NASA TM-100670

NASA Technical Memorandum 100670

FINITE ELEMENT COMPOSITE ANALYSIS PROGRAM (FECAP) FOR A MICROCOMPUTER

David E. Bowles

July 1988

NASA
National Aeronautics and
Space Administration

Langley Research Center
Hampton, Virginia 23665

FINITE ELEMENT COMPOSITE ANALYSIS PROGRAM (FECAP) FOR A MICROCOMPUTER

David E. Bowles
NASA-Langley Research Center
Hampton, Virginia 23665

ABSTRACT

A special purpose finite element composite analysis program (FECAP) for analyzing composite material behavior with a microcomputer is described. The formulation assumes a state of generalized plane strain in a material consisting of two or more orthotropic phases. Loading can be mechanical and/or thermal. The theoretical background, computer implementation, and program users guide are described in detail. A sample problem is solved showing the required user input and computer generated output.

NOMENCLATURE

Matrix Notation:

[ ]	rectangular or square matrix
$[\ ]^{-1}$	matrix inverse
$[\ ]^T$	matrix transpose
{ }	column vector
$\{\ \}^T$	row vector (transpose of column vector)

Symbol Definitions:

All symbols are defined in the text.

INTRODUCTION

Advanced composite materials have gained wide use in the aerospace industry over the last 20 years because of their high specific strength and stiffness, and low coefficient of thermal expansion (CTE). Design of composite structures requires the analysis of composite material behavior. Composite materials, for the purpose of this report, are defined as the combination of at least two distinct materials to form one nonhomogeneous anisotropic material. The anisotropic nonhomogeneous nature of composites has limited the number of closed-form analytical solutions available to analyze their behavior. Numerical techniques have been widely applied to many problems in composite material behavior. The finite element method has been one of the most popular of the numerical techniques.

Finite element analyses generally require large amounts of computer memory, and therefore in the past have been mainly used on large main-frame computer systems. However, recent advances in microcomputer technology have made finite element analysis on desk-top machines practical. Many large general purpose finite element programs are now commercially available for microcomputers. This report will describe a special purpose finite element composite analysis program (FECAP) developed at NASA-LaRC to analyze composite material behavior on a microcomputer. The theoretical formulation is presented first, followed by a description of the computer program and a users guide. A sample problem is solved showing the required user input and computer generated output.

THEORETICAL BACKGROUND

Problem Description

The problem under consideration is an orthotropic body, consisting of one or more orthotropic materials, in a state of generalized plane strain. The material is assumed to be linear elastic and temperature independent. The coordinate system, and the different levels of analysis possible for a continuous fiber reinforced composite (i.e. laminate or constituent level), for this problem are shown in fig. 1. The displacement field for this problem may be written as:

$$
\begin{aligned}
u(x,y,z) &= U(y,z) + (\varepsilon_{xo})x \\
v(x,y,z) &= V(y,z) \\
w(x,y,z) &= W(y,z)
\end{aligned}
\tag{1}
$$

where u, v, and w are the total displacements in the x,y, and z directions, respectively, and U,

V, and W are the unknown displacement functions to be determined. ε_{xo} is a constant uniform strain in the x direction, and may be treated as either a known or unknown quantity. The displacement functions U, V, and W are only functions of y and z. Therefore the problem becomes two dimensional even though all three components of displacement, and all six stress and strain components are present. This type of problem is often referred to as quasi-three dimensional.

Finite Element Geometry

The finite element formulation [1] is presented for both 4-node and 8-node isoparametric elements. The global and local coordinate systems for this type of element are shown in fig. 2. The relationship between the global and local coordinates is written as

$$
y = N_i y_i \quad i=1,4 \text{ or } 1,8
$$

$$
z = N_i z_i \quad i=1,4 \text{ or } 1,8
\tag{2}
$$

where the interpolation functions N_i are functions of this local coordinates (ξ, η), and y_i and z_i are the global coordinates of the nodes.

The interpolation functions for the 4-node element are linear, and are defined as

$$
N_i = (1/4)(1+\xi_0)(1+\eta_0), \quad i=1,4
\tag{3}
$$

where $\xi_0 = (\xi)(\xi_i)$, $\eta_0 = (\eta)(\eta_i)$, and ξ and η range from -1 to +1 at the corners. The interpolation functions for the 8-node elements are quadratic, and are defined as

        Corner nodes -
$$
N_i = (1/4)(1+\xi_0)(1+\eta_0)(\xi_0+\eta_0-1), \quad i=1,3,5, \text{ and } 7
\tag{4}
$$
        Mid-side nodes -
$$
N_i = (1/2)(1-\xi^2)(1+\eta_0), \quad \xi_i=0, \ i=2 \text{ and } 6
\tag{5}
$$

$$N_i = (1/2)(1+\xi_0)(1-\eta^2), \eta_i=0, i=4 \text{ and } 8 \tag{6}$$

Both ξ_0 and η_0 have the same definition as given for the 4-node interpolation functions.

Isoparametric elements use the same interpolation functions for the geometry and the nodal variables (i.e. displacements). Therefore, the unknown displacement functions of equ. (1) may be written as

$$
\begin{aligned}
U(y,z) &= N_i u_i \\
V(y,z) &= N_i v_i \\
W(y,z) &= N_i w_i
\end{aligned}
\tag{7}
$$

where u_i, v_i, and w_i are the unknown displacements at the nodes.

Finite Element Formulation

The finite element formulation used in FECAP is displacement based and requires the minimization of the total potential energy, PE, for each element with respect to the unknown variables. This procedure leads to a set of linear simultaneous equations relating the unknown nodal displacements to the applied loads. The equations for each element are assembled into a global system, the boundary conditions are applied, and the system is solved for the nodal displacements. A detailed description of the entire finite element formulation is given in reference [1]. A brief description is given in this report.

The total PE of an element is the sum of the strain energy, U_S, and the work of the external loads, W_L. The strain energy for an element is written as

$$U_S = 1/2 \int_{vol} \{\sigma\} \{\varepsilon_m\}^T dvol \tag{8}$$

where $\{\sigma\}$ and $\{\varepsilon_m\}$ are the element stresses and mechanical strains, respectively, and have the form

$$\{\sigma\}^T = \{\sigma_x, \sigma_y, \sigma_z, \tau_{yz}, \tau_{zx}, \tau_{xy}\}^T \tag{9}$$

$$\{e_m\}^T = \{\varepsilon_{mx}, \varepsilon_{my}, \varepsilon_{mz}, \gamma_{myz}, \gamma_{mzx}, \gamma_{mxy}\}^T \tag{10}$$

The work of the external loads is written as

$$W_L = -\{q\}\{f\}^T - \{\varepsilon x_0\}(F_{xo}) \tag{11}$$

where $\{q\}$ are the nodal displacements, $\{f\}$ are the applied nodal loads, and F_{xo} is the average axial force.

The total strain for a linear elastic system is written as the sum of the mechanical strain and the free thermal strain, or

$$\{\varepsilon\} = \{\varepsilon_m\} + \{\varepsilon_t\} \tag{12}$$

Rearranging to solve for the mechanical strain leads to

3

$$\{\varepsilon_m\} = \{\varepsilon\} - \{\varepsilon_t\} \tag{13}$$

Minimization of the total PE with respect to the nodal displacements requires that $\{\sigma\}$ and $\{\varepsilon_m\}$ in equ. (8) be expressed in terms of the displacements, and is accomplished by using the strain-displacement relationships of linear strain theory. These relationships are written in matrix form as

$$\{\varepsilon\} = [L]\{u\} \tag{14}$$

where $\{u\}$ is the vector of total displacements, and the matrix [L] is defined as

$$
\begin{bmatrix}
\partial/\partial x & 0 & 0 \\
0 & \partial/\partial y & 0 \\
0 & 0 & \partial/\partial z \\
0 & \partial/\partial z & \partial/\partial y \\
\partial/\partial z & 0 & \partial/\partial x \\
\partial/\partial y & \partial/\partial x & 0
\end{bmatrix}
\tag{15}
$$

Note that engineering shear strain has been used in the above relationships.

Combining equs. (1) and (7) and substituting into equ. (14) leads to an expression for strain in terms of the nodal displacements written as

$$\{\varepsilon\} = [B]\{q\} + \{\varepsilon_{xo}\} \tag{16}$$

where

$$\{q\}^T = \{u_1, v_1, w_1, \ldots u_n, v_n, w_n\} \tag{17}$$

and

$$\{\varepsilon_{xo}\}^T = \{\varepsilon_{xo}, 0, 0, 0, 0, 0\} \tag{18}$$

The matrix [B] (6 x 3n) is defined as the matrix product of [L] and a partitioned matrix involving the interpolation functions written as:

$$[B] = [L]\, [[I](N_1)\vdots\ [I](N_2)\vdots\ \ldots\ [I](N_n)] \tag{19}$$

where I is a 3x3 identity matrix. The subscript n in equs. (17-19) takes on values of 4 or 8, for the 4-node or 8-node elements, respectively. Substitution of equ. (16) into equ. (13) leads to an expression for the mechanical strains in terms of the displacements, written as

$$\{\varepsilon_m\} = [B]\{q\} + \{\varepsilon_{xo}\} - \{\varepsilon_t\} \tag{20}$$

where the free thermal strain vector is defined as

$$\{\varepsilon_t\} = \{\alpha\}(\Delta T) \tag{21}$$

4

and

$$\{\alpha\}^T = \{\alpha_x, \alpha_y, \alpha_z, \alpha_{yz}, \alpha_{zx}, \alpha_{xy}\} \tag{22}$$

and ΔT is a uniform change in temperature.

The stresses are expressed in terms of the displacements through the material constitutive equations, written in general form as

$$\{\sigma\} = [Cbar]\{\varepsilon_m\} \tag{23}$$

The specific form of [Cbar] will be discussed in the next section.

Equations (20) and (23) are substituted into equ. (8) to write an expression for the strain energy in terms of the displacements as

$$U_S = 1/2 \int_{vol} (([B]\{q\}+\{\varepsilon_{xo}\}-\{\varepsilon_t\})^T[Cbar]$$
$$([B]\{q\}+\{\varepsilon_{xo}\}-\{\varepsilon_t\}))dvol \tag{24}$$

Two cases must be considered for the minimization of the total PE (equs. (11) and (24)) with respect to the unknowns. First, for the case of a known ε_{xo}, the PE is minimized with respect to only the nodal displacements, $\{q\}$. This minimization results in an expression written as

$$[K]\{q\} = \{F\} \tag{25}$$

where

$$[K] = \int_{vol} [B]^T[Cbar][B]dvol \tag{26}$$

and

$$\{F\} = -\int_{vol} [B]^T[Cbar](\{\varepsilon_{xo}\}-\{\varepsilon_t\})dvol + \{f\} \tag{27}$$

The case of an unknown ε_{xo} requires that the PE be minimized with respect to both the nodal displacements, $\{q\}$, and ε_{xo}. This results in an expression of the general form

$$[K^*]\{q^*\} = \{F^*\} \tag{28}$$

which may be expanded and written in partitioned form as

$$\begin{bmatrix} [K] & \vdots & \{K_x\} \\ \cdots\cdots & \vdots & \cdots\cdots \\ \{K_x\}^T & \vdots & K_{xo} \end{bmatrix} \begin{Bmatrix} \{q\} \\ \cdots \\ \varepsilon_{xo} \end{Bmatrix} = \begin{Bmatrix} \{F\} \\ \cdots\cdots \\ F_{xo} - F_x \end{Bmatrix} \tag{29}$$

where [K], $\{q\}$, and $\{F\}$ have the same definitions as previously given, and

$$\{K_x\} = 1\text{'st column of } \int_{vol}[B]^T[Cbar]dvol \tag{30}$$

5

$$K_{xo} = \int_{vol}[Cbar_{11}]dvol \tag{31}$$

and

$$F_x = 1\text{'st element of } \int_{vol}[Cbar]\{\varepsilon_t\}dvol \tag{32}$$

Two steps are required to compute the elements of either [K] or [K*] and {F} or {F*}. First, the derivatives of the shape functions, N_i, with respect to the global coordinates, (y,z), must be evaluated for terms involving [B]. However, the shape functions are in terms of the local coordinates, (ξ,η). The derivative evaluation is acomplished by use of the Jacobian matrix in an expression written as

$$\begin{Bmatrix} \partial N_i/\partial y \\ \partial N_i/\partial z \end{Bmatrix} = [J]^{-1} \begin{Bmatrix} \partial N_i/\partial \xi \\ \partial N_i/\partial \eta \end{Bmatrix} \qquad i=1,n \tag{33}$$

where the Jacobian matrix is defined as

$$[J] = \begin{bmatrix} \partial N_1/\partial \xi & ... & \partial N_n/\partial \xi \\ \\ \partial N_1/\partial \eta & ... & \partial N_n/\partial \eta \end{bmatrix} \begin{bmatrix} y_1 & z_1 \\ . & . \\ . & . \\ . & . \\ y_n & z_n \end{bmatrix} \tag{34}$$

The subscript n in equs. (33) and (34) takes on values of 4 and 8 for the 4-node and 8-node elements, respectively.

The second step is to numerically evaluate the integrals in equs. (26-32). This is accomplished by first transforming the integral into the (ξ,η) coordinate system by noting that

$$dvol = |J| (d\xi)(d\eta) \tag{35}$$

where $|J|$ is the determinate of the Jacobian matrix. The limits of integration become -1 to +1 on both ξ and η. Gauss quadrature is used for the integration and all of the integrals can be written in the form

$$Integral = \sum_{i=1}^{n} \sum_{j=1}^{n} H_iH_jG(\xi_i,\eta_j) |J| \tag{36}$$

where H_i and H_j are the weight functions and $G(\xi,\eta)$ is the function to be integrated. A 2x2 Gauss quadrature (n=2) is used for both the 4-node and 8-node elements. The weight factors and coordinates of the Gauss points are given in [1].

Equations (25) and (29) represent a system of linear simultaneous equations for one element. These element equations are assembled into a global system of equations by requiring continuity of the displacements at the nodes of adjoining elements. The prescribed boundary conditions are imposed and the resulting system of simultaneous linear equations is solved for

the unknown nodal dispalcements, and if necessary ε_{xo}. The element strains and stresses are computed from the nodal displacements by using equs. (14) and (23). The strain energy density (SED) is computed from

$$SED = [\sigma_x(\varepsilon_x - \alpha_x \Delta T) + \sigma_y(\varepsilon_y - \alpha_y \Delta T) + \sigma_z(\varepsilon_z - \alpha_z \Delta T) + \sigma_{yz}(\gamma_{yz} - \alpha_{yz} \Delta T) \qquad (37)$$

$$+ \sigma_{zx}(\gamma_{zx} - \alpha_{zx} \Delta T) + \sigma_{xy}(\gamma_{xy} - \alpha_{xy} \Delta T)]/2$$

Resultant internal nodal forces [1] are computed from

$$\{P\} = \int_{vol} [B]^T \{\sigma\} dvol \qquad (38)$$

where $\{P\}$ is the vector of resultant nodal forces for an element. The integration in equ. (38) is evaluated numerically by rewriting equ. (38) in the form of equ. (36). The summation of these nodal forces in the x, y, and z directions, respectively, must equal zero for static equilibrium, and is used as a check on the validity of the solution.

It should be noted that the formulation presented above (i.e. minimization of the total potential energy) insures that [K] and [K*] will be symmetric and positive definite, after the imposition of boundary conditions which prohibit rigid-body motion. Therefore the inverse of [K] and/or [K*] exists and a solution can be found.

Material Constitutive Equations

A general form of the material constitutive relations (equ. (23)) was given in the preceding section. The specific form of these relations depends upon the relative orientation of the principal material axes with respect to the global axes. FECAP can analyze orthotropic materials (i.e. materials with 3 principal planes of symmetry) with two types of orientations relative to the global axes. These orientations are refered to as plane and cylindrical, and are shown in fig. 3. The plane orientation involves a rotation about the z or 3 axis, and the cylindrical orientation involves a rotation about the x or 1 axis. The elements of [Cbar] for each orientation are given in the Appendix A.

The angle of rotation, θ, for the cylindrical orientation varies with position in the y-z plane, and therefore varies within a given element. FECAP accounts for this variation by computing theta from the global coordinates at each of the Gauss points used for the numerical integration in equs. (26-27) and equs. (30-32). FECAP can also analyze materials having a combination of both orientations.

The matrix $\{\alpha\}$ also depends upon the relative orientation of the principal material axes with respect to the global axes. The specific forms of $\{\alpha\}$ for the two orientations described above are also given in Appendix A.

Solution Procedure

As discussed above, the solution of equs. (25) or (29) first requires the imposition of boundary conditions on the system. The three types of boundary conditions allowed in FECAP are prescribed nodal displacements, applied nodal forces, and constrained nodal displacements. The first two involve standard procedures for appropriately modifying the system stiffness

7

matrix ([K] or [K*]) and load vector ({F} or {F*}), and are well documented in texts on finite element analysis [1]. The application of constrained nodal displacements is not as well documented and requires further explanation.

Constrained nodal displacements refer to a series of nodes whose displacements are proportional to one another but whose absolute values are unknown (e.g. $v_1 = v_3 = 4v_6$). This effectively reduces the total number of unknowns in the system. The system stiffness matrix and load vector are modified by combining the contributions of the affected nodes into a single row and/or column. These modifications do not destroy the symmetry of the stiffness matrix, but can increase the bandwidth (to be discussed subsequently). The specifics of the necessary modifications are given in reference [2].

Determining the nodal displacements from equs. (25) or (29) requires finding the solution to a system of simultaneous linear algebraic equations. As stated earlier, [K] and [K*] will be symmetric and positive definite, and therefore an inverse will exist, and a solution is possible. The matrix [K] will also be banded (fig. 4) due to the assembly procedure in which a nodal displacement affects only those elements adjoining that node. Numerous solution algorithms exist for banded symmetric systems. These algorithms require storage of only the terms that lie on or above the diagonal and within the band, thus greatly reducing the amount of computer memory needed for the solution. However, [K*] is not banded due to the fact that every element contributes to ε_{xo}. This type of matrix is sometimes refered to as an "arrowhead" matrix, and is shown schematically in fig. 4. A special solution algorithm was developed for FECAP that allows a banded storage scheme to also be used for [K*]. This is accomplished by storing the terms of [K*] which contribute to ε_{xo} as a separate vector. A standard triangular ([L][U]) decomposition ([1]) is then used to solve the banded system for the unknowns, with terms from this separate vector used when needed.

COMPUTER IMPLEMENTATION

System Requirements

FECAP uses the finite element formulation presented in the preceding section. FECAP is written in Hewlett Packard BASIC 3.0, and can run on any Hewlett Packard 9000 Series 200 microcomputer with the BASIC 3.0 operating system. Although no special hardware or software enhancements are necessary, certain enhancements greatly increase program capability and performance, and are strongly recommended. The suggested system configuration is shown in Table 1.

The suggested size RAM in Table 1. is 2 Mbytes, and controls the size of problems that can be solved with FECAP. An in-core solution scheme is used in the program and the size of the stiffness matrix, [K], is the major factor in determining the size of the problem. The amount of memory needed to store [K] can be computed from

$$[K] \text{ Memory(bytes)} = \big((n)(3)(SBW)\big)8 \qquad (39)$$

where n is the number of nodes, 3 is the number of variables per node, SBW is the semi-bandwidth (fig. 4), and 8 is the number of bytes per word. For example, a medium size problem with 300 nodes and a semi-bandwidth of 75 requires 540 Kbytes. Loading the BASIC 3.0 operating system, and loading and compiling FECAP requires an additional 720 Kbytes. Therefore, the total RAM required for execution of this example problem is 1.26 Mbytes. The suggested RAM size of 2 Mbytes will provide enough memory for problems of

8

this size and larger. It should be noted that various factors can affect the semi-bandwidth, such as node numbering and constrained nodal displacement boundary conditions, and thus affect the memory requirements for a given problem.

Pre- and Post-Processing

Pre-processing for a finite element program involves generating the global nodal coordinate data and the element connectivity data for the finite element mesh. FECAP is written such that this data is read from a file that the user has previously generated. This file can be produced from relatively simple user written programs, or from one of the many commercially available mesh generation programs. Data files from commercially available programs will have to be modified so that the data is in the proper format required for FECAP. The required format is given in Appendix B. A very simple mesh generation program was written for the example problems presented in this report.

Post-processing in finite element analysis involves computing secondary quantities such as stresses and strains, and then displaying these quantities in tabular and/or graphic form. FECAP computes the nodal stresses and strains for each element from the nodal displacements, and displays these results in tabular form. FECAP also has the capability to store these values in a data file. The format of this data file is given in Appendix B. This data can then be used in a contour plotting program for graphical display of results. An interactive contour plotting program that is easily modified to work with FECAP can be found in reference [3].

User's Guide

FECAP runs interactively, and thus requires responses from the user during execution. A description of the computer prompts and appropriate user replies are given below. This description assumes a working knowledge of the HP BASIC 3.0 operating system.

Load file FECAP (compile if compiler is available) and begin execution by pressing the RUN key. FECAP will begin execution with the data input softkey menu appearing at the bottom of the screen and a computer prompt that reads:

DATA INPUT - Select a key

The user responds by pressing on of the seven defined softkeys. Softkeys K0 through K4 must be selected at least once and in any order, but must be selected before K5 or K6. Softkeys K0 through K4 may be selected more than once if necessary to correct data. The selection of softkey K5 is optional, but must occur before the selection of K6. Softkey K6 must be the last softkey selected. A description of each of the softkeys is given below. All user responses are entered by pressing the ENTER key. Multiple entries in the user response are seperated by commas.

Title (K0):
Computer prompt -   Input Problem Title, (<= 80 char.)

User response -   Problem title, must be less than or equal to 80 characters in length.

Material Props (K1):
Computer prompt -   Input Number of Materials

9

User response -	Number of different materials, must be consistent with mesh data file.
Computer prompt -	Is a Cylindrical Material Orientation Required? (Y/N)
User response -	Is a cylindrical material orientation required for this problem, yes (Y) or no (N). Note, a cylindrical material orientation depends upon the mesh geometry and effects every element. Plane orientations can be used in conjunction with the cylindrical orientation.
Computer prompt -	For Material No. n (top of screen), Input Mat. Prop's. E1,E2,E3,G23,G13,G12,NU23,NU13,NU12
User response -	Elastic orthotropic Young's moduli, shear moduli, and Poisson's ratios in principal material coordinates for n'th material.
Computer prompt -	For Material No. n (top of screen), Input Thermal Prop's. ALPHA1,ALPHA2,ALPHA3
User response -	Orthotropic coefficients of thermal expansion in principal material coordinates for n'th material.
Computer prompt -	For Material No. n (top of screen), Input Number of Angles
User response -	Number of different plane orientation angles for n'th material, must be consistent with mesh data file.
Computer prompt -	For material No. n (top of screen), Input Angles in degrees (one at a time followed by ENTER key)
User response -	Plane orientation angles for n'th material.

Note: The above four prompts are repeated n times, where n is the
number of materials

Mesh Data (K2):

Computer prompt -	Input Mesh Data File Name and MSUS (Ex.Fn,"700,0")
User response -	Mesh data file name and mass storage unit specifier (MSUS). The proper format for the mesh data file is given in Appendix B.
Computer prompt -	Input the Maximum Allowable Semi-Bandwidth (Must be <= Array dimensions)
User response -	Maximum allowable semi-bandwidth, must be consistent with array dimensions in main program.

Boundary Data (K3):
The boundary condition input softkey menu is displayed with the following prompt

BOUNDARY CONDITION DATA - Select a Key

These softkeys control how the boundary condition data is entered. Softkey K0 is for input from a previously defined file. Softkey K1 is for keyboard input. A description of each is given below.

File (K0):

Computer prompt -	Input Boundary Condition File Name and MSUS (Ex. Fn,"700,0")
User response -	Boundary condition file name and mass storage unit specifier (MSUS). The proper format for the boundary condition data file is given in Appendix B.

1 0

Keyboard (K1):
The keyboard boundary condition softkey menu is displayed with the following prompt

NODAL BOUNDARY CONDITIONS - Select a Key

These softkeys control the keyboard input of the allowable types of nodal boundary conditions. Softkeys K0 through K2 may be selected in any order, and more than once if corrections are necessary. Softkey K3 is used to store the current boundary conditions in memory to a file. Softkey K4 returns the program to the data input softkey menu after completion of boundary condition input. A description of each softkey is given below.

Specified Disp (K0):

Computer prompt -	Specified Nodal Displacements: Input Direction of Specified Nodal Displacement (1-X,2-Y,3-Z) (Input 0 for End of Specified Nodal Displacements)
User response -	Global direction of specified nodal displacement, 1 for x direction, 2 for y direction, and 3 for z direction. A response of 0 is used to end specified nodal displacement input.
Computer prompt -	Input Beg'ning node #, Ending node #, Increment, and Specified Value
User response -	Beginning node number, ending node number, node number increment, and specified nodal value for the direction specified in the previous prompt.

Note: The above two prompts are repeated until a 0 is input for the direction at which point the program returns to the boundary condition softkey menu.

Applied Force (K1):

Computer prompt -	Applied Nodal Forces: Input Direction of Applied Nodal Force (1-X,2-Y,3-Z) (Input 0 for End of Applied Nodal Forces)
User response -	Global direction of applied nodal forces, 1 for x direction, 2 for y direction, and 3 for z direction. A response of 0 is used to end applied nodal force input.
Computer prompt -	Input Beg'ning node #, Ending node #, Increment, and Specified Value
User response -	Beginning node number, ending node number, node number increment, and specified nodal force for direction specified in previous prompt.

Note: The above two prompts are repeated until a 0 is input for the direction at which point the program returns to the boundary condition softkey menu.

Constraint Eqs (K2):

Computer prompt -	Constrained Nodal Displacements: Input Number of Different Constraint Conditions (Input 0 for No Constraint Conditions)
User reponse -	Number of different nodal constraint conditions. A response of 0 is used when there are no constraint conditions.
Computer prompt -	Input Direction of Constrained Nodal Displacement (1-X,2-Y,3-Z) for Constraint n
User response -	Direction of constrained nodal displacement for n'th constraint condition, 1 for x direction, 2 for y direction, and 3 for z direction.
Computer prompt -	Input Beg'ning node #, Ending node #, Increment, and Scale Factor

User response -	Beginning node number, ending node number, node number increment, and constraint scale factor (i.e ratio of displacements) for the direction specified in the previous prompt.
Computer prompt -	More nodes for this direction ? (Y/N)
User response -	Are there additional constrained nodes for the direction specified above, yes (Y) or no (N). If the response is yes the previous prompt is repeated.
Computer prompt -	More nodes for this constraint condition ? (Y/N)
User response -	Are there additional node directions for the n'th constraint condition, yes (Y) or no (N). If the response is yes then execution returns to the constraint direction prompt.

Note: The above four prompts are repeated n times, where n is the number of constraint conditions, after which the program returns to the boundary condition softkey menu.

Save Data (K3):
Computer prompt -	Input Boundary Condition File Name and MSUS (Ex. Fn,"700,0")
User response -	Boundary condition file name and mass storage unit specifier (MSUS) for storing current boundary condition data.

Exit (K4):
This softkey returns the program to the data input softkey menu. The description of the data input softkey menu is continued below.

Global Loads (K4):
The global load softkey menu is displayed with the following prompt

GLOBAL LOAD CONDITIONS - Select a Key

Softkeys K0 and K1 define the type of global loading for the problem, and only one of these softkeys should be selected. Softkey K2 returns the program to the data input menu. A description of each softkey is given below.

Axial Strain (K0):
Computer prompt -	Input Uniform Axial Strain and Temperature Change
User response -	Uniform applied axial strain and uniform change in temperature.

Axial Force (K1):
Computer prompt -	Input Uniform Axial Force and Temperature Change
User response -	Uniform applied axial force and uniform change in temperature.

Exit (K2):
This softkey returns the program to the data input softkey menu, the description of which is continued below.

Print Input (K5):
Computer prompt -	Input Printer Code for Output - CRT(1), PRINTER(701)
User response -	Printer code for output results, 1 for CRT display and 701 for printer at address 701.
Computer prompt -	Store Results on Output File ? (Y/N)

1 2

User response -	Store results in an output file for later use with post-processor program, yes (Y) or no (N). If response is no the following prompt is skipped.
Computer prompt -	Input File Name and MSUS for Output Results (Ex. Fn,"700,0")
User response -	File name and mass storage unit specifier (MSUS) for output results.
Computer prompt -	Print Nodal Coordinates ? (Y/N)
User response -	Print the global y and z coordinates of the nodes, yes (Y) or no (N).
Computer prompt -	Print Element Connectivity Data ? (Y/N)
User response -	Print the element connectivity data (i.e node numbers of each element), yes (Y) or no (N).
Computer prompt -	Print Boundary Condition Data ? (Y/N)
User response -	Print the nodal boundary condition data, yes (Y) or no (N).

After the above prompt the following message appears on the screen

Press Execute Softkey to Begin Execution

Pressing the Execute softkey in the data input menu starts the finite element calculations. During execution the following message appears flashing on the screen

***** PROGRAM EXECUTING, DO NOT DISTURB *****

Execution can take from minutes to hours depending upon the size of the problem. The softkey menu returns to the default (pre-program) definitions. The remaining prompts occur after completion of the finite element calculations, and control the printing of output results.

Computer prompt -	Print Nodal Displacements ? (Y/N)
User response -	Print the nodal u, v, and w displacements, yes (Y) or no (N). Note the u displacement is not the total displacement, but only that part which is a function of y and z (i.e. $U(y,z)$).
Computer prompt -	Print Element Stresses and Strains ? (Y/N)
User response -	Print the element stresses, strains, strain energy density, and resultant nodal forces, yes (Y) or no (N). These quantities are computed and printed at each node of each element.

The last line printed by FECAP for a normal execution is

***** Problem Finished, Press RUN key for a New Problem *****

SAMPLE PROBLEM

The problem selected for demonstrating the proper program input and output is a [0/90] symmetric graphite/epoxy plate exposed to a unifrom change in temperature. The region modeled by FECAP is shown in fig. 5. The proper user input is given below. User responses are followed by (ENTER) unless otherwise specified.

DATA INPUT PHASE - Select a Key
(Press softkey K0)
Input Problem Title (<= 80 char.)

13

SAMPLE PROBLEM - [0/90] SYMMETRIC LAMINATE WITH THERMAL LOAD
(ENTER)
(Press softkey K1)
Input Number of Materials
1 (ENTER)
Is a Cylindrical Material Orientation Required ? (Y/N)
N (ENTER)
For Material No. 1 (Displayed at top of screen)
Input Mat Prop's. E1,E2,E3,G23,G13,G12,NU23,NU13,NU12
19.26E6,1.56E6,1.56E6,.487E6,.82E6,.82E6,.49,.238,.238 (ENTER)
Input Thermal Prop's. ALPHA1,ALPHA2,ALPHA3
-.06E-6,15E-6,15E-6 (ENTER)
Input Number of Angles
2 (ENTER)
Input Angles in degrees (one at a time)
90 (ENTER)
Input Angles in degrees (one at a time)
0 (ENTER)
DATA INPUT PHASE - Select a Key
(Press softkey K2)
Input Mesh Data File Name and MSUS (Ex. Fn,"700,0")
MESH_4E,"700,0"
Input the Maximum Allowable Semi_bandwidth (Must be <= Array Dimensions)
25 (ENTER)
DATA INPUT PHASE - Select a Key
(Press softkey K3)
BOUNDARY CONDITION DATA - Select a Key
(Press softkey K1)
NODAL BOUNDARY CONDITIONS - Select a Key
(Press softkey K0)
Specified Nodal Displacements:
Input Direction of Specified Nodal Displacement(1-X,2-Y,3-Z)
(Input 0 for End of Specified Nodal Displacements)
1 (ENTER)
Input Beg'ning node #, Ending node #, Increment, and Specified Value
1,1,1,0 (ENTER)
Specified Nodal Displacements:
Input Direction of Specified Nodal Displacement(1-X,2-Y,3-Z)
(Input 0 for End of Specified Nodal Displacements)
2 (ENTER)
Input Beg'ning node #, Ending node #, Increment, and Specified Value
1,7,3,0 (ENTER)
Specified Nodal Displacements:
Input Direction of Specified Nodal Displacement(1-X,2-Y,3-Z)
(Input 0 for End of Specified Nodal Displacements)
3 (ENTER)
Input Beg'ning node #, Ending node #, Increment, and Specified Value
1,3,1,0 (ENTER)
Specified Nodal Displacements:
Input Direction of Specified Nodal Displacement(1-X,2-Y,3-Z)
(Input 0 for End of Specified Nodal Displacements)
0 (ENTER)
NODAL BOUNDARY CONDITIONS - Select a Key
(Press softkey K2)
Constrained Nodal Displacements:

1 4

Input Number of Different Constraint Conditions
(Input 0 for No Constraint Conditions)
1 (ENTER)
Input Direction of Constrained Nodal Displacement(1-X,2-Y,3-Z) for Constraint 1
2 (ENTER)
Input Beg'ning node #, Ending node #, Increment, and Scale Factor
3,9,3,1 (ENTER)
More Nodes for this Direction ? (Y/N)
N (ENTER)
More Nodes for this Constraint Condition ? (Y/N)
N (ENTER)
NODAL BOUNDARY CONDITIONS - Select a Key
(Press softkey K4)
DATA INPUT PHASE - Select a Key
(Press softkey K4)
GLOBAL LOAD CONDITIONS - Select a Key
(Press softkey K1)
Input Average Axial Force and Temperature Change
0,1 (ENTER)
GLOBAL LOAD CONDITIONS - Select a Key
(Press softkey K3)
DATA INPUT PHASE - Select a Key
(Press softkey K5)
Input Printer Code for Output - CRT(1), PRINTER(701)
701 (ENTER)
Store Results on Output File ? (Y/N)
N (ENTER)
Print Nodal Coordinates ? (Y/N)
Y (ENTER)
Print Element Connectivity Data ? (Y/N)
Y (ENTER)
Print Boundary Condition Data ? (Y/N)
Y (ENTER)
Press Execute Softkey to Begin Execution
(Press softkey K6)
***** PROGRAM EXECUTING, DO NOT DISTURB *****
Print Nodal Displacements ? (Y/N)
Y (ENTER)
Print Element Stresses and Strains ? (Y/N)
Y (ENTER)

The output from the sample problem is given in Appendix C. It should be noted that the solution for this problem can be obtained from classical laminated plate theory (LPT) [5]. The stresses and strains predicted from FECAP are within 0.01 percent of the values predicted from LPT.

REFERENCES

1. Zienkiewicz, O. C.: The Finite Element Method. McGraw-Hill Book Company, New York, NY, 1977.

2. Adams, D. F.; and Crane, D. A.: Finite Element Micromechanical Analysis of a Unidirectional Composite Including Longitudinal Shear Loading. Computers and Structures, Vol. 18, No. 6, 1984, pp. 1153-1165.

3. Yeo, M. F.: An Interactive Contour Plotting Program. Engineering Computations, Vol. 1, No. 3, Sept. 1984, pp. 273-279.

4. Frederick, D.; and Chang, T. S.: Continuum Mechanics. Scientific Publishers, Inc., Boston, MA, 1972.

5. Jones, R. M.: Mechanics of Composite Materials. Scripta Book Company, Washington,   D.C., 1975.

APPENDIX A

Elements of [Cbar] Matrix

The specific form of the [Cbar] matrix is obtained using standard 4'th order tensor transformations [4]. In contracted form [Cbar] can be written as a 6x6 symmetric matrix. The elements of [Cbar] for the plane orientation (fig. 3) are

$$Cbar_{11} = m^4 C_{11} + 2m^2 n^2 (C_{12} + 2C_{66}) + n^4 C_{22}$$
$$Cbar_{12} = m^2 n^2 (C_{11} + C_{22} - 4C_{66}) + (m^4 + n^4)C_{12}$$
$$Cbar_{13} = m^2 C_{13} + n^2 C_{23}$$
$$Cbar_{16} = mn(m^2(C_{11} - C_{12} - 2C_{66}) + n^2(C_{12} - C_{22} + 2C_{66}))$$
$$Cbar_{22} = n^4 C_{11} + 2m^2 n^2 (C_{12} + 2C_{66}) + m^4 C_{22}$$
$$Cbar_{23} = n^2 C_{13} + m^2 C_{23}$$
$$Cbar_{26} = mn(n^2(C_{11} - C_{12} - 2C_{66}) + m^2(C_{12} - C_{22} + 2C_{66}))$$
$$Cbar_{33} = C_{33}$$
$$Cbar_{36} = mn(C_{13} - C_{23})$$
$$Cbar_{44} = m^2 C_{44} + n^2 C_{55}$$
$$Cbar_{45} = mn(C_{55} - C_{44})$$
$$Cbar_{55} = n^2 C_{44} + m^2 C_{55}$$
$$Cbar_{66} = m^2 n^2 (C_{11} - 2C_{12} + C_{22}) + (m^2-n^2)(m^2-n^2)C_{66}$$

(A.1)

where all other $Cbar_{ij} = 0$, and $m = \cos(\theta)$ and $n = \sin(\theta)$. Note that $Cbar_{ij} = Cbar_{ji}$ for i and j = 1,6. The elements of [C], the stiffness matrix in the principal material coordinate system, for an orthotropic material are given by [5]

$$C_{11} = \frac{1 - v_{23}v_{32}}{E_2 E_3 Del} \quad C_{12} = \frac{v_{12} + v_{32}v_{13}}{E_1 E_3 Del} \quad C_{13} = \frac{v_{13} + v_{12}v_{23}}{E_1 E_2 Del}$$

$$C_{22} = \frac{1 - v_{13}v_{31}}{E_1 E_3 Del} \quad C_{23} = \frac{v_{23} + v_{21}v_{13}}{E_1 E_2 Del} \quad C_{33} = \frac{1 - v_{12}v_{21}}{E_1 E_2 Del}$$

(A.2)

$$C_{44} = G_{23} \quad C_{55} = G_{31} \quad C_{16} = G_{12}$$

all other $C_{ij} = 0$

where

$$Del = \frac{1 - v_{12}v_{21} - v_{23}v_{32} - v_{13}v_{31} - 2v_{21}v_{32}v_{13}}{E_1 E_2 E_3}$$

(A.3)

17

and $C_{ij} = C_{ji}$. E_{ij}, G_{ij}, and v_{ij} are the orthotropic Young's moduli, shear moduli and Poisson's ratios, respectively, in the principal material coordinates.

Note that for an orthotropic material there is a relationship between E and v which can be written as

$$v_{ij}/E_i = v_{ji}/E_j \tag{A.4}$$

The elements of [Cbar] for the cylindrical orientation (fig. 3) are written as

$$Cbar_{11} = C'_{11}$$
$$Cbar_{12} = m^2 C'_{12} + n^2 C'_{13}$$
$$Cbar_{13} = n^2 C'_{12} + m^2 C'_{13}$$
$$Cbar_{14} = mn(C'_{12} - C'_{13})$$
$$Cbar_{15} = nC'_{16}$$
$$Cbar_{16} = mC'_{16} \tag{A.5}$$
$$Cbar_{22} = m^4 C'_{22} + m^2 n^2 (2C'_{23} + 4C'_{44}) + n^4 C'_{33}$$
$$Cbar_{23} = m^2 n^2 (C'_{22} - 4C'_{44} + C_{33}) + (m^4 + n^4) C'_{23}$$
$$Cbar_{24} = m^3 n (C'_{22} - C'_{23} - 2C'_{44}) + mn^3 (C'_{23} + 2C'_{44} - C'_{33})$$
$$Cbar_{25} = m^2 n (C'_{26} - 2C'_{45}) + n^3 C'_{36}$$
$$Cbar_{26} = m^3 C'_{26} + mn^2 (2C'_{45} + C'_{36})$$
$$Cbar_{33} = m^4 C'_{33} + m^2 n^2 (2C'_{23} + 4C'_{44}) + n^4 C'_{22}$$
$$Cbar_{34} = m^3 n (C'_{23} + 2C'_{44} - C'_{33}) + mn^3 (C'_{22} - C'_{23} - 2C'_{44})$$
$$Cbar_{35} = m^2 n (C'_{36} + 2C'_{45}) + n^3 C'_{26}$$
$$Cbar_{36} = m^3 C'_{36} + mn^2 (C'_{26} - 2C'_{45})$$
$$Cbar_{44} = m^4 C'_{44} + m^2 n^2 (C'_{22} - 2C'_{23} - 2C'_{44} + C'_{33}) + n^4 C'_{44}$$
$$Cbar_{45} = m^3 C'_{45} + mn^2 (C'_{26} - C'_{45} - C'_{36})$$
$$Cbar_{46} = m^2 n (C'_{26} - C'_{45} - C'_{36}) + n^3 C'_{45}$$
$$Cbar_{55} = m^2 C'_{55} + n^2 C'_{66}$$
$$Cbar_{56} = mn(C'_{66} - C'_{55})$$
$$Cbar_{66} = m^2 C'_{66} + n^2 C'_{55}$$

where all other $Cbar_{ij} = 0$, and m and n have the same definition as previously given. The elements of [C'] are either the elements of [Cbar] given by equ. (A.1) when a plane orientation is used in combination with a cylindrical orientation, or the elements of [C] when the cylindrical orientation is used alone. For the later case certain elements of [C'] used in equ. (A.5) are zero.

The elements of $\{\alpha\}$ are obtained using a transformation that is consistent with the definition of engineering shear strain, as apposed to tensor shear strain. The plane orientation results in an $\{\alpha\}$ with elements written as

18

$$\alpha_x = m^2\alpha_1 + n^2\alpha_2$$
$$\alpha_y = n^2\alpha_1 + m^2\alpha_2 \qquad\qquad (A.6)$$
$$\alpha_z = \alpha_3$$
$$\alpha_{xy} = 2mn(\alpha_1 - \alpha_2)$$

where all other $\alpha = 0$, and m and n have the same definitions as previously given.

The elements of $\{\alpha\}$ for the cylindrical orientation are written as

$$\alpha_x = \alpha'_x$$
$$\alpha_y = m^2\alpha'_y + n^2\alpha'_z$$
$$\alpha_z = n^2\alpha'_y + m^2\alpha'_z \qquad\qquad (A.7)$$
$$\alpha_{yz} = 2mn(\alpha'_y - \alpha'_z)$$
$$\alpha_{zx} = n\alpha'_{xy}$$
$$\alpha_{xy} = m\alpha'_{xy}$$

where $\{\alpha'\}$ is equal to $\{\alpha\}$ from equ. (A.6) when a combination of plane and cylindrical orientations are used, and $\{\alpha'\}$ is defined as

$$\{\alpha'_x, \alpha'_y, \alpha'_z, \alpha'_{yz}, \alpha'_{zx}, \alpha'_{xy}\} = \{\alpha_1,\alpha_2,\alpha_3,0,0,0\} \qquad\qquad (A.9)$$

when only a cylindrical orientation is used.

Format for Mesh Data File

The proper format for creating the mesh data file used in FECAP is given below. The statements are written in HP Basic 3.0.

```
CREATE BDAT Fn_meshdat$&":,"&Msus_meshdat$,1,
2*(Npe+2)*Nem+8*2*Nods+6
ASSIGN @File1 TO Fn_meshdat$&":,"&Msus_meshdat$
OUTPUT @FILE1;Nem,Nods,Npe
FOR I=1 TO Nods
OUTPUT @File1;Y(I),Z(I)
NEXT I
FOR I=1 TO Nem
IF Npe=4 THEN OUTPUT @File1;Nod(I,1),Nod(I,2),Nod(I,3),
Nod(I,4),Imat(I),Iang(I)
IF Npe=8 THEN OUTPUT @File1,Nod(I,1),Nod(I,2),Nod(I,3),
Nod(I,4),Nod(I,5),Nod(I,6),Nod(I,7),Nod(I,8),Imat(I),Iang(I)
NEXT I
ASSIGN @File1 TO *
```

The variables in the above statements have the following definitions

Fn_meshdat$	= File name of mesh data
Msus_meshdat$	= Mass storage unit specifier
Nods	= Number of nodes
Nem	= Number of elements
Npe	= Number of nodes per element, 4 or 8
Y(I),Z(I)	= Global nodal coordinates
Nod(I,J)	= Node numbers of I'th element, J=1,4 or 1,8 (Must be in consecutive CCW order around element)
Imat(I)	= Material number of I'th element
Iang(I)	= Angle number of material of I'th element

Format for Boundary Condition Data File

The boundary condition data file is created with the following format

```
Mxnsl=MAX(Nsl(*))
CREATE BDAT Fn_bcdat$&":,"&Msus_bcdat$,1,8*(3*Nsdf+3*Nsbf+
Ncd*3*Mxnsl+Ncd+3)
ASSIGN @File3 TO Fn_bcdat$&":,"&Msus_bcdat$
OUTPUT @File3;Nsdf,Nsbf,Ncd
FOR I=1 TO Nsdf
OUTPUT @File3;Ndd(I),Vbdf(I),Ibdf(I)
NEXT I
FOR I=1 TO Nsbf
OUTPUT @File3;Ndb(I),Vbsf(I),Ibsf(I)
NEXT I
FOR I=1 TO Ncd
OUTPUT @File3;Nsl(I)
```

```
FOR J=1 TO Nsl(I)
OUTPUT @File3;Nds(I,J),Ibcn(I,J),Vbcn(I,J)
NEXT J
NEXT I
ASSIGN @File3 TO *
```

The variables in the above statements have the following definitions

Fn_bcdat$	= File name of boundary condition data
Msus_bcdat$	= Mass storage unit specifier
Nsdf	= Number of specified nodal displacements
Nsbf	= Number of applied nodal forces
Ncd	= Number of nodal constraint conditions
Ndd(I)	= Node numbers of specified nodal displacements
Vbdf(I)	= Magnitudes of specified nodal displacements
Ibdf(I)	= Directions of specified nodal displacements
Ndb(I)	= Node numbers of applied nodal forces
Vbsf(I)	= Magnitudes of applied nodal forces
Ibsf(I)	= Directions of applied nodal forces
Nsl(I)	= Number of nodes for each constraint condition
Nds(I,J)	= Node numbers for each constraint condition
Ibcn(I,J)	= Directions for each constraint condition
Vbcn(I,J)	= Scale factors for each constraint condition

Format for Output Data File

The output data file is created with the following format

```
Max_len=INT((((Nods*3+Nem*(Npe*13+Npe*3)+3)*8+96)/256)
CREATE BDAT Fn_outdat$&":,"&Msus_outdat$,Max_len+1,256
ASSIGN @File2 TO Fn_outdat$&":,"&Msus_outdat$
OUTPUT @File2;T$,Nem,Nods,Npe
FOR I=1 TO Nods
L=3*I-2
M=3*I-1
N=3*I
OUTPUT @File2;Gf(L),Gf(M),Gf(N)
NEXT I
FOR N=1 TO Nem
FOR K=1 TO Npe
OUTPUT @File2;sig(1),sig(2),sig(3),sig(4),sig(5),sig(6)
OUTPUT @File2;Eps(1),Eps(2),Eps(3),Eps(4),Eps(5),Eps(6),Energy
NEXT K
OUTPUT @File2;Fxyz(*)
NEXT N
OUTPUT @File2;Fxtot,Fytot,Fztot
ASSIGN @File2 to *
```

The variables in the above statements have the following definitions

Fn_outdat$	= File name of output data
Msus_outdat$	= Mass storage unit specifier
T$	= Problem title
Gf(*)	= Nodal displacements
sig(*)	= Element nodal stresses

Eps(*)   = Element nodal strains
Energy   = Element nodal strain energy density
Fxyz(*)   = Element nodal resultant forces
Fxtot   = Summation of resultant x-direction nodal forces
Fytot   = Summation of resultant y-direction nodal forces
Fztot   = Summation of resultant z-direction nodal forces

Those variables not defined above have the same definitions as given previously in this appendix.

APPENDIX C


```
***************************************************************************

                            FECAP PROGRAM
                27 Jul 1988              14:51:12

***************************************************************************

SAMPLE PROBLEM - [0/90] SYMMETRIC LAMINATE WITH THERMAL LOAD

***************************************************************************
                              INPUT DATA
***************************************************************************


    CONTROL INFORMATION:

        NUMBER OF ELEMENTS...........................   4
        NUMBER OF NODES..............................   9
        NUMBER OF NODES PER ELEMENT..................   4
        NUMBER OF MATERIALS..........................   1
        MESH DATA FILE NAME..........................MESH_4E
        BOUNDARY CONDITION DATA FILE NAME........
        OUTPUT RESULTS FILE NAME.................


    MATERIAL PROPERTIES:

        MATERIAL NO. 1
        E1.......................................... 1.920E+07
        E2.......................................... 1.560E+06
        E3.......................................... 1.560E+06
        G23......................................... 4.870E+05
        G13......................................... 8.200E+05
        G12......................................... 8.200E+05
        NU23........................................ 4.900E-01
        NU13........................................ 2.380E-01
        NU12........................................ 2.380E-01
        ALPHA1......................................-6.000E-08
        ALPHA2...................................... 1.500E-05
        ALPHA3...................................... 1.500E-05


        ANGLE INFORMATION FOR MAT. NO. 1
        ANGLE NO.         THETA(DEG.)
            1               90.00
            2                0.00


    NODAL COORDINATES:

        NODE NO.      Y-COORDINATE        Z-COORDINATE
            1          0.000E+00           0.000E+00
            2          5.000E-01           0.000E+00
            3          1.000E+00           0.000E+00
            4          0.000E+00           5.000E-01
            5          5.000E-01           5.000E-01
            6          1.000E+00           5.000E-01
```


23

```
                7          0.000E+00             1.000E+00
                8          5.000E-01             1.000E+00
                9          1.000E+00             1.000E+00


ELEMENT CONNECTIVITY DATA:

   ELEMENT NO.              NODE NUMBERS              MAT. NO.      ANGLE NO.
        1                   1   2   5   4                1              1
        2                   2   3   6   5                1              1
        3                   4   5   8   7                1              2
        4                   5   6   9   8                1              2


SPECIFIED DEGREES OF FREEDOM:

   NODE NUMBER        DIRECTION        VALUE
        1                 1            0.000E+00
        1                 2            0.000E+00
        4                 2            0.000E+00
        7                 2            0.000E+00
        1                 3            0.000E+00
        2                 3            0.000E+00
        3                 3            0.000E+00


CONSTRAINED NODAL DISPLACEMENTS:

   NODE NUMBER        DIRECTION        SCALE FACTOR
        3                 2            1.000E+00
        6                 2            1.000E+00
        9                 2            1.000E+00

BANDWIDTH IS  21

UNIFORM APPLIED AXIAL FORCE................. 0.000E+00

UNIFORM APPLIED DELTA TEMP.................. 1.000E+00


UNIFORM COMPUTED AXIAL STRAIN .............. 1.293E-06


EXECUTION TIME IS 5.484E+01 SECONDS
```

****
                                OUTPUT RESULTS
****

```
NODAL DISPLACEMENTS:

 NODE NO.     U-DISPLACEMENT      V-DISPLACEMENT      W-DISPLACEMENT
    1           0.0000E+00          0.0000E+00          0.0000E+00
    2          -1.5165E-21          6.4632E-07          0.0000E+00
    3          -3.0330E-21          1.2926E-06          0.0000E+00
    4          -7.9810E-22          0.0000E+00          1.0665E-05
    5          -1.5165E-21          6.4632E-07          1.0665E-05
    6          -2.2349E-21          1.2926E-06          1.0665E-05
    7          -1.3369E-21          0.0000E+00          2.1329E-05
    8          -1.5165E-21          6.4632E-07          2.1329E-05
    9          -1.6961E-21          1.2926E-06          2.1329E-05
```

ELEMENT STRESSES, STRAINS, STRAIN ENERGY, AND RESULTANT NODAL FORCES:

```
ELEMENT NO.   1.      GLOBAL NODES -      1        2        5        4
                      Y COORD         0.00E+00  5.00E-01  5.00E-01  0.00E+00
                      Z COORD         0.00E+00  0.00E+00  5.00E-01  5.00E-01

  NODE      EPSXX/SIGXX              EPSYY/SIGYY              EPSZZ/SIGZZ
            GAMMAYZ/TAUYZ           GAMMAZX.TAUZX           GAMMAXY/TAUXY
   1    1.293E-06/-2.098E+01    1.293E-06/ 2.098E+01    2.133E-05/ 2.665E-14
        0.000E+00/-3.255E-32   -1.596E-21/-7.773E-16   -3.033E-21/ 8.194E-17
             SED = 1.580E-04
   2    1.293E-06/-2.098E+01    1.293E-06/ 2.098E+01    2.133E-05/ 5.329E-15
        0.000E+00/-7.670E-48   -3.762E-37/-1.832E-31   -3.033E-21/ 8.194E-17
             SED = 1.580E-04
   5    1.293E-06/-2.098E+01    1.293E-06/ 2.098E+01    2.133E-05/ 5.329E-15
       -1.016E-20/-8.335E-15   -3.762E-37/-3.904E-31   -1.437E-21/ 1.391E-15
             SED = 1.580E-04
   4    1.293E-06/-2.098E+01    1.293E-06/ 2.098E+01    2.133E-05/ 2.665E-14
       -1.016E-20/-8.335E-15   -1.596E-21/-7.773E-16   -1.437E-21/ 1.391E-15
             SED = 1.580E-04

  RESULTANT NODAL FORCES -
  NODE      FX          FY          FZ
     1   4.129E-31  -5.244E+00  -5.059E-15
     2   1.943E-16   5.244E+00  -4.711E-15
     5   1.739E-16   5.244E+00   2.106E-15
     4  -3.682E-16  -5.244E+00   7.664E-15

ELEMENT NO.   2.      GLOBAL NODES -      2        3        6        5
                      Y COORD         5.00E-01  1.00E+00  1.00E+00  5.00E-01
                      Z COORD         0.00E+00  0.00E+00  5.00E-01  5.00E-01

  NODE      EPSXX/SIGXX              EPSYY/SIGYY              EPSZZ/SIGZZ
            GAMMAYZ/TAUYZ           GAMMAZX.TAUZX           GAMMAXY/TAUXY
   2    1.293E-06/-2.098E+01    1.293E-06/ 2.098E+01    2.133E-05/ 5.329E-15
        0.000E+00/-7.670E-48   -3.762E-37/-1.832E-31   -3.033E-21/ 8.194E-17
             SED = 1.580E-04
   3    1.293E-06/-2.098E+01    1.293E-06/ 2.098E+01    2.133E-05/ 1.954E-14
        0.000E+00/ 3.255E-32    1.596E-21/ 7.773E-16   -3.033E-21/ 8.194E-17
             SED = 1.580E-04
   6    1.293E-06/-2.098E+01    1.293E-06/ 2.098E+01    2.133E-05/ 1.954E-14
        6.776E-21/ 5.557E-15    1.596E-21/ 7.773E-16   -1.437E-21/ 1.391E-15
             SED = 1.580E-04
   5    1.293E-06/-2.098E+01    1.293E-06/ 2.098E+01    2.133E-05/ 5.329E-15
        6.776E-21/ 5.557E-15   -3.762E-37/-4.502E-32   -1.437E-21/ 1.391E-15
             SED = 1.580E-04

  RESULTANT NODAL FORCES -
  NODE      FX          FY          FZ
     2  -1.943E-16  -5.244E+00  -3.163E-15
     3   1.726E-31   5.244E+00  -3.054E-15
     6   3.682E-16   5.244E+00   4.790E-15
     5  -1.739E-16  -5.244E+00   1.427E-15

ELEMENT NO.   3.      GLOBAL NODES -      4        5        8        7
                      Y COORD         0.00E+00  5.00E-01  5.00E-01  0.00E+00
                      Z COORD         5.00E-01  5.00E-01  1.00E+00  1.00E+00

  NODE      EPSXX/SIGXX              EPSYY/SIGYY              EPSZZ/SIGZZ
            GAMMAYZ/TAUYZ           GAMMAZX.TAUZX           GAMMAXY/TAUXY
   4    1.293E-06/ 2.098E+01    1.293E-06/-2.098E+01    2.133E-05/ 2.665E-14
       -1.016E-20/-4.950E-15   -1.078E-21/-3.837E-16   -1.437E-21/-1.178E-15
             SED = 1.580E-04
   5    1.293E-06/ 2.098E+01    1.293E-06/-2.098E+01    2.133E-05/ 5.329E-15
       -1.165E-20/-5.672E-15   -1.505E-36/-1.234E-30   -1.437E-21/-1.178E-15
```

25

```
                          SED = 1.580E-04
   8   1.293E-06/ 2.098E+01    1.293E-06/-2.098E+01    2.133E-05/ 5.329E-15
      -2.033E-20/-9.900E-15   -1.505E-36/-1.234E-30   -3.592E-22/-2.946E-16
                          SED = 1.580E-04
   7   1.293E-06/ 2.098E+01    1.293E-06/-2.098E+01    2.133E-05/ 2.665E-14
      -2.033E-20/-9.900E-15   -1.078E-21/-8.837E-16   -3.592E-22/-2.946E-16
                          SED = 1.580E-04

 RESULTANT NODAL FORCES -
 NODE      FX          FY          FZ
    4   3.682E-16   5.244E+00  -3.467E-15
    5  -1.473E-16  -5.244E+00  -4.527E-15
    8  -2.209E-16  -5.244E+00   9.237E-17
    7  -1.972E-31   5.244E+00   7.901E-15

   ELEMENT NO.    4,      GLOBAL NODES -      5         6         9         8
                         Y COORD        5.00E-01  1.00E+00  1.00E+00  5.00E-01
                         Z COORD        5.00E-01  5.00E-01  1.00E+00  1.00E+00

   NODE     EPSXX/SIGXX          EPSYY/SIGYY          EPSZZ/SIGZZ
            GAMMAYZ/TAUYZ        GAMMAZX.TAUZX        GAMMAXY/TAUXY
    5   1.293E-06/ 2.098E+01    1.293E-06/-2.098E+01    2.133E-05/ 7.105E-15
        5.294E-21/ 2.578E-15   -1.505E-36/-1.234E-30   -1.437E-21/-1.178E-15
                          SED = 1.580E-04
    6   1.293E-06/ 2.098E+01    1.293E-06/-2.098E+01    2.133E-05/ 7.105E-15
        8.470E-21/ 4.125E-15    1.078E-21/ 8.837E-16   -1.437E-21/-1.178E-15
                          SED = 1.580E-04
    9   1.293E-06/ 2.098E+01    1.293E-06/-2.098E+01    2.133E-05/ 7.105E-15
        6.776E-21/ 3.300E-15    1.078E-21/ 8.837E-16   -3.592E-22/-2.946E-16
                          SED = 1.580E-04
    8   1.293E-06/ 2.098E+01    1.293E-06/-2.098E+01    2.133E-05/ 7.105E-15
        6.776E-21/ 3.300E-15   -1.505E-36/-1.234E-30   -3.592E-22/-2.946E-16
                          SED = 1.580E-04

 RESULTANT NODAL FORCES -
 NODE      FX          FY          FZ
    5   1.473E-16   5.244E+00  -2.801E-15
    6  -3.682E-16  -5.244E+00  -1.640E-15
    9  -2.650E-31  -5.244E+00   3.909E-15
    8   2.209E-16   5.244E+00   5.320E-16


 EQUILIBRIUM CHECK - SUM OF NODAL FORCES:

 Fx =  7.396E-32  Fy =  3.553E-15  Fz = -9.861E-32

***************************************************************************
        ***** Problem Finished. Press RUN Key for New Problem *****
***************************************************************************
```

26

Table 1. Suggested System Requirements

Computer:

       o HP9000 Series 200 computer with HP Basic 3.0 operating system
       o 2Mbyte RAM
       o Floating point math co-processor card
       o Basic 3.0 compiler software

Peripherals:

       o Dual 3.5 or 5.25 inch floppy disc drive
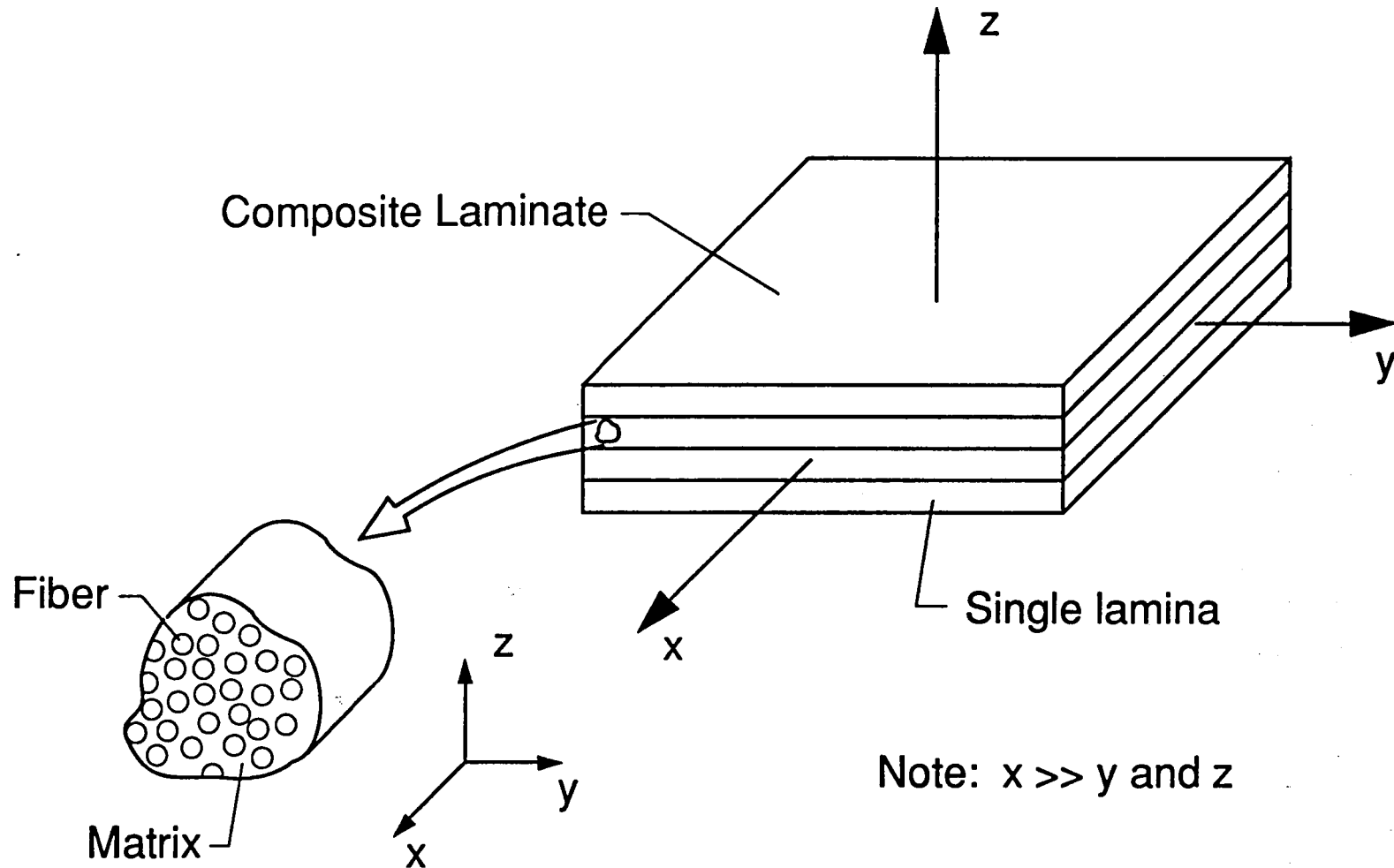   o 15Mbyte hard disc
       o Dot-matrix printer

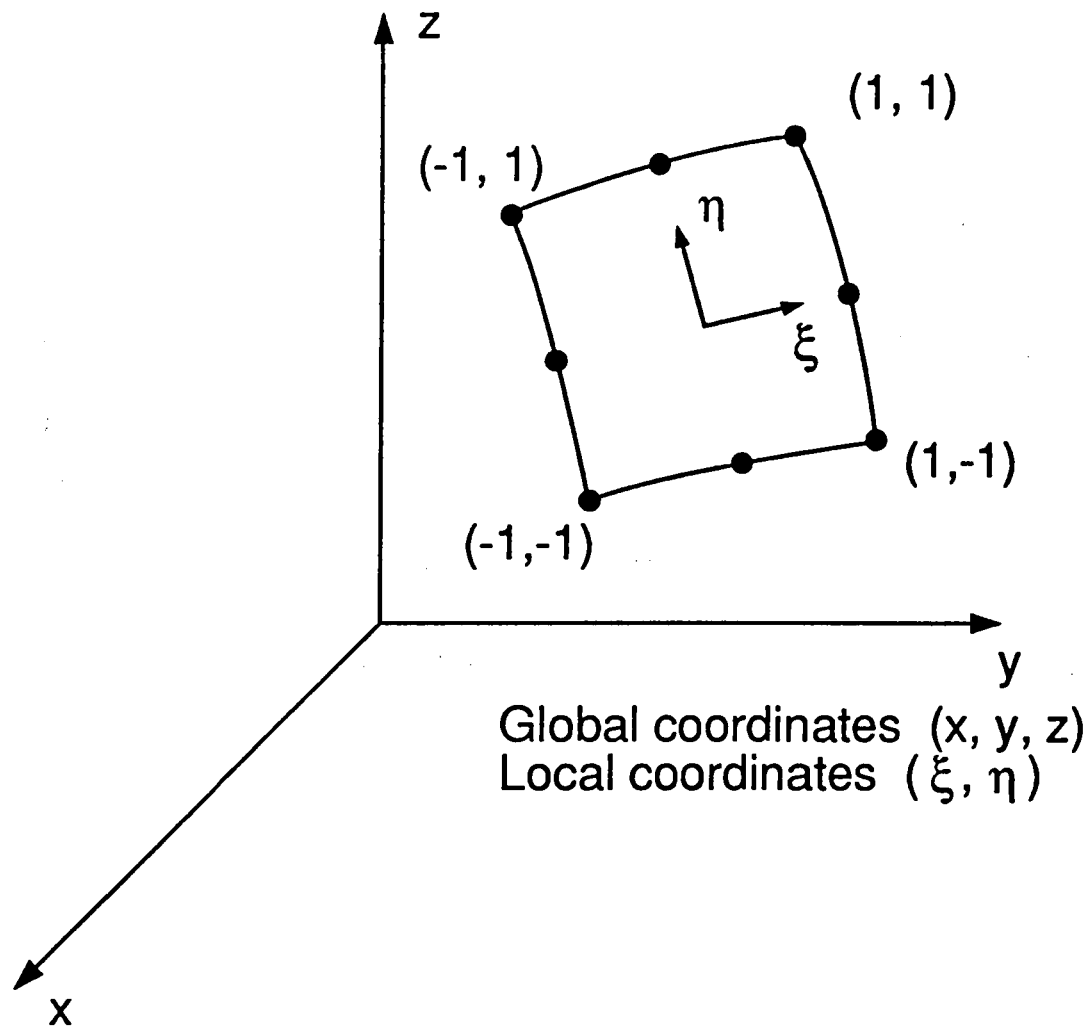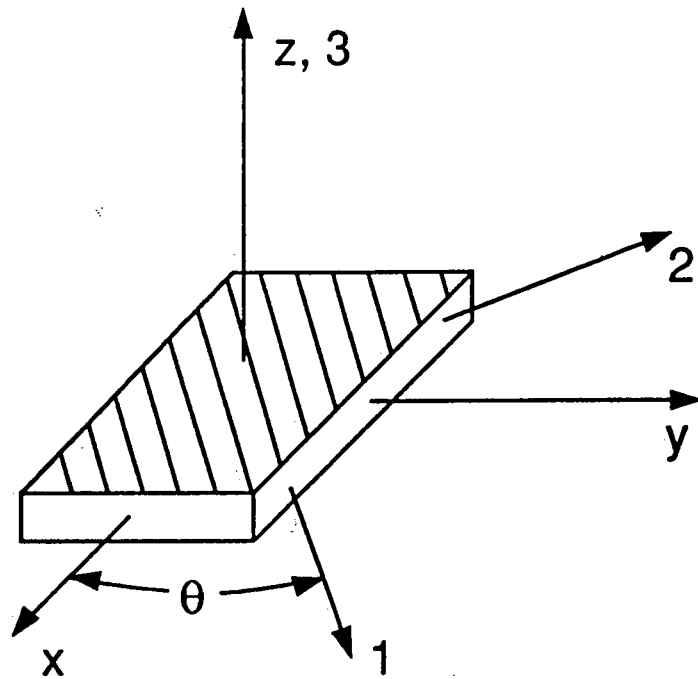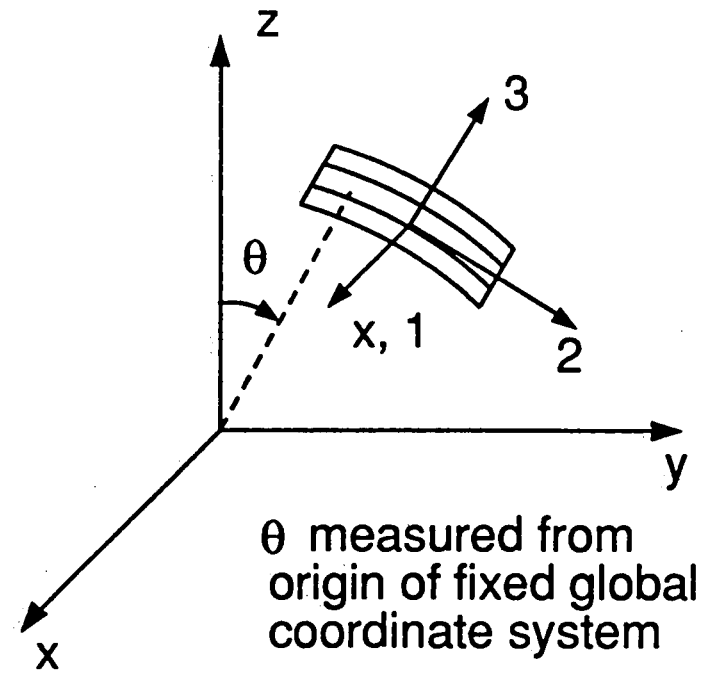Figure 1. Coordinate system for generalized plane strain analysis of fiber reinforced composites.

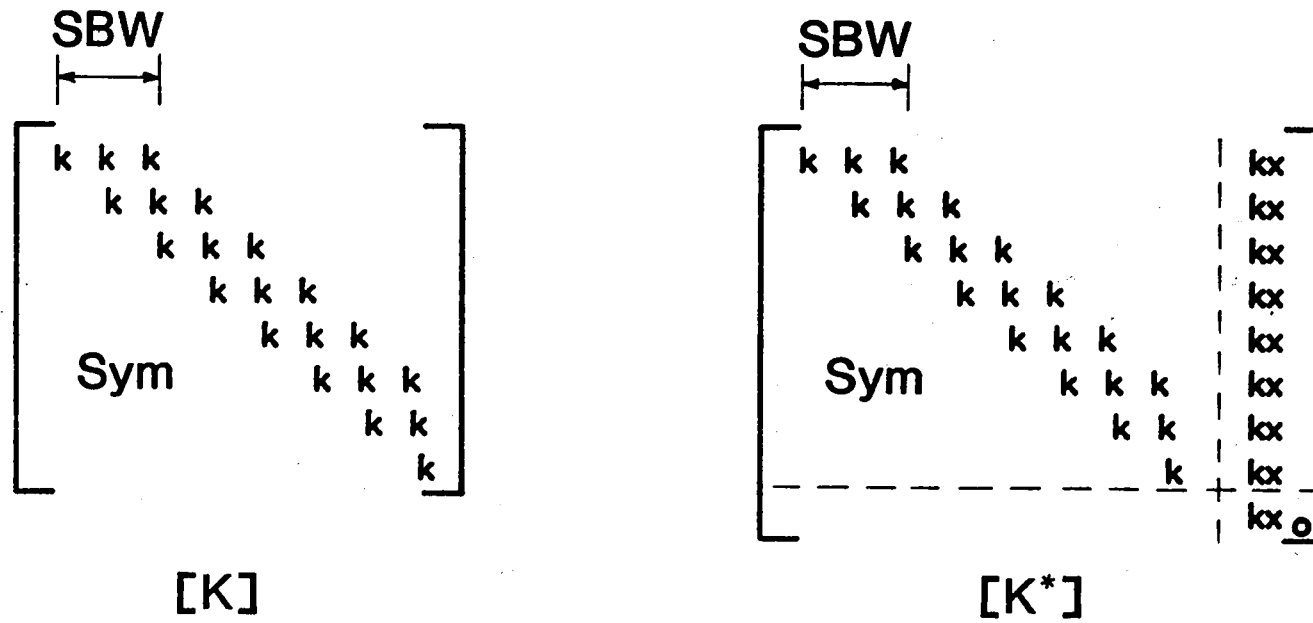**Figure 2.** Local and global coordinate systems for 4–node and 8–node isoparametric elements.

Plane orientation   Cylindrical orientation

Figure 3. Material orientation definitions.

Figure 4. Schematic diagram of stiffness matricies.
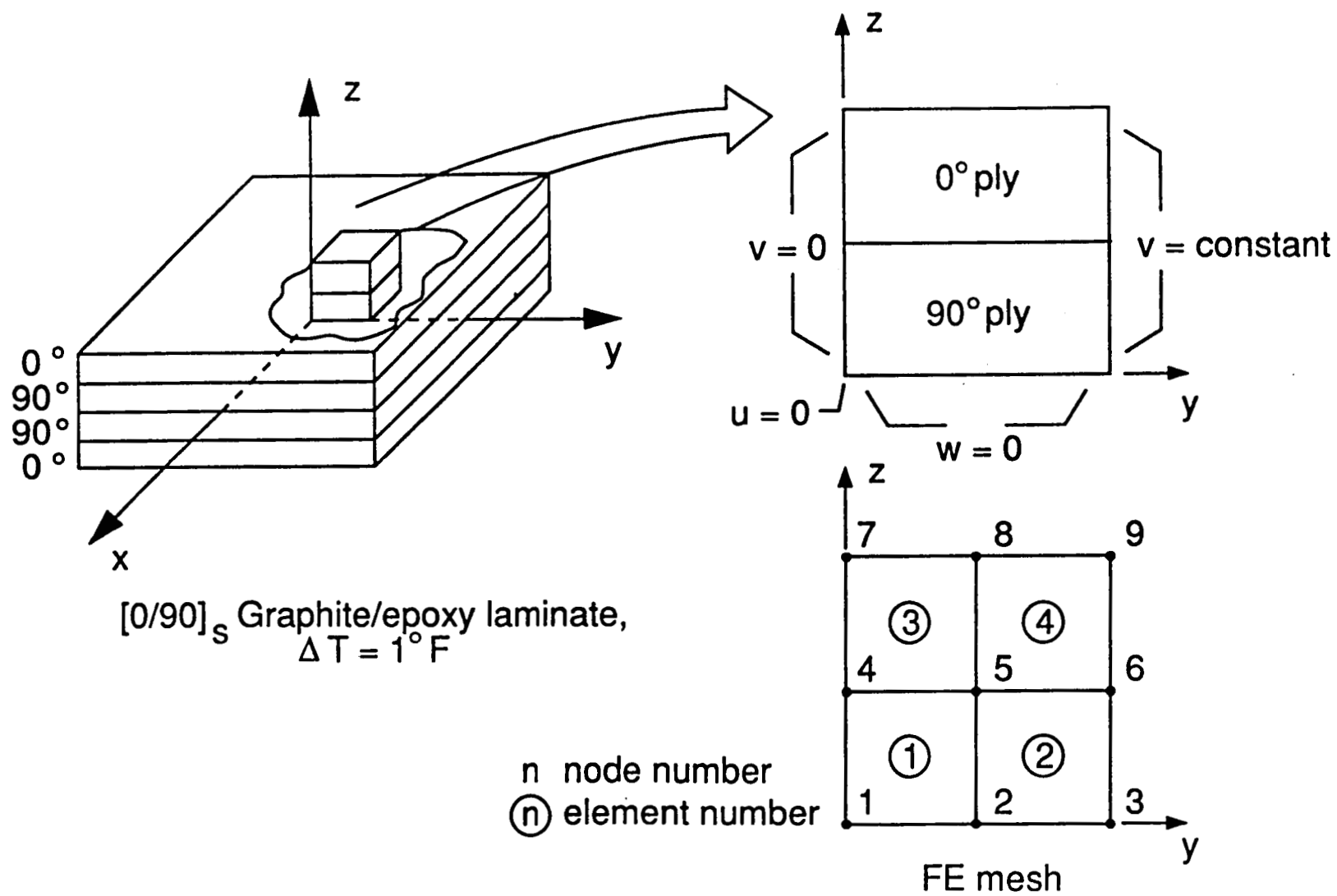
z

z

0° ply

$v = 0$ $v = $ constant

90° ply

$u = 0$ $w = 0$ y

0°
90°
90°
0°

y

x

[0/90]$_s$ Graphite/epoxy laminate,
$\Delta T = 1° F$

z

7   8   9

③   ④

4   5   6

n node number
(n) element number

①   ②

1   2   3

y

FE mesh

Figure 5. Sample problem geometry and FE mesh.

NASA

Federal
ganisati... Agrimini... ser...

Report Documentation Page

1. Report No. NASA TM-100670	2. Government Accession No.	3. Recipient's Catalog No.
4. Title and Subtitle Finite Element Composite Analysis Program (FECAP) for a Microcomputer		5. Report Date July 1988
		6. Performing Organization Code
7. Author(s) David E. Bowles		8. Performing Organization Report No.
		10. Work Unit No. 506-43-21-04
9. Performing Organization Name and Address NASA Langley Research Center Hampton, Virginia 23665-5225		11. Contract or Grant No.
		13. Type of Report and Period Covered Technical Memorandum
12. Sponsoring Agency Name and Address National Aeronautics and Space Administration Washington, DC 20546-0001		14. Sponsoring Agency Code

15. Supplementary Notes

16. Abstract

A special purpose finite element composite analysis program (FECAP) for analyzing composite material behavior with a microcomputer is described. The formulation assumes a state of generalized plane strain in a material consisting of two or more orthotropic phases. Loading can be mechanical and/or thermal. The theoretical background, computer implementation, and program users guide are described in detail. A sample problem is solved showing the required user input and computer generated output.

17. Key Words (Suggested by Author(s)) composite materials, finite elements, stress analysis, micromechanics, macromechanics	18. Distribution Statement Unclassified-Unlimited Subject Category 24		
19. Security Classif. (of this report) Unclassified	20. Security Classif. (of this page) Unclassified	21. No. of pages 33	22. Price A03

NASA FORM 1626 OCT 86