ELASTO-STATIC PROBLEM INCLUDING AUTOMATIC MESH GENERATION

Jaw-Ping Pan
Master of Science
Youngstown State University

The purpose of this thesis is to present the Finite Element Method as a working tool for the stress analyst. This work is an attempt to explain the subject with sufficient clarity so that the reader can master the fundamentals of the subject from theory through practical use, thus enabling him to develop programs for his own particular applications in his own area of specialty.

In this thesis, eight main programs dealing with elasticity problems are presented including the axial bar element, the triangular plane stress-plane strain element for both constant stress and linearly varying stress, the combined bar element and plane stress triangular element, the transverse plate bending element, the automatic mesh generation code, and lastly the automatic mesh generation for both the bar element and the triangular element.

Emphasis is placed on the triangular plane stress element since it has a history of good mathematical

performance; therefore, it is utilized to demonstrate all the features of a typical finite element computer code.

For each computer code that is constructed a flow chart is presented, a guide for the data input to the code is summarized, and a sample problem is solved.

Input data, program output, and numerical interpretation and comparisons are summarized for each problem considered.

ACKNOWLEDGEMENTS

The author wishes to express his deep gratitude to his thesis advisor, Dr. Paul X. Bellini, whose guidance, effort, time and encouragement directly contributed in the completion of this thesis. Thanks are also due to the review committee, Dr. John Cernica and Professor John F. Ritter, for their careful review and their valuable time toward the completion of the requirements of this thesis.

Special appreciation is given to my fiancee , Miss Barbara J. Fulmer, for her very skillful typing of the manuacript.

The author is particularly indebted to his parents, Mr. and Mrs. W. H. Pan, for their full support and encouragement to his studies.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER I

INTRODUCTION

1-1 Background

The limitation of human minds is such that it cannot grasp the behaviour of its complex surroundings and creations in one operation. Thus, the process of subdividing any systems into its individual components or 'elements', whose behaviour is readily understood, and then rebuilding the original system from such components is a natural way in which the engineer proceeds to study the system's behaviour.

To overcome the intractability of the realistic type of continuum problems, various methods have from time to time been proposed both by engineers and mathematicians. The engineer approaches the problem more intuitively by creating an analogy between discrete elements and finite portions of the continuum domain. A major innovation in the last decade in the development of realistic techniques to solve continuum problems using discretized methods has been made possible by the evolution of Finite Element Methods of solution.

1-2 Historical Review

In 1941, Hrennikoff(1)* presented a useful but limited method of solving plane stress problems

* Number in parentheses refers to literature cited in the Bibliography.

by replacing plate elements with bars thus reducing a plate problem to one in structural analysis. In 1954, Williams[2] had come very close to discovering the Finite Element Method, but his method still relied heavily on the finite difference approximation. The first complete presentation of the Finite Element Method was made in 1956 by Turner, Clough, Martin and Topp[3]. The Finite Element Methods would not be practical without the availability of high speed electronic computers with large amounts of storage capacity. The development of this analytical method and the 'computer' has progressed simultaneously.

Computers and programs in Finite Element Methods have recently become the most powerful tool available to engineers for the analysis of stress. The use of programs which determine stress and strain in various geometrical shapes is essential in designing more refined and reliable manufactured products. Stress analysis problems which a decade ago would have baffled the engineer are now solved with relative ease and in a routine manner by simply utilizing a standard finite element computer program which processes input data defining geometry, element material properties, and loading conditions.

1-3 Thesis Intent

The purpose of this thesis is threefold:

(i) The development of finite element computer programs for the analysis for two and three-dimensional elasticity problems including plane-stress, plane-strain, and plate-bending problems.

(ii) The introduction of mesh generation techniques and three mesh generation programs which greatly reduce the time required to input the necessary data into the computer program.

(iii) The solution of a set of sample examples to illustrate the use of each program and the interpretation of resulting solutions.

The basis of all the finite element computer programs contained in this thesis are given by Bowes and Russell[4]. Modifications of these programs have been made to suit the requirements of the YSU computer system. The background for the development of the mesh generation schemes is taken from Cheung and Yeo[5].

1-4 Thesis Overview

The finite element programs dealing with the bar element, the constant strain triangle, the linear strain triangle, the reinforced linear strain triangle, and the plate-bending element are presented in Chapter II, III, IV, V, VI, respectively. Each chapter is a self-contained entity with emphasis on various practical aspects of the computer method including: (i) a description of various techniques used in the choice of displacement functions, stiffness formulation, assumble, solution and mesh generation; (ii) a detailed explanation of a sample program at various levels of development together with comments for important FORTRAN program statements; (iii) a description of a typical element problem and the instructions for data input preparation, including nodal quantities such as geometric coordinates, boundary condition data, and loading condition, and element quantities such as connection data, and material properties.

The main program associated with the particular element is included in the Appendix for convenience. All subroutines which are associated with the main programs are listed in the Appendix B. Each subroutine is designed to compute one or at most a few basic steps in the finite element solution process, including forming the global stiffness matrix, inverting the matrix, solving equations, and printing results. This set of subroutines makes it possible for the analyst to use a complete set of working programs, one per element type for the solution of stress problems. After a familiarity is formed by the reader, these subroutines may be utilized to formulate programs for a variety of elements not included in this thesis.

For reasonable solution to many practical stress problems, thousands of elements and nodes are involved.

The task of preparing the data input becomes extremely time-consuming and tedious. Moreover, during the preparation of thousands of data cards, human error may be introduced and remain undetected in spite of the checks which are usually made. The presence of such errors which inevitably bring about incorrect results, require an additional computer run after correction of the data. However, if the errors stay undetected and such incorrect results are used for making decisions and judgements, there may be serious repercussions. It is therefore important to eliminate such data error, and this is achieved to a large extent by automatic mesh generation. In this process nodal numbers and their coordinates, together with element numbers and their definition, are prepared automatically by the computer program, using as input data a minimum amount of information necessary to describe the geometry of the domain and the desired fineness of the mesh generation. In the last few years, considerable effort has been expended in developing mesh generation routines in order to minmize the time required to prepare the data input, and to eliminate data input errors. An early literature survey of a number of such routines is given by Buell and Bush[(6)]. The first generator for triangular mesh code program was written, but never published, In 1958 by Mr. R. Maclean for an IBM 704 computer. Since then, much of the information developed on mesh generation techniques has been utilized in the various finite element programs.

Mesh generation programs written by this author in this thesis include programs for constant and linear strain triangular elements and a three dimensional space truss. All programs contain node points generation and element generation. An example of use of these programs is given in each case.

CHAPTER II

THE BAR ELEMENT

2-1 Formulation of Stiffness Matrix

For a uniaxial, two dimensional structural member, two nodal forces, each having two components, must be considered. These four force components are related to the associated four components of nodal displacements. The stiffness matrix of the structural element is expressed as a (4x4) stiffness matrix, $[K]$, which relates force to the displacement components in the matrix symbolic form

$$\{f\} = [k]\{\delta\} \qquad (2\text{-}1)$$

Referring to Figure (2-1) the stiffness matrix of the element is described first with respect to the element coordinate axes X'-Y' as shown. The functions that defined the u' and v' displacements at any point P on the undeformed element to the point P* on the deformed element is assumed in the matrix form as

$$\begin{Bmatrix} u' \\ v' \end{Bmatrix} = \begin{bmatrix} 1 & x' & 0 & 0 \\ 0 & 0 & 1 & x' \end{bmatrix} \begin{Bmatrix} \alpha_1 \\ \alpha_2 \\ \alpha_3 \\ \alpha_4 \end{Bmatrix} \qquad (2\text{-}2)$$

The four unknown coefficients $\alpha_1, \alpha_2, \alpha_3, \alpha_4$ are determined by the four displacements at the node point of the element. These are given as:

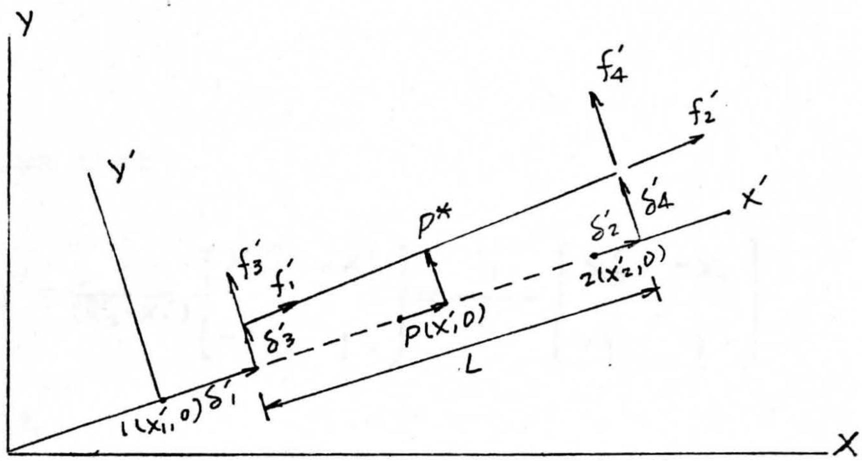


Figure (2-1) Bar Element and Global Frames

$u'_{(x'=x'_1)}=\delta'_1\ ,\quad u'_{(x'=x'_2)}=\delta'_2\ ,\quad v'_{(x'=x'_1)}=\delta'_3\ ,\quad v'_{(x'=x'_2)}=\delta'_4$
which when substituted into Equation (2.2) give

$$\begin{Bmatrix}\delta'_1\\ \delta'_2\\ \delta'_3\\ \delta'_4\end{Bmatrix}=\begin{bmatrix}1 & x'_1 & 0 & 0\\ 1 & x'_2 & 0 & 0\\ 0 & 0 & 1 & x'_1\\ 0 & 0 & 1 & x'_2\end{bmatrix}\begin{Bmatrix}\alpha_1\\ \alpha_2\\ \alpha_3\\ \alpha_4\end{Bmatrix}$$

or,

$$\{\delta'\}=[A]\{\alpha\}$$

Inversion of the latter equation yields

$$\{\alpha\}=[A]^{-1}\{\delta\} \qquad (2.3)$$

and in matrix partitioned form becomes

$$[A]^{-1}=\left[\begin{array}{c|c}[\bar{A}]^{-1} & 0\\ \hline 0 & [\bar{A}]^{-1}\end{array}\right] \qquad \text{where } [\bar{A}]=\begin{bmatrix}1 & x'_1\\ 1 & x'_2\end{bmatrix}$$

It follows that

$$[\bar{A}]^{-1} = \frac{1}{(X_2' - X_1')} \begin{bmatrix} X_2' & -X_1' \\ -1 & 1 \end{bmatrix} = \frac{1}{L} \begin{bmatrix} X_2' & -X_1' \\ -1 & 1 \end{bmatrix}$$

and hence,

$$[A]^{-1} = \frac{1}{L} \begin{bmatrix} X_2' & -X_1^2 & 0 & 0 \\ -1 & 1 & 0 & 0 \\ 0 & 0 & X_2' & -X_1' \\ 0 & 0 & -1 & 1 \end{bmatrix} . \qquad (2.4)$$

Substituting Equation (2.3) into Equation (2.2), one obtains

$$\begin{Bmatrix} u' \\ v' \end{Bmatrix} = \begin{bmatrix} 1 & X_1' & 0 & 0 \\ 0 & 0 & 1 & X' \end{bmatrix} [A]^{-1} \{\delta'\} \qquad (2.5)$$

From elementary strength of material, the axial strain is given as

$$\varepsilon = \frac{du'}{dX'} = \left\{ \frac{d}{dX'} \quad 0 \right\} \begin{Bmatrix} u' \\ v' \end{Bmatrix}$$

Using Equation (2.5) in the latter equation gives

$$\begin{aligned} \varepsilon &= \left\{ \frac{d}{dX'} \quad 0 \right\} \begin{bmatrix} 1 & X' & 0 & 0 \\ 0 & 0 & 1 & X' \end{bmatrix} [A]^{-1} \{\delta'\} \\ &= \{ 0 \quad 1 \quad 0 \quad 0 \} [A]^{-1} \{\delta'\} \\ &= [B][A]^{-1} \{\delta'\} \qquad (2.6a) \end{aligned}$$

$$\text{where} \quad [B] \equiv \{ 0 \quad 1 \quad 0 \quad 0 \} \qquad (2.6b)$$

The linear elastic Hooke's law in its one-dimensional form becomes $\sigma = E\varepsilon$ where all quantities are scalars. This equation is written in matrix form as

$$\{\sigma\} = [D]\{\varepsilon\} \tag{2.7}$$

where in the present case each matrix in Equation (2.7) is a (1x1) matrix and hence, a scalar quantity. Substituting Equation (2.6) into Equation (2.7) yields the state of stress in the element as

$$\{\sigma\} = [D][B][A]^{-1}\{\delta'\} \tag{2.8}$$

At this stage, the element is given an additional virtual displacement $\{\delta'^*\}$: This is assumed as an arbitrary quantity produces an additional strain into fibers given by

$$\{\varepsilon^*\} = [B][A]^{-1}\{\delta'^*\} \tag{2.9}$$

The nodal force components of $\{f'\}$ move through distances $\{\delta'^*\}$ and do an amount of external work, δW_E, during the virtual displacement, which is **defined as**

$$\delta W_E = \delta_1'^* f_1' + \delta_2'^* f_2' + \delta_3'^* f_3' + \delta_4'^* f_4' = \{\delta'^*\}^T\{f'\}$$

at the same time, internal work is done by existing stress in the fibers as the virtual strain is applied. This change in internal work, δW_I, is given by

$$\delta W_I = \int_{VOL} [\varepsilon^*]^T[\sigma]\, dV$$

Combining Equation (2.8) and (2.9), one obtains

$$\delta W_I = \int_{VOL} [[B][A]^{-1}[\delta'^*]]^T [D][B][A]^{-1}\{\delta'\}\, dV$$

$$= \int_{VOL} [\delta'^*]^T [A^{-1}]^T [B]^T [D][B][A]^{-1}\{\delta'\}\, dV$$

Since all terms to be integrated are constant in this case, it follows that

$$W_I = \{\delta'^*\}^T [A^{-1}]^T [B]^T [D][B][A]^{-1}\{\delta'\} \times VOL$$

For statical equilibrium the total virtual work must balance, hence,

$$\delta W_E = \delta W_I$$

or

$$\{\delta'^*\}^T \{f'\} = \{\delta'^*\}^T [A^{-1}]^T [B]^T [D][B][A]^{-1}\{\delta'\}(VOL)$$

hence,

$$\{f'\} = (VOL)[A^{-1}]^T [B]^T [D][B][A^{-1}]\{\delta'\} \qquad (2.10)$$

Comparing Equation (2.10) with Equation (2.1), the element stiffness matrix described with respect to the element axes becomes

$$[K'] = (VOL)[A^{-1}]^T [B]^T [D][B][A]^{-1} \qquad (2.11)$$

The volume term is defined as $VOL = \int_V dV$ which for constant cross-section becomes $(VOL) = AL$.

Equation (2.11) defines the stiffness of the element with respect to the element coordinate system which is with respect to the global coordinate frame (see Figure (2-1)). The analysis of any structure is made with reference to the system global axes and hence the element stiffness matrix must be transformed to the global frame.

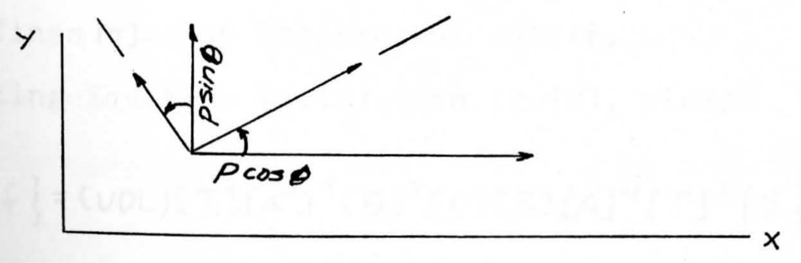


Figure (2-2) Vector Components - Element and Global Frames

Based on Figure (2-2), the force components in the global X and Y directions are written in matrix form as

$$\{f\} = \begin{bmatrix} \cos\theta & 0 & -\sin\theta & 0 \\ 0 & \cos\theta & 0 & -\sin\theta \\ \sin\theta & 0 & \cos\theta & 0 \\ 0 & \sin\theta & 0 & \cos\theta \end{bmatrix} \{f'\}$$

or $\{f\} = [T]\{f'\}$

Similarly for the displacment components

$$\{\delta\} = [T]\{\delta'\}$$

hence

$$\{f'\} = [T]^{-1}\{f\} \quad , \quad \{\delta'\} = [T]^{-1}\{\delta\} \qquad (2.12)$$

Noting the determinant of $[T]$ is equal to unity, it follows that,

$$[T]^{-1} = \begin{bmatrix} \cos\theta & 0 & \sin\theta & 0 \\ 0 & \cos\theta & 0 & \sin\theta \\ -\sin\theta & 0 & \cos\theta & 0 \\ 0 & -\sin\theta & 0 & \cos\theta \end{bmatrix} \qquad (2.13)$$

It may be shown that

$$[T]^{-1} = [T]^{T} \qquad (2.14)$$

which defines $[T]$ as an orthonormal matrix.

Substituting Equation (2.12) into (2.10), yields

$$\{f\} = (VOL)[T][A^{-1}]^{T}[B]^{T}[D][B][A]^{-1}[T]^{T}\{\delta\}$$

The global stiffness matrix takes the symbolic form

$$[K] = (VOL)[T][A^{-1}]^T[B]^T[D][B][A]^{-1}[T]^T$$

A matrix component values on the right side of the latter equation are known. Substituting these definitions the global element stiffness matrix in component form becomes

$$[K] = \left(\frac{AE}{L}\right)\begin{bmatrix} \cos^2\theta & -\cos^2\theta & \sin\theta\cos\theta & -\sin\theta\cos\theta \\ -\cos^2\theta & \cos^2\theta & -\sin\theta\cos\theta & \sin\theta\cos\theta \\ \sin\theta\cos\theta & -\sin\theta\cos\theta & \sin^2\theta & -\sin^2\theta \\ -\sin\theta\cos\theta & \sin\theta\cos\theta & -\sin^2\theta & \sin^2\theta \end{bmatrix} \quad (2\text{-}15)$$

wherein the cross-sectional area is assumed constant. It should be noted that $[K] = [K]^T$ and $|[K]| = 0$.

2-2 Generation of the Structure Stiffness Matrix

If the structure has N nodes, it possesses a total of 2N components of displacement and 2N components of forces. For the three nodes structure shown in Figure (2-3), the nodal displacement vector, $\{U\}$, containing all nodal displacements, $\{F\}$ containing all nodal force components, are each (6x1) matrices.

Consequently,

$$\{F\} = [K_A]\{U\} \quad (2.16)$$

where $[K_A]$ is a (6x6) structure stiffness matrix.

The horizontal force component at node M is in position 2M-1 in the force vector and vertical component is in position 2M. Components of displacement are similarly

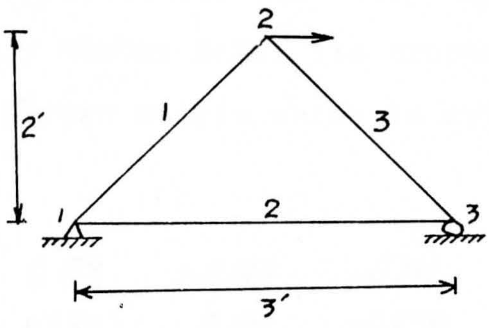


Figure (2-3) Sample Example-Truss System

located. Each structural member contributes certain elements to the structure stiffness matrix. Totaling these contribution from all members, a complete structure stiffness matrix is obtained. From Equation (2.15), the element stiffness matrix of member 2 in Figure (2-3) is written as

$$\begin{Bmatrix} f_1 \\ f_5 \\ f_2 \\ f_6 \end{Bmatrix} = \begin{matrix} \\ 1 \\ 5 \\ 2 \\ 6 \end{matrix} \begin{matrix} 1 \quad\quad 5 \quad\quad 2 \quad\quad 6 \\ \begin{bmatrix} K_{11} & K_{15} & K_{12} & K_{16} \\ K_{51} & K_{55} & K_{52} & K_{56} \\ K_{21} & K_{25} & K_{22} & K_{26} \\ K_{61} & K_{65} & K_{62} & K_{66} \end{bmatrix} \end{matrix} \begin{Bmatrix} U_1 \\ U_5 \\ U_2 \\ U_6 \end{Bmatrix}$$

The entries approaching in column 1 of the [K] in the latter equation appear in column 1 of the structure stiffness matrix; those of column 2 appear in column 5; etc. Similarly, the first row of [K] appear in the first row of $[K_A]$, etc. At this stage, $[K_A]$ is written as

$$[K_A] = \begin{bmatrix} K_{11} & K_{12} & 0 & 0 & K_{15} & K_{16} \\ K_{21} & K_{22} & 0 & 0 & K_{25} & K_{26} \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ K_{51} & K_{52} & 0 & 0 & K_{55} & K_{56} \\ K_{61} & K_{62} & 0 & 0 & K_{65} & K_{66} \end{bmatrix}$$

By repeating this process for all other members of the assembly, each and every member makes its proper contribution to the structure stiffness matrix which is obtained in complete form as

$$[K_A] = AE \begin{bmatrix} 0.477 & 0.192 & -0.144 & -0.192 & -0.333 & 0 \\ 0.192 & 0.256 & -0.192 & -0.256 & 0 & 0 \\ -0.144 & -0.192 & 0.288 & 0 & -0.144 & 0.192 \\ -0.192 & -0.256 & 0 & 0.512 & 0.192 & -0.256 \\ -0.333 & 0 & -0.144 & 0.192 & 0.477 & -0.192 \\ 0 & 0 & 0.192 & -0.256 & -0.192 & 0.256 \end{bmatrix}$$

2-3 Condensation of Stiffness Matrix

In most stiffness matrices the non-zero components are clustered near the main diagonal, and outside this band are zero values (see Figure (2-4a)). Such matrices are called banded matrices and for most structural problems in which the nodes have been numbered efficiently, the stiffness matrix possesses a minimum band width. The symmetry and banded nature of the stiffness matrix is utilized in defining a new smaller size matrix storage, in which only the upper half (or lower half) of the band is stored as rectangular array, $[S]$, as shown in Fighre (2-4), the storage requirement is significantly reduced becouse all the zero components outside the band are not stored in memory.

If a particular member connects node L to node M, the column range of its components would be from columns 2L-1 to 2M and the band width would be $\left[2M-(2L-1)+1\right]$. Such an element

$$\begin{bmatrix} a_{11} & a_{12} & a_{13} & 0 & 0 & 0 & 0 \\ a_{21} & a_{22} & 0 & a_{24} & 0 & 0 & 0 \\ a_{31} & 0 & a_{33} & a_{34} & a_{35} & 0 & 0 \\ 0 & a_{42} & a_{43} & a_{44} & 0 & a_{46} & 0 \\ 0 & 0 & a_{53} & 0 & a_{55} & a_{56} & a_{57} \\ 0 & 0 & 0 & a_{64} & a_{65} & 0 & 0 \\ 0 & 0 & 0 & 0 & a_{75} & 0 & 0 \end{bmatrix}$$

Figure (2-4a) Typical Banded Stiffness Matrix

$$\begin{bmatrix} a_{11} & a_{22} & a_{33} & a_{44} & a_{55} & 0 & 0 \\ a_{12} & 0 & a_{34} & 0 & a_{56} & 0 & 0 \\ a_{13} & a_{24} & a_{35} & a_{46} & a_{51} & 0 & 0 \end{bmatrix}$$

Figure (2-4b) Storage Matrix for Banded Stiffness Matrix

may span a different number of columns and the maximum span determines the actual band width=NBAND. As each element is considered the difference between the end node numbers recorded, the maximum difference, NDIFF, is selected, and hence the band width is calculated as

$$NBAND=2(NDIFF+1)$$

It should be noted in this case that each node has two degrees of freedom. By a skillful assignment of node numbers the band width may be minimized. A simple procedure is to number across the small dimension at one extremity of the structure and then in succeeding adjacent rows until the entire structure has been covered.

Since the algebra for substituting [S] into the stiffness equation formulation has not yet been developed, calculation are performed using $[K_A]$. Each numerical value of $[K_A]$ is extracted as required from the correponding location in S .

2-4 Determination of the Displacement Unknown

For the case shown in Figure (2-3), the support conditions require that U_1 , U_2 , U_3 = 0; while the loading conditions define F_1 ,F_2 , and F_3 as unknown reaction components. In general, this correspondence between the number of knowns and unknowns in {F} and {U} always exists. The Equation (2.10) expresses in matrix form a system of six simultaneous equations with six unknowns. This form differs from a set of classical simultaneous equations in that the unknowns are not all the {U} vector. A practical method of solution is given by Payne and Irons [7], which uses fictitious components of force in place of the unknowns, makes same changes in $[K_A]$ and solves as though all the {U} components are unknown. Let us say U_i is a known component. Then a fictitous component is put into the force vector given by

$$F_i = C K_{Aii} U_i$$

where C is a large number. (In practice, $C=10^{12}$ is satisfactory.)

The stiffness matrix is changed by substituting $C[K_{Aii}]$ for $[K_{Aii}]$. The ith equation is now treated in the same way as all the others, the changes being such that they ensure that the equations when solved give the original known value for $\{U_i\}$. By treating each row that contains a known $\{U\}$ component in this way the problem is changed to one in which all the $\{F\}$ components have numerical values and all the $\{U\}$ components are treated as unknowns. Solving this classical problem gives all values of $\{U\}$. A simple substitution of the known $\{U\}$ into Equation (2.16) using the original unmodified $[K_A]$ gives all $\{F\}$ components. This practical method is a solution type recommended by Payne and Irons(7) for large scale systems.

2-5 Determination of Element Stresses

Substituting Equation (2.15) into (2.13) into (2.8), one obtains the expression for stress as

$$\{\sigma\} = [D][B][A]^{-1}[T]^T\{\delta\}$$

where $\{\delta\}$ components of each member have been determined by selecting the appropriate components from the $\{U\}$ vector. Hence, the stress can be calculated for each member.

2-6 Pin-Jointed Truss Program

The main program, referred to as PT10B, solves for stresses in the members of a truss under any conceivable system of loads, for any degree of redundancy of the structure. The FORTRAN statements of PT10B are listed in Appendix A.

Figure (2-5) gives the PT10B Flow Chart; Figure (2-6), Instructions for PT10B Data Deck Preparation; and Figure (2-8). Typical Output for a problem proccessed by PT10B. These are given here to provide the reader a better understanding of PT10B executing procedures.

Some important observations for data deck preparation are:

1). Cases in which the Modulus of Elasticity is not the same for all members can not be treated by this program.

2). The node numbers and element numbers must be positive and run sequentially beginning at 1.

3). Only non-zero load components need be read in as data. The program was arranged this way in order to substantially reduce the input data deck in cases where a large number of nodes are not loaded; zero displacements must be read in.

4). If there is known settlement at any support this can be treated by reading the amount of the settlement as the value of known displacement component.

5). If the user wants to study the effect of a different loading or a support system, the program can be returned from the point where known load components are read. The user also has the option of executing the program from the beginning with a complete new data deck. The option that the user elects is indicated by a number "NEXT" in another data card.

1. PRINT TITLE

READ AND PRINT CASE TITLE

READ AND PRINT COORDINATES OF NODES (RC02B)

READ AND PRINT ELEMENT DATA (CN10B)

ZERO [S]

REWIND 4

DO FOR ALL ELEMENTS

  CALCULATE STIFFNESS AND ADD TO S (ES10B)

WRITE [S] ON 4

2 READ AND PRINT KNOWN FORCES (KF01B)

READ AND PRINT KNOWN DISPLACEMENT (KU01B)

SOLVE FOR UNKNOWN DISPLACEMENTS (GE02B)

REWIND 4

READ [S] FROM 4

SOLVE FOR UNKNOWN FORCES. PRINT FORCES AND DISPLACEMENT (FU10B)

CALCULATE AND PRINT STRESSES (ST10B)

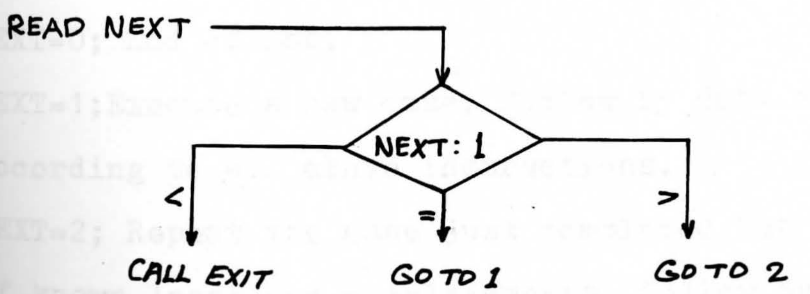


Figure (2-5) PT10B Flow Chart

DATA DECK

One card containing case title.

One card containing Young's Modulus Format: (F10.5)

One card for each node, containing;

Node number, x coord., y coord., Format: (I5, 2F10.5)

One blank card to indicate end of node data.

One card for each element containing:

Element numbers, cross-section area, node numbers at ends of element, Format: (I5, F10.5, 2I5)

One blank card to indicate end of element data.

One card for each unknown, non-zero force component, containing:

Component number, Force, Format: (I5, F10.5)

One blank card to indicate end of force data.

One card for each known dispacement component, containing:

Component number, Dispacement, Format: (I5, F10.5)

One blank card to indicate end of dispacement data.

One card containing NEXT. Format: (I5)

NEXT=0; End of job.

NEXT=1;Execute a new case. Follow by data cards prepared according to all above instructions.

NEXT=2; Repeat the case just completed but with a new set of known loads and displacements. Follow by data cards desribed above starting with force data cards.

Figure (2-6) Instruction For PT10B Data Decks Preparation

2-7 Sample Example- Pinned Truss

The King Post truss shown in Figure (2-7) is used as example.

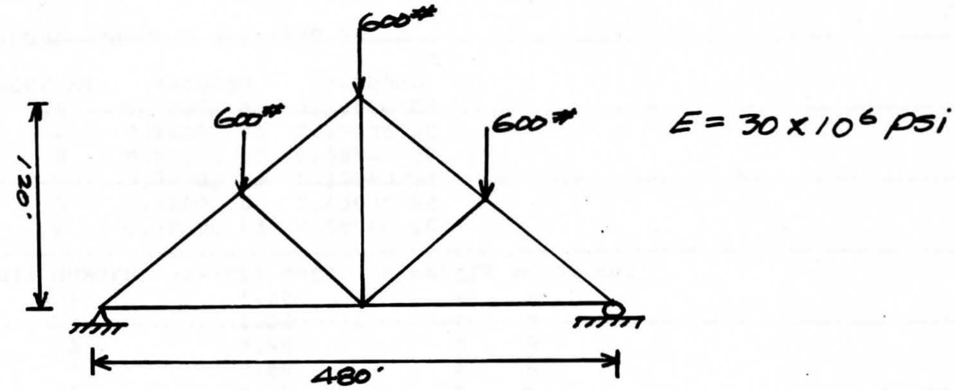


Figure (2-7) King Post Truss

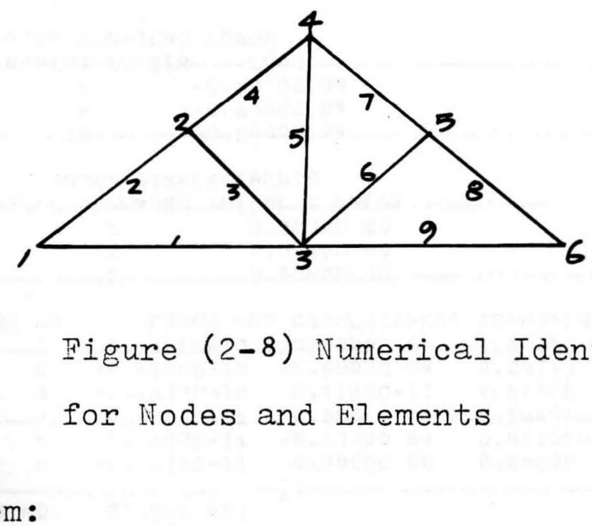


Figure (2-8) Numerical Identification System for Nodes and Elements

a). Problem:

To determine the force and displacement components at each panel point, the stress in each member, and the reaction force; compare the member stresses as found by the Finite Element Method with theoretical member stresses.

b). Solution:

To start to solve any problem by the Finite Element Method, the stressed body must be subdivided into elements.

```
MAIN PT10B
STRESS IN A PIN-JOINTED TRUSS

CASE TITLE  ---  STRESSES IN KING POST TRUSS

YOUNGS MODULUS = 0.300D 08

NODE NO.    X-COORD       Y-COORD
    1     0.0000D 00   0.0000D 00
    2     0.1200D 03   0.6000D 02
    3     0.2400D 03   0.0000D 00
    4     0.2400D 03   0.1200D 03
    5     0.3600D 03   0.6000D 02
    6     0.4800D 03   0.0000D 00

BAR NUMBER    X-SECT AREA    CONNECTS NODES NO.
    1            1.00           1    3
    2            1.20           1    2
    3            0.40           2    3
    4            0.80           2    4
    5            0.30           3    4
    6            0.40           3    5
    7            0.80           4    5
    8            1.20           5    6
    9            1.00           3    6

BAND WIDTH=      8

  KNOWN NON-ZERO LOADS
COMPONENT NUMBER      LOAD
       4         -0.6000D 04
       8         -0.6000D 04
      10         -0.6000D 04

    KNOWN DISPLACEMENTS
COMPONENT NUMBER  DISPLAC EMENT
       1            0.0000D 00
       2            0.0000D 00
      12            0.0000D 00

NODE NO.       FORCE AND DISPLACEMENT COMPONENTS
    1     0.4366D-10   0.9000D 04  -0.1325D-27  -0.1677D-12
    2    -0.4184D-10  -0.6000D 04   0.2479D 00  -0.6634D 00
    3    -0.1637D-10   0.1182D-10   0.1440D 00  -0.7034D 00
    4    -0.6366D-11  -0.6000D 04   0.1440D 00  -0.6234D 00
    5     0.5457D-11  -0.6000D 04   0.4015D-01  -0.6634D 00
    6    -0.7276D-11   0.9000D 04   0.2880D 00  -0.1677D-12

BAR NO.    STRESS PSI
    1     0.1800D 05
    2    -0.1677D 05
    3    -0.1677D 05
    4    -0.1677D 05
    5     0.2000D 05
    6    -0.1677D 05
    7    -0.1677D 05
    8    -0.1677D 05
    9     0.1800D 05

STATEMENTS EXECUTED=      3509
```

Figure (2-9) Typical Output for a Problem Processed by PT10B

	stress (psi)	
Member	Finite Element Method	Theoretical Value
1	18,000	18,000
2	-16,770	-16,770
3	-16,770	-16,770
4	-16,770	-16,770
5	20,000	20,000
6	-16,770	-16,770
7	-16,770	-16,770
8	-16,770	-16,770
9	18,000	18,000

Table (2-1) Comparison of Finite Element Method and Theoretical solutions

In this problem, since we have only the bar element to use, it is obvious that each bar member should be an element and the pin joint a node. A convenient numerical identification system for nodes and elements is shown in Figure (2-8). The information consisting of dimensions, loads and boundary condition is entered into the computer according to the instruction in Figure (2-6), and is printed out together with the solution as shown in Figure (2-9).

c). Comments on solution:

A comparison of the computer results and exact answers (see Table (2-1)) shows that the Finite Element Method results are exact in this case. The element developed in this chapter and used in the above problem, the pin-jointed bar element, because of the assumed displacement function, can represent exactly a linear axial displacement. If the members are of constant cross-section and have only axial loads, the resulting strain is constant and hence the displacement is linear. Because the assumed displacement and the actual displacement of bar element are both linear, the finite element solution yields, except for computer round-off, the exact solution.

CHAPTER III

THE CONSTANT STRAIN TRIANGULAR ELEMENT

3-1 Displacement Functions For A Triangular Element

Two-dimensional elastic problems were the first successful examples of the application of the Finite Element Method. We have already illustrated the basis of bar elements of the finite element formulation in Chapter II, where the general relationships were derived. Now, this is extended so that inplane loaded plate problems can be solved for displacements and stresses.

Consider a typical triangular element shown in Figure (3-1). The triangle is assigned displacement components and force components $\{f\}$. The displacements are calculated if the stiffness matrix, $[K]$, is known by the equation

$$\{f\}=[k]\{\delta\} \qquad (3.1)$$

To determine all component values in K, displacement functions u and v are assumed which give the horizontal and vertical displacements at any point P(x,y) within the triangle. The horizontal and vertical displacements are defined only at three nodal points (see Figure (3-1)) that is, the nodes of the triangle. Hence, no more than three coefficients are included in the polynomial fuction used to express the dis-

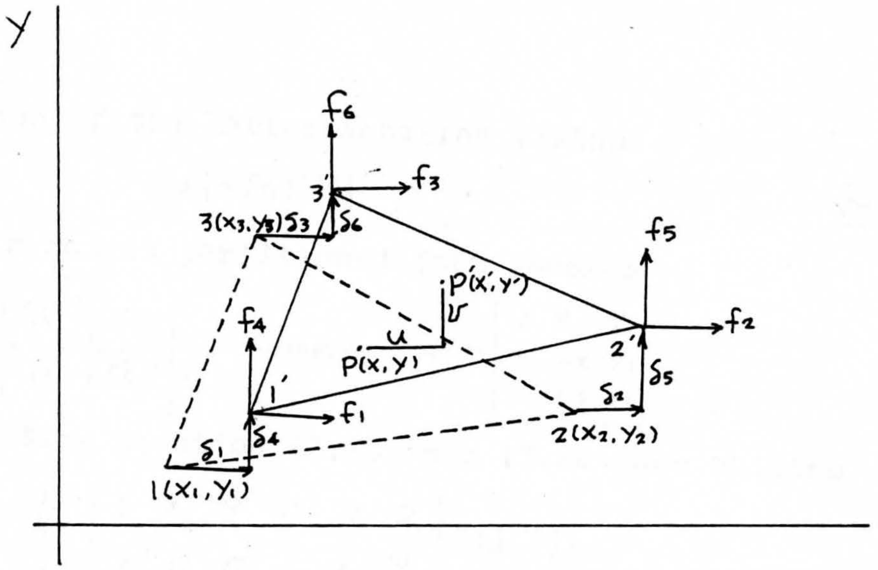


Figure (3-1) Displacement of Triangular Element

placement functions which in matrix form are

$$\begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} 1 & x & y & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & x & y \end{bmatrix} \begin{Bmatrix} \alpha_1 \\ \alpha_2 \\ \alpha_3 \end{Bmatrix} \qquad (3.2a)$$

or in symbolic matrix form

$$\{u\} = [P]\{\alpha\} \qquad (3.2b)$$

By applying the boundary conditions, it follows that

$$\begin{Bmatrix} \delta_1 \\ \delta_2 \\ \delta_3 \\ \delta_4 \\ \delta_5 \\ \delta_6 \end{Bmatrix} = \begin{bmatrix} 1 & x_1 & y_1 & 0 & 0 & 0 \\ 1 & x_2 & y_2 & 0 & 0 & 0 \\ 1 & x_3 & y_3 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & x_1 & y_1 \\ 0 & 0 & 0 & 1 & x_2 & y_2 \\ 0 & 0 & 0 & 1 & x_3 & y_3 \end{bmatrix} \begin{Bmatrix} \alpha_1 \\ \alpha_2 \\ \alpha_3 \\ \alpha_4 \\ \alpha_5 \\ \alpha_6 \end{Bmatrix} \qquad (3.3)$$

or,

$$\{\delta\} = [A]\{\alpha\} \qquad (3.4)$$

Inversion of the latter equation yields

$$\{\alpha\} = [A]^{-1}\{\delta\} \tag{3.5}$$

which in matrix partitioned form becomes

$$[A]^{-1} = \left[\begin{array}{c|c} [\bar{A}]^{-1} & 0 \\ \hline 0 & [\bar{A}]^{-1} \end{array}\right] \qquad \text{where} \quad [\bar{A}] = \begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{bmatrix}$$

Substituting Equation (3.5) into (3.2a) one obtains

$$\begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} 1 & x & y & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & x & y \end{bmatrix} [A]^{-1}\{\delta\} \tag{3.6}$$

The chosen displacement functions guarantee continuity of displacements with adjacent elements because the displacements vary linearly along any side of triangle, and with identical displacements imposed at the nodes, the same displacements exist all along the interface.

3-2 Strain and Stress in a Triangular Element

In the x-y plane, the total strain at any point within the element is defined by its three components which contribute to internal work as follows,

$$\{\varepsilon\} = \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{xy} \end{Bmatrix} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \begin{Bmatrix} u \\ v \end{Bmatrix} \tag{3.7a}$$

When displacement components u and v are substituted from Equation (3.6), this becomes

$$\{\varepsilon\} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \begin{bmatrix} 1 & x & y & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & x & y \end{bmatrix} [A]^{-1}\{\delta\} \tag{3.7b}$$

Inversion of the latter equation yields

$$\{\alpha\} = [A]^{-1}\{\delta\} \tag{3.5}$$

which in matrix partitioned form becomes

$$[A]^{-1} = \left[\begin{array}{c|c} [\bar{A}]^{-1} & 0 \\ \hline 0 & [\bar{A}]^{-1} \end{array}\right] \quad \text{where} \quad [\bar{A}] = \begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{bmatrix}$$

Substituting Equation (3.5) into (3.2a) one obtains

$$\begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} 1 & x & y & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & x & y \end{bmatrix} [A]^{-1}\{\delta\} \tag{3.6}$$

The chosen displacement functions guarantee continuity of displacements with adjacent elements because the displacements vary linearly along any side of triagle, and with identical displacements imposed at the nodes, the same displacements exist all along the interface.

3-2 Strain and Stress in a Triangular Element

In the x-y plane, the total strain at any point within the element is defined by its three components which contribute to internal work as follows,

$$\{\varepsilon\} = \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{xy} \end{Bmatrix} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \begin{Bmatrix} u \\ v \end{Bmatrix} \tag{3.7a}$$

When displacement components u and v are substituted from Equation (3.6), this becomes

$$\{\varepsilon\} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \begin{bmatrix} 1 & x & y & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & x & y \end{bmatrix} [A]^{-1}\{\delta\} \tag{3.7b}$$

then,

$$\{\varepsilon\} = \begin{bmatrix} 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \\ 0 & 0 & 1 & 0 & 1 & 0 \end{bmatrix} [A]^{-1}\{\delta\} \tag{3.7c}$$

or,

$$\{\varepsilon\} = [B][A]^{-1}\{\delta\} \tag{3.7d}$$

where

$$[B] = \begin{bmatrix} 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \\ 0 & 0 & 1 & 0 & 1 & 0 \end{bmatrix}$$

Equation (3.7d) gives the strains at any point P(x,y) in the triangular element. Knowing that all the entries of the equation are constant, the values of strain are constant at all points in the triangle. Hence, the triangle is referred to as the "constant strain triangle".

In elementary strength of materials, the relationships between strain and stress in an isotropic material are expressed in matrix form as

$$\begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{xy} \end{Bmatrix} = \begin{bmatrix} \frac{1}{E} & -\frac{\nu}{E} & 0 \\ -\frac{\nu}{E} & \frac{1}{E} & 0 \\ 0 & 0 & \frac{2(1+\nu)}{E} \end{bmatrix} \begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{Bmatrix} \tag{3.8a}$$

The equation for stress is obtained by matrix inversion as

$$\begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{Bmatrix} = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix} \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{xy} \end{Bmatrix} \tag{3.8b}$$

or,

(3.8c)

where

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

Substituting Equation (3.7d) into (3.8d), one obtains

$$\{\sigma\}=[D][B][A]^{-1}\{\delta\} \qquad (3.9)$$

Note that the elastic properties, the coordinates of the nodes and the nodal displacements of the triangular element are all knowns at this stage, thus, the stress in the element is calculated from the Equation (3.9) and hence the stresses are constant throughout the element.

3-3 The Stiffness Matrix of a Triangular Element

We follow the procedure used in Chapter II to establish the stiffness matrix of the element.

It is assumed that the element is given an additional virtual displacement $\{\delta^*\}$ which must be small enough so that the forces do not change significantly during the displacement. The external work done by the nodal force which produces the virtual displacement is given by

$$W_E=\delta_1^* f_1+\delta_2^* f_2+\delta_3^* f_3+\delta_4^* f_4+\delta_5^* f_5+\delta_6^* f_6=\{\delta^*\}^T\{f\}$$

At this state, the strain is given by Equation (3.7) as

$$\{\varepsilon^*\}=[B][A]^{-1}\{\delta^*\} \qquad (3.10)$$

The stress given by Equation (3.9) does internal work that is calculated for an incremental volume as

$$dW_I=\varepsilon_{xx}^*\sigma_{xx}\,dV+\varepsilon_{yy}^*\sigma_{yy}\,dV+\varepsilon_{xy}^*\sigma_{xy}\,dV=\{\varepsilon^*\}^T\{\sigma\}\,dV$$

Substituting from Equation (3.9) and Equation (3.10), the latter expression becomes

$$dW_I = [[B][A]^{-1}\{\delta^*\}]^T[D][B][A]^{-1}\{\delta\}\,dV$$
$$= \{\delta^*\}^T[A^{-1}]^T[B]^T[D][B][A]^{-1}\{\delta\}\,dV$$

Integrating over the entire element, the internal work done becomes

$$W_I = \int_{VOL} \{\delta^*\}^T[A^{-1}]^T[B]^T[D][B][A]^{-1}\{\delta\}\,dV$$

Since all quantities in the matrices to be integrated are constant, it follows that

$$W_I = \{\delta^*\}^T[A^{-1}]^T[B]^T[D][B][A]^{-1}\{\delta\}\,(VOL)$$

Since there is no loss of energy to the internal work done, hence

$$\{\delta^*\}^T\{f\} = \{\delta^*\}^T[A^{-1}]^T[B]^T[D][B][A]^{-1}\{\delta\}\,(VOL)$$

which yields

$$\{f\} = [A^{-1}]^T[B]^T[D][B][A]^{-1}(VOL)\{\delta\} \qquad (3.11a)$$

This is of the same form as Equation (3.1) with

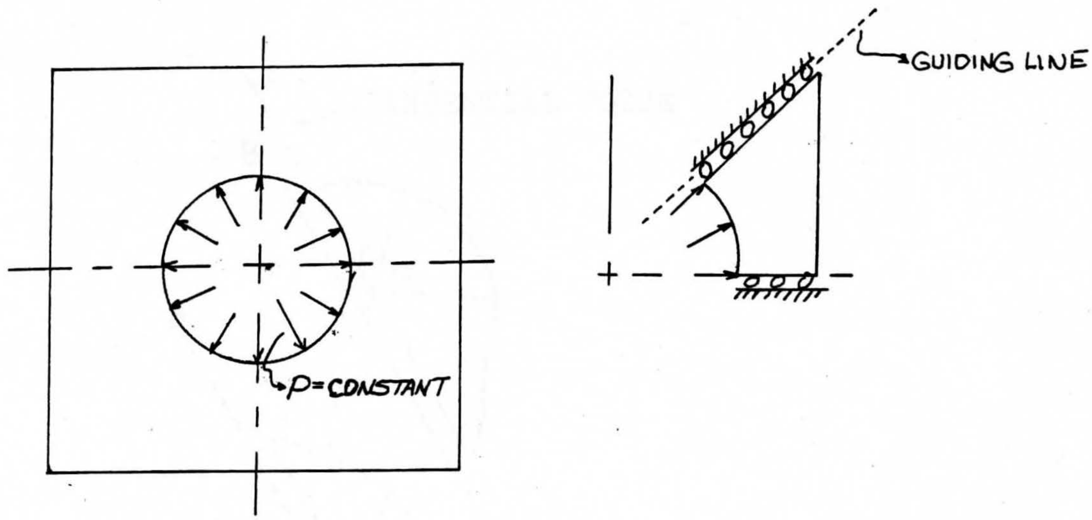
$$[K] = [A^{-1}]^T[B]^T[D][B][A]^{-1}(VOL) \qquad (3.11b)$$

Every term in the right side matrices of latter equation is known, so one may calculate the stiffness matrix for each triangular element. As each element stiffness matrix is determined, it is added to the general stiffness matrix in the manner similar to that of bar elements of Chapter II. For a loaded plate with a given supporting system, part of the force and displacement components are known. This is the same situation that exists in the truss structure and the methods for determining the unknown force and displacement components are identical. By using Equation (3.9), the stress in each element is readily determined.

3-4 Problem Reduction Using Axes of Symmetry

The displacement components are always considered to be parallel to the X and Y coordinate-axes in the theory that has been developed. Hence, by specifying one displacement component at a node and leaving the other component as an unknown, the node may be constrained to move in the horizontal or vertical direction. This enables one to take the advantage of axes of symmentry and thereby reduce the size of the problem to be solved. For the plate problem shown in Figure (3-2a), it is possible to constrain the nodes to move along the inclined radial lines. Thus, the problem is solved by dealing with only that portion shown in Figure (3-2b). Advantage is taken of the symmetry that exists along the radials. This saves both computer time and memory space required to solve the problem.

It is usually required to treat a multiple number of nodes that are guided to move along the guiding line. The treatment that follows deals with only one node, but for several guided nodes the process is merely repeated. To make the following more general, a known normal displacement to the guiding line, which usually is taken as zero in practice, will be considered and a known tangential force pro-



(a) Total Plate (b) Plate Segment

Figure(3-2) Nodes Constrained to Move Along Guiding Line

vided for.

Consider node N in Figure (3-3). The unload position of the node is given by C, it moves to C' after loading, the amount of displacement normal to AB is known. To consider the axes at node C to be locally rotated through an angle α, as shown in Figure (3-3b). The relationship between the two displacement systems is given from Figure(3-3b) as

$$\begin{Bmatrix} u_{2N-1} \\ u_{2N} \end{Bmatrix} = \begin{bmatrix} \cos\alpha & -\sin\alpha \\ \sin\alpha & \cos\alpha \end{bmatrix} \begin{Bmatrix} u'_{2N-1} \\ u_{2N} \end{Bmatrix} \tag{3.12a}$$

or,

$$\{u\} = [T]\{u'\} \tag{3.12b}$$

where

$$[T] = \begin{bmatrix} \cos\alpha & -\sin\alpha \\ \sin\alpha & \cos\alpha \end{bmatrix} \tag{3.12c}$$

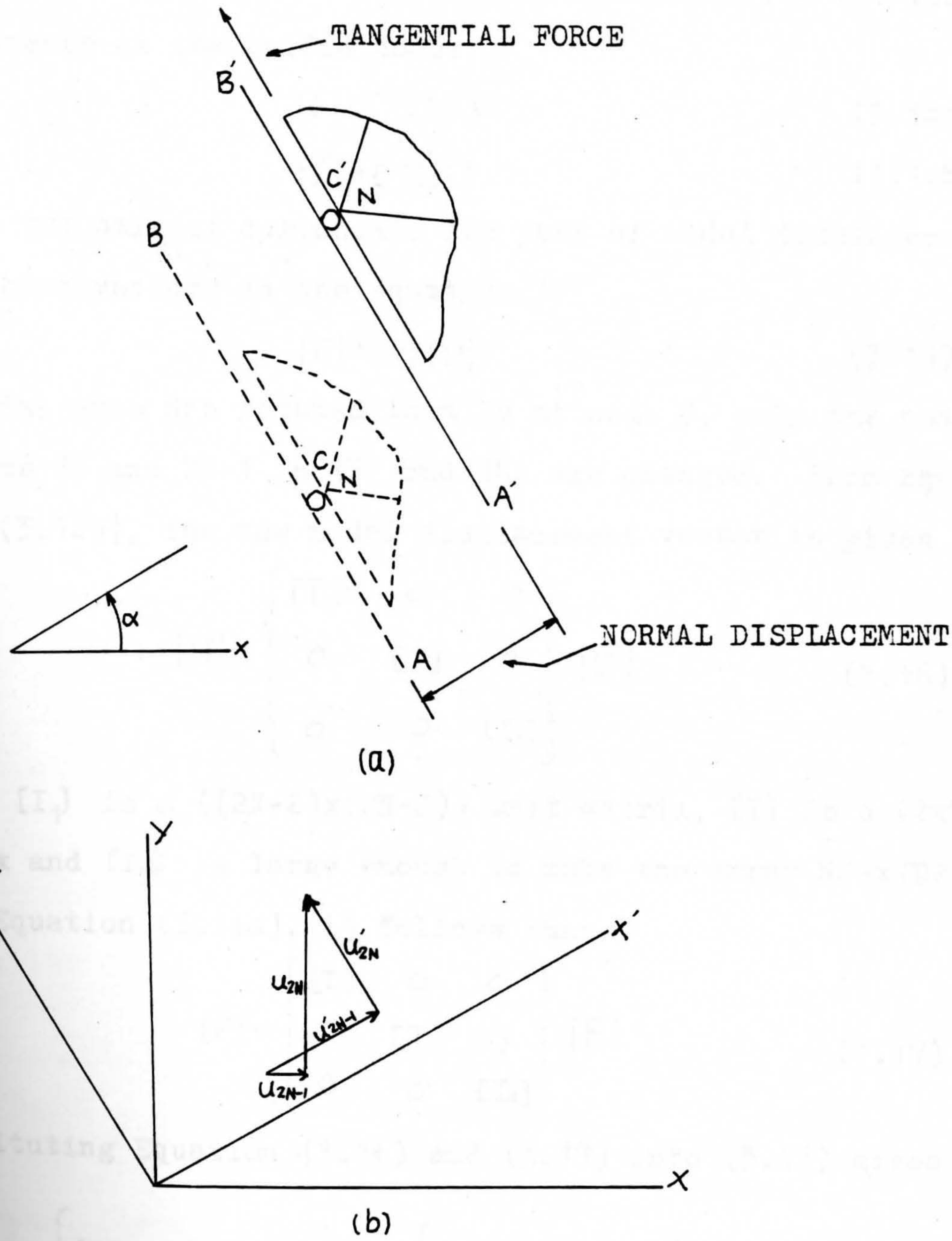


Figure (3-3) Guided Node

Hence,

$$\{u'\} = [T]^{-1}\{u\} = [T]^T\{u\} \qquad (3.13)$$

since [T] is an orthonormal matrix. Similarly for the force components at the guided node:

$$\{f\}=[T]\{f'\} \tag{3.14a}$$

and

$$\{f'\}=[T]^T\{f\} \tag{3.14b}$$

These two pair of components are part of nodal displacement and force vectors in the equation

$$\{F\}=[K]\{U\} \tag{3.15}$$

When the axes are rotated locally at node C, only the components 2N and 2N-1 in {F} and {U} are changed. From Equation (3.12b), the new nodal displacement vector is given by

$$\{U\}=\begin{bmatrix}[I_1] & 0 & 0\\ 0 & [T] & 0\\ 0 & 0 & [I_2]\end{bmatrix}\{U'\} \tag{3.16}$$

Where $[I_1]$ is a ((2N-2)x(2N-2)) unit matrix, [T] is a (2x2) matrix and $[I_2]$ is large enough to make the array NDFxNDF. From Equation (3.14a), it follows that

$$\{F\}=\begin{bmatrix}[I_1] & 0 & 0\\ 0 & [T] & 0\\ 0 & 0 & [I_2]\end{bmatrix}\{F'\} \tag{3.17}$$

Substituting Equation (3.16) and (3.17) into (3.15) gives

$$\{F'\}=\begin{bmatrix}[I_1] & 0 & 0\\ 0 & [T]^T & 0\\ 0 & 0 & [I_2]\end{bmatrix}[K]\begin{bmatrix}[I_1] & 0 & 0\\ 0 & [T] & 0\\ 0 & 0 & [I_2]\end{bmatrix}\{U'\} \tag{3.18}$$

or

$$\{F'\} = [K']\{U'\} \tag{3.19}$$

where

$$[K'] = \begin{bmatrix} [I_1] & 0 & 0 \\ 0 & [T]^T & 0 \\ 0 & 0 & [I_2] \end{bmatrix} [K] \begin{bmatrix} [I_1] & 0 & 0 \\ 0 & [T] & 0 \\ 0 & 0 & [I_2] \end{bmatrix} \tag{3.20}$$

By using Equation (3.20), one may modify the stiffness matrix to deal with axes at one node that are locally rotated. When several nodes are so constrained, the operation given in Equation (3.20) is simply repeated for each constrained node. The altered matrix [K'] together with the known components of {U'} and {F'} are then used to solve for the remaining components of {U'} . Since stress is calculated only from the displacement components of the original X-Y axes, the {U'} components are used to determine all {U} by Equation (3.16). Then recalling the original matrix {K} , and using {U} , the remaining {F} components are calculated.

3-5 Stiffness Matrix For a General Finite Element

The stiffness matrix of triangular element has been calculated and the stresses in each element of the assembly are found. For a general element a provision for strain that is not the result of stress is included. This will allow the possibility of treating cases involving thermal strain, creep, etc.

Consider the element, irrespective of its shape or number of components in its stress and strain vector, with strain $\{\varepsilon_o\}$, which is not related to load, but may change value freely from temperature change or for other reason. This strain ε_o will cause certain displacements which may not be ignored. When force $\{f\}$ is applied, an additional displacement will occur giving a total displacement represented by the displacement function $\{u\}$. Then the total strain $\{\varepsilon\}$ is found from the displacement function by performing certain differential operations given in an operator matrix, $[\Delta]$, that is,

$$\{\varepsilon\} = [\Delta]\{u\} \tag{3.21}$$

The elastic strain caused by force $\{f\}$ is given by $\{\varepsilon - \varepsilon_o\}$, and hence the stress is

$$\{\sigma\} = [D]\{\varepsilon - \varepsilon_o\} \tag{3.22}$$

The displacement function $\{u\}$ is expressed using a polynomial functions as

$$\{u\} = [P]\{\alpha\} \tag{3.23}$$

When the boundary conditions are applied the coordinates of the nodes when substituted into $[P]$ give displacement functions at the nodes, $\{\delta\}$, as

$$\{\delta\} = [A]\{\alpha\} \tag{3.24}$$

Solving Equation (3.24) gives the coefficients in the polynomials in the form $\{\alpha\} = [A]^{-1}\{\delta\}$. Substituting this into Equation (3.23) and then into Equation (3.21) yields

$$\{\varepsilon\} = [\Delta][P][A]^{-1}\{\delta\} \tag{3.25}$$

The operation $[\Delta]$ is performed on p giving a matrix B ; thus one obtains

$$\{\varepsilon\} = [B][A]^{-1}\{\delta\}$$

where

$$[B] = [\Delta][P] \qquad (3.26)$$

which, when substituted into Equation (3.22) gives

$$\{\sigma\} = [D][B][A]^{-1}\{\delta\} - [D]\{\varepsilon_0\} \qquad (3.27)$$

At this stage we have an element that has its nodes displaced by $\{\delta\}$ and is held in this position by some unknown force f . From this state let us give the element a virtual displacement $\{\delta^*\}$. This is assumed as an arbitrary quantity which produces an additional strain into fibers given by

$$\{\varepsilon^*\} = [B][A]^{-1}\{\delta^*\} \qquad (3.28)$$

The additional strain energy is produced by existing stress in the fibers as the virtual strain is applied. This increment in internal work, δW_I, is given by

$$\delta W_I = \int_{VOL} [[B][A]^{-1}\{\delta^*\}]^T [[D][B][A]^{-1}\{\delta\} - [D]\{\varepsilon_0\}] dV$$

$$= \int_{VOL} [\{\delta^*\}^T [A^{-1}]^T [B]^T][[D][B][A]^{-1}\{\delta\} - [D]\{\delta\}] dV \qquad (3.29)$$

When this virtual displacement is imposed, the external forces do an amount of external work, δW_E, which becomes

$$\delta W_E = \{\delta^*\}^T \{f\}$$

Equating external work to the increase in strain energy and removing constant terms from the integration process gives

$$\{f\} = [A^{-1}]^T \int [B]^T [D][B] dV [A]^{-1} \{\delta\} - [A^{-1}]^T \int [B]^T [D]\{\varepsilon_0\} dV \qquad (3.30)$$

Defining

$$[K] = [A^{-1}]^T \int [B]^T [D][B] dV [A]^{-1} \qquad (3.31)$$

then,

$$\{f\}=[K]\{\delta\}-[A^{-1}]^T\int[B]^T[D]\{\varepsilon_0\}\,dV \qquad (3.32)$$

Equation (3.32) relates force to displacement through stiffness matrix [K] determined by Equation (3.31). Defining

$$\{f_0\}=[A^{-1}]^T\int[B]^T[D]\{\varepsilon_0\}\,dV \qquad (3.33)$$

Then Equation (3.32) is written as

$$\{f\}+\{f_0\}=[K]\{\delta\} \qquad (3.34)$$

Thus, the components of {f} are the actual loads on the element, while the $\{f_0\}$ components are fictitious forces that would produce the strain $\{\varepsilon_0\}$. To solve a problem, one calculates $\{f_0\}$ for each element, treats it as a real load, and adds its components to the proper components of the force vector {f}.

After all displacements for the assembly have been determined, those that apply to each element are selected in turn and the stress found by substituting into Equation (3.27)

3-6 The Program to Determine Stresses in a Plate Due to Temperature Change and In-plane Loads

Many of the programmed steps of the method described in this chapter are quite similar to those already described in detail for PT01B in Chapter II. The main program SP23B which solves a wide range of plane stress problems is presented in Appendix 2. While reading the following, frequent reference should be made repeatedly to Figure

(3-4), SP23B Flow Chart, Figure (3-5), Instructions for SP23B Data Deck Preparation, and the computer output as given in the examples of the next section.

Having sufficient data , the program proceeds to calculate for each element, in turn, its stiffness and force that is equivalent to the thermal expansion. These are accumulated in [S] and {F} so that when all elements have been treated the [S] is complete and {F} contains the total thermal equivalent load. As each element is dealt with, the matrix $[A]^{-1}$ is determined and used. Since it is required later in the program when stresses are calculated by Equation (3.27), it is stored in memory. Since it will also be necessary to recall [S], it also is written into storage. External loads are now accepted by reading load cards containing the component number and the magnitude of the force component. These are read until a blank card is encountered which signals the computer to begin reading known displacements. All of those known displacement components must be parallel to the coordinates axes. The {IFX} is used to record which components are known.

In the next phase, guided nodes are dealt with. A card contains the node number, the inclination of the guiding plane, the displacement of the plane normal to its surface, and any load component that is parallel to the surface. As each card is read, {IFX} is altered to indicate that a certain component is known. The **values of the** trigonometric function in Equation (3.12) are determined and are saved in TRIG for further use.

At this stage [K] and some components of {U} and some of {F} are known, as described in Chapter II. The method of Payne and Irons[7] is used to solve for the unknown displacements. However, some of these components where there are guided nodes, refer to locally rotated axes. These are used with data in TRIG to determine the X and Y components by Equation (3.17). Using the restored {U} and the original [S] which is recalled from storage, all {F} components are calculated.

For any element, the stress obtained applies to all points in the element so at the interface between elements there is a step in stress level. Some very good results have been obtained by averaging the stresses of all elements that meet at a node and treating this as the stress at the nodal location. The other simple interpretation scheme is to consider that the stress in the element applies to the location of its centroid. This has the disadvantage of never giving stress at a boundary but boundary stresses can be approximated by some form of extrapolation.

1 PRINT TITLE

READ AND PRINT CASE TITLE

READ AND PRINT; Y.M., P.R., INITIAL TEMP., COEFF. OF EXP.

FILL IN [D] AND [B] (DB02B)

READ COORDINATES OF NODES (RC02B)

READ ELEMENT DATA (CN02B)

ZERO; {F}, {TRIG} AND [S]

REWIND 1

DO FOR ALL ELEMENTS

  CALCULATE NO-LOAD STRAIN

  FILL IN [A] (AB01B)

  INVERT [A] (IN12B)

  FILL IN $[A^{-1}]$ (AI01B)

  WRITE $[A]^{-1}$ ON 1

  CALC. ELEM. STIFFNESS. [E], AND THERMAL FORCE. {FTE} (ES03B)

  ADD [E] TO [S] AND {FTE} TO {F} (AS03B)

REWIND 1 AND 4

WRITE [S] ON 4

REWIND 4

READ KNOWN FORCES (KF02B)

READ KNOWN DISPLACEMENTS (KU01B)

READ GUIDED NODE DATA (GN02B)

SET IFX COMPONENTS (GN02B)

ALTER FOR ROTATED AXES: {F}-{F'} AND [S]-[S'] (GN02B)

SOLVE FOR UNKNOWN COMPONENTS IN {U} (GE02B)

READ [S] FROM 4

RESTORE {Ú}-{U} (RU01B)

SOLVE FOR ALL {F}. PRINT {F} AND {U} (FU10B)

DO FOR ALL ELEMENTS

READ [A] FROM 1

CALC. NO-LOAD STRAIN

CALC. AND PRINT STRESSES (ST03B)

READ NEXT

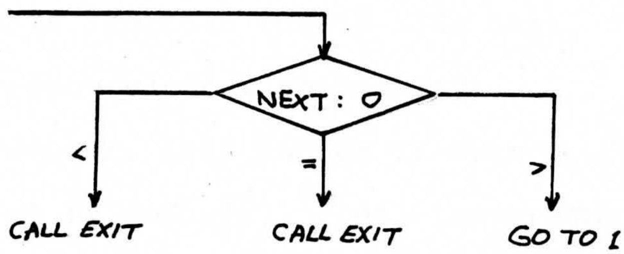


Figure (3-4) SP23B Flow Chart

Data Deck:

A card containing the Case Title.

A card containing physical properties:

Young's modulus, Poisson's ratio, initial temperature, Coefficient of linear thermal expansion. Format: (4F10.4)

One card for each node containing:

Node number, x coordinate, y coordinate. Format: (I5,2F10.5)

A blank card to indicate end of node data.

One card for each element containing:

Element number, Node number at apexes. Thickness of element(=1 by default), Final temperature. Format: (4I5,2F10.5)

A blank card to indicate end of element data.

One card for each known, nonzero load component containing;

Component number(Twice node number minus one for x component or twice for y component), Magnitude of force component. Format: (I5,F10.5)

Note: It is not necessary to read in zero components.

A blank card to indicate end of load data.

One card for each known displacement component containing:

Component number, Displacement. Format: (I5, F10.5)

Note: Zero displacements must be entered.

A blank card to indicate end of displacement data.

One card for each guided node containing:

Node number, A angle (alpha) in degrees defining direction of guiding plane, Normal displacement of guiding plane, load tangential to guiding plane.

Format: (I5, 3F10.5)

Note: Consider a set of axes, x' and y', rotated so that x' is normal to the guiding plane. The angle of the x' axis when measured counter clock-wise from the x axis gives alpha. Normal displacement is positive if the plane moves in the ⊕x' direction. The load is positive if it is in the ⊕y' direction.

A blank card to indicate end of guided node data.

A card containing NEXT. Format: (I5)

To process another case, enter any positive integer and follow by case data deck prepared in accordance with all instruction given above. Use a blank card to CALL EXIT.

Figure (3-5) Instructions for SP23B Data Deck Preparation

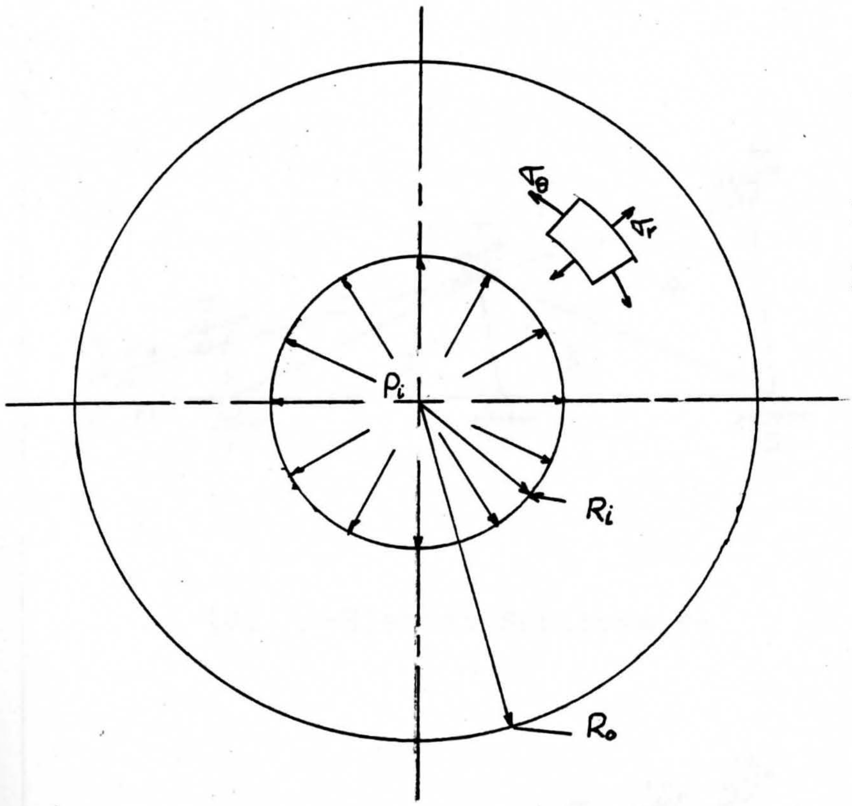
3-7 Example

The example of a thin disc has been selected for this chapter to show the form of the computer output and to give an indication of the accuracy obtainable. The computer analysis is performed with a very few elements and then repeated with a larger number of elements. The results are then compared graphically with the theoretical analysis.

Problem:

For the thin disc shown in Figure (3-6) determine the radial and circumferential stress by the finite element method and compare the results with the theoretical solution.

To solve this problem by the finite element method, first one has to subdivide the problem into elements. In order to save computer time and space, it is worthwhile to take advantage of the polar symmetry of this problem and hence reduce the problem to one of finding the stresses in a wedge with the appropriate boundary conditions. Almost any size of wedge could be chosen, a practical size will have an included angle of 15^{o} . The next step is to choose the number of elements. First the problem is solved with 4 elements and then with 12 elements in order to obtain some appreciation of the resulting increase in accuracy. The subdivision into elements for both cases is shown in Figure (3-7). Since we are using a constant strain triangle, nodal point loads, statically equivalent to the internal pressure, are applied as shown (note $F_1=F_2=1/2(P_i(t)R_i\,\theta_{RAD})$). The boundary conditions are determined from the nature of the problem and



P_i=1,000 psi

E =30x10^{6}psi

t =1.0

R_i =2 in.

R_o =5 in.

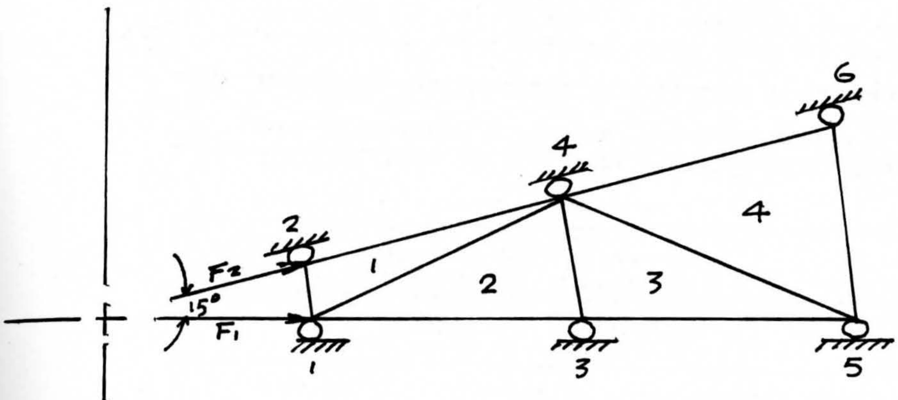
Figure (3-6) The Thin Disc with Constant Internal Pressure

because of symmetry, radial motion along any line of symmetry is permitted. The boundary constraint is therefore: no normal displacement permitted along the sides of the wedge.

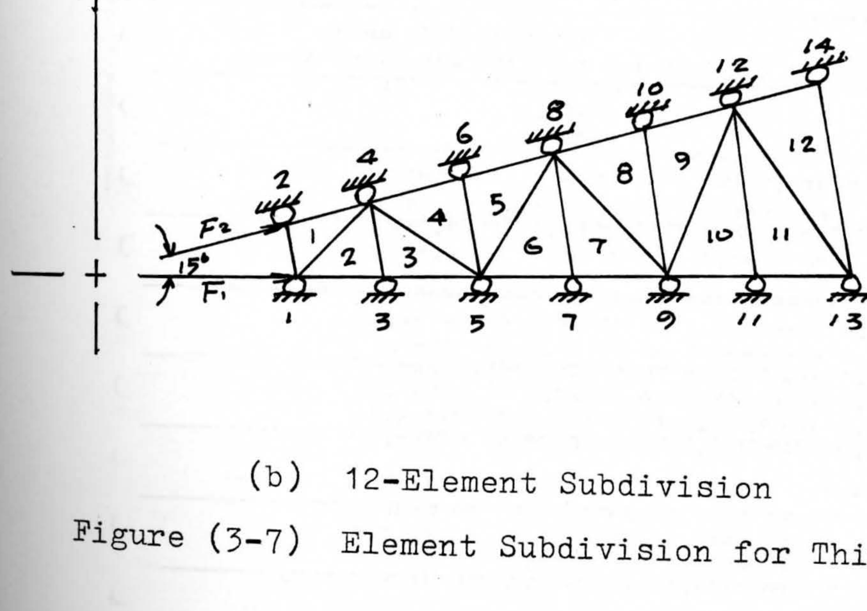
Solutions from computer output for the two cases are shown in Figure (3-8).

Results:

The computer output is shown in Figure (3-8). It is interpreted graphically in Figure (3-9). It is good practice to plot stresses at points on the critical section and to draw a smooth curve. Stress points "zig-zagging" above and below the exact curve is typical of the finite element



(a) 4-Element Subdivision



(b) 12-Element Subdivision

Figure (3-7) Element Subdivision for Thin Disc

solutions, but a smooth curve running between the points usually gives accurate results. If the total load on the critical section is known, it is good practice to check equilibrium. It is easily seen that the Finite Element

```
           MAIN SP23B    SEP  20 1979
STRESSES IN A PLATE DUE TO TEMPERATURE AND IN-PLANE LOADS
BY FINITE ELEMENT METHODS USING CONSTANT STRAIN TRIANGLES

CASE TITLE--- STRESSES IN A DISC WITH INTERNAL PRESSURE (4-ELEMENT SUBDIVIS

YOUNGS MODULUS= 0.300D 08   POISSONS RATIO= 0.300
INITIAL TEMP= 300.0     COEF.OF EXP.= 0.500D-05

NODE NO.    X-COORD      Y-COORD
     1    0.2000D 01   0.0000D 00
     3    0.3500D 01   0.0000D 00
     5    0.5000D 01   0.0000D 00
     2    0.1932D 01   0.5180D 00
     4    0.3381D 01   0.9060D 00
     6    0.4830D 01   0.1294D 01

ELEM.NO.        CONNECTING NODES NUMBERD THICKNESS TEMP
     1            1        4        2         1.00     300.0
     2            1        3        4         1.00     300.0
     3            3        5        4         1.00     300.0
     4            5        6        4         1.00     300.0

BAND WIDTH=      8

     KNOWN NON-ZERO LOADS
COMPONENT NUMBER    LOAD
         1          0.2618D 03

        KNOWN DISPLACEMENTS
COMPONENT NUMBER   DISPLACEMENT
          2          0.0000D 00
          6          0.0000D 00
         10          0.0000D 00

NODE NO.    ALPHA(DEG)      KNOWN U      TANG. FORCE
     2        -75.0       0.000D 00    0.262D 03
     4        -75.0       0.000D 00    0.000D 00
     6        -75.0       0.000D 00    0.000D 00

NODE NO.        FORCE AND DISPLACEMENT COMPONENTS
     1    0.2618D 03-0.9860D 03 0.9421D-04 0.1999D-16
     2    0.3471D 00 0.1011D 04 0.9659D-04 0.2588D-04
     3   -0.8442D-12-0.7498D 03 0.6876D-04 0.1211D-16
     4   -0.1991D 03 0.7430D 03 0.6303D-04 0.1689D-04
     5   -0.2842D-12-0.2535D 03 0.5707D-04 0.1156D-16
     6   -0.6307D 02 0.2354D 03 0.5642D-04 0.1512D-04

ELEM.NO.     SXX          SYY          SXY        THETA       PS1          PS2
     1   -0.307D 03   0.133D 04 -0.201D 03              6.9-0.332D 03 0.136D 04
     2   -0.375D 03   0.447D 03 -0.988D 02              6.8-0.387D 03 0.458D 03
     3   -0.727D 02   0.537D 03 -0.849D 02              7.8-0.843D 02 0.549D 03
     4   -0.308D 02   0.325D 03 -0.607D 02              9.4-0.409D 02 0.335D 03
```

Figure (3-8a) Computer output by SP23B for the 4-Element Case

MAIN SP23B SEP 20 1979

STRESSES IN A PLATE DUE TO TEMPERATURE AND IN-PLANE LOADS
BY FINITE ELEMENT METHODS USING CONSTANT STRAIN TRIANGLES

CASE TITLE--- STRESSES IN A DISC WITH INTERNAL PRESSURE (12-ELEMENT SUBDIVI

YOUNGS MODULUS= 0.3000 08 POISSONS RATIO= 0.300
INITIAL TEMP= 300.0 COEF.OF EXP.= 0.500D-05

NODE NO.	X-COORD	Y-COORD
1	0.2000D 01	0.0000D 00
2	0.1932D 01	0.5180D 00
3	0.2500D 01	0.0000D 00
4	0.2415D 01	0.6470D 00
5	0.3000D 01	0.0000D 00
6	0.2898D 01	0.7760D 00
7	0.3500D 01	0.0000D 00
8	0.3381D 01	0.9060D 00
9	0.4000D 01	0.0000D 00
10	0.3864D 01	0.1035D 01
11	0.4500D 01	0.0000D 00
12	0.4347D 01	0.1165D 01
13	0.5000D 01	0.0000D 00
14	0.4830D 01	0.1294D 01

ELEM.NO.	CONNECTING NODES NUMBERED			THICKNESS	TEMP
1	1	4	2	1.00	300.0
2	1	3	4	1.00	300.0
3	3	5	4	1.00	300.0
4	5	6	4	1.00	300.0
5	5	8	6	1.00	300.0
6	5	7	8	1.00	300.0
7	7	9	8	1.00	300.0
8	9	10	8	1.00	300.0
9	9	12	10	1.00	300.0
10	9	11	12	1.00	300.0
11	11	13	12	1.00	300.0
12	13	14	12	1.00	300.0

BAND WIDTH= 8

KNOWN NON-ZERO LOADS

COMPONENT NUMBER	LOAD
1	0.2618D 03

KNOWN DISPLACEMENTS

COMPONENT NUMBER	DISPLACEMENT
2	0.0000D 00
6	0.0000D 00
10	0.0000D 00
14	0.0000D 00
18	0.0000D 00
22	0.0000D 00
26	0.0000D 00

NODE NO.	ALPHA(DEG)	KNOWN U	TANG. FORCE
2	-75.0	0.0000 00	0.262D 03
4	-75.0	0.0000 00	0.0000 00
6	-75.0	0.0000 00	0.0000 00
8	-75.0	0.0000 00	0.0000 00

```
    10        -75.0        0.000D 00    0.000D 00
    12        -75.0        0.000D 00    0.000D 00
    14        -75.0        0.000D 00    0.000D 00
NODE NO.       FORCE AND DISPLACEMENT COMPONENTS
         1    0.2618D 03-0.3442D 03 0.1086D-03 0.1493D-16
         2    0.1645D 03 0.3984D 03 0.1076D-03 0.2882D-04
         3   -0.4078D-12-0.4681D 03 0.9301D-04 0.1138D-16
         4   -0.1220D 03 0.4552D 03 0.8817D-04 0.2362D-04
         5   -0.3833D-11-0.3572D 03 0.8043D-04 0.9130D-17
         6   -0.9230D 02 0.3445D 03 0.7865D-04 0.2108D-04
         7   -0.4419D-12-0.2873D 03 0.7389D-04 0.7158D-17
         8   -0.7410D 02 0.2766D 03 0.7070D-04 0.1894D-04
         9   -0.1926D-11-0.2418D 03 0.6832D-04 0.5998D-17
        10   -0.6291D 02 0.2348D 03 0.6642D-04 0.1780D-04
        11   -0.1106D-11-0.2119D 03 0.6532D-04 0.5002D-17
        12   -0.5450D 02 0.2034D 03 0.6276D-04 0.1682D-04
        13   -0.3411D-12-0.7898D 02 0.6276D-04 0.3911D-17
        14   -0.2051D 02 0.7655D 02 0.6084D-04 0.1630D-04
ELEM.NO.     SXX          SYY          SXY        THETA     PS1          PS2
         1  -0.744D 03   0.135D 04 -0.367D 03             9.7-0.806D 03 0.141D 04
         2  -0.666D 03   0.896D 03 -0.134D 03             4.9-0.678D 03 0.907D 03
         3  -0.468D 03   0.955D 03 -0.124D 03             5.0-0.479D 03 0.966D 03
         4  -0.355D 03   0.661D 03 -0.194D 03            10.5-0.391D 03 0.696D 03
         5  -0.251D 03   0.695D 03 -0.180D 03            10.4-0.284D 03 0.728D 03
         6  -0.225D 03   0.560D 03 -0.604D 02             4.4-0.230D 03 0.564D 03
         7  -0.160D 03   0.579D 03 -0.575D 02             4.4-0.165D 03 0.584D 03
         8  -0.105D 03   0.458D 03 -0.111D 03            10.8-0.126D 03 0.479D 03
         9  -0.642D 02   0.471D 03 -0.106D 03            10.8-0.844D 02 0.491D 03
        10  -0.548D 02   0.417D 03 -0.345D 02             4.2-0.574D 02 0.419D 03
        11  -0.260D 02   0.425D 03 -0.332D 02             4.2-0.284D 02 0.428D 03
        12   0.504D 01   0.363D 03 -0.718D 02            10.9-0.884D 01 0.376D 03
```

Figure (3-8b) Computer Output by SP23B for the 12-Element Case

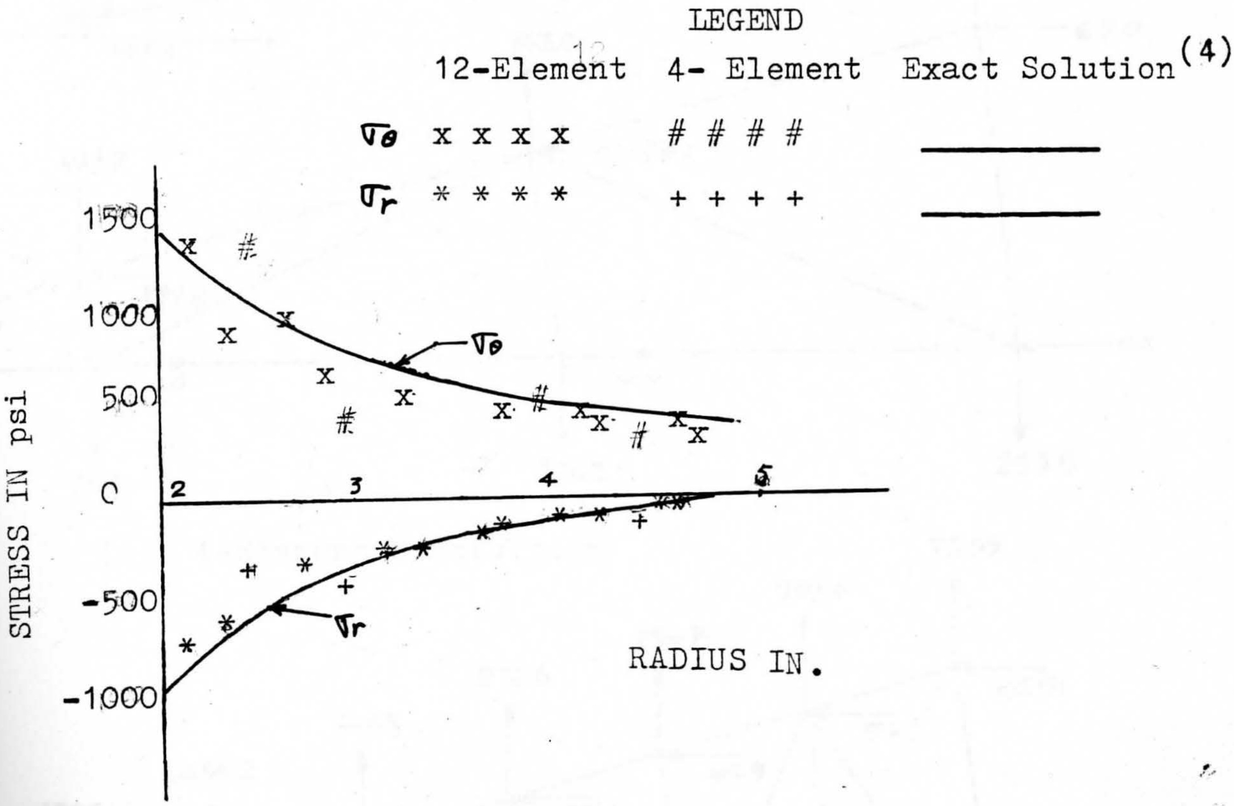
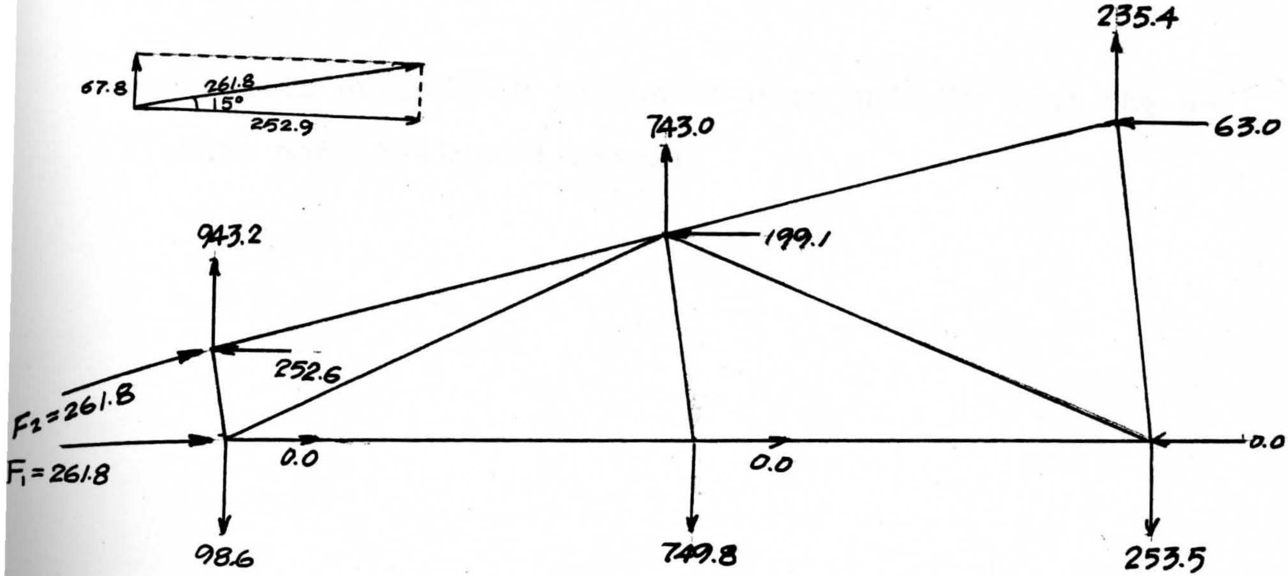
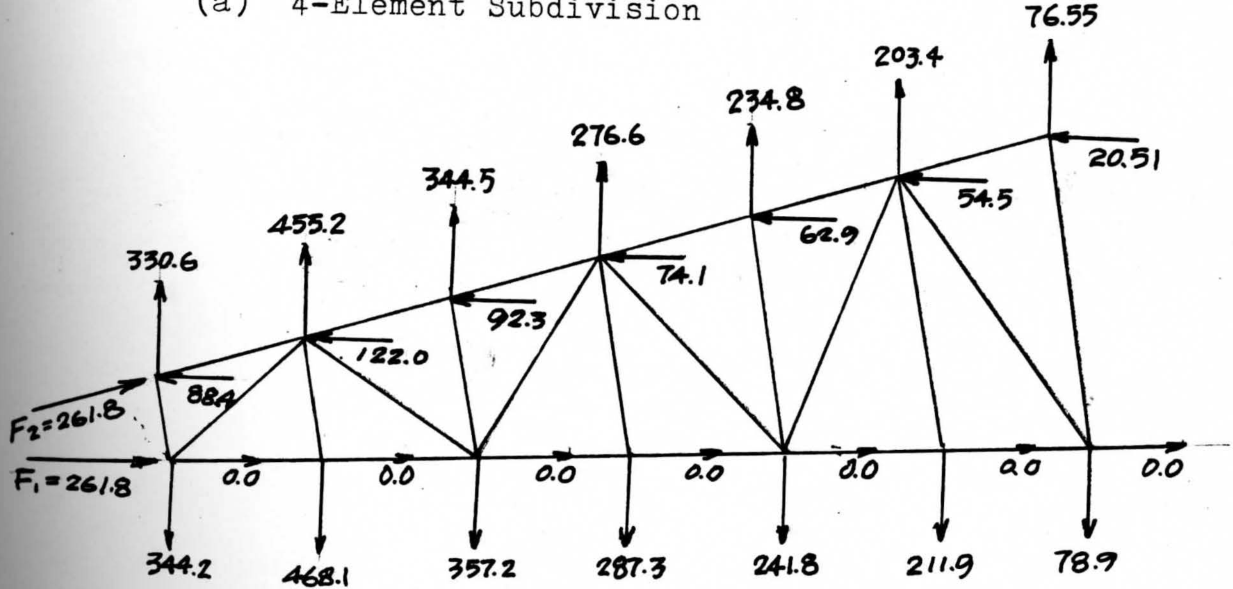


Figure (3-9) Stress in a Disc with Internal Pressure the method would give better results with a further moderate increase in the number of elements. The sketches of the final nodal force components for both 4-element and 12-element subdivisions are shown in Figures (3-10a) and (3-10b). Both diagrams satisfy the conditions of equilibrium. The force distribution on the horizontal edge of the 12-element subdivision is more close to the theoretical results than that of 4-elment subdivision based on two observations. Firstly, it possesses a more uniform distribution, and secondly, the change in magni-



(a) 4-Element Subdivision



(b) 12-Element Subdivision

Figure (3-10) The Diagram of Final Nodal Forces

tudes of adjacent force components is greatly reduced. Also, it should be noted that the zero force components in x-direction along the horizontal boundary match up with the assumed support condition on that boundary. That is, the roller support is placed at each node in both the 4-element and 12-element cases.

CHAPTER IV

THE LINEAR STRAIN TRIANGULAR ELEMENT

4-1 Displacement Function of Linear Strain Triangles

By using a more advanced element, in which the stress varies within the elements,more accurate results may be obtained with the same number of elements of constant strain triangles (CST). This is observed by comparing the numerical results of two different types of elements utilized in examples illustrated in this chapter.

For the condition in which the strain, and hence the stress, varies linearly, the displacement function is assumed in the matrix form as

$$\begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} 1 & x & y & x^2 & xy & y^2 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & x & y & x^2 & xy & y^2 \end{bmatrix} \begin{Bmatrix} \alpha_1 \\ \vdots \\ \alpha_{12} \end{Bmatrix}$$

or,

$$\begin{Bmatrix} u \\ v \end{Bmatrix} = [P]\{\alpha\} \tag{4.1}$$

The twelve unknown coefficients in $\{\alpha\}$ are determined by applying the boundary conditions at the node points of the element. This is accomplished by introducing nodes at the midpoints of the element sides as shown in Figure (4-1). To perform the intergration in the stiffness formula (3.31) more easily, the local axes are established

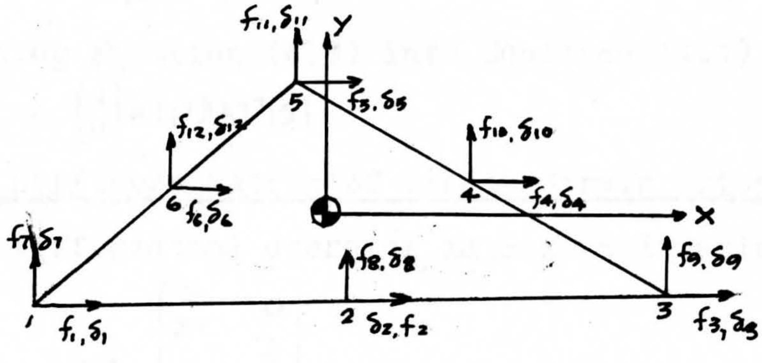


Figure (4-1) Nodal Force and Displacement Components of Linear Strain Triangular Element

with origin at the centroid of the element. Using Equation (3.1) and applying the boundary conditions at the six node points, one obtains

$$\begin{Bmatrix} \delta_1 \\ \delta_2 \\ \delta_3 \\ \delta_4 \\ \delta_5 \\ \delta_6 \\ \delta_7 \\ \delta_8 \\ \delta_9 \\ \delta_{10} \\ \delta_{11} \\ \delta_{12} \end{Bmatrix} = \begin{bmatrix} 1 & x_1 & y_1 & x_1^2 & x_1y_1 & y_1^2 & 0 & 0 & 0 & 0 & 0 & 0 \\ 1 & x_2 & y_2 & x_2^2 & x_2y_2 & y_2^2 & 0 & 0 & 0 & 0 & 0 & 0 \\ 1 & x_3 & y_3 & x_3^2 & x_3y_3 & y_3^2 & 0 & 0 & 0 & 0 & 0 & 0 \\ 1 & x_4 & y_4 & x_4^2 & x_4y_4 & y_4^2 & 0 & 0 & 0 & 0 & 0 & 0 \\ 1 & x_5 & y_5 & x_5^2 & x_5y_5 & y_5^2 & 0 & 0 & 0 & 0 & 0 & 0 \\ 1 & x_6 & y_6 & x_6^2 & x_6y_6 & y_6^2 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & x_1 & y_1 & x_1^2 & x_1y_1 & y_1^2 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & x_2 & y_2 & x_2^2 & x_2y_2 & y_2^2 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & x_3 & y_3 & x_3^2 & x_3y_3 & y_3^2 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & x_4 & y_4 & x_4^2 & x_4y_4 & y_4^2 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & x_5 & y_5 & x_5^2 & x_5y_5 & y_5^2 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & x_6 & y_6 & x_6^2 & x_6y_6 & y_6^2 \end{bmatrix} \begin{Bmatrix} \alpha_1 \\ \alpha_2 \\ \alpha_3 \\ \alpha_4 \\ \alpha_5 \\ \alpha_6 \\ \alpha_7 \\ \alpha_8 \\ \alpha_9 \\ \alpha_{10} \\ \alpha_{11} \\ \alpha_{12} \end{Bmatrix} \qquad (4.2)$$

or,

$$\{\delta\} = [A]\{\alpha\} \qquad (4.3)$$

Inversion of the latter equation yields

$$\{\alpha\} = [A]^{-1}\{\delta\} \tag{4.4}$$

Substituting Equation (4.4) into Equation (4.1) yields

$$\begin{Bmatrix} u \\ v \end{Bmatrix} = [P][A]^{-1}\{\delta\} \tag{4.5}$$

4-2 <u>The Stiffness Matrix of Linear Strain Triangle</u>

The differential operator matrix in Equation (3.25) is

$$[\Delta] = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix}$$

From Equation (3.26) and the polynomial matrix given in Equation (4.1), one obtains

$$[B] = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \begin{bmatrix} 1 & x & y & x^2 & xy & y^2 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & x & y & x^2 & xy & y^2 \end{bmatrix}$$

$$= \begin{bmatrix} 0 & 1 & 0 & 2x & y & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & x & 2x \\ 0 & 0 & 1 & 0 & x & 2y & 0 & 1 & 0 & 2x & y & 0 \end{bmatrix} \tag{4.6}$$

With [D] given by Equation (3.8), all matrices in the stiffness Equation (3.31) are known. The increment of volume dV for an element of uniform thickness, t, is written as

$$dV = t \cdot dA$$

and Equation (3.31) becomes

$$[K] = [A^{-1}]^T t \cdot \int_A [B]^T [D][B]\, dA\, [A]^{-1} \tag{4.7}$$

Therefore, one is able to evaluate the stiffness matrix for any element using the elastic constants of the material, the element thickness, and the coordinates of the nodes based on axes having origin at the centroid of the element.

4-3 Stresses in the Linear Strain Triangle

The stiffness matrix of each element is accumulated in turn as in the cases described in Chapter II and Chapter III to give a stiffness matrix for the assemblage of elements. Loads and constraints are applied as before and all displacements, $\{U\}$, are found. Taking each element in turn, the values in $\{\delta\}$ are selected from $\{U\}$ and used to calculate stress by

$$\{\sigma\} = [D][B][A]^{-1}\{\delta\} \qquad (4.8)$$

The coordinate variables in [B] are all linear in functions x and y, hence, the stress varies linearly within the element. Thus, when solving Equation (4.7) the point within the element at which the stress is to be calculated must be indicated by specifying its location through x and y values inserted into [B].

4-4 Consistent Load Vector

A distributed load acting on an element edge is more difficult to deal with than a concentrated load which is handled simply by placing a node at the point of load application. In the CST, a distributed load could be treated as concentrated loads at the two end nodes by making two systems statically equivalent. But with Linear Strain Triangle (LST), there are three nodes on any edge and the principle of static equivalence is not sufficient to determine the distribution uniquely. The solution of this problem is carried out by using energy principles.

Consider an element boundary with nodes 1,2,3 shown in Figure (4-2). Let the boundary be parallel to the y axis which defines all x as constants in the horizontal displacement function. Therefore, u is a parabolic function in y only. Any new edge location may be taken as 1'-2'-3' as shown by the broken lines. The edge displacement is given as

$$U = AY^2 + BY^2 + C = \{Y^2 \quad Y \quad 1\} \begin{Bmatrix} A \\ B \\ C \end{Bmatrix}$$

When this displacement accurs, the distributed edge load, p, does work on the system. Over an increment of edge distance dy, the work done is

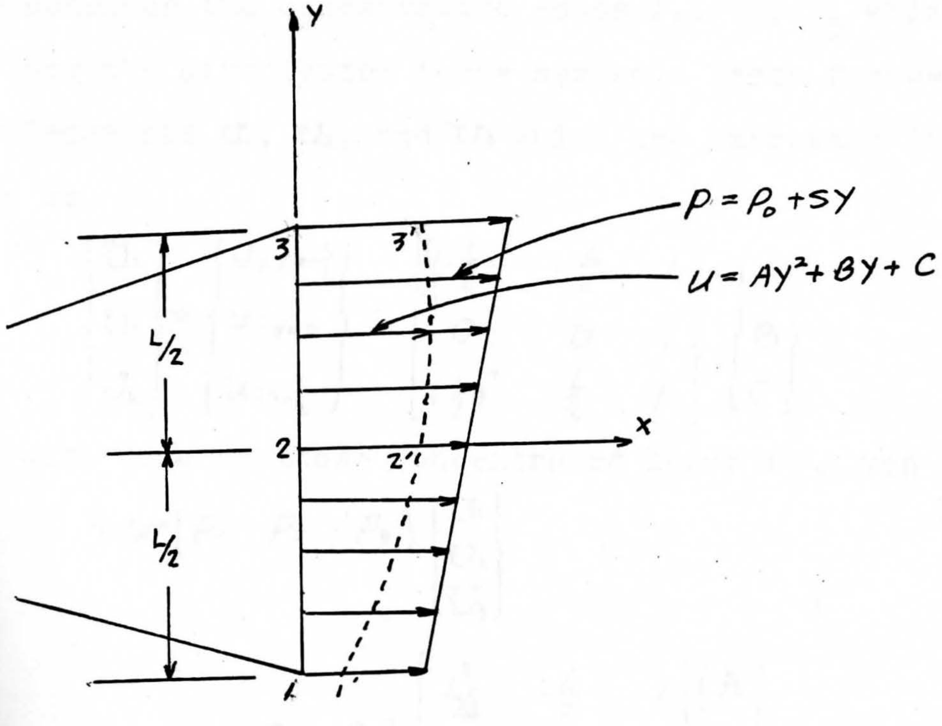
$$dW = PUt \cdot dY$$

$$= (P_o + SY)\{Y^2 \quad Y \quad 1\} \begin{Bmatrix} A \\ B \\ C \end{Bmatrix} \cdot t \cdot dY$$

$$= \{P_o Y^2 + SY^3 \quad P_o Y + SY^2 \quad P_o + SY\} \begin{Bmatrix} A \\ B \\ C \end{Bmatrix} \cdot t \cdot dY$$

and the work done over the entire edge is

$$W_P = \int_{-L/2}^{L/2} dW_P = t\{P_o L^3/12 \quad SL^3/12 \quad P_o L\} \begin{Bmatrix} A \\ B \\ C \end{Bmatrix}$$



(a) Distributed Load and Edge Displacement

(b) Concentrated Loads and Nodal Displacement

Figure (4-2) Distributed Load and Equivalent Consistent Load Vector

Now consider the concentrated loads P_1, P_2, P_3 which would replace the distributed force system. These forces produce displacements U_1, U_2, and U_3 which are expressed in matrix form as

$$\begin{Bmatrix} U_1 \\ U_2 \\ U_3 \end{Bmatrix} = \begin{Bmatrix} u_{1,y=-\frac{L}{2}} \\ u_{2\,y=0} \\ u_{3\,y=\frac{L}{2}} \end{Bmatrix} = \begin{bmatrix} (-\frac{L}{2})^2 & -\frac{L}{2} & 1 \\ 0 & 0 & 1 \\ (\frac{L}{2})^2 & \frac{L}{2} & 1 \end{bmatrix} \begin{Bmatrix} A \\ B \\ C \end{Bmatrix}$$

The work done by those concentrated loads is given by

$$W_P = \{P_1 \quad P_2 \quad P_3\} \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \end{Bmatrix}$$

$$= \{P_1 \quad P_2 \quad P_3\} \begin{bmatrix} L^2/4 & -\frac{L}{2} & 1 \\ 0 & 0 & 1 \\ L^2/4 & \frac{L}{2} & 1 \end{bmatrix} \begin{Bmatrix} A \\ B \\ C \end{Bmatrix}$$

Equating the work done by two equivalent loading systems gives

$$t\{P_oL^3/12 \quad SL^3/12 \quad P_oL\} \begin{Bmatrix} A \\ B \\ C \end{Bmatrix} = \{P_1 \quad P_2 \quad P_3\} \begin{bmatrix} L^2/4 & -L/2 & 1 \\ 0 & 0 & 1 \\ L^2/4 & L/2 & 1 \end{bmatrix} \begin{Bmatrix} A \\ B \\ C \end{Bmatrix}$$

Postmultiplying both sides of the latter equation by the inverse of the coefficient matrix

$$\begin{bmatrix} L^2/4 & -L/2 & 1 \\ 0 & 0 & 1 \\ L^2/4 & L/2 & 1 \end{bmatrix}$$

one obtains

$$\{P_1 \quad P_2 \quad P_3\} = t\,\{P_oL/6 - SL^2/12 \quad 2P_oL/3 \quad P_oL/6 + SL^2/12\}$$

Hence,

$$P_1 = (tL/6)(P_0 - SL/2)$$
$$P_2 = (2tL/3)P_0 \qquad (4.9)$$
$$P_3 = (tL/6)(P_0 + SL/2)$$

where tL is the edge area

P_1, P_2, and P_3 are the components of the consistent load vector and represent the intensity of the distributed load at points 1,2, and 3. They do the same amount of work during the displacements as the distributed force system.

4-5 The Program to Determine Stresses in a Plate by Linear Strain Triangles

Main Program SP33B solves in-plane loaded plate problems by using linear strain triangles. While reading this section, reference should be made to Figure (4-3), SP33B Flow Chart; SP33B Fortran program; Appendix 3; Figure (4-4), Instruction for SP33B Data Preparation, and the computer output as given in the examples. As the data cards for each node and element are read, the coordinates are recalculated with the origin at the centroid of the element. The stiffness matrix of each element is calculated as described earlier and the general stiffness matrix formed by accumulation. Then the known forces are read from the cards, concentrated loads are entered directly; but for distributed load, it must be transferred to the consistent load vector components from Equation (4.9).

The known displacements and guided nodes are treated as before and the unknowns are solved as previously. Stresses are determined at the location of centroid of the element by using Equation (4.8), where all the values of x and y in B are taken as zero. This gives the stresses at the centroid of the element without any other option for locating the stress.

1 PRINT TITLE

READ AND PRINT CASE TITLE

READ AND PRINT YOUNG'S MODULUS AND POISSON'S RATIO

SET CONSTANTS IN [D], [B] AND [D][B][D] (DB08B)

CALCULATE [D][B] AT CENTROID (RC06B)

READ COORDINATES OF NODES (CN06B)

READ ELEMENT DATA

ZERO {TRIG} [S]

REWIND 1

DO FOR ALL ELEMENTS

DETERMINE $[A]^{-1}$ AND {X}, {Y} FOR ORIGIN AT CENTROID (AI02B)

WRITE $[A]^{-1}$ ON 1 (IN12B)

DETERMINE ELEMENT STIFFNESS, [E] (ES06B)

ADD [E] TO [S] (AS06B)

REWIND 4

WRITE [S] ON 4

2 READ KNOWN FORCES (KF01B)

READ KNOWN DISPLACEMENTS (KU01B)

READ; GUIDED NODE NO., α, DISP., FORCE, (GN02B)

SET {IFX} COMPONENT (GN02B)

ALTER, FOR ROTATED AXES (GN02B)

SOLVE FOR UNKNOWN COMPONENT IN U' (GE02B)

REWIND 4

READ [S] FROM 4

RESTORE $\{U\} \rightarrow \{U\}$ (RU01B)

SOLVE FOR ALL FORCE COMPONENTS PRINT {F} AND {U} (RU10B)

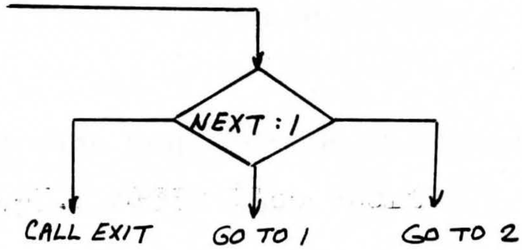
REWIND 1

DO FOR ALL ELEMENTS

READ $[A]^{-1}$ FROM 1

CALCULATE AND PRINT STRESSES AT CENTROID (ST06B)

READ NEXT



Figure(4-3) SP33B Flow Chart

Data Deck:

A card containing the Case Title.

A card containing physical properties: Young's Modulus and Poisson's Ratio, Format: (2F10.5)

A card containing the number of nodes in the system. Format: (I5)

One card for each node locate at the corner of a triangle containing;

Node number, x coordinate, y coordinate. Format: (I5, 2F10.5)

A blank card to indicate end of node data.

One card for each element containing:

Element number, thickness of element (=1 by default), node numbers on sides of triangle, starting at a corner node counterclockwise. Format: (I5, F10.5, 6I5).

A blank card to indicate end of element data

One card for each known, nonzero load component containing:

Component number, magnitude of force Format: (I5, F10.5)

A blank card to indicate end of load data.

One card for each known displacement component containing:

Component number, displacement, Format: (I5, F10.5)

A blank card to indicate end of displacement data.

One card for each guided node, containing; Node number, angle (alpha) in degrees defining direction of guiding plane, normal displacement of guiding plane, load tangential to guiding plane. Format: (I5, 3F10.5)

A blank card to indicate end of guided node data.

A card containing NEXT. Format (I5)

NEXT=0; End of job

NEXT=1; Execute program again starting a new case; follow by an entire deck prepared in accordance with the above instructions starting with the case title.

NEXT=2; Repeat the case completed but with a new set of known loads and displacements. Follow by force data cards and all following cards described above.

Figure (4-4) Instructions for SP33B Data Deck Preparation

4-6 Example

The finite element results are expected to be more accurate when LST is used as compared to the results using the CST. For this reason, the sample example for this chapter is the same as that of Chapter III. This enables a comparison of LST and CST with the theoretical results.

Problem:

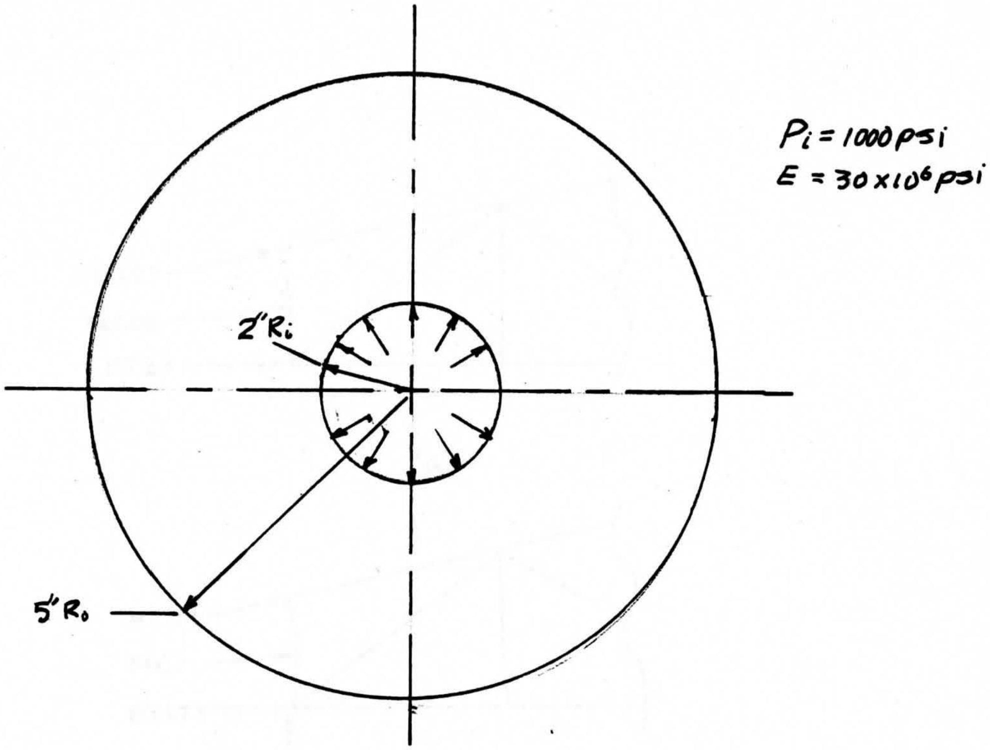
Determine the radial and circumferential stresses by the Finite Element Method for the thin disc shown in Figure (4-5), using a linear strain triangle. Compare the results with those found using the constant strain triangle of Chapter III, and with theoretical results.

Solutions:

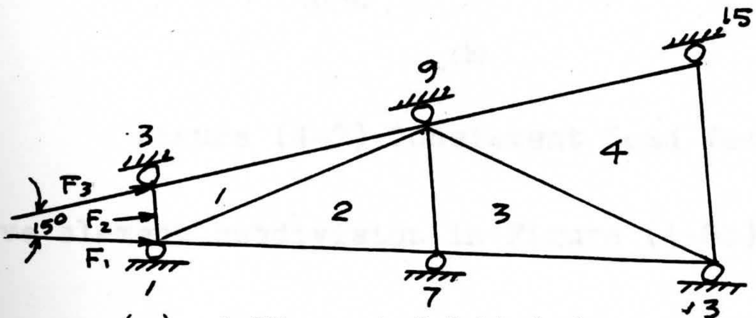
The element subdivision of the disc is made identically to the subdivision used for the example in Chapter III as shown in Figure (4-6) to provide a means of comparison. .

The consistent nodal point loads as calculated using Equation (4.19) are shown in Figure (4-7a). The x and y components, excepting for the load tangential to the guiding plane, are read into the computer in the normal manner. The nodal loads tangent to the guiding plane are read in separately under the tangential force listing. The actual components read into the computer are shown in Figure (4-7b).

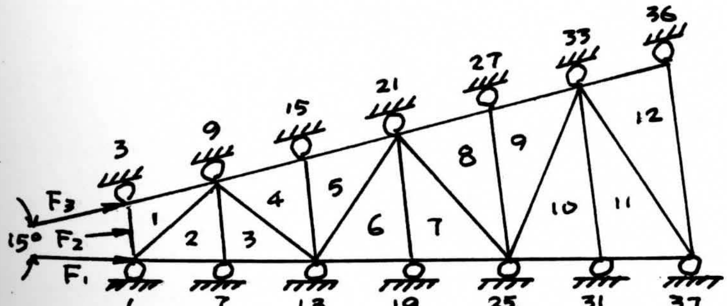
The complete input-output information for the four element solution is given in Figure (4-8a), and for the



Figure(4-5) Thin Disc with Internal Pressure



(a) 4-Element Subdivision



(b) 12-Element Subdivision

Fighre(4-6) Element Subdivision for Thin Disc

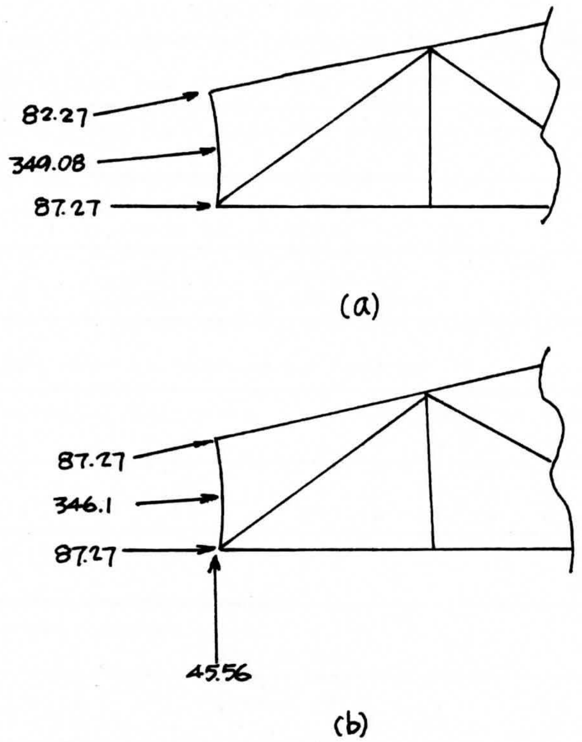


Figure (4-7) Consistent Load Vector

twelve element subdivision in Figure (4-8b).

MAIN SP33B   SEP 25,1979

STRESSES IN IN-PLANE LOADED PLATE USING LINEAR STRAIN TRIANGLES (GUIDED NODES)

CASE TITLE --- STRESSES IN A DISC WITH INTERNAL PRESSURE

YOUNGS MODULUS= 0.300D 08   POISSONS RATIO= 0.300

NODE NO.	X-COORD	Y-COORD
1	0.2000D 01	0.0000D 00
7	0.3500D 01	0.0000D 00
13	0.5000D 01	0.0000D 00
3	0.1932D 01	0.5180D 00
9	0.3381D 01	0.9060D 00
15	0.4830D 01	0.1294D 01

SYSTEM HAS 30 DEGREES OF FREEDOM

ELEM.NO.	THICKNESS	CONNECTING NODE NUMBERS					
1	1.00	1	5	9	6	3	2
2	1.00	1	4	7	8	9	5
3	1.00	7	10	13	11	9	8
4	1.00	13	14	15	12	9	11

BAND WIDTH= 18

KNOWN NON-ZERO LOADS

COMPONENT NUMBER	LOAD
1	0.8727D 02
3	0.3461D 03
4	0.4556D 02

KNOWN DISPLACEMENTS

COMPONENT NUMBER	DISPLACEMENT
2	0.0000D 00
8	0.0000D 00
14	0.0000D 00
20	0.0000D 00
26	0.0000D 00

NODE NO.	ALPHA(DEG)	KNOWN U	TANG. FORCE
3	-75.0	0.000D 00	0.873D 02
6	-75.0	0.000D 00	0.000D 00
9	-75.0	0.000D 00	0.000D 00
12	-75.0	0.000D 00	0.000D 00
15	-75.0	0.000D 00	0.000D 00

NODE NO.	FORCE AND DISPLACEMENT COMPONENTS			
1	0.8727D 02	-0.3649D 03	0.1101D-03	0.7400D-17
2	0.3461D 03	0.4556D 02	0.1107D-03	0.1484D-04
3	-0.9740D 01	0.3737D 03	0.1082D-03	0.2898D-04
4	-0.3149D-12	-0.8047D 03	0.8710D-04	0.1048D-16
5	-0.5627D-11	-0.1364D-11	0.8597D-04	0.1437D-04
6	-0.2105D 03	0.7856D 03	0.8342D-04	0.2235D-04
7	-0.1693D-12	-0.2978D 03	0.7388D-04	0.4811D-17
8	-0.6318D-11	-0.1865D-11	0.7373D-04	0.9661D-05
9	-0.7407D 02	0.2764D 03	0.7161D-04	0.1919D-04
10	-0.4313D-12	-0.4485D 03	0.6690D-04	0.5079D-17
11	-0.4624D-11	0.2309D-12	0.6691D-04	0.7234D-05
12	-0.1171D 03	0.4370D 03	0.6491D-04	0.1739D-04
13	0.4547D-12	-0.8428D 02	0.6325D-04	0.3842D-17

14	0.6416D-12	0.3483D-13	0.6280D-04	0.8288D-05
15	-0.2195D 02	0.8194D 02	0.6087D-04	0.1631D-04

ELEM.NO.	SXX	SYY	SXY	THETA	PS1	PS2
1	-0.555D 03	0.959D 03	-0.280D 03	10.1	-0.605D 03	0.101D 04
2	-0.351D 03	0.724D 03	-0.106D 03	5.6	-0.361D 03	0.735D 03
3	-0.105D 03	0.491D 03	-0.470D 02	4.5	-0.109D 03	0.495D 03
4	-0.359D 02	0.416D 03	-0.793D 02	9.7	-0.495D 02	0.429D 03

Figure (4-8a) Computer Output by SP33B for the 4-Element Case

68	0.0000D 00
74	0.0000D 00

NODE NO.	ALPHA(DEG)	KNOWN U	TANG. FORCE
3	-75.0	0.000D 00	0.873D 02
6	-75.0	0.000D 00	0.000D 00
9	-75.0	0.000D 00	0.000D 00
12	-75.0	0.000D 00	0.000D 00
15	-75.0	0.000D 00	0.000D 00
18	-75.0	0.000D 00	0.000D 00
21	-75.0	0.000D 00	0.000D 00
24	-75.0	0.000D 00	0.000D 00
27	-75.0	0.000D 00	0.000D 00
30	-75.0	0.000D 00	0.000D 00
33	-75.0	0.000D 00	0.000D 00
36	-75.0	0.000D 00	0.000D 00
39	-75.0	0.000D 00	0.000D 00

NODE NO.	FORCE AND DISPLACEMENT COMPONENTS			
1	0.8727D 02	-0.1294D 03	0.1115D-03	0.5611D-17
2	0.3461D 03	0.4556D 02	0.1113D-03	0.1477D-04
3	0.5057D 02	0.1486D 03	0.1079D-03	0.2892D-04
4	-0.3491D-11	-0.3748D 03	0.1013D-03	0.7636D-17
5	-0.7901D-11	-0.6253D-12	0.1008D-03	0.1476D-04
6	-0.9729D 02	0.3631D 03	0.9783D-04	0.2621D-04
7	-0.1066D-11	-0.1615D 03	0.9322D-04	0.3925D-17
8	-0.4937D-11	0.1495D-12	0.9303D-04	0.1215D-04
9	-0.4052D 02	0.1512D 03	0.9011D-04	0.2415D-04
10	-0.4092D-11	-0.2719D 03	0.8686D-04	0.4485D-17
11	-0.4886D-11	0.5530D-12	0.8680D-04	0.1037D-04
12	-0.7049D 02	0.2631D 03	0.8399D-04	0.2250D-04
13	-0.2640D-11	-0.1180D 03	0.8181D-04	0.3016D-17
14	-0.5186D-11	-0.4617D-12	0.8154D-04	0.1080D-04
15	-0.3166D 02	0.1182D 03	0.7901D-04	0.2117D-04
16	-0.2215D-11	-0.2130D 03	0.7767D-04	0.4463D-17
17	-0.4604D-11	-0.4547D-12	0.7728D-04	0.1097D-04
18	-0.5514D 02	0.2058D 03	0.7499D-04	0.2009D-04
19	-0.3586D-11	-0.9832D 02	0.7426D-04	0.2450D-17
20	-0.1644D-11	-0.5512D-13	0.7398D-04	0.9687D-05
21	-0.2454D 02	0.9159D 02	0.7174D-04	0.1922D-04
22	-0.4471D-11	-0.1759D 03	0.7146D-04	0.2967D-17
23	-0.2567D-11	0.1295D-12	0.7125D-04	0.8747D-05
24	-0.4553D 02	0.1699D 03	0.6904D-04	0.1850D-04
25	-0.5894D-11	-0.8008D 02	0.6918D-04	0.1987D-17
26	-0.5491D-11	-0.2634D-12	0.6884D-04	0.9101D-05
27	-0.2148D 02	0.8018D 02	0.6680D-04	0.1790D-04
28	-0.1494D-11	-0.1512D 03	0.6727D-04	0.2981D-17
29	-0.4249D-11	0.3979D-12	0.6686D-04	0.9331D-05
30	-0.3912D 02	0.1460D 03	0.6496D-04	0.1741D-04
31	-0.5121D-11	-0.7228D 02	0.6569D-04	0.1706D-17
32	-0.4619D-11	0.7980D-12	0.6536D-04	0.8577D-05
33	-0.1813D 02	0.6765D 02	0.6345D-04	0.1700D-04
34	-0.4924D-11	-0.1338D 03	0.6440D-04	0.2149D-17
35	-0.3569D-11	0.6630D-12	0.6412D-04	0.7992D-05
36	-0.3467D 02	0.1294D 03	0.6218D-04	0.1666D-04
37	-0.1705D-12	-0.2014D 02	0.6334D-04	0.9972D-18
38	-0.2930D-11	-0.4918D-12	0.6301D-04	0.8315D-05
39	-0.5355D 01	0.1998D 02	0.6114D-04	0.1638D-04

ELEM.NO.	SXX	SYY	SXY	THETA	PS1	PS2

MAIN SP33B   SEP 25,1979

STRESSES IN IN-PLANE LOADED PLATE USING LINEAR STRAIN TRIANGLES   (GUIDED NODES)

CASE TITLE --- STRESSES IN A DISC WITH INTERNAL PRESSURE (12-ELEMENT SUBDIVISIO

YOUNGS MODULUS= 0.300D 08   POISSONS RATIO= 0.300

NODE NO.	X-COORD	Y-COORD
1	0.2000D 01	0.0000D 00
7	0.2500D 01	0.0000D 00
13	0.3000D 01	0.0000D 00
19	0.3500D 01	0.0000D 00
25	0.4000D 01	0.0000D 00
31	0.4500D 01	0.0000D 00
37	0.5000D 01	0.0000D 00
3	0.1932D 01	0.5180D 00
9	0.2415D 01	0.6470D 00
15	0.2898D 01	0.7760D 00
21	0.3381D 01	0.9060D 00
27	0.3864D 01	0.1035D 01
33	0.4347D 01	0.1165D 01
39	0.4830D 01	0.1294D 01

SYSTEM HAS 78 DEGREES OF FREEDOM

ELEM.NO.	THICKNESS	CONNECTING NODE NUMBERS					
1	1.00	1	5	9	6	3	2
2	1.00	1	4	7	8	9	5
3	1.00	7	10	13	11	9	8
4	1.00	13	14	15	12	9	11
5	1.00	13	17	21	18	15	14
6	1.00	13	16	19	20	21	17
7	1.00	19	22	25	23	21	20
8	1.00	25	26	27	24	21	23
9	1.00	25	29	33	30	27	26
10	1.00	25	28	31	32	33	29
11	1.00	31	34	37	35	33	32
12	1.00	37	38	39	36	33	35

BAND WIDTH=   18

KNOWN NON-ZERO LOADS

COMPONENT NUMBER	LOAD
1	0.8727D 02
3	0.3461D 03
4	0.4556D 02

KNOWN DISPLACEMENTS

COMPONENT NUMBER	DISPLAC EMENT
2	0.0000D 00
8	0.0000D 00
14	0.0000D 00
20	0.0000D 00
26	0.0000D 00
32	0.0000D 00
38	0.0000D 00
44	0.0000D 00
50	0.0000D 00
56	0.0000D 00
62	0.0000D 00

1	-0.767D 03	0.115D 04	-0.352D 03	10.1	-0.830D 03	0.121D 04
2	-0.674D 03	0.106D 04	-0.160D 03	5.2	-0.688D 03	0.107D 04
3	-0.470D 03	0.855D 03	-0.111D 03	4.8	-0.480D 03	0.864D 03
4	-0.371D 03	0.753D 03	-0.199D 03	9.7	-0.405D 03	0.787D 03
5	-0.254D 03	0.638D 03	-0.166D 03	10.2	-0.284D 03	0.668D 03
6	-0.231D 03	0.614D 03	-0.779D 02	5.2	-0.238D 03	0.621D 03
7	-0.158D 03	0.541D 03	-0.595D 02	4.8	-0.163D 03	0.546D 03
8	-0.114D 03	0.497D 03	-0.109D 03	9.8	-0.133D 03	0.516D 03
9	-0.656D 02	0.449D 03	-0.956D 02	10.2	-0.828D 02	0.466D 03
10	-0.579D 02	0.441D 03	-0.456D 02	5.2	-0.621D 02	0.445D 03
11	-0.239D 02	0.407D 03	-0.364D 02	4.8	-0.270D 02	0.410D 03
12	-0.983D-01	0.383D 03	-0.693D 02	9.9	-0.123D 02	0.395D 03

```
STATEMENTS EXECUTED=   177104

CORE USAGE        OBJECT CODE=   30120 BYTES,ARRAY AREA=   85656 BYTES,TOTAL AREA

DIAGNOSTICS        NUMBER OF ERRORS=        0, NUMBER OF WARNINGS=       0, NUMBER

COMPILE TIME=     0.83 SEC,EXECUTION TIME=      5.18 SEC,      15.06.02     TUESDAY

          C$STOP
```

Figure (4-8b) Computer Output By Sp33B for the 12-Element Case

Results:

The computer output for this example is shown and interpreted graphically in the figures that follow. Referring to Figure (4-9a) and (4-9b), the exact elasticity solutions are shown in solid lines for normal stress and . Results of the four elements and twelve elements solutions are superimposed on the exact values. The four CST elements yields solutions which are from a practical standpoint inaccurate. The four LST elements produce solutions which give reasonably good accuracy. The latter observations hold for the twelve elements CST and LST solutions. However, the accuracy obtained using four LST elements is superior to that obtained using twelve CST elements.

Figure (4-9a) and (4-9b) show that the area of steepest stress gradient is in the vicinity of the inner radius. Thus, the element mesh should be refined to include more triangular elements in this area.

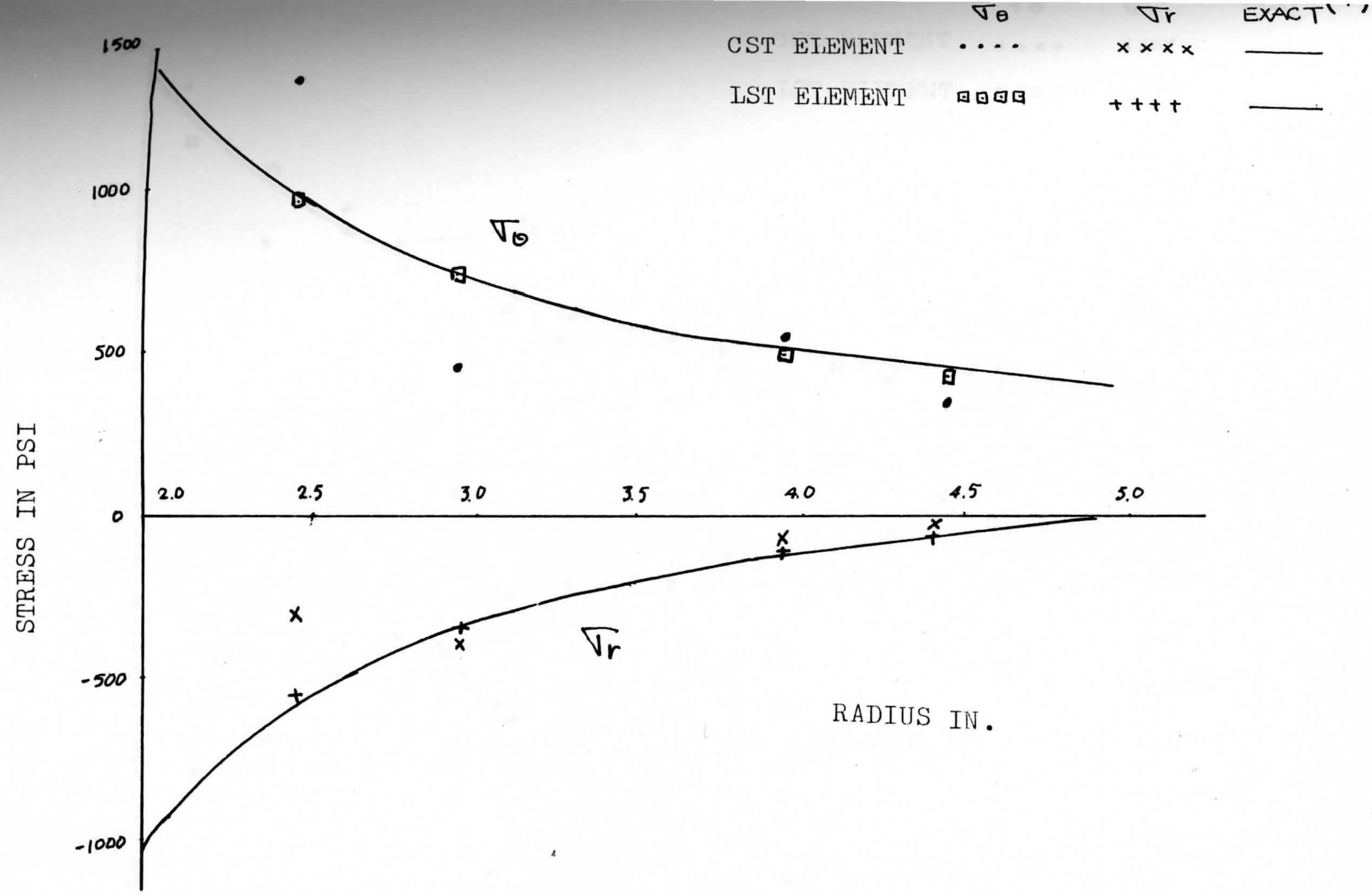


Figure (4-9a) Stress in a Disc with Internal Pressure : 4 Elements

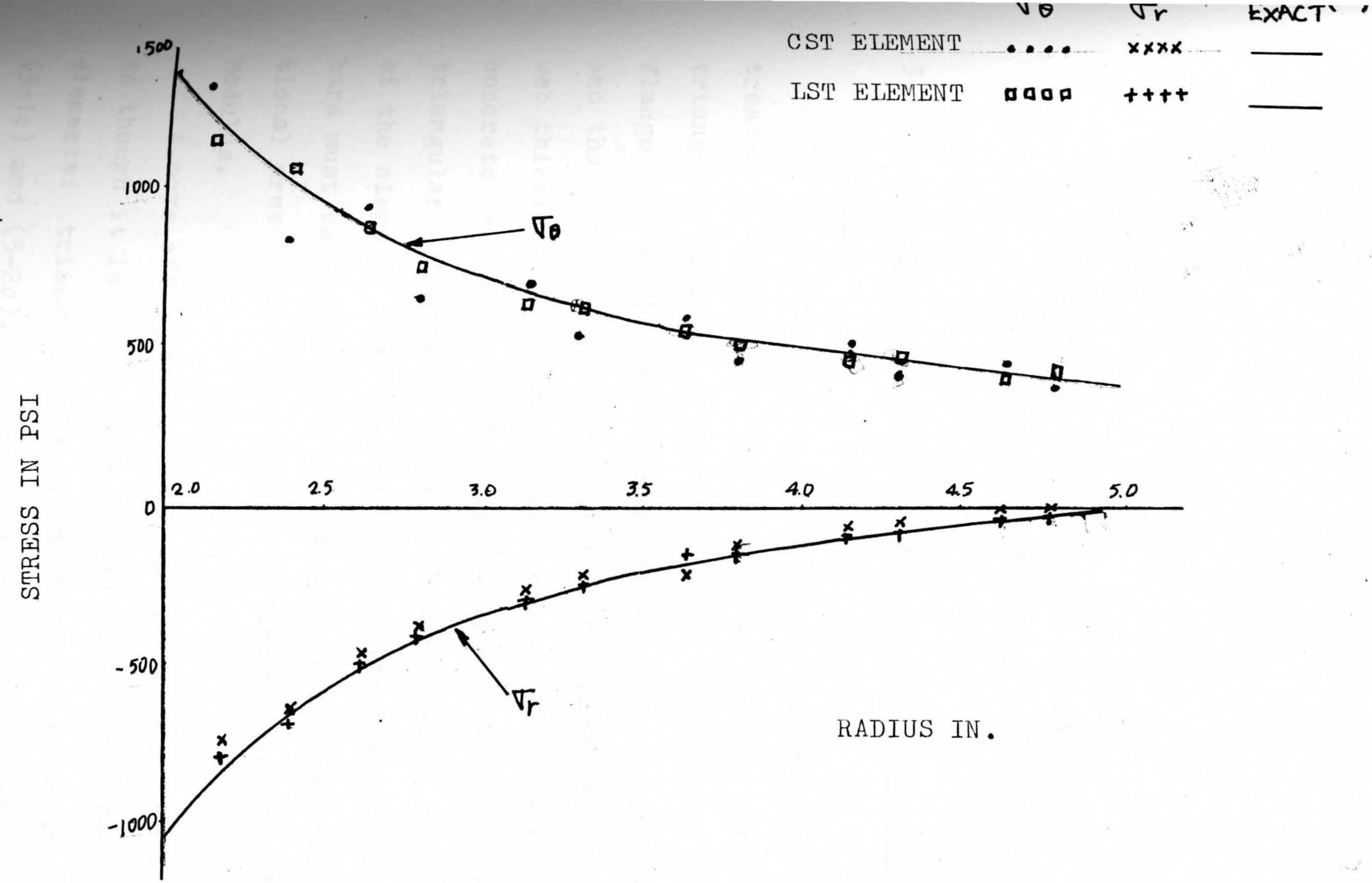


Figure (4-9b) Stress in a Disc with Internal Pressure : 12 Elements

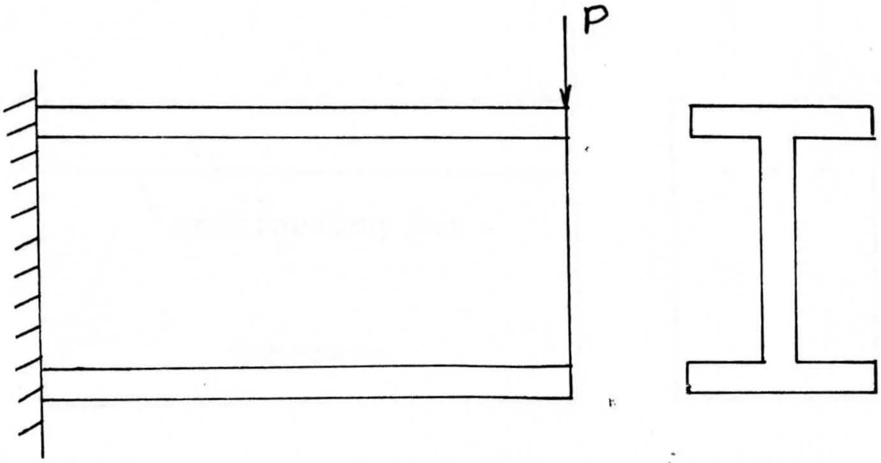
CHAPTER V

REINFORCED BEAMS BY BAR ELEMENTS AND LINEAR STRAIN TRIANGULAR ELEMENTS

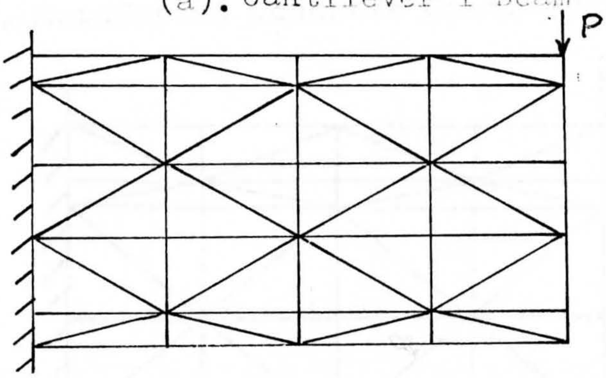
5-1 The Composite Structure and Associated Stiffness Matrix

The problems of the I beam in Figure (5-1a) may be treated by considering the beam to be a plate composed of triangular elements of Figure (5-1b), with those in the flange area having a thickness equal to the width of flange and those in the web area having a thickness equal to the web thickness. Similarly, the problem of the reinforced concrete cantilever in Figure (5-2a) may be solved by triangular elements as shown in Figure (5-2b), the thickness of the elements that are substituted for the reinforcement bars must be adjusted to take into account the cross-sectional area of the bars and the difference in Young's Modulus.

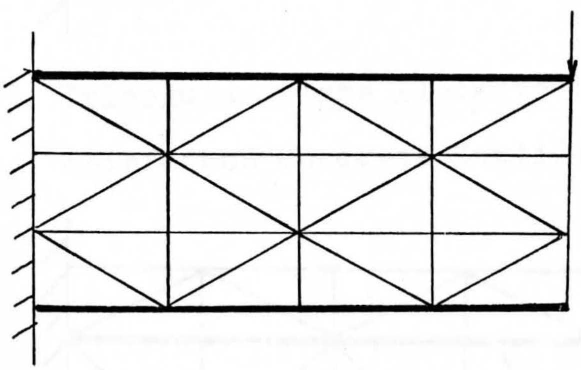
A more efficient approach is to consider the structure as though it is a composite of two different types of elements: triangular plates and bars, as shown in Figure (5-1c) and (5-2c). A large reduction in the number of degrees of freedom is accomplished when bar elements are



(a). Cantilever I Beam

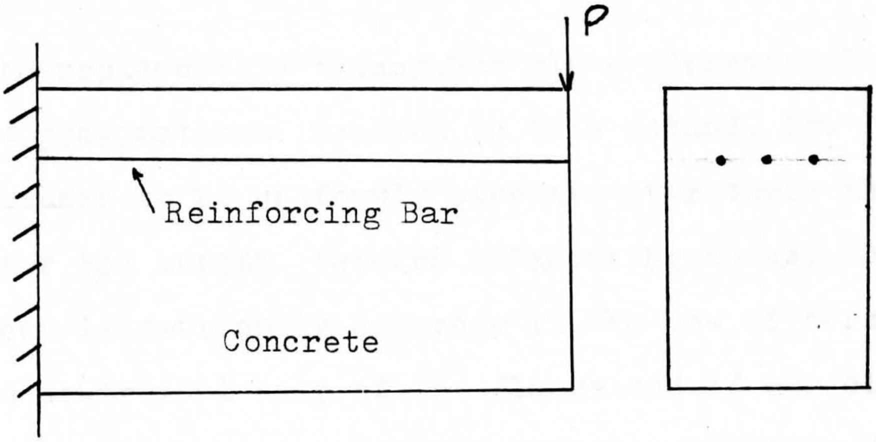


(b). Triangular Plate to Simulate I Beam

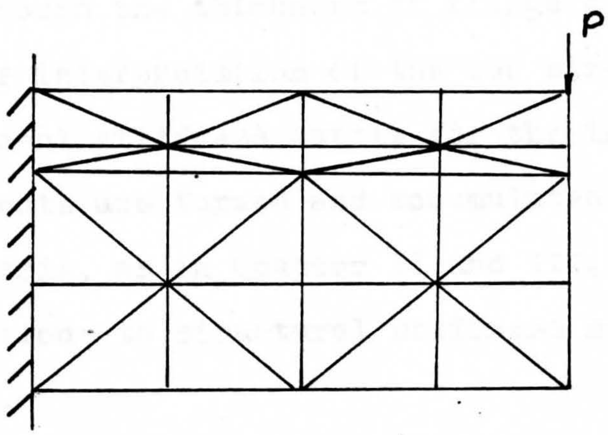


(c). I Beam Approximated by Plate and Bar Elements

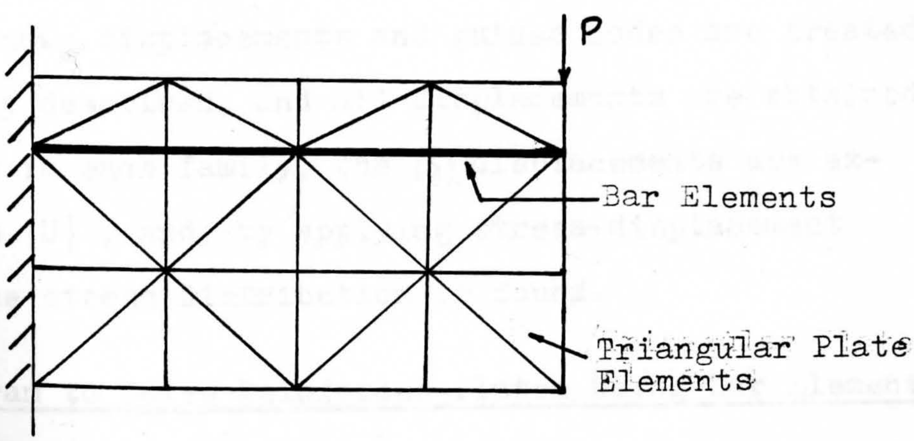
Figure (5-1) Cantilever I Beam



(a). Reinforced Concrete Cantilever



(b). Triangular Plate Elements to Simulate Reinforced Concrete Cantilever



(c). Reinforced Concrete Cantilever Approximated by Plate and Bar Elements

Figure (5-2) Reinforced Concrete Cantilever used to replace thin triangular plate elements. There are some approximations involved in this method. The flanges are assumed to be uniformly stressed over their thickness and over the length between neighboring nodes. This approximation is reasonably accurate if the row of bars is placed at the centroidal axis of the flange and if bar elements are kept short in regions of rapid change in moment. Variation in stress through the thickness of flange can be allowed for by proper interpretation of the bar stresses.

The element stiffness matrix for the triangular elements and bar elements are formed and accumulated in the structural stiffness matrix, as in Chapter II and III, independent of the contributions to structural stiffness matrix that other members make.

5-2 Stresses in a Composite Structure

When the structural stiffness matrix has been obtained, the loads, known displacements and guided nodes are treated as previously described, and all displacements are obtained. For elements in each family, the $\{\delta\}$ displacements are extracted from $\{U\}$, and by applying stress-displacement formulas, the stress distribution is found.

5-3 A Program to Solve Reinforced Plates Using Bar Element and Linear Strain Triangles

Main program SP44B, is presented in Appendix A and is not described in detail here since it is in essence a merger of PT01B with SP33B. Reference should be made to Figure (5-3), SP44B Flow Chart, Figure (5-4), instructions for SP44B Data Deck Preparations, and the computer output given in Figure (5-7).

1 READ TITLE

READ AND PRINT CASE TITLE

READ AND PRINT Y.M. AND P.R. FOR PLATE MATERIAL

SET CONSTANT VALUES IN [D], [B] AND [B] [D] [B] (DB03B)

CALCULATE [D][B]

READ COORDINATES OF NOTES (RC06B)

READ PLATE ELEMENT DATA

READ AND PRINT Y.M. OF BAR MATERIAL

READ BAR ELEMENT DATA (CN10B)

ZERO {TRIG} AND [S]

REWIND 1

DO FOR ALL PLATE ELEMENT

  DETERMINE $[A]^{-1}$, {X} AND {Y} FOR ORIGIN AT CENTROID (AI02B),

  WRITE $[A]^{-1}$ ON 1 (1N12B)

  DETERMINE ELEMENT STIFFNESS, [E] (ES06B)

  ADD [E] TO [S] (AS06B)

DO FOR ALL BAR ELEMENTS

  CALCULATE STIFFNESS AND ADD TO [S] (AS10B)

REWIND 4

WRITE [S] ON 4

2 READ KNOWN FORCES (KF01B)

READ KNOWN DISPLACEMENTS (KU01B)

READ GUIDED NODE DATA

SET {IFx} COMPONENTS

ALTER, FOR ROTATED AXES: {F} → {F′} AND [S] → [S′] (GN02B)

SOLVE FOR UNKNOWN COMPONENTS IN $\{U'\}$ (GEO2B)

REWIND 4

READ $[S]$ FROM 4

RESTORE $\{U'\} \rightarrow \{U\}$ (R 01B)

SOLVE FOR ALL FORCE COMPONENTS, PRINT $\{F\}$ AND $\{U\}$(F 10B)

REWIND 1

DO FOR ALL PLATE ELEMENTS

READ $[A]^{-1}$ FROM 1

CALCULATE AND PRINT STRESSES AT CENTROID (ST06B)

CALCULATE AND PRINT STRESSES IN BARS (ST10B)

READ NEXT

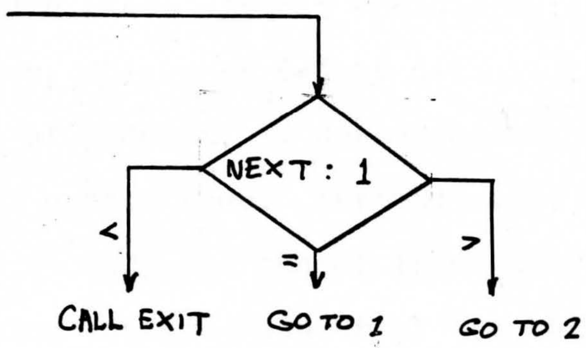


Figure (5-3) SP44B Flow Chart

Data Deck:

A card containing the Case Title.

A card containing the elastic constants of the plate:
Young's Modulus, Poisson's Ratio. Format: (2F10.5)

A card containing the number of nodes in the system. Format: (I5)

One card for each node that is located at the corner of a triangle, containing:
Node number, X coorinate, Y coordinate. Format: (I5, 2F105)

One blank card to indicate end of node data.

One card for each plate element containing:
Element number, Thickness (=1 by default), Node numbers on periphery of element, in sequence around the periphery starting at a corner node. Format: (I5, F10.5, 6I5)

One blank card to indicate end of triangular element data.

A card containing Young's Modulus of the bar element.

One card for each element, containing:
Element number, Cross-sectional area, Node number at each end of element. Format: (I5, F10.5, 2I5).

One blank card to indicate the end of bar element data.

One card for each known, nozero load component; containing
Component number, Magnitude of force, Format: (I5, F10.5)

One blank card to indicate end of load data.

One card for each known displacement component, containing:
Component number, Displacement. Format: (I5, F10.5)

One blank card to indicate end of load data.

One card for each guided node, containing:

Node number, Angle (alpha) in degrees defining the direction of the guiding plane, normal displacement of guiding plane, load tangential to guiding plane.

Format: (I5,3F10.5)

One blank card to indicate end of guided node data.

A card containing NEXT. Format: (I5)

NEXT=0;End the job.

Next=1; Execute a new case. Follow by data cards prepared according to all above instructions.

NEXT=2; Repeat the case just completed but with a new set of known loads and displacements. Follow by data cards described above starting with force data cards.

Figure (5-4) Instruction for SP44B Data Deck Preparation

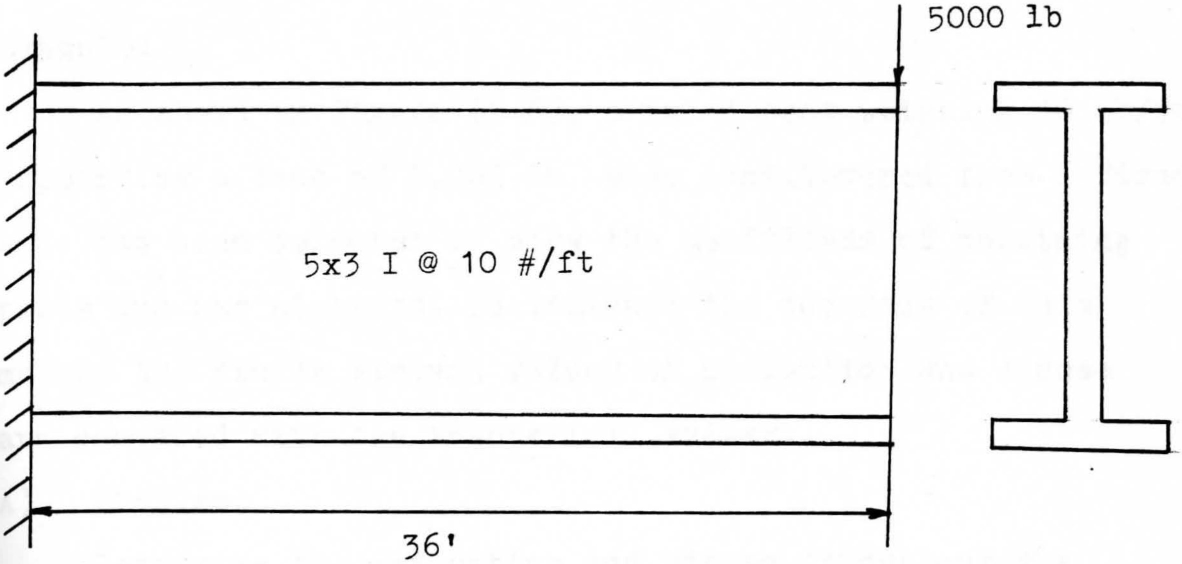


Figure (5-5) Cantilevered I Beam

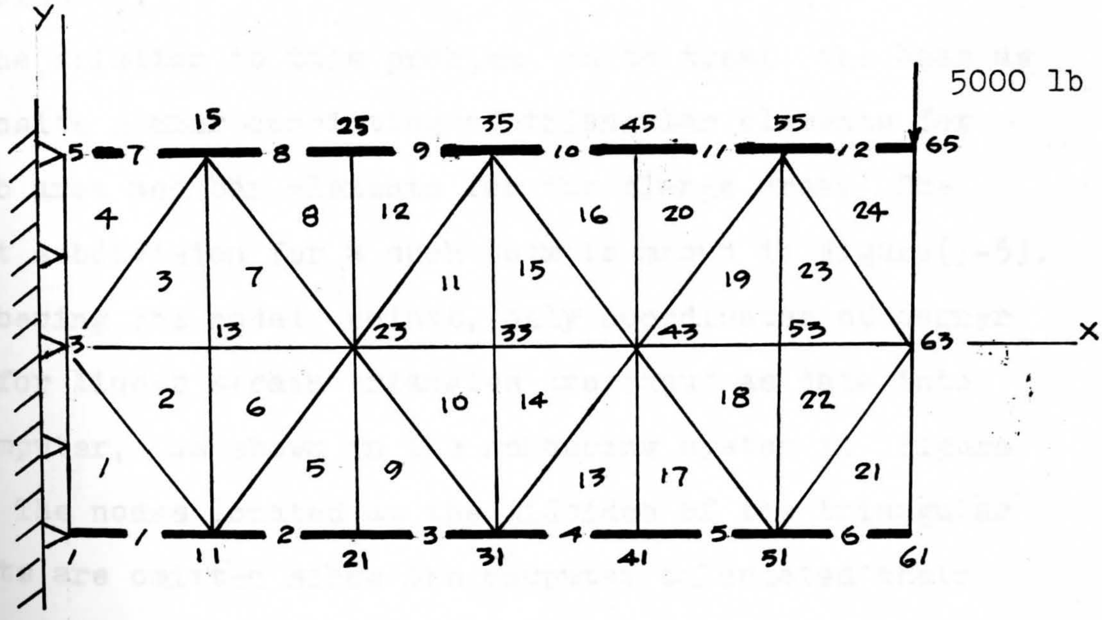


Figure (5-6) Element Subdivision for Cantilevered I Beam

Example:

As shown in Figure (5-6), a 5x3 I beam weighing 10 lb/ft supporting a load of 5,000 lb. when cantilevered from a fixed end, has been selected to show the usefulness of combining plate and bar elements. To indicate the accuracy of this method the finite element values of deflection and stress are compared with the theoretical values.

A). Problem:

Determine the deflection and stress throughout the beam, shown in Figure (5-5), by the Finite Element Method and compare the results with the theoretical solution.

B). Solution:

The solution to this problem is to treat the beam as a composite member consisting of triangular elements for the web area and bar elements for the flange area. The element subdivision for a such beam is shown in Figure(5-5). In numbering the nodal points, only coordinates of corner nodes for linear strain triangles are input as data into the computer, as shown in the numbering system in Figure (5-6). The nodes located at the midsides of the triangular elements are omitted since the computer calculated their location automatically.

The loading for this example, 5,000 lb. applied at the free end of the beam, is applied to the Finite Element Method model as a vertical nodal point load at node 65 as shown in Figure (5-6).

The boundary conditions for this cantilever beam are represented by pin connections at node 1 to 5 inclusive, thus, constraining the model in the same manner as the original fixed end beam. The computer input and output for this example is shown in Figure (5-7).

MAIN SP44B OCT 5 1979

STRESSES IN IN-P LANE LOADED PLATE (LST REINFORCING BARS GUIDED NODES)

CASE TITLE ---STRESSES IN A CATILEVERED I-BEAM

YOUNGS MODULUS= 0.300D 08 POISSONS RATIO= 0.300

NODE NO.	X-COORD	Y-COORD
1	0.0000D 00	-0.2337D 01
3	0.0000D 00	0.0000D 00
5	0.0000D 00	0.2337D 01
11	0.6000D 01	-0.2337D 01
13	0.6000D 01	0.0000D 00
15	0.6000D 01	0.2337D 01
21	0.1200D 02	-0.2337D 01
23	0.1200D 02	0.0000D 00
25	0.1200D 02	0.2337D 01
31	0.1800D 02	-0.2337D 01
33	0.1800D 02	0.0000D 00
35	0.1800D 02	0.2337D 01
41	0.2400D 02	-0.2337D 01
43	0.2400D 02	0.0000D 00
45	0.2400D 02	0.2337D 01
51	0.3000D 02	-0.2337D 01
53	0.3000D 02	0.0000D 00
55	0.3000D 02	0.2337D 01
61	0.3600D 02	-0.2337D 01
63	0.3600D 02	0.0000D 00
65	0.3600D 02	0.2337D 01

SYSTEM HAS 130 DEGREES OF FREEDOM

ELEM.NO.	THICKNESS	CONNECTING NODE NUMBERS					
1	0.21	1	6	11	7	3	2
2	0.21	11	12	13	8	3	7
3	0.21	3	8	13	14	15	9
4	0.21	15	10	5	4	3	9
5	0.21	11	16	21	22	23	17
6	0.21	23	18	13	12	11	17
7	0.21	13	18	23	19	15	14
8	0.21	23	24	25	20	15	19
9	0.21	21	26	31	27	23	22
10	0.21	31	32	33	28	23	27
11	0.21	23	28	33	34	35	29
12	0.21	23	29	35	30	25	24
13	0.21	31	36	41	42	43	37
14	0.21	31	37	43	38	33	32
15	0.21	33	38	43	39	35	34
16	0.21	43	44	45	40	35	39
17	0.21	41	46	51	47	43	42
18	0.21	51	52	53	48	43	47
19	0.21	43	48	53	54	55	49
20	0.21	43	49	55	50	45	44
21	0.21	51	56	61	62	63	57
22	0.21	51	57	63	58	53	52
23	0.21	53	58	63	59	55	54
24	0.21	63	64	65	60	55	59

BAND WIDTH= 26

YOUNGS MODULUS OF BARS = 0.300D 08

BAR NUMBER	X-SECT AREA	CONNECTS	NODES NO.
1	0.98	1	11
2	0.98	11	21
3	0.98	21	31
4	0.98	31	41
5	0.98	41	51
6	0.98	51	61
7	0.98	5	15
8	0.98	15	25
9	0.98	25	35
10	0.98	35	45
11	0.98	45	55
12	0.98	55	65

BAND WIDTH= 26

KNOWN NON-ZERO LOADS

COMPONENT NUMBER	LOAD
130	-0.5000D 04

KNOWN DISPLACEMENTS

COMPONENT NUMBER	DISPLAC EMENT
1	0.0000D 00
2	0.0000D 00
3	0.0000D 00
4	0.0000D 00
5	0.0000D 00
6	0.0000D 00
7	0.0000D 00
8	0.0000D 00
9	0.0000D 00
10	0.0000D 00

NODE NO. ALPHA(DEG) KNOWN U TANG. FORCE

NODE NO.	FORCE AND DISPLACEMENT COMPONENTS			
1	0.3569D 05	0.3664D 04	-0.3814D-14	-0.3915D-15
2	0.5641D 04	-0.1076D 04	-0.4745D-15	0.4310D-16
3	-0.6598D-03	-0.1776D 03	0.7399D-22	0.9488D-17
4	-0.5641D 04	-0.1076D 04	0.4745D-15	0.4310D-16
5	-0.3569D 05	0.3664D 04	0.3814D-14	-0.3915D-15
6	0.1273D-10	0.2491D-10	-0.3700D-02	-0.4017D-02
7	-0.4948D-11	0.1248D-10	-0.1746D-02	-0.3804D-02
8	-0.3802D-11	0.2028D-10	-0.9292D-10	-0.3740D-02
9	0.9095D-11	0.3039D-10	0.1746D-02	-0.3804D-02
10	-0.3638D-11	-0.4252D-11	0.3700D-02	-0.4017D-02
11	-0.1182D-10	0.9636D-10	-0.6140D-02	-0.1183D-01
12	-0.1013D-10	0.1985D-09	-0.3097D-02	-0.1158D-01
13	0.1936D-10	0.3511D-09	0.2720D-09	-0.1150D-01
14	0.6258D-11	0.3536D-09	0.3097D-02	-0.1158D-01
15	0.4532D-10	-0.7944D-10	0.6140D-02	-0.1183D-01
16	0.8573D-10	0.4134D-09	-0.9405D-02	-0.2332D-01
17	-0.4366D-10	0.1129D-08	-0.4605D-02	-0.2304D-01
18	-0.2925D-10	0.6856D-09	-0.1283D-08	-0.2293D-01
19	-0.2464D-11	0.4949D-09	0.4605D-02	-0.2304D-01
20	0.4547D-11	0.3638D-11	0.9405D-02	-0.2332D-01
21	0.1637D-10	-0.4316D-09	-0.1120D-01	-0.3795D-01
22	-0.8480D-10	0.1055D-08	-0.5661D-02	-0.3775D-01

23	-0.2547D-10	0.1110D-09	-0.7151D-08	-0.3769D-01
24	-0.3312D-10	0.8764D-09	0.5661D-02	-0.3775D-01
25	0.5730D-10	-0.7985D-09	0.1120D-01	-0.3795D-01
26	0.1746D-09	0.7440D-09	-0.1390D-01	-0.5546D-01
27	-0.7621D-10	0.8742D-09	-0.6851D-02	-0.5536D-01
28	0.1516D-10	0.1074D-08	-0.1407D-06	-0.5533D-01
29	-0.1455D-10	0.1586D-08	0.6851D-02	-0.5536D-01
30	-0.1057D-09	0.4908D-09	0.1390D-01	-0.5546D-01
31	-0.1601D-09	0.2664D-09	-0.1512D-01	-0.7560D-01
32	0.1087D-09	0.5562D-09	-0.7595D-02	-0.7546D-01
33	0.1095D-09	0.1524D-08	-0.2362D-06	-0.7541D-01
34	0.1200D-10	0.3838D-09	0.7595D-02	-0.7546D-01
35	0.7276D-10	-0.1216D-10	0.1512D-01	-0.7560D-01
36	0.7218D-09	0.1762D-08	-0.1727D-01	-0.9791D-01
37	-0.2910D-09	0.9841D-09	-0.8536D-02	-0.9772D-01
38	-0.3747D-09	0.2287D-08	-0.1819D-05	-0.9765D-01
39	0.8714D-10	0.1278D-08	0.8533D-02	-0.9772D-01
40	-0.4366D-09	-0.2232D-08	0.1726D-01	-0.9790D-01
41	0.5821D-10	0.8649D-09	-0.1793D-01	-0.1219D 00
42	-0.2475D-09	0.2675D-08	-0.9034D-02	-0.1218D 00
43	-0.7276D-10	0.2194D-08	-0.6732D-05	-0.1218D 00
44	-0.1101D-09	0.1165D-08	0.9024D-02	-0.1218D 00
45	0.4366D-09	-0.3110D-08	0.1794D-01	-0.1219D 00
46	0.6257D-09	-0.8095D-10	-0.1952D-01	-0.1473D 00
47	-0.8061D-10	0.3223D-08	-0.9676D-02	-0.1473D 00
48	0.8302D-10	0.4245D-08	-0.2465D-04	-0.1473D 00
49	-0.2910D-10	0.2507D-08	0.9638D-02	-0.1473D 00
50	-0.6863D-09	0.5888D-09	0.1949D-01	-0.1473D 00
51	0.4366D-10	0.2679D-09	-0.1959D-01	-0.1740D 00
52	0.1049D-09	0.3165D-08	-0.9887D-02	-0.1739D 00
53	0.7478D-10	0.1412D-08	-0.7704D-04	-0.1739D 00
54	-0.1641D-09	0.2265D-08	0.9787D-02	-0.1739D 00
55	0.4802D-09	-0.1870D-08	0.1963D-01	-0.1739D 00
56	0.6178D-09	0.2319D-08	-0.2068D-01	-0.2013D 00
57	-0.5239D-09	0.2797D-08	-0.1036D-01	-0.2012D 00
58	-0.5164D-09	0.3265D-08	-0.1665D-03	-0.2012D 00
59	-0.2443D-10	0.3054D-08	0.1004D-01	-0.2012D 00
60	-0.3056D-09	-0.6276D-10	0.2051D-01	-0.2013D 00
61	0.3347D-09	-0.2065D-09	-0.2009D-01	-0.2281D 00
62	-0.5478D-09	-0.3257D-08	-0.1024D-01	-0.2284D 00
63	0.1586D-09	-0.7676D-08	-0.2382D-03	-0.2288D 00
64	0.3781D-09	-0.1941D-08	0.1003D-01	-0.2292D 00
65	0.1892D-09	-0.5000D 04	0.2030D-01	-0.2298D 00

ELEM.NO.	SXX	SYY	SXY	THETA	PSI	PS2
1	-0.228D 05	-0.322D 04	-0.447D 04	12.3	-0.238D 05	-0.225D 04
2	-0.906D 04	-0.193D 03	-0.571D 04	26.1	-0.119D 05	0.260D 04
3	0.906D 04	0.193D 03	-0.571D 04	-26.1	0.119D 05	-0.260D 04
4	0.228D 05	0.322D 04	-0.447D 04	-12.3	0.238D 05	0.225D 04
5	-0.150D 05	0.150D 04	-0.445D 04	14.2	-0.161D 05	0.263D 04
6	-0.100D 05	0.184D 03	-0.574D 04	24.2	-0.126D 05	0.276D 04
7	0.100D 05	-0.184D 03	-0.574D 04	-24.2	0.126D 05	-0.276D 04
8	0.150D 05	-0.150D 04	-0.445D 04	-14.2	0.161D 05	-0.263D 04
9	-0.149D 05	-0.150D 04	-0.446D 04	16.8	-0.163D 05	-0.157D 03
10	-0.501D 04	-0.159D 03	-0.573D 04	33.5	-0.881D 04	0.364D 04
11	0.501D 04	0.157D 03	-0.573D 04	-33.5	0.880D 04	-0.364D 04
12	0.149D 05	0.150D 04	-0.446D 04	-16.8	0.163D 05	0.157D 03
13	-0.755D 04	0.153D 04	-0.448D 04	22.3	-0.938D 04	0.337D 04
14	-0.623D 04	0.168D 03	-0.574D 04	30.4	-0.960D 04	0.354D 04
15	0.620D 04	-0.147D 03	-0.573D 04	-30.5	0.957D 04	-0.352D 04

```
    16      0.752D 04     -0.147D 04     -0.443D 04      -22.3     0.934D 04     -0.329D 04
    17     -0.750D 04     -0.152D 04     -0.453D 04       28.3    -0.994D 04      0.919D 03
    18     -0.152D 04      0.126D 02     -0.594D 04       41.3    -0.674D 04      0.524D 04
    19      0.983D 03      0.347D 03     -0.553D 04      -43.4     0.620D 04     -0.487D 04
    20      0.736D 04      0.149D 04     -0.438D 04      -28.1     0.970D 04     -0.845D 03
    21     -0.489D 03     -0.157D 04     -0.403D 04      -41.2     0.304D 04     -0.509D 04
    22     -0.348D 04     -0.207D 03     -0.585D 04       37.2    -0.791D 04      0.423D 04
    23      0.122D 04     -0.552D 03     -0.559D 04      -40.5     0.600D 04     -0.533D 04
    24     -0.733D 03     -0.602D 04     -0.491D 04      -30.8     0.220D 04     -0.895D 04

BAR NO.    STRESS PSI
    1    -0.3070D 05
    2    -0.2530D 05
    3    -0.1960D 05
    4    -0.1407D 05
    5    -0.8291D 04
    6    -0.2468D 04
    7     0.3070D 05
    8     0.2530D 05
    9     0.1960D 05
   10     0.1408D 05
   11     0.8460D 04
   12     0.3339D 04

STATEMENTS EXECUTED=  320733

CORE USAGE          OBJECT CODE=   34048 BYTES,ARRAY AREA=  118376 BYTES,TOTAL AREA

DIAGNOSTICS            NUMBER OF ERRORS=        0, NUMBER OF WARNINGS=        0, NUMBER

COMPILE TIME=      0.95 SEC,EXECUTION TIME=      8.47 SEC,        6.09.57    WEDNESDA

        C$STOP
```

Figure (5-7) Computer Output Processed by SP44B

Results of the computer outputs are interpreted in Figure (5-8) and Table (5-1) and (5-2). The theoretical values of the flexural stress and the finite element values are compared in Figure (5-8). The maximum values are compared in Table (5-1). The percentage error of the stress at the center of gravity of the flange is 3.2%; this percentage may be reduced by inceasing the number of elements across the depth of the beam.

The indication of the accuracy for the deflections at the free end of beam obtained by the Finite Element Method and those obtained theoretically are compared in Table (5-2). Table (5-1) shows that the method gives more accurate deflection values than the theoretical one when the deflection due to shear is not considered. When the shear deflection is combined with the deflection due to bending, the Finite Element Method value is within 0.5% of the theoretical value.

Even though the number of elements used in this example is small, the results indicated the nature of the accuracy obtainable using the Finite Element Method.

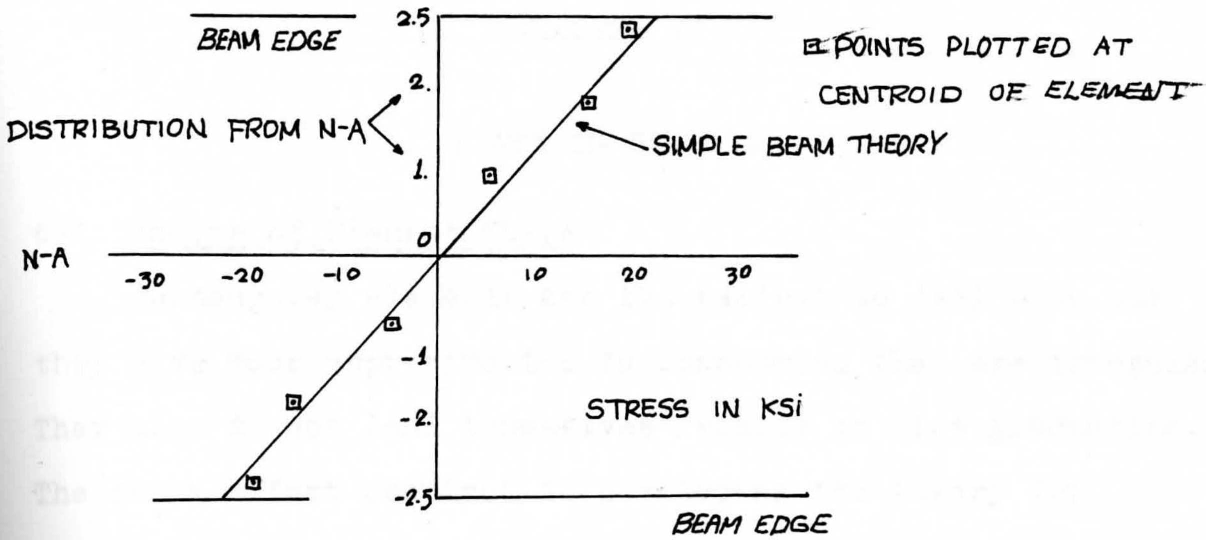


Figure (5-8) Stress in I Beam - Section x=0

Theoretical	20.91 ksi
Finite Element Method	20.54 ksi

Table (5-1) Maximum Bending Stress at x=0

	Maximum Deflection in inches		
	Bending	Shear	Total
Theoretical	-0.2142	-0.0171	-0.2312
Finite Element Method	--------	--------	-0.2301

Table (5-2) Deflections at Free End of I Beam

CHAPTER VI

PLATE IN BENDING

6-1 Choice of Element Shape

Rectangular elements are the easiest to deal with but they give poor approximation to boundaries that are irregular. They also do not lend themselves readily to size graduation. The extra effort required in developing the theory for triangular elements is justified by the ease with which they may be molded to fit irregular boundaries. They may also be graduated in size to permit small triangles in regions of large gradient.

Triangular elements having nine degrees of freedom have been used. As the number of degrees of freedom is increased, the assumed displacement function may be of higher order, hence the accuracy is improved but the analysis becomes more complex. A good compromise is reached at eighteen degrees of freedom where the accuracy is quite adequate for engineering solution, while the analysis is still manageable. The merits of various elements are discussed in Chapter 5 and 7 of Holand and Bell[10].

6-2 Triangular Plate Bending Element Having 18 Degrees of Freedom

The theoretical development of the 18 degrees of freedom triangle is presented in a very clear and concise manner by Bell[8]. A brief outline of his procedure will be given here, which should be adequate for those who wish to use the program of this chapter.

For the deflection normal to the plane of the plate, only one displacement, w, is required. This means that the displacement function can be simplified to the form as

$$w = \{ 1 \quad x \quad y \quad x^2 \quad xy \quad y^2 \cdots \} \begin{Bmatrix} \alpha_1 \\ \alpha_2 \\ \alpha_3 \\ \vdots \\ i \end{Bmatrix}$$

If the transverse displacements at the nodes were the only degrees of freedom the displacement function could contain only the first three polynomial terms, which would be far from adequate. More degrees of freedom could be introduced by taking a number of the derivatives of displacement as degrees of freedom. Thus, at each corner node, we may take as degrees of freedom,

$$\overset{(1)}{w}, \quad \overset{(2)}{\frac{\partial w}{\partial x}}, \quad \overset{(3)}{\frac{\partial w}{\partial y}}, \quad \overset{(4)}{\frac{\partial^2 w}{\partial x^2}}, \quad \overset{(5)}{\frac{\partial^2 w}{\partial x \partial y}}, \quad \overset{(6)}{\frac{\partial^2 w}{\partial y^2}}$$

Then, it is possible to evaluate constants for eighteen polynomial terms in the displacement function. 18 terms allows complete polynomials up to fourth degree and three from the six fifth-degree terms. To include the complete family of fifth-degree terms, three more degrees of freedom are needed. These can be found by introducing mid-side nodes and using the normal derivatives at these nodes as degree of freedom. Thus, an element is obtained that has complete fifth degree polynomial displacement and twenty-one degrees of freedom.

The mid-side nodes cause difficulties in geometrically defining an element by increasing the band-width and the total number of degrees of freedom. Thus, there is a strong incentive for eliminating them. This is accomplished by assuming that the variation of normal derivative on each edge is a cubic polynomial. The normal derivative at the mid-side node can then be expressed in terms of degrees of freedom at the end node. This enables the mid-side degrees of freedom to be eliminated from the system and provides an 18x18 stiffness matrix obtained for an element having 18 degrees of freedom.

The stiffness matrix for the assembly of triangular elements is accumulated in the usual manner. A concentrated point load at a node represents a known force component with the same subscript as w at that node. Distributed load is more difficult to deal with as it must be replaced by components corresponding to all six degrees of freedom at each node. This is done by determining the consistent load vector, which must be found by the computer as it is impractical to evaluate by hand, by processes similar to those developed in Chapter IV.

Boundaries that are fixed and those that are simply supported are quite easy to define by specifying the known displacement, slope, and curvature components at the nodes. Care must be taken to state all known components. For example, if an edge parallel to the X axis is fixed, then at each node on the edge it must be specified that component numbers 1,2, 3,4, and 5 are zero. Whenever there is an axis of symmetry, this wraping component 5, must be specified as zero as well as the normal slope, number 2 or 3. Some ambiguity may arise at corners where the known displacement component numbers depend on which edge the corner node is associated with. If the element at such a corner is large, the results can be significantly altered by the arbitrary choice of inputs. The best technique is to provide small elements at ambiguous corners.

Not all edges in real problems can be arranged to be parallel to one of the coordinate axes. To accommodate such an edge one must rotate the axes locally. When there is local rotation of axes, the stiffness matrix must be altered in a manner similar to that given in Chapter III for guided nodes.

A free edge or one on which the bending moment is known cannot be treated directly. The bending moment on the edge is dertermined by components 4 and 6 linked through the Poisson effect. Thus, a zero moment does not mean that either curvature is zero but rather that a combination of the two must be zero. This can be handled by a transformation which in effect makes a new component 4 equivalent to the old 4 and 6 linked through Poisson's ratio. Thus, the new component 4 is proportional to the bending moment in the X direction. Simultaneously, curvature component 6 is also modified.

6-3 The Program to Solve a Plate in Bending

The main program PB11B which solves plate-bending problems by the methods just described is presented in Appendix A. Reference should be made to Figure (6-1), PB11B flow chart; Figure (6-2), instruction for PB11B data deck preparation; and the computer output given for the example at the end of chapter as Figure (6-4).

The program is able to deal with concentrated normal forces and normal pressure that vary linearly over the surface of the element. For higher order pressure functions, one must approximate the pressure by specifying as equivalent pressure at the centroid and a rate of change in the coordinate directions.

For known edge moments, one must specify the nodes at which modified curvatures are required. The curvatures are combined to give a modified curvatures such that the modified curvature multiplied by the flexural rigidity is equal to moment intensity. Thus, for a given edge moment the modified curvature can be determined and used as a boundary condition.

1 PRINT TITLE

READ AND PRINT CASE TITLE

READ AND PRINT ELASTIC CONSTANTS AND PLATE THICKNESS

READ COORDINATES OF NODES (RC02B)

READ AND PRINT ELEMENT DATA INCLUDING PRESSURE (CN08B)

DO FOR ALL ELEMENTS

  DETERMINE ELEMENT STIFFNESS [E](AM21B) (IN12B) (EV02B)

  DETERMINE CONSISTENT LOAD VECTOR FOR PRESSURE, {PF}

  (TI01B) (ES18B) (PF01B)

  ADD [E] TO [S] AND {PF} TO {F}(AS03B)

READ DATA FOR LOCAL ROTATION OF AXES

FILL IN TRANSFORMATION MATRIX FOR ROTATION (TM02B)

ALTER [S] FOR ROTATED AXES (LR01B)

ALTER {F} FOR ROTATED AXES (LR02B)

READ NUMBERS OF NODES REQUIRING MODIFIED CURVATURE

ALTER [S] FOR MODIFIED CURVATURES (LR01B)

ALTER {F} FOR MODIFIED CURVATURES (LR02B)

READ KNOWN FORCES (KF02B)

READ KNOWN DISPLACEMENTS (KU01B)

SOLVE FOR UNKNOWN COMPONENTS IN {U} (GE02B)

CHANGE {U} FOR RESTORED CURVATURES (RU02B)

PRINT DISPLACEMENTS

CALCULATE AND PRINT MOMENTS AND STRESSES (ST19B)

READ NEXT

NEXT=0----CALL EXIT, NEXT=1----GO TO 1

Figure (6-1) PB11B Flow Chart

DATA DECK

One card containing Case Title.

One card containing:

Young's Modulus, Poisson's Ratio, Plate Thickness. (Format:2F10.5)

One card for each node containing:

Node Number, X coordinate, Y coordinate. (Format:I5, 2F10.5)

One blank card to indicate end of node data.

One card for each element containing:

Element number,Node number at corners of triangle in counter-clockwise order,Pressure at centroid, Rate of pressure change in X direction, Rate of pressure change in Y direction. (Format: 4I5, 3F10.5)

One blank card to indicate end of element data.

One card for each node having local axes rotated, containing:

Node number, angle of rotation in degrees. (Format: I5, F10.5)

One blank card to indicate end of rotation data.

One card for each node at which curvatures are to be modified.

Node number. (Format: I5)

One blank card to indicate end of modified curvatures data.

One card for each known, concentrated load component containing:

Component number, Load component. (Format: I5, F10.5)

One blank card to indicate end of concentrated load data.

One card for each known displacement component containing:

Component number, Displacement. (Format: I5, F10.5)

One blank card to indicate end of known displacements.

One card containing NEXT. (Format:I5)

NEXT=0;End of job. NEXT=1; Execute a new case. Follow by cards prepared according to all above instruction.

Figure(6-2) Instruction for PB11B Data Preparation

6-4 Example Problem

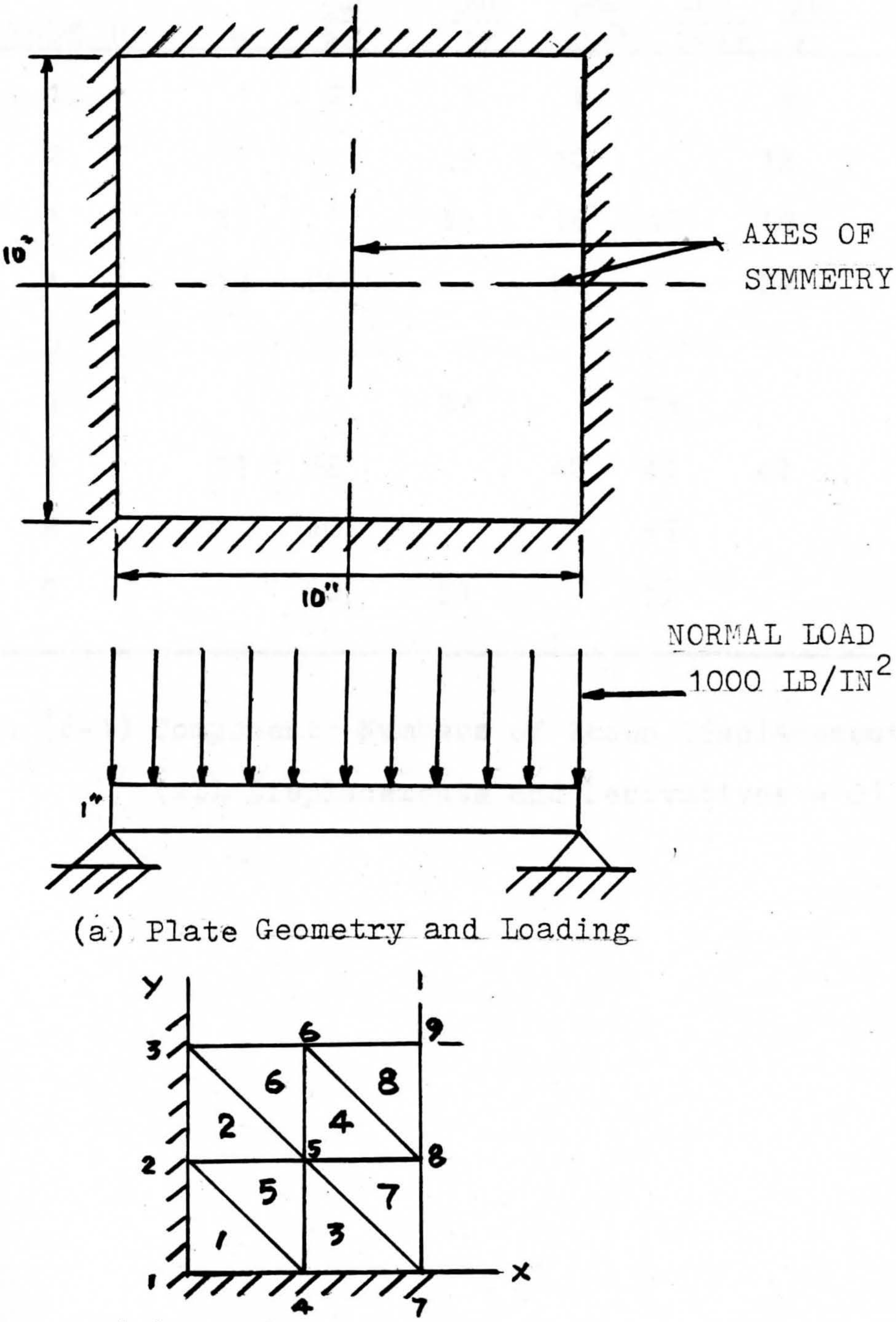
The sample example of plate in bending with simply supported edge condition has been selected for this chapter to illustrate the use of Finite Element Method and to show the form of computer output.

a). Problem:

For a 10"x10" steel plate with a uniform load of 1,000 pounds per square inch as shown in Figure (6-3a), determine the maximum deflection and maximum bending stress in the plate. The plate is one inch thick and is simply supported on all four edges.

b). Solution:

It is necessary to solve only one-quarter of the plate since the plate and load have two axes of symmetry. This section of the plate is divided into eight elements as shown in Figure(6-3b). No local rotations and no modified curvatures are required for this problem. The boundary conditions are interpreted as component numbers of known zero displacements, slopes, and curvatures are shown in Table (6-1). This tabular arrangement enables a comparison to be made to ensure that nodes have been treated consistently.



(a) Plate Geometry and Loading

(b) Element Subdivision

Figure (6-3) Square Plate with Normal Load

NODE NO	w	$\frac{\partial w}{\partial x}$	$\frac{\partial w}{\partial y}$	$\frac{\partial^2 w}{\partial x^2}$	$\frac{\partial^2 w}{\partial x \partial y}$	$\frac{\partial^2 w}{\partial y^2}$
1	1	2	3	4		6
2	7		9	10		12
3	13		15	16	17	18
4	19	20		22		24
5						
6			33		35	
7	37	38		40	41	42
8		40			47	
9		50	51		53	

Table (6-1) Components Numbers of Known Displacement
(All Displacements and Derivatives = 0)

MAIN PR11B NOV 14 1979

PLATE IN BENDING B Y TRIANGULAR ELEMENTS (18 DEG. OF F.)

CASE TITLE --- SQAURE PLATE UNIFORM LOAD SIMPLY SUPPORTED

YOUNGS MODULUS= 0.300000000D 08

POISSONS RATIO= 0.300000000D 00

PLATE THICKNESS= 0.100000000D 01

NODE NO.	X-COORD	Y-COORD
1	0.00000D 00	0.00000D 00
2	0.10000D-02	0.25000D 01
3	0.20000D-02	0.50000D 01
4	0.25000D 01	0.00000D 00
5	0.25010D 01	0.25000D 01
6	0.25020D 01	0.50000D 01
7	0.50000D 01	0.00000D 00
8	0.50010D 01	0.25000D 01
9	0.50020D 01	0.50000D 01

ELEM.NO.	CORNER NODE NUMBERS			PRESSURE	DP/DX	DP/DY
1	1	4	2	0.10000D 04	0.00000D 00	0.00000D 00
2	2	5	3	0.10000D 04	0.00000D 00	0.00000D 00
3	4	7	5	0.10000D 04	0.00000D 00	0.00000D 00
4	5	8	6	0.10000D 04	0.00000D 00	0.00000D 00
5	2	4	5	0.10000D 04	0.00000D 00	0.00000D 00
6	3	5	6	0.10000D 04	0.00000D 00	0.00000D 00
7	5	7	8	0.10000D 04	0.00000D 00	0.00000D 00
8	6	8	9	0.10000D 04	0.00000D 00	0.00000D 00

BAND WIDTH= 24

NUMBER OF DIFFERENT ELEMENT TYPES = 2

LOCAL ROTATED AXES
ODE NO. THETA(DEG)

NODES HAVING MODIFIED CURVATURES

KNOWN NON-ZERO LOADS

COMPONENT NUMBER LOAD

KNOWN DISPLACEMENTS

COMPONENT NUMBER	DISPLAC EMENT
1	0.00000D 00
2	0.00000D 00
3	0.00000D 00
4	0.00000D 00
7	0.00000D 00
6	0.00000D 00
9	0.00000D 00
10	0.00000D 00
12	0.00000D 00
13	0.00000D 00
15	0.00000D 00
16	0.00000D 00
17	0.00000D 00
18	0.00000D 00

19	0.00000 00
20	0.00000 00
22	0.00000 00
24	0.00000 00
33	0.00000 00
35	0.00000 00
37	0.00000 00
38	0.00000 00
40	0.00000 00
41	0.00000 00
42	0.00000 00
44	0.00000 00
47	0.00000 00
50	0.00000 00
51	0.00000 00
53	0.00000 00

NODE NO. DISPLACEMENTS REFERRED TO GLOBAL AXES

NODE NO.						
1	-0.1733D-15	0.3984D-16	0.4270D-16	0.1664D-15	0.1643D-02	0.1650D-15
2	0.1300D-15	0.3515D-02	0.2151D-16	-0.1521D-15	0.9825D-03	0.2259D-15
3	0.2434D-15	0.4827D-02	-0.4652D-16	-0.2398D-15	0.3341D-15	0.2379D-15
4	0.1658D-15	0.4735D-16	0.3590D-02	0.2722D-15	0.1105D-02	-0.1122D-15
5	0.7662D-02	0.2273D-02	0.2294D-02	-0.4962D-03	0.7103D-03	-0.7061D-03
6	0.1070D-01	0.3157D-02	0.3243D-15	-0.1160D-02	0.2147D-15	-0.9831D-03
7	0.1665D-15	0.7517D-16	0.4906D-02	-0.1127D-15	0.4961D-15	-0.3604D-15
8	0.1060D-01	0.3179D-15	0.3194D-02	-0.9310D-03	0.2072D-15	-0.1079D-02
9	0.1475D-01	0.4223D-15	0.4304D-15	-0.1317D-02	-0.1591D-14	-0.1366D-02

NODE NO.	MXX	MYY	MXY	SXX	SYY	SXY	THETA	PS1	PS2
1	0.593D-09	0.590D-09	0.316D 04	-0.356D-08	-0.354D-08	-0.190D 05	45.0	-0.190D 05	0.190D 05
2	-0.232D-09	0.495D-09	0.189D 04	0.139D-09	-0.297D-09	-0.113D 05	-45.0	0.113D 05	-0.113D 05
3	-0.463D-09	0.456D-09	0.643D-09	0.279D-08	-0.273D-08	-0.386D-08	-27.2	0.476D-08	-0.472D-08
4	0.655D-09	-0.839D-10	0.212D 04	-0.393D-08	0.503D-09	-0.127D 05	45.0	-0.127D 05	0.127D 05
5	-0.195D 04	-0.235D 04	0.137D 04	0.117D 05	0.141D 05	-0.820D 04	40.8	0.460D 04	0.212D 05
6	-0.400D 04	-0.366D 04	0.413D-09	0.240D 05	0.219D 05	-0.248D-08	-0.0	0.240D 05	0.219D 05
7	-0.607D-09	-0.108D-08	0.935D-09	0.364D-08	0.650D-08	-0.561D-08	37.9	-0.720D-09	0.109D-07
8	-0.345D 04	-0.373D 04	0.398D-09	0.207D 05	0.224D 05	-0.239D-08	0.0	0.207D 05	0.224D 05
9	-0.474D 04	-0.484D 04	-0.306D-08	0.285D 05	0.290D 05	0.184D-07	-0.0	0.285D 05	0.290D 05

Figure (6-4) Computer Output Processed by PB11B

	MAXIMUM DEFLECTION	BENDING MOMENT
	$W_{max} = \alpha q a^4/D$	$M_{x_{max}} = M_{y_{max}} = \beta q a^2$
FINITE ELEMENT METHOD	0.01475	4740
EXACT SOLUTION (11)	0.01478	4789
SOLUTION BY BELL (10)	0.01479	4791

Table (6-2) Comparisons of Centerline Deflection and Moment for the Square Plate in Bending

c). Results:

The computer output for this problem is shown in Figure (6-4). The sample used in this exercise is the same as that used by Bell[8]. The results agree with those of Bell and theoretical solutions. The largest error occurred at node 1 where M_{xx} is 1.4 percent in error in comparison with Bell. Considering the wide variation in moment over the region of the chosen elements, it is remarkable that this accuracy is attained.

Comparisons of the centerline deflection and moments are made in Table (6-2) which includes the reults of the finite element solutions of Bell as well as the exact solutions of Timoshenko[11].

CHAPTER VII

AUTOMATIC MESH GENERATION AND ITS PRACTICAL USE

7-1 Introduction

The development of Finite Element Method provides engineers with a versatile and general-method tool for the analysis of continuous domain in the field of structural mechanics, heat transfer, or fluid mechanics. The ease of automating input data and the trend in computer industry to larger and faster machines increases the value of this method. In any of the available practical computer programs for these analyses, the preparation of input data is a tedious and time consuming task. Automatic mesh generation then is an attempt to simplify input data for the Finite Element Method programs.

Some of the advantages of automatic mesh generation are obvious. Once the programs has been prepared it can be used for various problems by simply changing the few parameters involved according to the geometrical condition of the structure to be solved. The saving in the hand-labor time is significant. In addition, there is a reduction in the probability of human error involved in the preparation of data; this is because the attention of the users can be concentrated on a few parameters rather than on handling of thousands of data associated with the nodes and elements. Thus, if the users can communicate with the computer in a better, faster, error-free environment, money and labor will be substantially saved. Other advantages include the following: simplication of a parameter study, insured regularity of the mesh, ease in using other types of elements, and closer control on errors in stress values.

Zienkiewicz[9], reiterates the importance of mesh generation when he writes: " Automatic mesh generation...... are items on which efforts must be continued. If further break-throughs are to be expected, it is in this area that they will occur."

Because of the many mesh generation programs coming into existence, this chapter is limited to the discussion of pertinent schemes in automatic mesh generation for triangular and bar elements since both are most useful element types in practical problems. The investigation is conveniently divided into two separate parts: node point generation and element generation. The triangular element mesh generation program for LST elements will be used practically by combining the program developed in Chapter IV for the problem of compressed circular ring plate to indicate the incentives to automating the input data.

7-2 Triangular Element Mesh Generation and Computer Program

The input data for generating a triangular mesh consists of the following information:

NY - the total number of generation lines

CON - a weighting factor

NX(I)- the required number of intervals for each generating line

XY(I)
YF(I) - coordinates of the endpoints of each generating line
XL(I)
YL(I)

The output information consists of nodal point and element definitions which are the x and y coordinates of each nodal point with its correponding node number and the element number with its three corner node numbers. Both nodal and element numbers start with initial value 1 and are arranged in the sequential order.

In order to minimize the band width of the resulting stiffness matrix, the generating line should always traverse the shorter direction of the domain. In order to obtain the more accurate solutions in area of steep stress gradient a larger number of elements which means a finer mesh is necessary.

An efficient technique incorporated into the program is the weighting factor, CON. By varying this parameter the node intervals along a generating line are made to become progressively shorter, stay equal, or progressively

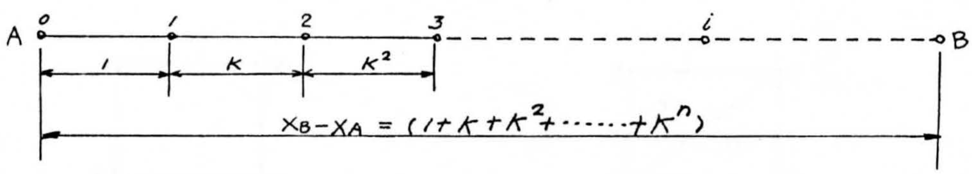


Figure (7-1) Effect of Weighting Factor upon Nodal Intervals

longer.

From Figure (7-1) it can be seen that the coordinates of point i along a generating line AB can be obtained by

$$x_i = (x_B - x_A)\frac{\sum_{j=1}^{i} K^{j-1}}{\sum_{j=1}^{n} K^{j-1}} + x_A$$

and analogously

$$y_i = (y_B - y_A)\frac{\sum_{j=1}^{i} K^{j-1}}{\sum_{j=1}^{n} K^{j-1}} + y_A$$

It should be noted that for K=1 the nodes are equally spaced, for K<1 the spacing decreases for subsequent node points, for K>1 the spacing increases. The node numbers are labelled sequentially from one generating line to the next, by systematically increasing the node number by a value one at every defined node point. Numbering the node points finishes automatically after the last point of the structure is reached. All nodal points of the structure have unique names at this stage.

The nodal points must then be connected to define elements, and the assemblage of the elements forms the continuous structure. The element definitions are formulated by taking the nth and (n+1)th generating line,

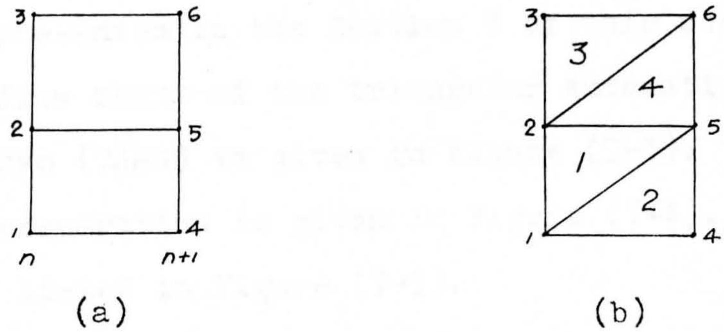


Figure (7-2) Steps of Element Generation

and forming a quadrilateral by connecting the nodal points of nth and (n+1)th generating line starting from line n=1 (see Figure (7-2a)). The quadrilateral elements are then divided into two triangles by connecting the first nodal point of the nth generating line to the second nodal point of the (n+1)th generating line as shown in Figure(7-2b). Triangular element 1 simply connects the nodal points 1,2,5, with clockwise ordering of the node point numbers. All element numbers are defined by adding 1 to the number of the preceding element until the last element is encountered.

The program is efficiently written so that a number of features may be changed or added to the program without a great deal of difficulty. This program may be altered to include the linear strain triangle problem which has six nodal points per element by simply making some minor changes in numbering nodal point system to provide an extra nodal point at the midpoint of each element edge. This has been done by the author and used practically in SP33B (LST) program which is presented in the Section 3 of this chapter.

The flow chart of the triangular automatic mesh generation program (TAMG) is given in Figure (7-3). The instructions for data preparation is given in Figure (7-4). The program (TAMG) is listed in Figure (7-5).

READ AND WRITE NUMBERS OF GENERATING LINES AND WEIGHTING FACTOR.

READ AND WRITE NUMBERS OF INTERVALS, AOORDINATES OF STARTING AND ENDING POINTS OF GENERATING LINES.

CALCULATE AND PRINT THE NODE NUMBERS AND ITS CORRESPONDING X, Y COORDINATES.

CALCULATE AND PRINT THE ELEMENT NUMBERS AND ITS THREE CONNECTING NODES.

Figure (7-3) Flow Chart for TAMG Program

Data Deck:

A card containing:

The number of generating lines and weighting factor.

Format: (I5, F10.5)

One card for each generating line containing:

Number of intervals, starting X coordinate, starting Y coordinate, ending Xcoordinate, ending Y coordinate.

Format: (I5, 4F10.5)

Figure (7-4) Instructions for TAMG Data Deck Preparation

```
$JOB
      DIMENSION NX(20).YL(20).XF(20).YF(20).NOD(4).SUM1(15).XL(20)
      READ(5.1000) NY.CON
      WRITE(6.1001) NY.CON
      DO 100 I=1.NY
      READ(5.1000) NX(I).XF(I).YF(I).XL(I).YL(I)
      WRITE(6.1000) NX(I).XF(I).YF(I).XL(I).YL(I)
  100 CONTINUE
      WRITE(6.1431)
      N=0
      DO 350 I=1.NY
      NXI=NX(I)+1
      SUM1(1)=0.0
      SUM1(2)=1.0
      SUM=1.0
      IF(NXI-2) 190.291.190
  190 DO 250 K=3.NXI
      SUM1(K)=SUM1(K-1)*CON
      SUM=SUM+SUM1(K)
  250 CONTINUE
  291 CONTINUE
      X=XF(I)
      Y=YF(I)
      DO 300 J=1.NXI
      N=N+1
      X=(XL(I)-XF(I))*SUM1(J)/SUM+X
      Y=(YL(I)-YF(I))*SUM1(J)/SUM+Y
      WRITE(6.1430) X.Y.N
  300 CONTINUE
  350 CONTINUE
      N=0
      NSUM=0
      NXI=NY-1
      WRITE(6.1432)
      DO 600 I=1.NXI
      NXI=NX(I)
      DO 500 J=1.NXI
      IF(J-NXI) 379.371.379
  371 IF(NX(I+1)-NX(I)) 380.379.401
  379 NOD(1)=J+NSUM
      NOD(2)=NOD(1)+1
      NOD(3)=NOD(2)+NXI+1
      NOD(4)=NOD(3)-1
      GO TO 412
  380 NOD(1)=NOD(2)
      NOD(2)=NOD(1)+1
      NOD(4)=0
      GO TO 412
  401 NOD(1)=J+NSUM
      NOD(2)=NOD(1)+1
      NOD(3)=NOD(2)+NXI+1
      NOD(4)=NOD(3)-1
      N=N+1
      WRITE(6.1470) N.NOD(1).NOD(2).NOD(3)
      N=N+1
      WRITE(6.1470) N.NOD(1).NOD(3).NOD(4)
      NOD(1)=NOD(2)
      NOD(2)=NOD(3)+1
      NOD(4)=0
  412 N=N+1
```

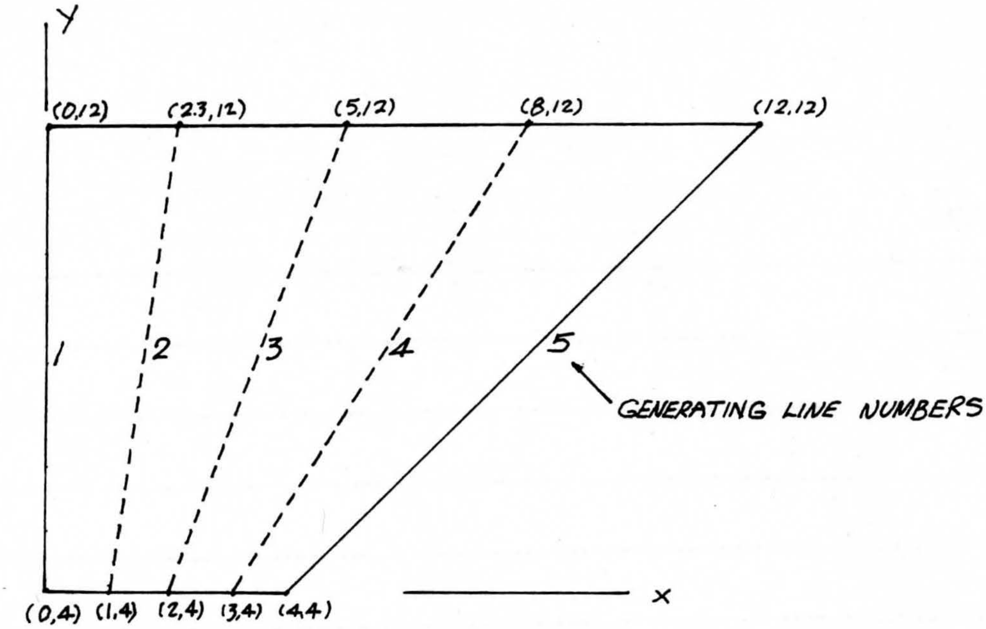
```
      WRITE(6.1470) N.NOD(1).NOD(2).NOD(3)
      IF(NOD(4))433.434.433
  433 N=N+1
      WRITE(6.1470) N.NOD(1).NOD(3).NOD(4)
  434 CONTINUE
  500 CONTINUE
      NSUM=NSUM+NXI+1
  600 CONTINUE
      STOP
 1000 FORMAT(I5.5X.6F10.5)
 1001 FORMAT('1 CONTROL DATA'./.I5.5X.6F10.5)
 1431 FORMAT('1  X CO-ORDINATE    Y CO-ORDINATE   NODAL POINT')
 1430 FORMAT(2F16.8.I4)
 1432 FORMAT('1  ELEMENT  NO.   NODAL POINTS OF THE ELEMENTS')
 1470 FORMAT(I8.5X.3I8)
      END

$ENTRY
```

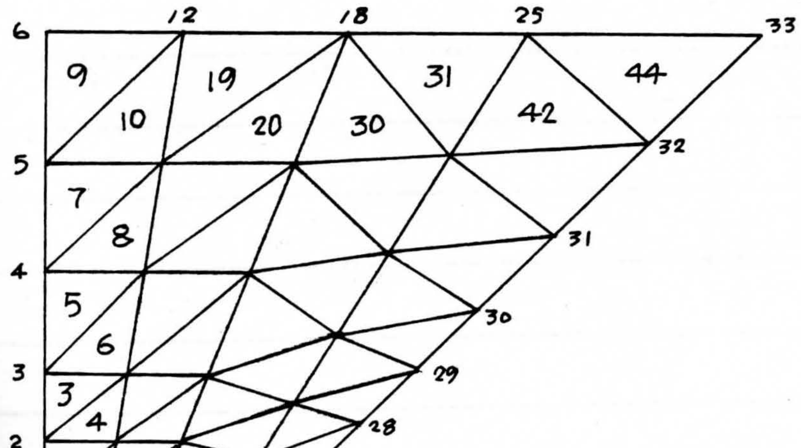
Figure (7-5) Program for Triangular Mesh Generation

Sample problem:

Consider the area shown in Figure (7-6a). The TAMG program is used to mesh the area into finite elements. The mesh includes five generating lines with five node points on the first three lines and six node points on the last two lines. The value of weighting factor is taken as 1.2 so that the node distances increase. The result of mesh generation is graphically shown in Figure (7-6b) in which thirty-three (33) nodes are defined together with forty-five (45) triangular elements. The input data, node point coordinate generation output, and element generation output are shown in Figure (7-6c).



(a)



(b)

Figure(7-6) Sample Mesh Generation

CONTROL DATA

5	1.20000			
5	0.00000	4.00000	0.00000	12.00000
5	1.00000	4.00000	2.30000	12.00000
5	2.00000	4.00000	5.00000	12.00000
6	3.00000	4.00000	8.00000	12.00000
7	4.00000	4.00000	12.00000	12.00000

X CO-ORDINATE	Y CO-ORDINATE	NODAL POINT
0.00000000	4.00000000	1
0.00000000	5.07503700	2
0.00000000	6.36508300	3
0.00000000	7.91313700	4
0.00000000	9.77080100	5
0.00000000	11.99999000	6
1.00000000	4.00000000	7
1.17469300	5.07503700	8
1.38432500	6.36508300	9
1.63588300	7.91313700	10
1.93775300	9.77080100	11
2.29999700	11.99999000	12
2.00000000	4.00000000	13
2.40313900	5.07503700	14
2.88690500	6.36508300	15
3.44742500	7.91313700	16
4.16406900	9.77080100	17
4.99999700	11.99999000	18
3.00000000	4.00000000	19
3.50352900	4.80564500	20
4.10776300	5.77242100	21
4.83284400	6.93255100	22
5.70284100	8.32470700	23
6.74705700	9.99528300	24
7.99999700	11.99999000	25
4.00000000	4.00000000	26
4.61939100	4.61939100	27
5.36266100	5.36266100	28
6.25458500	6.25458500	29
7.32489200	7.32489200	30
8.60926200	8.60926200	31
10.15050000	10.15050000	32
11.99999000	11.99999000	33

ELEMENT NO.	NODAL POINTS OF THE ELEMENTS		
1	1	2	8
2	1	8	7
3	2	3	9
4	2	9	8
5	3	4	10
6	3	10	9
7	4	5	11
8	4	11	10
9	5	6	12
10	5	12	11
11	7	8	14
12	7	14	13
13	8	9	15
14	8	15	14
15	9	10	16
16	9	16	15
17	10	11	17
18	10	17	16
19	11	12	18
20	11	18	17
21	13	14	20
22	13	20	19
23	14	15	21
24	14	21	20
25	15	16	22
26	15	22	21
27	16	17	23
28	16	23	22
29	17	18	24
30	17	24	23
31	18	25	24
32	19	20	27
33	19	27	26
34	20	21	28
35	20	28	27
36	21	22	29
37	21	29	28
38	22	23	30
39	22	30	29
40	23	24	31
41	23	31	30
42	24	25	32
43	24	32	31
44	25	33	32

```
STATEMENTS EXECUTED=      616

CORE USAGE        OBJECT CODE=    3144 BYTES,ARRAY AREA=     476 BYTES,TOTAL AREA

DIAGNOSTICS           NUMBER OF ERRORS=        0, NUMBER OF WARNINGS=        0, NUMBER

COMPILE TIME=      0.13 SEC,EXECUTION TIME=     0.09 SEC,      16.21.54    TUESDAY

      C$STOP
```

Figure (7-6c) Output of TAMG Program

7-3 Bar Element Mesh Generation of Space Truss

Three dimensional truss or space frame structures have practical use in shopping mall and sports stadium construction where long span requirements must be obtained. In many realistic situations thousands of members must be considered. Automatic mesh generation for the analysis of these structures can greatly reduce the cost of data preparation for structural analysis and design.

To date little emphasis has been placed thus far, on developing three dimensional bar element mesh generation. This situation exists because there is no "one-fits-all" computer user's program available that would efficiently satisfy its use for every type of truss structure. Since a computer user's program considers each bar member as an element in analysis, the associated geometrical configuration produces an infinite number of types of assemblage. Fortunately, one who is familiar with the Fortran programming and the Finite Element Method can write a mesh generation program for the truss structure with a certain bar configuration without serious difficulty.

Consider the space truss shown in Figure (7-7) which is defined as a single unit assembly of a larger structure. This simple model is used to demonstrate the scheme for bar element mesh generation. The intersection points of the truss members conveniently define the node point location.

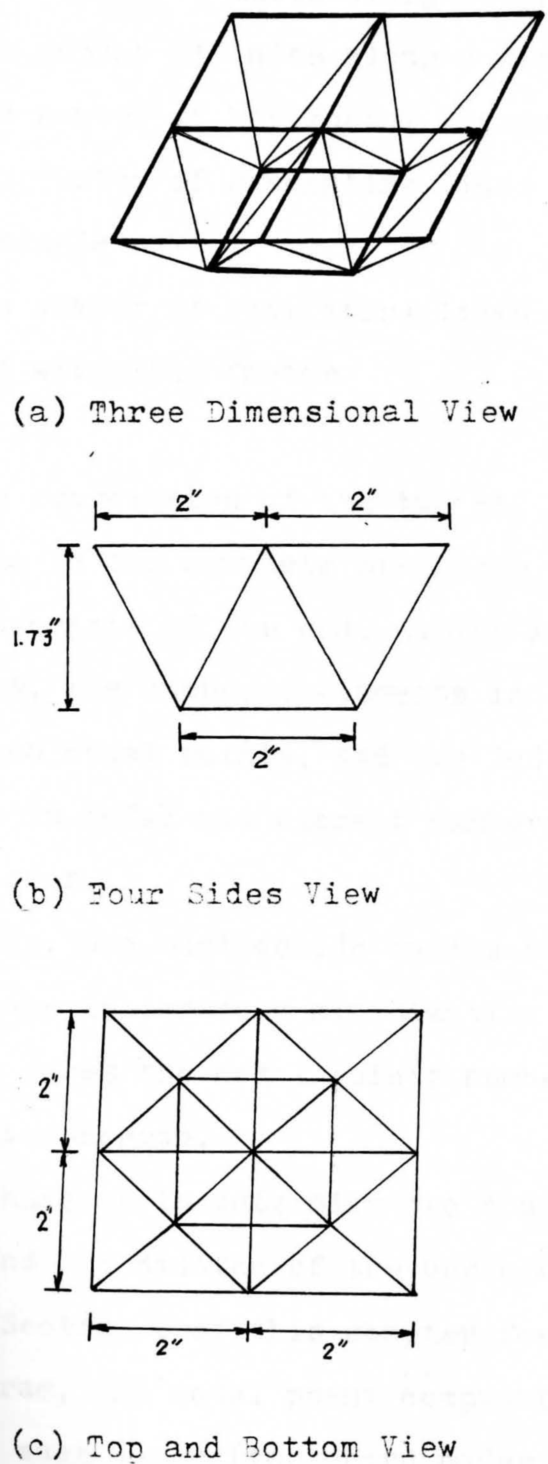


Figure (7-7) Sample Space Truss Unit

The input data consists of information containing:

LY - the number of units along y direction

LX - the number of units along x direction

NL - the number of horizontal layers of node points

NY - the number of generating lines for each layer in a single unit

NX - the number of generating lines

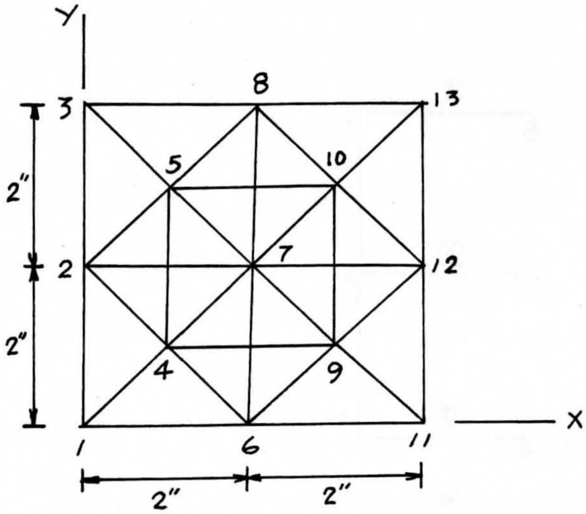
CON - the weighting factor

XF,XL
YF,YL } - the coordinates of the two end points in each generating
ZF,ZL line in the complete structure

The output consists of the node number including its x,y and z coordinates, the numbered elements including the numbers of its two end nodal points, and the index of the element location. Both nodal and element numbers are arranged in sequential order.

Initially, one must decide on the best numbering system for node points to insure a minimization of the bandwidth size. Figure (7-8) shows the nodal points numbering used by the author in this program.

The techniques to determine the nodal number along with its correspond coordinates of the bar element is the same as the one used in Section 2 of this chapter for triangular element. In this program, the nodal point computation operates twice, one time for each layer (top chord nodes and bottom chord nodes). For the top layer, after numbering the nodal points



TOP LAYER NODES
1,2,3,6,7,8,11,12,13

BOTTOM LAYER NODES
4,5,9,10

Figure (7-8) Nodal Point Numbering of the Space Truss

along nth generating line,the program skips the numbering process by factor (LYx2) and continues to node number the (n+1)th generation line. The (LYx2) nodal numbers are reserved numbers for the nth generating line in bottom layer. After all nodal points on top layer are defined, the nodal points on bottom layer are defined using those reserved numbers beginning with number ((LYx2)+2). The program stops numbering nodal points when last nodal point on the bottom layer is defined by number 2LX(4LY+1).

The element generation is developed by the eight step process as illustrated in Figure (7-9). Each step is performed by one DO loop statement, all eight DO loop statements are nested into a single master DO loop operation. The master loop executes as many times as the number of units in the x direction plus one (i.e.(2LY+1)). The complete structure mesh can be formed by repeating eight mesh steps since the

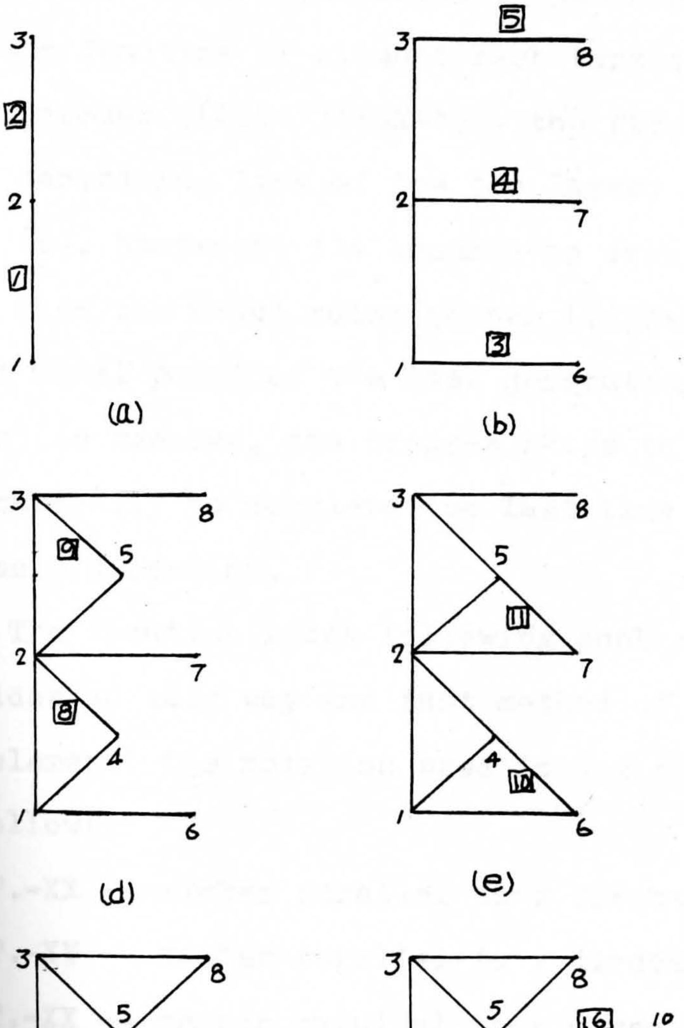


Figure (7-9) Eight Step Process Element Generation unit structure is symmetrical. Statement 70 of the program has the function of element mesh termination when the nodal point number ((4LY+1)x2LX+1) - the first nodal point of the last generating line of the top layer) is reached in executing step (b). Statement 114 ensures no over mesh is made by step (h). When the nodal point number ((4LY+1)x2LX-2LY+1) - the first nodal point of the last generating line of the bottom layer) is reached, the program skips to execute step (a) automatically to complete the last line of top chord elements in the y direction.

The location index following each element definition provides an easy way and fast method of locating a particular bar element. The notation used in the index is explained as follow:

TOP.-XX -- member parallel to x direction in the top layer

TOP.-YY -- member parallel to y direction in the top layer

BOT.-XX -- member parallel to x direction in the bottom layer

BOT.-YY -- member parallel to y direction in the bottom layer

DIA.-DL -- diagonal member, down to the left (from a top view of the central connecting point)

DIA.-DR -- diagonal member, down to the right

DIA.-UL -- diagonal member, up to the left

DIA.-UR -- diagonal member, up to the right

For example, TOP.-YY(2,3) indicates a member in the top layer parallel to the y-axis with position in the second column, third row.

This program can deal with any combination of space truss units by inputing the number of units along x and y direction.

The flow chart for the program is given in Figure (7-10). A listing of the data preparation process is given in Figure (7-11). The space truss automatic mesh generation program (STAMG) is listed in Figure (7-12).

11 READ AND WRITE THE NUMBER OF LAYERS AND THE NUMBER OF UNITS
IN X DIRECTION, THE NUMBER OF UNITS IN Y DIRECTION

READ AND WRITE THE NUMBER OF GENERATING LINES OF EACH LAYER
IN A SINGLE UNIT AND THE WEIGHTING FACTOR

READ AND WRITE THE NUMBER OF ORDER OF GENERATING LINE AND
THE COORDINATES OF TWO END POINTS OF EACH GENERATING LINE

CALCULATE AND DEFINE THE NODE NUMBERS AND ITS CORRESPONDING
X,Y,Z COORDINATES

CALCULATE AND DEFINE THE ELEMENT NUMBERS AND ITS TWO
CONNECTING NODAL POINTS

READ NEXT

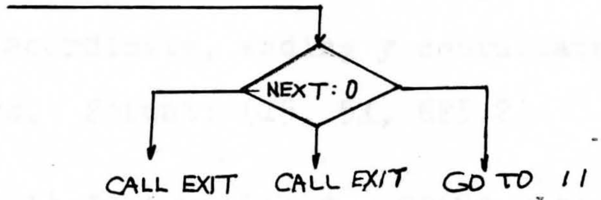


Figure (7-10) Flow Chart for STAMG Program

Data Deck:

A card containing:

the number of layers, the number of units in y direction, the number of units in x direction. Format: (3I5)

A card containing:

the number of generating lines of each layer in a single unit and the weighting factor. Format: (I5,5X, F5.2)

One card for each generating line containing:

the number of order of generating line, starting x coordinate, strating y coordinate, starting z coordinate, ending x coordinate, ending y coordinate, ending z coordinate. Format: (I5, 5X, 6F5.2)

Figure (7-11) Instruction for STAMG Data Preparation

```
$JOB
      DIMENSION NX(40).XF(40).XL(40).YF(40).YL(40).ZF(40).ZL(40)
      DIMENSION NY(40).CON(40).NOD(6).SUM1(15)
   11 READ(5.999) NL.LY.LX
      WRITE(6.998) LY.LX
      MY=LY*2
      MX=LX*2
      DO 99 L=1.NL
      WRITE(6.1002) L
      READ(5.1000) NY(L).CON(L)
      WRITE(6.1001) NY(L).CON(L)
      IF(L-2) 14.6.14
   14 NYL=MX+1
      GO TO 10
    6 NYL=MX
   10 DO 109 I=1.NYL
      READ(5.1000) NX(I).XF(I).XL(I).YF(I).YL(I).ZF(I).ZL(I)
      WRITE(6.1000) NX(I).XF(I).XL(I).YF(I).YL(I).ZF(I).ZL(I)
      IF(L-2) 1.2.1
    1 N=0
      GO TO 109
    2 N=MY+1
  109 CONTINUE
      WRITE(6.1431)
      DO 98 I=1.NYL
      IF(L-2) 7.8.7
    7 NXI=MY+1
      GO TO 30
    8 NXI=MY
   30 SUM1(1)=0.0
      SUM1(2)=1.0
      SUM=1.0
      IF(NXI-2) 190.291.190
  190 DO 250 K=3.NXI
      SUM1(K)=SUM1(K-1)*CON(L)
      SUM=SUM+SUM1(K)
  250 CONTINUE
  291 CONTINUE
      X=XF(I)
      Y=YF(I)
      Z=ZF(I)
      DO 300 J=1.NXI
      N=N+1
   20 X=(XL(I)-XF(I))*SUM1(J)/SUM+X
      Y=(YL(I)-YF(I))*SUM1(J)/SUM+Y
      Z=(ZL(I)-ZF(I))*SUM1(J)/SUM+Z
      WRITE(6.1430) N.X.Y.Z
  300 CONTINUE
      IF(L-2) 3.4.3
    3 N=N+MY
      GO TO 98
    4 N=N+MY+1
   98 CONTINUE
   99 CONTINUE
      WRITE(6.1432)
      N=1
      NSUM1=0
      NYI1=MX+1
      DO 5 I1=1.NYI1
      NXI1=MY
```

```
      DO 15 J1=1,MXI1
      NOD(1)=J1+NSUM1
      NOD(2)=NOD(1)+1
      N=N+1
      WRITE(6,1003) N,NOD(1),NOD(2),I1,J1
   15 CONTINUE
      MXI2=MY+1
      DO 25 J2=1,MXI2
      NOD(1)=J2+NSUM1
      NOD(2)=NOD(1)+MY*2+1
      NOD(3)=NOD(1)-(((MY+1)+MY)*MX+1)
      IF(NOD(3)) 13,5,5
   13 N=N+1
      WRITE(6,1007) N,NOD(1),NOD(2),I1,J2
   25 CONTINUE
      MXI3=MY
      DO 35 J3=1,MXI3
      NOD(1)=J3+NSUM1
      NOD(2)=NOD(1)+MY+1
      N=N+1
      WRITE(6,1005) N,NOD(1),NOD(2),I1,J3
   35 CONTINUE
      MXI4=MY
      DO 45 J4=1,MXI4
      NOD(1)=J4+1+NSUM1
      NOD(2)=NOD(1)+MX
      N=N+1
      WRITE(6,1006) N,NOD(1),NOD(2),I1,J4
   45 CONTINUE
      MXI5=MY
      DO 55 J5=1,MXI5
      NOD(1)=J5+MY+1+NSUM1
      NOD(2)=NOD(1)+MY
      N=N+1
      WRITE(6,1008) N,NOD(1),NOD(2),I1,J5
   55 CONTINUE
      MXI6=MY
      DO 65 J6=1,MXI6
      NOD(1)=J6+MY+1+NSUM1
      NOD(2)=NOD(1)+MY+1
      N=N+1
      WRITE(6,1009) N,NOD(1),NOD(2),I1,J6
   65 CONTINUE
      MXI7=MY-1
      DO 75 J7=1,MXI7
      NOD(1)=J7+MY+1+NSUM1
      NOD(2)=NOD(1)+1
      N=N+1
      WRITE(6,1010) N,NOD(1),NOD(2),I1,J7
   75 CONTINUE
      MXI8=MY
      DO 85 J8=1,MXI8
      NOD(1)=J8+MY+1+NSUM1
      NOD(2)=NOD(1)+MY*2+1
      NOD(3)=NOD(1)-(((MY+1)+MY)*MX-(MY-1))
      IF(NOD(3)) 23,85,85
   23 N=N+1
      WRITE(6,1011) N,NOD(1),NOD(2),I1,J8
   85 CONTINUE
      NSUM1=NSUM1+MY*2+1
```

```
         5 CONTINUE
           READ(5.1004) NEXT
           IF(NEXT-1) 12.11.11
        12 CALL EXIT
       998 FORMAT('1'.'MESH GENERATION FOR '.I2.'X'.I2.' UNITS SPACE TRUSS')
       999 FORMAT(3I5)
      1001 FORMAT('0 CONTROL CARD'./.'NO. OF GENERATING LINES ='. I2.
          1':CON ='.F3.1)
      1000 FORMAT(I5.5X.6F5.2)
      1005 FORMAT(I8.5X.2I8.17X.'DIA.-DL'.'('.I3.'.'.I3.')')
      1011 FORMAT(I8.5X.2I8.17X.'BOT.-XX'.'('.I3.'.'.I3.')')
      1010 FORMAT(I8.5X.2I8.17X.'BOT.-YY'.'('.I3.'.'.I3.')')
      1009 FORMAT(I8.5X.2I8.17X.'DIA.-UR'.'('.I3.'.'.I3.')')
      1008 FORMAT(I8.5X.2I8.17X.'DIA.-DR'.'('.I3.'.'.I3.')')
      1006 FORMAT(I8.5X.2I8.17X.'DIA.-UL'.'('.I3.'.'.I3.')')
      1007 FORMAT(I8.5X.2I8.17X.'TOP.-XX'. '('.I3.'.' .I3.')')
      1003 FORMAT(I8.5X.2I8.17X.'TOP.-YY'.'('.I3.'.'.I3.')')
      1002 FORMAT('0'.'LAYER='.1X.I2)
      1004 FORMAT(I5)
      1430 FORMAT(I7.5X.3F16.8)
      1431 FORMAT('0'.' NODAL POINT  X COORDINATE  Y COORDINATE   Z COORDINA
          1TE')
      1432 FORMAT('1 ELEMENT NO  NODAL POINTS OF THE ELEMENT  LOCATION')
           STOP
           END

   $ENTRY
```

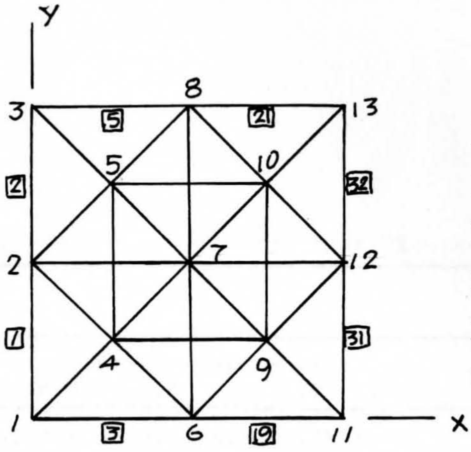
Figure (7-12) STAMG Main Program

Sample problem:

Consider the space truss problems shown in Figure (7-13). A one unit truss is shown in Figure (7-13a) with LY=LX=1. It consists of thirteen (13) nodal points and thirty-two (32) members. The unit has two horizontal layers of nodes, NL=2. In the top layer the unit has three (3) generating lines for nodal point coordinate definition, NY(1)=3; in the lower layer the unit has two (2) generating lines NY(2)=2. Since the nodal points are equally spaced, CON=1.0. The computer input data and output computations are shown in Figure(7-13c). Given the geometric data for ten starting and end points (i.e, five (5) generating lines), the program calculates the node point coordinates for the remaining three node points. In addition, the thirty-two members are numbered and each pair of element nodes are defined.

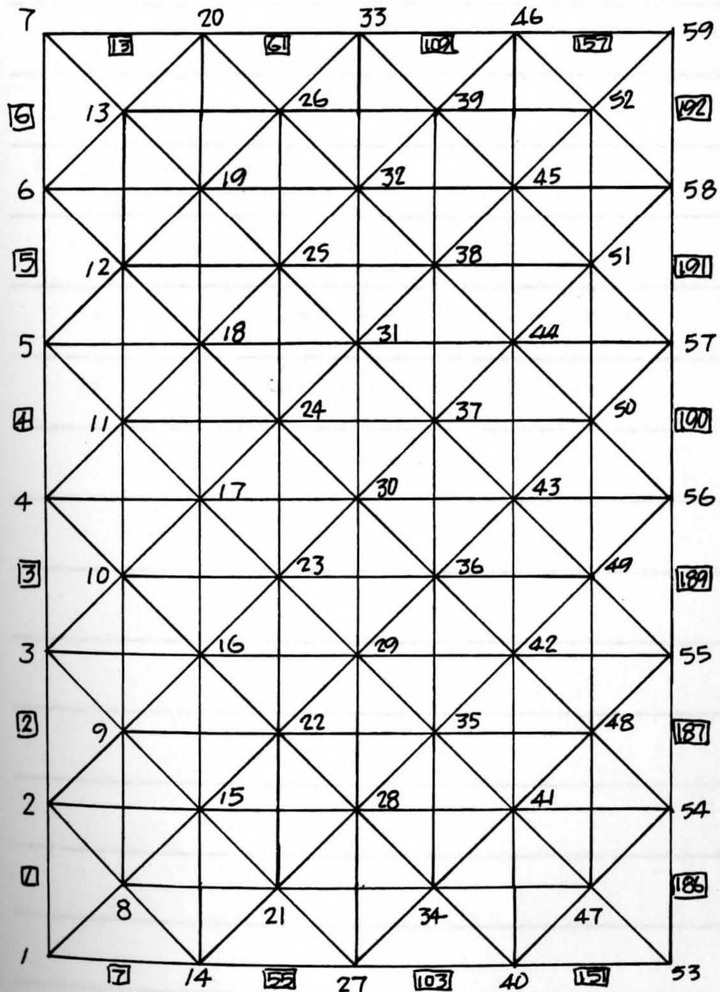
A more complex geometrical truss is considered in Figure (7-13b), with fifty-nine (59) nodes and one hundred and ninety-two (192) members. Eighteen (18) nodal points coordinates are input. The program calculates the remaining forty one (41) nodal points as well as numbering the 192 elements and giving their indicated position.

A (13x20) structural system (i.e, 260 units with 2147 nodal points and 6175 members) has been investigated by the author. Since the computer output is somewhat lengthy, it is not listed in this thesis.



NL = 2
LY = 1
LX = 1
NY(1)= 3
NY(2)= 2
CON = 1.0

Figure (7-13a) Final Mesh of A One Unit Space Truss



NL = 2
LY = 3
LX = 2
NY(1)= 5
NY(2)= 4
CON = 1.0

Figure (7-13a) Final Mesh of 3x2 Unit Space Truss

Figure (7-13) Sample Examples of Space Truss

MESH GENERATION FOR 1X 1 UNITS SPACE TRUSS

LAYER= 1

CONTROL CARD
NO. OF GENERATING LINES = 3:CON =1.0

```
1      0.00 0.00 0.00 4.00 1.73 1.73
2      2.00 2.00 0.00 4.00 1.73 1.73
3      4.00 4.00 0.00 4.00 1.73 1.73
```

NODAL POINT	X COORDINATE	Y COORDINATE	Z COORDINATE
1	0.00000000	0.00000000	1.72999900
2	0.00000000	2.00000000	1.72999900
3	0.00000000	4.00000000	1.72999900
6	2.00000000	0.00000000	1.72999900
7	2.00000000	2.00000000	1.72999900
8	2.00000000	4.00000000	1.72999900
11	4.00000000	0.00000000	1.72999900
12	4.00000000	2.00000000	1.72999900
13	4.00000000	4.00000000	1.72999900

LAYER= 2

CONTROL CARD
NO. OF GENERATING LINES = 2:CON =1.0

```
1      1.00 1.00 1.00 3.00 0.00 0.00
2      3.00 3.0010.00 3.00 0.00 0.00
```

NODAL POINT	X COORDINATE	Y COORDINATE	Z COORDINATE
4	1.00000000	1.00000000	0.00000000
5	1.00000000	3.00000000	0.00000000
9	3.00000000	10.00000000	0.00000000
10	3.00000000	3.00000000	0.00000000

ELEMENT NO	NODAL POINTS OF THE ELEMENT		LOCATION
1	1	2	TOP.-YY(1. 1)
2	2	3	TOP.-YY(1. 2)
3	1	6	TOP.-XX(1. 1)
4	2	7	TOP.-XX(1. 2)
5	3	8	TOP.-XX(1. 3)
6	1	4	DIA.-DL(1. 1)
7	2	5	DIA.-DL(1. 2)
8	2	4	DIA.-UL(1. 1)
9	3	5	DIA.-UL(1. 2)
10	4	6	DIA.-DR(1. 1)
11	5	7	DIA.-DR(1. 2)
12	4	7	DIA.-UR(1. 1)
13	5	8	DIA.-UR(1. 2)
14	4	5	BOT.-YY(1. 1)
15	4	9	BOT.-XX(1. 1)
16	5	10	BOT.-XX(1. 2)
17	6	7	TOP.-YY(2. 1)
18	7	8	TOP.-YY(2. 2)
19	6	11	TOP.-XX(2. 1)
20	7	12	TOP.-XX(2. 2)
21	8	13	TOP.-XX(2. 3)
22	6	9	DIA.-DL(2. 1)
23	7	10	DIA.-DL(2. 2)
24	7	9	DIA.-UL(2. 1)
25	8	10	DIA.-UL(2. 2)
26	9	11	DIA.-DR(2. 1)
27	10	12	DIA.-DR(2. 2)
28	9	12	DIA.-UR(2. 1)
29	10	13	DIA.-UR(2. 2)
30	9	10	BOT.-YY(2. 1)
31	11	12	TOP.-YY(3. 1)
32	12	13	TOP.-YY(3. 2)

```
MESH GENERATION FOR  3X 2 UNITS SPACE TRUSS

LAYER=  1

CONTROL CARD
0. OF GENERATING LINES = 3:CON =1.0
  1        0.00 0.00 0.0012.00 1.73 1.73
  2        2.00 2.00 0.0012.0010.30 1.73
  3        4.0040.00 0.0012.00 1.73 1.73
  4        6.00 6.00 0.0012.00 1.73 1.73
  5        8.00 8.00 0.0012.00 1.73 1.73
```

NODAL POINT	X COORDINATE	Y COORDINATE	Z COORDINATE
1	0.00000000	0.00000000	1.72999900
2	0.00000000	2.00000000	1.72999900
3	0.00000000	4.00000000	1.72999900
4	0.00000000	6.00000000	1.72999900
5	0.00000000	8.00000000	1.72999900
6	0.00000000	10.00000000	1.72999900
7	0.00000000	12.00000000	1.72999900
14	2.00000000	0.00000000	10.30000000
15	2.00000000	2.00000000	8.87166600
16	2.00000000	4.00000000	7.44333300
17	2.00000000	6.00000000	6.01500000
18	2.00000000	8.00000000	4.58666700
19	2.00000000	10.00000000	3.15833300
20	2.00000000	12.00000000	1.73000000
27	4.00000000	0.00000000	1.72999900
28	10.00000000	2.00000000	1.72999900
29	16.00000000	4.00000000	1.72999900
30	22.00000000	6.00000000	1.72999900
31	28.00000000	8.00000000	1.72999900
32	34.00000000	10.00000000	1.72999900
33	40.00000000	12.00000000	1.72999900
40	6.00000000	0.00000000	1.72999900
41	6.00000000	2.00000000	1.72999900
42	6.00000000	4.00000000	1.72999900
43	6.00000000	6.00000000	1.72999900
44	6.00000000	8.00000000	1.72999900
45	6.00000000	10.00000000	1.72999900
46	6.00000000	12.00000000	1.72999900
53	8.00000000	0.00000000	1.72999900
54	8.00000000	2.00000000	1.72999900
55	8.00000000	4.00000000	1.72999900
56	8.00000000	6.00000000	1.72999900
57	8.00000000	8.00000000	1.72999900
58	8.00000000	10.00000000	1.72999900
59	8.00000000	12.00000000	1.72999900

```
LAYER=  2

CONTROL CARD
0. OF GENERATING LINES = 2:CON =1.0
  1        1.00 1.00 1.0011.00 0.00 0.00
  2        3.00 3.00 1.0011.00 0.00 0.00
  3        5.00 5.00 1.0011.00 0.00 0.00
  4        7.00 7.00 1.0011.00 0.00 0.00
```

NODAL POINT	X COORDINATE	Y COORDINATE	Z COORDINATE
8	1.00000000	1.00000000	0.00000000

9	1.00000000	3.00000000	0.00000000
10	1.00000000	5.00000000	0.00000000
11	1.00000000	7.00000000	0.00000000
12	1.00000000	9.00000000	0.00000000
13	1.00000000	11.00000000	0.00000000
21	3.00000000	1.00000000	0.00000000
22	3.00000000	3.00000000	0.00000000
23	3.00000000	5.00000000	0.00000000
24	3.00000000	7.00000000	0.00000000
25	3.00000000	9.00000000	0.00000000
26	3.00000000	11.00000000	0.00000000
34	5.00000000	1.00000000	0.00000000
35	5.00000000	3.00000000	0.00000000
36	5.00000000	5.00000000	0.00000000
37	5.00000000	7.00000000	0.00000000
38	5.00000000	9.00000000	0.00000000
39	5.00000000	11.00000000	0.00000000
47	7.00000000	1.00000000	0.00000000
48	7.00000000	3.00000000	0.00000000
49	7.00000000	5.00000000	0.00000000
50	7.00000000	7.00000000	0.00000000
51	7.00000000	9.00000000	0.00000000
52	7.00000000	11.00000000	0.00000000

ELEMENT NO.	NODAL POINTS OF THE ELEMENT		LOCATION
1	1	2	TOP.-YY(1. 1)
2	2	3	TOP.-YY(1. 2)
3	3	4	TOP.-YY(1. 3)
4	4	5	TOP.-YY(1. 4)
5	5	6	TOP.-YY(1. 5)
6	6	7	TOP.-YY(1. 6)
7	1	14	TOP.-XX(1. 1)
8	2	15	TOP.-XX(1. 2)
9	3	16	TOP.-XX(1. 3)
10	4	17	TOP.-XX(1. 4)
11	5	18	TOP.-XX(1. 5)
12	6	19	TOP.-XX(1. 6)
13	7	20	TOP.-XX(1. 7)
14	1	8	DIA.-DL(1. 1)
15	2	9	DIA.-DL(1. 2)
16	3	10	DIA.-DL(1. 3)
17	4	11	DIA.-DL(1. 4)
18	5	12	DIA.-DL(1. 5)
19	6	13	DIA.-DL(1. 6)
20	2	6	DIA.-UL(1. 1)
21	3	7	DIA.-UL(1. 2)
22	4	8	DIA.-UL(1. 3)
23	5	9	DIA.-UL(1. 4)
24	6	10	DIA.-UL(1. 5)
25	7	11	DIA.-UL(1. 6)
26	8	14	DIA.-DR(1. 1)
27	9	15	DIA.-DR(1. 2)
28	10	16	DIA.-DR(1. 3)
29	11	17	DIA.-DR(1. 4)
30	12	18	DIA.-DR(1. 5)
31	13	19	DIA.-DR(1. 6)
32	8	15	DIA.-UR(1. 1)
33	9	16	DIA.-UR(1. 2)
34	10	17	DIA.-UR(1. 3)
35	11	18	DIA.-UR(1. 4)
36	12	19	DIA.-UR(1. 5)
37	13	20	DIA.-UR(1. 6)
38	8	9	BOT.-YY(1. 1)
39	9	10	BOT.-YY(1. 2)
40	10	11	BOT.-YY(1. 3)
41	11	12	BOT.-YY(1. 4)
42	12	13	BOT.-YY(1. 5)
43	8	21	BOT.-XX(1. 1)
44	9	22	BOT.-XX(1. 2)
45	10	23	BOT.-XX(1. 3)
46	11	24	BOT.-XX(1. 4)
47	12	25	BOT.-XX(1. 5)
48	13	26	BOT.-XX(1. 6)
49	14	15	TOP.-YY(2. 1)
50	15	16	TOP.-YY(2. 2)
51	16	17	TOP.-YY(2. 3)
52	17	18	TOP.-YY(2. 4)
53	18	19	TOP.-YY(2. 5)
54	19	20	TOP.-YY(2. 6)
55	14	27	TOP.-XX(2. 1)
56	15	28	TOP.-XX(2. 2)
57	16	29	TOP.-XX(2. 3)
58	17	30	TOP.-XX(2. 4)
59	18	31	TOP.-XX(2. 5)

60	19	32	TOP.-XX(2. 6)
61	20	33	TOP.-XX(2. 7)
62	14	21	DIA.-DL(2. 1)
63	15	22	DIA.-DL(2. 2)
64	16	23	DIA.-DL(2. 3)
65	17	24	DIA.-DL(2. 4)
66	18	25	DIA.-DL(2. 5)
67	19	26	DIA.-DL(2. 6)
68	15	19	DIA.-UL(2. 1)
69	16	20	DIA.-UL(2. 2)
70	17	21	DIA.-UL(2. 3)
71	18	22	DIA.-UL(2. 4)
72	19	23	DIA.-UL(2. 5)
73	20	24	DIA.-UL(2. 6)
74	21	27	DIA.-DR(2. 1)
75	22	28	DIA.-DR(2. 2)
76	23	29	DIA.-DR(2. 3)
77	24	30	DIA.-DR(2. 4)
78	25	31	DIA.-DR(2. 5)
79	26	32	DIA.-DR(2. 6)
80	21	28	DIA.-UR(2. 1)
81	22	29	DIA.-UR(2. 2)
82	23	30	DIA.-UR(2. 3)
83	24	31	DIA.-UR(2. 4)
84	25	32	DIA.-UR(2. 5)
85	26	33	DIA.-UR(2. 6)
86	21	22	BOT.-YY(2. 1)
87	22	23	BOT.-YY(2. 2)
88	23	24	BOT.-YY(2. 3)
89	24	25	BOT.-YY(2. 4)
90	25	26	BOT.-YY(2. 5)
91	21	34	BOT.-XX(2. 1)
92	22	35	BOT.-XX(2. 2)
93	23	36	BOT.-XX(2. 3)
94	24	37	BOT.-XX(2. 4)
95	25	38	BOT.-XX(2. 5)
96	26	39	BOT.-XX(2. 6)
97	27	28	TOP.-YY(3. 1)
98	28	29	TOP.-YY(3. 2)
99	29	30	TOP.-YY(3. 3)
100	30	31	TOP.-YY(3. 4)
101	31	32	TOP.-YY(3. 5)
102	32	33	TOP.-YY(3. 6)
103	27	40	TOP.-XX(3. 1)
104	28	41	TOP.-XX(3. 2)
105	29	42	TOP.-XX(3. 3)
106	30	43	TOP.-XX(3. 4)
107	31	44	TOP.-XX(3. 5)
108	32	45	TOP.-XX(3. 6)
109	33	46	TOP.-XX(3. 7)
110	27	34	DIA.-DL(3. 1)
111	28	35	DIA.-DL(3. 2)
112	29	36	DIA.-DL(3. 3)
113	30	37	DIA.-DL(3. 4)
114	31	38	DIA.-DL(3. 5)
115	32	39	DIA.-DL(3. 6)
116	28	32	DIA.-UL(3. 1)
117	29	33	DIA.-UL(3. 2)
118	30	34	DIA.-UL(3. 3)
119	31	35	DIA.-UL(3. 4)

120	32	36	DIA.-UL(3. 5)
~~121~~	~~33~~	~~37~~	~~DIA.-UL(3. 6)~~
122	34	40	DIA.-DR(3. 1)
123	35	41	DIA.-DR(3. 2)
~~124~~	~~36~~	~~42~~	~~DIA.-DR(3. 3)~~
125	37	43	DIA.-DR(3. 4)
126	38	44	DIA.-DR(3. 5)
~~127~~	~~39~~	~~45~~	~~DIA.-DR(3. 6)~~
128	34	41	DIA.-UR(3. 1)
129	35	42	DIA.-UR(3. 2)
~~130~~	~~36~~	~~43~~	~~DIA.-UR(3. 3)~~
131	37	44	DIA.-UR(3. 4)
132	38	45	DIA.-UR(3. 5)
~~133~~	~~39~~	~~46~~	~~DIA.-UR(3. 6)~~
134	34	35	BOT.-YY(3. 1)
135	35	36	BOT.-YY(3. 2)
~~136~~	~~36~~	~~37~~	~~BOT.-YY(3. 3)~~
137	37	38	BOT.-YY(3. 4)
138	38	39	BOT.-YY(3. 5)
~~139~~	~~34~~	~~47~~	~~BOT.-XX(3. 1)~~
140	35	48	BOT.-XX(3. 2)
141	36	49	BOT.-XX(3. 3)
~~142~~	~~37~~	~~50~~	~~BOT.-XX(3. 4)~~
143	38	51	BOT.-XX(3. 5)
144	39	52	BOT.-XY(3. 6)
~~145~~	~~40~~	~~41~~	~~TOP.-YY(4. 1)~~
146	41	42	TOP.-YY(4. 2)
147	42	43	TOP.-YY(4. 3)
~~148~~	~~43~~	~~44~~	~~TOP.-YY(4. 4)~~
149	44	45	TOP.-YY(4. 5)
150	45	46	TOP.-YY(4. 6)
~~151~~	~~40~~	~~53~~	~~TOP.-XX(4. 1)~~
152	41	54	TOP.-XX(4. 2)
153	42	55	TOP.-XX(4. 3)
~~154~~	~~43~~	~~56~~	~~TOP.-XX(4. 4)~~
155	44	57	TOP.-XX(4. 5)
156	45	58	TOP.-XX(4. 6)
~~157~~	~~46~~	~~59~~	~~TOP.-XX(4. 7)~~
158	40	47	DIA.-DL(4. 1)
159	41	48	DIA.-DL(4. 2)
~~160~~	~~42~~	~~49~~	~~DIA.-DL(4. 3)~~
161	43	50	DIA.-DL(4. 4)
162	44	51	DIA.-DL(4. 5)
~~163~~	~~45~~	~~52~~	~~DIA.-DL(4. 6)~~
164	41	45	DIA.-UL(4. 1)
165	42	46	DIA.-UL(4. 2)
~~166~~	~~43~~	~~47~~	~~DIA.-UL(4. 3)~~
167	44	48	DIA.-UL(4. 4)
168	45	49	DIA.-UL(4. 5)
~~169~~	~~46~~	~~50~~	~~DIA.-UL(4. 6)~~
170	47	53	DIA.-DR(4. 1)
171	48	54	DIA.-DR(4. 2)
~~172~~	~~49~~	~~55~~	~~DIA.-DR(4. 3)~~
173	50	56	DIA.-DR(4. 4)
174	51	57	DIA.-DR(4. 5)
~~175~~	~~52~~	~~58~~	~~DIA.-DR(4. 6)~~
176	47	54	DIA.-UR(4. 1)
177	48	55	DIA.-UR(4. 2)
~~178~~	~~49~~	~~56~~	~~DIA.-UR(4. 3)~~
179	50	57	DIA.-UR(4. 4).

```
180          51       58                DIA.-UR(  4.  5)
181          52       59                DIA.-UR(  4.  6)
182          47       48                BOT.-YY(  4.  1)
183          48       49                BOT.-YY(  4.  2)
184          49       50                BOT.-YY(  4.  3)
185          50       51                BOT.-YY(  4.  4)
186          51       52                BOT.-YY(  4.  5)
187          53       54                TOP.-YY(  5.  1)
188          54       55                TOP.-YY(  5.  2)
189          55       56                TOP.-YY(  5.  3)
190          56       57                TOP.-YY(  5.  4)
191          57       58                TOP.-YY(  5.  5)
192          58       59                TOP.-YY(  5.  6)

STATEMENTS EXECUTED=     2323

CORE USAGE        OBJECT CODE=     6336 BYTES,ARRAY AREA=     1524 BYTES,TOTAL AREA AV

DIAGNOSTICS         NUMBER OF ERRORS=      0, NUMBER OF WARNINGS=      0, NUMBER O

COMPILE TIME=      0.23 SEC,EXECUTION TIME=      0.31 SEC,     15.21.15     MONDAY

        C$STOP
```

Figure (7-13c) Output Processed by STAMG Program

7-4 Computer Program for the LST with Mesh Generation

In this section a combination of program SP33B developed in Chapter IV and the techniques of automatic mesh generation for the LST are combined into one compact and efficient program. The program may be used to analyse any two dimensional plane stress problem. The program is given in Figure (7-14). The mesh generation scheme is inserted into the program starting at line 24 and ending at line 101. The remainder of the program is essentially program SP33B with some minor changes in DIMENSION statements and CALL statements.

To prepare data input for the program, one uses a similar format given in Section 2 of this Chapter which replaces the data input used in program SP33B.

```
$JOB              JP.TIME=(3)
C            *****     MAIN SP33B     *****        SEP 25.1979
C   STRESSES IN A PLATE USING LINEAR STRAIN TRIANGLES    JP PAN
C        GUIDED NODES
      DIMENSION NX(20).YL(20).XF(20).YF(20).XL(20).NOD(10).SUM1(15)
      DIMENSION NCON(6.150).T(150)
      DIMENSION F(300).U(300).IFX(300).TRIG(300).XY(2.150).S(60.300)
      DIMENSION NI(12)
      DIMENSION E(12.12).B(3.12).AI(12.12).X(3).Y(3).NAME(20)
      DIMENSION DB(3.12).D(3.3).BDB(12.12)
      DOUBLE PRECISION XX.YY
      DOUBLE PRECISION S.F.U.XY.TRIG.T.E.B.AI.X.Y
      DOUBLE PRECISION XL.XF.YL.YF.SUM1.CON
      DOUBLE PRECISION DB.D.BDB.YM.GNU.DBIJ.V
      LBAND=60
    1 WRITE(6.102)
      READ(5.100)NAME
      WRITE(6.101)NAME
      READ(5.103)YM.GNU
      WRITE(6.104)YM.GNU
      CALL DB03B(YM.GNU.D.B.BDB)
      DO 4 I=1.3
      DO 4 J=1.12
      DBIJ=0.
      DO 3 IJ=1.3
    3 DBIJ=DBIJ+D(I.IJ)*B(IJ.J)
    4 DB(I.J)=DBIJ
      READ(5.1000) NY.CON
      WRITE(6.1001) NY.CON
      DO 99 I=1.NY
      READ(5.1000) NX(I).XF(I).XL(I).YF(I).YL(I)
      WRITE(6.1000) NX(I).XF(I).XL(I).YF(I).YL(I)
   99 CONTINUE
      WRITE(6.1431)
      N=-1
      NE=2
      DO 350 I=1.NY
      NXI=NX(I)+1
      SUM1(1)=0.D0
      SUM1(2)=1.D0
      SUM=1.D0
      IF(NXI-2) 190.291.190
  190 DO 250 K=3.NXI
      SUM1(K)=SUM1(K-1)*CON
      SUM=SUM+SUM1(K)
  250 CONTINUE
  291 CONTINUE
      XX=XF(I)
      YY=YF(I)
      DO 300 J=1.NXI
      N=N+2
      XX=(XL(I)-XF(I))*SUM1(J)/SUM+XX
      YY=(YL(I)-YF(I))*SUM1(J)/SUM+YY
      WRITE(6.1430) N.XX.YY
      XY(1.N)=XX
      XY(2.N)=YY
  300 CONTINUE
      N=N+2*NXI-2
  350 CONTINUE
      READ(5.1002) NN
```

```
      NDF=NE*NN
      WRITE(6,1999)NDF
 1002 FORMAT(I5)
 1999 FORMAT('0SYSTEM HAS',I4,'DEGREES OF FREEDOM')
      WRITE(6,1432)
 1432 FORMAT('0ELEM. NO.   THICHNESS       CONNECTING NODE NUMBERS')
      NE=0
      NBAND=0
      NN1=NN-1
      N=0
      NSUM=0
      NYI=NY-1
      DO 600 I=1,NYI
      NSUM1=0
      NXI=NX(I)
      DO 500 J=1,NXI
      NOD(1)=J+NSUM+NSUM1
      NOD(2)=NOD(1)+2*NXI+1
      NOD(3)=NOD(2)+2*NXI+1
      NOD(4)=NOD(3)+1
      NOD(5)=NOD(4)+1
      NOD(6)=NOD(1)+2*NXI+2
      NOD(7)=NOD(1)+1
      NOD(8)=NOD(7)+1
      NOD(9)=NOD(8)+2*NXI+1
      N=N+1
      NI(1)=NOD(1)
      NI(2)=NOD(6)
      NI(3)=NOD(5)
      NI(4)=NOD(9)
      NI(5)=NOD(8)
      NI(6)=NOD(7)
      CALL CN06B(LBAND,NN,NE,NCON,T,NE,NBAND,NN1,N,NI)
      N=N+1
      NI(1)=NOD(1)
      NI(2)=NOD(2)
      NI(3)=NOD(3)
      NI(4)=NOD(4)
      NI(5)=NOD(5)
      NI(6)=NOD(6)
      CALL CN06B(LBAND,NN,NE,NCON,T,NE,NBAND,NN1,N,NI)
      NSUM1=NSUM1+1
  500 CONTINUE
      NSUM=NSUM+4*NXI+2
  600 CONTINUE
      NBAND=NE*(NBAND+1)
      WRITE(6,1105) NBAND
      IF(LBAND-NBAND) 122,124,124
  122 WRITE(6,1106)
 1105 FORMAT('0BAND WIDTH=',I5)
 1106 FORMAT('0 **** BAND WIDTH EXEEDS LIMIT ****')
 1000 FORMAT(I5,5X,6F10.5)
 1001 FORMAT('0 CONTROL DATA',/,I5,5X,6F10.5)
 1431 FORMAT('0NODAL POINT    X-COORDINATE     Y-COORDINATE')
 1430 FORMAT(I4,2F16.8)
  124 DO 5 J=1,NDF
      TRIG(J)=0.
      DO 5 I=1,NBAND
    5 S(I,J)=0.
      REWIND 1
```

```
      DO 25 IE=1.NE
      CALL AI02B(IE.NCON.XY.T.AI.X.Y)
      WRITE(1) AI
      V=T(IE)*.5*(X(1)*(Y(2)-Y(3))+X(2)*(Y(3)-Y(1))+X(3)*(Y(1)-Y(2)))
      CALL ES06B(V.AI.BDB.X.Y.E)
   25 CALL AS063(IE.NCON.E.S.2.6.12.LBAND)
      END FILE 1
      REWIND 4
      DO 26 I=1.NBAND
   26 WRITE(4)(S(I.J).J=1.NDF)
      END FILE 4
    2 CALL KE013(E.NDF)
      CALL KU01B(U.IEX.NDF)
      CALL GN02B(S.U.E.IEX.NBAND.LBAND.TRIG)
      CALL GE02B(E.U.S.IEX.NBAND).NDF.LBAND)
      REWIND 4
      DO 30 I=1.NBAND
   30 READ(4)(S(I.J).J=1.NDF)
      CALL RU01B(U.TRIG.NDF)
      CALL EU103(S.U.NBAND.NDF.2.LBAND)
      REWIND 1
      WRITE(6.106)
      DO 45 IE=1.NE
      READ(1) AI
   45 CALL ST06B(DB.AI.U.NCON.IE)
      READ(5.108)NEXT
      IF(NEXT-1)47.1.2
   47 CALL EXIT
  100 FORMAT(20A4)
  101 FORMAT('0CASE TITLE  --- '.20A4)
  102 FORMAT('1                MAIN SP33B   SEP 25.1979'./.' STRESSES IN IN-P
     1LANE LOADED PLATE USING LINEAR STRAIN TRIANGLES  (GUIDED NODES)')
  103 FORMAT(2F10.5)
  104 FORMAT('0YOUNGS MODULUS='.F10.3.'  POISSONS RATIO='.F6.3)
  106 FORMAT('0ELEM.NO.          SXX            SYY            SXY          THETA
     1          PS1           PS2')
  108 FORMAT(I5)
      STOP
      END
C
C
C
C
C
C
C
C
C
C
C
C
C
C
C
C
C
C
C
C
C
```

Figure (7-14) Computer Program for LST with Mesh Generation

7-5 Application of Automatic Mesh Generation with the LST

A circular ring compressed by two equal and opposite concentrated forces acting along a diametrical line as shown in Figure (7-15) is considered. This problem is specifically chosen to illustrate the efficiency of using the mesh generation scheme. In addition a comparison of accuracy between the LST finite element solution and the exact functional solution as given by Timoshenko[11] is made.

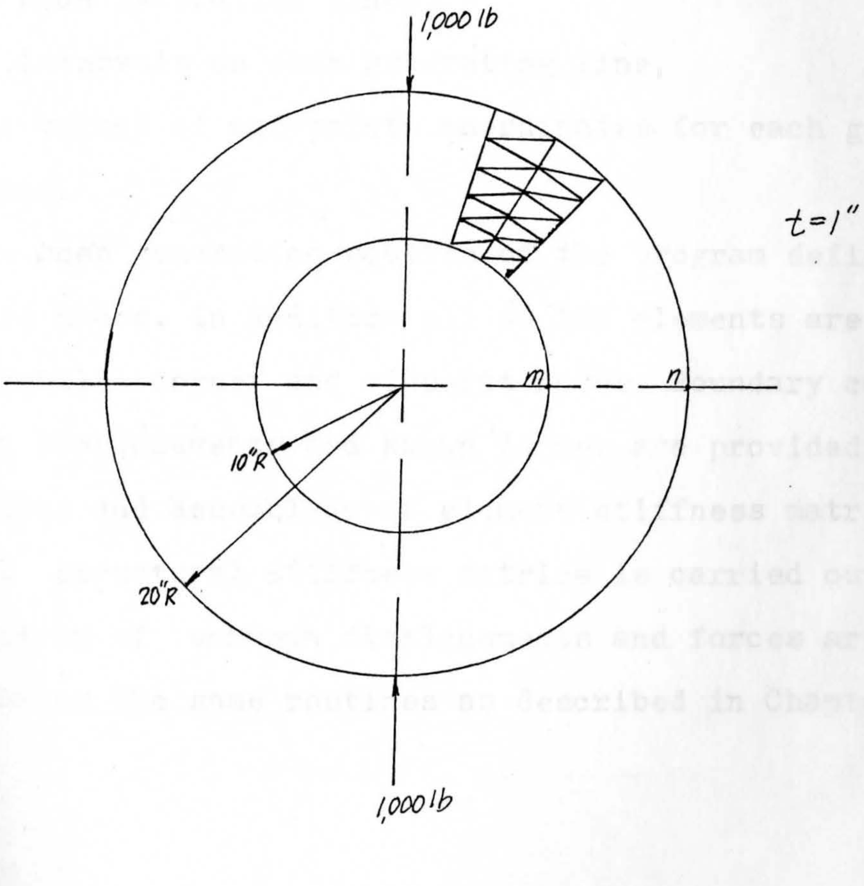


Figure (7-15) Thin Circular Ring with Concentrated Loads

As shown in Figure (7-15), the concentrated loads have two axes of symmetry, vertical and horizontal. Thus, one may investigate only one-quarter of disc. Due to computer capacity constraints at YSU, a maximum of sixty (60) elements are defined for this problem. The final nodal point and element mesh (Figure(7-16)) is constructed using the following input data:

1). 7 node generation lines,

2). 5 intervals on each generating line,

3). 14 values of end points coordinates for each generating line.

The mesh generation routine of the program defines all 42 corner nodes. In addition all 60 LST elements are numbered with essential corner and midpoint nodes. Boundary conditions on known displacements and known forces are provided. The computation and assemblage of element stiffness matrices into the structural stiffness matrice is carried out and the solution of unknown displacements and forces are solved by following the same routines as described in Chapter IV.

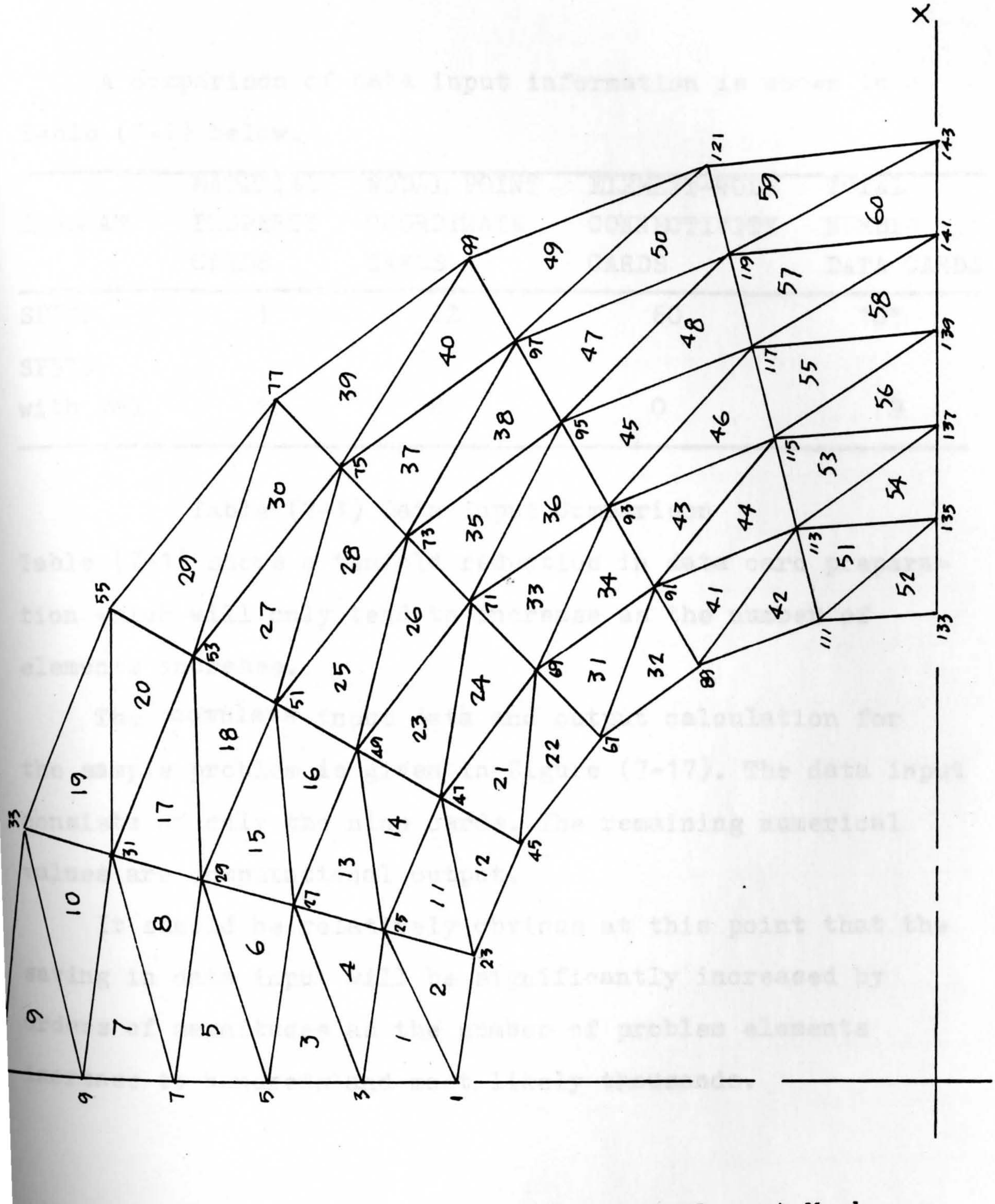


Figure (7-16) Final Nodal Point and Element Mesh

A comparison of data input information is shown in Table (7-1) below.

PROGRAM	MATERIAL PROPERTY CARDS	NODAL POINT COORDINATE CARDS	ELEMENT-NODE CONNECTIVITY CARDS	TOTAL NUMBER DATA CARDS
SP33B	1	42	60	103
SP33B with AMG	1	8	0	9

Table (7-1) Data Input Comparison

Table (7-1) shows a tenfold reduction in data card preparation which will only tend to increase as the number of elements increase.

The complete input data and output calculation for the sample problem is given in Figure (7-17). The data input consists of only the nine cards. The remaining numerical values are computational output.

It should be relatively obvious at this point that the saving in data input will be significantly increased by orders of magnitudes as the number of problem elements increase to hundreds and most likely thousands.

STRESSES IN IN-PLANE LOADED PLATE USING LINEAR STRAIN TRIANGLES (GUIDED NODES)

CASE TITLE --- STRESSES ANALYSIS IN A DISC PLATE USING LST AND AMG

YOUNGS MODULUS= 0.300D 08 POISSONS RATIO= 0.300

CONTROL DATA

7	1.00000			
5	0.00000	0.00000	10.00000	20.00000
5	2.56000	5.18000	9.66000	19.31000
5	5.00000	10.00000	8.66000	17.32000
5	7.07000	14.14000	7.07000	14.14000
5	8.66000	17.32000	5.00000	10.00000
5	9.66000	19.32000	2.59000	5.18000
5	10.00000	20.00000	0.00000	0.00000

NODAL POINT	X-COORDINATE	Y-COORDINATE
1	0.00000000	10.00000000
3	0.00000000	12.00000000
5	0.00000000	14.00000000
7	0.00000000	16.00000000
9	0.00000000	18.00000000
11	0.00000000	20.00000000
23	2.56000000	9.66000000
25	3.08400000	11.59000000
27	3.60800000	13.52000000
29	4.13200000	15.45000000
31	4.65600000	17.38000000
33	5.18000000	19.31000000
45	5.00000000	8.66000000
47	6.00000000	10.39200000
49	7.00000000	12.12400000
51	8.00000000	13.85600000
53	9.00000000	15.58800000
55	10.00000000	17.32000000
67	7.07000000	7.07000000
69	8.48400000	8.48400000
71	9.89800000	9.89800000
73	11.31200000	11.31200000
75	12.72600000	12.72600000
77	14.14000000	14.14000000
89	8.66000000	5.00000000
91	10.39200000	6.00000000
93	12.12400000	7.00000000
95	13.85600000	8.00000000
97	15.58800000	9.00000000
99	17.32000000	10.00000000
111	9.66000000	2.59000000
113	11.59200000	3.10800000
115	13.52400000	3.62600000
117	15.45600000	4.14400000
119	17.38800000	4.66200000
121	19.32000000	5.18000000
133	10.00000000	0.00000000
135	12.00000000	0.00000000
137	14.00000000	0.00000000
139	16.00000000	0.00000000
141	18.00000000	0.00000000
143	20.00000000	0.00000000

SYSTEM HAS 286DEGREES OF FREEDOM

ELEM. NO.	THICHNESS	CONNECTING NODE NUMBERS					
1	1.00000	1	13	25	14	3	2
~~2~~	~~1.00000~~	~~1~~	~~12~~	~~23~~	~~24~~	~~25~~	~~13~~
3	1.00000	3	15	27	16	5	4
4	1.00000	3	14	25	26	27	15
~~5~~	~~1.00000~~	~~5~~	~~17~~	~~29~~	~~18~~	~~7~~	~~6~~
6	1.00000	5	16	27	28	29	17
7	1.00000	7	19	31	20	9	8
~~8~~	~~1.00000~~	~~7~~	~~18~~	~~29~~	~~30~~	~~31~~	~~19~~
9	1.00000	9	21	33	22	11	10
10	1.00000	9	20	31	32	33	21
~~11~~	~~1.00000~~	~~23~~	~~35~~	~~47~~	~~36~~	~~25~~	~~24~~
12	1.00000	23	34	45	46	47	35
13	1.00000	25	37	49	38	27	26
~~14~~	~~1.00000~~	~~25~~	~~36~~	~~47~~	~~48~~	~~49~~	~~37~~
15	1.00000	27	39	51	40	29	28
16	1.00000	27	38	49	50	51	39
~~17~~	~~1.00000~~	~~29~~	~~41~~	~~53~~	~~42~~	~~31~~	~~30~~
18	1.00000	29	40	51	52	53	41
19	1.00000	31	43	55	44	33	32
~~20~~	~~1.00000~~	~~31~~	~~42~~	~~53~~	~~54~~	~~55~~	~~43~~
21	1.00000	45	57	69	58	47	46
22	1.00000	45	56	67	68	69	57
~~23~~	~~1.00000~~	~~47~~	~~59~~	~~71~~	~~60~~	~~49~~	~~48~~
24	1.00000	47	58	69	70	71	59
25	1.00000	49	61	73	62	51	50
~~26~~	~~1.00000~~	~~49~~	~~60~~	~~71~~	~~72~~	~~73~~	~~61~~
27	1.00000	51	63	75	64	53	52
28	1.00000	51	62	73	74	75	63
~~29~~	~~1.00000~~	~~53~~	~~65~~	~~77~~	~~66~~	~~55~~	~~54~~
30	1.00000	53	64	75	76	77	65
31	1.00000	67	79	91	80	69	68
~~32~~	~~1.00000~~	~~67~~	~~78~~	~~89~~	~~90~~	~~91~~	~~79~~
33	1.00000	69	81	93	82	71	70
34	1.00000	69	80	91	92	93	81
~~35~~	~~1.00000~~	~~71~~	~~83~~	~~95~~	~~84~~	~~73~~	~~72~~
36	1.00000	71	82	93	94	95	83
37	1.00000	73	85	97	86	75	74
~~38~~	~~1.00000~~	~~73~~	~~84~~	~~95~~	~~96~~	~~97~~	~~85~~
39	1.00000	75	87	99	88	77	76
40	1.00000	75	86	97	98	99	87
~~41~~	~~1.00000~~	~~89~~	~~101~~	~~113~~	~~102~~	~~91~~	~~90~~
42	1.00000	89	100	111	112	113	101
43	1.00000	91	103	115	104	93	92
~~44~~	~~1.00000~~	~~91~~	~~102~~	~~113~~	~~114~~	~~115~~	~~103~~
45	1.00000	93	105	117	106	95	94
46	1.00000	93	104	115	116	117	105
~~47~~	~~1.00000~~	~~95~~	~~107~~	~~119~~	~~108~~	~~97~~	~~96~~
48	1.00000	95	106	117	118	119	107
49	1.00000	97	109	121	110	99	98
~~50~~	~~1.00000~~	~~97~~	~~108~~	~~119~~	~~120~~	~~121~~	~~109~~
51	1.00000	111	123	135	124	113	112
52	1.00000	111	122	133	134	135	123
~~53~~	~~1.00000~~	~~113~~	~~125~~	~~137~~	~~126~~	~~115~~	~~114~~
54	1.00000	113	124	135	136	137	125
55	1.00000	115	127	139	128	117	116
~~56~~	~~1.00000~~	~~115~~	~~126~~	~~137~~	~~138~~	~~139~~	~~127~~
57	1.00000	117	129	141	130	119	118

58	1.00000	117	128	139	140	141	129
59	1.00000	119	131	143	132	121	120
60	1.00000	119	130	141	142	143	131

BAND WIDTH= 50

KNOWN NON-ZERO LOADS

COMPONENT NUMBER LOAD

KNOWN DISPLACEMENTS

COMPONENT NUMBER	DISPLAC EMENT
266	0.0000D 00
268	0.0000D 00
270	0.0000D 00
272	0.0000D 00
274	0.0000D 00
276	0.0000D 00
278	0.0000D 00
280	0.0000D 00
282	0.0000D 00
284	0.0000D 00
286	0.0000D 00

NODE NO.	ALPHA(DEG)	KNOWN U	TANG. FORCE
1	0.0	0.0000D 00	0.000D 00
2	0.0	0.0000D 00	0.0000D 00
3	0.0	0.0000D 00	0.000D 00
4	0.0	0.0000D 00	0.000D 00
5	0.0	0.0000D 00	0.0000D 00
6	0.0	0.0000D 00	0.000D 00
7	0.0	0.0000D 00	0.000D 00
8	0.0	0.0000D 00	0.0000D 00
9	0.0	0.0000D 00	0.000D 00
10	0.0	0.0000D 00	0.000D 00
11	0.0	0.0000D 00	-0.5000D 00

NODE NO.	FORCE AND DISPLACEMENT COMPONENTS			
1	-0.5623D-01	0.1057D-13	0.1942D-20	-0.1461D-06
2	-0.1146D 00	-0.2278D-14	0.1611D-20	-0.1470D-06
3	-0.7961D-01	0.9298D-14	0.1723D-20	-0.1478D-06
4	-0.1441D 00	0.1486D-14	0.2851D-20	-0.1487D-06
5	-0.7139D-01	0.3220D-14	0.1883D-20	-0.1496D-06
6	-0.9615D-01	0.5107D-14	0.2017D-20	-0.1510D-06
7	-0.3123D-01	0.7147D-14	0.8415D-21	-0.1527D-06
8	0.1881D-02	0.4741D-14	-0.3962D-22	-0.1568D-06
9	0.2993D-01	0.7883D-14	-0.8041D-21	-0.1625D-06
10	0.2127D 00	0.4870D-14	-0.4473D-20	-0.1761D-06
11	0.3488D 00	-0.5000D 00	-0.2231D-19	-0.1940D-06
12	0.3676D-14	0.4099D-13	0.1379D-07	-0.1431D-06
13	-0.1554D-14	-0.3829D-13	0.1248D-07	-0.1436D-06
14	0.1543D-15	-0.6624D-13	0.1065D-07	-0.1442D-06
15	-0.2442D-14	0.1479D-13	0.8892D-08	-0.1445D-06
16	-0.5573D-15	0.7566D-14	0.6672D-08	-0.1454D-06
17	0.3331D-14	0.1639D-13	0.5097D-08	-0.1453D-06
18	0.2072D-14	0.5099D-14	0.3134D-08	-0.1467D-06
19	0.4663D-14	0.1359D-13	0.1234D-08	-0.1468D-06
20	0.7176D-14	0.1416D-13	-0.1282D-08	-0.1490D-06
21	0.5329D-14	0.1582D-13	-0.6596D-08	-0.1476D-06
22	-0.4394D-14	-0.8511D-14	-0.1652D-07	-0.1478D-06
23	0.8007D-14	0.1824D-13	0.2630D-07	-0.1359D-06

ELEM.NO.	SXX	SYY	SXY	THETA	PS1	PS2
1	0.247D 00	0.907D-02	-0.913D-02	-2.2	0.247D 00	0.872D-02
2	0.244D 00	0.886D-02	-0.310D-01	-7.4	0.248D 00	0.484D-02
3	0.152D 00	0.497D-02	0.121D-01	4.7	0.153D 00	0.398D-02
4	0.172D 00	0.350D-02	0.661D-02	2.2	0.172D 00	0.324D-02
5	0.619D-01	-0.249D-01	0.278D-01	16.3	0.700D-01	-0.330D-01
6	0.879D-01	-0.155D-01	0.375D-01	18.0	0.100D 00	-0.277D-01
7	-0.245D-01	-0.783D-01	0.479D-01	30.3	0.346D-02	-0.106D 00
8	0.425D-02	-0.436D-01	0.573D-01	33.7	0.424D-01	-0.818D-01
9	-0.166D 00	-0.147D 00	0.911D-01	-42.0	-0.248D 00	-0.645D-01
10	-0.986D-01	-0.544D-01	0.657D-01	-35.7	-0.146D 00	-0.718D-02
11	0.139D 00	0.331D-02	-0.212D-01	-8.7	0.143D 00	0.706D-04
12	0.963D-01	0.845D-02	-0.290D-01	-16.7	0.105D 00	-0.277D-03
13	0.109D 00	-0.182D-01	0.252D-01	10.8	0.114D 00	-0.230D-01
14	0.936D-01	-0.198D-01	0.966D-02	4.8	0.944D-01	-0.207D-01
15	0.583D-01	-0.356D-01	0.579D-01	25.5	0.859D-01	-0.631D-01
16	0.701D-01	-0.413D-01	0.451D-01	19.5	0.861D-01	-0.573D-01
17	-0.861D-02	-0.370D-01	0.677D-01	39.1	0.464D-01	-0.920D-01
18	0.236D-01	-0.439D-01	0.600D-01	30.3	0.587D-01	-0.790D-01
19	-0.749D-01	-0.997D-02	0.412D-01	-25.9	-0.949D-01	0.999D-02
20	-0.339D-01	-0.252D-01	0.487D-01	-42.4	-0.784D-01	0.194D-01
21	0.215D-01	-0.234D-01	0.930D-02	11.3	0.234D-01	-0.252D-01
22	-0.144D-01	-0.300D-01	0.222D-01	35.3	0.136D-02	-0.458D-01
23	0.457D-01	-0.520D-01	0.248D-01	13.4	0.516D-01	-0.579D-01
24	0.179D-01	-0.570D-01	0.236D-01	16.1	0.248D-01	-0.638D-01
25	0.470D-01	-0.641D-01	0.372D-01	16.9	0.584D-01	-0.754D-01
26	0.393D-01	-0.729D-01	0.269D-01	12.8	0.454D-01	-0.790D-01
27	0.227D-01	-0.509D-01	0.376D-01	22.8	0.385D-01	-0.667D-01
28	0.343D-01	-0.637D-01	0.267D-01	14.3	0.411D-01	-0.705D-01
29	-0.138D-01	-0.220D-01	0.243D-01	40.2	0.674D-02	-0.426D-01
30	0.949D-02	-0.332D-01	0.183D-01	20.3	0.163D-01	-0.399D-01
31	-0.328D-01	-0.867D-01	0.507D-01	31.0	-0.235D-02	-0.117D 00
32	-0.457D-01	-0.111D 00	0.671D-01	32.0	-0.383D-02	-0.153D 00
33	-0.728D-04	-0.924D-01	0.290D-01	16.1	0.827D-02	-0.101D 00
34	-0.190D-01	-0.110D 00	0.390D-01	20.4	-0.459D-02	-0.124D 00
35	0.221D-01	-0.831D-01	0.127D-01	6.8	0.236D-01	-0.846D-01
36	0.690D-02	-0.952D-01	0.127D-01	7.0	0.846D-02	-0.968D-01
37	0.249D-01	-0.530D-01	0.281D-02	2.1	0.250D-01	-0.531D-01
38	0.191D-01	-0.631D-01	-0.287D-02	-2.0	0.192D-01	-0.632D-01
39	0.132D-01	-0.557D-02	-0.567D-02	-15.6	0.148D-01	-0.714D-02
40	0.172D-01	-0.142D-01	-0.115D-01	-18.1	0.210D-01	-0.179D-01
41	-0.331D-01	-0.169D 00	0.632D-01	21.5	-0.823D-02	-0.194D 00
42	-0.289D-01	-0.203D 00	0.674D-01	18.9	-0.587D-02	-0.226D 00
43	-0.200D-01	-0.134D 00	0.281D-01	13.1	-0.135D-01	-0.141D 00
44	-0.251D-01	-0.163D 00	0.359D-01	13.7	-0.164D-01	-0.172D 00
45	-0.565D-02	-0.888D-01	-0.254D-03	-0.2	-0.565D-02	-0.888D-01
46	-0.167D-01	-0.109D 00	0.532D-02	3.3	-0.164D-01	-0.109D 00
47	0.503D-02	-0.343D-01	-0.153D-01	-19.0	0.103D-01	-0.395D-01
48	-0.616D-02	-0.483D-01	-0.128D-01	-15.7	-0.256D-02	-0.519D-01
49	0.106D-01	0.300D-01	-0.219D-01	33.1	-0.367D-02	0.442D-01
50	0.340D-02	0.181D-01	-0.205D-01	35.1	-0.110D-01	0.325D-01
51	-0.160D-01	-0.231D 00	0.343D-01	8.8	-0.107D-01	-0.237D 00
52	-0.833D-02	-0.258D 00	0.214D-01	4.9	-0.650D-02	-0.260D 00
53	-0.245D-01	-0.164D 00	0.147D-01	5.9	-0.230D-01	-0.166D 00
54	-0.211D-01	-0.195D 00	0.113D-01	3.7	-0.204D-01	-0.196D 00
55	-0.243D-01	-0.889D-01	-0.167D-02	-1.5	-0.242D-01	-0.889D-01
56	-0.274D-01	-0.116D 00	0.149D-02	1.0	-0.274D-01	-0.116D 00
57	-0.150D-01	-0.151D-01	-0.105D-01	-45.0	-0.455D-02	-0.255D-01
58	-0.221D-01	-0.373D-01	-0.440D-02	-15.1	-0.209D-01	-0.385D-01
59	-0.150D-02	0.602D-01	-0.138D-01	12.1	-0.446D-02	0.632D-01

24	-0.3200D-14	-0.1058D-12	0.2333D-07	-0.1358D-06
25	-0.2526D-14	-0.2911D-14	0.2078D-07	-0.1356D-06
26	-0.2559D-14	0.1756D-13	0.1678D-07	-0.1352D-06
27	-0.1027D-14	-0.7965D-14	0.1360D-07	-0.1347D-06
28	-0.2885D-14	0.9608D-14	0.9801D-08	-0.1338D-06
29	-0.3081D-14	0.2716D-14	0.6516D-08	-0.1329D-06
30	-0.1939D-14	0.1290D-13	0.2034D-08	-0.1312D-06
31	-0.1388D-16	-0.4940D-14	-0.3221D-08	-0.1289D-06
32	-0.7103D-15	0.1331D-13	-0.1290D-07	-0.1241D-06
33	-0.4151D-14	-0.1695D-13	-0.2851D-07	-0.1171D-06
34	0.5582D-14	0.2024D-13	0.3937D-07	-0.1245D-06
35	0.3997D-14	-0.6466D-13	0.3608D-07	-0.1230D-06
36	-0.2492D-14	-0.5941D-13	0.3249D-07	-0.1225D-06
37	-0.6661D-14	0.1980D-13	0.2773D-07	-0.1198D-06
38	-0.2814D-14	0.6178D-14	0.2275D-07	-0.1190D-06
39	-0.1443D-14	0.1456D-13	0.1800D-07	-0.1151D-06
40	-0.1942D-14	0.1385D-15	0.1260D-07	-0.1139D-06
41	-0.7355D-15	0.1296D-13	0.6781D-08	-0.1083D-06
42	-0.8777D-15	0.1303D-13	-0.8285D-09	-0.1057D-06
43	-0.9021D-15	0.3873D-14	-0.1161D-07	-0.9658D-07
44	-0.8657D-14	-0.4483D-14	-0.2599D-07	-0.9101D-07
45	0.3414D-14	0.2359D-14	0.4977D-07	-0.1108D-06
46	0.3527D-14	-0.7882D-13	0.4604D-07	-0.1088D-06
47	0.1351D-14	-0.5967D-14	0.4271D-07	-0.1069D-06
48	0.8561D-15	0.1086D-13	0.3705D-07	-0.1037D-06
49	-0.1732D-14	-0.4260D-14	0.3225D-07	-0.1008D-06
50	-0.7349D-14	0.5226D-14	0.2610D-07	-0.9673D-07
51	-0.2631D-14	0.6948D-14	0.2047D-07	-0.9273D-07
52	-0.2606D-14	-0.4630D-15	0.1304D-07	-0.8720D-07
53	-0.3200D-14	0.4411D-14	0.4817D-08	-0.8104D-07
54	-0.1627D-14	0.6494D-14	-0.6995D-08	-0.7267D-07
55	-0.2849D-14	-0.6268D-14	-0.2099D-07	-0.6329D-07
56	-0.6072D-14	0.5366D-14	0.6126D-07	-0.9643D-07
57	0.8216D-14	-0.2978D-13	0.5818D-07	-0.9323D-07
58	0.1397D-14	-0.3715D-14	0.5470D-07	-0.9126D-07
59	0.6939D-15	0.8701D-14	0.4990D-07	-0.8589D-07
60	-0.1220D-14	-0.7764D-14	0.4444D-07	-0.8301D-07
61	-0.5135D-15	0.4011D-14	0.3917D-07	-0.7633D-07
62	-0.2639D-14	-0.3054D-14	0.3249D-07	-0.7279D-07
63	-0.1152D-14	0.3594D-14	0.2612D-07	-0.6436D-07
64	-0.8872D-15	-0.3135D-14	0.1702D-07	-0.5911D-07
65	-0.6566D-15	0.2310D-14	0.7127D-08	-0.4749D-07
66	-0.3031D-14	0.1681D-14	-0.7040D-08	-0.3885D-07
67	-0.5931D-14	-0.5551D-15	0.7017D-07	-0.8083D-07
68	0.1131D-13	-0.3209D-13	0.6743D-07	-0.7765D-07
69	0.3617D-15	-0.7067D-14	0.6489D-07	-0.7476D-07
70	0.7415D-16	0.3152D-14	0.6031D-07	-0.6964D-07
71	-0.5661D-14	0.5085D-14	0.5622D-07	-0.6511D-07
72	0.1578D-14	-0.1303D-14	0.5065D-07	-0.5905D-07
73	-0.2143D-14	0.2907D-14	0.4541D-07	-0.5336D-07
74	-0.7491D-15	0.3906D-15	0.3846D-07	-0.4590D-07
75	-0.1749D-14	0.5012D-14	0.3111D-07	-0.3810D-07
76	-0.1625D-14	0.1173D-14	0.2085D-07	-0.2750D-07
77	0.1071D-14	-0.1356D-14	0.9053D-08	-0.1562D-07
78	-0.3283D-13	-0.6253D-15	0.7984D-07	-0.6656D-07
79	-0.4524D-14	-0.2928D-14	0.7816D-07	-0.6281D-07
80	0.1852D-13	0.8619D-14	0.7597D-07	-0.6025D-07
81	0.2484D-14	-0.2193D-14	0.7316D-07	-0.5403D-07
82	-0.3729D-14	-0.3970D-15	0.6927D-07	-0.5014D-07
83	-0.2442D-14	0.2956D-14	0.6585D-07	-0.4272D-07

84	-0.4308D-14	0.2029D-14	0.6053D-07	-0.3797D-07
85	-0.1799D-14	0.2526D-14	0.5619D-07	-0.2914D-07
86	-0.2820D-14	-0.7199D-15	0.4879D-07	-0.2256D-07
87	-0.2693D-14	-0.1388D-15	0.4222D-07	-0.1076D-07
88	-0.5128D-15	0.9301D-16	0.3106D-07	-0.2524D-09
89	-0.1832D-13	-0.1011D-13	0.8724D-07	-0.5150D-07
90	0.4440D-13	0.5566D-14	0.8614D-07	-0.4837D-07
91	0.4304D-14	0.1388D-15	0.8505D-07	-0.4551D-07
92	0.6811D-14	0.7921D-14	0.8289D-07	-0.4042D-07
93	-0.4715D-14	0.3960D-14	0.8078D-07	-0.3593D-07
94	-0.5260D-14	0.1709D-14	0.7763D-07	-0.2993D-07
95	-0.3486D-14	0.3894D-14	0.7448D-07	-0.2437D-07
96	-0.9187D-14	-0.7832D-15	0.7013D-07	-0.1716D-07
97	-0.3212D-14	0.3137D-14	0.6548D-07	-0.9754D-08
98	-0.7526D-14	-0.2135D-14	0.5901D-07	0.2605D-09
99	0.4718D-14	-0.6713D-15	0.5166D-07	0.1141D-07
100	-0.3818D-13	-0.3611D-14	0.9415D-07	-0.3836D-07
101	0.5061D-13	0.3747D-15	0.9400D-07	-0.3531D-07
102	0.3807D-14	0.4917D-15	0.9338D-07	-0.3329D-07
103	-0.1554D-13	-0.2359D-14	0.9291D-07	-0.2831D-07
104	0.6342D-14	0.5190D-14	0.9133D-07	-0.2514D-07
105	-0.6647D-14	0.8327D-16	0.9032D-07	-0.1934D-07
106	-0.3982D-14	-0.1554D-14	0.8755D-07	-0.1534D-07
107	-0.5109D-15	-0.3608D-15	0.8582D-07	-0.8697D-08
108	-0.1613D-14	-0.2738D-14	0.8160D-07	-0.3095D-08
109	-0.3678D-14	-0.3900D-14	0.7871D-07	0.5500D-08
110	0.4133D-14	0.3521D-15	0.7236D-07	0.1439D-07
111	-0.1627D-13	0.1679D-14	0.9898D-07	-0.2475D-07
112	0.6741D-13	-0.1244D-14	0.9929D-07	-0.2283D-07
113	-0.8743D-15	-0.4982D-14	0.9948D-07	-0.2108D-07
114	-0.2910D-14	-0.1510D-14	0.9952D-07	-0.1794D-07
115	-0.1187D-14	-0.2123D-14	0.9929D-07	-0.1517D-07
116	-0.4703D-14	-0.2133D-14	0.9855D-07	-0.1146D-07
117	-0.1007D-13	-0.2373D-14	0.9753D-07	-0.8037D-08
118	-0.7903D-14	-0.5170D-14	0.9585D-07	-0.3596D-08
119	-0.1356D-13	-0.3816D-14	0.9391D-07	0.9251D-09
120	-0.2667D-14	-0.3593D-15	0.9113D-07	0.7030D-08
121	0.3608D-14	-0.4066D-14	0.8795D-07	0.1376D-07
122	-0.4036D-13	0.4168D-14	0.1020D-06	-0.1243D-07
123	0.4314D-13	-0.5551D-14	0.1028D-06	-0.1107D-07
124	0.6439D-13	-0.1768D-14	0.1033D-06	-0.1052D-07
125	-0.3275D-14	-0.2887D-14	0.1042D-06	-0.8355D-08
126	-0.5941D-14	0.1593D-14	0.1043D-06	-0.7427D-08
127	-0.1489D-13	-0.5551D-14	0.1047D-06	-0.5035D-08
128	0.4008D-14	0.3092D-14	0.1040D-06	-0.3684D-08
129	-0.1040D-13	-0.4441D-14	0.1036D-06	-0.1210D-08
130	-0.1177D-14	0.9373D-15	0.1022D-06	0.9866D-09
131	-0.1249D-13	-0.4219D-14	0.1011D-06	0.3822D-08
132	0.1040D-13	-0.3256D-14	0.9872D-07	0.7638D-08
133	0.3303D-14	0.4731D-01	0.1032D-06	-0.1475D-20
134	-0.6502D-14	0.1100D 00	0.1040D-06	-0.1378D-20
135	0.7300D-14	0.7707D-01	0.1047D-06	-0.1478D-20
136	-0.3212D-14	0.1580D 00	0.1056D-06	-0.2614D-20
137	-0.6939D-16	0.8200D-01	0.1061D-06	-0.1879D-20
138	0.8979D-15	0.1288D 00	0.1064D-06	-0.2178D-20
139	0.2776D-14	0.4982D-01	0.1063D-06	-0.1164D-20
140	0.1321D-14	0.3298D-01	0.1057D-06	-0.5492D-21
141	-0.6523D-15	-0.1478D-01	0.1049D-06	0.3447D-21
142	-0.4278D-14	-0.1299D 00	0.1036D-06	0.2143D-20
143	0.2678D-14	-0.4130D-01	0.1021D-06	0.2296D-20

```
  60   -0.915D-02    0.393D-01    -0.670D-02      7.7    -0.101D-01    0.402D-01
STATEMENTS EXECUTED= 1538321
CORE USAGE        OBJECT CODE=   32224 BYTES,ARRAY AREA=  170136 BYTES,TOTAL AREA
DIAGNOSTICS          NUMBER OF ERRORS=       0, NUMBER OF WARNINGS=       0, NUMBER
COMPILE TIME=     0.96 SEC,EXECUTION TIME=    34.54 SEC,        4.03.13     THURSDAY
       C$STOP
```


Figure (7-17) Output for 60 LST Elements withAAMG

****

For the purposes of comparison the circumferential stress is plotted in Figure (7-18) from the LST/AMG output and from the classical functional solution. Reasonably accurate approximations to the exact solution are obtained.

It is observed from the computed coordinate points that more accuracy could be obtained by a finer mesh of elements close to the inside radius as well as a finer mesh of elements adjacent to the outer radius which is the area of steepest stress gradient.

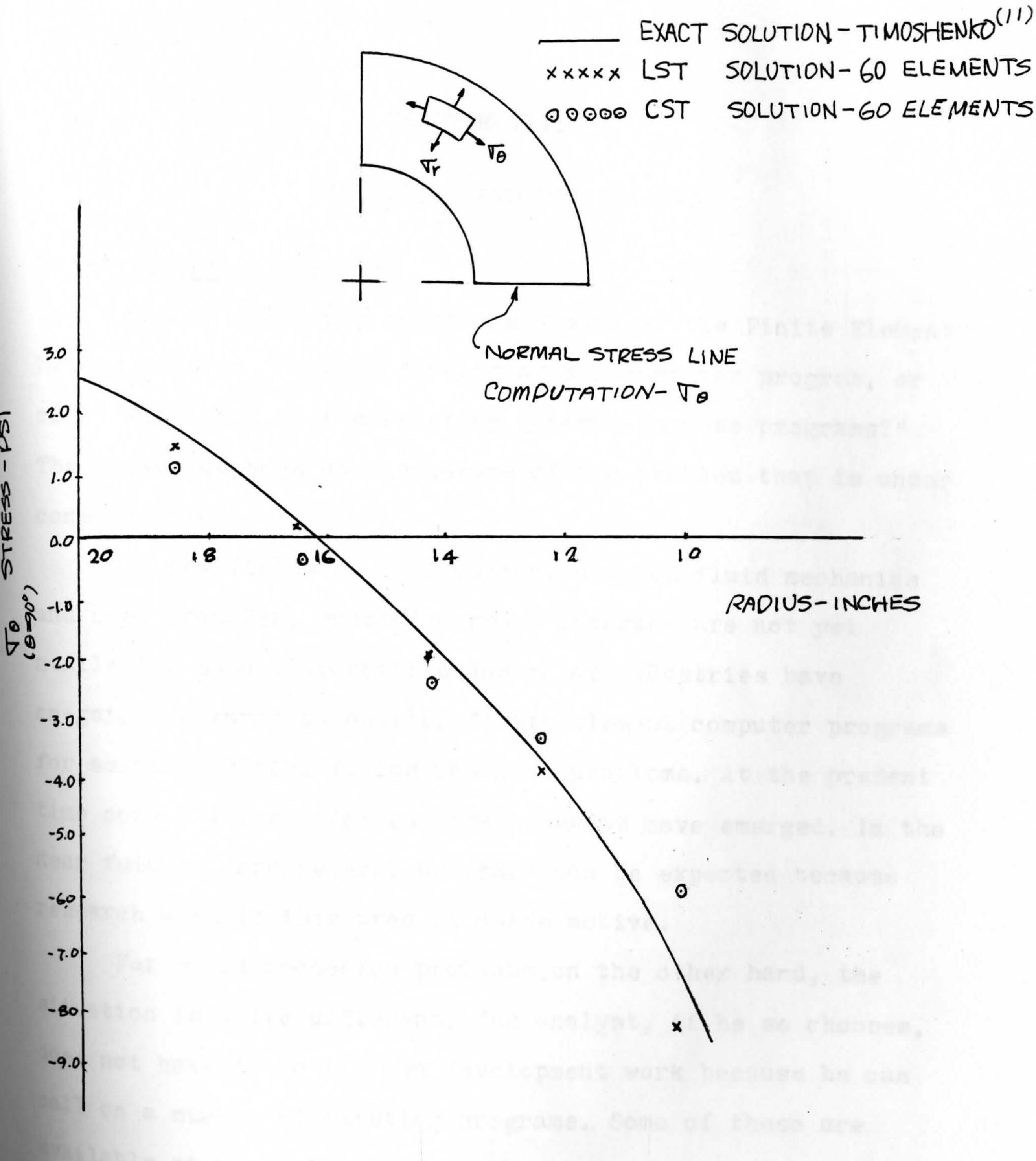


Figure (7-18) Stresses in a Disc with Concentrated Loads

CHAPTER VIII

DISCUSSION AND CONCLUSIONS

8-1-1 General Discussion

One of the first questions a user of the Finite Element Method asks is " must I develop my own computer program, or can I adapt one of the existing general-purpose programs?" The answer depends on the nature of the problem that is under consideration.

In new fields of application, such as fluid mechanics and heat transfer, general purpose programs are not yet available. Some universities and major industries have on-going research to develop finite element computer programs for solving general fluids and heat problems. At the present time only a few special-purpose programs have emerged. In the near future, more general programs can be expected because research work in this area is quite active.

For solid mechanics problems,on the other hand, the situation is quite different. The analyst, if he so chooses, does not have to do his own development work because he can call on a number of existing programs. Some of these are available at nominal cost from the government-sponsored COSMIC library at the University of Georgia. The extensive documentation on these programs makes it relatively easy to learn how to use them. However, it should be borne in mind that to use a versatile general-purpose program to solve a specialized problem is often far more costly (in computer time) than to write a program expressly for solving the specialized problem. Therefore, if a particular type of problem is to be solved repeatedly, the analyst should consider writing his own program for the job. The sample computer programs presented in Chapter II through VII illustrate how the real problems are actually solved by the Finite Element Method. Once the reader has gained an understanding of the Finite Element Method fundamentals presented in the preceding chapters and has learned how these fundamentals are implemented on a digital computer, one has at one's command a powerful engineering analysis tool.

The most important numerical computation performed in a computer code is the solution of the resulting simultaneous equations for displacements and forces. In this thesis a Gaussion elimination technique is adopted. Though the subject of matrix equation solvers is beyond the scope of this thesis, the author suggests the references in Chapter II where pertinent information can be found on this subject.

When the finite element mesh contains hundreds or even thousands of elements and nodes, a cumbersome aspect of obtaining a finite element solution to a problem is the preparation of the input data. Because of this situation, many serious users of the Finite Element Method have developed or adopted automatic mesh generation techniques. All of these are labor-saving devices, but none of the existing user's programs is completely general. In Chapter VII the triangular element and bar member element are used as examples for automatic mesh generation programs. The reader can extend his/her knowledge of mesh generation methods from the given algorithms and include the mesh generation process with any element type. Samples of the different kinds of approaches to development of automatic mesh generation schemes can be found in Buell and Bush[6]. This text provides a survey of many of the useful mesh generation schemes, and is a good source of additional reference information on the subject.

8-1-2 Limits Imposed by Dimensions of Arrays

All computer programs in this thesis begin with a series of DIMENSION statements, which reserve computer memory arrays needed in the calculation process. For example, in program PT10B, the DIMENSION statement for [S] is (20,120) which means the storage has been allocated for problems with up to 60 nodes (2 DOF/node). Also, the bandwidth cannot exceed 20. Larger problems can be considered simply by increasing the dimensions of these matrices. The limit of problem size is dictated by the core storage of available machine. Before proceeding to input data describing the geometrical and loading conditions, all the matrix arrays are initialized by setting all the terms in the arrays to zero. This removes extraneous numbers which could be troublesome later. In addition to the DIMENSION statements, only one other change is required: the LBAND = statement must be altered so that the value of LBAND matches the first value in the S arrays. The unmatched values will cause termination in calculation by the computer.

8-1-3 Reliability of Input Data and Interpretation of Output Results

When solving problems with large numbers of nodes and elements, there is a high probability that a data input error will be made using either the mesh generation scheme or manual generation. The program prints all input data information, but an error may still escape notice. Often the presence of an error is obvious from printed stresses; but, if the problem is one in which a good estimate of the solution is not known in advance, the error caused by faulty data may stay undetected. The best practical method for ensuring a reliable data deck is using computer graphics facilities to have the mesh displayed graphically. Then, a quick visual check is all that is needed to detect unwanted errors. Where this equipment is not available, several useful data checking subroutine programs are provided by Hinton and Owen(12).

When very large numbers of elements are used, some important stress values may escape being noticed in the volume of printed values. It may then be worthwhile to alter the program so that the stresses are card punched as well as printed. The punched output along with the input data cards can then be processed by another program to plot the stresses directly to scale on the figure. The best representation of stress is a plot of stress contours on the diagram of the elastic body. However, the programming required for this is quite lengthy.

8-1-4 Round-Off Error and Double Precision Statement

When solving for unknown displacement components by the method of Payne and Irons and Gaussian elimination scheme, we are in effect inverting a stiffness matrix. Inverting a singular matrix is impossible and if the conditions that would produce a singular matrix exists in a problem, a division by zero will be encountered when attempting to carry out the elimination process. In stress problems, unless an error has been made, the matrix will never be singular; it can, however, approach this state, and the round-off errors will drastically reduce the number of significant figures in the output.

As was previously recommended that each problem be solved several times by increasing the number of elements with each repetition. With each step the numerical solution approaches more closely the theoretically correct solution. However, with each step the matrix comes closer to being singular and so round-off errors become more important. In extreme cases the round-off errors can completely dominate so that there are no significant figures in the output. This imposes a limit on the fineness of the mesh subdivision that can be used when the body is being discretized into elements.

In most applications, if double precision is used, the effect of the round-off does not seriously influence the numerical values of stress. This topic is discussed by Rosanoff and Ginsberg[13], where a method is presented which determines the amount of round-off error. Unless a study is made to determine the effect of round-off, it is recommended that double precision be used.

8-2 Conclusions

For the problem of pin-joint bar element with axial loads, the finite element solution yields the exact classical solution, because the assumed displacement function (used to derive the stiffness matrix of the element) and actual displacement are the identical linear functions.

Comparing the results of problem solutions obtained by using CST and LST, it can be seen the LST elements normally produce more accurate and realistic solutions. The assumed displacement function of LST element yields a condition of linearly varying strain and stress within the element. Hence, the element is more adaptable to problems involving stress gradient.

In Chapter V, results of computer program for a composite structure of bar elements and LST elements yield solutions which are within three to four percent accuracy with those of the classical solution.

The primary advantage obtained by using the 18 degrees of freedom plate element is that it permits the application of boundary conditions to be satisfied more accurately. Care must still be taken that the elements at the plate corners be as small as possible.

The automatic mesh generation process for data input has been shown to reduce significantly the number of lines of data input by at least an order of magnitude for relatively small problems. It has the potential for reduction of data input of the order of magnitude of two, three, and higher for larger problems. The prime intent of this process is to minimize the human error in data input preparation.

BIBLIOGRAPHY

1. Hrennikoff, A., " Solutions in Problems of Elasticity by the Framework Method", Journal Applied Mechnics 8(4) (December, 1969)..
2. Williams,D., " A General Method (depending on the aid of a digital computer) for Deriving the Structural Influence Coefficients of Aeroplane Wings,"Report No. Structures 168, Royal Aircraft Establishment, U.K., November 1954.
3. Turner,M.J.;R.W. Clough;H.C.Martin; and L.J.Topp, " Stiffness and Deflection Analysis of Complex Structures," Journal of the Aeronautical Sciences 23(9) (September 1956) 805-823.
4. Bowes,W.H.; Russell L.T.," Stress Analysis by Finite Element Method for Practicing Engineers,"Lexington Books, Lexington, MA. 1975.
5. Cheung,Y.K.; Yeo,M.F.," A Practical Introduction to Finite Element Analysis," Pitman Publishing Limited, London, 1979.
6. Buell,W.R.; Bush,B.A.," Mesh Generation - a Survey," Journal of Engineering for Industry, Serious B,vol.95,no.1, Feb 1973,pp.332-338.
7. Payne and Irons, referred to as private communication in the Finite Element Method by Zienkiewicz.

```
$JOB
C       *****    MAIN PT10B     *****           SEP 20 1979
C         PIN-JOINTED TRUSS        J P PAN
        DIMENSION S(20,120),U(120),F(120),IFX(120),XY(2,60)
        DIMENSION ARB(100),NCB(2,100)
        DIMENSION NAME(20)
        DOUBLE PRECISION S,U,F,XY,ARB,YMB
        LBAND=20
      1 WRITE(6,100)
        READ(5,102) NAME
        WRITE(6,103) NAME
        READ(5,106) YMB
        WRITE(6,107) YMB
        CALL RC02B(2,XY,NDF)
        NBAND=0
        CALL CN10B(NCB,ARB,NB,NBAND)
        IF(NBAND-LBAND)5,5,4
      4 WRITE(6,104)
        CALL EXIT
      5 DO 6 I=1,NBAND
        DO 6 J=1,NDF
      6 S(I,J)=0.D0
        REWIND 4
        DO 8 IB=1,NB
      8 CALL ES10B(IB,NCB,XY,ARB,YMB,S,LBAND)
        DO 10 I=1,NBAND
     10 WRITE(4)(S(I,J),J=1,NDF)
        END FILE 4
      2 CALL KF01B(F,NDF)
        CALL KU01B(U,IFX,NDF)
        CALL GE02B(F,U,S,IFX,NBAND,NDF,LBAND)
        REWIND 4
        DO 20 I=1,NBAND
     20 READ(4)(S(I,J),J=1,NDF)
        CALL FU10B(S,U,NBAND,NDF,2,LBAND)
        CALL ST10B(NB,NCB,XY,U,YMB)
        READ(5,108) NEXT
        IF(NEXT-1) 24,1,2
     24 CALL EXIT
    100 FORMAT('1 MAIN PT10B',/,' STRESS IN A PIN-JOINTED TRUSS')
    102 FORMAT(20A4)
    103 FORMAT('0CASE TITLE  ---   ',20A4)
    104 FORMAT('0BAND WIDTH TOO LARGE')
    106 FORMAT(F10.5)
    107 FORMAT('0YOUNGS MODULUS =',E10.3)
    108 FORMAT(I5)
        STOP
        END
C
C
C
C
C
C
C
C
C
C
C
C
```

```
$JOB
C       *****     MAIN SP23B       *****        SEP 20 1979
C       STRESSES IN A PLATE DUE TO TEMPRATURE AND IN-PLANE LOADS
C       TRIANGLES
        DIMENSION E(6,6),B(3,6),D(3,3),AB(3,3),AI(6,6),FTE(6),NAME(20)
        DIMENSION S(30,120)
        DIMENSION F(120),U(120),IFX(120),TRIG(120),XY(2,60)
        DIMENSION NCON(3,100),T(100),DE(100)
        DOUBLE PRECISION E,B,D,AB,AI,S,F,U,TRIG,XY,T,DE
        DOUBLE PRECISION GNU,DEI,CE,ET,VOL,FTE
        DOUBLE PRECISION YM
        LBAND=30
      1 WRITE(6,111)
        WRITE(6,112)
        READ(5,100)NAME
        WRITE(6,101)NAME
        READ(5,110)YM,GNU,DEI,CE
        WRITE(6,104)YM,GNU
        WRITE(6,105)DEI,CE
        CALL DB02B(YM,GNU,D,B)
        CALL RC02B(2,XY,NDF)
        CALL CN02B(LBAND,NCON,T,DE,NE,NBAND)
        DO 5 J=1,NDF
        F(J)=0.D0
        TRIG(J)=0.D0
        DO 5 I=1,NBAND
      5 S(I,J)=0.D0
        REWIND 1
        DO 25 IE=1,NE
        ET=CE*(DE(IE)-DEI)
        CALL AB01B(IE,NCON,XY,T,AB,VOL)
        CALL IN12B(AB,3)
        CALL AI01B(AB,AI)
        WRITE(1) AI
        CALL ES03B(AI,B,D,VOL,ET,E,FTE)
     25 CALL AS03B(IE,NCON,E,FTE,S,F,2,3,6,LBAND,1)
        END FILE 1
        REWIND 1
        REWIND 4
        DO 26 I=1,NBAND
     26 WRITE(4)(S(I,J),J=1,NDF)
        END FILE 4
        REWIND 4
        CALL KE02B(F)
        CALL KU01B(U,IFX,NDF)
        CALL GN02B(S,U,F,IFX,NBAND,LBAND,TRIG)
        CALL GE02B(F,U,S,IFX,NBAND,NDF,LBAND)
        DO 30 I=1,NBAND
     30 READ(4) (S(I,J),J=1,NDF)
        CALL RU01B(U,TRIG,NDF)
        CALL EU10B(S,U,NBAND,NDF,2,LBAND)
        WRITE(6,106)
        DO 45 IE=1,NE
        READ(1) AI
        ET=CE*(DE(IE)-DEI)
     45 CALL ST03B(D,B,AI,ET,U,NCON,IE)
        READ(5,108)NEXT
        IF(NEXT)47,47,1
     47 CALL EXIT
    100 FORMAT(20A4)
```

```
      101 FORMAT('0CASE TITLE--- ',20A4)
      104 FORMAT('0YOUNGS MODULUS=',F10.3,'  POISSONS RATIO=',F6.3)
      105 FORMAT(' INITIAL TEMP=',F6.1,'   COEF.OF EXP.=',F10.3)
      106 FORMAT('0ELEM.NO.     SXX          SYY          SXY      THETA      PS1
     1        PS2')
      108 FORMAT(I5)
      110 FORMAT(4F10.5)
      111 FORMAT('1              MAIN SP23B   SEP  20 1979')
      112 FORMAT(' STRESSES IN A PLATE DUE TO TEMPERATURE AND IN-PLANE LOA
     1',/,' BY FINITE ELEMENT METHODS USING CONSTANT STRAIN TRIANGLES
          STOP
          END
          SUBROUTINE DB02B(YM,GNU,D,B)
          DIMENSION D(3,3),B(3,6)
          DOUBLE PRECISION YM,GNU,D,B
          D(1,1)=YM/(1.D0-GNU*GNU)
          D(1,2)=GNU*D(1,1)
          D(1,3)=0.D0
          D(2,1)=D(1,2)
          D(2,2)=D(1,1)
          D(2,3)=0.D0
          D(3,1)=0.D0
          D(3,2)=0.D0
          D(3,3)=.5D0*(1.D0-GNU)*D(1,1)
          DO 4 I=1,3
          DO 4 J=1,6
        4 B(I,J)=0.D0
          B(1,2)=1.D0
          B(2,6)=1.D0
          B(3,3)=1.D0
          B(3,5)=1.D0
          RETURN
          END
          SUBROUTINE RC02B(NE,XY,NDF)
          DIMENSION XY(2,1)
          DOUBLE PRECISION X,Y,XY
          NDF=0
          WRITE(6,100)
        2 READ(5,102)I,X,Y
          IF(I)20,20,4
        4 XY(1,I)=X
          XY(2,I)=Y
          NDF=NDF+NF
          WRITE(6,104)I,X,Y
          GO TO 2
       20 RETURN
      100 FORMAT('0NODE NO.    X-COORD      Y-COORD')
      102 FORMAT(I5,2F10.5)
      104 FORMAT(I6,2X,2F12.4)
          END
          SUBROUTINE CN02B(LBAND,NCON,T,DE,NE,NBAND)
          DIMENSION NCON(3,100),T(100),DE(100)
          DOUBLE PRECISION T,DE,TEMP,TH
          WRITE(6,100)
          NE=0
          NBAND=0
        2 READ(5,102)I,N1,N2,N3,TH,TEMP
```

```
$JOB
C           *****     MAIN SP33B     *****     SEP 25,1979
C    STRESSES IN A PLATE USING LINEAR STRAIN TRIANGLES    JP PAN
C         GUIDED NODES
          DIMENSION S(42,200),F(200),U(200),IFX(200),XY(2,100)
          DIMENSION TRIG(200)
          DIMENSION NCON(6,50),T(50)
          DIMENSION E(12,12),B(3,12),AI(12,12),X(3),Y(3),NAME(20)
          DIMENSION DB(3,12),D(3,3),BDB(12,12)
          DOUBLE PRECISION S,F,U,XY,TRIG,T,E,B,AI,X,Y
          DOUBLE PRECISION DB,D,BDB,YM,GNU,DBIJ,V
          LBAND=42
        1 WRITE(6,102)
          READ(5,100)NAME
          WRITE(6,101)NAME
          READ(5,103)YM,GNU
          WRITE(6,104)YM,GNU
          CALL DB03B(YM,GNU,D,B,BDB)
          DO 4 I=1,3
          DO 4 J=1,12
          DBIJ=0.
          DO 3 IJ=1,3
        3 DBIJ=DBIJ+D(I,IJ)*B(IJ,J)
        4 DB(I,J)=DBIJ
          CALL RC06B(2,XY,NDF)
          CALL CN06B(LBAND,6,2,NCON,T,NE,NBAND)
          DO 5 J=1,NDF
          TRIG(J)=0.
          DO 5 I=1,NBAND
        5 S(I,J)=0.
          REWIND 1
          DO 25 IE=1,NE
          CALL AI02B(IE,NCON,XY,T,AI,X,Y)
          WRITE(1) AI
          V=T(IE)*.5*(X(1)*(Y(2)-Y(3))+X(2)*(Y(3)-Y(1))+X(3)*(Y(1)-Y(2)))
          CALL ES06B(V,AI,BDB,X,Y,E)
       25 CALL AS06B(IE,NCON,E,S,2,6,12,LBAND)
          END FILE 1
          REWIND 4
          DO 26 I=1,NBAND
       26 WRITE(4)(S(I,J),J=1,NDF)
          END FILE 4
        2 CALL KF01B(F,NDF)
          CALL KU01B(U,IFX,NDF)
          CALL GN02B(S,U,F,IFX,NBAND,LBAND,TRIG)
          CALL GE02B(F,U,S,IFX,NBAND,NDF,LBAND)
          REWIND 4
          DO 30 I=1,NBAND
       30 READ(4)(S(I,J),J=1,NDF)
          CALL RU01B(U,TRIG,NDF)
          CALL EU10B(S,U,NBAND,NDF,2,LBAND)
          REWIND 1
          WRITE(6,106)
          DO 45 IE=1,NE
          READ(1) AI
       45 CALL ST06B(DB,AI,U,NCON,IE)
          READ(5,108)NEXT
          IF(NEXT-1)47,1,2
       47 CALL EXIT
      100 FORMAT(20A4)
```

```
  101 FORMAT('0CASE TITLE  --- ',20A4)
  102 FORMAT('1                MAIN SP33B    SEP 25,1979',/,' STRESSES IN IN-P
     1LANE LOADED PLATE USING LINEAR STRAIN TRIANGLES  (GUIDED NODES)')
  103 FORMAT(2F10.5)
  104 FORMAT('0YOUNGS MODULUS=',F10.3,'   POISSONS RATIO=',F6.3)
  106 FORMAT('0ELEM.NO.     SXX              SYY              SXY        THETA
     1    PS1       PS2')
  108 FORMAT(I5)
      STOP
      END
      SUBROUTINE DB03B (YM,GNU,D,B,DBD)
      DIMENSION D(3,3),B(3,12),DBD(12,12)
      DOUBLE PRECISION YM,GNU,D,B,DBD
      D(1,1)=YM/(1.-GNU*GNU)
      D(1,2)=GNU*D(1,1)
      D(1,3)=0.
      D(2,1)=D(1,2)
      D(2,2)=D(1,1)
      D(2,3)=0.
      D(3,1)=0.
      D(3,2)=0.
      D(3,3)=.5*(1.-GNU)*D(1,1)
      DO 4 I=1,3
      DO 4 J=1,12
    4 B(I,J)=0.
      B(1,2)=1.
      B(2,9)=1.
      B(3,3)=1.
      B(3,8)=1.
      DO 6 I=1,12
      DO 6 J=1,12
    6 DBD(I,J)=0.
      DBD(2,2)=D(1,1)
      DBD(9,2)=D(1,2)
      DBD(2,9)=DBD(9,2)
      DBD(3,3)=D(3,3)
      DBD(8,3)=DBD(3,3)
      DBD(3,8)=DBD(8,3)
      DBD(8,8)=DBD(3,3)
      DBD(9,9)=DBD(2,2)
      RETURN
      END
      SUBROUTINE RC06B(NE,XY,NDE)
      DIMENSION XY(2,1)
      DOUBLE PRECISION XY,X,Y
      WRITE(6,100)
      READ(5,102)NN
      NDE=NE*NN
    2 READ(5,102)I,X,Y
      IF(I)20,20,4
    4 XY(1,I)=X
      XY(2,I)=Y
      WRITE(6,104)I,X,Y
      GO TO 2
   20 WRITE(6,106)NDE
      RETURN
  100 FORMAT('0NODE NO.    X=COORD      Y=COORD')
  102 FORMAT(I5,2F10.5)
```

```
$JOB
C       ***** MAIN SP44B    *****       OCT. 5,1979
C       STRESSES IN A PLATE USING LINEAR STRAIN TRIANGLES    J P PAN
C       (REIANFORCING BARS      GUIDED NODES)
        DIMENSION S(46.260).F(260).U(260).IFX(260).XY(2.130).TRIG(260)
        DIMENSION NCON(6.90).T(90)
        DIMENSION NCB(2.40).ARB(40)
        DIMENSION E(12.12).B(3.12).AI(12.12).X(3).Y(3).NAME(20)
        DIMENSION DB(3.12).D(3.3).BDB(12.12)
        DOUBLE PRECISION S.F.U.XY.TRIG.T.ARB.E.B.AI.X.Y.DB.D
        DOUBLE PRECISION BDB.YM.GNU.DBIJ.YMB.V
        LBAND=46
      1 WRITE(6.102)
        READ(5.100)NAME
        WRITE(6.101) NAME
        READ(5.103)YM.GNU
        WRITE(6.104)YM.GNU
        CALL DB03B(YM.GNU.D.B.BDB)
        DO 8 I=1.3
        DO 8 J=1.12
        DBIJ=0.
        DO 7 IJ=1.3
      7 DBIJ=DBIJ+D(I.IJ)*B(IJ.J)
      8 DB(I.J)=DBIJ
        CALL RC06B(2.XY.NDF)
        CALL CN06B(LBAND.6.2.NCON.T.NE.NBAND)
        READ(5.103)YMB
        WRITE(6.107)YMB
        CALL CN10B(NCB.ARB.NB.NBAND)
        DO 9 J=1.NDF
        TRIG(J)=0.
        DO 9 I=1.NBAND
      9 S(I.J)=0.
        REWIND 1
        DO 15 IE=1.NE
        CALL AI02B(IE.NCON.XY.T.AI.X.Y)
        WRITE(1) AI
        V=T(IE)*.5*(X(1)*(Y(2)-Y(3))+X(2)*(Y(3)-Y(1))+X(3)*(Y(1)-Y(2)))
        CALL ES06B(V.AI.BDB.X.Y.E)
        CALL AS06B(IE.NCON.E.S.2.6.12.LBAND)
     15 CONTINUE
        END FILE 1
        IF(NB)21.21.19
     19 DO 20 IB=1.NB
     20 CALL ES10B(IB.NCB.XY.ARB.YMB.S.LBAND)
     21 REWIND 4
        DO 26 I=1.NBAND
     26 WRITE(4)(S(I.J).J=1.NDF)
        END FILE 4
      2 CALL KF01B(F.NDF)
        CALLKU01B(U.IFX.NDF)
        CALL GN02B(S.U.F.IFX.NBAND.LBAND.TRIG)
        CALL GE02B(F.U.S.IFX.NBAND.NDF.LBAND)
        REWIND 4
        DO 30 I=1.NBAND
     30 READ(4)(S(I.J).J=1.NDF)
        CALL RU01B(U.TRIG.NDF)
        CALL FU10B(S.U.NBAND.NDF.2.LBAND)
        REWIND 1
        WRITE(6.106)
```

```
      DO 45 IE=1,NE
      READ(1) AI
   45 CALL ST06B(DB,AI,U,NCON,IE)
      CALL ST10B(NB,NCB,XY,U,YMB)
      READ(5,108)NEXT
      IF(NEXT-1)47,1,2
   47 CALL EXIT
  100 FORMAT(20A4)
  101 FORMAT('OCASE TITLE  ---',20A4)
  102 FORMAT('1                MAIN SP44B  OCT 5 1979',/,' STRESSES IN IN-P
     1LANE LOADED PLATE ( LST   REINFORCING BARS    GUIDED NODES )')
  103 FORMAT(2F10.5)
  104 FORMAT('OYOUNGS MODULUS=',E10.3,' POISSONS RATIO=',F6.3)
  106 FORMAT('OELEM.NO.     SXX           SYY           SXY          THETA      PSI
     1    PS2')
  107 FORMAT('OYOUNGS MODULUS OF BARS =',E10.3)
  108 FORMAT(I5)
      STOP
      END

      SUBROUTINE DB03B (YM,GNU,D,B,DBD)
      DIMENSION D(3,3),B(3,12),DBD(12,12)
      DOUBLE PRECISION YM,GNU,D,B,DBD
      D(1,1)=YM/(1.-GNU*GNU)
      D(1,2)=GNU*D(1,1)
      D(1,3)=0.
      D(2,1)=D(1,2)
      D(2,2)=D(1,1)
      D(2,3)=0.
      D(3,1)=0.
      D(3,2)=0.
      D(3,3)=.5*(1.-GNU)*D(1,1)
      DO 4 I=1,3
      DO 4 J=1,12
    4 B(I,J)=0.
      B(1,2)=1.
      B(2,9)=1.
      B(3,3)=1.
      B(3,8)=1.
      DO 6 I=1,12
      DO 6 J=1,12
    6 DBD(I,J)=0.
      DBD(2,2)=D(1,1)
      DBD(9,2)=D(1,2)
      DBD(2,9)=DBD(9,2)
      DBD(3,3)=D(3,3)
      DBD(8,3)=DBD(3,3)
      DBD(3,8)=DBD(8,3)
      DBD(8,8)=DBD(3,3)
      DBD(9,9)=DBD(2,2)
      RETURN
      END

      SUBROUTINE RC06B(NF,XY,NDF)
      DIMENSION XY(2,1)
      DOUBLE PRECISION XY,X,Y
      WRITE(6,100)
      READ(5,102)NN
      NDF=NF*NN
    2 READ(5,102)I,X,Y
```

```
$JOB
C     *****      MAIN PB11B      *****NOV. 14 1979
C     PLATE BENDING      TRIANGULAR ELEMENTS WITH 18 DOF
      DIMENSION S(42,150),F(150),U(150),IFX(150)
      DIMENSION XY(2,25),IROT(25),IM(25)
      DIMENSION NCON(3,40),PR(3,40)
      DIMENSION PE(18),D(3,3),A(21,21),P(7,7),G(21,18),XL(3),YL(3)
      DIMENSION X(4),Y(4),CO(3),SI(3),EL(3),NAME(20),E(18,18),CQ(21,3)
      DIMENSION T(5,5),TC(3,3),ND(4),NDL(3)
      DOUBLE PRECISION S,F,U,XY,PR,PE,D,A,P,G,XL,YL,X,Y
      DOUBLE PRECISION CO,SI,EL,E,CQ,T,TC,YM,GNU,TH,C
      DOUBLE PRECISION XC,YC,XD,YD,DABS,THE
      LBAND=42
      DO 2 J=1,3
      XL(J)=0.D0
      YL(J)=0.D0
      NDL(J)=0
      DO 2 I=1,21
    2 CQ(I,J)=0.D0
      DO 3 KI=1,4
      DO 3 LI=1,KI
      I=(KI*(KI+3))/2+3+LI
      CQ(I,1)=(LI*(LI+1))/2
      CQ(I,2)=((KI+1-LI)*(KI+2-LI))/2
    3 CQ(I,3)=LI*(KI+1-LI)
      DO 4 I=1,5
      DO 4 J=1,5
    4 T(I,J)=0.D0
    1 WRITE(6,100)
      READ(5,102) NAME
      WRITE(6,104) NAME
      READ(5,106)YM,GNU,TH
      WRITE(6,108)YM
      WRITE(6,121)GNU
      WRITE(6,122)TH
      C=YM*TH**3/(12.D0*(1.D0-GNU*GNU))
      D(1,1)=C
      D(1,2)=C*GNU
      D(1,3)=0.D0
      D(2,1)=C*GNU
      D(2,2)=C
      D(2,3)=0.D0
      D(3,1)=0.D0
      D(3,2)=0.D0
      D(3,3)=C*(1.D0-GNU)*.5D0
      C=1.D0/(1.D0-GNU*GNU)
      TC(1,1)=C
      TC(1,2)=0.D0
      TC(1,3)=-GNU*C
      TC(2,1)=0.D0
      TC(2,2)=1.D0
      TC(2,3)=0.D0
      TC(3,1)=-GNU*C
      TC(3,2)=0.D0
      TC(3,3)=C
      CALL RC02B(6,XY,NDF)
      NN=NDF/6
      CALL CNO8B(LBAND,NN,[illegible],NE,NCON,PR)
      DO 5 J=1,NDF
      F(J)=0.D0
```

```
      DO 5 I=1.NBAND
    5 S(I.J)=0.D0
      NET=0
      DO 15 IF=1.NF
      XC=0.D0
      YC=0.D0
      DO 6 I=1.3
      N=NCON(I.IF)
      ND(I)=N
      X(I)=XY(1.N)
      Y(I)=XY(2.N)
      XC=XC+X(I)/3.D0
    6 YC=YC+Y(I)/3.D0
      ND(4)=ND(1)
      DO 7 I=1.3
      ND(I)=(ND(I+1)-ND(I))/IABS(ND(I+1)-ND(I))
      X(I)=X(I)-XC
    7 Y(I)=Y(I)-YC
      X(4)=X(1)
      Y(4)=Y(1)
      DO 10 I=1.3
      XD=DABS(X(I)-XL(I))
      IF(XD-1.0D-3)8.8.12
    8 YD=DABS(Y(I)-YL(I))
      IF(YD-1.0D-03)9.9.12
    9 IF(ND(I)-NDL(I))12.10.12
   10 CONTINUE
      GO TO 14
   12 NET=NET+1
      DO 13 I=1.3
      NDL(I)=ND(I)
      XL(I)=X(I)
   13 YL(I)=Y(I)
      CALL AM21B(X.Y.A.CO.SI.EL.ND)
      CALL IN12B(A.21)
      CALL EV02B(A.CO.SI.EL.ND.G)
      CALL TI01B(7.X.Y.P)
      CALL FS18B(CO.D.G.P.F)
   14 CALL PE01B(IF.PR.P.G.PF)
   15 CALL AS03B(IF.NCON.F.PF.S.E.6.3.18.LBAND.2)
      WRITE(6.116)NET
      REWIND 1
      IROT(1)=0
      J=1
      WRITE(6.113)
   22 READ(5.114)N.THE
      IF(N)26.26.24
   24 WRITE(6.115)N.THE
      IROT(J)=N
      J=J+1
      IROT(J)=0
      CALL TM02B(THE.T)
      WRITE(1)T
      NU=6*N-5
      CALL LR01B(S.LBAND.NBAND.NU.T.5)
      CALL LR02B(E.NU.T.5)
      GO TO 22
   26 END FILE 1
      REWIND 1
      IM(1)=0
```

```
      J=1
      WRITE(6.118)
   27 READ(5.112)N
      IF(N)29.29.28
   28 WRITE(6.112)N
      IM(J)=N
      J=J+1
      IM(J)=0
      NU=6*N-3
      CALL LR01B(S.LBAND.NBAND.NU.TC.3)
      CALL LR02B(F.NU.TC.3)
      GO TO 27
   29 CALL KF02B(F)
      CALL KU01B(U.IFX.NDF)
      CALL GE02B(F.U.S.IFX.NBAND.NDF.LBAND)
      DO 31 J=1.NN
      N=IM(J)
      IF(N)35.35.30
   30 NU=6*N-3
   31 CALL RU02B(U.NU.TC.3)
   35 DO 37 J=1.NN
      N=IROT(J)
      IF(N)38.38.36
   36 READ(1) T
      NU=6*N-5
   37 CALL RU02B(U.NU.T.5)
   38 WRITE(6.119)
      DO 40 I=1.NN
      IF=6*I
      IS=IF-5
   40 WRITE(6.120) I.(U(II).II=IS.IF)
      WRITE(6.110)
      CALL ST18B(D.U.NCON.TH.NN)
      READ(5.112)NEXT
      IF(NEXT-1)52.1.1
   52 CALL EXIT
  100 FORMAT('1          MAIN PR11B    NOV 14 1979 './.' PLATE IN BENDING B
     1Y TRIANGULAR ELEMENTS (18 DEG. OF F.)')
  102 FORMAT(20A4)
  104 FORMAT('0CASE TITLE --- '.20A4)
  106 FORMAT(3F10.5)
  108 FORMAT('0YOUNGS MODULUS='.D20.8)
  121 FORMAT('0POISSONS RATIO='.D20.8)
  122 FORMAT('0PLATE THICKNESS='.D20.8)
  110 FORMAT('0NODE NO.     MXX          MYY          MXY          SXX          SYY
     1          SXY          THETA      PS1          PS2')
  112 FORMAT (I5)
  113 FORMAT('0LOCAL ROTATED AXES'./.'NODE NO.    THETA(DEG)')
  114 FORMAT(I5.F10.5)
  115 FORMAT(I5.5X.F8.1)
  116 FORMAT('0NUMBER OF DIFFERENT ELEMENT TYPES ='.I4)
  118 FORMAT('0NODES HAVING MODIFIED CURVATURES')
  119 FORMAT('0NODE NO.      DISPLACEMENTS REFERRED TO GLOBAL  AXES')
  120 FORMAT(I6.3X.6E11.4)
      STOP
      END

      SUBROUTINE AS03B(IE.NCON.F.FF.S.E.NE.NN.KE.LBAND.IT)
      DIMENSION NCON(NN.1).F(KE.KE).FF(1).S(LBAND.1).E(1)
      DOUBLE PRECISION F.FF.S.E
```

```
$JOB
      SUBROUTINE AB01B(IE,NCON,XY,T,AB,VOL)
C   GENERATES A-BAR MATRIX AND VOLUME
      DIMENSION NCON(3,1),XY(2,1),AB(3,3),T(1)
      DOUBLE PRECISION XY,T,AB,VOL,AREA
      DO 4 I=1,3
      NC=NCON(I,IE)
      AB(I,1)=1.D0
      AB(I,2)=XY(1,NC)
    4 AB(I,3)=XY(2,NC)
      AREA=(AB(2,2)-AB(1,2))*(AB(3,3)-AB(1,3))-(AB(3,2)-AB(1,2))*(AB(2,
     1)-AB(1,3))
      AREA=.5D0*AREA
      VOL=AREA*T(IE)
      IF(AREA)6,6,8
    6 VOL=-VOL
    8 RETURN
      END

      SUBROUTINE AI01B(AB,AI)
C   GENERATES A-INVERSE
      DIMENSION AB(3,3),AI(6,6)
      DOUBLE PRECISION AB,AI
      DO 2 I=1,3
      DO 2 J=1,3
      AI(I,J)=AB(I,J)
      AI(I+3,J)=0.D0
      AI(I,J+3)=0.D0
    2 AI(I+3,J+3)=AB(I,J)
      RETURN
      END

      SUBROUTINE AI02B(IE,NCON,XY,T,AI,X,Y)
C   GENERATES A-INVERSE AND LOCAL COORDS OF CORNERS OF LINEAR
C   STRAIN TRIANGLE
      DIMENSION NCON(6,1),XY(2,1),T(1),AI(12,12),X(3),Y(3),AB(6,6)
      DIMENSION XX(6),YY(6)
      DOUBLE PRECISION XY,T,AI,XX,YY,XC,YC,AB,X,Y
      XC=0.
      YC=0.
      DO 4 I=1,3
      NC=NCON(2*I-1,IE)
      X(I)=XY(1,NC)
      Y(I)=XY(2,NC)
      XC=XC+X(I)/3.
    4 YC=YC+Y(I)/3.
      DO 6 I=1,3
      X(I)=X(I)-XC
      Y(I)=Y(I)-YC
      II=2*I-1
      XX(II)=X(I)
    6 YY(II)=Y(I)
      XX(2)=(XX(1)+XX(3))*.5
      XX(4)=(XX(3)+XX(5))*.5
      XX(6)=(XX(5)+XX(1))*.5
      YY(2)=(YY(1)+YY(3))*.5
      YY(4)=(YY(3)+YY(5))*.5
      YY(6)=(YY(5)+YY(1))*.5
      DO 7 I=2,6,2
      NCT=NCON(I,IE)
```

```
      XY(1,NCT)=XX(I)+XC
    7 XY(2,NCT)=YY(I)+YC
      DO 8 I=1,6
      AB(I,1)=1.
      AB(I,2)=XX(I)
      AB(I,3)=YY(I)
      AB(I,4)=XX(I)*XX(I)
      AB(I,5)=XX(I)*YY(I)
    8 AB(I,6)=YY(I)*YY(I)
      CALL IN12B(AB,6)
      DO 10 I=1,6
      DO 10 J=1,6
      AI(I,J)=AB(I,J)
      AI(I+6,J)=0.
      AI(I,J+6)=0.
   10 AI(I+6,J+6)=AB(I,J)
      RETURN
      END
      SUBROUTINE AM21B(XX,YY,A,CO,SI,FL,ND)
C     GENERATES A-MATRIX ETC FOR TRIANGULAR ELEMENT (18 DOF) IN
C     BENDING
      DIMENSION XX(4),YY(4),A(21,21),CO(3),SI(3),FL(3),F(21),ND(4)
      DOUBLE PRECISION XX,YY,A,CO,SI,FL,F,X,Y,XY,C,FF,DX,DY,DSQRT,S
      DO 40 III=1,2
      DO 40 II=1,3
      GO TO (1,2),III
    1 X=.5D0*(XX(II)+XX(II+1))
      Y=.5D0*(YY(II)+YY(II+1))
      IL=3
      GO TO 3
    2 X=XX(II)
      Y=YY(II)
      IL=6
    3 DO 30 I=1,IL
      IA=6*(II-1)+I
      J=0
      DO 30 K=1,6
      DO 30 L=1,K
      J=J+1
      IX=K-L
      IY=L-1
      IF(IX)4,4,6
    4 F(J)=1.D0
      IF(IY)10,10,5
    5 F(J)=Y**IY
      GO TO 10
    6 IF(IY)7,7,9
    7 F(J)=X**IX
      GO TO 10
    9 F(J)=X**IX*Y**IY
   10 GO TO (11,12,13,14,15,16),I
   11 C=1.D0
      IF=J
      GO TO 20
   12 C=IX
      IF=J-K+1
      GO TO 20
   13 C=IY
      IF=J-K
```

```
      GO TO 18
   14 C=IX*(IX-1)
      IF=J-2*K+3
      GO TO 20
   15 C=IX*IY
      IF=J-2*K+2
      GO TO 18
   16 C=IY*(IY-1)
      IF=J-2*K+1
   18 IF(IF) 19,19,20
   19 FF=0.D0
      GO TO 21
   20 FF=F(IF)
   21 A(IA,J)=C*FF
   30 CONTINUE
      GO TO (32,40),III
   32 DX=XX(II+1)-XX(II)
      DY=YY(II+1)-YY(II)
      XY=DSQRT(DX*DX+DY*DY)
      FL(II)=XY
      C=DX/XY
      S=DY/XY
      CO(II)=C
      SI(II)=S
      IL=II+18
      IR=6*II-4
      IRR=6*II-3
      DO 38 J=1,21
   38 A(IL,J)=(-S*A(IR,J)+C*A(IRR,J))*ND(II)
   40 CONTINUE
      RETURN
      END

      SUBROUTINE AS02B(E,S,K,LBAND)
      DIMENSION S(LBAND,1),E(6,6)
      DOUBLE PRECISION S,E
      DO 2 J=1,6
      DO 2 I=J,6
      JJ=3*(K-1)+J
      II=I-J+1
    2 S(II,JJ)=S(II,JJ)+E(I,J)
      RETURN
      END

      SUBROUTINE AS03B(IE,NCON,E,FE,S,F,NF,NN,KE,LBAND,II)
      DIMENSION NCON(NN,1),E(KE,KE),FE(1),S(LBAND,1),F(1)
      DOUBLE PRECISION E,FE,S,F
      GO TO (1,2),II
    1 K1=NN
      K2=1
      K3=NN
      GO TO 3
    2 K1=1
      K2=NF
      K3=NF
    3 DO 8 JA=1,NF
      DO 8 JB=1,NN
      J=K1*JA+K2*JB-K3
      JJ=NF*(NCON(JB,IE)-1)+JA
      F(JJ)=F(JJ)+FE(J)
```

```
      DO 8 IA=1,NF
      DO 8 IB=1,NN
      I=K1*IA+K2*IB-K3
      II=NF*(NCON(IB,IE)-1)+IA-JJ+1
      IF(II)8,8,6
    6 S(II,JJ)=S(II,JJ)+E(I,J)
    8 CONTINUE
      RETURN
      END

      SUBROUTINE AS06B(IE,NCON,E,S,NF,NN,KE,LBAND)
      DIMENSION NCON(NN,1),S(LBAND,1),E(KE,KE)
      DOUBLE PRECISION E,S
      DO 5 IA=1,NF
      DO 5 IB=1,NN
      DO 5 JA=1,NF
      DO 5 JB=1,NN
      JJ=NF*(NCON(JB,IE)-1)+JA
      II=NF*(NCON(IB,IE)-1)+IA-JJ+1
      IF(II)5,5,3
    3 I=NN*(IA-1)+IB
      J=NN*(JA-1)+JB
      S(II,JJ)=S(II,JJ)+E(I,J)
    5 CONTINUE
      RETURN
      END

      SUBROUTINE CN06B(LBAND,NN,NF,NCON,T,NE,NBAND)
      DIMENSION NCON(NN,1),T(1),N(12)
      DOUBLE PRECISION T,TH
      WRITE(6,100)
      NE=0
      NBAND=0
      NN1=NN-1
    2 READ(5,102)I,TH,N
      IF(I)20,20,3
    3 DO 4 J=1,NN
    4 NCON(J,I)=N(J)
      T(I)=1.
      IF(TH)6,6,5
    5 T(I)=TH
    6 WRITE(6,103)I,T(I),(N(J),J=1,NN)
      DO 10 J=1,NN1
      JJ=J+1
      DO 10 I=JJ,NN
      ND=IABS(N(J)-N(I))
      IF(ND-NBAND)10,10,8
    8 NBAND=ND
   10 CONTINUE
      NE=NE+1
      GO TO 2
   20 NBAND=NF*(NBAND+1)
      WRITE(6,105)NBAND
      IF(LBAND-NBAND)22,24,24
   22 WRITE(6,106)
      CALL EXIT
   24 RETURN
  100 FORMAT('0ELEM.NO.    THICKNESS     CONNECTING NODE NUMBERS')
  102 FORMAT(I5,F10.5,12I5)
  103 FORMAT(I6,6X,F6.2,5X,12I4)
```

```
  105 FORMAT('0BAND WIDTH=',I5)
  106 FORMAT('0   ****   BAND WIDTH EXCEEDS LIMIT   ****')
      END

      SUBROUTINE CN05B(LBAND,NCON,NE,NBAND)
      DIMENSION NCON(4,1),X(4),Y(4),K(4)
      DOUBLE PRECISION X,Y
      NBAND=0
      NE=0
      WRITE(6,100)
    6 READ(5,102)N,K
      IF(N)20,20,8
    8 NE=NE+1
      DO 10 I=1,3
      II=I+1
      DO 10 J=II,4
      IF(IABS(K(I)-K(J))-NBAND)10,10,9
    9 NBAND=IABS(K(I)-K(J))
   10 CONTINUE
      DO 11 I=1,4
   11 NCON(I,N)=K(I)
      WRITE(6,104)N,K
      GO TO 6
   20 NBAND=2*(NBAND+1)
      IF(LBAND-NBAND)22,24,24
   22 WRITE(6,107)
      CALL EXIT
   24 WRITE(6,108)NBAND
      RETURN
  100 FORMAT('0ELEM.NO.       CORNER NODE NUMBERS')
  102 FORMAT(5I5)
  104 FORMAT(I6,5X,4I5)
  107 FORMAT('0BAND WIDTH TOO LARGE. EXECUTION TERMINATED')
  108 FORMAT('0BAND WIDTH=',I3)
      END

      SUBROUTINE BC01B(B)
      DIMENSION B(4,6)
      DOUBLE PRECISION B
      DO 2 I=1,4
      DO 2 J=1,6
    2 B(I,J)=0.D0
      B(1,2)=1.D0
      B(3,5)=2.D0
      RETURN
      END

      SUBROUTINE BV01B(B,X1,X2,Y1,Y2,FL,Z)
      DIMENSION B(4,6)
      DOUBLE PRECISION B,X1,X2,Y,Y1,Y2,Z,RY,XP,SP,FL,CP
      Y=Y1+Z*(Y2-Y1)
      IF(Y-.001)2,2,4
    2 Y=.001D0
    4 RY=1.D0/Y
      XP=Z*FL
      SP=(Y2-Y1)/FL
      CP=(X2-X1)/FL
      B(2,1)=SP*RY
      B(4,4)=B(2,1)
      B(2,2)=XP*B(2,1)
```

```
      B(4,5)=2.D0*B(2,2)
      B(4,6)=1.5D0*XP*B(4,5)
      B(2,3)=CP+RY
      B(2,4)=XP*B(2,3)
      B(2,5)=XP*B(2,4)
      B(2,6)=XP*B(2,5)
      B(3,6)=6.D0*XP
      RETURN
      END

      SUBROUTINE CC01B(C,FL)
      DIMENSION C(6,6)
      DOUBLE PRECISION C,FL
      DO 2 I=1,6
      DO 2 J=1,6
    2 C(I,J)=0.D0
      C(1,1)=1.D0
      C(2,3)=1.D0
      C(3,4)=1.D0
      C(4,1)=1.D0
      C(5,3)=1.D0
      C(6,4)=1.D0
      C(4,2)=FL
      C(5,4)=FL
      C(5,5)=FL**2
      C(5,6)=FL**3
      C(6,5)=2.D0*FL
      C(6,6)=3.D0*FL**2
      RETURN
      END

      SUBROUTINE CN02B(LBAND,NCON,T,DF,NE,NBAND)
      DIMENSION NCON(3,100),T(100),DF(100)
      DOUBLE PRECISION T,DF,TEMP,TH
      WRITE(6,100)
      NE=0
      NBAND=0
    2 READ(5,102)I,N1,N2,N3,TH,TEMP
      IF(I)20,20,3
    3 NCON(1,I)=N1
      NCON(2,I)=N2
      NCON(3,I)=N3
      T(I)=1.D0
      DF(I)=TEMP
      IF(TH)4,5,4
    4 T(I)=TH
    5 ND=IABS(N1-N2)
      IF(NBAND-ND)6,7,7
    6 NBAND=ND
    7 ND=IABS(N2-N3)
      IF(NBAND-ND)8,9,9
    8 NBAND=ND
    9 ND=IABS(N3-N1)
      IF(NBAND-ND)10,11,11
   10 NBAND=ND
   11 WRITE(6,103)I,N1,N2,N3,T(I),TEMP
      NE=NE+1
      GO TO 2
   20 NBAND=2*NBAND+2
      WRITE(6,105)NBAND
```

```
      IF(LBAND-NBAND)22,24,24
   22 WRITE(6,106)
      CALL EXIT
   24 RETURN
  100 FORMAT('0ELEM.NO.       CONNECTING NODES NUMBERD THICKNESS TEMP')
  102 FORMAT(4I5,2F10.5)
  103 FORMAT(I6,4X,3I8,7X,F6.2,2X,F6.1)
  105 FORMAT('0BAND WIDTH=',I5)
  106 FORMAT('0BAND WIDTH IS TOO LARGE')
      END

      SUBROUTINE CP01B(D,B,X1,X2,Y1,Y2,FL,A,H,P)
      DIMENSION B(4,6), D(4,4),DBY(4,6),P(6,6),A(6),H(6)
      DOUBLE PRECISION B,D,DBY,P,A,H,PL,FL,Z,X,Y
      DOUBLE PRECISION Y1,Y2,X1,X2,DIJ,PIJ
      PL=3.14159D0*FL
      DO 2 I=1,6
      DO 2 J=1,6
    2 P(I,J)=0.D0
      DO 30 IZ=1,6
      Z=A(IZ)
      X=.5D0*FL*(Z+1.D0)
      Y=Y1+X*(Y2-Y1)/FL
      B(2,1)=(Y2-Y1)/(FL*Y)
      B(2,2)=X*B(2,1)
      B(2,3)=(X2-X1)/(FL*Y)
      B(2,4)=X*B(2,3)
      B(2,5)=X*B(2,4)
      B(2,6)=X*B(2,5)
      B(3,6)=6.D0*X
      B(4,4)=B(2,1)
      B(4,5)=2.*X*B(4,4)
      B(4,6)=1.5*X*B(4,5)
      DO 12 I=1,4
      DO 12 J=1,6
      DIJ=0.D0
      DO 10 IJ=1,4
   10 DIJ=DIJ+D(I,IJ)*B(IJ,J)*Y
   12 DBY(I,J)=DIJ
      DO 22 I=1,6
      DO 22 J=1,6
      PIJ=0.D0
      DO 20 IJ=1,4
   20 PIJ=PIJ+B(IJ,I)*DBY(IJ,J)*H(IZ)
   22 P(I,J)=P(I,J)+PIJ
   30 CONTINUE
      DO 40 I=1,6
      DO 40 J=1,6
   40 P(I,J)=PL*P(I,J)
      RETURN
      END

      SUBROUTINE CN08B(LBAND,NBAND,NE,NCON,PR)
      DIMENSION NCON(3,1),PR(3,1),NC(4),PCXY(3)
      DOUBLE PRECISION PCXY,PR
      WRITE(6,100)
      NE=0
      NBAND=0
    2 READ(5,102)IE,(NC(I),I=1,3),PCXY
      IF(IE) 20,20,6
```

```
    6 WRITE(6,104) IE,(NC(I),I=1,3),PCXY
      NE=NE+1
      DO 8 I=1,3
      NCON(I,IE)=NC(I)
    8 PR(I,IE)=PCXY(I)
      NC(4)=NC(1)
      DO 12 I=1,3
      ND=IABS(NC(I+1)-NC(I))
      IF(ND-NBAND)12,12,10
   10 NBAND=ND
   12 CONTINUE
      GO TO 2
   20 NBAND=(NBAND+1)*6
      IF(LBAND-NBAND)22,24,24
   22 WRITE(6,106)
      CALL EXIT
   24 WRITE(6,108)NBAND
      RETURN
  100 FORMAT('0ELEM.NO.   CORNER NODE NUMBERS   PRESSURE   DP/DX
     1 DP/DY')
  102 FORMAT(4I5,4E10.5)
  104 FORMAT(I5,3X,3I7,4E12.4)
  106 FORMAT('0BAND WIDTH OVER LIMIT EXECUTION TERMINATED')
  108 FORMAT('0BAND WIDTH=',I3)
      END

      SUBROUTINE CN108(NCB,ARB,NB,NBAND)
      DIMENSION NCB(2,1),ARB(1)
      DOUBLE PRECISION ARB,AREA
      NB=0
      WRITE(6,102)
    2 READ(5,104)IB,AREA,NN1,NN2
      IF(IB)10,10,4
    4 WRITE(6,105)IB,AREA,NN1,NN2
      ARB(IB)=AREA
      NCB(1,IB)=NN1
      NCB(2,IB)=NN2
      NB=NB+1
      ND=2*(IABS(NN1-NN2)+1)
      IF(ND-NBAND)2,2,6
    6 NBAND=ND
      GO TO 2
   10 WRITE(6,110)NBAND
      RETURN
  102 FORMAT('0BAR NUMBER   X-SECT AREA   CONNECTS NODES NO.')
  104 FORMAT(I5,F10.5,2I5)
  105 FORMAT(I7,9X,F6.2,7X,2I5)
  110 FORMAT('0BAND WIDTH=',I5)
      END

      SUBROUTINE DB02B(YM,GNU,D,B)
      DIMENSION D(3,3),B(3,6)
      DOUBLE PRECISION YM,GNU,D,B
      D(1,1)=YM/(1.D0-GNU*GNU)
      D(1,2)=GNU*D(1,1)
      D(1,3)=0.D0
      D(2,1)=D(1,2)
      D(2,2)=D(1,1)
      D(2,3)=0.D0
      D(3,1)=0.D0
```

```
      D(3,2)=0.D0
      D(3,3)=.5D0*(1.D0-GNU)*D(1,1)
      DO 4 I=1,3
      DO 4 J=1,6
    4 B(I,J)=0.D0
      B(1,2)=1.D0
      B(2,6)=1.D0
      B(3,3)=1.D0
      B(3,5)=1.D0
      RETURN
      END

      SUBROUTINE DB03B (YM,GNU,D,B,DBD)
      DIMENSION D(3,3),B(3,12),DBD(12,12)
      DOUBLE PRECISION YM,GNU,D,B,DBD
      D(1,1)=YM/(1.-GNU*GNU)
      D(1,2)=GNU*D(1,1)
      D(1,3)=0.
      D(2,1)=D(1,2)
      D(2,2)=D(1,1)
      D(2,3)=0.
      D(3,1)=0.
      D(3,2)=0.
      D(3,3)=.5*(1.-GNU)*D(1,1)
      DO 4 I=1,3
      DO 4 J=1,12
    4 B(I,J)=0.
      B(1,2)=1.
      B(2,9)=1.
      B(3,3)=1.
      B(3,8)=1.
      DO 6 I=1,12
      DO 6 J=1,12
    6 DBD(I,J)=0.
      DBD(2,2)=D(1,1)
      DBD(9,2)=D(1,2)
      DBD(2,9)=DBD(9,2)
      DBD(3,3)=D(3,3)
      DBD(8,3)=DBD(3,3)
      DBD(3,8)=DBD(8,3)
      DBD(8,8)=DBD(3,3)
      DBD(9,9)=DBD(2,2)
      RETURN
      END

      SUBROUTINE ES02B(P,C,AM,E)
      DIMENSION P(6,6),C(6,6),AM(6,6),E(6,6),X(6,6)
      DOUBLE PRECISION P,C,AM,E,X,XIJ,EIJ
      DO 3 I=1,6
      DO 3 J=1,6
      XIJ=0.D0
      DO 2 IJ=1,6
    2 XIJ=XIJ+C(I,IJ)*AM(IJ,J)
    3 X(I,J)=XIJ
      DO 5 I=1,6
      DO 5 J=1,6
      EIJ=0.D0
      DO 4 IJ=1,6
    4 EIJ=EIJ+P(I,IJ)*X(IJ,J)
    5 E(I,J)=EIJ
```

```
      DO 7 I=1,6
      DO 7 J=1,6
      XIJ=0.D0
      DO 6 IJ=1,6
    6 XIJ=XIJ+C(IJ,I)*E(IJ,J)
    7 X(I,J)=XIJ
      DO 9 I=1,6
      DO 9 J=1,6
      EIJ=0.D0
      DO 8 IJ=1,6
    8 EIJ=EIJ+AM(IJ,I)*X(IJ,J)
    9 E(I,J)=EIJ
      RETURN
      END

      SUBROUTINE ES03B(AI,B,D,VOL,ET,E,FTE)
      DIMENSION D(3,3)
      DIMENSION AI(6,6),B(3,6),E(6,6),P1(3,6),P2(3,6),FTE(6),DE(3)
      DOUBLE PRECISION AI,B,D,VOL,ET,E,FTE,P1IJ,P2IJ,EIJ,DE,P1,P2
      DO 3 I=1,3
      DO 3 J=1,6
      P1IJ=0.D0
      DO 2 IJ=1,6
    2 P1IJ=P1IJ+B(I,IJ)*AI(IJ,J)
    3 P1(I,J)=P1IJ
      DO 5 I=1,3
      DO 5 J=1,6
      P2IJ=0.D0
      P2IJ=0.D0
      DO 4 IJ=1,3
    4 P2IJ=P2IJ+D(I,IJ)*P1(IJ,J)
    5 P2(I,J)=P2IJ
      DO 8 I=1,6
      DO 8 J=1,6
      EIJ=0.D0
      DO 6 IJ=1,3
    6 EIJ=EIJ+P1(IJ,I)*P2(IJ,J)
    8 E(I,J)=VOL*EIJ
      DO 10 I=1,3
   10 DE(I)=ET*(D(I,1)+D(I,2))
      DO 12 I=1,6
      FTE(I)=0.D0
      DO 12 IJ=1,3
   12 FTE(I)=FTE(I)+P1(IJ,I)*DE(IJ)*VOL
      RETURN
      END

      SUBROUTINE ES06B(V,AI,BDB,X,Y,E)
      DIMENSION AI(12,12),BDB(12,12),P(12,12),X(3),Y(3),E(12,12)
      DOUBLE PRECISION V,AI,BDB,X,Y,E,P,XY,XX,YY
      XX=0.
      YY=0.
      XY=0.
      DO 2 I=1,3
      XX=XX+X(I)*X(I)
      YY=YY+Y(I)*Y(I)
    2 XY=XY+X(I)*Y(I)
      XX=XX/12.
      YY=YY/12.
      XY=XY/12.
```

```
      BDB(4,4)=4.*XX*BDB(2,2)
      BDB(5,4)=2.*XY*BDB(2,2)
      BDB(4,5)=BDB(5,4)
      BDB(11,4)=2.*XX*BDB(9,2)
      BDB(4,11)=BDB(11,4)
      BDB(12,4)=4.*XY*BDB(9,2)
      BDB(4,12)=BDB(12,4)
      BDB(5,5)=YY*BDB(2,2)+XX*BDB(3,3)
      BDB(6,5)=2.*XY*BDB(3,3)
      BDB(5,6)=BDB(6,5)
      BDB(10,5)=2.*XX*BDB(3,3)
      BDB(5,10)=BDB(10,5)
      BDB(11,5)=XY*(BDB(9,2)+BDB(3,3))
      BDB(5,11)=BDB(11,5)
      BDB(12,5)=2.*YY*BDB(9,2)
      BDB(5,12)=BDB(12,5)
      BDB(6,6)=4.*YY*BDB(3,3)
      BDB(10,6)=4.*XY*BDB(3,3)
      BDB(6,10)=BDB(10,6)
      BDB(11,6)=.5*BDB(6,6)
      BDB(6,11)=BDB(11,6)
      BDB(10,10)=2.*BDB(10,5)
      BDB(11,10)=BDB(6,5)
      BDB(10,11)=BDB(11,10)
      BDB(11,11)=XX*BDB(2,2)+YY*BDB(3,3)
      BDB(12,11)=BDB(5,4)
      BDB(11,12)=BDB(12,11)
      BDB(12,12)=4.*YY*BDB(2,2)
      DO 6 I=1,12
      DO 6 J=1,12
      P(I,J)=0.
      DO 6 IJ=1,12
    6 P(I,J)=P(I,J)+BDB(I,IJ)*AI(IJ,J)
      DO 10 I=1,12
      DO 10 J=1,12
      E(I,J)=0.
      DO 8 IJ=1,12
    8 E(I,J)=E(I,J)+AI(IJ,I)*P(IJ,J)
   10 E(I,J)=V*E(I,J)
      RETURN
      END

      SUBROUTINE ES1OB(IB,NCB,XY,ARB,YMB,S,LBAND)
      DIMENSION NCB(2,1),XY(2,1),ARB(1),S(LBAND,1)
      DIMENSION BK(4,4),XXYY(4)
      DOUBLE PRECISION XY,ARB,YMB,S,BK,XXYY,X,Y,DSQRT
      NN1=NCB(1,IB)
      NN2=NCB(2,IB)
      X=XY(1,NN2)-XY(1,NN1)
      Y=XY(2,NN2)-XY(2,NN1)
      AEL=ARB(IB)*YMB/((DSQRT(X**2+Y**2))**3)
      XXYY(1)=-X
      XXYY(2)=X
      XXYY(3)=-Y
      XXYY(4)=Y
      DO 6 I=1,4
      DO 6 J=1,4
    6 BK(I,J)=AEL*XXYY(I)*XXYY(J)
      DO 10 I1=1,2
      DO 10 I2=1,2
```

```
      DO 10 J1=1,2
      DO 10 J2=1,2
      JJ=2*NCB(J2,IB)-2+J1
      II=2*NCB(I2,IB)-1+I1-JJ
      IF(II)10,10,8
    8 I=2*(I1-1)+I2
      J=2*(J1-1)+J2
      S(II,JJ)=S(II,JJ)+BK(I,J)
   10 CONTINUE
      RETURN
      END
      SUBROUTINE ES18B(CQ,D,G,P,E)
      DIMENSION G(21,18),D(3,3),P(7,7),E(18,18),CQ(21,3)
      DIMENSION CI(21),CJ(21),Q(21,21)
      DOUBLE PRECISION CQ,D,G,P,E,CI,CJ,Q,CV,C
      DO 8 I=1,21
      DO 8 J=1,21
    8 Q(I,J)=0.D0
      DO 32 KT=1,5
      GO TO (1,2,3,4,5),KT
    1 CV=4.D0*D(1,1)
      LII=0
      LJJ=0
      JCI=1
      JCJ=1
      GO TO 20
    2 CV=4.D0*D(1,2)
      LII=-2
      JCI=2
      GO TO 20
    3 CV=4.D0*D(1,1)
      LJJ=-2
      JCJ=2
      GO TO 20
    4 CV=4.D0*D(1,2)
      LII=0
      JCI=1
      GO TO 20
    5 CV=4.D0*D(3,3)
      LII=-1
      LJJ=-1
      JCI=3
      JCJ=3
   20 CONTINUE
      DO 30 KI=1,4
      III=(KI*(KI+3))/2+3
      DO 30 KJ=KI,4
      JJJ=(KJ*(KJ+3))/2+3
      DO 30 LI=1,KI
      ICQ=III+LI
      I=ICQ+LII
      DO 30 LJ=1,KJ
      JCQ=JJJ+LJ
      J=JCQ+LJJ
      II=KI+KJ+1-LI-LJ
      JJ=LI+LJ-1
   30 Q(I,J)=Q(I,J)+CV*CQ(ICQ,JCI)*CQ(JCQ,JCJ)*P(II,JJ)
   32 CONTINUE
      DO 34 I=1,21
```

```
      DO 34 J=1,21
   34 Q(J,I)=Q(I,J)
      DO 40 J=1,18
      DO 38 IC=1,21
      C=0.D0
      DO 36 IJ=1,21
   36 C=C+Q(IC,IJ)*G(IJ,J)
   38 CJ(IC)=C
      DO 40 I=1,18
      C=0.D0
      DO 39 IJ=1,21
   39 C=C+G(IJ,I)*CJ(IJ)
   40 E(I,J)=C
      RETURN
      END

      SUBROUTINE EV02B (A,CO,SI,FL,ND,G)
      DIMENSION A(21,21),CO(3),SI(3),FL(3),G(21,18),H(3,18),ND(4)
      DIMENSION D(3,3)
      DOUBLE PRECISION A,CO,SI,FL,G,H,F,S,SIGN,C,T
      DO 2 I=1,3
      DO 2 J=1,18
    2 H(I,J)=0.D0
      DO 10 I=1,3
      S=SI(I)
      C=CO(I)
      F=FL(I)
      SIGN=ND(I)
      IJ=6*(I-1)
      IJJ=IJ+6
      IF(IJJ-18)5,4,4
    4 IJJ=IJJ-18
    5 H(I,IJ+2)=-.5D0*S*SIGN
      H(I,IJJ+2)=H(I,IJ+2)
      H(I,IJ+3)=.5D0*C*SIGN
      H(I,IJJ+3)=H(I,IJ+3)
      H(I,IJ+4)=-.125D0*F*S*C*SIGN
      H(I,IJJ+4)=-H(I,IJ+4)
      H(I,IJ+5)=.125D0*F*(C*C-S*S)*SIGN
      H(I,IJJ+5)=-H(I,IJ+5)
      H(I,IJJ+6)=-H(I,IJ+4)
   10 H(I,IJJ+6)=-H(I,IJ+6)
      DO 20 I=1,21
      DO 20 J=1,18
      T=A(I,J)
      DO 18 IJ=1,3
   18 T=T+A(I,IJ+18)*H(IJ,J)
   20 G(I,J)=T
      RETURN
      END

      SUBROUTINE FP01B(PR,Y1,Y2,C,FL,AM,KE,F)
      DIMENSION  C(6,6),AM(6,6),F(1),FP(6),CFP(6)
      DOUBLE PRECISION C,AM,F,FP,CFP,PR,FL,Y1,Y2
      FP(3)=PR*.5D0*(Y1+Y2)*FL
      FP(4)=PR/6.D0*(Y1+2.D0*Y2)*FL**2
      FP(5)=PR/12.D0*(Y1+3.D0*Y2)*FL**3
      FP(6)=PR*0.05D0*(Y1+4.D0*Y2)*FL**4
      DO 4 I=1,6
      CFP(I)=0.D0
```

```
      DO 4 J=3,6
    4 CFP(I)=CFP(I)+C(J,I)*FP(J)
      DO 8 I=1,6
      FP(I)=0.D0
      DO 8 J=1,6
    8 FP(I)=FP(I)+AM(J,I)*CFP(J)
      DO 10 J=1,6
      JJ=3*KE-3+J
   10 F(JJ)=F(JJ)+FP(J)
      RETURN
      END

      SUBROUTINE FU10B(S,U,NBAND,NDF,NCN,LBAND)
      DIMENSION S(LBAND,1),U(1),FC(6),UC(6)
      DOUBLE PRECISION S,U,FORCE,UC,FC
      WRITE(6,100)
      NN=NDF/NCN
      DO 20 IN=1,NN
      DO 16 II=1,NCN
      IJ=(IN-1)*NCN+II
      FORCE=S(1,IJ)*U(IJ)
      DO 12 I=2,NBAND
      JL=IJ+1-I
      IF(JL)8,8,5
    5 FORCE=FORCE+S(I,JL)*U(JL)
    8 JR=IJ-1+I
      IF(NDF-JR)12,10,10
   10 FORCE=FORCE+S(I,IJ)*U(JR)
   12 CONTINUE
      UC(II)=U(IJ)
   16 FC(II)=FORCE
   20 WRITE(6,102) IN,(FC(II),II=1,NCN),(UC(II),II=1,NCN)
      RETURN
  100 FORMAT('0NODE NO.      FORCE AND DISPLACEMENT COMPONENTS')
  102 FORMAT(2X,I5,3X,10(E11.4,2X),/,98X,2(E11.4,2X))
      END

      SUBROUTINE GD02B(E,P,T,A,H,D)
      DIMENSION A(6),H(6),D(4,4)
      DOUBLE PRECISION A,H,D,E,T,P
      A(1)=-.93247D0
      A(2)=-.66121D0
      A(3)=-.23862D0
      A(4)=-A(3)
      A(5)=-A(2)
      A(6)=-A(1)
      H(1)=.17132D0
      H(2)=.36076D0
      H(3)=.46791D0
      H(4)=H(3)
      H(5)=H(2)
      H(6)=H(1)
      DO 5 I=1,4
      DO 5 J=1,4
    5 D(I,J)=0.D0
      D(1,1)=E*T/(1.D0-P*P)
      D(2,2)=D(1,1)
      D(1,2)=P*D(1,1)
      D(2,1)=D(1,2)
      D(3,3)=T*T/12.*D(1,1)
```

```
      D(4,4)=D(3,3)
      D(3,4)=P*D(3,3)
      D(4,3)=D(3,4)
      WRITE(6,100)E,P,T
      RETURN
  100 FORMAT('0YOUNGS MODULUS    =',E10.2,/,' POISSONS RATIO    =',F6.3,
     1/,' THICKNESS          =',F6.3)
      END

      SUBROUTINE GD03B(YM,GNU,A,H,D)
      DIMENSION A(3),H(3),D(4,4)
      DOUBLE PRECISION A,H,D,YM,GNU
      A(1)=.23861919
      H(1)=.46791393
      A(2)=.66120939
      H(2)=.36076157
      A(3)=.93246951
      H(3)=.17132449
      DO 4 I=1,4
      DO 4 J=1,4
    4 D(I,J)=0.
      D(1,1)=YM*(1.-GNU)/((1.+GNU)*(1.-2.*GNU))
      D(2,2)=D(1,1)
      D(3,3)=D(1,1)
      D(4,4)=.5*YM/(1.*GNU)
      D(1,2)=D(1,1)*GNU/(1.-GNU)
      D(1,3)=D(1,2)
      D(2,1)=D(1,2)
      D(2,3)=D(1,2)
      D(3,1)=D(1,2)
      D(3,2)=D(1,2)
      RETURN
      END

      SUBROUTINE GE02B(F,U,S,IFX,NBAND,NDF,LBAND)
      DIMENSION S(LBAND,NDF),F(NDF),U(NDF),IFX(1)
      DOUBLE PRECISION CP,F,U,S,R,C
      CP=1.0D12
      DO 6 I=1,NDF
      IF(IFX(I))6,6,4
    4 S(1,I)=CP*S(1,I)
      F(I)=S(1,I)*U(I)
    6 CONTINUE
      IJ=1
      NA=NBAND
      NR=NDF-NBAND
   10 R=1.D0/S(1,IJ)
      F(IJ)=R*F(IJ)
      IF(NA-1)30,30,12
   12 DO 20 II=2,NA
      C=S(II,IJ)
      S(II,IJ)=R*S(II,IJ)
      DO 18 JJ=2,II
      I=II+1-JJ
      J=IJ-1+JJ
   18 S(I,J)=S(I,J)-C*S(JJ,IJ)
      IIF=IJ-1+II
   20 F(IIF)=F(IIF)-C*F(IJ)
      NR=NR-1
      IF(NR)22,24,24
```

```
   22 NA=NA-1
   24 IJ=IJ+1
      GO TO 10
   30 NA=1
      IJ=NDF
      U(IJ)=F(IJ)
   32 IJ=IJ-1
      IF(IJ)34,34,36
   34 RETURN
   36 IF(NA-NBAND)38,40,40
   38 NA=NA+1
   40 C=F(IJ)
      DO 44 I=2,NA
      J=IJ-1+I
   44 C=C-S(I,IJ)*U(J)
      U(IJ)=C
      GO TO 32
      END

      SUBROUTINE GL01B(X1,X2,Y1,Y2,AM,FL)
      DIMENSION AM(6,6)
      DOUBLE PRECISION AM,DSQRT,FL,Y1,Y2,X1,X2
      DO 2 I=1,6
      DO 2 J=1,6
    2 AM(I,J)=0.D0
      FL=DSQRT((X2-X1)**2+(Y2-Y1)**2)
      AM(1,1)=(X2-X1)/FL
      AM(1,2)=(Y2-Y1)/FL
      AM(2,1)=-AM(1,2)
      AM(2,2)=AM(1,1)
      AM(3,3)=1.D0
      DO 4 I=1,3
      DO 4 J=1,3
    4 AM(I+3,J+3)=AM(I,J)
      RETURN
      END

      SUBROUTINE GN02B(S,U,F,IFX,NB,LBAND,TRIG)
      DIMENSION S(LBAND,1),U(1),F(1),IFX(1),TRIG(1)
      DOUBLE PRECISION S,U,F,TRIG,ALP,UK,FT,C,SN,FS,CS
      DOUBLE PRECISION S11,S22,C2,S2,S21,S1,DSIN,DCOS
      WRITE(6,100)
    2 READ(5,102)NN,ALP,UK,FT
      IF(NN)4,4,6
    4 RETURN
    6 NN2=2*NN
      NN21=NN2-1
      IFX(NN21)=1
      U(NN21)=UK
      WRITE(6,106)NN,ALP,UK,FT
      S11=S(1,NN21)
      S21=S(2,NN21)
      S22=S(1,NN2)
      ALP=3.1415900/180.D0*ALP
      C=DCOS(ALP)
      TRIG(NN21)=C
      C2=C*C
      SN=DSIN(ALP)
      TRIG(NN2)=SN
      S2=SN*SN
```

```
      CS=C*SN
      FS=F(NN21)
      F(NN21)=C*F(NN21)+SN*F(NN2)
      F(NN2)=-SN*FS+C*F(NN2)+FT
      S(1,NN21)=C2*S11+2.D0*CS*S21+S2*S22
      S(2,NN21)=-CS*S11+(C2-S2)*S21+CS*S22
      S(1,NN2)=S2*S11-2.D0*CS*S21+C2*S22
      NB1=NB-1
      DO 30 I=2,NB1
      I1=I+1
      J=NN2-I
      IF(J)20,20,10
   10 S1=S(I,J)
      S2=S(I1,J)
      S(I,J)=C*S1+SN*S2
      S(I1,J)=-SN*S1+C*S2
   20 S1=S(I1,NN21)
      S2=S(I,NN2)
      S(I1,NN21)=C*S1+SN*S2
   30 S(I,NN2)=-SN*S1+C*S2
      GO TO 2
  102 FORMAT(I5,3F10.5)
  106 FORMAT(I6,F11.1,4X,E10.3,2X,E10.3)
      END

      SUBROUTINE IN12B(A,N)
      DIMENSION A(N,N),B(21,21)
      DOUBLE PRECISION A,B,SUM,FN,DABS,AVA,RER,R,C
      FN=N
      SUM=0.D0
      DO 4 I=1,N
      DO 2 J=1,N
    2 B(I,J)=0.D0
      SUM=SUM+DABS(A(I,I))
    4 B(I,I)=1.D0
      AVA=SUM/FN
      RER=.000001*AVA
      DO 40 I=1,N
      IF(DABS(A(I,I))-RER)6,6,20
    6 IF(I-N)10,7,10
    7 WRITE(6,100)
      CALL EXIT
   10 DO 12 II=I,N
      IF(DABS(A(II,I))-RER)12,12,14
   12 CONTINUE
      GO TO 7
   14 DO 16 JJ=1,N
      A(I,JJ)=A(I,JJ)+A(II,JJ)
   16 B(I,JJ)=B(I,JJ)+B(II,JJ)
   20 R=1.D0/A(I,I)
      DO 22 J=1,N
      A(I,J)=R*A(I,J)
   22 B(I,J)=R*B(I,J)
      DO 37 II=1,N
      IF(II-I)32,37,32
   32 C=-A(II,I)
      DO 35 J=1,N
      A(II,J)=A(II,J)+C*A(I,J)
      B(II,J)=B(II,J)+C*B(I,J)
```

```
      35 CONTINUE
      37 CONTINUE
      40 CONTINUE
         DO 42 I=1,N
         DO 42 J=1,N
      42 A(I,J)=B(I,J)
         RETURN
     100 FORMAT('0     *****   MATRIX IS SINGULAR   *****')
         END

         SUBROUTINE KF01B(F,NDF)
         DIMENSION F(120)
         WRITE(6,100)
         DO 2 I=1,NDF
       2 F(I)=0.
       4 READ(5,104)I,FT
         IF(I)10,10,5
       5 F(I)=FT
         WRITE(6,106)I,FT
         GO TO 4
      10 RETURN
     100 FORMAT('0  KNOWN NON-ZERO LOADS',/,' COMPONENT NUMBER  LOAD')
     104 FORMAT(I5,F10.5)
     106 FORMAT(5X,I5,7X,E11.4)
         END

         SUBROUTINE KU01B(U,IHLD,NDF)
         DIMENSION U(100),IHLD(100)
         DOUBLE PRECISION U,UT
         WRITE(6,100)
         DO 2 I=1,NDF
         U(I)=0.D0
       2 IHLD(I)=0
       4 READ(5,104)I,UT
         IF(I)10,10,5
       5 IHLD(I)=1
         U(I)=UT
         WRITE(6,106)I,UT
         GO TO 4
      10 RETURN
     100 FORMAT('0         KNOWN DISPLACEMENTS',/,' COMPONENT NUMBER  DISP
        1EMENT')
     104 FORMAT(I5,F10.5)
     106 FORMAT(7X,I5,8X,E11.4)
         END

         SUBROUTINE KF02B(F)
         DIMENSION F(1)
         DOUBLE PRECISION F,FT
         WRITE(6,100)
       4 READ(5,104)I,FT
         IF(I)10,10,5
       5 F(I)=F(I)+FT
         WRITE(6,106)I,FT
         GO TO 4
      10 RETURN
     100 FORMAT('0  KNOWN NON-ZERO LOADS',/,' COMPONENT NUMBER  LOAD')
     104 FORMAT(I5,F10.5)
     106 FORMAT(5X,I5,7X,E11.4)
         END
```

```
      SUBROUTINE KF01B(F,NDF)
      DIMENSION F(300)
      DOUBLE PRECISION F,FT
      WRITE(6,100)
      DO 2 I=1,NDF
    2 F(I)=0.D0
    4 READ(5,104)I,FT
      IF(I)10,10,5
    5 F(I)=FT
      WRITE(6,106)I,FT
      GO TO 4
   10 RETURN
  100 FORMAT('0  KNOWN NON-ZERO LOADS',/,' COMPONENT NUMBER LOAD')
  104 FORMAT(I5,F10.5)
  106 FORMAT(5X,I5,7X,E11.4)
      END
**WARNING**  SUBPROGRAM KF01B  WAS PREVIOUSLY DEFINED. FIRST DEFINITION IS USED

      SUBROUTINE KU01B(U,IHLD,NDF)
      DIMENSION U(300),IHLD(300)
      DOUBLE PRECISION U,UT
      WRITE(6,100)
      DO 2 I=1,NDF
      U(I)=0.D0
    2 IHLD(I)=0
    4 READ(5,104)I,UT
      IF(I)10,10,5
    5 IHLD(I)=1
      U(I)=UT
      WRITE(6,106)I,UT
      GO TO 4
   10 RETURN
  100 FORMAT('0      KNOWN DISPLACEMENTS',/,' COMPONENT NUMBER  DISPLAC
     1EMENT')
  104 FORMAT(I5,F10.5)
  106 FORMAT(7X,I5,8X,E11.4)
      END
**WARNING**  SUBPROGRAM KU01B  WAS PREVIOUSLY DEFINED. FIRST DEFINITION IS USED

      SUBROUTINE LR01B(S,LBAND,NBAND,NU,T,NF)
      DIMENSION S(LBAND,1),T(NF,NF),P(6),TJ(6),FK(6,6)
      DOUBLE PRECISION S,T,P,TJ,FK,SUM
      NBF=NBAND-NF+1
      DO 20 J=1,NF
      DO 4 I=1,NF
    4 TJ(I)=T(I,J)
      DO 12 I=1,NF
      SUM=0.D0
      DO 10 IJ=1,NF
      JJ=NU+IJ
      II=I+1-IJ
      IF(II)8,8,10
    8 JJ=JJ+II-1
      II=2-II
   10 SUM=SUM+S(II,JJ)*TJ(IJ)
   12 P(I)=SUM
      DO 20 IK=1,NF
      SUM=0.D0
      DO 14 IJ=1,NF
   14 SUM=SUM+T(IJ,IK)*P(IJ)
```

```
   20 FK(IK,J)=SUM
      DO 30 J=1,NF
      JJ=NU+J
      DO 30 I=J,NF
      II=I+1-J
   30 S(II,JJ)=FK(I,J)
      DO 40 IS=2,NBF
      JJ=NU+2-IS
      IF(JJ)41,41,31
   31 DO 36 IP=1,NF
      SUM=0.D0
      DO 32 IJ=1,NF
      II=IS-1+IJ
   32 SUM=SUM+T(IJ,IP)*S(II,JJ)
   36 P(IP)=SUM
      DO 40 IJ=1,NF
      II=IS-1+IJ
   40 S(II,JJ)=P(IJ)
   41 DO 50 IS=2,NBF
      DO 46 IP=1,NF
      SUM=0.D0
      DO 42 IJ=1,NF
      JJ=NU+IJ
      II=IS+NF-IJ
   42 SUM=SUM+S(II,JJ)*T(IJ,IP)
   46 P(IP)=SUM
      DO 50 IJ=1,NF
      JJ=NU+IJ
      II=IS+NF-IJ
   50 S(II,JJ)=P(IJ)
      RETURN
      END

      SUBROUTINE LR02B(F,NU,T,NF)
      DIMENSION F(1),T(NF,NF),FF(10)
      DOUBLE PRECISION F,T,FF,SUM
      DO 2 I=1,NF
    2 FF(I)=F(NU+I)
      DO 6 I=1,NF
      SUM=0.D0
      DO 4 IJ=1,NF
    4 SUM=SUM+T(IJ,I)*FF(IJ)
    6 F(NU+I)=SUM
      RETURN
      END

      SUBROUTINE RC02B(NF,XY,NDF)
      DIMENSION XY(2,1)
      DOUBLE PRECISION XY,X,Y
      NDF=0
      WRITE(6,100)
    2 READ(5,102)I,X,Y
      IF(I)20,20,4
    4 XY(1,I)=X
      XY(2,I)=Y
      NDF=NDF+NF
      WRITE(6,104) I,X,Y
      GO TO 2
   20 RETURN
  100 FORMAT('ONODE NO.    X-COORD        Y-COORD')
```

```
  102 FORMAT(I5,2F10.5)
  104 FORMAT(I6,2X,2E12.4)
      END

      SUBROUTINE RC03B(XY,NE,NDF)
      DIMENSION XY(2,6J)
      DOUBLE PRECISION XY,X,Y
      NE=-1
      WRITE(6,100)
    2 READ(5,102)I,X,Y
      IF(I)20,20,4
    4 XY(1,I)=X
      XY(2,I)=Y
      NE=NE+1
      WRITE(6,104)I,X,Y
      GO TO 2
   20 NDF=3*NE+3
      RETURN
  100 FORMAT('ONODE NO.   X-COORD      Y-COORD')
  102 FORMAT(I5,2F10.5)
  104 FORMAT(I6,2X,2E12.4)
      END

      SUBROUTINE RC06B(NF,XY,NDF)
      DIMENSION XY(2,1)
      DOUBLE PRECISION XY,X,Y
      WRITE(6,100)
      READ(5,102)NN
      NDF=NF*NN
    2 READ(5,102)I,X,Y
      IF(I)20,20,4
    4 XY(1,I)=X
      XY(2,I)=Y
      WRITE(6,104)I,X,Y
      GO TO 2
   20 WRITE(6,106)NDF
      RETURN
  100 FORMAT('ONODE NO.   X-COORD      Y-COORD')
  102 FORMAT(I5,2F10.5)
  104 FORMAT(I6,2X,2E12.4)
  106 FORMAT( 'OSYSTEM HAS',I4,' DEGREES OF FREEDOM')
      END

      SUBROUTINE PF01B(IE,PR,P,G,PF)
      DIMENSION PR(3,1),P(7,7),PF(18),PP(21),G(21,18)
      DOUBLE PRECISION PR,P,PF,PP,G,PC,PX,PY,C
      PC=PR(1,IE)
      PX=PR(2,IE)
      PY=PR(3,IE)
      I=0
      DO 2 K=1,6
      DO 2 JJ=1,K
      II=K+1-JJ
      I=I+1
    2 PP(I)=PC*P(II,JJ)+PX*P(II+1,JJ)+PY*P(II,JJ+1)
      DO 6 I=1,18
      C=0.D0
    6 PF(I)=C
      RETURN
      END
```

```
      SUBROUTINE RU01B(U,TRIG,NDF)
      DIMENSION U(1),TRIG(1)
      DOUBLE PRECISION U,TRIG,US
      DO 20 I=2,NDF,2
      IF(TRIG(I))5,20,5
    5 I1=I-1
      US=U(I1)
      U(I1)=TRIG(I1)*U(I1)-TRIG(I)*U(I)
      U(I)=TRIG(I)*US+TRIG(I1)*U(I)
   20 CONTINUE
      RETURN
      END

      SUBROUTINE RU02B(U,NU,T,NF)
      DIMENSION U(1),T(NF,NF),UP(10)
      DOUBLE PRECISION U,T,UP,SUM
      DO 2 I=1,NF
      IU=NU+I
    2 UP(I)=U(IU)
      DO 6 I=1,NF
      SUM=0.D0
      DO 4 IJ=1,NF
    4 SUM=SUM+T(I,IJ)*UP(IJ)
      IU=NU+I
    6 U(IU)=SUM
      RETURN
      END

      SUBROUTINE ST03B(D,B,AI,ET,U,NCON,IE)
      DIMENSION D(3,3),B(3,6),AI(6,6),U(1),NCON(3,1),UE(6),STR(3),P(6),
     1P(3)
      DOUBLE PRECISION D,B,AI,ET,U,UE,STR,P,EP,C,A,R,TH,DATAN
      DOUBLE PRECISION DSQRT,S1,S2
      DO 2 K=1,2
      DO 2 IH=1,3
      I=3*(K-1)+IH
      II=2*NCON(IH,IE)+K-2
    2 UE(I)=U(II)
      DO 4 I=1,6
      P(I)=0.D0
      DO 4 IJ=1,6
    4 P(I)=P(I)+AI(I,IJ)*UE(IJ)
      EP(1)=-ET
      EP(2)=-ET
      EP(3)=0.D0
      DO 6 I=1,3
      DO 6 IJ=1,6
    6 EP(I)=EP(I)+B(I,IJ)*P(IJ)
      DO 8 I=1,3
      STR(I)=0.D0
      DO 8 IJ=1,3
    8 STR(I)=STR(I)+D(I,IJ)*EP(IJ)
      A=.5D0*(STR(1)+STR(2))
      C=STR(1)-A
      R=DSQRT(C**2+STR(3)**2)
      IF(C)9,10,10
    9 R=-R
   10 S1=A+R
      S2=A-R
      TH=90.D0/3.14159D0*DATAN(STR(3)/C)
```

```
      WRITE(6,100)IE,(STR(I),I=1,3),TH,S1,S2
      RETURN
  100 FORMAT(I6,2X,3E11.3,F10.1,2E10.3)
      END

      SUBROUTINE ST06B(DB,AI,U,NCON,IE)
      DIMENSION DB(3,12),AI(12,12),U(1),NCON(6,1),UE(12),STR(3),P(12)
      DOUBLE PRECISION DB,AI,U,UE,STR,P,DSQRT,S1,S2,DATAN
      DOUBLE PRECISION C,A,R,TH
      DO 2 IA=1,2
      DO 2 IB=1,6
      I=6*(IA-1)+IB
      II=2*NCON(IB,IE)+IA-2
    2 UE(I)=U(II)
      DO 4 I=1,12
      P(I)=0.
      DO 4 IJ=1,12
    4 P(I)=P(I)+AI(I,IJ)*UE(IJ)
      DO 6 I=1,3
      STR(I)=0.
      DO 6 IJ=1,12
    6 STR(I)=STR(I)+DB(I,IJ)*P(IJ)
      A=.5*(STR(1)+STR(2))
      C=STR(1)-A
      R=DSQRT(C**2+STR(3)**2)
      IF(C)9,10,10
    9 R=-R
   10 S1=A+R
      S2=A-R
      TH=90./3.14159*DATAN(STR(3)/C)
      WRITE(6,100)IE,(STR(I),I=1,3),TH,S1,S2
      RETURN
  100 FORMAT(I6,3X,3(E10.3,3X),F6.1,3X,E10.3,3X,E10.3)
      END

      SUBROUTINE TI01B(NP,X,Y,P)
      DIMENSION X(4),Y(4),P(NP,NP)
      DOUBLE PRECISION X,Y,P,X1,X2,X3,Y1,Y2,Y3,A,C
      NL=NP-1
      X1=X(1)
      X2=X(2)
      X3=X(3)
      Y1=Y(1)
      Y2=Y(2)
      Y3=Y(3)
      A=0.D0
      DO 1 I=1,3
    1 A=A+.5D0*(X(I)*Y(I+1)-X(I+1)*Y(I))
      P(1,1)=1.D0
      P(1,2)=0.D0
      P(2,1)=0.D0
      DO 14 N=2,6
      IF(N-NL)3,3,20
    3 K=N-1
      GO TO (4,5,5,7,8),K
    4 C=1.D0/12.D0
      GO TO 10
    5 C=1.D0/30.D0
      GO TO 10
    7 C=2.D0/105.D0
```

```
      GO TO 10
    8 C=1.D0
   10 NE=N+1
      DO 12 I=2,N
      I1=I-1
      J1=NE-I
      J=J1+1
   12 P(I,J)=C*(X1**I1*Y1**J1+X2**I1*Y2**J1+X3**I1*Y3**J1)
      P(1,NE)=C*(Y1**N+Y2**N+Y3**N)
      P(NE,1)=C*(X1**N+X2**N+X3**N)
   14 CONTINUE
      P(7,1)=P(7,1)/84.D0+15.D0/14.D0*P(5,1)*P(3,1)
      P(6,2)=-P(6,2)/168.D0+15.D0/14.D0*(2.D0*P(5,1)*P(2,2)
     1+5.D0*P(4,1)*P(3,2)+2.D0*P(3,1)*P(4,2))
      P(5,3)=-11.D0*P(5,3)/840.D0+1.D0/14.D0*(6.D0*P(5,1)*P(1,3)
     1+24.D0*P(3,1)*P(3,3)+30.D0*P(4,1)*P(2,3)+
     148.D0*P(4,2)*P(2,2)+75.D0*P(3,2)**2)
      P(4,4)=-P(4,4)/56.D0+1.D0/28.D0*(25.D0*P(4,1)*P(1,4)
     1+225.D0*P(3,2)*P(2,3)+36.D0*P(3,1)*P(2,4)+
     136.D0*P(1,3)*P(4,2)+108.D0*P(2,2)*P(3,3))
      P(3,5)=-11.D0*P(3,5)/840.D0+1.D0/14.D0*(6.D0*P(1,5)*P(3,1)
     1+24.D0*P(1,3)*P(3,3)+30.D0*P(1,4)*P(3,2)
     1+48.D0*P(2,4)*P(2,2)+75.D0*P(2,3)**2)
      P(2,6)=-P(2,6)/168.D0+15.D0/14.D0*(2.D0*P(1,5)*P(2,2)+
     15.D0*P(1,4)*P(2,3)+2.D0*P(1,3)*P(2,4))
      P(1,7)=P(1,7)/84.D0+15.D0/14.D0*P(1,5)*P(1,3)
   20 DO 24 I=1,NP
      JL=NP+1-I
      DO 24 J=1,JL
   24 P(I,J)=A*P(I,J)
      RETURN
      END

      SUBROUTINE TM02B(TH,T)
      DIMENSION T(5,5)
      DOUBLE PRECISION T,TH,C,DCOS,S,DSIN,CC,SS,CS
      TH=3.14159D0*TH/180.D0
      C=DCOS(TH)
      S=DSIN(TH)
      CC=C*C
      CS=C*S
      SS=S*S
      T(1,1)=C
      T(1,2)=-S
      T(2,1)=S
      T(2,2)=C
      T(3,3)=CC
      T(3,4)=-2.D0*CS
      T(3,5)=SS
      T(4,3)=CS
      T(4,4)=CC-SS
      T(4,5)=-CS
      T(5,3)=SS
      T(5,4)=2.D0*CS
      T(5,5)=CC
      RETURN
      END

      SUBROUTINE ST19B(D,U,NCON,TH,NN)
      DIMENSION U(1),D(3,3),EM(3),STR(3),C(3),NCON(NN,1)
```

```
      DOUBLE PRECISION D,U,TH,EM,STR,C,S,A,B,R,DSQRT,THE,DATAN,S1,S2
      DO 20 IN=1,NN
      C(1)=U(6*IN-2)
      C(2)=U(6*IN)
      C(3)=2.D0*U(6*IN-1)
      DO 8 I=1,3
      S=0.D0
      DO 6 IJ=1,3
    6 S=S+D(I,IJ)*C(IJ)
      EM(I)=S
    8 STR(I)=-6.D0*S/TH**2
      A=.5D0*(STR(1)+STR(2))
      B=STR(1)-A
      R=DSQRT(B**2+STR(3)**2)
      IF(B)9,10,10
    9 R=-R
   10 S1=A+R
      S2=A-R
      THE=90.D0/3.14159D0*DATAN(STR(3)/B)
   20 WRITE(6,100)IN,EM,STR,THE,S1,S2
      RETURN
  100 FORMAT(I6,3X,6E10.3,F9.1,2E10.3)
      END
      SUBROUTINE ST10B(NB,NCB,XY,U,YMB)
      DIMENSION NCB(2,1),XY(2,1),U(1)
      DOUBLE PRECISION X,Y,U,YMB,EL,XY,STR
      WRITE(6,100)
      IF(NB)6,6,2
    2 DO 4 IB=1,NB
      NN1=NCB(1,IB)
      NN2=NCB(2,IB)
      X=XY(1,NN2)-XY(1,NN1)
      Y=XY(2,NN2)-XY(2,NN1)
      EL=YMB/(X**2+Y**2)
      STR=EL*(X*(U(2*NN2-1)-U(2*NN1-1))+Y*(U(2*NN2)-U(2*NN1)))
    4 WRITE(6,102)IB,STR
    6 RETURN
  100 FORMAT('0BAR NO.     STRESS PSI')
  102 FORMAT(I6,3X,E12.4)
      END
C
C
C
C
C
C
C
C
C
C
C
C
C
C
C
C
C
C
C
```